Nonlinear Modeling of Structures

(using PERFORM 3D)

CSI Seminar by Graham H. Powell

Notes: Fawad A. Najam

PERFORMANCE BASED DESIGN USING NONLINEAR ANALYSIS

PERFORM 3D SEMINAR

Session 1

Important aspects of Nonlinearity:

- Material nonlinearity
  - Caused by inelastic material behavior
  - Gap opening/closing etc.
- Geometric nonlinearity
  - Caused by change in shape of structure
    - PΔ theory → adequate for seismic Analysis
    - Large displacement theory

Material nonlinearity is more complicated.

For a structure F = load, D = deflection

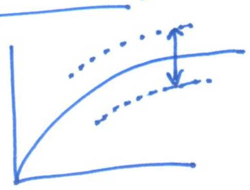
for a component F depends on component type

D is corresponding deformation eg for plastic hinge M and θ.

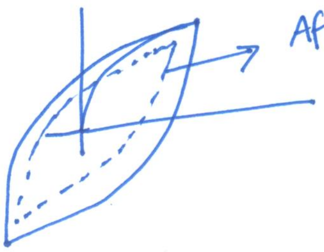
Input in computer programs → Component FD relationships (Known)

Output from " " → Structure FD from analysis.

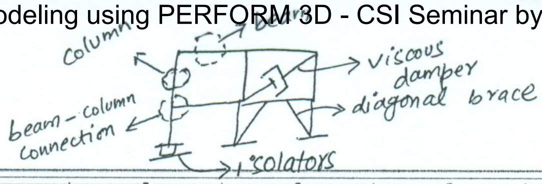
Complications:

a) 

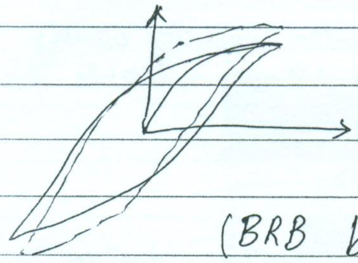
There can be substantial uncertainity. We dont know f_y, f_c' certainly.

b) 

After few cycles, strength and stiffness degrades.



c)

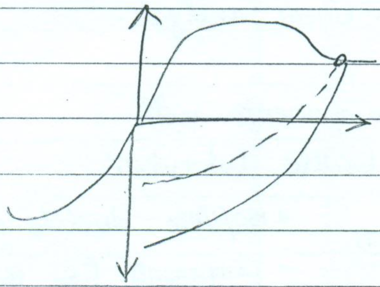


cyclic strength gain

(with cyclic deformation, the strength may increase)

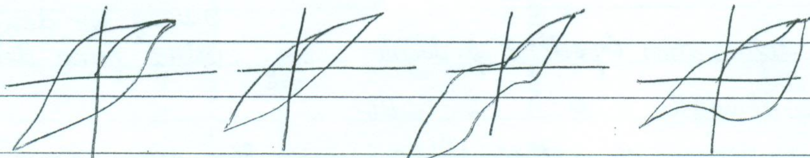
(BRB behaviour exhibits this)

d)



effect of strength loss.
(Strength in opposite direction is not affected by strength loss in this direction or can affect--)

e)

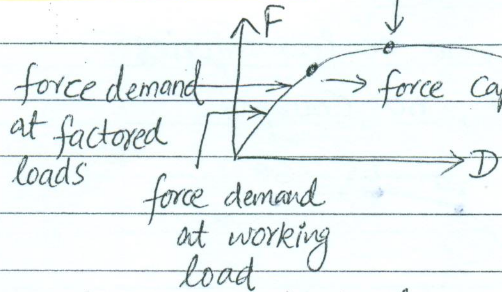


Non degrading ; degrading ; pinched ; Buckling --- and other possibilities

→ The goal is not to get accurate prediction. → Impossible
→ Linear analysis is far from accurate.
→ Non " " " much more rational.

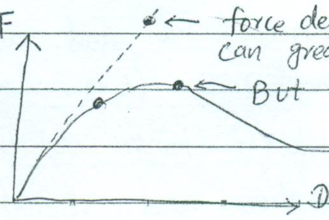
Strength Based Vs. Deformation Based Design :-

Strength Based Design :-



- Check D/C at component level, OK or Not
- Use linear analysis to calculate demand. Usually OK because even at factored loads, behaviour is somewhat linear.

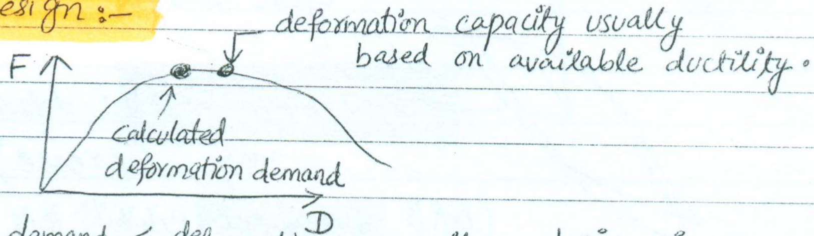
EQ forces are different:-



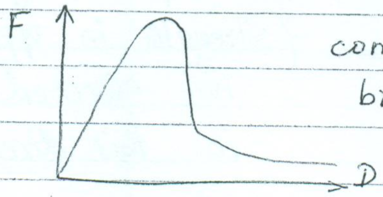
force demands on an elastic structure can greatly exceed force capacity.

But deformation demand on an inelastic structure may not be excessive.

Deformation Based design :-



if Deformation demand < deformation capacity, design is OK
must be calculated using non linear analysis.



components with brittle behaviour
- No/little ductile capacity
- Components like this must usually be designed using force demand-capacity.

Summary :-

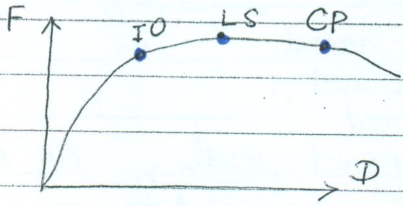
a) Define deformation capacities for ductile compon-ents
b) " strength " " brittle "
c) Use non linear st. analysis to calculate the deformation and strength demands
d) $D/C \leq 1$ ok.

Performance based design :-

Provide assurance "NOT A GAURANTEE" that a design will satisfy a specific performance level.

Levels :-

1) IO : Little or no damage
2) LS : Some damage, no or few injuries
3) CP : More damage, no collapse.



Key steps for performance based design :-

a) Choose performance level and design loads
b) Define D/C measures. (Engineering decision parameters EDPs)
Drift, Plastic hinge rotation, Shear strength etc.
c) Get the deformation and strength capacities

Strength : AISC, ACI, etc codes
Deformation: ASCE41 (FEMA 356), experimental results

d) Calculate the deformation and strength demands.
Use structural analysis. (NL)

e) Calculate D/C. if >1
(i) change the design
(ii) Sharpen the pencil on the demand side or (calculate more precisely)
(iii) " " " " " Capacity ".

ASCE41 :- Guidelines :-

(i) Performance based design for seismic rehabilitation of existing buildings.
(ii) However, can be applied to new construction
(iii) Provides deformation capacities for wide range of structural components, for the IO, LS and CP levels.
(iv) Modeling guidelines are rather simplistic; the deformation capacities may be too conservative.

Engineering demand parameters → EDP's.
(V) essential to establish formal EDPs.

CAPACITY DESIGN

→ Some components in a structure can safely be allowed to yield, others should remain elastic.

→ If we do not know which components yield and which donot, we have to rely on analysis to tell us. This can be dangerous.
We are not analyzing the actual structure, we are only analysing model of a structure.

→ Better to decide in advance which components can yield and which must remain elastic.

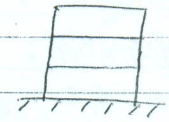
→ We can then design the yielding components to have sufficient ductility, and the elastic components to have sufficient strength. because you know where the yield is going to occur.

→ Major advantages.
- We can set up more reliable NL Models
- We can calculate more " demands
- The final design is likely to be more forgiving.

→ Capacity design is not always possible, but it is a desirable goal.

For frames,

use deformation D/C
- beams can yield in bending
- Columns at base and at roof



Should not yield
Use strength D/C
- Columns except at base and roof
- Connections
- Beams and columns in Shear
- Foundations

For Shear walls,

Deformation D/C
- Flexural hinging at base of wall.
- Shear in coupling beams

Strength D/C
- Bending in wall at all levels except the base.
- Shear in wall at all levels, especially in hinge region.
- Foundations.

Capacity Design without Analysis (Park and Paulay)

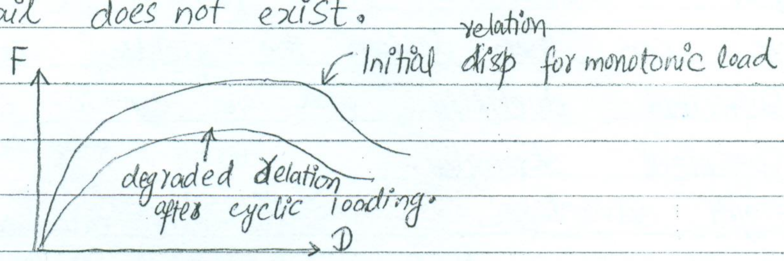
- NL analysis is not essential for capacity design; essential requirement is that all D/C < 1
- Components that yield can be detailed to provide deformation capacities that the engineer believes are sufficient, using judgment or simplified analysis.

Modeling for Practical Nonlinear Analysis

The goal of structural analysis is not to get an "exact" simulation of behavior.

→ "Holy grail" is the exact prediction of behavior and that holy grail does not exist.

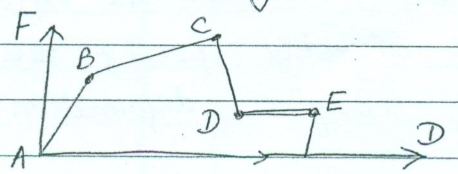
We need FD relationship For Analysis :-



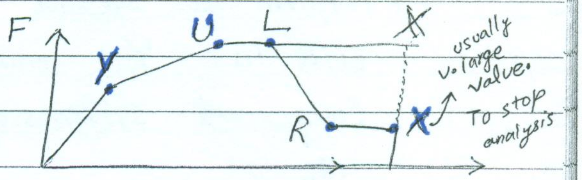
→ The current practice is to use a degraded relationship, allowing for a reasonable amount of cycling.

→ In the future, we may have "Initial" and "fully degraded" relationships, and transition between them as the amount of cycling increases.

Practical F-D Relationships :- to just capture the "main" aspects of non-linearity.

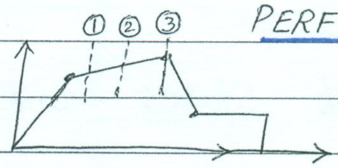


ASCE 41



PERFORM

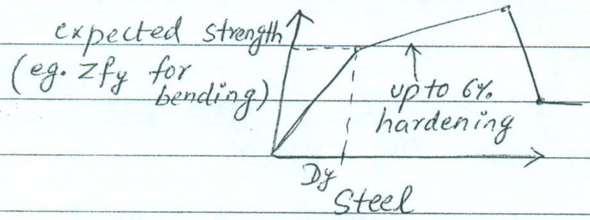
Typical ASCE 41 Capacities :-
(wide range of components)



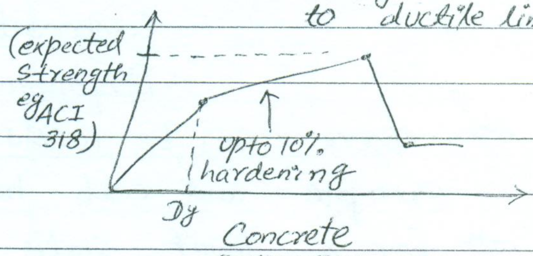
① IO capacity, usually at onset of visible damage.
② LS capacity, usually about 75% of CP capacity.
③ CP capacity, usually at or close to ductile limit.

Perform 3D uses "Capacities"
ASCE 41 " "Acceptance Criteria"

Steel vs. Concrete in ASCE 41 :-



Steel

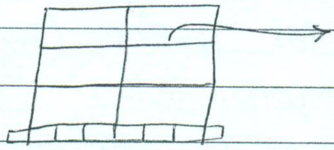


Concrete

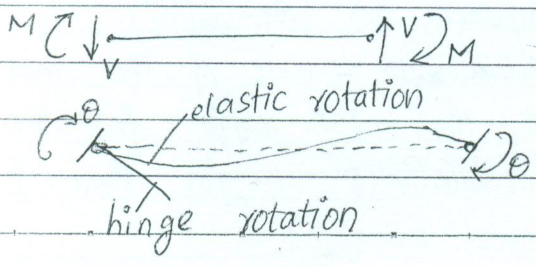
Deformation Capacities are multiples of D_y.

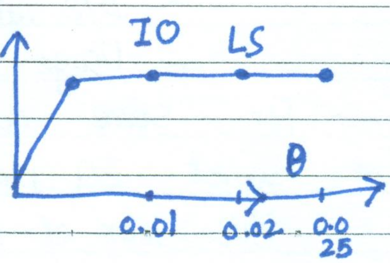
Deformation capacities are $D-D_y$.

- for example, RC beam :-



Performance Assessment:
Bending behavior is ductile
Use hinge rotation D/C ratio.





PERFORM 3D Hands-on Session :

Perform 3D :- Two phases
a) Modeling
b) Analysis

"Component properties" , "Limit states"
↓
no of inelastic components available

→ Define/plot hysterisis loops for inelastic materials.
→ FEMA Beam element
→ Assign to elements
→ Perform 3D doesnot have the ASCE 41 equations built-in. You must directly specify the component stiffnesses, strengths, deformation capacities etc.

In strain Capacities

Level 1	IO
2	LS
3	CP

Limit States and D/C ratios

"PERFORM" limit states organize D/C ratios into sets, by :
- Element (member) type
- Component type and mode of action
- Performance level

→ There can be 1000s of plastic hinges for example and each has one D/C ratio.

→ For a limit state, the simplest and (most conservative) way to get the "Usage Ratio" is to take the largest of those D/C ratios.

→ However there are other ways, for example
- Run analysis for N EQs (Say N=7
- Consider each component in turn — infact 28 → 7EQ for each direction + with + and − sign.
- Calculate the mean D/C ratio over the N EQs.
- D/C ratio for component = This mean value.
- Usage ratio for limit state = Max of the mean values.

→ For some components, the D/C ratio for the worst EQ may be substantially larger than 1. But max of mean should be < 1

→ ASCE 41 allows this method and is included as optional in PERFORM 3D.

So

Usage Ratio → Decesion Making
→ distills the results

Each limit state covers certain elements, components, modes of action and performance levels.

If Usage Ratio < 1, Limit state is satisfied.

Dynamic Analysis vs. Static Pushover Analysis

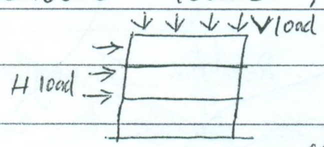
Steps for Dynamic Analysis:-

- Apply gravity loads
- The EQ load is defined by ground motions (seismograms). Can be both vertical and horizontal but usually only H motions are considered.
- Run a nonlinear step-by-step analysis. A typical time step is 0.01 or 0.02 seconds.
- At each step calculate demands → D/C → Usage ratios
- If all usage ratios < 1 at the end of analysis, the performance requirements are satisfied.

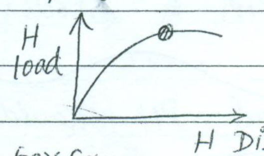
If 20 sec EQ, 1000-2000 steps, 1000-2000 times more time than Linear analysis.

Steps for Static Pushover Analysis:-

- Choose loads, vertical + horizontal load pattern



- Apply V load. Then apply H load and calculate pushover curve.
  constant



- Using RS, estimate performance point or Target disp.
- Calculate the limit States usage ratios < 1, OK

Pushover Advantages and Disadvantages:-

- Uses RS instead of ground motions
- A static analysis model is simpler than a dynamic model.
- Less computer time

However,

- Approximate - Uses static analysis to represents dynamic loads and cyclic deformations
- Not work well for tall and complex structures.

(To be continued)

PBD Seminar by Graham H. Powel
Session 1

Present Vs. Future

a) Future methods are likely to based on probability.
eg a limit state might be statisfied if there only a 5% probability that its usage ratio > 1

b) Future methods may also use performance measures such as cost of repair ("dollars, down~~town~~time and deaths") (different from deformations or hinge rotations).

c) For the present, deterministic methods are enough of a challenge.

Session 1 ends.

PERFORM 3D SEMINAR Continued from page 43

Graham H. Powell

→ You can create analysis series.
eg one with different mass and one with difference
or one including PΔ and one without PΔ.

Session Two

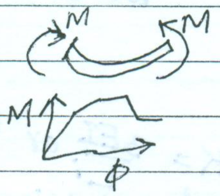
Modeling and D/C calculations for beams in bending, shear; columns, connections, panel zones and diagonal braces

→ For each component type, we need
a) → nonlinear F-D relationship
b) → deformation and/or strength capacities
→ Chose realistic D/C measure

A) Beam in bending: 5 models

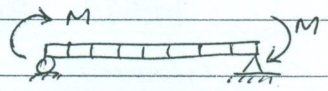
① "Exact" model :-

- Take short length of beam → Calculate Mϕ relationship. eg for elastic beam you need EI only and use BM as D/C measure.



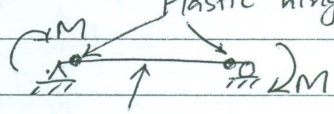
- For nonlinear you may use NL Mϕ and use ϕ as D/C measure.
- Not practical (exact beam theory)

② Finite element model:-

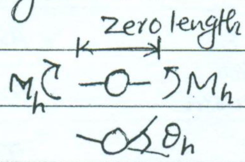


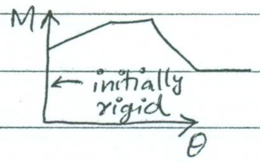
- For NL → sensitive to mesh
- Computationally expensive.

③ Plastic hinge Model :-



inelastic All def are assumed to be in PHs.

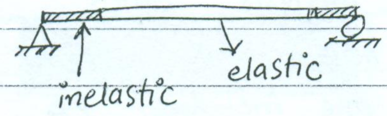




Like a rusty hinge

- All def is assumed to be concentrated in PH
- Def measure is hinge rotations

(4) Plastic zone model :-

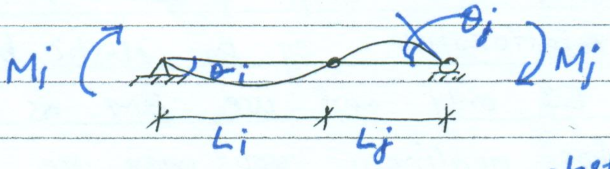


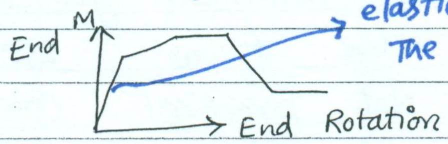
- Average curvature in zone X Plastic zone length
  = Its D/C measure
  or average rotation
- The length of plastic zone should be such that you can use actual $M\phi$ for beam. → ($0.5 \times d$, thumb rule)
- In perform you can use
  - Curvature hinge (uses $M\phi$ curve)
  - Fiber section (here $M\phi$ follows from the behavior of fiber section)

$L_p = 0.08\,H + 0.5 d f_y$
(Paulay, Priestley)

(5) Chord Rotation Model :-

- FD relationship is now directly End M vs End Rotation (No PH rotations, No curvature)





elastic $K = \frac{3EI}{L_i}$ or $\frac{3EI}{L_j}$

The usual assumption is $L_i = L_j = L/2$

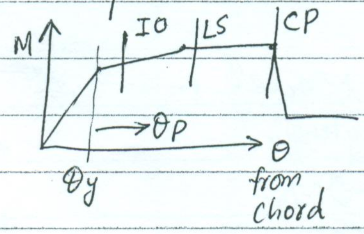
so $K = \frac{6EI}{L}$

Only model covered in FEMA 356

FEMA Beam → covered in FEMA 356

- Generally inflection pt is middle.
- Def measure θ (Total end rotation, not PH θ)

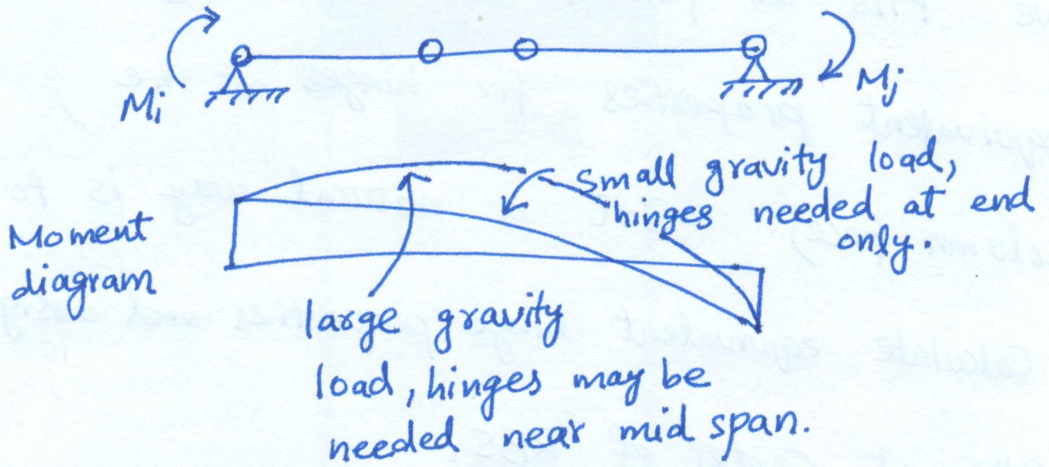
→ ASCE Def capacities :-



- For all component types
- For beams and columns, ASCE 41 gives capacities only for chord rotation model.

	IO	LS	CP
Steel beam	$\frac{\theta_P}{\theta_y} = 1$	6	8
RC beam Low shear	$\theta_p = 0.01$	0.02	0.025
less brittle ← high Shear	$\theta_p = 0.005$	0.01	0.02

Plastic Hinge Model with Gravity loads

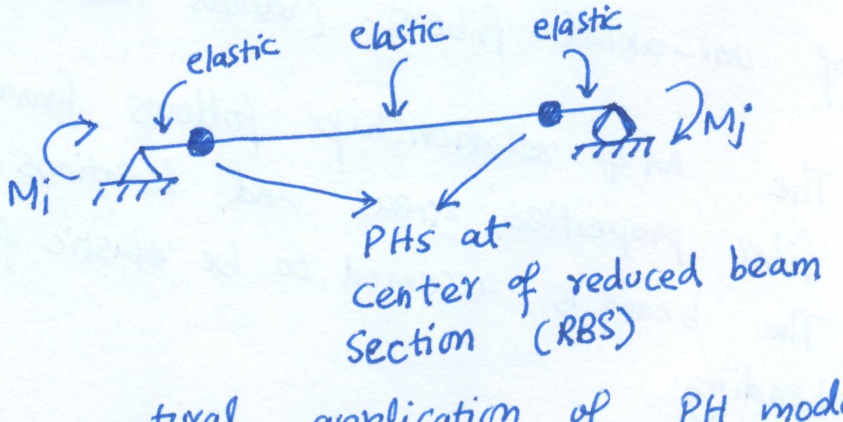


Problem :

If a hinge forms near midspan, under lateral cyclic load, the hinge rotation will progressively increase and the beam will sag.

This you may not want, so in that case it is not a good idea to use PHs in the middle of the beam.

Plastic hinge model for reduced beam Section

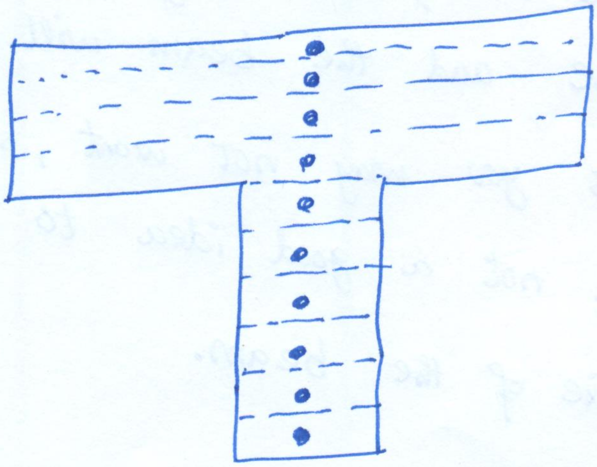


This is a natural application of PH model

Experimental results can be processed to give the required hinge properties.

Some people process experimental results and give PHs at face of columns (giving equivalent properties for hinges at the column face). But a natural way is to calculate equivalent hinge properties and assign PHs at center of RBS.

Fiber Section for a beam :-



Cross-section is represented by a number of uni-axial fibers. (various materials type)

The $M\phi$ relationship follows from the fiber properties, areas and locations.

The beam is assumed to be elastic for horizontal bending.

→ Summary :-

Practical models for beams

1) Chord Rotation model
 - first choice + ASCE 41 gives capacities
2) Plastic hinge Model (2nd choice)
3) Plastic zone model (3rd choice)

Impractical models in most cases

4) Detailed FEM (might be used for Special cases)
5) Exact beam theory

→ Fiber sections :-

- Do not apply for the chord rotation models
- Most likely used in plastic zone models
- Zero length hinges can be approximated using short plastic zones
- Fiber sections can also be used for "FE" and "Exact" models.
- D/C measure = Rotation over the plastic zone. Rotation = Average curvature X plastic zone length
 DC measure can also be curvature or fiber strain
- In PERFORM 3D, fibers can be Concrete, Steel, buckling and/or tension-only type.

- PERFORM 3D Shear wall element uses fibers

Note :- In concrete beams, as beams cracks, the neutral axis shifts and hence the beam must grow axially. If the axial growth is restrained the beam goes in to compression and its strength may be increased. The compression face can cause extra shears in adjacent columns. (Some engineers believe that this can be a substantial effect). If a beam is modeled using hinge elements, it doesnot grow. If it is modelled using fiber sections, it does.

This growth ↓
It can tear the floor diaphragm apart (some engineers think).

Strength Section :— in PERFORM 3D

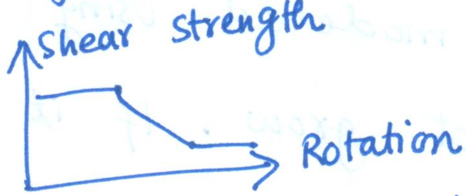
Moment S.S.

Shear Strength Section

If you dont allow
yielding in bending
but just want to
make sure that strength D/C < 1
Just give strength directly.
so → no hinge formed.

One of the complications in nonlinear behavior
is "Interactions".

eg Larger the hinge Rotation → smaller the Shear Strength



So you have to account for that.

In PERFORM You can consider this interaction in shear strength sections (but not in shear hinge, in current version).

continued from page 98

B) Modeling and D/C calculation for beam in shear :-

Steel beams
- Rarely yield in shear
- One exception is shear link in eccentrically braced fram.

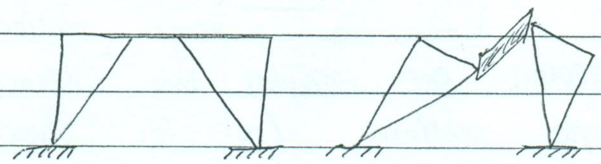
Concrete beams
- Usually brittle in shear
- Hence usually design for strength (elastic)

→ If Inelastic behavior in shear is allowed, it requires a "shear hinge".

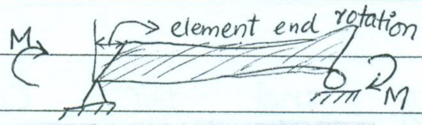
The D/C measure for deformation can be shear disp across hinge.

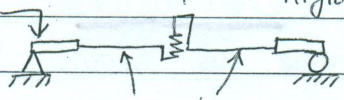
Alternatively it can be shear strain = Shear disp / member length.

→

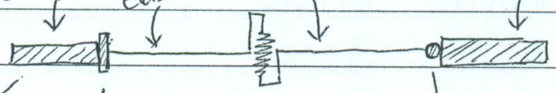


Shear link in steel EBF





ASCE 41 Def Capacity :-



If you intent shear link does not yield in bending.

If you think shear link can yield in bending.

Moment strength section, if moment hinge is not allowed check strength $\frac{D}{C} < 1$ OR Moment hinge if inelastic bending is allowed.

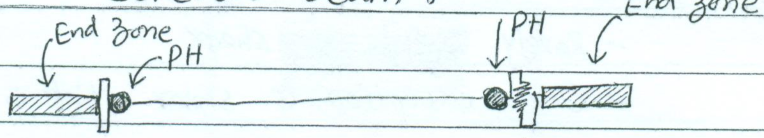
Plastic Shear strain for yield in Shear

IO = 0.005 radian , LS = 0.11 , CP = 0.14

Shear disp across hinge = Shear strain × Clear span Length

If inelastic bending is allowed, the rotation capacity is the same as for a regular steel beam.

Shear in concrete beam :-

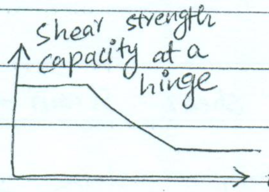


Shear strength section if inelastic shear strength is not allowed
Calculate shear force and strength $\frac{D}{C}$ ratio

OR

Rigid plastic shear hinge if inelastic shear is allowed.
Calculate shear disp and deformation $\frac{D}{C}$ ratio.
[unlikely for concrete beams]

An issue :-



(larger the hinge rotation, smaller the shear strength)

PERFORM can consider this interaction only for shear strength sections (not for shear hinges where shear can be inelastic)

c) Modeling and D/C calculations for COLUMNS

Basic models are same as beam.

- Chord rotation
- Plastic hinge
- Plastic zone
- Finite element or Exact theory.

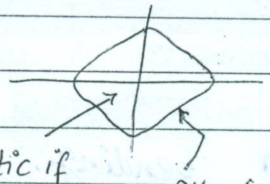
But the behavior is complicated by P-M-M interaction.

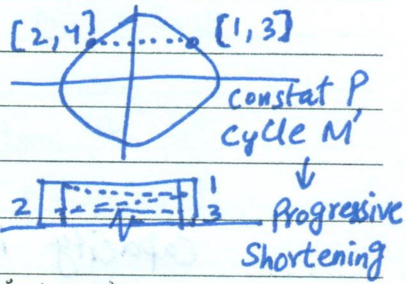
- M strength depends on P and vice versa
- Moments act about two axes.
- Hinge behavior after yield is complex - not just rotation.

Shear behavior is also more complex

- Shears act along 2 axes.
- Shear strength can be affected by P and M. also by hinge rotation

Beam = Uniaxial bending.
Column = Biaxial ".

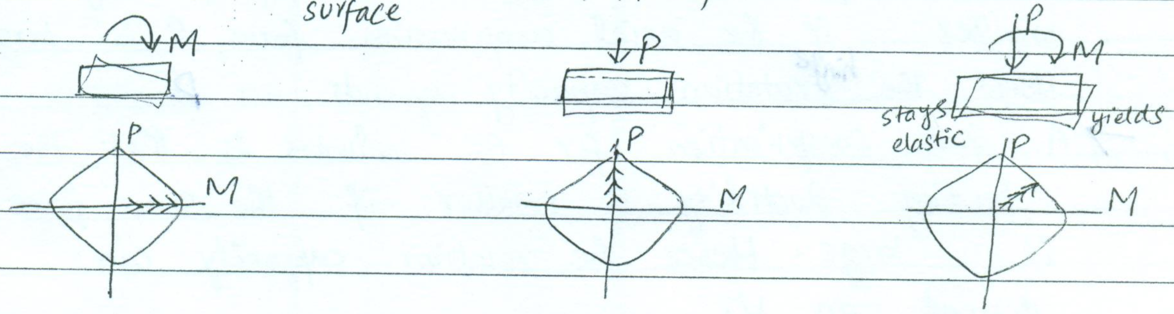




elastic if inside P-M surface

PH forms when on P-M surface (inelastic)

Steel Behavior:-



Only bending like a Moment hinge in a beam

Only axial deformation like an axial hinge (Theoratically)

Combined bending and axial deformation Combined moment and axial hinge.

P-M models for column elements :-

→ - each model requires two F-D relationships, linked by plasticity theory.

a) Chord rotation model
- End moment vs end rotation
- Axial force vs. axial deformation

b) Plastic hinge model
- Hinge moment vs. rotation
- Hinge axial force vs. axial deformation

Axial deformation of a zero-length hinge means infinite strain. This is OK.
Rotation of a zero length hinge means infinite curvature.

If we have Plasticity theory we can link Moment action-def with axial.
The advantage of fiber section is that we dont need Plasticity theory. (discussed later)

c) Plastic zone, FE and exact models
- Moment vs. Curvature
- Axial Force vs. Axial strain

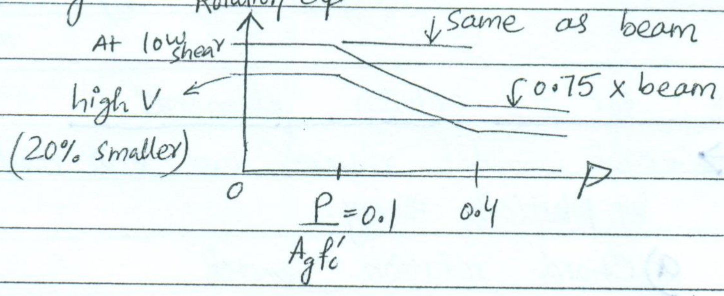
Column deformation Capacity:-

Because of PM interaction, column hinges or plastic zones have both axial and bending deformations. The deformation D/C measure is usually bending deformation (rotation) only, not on axial deformation. The capacity is based on bending.

→ One complication is that the bending ductility is smaller if the axial compression force, P is large. Hence the hinge rotation capacity depends on P.

→ A 2nd complication for RC columns is that the bending ductility is smaller if the shear force, V is large. Hence the rotation capacity can depend on V.

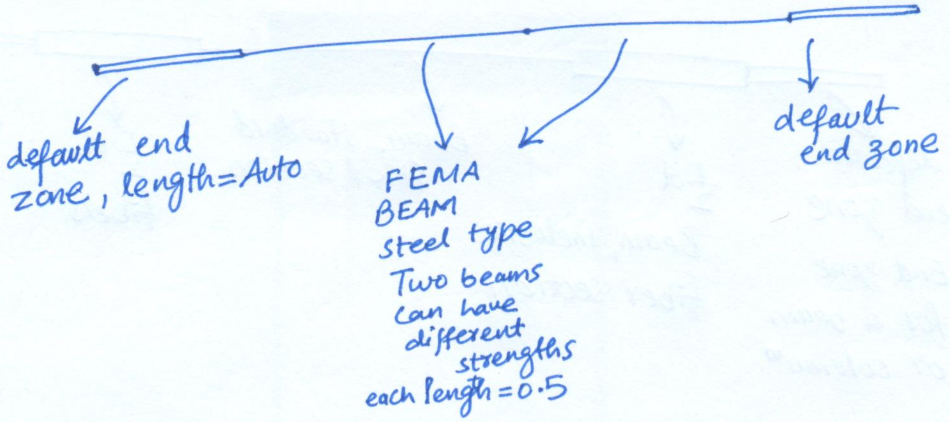
ASCE 41 gives bending capacities for the chord rotation model only.



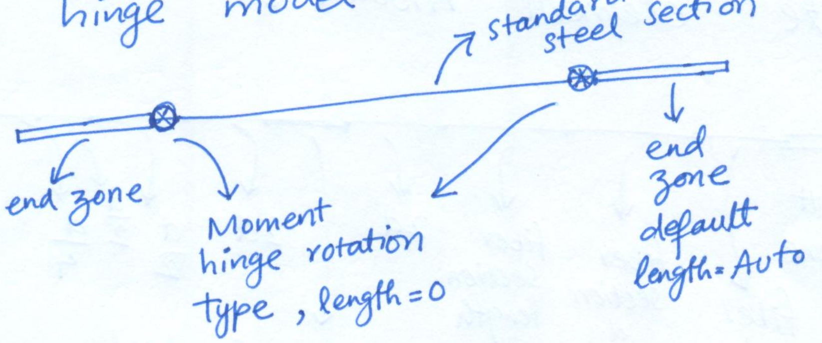
To be contd...

Nonlinear modeling of beams in PERFORM 3D :-

(a) Chord Rotation model



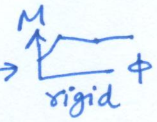
(b) Plastic hinge model



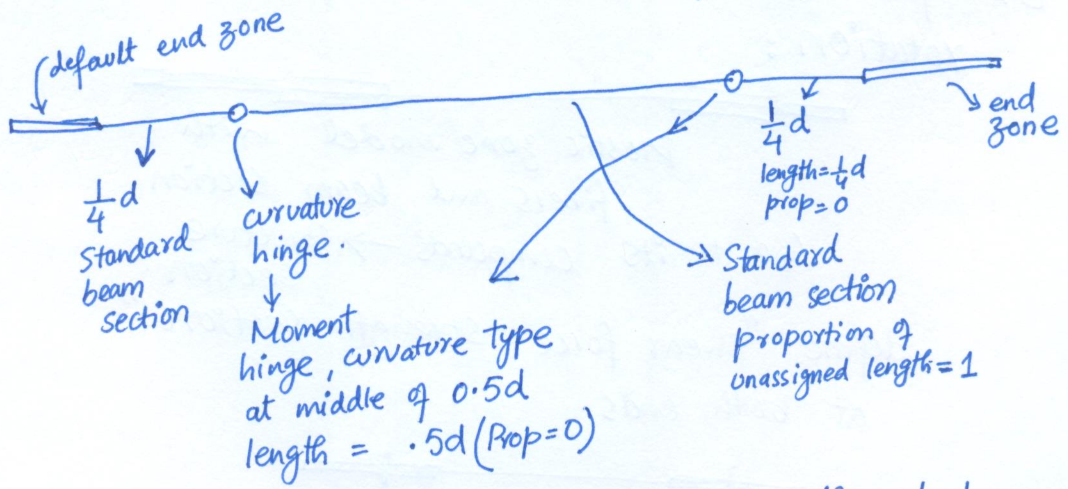
(c) Plastic zone, curvature hinge model

a variation of rigid plastic hinge

For plastic hinge →



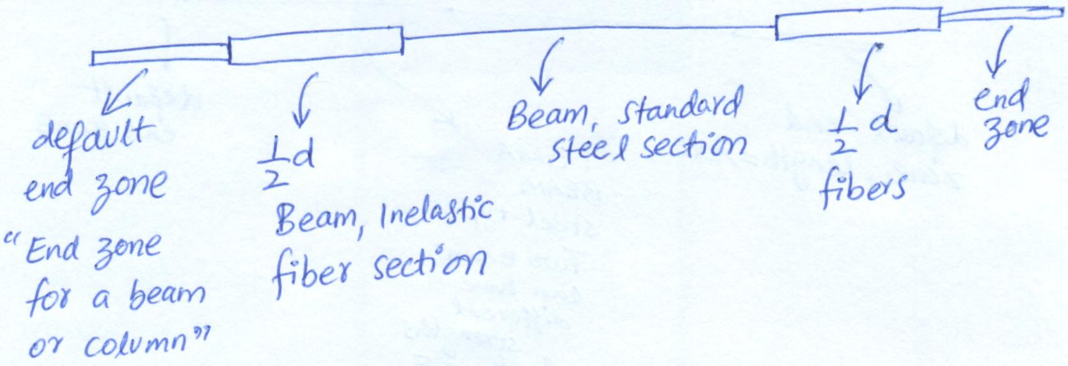
For curvature hinge →



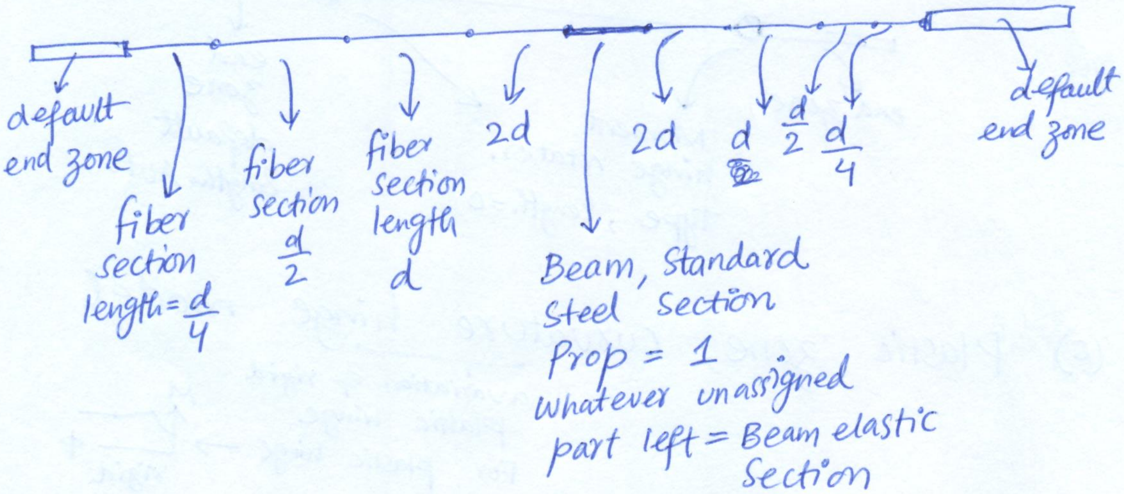
Curvature hinge has tributary length $= \frac{1}{2}d$
it means lets use a plastic zone here with $M\phi$ relationship designed by this hinge and lets make the length of plastic zone as $\frac{1}{2}d$.

(d) Plastic zone model with fiber

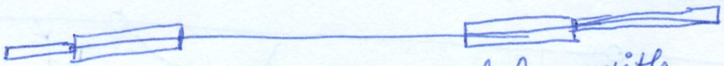
Section :-



(e) Finite element model :-



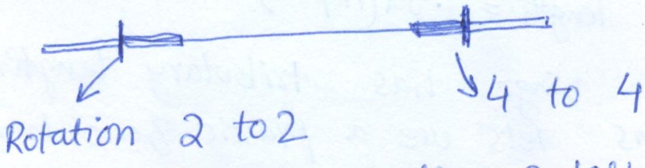
(f) Example where Shear strength depends on the rotation :



Plastic zone model with fibers and beam section

Go to its compounds → Structure section

define "Shear force strength section" at both ends



2 = Fiber section of left
4 = " " " Right

In the "strength" tab of strength sections,
give V_c, V_0, V_T

In the "Rotation effect" tab of strength sections,
give Rotations and shear strength reduction factors

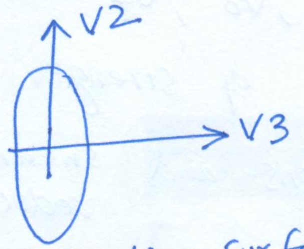
So the strength of beam will be modified depending upon the rotation.

So first strength section with 2 to 2 is saying that use the rotation from component no 2 to 2 (fibers) and calculate shear strength capacity based on that rotation. The current rotation will go to the diagram in "Rotation effect" of strength Section and find strength reduction factor for shear and calculate "reduced Shear Strength" and use this for calculating D/C ratio.

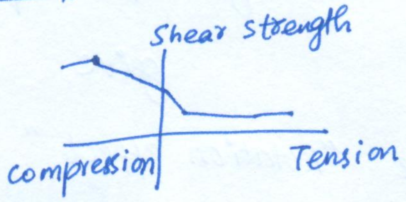
<u>An Alternative to Plasticity theory - Fiber Section</u>:-

- PM Interaction is taken in to account through the behavior of the fibers. No need of plasticity theory. So can be equally applied to concrete as well as steel. It also models the cracking correctly. (b/c doesn't use plasticity theory)
- The cross section is represented by a number of uni-axial fibers.
- The moment - curvature and Axial load - curvature relationships for short length of columns follow from the fiber properties, areas and locations.
- PM Interaction is automatic. It is not necessary to use plasticity theory.

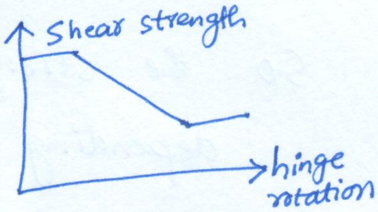
Shear in concrete columns :-



Interaction surface



Axial force effect



Hinge rotation effect

You have some control on these shapes.

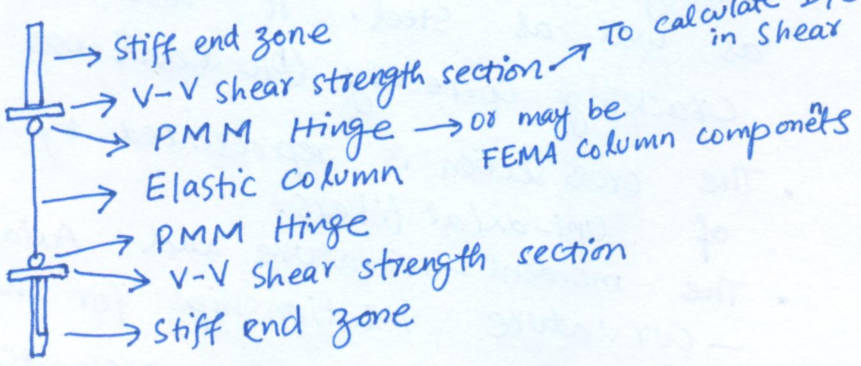
Case A:

→ If inelastic shear is allowed, use a shear hinge with V-V Interaction. In this case PERFORM does not consider axial force and hinge rotation effects.

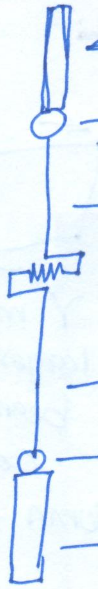
Case B:

→ If inelastic shear is not allowed, use a "Shear ~~hinge with~~ strength section". In this case PERFORM does consider axial force and hinge rotation effects.

Possible column models with Shear :-



(a) Shear must be elastic



← Stiff end zone

→ PMM hinge

→ Elastic column

→ V-V Shear hinge — but doesnt have axial force effect and hinge rotation effect.

→ Elastic column

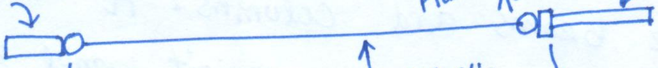
→ PMM Hinge

→ Stiff end zone

(b) If you wanna have inelastic shear.

Considering inelastic shear in concrete columns is not a good idea. (not likely to have).

D) Beam-to-column connections:-

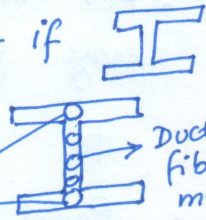


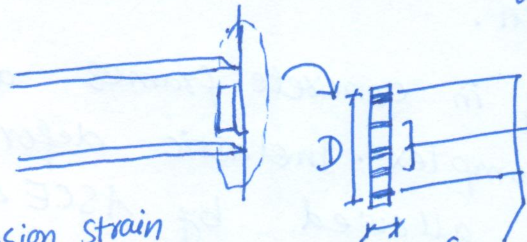
Nonlinear Semi rigid connection component

OR

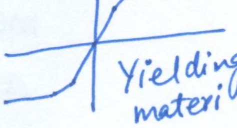
Short segment (eg 1") with a fiber Cross section

eg if





→ web connection ductile so

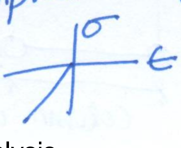


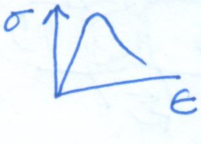
Tension strain in flange fiber is essentially = Connection Rotation × $\frac{D}{L}$

flange weld

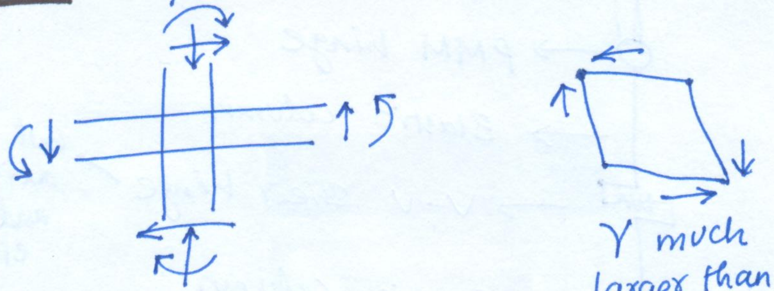
Concrete fiber for compression only

Tension-only fiber for tearing





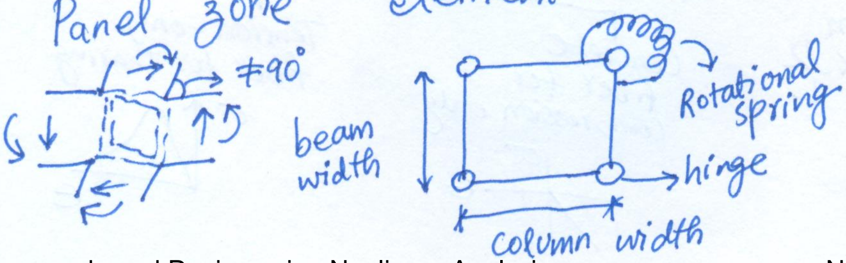
E) Connection Panel zones:-



Effect of panel zone deformation:- angle b/w beam and column $\neq 90°$

- Connection panel zones can have much larger Shear stresses than the adjacent beams and column members.
- For elastic analysis the shear stresses Can be Calculated from the forces in the beams and columns. It is not necessary to have explicit panel zone elements.
- For inelastic analysis, the panel zone may yield. In this case, it is necessary to have explicit panel zone elements.
- Panel zones for steel frames are fairly Simple. Deformation capacities are given by ASCE 41.
- Panel zones in concrete frames are more complex. Inelastic deformations are not allowed by ASCE 41.

Panel zone element:-

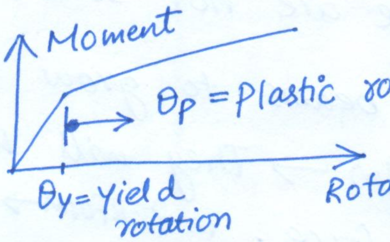


(Models the shear stiffness and strength of Panel zone)

Deformation, D = Spring rotation
= Shear strain in panel zone

F = Moment in spring
= Moment transferred from beam to column elements

Also F = Panel zone horizontal shear Force X Beam depth.



Guidelines	Measure	IO	LS	CP
ASCE 41	θ_p/θ_y	1	8	11
FEMA 273	θ_p/θ_y (F_y = 50 ksi)	0.6	10	17

Software Demonstration :-

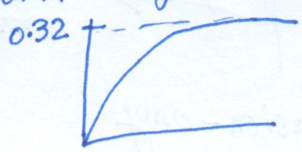
Inelastic fiber sections to model Connections :-

fracturing connection
↓
Fibers
- → Some are with Type = Tension only (brittle to measure tension in flange weld material)
- → Concrete Comp only for flange weld material (exactly same coordinates as tension only fiber)
- → Steel (for web connection material → ductile)

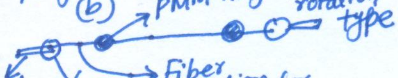
Lets consider 3 cases:

(a) Inelastic fibers to model Connections

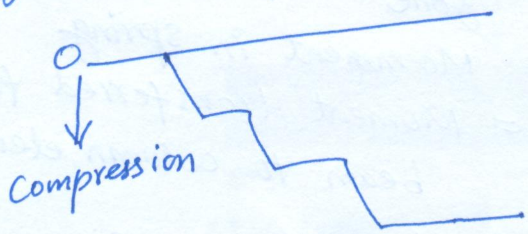
(b) include PMM Hinge So axial force will be limited by PMM hinge



(c) Apply axial releases in (b)



In (a) the beam is fully restrained, cannot grow in length. (Rigid diaphragm) So its axial force increase in compression.



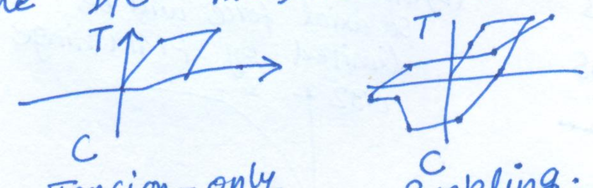
In (b) → we "a sort of" accounted for axial force effects. but we are not sure.

In (c) → we allow beams to grow axially even in rigid diaphrgm → They will have zero horizontal force. In PERFORM → Linear P/V/M Hinge or release → Axial Stiffness = 0

So Graham powell suspects,
" There is some significant restraint being offered by floor diaphrams (not necessarily to be rigid). The building with these connections is developing substaintial compression in beams which is stopping axial connections to open up and this is the "may be" saving the building". → (because otherwise → (c) 0.16 V/W)

F) Diagonal braces :-

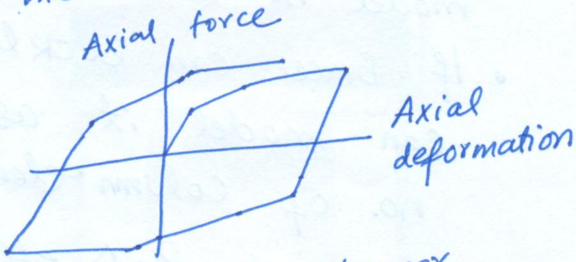
- Treat as bar element, with axial force only → forget about bending
- If very slender → Use tension only material
- If moderately selender → use buckling material
- The D/C measure is axial deformation.



Tension-only Buckling.

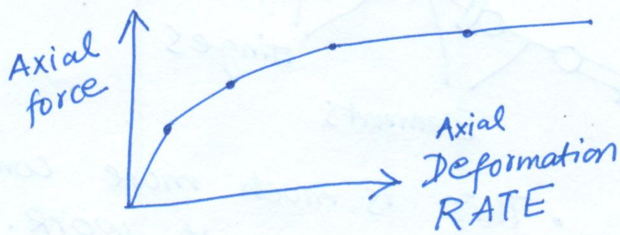
BRB :-

- The PERFORM BRB element includes "Isotropic hardening"
- The D/C measure is axial deformation.

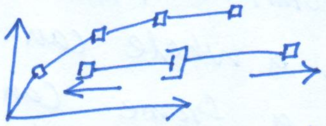


In tension it gets stronger.

Fluid Damper :-



- In PERFORM element the relationship is defined by a number of linear segments as shown below.
- The D/C measures are axial deformation and/or axial force.
- In PERFORM, the extra damping provided by fluid dampers can be estimated in push-over analysis.



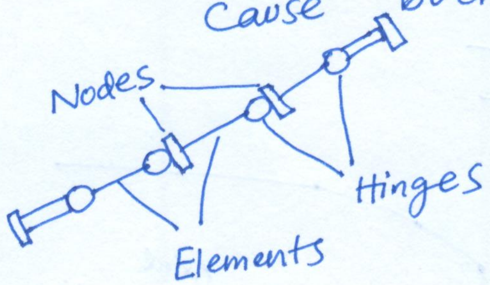
will give a rough idea even in just pushover.

In pushover → no dynamic effect so no strength in Pushover from dampers.

The effect of damping comes when you Calculate the performance point. So it can't effect the pushover curve but it can effect the target displacement.

Non-Slender Brace :-

- If brace doesnot buckle, model it like a column.
- If brace can buckle, you can model it as a no. of column elements and rely on P-Δ effects to cause buckling.



- This is much more complex and may not work.

→ Next session will be about Shear walls → very different than frame elements.

→ Foundation uplifts → GAP type element.
Pounding → " " .

→ A typical column in MRF don't buckle, a frame as a whole can buckle → PΔ effect in a frame can cause it as a whole to buckle but it is very hard to press a column in MRF to buckle.
But if it is still necessary to model you can ~~use~~ consider it as a brace but then no moment. Or you can make two parallel elements. One with moment hinges but no axial Stiffness and other Brace with buckling material.

→ Deformable diaphragms → using elements
or links b/w diaphrams to
increase K in case of set backs
→ will Talk later.

→ When we use a plastic zone and use strain or curvature as a D/C measure. we are saying that "average strain" or "Av. ϕ" over that zone is my D/C measure.
So define gauge length is very important.

→ If you somehow have to use a FE model then the length of that FE $\not\leq \frac{d}{2}$ → then OK, then you will trust E results.
so for beams
Chord rotation
PH
P Zone
FE

my choice.

→ If a lot of internal gravity framing. then to account for PΔ.
1) Model internal frame directly. put loads on gravity columns
2) Not model internal gravity framing but model a special PΔ column then put all loads that on internal columns on it. and that provides you PΔ effect.

Dont leave.

→ If Foundation not going any where → Rigid supports / Elastic springs
but if significant NL → model piles as columns / Use bars to model Py springs for lateral disp.

→ Within same structure → we can have different models for different elements.

→ How Perform 3D calculates effective (additional) damping provided by damper?

At any step of pushover we have displacement of dampers. At any point, we can also determine T_{sec} (from capacity spectrum) method. Using these 2 values, we can calculate the energy dissipation. We have energy dissipation from only nonlinear behavior so we can get additional damping from fluid dampers but this all is tied in capacity spectrum method.

→ FRP Jacketing → may be lumped fiber or may be full FE

→ The rotation capacities given in ASCE 41 for Chord rotation model → can we use the same values for hinge model ???

For RC Concrete beam or column, Yes we can. so PMM hinge same capacities = chord rotation model.

hinge rotations = Rotation capacities for Chord rotation model.

For Steel beam or column → cannot. because its defined differently.

so In RC beams, columns you use plastic/ PMM hinges and use same rotation capacities which you would have used for FEMA beam (chord rotation).

Why not for steel ??? → The capacities given for chord rotation model in ASCE 41 are the Plastic end rotations directly and same as Plastic hinge rotations. So FEMA Beam model = PH model (RC) but for steel beam, the capacities are multiples of yield (not simply taking the same capacities, it depends on aspect ratio of beam).

So PERFORM has 2 types of hinges < Rotation hinges. / Curvature hinges.

Rotation hinge → you will use in RC.

Curvature hinge → you have to use in case of steel.

So using curvature hinge → you can use FEMA Chord rotation values for steel "without much effort". → See documentation (for how FEMA model can be implemented using plastic hinges → may be using reverse engineering)

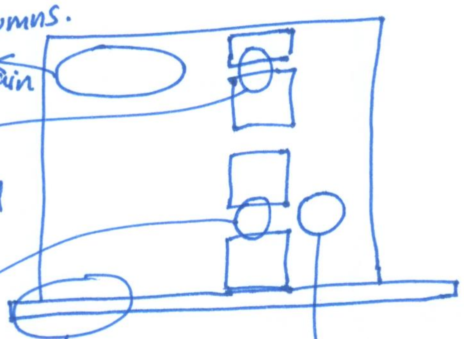
Shear Walls

- Main aspects of behavior for 2D walls
- Modeling of 2D elastic
  - 2D inelastic – bending
  - 2D inelastic – Shear
  - 3D walls.
- D/C calculation – Axial-bending deformation
  - – Shear strength
  - – Shear deformation
- Other aspects (P-Δ effects, out-of-plane behavior, walls supported on columns, etc)

Main aspects of behavior for 2D walls :-

Piers act much like columns. Plane sections essentially remain plane. May be governed by bending or shear.

Selender coupling beams may be governed by bending or shear

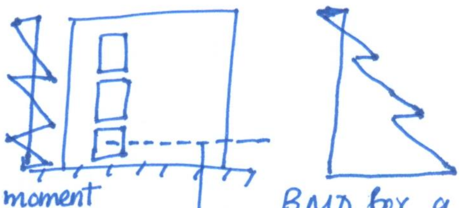


deep coupling panels are probably governed by shear.

Foundations may not be rigid.

Panel zones are usually not highly stressed

Pier behavior →



Bending moment diagram for a Selender pier may be like a frame

BMD for a wide pier will be more like a Cantilever.

Shear stresses theoratically vary over pier width, but for design purposes use average value.

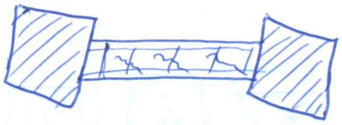
Coupling beam behavior - Bending :-

Elastic behavior

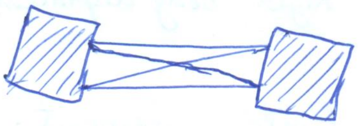
Actual Behavior

Coupling beam behavior - Shear :-

Elastic behavior:-
- Compression diagonal shortens
- Tension diagonal extends
- Beam as a whole doesnot extend



Actual behavior with conventional Rft:-
- Vertical steel yields
- Horizontal steel doesnot yield
- Beam as a whole doesnot extend



Actual behavior with diagonal rft:-
- Tension diagonal yields
- Compression diagonal has a much smaller deformation
- Beam as a whole must increase in length.

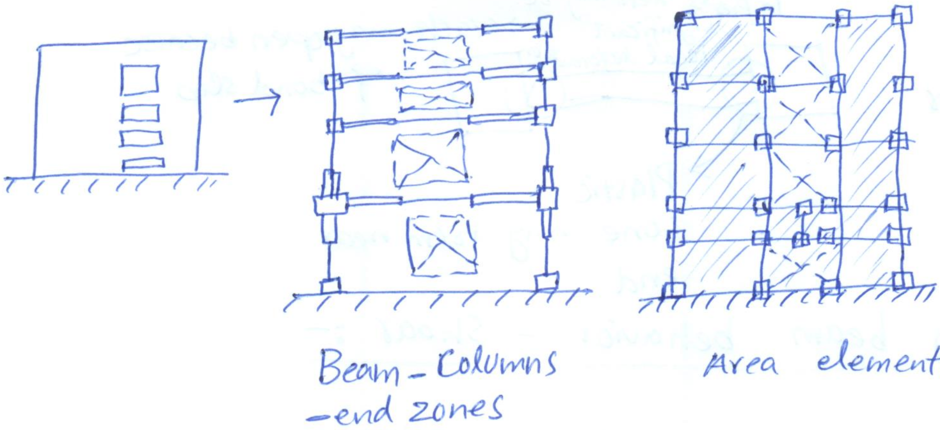
Squat or irregular walls :-

These can be difficult to model.

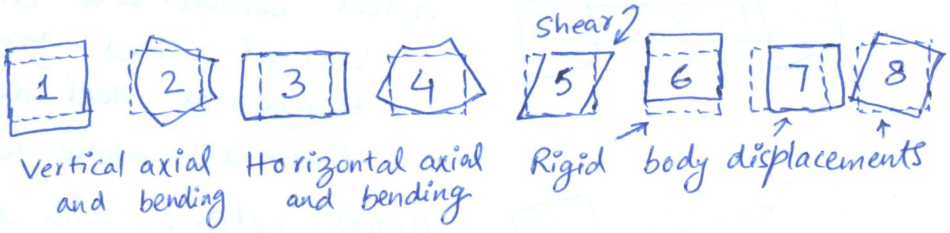
Modeling of Elastic 2D Walls

Frame or wall model :-

- A model with wall elements is better.
- This seminar doesnot consider frame models.



PERFORM elastic wall element :-

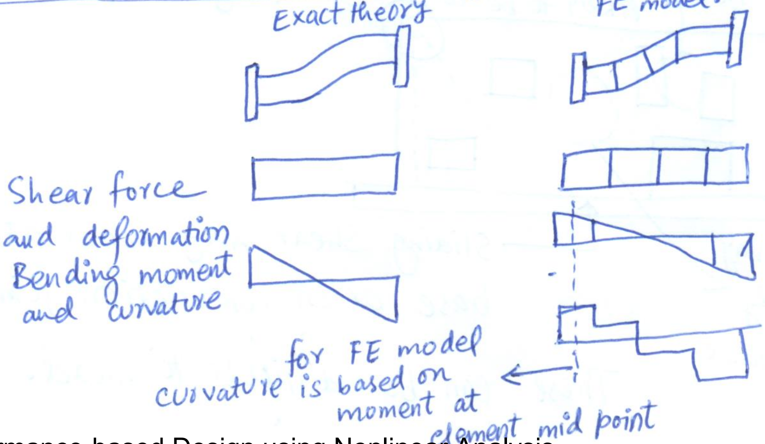


This is a simple 4- node finite element.

Axial deformation, curvature and shear strain are all constant over the element.

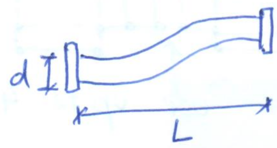
Better to keep them roughly rectangular.

Approximation for bending behavior :-



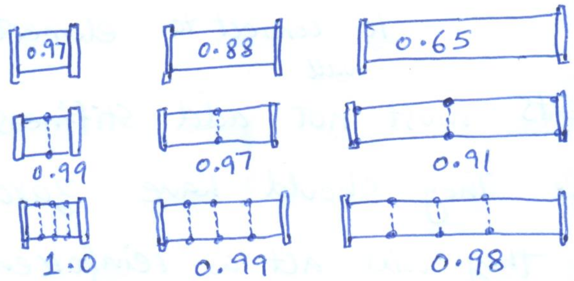
This is for a Coupling beam, a slender pier is similar.

Effect of element mesh on accuracy :-



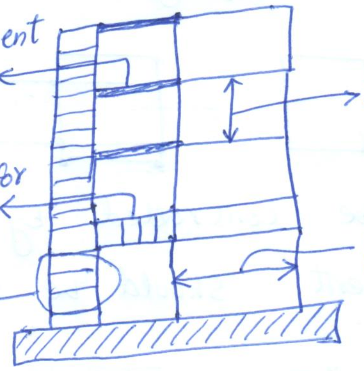
Elastic beam, rectangular section. Shear modulus, $G = 0.4E$
Assume shear area = actual area.
(should be 5/6 of actual)

Ratio of finite element deflection to exact deflection for different d/L ratios and no of elements.



(e.g for coupling beams)

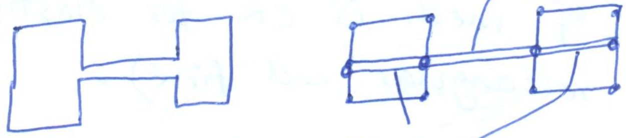
Reasonable mesh for elastic analysis :-



Single element over story height should be OK

Plane sections stays essentially plane. Single element over pier width should be OK. If in doubt, use 2 or 3 elements.

Model for slender coupling beam :- (in PERFORM)

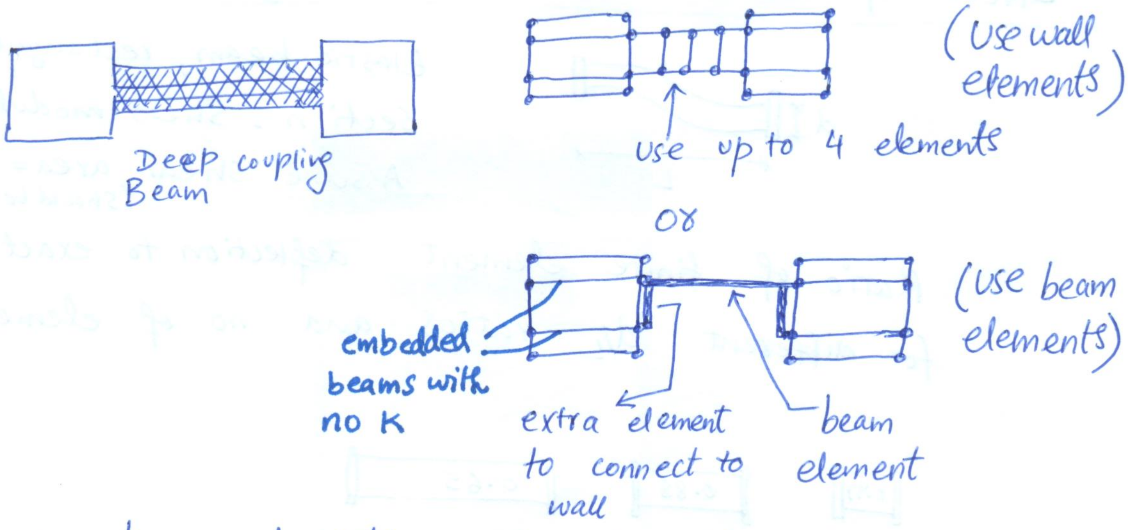


Extra imbedded elements to connect beam to wall. (otherwise pin connection b/w beam and wall)

The coupling beam can have plastic hinges for bending and/or shear. (depending upon controlling factor)

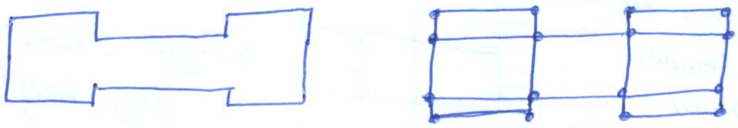
[The imbedded elements must not add stiffness to the wall elements.]

Models for deeper coupling beams :-



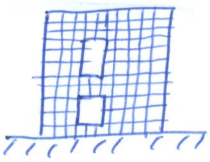
The extra beam elements must not add stiffness to the wall elements. They should have zero axial stiffness otherwise they will act as reinforcement in wall. In bending they should be stiff.

Model for a deep coupling panel :-



The panel will be controlled by shear so a single element should be sufficient.

"Exact" FE Mesh :-



generally thought,

finer mesh → FE results more close to elastic.

This type of mesh is OK for elastic analysis (uniform, rectangular and fine).

For inelastic analysis, it is not such a good idea. (computer time).

The analysis results (e.g stresses etc) may be accurate for an elastic material but not for RC.

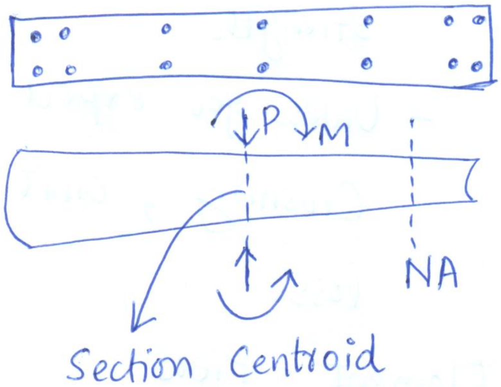
Nevertheless, for "true strength based design," RC cross sections designed using the analysis results are probably OK. The problem is that these cross sections may not be OK for earthquake design, where there can be inelastic behavior. **The stresses may not be accurate but P,V,M may be OK.**

You calculate P,V and M from those stresses and design based on those. (These P,V,M may be OK for strength based elastic design). You design shear reinforcement based on average value over width of pier.

A Key Aspect : P-M Interaction :

Modeling of 2D Inelastic Walls (Bending)

A key Aspect : P-M Interaction



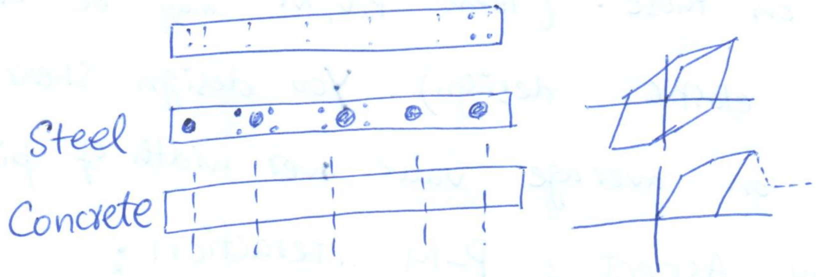
The NA location shifts, depending upon the P/M ratio and the amount of cracking and yielding.

Shift in NA is not modeled in elastic Analysis.

For inelastic Analysis in PERFORM 3D, P-M interaction is modeled using fiber sections, not PM interaction surfaces.

Interaction surfaces and Plasticity theory is not applicable to RC.

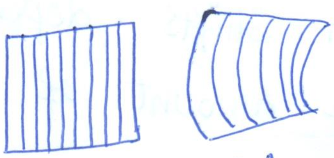
Fiber Section :-



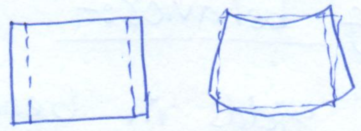
Steel fibers can yield and may degrade under cyclic loading.

Concrete fibers can crack and possibly crush.
- Usually specify zero tension Strength
- Unless you expect substantial Crushing, omit brittle strength loss.

Inelastic Wall Element - Piers

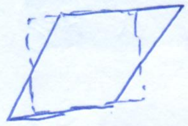


Vertical Axial and Bending.



(assumed to be elastic)

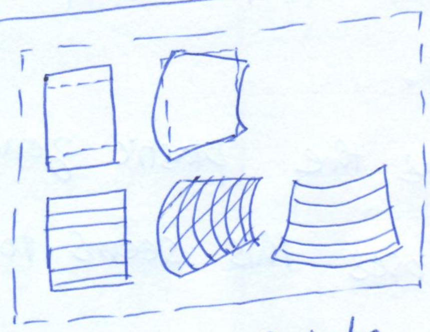
Horizontal axial and bending

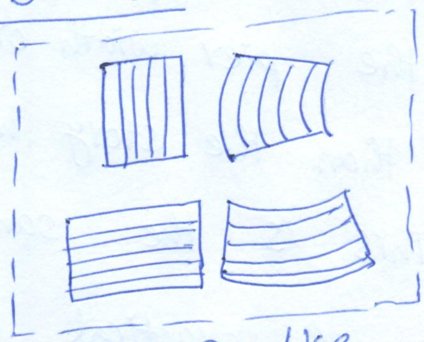


(can be elastic and inelastic)

Shear

Inelastic Wall Element - Beams



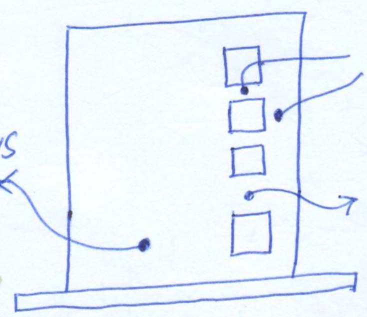


Option 1 : Rotate element so that fibers are horizontal (shear wall Element)

Option 2 : Use fiber sections in both directions. In PERFORM its called "General Wall Element"

Or you can use beam Element.

How many fibers do we need?



Fewer fibers are needed for a beam or a selender pier

8-12 Steel fibers and 8-12 Concrete fibers should be OK for the full width of a wide pier

Its a deep panel controlled by shear so axial-bending behavior could be elastic. So inelastic behavior can only be allowed in shear.

You may need to experiment with a small model of a single beam or pier to decide on the no. of fibers.

A key Aspect for bending behavior:-

→ When a wall element yields in bending, it acts like a plastic zone in a beam or column.

→ As with a plastic zone, a key parameter is the zone length.

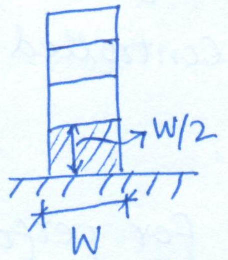
→ ASCE 41 Specifies a "hinge length" of 0.5 times the pier width or beam depth, but not greater than the story height.

→ This is the same as the plastic zone length.

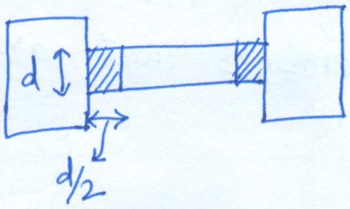
→ For ~~th~~ practical purposes this seems to be a reasonable rule.

→ In a beam or column, if the plastic zone length is "correct" the actual $M\phi$ relationship can be used. In a wall, if the hinge length is "correct" a fiber model of the cross-section can be used.

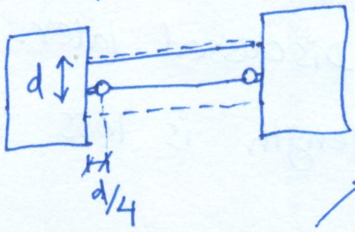
Hinge Length : Simple Examples :-



Solid wall with hinge at base.

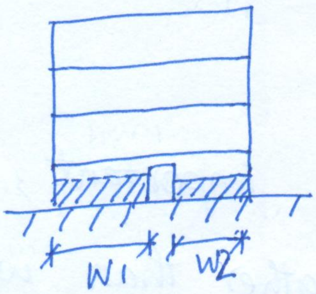


Coupling beam using wall elements

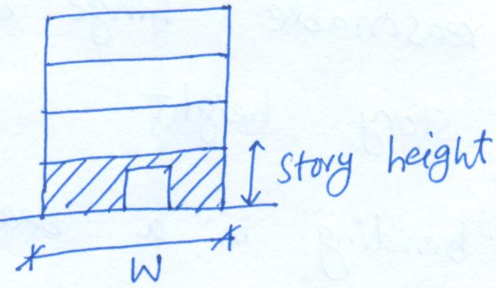


Coupling beam using beam element. (if governed by bending)

Hinge Length : Not so Simple :-



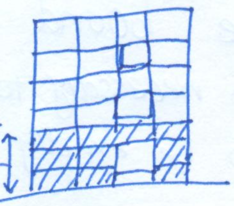
Weak Coupling – Two hinges?



Strong Coupling – One hinge?

Decision is yours.

Hinge length may not match element mesh :-



The hinge length may extend over several elements

It is convenient that the FE mesh matches the hinge length. Generally not possible and is not necessary.

Imp point → Rotation Gauges → discussed later.

Key Points and Suggested Procedure :-

The hinge length is for bending only. It is not needed for members that are controlled by shear.

The hinge length is important for performance assessment → Discussed later.

The hinge length is less important for the FE mesh.

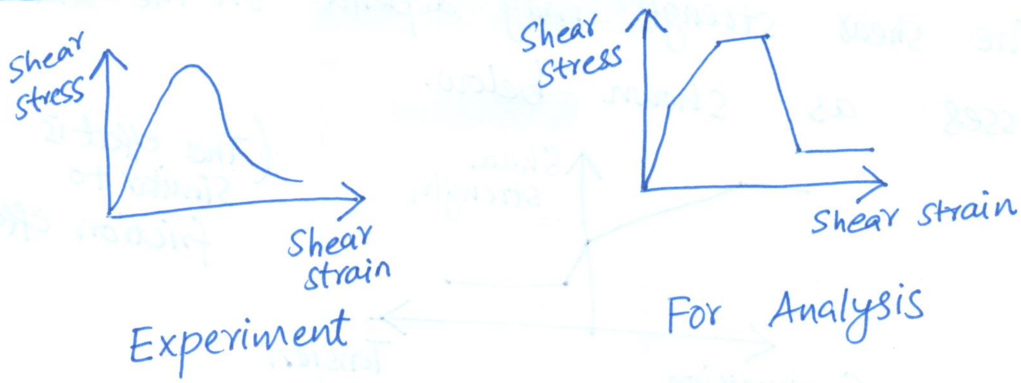
If bending in a pier is important, estimate a reasonable hinge length. Often it can be the story height.

If bending in a beam is important, consider using a beam element rather than wall elements. For wall elements, use one half the beam depth as a reasonable hinge length.

Use a reasonable element mesh. Consider the hinge length but do not be bound by it.
(no necessary to be bound, to be exactly same)

Modeling of 2D Inelastic walls (Shear)

Inelastic shear Material:-



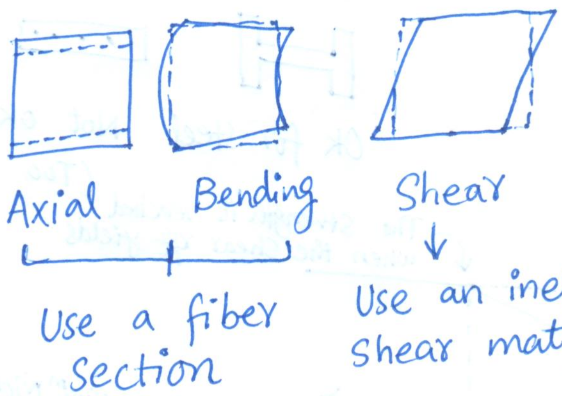
Shear behavior tends to be brittle.

In PERFORM you have 2 options.

a) Assume elastic shear behavior and check Shear strength using Strength sections.

b) Use and allow inelastic shear behavior using Inelastic shear material and check deformation.

Wall Element (for shear):-

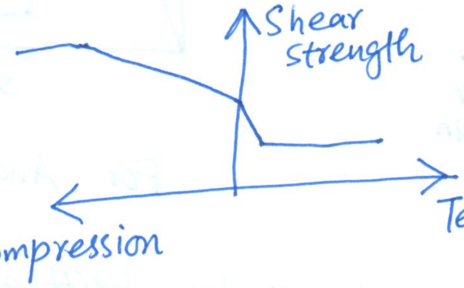


Shear stress and strain are constant over the element. In a pier (or beam both) the shear force is actually constant over the member length (unlike bending moment). Hence there is no need of a "hinge length" for shear.

Shear strength :-

Calculate the shear strength using ACI or other formulas.

The shear strength may depend on the axial stresses as shown below.



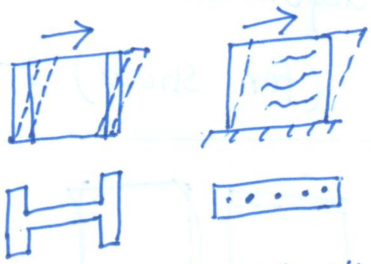
(The effect is similar to friction effect)

The PERFORM Inelastic Shear material currently does not account for this effect. You must account for it in the strength calculation.

PERFORM allows to account for this effect if you assume shear to be elastic

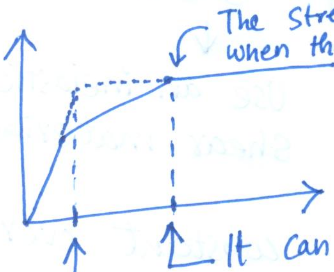
Shear Modulus :-

Usual formula: $G = \frac{E}{2(1+\nu)} = 0.4E$



Ok for steel

Not ok for concrete (Too big)



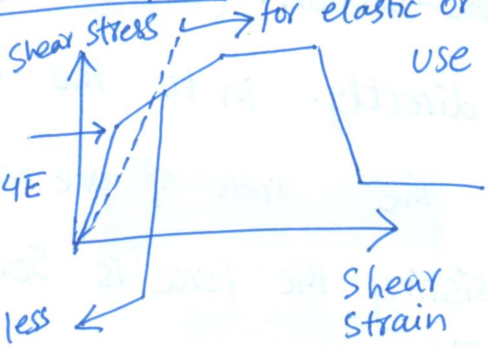
The strength is reached when the shear γ_{ft} yields

It can be shown (from Mohr's circle for strain that this shear strain is about 2 times the steel yield strain, or about 0.004)

The strain based on ~~0.4~~ $G = 0.4E$ is much smaller than 0.004 (e.g for 0.5% shear γ_{ft} it is about · 0.0003)

1/10 th or even less than this.

Shear FD Relationship:-



→ for elastic or bilinear inelastic, shear use a reduced modulus. (substaintially smaller)

before cracking $G=0.4E$ is OK

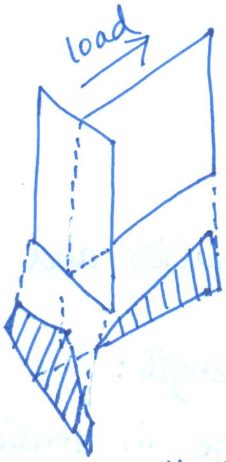
much less stiff after cracking

↓ In many cases the amount of shear def doesnt matter very much so if you are off by a factor of 10 it doesnt affect much. But in walls where shear defs do tend to be more significant part of total defs → This could be serious error.

ASCE 41 specifies $G=\frac{E}{2(1+\nu)}$. If shear deformations are significant, this is not accurate.

Modeling of 3D Walls:-

Here we get into the shear lag effects.



Short wall - substantial shear lag.

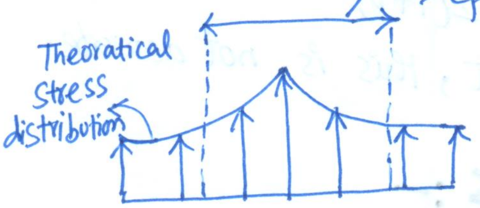
(The vertical stress at the edge of flanges are substaintially less than the location when this flange connects to the web)

Taller wall - little or no shear lag. (more uniform)

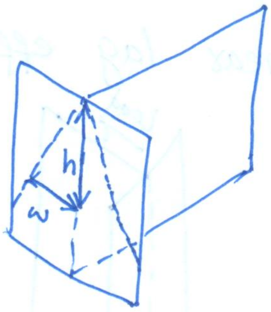
Bending stresses at base (Elastic Behavior)

[Shear Lag effect is caused when a member is connected to another member by only a portion of its cross-section. (because the forces cannot be transmitted directly in to the entire cross-section). This means that the area of one member that is effective in resisting the force is something less than the total area.]

Effective flange Width :-



effective width is as maximum (at middle) width which has same stress and same area. Means if you select flange of effective width and ignore shear lag effects, you will get same effect if you choose true width and do account for shear lag effects.



$\frac{w}{h}$ = effective width ratio

Must account for inelastic behavior and cyclic loading.

→ Paulay and Priestley for strength:

$\frac{w}{h} = 0.5$ for flange in tension

0.15 for flange in compression

→ ACI 318 : $\frac{w}{h} = 0.25$ for strength (based on flange in tension)

→ FEMA 273 : $\frac{w}{h} = 0.1$ for strength, 0.2 for stiffness

→ ASCE 41 is unclear.

A note about Paulay and Priestley values :-

The reccomendations from them are a bit counter intuitive. When the flange cracks in tension, you would expect the shear modulus to be smaller and the amount of shear lag would be larger so you expect the effective width to be smaller

but

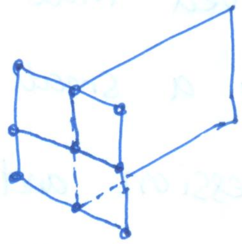
Paulay and priestley observed that in compression after few cycles, there is only a small amount of concrete which is in compression and other is cracked so effective width is smaller. However in tension, It can tend to bring in quite a lot of tensile reinforcement and its not a bad idea ~~to~~ in strength-based design to may be overestimate it a little bit so you are also overestimating the concrete stresses.

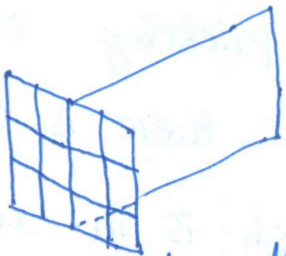
$$\frac{w}{h} = 0.5 \quad (\text{tension})$$

$$\frac{w}{h} = 0.15 \quad (\text{compression}).$$

Effective width depends on:-

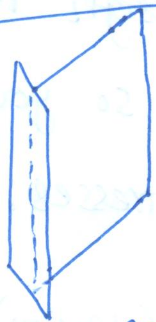
- The G (smaller G → smaller w)
- Whether the flange is in tension or Compression
- The amount of steel yielding and concrete cracking.
- The amount of cyclic loading
- Whether there is a biaxial load on the wall.
- In an analysis, the finite element mesh.



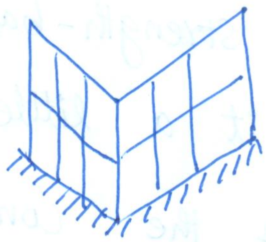


More elements across width tends to give more Shear lag (smaller effective width).

What do we do?

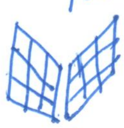


If its a sclender building, use the full Section. Assume full Section is effective



If a squat building, you model full structure and hope that modeling will take care of shear lag effects.

Or Model as separate walls → with extra material for flanges.



May be first option is better.

Rigid floor Constraint in PERFORM 3D

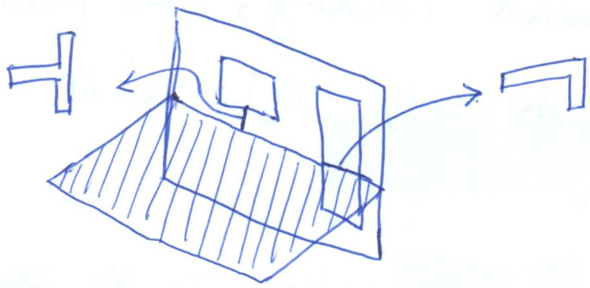
The nodes you want to assign rigid diaphragm, they must all have same V coordinate (i-e the floor must be level).

The first constraint type, "Horizontal Rigid floor (H1, H2, RV disps)" makes the H1 and H2 Translations and the V rotation (the rotation about the V axis) SAME at all those nodes. The V translation and H1 and H2 rotations are not affected.

→ All beams (typical models) in a rigid floor will have zero axial deformations. However if you use a beam model with fiber sections or PMM Concrete type hinges, the beam element may want to extend. Since you are restraining this extension, there may be compression forces in the beams.

→ If Applied support to a node → cannot be included in rigid diaphragm.

The effect of floor slab on Beams:-



If there is a composite action, it must be considered when coupling beams are modeled. Piers are not affected. Care must be taken if the floors are assumed to be rigid diaphrams.

Deformable floor diaphragms:-

A fine mesh of floors is OK for linear analysis.
For nonlinear analysis → fewer elements should be used.

These elements might have only shear stiffness (PERFORM Infill panel elements), and be combined with bar elements to model chords. (means may be you just use bar elements rather than wall elements).

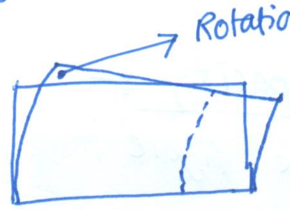
D/C Calculation – Axial-Bending Deformation

Since inelastic so we will not talk about strength D/C but deformation D/C.

- The main considerations for piers and beams are
  - a) Axial-bending behavior
  - b) Shear behavior

- Axial-bending behavior usually involves steel yielding, concrete cracking, and possibly concrete crushing (preferably not), so it will usually be inelastic.
- Inelastic axial-bending behavior in the piers may be limited to a hinge region (usually at the base), using capacity design.
- In a selender wall, shear in the piers is usually designed to remain essentially elastic (including in the hinge region).
- Shear in the coupling beams and panels may be inelastic.
- Coupling beams may be regarded as sacrificial (secondary) components, allowing large inelastic deformations. In ASCE41, for primary components, the deformation capacity have to be no larger than the ductile limit. If you consider as secondary component, you can allow larger deformations than ductile limit (means you can reduce strength). This is a design decision.

ASCE 41 D/C :-

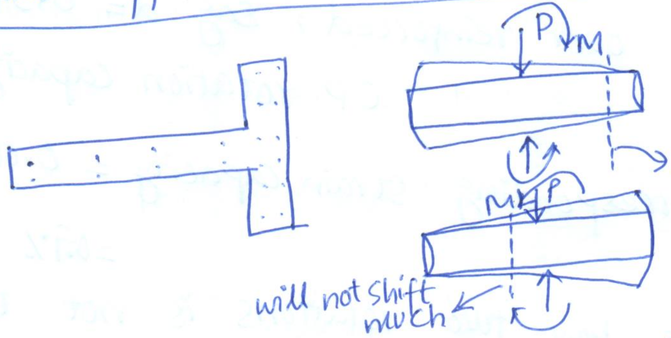


Smaller of $\frac{1}{2}$ of width or story height

In ASCE 41 the $\frac{D}{C}$ measure for axial-bending is rotation. This is similar to the rotation of a hinge or plastic zone in a column.

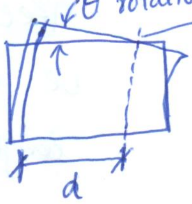
As for a concrete beam or column, ASCE 41 gives capacities in terms of plastic rotation, θ_p. (after yield)

Factors that affect rotation Capacity :-



The rotation capacity for (a) is larger than for (b). Case (b) acts like an over reinforced cross-section. A large compression force (P) reduces the capacity. A large shear force in the hinge region reduces the capacity.

Alternative Measure - Strain :



Fiber strain $= \frac{\theta d}{L}$

It is not difficult to convert rotation to strain.

Equivalent strain Capacity :-

Assume same figure,
hinge Length = $\frac{1}{2}W$ and consider extreme tension fiber.

Under reinforced, say $d = 0.85W$
CP rotation capacity from ASCE 41 = 1.5%

Corresponding strain capacity $\left(\varepsilon = \frac{\theta d}{L}\right)$

$$= 0.015 \times \frac{0.85W}{0.5W}$$

$$= 2.6\%$$

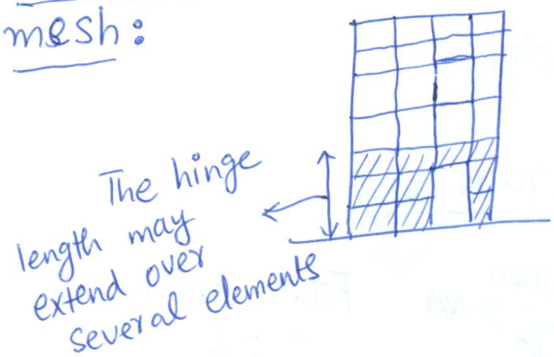
For over reinforced, say $d = 0.5W$
CP rotation capacity = 0.9%

Corresponding strain capacity $= 0.009 \times \frac{0.5W}{0.5W}$

$$= 0.9\%$$

The difference b/w two rotations is not that large but the difference b/w two strains capacities is quite substantial.

Problem : Hinge length may not match element mesh :



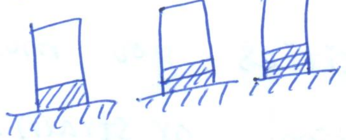
In PERFORM 3D, the rotation or strain in individual elements can be used for D/C calculation.

However the element sizes (the effective gage length) may not be equal to the hinge length.

If the mesh is subdivided and the elements get smaller, the calculated rotation and strain demands get increasingly larger.

Effect of element mesh and gage length on behavior:

eg we have
1) 1 element for full story (hinge length)
2) 2 elements over hinge length
3) 3 " " " "



Calculate
a) Strain at extreme fiber in lowest element
b) " " " over story height using a strain gage
c) Rotation over story height using a rotation gage.

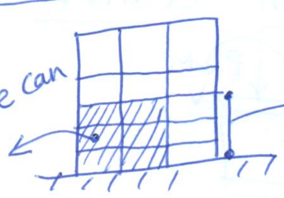
subjected to PO, dynamic EQ.

No of elements ↑ (a) ↑ (b) Remains Same (c) Remains Same

So (a) dependent on mesh, not a good D/C measure. Designing based on worst element may not be a good idea (concentration of strains in smaller mesh).

Solution - Rotation and strain gauges:

4-noded rotation gauge can cover several wall elements over the gage height.



2-noded strain gauge can cover several wall elements. It measures average strain over the gage length.

For D/C calculation, it is better to use rotation and/or strain gauges, not

rotations and strains in individual elements.
They dont have any effect on K matrix/behavior.

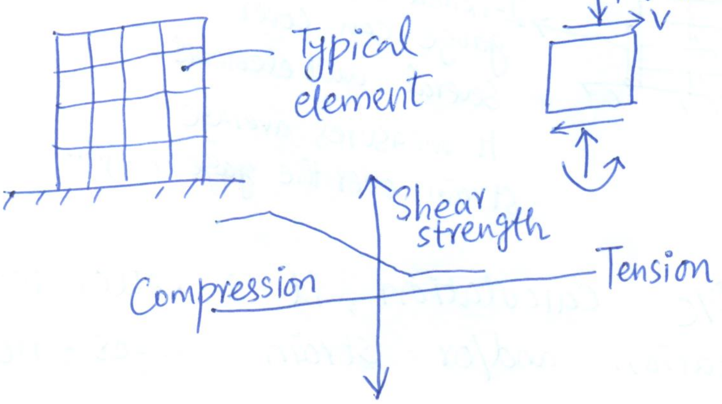
Procedure for D/C Calculation :-

- Use strain or rotation as D/C measure.
- The PERFORM gage capacities are total strain or rotation not plastic. To use the ASCE 41 Capacities, you must add on a yield rotation or strain.
- You can also use experimental results to get capacities.
- When you specify strain or rotation Capacities you must account for the amount of reinforcement, the axial force and the shear force. PERFORM does not do this.

D/C Calculation - Shear strength :-

We are assuming that you are designing the structure to remain elastic in shear.

Shear strength D/C for an element :-



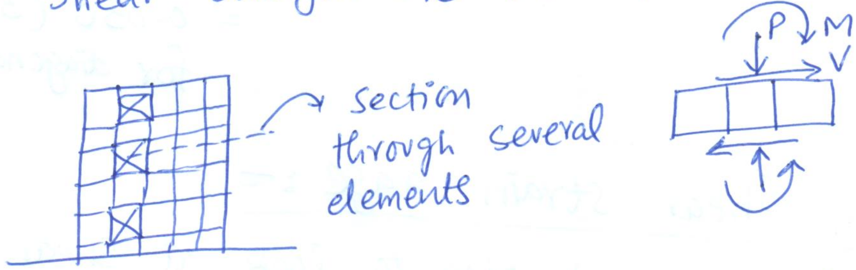
Calculate shear strength using ACI code or other sources.

In PERFORM, assign this strength to the shear material for the element. At each step of the analysis PERFORM calculates V and P and hence calculates the D/C ratio for shear strength.

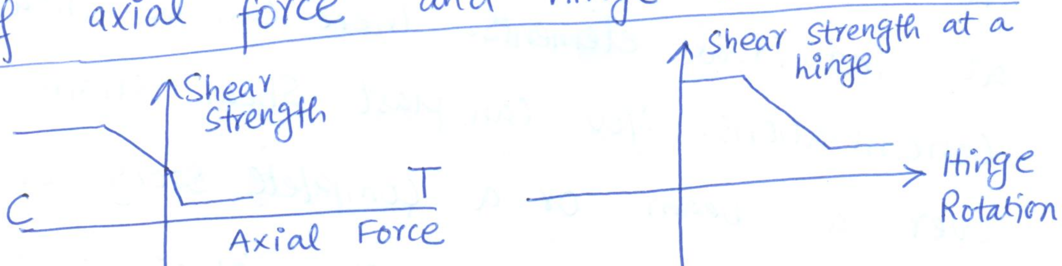
Shear strength D/C for a cross section :-

Usually it is better to check the shear strength for a cross section that covers several elements, because there may be shear stress concentrations.

In PERFORM 3D, you can assign a shear strength to a cross-section. At each step of the analysis PERFORM calculates V and P and hence calculates the shear strength D/C ratio for the section.



Effect of axial force and hinge rotation :-



- axial force effects can be considered in PERFORM 3D for single elements and cross-sections.

→ Hinge rotation effects cannot be considered for walls (they can be considered for beams)

→ When you calculate the shear strength capacity, at a hinge, you mush account for the expected hinge rotation.

D/C calculation - shear deformation :-

Shear deformation measure :

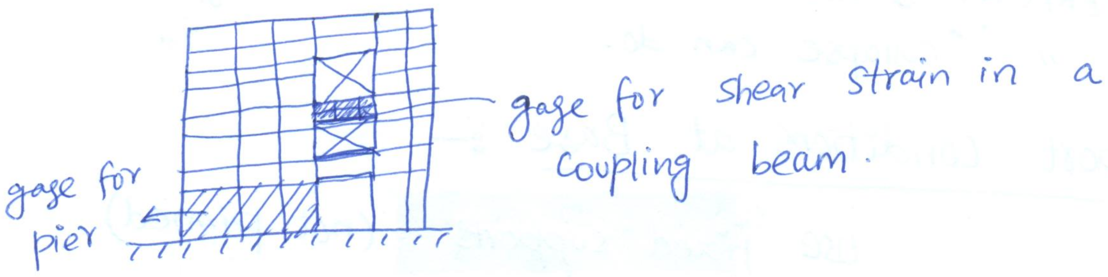
= Shear rotation (= Shear strain)

Typical ASCE 41 Capacity for a pier = 0.0075
= 0.75%
for CP Level.

Typical ASCE 41 ~~Cap~~ shear strain capacity for a coupling beam = 0.016 (1.6%)
for conventional rft.
= 0.030 (3%)
for diagonal rft.

PERFORM Shear strain gage :-

It is not a good idea to look at shear strain at individual elements. There can be shear strain concentrations. You can plast shear strain gage over a beam or a complete story or over some portion and use shear strain in that gage as D/C measure rather than individual elements.



Other Aspects:-

You can specify P-Δ effects directly in the wall elements
OR
you can specify use a separate "P-Δ" column."

First run without adding (then add and run, always good to compare)

[Also you can supress strength loss and add and compare]

Out-of-plane behavior :- (involves plate bending)

PERFORM assumes that this is elastic and separate (uncoupled) from the in-plane behavior.
You may want to ignore the out-of-plane bending contributions to the wall stiffness and strength. One way you might do this is to make the wall very thin for out-of-plane bending. Do not do this if you consider PΔ effects in the wall elements - the analysis is likely to predict that the wall buckles.
The out-of-plane stiffness is small. Usually you can use the actual wall thicknesses. You may specify a reduced E for out-of-plane bending, to account for cracking (say 0.5 E).

PERFORM 3D doesn't do cracked slab analysis.
" collapse can do.

Support conditions at Base :-

use fixed supports (not pinned)

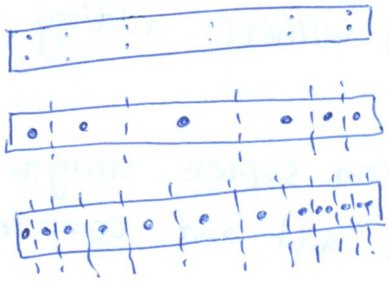
PERFORM 3D Wall element may have potential problem with out-of-plane bending (if pinned supports)

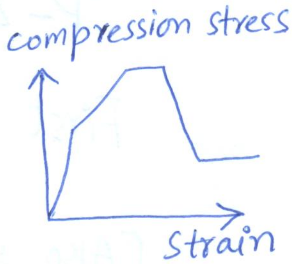
More realistic than Piano hinge behavior for out-of-plane bending.

Concrete Crushing :-

Wall section

Possible Concrete fibers (a) (b)





In an actual section, cracking and crushing spread continuously across the section. The NA also moves continuously.

In a fiber section, they occur discontinuously - fiber by fiber.

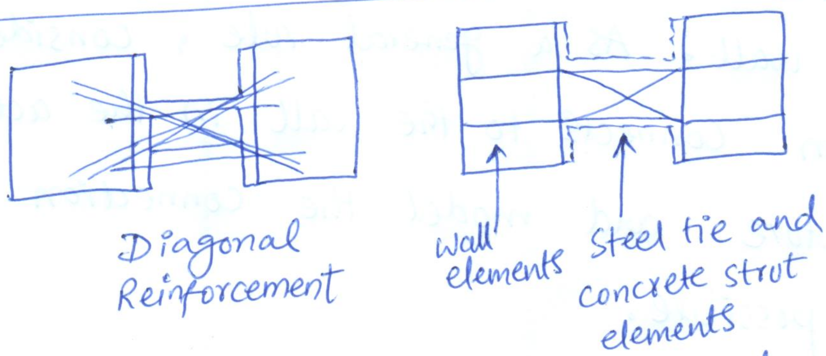
In a good design, there is likely to be little or no concrete crushing. If there is no crushing concrete fibers such as in (a) should be OK. The NA location is approximate but should be sufficiently accurate

If crushing occurs, it may requires more concrete fibers such as in (b).

One problem is that as the elements are made smaller, the calculated strains increase. Hence crushing occurs early.

This is a difficult topic. If possible, assume the concrete is elastic in compression. If not, run small analysis to check the section behavior.

Diagonally reinforced coupling panels:-



These are usually modelled with wall elements with conventional shear.

However they can also be modeled using steel tie and concrete strut elements. (in parallel)

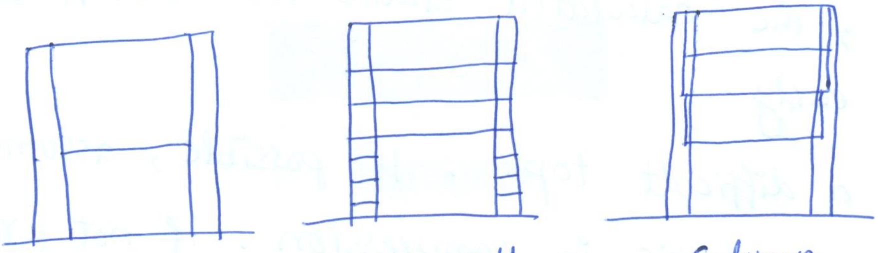
A diagonally reinforced panel must grow axially as the tension diagonal yields. This may be a problem if axial growth is restrained.

Irregular Meshes:- No auto mesh in PERFORM 3D

Distorted Elements:- Wall elements do not have to be exactly rectangular. However they should not be badly distorted. A rectangular mesh may match the reinforcement pattern and may be the best.

If irregular holes → can be non rectangular → OK.

Wall Supported on columns:-



If you use column elements, you must use imbedded elements to connect the columns to the wall. As a general rule, consider how the Column connects to the wall in the actual structure and model the connection as directly as possible.

Foundation flexibility :-

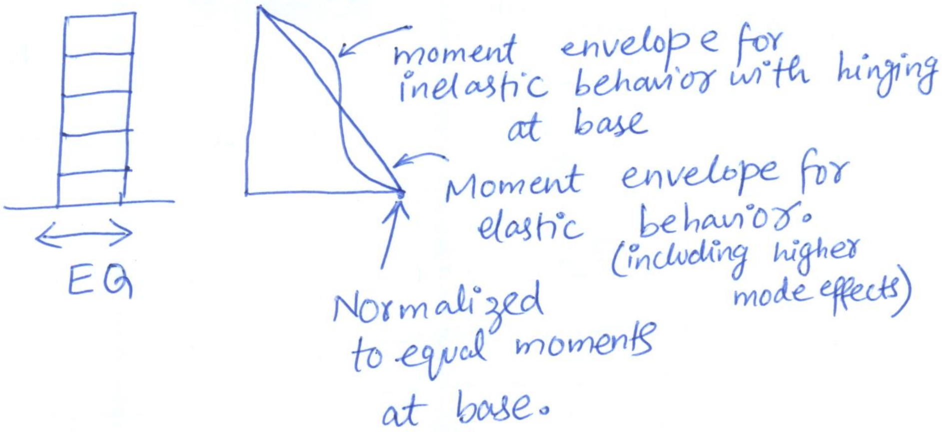
One approach is to estimate the foundation stiffness using and model it using support spring elements.

Alternatively, model the foundation (beams, piles etc.) explicitly.

Bending Moment Envelopes :- When a wall hinges at the base the maximum BMs can be very different from the case when the wall remains elastic. One explanation is that the higher modes are different from

When a hinge forms at base, and they can have a larger effect.

Elastic analysis can give an incorrect picture of the behavior.



So if you design the upper part of structure using the forces, Ms from elastic analysis, it may be underdesigned (may hinge up there). Same is true for shear.

CSi

PRESENTS

NONLINEAR ANALYSIS AND PERFORMANCE ASSESSMENT USING PERFORM-3D

SESSION 4 – NONLINEAR ANALYSIS

CSi

Topics for this Session

- Static push-over analysis.
  - Displacement control and event-to-event method.
  - Some more advanced aspects.
  - How is it used ?
  - Advantages and disadvantages.
- Dynamic step-by-step analysis.
  - Advantages and disadvantages.
  - Is it feasible (how much computer time)?
  - Hysteresis loops.
  - Damping.
  - Step-by-step integration.

CSi

2

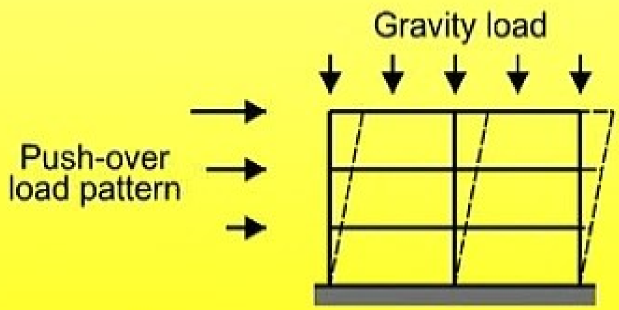
NEXT TOPIC

- Static push-over analysis.
  - Displacement control and event-to-event method.
  - Some more advanced aspects.
  - How is it used ?
  - Advantages and disadvantages.
- Dynamic step-by-step analysis.
  - Advantages and disadvantages.
  - Is it feasible (how much computer time)?
  - Hysteresis loops.
  - Damping.
  - Step-by-step integration.

CSi

3

Load Sequence

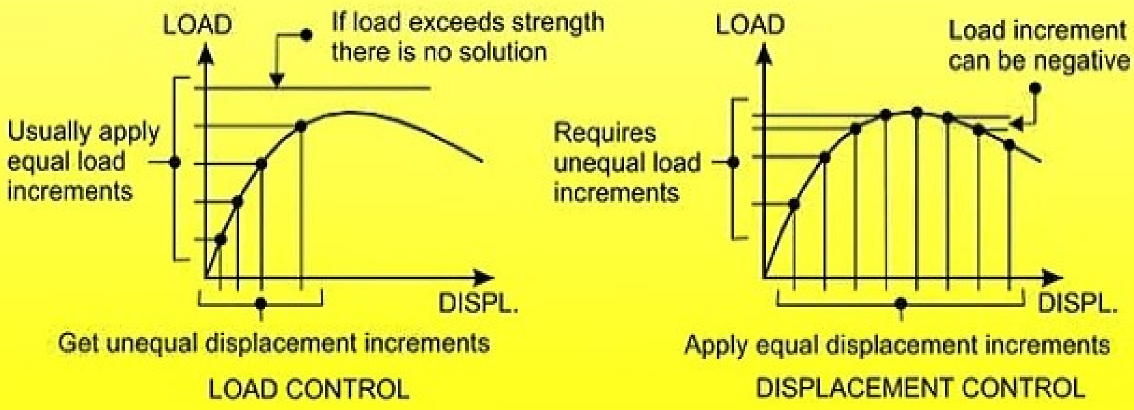


- Apply gravity loads first. Then add push-over loads, keeping the gravity loads constant.
- Specify the push-over load pattern only, not its magnitude. The magnitude is found during the analysis.
- The gravity load analysis is simple, and usually linear. The complicated part is the push-over analysis.
- The push-over analysis uses displacement control.

CSi

4

Load Control vs. Displacement Control

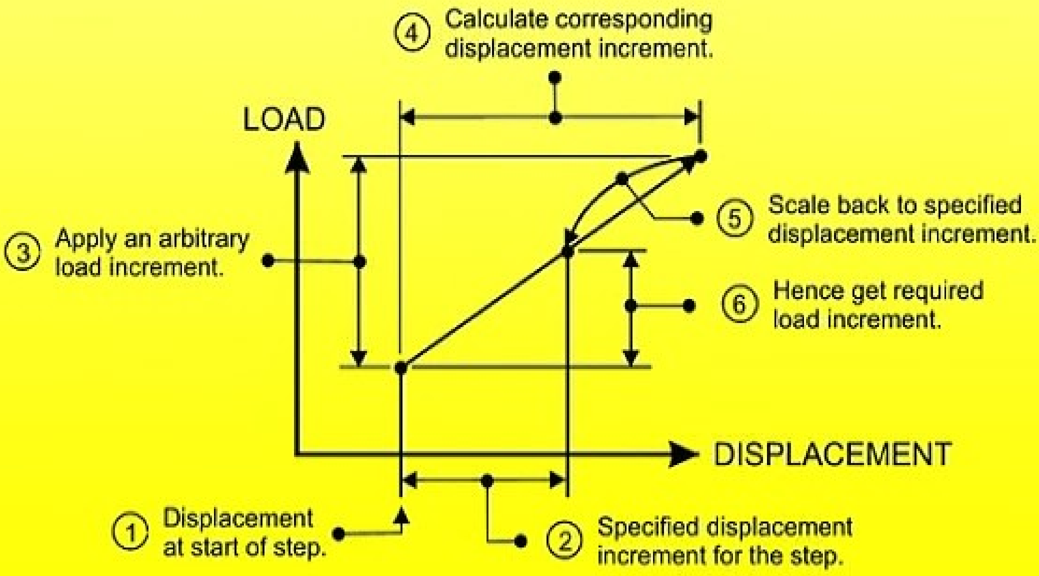


- For load control the load increment is specified for each step.
- For displacement control, the load increment is not specified. Instead, the displacement increment is specified, and the required load increment is calculated (see the next slide).
- Displacement control is not the same as imposed displacements. The loads are specified, not the displaced shape.

CSi

5

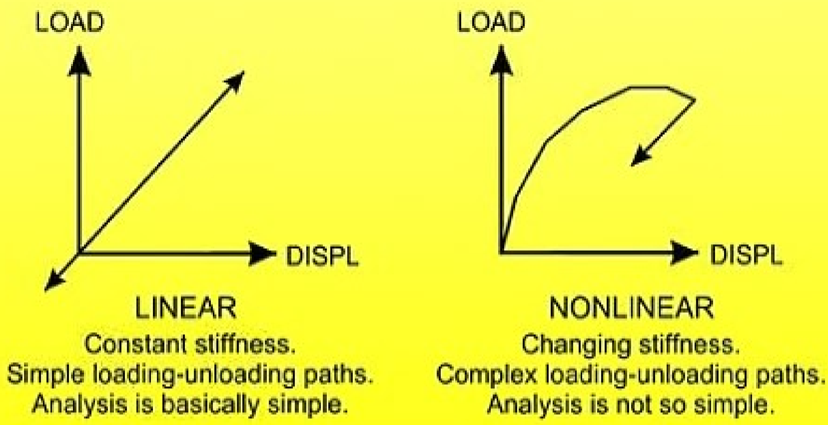
Displacement Control Procedure



CSi

6

Analysis With Nonlinear Events

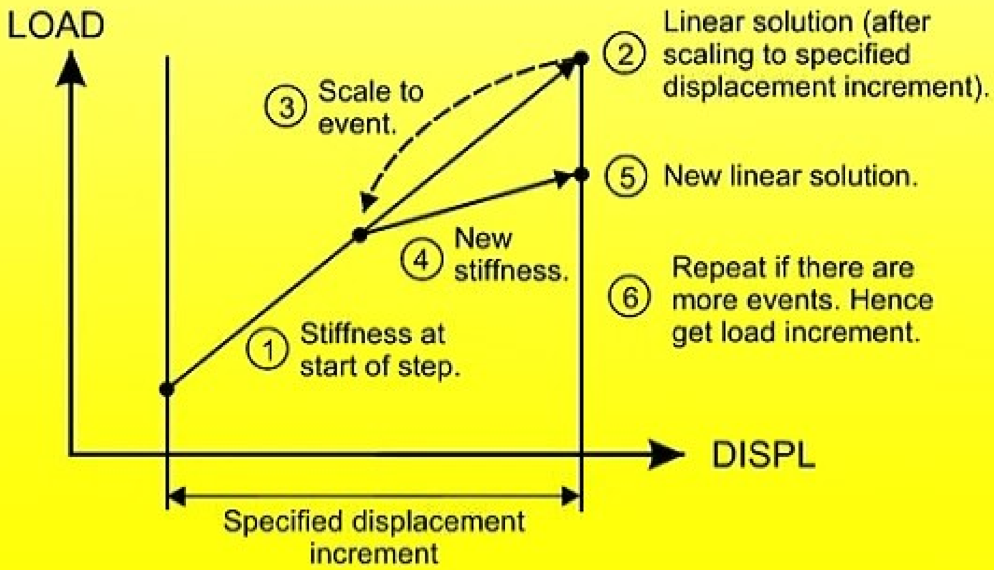


- A significant stiffness change is a nonlinear "event".
- PERFORM uses the event-to-event strategy. Essentially, the complete nonlinear behavior is traced out, from event to event.
- The most common events are yielding (stiffness reduces) and unloading (stiffness increases), but there are many others.

CSi

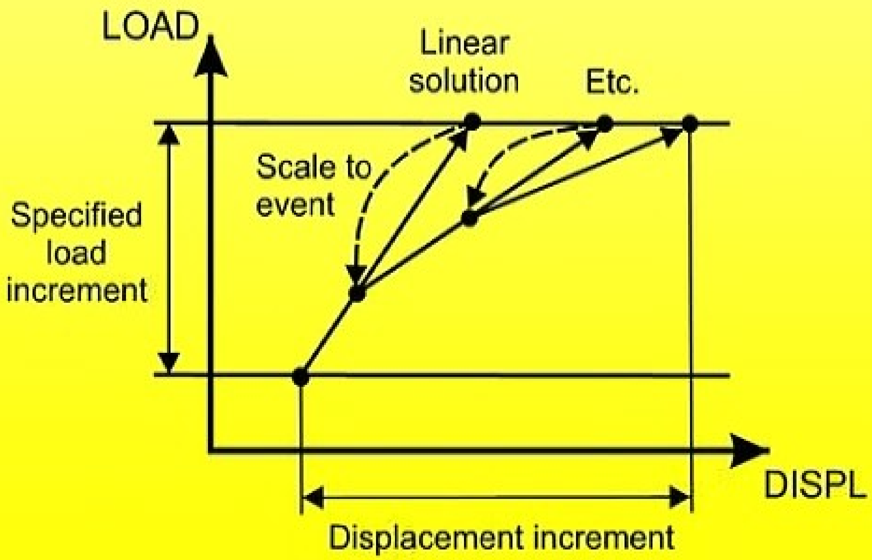
7

Displacement Control with Event-to-Event



CSi 8

Load Control with Event-To-Event



CSi 9

Reliability and Efficiency

- The event-to-event method is the most flexible and reliable method.
- It can be used for displacement controlled or load controlled analysis.
- Its major disadvantage is that the stiffness must be modified at each event (but not completely reformed).
  - As the structure size increases, the number of events tends to increase.
  - Also, the computer time to modify the stiffness increases.
  - Hence the computation cost tends to increase exponentially as the structure gets larger.
- There are many other nonlinear solution strategies. Sometimes they can be faster, but they are more difficult to use and less reliable.
- PERFORM-3D is very reliable, and is competitive in speed.

CSi

10

NEXT TOPIC

- Static push-over analysis.
  - Displacement control and event-to-event method.
  - Some more advanced aspects.
  - How is it used ?
  - Advantages and disadvantages.
- Dynamic step-by-step analysis.
  - Advantages and disadvantages.
  - Is it feasible (how much computer time)?
  - Hysteresis loops.
  - Damping.
  - Step-by-step integration.

CSi

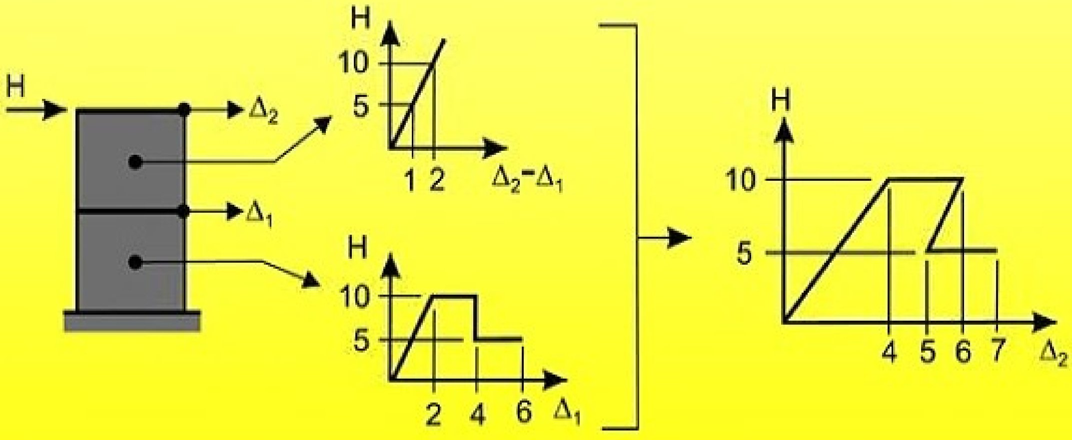
11

Controlled Displacements

- In a displacement controlled analysis, the displacement is increased in equal increments. A structure has many displacements. Which one should be incremented?
- For a PERFORM push-over analysis there are two key displacements, namely (1) the Reference Displacement, and (2) one or more Controlled Displacements.
- The reference displacement is usually the roof displacement. When a push-over curve is drawn, the X axis is the reference displacement and the Y axis is the horizontal load.
- If there is only one controlled displacement this is also usually the roof displacement. This is incremented during the analysis, in equal steps.
- However, the roof displacement may not be "well behaved", and it may not be possible to increase it in equal steps. In this case it is necessary to use multiple controlled displacements.
- This is illustrated in the next slide.

12

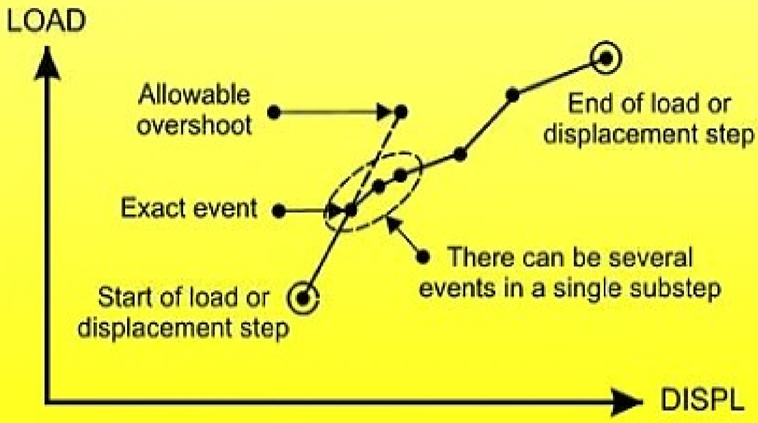
A Badly Behaved Displacement



- The top story is elastic. The bottom story yields and loses strength.
- The load vs. roof displacement curve doubles back on itself.
- The roof displacement is "badly behaved". If this is the only controlled displacement, there may not be a solution.
- For cases like this it is essential to use several controlled displacements, not just one. In this example use both Δ_2 and Δ_1.

13

Event Overshoot

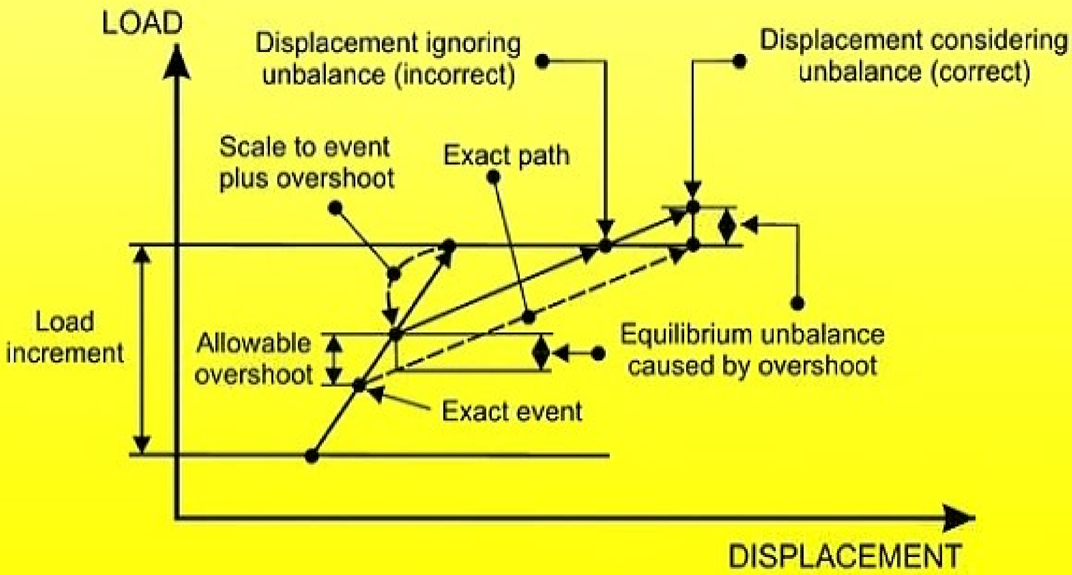


- The computation cost can be reduced by allowing event overshoot, so that several events can occur in a single substep.
- The disadvantage is that there is a temporary equilibrium unbalance.
- In PERFORM the overshoot tolerance is typically 1% of the component strength. However, larger (and also smaller) values can be specified.
- Larger overshoot tolerances can be allowed for larger structures – they can usually absorb larger equilibrium unbalances.

CSi

14

Event Overshoot for Load Control



Overshoot can also be used with displacement control

CSi

15

In PERFORM-3D You Must Specify :

- The push-over load pattern. The choices are (a) specified loads, (b) loads based on masses and a displaced shape, or (c) loads based on masses and mode shapes.
- The maximum displacement (PERFORM uses drift ratio). Specify a reasonable value (a 20% drift is not reasonable).
- The number of displacement steps. Suggested = 50. Do not use very large numbers (e.g., not 500).
- The maximum number of events in any step. This will stop the analysis if there are convergence problems.
- A limit state to stop the analysis if the deformations become unreasonably large and there is no point in continuing. In general, use the default (stop if any component is deformed past the X point on its F-D relationship).
- The controlled displacements. Do not rely on a single controlled displacement, especially if there is strength loss.
- The amount of event overshoot. Default = 1%. You can often increase this for large structures.

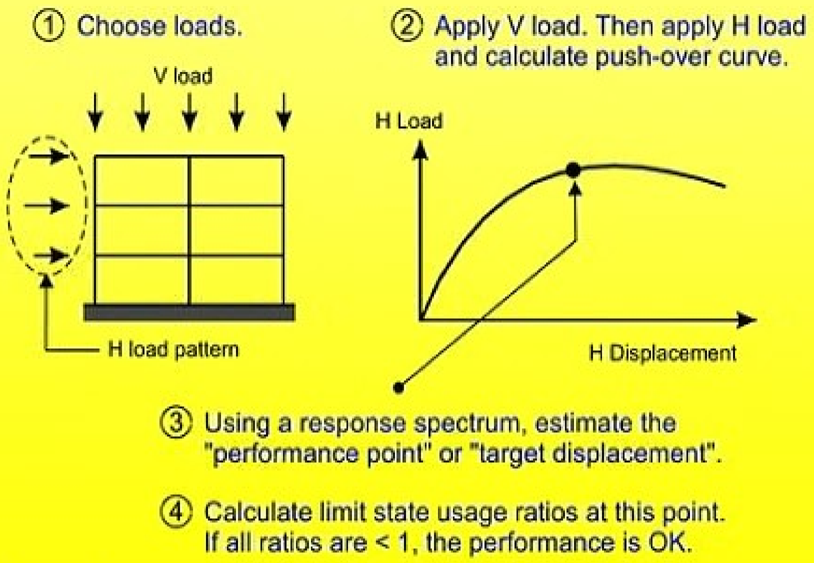
CSi 16

NEXT TOPIC

- Static push-over analysis.
  - Displacement control and event-to-event method.
  - Some more advanced aspects.
  - How is it used ?
  - Advantages and disadvantages.
- Dynamic step-by-step analysis.
  - Advantages and disadvantages.
  - Is it feasible (how much computer time)?
  - Hysteresis loops.
  - Damping.
  - Step-by-step integration.

CSi 17

Overall Steps for Performance Assessment

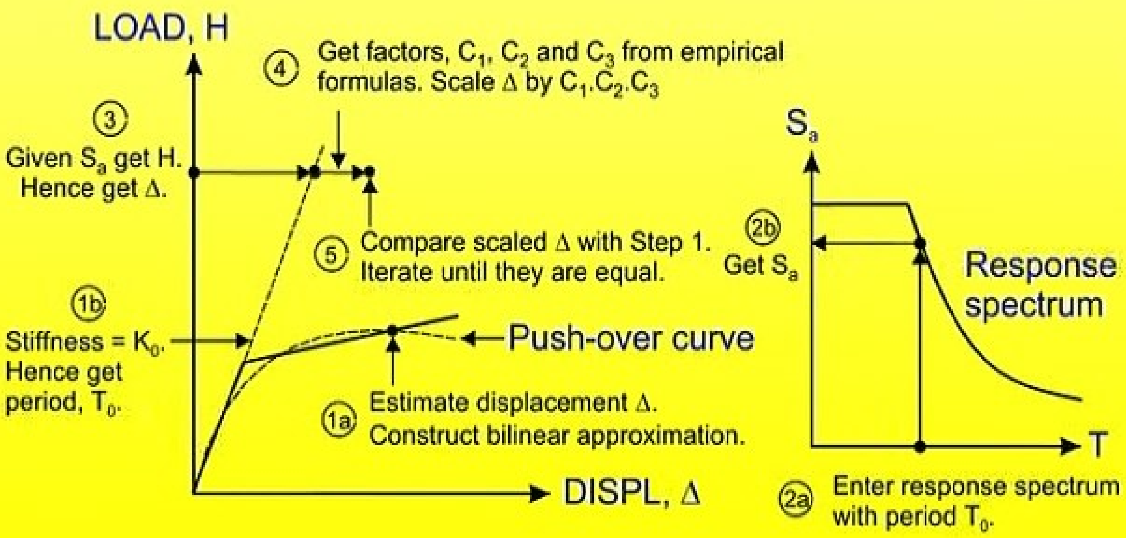


- The push-over analysis just gives a push-over curve.
- For performance evaluation the performance point (target displacement) must be determined.

CSi

18

Target Displacement Calculation "Coefficient" Method

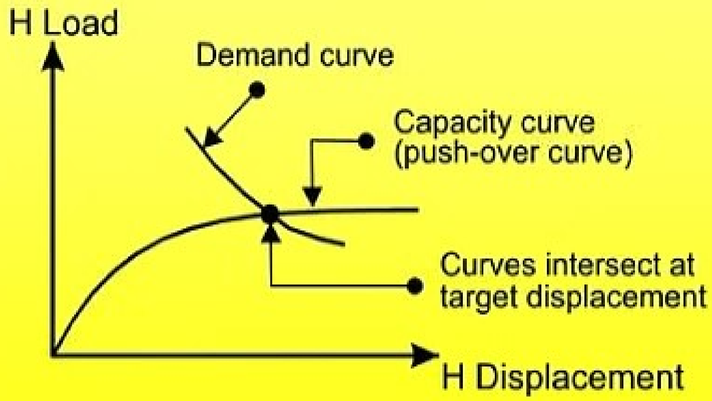


See seminar notes for other methods.

CSi

19

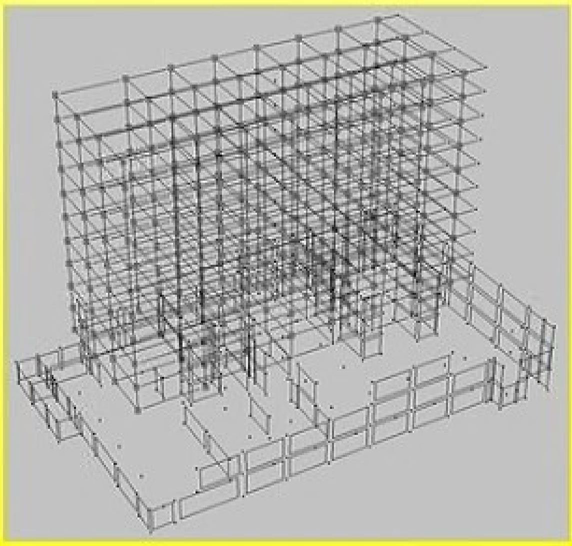
PERFORM-3D Procedure



- PERFORM-3D includes a number of methods for calculating the target displacement. A demonstration was given in Session 1.
- PERFORM-3D can then calculate limit state usage ratios at the target displacement. If all usage ratios are <1, the performance requirements are satisfied.

CSi 21

PERFORM-3D Computer Time



- 11 story steel frame.
- 3 bays x 7 bays.
- 720 nodes, 2100 DOFs.
- 960 inelastic beam and column elements.
- 320 inelastic panel zone elements.
- 1000 steps @ 0.02 seconds.
- 3 Ghz PC.
- Time about 10 minutes for one earthquake.

CSi 29

NEXT TOPIC

- Static push-over analysis.
  - Displacement control and event-to-event method.
  - Some more advanced aspects.
  - How is it used ?
  - Advantages and disadvantages.
- Dynamic step-by-step analysis.
  - Advantages and disadvantages.
  - Is it feasible (how much computer time)?
  - Hysteresis loops.
  - Damping.
  - Step-by-step integration.

CSi

22

Push-Over Analysis : Advantages

- It uses a response spectrum. Dynamic analysis requires ground motions.
- It requires less computer time than dynamic analysis.
- It can provide sensitivity information on the effects of changing the strength and stiffness.

CSi

23

Push-Over Analysis : Disadvantages

- It does not account directly for the dynamic nature of earthquake loads.
- It does not account directly for hysteresis loops. Stiffness degradation and energy dissipation are considered only indirectly.
- It can work well if the structure responds in essentially a single mode of vibration. It is less accurate, and possibly inaccurate, for tall buildings.
- Static push-over analysis is a valuable part of the design process. However, it has limitations for performance assessment.

CSi

24

NEXT TOPIC

- Static push-over analysis.
  - Displacement control and event-to-event method.
  - Some more advanced aspects.
  - How is it used ?
  - Advantages and disadvantages.
- Dynamic step-by-step analysis.
  - Advantages and disadvantages.
  - Is it feasible (how much computer time)?
  - Hysteresis loops.
  - Damping.
  - Step-by-step integration.

CSi

25

Dynamic Analysis Disadvantages

- A dynamic analysis model is rather more complex than a static push-over model.
- A response spectrum can not be used. The analysis requires ground motions.
- The response can be sensitive to changes in the ground motion. Analyses must be carried out for a number of earthquakes.
- It requires more computer time than push-over analysis.
- The analysis does not give sensitivity information. Hence it is more difficult to evaluate alternative designs.

CSi 26

Dynamic Analysis Advantages

- It applies to structures of all types.
- It accounts directly for the dynamic nature of earthquake loads.
- It accounts directly for hysteresis loops and energy dissipation.

- Dynamic analysis is more general and more accurate than push-over analysis.

CSi 27

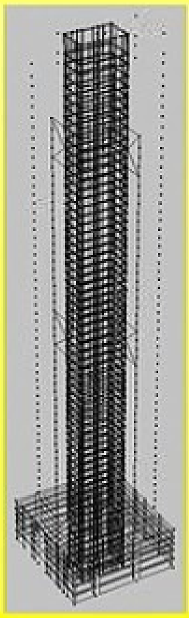
NEXT TOPIC

- Static push-over analysis.
  - Displacement control and event-to-event method.
  - Some more advanced aspects.
  - How is it used ?
  - Advantages and disadvantages.
- **Dynamic step-by-step analysis.**
  - Advantages and disadvantages.
  - **Is it feasible (how much computer time)?**
  - Hysteresis loops.
  - Damping.
  - Step-by-step integration.

CSi

28

PERFORM-3D Computer Time

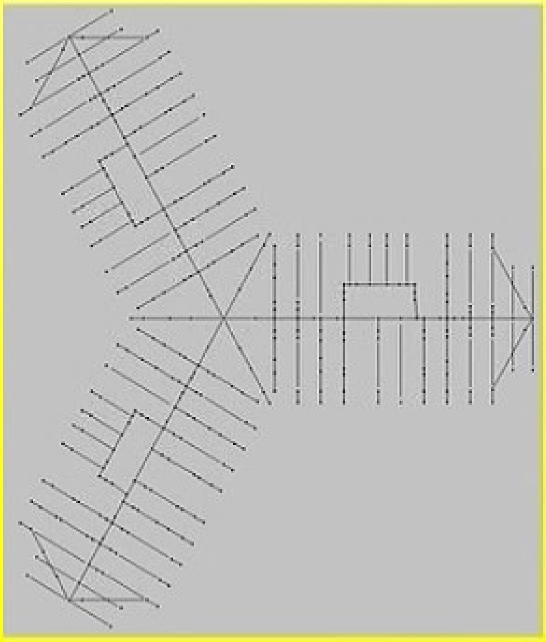


- 61 stories.
- 2250 nodes, 6200 DOFs.
- 1200 elastic wall elements.
- 120 inelastic wall elements.
- 320 beams and columns.
- 16 BRBs.
- 1000 steps @ 0.02 seconds.
- 3GHz PC.
- Time about 2 hours for one earthquake.

CSi

30

PERFORM-3D Computer Time



- Very large and complex.
- 50 stories.
- 16,700 nodes.
- 48,000 DOFs.
- 12,500 inelastic wall elements.
- 9,400 inelastic beam elements.
- 1000 steps @ 0.02 seconds.
- 2 GHz PC.
- Time about 3 days for one earthquake.

CSi

31

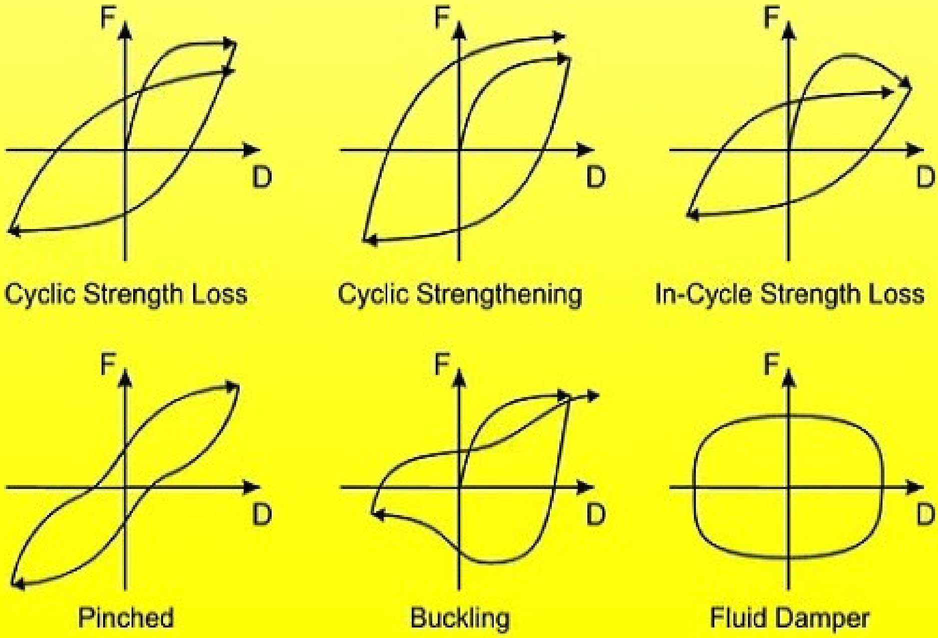
NEXT TOPIC

- Static push-over analysis.
  - Displacement control and event-to-event method.
  - Some more advanced aspects.
  - How is it used ?
  - Advantages and disadvantages.
- Dynamic step-by-step analysis.
  - Advantages and disadvantages.
  - Is it feasible (how much computer time)?
  - Hysteresis loops.
  - Damping.
  - Step-by-step integration.

CSi

33

Some Hysteresis Loop Shapes



CSi

Hysteresis Loops : Main Issues

- Under cyclic loading, inelastic components dissipate energy. Dissipated energy = area under hysteresis loop.
- The sizes and shapes of the loops can significantly affect the response of the structure.
- In static push-over analysis, hysteresis loops are considered only implicitly (there is no cyclic loading in push-over analysis).
- In dynamic analysis hysteresis loops can be (and must be) considered explicitly.
- Different components can have very different hysteresis loops.

CSi

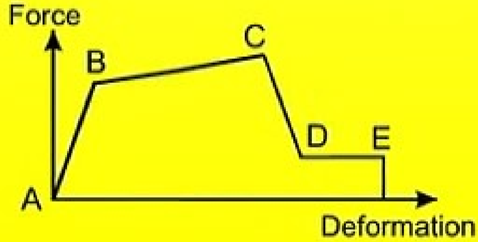
Hysteresis Loops in ASCE 41

ASCE 41 states the following :

"The complete hysteretic behavior of each component shall be modeled using properties verified by experimental evidence."

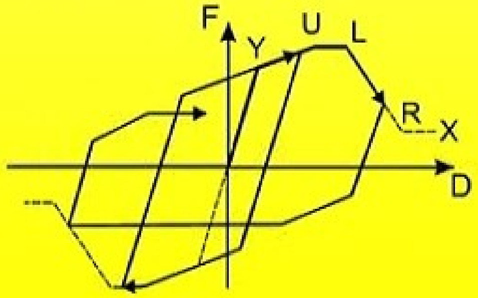
"The use of the (F-D relationship shown below) to represent the envelope relation for the analysis shall be permitted."

"Unloading and reloading properties shall represent significant stiffness and strength degradation characteristics."

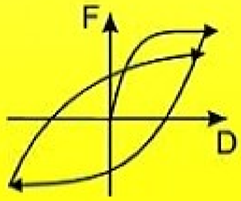


Basic Hysteresis Loop in PERFORM

- PERFORM-3D gives you substantial control over the hysteresis loop (details in the next few slides).
- The PERFORM loop is based on the YULRX "backbone" relationship. The ASCE 41 relationship is a special case.
- The loop does not account for progressive strength loss under constant amplitude cycling. This must be accounted for in the backbone relationship.



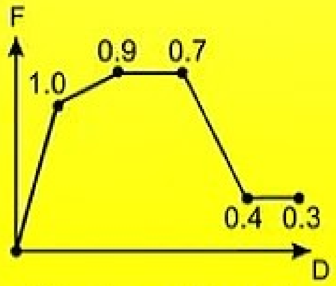
The backbone relationship envelopes the hysteresis loops.



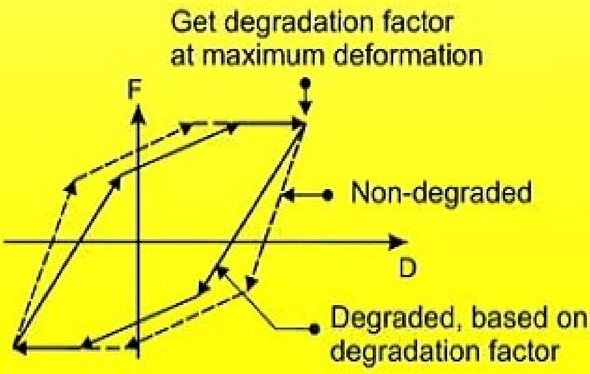
Cyclic strength loss is not considered directly.
It must be accounted for in the backbone relationship.

PERFORM Loop : Energy Degradation

- "Degradation factors" can be specified. They depend on the maximum deformation, for example as shown below.
- Area of degraded loop = (non-degraded area) x (degradation factor).
- PERFORM automatically adjusts the unloading-reloading stiffnesses to give the required energy degradation.

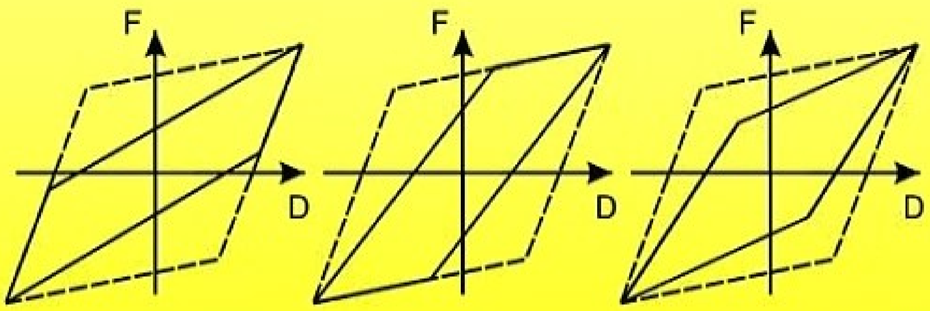


Possible degradation factors, (e.g., based on experiments)



Hysteresis loop

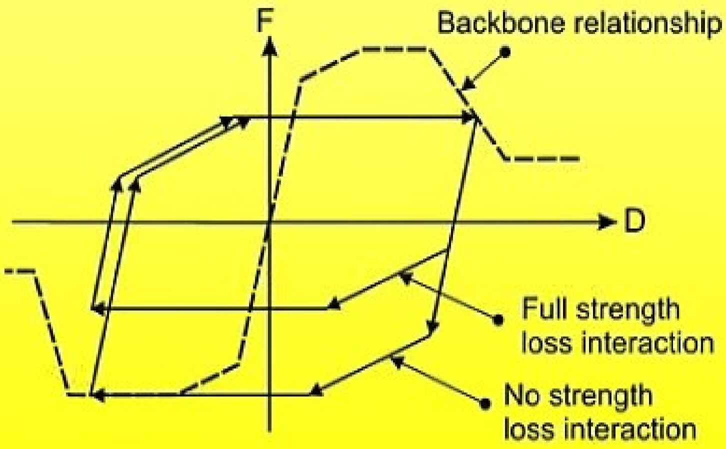
PERFORM Loop : Unloading Stiffness



Loop A Loop B Loop C

- All three loops have the same energy degradation.
- Loop A has the maximum unloading stiffness and the smallest elastic range.
- Loop B has the smallest unloading stiffness and the largest elastic range.
- Loop C is between Loops A and B.
- In PERFORM you can control the loop shape, with Loops A and B as the extremes.
- The figures show a bilinear F-D relationship. The trilinear case is similar.

PERFORM Loop : Strength Loss Interaction



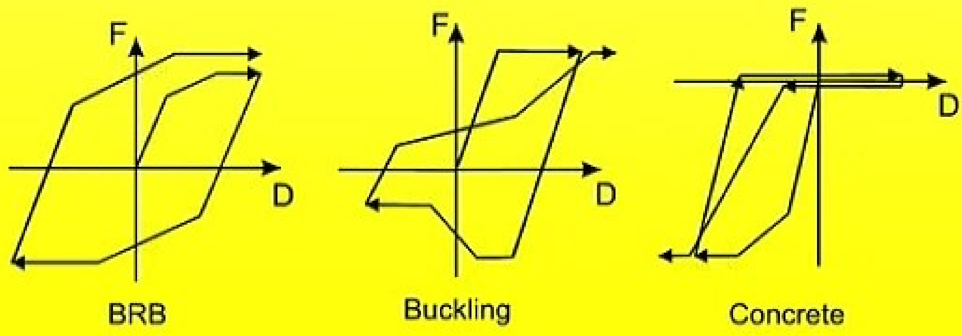
- With no strength loss interaction, if strength loss occurs in one direction there is no loss in the other direction.
- With full strength loss interaction, if strength loss occurs in one direction, the same loss occurs in the other direction.
- In PERFORM you can control the loop shape, with full and no interaction as the extremes.

CSi

40

PERFORM Loop : Special Cases

- The loop for a Buckling Restrained Brace includes cyclic expansion (“isotropic hardening”).
- A buckling strut has a special loop.
- A concrete material cracks in tension and has special unloading-reloading behavior.

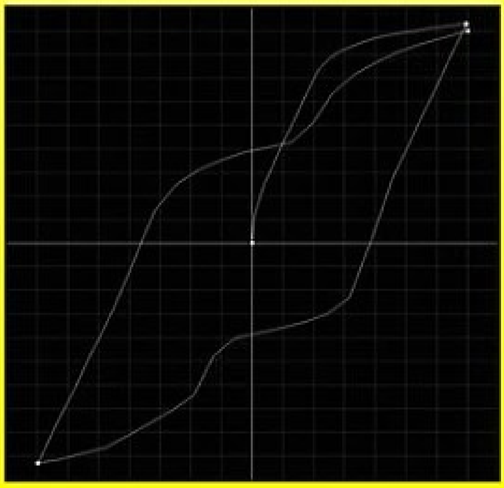


CSi

41

PERFORM Loop : Fiber Section

- The PERFORM basic loop does not allow a pinched shape.
- However, reinforced concrete fiber sections (for beams, columns and walls) do have pinched loops.
- The figure shows an example (from a PERFORM analysis of a wall).



CSi

42

NEXT TOPIC

- Static push-over analysis.
  - Displacement control and event-to-event method.
  - Some more advanced aspects.
  - How is it used ?
  - Advantages and disadvantages.
- **Dynamic step-by-step analysis.**
  - Advantages and disadvantages.
  - Is it feasible (how much computer time)?
  - Hysteresis loops.
  - **Damping.**
  - Step-by-step integration.

CSi

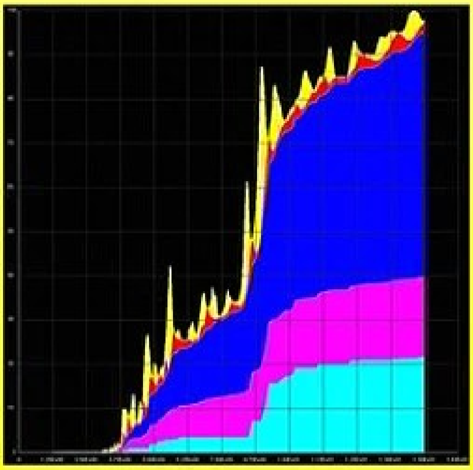
43

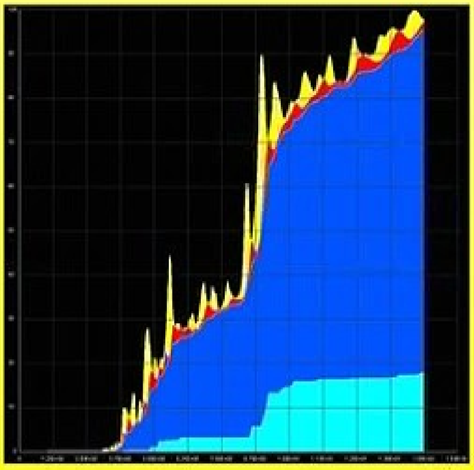
The Purpose of Damping

- An elastic structure dissipates energy by a variety of mechanisms. This is almost always modeled as viscous damping.
- When a structure yields it dissipates energy more directly, through inelastic action. The dissipated energy for a component is the area under its hysteresis loops.
- However, this does not account for all energy dissipation – there is still a lot of "elastic" energy dissipation.
- Again, this is almost always modeled as viscous damping.
- For nonlinear analysis there are no mode shapes, so the usual "5% modal damping" can not be used. (It could be, but the mode shapes would have to be re-calculated at every nonlinear event, which is impractical.)
- PERFORM has two models for elastic energy dissipation, namely Rayleigh damping and "Modal" damping.

44

Energy Balance Example 2





- 10 story concrete shear wall.
- Left figure = 5% Rayleigh at $0.2T_1$ and $0.9T_1$.
  Total energy = 4762. Inelastic energy = 1024. Roof drift = 1.71%
- Right figure = 5% Modal + very small Rayleigh.
  Total energy = 4460. Inelastic energy = 789. Roof drift = 1.77%
- The differences are significant. You may have to try both methods.

56

Damping Matrix and Damping Forces

At any point in time, the dynamic analysis satisfies the following equilibrium equation :

$$\mathbf{R}_{\text{external}} = \mathbf{R}_{\text{internal}} = \mathbf{R}_K + \mathbf{R}_M + \mathbf{R}_C$$

For linear analysis : $\mathbf{R}_K$ = (stiffness) x (displacement) = $\mathbf{K}\,\mathbf{r}$

For nonlinear analysis : $\mathbf{R}_K = \sum_{\text{all steps}} \mathbf{K}_{\text{tangent}}\,\Delta\mathbf{r}$

For both linear and nonlinear analysis :

$\mathbf{R}_M$ = (mass) x (acceleration) = $\mathbf{M}\,\ddot{\mathbf{r}}$

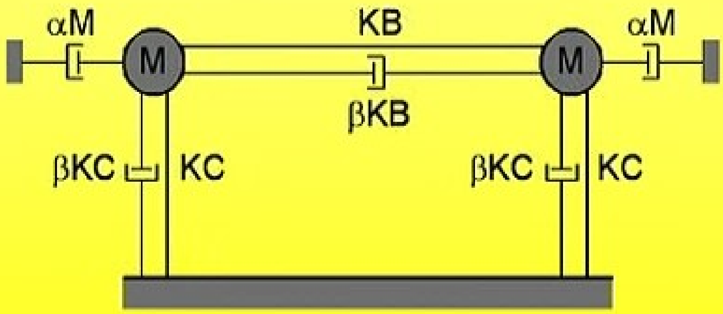
and usually $\mathbf{R}_C$ = (damping) x (velocity) = $\mathbf{C}\,\dot{\mathbf{r}}$

The mass and stiffness matrices, **M** and **K**, are well defined.

For the damping forces we need a damping matrix, **C**.

45

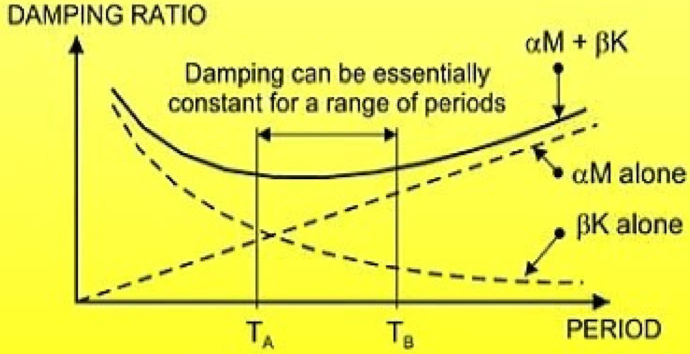
Rayleigh Damping



- The αM dampers connect the masses to the ground. They exert external damping forces.
- The βK dampers act in parallel with the elements. They exert internal damping forces.
- The damping matrix is $\mathbf{C} = \alpha\mathbf{M} + \beta\mathbf{K}$.
- The physical meaning of the model is clear.
- How to choose the α and β parameters is not so clear.

46

Rayleigh Damping : Linear Analysis

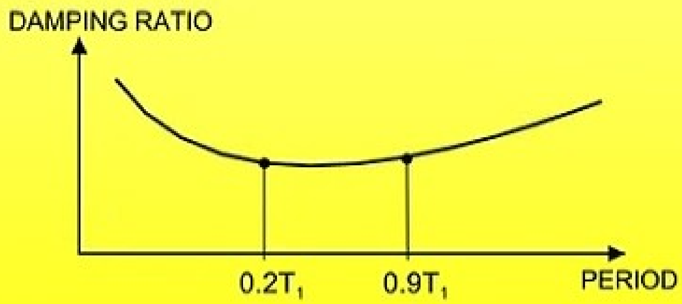


- For linear analysis, the coefficients α and β can be chosen to give essentially constant damping over a range of mode periods, as shown.
- A possible method is as follows :
  - Choose T_B = 0.9 times the first mode period.
  - Choose T_A = 0.2 times the first mode period.
  - Calculate α and β to give 5% damping at these two values.
- The damping ratio is essentially constant in the first few modes.
- The damping ratio is higher for the higher modes.

CSi

47

Rayleigh Damping in PERFORM-3D Suggested Procedure

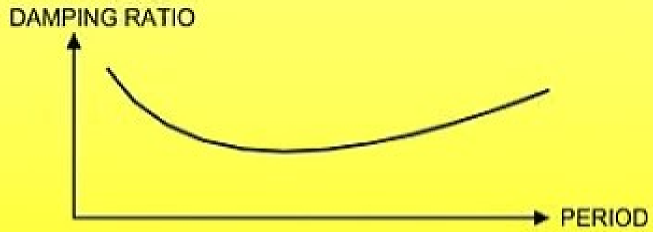


- Choose the damping ratio, typically 5%.
- Specify that this damping ratio is to apply at 0.2 and 0.9 times the elastic first mode period (or you can choose two other values).
- PERFORM-3D calculates the first mode period, then calculates the required α and β values.

CSi

48

An Interesting Contradiction



- From the diagram, a longer period means a larger damping ratio.
- As a structure yields, its periods tend to increase. Hence, we might expect the viscous energy dissipation to increase.
- However, for a given amplitude of vibration :

  A longer period means smaller deformation rates.
  The damping matrix, $C = \alpha M + \beta K$ is fixed.
  Hence the damping forces decrease as the period increases.
  Hence, the energy dissipation decreases.

- The βK damping does damp out high frequency vibrations. This is usually good, because they can cause numerical difficulties.

CSi

Modal Damping : Linear Analysis

- In a linear analysis the modes are uncoupled. The structure vibrates independently in each mode, and each mode is damped independently.
- Modal damping corresponds to a specific damping matrix, as shown below.
- Modal damping gives both external and internal damping forces. These are not the same as the forces from Rayleigh damping.

For n^{th} mode : $\mathbf{C}_n = \xi_n \dfrac{4\pi}{T_n} (\mathbf{M}\,\phi_n)(\mathbf{M}\,\phi_n)^T$

For structure : $\mathbf{C} = \Sigma\, \mathbf{C}_n$

Where : ξ = damping ratio, T = period,
$\mathbf{M}$ = mass matrix, ϕ = mode shape.

CSi

Modal Damping in PERFORM-3D

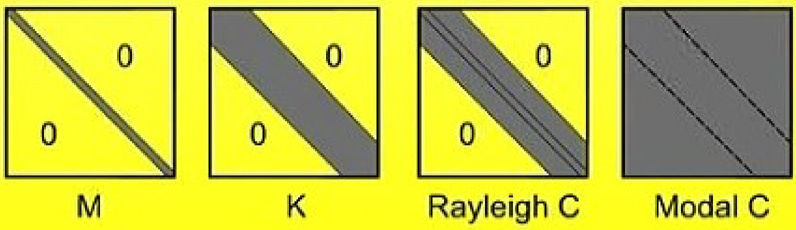
- PERFORM uses the same damping matrix as for linear analysis, based on the mode shapes and periods for the initial elastic structure.
- The damping matrix stays constant.

- As the structure yields, instantaneous mode shapes and periods exist. The mode shapes and damping matrix could be recalculated.
- However, this would be very expensive computationally.
- Also, changing the damping matrix as the structure yields is not necessarily a good idea. For example, there is a sudden equilibrium unbalance when the damping matrix changes.

CSi

Modal Damping Effect on Stiffness Matrix

- The mass matrix, M, is diagonal.
- The stiffness matrix, K, is banded.
- For Rayleigh damping, $C = \alpha M + \beta K$, so C is banded.
- For modal damping, C is generally a full matrix (not banded).



- Modal damping terms that are outside the stiffness matrix band are "right hand sided" in the analysis. This causes equilibrium errors in each time step that are corrected in the next step.
- This does not seem to cause any problems for frame structures. However, be careful with shear walls – compare with Rayleigh damping.

CSi

Combined Modal and Rayleigh Damping

- Suppose a structure has N total degrees of freedom (say N = 10000) and the damping matrix is based on M modes (say M = 20).
- There are many undamped degrees of freedom (10000 – 20 = 9980). This is not good.
- Always add a small amount of Rayleigh βK damping, to damp out high frequency vibrations.
- For example, specify 5% modal damping plus enough βK damping to give 0.02% damping at the first mode period.

CSi

53

Energy Balance

- Check energy balance, to see if the viscous energy looks reasonable.
- Unfortunately there are currently no guidelines.

CSi

54

Energy Balance Example 1

- 3-story steel frame building.
- Left figure = 5% Rayleigh at $0.2T_1$ and $0.9T_1$.
  Total energy = 5880. Inelastic energy = 3030. Roof drift = 2.97%
- Right figure = 5% Modal + very small Rayleigh.
  Total energy = 5850. Inelastic energy = 3080. Roof drift = 2.97%
- No significant difference.

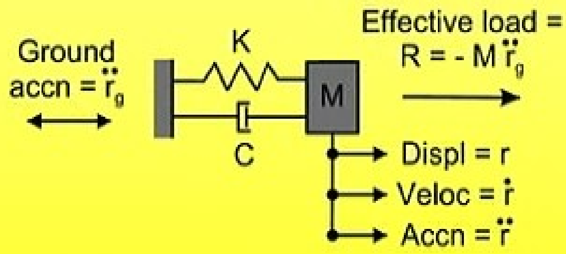
55

NEXT TOPIC

- Static push-over analysis.
  - Displacement control and event-to-event method.
  - Some more advanced aspects.
  - How is it used ?
  - Advantages and disadvantages.
- Dynamic step-by-step analysis.
  - Advantages and disadvantages.
  - Is it feasible (how much computer time)?
  - Hysteresis loops.
  - Damping.
  - Step-by-step integration.

57

Dynamic Equilibrium



- At any point in time, dynamic equilibrium is :

$$M\ddot{r} + C\dot{r} + Kr = R$$

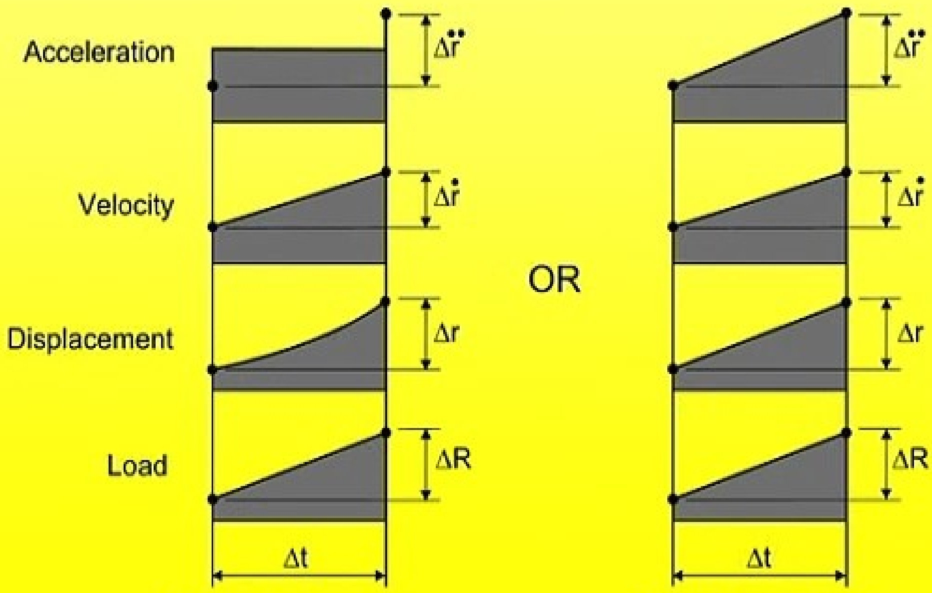
- Over a time step, Δt, dynamic equilibrium is :

$$M\,\Delta\ddot{r} + C\,\Delta\dot{r} + K\,\Delta r = \Delta R$$

- This equation can be solved by step-by-step methods.
- There is one equation with three unknowns ($\Delta\ddot{r}$, $\Delta\dot{r}$ and Δr), so assumptions must be made and the solution is approximate.
- There are many step-by-step methods. PERFORM uses the Constant Average Acceleration (CAA) method.

58

Assumptions for CAA Method

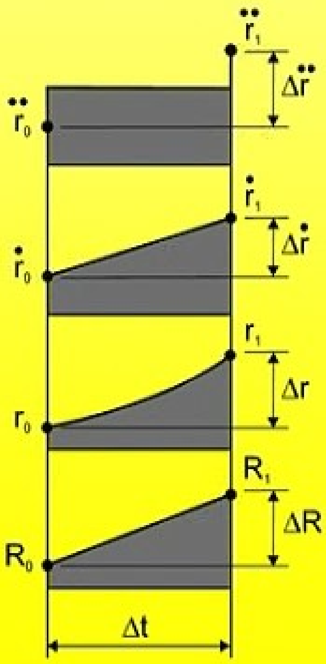


Satisfies kinematics, violates equilibrium

Satisfies equilibrium, violates kinematics

59

Equations for CAA Method



Equilibrium :

$$M\,\Delta\ddot{r} + C\,\Delta\dot{r} + K\,\Delta r = \Delta R$$

From kinematics :

$$\Delta\dot{r} = \frac{\Delta t}{2}(\ddot{r}_0 + \ddot{r}_1) \;=\; \frac{\Delta t}{2}(2\ddot{r}_0 + \Delta\ddot{r})$$

$$\Delta r = \frac{\Delta t}{2}(\dot{r}_0 + \dot{r}_1) \;=\; \frac{\Delta t}{2}(2\dot{r}_0 + \Delta\dot{r})$$

Hence get effective stiffness and load :

$$K_{eff} = \frac{4}{\Delta t^2}M + \frac{2}{\Delta t}C + K$$

$$\Delta R_{eff} = -M\ddot{u}_g + M(2\ddot{r}_0 + \frac{4}{\Delta t}\dot{r}_0) + 2C\dot{r}_0$$

Solve $K_{eff}\,\Delta r = \Delta R_{eff}$. Then :

$$\Delta\dot{r} = -\,2\dot{r}_0 + \frac{2}{\Delta t}\Delta r$$

$$\Delta\ddot{r} = -\,2\,\ddot{r}_0 + \frac{2}{\Delta t}\Delta\dot{r}$$

CSi

PERFORM Step-By-Step Procedure

- PERFORM-3D uses the CAA method.
- The time step, Δt, is the same for all steps.
- A variable time step does not seem to be needed, and is not implemented in PERFORM.
- In each time step the load increment is known, and the analysis uses load control. Displacement control is not suitable for dynamic step-by-step analysis.
- Within each step the event-to-event strategy is used (with overshoot).

CSi

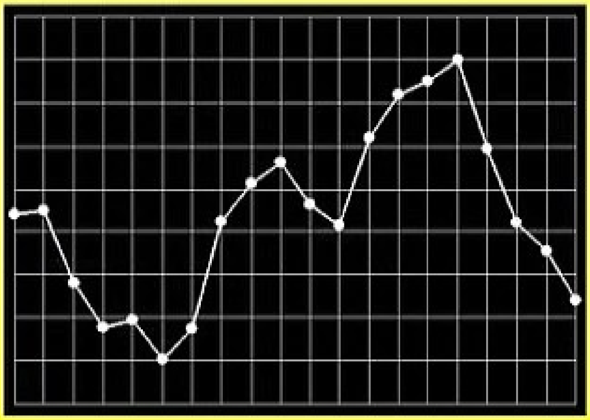
Choice of Time Step

- The time step must be short enough to do two things :
  (1) Capture the response of the structure.
  (2) Capture the earthquake ground motion.
- Structure response : the time step (Δt) should be no larger than about 1/10 of the shortest significant period (the period of the highest mode that contributes significantly to the response).
- If the first mode period is T_1, the period of the highest significant mode might be about $0.1T_1$. Hence Δt should be no larger than about $0.01T_1$. For example, if T_1 = 2 secs, Δt should be no larger than 0.02 secs.
- If the ground motion is discretized at, say 0.02 second intervals, do not use a time step larger than 0.02 seconds.
- It is wise to use a time step that is an exact subdivision of the discretization interval, to avoid missing ground motion peaks (e.g., in the above case, use 0.02, 0.01, 0.005, etc.).

CSi

62

Choice of Time Step (Contd.)

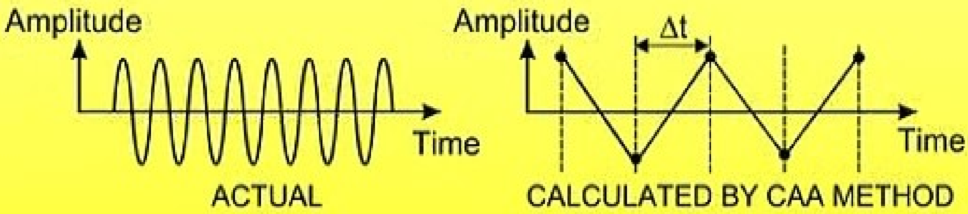


- The above is a sample of an actual record, discretized at 0.02 second intervals.
- A time step of 0.04 seconds misses some peaks.
- A time step that is not an exact subdivision of 0.02 (say 0.015) also misses some peaks.
- Be careful. You may need to experiment with the time step.

CSi

63

High Frequency Vibrations



- Short period (high frequency) vibrations will always be present.
- Indeed, a typical structure will have many deformation modes that have zero mass, and hence zero period.
- If the vibration period is smaller than the time step, the CAA method calculates oscillating results essentially as shown.
- Obviously this is not accurate. However, the amplitudes are small, so it does not matter that they are inaccurate.
- Short period vibrations are damped out if there is βK damping. This is usually desirable. These vibrations usually have little effect on the response, but they can cause numerical instability.

CSi

64

Please Keep In Mind

- Structural analysis is not an end in itself. It is a tool for use in design.
- There will always be uncertainties, in the properties of the structure and in the loads.
- The goal is not to get an "exact" simulation of behavior.
- The goal is to get D/C ratios that are accurate enough for decision making.

Questions?

CSi

66

Structural Models for GLD RC Framed Buildings

MS Thesis

Three major characteristics of GLD structures,

a) Irregularities

b) Poor proportioning of beams and Columns

c) Non-ductile reinforcement details.

Analytical Models to simulate behavior of isolated GLD RC members.
- FE Models
- Lumped plasticity Models.

Structural model is composed of NL element, the response of which can be exactly defined by closed form solutions of mathematical expressions after imposing an appropriate set of assumptions.

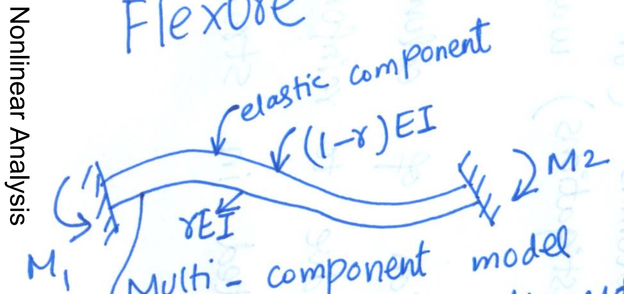
- Shear failure of Columns (typically found during post-earthquake investigations) was never reported in literature.
- Rigid body rotation of columns as a result of bar slippage due to reinforcement lap-splice was not incorporated in structural models.

Analytical Models

FRAME Members

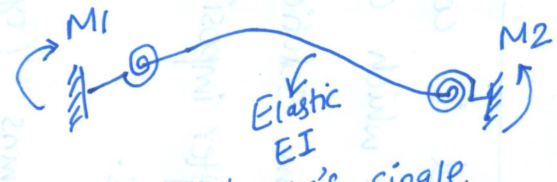
Beam-Column Joints.

Flexure

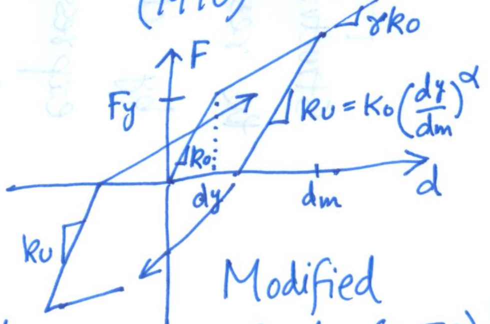


Multi-component model with bilinear hysteretic $M\phi$ (Clough 1965)

a rotational spring at each end of this elasto-plastic component. These springs are fully rigid before yielding and rotate freely after yielding. In b/w springs, elastic portion with γEI. After yielding of EPP component, flexural K of whole member = of elastic component which represents strain hardening.



Giberson's single component model with bilinear hysteresis (1970)



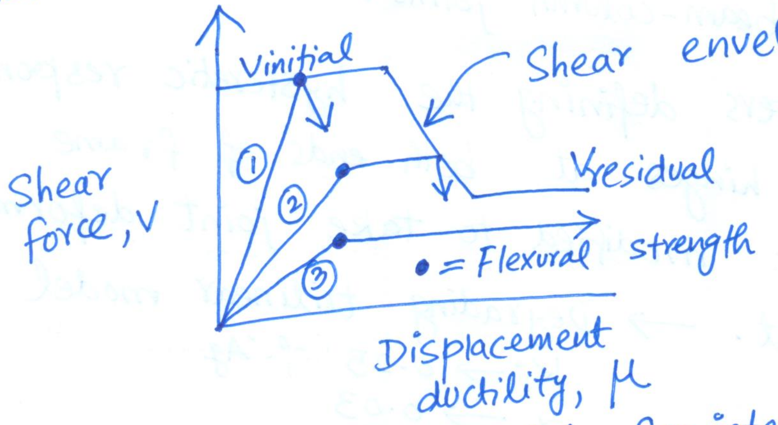
Modified Takeda (1970)

Shear

Axial Load Collapse

Shear :-

In 1981 it was first recognized that Shear strength degrades as displacement ductility demand increases.



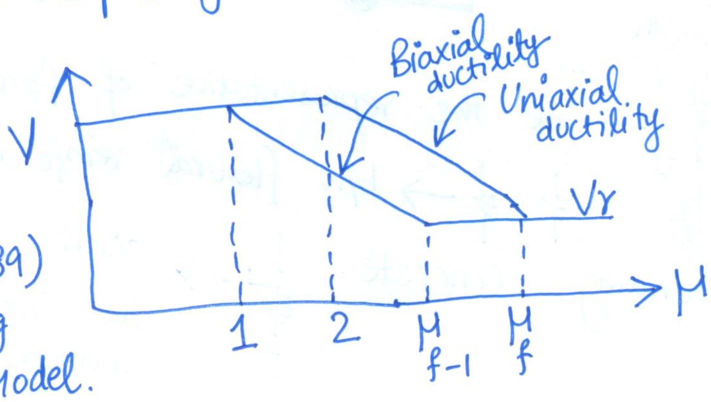
ATC-6 Model for interaction b/w Shear strength and flexural ductility. (1981)

Considering only this, 3 failure paths are possible.

③ → desired behavior of frame element.
Vresidual always kept higher than flexural Strength.

① and ② show premature failure due to shear in which either required flexural strength or displacement ductility cannot be developed.

So Shear Capacity model is required.



Ang (1989) and Wong (1993) Model.

Analytical Models of beam-column joints :-

Anderson and Townsend (1977) proposed a method of incorporating both bar-slip and shear-deformation occuring in beam-column joints.

The parameters defining the hysteretic response of plastic hinges at both ends of frame members were modified to take joint deformations into account. → Degrading trilinear model

$K_i \rightarrow 0.25$ of A_g

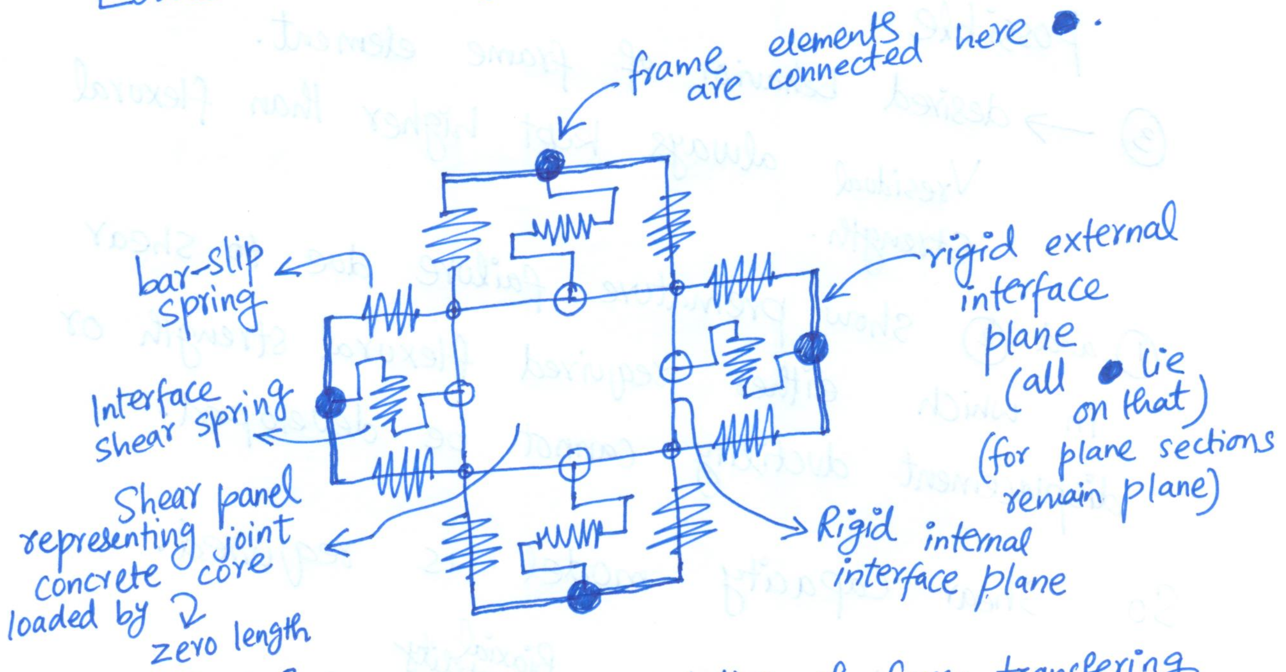
$\alpha \rightarrow 0.03$

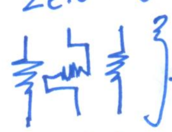
$K_{unloading} = K_{loading}$

$K_{reloading} = \sqrt{1/U_c} \times K_i$

max curvature ductility attained.

Lowes et. al. (2002) → Macro-model




→ are the representative of force transfering ability.

→ b/w flexural reinforcement and surrounding concrete.

→ Shear stransfering ~~b/w~~ at the interface b/w frame members and joint.

The slip of a bar as a function of bar force can be found by integrating unit axial deformation of bar along its length. So using experimental data, the constitutive models for axial springs can be defined.

For Shear panel, modified compression field theory (MCFT) was used to determine backbone curve. MCFT is a theory used to construct stress-strain relationship of ~~an~~ RC panel subjected to in-plane shear and axial force, proposed by Vecchio and Collins (1986).