



# The RAM Structural System™

# Tutorial

**Version 11.0**

July 2006

RAM International
2744 Loker Avenue West, Suite 103
Carlsbad, CA 92010
Telephone: (760) 431-3610
Toll Free: (800) 726-7789
Fax: (760) 431-5214

DAA036840-1/0001

# DISCLAIMER

The software and related documentation, including this documentation, are protected by both United States copyright law and international treaty provisions. Any unauthorized copying or reproduction is strictly prohibited and subject to civil and criminal penalties. Please refer to the License Agreement for authorization to make a backup copy of the software. You may not sell this software or documentation or give copies of them away to anyone else.

Except as expressly warranted in the License Agreement, RAM International disclaims all warranties, expressed or implied, including but not limited to implied warranties of merchantability and fitness for a particular purpose, with respect to the software, the accompanying written materials, and any accompanying hardware. All results should be verified to the user's satisfaction. The contents of these written materials may include technical inaccuracies or typographical errors and may be revised without prior notice.

Copyright attribution: © 2006, Bentley Systems, Incorporated. All rights reserved.

Trademark attribution: RAM Structural System, RAM Manager,

RAM Modeler, RAM Steel, RAM Frame, RAM Foundation and RAM Concrete are either registered or unregistered trademarks or service marks of Bentley Systems, Incorporated or one of its direct or indirect wholly-owned subsidiaries. Other brands and product names are trademarks of their respective owners.

# RAM Structural System Tutorial

## Table of Contents

# Introduction

The RAM Structural System can be used to design nearly every structural component of a building structure, from the foundations to the gravity beams and columns to the lateral framing system. This tutorial provides you with step-by-step instructions for using the RAM Structural System. Every attempt has been made to create a tutorial that addresses the many different uses of the program, but some features are still not covered. Refer to the on-line documentation for each of the program design modules for more complete information. The tutorial includes a chapter for each design module. If you do not own licenses for all of the modules, or if you do not wish to perform every chapter, then you can skip sections that do not apply. If you do not wish to model the structure from scratch, a completed model has been included in the installation. Just open the file called RAMTutorial_v10_US.rss in the default Data directory and skip to the design chapter that you are interested in.

Throughout this tutorial, action items are made in bulleted lists. References to menu commands are made in bold text and a hyphen between words indicates a sub-menu selection. For example:

- Select **File - New**.

Means to click on the "File" menu at the top of the screen and from the sub-menu items pick "New". When you are expected to type specific the words in a particular edit box the instructions are written in courier font as follows:

- Type `MyTutorial` in the File name field.
- Click **[OK]**.

The term "Click" is used to indicate a single left-click with the mouse and "Fence" is another way of saying hold the left mouse down while you window an area. It should be noted that nearly every menu command that is available in the program is also available as a toolbar icon. These buttons allow you to access the various commands more rapidly, saving time. To guide you, each button is equipped with a pop – up flag indicating its function.. This will appear as you position the cursor over the icon.

The tutorial is written for use in with both the English (Imperial) and SI systems of units. When input is required by the user the English value will be given followed by an SI equivalent in parenthesis. For the English units model the IBC 2003, AISC-LRFD and ACI 318-02 codes are implemented. For the SI model, the latest British codes are used.

## *RAM Structural System Outline*

The RAM Structural System is an assembly of several different RAM International programs. These individual programs each provide different capabilities and functionality to the RAM Structural System. The RAM Structural System is comprised of up to eight programs, all of which are described below:

### RAM Manager

The RAM Manager is the hub through which all of the RAM Structural System (RAM SS) design modules are activated. Using the RAM Manager you will create or open your structural model and establish your global criteria (such as units). Some of the output generated by the other RAM SS programs can also be printed from the RAM Manager. Whenever you open one of the design modules, the RAM model or database is locked until you close that module and return to the RAM Manager. Traffic signals in the center of the RAM Manager screen indicate the status of the current model.

### 3-D viewer

The 3-D viewer is a program that can be launched directly from RAM Manager or from RAM Modeler once a file has been opened. It allows you to view the entire structure in 3D with a variety of view controls. Some of the design modules (e.g. RAM Steel Column and RAM Concrete) have been incorporated into an interface that is identical in appearance to the 3-D viewer. No software license is required to view the model in 3-D.

### RAM Modeler

The RAM Modeler is used in the creation and modification of all RAM Structural System models. Modeling a structure from scratch is done by defining the floors or levels and then specifying the floor-to-floor heights in the story data. The program is set up for the layout of building structures but with a little ingenuity other types of structures can also be modeled.

### RAM Steel Beam Design

This module is a powerful tool to perform a rapid, interactive, design of all your steel gravity beams using a variety of steel design codes. The program optimizes steel composite and noncomposite beams, open web steel joists and Smartbeams$^{TM}$. Tributary loads for each member are automatically

calculated by the program based on the geometry of the members and the decking. The program can also be used to evaluate user selected members.

## RAM Steel Column Design

This module designs of all your steel gravity columns and base plates. Tributary loading, including the effects of pattern loading is automated and the columns are checked not only for axial loading but for the effects of connection eccentricity as well. The Steel Column module and the Steel Beam module are always licensed together and are sometimes referred to simply as RAM Steel.

## RAM Frame

This module provides an interactive lateral analysis of the structure with braced frames, moment frames or shear walls in any material. It is further subdivided into sub-modules which are licensed separately. These include the **Steel Standard Provisions** mode and the **Steel Seismic Provisions** mode for aiding in the design of steel lateral systems as well as the **Drift Control** mode for performing a Virtual Work analysis of the structure to determine the hardest working components of any system. A special interface for reviewing shear wall forces in more detail referred to as the **Shear Wall module** can be launched from the RAM Frame as well.

## RAM Concrete

This module focuses on the analysis and design of one-way concrete framing systems. It can design all of the gravity resisting concrete beams and columns as well as concrete moment frames when used in conjunction with RAM Frame. The exact loading of each member is automatically calculated through finite element analysis with options for patterning the live load and retrieving lateral forces from RAM Frame. Similar to RAM Frame, the RAM Concrete design module is broken up into separate modes for **Concrete Analysis, Concrete Beam** design and **Concrete Column** design. When designing flat slab structures (post-tensioned or conventional reinforcing), the user can also import the concrete column reactions from a **RAM Conce*pt*** file and use those forces in the Concrete Column design. Only the US concrete codes are incorporated in RAM Concrete at this time.

## RAM Foundation

This module performs an interactive design of concrete spread footings, continuous footings and pile cap foundations. Loads are automatically

calculated based on the results of the gravity and lateral analysis. RAM Foundation requires at least one of the analysis modules above (RAM Steel, RAM Frame or RAM Concrete).

## Links with Other Programs

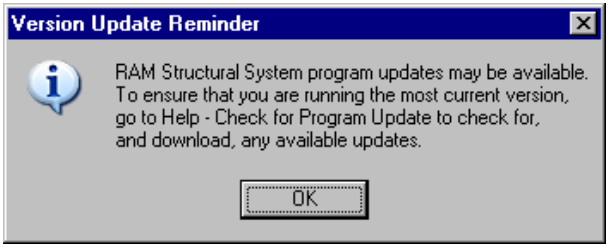
In addition to the design modules outlined above, RAM Structural System models can also link with other programs. Here's a list of the available external links to software from RAM International:

- A completed model may be exported into **RAM Advans*e*** for general purpose finite element analysis.
- Connection design for shear, moment and gusset plate connections can be performed on a RAM SS model using **RAM Connection**.
- Concrete floors can be exported into **RAM Conce*pt*** for 2-way slab design with or without post-tensioning. The revised column forces from the RAM Conce*pt* model can then be re-imported for the final Concrete Column Design.
- Mat Foundations can also be exported into **RAM Conce*pt***.
- The **RAM CADstudio** program can be used to generate high quality drawing files of the RAM Structural System model with electronic coordination between analysis and drafting models throughout the design cycle.
- The **RAM BasePlate** program can be used to design the base plates for lateral steel columns with considerations for shear and moment at the base.

Furthermore, certain 3rd party software can also be linked with RAM Structural System models. For more information on our partner companies, please visit our website at www.ramint.com.

# RAM Manager

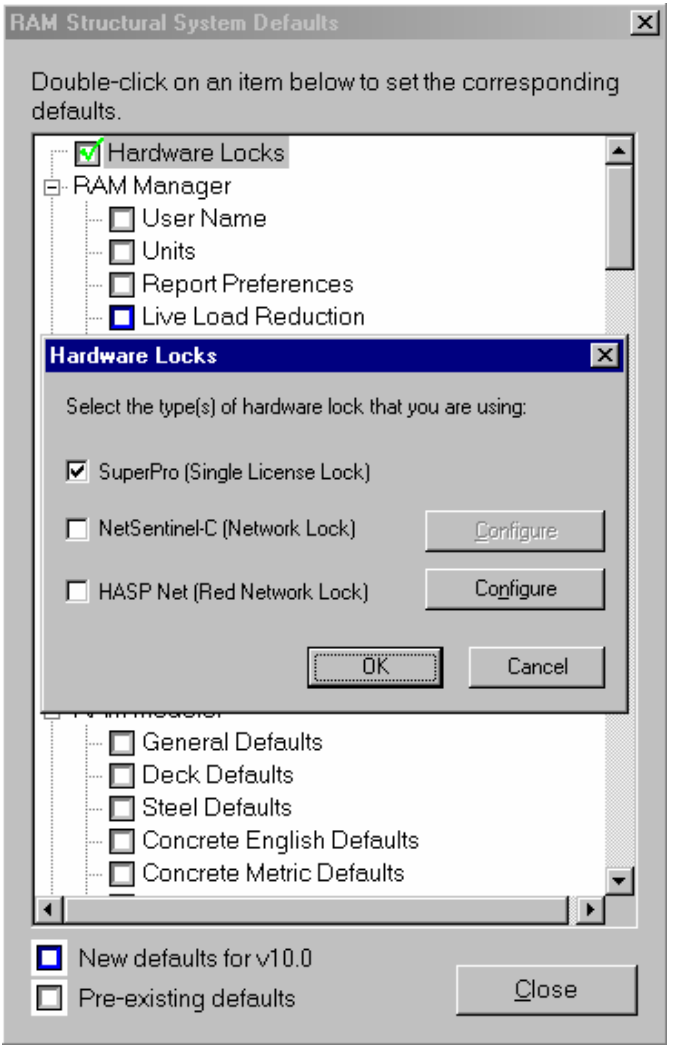
To begin the tutorial, double click on the RAM Structural System icon on the desktop. The screen that appears is the RAM Manager program. In the middle of the screen there should be a RAM International logo which indicates that you are running version 10.0 (or later). You may receive a Version Update Reminder when you start the program.




- Click **[OK]** to the warning then select **Help – Check for Program Update**… to initiate an automatic installation of the latest RAM SS patch. If you do not currently have internet access then you can download the patch from our website at another time and run it later.

Before you even create your first model you can adjust the program defaults. The defaults cover everything from User Name, to Steel preferences and beyond. If you had another version of RAM SS prior to installing version 10, then those previous default settings should still be set. With each new version there are new defaults for new features which should be checked however.

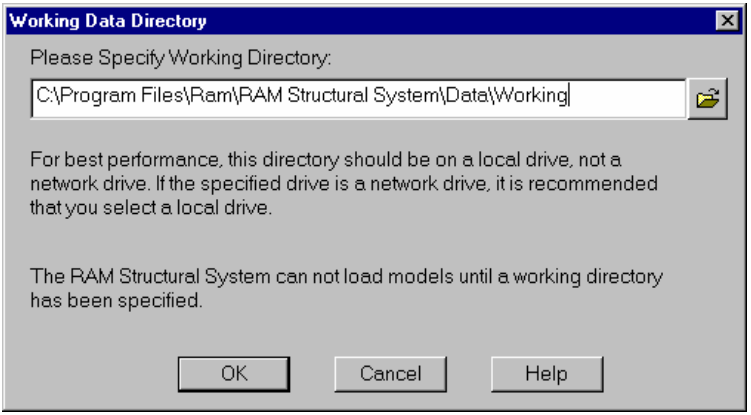
- Select **Tools – RAM Defaults Utility**. A message pops up indicating that the defaults only affect new models. For existing models, the various criteria menu options throughout the program modules can me used to modify the model. Click **[OK]** to close the warning.
- In the Default utility double-click on the option labeled Hardware Locks.



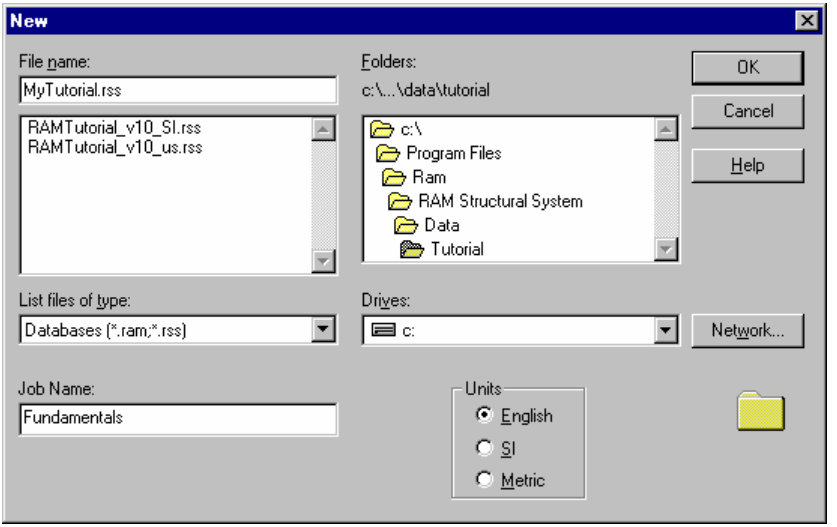

- In the Hardware Lock options make sure that the appropriate lock type is selected. If you have any difficulty with the hardware lock please refer to the installation documentation provided on the installation CD.
- Feel free to adjust any of the remaining defaults.

Once that is complete it's time to create your tutorial model.

- Select **File - New**.
- If this is the first time using the program then the following dialog box will pop-up asking you to select a working directory. This is where temporary files will be saved while you are working on a RAM Model. The directory should be a local directory on the hard drive with complete read and write privileges. It needs to be set only once.




- If the default working directory is not what you would like, click the open folder icon on the right and then browse to a directory you do want to use and Click **[OK]**.




- When the "New" dialog box opens, type `MyTutorial.rss` in the File name field.
- In the Job Name field type `Fundamentals`.
- Select the desired Units. Note; the units you select now set all the defaults for the model you are creating. There are a complete set of English (Imperial) defaults and another set of Metric/SI defaults which can be set in the Defaults Utility. There are subtle differences between SI and Metric units, but the default values are all the same. When using "English" units you can switch between feet and inches while modeling.
- Click **[OK]**.

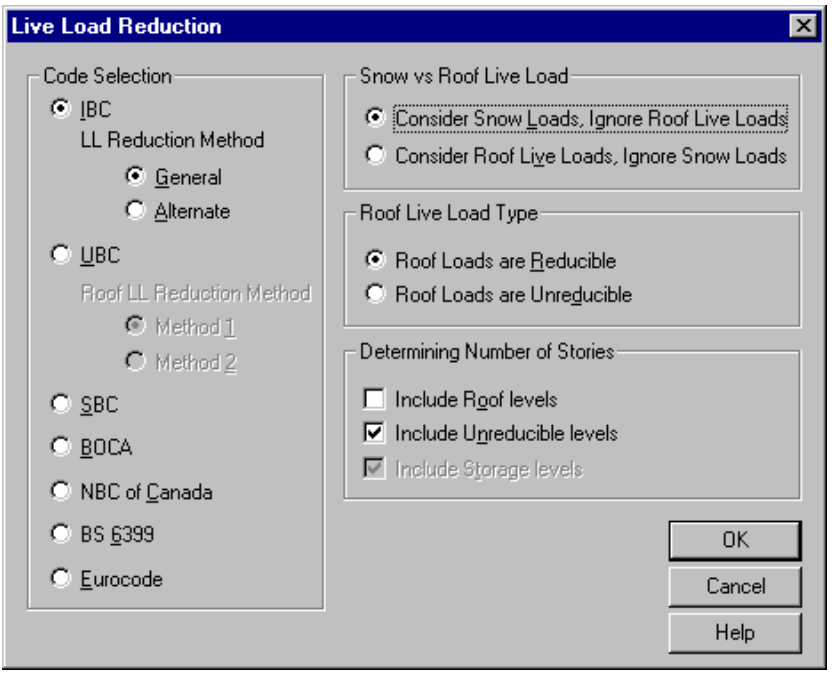
## *Selecting Criteria*

There are several different criteria that can be set from the RAM Manager. These typically have an impact on all of the various design modules. Other, more specific design criteria are set within the individual design modules.

### Live Load Reduction

To pick the appropriate code for Live Load Reduction:

- Select **Criteria - Live Load Reduction**




- Select IBC LL reduction Method – General
- Under Snow vs. Roof Live Load select "Consider Snow Loads, Ignore Roof Live loads". While the program will allow you to model both kinds of loads it can only consider one type or the other in the design at one time.

Before clicking [OK] to this dialog box, notice that there are several Building Codes available for Live Load Reduction, including SBC, BOCA, IBC, NBC of Canada, BS 6399 and Eurocode. Selecting certain codes will

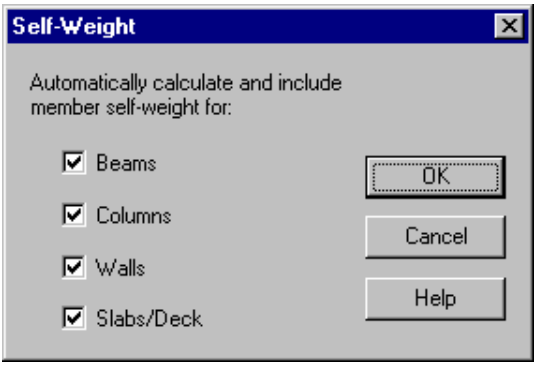
make additional criteria specific to these building codes available in the Criteria Menu.

Also, there are three check boxes under Determining Number of Stories. If you have not changed the installation defaults, the bottom two check boxes will be checked. This setting is used by the program to determine how many levels are supported by a column. Some codes establish a maximum Live Load Reduction depending on the number of floors supported by the column and differentiate between floor levels and roof levels. For more information on the use of this setting, refer to the RAM Manager on-line documentation.

## Self – Weight Options

To control whether or not the program automatically includes member self weight as part of the Dead Load case:

- Select **Criteria - Self-Weight**.



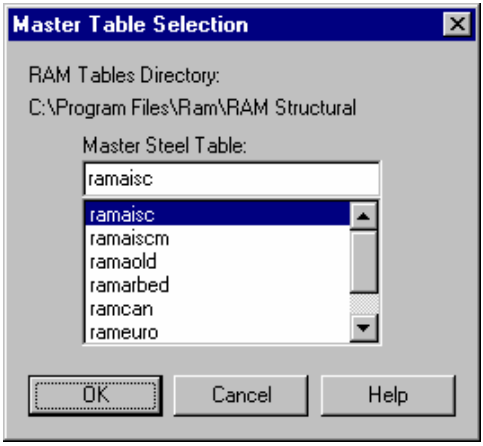

- Check all four items and click **[OK]**.

Note; the program will now apply a vertical load to all members based on self-weight. The self weight forces generated by the program are automatically included as part of the Dead Load case (downward loads). The masses used in the seismic force determination and dynamic behavior are assigned separately in the modeler and an estimated mass for beams, columns and slabs must always be input as part of the loads. If the self weight of walls is checked above, then the mass of gravity and lateral walls will be included in the building mass based on the full story height. Calculated wall dead load and mass are not adjusted for openings in the walls. If your walls have a large percentage of opening then you should manually apply appropriate line loads rather than using self weight.

## *Selecting Tables*

The program references tables in the analysis and design of all models. The **Master Steel Table** includes all section designations and section properties for all available steel shapes. The installed master tables reflect the current state of available sections. When changes to the section properties or availability are made by AISC or other governing organization we make every attempt to update the files accordingly. You may modify the master tables yourself with any text editor (e.g. Microsoft Word Pad), but we recommend that you make a copy of the file for alterations first, as the default files will be overwritten when the program is reinstalled. Master table files have the extension ".TAB" and can be found in the Tables subfolder wherever the RAM Structural System was installed. For more information on table format, please refer to the RAM Manager on-line manual.

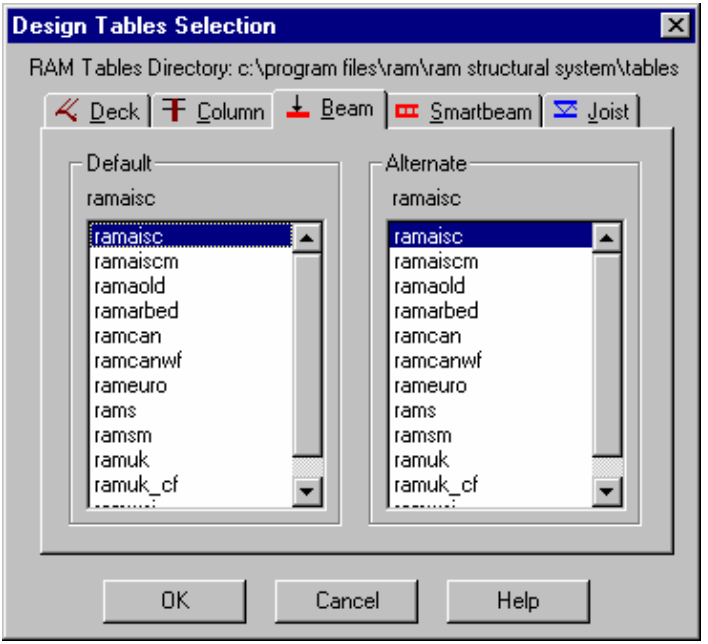
- Select **Criteria - Master Steel Table**.



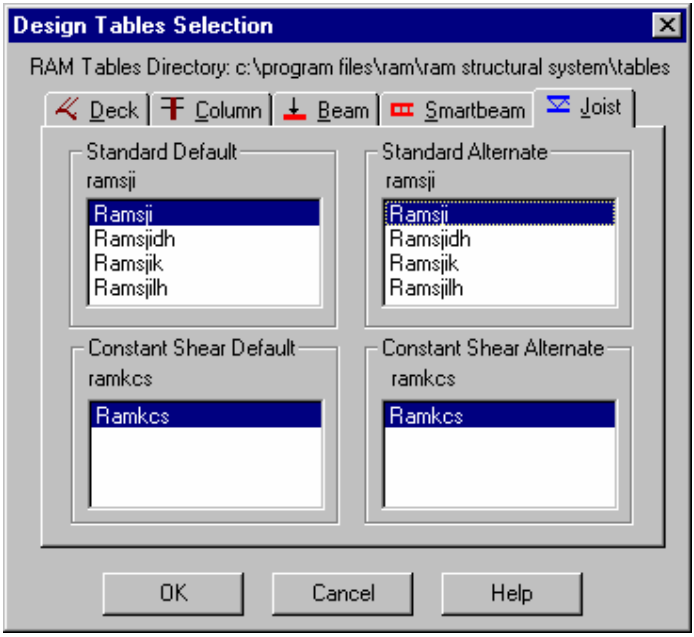

- Select RAMAISC (RAMUK) as the master table and Click **[OK]**.

Besides the master table of sections, the program also utilizes **Design Steel Tables** in the optimization of the steel members. There are Design Steel Tables for Deck, Column, Beam, Smartbeam and Joist; their extensions are ".DCK", ".COL", ".BMS", ".CAS", ".JST" (and ".KCS") respectively. The Design Steel Tables include a listing of the sections in a particular order which affects the program optimization. You can customize these tables to meet your specific needs or create new tables. Again, the RAM Manager on-line documentation has a complete description of the format of the design tables.

- Select **Criteria – Steel Design Tables**.
- Under the Decks tab choose RAMDECKS (RAMUK).
- Under the Column tab choose RAMAISC (RAMUK).
- Under the Beam tab choose RAMAISC (RAMUK) for both default and alternate tables.

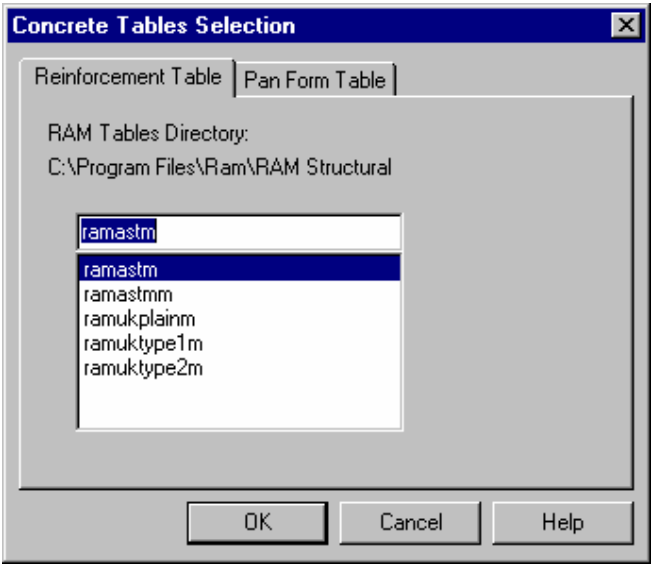



- Under the Smartbeam tab select RAMSMI for all four tables.
- Under the Joist tab select RAMSJI for the two standard tables and RAMKSC for the two constant shear tables and Click **[OK]**.




For concrete members, the gross dimensions are always created directly in the model, but there is a **Concrete Table** which contain a list of reinforcement bar designations (names) along with cross sectional areas and diameters. All the reinforcement in the table will be available for use in the design of both concrete beams and columns. The **Pan Joist Table** contains a list of common pan sizes for use in pan joist construction. The reinforcement tables have extension ".REN" and the pan joist tables have extension ".PNJ".

- Select **Criteria - Concrete Tables**.
- Under Reinforcement Table select RAMASTM (RAMUKTYPE1M)
- Under Pan Form Table select RAMCECO and Click **[OK]**.




## Selecting Units

The units for any model may be altered at any time. Changing the units does not affect the physical model size of an existing model, it only alters the reporting that the program does.. It is important to note that when you create a new file and pick the units you are also selecting which set of defaults to use. Changing the units after the model is created **does not affect which set of defaults the program uses**.

- Select **Criteria - Units**.
- Select English (or SI).
- Click **[OK]**.

## Additional Commands

There are several other additional commands and settings which do not need to be invoked for the tutorial model at this time. These may be found under the Post-Processing and Tools menus in the menu bar. They include more criteria settings, report styles defaults, and export options, etc. Feel free to examine or adjust the other items now.

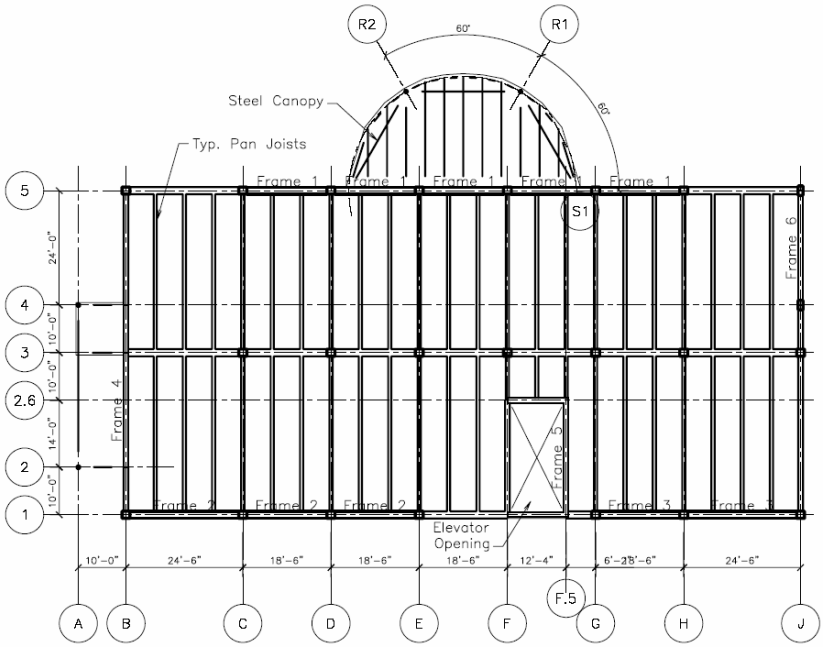
# RAM Modeler

During this section of the tutorial, you will construct a 4 story building with a variety of member types and configurations. You will start by creating the 2nd Floor Plan as shown below. Then you will copy and edit that data to create additional floors. The model is intended to show many of the capabilities of the software all in this one model and as such may be somewhat eclectic.

To invoke the RAM Modeler, from the RAM Manager Menu Bar,

- Select **Model** (or click the Square button labeled Model in the upper left corner).

The lowest level of the tutorial model is depicted below for your reference. Refer to it when the written instructions are not clear.



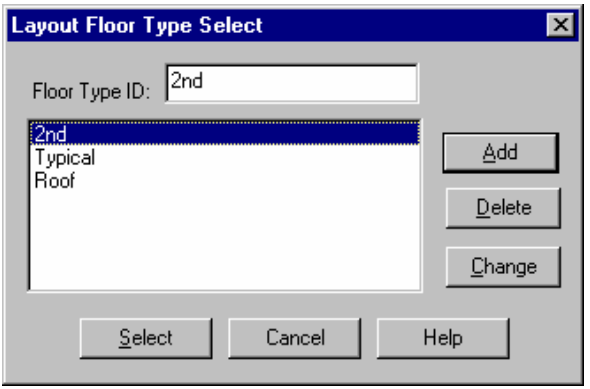

2nd Floor Plan Figure

## *Floor Layout Type*

Each unique floor should be modeled as a different "layout type" in the RAM Modeler. The power of the Floor Layout Type is that it allows the program to consider floor framing layouts the same way the Engineer considers floor plans in construction drawings. That is, a typical floor layout may occur at multiple levels in the structure. The RAM Structural System takes advantage of this same practice by employing floor layout types. You will need to create at least one floor layout type for every model you create.

To create and select a floor type:

- Select **Layout - Type - Select**.




- Type `2nd` for Floor Type ID and Click **[Add]**.
- Type `Typical` for Floor Type ID and Click **[Add]**.
- Type `Roof` for Floor Type ID Click **[Add]**.
- Highlight 2nd and Click **[Select]**.

Note; from this point on all the elements that are created are associated with the floor type labeled “2nd”. The Layout Type dropdown list located in the toolbars indicates the currently active layout type. This drop down list can also be used to switch between layout types.

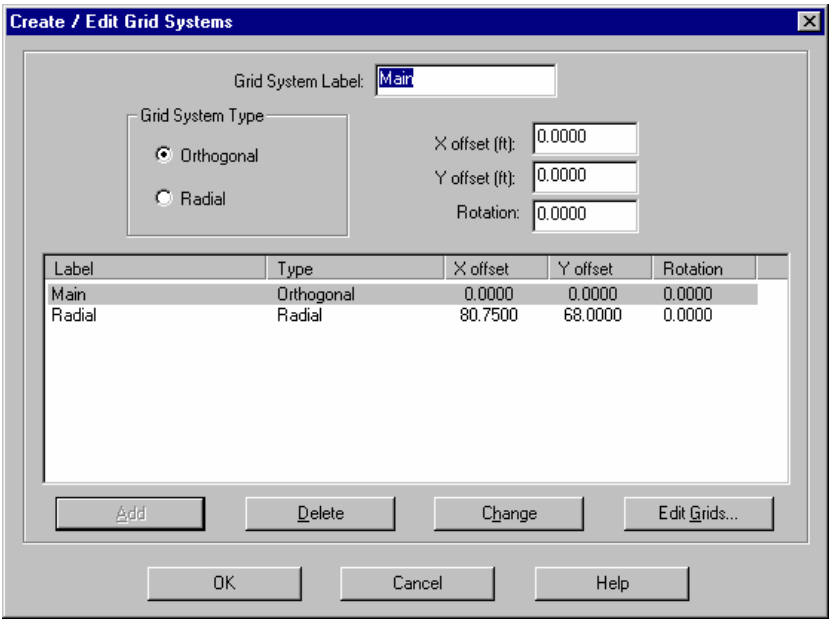
At this point, there is the opportunity to import an AutoCAD .DXF file to generate the grids, beams and columns of the current floor type. This option is not going to be used in order to illustrate the step-by-step approach instead.

## *Grid Layout*

The modeling of a structural floor plan or layout typically begins with the layout of the grid lines, just as you would start a framing plan drawing. The primary gridlines are used to locate the columns and walls. The grids can be adjusted later and the model can be automatically stretched in the process. Construction grids are a new feature ideal for locating other items like beams or loads. The grids for this model are shown in the 2nd Floor Plan figure.

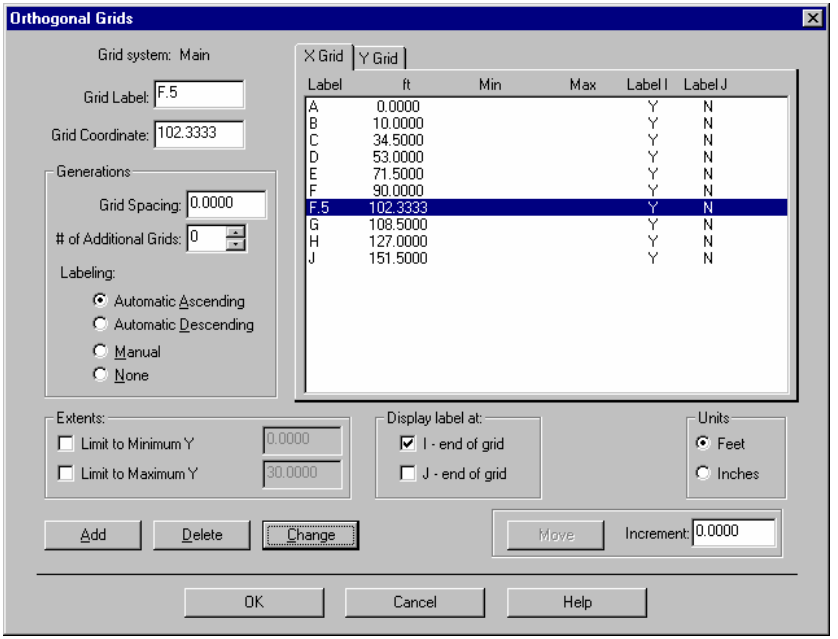
To create the Grid Systems:

- Select **Layout - Grids - Create / Edit**.



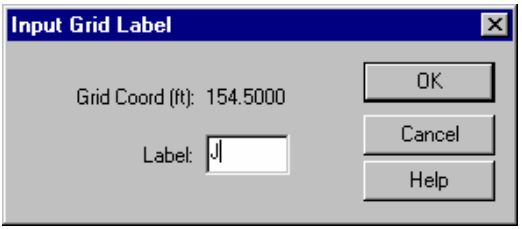

- Type `Main` for Grid System Label.
- Set Grid System Type to Orthogonal.
- X and Y offset and Rotation should all be set to 0.
- Click **[Add]**.
- Type `Radial` for Grid System Label.
- Set Grid System Type to Radial.

- Type 80.75 (24.875) for X offset.
- Type 68 (21) for Y offset.
- Type 0 for Rotation.
- Click **[Add]**.
- Highlight Main from the list box.
- Click **[Edit Grids]** and the Grids dialog box should appear. The X grids are the grids that run up-and-down the screen. They are located with a horizontal measurement from the grid system origin.

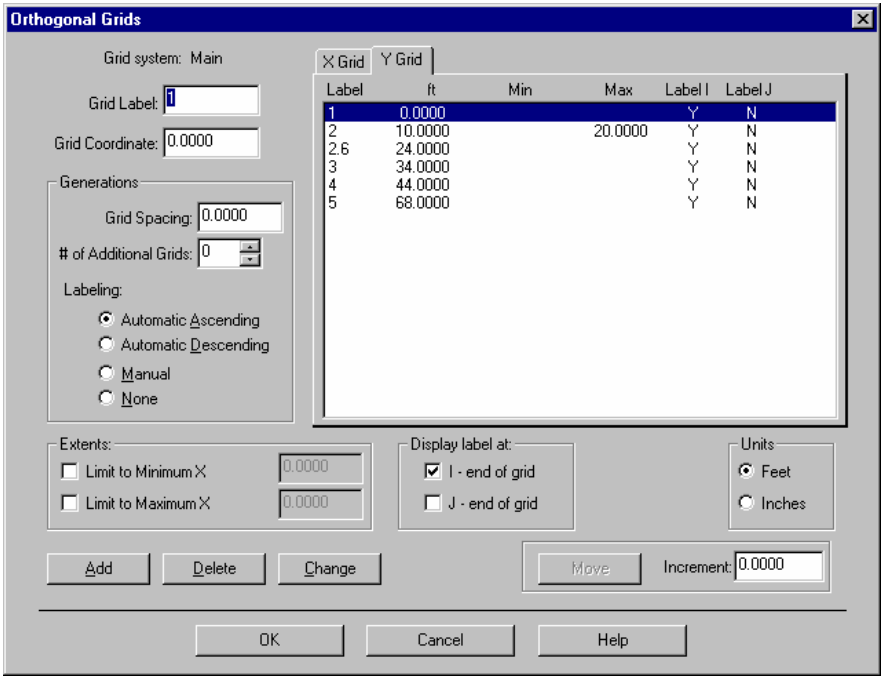



- Type A for Grid Label.
- Leave the Grid Coordinate set to 0.
- Click **[Add]**.
- Type B for Grid Label.
- Set the Grid Coordinate set to 10 (3).
- Click **[Add]**.

- Type C for Grid Label.
- Set the Grid Coordinate set to 34.5 (10.5).
- Click **[Add]**.
- While Grid C is still highlighted in the main window, type 18.5 (5.75) in the Grid Spacing field.
- Set the # of Additional Grids to 5 and set the Labeling to Automatic.
- Click **[Add]**. The program should generate grids D,E,F,G and H at once.
- Change the Grid Spacing to 24.5 (7.5)
- Set the # of Additional Grids to 1.
- Set the Labeling to Manual
- Click **[Add]**. The program will prompt you to enter the new grid Label

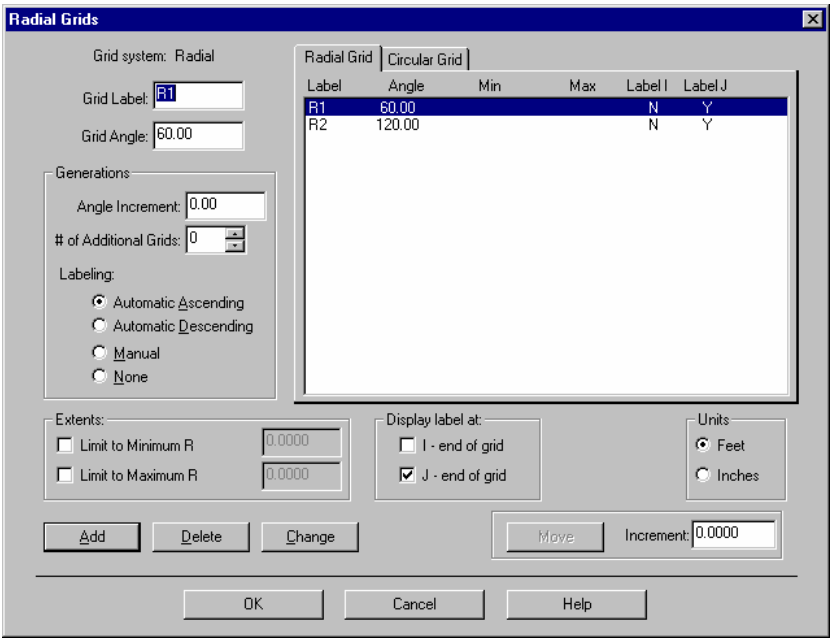



- Type J in the Label box and Click **[OK]** to add grid J.
- Set # of Additional Grids to 0 again.
- Type F.5 for Grid Label.
- Set the Grid Coordinate set to 102.3333 (31.5833).
- Click **[Add]**. Your screen should now have the grids shown above.
- Select the Y Grid tab.



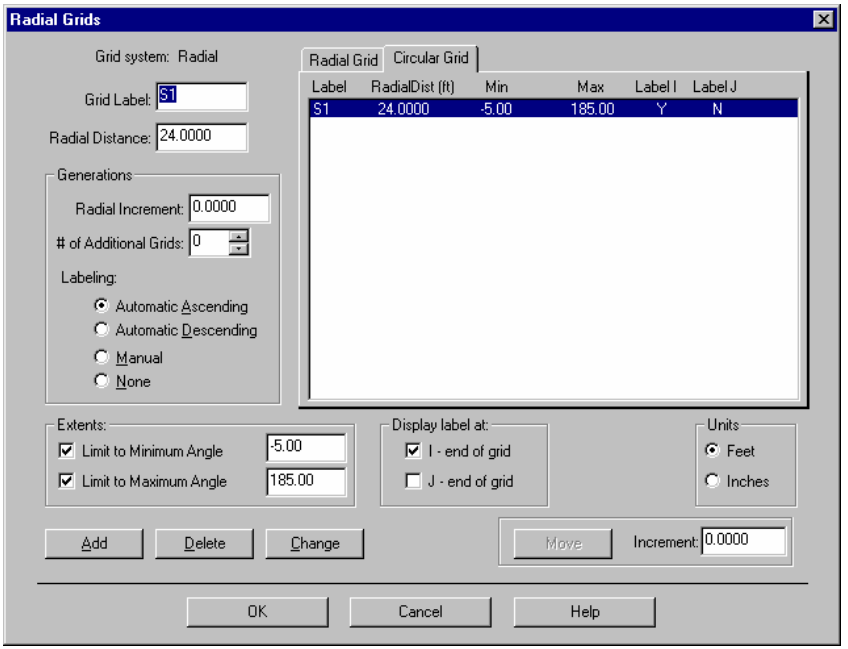

- Type 1 for Grid Label.
- Type 0 for Grid Coordinate and click **[Add]**.
- Type 2 for Grid Label.
- Type 10 (3) for Grid Coordinate and click **[Add]**.
- Type 2.6 for Grid Label.
- Type 24 (7.5) for Grid Coordinate and click **[Add]**.
- Type 3 for Grid Label.
- Type 34 (10.5) for Grid Coordinate and click **[Add]**.
- Type 4 for Grid Label.
- Type 44 (13.5) for Grid Coordinate and click **[Add]**.
- Type 5 for Grid Label.
- Type 68 (21) for Grid Coordinate and click **[Add]**.

- Before exiting the Orthogonal grids dialog box, select grid line 2 in the main window so that 2 appears in the Grid Label field.
- In the lower left corner, check the box called Limit to Maximum X.
- Type 20 (6) in the box to the right and click **[Change]**. This will cause the grid line #2 to be trimmed beyond X=20'.There is no harm in having the grid extend across the entire building, but if you have many grids close together, trimming the grids can make the modeling easier. You can also alter the side at which the bubble for the grid appears. Using the I end default results in the bubbles being along the bottom and the left sides of the plan.
- Click **[OK]**.
- This takes you back to the Create / Edit Grid Systems dialog box.
- Highlight Radial from the main list box.
- Click **[Edit Grids]**. The Radial Grids dialog box should appear:




- Type R1 for Grid Label.

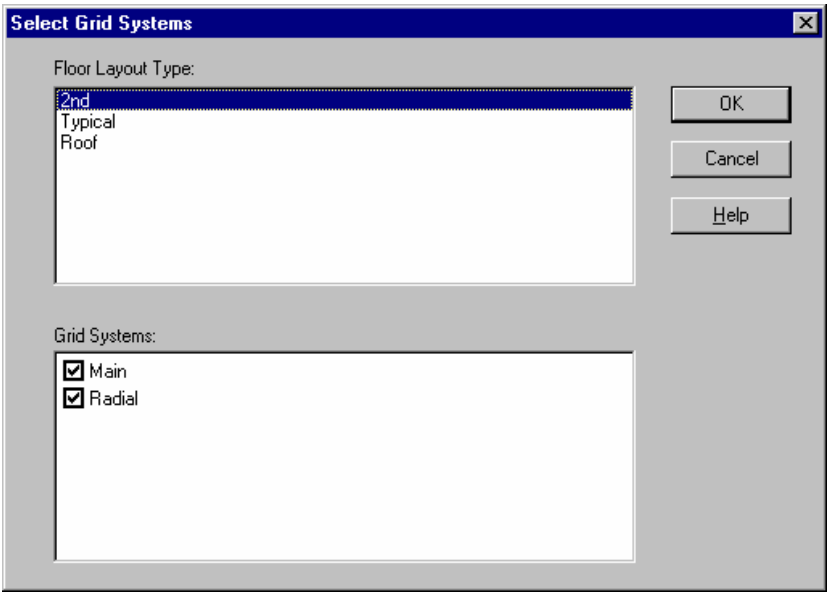
- Type 60 for Grid Angle and click **[Add]**.
- Type R2 for Grid Label.
- Type 120 for Grid Angle and click **[Add]**.
- Select the Circular Grid tab.




- Type S1 for Grid Label.
- Type 24 (7.5) for Radial Distance.
- Check the Limit to Minimum angle and type -5 for the angle.
- Set the Limit to Maximum Angle at 185 and click **[Add]**.
- Click **[OK]** to go back to the Create / Edit Grid Systems Dialog box.
- Click **[OK]** to finish the grid creation. If you made any mistakes, you can reopen the grids control and change the coordinates or other property of the grids. Just select the grid first, make the desired changes and click **[Change]**.

To select the grid systems to be used for the various levels:

- Select **Layout - Grids - Select**. The Select Grid Systems dialog box should appear:




- Highlight 2nd under Layout Floor Type if it is not already.
- Check both Main and Radial boxes under Grid Systems.
- Click **[OK]**. Your screen should now show the grids as depicted in the 2nd Floor Plan figure at the beginning of this chapter.

It is a good idea to save your work periodically, to do so now:

- Select **File - Save**.

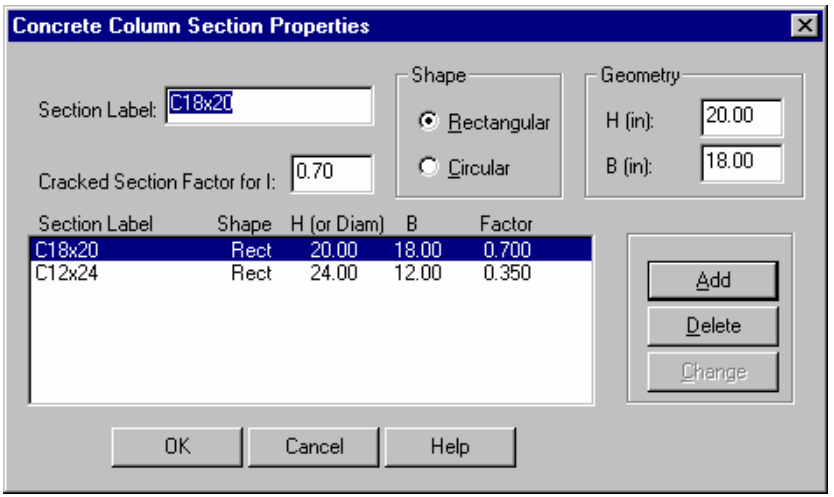
## *Concrete Beam and Column Section Properties*

The 2nd Floor Type consists of a combination of different concrete members. To prepare for laying out the beams and column, the Material must first be set to Concrete. To do this:

- Select **Material – Concrete** or pull down the material drop down list and select "Concrete" (it likely indicates "Steel" initially).

To enter Column Sections:

- Select **PropTable – Column Sections** and the Concrete Column Sections Dialog will appear.



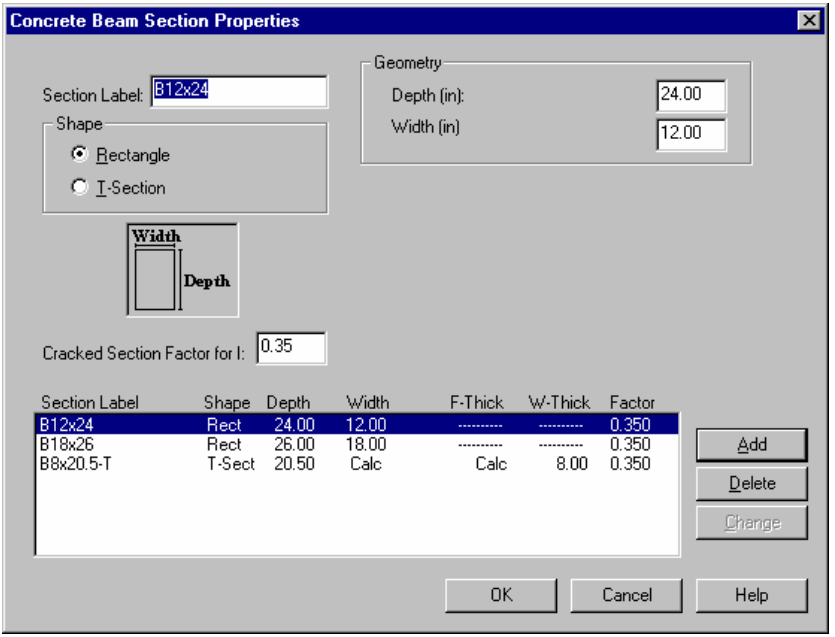

There are two concrete column sizes required for this model:

- Type `C18x20 (C45x50)` for Section Label.
- Enter 20 (500) for H.
- Enter 18 (450) for B.
- Click **[Add]**.
- Type `C12x24 (C30x60)` for Section Label.
- Change the Cracked Section Factor for I from 0.70 to 0.35. 0.7 is the default value for columns, but the C12x24 columns will be used like pilasters at the ends of a wall. In practice you may want to establish the cracked section factors more accurately.

- Enter 24 (600) for H.
- Enter 12 (300) for B.
- Click **[Add]**.
- Click **[OK]**.

To enter the concrete beam properties:

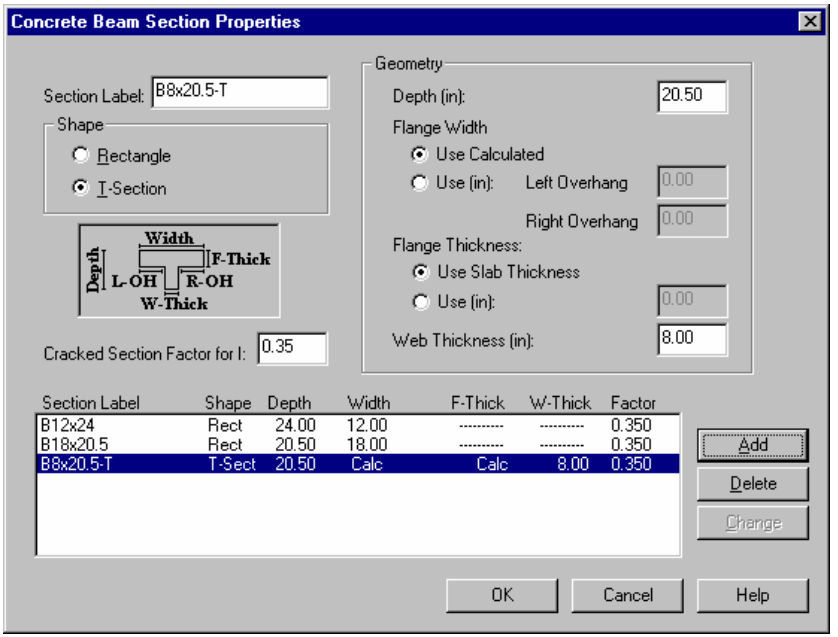
- Select **PropTable - Beam Sections**. The Concrete Beam Sections Dialog will appear.




There will be three different concrete beam sections in this model. Enter them as follows:

- Enter `B12x24 (B30x60)` for Section Label.
- Set the Shape option to Rectangular.
- Enter 24 (600) for the Depth and 12 (300) for the Width
- Click **[Add]**.
- Enter `B18x20.5 (B45x50)` for Section Label.

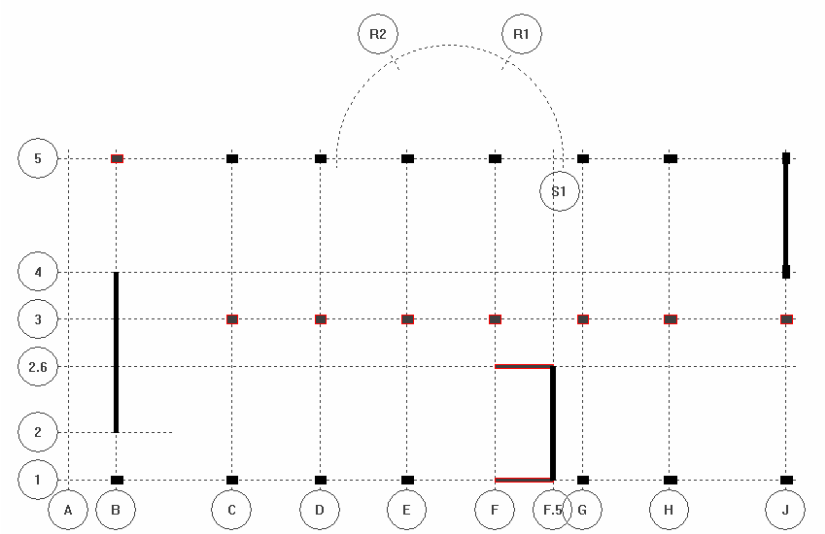
- Enter 20.5 (500) for the Depth and 18 (450) for the Width
- Click **[Add]**.




- Enter `B8x20.5-T (B20x40-T)` for Section Label.
- Set the Shape option to T-Section.
- Type 20.5 (515) for Depth
- Set the Flange Width to Use Calculated (the program will determine the individual beam widths based on the span, the distance to a slab edge or the distance to an adjacent beam, whichever controls).
- Set the Flange Thickness to Use Slab Thickness.
- Set the Web Thickness to 8” (200). Note; the program does not count any tapering of the T beam stem.
- Click **[Add]** and click **[OK]**.

## *Concrete Column Layout*

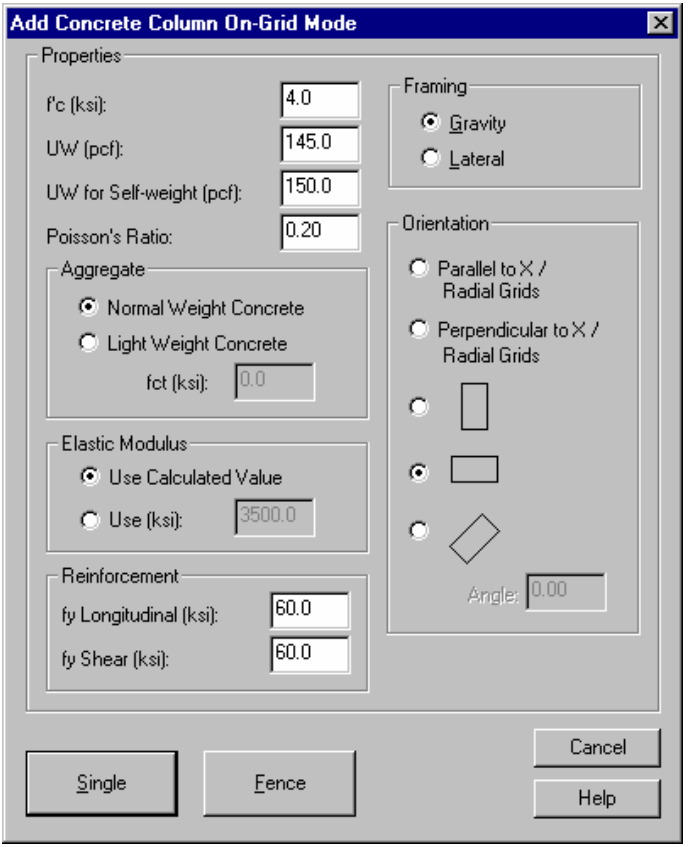
Columns can be located at the intersections of gridlines ( On-Grid) or at any other location on the layout ( Off-Grid). Throughout the program output, columns are identified by the grid intersection they are on, so it is recommended to place the column On-Grid. Placing columns On-Grid is also the best way to assure that they line up from level to level. If a column is at the confluence of three or more grid lines, you may want to stop all but two of the grid lines short of the intersection so that the program can identify the column correctly.




Concrete Column and Wall layout

To begin modeling the columns:

- Select **Layout - Columns - Add On-Grid**. The Add Concrete Column On-Grid Mode dialog box should appear.



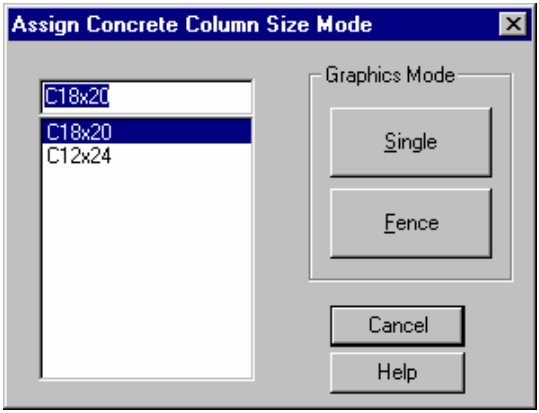

- Enter 4 for f'c (27.6 for fcu).
- Enter 145 (2320) for Unit Weight
- Enter 150 (2400) for Unit Weight for Self-Weight
- Enter 0.20 for Poisson's Ratio
- Select Normal Weight Concrete
- Select Use Calculated Value for Elastic Modulus
- For Reinforcement, enter 60 (410) for fy Longitudinal and 60 (410) for fy Shear
- Set Framing to Gravity.

- Set Orientation to ■ (this is zero degree orientation).
- Click **[Single]**. This will take you back to the graphics screen:
- With the cursor, place a column (Left Click one time) at these grid intersections: B-5, C-3, D-3, E-3, F-3, G-3, H-3 and J-3.
- Right Click with the mouse to return to the Add Concrete Column On-Grid Mode dialog box (right clicking in the Modeler always returns you to the previous dialog box):
- Change Framing from Gravity to Lateral.
- Click **[Single]** to return to the graphics screen:
- Place lateral columns by clicking at Grids B-1, C-1, D-1, E-1, G-1, H-1, J-1C-5, D-5, E-5, F-5, G-5, and H-5. Do not place columns at F-1 or F.5-1.
- Right Click to return to the Add Concrete Column On-Grid Mode dialog box.
- Change Orientation to ▌ (90 degrees).
- Click **[Single]** to return to the graphics window.
- Place 2 lateral columns at the ends of the wall on Grid J.

Note; if you accidentally place a member with an incorrect setting, you can undo the last command Using Edit – Undo. If it's too late to undo the command, then you can delete the erroneous column and model it again, or you can fix it by using the Change Properties command. This is covered later in the tutorial.

You can now assign concrete column section sizes.

- Select the **Layout – Columns – Assign Size** Command. The Assign Concrete Column Size Mode dialog box will appear.



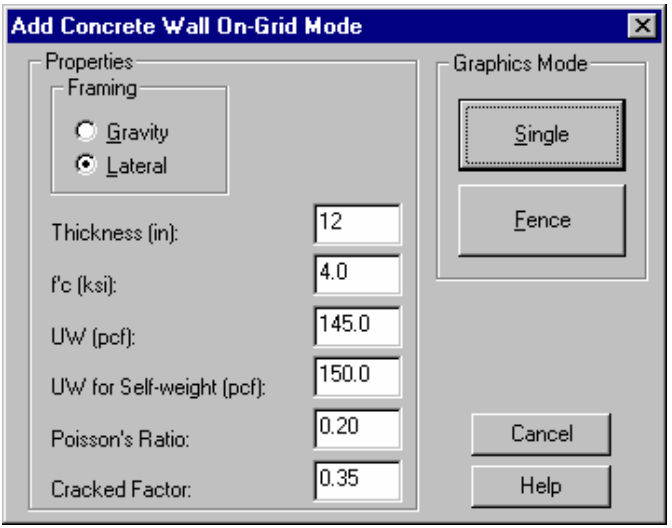

- Select C18x20 (C45x50) from the list box and click **[Fence]**. This will take you back to the graphics screen:
- With the cursor drag a window large just enough to encompass all of the columns. Window selections in the modeler only work on items entirely within the window.
- Right click to return to the Assign Size command.
- Select the C12x24 (C30x60) from the list and click **[Fence]** again.
- This time select only the tow lateral columns at the ends of the wall on Grid J. This will override the previous assignment.

This completes the layout of concrete columns on the 2nd Floor. We have a few steel columns to add, but we are going to put the concrete walls in first.

## *Concrete Wall Layout*

Like columns, walls can be placed on or off of the established grids. You will only use the On-Grid feature in this section. Walls can only be placed while the current material is set to either “Concrete” or “Other”.

- Select **Layout – Walls – Add On-Grid**. The Add Concrete Wall On-Grid Mode dialog box should appear:




- Set Framing to Lateral.
- Type 12 (300) for Thickness
- Type 4 for f’c (27.5 for fcu).
- Type 145 (2400) for UW (this is used to calculate E).
- Type 150 (2400) for UW for Self – Weight (this is only used if the RAM Manager self weight criteria has walls checked).
- Type 0.2 for Poisson’s Ratio
- Type 0.35 for Cracked Factor
- Click **[Single]** This will take you back to the graphics screen:

- With the cursor, place a wall by clicking once at the beginning and once at the end of the wall. Start by clicking at grid B-2 and then stop at grid B-3. Then place a second wall continuing north from B-3 and stopping at B-4. This wall is being placed in two pieces because the braced frame above has a columns at B-3. Otherwise, a single wall segment would have sufficed.
- Add another wall from Grid F.5-1 to F.5-2.6.
- Add another lateral wall from J-4 to J-5.

Note; in the modeler, walls and columns are placed according to the center location. In reality a concrete wall between two pilasters may be shifted slightly from the column centerline or gridline. For simplicity of modeling and analysis, it is highly recommended to place the columns and walls on the same centerline regardless.

- Right click to return to the Layout Walls dialog box.
- Change the framing from Lateral to Gravity.
- Click **[Single]** to return to the graphics mode and add two walls from F-1 to F.5-1 and from F-2.6 to F.5-2.6. The screen should then look like the figure below.

Note; these “flange walls” could be modeled as either gravity members or lateral members. As gravity members they will not participate in the lateral finite element analysis in RAM Frame. Consequently, the wall on grid F.5 will resist the lateral forces in the north south direction directly. If these two walls were modeled as lateral elements instead, then the three walls together would act like a big “C” shape. Such a wall group would be significantly stiffer in that case.

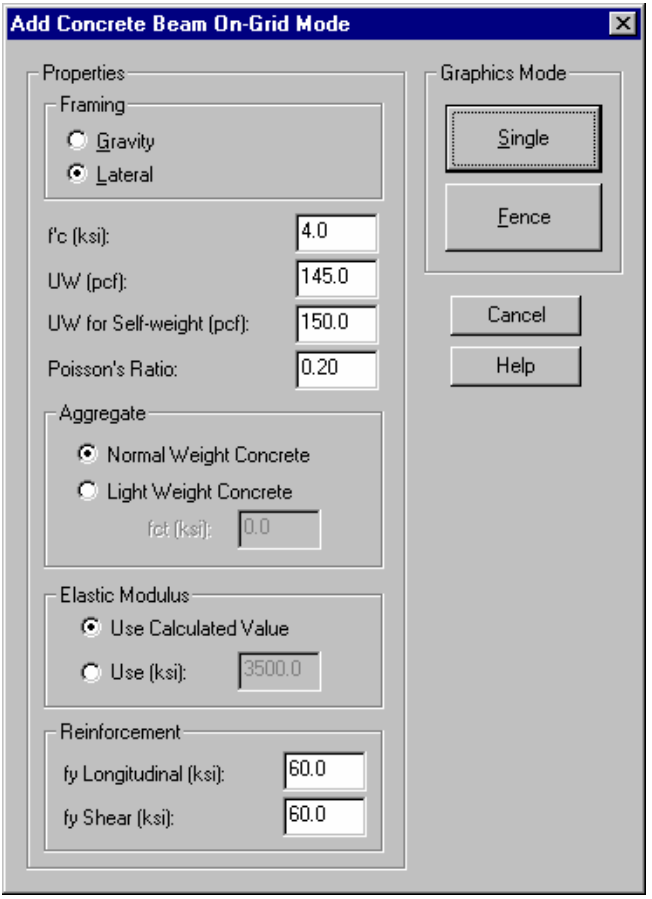
## *Concrete Beam Layout*

Beams can either be located between the intersection of gridlines or existing members, or Off-Grid, which is usually the case for secondary framing. Beams can also be automatically generated by the program at regular spacing. Cantilevers can be added to beams after the main span is modeled. Not all beam layout commands will be illustrated in this tutorial, but all are explained in the on-line help or RAM Modeler documentation.


Concrete Beam layout

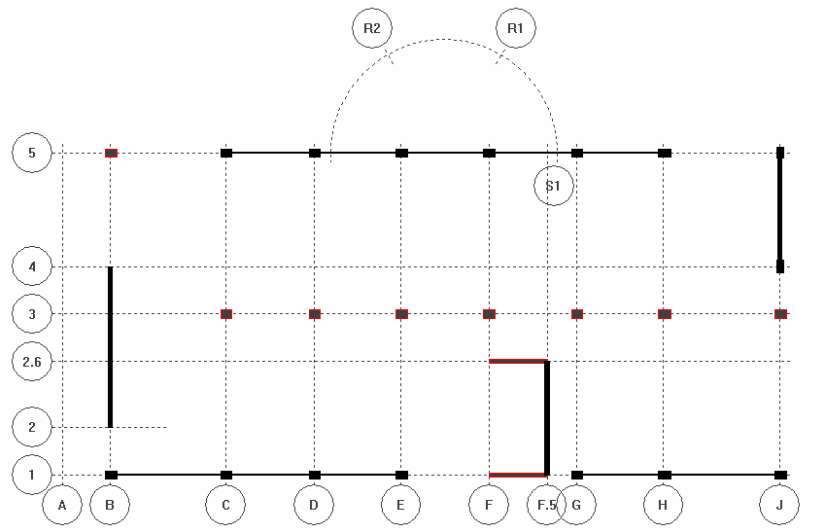
To add beams to the plan:

- Select **Layout - Beams - Add On-Grid**. The Add Concrete Beam On-Grid Mode dialog box should appear.



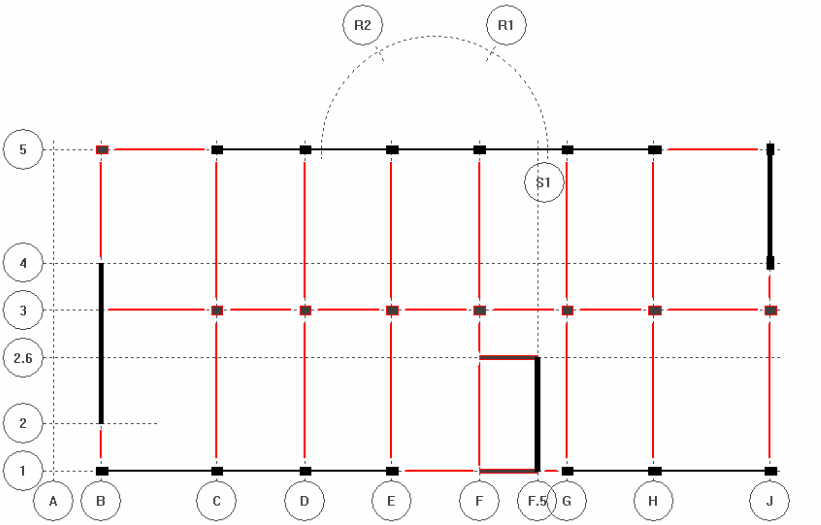

- Set Framing to Lateral.
- Type 4 (27) for f'c.
- Type 145 (1850) for UW.
- Type 150 (1900) for UW for Self-Weight.
- Type 0.2 for Poisson's Ratio
- Select Normal Weight Concrete
- Select Use Calculated Value for Elastic Modulus

- For Reinforcement, enter 60 (410) for fy Longitudinal and 60 (410) for fy Shear
- Click **[Fence]**. This will take you back to the graphics screen:
- With the mouse drag a narrow window around the columns from Grid C-5 to H-5. This should create the five beams of Frame 1.
- Drag another narrow fence around the columns from B-1 to E-1.
- Drag another fence from G-1 to J-1. This completes the lateral framing of the lowest level. At this point, please verify that the model is identical to the following figure




Lateral Concrete Beam layout

- Right click to return to the Add Beam On-Grid dialog box.
- Change the Framing from Lateral to Gravity.
- Click **[Single]** to return to the graphics window.
- Connect the all columns together with beams and connect the ends of the walls to columns with gravity concrete beams.



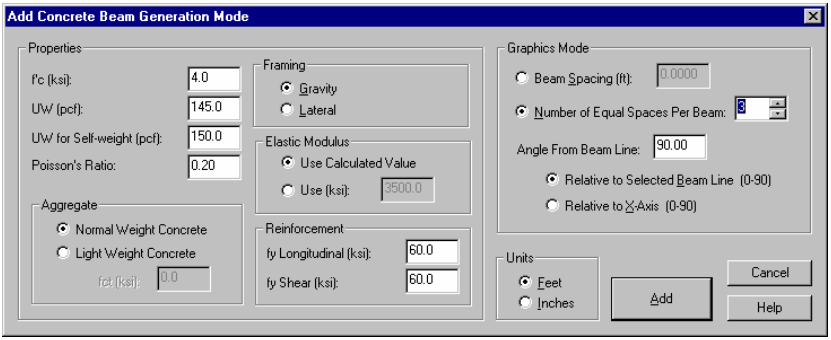

Concrete Beams On-Grid

Note; in single mode, a white line appears after selecting the start point of a beam or wall. This “rubber band” indicates the line of the before the other end is located. The same graphic also appears when laying out slab edges, deck polygons, line loads, etc.

A word about the Fence option: this will add beams from column to column (on grid) within the fenced area. If you incorrectly add a beam, it can be deleted or you can Undo the last steps. Fence mode is extremely useful for adding beams in regular structures.

Now it’s time to add the infill beams. These beams could be modeled as either beams or pan joists. In the end, there is no difference between beams and pan joists, it’s just an optional way of modeling them. When adding pan joists, you must initially assign a size to the edge beams, then specify the pan size or the spacing between the beams. If the spacing does not work out perfectly, then there will be one odd space. When laying out beams, on the other hand, it is the centerline that you are defining. Since this tutorial needs to work for two system of units, we will add beams rather than pan joists. To add the intermediate, infill beams:

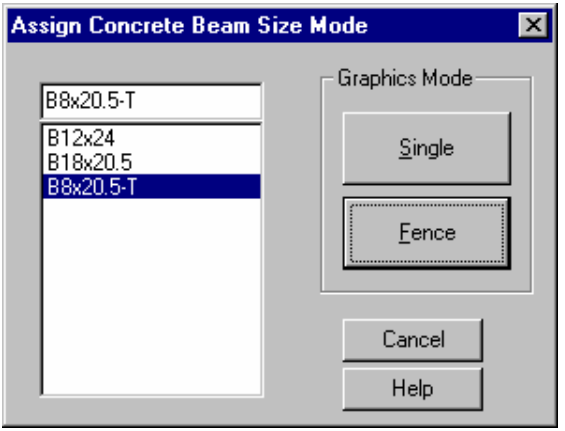
- Select **Layout – Beams – Add Generation**. The following dialog box will appear:




- Change the number of Equal Spaces per beam from 2 to 3.
- Click **[Add]** to return to the graphics window.
- Click once at the column on Grid C-3 where the generation will start from. Then click a Second time at Grid H-3 to terminate the string of beams. Then click to the North side of line 3 to project the new beams in that direction. Notice the command line at the bottom of your screen. The brief instructions typically lead you through the layout steps
- Repeat the steps above but add the beams to the south side of Grid 3.
- Right click to return to the Add Generation command.
- Change the spaces from 3 to 4 and Click **[Add]**.
- In the graphics mode click at the left, then the right end of the beam between B-3 and C-3, then add the new beams to the north side.
- Repeat for the south side and do the same for the girder from H-3 to J-3.

This completes the layout of all the Concrete members. Like concrete columns, all concrete beams must be assigned a preliminary size in order to perform a complete analysis. Before assigning sizes, it's best to turn on the size display:

- Select **Options – Show Sizes** (you won't see anything different).
- Select **Layout – Beam – Assign Size**.




- In the Assign Size dialog box, highlight the B8x20.5-T section (B20x40-T) and click **[Fence]**.
- In the graphics mode, fence the entire floor plan.
- Right click to return to the previous window.
- Highlight the B18x20.5 (B45x50) section and click **[Single]**.
- With the target cursor, select each of the girders or beams on Lines 1, 3 and 5 (a [Fence] could also be used).
- Right click to return to the previous window.
- Highlight the B12x24 (B30x60) section and click **[Single]**.
- Pick each of the perimeter beams on Grids B and J.
- Also pick the lintel beam on grid F at the opening of the elevator.
- Select **File - Save**.

That completes the layout of concrete members in the Tutorial model. If you purchased the RAM Concrete design module only, and do not have a license for RAM Steel then you should substitute concrete beams and columns for steel beams and columns in the following sections, or omit those members altogether. You are allowed to model steel members even if you don't have a license for RAM Steel, but you must assign a specific size to all of those members prior to running the model in RAM Concrete.

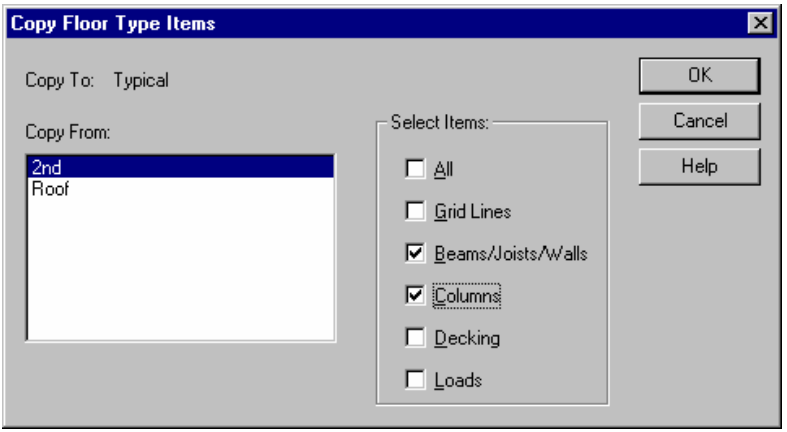
## *Copy Floors Types*

In many buildings the various floor types are similar. The program has a feature for copying information from one level type to another. The feature only works when you have a blank level type current.

- Select **Layout – Type – Select**.
- Highlight the Level called "Typical" and Click **[Select]**.

The graphics view is altered to show you the Typical floor type. It is completely blank at this time. In the menu bar at the top of the modeler, you'll notice that the current level type has changed from 2nd to Typical. This pull down menu is the quickest way to switch from level to level.

- Select **Layout – Type – Copy**.




- Under Copy from Highlight 2nd.
- Under Select Items check both Beams/Joists/Walls and Columns.
- Click **[OK]**.

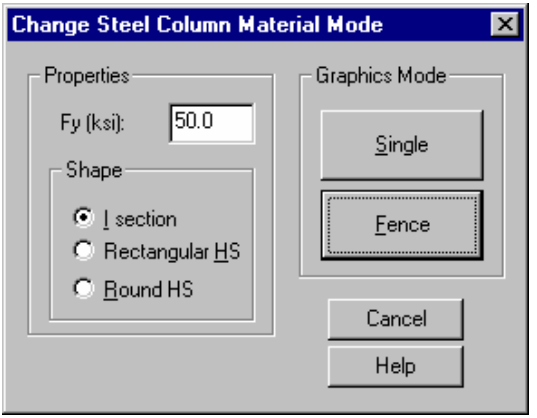
The typical level will now be an exact duplicate of the 2nd floor. Any changes made from this point on will be to one level only. We'll copy to the roof later.

- Select **Layout – Grids – Select**.
- For the Typical level, check the grid system called Main only. The radial grid system is not needed on the upper levels.

## *Steel Column Layout*

Modeling Steel members is identical to modeling concrete members, but the gravity only steel members do not have to be assigned any specific size. The RAM Steel design modules will select an optimum size for each. We will be modeling some steel columns from scratch, but first we need to change some of the existing columns on the Typical level into steel.

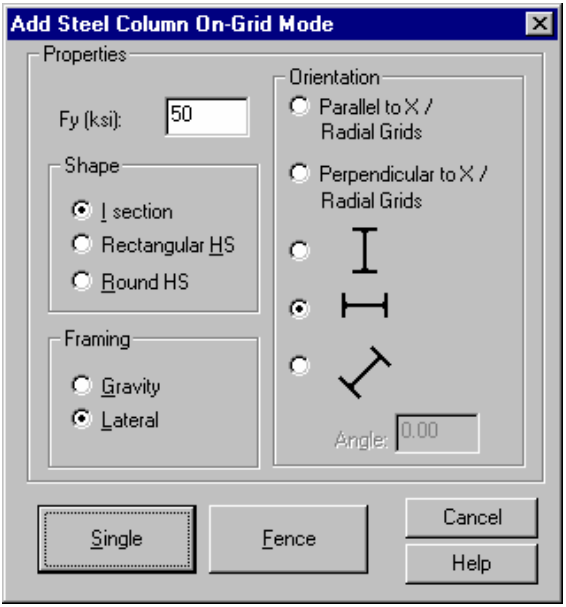
- Select **Material - Steel** (or use the pull-down list of materials).
- While on the Typical floor type select **Layout - Columns - Change Material**.




- For the Fy value type 50 (355 for py).
- Set Shape to I section.
- Click **[Fence]** and in the graphics mode fence the entire framing. All of the columns are changed into Steel Wide flange shapes. The framing and orientation of the original columns are maintained.

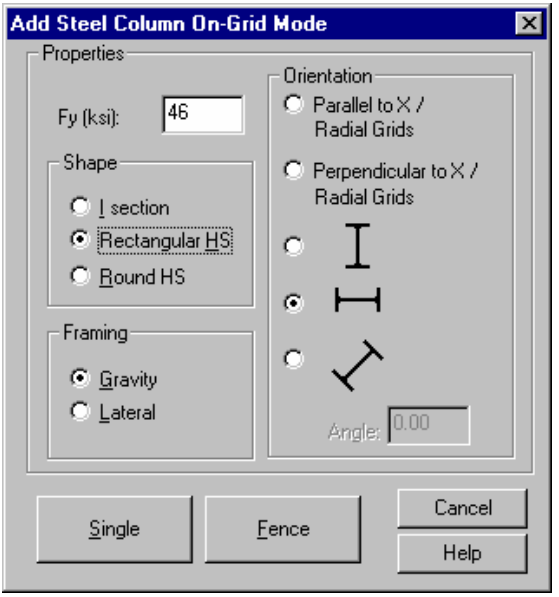
Now it's time to add the extra columns for the stair tower and the braced frames which will replace the walls on the upper levels, as well as the canopy on the 2nd level.

- Select **Layout - Columns - On-Grid**.
- Set the Fy value to 50 (355).
- Change the Framing type to Lateral.
- Click **[Single]**.

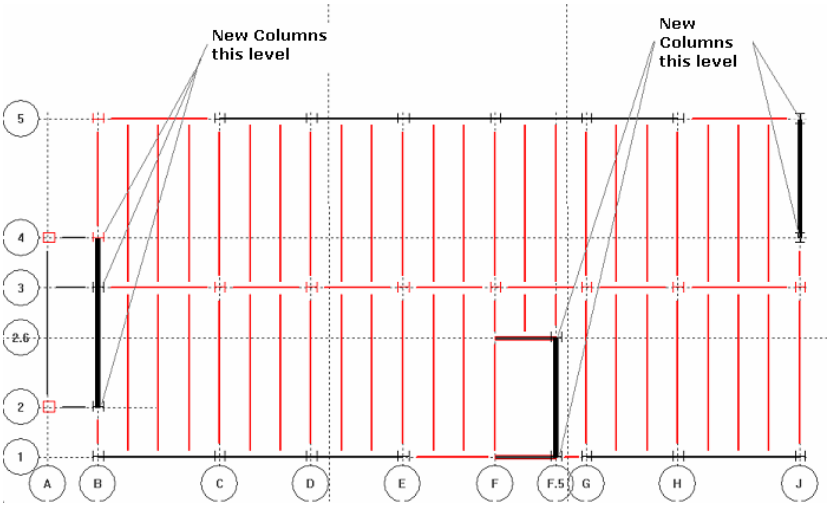



- With the "+" cursor, add lateral steel columns to the ends of the lateral walls at Grids B-2, B-3 (within the wall), F.5-1 and F.5-2.6.
- Right click to return to the Add Column On-Grid dialog box.
- Change the Framing from Lateral back to Gravity.
- Click Single and place a column at B-4 (end of the wall).
- Right click to return to the options.
- Change the Fy (py) value to 46 ksi (355).
- Set the Shape to Rectangular HS. Note – the choice to use HSS sections or TS sections is determined by the master table and design table you select in the RAM Manager. The tables called "RAMAISC" have the latest HSS information, while the "RAMAOLD" tables use the older TS and Pipe designations.
- Leave the orientation set to "H". Note – even though the tube may be square the program still keeps track of major axis and minor axis forces. If the tube is part of a moment frame, it is recommended to orient the column in the plane of the frame. The orientation symbols in the dialog box are draw as Wide Flange shapes for clarity.

- Click **[Single]**.

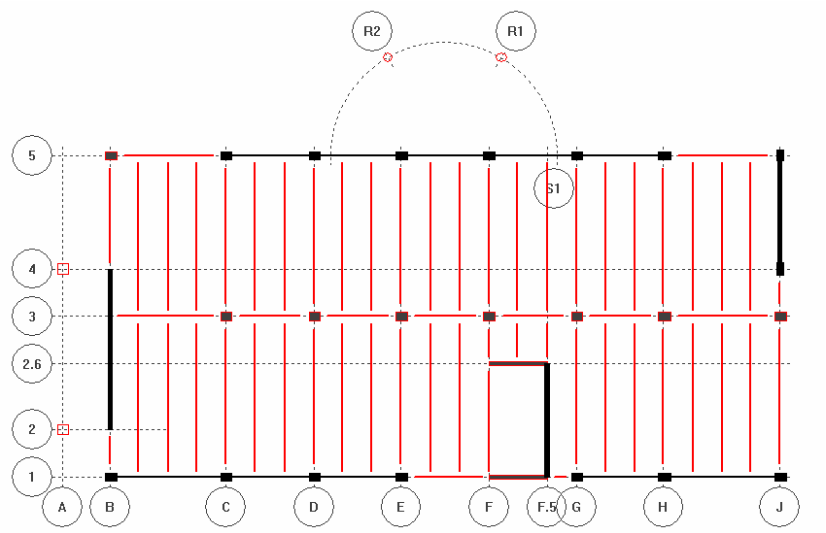



- With the "+" cursor, click once at A-2 and A-4.




Typical level Column layout

- Change the current level from Typical to 2nd using the drop down list on the top menu bar.
- Right click to resume adding columns.
- Add the same tube columns to grids A-2 and A-4 on this level.
- Right click to return to the Add Columns dialog box.
- Change the Fy from 46 (355) to 42 (355).
- Change the Shape from Rectangular HS to Round HS.
- Click **[Single]**.
- Add columns on the radial grid intersections at R1-S1 and R2-S1.




2nd level Columns and Concrete beams

## *Steel Beam Layout – 2nd Level*

Steel beams, like concrete beams must be modeled to have two supports. Beams with cantilevers are modeled as simple-span beams first, then the cantilevers are added to the end. Do not attempt to add a long beam from the tip of the cantilever to the other end of the beam as it will cross the supporting beam or column which is not allowed. When you have indeterminate systems, like two cantilevers meeting at a point, simplifications may have to be made. To begin laying out beams:

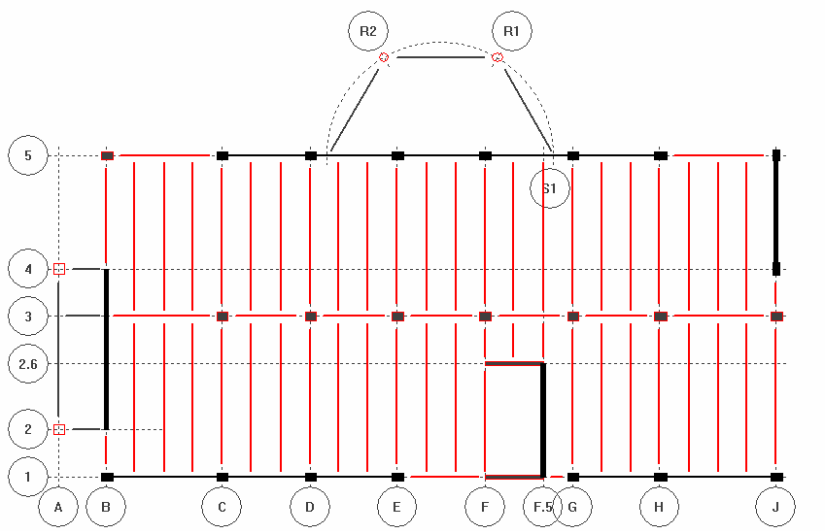
- Select **Layout - Beams - Add On-Grid**.


- Set the Composite framing flag to Noncomposite.

Note – in order for a beam to be designed as a composite beam, it not only needs to be defined as a composite section, but it also needs to have a composite deck on top of it for the entire beam length. If the beam is covered with noncomposite deck (or no deck at all) then it will always get designed as noncomposite.

- Click **[Single]**.

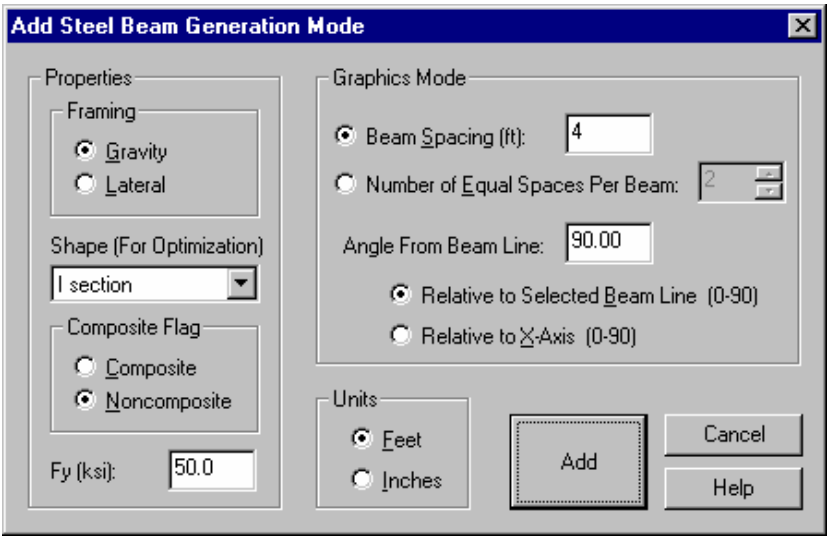
Add steel beam connecting the new steel columns to the concrete framing as shown in the following figure. Note – the direction you add the beam does not matter. Beams are always assigned a left end and a right end based on the geometry.




2nd Floor Steel Beams On-Grid

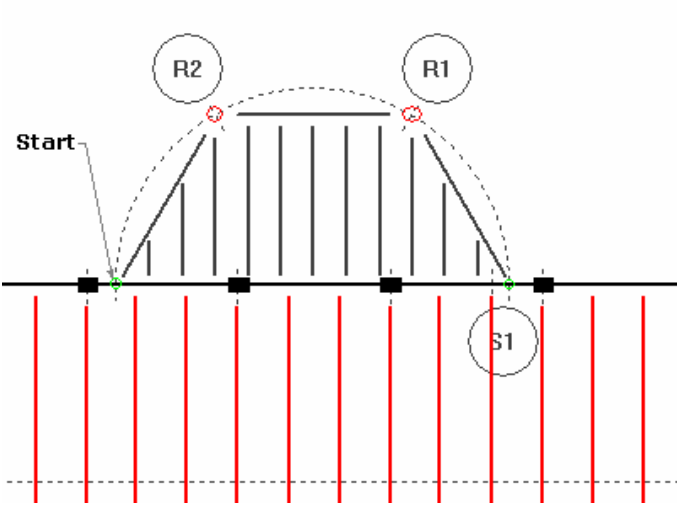
The rest of the Canopy beams will be added using a generation of beams. This is useful for any beams or joists of regular spacing.

- Select **Layout - Beams - Add Generation**.



- Set the Graphics mode to Beam Spacing and set the value to 4 ft (1.25).
- Click **[Add]**

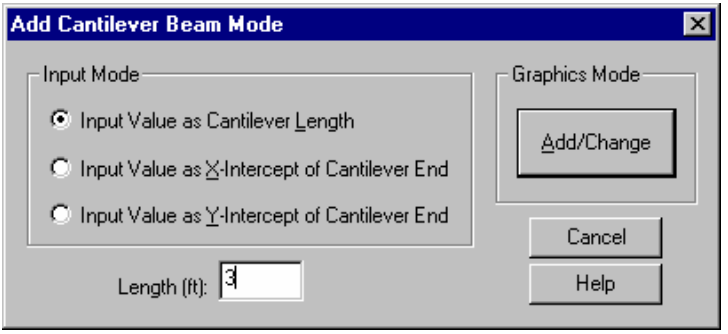
Click along the diameter of the circle, starting and stopping at the two points where Grid S1 crosses Grid 5. Then click to the north to add equally spaced beams. Two of those beams should be perfectly aligned with the columns. In practice, that might be a complex connection, but it's fine for now.



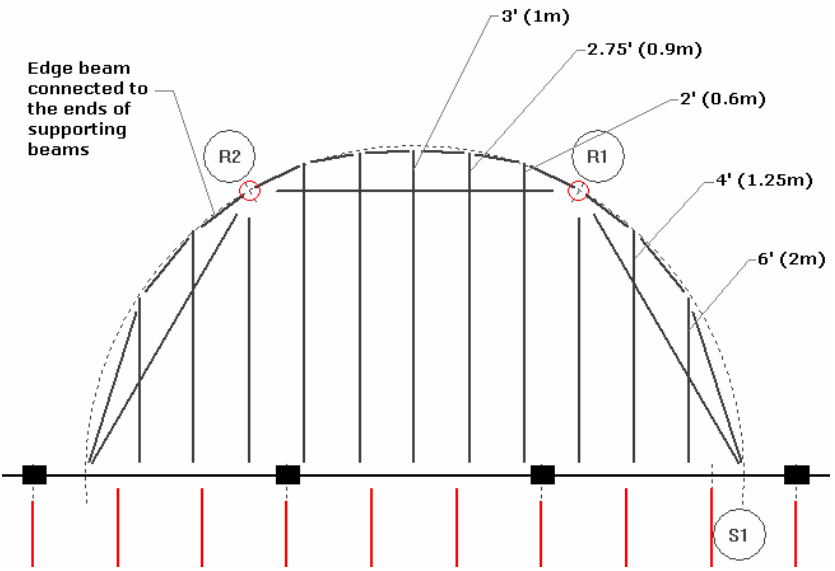

Generated Beams

Now you are going to add cantilevers to these new beams to frame out the circular shape. The following cantilever lengths are rounded for simplicity.

- Select **Layout - Beams - Add Cantilever**.



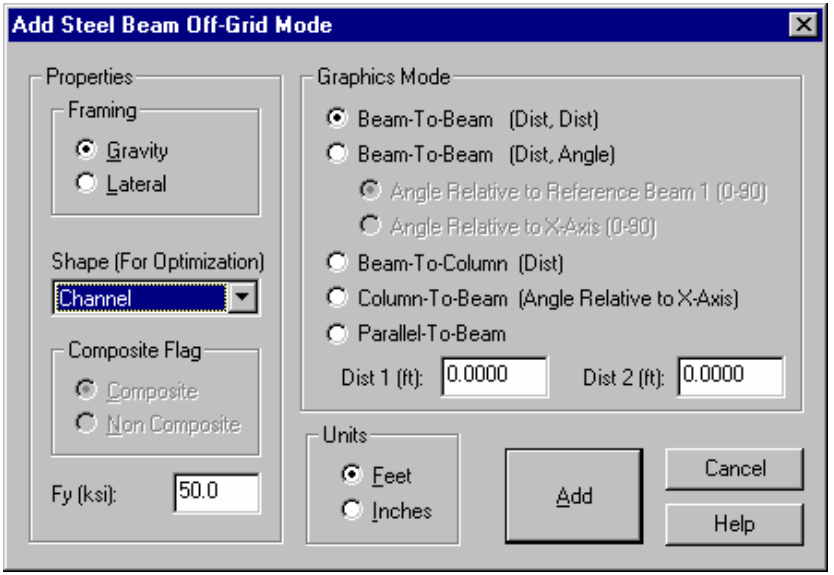
- Type 3 (1) for the cantilever length. Note; there is an option to project cantilever beams to a global X or Y intercept. When you know where you want the cantilever to stop, but you don't know how long the cantilever should be, that is the preferred method.
- Click **[Add/Change]**.
- With the target cursor, select the center beam of the canopy and then click at the north end of that beam to project the 3' cantilever in that direction.
- Right click to return to the Add Cantilever dialog box. Change the value to 2.75 (0.9) and add cantilevers to the two beams on either side of the center beam.
- Repeat these steps using the cantilever length indicated in the figure below.




Canopy framing with Cantilevers and Edge beams

We need an edge beam in order to define the edge of the floor slab. These beams need to be modeled from one cantilever to the next, otherwise the system is indeterminate. To model those beams:

- Select **Layout - Beams - Add Off-Grid**.




- Change the Shape drop down list to Channel.

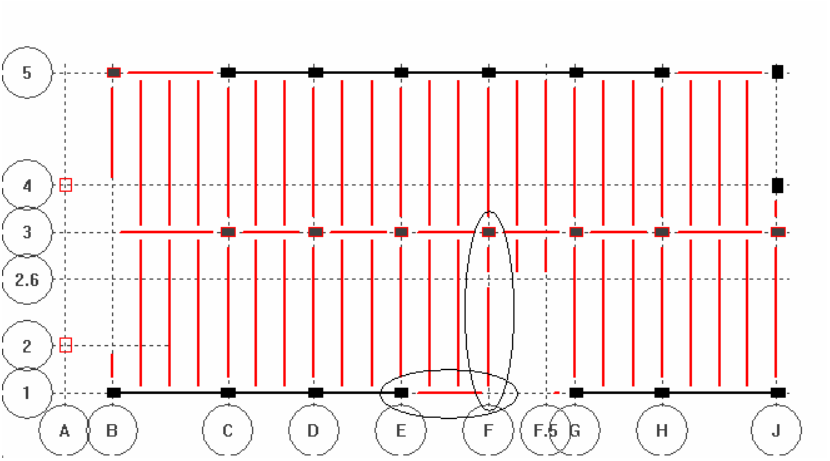
There are many options for adding beams off-grid. Often it is a matter of adding a new beam parallel to an existing beam with some specified offset. This can be done even if the new beam is in a different bay. In this case the beams will be added from then end of one beam to the end of another.

- Set the Graphics model to Beam-To-Beam (Dist, Dist) and set Dist 1 and Dist 2 to 0. The will allow us to add new beams from then end of one beam to the end of another. It should be noted that the method of layout out beams On-Grid would also work in this case, not because of the curved grid line, but because the mouse will always snap to the end of an existing beam even if there is no actual grid line at that location.
- Click **[Add]**.
- With the target cursor, select any one of the cantilever beams, then click at the north end of that beam to set the start point.
- Now select the next beam over and pick the north end of that beam. A new beam will be added from the end of one cantilever to the next
- Continue selecting outer ends of the beams until the arc is defined. At the end, you can select the girder as the reference beam.

## *Steel Beam Layout – Typical Level*

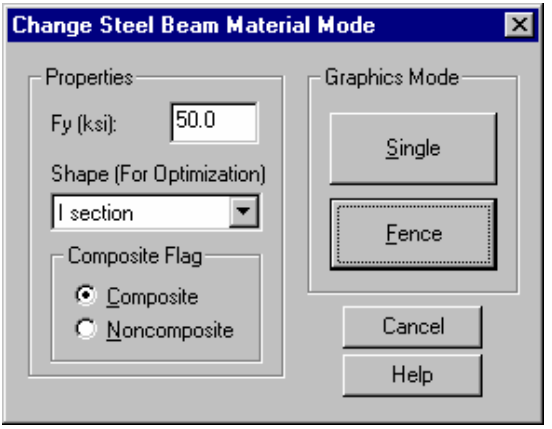
That completes the layout of beams on the 2nd level, but there is some more work to do on the other level types.

- Select the Typical level type.
- Change the current material from Steel to Concrete
- Select **Layout - Walls - Delete**.
- Click **[Fence]** and fence all the walls. Now there are a few beams that we need to delete.
- Select **Layout - Beams - Delete**.
- Select **[Single]**.
- In the graphics mode, delete the beams circled below.



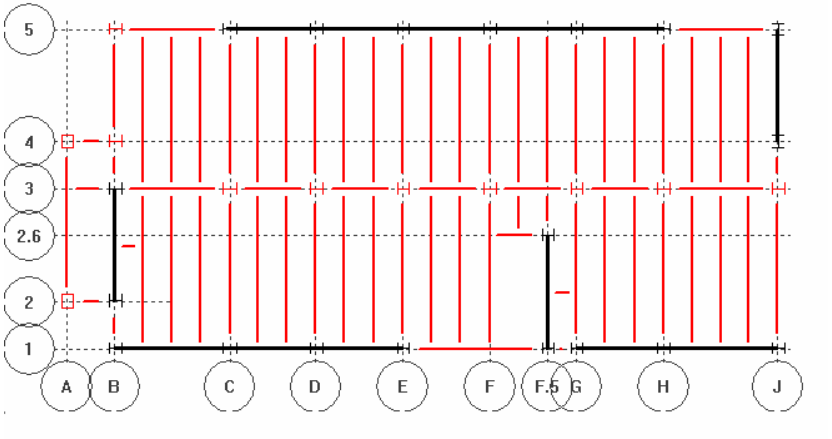

Concrete Beams to be Deleted from Typical Level

- Change the current material from Concrete to Steel.
- Select **Layout - Beams - Change Material**.
- Change Fy to 50 ksi (355).
- Change the Shape to "I" section.
- Set the framing to Composite.



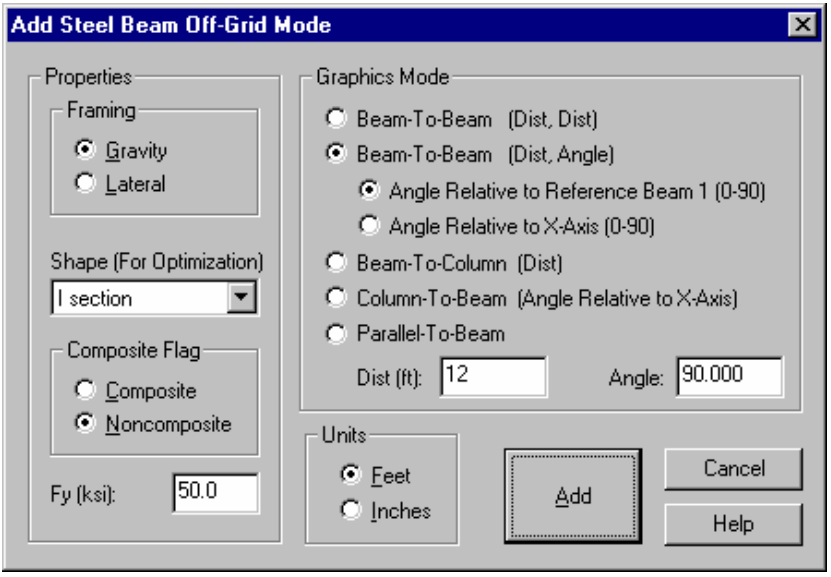

- Click **[Fence]** and fence the whole floor to change all of the concrete beams into steel.

Now you just need to complete the steel framing. The new steel beams of this level can be added a number of different ways. The image below indicates the final beam layout of the Typical level. The darker beams are lateral beams while the rest are all Composite gravity beams.




Complete Framing of the Typical Level

- Select **Layout – Beams – Add On-Grid**.
- Change the Framing from Gravity to Lateral.
- Click **[Single]** and add beams in place of the walls from Grid B-2 to B-3, F.5-1 to F.5-2.6 and from J-4 to J-5.
- Right click to return to the Layout – Beams – Add On-Grid.
- Change the Framing to Gravity.
- Add the necessary gravity beams needed to complete the framing indicated in the previous picture, except for the short beams at the mid-span of the braces which you will add next.
- Select **Layout Beams – Add Off-Grid**.



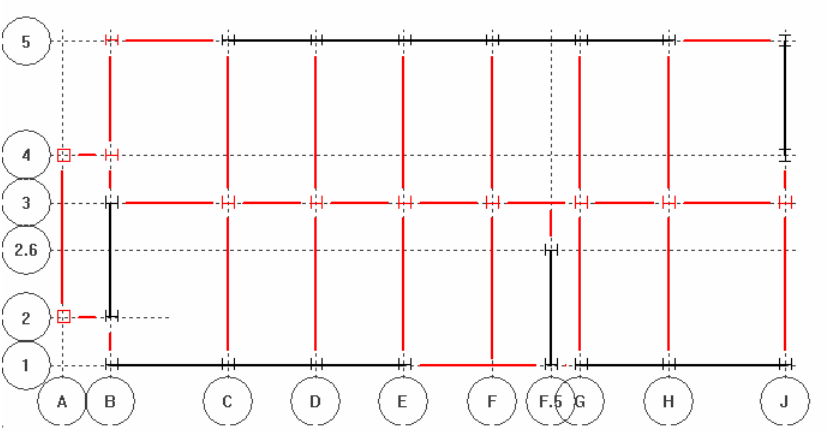

- Change the Framing back to Gravity and the Composite flag back to Noncomposite.
- Select the option Beam-To-Beam (Dist, Angle).
- Set the Dist. To 12 (3.75) and the angle 90 degrees, measured relative to the Reference beam.
- Click **[Add]**.
- With the target cursor, select the lateral beam from grid B-2 to B-3.

- Click to the south end of that beam.
- A white horizontal line will appear 12’ from the south end of that beam. Click to the right side and the new beam will be placed on that side. This beam will brace the top and bottom flange of the lateral beam which will improve the design.
- Repeat for the lateral beam on Grid F.5.

## *Steel Joist Layout – Roof*

For the roof level, you can again copy information from another level to begin.

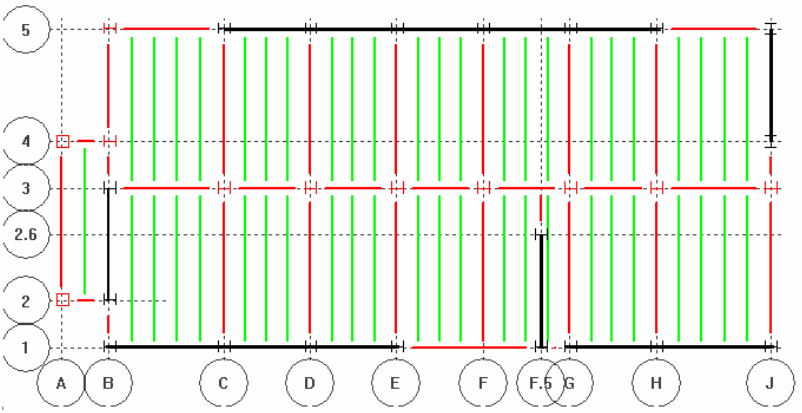
- Select the Roof level using the Level type pull-down list.
- Select **Layout - Type - Copy**.
- Copy the Grids, Beams/Walls/Joists, and Columns from the Typical level.
- Select **Layout - Beams - Delete**.
- Click **[Single]**.
- Delete all of the infill beams so that the plan looks like the picture below.




Remaining Steel Roof Beams

- Change the Current Material to Steel Joist
- Select **Layout - Joists - Change Material**.
- Click **[Fence]**.
- Fence all the center beams along Grid line 3.
- Select **Layout - Joist - Add Generation**.

- Set the Graphics Mode to Number of Equal Spaces Per Beam and change the value to 4.
- Click **[Add]**.
- In the graphics mode, drag along the center beam from C-3 to H-3, then click to the north side of the Joist Girders.
- Repeat to the south side of Grid 3.
- Right click and change the number of spaces from 4 to 5.
- Click **[Add]**.
- Click along the Joist Girders from Grid B to C and from H to I adding joists on both sides of Grid C.
- Change the Number of spaces to 2 then click **[Add]**.
- Drag along the short beam between A-2 and B-2, then click to the north side to cover the stair.

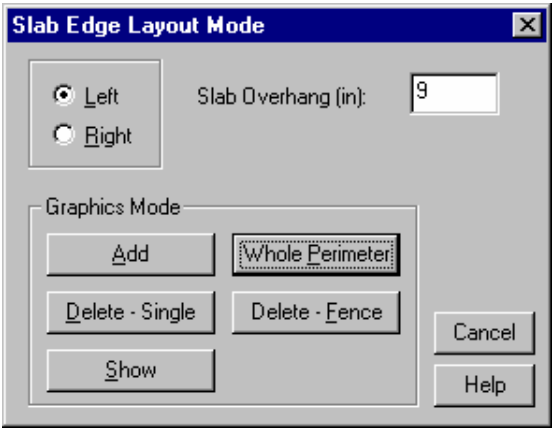



Roof Beam and Joist Framing

## *Slab Edge*

Now that the floor framing layout is complete, the slab edges and slab openings must be laid out. Regardless what type of deck you are using, the level needs to have a slab edge. The deck and surface loads need the slab edges to define their boundaries. To layout the slab edge:

- Select the 2nd level type.
- Select **Layout – Slab – Slab Edge**.



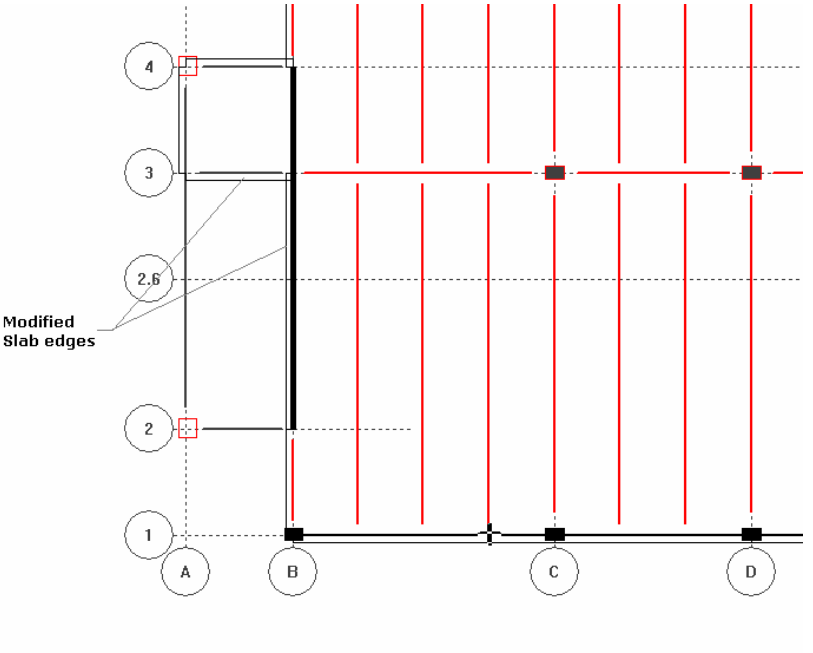

- Type 9 (225) for Slab Overhang. The slab edge overhang is measured from the beam centerline and must be at least 1".
- Click **[Whole Perimeter]**.

You should see the slab edge (a green outline) surrounding entire perimeter of the 2nd Floor Plan. The slab edge must create a single, complete loop. If there are two or more separate floor areas on one floor, then those areas must either be linked together with beams or modeled on separate levels at this time.

If you receive an error applying the slab edge, then there may be errors in the framing (e.g. unsupported beams). You can use the Layout – Beams – Delete command to delete any beams that have not been framed correctly.

- Right click to return to the Slab Edge Layout.
- Click **[Delete Single]**.

- Pick the short slab edge segments on the west and south sides of the stair tower.
- Right click to return to the Slab Edge Layout.

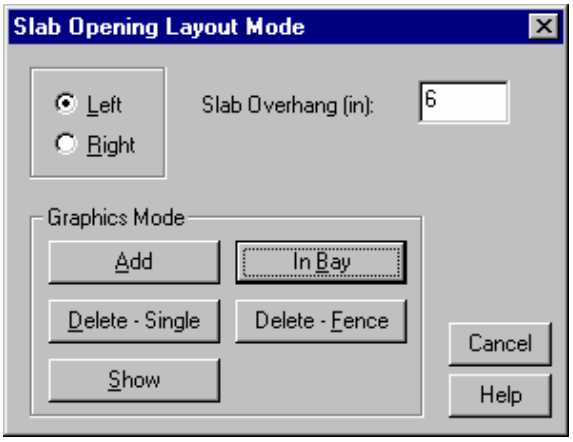



- While the Layout mode is set to Left, click **[Add]**.
- In the graphics mode click at B-2, then click again at B-3.
- Click again at B-3 and then A-3
- Click again at A-3 and close the slab edge by clicking at A-4. In this case, the order that you select these locations does make a difference. So long as the layout mode is set to “Left”, the slab edge must be applied in a clock-wise fashion around the structure. The slab edges will always be offset to the outside of the structure.
- Repeat these exactly steps at the Typical floor level.
- Repeat these steps for the Roof level, but leave the slab edge around the whole perimeter.

## *Slab Opening*

Now that the slab edge is complete, you can now add an interior opening:

- Select the 2nd level type.
- Select **Layout – Slab – Slab Openings**.




- Type 6 (150) for Slab Overhang.
- Click **[In Bay]**.
- With the target cursor, click once in the center of the Elevator Shaft (rectangular bay defined just east of Grid F). You should see the four edges of each opening drawn along the adjacent framing.

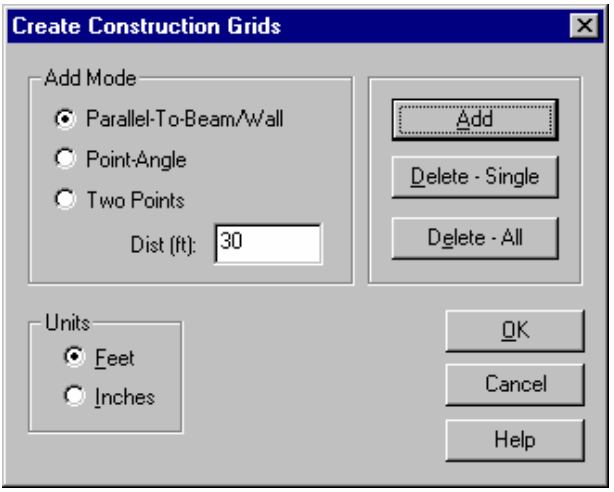
Slab edge openings, like slab edge overhangs are always offset from the center line of the framing, but these edges are offset towards the center of the opening.

- Repeat these steps exactly on the Typical floor level.

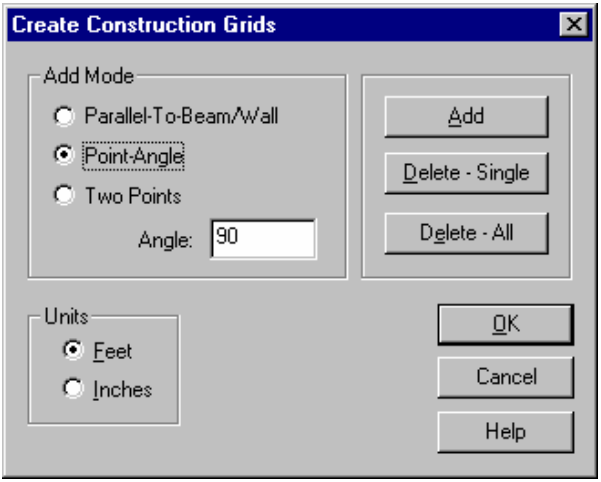
## *Construction Grids*

In addition to these two grid systems, the model also needs a few construction grids to aid in the modeling of certain features. Construction grids are useful for modeling loads or deck areas that do not fall exactly on the framing. Construction grids will not appear in any other design module and that are not labeled. To create a construction grid:

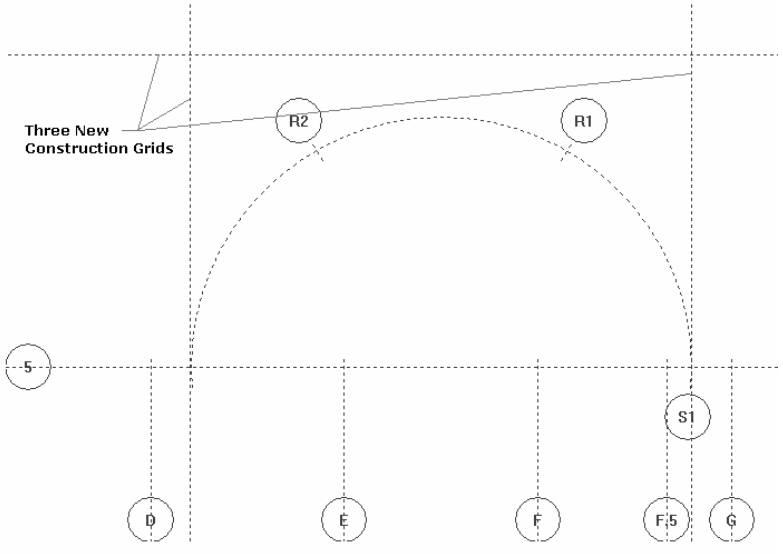
- Select **Layout - Grids - Construction Grids**.




- With the Add Mode set to Parallel-To-Beam/Wall type 30 (10) for the distance.
- Click **[Add]**.
- With the cursor, select any of the beams along the north Grid 5.
- Two parallel white lines 30 ft from grid 5 will appear. Click on or near the top line to create the new grid at that location.
- Right click to return to the Construction Grids dialog box.
- Change the Add Mode to Point-Angle and set the angle to 90 degrees.
- Click **[Add]**.




- With the "+" cursor, click at one of the two intersections of Grid S1 and Grid 5.
- Click to either side to finish creating the grid line.
- Repeat for the opposite intersection of Grid S1 and Grid 5.

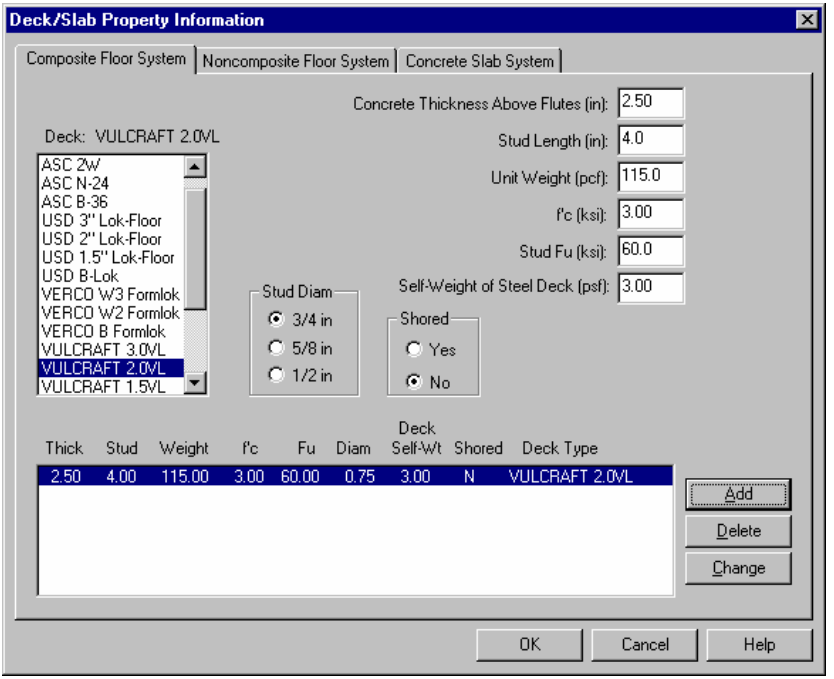



Construction Grids

## *Floor Slabs and Deck*

To complete the physical model it is necessary to apply a slab or deck to the structure. The slab and the deck properties are necessary both to determine composite beam section or T-Beam properties as well as to facilitate the distribution of the gravity load. You do this in two stages, first you will define the properties of the decks and slabs, and then you will layout the slab on the floor. To define the Slab Properties:

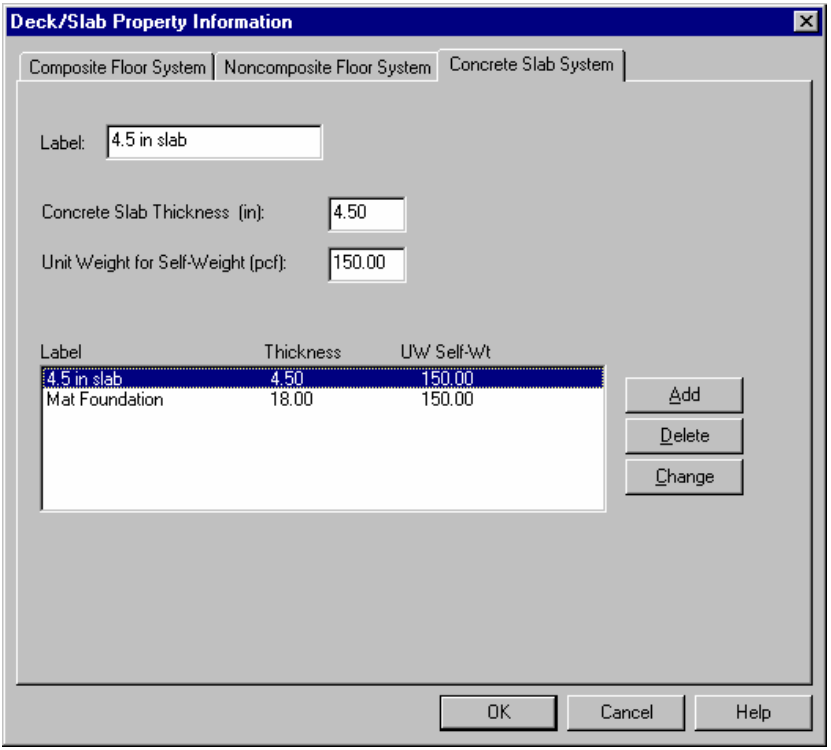
- Select **PropTable - Decking**.




On the Composite Floor System tab select VULCRAFT 2.0VL as the deck type. Set the other attributes as follows:

- Concrete Thickness Above Flutes: 2.5 (65)
- Stud Length: 4 (100).
- Unit Weight: 115 (2400).
- f'c: 3 (fcu: 27.5).
- Stud Fu: 60 (450).

- Self-Weight of Steel Deck: 3 (0.15).
- Set the Stud Diam to 3/4 (19).
- Set the Shored option to No.
- Click **[Add]**.
- Click the Noncomposite Floor System tab.
- Enter a deck Unit Weight for Self-Weight of 3 (0.15). Note – there are no other properties entered for noncomposite deck types. The program is not designing decks at this time, only using the deck self weight in the loads. It is up to the user to make sure the deck selected can span the required distances in the model.
- Select the Concrete Floor System Tab.

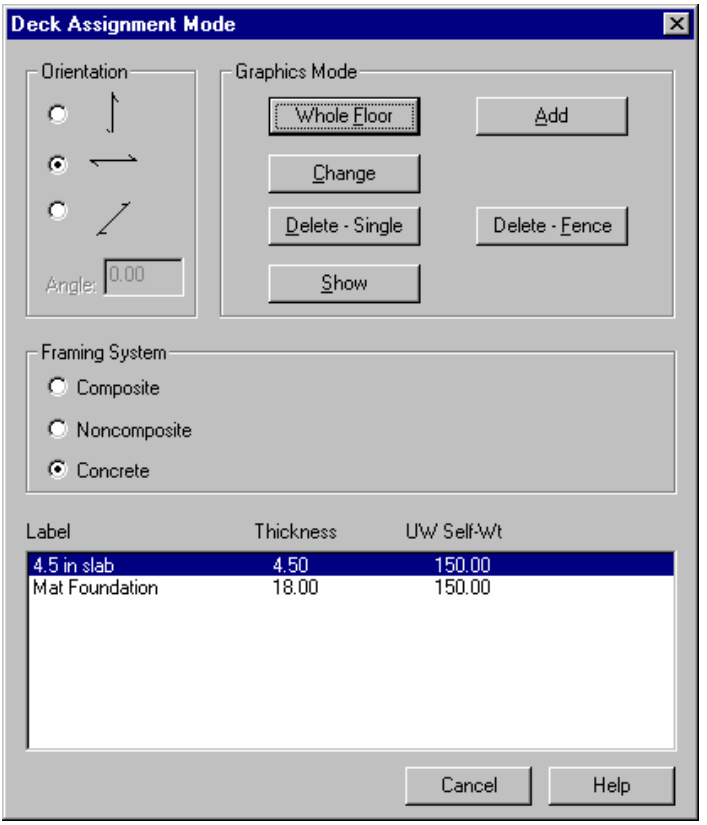



- Type 4.5 in slab for the Label.
- Type 4.5 (115) for Concrete Slab Thickness.

- Click **[Add]** and the information will be added to the large window..
- Type `Mat Foundation` for the Label.
- Type 18 (450) for Concrete Slab Thickness.
- Click **[Add]**.
- Click **[OK]**.

You have now defined the slab to be applied to the floor plan.

- Select the 2nd level type.
- Select **Layout - Slab - Deck Assign**.



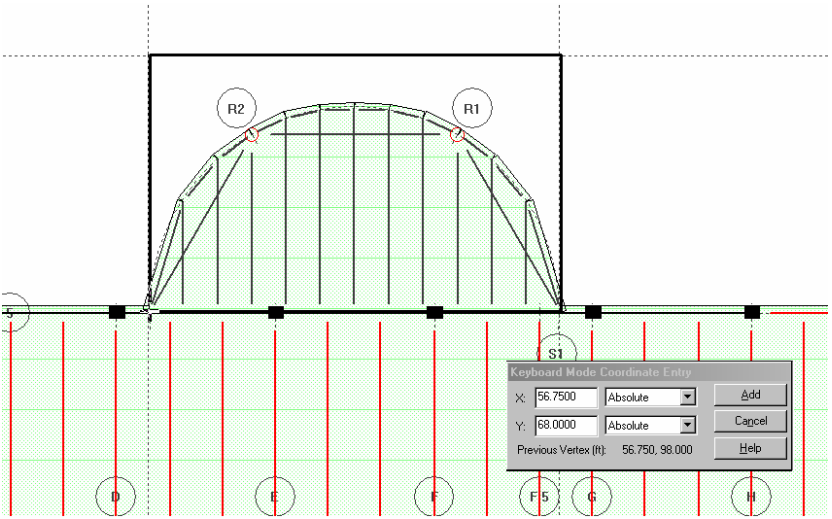
- Select the horizontal orientation option button for the span direction (zero degrees).
- Select the Concrete Framing System Option
- Highlight the concrete slab that you defined from the list box.
- Click **[Whole Floor]**.

This takes you back to the graphics screen where the entire floor area is now hatched with an indication of the decking. Long lines in the hatch pattern indicate the span direction of the slab.

- Right click to return to the Deck Assignment dialog box.
- Change the Framing System to Noncomposite.
- Click **[Add]**.

This will take you back to the graphics screen. If the construction grids have been turned off, turn them back on by selecting Layout – Grids – Show construction grids (this will not terminate the add deck command).

With the "+" cursor, click at the 4 corners of the rectangle defined by the three construction grids and Grid 5. A continuous white line should appear as you trace out this polygon. Once the polygon is closed the area will be hatched differently that the concrete slab area




2nd Level Deck Areas

The entire floor, except the slab openings, should now be hatched indicating the slab placement. (If the program does not display the deck, it means your slab edge or one of the slab openings are not closed properly. Return to fix that now.). The deck itself is always truncated to the slab edge so it is fine to add deck areas larger than the floor plan as you did here. By making the deck extra large you are assured that the correct deck extends out to the slab edge overhang. This is recommended to prevent unwanted decking on the overhang that might incorrectly brace the perimeter beams.

The points that define the deck or slab assignment area can also be established using the Keyboard Mode Coordinate Entry dialog box that appears on screen. The X and Y fields are related to the global coordinate system of the model. Once you have established a node of the deck polygon, those values can also be used to generate an offset, or relative displacement for the next node. The dialog box can be moved if it is in your way when graphically selecting points.

Now you can assign decking to the other levels.

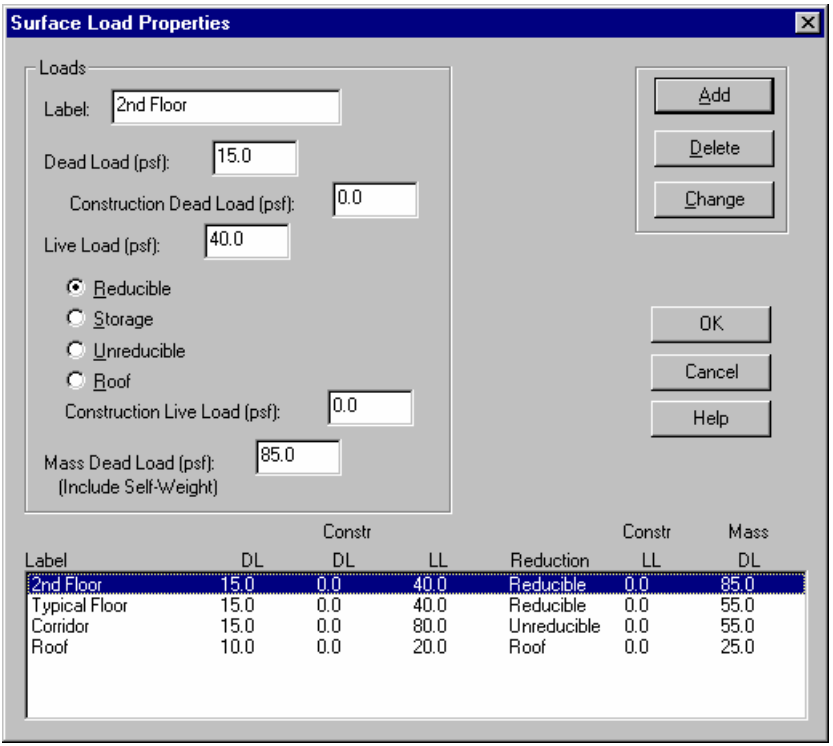
- Select the Typical level type.
- Select **Layout – Slab – Deck Assign**.
- Select the Composite Floor deck.
- Click **[Whole Floor]**.
- Select the Roof level type.
- Select **Layout – Slab – Deck Assign**.
- Select the Noncomposite deck.
- Click **[Whole Floor]**.

## *Defining Loads*

You will now define and apply the gravity loads for which the floor's gravity system will be designed. Loads must first be defined and then applied to the model.

The applied loads on a typical floor consist of surface loads, including tapered snow loads, and line loads. The gravity loads will be defined in the Load Properties dialog boxes. There are separate dialog boxes for defining surface loads, line loads, point loads and snow loads. To define the Surface Loads:

- Select **PropTable - Loads - Surface**.

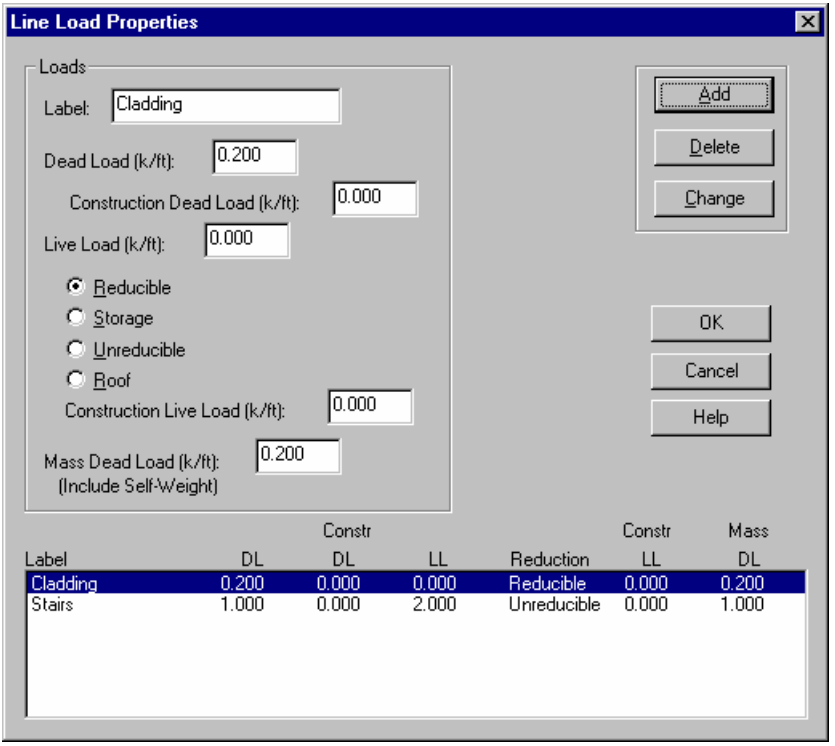



- Type `2nd Floor` for Label.
- Type 15 (0.75) for Dead Load. These are only the superimposed dead loads on the slab. The weight of the beams, and slab are already accounted for in the self weight.

- Type 40 (2) for Live Load.
- Set the Live Load Type to Reducible.
- Leave the Construction Dead Load and Construction live loads at zero. These are only used composite steel beam design and they are only required when there are construction loads greater than the self weight.
- Type 85 (4) for Mass Dead Load. Mass Dead Load data is required in order to generate mass properties for lateral analysis. **The program does not currently add the beam, column or slab self weights into the Mass DL of the structure. Include an estimated self weight when entering this field.**
- Click **[Add]**.
- Change the Label to Typical Floor
- Change the Mass DL to 55 (2.5) and click **[Add]**.
- Change the Label to Corridor
- Change the Live load to 80 (4) and change the type of load to Unreducible.
- Click **[Add]**.
- Change the Label to Roof
- Change the DL to 10 (0.5)
- Change the Live load to 20 (1) and change the type of load to Roof.
- Change the Mass DL to 25 (1.25) and click **[Add]**.

To define the Line Loads:

- Select **PropTable – Loads – Line**. The Line Load Properties dialog box should appear:

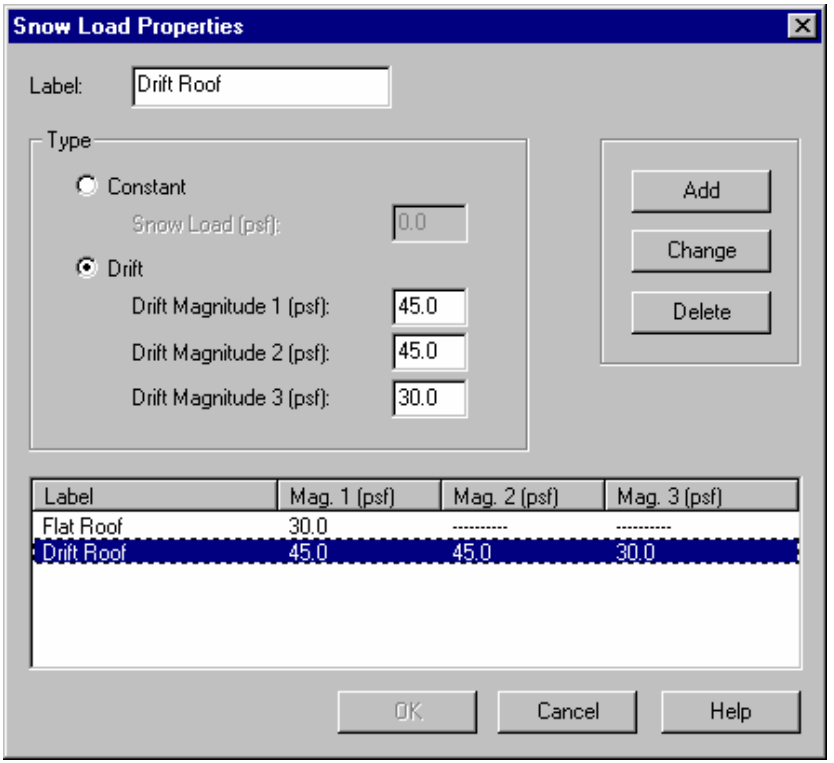



- Type `Cladding` for Label.
- Type 0.2 (3) for Dead Load.
- Type 0.2 (3) for Mass Dead Load.
- Click **[Add]**.
- Change the Label to Stairs.
- Change Dead Load and Mass DL to 1 (15).
- Change Live Load to 2 (30) and set the type to Unreducible.
- Click **[Add]**.
- Click **[OK]**.

Point load properties can be defined in a similar manner as the line and surface loads. Feel free to do so now. To define the snow loads:

- Select **Property Table – Loads – Snow**.

- Type `Flat Roof` for the Label.
- Set the Type to Constant and enter 30 (1.5) for the Snow load magnitude.
- Click **[Add]**.

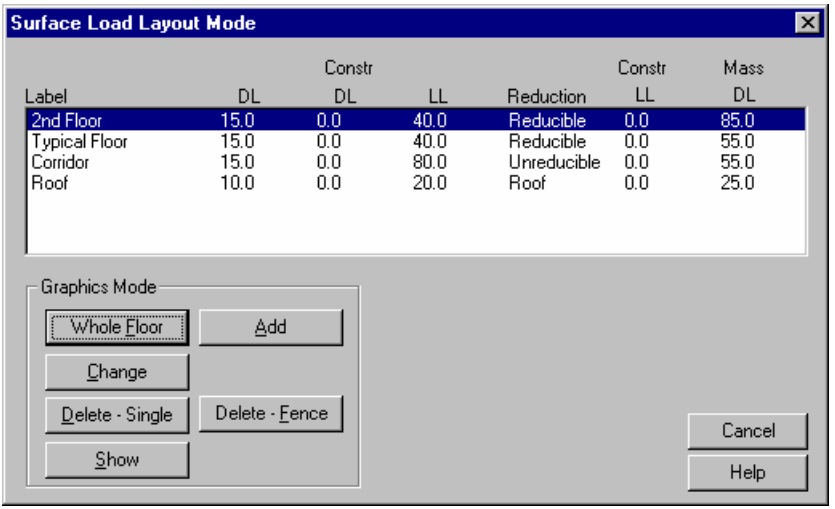



- Change the Label to Drift Roof
- Change the Type to Drift and enter the three magnitudes 45, 45 and 30 (2, 2, 1.5). When you layout the loads you can choose which corner of the load should be associated with the values entered here. The snow loads you apply may have more than 3 corners, but only three are required to define the load plane.
- Click **[Add]** and click **[OK]**.

## *Applying Loads*

Now that the loads are defined, it's time to apply them to the floor.

- Select the 2nd level type.
- Select **Layout - Loads - Surface Loads**.




- Highlight 2nd Floor from the list box in the Surface Loads Layout Mode dialog box.
- Click **[Whole Floor]**.

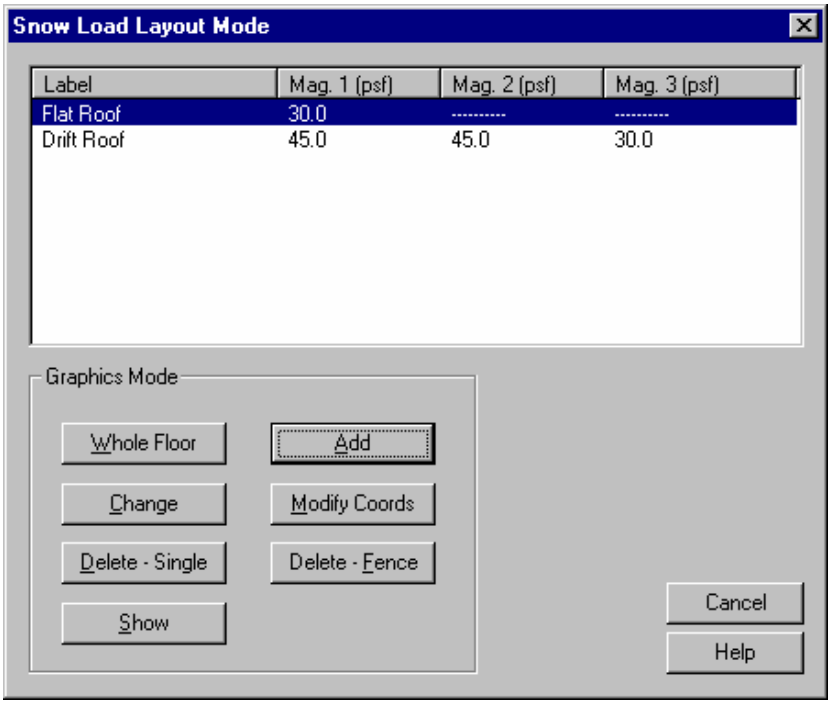
You should see the entire floor covered with a hatch pattern. At this stage you have the floor load over the entire floor area. You could now place other loads over a portion of the floor (as in the case of corridor, storage or equipment loads) and those loads would supersede (replace) the floor load in that area. **Surface loads are not cumulative.**

- Right click to return to the surface loads dialog box.
- Select Corridor and click **[Add]**.
- Click at the four corners of the corridor defined by the grids A-3, A-4, J-4 and J-3.
- Right click to return to the surface loads dialog box.
- Select Roof and click **[Add]**.

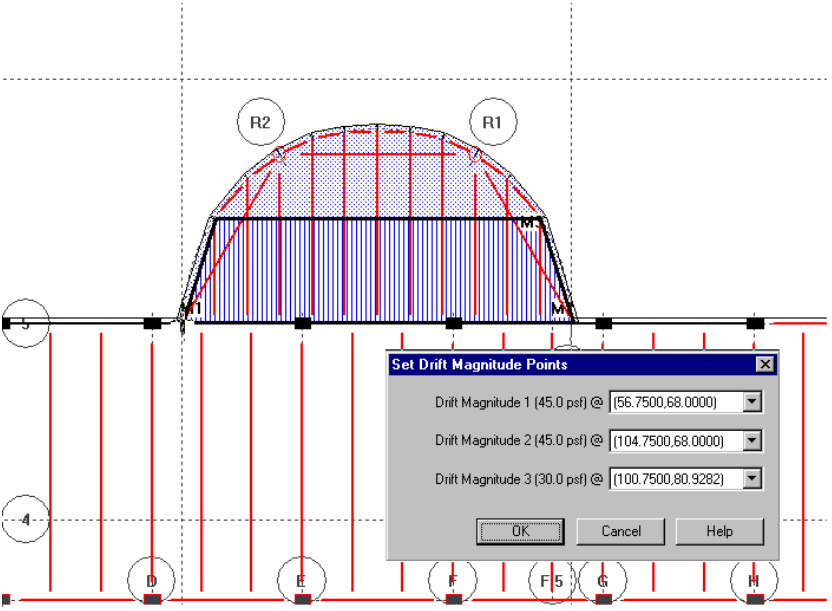
- In the graphics mode select the area of the canopy where the noncomposite deck was previously applied.

Snow loads are applied in much the same way as surface loads.

- Select **Layout - Loads - Snow**.




- Highlight the Flat Roof load and click **[Add]**
- In the graphics mode, select the entire Canopy area using the constructions grids as you did with the deck.
- Right click and change to the Drift Roof load type and click **[Add]**.
- Start by clicking at the lower left corner of the canopy, then select the lower right corner, then continue around counter-clockwise to the end of the longest edge beam, then move left to end of the beam on the other side and finally close the polygon as indicated below.

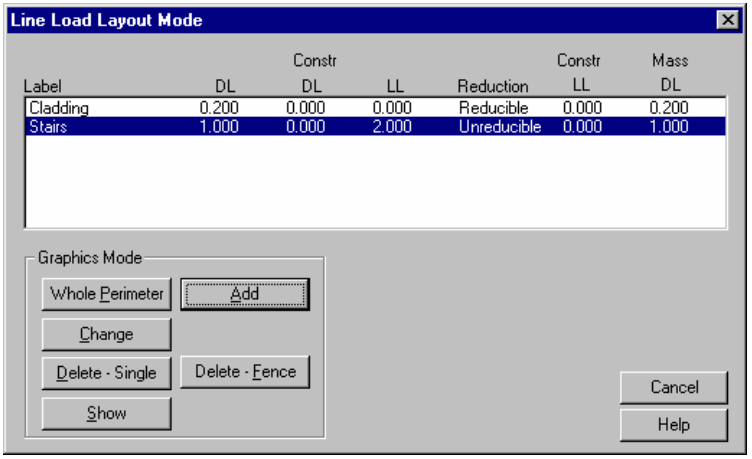



Snow Loads on the Canopy

- You will be prompted to select the 3 defining nodes of the snow load. Select the first three nodes that you picked, in-order so that the load has the highest magnitude across the south edge.

To layout the Line Loads:

- Select **Layout - Loads - Line Loads**.
- Highlight Cladding load from the list box.
- Click **[Whole Perimeter]**. A line indicating the load will appear along all perimeter beams. The line will be broken up into segments, but this not a requirement when modeling line loads.
- Right click to return to the Line Load Layout Mode.




- Highlight Stairs and click **[Add]**.
- Trace along the stair beam from Grid A-3 to B-3.

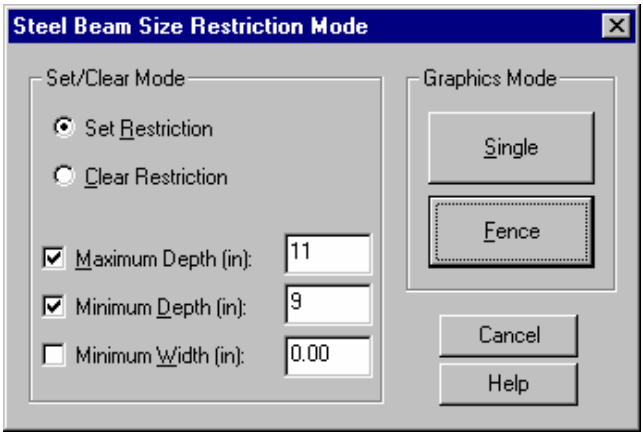
Note; point Loads and Line Loads do not have to be placed directly on a beam for them to be recognized by the program. When those loads are placed directly on the deck the program will distribute a portion of the load to the adjacent framing based on the angle of the deck and the location of the load.

- Select the Typical level type.
- Select **Layout – Loads – Surface Loads**.
- Assign the Typical Floor load to the **[Whole Floor]**.
- Repeat the steps above for adding a Corridor load to this level.
- Select **Layout – Loads – Line loads**.
- Repeat the steps above for adding the Cladding and Stair loads to the Typical level.
- Select the Roof level type.
- Assign the Roof surface load to the whole floor.
- Select **Layout – Loads – Line Loads** and assign the Cladding load to the **[Whole Perimeter]**.
- Select **File – Save**.

## *Steel Beam Size Restrictions*

You can assign specific sizes to steel beams using Layout – Beam – Assign Size which is nice for evaluating existing structures. In this case, we want the program to pick the beam sizes for us, but we want to make sure all the cantilever beams are the same Note; when a gravity beam cantilevers over another beam or column, the behavior is assumed to be that of a fulcrum. No moment is transferred into the supporting member. If you want moment to be transferred as in the case of a fully restrained moment connection, then the beam and the support should be modeled as Lateral members. With lateral members, you can control the end fixity of all the members.

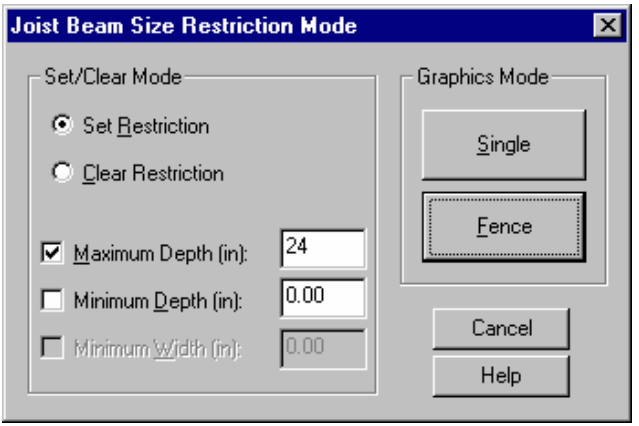
- Select the 2nd level type.
- Set the current material to Steel.
- Select **Layout – Beams – Size Restrictions**.




- Check the Maximum depth limit and type 11 (275)
- Also check the Minimum depth option and set it to 9 (225). Note; these limits are literal limits on the beam depth. We are using a depth range here to insure that we get nominal 10" beam.
- Click **[Fence]** and fence the entire canopy. Note; while you are in the Steel material mode you cannot alter concrete beams.

Hint: With the Options – Set Show Options command, you can confirm the attributes assigned to the model graphically. For example, you can use that feature to highlight all beams with an assigned size restriction.

- Select the Roof level type.
- Set the Current material to Steel Joist.
- Right click and change the size restriction dialog box.



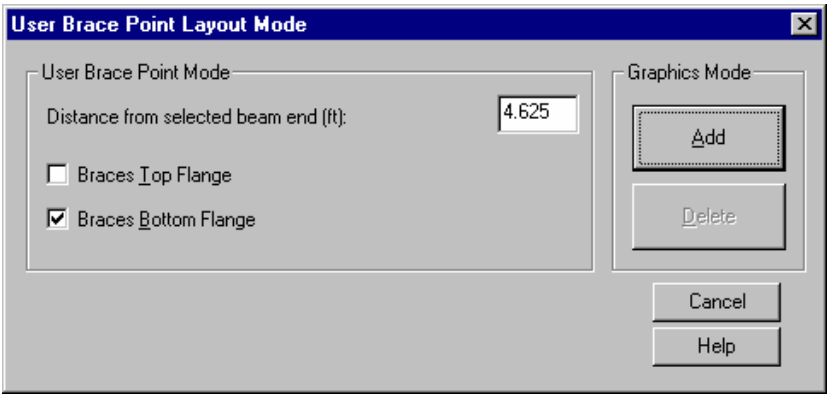

- Check to use only a Maximum Depth of 24 (600).
- Click **[Fence]** and select all of the joist girders on Line 3.

Note – for joist girders, the maximum depth field is used explicitly in labeling the member. These joists will not be deeper than 24", they will be exactly 24" deep. For more on the design labeling of joists and joist girders, refer to the online documentation for the Steel beam module.

## *Brace Points*

In the design of steel beams, the program automatically determines the unbraced length of the top and bottom flanges. When a beam frames into a girder, that girder is braced on the top and bottom flange at that location, but when a joist frames into a girder, only the top flange is braced by default.

- Set the Current Material back to Steel.
- Select **Layout - Beams - Brace Points**.




- Type 4.625 (1.4375) for the Distance from selected end.
- Check the Braces Bottom Flange box only.
- Click **[Add]**.

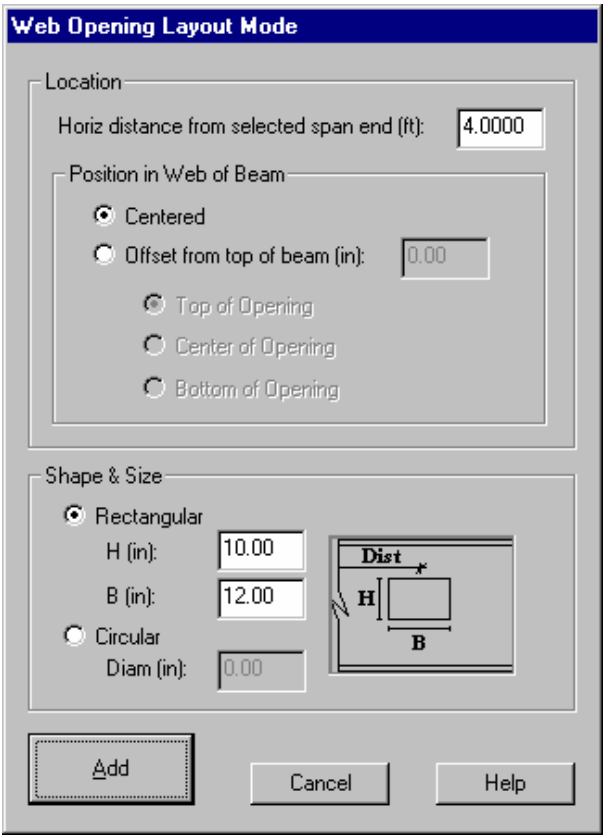
In the graphics mode, select one of the frame beams on Grid Line 1 and then pick at the left end. A yellow triangle on the underside of the beam will appear. For top flange brace points the triangle will appear on top.

- Repeat for the other end of the same lateral beam.
- Repeat for both ends of all the moment frame beams on Grids 1 and 5. Note; the spacing of the joists in the first and last bay is not exactly 4.5 ft, but it's close enough for this tutorial.
- Right click and change the Distance to 9.25 (2.875).
- Continue adding brace points until there is one at or near each joist location on all lateral moment frame beams. If you make a mistake select Edit Undo or use the Delete brace point option and remodel.

## *Web Openings*

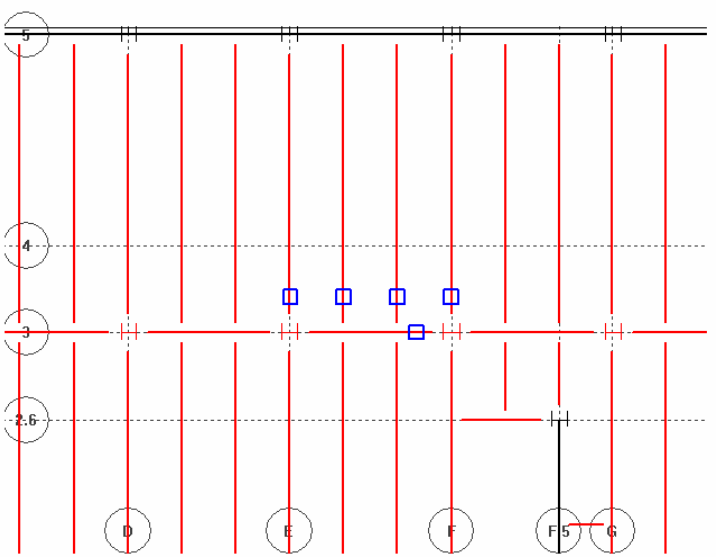
Steel beams can have openings modeled in the web. The size and the location of the opening must be specified, but then the program can optimize the beam and the stiffener plates (when required). To model an opening:

- Select the Typical level type.
- Select **Layout - Beams - Web Openings - Add**.




- For the Location, set the Horiz distance to 4 (1.25).
- Set the Position in Web to Centered
- Set the Shape and Size to Rectangular and make the opening 10 (250) high by 12 wide (300).

- Click **[Add]**.
- With the target cursor, select each one of the steel beams in the bay from E to F, 3. to 5.
- Pick the south end of that beam to place the opening relative to that end.
- Repeat for the rest of the beams indicated below.




Web Openings

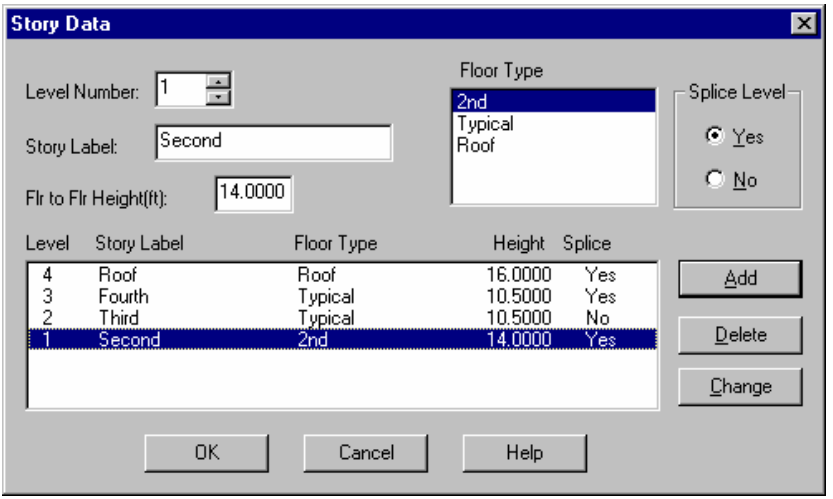
Web openings which occur near the end of the beam or within a small distance of a supported beam will always generate a warning in the design. For more information on web openings, refer to the Steel Beam module on-line documentation.

Also Note; the program is equipped to model and design Smartbeams with repeated hexagonal or circular openings down the length of the beam. This tutorial will not use any Smartbeams, however.

## *Story Data*

With all of the floor types defined, you can now designate the arrangement of these floor types in the building. This is called the story data:

- Select **Story**.

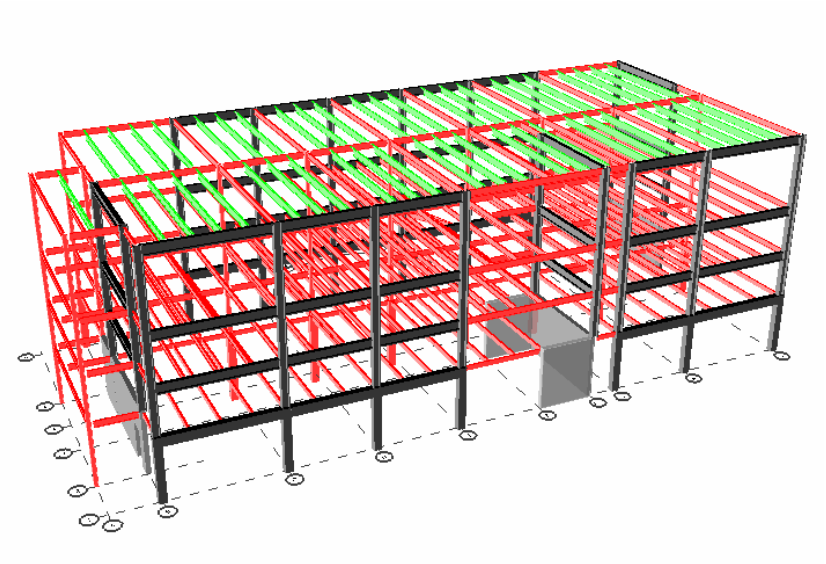



- Type Second for Story Label.
- Type 14 (4.25) for Flr to Flr Height. Note; it is recommended to reference deck bearing when entering the story data.
- Highlight 2nd in the Floor Type list box.
- Select Yes for Splice Level.
- Click **[Add]**.
- Type Third for Story Label.
- Type 10.5 (3.2) for Flr to Flr Height.
- Highlight Typical in the Floor Type list box.
- Select No for Splice Level.
- Click **[Add]**.
- Type Fourth for Story Label.
- Type 10.5 (3.2) for Flr to Flr Height.
- Highlight Typical in the Floor Type list box.
- Select Yes for Splice Level.
- Click **[Add]**.
- Type Roof for Story Label.
- Type 16 (4.75) for Flr to Flr Height.

- Highlight Roof in the Floor Type list box.
- Select Yes for Splice Level. (the splice above the top level does not matter)
- Click **[Add]**.
- If your data appears like the figure above Click **[OK]**, otherwise highlight the incorrect level, modify the required settings and click **[Change]**.

A full 3-D model of your structure has now been developed, albeit not quite complete. To view your 3D model:

- Select **View - 3D-View**.
- You should now see a 3D view of your model. Feel free to investigate some of the features in the 3D Viewer.



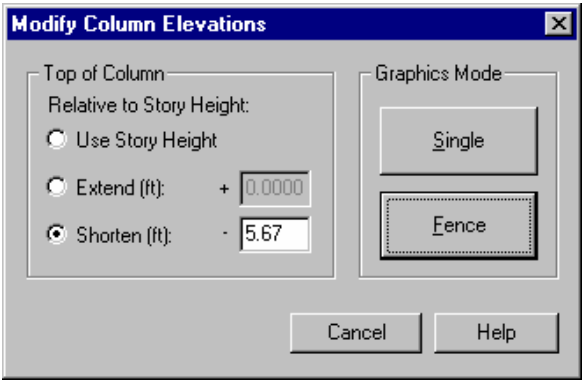
- Press the arrow keys on the keyboard to rotate the model.
- Press the F2 function key to see other keyboard shortcuts.
- To exit 3-D View and return to Plan View, select **File - Exit**.

Note; the environment and interface of the 3D view is also used in some of the design modules.

## *Sloping the Roof*

The Modeler also has the ability to modify the slope of a level. This is done by adjusting the columns or walls from the story reference datum. Foundations can also be raised or lowered. Before modifying the elevations, you will want to turn on the display of the column elevations.

- Select **Options – Show Property – Column and Wall Elevations**. If you have difficulty reading the text on the screen you can increase or decrease the text size with the Options – Scale Text command.
- Select the Roof level type.
- Select **Layout – Columns – Modify Elevation**.

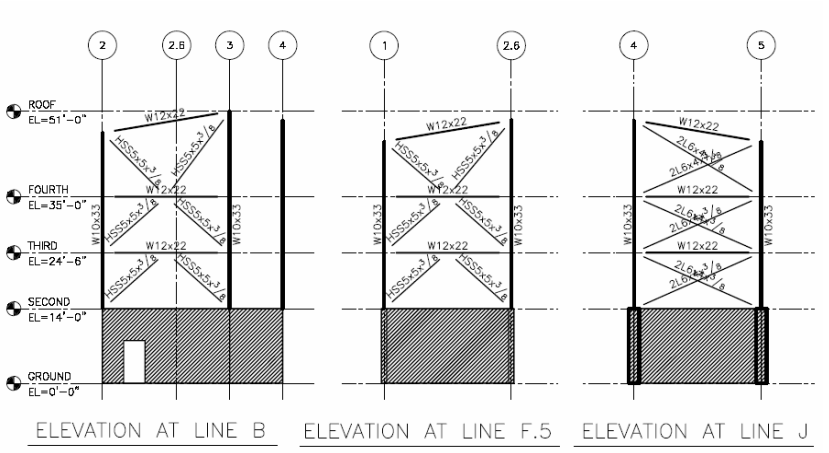



- Set the Top of Column to Shorten and type 5.67 (1.75) for the value (you do not need to use a negative sign).
- Click **[Fence]**.
- In the graphics mode, fence all of the columns on Grids 1 and 5.
- Right click to return to the Modify Column Elevations dialog.
- Change the value to 4 (1.25) and Fence the Columns on Grid 2
- Repeat the command and shorten the columns on grids 2.6 and 4 by 1.67 (0.5).
- De-select **Options – Show Property – Column and Wall Elevations**.

If you review the 3D view now you should be able to see the slope of the Roof.

## *Layout Bracing*

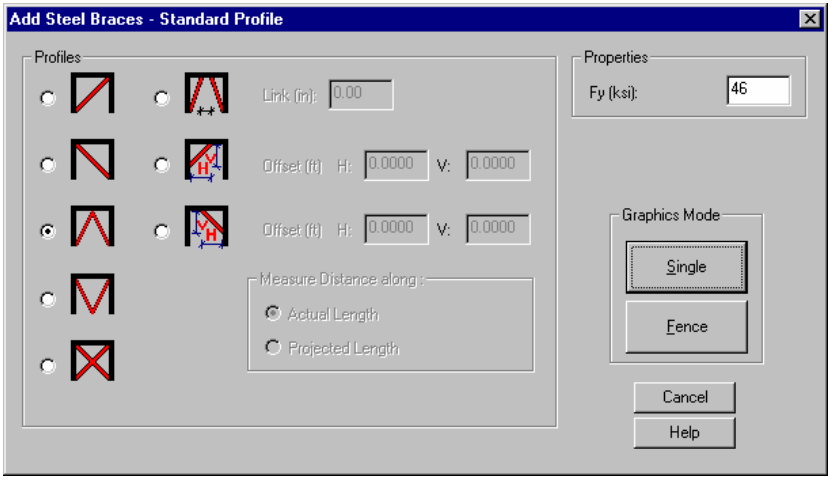
The next step in the layout of this structure is to define the vertical bracing system. The layout of vertical bracing is performed in then elevation mode of the Modeler. Below is the final layout of the braced frames you are going to model. The E-W frames are moment frames in this model.




- Select **View – Elevation Mode**.
- With the cursor, select the frame on Grid B by selecting the Frame Beam or Shear wall. An elevation of your frame will appear. Only the lateral members are displayed in this view.

In elevation mode, a new menu of commands is available to you. While some of the commands from plan mode are included, other commands, such as Layout – Braces, are unique to elevation mode. To add braces:

- Select **Layout – Braces – Add Standard**.
- Select the third Profile for chevron (inverted V) braces.
- Enter 46 (315) for Fy.
- Click **[Single]**.




This will take you back to the elevation view:

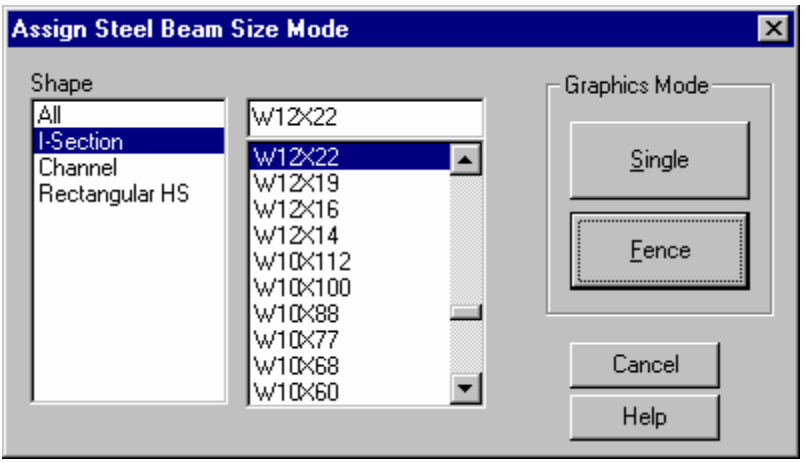
- With the target cursor, click in the middle of the bay defined by the columns and beams of the third level. Do the same for the 4th level.
- Right click to return to the Layout Brace command and change the profile to the fourth, “V” shaped configuration.
- Click **[Single]** and select the top level of the frame.
- Select **View – Plan Mode** to return to the Plan View.
- Repeat the last set of actions for the Frame on Grid F.5.
- Select **View – Plan Mode** to return to the Plan View.
- Select **View – Elevation View** and pick the Frame on Line J.
- Select **Layout – Braces – Add Standard**.
- Select the 5th, “X” shaped profile.
- Enter 36 for Fy (275 for py) and click **[Fence]**.
- Fence the entire frame to put an X brace in the top three levels.
- Select **View – Plan Mode** to return to plan mode.

## *Assigning Lateral Member Sizes*

The RAM Steel Beam Design and RAM Steel Column Design modules optimize the design of the steel gravity members. It is not necessary to assign sizes to these members. However, if you want the program to use (check) a specific size for a particular gravity beam, the Assign Sizes command can be used to assign a size to an individual gravity member. The selected member size would be checked for adequacy and no design optimization will be performed for that member. If the steel beam is specified as composite then RAM Steel Beam Design module will determine the number of studs required to meet the design criteria.

The lateral analysis performed in RAM Frame, on the other hand, requires that preliminary sizes be assigned to the Lateral members in order to analyze those frames. The preliminary sizes can be assigned manually in the RAM Modeler using the Assign Size commands, in RAM Frame using similar Assign Size commands, or member size can be left out and the program will then automatically assign a size adequate for the gravity loads only when the RAM Steel Beam and RAM Steel Column modules are executed.

- Select the elevation view of the braced frame on Grid B.
- Select **Layout - Beams - Assign Size**.




- From the list of sizes, select a W12x22 (UB305x102x33). Note; you can simply type the section name in the box above the list or browse to it.
- Click **[Fence]**.

- Fence all the beams of this frame.
- Select **Layout – Columns – Assign Size**.
- From the list of sizes, select a W10x33 (UC254x254x73) and click **[Fence]**.
- Fence all the columns of this frame.
- Select **Layout – Braces – Assign Size**.
- From the list of sizes, select a HSS5x5x3/8 (SHS120x120x6.3) and click **[Fence]**.
- Fence all the braces of this frame.
- Repeat the same steps for the braced frame on line F.5.
- Repeat the same steps for the braced frame on line J, except assign the braces to be Double Angles 2L6x4x3/8 (2L150x90x12).

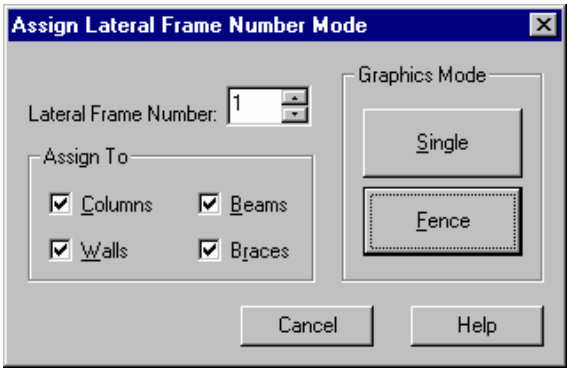
That completes the braced frames, now you can move on to the moment frames.

- Select the elevation view of the Moment Frame on Grid 5.
- Assign all of the columns to be W14x48 (UC356x368x153) with a fence.
- Assign all the beams to be W16x40 (UB406x178x60).
- Repeat for the other two moment frames on Grid 1.

## *Assign Frame Numbers*

Reference numbers, referred to as Frame Numbers, can be assigned to some or all the members of a frame. RAM Frame uses these numbers to organize output to printed reports and screen output. While frame numbers are not required by the program, they are an excellent way to organize output.

- Select **View - Elevation Mode**.
- With the cursor, select the frame on Grid 5.
- Select **Options - Show Property - Frame Number**. This command will cause the frame numbers of each member to be displayed on the screen. Having these active while you make frame number assignments provides visual confirmation.
- Select **Layout - Frame Numbers**.




- Check all four settings for Columns, Beams, Walls and Braces.
- Type `1` for Lateral Frame Number.
- Click **[Fence]**.
- Fence the entire frame. The frame numbers will change from 0 to 1. Any lateral member that is not assigned a specific frame number is, by default, set to be part of Frame 0.
- Label the other frames as follows: Moment frame on Grid 1 from B to E is #2, MF from G-J is #3, Braced Frame on Line B is #4, BF on Gird F.5 is #5 and BF on Grid J is #6. (Refer to 2nd Floor Framing figure at the beginning of the Modeler section).

## *Frame Fixity*

To change or review the member end fixities it is recommended that you display the end fixity on screen:

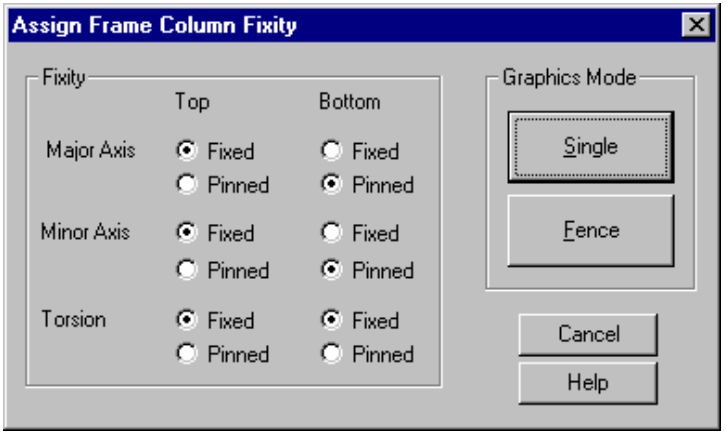
- Select **Options – Show Frame Fixity – All**.

The default element fixities for lateral beams and columns is for the members to be fixed in their major and minor axes as well as about the torsion axis. Gravity steel beams are always pinned, though concrete gravity beams can be fixed. The default for braces is to be pinned in the major and minor axes as well as pinned against torsion.

The fixed condition is indicated by an "X" while a pinned end condition is indicated with an "O". When all fixity conditions are shown simultaneously, for the left or top end of a member, the first character (reading left to right) is for the major axis, the second is for the minor axis and the third is for the torsion axis. For the bottom or right end of the member the opposite order applies.

If the Frame numbers are interfering with the end fixity symbols then turn off the frame numbers by again selecting **Options – Show Property – Frame Number**.

- Select the Elevation View of the Frame on Line B.
- Select **Layout – Columns – Assign Frame Fixity**.




The Assign Frame Column Fixity dialog box should appear. It has two sets of option buttons that give you the choice of setting the column ends to fixed or pinned. One set is for the top of the column and the other for the bottom.

- Set Major and Minor Axes of the Bottom to Pinned.
- Click **[Single]**.
- Select the third level columns (the bottom level of steel). Notice that the fixity display changes.

To change the Frame Fixity for the ends of the beams:

- Select **Layout – Beams – Assign Frame Fixity**.
- Set Major and Minor Axes of the Left and Right ends to Pinned.
- Click **[Fence]**.
- Fence the entire elevation.
- Select **View – Plan Mode** and repeat the last set of actions for the other braced frames.

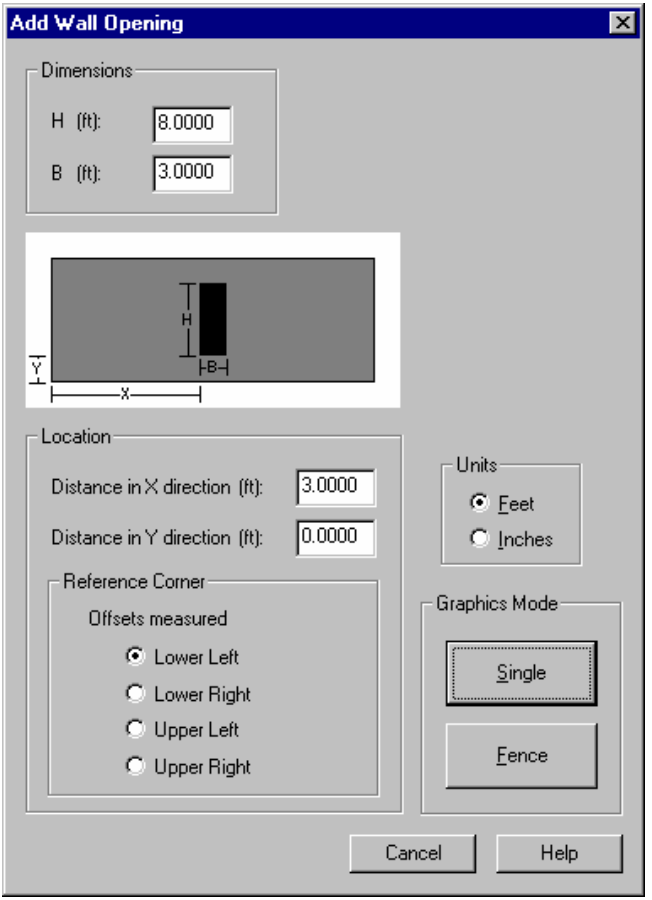
This completes the layout of the frames. Now you can assign the fixity for all concrete beams.

- Select the 2nd level plan.
- Change the current material to Concrete.
- Select **Layout – Beams – Fixity**.
- Make sure all degrees of freedom are set to Fixed and **[Fence]** the whole floor.
- Select **File – Save**.

## *Wall Openings*

A lateral wall can be modeled with openings. Opening for doors and windows can easily be placed in the walls. Openings can also be modeled that cross the edge boundaries of wall elements, but if the top edge of the wall is going to be clipped by the opening we recommend that the wall be split into separate pieces. Otherwise the floor framing might frame into the opening. To model an opening:

- Select **View - Elevation Mode** and select the wall on line B.
- Select **Layout - Walls - Wall Openings - Add**.




- For the Dimensions, type 8 (2.5) for H and 4 (1) for B

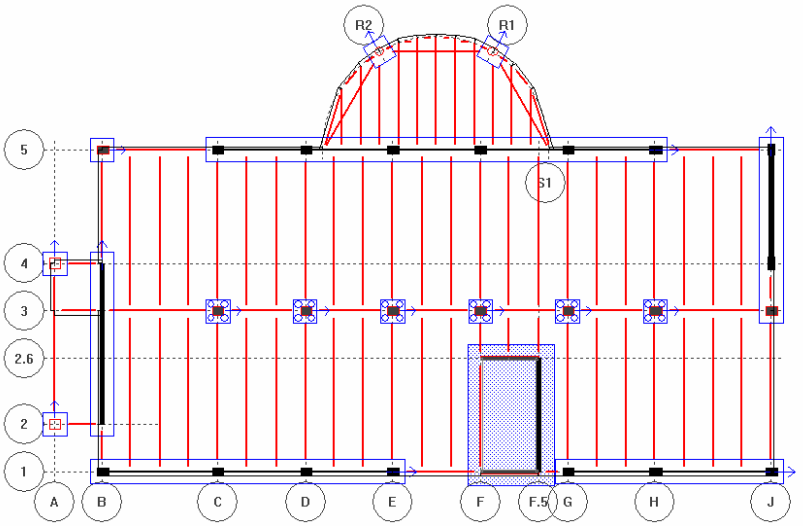
- For the Location, set the Reference corner to the Lower Left and type 3 (1) for the Distance in the X direction.
- Click **[Single]**.
- In the graphics mode, select the longer wall under the braced frame.

The opening will show as a black rectangle on the wall near the south edge (see the Frame elevation figure). The opening is tied to this wall and if it should be altered (due to a change in the story height or a modification to the grids for example) then the opening will maintain its position to the Reference corner. When an opening spans across more than one wall segment, it is still associated with one wall or the other.

Wall openings can be changed using the command Layout – Walls – Wall Openings – Change. They can also be deleted or reviewed using the delete and show options respectively. The wall openings are assigned numbers and the numbers can be shown using Options – Show Property – Wall opening numbers.

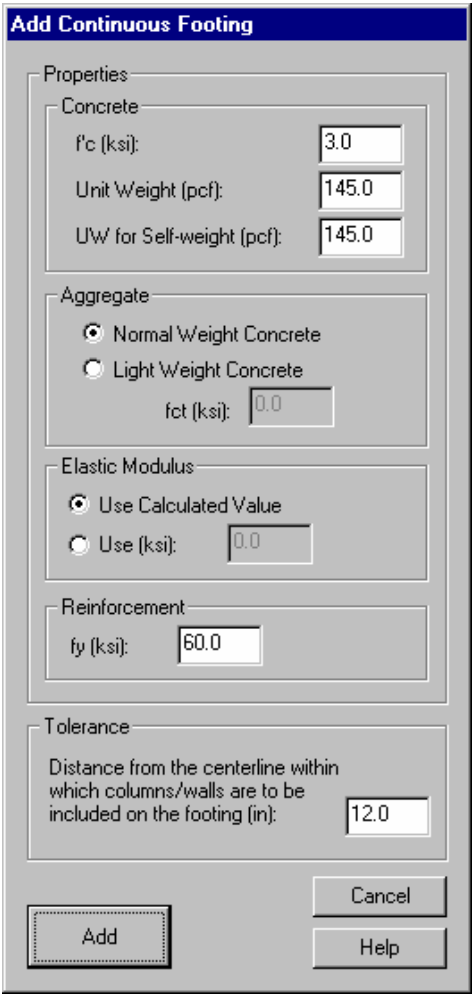
## *Layout Foundations*

The RAM Structural System also includes a module for the design of foundations. Like the other modules, the foundations need to be modeled in the Modeler before they can be designed and they can be modeled even if no license for the Foundation Design module is available. Below is a reference for the final foundation layout. Note that the foundations are modeled on the lowest framed level, in this case the 2nd floor. There is no need to define a separate foundation level.

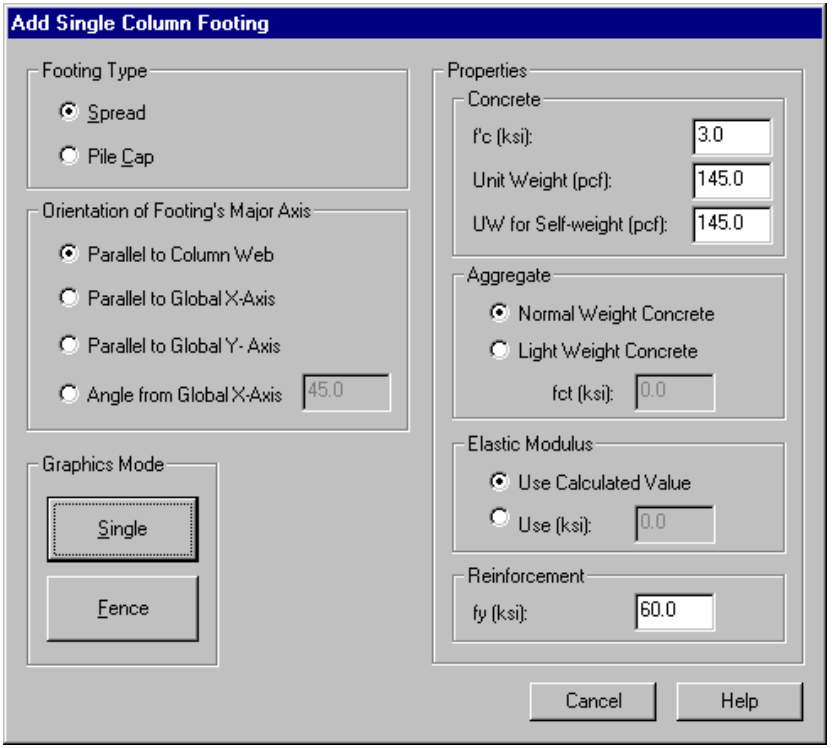



Foundation Layout

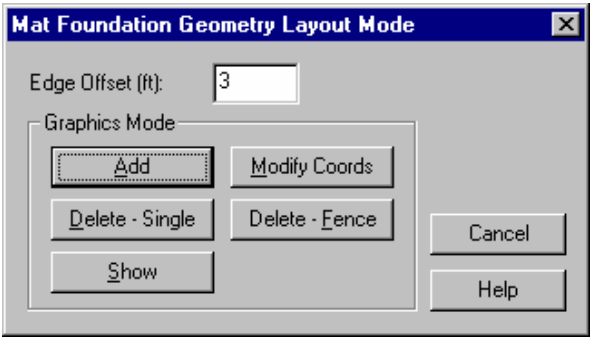
- Select **Layout - Foundations - Continuous - Add**.
- Set the f'c to 3 (27.5).
- Set Unit Weight to 145 (2400).
- Set UW for Self-weight to 145 (2400).
- Set Fy to 60 (425).



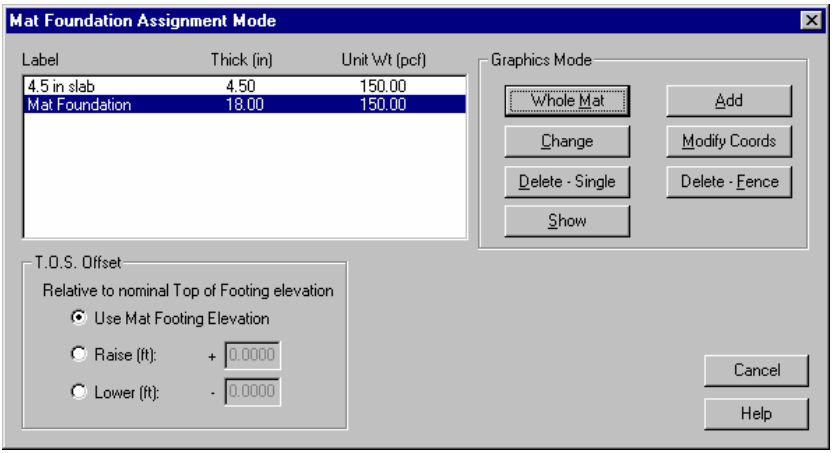

- Click **[Add]**.
- In the graphics mode add a continuous footing by clicking once at Grid B-1 and again at Grid E-1.
- Add another footing under each of the moment frames.
- Select **Layout – Foundations – Single Column – Add**.




- Select all the same variables used with the continuous foundation.
- For the Orientation of the Footing's Major axis, use Parallel to Column Web.
- Click **[Single]**.
- Add a footing under each steel column and also the concrete column at B-5.
- Right click to return to the Add Single command.
- Change the Footing Type to Pile Cap and click **[Fence]**.
- Fence the interior columns on Grid 3.
- Select **Layout - Foundations - Mat - Geometry**.
- Type 3 (1) for the Edge Offset and click **[Add]**.




- With the cursor, click at each of the 4 corners of the elevator. The program will add the perimeter of the mat foundation to an area 3' larger than the pit on all sides.
- Select **Layout – Foundations – Mat – Properties**.




- Select the Mat Foundation slab and click **[Whole Mat]**.
- In the graphics mode select anywhere within the mat perimeter and it should be filled in with a hatch pattern.

Note; foundations should be placed at the level where the column or wall stops. This is typically the lowest level of a structure, but foundations may also be placed at elevated levels (e.g. in cases where you are modeling a partial basement). The bottom of all columns and walls will always be supported whether a foundation is modeled or not.

## *Renumber Members & Data Check*

The program allows you to automatically reorder the members so that the first beam occurs in the lower-left hand corner of the plan. The member numbers increase as you move left-to-right and bottom-to-top across the screen. To renumber the members in the structure:

- Select **Options – Renumber Members**.

If you would like to see the member numbers on screen:

- Select **Options – Show Property** and choose from Column Numbers, Beam Numbers, Wall numbers, etc.

The modeler includes a Data Check feature that verifies the layout of the model. If there are errors in your model, the Data Check will print a detailed list of the errors and the steps necessary to correct the errors. The Data Check can be invoked at any time during modeling to check for errors. **Only the levels that are included in the Story Data will be checked by the Data Check**. To perform the Data Check:

- Select **Data Check**.

The Data Check Options dialog box will appear:

- Select Integrated.
- Click **[OK]**.

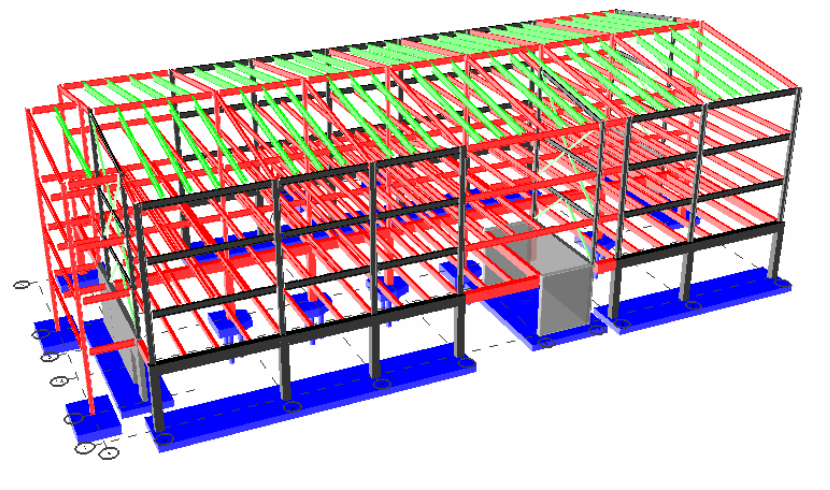
If you have only the RAM Steel or RAM Concrete Design Modules, or if you only want to check the gravity design sections of a model then select Gravity Only. The Frame Only option is used to verify that lateral members have an assigned size and that all lateral members are supported by other lateral members all the way down to the ground (plus a few other checks). The integrated data check does both.

If you receive any errors or warnings they can be viewed on screen. Review those aspects of the model and follow the instructions given on how to solve the problem. Refer to the technical portion of the RAM Modeler manual if you need further assistance.

Your model is now ready to be used to design the gravity and/or the lateral system. You can now proceed to Beam, Column or Frame tutorials from here depending on the Modules you have licensed and information you want to review.

- Select **File - Save**.
- Select **File - Exit**.

Note; if you have not completed the modeling of the structure as documented in the tutorial you may also close the Modeler and open the file called RAMTutorial_v10_US.rss which is installed with the program. It should be located in a sub-folder of default Data directory called Tutorial.



The completed model.

# RAM Steel Beam Design

The RAM Steel Beam/Joist Module optimizes steel beams, Smartbeams and open web steel joists. Unsized lateral beams are also assigned an optimum preliminary size based on gravity loads only (and without consideration of end fixity or braces). Gravity beams and Smartbeams may be composite or non–composite but must be covered entirely with composite deck to be designed as such. All beams in this module are assumed to be simple supported and they are designed for Dead and Live Loads only. This section can only be completed if you have installed the RAM Steel program and have a license available.

To invoke the gravity beam module from RAM Manager:

- Select **Design – RAM Steel Beam**. You can also click the second square button on the left side depicting a steel beam.

When the Beam program is invoked for the first time after saving the model in the Modeler, the Framing Table Options dialog box appears.

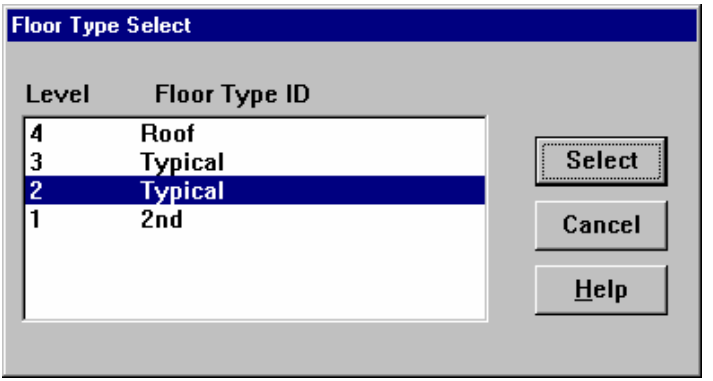

At this stage the program will apply surface, line and point loads to members, transfer loads to supporting members and calculate Live Load Reduction factors (this is referred to as “Building the Framing Tables”). The option to design the members is presented to the user before building the framing tables. Since the design criteria have not been reviewed:

- Leave the option unchecked and click **[OK]**.

## *Selecting a Floor type*

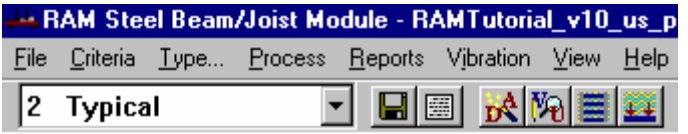
At first the graphical screen is empty. To see the typical floor in the model:

- Select **Type** from the menu bar. The Floor Type Select dialog box should appear:




- Highlight the level 2 Typical from the list box.
- Click **[Select]**.

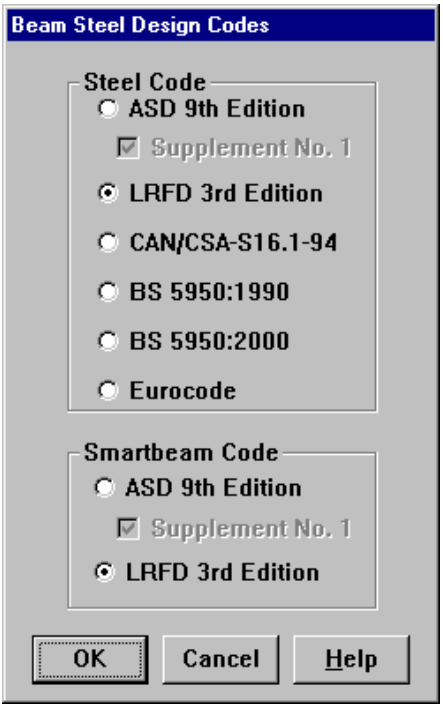
This sets the Typical floor to the current floor in much the same way as it is done in the modeler. The same can be accomplished by using the drop-down list box in the menu bar.



## *Design Codes*

Steel beams may be designed per requirements of five different codes including Allowable Stress Design (ASD), Load and Resistance Factor Design (LRFD), Canadian (CAN/CSA – S16.1 – 94), British (BS5950:1990 or 2000) and Eurocode. Smartbeams may only be design according to the ASD and LRFD codes. Joists are not designed by the program but rather selected from the manufacturer's table of Total and Live Load capacities. These tables are in the RAM\Tables directory, they have the extension .JST and may be edited with any text editor. For more information about design codes see the Design Codes Section in the RAM Steel Beam on-line documentation. For more information about joist tables see the Tables section in the Manager manual. To establish the design code for steel beams:

- Select **Criteria - Steel Design Codes**.

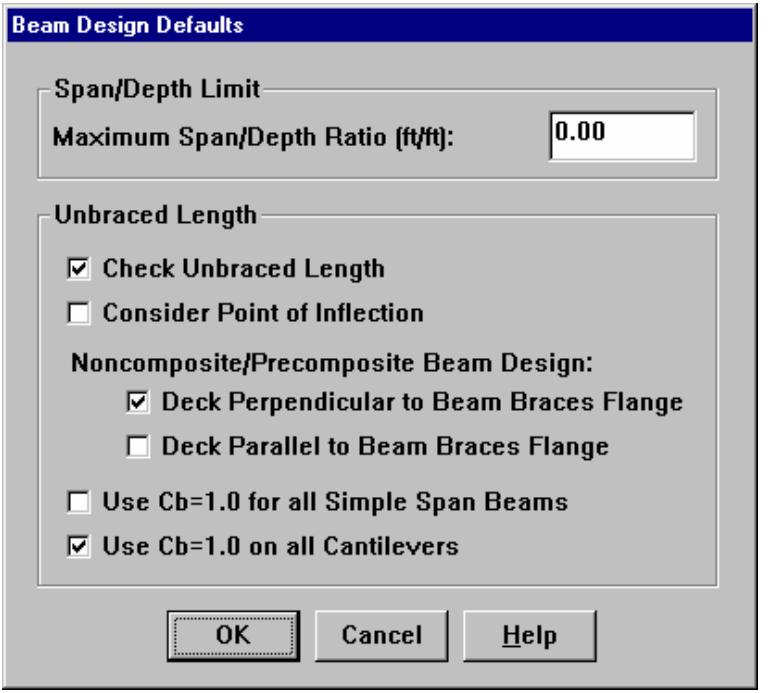



- Select LRFD 3rd Edition Steel Code (or use BS 5950:2000)
- Click **[OK]**.

## *Design Criteria*

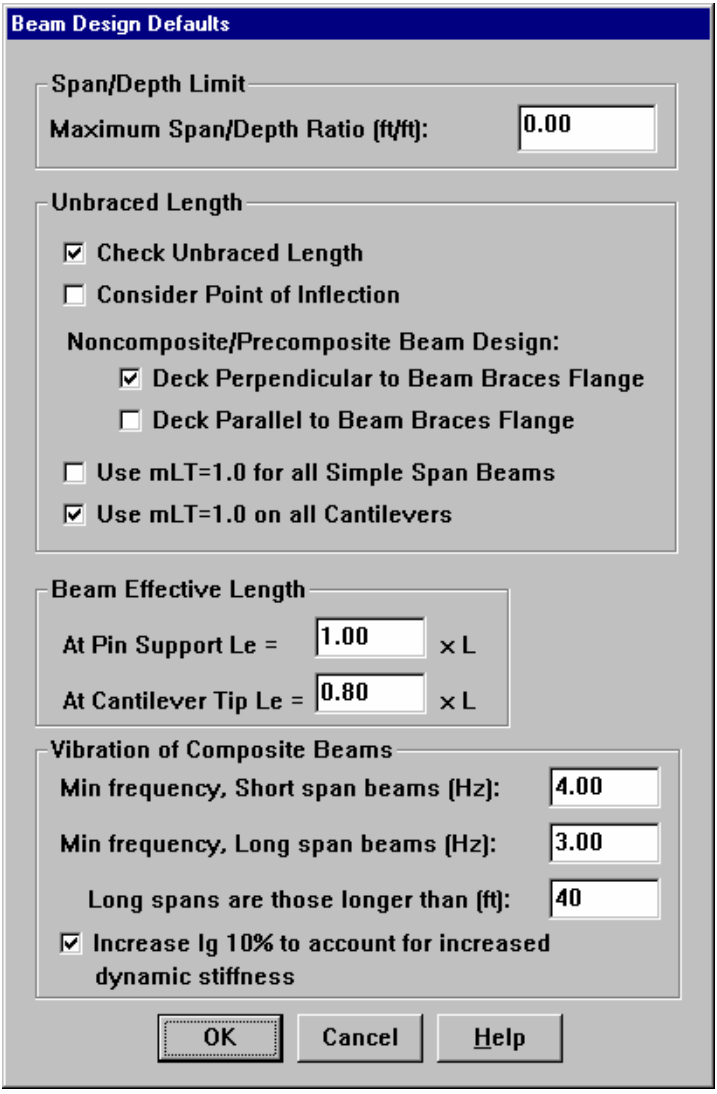
The designs of beams is guided by the various criteria set under the Criteria menu. For each of the design modules it is critical that the user understand the criteria completely. In this tutorial you can use the default criteria of modify the settings to match the values shown hereafter. Refer to the Steel Beam Module documentation for a more detailed explanation of the various criteria. Note; these criteria can also be set through the RAM Defaults Utility in RAM Manager so that future projects use the setting you choose.

- Select **Criteria – Design Defaults**.




- Modify the defaults as indicated if using LRFD.
- Click **[OK]**.

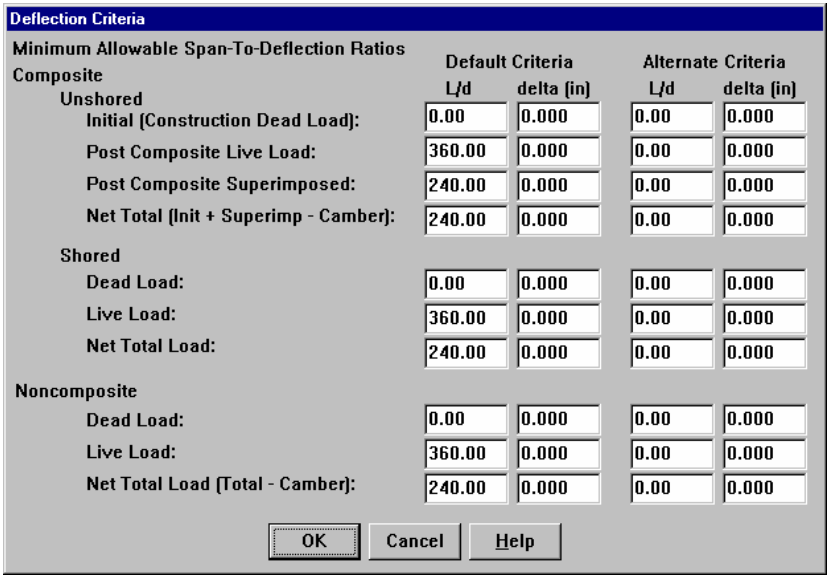
Note; when using different design codes, the wording of the criteria dialog boxes may vary slightly. In some cases there are even code-specific options to choose. Below are the same options when using the BS 5950:2000 code.




- Modify the defaults as indicated if using BS 5950.
- Click **[OK]**.

When the Design Criteria are altered, a warning will pop up indicating the following:

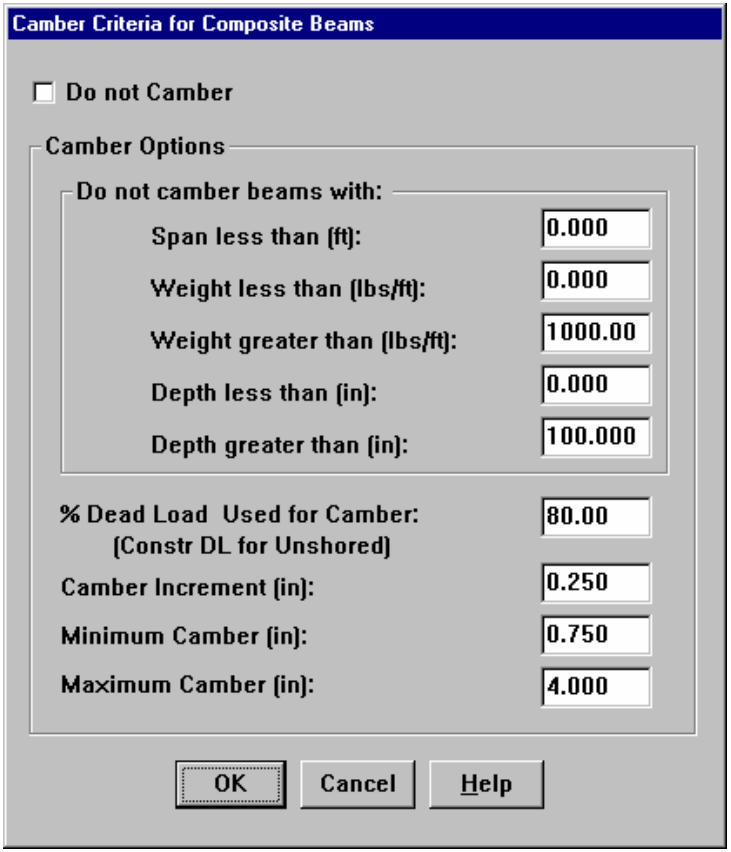

- Click **[OK]** to proceed.
- Select **Criteria - Deflection Criteria**.




- Set the L/d Default criteria for Unshored construction to L/360 for Post Composite Live Load, 240 for Post Composite Superimposed and 240 for Net Total.
- Set the Noncomposite criteria for Live load to 360 and the Net Total to 240.

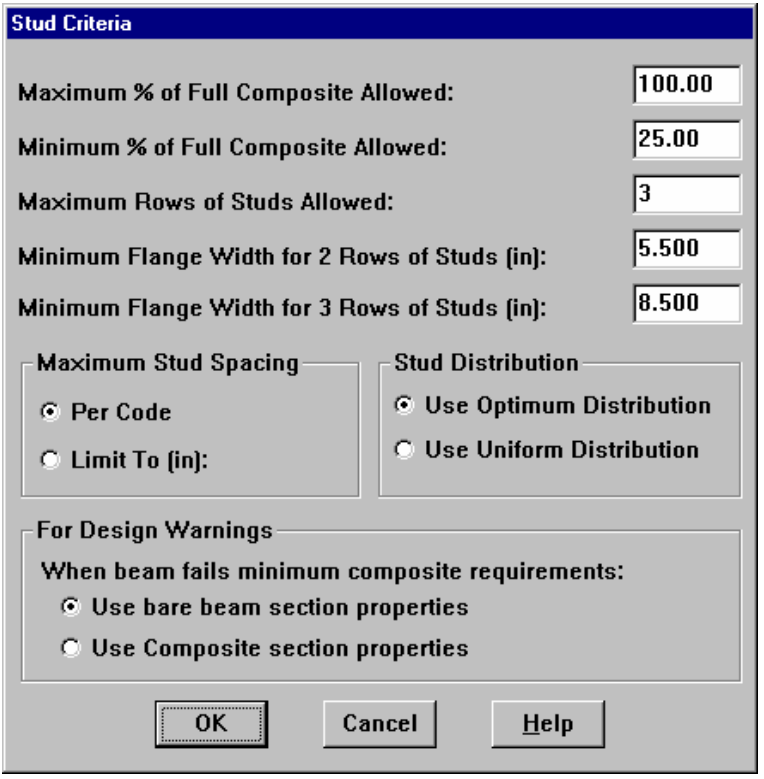
Note that this tutorial has noncomposite steel beams (and joists) as well as composite, unshored beams. There are no shored beams because there is no shored deck defined in the model.

- Select **Criteria – Camber – Composite**.



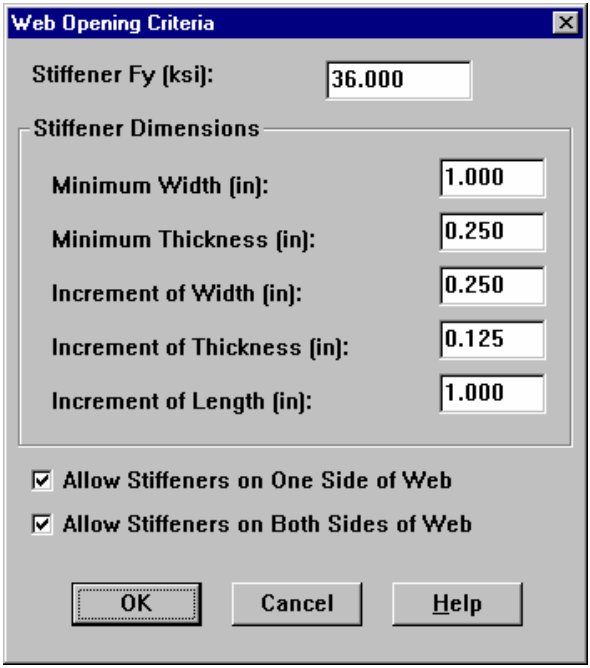

- Modify the defaults as indicated above (similar for SI models).
- Click **[OK]**.
- Select **Criteria – Camber – Noncomposite**
- Check the box that says Do not Camber.
- Click **[OK]**.

- Select **Criteria - Stud Criteria**.



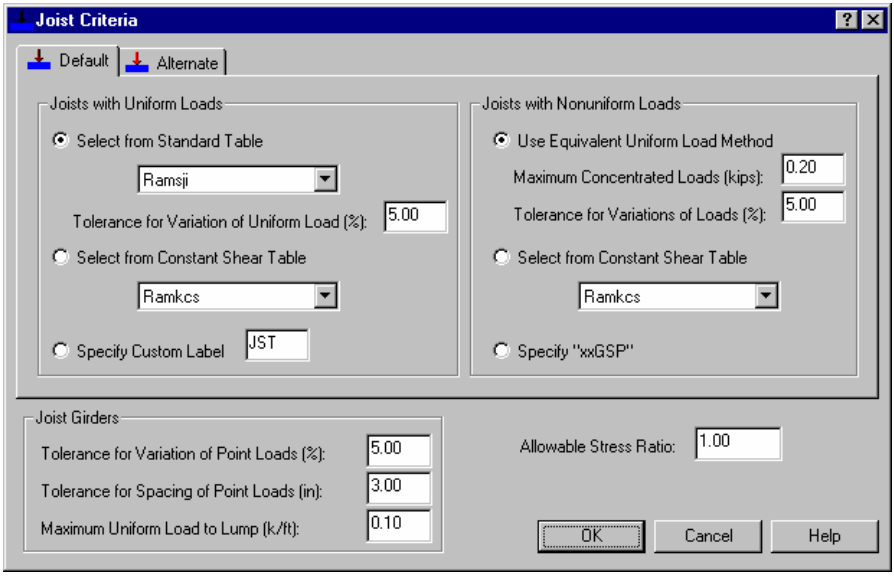

- Modify the defaults as indicated above (similar for SI models).
- Click **[OK]**.

- Select **Criteria - Web Openings**.




- Modify the defaults as indicated above (similar for SI models) and click **[OK]**.

- Select **Criteria - Joists**.




- Modify the defaults as indicated above.
- Click **[OK]**.

## *Design and Investigation*

Beam members are now ready to be designed. The **Process – Design All** menu item may be selected to perform the design of all beam members or the **Process – View/Update** menu item may be selected to look at one particular member.

- Select Process - Design All.

The program will design all beams, Smartbeams and joists. In order to track beam self weight reactions, the program designed the top level infill beams first, then moves to the girders and then repeats for the levels below.

A dialog box may appear indicating that there are design warnings in the model. If so click [Yes] to review the Design Warnings report. Click the "X" in the corner to close the report. Design warnings always include the beam number. Beam numbers can be displayed in the Steel Beam module using the command View – Show Beam Numbers. A beam can be also be found in the Modeler or in the 3D view with a special find-member command.

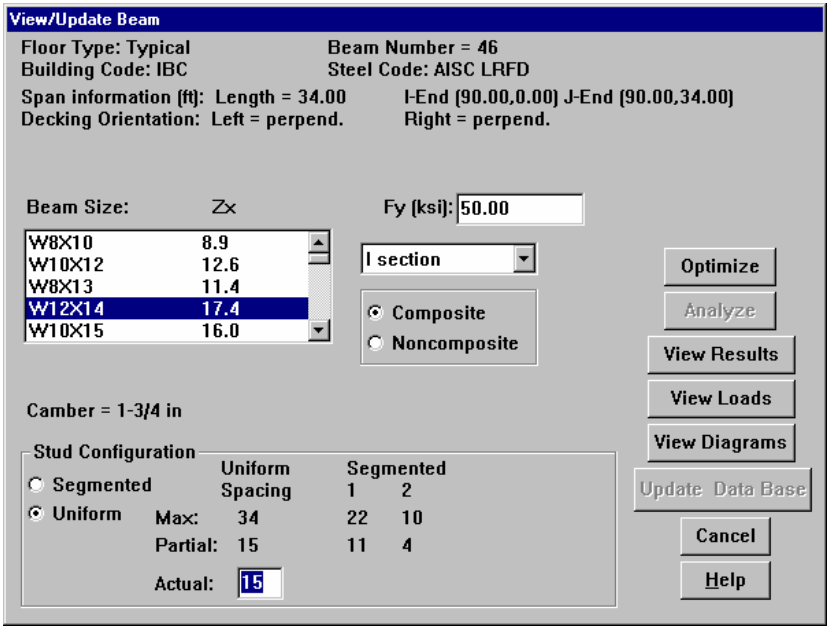
- Select View - Show Designs.

Note; the View – Zoom command may be used to better see the selected beam sizes. The View – Scale Text command can be used to adjust the text size as necessary. All of these controls can also be accessed through toolbar buttons.

To investigate a selected member size:

- Select Process - View/Update.
- Notice that the cursor now has a "target" shape. Also notice that at the bottom left corner of the screen there is a prompt for the expected action saying "Select a Beam to Edit or Update".
- Click on the beam on Grid F from 1 to 3 (in front of the elevator).

The View/Update Dialog box pops up. This dialog box is used to investigate, modify and update the design of single members. You can change size, shape, yield stress, section type, composite flag or stud configuration and re-analyze the beam. For a complete explanation of how the program designs beams refer to the RAM Steel Beam documentation.




To view the loads on the beam:

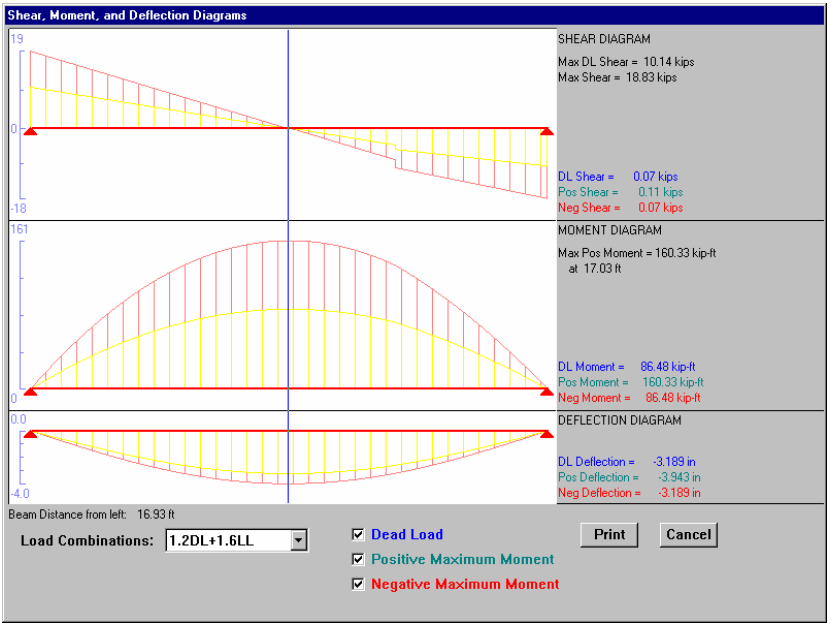
- Click **[View Loads]**.

A Loads Report will come up showing the line and point loads on the beam. When there is a change in the uniform load at a given location, the value is listed just to the left and right of that point.

- Click the circled **X** to exit the report.

To view the shear moment and deflection diagrams for the beam:

- Click **[View Diagrams]** The following interactive dialog box appears:




Notice that force and deflection magnitudes on the right hand side reflect the cursor location on the graphic. The diagrams for this beam may be printed by clicking [Print] button in this screen.

- Click **[Cancel]** to return to the View/Update dialog box.

To inspect the complete design results for the beam:

- Click **[View Results]**.

A Design Report will come up showing various calculations for the beam. This includes information about the composite properties of the member, controlling moments used in the design (along with the applicable unbraced length) and deflections. If the report is more than one page long, click the forward triangle or press the Page Down key to see the next page. Take time to become familiar with this report.

- Click the circled **X** to exit the report.

To check the design of a different section:

- Select W14x22 (UB356x127x33) from the Beam Size list box.
- Click **[Analyze]**. Another dialog box will pop up showing the design. In this case the required number of studs is less than in the

original design. In other situations, the user selected size might produce a design warning. Click **[OK]** to accept the new stud configuration.

At this point you can go on to review the loads, diagrams or design results for the new trial section. If you are satisfied with the results and you would like to override the optimum selection,

- Click **[Update Data Base]**.
- A warning will appear indicating that the new information will override optimization.


- Click **[OK]**.

Note; when the Update Data Base command is performed, the new section size will be assigned to that beam permanently (the design is "frozen"). You will receive a design warning if that design should ever fail the strength or deflection checks that the program performs, but a new size will not be selected unless you clear the beam size. This can be done from within the Modeler using a Clear Size Command, or in the Beam Module by selecting Process – Clear Size – Single.

At this point you may wish to review other beams or joists in the model. The cantilevered beams on the 2nd level and the Joists on the Roof level are of particular interest.

When finished, look into the lower-right corner of the screen. A circle (a.k.a. traffic light) indicates the status of the model. If the circle is yellow, that indicates that there has been some change to the model that may affect reactions on other members. To make sure all the designs are current:

- Select **Process - Design All**.

## *Reports*

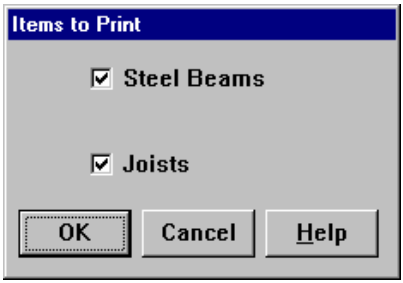
All printable reports except for the shear, moment, and deflection diagrams are available under the Reports menu item. The Reports – Map Fence and Reports – Map Floor reports print the plan view of a floor type with options to show surface, line and point loads together with beam designs. The Framing Check and Connection Check reports check the intersection of beams and report possible fit-up problems. For a better explanation of the various reports available see the Beam Manual.

To see any of these reports on the screen first:

- Pull down the repots menu. A check mark should appear next the Screen menu item. If not, select Screen now. The other options (Printer, Text File, Viewer file) can be used to send the reports directly to the print or to file.

The Summary Design report lists all gravity beams by Floor and by Number with the controlling Moments, Section Size, Studs, etc. To see the Summary of beam designs for the tutorial model:

- Select **Reports - Summary**.




- When the "Items to Print" windows appears, check both Steel Beams and Joists, then Click **[OK]**
- Click the circled **X** to exit the report.
- Feel free to review other design reports at this time. Of particular interest is the Material Takeoff report. The Connection design report only functions after a connection table is written. Refer to the RAM Manager documentation for instructions on creating connection check tables.
- Select **File - Exit**.

# RAM Steel Column Design

The RAM Steel Column Module is the module where gravity columns and gravity base plates are optimized. Unsized lateral columns are also assigned a preliminary size. All columns in this module are assumed to be simply supported and they are designed for Dead and Live Load only. Base plates are optimized for Dead and Live Load only although the columns will be designed for moments induced by the eccentricity of the supported beam connections. This section can only be completed if you have installed the RAM Steel program and have a license available. To invoke the Steel Column module from RAM Manager:

- Select **Design – RAM Steel Column** from the Manager. You can also click the third square button on the left side depicting a steel column and Base Plate.

The graphical area will then show a 3D view of the building with the columns colored yellow and the other members in grey. This coloration scheme is what we call Design Colors. You might think of them as status colors. The various colors and the meanings are listed below:

Yellow – Columns are ready to be designed

Green – Successfully designed columns, selected by the program

Blue – Columns selected by the user that pass all code checks.

Red – Columns that fail one or more code checks (red overrides green or blue)

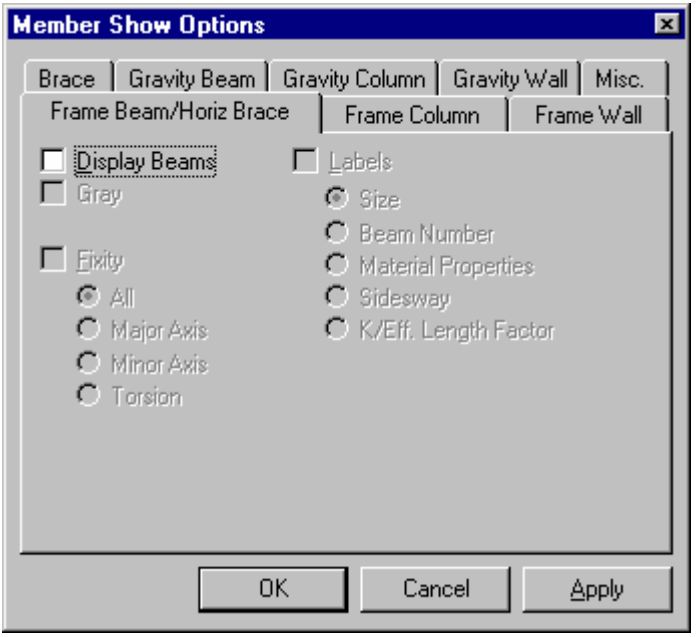
- Select **View – Colors – Model Colors**.

The graphics will change to the pen colors used in the Modeler. These can be altered on a model-by-model basis or they can be altered in the defaults utility, by the way.

The third Color option called Interaction Colors is not available until after the columns have been designed.

Besides altering colors, you may also want to change the information or members displayed in the graphics window. To alter the text or members that are shown further:

- Select **View – Members**.




This dialog box is organized by the various types of members. Each can be labeled with a variety of labels or turned off completely.

- Click on the Misc. tab and Uncheck the Display of foundations.
- Click **[OK]**.

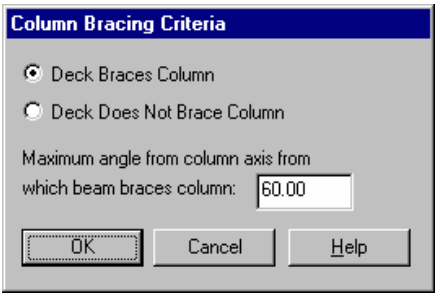
Sometimes it's easier to work in Plan or Elevation mode than in 3D. If that's the case you can select View – Elevation or View – Plan. An additional option in the Steel Column module is to View – Column Plan. This takes you to a compressed aerial view where you can see the entire 3D structure from above with no perspective.

As with the Beam design module, a light in the lower right corner indicates the current status of the file. When the designs are all current the light will be green.

## *Brace Points and Splice Levels*

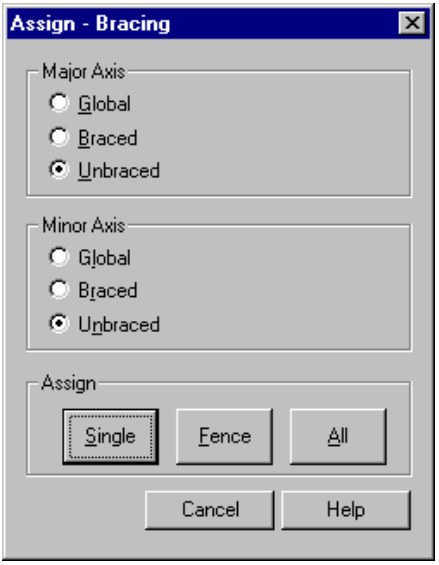
The RAM Steel Column module automatically detects the brace points for columns based on the framing. When a beam or joist frames into a column it braces the column in that axis. In the case of skewed framing, there is a maximum angle for which this is true.

- Select **Criteria – Bracing**.




Notice that the deck can also be sufficient to brace interior columns when there are no beams framing into it. In some cases you may wish to alter the program determined brace points. To do so now:

- Select **Assign – Bracing**.



As soon as the command is started, the graphic will indicate the current brace locations using green triangles in the plan of the bracing.

- Set the Major and Minor Axis to Unbraced and click **[Single]**.
- Select each of the lowest three levels of the column at A-2 to illustrate the process. Do not select the top level.

Column splices are also preset when entering the Steel Column Module. When we defined the story data in the modeler, this also established the splice levels. When the story data was specified to have a splice at the 4th floor (story #3), that tells the program to change size for the columns above this level. The columns between 3rd and 4th floors won't change size. There are exceptions to this rule, places where the program will splice the column at non-splice levels. These automatic or temporary splice location result whenever the column changes material, shape or orientation between levels. Lateral columns can also have splice at non-splice levels since the column sizes are specified level-by-level.

- To view the splice locations Select **View - Splicing**.

The splices are indicated as red squares at the levels where the splice occurs. In practice, the splice will be a few feet above the floor but for the purpose of design, the column size changes just above the level.

- Turn off the display of Splices using the toolbar button or use the View menu option again.

## *Design Criteria*

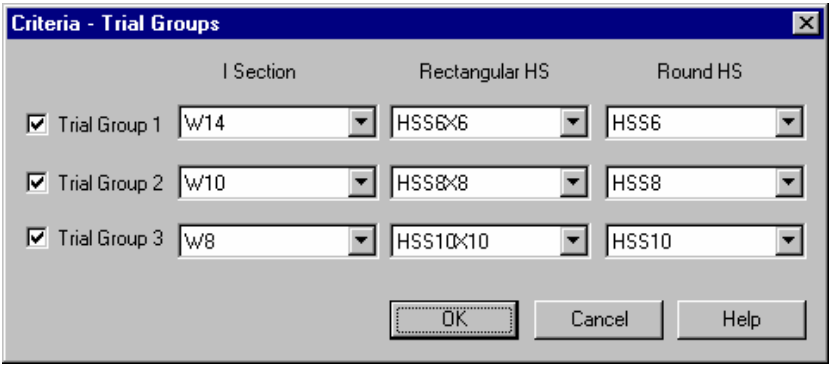
Columns and Base Plates may be designed per requirements of five different codes including Allowable Stress Design (ASD), Load and Resistance Factor Design (LRFD), Canadian (CAN/CSA - S16.1 - 94), British (BS5950:1990 or 2000) and Eurocode. For more information about design codes see the Design Codes Section in the RAM Steel Column documentation. To set the design codes:

- Select **Criteria – Steel Design Codes**.
- Select LRFD 3rd Edition (BS 5950:2000) for the Column and Base Plate Steel Design Codes.
- Click **[OK]**.

The Gravity Column program uses trial groups to optimize column sizes. In the Modeler, columns are assigned shape (I, HSS, Pipe), material strength and orientation. The design of the columns is performed by selecting the lightest adequate column size from a maximum of three column groups.

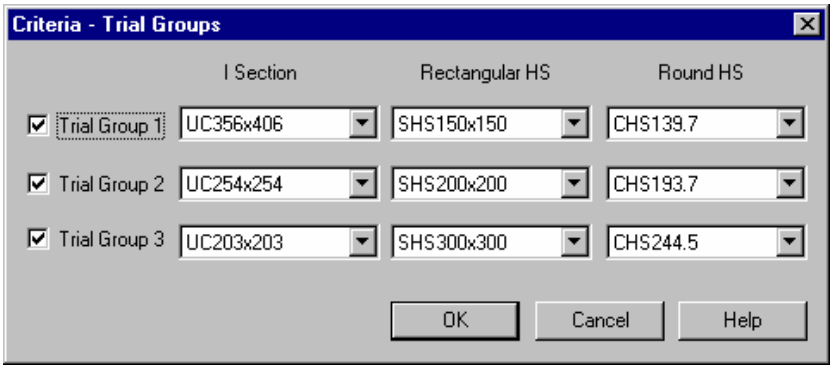
Column groups are identified by the section depth in the designation (e.g. W12, UC254, etc.). The program will select the lightest column from the available column groups which are defined in the Column Design Table.

- Select **Criteria – Trial Group Defaults**.




- Check to use each of the three trial groups. If you want to consider fewer options, uncheck one of the Trial groups on the left.
- For each shape there are a number of trial groups available to choose from.

- For US models, select the trial groups listed in the figure above. For SI models using the RAMUK tables, select the following:




- Click **[OK]**.

If any changes are made to the trial groups, you will receive a message asking if you want to assign new trial groups to existing column lines.

- Click **[Yes]**.

Note; this criteria applies to the whole model globally. When one column line needs to be designed using a different sizes, they may be assigned to that column line by selecting the **Assign – Trial Groups** command.

## *Column Design and Investigation*

The columns are now ready to be designed. The **Process – Design All** menu item may be selected to perform the design of all columns or the View/Update menu item may be selected to look at one particular column line.

- Select **Process – Design All**.

Once complete, the view will automatically shift to showing the interaction colors. A pop-up box labeled Color Scale appears. Note; this box may be moved if it is in the way of a menu item or icon you wish to use.

- Click **[Show Values]** to see the interaction values on screen.

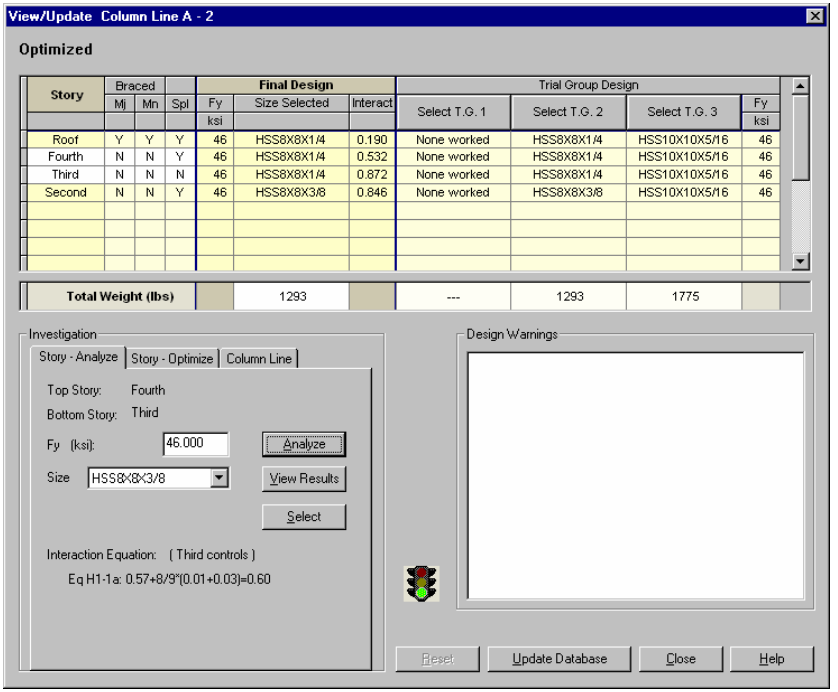
Notice that all of the columns are checked, even the lateral columns. It is important to note that the design of the lateral columns is not complete. So far, the columns have been checked for gravity loads only (and those loads are based on simple, tributary areas). The effect of moment connections, braces, and lateral loads is only examined in RAM Frame.

When a column spans multiple levels, each segment of the column will be checked independently, but each will use the overall, multi-story unbraced length in the design. To review the results for a particular member:

- Select **Process – View/Update**.
- Click any part of the Column at Grid A-2.

The interactive View/Update dialog box that appears has many important functions. In the upper left corner is the final design, the sizes selected by the program for each level. If there are more than 8 stories, the slider on the right side will allow you to scroll down. Information about the splicing, bracing and yield stress of the columns as well as the current interaction ratio also appear in the upper left area.

In the upper right area are the results for the three trial groups (assuming three were used). The program will select the lighted working design from the three trial groups. In this case, there were no HSS6x6 columns sufficient to support the loads with no intermediate bracing so the Trial Group 1 indicates “None Worked”. Either HSS8x8 or HSS10x10 will work, but the 8x8 sections are lighter, so they were selected by default.




When you initially open a particular column, the top segment of the column will be highlighted in white between splices. In this case, that is the roof level only.

- Click on the Story cell labeled Fourth.

Doing so causes the program to highlight the Fourth and Third levels together.

- Click **[Analyze]**.

The result of the controlling column segment (either the Third level or the Fourth) will be indicated in the space below.

- In the Size Drop down list, change the section size to HSS8x8x3/8 (SHS200x200x8.0).
- Click **[Analyze]** again.
- The interaction value listed is adjusted for the new size.

- Click **[Select]** to move that size up to the final design section for those two levels. The interaction values above will then be adjusted. If the design should fail, the interaction value will be written in red. A design warning will also appear in the lower right corner.
- Click in the Roof cell under Story to select the Roof level.
- Change the size to HSS8x8x3/8 (SHS200x200x8.0) and **[Select]** that for the top level.

To review the results for the selected column:

- Click **[View Results]**.

The design report starts with the top level. Click page down to proceed to the other levels. Notice that the columns are being designed for axial loads and moments using the full unbraced length. The moments are a result of the eccentricity of the beam connections. The column eccentricity may be set in the Modeler using **Layout – Columns – Assign Eccentricity**. The program considers all possible patterns of live load when evaluating the column design and the governing combination of patterns used at the top and bottom of the column determines the design data that is reported here. To turn off the skip loading altogether, go to Criteria – Design Defaults. Refer to the technical notes in the RAM Steel Column documentation for an explanation of the design methods and results.

- Click the circled **X** to exit the report.
- If all levels indicate the larger size in the final design area, click **[Update Database]** to make the changes for the whole column line permanent.

In the graphics mode, the column now appears with the modified interaction colors. Feel free to view/update other columns in the model.

The Story – Optimize and Column Line tabs provide other options for investigation. Using the Story – Optimize tab for example, you can see what size “I” section would work for the column at A-2 even though it was initially modeled as a Tube.

## *Copy and Clear User Sizes*

If you have one column saved with the sizes you want to keep, that column line design can be easily copied to other column lines.

- Select **Process - Copy** from the menu bar.
- A target cursor with an arrow pointing to 2 o'clock now appears. At the bottom left corner of the screen the program indicates that you should select the column line to copy from:
- Select the Column at Grids A-2. The cursor changes slightly and a new message at the bottom left corner of the screen indicates to now select the column line to copy to.
- Click the Column at Grids A-4 (you can rotate the model or view in it in elevation mode if necessary).

At this point, the program is going to make the column at A-4 just like the column at A-2. This saves you the step of performing a View/Update on all columns with identical designs. Note; if the new design is insufficient for the column at A-4, then a warning message will pop up asking if you want to cancel the operation.

At certain stages of the project you might want to freeze the column designs without making any changes to the program selected sizes.

Select Process – Freeze Design – Column line.

The Target cursor will appear and any column line you select will be frozen. The sizes will not be re-optimized by the program unless cleared first. Using the [Update Database] button in the View/Update dialog box is the same as freezing a column line.

User assigned or frozen column sizes may be cleared by using one of the Process – Clear Design commands from the menu. The columns can then be redesigned using **Process – Design All**.

## *Reports*

Several reports are available from the Reports Menu including all Summary Reports for column, column loads and base plate design.

Because the program alternates Live Loads on columns to obtain the controlling combination of unbalanced moment and axial loads, the Column Summary report should not be used to obtain maximum loads at column bases. The total loads are reported in the Loads Report or Loads Summary report. A steel takeoff of all gravity columns is also available from the Reports menu:

- Select **Reports – Takeoff**.

Click the circled **X** icon to close the report.

- Select **File – Exit**.

Note; you can save your model at any time form any module, but you are not required to save before exiting the Modeler or any design module. If you close the Manager completely or change to another model, then the program will prompt you to save any recent changes.

# RAM Frame – Analysis

This section illustrates the analysis and design of the lateral frame elements in an integrated model. This section can only be completed if you have licensed and installed the RAM Frame module. You may begin with the model that you generated in the previous portions of this tutorial, or you may open the model called RAMTutorial_v10_US.RSS from the RAM Manager.

## RAM Frame Basics

A little background information is needed before beginning work with the RAM Frame program. RAM Frame has three modes of operation, Analysis mode, Steel Post Processing Mode and Drift Control Mode.

In **Analysis Mode**, the structure is analyzed for individual load cases. Results for the member forces, reactions, drift etc can be obtained for the individual load cases in the **Load Cases** sub-mode, or the results can be combined in the **Load Combinations** sub-mode.

In **Steel Mode**, the previously analyzed load cases are combined and used to determine their design status of steel members. Various design codes can be selected to perform code checks. The steel Mode is sub-divided in to **Standard Steel Provisions** (e.g. LRFD 3rd edition) and a **Special Seismic Provisions** (e.g. AISC Seismic Provisions for Structural Buildings).

The third mode, **Drift Control** allows the user to investigate the relative participation of the various members with the structure related to the control of drift. The Steel mode and Drift Control modes are discussed in the following sections of this tutorial.

Concrete members that are designated as Frame Members are analyzed in RAM Frame, but their design is performed in RAM Concrete.

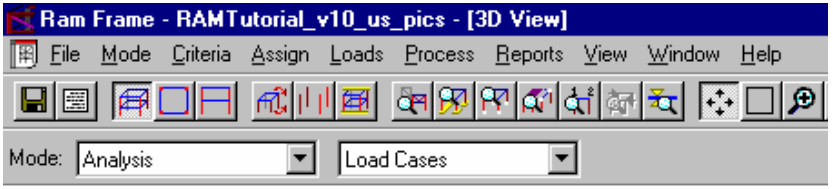
To invoke RAM Frame from the RAM Manager:

- Select **Design – RAM Frame** or click the 4th square button that depicts part of a braced frame.

When the framing process is complete, a three dimensional wire-frame view of your model will appear on the screen.

The RAM Frame program has a toolbar from which many commands can be issued with just a click of the mouse button. As with the other modules, this tutorial will present the commands as selected through the menus. There are

two pull-down lists in the second row of buttons than can be used to switch the program mode and sub-mode. They are also useful for checking the mode you are currently in. The Status Bar at the bottom of the screen also tells you which RAM Frame mode you are currently working in. It also has a light to indicate the status of the current model. If the status indicator light is red, the model has not yet been analyzed. A yellow light is used when the results are available, but may not be absolutely current due to a change in member size for example.




Note; upon entry into the RAM Frame program, you are always placed in Analysis Mode – Load Cases Sub-Mode. An analysis of the load cases is required before the other modes can be entered.

The View menu has controls for displaying general model information such as finite element node numbers or to modify the rotation of the 3d view. To view the Wall Mesh for example:

- Select **View - Meshed Walls**.

Notice how the mesh works around the opening in the West wall.

Help is available in any of the following ways:

Allowing the cursor to rest on top of a toolbar button causes a Tool Tip to appear, as well as a brief explanation of the command in the status bar.

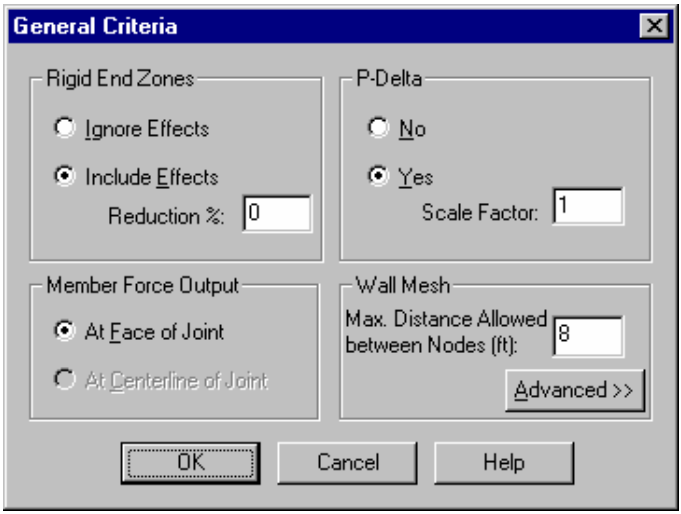
Each dialog box has direct access to its related help topic via a help button.

The Help index is also accessible from the main menu.

## *General Analysis Criteria*

Before performing the analysis you should always establish your criteria using the Criteria menu. To set the general analysis criteria:

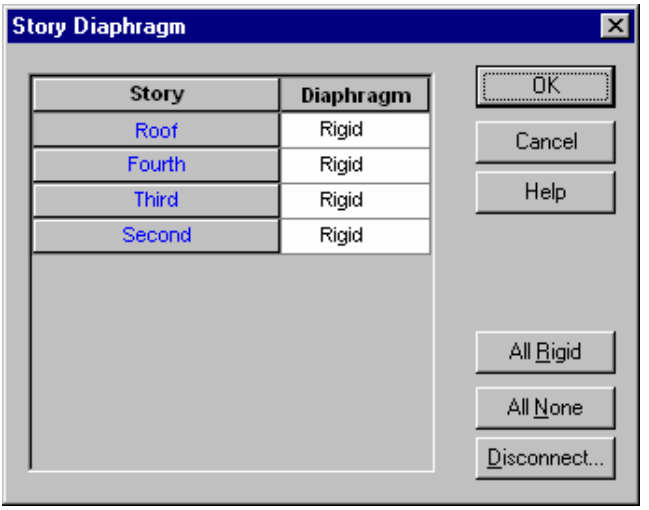
- Select **Criteria - General**.




- Set Rigid End Zones to Include Effects and Type 0 for Reduction %. This means that the beams in the rigid frames will be shorted in the analysis to the face of the column.
- Set Member Force Output to At Face of Joint (when using rigid end zones you have no choice).
- Set P-Delta to Yes and Type 1 for Scale Factor. This means that second-order, P-Delta effects will be calculated for all load cases utilizing the building mass DL as the P in the P-delta calculations. If you wanted to also consider part of the live loads in the P-Delta calculations, then you can increase the value somewhat.
- For the Wall Mesh set the Max. Distance Allowed between Nodes to 8 (2.5). This means that the program will mesh the walls in such a way that no single element is more than 8' on a side (though they may be smaller due to geometric constraints). More Mesh options are available by clicking the [Advanced] button.
- Click **[OK]**.

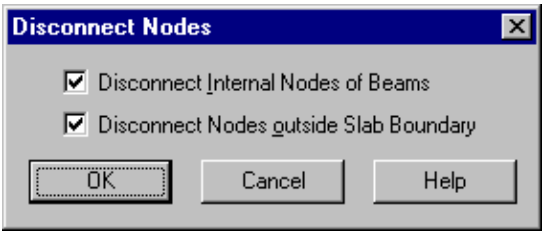
## *Diaphragm Criteria*

- Select **Criteria - Diaphragm**.




By default, all floor levels are assumed to be Rigid diaphragms. A rigid diaphragm creates a horizontal constraint for all of the nodes connected to it. For sloped levels, the diaphragm constraint is still horizontal. The size of the diaphragm is dictated by the extent of the slab edge.

- Click **[Disconnect]**.

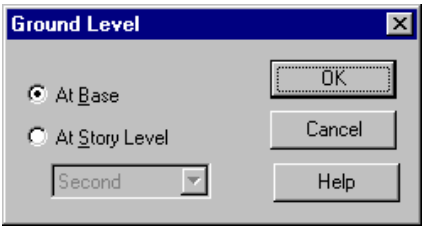



- Make sure both options are checked. This means that any lateral elements that fall outside of the slab edge will automatically be disconnected from that diaphragm. Internal nodes on beams occur in places like our chevron braces where the beam is intersected by another lateral member.
- Click **[OK]** to both windows.

## *Ground Level*

The Criteria – Ground Level command is used to specify the level at which the ground intersects the structure.

- Select **Criteria - Ground Level**.



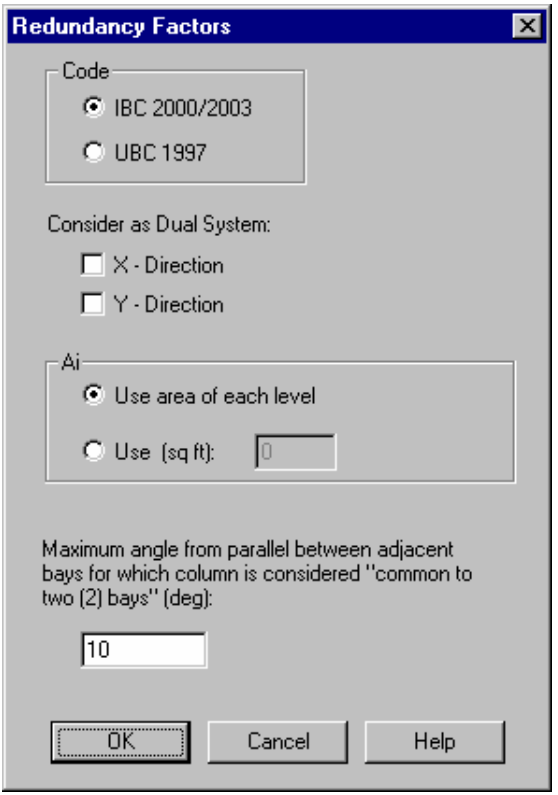

The default is for the ground level to be at the base of the model. In this tutorial the default will be accepted and no changes need be made. When a level other than the base is selected as the ground level two things will happen. First, the program generated loads will be adjusted for the new building height and the forces will be applied to the above ground levels only. Second, the ground level and any level below grade will be laterally restrained as if by a vertical roller.

- Click **[OK]**.

## *Redundancy Factors*

Redundancy factors are determined by the program and then used to modify the load factors applied to the seismic load cases in the generated load combinations. It's important to note that the redundancy factors always calculated for every seismic load case, even if your model is in an area of low seismic activity. They only apply to US codes.

- Select **Criteria – Redundancy Factors**.



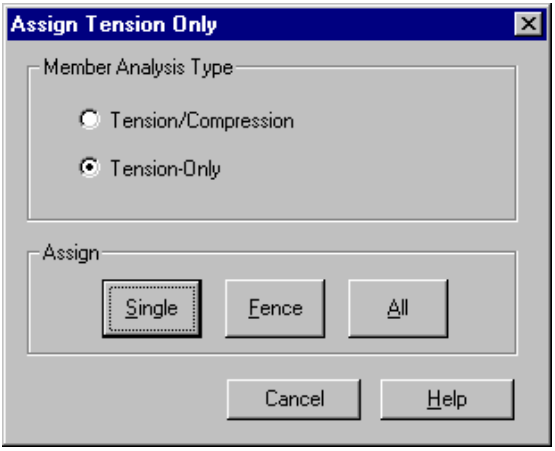

- For the Code choose IBC 2000/2003 (when code references are separated with a slash, it indicates that the two codes are identical).
- Set the other variable as indicated above and click **[OK]**.

## *Assign Menu Options*

In order for the program to perform a Finite Element Analysis, every lateral member must have an assigned size. For our model, that was done in the Modeler, but the assign menu in RAM Frame can also be used to assign sizes to lateral.

The assign menu has several other functions as well. To assign certain braces to be tension-only braces:

- Select **Assign – Braces – Tension-Only**.




- Set the Member Analysis Type to Tension-Only.
- Click Assign **[Single]**.
- In the graphics mode, select each of the "X" braces in Braced Frame 6. When you click on a brace a symbol showing an abstract turnbuckle should appear near the middle of the brace. When complete if you want to turn off the labels:
- Select **View – Reset Model**.

A variety of information, including the tension-only symbols, the member sizes, end fixity, etc. can all be displayed on screen using the command View – Members similar to the Steel Column module command of the same name.

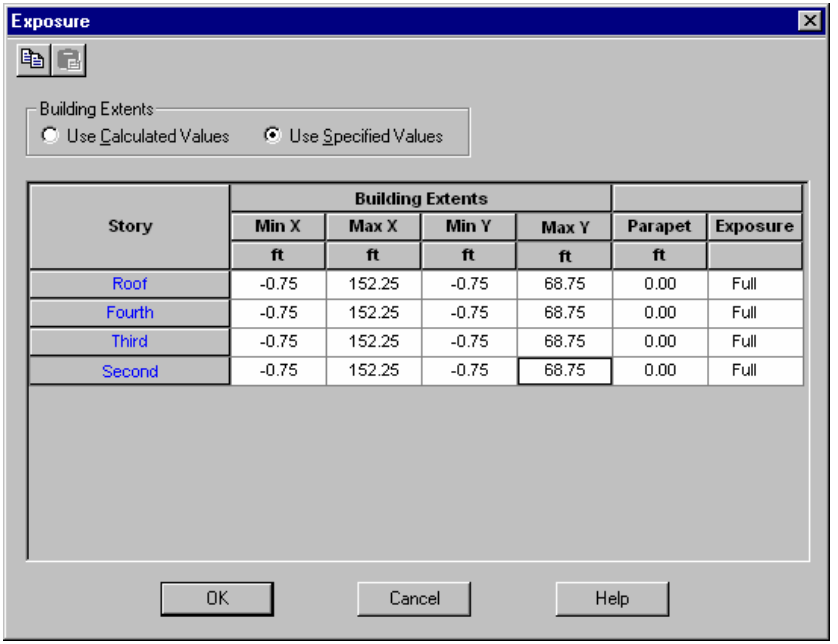
The assign menu commands can also use used to assign diaphragm connections, frame numbers, wall group numbers and foundations springs.

## *Mass and Exposure*

The load cases for which the structure is to be analyzed and designed must be defined. User defined and program generated load cases can be created. If the program is to automatically generate seismic load cases it requires information on the structure's mass properties, and for the program to generate wind loads, the building's dimensional properties are required. (Mass information is also used any time that second order analysis is performed).

In this model all of the information required by the program to generate seismic and wind forces is already provided. If desired, this data can be overridden. Only the building extents affecting wind generated forces will be modified in this example.

- Select **Loads - Exposure**.



The extents that were determined by the program using the slab edges of the model are listed in the Building Extents portion of the window. To change those values:

- Select Use Specified Values.
- On the Second Level, change the Max Y extent to be the same as the level above (we're removing the E-W wind loads applied to the canopy).

The Parapet field is used to indicate the height of the parapet. If a parapet height is specified, additional wind loads will be attributed to that level. If there is another level above the level with a parapet, then the additional wind loads will only get applied to the portion of the lower level that is wider than the level above.

The Exposure column is used to indicate if a particular level does not resist wind loads at all. This might be the case if there is only a partial slab at that level (such as in a mezzanine or stair landing between floors). By changing the exposure flag from Full to None a level is designated as having no exposure, and the wind force is distributed to the adjacent levels instead.

- Click **[OK]**.

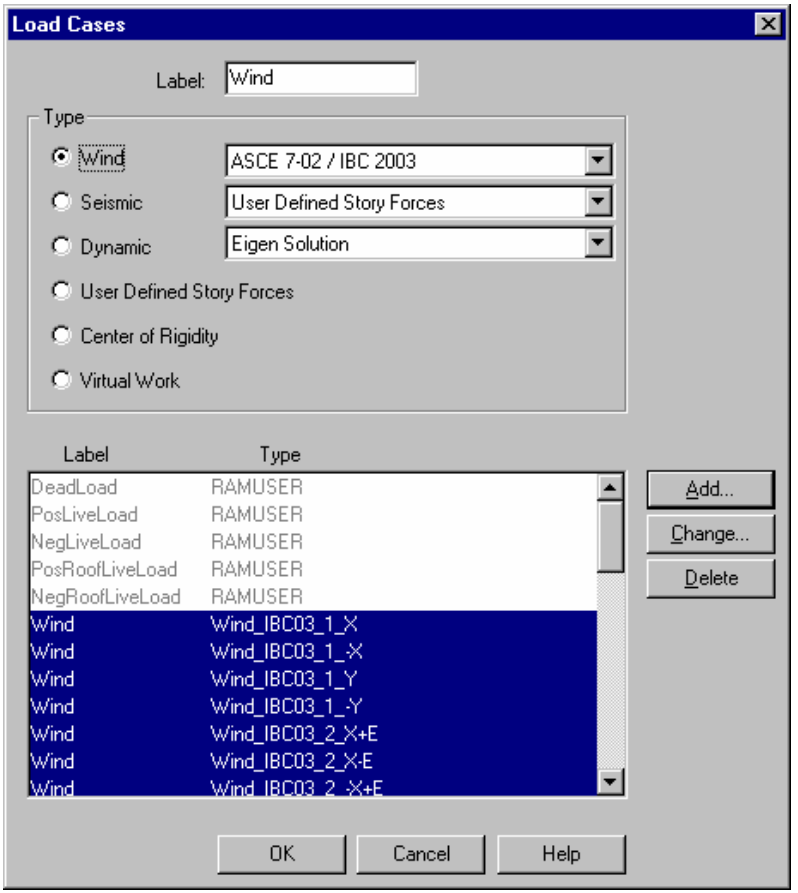
It's important to note that the program does not currently calculate wind forces on sloping roofs, nor does it calculate uplift pressures. The program only calculates horizontal wind forces, and the magnitude of the force at the roof level is based solely on the original story height. Modifying the elevations of the columns has no direct impact on the horizontal wind force.

For structures that are partially shielded or structures that have more than one windward and leeward face, the wind loads will have to be entered as User Defined Story Forces (or Nodal Loads), rather than using the code generated lateral loads as done in the next section.

## *Wind Load*

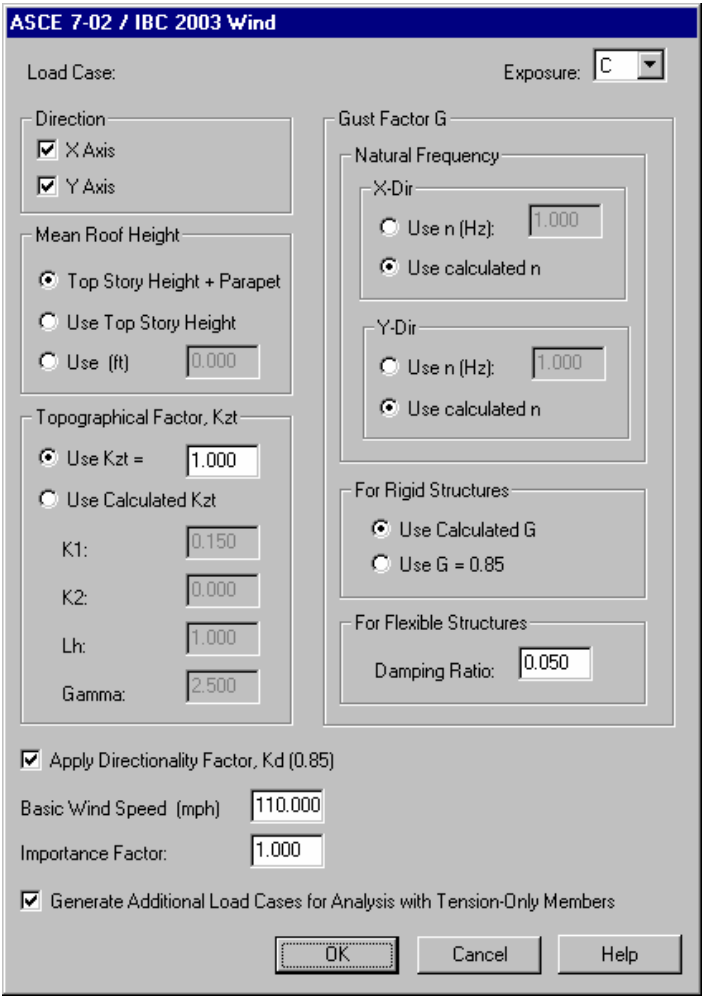
This model already includes the gravity load cases created in the Modeler. These loads cannot be modified or deleted within the RAM Frame program. If other load cases are to be considered, such as Wind Loads, Seismic Loads or Dynamic Loads, they must be created in RAM Frame. To define a Wind Load case:

- Select **Loads - Load Cases**.




- Type `Wind` in the Label edit box.
- Click the Wind option button under Type.

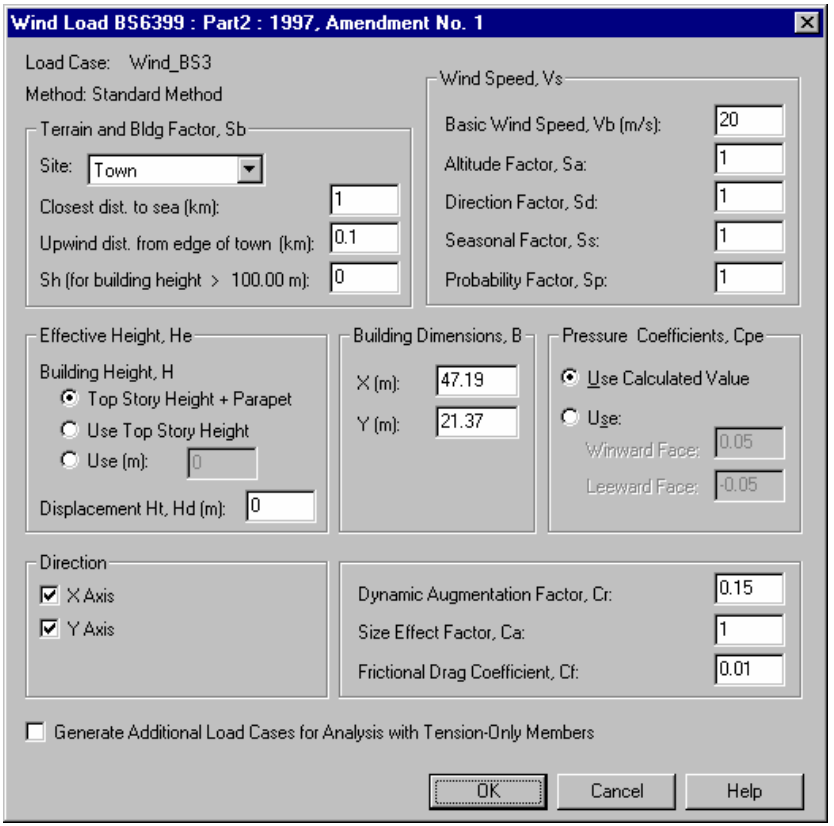
- Click the down arrow to access the wind load building code selection list and select ASCE 7-02 / IBC 2003 (BS6399: Part 2: 1997)
- Click **[Add]**. The Wind dialog box should appear allowing you to define specific characteristics of the wind load:




- Fill out all the fields as indicated in the figure above for US models. Be sure to check the box for Additional tension-only load cases.

Note; in the section marked Natural Frequency, the load is set to use the calculated n. In order to calculate the building frequency for Wind Load (or period for seismic load) the model must have rigid diaphragm levels with masses defined. If diaphragm masses are zero the load case will not run.

- For SI models, select the BS6399: Part 2: 1997 as the building code rather than ASCE 7-02 and set up the load as indicated below.



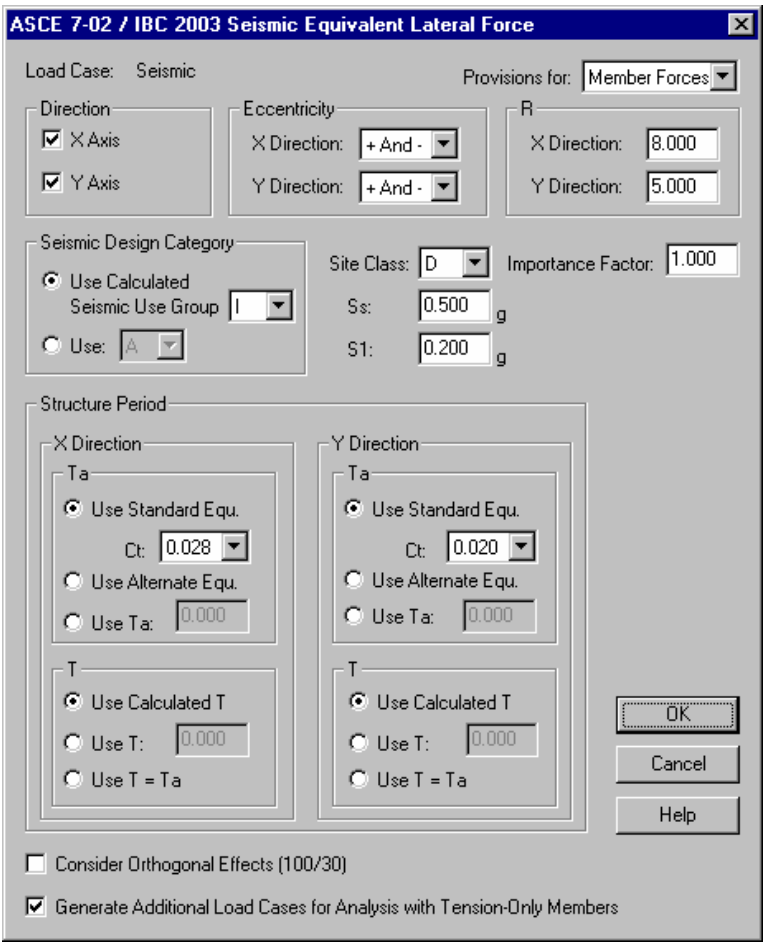

- Click **[OK]**.

New load cases are now added to the Load Cases list box, one for each direction selected in the direction box. Some building codes require consideration of winds eccentric to the building. This results in additional wind load cases. Since the wind load here was created with additional tension-only load cases, both the positive and negative direction load cases appear in the list.

## *Seismic Load*

Seismic loads are input similar to wind loads.

- In the Load Cases window, type `Seismic` in the Label edit box.
- Select the Seismic option under Type.
- Click the down arrow to access various building codes and select ASCE 7-02/IBC03 Equivalent Lateral Force option.
- Click **[Add]**.



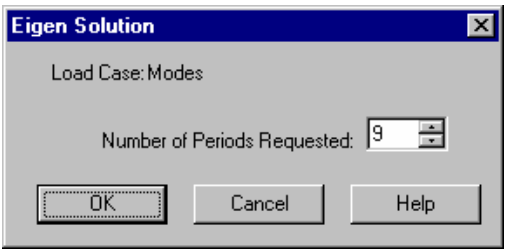
- In the window that appears, fill in the fields as indicated in the previous figure.

Note; Ss and S1 are percentage of gravity values. Also note that the program is set to use the calculated period, but if the calculated period should exceed the upper bound limitation of the code, that maximum period will be used.

- Click **[OK]**.

8 new load cases are added to the Load Cases list box, this covers the different directions and horizontal eccentricities.

- Without closing the Load Cases window, type `Modes` in the Label edit box.
- Select the Dynamic option under Type.
- Leave the drop down list set to Eigen Solution.
- Click **[Add]**. The Eigen Solution dialog box should appear allowing you to define the number of periods you wish to generate.



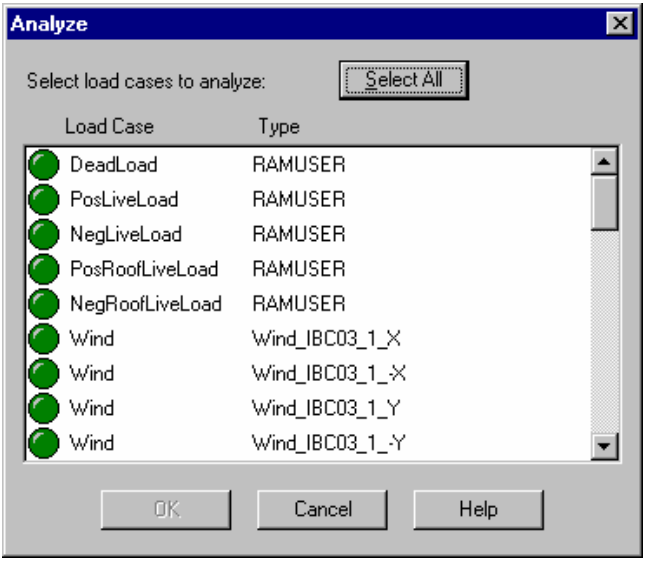

- Click **[OK]** to select the default number of periods.
- Without closing the Load Cases window, type `Center` in the Label edit box.
- Select the Center of Rigidity option under Type.
- Click **[Add]**. Note; the center of rigidity load case has no options and does not affect the analysis at all, it simply allows you to review the center of stiffness graphically in the model.
- Click **[OK]** again to close the Load Cases Dialog box completely.

If you need to edit any of the load cases you can select the load case in the Load List at the bottom of the Load cases window and click Change. When a load with multiple cases is selected, like wind load, the full set of loads will be edited together.

## *Analysis*

You have now defined all the load cases for which the frames will be analyzed. To start the analysis and display the Analyze dialog box:

- Select **Process - Analyze**.




The Analyze dialog box displays all the load cases available for analysis. Those load cases which are preceded by a green dot are available to be analyzed. A red light means that something is preventing that load case from being analyzed (e.g. no diaphragm or diaphragm masses are defined). In this example all load cases should be available.

- Click **[Select All]**.
- Click **[OK]**.

The analysis will commence and a status message box will keep you informed of the progress. Upon completion of the analysis, click **[OK]** and notice that the status indicator light on the status bar turns green (if the self weight reactions of beams and columns are not current then he light will be yellow – see the RAM Manager documentation for a complete explanation on status lights). This indicates that the structure has now been analyzed for each of the load cases selected, and analysis results can be viewed for each load case separately.

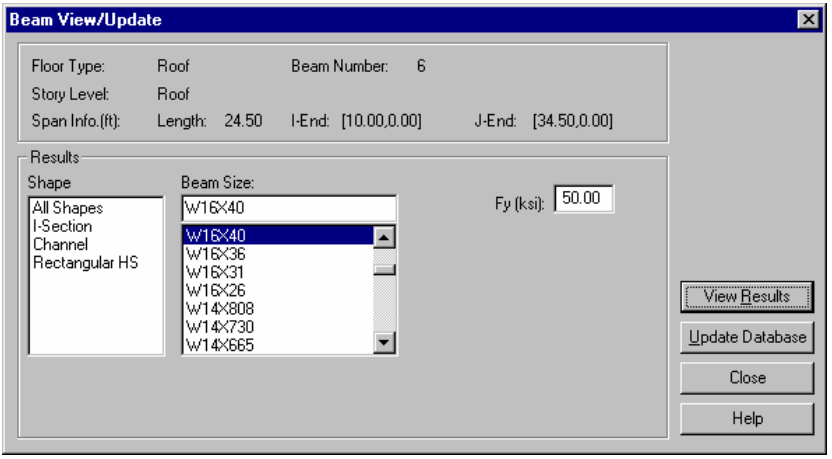
If there is a stability problem with your model it will result in a warning during the analysis. Adjusting the member fixity can usually correct a stability problem. If there is too little stiffness in your structure, and the analysis is being performed with the P-Delta consideration, that might result in an excess P-Delta warning. The tutorial model should not produce any analysis warnings.

The results of the analysis of each load case can now be viewed either on-screen or in printed reports. In either case you can select the load case(s) for which you want results displayed. If you have analyzed many load cases you may find your reports to be quite lengthy. The Reports – Select Cases command acts as an "Output Filter", allowing you to select which load cases will appear in the output.

## *View/Update*

The Process – View/Update command provides information about an individual selected member. While in the Analysis mode for Load Cases, it provides access to the individual member results for each load case analyzed. The View/Update command also allows you to change the member size if desired. To review the results for an individual member:

- Select **Process - View/Update**.
- Click the cursor on one of the roof beams in a moment frame.




- Click **[View Results]**.
- Scroll through the report and get familiar with format of the output.
- Click the circled **X** icon to exit the report and return to the View/Update dialog box.
- Selecting another member size does not immediately affect the member forces, but if the member size is updated, the status light will change to yellow until the analysis is performed again.

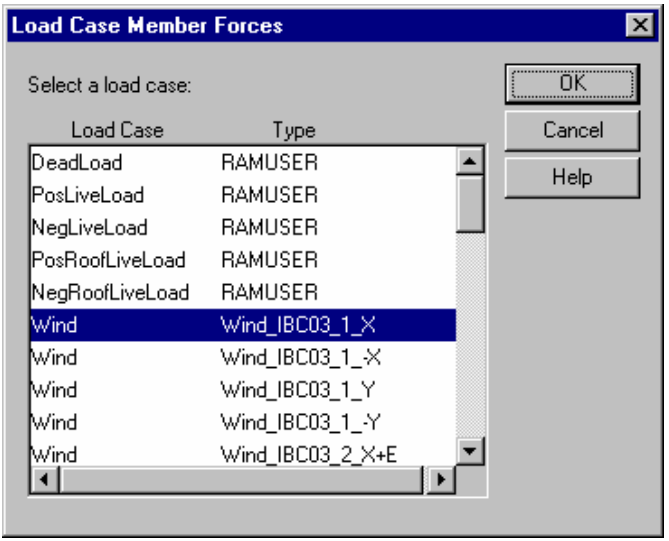
It is not necessary to close the dialog box before selecting another member to View. Just click the target cursor on any other lateral member and the dialog box will be updated.

- Click **[Close]** to close the View/Update dialog box.

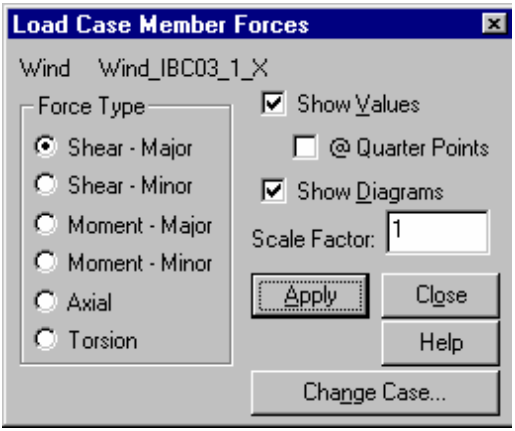
## *Member Forces*

The analysis results may also be displayed on-screen. While in Load Cases mode all results, whether on-screen or in reports, are for the individual, unfactored load cases. By switching to the Load Combinations mode and generating load combinations, you can also review the combined values for reactions, member forces and deflections.

- Select **Process - Results - Member Forces**.




- In the Load Case Member Forces dialog box that appears, select the first Wind Load case.
- Click **[OK]**.
- In the Load Case Member Forces window that appears, select Shear – Major as the Force type.
- Check the box labeled Show Diagrams with a Scale Factor of 1.
- Click **[Apply]**.
- The 3D graphic will now indicate the member strong axis (Major) shear values diagrammatically.



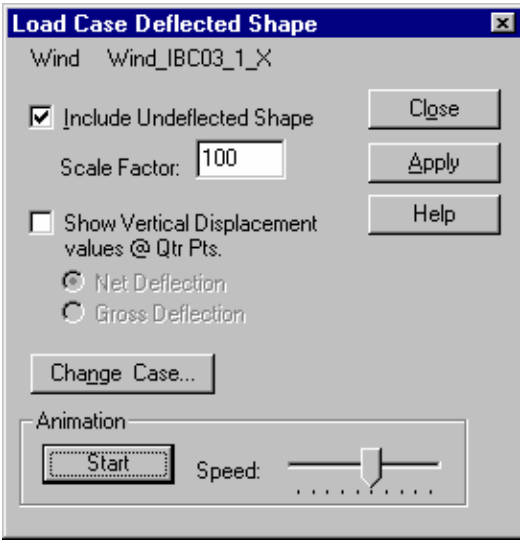

Note; the commands in the View menu, such as Zoom, View – Extents, View – Options – Scale Text and View – Members, can be used to make the output on the screen more readable (see the on-line help for instructions).

- To review those same forces in elevation view, select View – Elevation and select one of the Moment Frame beams.
- To print the screen, select Reports – Print Screen.
- To view the member forces that resulted from a different load case click the **[Change Case]** button in the Load Case Member Forces dialog box.
- Highlight the Dead Load.
- Click **[OK]**.
- Set Force Type to Axial.
- Click **[Apply]**.
- Click **[Close]** to close the Load Case Member Forces dialog box. Your last force display remains on the screen. Use the View – Rest Model command to clear the display.

## *Deflected Shape:*

The deflected shape which results from applying any of the load cases can be viewed on screen. This is a great way to identify any unusual model behavior.

- Select **Process - Results - Deflected Shape**.
- In the Load Case Deflected Shape dialog box that appears highlight the first Wind load case, Wind_IBC03_1_X.
- Click **[OK]**.




- In the Load Case Deflected Shape dialog type 100 for Scale Factor.
- Click **[Apply]**. Note; if the Include Undeflected Shape box is checked, then both the deflected and undeflected shapes will be displayed.
- Click **[Start]**. Note that the scale factor and speed can be altered and the animation can be restarted if necessary.
- Click **[Stop]**.

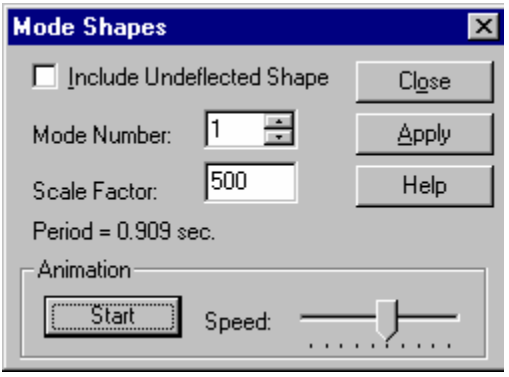
The deflected shape for other load cases can be displayed by clicking the [Change Case] button and making a new selection from the list box.

- Click **View - Reset Model**.

## *Mode Shape*

If an Eigen Solution load case has been analyzed, then the Modal Shapes menu choice can be selected:

- Select **Process - Results - Mode Shapes**.




- In the Mode Shapes dialog box that appears click **[Start]**.

The on-screen graphic will be animated to show the first principal mode of the structure. In this tutorial, that's the principal X mode, the period of which will is used in the seismic load calculations.

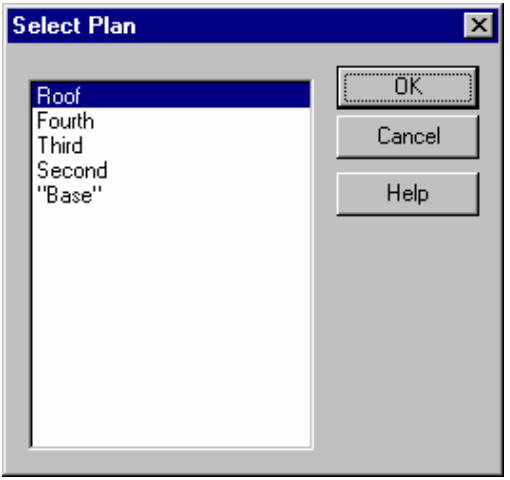
- When finished, click **[Stop]**.
- Change the Mode number to 2 and click **[Apply]** or **[Start]**.
- When finished, click **[Close]**.

As with the Member Forces and Deflected Shapes, **[Close]** removes the dialog box from the screen and the View – Reset Model returns you to the screen display you had prior to issuing this command.

## *Drift*

A drift report for all the load cases can be displayed on screen for any point on the model. The Drift report will only include the load cases selected using the Select Cases button. The drift is only reported for locations within the diaphragm. For points outside the diaphragm or points on levels with no diaphragm, zero drift is always reported. In those cases, the report – Nodal Displacements can be used. To view a Drift Report for a point:

- Select **Process - Results - Drift - At a Point**.




- In the Select Plan dialog box that appears, Highlight Roof from the list box.
- Click **[OK]**.
- This takes you back to the graphics screen. Click the cursor on any point on the floor plan to view drift for that point for all load cases.
- This report starts with a listing of the load cases by name. Click the Forward arrow "►" to continue to the next page where the results begin. The report includes the total displacement as well as the inter-story displacement.
- Click the circled **X** icon to exit the report.

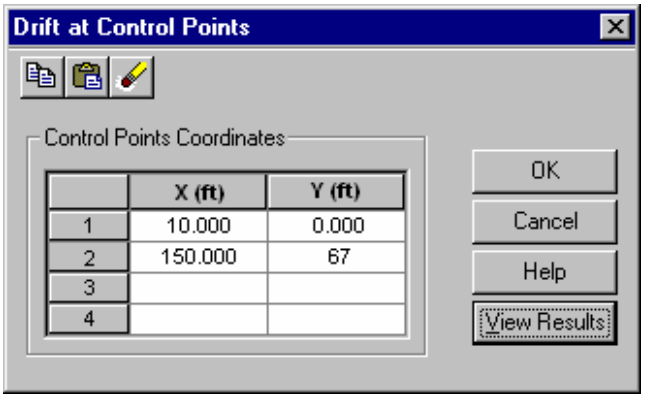
Upon exiting the drift report, you are returned to the floor plan with the target cursor indicating that you are still in Results – Drift – At a Point mode.

You can either continue investigating drift at other points or issue other commands.

A drift report for up to four predefined points, can also be displayed on screen. To Obtain a Drift Report at Control Points:

- Select **Process - Results - Drift - At Control Points**.

The Drift at Control Points dialog box will appear on the screen:

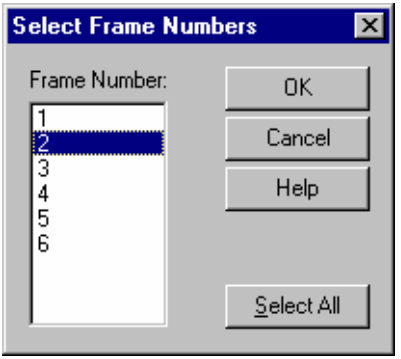



- In the row marked 1 type 10 (3) for X and 0 for Y (you can use the arrow or tab keys to move from cell-to-cell..
- In the second row type 150 (46) for X and 67 (31) for Y.
- Click **[View Results]** to get the drift values at these two corners of the building.
- Click the circled **X** icon to exit the report.

## *Reports*

Many different reports can be generated from the results of the analysis of the structure subject to the various load cases. All reported values are for the unfactored, uncombined load cases while in Load Cases Mode. In the next section you'll see how to get combined results. Printed output is generated using the Reports menu, but the reports can be viewed on screen as well.

- Select **Reports - Screen** (if it's not already checked).
- Select **Reports - Reactions**.




- Click **[Unselect All]**.
- Highlight 2 from the Frame Number list box.
- Click **[OK]**. The Frame Reactions report should come up for the support nodes of Frame #2 only.

Note; the report can be printed from within the report viewer or it can be printed directly to the printer by selecting Reports – Printer, rather than Reports – Screen.

- Click the circled **X** icon to exit the report.
- Feel free to review other reports. Some of the reports can be quite long and make take some time to generate onto the screen.

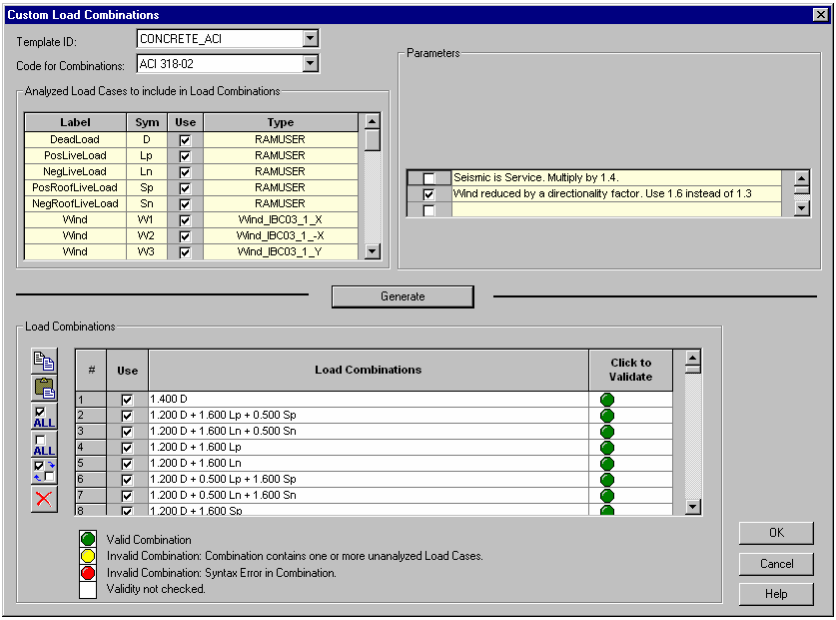
## *Load Combinations*

The Load Combinations Mode allows you to manually define or generate a set of custom load combinations. The member forces and other analytical results from those combinations of load cases can then be displayed on screen or reported. If your model is made of steel frames and if you are going to be using the RAM Frame – Steel Mode for the design of those frames, then there will be a separate set of load combinations defined in that mode which are used only in the steel member design. To initiate the Load Combinations Mode:

- Select **Mode – Analysis – Load Combinations**.

The mode with change and some of the menu options will be affected.

- Select the new menu item **Combinations – Custom Combinations**.




The program can generate load combinations using templates. The templates are text files stored along with the rest of program tables. The templates are grouped in categories like Concrete combos, Soil Combos, LRFD combos, etc. Those templates are further broken down by building code (at least in the US).

- In the Custom Load Combinations dialog box that appears select CONCRETE_ACI (CONCRETE_BRITISH) as the template ID.
- In the Code for Combinations select ACI 318-02 (BS8110 1997).

Different codes modify load combinations in various ways and the program doesn't always know which approach to take. In the case of ACI 318-02 load combinations, the program needs to know if the seismic load was initially defined as a service level load case or an ultimate load case. In our case, the ASCE 7-02 seismic load case we defined was an ultimate level load case.

- Uncheck the first box labeled "Seismic is Service. Multiply by 1.4
- Leave the second box Checked, this model did utilize the Kd factor in our wind load case (for British code users check the box labeled Use 0.9 Instead of 1.0 for Dead Load Factor).
- Click **[Generate]** and the program will generate the full set of code combos. This could result in several hundred load combinations. The new load combinations are automatically checked to be Used. You can uncheck any of the boxes in the "Use" column to inactivate a particular load combination without deleting it entirely.

Note; if you generate more load combinations in the Custom Load Combinations dialog box, those additional combos are appended to the end of the list.

Now the analysis results of the selected load combinations can be viewed using the same commands as described in previous sections.

- Select **Report – Member Force Envelope – Single**.
- Select any lateral member.

The program will generate a report of the minimum and maximum forces in that member for any load combination. The envelope values reports are algebraic. In other words, the minimum moment might be a bigger negative number than the maximum, positive moment. Note that the maximum value may occur at the end or at any point along the length of the member. The location of the maximum force, along with the load combination which produces it are both reported.

- Select **Mode – Analysis – Load Cases** to return to that mode.

You have completed the analysis of the structure for each of the individual load cases. You will now proceed to design your frame members using combinations of those loads.

# RAM Frame – Shear Wall Analysis Module

Whether your shear walls have opening in them or not, the RAM Frame – Shear Wall Analysis module can be used to get more precise information about the force in the lateral walls.

To start the Shear Wall Analysis Module:

- Select **Process – Results – Wall Analysis Results**.

The program launches a separate application in the 3D view akin to the Steel Column module. Most of the actions in this mode are done from an elevation view of a shear wall.

- Select **View – Elevation** and select the wall elevation on Grid B.

Note; if you start this program from an elevation view in the RAM Frame analysis mode, the same elevation will automatically come up.

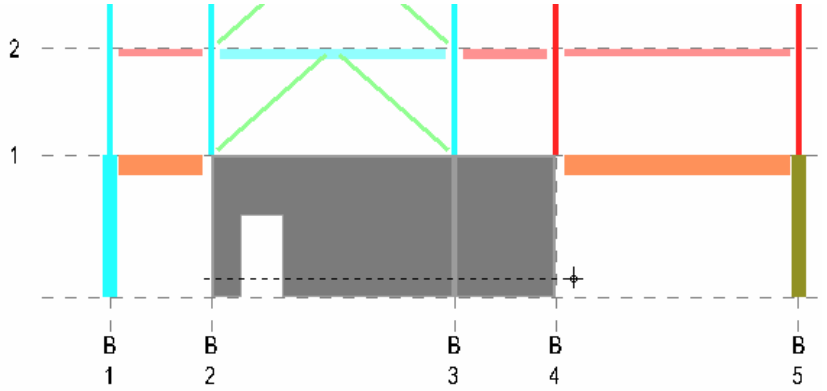
## *Section Cuts*

The Shear Wall Module works by reporting the forces on user-specified section cuts through walls. The section forces can be obtained graphically or through reports. When the program is launched from the RAM Frame Analysis Mode – Load Cases, then the results are for individual load cases. When the program is launched from the Load Combinations mode, then the section results are all given for the combinations.

- Select **Assign - Section Cuts - Add**.

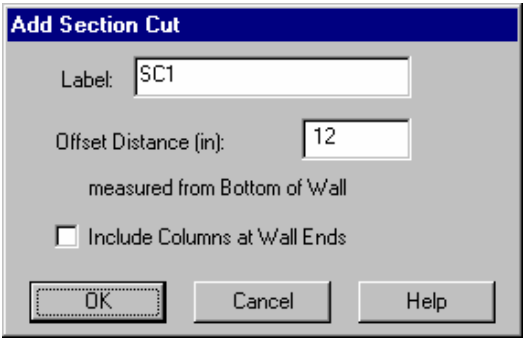
The cursor will change into a cross-hair type cursor at this point.

- Click and hold the mouse just left of the wall and drag the mouse horizontally through the entire length of the wall as pictured below.




First Section Cut

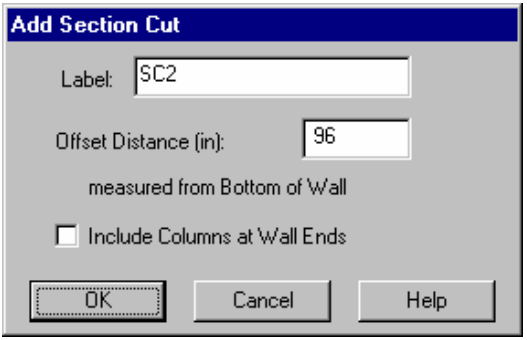
When you let go of the cursor, an options box will pop up.



- Set the offset distance to 12 (300).
- Click **[OK]**.

The Green section cut will be adjusted to be exactly 12” from the lowest left corner of the wall where the section cut started.

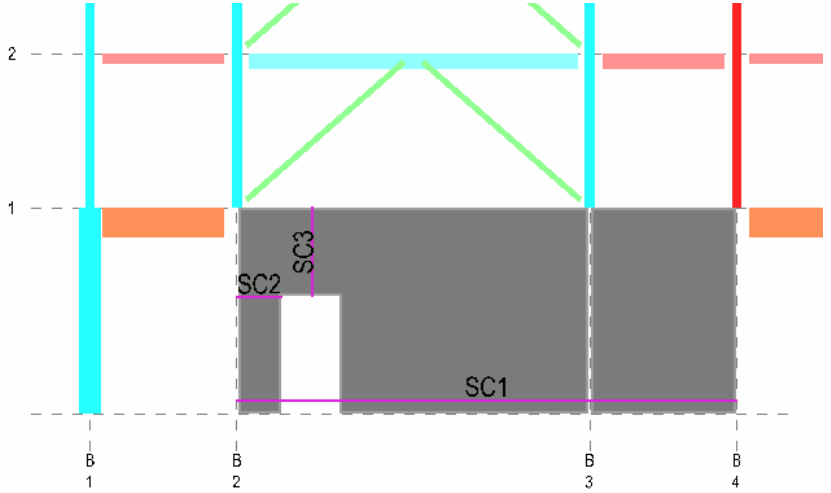
- Create a second section cut by dragging horizontally through the left part of the wall only, just below the top of the opening.




- Set the elevation of the section cut to 96 (2500) and click **[OK]**.
- Now drag a slice vertically through the lintel near the middle of the opening, starting inside the opening and dragging up.
- For this cut the distance is measured relative to the corner of the opening (since the slice started in the opening). Set the offset distance to 24 (600) and click **[OK]**.
- To see the labels on the section cuts, select **View - Section Cut Labels**.

Note; section cuts can be altered after created using the command Assign – Section Cuts – Change. To review a table of the section cuts all at once, select Assign – Section Cuts – List.

Once complete, the three completed cuts should look like the following figure.




Completed Section Cuts

To see the shear forces on the various section cuts:

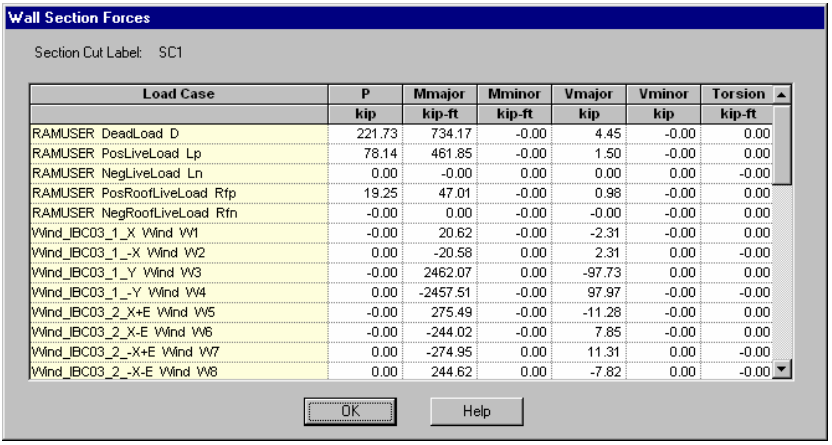
- Select **Process - Results - Display**.


- In the options box that appears, select the first wind load case in the Y direction (Wind_IBC03_1_Y).
- For the Force Type select Shear – Major. At this point, the shear forces will appear on the graphic. If the text is too large or too small to read, it can be adjusted using the buttons to increase or decrease text size in the top menu bar.

- When finished, click **[Close]**.

To see a table of the forces on an individual section:

- Select **Process – Results – Wall Section Forces**.
- With the Target cursor, select one of the section cuts.




| Load Case | P | Mmajor | Mminor | Vmajor | Vminor | Torsion |
|---|---|---|---|---|---|---|
| | kip | kip-ft | kip-ft | kip | kip | kip-ft |
| RAMUSER DeadLoad D | 221.73 | 734.17 | -0.00 | 4.45 | -0.00 | 0.00 |
| RAMUSER PosLiveLoad Lp | 78.14 | 461.85 | -0.00 | 1.50 | -0.00 | 0.00 |
| RAMUSER NegLiveLoad Ln | 0.00 | -0.00 | 0.00 | 0.00 | 0.00 | -0.00 |
| RAMUSER PosRoofLiveLoad Rfp | 19.25 | 47.01 | -0.00 | 0.98 | -0.00 | 0.00 |
| RAMUSER NegRoofLiveLoad Rfn | -0.00 | 0.00 | -0.00 | -0.00 | -0.00 | 0.00 |
| Wind_IBC03_1_X Wind W1 | -0.00 | 20.62 | -0.00 | -2.31 | -0.00 | 0.00 |
| Wind_IBC03_1_-X Wind W2 | 0.00 | -20.58 | 0.00 | 2.31 | 0.00 | -0.00 |
| Wind_IBC03_1_Y Wind W3 | -0.00 | 2462.07 | 0.00 | -97.73 | 0.00 | 0.00 |
| Wind_IBC03_1_-Y Wind W4 | 0.00 | -2457.51 | -0.00 | 97.97 | -0.00 | -0.00 |
| Wind_IBC03_2_X+E Wind W5 | -0.00 | 275.49 | -0.00 | -11.28 | -0.00 | 0.00 |
| Wind_IBC03_2_X-E Wind W6 | -0.00 | -244.02 | -0.00 | 7.85 | -0.00 | 0.00 |
| Wind_IBC03_2_-X+E Wind W7 | 0.00 | -274.95 | 0.00 | 11.31 | 0.00 | -0.00 |
| Wind_IBC03_2_-X-E Wind W8 | 0.00 | 244.62 | 0.00 | -7.82 | 0.00 | -0.00 |

A table of the section forces will appear. The forces are separated into axial forces P (positive = compression), overturning moment (Mmajor), out-of-plane moment (Mminor), shear along the length of the cut (Vmajor), shear perpendicular to the wall (Vminor) and torsion. For a wall with no included lateral columns, the minor axis shear and moment values (as well as torsion) will be zero since the walls are not assumed to have any significant out-of-plane stiffness in the analysis.

- When finished, Click **[OK]**.

## *Reports*

The reports menu has printable versions of the section cut information. For a printable version of the section cut forces:

- Select **Reports – Wall Section Forces – Single**.
- Select a single section cut to generate a printable report.
- Close the report when finished.

Another useful report is the Envelope report. This report includes the maximum and minimum forces on each section cut similar to the force envelope reports in the analysis mode of RAM Frame. It is more useful to generate this report while working with load combinations. To get such a report:

- Select **File – Exit** to exit the Shear Wall Module and return to the primary RAM Frame window.
- Select **Mode – Analysis – Load Combinations** to switch into that mode.
- Select **Process – Results – Wall Analysis Results** to reopen the Shear Wall Module now in Load Combinations mode.
- Select **View – Elevation** and reselect the west shear wall.
- Select **Reports – Envelope – Single**.
- Select one of the section cuts previously defined.
- Close the report when finished.
- Select **File – Exit** to close the module and return to RAM Frame.

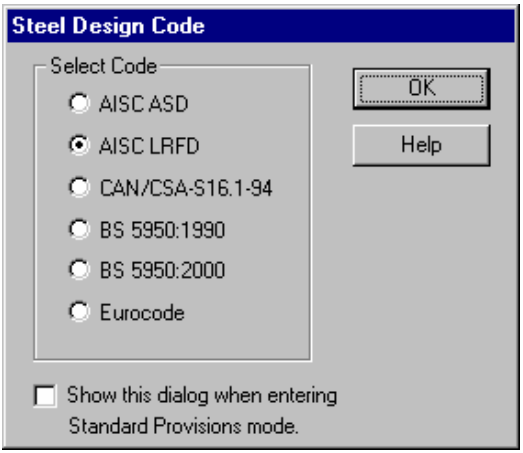
# RAM Frame Steel – Standard Provisions

Within RAM Frame there are two steel post-processing modes: the Standard Provisions mode and the Seismic Provisions mode. In Standard Provisions mode the ability of the structural members to carry the applied gravity and lateral loads (including seismic) is checked according to the primary design code. In Seismic Provisions mode the structure is checked for the seismic detailing and design requirements of the selected seismic code (US codes only).

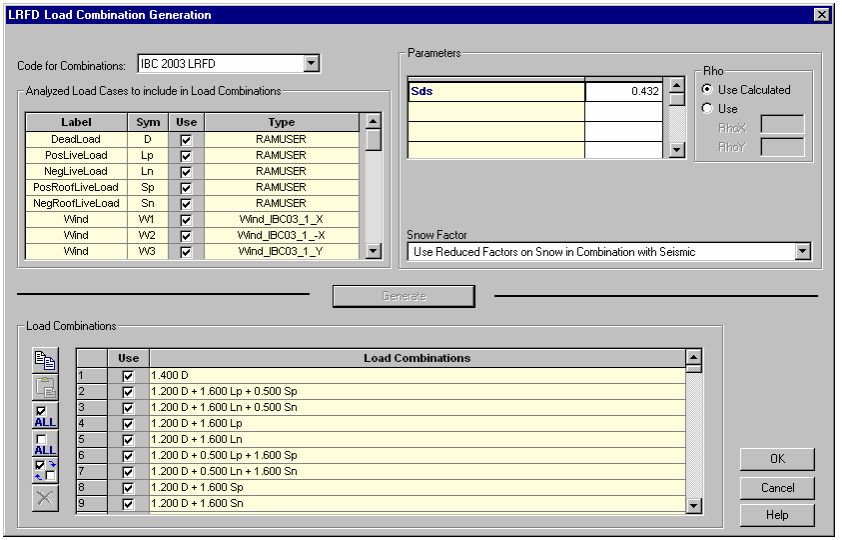
These are optional modules and are not always included with the RAM Frame basic module. In order to perform this section of the tutorial you must have the RAM Frame – Standard Provision Module installed and licensed.

To switch to the Steel – Standard Provision mode:

- Select **Mode - Steel - Standard Provisions**.




- In the Steel Design Code dialog box that appears, check the button for AISC LRFD. If your model was built using SI and British code selections, feel free to select the BS 5950:2000 code.
- Click **[OK]**.
- This dialog can be displayed any time while in Standard Provision mode by selecting the Criteria – Codes menu item.
- The LRFD Load Combination Generation dialog box should appear:




- Select IBC 2003 LRFD (BS 5950) from the Code Combo drop-down list.
- For Sds type 0.432 (this was taken from the loads and Applied Force report in Analysis Mode – Load Cases).
- Select Use Calculated for Rho.
- For Snow Factor, select Use Reduced Factors on Snow in Combinations with Seismic (for British code models Generate the combinations using the default criteria).
- Click **[Generate]**.
- Click **[OK]**.

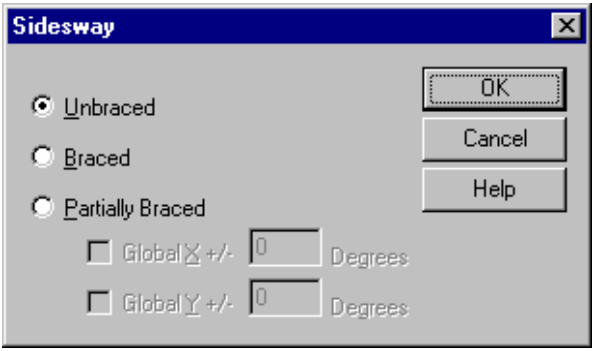
Note; Load combinations will only be generated using the load cases checked in the Analyzed Load Cases to include section. If the program generates any load combinations that you do not wish to be considered, uncheck the Use box for that combo.

Additional Customized Load Combinations can be defined and selected in the same way that they are in Analysis – Load Combinations Mode. The load combinations created within each mode are unique to that mode. Load combinations can be copied from one custom combination dialog to another using the cut and paste tool buttons in the dialog box.

## *Design Criteria*

As in the analysis mode, the Steel Mode has a set of criteria that govern effect the design results of the structure. A complete discussion of these criteria appears in the RAM Frame documentation. It is recommended that you review and understand each of these criteria before accepting RAM Frame design results.

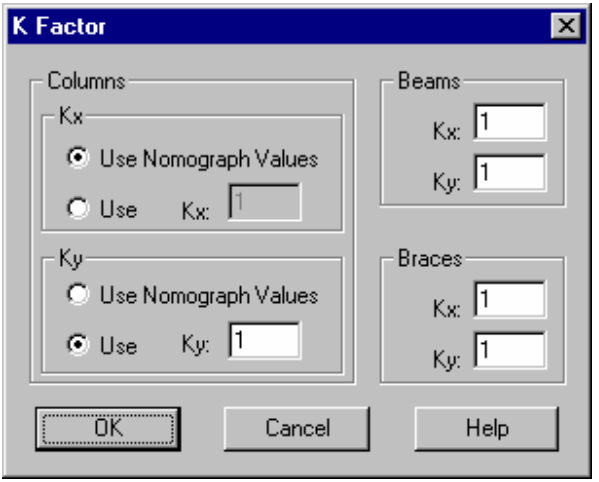
- To assign the Sidesway characteristic of the building:
- Select **Criteria – Sidesway**.




- In the Sidesway dialog box that appears, select Unbraced. Some columns are braced, but those will have to be assigned next.
- Click **[OK]**.

To assign Global Effective Length or K factors,

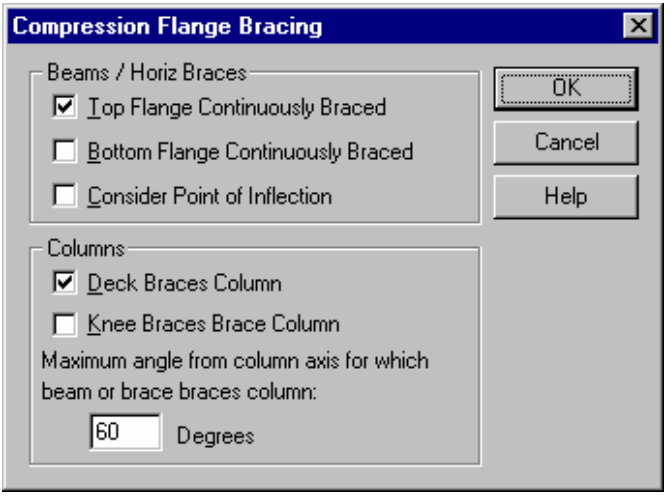
- Select **Criteria – K Factors** ( Effective Length).
- In the K Factor dialog box that appears, set the Column Kx value to Use Nomograph and set all other values to 1. You will assign a K factor of 1 to the strong axis of the braced frames in the next section. (for British code models use the defaults).




- Click **[OK]**.

To set the flange bracing criteria for beams and columns:

- Select **Criteria - Flange Bracing**.



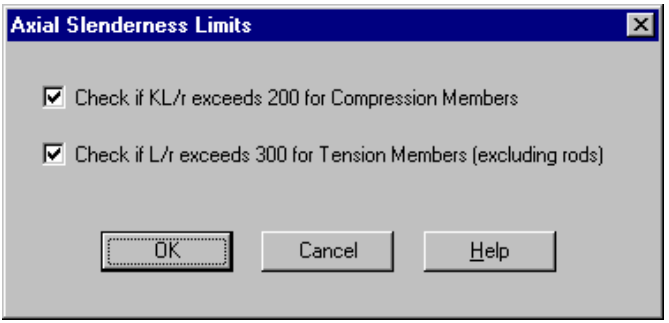

- Check Top Flange Continually Braced for Beams.
- Check Deck Braces Column for Columns.

- Uncheck the other options.
- Type 60 for Maximum angle from column axis for which beam or brace braces column.
- Click **[OK]**.

The Column Moments criteria can be used if you wish for the moment connections in the building to transfer only a portion of the end moment to the supporting column. This can be helpful in designing partially restrained, Type 2 connections (Refer to AISC Steel Specifications for more information), but will not be covered in this Tutorial.

To turn off the Axial Slenderness limits prescribed by the code:

- Select **Criteria – Axial Slenderness Limits**.



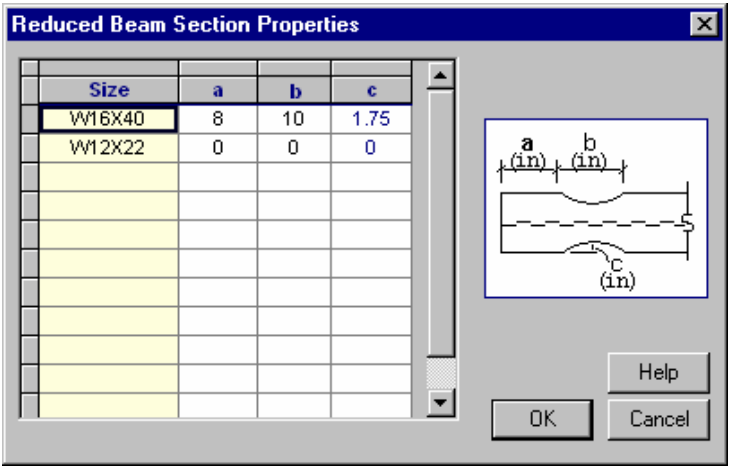

- Check both options.
- Click **[OK]**.

Besides member design criteria, RAM Frame also checks moment connections for stiffener and web plate (doubler) requirements. To assign the criteria to be used in the Joint Design:

- Select **Criteria – Joints**.
- Click **[OK]** to accept all the defaults.

For the Special Moment Frames in this model, we can also assign Reduced Beam Section Criteria (US models only).

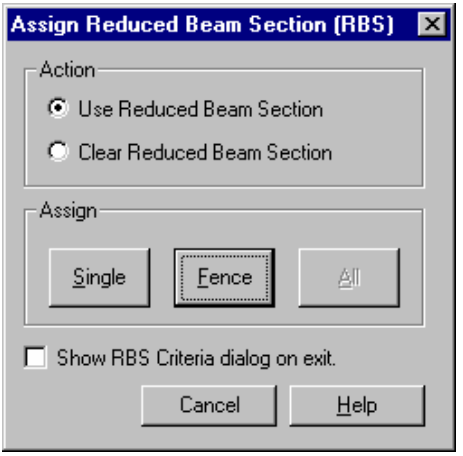
- Select **Criteria – Reduced Beam Sections**.
- For the W16X40, type 8,10 and 1.75 for a, b and c respectively.




- Click **[OK]**.

In order to have the reduced beam sections considered an assignment must be made to the Beams in the Special Moment Frames.

- Select **View - Elevation** and pick one of the braced frames.
- Select **Assign - Beams - Reduced Beam Sections**.

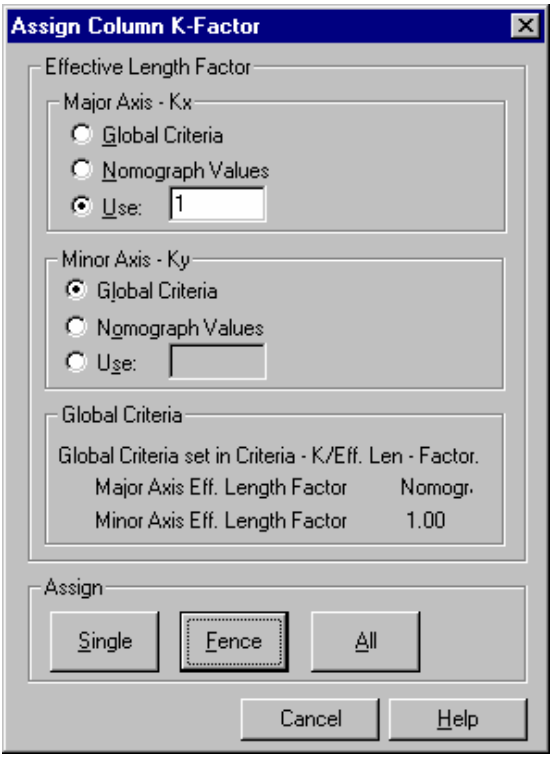



- Set the Action to Use Reduced Beam Section and select the beams with a single or fence option.
- Repeat for the other Moment Frame beams.

## *Assign Effective Length - K Factors*

Since some of the lateral columns in this model are part of braced frames, it is necessary to override the global Nomograph selection of Kx. This global selection was made previously using the Criteria – K Factor menu item. To assign a K factor to an individual member:

- Select **View - Elevation** and pick one of the braced frames.
- Select **Assign - Columns - K Factor** (Effective Length).




- In the Assign Column K Factor dialog box that appears, set Major Axis – Kx (Le / L) to Use and Type 1.0 for the value.
- Click **[Fence]**.
- With the cursor fence all of the steel columns in each of the braced frames.

(Note; it is OK to also fence the concrete columns on those frames, as these commands will only be assigned to steel columns. Concrete column slenderness parameters are set within the Concrete Design Module).

- Repeat for the other two frames.

Note; the assignment can also be made in 3D view provided you can select the column of the braced frames without selecting the columns in the moment frames. The rotation angle of the 3D view can be altered using view – rotate.

The screen should now display the K Factor setting that has been applied. To clear the screen,

- Select **View – Reset Model**.

## *Member Code Check – Standard Provisions*

You will now perform a code check on all lateral members using the load combinations you selected previously. To perform a code check:

- Select **Process - Member Code Check**.

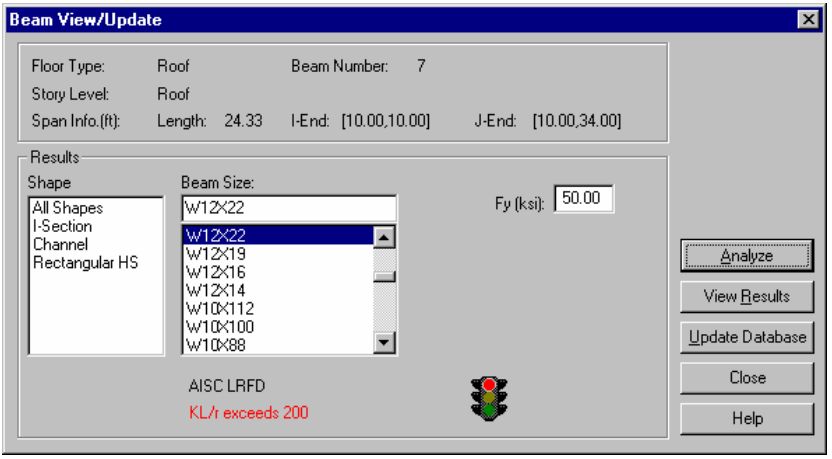
A progress bar will keep you informed of the progress of the code check. Because the program needs to check each unbraced segment of each member for each load combination, the process can take several minutes.

When the code check is complete, the lateral system of the model will appear color coded to indicate to what degree the members are stressed. A color scale correlates the colors to the interaction values of the members. The display of each member is based on the controlling Interaction Equation from all the selected load combinations.

- Click **[Show Values]** to display the interaction equation values on screen. Note; an interaction value of "–1" indicates that the member fails some prescriptive limitation (e.g. KL/r > 200).

The View/Update command can be used to interactively modify individual member sizes to improve the design.

- Select **Process - Member View/Update**.
- With the cursor, select one of the red beams in the braced frames at the roof level.




The Beam View/Update dialog box should appear. In the lower left hand corner is an indication of the maximum interaction equation or design

warning. In this case, the bottom flange of the beam is completely unbraced. When there is compression in the member it will thus fail the KL/r limit of 200 (US codes) or it will fail the L/r limit of 300 when the member is in tension. The stoplight in the box will be red to indicate when the member has a design warning.

- Click **[Close]**.
- Select one of the Moment Frame beams.
- This beam should pass and the interaction value will be listed at the bottom.
- To see the details of the design Click **[View Results]**.

The Member Code Check Report should come up. Review the report. In particular, take note of the unbraced length for bending – Y-axis value.. A beam is typically broken into several unbraced segments, each with a different unbraced length. The single segment that yields the highest interaction ratio is the one segment that gets reported on the design report. Note; when a member fails a prescriptive limit, the reports results are for the first combo that causes the problem. The reported forces are not necessarily the critical forces.

For more information on the design performed in this check, refer to the technical portion of the RAM Frame Manual.

- Click the circled **X** to exit the report and return to the View/Update dialog box.
- Select a different section from the Beam Size list box.
- Click **[Analyze]**. The interaction equation and signal light will be updated to reflect the results of the analysis.
- Click **[Close]** to exit without saving the modified design. (Click [Update Database] if you want to make the change permanent.)

If you modify any member size using the view/update command, the color of the member will change to indicate the new level of stress. As with View/Update in Analysis Mode, a second member can be selected without closing the View/Update dialog box.

Note; the program uses the member forces from the original analysis, but uses the section properties of the currently selected size when a redesign is performed. Once sizes have been changed the analysis of the model is no

longer technically valid. Notice that the status indicator light on the status bar in the lower right corner has turned yellow to indicate that you should view all results with caution as they no longer represent the analysis of the current member sizes in the database. Whenever member sizes are changed, the analysis should be re-run to insure accuracy. This applies to all of the View/Update commands in RAM Frame.

To rerun the analysis and verify the new sizes:

- Select **Mode – Analysis – Load Cases**.
- Select **Process – Analyze**.
- When the Analyze dialog box appears, make sure that all of load cases are selected and Click **[OK]**.
- Select **Mode – Steel – Standard Provisions**.
- If you want to skip the slenderness limit checks this time, select **Criteria – Axial Slenderness Limits** and turn off both parameters.
- Select **Process – Member Code Check**.

The new code check results should be indicated on screen. By working through a few cycles of this process, you can quickly converge on an efficient working design.

## *Joint Code Check – Standard Provisions*

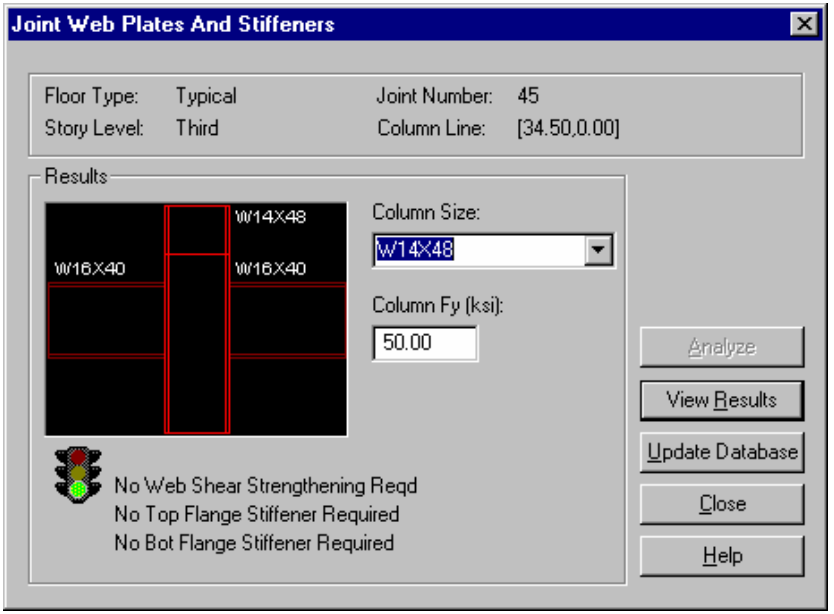
You will now perform a code check on the rigid beam to column connections. To perform a joint code check:

- Select **Process - Joint Code Check**.

A progress bar will keep you informed of the progress of the code check. When the code check is complete, the nodes of the lateral system will appear as colorful dots indicating the status of the joint. The status represents if the joint is valid (refer to Technical Section in the RAM Frame manual), and what strengthening is required of the joint.

The View/Update command can be used to interactively modify individual column sizes to improve the connection design.

- Select **Process - Joint View/Update**.
- With the cursor, select one of the green colored nodes.




The Joint Web Plates and Stiffeners dialog box should appear. A graphic shows the various member sizes that meet at the selected joint and the signal light indicates the status of the column. The size of stiffeners and web plate are listed when necessary.

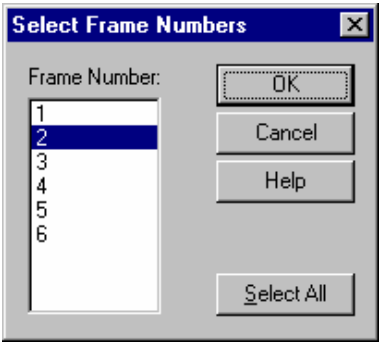
Note; the graphic displayed in the View/Update dialog represents a view from the minor axis direction of the column. The graphic shows the steel beams that frame into the flanges of the steel column. No members supported on the web of the column are shown.

As with Member – View/Update, this dialog box allows you to select other column sizes and investigate the stiffener plate requirements by clicking [Analyze]. You can view the Joint Code Check Report by selecting [View Results] and you can update the column size if desired by Clicking [Update Database].

## *Reports – Standard Provisions*

Many different reports can be generated from the design results of the structure subject to the various load combinations. The printed output is generated using the Reports menu. The print options available in Steel Mode are different that those available in Analysis Mode. The forces and reactions printed while in Steel Mode are based on load combinations in that mode. To print the minimum and maximum reactions for any load combination:

- Select **Reports - Screen**.
- Select **Reports - Reaction Envelope**.




- In the Select Frame Numbers dialog box that appears, choose which frames you would like to review and click **[OK]**.

The Reactions Envelope report should come up. Hit the Page Down Key to see the rest of the report. Notice that the reaction values are given for the worst case among all load combinations in that mode.

- Click the circled **X** to close the report.

To print a summary of the results for all the member code checks:

- Select **Reports - Member Check Summary**.
- Select a frame or frames and click **[OK]**.
- The Member Check Summary report should come up.
- Click the circled **X** to close the report.

Take time now to review the various reports available in the Reports menu.

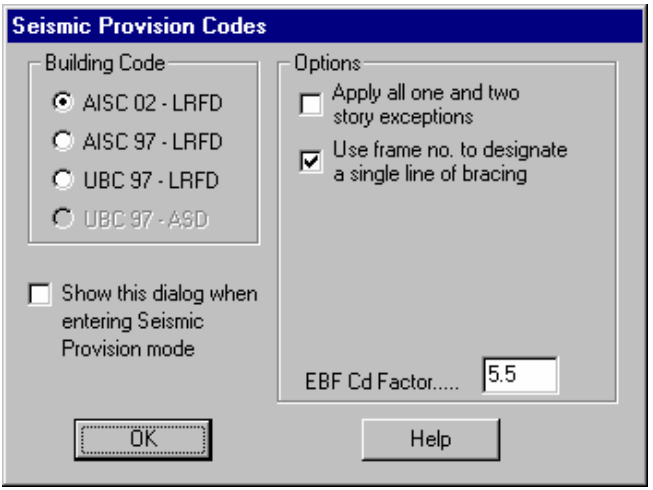
# RAM Frame Steel – Seismic Provisions

In Seismic Provision mode the structure is checked in accordance with the detailing and design requirements of a selected seismic code. The checks performed in this mode rely heavily on the load combinations and design criteria established in the Standard Provision Mode. Note that only the AISC 02 - LRFD post processor will be used in this tutorial although the AISC - 97 and UBC - 97 codes work in a similar manner.

The Seismic Provision Mode is only accessible if one of the AISC Steel Design codes (ASD or LRFD) is used in Standard Provision Mode. There must also be at least one seismic load case defined. If your model is based on the British code or if you do not have a license of the Special Seismic provisions module, skip this entire section.

To switch to the Steel – Seismic Provision mode:

- Select **Mode - Steel - Seismic Provisions**.

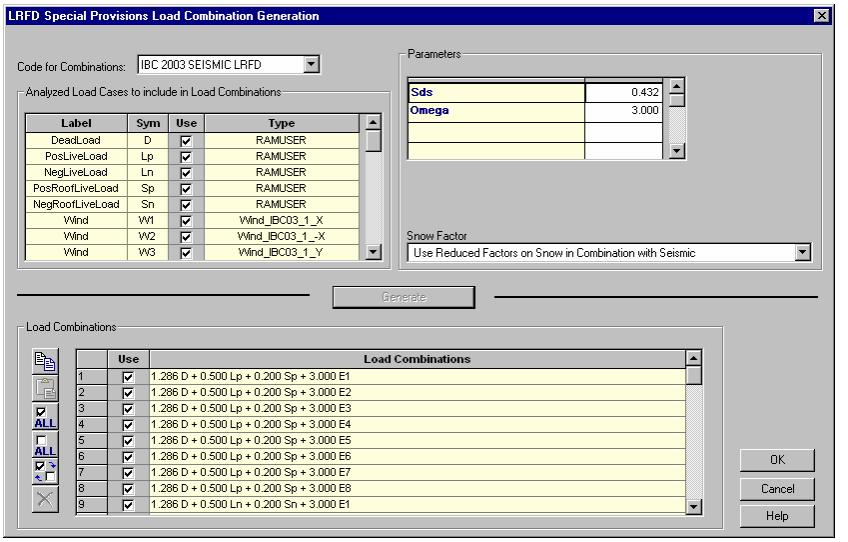



- In the Special Seismic Provision Codes dialog box that appears, select the button for AISC 02 - LRFD.
- Under Options, Check Use Frame No. To Designate a Single Line of Bracing
- Type 5.5 for the EBF Cd Factor.

Note that you are not able to change to the UBC 97 – ASD code because this requires that the AISC - ASD Steel Design Code be selected in Standard Provision mode prior to entering the seismic mode.

This dialog can be displayed any time while in Seismic Provision mode by selecting the Criteria - Codes menu item.

- Click **[OK]**.




In the Seismic Provisions Load Combinations dialog box that appears

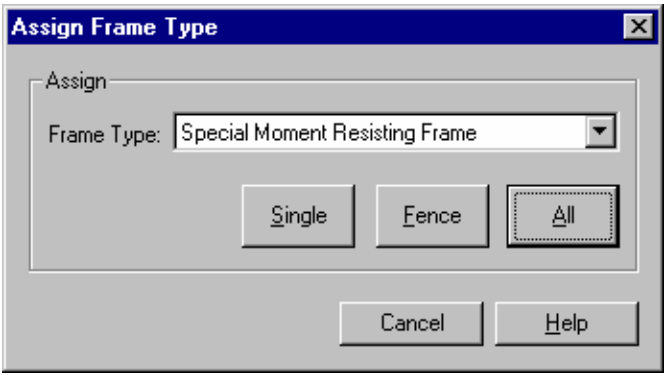
- Set the Sds value to 0.432
- Set Omega to 3.
- Click **[Generate]**.
- Click **[OK]**.

Additional Customized Load Combinations can be defined and selected in the same way that they are in Standard Provision mode.

## *Assign Frame Type*

In order to check the members and joints for these provisions you must first define what type of lateral frames exist in the model:

- Select **Assign – Frame Type**.




- In the Assign Frame Type dialog box that appears select Special Moment Resisting Frame as the Frame Type.
- Click **[All]**.
- Select **View – Elevation** and pick the braced frame on Grid B, if not already selected.
- Select **Assign – Frame Type** again.
- Change the Frame Type to Concentric Brace Frame – Chevron.
- Click Fence and fence all of the steel members in the braced frame on Grid B.
- Repeat for the braced frames on grid F.6.
- Do not assign any frame type to the tension-only X brace at this time.

## *Member Code Check – Seismic Provisions*

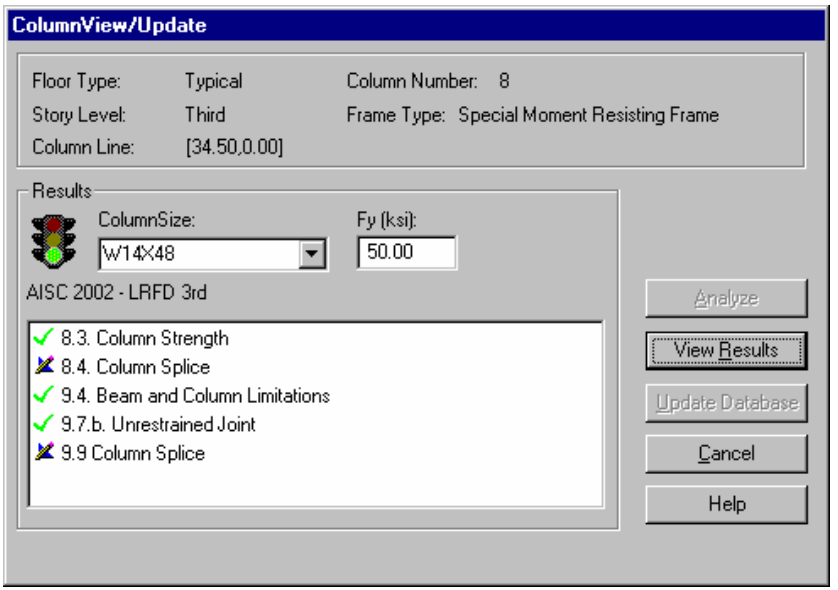
To check the member design against the special seismic provisions, you will follow the same steps as for standard provisions: To start the code check:

- Select **Process - Member Code Check**.

A progress bar will keep you informed of the progress of the code check. When the code check is complete, the lateral system of the model will appear color coded to indicate the design status of the members.

The View/Update command can be used to interactively modify individual member sizes to improve the design.

- Select **Process - Member View/Update**.
- With the cursor, select one of the lowest steel columns in the south moment frame.




The Column View/Update dialog box should appear. In the lower box is a list of the various design provisions and an indication of which checks the member has passed, or failed. The signal light in the box will be lit red to indicate the member has failed one of these provisions.

Note; double click on any icon in the results list box to view a short explanation of that icon. Refer to the technical manual for a description of all the View/Update icons.

- To see the details of the design Click **[View Results]**.

The Seismic Provisions Member Code Check Report should come up. Review the report. For more information on the designs performed in this check refer to the technical portion of the RAM Frame – Special Seismic Provisions documentation.

- Click the circled **X** to exit the report and return to the View/Update dialog box.
- Select the W14x43 section from the Column Size list box.
- Click **[Analyze]**. The design checks and signal light will be updated to reflect the results of the analysis.
- Click **[Cancel]** to revert to the original size.

Note; some of the beams and braces in the chevron braced frames will fail the special seismic provisions code check. Feel free to increase the size of those members now. An HSS6x6x3/8 should work for the braces and a W12x30 should work for the beams. Besides using the View/Update command, member sizes can also be assigned directly using the assign menu options.

## *Joint Code Check – Seismic Provisions*

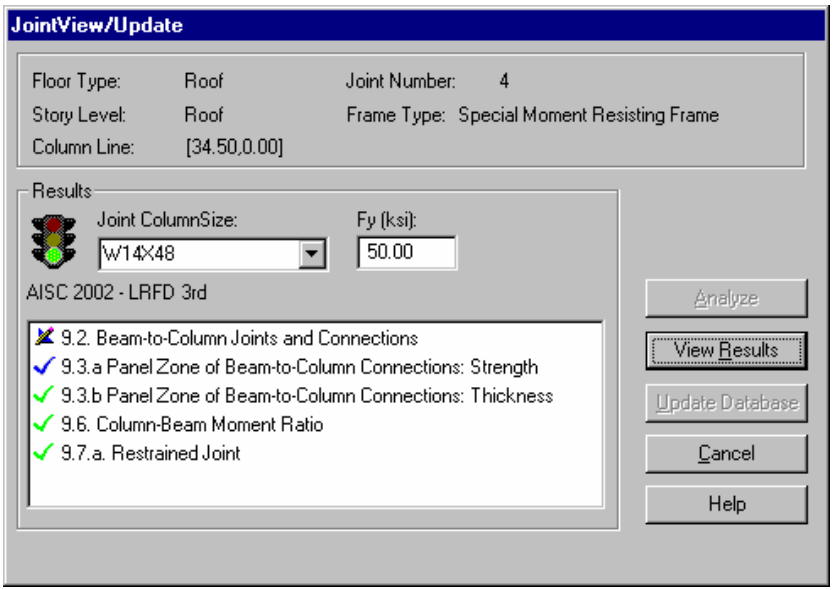
You will now perform a code check on of the rigid beam to column connections according to the Special Seismic Provisions of the selected code. To perform a joint code check:

- Select **Process - Joint Code Check**.

When the code check is complete, the steel beam - column joints of the lateral system will appear as colorful dots indicating the status of the joint.

The View/Update command can be used to interactively modify individual column sizes to improve the connection design.

- Select **Process - Joint View/Update**.
- With the cursor, select one of the top floor (blue) nodes in the moment frame (Frame on Grid F).




- To see the details of the design Click **[View Results]**.
- Feel free to investigate other sizes and click **[Cancel]** when finished. If you make the column slightly larger you can possibly avoid the need for stiffener plates.

## *Reports – Seismic Provisions*

Many different reports can be generated from the seismic code check of the structure. The printed output is mostly generated using the Reports menu. The print options available in Seismic Provisions Mode are different than those available in Analysis Mode. To print the detailed seismic code check results for one or more member:

- Select **Reports – Member Code Check – Single**.
- Pick any lateral member.
- Close the report.

To print a summary of the results for all the member code checks:

- Select **Reports – Member Check Summary**.

A similar output is available for the joint code checks.

- Select **Reports – Joint Check Summary**.

Take time now to review the various reports available in the Reports menu.

# RAM Frame – Drift Control

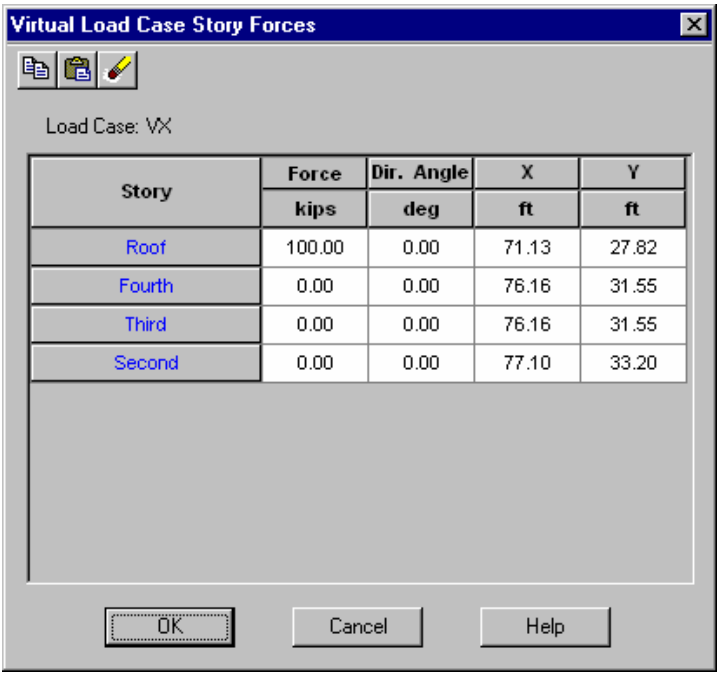
In addition to the member design provisions of each design code, building structures are also required to meet certain drift limitations. The Drift Control Module provides the user with a means to see how each of the lateral members contributes to the resistance of that drift. For the Tutorial, drift at the Roof Level in the X and Y directions is the primary concern. In this section you will define a virtual load case in both directions, pair those load cases with the governing seismic load cases and review the results in order to determine which members provide the greatest resistance to that drift. You will also determine how to improve the overall structural performance.

This is an optional module and is not included with the RAM Frame basic module. In order to perform this section of the Tutorial you must have the RAM Frame – Drift control Module installed and the hardware lock programmed for that module. You can skip this section otherwise.

## *Defining Virtual Load Cases*

To define the virtual load cases for analyzing roof drift:

- Select **Mode – Analysis – Load Cases**.
- Select **Loads – Load Cases**. The Load Cases dialog box should appear:
- Type VX in the Label edit box.
- Click the Virtual Work option button.
- Click **[Add]** and the Virtual Load Case Story Forces dialog box should appear:




- For the Roof Level Type 100 in the Force column (100 can also be used for SI models).
- Leave the Dir. Angle set to 0.
- Leave the X and Y coordinates at the default. This represents the calculated center of mass for the respective level.

- Leave the forces for the other floors set to 0.
- Click **[OK]**. This returns you to the Load Cases dialog box.
- Type VY in the Label edit box.
- Click the Virtual Work option button.
- Click **[Add]**.
- For the Roof Level Type 100 (100) in the Force column.
- Type 90 for Dir. Angle.
- Click **[OK]**.
- Click **[OK]** to dismiss the Load Cases dialog box.

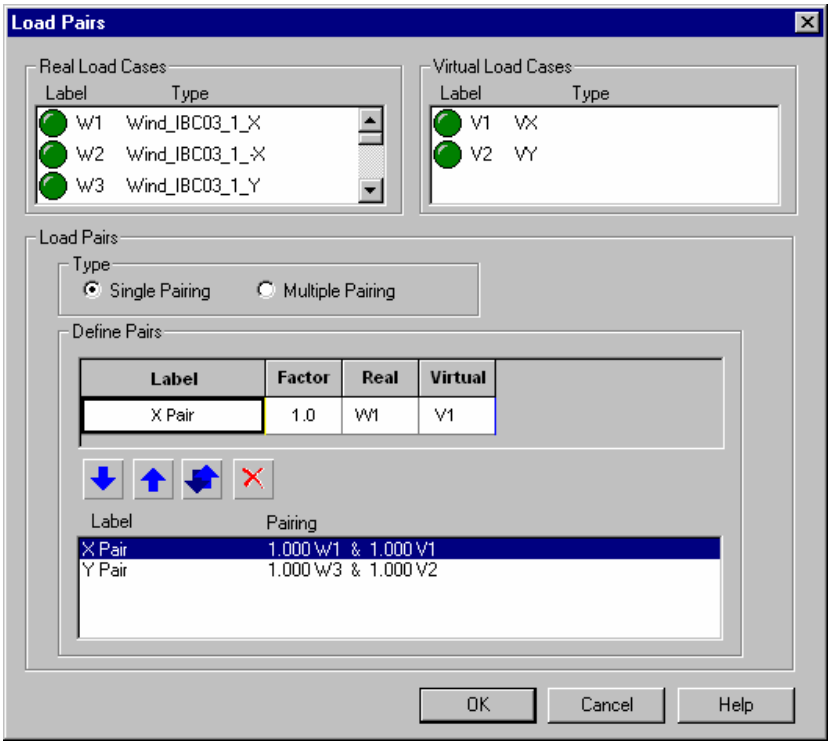
Next, you need to analyze the new virtual load cases:

- Select **Process – Analyze**.
- When the Analyze dialog box appears, make sure that all of load cases are selected by clicking **[Select All]**.
- Click **[OK]**.

## *Defining Load Pairs*

In Order to pair the virtual loads with real load cases and perform the Drift Control Analysis, you must now enter the Drift Control Mode of RAM Frame:

- Select **Mode - Drift Control**.
- Select **Loads - Load Pairs**. The Load Pairs dialog box should appear:




In the center portion of the box under Define Pairs:

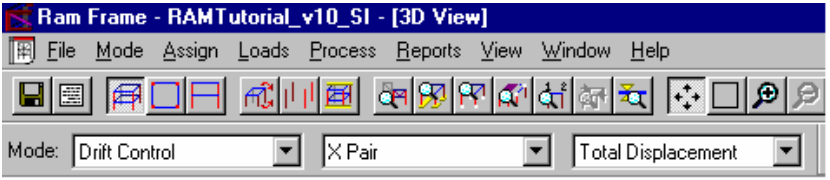
- Type `X Pair` for Label.
- Type 1.0 for Factor.
- Type or select W1 for Real column.
- Type or select V1 for the Virtual column.

- Click the Blue Down Arrow to establish the pair.
- Type `Y Pair` for Label.
- Type 1.0 for Factor.
- Type W3 for Real.
- Type V2 for Virtual.
- Click the Blue Down Arrow to establish the pair.
- Click **[OK]**.

This takes you back to the graphics screen. To analyze the load pairs:

- Select **Process – Analyze**.

The screen should now display a color coded image of the structure. Members shown in warm colors (i.e. red, orange, yellow…) are participating more in the resistance of the roof drift based on the current load pair (X Pair) and the current evaluation method (Total Displacement). In this case, you can see the beams of the north moment frame do the most work, indicated by their color.




To review the results for the other load pair:

- Click the center drop-down list in the toolbar that indicates the current load pair. Select Y Pair from this list. The screen should be updated to reflect the results of Y direction loads.

Another way of reporting participation is to divide a member's participation by its own volume. In this way you can see where increasing a member provides the most benefit. To review the results as Total Displacement/Volume:

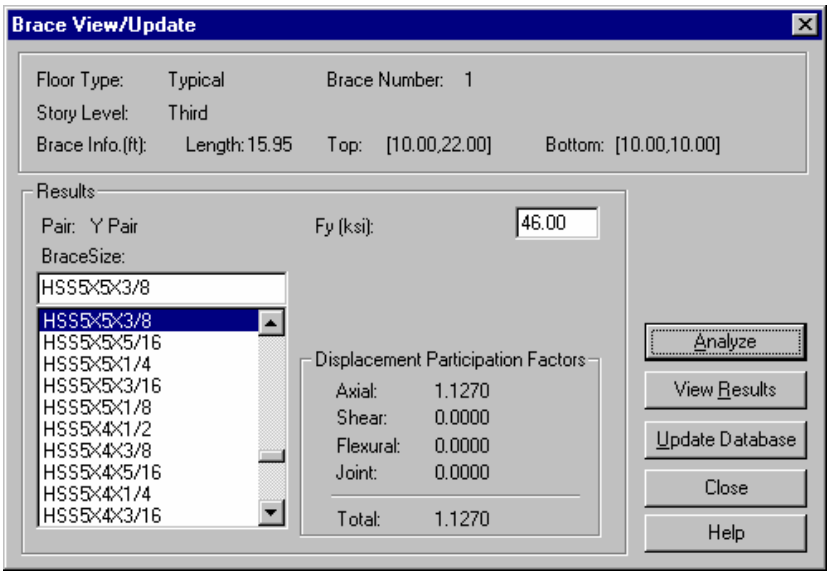
- Select **Process – Results – Total Displacement/Volume** or use the third drop-down list.

A dramatic change should occur in the screen output. The walls that were red or yellow are now blue. This happens because while that member does a lot to resist drift in the Y direction (even at the roof), it is also has a much larger volume than the steel members. Now the lower level braces the frames should be the highest participating members by volume.

There are several other features in the Drift Control Module as described in the Drift Control portion of the RAM Frame manual. Examine some of those options now.

To review an individual member to evaluate its participation:

- Select **Process - View/Update**.
- With the cursor select the lowest level beam in the aforementioned braced frame.




The Beam View/Update dialog box should appear. In the center of the box you can see the participation of that beam due to the current load pair.

As in the other modes, you can investigate other member sizes by selecting them from the list and clicking analyze.

- Select another brace size from the list.
- Click **[Analyze]**.
- The participation values will change slightly.
- Click **[Update Database]** if you want to make the change official.
- Click **[Close]** to dismiss the View/Update dialog box.

The fact that one or more members are drawn red is only an indication that those members are working the hardest on a relative scale. It does not mean that they are failing in any way. An optimized structure in terms of drift is one where the majority of the members are all performing equally. If a model has a few red members and the rest are blue, that is an indication that the red members are overworked while the rest of your framing isn't helping that much with respect to drift control.

As with any Update Database command, the analysis results considered when recalculating the participation factors are from the previous analysis run, and those results are invalidated by any modification to the stiffness matrix. Another analysis run should be performed before the Drift Control results are accepted.

To rerun the analysis and review the new sizes:

- Select **Mode – Analysis – Load Cases**.
- Select **Process – Analyze**.
- When the Analyze dialog box appears, make sure that all of load cases are selected and Click **[OK]**.
- Select **Mode – Drift Control**.
- Select **Process – Analyze**.

The participation factors should be shown on screen. The colors may be a little different than they appeared before.

## *Reports*

Reports can be generated from the Drift Control Mode just like the other modes. The printed output is generated using the Reports menu. To print the Displacement Participation / Volume Summary report:

- Select **Reports - Displacement/Volume Summary**.

The Displacement Participation / Volume Summary report should appear:

- Hit the Page Down Key to see the rest of the report.
- Click the circled **X** to close the report.

Take time now to review the various reports available in the Reports menu.

- Select **File - Exit** to exit RAM Frame and return to RAM Manager.

This completes the RAM Frame portion of the tutorial. Proceed to the next section to perform the concrete member design. If you do not own that module you can skip ahead to the RAM Foundation section of the Tutorial.

# RAM Concrete Analysis

RAM Concrete is completely integrated into the RAM Structural System. It uses the same model and database as RAM Frame and RAM Steel. While most of the information needed for concrete gravity analysis and design is taken from the database, some data does need to be entered in the Concrete program itself.

You must have a license to RAM Concrete to perform this part of the tutorial. If no license is available, skip to the next section, RAM Foundation. At this point, only the US codes are implemented in the RAM Concrete module. SI units will not be presented in this section, but an SI units model can still be used.

To invoke RAM Concrete from the RAM Manager:

- Select **Design – RAM Concrete** or select the 5th square button depicting a concrete beam and column.

In order for the program to design the concrete members correctly, a separate gravity analysis needs to be performed. The lateral force results from RAM Frame are added to these gravity results automatically. For this reason you should always perform a RAM Frame analysis prior to running RAM Concrete.

## Concrete Program Organization

The RAM Concrete program is comprised of three modes: the Concrete Gravity Analysis Module, the Concrete Beam Design module and the Concrete Column Design module. When you enter RAM Concrete you will always be placed in the Concrete Gravity Analysis Module. These modules function as follows:

## Concrete Gravity Analysis Module

When you select Design – RAM Concrete you will be placed in the Concrete Gravity Analysis Mode. The mode drop-down on the upper left toolbar will indicate you are in this mode.

In this mode a finite element analysis of each story is performed. This analysis determines the gravity forces for both concrete column and beam design. In this module the user has some control over the finite element analysis which will be discussed. The user also assigns beam lines which affects how the beams are detailed.

It's important to note that the concrete analysis requires all members, even steel members to have an assigned size. If a model includes optimized members, then the RAM Steel Beam and Column design modules must be run prior to analyzing the model in RAM Concrete.

## Concrete Beam Design Module

In this module the design of all concrete beams is performed. The gravity analysis forces are automatically combined with the lateral forces (from RAM Frame) to obtain design beam forces. The program automatically designs reinforcement considering the user preferences and code requirements. An interactive view/update command provides a means of modifying the programs design.

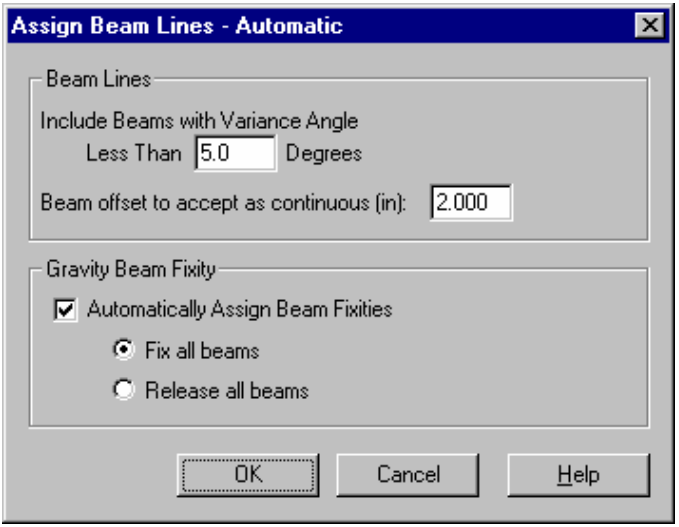
## Concrete Column Design Module

In this module the design of all concrete columns is performed. The gravity analysis forces are automatically combined with the lateral forces (from RAM Frame) to obtain design column forces. Column forces from the external program RAM Conce*pt* can also be utilized. A comprehensive moment interaction surface is calculated for the design of each column. Multiple reinforcement patterns can be considered for each column line, and the program automatically selects the optimal pattern and reinforcement size. An interactive view/update dialog provides a means of altering the programs designs.

## Assign Beam Line Numbers

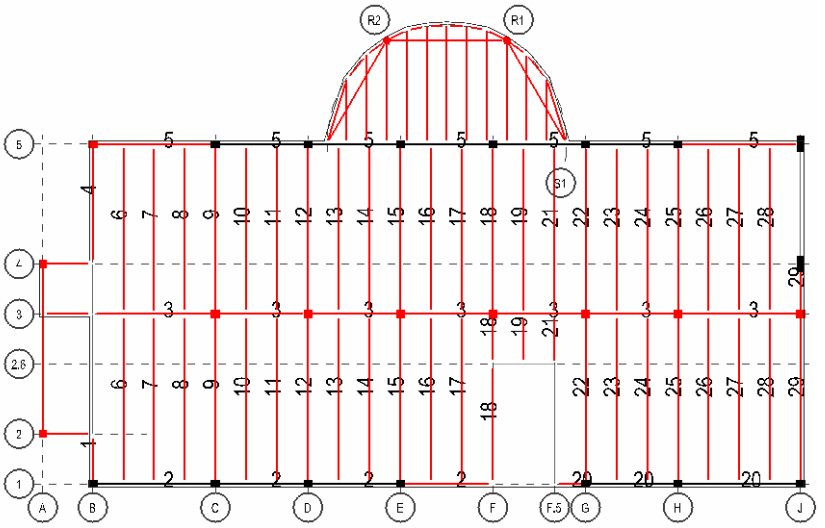
In order to establish which beams will be skip loaded, and which gravity concrete beams should be considered continuous for analysis, the user must first assign beam line numbers to the beams. A beam line number defines which beams will be designed as a single continuous beam in the beam program. Beams in a contiguous line are typically assigned the same beam line number, but the user can manually assign almost any continuous line of beams to the same beam line as described below.

- Select **Assign – Beam Line – Automatic** from the menu bar to display the dialog box.




- Our model should not have any skewed beams but the beams on either side of a girder might be slightly off center. Set the Beam offset to 2.
- In the Gravity Beam Fixity area, check the box to Automatically Assign Fixity and Fix all beams.
- Click **[OK]**.
- Select **View – Plan** and select the Second story.

The second story (the story with concrete beams) will have the following beam lines automatically generated. Notice that the only concrete beams are assigned beam line numbers.

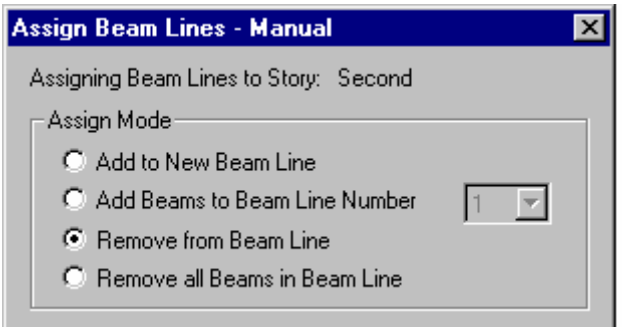



Generated Concrete Beam Lines

We will now the last segment of beam line two and assign it to be a separate line manually. Zoom into the area if you cannot easily see the beam line numbers using View – Zoom Fence.

- Select **Assign - Beam Line - Manual** from the menu bar.

If you are not already in plan mode you will be asked to select a floor plan, select the 2nd floor and click [OK]. The Assign Beam Line manual dialog will appear.




- Select the option to Remove from Beam Line.

- Click **[Single]**.
- Click on the beam between grids E and F on Grid 1.

Hint: Beams in a single beam line must be must be continuous. If you select two beams, that are not continuous, to be in the same beam line you will be issued an error message.

- Right click to return to the previous window
- Set the Assign Mode to Add to New Beam Line.
- Click **[Single]** and pick on the same beam again.

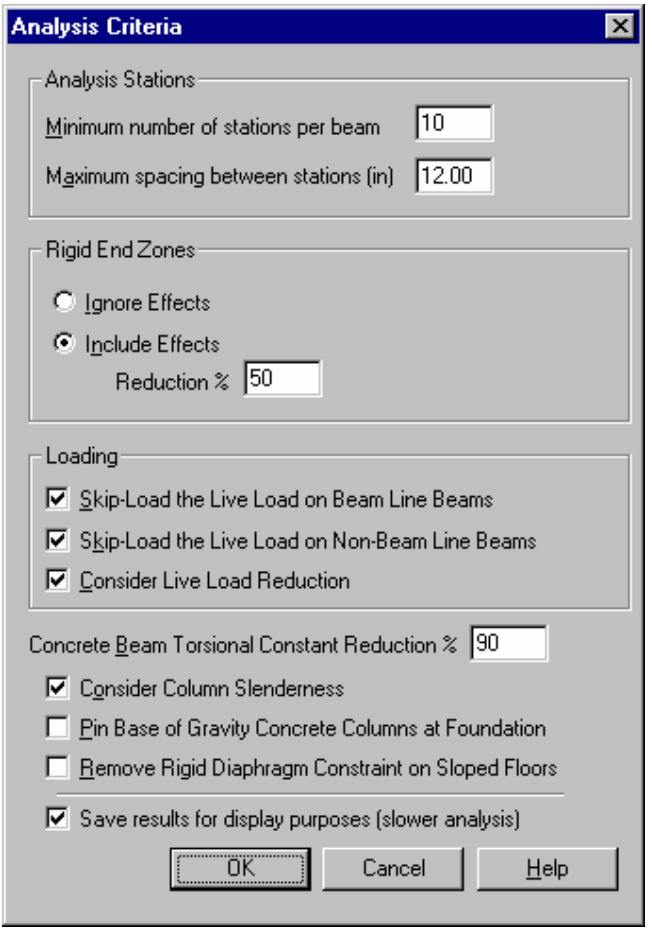
This span will now be detailed separately from the beams in the moment frame. To clear up the display:

- Select **View - Reset Model**.

## *Analysis Criteria*

Before the analysis is performed make sure the analysis criteria are appropriate.

- Select **Criteria - Analysis**. The analysis criteria dialog will appear.




Set the various analysis criteria as follows:

- Minimum number of stations per beam: 10.
- Maximum spacing between stations: 12in (250mm).
- Rigid End Zones: Include effects.

- Reduction 50%.
- Skip - load live load on Beam Line beams: Yes.
- Skip - load live load on non-beam line beams: Yes.
- Consider live load reduction: Yes.
- Concrete beam torsional constant reduction: 90%. This prevents girders with unbalanced loads from taking too much torsion. It accounts for the additional cracking in the concrete.
- Consider column slenderness: Yes.
- Pin base of gravity concrete columns at foundation: No.
- Remove Rigid Diaphragm Constrain on Sloped Floors: No.
- Save results for display purposes (slower analysis).
- Click **[OK]**.

Note; many of the settings in this dialog can have a significant effect on the speed of the analysis and design. In particular the greater the number of stations per beam and the larger the number of load cases to be analyzed the longer the analysis and design. If you are only interested in performing a quick preliminary design, increase the maximum space between stations and switch off skip loading. You can also turn off the option to Save results for display purposes – this won't prevent you from designing the members.

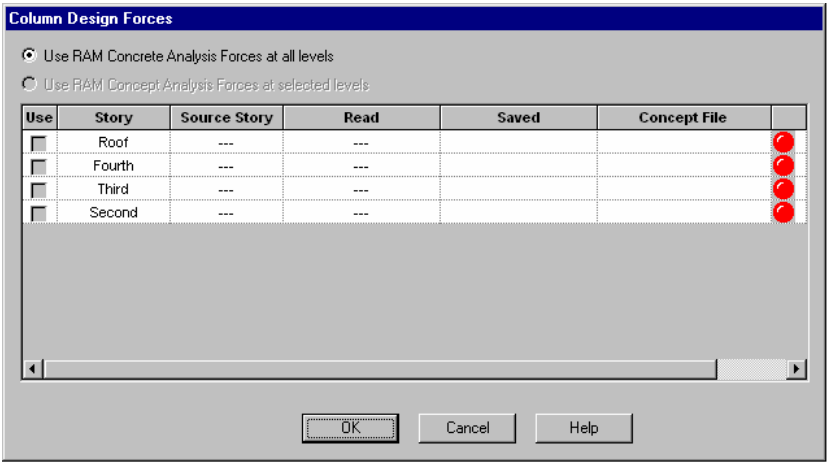
## *Other Criteria*

The other criteria should also be set before running the analysis. Start with the design code.

- Select **Criteria - Code**.
- Select ACI 318-02 for the code.
- Click **[OK]**.

The criteria for column Design Forces allows you to choose whether to use the forces from the RAM Concrete analysis or to use the column design forces from one or more RAM Conce*pt* models instead.

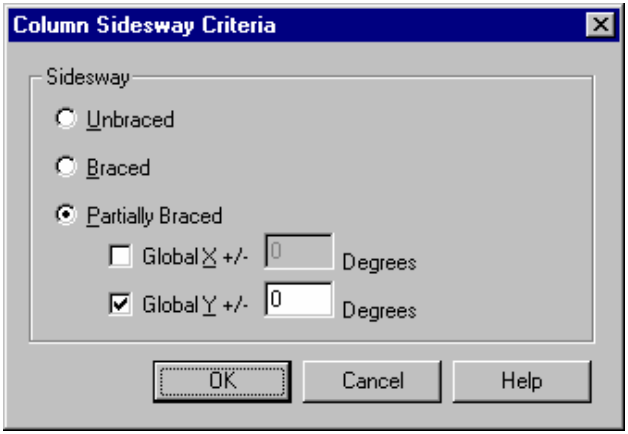
- Select **Criteria - Column Forces**.




- Click **[OK]**.

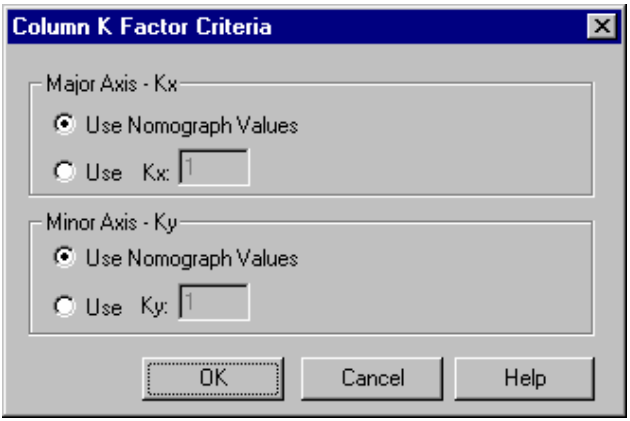
The sidesway criteria establishes whether the column should be considered Braced or Unbraced.

- Select **Criteria - Sidesway**.




- Select the option for Partially Braced.
- Check the second box for Global Y axis. This means that our moment frames will be unbraced in the X axis and braced in the Y where there are shear walls.
- Click **[OK]**.

- Select **Criteria – K Factor**.



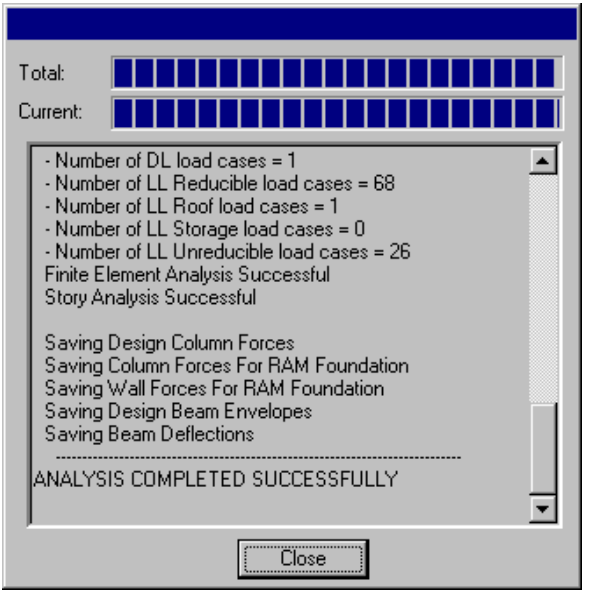

- Select to use the Nomograph in both axes.
- Click **[OK]**.

## *Gravity Analysis*

Now that the various criteria are set it time to perform the analysis. To perform the analysis:

- Select **Process - Analyze**.

The analysis progress window will appear:




This window shows the progress of the analysis and the number of load cases that were automatically generated for each story. The two progress bars at the top of the window show the progress of the analysis.

It is required that all members have an assigned size to complete the concrete analysis. In order to assure that all the steel members have an assigned size, **be sure to run the steel beam and columns modules first**.

In the RAM Concrete analysis, a finite element model is generated for each story and skip load patterns are automatically generated for the live loads. For each story a full finite element analysis is performed and the reactions carried down to be applied to the analysis of the story below. For this reason mainly, the results obtained from RAM Concrete may differ from the results for the gravity loads in RAM Frame or in RAM Steel for that matter.

- When the analysis is complete click **[Close]** to close the window.

## *On-Screen Results*

Once the analysis is complete you can display several different analysis results on the screen. Depending on the current view and the model size the screen could be extremely cluttered when results are displayed. It may be desirable to view the results in elevation or plan view and/or with the extruded shape setting removed (i.e. in stick drawing mode).

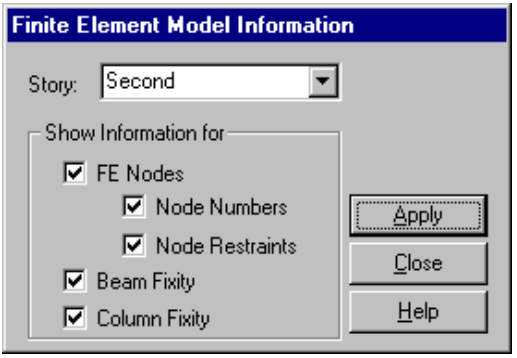
To ready the model view for best display of results on the entire structure:

- Select **View - 3D**.
- Select **View - Zoom - Full**.
- Select **View - Resolution - Low**.

### Results – Finite Element Model information

To view the finite element model for a particular story:

- Select **Process - Results - FE Model**.




The FE Model Info Window will appear. To view the finite element model that was created and analyzed for the second story:

- Select Second from the Story dropdown box.
- Keep all the display options selected.
- Click **[Apply]**.

The display will change to show only those members that were considered in the analysis of the second floor. The member fixity conditions (released, fixed) are displayed, as are the finite elements, node numbers and restraint

conditions. Note that nodes are restrained, at the levels above and below the current story, where they are braced according to the column bracing criteria.

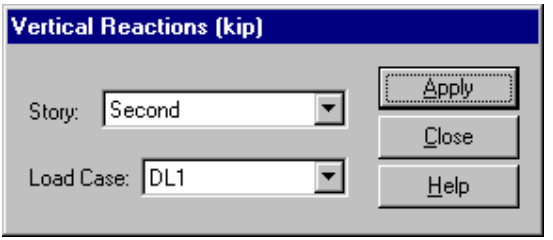
You can zoom, rotate and print the screen display while the FE Model Info window is displayed. Closing the window will reset the model and remove all the finite element settings.

If there is an error during the analysis it is possible that a node number (or location) will be provided in the error message. The analysis will terminate, but the Finite Element Model of the story under analysis should be available so that you can identify the location (node) of the problem and perform corrective action.

## Results – Vertical Reactions

To view the vertical reactions for each load case on a particular story:

- Select **Process - Results - Vertical Reactions**.




The Vertical Reactions Window will appear. To view the vertical reactions from each load case in the analysis of the second story:

- Select Second from the Story dropdown box.
- Select DL1 from the Load Case dropdown box.
- Click **[Apply]**.

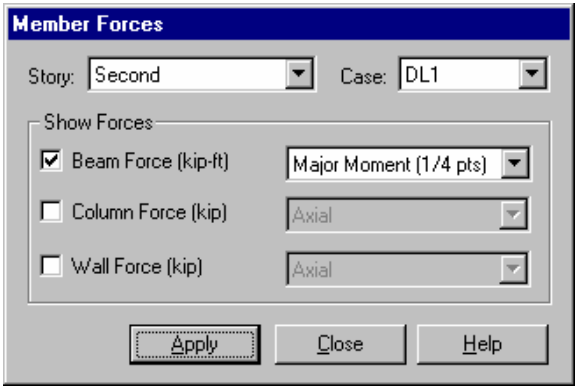
The display will change to show only the members used in the selected stories analysis, and the vertical reactions from the selected load case. The reactions displayed represent the reactions for the current story and load case analysis. However, one or more of the load cases will include the reactions from all the upper stories analyses. For dead load the loads from the previous stories analysis will have been transferred into the analysis for the current story. The reactions for the DL1 case therefore include the loads that are transferred from the upper stories.

Notice that the Live Loads have been separated into several load cases to account for load patterns. The total live load reaction can be obtained more easily through a report.

## Results – Member Forces

To view the beam and column forces for each load case on a particular story:

- Select **Process - Results - Member Forces**.




The Member Forces Window will appear. To view the member forces from each load case in the analysis of the second story:

- Select Second from the Story dropdown box.
- Select DL1 from the Load Case dropdown box.
- Check Beam Force Major Moments (1/4 pts).
- Uncheck Column Force and Wall Force.
- Click **[Apply]**.
- Select View – Plan or zoom in if the text is difficult to read.

Only significant member forces are shown for each load case selected.. If beams are skip-loaded then each type (reducible, unreducible, roof or storage) of live load on a single span will be considered its own load case. The loads on one member will affect other members due to the continuity, but the forces are not displayed when the values are insignificant.

## *View Options*

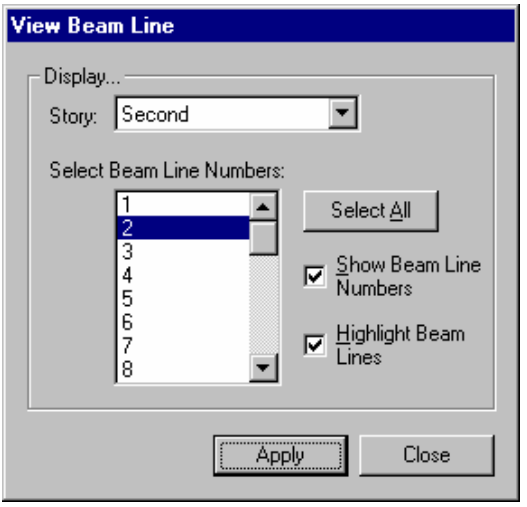
In the concrete gravity analysis mode there are several other useful display options under the view menu. Take the time to select the various menu options to view different model data or adjust your view of the model. Two menu options are particularly relevant to the Concrete Gravity Analysis mode, namely the command for viewing Beam Lines and for viewing Gravity Loads.

### View Beam Lines

To view the currently assigned Beam Line Numbers:

- Select **View - Beam Lines**.

The View Beam Line window will be displayed.

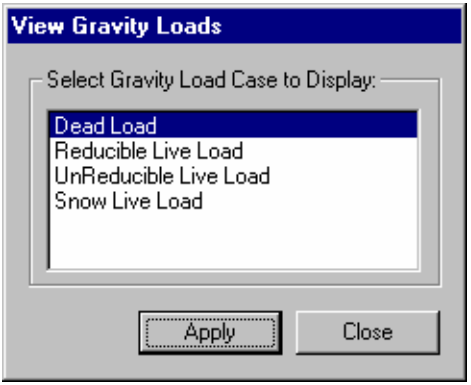



- Select Second from the Story dropdown box.
- Select the beam line numbers whose beams you want to highlight and numbered.
- Click **[Apply]**.

### View Gravity Loads

To view all the member line and point loads that were applied by the user and also calculated by the program:

- Select **View - Gravity Loads**.




- Select one of the gravity load types that you would like to view.
- Click **[Apply]**.

Note; depending on the current view and the selected load to display the screen could be extremely cluttered. It may be desirable to view the gravity loads in elevation view and/or with the extruded shape setting removed (i.e. in low resolution drawing mode).

More complete gravity load information can be obtained in report form. Select the Reports – Load Diagram command to obtain beam gravity loads.

## Reports

Reports can be generated from the Concrete Analysis just like the other modes. The printed output is generated using the Reports menu. There are several important and useful reports generated in the Concrete Gravity Analysis Mode, some of which are discussed below.

To view the analytical member properties used in the finite element analysis (this includes the calculation of effective flange width for concrete T beams and consideration of cracked section factors):

- Select **Reports – Member Analysis Properties**.
- After a few seconds the Analysis Member Properties report should appear. Hit the Page Down Key to see the rest of the report.
- Click the circled **X** to close the report.

For column design the program considers the worst skip loaded condition for live loads about each axis of the column, both top and bottom. To view the gravity column forces that will be used in the column design mode:

- Select **Reports – Column Design Forces – Single**.
- Click on one of the concrete columns below the second floor.

For beam design, the program also considers an envelope of results. To view the gravity beam envelope forces that will be considered in the beam design mode:

- Select **Reports – Beam Line Force Envelope – Single**.
- Click on one of the concrete columns below the second floor

Note; for lateral beams and columns the gravity forces shown in these reports will normally be combined in the Design Modes with lateral forces from the analysis performed in RAM Frame.

Take the time now to review the various other reports available in the Reports menu.

# RAM Concrete Beam

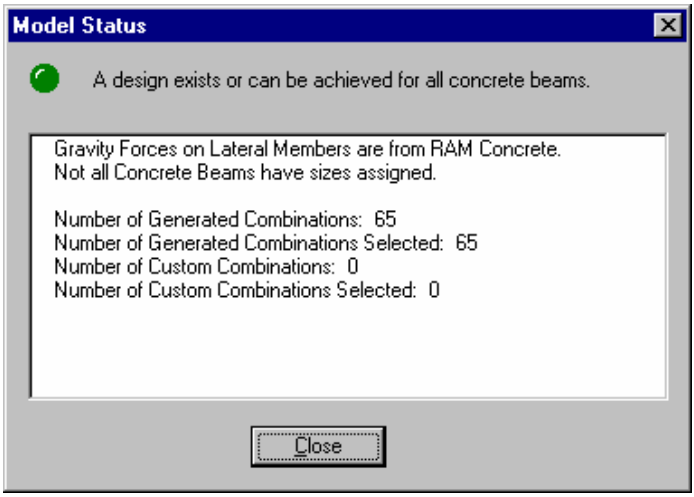
To design all your concrete beams you need to be in the Concrete Beam Design Mode. To switch to the concrete beam mode:

- Select **Mode - Concrete Beam**.

The mode drop-down on the toolbar will indicate when you are in the Concrete Beam Design Mode. The same toolbar can be used to switch between modes rather than using the menus.

Note; before you can enter the concrete beam design mode, the concrete gravity analysis must be performed. If you have lateral concrete beams you will also want to make sure your model has been analyzed in RAM Frame. If you are unsure of the current status of the model:

- Select the **File - Model Status**.

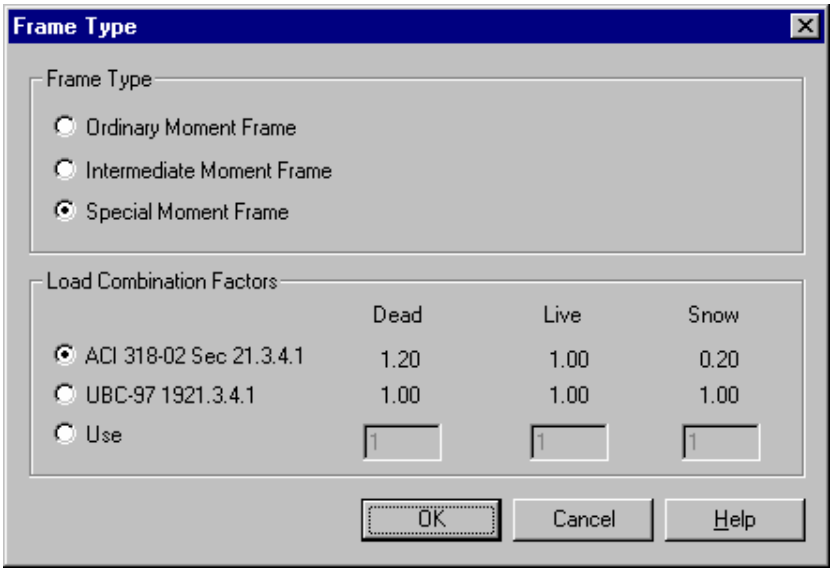



The window that appears will indicate anything that is missing.

## *Beam Design Criteria*

The program has an extensive set of design defaults to customize the way concrete beams are designed and detailed. First you need to designate the concrete moment frame type:

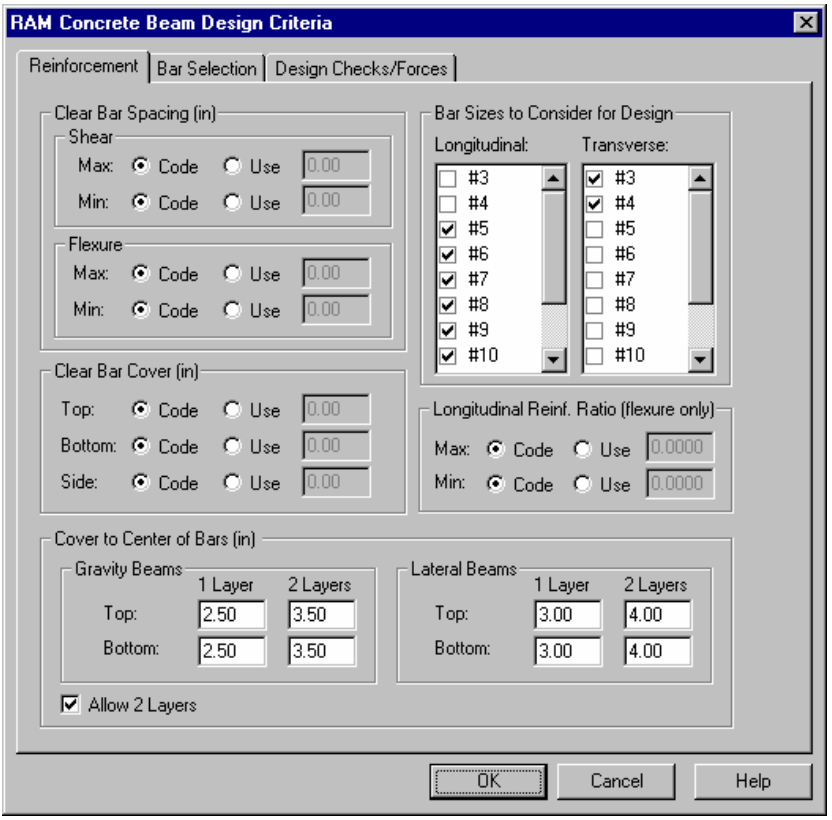
- Select **Criteria – Frame Type**.




- Set the Frame Type to Special Moment Frame.
- Set the Load Combination Factors to ACI 318-02 Sec. 21.3.4.1.
- Click **[OK]**.

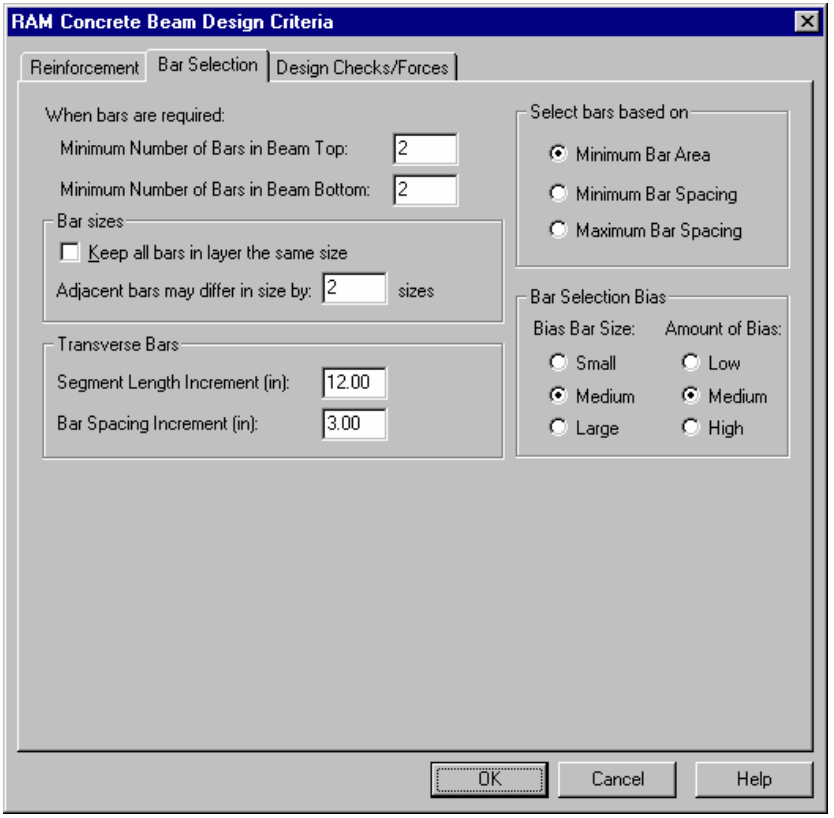
To set up the beam design defaults which will be used for all beams:

- Select **Criteria - Beam Design**.



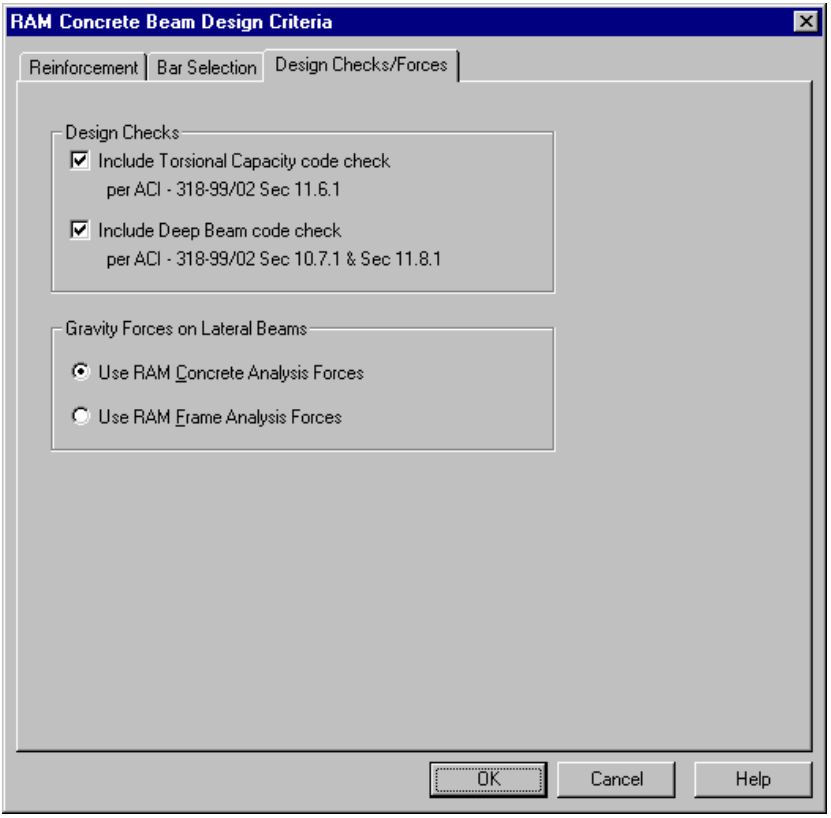

- Set the program to use the code limits for all cover and spacing.
- Check the Longitudinal Bars to Consider from #5 to #10 only.
- Check Transverse Bars #3 and #4 only.
- Use the defaults for Cover to Center of Bars. This is the value used by the program to establish the depth of the reinforcing.

- Click on the Bar Selection Tab to set the more beam design defaults:




- Set the Minimum number of bars Top and Bottom to 2.
- Deselect Keep all bars in layer the same size.
- Type 2 for Adjacent bars may differ in size by.
- Type 12 for Transverse bar Segment Length.
- Type 3 for Spacing Increment.

- Click on the Design Checks/Forces tab:



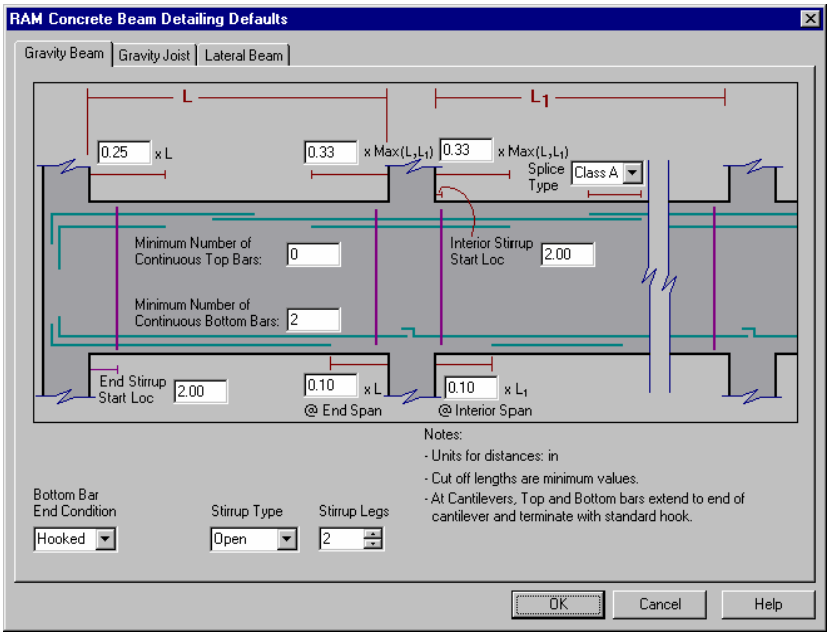

- For now, include both Design Checks. At this time, the concrete beam design program does not design beams for additional torsional reinforcing. Nor does it consider the special detailing requirements of deep beams. When a beam exceeds the torsional limit of the concrete or the maximum depth to span ratio a warning will be given. Some users prefer not to get these warnings so that they can concentrate of other types of design issues.
- For Gravity Forces on Lateral Beams. select to Use RAM Concrete Analysis Forces. In rare cases, the 3D finite element analysis performed by RAM Frame may actually be more accurate than the floor-by-floor approach used in the Concrete Analysis mode.
- Click **[OK]**.

## *Detailing Defaults*

To set the defaults for concrete beam bar placement

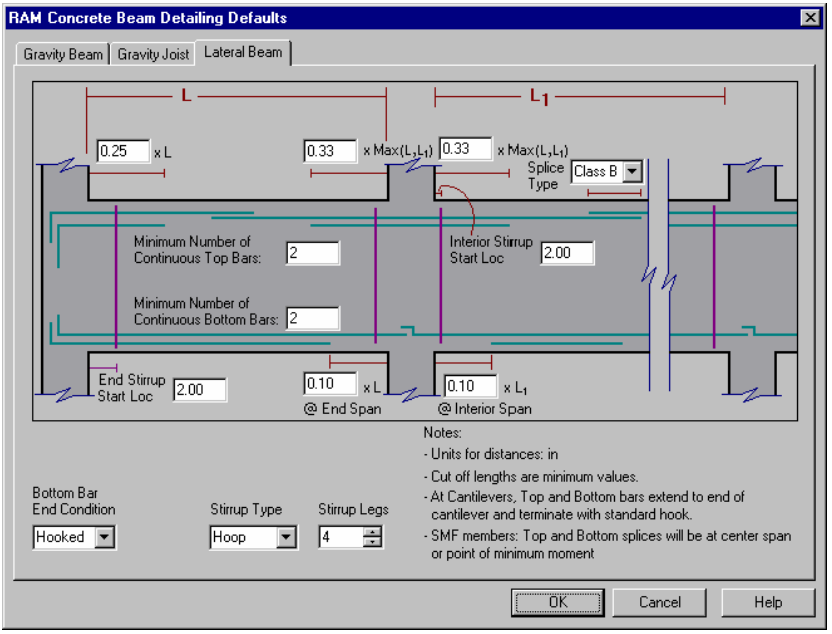
- Select **Criteria - Detailing Defaults**.

The Detailing Defaults window will appear with separate tabs for Gravity Beams, Gravity Joists and Lateral Beams. Set the various bar lengths and Stirrup start Locations to the values shown in the figure below.




- Set the Longitudinal Bar End Condition to be Hooked.
- Set the Stirrup Type to be Closed.
- Set the number of Stirrup Legs to be 2.

- Click on the Lateral Beam tab.



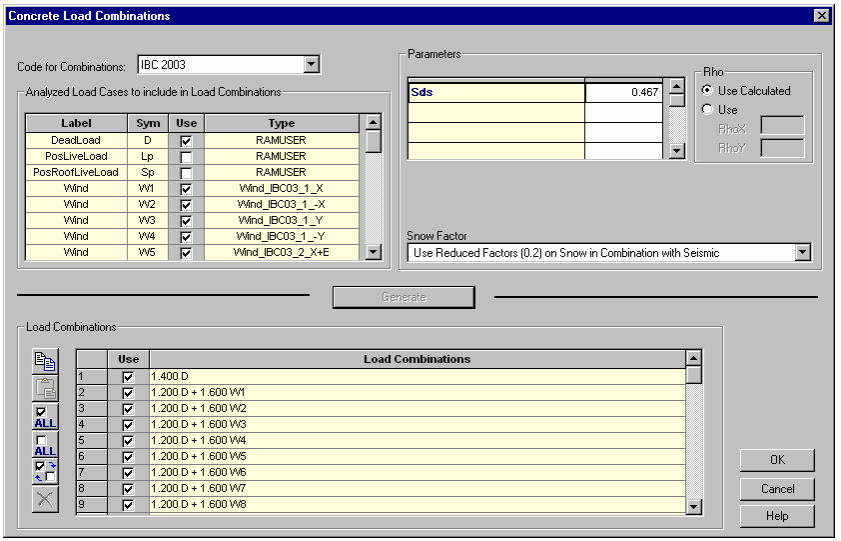

- Set the Minimum number of Continuous Top Bars to 2.
- Change the Splice Type to Class B
- Change the Stirrup Type to Hoop.
- Set the Stirrup legs to 4.
- Click **[OK]**.

At this point you could also choose to make individual member assignments using the options under the Assign Menu, but for this example we will allow the default settings to apply to all members.

## Load Combinations

Before any beams can be designed, the user must specify the load combinations to consider. These may be created manually as User Defined Load Combinations or Code Generated Combinations can be utilized. To establish the Combinations for Concrete Design:

- Select **Combinations – Generated.**




- Select IBC 2003 from the Code Combo list box.
- Type 0.432 for Sds.
- Set Rho to Use Calculated.
- Set the Snow Factor to Use Reduced Factor (0.2)…
- Click **[Generate]**. The Load Combinations box should be filled with load combinations and each should be checked to Use.
- Click **[OK]**.

## *Design All and View/Update*

With the design criteria set and the load combinations defined, we are ready to design the concrete beams. At this point, the beams should have changed from a light blue color to yellow, indicating that they are ready to be designed. To design the concrete beams:

- Select **Process – Design All**.

The design process requires numerous code checks for all of the Load Combinations and may take some time to run on large concrete models. The status indicator again displays the progress of the design and indicates when the design is complete. Click [Close] to dismiss this window. The Model graphics will be updated to display either Green or Red beams.

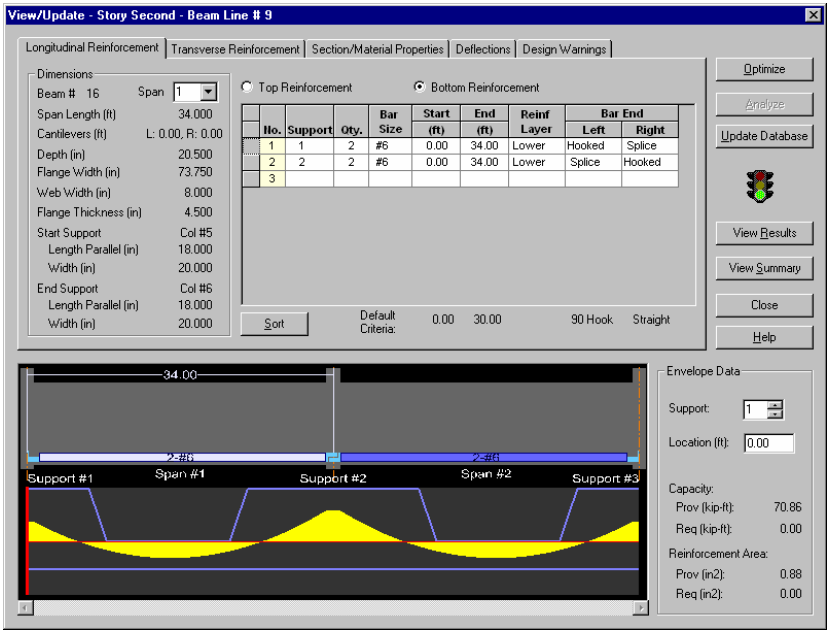
Green – Indicates that the beam was designed successfully with no design warnings.

Red – Indicates some aspect of the Concrete Beam design is insufficient or incomplete. Design warnings that elaborate on why a beam design failed can be seen in the View/Update dialog.

Blue – Indicates a successful design is Frozen by the user. (A beam that has failed will always appear red even if it is frozen.) In either case, the way to review the details of the design and make changes if desired is through the View/Update command.

- Select **View – Plan** and choose the Second Floor plan.
- Select **Process – View/Update**.
- With the mouse, select the interior beam line on Grid C.

The View/Update dialog box will appear as shown below. There are many options available to the user from this window. For a complete explanation of the View/Update window see the RAM Concrete manual. For this example, we will touch a couple of the highlights.




The View\Update dialog is broken into 5 separate tabs. The first tab for Longitudinal reinforcement shows by default. Here you can see the layout of the longitudinal top Reinforcement.

Notice the diagram at the bottom which graphically indicates the reinforcing and plots the demand envelope (yellow diagram) and capacity (blue line) for the length of the member.

To switch the display to show Bottom Reinforcement instead:

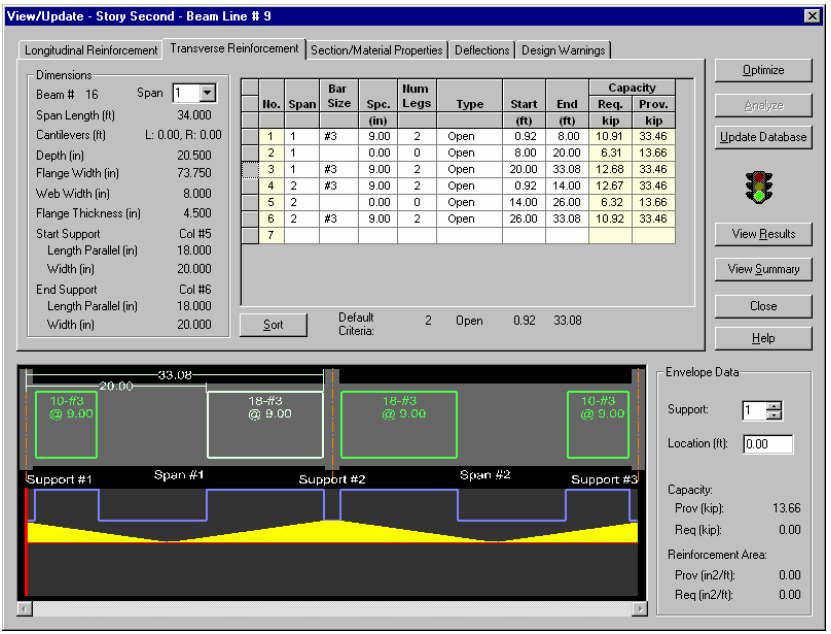
- Click the radio button near the top labeled Bottom Reinforcement. The graphic along with the reported Capacity information will be updated.

To see the capacity at a particular location:

- Click on the bottom graphic where the moment diagram is displayed somewhere near the greatest moment. The red line slider will move to that position and the required and provided member capacity information on the right side will be updated to reflect that particular location.

Changes to the reinforcement can be made in the spreadsheet of reinforcement in the top portion of the window.

- While reviewing Bottom Reinforcement, click in the Bar Size field for the first bar set and increase the bars by one size (#6). After doing so, the traffic light changes to yellow indicating that the results are no longer current.
- Do the same thing for the second bar set.
- Click **[Analyze]** to have the results updated. The light should still be green in this case.
- Click on the Transverse Reinforcement tab. Notice that the bottom graphic is adjusted to show shear information rather than bending now.




- Here the stirrups are grouped into three sections. The first section has #3 stirrups at 9” on center and continues for 8’ from the face of the column.
- Click in the second row where the bar size is blank.

- Reduce the End location by 3 ft (e.g. from 23 to 20). The third row start location will automatically be bumped back as well, making the third bar set longer.
- Click in the 4th row and increase the end location by 3 ft (e.g. from 11 to 14).
- Click **[Analyze]** again to review the altered design.
- To see the reported details of the design click **[View Results]**.

The Concrete Beam Design Report should come up. Review the report. For more information on the designs performed in this check refer to the technical portion of the RAM Concrete Manual.

- Close the report and return to the View/Update dialog box.
- Assuming the modified design works, click **[Update Database]** to save the changes.
- Click **[Close]** to return to the screen display.

If the new design is acceptable, the color of the member will change to dark blue indicating that the design has been modified by the user. This design will not change again unless the user changes it manually, or clears the design using the Process – Clear Beam Design command.

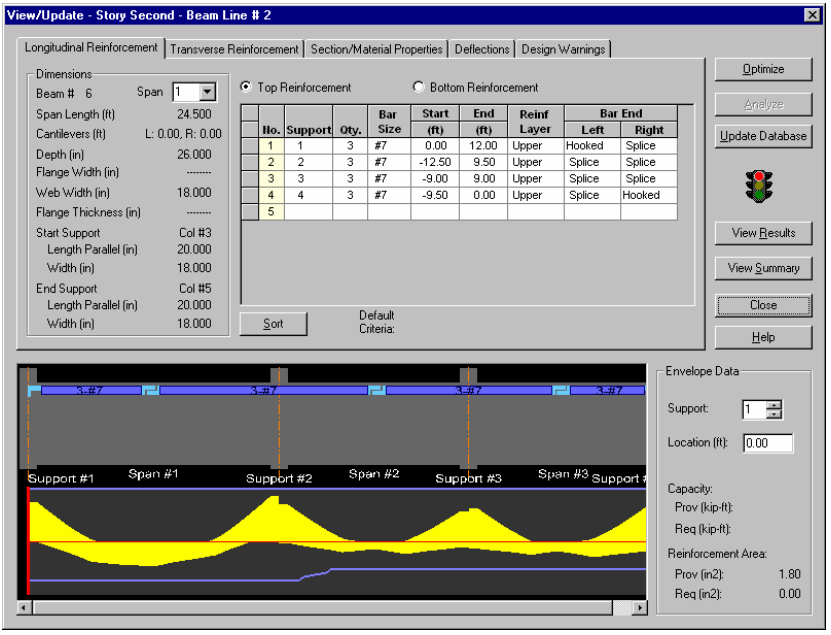
- Now select one of the moment frame beams with View/Update.

There are two significant differences about this beam line. First note that the envelope of the demand moments (the yellow diagram) is thicker than the earlier gravity beam. That's because this is a lateral member with more load combinations to consider. The second is the traffic light. If the traffic light is red then some aspect of the design has a warning.

To see what that warning is:

- Click the Design Warnings tab at the top of the window. This beam happens to have the following warning for each span:

  * Torsional Reinforcement Required. Section Torsional Capacity Not Acceptable (Ld/Cap = 2.72) per ACI 318-02 Sec. 11.6.4.1 a.




This warning will occur whenever the member Torsion exceeds the minimal capacity of the concrete as specified in the code. It is important to note that the rest of the design is still applicable, despite this warning. To avoid seeing torsion warnings at all:

- Select **Criteria - Beam Design**.
- Click on the Design Checks/Forces tab.
- Uncheck both optional design checks for Torsion and Deep Beams.
- Click **[OK]**.

When you change one of the design criteria you will get a reminder that the designs will have to be rechecked.

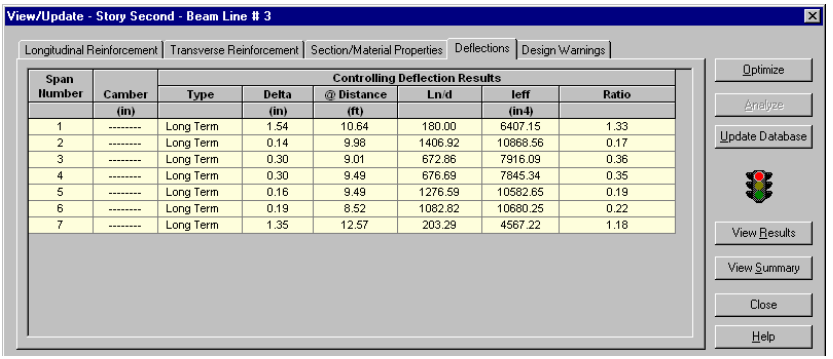
- Select **Process - Design All**.

## *Concrete Beam Deflection*

Any beams that are still red at this point have some other kind of warning.

- Select **Process – View/Update**.
- Pick the center girder on line 3.
- Click on the Design Warnings tab and you should see this warning:

  * Beam 7 fails Long Term Deflection check. Deflection ratio=1.33
- Click on the Deflections tab.




| Span Number | Camber | Controlling Deflection Results | | | | | |
|---|---|---|---|---|---|---|---|
| | | Type | Delta | @ Distance | Ln/d | Ieff | Ratio |
| | (in) | | (in) | (ft) | | (in4) | |
| 1 | -------- | Long Term | 1.54 | 10.64 | 180.00 | 6407.15 | 1.33 |
| 2 | -------- | Long Term | 0.14 | 9.98 | 1406.92 | 10868.56 | 0.17 |
| 3 | -------- | Long Term | 0.30 | 9.01 | 672.86 | 7916.09 | 0.36 |
| 4 | -------- | Long Term | 0.30 | 9.49 | 676.69 | 7845.34 | 0.35 |
| 5 | -------- | Long Term | 0.16 | 9.49 | 1276.59 | 10582.65 | 0.19 |
| 6 | -------- | Long Term | 0.19 | 8.52 | 1082.82 | 10680.25 | 0.22 |
| 7 | -------- | Long Term | 1.35 | 12.57 | 203.29 | 4567.22 | 1.18 |

Here the table lists the controlling deflection criteria. The first and long spans of the beam line are currently exceeding the long term deflection limits set in the deflection criteria. To solve the problem we could try a larger size beam or possibly re-design the beam using an effective "T" section.

- For now, just click **[Close]** to return to the main view.

To see the deflection ratios for all beams at once:

- Select **Process – Results – Deflections**.

The screen will be updated to display the deflection using a color scale. The graphic can also be set to show Span to deflection ratio, Effective moment of Inertia or Deflection ratio.

- Click **[Close]** when finished.
- Select **View – Reset Model** to clear the screen.

## *Copy Design*

Because the design and detailing of concrete beams can be complicated, it is often desired to use an identical bar layout for typical beams. In this example, many of the beams can be detailed the same. To apply identical design of one beam to another:

- Select **Process – Copy Design – Single to Single**.

A target cursor with an arrow will appear with the arrow pointing up indicating that the program is expecting you to select the beam to copy the design from.

- Click on the first beam previously designed and updated.

The arrow on the cursor will now point down indicating that the program expects you to select the beam to copy that design to. When using the Single-to-Fence option the cursor works by windowing all the beams you wish to be affected.

- Click on a similar beam on the next column line (e.g. Grid D).

If the design is acceptable, the color of that beam will change to Blue indicating the design is frozen. If the same rebar is insufficient in the applied beam, then it will be painted red to let you know there is a problem. The design can be restored to the optimum design using the Process – Clear Beam Design command as before.

Note; you can only copy from one beam to another if the geometry is identical. You cannot copy the beam reinforcing from the beam on the column line to one of the infill beams because the support conditions are different. This makes the Single to Fence and Single to All copy options quite useful because you don't have to worry about copying rebar that won't fit in the various beams.

- Select **Process – Copy Design – Single to All**.
- Click on the same beam as before.
- Now all the column line beams will be adjusted at once but the other beams are left alone.

## *Reports*

Many different reports can be generated from the Concrete Beam Module. The printed output is mostly generated using the Reports menu.

To print the load combinations used in concrete design:

- Select **Reports – Screen** (if it is not already selected).
- Select **Reports – Load Combinations**.
- Click the circled **X** to close the report.

To print a summary of the concrete beam designs:

- Select **Reports – Beam Design Summary**.
- Close the report.

A similar output is available for a single member through the View/Update dialog box by clicking [View Summary]. The complete design results are also available from the report menu.

Take time now to review the various reports available in the Reports menu.

# RAM Concrete Column

To design all your concrete column you need to be in the Concrete Column Design Mode. To switch to the concrete beam mode:

- Select **Mode – Concrete Column**.

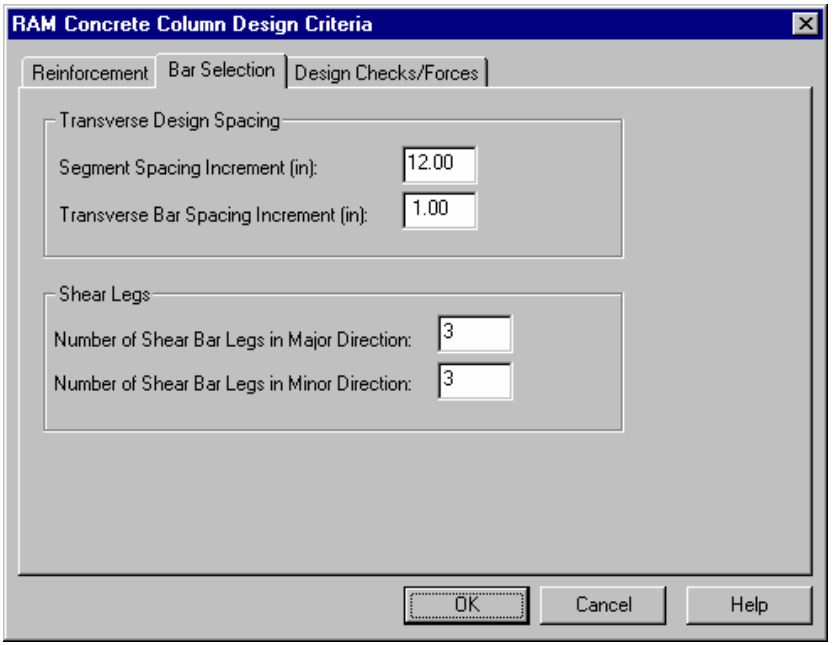
Before you can enter the concrete column design mode the concrete gravity analysis should have been performed. If you have lateral concrete columns you will also want to make sure your model has been analyzed in RAM Frame. Finally, if you are using Special Moment resisting frames, all of the concrete beams must be designed before designing the columns.

The mode drop-down on the toolbar will indicate you are in column design mode.

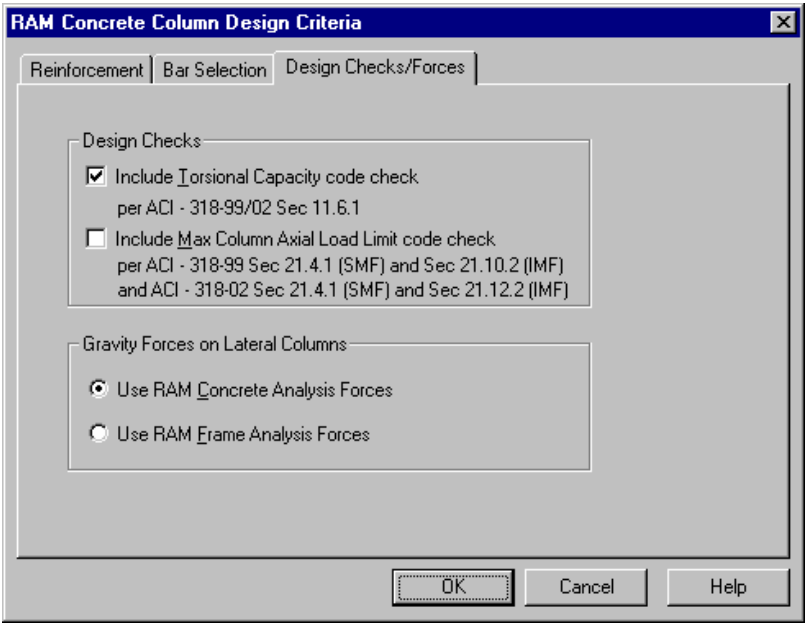
## *Column Design Criteria*

The program has an extensive set of design defaults to customize the way concrete columns are designed and detailed. To set up the column design defaults which will be used for all columns in the model:

- Select **Criteria – Column Design**.
- Select the Code limit for each of the options under the Reinforcement tab.



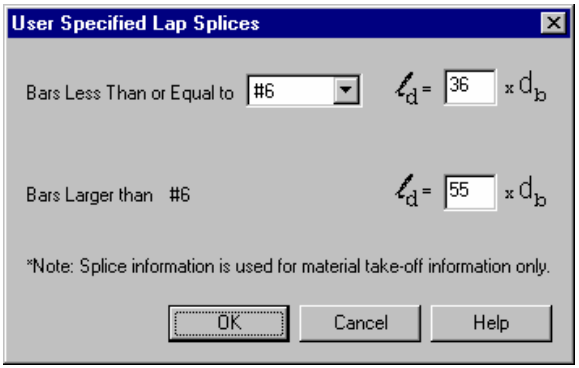

- Click on the Bar Selection Tab to set more column design defaults.
- Type 12 for Transverse Design Spacing Segment Spacing Increment
- Type 1 for Transverse Bar Spacing Increment (the special moment frame columns may require tie spacing other than 3', 6" etc.)
- Type 3 for the Number of Shear Bar Legs in each.
- Click on the Design Checks/Forces tab.




- Uncheck the option to Include the Max Column Axial Load Limit.
- Click **[OK]**.

The other design criteria for Reinforced Concrete Columns is the Lap Spacing which can be customized by the user.

- Select **Criteria - Column Lap Splice**.

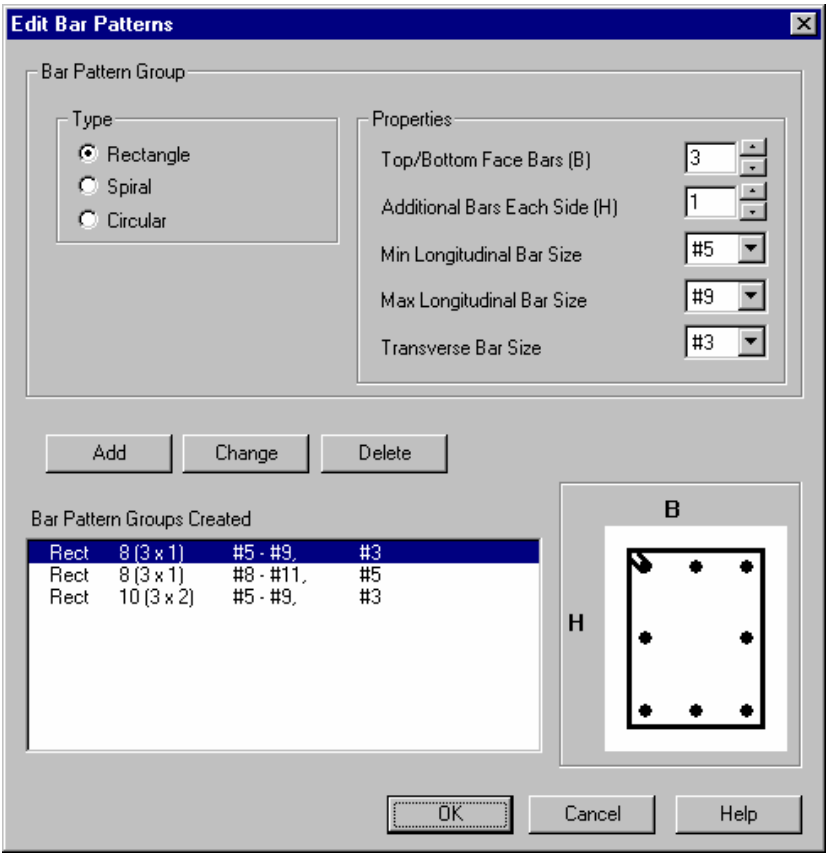



- Use the Lap Splice options shown above.

## Assign Bar Patterns

Before the Columns can be designed we must specify a bar patterns that we wish to use. You can define any number of bar patterns or layout and the program can consider up to three of these layout for each column that it designs. To define the bar patterns:

- Select **Assign - Edit Bar Patterns**.




For this example we will define three bar patterns.

- Select 3 for Top/Bottom Face Bars (B).
- Select 1 for Additional Bars Each Side (H).
- Select #5 for Min. Longitudinal Bar Size.

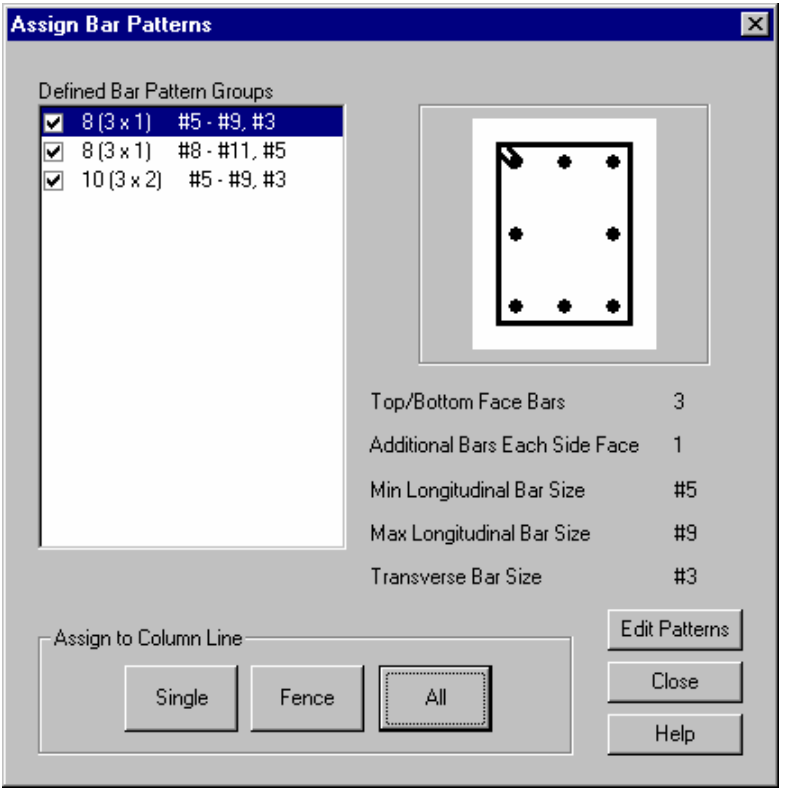
- Select #9 for Max Longitudinal Bar Size.
- Select #3 for Transverse Bar Size.
- Click **[Add]**. A new line of data will appear in the Bar Pattern Groups Created Window.
- Increase the Min Longitudinal Bar size to #8.
- Increase the Max Longitudinal Bar size to #11.
- Increase the Transverse Bar Size to #5.
- Click **[Add]** to create the second pattern.
- Increase the Additional Bars Each Side to 2.
- Select #5 for Min. Longitudinal Bar Size.
- Select #9 for Max Longitudinal Bar Size.
- Select #3 for Transverse Bar Size.
- Click **[Add]** to create the last pattern of 10 total bars.
- Click **[OK]**.

Now the bar patterns we just defined must be assigned to the concrete columns in the model. To do this:

- Select **Assign – Bar Patterns**.

The Assign Bar Patterns dialog box appears:

- Check the box next to each of the patterns that we defined. The graphic on the side indicates the pattern information for the highlighted pattern.
- Click **[All]** to assign those two pattern options for design to all of the concrete columns.




Note; you can access the Edit Bar Patterns dialog box directly from this window as well.

At this point you could generate the Load Combinations for design, but the combinations used in the Concrete Beam Module are already defined so we can skip ahead to designing the columns.

## *Design All and View/Update*

With the design criteria set and the load combinations defined, we are ready to design the concrete columns. At this point, the columns should have changed from a light blue color to yellow, indicating that they are ready to be designed. To design the concrete columns:

- Select **Process - Design All**.

The design process requires numerous code checks for all of the Load Combinations and may take some time to run on large concrete models. The status indicator again displays the progress of the design and indicates when the design is complete.

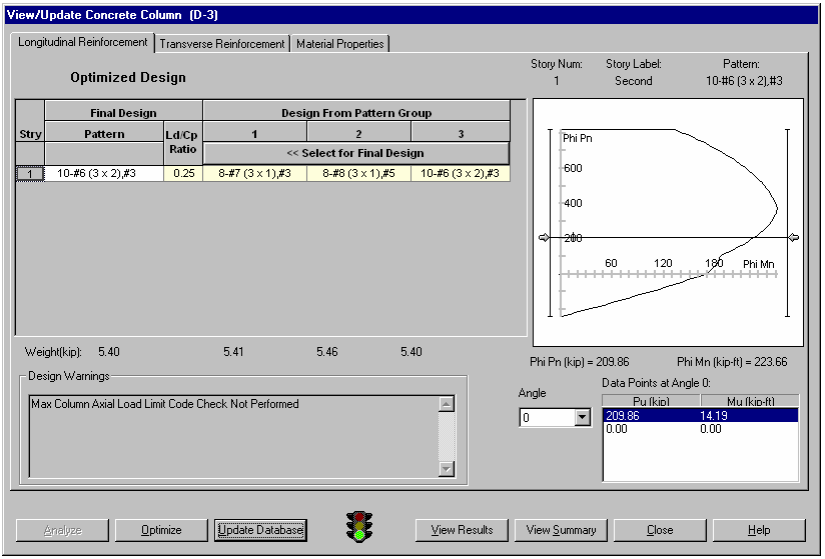
- Click **[Close]** to dismiss this window.

The Model graphics will be updated to display either Green or Red columns.

> Green – Indicates that the column was designed successfully with no design warnings.
>
> Red – Indicates some aspect of the Concrete Column design is insufficient or incomplete. Complete design warnings can be seen in View/Update.
>
> Blue – Indicates a successful design that is Frozen or Updated by the user. (Columns that have failed will always appear red even if they are frozen.)

- Select **Process - View/Update**.
- With the Target cursor select one of the interior gravity columns (this can be done from any plan view, elevation view or directly from the 3D view).

The View/Update dialog box will appear as shown below. There are many options available to the user from this window. For a complete explanation of the View/Update window see the RAM Concrete manual.



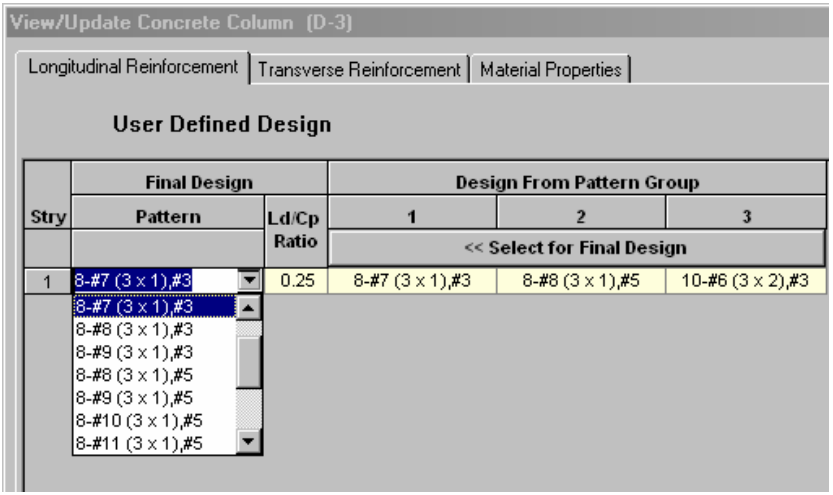

The Column View/Update dialog box is separated into three tabs for Longitudinal Reinforcement which is initially displayed, Transverse Reinforcement and Material Properties.

The program determined bar pattern is listed in the upper left corner. This reflects the optimum working design from the assigned bar pattern groups. The design can be changed by simply clicking in the cell with the design and picking another set of bars.

On the right side is an interaction surface for the currently selected level of the column. The program creates an interaction surface for each 2-degree increment around the column. When designing the column, the program investigates each data point (a set of axial forces and moments from a particular load combination or pattern). The data point that is closest to the interaction surface is the point initially shown in the diagram.

At the bottom of the widow is an area for design warnings related to longitudinal reinforcement. There is a separate area for transverse design warnings on the second tab.

- Click in the Final Design Pattern cell for the 1st level
- From the drop down list that appears, select 8-#7 (3x1), #3.

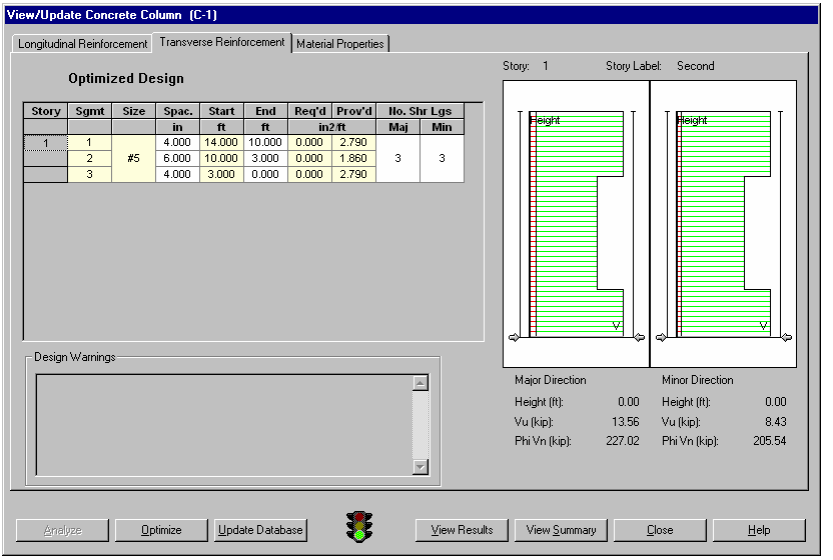



- This invalidates the current results.
- Click **[Analyze]** to update the display for the modified design.
- The program picked 10#6 bars because that requires less total steel than 8-#7 bars, but the 8-#7 bars pattern might be preferable. If you want to stick with this selection, click **[Update Database]**.
- Click **[Close]** to return to the main graphic.

The lateral columns in this model have many more design requirements, not only because of the additional loads, but also because they were designated as Special Moment Frames.

- While in View/Update mode, select one of the moment frame columns.

In this case, the program selects one of the bar sets with #5 ties. To review the transverse reinforcement click on the Transverse Reinforcement Tab.




| Story | Sgmt | Size | Spac. | Start | End | Req'd | Prov'd | No. Shr Lgs | |
|---|---|---|---|---|---|---|---|---|---|
| | | | in | ft | ft | in2/ft | | Maj | Min |
| 1 | 1 | #5 | 4.000 | 14.000 | 10.000 | 0.000 | 2.790 | 3 | 3 |
| | 2 | | 6.000 | 10.000 | 3.000 | 0.000 | 1.860 | | |
| | 3 | | 4.000 | 3.000 | 0.000 | 0.000 | 2.790 | | |

Notice that the shear reinforcement has to be quite concentrated near the ends of the column in order to meet the special detailing requirements of the code.

To review the complete design results.

- Click **[View Results]**.

The Concrete Column Design Report will open. Review the report. For more information on the designs performed in this check refer to the technical portion of the RAM Concrete Manual. Note; with multi-level columns, the report will start from the top level and work down to the base.

- Close the report to return to the View/Update dialog box.
- Click **[Close]** to return to the main graphics window.

## *Copy Design*

As with concrete beams, it is often desired to use an identical bar layout for typical columns. To apply the same design to another column:

- Select **View – Elevation**.
- Click on any beam on Grid 3 to see the interior columns.
- Select **Process – Copy Column Line – Single to Fence**.

A target cursor with an arrow will appear with the arrow pointing up indicating that the program is expecting you to select the column to copy the design from.

- Click on the gravity column previously modified.
- The fence cursor will then be ready for you to select all other columns that you wish to have use this same design. Fence all the interior gravity columns from C-3 to H-3.
- Click **[Close]** to close the pop-up window.

If the design is acceptable, the color of that column will change to Blue indicating the design is frozen. The design can be restored to the optimum design using the Process – Clear Column Line command. Unlike the copy beam design command, the copy column design command transfers not only the reinforcement, but the column size as well, so do not apply the change to columns that need to remain a different size. It should also be noted that the entire column line is copied with this command so you will not be able to copy column of different height or number of stories.

## *Reports*

Many different reports can be generated from the Concrete Column Module. The printed output is mostly generated using the Reports menu.

To print a summary of the concrete column designs:

- Select **Reports – Column Design Summary**.
- Close the report.

A similar output is available for a single member through the View/Update dialog box by clicking View Summary. The complete column design results are also available from the report menu.

Take time now to review the various reports available in the Reports menu.

- Click **File – Save**.
- Select **File – Exit**.

This completes the Tutorial for RAM Concrete. Proceed to the next section in order to work the RAM Foundation Tutorial.

# RAM Foundation

This section illustrates the analysis and design of the spread footings and continuous foundations in an integrated model. This section can only be completed if you have licensed and installed the RAM Foundation module. You may begin with the model that you generated in the previous portions of this tutorial, or you may open the model called RAMTutorial_v10_US.rss from the RAM Manager.

## RAM Foundation Basics

To invoke RAM Foundation from the RAM Manager:

- Select **Design – RAM Foundation** or click the last square button on the left.

Because RAM Foundation is integrated into the RAM Structural System, it uses the same model and database as RAM Frame and RAM Steel when designing foundations. In order for the program to design the foundations, the loads must be determined by running the gravity column design and/or lateral analysis in RAM Frame. If this has not been done, do so now, then proceed with this Tutorial. While most of the information needed for foundation design is taken from the database, some data does need to be entered in the Foundation program itself.

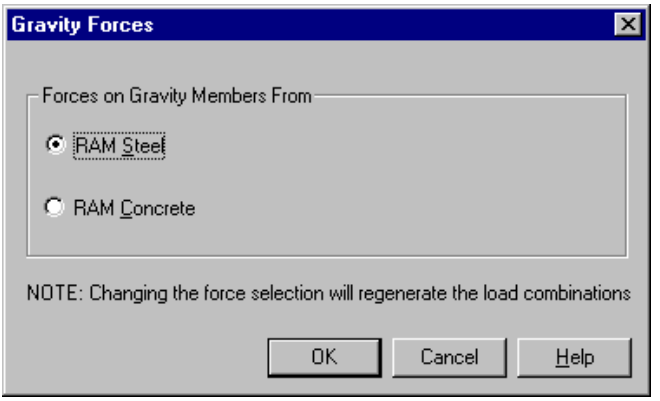
Before foundations can be designed, the following must be defined:

1. Soil Capacity.
2. Base plate size for lateral steel columns.
3. Width for Continuous foundations.
4. At least one load combination for Concrete.
5. At least one load combination for Soil.
6. Pile capacity information and layout.

Once this is complete, a design can be performed, but we recommend that you review the design and optimization criteria closely first.

To establish the source for the design loads:

- Select **Criteria - Forces**.




Leave the Forces on Gravity Members criteria set to use RAM Steel. This means that the program will use the simple, tributary analysis of the Steel Column module for determining the foundation loads. The RAM Concrete Forces could also be used and there should be only small variations in the results for this model.

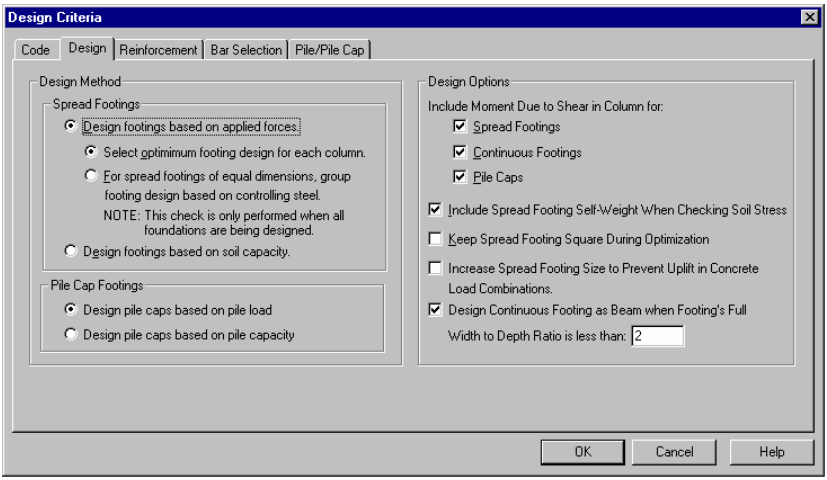
To set the design criteria:

- Select **Criteria - Design**.
- Choose ACI 318-02 as the desired code on the Code Tab
- Click the Design tab.

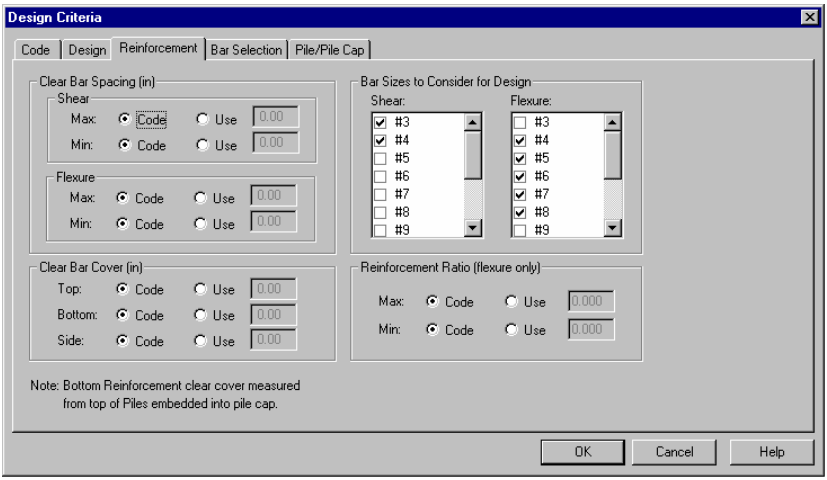
These parameters have a significant effect on the way the foundations are designed. For a complete description of the option see the RAM Foundation manual. For this example:

- Set the Spread Footings to be designed based on applied forces (rather than soil capacity).
- Set the Pile Caps to use the pile load as well.
- Check all three footings to Include Moments Due to Shear in Column.
- Check to Include Spread Footing Self-Weight When Checking Soil Stress.
- Check to Keep Spread Footing Square During Optimization.

- Uncheck to Increase Spread Footing Size to Prevent Uplift in Concrete Load Combinations.
- Check to Design Continuous Footing as Beam when Footing's Full Width to Depth Ratio is less than 2.




- Click on the reinforcement tab

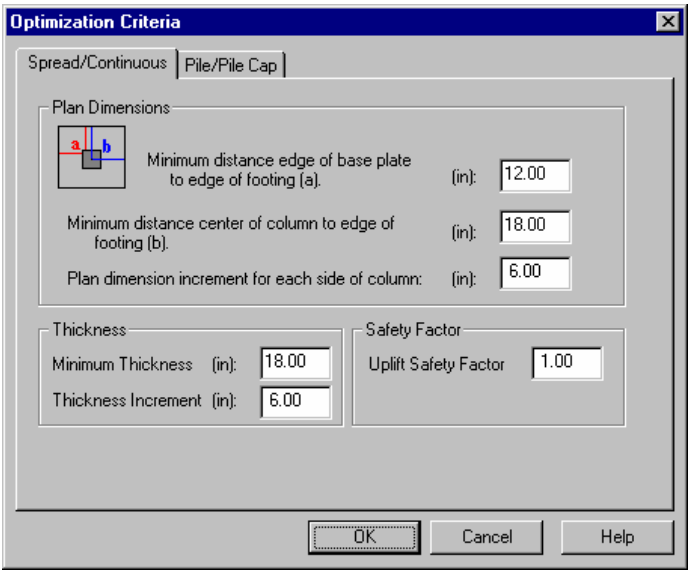



- Check the #3 and #4 bars for shear (f08 and F10) and the bars from #4 to #8 for flexure (F10-F20).
- Click **[OK]**.

## *Optimization Criteria*

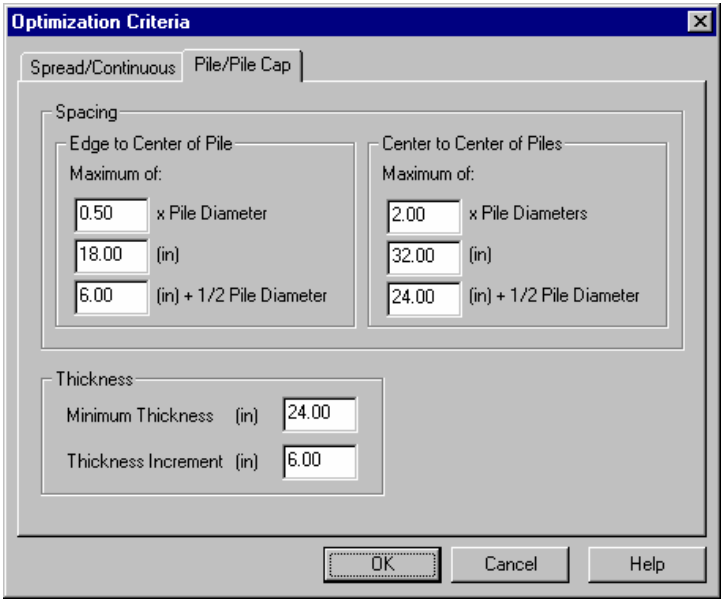
There are additional criteria that affect the design of the foundations. To set those options now:

- Select **Criteria - Optimize**.




- Set the Minimum distances from the base plate to 12 (250).
- Set the minimum from the column center to 18 (500).
- Set the Plan dimension increment to 6 (125).
- Set the minimum Thickness to 18 (500).
- Set the Thickness Increment to 6 (250).
- Set the Uplift Safety Factor to 1.

- Click on the Pile/Pile Cap tab.




- Set the spacing variables to 0.5, 18 and 6 (0.5, 450 and 150).
- Set the Center to Center options to 2, 32, and 24 (2, 800, 600).
- Set the Thickness options to 24, 6 (600, 150).
- Click **[OK]**.

## *Assign Soil Capacity*

To assign soil capacity to this model

- Select **Assign – Soil**.

Initially, only the Allowable Bearing Capacity option for entering the soil capacity is available. This option allows the engineer to idealize the soil capacity by using just one value to represent the soil capacity regardless of dimensions of the foundation or depth of the soil.

You can also create a soil table by using the wizard. This feature allows you to assign the modulus of subgrade reaction that affects the springs that will be used in the analysis of the footing.

To assign an Allowable Bearing Capacity:

- Enter 4 (200) in the Allowable Bearing Capacity edit box.
- Click **[All]**.

Note; All of the Assign commands in RAM Foundation give the option of Single, Fence or All for making assignments. [Single] will change the cursor to the target and allow for assignments on a foundation - by - foundation basis. [Fence] will change the cursor to a rubber band. Assignments are made to foundations completely enclosed within the rubber band. [All] assigns the value to all foundations within the model. If a new foundation is added to the model it must also have a soil assignment made.

## Assign Base Plate Sizes to Lateral Columns

The Assign – Base Plate Size command allows the engineer to assign different base plate sizes to different columns on a continuous foundation.

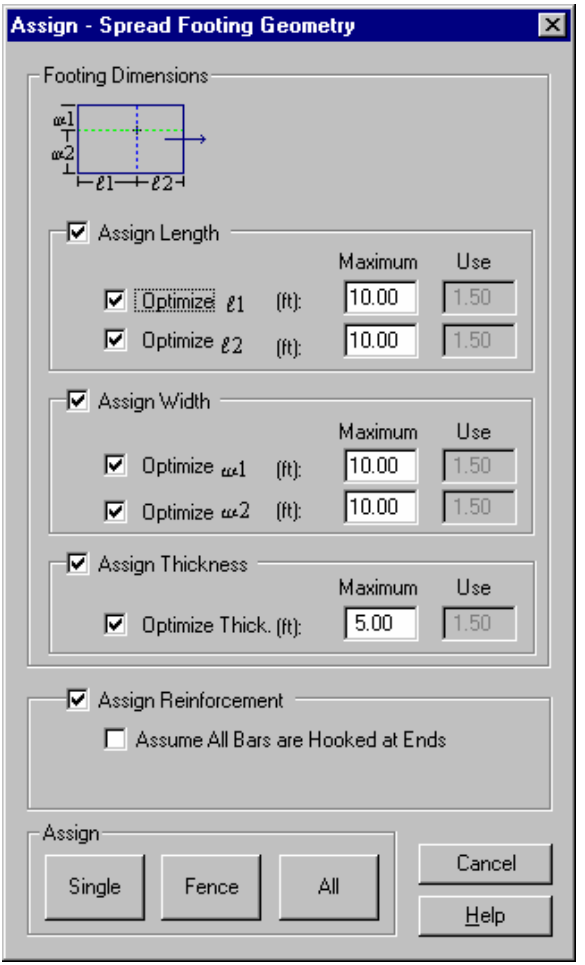
In our example the columns are mostly concrete columns with a specified size. For the gravity steel columns, base plates are already provided by the RAM Steel Column design. These sizes can be overridden with the Assign – Base Plate Size command. This size will only be used in RAM Foundation. It will NOT be exported back to RAM Steel Column Design. Additionally, once an override is made in the foundation module, this size will continue to be used by RAM Foundation even if the size is changed in RAM Steel Column Design. To update a base plate so that that the optimum base plate size is again obtained from the RAM Steel Column Design module, the "Clear User Defined Base Plate Size" option must be selected and assigned. Upon re-running the RAM Steel Column Design, the new optimized base plate size will automatically be assigned.

## *Assign Geometry*

The footing module allows you to assign any of the footing dimensions that you do not wish to have optimized. For spread footings, all of the dimensions may be optimized. Continuous footing design requires the user to assign at least the width.

To assign the geometry of the spread foundations:

- Select **Assign - Geometry - Spread**.

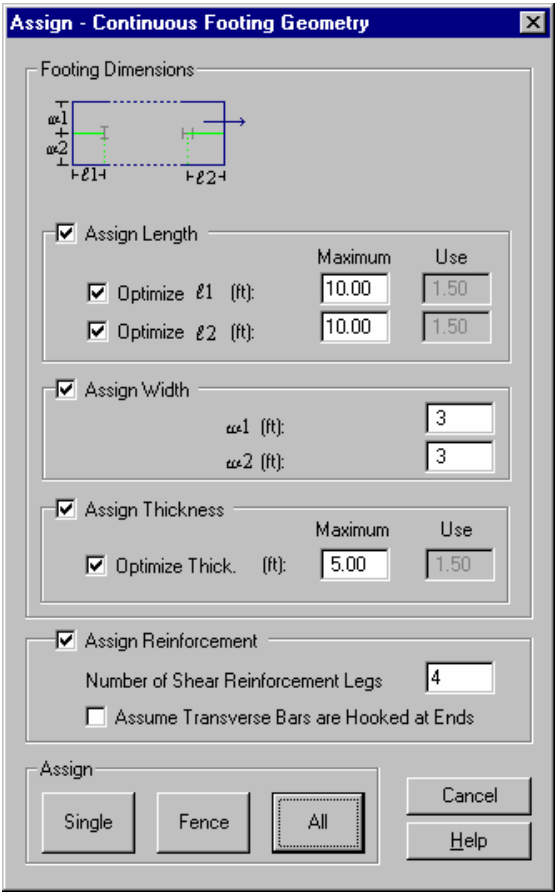


- In the Assign – Spread Footing Geometry dialog box that appears, Check the Assign boxes for all three dimensions.
- Check the Optimize box for all variables.
- Click **[All]**.

Note; the footing need not be centered under the column. Any of the length dimensions may be set or limited by the user and the eccentricity will be considered in the design.

To assign the geometry of the continuous foundations:

- Select **Assign – Geometry – Continuous**.



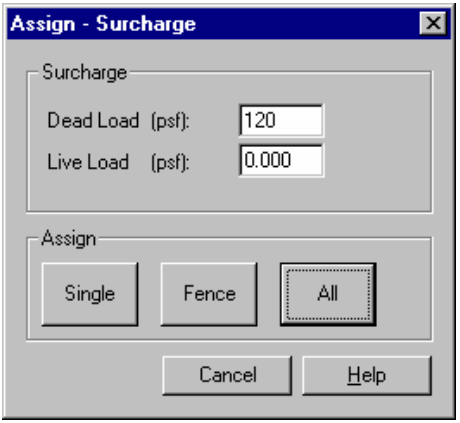
- Check the Optimize box for lengths and thickness.
- Type 3 (1) for both width dimensions.
- Type 4 for the Number of Shear Reinforcement Legs.
- Click **[All]**.

## Assign Surcharge

Since the footing is typically underground, the user can assign a surcharge load to be considered in the design. The surcharge will affect the soil check and the possibility of uplift, but does not enter into the design calculations for the foundation itself. The self weight of the foundation is automatically accounted for by the program.

To assign a surcharge:

- Select **Assign – Surcharge**.

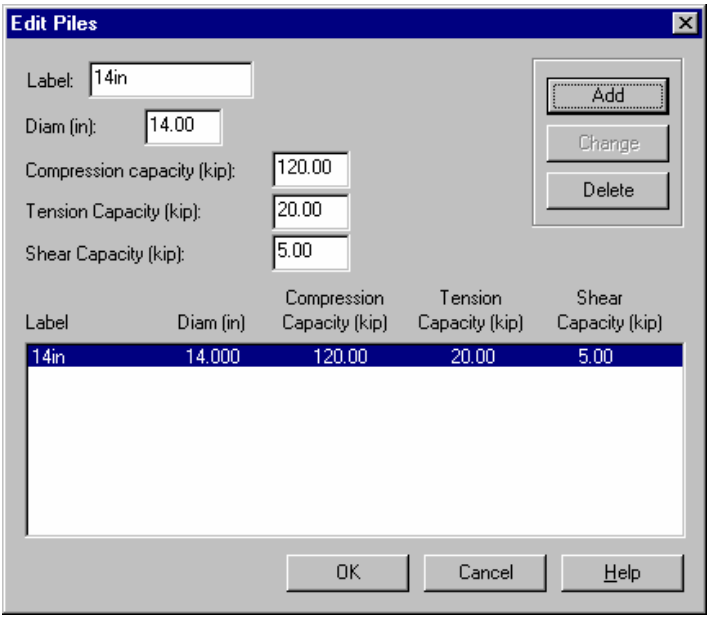



- Type 120 (6) for the Dead Load Surcharge.
- Click **[All]**.

## Assign Pile Geometry

In order for the program to check the pile design, the user must provide the unfactored capacity of the piles and the layout which should be used.

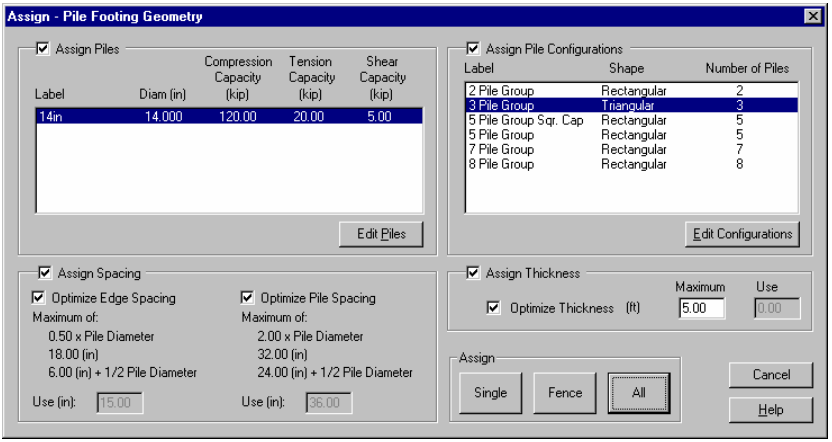
- Select **Assign - Edit Piles**.




- For the Label type `14in`.
- Set the Diam to 14 (300)
- Compression capacity to 120 (500)
- Tension Capacity to 20 (100)
- Shear capacity to 5 (25)
- Click **[Add]**.
- Click **[OK]**.

To assign the pile cap geometry:

- Select **Assign – Geometry – Pile Cap**.

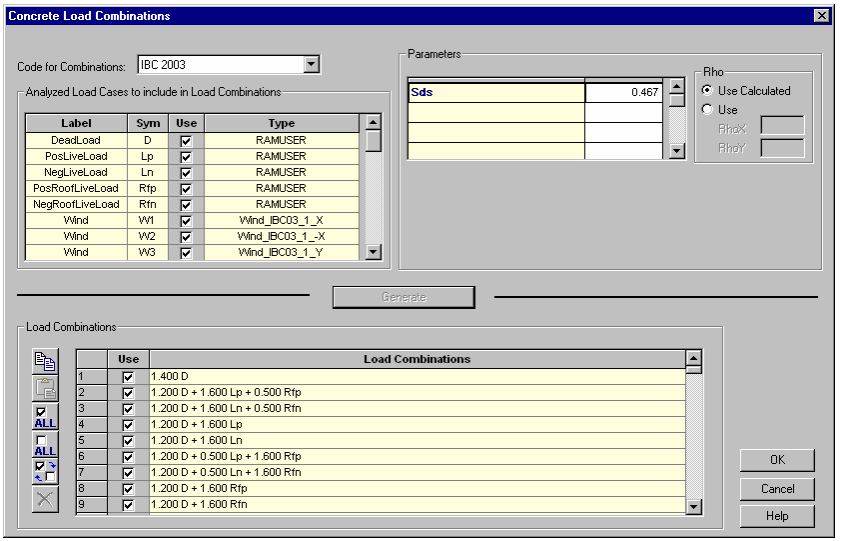



- Select the 14in pile on the left and the 3 pile Group on the right
- Leave the other options in the default position and click **[All]**.

## Load Combinations

Before any foundations can be defined, the user must specify the load combinations to consider. These may be created manually as User Defined Load Combinations or Code Generated Combinations can be utilized..

- Select **Combinations - Generate for Concrete**.




- Select IBC 2003 from the Code Combo list box.
- Type 0.432 for Sds.
- Set Rho to Use Calculated.
- Click **[Generate]**. The Load Combinations box should be filled with load combinations and each should be checked to Use.
- Click **[OK]**.
- Select Combinations – Generate for Soil.
- Select IBC 2003 from the Code Combo list box.
- Set Rho to Use Calculated.
- Click **[Generate]** and click **[OK]**.

## *Design All and View/Update*

The foundations should now appear yellow indicating that the foundations are all ready to be designed (with the exception of the mat foundation which can only be designed using RAM Conce*pt*). To design the foundations:

- Select **Process - Design - All Footings**.

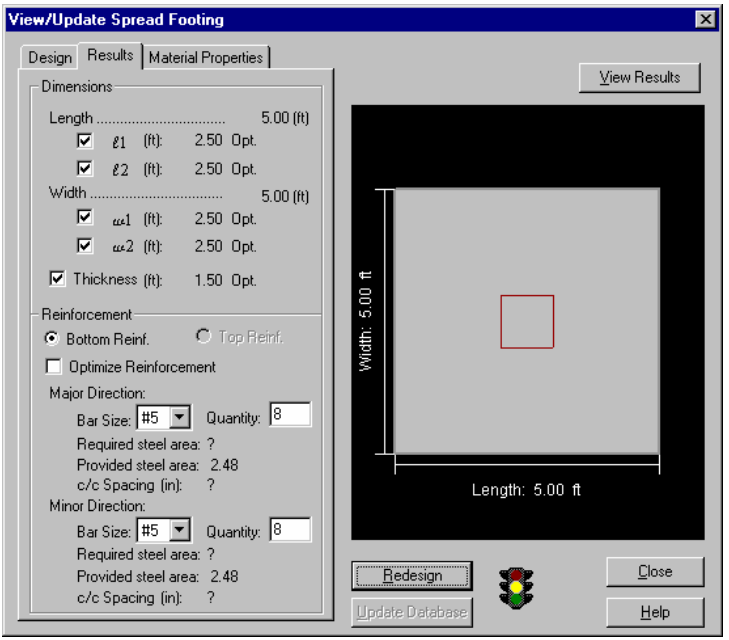
The program will calculate the optimum sizes for all of the footings. When finished, each should be drawn in green and sized appropriately. Any foundations that appear in red could not be successfully designed.

This may take a few minutes to complete. Keep in mind that the program is running an individual finite element analysis of each continuous foundation considering compression-only springs representing the soil below. The progress is indicated in the status bar in the lower left hand corner of the screen. For a complete explanation of how foundations are designed by the program refer to the RAM Foundation manual.

To investigate the design of individual spread footing:

- Select **Process - View/Update**.

With the target cursor, select the spread footing at Grid A - 4.



The dialog box will open to the Results Tab provided a successful design was accomplished. From here you can see the size of the footing and the reinforcement required. The signal light indicating the status of the design is shown. From here you can change the sizes or the reinforcement, redesign the footing and update the model if desired. Note; the orientation of the foundation in the view/update dialog box is adjusted to so that the footing length points to the right. This may not match the orientation on the plan.

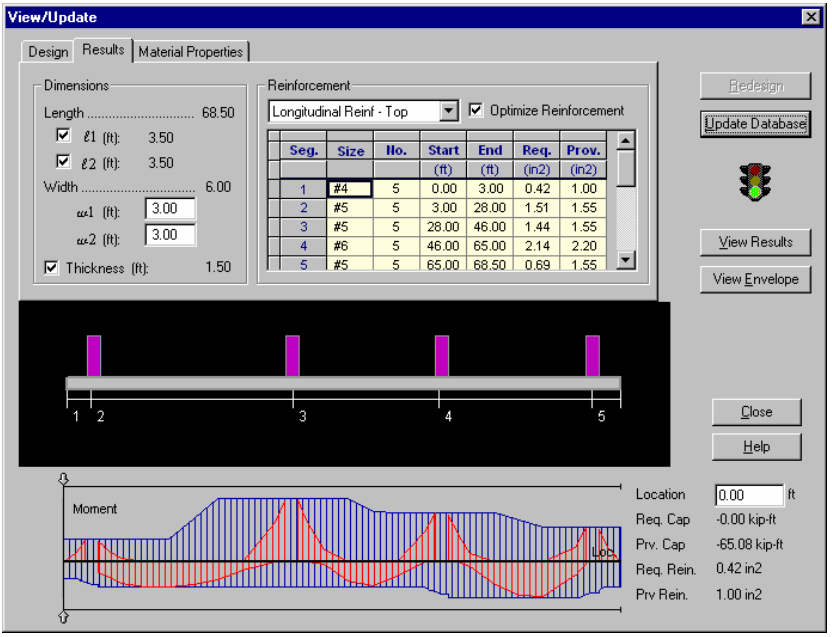
- Uncheck the box marked Optimize Reinforcement. New text boxes will appear allowing you to specify the reinforcement to use.
- Select #5 for Major and Minor Axis Bar Size. Notice that the provided steel area is immediately updated and it appears red since the quantity of bars is no longer adequate.
- Type 8 for Major and Minor Axis Bar Quantity.
- Click **[Redesign]**.
- Take a moment to become familiar with the contents of the other two Tabs: Design and Material Properties. The Design tab summarizes the current design. When there is a problem with the foundation design this page will indicate why the footing could not be designed. There is no need to change any of the Material Properties at this time.
- Click **[View Results]**.
- The Spread Footing Design Report should appear. Read through the results.
- Close the report and return to View/Update.
- Click **[Update Database]**.
- Click **[Close]**.

The foundation will now appear blue since it is user assigned. Note that the arrow should still appear green. This indicates that the design is still satisfactory. If something should change later causing the foundation to fail, the arrow will change to red.

To investigate the design of individual continuous foundations:

- Select **Process - View/Update**.

- With the target cursor, select the continuous footing along Grid 1.




The dialog box will open to the Results Tab provided a successful design was accomplished. From here you can see the size of the footing and the reinforcement required. The signal light indicated the status of the design as shown. In this case you have optimized for the length of the footing beyond the last column as well as the thickness. Furthermore, the top and bottom reinforcement has been selected. The reinforcement box shows what reinforcing bars have been selected. It defaults to the Longitudinal Top bars.

To change the selection:

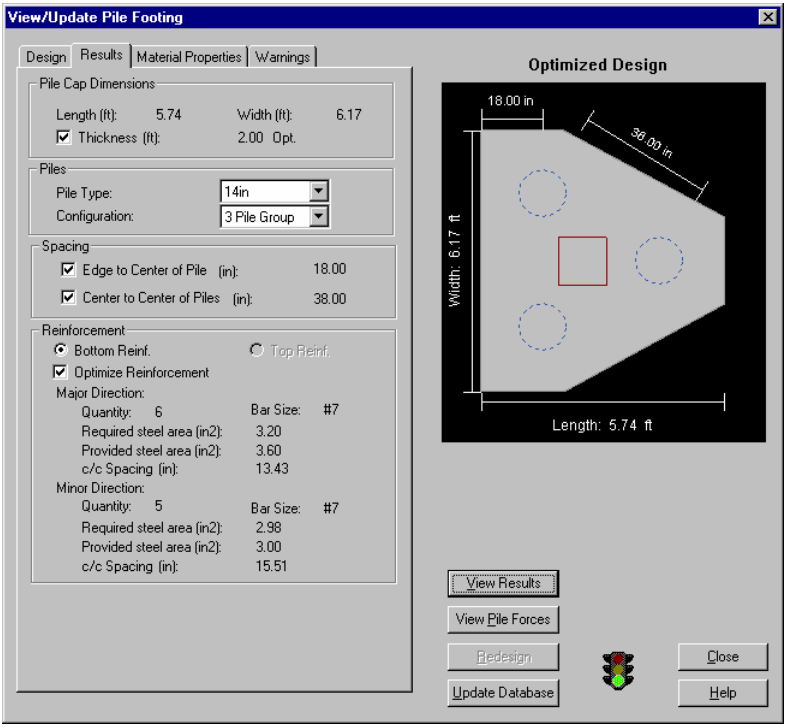
- Select Longitudinal Reinf – Bottom from the drop down list. The data in the box should be updated for bottom bars. You can also view the transverse and shear reinforcement in this way.
- If you wish to change the bar selection – uncheck the Optimize Reinforcement box. Then you can change the numbers in the table directly.
- When finished Click **[Redesign]** to have the foundation checked.

Take a moment to become familiar with the contents of the other two Tabs: Design and Material Properties.

- Click **[View Results]** to see a complete design report for the foundation.
- Close the report when finished and click **[Close]** to exit the View/Update dialog box without making any changes.

To investigate the design of individual continuous foundations:

- Select **Process - View/Update**.
- With the target cursor, select one of the pile cap foundations.




As with the other foundations, the pile cap thickness or rebar may be altered through the view/update command.

- To see the design results, select **[View Results]**.
- To see the maximum individual pile forces click **[View Pile Forces]**.
- Click **[Close]** when finished.

## *Reports*

Many different reports can be generated from the Foundation Module. The printed output is mostly generated using the Reports menu.

To print the service loads for a foundation design:

- Select **Reports – Screen**.
- Select **Reports – Foundation Loads – Single**.

Click the same continuous footing along Grid 1. The Foundations Load report should appear. Notice that the various elements that are supported by the foundation are listed separately. For an explanation of sign convention see the Foundation manual.

- Close the report when finished.

To print a summary of the spread footing designs:

- Select **Reports – Spread Footing Design Summary**.
- Close the report.

For a continuous foundation design report:

- Select **Reports – Continuous Footing – Single**.
- Select one of the continuous foundations.
- Close the report.

The foundation envelope report is also available from the reports menu. This report lists the maximum design forces and soil stresses along the length of the foundation. This is the same report that you see when you select [View Envelope] from the View/Update dialog box.

Take time now to review the various reports available in the Reports menu.

*Thank you for taking time to complete this tutorial for the RAM Structural System. Refer to the various program manuals for additional information on any aspect of the program that you do not fully understand.*