CES 4702 - CONCRETE DESIGN
TERM PROJECT

MULTI-FAMILY HOUSE IN FORT MYERS, FL

submitted to:

Professor Jerry W. Wekezer, P.E.
Brandon Baggett, TA,
Cezary Bojanowski, RA

by

DESIGN GROUP No. 18:

Team members	Requested max credit distribution:
Name 1	10 ?
Name 2	20 ?
Name 3	30 ?
TOTAL:	60 points

December 5, 2007

PROBLEM STATEMENT

Use RISA 3-D commercial software for structural analysis of the assigned multi-family house in Fort Myers. Live load and dead load calculation for this building was explained in class and this data shall be used for this part of the project. Consider a typical 2-D frame for analysis (3-D is not required) and calculate maximum and minimum values of required internal forces:

- o bending moments M and shear forces V for selected beams, and
- o bending moments and axial forces for selected columns.

BUILDING ASSIGNMENTS

Group	Location	Applicable building parameters
18	Fort Myers, FL	3rd floor, public rooms in multi-family house f'_c = 4 ksi, x = 15 ft, y = 40 ft, z = 20 ft

|	
UNFACTORED LOAD: Dead load: D = 2,250 lb/ft Live load: L = 2,000 lb/ft Wind load: W = 654 lb/ft	

1. MODIFY THE GRID TO EASE THE ELEMENTS GENERATION

Click on the drawing toolbar to redefine grid. The drawing grid may be specified in plan or in either elevation. For our model we will leave the drawing grid in the XY plane.
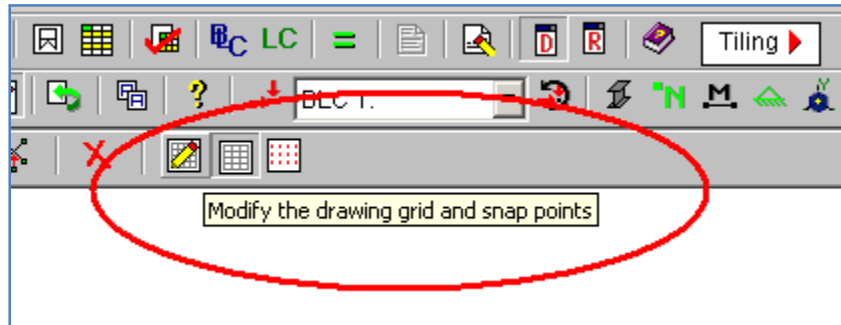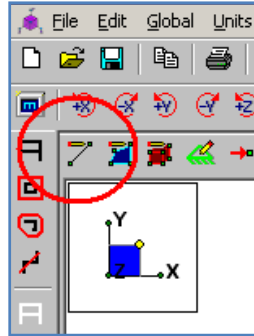

Now type in the X axis column '3@40' to get three bays each 40ft long. In Y axis column type 4@15 to get four stories each 15 ft high. The dialog window should look like this:
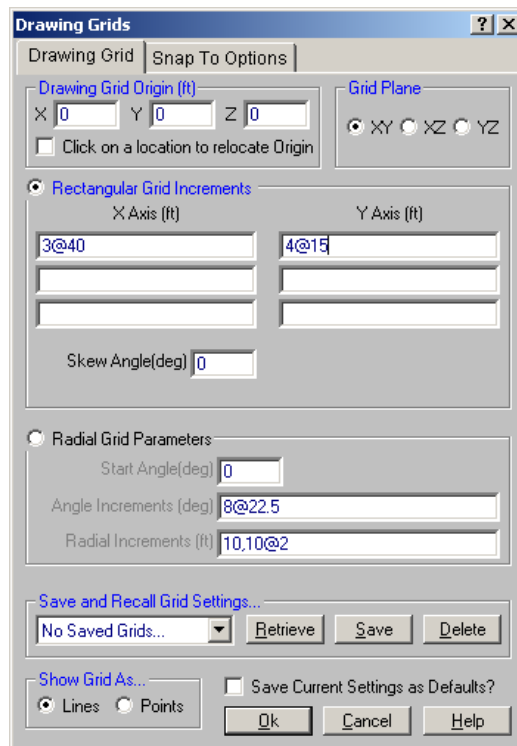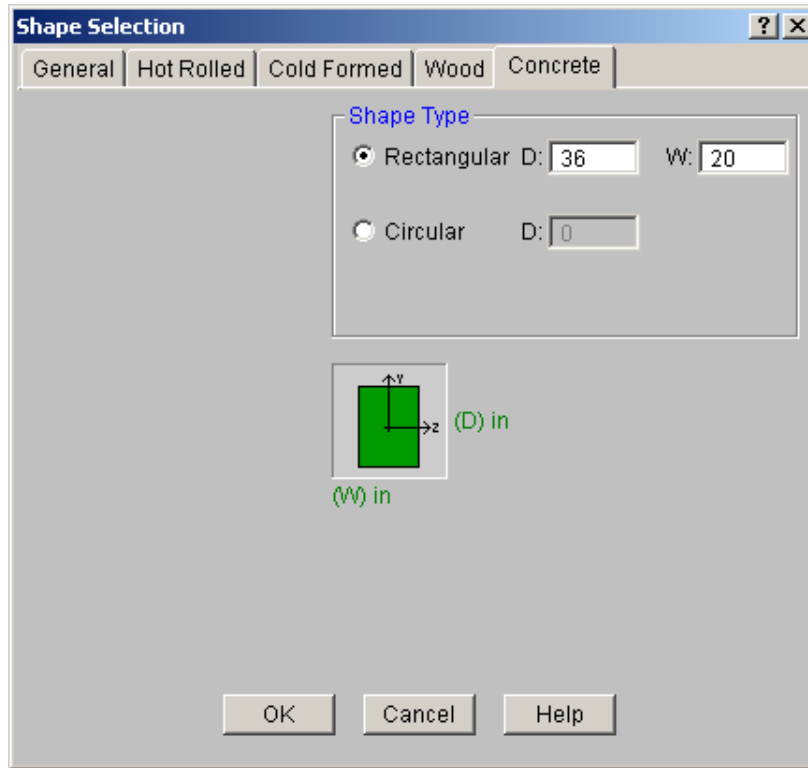

2. DRAWING MEMBERS

Elements will be drawn using created grid. We will start with drawing beam elements. Click button to start drawing.

We will see the **member properties** dialog. Select Concrete from Member Material Type. In the release codes select Fully Fixed at Both Ends. Pinned structure would be related to truss structure. Our frame has fixed joints. Click Apply to draw members. To assign the shape directly click on button encircled in the figure below.
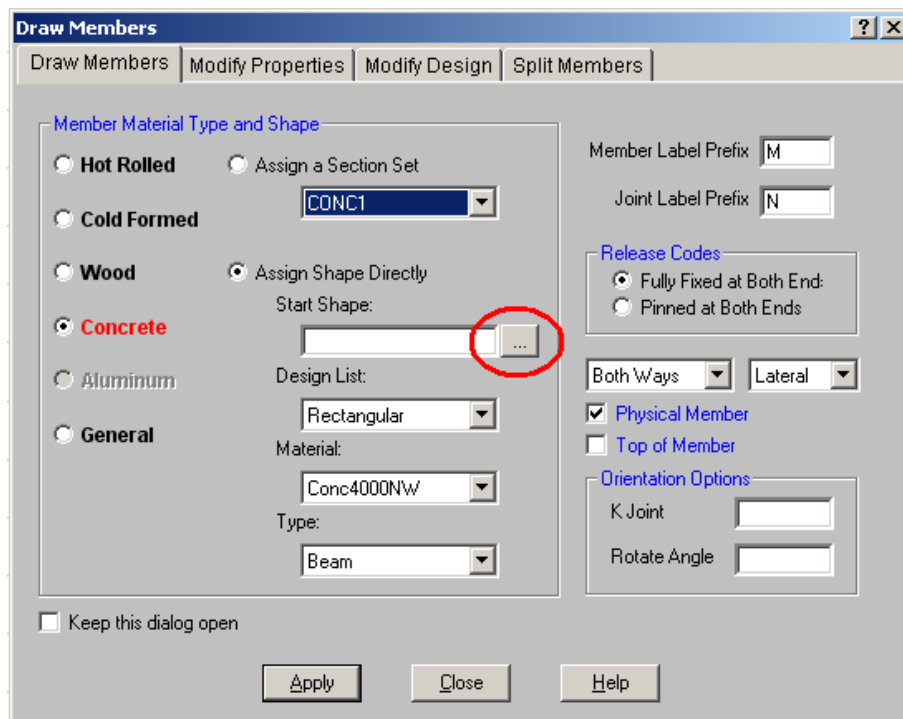
Define size of the Beam element cross section as your initial design assumptions show. In our case outer dimensions of all beams are 36in (depth) and 20in (width).

Clicking OK and then Apply cursor changes into . Placing it in proximity of grid intersections symbol appears. Clicking with left button of mouse we start drawing members. We end drawing with the same method. Right click terminates the continuous drawing. Clicking we get numbering of nodes and members. Current view looks like this:
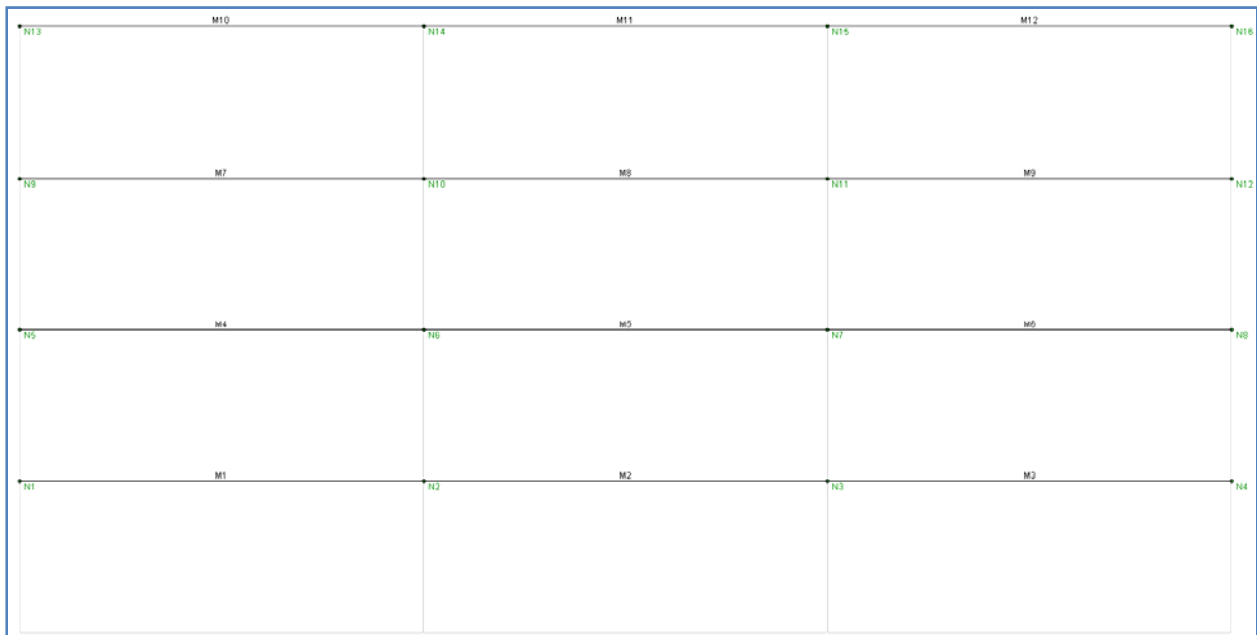
Click once again Draw new member button to create initial section for the column. Repeat steps and create cross section 24in x 24in not changing other parameters. Click on and to get isometric and rendered view of frame.
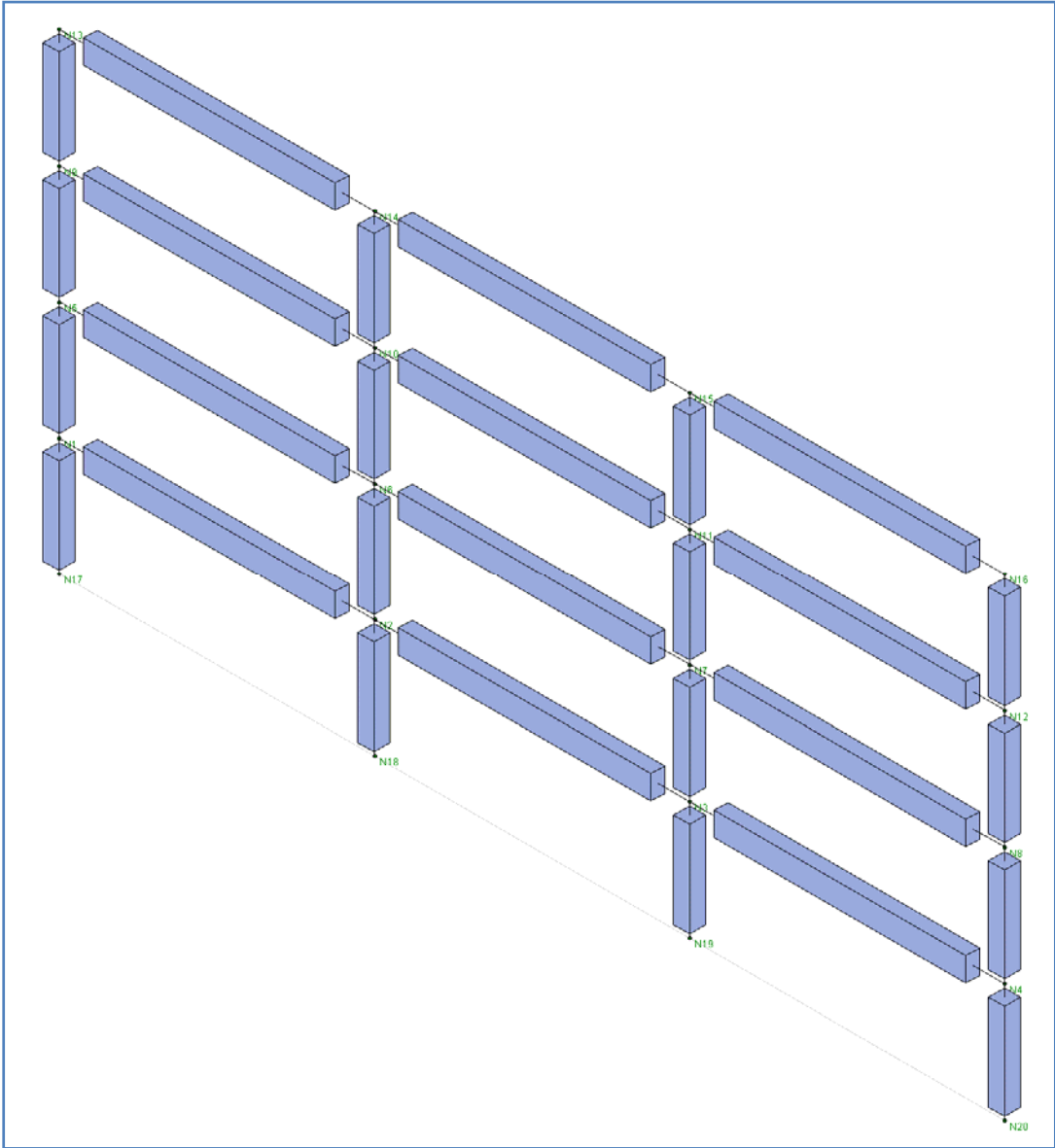
Click to go back to XY plane.

3. CREATING BOUNDARY CONDITIONS

Boundary conditions can be easily created using Spreadsheets. Choose from the list of spreadsheets Boundary conditions.

The table Joint Boundary Conditions will pop up. The entry "ALL" may be entered in the **Joint Label** field. The boundary 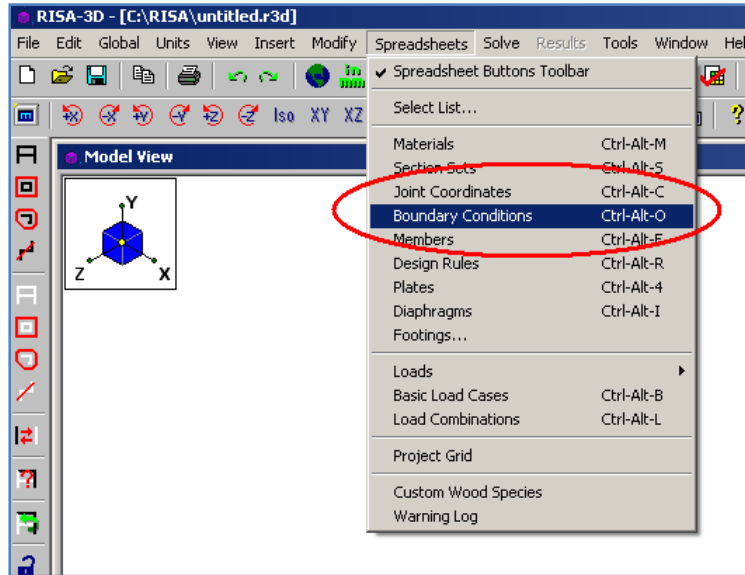conditions entered on this line will be applied to ALL the joints not otherwise listed. This is useful if you should want to lock certain directions of movement for all or most of the joints. We are solving a 2D frame defined in the XY plane and we're only interested in the planar action, you could enter "ALL" and put an "F" (for Fixed) for Z translation, X Rotation and Y Rotation. We are using option without calculation of reactions to not to mess up the reports. See the following figure:
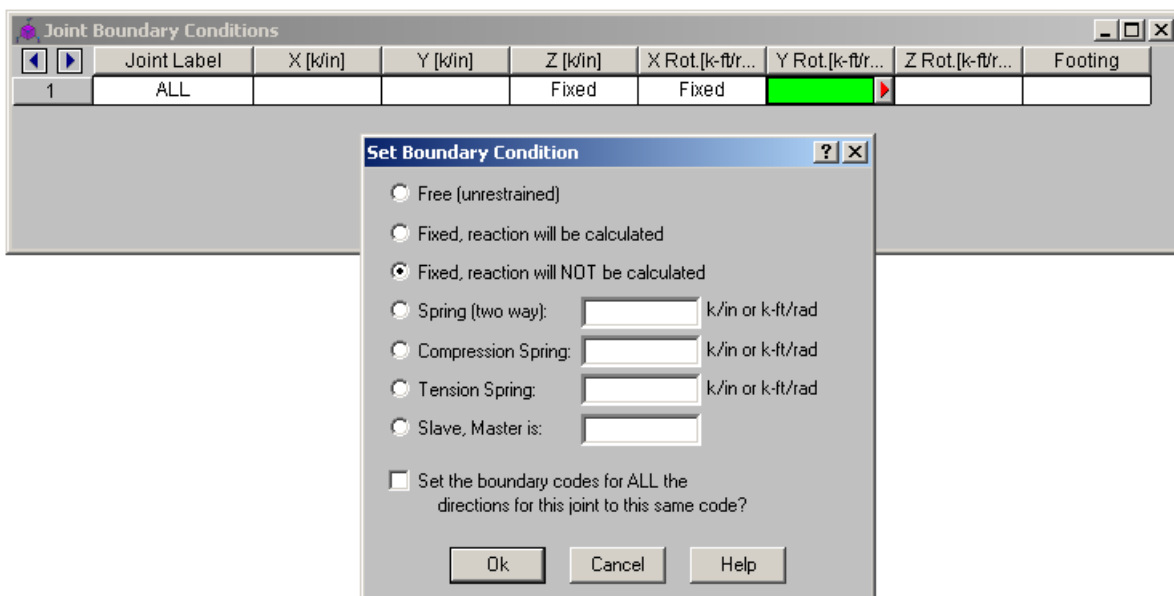
Enter all values, hit OK and end the Joint Boundary Conditions window. Note: If a joint is explicitly listed with boundary conditions, those boundary conditions override the "ALL" conditions for all 6 directions. The "ALL" specified boundary codes apply only to those joints NOT otherwise listed on the **Boundary Conditions Spreadsheet**. This is why joints 1 and 2 in the figure above also have the Fixed code in the Z translation, 2x Rotation and 2y Rotation fields.

Rest of the Boundary Conditions we will enter with other method. Find button to start modifying boundary conditions for selected nodes. Click fixed and conditions are filled in for you.
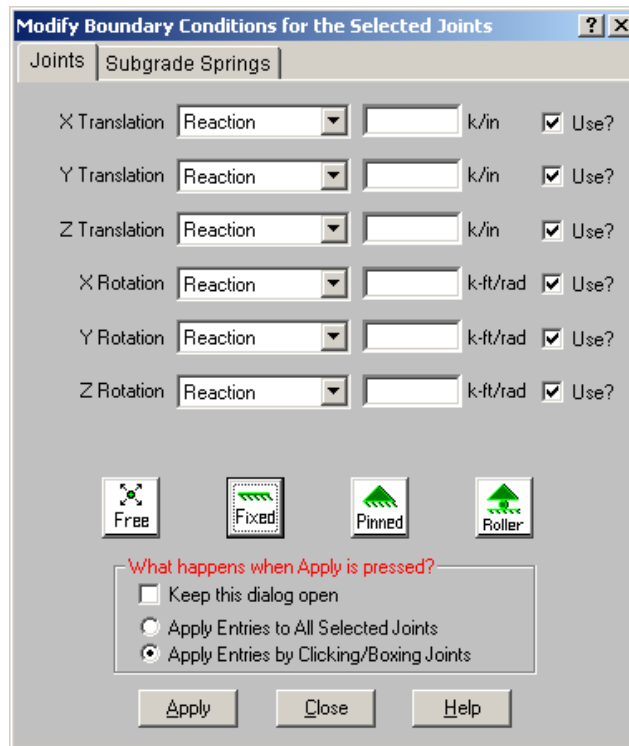
Hit Apply. The mouse cursor now looks like this , which indicates that you are ready to apply the boundary condition to any joint that you click on. We want to apply the boundary condition to the base of both columns at joints N17 to N20. Right click to end the command.

4. DEFINING LOADS

Open the **Basic Load Cases** spreadsheet by selecting it on the **Spreadsheet** menu. Enter following BLC descriptions in first column:
1. Dead Load
2. Live Load 1
3. Live Load 2
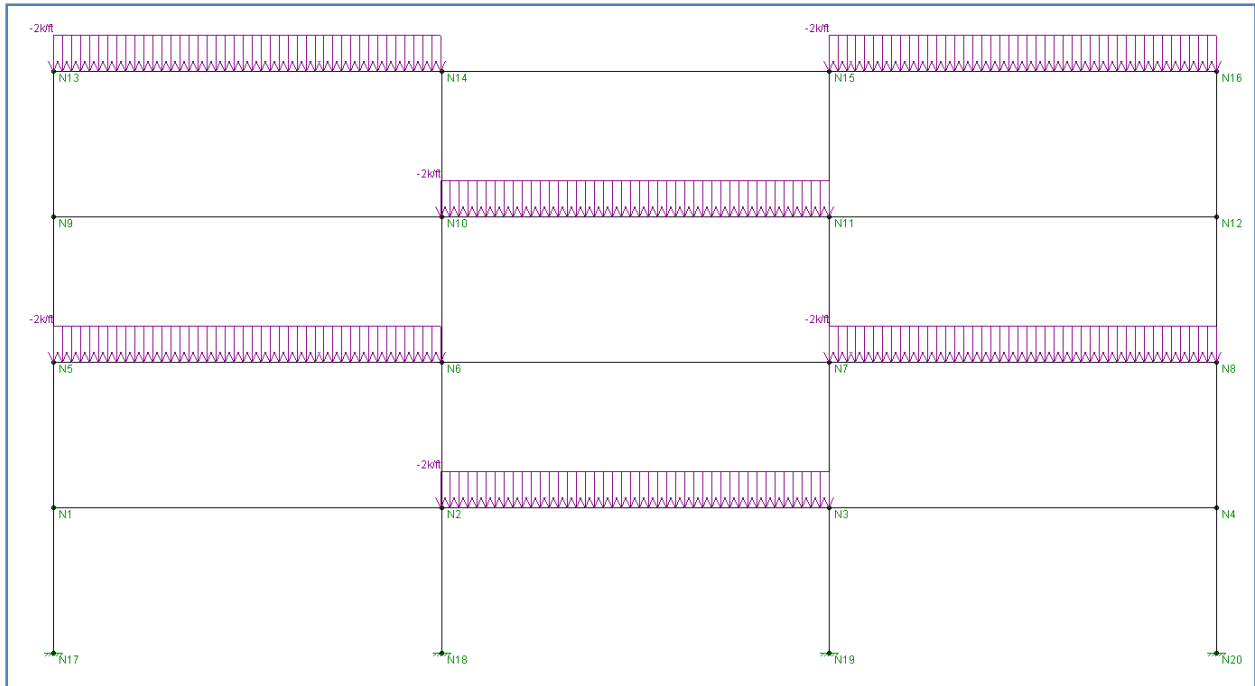4. Live load 3
5. Wind Load

8

Basic Load Cases

	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Joint
1	Dead Load	None				
2	Live Load 1	None				
3	Live Load 2	None				
4	Live Load 3	None				
5	Wind Load	None				

Close the window and Click the **Distributed Loads** button. Change the value in the **Start Magnitude** so that it reads "-2.250". Notice how the value for the **End Magnitude** automatically defaults to this same value. Be sure the value is correct unit wise and select correct Basic Load Combination from the list. In this case it is Dead Load.

Apply load by clicking on all beams. After finishing do the "right click" twice and from the menu select Recall Last Dialog. You will see the same window as above. Now we will define three different cases of Live Load combinations.

- First will be for finding greatest forces in columns – Live Load on all beams
- Second will be for finding greatest moment at midspan
- Third case will be used for winding maximum bending moment above support

This is the load pattern for second case, loads in Live Load 2 group:

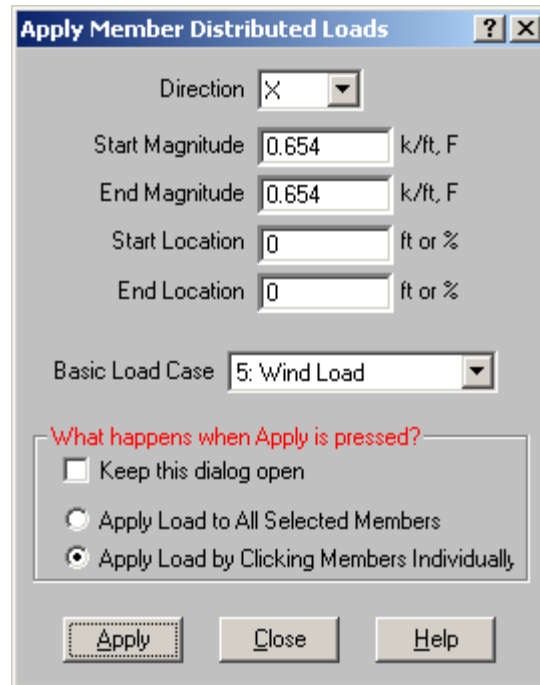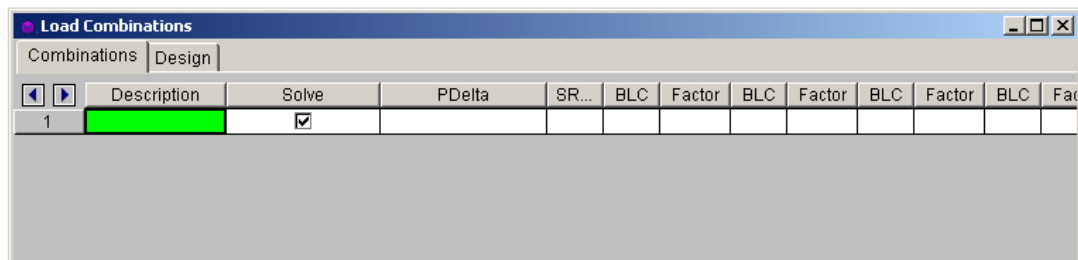This is the load pattern for third case, loads in Live Load 3 group:
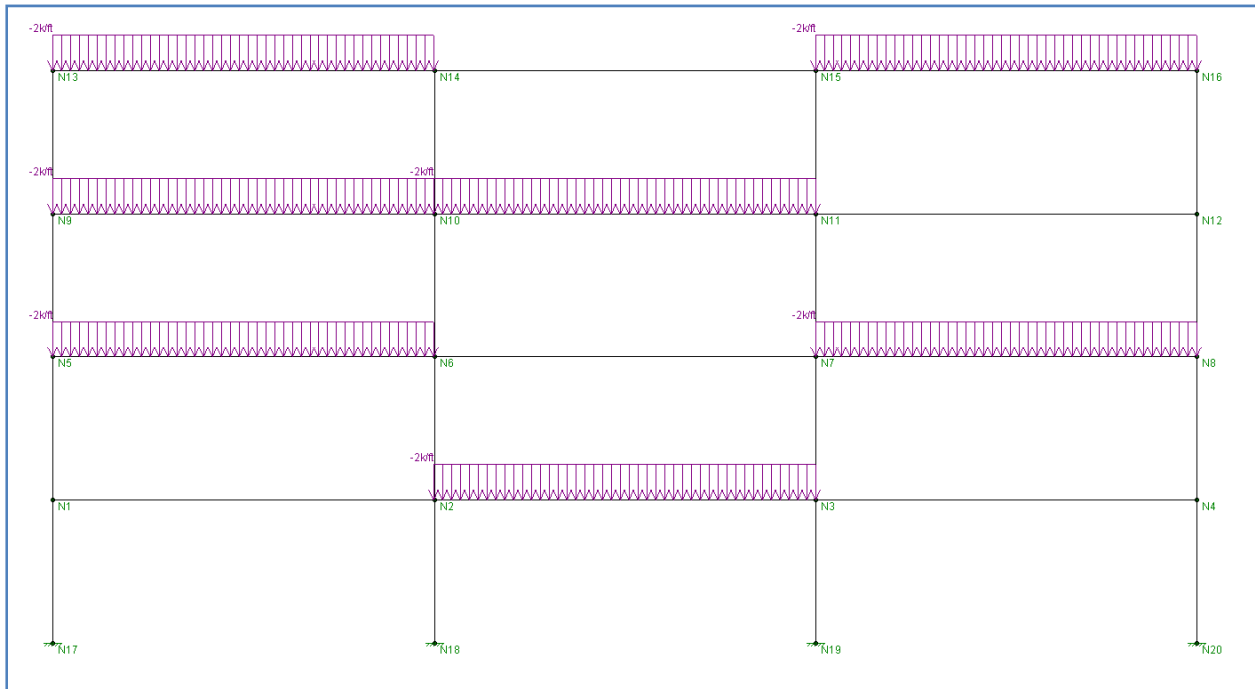
Repeat the same procedure for Wind load and apply them to columns at one side with load directed into the structure in X direction.

1) DEFINING LOAD COMBINATIONS

Go to spreadsheets and select Load Combinations. The window like this will appear:

Now we will have to create combinations of loads defined by design code. We will have repeated some of them three times with different cases of live load distribution. Let's consider here two of them:

1) U = 1.2 D + 1.6 L
2) U = 1.2 D + 1.6 W + 1.0 L

Where D is a Dead Load, L is one of our three cases of Live Load and W is a Wind Load. Give a name to new combination: "Combination 1a". Then at the first field BLC click the red triangle and window Set BLC Entry window will pop up. At Basic Load Case list select 1: Dead Load in next column set Factor to 1.2. For second BLC choose @: Live Load 1 and set multiplying factor to 1.6. Repeat the same process for other live loads. After doing it three times your window should look like this:

Now select all data like in excel program. We will copy it and use for next three remaining combinations involving wind load. Click Enter on field Combination 1c and paste copied data. Change 1 into 2 in description of pasted rows. Change multiplying factor for live loads into 1.0 and as a next BLC put 5 which in our case stand for wind load. Multiplying factor for Wind Load is 1.6 according to code. If you do everything correctly your table will be now:

6) SOLVING AND VIEWING RESUTS

To solve the problem click [=]. From the window that pops up choose Envelope of Marked Combinations. Looking at previous figure automatically all combinations were marked to be taken into account in calculations.
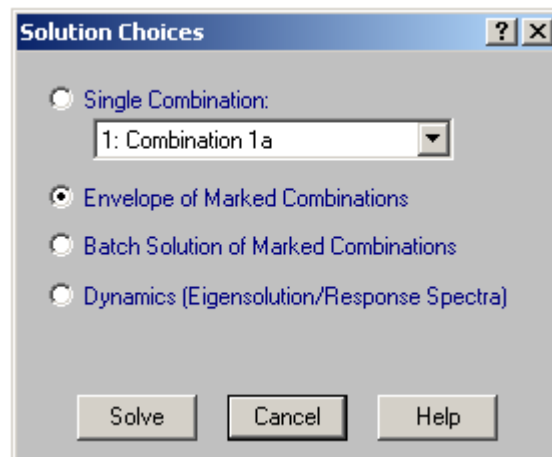
Now problem should be calculated. You should see new commands window:
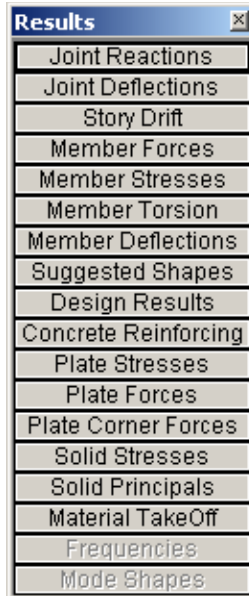
It will give you access to different spreadsheets with various results. We will be mostly interested in Member Forces. Click on button to see table like below:

|	Member	S...		Axial[k]	lc	y Shear[k]	lc	z Shea...	lc	Torque...	lc	y-y Mo...	lc	z-z Moment[k-ft]	lc
1	M1	1	max	-4.444	4	109.799	1	0	1	0	1	0	1	561.124	1
2			min	-28.244	2	42.039	6	0	1	0	1	0	1	122.948	6
3		2	max	-4.444	4	51.099	1	0	1	0	1	0	1	-90.872	2
4			min	-28.244	2	15.339	6	0	1	0	1	0	1	-257.589	4
5		3	max	-4.444	4	-7.167	2	0	1	0	1	0	1	-152.701	2
6			min	-28.244	2	-11.612	4	0	1	0	1	0	1	-460.864	1
7		4	max	-4.444	4	-33.867	2	0	1	0	1	0	1	64.551	5
8			min	-28.244	2	-66.301	1	0	1	0	1	0	1	-91.358	1

For member M1 we see results for 5 different sections. Number of sections through the element is defined in Global Parameters in solution card. Sometimes default value of 5 is not enough. Increase slightly that value for your project (let's say 7 or 9) and repeat the solution. For each section we get min and max value of axial force, shear forces and moments. Also we get in columns lc the load combination that caused extreme value of internal force.
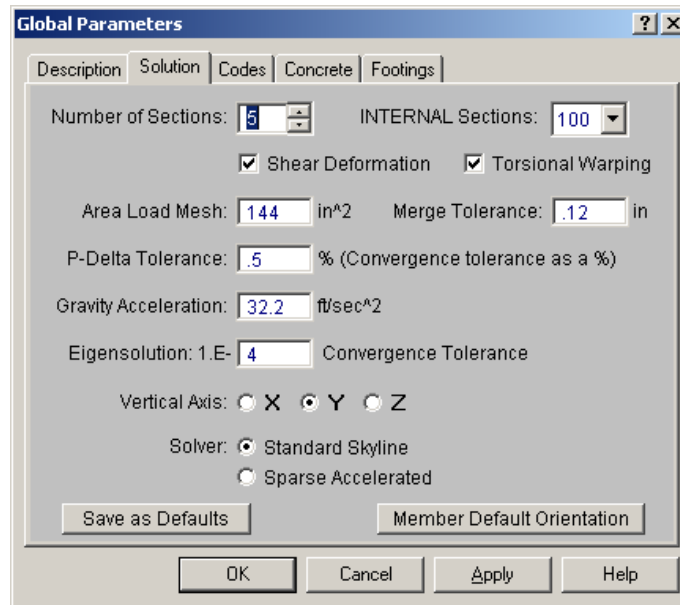
To draw envelope of bending moments we click on button and new window Set Options for Current View will appear on the screen.
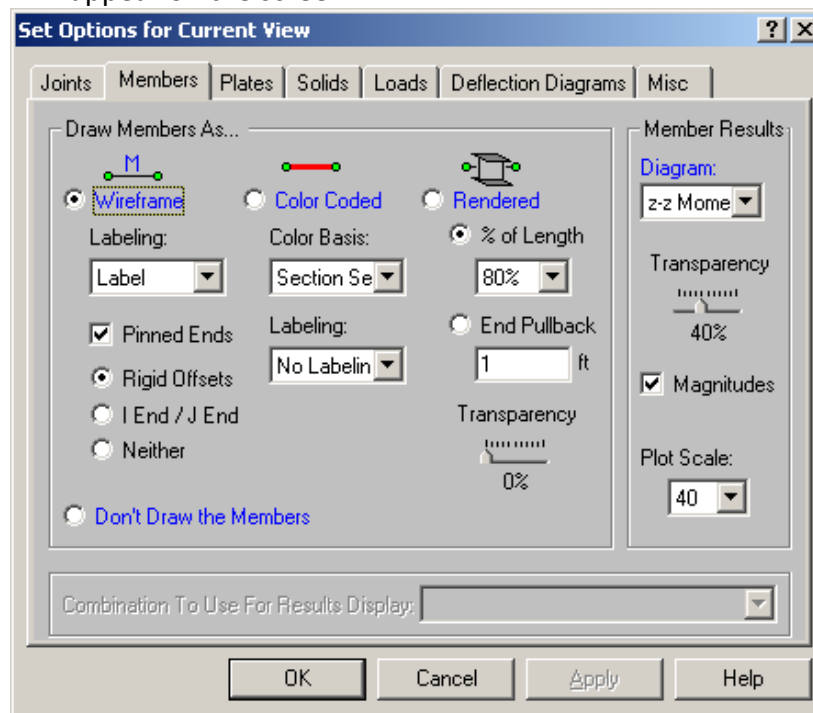
From Members card choose z-z Moments in Diagram and check Magnitudes. By applying that we should get envelopes of bending moment envelope for all members like in the figure below.
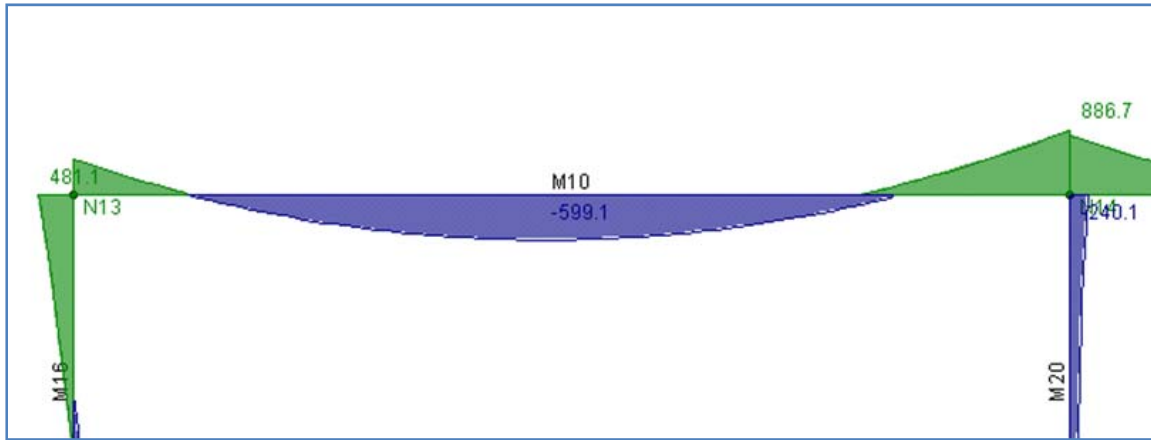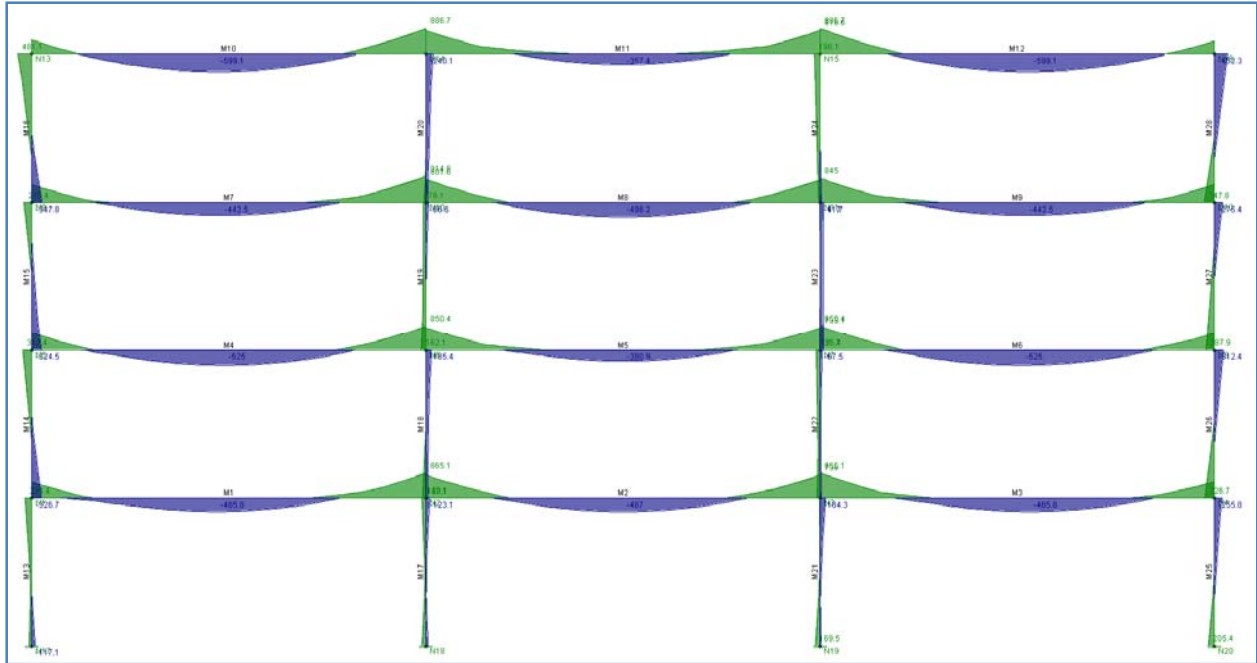
For design purposes we would like to have more detailed data than those presented in figures above. Unfortunately the RISA program does not give the option to show detailed report after envelope solution type of analysis.
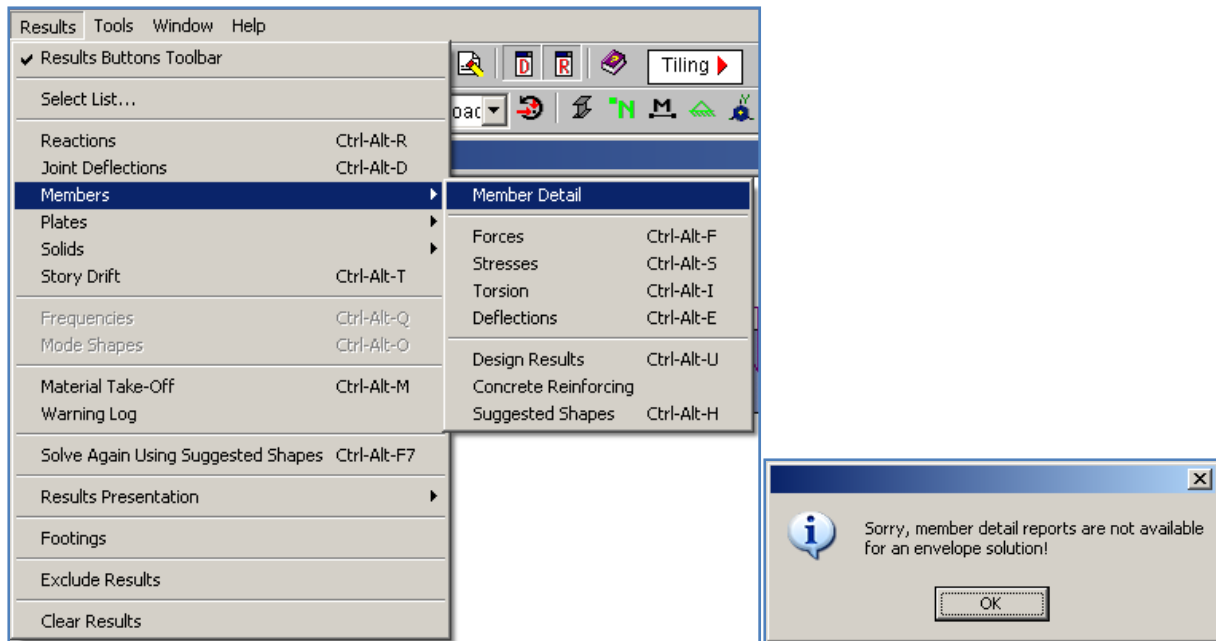
To get detailed data about separate members we will have to perform several analyses for separate load cases. Let's take a look back to spreadsheet of Results > Members > Forces.

Knowing the load combination # that caused extreme value of force we can perform quickly once again analysis for that particular combination. For member 1 (beam in bending) we are interested in biggest absolute values of shear and moment. We will be considering:

- Max positive shear 109.8k occurred at combination 1 in section 1
- Max negative shear -125.0 k occurred at combination 1 in section 5
- Max positive moment 561.1 k-ft at load combination 1 in section 1(left end)
- Max positive moment 865.1 k-ft at load combination 1 in section 5 (right end)
- Max negative moment -460.9 k-ft at load combination 1 in section 3 (middle of the beam)

For that particular member all extreme forces come from LC 1. Let's perform analysis for that LC. Click to solve button and choose Single Combination 1: Combination 1a.
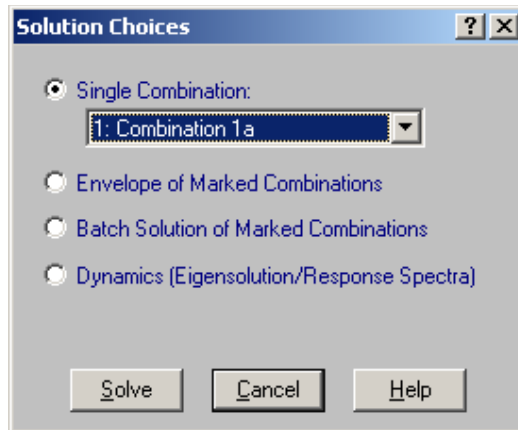
Click Solve and then Yes when asked for clearing current results. Table with results for that LC should appear. Also new command appeared Detail Report for Current Item . In table make active some window at level of Member 1. Then click on that new command. You should see:

Beam: **M1**

Shape: **CRECT36X20**	Concrete Stress Block:	**Rectangular**
Material: **Conc4000NW**	Cracked Sections Used:	**Yes**
Length: **40 ft**	Cracked 'I' Factor:	**.35**
I Joint: **N1**		
J Joint: **N2**		

Code Check: **1.075 (bending)**
Report Based On 97 Sections

Beam Design does not consider any 'T' & 'My' Moments, nor 'A' & 'Vz' Forces.

ACI 318-02 Code Check

|					
Top Bending Check	**1.075**	Bot Bending Check	**0.860**	Shear Check	**0.977 (y)**
Location	**39.167 ft**	Location	**18.75 ft**	Location	**36.25 ft**
Gov Muz Top	**763.019 k-ft**	Gov Muz Bot	**-465.779 k-ft**	Gov Vuy	**102.988 k**
phi*Mnz Top	**709.81 k-ft**	phi*Mnz Bot	**541.915 k-ft**	phi*Vny	**105.421 k**
Tension Bar Fy	**60 ksi**	Concrete Weight	**.145 k/ft^3**	Top Cover	**1.5 in**
Shear Bar Fy	**60 ksi**	Concrete Type	**Normal WT**	Bottom Cover	**1.5 in**
F'c	**4 ksi**	E_Concrete	**3644 ksi**	Side Cover	**1.5 in**
Flex. Rebar Set	**ASTM A615**	Min 1 Bar Dia Spac.	**No**	Legs/Stirrup	**2**
Shear Rebar Set	**ASTM A615**				

When you scroll down in that window you will also have detailed design solution. It may be used by you as a good check to your hand calculations.
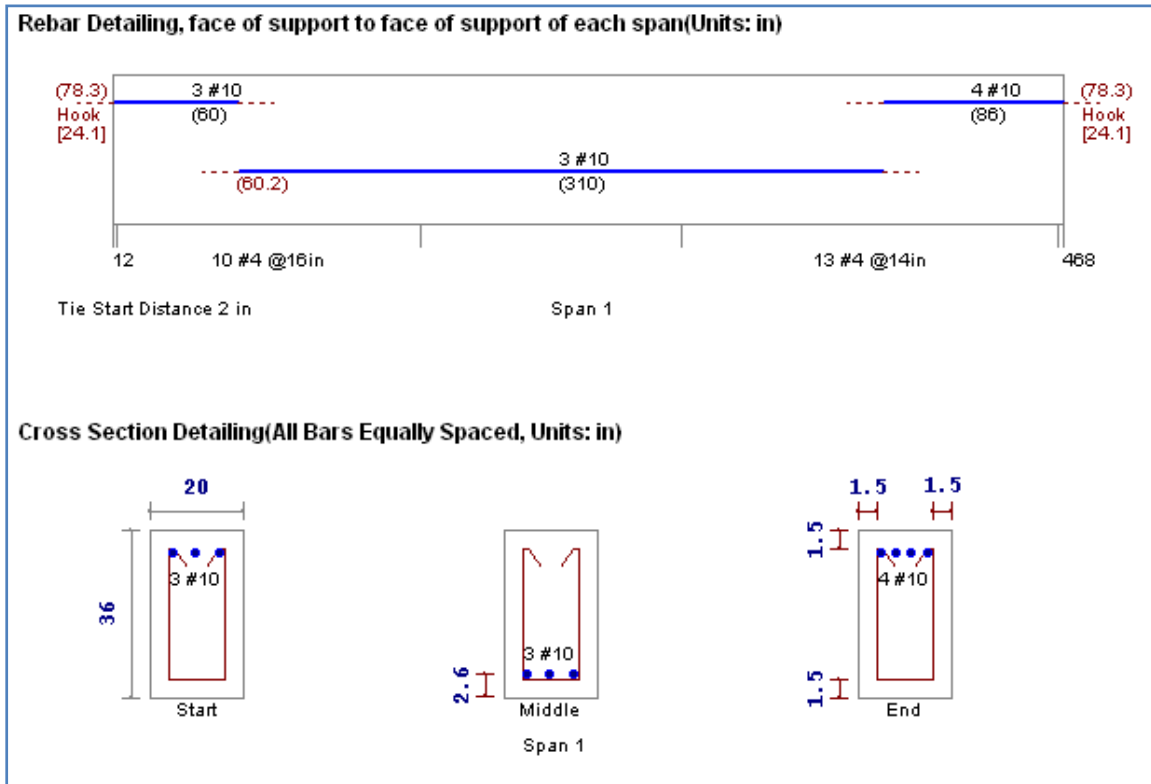
In the table at the top of report when you click on the graph you will get enlarged detailed diagrams of internal forces
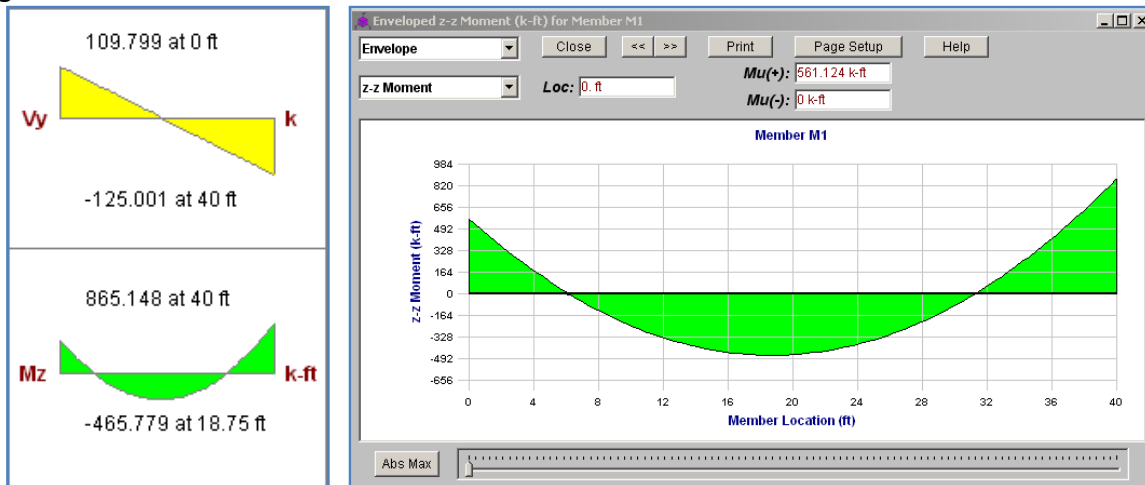
In this case you don't need to perform another analysis but it can happen that bottom of the beam will be designed for load from one LC and the top from the other. So be aware of that.