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.
- 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

Release 2.4.2, December 2007

Developed by
Andreas Schellenshagen
Dep. of Civil and Env. Eng.
University of California, Berkeley

Tony Yong, 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.
Define geometry: new model template

- Stick Model
- Beam Model
- EBF Model
- Zipper Frame
- Inverted-V Braced Frame
- Moment Frame
- Single Area Mesh
Define geometry: Zipper braced frame

<table>
<thead>
<tr>
<th>Dimension (ndm)</th>
<th>2d</th>
</tr>
</thead>
<tbody>
<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>
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

<table>
<thead>
<tr>
<th>Dimension (ndm)</th>
<th>2d</th>
</tr>
</thead>
<tbody>
<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: display
View geometry: set display options

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

Element:
- Tags
- Types
- GeoTrans
- Local Axes
- Zero Length
- Loads/Deformations

General:
- Model
- Global Axes
- Grid Lines
View geometry: display
Edit geometry

Node:
- Add
- Delete
- Move

Element:
- Add
- Delete
- Divide/Join
- Add/Delete ZeroLength
Define material: uniaxial materials

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

Material properties:
- $F_y = 50$ ksi
- $E = 29000$ ksi
- $b = 0.05$
Define uniaxial material: Steel01

<table>
<thead>
<tr>
<th>Material Name</th>
<th>50</th>
</tr>
</thead>
<tbody>
<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>
</tbody>
</table>

Optional Parameters:
- 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
Define material: uniaxial materials

- Add Material: BoucWen
- Modify/Show Material: ElasticDefault
- Delete Material: ElasticDefault, A50
Define uniaxial material: Steel01

<table>
<thead>
<tr>
<th>Material Name</th>
<th>A50</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield Stress (F_y)</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>
</tbody>
</table>

Optional Parameters:

- 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
Define material: uniaxial materials
Define material: uniaxial materials
Define material: nD materials

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

Under development:
- NewTemplate3DElastoPlastic
- ModelsLargeDeformation
Define section: line sections

Templates:
- Aggregator
- Elastic
- Fiber
- Uniaxial
Define line section: elastic section

If the model is 3D
Define fiber section: Composite patch
Define line section: fiber section

<table>
<thead>
<tr>
<th>Define Fiber Section</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Section Name</strong></td>
<td><strong>Add</strong></td>
</tr>
<tr>
<td><strong>Add Fiber</strong></td>
<td>Fiber</td>
</tr>
<tr>
<td><strong>Modify Fiber</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Delete Fiber</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Add Patch</strong></td>
<td>Quadrilateral</td>
</tr>
<tr>
<td><strong>Modify Patch</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Delete Patch</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Add Layer</strong></td>
<td>Straight</td>
</tr>
<tr>
<td><strong>Modify Layer</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Delete Layer</strong></td>
<td></td>
</tr>
</tbody>
</table>
Define line section: quadrilateral patch

<table>
<thead>
<tr>
<th>Patch Name</th>
<th>CoreConcrete</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material Type</td>
<td>ConfinedConcrete</td>
</tr>
<tr>
<td>Lower Left Corner</td>
<td>ConfinedConcrete</td>
</tr>
<tr>
<td>(yL,zL)</td>
<td>[0 0]</td>
</tr>
<tr>
<td>Lower Right Corner</td>
<td>UnconfinedConcrete</td>
</tr>
<tr>
<td>(yJ,zJ)</td>
<td>[0 0]</td>
</tr>
<tr>
<td>Upper Right Corner</td>
<td>[0 0]</td>
</tr>
<tr>
<td>(yK,zK)</td>
<td></td>
</tr>
<tr>
<td>Upper Left Corner</td>
<td>[0 0]</td>
</tr>
<tr>
<td>(yL,zL)</td>
<td></td>
</tr>
<tr>
<td>Number of Fibers in</td>
<td>1</td>
</tr>
<tr>
<td>I-J dir (nFlJ)</td>
<td></td>
</tr>
<tr>
<td>Number of Fibers in</td>
<td>1</td>
</tr>
<tr>
<td>J-K dir (nFJK)</td>
<td></td>
</tr>
<tr>
<td>Optional Arguments</td>
<td></td>
</tr>
<tr>
<td>Counter-Clockwise</td>
<td>0.</td>
</tr>
<tr>
<td>Rot (Theta)</td>
<td></td>
</tr>
</tbody>
</table>
Define fiber section: AISC patch

<table>
<thead>
<tr>
<th>Patch Name</th>
<th>Patch01</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 along dw (nfdw)</td>
<td>10</td>
</tr>
<tr>
<td>Number of Fibers along tw (nftw)</td>
<td>1</td>
</tr>
<tr>
<td>Number of Fibers along bf (nfbf)</td>
<td>10</td>
</tr>
<tr>
<td>Number of Fibers along tf (nftf)</td>
<td>1</td>
</tr>
<tr>
<td>Optional Arguments</td>
<td></td>
</tr>
<tr>
<td>Counter-Clockwise Rot (Theta)</td>
<td>0.</td>
</tr>
</tbody>
</table>
Define section: area section

Templates:
- Bidirectional
- ElasticMembranePlate
- PlateFiber
Define element: line element

Templates:
- CorotationalTruss
- DispBeamColumn
- ElasticBeamColumn
- ForceBeamColumn
- HingeBeamColumn
- Truss
- ZeroLength
Define line element: ElasticBeamColumn

**Define ElasticBeamColumn Element**

<table>
<thead>
<tr>
<th>Element Name:</th>
<th>EColumn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modulus of Elasticity (E):</td>
<td>29000</td>
</tr>
<tr>
<td>Cross-Sectional Area (A):</td>
<td>13.3</td>
</tr>
<tr>
<td>Moment of Inertia (Iz):</td>
<td>248</td>
</tr>
</tbody>
</table>

**Select Section from Database**

<table>
<thead>
<tr>
<th>Database:</th>
<th>AISC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section Name:</td>
<td>W10X45</td>
</tr>
<tr>
<td>Direction:</td>
<td>strong</td>
</tr>
</tbody>
</table>
Define line element: ForceBeamColumn

Define ForceBeamColumn Element

<table>
<thead>
<tr>
<th>Element Name</th>
<th>1stStoryColumn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number Integration Points (NIP)</td>
<td>5</td>
</tr>
<tr>
<td>Section Type</td>
<td>1stStoryColumn</td>
</tr>
</tbody>
</table>

Optional Arguments:

<table>
<thead>
<tr>
<th>Mass Density (massDens)</th>
<th>0.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum Iterations (maxIters)</td>
<td>10</td>
</tr>
<tr>
<td>Tolerance (tol)</td>
<td>1E-8</td>
</tr>
</tbody>
</table>
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:

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

Templates:
- Plain
- UniformExcitation
- MultipleSupport
Define LoadPattern: UniformExcitation

<table>
<thead>
<tr>
<th>LoadPattern Name</th>
<th>SACNF01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TimeSeries Type</td>
<td>SACNF01</td>
</tr>
<tr>
<td>Direction of Excitation (dir)</td>
<td>1</td>
</tr>
</tbody>
</table>
Define recorder

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

Defaults:
- DefoShape
- EigenVector

load-pattern(2) with name "SACNF01" has been defined/modified successfully
Define recorder: node recorder
Define recorder: BeamColumn recorder
Define analysis options

- StaticDefault
- TransientDefault
- EigenDefault
Define analysis options: new analysis

**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
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

Assign Element Types
- Assign Element Types: [Assign, Display]
- Element Number(s): 17:24
- Element Type: Brace

Assign Element Geometric Transformations
- Assign Geometric Transformations: [Assign, Display]
- Element Number(s): 17:24
- Geometric Transformation: Corotational
Display assigned properties

Nodal Masses

Element Types
Define analysis case

**Defaults:**
- StaticDefaultCase
- EigenDefaultCase
Define analysis case: new analysis case

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

- **expControl**
  - Interfaces to the different control and data acquisition.

- **expSetup**
  - Transforms between the experimental element degrees of freedom and the actuator degrees of freedom (linear vs. non-linear transformations).

- **expSite**
  - Stores data and provides communication methods for distributed testing.

- **expElement**
  - Represents the part of the structure that is physically tested and provides the interface between the FE-software and the experimental software framework.
Define expControl:
Define expControl: MTSCsi
Define expSetup:

Templates:
- InvertedVBrace
- InvertedVBraceJntOff
- Aggregator
- NoTransformation
- OneActuator
- TwoActuators
- ThreeActuators
- ThreeActuatorsJntOff
Define **expSetup**: *InvertedVBrace*

<table>
<thead>
<tr>
<th>Setup Name</th>
<th>BraceExpSetupJntOff</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Experimental Control Type</strong></td>
<td>BraceExpCtrlXPC</td>
</tr>
<tr>
<td><strong>Geometry Type</strong> (nlGeomFlag)</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**:

<table>
<thead>
<tr>
<th>Dsp Control Factor (dspCtrlFact)</th>
<th>[1 1 1]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vel Control Factor (velCtrlFact)</td>
<td>[1 1 1]</td>
</tr>
<tr>
<td>Acc Control Factor (accCtrlFact)</td>
<td>[1 1 1]</td>
</tr>
<tr>
<td>Dsp Daq Factor (dspDaqFact)</td>
<td>[1 1 1 1 1]</td>
</tr>
<tr>
<td>Force Daq Factor (frcDaqFact)</td>
<td>[1 1 1 1 1 1]</td>
</tr>
</tbody>
</table>
Define expSite:

Templates:
- LocalSite
- RemoteSite
- ActorSite
Define expSite: LocalSite
Define `expElement`: `InvertedVBrace`

<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 0 0 0 0 0</td>
</tr>
</tbody>
</table>

**Optional Arguments:**

- `l-Modification (iMod)`: no
- `Is Copy (isCopy)`: no
- Mass Density 1 (massDens1): 0
- Mass Density 2 (massDens2): 0

---

Acquired forces

<table>
<thead>
<tr>
<th>d1, q1</th>
<th>d2, q2</th>
<th>d3, q3</th>
</tr>
</thead>
<tbody>
<tr>
<td>d4, q4</td>
<td>d5, q5</td>
<td>d6, q6</td>
</tr>
</tbody>
</table>

Controlled displacements

<table>
<thead>
<tr>
<th>d7, q7</th>
</tr>
</thead>
</table>
Define expElement: InvertedVBrace
Run OpenSees: set OpenSees.exe path

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

Steps:
2. Write OpenSees Input Files (writes TCL files)
3. Run OpenSees
Post processing: load results

First:
Load OpenSees Results
Post processing: load results

First:
Load OpenSees Results into Matlab
Post processing: output

Now these are enabled
Post processing: plot deformed shape
Post processing: plot element forces
Post processing: plot mode shape
Post processing: plot response histories
Post processing: plot response spectra

![Response Spectra Graph]

- **Spectra Name**: FirstFloorSpectra
- **Response**: psdAcc
- **Damping**: 0.05
- **Axis Scale**: Linear
Post processing: animate response
Post processing: principal $\sigma$ and $\varepsilon$
Design: AISC design toolbox

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

Design Checks:
- Bending Capacity
- Compression Capacity
- Shear Capacity
- PMM Interaction
AISC design toolbox: section properties

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
I_w = 1820
I_y = 70.4
AISC design toolbox: matching sections

![Image of AISC design toolbox](attachment:image.png)

Select Section Shape: WSM

Add Parameter:
- **Depth**: Min: 0, Max: 0

Modify Parameter:
- **Area-A**: Min: 2, Max: 6

Delete Parameter:
- **Area-A**

Sort by Parameter:
- **Area-A**

There are a total of 19 sections available:
- W12 x 17.5
- W12 x 17.7
- W18 x 8.5
- W18 x 9
- W18 x 9.5
- W10 x 10
- W10 x 10
- W12 x 10
- W12 x 10
- W12 x 12.5
- W12 x 13
- W13 x 13
- W14 x 14
- W16 x 16
- W16 x 17.25
- W18 x 18.4
- W18 x 18.9
- W20 x 20
- W20 x 20
AISC design toolbox: bending capacity

Note: The AISC Bending Capacity check is only applied to AISC rolled W, I, and HSS sections.

Bending capacity for section W24x68:
With Lb = 120 in
Cb = 1
Fy = 50 ksi
K = 29000 ksi

phiL = 0.9
Mp = 8840
Mr = 8160
Lp = 99.2626
bL = 208.7244
Flange_Compactness = Compact
Web_Compactness = Compact
Capacity = 7283.19
FailureMode = Lateral torsional buckling
## AISC design toolbox: compression cap.

### AISC Compression Capacity

<table>
<thead>
<tr>
<th>Compression Capacity of AISC Section</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Section Shape:</strong> W14x68</td>
</tr>
<tr>
<td><strong>Effective Length (kLx):</strong> 144 [in]</td>
</tr>
<tr>
<td><strong>Effective Length (kLy):</strong> 144 [in]</td>
</tr>
<tr>
<td><strong>Yield Stress (Fy):</strong> 50 [ksi]</td>
</tr>
<tr>
<td><strong>Modulus of Elasticity (E):</strong> 29000 [ksi]</td>
</tr>
</tbody>
</table>

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

### Output

**AISC Compression Capacity**

Compression capacity for section W14x68:

- kLx = 144 in
- kLy = 144 in
- Fy = 50 ksi
- E = 29000 ksi

- Section Slenderness = None Slender
- phi = 0.76
- Failure Mode = Inelastic buckling (Qs(flange) = 1, Qa(web) = 1)
- Capacity = 661.6242

**OK**
AISC design toolbox: shear capacity

Shear Capacity of AISC Section

Section Shape: W24x68
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 WS/SM sections.

The shear capacity parameters are:
Ttw_p = 78.2284
Ttw_r = 91.4274
Ttw = 49.8795
phi1 = 0.9
Capacity = 265.5585
Failure Mode = Reaching yielding capacity 0.6\*Fy

OK
AISC design toolbox: PMM interaction

### PMM Interaction Check of AISC Section

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section Shape</td>
<td>W24x68</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>
<tr>
<td>Applied Axial Force (Pu)</td>
<td></td>
</tr>
<tr>
<td>Applied Moment about X axis (Mux)</td>
<td></td>
</tr>
<tr>
<td>Applied Moment about Y axis (Muy)</td>
<td></td>
</tr>
<tr>
<td>Effective Length (kLx)</td>
<td></td>
</tr>
<tr>
<td>Effective Length (kLy)</td>
<td></td>
</tr>
<tr>
<td>Unbraced Length (Lb)</td>
<td></td>
</tr>
<tr>
<td>Bending Coefficient (Cb)</td>
<td>1 [-]</td>
</tr>
</tbody>
</table>

**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.
Dear OpenSees Navigator users,

Thanks for your interest in OpenSees Navigator. This program is intended to be self-explanatory, nevertheless a basic user manual will be added to the website shortly. We are very happy to have the opportunity to distribute this software for OpenSees Navigator users. We encourage everyone to try out all of the functions of the program and send us criticism, corrections or suggestions to improve future versions. We also encourage users to e-mail us at either andreas.schellenberg@gmail.com or yangtony2004@gmail.com so that we can add the e-mail addresses to the OpenSees Navigator user list. We will use such list to contact everyone about new releases or major updates. We will try our best to improve the next release.

Thank you

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

Andreas Schellenberg & Tony Yang
**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 2.4.2 is available at
http://peer.berkeley.edu/OpenSeesNavigator

Contacts:
Dr. Eiji Kohama: kohama-e83ab@pa.ktr.mlit.go.jp
Dr. Andreas Schellenberg: andreas.schellenberg@gmail.com
Dr. Tony Yang: yangtony2004@gmail.com