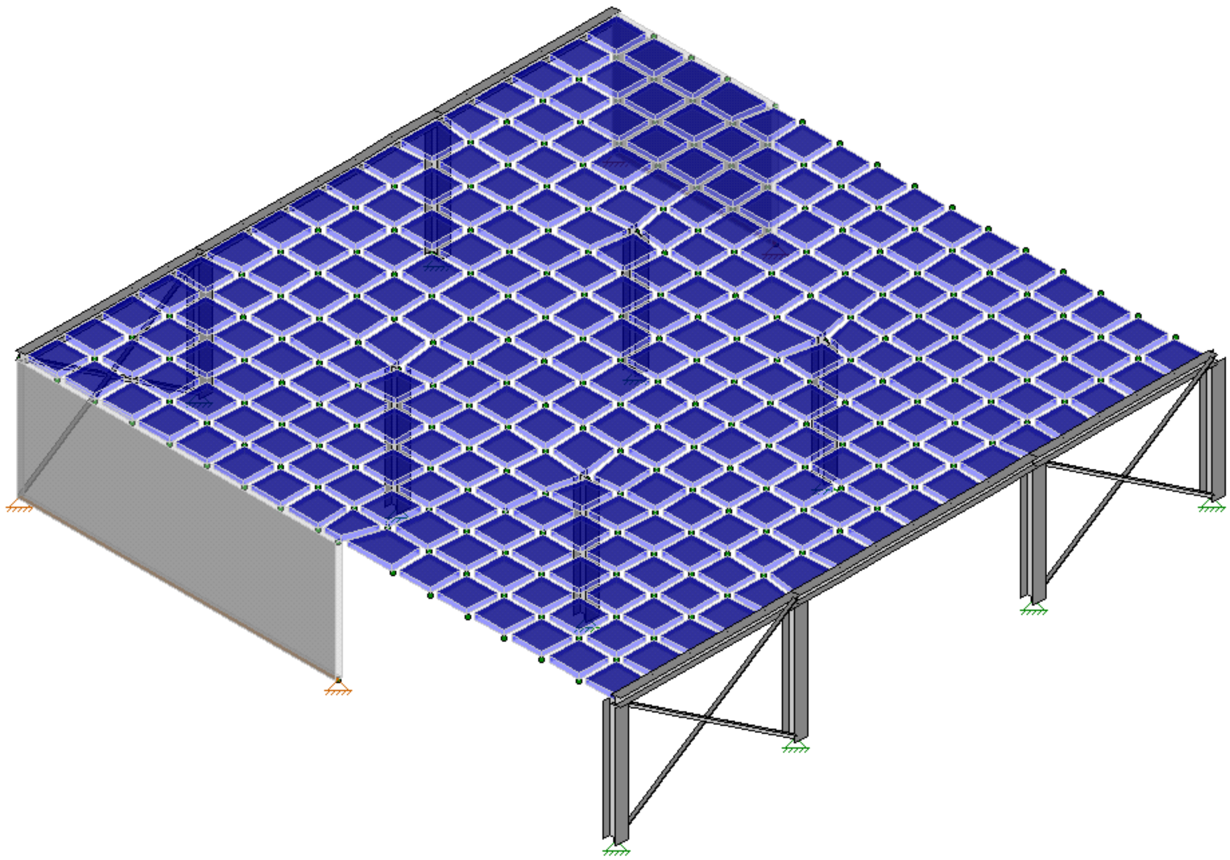
Example 6: Discretizing Floor Slab Mass

This is a model that will introduce us to the Response Spectra Analysis Scaling Factors. We will review these factors and discuss how to apply them to a model.



Project Definition: DYN_SHEAR.R3D

Modeling Description

This model is a one story structure with a braced frame in one direction and a shear wall in the other. There is significant Torsional irregularity, so a response spectra analysis would likely be required by code.

The class exercise will consist of building this model and running a response spectra analysis using two different ways for modeling the mass of the structure. The first way is to use the self weight of the structure as the basis of the dynamic mass. The second way is to lump the weight of the structure at discrete locations to reduce the response of local modes.

Modeling Procedure

Let's start by creating steel frames in the YZ plane

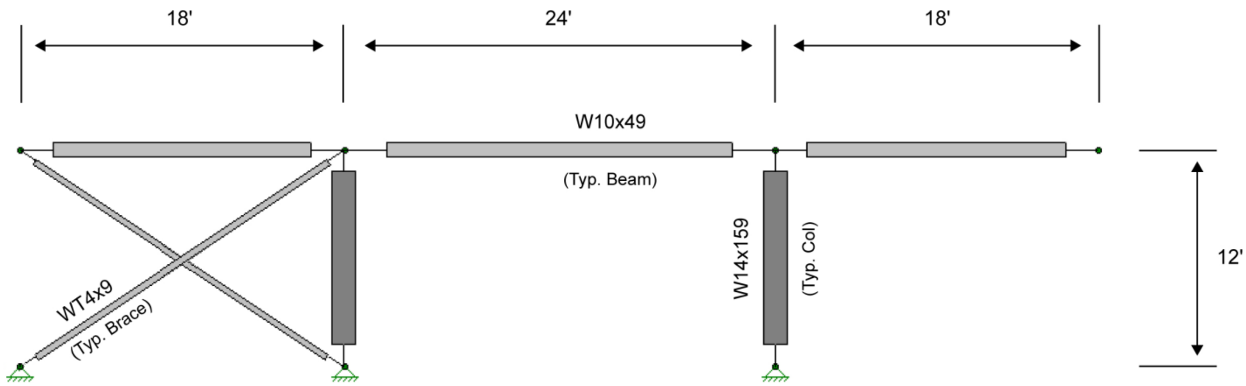
Click on the Section Sets button of the Data Entry toolbar. Make sure the Hot Rolled tab is selected and enter the information shown below:

Hot Rolled Steel Section Sets

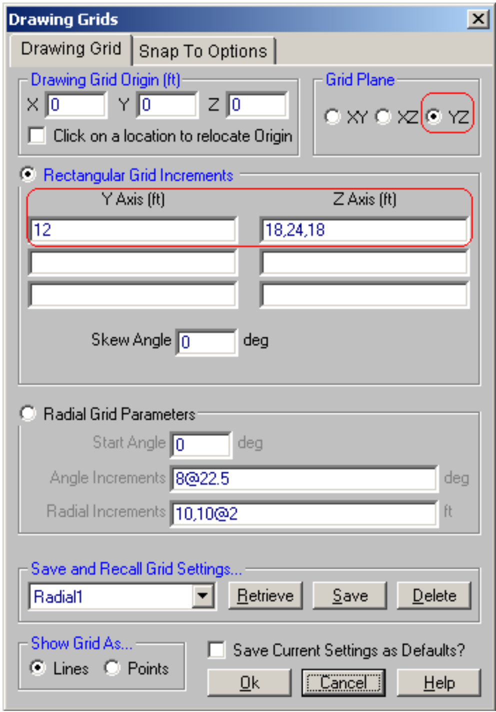
Hot Rolled | Cold Formed | Wood | Concrete | Aluminum | General

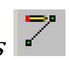
	Label	Shape	Type	Design List	Material	Design Rules	A [in2]	Iyy [in4]	Izz [in4]	J [in4]
1	Column	W14x159	Column	Wide Flange	A992	Typical	46.7	748	1900	19.7
2	Beam	W10x49	Beam	Wide Flange	A992	Typical	14.4	93.4	272	1.39
3	Brace	WT4x9	VBrace	Wide Flange	A992	Typical	2.63	3.98	3.41	.0855

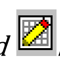
The frame at X=0 in the YZ frame is shown:

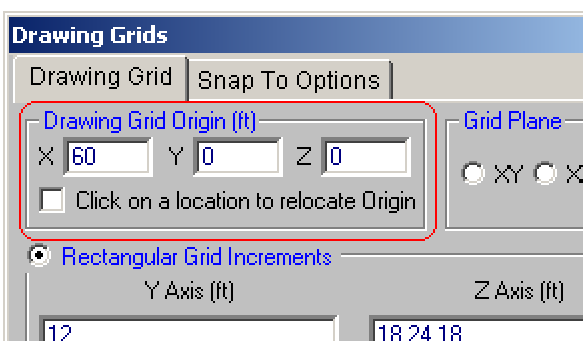


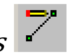
From the Main Menu toolbar select Insert – Drawing Grid and enter the information shown on the left below:

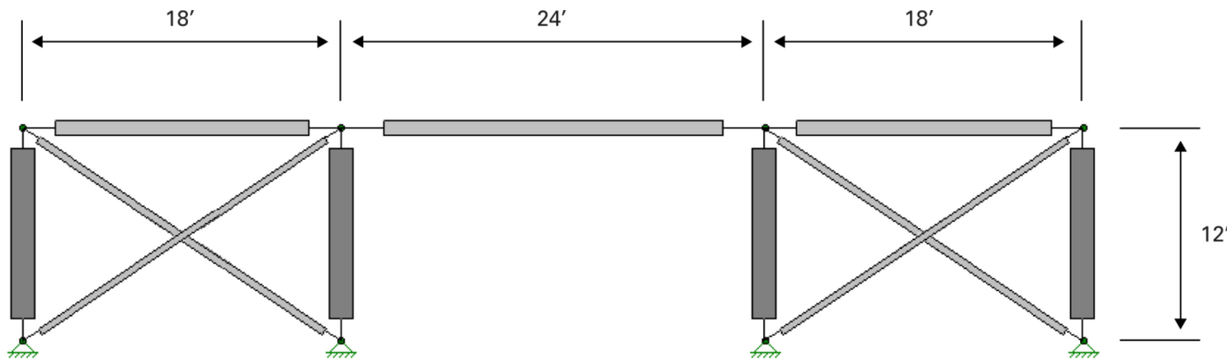


Select Ok and start drawing in the frame. Click on the Draw Members  *button and start drawing in the column, beam and brace members. Make sure to set the Release Codes to Pinned at Both Ends for all members (even the columns).*

Switch to an Iso (isometric view) to see the frame. Then click on the edit drawing grid  *button and change the Drawing Grid Origin to X = 60 as shown below:*



Select Ok and start drawing in the second frame. Click on the Draw Members  *button and start drawing in the column, beam and brace members shown below:*

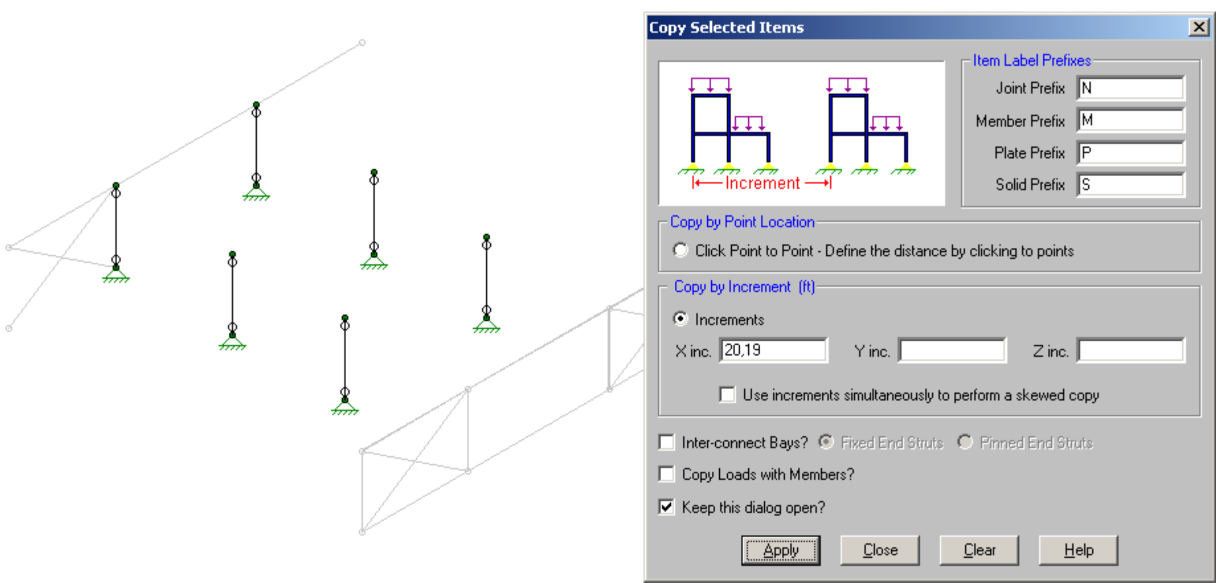


> **Note:** The columns, beams, and braces use the same member sizes as the previous frame. This is why we set up section sets to define these members.

Use the Draw / Modify Boundary Conditions button to  *to change the boundary conditions at the base of all the columns to pinned.*

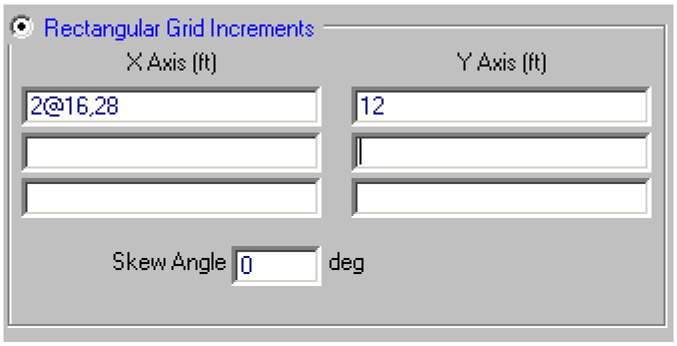
Defining the interior columns

Unselect the whole model except for the two middle column on the back frame. Then select Modify – Copy from the Main Menu toolbar and enter the information shown below (X increments of 20 and 19) to create the interior columns:

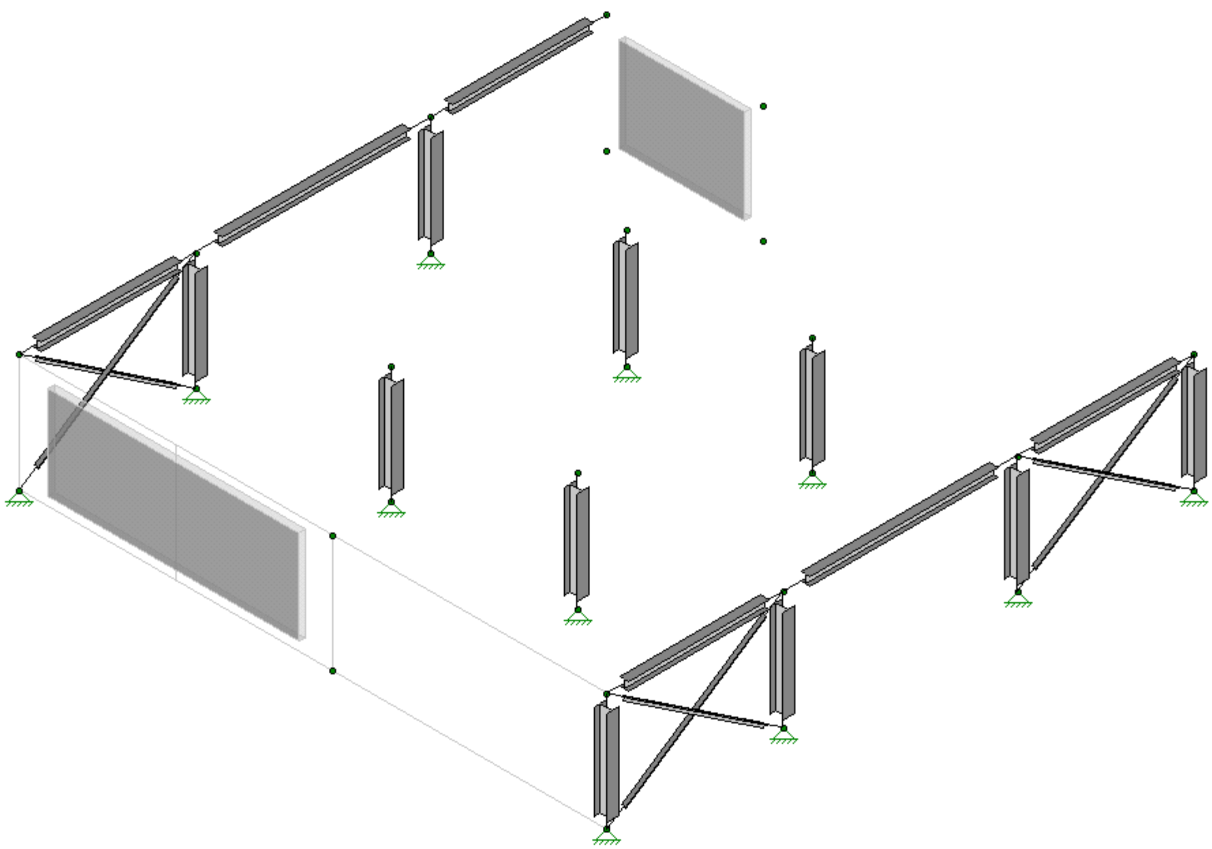


Drawing in the Wall Panels

Click on the edit drawing grid button and change the Drawing Grid Origin back to X = 0. Change the Grid Plane to XY and enter in the grid information shown below:



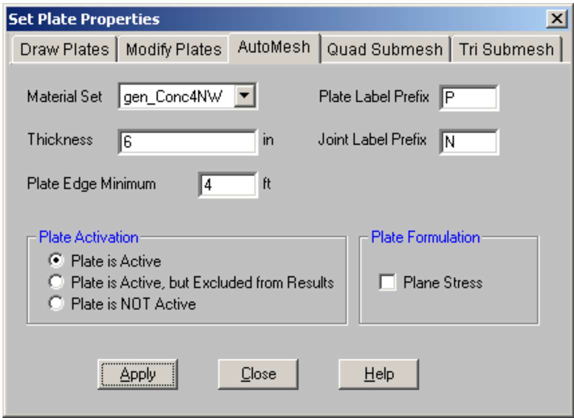
Click the Draw / Edit Wall Panels button to create a concrete wall panel (Conc4000NW) which is 8 inches thick. Draw in a 16 foot long wall panel at Z = 0. Then change the drawing grid to start at Z = 60 and draw in a 32 foot long wall panel. When you are done, select the entire model and then turn on rendering. The model should look like the image shown below:



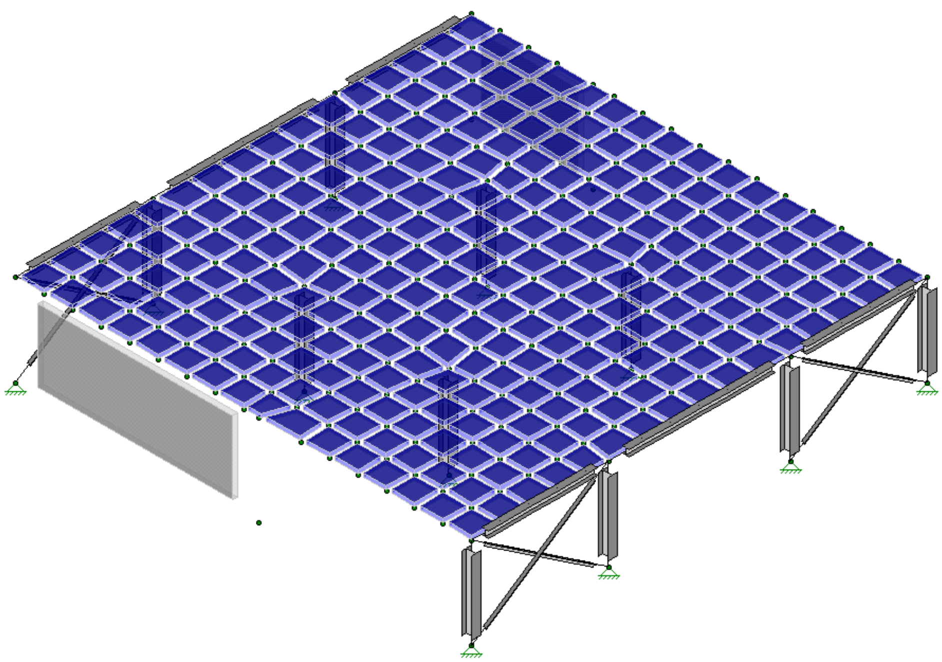
Note: When we solve, the program will automatically add in continuously pinned boundary conditions at the base of each wall.

Drawing in the Floor Slab

Click on the Draw Plates button  *and select the AutoMesh tab. Enter the information shown below:*



*Click **Apply** and then click on the four corners of the slab, double-clicking on the last corner to complete the action. When you are done, the slab / structure should look similar to the following image:*

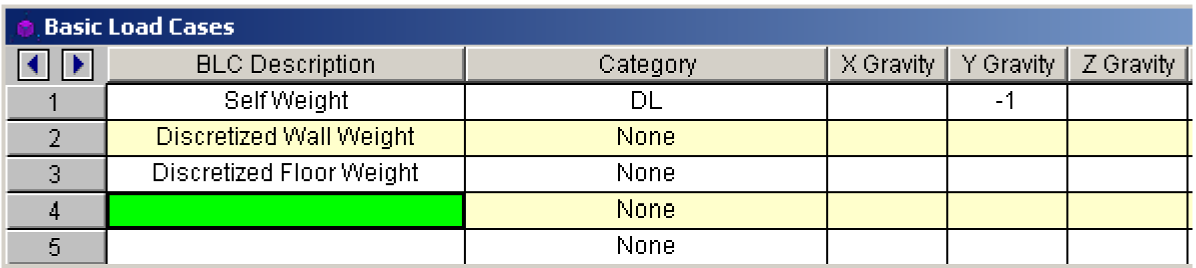


The AutoMesh utility generates a slab with the general concrete properties with a mesh size of approximately 4ft x 4ft. But, it locally adjusts the mesh so that it attaches to the currently selected joints on the walls and tops of columns.

Creating the Basic Load Cases

Now that the model is created, we will add some loading onto the structure. The basic loading is just the self weight of the structure. However, we will also be creating a set of discretized loads to represent the slab and wall self weight. Later we will compare the results produced when running a dynamic analysis with the true self weight versus the discretized self weight.

*Open the **Basic Load Cases** spreadsheet. Enter "Self Weight", "Discretized Wall Weight" and "Discretized Floor Weight" as the BLC descriptions for the first three basic load cases as shown below:*

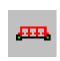

Basic Load Cases

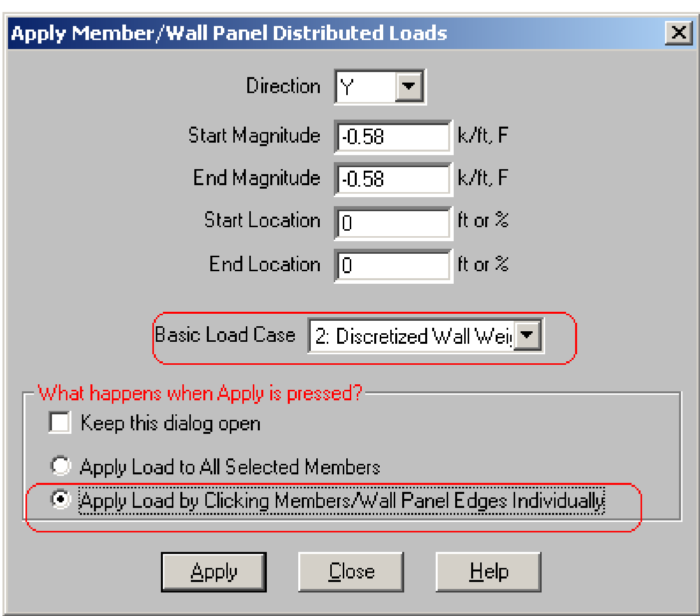
	BLC Description	Category	X Gravity	Y Gravity	Z Gravity
1	Self Weight	DL		-1	
2	Discretized Wall Weight	None			
3	Discretized Floor Weight	None			
4		None			
5		None			

Enter a Load Category of DL for the Self Weight load case along with a Y Gravity factor of -1.0. Leave the Category and Gravity factors for the discretized wall and floor weights blank.

Calculating the Discretized Wall Weight

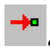
The walls are 12 feet tall and 8 inches thick. That give a total wall weight of 0.145 kcf * (8/12)ft *12ft = 1.16 kips per food of wall length. However, 50% of the wall weight is tributary to the support. So, we will apply a discretized load of 0.58 k/ft along the top of each wall.

Click on the Apply Distributed Load icon  *to open up the following dialog. Enter the information shown below. Be sure to select the option to **Apply Load by Clicking Members / Wall Panel Edges Individually**. Then click Apply and draw in the loads at the top of the two walls.*



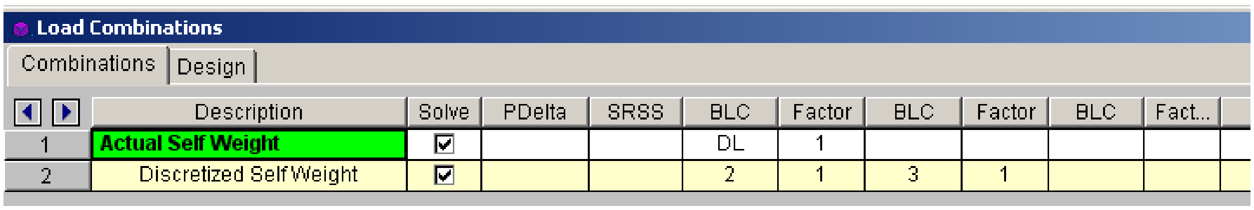
Calculating the Discretized Slab Weight

The total slab weight is 0.145 kcf * 0.5 ft * 60 ft * 60 ft = 261 kips. We could get really precise and try to determine how much axial force goes into wall and column when we solve the gravity load combination. And, that is probably the most accurate way to go. However, the easiest solution is to say that each interior column get approximately 38 kips of load, each edge column about 19 kips and each corner column about 9.5 kips. That would result in a total load of 4*38 + 4*19 + 4*9.5 = 266 kips, which is close enough to the 261 kips we were trying to match. The extra few kips can be considered to include some percentage of weight of the steel members.

Click on the Apply Joint Loads icon  *and select Basic Load Case 3 (Discretized Slab Weight). Then use this dialog to apply -38 kips of load to the tops of the interior columns, -19 kips to the tops of the edge columns and -9.5 kips to the tops of the corner columns.*

Create the Load Combinations & Review Loading

Change the model view to a wire frame view. Then select Load Combinations from the data entry toolbar. Enter the information shown below:

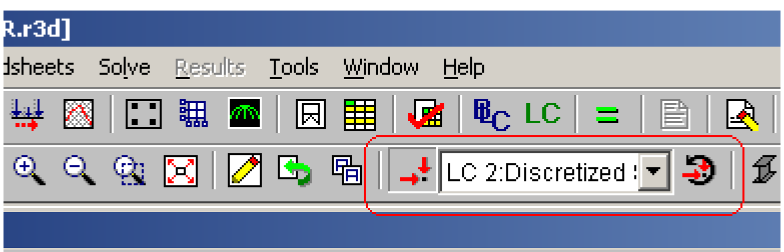


Load Combinations

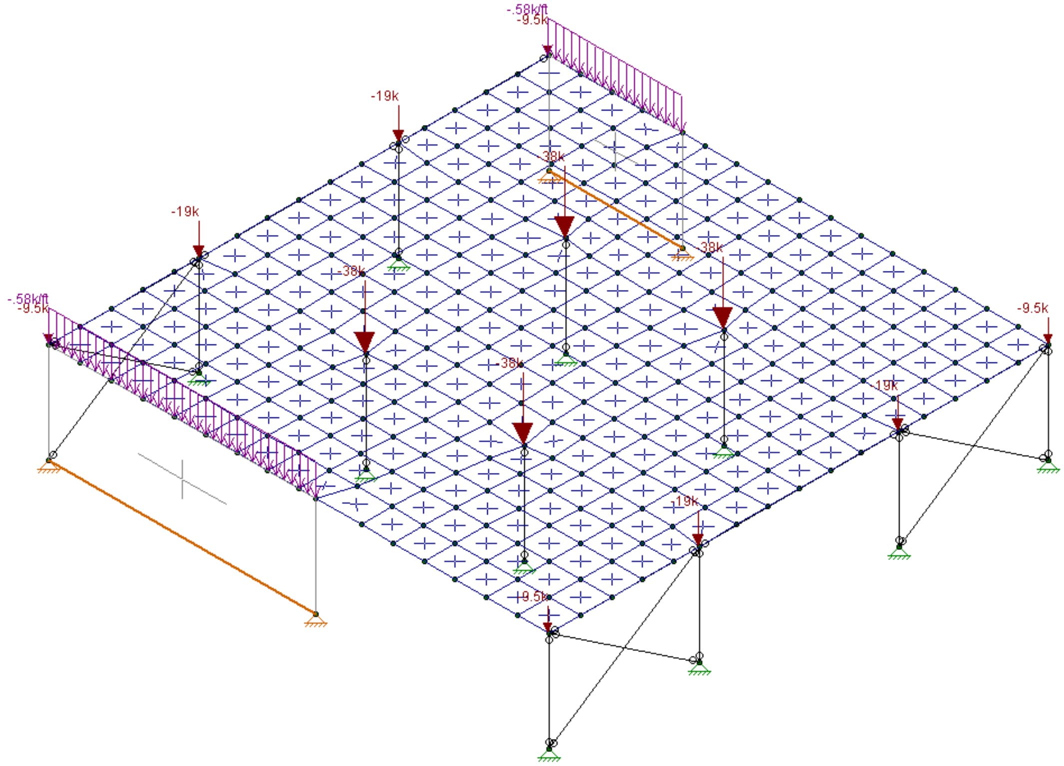
Combinations | Design

	Description	Solve	PDelta	SRSS	BLC	Factor	BLC	Factor	BLC	Fact...
1	Actual Self Weight	☑			DL	1				
2	Discretized Self Weight	☑			2	1	3	1		

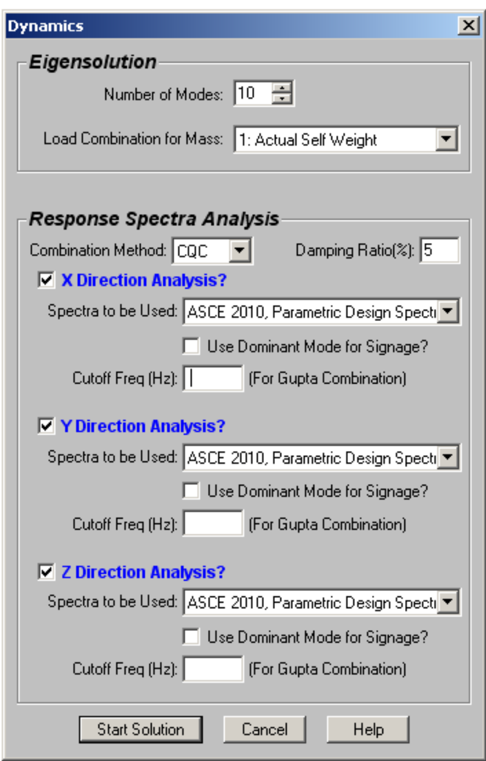
Use the controls shown below to toggle the display of loads on and to select the display of LC 2: Discretized Self Weight.



Return to the model view and your structure should look similar to the following:



Solve for Dynamics and Compare Results



Click Solve –Dynamics from the Main Menu toolbar to bring up the Dynamic solution options. Set the LC for dynamic mass to LC#1 (Actual Self Weight) and request 10 modes.

Request response spectra results for the X, Y and Z directions and click Start Solution to bring up the dynamic results shown on the next page.

Notice how the mass participation in the X direction is insignificant compared to the mass participation in the Y direction. This is because the shear walls create a very stiff system in the X direction whereas the vertical direction has a number of local modes associated with the slab vibrating. These modes do not significantly affect the overall seismic response. Therefore, we would like to suppress these local modes. But, we must do so in a way which does not adversely affect the lateral modes.

Frequencies and Participation

	Mode	Frequency (Hz)	Period (Sec)	SX Par...	SY Par...	SZ Par...
1	1	3.3247	.3008		5.6843	
2	2	5.2368	.191		2.5007	
3	3	5.7947	.1726		17.5071	.0267
4	4	6.1114	.1636	.0231	.0129	93.2254
5	5	6.297	.1588		.3644	
6	6	6.6917	.1494		8.2476	
7	7	7.1929	.139		.0807	.0101
8	8	9.0559	.1104		2.0097	.0732
9	9	9.6271	.1039		18.3636	.037
10	10	9.7991	.1021		1.0585	.0391
11	Totals:			.041	55.8296	93.4144

One way to do this is to remove some of the mass from the locations which contribute to local modes. But, we want to do this in a way which still preserves the overall spatial distribution of the slab. This way we will still capture the global Torsional modes of the model. We have already done that by creating LC#2 (Discretized Self Weight).

Click Solve –Dynamics from the Main Menu toolbar to bring up the Dynamic solution options. Set the LC for dynamic mass to LC#2 (Discretized Self Weight) and request 10 modes. Click Start Solution to bring up the dynamic results shown below:

Frequencies and Participation

	Mode	Frequency (Hz)	Period (Sec)	SX Par...	SY Par...	SZ Par...
1	1	6.2035	.1612	.0296		99.6674
2	2	16.6379	.0601	67.1201	.0364	.1704
3	3	25.5322	.0392	27.0841		.1198
4	4	48.2336	.0207	.093	.9167	
5	5	49.2332	.0203		50.2533	
6	6	49.2791	.0203		1.2214	
7	7	49.303	.0203		.2818	
8	8	49.3597	.0203			
9	9	50.7684	.0197	.5734	1.7854	.0224
10	10	51.6777	.0194	.0334	.9802	
11	Totals:			94.9336	55.4812	99.9878

Notice how the Z direction mode has essentially the same basic period / frequency as before. We were also able to achieve 90% mass participation in both directions within 3 modes. Using the true self weight, it would have taken approximately 50 modes.

Notes: