OpenSees Navigator

Andreas Schellenberg
Research Engineer
Pacific Earthquake Engineering Research Center

Tony Yang
Assistant Professor
Department of Civil Engineering
University of British Columbia, Vancouver, Canada

Eiji Kohama
Head of Group
Earthquake and Structural Dynamics Group
Port and Airport Research Institute, Japan
Introduction

- MATLAB based graphical user interface (GUI).
- Pre- and post-processing for OpenSees and OpenFresco.
- Design toolboxes: NSP, PBEE, AISC design checks, AISC database, response spectra for linear and bilinear systems and signal filtering.
- Both MATLAB Pcode and self-executable versions are available for Windows & Mac.
- Being used by researchers from Asia, US, Canada, South America and Europe.
Motivations

Replace the TCL text input with graphical input.

Most researchers use MATLAB to do the post-processing, and MATLAB/Simulink is the typical framework for implementing hybrid simulation tests.

OpenSees Navigator will create the OpenSees (analytical/hybrid) model and graphically display the results before, during or after a test.

Provides many robust plotting algorithms and is very effective in generating the plots for engineering applications.

Flexible to use and requires no programming skill.
OpenSees Navigator

Release 2.5.2, April 2013

Developed by
Andreas Schellenberg, Ph.D., P.E.
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.
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>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: display
Conversion: SAP2000 to OpenSees

- Export SAP2000 model to .s2k file
- Start OpenSees Navigator program
- Select “Open Model from File...” and choose the earlier exported .s2k file
- There is no one-to-one relationship between SAP2000 and OpenSees objects, therefore
- Carefully check the converted model
- Element loads and nodal constraints are currently not converted yet
Export to s2k file
Export to s2k file

Choose Tables for Export to Text File

- MODEL DEFINITION (65 of 65 tables selected)
  - System Data
  - Property Definitions
  - Load Pattern Definitions
  - Other Definitions
  - Load Case Definitions
  - Bridge Data
  - Connectivity Data
  - Joint Assignments
  - Frame Assignments
  - Solid Assignments
  - Link Assignments
  - Options/Preferences Data
  - Miscellaneous Data

Load Patterns [Model Def.]
- Select Load Patterns...
- 1 of 1 Selected

Options
- Selection Only
- Open File After Export
  - Use Text Editor
  - Use Microsoft Word
- Expose All Input Tables

Named Sets
- Save Named Set...
- Show Named Set...
- Delete Named Set...

OK  Cancel

nees@berkeley
Define geometry: import from SAP2000
Define geometry: import from SAP2000
Define geometry: import from SAP2000

XY plane

Flip through plan views
Define geometry: import from SAP2000

XZ plane
Flip through elevation views
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
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
Define uniaxial material: Steel01

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

- **Material Name**: A50
- **Yield Stress (Fy)**: 50
- **Modulus of Elasticity (E)**: 29000
- **Hardening Ratio (b)**: 0.05

**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: nD materials

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

![Define Fiber Section dialog box](image-url)
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 (yL,zL)</td>
<td>ConfinedConcrete</td>
</tr>
<tr>
<td>Lower Right Corner (yJ,zJ)</td>
<td>UnconfinedConcrete</td>
</tr>
<tr>
<td>Upper Right Corner (yK,zK)</td>
<td>[0 0]</td>
</tr>
<tr>
<td>Upper Left Corner (yL,zL)</td>
<td>[0 0]</td>
</tr>
<tr>
<td>Number of Fibers in I-J dir (nflJ)</td>
<td>1</td>
</tr>
<tr>
<td>Number of Fibers in J-K dir (nfJK)</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 fiber section: 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

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

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 line element: ForceBeamColumn

<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**

- Mass Density (massDens): 0.
- Maximum Iterations (maxIters): 10
- Tolerance (tol): 1E-8
Define element: area and solid elements

Templates (area):
- Quad
- BbarQuad
- EnhancedQuad
- SSPQuad
- Shell
- ShellNL

Templates (solid):
- StdBrick
- BbarBrick
- SSPBrick
Define TimeSeries:

Templates:
- Constant
- Interpolated GM
- Linear
- PathFile
- PathFileFiltered
- PathValue
- Plain GM
- Pulse
- Rectangular
- Sine
- Triangle
Define TimeSeries: PathFile

![Image of Define PathFile Time Series window]

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

![Graph of Ground-Acceleration-Time-History (SAC NFD1 (1978 Talbse))]

- **Ground Acceleration [m/sec^2]**
- **Time [sec]**
  - 0 to 30
  - 0 to 300
Plot TimeSeries

A plot showing a TimeSeries with a factor ranging from -0.4 to 0.3 over a time period from 0 to 25 seconds. The graph is labeled with "M1" and includes options for plotting, starting, stopping, and clearing. The MATLAB menu is visible at the top of the window.
Define LoadPattern:

- Plain
- UniformExcitation
- MultipleSupport
Define LoadPattern: UniformExcitation

<table>
<thead>
<tr>
<th>Define UniformExcitation Load Pattern</th>
</tr>
</thead>
<tbody>
<tr>
<td>LoadPattern Name:</td>
</tr>
<tr>
<td>TimeSeries Type:</td>
</tr>
<tr>
<td>Direction of Excitation (dir):</td>
</tr>
</tbody>
</table>
Define recorder

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

Defaults:
- DefoShape
- Reactions
- EigenVector
Define recorder: node recorder

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

![BeamColumn Element Recorder Window](image.png)

<table>
<thead>
<tr>
<th>Recorder Name</th>
<th>ElemForces</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Number(s)</td>
<td>all</td>
</tr>
<tr>
<td>Arguments</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Global Resisting Forces</td>
</tr>
<tr>
<td></td>
<td>Local Resisting Forces</td>
</tr>
<tr>
<td>Section Response</td>
<td>Section</td>
</tr>
<tr>
<td>Section Number(s)</td>
<td>15</td>
</tr>
<tr>
<td>Arguments</td>
<td>Forces, Deformations</td>
</tr>
<tr>
<td>Fiber Response</td>
<td>Fiber</td>
</tr>
<tr>
<td>Arguments</td>
<td>Stress/Strain, Y-Coor, Z-Coor</td>
</tr>
</tbody>
</table>
Define analysis options

Defaults:
- StaticDefault
- TransientDefault
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
- Imposed Motions

Element:
- Types
- GeoTrans
- Rotations
- Local Axes
- Loads
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:
  - Select

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 New Damping Parameters

Damping Parameter Set Name: DampingParam01
Region Defined by: Nodes
Node or Element Number(s): all
Mass Prop. Damping (alphaM): 1.4218
Kcurr Prop. Damping (betaK): 0.
Kinit Prop. Damping (betaKinit): 0.
Kcom Prop. Damping (betaKcom): 0.001728

Graph showing damping parameters vs frequency.
User Defined Analysis Script

```bash
# set the test parameters
set testType NormDispIncr
set testTol 1.0e-8;
set testIter 28;
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 < $tFinal) {
  if (round($tCurrent,1) < 1.0e-15) {
    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

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

Select Analysis Case(s):
- EigenDefaultCase
- SACHF01Case01

Recorder(5) with name "EigenVector" has been defined/modified successfully.
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 Plot]

- **Spectra Name:** FirstFloorSpectra
- **Response:** psdAcc
- **Damping:** 0.05
- **Axis Scale:** Linear
Post processing: animate response
Post processing: animate response
Post processing: animate response
Post processing: principal stress and strain
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
b = 23.7
tw = 1.060
Iy = 79.4
AISC design toolbox: matching sections
AISC design toolbox: bending capacity

Bending Capacity of AISC Section

| Section Shape : | W24x68 |
| Unbraced Length (Lb) : | 40 [in] |
| Bending Coefficient (Cb) : | 1 [-] |
| Yield Stress (Fy) : | 50 [ksi] |
| Modulus of Elasticity (E) : | 290000 [ksi] |
| Direction : | strong |

Note: The AISC Bending Capacity check is only applied to AISC rolled W/IM/HSS sections.

Bending capacity for section W24x68:
With Lb = 120 in
Cb = 1
Fy = 50 ksi
E = 290000 ksi

phiL = 0.9
Mp = 8840
Mr = 8160
Lp = 79.2626
Dr = 208.7244
Flange_Compactness = Compact
Web_Compactness = Compact
Capacity = 7203.19
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>Section Shape : W14x68</td>
</tr>
<tr>
<td>Effective Length (kLx) : 144 [in]</td>
</tr>
<tr>
<td>Effective Length (kLy) : 144 [in]</td>
</tr>
<tr>
<td>Yield Stress (Fy) : 50 [ksi]</td>
</tr>
<tr>
<td>Modulus of Elasticity (E) : 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:
with kLx = 144 in
  kLy = 144 in
  Fy = 50 ksi
  E = 29000 ksi
Section Slenderness = None slender
phi = 0.75
Failure Mode = Inelastic buckling (Qs(flange) = 1, Qa(web) = 1)
Capacity = 661.6242

OK
AISC design toolbox: shear capacity

**Shear Capacity of AISC Section**

<table>
<thead>
<tr>
<th>Section Shape</th>
<th>W24x68</th>
</tr>
</thead>
<tbody>
<tr>
<td>Distance between Stiffeners (a)</td>
<td>24 [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 Shear Capacity check is only applied to AISC rolled W/S/M sections.

**Output**

The Shear Capacity parameters are:

- Tw_p = 78.2264
- Tw_t = 91.4074
- Tw = 49.8995
- phi = 0.9
- Capacity = 265.5585
- Failure Mode = Reaching yielding capacity 0.6*Fy
# AISC design toolbox: PMM interaction

### AISC PMM Interaction Check

**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><strong>Demand</strong></td>
<td></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><strong>Compression</strong></td>
<td></td>
</tr>
<tr>
<td>Effective Length (kLx)</td>
<td></td>
</tr>
<tr>
<td>Effective Length (kLy)</td>
<td></td>
</tr>
<tr>
<td><strong>Bending</strong></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 (32 bit and 64 bit) and self-executable versions for Windows & Mac 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.