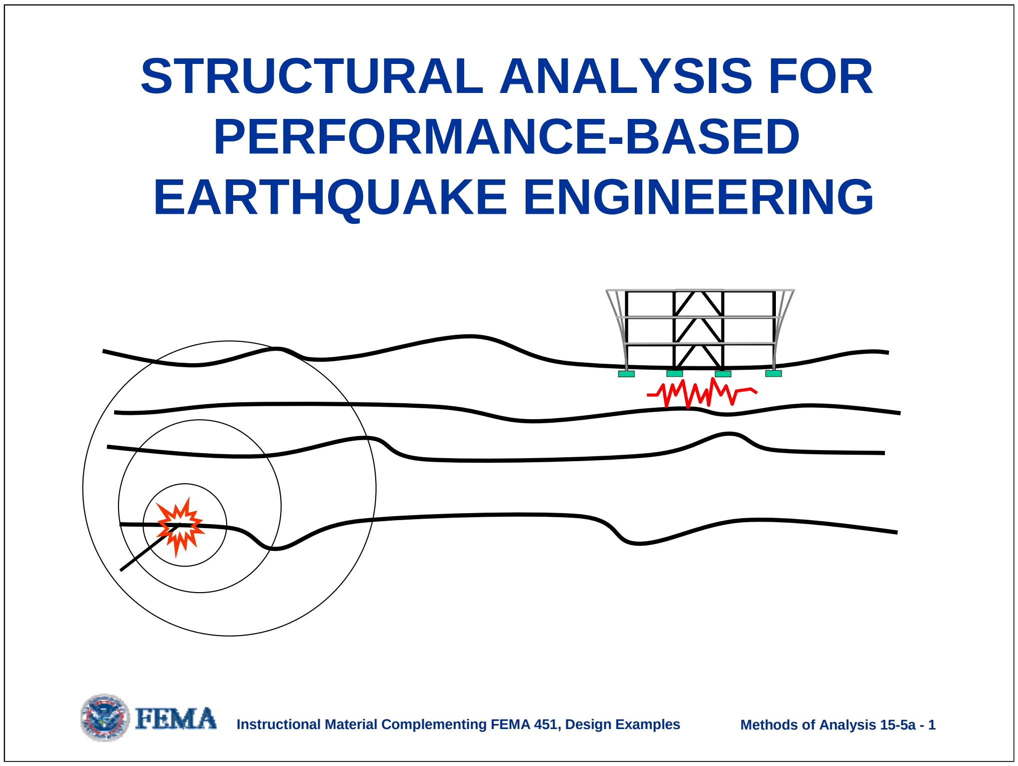
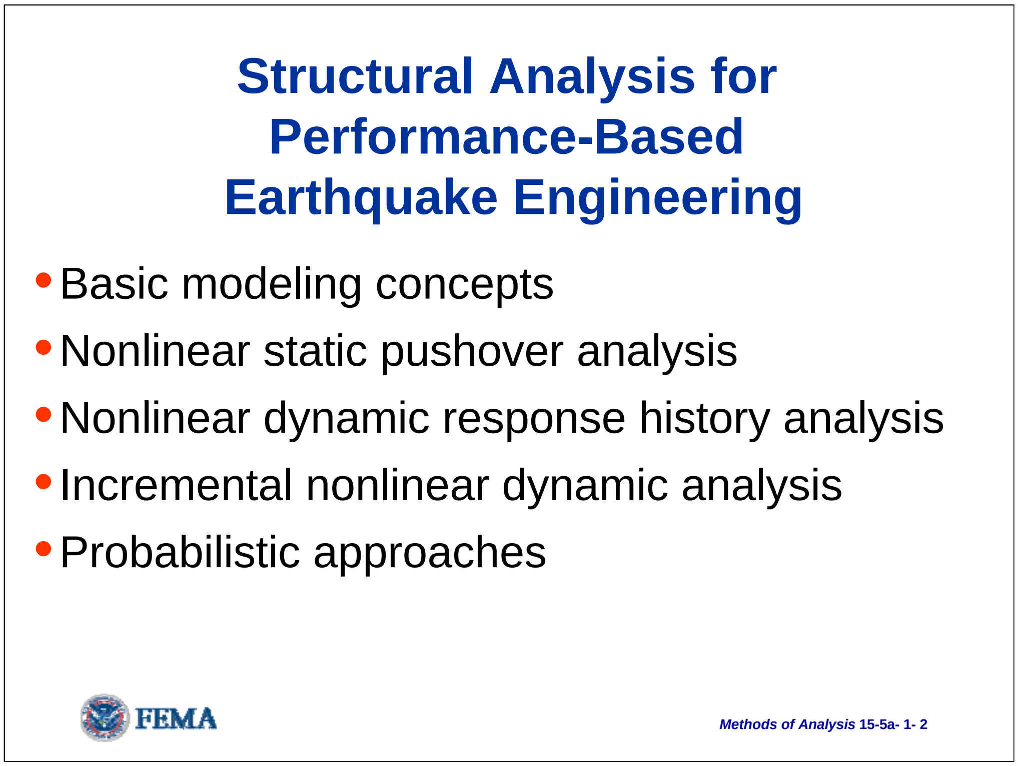


STRUCTURAL ANALYSIS FOR PERFORMANCE-BASED EARTHQUAKE ENGINEERING

FEMA Instructional Material Complementing FEMA 451, Design Examples

This topic addresses structural analysis requirements in performance-based earthquake engineering. Such analysis must typically include a variety of nonlinear effects, both material and geometric. This topic provides an overview of nonlinear analysis methodologies.



Structural Analysis for Performance-Based Earthquake Engineering

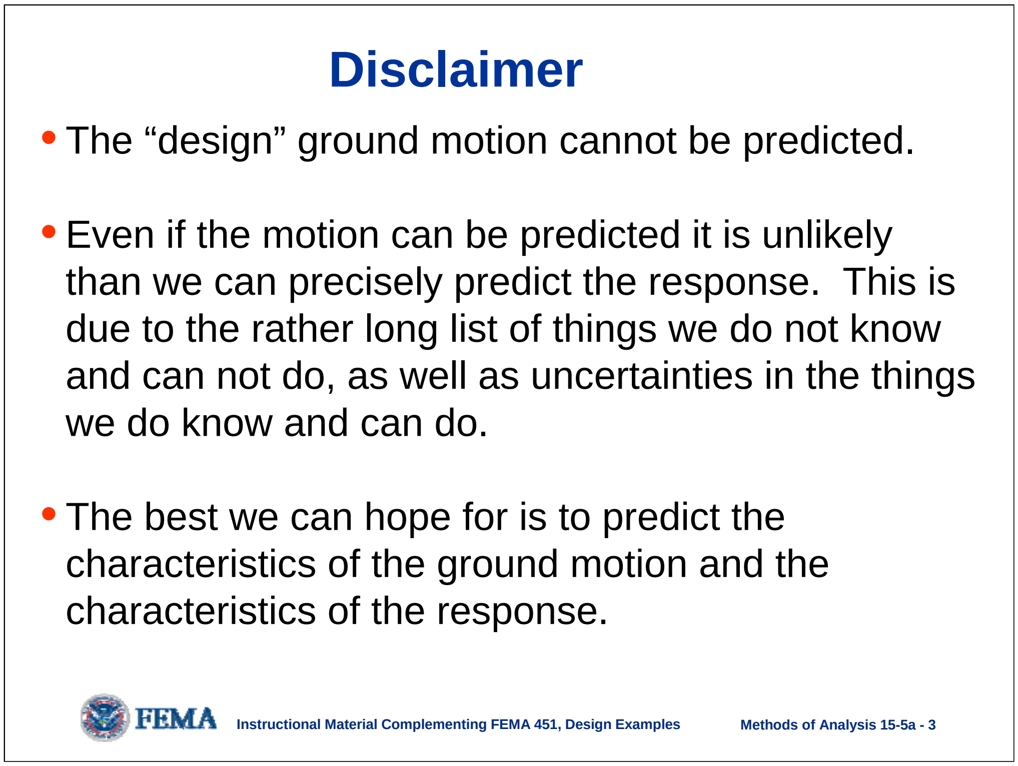
- Basic modeling concepts
- Nonlinear static pushover analysis
- Nonlinear dynamic response history analysis
- Incremental nonlinear dynamic analysis
- Probabilistic approaches

Methods of Analysis 15-5a- 1- 2

This is a summary of the topics covered. It should be emphasized that this slide (third bullet) and subsequent slides use the terminology “Response History Analysis” instead of “Time History Analysis”. Response history is a more accurate description of what is being done. Analyzing the history of time makes little sense, whereas the history of the response of a structure is meaningful.

Incremental Dynamic Analysis (IDA) is a relatively new approach that uses response history analyses in a systematic manner to assess the behavior of a structure subjected to a suite of ground motions. The structure is repeatedly analyzed for each motion scaled for gradually increasing intensities. For each intensity analyzed, certain damage measures are recorded and plotted against intensity to produce “IDA Curves”.

We will only briefly review the probabilistic approaches because the approach is quite new, and has not been fully formulated. We will describe the probabilistic approach used by FEMA 350, and the broader approach being suggested by PEER.



Disclaimer

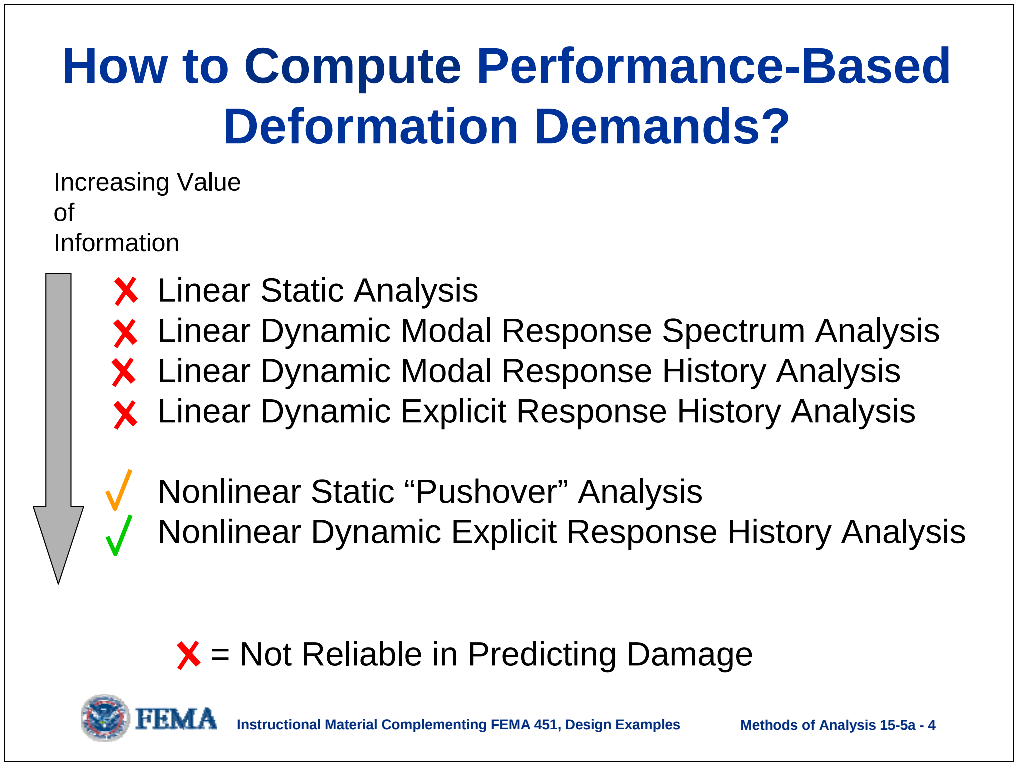
- The “design” ground motion cannot be predicted.
- Even if the motion can be predicted it is unlikely than we can precisely predict the response. This is due to the rather long list of things we do not know and can not do, as well as uncertainties in the things we do know and can do.
- The best we can hope for is to predict the characteristics of the ground motion and the characteristics of the response.

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 3

It must be made very clear that the purpose of analysis (in the context of this Topic) is NOT to accurately predict the response of a certain structure to a certain ground motion. This is impossible due to the large number of uncertainties in modeling, loading, analysis, and interpretation of results. Also, what is being predicted will never occur since the actual ground motion is not known.

What we are trying to do is to use analysis to get a handle on the likely behavior of a structure, and to estimate whether or not such behavior will meet pre-established performance objectives.



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

Instructional Material Complementing FEMA 451, Design Examples

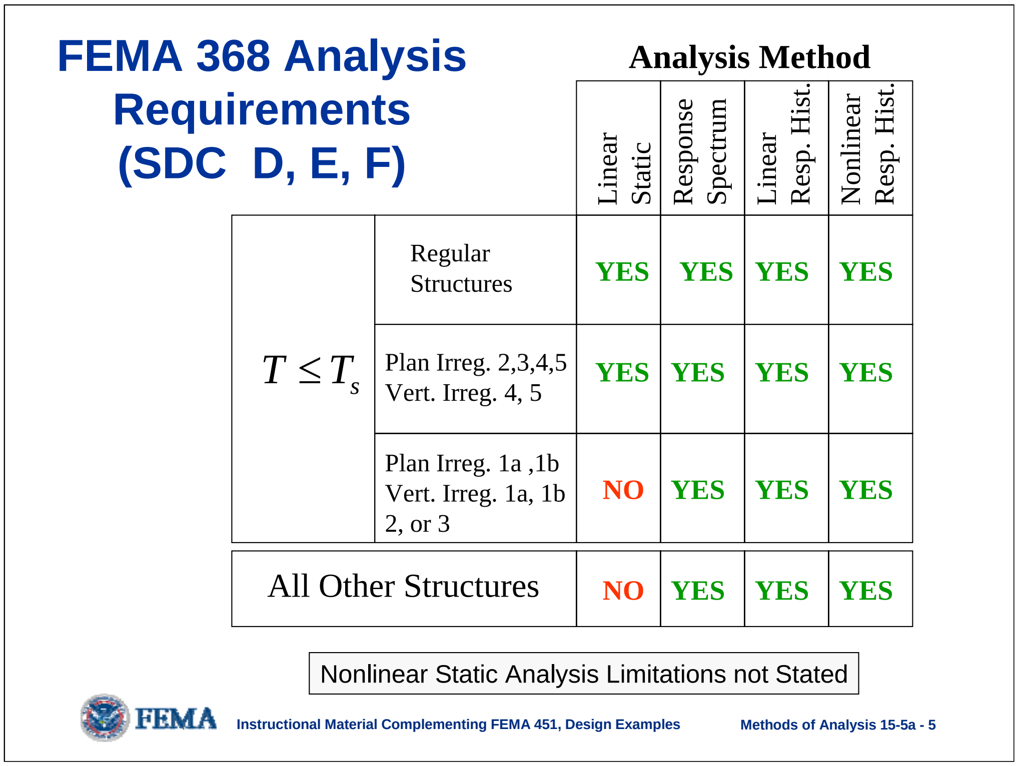
Methods of Analysis 15-5a - 4

In earthquake engineering it has been said that “Strength is essential, but otherwise unimportant”. This is true because the basic requirement in seismic resistant design is that deformation demand must be less than or equal to the deformation capacity of the system’s elements and components. The design objective is to provide sufficient strength to keep deformation demands below the capacity.

In structural analysis for performance-based engineering, therefore, the emphasis is on predicting deformation demands. Because the response is almost certainly inelastic, the analysis must explicitly include inelastic effects. Thus the first four analysis methods listed are not applicable.

Pushover analysis was originally adopted as a “practical” replacement for more time-consuming response history analysis. However, the method has its limitations, and is falling out of favor with many researchers.

While nonlinear response history analysis has certain advantages, the down side is that multiple ground motions must be considered, and that response can vary widely for the same system analyzed for a suite of reasonably scaled motions. Response can also vary considerably for minor variations of the same system responding to the same ground motion.



FEMA 368 Analysis Requirements (SDC D, E, F)

		Analysis Method			
		Linear Static	Response Spectrum	Linear Resp. Hist.	Nonlinear Resp. Hist.
$T \leq T_s$	Regular Structures	YES	YES	YES	YES
	Plan Irreg. 2,3,4,5 Vert. Irreg. 4, 5	YES	YES	YES	YES
	Plan Irreg. 1a ,1b Vert. Irreg. 1a, 1b 2, or 3	NO	YES	YES	YES
All Other Structures		NO	YES	YES	YES

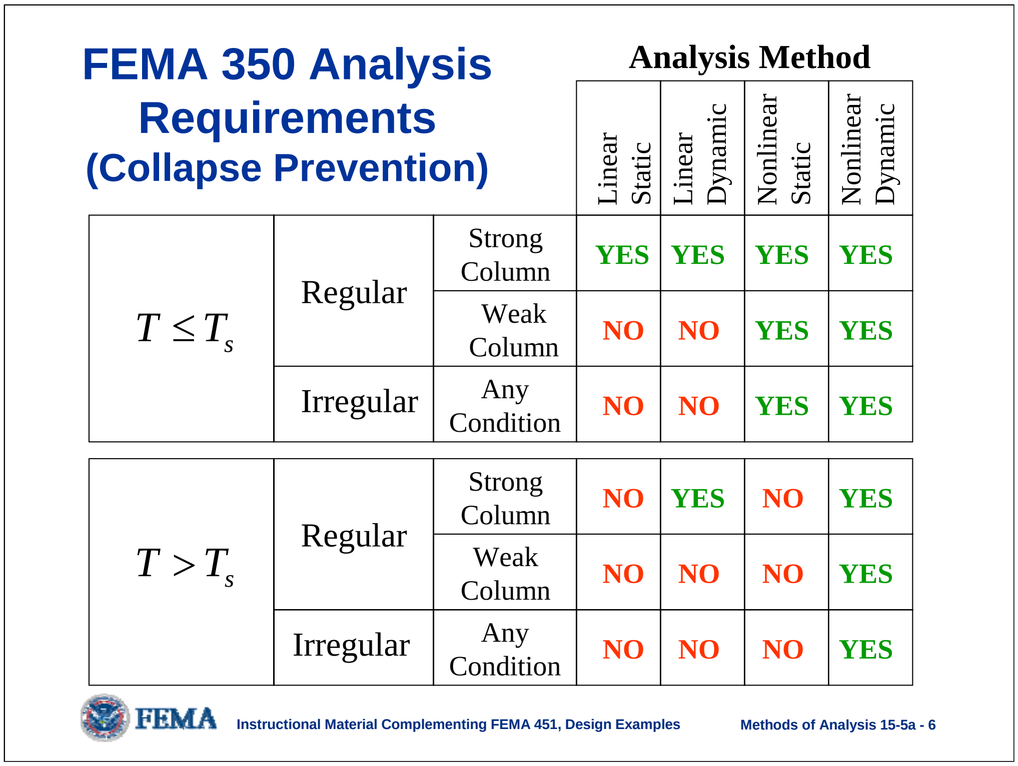
Nonlinear Static Analysis Limitations not Stated

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 5

The 2003 *NEHRP Recommended Provisions* is not a performance-based document and, hence, has no requirements for nonlinear analysis. An Appendix to Chapter 5 of the *Provisions,* however, uses nonlinear analysis in the context of a FEMA 273 nonlinear static pushover approach.

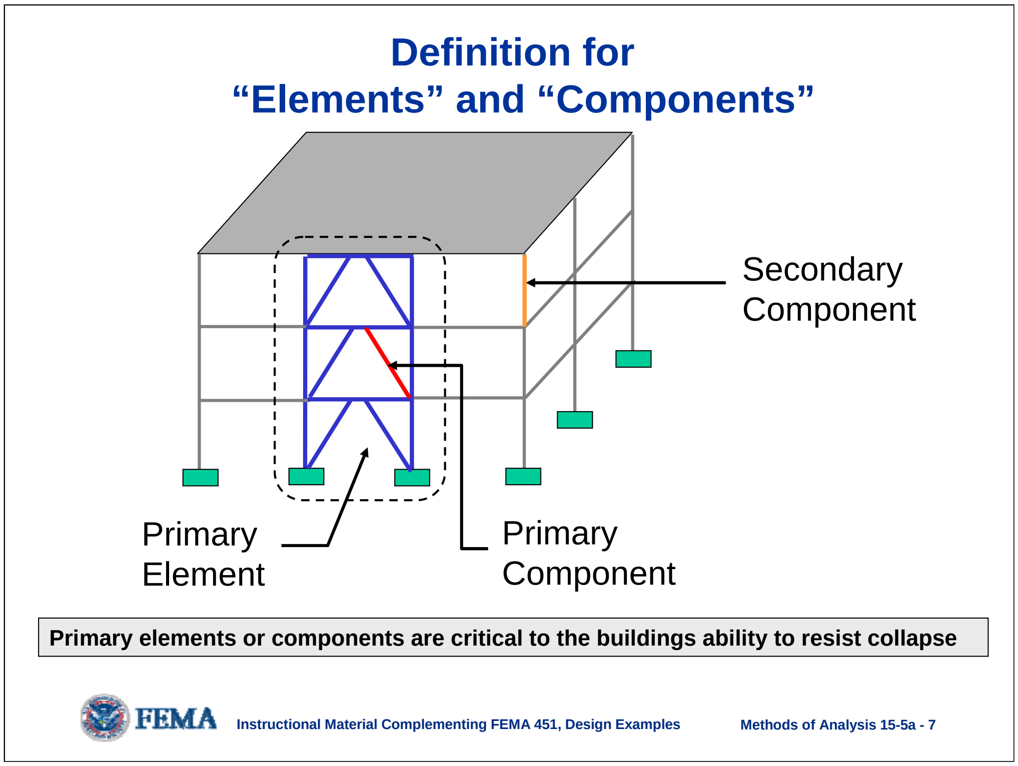
Note that T_s is the point on the design spectrum where the constant acceleration portion of the spectrum crosses the constant velocity (inversely proportional to T) portion of the spectrum. $T_S=S_{D1}/S_{DS}$



FEMA 350 Analysis Requirements (Collapse Prevention)

			Analysis Method			
			Linear Static	Linear Dynamic	Nonlinear Static	Nonlinear Dynamic
$T \leq T_s$	Regular	Strong Column	YES	YES	YES	YES
		Weak Column	NO	NO	YES	YES
	Irregular	Any Condition	NO	NO	YES	YES
$T > T_s$	Regular	Strong Column	NO	YES	NO	YES
		Weak Column	NO	NO	NO	YES
	Irregular	Any Condition	NO	NO	NO	YES

FEMA 350 is a performance-based document, and has specific requirements for nonlinear analysis. Note the situations where nonlinear dynamic response history analysis is *required.* The first table is for short period buildings and the second is for longer period buildings.



Definition for “Elements” and “Components”

Secondary Component

Primary Element

Primary Component

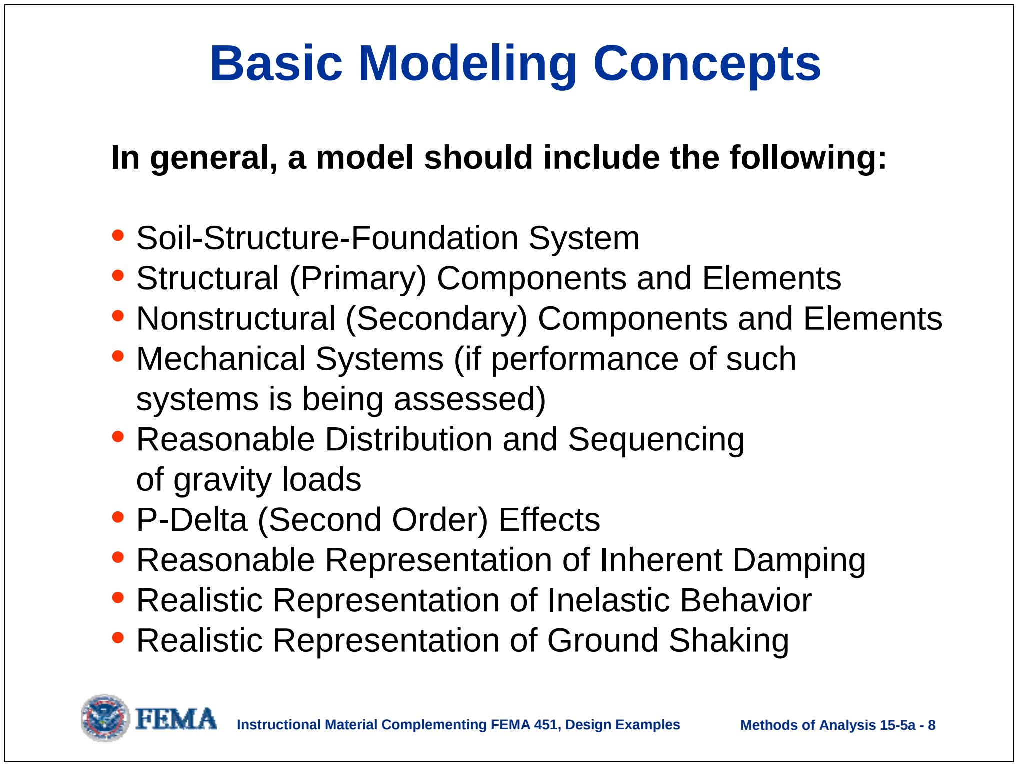
Primary elements or components are critical to the buildings ability to resist collapse

FEMA

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 7

FEMA 273 uses terminology which may be confusing. An “Element” is really a system, such as a moment frame, braced frame, and so-on. A “Component” is a particular member of the “Element”. The confusion lies in the fact that the word element is commonly used to refer to an individual member in the context of a finite element analysis.

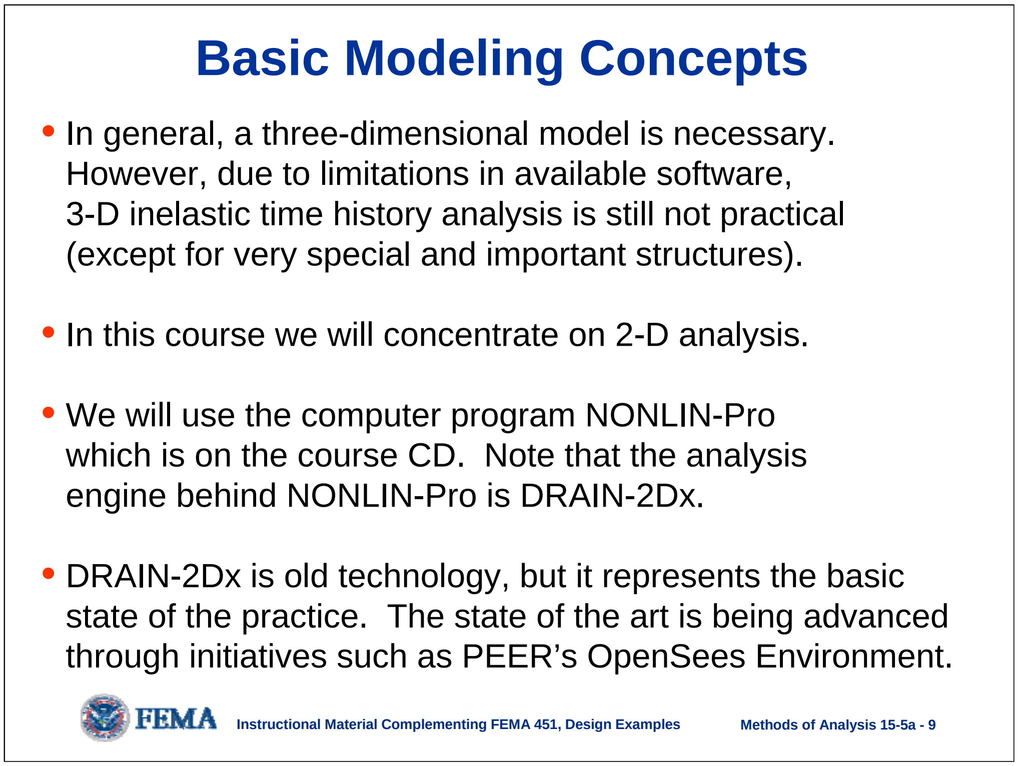


Basic Modeling Concepts

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

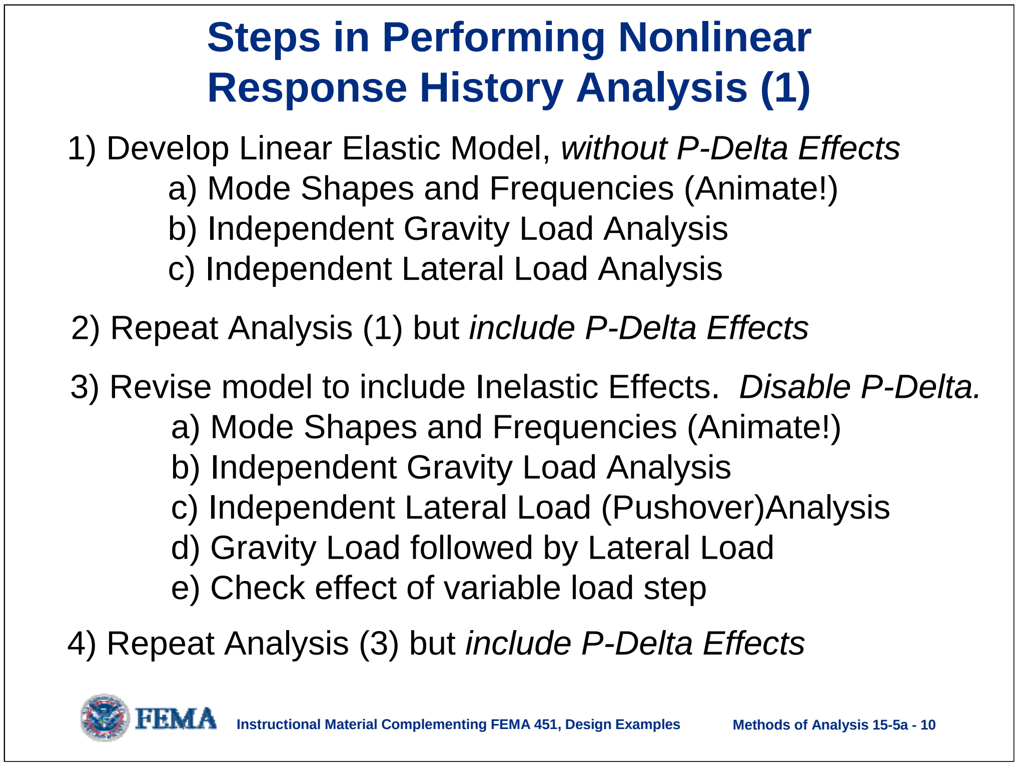
This list gives the idealized requirements of an analytical model. Unfortunately, sufficient information is often not available, and when the information is available, very significant uncertainties make choices difficult. If the certainties can be identified and quantified, several analyses with a variety of properties may be required to adequately bound the response.



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.

Three dimensional nonlinear dynamic analysis is becoming more available with the release of SAP 2000 Version 8, as well as a 3D version of RAM Perform. However, it will still take several years for these programs to supercede DRAIN 2Dx and perhaps RAM Perform 2D as the “state of the practice”.



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

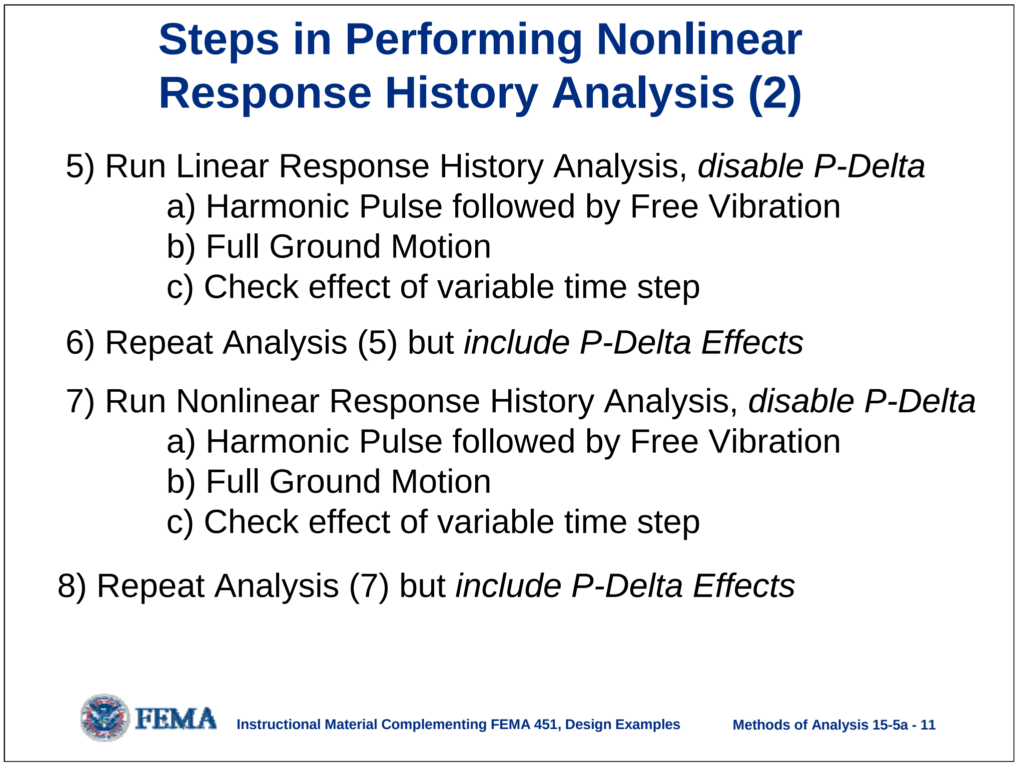
2) Repeat Analysis (1) but *include P-Delta Effects*

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*

Running a nonlinear dynamic response history analysis of a structure is one of the most complex tasks a structural engineer has to do. Many engineers are too rushed to perform the analysis, and do not take the time to perform the steps outlined on this and the next slide. The result can be a meaningless analysis.

Before any nonlinear analysis is run, a linear analysis must be performed to check the model. Similarly, before any dynamic analysis is run a static analysis must be performed on the same model. After each analysis a reasonableness check must be performed. Are the frequencies and mode shapes realistic. Do P-Delta effects make the period longer? How does the presence of gravity loads affect hinge sequencing? Do the results of the pushover analysis depend on the size of the load step? Does the dynamic pulse loading produce the appropriate free vibration response (check period and damping). This is only a short list of items that should be checked.



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*

Continuation of previous slide.

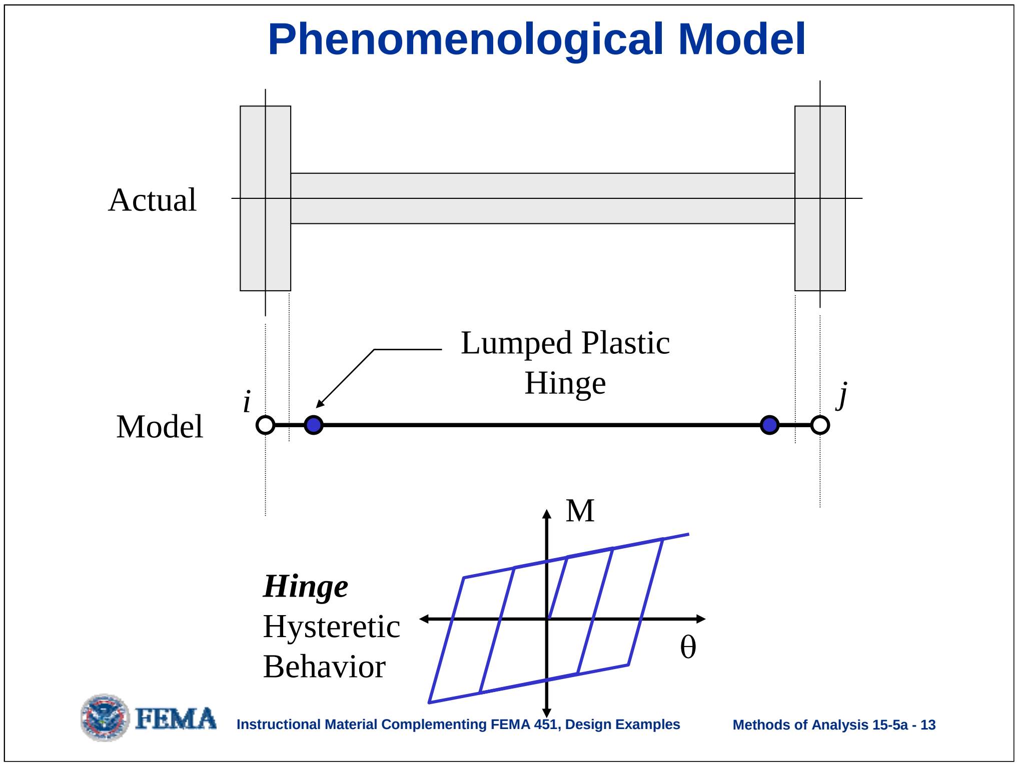
Basic Component Model Types

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.

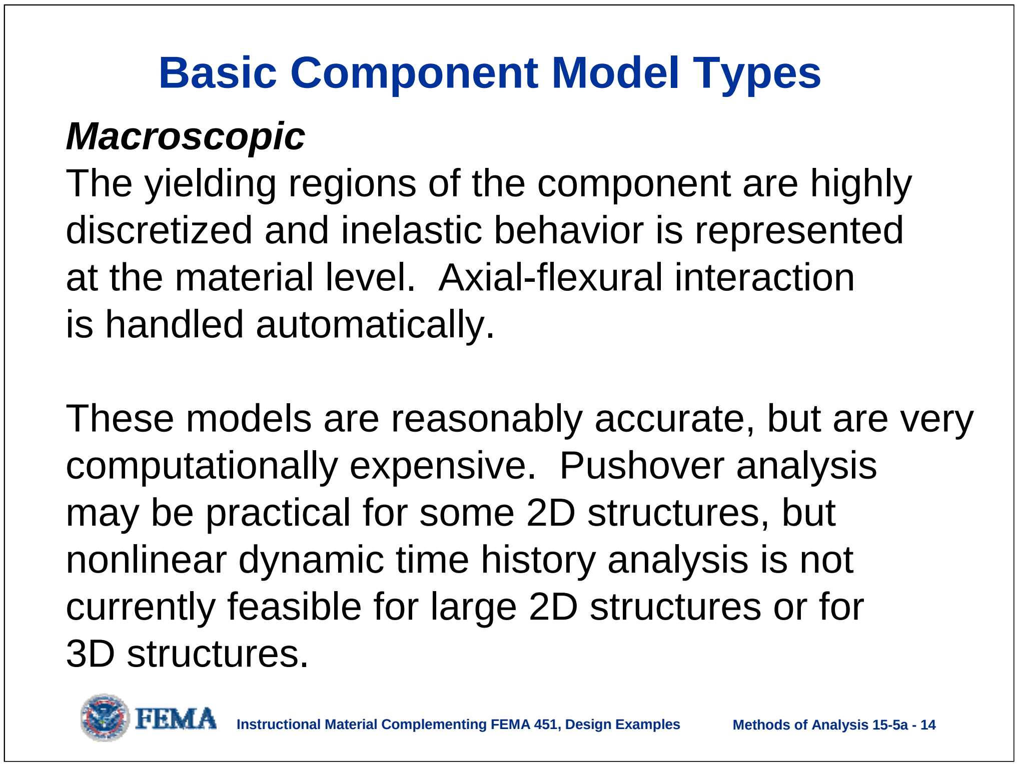
Very large structures may be modeled using this approach. Nonlinear dynamic analysis is practical for most 2D structures, but may be too computationally expensive for 3D structures.

The two basic element modeling types are Phenomenological and Macroscopic. Phenomenological models, explained here, are much more practical and are the norm for most nonlinear dynamic response history programs.



Phenomenological Model

Phenomenological models are typically used to represent a region of a component, such as a plastic hinge in a beam. Modeled behavior may include axial-flexural-shear interaction, or may be limited to simple flexural behavior as shown here. Because all of the inelastic activity is limited to the (typically zero length) hinge, a phenomenological model may also be referred to as a “lumped” plasticity model. In the diagram shown above the expected plastic behavior at each end of the beam is modeled as a simple plastic hinge. Note that the hinge is located some distance in from the ends of the beam.



Basic Component Model Types

Macroscopic

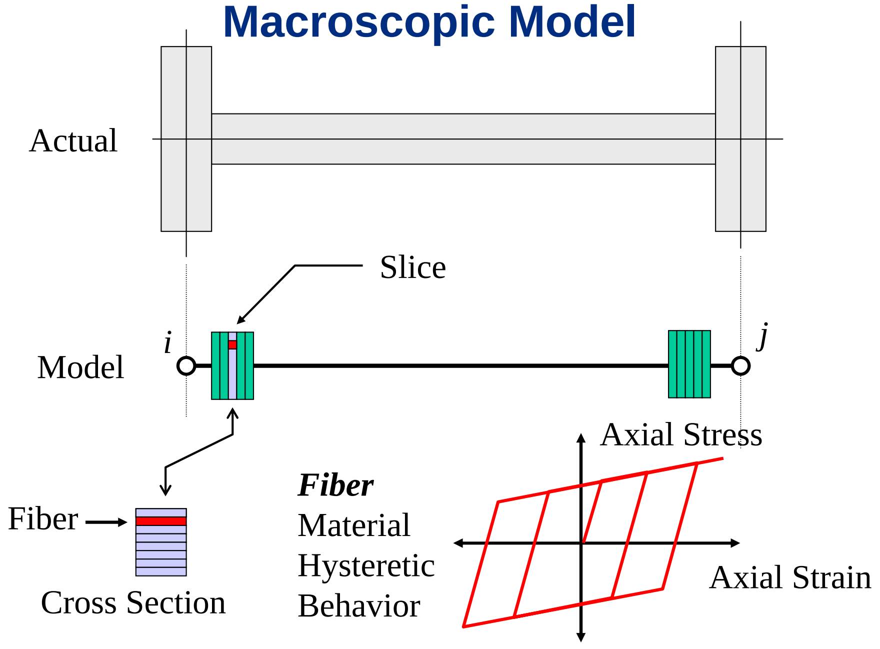
The yielding regions of the component are highly discretized and inelastic behavior is represented at the material level. Axial-flexural interaction is handled automatically.

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.

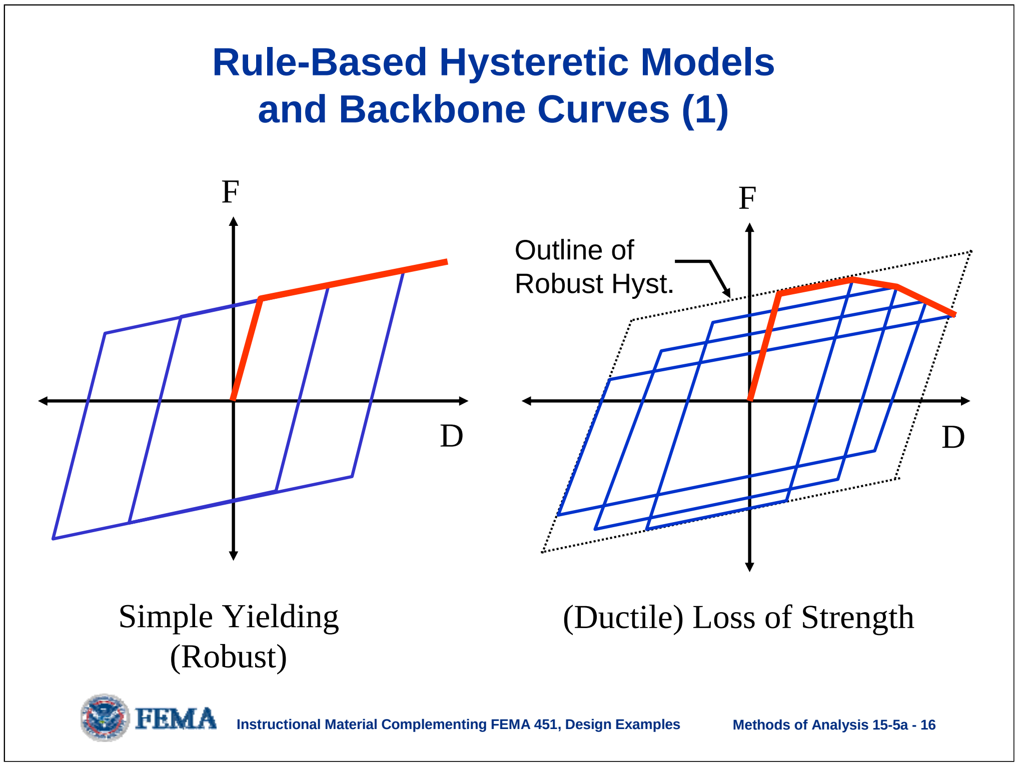
FEMA Instructional Material Complementing FEMA 451, Design Examples Methods of Analysis 15-5a - 14

A macroscopic model attempts to represent behavior down to the “fiber” level. These models have the advantage of automatically including some aspects of complex behavior, such as axial-flexural interaction. Unfortunately, macroscopic models are still prohibitively expensive if used on a large scale. However, it may be reasonable to use a mixture of phenomenological and macroscopic models in a single structure. For example, simple plastic hinges may be used at the ends of beams, and a more refined fiber model used at the base of a critical shear wall.

Macroscopic Model



Here, a plastic hinge is represented as a series of slices (along the length of the beam) and a series of layers (through the depth of the beam). For a concrete structure special layers may be used to represent both concrete and reinforcing steel with different constitutive laws used for unconfined concrete, confined concrete, and steel. This type of model automatically represents growth in the length of plastic hinges, as well as neutral axis migration. Some analysts refer to this type of model as a spread plasticity model.

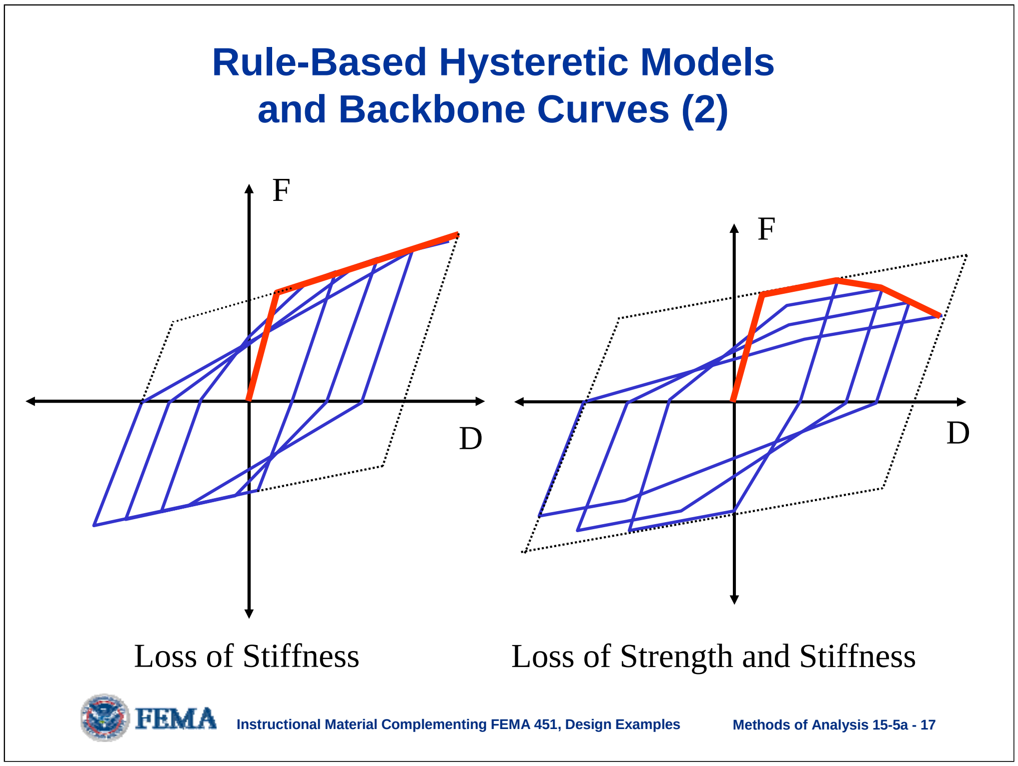


Rule-Based Hysteretic Models and Backbone Curves (1)

Simple Yielding (Robust)

(Ductile) Loss of Strength

Whether used to represent an entire plastic hinge or a single fiber, it is necessary to have computational rules for tracking hysteretic behavior. There are a nearly infinite number of behaviors that can be so represented. These models represent stable hysteretic behavior (left; an unbonded brace, for example) and a system with gradual strength loss (right: a plastic hinge in a wide-flange beam, for example).



Rule-Based Hysteretic Models and Backbone Curves (2)

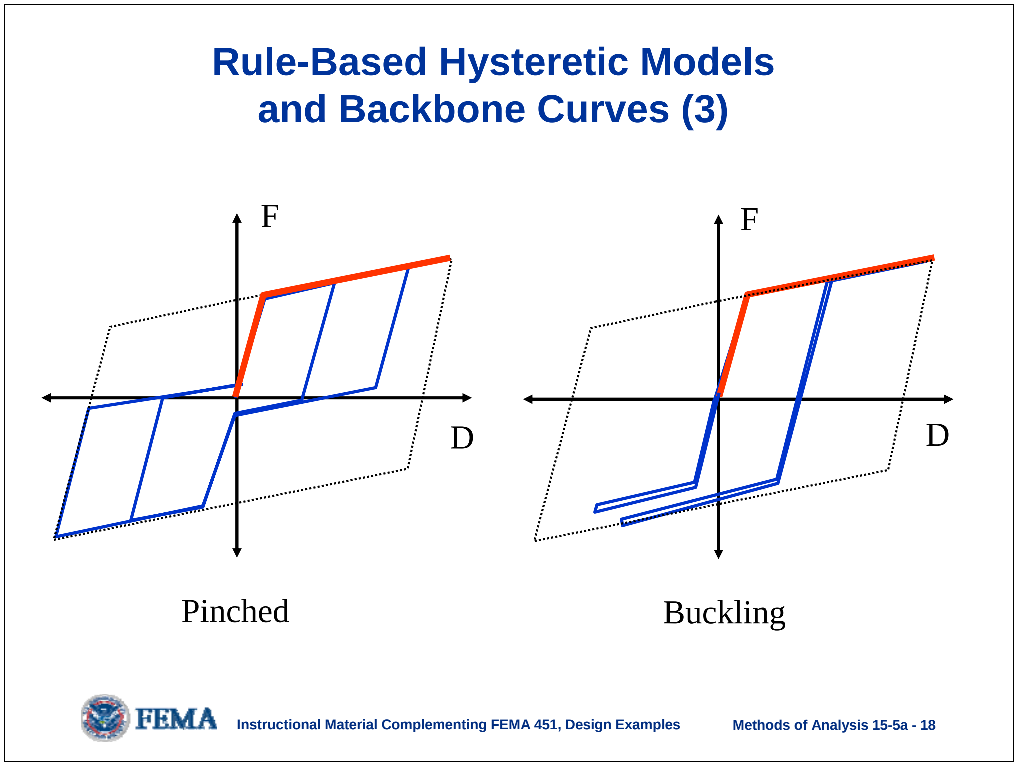
Loss of Stiffness

Loss of Strength and Stiffness

FEMA Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 17

Hysteresis rules may also include loss of stiffness with sustained strength (left: a well confined reinforced concrete beam) or a loss of both strength and stiffness (right: a poorly confined concrete beam).



Rule-Based Hysteretic Models and Backbone Curves (3)

Pinched

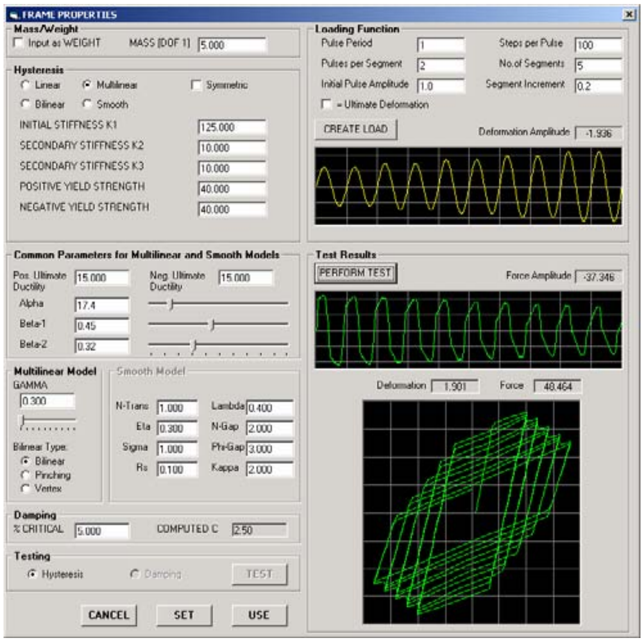
Buckling

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 18

Hysteresis rules may also include pure pinching (left: a self-centering device) or buckling and tension yielding (right: a slender brace).

Sivaselvan and Reinhorn Models in NONLIN (MDOF MODEL)



Mass/Weight
☐ Input as WEIGHT
MASS [DOF 1] 5.000

Hysteresis
○ Linear ● Multilinear ☐ Symmetric
○ Bilinear ○ Smooth
INITIAL STIFFNESS K1 125.000
SECONDARY STIFFNESS K2 10.000
SECONDARY STIFFNESS K3 10.000
POSITIVE YIELD STRENGTH 40.000
NEGATIVE YIELD STRENGTH 40.000

Common Parameters for Multilinear and Smooth Models
Pos. Ultimate Ductility 15.000
Neg. Ultimate Ductility 15.000
Alpha 17.4
Beta-1 0.45
Beta-2 0.32

Multilinear Model
GAMMA 0.300
Bilinear Type:
● Bilinear
○ Pinching
○ Vertex

Smooth Model
N-Trans 1.000
Eta 0.300
Sigma 1.000
Rs 0.100
Lambda 0.400
N-Gap 2.000
Phi-Gap 3.000
Kappa 2.000

Damping
% CRITICAL 5.000
COMPUTED C 2.50

Testing
● Hysteresis ○ Damping

Loading Function
Pulse Period 1
Pulses per Segment 2
Initial Pulse Amplitude 1.0
Steps per Pulse 100
No.of Segments 5
Segment Increment 0.2
☐ = Ultimate Deformation
Deformation Amplitude -1.936

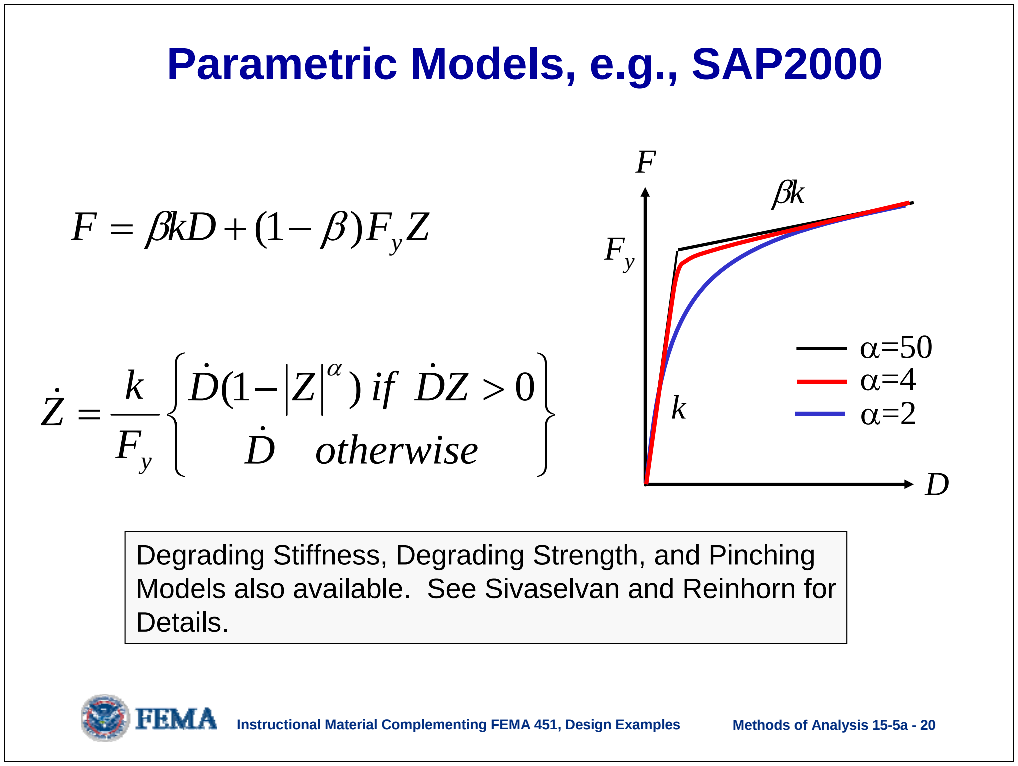
Test Results
Force Amplitude -37.346
Deformation 1.901
Force 40.464

NONLIN

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 19

Sivaselvan and Reinhorn have developed a nice family of multilinear models. This slide is a screen capture from NONLIN's MDOF modeler.



Parametric Models, e.g., SAP2000

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

$$\dot{Z} = \frac{k}{F_y}\begin{Bmatrix} \dot{D}(1-|Z|^{\alpha}) \; if \; \dot{D}Z > 0 \\ \dot{D} \quad otherwise \end{Bmatrix}$$

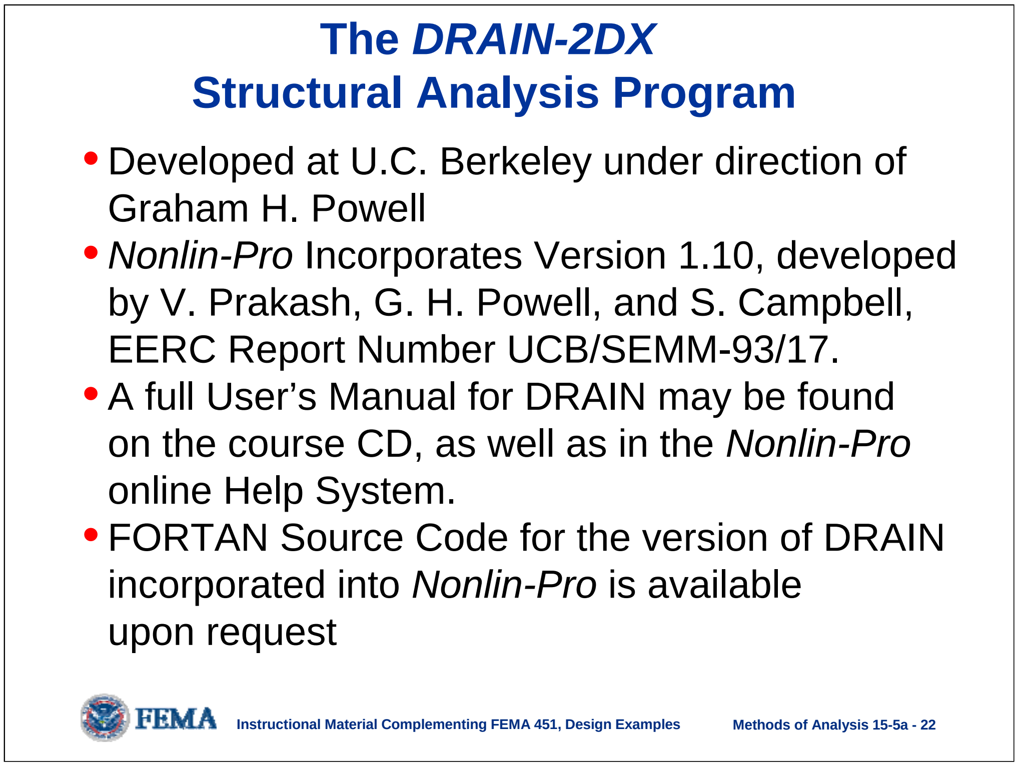
Degrading Stiffness, Degrading Strength, and Pinching Models also available. See Sivaselvan and Reinhorn for Details.

There are also a variety of smooth models. The one shown here is used in SAP2000. Much more elaborate models are available. The Sivaselvan/Reinhorn smooth models have been incorporated into NONLIN’s MDOF model.

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.

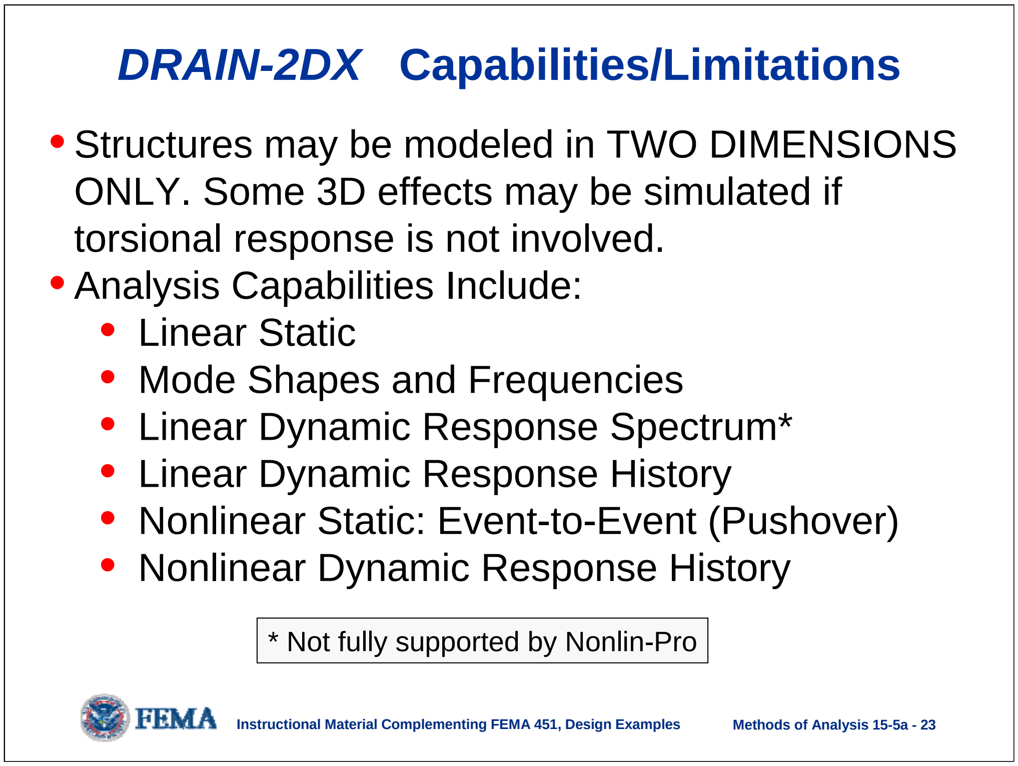
Basic info on NONLIN-Pro (1).



The *DRAIN-2DX* Structural Analysis Program

- Developed at U.C. Berkeley under direction of Graham H. Powell
- *Nonlin-Pro* Incorporates Version 1.10, developed by V. Prakash, G. H. Powell, and S. Campbell, EERC Report Number UCB/SEMM-93/17.
- 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

Basic info on Nonlin-Pro (2).

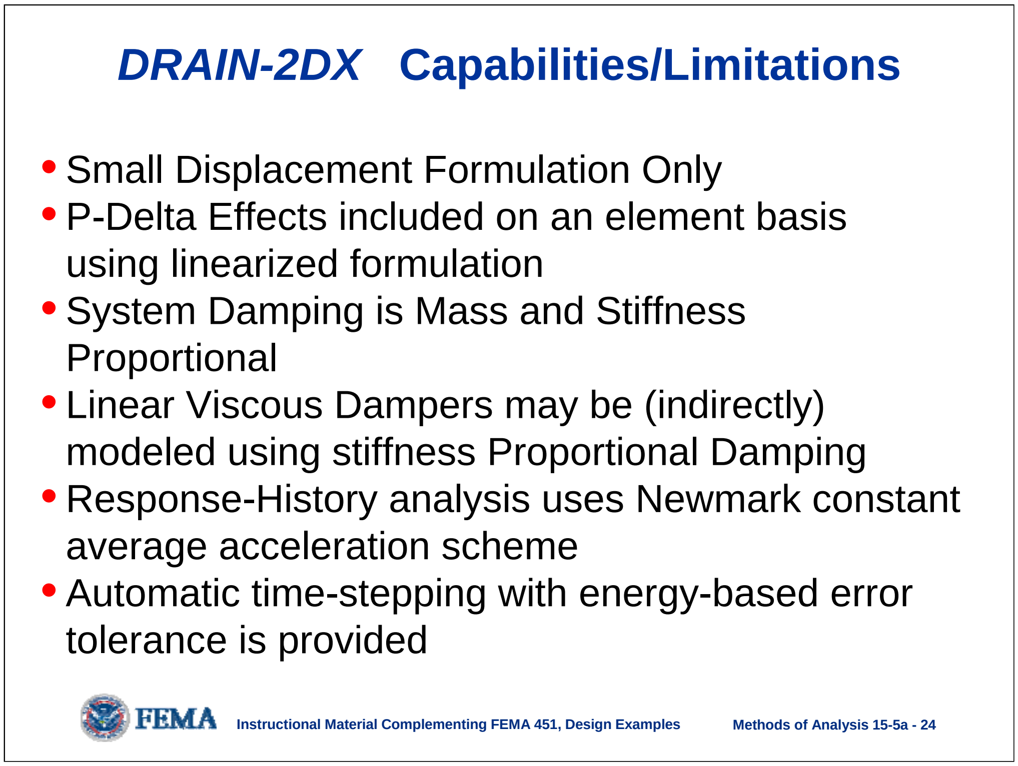


DRAIN-2DX Capabilities/Limitations

- 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

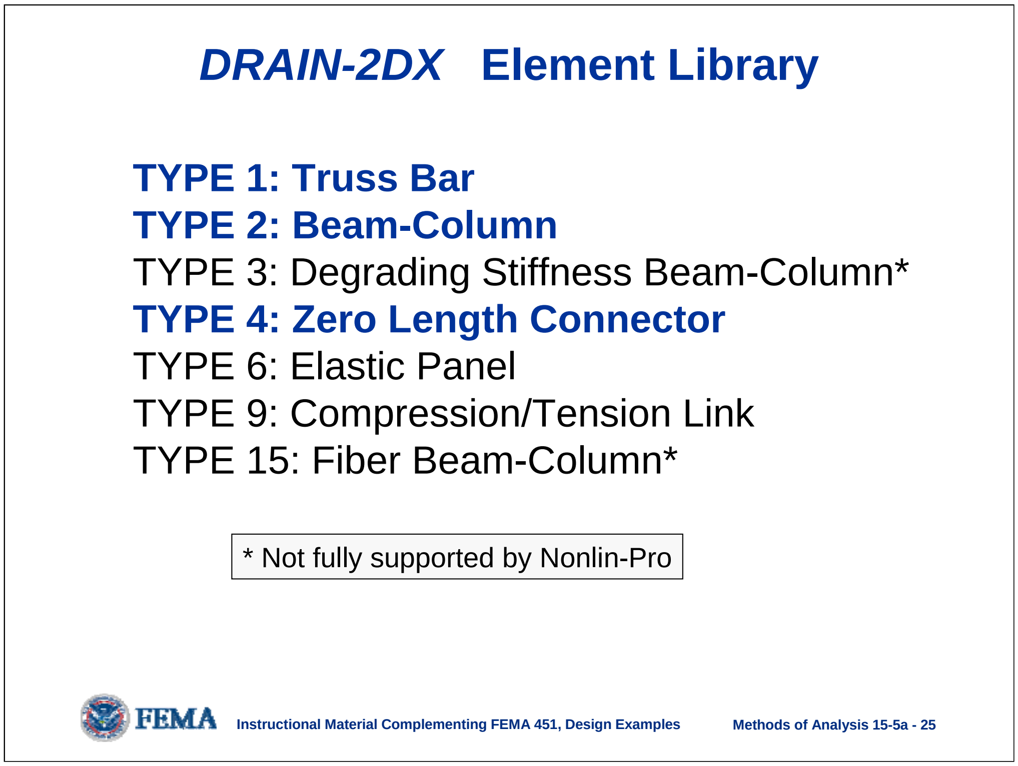
Basic info on Nonlin-Pro (3).



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

Basic info on Nonlin-Pro (4).

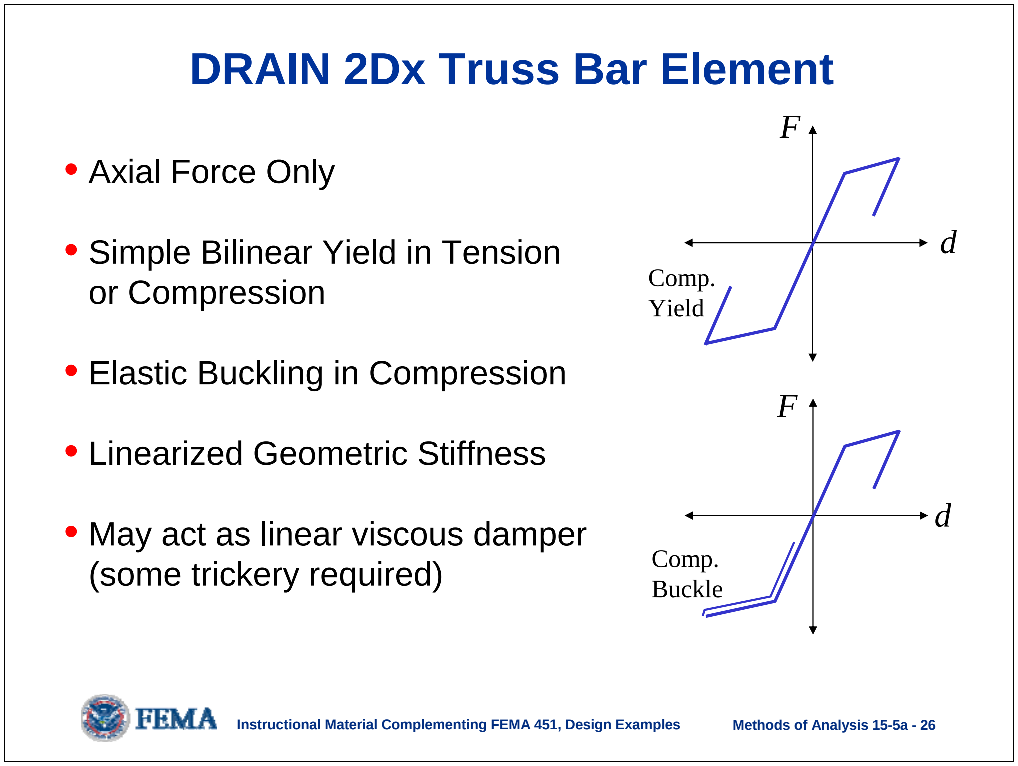


DRAIN-2DX Element Library

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

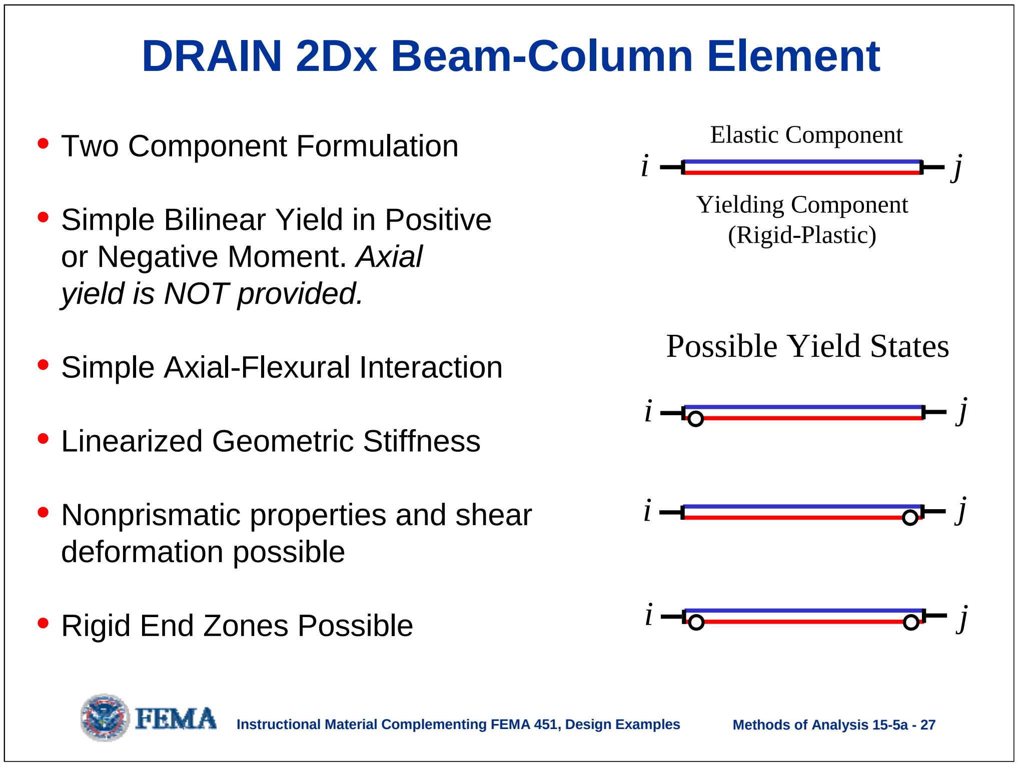
This in the NONLIN-Pro element library. All of the elements are provided by DRAIN 2Dx, but only those indicated elements are supported by the graphic pre- and post processors. The fiber element is supported but is not particularly dependable.



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)

The axial truss bar element is useful for any kind of axial link, and is also used to model linear viscous damper elements. The damper element is modeled by setting a very low stiffness, and then setting a very high stiffness proportional damping constant (beta). The product of beta and the stiffness is the desired damping constant C (units = force-time/length). Note that the Type 4 element may be used in lieu of the truss element when a zero-length device is required.

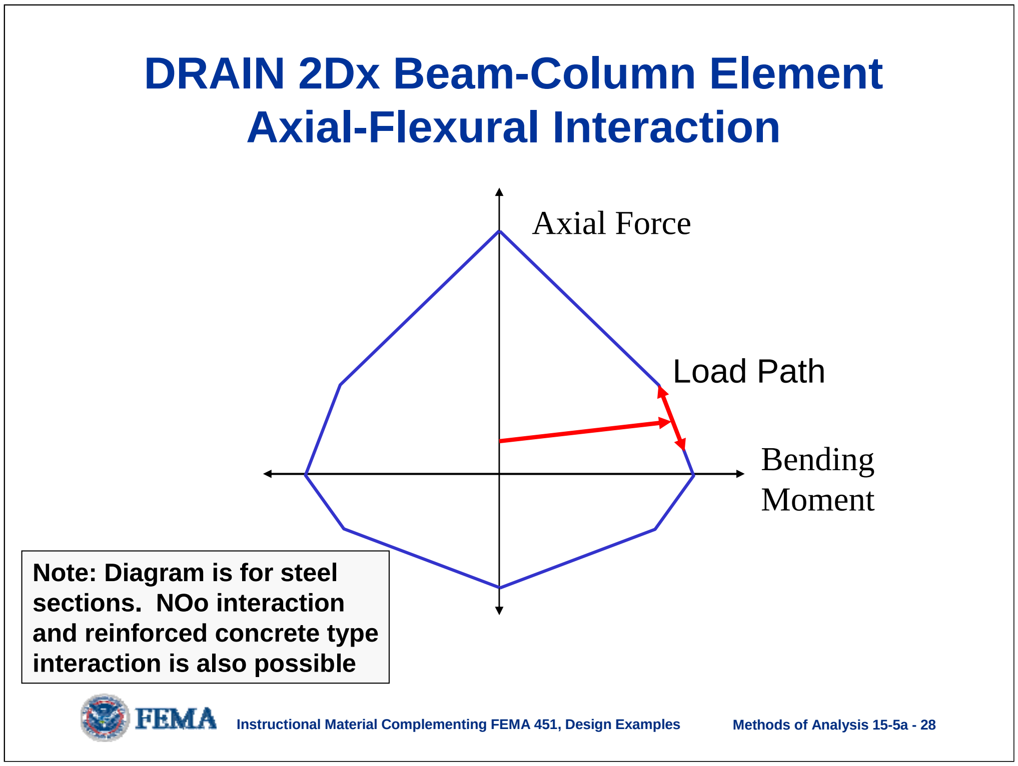


DRAIN 2Dx Beam-Column Element

- 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

FEMA Instructional Material Complementing FEMA 451, Design Examples Methods of Analysis 15-5a - 27

This is the “standard” beam-column element provided by DRAIN. The only advantage of this element is it’s ability to model axial-flexural interaction. Unfortunately, the interaction model does not work very well. For beams with flexural yielding that is independent of axial force, it is better to explicitly model the hinges using type-4 elements as explained later.



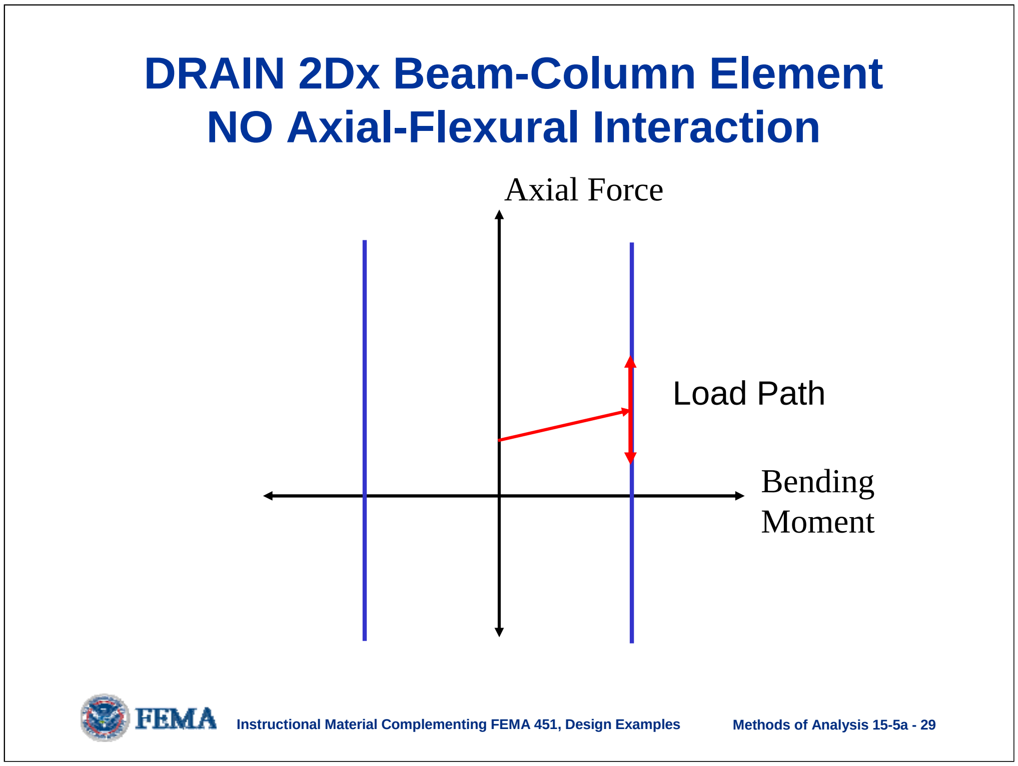
DRAIN 2Dx Beam-Column Element Axial-Flexural Interaction

Note: Diagram is for steel sections. NOo interaction and reinforced concrete type interaction is also possible

Instructional Material Complementing FEMA 451, Design Examples

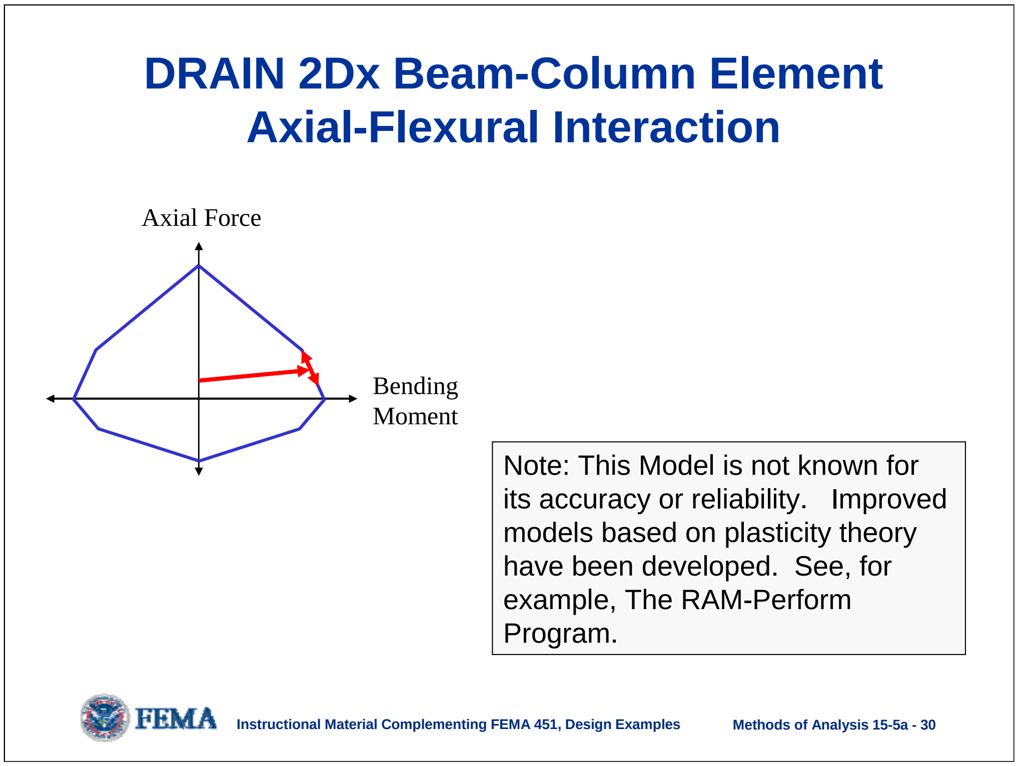
Methods of Analysis 15-5a - 28

This is the theoretical axial-flexural interaction relationship for the two-component beam-column element. When the load path intersects the yield surface, a *flexural* hinge is placed in the yielding component of the element. While yielding, the element is required to load along the path as indicated.



DRAIN 2Dx Beam-Column Element NO Axial-Flexural Interaction

For beams, the element simply yields when the moment at an end of the element reaches the flexural yield point.



DRAIN 2Dx Beam-Column Element Axial-Flexural Interaction

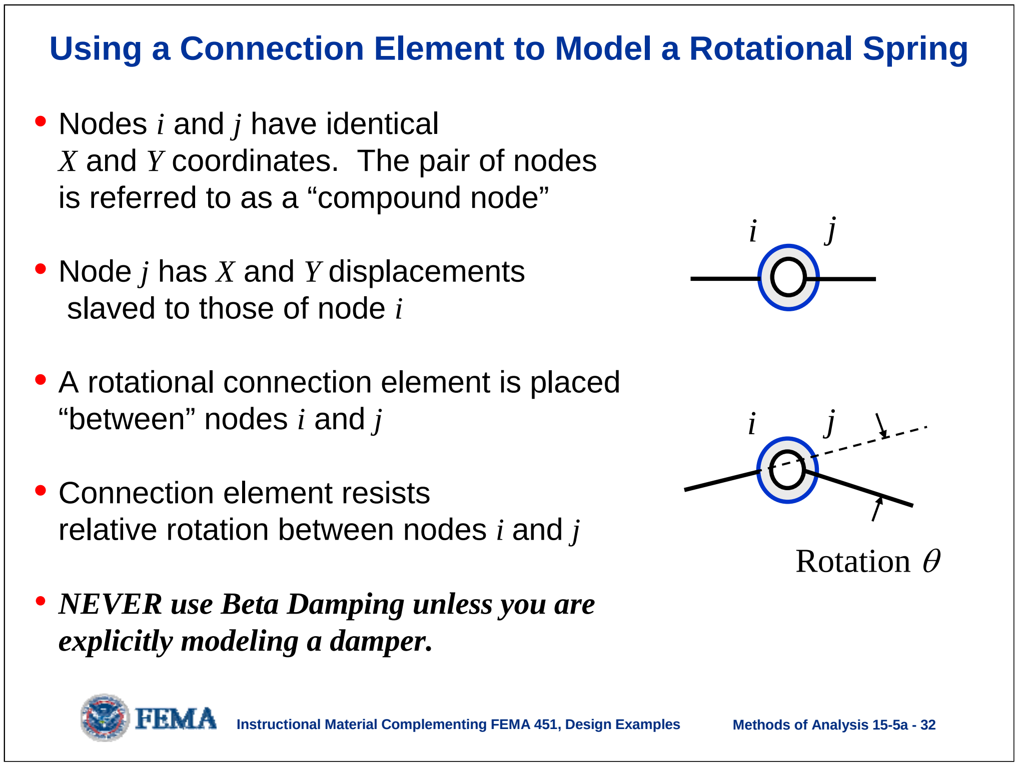
Note: This Model is not known for its accuracy or reliability.   Improved models based on plasticity theory have been developed. See, for example, The RAM-Perform Program.

Self explanatory.

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

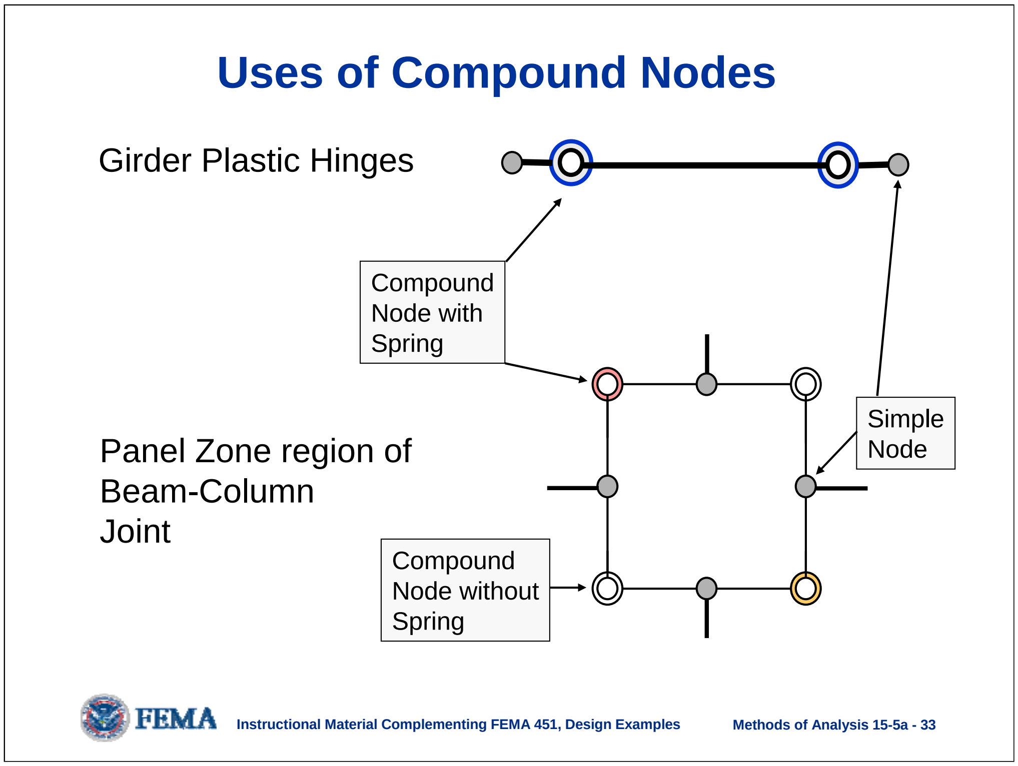
The connection element is one of the most important elements in DRAIN or in any nonlinear analysis program. This is a zero length element, and is therefore connected to two nodes that share the same X-Y coordinates. These nodes are referred to as “compound nodes”.



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.***

This slide illustrates the use of a pair of nodes and a connection element to model a plastic hinge. A real moment-free hinge may be modeled by elimination of the rotational spring. The last bullet item is a preliminary caution. The reason for the caution will be explained in some detail later in the topic.



Uses of Compound Nodes

Girder Plastic Hinges

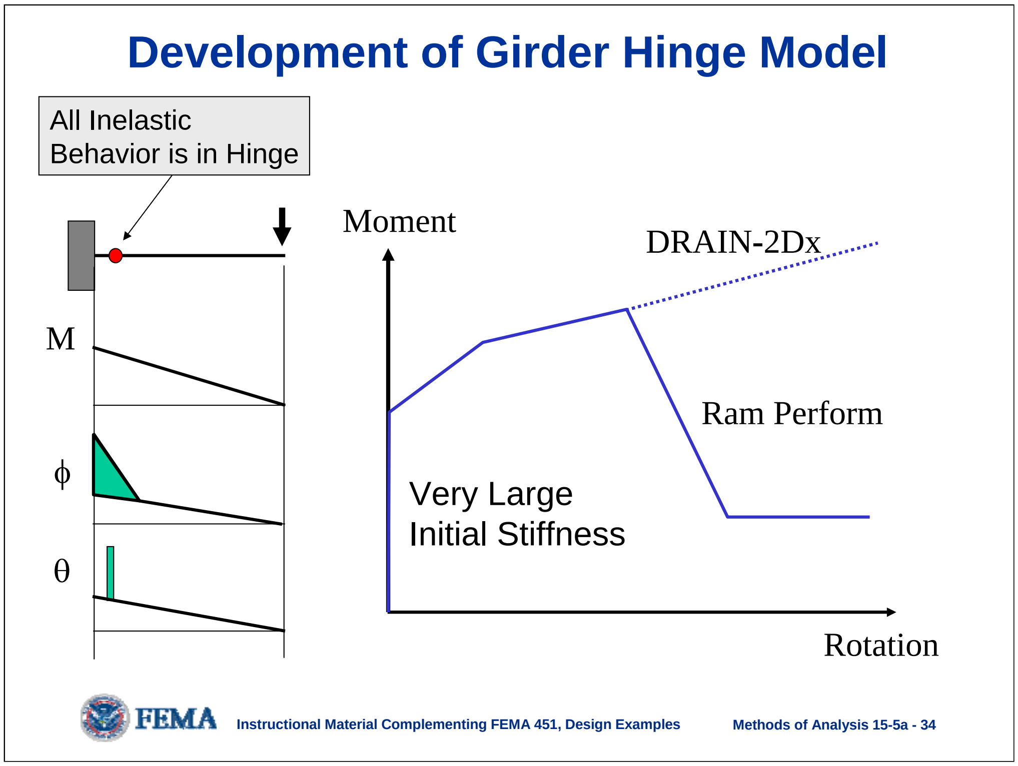
Compound Node with Spring

Simple Node

Panel Zone region of Beam-Column Joint

Compound Node without Spring

Here are two applications for a compound node. In the first case, compound nodes with rotational springs are placed at each end of a beam. In the second case, four compound node sets are used to develop a “Krawinkler” beam-column joint deformation model. The Krawinkler model and a much simpler Scissors model are described in some detail later.



Development of Girder Hinge Model

All Inelastic Behavior is in Hinge

Very Large Initial Stiffness

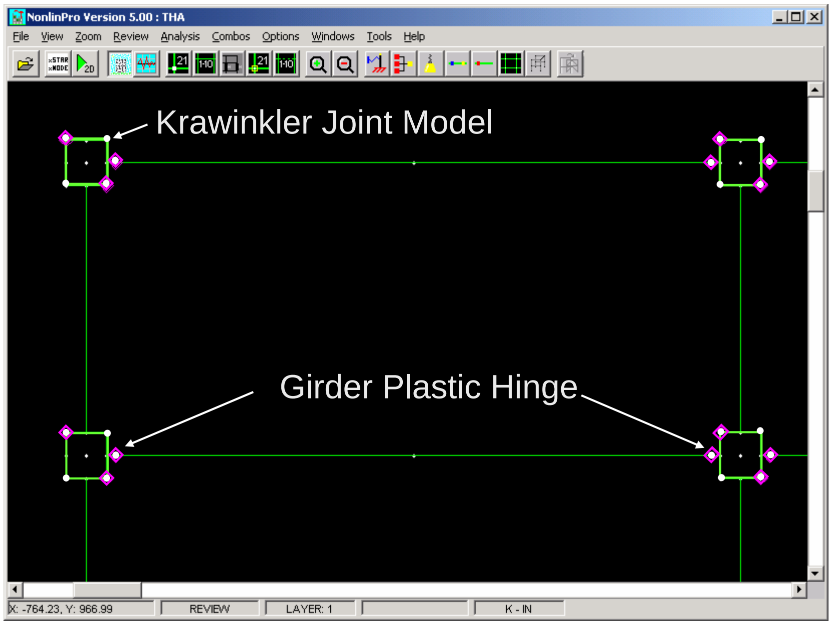
Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 34

When modeling beams it is preferable to use a concentrated plastic hinge. This is done using rotational connection elements in DRAIN, and is done automatically in PERFORM. The key point here is that 100% of the inelastic rotation is assumed to occur in the rotational plastic hinges. The initial stiffness of the rotational spring should be set to a large number (say 1000*4EI/L).

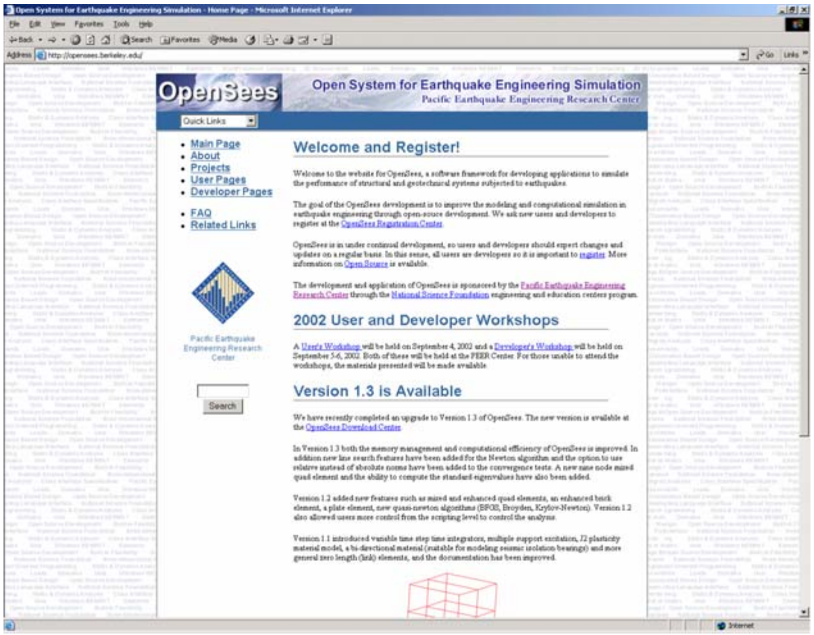
Note that DRAIN does not have the capability to model loss of strength after first yielding. RAM-Perform does. This capability is only important for modeling existing buildings that are not expected to perform well.

Girder and Joint Modeling in NONLIN-Pro



This slide is an image capture from the NONLIN-Pro program. It shows the modeling of beam-column joints using the Krawinkler model and Girder Hinges.

The OpenSees Computational Environment



Welcome and Register!

Welcome to the website for OpenSees, 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 earthquake engineering through open-source development. We ask new users and developers to register at the OpenSees Registration Center.

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

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

2002 User and Developer Workshops

A User's Workshop will be held on September 4, 2002 and a Developer's Workshop will be held on September 5-6, 2002. Both of these will be held at the PEER Center. For those unable to attend the workshops, 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 is 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 nine node mixed quad element and the ability to compute the standard eigenvalues have also been added.

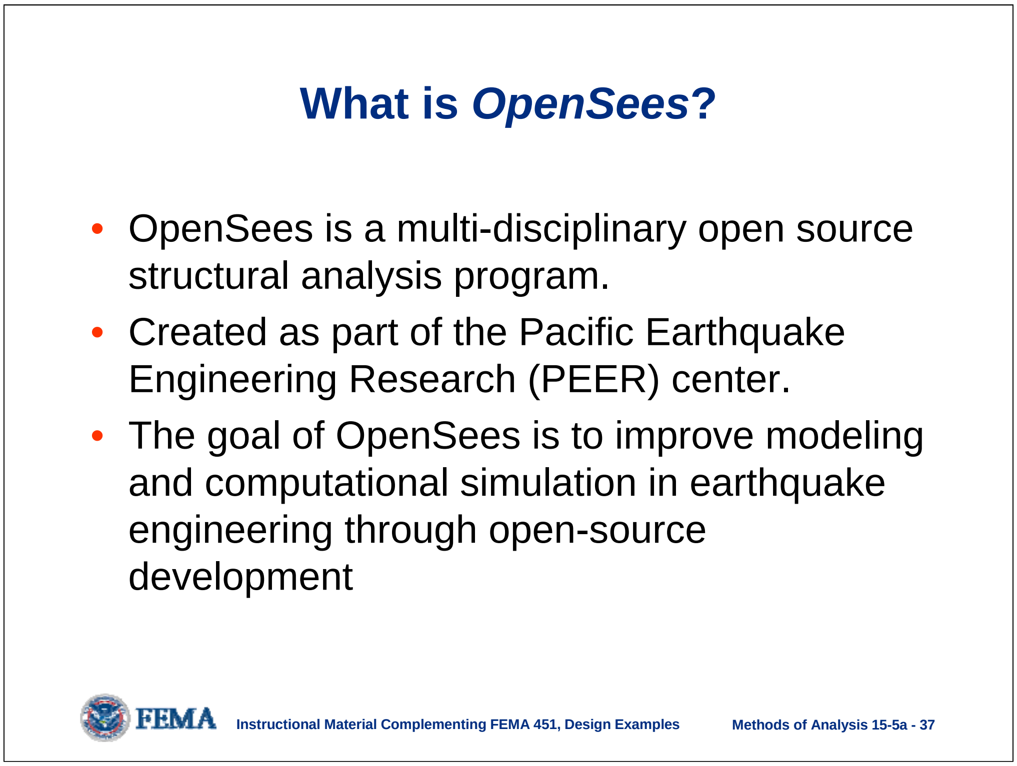
Version 1.2 added new features such as mixed and enhanced quad elements, an enhanced brick element, a plate element, new quasi-newton algorithms (BFGS, Broyden, Krylov-Newton). Version 1.2 also allowed users more control from the scripting level to control the analysis.

Version 1.1 introduced variable time step time integrators, multiple support excitation, J2 plasticity material model, a bi-directional material (suitable for modeling seismic isolation bearings) and more general zero length (link) elements, and the documentation has been improved.

FEMA Instructional Material Complementing FEMA 451, Design Examples Methods of Analysis 15-5a - 36

The program that may ultimately replace DRAIN is the OpenSees environment being developed at PEER (primarily Professor Fenves at Berkeley and Professor Deierlein at Stanford). This is an open-source object oriented C++ code. Getting into the “guts” of the program is not for the timid (even though the OpenSees web site gives a pretty good description of the development environment). The web site has a complete users manual and several examples. Pre-and post-processing still leave a lot to be desired.

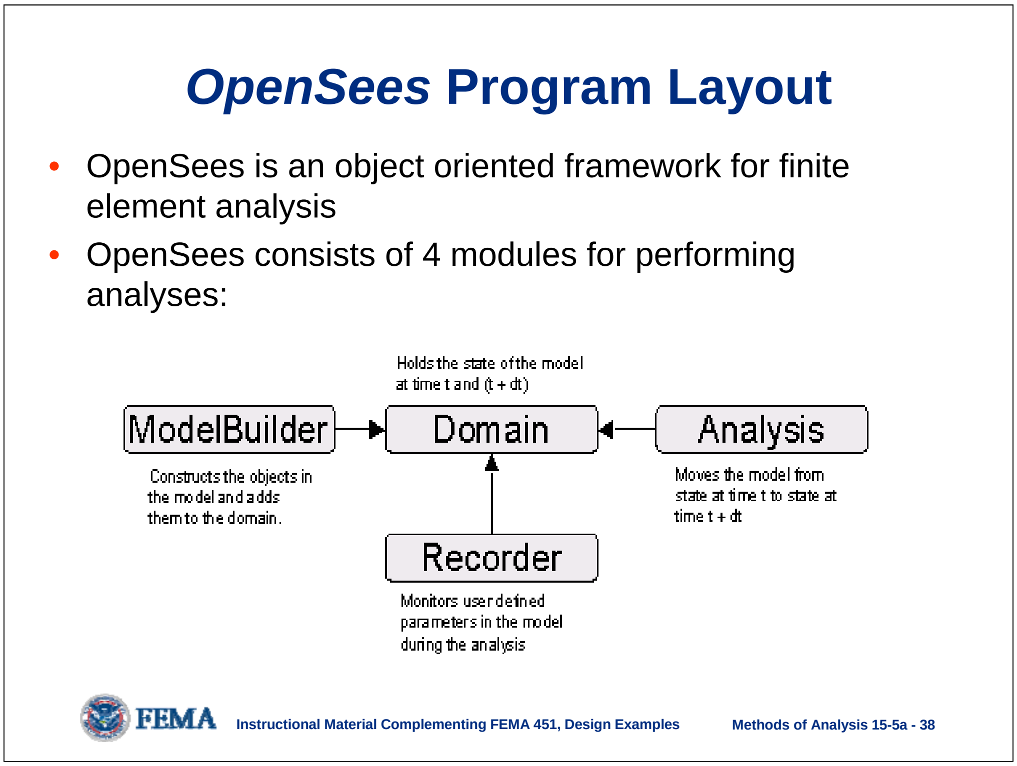
The NEES equipment sites that utilize hybrid resting (e.g. UC Boulder) may rely on OpenSees as the analytical counterpart to the physical testing equipment.



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

It is envisioned that OpenSees will replace DRAIN-2Dx as the analysis method of choice among researchers. This will happen with time, but the program has a way to go before it can be readily accepted.



OpenSees Program Layout

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

Holds the state of the model at time t and (t + dt)

ModelBuilder

Domain

Analysis

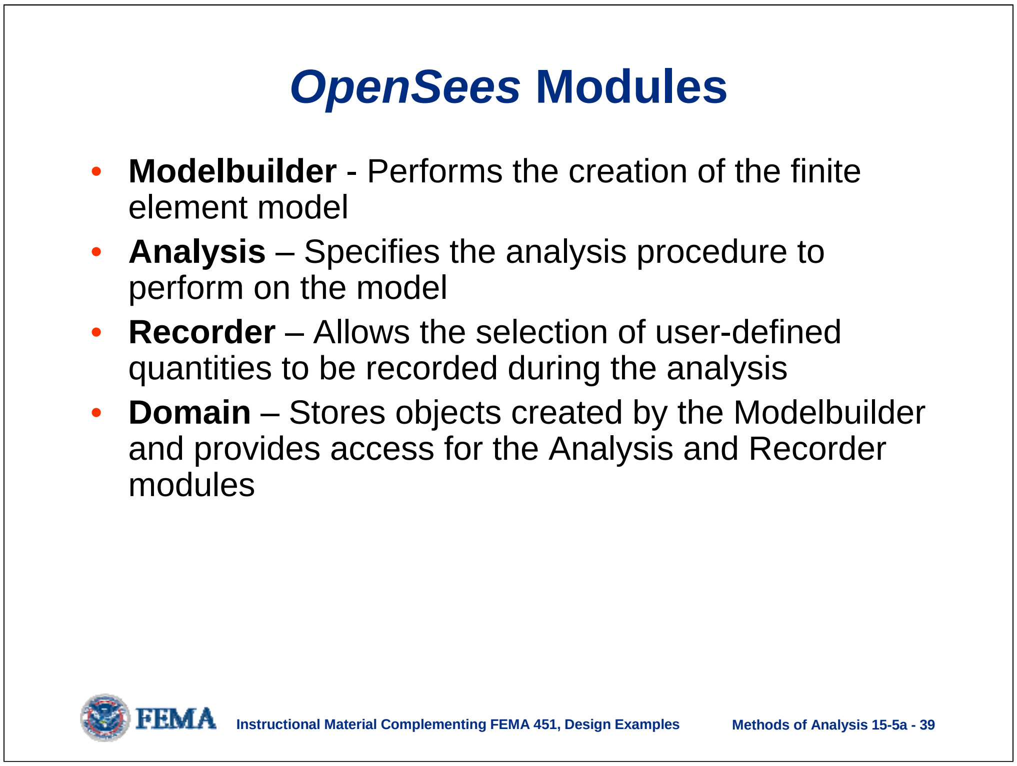
Constructs the objects in the model and adds them to the domain.

Moves the model from state at time t to state at time t + dt

Recorder

Monitors user defined parameters in the model during the analysis

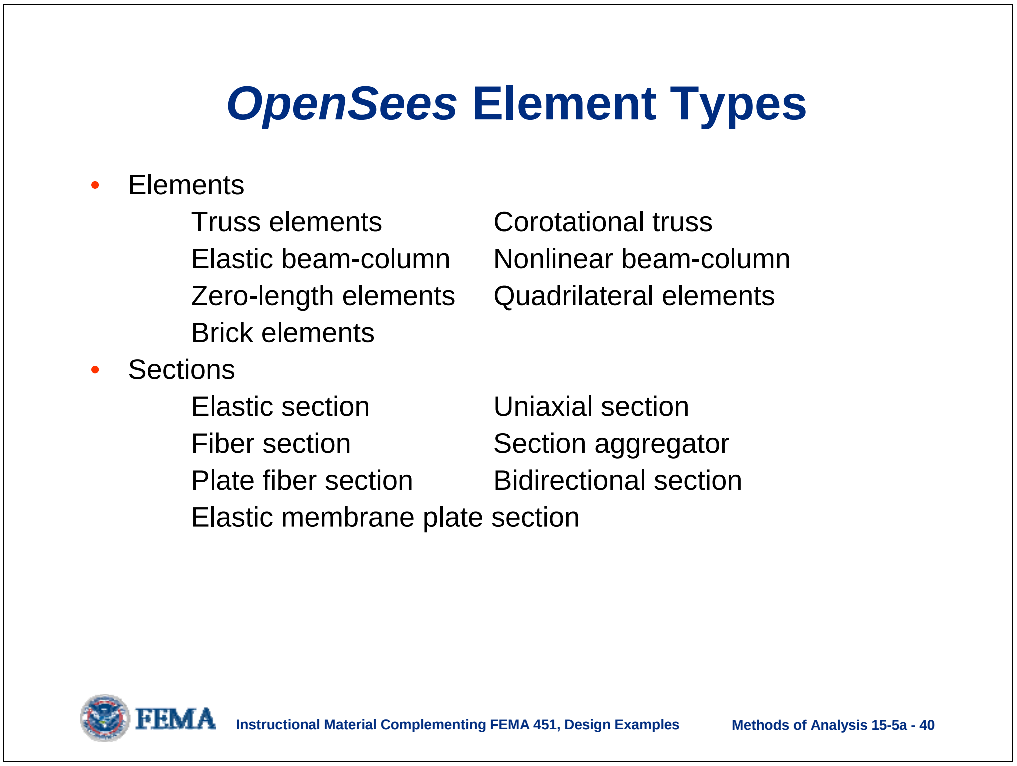
One of the strengths of OpenSees is also a weakness. It is written in object oriented C++ which is good from the original programmer's prospective, but is not so good for the typical engineer that would like to get his or her hands into the code. Few engineers master C++, not to mention the object oriented programming concepts necessary for contributing to the OpenSees project.



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

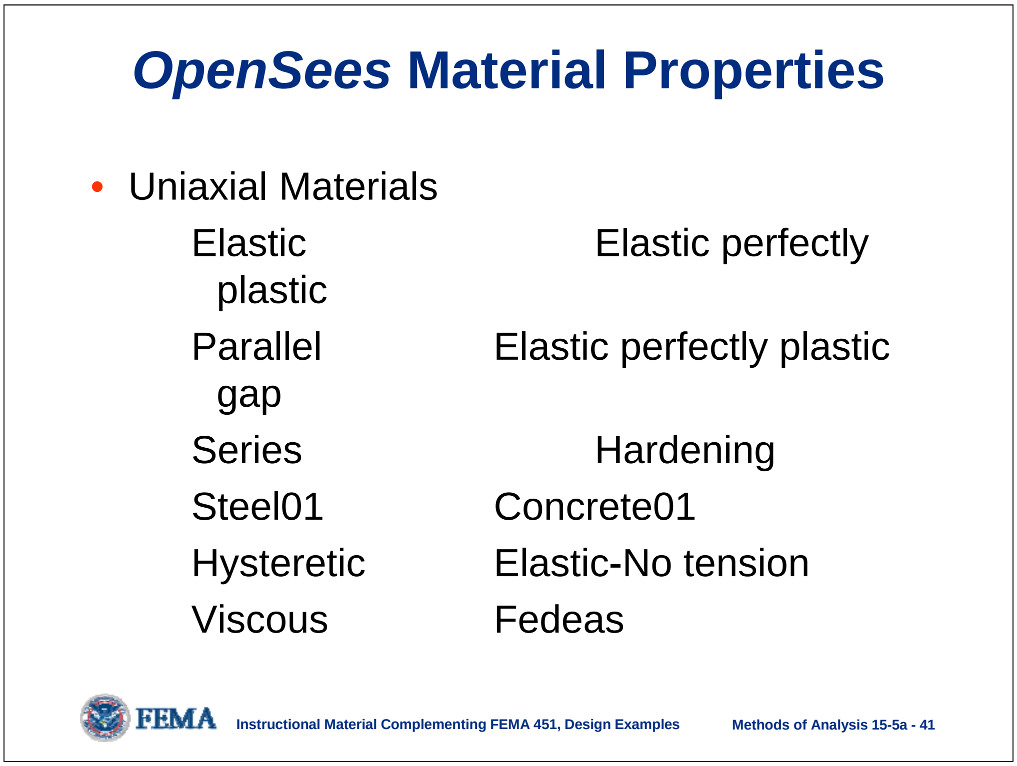
No annotation.



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

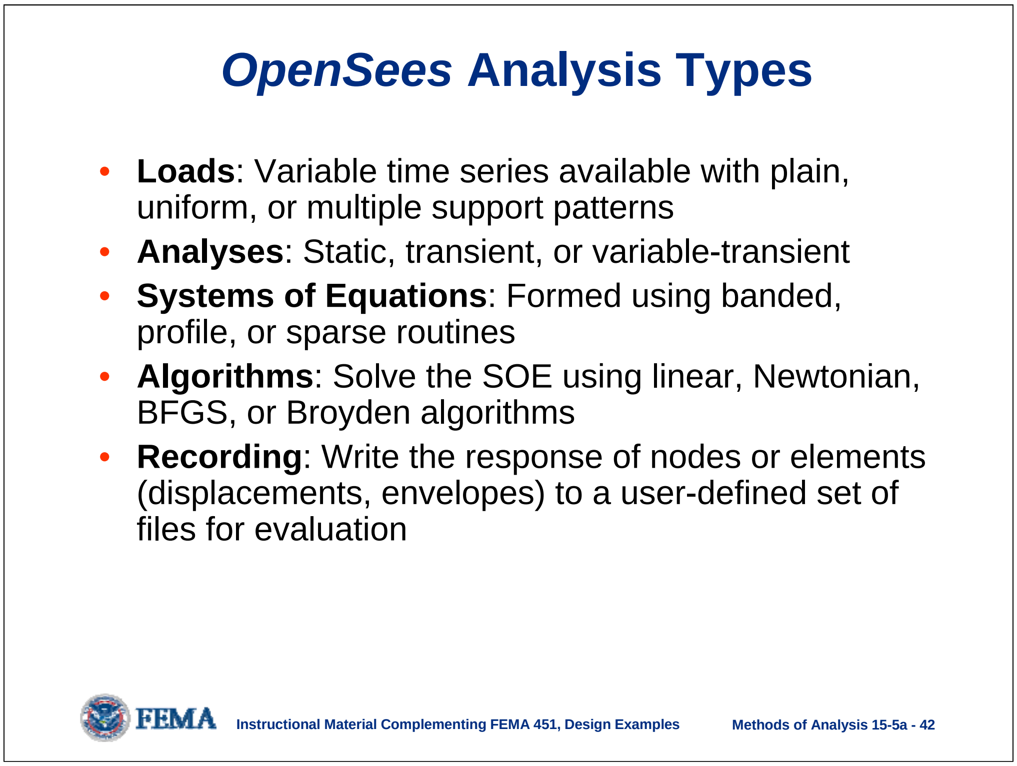
No annotation.



OpenSees Material Properties

- Uniaxial Materials
  - Elastic
  - Parallel
  - Series
  - Steel01
  - Hysteretic
  - Viscous
  - Elastic perfectly plastic
  - Elastic perfectly plastic gap
  - Hardening
  - Concrete01
  - Elastic-No tension
  - Fedeas

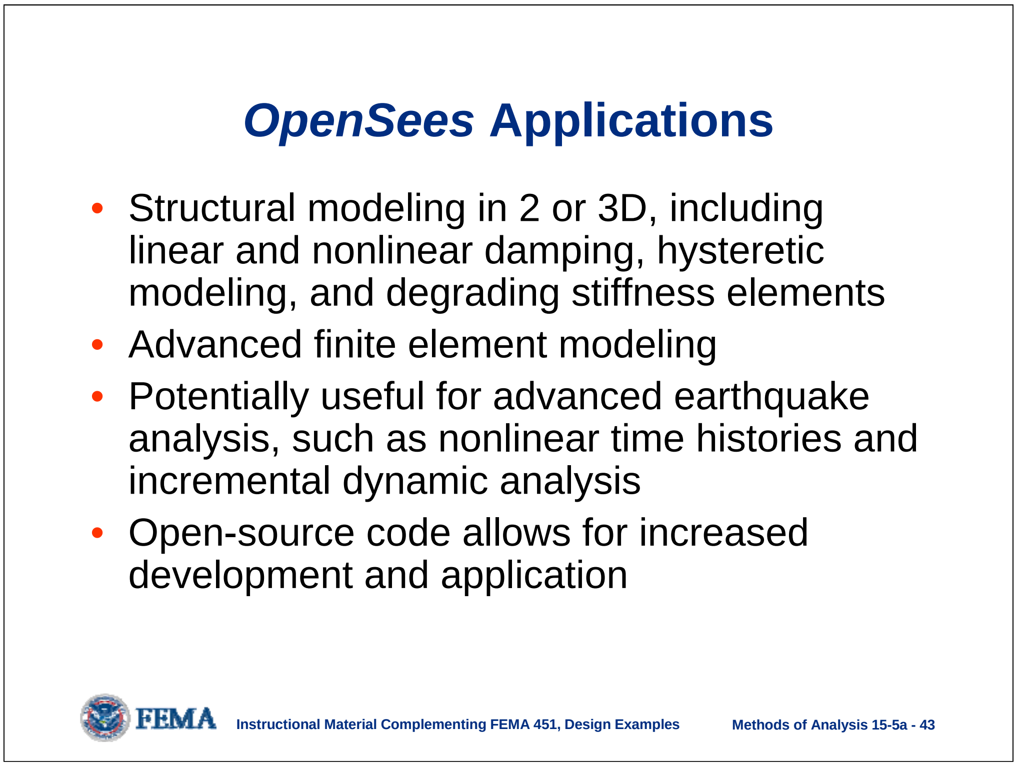
No annotation.



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

One of the strengths of OpenSees is the large variety of solvers available.



OpenSees Applications

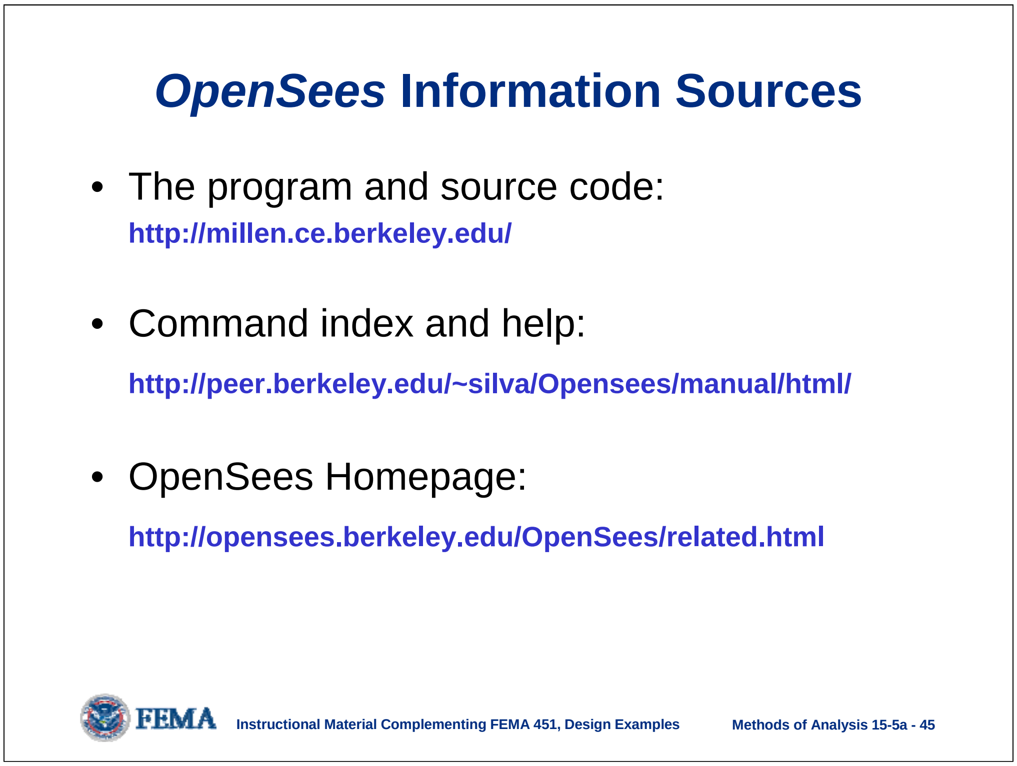
- Structural modeling in 2 or 3D, including linear and nonlinear damping, hysteretic modeling, and degrading stiffness elements
- Advanced finite element modeling
- Potentially useful for advanced earthquake analysis, such as nonlinear time histories and incremental dynamic analysis
- Open-source code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

No annotation.

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.

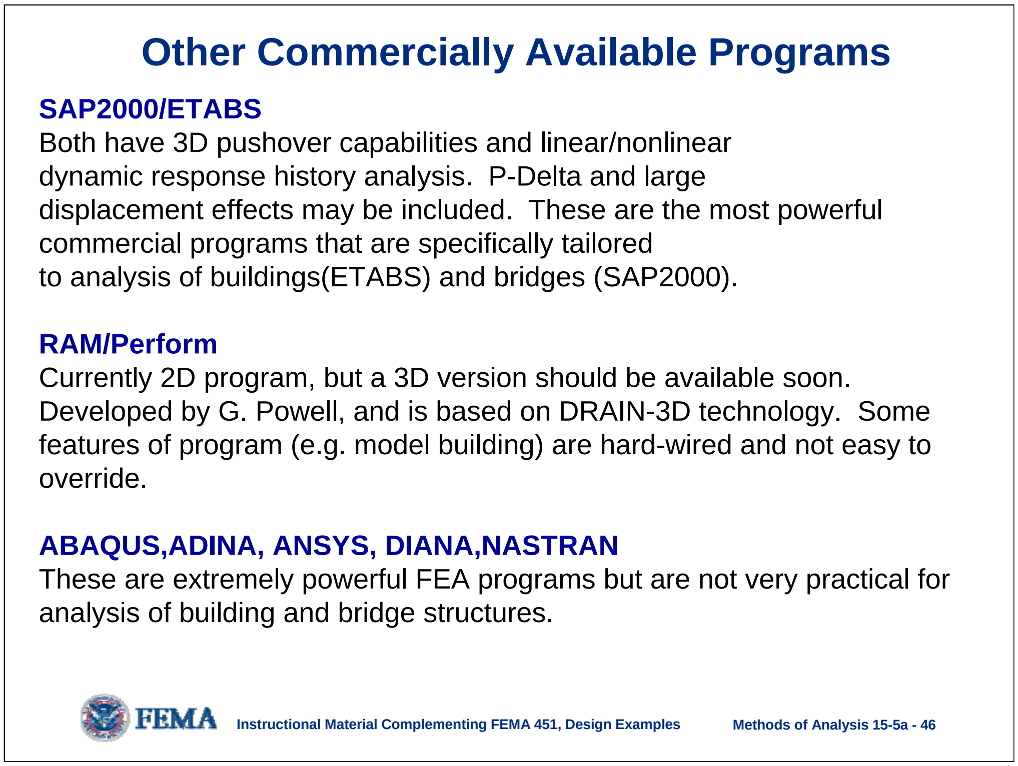
No annotation.



OpenSees Information Sources

- The program and source code:
  http://millen.ce.berkeley.edu/
- Command index and help:
  http://peer.berkeley.edu/~silva/Opensees/manual/html/
- OpenSees Homepage:
  http://opensees.berkeley.edu/OpenSees/related.html

No annotation. Provided for student reference only.



Other Commercially Available Programs

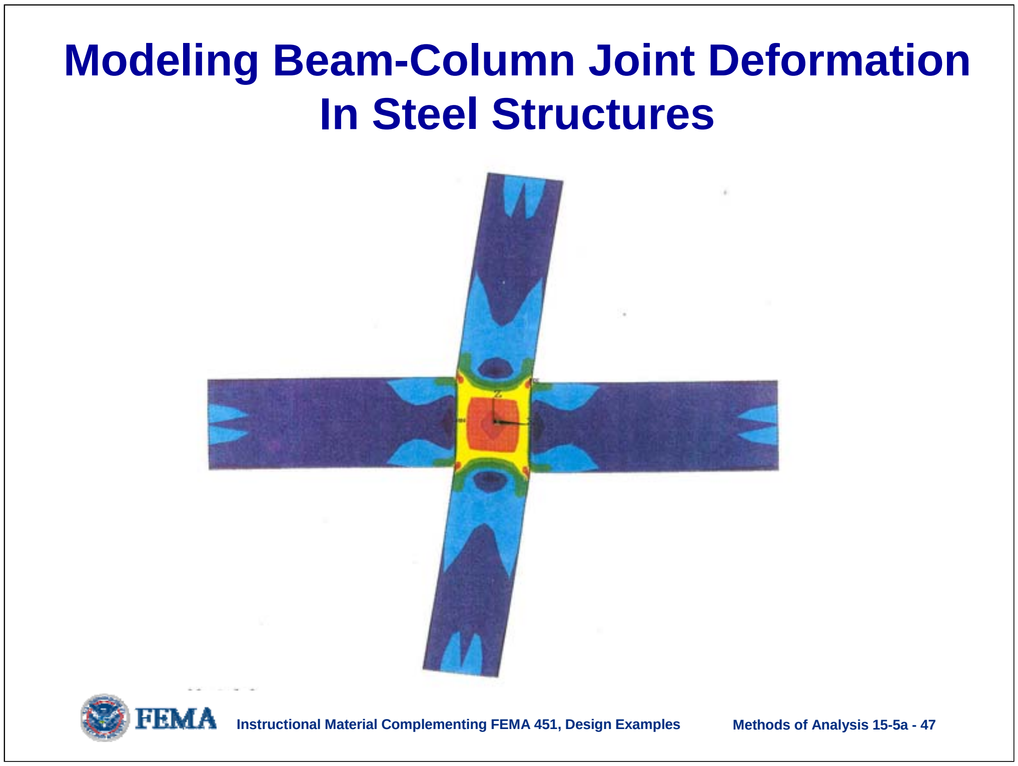
SAP2000/ETABS
Both have 3D pushover capabilities and linear/nonlinear dynamic response history analysis. P-Delta and large displacement effects may be included. These are the most powerful commercial programs that are specifically tailored to analysis of buildings(ETABS) and bridges (SAP2000).

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.

SAP2000 has grown tremendously in power with the release of Version 8. The most important new capabilities include full nonlinear dynamic response history analysis and large displacement effects.

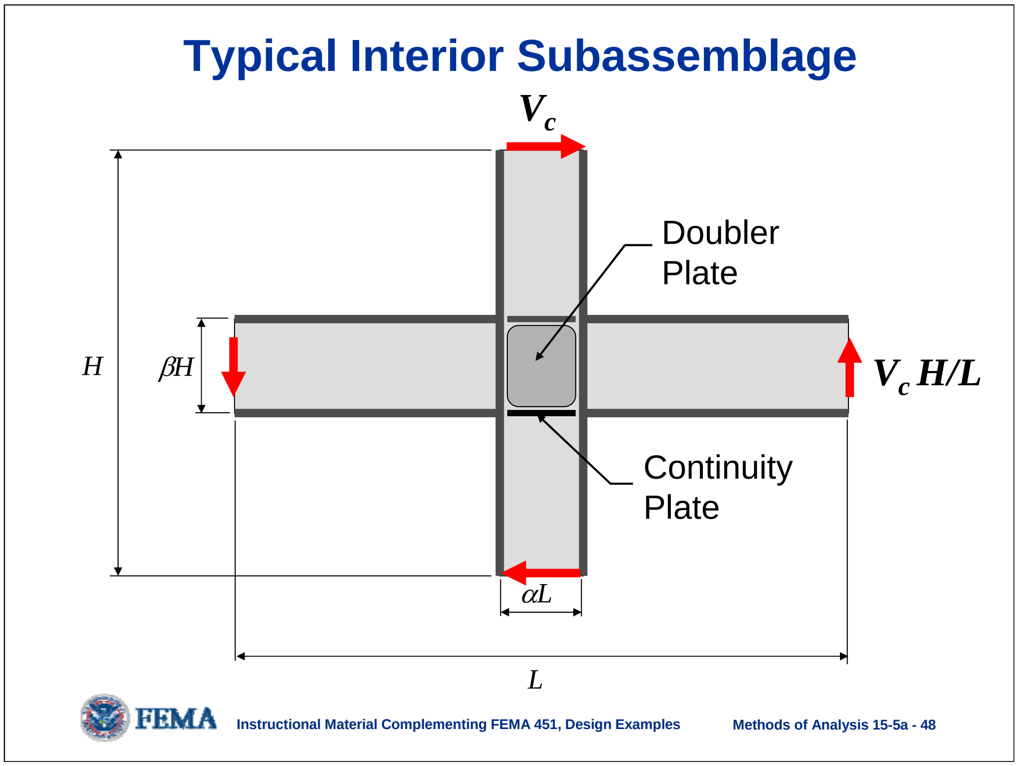
Programs like ABAQUS are very powerful, but typically do not have capabilities to easily model building and bridge type structures. Deficiencies include lack of standard section databases, lack of phenomenological models, inability to conveniently apply ground motions, and inability to apply load combinations. Also, these programs are very expensive (ABAQUS is $25K per year), and VERY HARD TO LEARN.



Modeling Beam-Column Joint Deformation In Steel Structures

FEMA Instructional Material Complementing FEMA 451, Design Examples Methods of Analysis 15-5a - 47

Modeling of the beam-column joint region in steel moment frames is presented in the next several slides. The image shown on this slide is an image from ANSYS showing the shear stresses in a typical subassemblage. Note the very high shear stress in the panel zone region. Such stresses and associated strains may be responsible for as much as 40 percent of the total drift in steel frame structures.



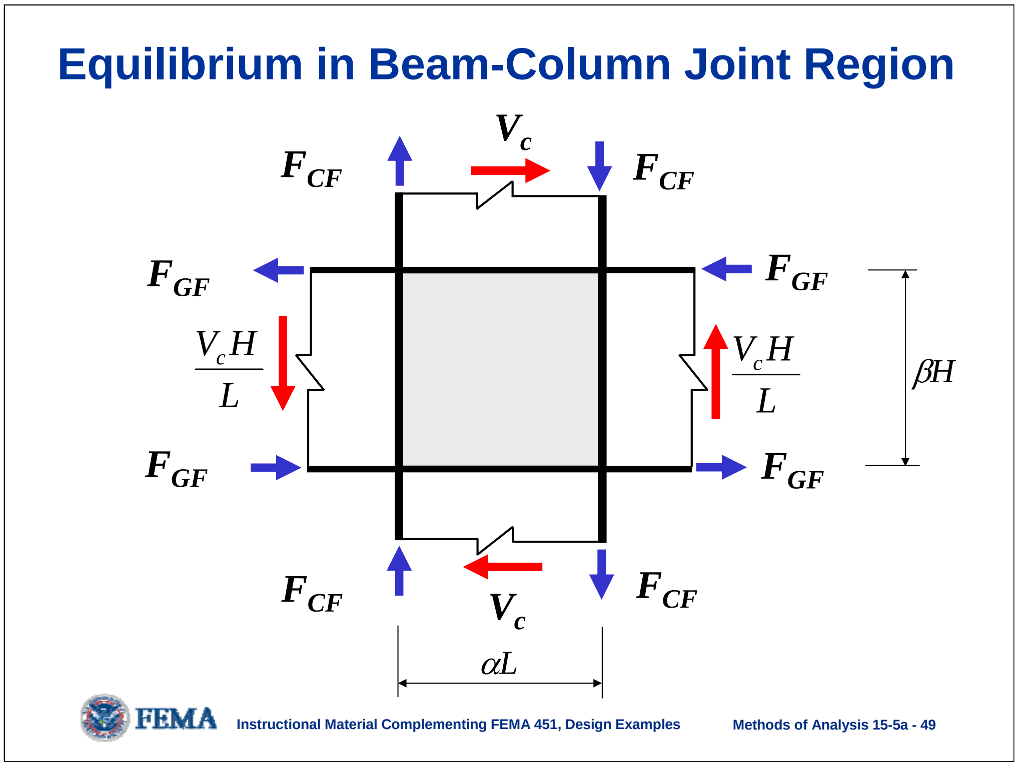
Typical Interior Subassemblage

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 48

This is a typical interior subassemblage. Note the dimensionless terms alpha and beta. The effective girder and column depth is taken as the distance between flanges.

Note that the columns always pass through the floor, and that the continuity plates are almost always present. Current AISC seismic provisions call for a strong panel zone, so the doubler plate will often be present. Such plates are extremely effective in reducing beam-column joint deformations. Unfortunately the cutting and welding of the plates is very expensive.

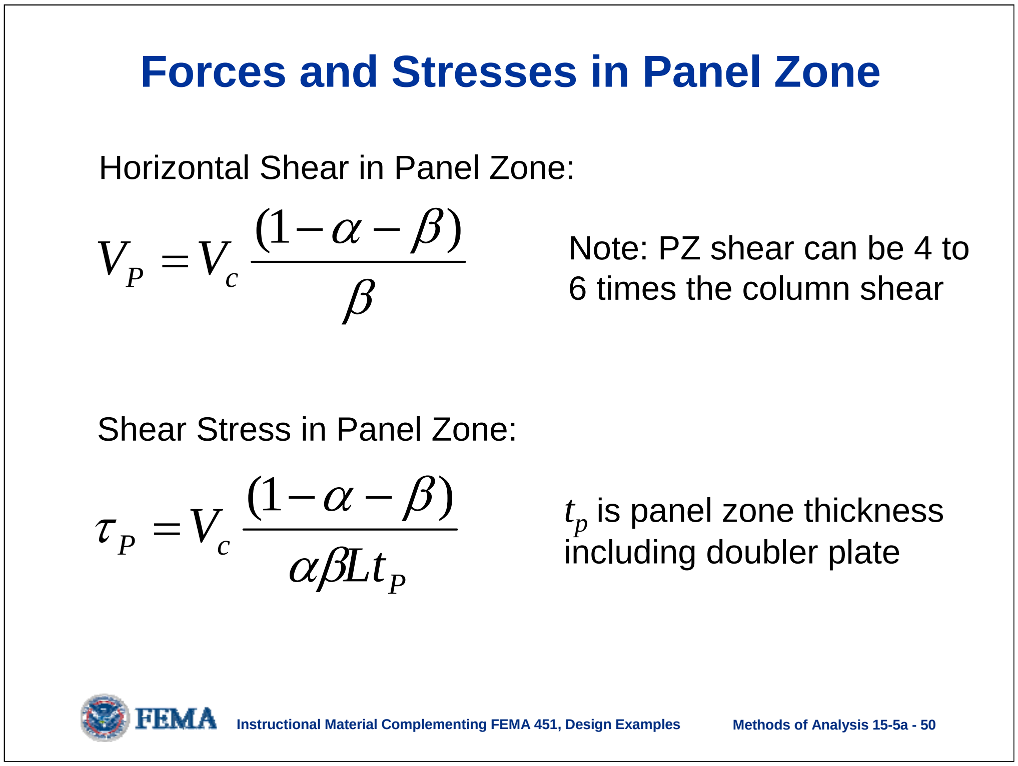


Equilibrium in Beam-Column Joint Region

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 49

Here the forces on the joint are determined. The main simplifying assumption is that the girder and column moments may be represented by a couple with all of the moment being resisted by flange forces.



Forces and Stresses in Panel Zone

Horizontal Shear in Panel Zone:

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

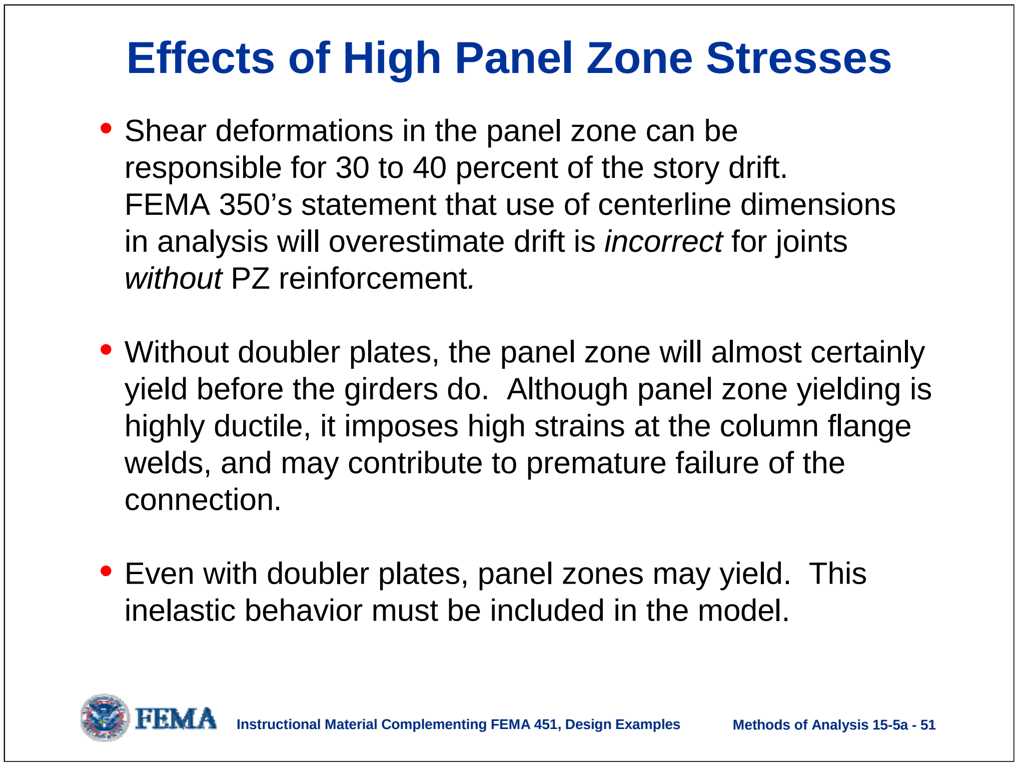
Note: PZ shear can be 4 to 6 times the column shear

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

The shear force in the panel zone is given by the upper equation. It is easy to see that the shear force in the joint may be several times the shear force in the column above and below the joint.

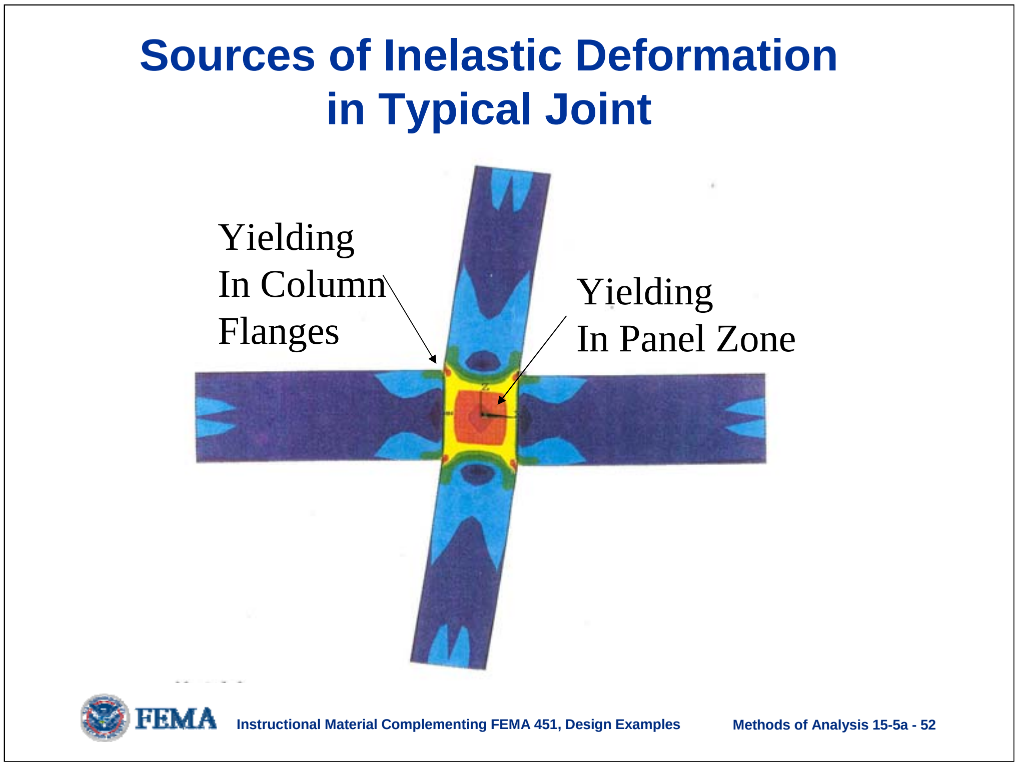


Effects of High Panel Zone Stresses

- 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.

Instructional Material Complementing FEMA 451, Design Examples **Methods of Analysis 15-5a - 51**

Points are self explanatory.

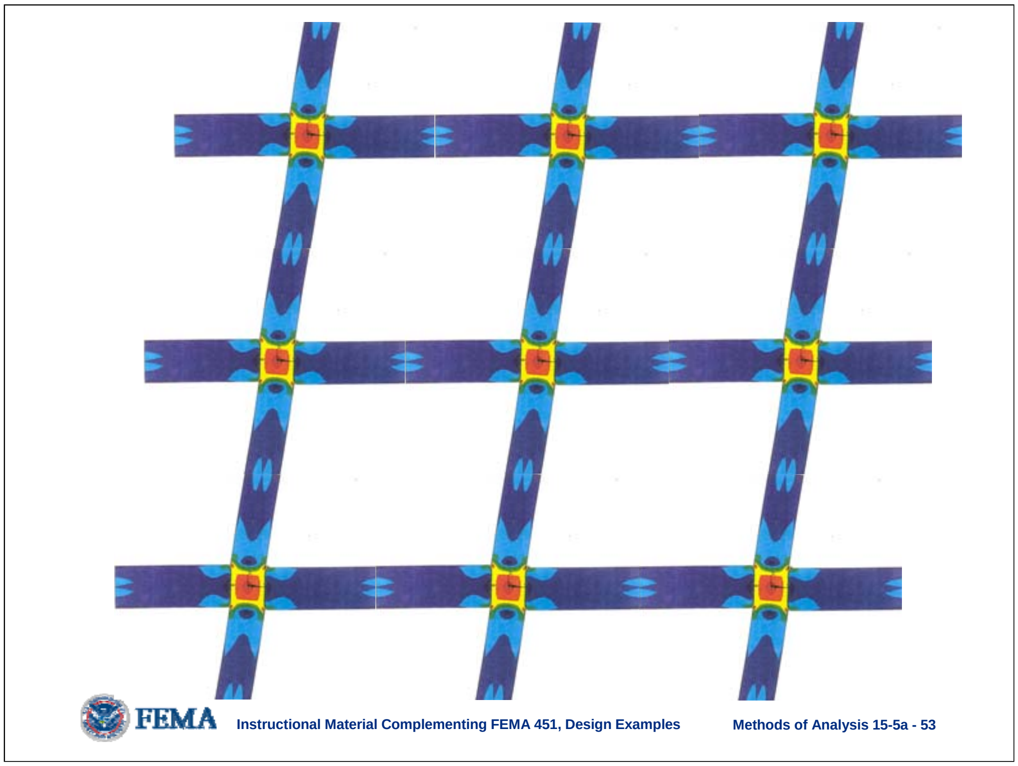


Sources of Inelastic Deformation in Typical Joint

Yielding In Column Flanges

Yielding In Panel Zone

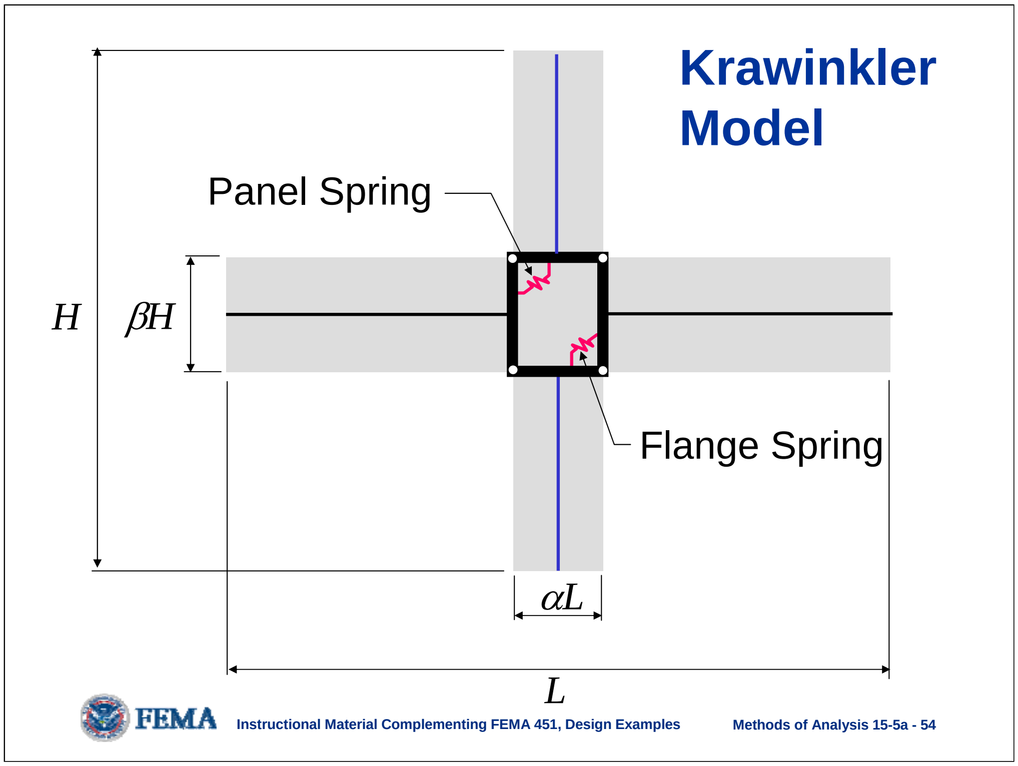
ANSYS results illustrate the previous points.



Instructional Material Complementing FEMA 451, Design Examples

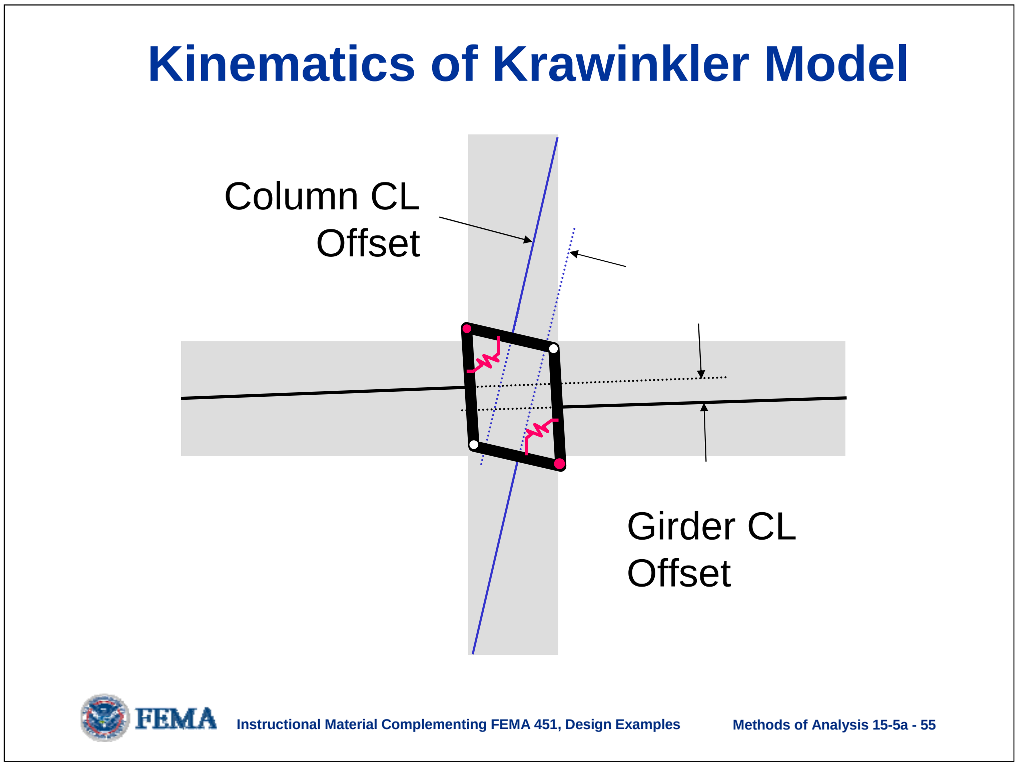
Methods of Analysis 15-5a - 53

This slide is a composite of the ANSYS results of a single subassemblage. Note that the dominate source of deformation in this frame is shear deformation in the panel zones and in the webs of the beams.



Krawinkler Model

One of the best idealizations for beam-column behavior is the model developed by Krawinkler. The basic model consists of four links which frame the joint. The links are connected at the corners by true hinges or by rotational springs. The stiffness and strength of the joint is represented by these springs.



Kinematics of Krawinkler Model

Column CL Offset

Girder CL Offset

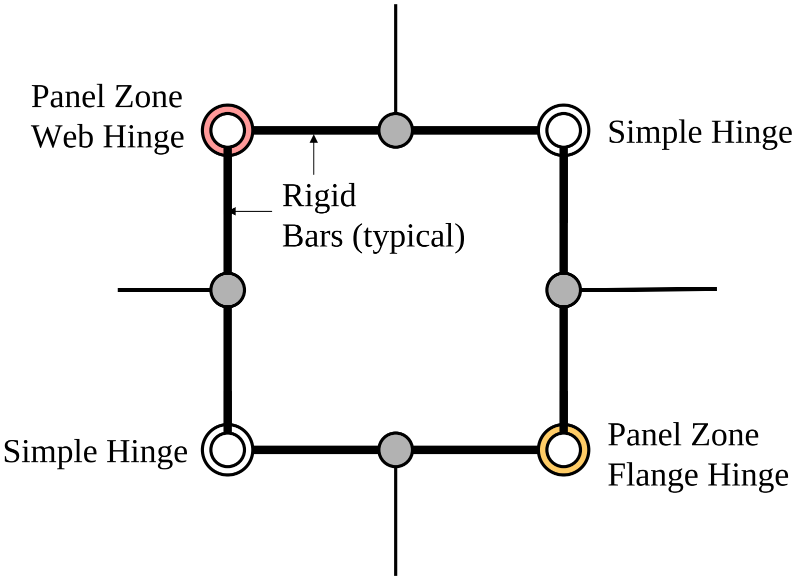
FEMA

Instructional Material Complementing FEMA 451, Design Examples

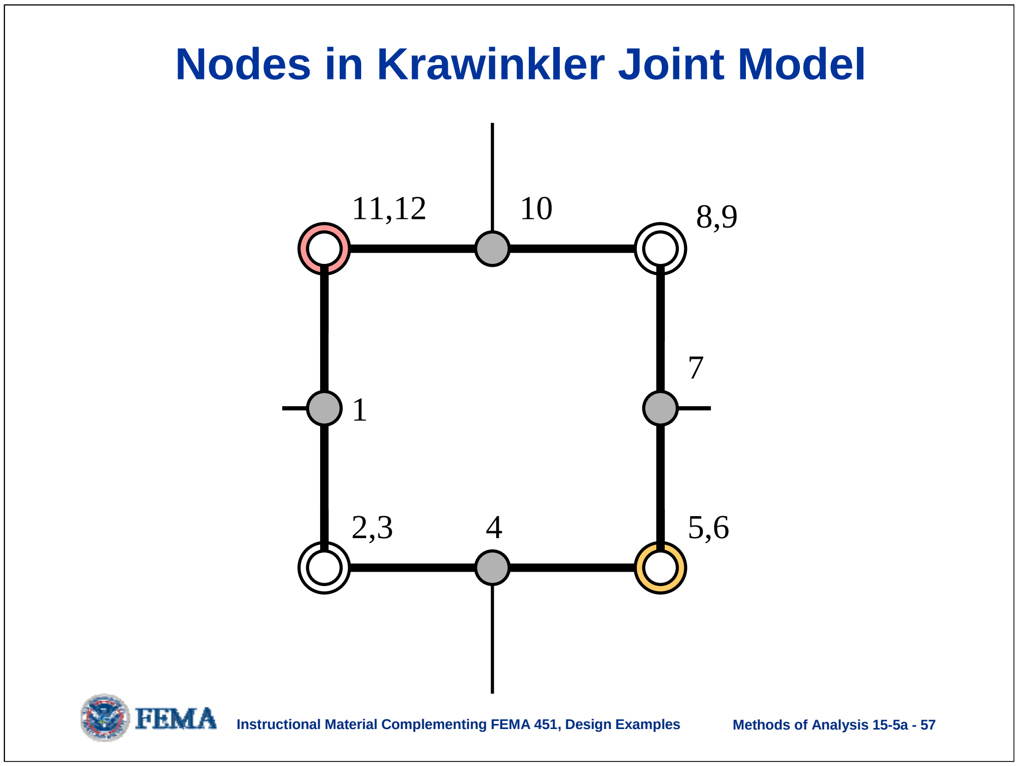
Methods of Analysis 15-5a - 55

This illustration shows the Krawinkler model in its deformed state (with the beams and columns remaining relatively rigid). Note how the joint “rotates” in the opposite direction than would be expected. Note also that significant “offsets” occur in the centerlines of the columns and girders. As mentioned later, this kinematic effect does not occur in the simpler “Scissors Model”.

Krawinkler Joint Model

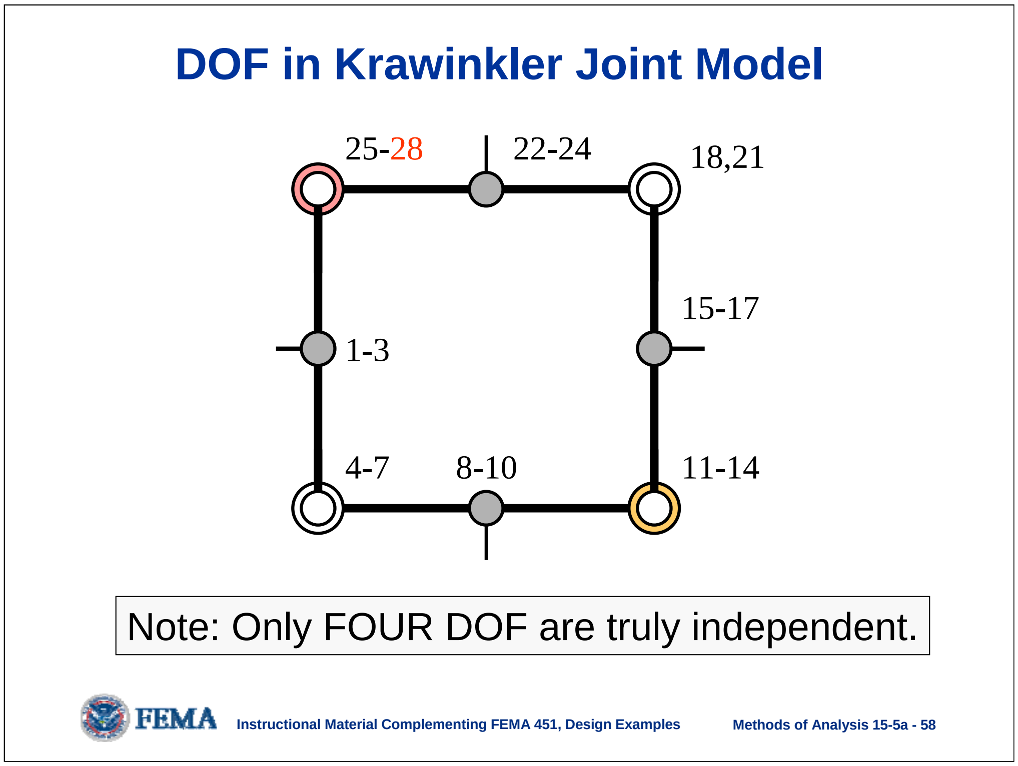


The Krawinkler Model is a phenomenological model of a beam column joint. When modeled in DRAIN it consists of a “frame” of Type-2 beam-column elements connected at the four corners by compound nodes. The upper left compound node utilizes a rotational Type 4 spring to represent the panel zone web stiffness and strength. The lower right compound node utilizes a Type-4 rotational spring to represent column flange contributions. The other two compound nodes are simple flexural hinges.



Nodes in Krawinkler Joint Model

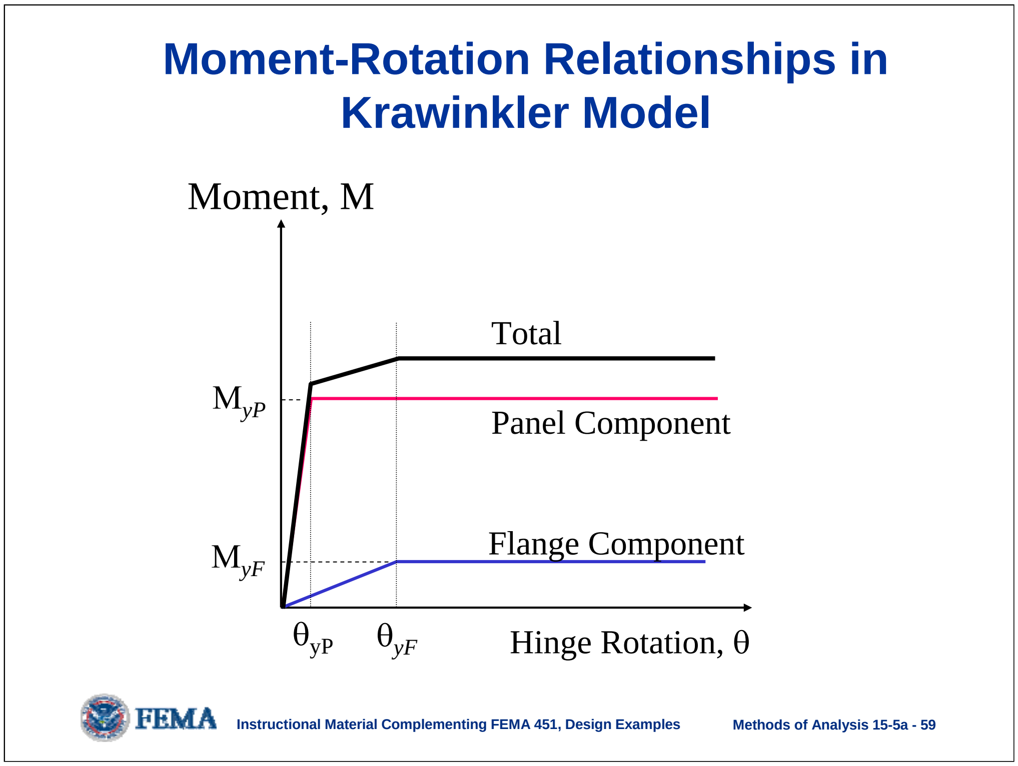
It takes twelve nodes to represent a single Krawinkler joint. Note that the corners each contain two nodes that have constrained X-Y degrees of freedom and independent rotational degrees of freedom.



DOF in Krawinkler Joint Model

Note: Only FOUR DOF are truly independent.

If no constraints are used the Krawinkler model has 28 degrees of freedom. However, only four of the degrees of freedom are truly independent. These degrees of freedom are rigid body X and Y translation, rigid body rotation, and racking. Unfortunately, DRAIN makes it difficult to impose the constraints required to minimize the number of degrees of freedom. Fortunately. experience has shown that reasonable solution times can be obtained in response history analysis of multiple story-multiple bay frames.

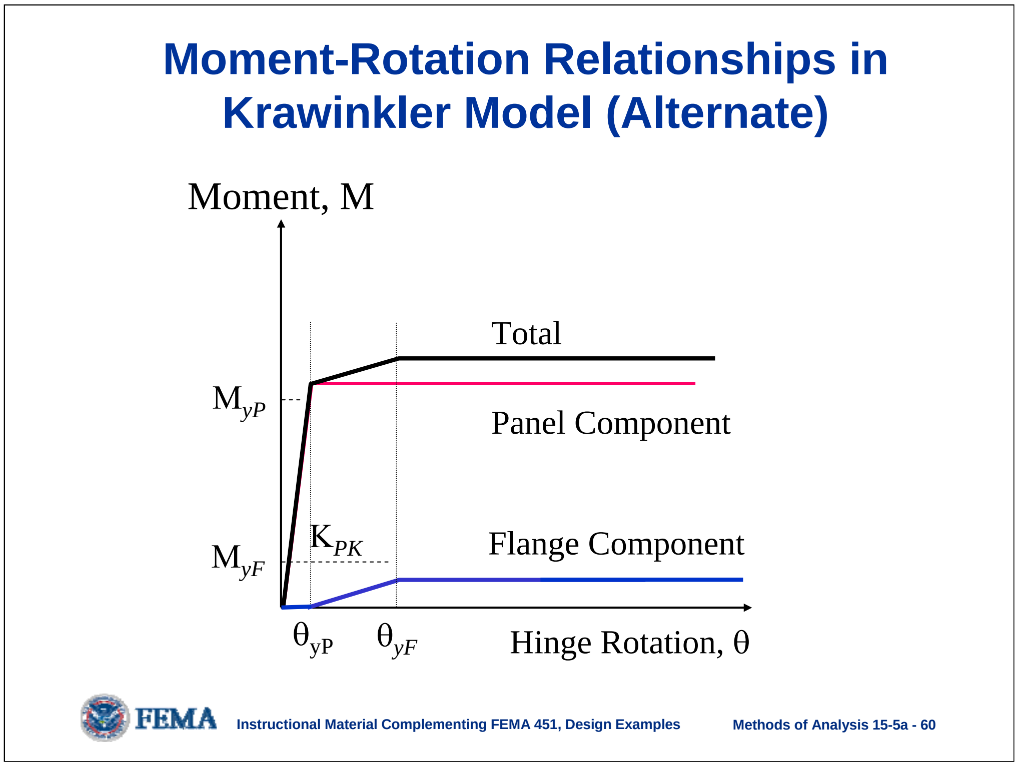


Moment-Rotation Relationships in Krawinkler Model

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 59

This slide shows the simple moment-rotation relationships for the two rotational springs in the Krawinkler Model. The Panel Component (shown in red) represents the stiffness and strength of the panel zone, including the doubler plate if present. The Flange component (shown in blue) arises from the eventual formation of plastic hinges in the flanges of the columns the panel tries to rack. In the model shown here the flange component contributes to the initial stiffness of the joint. On the basis of test results it is typically assumed that the flange components yields at four times the yield rotation of the panel component.

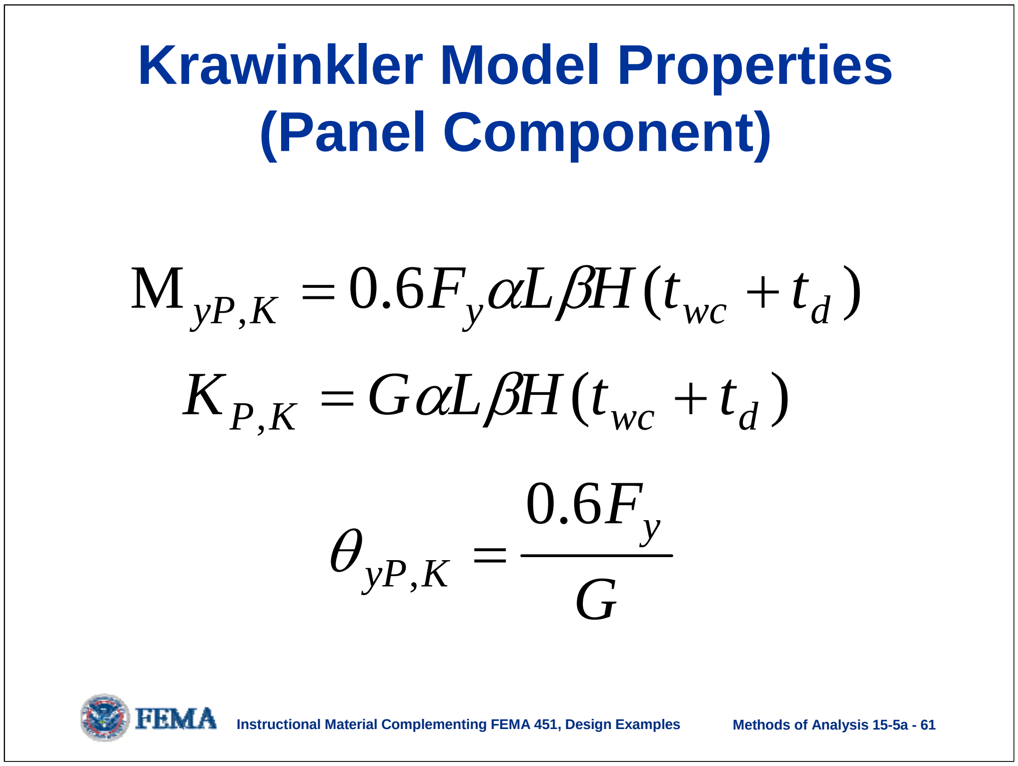


Moment-Rotation Relationships in Krawinkler Model (Alternate)

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 60

An alternate model assumes that the flange component has no initial stiffness, picking up force only after the panel component has yielded.



Krawinkler Model Properties (Panel Component)

$$\mathrm{M}_{yP,K} = 0.6F_y \alpha L \beta H (t_{wc} + t_d)$$

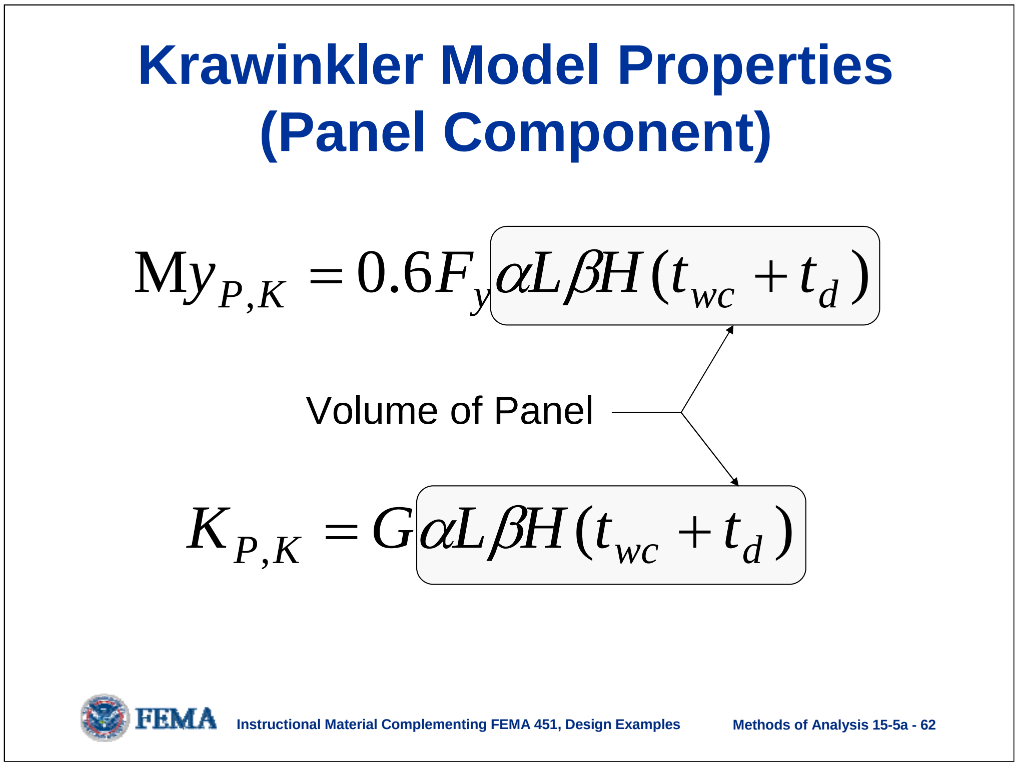
$$K_{P,K} = G \alpha L \beta H (t_{wc} + t_d)$$

$$\theta_{yP,K} = \frac{0.6F_y}{G}$$

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 61

This slide shows the required properties for the Panel Component. Note that t_{wc} and t_d are the thicknesses of the web of the column and the doubler plate, respectively.



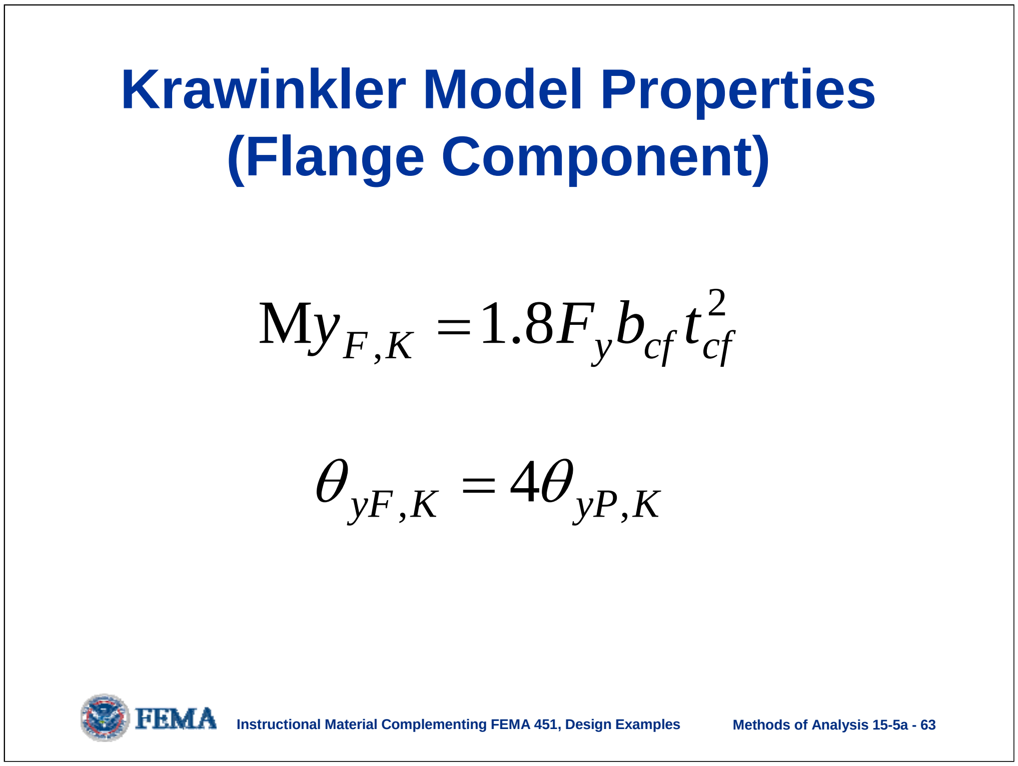
Krawinkler Model Properties (Panel Component)

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

Volume of Panel

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

It is easy to memorize the formulas for the panel component if it is recognized that the grouped terms represent the volume of the panel zone.

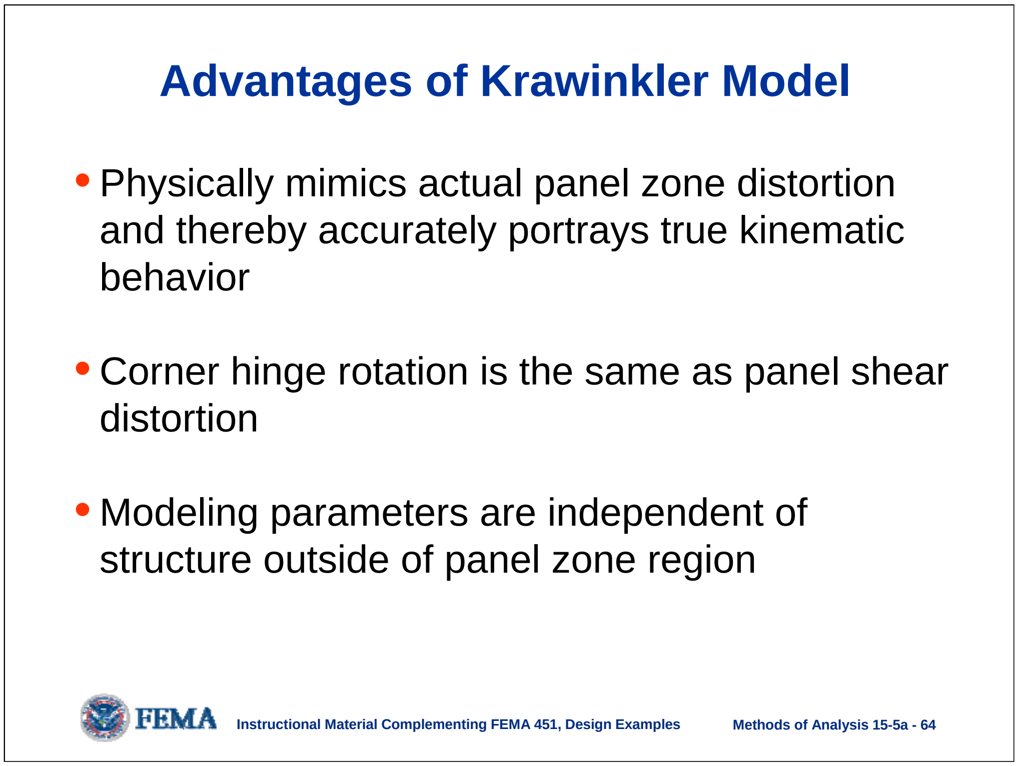


Krawinkler Model Properties (Flange Component)

$$\mathrm{M}y_{F,K} = 1.8F_y b_{cf} t_{cf}^2$$

$$\theta_{yF,K} = 4\theta_{yP,K}$$

Here the properties of the rotational spring used to represent the flange component are shown. The stiffness of this component is back-calculated from the rotation relationship.



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

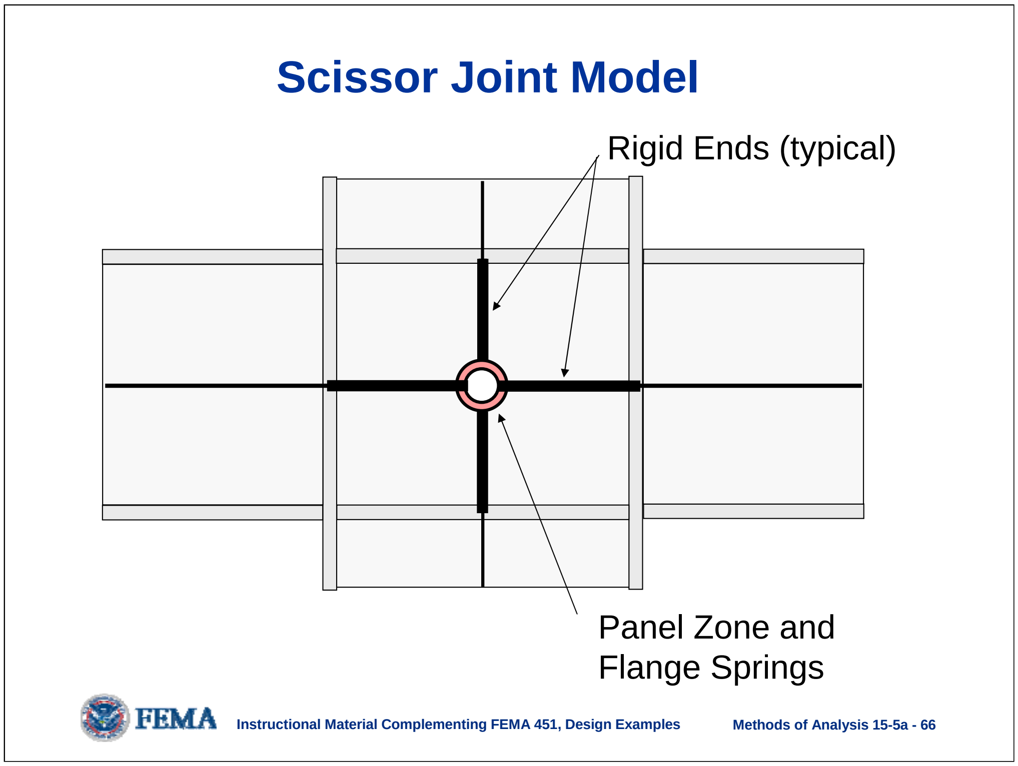
This slide lists the main advantages of the Krawinkler Model.

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.

This slide lists the main disadvantages of the Krawinkler Model.

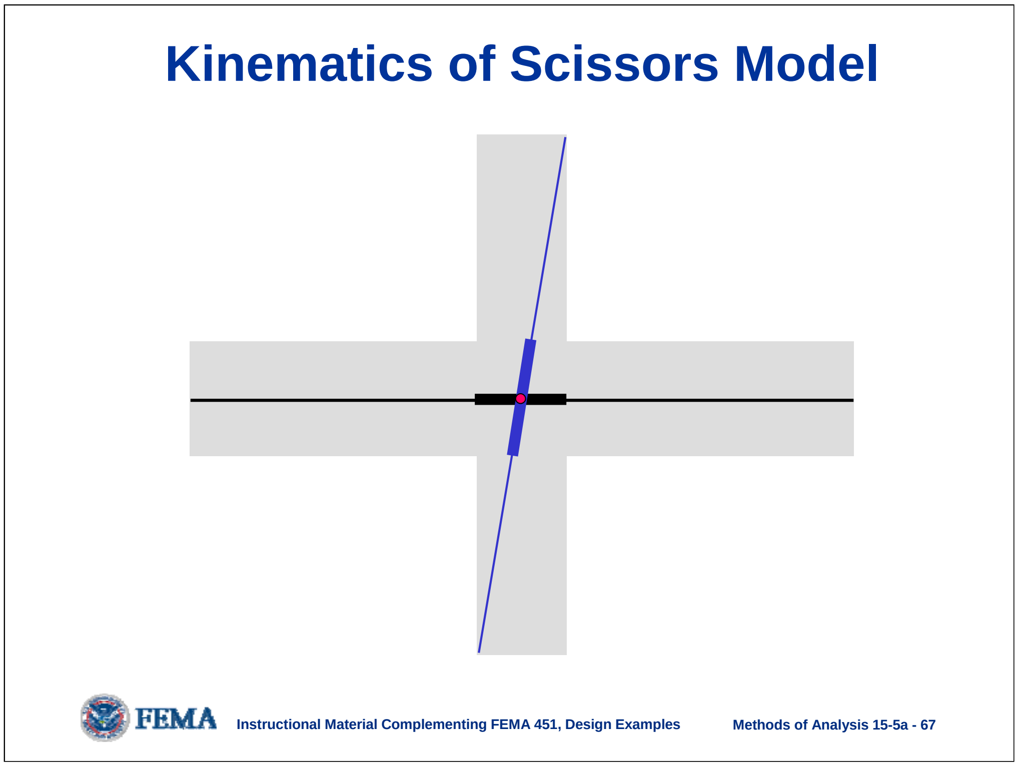


Scissor Joint Model

Rigid Ends (typical)

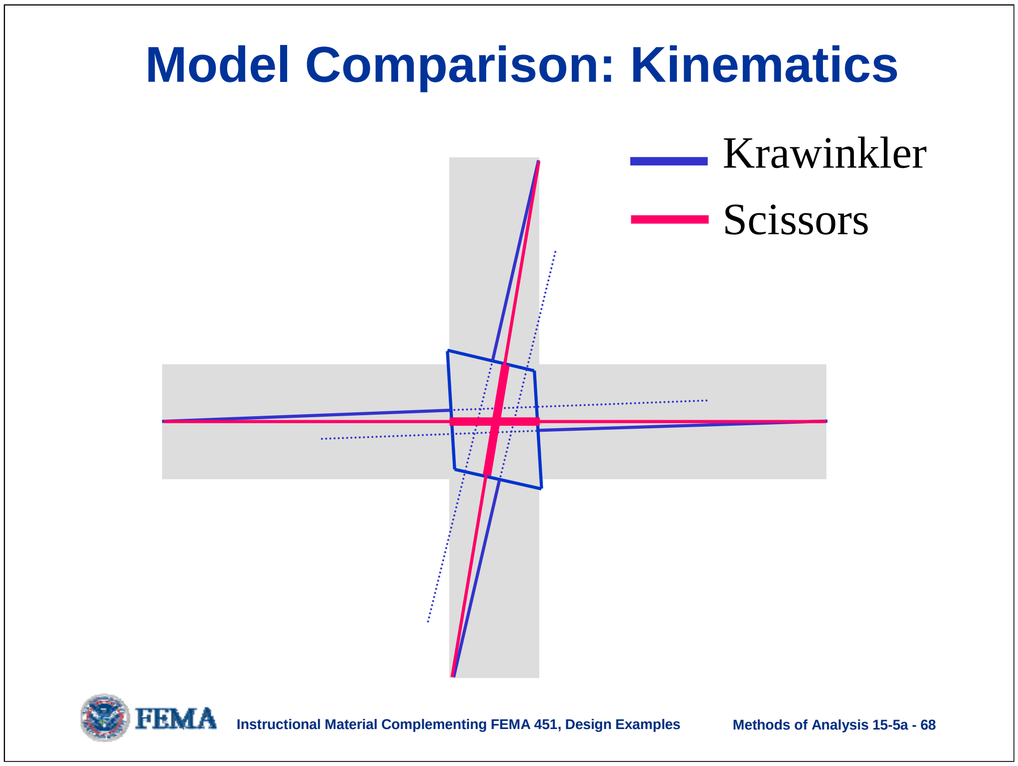
Panel Zone and
Flange Springs

An attractive alternative to the Krawinkler Model is the so-called Scissors Model. Here, a single compound node is used to represent the panel zone. The rigid end zones are extensions of the beams and columns that frame into the joint. A pair of rotational springs are used to represent panel zone and column flange effects.



Kinematics of Scissors Model

The kinematics of the Scissors Model is quite different that that of the Krawinkler Model. This is seen more clearly on the next slide.



Model Comparison: Kinematics

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 68

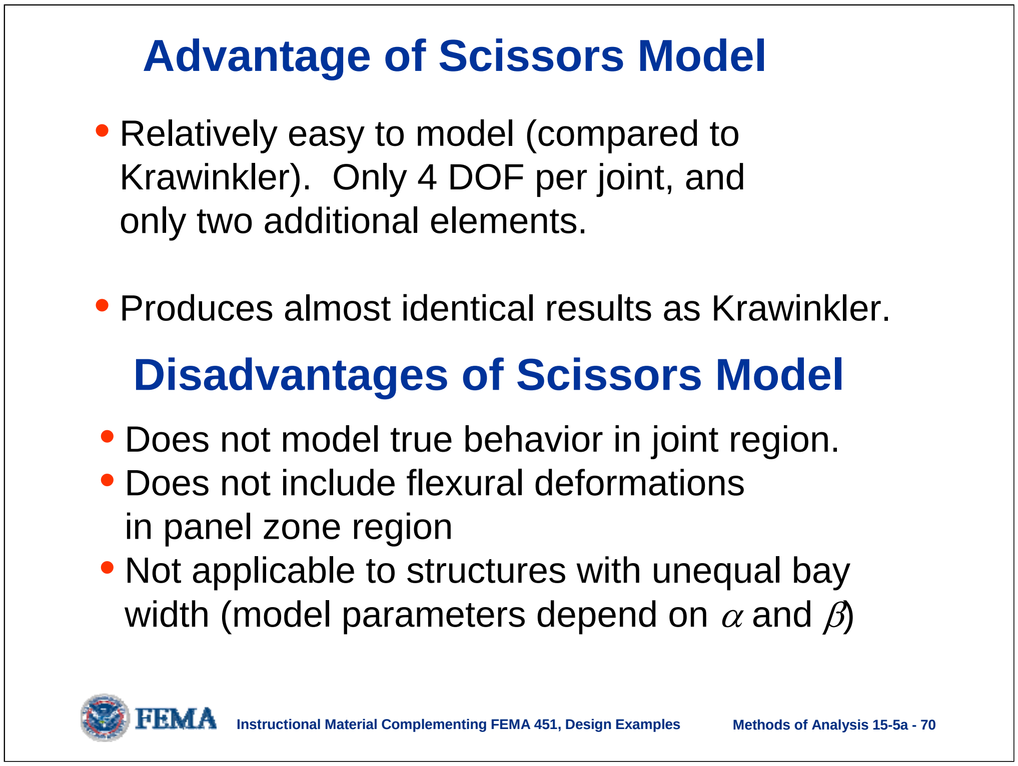
Here the kinematic differences between the two models can be seen more clearly. Note that the centerline offsets in the Krawinkler model are not evident in the Scissors Model. Interestingly, exhaustive testing using DRAIN 2D has shown that the kinematic differences do not have a significant effect on the response.

Mathematical Relationship Between Krawinkler and Scissors Models

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

$$M_{y,Scissors} = \frac{M_{y,Krawinkler}}{(1-\alpha-\beta)}$$

The properties for the Scissors model are most conveniently derived on the basis of the equivalent Krawinkler properties. It is very important to note that many engineers are under the impression that the Scissors model properties are identical to the Krawinkler properties. This is NOT TRUE as indicated by the equations on this slide.



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

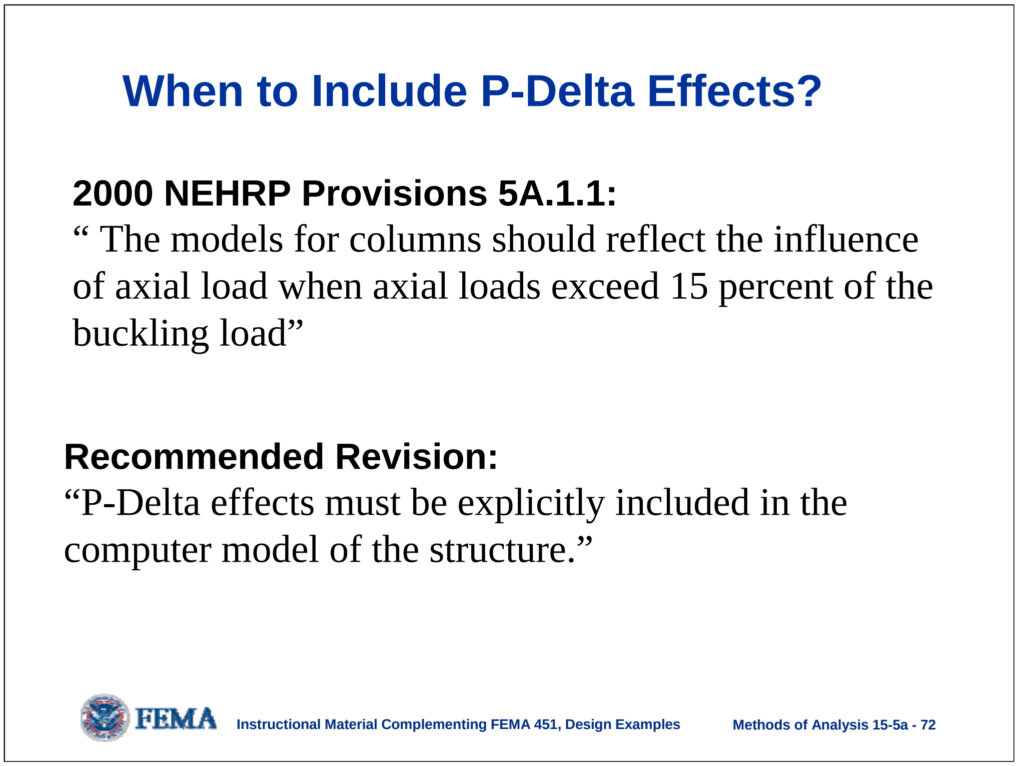
- Does not model true behavior in joint region.
- Does not include flexural deformations in panel zone region
- Not applicable to structures with unequal bay width (model parameters depend on α and β)

Self explanatory. It should be noted that a method has been developed (by Professor Charney) to include panel zone flexural deformations in both the Krawinkler and Scissors models. Before this approach may be released however, careful calibration with FEA models is required, and this calibration is not yet complete.

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.

Modeling of joint deformations in concrete structures is much more difficult than in steel structures.

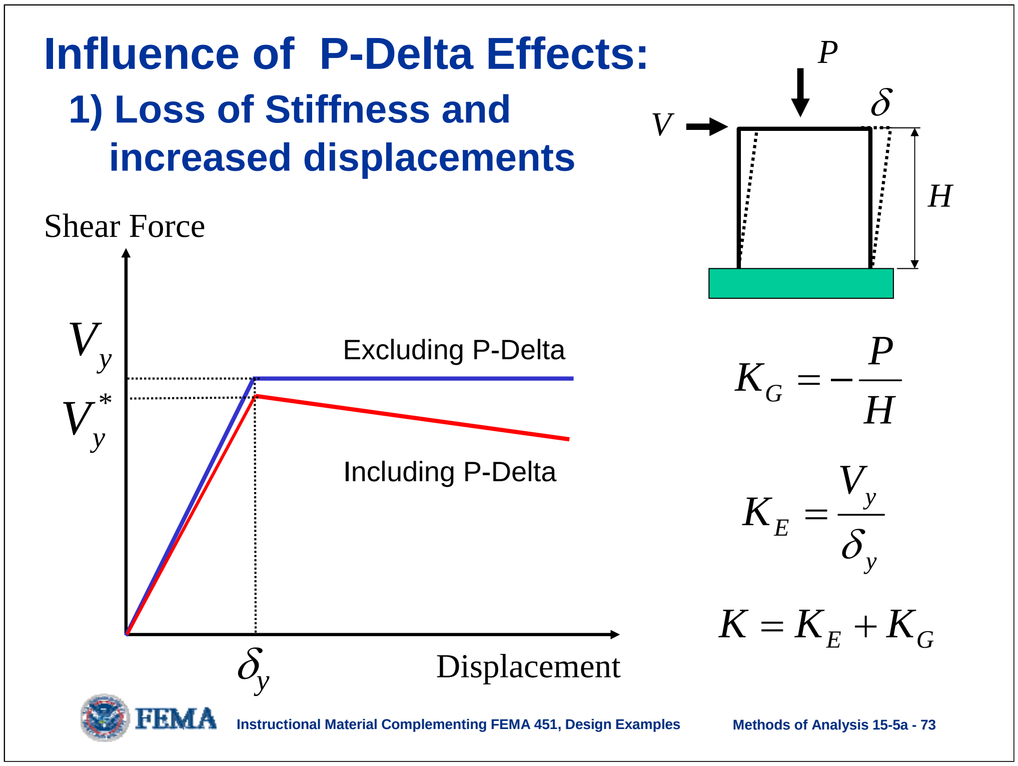


When to Include P-Delta Effects?

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.”

It is essential that P-Delta effects be included in **any** nonlinear analysis. While such effects may have a negligible influence on an elastic response, the influence on the inelastic response of the same structure may be profound. This is particularly true if the structure has little overstrength and if the post-elastic portion of the pushover curve is nearly flat or is descending. Hence the revised recommendation.



Influence of P-Delta Effects:

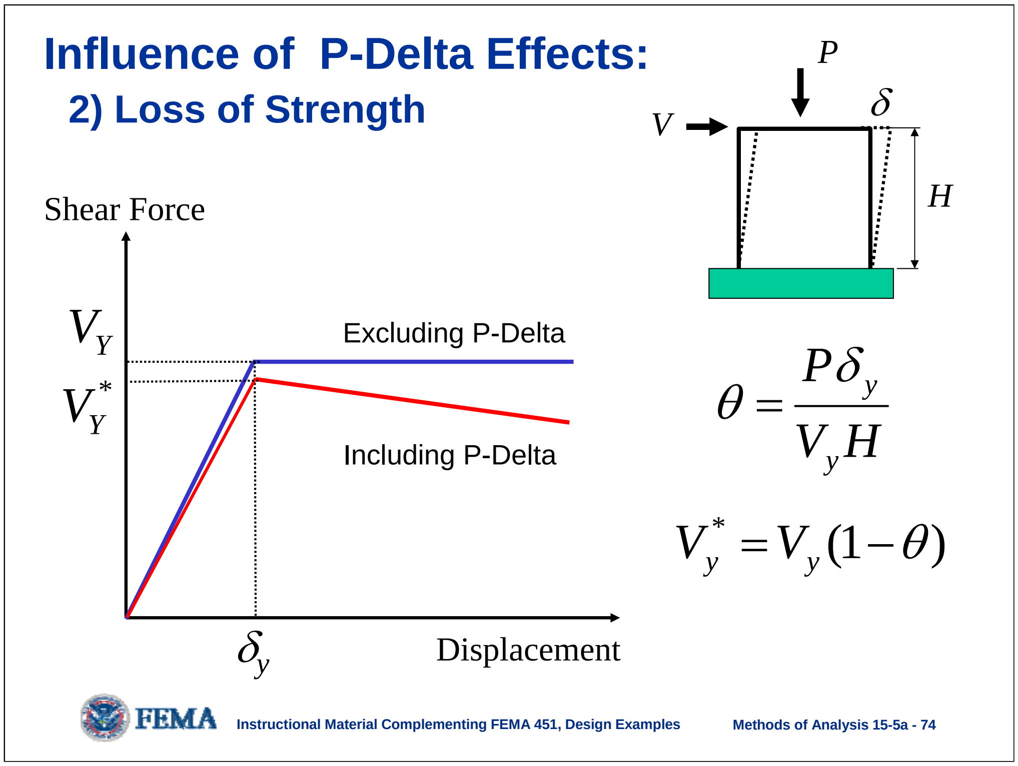
1) Loss of Stiffness and increased displacements

$$K_G = -\frac{P}{H}$$

$$K_E = \frac{V_y}{\delta_y}$$

$$K = K_E + K_G$$

In the simplest sense, P-Delta effects lead to a reduction in stiffness and strength of a structure. In this slide a linearized version of the P-Delta effect is shown. The term K_G refers to the "Linearized Geometric Stiffness" of the structure. The term LINEAR is used because it is assumed the column has a straight-line deflection for consideration of P-Delta effects. Hence, only the rigid-body rotation of the column is considered in the formulation. The actual deformation of the column is not included.



Influence of P-Delta Effects:

2) Loss of Strength

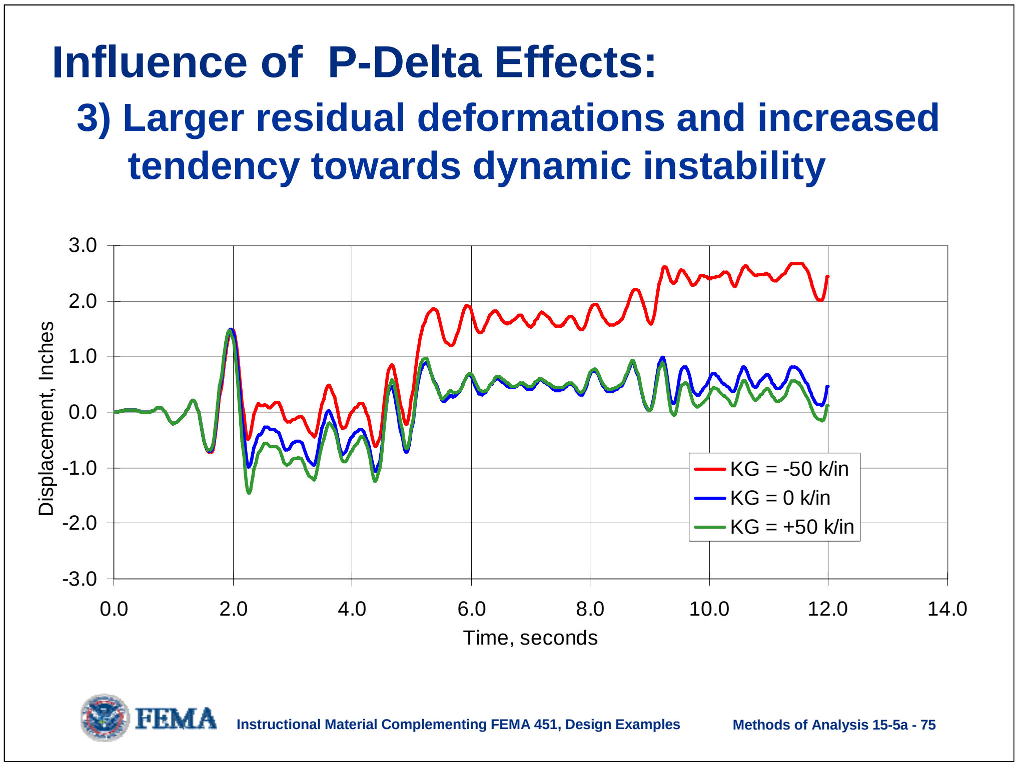
$$\theta = \frac{P\delta_y}{V_y H}$$

$$V_y^* = V_y(1-\theta)$$

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 74

P-Delta effects also reduce the lateral strength of a structure. In essence, the P-Delta effects are imposing a lateral load on the structure, hence, it takes a lower additional lateral load to cause yielding.



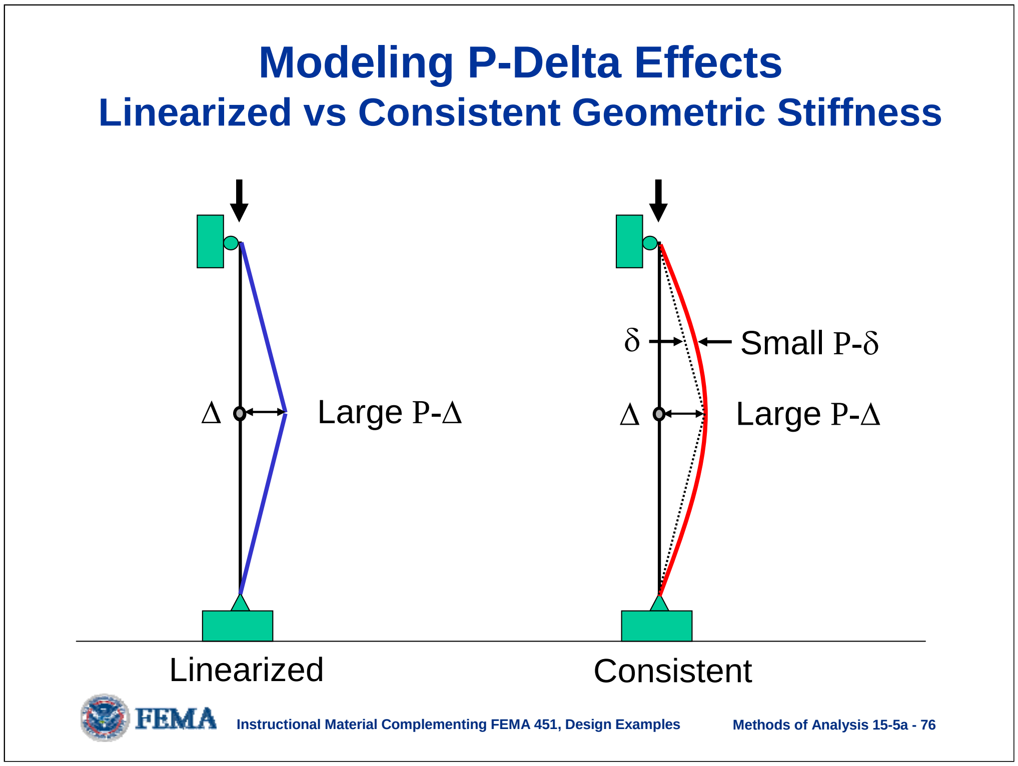
Influence of P-Delta Effects:

3) Larger residual deformations and increased tendency towards dynamic instability

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 75

The most profound influence of the P-Delta effect is on the dynamic response of structures. This plot shows the response history of a simple SDOF system with three different assumptions regarding the post-yield stiffness. All three systems have the same yield strength. A slightly decreased KG (to a value of -75 k/in) may have caused a complete dynamic instability of the system.



Modeling P-Delta Effects

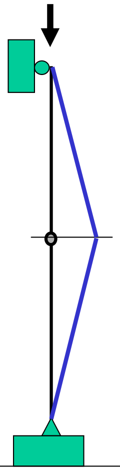
Linearized vs Consistent Geometric Stiffness

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 76

Most analyst include only the “Large” P-Delta effect as shown at the left. This model does a good job of representing the story effect, but does not represent the additional softening that can be produced by consideration of the actual deformation of the component (as shown at the right).

Modeling P-Delta Effects
Linearized Geometric Stiffness



Linearized

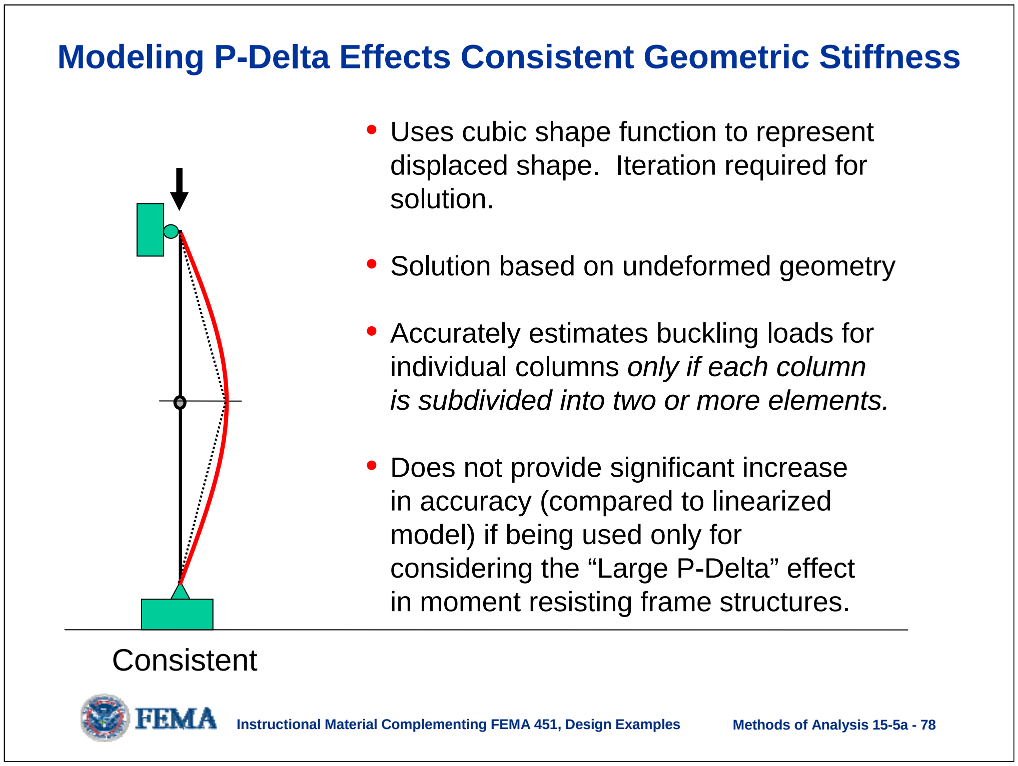
- Uses linear shape function to represent displaced shape. No iteration required for solution.
- Solution based on undeformed geometry
- Significantly overestimates buckling loads for individual columns
- Useful ONLY for considering the “Large P-Delta” Effect on a story-by-story basis

DRAIN 2D and Perform include only the linearized geometric stiffness. This is probably sufficient because most of the structural deformation (interstory drift) is due to rotation in plastic hinges… the columns stay relatively straight between ends and are in double curvature (minimizing the magnitude of the small delta).

The equilibrium equations are formulated on the basis of the undeformed geometry of the structure. Hence, large displacement effects are not considered. This is not a significant source of error in the analysis of framed structures.

Iteration is not required when linearized geometric effects are included because it is the total story (gravity) load that induces the instability. Under lateral load the sum of column forces in a story is zero, hence there is no story P-delta effect.

Linearized geometric stiffness should never be used when computing elastic buckling loads in structures. Buckling loads so predicted may be much higher than the actual buckling load. Improved accuracy may be obtained by subdividing the columns into several (at least four) segments.



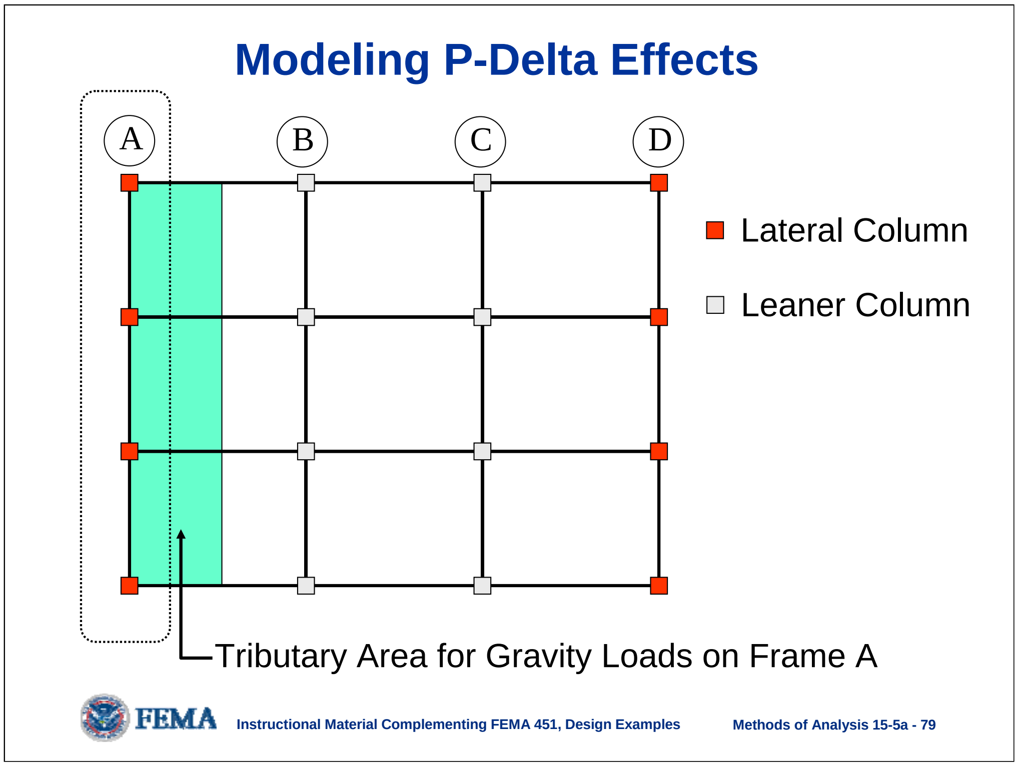
Modeling P-Delta Effects Consistent Geometric Stiffness

- 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.

Consistent

Consistent geometric stiffness uses cubic polynomials to represent the displaced shape between element ends. The word “consistent” arises from the fact that exactly the same polynomials are used in the (virtual displacement) derivation of the element elastic stiffness. As with linearized geometric stiffness, the equilibrium equations are formulated on the basis of the undeformed geometry of the structure.

For buckling analysis, it is still required to subdivide columns. However, only two segments are needed when consistent geometric stiffness is used.



Modeling P-Delta Effects

🟥 Lateral Column

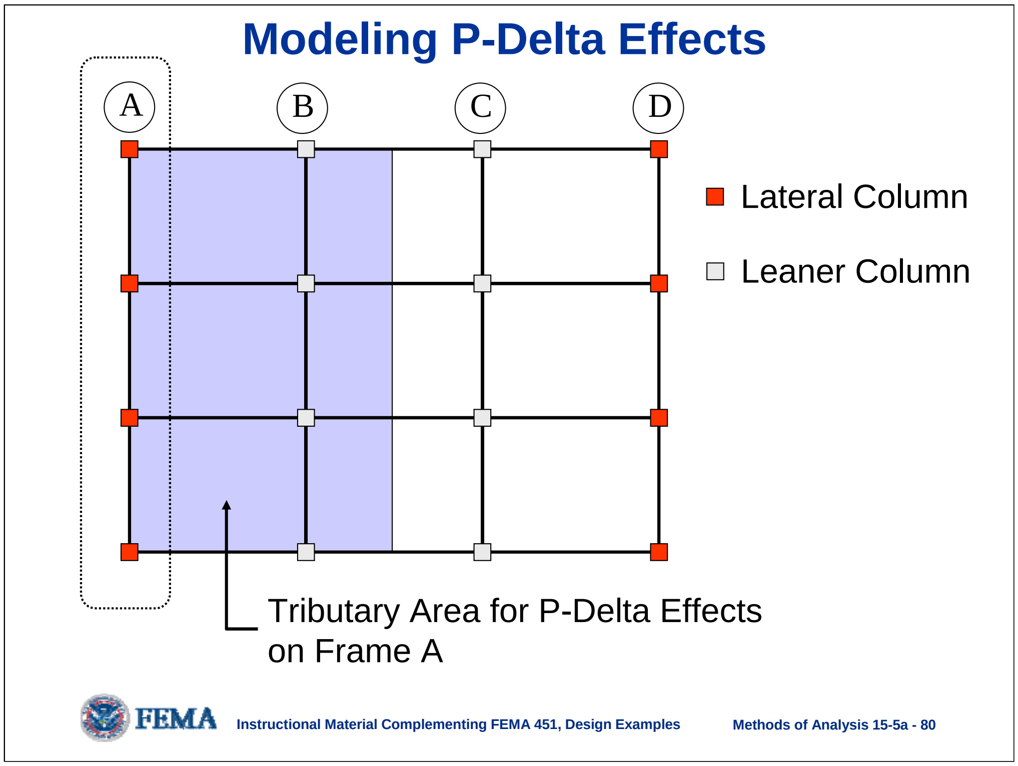
⬜ Leaner Column

Tributary Area for Gravity Loads on Frame A

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 79

When using 2D analysis to analyze a single frame in a 3D structure it is very important to accurately represent the TOTAL P-Delta effects on the frame. In the frame shown here, moment resisting frames are deployed on Lines A and D only. Frames on lines B and C have only simple gravity loads, and are referred to a “leaner” columns. For the structure shown, Frame A is to be analyzed alone using a program like DRAIN 2D. The area shown in green is the tributary gravity load for the frame. However, the tributary load for modeling P-delta effects is much larger as shown on the following slide.



Modeling P-Delta Effects

🟥 Lateral Column

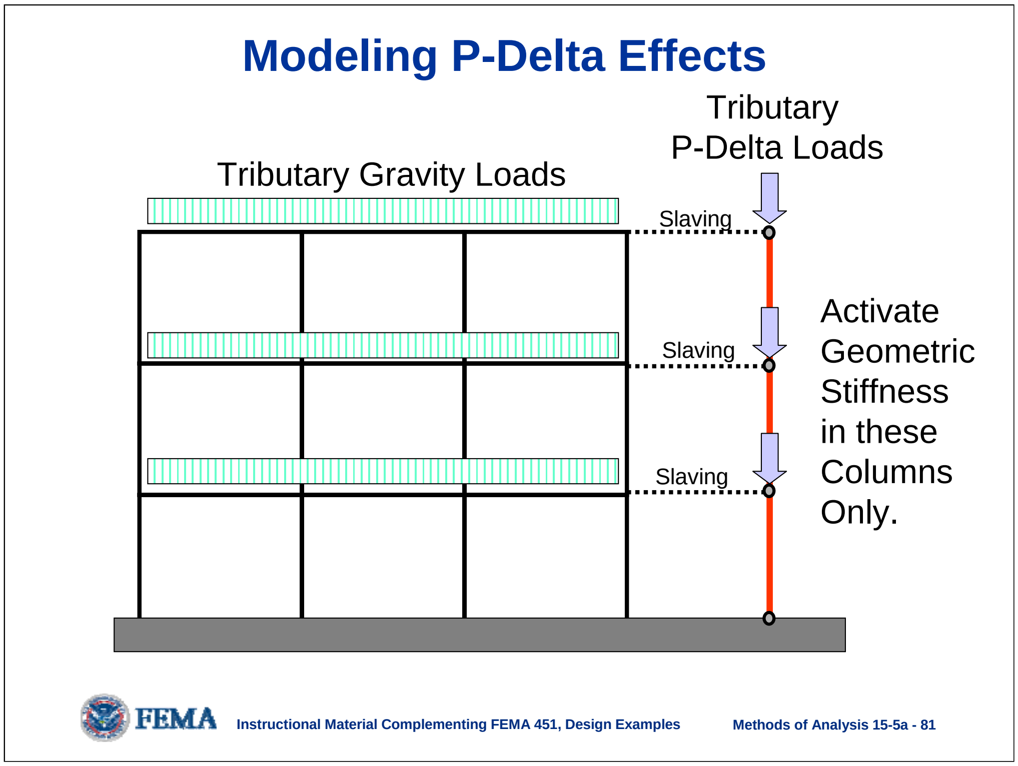
☐ Leaner Column

Tributary Area for P-Delta Effects on Frame A

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 80

When Frame A drifts laterally the entire structure drifts with it. Hence the entire gravity load is producing the P-Delta effect. Thus, the geometric stiffness of the columns in Frame A must be based on the shaded blue area.



Modeling P-Delta Effects

Tributary Gravity Loads

Tributary
P-Delta Loads

Slaving

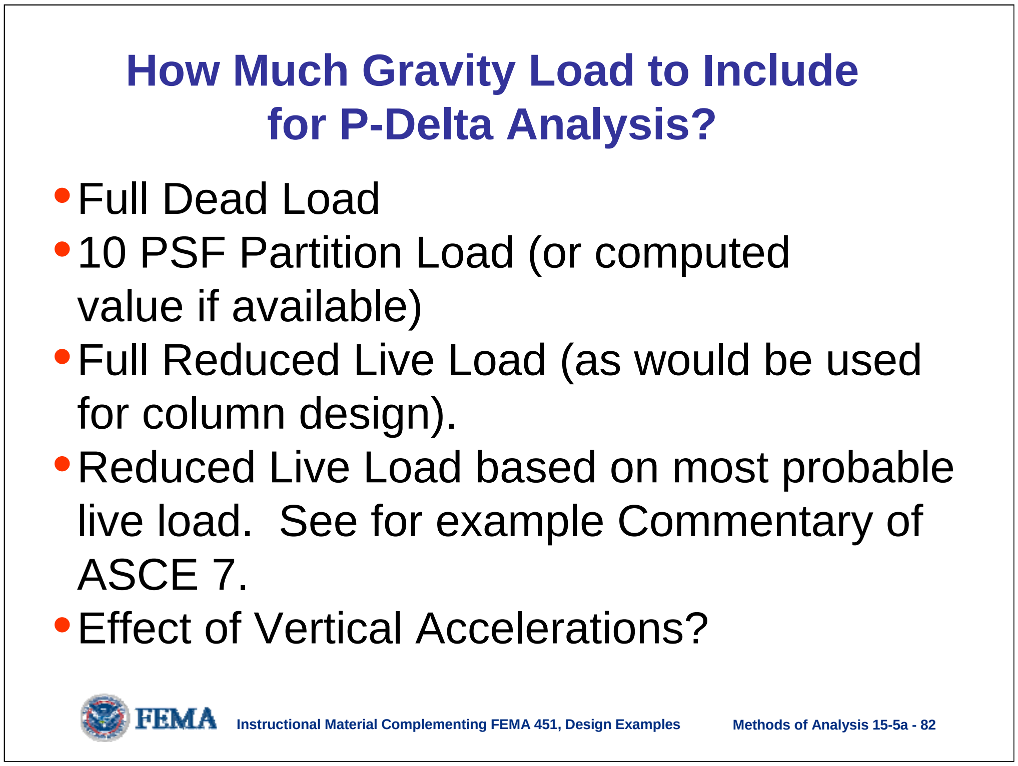
Slaving

Slaving

Activate
Geometric
Stiffness
in these
Columns
Only.

Because different tributary loads are required for frame gravity effects and for system P-Delta effects, it is necessary to model P-Delta effects through the use of a special outrigger or “ghost” frame. In DRAIN, these columns would likely be Type-1 truss elements, with one element being used for each story. The lateral DOF at each story of the ghost frame are slaved to the appropriate story in the main frame. Story gravity loads are applied as shown.

If all of the story gravity load is applied to the ghost column, the P-Delta effect would be TURNED OFF in the main frame columns.



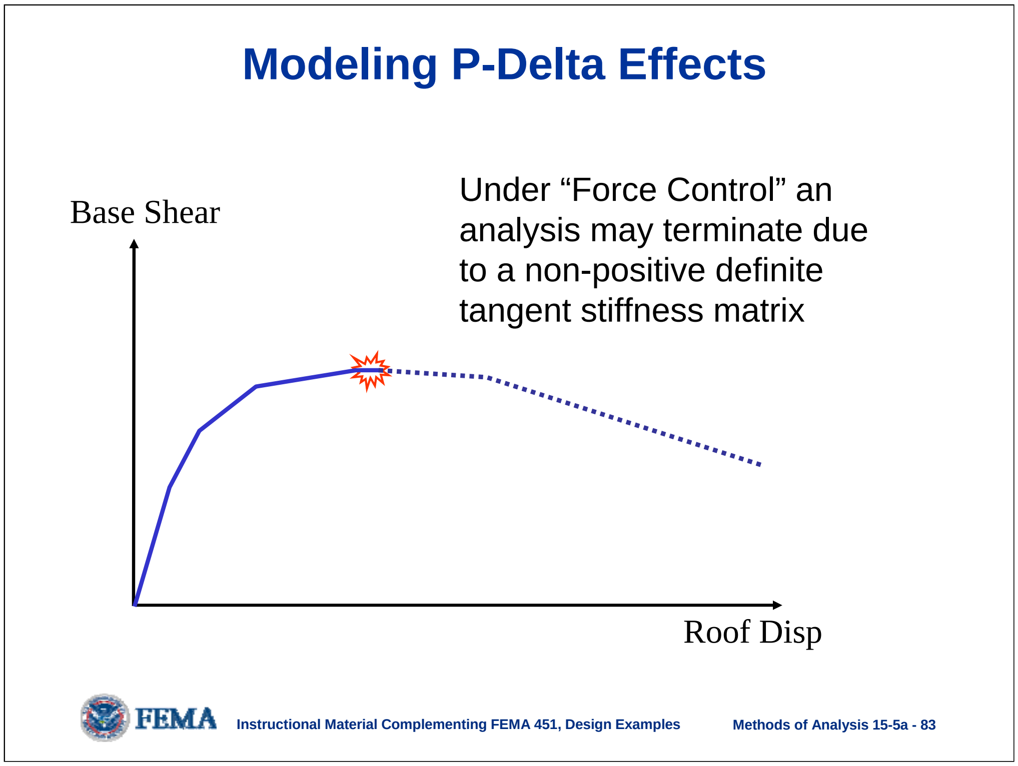
How Much Gravity Load to Include for P-Delta Analysis?

- 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?

How much gravity load to include for P-Delta analysis? This slide gives some recommendations.

It is certainly overly conservative to include full live load. Even fully reduced live load may be too conservative. See for example Table C4-2 of ASCE7-02 which suggests an average of 10.9 psf for offices, with a standard deviation of 5.9 psf.

Vertical accelerations may have an important effect on system stability, depending on phasing effects. It is not known if any research has been done in this area.

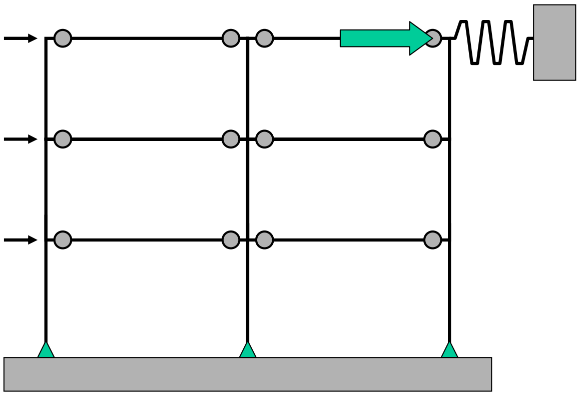


Modeling P-Delta Effects

Under “Force Control” an analysis may terminate due to a non-positive definite tangent stiffness matrix

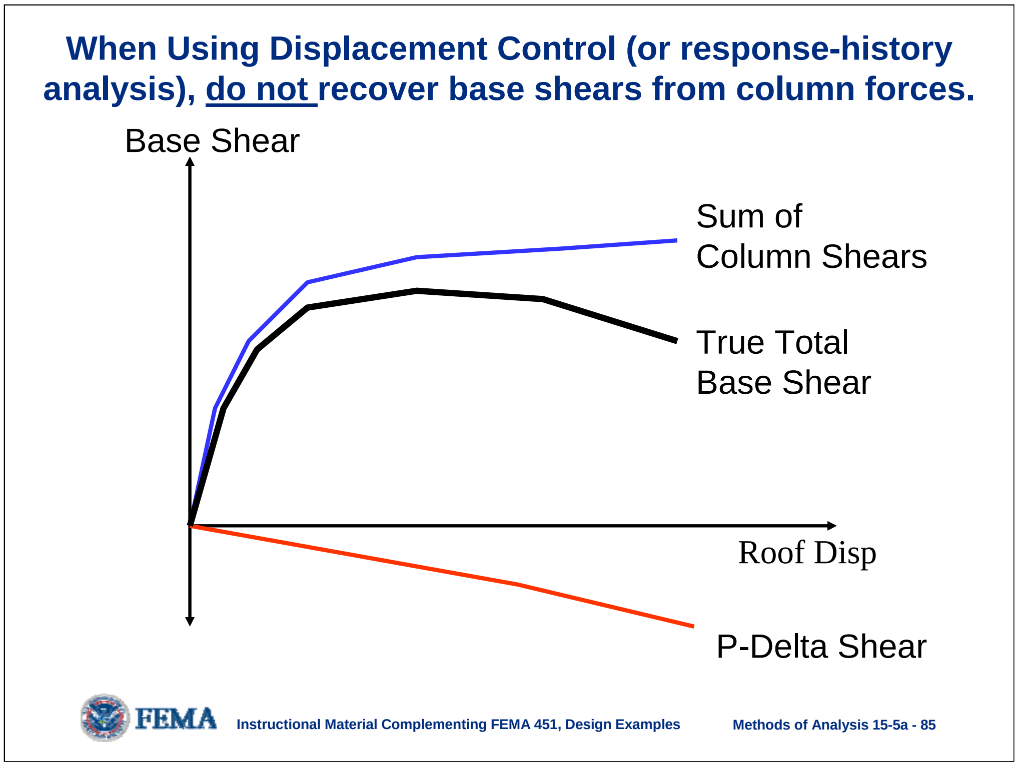
When running a pushover analysis the user has the option of performing the analysis under “Force Control” or under “Displacement Control”. If the analysis is being executed under force control the analysis will terminate with an error as soon as the incremental tangent stiffness is negative. (Actually, the determinant of the tangent stiffness will be negative.) If it is desired to track behavior beyond the point where the tangent stiffness is negative, it is necessary to use a displacement controlled analysis.

Must Use Displacement Controlled Analysis to Obtain Complete Response



In essence, a displacement controlled analysis uses a stiff spring, which when added to the system's tangent stiffness, results in a positive definite system stiffness. Displacement control algorithms maintain the desired lateral force pattern for all analysis steps. It must be noted, however, that the response of a statically loaded system beyond the point where the tangent stiffness of the original system goes negative is completely fictitious. A real structure, statically loaded, would immediately collapse at that point.

Under dynamic loads, the system tangent stiffness may be negative, but the effective system stiffness may be positive due to inertial effects. This is evident from the fact that the incremental tangent stiffness is the actual tangent stiffness + Mass/ΔT^2 + Damping/ΔT. The Mass term is always positive. The damping term will also be positive if the damping matrix is based on mass and initial system stiffness.



When Using Displacement Control (or response-history analysis), do not recover base shears from column forces.

Instructional Material Complementing FEMA 451, Design Examples

Methods of Analysis 15-5a - 85

When using response history analysis or displacement controlled static analysis, the system base shear must be recovered from the sum of the column shears PLUS the story wise P-Delta shears. Under force controlled static analysis, the base shear can be computed directly from the applied loads or from the process shown above (which should give the same answer).

Proceed to Topic 15-5b.