

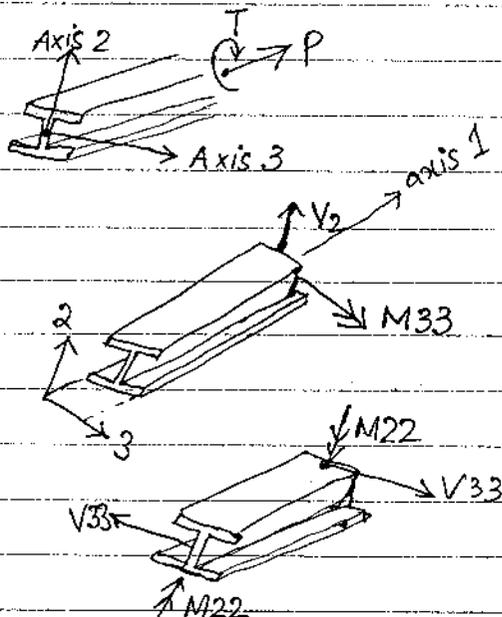
SAP 2000 CSI watch and Learn

→ General:

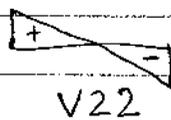
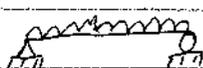
- No "draw columns using clicks" button
- Select nodes → Assign → Releases/Partial fixity → and check start and end boxes for Moment M33 in Release.] → To ^{make} pin joints like Truss joints.
- To replicate, Select object → Edit → Replicate and enter dx, dy or dz and number.
- Select frame element, Assign → Frame → Automatic frame mesh. Intersection with other frames.
- Select → Coordinate Specification → Specified Coordinate Range. To select very specific Object.
- Display → Show Member Forces → Frames

Components

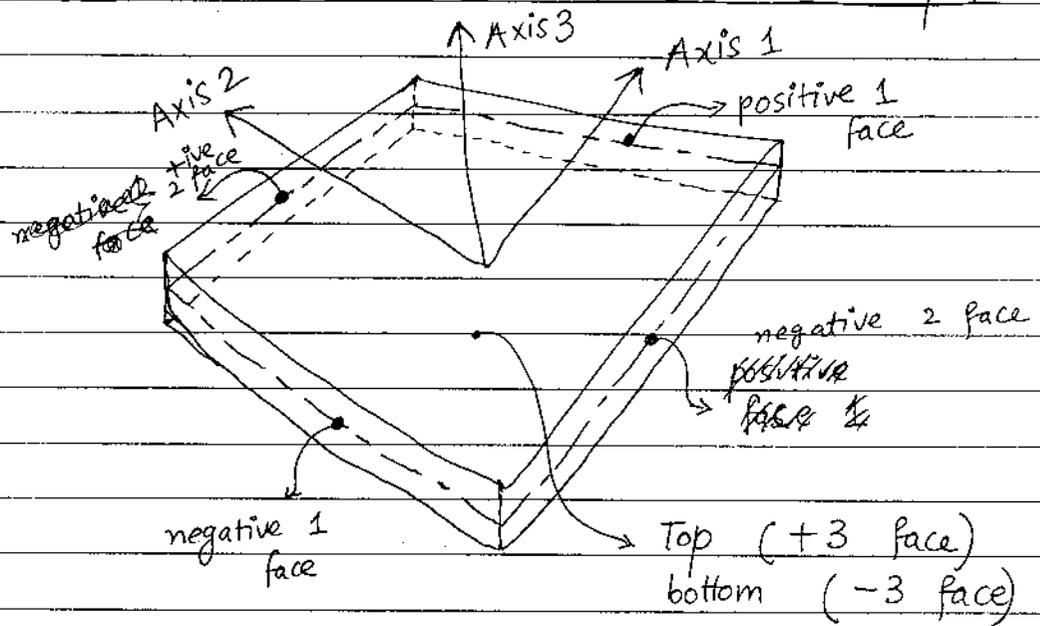
- | | |
|---------------|-----------|
| ○ Axial Force | ○ Torsion |
| ○ V22 | ○ M22 |
| ○ V33 | ○ M33 |



50



Shell Internal Force and Stress Output



Stresses :-

- In-plane direct stresses : S_{11} and S_{22}
- In-plane shear stress : S_{12}
- Transverse shear stresses : S_{13} , S_{23}
- Transverse direct stress : S_{33} (always assumed to be zero)

These shell element stresses are the forces per unit area that act "within" the volume of the element to resist the loading.

The Three in-plane stresses are assumed to be constant or to vary linearly through the element thickness.

The two transverse shear stresses are assumed to be constant through the thickness. The actual shear stress distribution is parabolic, being zero at the top and bottom surfaces and taking a max or min value at the mid surface of the element.

Stress Resultants (Shell element internal forces):

These are the forces and moments that

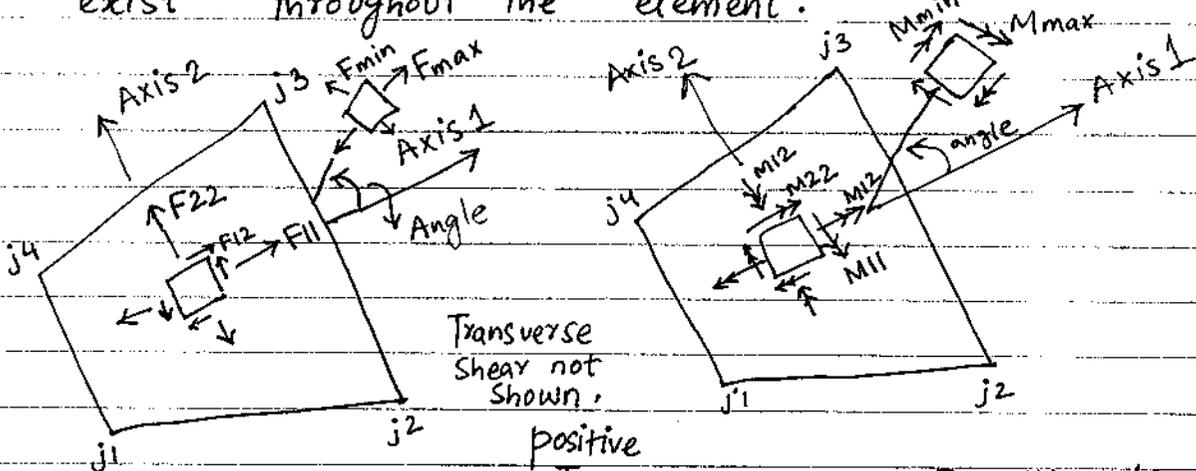
result from integrating the stresses over the element thickness.

- Membrane direct forces F_{11} and F_{22}
- Membrane Shear force F_{12}
- Plate bending moments M_{11} and M_{22}
- Plate twisting moment M_{12}
- Plate transverse shear forces V_{13} , V_{23}

These are forces and moments per unit of in-plane length. They are present on every point on the mid surface of the element.

Positive internal forces correspond to a state of positive stress that is constant through the thickness. Positive internal moments correspond to a state of stress that varies linearly throughout the thickness and is tensile at bottom.

The stresses and internal forces are evaluated at the standard 2×2 Gauss integration points of the element and extrapolated to the joints. Although they are reported at the joints, the stresses and internal forces exist throughout the element.

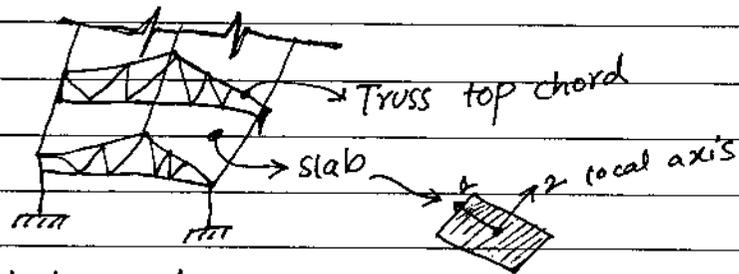


Stresses and membrane forces. S_{ij} has same direction as F_{ij} .

positive
Transverse Shear forces and stresses acting on positive faces point toward the viewer.

Forces/Moments are per unit of in-plane length.

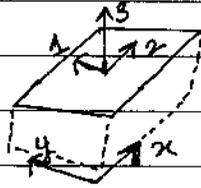
→ General (Continued) :-



You don't want area objects (slab) to act as flanges for the top chord of Trusses.

Select all slabs → Assign → Area → Area Stiffness Modified
Set membrane f_{22} Modifier as "0". This prohibits in-plane axial loads.

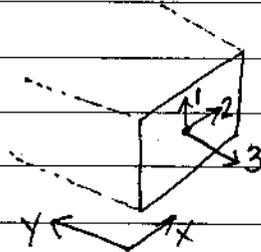
- For horizontal area, local axis 2 is parallel to the global x-axis.



red = 2 (|| to x axis)
white = 1 (|| to y axis)

Blue (cyan) = Normal to Surface of area, 3 axis.

- For vertical area,



→ Mass Modal Analysis :-

The masses used in program are typically available from

- Object material properties.

- Applied vertical loads.

- Assigned Object Masses.

a) Materials :- User give wt/vol → program converts it to mass/vol and \times by vol. The mass is then lumped at joints.

b) One of the unique features of SAP 2000 is its ability to convert mass from applied loads. Only loads in the negative z direction may be used to calculate mass. If that option

[A load of 32.2 lb in \downarrow dir converts to a mass of $1 \frac{\text{lb s}^2}{\text{in}}$ in the X, Y and Z directions]

is selected, the load will be divided by g and applied as mass in x, y and z direction.

c) To add mass, ^{select} joints \rightarrow Assign \rightarrow joint \rightarrow Masses
Coordinate \rightarrow Global, Global X \rightarrow enter value.

- Define \rightarrow Mass Source (To specify, where the program should find the mass)
 - from element and additional masses
 - from loads \rightarrow then user have to give multiplier.
 - from both
- Default load case \rightarrow Modal
modify it \rightarrow Eigen, no of modes. etc.
- Run analysis \rightarrow 1st mode will be displayed.
- Display \rightarrow show tables \rightarrow Joint output
on table list, Assembled joint masses,
if you assigned some mass to your joint
in any direction, you can check that here.
- Display \rightarrow show tables \rightarrow Structure output
from list \rightarrow Modal load participation ratios.

\rightarrow P-Delta effects in SAP :- $P\Delta$ is a non-linear effect.

Typically two options for $P\Delta$.

a) Run/set each load combination as a static nonlinear load case with $P\Delta$ included.

This approach is accurate but limited to static analysis.

There is no option of including $P\Delta$ in Modal, RS and Time history load case types.

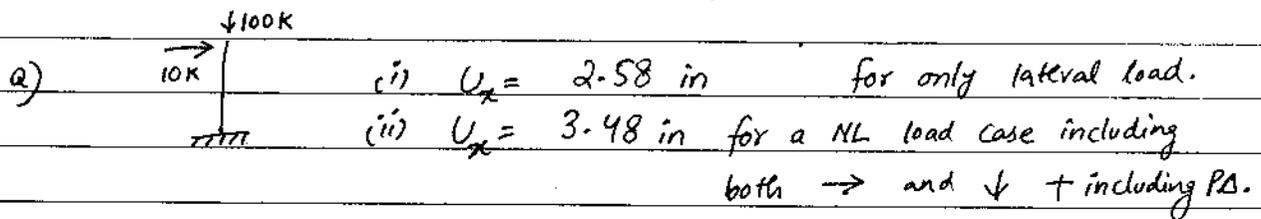
b) Set up a single "Initial" $P\Delta$ load case (with only dead load) + Make it static nonlinear + include $P\Delta$.

For other load cases, select "Stiffness at end of NL case" and select this initial $P\Delta$ case.

so you are using modified stiffness for other analyses.

Benefits :-

- It requires only one nonlinear analysis
- It can be used with dynamics. (modal, NLRHA, RS)



b) Add a load case named \rightarrow "Initial PD" Static + NL + Include PD
(only dead load, multiplier 1)

Add a 2nd load case \rightarrow Static, linear, Select "Stiffness at end of NL case \rightarrow select "Initial PD".
(add both \rightarrow and \downarrow in this case).

$$U_x = 3.48 \quad \text{for (ii)}$$

$$U_x = 3.48 \quad \text{for 2nd load case in (b).}$$

The default modal analysis case uses "Zero Initial Conditions". If you change (or add a new load case) to "Stiffness at end of NL case \rightarrow "Initial PD", the time periods will be lengthened due to compression softening.

\rightarrow Interactive database Editing :-

Edit \rightarrow Interactive database editing

▣ Property definitions

▣ Connectivity data.

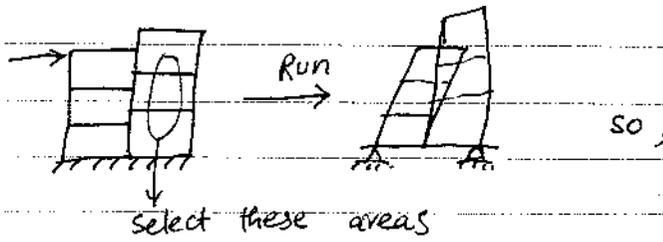
In table select "Material properties" etc. \rightarrow Can edit.

Click "To Excel" \rightarrow Change \rightarrow "From Excel" \rightarrow \downarrow "Apply to model"

▣ Joint Assignments \rightarrow joint loads \rightarrow Change it.

\rightarrow Area Edge Constraints :- Sap constraints the edges of area objects so that you don't need to create

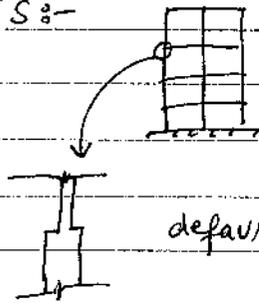
a transition mesh.



Select these areas

↓
Assign → Area → Generate edge constraints
→ Create constraints.

→ Cardinal points :-



default Cardinal point is "Centroid"

Select frame elements → Assign → frame → Insertion point.
to change Cardinal point.

→ Response Spectrum Analysis :-

- In "Modal" load case → Select no of modes required.
- Define → Functions → RS → ^{select.} Code

Give Parameters → eg in case of IBC 2009, select from
Where program should pick S_s and S_1 values.
Site class etc etc.

- Acceleration is in "g" units.
- Add a new load case → RS as type.
- Select Combination for both modes as well as direction.

Use modes from "Modal" case.

- Load type → Accel, U1 direction, fn, Scale
- Scale to "g" units to "inch" units eg.
Double click + Shift → Calculator.

$$32.2 \times 12 = 386.4$$

because eg $1g \rightarrow 1 \times 32.2 \text{ ft} \rightarrow 1 \times 32.2 \times 12 \frac{\text{in}}{\text{sec}^2}$

- For Damping → 3 options.
↳ Constant Interpolated
↳ Mass and K proportional.

→ In Results, all U_1 deformation values are positive because "SRSS".

→ Modal Time History Analysis :-

Modal THA can be more computationally efficient than direct integration.

- Define → functions → Time History → ^{give} Ground motion parameters.
- Define → load case → Time History
 - Linear
 - Modal

Accel → U_1 → function → 386.4

↳ (32.2x12)

No of Output time steps ✓

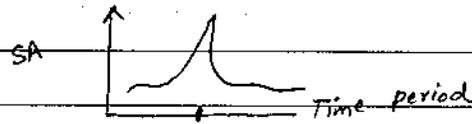
Output time step size ✓

- Display → forces → at a certain time step
- Display → deformed shape → at a certain time step.
- File → Create video → Create multistep Animation

☑ Show graph.

- Select a point on undeformed shape.

↳ Display → Show RS Curves → check all tabs.



↳ period of forcing fn (sine in this case)

- Select any joint, Display → Show plot functions
Define plot function, Vertical functions, Hor. fns
"Save Named Set", "Display"

→ Non-Linear GAP Element :-

→ The link/support Element :-

link → connect two joints

Support → Connect one joint to ground → A one-joint grounded spring

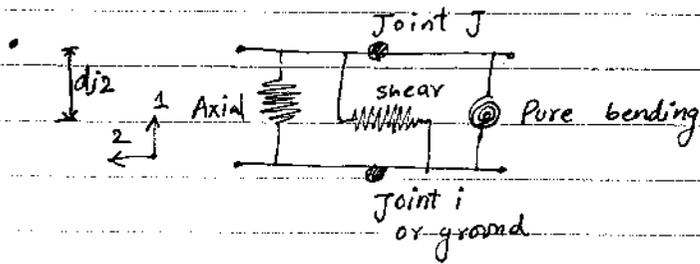
- Three types of behavior
 - Linear
 - NL

frequency dependent. (optional) K, c → for all free dependent analysis.

each element → Six separate Springs ^{each} and dashpots ^{Several Springs} [can be coupled]

- Viscoelastic Damping
- Hook (Tension only)
- Uniaxial Plasticity
- Gap (Compression Only)
- Multi-linear Uniaxial Elasticity
- " " " Plasticity with Several hysteretic behavior. Kinematic Takeda and Pivot.

- Biaxial plasticity base isolator
- Friction pendulum base isolator, with or without uplift prevention. This can also be used for modeling gap-friction contact behavior.



[Three of six independent springs hinges in a link/support element]

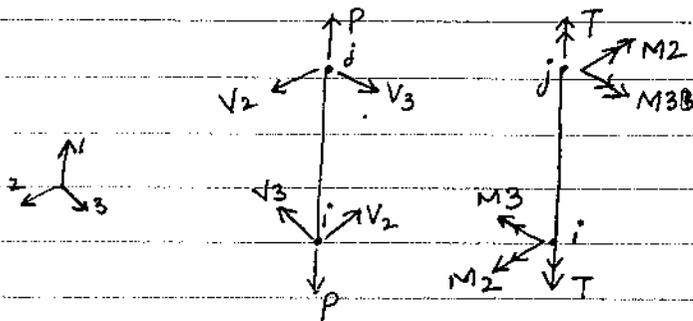
• Six FD relationships:-

- Axial f_{u1} vs d_{u1}
- Shear f_{u2} vs d_{u2} , f_{u3} vs d_{u3}
- Torsional f_{r1} vs d_{r1}
- Pure bending f_{r2} vs d_{r2} , f_{r3} vs d_{r3}

$f_{u1}, f_{u2}, f_{u3} \rightarrow$ internal spring forces

$f_{r1}, f_{r2}, f_{r3} \rightarrow$ Internal spring Moments.

Each relationships may be $\begin{cases} \text{Zero} \\ \text{Linear only} \\ \text{Linear/NL} \end{cases}$
 Independent or coupled related to the def rates (Velocities) as well as to the deformations.



Axial = $f_{u1} = : P = f_{u1}$

Shear in 1-2 plane: $v_2 = f_{u2}$, $M_{3s} = (d - d_{j2}) f_{u2}$

Shear in 1-3 plane: $v_3 = f_{u3}$, $M_{2s} = (d - d_{j3}) f_{u3}$

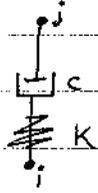
Torsion: $T = f_{r1}$

Pure bending in 1-3 plane: $M_{2b} = f_{r2}$

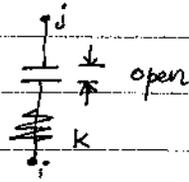
" " " 1-2 plane: $M_{3b} = f_{r3}$

$d =$ distance from joint J.

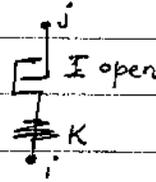
$M_2 = M_{2s} + M_{2b}$, $M_3 = M_{3s} + M_{3b}$



NL Viscous Damper



Gap



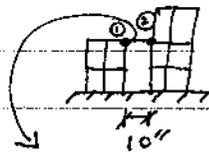
Hook

GAP :-

"open" is the initial gap opening. (0 or positive)

NL FD is given by,

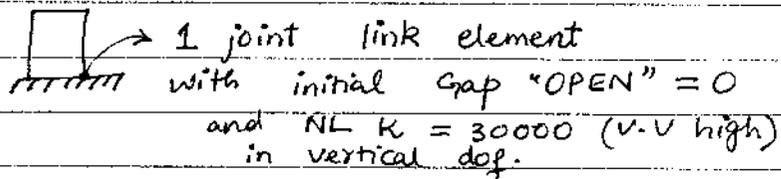
$$f = \begin{cases} K(d + \text{open}) & \text{if } d + \text{open} < 0 \\ 0 & \text{otherwise} \end{cases}$$



Gap link with "open" = 10 inch, NL $K = 1000$

[If ^{total internal} deformation + OPEN GAP] < 0 means nodes ① and ② come closer to each other more than initial gap \rightarrow means two buildings collide \rightarrow
 \rightarrow Then stiffness will have some value eg 1000. otherwise $K = 0$.

It means whenever this gap link will be compressed more than 10 in, it will start acting as a member with given FD (or K)
 Whenever compression is < 10 in $\rightarrow K = 0$



So whenever this point uplifts, $(d + \text{open}) > 0$
 So $K = 0$ (means uplift allowed) but whenever this point want to sink into ground $\rightarrow (d + \text{open}) < 0$
 \rightarrow so $K = 30000$, means negligible ≈ 0 downward movement.

Linear eff. K and Linear eff $C = 0$

HOOK :- (Tension Only)

$$f = \begin{cases} k(d-open) & \text{if } d-open > 0 \\ 0 & \text{otherwise} \end{cases}$$

→ Nonlinear Shear Walls : (NL Shell element)

- Define → materials Show advanced properties.
↳ Models OE

- Define → Section → Area → Add new section.

⊙ Shell- Layered / Non-linear

"Modify / show layer definition"

S22 NonLinear → Vertical inplane behavior linear
but others → non linear.

Out-of-plane element ^{behavior} → Linear

- If you are modeling Only SW → but actual behavior you want is "as if it is connected to Rigid Diaphragms" → Select all joints except base
go to Assign → joint → Constraints
Select "Diaphragm" from list → Add new on
and Constraints axis is "Z Axis".

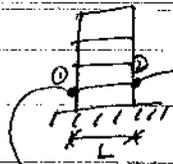
Assign a diff diaphragm constraint to each different selected Z level.

Go to define → Joint Constraints (can see constraints)

- If modeling only wall → Apply concentrated loads (using tributary area) and Define → mass source → from loads.
- Drift at roof :- Can be plotted during time history using Define → "Generalized Displacements"

After analysis, go to Define → Generalized Disps → Add New
give joint number and Scale factor to any specific dof (eg in roof drift case, give top node number and give its ux a scale factor of 1/H)

Go to Display → Show plot functions → Select time history load case, Define plot functions → select "Add generalized Displacements" select previously defined drift roof gen. disp
For strain in a member vs time → In gen. disp, give U3 a factor of 1/length of element.



+ add this with factor $-\frac{1}{L}$.

for rotation in first storey \rightarrow set ^{Scale} factor of U3 for this node as $1/L$.

$$\theta = \frac{\Delta_1 - \Delta_2}{L}$$

for determining M at bottom story, select story + Assign \rightarrow Assign to group.

Define \rightarrow section cuts \rightarrow Add new. Select this group.

Display \rightarrow show plot fns \rightarrow Add two functions now, One from gene. disps (θ) and one from section cut forces (M2).
Plot.

\rightarrow Non Linear Static Procedures :- (Pushover)

To assign auto hinges to frame objects.

- Select all beams \rightarrow Assign \rightarrow frame \rightarrow Hinges
Hinge property Relative distance

Hinges can be from FEMA ^{Auto} or CALTRANS ¹ Add
Select table, eg "Steel beams - flexure" etc.

$$\text{DOF} = \text{M3}$$

- Select all Columns \rightarrow same process but diff table.
- Go to Define \rightarrow Section properties \rightarrow Hinge properties.
 Show generated props.

Modify any hinge property to review "Moment - rotation Curve".

- Select All beams and columns \rightarrow Assign \rightarrow Frame \rightarrow
 \rightarrow Hinge Overwrites.

Auto Subdivide line objects At hinges
(Discretize to give better results).

- Make Dead load case as Nonlinear and use this NL case as starting point of

"Pushover" case.

- Define → Load Case, Static, NL, continue from NL Dead
Load type → Accel, UX, -1 scale factor (results in
disp in +X direction).
- Load Application → Displacement controlled → Use monitored
Displacement 1 foot at U1 dof at Joint 4 (eg).
- Results saved @ Multiple stages. (control node)
- Run → Deformed Shape → see PHS and their location on
MO graph.
- Display → Pushover curve.

Select ATC 40 Cap demand curve blue > Performance
Spectrum: Capacity Curve Green > point.

- Check period of performance point → T_{eff} .

- File → Display Tables. Select Pushover curve D/C ATC 40
see at which time/^{push} step, this T_{eff} occurs.

go to display deformed shape and Toggle up to
that time step → check the largest hinge at that
step. If within limits → OK.

- Display → Show Hinge Results. Select hinge.
□ plot control parameters, select steps.