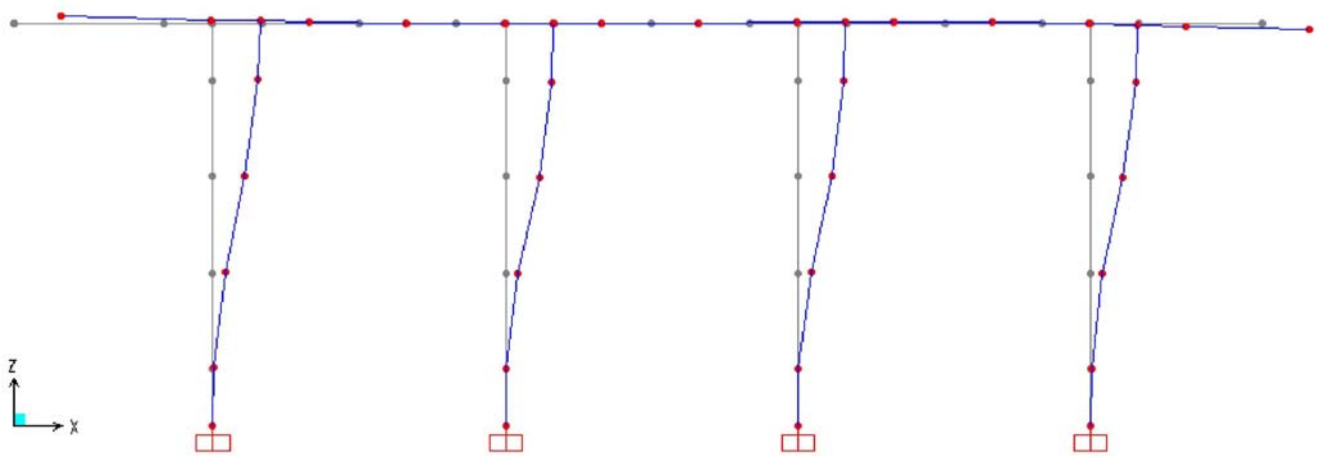
EARTHQUAKE ANALYSIS

with

SAP2000

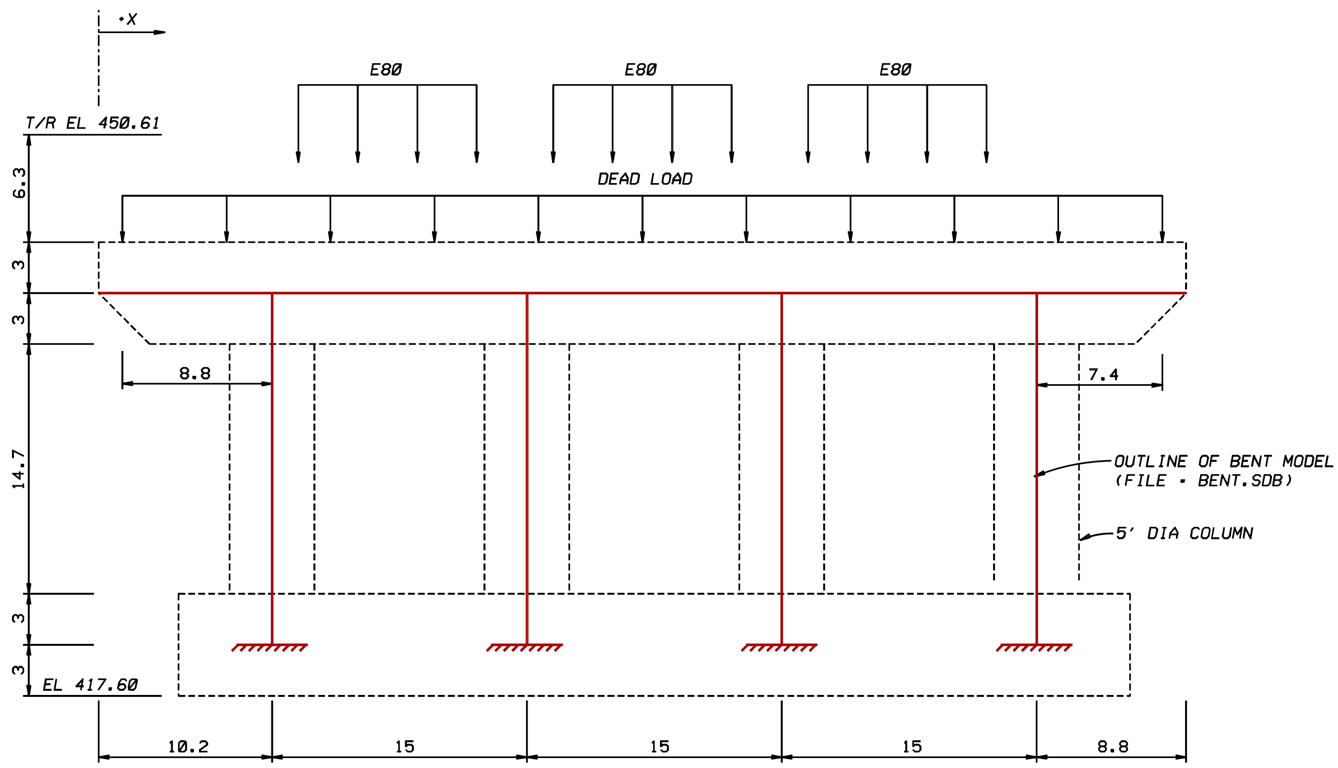


Prepared by

Bob Matthews

EARTHQUAKE ANALYSIS EXAMPLE

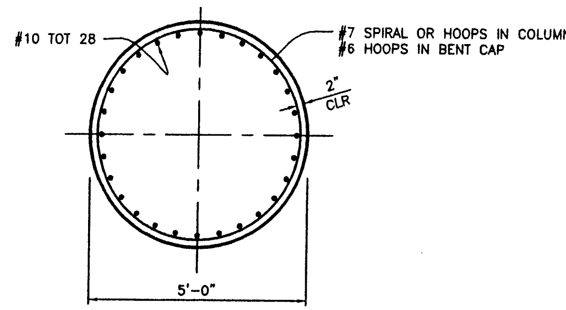
The earthquake analysis capabilities of SAP2000 are demonstrated using a railroad bridge bent designed in accordance with the AREMA Manual for Railway Engineering.



Prepare finite element model

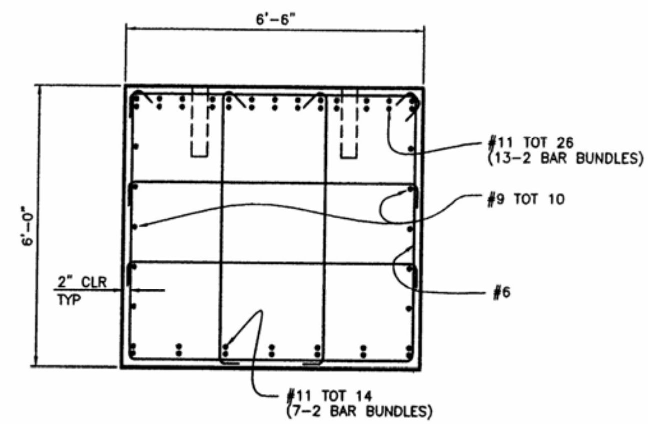
- Column properties:

    5’ diameter
    Assume $I_e = 0.35\ I_g$
    Use $I_e = 10\ I_g$ for stiffness in joint areas



- Bent cap properties:

    6.5’ wide x 6’ deep
    Assume $I_e = 0.75\ I_g$
    Use $I_e = 10\ I_g$ for stiffness in joint areas



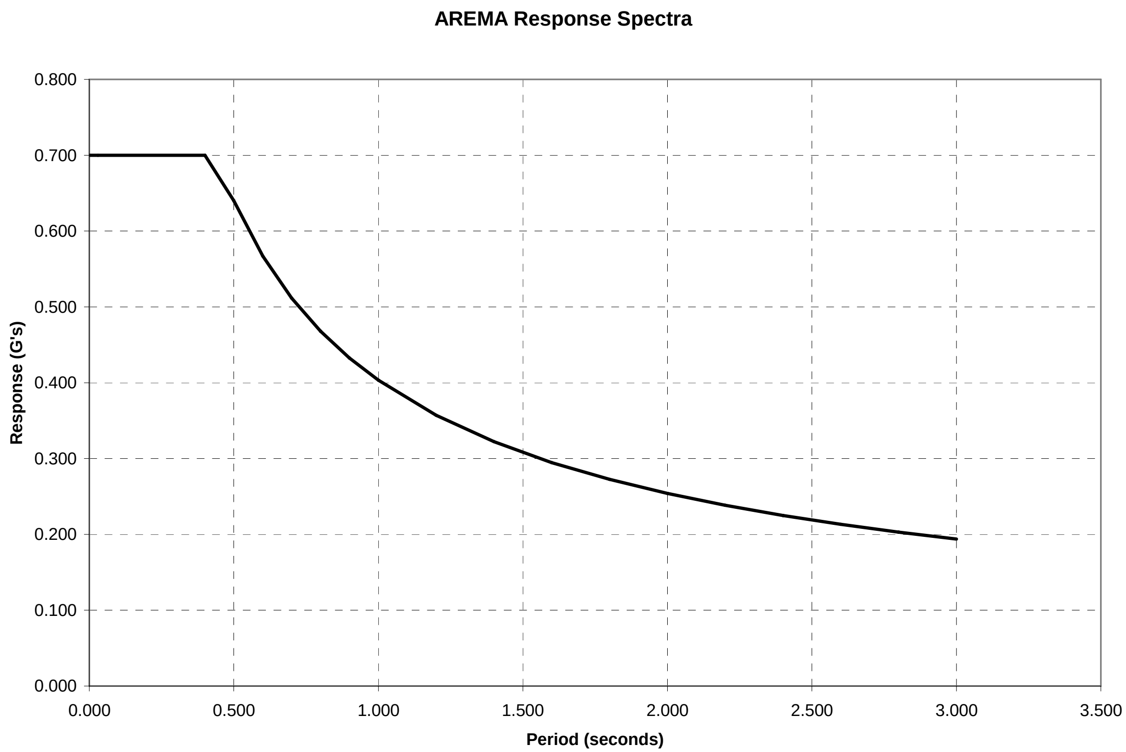
- Dead loads:

    Superstructure load = 2418 kips/61.2 ft = 39.5 kips/ft

Response spectrum analysis

Response spectrum analysis is performed to verify that the column design is adequate.

Period (sec)	Response (G's)
0.001	0.700
0.030	0.700
0.087	0.700
0.100	0.700
0.200	0.700
0.300	0.700
0.400	0.700
0.500	0.640
0.600	0.567
0.700	0.511
0.800	0.468
0.900	0.433
1.000	0.403
1.200	0.357
1.400	0.322
1.600	0.295
1.800	0.272
2.000	0.254
2.200	0.238
2.400	0.225
2.600	0.213
2.800	0.203
3.000	0.194



- Input steps

1. Define mass

   a. Select menu item: *Define > Mass Source*
   b. Select mass definition option: *From Loads*
   c. Add load case: *Dead (with multiplier of 1)*

   Note: The dead load is automatically divided by gravity.

2. Define Response Spectrum Function

   a. Select menu item: *Define > Functions > Response Spectrum*
   b. Choose function type: *Spectrum from File*
   c. Click: *Add New Function*
   d. Type function name: AREMA
   e. Select option: *Period vs. value*
   f. Browse to and select “spectra.txt” file
   g. Click: *Display graph*

3. Define Modal Analysis Case

   a. Select menu item: *Define > Analysis Cases*
   b. Select case: *MODAL*
   c. Click: *Modify/Show Case*
   d. Enter maximum number of modes = 2

4. Define Response Spectra Analysis Case

   a. Select menu item: *Define > Analysis Cases*
   b. Click: *Add New Case*
   c. Type case name = AREMA
   d. Select analysis case type: *Response Spectrum*
   e. Select loads applied from AREMA function
   f. Enter Scale Factor = 32.2 (response values in G's)
   g. Click: *Add*

5. Define Load Combination

    a. Select menu item: *Define* > *Combinations*
    b. Click: *Add New Combo*
    c. Select DEAD and AREMA cases
    d. Click: *Add*

    Note: Response spectrum will automatically give +/- values

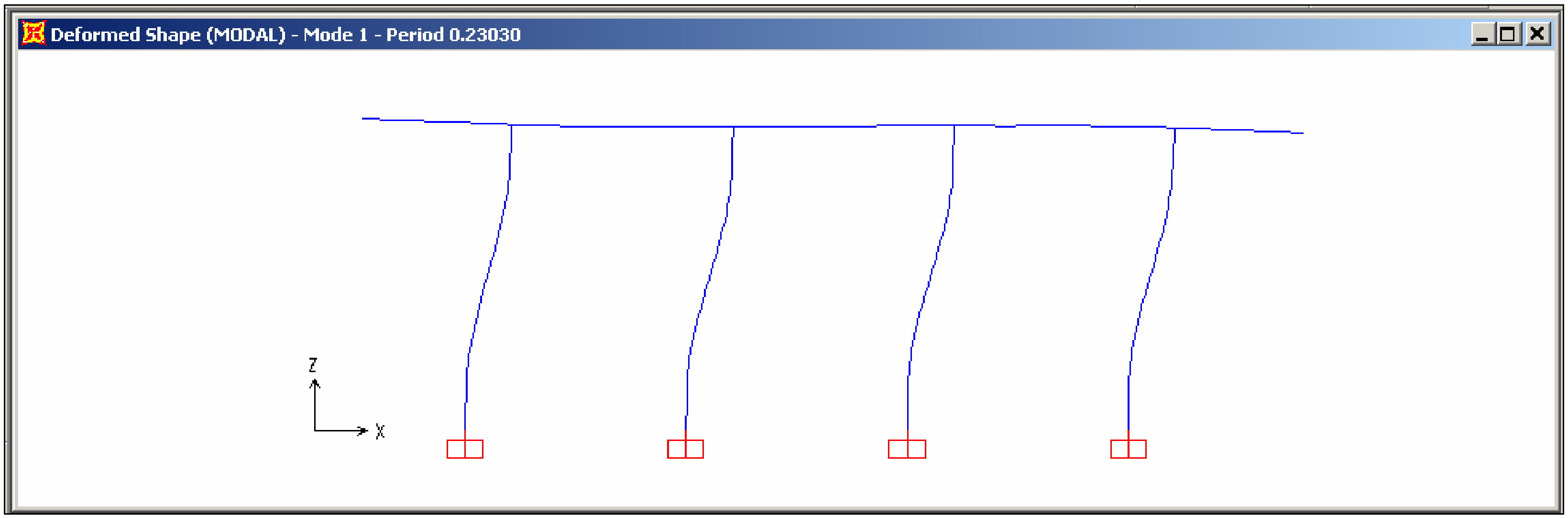
6. Perform Response Spectrum Analysis

    a. Select menu item: *Analyze* > *Run Analysis*
    b. Click: *Run Now*

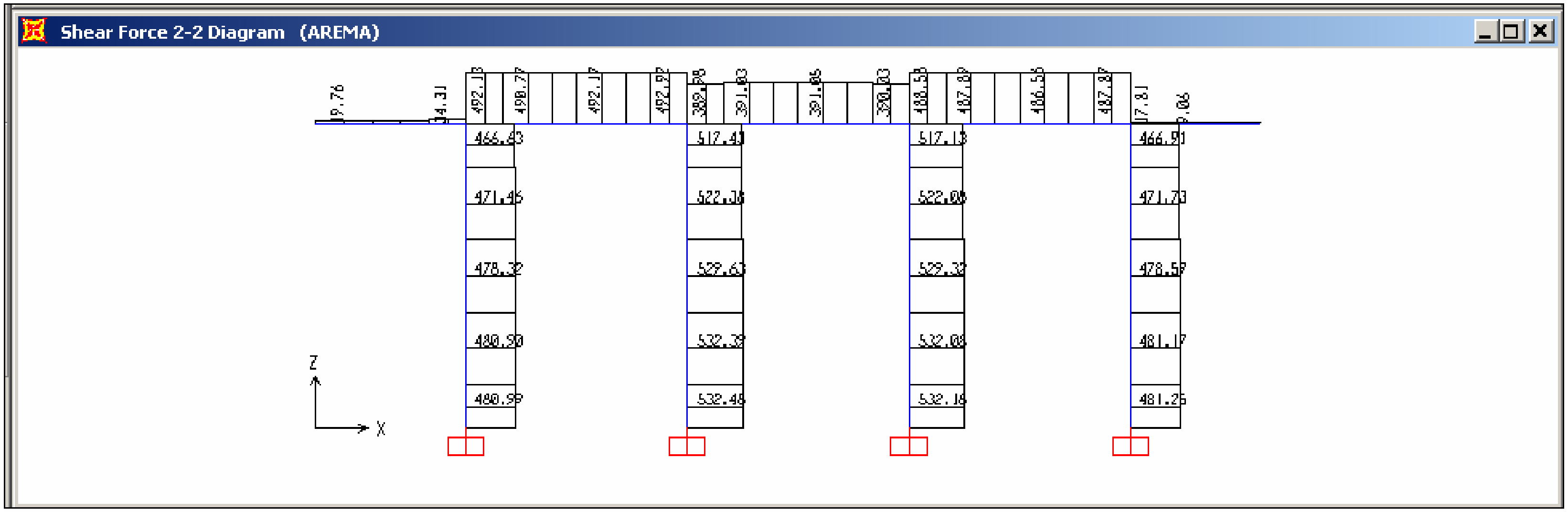
- Verify results

  1. Display results graphically

     a. Select menu item: *Display > Deformed Shape*



b. Select menu item: *Display > Show Forces/Stresses*



2. Display results in tabular form

   a. Select menu item: *Display > Show Tables*
   b. Select output to verify
   c. Select analysis cases to verify

Joint Reactions

File View Options Format

Units: As Noted — Joint Reactions

Joint Text	OutputCase Text	CaseType Text	StepType Text	U1 Kip	U2 Kip	U3 Kip	R1 Kip-ft	R2 Kip-ft	R3 Kip-ft
101	DEAD	LinStatic		-10.316	0	834.134	0	-81.1603	0
101	AREMA	LinRespSpec	Max	480.988	0	520.814	0	5099.7767	0
201	DEAD	LinStatic		2.817	0	684.529	0	27.9762	0
201	AREMA	LinRespSpec	Max	532.483	0	104.167	0	5526.7034	0
301	DEAD	LinStatic		2.111	0	705.054	0	21.9008	0
301	AREMA	LinRespSpec	Max	532.176	0	99.318	0	5523.5348	0
401	DEAD	LinStatic		5.388	0	741.227	0	49.0556	0
401	AREMA	LinRespSpec	Max	481.254	0	508.889	0	5100.1768	0

Record: 1 of 8 — Done

3. Verification computations

a. Verify: dead load reaction = total weight

$$Weight = 0.15\left(\frac{\pi(5)^2}{4}\times 14.7\times 4 + 6.5\times 6\times 64\right) + 2418 = 2965.6\ kips$$

$$Dead\ load\ reaction = 834.1 + 684.5 + 705.1 + 741.2 = 2964.9\ kips$$

b. Verify: total mass = total dead load / gravity

Mass = 92.1 (Export joint masses to Excel and sum)
Weight / Gravity = 2964.9 / 32.2 = 92.1

c. Verify: modal participating mass ratio > 90%

Ratio = 97% for mode 1

d. Verify: response spectra base shear = response x weight

Base shear = 481 + 532 + 532 + 481 = 2026 kips
Response for period = 0.247 sec = 0.7 G
0.7 x (2965 – 173 / 2) = 2015 kips (within 1%)

e. Verify: structure period $T = 2\pi\sqrt{\frac{W}{gK}}$

K = 40 kips / 0.0006 ft = 66667 k/ft (from unit load case)
T = 2 x 3.1416 x [2965 / (32.2 x 66667)]$^{1/2}$ = 0.23 sec
(within 6% of SAP2000 analysis)

- Check column design

1. Verify that column reinforcement is defined

   a. Select menu item: *Define > Frame Sections*
   b. Select property: *COL*
   c. Click: *Modify/Show Property*
   d. Click: *Concrete Reinforcement*

2. Analyze column

   a. Select menu item: *Options > Preferences > Concrete Frame Design*
   b. Select design code: *AASHTO*
   c. Select menu item: *Design > Concrete Frame Design > Select Design Combos*
   d. Remove all generated combos and add COMB1
   e. Select column members to design

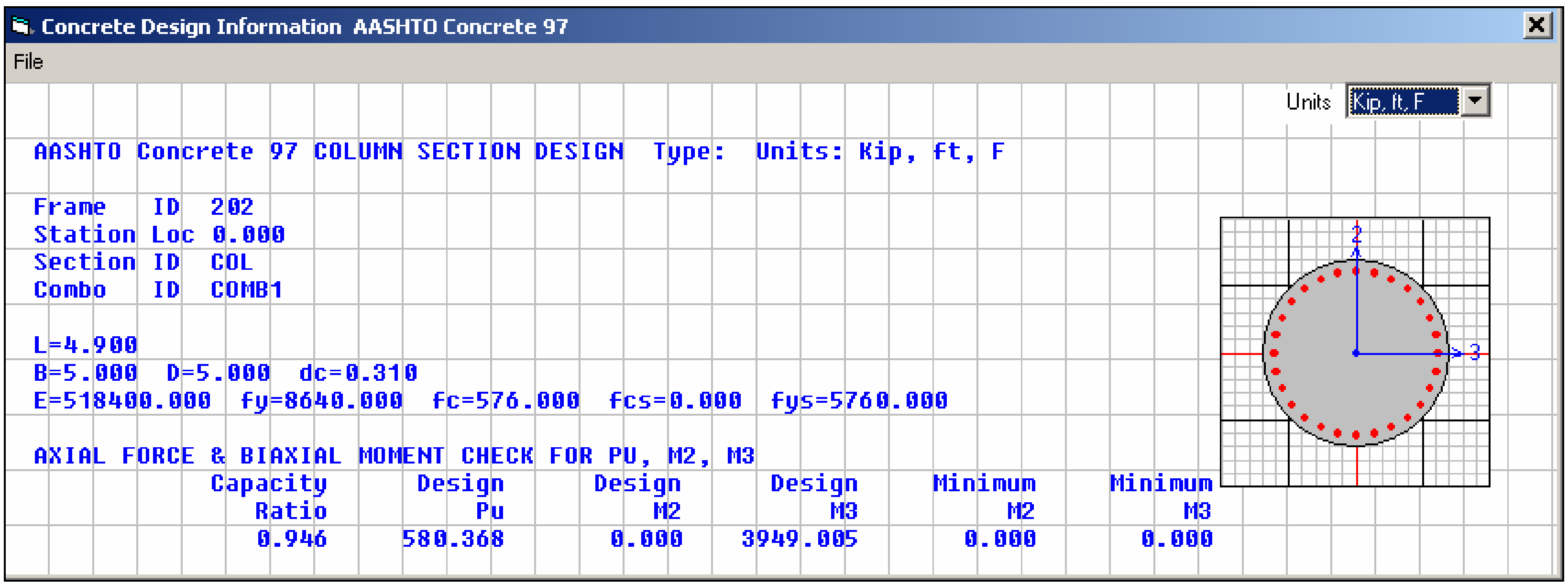
f. Select menu item: *Design > Concrete Frame Design > Start Design/Check of Structure*

g. Select menu item: *Design > Concrete Frame Design > Display Design Info*

h. Select: *Column P-M-M Interaction Ratio*

i. Right click on member with max ratio

j. Click: *Details*



EARTHQUAKE ANALYSIS EXAMPLE - PART 2

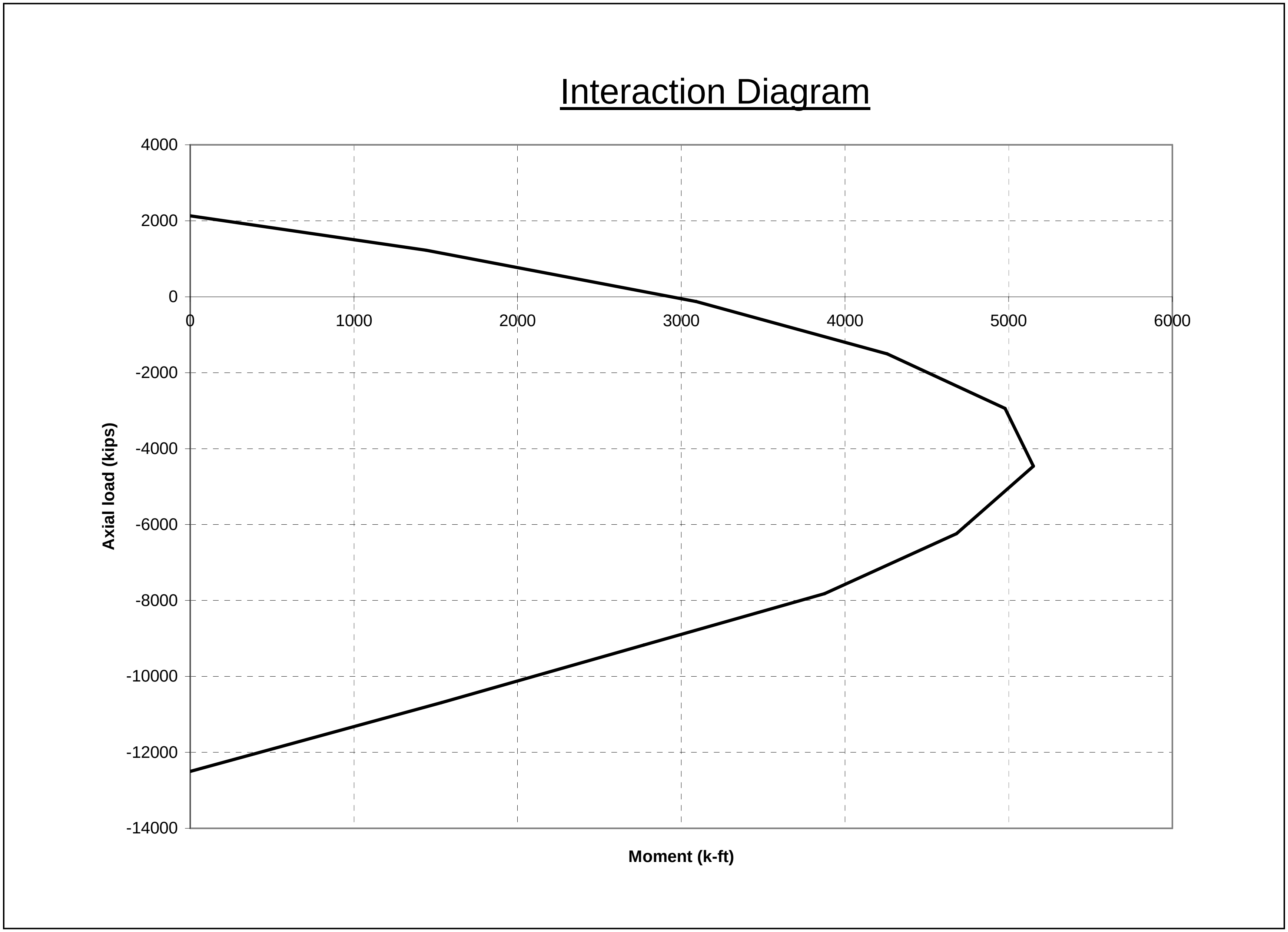
Nonlinear static analysis

Nonlinear static analysis is performed to verify that the bent cap and footing design is adequate.

- Develop plastic hinge properties

  1. The plastic hinge properties are based on 1.3 times the nominal moments from an axial-moment interaction diagram (Refer to AREMA Chapter 9 section 1.4.7.3.1 *Weak Column Provisions*). Similar requirements are contained in AASHTO 16th edition, Division IA Section 7.2.2 *Forces Resulting From Plastic Hinging In The Columns, Piers, Or Bents*.

  2. Define a 3D P-M-M interaction surface with at least 3 curves (0°, 45°, 90°) using available software. Hint: For a circular member, the 45 degree curve can be obtained by dividing the 0 degree moment values by the square root of 2.


Interaction Diagram
4000
2000
0
-2000
-4000
-6000
-8000
-10000
-12000
-14000
0
1000
2000
3000
4000
5000
6000
Axial load (kips)
Moment (k-ft)

Interaction Values

Curve 1	0. deg		Curve 2	45. deg		Curve 3	90. deg	
P	M3	M2	P	M3	M2	P	M3	M2
-12501	0	0	-12501	0	0	-12501	0	0
-10689	2110	0	-10662	1536	1536	-10689	0	2110
-9175	3872	0	-9148	2770	2770	-9175	0	3872
-7818	5452	0	-7791	3876	3876	-7818	0	5452
-6240	6610	0	-6236	4683	4683	-6240	0	6610
-4461	7272	0	-4480	5151	5151	-4461	0	7272
-2943	7010	0	-2975	4978	4978	-2943	0	7010
-1508	5967	0	-1551	4258	4258	-1508	0	5967
-129	4258	0	-198	3092	3092	-129	0	4258
1225	1915	0	1165	1442	1442	1225	0	1915
2134	0	0	2134	0	0	2134	0	0

Notes:

1. P values are modified to remove kink in interaction curve at maximum allowed compression per AREMA code.
2. Compression is negative for SAP2000.

- Input steps

1. Define hinge properties

    a. Select menu item: *Define > Hinge Properties*
    b. Select defined hinge properties: *Default-PMM*
    c. Click: *Define New Property*
    d. Type property name: *colhng*
    e. Uncheck default
    f. Click: *Modify/Show for PMM*
    g. Select scale factor for rotation: *User SF* (leave value = 1)
    h. Click: *Modify/Show Moment Rotation Curve Data*

i. Enter values shown below and click OK

Point	M/My	Rot/SF
A	0	0
B	1	0
C	1	0.02
D	0.2	0.02
E	0.2	0.06

Note: Ultimate rotation from moment-curvature program is not critical for this example, except must be high enough to allow all hinges to form prior to failure.

j. Click: *Modify/Show PMM Interaction Surface Data*
k. Click: *Define/Show User Interaction Surface*
l. Change number of curves to 3
m. Enter scale factors = 1.3 for M2 and M3
n. Enter first and last P points for all curves
o. Type or copy and paste remaining undefined values and click OK

2. Assign hinges to members

    a. Select all members at column bottom
    b. Select menu item: *Assign > Frame/Cable/Tendon > Hinges*
    c. Add *colhng* with relative distance = 0
    d. Select all members at column top
    e. Select menu item: *Assign > Frame/Cable/Tendon > Hinges*
    f. Add *colhng* with relative distance = 1

3. Define unit load case for nonlinear static analysis

    a. Select menu item: *Define > Load Cases*
    b. Type in *FX*, select *OTHER* and click *Add New Load*
    c. Select nodes at top of columns
    d. Select menu item: *Assign > Joint Loads > Forces*
    e. Select *FX* and type in unit load of 10 kips in global X dir

4. Define analysis cases

   a. Select menu item: *Define > Analysis Cases*
   b. Select *DEAD* and Click *Modify/Show Case*
   c. Change analysis type to nonlinear and click *OK*
   d. Click: *Add New Case*
   e. Type case name: *PUSH*
   f. Change analysis type to nonlinear
   g. Select initial condition: *Continue from State at end of Nonlinear Case* (DEAD)
   h. Add FX for loads applied
   i. Click *Modify/Show* load application
   j. Select: *Displacement Control*
   k. Enter displacement magnitude = 0.4
   l. Click *Modify/Show* results saved
   m. Select: *multiple states*

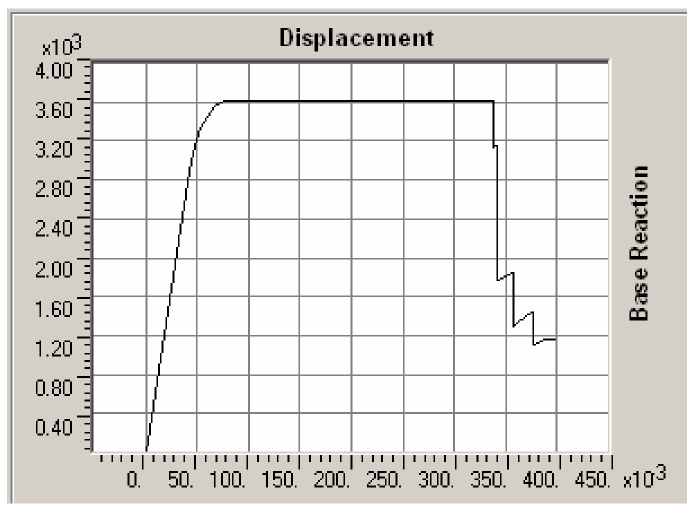
5. Perform nonlinear static analysis

   a. Select menu item: *Analyze > Run Analysis*
   b. Click: *Run Now*

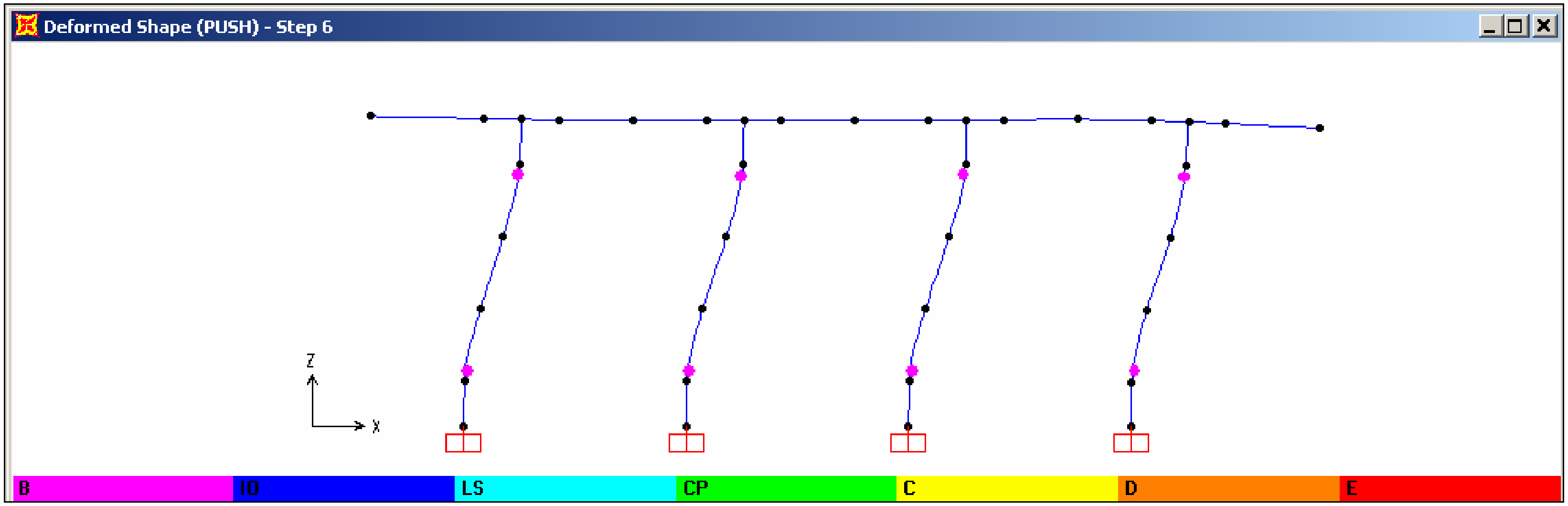
- Verify results

1. Display results

   a. Select menu item: *Display > Show Static Pushover Curve*



b. Select menu item: *Display > Deformed Shape*



c. Select menu item: *File > Print Tables*

Table: Element Forces - Frames

Frame	Station	OutputCase	StepNum	P	V2	M3
Text	ft	Text	Unitless	Kip	Kip	Kip-ft
102	0.0000	PUSH	6.000000	-58.028	737.795	5493.2704
202	0.0000	PUSH	6.000000	-802.655	919.764	6825.4076
302	0.0000	PUSH	6.000000	-323.195	853.890	6328.7005
402	0.0000	PUSH	6.000000	-1781.067	1093.187	8028.0166

2. Verification computations

a. Verify: P-M3 values match 1.3 x interation diagram values

P	1.3 x M	M3	Difference
-2943	9114		
-1781	**8277**	**8028**	**3%**
-1508	7758		
-803	**6890**	**6825**	**1%**
-323	**6034**	**6329**	**5%**
-129	5535		
-58	**5513**	**5493**	**0%**
1225	2489		

- Check bent cap and footing design (not shown)