OpenSees Navigator & Hybrid Simulation

Andreas Schellenberg & Tony Yang

Department of Civil and Environmental Engineering
University of California, Berkeley
Introduction

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

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

- Stick Model
- Beam Model
- EBF Model
- Zipper Frame
- Inverted-V Braced Frame
- Moment Frame
- Single Area Mesh
### Define Geometry: Zipper Frame

#### Define Zipper Frame Geometry

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimension (ndm)</td>
<td>2d</td>
</tr>
<tr>
<td>Number of Stories (NOS)</td>
<td>3</td>
</tr>
<tr>
<td>Number of Bays (NOB)</td>
<td>1</td>
</tr>
<tr>
<td>Story Height (SH)</td>
<td>52</td>
</tr>
<tr>
<td>Bay Width (BW)</td>
<td>80</td>
</tr>
<tr>
<td>Boundary Condition (BC)</td>
<td>pinned</td>
</tr>
<tr>
<td>Brace Bay Config (BraceBay)</td>
<td>BraceBay</td>
</tr>
<tr>
<td>Num Segments in Col (NSC)</td>
<td>1</td>
</tr>
<tr>
<td>Num Segments in Beam (NSB)</td>
<td>1</td>
</tr>
<tr>
<td>Num Segments in Brace (NSBR)</td>
<td>2</td>
</tr>
<tr>
<td>Num Segments in Z-Col (NSZC)</td>
<td>1</td>
</tr>
<tr>
<td>Brace Offset (BraceOffset)</td>
<td>None</td>
</tr>
</tbody>
</table>
View Geometry: Set Display Options

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

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

General:
- Model
- Global Axes
- Grid Lines
View Geometry: Display
Edit Geometry

Node:
- Add
- Delete
- Move

Element:
- Add
- Delete
- Divide/Join
- Add/Delete ZeroLength

Zipper frame geometry has been generated successfully
Define Material: Uniaxial Materials

Templates:
- BoucWen
- Concrete01
- Concrete02
- Concrete03
- Elastic
- ElasticNoTension
- ElasticPP
- ElasticPPGap
- Fatigue
- Hardening
- Hysteretic
- MinMax
- Parallel
- Series
- Steel01
- Steel02
- Viscous
Define Steel01 Material: A50

Material Properties
- $F_y = 50$ ksi
- $E = 29000$ ksi
- $b = 0.05$
Define Steel01 Material: A50

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material Name</td>
<td>A50</td>
</tr>
<tr>
<td>Yield Stress (Fy)</td>
<td>50</td>
</tr>
<tr>
<td>Modulus of Elasticity (E)</td>
<td>29000</td>
</tr>
<tr>
<td>Hardening Ratio (b)</td>
<td>0.05</td>
</tr>
<tr>
<td>Iso Hardening Parameter (a1)</td>
<td>0.0</td>
</tr>
<tr>
<td>Iso Hardening Parameter (a2)</td>
<td>1.0</td>
</tr>
<tr>
<td>Iso Hardening Parameter (a3)</td>
<td>0.0</td>
</tr>
<tr>
<td>Iso Hardening Parameter (a4)</td>
<td>1.0</td>
</tr>
</tbody>
</table>
Define Material: nD Materials

Templates:
- ElasticIsotropic
- J2Plasticity
- PlaneStress
- PlateFiber
Define Section: Line Sections

Templates:
- Aggregator
- Elastic
- Fiber
- Uniaxial
Define Fiber Section: 1stFloorBeam

<table>
<thead>
<tr>
<th>Define Fiber Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section Name: 1stFloorBeam</td>
</tr>
<tr>
<td>Add Fiber: Fiber</td>
</tr>
<tr>
<td>Modify Fiber:</td>
</tr>
<tr>
<td>Delete Fiber:</td>
</tr>
<tr>
<td>Add Patch: AISC</td>
</tr>
<tr>
<td>Modify Patch:</td>
</tr>
<tr>
<td>Delete Patch:</td>
</tr>
<tr>
<td>Add Layer: Straight</td>
</tr>
<tr>
<td>Modify Layer:</td>
</tr>
<tr>
<td>Delete Layer:</td>
</tr>
</tbody>
</table>
## Define Fiber Section: AISC Patch

![Define AISC Patch Window](image)

<table>
<thead>
<tr>
<th>Patch Name</th>
<th>01 Patch</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material Type</td>
<td>A50</td>
</tr>
<tr>
<td>AISC Section Name</td>
<td>W24X68</td>
</tr>
<tr>
<td>Number of Fibers</td>
<td></td>
</tr>
<tr>
<td>dw (nfdw)</td>
<td>10</td>
</tr>
<tr>
<td>tw (nftw)</td>
<td>1</td>
</tr>
<tr>
<td>bf (nfbf)</td>
<td>10</td>
</tr>
<tr>
<td>tf (nftf)</td>
<td>1</td>
</tr>
<tr>
<td>Optional Arguments</td>
<td></td>
</tr>
<tr>
<td>Counter-Clockwise Rot (Theta)</td>
<td>0.0000</td>
</tr>
</tbody>
</table>
Define Section: Area Section

Templates:
- Bidirectional
- ElasticMembranePlate
- PlateFiber
Define Element: Line Element

Templates:
- CorotationalTruss
- DispBeamColumn
- ElasticBeamColumn
- ExpBeamColumn
- ExpChevronBrace
- ExpTruss
- ExpZeroLength
- ForceBeamColumn
- HingeBeamColumn
- Truss
- ZeroLength
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
Database: AISC
Section Name: W10X45
Direction: strong
Define ForceBeamColumn Element

![Define ForceBeamColumn Element dialog box]

- **Element Name**: 1stStoryColumn
- **Number Intergration Points (NIP)**: 5
- **Section Type**: 1stStoryColumn
- **Optional Arguments**:
  - **Mass Density (massDens)**: 0.0
  - **Maximum Iterations (maxIters)**: 10
  - **Tolerance (tol)**: 1E-8
Define TimeSeries

Templates:
- Constant
- Linear
- PathFile
- PathFileFiltered
- PathValue
- Pulse
- Rectangular
- Sine
- Triangle
Define PathFile TimeSeries: SACNF01

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

Ground Acceleration Time History (SAC NF01 (1978 Tabas))
Define LoadPattern

Templates:
- Plain
- UniformExcitation
Define UniformExcitation LoadPattern

Define UniformExcitation Load Pattern

- LoadPattern Name: SACNF01
- TimeSeries Type: SACNF01
- Direction of Excitation (dir): 1
Define Recorder

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

Defaults:
- DefoShape
- EigenVector
## Define Node Recorder

<table>
<thead>
<tr>
<th>Recorder Name</th>
<th>DefoShape</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node Number(s)</td>
<td>all</td>
</tr>
<tr>
<td>Deformations</td>
<td></td>
</tr>
<tr>
<td>Displacements</td>
<td>✓</td>
</tr>
<tr>
<td>Velocities</td>
<td>✓</td>
</tr>
<tr>
<td>Accelerations</td>
<td>✓</td>
</tr>
<tr>
<td>Incremental Displacements</td>
<td></td>
</tr>
<tr>
<td>Incremental Delta Displacements</td>
<td></td>
</tr>
<tr>
<td>Eigenvectors</td>
<td></td>
</tr>
<tr>
<td>Forces</td>
<td></td>
</tr>
<tr>
<td>Reaction Forces Without Inertia</td>
<td></td>
</tr>
<tr>
<td>Reaction Forces Including Inertia</td>
<td></td>
</tr>
<tr>
<td>Unbalanced Loads Without Inertia</td>
<td></td>
</tr>
<tr>
<td>Unbalanced Loads Including Inertia</td>
<td></td>
</tr>
</tbody>
</table>

![Define Node Recorder interface](image-url)
Define BeamColumn Recorder

<table>
<thead>
<tr>
<th>Recorder Name</th>
<th>ElemForces</th>
<th>Add</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Number(s)</td>
<td>all</td>
<td></td>
</tr>
<tr>
<td>Arguments</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Global Resisting Forces</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Local Resisting Forces</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Section Response</td>
<td>Section</td>
<td></td>
</tr>
<tr>
<td>Section Number(s)</td>
<td>15</td>
<td></td>
</tr>
<tr>
<td>Arguments</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Forces</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deformations</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stiffness</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fiber Response</td>
<td>Fiber</td>
<td></td>
</tr>
<tr>
<td>Arguments</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stress/Strain</td>
<td>0.</td>
<td>Y-Coor</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.</td>
</tr>
</tbody>
</table>
Define Analysis Options

Defaults:
- StaticDefault
- TransientDefault
- EigenDefault
Define New Analysis Options

**Integrator Type:**
For example use AlphaOS Method for Hybrid Simulation

**Solution Algorithm:**
The AlphaOS Method requires a Linear solution algorithm
Assign Menu

Node:
- SP Constraints
- MP Constraints
- Masses
- Loads
- Displacements

Element:
- Types
- GeoTrans
- Rotations
- ZeroLength Axis

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

Assign Nodal Masses

Replace/Add/Delete Masses:
- Replace
- Add
- Delete
- Display

Node Number(s):
- 2 3 5 6

Mass X-dir:
- 0.5

Mass Y-dir:
- 0.5

Mass Moment of Inertia Z-dir:
- 0.

Select Nodes

X-Coordinate:
- 240

Y-Coordinate:
- 0.

Assign Element Types

Assign Element Types:
- Assign

Element Number(s):
- 17:24

Element Type:
- Beam

Assign Element Geometric Transformations

Assign Geometric Transformations:
- Assign

Element Number(s):
- 17:24

Geometric Transformation:
- Corotational
Assigned Properties

Nodal Masses

Element Types
Define Analysis Cases

Defaults:
- StaticDefaultCase
- EigenDefaultCase
Define New Analysis Case

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

Steps:
1. Set OpenSees.exe Path (needs to be done only once)
To Run OpenSees

Steps:

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

- Experimental Control
  - SCRAMNet, dSpace, xPC Target

- Experimental Setup
  - ChevronBrace, ChevronBraceJntOff, NoTransformation, OneActuator, ThreeActuators, TwoActuators

- Experimental Site
  - ActorSite, LocalSite, RemoteSite

- Experimental Element
Define Experimental Control

Templates:
- SCRAMNET
- SimChevronBrace
- Simulation
- dSpace
- xPC Target
Define ExpControl: xPC Target

<table>
<thead>
<tr>
<th>Control Name</th>
<th>BraceExpCtrlXPC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Setups</td>
<td>1</td>
</tr>
<tr>
<td>Predictor-Corrector</td>
<td>Dsp</td>
</tr>
<tr>
<td>Type</td>
<td>Dsp</td>
</tr>
<tr>
<td>xPC Target IP Address</td>
<td>192.168.2.20</td>
</tr>
<tr>
<td>xPC Target IP Port</td>
<td>22222</td>
</tr>
<tr>
<td>Application Name</td>
<td>HybridControllerPoly3</td>
</tr>
<tr>
<td>Application Path</td>
<td>TestModels\c\mCode\xPCTarget-STS\</td>
</tr>
</tbody>
</table>
Define Experimental Setup

Templates:
- ChevronBrace
- ChevronBraceJntOff
- NoTransformation
- OneActuator
- ThreeActuators
- TwoActuators
## Define ExpSetup: ChevronBrace

<table>
<thead>
<tr>
<th>Setup Name :</th>
<th>BraceExpSetupJointOff</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental Control Type :</td>
<td>BraceExpCtrlXPC</td>
</tr>
<tr>
<td>Geometry Type (ntGeomFlag) :</td>
<td>nonlinear, horizontal right</td>
</tr>
<tr>
<td>Actuator Length 1 (La1) :</td>
<td>124.5</td>
</tr>
<tr>
<td>Actuator Length 2 (La2) :</td>
<td>176.625</td>
</tr>
<tr>
<td>Actuator Length 3 (La3) :</td>
<td>176.625</td>
</tr>
<tr>
<td>Rigid Link Length 1 (L1) :</td>
<td>53</td>
</tr>
<tr>
<td>Rigid Link Length 2 (L2) :</td>
<td>108</td>
</tr>
<tr>
<td>Rigid Link Length 3 (L3) :</td>
<td>108</td>
</tr>
<tr>
<td>Rigid Link Length 4 (L4) :</td>
<td>53</td>
</tr>
<tr>
<td>Rigid Link Length 5 (L5) :</td>
<td>24.625</td>
</tr>
<tr>
<td>Rigid Link Length 6 (L6) :</td>
<td>24.625</td>
</tr>
</tbody>
</table>

**Optional Parameters :**
- Dsp Control Factor (dspCtrlFact) : [1 1 1]
- Vel Control Factor (velCtrlFact) : [1 1 1]
- Acc Control Factor (accCtrlFact) : [1 1 1]
- Dsp Daq Factor (dspDaqFact) : [1 1 1 1 1]
- Force Daq Factor (frcDaqFact) : [1 1 1 1 1]
Define Experimental Site

Templates:
- LocalSite
- RemoteSite
Define ExpSite: RFS
Define ExpElement: ChevronBrace

<table>
<thead>
<tr>
<th>Element Name</th>
<th>ExpChevronBrace01</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental Site Type</td>
<td>RFS</td>
</tr>
<tr>
<td>Initial Stiffness (initStif)</td>
<td>0 0 0</td>
</tr>
<tr>
<td></td>
<td>0 0 0</td>
</tr>
<tr>
<td>Optional Arguments</td>
<td></td>
</tr>
<tr>
<td>I-Modification (IMod)</td>
<td>no</td>
</tr>
<tr>
<td>Is Copy (isCopy)</td>
<td>no</td>
</tr>
<tr>
<td>Mass Density 1 (massDens1)</td>
<td>0</td>
</tr>
<tr>
<td>Mass Density 2 (massDens2)</td>
<td>0</td>
</tr>
</tbody>
</table>

Diagram showing controlled displacements and acquired forces for the ChevronBrace element.
Define ExpElement: ChevronBrace
Post-Processing: Output

First:
Load OpenSees Results into Matlab
Post-Processing: Output

Now these are enabled
Plot Deformed Shape

- Time Step
- AnalysisCase
- Recorder
- Order
- Magnification
Plot Element Forces: Axial Forces
Plot Mode Shape: 1\textsuperscript{st} Mode
Plot Response

[Image of a plot response diagram with various labels and options for analysis case, parameter, node/element, DOF, recorder, parameter, node, element, DOF, DefoShape, Dsp, 16, Sec, 1, LA22YY, Time = 0.000]
Plot Response Spectra

- X-Axis: FirstFloorSpectra - T - Damping - Linear
- Y-Axis: FirstFloorSpectra - psdAcc - 0.05 - Linear

Spectra Name - Response - Damping - Axis Scale
AISC Design Toolbox

**Database Inquiries:**
- Show Available Sections
- Find Section Properties
- Find Matching Sections

**Design Checks:**
- Bending Capacity
- Compression Capacity
- Shear Capacity
- PMM Interaction
AI SC Toolbox: Find Section Prop.

### Find AISC Section Properties

**Section Shape:**
- W24x68

**Section Parameters:**
- Area - A
- Depth - d
- Width - bf
- Thickness of the web (W,M,S only) - tw
- Thickness of the flange (W,M,S only) - tf
- Moment of inertia - Ix

### Output

The requested parameters are:
- name = W24x68
- shape = W
- A = 20.1
- d = 23.7
- Ix = 1830
- Iy = 70.4

[OK]
AI SC Toolbox: Find Matching Sec.

Output:

There are a total of 19 sections available:

S3X7.5
S4X7.7
S5X8.5
S6X9
S4X9.5
S5X10
S6X10
S6X12
S5X12.5
S4X13
S5X13
S6X15
S5X16
S6X16
S6X17.25
S8X19.4
S5X18.9
S5X19
S6X20
AISC Toolbox: Bending Capacity

Bending Capacity of AISC Section

- Section Shape: W24 x 68
- Unbraced Length (Lb): 40 [in]
- Bending Coefficient (C_b): 1 [-]
- Yield Stress (F_y): 50 [ksi]
- Modulus of Elasticity (E): 29000 [ksi]
- Direction: strong

Note: The AISC Bending Capacity check is only applied to AISC rolled WIS/MHSS sections.

Output

Bending capacity for section W24x68:
With Lb = 120 in
C_b = 1
F_y = 50 ksi
e = 29000 ksi
phi = 0.9
M_p = 8680
Mr = 8160
L_p = 79.2626
L_r = 208.7244
Flange_Compactness = Compact
Web_Compactness = Compact
Capacity = 7203.19
FailureMode = Lateral torsional buckling
AISC Toolbox: Compression Capacity

**AISC Compression Capacity**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section Shape</td>
<td>W14x88</td>
</tr>
<tr>
<td>Effective Length (kLx)</td>
<td>144 [in]</td>
</tr>
<tr>
<td>Effective Length (kLx)</td>
<td>144 [in]</td>
</tr>
<tr>
<td>Yield Stress (Fy)</td>
<td>50 [ksi]</td>
</tr>
<tr>
<td>Modulus of Elasticity (E)</td>
<td>29000 [ksi]</td>
</tr>
</tbody>
</table>

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

**Output**

Compression capacity for section W14x88:
- With kLx = 144 in
- kLy = 144 in
- Fy = 50 ksi
- E = 29000 ksi

Section Slenderness = None Slender
phi = 0.85
Failure Mode = Inelastic buckling (Qs(flange) = 1, Qs(web) = 1)
Capacity = 661.6242
### AISC Toolbox: Shear Capacity

**Shear Capacity of AISC Section**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section Shape</td>
<td>W24x68</td>
<td></td>
</tr>
<tr>
<td>Distance between Stiffeners</td>
<td>24</td>
<td>[in]</td>
</tr>
<tr>
<td>Yield Stress (Fy)</td>
<td>50</td>
<td>[ksi]</td>
</tr>
<tr>
<td>Modulus of Elasticity (E)</td>
<td>29000</td>
<td>[ksi]</td>
</tr>
</tbody>
</table>

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

**Output**

```
The Shear Capacity parameters are:
T_tw, p = 78.2264
T_tw, c = 91.4274
T_tw = 49.8795
phi = 0.9
Capacity = 265.5565
FailureMode = Reaching yielding capacity 0.6*fy
```

OK
### AISC Toolbox: PMM Interaction

#### AISC PMM Interaction Check

<table>
<thead>
<tr>
<th>Section Shape</th>
<th>W24x68</th>
<th><strong>Calculate</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield Stress (Fy)</td>
<td>50</td>
<td>[ksi]</td>
</tr>
<tr>
<td>Modulus of Elasticity (E)</td>
<td>29000</td>
<td>[ksi]</td>
</tr>
</tbody>
</table>

#### 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.
OpenSees Navigator provides:

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

Thanks for your interest in OpenSees Navigator. This program is intended to be self-explanatory, nevertheless a basic user manual will be added to the website shortly. We are very happy to have the opportunity to distribute this software for OpenSees Navigator users. We encourage everyone to try out all of the functions of the program and send us criticism, corrections or suggestions to improve future versions. We also encourage users to e-mail us at either andreas.schellenberg@gmail.com or yangtony2004x@gmail.com so that we can add the e-mail addresses to the OpenSees Navigator user list. We will use such list to contact everyone about new releases or major updates. We will try our best to improve the next release.

Thank you.

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

Andreas Schellenberg & Tony Yang
**OpenSees Navigator**

<table>
<thead>
<tr>
<th>Home</th>
<th>Introduction</th>
<th>Manuals</th>
<th>Tutorials</th>
<th>Presentations</th>
<th>Discussion</th>
<th>Updates</th>
<th>Downloads</th>
<th>Links</th>
</tr>
</thead>
</table>

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

OpenSees Navigator 2.0 is available at
http://peer.berkeley.edu/OpenSeeNavigator

Andreas Schellenberg:
andreas.schellenberg@gmail.com

Tony Yang:
yangtony2004@gmail.com