PERFORM 3D

CSI Watch and Learn

→ Two ways to start model

a) Importing from SAP
- Geometry
- Materials
- Sections
- Loads

In SAP,

file → Export → PERFORM 3D Structure

and open in PERFORM.

b) Directly in PERFORM 3D.

→ Two phases,

a) Modeling
- Overall information
- Nodes
- components
- Elements
- Frames
- Load patterns
- Import/Export
- Drifts / Δs
- Structure sections
- Limit states
- Delete elements for gravity loads.

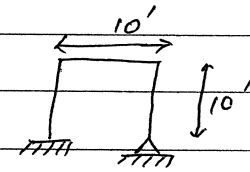
b) Analysis
- load case
- Analysis Series Run
- Mode Shapes
- Energy balance
- LS groups
- Deflected shaps
- Time history
- hysteresis loop
- M and V diagrams
- General Pushover
- S_T pushover
- Usage Ratio plots
- Combinations and Envelopes

→ Nodes
- Nodes
- Supports
- Masses
- Slaving

→ Components
- Cross Sections
- Inelastic
- Elastic
- Materials
- Strength Sects
- Compound (various components are assembled)

PERFORM 3D CSI Watch and Learn

Starting a new structure



→ Nodes → Grid tab

Select orientation = Vertical

Start coordinates = 0, 0, 0

Bays = 1

Stories = 1

Bay width = 10'

Story height = 10'

Supports tab :- Select lower left node

All ✓ fixed

Right node → allow rotations

Masses tab :- Define a new "mass pattern"

eg "nodal mass" $\left(\frac{50\text{lb}}{\text{ft}} \times 10\text{ft}\right)$ (tributary length)

$= 500\text{lb}$

Using weight units ⊙ Weight units.

H1	500
H2	Same as H1
V	500

→ Components

Cross sections tab : → Beam type → Beam, Standard steel section

Section type → New.

Section ID

Dimensions and Stiffness , Inelastic Strength , Elastic

↓ Automatically populated

↓ Ultimate Strengths are specified while developing "inelastic beams"

↓ for strength limit states.

Similarly → Column type

Compound → New → "Frame member compound component"

Basic Components | Strength Sections | Self Wt

↓ type, Component name, Length Value = 1, Add

↓ for Limit States

↓ enter 50 lb/ft

Make two compound → "Beam", "Column"

→ Elements → here we assign compound components to model.

New, Group, type = Beam (Data) and Columns, OK

Add elements.

Current group = Beams ⊙ Straight line

Select two nodes → Test → OK

Current group = Columns ⊙ Grid

Select bottom two joints + then top 2 joints

test → OK

Orientations Tab :- current group = beam

Select beam → ⊙ Vertical up

Group = Columns → ⊙ +H1 option.

Properties Tab :-

Current group = Column, Select columns

"Assign Component" → select component type etc

Click "assign". do it for beams.

→ Load Pattern :-

New → "Quake"

Nodal loads, select node, enter load

"Element loads" → New "Live"

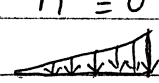
Chose element loads group group "Beam"

⊙ Define a new subgroup

Done

"Add loads" → Load type

$F_1 = 0$ $F_2 = 10$ $L_1/L = 0$ $\frac{L_2}{L} = 1$



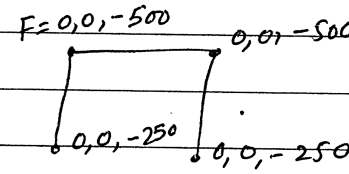
OK

Self weight tab → "New" "Dead"

Element group = Beam , Select beam → OK

" " " Column , " Columns → OK

Plot loads



ANALYSIS PHASE :-

Load Cases → type = Gravity , → New "Live load" OK

Analysis method = Linear

Load type = Element. → "Add" → "Save"

Run Analysis → "Check Structure"

Name = Live load eg

Scale factor for mass pattern = 1 , No of modes = 5

OK.

Add "live load" to list of "Analysis list"

"GO"

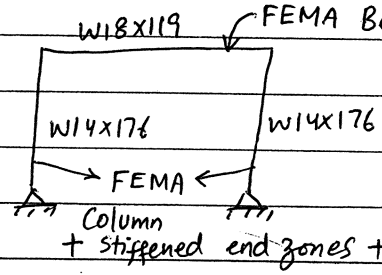
(All other tasks deal with

Modal Analysis → mode no, chose plot, Animate

Moment and Shear diagrams → Single Element, group

result type → Plot

Compound Components for FEMA steel frames



→ Cross Sections → define W18x119 beam → go to "Inelastic Strength" Tab Select ⊙Yes for

bending (because our beam will have flexural hinges) Set f_y = 36 ksi

Shear ⊙NO (we dont want shear yielding)

In "Elastic strength" Tab, → leave blank.

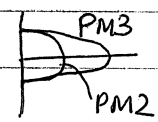
Define a new column W14x176 → Go to "inelastic strength" Same Under "Axial/Bending"

Click "Yes"

f_y Tension = 36 ksi BM about 2 = 36 ksi

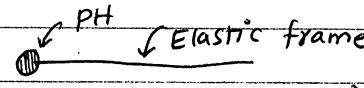
Comp = 36 ksi BM " 3 = " "

"Plot" → PM_2, PM3



leave "Shear", "Torsion" blank

→ "Inelastic" Tab :-



(FEMA Steel beam)

Select Type = FEMA Beam Steel type "New"

Select "Section and Dimensions" tab

Use Cross Section = Yes + Select our defined beam cross section

Select "Basic FD relationship" Tab

Shape of relationship = EPP

+ive Rotation AT X = 15 → means DX = 15DY (for columns enter 15 for all 4 cases)

NO -ive → because of Symmetry

"Deformation Capacities" Tab

" " = Yes

enter D/DY

2 => IO

6 => LS

8 => CP

Strength loss = NO "Tab remains blank"

Cyclic Degradation = " " " "

Check → Save

Repeat process for columns:

- enter 15 $\frac{DX}{DY}$ for ~~def~~ capacity basic FD relationships.
- In "def Capacities"

$\frac{2}{6}$ $\frac{2}{8}$ [1] axis $\frac{2}{3}$ ratio

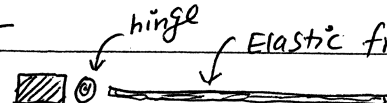
- No strength loss, No L/L bounds, No Cyclic deg.
- Check → Save

→ "Elastic" Tab → we will define two elastic components.

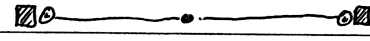
a) Select "End zone for a beam or Column"
↳ Used to increase member stiffnesses where beams and columns overlap.
Enter multiplier of 10 for stiffness

b) Select "Linear P/V/M Hinge or Release" for "Moment releases at column bases".
Releases
☑ M about 3 (1-2 plane) [0.001]
↓ small value for numerical stability.

→ "Compound" Tab :-



Stiffened End zone (FEMA basic Component)

A FEMA "Compound" component must contain two FEMA "basic" components connected at the point of inflexion 
we will assume this point of inflection to be midspan of beam and a bottom of pinned Columns.

Select "Frame member compound Component"
- Select "End zone for a beam or Column"
Select Component name

Select "Length type" as "Based on adjacent beam or column size" → Add.

- Select Again "FEMA Beam, steel type".
  Select component name.
  Length type = "Proportion of unassigned length"
  Length value = 0.5 (for inflection at mid point)
  Add
- Same "Add" for 2nd half.
- Select again "End zone", length type = Auto
  Check → Save.

Click again "New" for column compound Component. (Working from bottom up)

- 1st Add "Linear P/V/M hinge or release" with zero length at bottom of column.
- Then "FEMA Column Component"
  Since inflection is at bottom, so
  Length type = "Proportion of unassigned length"
  with length value = 0.02
  ↓ v. small value.
- Add 2nd fema component with length = 0.98
  ↓ nearly the entire length.
- Add Stiffened end zone at top of column.

Check → Save