

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

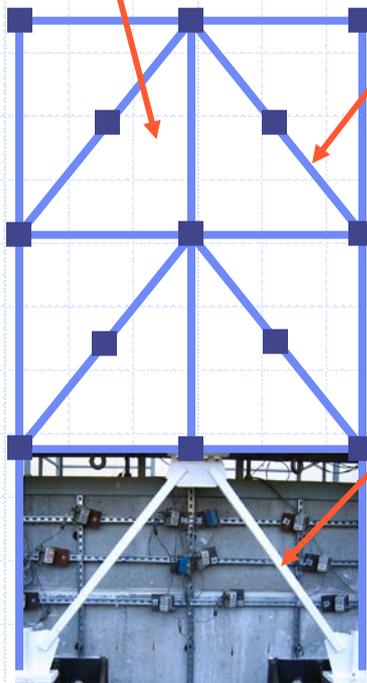


Hybrid Simulation

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

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

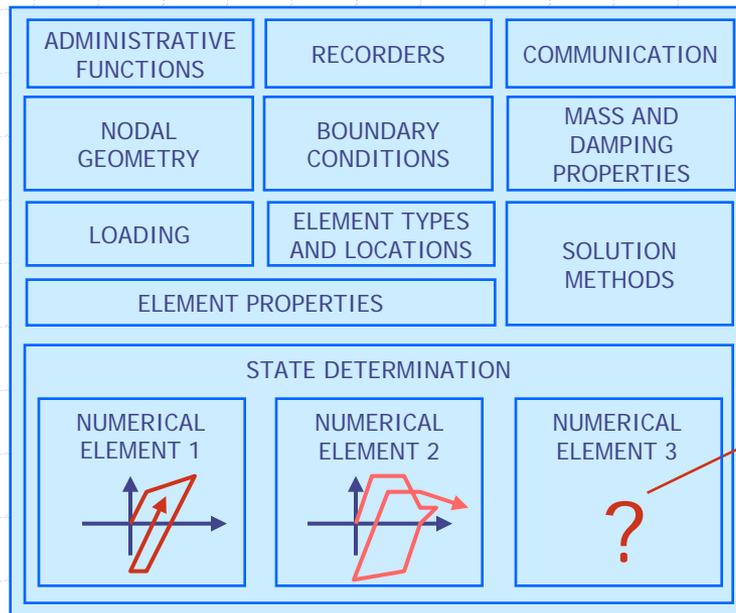
- Abacus
- Ansys (soon)
- JumpStart (soon)
- LS-Dyna
- Matlab
- OpenSees
- SimCor
- Simulink

◆ Control Systems

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

Implementation strategy

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

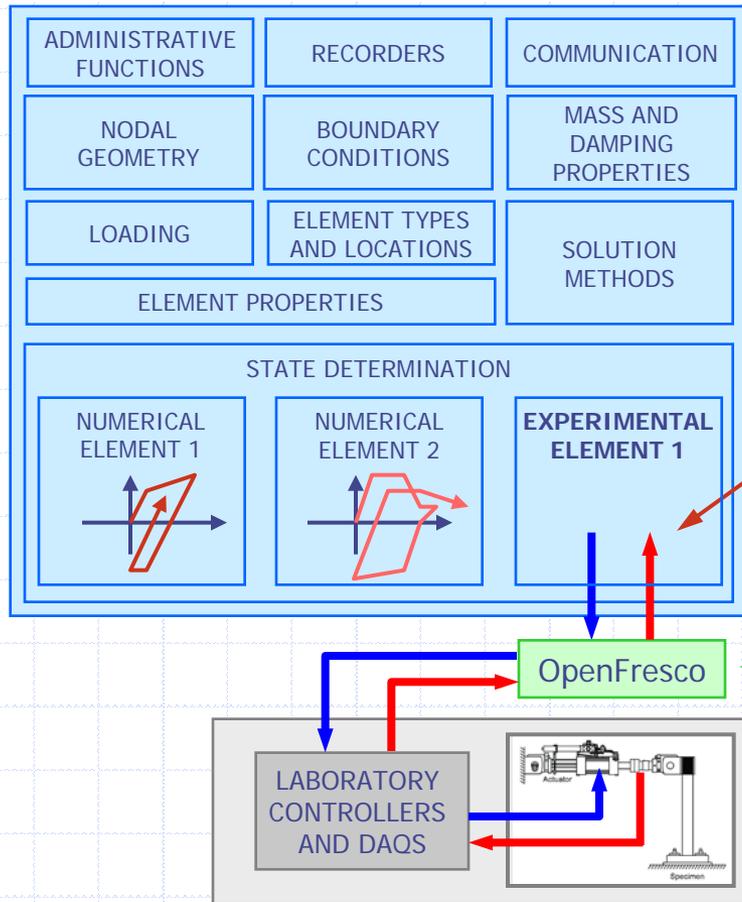


Typical features of an analysis framework

Proper numerical model uncertain

Implementation strategy

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



Typical features of an analysis framework

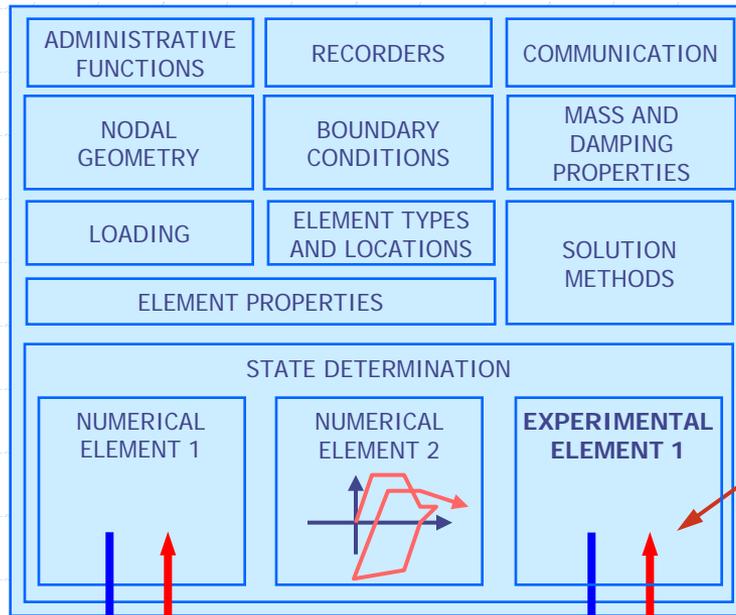
Define element as an "Experimental Element"

OpenFresco

Laboratory

Implementation strategy

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

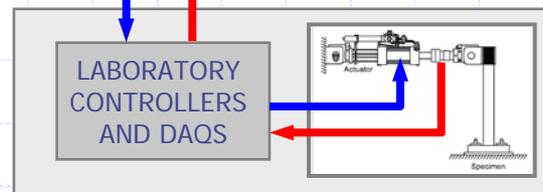
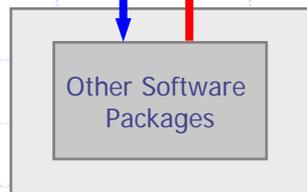


Typical features of an analysis framework

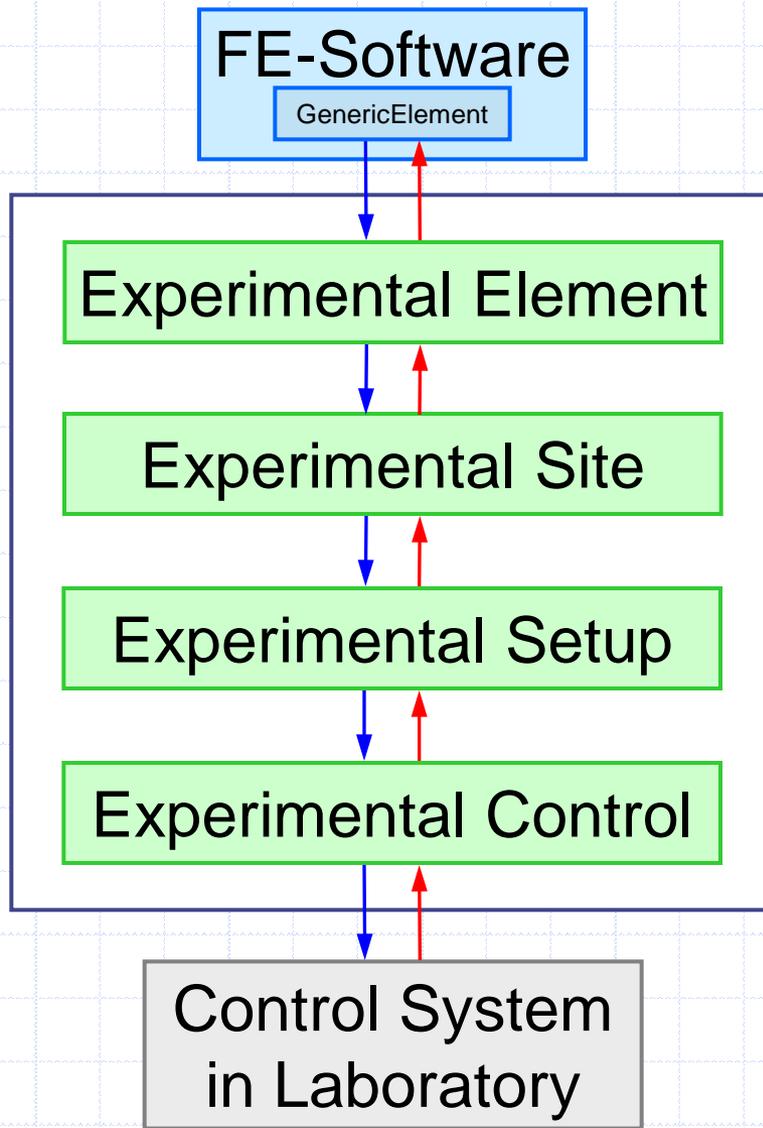
Define element as an "Experimental Element"

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

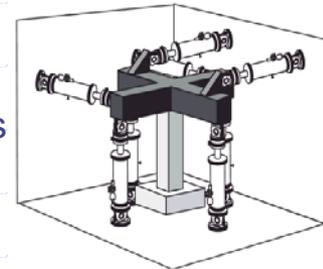
stores data and provides communication methods for distributed testing



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 and Tony Yang
Department of Civil and Environmental Engineering
University of California, Berkeley

Eiji Kohama
Port and Airport Institute, Japan
Earthquake Engineering Research Center (EERC)
University of California, Berkeley



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

- ◆ Graphical input is more flexible than TCL text 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.
- ◆ 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 | 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:

- Stick Model**: A vertical line with a horizontal line at the top.
- Beam Model**: A horizontal line with a vertical line at the left end.
- EBF Model**: A rectangular frame with diagonal bracing.
- Zipper Frame**: A rectangular frame with diagonal bracing in a zipper pattern.
- Inverted-V Braced Frame**: A rectangular frame with diagonal bracing in an inverted-V pattern.
- Moment Frame**: A rectangular frame with diagonal bracing in a moment-resisting pattern.
- Single Area Mesh**: A 3x3 grid of squares.

The software interface includes a menu bar with options: File, Edit, View, Define, Assign, Analyze, Output, Design, Help, and MATLAB Menu. 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".

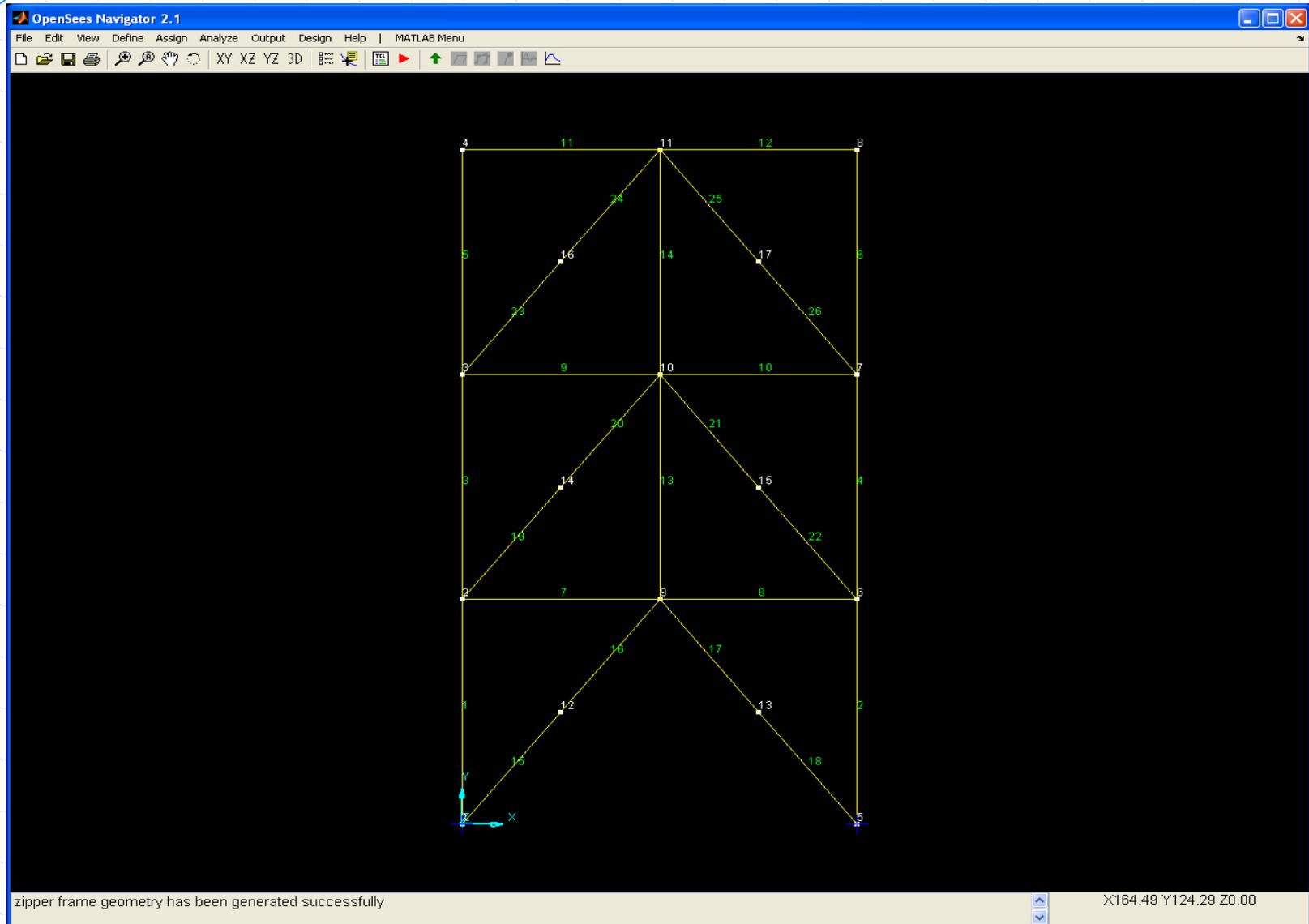
At the bottom of the window, a status bar displays the message: "Model has been closed and workspace has been reinitialized successfully" and coordinates: "X0.00 Y0.00 Z0.00".

Define geometry: Zipper braced frame

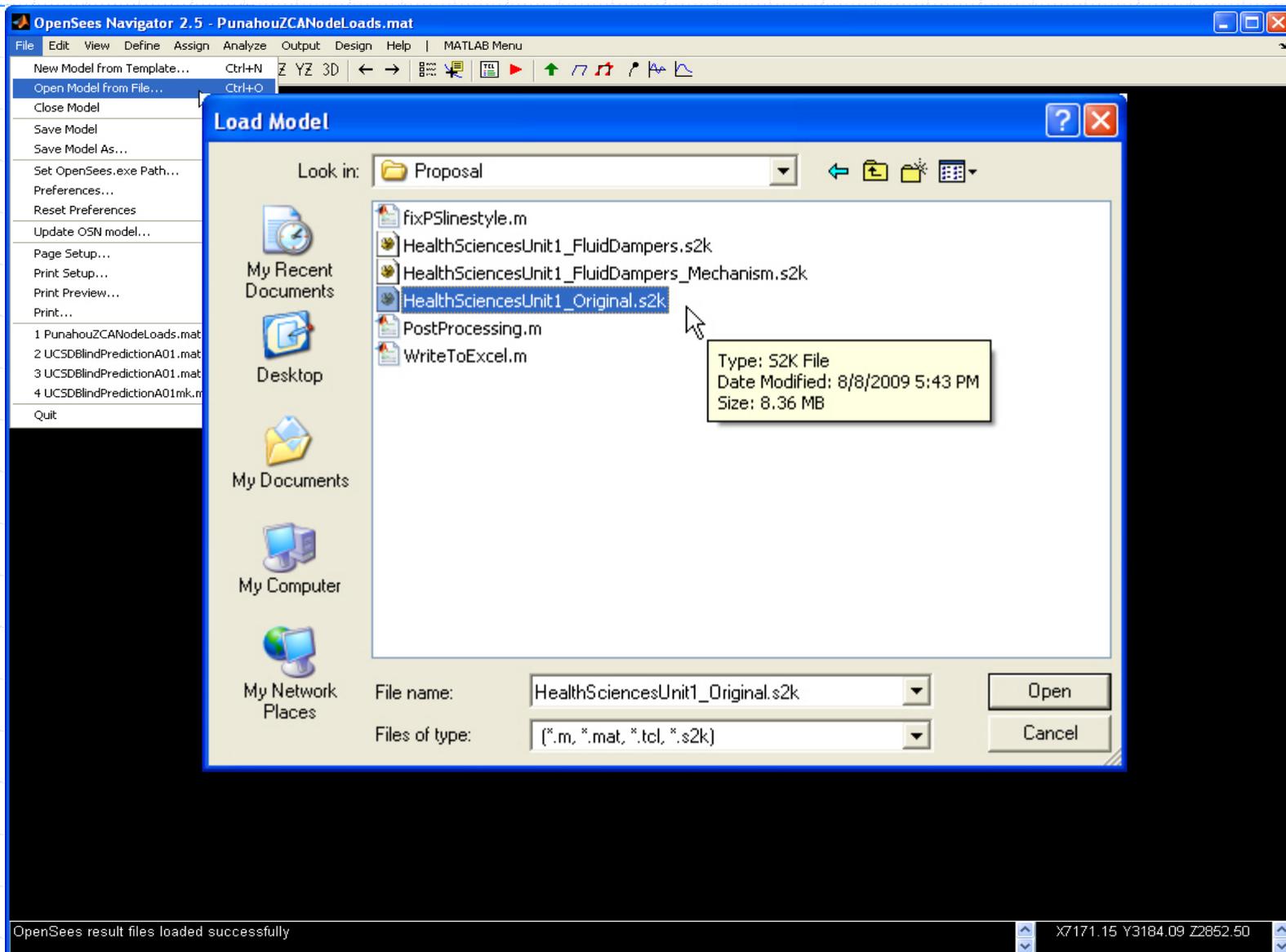
Define Zipper Frame Geometry

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) :	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	

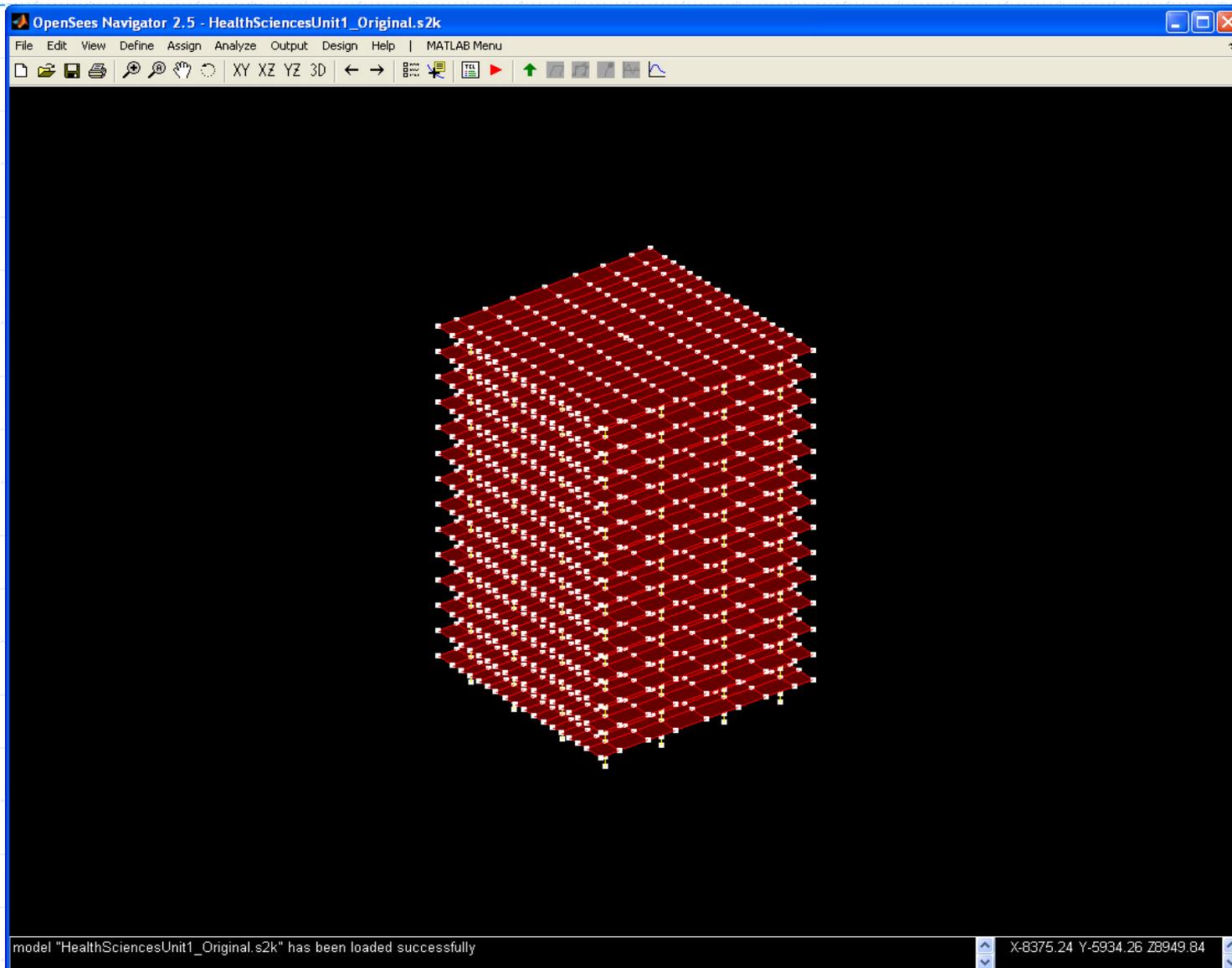
View geometry: display



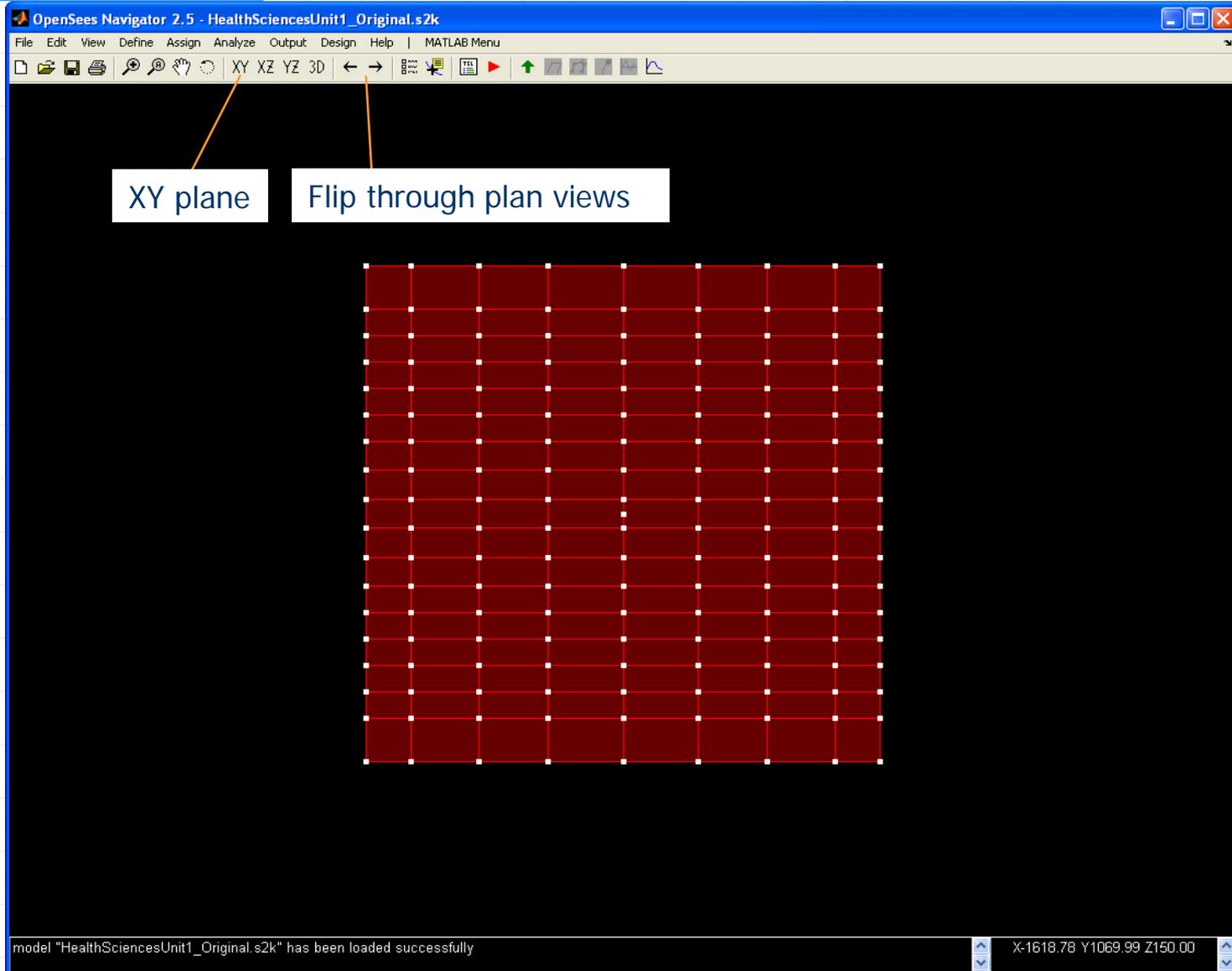
Define geometry: import from SAP2000



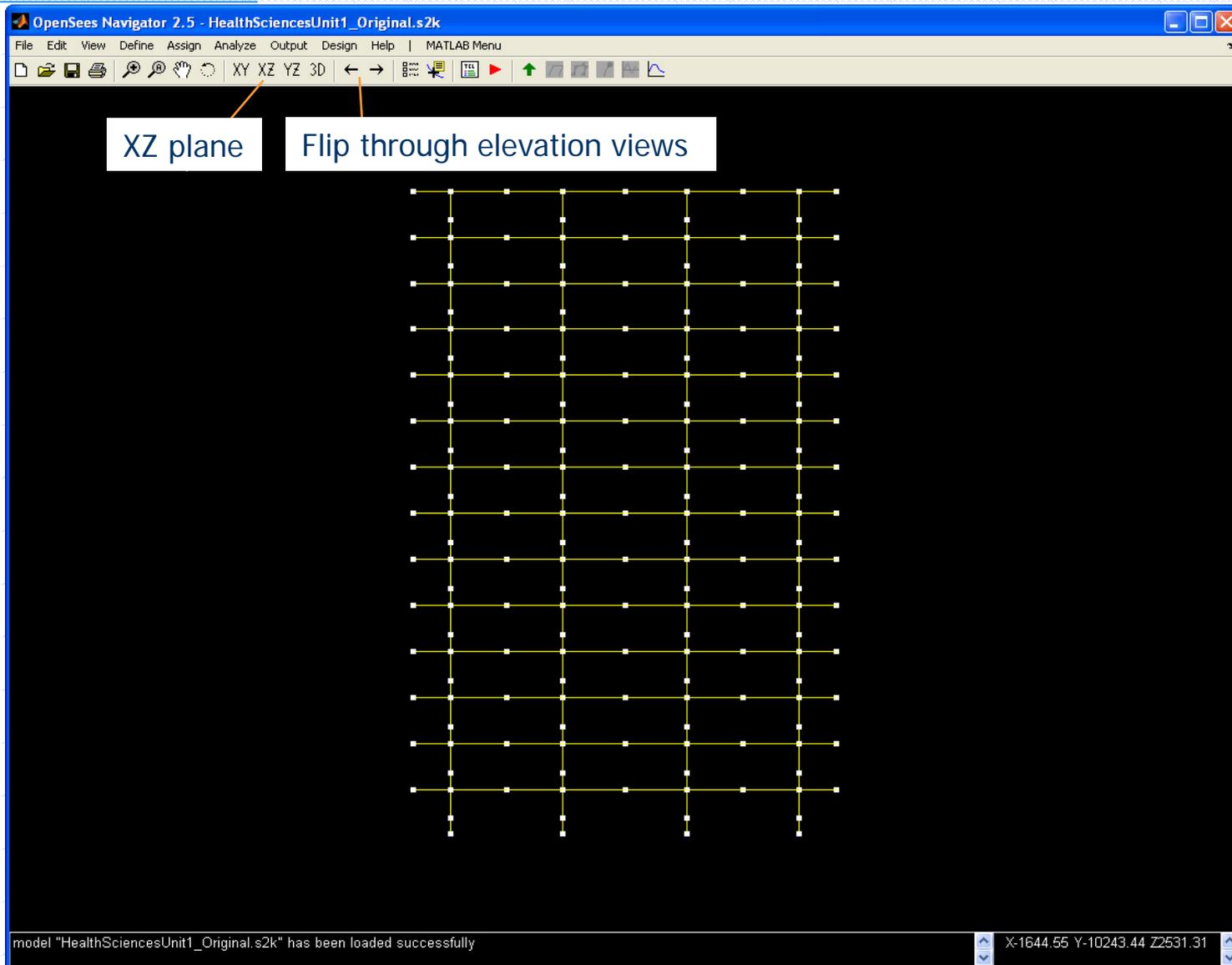
Define geometry: import from SAP2000



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)

Set Grid Lines...
 Clear all
 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 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 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
- Concrete03
- Concrete04
- Elastic
- ElasticNoTension
- ElasticPP
- ElasticPPGap
- Fatigue
- Hardening
- Hysteretic
- MinMax
- Parallel
- ReinforcingSteel
- Series
- Steel01
- Steel02
- Viscous

The screenshot displays the OpenSees Navigator 2.1 software interface. The main window shows a zipper frame geometry with nodes numbered 4, 11, 8, 5, 16, 14, 17, and 6. A 'Define Uniaxial Material' dialog box is open in the foreground, containing the following fields:

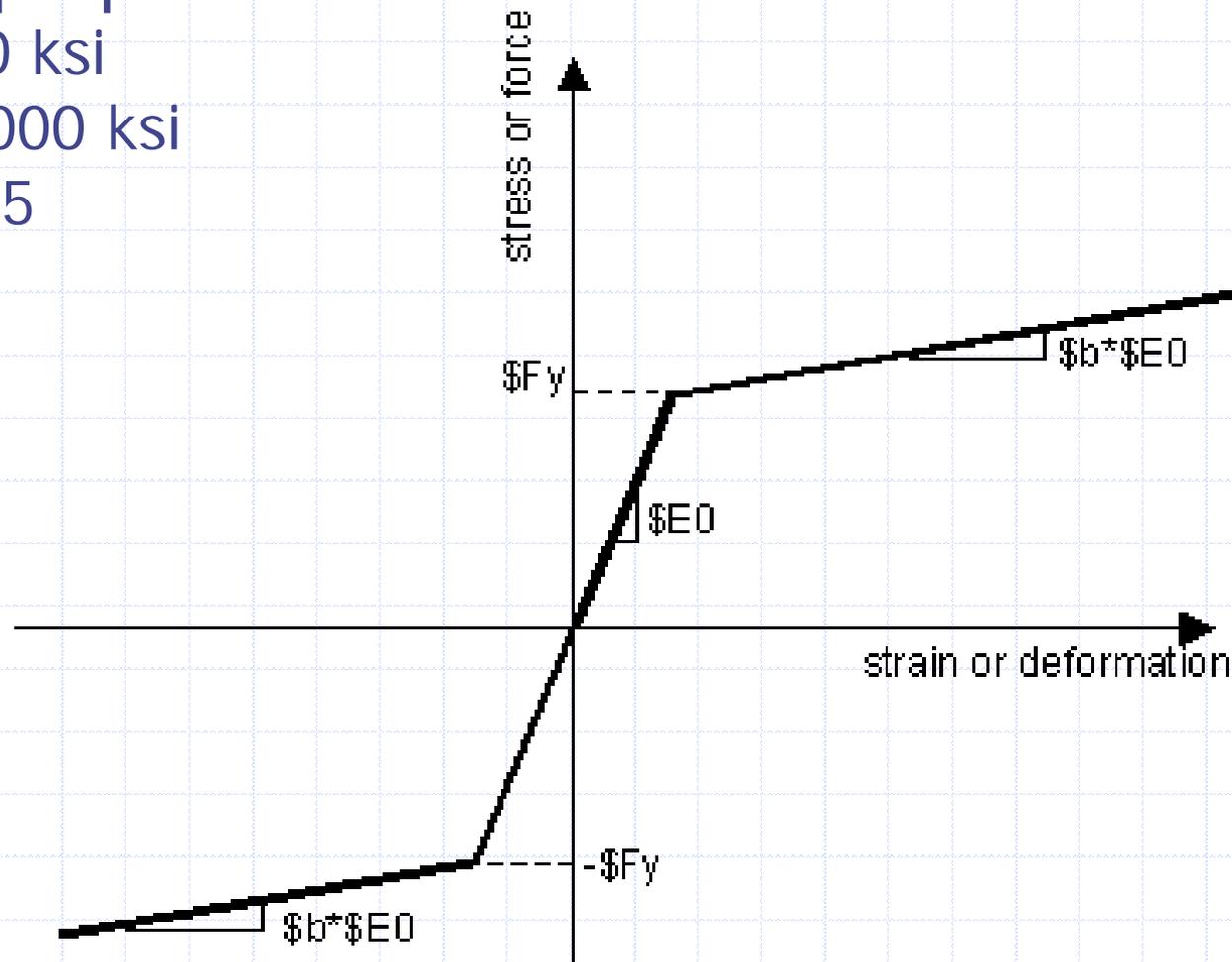
Field	Value
Add Material :	BoucWen
Modify Material :	ElasticDefault
Delete Material :	ElasticDefault

The status bar at the bottom of the software window displays the message: "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 8. A 'Define Uniaxial Material' dialog box is open, showing the following settings:

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

The status bar at the bottom indicates: zipper frame geometry has been generated successfully. X-56.17 Y172.73 Z0.00

Define material: nD materials

OpenSees Navigator 2.3

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

Materials ▶ Uniaxial Materials...
Friction Models...
Sections ▶
Experimental ▶
Elements ▶
Time Series...
Load Pattern...
Recorders...
Analyses Options

Define nD Material

Add Material : ElasticIsotropic

Modify Material :

Delete Material :

single solid-mesh geometry has been generated successfully X0.00 Y0.00 Z0.00

Templates:

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

Under development:

- NewTemplate3DElastoPlastic
- ModelsLargeDeformation

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, 23, 24, 25, 26, and 1. 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

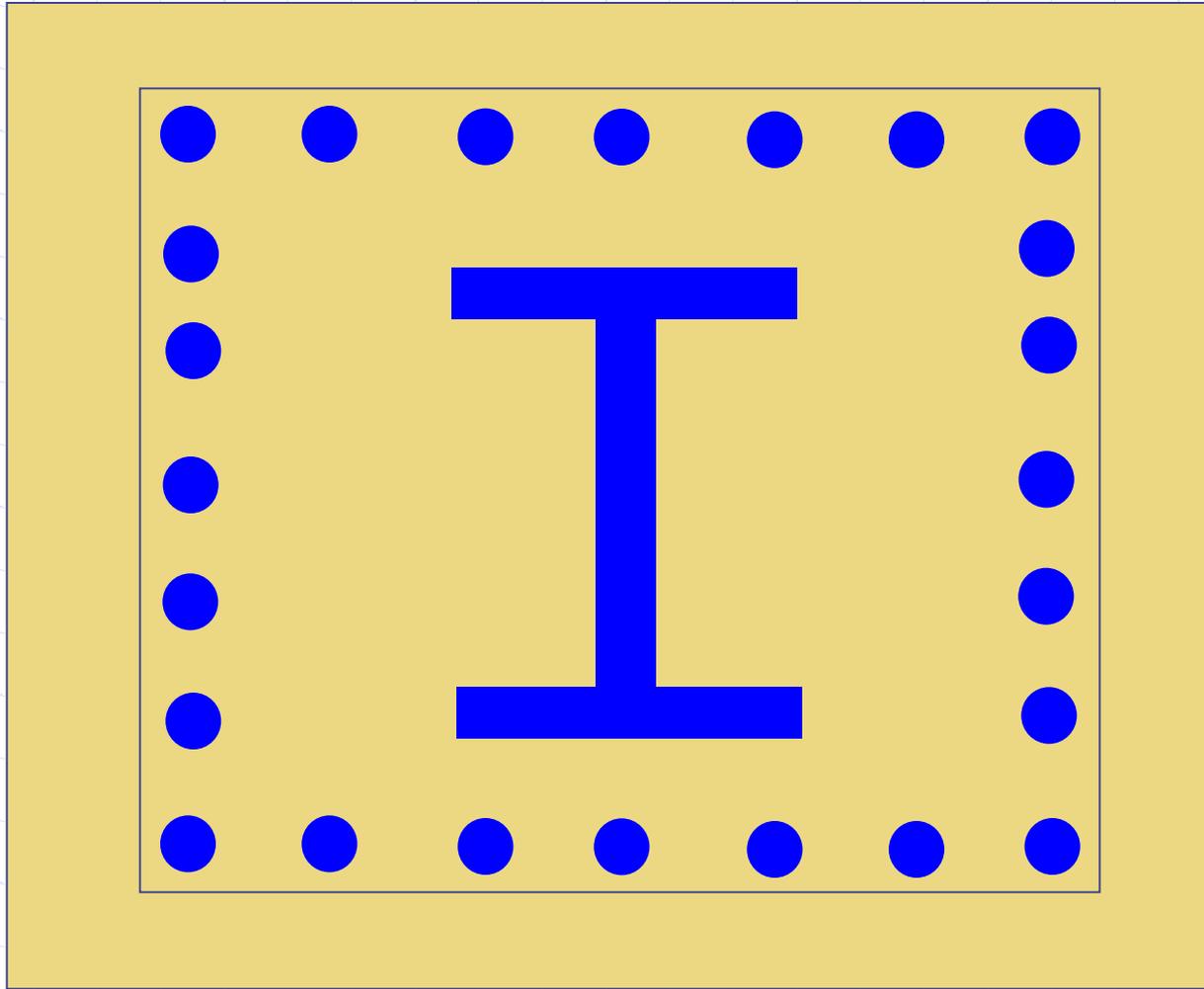
Section Name :	Section01	Add
Modulus of Elasticity (E) :	29000	Database
Cross-Sectional Area (A) :	20	
Moment of Inertia (Iz) :	2000	

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	<input type="button" value="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 software interface. The main window shows a structural model with nodes and elements. A menu is open, highlighting 'Area Sections...'. A dialog box titled 'Define Area Section' is overlaid on the model. The dialog box contains three dropdown menus: 'Add Section :', 'Modify Section :', and 'Delete Section :'. The 'Add Section :' dropdown is currently set to 'Bidirectional'. The status bar at the bottom of the window displays the message: 'section(2) with name "1stFloorBeam" has been defined/modified successfully' and the 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 :

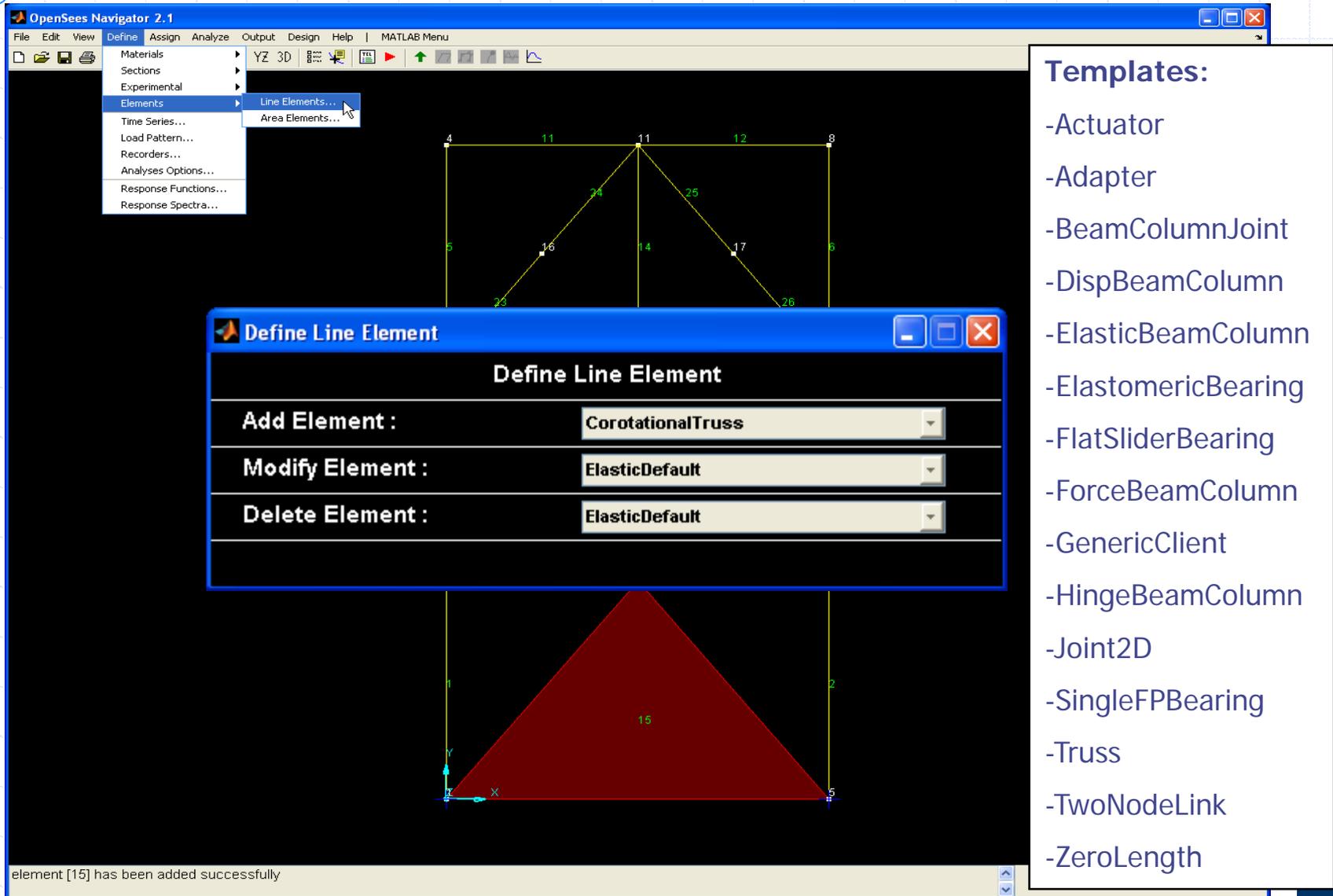
section(2) with name "1stFloorBeam" has been defined/modified successfully

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, and 'Line Elements...' is 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, 23, 25, and 26. A red triangle representing element 15 is highlighted in the foreground. The status bar at the bottom 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

Modulus of Elasticity (E) : 29000

Cross-Sectional Area (A) : 13.3

Moment of Inertia (Iz) : 248

Select Section from Database

Select Section from Database

Database : AISC

Section Name : W10X45

Direction : strong

Define line element: ForceBeamColumn

Define ForceBeamColumn Element

Element Name : 1stStoryColumn

Number Intergration Points (NIP) : 5

Section Type : 1stStoryColumn

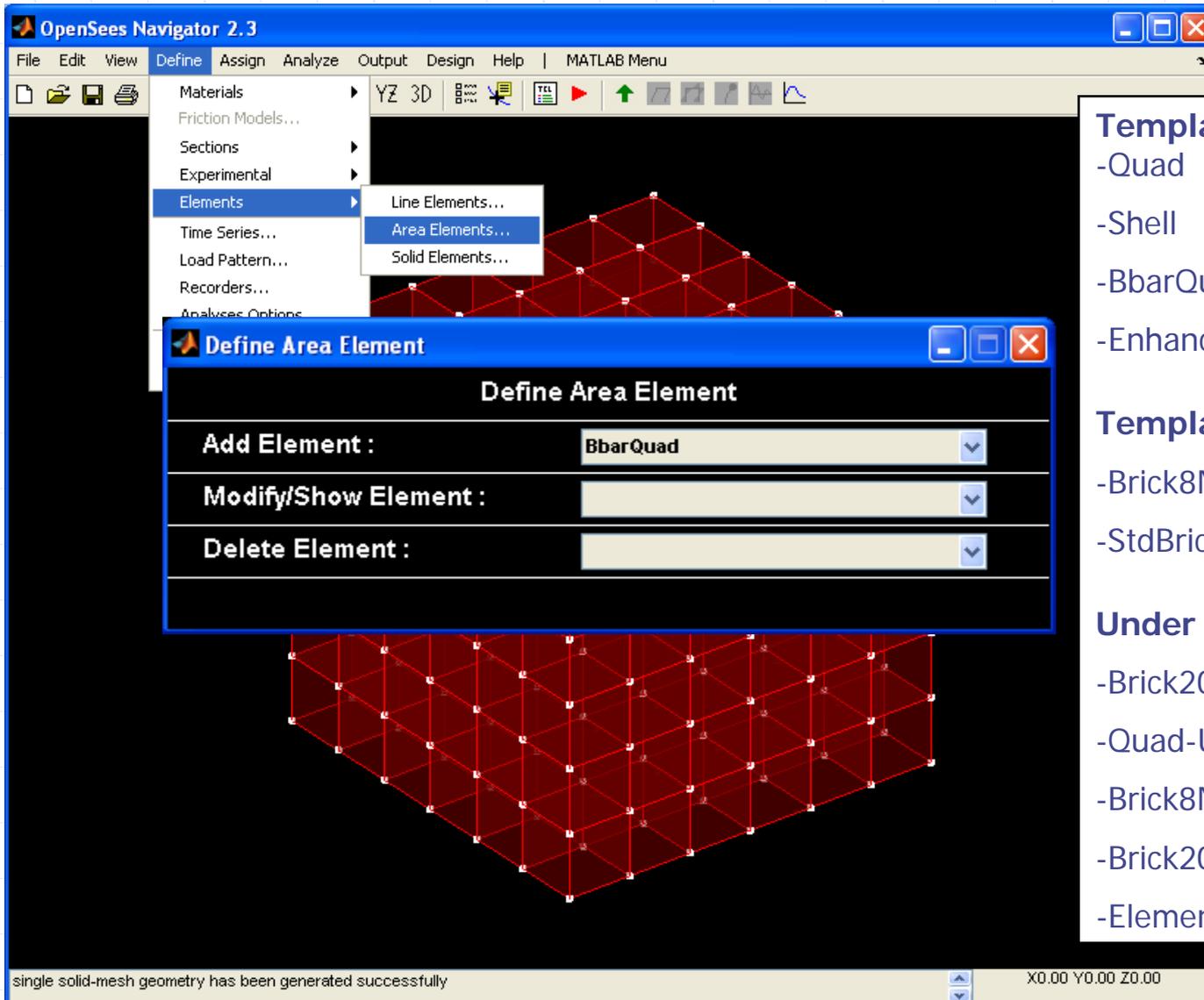
Optional Arguments :

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, and the 'Time Series...' option is selected. A 'Define TimeSeries' dialog box is displayed in the foreground, showing three rows of dropdown menus:

Operation	Selected Template
Add TimeSeries :	Constant
Modify TimeSeries :	LinearDefault
Delete TimeSeries :	LinearDefault

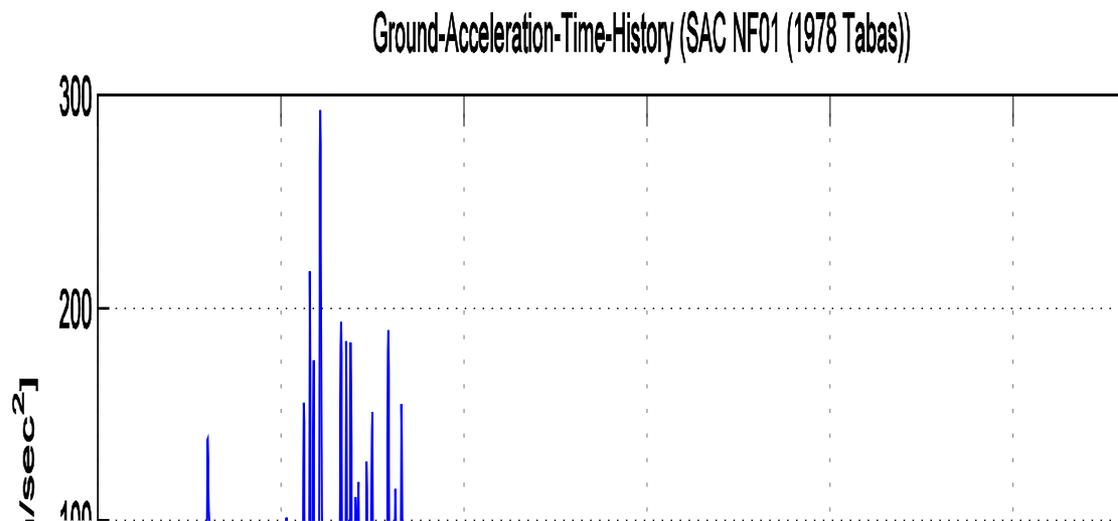
The background shows a 3D model of a truss structure. A status bar at the bottom left indicates 'element [15] has been added successfully' and the bottom right shows coordinates 'X-85.20 Y173.55 Z0.00'.

- Templates:**
- Constant
 - Linear
 - PathFile
 - PathFileFiltered
 - PathValue
 - Pulse
 - Rectangular
 - Sine
 - Triangle

Define TimeSeries: PathFile

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:

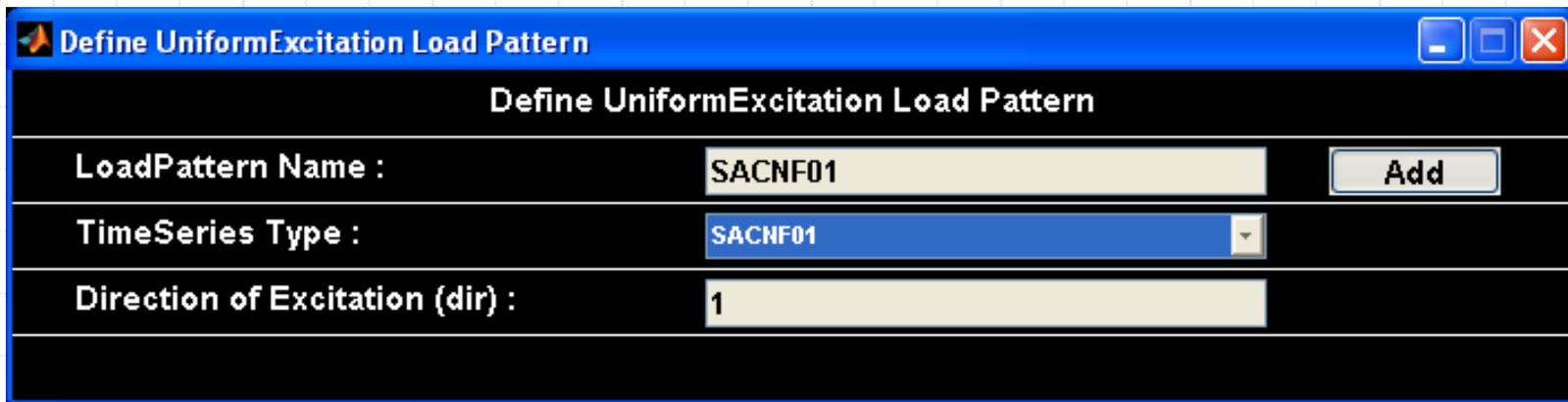
The screenshot shows the OpenSees Navigator 2.0 interface. The 'Define' menu is open, with 'Load Pattern...' selected. A dialog box titled 'Define LoadPattern' is displayed in the foreground, containing three dropdown menus:

Operation	Selected Value
Add LoadPattern :	Plain
Modify LoadPattern :	PlainDefault
Delete LoadPattern :	PlainDefault

The background shows a 2D truss model with a red triangular load applied to the top node. 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 LoadPattern: UniformExcitation



Define UniformExcitation Load Pattern

LoadPattern Name : SACNF01 Add

TimeSeries Type : SACNF01

Direction of Excitation (dir) : 1

Define recorder

The screenshot shows the OpenSees Navigator 2.0 interface. The main window displays a structural model with a red triangle and yellow lines. A 'Define Recorder' dialog box is open, showing the following settings:

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

The status bar at the bottom of the main window displays the message: "load-pattern(2) with name "SACNF01" has been defined/modified successfully". The coordinates "X-89.51 Y173.55 Z0.00" are also visible.

Templates:

- AreaElement
- BeamColumn Element
- Bearing Element
- Display
- Experimental Element
- Joint2D Element
- Node
- Solid Element
- Truss Element
- TwoNodeLink 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

Define BeamColumn Element Recorder

Recorder Name : ElemForces

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...

Define Analysis Options

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

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

The screenshot displays the OpenSees Navigator 2.0 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 :

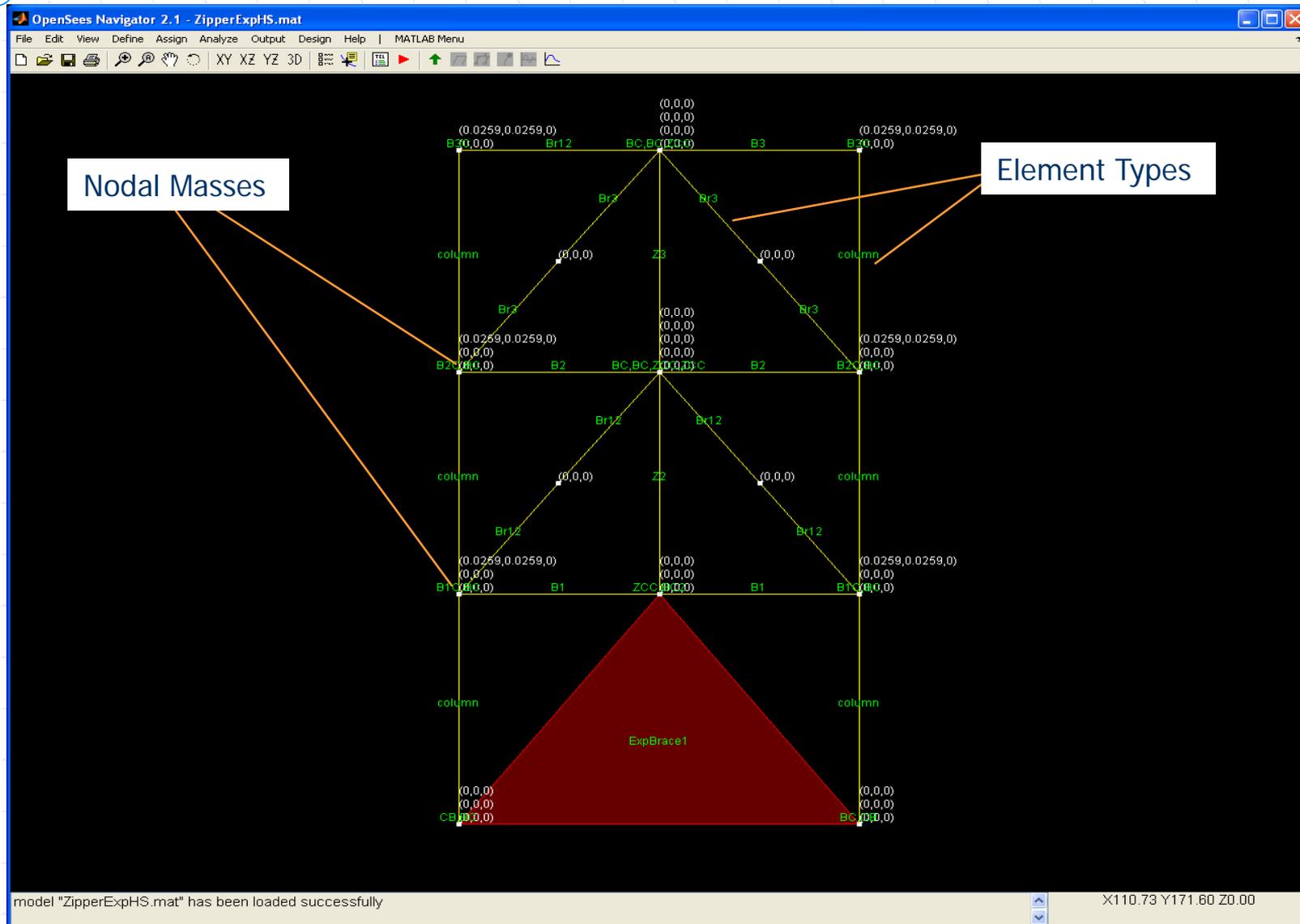
Assign Element Geometric Transformations

Assign Geometric Transformations :

Element Number(s) :

Geometric Transformation :

Display assigned properties



Define analysis case

The screenshot displays the OpenSees Navigator 2.1 interface for the file 'ZipperExpHS.mat'. The 'Analyze' menu is open, showing options: 'Define Analysis Cases...', 'Write OpenSees Input Files... (Ctrl+W)', and 'Run OpenSees (Ctrl+R)'. A 'Define Analysis Case' dialog box is open in the foreground, containing the following fields:

- Add Analysis Case :** New Analysis Case
- Modify Analysis Case :** EigenDefaultCase
- Delete Analysis Case :** EigenDefaultCase

The background shows a structural model with a red triangle at the bottom and a yellow trapezoidal shape above it. The status bar at the bottom indicates 'model "ZipperExpHS.mat" has been loaded successfully' and 'X-46.36 Y171.60 Z0.00'.

Defaults:

- StaticDefaultCase
- EigenDefaultCase

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 (with an **Add** button)
- 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 (with an **Add** button)
- Region Defined by :** Nodes (dropdown menu)
- Node or Element Number(s) :** all
- Mass Prop. Damping (alphaM) :** 0. (with a **Calculate** button)
- Kcurr Prop. Damping (betaK) :** 0.
- Kinit Prop. Damping (betaKinit) :** 0.
- Kcom Prop. Damping (betaKcom) :** 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

OpenSees Navigator 2.1 - ZipperExpHS.mat

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

New Model from Template... Ctrl+N
Open Model from File... Ctrl+O
Close Model
Save Model Ctrl+S
Save Model As...
Set OpenSees.exe Path...
Page Setup...
Print Setup...
Print Preview...
Print... Ctrl+P
1 ZipperExpHS.mat
2 Test.mat
3 Test.mat
4 tenpozan.tcl
Quit Ctrl+Q

Steps:

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

recorder(5) with name "EigenVector" has been defined/modified successfully X-49.00 Y170.79 Z0.00

Run OpenSees: write TCL files

The screenshot shows the OpenSees Navigator 2.1 software interface. The main window is titled "OpenSees Navigator 2.1 - ZipperExpHS.mat" and has a menu bar with "File", "Edit", "View", "Define", "Assign", "Analyze", "Output", "Design", "Help", and "MATLAB Menu". Below the menu bar is a toolbar with various icons. A white dialog box titled "Running OpenSees" is overlaid on the main window. The dialog box 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 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.

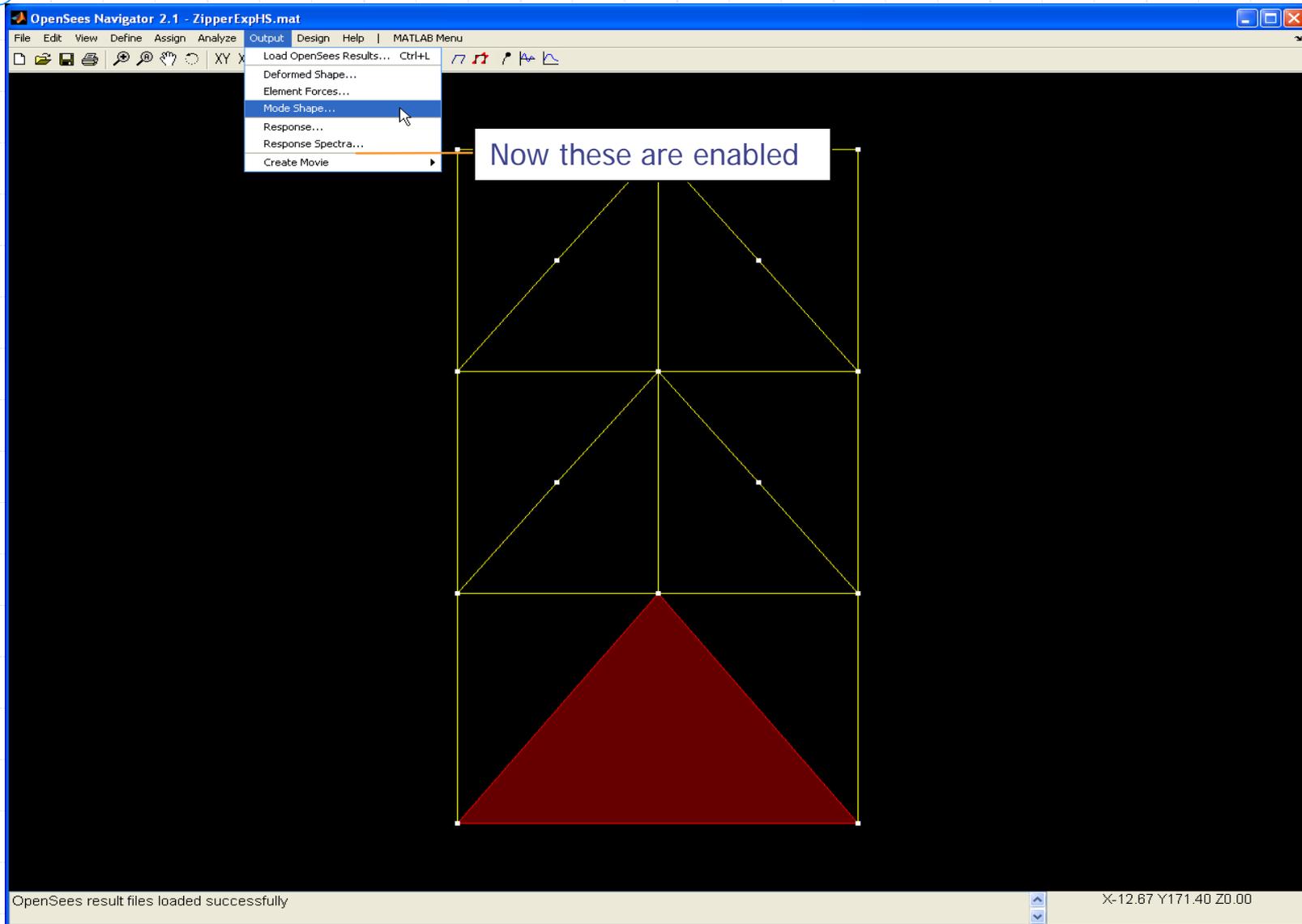
Steps:

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

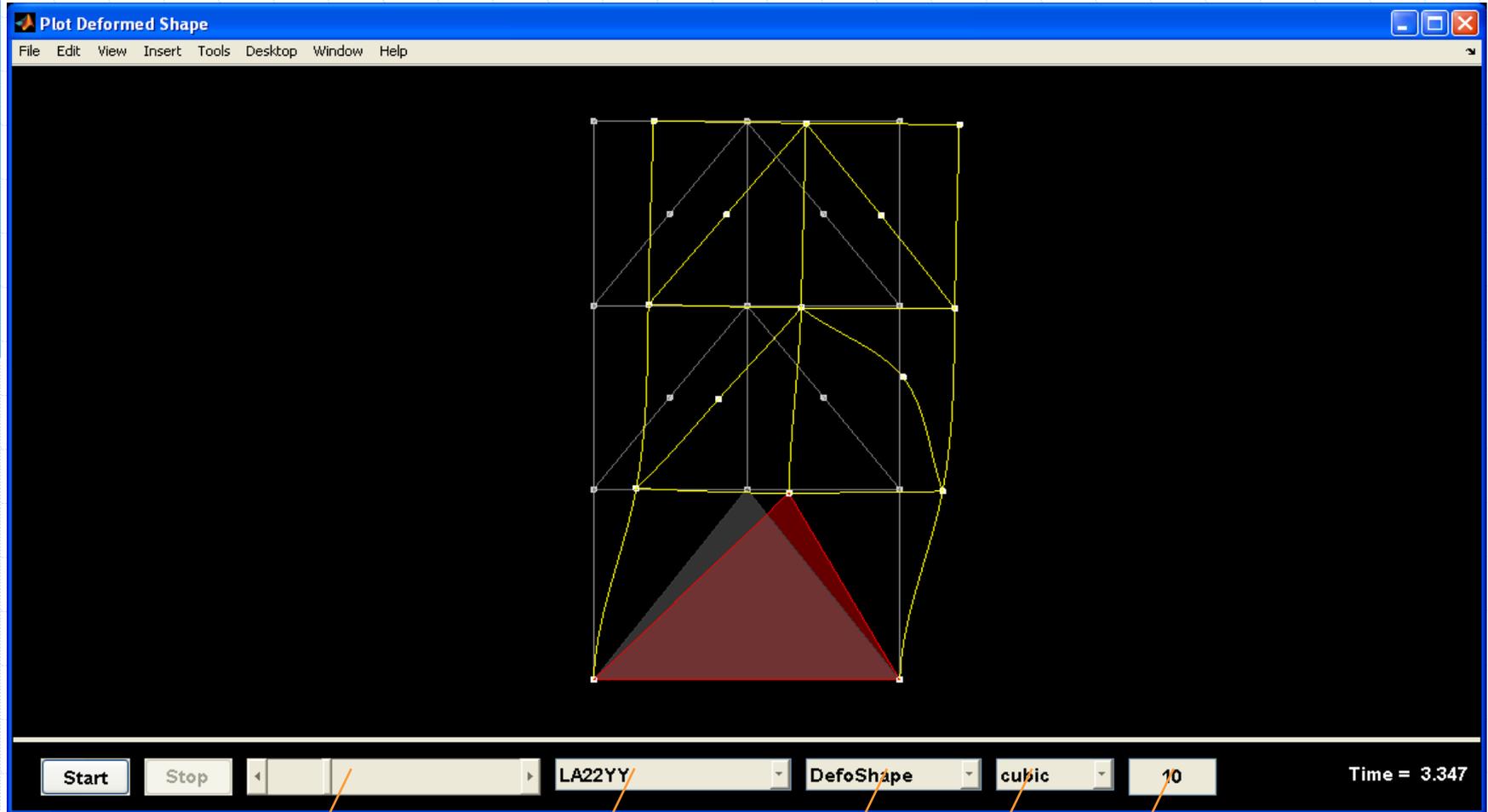
Post processing: load results

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 white 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) :" field contains a list with "EigenDefaultCase" and "SACNF01Case01". The background of the main window shows a red triangle on a black field, with yellow lines indicating a coordinate system. The status bar at the bottom left shows "recorder(5) with name 'Eigenvector' has been defined/modified successfully" and the bottom right shows "X-45.55 Y171.60 Z0.00".

Post processing: output



Post processing: plot deformed shape



Time Step

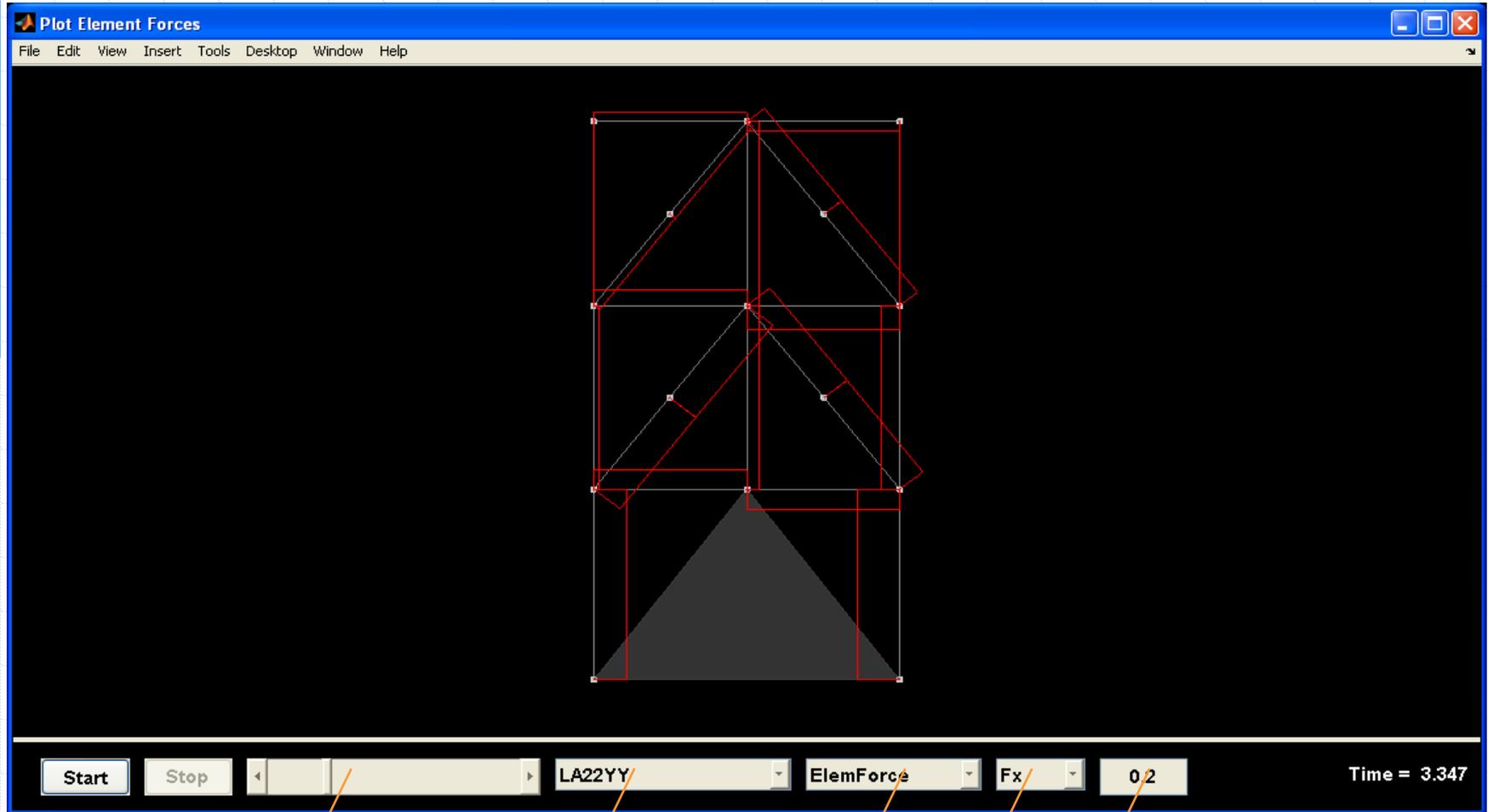
AnalysisCase

Recorder

Order

Magnification

Post processing: plot element forces



Time Step

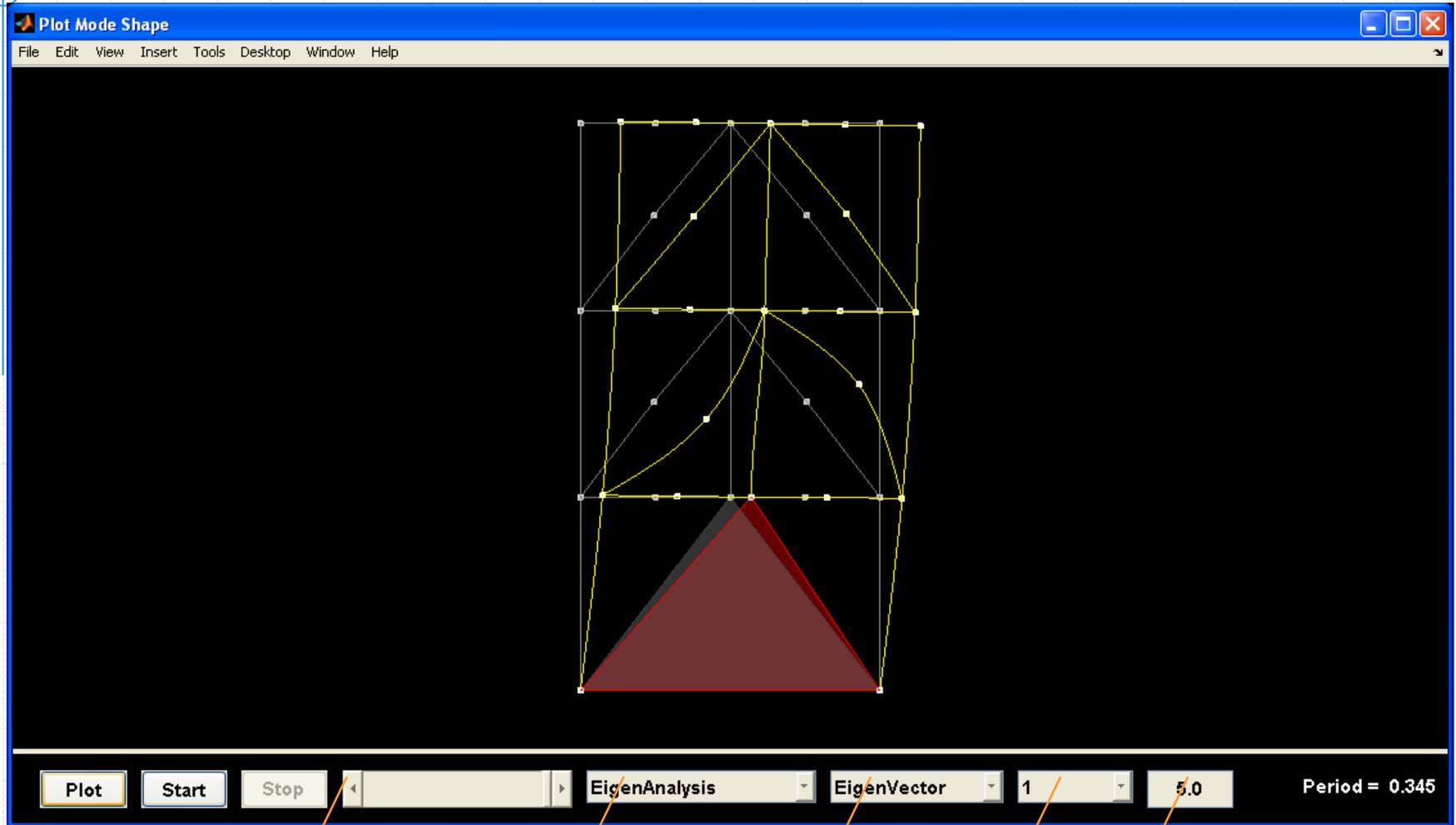
AnalysisCase

Recorder

Response

Magnification

Post processing: plot mode shape



Animation Speed

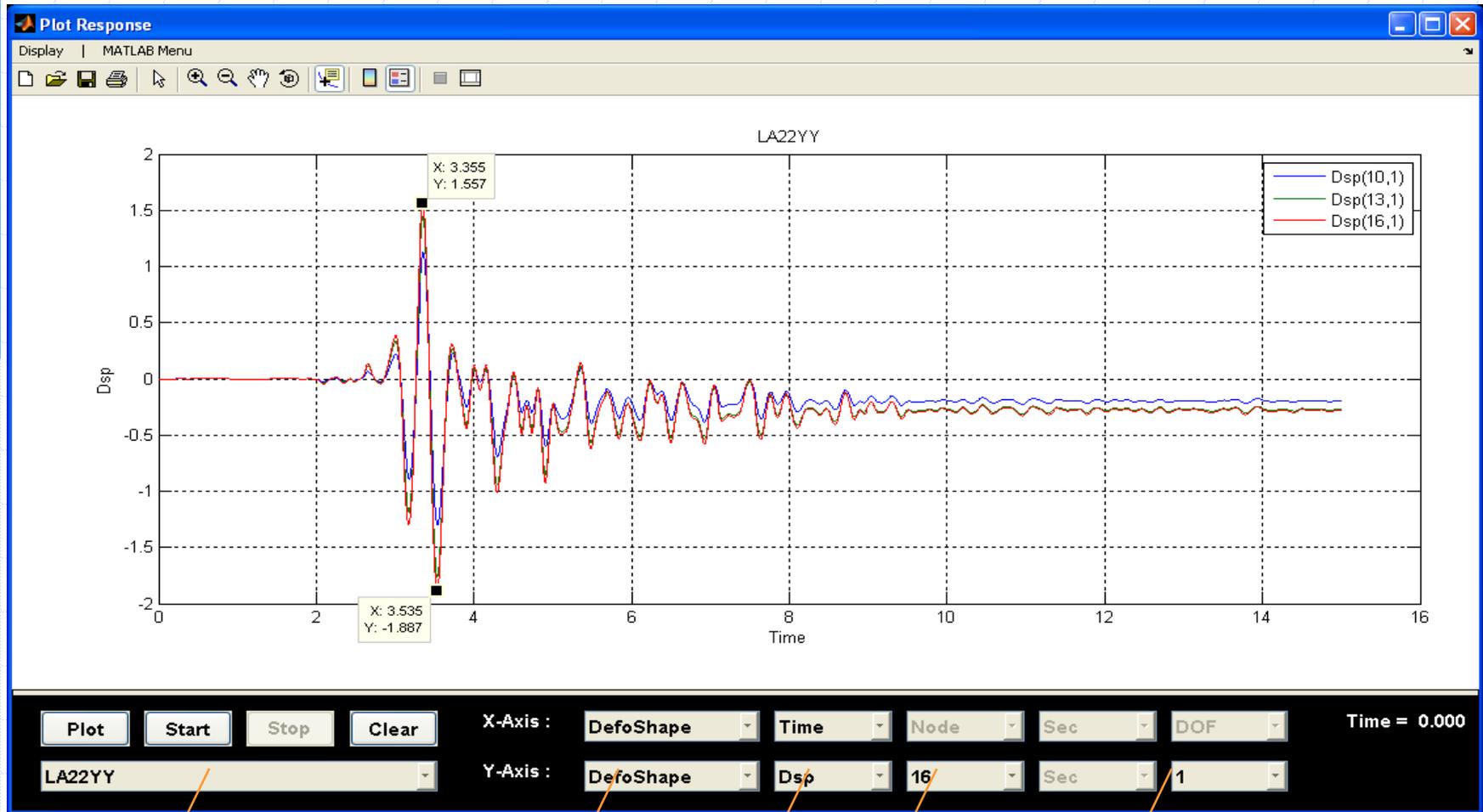
AnalysisCase

Recorder

Mode

Magnification

Post processing: plot response histories



AnalysisCase

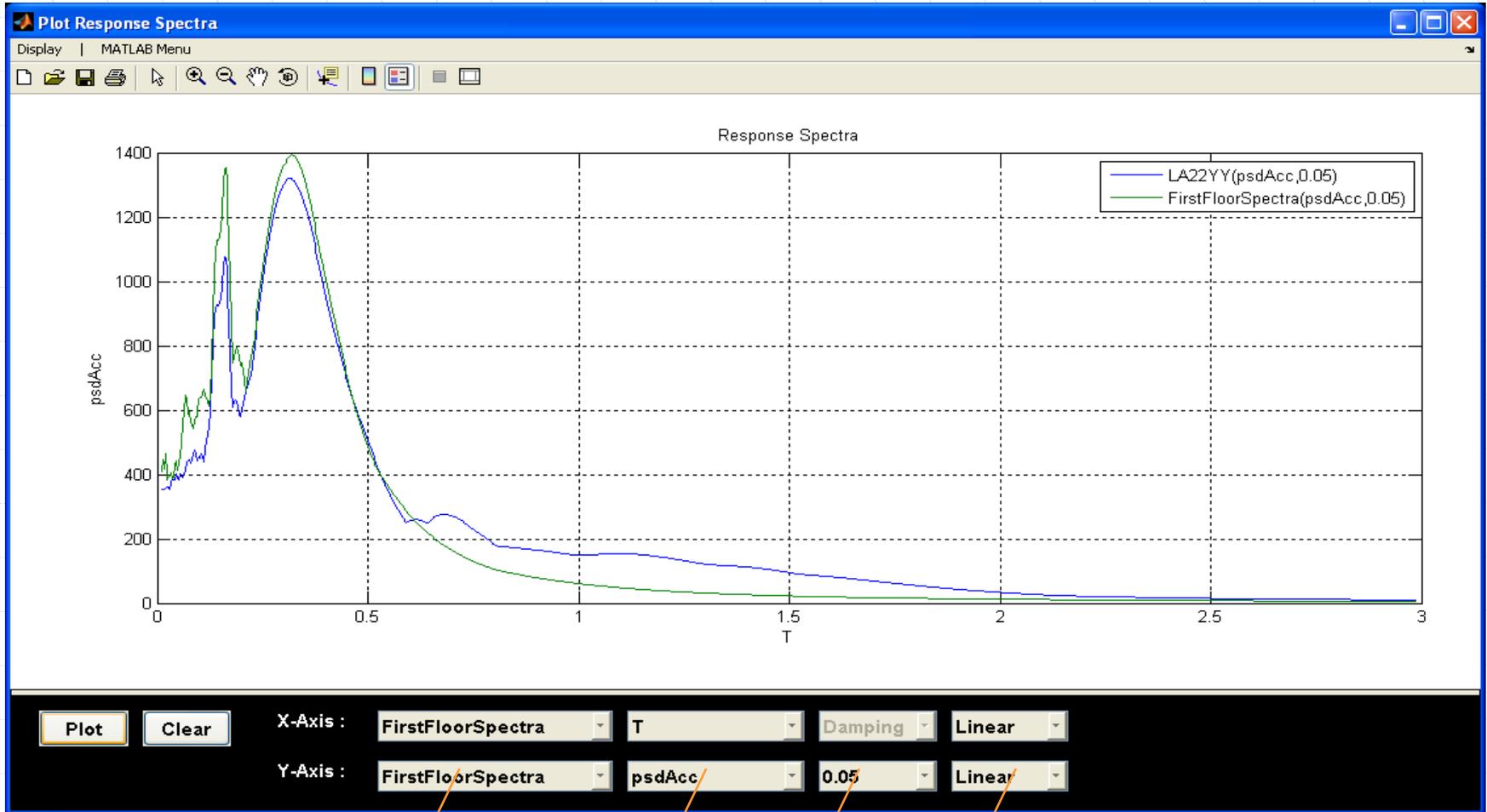
Recorder

Parameter

Node/Element

DOF

Post processing: plot response spectra



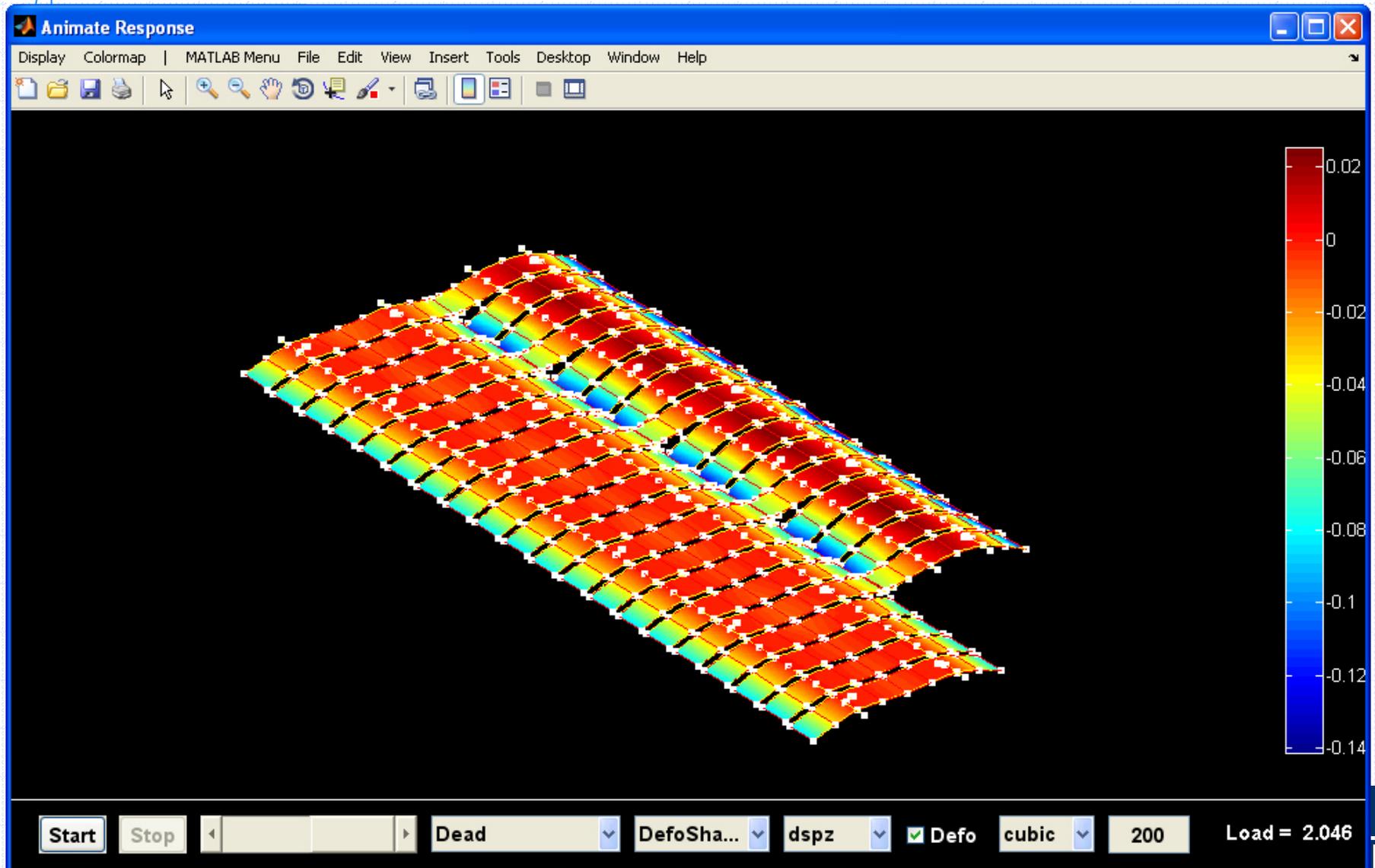
Spectra Name

Response

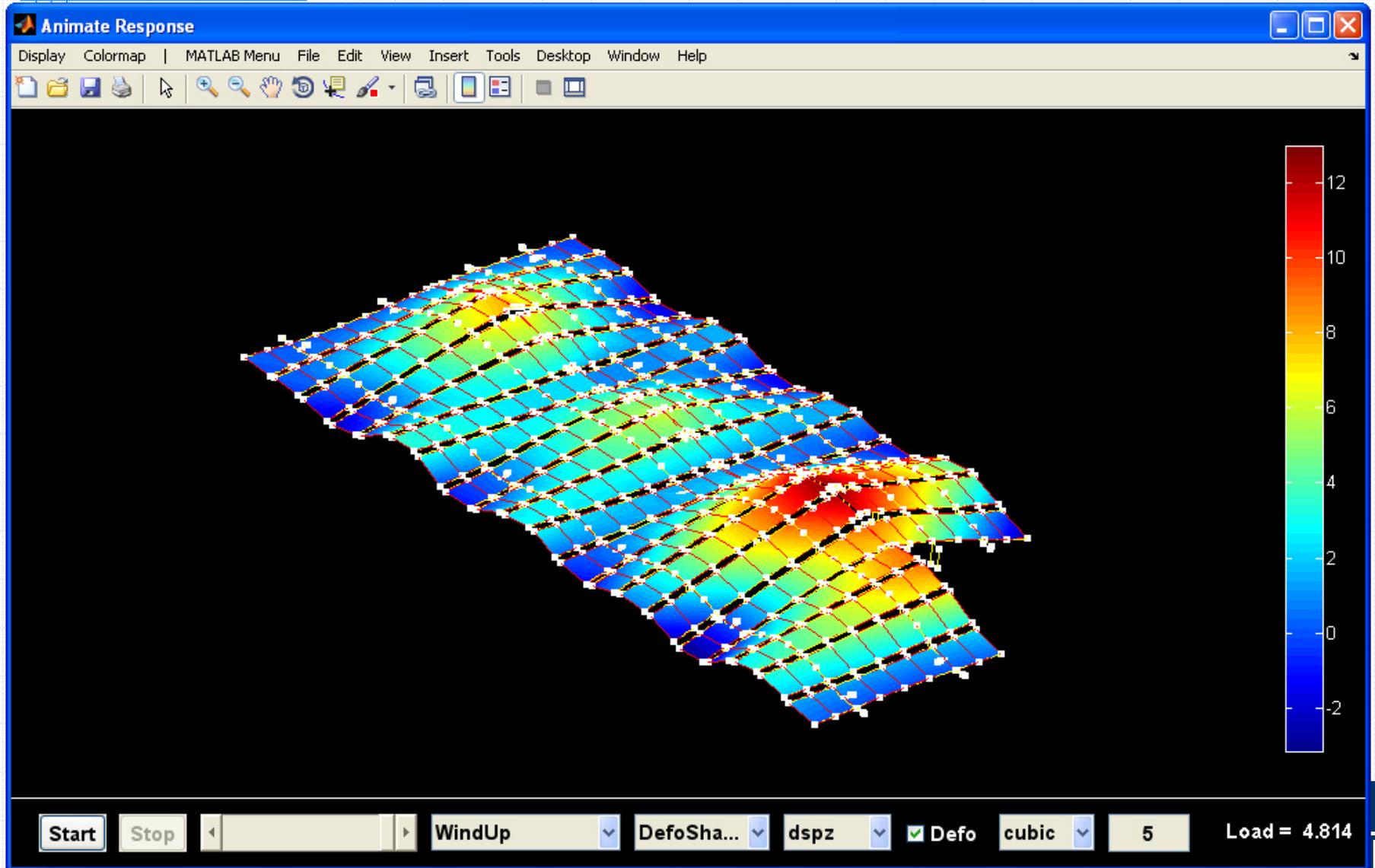
Damping

Axis Scale

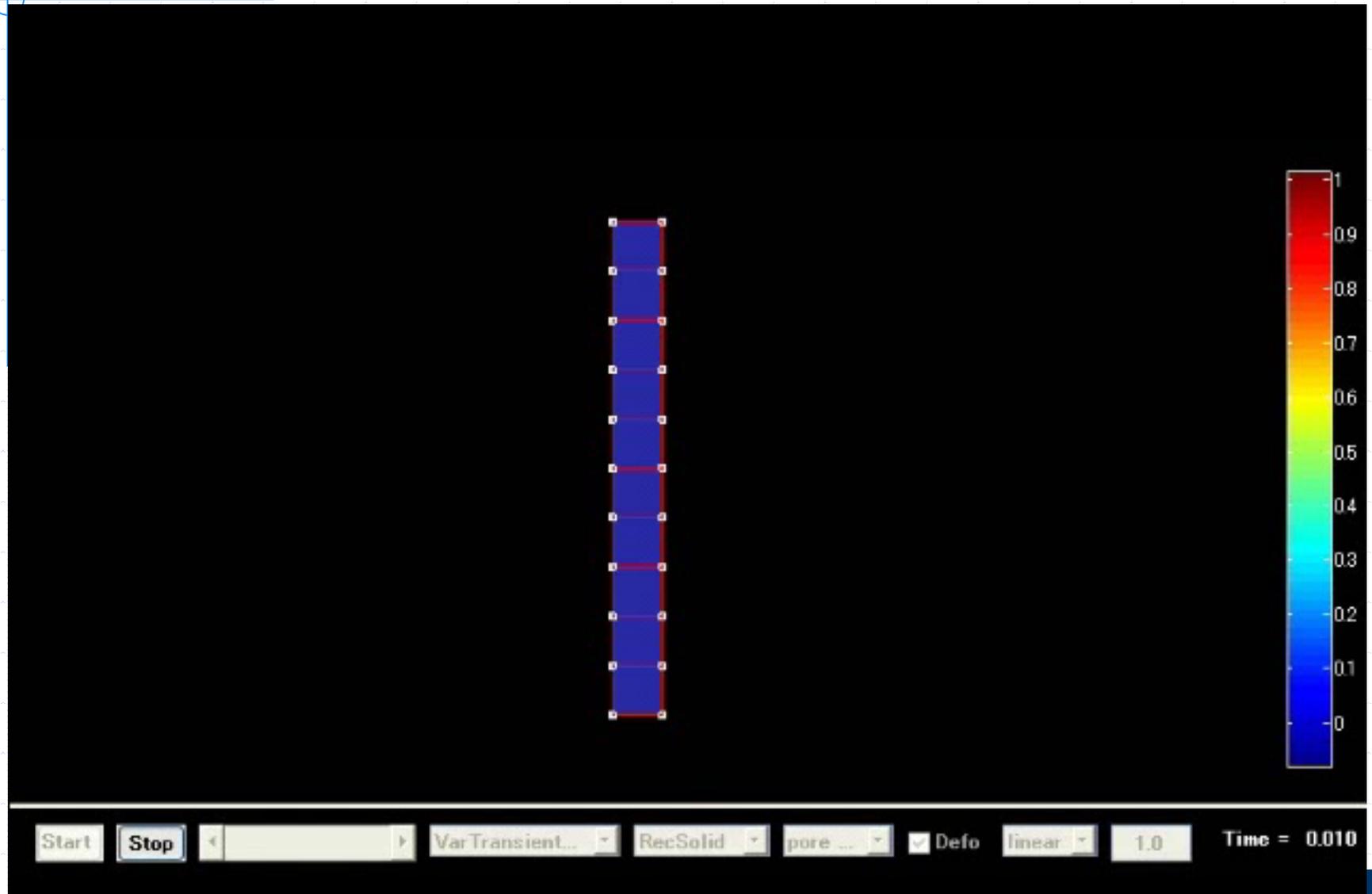
Post processing: animate response



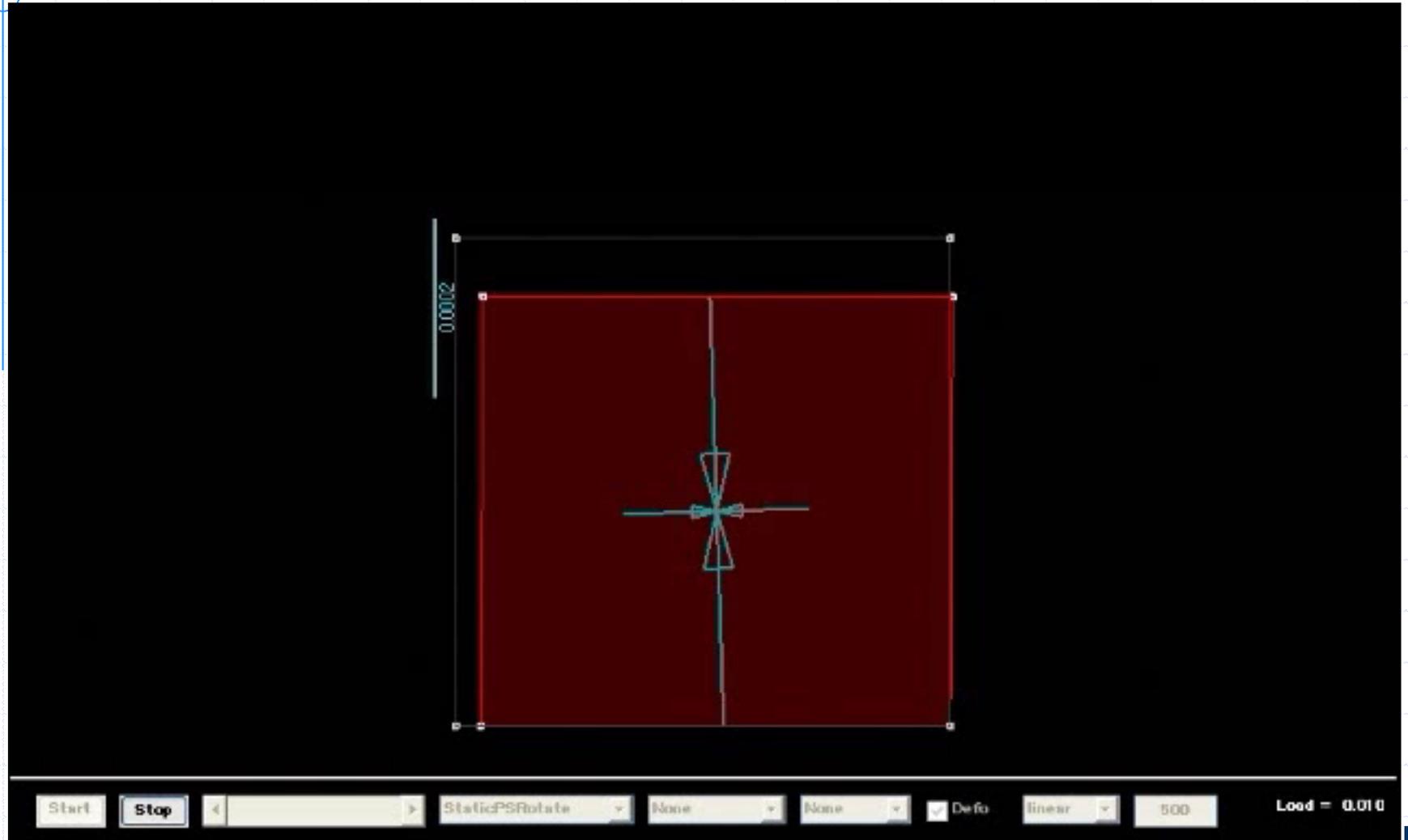
Post processing: animate response



Post processing: animate response



Post processing: principal σ and ε



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

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

Output

AISC Shear Capacity

```
The Shear Capacity parameters are :  
T_tw_p = 78.2264  
T_tw_r = 97.4274  
T_tw = 49.8795  
phi = 0.9  
Capacity = 265.5585  
FailureMode = Reaching yielding capacity 0.6*Fy
```

OK

AISC design toolbox: PMM interaction

AISC PMM Interaction Check

PMM Interaction Check of AISC Section

Section Shape :	W24x68	<input type="button" value="Calculate"/>
Yield Stress (Fy) :	50	[ksi]
Modulus of Elasticity (E) :	29000	[ksi]
Demand :		
Applied Axial Force (Pu) :		[kips]
Applied Moment about X axis (Mux) :		[kips - in]
Applied Moment about Y axis (Muy) :		[kips - in]
Compression :		
Effective Length (kLx) :		[in]
Effective Length (kLy) :		[in]
Bending :		
Unbraced Length (Lb) :		[in]
Bending Coefficient (Cb) :	1	[-]

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 and self-executable versions are available.

Website: home

OPENSEES NAVIGATOR	
HOME INTRODUCTION MANUALS TUTORIALS PRESENTATIONS DISCUSSION UPDATES DOWNLOADS LINKS	
Home> <input type="text"/> <input type="button" value="Search"/>	
Introduction	<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 andreas.schellenberg@gmail.com or yangtony2004@gmail.com so that we can add the e-mail addresses to the OpenSees Navigator user list. We will use such list to contact everyone about new releases or major updates. We will try our best to improve the next release.</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 & Tony Yang</p>
Manuals	
Tutorials	
Presentations	
Discussion	
Updates	
Downloads	
Links	
<input type="checkbox"/> Hit Counter	OpenSees Navigator ©2004-2006 UC Berkeley . All rights reserved. Please contact us with any questions or comments.

Website: download

OPENSEES NAVIGATOR

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

Home > Downloads > Stand-Alone Windows >

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

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