

# OpenSees Navigator

**Tony Yang**

*Assistant Professor*

**Department of Civil Engineering**

**University of British Columbia, Vancouver, Canada**

**Andreas Schellenberg**

*Project Engineer*

**Rutherford & Chekene Structural and Geotechnical Engineers**

**Eiji Kohama**

*Head of Group*

**Earthquake and Structural Dynamics Group**

**Port and Airport Research Institute**



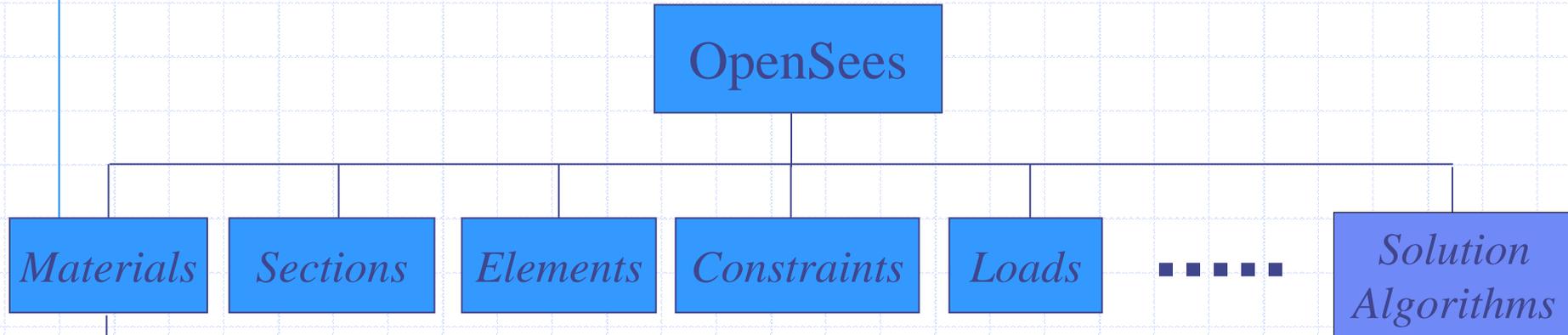
# What is OpenSees?

- ◆ **Open System for Earthquake Engineering Simulation.**
- ◆ A software *framework* for developing sequential and parallel, grid-enabled finite element applications in earthquake engineering.
- ◆ It is open source. Hence everybody can download and contribute to the code.
- ◆ Mostly used as a research tool, but it is fairly stable for regular use.



# OpenSees

- ◆ A robust finite element software.

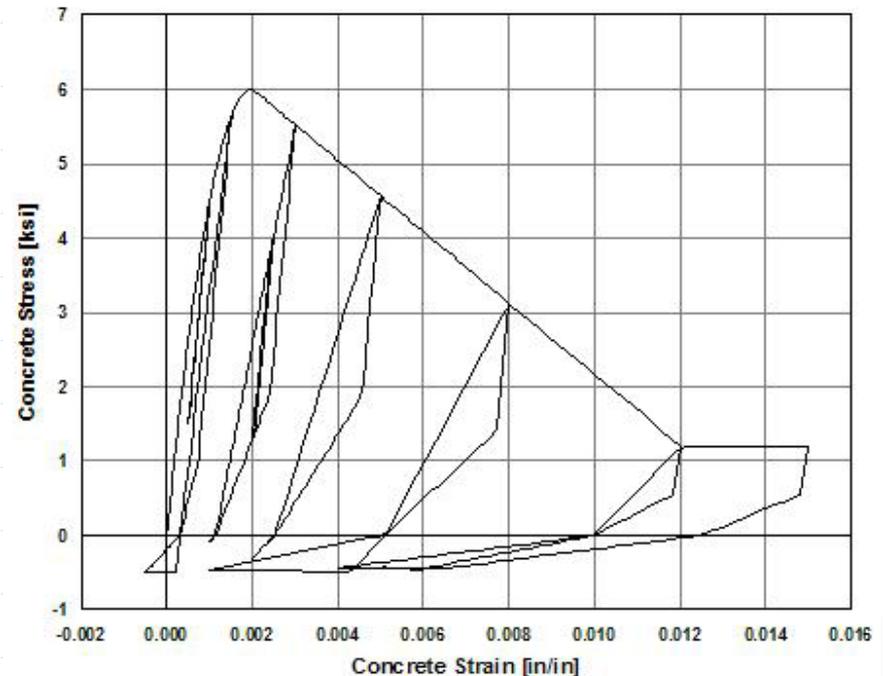


## *Uniaxial material*

- Elastic, -BoucWen, -Steel02,
- Concrete02, -Gap, -Fatigue,
- Hardening, -Hysteretic,
- Viscous, -MinMax, -Parallel,
- Series, ....

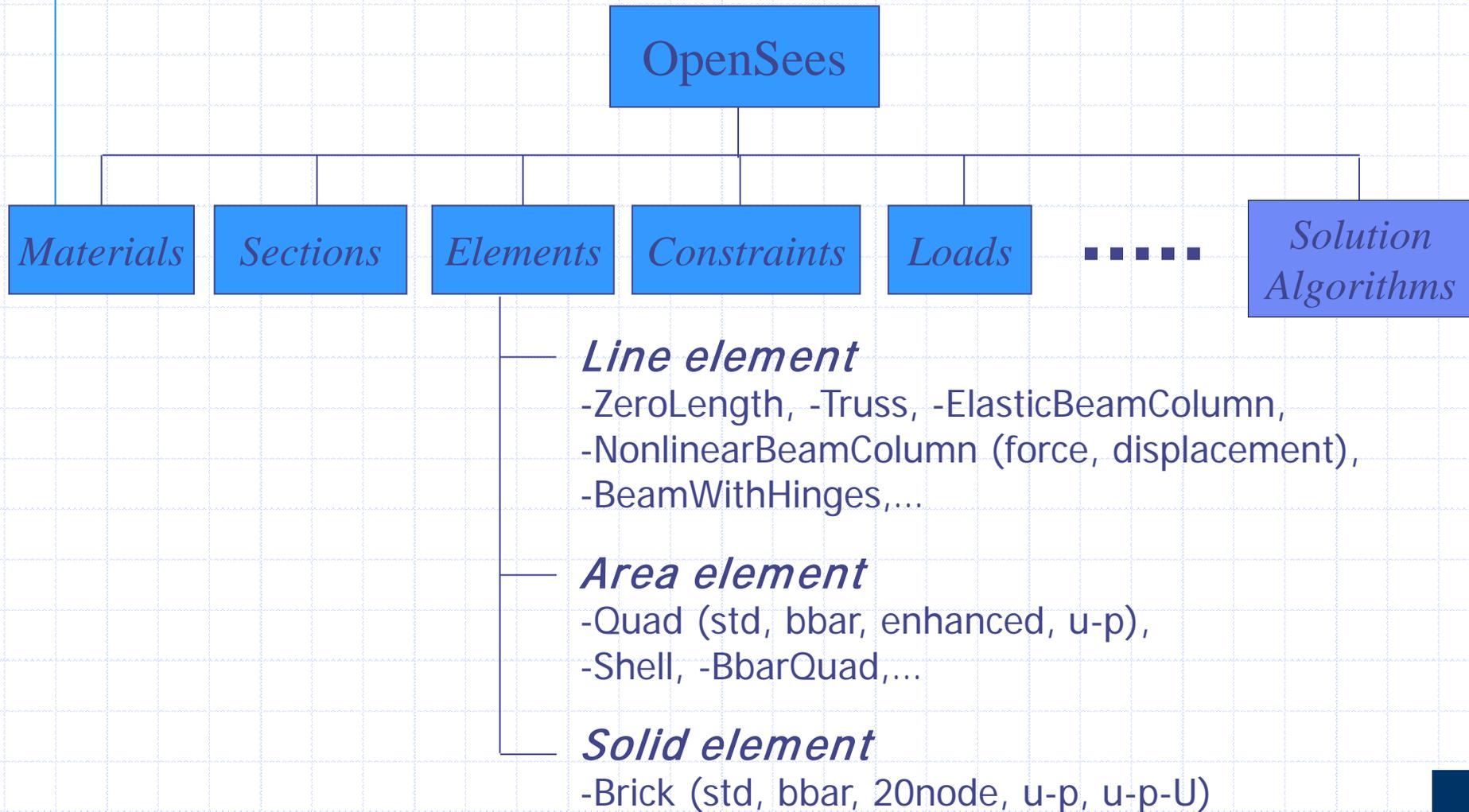
## *nD material*

- PlaneStress
- PressureDependMultiYield
- Template3DElastoPlastic, ...



# OpenSees

◆ A robust finite element software.



# Running OpenSees

- ◆ Tcl (tool command language) programming language
  - It is a string based command language.
  - Variables and variable substitution.
  - Expression evaluation.
  - Basic control structures (if , while, for, foreach).
  - Procedures (subroutine).
- ◆ Requires some time and learning effort to use it.



# TCL scripting models

◆ <sup>with</sup> Difficult to visualize the model.

2. source Units.tcl; # define units

◆ source ParamList.tcl; # load up parameter values (Model information)

4. source GMFiles.tcl; # load up ground-motion filenames

◆ Difficult to review the structural response.

5. foreach Xframe \$iXframe Hcol \$iHcol Lcol \$iLcol Lbeam \$iLbeam FRAME

Gblc \$iGblc GrhoCol \$iGrhoCol GPcol \$iGPcol GMfact \$iGMfact {

6. { source Static.tcl; # load procedure for static analysis }

7. { source Dynamic.tcl; # load procedure for dynamic analysis }

8. puts FRAME\$Xframe.....FRAME\$Xframe.....

9. puts STATIC\_ANALYSIS

10. Static \$Xframe \$Hcol \$Lcol \$Lbeam \$Gblc \$GrhoCol \$GPcol \$GMfact ;

11. puts DYNAMIC\_ANALYSIS

12. foreach GroundFile \$iGroundFile { GROUND MOTION

13. puts GroundMotion\$GroundFile

14. Dynamic \$Xframe \$Hcol \$Lcol \$Lbeam \$Gblc \$GrhoCol \$GPcol \$GMfact \$GroundFile;

15. }

16. }

# OpenSees Navigator

- ◆ Graphical user interface (GUI) for OpenSees.
- ◆ Pre- and post- processing for OpenSees.
- ◆ Design toolboxes: NSP, PBEE, AISC design checks, AISC database, response spectra for linear and bilinear systems, system identification and signal filtering.
- ◆ Can be used as a standalone application or coupled with MATLAB for development purpose.
- ◆ Being used by researchers from Asia, north and south America and Europe.

<https://www.google.com/analytics/reporting/maps?id=20238794&pdr=20100201-20100303&cmp=average#lts=1267723466859>



# Motivations

- ◆ Replace the TCL text input with graphical input.
- ◆ This will provide many robust plotting algorithms and it is very effective in generating the plots for engineering applications.
- ◆ OpenSees Navigator will create the OpenSees model and graphically display the results before, during or after a test.
- ◆ 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

Andreas Schellenberg Dep. of Civil and Env. Eng. University of California, Berkeley	Tony Yang, Ph.D. Dep. of Civil and Env. Eng. University of California, Berkeley	Eiji Kohama, Dr. Eng. 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 I

X0.47 Y0.49 Z0.00

# Define geometry: new model template

The screenshot displays the OpenSees Navigator 2.1 software interface. The main window is titled "Geometry Templates" and contains several structural models arranged in a grid. Each model is represented by a small diagram with a corresponding label in a white box with a blue border. The labels are: "Stick Model" (top left), "Beam Model" (top right), "EBF Model" (middle right), "Zipper Frame" (middle right), "Inverted-V Braced Frame" (bottom right), "Moment Frame" (bottom center), and "Single Area Mesh" (bottom left). The "Geometry Templates" window is overlaid on the main application window, which shows a menu bar with options like "File", "Edit", "View", "Define", "Assign", "Analyze", "Output", "Design", and "Help". The "File" menu is open, showing options such as "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 of the application window 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

**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) :**

**Num Segments in Col (NSC) :** 1

**Num Segments in Beam (NSB) :** 1

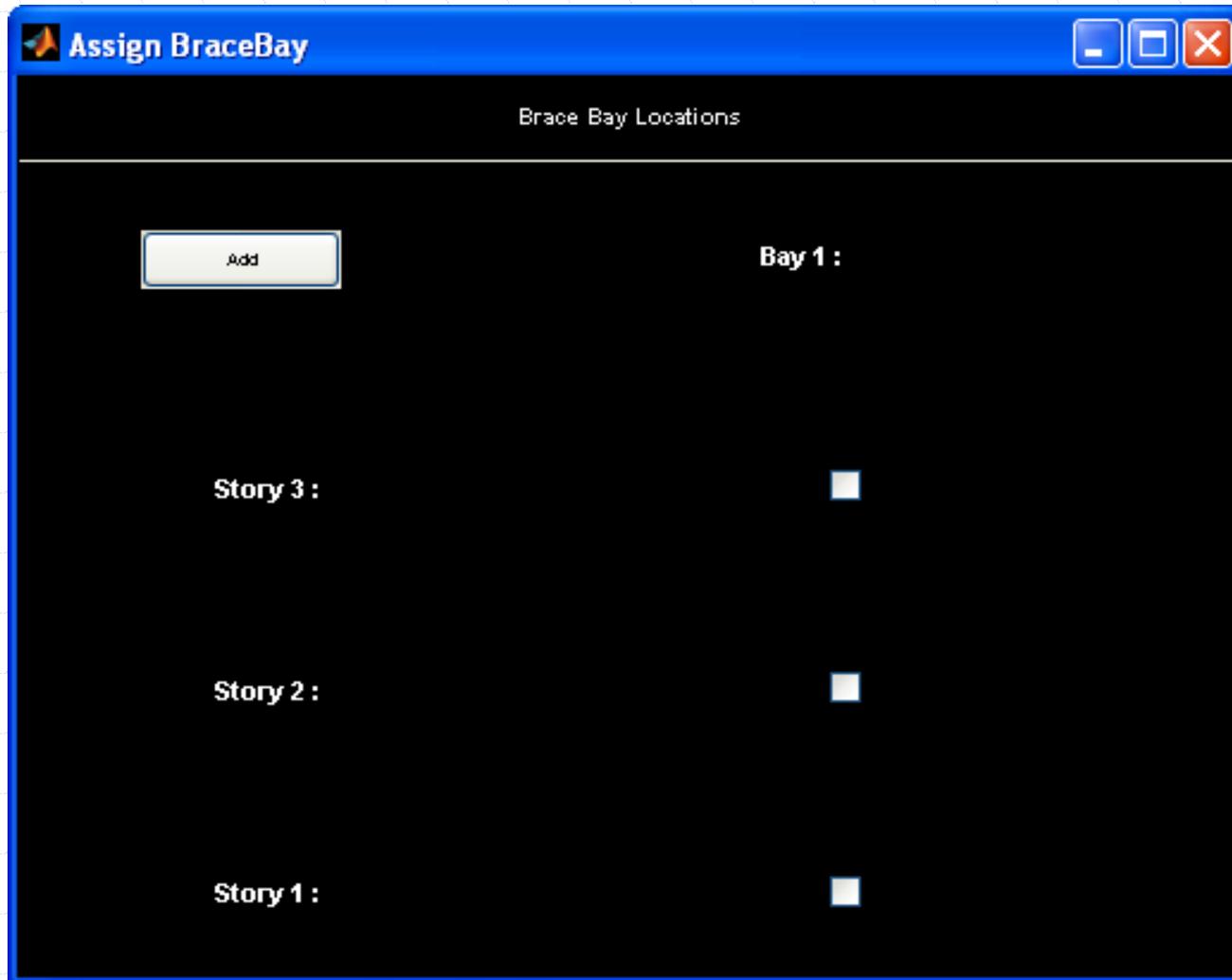
**Num Segments in Brace (NSBR) :** 2

**Num Segments in Z-Col (NSZC) :** 1

**Brace Offset (BraceOffset) :** None



# Define geometry: Zipper braced frame



# Define geometry: Zipper braced frame



# Define geometry: Zipper braced frame



# Define geometry: Zipper braced frame



# Define geometry: Zipper braced frame

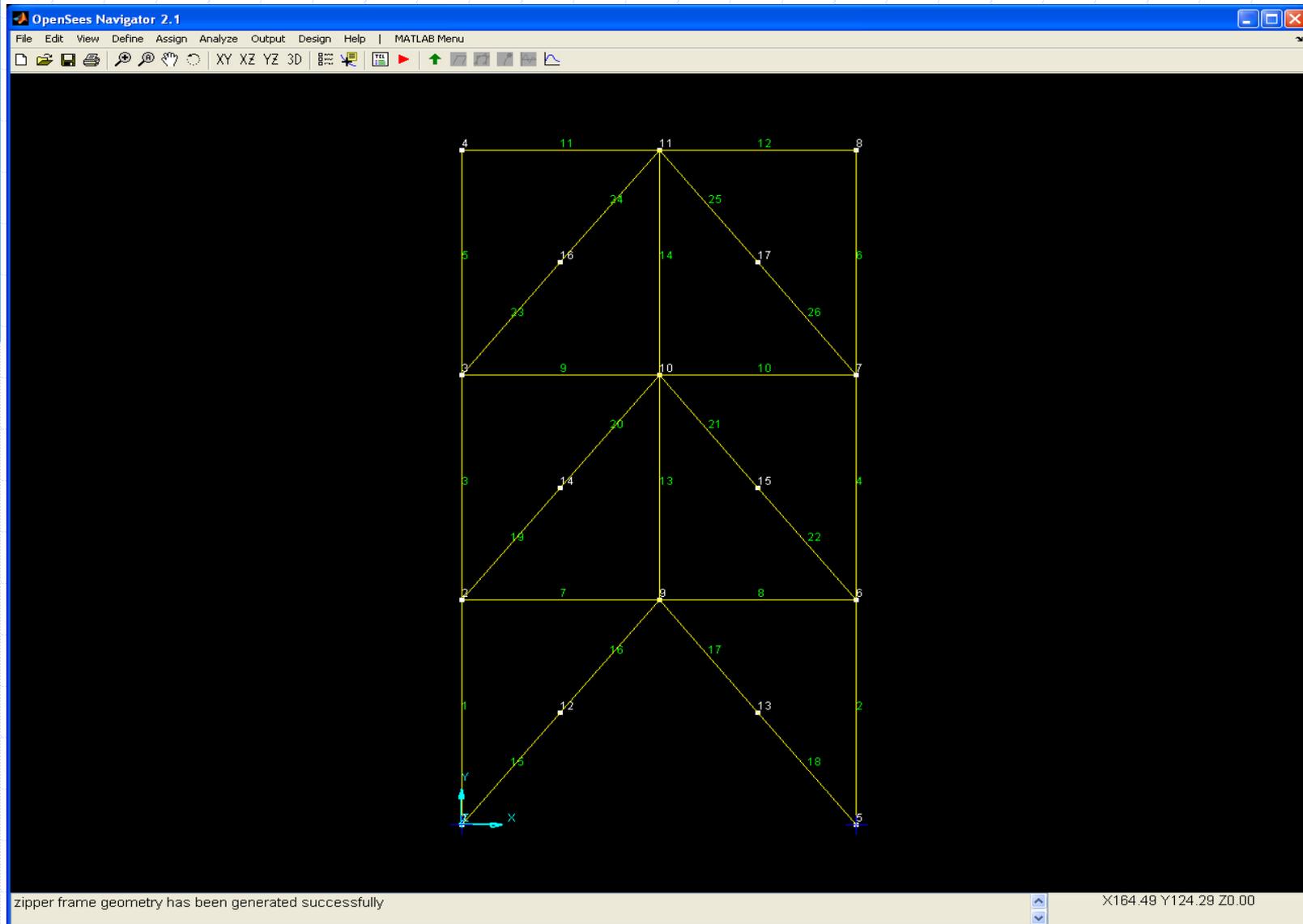
**Define Zipper Frame Geometry**

**Define Zipper Frame Geometry**

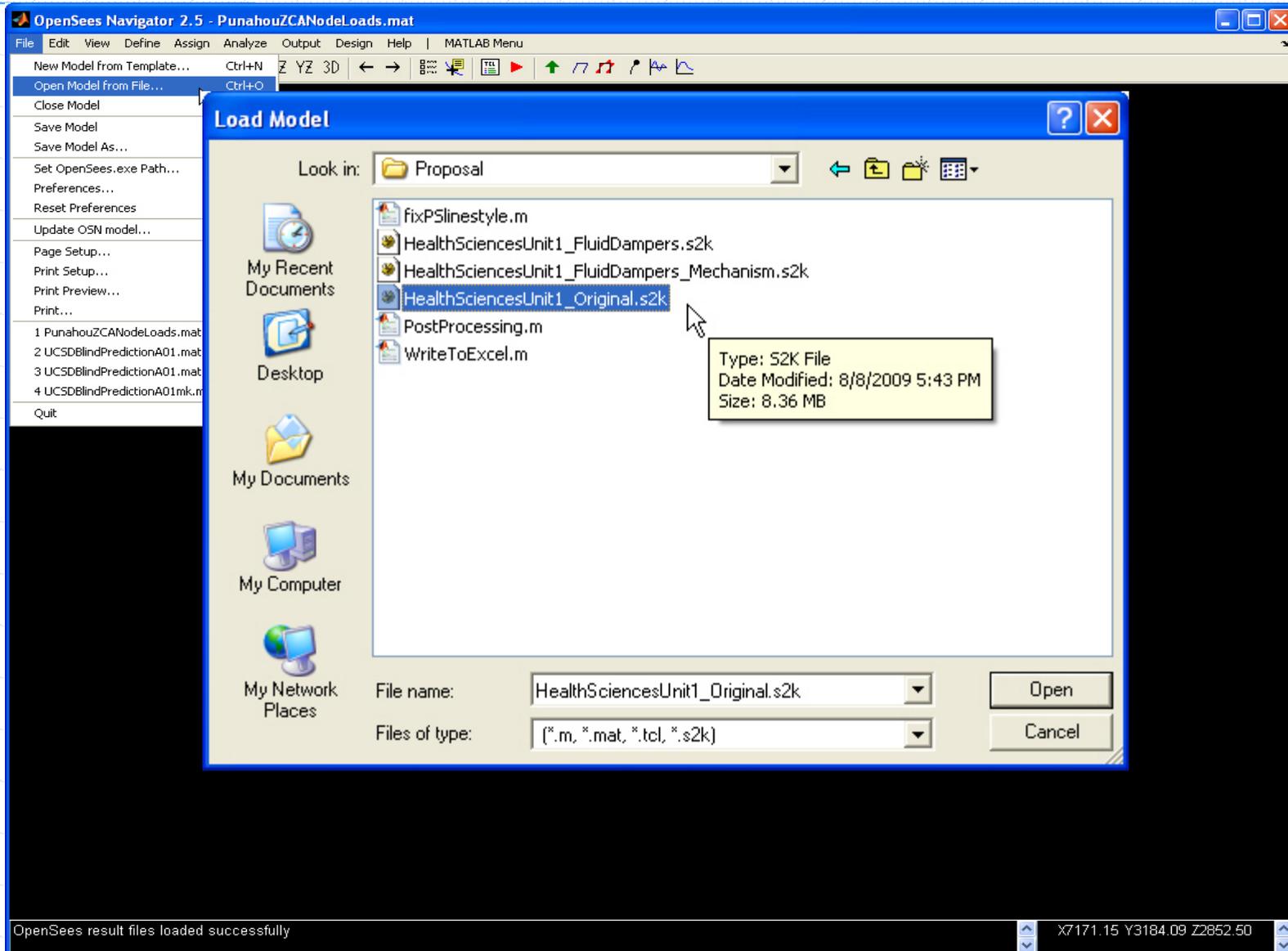
Dimension (ndm) :	2d	<input type="button" value="Generate"/>
Number of Stories (NOS) :	3	
Number of Bays (NOB) :	1	
Story Height (SH) :	52	
Bay Width (BW) :	80	
Boundary Condition (BC) :	pinned	
Brace Bay Config (BraceBay) :	<input type="button" value="BraceBay"/>	
Num Segments in Col (NSC) :	1	
Num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



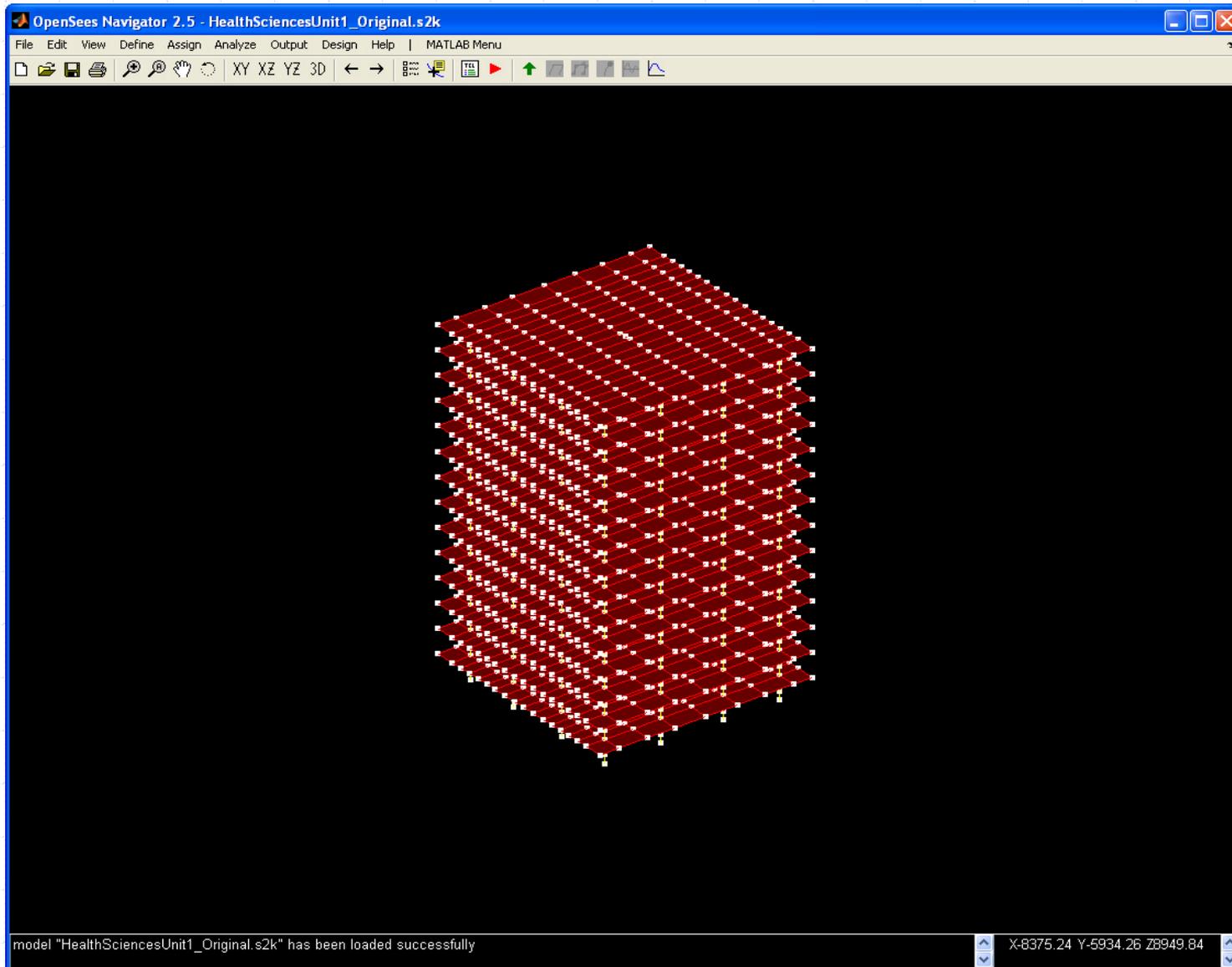
# View geometry: display



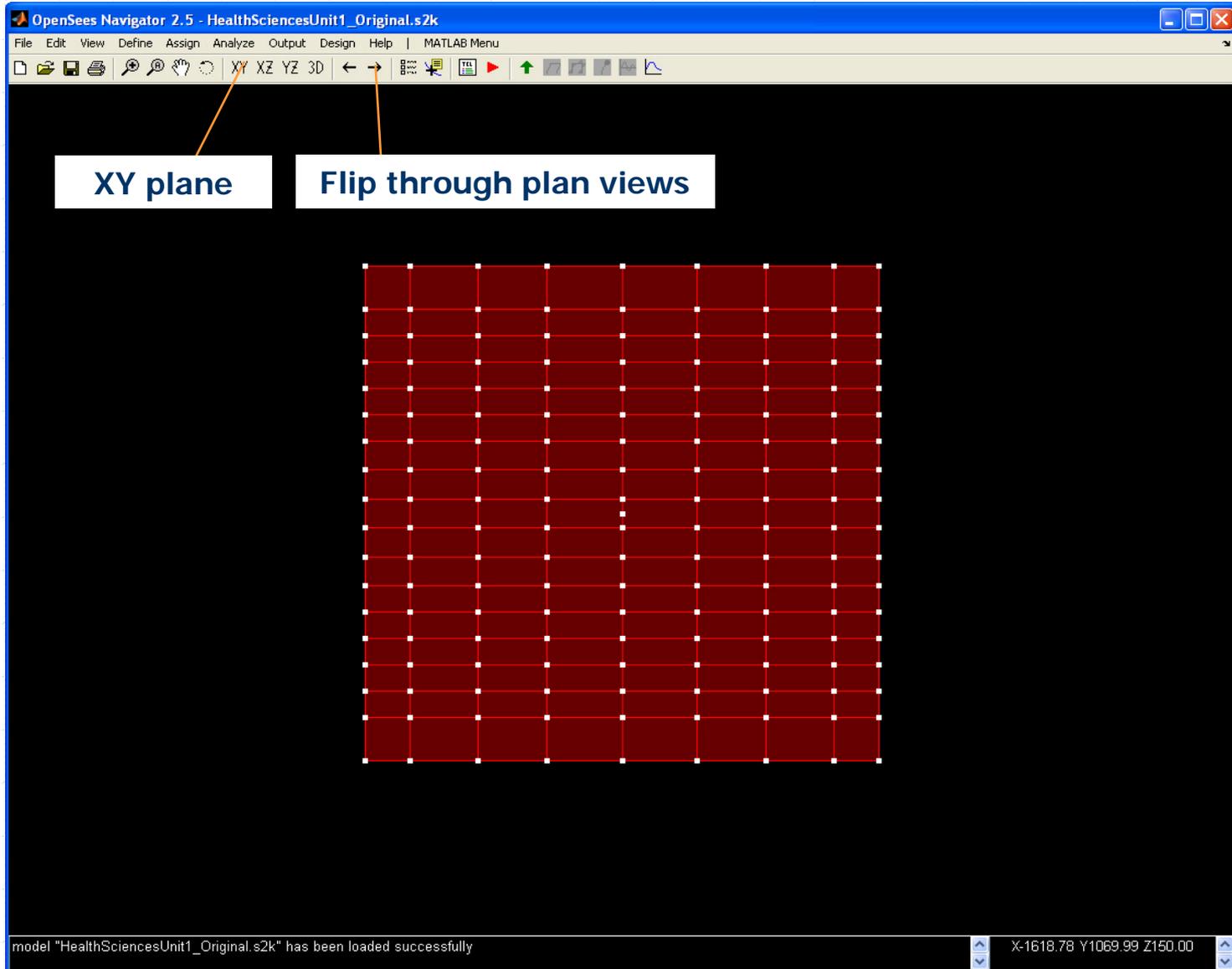
# Define geometry: \*.m, \*.mat, \*.tcl, \*.s2K



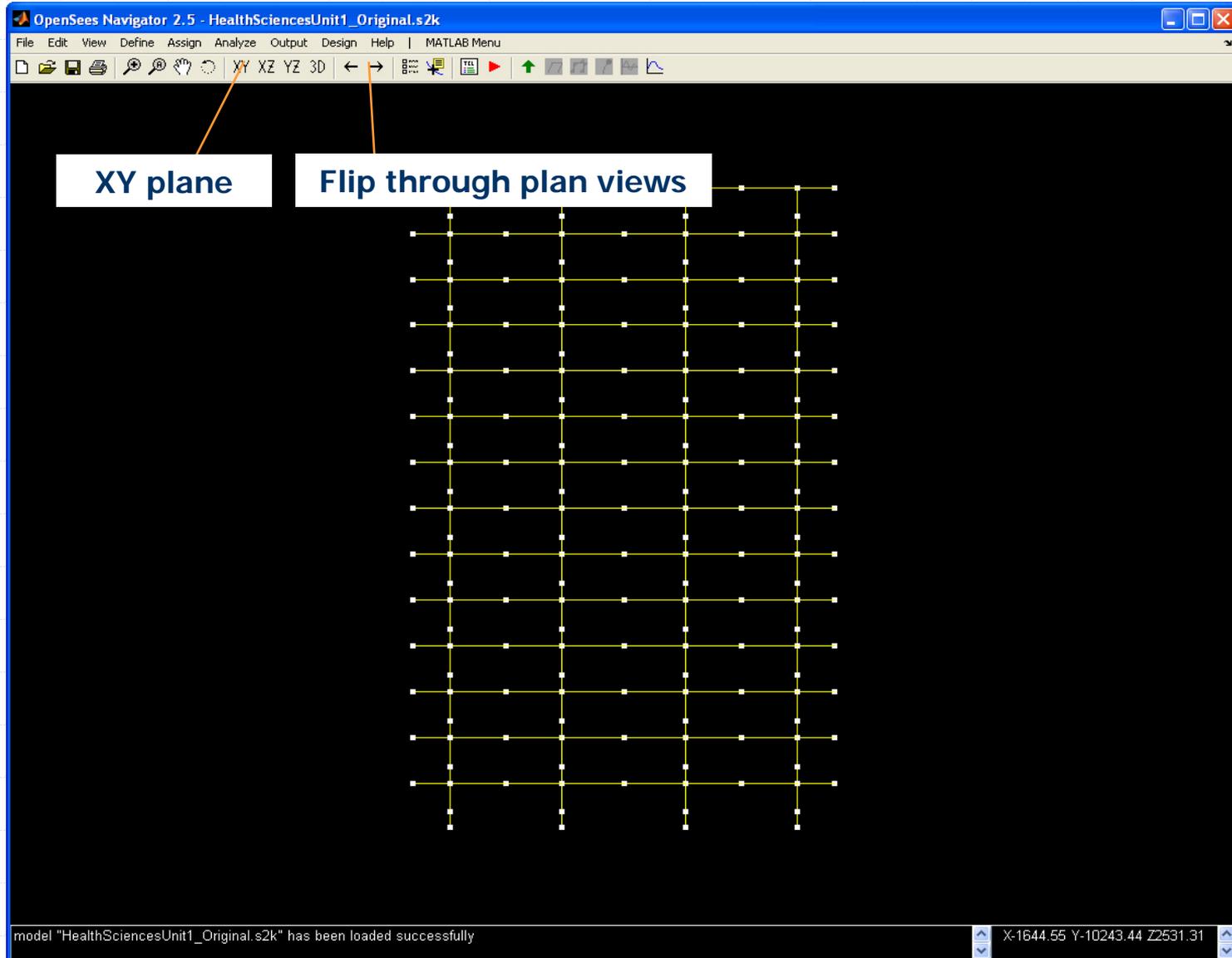
# Define geometry: import from SAP2000



# Define geometry: Plan



# Define geometry: plan



# 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

Clear all

Set Grid Lines...

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 (Ctrl+E)

**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.

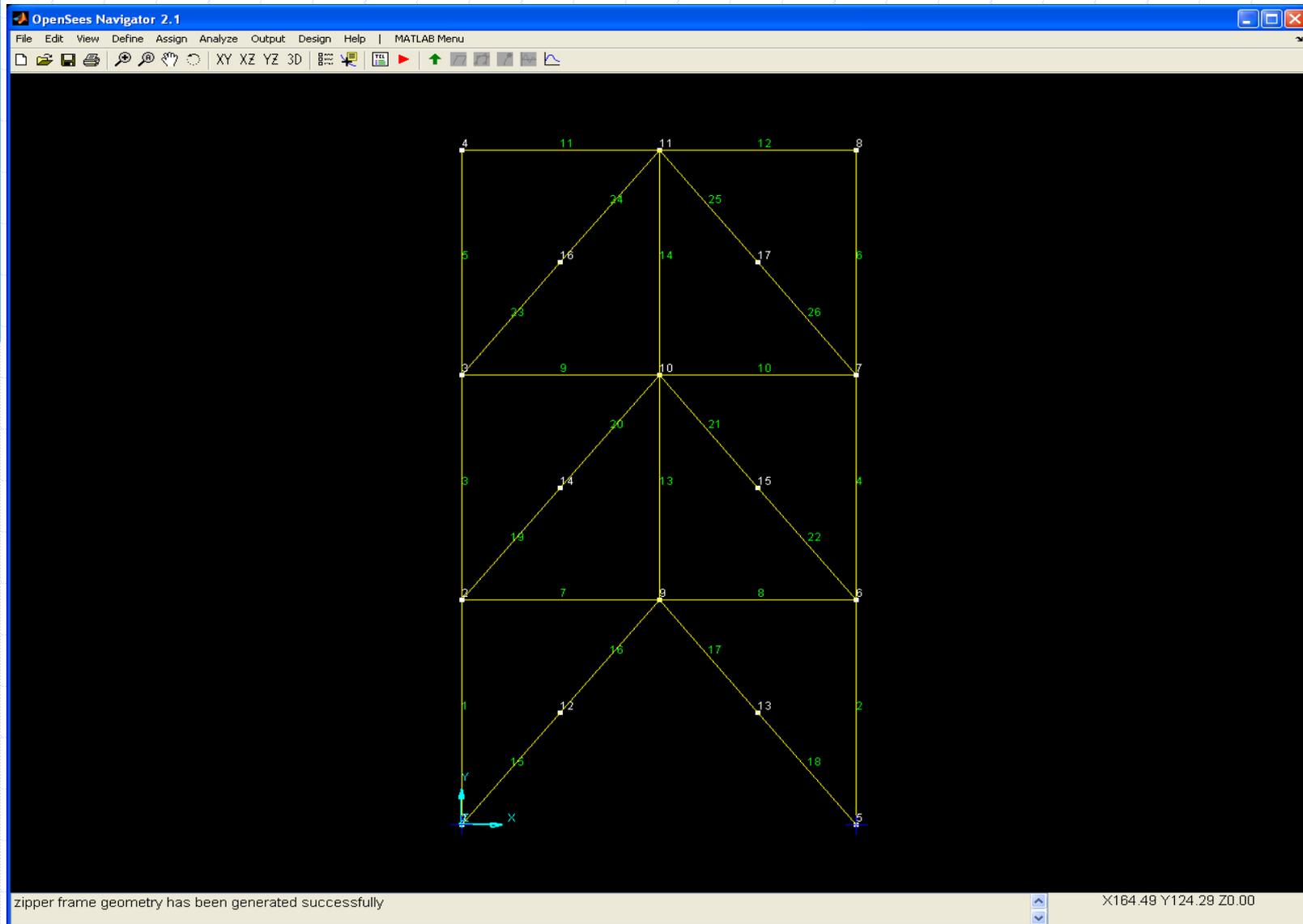
## Element:

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

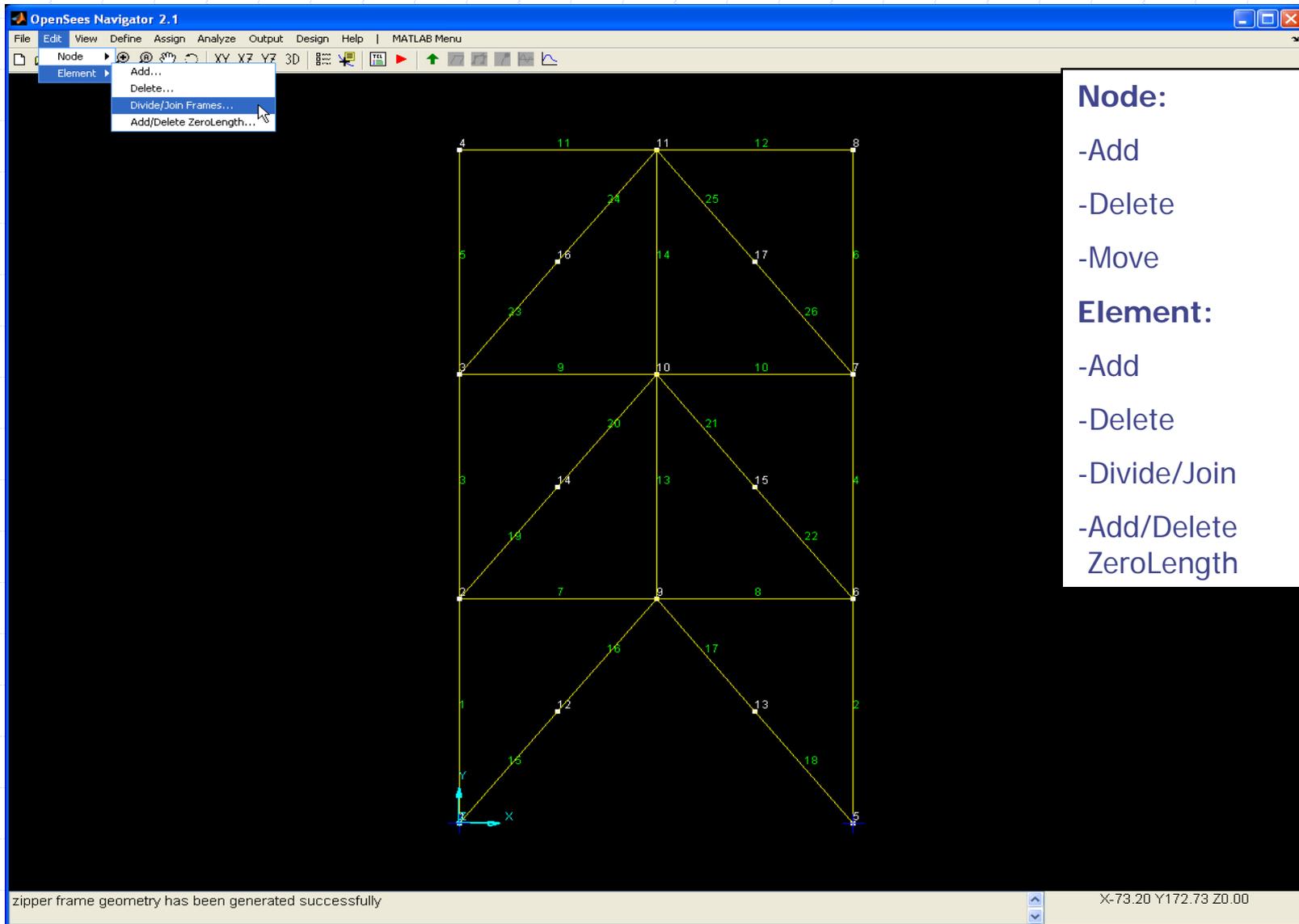
## General:

- Model
- Global Axes
- Grid Lines

# View geometry: display



# Edit geometry



The screenshot displays the OpenSees Navigator 2.1 interface. The main window shows a zipper frame geometry with nodes numbered 1 through 17 and elements numbered 1 through 30. A context menu is open over element 10, showing options: Add..., Delete..., Divide/Join Frames..., and Add/Delete ZeroLength... The status bar at the bottom indicates "zipper frame geometry has been generated successfully" and "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
- Concrete03
- Elastic
- ElasticNoTension
- ElasticPP
- ElasticPPGap
- Fatigue
- Hardening
- Hysteretic
- MinMax
- Parallel
- Series
- Steel01
- Steel02
- Viscous

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...

Define Uniaxial Material

Define Uniaxial Material

Add Material : BoucWen

Modify Material : ElasticDefault

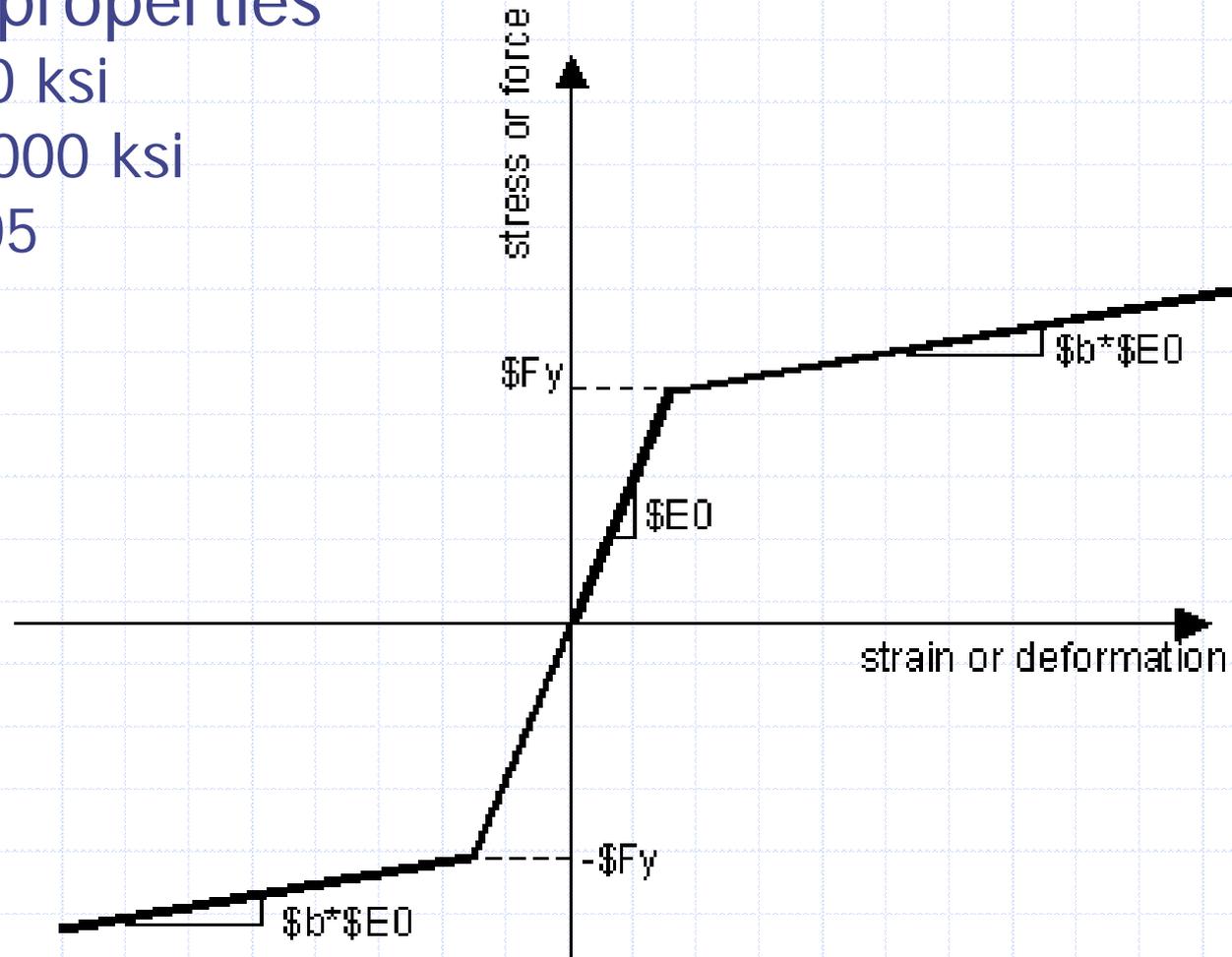
Delete Material : ElasticDefault

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 software interface. The main window shows a zipper frame geometry with nodes and elements. A menu is open, highlighting 'Uniaxial Materials...'. A dialog box titled 'Define Uniaxial Material' is overlaid on the main window. The dialog box contains three rows of controls:

Control	Value
Add Material :	BoucWen
Modify Material :	ElasticDefault
Delete Material :	ElasticDefault

The status bar at the bottom of the window displays the message: 'zipper frame geometry has been generated successfully'. The version information 'X-56.17 Y172.73 Z0.00' is visible in the bottom right corner.

# Define material: uniaxial materials

The screenshot displays the OpenSees Navigator 2.1 interface. The 'Define' menu is open, and the 'Uniaxial Materials...' option is selected. A dialog box titled 'Define Uniaxial Material' is in the foreground. The dialog contains the following fields:

Field	Value
Add Material :	BoucWen
Modify/Show Material :	ElasticDefault
Delete Material :	ElasticDefault A50

The background shows a zipper frame geometry with nodes and elements. The status bar at the bottom left reads 'zipper frame geometry has been generated successfully' and the bottom right shows 'X-56.17 Y172.73 Z0.00'.

# 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 'Define' menu is open, with 'Uniaxial Materials...' selected. A dialog box titled 'Define Uniaxial Material' is in the foreground. It contains the following fields:

- Add Material :** BoucWen
- Modify/Show Material :** ElasticDefault
- Delete Material :** ElasticDefault (with a dropdown menu showing 'ElasticDefault' and 'A50')

The background shows a zipper frame geometry with nodes and elements. The status bar at the bottom indicates 'zipper frame geometry has been generated successfully' and 'X-56.17 Y172.73 Z0.00'.

# Define material: uniaxial materials

The screenshot displays the OpenSees Navigator 2.1 software interface. The main window shows a zipper frame geometry with nodes numbered 1 through 18. The 'Define' menu is open, and the 'Uniaxial Materials...' option is selected. Two dialog boxes are overlaid on the main window:

- Define Uniaxial**: A dialog box with three buttons: 'Add Material', 'Modify/Show', and 'Delete Material'.
- Confirm deleting material**: A confirmation dialog box with a question mark icon and the text 'Are you sure you want to delete ElasticDefault?'. It has 'Yes' and 'No' buttons.

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

# Define material: uniaxial materials

The screenshot displays the OpenSees Navigator 2.1 interface. The 'Define' menu is open, with 'Uniaxial Materials...' selected. A dialog box titled 'Define Uniaxial Material' is in the foreground, containing the following settings:

Field	Value
Add Material :	BoucWen
Modify/Show Material :	A50
Delete Material :	A50

The background shows a structural model of a zipper frame with nodes and elements. The status bar at the bottom left reads 'zipper frame geometry has been generated successfully' and the bottom right shows 'X-56.17 Y172.73 Z0.00'.

# Define material: nD materials

The screenshot shows the OpenSees Navigator 2.3 software interface. The main window displays a 3D mesh model of a structure. A 'Define nD Material' dialog box is open in the foreground, allowing the user to define a new material. The dialog box has three rows of controls:

- Add Material :** A dropdown menu currently showing 'ElasticIsotropic'.
- Modify Material :** An empty dropdown menu.
- Delete Material :** An empty dropdown menu.

The background window shows the 'Materials' menu with 'nD Materials...' selected. The status bar at the bottom indicates 'single solid-mesh geometry has been generated successfully' and shows coordinates 'X0.00 Y0.00 Z0.00'.

## Templates:

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

## Under development:

- NewTemplate3DElastoPlastic
- ModelsLargeDeformation

# Define section: line sections

The screenshot displays the OpenSees Navigator 2.1 interface. The 'Define' menu is open, showing 'Line Sections...' selected. A 'Define Line Section' dialog box is open in the foreground, showing the following configuration:

Field	Value
Add Section :	Aggregator
Modify Section :	ElasticDefault
Delete Section :	ElasticDefault

The background shows a zipper frame geometry with nodes and elements. The status bar at the bottom indicates 'zipper frame geometry has been generated successfully' and 'X-50.42 Y171.91 Z0.00'.

## Templates:

- Aggregator
- Elastic
- Fiber
- Uniaxial

# Define line section: elastic section

The screenshot shows a dialog box titled "Define Elastic Section" with a blue title bar and standard window controls. The dialog contains the following fields and buttons:

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

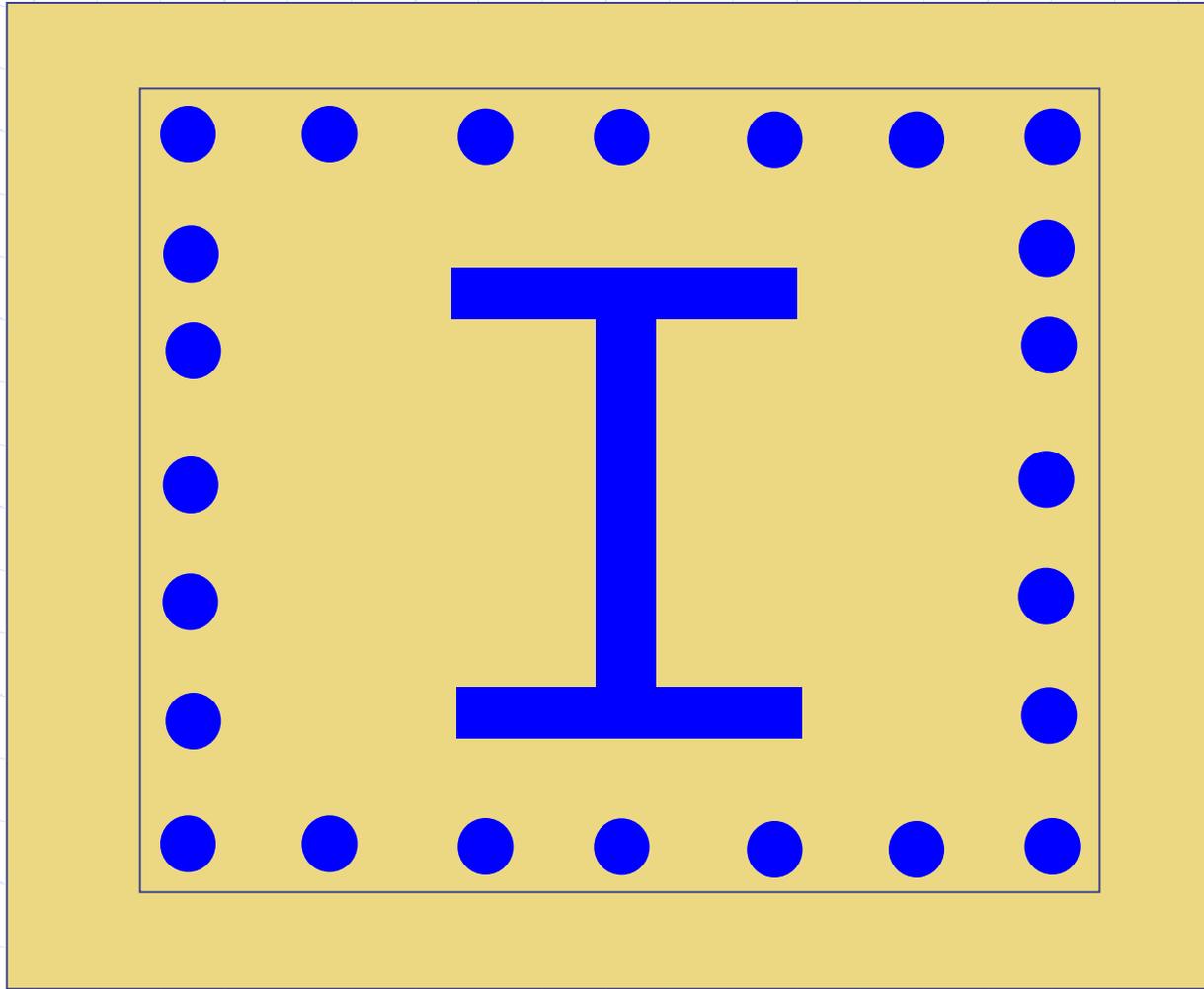
If the model is 3D

The screenshot shows a dialog box titled "Define Elastic Section" with a blue title bar and standard window controls. The dialog contains the following fields and buttons:

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**

Patch Name : CoreConcrete

Material Type : ConfinedConcrete

Lower Left Corner (yI,zI) : A50

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

*Optional Arguments :*

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 in the foreground, with 'Bidirectional' selected in the 'Add Section' dropdown. The background shows a structural model with nodes and elements. A status bar at the bottom indicates: 'section(2) with name "1stFloorBeam" has been defined/modified successfully' and coordinates 'X-47.75 Y173.55 Z0.00'.

**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 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 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, 11, 8, 5, 16, 14, 17, 24, 25, 7, 9, 6, 2, 1, 3, 5. A red triangle element 15 is highlighted. The status bar at the bottom indicates 'element [15] has been added successfully' and 'X-17.78 Y166.16 Z0.00'.

## Templates:

- CorotationalTruss
- DispBeamColumn
- ElasticBeamColumn
- ForceBeamColumn
- HingeBeamColumn
- Truss
- ZeroLength

# Define line element: ElasticBeamColumn

**Define ElasticBeamColumn Element**

Define ElasticBeamColumn Element

Element Name :	<input type="text" value="EColumn"/>	<input type="button" value="Add"/>
Modulus of Elasticity (E) :	<input type="text" value="29000"/>	<input type="button" value="Database"/>
Cross-Sectional Area (A) :	<input type="text" value="13.3"/>	
Moment of Inertia (Iz) :	<input type="text" value="248"/>	

**Select Section from Database**

Select Section from Database

Database :	<input type="text" value="AISC"/>	<input type="button" value="Select"/>
Section Name :	<input type="text" value="W10X45"/>	
Direction :	<input type="text" value="strong"/>	



# Define line element: ForceBeamColumn

**Define ForceBeamColumn Element**

Element Name : 1stStoryColumn

Number Intergration Points (NIP) : 5

Section Type : 1stStoryColumn

*Optional Arguments :*

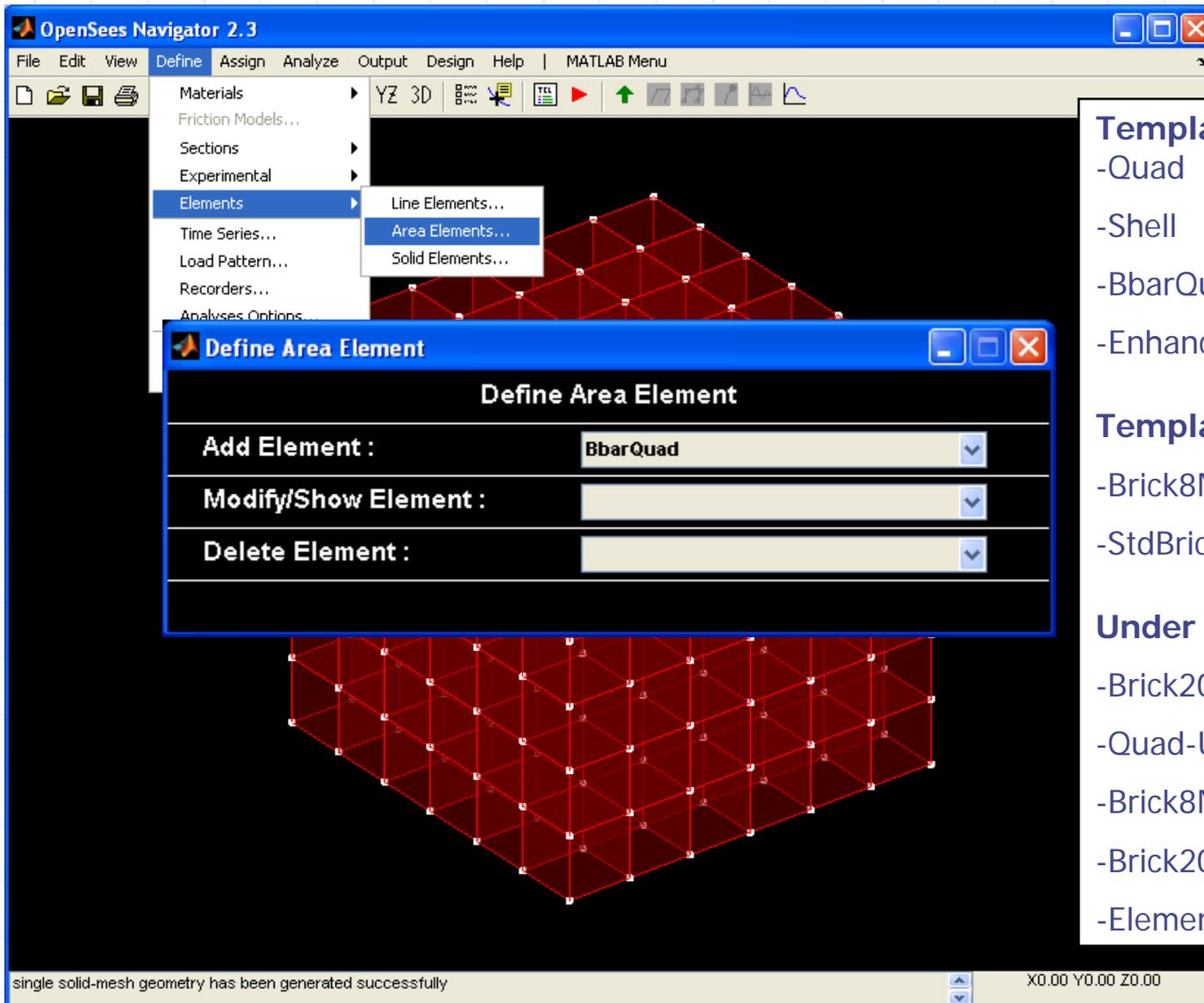
Mass Density (massDens) : 0.

Maximum Iterations (maxlters) : 10

Tolerance (tol) : 1E-8



# Define element: area and solid elements



## Templates (area):

- Quad
- Shell
- BbarQuad
- EnhancedQuad

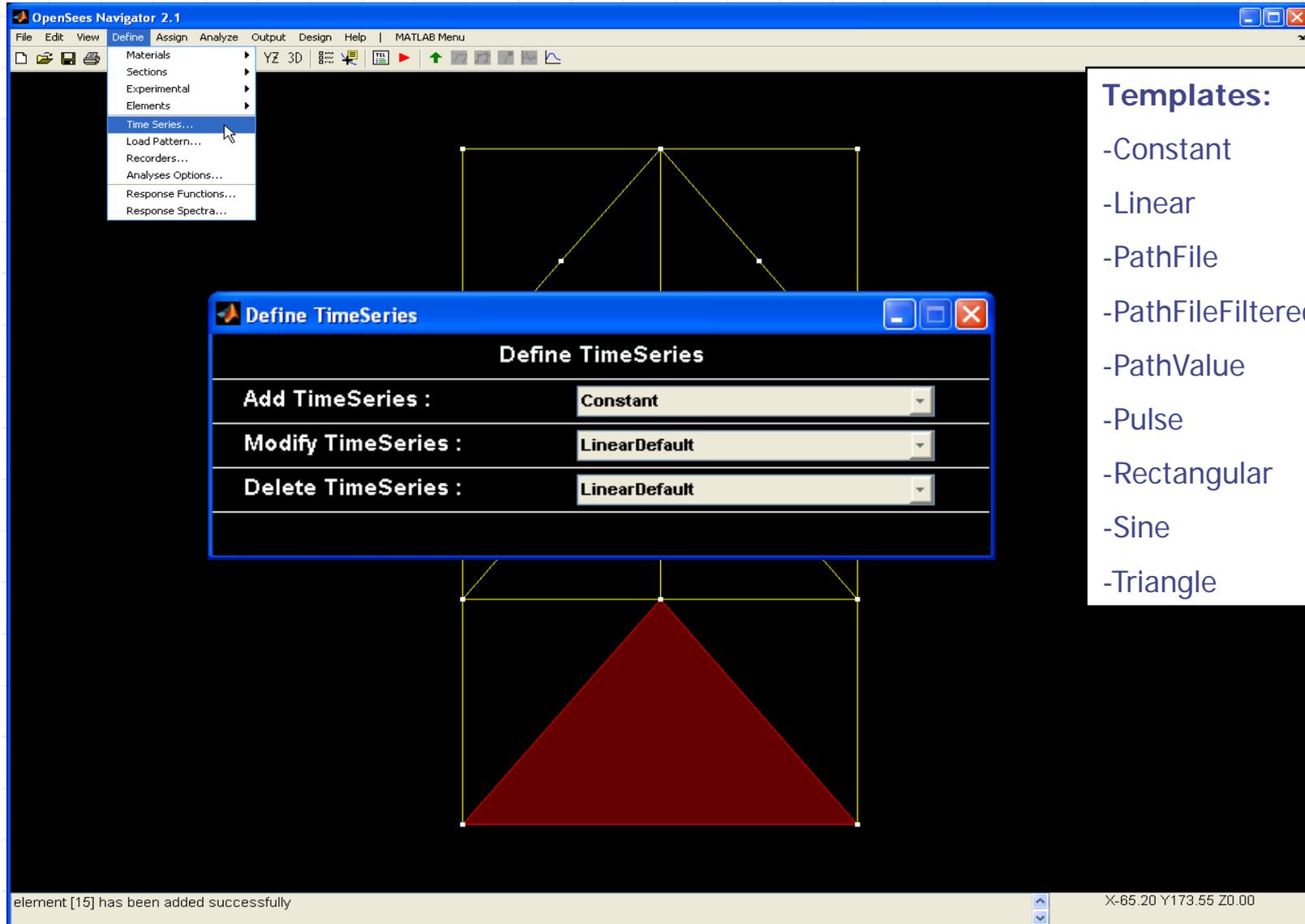
## Templates (solid):

- 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, and the 'Time Series...' option is selected. A 'Define TimeSeries' dialog box is displayed in the foreground, showing three dropdown menus: 'Add TimeSeries' set to 'Constant', 'Modify TimeSeries' set to 'LinearDefault', and 'Delete TimeSeries' set to 'LinearDefault'. The background shows a 2D truss structure with a red triangular element highlighted. A status bar at the bottom indicates 'element [15] has been added successfully' and 'X-65.20 Y173.55 Z0.00'.

**Templates:**

- Constant
- Linear
- PathFile
- PathFileFiltered
- PathValue
- Pulse
- Rectangular
- Sine
- Triangle

# Define TimeSeries: PathFile

**Define PathFile Time Series**

TimeSeries Name  
Time Interval  
Time File Name  
Value File Name  
Load Factor

**Choose Value File**

Look in: MATLAB

My Recent Documents  
Desktop  
My Documents  
My Computer  
My Network Places

File name: \*.txt;\*.dat;\*.acc;\*.thf;\*.AT2;\*.VT2;\*.DT2  
Files of type: (\*.txt;\*.dat;\*.acc;\*.thf;\*.AT2;\*.VT2;\*.DT2)

Open  
Cancel

Ground Acceleration [in/sec<sup>2</sup>]  
Time [sec]

Time [sec]	Ground Acceleration [in/sec <sup>2</sup> ]
5.5	-250
6.0	-250
6.5	-250
7.0	-250
7.5	-250
8.0	-250
8.5	-250
9.0	-250
9.5	-250



# Define LoadPattern:

The screenshot shows the OpenSees Navigator 2.0 interface. The 'Define' menu is open, with 'Load Pattern...' selected. A dialog box titled 'Define LoadPattern' is displayed in the foreground, containing three rows of controls:

Control Type	Value
Add LoadPattern :	Plain
Modify LoadPattern :	PlainDefault
Delete LoadPattern :	PlainDefault

The background shows a 2D structural model of a truss. A triangular load pattern is applied to the top chord, highlighted in red. The status bar at the bottom indicates: 'time-series(2) with name "SACNF01" has been defined/modified successfully' and 'X-41.59 Y157.54 Z0.00'.

- Templates:**
- Plain
  - UniformExcitation
  - MultipleSupport

# Define recorder

The screenshot shows the OpenSees Navigator 2.0 interface. The 'Define' menu is open, and the 'Recorders...' option is selected. A 'Define Recorder' dialog box is displayed in the foreground, showing the following configuration:

Define Recorder	
Add Recorder :	BeamColumn Element
Modify Recorder :	DefoShape
Delete Recorder :	DefoShape

The background shows a structural model of a truss with a red triangular element highlighted. The status bar at the bottom indicates: 'load-pattern(2) with name "SACNF01" has been defined/modified successfully' and 'X-69.51 Y173.55 Z0.00'.

## Templates:

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

## Defaults:

- DefoShape
- Eigenvector

# Define recorder: node recorder

**Define Node Recorder**

Recorder Name : DefoShape **Add**

Node Number(s) : all  Envelope

Deformations :

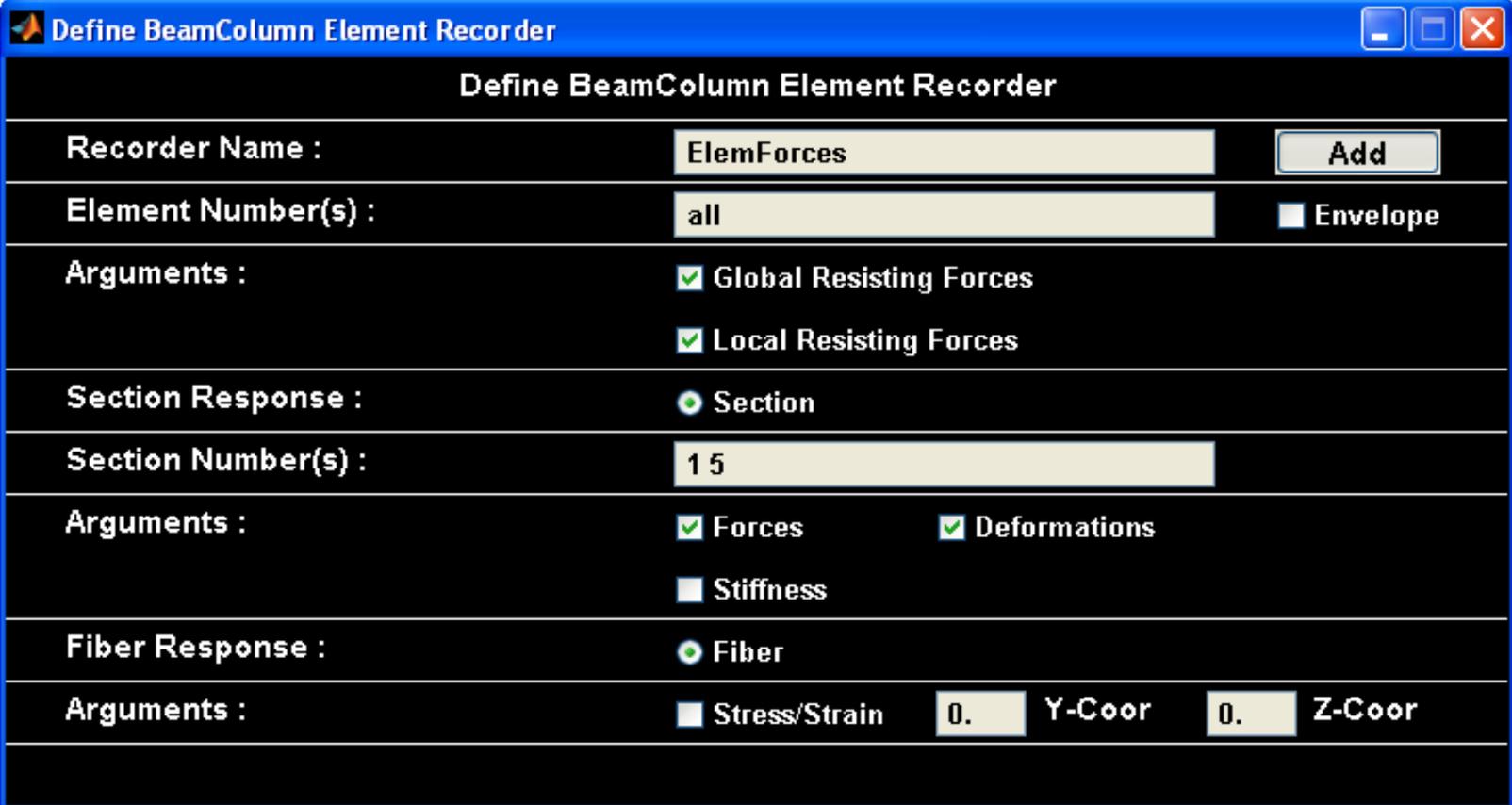
- Displacements
- Velocities
- Accelerations
- Incremental Displacements
- Incremental Delta Displacements
- Eigenvectors

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

The screenshot shows the OpenSees Navigator 2.0 interface. The 'Define' menu is open, and 'Analyses Options...' is selected. A dialog box titled 'Define Analysis Options' is displayed in the foreground. The dialog box contains three rows of controls:

- Add Analysis Options :** A button labeled 'New Analysis Options'.
- Modify Analysis Options :** A dropdown menu currently showing 'StaticDefault'.
- Delete Analysis Options :** A dropdown menu currently showing 'StaticDefault'.

In the background, a structural model is visible, consisting of a rectangular frame with a diagonal member. The bottom portion of the model is highlighted in red. A status bar at the bottom of the window displays the message: 'load-pattern(2) with name "SACNF01" has been defined/modified successfully'. The system clock in the bottom right corner shows 'X-79.16 Y173.55 Z0.00'.

## Defaults:

- StaticDefault
- TransientDefault
- EigenDefault

# 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:**

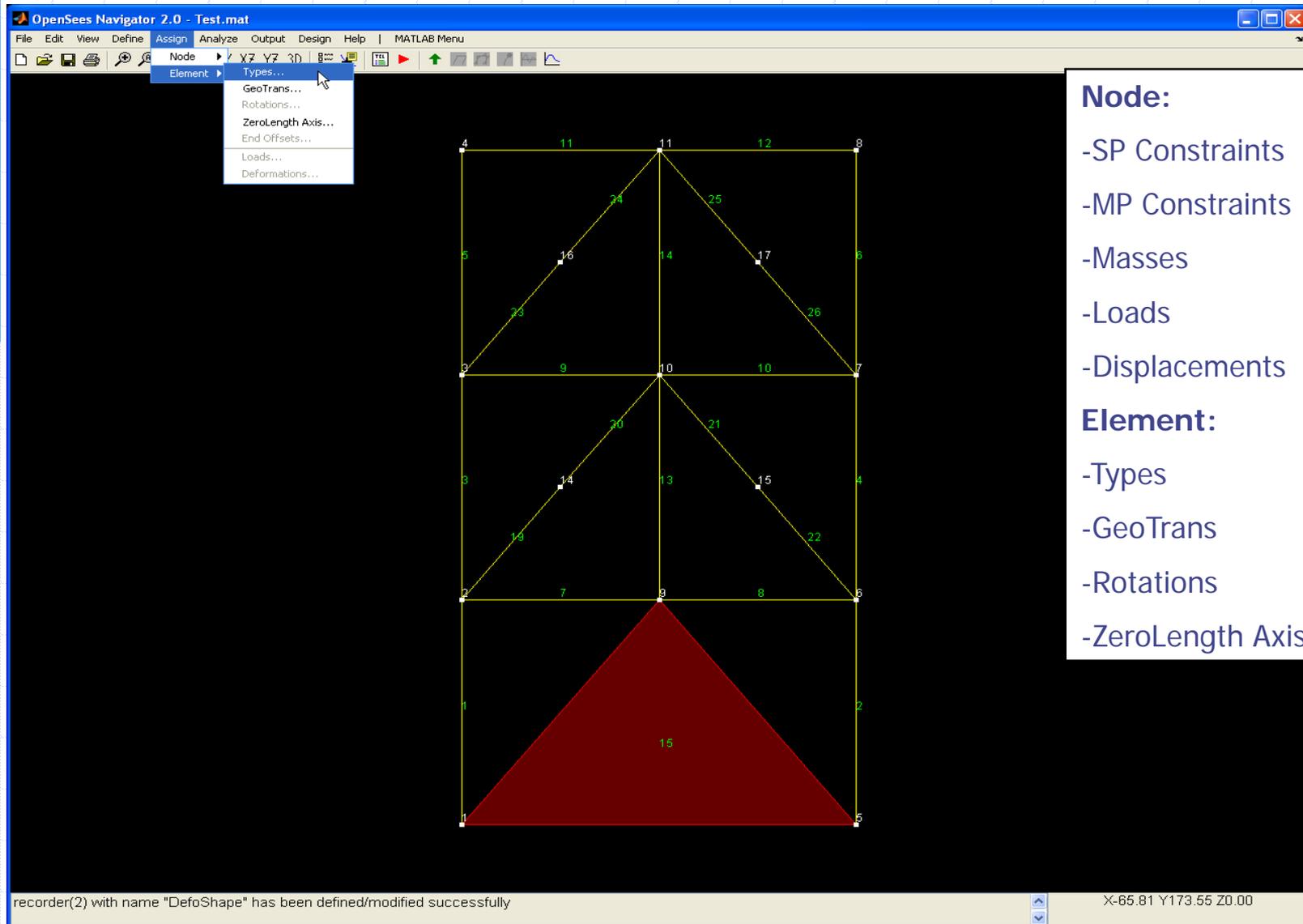
For example use AlphaOS Method for Hybrid Simulation

## **Solution Algorithm:**

The AlphaOS Method requires a Linear solution algorithm



# Assign menu



OpenSees Navigator 2.0 - Test.mat

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

Node ▾ X Y Z 3D

Element ▾

- Types...
- GeoTrans...
- Rotations...
- ZeroLength Axis...
- End Offsets...
- Loads...
- Deformations...

**Node:**

- SP Constraints
- MP Constraints
- Masses
- Loads
- Displacements

**Element:**

- Types
- GeoTrans
- Rotations
- ZeroLength Axis

recorder(2) with name "DefoShape" has been defined/modified successfully

X-65.81 Y173.55 Z0.00

# 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**

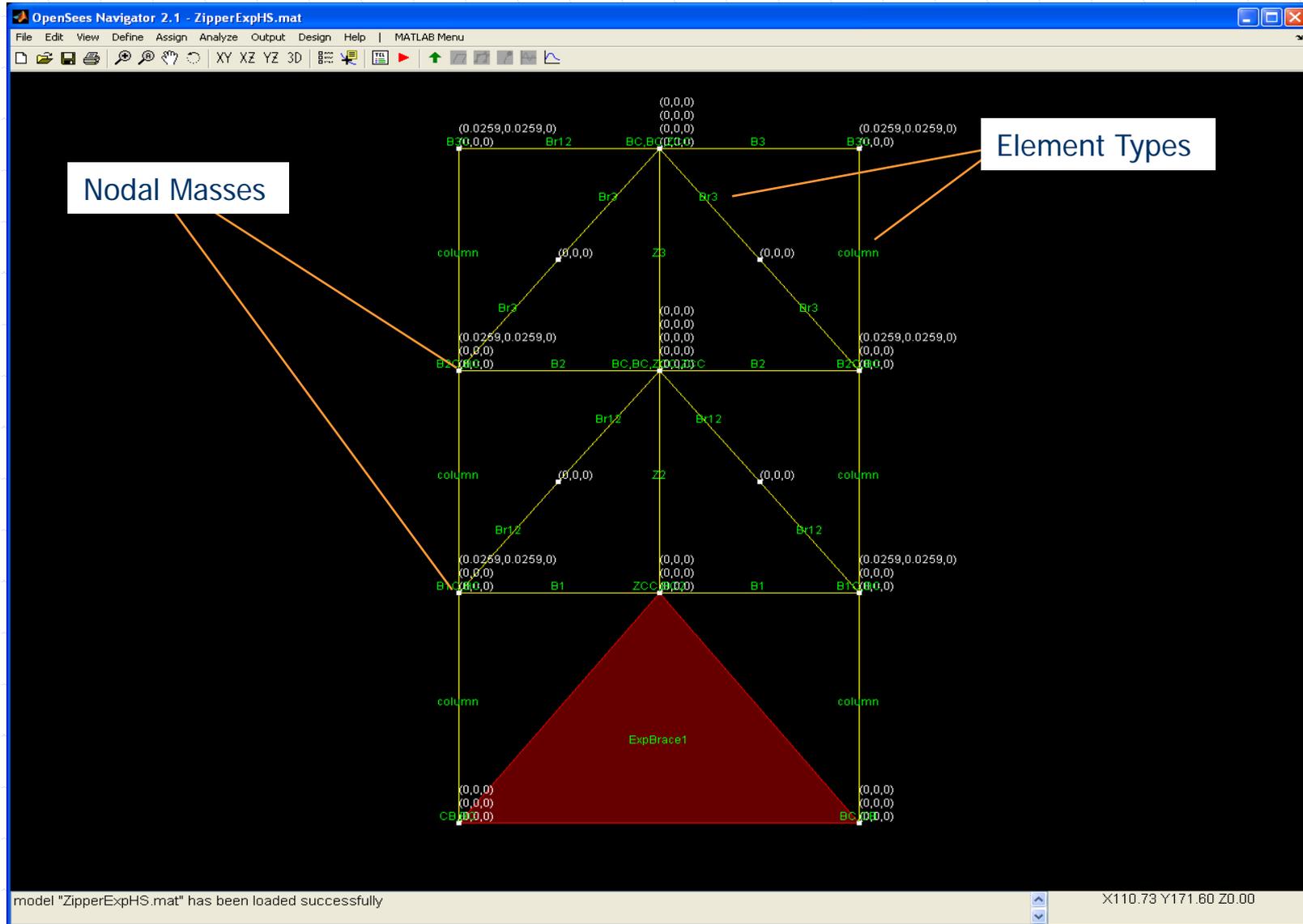
Assign Geometric Transformations :

Element Number(s) :

Geometric Transformation :



# Display assigned properties



# Define analysis case

The screenshot displays the OpenSees Navigator 2.1 interface. The main window shows a structural model of a truss with a red triangular element highlighted. A 'Define Analysis Case' dialog box is open, showing the following options:

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

The status bar at the bottom indicates: model "ZipperExpHS.mat" has been loaded successfully. The system information shows: X-46.36 Y171.80 Z0.00.

## Defaults:

- StaticDefaultCase
- EigenDefaultCase

# Define analysis case: new analysis case

**Define New Analysis Case**

Analysis Case Name :	EQ1	Add
Start from Previous Analysis Case :	WhiteNoise	
Load Pattern Name(s) :	EQ1 EQ2 EQ3	
Recorder Name(s) :	Elem01FibC Elem01FibCc None	
Analysis Options Name :	TransientDefault	
User Defined Analysis Script :	THAOption.txt	Browse
Damping Parameters :	Damping Parameters	
Geotechnical Parameters :	Geotechnical Parameters	
Number of Load Steps (numIncr) :	20480	
Time Step Increment (dt) :	0.00390625	

For Example:

Run the Time-History Analysis after the gravity load

# 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
    }
  }
}
```

# Define analysis case: new analysis case

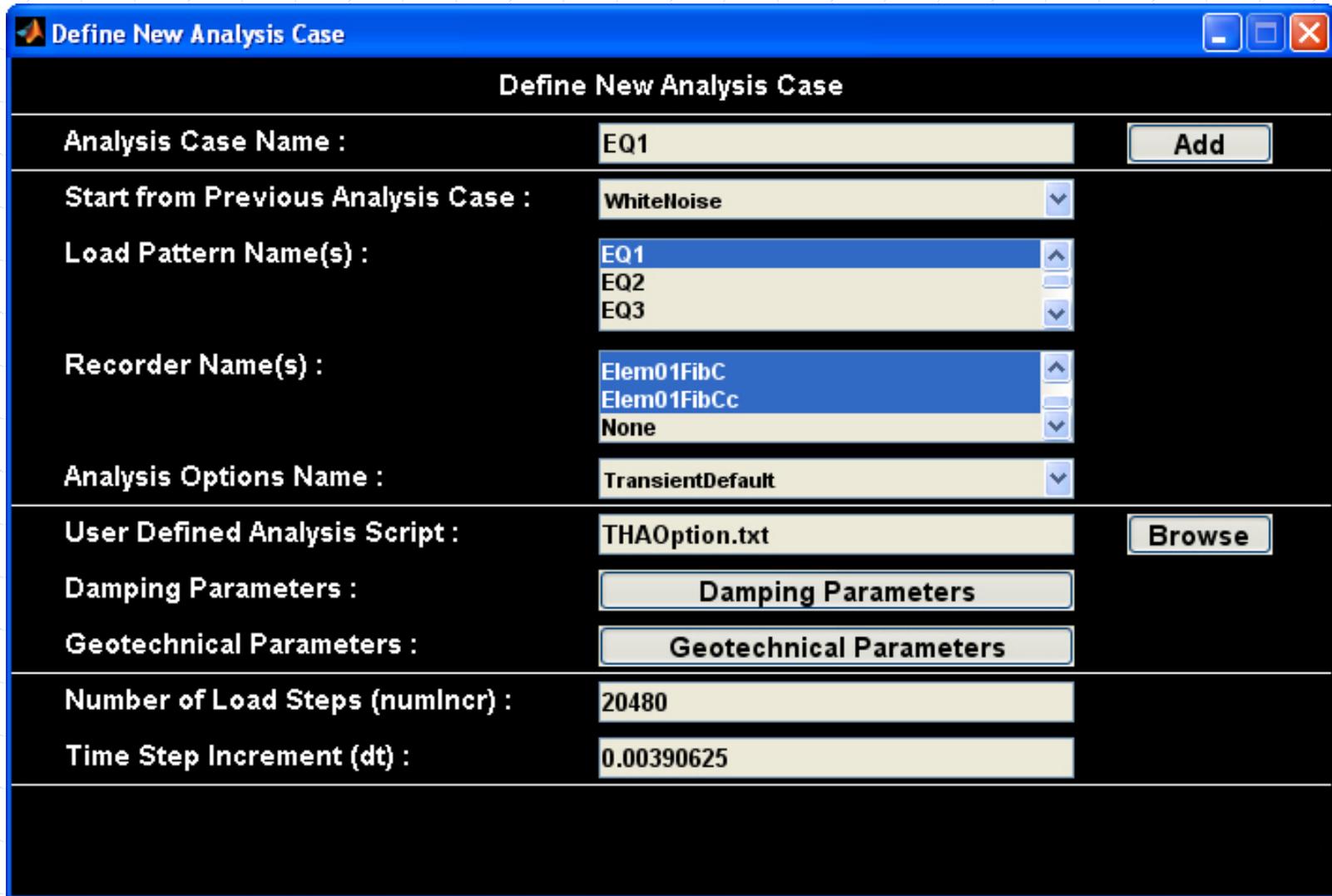
The image shows a software interface for defining a new analysis case. The main dialog box, titled "Define New Analysis Case", has a blue header bar. It contains the following fields and controls:

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

A sub-dialog box, titled "Define New Damping Parameters", is overlaid on top of the main dialog. It also has a blue header bar and contains the following fields and controls:

- Damping Parameter Set Name :** DampingParam01 (text input field)
- Add** (button)
- Region Defined by :** Nodes (dropdown menu)
- Node or Element Number(s) :** all (text input field)
- Mass Prop. Damping (alphaM) :** 0. (text input field)
- Kcurr Prop. Damping (betaK) :** 0. (text input field)
- Kinit Prop. Damping (betaKinit) :** 0. (text input field)
- Kcom Prop. Damping (betaKcom) :** 0. (text input field)
- Calculate** (button)

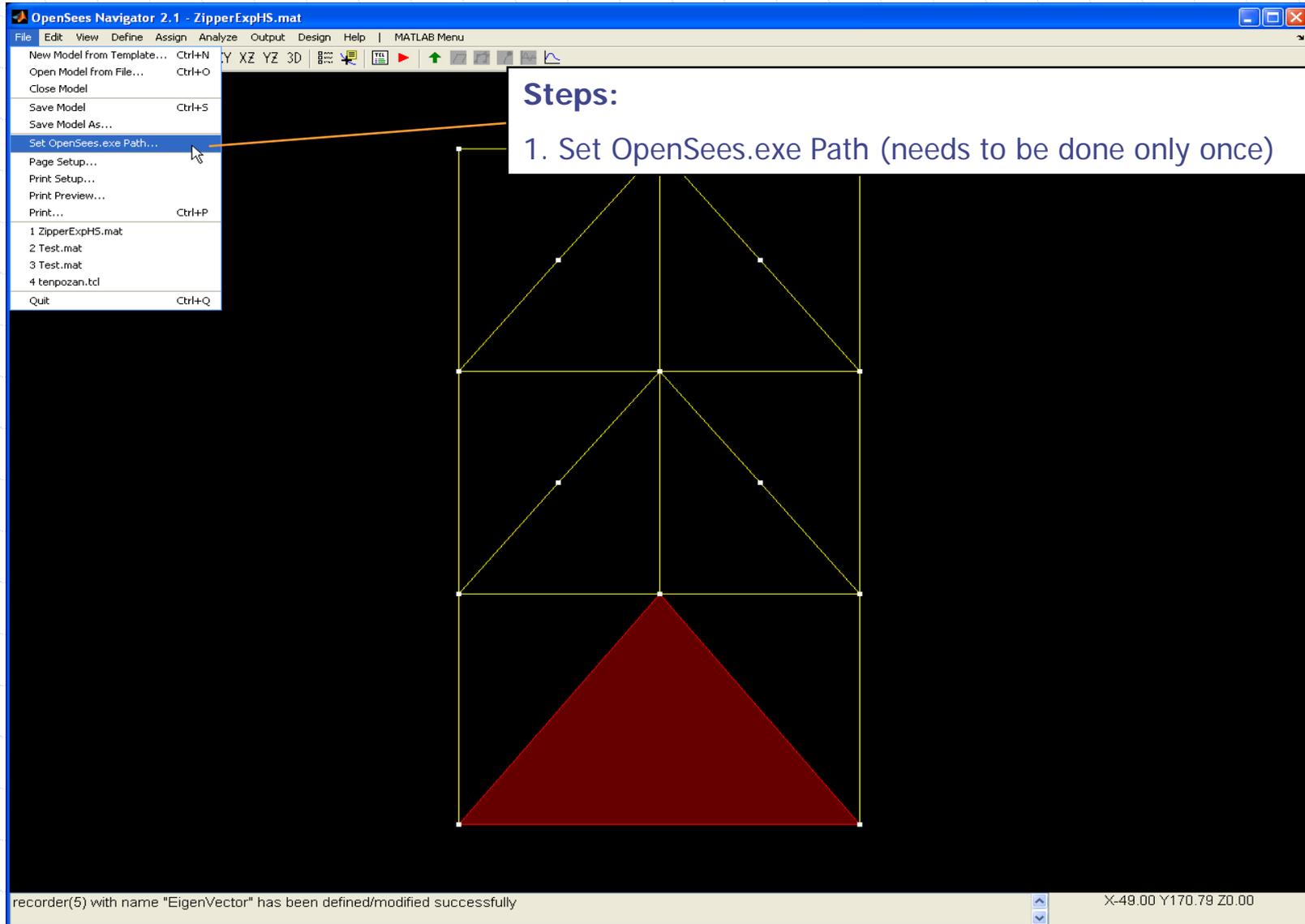
# Define analysis case: new analysis case



**Define New Analysis Case**

Analysis Case Name :	EQ1	Add
Start from Previous Analysis Case :	WhiteNoise	
Load Pattern Name(s) :	EQ1 EQ2 EQ3	
Recorder Name(s) :	Elem01FibC Elem01FibCc None	
Analysis Options Name :	TransientDefault	
User Defined Analysis Script :	THAOption.txt	Browse
Damping Parameters :	Damping Parameters	
Geotechnical Parameters :	Geotechnical Parameters	
Number of Load Steps (numIncr) :	20480	
Time Step Increment (dt) :	0.00390625	

# Run OpenSees: set OpenSees.exe path



The screenshot shows the OpenSees Navigator 2.1 interface. The menu bar includes File, Edit, View, Define, Assign, Analyze, Output, Design, Help, and MATLAB Menu. The File menu is open, and the option 'Set OpenSees.exe Path...' is highlighted. A callout box points to this option with the text 'Steps: 1. Set OpenSees.exe Path (needs to be done only once)'. The main workspace displays a 3D model of a structure with a red triangular base and yellow lines representing the rest of the model. The status bar at the bottom shows 'recorder(5) with name "EigenVector" has been defined/modified successfully' and coordinates 'X-49.00 Y170.79 Z0.00'.

**Steps:**

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

# Run OpenSees: write TCL files

The image shows a screenshot of 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 dialog box titled "Steps:" is overlaid on the main window, listing the following steps:

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

Below the "Steps:" dialog box, a smaller dialog box titled "Running OpenSees" is shown. It contains a text area with the message "running OpenSees analysis, please wait ....." and an "OK" button at the bottom. A red trapezoidal shape is positioned below the "Running OpenSees" dialog box. At the bottom of the main window, a status bar displays the text "recorder(5) with name 'EigenVector' has been defined/modified successfully" on the left and "X-45.55 Y171.60 Z0.00" on the right.

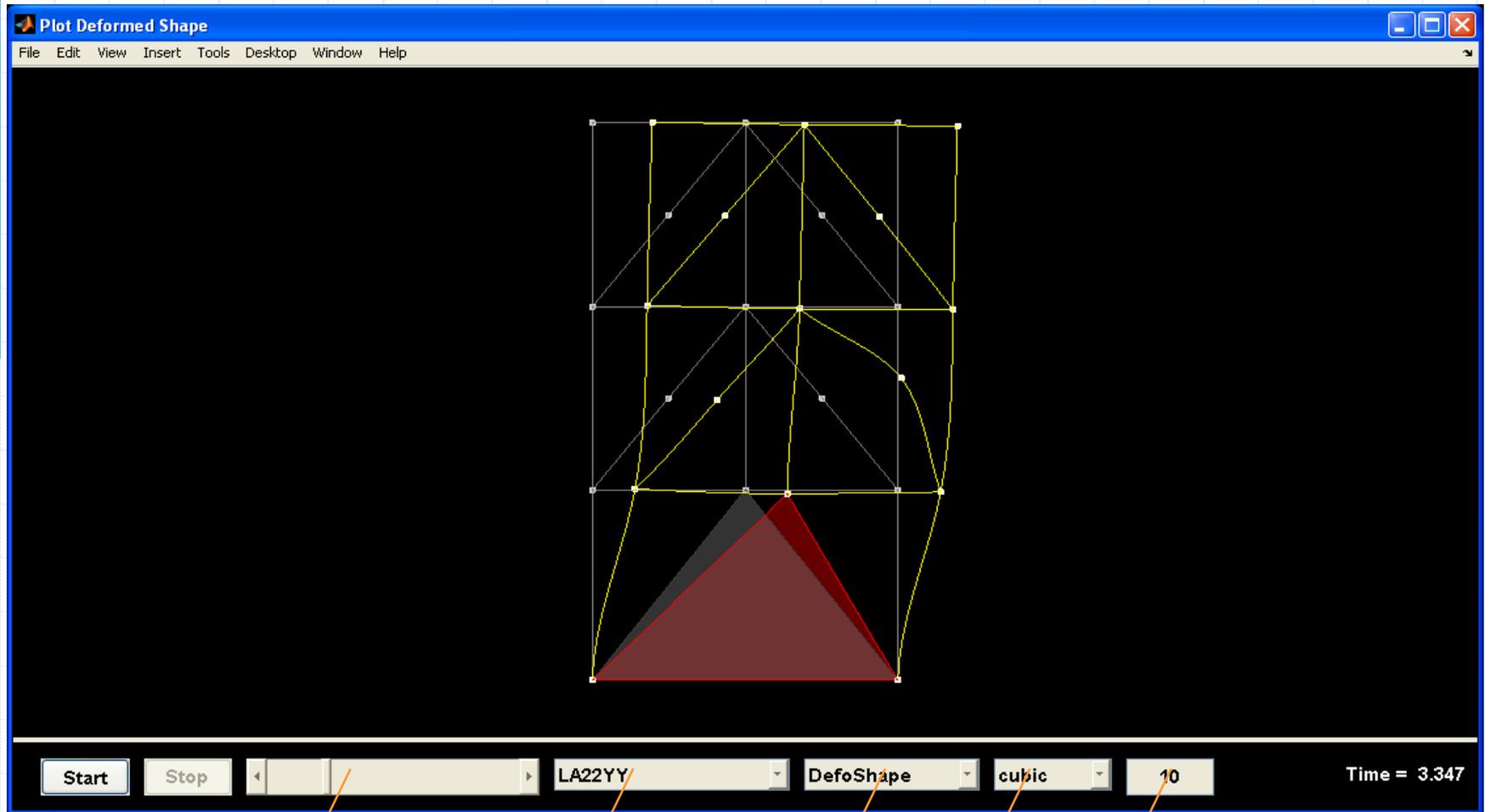
# Post processing: load results

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 callout box with the text "First: Load OpenSees Results" points to the "Load" button in the "Load OpenSees Results" dialog box. The dialog box has a title bar "Load OpenSees Results" and a "Load" button. The "Select Analysis Case(s) :" label is followed by a list box containing "EigenDefaultCase" and "SACNF01Case01". The background view shows a dark workspace with a red triangle and a yellow wireframe structure. The status bar at the bottom left reads "recorder(5) with name 'EigenVector' has been defined/modified successfully" and the bottom right shows coordinates "X-45.55 Y171.60 Z0.00".

# Post processing: output

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 "Output" menu is open, showing options: "Load OpenSees Results...", "Deformed Shape...", "Element Forces...", "Mode Shape..." (highlighted with a mouse cursor), "Response...", "Response Spectra...", and "Create Movie". A white callout box with the text "Now these are enabled" points to the "Mode Shape..." option. The main workspace shows a structural model with a red triangular base and yellow lines representing the rest of the structure. The status bar at the bottom left says "OpenSees result files loaded successfully" and the bottom right shows coordinates "X-12.67 Y171.40 Z0.00".

# Post processing: plot deformed shape



Time Step

AnalysisCase

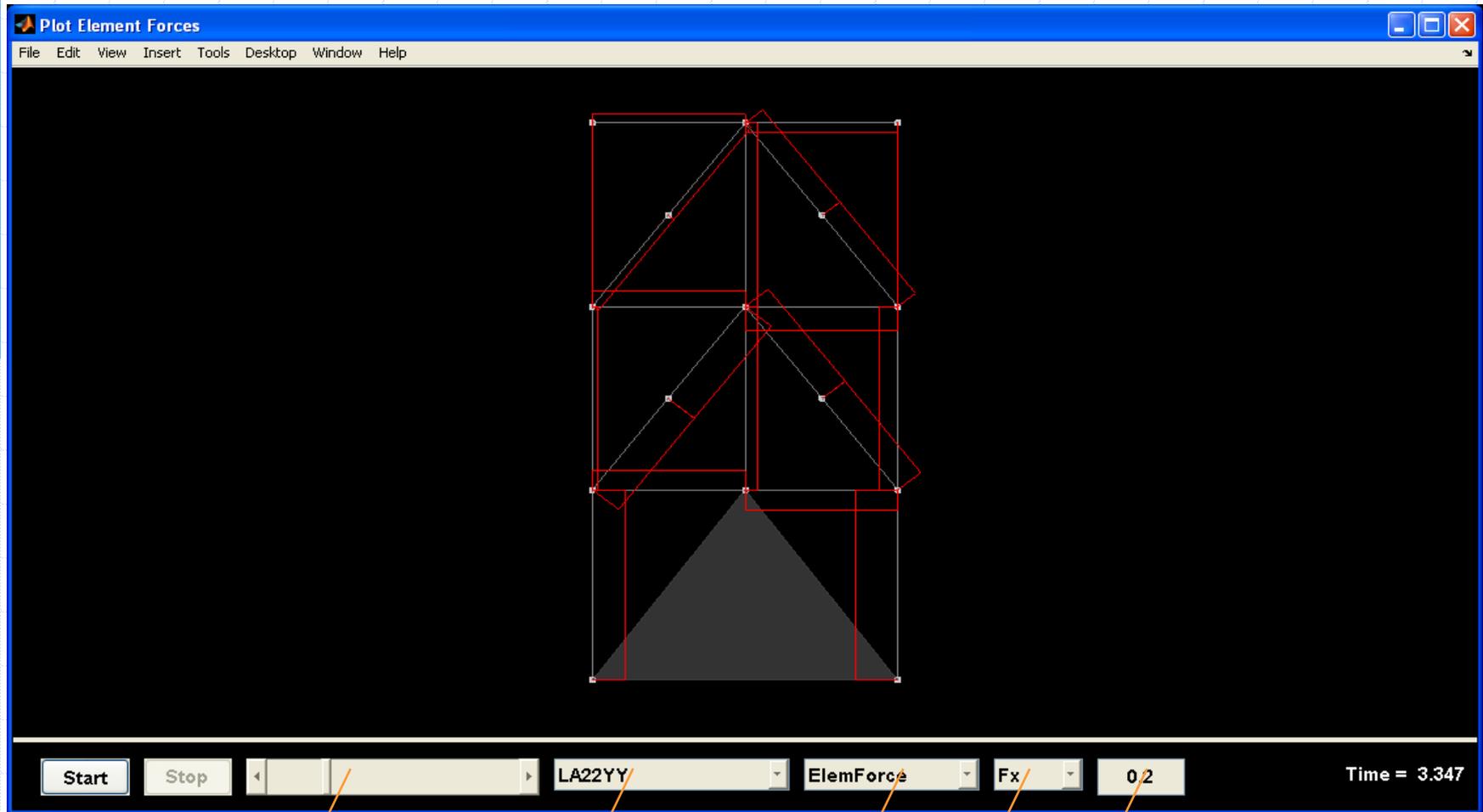
Recorder

Order

Magnification



# Post processing: plot element forces



Time Step

AnalysisCase

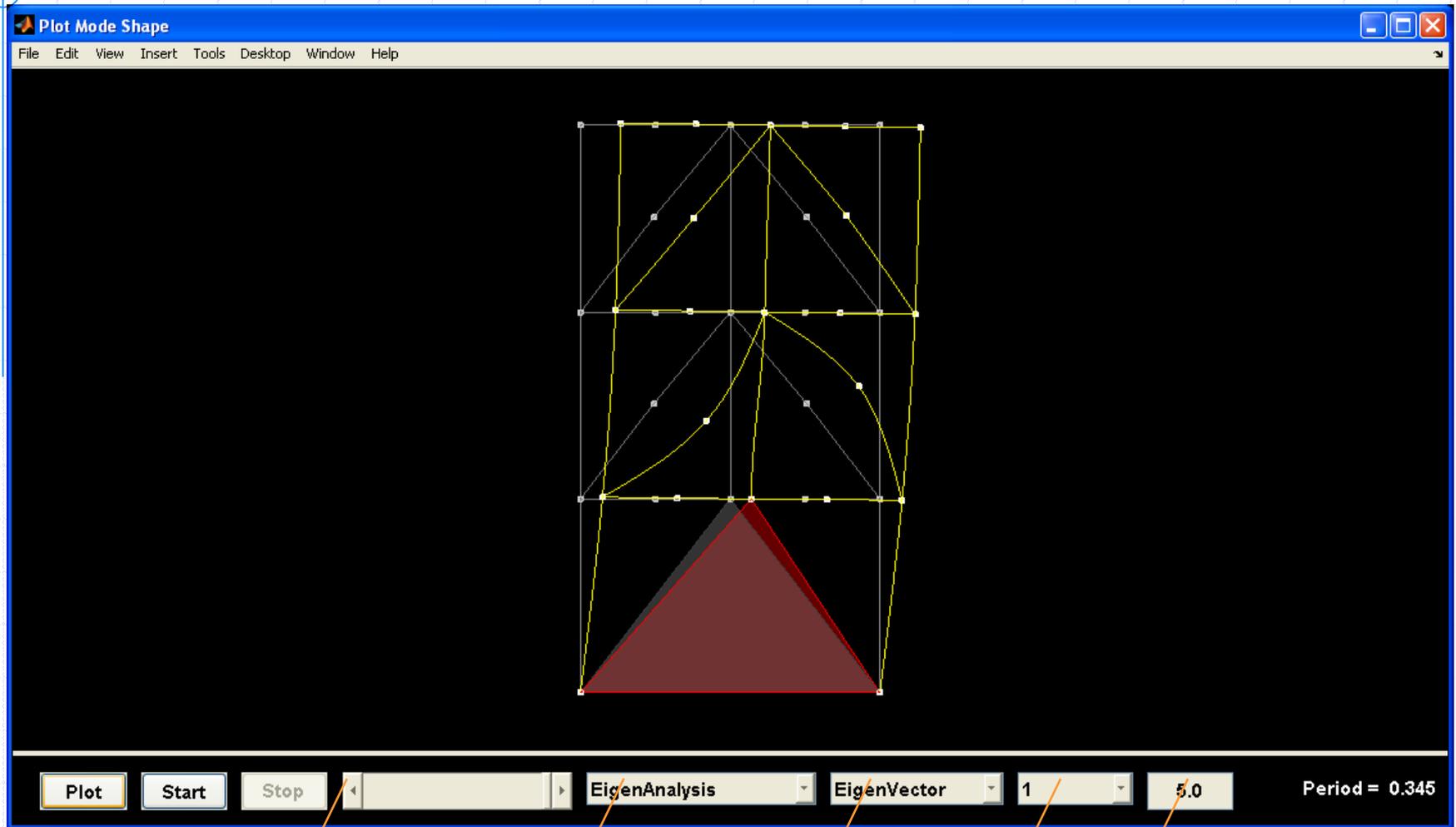
Recorder

Response

Magnification



# Post processing: plot mode shape



Animation Speed

AnalysisCase

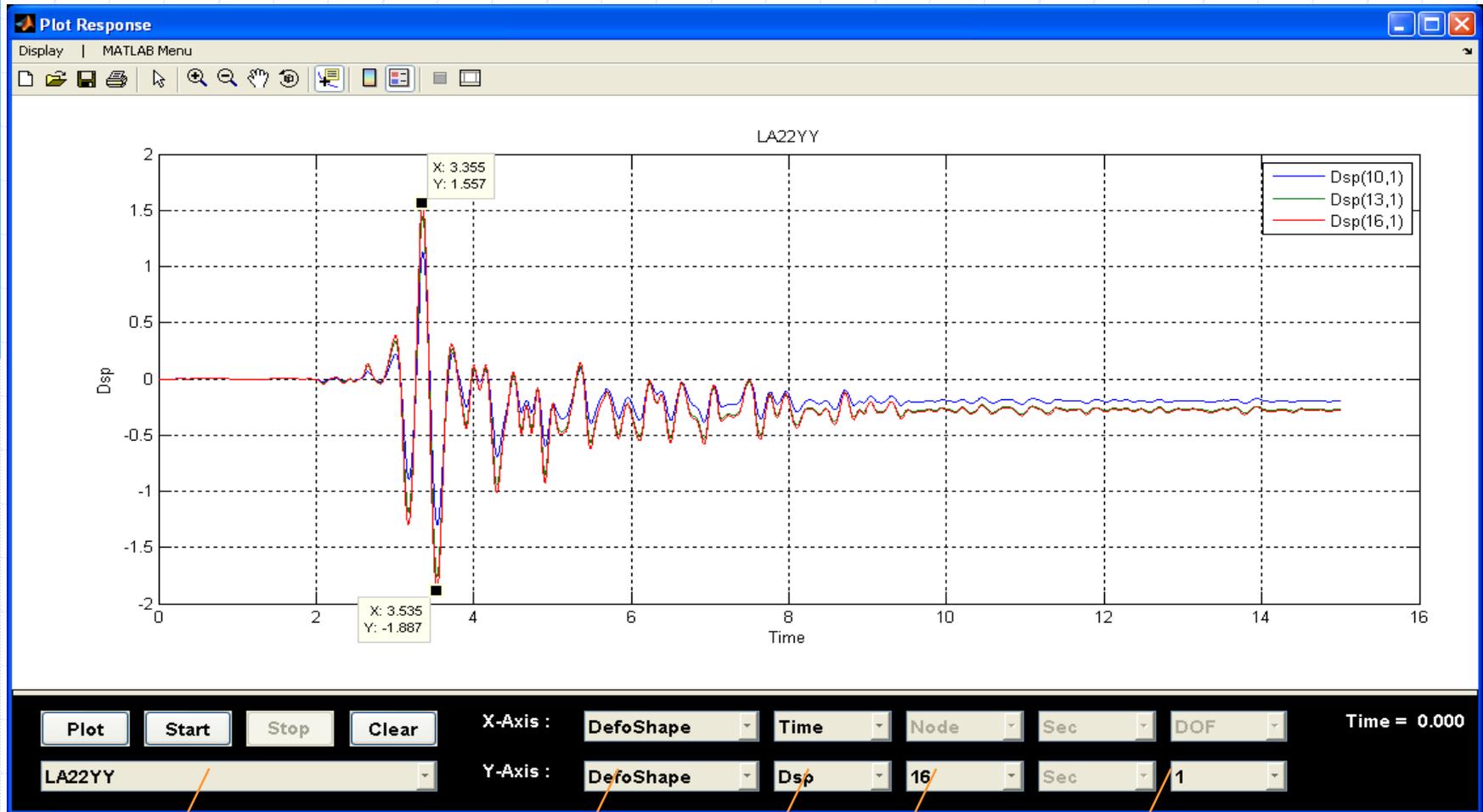
Recorder

Mode

Magnification



# Post processing: plot response histories



AnalysisCase

Recorder

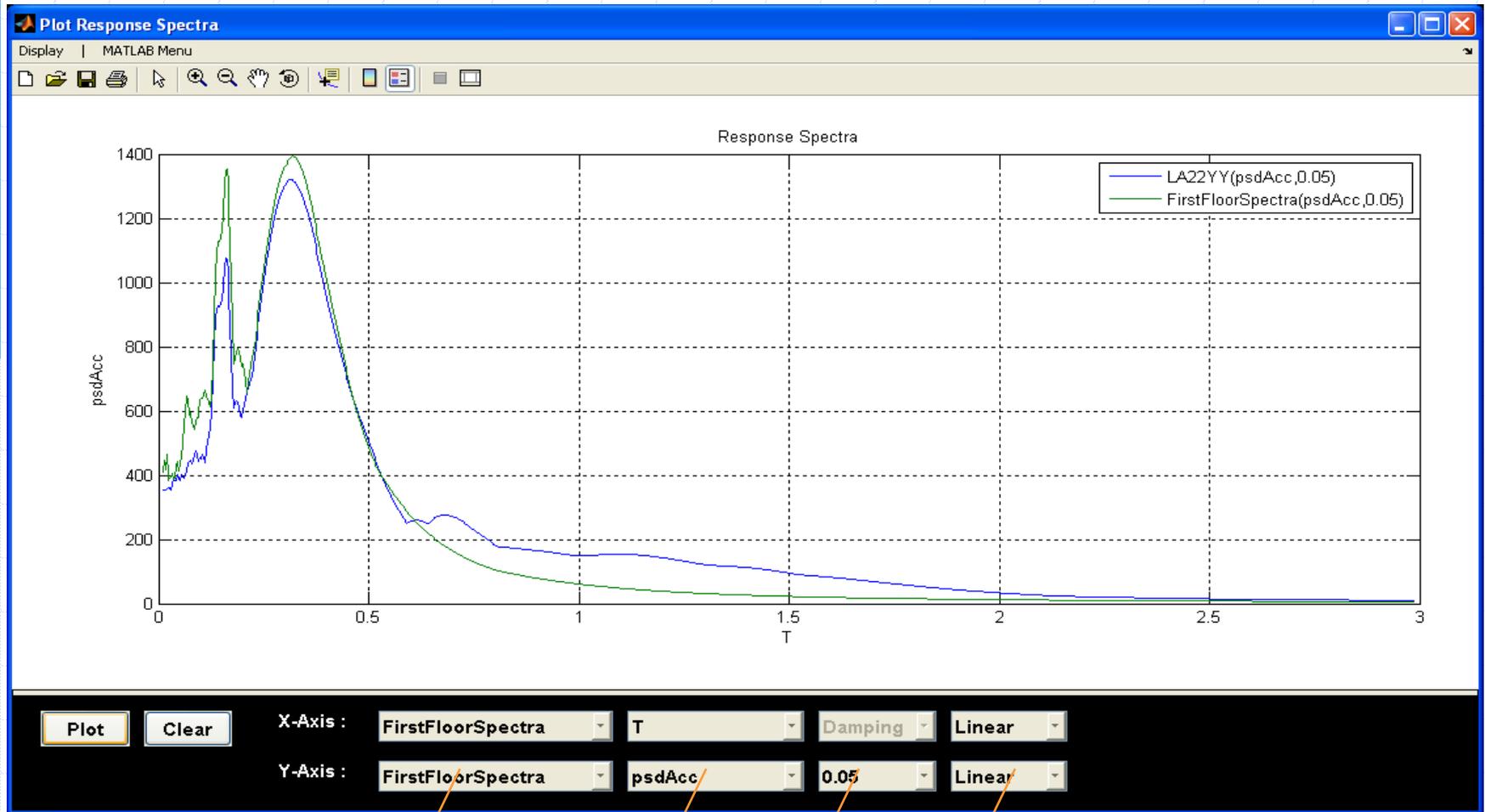
Parameter

Node/Element

DOF



# Post processing: plot response spectra



Spectra Name

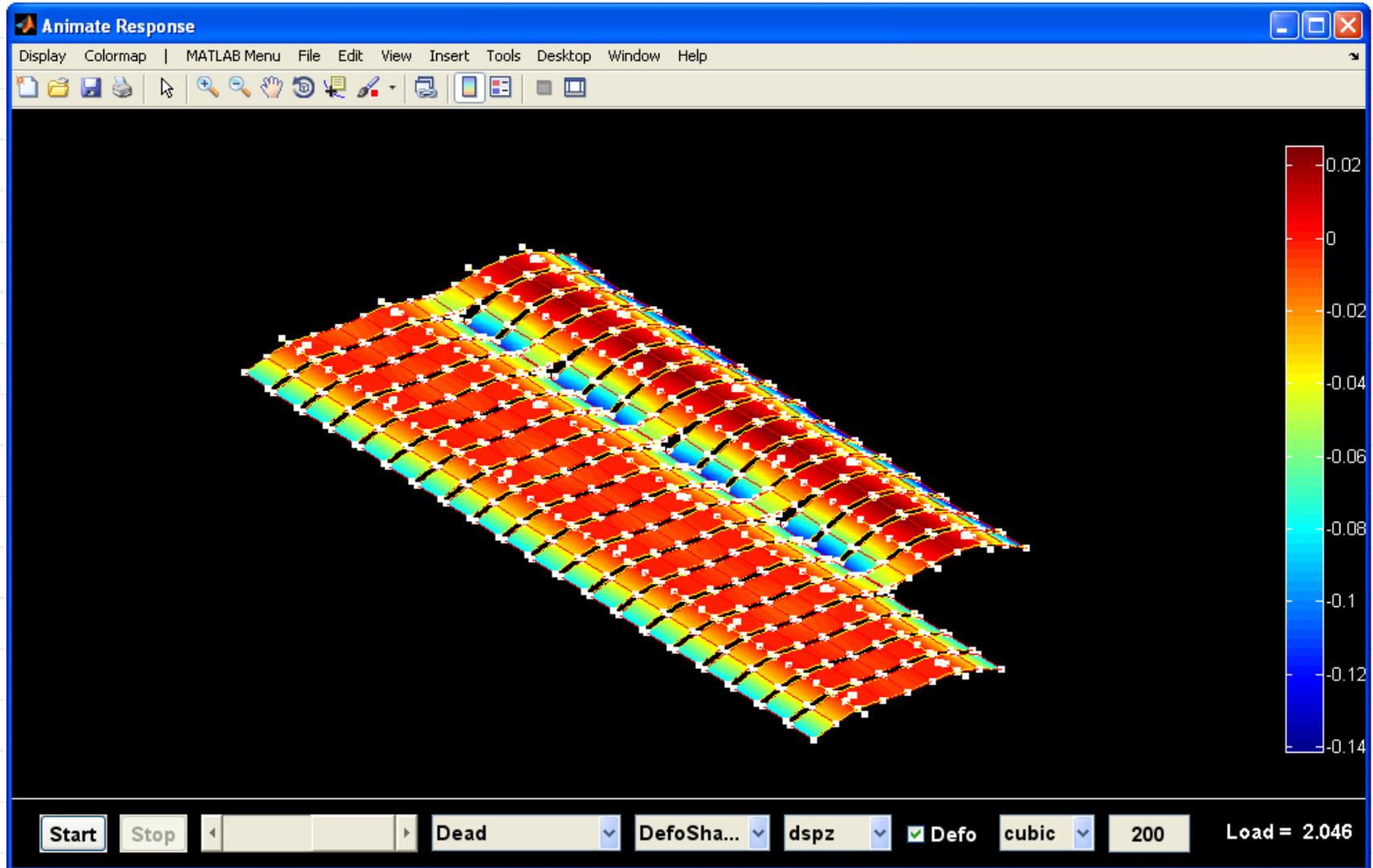
Response

Damping

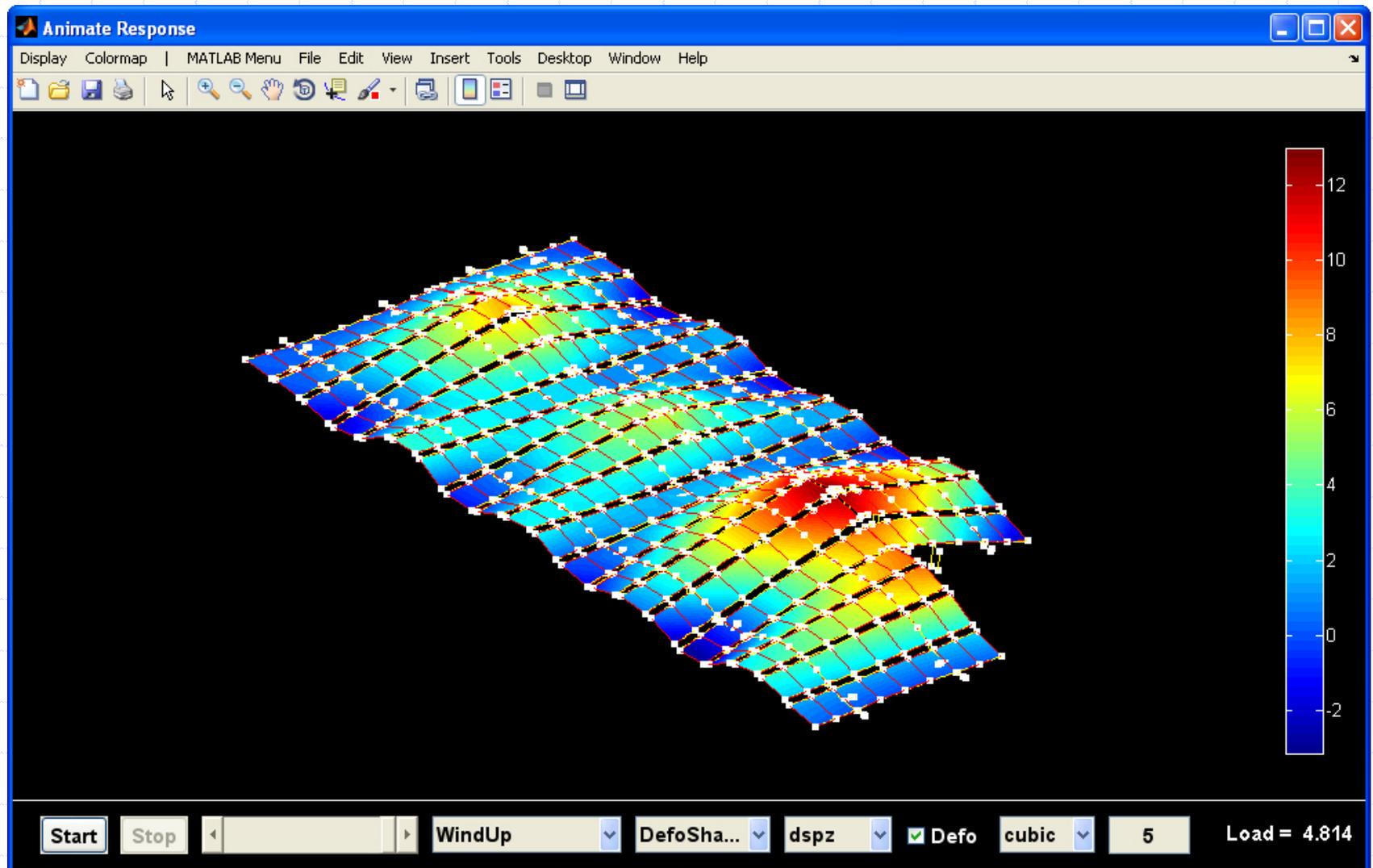
Axis Scale



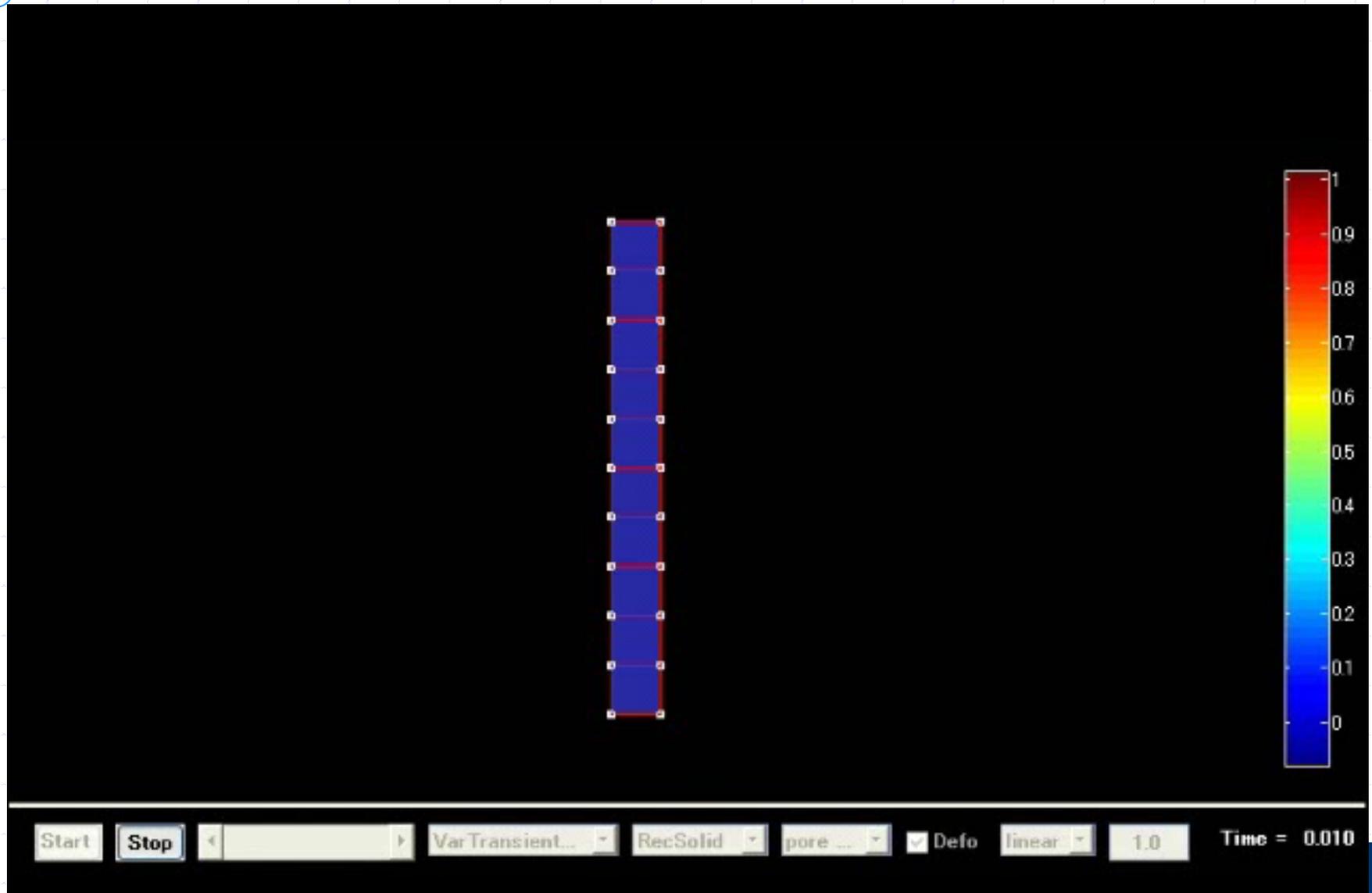
# Post processing: animate response



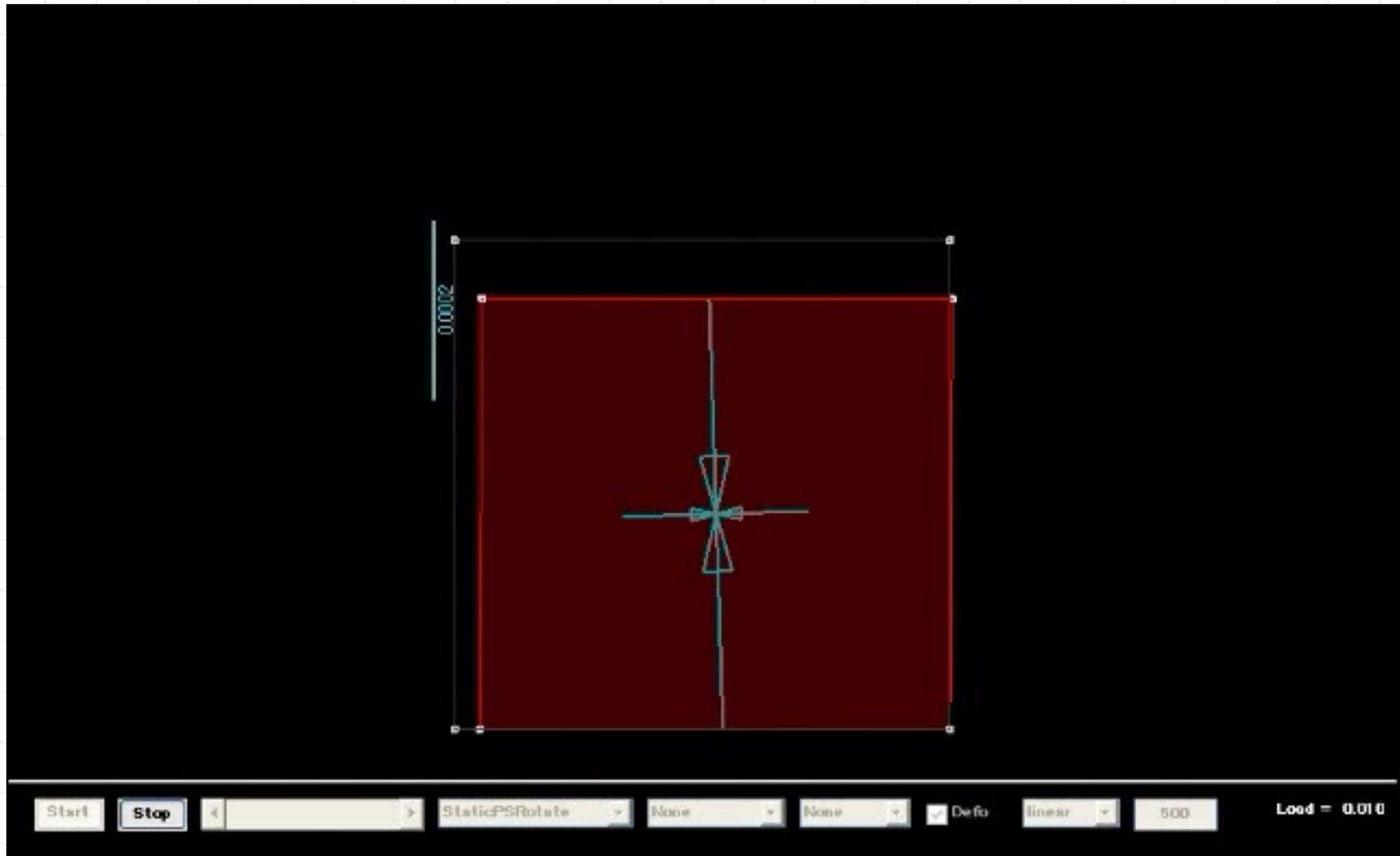
# Post processing: animate response



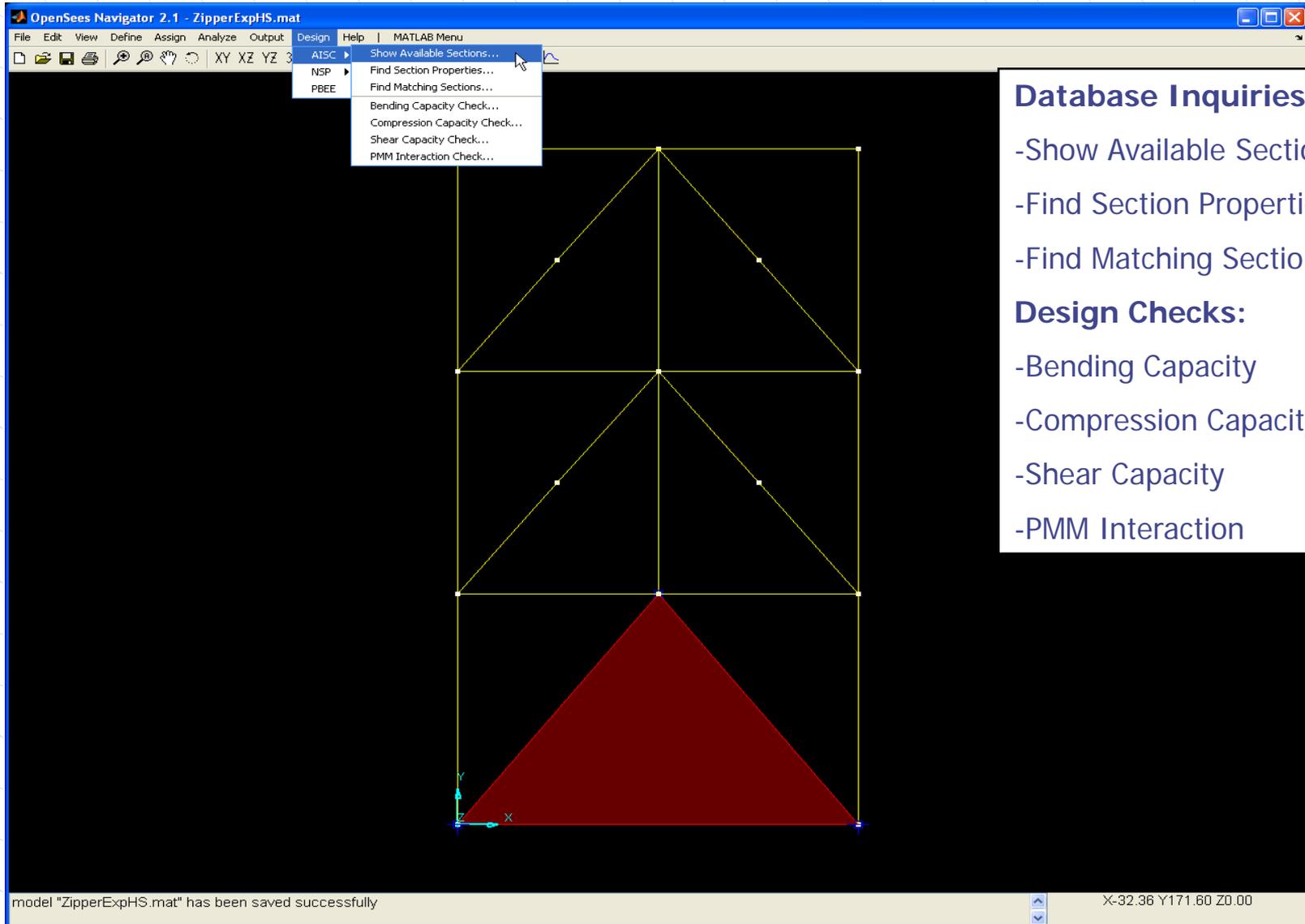
# Post processing: animate response



# Post processing: principal $\sigma$ and $\varepsilon$



# 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 and a yellow wireframe structure above it. 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	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 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	<input type="button" value="Calculate"/>
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.



# OpenSees Navigator – Hybrid simulation

The screenshot displays the OpenSees Navigator 2.1 software interface, which is used for hybrid simulation. The interface includes several key components:

- Station Manager:** Shows the current station setup, including channels, actuators, and sensors.
- Function Generator:** Configures the displacement signal, including target setpoint, amplitude, frequency, and phase.
- Actuator 1 Displacement:** Displays real-time data for the actuator, such as displacement (-0.2221 in) and force (1.242 kip).
- Manual Controls:** Provides manual control options for the actuator, including active mode and manual command.
- Limit Detectors:** Configures upper and lower limits for displacement and force, with associated actions like interlock.
- Plot Window:** Shows a graph of Actuator 1 Command (in) over time (Sec), with a trace time of 20.000 seconds.
- OpenSees Computer Simulation Configurator:** Configures the simulation parameters, including control points and degrees of freedom.

On the right side of the interface, there is a photograph of the physical test rig, which is a large, complex mechanical structure used for testing. The rig is labeled with various components and sensors, including:

- ExpBrace1:** A red triangular brace structure.
- CBBC:** A component at the bottom left of the brace.
- ESCP:** A component at the bottom right of the brace.
- B3:** A sensor or actuator at the top right.
- B2:** A sensor or actuator in the middle right.
- B12:** A sensor or actuator in the middle left.
- B1:** A sensor or actuator at the bottom left.
- COLU:** A vertical column supporting the structure.

At the bottom of the interface, a status bar indicates "OpenSees result files loaded successfully" and shows the coordinates "X-11.45 Y170.18 Z0.00".

# Summary and conclusions

## ◆ OpenSees

- A powerful software framework which is capable for conducting sequential and parallel, grid-enabled finite element applications.

## ◆ Hybrid simulation interface (OpenFresco)

## ◆ OpenSees Navigator

- Flexible and user friendly graphical user interface.
- Great tool to visualize structural behavior.
- Easy way to study material, section, element or system behavior.
- Many design toolboxes: NSP, PBEE, AISC design checks, AISC database, response spectra for linear and bilinear systems and signal filtering.

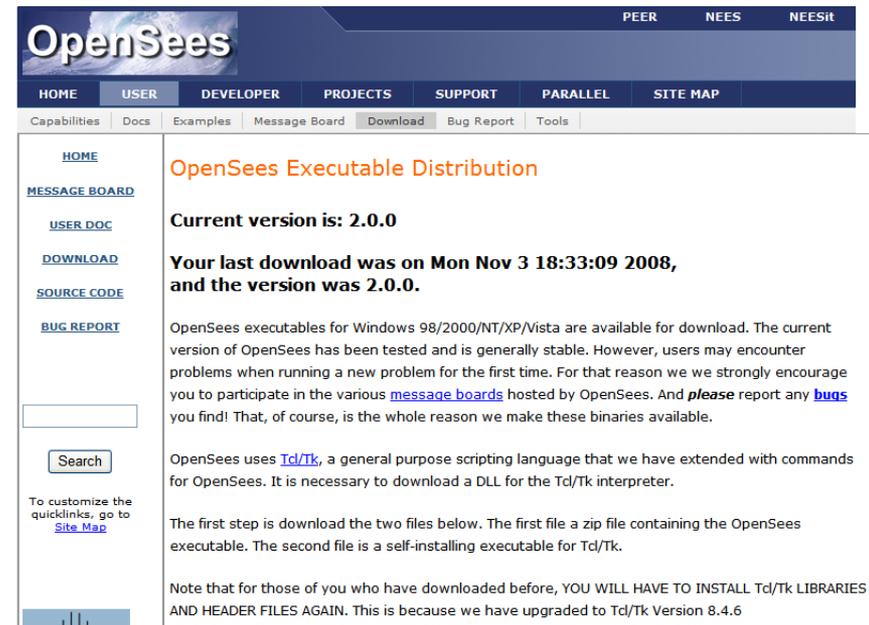
## ◆ Both MATLAB Pcode and self-executable versions are available



# Websites:



The screenshot shows the OpenSees Navigator website. The header includes the title "OPENSEES NAVIGATOR" and a navigation menu with links for HOME, INTRODUCTION, MANUALS, TUTORIALS, PRESENTATIONS, DISCUSSION, UPDATES, DOWNLOADS, and LINKS. A search bar is located below the menu. The main content area features a left sidebar with a list of navigation links (Introduction, Manuals, Tutorials, Presentations, Discussion, Updates, Downloads, Links) and a main text area. The text area contains a welcome message: "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](mailto:andreas.schellenberg@gmail.com) or [yangtony2004@gmail.com](mailto: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." Below this, it says "Thank you. Please feel free to visit our websites to discover in what other fun research we are involved." and lists "Andreas Schellenberg & Tony Yang". At the bottom, there is a "Hit Counter" and a copyright notice: "OpenSees Navigator ©2004-2006 UC Berkeley. All rights reserved. Please [contact us](#) with any questions or comments."



The screenshot shows the OpenSees website. The header includes the title "OpenSees" and a navigation menu with links for HOME, USER, DEVELOPER, PROJECTS, SUPPORT, PARALLEL, and SITE MAP. A secondary menu below includes links for Capabilities, Docs, Examples, Message Board, Download, Bug Report, and Tools. The main content area features a left sidebar with a list of navigation links (HOME, MESSAGE BOARD, USER DOC, DOWNLOAD, SOURCE CODE, BUG REPORT) and a main text area. The text area contains the following information: "OpenSees Executable Distribution", "Current version is: 2.0.0", "Your last download was on Mon Nov 3 18:33:09 2008, and the version was 2.0.0.", "OpenSees executables for Windows 98/2000/NT/XP/Vista are available for download. The current version of OpenSees has been tested and is generally stable. However, users may encounter problems when running a new problem for the first time. For that reason we we strongly encourage you to participate in the various [message boards](#) hosted by OpenSees. And *please* report any [bugs](#) you find! That, of course, is the whole reason we make these binaries available.", "OpenSees uses [Td/Tk](#), a general purpose scripting language that we have extended with commands for OpenSees. It is necessary to download a DLL for the Td/Tk interpreter.", "The first step is download the two files below. The first file a zip file containing the OpenSees executable. The second file is a self-installing executable for Td/Tk.", "Note that for those of you who have downloaded before, YOU WILL HAVE TO INSTALL Td/Tk LIBRARIES AND HEADER FILES AGAIN. This is because we have upgraded to Td/Tk Version 8.4.6". At the bottom, there is a "Hit Counter" and a copyright notice: "OpenSees Navigator ©2004-2006 UC Berkeley. All rights reserved. Please [contact us](#) with any questions or comments."

Download information:

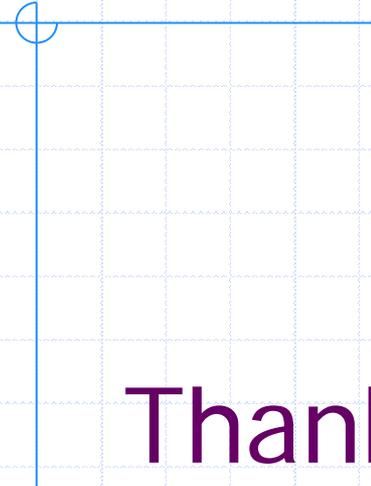
OpenSees 2.2 is available at

<http://opensees.berkeley.edu>

OpenSees Navigator 2.5 is available at

<http://peer.berkeley.edu/OpenSeesNavigator>





# Thank you for your attention!

Contact information:

Tony Yang, [yang@civil.ubc.ca](mailto:yang@civil.ubc.ca),  
<http://peer.berkeley.edu/~yang>

Andreas Schellenberg, [andreas.schellenberg@gmail.com](mailto:andreas.schellenberg@gmail.com)

Eiji Kohama, [ekohama@gmail.com](mailto:ekohama@gmail.com)

