STRUCTURAL ANALYSIS FOR PERFORMANCE-BASED EARTHQUAKE ENGINEERING
Structural Analysis for Performance-Based Earthquake Engineering

• Basic modeling concepts
• Nonlinear static pushover analysis
• Nonlinear dynamic response history analysis
• Incremental nonlinear dynamic analysis
• Probabilistic approaches
Disclaimer

• The “design” ground motion cannot be predicted.

• Even if the motion can be predicted it is unlikely than we can precisely predict the response. This is due to the rather long list of things we do not know and can not do, as well as uncertainties in the things we do know and can do.

• The best we can hope for is to predict the characteristics of the ground motion and the characteristics of the response.
How to Compute Performance-Based Deformation Demands?

Increasing Value of Information

- × Linear Static Analysis
- × Linear Dynamic Modal Response Spectrum Analysis
- × Linear Dynamic Modal Response History Analysis
- × Linear Dynamic Explicit Response History Analysis

- ✔ Nonlinear Static “Pushover” Analysis
- ✔ Nonlinear Dynamic Explicit Response History Analysis

× = Not Reliable in Predicting Damage
# FEMA 368 Analysis Requirements (SDC D, E, F)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Regular Structures</strong></td>
<td>YES</td>
<td>YES</td>
<td>YES</td>
<td>YES</td>
</tr>
<tr>
<td>$T \leq T_s$</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Plan Irreg. 2,3,4,5</td>
<td>YES</td>
<td>YES</td>
<td>YES</td>
<td>YES</td>
</tr>
<tr>
<td>Vert. Irreg. 4, 5</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Plan Irreg. 1a,1b</td>
<td>NO</td>
<td>YES</td>
<td>YES</td>
<td>YES</td>
</tr>
<tr>
<td>Vert. Irreg. 1a, 1b</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2, or 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>All Other Structures</td>
<td>NO</td>
<td>YES</td>
<td>YES</td>
<td>YES</td>
</tr>
</tbody>
</table>

Nonlinear Static Analysis Limitations not Stated
<table>
<thead>
<tr>
<th>Condition</th>
<th>Regular</th>
<th>Irregular</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T \leq T_s$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Strong Column</td>
<td>YES</td>
<td>YES</td>
</tr>
<tr>
<td>Weak Column</td>
<td>NO</td>
<td>NO</td>
</tr>
<tr>
<td>Any Condition</td>
<td>NO</td>
<td>NO</td>
</tr>
<tr>
<td>$T &gt; T_s$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Strong Column</td>
<td>NO</td>
<td>YES</td>
</tr>
<tr>
<td>Weak Column</td>
<td>NO</td>
<td>NO</td>
</tr>
<tr>
<td>Any Condition</td>
<td>NO</td>
<td>NO</td>
</tr>
</tbody>
</table>
Definition for “Elements” and “Components”

Primary elements or components are critical to the building's ability to resist collapse.
Basic Modeling Concepts

In general, a model should include the following:

• Soil-Structure-Foundation System
• Structural (Primary) Components and Elements
• Nonstructural (Secondary) Components and Elements
• Mechanical Systems (if performance of such systems is being assessed)
• Reasonable Distribution and Sequencing of gravity loads
• P-Delta (Second Order) Effects
• Reasonable Representation of Inherent Damping
• Realistic Representation of Inelastic Behavior
• Realistic Representation of Ground Shaking
Basic Modeling Concepts

• In general, a three-dimensional model is necessary. However, due to limitations in available software, 3-D inelastic time history analysis is still not practical (except for very special and important structures).

• In this course we will concentrate on 2-D analysis.

• We will use the computer program NONLIN-Pro which is on the course CD. Note that the analysis engine behind NONLIN-Pro is DRAIN-2Dx.

• DRAIN-2Dx is old technology, but it represents the basic state of the practice. The state of the art is being advanced through initiatives such as PEER’s OpenSees Environment.
Steps in Performing Nonlinear Response History Analysis (1)

1) Develop Linear Elastic Model, *without P-Delta Effects*
   a) Mode Shapes and Frequencies (Animate!)
   b) Independent Gravity Load Analysis
   c) Independent Lateral Load Analysis

2) Repeat Analysis (1) but *include P-Delta Effects*

3) Revise model to include Inelastic Effects. *Disable P-Delta.*
   a) Mode Shapes and Frequencies (Animate!)
   b) Independent Gravity Load Analysis
   c) Independent Lateral Load (Pushover) Analysis
   d) Gravity Load followed by Lateral Load
   e) Check effect of variable load step

4) Repeat Analysis (3) but *include P-Delta Effects*
Steps in Performing Nonlinear Response History Analysis (2)

5) Run Linear Response History Analysis, *disable P-Delta*
   a) Harmonic Pulse followed by Free Vibration
   b) Full Ground Motion
   c) Check effect of variable time step

6) Repeat Analysis (5) but *include P-Delta Effects*

7) Run Nonlinear Response History Analysis, *disable P-Delta*
   a) Harmonic Pulse followed by Free Vibration
   b) Full Ground Motion
   c) Check effect of variable time step

8) Repeat Analysis (7) but *include P-Delta Effects*
Basic Component Model Types

*Phenomenological*

All of the inelastic behavior in the yielding region of the component is “lumped” into a single location. Rules are typically required to model axial-flexural interaction.

Very large structures may be modeled using this approach. Nonlinear dynamic analysis is practical for most 2D structures, but may be too computationally expensive for 3D structures.
Phenomenological Model

Actual

Model

Lumped Plastic Hinge

Hinge
Hysteretic Behavior

$\theta$

$M$

$i$

$j$
Basic Component Model Types

**Macroscopic**
The yielding regions of the component are highly discretized and inelastic behavior is represented at the material level. Axial-flexural interaction is handled automatically.

These models are reasonably accurate, but are very computationally expensive. Pushover analysis may be practical for some 2D structures, but nonlinear dynamic time history analysis is not currently feasible for large 2D structures or for 3D structures.
Rule-Based Hysteretic Models and Backbone Curves (1)

Simple Yielding (Robust)  
(Ductile) Loss of Strength
Rule-Based Hysteretic Models and Backbone Curves (2)

Loss of Stiffness

Loss of Strength and Stiffness
Rule-Based Hysteretic Models and Backbone Curves (3)

Pinched

Buckling
Sivaselvan and Reinhorn Models in NONLIN (MDOF MODEL)
Parametric Models, e.g., SAP2000

\[ F = \beta k D + (1 - \beta) F_y Z \]

\[ \dot{Z} = \frac{k}{F_y} \begin{cases} \dot{D} (1 - |Z|^\alpha) & \text{if } \dot{D}Z > 0 \\ \dot{D} & \text{otherwise} \end{cases} \]

Degrading Stiffness, Degrading Strength, and Pinching Models also available. See Sivaselvan and Reinhorn for Details.
The **NONLIN-Pro**
Structural Analysis Program

- A Pre-and Post-Processing Environment for DRAIN 2Dx
- Developed by Advanced Structural Concepts, Inc., of Blacksburg, Virginia
- Formerly Marketed as RAM XLINEA
- Provided at no cost to MBDSI Participants
- May soon be placed in the Public Domain through NISEE.
The *DRAIN-2DX*
Structural Analysis Program

- Developed at U.C. Berkeley under direction of Graham H. Powell
- A full User’s Manual for DRAIN may be found on the course CD, as well as in the *Nonlin-Pro* online Help System.
- FORTAN Source Code for the version of DRAIN incorporated into *Nonlin-Pro* is available upon request
DRAIN-2DX Capabilities/Limitations

• Structures may be modeled in TWO DIMENSIONS ONLY. Some 3D effects may be simulated if torsional response is not involved.

• Analysis Capabilities Include:
  • Linear Static
  • Mode Shapes and Frequencies
  • Linear Dynamic Response Spectrum*
  • Linear Dynamic Response History
  • Nonlinear Static: Event-to-Event (Pushover)
  • Nonlinear Dynamic Response History

* Not fully supported by Nonlin-Pro
DRAIN-2DX Capabilities/Limitations

• Small Displacement Formulation Only
• P-Delta Effects included on an element basis using linearized formulation
• System Damping is Mass and Stiffness Proportional
• Linear Viscous Dampers may be (indirectly) modeled using stiffness Proportional Damping
• Response-History analysis uses Newmark constant average acceleration scheme
• Automatic time-stepping with energy-based error tolerance is provided
DRAIN-2DX Element Library

TYPE 1: Truss Bar
TYPE 2: Beam-Column
TYPE 3: Degrading Stiffness Beam-Column*
TYPE 4: Zero Length Connector
TYPE 6: Elastic Panel
TYPE 9: Compression/Tension Link
TYPE 15: Fiber Beam-Column*

* Not fully supported by Nonlin-Pro
DRAIN 2Dx Truss Bar Element

• Axial Force Only

• Simple Bilinear Yield in Tension or Compression

• Elastic Buckling in Compression

• Linearized Geometric Stiffness

• May act as linear viscous damper (some trickery required)
DRAIN 2Dx Beam-Column Element

- Two Component Formulation
- Simple Bilinear Yield in Positive or Negative Moment. Axial yield is NOT provided.
- Simple Axial-Flexural Interaction
- Linearized Geometric Stiffness
- Nonprismatic properties and shear deformation possible
- Rigid End Zones Possible
DRAIN 2Dx Beam-Column Element
Axial-Flexural Interaction

Note: Diagram is for steel sections. NOo interaction and reinforced concrete type interaction is also possible
DRAIN 2Dx Beam-Column Element
NO Axial-Flexural Interaction

Axial Force

Bending Moment

Load Path
DRAIN 2Dx Beam-Column Element
Axial-Flexural Interaction

Note: This Model is not known for its accuracy or reliability. Improved models based on plasticity theory have been developed. See, for example, The RAM-Perform Program.
DRAIN 2Dx
Connection Element

• Zero Length Element

• Translational or Rotational Behavior

• Variety of Inelastic Behavior, including:
  Bilinear yielding with inelastic unloading
  Bilinear yielding with elastic unloading
  Inelastic unloading with gap

• May be used to model linear viscous dampers
Using a Connection Element to Model a Rotational Spring

- Nodes $i$ and $j$ have identical $X$ and $Y$ coordinates. The pair of nodes is referred to as a “compound node.”

- Node $j$ has $X$ and $Y$ displacements slaved to those of node $i$.

- A rotational connection element is placed “between” nodes $i$ and $j$.

- Connection element resists relative rotation between nodes $i$ and $j$.

- NEVER use Beta Damping unless you are explicitly modeling a damper.
Uses of Compound Nodes

Girder Plastic Hinges

Panel Zone region of Beam-Column Joint
Development of Girder Hinge Model

All Inelastic Behavior is in Hinge

\[ M \phi \theta \]

Moment

DRAIN-2Dx

Very Large Initial Stiffness

Ram Perform

Rotation
Girder and Joint Modeling in NONLIN-Pro

Krawinkler Joint Model

Girder Plastic Hinge
The OpenSees Computational Environment

Welcome and Register!

Welcome to the OpenSees website, a software framework for developing applications to simulate the performance of structural and geotechnical systems subjected to earthquakes.

The goal of the OpenSees development is to improve the modeling and computational simulation in engineering through open-source development. We ask all users and developers to register at the OpenSees Registration Center.

OpenSees is under continued development, so users and developers should expect changes and updates on a regular basis. In this sense, all users are developers as this is important to register. More information on OpenSees is available.

The development and application of OpenSees is sponsored by the Pacific Earthquake Engineering Research Center through the National Science Foundation's engineering and education center program.

2002 User and Developer Workshops

A User Workshop will be held on September 4, 2002, and a Developer Workshop will be held on September 5, 2002. Both of these will be held at the PEER Center. For those unable to attend the workshop, the materials presented will be made available.

Version 1.3 is Available

We have recently completed an upgrade to Version 1.3 of OpenSees. The new version is available at the OpenSees Download Center.

In Version 1.3 both the memory management and computational efficiency of OpenSees have been improved. In addition, new line search features have been added for the Newton algorithm and the option to use relative instead of absolute norms have been added to the convergence tests. A new strain-rate-based yield element and the ability to compute the standard deviations have also been added.

Version 1.3 adds new features such as mixed and enhanced yield elements, an enhanced brick element, a new strain-energy algorithm (SGRA), a system, and a new interface. Version 1.2 also allows users to more control the stepping level to control the analysis.

Version 1.1 introduces a new line search algorithm, additional support for evaluating and 3D plasticity model, a bi-directional material (suitable for modeling concrete inclusion beams) and more general zero-length (fem) elements, and the documentation has been improved.
What is *OpenSees*?

- OpenSees is a multi-disciplinary open source structural analysis program.
- Created as part of the Pacific Earthquake Engineering Research (PEER) center.
- The goal of OpenSees is to improve modeling and computational simulation in earthquake engineering through open-source development.
**OpenSees Program Layout**

- OpenSees is an object oriented framework for finite element analysis
- OpenSees consists of 4 modules for performing analyses:

![Diagram showing the program layout of OpenSees](image)
OpenSees Modules

- **Modelbuilder** - Performs the creation of the finite element model
- **Analysis** – Specifies the analysis procedure to perform on the model
- **Recorder** – Allows the selection of user-defined quantities to be recorded during the analysis
- **Domain** – Stores objects created by the Modelbuilder and provides access for the Analysis and Recorder modules
OpenSees Element Types

- **Elements**
  - Truss elements
  - Elastic beam-column
  - Zero-length elements
  - Brick elements
  - Corotational truss
  - Nonlinear beam-column
  - Quadrilateral elements

- **Sections**
  - Elastic section
  - Fiber section
  - Plate fiber section
  - Elastic membrane plate section
  - Uniaxial section
  - Section aggregator
  - Bidirectional section
## OpenSees Material Properties

<table>
<thead>
<tr>
<th>Uniaxial Materials</th>
<th>Elastic</th>
<th>Elastic perfectly plastic</th>
<th>Parallel gap</th>
<th>Elastic perfectly plastic</th>
</tr>
</thead>
<tbody>
<tr>
<td>Series</td>
<td>Steel01</td>
<td>Concrete01</td>
<td>Hysteretic</td>
<td>Elastic-No tension</td>
</tr>
<tr>
<td>Viscous</td>
<td></td>
<td></td>
<td></td>
<td>Fedelas</td>
</tr>
</tbody>
</table>
OpenSees Analysis Types

- **Loads**: Variable time series available with plain, uniform, or multiple support patterns
- **Analyses**: Static, transient, or variable-transient
- **Systems of Equations**: Formed using banded, profile, or sparse routines
- **Algorithms**: Solve the SOE using linear, Newtonian, BFGS, or Broyden algorithms
- **Recording**: Write the response of nodes or elements (displacements, envelopes) to a user-defined set of files for evaluation
OpenSees Applications

- Structural modeling in 2 or 3D, including linear and nonlinear damping, hysteretic modeling, and degrading stiffness elements
- Advanced finite element modeling
- Potentially useful for advanced earthquake analysis, such as nonlinear time histories and incremental dynamic analysis
- Open-source code allows for increased development and application
OpenSees Disadvantages

• No fully developed pre or post processors yet available for model development and visualization
• Lack of experience in applications
• Code is under development and still being fine-tuned.
OpenSees Information Sources

• The program and source code:
  http://millen.ce.berkeley.edu/

• Command index and help:

• OpenSees Homepage:
  http://opensees.berkeley.edu/OpenSees/related.html
Other Commercially Available Programs

**SAP2000/ETABS**
Both have 3D pushover capabilities and linear/nonlinear dynamic response history analysis. P-Delta and large displacement effects may be included. These are the most powerful commercial programs that are specifically tailored to analysis of buildings (ETABS) and bridges (SAP2000).

**RAM/Perform**
Currently 2D program, but a 3D version should be available soon. Developed by G. Powell, and is based on DRAIN-3D technology. Some features of program (e.g. model building) are hard-wired and not easy to override.

**ABAQUS, ADINA, ANSYS, DIANA, NASTRAN**
These are extremely powerful FEA programs but are not very practical for analysis of building and bridge structures.
Modeling Beam-Column Joint Deformation In Steel Structures
Typical Interior Subassemblage

\[ V_c \]

Doubler Plate

Continuity Plate

\[ H, \beta H, L, \alpha L, V_c H/L \]
Equilibrium in Beam-Column Joint Region

\[ F_{GF} \]

\[ F_{CF} \]

\[ \frac{V_c H}{L} \]

\[ V_c \]

\[ \alpha L \]

\[ \beta H \]
Forces and Stresses in Panel Zone

Horizontal Shear in Panel Zone:

\[ V_P = V_c \frac{(1 - \alpha - \beta)}{\beta} \]

Note: PZ shear can be 4 to 6 times the column shear

Shear Stress in Panel Zone:

\[ \tau_P = V_c \frac{(1 - \alpha - \beta)}{\alpha \beta L t_P} \]

\( t_P \) is panel zone thickness including doubler plate
Effects of High Panel Zone Stresses

• Shear deformations in the panel zone can be responsible for 30 to 40 percent of the story drift. FEMA 350’s statement that use of centerline dimensions in analysis will overestimate drift is incorrect for joints without PZ reinforcement.

• Without doubler plates, the panel zone will almost certainly yield before the girders do. Although panel zone yielding is highly ductile, it imposes high strains at the column flange welds, and may contribute to premature failure of the connection.

• Even with doubler plates, panel zones may yield. This inelastic behavior must be included in the model.
Sources of Inelastic Deformation in Typical Joint

Yielding in Column Flanges

Yielding in Panel Zone
Krawinkler Model

Panel Spring

Flange Spring

$H \quad \beta H \quad \alpha L \quad L$
Kinematics of Krawinkler Model

- Column CL Offset
- Girder CL Offset
Krawinkler Joint Model

Panel Zone Web Hinge

Simple Hinge

Rigid Bars (typical)

Simple Hinge

Panel Zone Flange Hinge
Nodes in Krawinkler Joint Model

11,12, 10, 8,9

1, 2,3

4, 5,6

7
DOF in Krawinkler Joint Model

Note: Only FOUR DOF are truly independent.
Moment-Rotation Relationships in Krawinkler Model

Moment, $M$

$M_{yP}$

$M_{yF}$

$\theta_{yP}$ $\theta_{yF}$

Hinge Rotation, $\theta$

Total

Panel Component

Flange Component
Moment-Rotation Relationships in Krawinkler Model (Alternate)

Moment, $M$

$M_{yP}$

$M_{yF}$

$K_{PK}$

$\theta_{yP}$

$\theta_{yF}$

Hinge Rotation, $\theta$
Krawinkler Model Properties
(Panel Component)

\[ M_{yP,K} = 0.6 F_y \alpha L \beta H (t_{wc} + t_d) \]

\[ K_{P,K} = G \alpha L \beta H (t_{wc} + t_d) \]

\[ \theta_{yP,K} = \frac{0.6 F_y}{G} \]
Krawinkler Model Properties (Panel Component)

$$M_y P,K = 0.6F_y \alpha L \beta H (t_{wc} + t_d)$$

Volume of Panel

$$K_{P,K} = G \alpha L \beta H (t_{wc} + t_d)$$
Krawinkler Model Properties
(Flange Component)

\[ M_{yF,K} = 1.8F_y b_{cf} t_{cf}^2 \]

\[ \theta_{yF,K} = 4\theta_{yP,K} \]
Advantages of Krawinkler Model

• Physically mimics actual panel zone distortion and thereby accurately portrays true kinematic behavior

• Corner hinge rotation is the same as panel shear distortion

• Modeling parameters are independent of structure outside of panel zone region
Disadvantages of Krawinkler Model

• Model is relatively complex

• Model does not include flexural deformations in panel zone region

• Requires 12 nodes, 12 elements, and 28 degrees of freedom

Note: Degrees of freedom can be reduced to four (4) through proper use of constraints, if available.
Scissor Joint Model

- Rigid Ends (typical)
- Panel Zone and Flange Springs
Kinematics of Scissors Model
Model Comparison: Kinematics

Krawinkler

Scissors
Mathematical Relationship Between Krawinkler and Scissors Models

\[
K_{\text{Scissors}} = \frac{K_{\text{Krawinkler}}}{(1 - \alpha - \beta)^2}
\]

\[
M_{y,\text{Scissors}} = \frac{M_{y,\text{Krawinkler}}}{(1 - \alpha - \beta)}
\]
Advantage of Scissors Model

- Relatively easy to model (compared to Krawinkler). Only 4 DOF per joint, and only two additional elements.
- Produces almost identical results as Krawinkler.

Disadvantages of Scissors Model

- Does not model true behavior in joint region.
- Does not include flexural deformations in panel zone region.
- Not applicable to structures with unequal bay width (model parameters depend on $\alpha$ and $\beta$).
Modeling Beam-Column Joint Deformation in Concrete Structures

• Accurate modeling is much more difficult (compared to structural steel) due to pullout and loss of bond of reinforcement and due to loss of stiffness and strength of concrete in the beam-column joint region.

• Physical models similar to the Krawinkler Steel Model are under development. See reference by Lowes and Altoontash.
When to Include P-Delta Effects?

2000 NEHRP Provisions 5A.1.1:
“The models for columns should reflect the influence of axial load when axial loads exceed 15 percent of the buckling load”

Recommended Revision:
“P-Delta effects must be explicitly included in the computer model of the structure.”
Influence of P-Delta Effects:
1) Loss of Stiffness and increased displacements

Shear Force

\[ V_y \]
\[ V^* \]
\[ V_y \]

Displacement

\[ \delta_y \]

\[ \delta \]

\[ H \]

Excluding P-Delta

Including P-Delta

\[ K_G = -\frac{P}{H} \]

\[ K_E = \frac{V_y}{\delta_y} \]

\[ K = K_E + K_G \]
Influence of P-Delta Effects:

2) Loss of Strength

\[ \theta = \frac{P \delta_y}{V_y H} \]

\[ V_y^* = V_y (1 - \theta) \]
Influence of P-Delta Effects:

3) Larger residual deformations and increased tendency towards dynamic instability
Modeling P-Delta Effects
Linearized vs Consistent Geometric Stiffness

Linearized

Consistent

Large $P-\Delta$

Small $P-\delta$

$\Delta$
Modeling P-Delta Effects

Linearized Geometric Stiffness

• Uses linear shape function to represent displaced shape. No iteration required for solution.

• Solution based on undeformed geometry

• Significantly overestimates buckling loads for individual columns

• Useful ONLY for considering the “Large P-Delta” Effect on a story-by-story basis
Modeling P-Delta Effects Consistent Geometric Stiffness

- Uses cubic shape function to represent displaced shape. Iteration required for solution.
- Solution based on undeformed geometry
- Accurately estimates buckling loads for individual columns only if each column is subdivided into two or more elements.
- Does not provide significant increase in accuracy (compared to linearized model) if being used only for considering the “Large P-Delta” effect in moment resisting frame structures.
Modeling P-Delta Effects

A

B

C

D

Lateral Column

Leaner Column

Tributary Area for Gravity Loads on Frame A
Modeling P-Delta Effects

Tributary Area for P-Delta Effects on Frame A

- Lateral Column
- Leaner Column
Modeling P-Delta Effects

Tributary Gravity Loads

Tributary P-Delta Loads

Activate Geometric Stiffness in these Columns Only.

Slaving
How Much Gravity Load to Include for P-Delta Analysis?

• Full Dead Load
• 10 PSF Partition Load (or computed value if available)
• Full Reduced Live Load (as would be used for column design).
• Reduced Live Load based on most probable live load. See for example Commentary of ASCE 7.
• Effect of Vertical Accelerations?
Modeling P-Delta Effects

Under “Force Control” an analysis may terminate due to a non-positive definite tangent stiffness matrix.
Must Use Displacement Controlled Analysis to Obtain Complete Response
When Using Displacement Control (or response-history analysis), do not recover base shears from column forces.