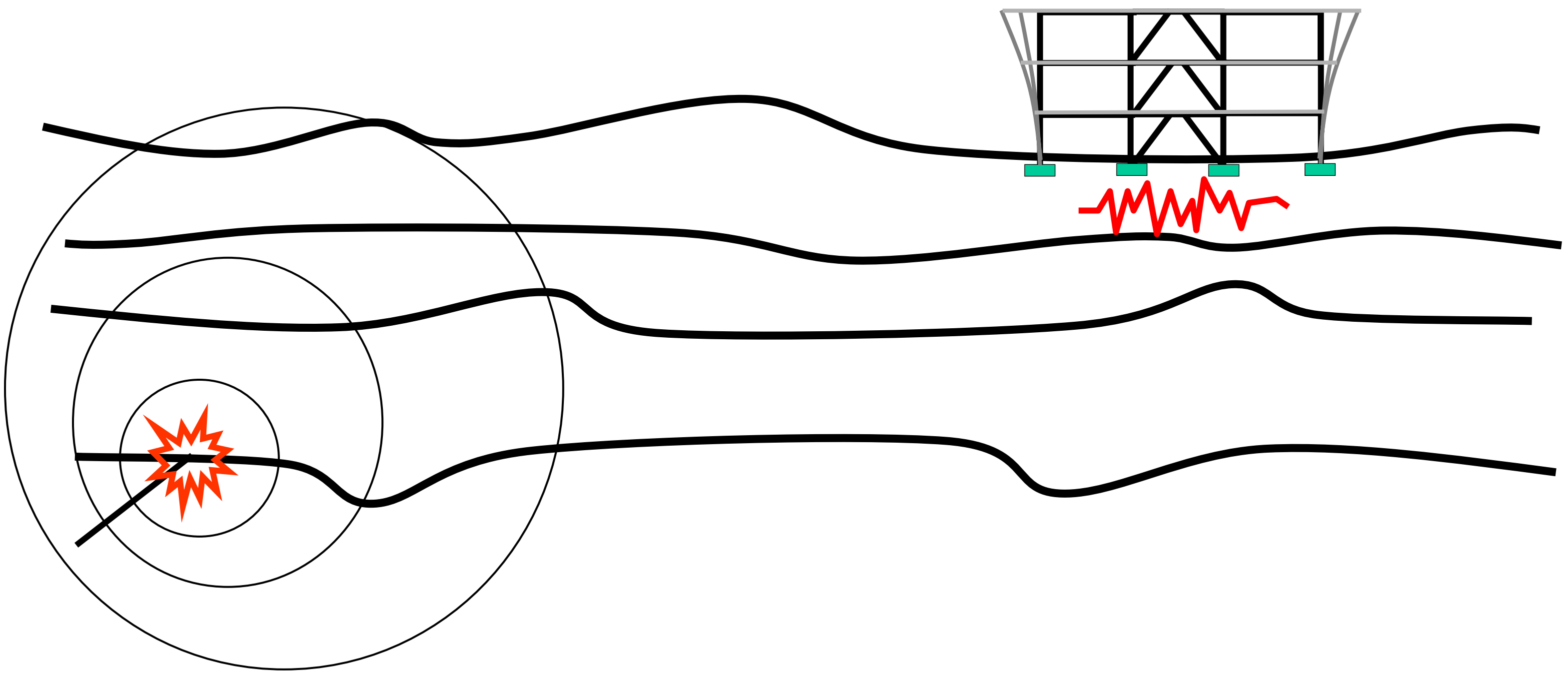
STRUCTURAL ANALYSIS FOR PERFORMANCE-BASED EARTHQUAKE ENGINEERING



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)

Analysis Method

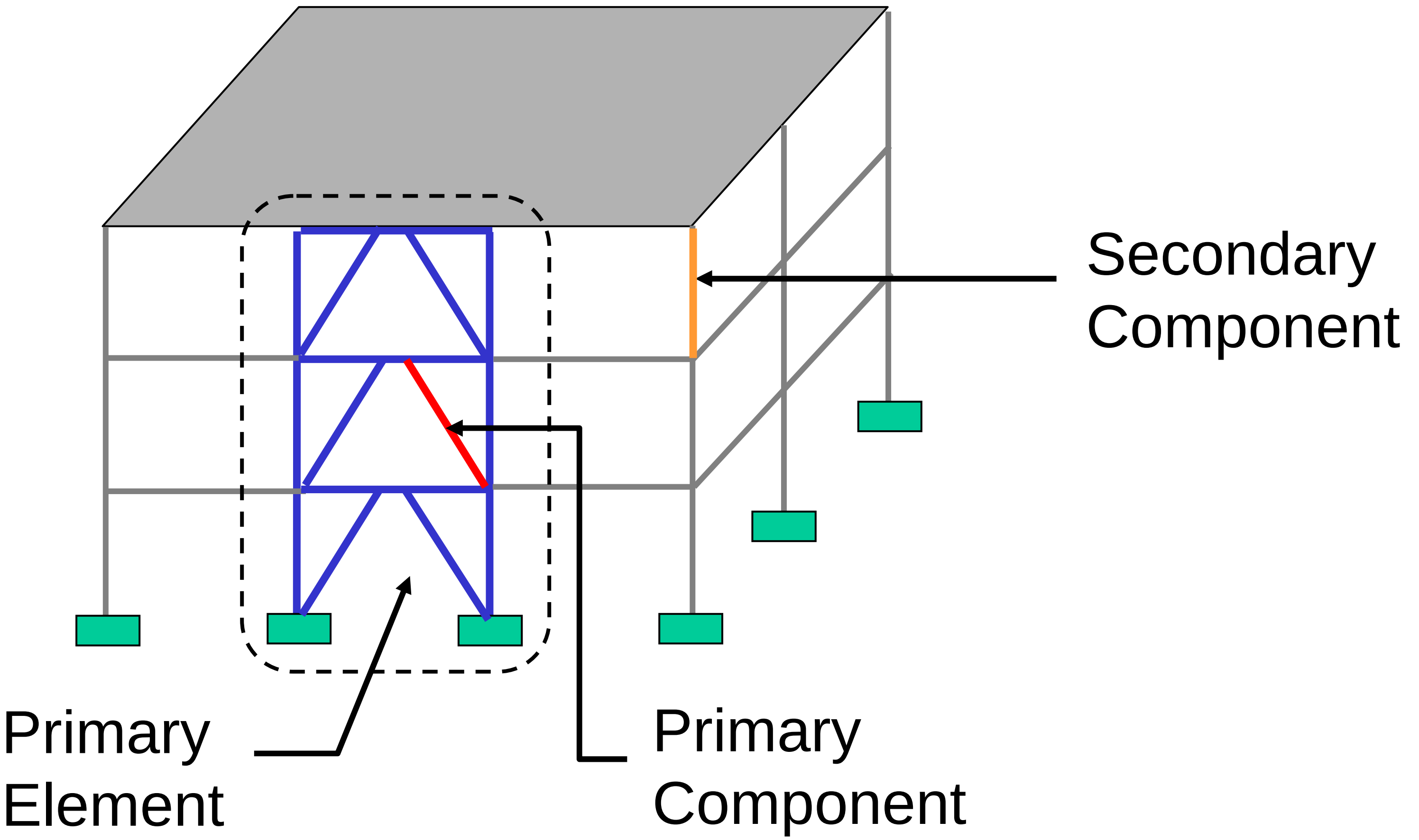
		Linear Static	Response Spectrum	Linear Resp. Hist.	Nonlinear Resp. Hist.
$T \le T_s$	Regular Structures	YES	YES	YES	YES
	Plan Irreg. 2,3,4,5 Vert. Irreg. 4, 5	YES	YES	YES	YES
	Plan Irreg. 1a ,1b Vert. Irreg. 1a, 1b 2, or 3	NO	YES	YES	YES
All Other Structures		NO	YES	YES	YES

Nonlinear Static Analysis Limitations not Stated

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

Definition for "Elements" and "Components"



Primary elements or components are critical to the buildings 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*

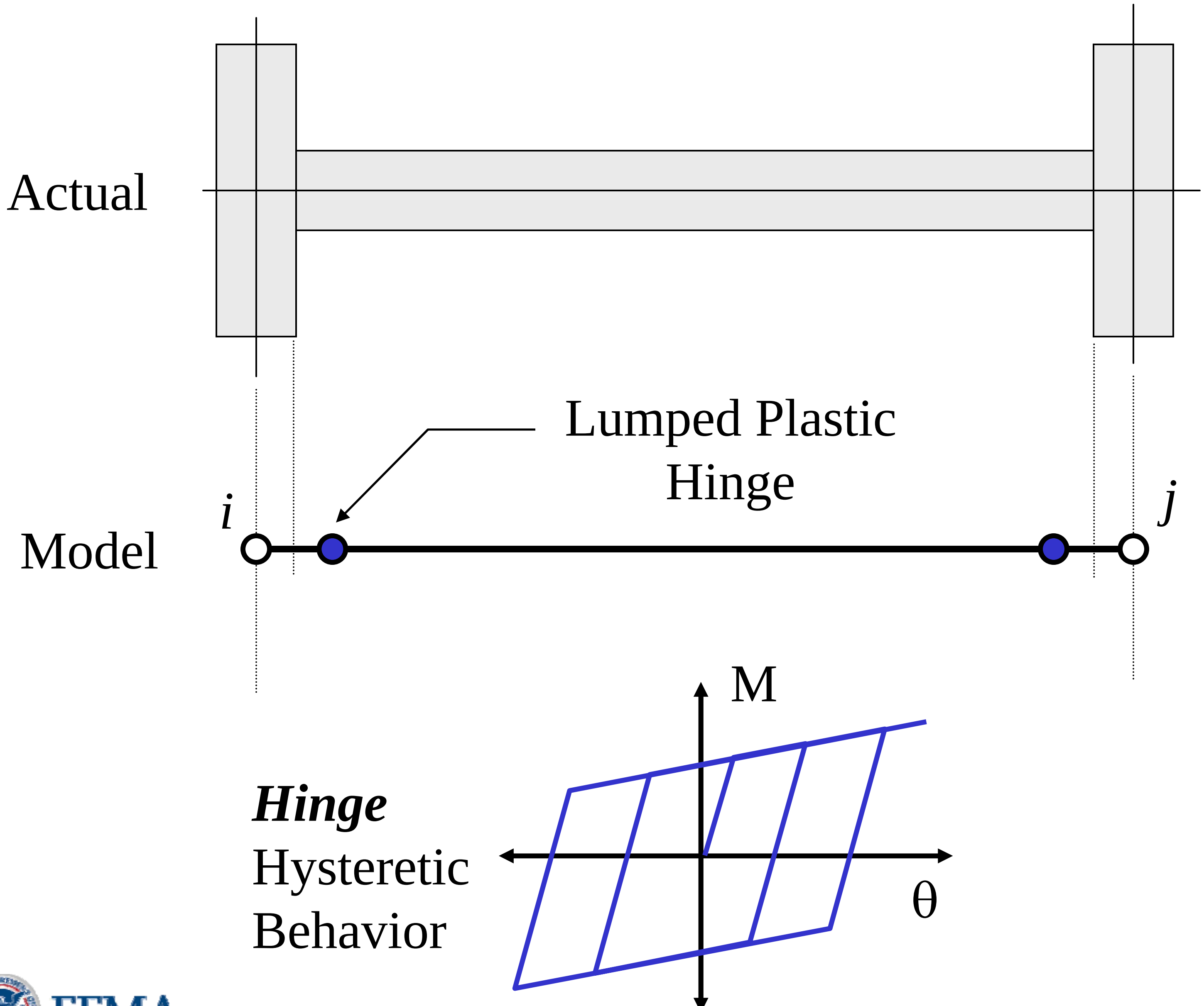
Basic Component Model Types

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



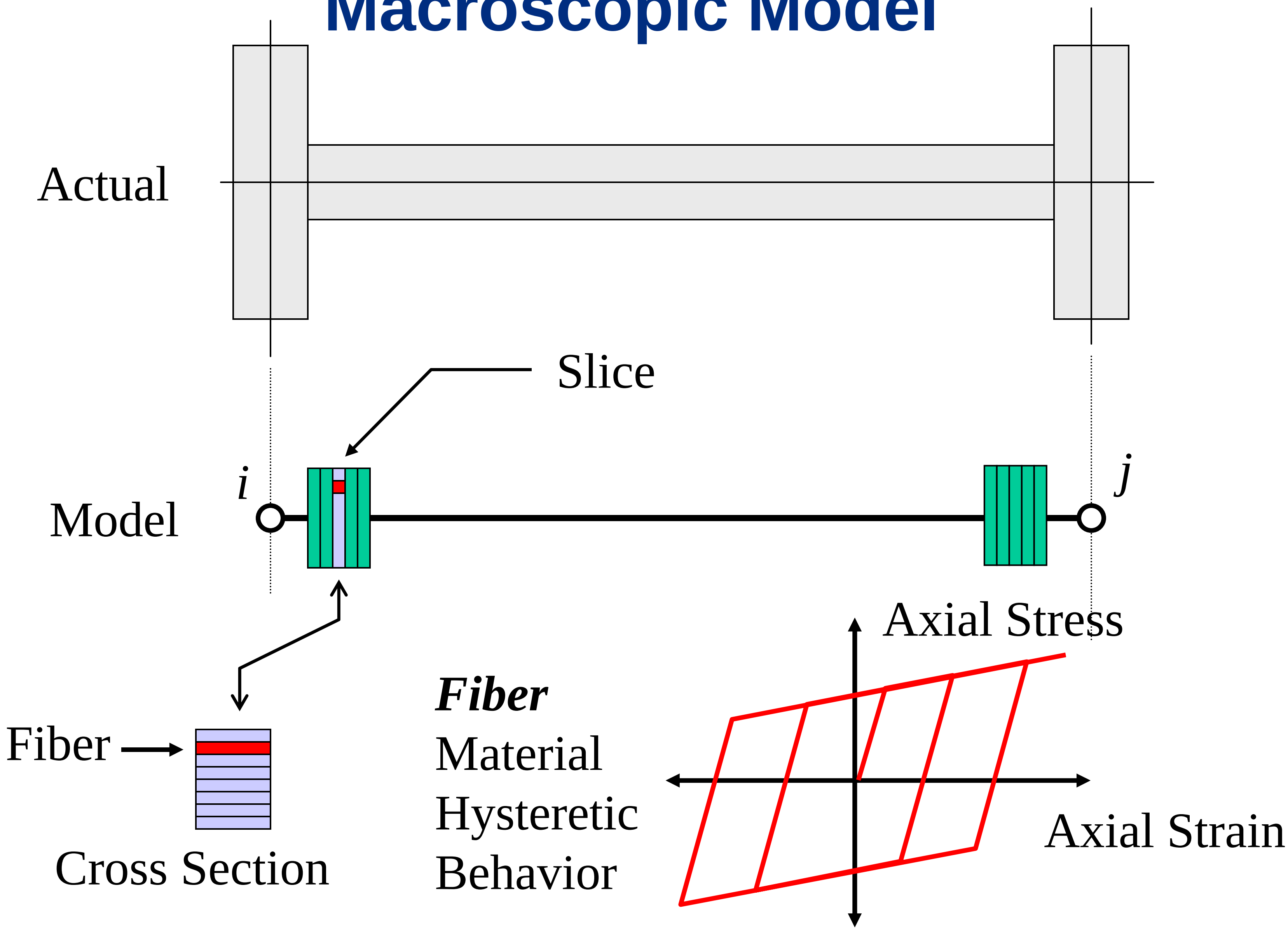
Basic Component Model Types

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.

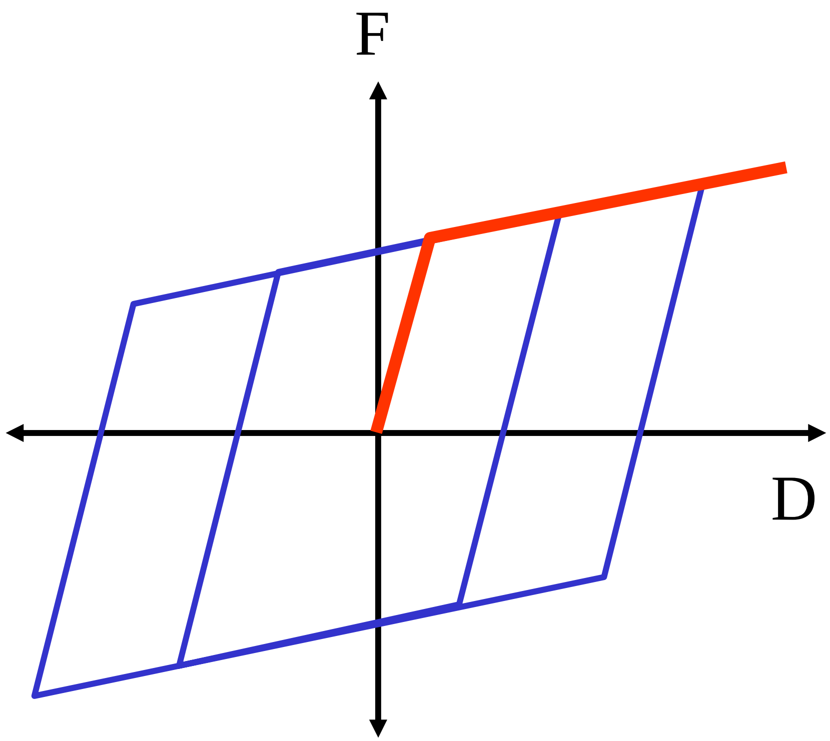
These models are reasonably accurate, but are very computationally expensive. Pushover analysis may be practical for some 2D structures, but nonlinear dynamic time history analysis is not currently feasible for large 2D structures or for 3D structures.

Macroscopic Model

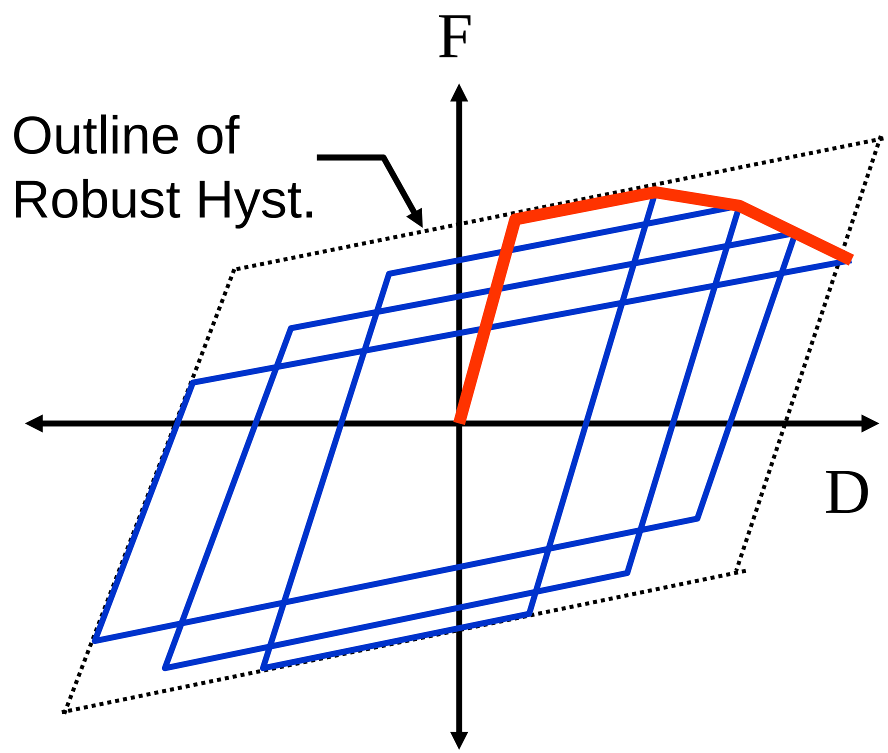


Fiber
Material
Hysteretic
Behavior

Rule-Based Hysteretic Models and Backbone Curves (1)

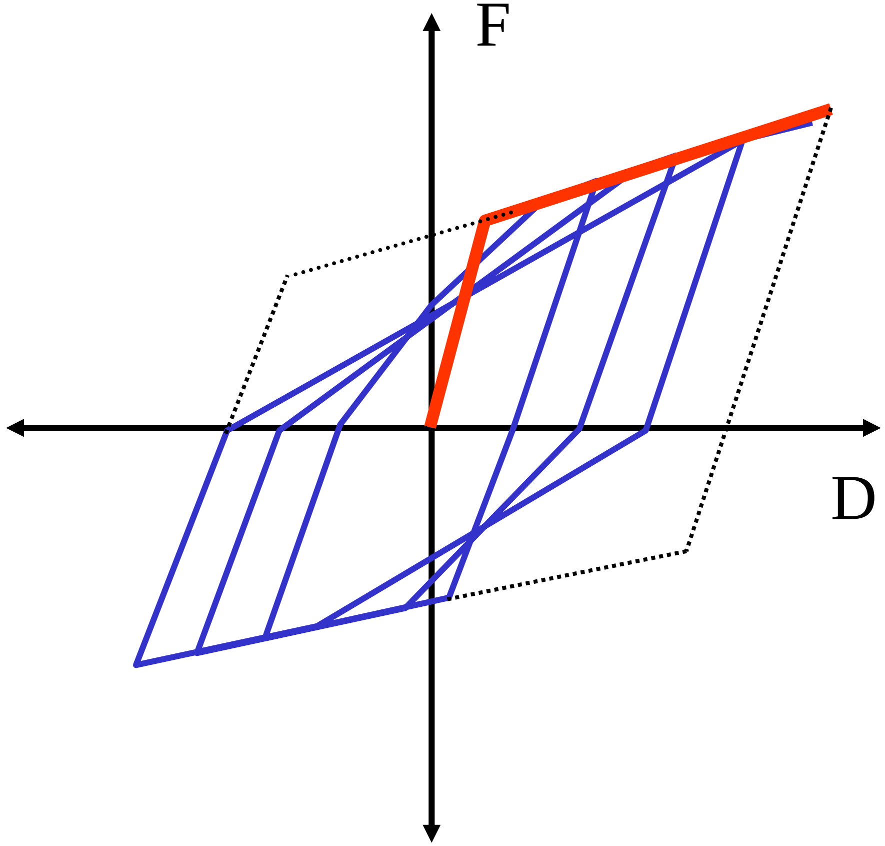


Simple Yielding
(Robust)

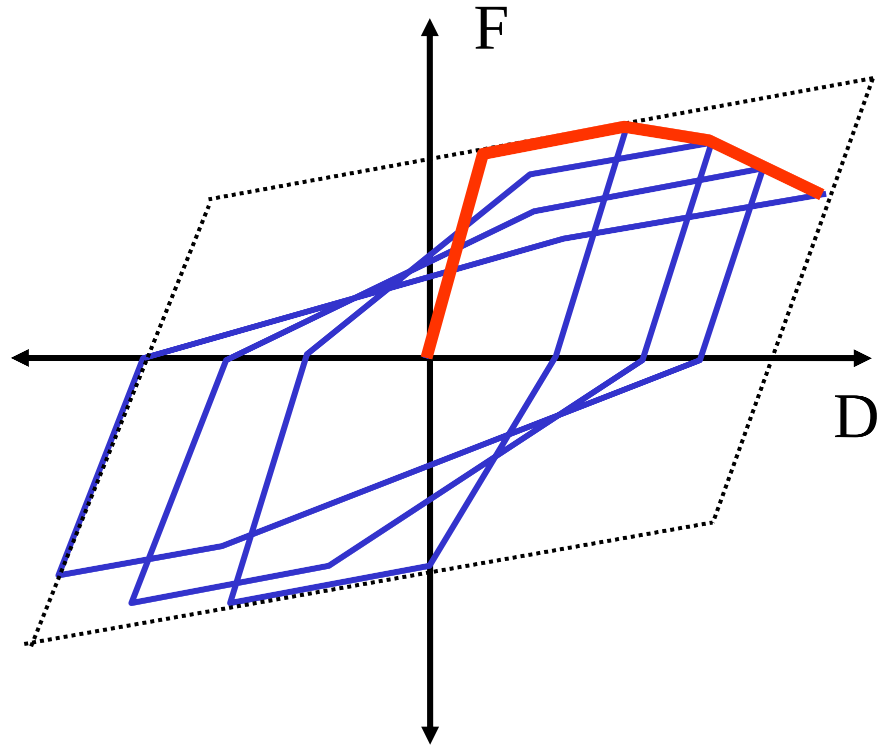


(Ductile) Loss of Strength

Rule-Based Hysteretic Models and Backbone Curves (2)

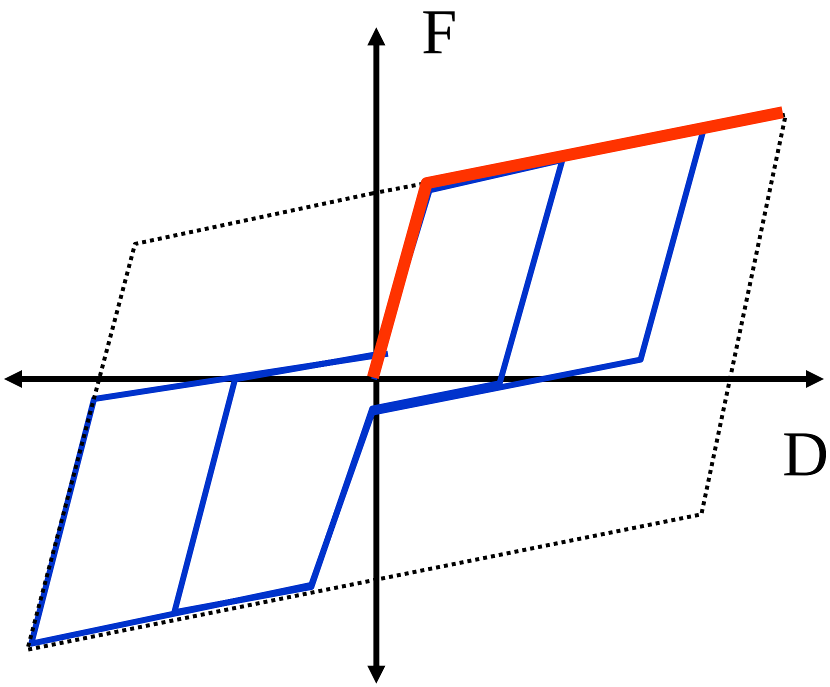


Loss of Stiffness

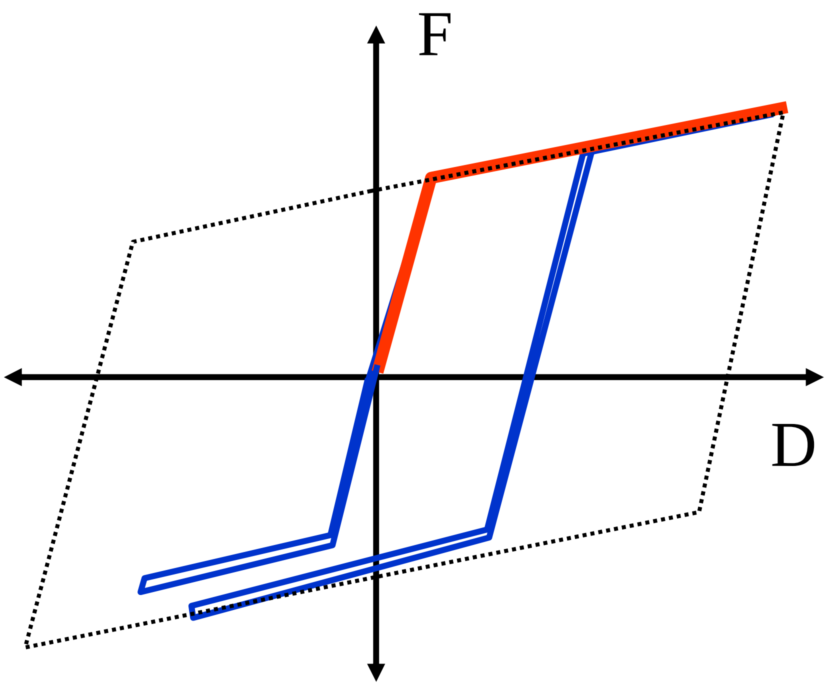


Loss of Strength and Stiffness

Rule-Based Hysteretic Models and Backbone Curves (3)

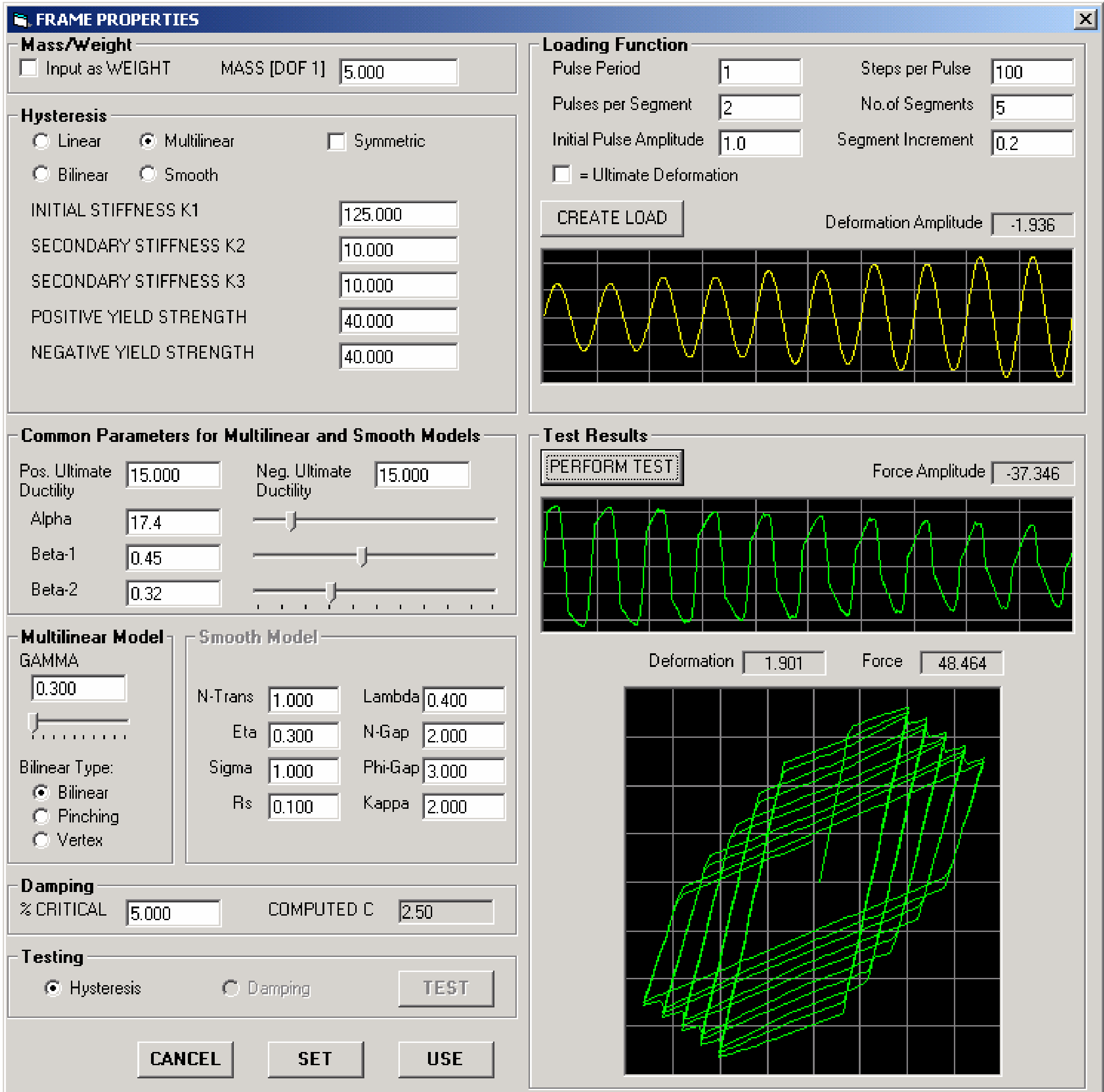


Pinched



Buckling

Sivaselvan and Reinhorn Models in NONLIN (MDOF MODEL)

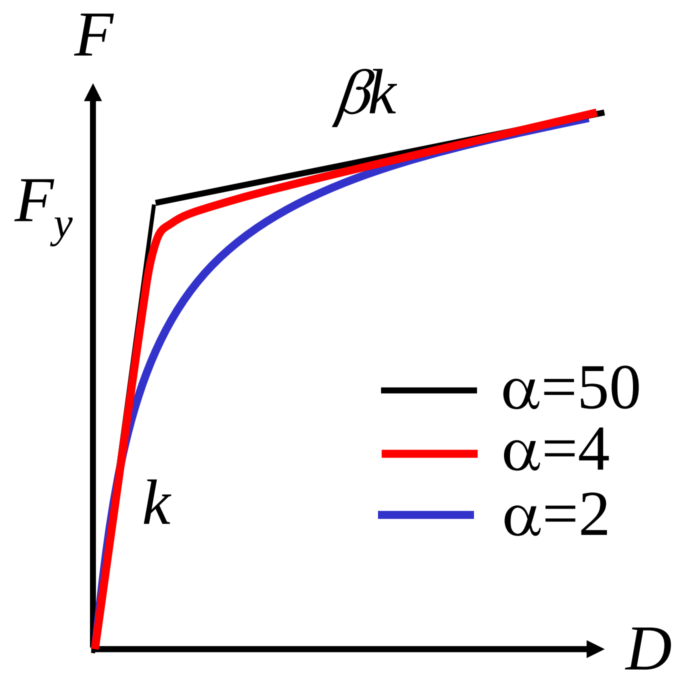


NONLIN

Parametric Models, e.g., SAP2000

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

$$\dot{Z} = \frac{k}{F_y} \left\{ \begin{array}{ll} \dot{D}(1-|Z|^{\alpha}) & if \;\; \dot{D}Z > 0 \\ \dot{D} & otherwise \end{array} \right\}$$



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
- *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

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

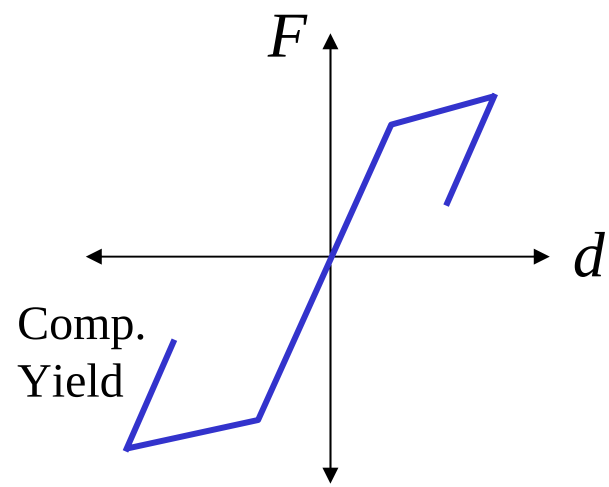
DRAIN-2DX Element Library

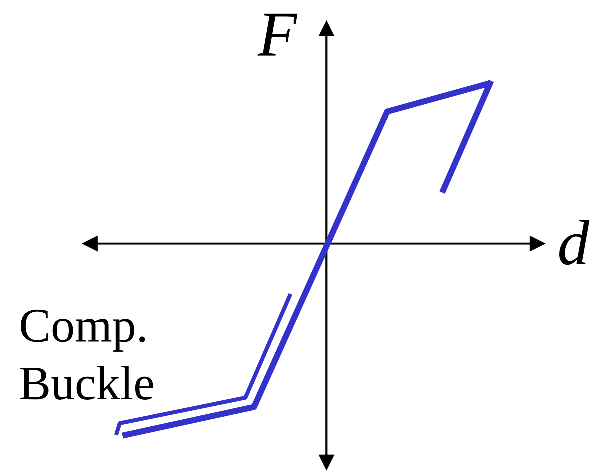
TYPE 1: Truss Bar
TYPE 2: Beam-Column
TYPE 3: Degrading Stiffness Beam-Column*
TYPE 4: Zero Length Connector
TYPE 6: Elastic Panel
TYPE 9: Compression/Tension Link
TYPE 15: Fiber Beam-Column*

* Not fully supported by Nonlin-Pro

DRAIN 2Dx Truss Bar Element

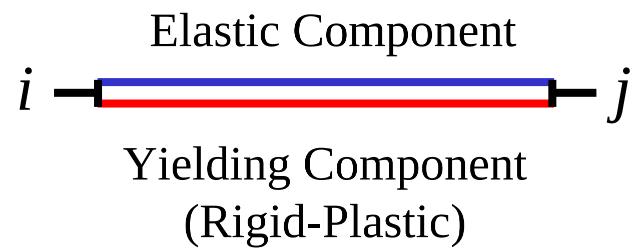
- Axial Force Only
- Simple Bilinear Yield in Tension or Compression
- Elastic Buckling in Compression
- Linearized Geometric Stiffness
- May act as linear viscous damper (some trickery required)



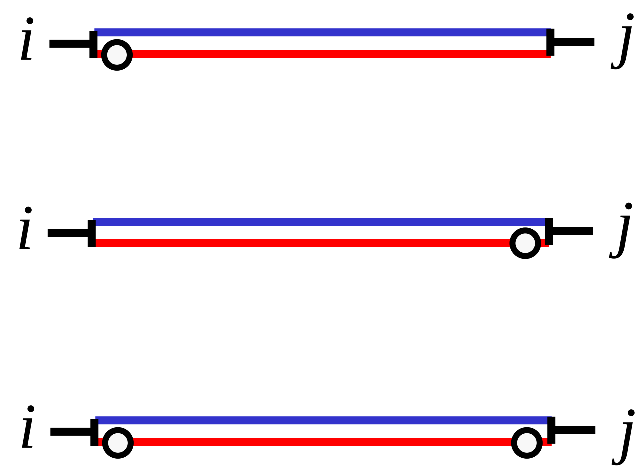


DRAIN 2Dx Beam-Column Element

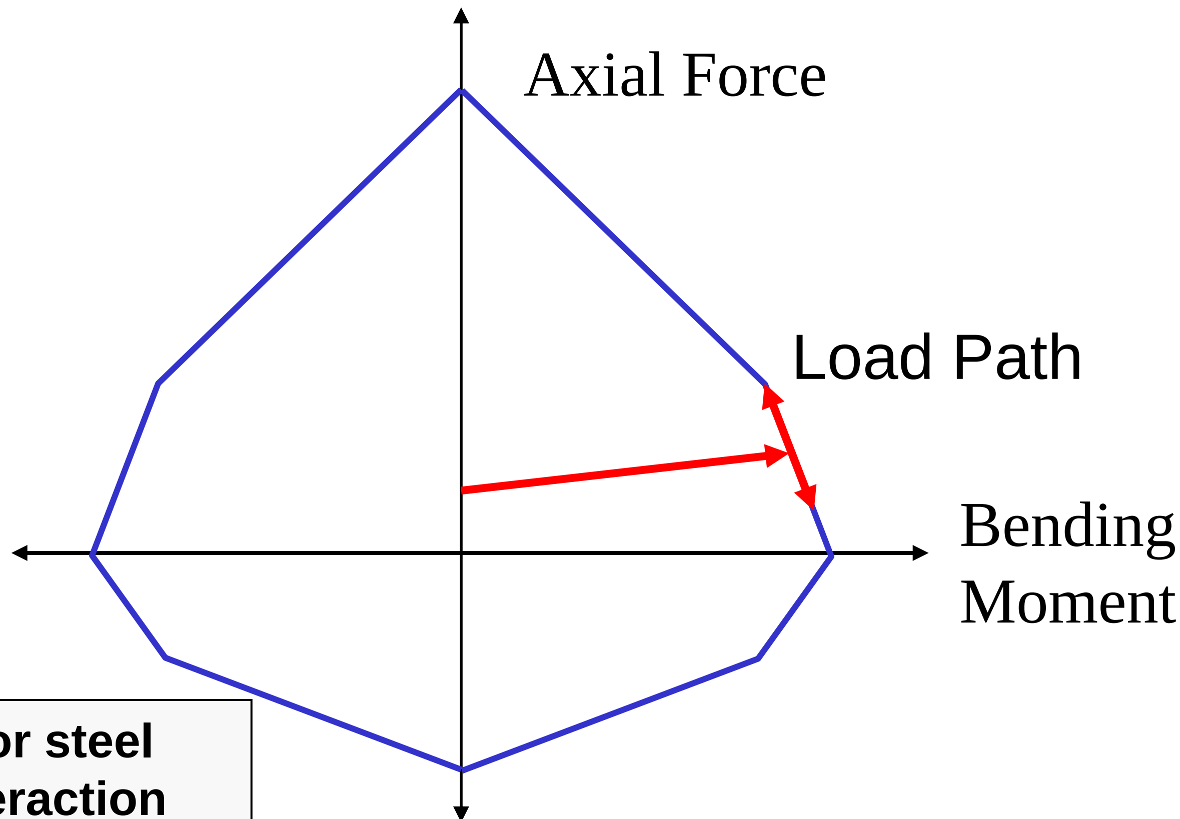
- Two Component Formulation
- Simple Bilinear Yield in Positive or Negative Moment. *Axial yield is NOT provided.*
- Simple Axial-Flexural Interaction
- Linearized Geometric Stiffness
- Nonprismatic properties and shear deformation possible
- Rigid End Zones Possible



Possible Yield States

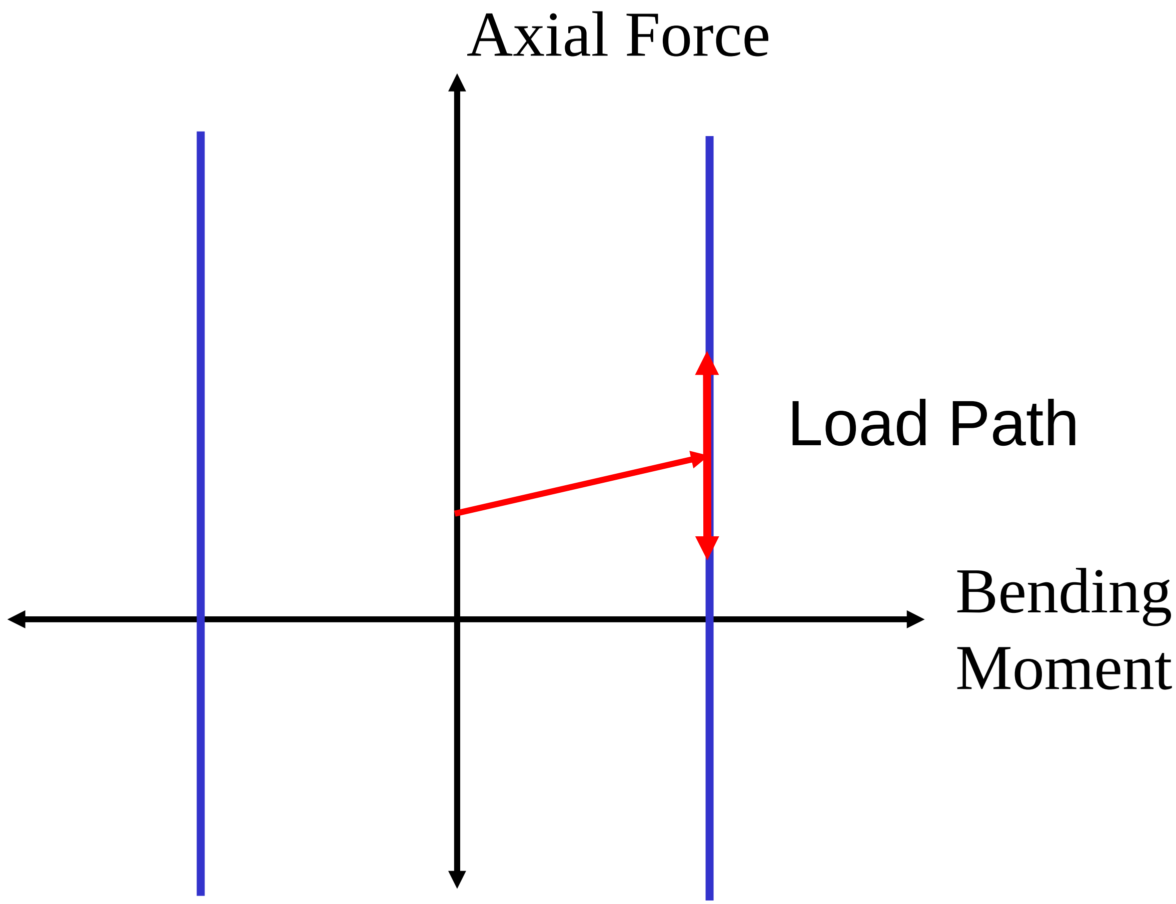


DRAIN 2Dx Beam-Column Element
Axial-Flexural Interaction

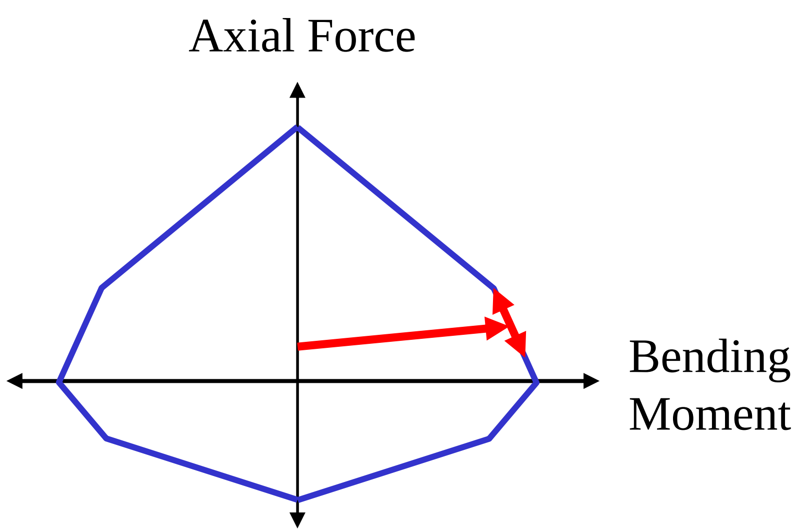


Note: Diagram is for steel sections. NOo interaction and reinforced concrete type interaction is also possible

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



DRAIN 2Dx Beam-Column Element Axial-Flexural Interaction



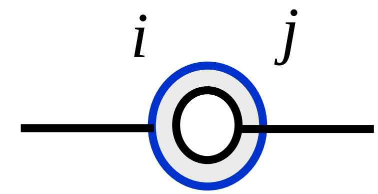
Note: This Model is not known for its accuracy or reliability. Improved models based on plasticity theory have been developed. See, for example, The RAM-Perform Program.

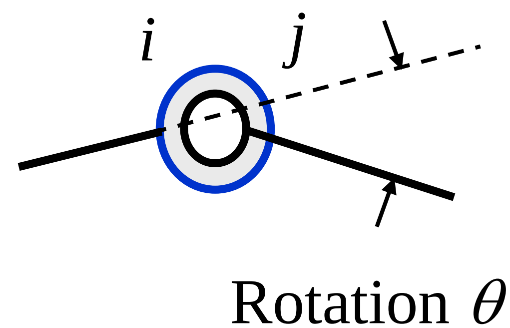
DRAIN 2Dx
Connection Element

- 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.***

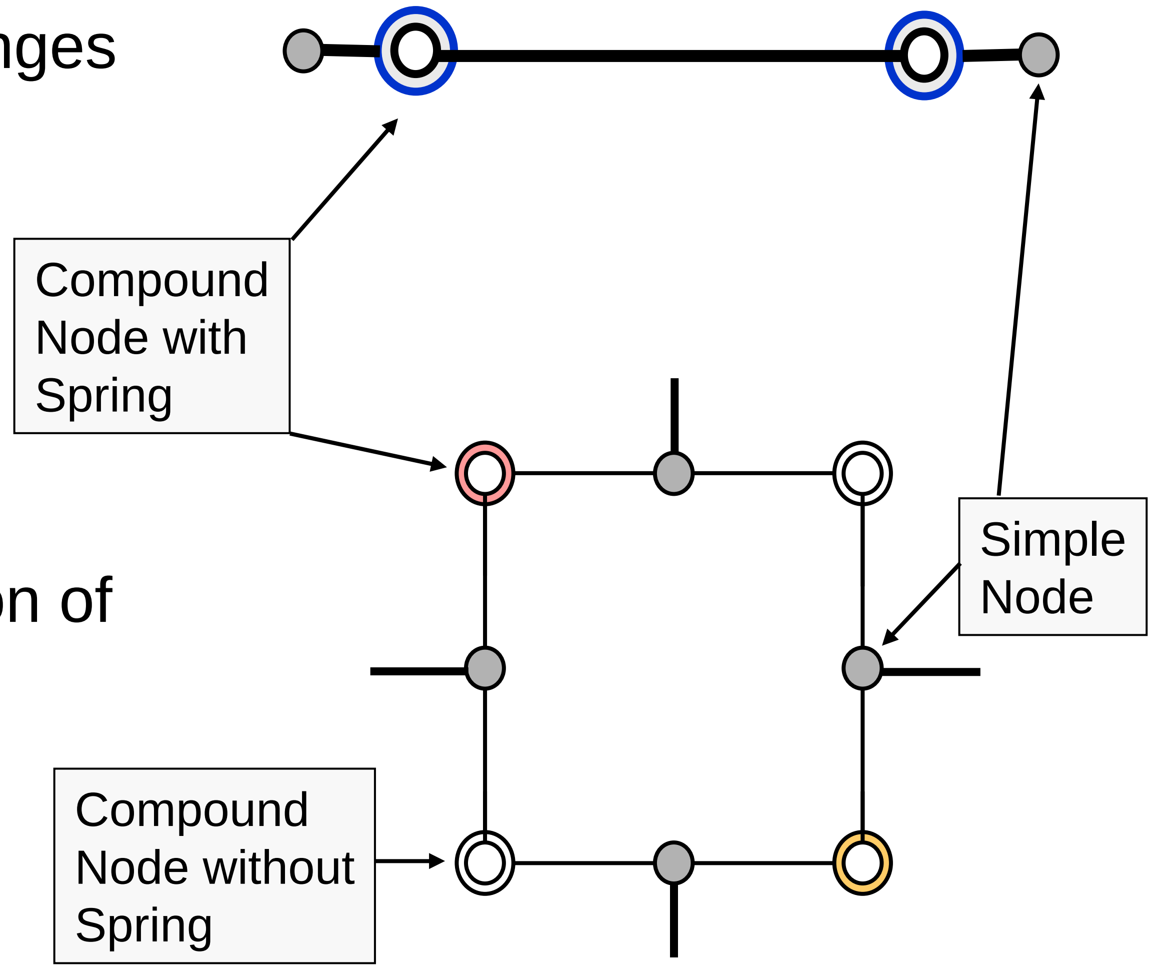




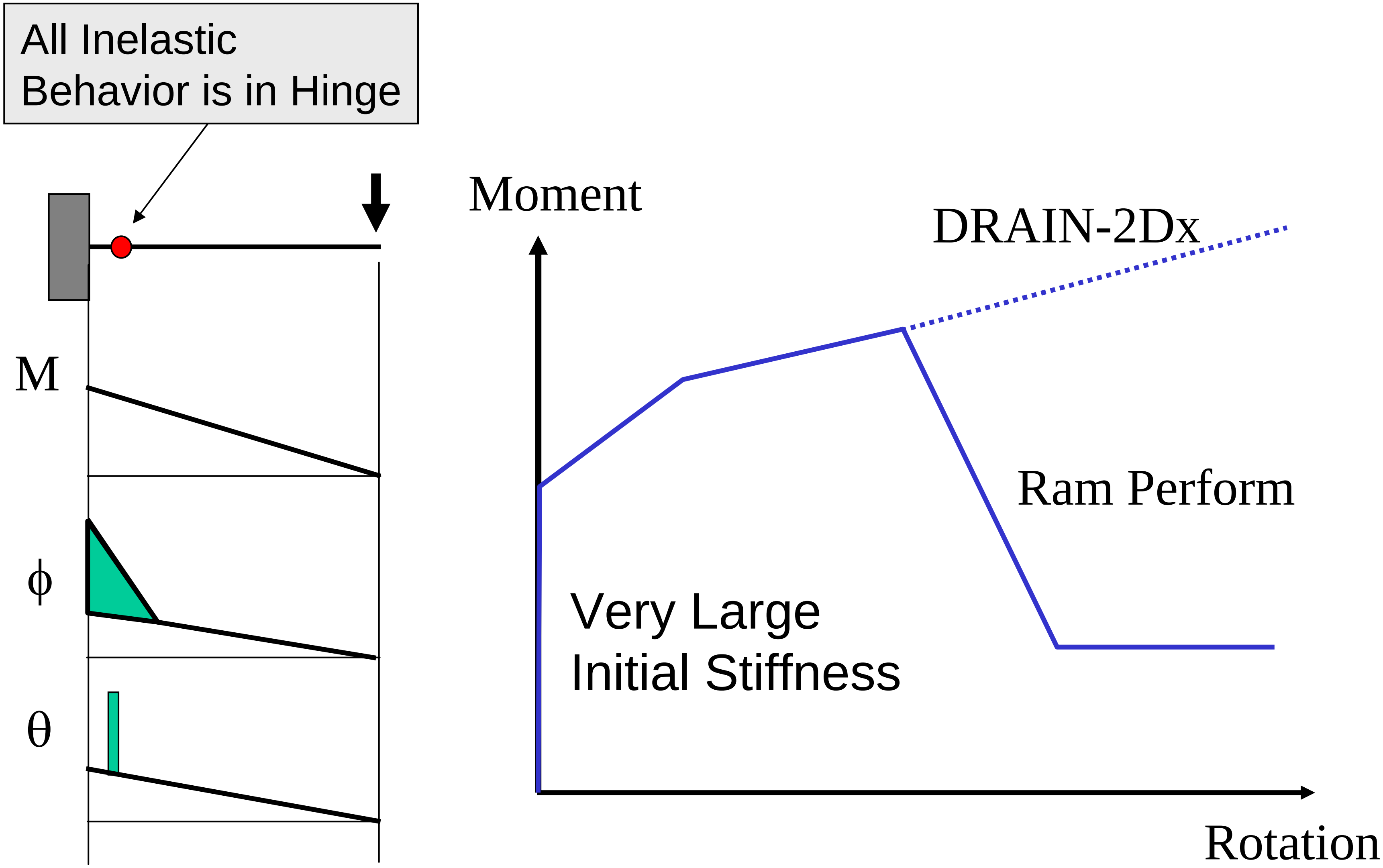
Uses of Compound Nodes

Girder Plastic Hinges

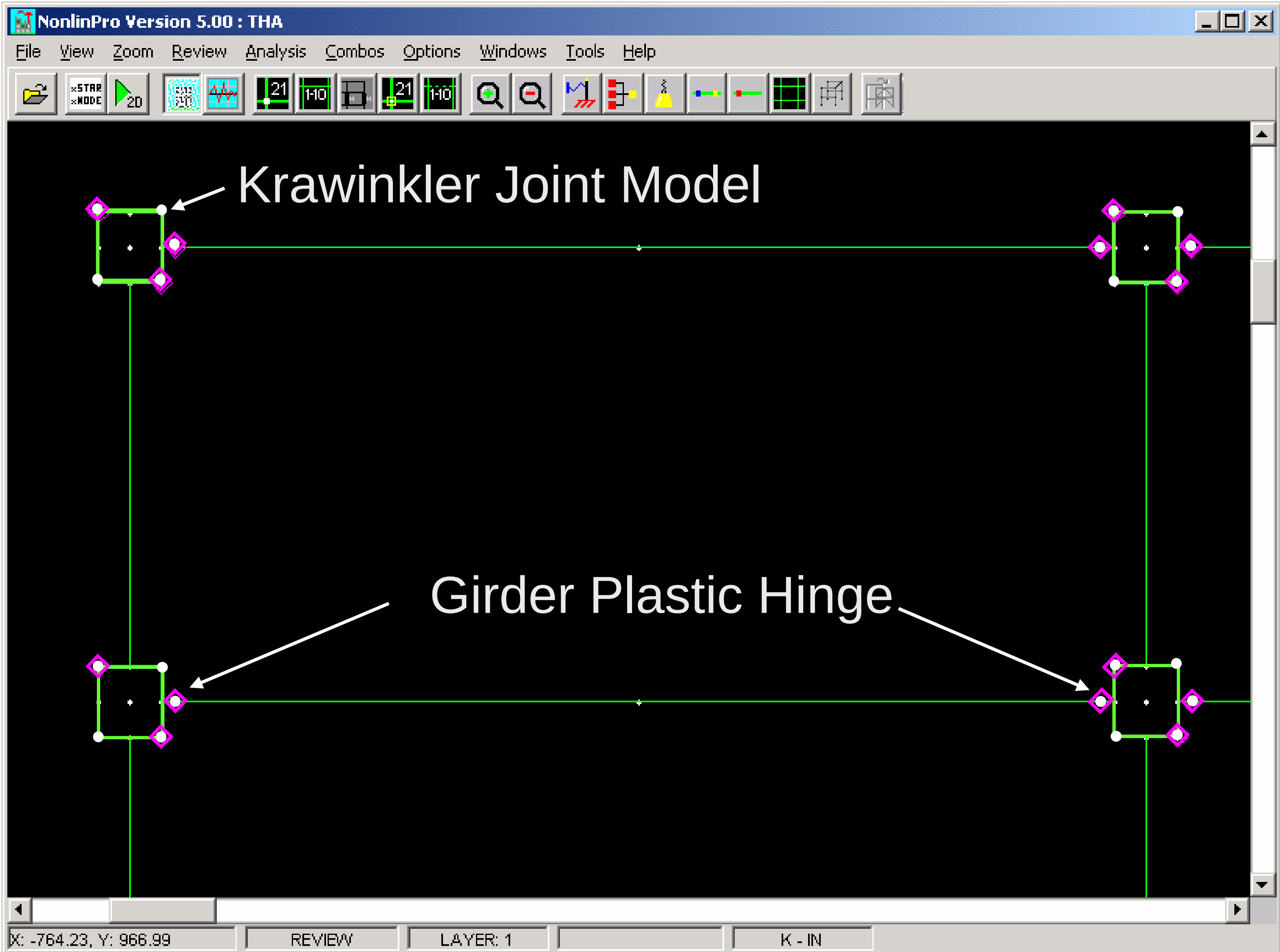
Panel Zone region of Beam-Column Joint



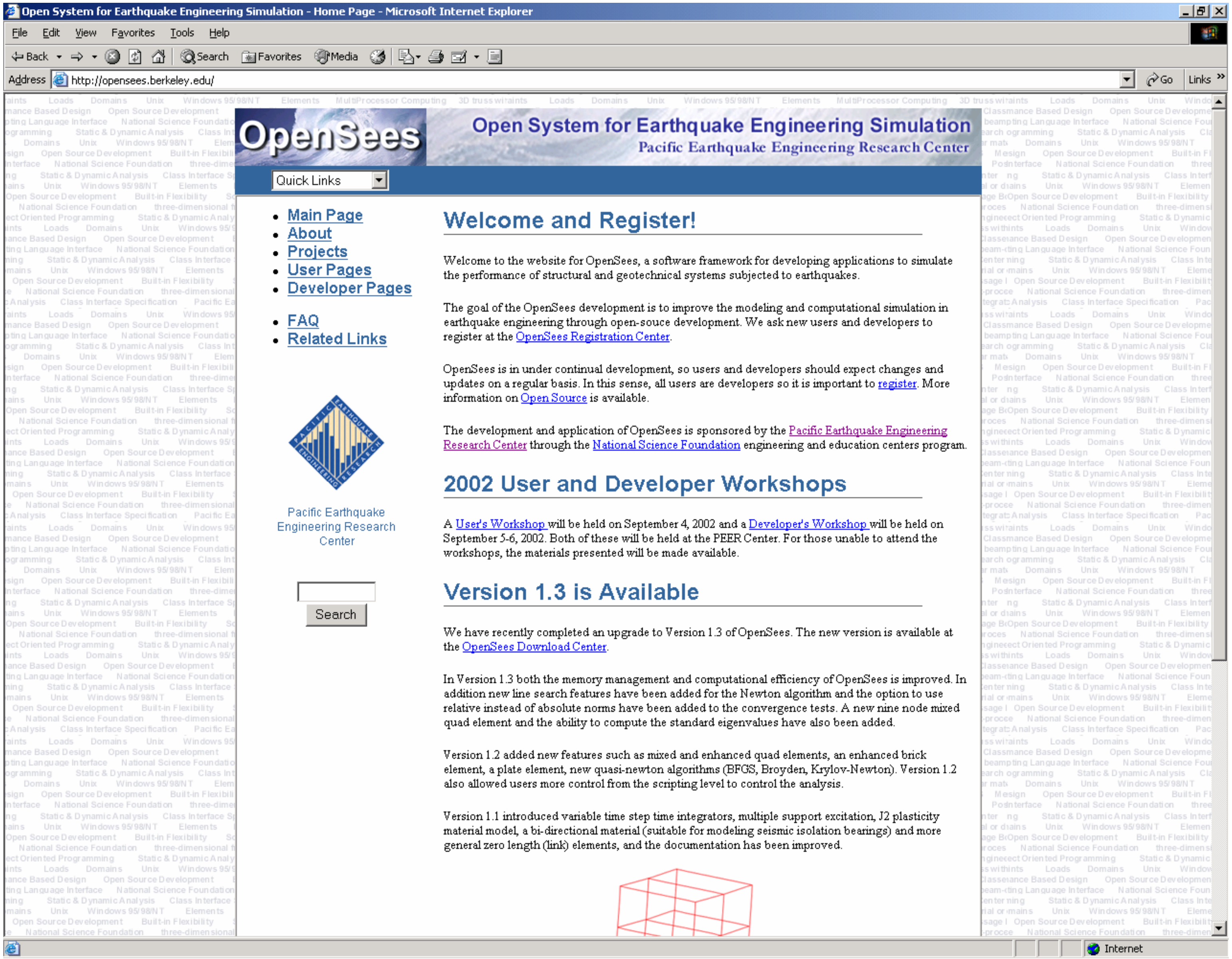
Development of Girder Hinge Model



Girder and Joint Modeling in NONLIN-Pro



The OpenSees Computational Environment

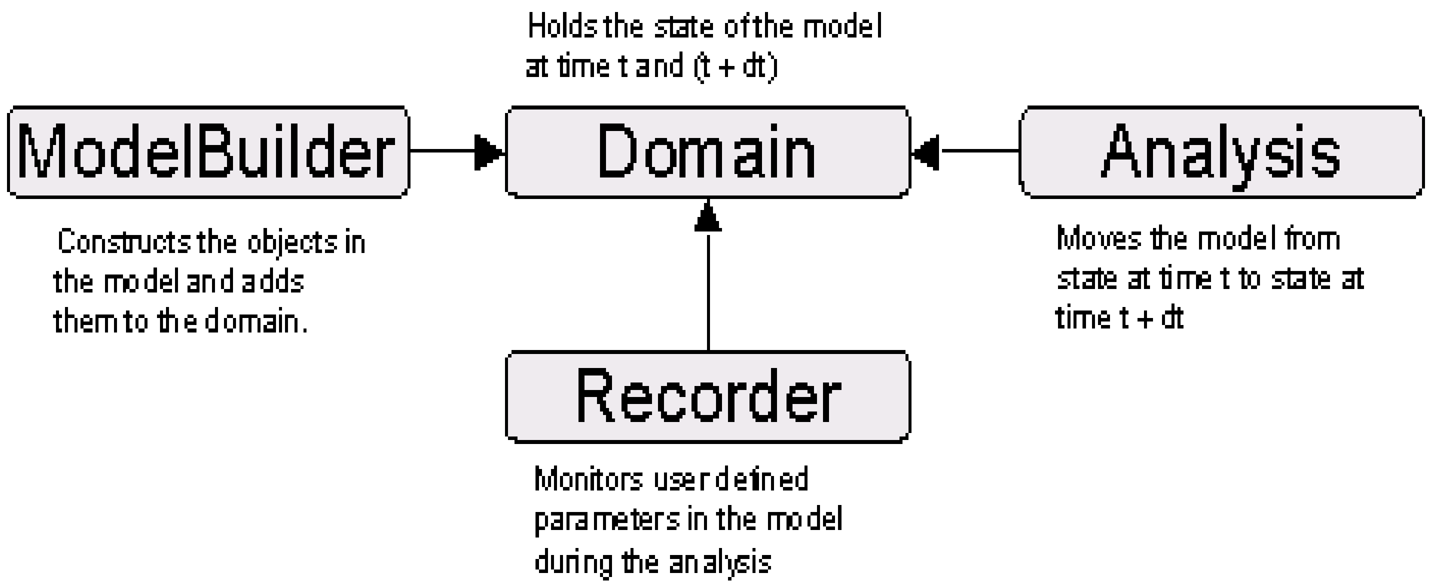


What is *OpenSees*?

- 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:



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

- Sections

Elastic section	Uniaxial section
Fiber section	Section aggregator
Plate fiber section	Bidirectional section
Elastic membrane plate section	

OpenSees Material Properties

- Uniaxial Materials

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

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:

  http://peer.berkeley.edu/~silva/Opensees/manual/html/

- 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).

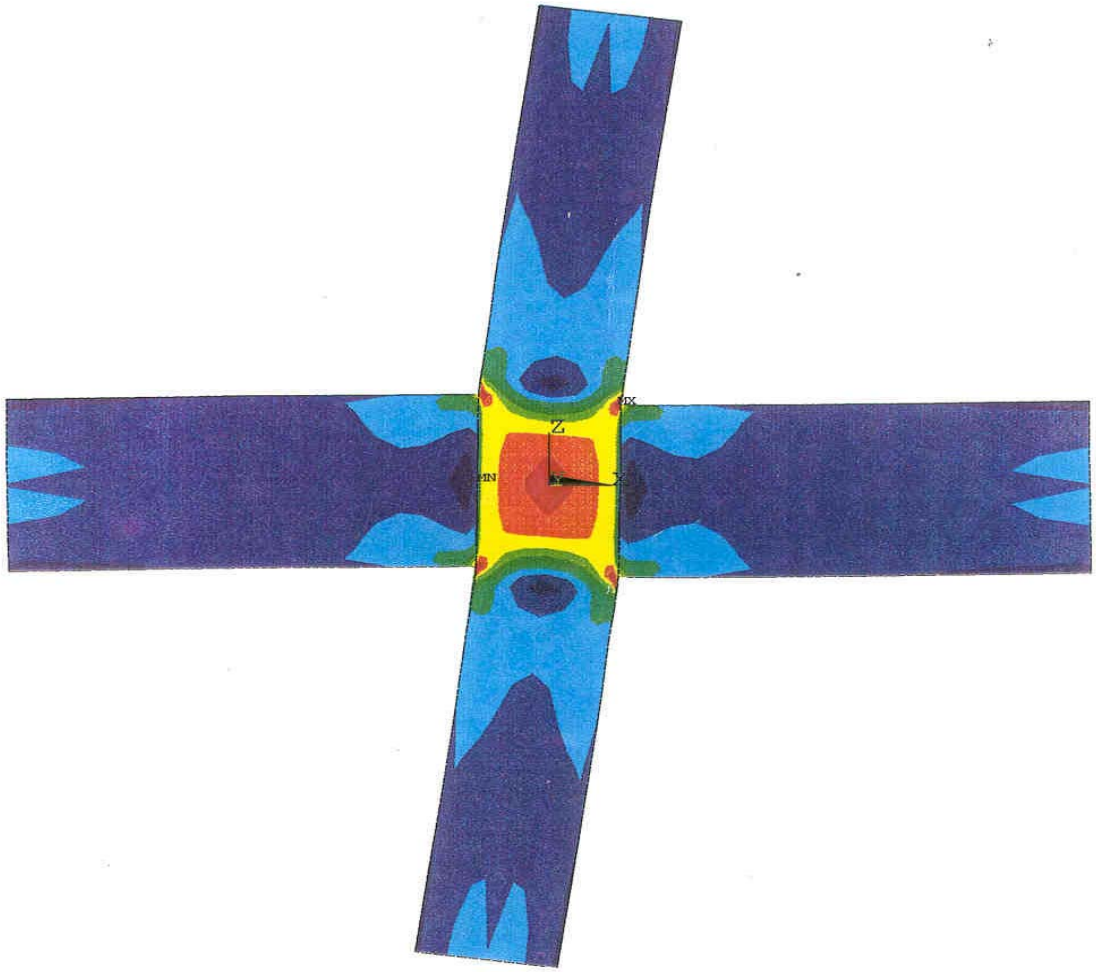
RAM/Perform

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.

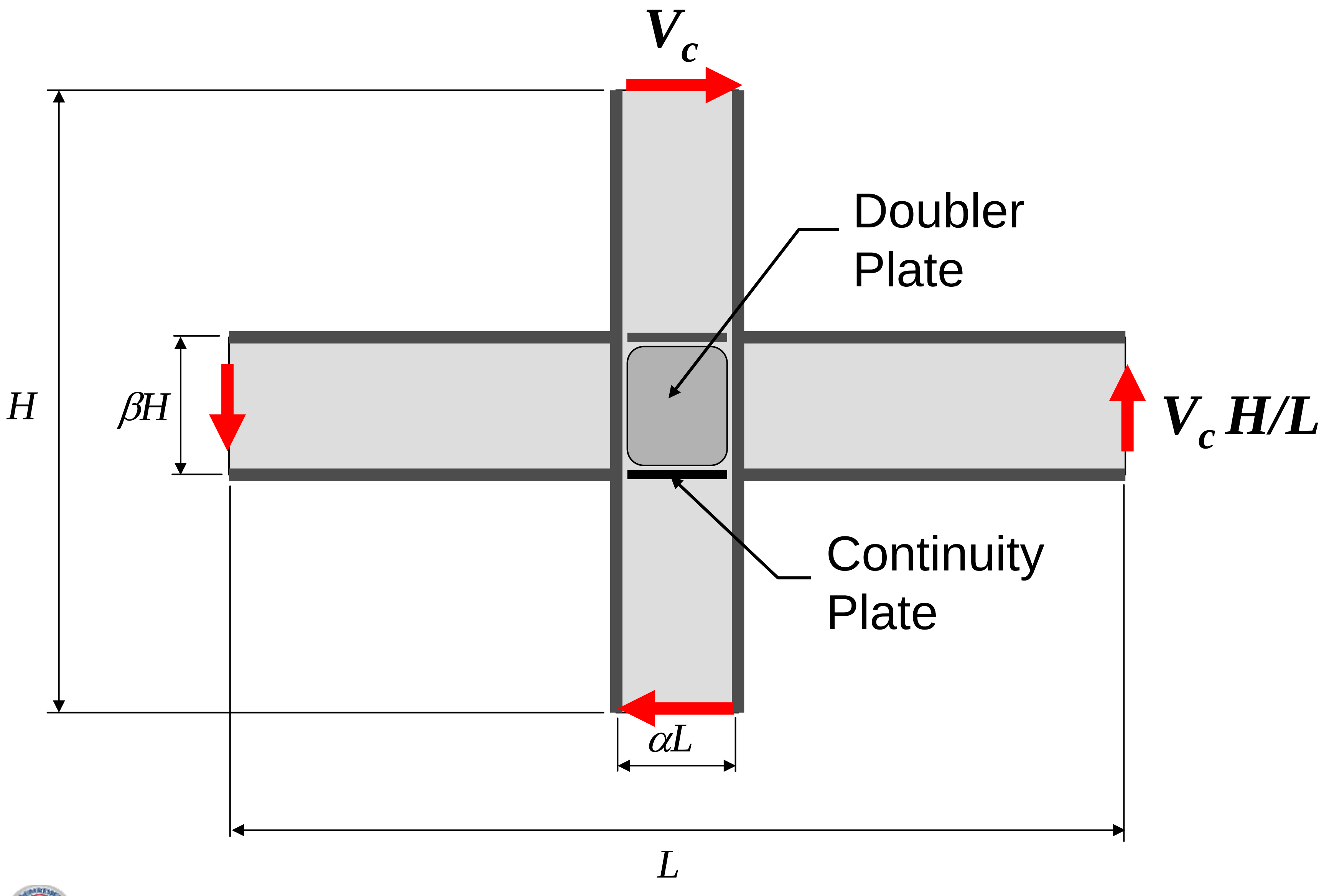
ABAQUS,ADINA, ANSYS, DIANA,NASTRAN

These are extremely powerful FEA programs but are not very practical for analysis of building and bridge structures.

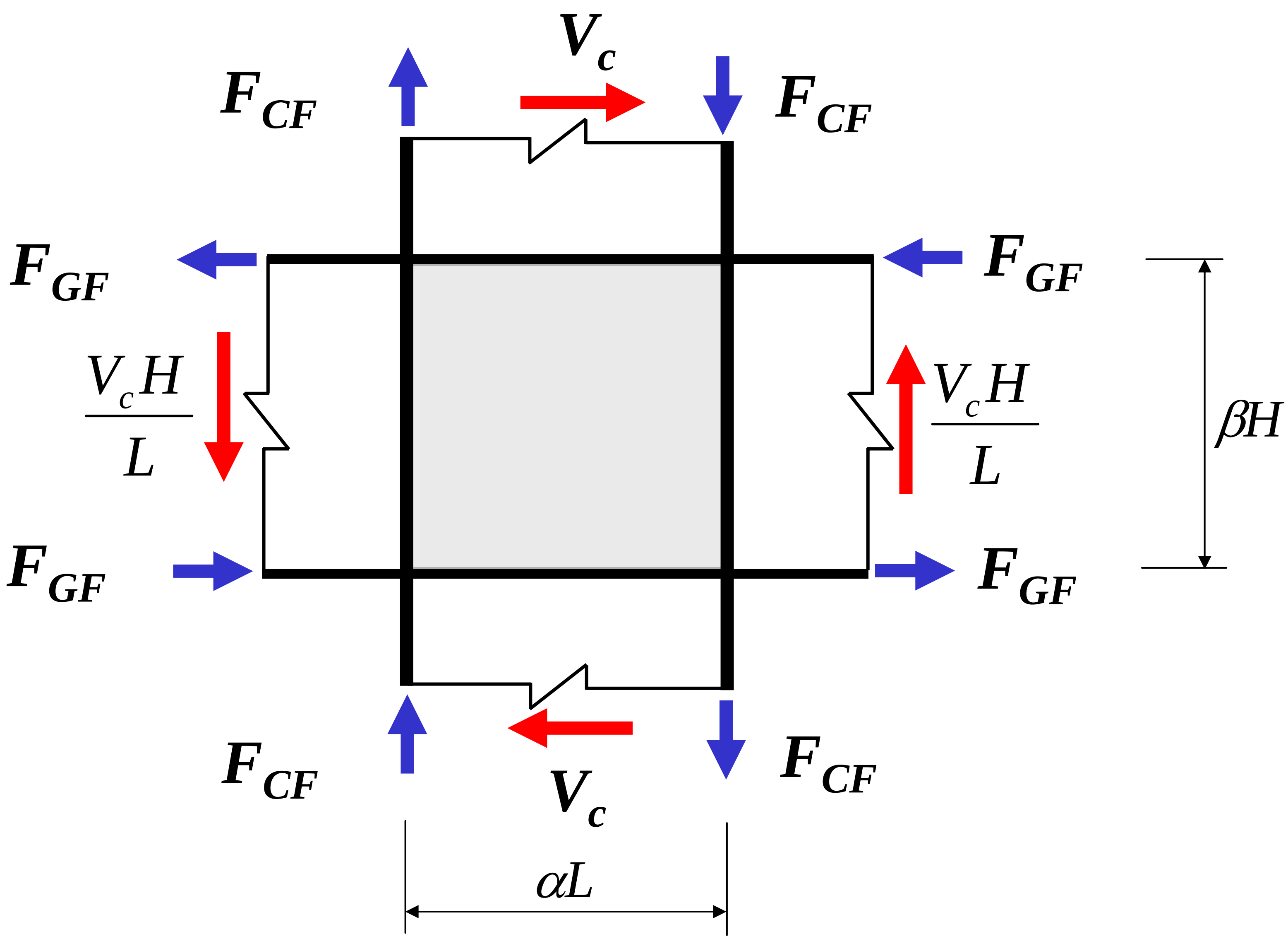
Modeling Beam-Column Joint Deformation In Steel Structures



Typical Interior Subassemblage



Equilibrium in Beam-Column Joint Region



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

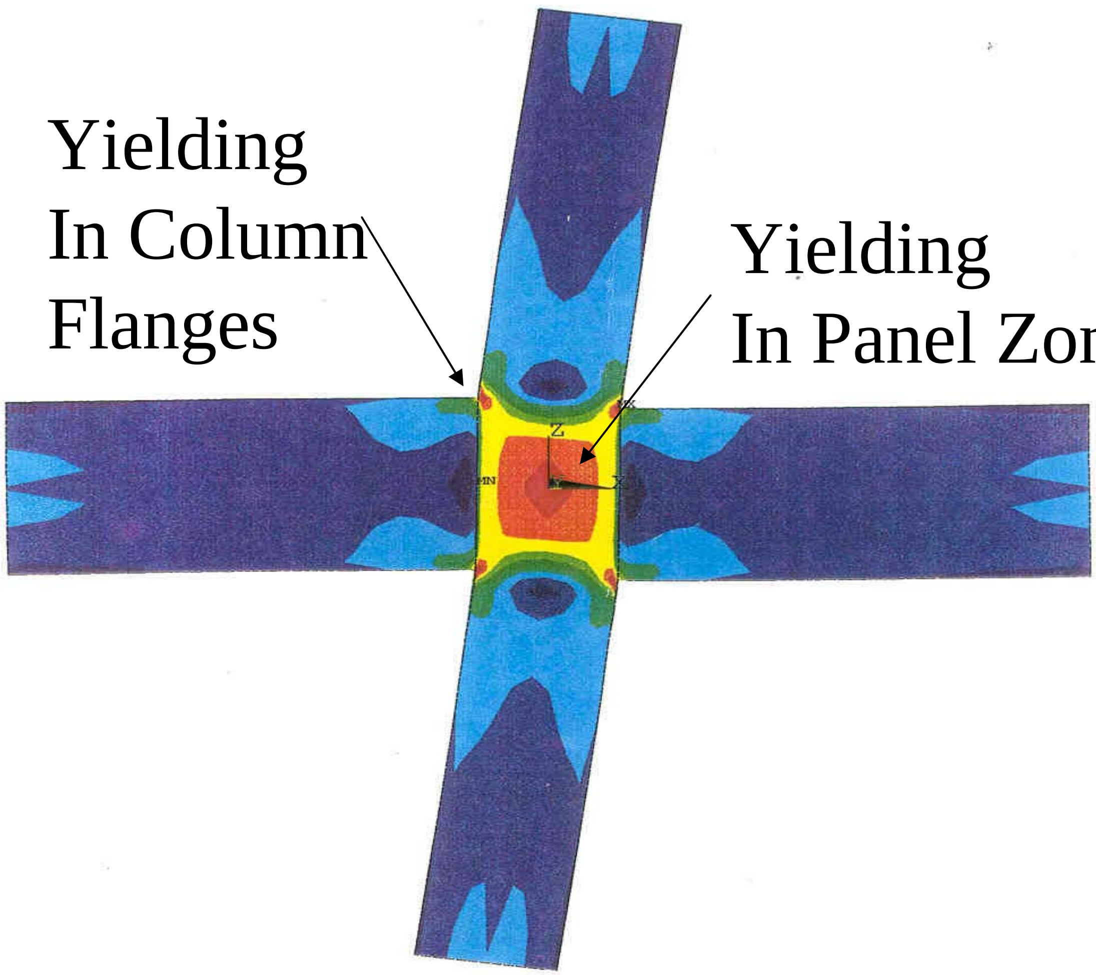
Effects of High Panel Zone Stresses

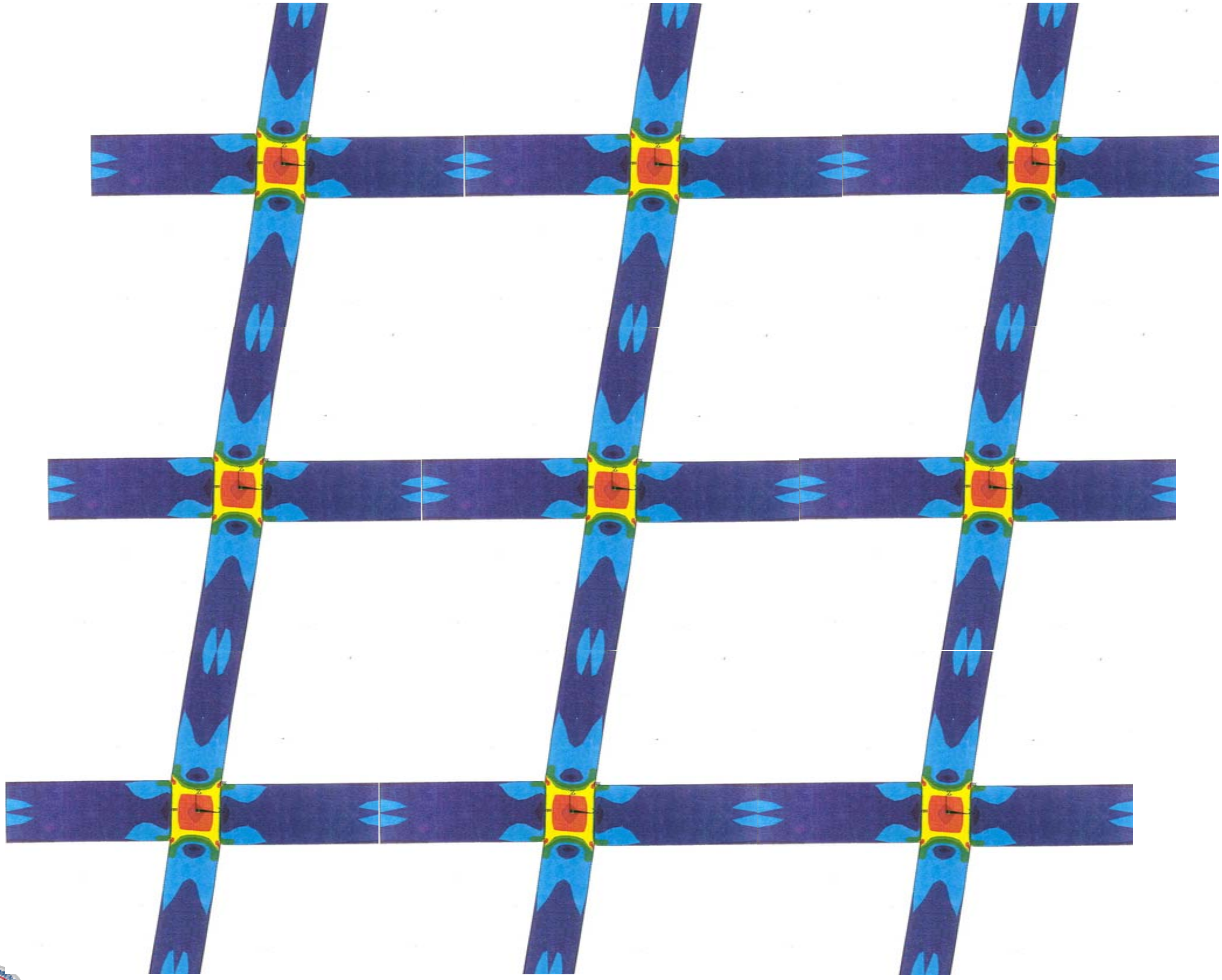
- Shear deformations in the panel zone can be responsible for 30 to 40 percent of the story drift. FEMA 350's statement that use of centerline dimensions in analysis will overestimate drift is *incorrect* for joints *without* PZ reinforcement.

- 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.

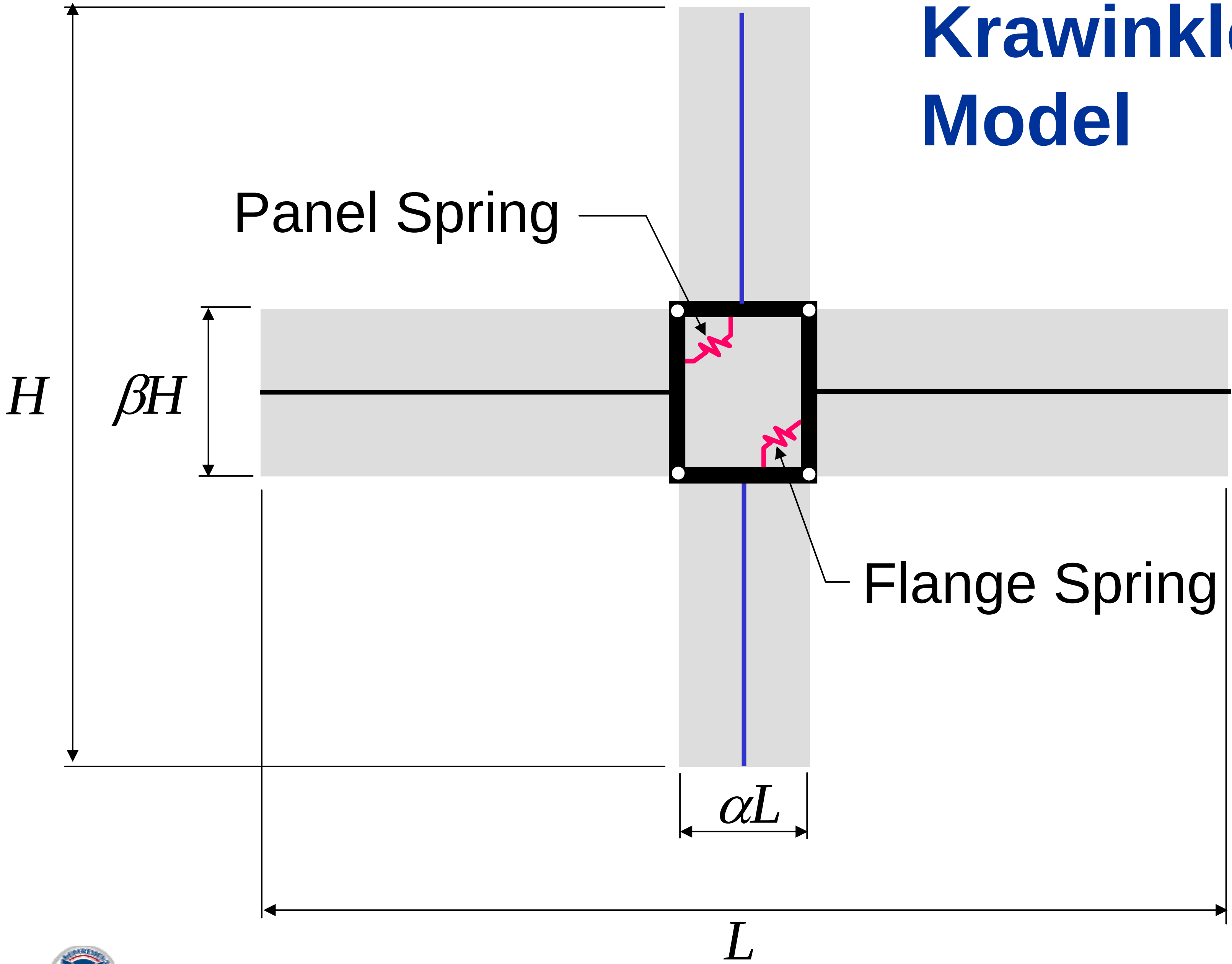
- Even with doubler plates, panel zones may yield. This inelastic behavior must be included in the model.

Sources of Inelastic Deformation in Typical Joint

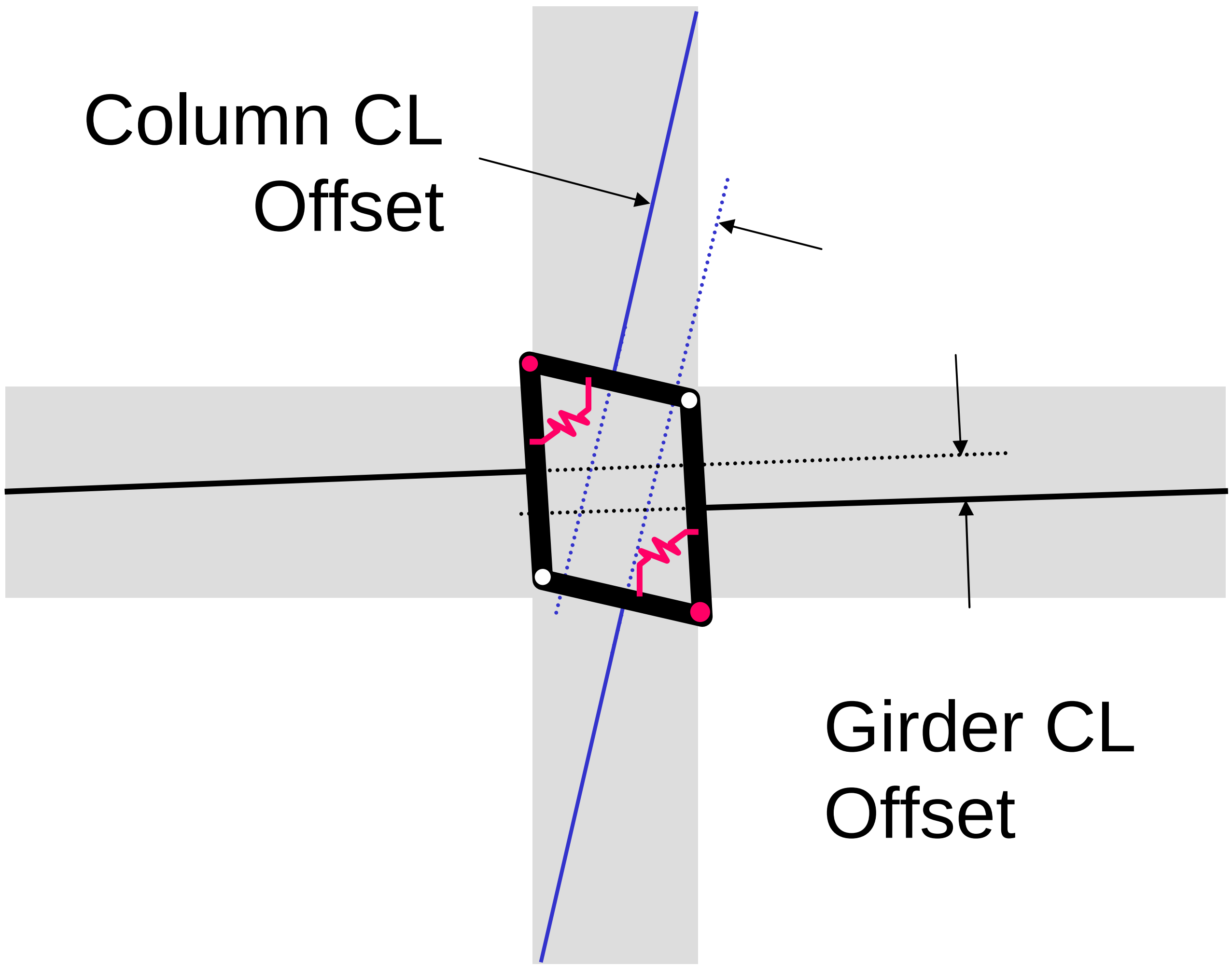




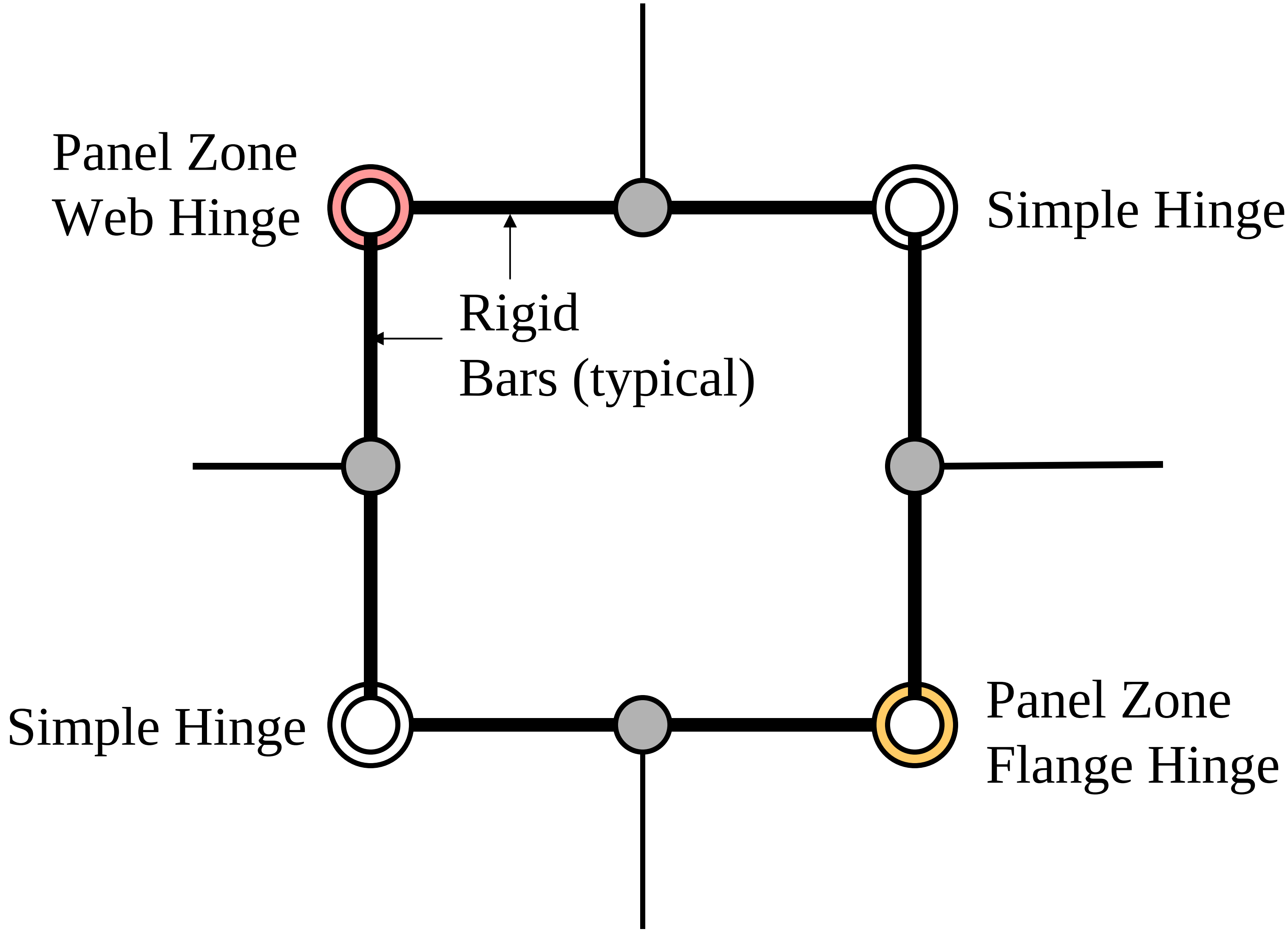
Krawinkler Model



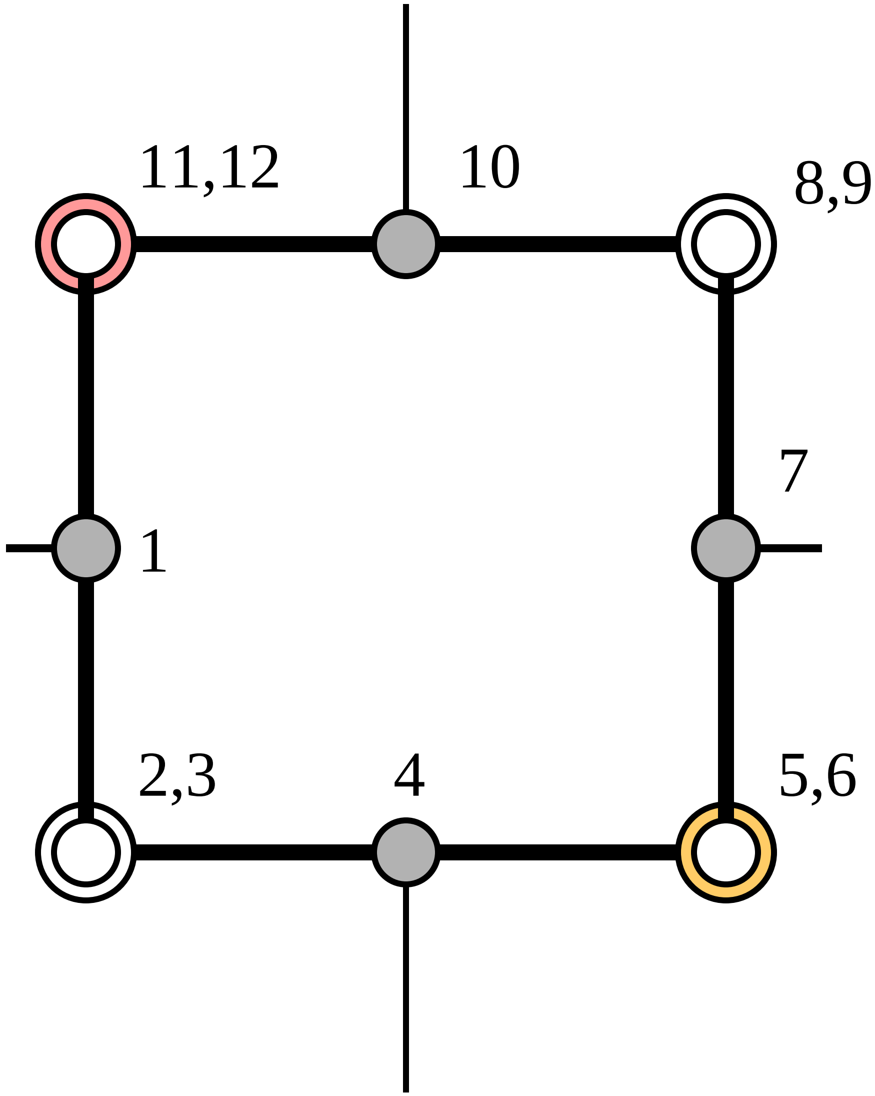
Kinematics of Krawinkler Model



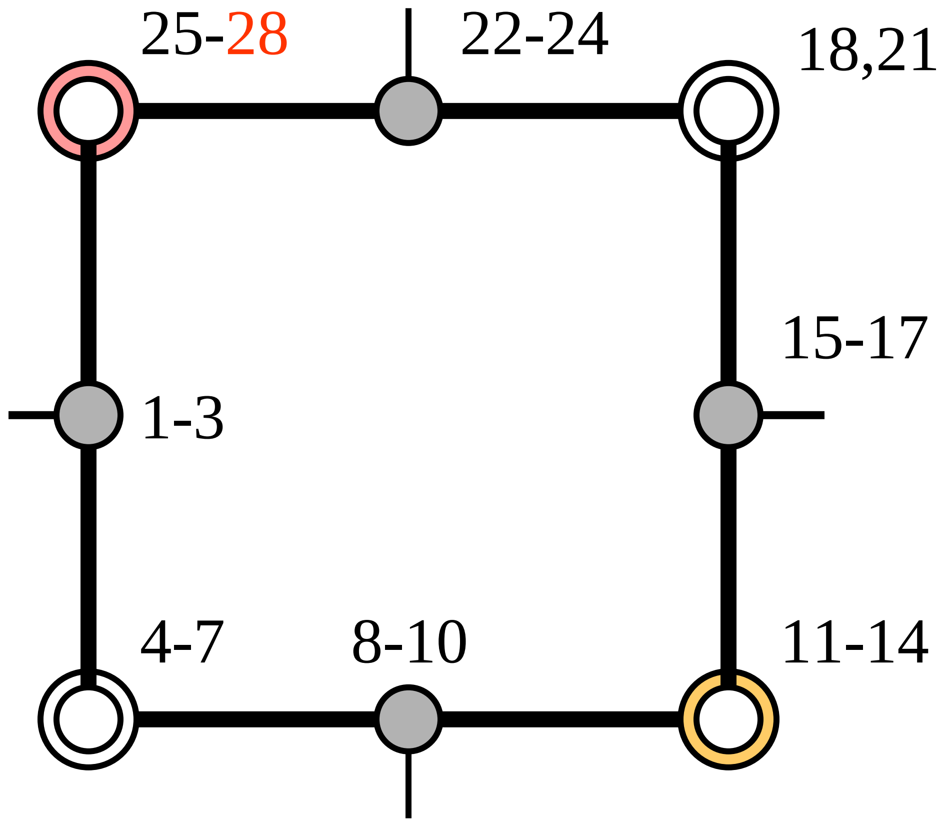
Krawinkler Joint Model



Nodes in Krawinkler Joint Model

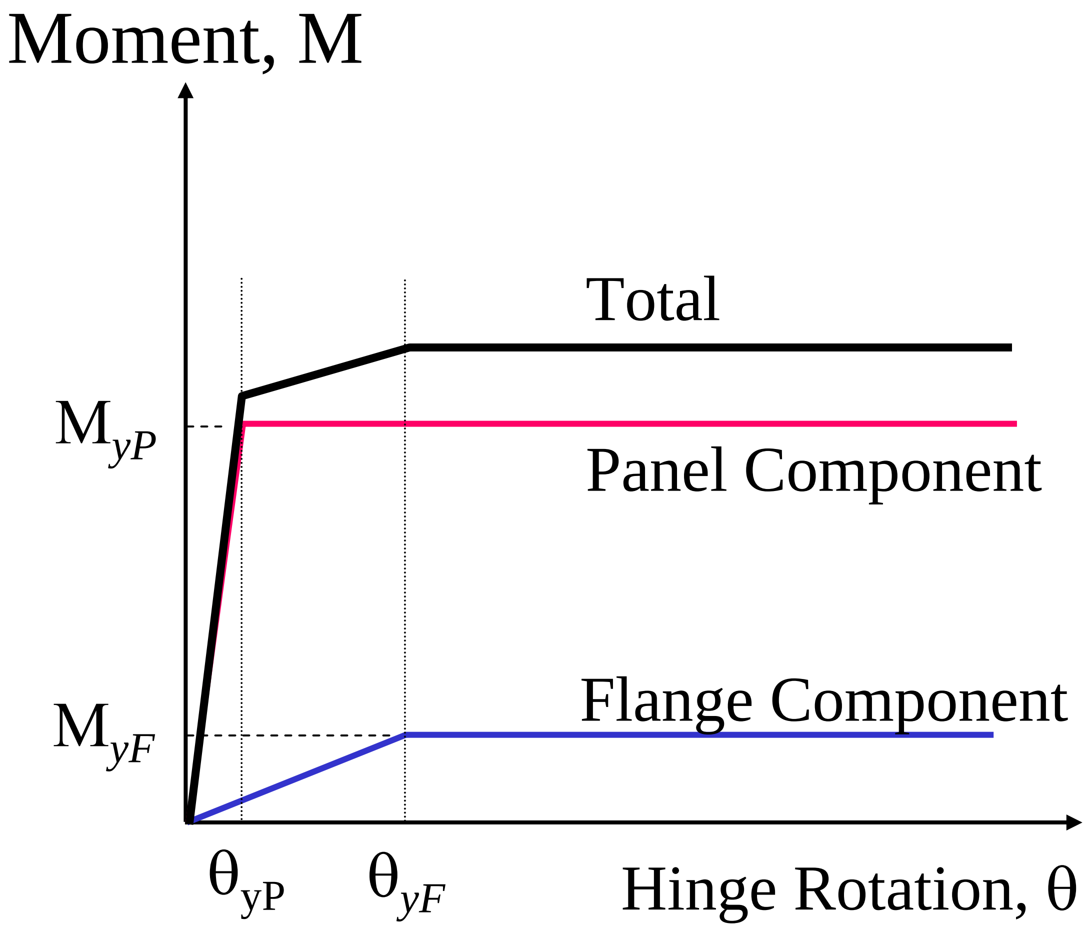


DOF in Krawinkler Joint Model

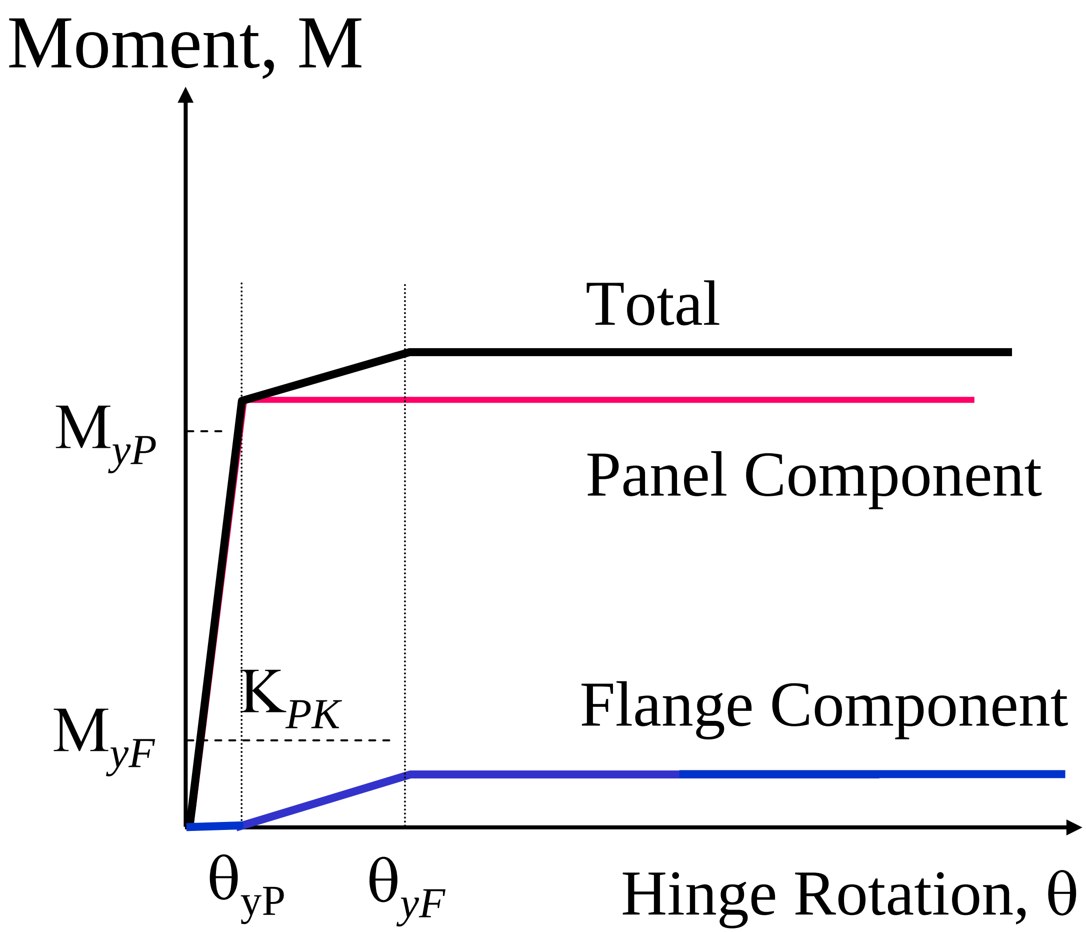


Note: Only FOUR DOF are truly independent.

Moment-Rotation Relationships in Krawinkler Model



Moment-Rotation Relationships in Krawinkler Model (Alternate)



Krawinkler Model Properties (Panel Component)

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

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

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

Krawinkler Model Properties (Panel Component)

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

Volume of Panel

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

Krawinkler Model Properties (Flange Component)

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

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

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

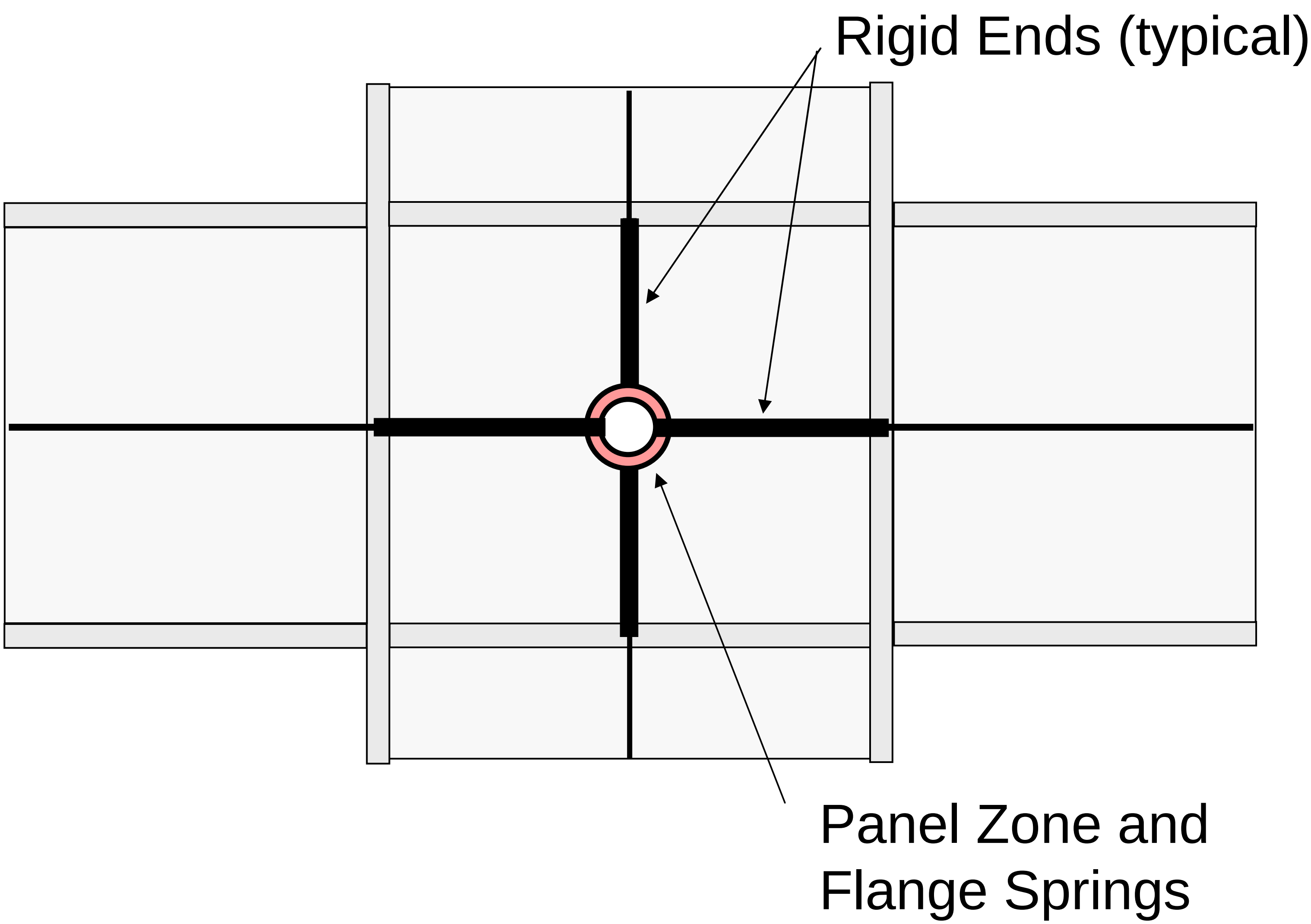
- Modeling parameters are independent of structure outside of panel zone region

Disadvantages of Krawinkler Model

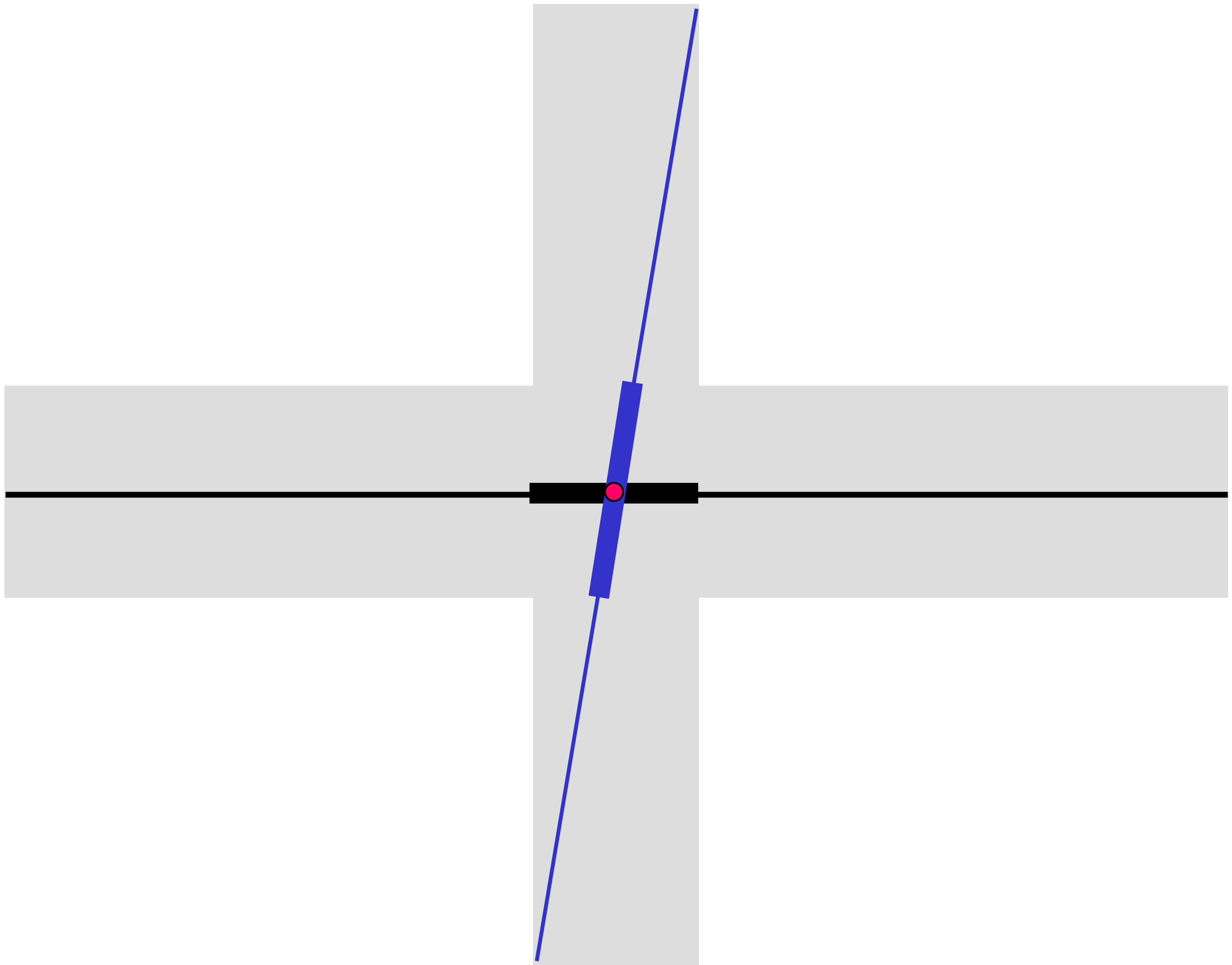
- Model is relatively complex
- Model does not include flexural deformations in panel zone region
- Requires 12 nodes, 12 elements, and 28 degrees of freedom

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

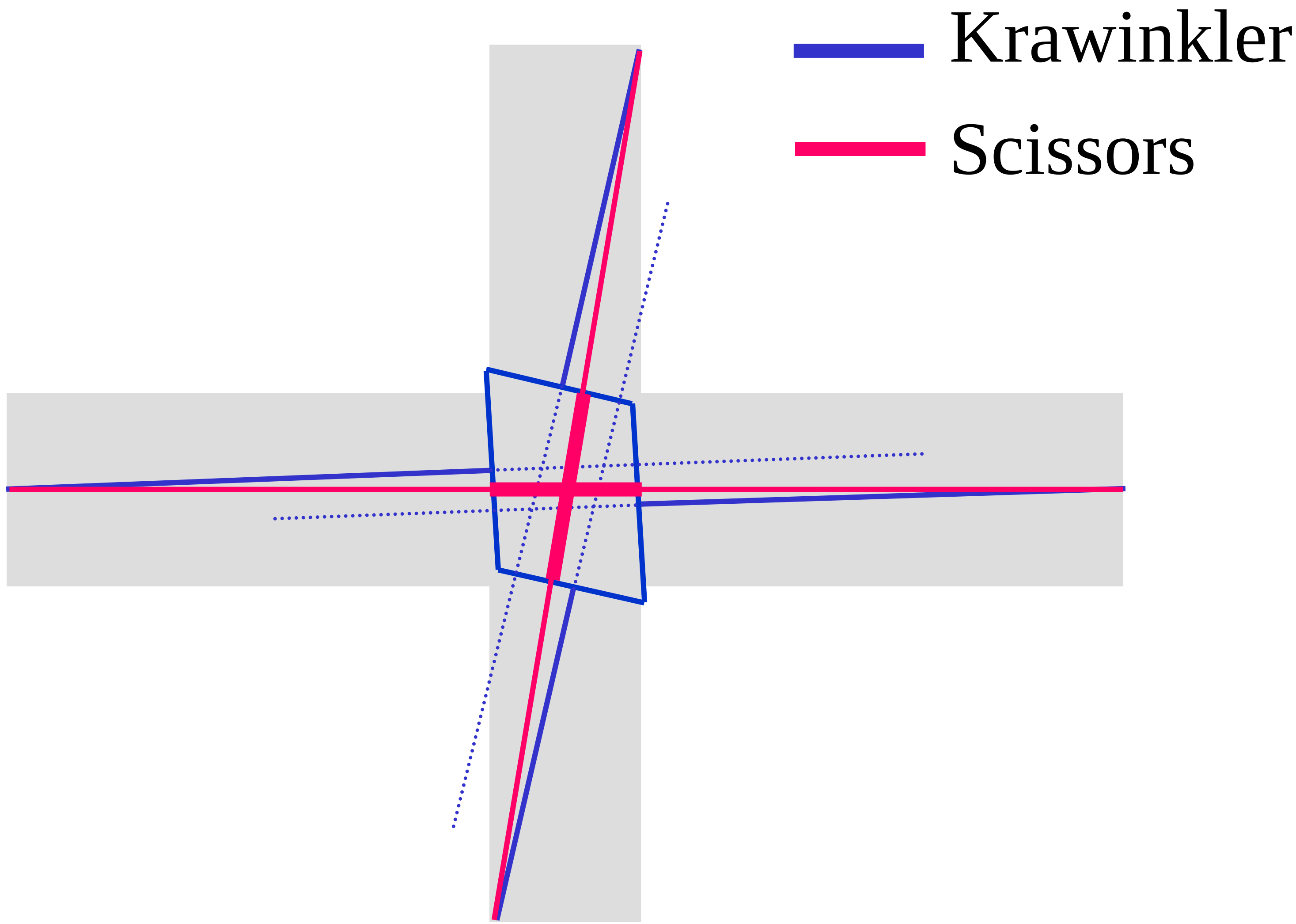
Scissor Joint Model



Kinematics of Scissors Model



Model Comparison: Kinematics



Mathematical Relationship Between Krawinkler and Scissors Models

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

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

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)

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:

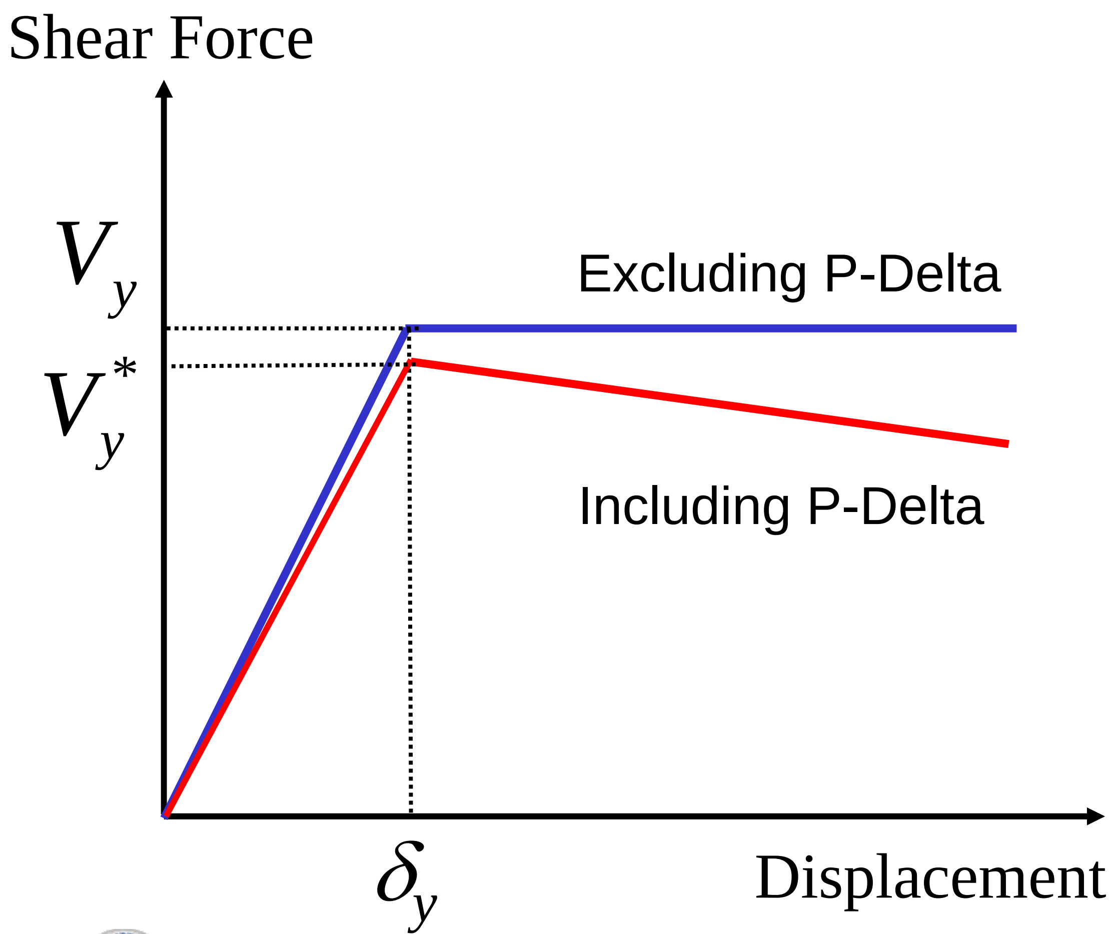
“ The models for columns should reflect the influence of axial load when axial loads exceed 15 percent of the buckling load”

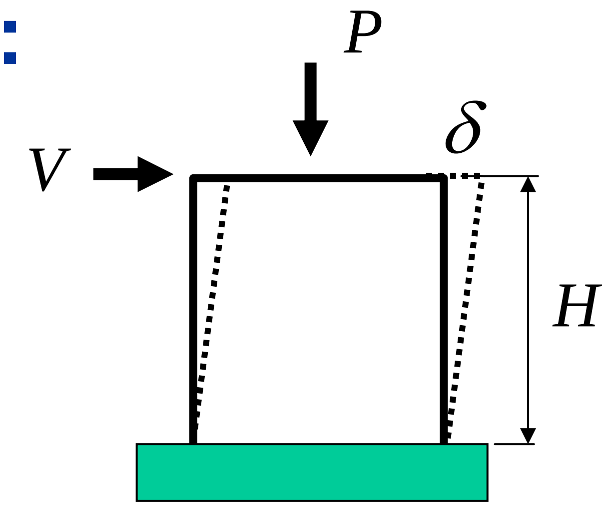
Recommended Revision:

“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





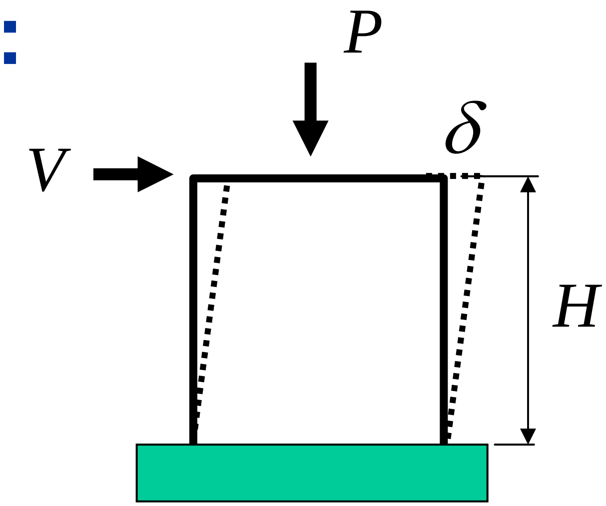
$$K_G = -\frac{P}{H}$$

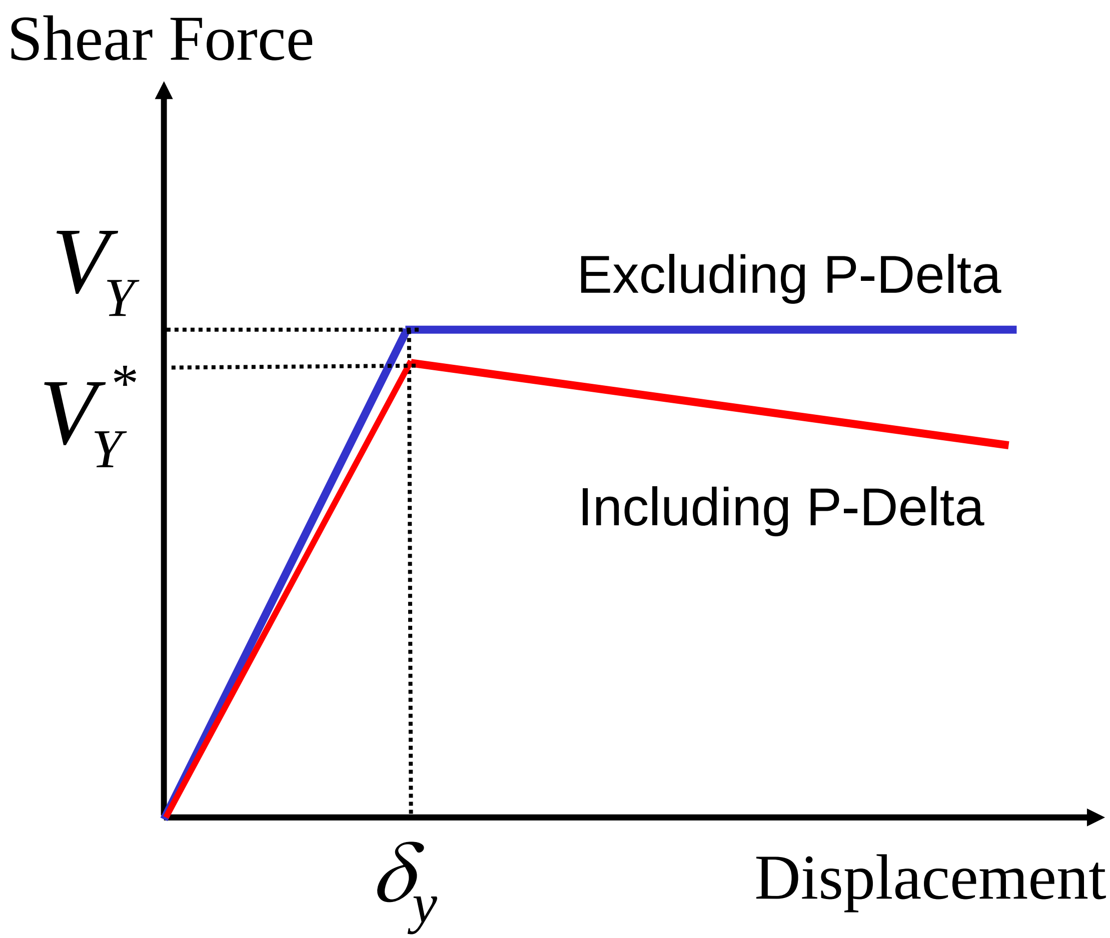
$$K_E = \frac{V_y}{\delta_y}$$

$$K = K_E + K_G$$

Influence of P-Delta Effects:

2) Loss of Strength



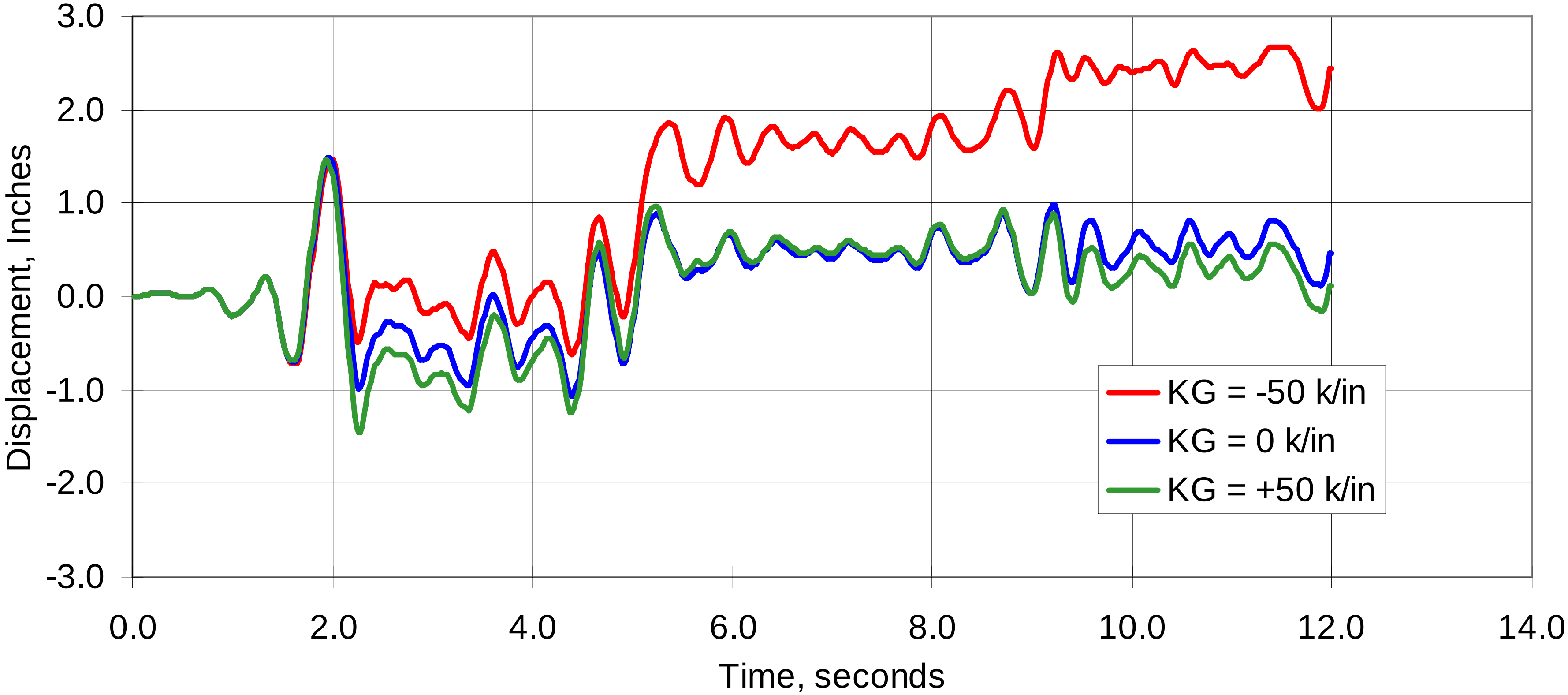


$$\theta = \frac{P\delta_y}{V_y H}$$

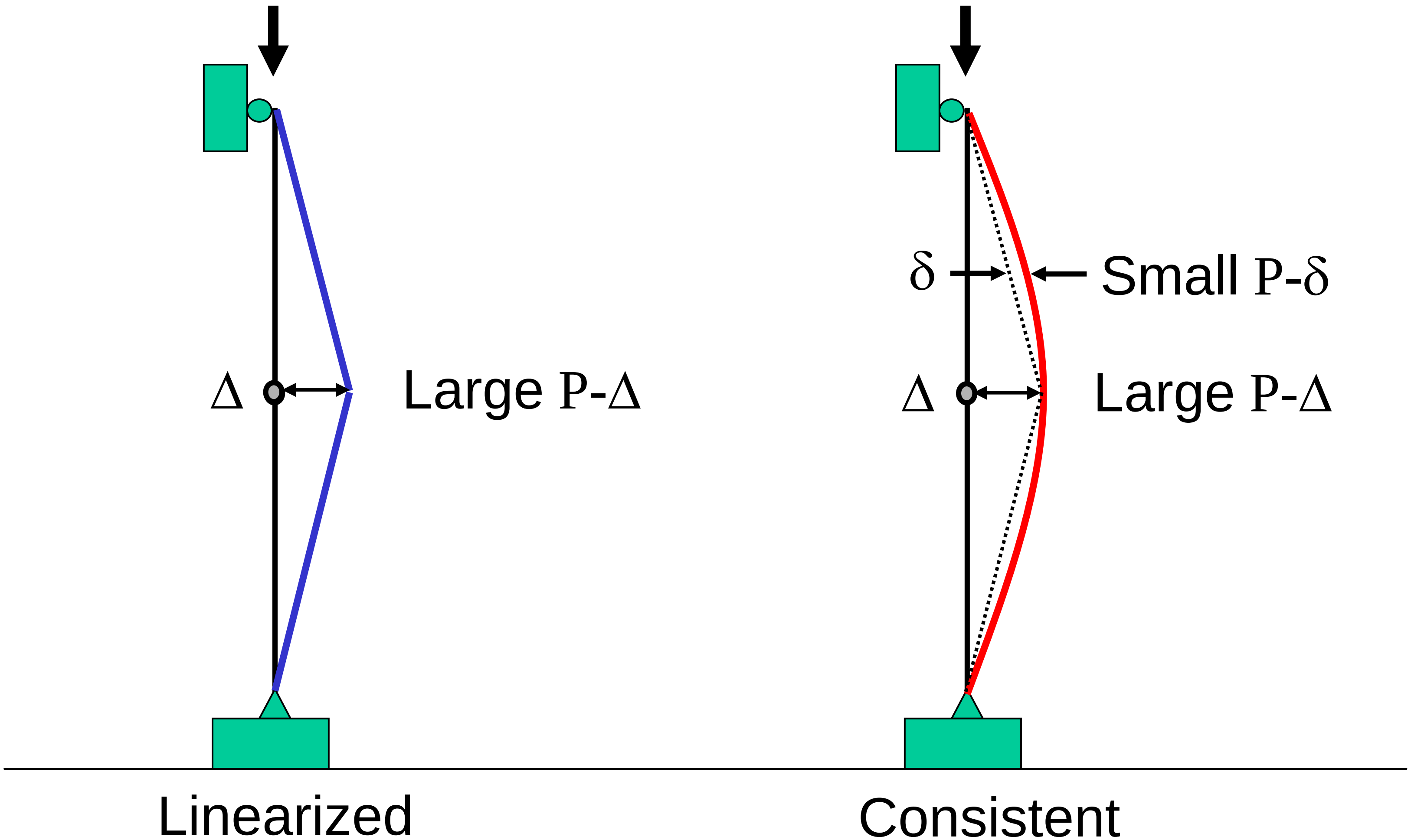
$$V_y^* = V_y(1-\theta)$$

Influence of P-Delta Effects:

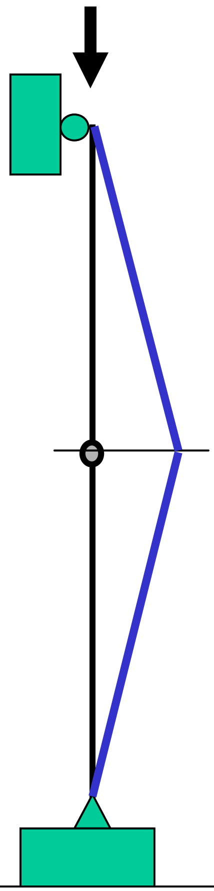
3) Larger residual deformations and increased tendency towards dynamic instability



Modeling P-Delta Effects
Linearized vs Consistent Geometric Stiffness



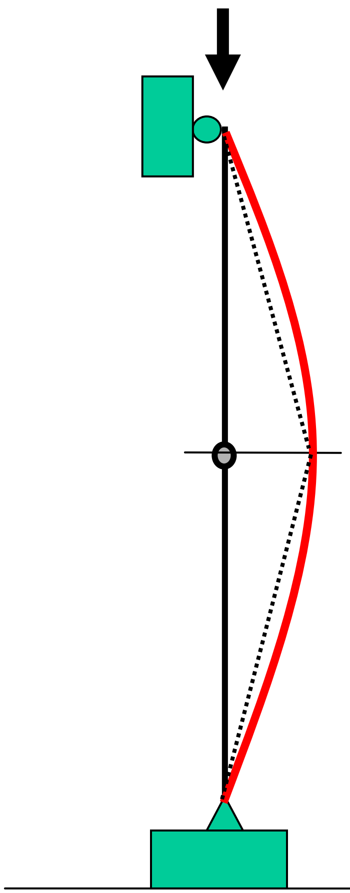
Modeling P-Delta Effects
Linearized Geometric Stiffness



- 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

Linearized

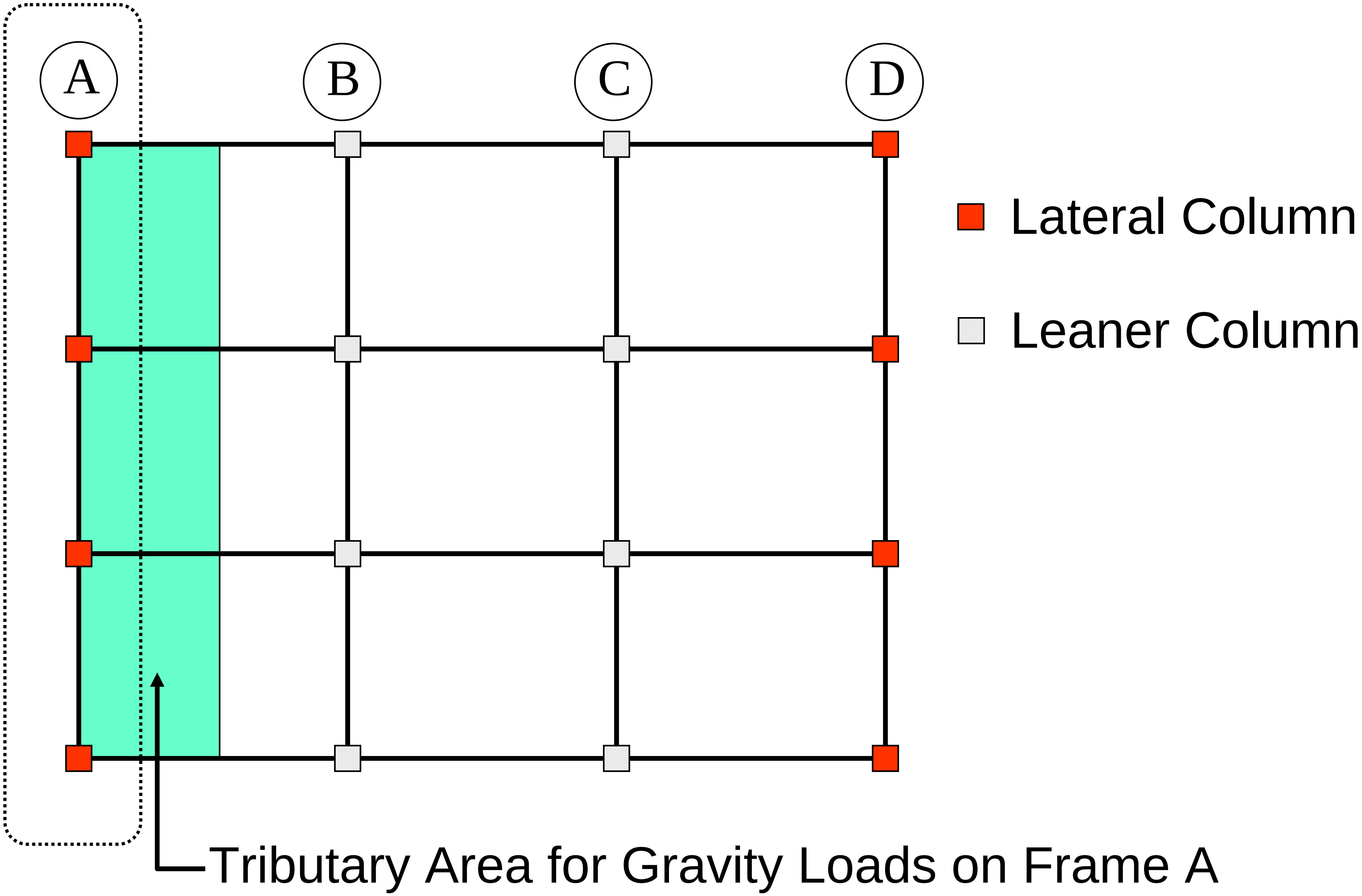
Modeling P-Delta Effects Consistent Geometric Stiffness



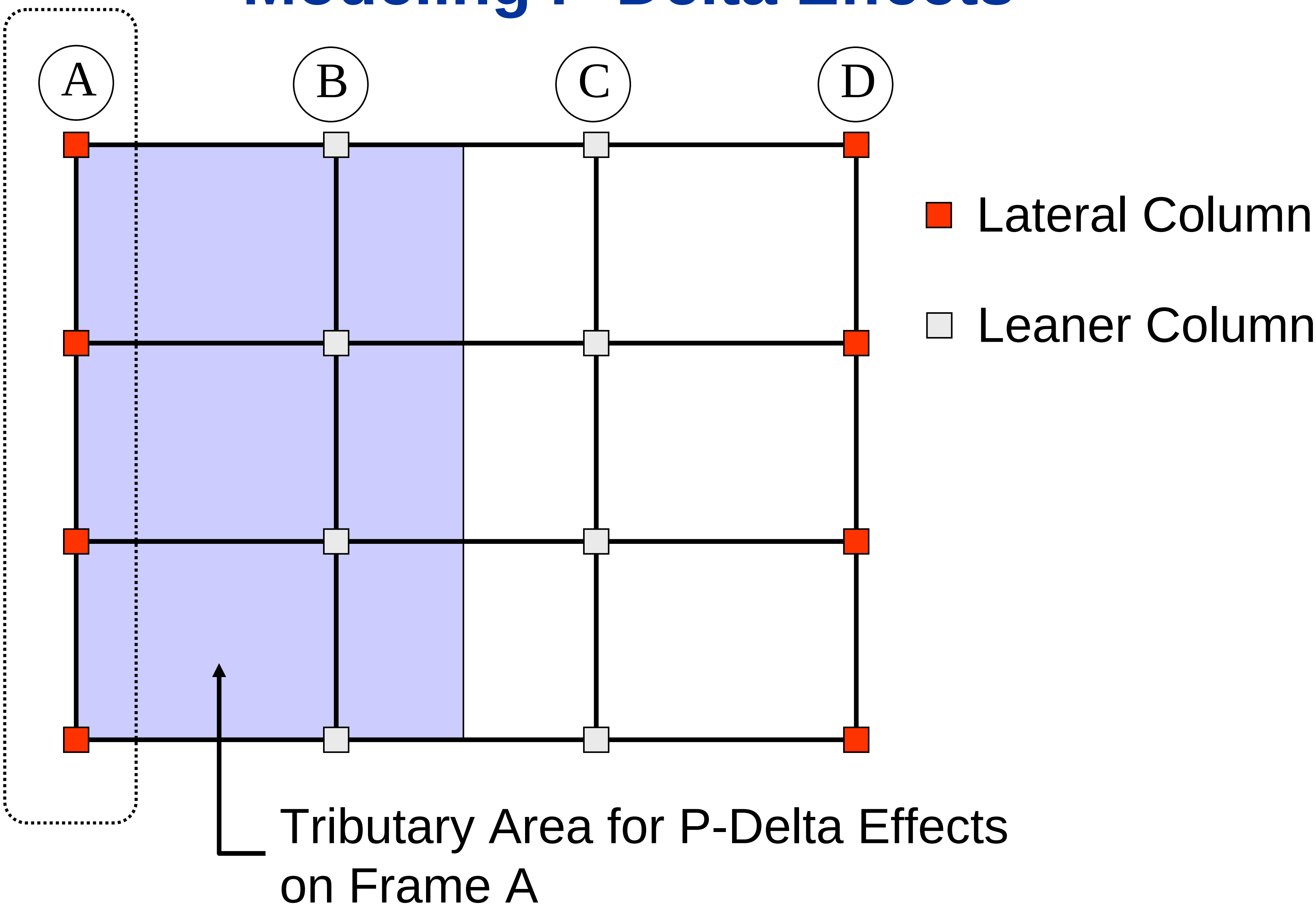
Consistent

- 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.

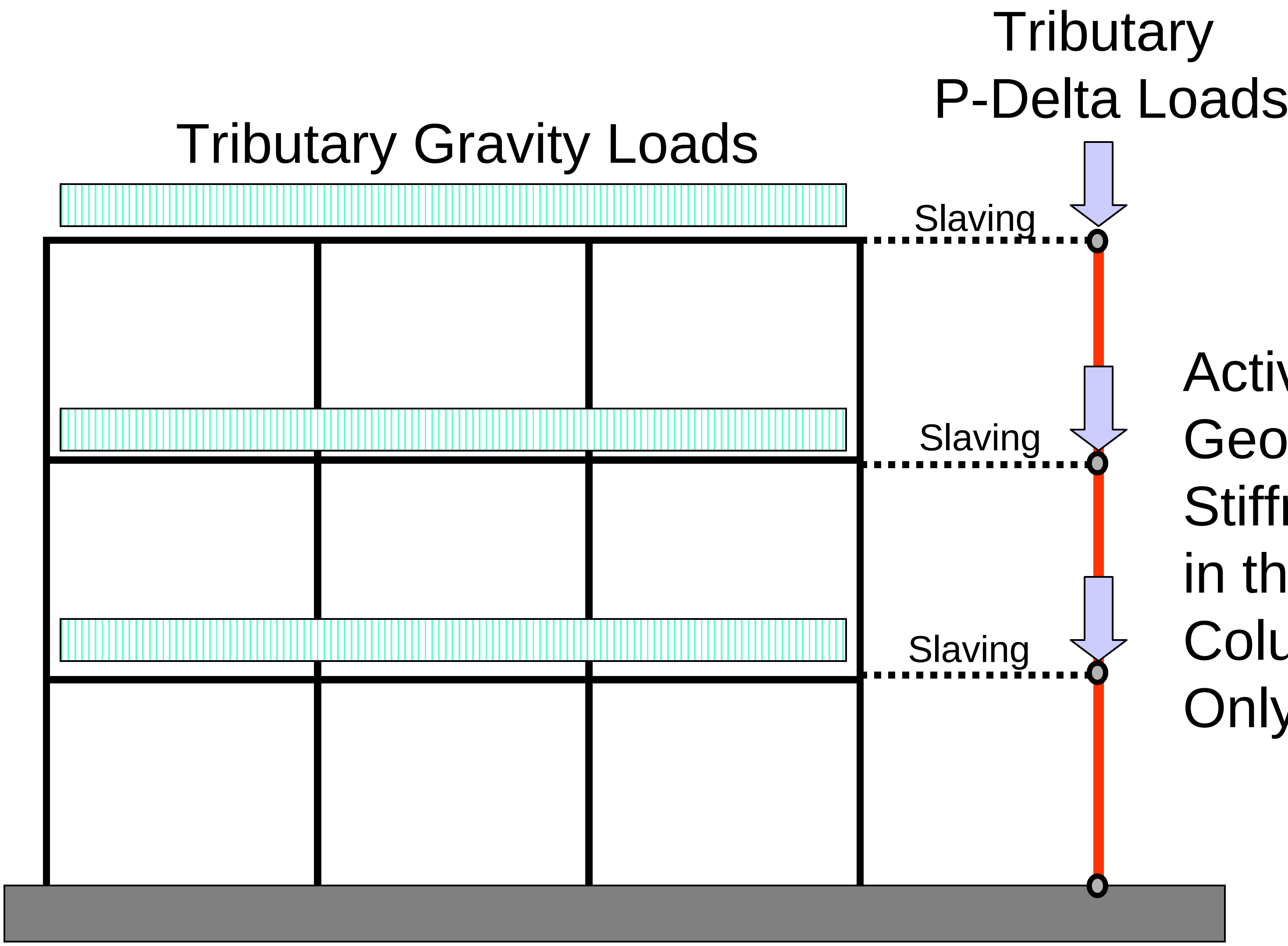
Modeling P-Delta Effects



Modeling P-Delta Effects



Modeling P-Delta Effects

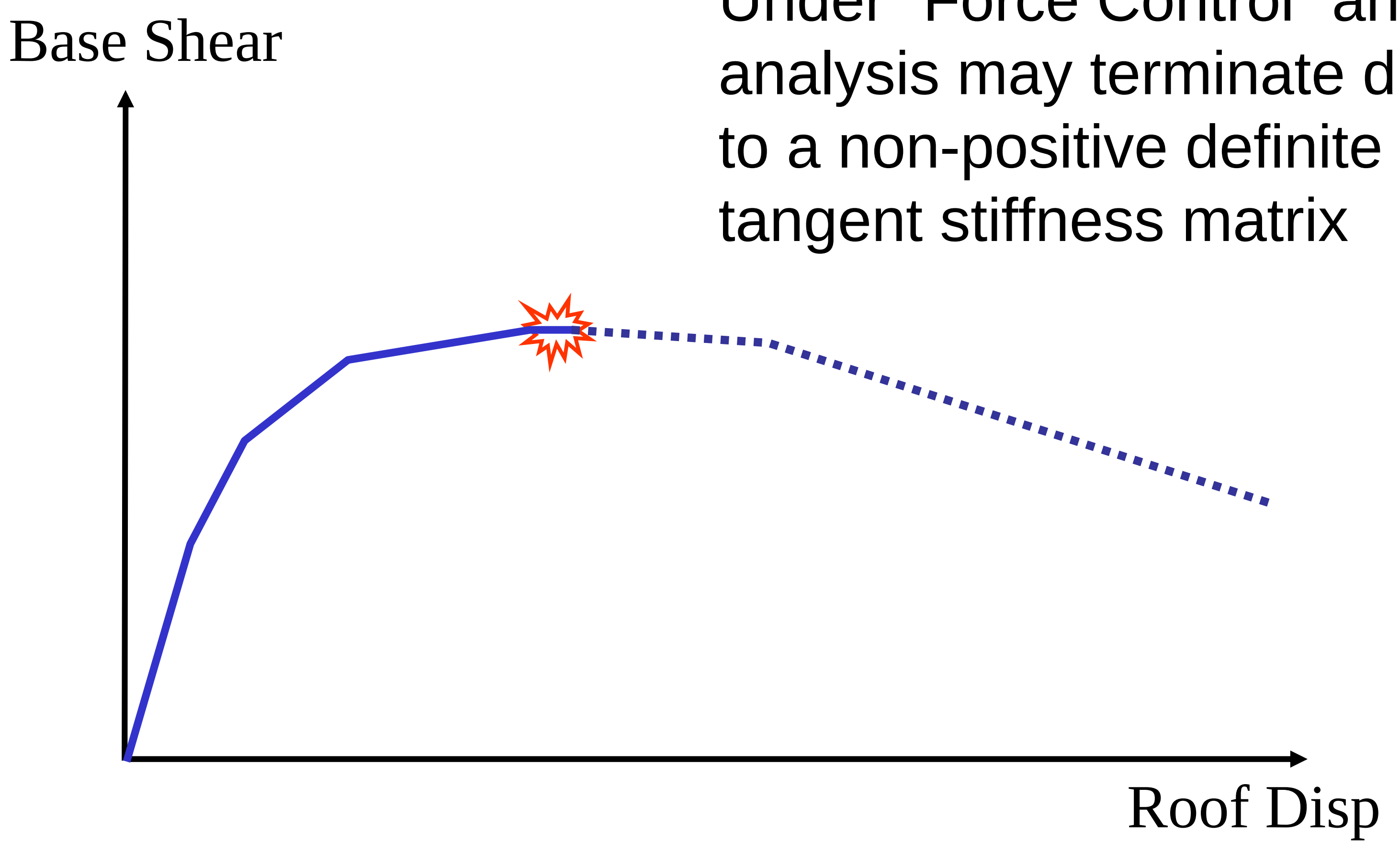


Activate Geometric Stiffness in these Columns Only.

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

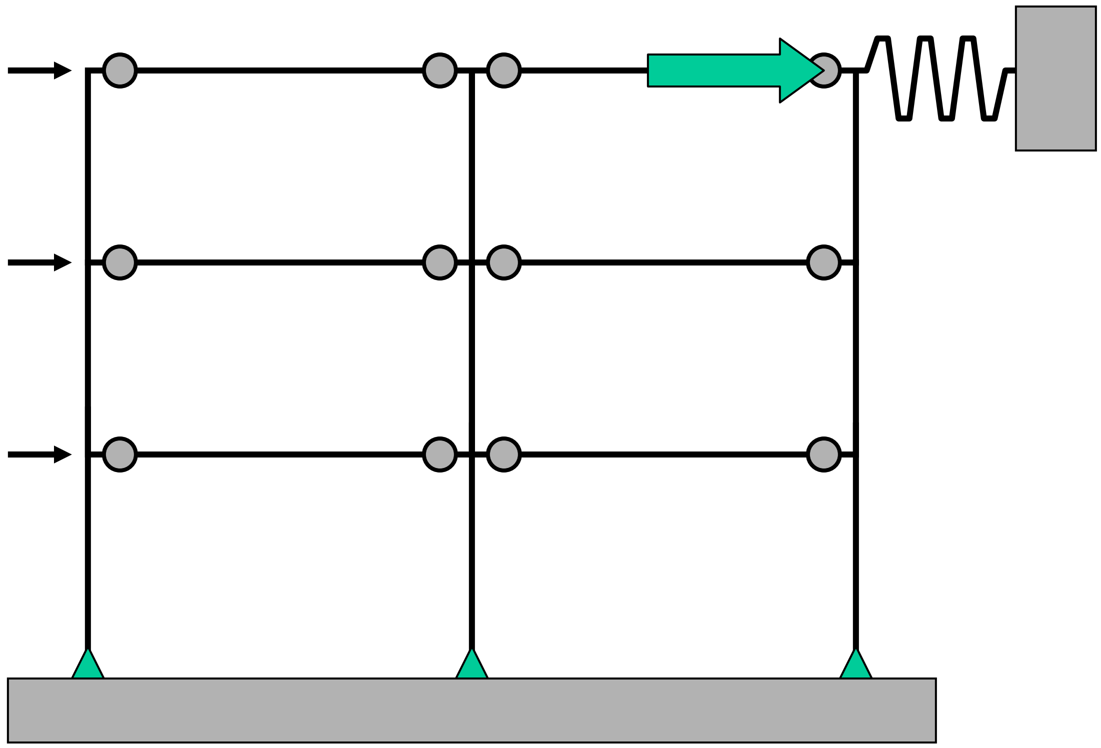
- Full Dead Load
- 10 PSF Partition Load (or computed value if available)
- Full Reduced Live Load (as would be used for column design).
- Reduced Live Load based on most probable live load. See for example Commentary of ASCE 7.
- Effect of Vertical Accelerations?

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.