RISAFoundation

Rapid Interactive Structural Analysis – Foundations

Version 15- General Reference

RISA Tech, Inc.

27442 Portola Pkwy, Suite 200

Foothill Ranch, California 92610

(949) 951-5815

(949) 951-5848 (FAX)

risa.com

Table of Contents

Before You Begin

Welcome to the RISAFoundation General Reference manual. Please read this topic prior to installing the program and pay particular attention to the License Agreement. If you are a first time user of RISAFoundation you should turn your attention to the Tutorials (a separate document) which are designed to get you up and running as fast as possible while still exposing you to the important features of the software.

The Tutorials are designed to be read in two ways. If you are already familiar with structural modeling in general you can skip the supporting text and read only the underlined action items to quickly move through the tutorial. If you want more thorough explanations of the modeling process you may read all or some of the supporting text as you see fit.

After you have gone through the Tutorials, use this General Reference for detailed information on any topic. The topics are arranged in alphabetical order and are thoroughly indexed.

Overview

RISAFoundation is a foundation analysis and design program. It has been developed to make the definition, solution and modification of foundation models as fast and easy as possible. Analysis, up to and including calculation of maximum deflections and stresses, may be done on mat slabs, spread footings, and grade beams. Complete reinforcing detailing is also included in the program.

RISAFoundation has full graphical modeling capability. You can draw your model on the screen and also perform extensive graphical editing simultaneously in multiple views. To modify your model data directly, RISAFoundation employs a powerful, proprietary spreadsheet. All this combined with flexible data generation algorithms makes modeling very easy. Graphic display of the model along with applied loads, boundary conditions and much more, is always available. The model can be rapidly edited, solved, viewed, modified, re-solved, etc. The truly interactive nature and design output of RISAFoundation is its primary strength.

You may access the features in RISAFoundation by using the menu system, or the toolbars. The best way to learn RISAFoundation is to go through the Tutorials. The advantage to this is that you are exposed to the tools RISAFoundation provides and the ways that you can take advantage of them.

Minimum System Requirements

One of the following operating systems is required:

- Microsoft Windows 11 (64 bit only)
- Microsoft Windows 10 (64 bit only)
- Microsoft Windows 8.1 (64 bit only)

Internet (Subscription License Only)

An internet connection is required to launch the program. The internet connection must be maintained as long as the program is open, although brief internet outages (a few minutes) do not affect the user's ability to keep the program open.

Hardware

The following hardware is required:

- 1 GHz or faster processor (x86-64)
- 1920x1080 monitor resolution recommended (1024x768 minimum)
- 2 (or more) button mouse, mouse wheel recommended
- 8 GB of RAM
- 4 GB of hard disk space

Program Limits

Software Limitations

Points	500,000
Beams	10,000
Materials	500
Point Loads	250,000
Line Loads	10,000
Area Loads	10,000
Load Combinations	5,000
Slabs	250
Soil Regions	1000
Design Strips	2000
Point Supports (Footings, Piles)	7,500
Pile Definitions	800
Footing Definition	800
Pile cap definition	800
Pedestals	7,500

Demonstration Version: You may build and solve any model, however you may not save a model. There are also limits on some of the design values. For example, the allowable bearing stress and subgrade modulus for soil can not be adjusted.

Also, the Demonstration Version will automatically shut down if left open for 24 continuous hours.

License Agreement

For the full license agreement, please visit: risa.com/eula

Maintenance (Perpetual License Only)

Program maintenance provides all upgrades to RISAFoundation.

When your maintenance expires, you will be given the opportunity to continue program maintenance on an annual basis. You are under no obligation to continue program maintenance, of course, but if you decide to discontinue maintenance you will no longer receive RISAFoundation program upgrades and technical support. Customers who discontinue maintenance will need to purchase a subscription license for future program upgrades and/or technical support is required .

Complete program support is available to registered owners of RISAFoundation and is included in the purchase price. This support is provided for the life of the program. See Technical Support for a list of your support options.

The "life of the program" is defined as the time period for which that version of the program is the current version. In other words, whenever a new version of RISAFoundation is released, the life of the previous version is considered to be ended.

RISA Tech, Inc. will provide technical support only for the current version of RISAFoundation.

Application Interface

The Tutorials (a separate document) contain a tutorial that leads you through the RISAFoundation interface with an actual model. Consider going through the tutorial if you have not done so already, as it is the fastest way to learn the program. Although it requires some time up front, the tutorial will save you time and brainpower in the long run.

The features that are available to you in RISAFoundation may be accessed through the main menu, shortcut menus, toolbars and shortcut keystrokes. You may use any or all of these vehicles to interact with the software. The main menu has the advantage of containing all of the program options and features and may initially be the simplest to use, letting you learn just one system. The toolbars contain more common options and invoke with one click. The shortcut menus present options relevant to the task at hand. The shortcut keys provide a fast way to access features should you use the program often enough to make them familiar to you. All of these features are discussed in the sections below. There are many ways to access features and the method that you will use will simply be a matter of personal preference. The good news is that you have the options.

The bar along the top of the screen is called the title bar and contains the name of the file that is currently open. The three buttons ▬|🗗|✕ on the far right side of the title bar are used to control the main window. The left button will shrink the main application window to a button on the taskbar. The middle button will shrink or maximize the window on your screen. The right button will close the window, prompting you to save changes if necessary. You will also see these buttons in other windows and they have basically the same functions there as well.

The actual work that you do will be in the main area on the screen, which is called the workspace. When you open a model view, a spreadsheet or a dialog it will be opened in the workspace and listed in the **Window** menu. You may have as many windows open as you like.

Main Menu

All of the program features may be accessed through the main menu system at the top of the screen beginning with **File** on the far left and ending with **Help** or possibly **Director** on the far right. Clicking on each of these menus (listed below) will display sub-menus that contain options that you may choose from. You may also select the main menus by using the ALT key along with the underlined letter in the menu you wish to choose. You may then continue to use the keyboard to choose from the menu options. In addition, some of the menu options will have hot key combinations listed to the right of the option. These hot keys allow you to use the keyboard to access features without using the menu system.

File Menu

New will close the current file, prompting for saving if necessary, and will open a new file.

Open will close the current file, prompting for saving if necessary, and will open an existing file.

Save will save the current file, prompting for a name if necessary.

Save As will save the current file, prompting for a name.

DXF Export will export the current file to a DXF file.

Print will access RISAFoundation printing options.

Page Setup will present page setup options for printing.

Recent Files The five most recent files will be listed at the bottom of the menu. Selecting one of these files will close the current file, prompting for saving if necessary, and will open the selected file.

Exit will close RISAFoundation, prompting for saving if necessary.

Edit Menu

Undo will undo the last edit that was applied to the model whether it was made graphically or in the spreadsheets. You may continue to apply Undo to remove up to 100 model edits.

Redo will reverse the last undo that was applied to the model. You may continue to apply Redo to remove up to 100 undo operations.

Select All will highlight an entire spreadsheet all at once.

Copy will copy the selected spreadsheet cells or model view from the active window to the clipboard.

Paste will paste data from the clipboard to the spreadsheet cells.

Insert Line will insert a new line in the spreadsheet beneath the current line.

Delete Line will delete the current spreadsheet line.

Repeat Line will insert a new line in the spreadsheet beneath the current line and copy the data from the current line.

Mark All Lines will select all of the lines in the spreadsheet.

Unmark Lines will unmark any currently marked lines.

Delete Marked Lines will delete the marked lines in the spreadsheet.

Find will locate an item on the spreadsheet by its label.

Sort will sort the column containing the active cell.

Fill Block will fill the marked block of cells with a valid entry.

Math on Block allows you to add, subtract, multiply or divide the values in the marked block of cells.

Settings

Settings opens the Model Settings.

Units

Units opens the Units settings.

View Menu

New Plan View will open a new model view window.

Save or Recall Views allows you to save a view or recall a view that has previously been saved.

Clone View makes a copy of the current view so you can modify one and maintain the other.

Refresh All will refresh all of the windows that are open in the workspace.

Select provides graphic select options that are also provided on the **Selection Toolbar**.

Unselect provides graphic unselect options that are also provided on the **Selection Toolbar**.

Save or Recall Selection States allows you to save a selection or recall a selection that has previously been saved.

Zoom provides options for zooming in and out of the current model view.

Rotate provides options to snap the model view to global planes or an isometric view.

Model Display Options opens the Model Display Options.

Render will turn rendering of the current model view on or off, depending on the current setting.

Drawing Grid will turn the display of the Drawing Grid on or off, depending on the current setting.

Project Grid will turn the display of the Project Grid on or off, depending on the current setting.

Axes will turn the display of the global axes in the model view on or off, depending on the current setting.

Loads will turn the display of the model loads on or off, depending on the current setting.

Point Labels will turn the display of the point labels on or off, depending on the current setting. A third setting is also available where the points themselves are not shown at all.

Beam Labels will turn the display of the beam labels on or off, depending on the current setting. However, if rendering is turned on, beam labels will not be visible in the model view.

Insert Menu

The **Insert Menu** will help you insert new items into the model. Most of the options will provide a graphical method of insertion but some will open spreadsheets where appropriate. See Graphic Editing for specific information.

Modify Menu

The **Modify Menu** will help you modify existing items in the model. Most of the options will provide a graphical method of modification but some will open spreadsheets where appropriate. The **Delete Items Dialog** may also be accessed via this menu. See Graphic Editing for specific information.

Spreadsheets Menu

The **Spreadsheets Menu** provides access to any of the input spreadsheets. See Spreadsheet Operations to learn how to work within the spreadsheets.

Solve Menu

Clicking on the **Solve Menu** will immediately begin a solution to the model. See Solution for more information.

Results Menu

The Results Menu provides access to any of the results spreadsheets. See Results Spreadsheets for more information.

Tools Menu

Delete All Unattached Points will delete any points that are not connected to foundation elements or loads.

Application Settings contain settings that let you customize the program. See Customizing RISA for more information.

Customize Toolbar allows you to modify the model view toolbar by adding, subtracting and re-ordering buttons. See the customizable toolbar section.

Reset All Program Defaults will reset all customized settings to the original factory settings.

Window Menu

In order to help you work with the model and the results, you are provided with many window arrangements to choose from. You may access them from the **Window Menu**. The best way to understand just what these 'tilings' do is to try them. Remember that once you choose a tiling you may adjust any of the windows as you wish.

Help Menu

Help Topics opens the RISAFoundation Help File so that you may search the contents and the index. See Help Options to learn about getting help.

Check for Update runs an internal check for possible program updates. If your program is up to date, you will receive a message saying you are up to date. If you are out of date, the check will offer you the option to email RISA Tech, Inc. for upgrade information if you are out of date for a major update. If you are out of date just a minor update, then we will send you to our website to upgrade.. This check is also offered during the installation process.

Licensing gives information about your subscription license and the option to Borrow or Return a license from your subscription.

About provides RISAFoundation version and hardware key information.

Director Menu

File Edit Global Units View Insert Modify Spreadsheets Solve Results Tools Window Help   Director

If you are working from within the RISA Building System (RBS), use this menu to switch between RISAFloor, RISA-3D and RISAFoundation. If you are not working within the RISA Building System, the **Director Menu** will not be shown.

The directory button is located at the far, far right hand side of the Main Menu Toolbar as shown in the image above.

Shortcut Menu

The **Shortcut Menu** is also referred to as the **Right-Click Menu**. This is because to access the shortcut menu you simply click the RIGHT mouse button where you are working to see options that are relevant to what you are doing. For example if you are working in a model view the right click menu will provide options to help you modify the view and edit the model graphically. If you are working in a spreadsheet the menu will provide editing tools for that spreadsheet.

This menu will appear wherever you RIGHT click the mouse. This way you do not need to move away from where you are working to select the features you want to use.

Toolbars

The **Toolbars** provide buttons to help you access common commands and popular options in the menu system discussed above. There are different toolbars that will appear as you work to build your model and browse your results. If at any time you are not sure what a particular button does, simply let your mouse hover over the button and a helpful tip will pop up and explain the button.

RISA Toolbar

The first horizontal toolbar located just below the Main Menu is called the **RISA Toolbar.** The buttons on this bar facilitate file and window access. You may use these buttons to open files and windows and also to analyze the model.

Window Toolbar

The **Window Toolbar** is the second horizontal toolbar located below the Main Menu. It gets its name because the buttons change as you move from window to window in order to help you with what you are currently doing. When

you are working in a model view the buttons provide viewing tools, such as **Rotate** and **Zoom**, to assist you with that view. There are also many other results and information display toggles, including some icons with the drop down arrow next to them. Clicking the arrow will show you the different view options for that icon. Clicking the icon itself will bring you back to the default view. Note that this model view toolbar is now fully customizable. See below for more information.

Other model view windows that are open will not be affected so that each may show different information. When you are working in a spreadsheet, editing tools are provided that are appropriate to that particular spreadsheet. Note that not all tools are available with all spreadsheets. In fact there are many tools that are provided for one spreadsheet only. See Spreadsheet Operations for more information.

Note:

- ![button] - This button will allow you to take a snapshot of the current detail report you are viewing so that it can be added to a report. View the Printing topic for more information.

Customizable Model View Toolbar

The model view toolbar is fully customizable. By creating your personalized toolbar, you can quickly access your most frequently used buttons. This can be done quickly and easily in just a few steps.

1. Go to Tools menu and select Customize Toolbar.
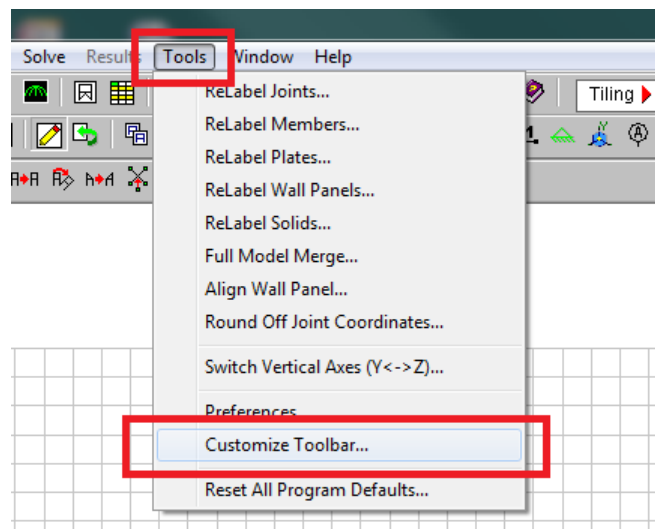
2. Select one of the toolbars by clicking in the box Available toolbar buttons, and click on Add to place them on the current toolbar.
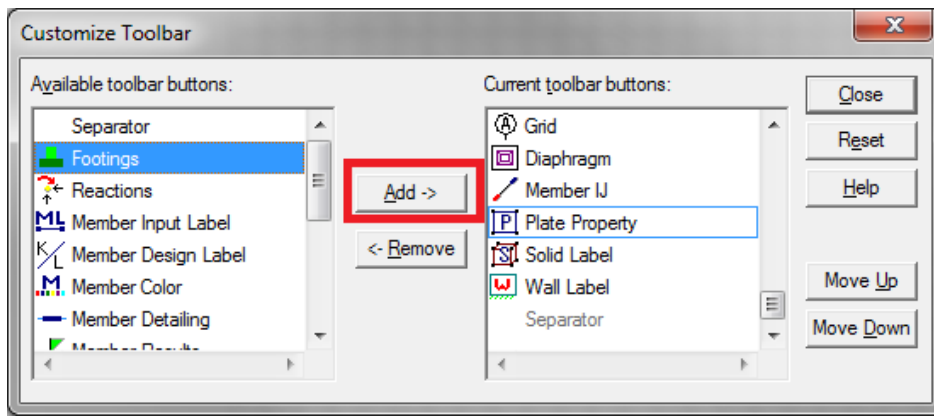
3. Once you've moved the buttons to the Current Toolbar, you can rearrange them by clicking on Move Up or Move Down.
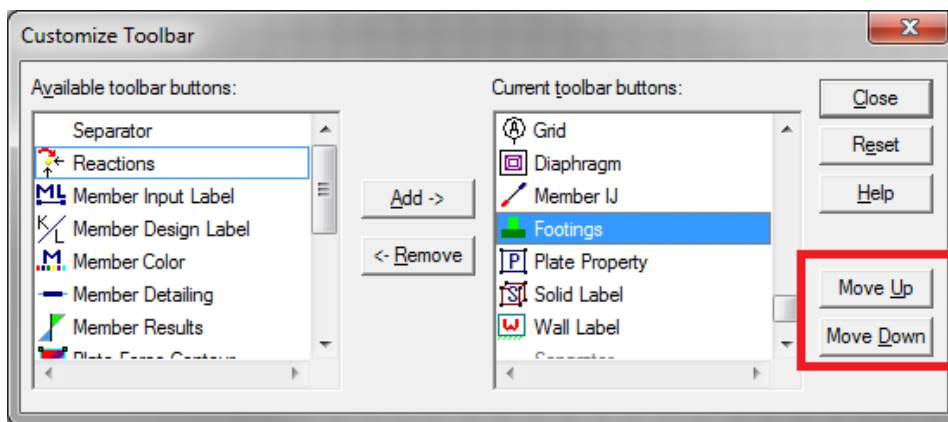
4. Click Close and you will see your selections on the model view toolbar.

Note:

- You must have a model view as the current view to see this toolbar.
- If you add more buttons than will fit on the toolbar the buttons that are at the end of the "Current toolbar buttons" will be cut off.
- The changes you have made will automatically be saved on a per-user (Windows User) basis, such that next time you open the program the toolbar will be arranged per your Application Settings. These saved changes are saved in the registry in the HKEY_CURRENT_USER\Software\RISA Technologies*RISAProgram* location with a String Value called "ToolbarState".

Selection Toolbar

The vertical toolbar on the left side of the screen is the **Selection Toolbar.** This toolbar will only be available when the active window is a model view. The buttons on this toolbar help you select and unselect items in the model in order to help you build and modify the model or view results. See Graphic Selection for more information.

Drawing Toolbar

Another toolbar that is available is the **Drawing Toolbar**. Unlike those mentioned above, this toolbar is located in the model view windows rather than in the main application window. This way the drawing tools stay close to where you are working. This toolbar controls modeling features that help you draw, load, and modify your model graphically. You may have more than one view open and a Drawing Toolbar for each view.

The **Drawing Toolbar** may be displayed in any model view window by clicking ⬚ on the **Window Toolbar** while in the model view window. Some of the buttons on the toolbar are for one-time applications such as modifying the drawing grid. Other buttons place you in an editing mode, such as Draw Members, that remains active until you cancel it. The current mode is indicated by the mouse pointer and by the state of the button. While in an editing mode the button will stay down until you click it again or choose another button. See Graphic Editing for more information.

This brings us to an important point. Some of the toolbar buttons remain down when you press them to indicate that you are in a certain mode or that something is either on or off. For example the **Box Zoom** ⬚ button will stay down to indicate that you are currently in the zooming mode. The **Show Drawing Toolbar** ⬚ button will remain down when you turn on this toolbar for the active window. You may be in more than one mode at the same time as long as they are not mutually exclusive.
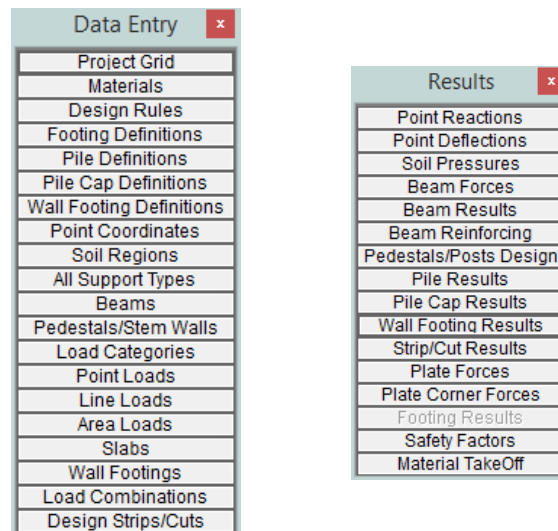
The **Data Entry Toolbar** is the vertical toolbar on the right side of the application window. It contains buttons that facilitate data entry through the spreadsheets. The buttons on this toolbar provide quick access to the spreadsheets that are also listed in the **Spreadsheets Menu**. You may open and close the toolbar by clicking the button on the **RISA Toolbar.**

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Data Entry**.

The **Results Toolbar** is the vertical toolbar on the right side of the application window that is placed over the **Data Entry Toolbar** after the model has been solved. The buttons on this toolbar provide quick access to the results spreadsheets that are also listed in the **Results Menu**. You may open and close the toolbar by clicking the button on the **RISA Toolbar.**

Dynamic View Controls

When your current window is a graphical model view, you can use the mouse wheel to dynamically zoom, pan, or rotate the graphical image.

Mouse Action	Model View Function
Rolling the Wheel Forward	Zoom In
Rolling the Wheel Backward	Zoom Out
Clicking and holding the Wheel Button	Grab the image and pan in the direction of mouse movement
Click and hold the Wheel button while pressing the Shift key	Dynamically rotate the structure in the direction of mouse movement

Dynamic Pan: Clicking and holding the mouse wheel button triggers the tool and allows the user to pan or drag the view to the limit of scroll bars.

Dynamic Zoom: This tool uses the wheel button on the mouse. Rotating forward zooms in and rotating backward zooms out.

Dynamic Rotate: This tool is triggered by clicking and holding the mouse wheel button while holding the Shift key. The rotational movement will be based on the how the user drags the mouse cursor over the screen and the projection of global axis on the screen. For rotation about X axis, drag the cursor perpendicular to the projection of the global X axis. The same logic applies for Y or Z axis rotations. When rotation is initiated, the system locks for rotation about that axis until the user releases the middle mouse button.

Zoom Previous/Next: Function keys F3 and F4 are associated with Zoom Previous and Zoom Next respectively. The system holds a doubly linked list of zoom info. This list has 10 zoom-states in the list. The F3 or F4 keystroke moves the active pointer forward or backward on the list. Each window has its own zoom list.

Dynamic Distance Tool: This tool triggers by pressing the F5 key. The user has to pick up two points on the screen and the system gives back the total and partial distance between points on the status bar.

Shortcut Keys and Hot Keys

Shortcut Keys and **Hot Keys** allow you to use the keyboard to quickly access features. The difference between the two is simply that the shortcut keys are related to a specific window and will only work in that window while the hot keys will perform at most any time.

General Hot Keys

Key Combination	Function
F1	Help on the active window
F5	Activates the Dynamic Distance Tool
Ctrl-F1	Main Help topics
Ctrl-F2	Create New view
F7, Ctrl-F7	Opens solution choices
Ctrl-C	Copy to the clipboard
Ctrl-V	Paste from clipboard
Ctrl-N	Start a new file
Ctrl-O	Open an existing file
Ctrl-S	Save the current file
Ctrl-P	Print
Ctrl-Z	Undo
Alt-	Access the menus by combining the Alt key with the underlined letter in the menu

Shortcut Keys available for Specific Windows

Key Combination	Model View Window	Spreadsheet
Ctrl-D	Open last graphic editing dialog	Delete Marked Lines
Ctrl-G	Toggle Drawing Toolbar	
Ctrl-A	Select All	Select All
Ctrl-U	Unselect all	
Ctrl-F		Block Fill
Ctrl-M		Block Math
Ctrl-I	Invert Selection	
Ctrl-L	Toggle Lock unselected	Unmark lines

Key Combination	Model View Window	Spreadsheet
Ctrl-Enter		Press cell ▶ button
F2	Open Model Display Options	Start/Stop Cell Edit
F3		Insert line
F4		Delete Line
F5	Initiates the "Distance" tool	Find
F8		Repeat Current Line
+	Zoom In	
-	Zoom Out	
←→↑↓ PgUp PgDwn	Scrolling	Scrolling

Spreadsheet Hot Keys that open spreadsheets

Key Combination	Unsolved Model	Solved Model
Ctrl-Alt-C	Point Coordinates	Corner Forces
Ctrl-Alt-D	Line Loads	Joint Deflections
Ctrl-Alt-G	Model Settings	
Ctrl-Alt-K		Solid Stresses
Ctrl-Alt-L	Load Combinations	Plate Forces
Ctrl-Alt-P	Point Loads	
Ctrl-Alt-R	Design Rules	Reactions

Status Bar

The **Status Bar** passes useful information to you as you work. It is divided into four parts located along the very bottom of the main application window, just beneath the workspace.

The left side of the status bar shows a solution flag to indicate the solved state of the model as follows:

Solution Type	Unsolved	Solved
Static	S	𝒮

To the right of the solution flags there are three message boxes.

The first and largest box lets you know what you are currently doing. If you are in a spreadsheet, this box will contain the explanation of the current cell. If you are working in a model view and select a graphic editing option, look to this box for information on how to use the feature.

The second box is used to pass you units of the current spreadsheet cell.

The third box indicates the coordinates of the mouse when a model view is active. The mouse coordinates that are displayed are the coordinates of the grid point or joint that is nearest to the mouse.

Windows

Modeling the structure will take place within model views and spreadsheets, each in their own window that may be moved around the workspace and sized as you wish. The ability to have multiple model views and multiple

spreadsheets open at one time is a powerful feature. The options in the **Window Menu** are provided to help you manage these windows.

These windows contain three buttons in the upper right corner to help you minimize, maximize and close the window, respectively. There are also scroll boxes to help you view information that is outside of the window viewing area. Click the scroll bar buttons or drag the scroll box to advance the display in one direction or another.

Model Views

Model View windows show a graphic view of the model. Open a new view with the button.

You may open as many model view windows as you like. This is especially helpful when working in close on large models. You might have one overall view and a few views zoomed in and rotated to where you are currently working. You may also have different information plotted in multiple views.

One thing to remember is that the toolbars that are displayed depends upon what window is active. The active window is the one with the blue title bar. For example, if you are looking for the zoom toolbar button and the active window is a spreadsheet you need to select a model view first before you can access the zooming tools.

Spreadsheets

Spreadsheet windows are made up of rows and columns of data cells. If you wish to add or edit data in a spreadsheet cell you click on the cell, making it the active cell, and then edit the cell. This active cell is simply the green cell that moves around the spreadsheet as you hit the cursor keys (← , →), Page Up, Page Down, Home, End, etc. There is always one and only one active cell, which is the cell that has the "attention" of the keyboard.

You may also select blocks of data to work on. You can select a block of data by clicking and holding the mouse button on the first cell in the block and then dragging the mouse to the opposite corner of the block and releasing the mouse.

Dialogs

A **Dialog** is a third type of window and is used to access a specific function within the program. Another powerful feature is that most of the dialogs may be left open while you edit the model, making it easy to make adjustments as you work. You will find that dialogs are very easy to work with. There are **Help** buttons that will bring you directly to the relevant topic in the help file.

Modes

There are three basic program modes (**View, Select,** and **Edit**) and a mode hierarchy to allow you to move between them easily. While you are **editing** the model you may **select** items to edit. When you are finished **selecting** you will be returned to **editing**. Likewise, while you are **selecting** items you can adjust the **view** and then be returned to **selecting**.

Different mouse cursors are used with each mode to make it clear what the current mode is.

View Mode is the upper level mode that allows you to adjust the view by zooming in and out, rotating and setting Model Display Options. This mode supersedes all other modes so that you may do these things at any time, and then be returned to the previous mode. This mode does not cancel other modes so that when you are finished adjusting the view you are returned to what you were doing. See Graphic Display for more information.

Select Mode is the middle level mode that allows you to make a graphic selection of joints, members and plates. This mode supersedes the **Edit Mode** but not the **View Mode**. This means that you can make a selection while in the middle of editing the view and when you are finished you are returned to the editing feature that you were using. It also means that you may adjust the view while remaining in the same **Select Mode**. See Graphic Selection for more information.

Edit Mode is the lower level mode that allows you to graphically edit the model. You may make selections and adjust the view while in the edit mode such that when you are finished selecting you will be returned to the **Edit Mode**. Some **Edit Mode** features have options on how you apply the edit. See Graphic Editing for more information.

Note

- The default mode is the mode you are in if you are not in any other mode and is indicated by the stand-ard ⏳ mouse cursor. The default mode is a selection mode where you can select/unselect individual items by clicking on them.
- You may use the ESC key or the right mouse button to cancel a mode.

Beams

RISAFoundation uses a [Physical Member](#) that is automatically sub-meshed into general-purpose beam elements. Beam data may be viewed and edited in two ways: graphically in a model view or in the **Beams Spreadsheet**. See [Beams Spreadsheet](#) for descriptions of the beam data.

Drawing Beams

To create new beams you can draw them using a [Drawing Grid](#), [Project Grid](#), or drawing "dot to dot" clicking existing points. You can set all of the beam properties up front or you can modify these properties after you draw the beam. Graphically modifying properties is discussed in the next section.
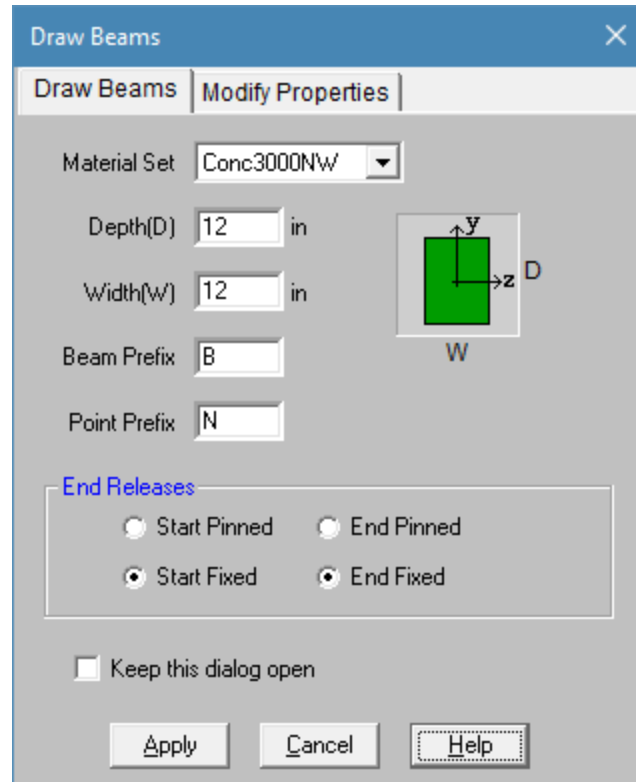
To Draw Beams

1. If there is not a model view already open then click ⊞ on the **RISA Toolbar** to open a new view and click ✎ to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you are not drawing between existing points, you will need to create a drawing grid or define points on the **Point Coordinates Spreadsheet**.
3. Click the ✎ button and set the beam properties.
4. Click **Apply** to start drawing beams by clicking on points or grids with the left mouse button. The coordinates of the closest point or grid intersection to your cursor are displayed in the lower right hand corner of the status bar.

   The first click will define the **Start Point** of the first beam. The second click, and each click thereafter, will define the **End Point** of the first member and also the **Start Point** of the next beam so that you may continue to draw as if your pencil is down. To "pick up" the pencil, click the right mouse button. You may then start drawing somewhere else with the left button.

   The new beams will be shown on screen and will be recorded in the **Beams Spreadsheet**.

5. To stop drawing altogether click the right mouse button or press the Esc key.

Note

- Press CTRL-D to quickly recall the last dialog accessed and make changes.
- You may also specify or edit beams in the **Beams Spreadsheet**.
- You may undo any mistakes by clicking the **Undo** button.

Modifying Beam Properties

There are a number of ways to modify beams. You may view and edit the beam data in the Beams Spreadsheet. You can use the **Modify Properties** tool to graphically modify a possibly large selection of beams.

The graphical **Modify Properties** tool discussed here lets you modify the properties of beams that already exist in your model. To use this, you will typically specify parameters you want changed, then select the beams that you want to modify. You can modify beams one at a time by selecting the **Apply by Clicking Individually** option and then click on the beams you wish to modify. You may also modify entire selections of beams by selecting the beams in the model view first and then use the **Apply to All Selected** option. See the Graphic Selection topic for more on selecting.

To Modify Beams

1. If there is not a model view already open then click ⬚ on the **RISA Toolbar** to open a new view and click ⬚ to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Draw/Modify Beams** ⬚ button and select the **Modify Properties** tab.
3. Set the parameters for the new beams. Check the **Use?** box for the items that are to be applied.
4. You may choose to modify a single beam at a time or an entire selection of beams.

    To modify a few beams, choose **Apply Entries by Clicking Items Individually**, click **Apply**, and click on the beams with the left mouse button.

    To modify a selection of beams, choose **Apply Entries to All Selected Items** and click **Apply**.

Note

- To modify more beams with different parameters, press CTRL-D to recall the **Modify Beams Dialog**.
- You may also modify beams in the **Beams Spreadsheet**.
- To relabel beams first sort the **Beams Spreadsheet** into the desired order, select the **Tools Menu** and choose **Relabel Beams**.
- You may undo any mistakes by clicking the **Undo** button.

The parameters shown are nearly the same as those used to define new beams.

The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected beams, if the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on beams without affecting all the rest of the properties. Note that if no value is entered in a field (I.e. the field is blank) and the corresponding check box is checked, clicking **Apply** will have the effect of clearing the data for these fields.

Beams Spreadsheet

The **Beams Spreadsheet** records the properties for the beam elements and may be accessed by selecting **Beams** on the **Spreadsheets Menu**.

The following data columns hold the data for the beams:

Beam Labels

You may assign a unique **Label** to any or all of the beams. You can then refer to the beam by its label. Each label has to be unique, so if you try to enter the same label more than once you will get an error message. You may relabel beams at any time with the **Relabel Beams** option on the **Right Click Menu**.

Beam Start and End Points

The **Start Point** and **End Point** entries define the start and end locations of the beams using point labels.

Shape

The **Shape** entry is used to set the beam size. To enter a shape, type the shape name directly or click the ▶ arrow in the cell to access the **Set Beam Size Dialog**. See [Beams - Database](#) for more information.

Material

The **Material** column displays the concrete material used for each beam. Available materials may be selected from the drop down list which contains all entries in the **Materials Spreadsheet**. See [Material Properties](#) for more information.

Design Rules

The **Design Rules** entry displays the selected design rule for each beam. Available design rules may be selected from the drop down list which contains all entries in the **Design Rules Spreadsheet**. See [Design Optimization](#) for more information.

Start and End Releases

The **Start** and **End Releases** allow you to set the member end releases to **Fixed** or **Pinned**. This will determine how the beam connects to other elements at the end nodes.

Flexural Layout

The **Flexural Layout** entry displays the selected custom rebar layout for beam flexural reinforcement. Available layouts may be selected from the drop down list which contains all **Bending Rebar Layouts** from the **Rebar Layout Database**. Select **Optimize** from the drop down list if you would like RISA to select the reinforcement in the beam. See [Rebar Layout Database](#) for more information.

Shear Layout

The **Shear Layout** entry displays the selected custom rebar layout for beam shear reinforcement. Available layouts may be selected from the drop down list which contains all **Shear Rebar Layouts** from the **Rebar Layout Database**. Select **Optimize** from the drop down list if you would like RISA to select the reinforcement in the beam. See [Rebar Layout Database](#) for more information.

Analysis Offset

This is a positive value that will move the beam downward in relation to any other element in the model. This not just a graphics change. The program is internally adjusting the location of the beam and connecting it to surrounding elements with rigid links. You can expect to see different distribution of forces due to the offset.

Icr Factor (Cracked Moment of Inertia Factor)

The **Icr Factor** is used to reduce the bending stiffness of beams. If this entry is left blank, the Icr factor will default to a value of 0.35.

Note

- The **Icr Factor** will be ignored if the **"Use Cracked Sections"** box is not checked on the **Solution** tab of the **(Global) Model Settings** dialog.

Service Level Stiffness

Due to cracking and material non-linearity, modeling the stiffness of concrete members is more complex than it is for steel or wood members.

For typical applications, ACI 318-14 Sesction 6.6.3.1 (ACI 318-11 Section 10.10.4) requires that member stiffness be reduced to account for the cracking that occurs when a member is subjected to ultimate level loads. As described in the previous section, RISA uses the **Icr Factor** to account for this stiffness reduction. However, for service level analysis, the level of cracking will be significantly less. Therefore, the stiffness used in your analysis should be representative of the reduced loading and reduced cracking. Per ACI 318-14 Section R6.6.3.1.1 (ACI 318-11 Section R10.10.4.1), the program will account for this increased stiffness by applying a factor of 1.43 to the cracked section properties for any load combination that has the **"Service"** flag checked in the **Load Combinations Spreadsheet.**

Physical Members

Physical Members provide fixity to all joints that occur along the length of the member, without breaking that member into multiple smaller members. You may use the physical member feature to avoid defining one 'field' member with multiple members in your model. This saves time in building and editing your model and in understanding your results.

Continuous beams are examples of continuous 'field' members that can be modeled with one Physical Member. You may define them from start to finish without having to explicitly define their connection to other elements or boundaries through intermediate points. Subsequently, when it comes time to make changes to a beam you can edit the properties of one Physical Member rather than the multiple beams that might otherwise represent it.

Physical Members are also effective in managing results because the results for one Physical Member are reported together in the results spreadsheets and the beam detail report.

Note

- You cannot convert multiple beams into one Physical Member. To achieve this you must delete the multiple beams and define a new Physical Member.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **What are Physical Members**.

The equation for calculating shear area varies for different shape types:

- For WT, LL, L, A_v =(depth*thickness)/1.2
- For channel, A_v =(depth*thickness)/1.0
- For rectangular tube, A_v =(2*depth*thickness)/1.0 for AISC 360-05 (13th edition) and AISC 360-10 (14th edition), but (2*(depth-3*web thickness^2))/1.0 for AISC 360-16 (15th edition).
- For round tube, A_v =A/2

Beams - Database

There is one type of concrete shape currently supported, Rectangular. If you're familiar with the shape definitions, you can type the name directly into the appropriate field. Alternately you may click the ▶ button to have the program generate the desired shape definition for you.
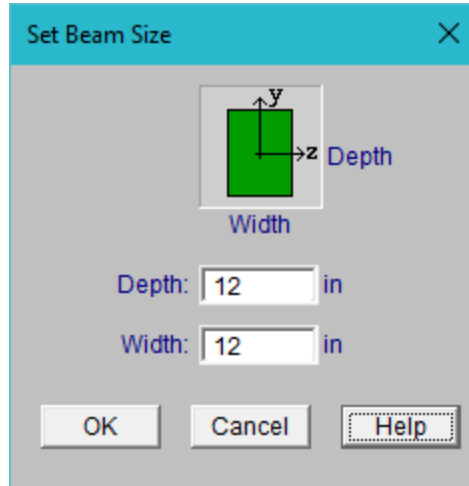
Rectangular Sections

Rectangular sections are defined using a parametric shape code since a rectangular shape may be any depth or width. The code is CRECT'depth'X'width', where 'depth' and 'width' are the values in the current dimension units. For example, if you wanted a beam that was 18" deep and 12" wide, you would enter "CRECT18X12". Note that the dimensions can also be decimal values like "18.25".

Custom Rebar Layout Database

Pressing the ⬚ button on the **RISA Toolbar** will open the database that is used for creating and storing custom rebar layouts. This allows the user to create multiple layers of bars and add in compression reinforcement or unusual bar arrangements.

These reinforcement layouts may be assigned to beam members in the same way as the other concrete design parameters are assigned. This can be done from the **Beams Spreadsheet**, **Design Strips Spreadsheet**, or **Design Cuts Spreadsheet**, or the **Modify Beams Dialog**.

These reinforcement layouts may also be assigned to pedestals in the **Pedestals/Stem Walls** spreadsheet or from the **Modify Pedestals** dialog.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Custom Rebar Layouts**.

Bending Rebar Layouts

Since beams are only designed for uniaxial bending, the only requirements for the beam layouts are that you specify the depth at which the bars are located and the size & number of the bars that are present at that depth. You can specify the depth with respect to the top surface of the beam or the bottom surface.

The **Start** and **End** locations dictate the location along the length of the beam where these bars will be present. You can use these entries to specify partial length bars that will only be present in locations with a higher moment demand. If the bar should be present for the entire length of the beam, the start location should be '0' and the end location should be '%100' as shown in the dialog above.

Note

- While the rebar layout sheet resembles one of RISA's spreadsheets in appearance it is NOT a spreadsheet and standard TAB controls will not work. Instead, the arrow keys or the new arrow buttons ◄ ► can be used to advance from cell to cell.

Shear Rebar Layouts

These rebar layouts are specified by giving the size and spacing of the bars as well as the location on the beam where that reinforcement will be present.

The **Start** and **End** locations dictate the location along the length of the beam where these bars will be present. You can use these entries to specify partial length reinforcement that will only be present in locations with a higher shear demand. If the reinforcement should be present for the entire length of the beam, the start location should be '0' and the end location should be '%100' as shown in the dialog above.

Note

- While the rebar layout sheet resembles one of RISA's spreadsheets in appearance it is NOT a spreadsheet and standard TAB controls will not work. Instead, the arrow keys or the new arrow buttons ◄ ► can be used to advance from cell to cell.

Rectangular Pedestal Rebar Layouts

Since pedestals are designed for biaxial bending, they require more information about the location and arrangement of the bars.

Normally, column bars are arranged in **layers**. One 'top' and one 'bottom' horizontal layer must **always** be defined, each containing at least two bars. These layers, as well as any additional horizontal layers, will be specified by entering a **x1** value to specify the depth from the top or bottom fiber to the center-line of the reinforcing steel. The number and size of the bars must then be entered. The **z1** and **z2** values dictate where the first and last bar in that layer are located. Additional bars in that layer will be placed so that they are evenly spaced in that layer.

Vertical layers can be specified by entering a **x1** value specifying the depth from the right or left most fiber to the center-line of the reinforcing steel. The number and size of the bars must then be entered. The **z1** and **z2** values are ignored for vertical layers because the bars will be assumed to be evenly spaced between the required top and bottom layers referred to previously.   If this is not desired, then the side bars should be entered individually as **custom single bars**.

Custom single bars are specified by their y and z coordinates measured from the local x and z-axis respectively. A positive x coordinate would place the bar closer to the top fiber and a negative y coordinate would place the bar closer to the bottom fiber (relative to the local axes of the pedestal). Similarly, a positive z coordinate would place the bar closer to the right side and a negative z coordinate would place the bar closer to the left side(relative to the local axes of the pedestal).

The **Start** and **End** locations dictate the location along the length of the member where these bars will be present. You can use these entries to specify partial length bars that will only be present in locations with a higher moment demand. If the bar should be present for the entire length of the member, the start location should be '0' and the end location should be '*%100*' as shown in the dialog above

Circular Pedestal Rebar Layouts

For circular pedestals, you may specify equally spaced concentric *rings* of bars at given depths, **z1**, measured from the exterior fiber of the column. You may also specify custom single bars.

Custom single bars are specified by their x and z coordinates measured from the local x and z axis respectively. A positive x coordinate would place the bar closer to the top fiber and a negative x coordinate would place the bar closer to the bottom fiber. Similarly, a positive z coordinate would place the bar closer to the right side and a negative z coordinate would place the bar closer to the left side.

The **Start** and **End** locations dictate the location along the length of the member where these bars will be present. You can use these entries to specify partial length bars that will only be present in locations with a higher moment demand. If the bar should be present for the entire length of the member, the start location should be '0' and the end location should be '*%100*' as shown in the dialog above.

Note

- While the rebar layout sheet resembles one of RISA's spreadsheets in appearance it is NOT a spreadsheet and standard TAB controls will not work. Instead, the arrow keys or the new arrow buttons ◄ ► can be used to advance from cell to cell.

Beams - Design

Beam design and optimization can be performed for rectangular concrete shapes based on the following codes:

- The 2019, 2014, 2011, 2008, 2005, 2002 and 1999 Editions of ACI 318
- The 1992 EuroCode (EC2) and the British publication of the 2014 and 2004 Eurocode (BSEN)
- The 1994 and 2004 Editions of the Canadian code (CSA A23.3)
- The 2004 Edition of the Mexican code (NTC-DF)

Note:

- Unless otherwise specified, all code references below are to ACI 318-14.
- Beams and Columns designed in RISA meet all of the requirements for Ordinary Moment Frames except for the additional requirements indicated in ACI 318-14 Section 18.3. Those provisions should be checked by hand outside of RISA.

The program will design the longitudinal and shear reinforcement for rectangular beams. These calculations encompass all the code requirements except those noted in the [Limitations](#) section of this document. The program also provides reinforcement detailing information for concrete beams in the beam detail reports.

To Apply a Concrete Design Code

1. On the **Design** tab of **Model Settings Dialog**, select the concrete code from the drop down list.
2. Click **Apply** or **OK**.

Beam Spans

RISAFoundation will automatically break a concrete physical member into spans based on the number of internal supports. Each internal point is NOT automatically treated as a support. Instead, we go through the whole model geometry to determine where a beam is supported. Note that for a physical member to see a support, there must be a point at that support point.

Beam members are supported by the following: Supports (Reaction, Spring, etc.) and other Beam Members that are supporting that member.

Note

- The program's ability to recognize spans is important because it will give you more relevant span to span information without overwhelming you with independent design results for each finite element segment that comprises your physical member.
- For continuous beam members, the program will evaluate the framing to determine which beams elements are supporting other beam elements so that only supporting members are treated as supports and not vice versa.

Parabolic vs. Rectangular Stress Blocks

You can specify whether you want your concrete design to be performed with a rectangular stress block, or with a more accurate parabolic stress block. While most hand calculations are performed using a rectangular stress block, the parabolic stress block is more accurate. In fact, most of the PCA design aids are based upon the parabolic stress distribution. A good reference on the parabolic stress block is the PCA Notes on ACI 318-99.

Concrete Strain   Parabolic Stress Distribution PCA Method   Rectangular Stress Block
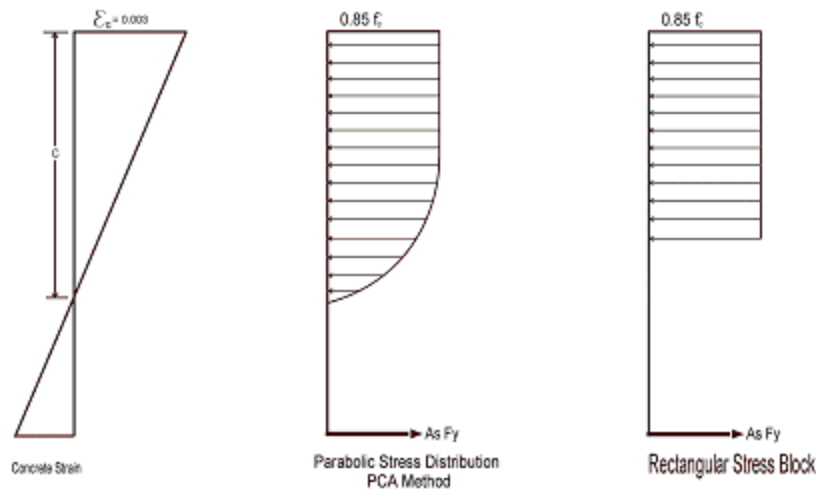
British Eurocode Design Parameters (BS EN 1992-1-1: 2004)

General

- f'_{ck} – Can not be more than 50 MPa (7252 psi) for normal strength concrete
- α_{cc} is assumed to be 1 (recommended value) : See 3.1.6
- Effective length of T and L: L_o=.7*span length and $b_{eff,i}$ =L_o/10
- $\varphi_{concrete}$ = 1.5
- φ_{rebar} = 1.15
- Maximum bar spacing for beams = 300 mm

Tension Development Length

- α_{ct}= 1 (assumed in Eq 3.16)

- η_1= η_2=1 (assumed in Eq 8.2 to calculate bond stress)

- λ_3, λ_5, λ_4=1 (assumed in Eq 8.4)

- C_d in Table 8.2 assumed to be 1 bar diameter rebar spacing

- Development length when hooks are provided uses same assumptions as BS 8110-1: 1997

Shear Capacity of Concrete

To compute the shear capacity of concrete the following recommended values are being used:

- $C_{Rd,c}$ =0.18/γ_c for Eq 6.2.a

- v_{min} =0.035 $k^{3/2}$ $f_{ck}^{1/2}$

- v =.6*[1- f_{ck} /250]

- V_{max} is calculated from Eq. 6.5.

- θ is assumed 45 degrees in Eq 6.8.

Limitations - General

Torsion – Beams ignore torsion with respect to the design of shear reinforcement. A message is shown in the detail report to remind you of this.

Creep / Long Term Deflections – No considerations are taken in the analysis to account for the effects of creep or long term deflections.

Cracking – The effects of concrete cracking are not considered in the Foundation analysis.

Beam Design – Beams are not designed for weak axis y-y bending, weak axis shear, or axial forces. A message is shown in the detail report to remind you of this. Beams currently do not consider any compression steel in the calculation of the moment capacity. Beam "skin reinforcement" per the requirements of ACI 318-14 Section 9.7.2.3 (ACI 318-11 Section) 10.6.7 for beams with "d" greater than 36" is currently not specified by the program. The provisions in ACI 318-14 Section 9.9 (ACI 318-11 Section 10.7) for deep beams are not considered.

Limitations - ACI

Shear Design –When ACI 318-19 is selected, the shear strength of concrete (Vc) uses equations in Table 22.5.5.1. Note that for members meeting the minimum shear reinforcement requirement (Av≥Av,min), Vc is taken as the larger of the results calculated by the equations (a) and (b) in the table. ACI 318-19 code suggests ρw may be taken as the sum of the areas of longitudinal bars located more than two-thirds of the overall member depth away from the extreme compression fiber. Therefore, RISA calculates ρw as the sum of the areas of longitudinal bars on the tension face.

When other ACI 318 editions are selected, the shear strength of the concrete alone is limited to the standard $2*\lambda*sqrt$ (f'c) equation from ACI 318-14 Section 22.5.5.1 (ACI 318-11 Section 11.2.1.1) and does not use the more detailed calculations of ACI 318-14 Table 22.5.5.1 (ACI 318-11 Section 11.2.2). Also, note that for members with significant axial tension (greater than 0) the program designs the shear reinforcement to carry the total shear per ACI 318-14 Section R22.5.7.1 (ACI 318-11 Section 11.2.1.3).

Deep Beam Design – The program does not design deep beams as defined in ACI 318-14 Section 9.9.1.1(a) (ACI 318-11 Section 10.7).

Limitations - Canadian Code

Concrete Stress Profile – Concrete stress strain curve (parabolic) is assumed same as PCA method for the Canadian codes.

Bi-Axial Bending - The program uses the simplified uniaxial solution provided in the Canadian specification rather than performing a complete biaxial condition.

Mid-Depth Flexural Strain for Shear Design - The program uses the code equation (per the General Method) to calculate ε_x with the moment and shear at the section taken from the envelope diagrams. The maximum ε_x for each span is conservatively assumed for the entire span. Currently the program has no option for pre-stressing, so V_p and A_p are both taken as zero.

Shear Design - The shear strength of concrete is calculated using β and θ, which are both calculated per the General Method (Clause 11.3.6.4 from the *2004 CSA A23.3*). S_{ze} is calculated per equation 11-10 and a_g is always assumed to equal 20 mm (maximum aggregate size).

Limitations - Saudi Code

Concrete Stress Profile – Concrete stress strain curve (parabolic) is assumed to be the same as the ACI code.

Shear Strength– The shear strength is based on 11.3.1.1 and does not include the more detailed provisions of section 11.3.1.2.

Yield Strength of Shear Ties - The yield strength of shear ties is not allowed to exceed 420MPa.

Shear Tie Spacing - Minimum spacing of shear ties is set to 50mm

Bi-Axial Bending – Both the Exact Integration and the PCA Load Contour methods for bi-axial bending are supported in the Saudi code.

Special Messages

In some instances code checks are not performed for a particular beam. A message is usually shown in the **Warning Log** and **Detail Report** explaining why the code check was not done. There are also instances where a code check is performed, but the results may be suspect as a provision of the design code was violated. In these cases, results are provided so that they can be examined to find the cause of the problem. Following are the messages that may be seen.

No Load Combinations for Concrete Design have been run.

All of the load combinations that were run had the **Service** box checked on the **Load Combinations Spreadsheet**. Since there are no concrete design specific load combinations, there are no results or force diagrams to show.

Warning: No design for spans with less than 5 sections.

Certain very short spans in physical members can end up with less than 5 design sections. No design is attempted without at least 5 sections because maximum values may be missed and an un-conservative design may result.

Warning: No design for spans less than 1 ft.

Certain very short spans in physical members can end up with lengths less than 1 foot. No design is attempted for these sections.

Warning: The shear tie spacing does not meet the code Minimum Requirement

This warning is stating that either minimum spacing or strength requirements are not being met for the shear reinforcement in the concrete member.

Rebar spacing exceeds maximum allowable (ACI 318-14 Section 13.3.4.4, ACI 318-11 Section 15.10.4)

The 2008 and newer editions of the ACI 318 forbid the placement of rebar in foundation slabs exceeding 18 inches. The program will still optimize rebar up to the largest spacing specified in the Design Rules spreadsheet, and may violate this ACI provision. Reduce maximum spacing in the Design Rules.

Beams - Design Results

You can access the **Beam Results Spreadsheets** by selecting the **Results Menu** and then selecting **Beams ▸ Beam Design** or **Beam Reinforcing**. Note that concrete results are always based on envelope results.

For beam flexural design, the required bars are based on the envelope moment diagrams. For beam shear steel design, the required bars are based on the enveloped shear force diagrams.

Beam Results

Beam results are shown in the three following spreadsheets: **Beam Design Results**, **Beam Bending Reinforcement**, and **Beam Shear Reinforcement**.

Beam Design Results Spreadsheet

The **Beam Design Results Spreadsheet** shows the governing maximum code check for the top and bottom of the beam for all spans.

	Beam	Shape	UC Max Top	Loc[ft]	LC	UC Max Bot	Loc[ft]	LC	Shear UC	Loc[ft]	LC	Phi*Mnz T...	Phi*Mnz B...	Phi*Vny[k]
1	B1	CRECT12X12	0	0	N/A	2.284	8.021	2	.867	8.021	2	0	55.564	46.83
2	B2	CRECT12X12	.101	11.958	3	3.539	7	2	1.526	7	2	34.825	55.564	46.83

These top and bottom code checks, **UC Max Top** and **UC Max Bot**, are based on the top/bottom moment capacities and maximum top/bottom moment. Currently the moment capacity is based only on the tension steel (NO compression steel is considered in the capacity calculation). The governing maximum shear check for all spans, **Shear UC**, is also shown. The capacities shown are only for the governing section. Capacities for each span, as well as beam reinforcement detailing diagrams, may be viewed on the Detail Report.

Beam Bending Reinforcement Spreadsheet

The **Beam Bending Reinforcement Spreadsheet** records the top and bottom flexural reinforcement steel required for the left, middle, and right locations of each beam. This spreadsheet may be accessed by selecting **Beams ▸ Beam Reinforcing** on the **Results Menu** and the results are listed on the **Beam Bending** tab.

Beam Bending Reinforcement (Envelope)

Beam Bending | Beam Shear

	Beam	Shape	Span	Left Top	Left Bot	Mid Top	Mid Bot	Right Top	Right Bot
1	B1	CRECT12X12	1		2 #5	2 #5			2 #5
2	B2	CRECT12X12	1				2 #5		
3	B3	CRECT18X12	1	2 #5			2 #5		2 #5
4	B4	CRECT18X12	1		2 #5			2 #5	
5	B5	CRECT18X12	1		2 #5				
6	B6	CRECT12X24	1						

The **Beam** column lists the beam label.

The **Shape** column displays the beam size. .

The **Span** column displays the span number corresponding to the reinforcement sections listed. Span '1' is the span beginning at the "start" of the beam and subsequent spans are numbered '2', '3', '4', and so forth moving from the "start" to the "end" of the beam.

The program assumes that the moment diagrams for all beam spans have two or fewer points of inflection. Therefore, each span is broken into **Left, Middle, and Right Reinforcement Sections** for flexural steel layout. Each section is further broken into **Top and Bottom Reinforcement Sections**. Note that a beam may have only two or

even one reinforcement section. In this case, the other reinforcement sections would be left blank in this spreadsheet.
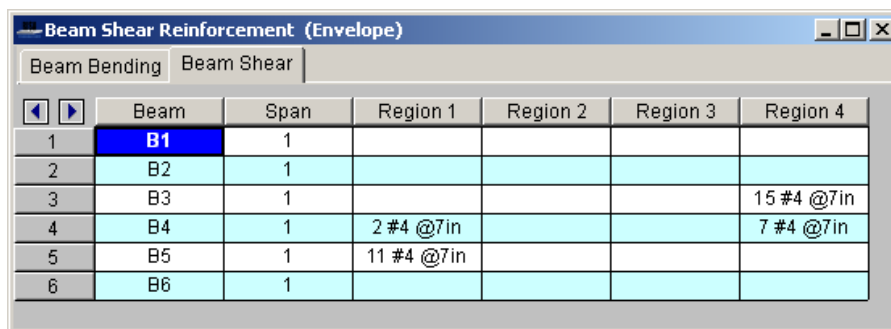
The **Left Top**, **Left Bot**, **Mid Top**, **Mid Bot**, **Right Top**, and **Right Bot** entries record the number and size of flexural reinforcement bars that are required in each of the six **Reinforcement Sections**. The first number indicates the number of parallel reinforcement bars in that section. The second number, preceded by the '#' sign, indicates the size of reinforcement bars used.

Note

- Only reinforcement bars selected by the program are listed in this spreadsheet. If a custom rebar layout is used for a particular beam, all six reinforcement section entries will report "CUSTOM" as a reminder that the user applied a custom rebar layout.
- Longitudinal reinforcement bars are assumed to be in a **single layer** at the top and/or bottom of the member.
- Longitudinal reinforcement bars for the left and right sides of adjacent spans have been "smoothed" such that the larger steel area is used for both sides.

Beam Shear Reinforcement Spreadsheet

The **Beam Shear Reinforcement Spreadsheet** records the shear reinforcement ties required in each shear region of each beam. This spreadsheet may be accessed by selecting **Beams** ‣ **Beam Reinforcing** on the **Results Menu** and the results are listed on the **Beam Shear** tab.

	Beam	Span	Region 1	Region 2	Region 3	Region 4
1	**B1**	1				
2	B2	1				
3	B3	1				15 #4 @7in
4	B4	1	2 #4 @7in			7 #4 @7in
5	B5	1	11 #4 @7in			
6	B6	1				

The **Beam** column lists the beam label.

The **Span** column displays the span number corresponding to the shear regions listed. Span '1' is the span beginning at the "start" of the beam and subsequent spans are numbered '2', '3', '4', and so forth moving from the "start" to the "end" of the beam.

Each beam's shear reinforcement layout is broken into either two or four **Shear Reinforcement Regions**. The user can control whether the program uses '2' or '4' regions from the **Design** tab of the **Model Settings Dialog**. The program will try to group the required shear ties/stirrups into two or four regions and will allow for a middle region to have no shear reinforcement if the shear force is lower than that for which the code requires shear reinforcement.

The **Region 1**, **Region 2**, **Region 3**, and **Region 4** entries record the number, size, and spacing of shear reinforcement ties/stirrups that are required in each of the **Reinforcement Regions**. The first number of each entry indicates the total number of ties/stirrups that are required in that region of the beam span. The second number, proceeded by the '#' sign, indicates the size of reinforcement bars used. The third number, proceeded by the "@" symbol, indicates the spacing of the ties/stirrups in that region of the beam span.

Note

- If '2' shear regions are selected on the **Design** tab of the **Model Settings Dialog**, the Region 2 and Region 3 entries in this spreadsheet will be left blank.

- The concrete code checks are only performed at the sections where the internal forces are calculated. The number of internal force calculations is based on the setting in the Model Settings dialog. Normally, this is acceptable for design and analysis. However, it is possible for the design locations (face of support for moment and "d" from the face of support for shear) to be located far enough away from the nearest internal force location that it could affect the code check results. If this happens, it may be advisable to use a larger number of internal sections. Or, the user may be forced to calculate the maximum Vu and Mu themselves.

Concrete Beam Detail Reports

The **Concrete Detail Reports** allow you to see the overall force state for any particular beam. Detail reports for concrete beams are not based on individual load combinations. Instead, they are based on an envelope of the solved load combinations.

Detail reports for concrete beams can, and often do, go more than one page in length due to the large amount of information that must be displayed for concrete design. One reason for this is that RISAFoundation figures out the number of spans for concrete beams based on the number of internal supports, thus one physical member may have several spans that all must be reported.

Note:

- - This button will allow you to take a snapshot of the current detail report you are viewing so that it can be added to a report. View the Printing topic for more information.

Beam Detail Reports

The image below is the first portion of a detail report for a concrete beam showing the member information, warnings, force diagrams, code checks, and span information.
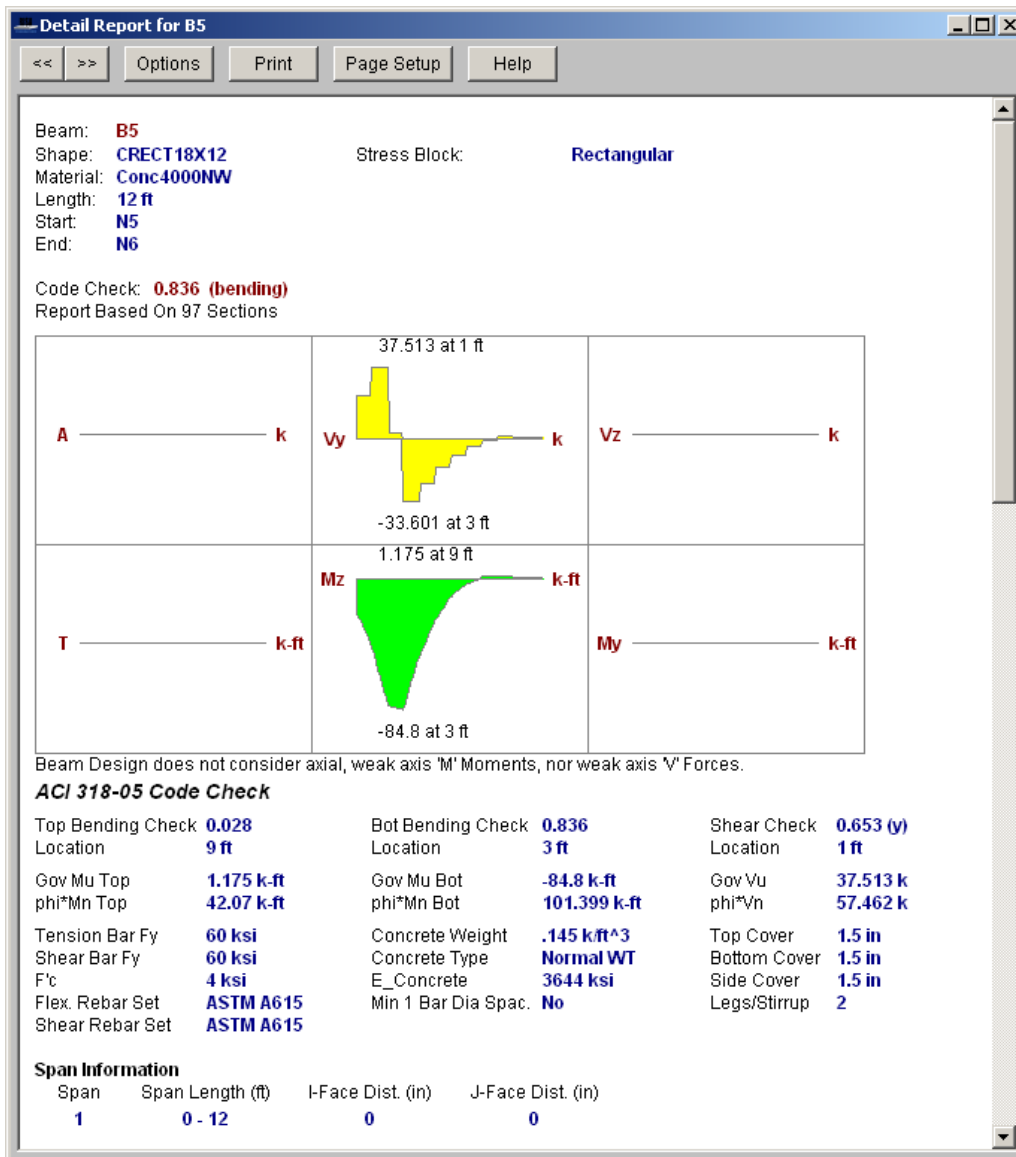
You can tell the **Member Type** by looking at the black title in the upper left corner next to the red member label. This title will always show the member type (Beam).

The **Member Information** in the text above the force diagrams shows basic member information as well as the **Concrete Stress Block** type used in the solution.

The next section of the detail report contains the **Member Force Diagrams**. The diagrams shown are envelope diagrams of all solved load combinations. Any **Unused Force Warnings** or critical **Design Warnings** will be shown directly below the force diagrams in the detail report.
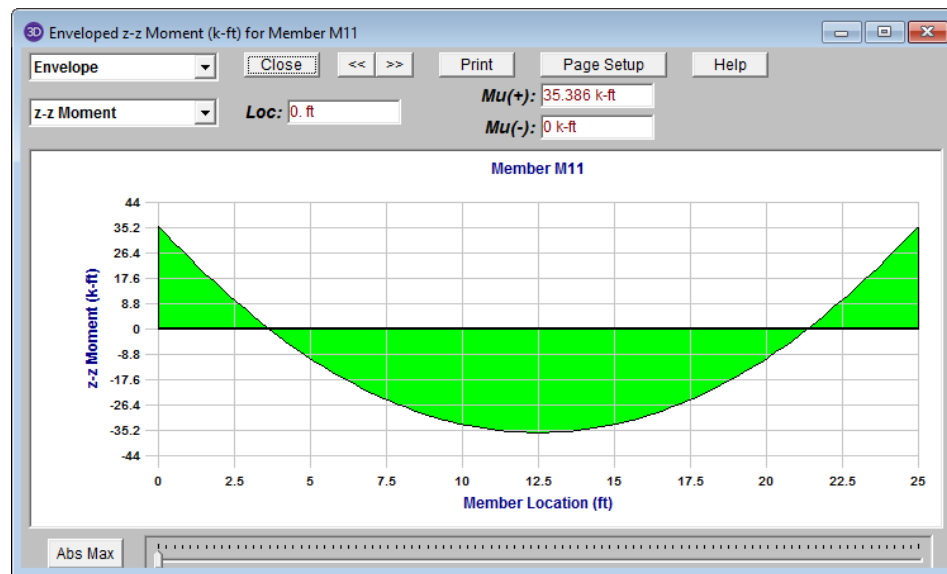
Each enlarged diagram will also have a slider bar at the bottom of the window for checking forces at all locations along the member. There is also an **Abs Max** button that will jump the slider bar to the absolute maximum value in the diagram. Note that once an enlarged diagram is opened, diagrams for other forces may be accessed via the pull down menu on the left.

The **Code Check Information** directly below the force diagrams is a summary of the governing checks for bending and shear, their location, and the section capacities at those locations. Separate bending checks for the most critical top and most critical bottom condition are given. **Gov Muz Top** and **Gov Muz Bot** represent the governing ultimate moment in the top and bottom of the beam respectively. **Gov Vuy** represents the governing ultimate shear along the local y axis of the beam.

The values **phi*Mnz Top** and **phi*Mnz Bot** represent the nominal moment strength in the top and bottom of the beam respectively, reduced by the appropriate **Strength Reduction Factor**, **Phi**, as indicated in the code. Likewise, the value **phi*Vuy** represents the nominal shear strength in the beam, reduced by the appropriate Phi Factor.

There is also general concrete, reinforcement, and bar cover information about the section provided which you would need if you were doing a hand check. **Concrete Type** (Normal Weight vs Light Weight) is automatically determined from the **Concrete Weight** density per the ACI code. λ is taken from the Materials spreadsheet. The **E_ Concrete** value shown here is either the value entered on the **Materials Spreadsheet** or is the calculated value based on the given f'c and weight density (if the 'E' value was left blank on the Materials Spreadsheet).

The **Span Information** gives the start and end of each span centerline within the member, as well as the distance from the support centerline to the face of the support for each end of the span.

The next portion of the detail report shown below contains detailed information for the placement of the **Bending Steel** and the **Bending Span Results** for each span. The bending capacity for the governing section in each span is shown as **Mnz**, the nominal moment strength. **Rho Min** and **Rho Max** are the minimum and maximum required reinforcement ratios at each location. These values are based on the minimum and maximum reinforcing requirements for flexural members as described in ACI 318-14 Sections 9.6.1 and 21.2.2/9.3.3.1 respectively (ACI 318-11 Sections 10.5.1 and 10.3.3/10.3.5 respectively). **Rho** is the ratio of reinforcement corresponding to the area of steel provided at each location, **As Prvd**. The **As Req** value is the area of steel required at each location.

Note:

- Per ACI 318-14 Section 9.6.1 (ACI 318-11 Section 10.5.3), the reinforcement ratio (ρ) chosen by the program can be less than ρ_{min} when A_s Provided exceeds A_s Required by more than 33%

The next portion of the detail report, shown above, contains detailed information for the placement of the **Shear Steel** and the **Shear Span Results** for each span. Shear results are shown by region within each span. The number of regions used is a function of the shear diagram, with the maximum number of regions being taken from the **Shear Regions** setting on the **Design** tab of the **Model Settings Dialog**. The number, size, and spacing of reinforcing bars is given for each region. Also indicated is the nominal shear strength, **Vn**, in each region. The portion of the nominal shear strength provided by the concrete and the steel, **Vc** and **Vs** respectively, is given for each region. The area of steel required, **As Reqd**, and the area of steel provided, **As Prvd**, are also given for each shear region and are reported as 'area of steel per unit dimension', i.e. in^2/in or mm^2/mm.

It should be noted that the values for **Mn** and **Vn** given in this section of the detail report are the UNREDUCED nominal capacities of the member at each span/region. The actual design capacities would be obtained by multiplying these values by their respective **Phi Factors** indicated in the code.

The last section of the detail report shows the **Beam Reinforcement Detailing Diagrams**. The **Rebar Detailing** portion of the report shows elevation views of the beam complete with top and bottom flexural reinforcement indicated for the left, middle, and right portions of each span. The number and size of bars required in each section is indicated on the top middle of each drawn bar. The required length of each bar is indicated on the bottom middle of each drawn bar in parenthesis. Development lengths are shown in parenthesis at one end of each bar and is represented by a dashed line. For bars at the ends of the beam, hook lengths are given in addition to the development lengths and are shown in brackets.

Note:

- Development lengths are calculated per ACI 318-14 Sections 25.4.2 and 25.4.3 (ACI 318-11 Sections 12.2.2 and 12.5.2. No additional factors are used, aside from the lightweight modification factor.

The values shown at the bottom corner of each span indicate the distance from the start of the beam to the face of a support. Flexural bars at the ends of the beam are measured beginning at the face of the support and bars at intermediate supports are measured to the center of the support.

The number, size, and spacing of shear reinforcement is also indicated below each span in the corresponding shear region. Each shear region is indicated by vertical lines at the bottom of the beam.
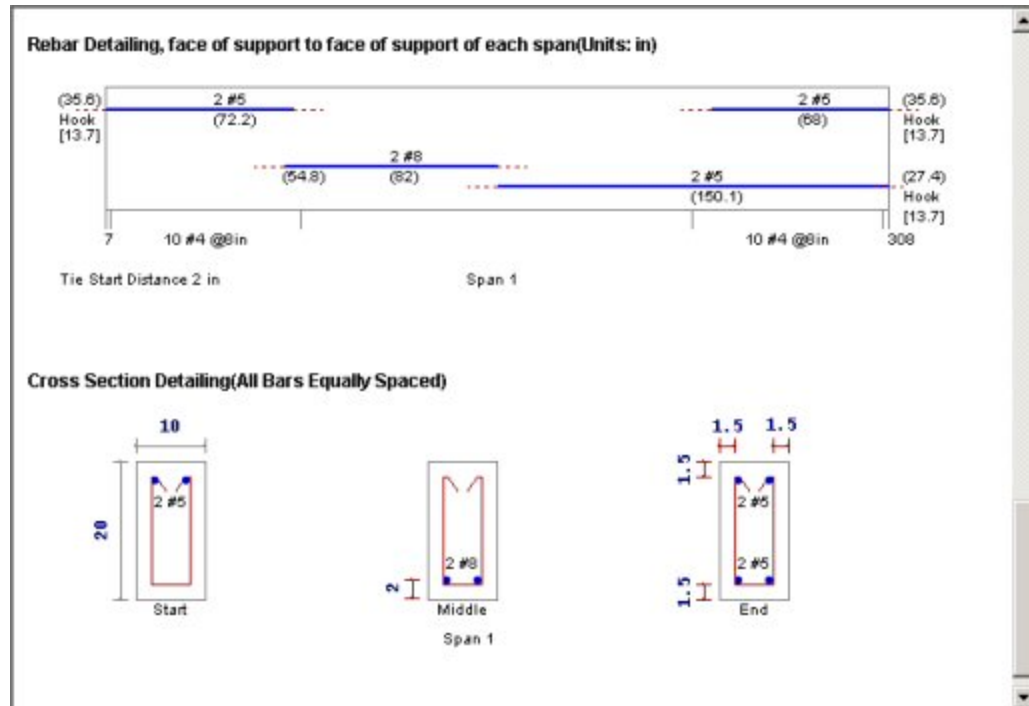
The **Cross Section Detailing** portion of the report shows cross sectional views for the start, middle, and end of each beam span. The number and size of flexural bars for each cross section are shown as well as the orientation of the shear ties/stirrups. The clear cover to each stirrup for the top and sides is shown. The overall beam dimensions for each span are indicated on the 'Start' cross section.

Beams - Forces

When the model is solved, the results are separated into specific design results and generic results. The generic beam forces results are discussed in this section. The design results are discussed in the Beams - Results section.

Number of Reported Sections

Note that the beam forces are only reported at the section locations. For example, if you set the **Number Of Sections** in the **Model Settings Dialog** to be '2', you will not get any results for the middle of your beam, you will only get results for the end points. If you have a point load applied to your member at a location that is not a section location, you will probably not report the maximum moment in the section if it does not occur at an endpoint.

To Adjust the Number of Sections

- In the **Model Settings Dialog**, select the **Solution** tab.
- Adjust the **Number of Sections** as needed.

Note

- Adjusting the number of sections affects the amount of output.

Number of Internal Sections

Internally, the program subdivides the beam into equally spaced sections to calculate forces, code checks, etc. The **Internal Sections** can be adjusted in the **Model Settings Dialog**. If this value were set to 100, this would mean that for a beam that is 100ft long, RISAFoundation will calculate these values at approximately every foot. These values are then used in the beam code checks, the diagrams in the model view, and in the detail reports. The locations of the maximum code checks are reported at a distance from the Start Point.

To Adjust the Number of Internal Sections

- In the **Model Settings Dialog**, select the **Solution** tab.
- Adjust the **INTERNAL Sections** as needed.

All other results are reported at the **Number of Sections** that you specify in the **Model Settings Dialog**. This controls how many places you receive **reported** or **printed** beam force results. These locations are also equally spaced so setting the value to 5 will give you 5 equally spaced sections; at the ends, the middle and the quarter points.

Note

- Adjusting the number of **Internal Sections** will not affect the amount of output in the results spreadsheets.
- You may want to stick with odd numbers for the **Number of Sections**. Setting the **Number of Sections** to an even number will not report forces at the midpoint of the beam, which is often the location of maximum moment.

Beam Force Results

Access the **Beam Section Forces** Spreadsheet by selecting the **Results** Menu and then selecting **Member Forces** then clicking the **Sections** tab**Beams ▶ Forces**.

These are the beam forces calculated along each beam. The number of sections for which forces are reported is controlled by the **Number Of Sections** specified in the **Model Settings Dialog**.

The units for the forces are shown at the top of each column. As for the sign convention, the signs of these results correspond to the beam's local axes, using the right hand rule. The left side forces at each section location are displayed. There are three force values for each section location.

These are shear parallel to the local y axis (Shear y-y), torque moment, and moment about the member's local z axis (Moment z-z).

These section forces may also be displayed graphically. Remember that the section forces used for the plot are the left side forces. For an example of what you would see for the graphic plot of the moment diagram for a member, please see below:
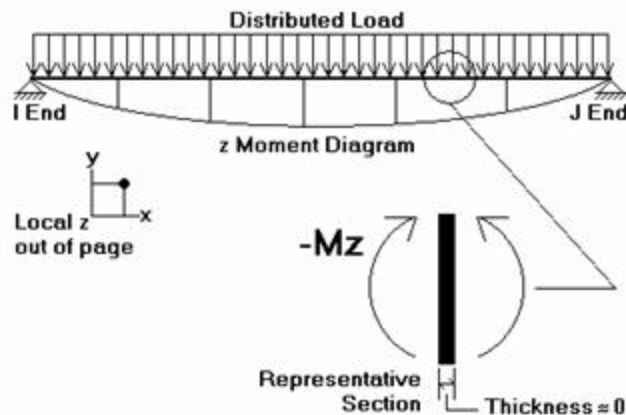
RISAFoundation uses the right hand rule **joint convention** and is always consistent with this convention. Since the left side moment is being used, a beam under negative Mz moment would have the "holds water" deflected shape, which is contrary to some **beam conventions**. The opposite is true for My moments which will tend to "hold water" under a positive moment and "shed water" under a negative moment.

The **2nd/1st Moment Ratios Tab** shows the ratio of maximum moment with and without secondary P-Delta effect. In general, the larger the ratio is, the more significant the secondary effect is. Notice that some design codes may have a limit on this ratio (e.g., ACI has this limit as 1.4), but this limit is not checked by the program.

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort**, and other options.
- See Model Display Options - Beams to learn how to plot beam results.

Customizing RISAFoundation

You may customize many of the default parameters, design and analysis options in RISAFoundation. In this way you can modify the program so that it best suits you and your work processes. All customization may be defined or redefined at any time. The **Application Settings** option on the **Tools Menu** provides you control over the behavior of the software. The **Save as Defaults** feature allows you to specify the default settings for new model files. These features are discussed below. Custom reports may also be defined and saved for future use. See [Printing](#) to learn how to build a custom report.

Save as Defaults

You may use the **Save as Defaults** feature in the following dialog boxes by entering the default information in the dialog and clicking the **Save as Defaults** button: **Model Settings, Units**, and **Drawing Grids**. This will cause the program to use these settings with any new files that are then created.

Many of the spreadsheets also provide the option to save the current data as the default and every subsequent new file will already have that data. Simply enter the data you want then save it as the default by clicking on the ⊞ button. This way the office standards that you might use in most of your models are already entered and available in new models. This feature is available in the following spreadsheets: **Materials**, **Custom Wood Species**, **Design Rules**,**Footing Definitions**, and **Load Combinations**.

Once you create a new file you may redefine any of the default data and settings for that particular file so the **Save as Defaults** feature may be used to give you a good starting point for new files but won't hold you to those settings.

Application Settings (Preferences)

Program options may be accessed by selecting **Application Settings** from the **Tools Menu** and are divided into the five sections described below. Many of the Application Settings themselves are self-explanatory.

General Application Settings (Preferences)

The general Application Settings are straightforward. For help on an item click ⍰ and then click that item. It may be a good idea to disable the **Automatically refresh…** option when working with large files or slower computers. You may also set the backup timing. See [Automatic Backup](#) to learn about the backup capabilities of RISAFoundation. The **Reset Customization Options** button will clear all of the Application Settings that you have set on any of the tabs.
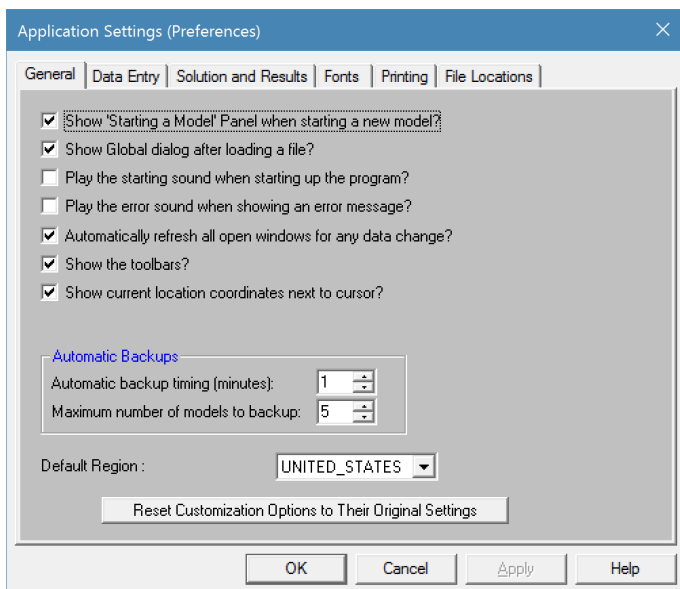
Show "Starting a Model" Panel when starting a new model – The New Model Dialog will be displayed when opening the program or selecting 'New File' from the File Menu.

Show Global dialog after loading a file – Displays the Model Settings settings automatically after loading a file.

Play the starting sound when starting up the program – A startup sound will be played when the program opens.

Play the error sound when showing error messages – An error sound will be played when an error is displayed.

Automatically refresh any open windows for any data change – Changes to the model will automatically be reflected in all windows – spreadsheets and model views. For large models you may want to limit the number of open windows or disable this feature altogether.

Show Toolbars – All toolbar commands may also be found in the menu system so if you want more work space you may disable the toolbars.

Show Exclude Results confirmation message – After solving the model you may use the Exclude Feature to graphically select items that you wish to see in the results spreadsheets, "excluding" other results. This enables a confirmation message warning you that some results are not shown.

Automatic backup timing – Automatic backup of the currently open model occurs at the specified interval. Each backup overwrites the previous one, such that only one backup of a given model exists at any time.

Maximum Number of Models to Backup - This controls how many models are kept in the backup folder. Once the folder becomes "full" according to this setting, the oldest backups will be deleted automatically. Setting this to zero turns off automatic backups.

Reset Customization Options – Choose this to return to the program defaults.

Default Region - Choose the default region for your projects.

Data Entry Application Settings (Preferences)

To use bigger or smaller fonts in the spreadsheets you may adjust the row heights. You may also specify the number of decimal places that are displayed. The one exception is the **Joint Coordinates**. RISAFoundation maintains the coordinates to 15 significant figures and the exact value is always displayed.

If you wish to use a prefix with your point, beam, and plate labels, such as "P" with points, you can specify the default prefix. These prefixes may be changed as you build your model.
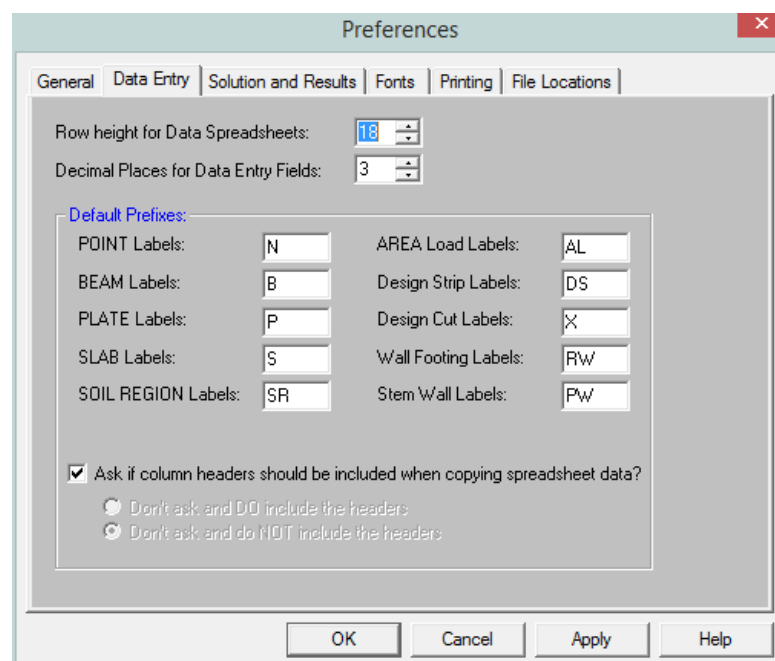
Row height for data spreadsheets – Sets the row height and font size for data spreadsheets.

Decimal places for data entry fields – Sets the number of decimal places to display in the data spreadsheets with a maximum of four places.

Default prefix for POINT labels – Sets the default prefix to be used in point labels.

Default prefix for BEAM labels – Sets the default prefix to be used in beam labels.

Default prefix for PLATE labels – Sets the default prefix to be used in plate labels.

Default prefix for SLAB labels - Sets the default prefix to be used in slab labels.

Default prefix for SOIL REGION labels - Sets the default prefix to be used in soil region labels.

Default prefix for AREA Load labels - Sets the default prefix to be used in area load labels.

Default prefix for Design Strip labels- Sets the default prefix to be used in design strip labels.

Default prefix for Design Cut labels - Sets the default prefix to be used in design cut labels.

Default prefix for Wall Footing labels - Sets the default prefix to be used in wall footing labels.

Default prefix for Stem Wall labels - Sets the default prefix to be used in stem wall labels.

Ask if column headers should be included when copying... - If this checkbox is checked then the program will ask if you want to copy the spreadsheet header any time you attempt a copy command in a spreadsheet. If this checkbox is not checked then you have the option of either always copying the spreadsheet headers, or never copying them.

Solution and Results Application Settings (Preferences)

RISAFoundation can provide a warning when clearing results. To use bigger or smaller fonts in the results spreadsheets you may adjust the row heights. You may also specify the number of decimal places that are displayed. The number of figures displayed may not be the actual number. Behind the scenes RISAFoundation maintains numbers to numerous decimal places.
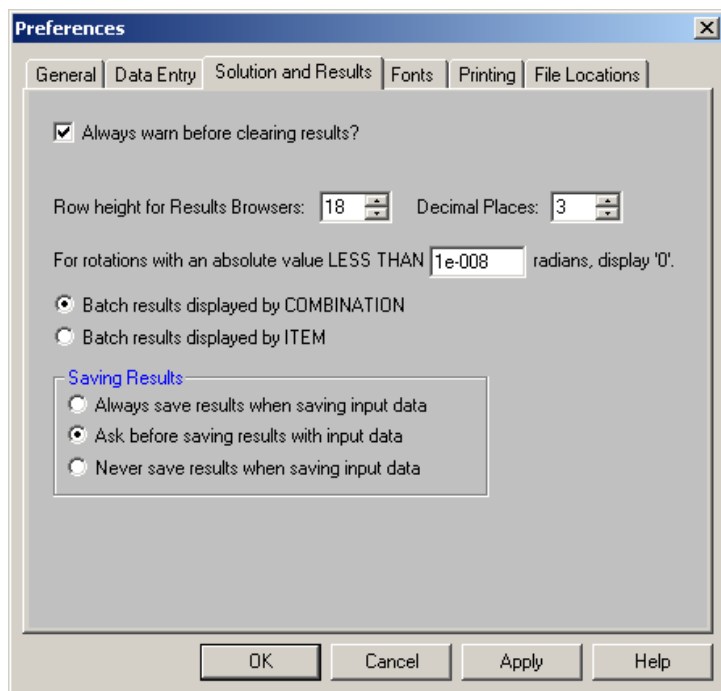
Always warn before clearing results? – Verifies that results are to be cleared to edit the model.

For additional advice on this topic, please see the RISA Tips & Tricks website: www.risa.com/post/support. Type in Search keywords: **Sum of Reactions**.

Row height for Results Browsers – Sets the row height and font size for results spreadsheets.

Decimal Places – Sets the number of decimal places to display in the results spreadsheets with a maximum of four places.

Rotation Limits – Shows 0 for the rotation when smaller than this value.

Batch Results Display - The results of a batch solution may be grouped by load combination or by item. For example you can group results for all members under each particular load combination or you can group results from each combination under a particular member. The setting here is merely a preference. Once you have solved a model you can switch back and forth using the **Results Menu**.

Saving Results – These options let you control what is done with the results when saving a file.

Font Application Settings (Preferences)

The font Application Settings can be used to adjust the fonts used by the spreadsheets, results browsers, and graphics. The font changes will affect both the on-screen displayed data and the printed data. The exceptions to this are the spreadsheet and browser fonts which may be changed for on-screen display but are hard-wired for printing purposes.
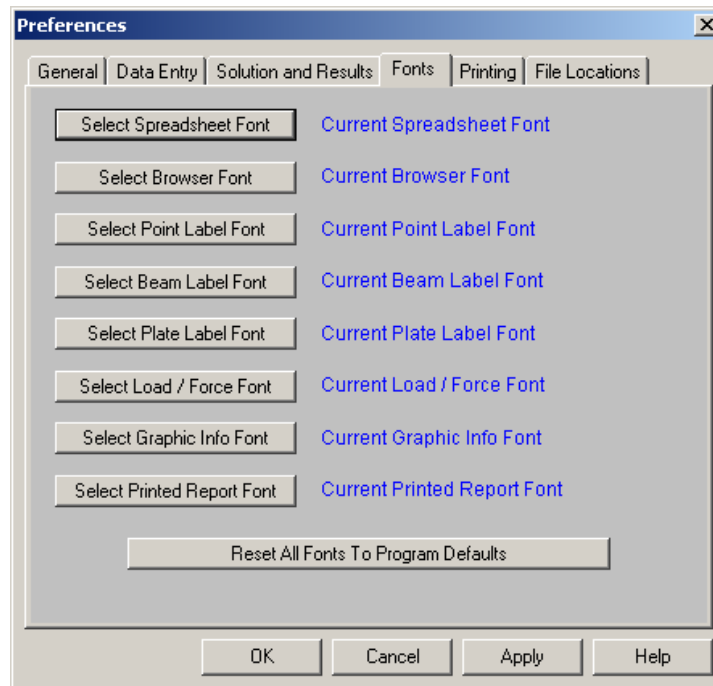
If the font data has be set to some unusual settings, then the user can click the **Reset All Fonts to Program Defaults** button to restore the fonts to what is normally expected for the RISA program.

Printing Application Settings (Preferences)

This tab gives many options for printing output. See Printing for additional information.
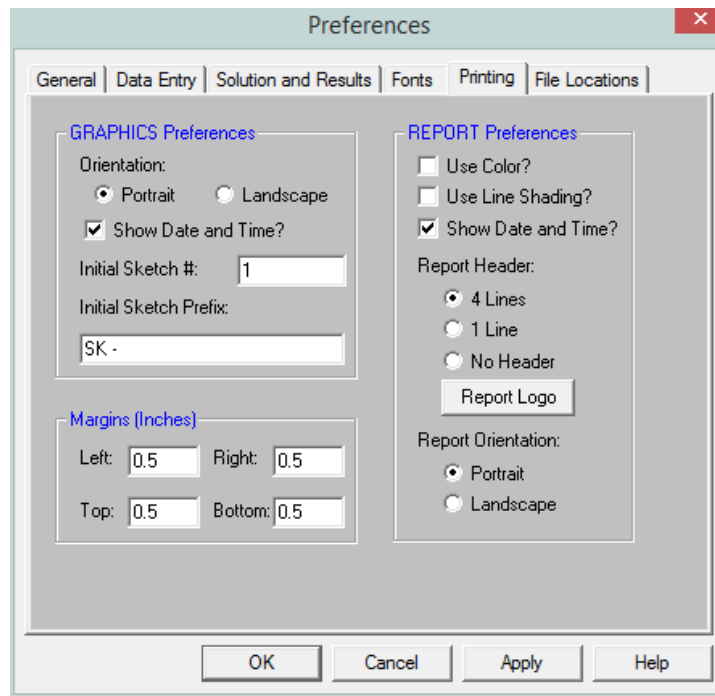
Orientation for graphics – Sets the default paper orientation for graphic printing.

Sketch Information - Sets sketch numbers and prefixes for graphic printing.

Margins – Sets the default printing margins.

Report Application Settings – These allow you to define information about colors, date and time, headers, report logo and overall report orientation.

Use Line Shading? - Defines whether the spreadsheet output shades every other line. This can give contrast to make the information easier to read.

Report Logo - Allows you to add either a RISA logo or any company logo to your printed output.
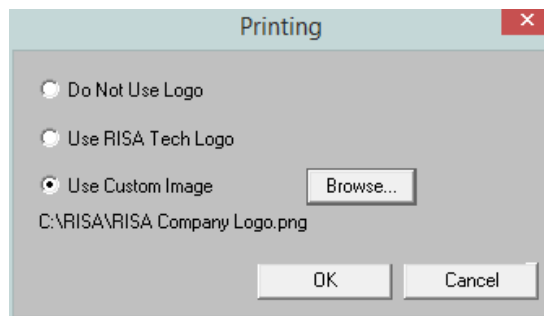
File Application Settings (Preferences)

The locations for data files, databases, temporary space, importing, and backing up may be specified separately by choosing from the list.
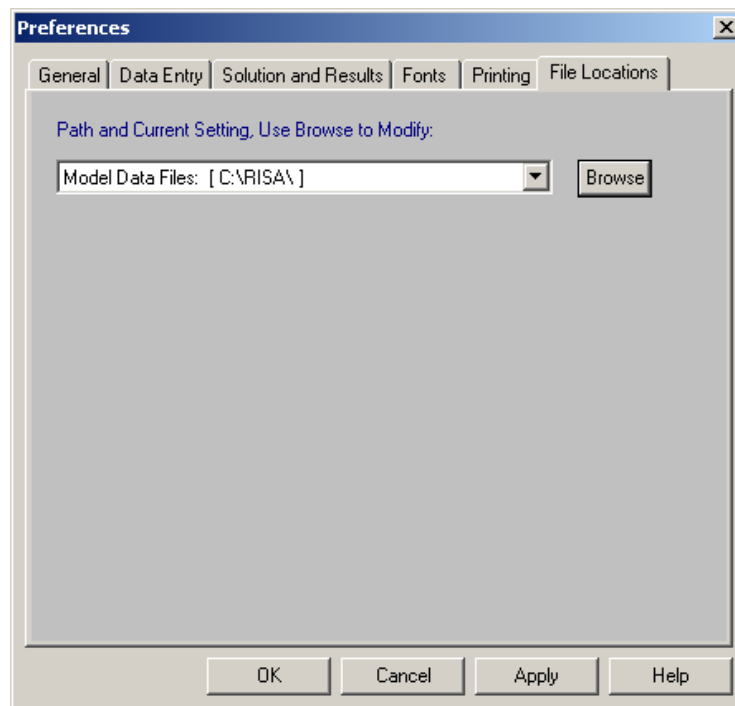
Path and current setting – For each file type in the list the current setting is displayed. Click the drop down list to view different file types. Click the browse button to choose a different location.

Design Optimization

RISAFoundation will optimize Concrete beams and slab reinforcement. The criteria used for this optimization are the selected design code and the **Design Rules** assigned to the beam or slab design strip/cut.

The **Member Design Rules Spreadsheet** records the parameters for the optimization. Optimization is performed for minimum weight, taking into consideration any depth, width, rebar limitations, etc. Note that the design rules input is one large spreadsheet, thus all of your design rules will be in the same place. Note that the dimensional rules and the reinforcement rules are all a separate entity. They have no interaction with each other in the program. They are simply all input into the same location. For example, your minimum member width rules will not be influenced by concrete beam reinforcement rules.

You can assign the design rules graphically as you draw beams and design strips or later as a modification. Design rules for slab design cuts may only be applied via the **Design Cuts Spreadsheet**. See Modifying Beam Properties, Slabs - Design Strips, and Slabs - Design Cuts for more information.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Redesign**.

Deflection (Beams) tab:

This only applies to Beam members.

Design Rules - General

You must assign a unique **Label** to the design rules. You then refer to the design rule by its label when assigning it to slabs or beams. The Label column is displayed on all tabs of the spreadsheet.

Max Bending and Shear Check

Enter the maximum bending and shear unity checks in the **Max Bending Chk** and **Max Shear Chk** columns of the spreadsheet. This should usually be specified as '1'. If you desire a larger factor of safety, provide a lower factor (i.e. '0.95').

Concrete Cover

The last two columns, **Top Cover** and **Bottom Cover**, are used to specify the clear cover measured to the slab or beam flexural reinforcing steel. The units are listed in the column headers. Note that **Side Cover** will be entered on the subsequent tabs of the spreadsheet.

Design Rules - Beam Rebar

Note

- If you would like to define specific flexural and shear rebar layouts for beams and columns, either use the **Explict** option below or see Rebar Layout Database for Custom Rebar layout.

Optimize

The options in this column are **Optimize** and **Explicit**. **Optimize** indicates the program will find the most optimal size and spacing of the reinforcement. **Explicit** indicates the exact number of and size of reinforcement bars. When Explicit is selected the additional columns are editable and allow you to enter the quantity for **Top/Bottom Bars**, **Add'l Side Bars**, and **Shear Bar Spacing**. These columns are greyed out when **Optimize** is selected.

Flexural Bars

Use the **Min Flex Bar** and **Max Flex Bar** columns to restrict bar sizes for your beam flexural reinforcing. Currently we support the ASTM A615 (imperial) and ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar) reinforcement standards. You may specify your rebar set in the Model Settings - Design. You can force the program to analyze one bar size by setting the Min and Max values to be the same bar size.

Shear Ties
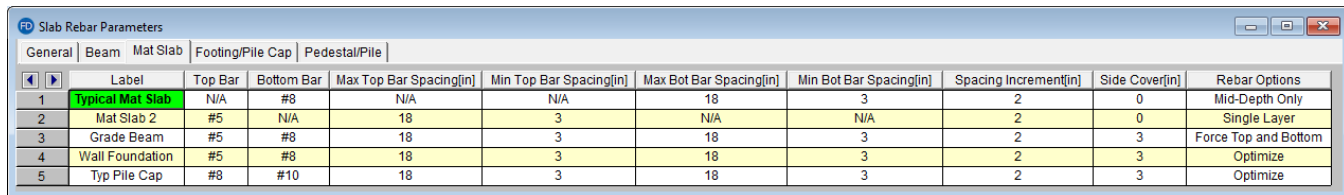
Use the **Shear Bar** column to enter the size of your shear ties. Currently we support the ASTM A615 (imperial) and ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar) reinforcement standards. You may specify your rebar set in the Model Settings - Design.

Use the **Legs per Stirrup** column to enter the specific information about how may legs each of your shear ties is expected to have.

Side Cover

The last column, **Side Cover**, is used to specify the clear cover measured to the shear reinforcing. The units are listed in the column header.

Design Rules - Slab Rebar

	Label	Top Bar	Bottom Bar	Max Top Bar Spacing[in]	Min Top Bar Spacing[in]	Max Bot Bar Spacing[in]	Min Bot Bar Spacing[in]	Spacing Increment[in]	Side Cover[in]	Rebar Options
1	Typical Mat Slab	N/A	#8	N/A	N/A	18	3	2	0	Mid-Depth Only
2	Mat Slab 2	#5	N/A	18	3	N/A	N/A	2	0	Single Layer
3	Grade Beam	#5	#8	18	3	18	3	2	3	Force Top and Bottom
4	Wall Foundation	#5	#8	18	3	18	3	2	3	Optimize
5	Typ Pile Cap	#8	#10	18	3	18	3	2	3	Optimize

Rebar Sizes

The **Top Bar** and **Bottom Bar** columns register the flexural reinforcement bar sizes that are to be used in the footing design. Currently we support the ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar) and CSA G30.18 (canadian) reinforcement standards. You may specify your rebar set in the [Model Settings - Design](#). Deformed welded wire sizes can be selected from the "Top Bar" and "Bottom Bar" dropdown list.

Rebar Spacing

The **Max Top Bar Spacing**, **Min Top Bar Spacing**, **Max Bot Bar Spacing**, and **Min Bot Bar Spacing** columns are used to set the maximum and minimum bar spacing for the top and bottom slab flexural reinforcement. The units are listed in the column headers.

The **Spacing Increment** column sets the rebar spacing increment that will be used by RISAFoundation when analyzing various rebar layouts between the maximum and minimum bar spacing set in the previous columns. The units are listed in the column header.

Side Cover

The **Side Cover** column is used to specify the clear cover from the edge of a design strip or cut measured to the nearest slab flexural bar. The units are listed in the column header. Note that this entry should probably be '0' for design strips or cuts that do not extend to the edges of a slab provided that they are not directly adjacent to another design strip or cut with a side cover greater than '0'.
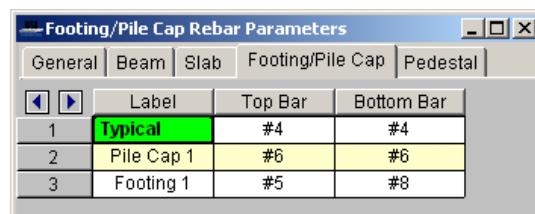
Rebar Options

The last column allows the user to place their slab reinforcement at the mid-depth of the slab, force it to be placed at the top and bottom, one single layer at any depth, or to be optimized by the program solution.

Note:

- For **mid-depth** rebar, the program will use the *bottom* bar size and spacing options.
- For **single layer** rebar, the design uses the *top* bar size, and spacing options. The top cover under the General tab is used to define the rebar placement from the top of the slab.

Design Rules - Footing/Pile Cap Rebar
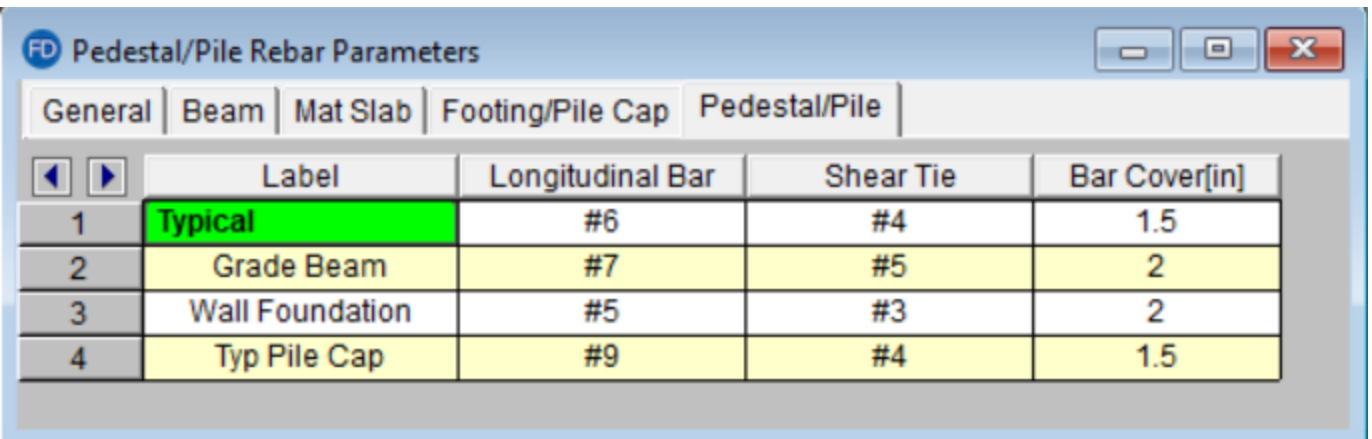
	Label	Top Bar	Bottom Bar
1	Typical	#4	#4
2	Pile Cap 1	#6	#6
3	Footing 1	#5	#8

Rebar Sizes

The **Top Bar** and **Bottom Bar** columns register the flexural reinforcement bar sizes that are to be used in the footing design. Currently we support the ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar) and CSA G30.18 (canadian) reinforcement standards. You may specify your rebar set in the Model Settings - Design. Deformed welded wire sizes can be selected from the "Top Bar" and "Bottom Bar" dropdown list.

Design Rules - Pedestal/Pile Rebar

Rebar Sizes

The **Top Bar** and **Bottom Bar** columns register the flexural reinforcement bar sizes that are to be used in the footing design. Currently we support the ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar) and CSA G30.18 (canadian) reinforcement standards. You may specify your rebar set in the Model Settings - Design.

Pedestal Bar Cover

The last column, **Pedestal Bar Cover**, is used to specify the clear cover measured to the shear reinforcing. The units are listed in the column header.

DXF Files

RISAFoundation provides the ability to export a drawing of your foundation plan (with footing and beam schedules), Footing Details or Slab Reinforcing to a DXF file.

Note

- You can also import a DXF as a Drawing Grid. See the Import DXF section of the Graphic Editing topic for more information.

Exporting Foundation Plan Drawing

Only the joint, member, wall and element geometry will be translated and used to create an ASCII DXF file. Any other information such as the boundary conditions, loads, etc. will not be translated at this time. You have several options available for controlling how DXF files are exported as follows:

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **DXF Foundation**.

DXF Units

Select the units you desire the CAD model to be created in. The options for the DXF units are none, inches, feet, mm, cm, and meters.

CAD Scale

Enter the scale factor that will cause the full scale RISA model to be scaled up or down to the desired drawing scale. For example, if you created a full scale model that you wanted scaled down to 1/4"=12", the factor would be 0.020833, which is (.25/12).

Standoff Distance

Enter the distance you wish to have the line entities "stand off" from the joints to which they are attached. The standoff distance is measured along the axis of the line. The distance will be in the DXF units, which is defined below. The distance will be used as entered and will not be scaled by the CAD Scale factor.

Note that if you create a DXF file with a non-zero standoff distance, it will be difficult to use the file for model geometry if you read the file back into RISA. (If you read such a file back in, you will end up with multiple joints at each member endpoint which will separated by the standoff distance)

Grid Bubbles

You can export the project grid bubbles either as Circles or Hexagons.

DXF Layer Options

These controls allow you to customize the layer names in the DXF output of your Foundation Plan Drawing.

Exporting Foundation Plot Plans

Once you have all these parameters set, press OK and the program will export a plan view of the foundation model, complete with slabs, footings, grade beams, pedestals, piles, pile caps and retaining walls. These come built in with footing, pedestal, pile, pile cap and retaining wall. Slab reinforcement can be found in the slab reinforcing details below.
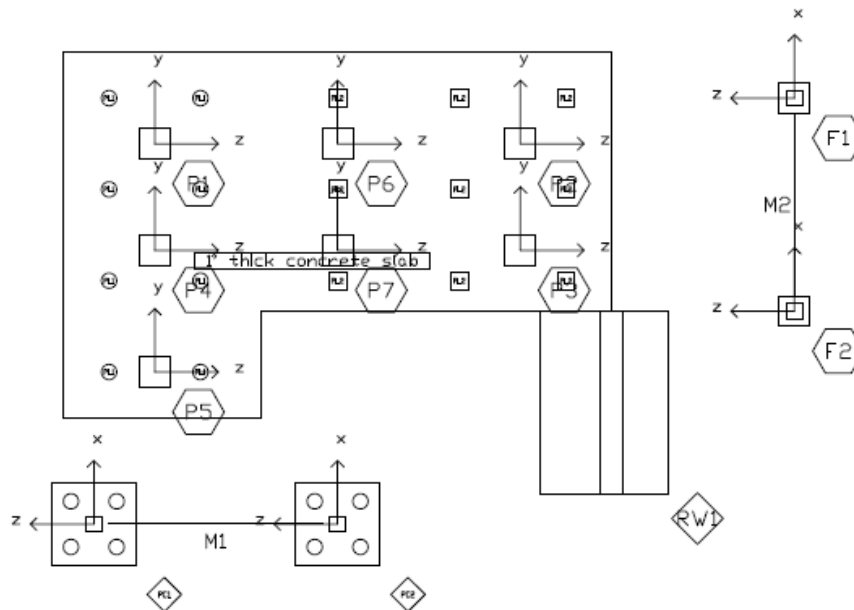
		Footing Size (ft x ft x in)	x-bars			z-bars		Footing Pedestal Schedule				
Mark	No.	(X Width x Z Width x Thk)	Top	Bottom	Top		Bottom	Height (in)	Size (in x in)	Flex. Bars	Ties	
F1	1	2 x 2 x 12	–	#6 @ 10.5 in max	–		#6 @ 10.5 in max	24	12 x 12	4 – #6	#4 @ 12 in max	
F2	1	2 x 2 x 12	–	#6 @ 10.5 in max	–		#6 @ 10.5 in max	24	12 x 12	4 – #6	#4 @ 12 in max	

Pedestal Schedule

Mark	No.	Height (in)	Size (in x in)	Flex. Bars	Shear Ties
P1	1	12	24 x 24	8 – #6	#4 @ 12 in max
P2	2	12	24 x 24	8 – #6	#4 @ 12 in max
P3	3	12	24 x 24	8 – #6	#4 @ 12 in max
P4	4	12	24 x 24	8 – #6	#4 @ 12 in max
P5	5	12	24 x 24	8 – #6	#4 @ 12 in max
P6	6	12	24 x 24	8 – #6 z-bars	#4 @ 12 in max

Pile Cap Schedule / **Pile Cap Pedestal Schedule**

Mark	No.	(X Width x Z Width x Thk)	Type	Top	Bottom	Top	Bottom	Height (in)	Size (in x in)	Flex. Bars	Ties
PC1	1	5.5 x 5.5 x 24	Pile 1	–	#6@10 in	–	#6@10 in	24	12 x 12	4 – #6	#4@12 in
PC2	1	5.5 x 5.5 x 24	Pile 1	–	#6@10 in	–	#6@10 in	24	12 x 12	4 – #6	#4@12 in

Pile Schedule

Mark	No.	Shape	Diameter/Side (in)	Pile Embed (in)
PL1	16	Round	12	4
PL2	9	Square	14	6

Retaining Wall Geometry Schedule

Mark	Wall Height (ft)	Wall Thick (in)	Toe Length (ft)	Heel Length (ft)	Foot Thick (in)	Key Depth(in)	Key Width(in)
RW1	14	18	3	4	18	0	0

Retaining Wall Reinforcement Schedule

Mark	Int Wall Verts	Ext Wall Verts	Wall Horz Bars	Foot Top Bars	Foot Bot Bars	Foot Long Bars	Dowel Bars
RW1	#7 @ 10 in	#8 @ 18 in	#4 @ 18 in	#6 @ 10 in	#6 @ 14 in	#4 @ 18 in	#7@10in,#8@18in

Grade Beam Schedule

	Size(in x in)			Flex Reinforcing		Shear Ties		
Mark	(Width x Depth)	Span	Location	Top	Bottom	Size	Region	Max. Spacing
M1	12 x 12	1	Start	2 – #5	–	–	–	–
			Middle	–	2 – #5	–	–	–
			End	2 – #5	–	–	–	–
						–	–	–
M2	12 x 12	1	Start	2 – #5	–	–	–	–
			Middle	–	2 – #5	–	–	–
			End	2 – #5	–	–	–	–
						–	–	–

Exporting Slab Reinforcing Details

The program can export a schedule of the horizontal and vertical slab reinforcing details. Here is an example of the horizontal slab reinforcing details.
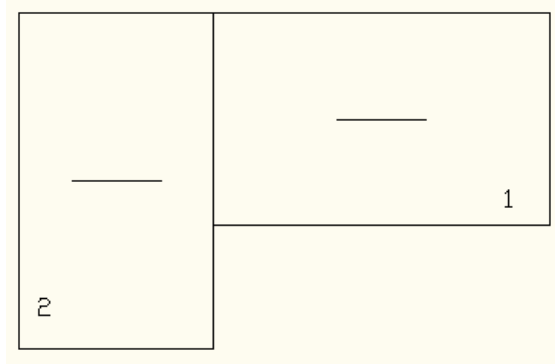
Plan Horizontal Design Strip Schedule

Mark	Flex Reinforcing	
	Top	Bottom
1	#5 @ 18 in max	#8 @ 18 in max
2	#5 @ 18 in max	#8 @ 18 in max

Exporting Footing Details

The program can export a sketch of each designed footing. The sketch will show a reinforcement plan, an elevation view and a pedestal reinforcement plan.

Footing 1 – N3
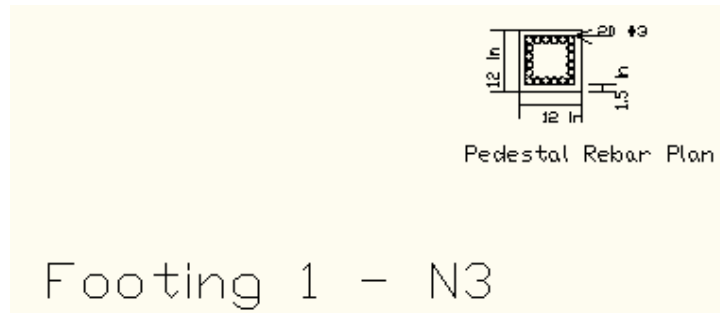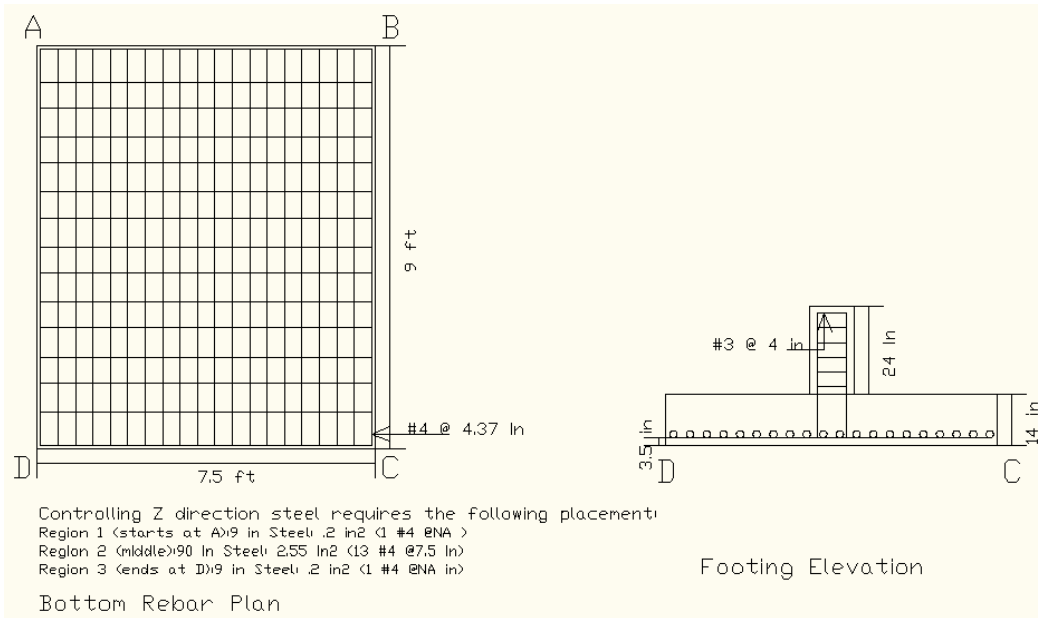
Exporting Pedestal Details

The program can export a sketch of each designed pedestal. The sketch will show the reinforcement, a plan view and an elevation view.
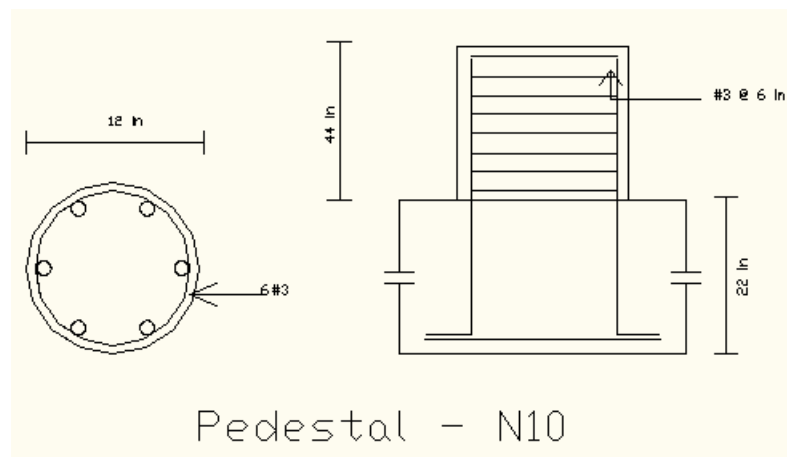
Pedestal – N10

Exporting Pile Cap Details

The program can export a detailed plans of each designed pile cap. The sketch will show a pile cap plan, reinforcement plans, an elevation view and a pedestal reinforcement plan.

Pile Cap 1 — N10
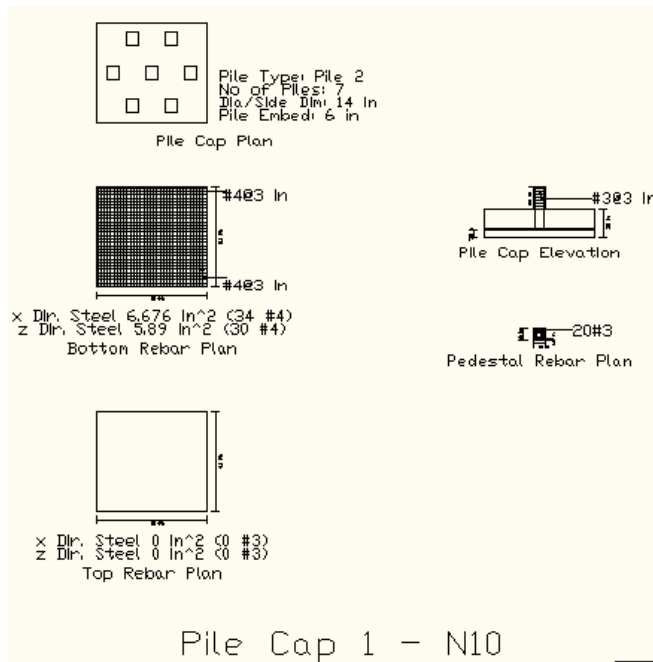
Exporting Retaining Wall Details

The program can export a detailed plan of each designed retaining wall. The sketch will show a section view complete with dimensions and reinforcement sizes and spacings.

RW1
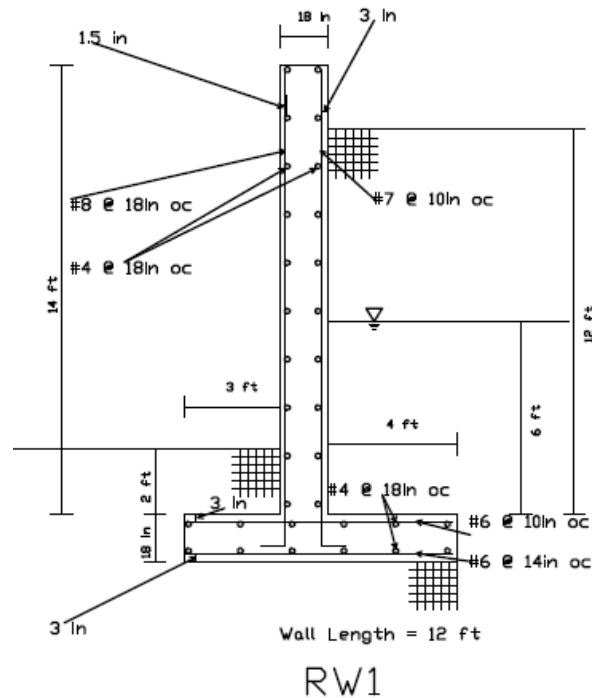
DXF Element Numbers

Different CAD packages handle ordering of geometric data in their DXF files in two basic ways.

Method 1:

Entities are written out into the DXF file based on the order in which they were created within the CAD program itself regardless of the order in which they were selected at the time the DXF file was made. Different operations such as copying, mirroring, arraying, etc. can produce unexpected results and it therefore becomes necessary to consult your CAD program documentation to understand how it stores and orders the geometry that you create via these various operations.

Method 2:

Entities are written out into the DXF file based on the order in which they were selected at the time the DXF file was made. AutoCAD is such a program. In order to control the ordering of the LINE entities, you must select the "Entities" option under the DXFOUT command and then select the lines in the order that you want them to appear in the RISA model.

Note

- Another option to help improve the ordering of the joints, members and elements in a model obtained from reading in a DXF file is to sort and relabel them once in RISA.

DXF File Format

The specific DXF file that you may read and write is the ASCII Drawing eXchange Files (DXF) file. Please note that AutoCAD has several different forms of DXF files available. ASCII is the default form and is the only form currently supported. The DXF read/write feature was written based on the DXF documentation for AutoCAD release 14. The feature has been tested with AutoCAD Versions 13 and 14.

The following is a short excerpt of the AutoCAD ASCII DXF format. This information is provided to help you debug any problems you may be having with DXF files that you are trying to read. For more complete information, consult your CAD documentation.

General

A DXF file is composed of sections of data. Each section of data is composed of records. Each record is stored on it's own line. Each particular item is stored as two records, the first record is a group code and the second record is the data or a keyword. RISA only reads the ENTITIES section.

Group Codes

Each 2 record item start with an integer group code. RISA recognizes the following group codes:

Group Code	Description
0	Identifies the following overall keywords: SECTION, ENDSEC, and EOF. Within the ENTITIES section it also identifies POINT, LINE, and 3DFACE.
2	Identifies a section name (I.e., ENTITIES)
8	Identifies a layer name.
10, 11, 12, 13	Identifies the X coordinate of the 1st, 2nd, 3rd and 4th points of an item.
20, 21, 22, 23	Identifies the Y coordinate of the 1st, 2nd, 3rd and 4th points of an item.

Group Code	Description
30, 31, 32, 33	Identifies the Z coordinate of the 1st, 2nd, 3rd and 4th points of an item.

First and Last Records for a DXF file

Each DXF file must start with the first record as the group code "0". The 2nd record must be the keyword "SECTION". Each DXF file must have the 2nd to last record as the group code "0". The last record must be the keyword "EOF".

Entities Section

The ENTITIES section will be identified by a group code of "0", followed in the next record by the keyword "SECTION". The next record will be the group code 2, followed in the next record by the keyword "ENTITIES".

Item Formats within the ENTITIES Section

The POINT format is started by a group code of "0" followed by the keyword "POINT". The layer name for the POINT will start with a group code record of 8, followed by a record with the actual layer name.

The coordinates for the point will be started by the 10, 20, and 30 group codes respectively for the X, Y, and Z coordinates. Other group codes and data may be present within the POINT data but these will be ignored.

The LINE format is started by a group code of "0" followed by the keyword "LINE". The layer name for the LINE will start with a group code record of 8, followed by a record with the actual layer name.

The coordinates for the first point will be started by the 10, 20, and 30 group codes respectively for the X, Y, and Z coordinates. The coordinates for the second point will be started by the 11, 21, and 31 group codes respectively for the X, Y, and Z coordinates. Other group codes and data may be present within the LINE data but these will be ignored by RISA-.

The 3DFACE format is started by a group code of "0" followed by the keyword "3DFACE". The layer name for the 3DFACE will start with a group code record of 8, followed by a record with the actual layer name.

The X, Y, and Z coordinates for the 1st through 4th points will be started by the 10, 20, and 30 through 14, 24, and 34 group codes respectively. Other group codes and data may be present within the 3DFACE data but these will be ignored.

AutoCAD Layer Names

The only valid characters in an AutoCAD layer name are the letters A to Z, the numbers 0 to 9, and the three following characters: the dollar sign "$", the underscore "_", and the dash "-".

File Operations

You may create, save, and recall files from the **File Menu** or from the toolbar buttons just beneath the menu on the **RISA Toolbar**. Importing and exporting other file types may also be accomplished from the **File Menu**.

Starting Off

When you open the program or start a new model, you are presented with the following options: Draw Beams, Draw Retaining Walls, Draw Slabs, Generate a Modeland Go to the File Open Dialog

The first group of options is to help you to create a new model by drawing members or plates or by generating with a template drawing beams, walls or slabs or by using a template to start your model. The second group of options helps you open existing files by showing you the most recent files or letting you find the model you want in the Open File Dialog.

Creating a New Model

- On the RISA toolbar click the **New Model** button. You will be prompted to 'Save' if you have made any changes to the current model that have not yet been saved.

Opening an Existing Model

1. On the RISA toolbar click the **Open File** button.
2. In the **Look In** field click the drive, folder, or location that contains the file.
3. In the folder list, double-click the folders until you open the folder that contains the file you want.
4. Double-click the file you want to open.

Tip

- To open a model you have used recently, click the **File Menu** and select from the file names at the bottom of the menu.

Saving a New Untitled Model

1. On the RISA toolbar click the **Save Model** button.
2. To save the model in a different folder, click a different drive in the **Save in** field, or double-click a different folder in the folder list. To save the model in a new folder click the **Create New Folder** button.
3. In the **File name** field type a name for the document. You can use long, descriptive filenames if you want.
4. Click **Save**.

Saving an Existing Model

1. On the RISA toolbar click the **Save Model** button.

Exporting to Excel Spreadsheet

You may export Result spreadsheets into Excel spreadsheet:

1. On the **File** menu click **Export**.
2. Choose **Excel File**.
3. Select export options , you can select to export an entire RISA result spreadsheet or an individual RISA result section.
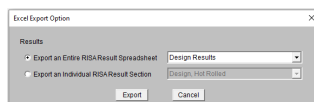
4. Select the folder to export the spreadsheet and export.

Automatic Backup

RISAFoundation provides an automatic backup system for systematically backing up your models. The purpose of this is to provide a means for you to recover your data in the event RISAFoundation or your computer closes unexpectedly. As you work within RISAFoundation your model is saved with a "_backup" suffix to a model backup folder. By default, this automatic save operation takes place every 5 minutes, but you can alter the timing and location with the **Application Settings** on the **Tools Menu**. Backups for the last 100 models that you worked on are saved, although you can control how many models are backed up with the **Application Settings** option on the **Tools Menu**.
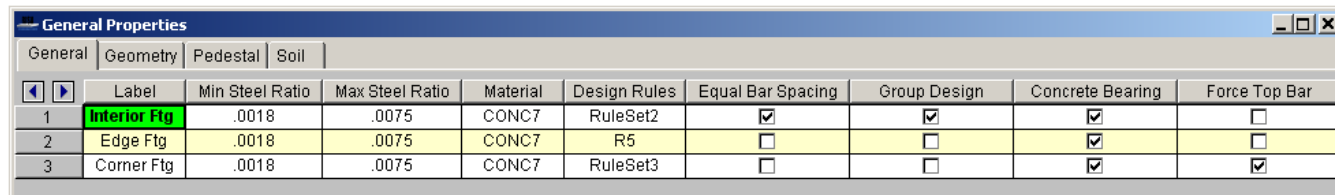
If RISAFoundation does close unexpectedly, the next time you start the program the following message will appear:
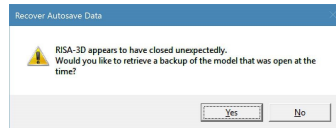
Footings - Design

Footing General Properties

The General tab contains material properties and other specific toggles.

Reinforcement Ratios

Acceptable steel ratios are controlled by ACI 318-14 Section 24.4.3 (ACI 318-11 Section 7.12.2.1) which stipulates that the minimum steel ratio for Grade 40 or 50 steel is .0020. For Grade 60 the ratio is 0.0018. The ratio cannot be less than 0.0014 for any grade of steel.

In the ACI 318-11 Section 7.12 explicitly states "gross concrete area" is to be used. Thus, for this and earlier versions of the code, RISA requires that the entire cross-section (top and bottom reinforcement together) meets this requirement.

For the ACI 318-14 code, the flexural minimum section has been updated to require this area of steel in each face where flexural steel exists. Therefore, for this version of the code, RISA enforces this minimum for each face individually. See ACI 318-14 Sections 8.6.1.1 and 7.6.1.1 for more information.

In addition, the codes limit the maximum steel that can be used. The 1995 code and 1999 code limit the steel ratio to 75% of the balanced condition. The 2002 and newer codes instead limit the minimum strain on the reinforcing steel to 0.004. Foundationdoesn't specifically enforce these maximums and instead relies on this Max Steel Ratio separately for each face.

Material

This allows you to define the material from the Materials spreadsheet.

Design Rules

This allows you to define the design rule from the Design Rules spreadsheet.

Force Bars to be Equally Spaced

Checking the **Equal Bar Spacing** option allows you to indicate that you do not want to specify banded reinforcement. If you check this box, RISAFoundation will determine the tightest spacing required per ACI 318-14 Section 13.3.3.3 (ACI 318-11 Section 15.4.4) and then use that spacing throughout the footing.

Enveloping Multiple Footing Designs

Checking the box in the **Group Design** column will cause all footings assigned to the current footing definition to use the same final size. The largest required size within the group of footings will be used for all footings within the group. The maximum required reinforcement for the footings and pedestals will also be enveloped.

Concrete Bearing Check
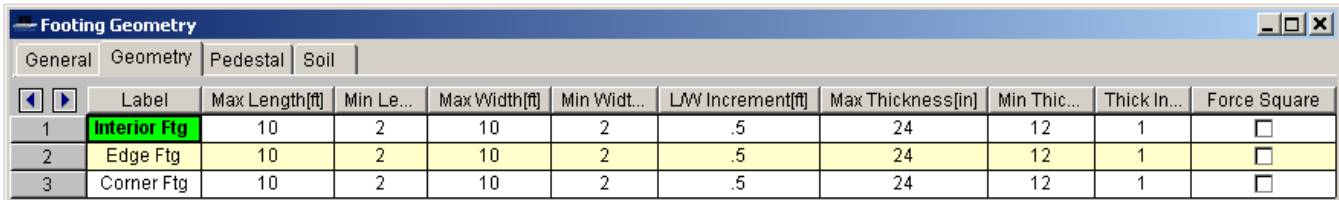
Checking the box in the **Concrete Bearing** column will force the program to check the concrete bearing failure between the footing and pedestal.

Force Top Bars

The **Force Top Bars** option allows the user to to indicate that top bars are required for this footing even for cases where the footing does not experience significant uplift or negative moment. One reason for checking this box would be to spread out the Temperature / Shrinkage reinforcement betwee the top and bottom layers of steel.

Footing Geometry

The Geometry tab contains the basic information that will control the size and thickness of the footing.

	Label	Max Length[ft]	Min Le...	Max Width[ft]	Min Widt...	L/W Increment[ft]	Max Thickness[in]	Min Thic...	Thick In...	Force Square
1	Interior Ftg	10	2	10	2	.5	24	12	1	☐
2	Edge Ftg	10	2	10	2	.5	24	12	1	☐
3	Corner Ftg	10	2	10	2	.5	24	12	1	☐

These parameters define the minimum and maximum length, width and thickness for the footing. When RISAFoundation sizes the footing, it optimizes for the smallest footing that fits within the parameters and does not exceed the allowable soil bearing pressure. RISAFoundation favors footings with the lowest length/width ratio (as close to a square as possible).

If you are investigating an existing footing size that you don't want RISAFoundation to change, you may either specify the same maximum and minimum length, width and/or thickness OR specify the maximum values and set the design increments to zero.

Note:
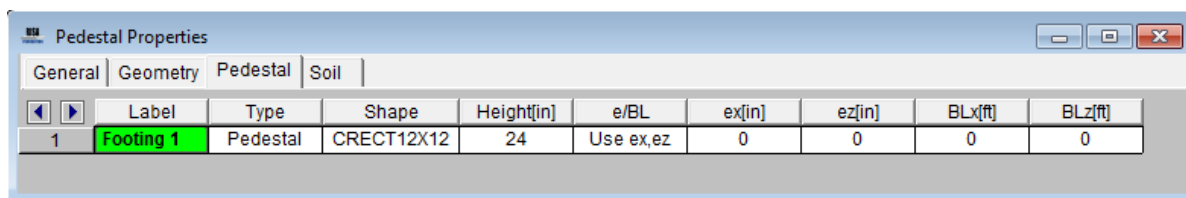
- The min/max length is always defined as the dimension in the local z axis. The min/max width is always defined as the dimension in the local x-axis.

Force Footing to be Square

Checking the box in the **Force Square** column will restrict the design to square footings only.

Footing Pedestal

The Pedestal tab contains the basic information that will control the width and height of the pedestal. It will also control the location of the pedestal within of the footing if you have an eccentric footing.

	Label	Type	Shape	Height[in]	e/BL	ex[in]	ez[in]	BLx[ft]	BLz[ft]
1	Footing 1	Pedestal	CRECT12X12	24	Use ex,ez	0	0	0	0

The pedestal geometry is entered in these fields. The type can be specified as either a pedestal or a post. The shape cell will prompt the following dialog box to specify the dimensions of the pedestal:
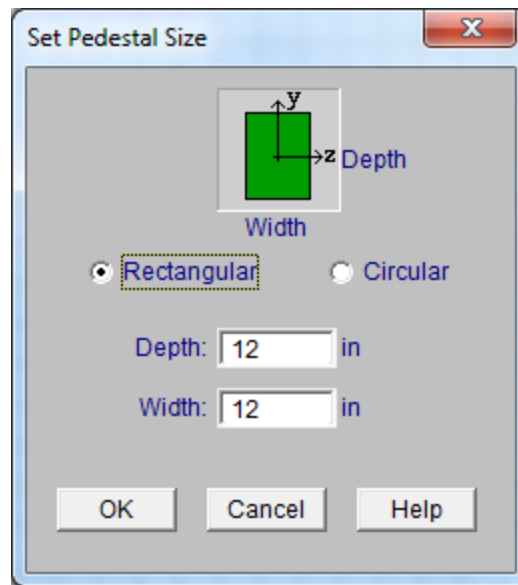
The height is the distance from the upper surface of the footing to the top of the pedestal. The pedestal location may be controlled in the Boundaries and Eccentricities section discussed below.

Boundaries and Eccentricities

When specifying an eccentric pedestal the **eX** and **eZ** values are the offsets from the center of the footing in the local X and Z directions, respectively. These values may be either positive or negative.

You may use the **BLx** or **BLz** entries to set boundary lines for the footing in one or two directions or you may specify the pedestal eccentricity from the centroid of the footing. If you set boundary lines, RISAFoundation will keep the sides of the footing from violating the boundary. These values may be either positive or negative.
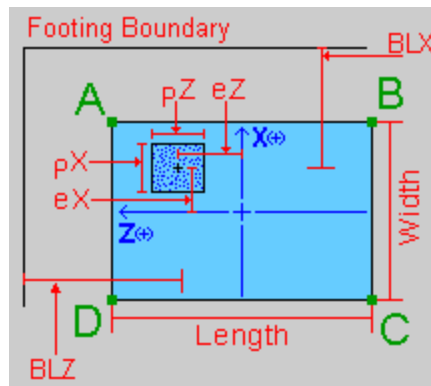
Soil Properties

The Soil Properties tab contains the basic information related to design for soil bearing.

	Label	Overburden[ksf]	Passive[k]	Friction Coeffi...	Gross/Net
1	Footing 1	.1	0	.3	Gross
2	Footing 2	.125	5	.3	Gross
3	Footing 3	.15	10	.3	Net

Soil Bearing

Overburden

Overburden is the weight of soil or other dead loads (e.g. pavement) which exist above the footing in its final (constructed) condition. RISAFoundation applies the overburden pressure uniformly on the footing, except for at the pedestal.

Passive Resistance of Soil

For additional sliding resistance you may enter the passive resistance of the soil as a lump sum value. RISAFoundation applies this value in both directions.

Coefficient of Friction

RISAFoundation multiplies this coefficient of friction by the total vertical forces to determine the soil sliding resistance.

Gross/Net

In RISAFoundation, the Allowable Soil Bearing value is controlled by the Soil Region. This value is always taken as a Gross allowable soil pressure by default.

Allowable Soil Bearing values are typically supplied by the Geotechnical Engineer as either *Gross* or *Net* allowable values. The soil bearing check uses the same load to check soil bearing (Footing Self Weight + Overburden + Applied Load) regardless of whether Gross or Net is specified.

When a Gross soil pressure is specified the load is compared directly against the Bearing Allowable (user-entered).

When a Net soil pressure is specified the load is compared against a modified soil capacity as shown below:

Soil Capacity = Bearing Allowable (user-entered) + Footing Self Weight + Overburden (user-entered)

If the sum of Footing Self Weight and Overburden *do not* equal the soil weight which was present above the footing/soil interface strata prior to excavation then the modified soil capacity provided by the program *is incorrect*. In this circumstance the Net Allowable Soil Bearing value should be specified as a Gross value in the program, and the magnitude of Bearing Allowable should be increased by that pre-excavation soil weight.

Local Axes

The "A,B,C,D" notation provides a local reference system for the footing. The local z axis is defined as positive from B towards A, and the local x axis is defined as positive from D towards A. The origin for the local axes is the geometric center of the footing.
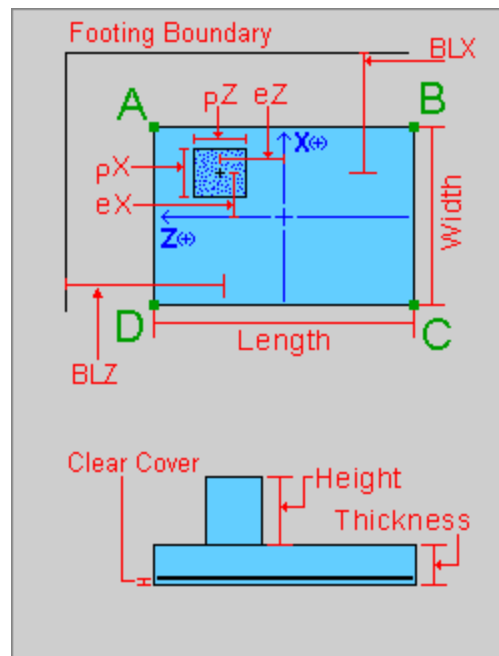
It is important to realize that the local footing axes do NOT necessarily correspond to the Global axis of the RISA model or the local axis of the column member.

Limitations

Non-Linear Effects

The Joint Reaction results can differ from the design results used for the OTM safety factor calculation. The reason for this is that the OTM safety factor needs to determine whether each component (P, V, or M) of each load creates a stabilizing or de-stabilizing moment.

Footing Results

Upon completion of the analysis and footing design the results are presented in results spreadsheets and a detail report for each footing. The detail report may be may customize and printed. You may choose which sections you wish to view or print by clicking the Report Options button. The results sections are shown below.

You can control what information is included in the detail report by clicking the Report Options button at the top of the detail report. Available report sections are listed for you to check those that you wish to include.

Solution Methodology

It is important to note that if the location of the load resultant is not within the foot print of the footing then the footing is unstable. In this instance the user is warned of the instability which must be corrected by increasing the footing size or overturning resisting force.

The method RISAFoundation utilizes to obtain the soil pressure distribution provides an exact solution. RISAFoundation assumes the footing behaves as a rigid body and satisfies equilibrium equations. The contribution of the soil to the above equations is determined through direct integration.

The calculation of the soil bearing profile for a two way (biaxial) load state is the most complex calculation RISAFoundation makes. The soil is considered to resist compression only. If uplift is present, RISAFoundation allows that part of the soil to "let go", so no soil tension is ever part of the soil bearing solution.

Soil bearing controls the plan dimensions of the footing; i.e. the footing is sized such that the actual maximum soil bearing pressure does not exceed the user defined allowable soil bearing. The calculations use the service load combinations only. When an allowable bearing increase factor is specified, the allowable soil bearing is scaled up by the factor (ABIF), typically a 1/3 increase.
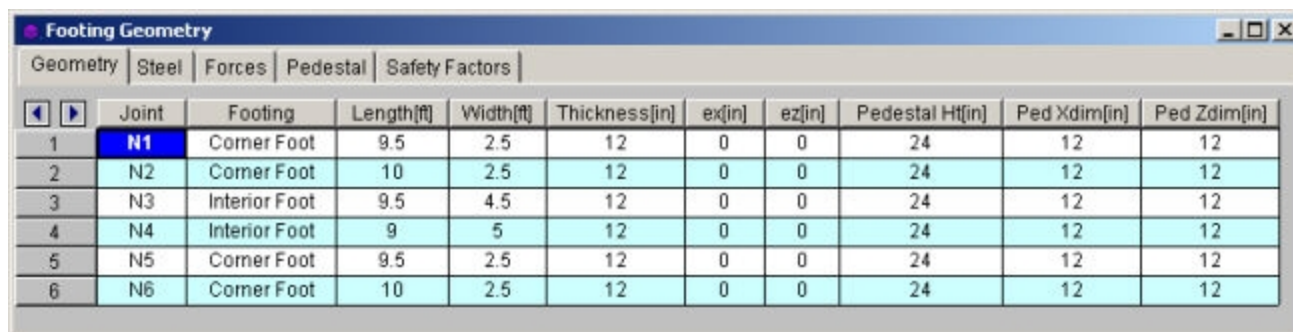
Once the soil stress distribution has been solved, the load on the footing is known at every point. Direct integration on the contributing portion of the soil stress provides the shears and moments at the lines of interest. See the Results section for more information on the calculations that RISAFoundation performs.

Sketch and Details

Note:

- ⊡ - This button will allow you to take a snapshot of the current detail report you are viewing so that it can be added to a report. View the Printing topic for more information.

RISAFoundation will optimize the dimensions of the footing for optimum concrete volume. The final footing geometry is presented in tabular form within Footing Results spreadsheet and graphically in the Footing detail report. The orientation of the footing is based on the footing's local x and z axes, respectively.

	Joint	Footing	Length[ft]	Width[ft]	Thickness[in]	ex[in]	ez[in]	Pedestal Ht[in]	Ped Xdim[in]	Ped Zdim[in]
1	N1	Corner Foot	9.5	2.5	12	0	0	24	12	12
2	N2	Corner Foot	10	2.5	12	0	0	24	12	12
3	N3	Interior Foot	9.5	4.5	12	0	0	24	12	12
4	N4	Interior Foot	9	5	12	0	0	24	12	12
5	N5	Corner Foot	9.5	2.5	12	0	0	24	12	12
6	N6	Corner Foot	10	2.5	12	0	0	24	12	12

Footing Sketch
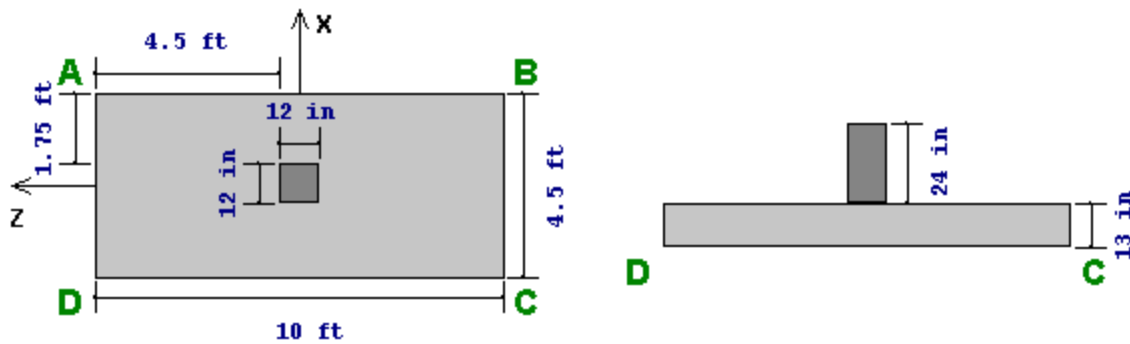
The detail report presents a sketch of the footing with dimensions (using the A,B,C,D reference system discussed in the geometry section).

Footing Details

Footing details, shown below, may be included in the report. The details may also be exported to a DXF CAD file using the Export option on the File menu.
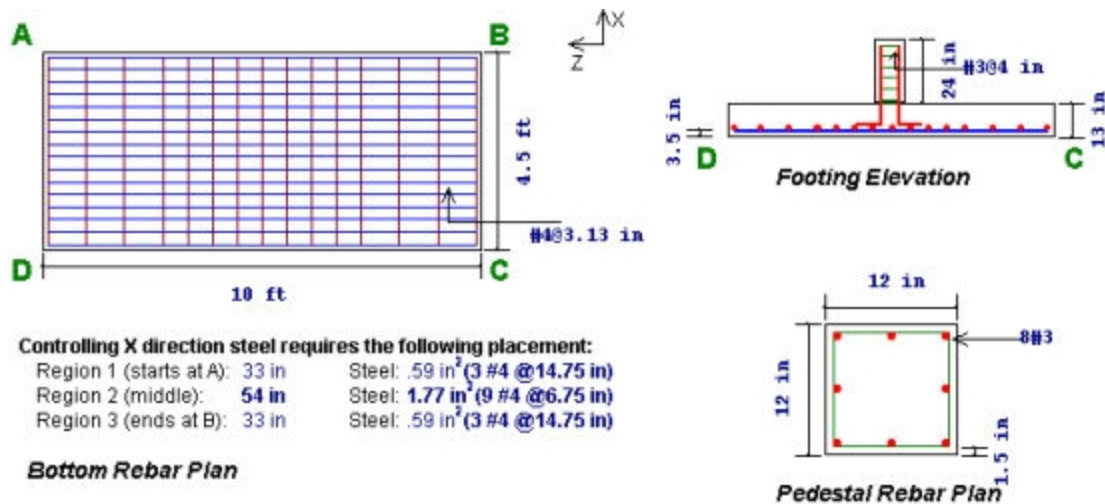
You may export the footing details to a DXF file for use in your CAD drawings.

Soil Bearing Results

RISAFoundation displays the Soil Bearing check in a spreadsheet format on the Code Checks tab of the Footing results spreadsheet. The RISAFoundation detail report presents the soil bearing results for all service combinations. A soil bearing contour plot for each combination can also be displayed.

Description	Categories and Factors	Gross Allow.(ksf)	Max Bearing (ksf)	Max/Allowable Ratio
ASCE 2 (b)	1DL+1LL+1SL	2.2483	2.2197 (C)	.9873
ASCE 2 (c)	1DL+1LL+1RL	2.2483	2.2197 (C)	.9873
ASCE 4 (a)	.6DL+1.5WL	2.2483	1.2717 (D)	.5656
ASCE 4 (b)	.6DL-1WL	2.2483	1.6084 (C)	.7154

1DL+1LL+1SL
QA: .5698 ksf
QB: 1.4164 ksf
QC: 2.2171 ksf
QD: 1.373 ksf
NAZ: 188.7615 in
NAX: 315.0307 in

1DL+1LL+1RL
QA: .5698 ksf
QB: 1.4164 ksf
QC: 2.2197 ksf
QD: 1.373 ksf
NAZ: 188.7615 in
NAX: 315.0307 in

.6DL+1.5WL
QA: .4224 ksf
QB: 0 ksf
QC: .5171 ksf
QD: 1.2717 ksf
NAZ: 121.3328 in
NAX: 170.6975 in

.6DL-1WL
QA: 0 ksf
QB: .7986 ksf
QC: 1.6084 ksf
QD: .6296 ksf
NAZ: 118.3107 in
NAX: 226.4333 in

RISAFoundation displays color contoured images of the soil bearing profile along with the bearing pressure magnitudes at each footing corner (QA through QD). Also provided is the location of the neutral axis (the line between compressive stress and no stress due to uplift). The location is given by NAX and NAZ values. These are the distances from the maximum stress corner to the neutral axis in the X and Z directions respectively.

Footing Flexure Design

The flexural design results include the maximum moment and required steel for each direction for each design load combination. The controlling required steel values are highlighted. This information is reported in the Forces and Steel tabs of the Footing Results spreadsheet and also from within the Footing Detail report. The direction of the xx and zz moments is based on the orientation of the footing and the footing's local x and z axes, respectively.

Minimum flexural steel is listed here (along with temperature and shrinkage minimum) even though temperature and shrinkage or 4/3 As_required will normally control the design of the footing steel.

Footing Flexure Design (Bottom Bars)

As-min x-dir (Top Flexure): **22.945 in^2**   As-min x-dir (T & S): **13.4784 in^2**
As-min z-dir (Top Flexure): **22.945 in^2**   As-min z-dir (T & S): **13.4784 in^2**
As-min x-dir (Bot Flexure): **22.945 in^2**
As-min z-dir (Bot Flexure): **22.945 in^2**

Description	Categories and Factors	Mu-xx UC Max	Mu-xx (k-ft)	z-Dir As Required (in^2)	z-Dir As Provided (in^2)	Mu-zz UC Max	Mu-zz (k-ft)	x-Dir As Required (in^2)	x-Dir As Provided (in^2)
ACI 9-1	1.4DL+1.7LL	.01346	35.27	.178	13.548	.01787	46.82	.237	13.548
ACI 9-2	1.05DL+1.275LL+1...	0	0	0	13.548	.01121	29.38	.148	13.548
ACI 9-3	.9DL+1.3WL	0	0	0	13.548	.00666	17.46	.088	13.548
ACI 9-2E	1.05DL+1.28LL+1.4..	.01441	37.77	.191	13.548	.02241	58.72	.297	13.548
ACI 9-3E	.9DL+1.43EL	.00709	18.57	.094	13.548	.0126	33.01	.167	13.548

Footing Flexure Design (Top Bars)

Description	Categories and Factors	Mu-xx (k-ft)	z Dir As (in²)	Mu-zz (k-ft)	x Dir As (in²)
SW+OB	1SW+1OB-(ACI 9-3,ACI 9-3)	69.4814	0	139.4879	0

Moment Capacity of Plain Concrete Section Along xx and zz= **816.1157k-ft,816.1157k-ft** Per Chapter 22 of ACI 318.

To select the required flexural reinforcing steel for the footing, RISAFoundation considers moments at the face of the pedestal on all four sides. The soil bearing profile from the edge of the footing to the face of the pedestal is integrated to obtain its volume and centroid, which are in turn used to calculate the moments and shears. This moment is then used to calculate the required steel using standard flexural methods.

RISAFoundation reports the reinforcing steel as a total required area for each direction. ACI 318 also requires that short direction reinforcing in a rectangular footing be concentrated under the pedestal (this is referred to as banding), per the requirements of ACI 318-14 Section 13.3.3.3 (ACI 318-11 Section 15.4.4). RISAFoundation will report the proper steel distribution for this circumstance.

The minimum steel required for the footing is maintained based on the ratio entered on the General Tab of the Footing Definitions spreadsheet.

Top reinforcing may be required when the footing goes into partial uplift resulting in a negative moment due to soil overburden and footing self weight. When this happens, RISAFoundation will first check to see if the plain (unreinforced) concrete section is sufficient to resist the negative moment per ACI 318-14 Chapter 14 (ACI 318-11 Chapter 22). If not, then top reinforcing will be provided. The Load Combination shown under "Categories and Factors" shows (in parentheses) the controlling Z direction, then the controlling X direction for Top bar design.

Footing Shear Check

An important part of footing design is insuring the footing can adequately resist the shearing stresses. Shear checks for each design combination are shown below. RISAFoundation checks punching shear as well as one way (beam) shear in each direction. The capacity and shear check results are presented, both as ultimate shear and as demand vs. capacity ratios where the actual shear is divided by the available shear capacity, thus any ratio above 1.0 would represent failure.

This information is displayed on the Forces tab of the Footing results spreadsheet also from within the Footing Detail report.

One-way and two-way shear are checked per ACI 318-14 Section 13.2.7 (ACI 318-11 Section 15.5). RISAFoundation checks shear assuming only the concrete resists the applied shear; the contribution of the reinforcing steel to shear resistance is ignored. Two-way punching shear checks for the pedestal are reported on the Pedestal tab of the footing results spreadsheet.

One way shear is calculated for a "cut" through the footing a distance "d" from the face of the footing, where "d" is the distance from the top of the footing to the centerline of the reinforcing steel. For RISAFoundation, the "d" distance value is equal to the footing thickness minus the rebar cover and one bar diameter (we do not know how the r/f is placed so we use the interface between the bars in each direction. The soil bearing profile from the edge of the footing to the cut location is integrated to obtain the total shearing force that must be resisted by the concrete. This is done for both directions (width cut and length cut) on both sides of the pedestal for a total of four one way shear calculations.

Pedestal Design

RISAFoundation will design the longitudinal and shear reinforcement for rectangular pedestals. This information is presented in tabular form within Footing Results spreadsheet and graphically in the Footing detail report.

Pedestal Design

Shear Check Results (Envelope):

	Vc	Vs	Vu	Vu/phi*Vn	phi
Shear Along x Direction:	0	43.9025	26.58	.7123	.85
Shear Along z Direction:	0	43.9025	0	0	.85
Pedestal Ties : #3 @ 3 in					

Bending Check Results (Envelope): PCA Load Contour Method (for biaxial)

Unity Check	: 1.5015	Phi	: .9	Parme Beta	: .65
Pu	: -38.599 k	Mux	: 0 k-ft	Muz	: 53.16 k-ft
Pn	: -28.5637 k	Mnx	: NC	Mnz	: 39.339 k-ft
Governing LC	: 22	Mnox	: NC	Mnoz	: NC
Pedestal Bars	: 20 #3	% Steel	: 1.534		

Pedestal Flexure

For flexure RISAFoundation uses a rectangular stress block. For biaxial bending situations the load contour method is used. RISAFoundation does not account for slenderness effects.

For flexure design RISAFoundation presents the required longitudinal reinforcement assuming an equal distribution of bars around the perimeter of the pedestal. Supporting information includes the axial and bending contributions to resistance Pn, Mnx and Mnz. The total factored axial and bending loads Pu, Mux and Muz are also listed. The contour method beta factor and Mnox and Mnoz are given, the moments representing the nominal uniaxial bending capacity at the axial load Pn. Finally, the Phi factor and the resulting unity check (demand vs. capacity) ratio is given.

For vertical reinforcing design, note that RISAFoundation will meet the provisions of the ACI 318-14 Section 16.3.4.1 (ACI 318-11 Section 15.8.2.1) that states that in pedestals you must have a minimum reinforcement of 0.5% of the pedestal cross section.

Pedestal Shear

For shear design in either direction RISAFoundation presents the required tie spacing. Also given are the concrete and steel contributions to resisting shear Vc and Vs. The total factored load Vu is listed. Finally, the Phi factor and the resulting demand vs. capacity ratios are given.

Pedestal Punching Shear

Punching shear is calculated for a perimeter a distance d/2 from the pedestal all the way around the pedestal. The soil bearing profile is integrated, minus the volume bounded by this perimeter, to obtain the total punching force to be resisted. For cases where the pedestal is near a corner of the footing, RISAFoundation considers the punching perimeter to extend out to the footing edge if the pedestal is within a distance d from the edge of the footing.

RISAFoundation presents the Vu and Vc values for punching and both one way shears as well as the shear demand vs. capacity ratio. So as long as this ratio is less than 1.0 the footing is adequate for shear. The **Forces** tab of the **Footings** spreadsheet gives the two way and one way shear capacities in tabular form.

Stability Results

RISAFoundation will check the stability results for overturning and sliding. This information is presented in tabular form within Footing Results spreadsheet and graphically in the Footing detail report.

Overturning Check (Service)

Description	Categories and Factors	Mo-XX (k-ft)	Ms-XX (k-ft)	Mo-ZZ (k-ft)	Ms-ZZ (k-ft)	OSF-XX	OSF-ZZ
ASCE 2 (b)	1DL+1LL+1SL	13.7738	122.2019	0	188.8576	8.8721	NA
ASCE 2 (c)	1DL+1LL+1RL	13.7738	122.2019	0	188.8576	8.8721	NA
ASCE 4 (a)	.6DL+1.5WL	33.392	62.8353	21.6066	89.3476	1.8817	4.1352
ASCE 4 (b)	.6DL-1WL	17.963	67.1336	0	103.752	3.7373	NA

Mo-XX: Governing Overturning Moment about AD or BC
Ms-XX: Governing Stablizing Moment about AD or BC
OSF-XX: Ratio of Ms-XX to Mo-XX

Sliding Check (Service)

Description	Categories and Factors	Va-XX (k)	Vr-XX (k)	Va-ZZ (k)	Vr-ZZ (k)	SR-XX	SR-ZZ
ASCE 2 (b)	1DL+1LL+1SL	13.3859	17.5139	4.9188	17.5139	1.3084	3.5606
ASCE 2 (c)	1DL+1LL+1RL	13.3859	17.5139	4.9188	17.5139	1.3084	3.5606
ASCE 4 (a)	.6DL+1.5WL	13.1902	8.9532	4.4009	8.9532	.6788	2.0344
ASCE 4 (b)	.6DL-1WL	13.1858	11.4855	6.3512	11.4855	.8711	1.8084

Va-XX: Applied Lateral Force to Cause Sliding Along XX Axis
Vr-XX: Resisting Lateral Force Against Sliding Along XX Axis
SR-XX: Ratio of Vr-XX to Va-XX

Overturning Check

Overturning calculations are presented as shown in the figure above. For all service combinations both the overturning moments and the stabilizing moments are given along with the calculated safety factor.

Overturning moments are those applied moments, shears, and uplift forces that seek to cause the footing to become unstable and turn over. Resisting moments are those moments that resist overturning and seek to stabilize the footing. The overturning safety factor (OSF) is the sum of resisting moments divided by the sum of overturning moments. Most codes require that this factor be greater than 1.5. Overturning safety factor calculations are based on the service load combinations only and are calculated in both the X and Z directions.

Overturning results from applied moments and applied shears and the summation of all these forces becomes the overall "overturning moment".

Note

- The OSF is calculated separately in both the X and Z directions as: resisting moment / overturning moment. Overturning is evaluated about all edges of the footing and the worst case in each direction is reported.
- The width and length of the footing are only increased or optimized to prevent stability checks failure if the "Optimize for OTM/Sliding" check box on the Design tab of the Model Settings has been checked.
- For Safety Factors greater than 1000, the footing is stable and the program will report 'NA' as the result.

Sliding Check

RISAFoundation provides sliding checks for all of the service combinations.

The applied forces and the resisting forces, due to friction and passive soil resistance, are calculated and displayed. The final result is a sliding ratio of resistance vs. applied force that indicates a footing susceptible to sliding when the ratio is less than 1.0.

Concrete Bearing Check

The concrete bearing and demand /capacity ratios for each design combination are presented.

IMPORTANT: These concrete bearing checks are only based on the vertical applied loads, i.e. bending effects that increase or decrease bearing area and pressure are not considered.

The concrete bearing check in RISAFoundation checks whether the footing concrete is adequate to resist in bearing the vertical loads imposed on the pedestal. These calculations are per ACI 318-14 Section 22.8 (ACI 318-11 Section 10.14). The results are presented as the demand vs. capacity ratio.

Footings - Combined

One of the strengths of the RISAFoundation program is the ability to easily create and design combined footings. There are many different types of combined foundations. Therefore, this discussion is meant only to address the basic cases. However, the principals discussed here are valid for more complex arrangements as well.

Strap Footings

One of the most common types of spread footings is the simple case of multiple spread footings that are connected together with a grade beam. The beam allows the two footings to share the load between them based on their relative stiffness and the stiffness of the connecting beam. A simple example of a Strap Footing is shown below:
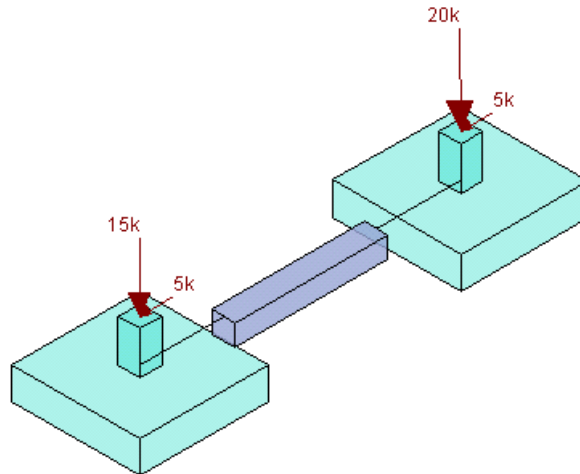
It's important to realize that each footing will still be designed as a single spread footing. It's just that the presence of the grade beam allows a re-distribution of the forces between the footings that can result in a more economical foundation. Refer to the sections on individual footing design and on Footing Design Results for more information. Refer to the Soil Parameters section for more information about how the program is using the soil parameters to assign vertical and rotational springs to the individual footing supports.

Using Slab Foundations to Model Combined Foundations

Rather than using a grade beam or strap beam to connect two individual footing together, it may be preferable to use less forming and pour a small mat foundation. While many people would refer to these types of foundations as combined footings, the program will treat them as small mat foundations.
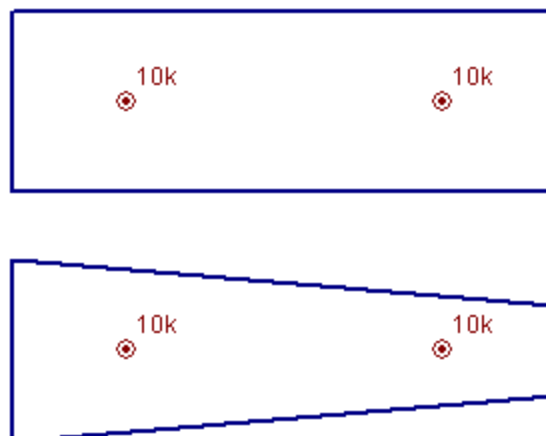
Because they are treated as mat foundations you will have to specify Design Strips or Design Cuts to get you reinforcement design. See the sections on Slab Design and on Design Tools for more information.

Wall Footings

Please see the [Modeling Tips](#) topic for more information on how to model a wall footing.

Footing Stability and Overturning Calculations

Calculation of OTM Stability Ratio

One of the important results from any footing analysis is a ratio of the stabilizing moments to the de-stabilizing moments. This is referred to as the Stability Ratio or the Safety Factor for overturning. RISAFoundation calculates this value differently than the way many engineers would traditionally approach a hand calculation. The RISA method is a bit more complicated, but will provide a more realistic representation of the true safety factor versus overturning whenever uplift loads are present. A comparison of the two methods is provided below.

The overturning safety factor (OSF) is the sum of resisting moments divided by the sum of overturning moments. RISAFoundation may provide default input for safety factors, but they need to be reviewed and updated if needed by project EOR per code requirements, foundation types, and project needs. Overturning safety factor calculations are based on the service load combinations only and are calculated in both the X and Z directions.

Traditional / Simplistic OTM Calculation

Most hand calculations are done on the assumption that all vertical loads are stabilizing loads that are only capable of producing a stabilizing moment whereas all lateral shear and moments will act to de-stabilize the structure. As an example consider the case where you have the following Loads applied to a 4x4 footing with a pedestal that extends 2 ft above the bottom of the footing.

Force Effect	Dead Load	Wind Load
Axial	25 kips	+/- 10 kips
Shear	0	5 kips

In the traditional method, the axial forces are combined together to form a total vertical load. Since the TOTAL EFFECT of the axial loads is considered a stabilizing effect the resisting moment is considered to be $P*L/2$. Since the only de-stabilizing effect is assumed to be the shear force the de-stabilizing moment would be calculated as $V*H$. These calculations are summarized in the table below.

Wind Effect	Stabilizing Moment	DeStabilizing Moment	OTM Safety Factor
Downward	70 k-ft	10 k-ft	7.0
Uplift	30 k-ft	10 k-ft	3.0

True Safety Factor Method (used by RISAFoundation)

The True Safety Factor method used by RISAFoundation involves taking a look at EVERY component of every load and deciding whether it has a stabilizing or a de-stabilizing effect. Overturning moments are those applied moments, shears, and uplift forces that seek to cause the footing to become unstable and turn over. Resisting moments are those moments that resist overturning and seek to stabilize the footing. These overturning checks are performed for overturning about each edge of the footing. This is important for footings with eccentric pedestals where the vertical loads will have different stabilizing effects for each face.

By summing up all the resisting moments and comparing them to the sum of all the overturning moments we get a better sense of the true tendency of the structure to overturn. Taking the example listed above, it is clear that when

the wind forces act in compression, the two method will produce identical results. That means that if the Wind shear is increased by a factor of 7.0, then the footing would be in net overturning.

However, the stabilizing moment in the uplift case would be viewed as the FULL dead load * L/2 rather than the reduced axial load used in the traditional method. Similarly, the de-stabilizing load would be viewed as the sum of the moments about the edge of the footing. This includes the de-stabilizing effect of the wind uplift and is equal to P_wind* L/2 + V*H.

The OTM Safety Factor calculations for the True Safety Factor Method are summarized in the table below:

Wind Effect	Stabilizing Moment	DeStabilizing Moment	OTM Safety Factor
Downward	70 k-ft	10 k-ft	7.0
Uplift	50 k-ft	30 k-ft	1.67

Comparing the Two Methods

Which is a more accurate representation of the safety factor of stability ratio for the uplift case? The best way to judge this is to multiple the wind load by a factor equal to the Safety factor and to see if the footing still works. If the wind effects (shear and uplift) are multiplied by a factor of 3.0 as given by the traditional method, then the footing will be unstable due to net uplift and net overturning. The Safety Factor would be calculated a negative value which has little physical meaning. If the wind effects (shear and uplift) are multiplied by a factor of 1.67 as given by the True Safety Factor method, then the stabilizing moment will equal the de-stabilizing moment and the safety factor becomes 1.0 as predicted.

Calculation of Moment and Shear Demand for Unstable Footings

Older versions of the program could not handle situations where the Ultimate Level load combinations resulted in net overturning of the Footing. This is because the previous versions always relied on the soil bearing pressure calculations to derive the moment and shear demand in the footing. For net overturning cases, the current version of the program assumes that the design shear and moment can be based on the total vertical load and the net eccentricity of that load.
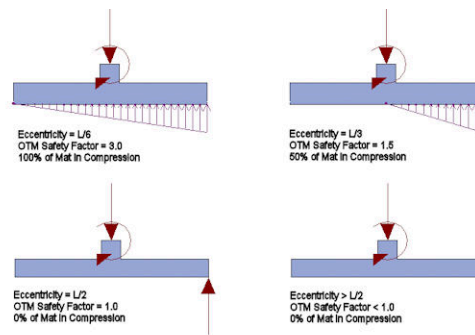

In these cases, where the resultant is off the footing, the design shear force in the footing will be constant equal to the net resultant minus any effects of self weight or over burden. Also, the basis for the design moment at the face of the pedestal will be the load resultant times the distance from the resultant to the face of the pedestal. Any negative moment effect due to the self weight of the footing slab and overburden would be subtracted out when determining the final design moment.

Generation

Circular Slab Generator

The circular slab generator allows users to quickly lay out a circular mat slab for design. There are also circular line and area load generation capabilities that allow you to automatically create typical loading for round structures; tanks for example.
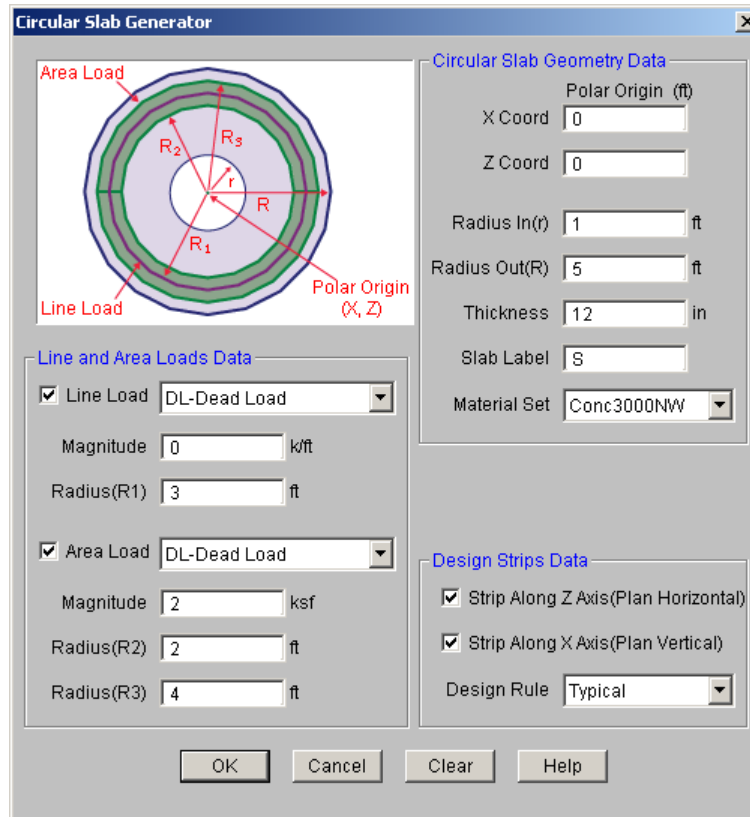
The polar origin defines the center coordinates of the slab. The inner radius, outer radius, thickness and material for the slab are also defined here.

The load generation information allows you to input the load category, the magnitude and the geometry of the loading.

Note:

- The program creates interior nodes in the slab for application of line and area loads. These nodes can be manipulated for line loads in the Line Loads spreadsheet. The generated Area Loads need to be deleted and redrawn to manipulate.

There is also a design strip generator that allows you to define a strip over the entire slab in either of the global axes and also choose the design rule that defines the reinforcement for these strips.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Circular Slab Generator**.

(Global) Model Settings

The **Model Settings** may be accessed through the menu or toolbar buttons highlighted below.
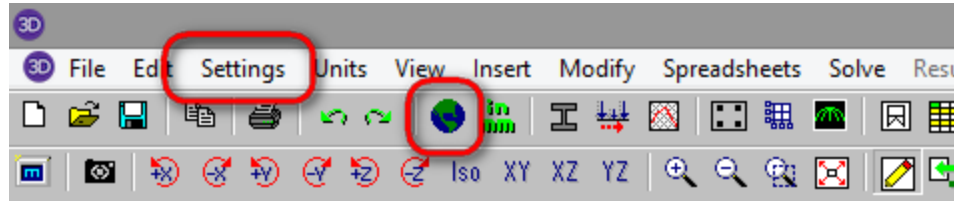
The Model Settings are settings specific to the model which is currently open. The information stored here is retained with the model, and will be retained even if the model is opened on a different computer.

You may save any of the settings in the Model Settings as the default setting, such that new models you create will begin with the settings that you saved as Defaults. To do this, simply enter the information that you want to save and click the **Save as Defaults** button.

Description

The entries under the **Description** tab are used to enter descriptive information such as a title for the particular model being defined, the name of the designer and a job number. The title may then be printed at the top of each sheet of the output, and on the graphic printouts of the model. Note that for any of these tabs, you can click on **Save as Defaults** and then your settings will be saved the next time you open the program.

The **Checked By** field is intended for the model reviewer to initial upon completion of a model review. Their initials will then be printed on each sheet of the output.

The **Notes** field is intended for information about the model that may be useful for the engineer or reviewer to refer to.

Solution

The entries under the **Solution** tab are used to control settings that affect the general solution of the model.

Number of Sections controls how many places you receive **reported** beam force, stress, and deflection results. This only affects the amount of data displayed in the results spreadsheets, and has no effect on the solution of the model or the code checks. See Printing and Member Results for more information.

Number of Internal Sections controls how many places along each beam the software calculates and stores results such as deflections and code checks. The beam force diagrams displayed in the model view and the detail plot are also drawn from these results. Increasing this value means that the program will make more "cuts" along the beam's length, which means it is more likely to hit the theoretical maximum and minimum values for code checks.

Note:

- Number of Sections cannot exceed 20. Also, Number of Internal Sections cannot be less than twice the Number of Sections. If unacceptable values are entered for either of these fields the program will automatically reset them to acceptable values.

In previous versions, the default **Subgrade Modulus** and **Allowable Bearing** were entered in the Model Settings. In version 10.0 and higher, the default subgrade modulus and allowable bearing pressure are entered in the **Soil**

Definitions spreadsheet. There is a check box in the Soil Definitions spreadsheet to determine which soil definition will be used as the default. The default soil definition properties will be applied to the entire model except where another soil region is drawn in the model. It is not necessary to draw a soil region for the default soil definition.

The **Mesh Size** is used to control the coarseness or fineness of the slab mesh when RISA auto-meshes slabs during solution.

The **"Use Cracked Sections"** check box will allow the program to use the cracked moment of inertia and reduce the bending stiffness of beams and slabs. The reduction factor can either be user defined (by the use of specifying the Icr Factor), or the program will automatically generate using the program defaults.

The **Maximum Iterations** defines the maximum number of iterations the program will undergo for tension and compression springs in a model. Soil regions are always modeled as compression springs, but piles may be tension springs, compression springs or both. In the first iteration the program assumes a compression spring for every support node in the model. The program then checks the forces (tension or compression) in these springs against the soil moduli or pile stiffness. Soil springs have no tension stiffness, and piles may or may not have tension stiffness. If a support location sees tension, then the springs in this location are switched to tension springs of the appropriate capacity. Soil springs will have a tension stiffness of zero and piles will have a stiffness defined in the **Pile Definitions** spreadsheet. At this point the model is re-run for its second iteration. The model is then checked again to make sure that tension springs are in tension and compression springs are in compression. If so, the solution is stopped. If not, the springs are updated to the proper tension or compression designation and the model is re-run again. At least 3 iterations are required to get acceptable behavior, though more may be needed in complicated loading scenarios.

The **Coefficient of Friction** is used in the slab sliding check to calculate the resisting force against sliding.

The **Merge Tolerance** is used as the maximum distance 2 points can be apart and still be merged together. It is also used when scanning for crossing members and for unattached joints along the spans of beams.

The **Solver** to be used during solution may be selected by clicking the radial button next to **Standard Skyline** or **Sparse Accelerated**. See Solution for more information on these two options.

The Use all available CPU cores for solution allows the program to speed up solution time by using all available processing power on the computer for some aspects of the solution. See Solution for more information.

The **LL Reduction** check boxe indicate whether LL Reduction will be considered. These check boxes are only available for models that are integrated with RISAFloor. And, they only affect elements that originated in the RISAFloor model.

Note:

- Wall results are not currently affected by LL Reduction.

Seismic

The options in the Seismic tab list factors that can be included in your seismic load combinations generate by the LC Generator.
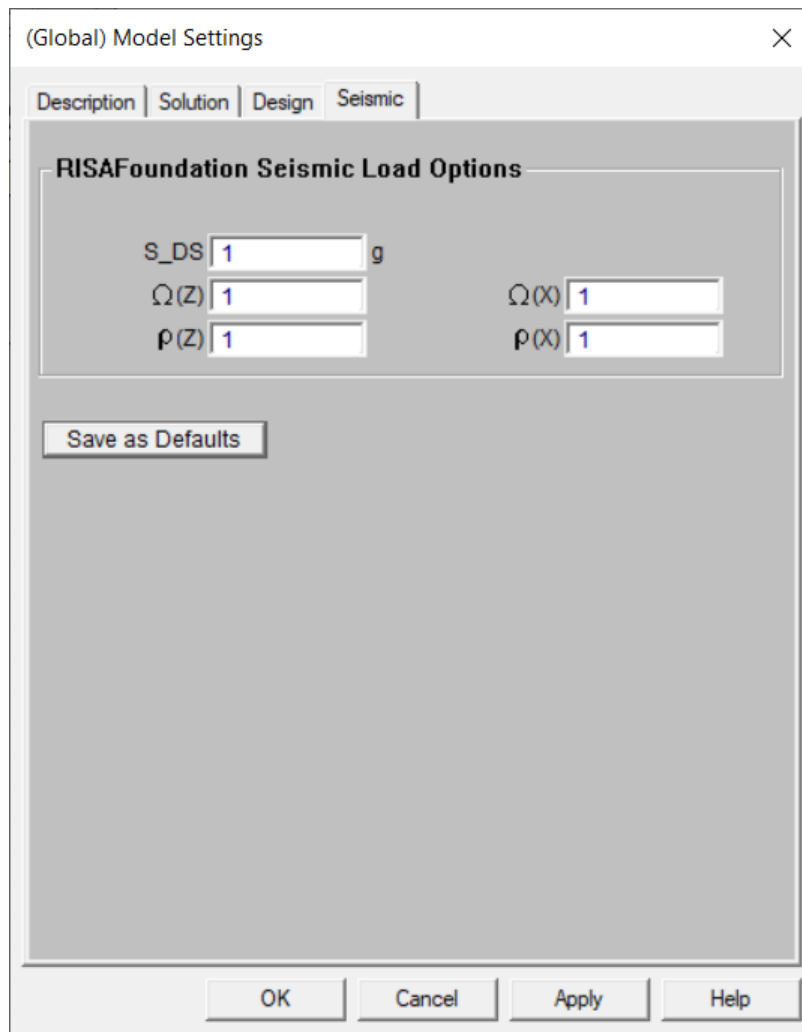
S_DS represents the 5% damped spectral response Design acceleration for short period response. This will be used if you include vertical seismic load effects in your seismic load combinations. This can be included in the seismic load combinations generated by the LC Generator.

(Ω) is the **Overstrength Factor**. This can be included in the seismic load combinations generated by the LC Generator.

The **Redundancy Factor** (ρ) is based on the extent of redundancy in your structure. This can be included in the seismic load combinations generated by the LC Generator.

Note

- The overstrength and redundancy factor indicated in the Z direction will be used for the ELZ basic load case. Likewise, the factors indicated in the X direction will be used for the ELX basic load case.
- Using the overstrength factor or redundancy factor with the EL basic load case will default to using the factors in the X direction.

Design

The entries under the **Design** tab contain options related to the analysis and design of concrete and masonry elements.

The **Design Codes** allow you to choose your specific design code for elements in your model.

Available options for concrete are:

- The 2014, 2011, 2008, 2005, 2002 and 1999 Editions of ACI 318
- The 1992 EuroCode (EC2) and the British publication of the 2014 and 2004 Eurocode (BSEN)
- The 2004 Edition of the Canadian code (CSA-A23.3)
- The 2004 Edition of the Mexican code (NTC-DF)
- The 2007 Edition of the Saudi Building Code (SBC 304)

Available options for masonry are:

- The ACI 530-13: ASD and Strength Codes:

Note:

- Masonry is only used for the wall in wall footing elements. See the Wall Footing Definitions topic for more information.

- A significant change for the 2014 ACI code is a change to the minimum flexural reinforcement for mat slabs. Please see ACI 318-14 Sections 7.6.1 and 8.6.1 (and Chapter 13's references to Chapter 7 and Chapter 8) for more information

The **Pile Design Codes** section allows the user to select the Hot Rolled Steel and Wood codes for pile design. The design code selected for piles will be used to calculate the pile tensile and compressive capacity when those materials are used for piles in the model.

The available design codes for Hot Rolled Steel are:

- AISC 360-10 (14th Edition) and AISC 360-05 (13th Edition) ASD and LRFD
- AISC LRFD (2nd and 3rd Editions)
- AISC ASD (9th Edition)
- The CSA S16-14, CSA S16-09, CSA S16-05, CSA S16-01, and CSA S16.1-94 Canadian codes

The available design codes for Wood are:

- The 2015, 2012, 2005/08, 2001, 1991/1997 Editions of the NDS wood design code for ASD
- The 2014 and 2009 Editions of the CSA O86 Canadian wood design codes.

The **Shear Tie Options** allow you to control the **Number of Shear Regions** that will be used when detailing a beam span. It also allows you to specify the increment that you'd like the program to use when increasing or reducing the spacing of the shear ties.

The **Parme Beta Factor** is used to approximate the column's 3D interaction surface when using the PCA Load Contour Method.

The **Pile Safety Factor** is used to calculate the static pile soil capacity. The user may modify this value to consider potential variances in soil and pile interaction

The **Minimum % Steel for Pedestal** allows you to choose the minimum percentage of reinforcement steel to be used in a concrete pedestal or pile section. The percentage entered will be multiplied by the gross area of the pedestal or pile section to determine the minimum amount of reinforcement required in each member. If left as blank, default minimum percentage of reinforcement steel will be used based on selected concrete design code. For example the default minimum percentage of steel for pedestal and pile is 0.5% under ACI 318-14.

The **Concrete Stress Options** allow you to choose what type of stress block to consider in your analysis. The options are the constant **Rectangular Stress Block** and the **Parabolic Stress Block**. See Parabolic vs. Rectangular Stress Blocks for more information.

Check the **Optimize Footings for OTM/Sliding** to get simple spread footings to be sized for overturning and sliding forces.

The **LL Reduction For Columns** check box determines whether the live load reduction based on the RISAFloor calculated tributary areas will be accounted for in the column reactions used in RISAFoundation. This check box is not available unless the model originated in RISAFloor.

Checking the **Minimum 1 Bar Dia Spacing** box will allow a minimum spacing of one bar diameter between parallel bars. Otherwise, RISA will default to a two bar diameter or one inch clear spacing, whichever is greater, to allow for lap splices and continue to maintain adequate spacing between parallel bars. This option applies to beams only.

The **Concrete Rebar Set** allows you to choose from the standard ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar), prENV 10080 (Euro), CSA G30.18 (Canadian), and AS/NZS 4671:2001 (Australian/New Zealand) reinforcement standards.

The **Include Welded Wire Reinf. (ASTMA1064) in Slab Design** allows you to design a slab using Welded Wire Reinforcement.

The **Mat Slab Design Option** offers a choice between two methods, namely the Construction method and the Theoretical method, for the design of the mat slab rebar. The Construction method, which is the default method

employed by the program for a significant period, determines the design of the slab based on an integer number of rebar. This method operates under the assumption that if the strip width divided by rebar spacing is not an integer, an additional rebar will be provided to guarantee the presence of a rebar at each end of the design strip. On the other hand, the Theoretical method is a new method added later that enables the program to calculate the rebar area based on the average value, where the rebar area equals As (single rebar) multiplied by the Strip Width divided by Spacing. However, it is crucial to note that this method may not always result in an integer number of rebar so unlike the Construction method, the results for Theoretical method will have no info of rebar amount.

Note:

- Users shall choose the correct material in the [Materials Spreadsheet](#) to account for ASTM A1064 steel.
- Welded wire reinforcement is only available for Mat Slabs, Footings, and Pile Caps.
- Only ASTM A1064 deformed bar sizes are available.

Graphic Display

RISAFoundation's robust graphics make building and understanding your model much easier. You may have numerous independent views open and active at the same time. Fast redraw times, responsive panning, true to scale rendering, animation capabilities, and graphic editing tools all help you build and understand your model. You may draw the model and specify loads and supports graphically. These items may be modified graphically as well. Verification of your model is made simple with true-to-scale rendering and color coded code checks. Results such as member force diagrams, deflected shapes, and animations may also be viewed for solved models.

Multiple Windows
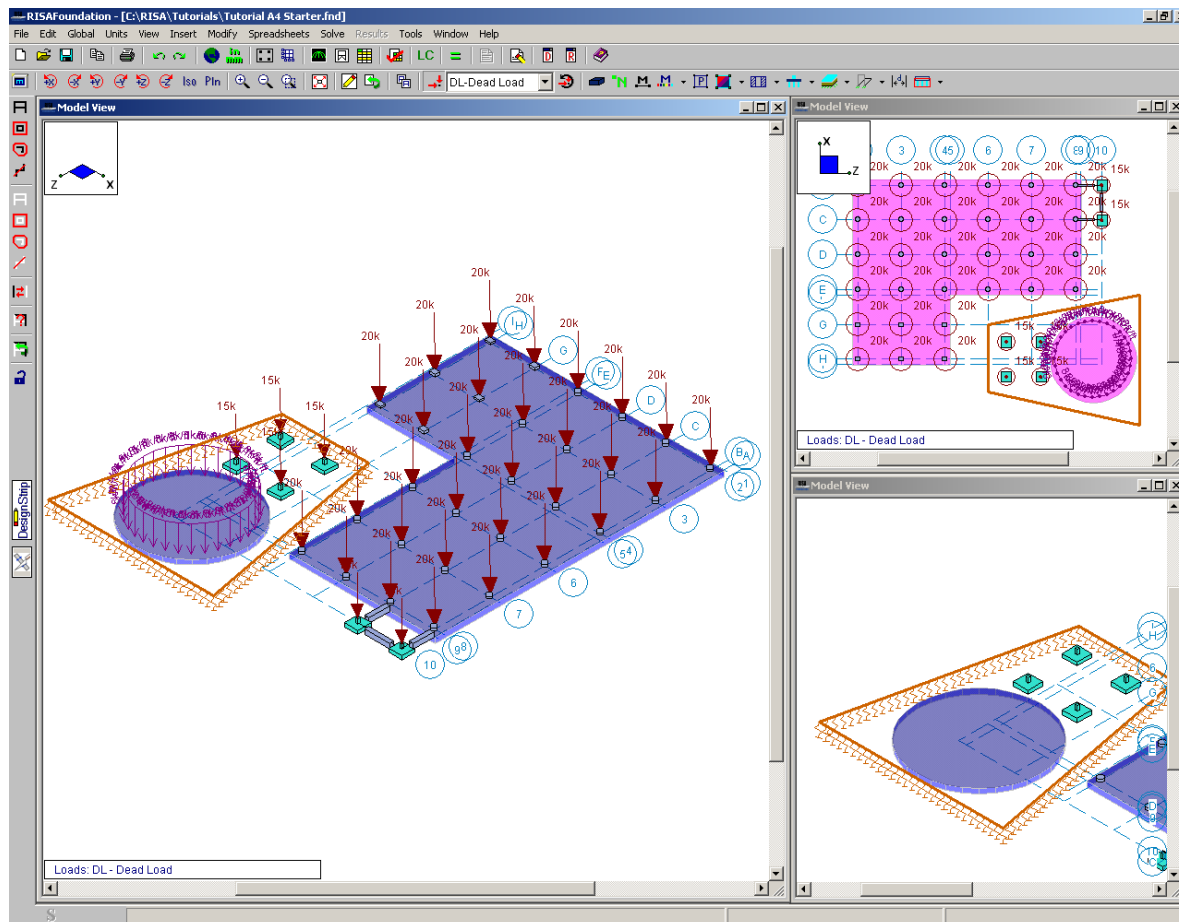
You may have multiple views of the model open at the same time, each with it's own characteristics. For example you can simultaneously view and work with slabs in one window and footings in another. You can have multiple spreadsheets open as well.

### To Open a New View or Clone an Existing View

- To open a new view of the model click the **New View** ▦ button. Alternately, you can open a new view that is an exact replica of the current view by clicking the **Clone View** ▦ button.

Note

- With the model view active you may then use the **Window Toolbar** to adjust the view of the model.

Controlling the Model View

The two main tools to help you manipulate the views of the model are the **Window Toolbar** and the **Model Display Options**.
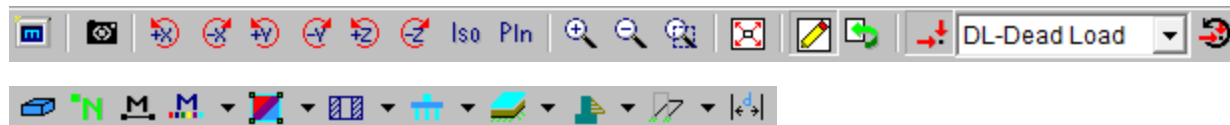
The **Window Toolbar,** discussed in this section, is the second toolbar from the top when a model view is active. You may use these buttons to rotate the model, zoom in and out, render the model, and display common items such as point labels and loads.

The **Model Display Options Dialog** (See Model Display Options for more information) is opened by the first toolbar button and provides various options not otherwise available on the toolbar. It allows you to specify how to plot points, beams, slabs, and loads and the information that is to accompany them. The tabs across the top of the dialog break things up into basic elements to help you find the options.

The **Snapshot** tool is a great way to add images of model views and detail reports in your report printing. See Printing for more information.

Adjusting the Viewpoint

The remaining buttons on the **Window Toolbar** help you change the viewpoint by rotating, snapping, and zooming the view. You may use the scroll bars and the mouse wheel (if you have one) to pan the view.

To Rotate a View

- When holding down the Shift key, the mouse wheel button can be used to dynamically rotate the structure. See Application Interface - Dynamic Controls for more information.
- To rotate a view of the model about a global axis, click any of the **Rotate** buttons on the **Window Toolbar:**
- To snap to an isometric or planar view of the model click any of the **Snap** buttons on the **Window Toolbar:**

Note

- As you rotate the view, the global X-Z axis orientation is shown in the view.
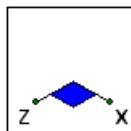
To Zoom In and Out

- Use the wheel button on the mouse to dynamically zoom in or out the model view. Rotating the wheel forward zooms in and rotating the wheel backward zooms out.
- To zoom in towards the center of the model view click the **Zoom In** button or press the **+** key.
- To zoom out from the center of the model view click the **Zoom Out** button or press the **-** key.
- To zoom in on a specific area of the model view click the **Zoom Box** button and then draw a box around the zoom area by clicking and dragging with the left mouse button.
- To snap the model to a full model view click the **Full Model View** button.

- The Function Keys, F3 and F4, are for the Zoom Previous and Zoom Next functions respectively. The system holds a total of 10 zoom-states. The F3 or F4 keystroke moves the active pointer forward or backward on the list of zoom states. If multiple model views are present, each window will have its own zoom list.

To Pan the View

- Clicking and holding the mouse wheel button triggers a dynamic pan and allows the user to drag the current model view to the limits of scroll bars.
- If you do not have a mouse wheel you may drag the scroll bars and click the arrows to pan the view.

Enabling Graphic Editing

The **Graphic Editing Toolbar** or **Drawing Toolbar** may be turned on to graphically edit the model at any time and it may be turned off to provide a larger view. See Graphic Editing for more information.

To Toggle the Graphic Editing Toolbar

- To toggle the graphic editing toolbar on or off, click the ⬚ button on the **Window Toolbar**.

Saving Views

Click the ⬚ button to save and recall views. If you have a view that you like to work with or have created a view that took some time to set up, save it with this feature. All of the Model Display Options are saved with the model for later recall. See Saving and Retrieving Model Views for more information.
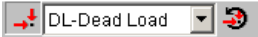
Cloning Views

You can clone a view. This allows you to preserve the original view and modify the clone. This is useful in achieving two views that have similarities in some way.

To Clone the Model View

- Click the **Clone View** ⬚ button on the **Window Toolbar.**

Displaying Loads

Use these buttons ⬚ DL-Dead Load ⬚ to control the display of loads. You may display an entire load category or load combination. The first button turns the display of loads on and off in the current view. The pull down list controls which category or combination is shown in the view. Click the last button to choose between plotting load categories or load combinations. More control over the display of loads is provided in the **Model Display Options Dialog** discussed in the next section.

Toggle Buttons

The last buttons on the **Window Toolbar** are toggles that plot information in the current view.

Click ⬚ to toggle model rendering. See Model Display Options for more information on rendering.

Click ⬚ to toggle the point labels.

Click ⬚ to toggle the beam labels on and off. If rendering is on, beam labels will not be visible.

Click ⬚ to toggle display of member color basis.

Click to toggle display of plate contours.

Click to toggle display of pile caps.

Click to toggle display of soil regions.

Click to toggle display of retaining walls.

Click to toggle display of deflected shape.

Click to use the distance tool.

Depth Effect

Right click in the model view and click **Depth Effect** on the **Shortcut Menu** to access the depth effect options. Depth effect is used to increase or decrease the "depth" effect of the plot. Increasing the depth effect causes points farther away from the screen to converge towards a vanishing point, giving the image a more realistic three-dimensional appearance. Removing the depth effect causes the vanishing point to become infinity (no vanishing point). The button resets the view to have no depth effect.

Viewing Part of the Model

You may want to view only part of your model, especially if it is a big one. You may use the selection tools to give you the view that you want. You can graphically unselect parts of the model that you don't wish to see, or you can use a range of coordinates or other criteria to specify what to view. See Graphic Selection for more information.

Saving and Retrieving Model Views

You can save and recall views for a model. Saved views are model dependent so any views you save will stay with the model. A saved view includes information such as the current view angle, zoom state, pan location, plot option settings, etc. Saved views do NOT include the selection state for the model. You can save selection states separately. See Saving Selections for more information.

To Save the Model View

- Click the **Save View** button on the **Window Toolbar.**
- Click the **Save** button and enter a name and then click **OK.**

To Retrieve a Saved Model View

- Click the **Save View** button on the **Window Toolbar.**
- From the drop down list, choose the view you wish to recall and then click the **Retrieve** button.

To Delete a Saved Model View

- Click the **Save View** button on the **Window Toolbar.**
- From the drop down list, choose the view you wish to delete and then click the **Delete** button.

Model Display Options

The Window toolbar (See **Graphic Display** for more information) offers some common graphical plotting choices, but these are just a few of the options available. Many more options are located in the **Model Display Options** dialog which may be accessed by clicking the [m] button on the Window toolbar or pressing the F2 function key.

The options are organized into groups of items. Access each group by clicking on the tabs along the top of the dialog box. Some of the options are mutually exclusive and others are conditional. Typically, only one option in a vertical row may be chosen at a time and indented options will not be applied if the heading option is not also applied.

Each of the groups presents the option to turn off those particular items in the view. This is not the same as unselecting the items. Turning items on and off is independent from selecting and unselecting. For example if you make a selection of members and then turn off members all together, the selection still stands for any modification applied to members.

Defaults

You may make any Plot Option settings the default start-up settings. To do this, go through all the Model Display Options tabs and set all the options to what you want as the start-up default. Once that is done, select the **Misc** tab and press the button labeled **Make Current Plot Settings the Default**.

Note:

- Remember, this button applies to ALL tabs in the Model Display Options dialog.
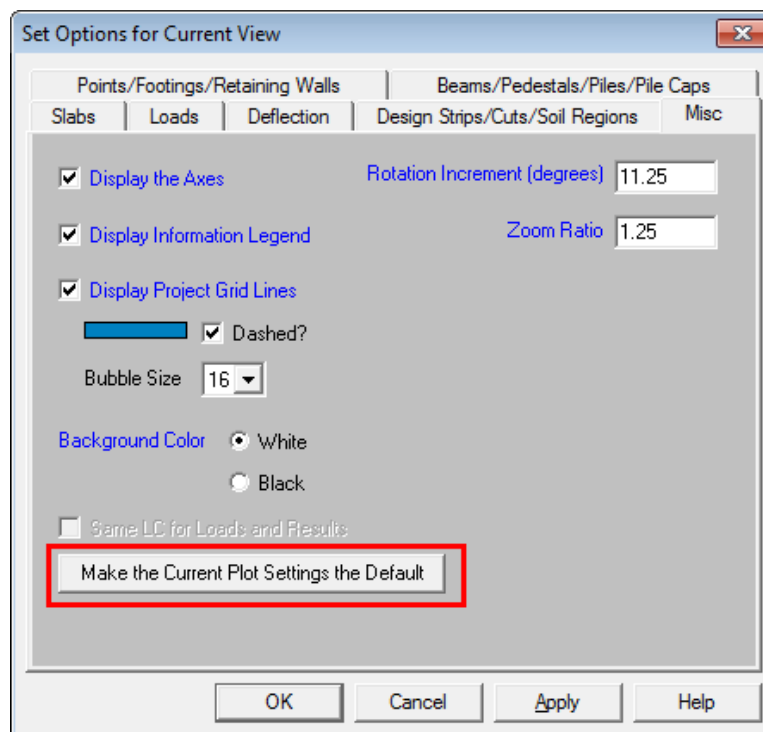
Beams/Pedestals/Piles/Pile Caps

Access the Model Display Options for **Beams/Pedestals** by clicking **Model Display Options** dialog [m] button on the Window toolbar and selecting the **Beams/Pedestals/Piles/Pile Caps** tab shown below.
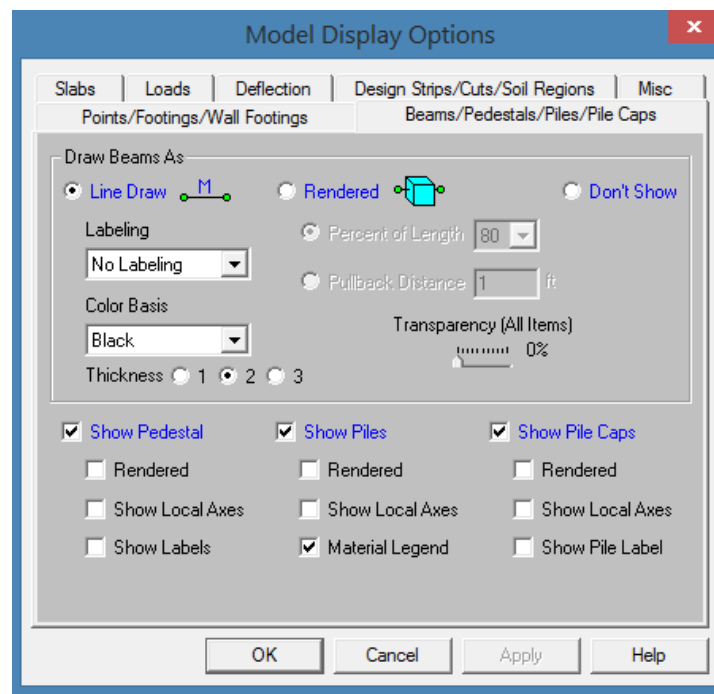
The **Draw Beams As...** section of this tab allows you to specify that beams are to be drawn as lines, or as rendered elements, or that they are not to be shown at all. For a line draw, you can specify the color basis of the line and the line thickness.

If the **Line Draw** option is selected, you may include textual information alongside each beam. The **Labeling** drop down list provides the choices you have as to what information will be displayed. None of these options are available if the **Rendered** display option is selected.

Color Basis draws the beams using various colors to represent particular information such as material set and shape group. The **Color Basis** drop down list provides the choices you have as to how the colors are to be assigned to each member.

If the **Rendered** option is selected, true to scale representations of the beams will be drawn. Selecting the **Percent of Length** option and a value from the drop down list will render only that percentage of the length of each beam and the left over portions at each end will be drawn as lines. Entering a percentage of '100' will render the full length of all beams. Selecting the **Pullback Distance** option and entering a value in the text box will render each beam with the specified length at each end drawn as lines. Entering a length of '0' will render the full length of all beams.

The rendered option is very useful for verifying member orientations and modifying the rendering options can provide a better understanding of intersection areas and a better view of the member cross-section.

Show Pedestal turns on the display of pedestals. If the **Rendered** option is selected, true to scale representations of the pedestals will be drawn. If the **Show Local Axes** box is checked, the local axes of the pedestals will be displayed.

Show Piles turns on the display of individual piles. If the **Rendered** option is selected, true to scale representations of the piles will be drawn. If the **Show Local Axes** box is checked, the local axes of the piles will be displayed. The **Material Legend** option displays a legend in the top right of the Model View of the pile materials by their graphical display color.

Show Pile Caps turns on the display of pile caps. If the **Rendered** option is selected, true to scale representations of the pile caps will be drawn. If the **Show Local Axes** box is checked, the local axes of the pile caps will be displayed.

Points/Footings/Retaining Walls

Access the graphical Model Display Options for Points, Footings, and Retaining Walls by clicking the **Model Display Options** button on the Window toolbar and selecting the **Points/Footings/Retaining Walls** tab.

Checking the **Show Points** box will turn on the display of points in the current model view. Point labels may also be turned on by checking the **Show Labels** box. Checking the **Include Generated Points** box will turn on the display of points generated by the slab auto-meshing. Note that these will only be displayed after a solution is performed.
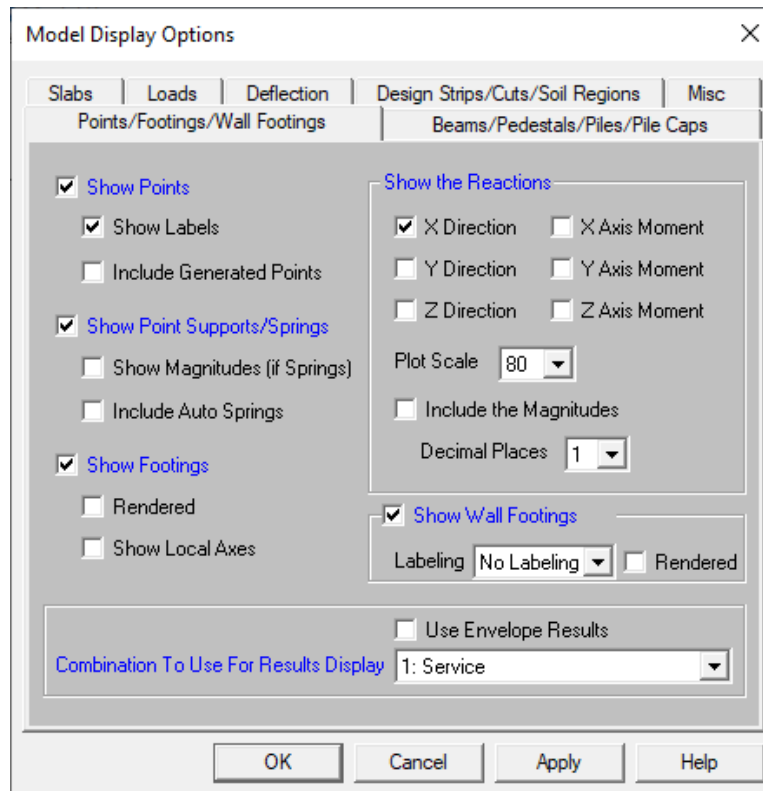
Checking the **Show Point Supports/Springs** box will turn on the display of any user entered point supports or springs in the current model view. The magnitudes of user entered springs may be displayed by checking the **Show Magnitudes** box. Checking the **Include Auto Springs** box will turn on the display of springs automatically generated at the meshed points of the slab. Note that these will only be displayed after a solution is performed.

Checking the **Show Footings** box will turn on the display of footings in the current model view. The footings can be rendered by checking the **Rendered** box. Checking the **Show Local Axes** box will display the local axes for each footing.

The **Reactions** section of this tab is used to control the display of point support, spring, footing, and beam reactions in the current model view. Checking the **Show Reactions** box will turn on the display of all reactions. The magnitudes of these reactions may be displayed by checking the **Include Magnitudes** box. The number of decimal places to be displayed is controlled with the **Decimal Places** pull down box.

Checking the **Show Retaining Walls** box will turn on the display of retaining walls in the current model view. The retaining walls can be rendered by checking the **Rendered** box. You can also show the **Retaining Wall Label** or the **Local Axes** from the drop-down list.

The **Combination To Use For Results Display** pull down box is used to select the load combination for which the results are displayed in the current model view. This box is available on any tab in the Model Settings dialog containing results display capabilities. The selection show in this box will be the selection for the display of all

results, even those whose display is controlled on another tab. You can also check the "Use Envelope Results" checkbox to show the Envelope Point Reactions.

Slabs

Access the graphical Model Display Options for slabs by clicking the **Model Display Options** button on the Window toolbar and selecting the **Slabs** tab shown below.
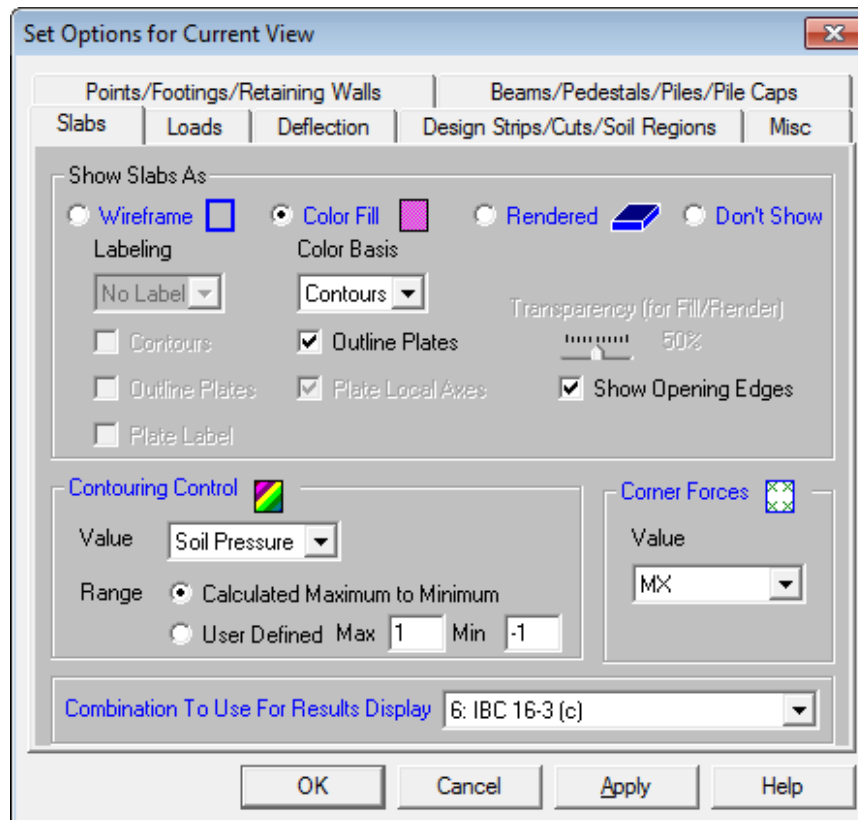
The **Show Slabs As...** section of this tab allows you to specify that slabs are to be drawn as wireframe, color filled, or rendered elements, or that they are not to be shown at all.

When the **Wireframe** option is selected, you may include information on the slabs when plotted. The **Labeling** drop down list provides the choices you have as to what information will be displayed on each slab, such as the slab label, material, thickness, or the slabnumber. The wireframe option also allows for the display of line contours representing the force or stress results for the slabs. Each color line represents a specific value. You must choose which load combination you want to view in the **Combination To Use For Results Display** drop down list.

When the **Color Filled** option is selected, slabs are plotted with different colors that are mapped in a key shown in the upper right corner of the model view. You may color the plates within a slab by material set or specify a uniform color for all plates. You also have the option to plot the stress and force results using filled color contours where each fill color represents a range of values. The color filled plates themselves may be presented with varying **Transparency** with 100% transparency meaning completely see through and 0% indicating completely solid. You must choose which combination you want to view.

Note:

- The transparency setting does not apply when "**Contour**" is selected as the color basis from the drop down list.

The **Color Contours** are plotted with a global smoothing algorithm that allows the contour to vary across the plates within a slab. Because of this, the plotted contour results will differ slightly from the tabulated results in the spreadsheets. This effect will be heightened in regions of a high rate of change such as loads or boundaries. The contours may be drawn as either lines or as color filled areas.

The **Contouring Control** section determines what force or stress result is to be contoured and how the contour colors are to be assigned. The **Value** drop down list is where you select the specific result to be contoured (QZ, MXZ, Soil Pressure, etc.). The **Range** controls determine how the contour colors are assigned. You can either contour the full range of the results by choosing **Calculated Maximum to Minimum**, or, if you are only interested in a specific range of values, you can choose **User Defined** and enter your own max and min values.

When the **Rendered** option is selected, true to scale representations of the slabs will be drawn with thickness.

Along with any of the above options you may also plot the plate **Corner Forces**. These are the global direction forces applied to the corners of the plates within a slab that hold them in equilibrium. The **Value** drop down list is used to pick which corner force (or moment) is to be displayed. You must choose which combination you want to view.

The **Transparency** setting described above for Color Filled plates also applies to Rendered plates.

Loads

Access the graphical Model Display Options for loads by clicking the **Model Display Options** button on the Window toolbar and selecting the **Loads** tab shown below.
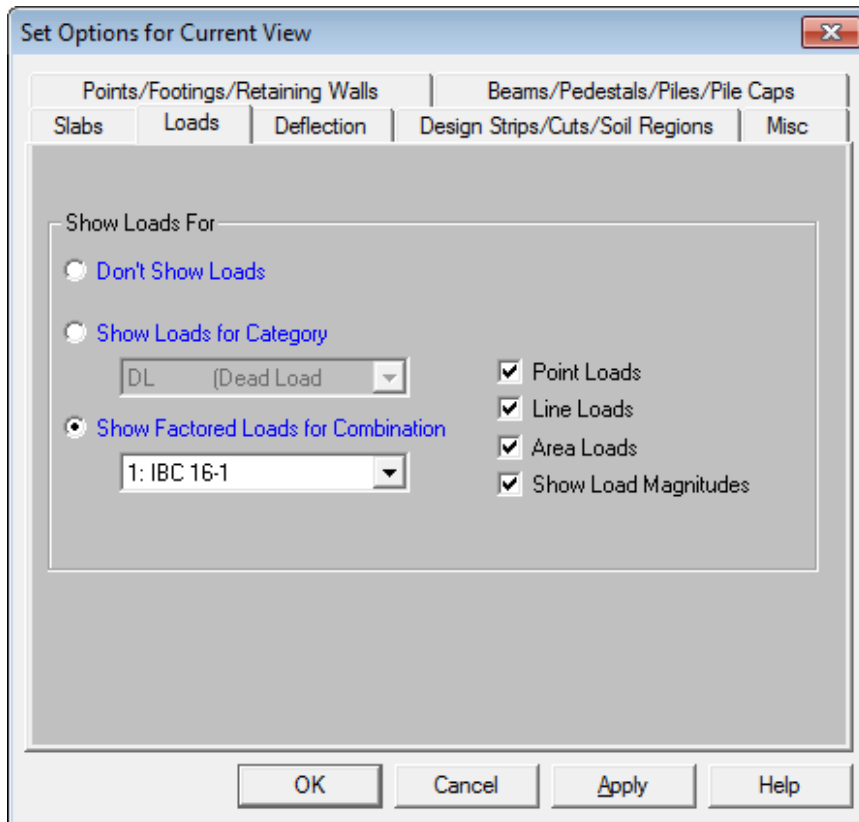
You may specify that the loads be drawn as Load Categories or Load Combinations by selecting the corresponding radio button. The load types to be displayed can be selected by checking the corresponding boxes.

Once you set the Model Display Options for loads, you may then run through all of the load categories or load

combinations by choosing them from the drop down list ![DL-Dead Load dropdown] on the Window toolbar.

Deflection

Access the Model Display Options for deflections by clicking the **Model Display Options** button on the Window toolbar and selecting the **Deflection** tab shown below.

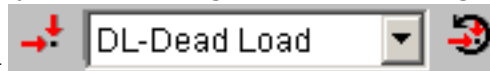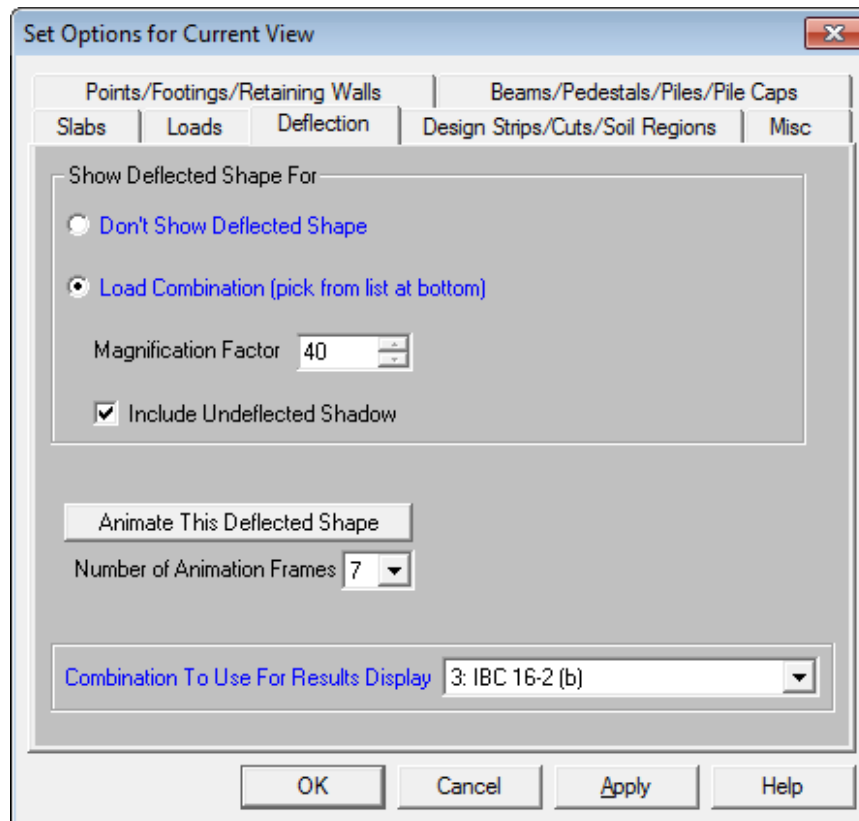In the **Show Deflected Shape For...** section of the tab you may specify that a deflected shape be drawn in the current model view based on a load combination. You can control the magnification of the deflections with the **Magnification Factor** text box and specify whether an undeflected shadow is to be shown by checking the **Include Undeflected Shadow** box. You must choose which combination you want to view in the **Combination To Use For Results Display** drop down list.

To animate a particular deflected shape, first select the deflected shape (as described above) and then simply click the **Animate This Deflected Shape** button. A new model view will be created with the animation. The **Number of Animation Frames** drop down list allows you to specify how many frames are used in the animation to move from the undeflected state of the model to the fully deflected state.

In the model view of the animated deflected shape, the **Capture** feature allows users to save video file format of the animated deflected shape. To use this feature, adjust to the desired number of frames and click on the video button. The user will be prompted to save the video file.

Note:

- The animation that is created is placed behind the original view. You may shrink the original view or use the **Window** menu to bring the animation into view.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Deflected**.

Miscellaneous

Access miscellaneous plotting options by clicking the **Model Display Options** button on the Window toolbar and selecting the **Misc** tab shown below.
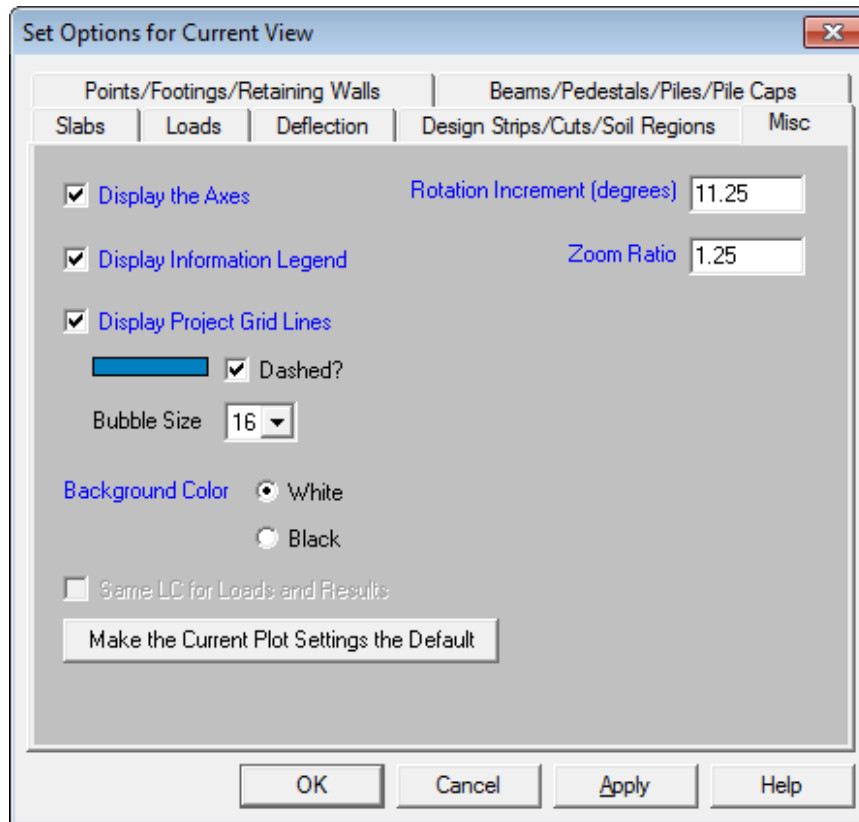
You may indicate whether the global axes icon is to be displayed in the upper left corner of the model view by checking the **Display the Axes** box. You also have the option of displaying the informational legend drawn in the bottom left corner of the model view by checking the **Display Information Legend** box. This legend has information such as which combination has been solved, which loads are being displayed, etc.

Checking the **Display Project Grid Lines** box will turn on the display of the project grid in the current model view. The color and line type for the grid may also be customized. Click the colored rectangle to choose a custom color. Check the **Dashed?** box to display grid lines as dashed lines. The **Bubble Size** drop down list allows you to choose a font size for the grid labels and the bubbles that encompass them.

You may specify whether the **Background Color** for the model view is to be black or white by clicking the corresponding radial button. A black background shows colors more vividly, but a white background shows a more realistic representation of what you'll get if you print the graphic view.

By entering values in the text boxes, you may control the **Rotation Increment** and **Zoom Ratio** that will be used with each click of the corresponding toolbar buttons on the **Window Toolbar**.

Finally, you may make any plot option settings the default startup settings. To do this, go through all the Model Display Options tabs and set all the options to what you want as the startup default. Once that is done, press the **Make Current Plot Settings the Default** button. This button applies to ALL the Model Display Options tabs.

Design Strips/Cuts/Soil Regions

Access the graphical Model Display Options for design strips by clicking the **Model Display Options** button on the Window toolbar and selecting the **Design Strips/Cuts/Soil Regions** tab shown below.
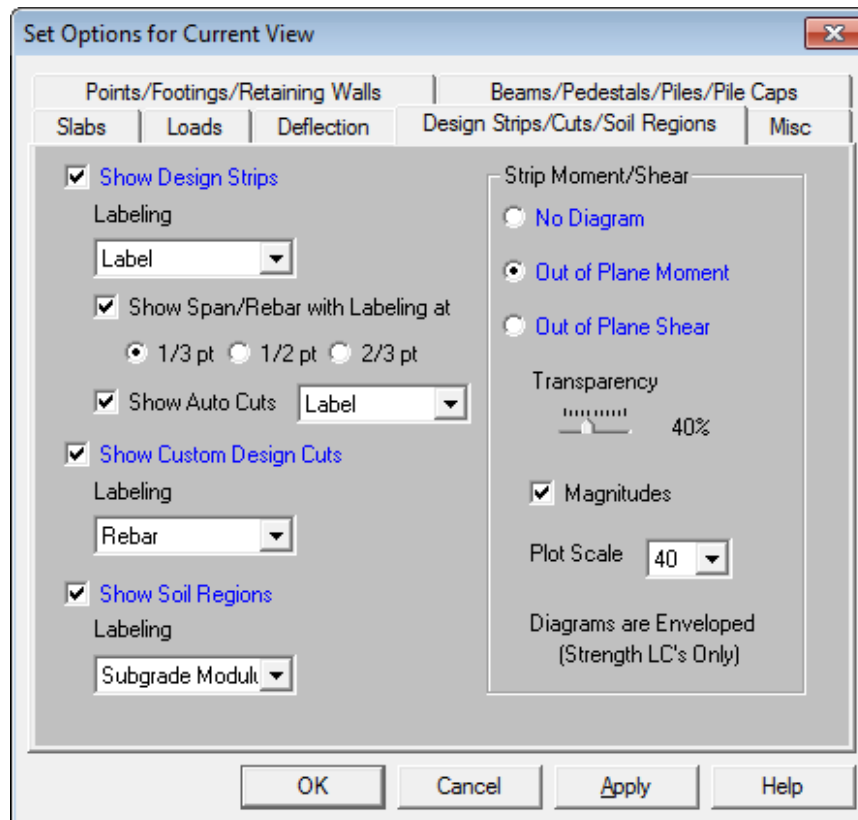
Checking the **Show Design Strips** box will turn on the display of design strips in the current model view. The **Labeling** drop down list allows you to label design strips with the **Label** name or the **Rebar** call out that is being used. Checking the **Span/Rebar** box toggles the graphical display of the rebar layout and you can also pick where you want the labeling to be displayed. This is helpful in showing your reinforcing with out having your labels overlap. Checking **Show Auto Cuts** toggles the display of cuts automatically made by the program during a solution.

Checking the **Show Custom Design Cuts** box will turn on the display of design cuts specified by the user. The **Labeling** drop down list allows you to display design cuts with the **Label** name or the **Rebar** call out that is being used.

Checking the **Show Soil Regions** box will turn on the display of user entered soil regions in the current model view. The **Labeling** pull down box can be used to turn on the display of soil region labels and to choose the labeling type. The choices are Label, Number, and Subgrade Modulus.

The **Strip Moment/Shear** section of this tab controls the display of moment and shear diagrams on each design strip. **Out of Plane Moment** diagrams and **Out of Plane Shear** diagrams can be plotted on each design strip by clicking the corresponding radial button. The diagrams themselves may be presented with varying degrees of **Transparency** with 100% transparency meaning an outline of the force diagram and 0% indicating a solidly filled diagram. The transparency is controlled by clicking and dragging the slider bar until the desired percentage is displayed below the bar. Checking the **Magnitudes** box will turn on the display of the force magnitudes

corresponding to each design cut within the design strips. The relative graphical scale of the moment and shear diagrams can be controlled by selecting a factor from the **Plot Scale** drop down list.

Note:

- The Moment and Shear Diagrams that are displayed are Enveloped to capture the maximum positive and negative force magnitudes along the length of each design strip.

Graphic Editing

You may draw, edit, and load your model directly in the model views. You can draw beams and slabs between existing points or draw to grid intersections and have the points created automatically. You may graphically select points to be supported and beams or slabs to be loaded. See Graphic Selection to learn how to make selections.

Drawing and Modification Features

There are several graphic editing features that make the creation and modification of models quite easy. Use the **Insert** and **Modify Menus** or the **Drawing Toolbar** to utilize these features in the model view. To create new beams, slabs, etc., you can draw them using a drawing grid or draw "dot to dot" from existing points. Once you have created these items you may use other graphic features to load the model and set supports.

All model data is automatically recorded in spreadsheets and the spreadsheets and model views are always in tune (unless you turn this feature off in the **Application Settings** on the **Tools Menu**). As you edit a model graphically the spreadsheets are automatically updated and as you make changes in the spreadsheets the model views reflect these changes immediately.

All of the graphical modeling tools may be found on the **Drawing Toolbar** shown below. This toolbar may be turned on and off by clicking the ![pencil icon] button on the **Window Toolbar**.

Where to Start

The Project Grid and Drawing Grid are often useful when you are starting a new model from scratch or adding a new section to a model. They allow you to set up grid lines which may then be used to define the beams, slabs, and footings. The differences between the two grids are simple. The Project Grid is part of the model and may be allowed to move model elements as grid lines are moved. The Drawing Grid is independent of the model so you may change the grid and place it anywhere, without affecting the model and whenever it is convenient.

There are also times when it is simpler to define points in the spreadsheet and then draw the members or slabs between them. This might be the case if you are working with just a few points, or if the structure is irregular and does not lend itself to a grid.

Apply Options

Some of the graphic editing options offer more than one way to apply a modification. This is because there are times when each option is useful. For example, changing the material of a beam from 3000 psi concrete to 4000 psi can be accomplished using the **Beams Spreadsheet**. If you had to apply this change to 100 beams however you would not want to do that for each of them. A better way to do this would be to graphically select all of the beams and then apply the changes all at once.

Use the following options to specify how you want to choose the items to modify. Choosing **Apply Entries to All Selected Items** allows you to use the tools on the **Selection Toolbar** to choose the items you want and then apply the modifications to all selected items at once. Choose **Apply Entries by Clicking Items Individually** to then click on the items you wish to modify individually. See Graphic Selection for more information on the selection tools.

Note

- The selection and viewing tools override the graphic editing modes so that as you are editing the model you can alter the selected state. You will be returned to the current editing mode when you exit a selection mode.
- You may also double click any joint, member, or plate to view and edit it's properties.

Undo Operations

RISAFoundation provides you with unlimited 'Undo' capability so that you may easily correct mistakes or just back up to try different possibilities. Simply click the button on the **RISA Toolbar** as many times as you wish to undo your previous actions. The model view and the spreadsheets will visually display the "undoing". Remember that spreadsheet edits are undone as well.

Note

- Changes made to the selection state of the model, the zoom level or rotation of the model view, or to the Model Display Options settings are NOT undone. Only a change to the model data can be undone.

Redo Operations

RISAFoundation provides you with unlimited 'Redo' capability so that you may easily redo any actions that were previously undone using the 'Undo' button. Simply click the button on the **RISA Toolbar** as many times as you wish to redo actions that were previously undone. The model view and the spreadsheets will visually display the "redoing". Remember that spreadsheet edits are redone as well.

 Note

- Changes made to the selection state of the model, the zoom level or rotation of the model view, or to the Model Display Options settings are NOT redone, since they cannot be undone to begin with. Only a change to the model data can be redone.

Project Grid

The Project Grid provides convenient snap points for modeling columns and walls in buildings and building-type structures. It also provides a convenient terminology to refer to locations in a model, such as "Grid Intersection C-4". The Project Grid is intended to be a permanent part of the model so unlike the Drawing Grid it is saved with the model.
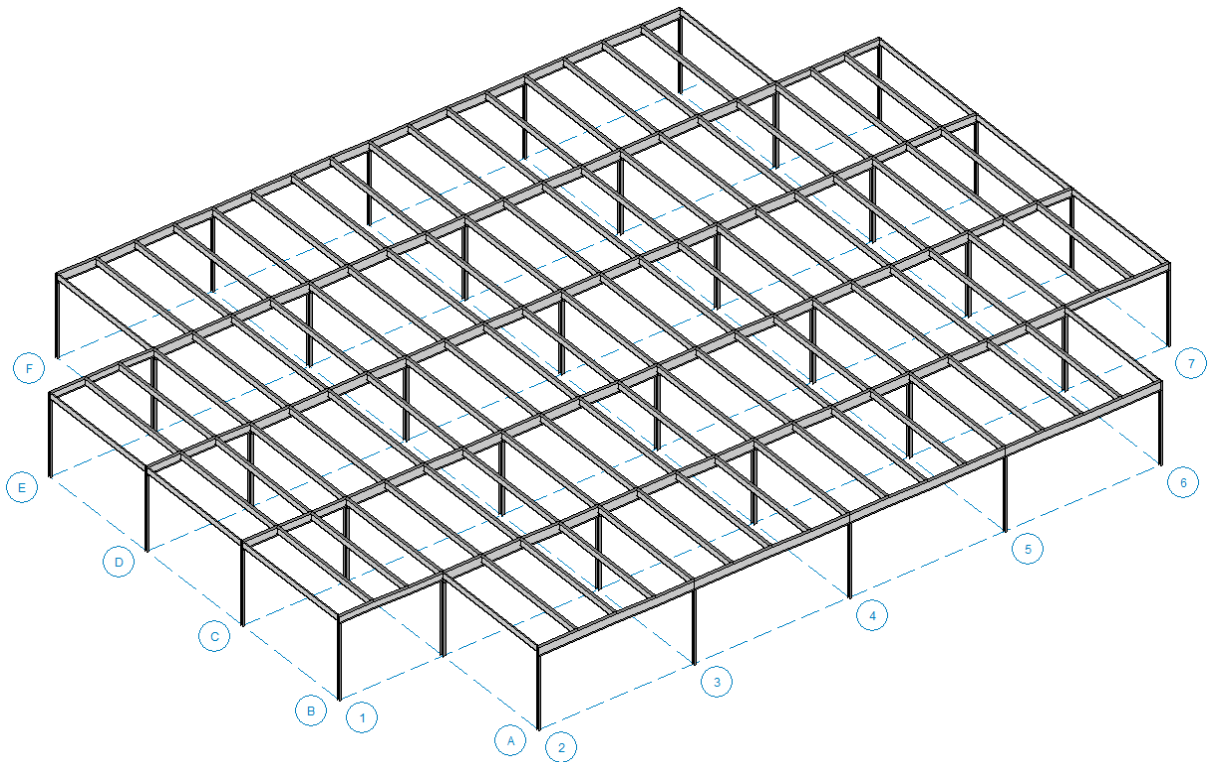
Project Grid Spreadsheet

Click the ⊞ button on the **RISA Toolbar**, or click the Project Grid button on the **Data Entry Toolbar** to open the **Project Grid Spreadsheet** shown below.

	Label	Start Z [ft]	End Z [ft]	Start X [ft]	End X [ft]	Start Bubble	End Bubble
1	A	0	0	0	125	☑	☑
2	B	20	20	0	125	☑	☑
3	C	50	50	0	125	☑	☑
4	D	80	80	0	125	☑	☑
5	E	110	110	0	125	☑	☑
6	F	140	140	0	125	☑	☑
7	G	160	160	0	125	☑	☑
8	H	170	170	0	125	☑	☑
9	1	0	170	0	0	☑	☑
10	2	0	170	25	25	☑	☑
11	3	0	170	50	50	☑	☑
12	4	0	170	75	75	☑	☑
13	5	0	170	100	100	☑	☑
14	6	0	170	125	125	☑	☑
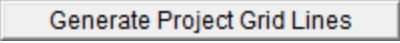
Each Grid Line (as opposed to Grid Arcs) is defined by Start and End coordinates along the X and Z axes. This method of defining grids allows them to be oriented in any direction, including skewed grids or non-parallel grids within the same building.

Detach Model

The columns and walls will move when the project grid is moved. The detach model check-box will allow you to move the project grid lines without moving the model.

Note: The model will move with the associate grid line/arc that moved. For example if you increase the radius of an Arc, the columns on that Arc grid will follow the arc. If you move a Line grid line, the columns/walls will move with that Line grid. This may mean that the column/wall is no longer at the intersection.

Project Grid Line Generation

The Project Grid Line generator can be accessed using the **Generate Project Grid Lines** button in the Project Grids spreadsheet, or by clicking the button on the graphic editing toolbar.

The generator provides the ability to generate an entire grid system at once rather than entering the grids in manually in the project grid spreadsheet.

You may use symbols such as "@", "/" and "," when entering the increments.

The "@" entry may be used to specify multiple, equally spaced, grid increments. For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.

The "/" entry subdivides a larger increment into smaller equal increments. For example, the entry "12/4" would create 4 increments of 3 units each.
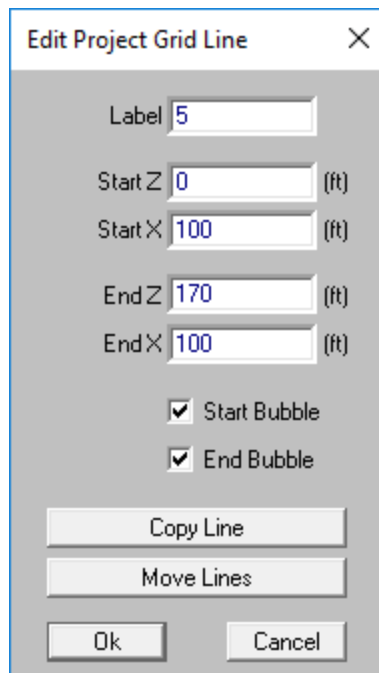
Use commas (",") to enter multiple increments in the increment field. For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.

The Apply and Undo buttons provide an easy way to preview a Project Grid before adding one to the model.

Once the project grid is specified and displayed in the model view, it will provide snap points while drawing your model.

Editing the Project Grid (Adding or Moving Grid Lines)
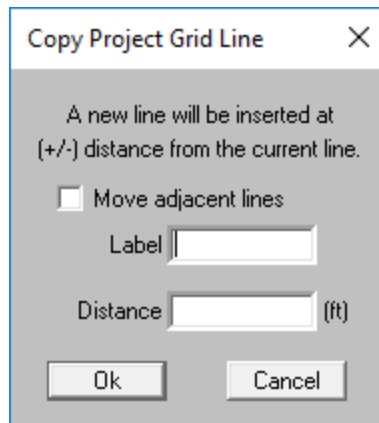
Once a Project Grid has been created the grid lines can be moved, or additional grid lines can be created, by double clicking on a grid line in the model view.
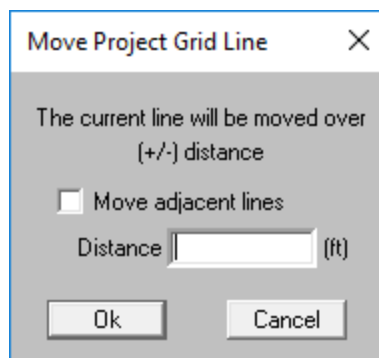
The Copy Line button allows a line to be created by offsetting an existing grid line. If the 'Move adjacent lines' is checked, a new grid line will be created and will shift existing grid lines in the same direction as well. If the 'Move adjacent lines' is unchecked, the new grid line will be created at the offset distance specified, without altering adjacent existing grid lines.

The Move Lines button allows a grid line to be moved by a specified distance. A positive distance will move the line in the positive global direction, while a negative distance will move the line in the opposite direction. If 'Move adjacent lines' is checked, all lines in the same direction will be moved by the same distance as well. This allows for a particular bay in a structure to be made larger/smaller without affecting the other bay sizes.

Drawing Grid

The **Drawing Grid** is a tool that lets you draw new beams, slabs, and footings in the model view. This grid is independent of the model, so you may change the grid as you build your model without changing any modeling that you have completed. This is because as you draw the beams, slabs, and footings the points used to define them are created automatically. These points remain in their locations if the grid is relocated.

Drawing Grid Dialog

Click the [icon] button on the **Drawing Toolbar** to open the **Drawing Grid Dialog** shown above. The display of the **Drawing Grid** in the current model view may be turned on and off by clicking the [icon] button on the **Drawing Toolbar**. If the Drawing Toolbar is not visible in the current model view, click the [icon] button on the **Window Toolbar**.
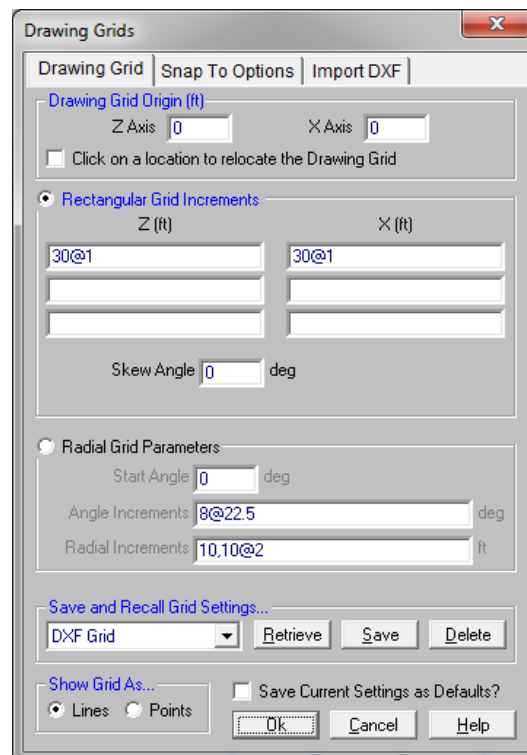
You may choose between a rectangular drawing grid or a radial drawing grid. Also, rectangular drawing grids may be skewed from perpendicular to the global axis plane.

The **Save and Recall Grid Settings...** section allows for drawing grids may be saved and recalled for later use. Saved drawing grids are model independent, i.e. when you save a grid, you can reuse it in any other model you are working with in the future. To save a drawing grid, click the **Save** button after defining the grid. You will be prompted for a name for the drawing grid and it will then be saved and added to the list of grids available for recall. To recall a previously defined drawing grid, select the grid name from the drop down list and click the **Retrieve** button. If you wish to delete a previously saved drawing grid, select the grid name you want to delete from the drop down list and click the **Delete** button.

The **Show Grid As...** section gives the option of displaying the drawing grid as lines from grid point to grid point or simply as a grid of points.

You may save any of the grid information as the default setting so that when you start a new model that information is already there. To do this, simply enter the information that you want to save in the **Drawing Grids Dialog**, check the **Save Current Settings as Defaults** box, and then click the **OK** button.

Rectangular Drawing Grid

The rectangular drawing grid is defined by increments in two directions. The **Drawing Grid Origin** is where you want the grid increments to start. The **Rectangular Grid Increments** are the distances between the grid points or lines in two global directions.
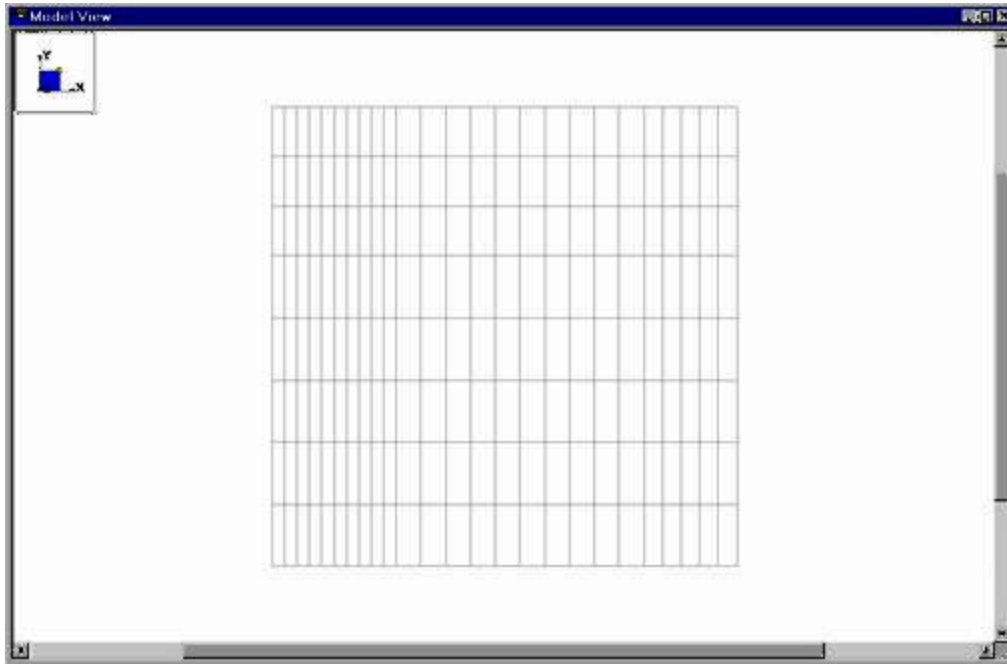
You may use symbols such as "@", "/" and "," when entering the drawing grid increments.

The "@" entry may be used to specify multiple, equally spaced, grid increments. For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.

The "/" entry subdivides a larger increment into smaller equal increments. For example, the entry "12/4" would create 4 increments of 3 units each.

Use commas (",") to enter multiple increments in the increment field. For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.

Once the drawing grid is specified and displayed in the model view, it will provide snap points while drawing your model.

Skewed Drawing Grid

The rectangular drawing grid can be skewed by specifying a skew angle. This option is available in the **Rectangular Grid Increments** section of the **Drawing Grids Dialog**. This skew angle will allow the creation of a regular rectangular drawing grid, but displayed in the model view at the specified angle, inclined from the global axis.

Radial Drawing Grid

Increments in two polar directions define a radial drawing grid. The **Drawing Grid Origin** is the point about which the grid will rotate. The default is at the global origin (0,0). The **Start Angle** defines the angle from the global axis that the first spoke will be drawn. The **Angle Increment** controls the number and angular spacing of the spokes in the grid. The **Radial Increments** controls the number and location of the rings in the grid.
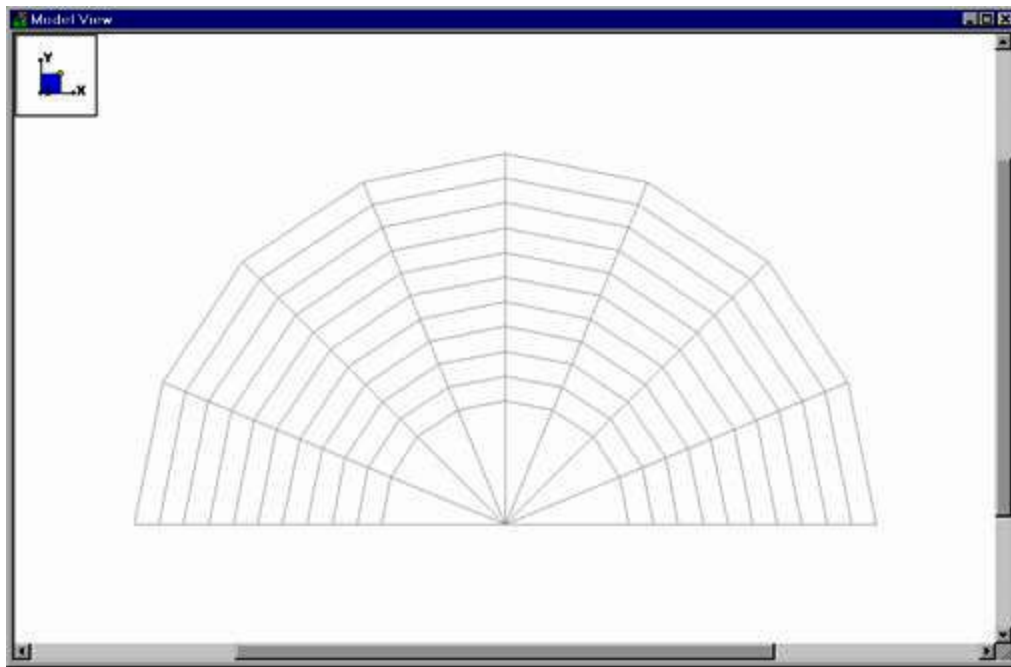
Relocating the Drawing Grid

The origin of the drawing grid may be specified in two ways. The first is to enter the exact global coordinates for the origin. This can be done by entering the values in the X and Z fields of the **Drawing Grid Origin** section. The second option is to specify the origin by clicking on an existing point in the model. This option is available by checking the **Click on a location to relocate Origin** box, clicking the **OK** button, and then clicking on the specific location in the model view where you wish the drawing grid origin to be located. The drawing grid origin will then be moved to this point.

Snap Points

Snap Points let you draw in the model view without the use of grids. To view or modify the snap point settings, click the ![button] button on the **Drawing Toolbar** and select the **Snap To Options** tab shown below.
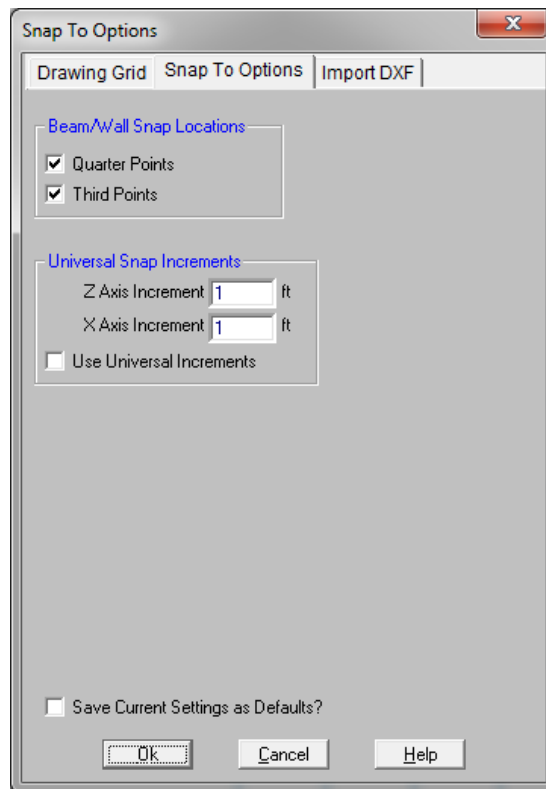
In the **Beam Snap Locations** section, you can set the program to automatically snap to the **Quarter Points** and/or **Third Points** of a beam by checking the appropriate boxes.

The **Universal Snap Increments** section is used to define a snap grid for "free" drawing to any incremental location on a plane. To activate this feature, check the **Use Universal Increments** box. Or, the feature may be toggled on and off in the current model view by clicking the ⊞ button on the **Drawing Toolbar**. If the Drawing Toolbar is not visible in the current model view, click the ✎ button on the **Window Toolbar**.

When snap points are activated, a red dot or asterisk will appear on your screen as you move your drawing cursor over one of these points. The exact coordinates of this point, and whether it is a 1/3 or 1/4 point of a beam, are reported in the status bar at the very bottom of the main application window just beneath the workspace.

Import DXF

Import DXF lets you import a DXF drawing into the model view as a grid. The DXF image is imported so that you can snap to any point or intersection to aid in the drawing of your model. The DXF Import Drawing Grid supports Lines, Polylines, Circles, Arcs, Polylines with Arcs, and Points.

To Import DXF Drawing Grid:

1. If there is not a model view already open then click 🖺 on the **RISA Toolbar** to open a new view and click ✎ to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the Drawing Grid Origin or click to checkbox to manually locate it after the dialogue closes.
3. Assign the DXF Import Options and DXF Plane.
4. Select Lines or Points for grid lines.
5. Select the DXF File from a file or from the list of recently used files. The program will remember the last 5 files imported.
6. Once the file is imported, you will be prompted to "Use All Layers" or "Select Layers from the List". You can check or uncheck selected layers, and select colors of the layers display by clicking on the gray box (defaults to gray).
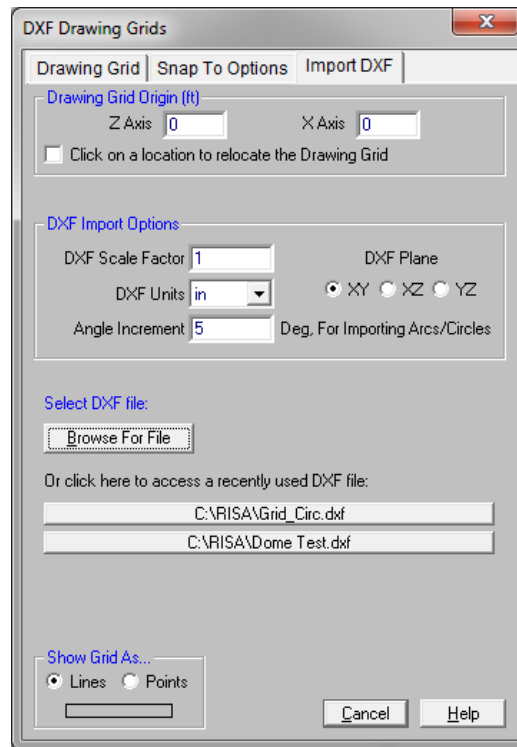
The program will default to use All Layers. Or you can select the color of the grid lines/points by clicking on the color box next to each layer name and select the color.
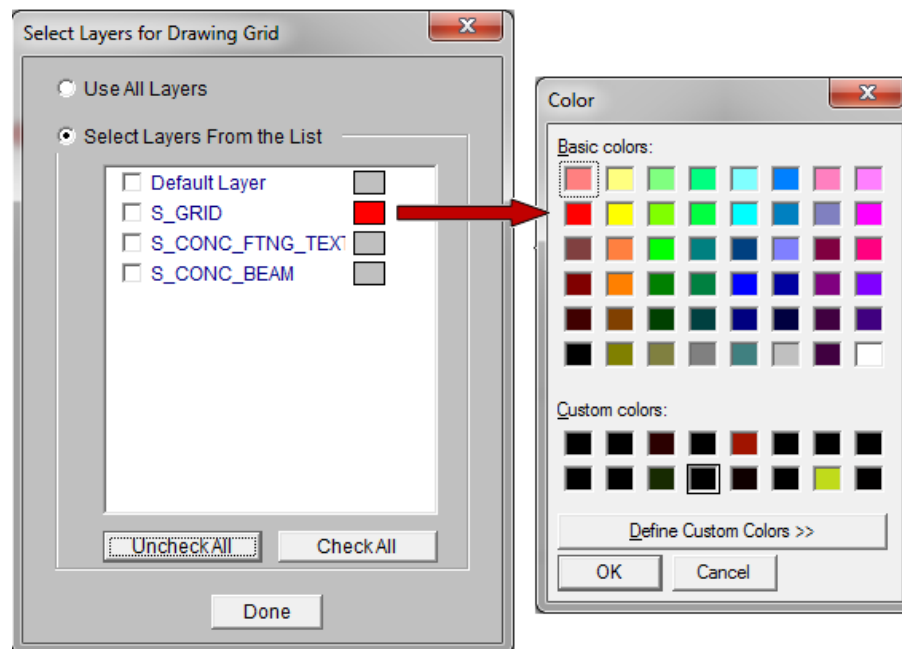
Locating the DXF Drawing Grid

The origin of the drawing grid may be specified in two ways. The first is to enter the exact global coordinates for the origin. This can be done by entering the values in the X and Z fields of the **Drawing Grid Origin** section. The second option is to specify the origin by clicking on an existing point in the model. This option is available by

checking the **Click on a location to relocate Origin** box, clicking the **OK** button, and then clicking on the specific location in the model view where you wish the drawing grid origin to be located. The drawing grid origin will then be moved to this point.

Drawing Plane

Select the RISA drawing plane for the DXF file shall be placed.

DXF Scale Factor

Enter the scale factor that will cause the DXF file to be scaled up or down to full scale. For instance, if you had created a scaled model in AutoCAD at a scale of 1/4"=12", then the appropriate scale factor to produce a full size RISA- model would be 48. The default is 1.0.

DXF Plane

Select the DXF drawing plane to import from the DXF file. The DXF file will be flattened to this plane and all lines/points visible from this plane will be imported.

DXF Units

Select the units you used in the CAD model from which you produced the DXF file. The supported DXF units are none, inches, feet, mm, cm, and meters.

Angle Increment

The program will break an arc or circle into straight line segments to allow you to snap to them. This indicates the number of degrees the that an imported arc or circle should be broken into.

Copying Model Elements

To Copy Selected Items

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to copy.
3. To copy the selection *linearly* click the **Copy** button and select one of the following options:
    - **Copy by Point Location**
    - **Copy by Increment**
    - **Copy by Angle**

Note:

- Use the "@" symbol to specify multiple equal increments. For example specifying "3@10" will give you 3 copies at 10 units apart.
- You may undo any mistakes by clicking the **Undo** button.

Copy by Point Location

This allows you to copy the selected items by clicking on any two points. The selected items will copy the distance between the two points in the direction of the first to second click.
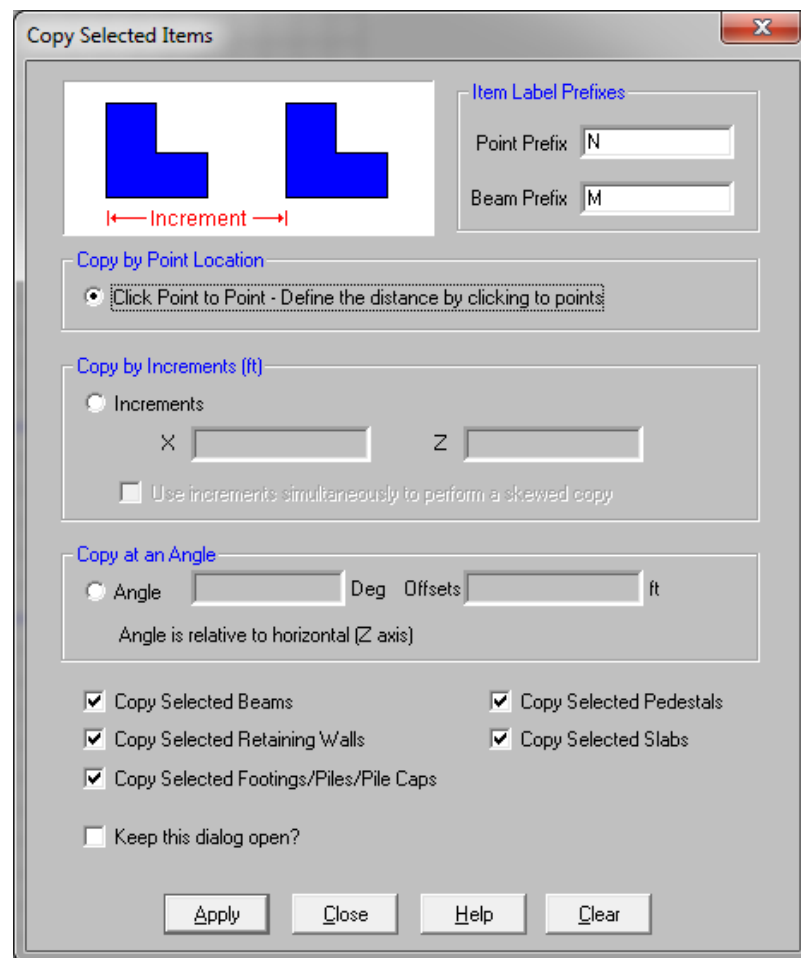
For example, if you click two points: A then B in the model below, the selected items will copy 10 feet shown below.
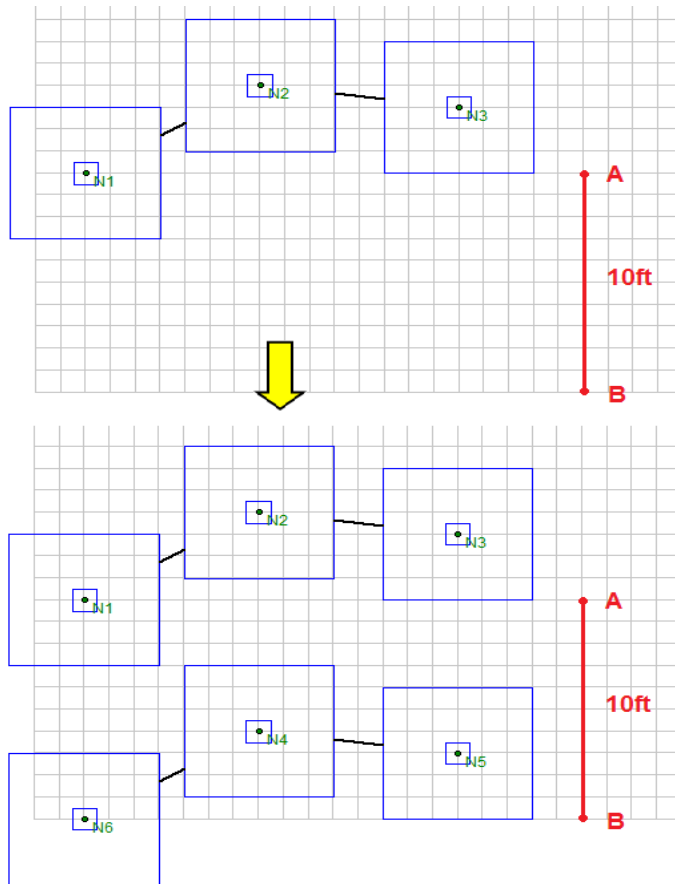
Copy by Increments

This allows you to copy the selected items by entering in increments in any or all of the global directions. The selected items will copy the increment distance(s) that you have entered.

Use the "@" symbol to specify multiple equal increments. For example specifying "3@10" will give you 3 copies at 10 units apart.
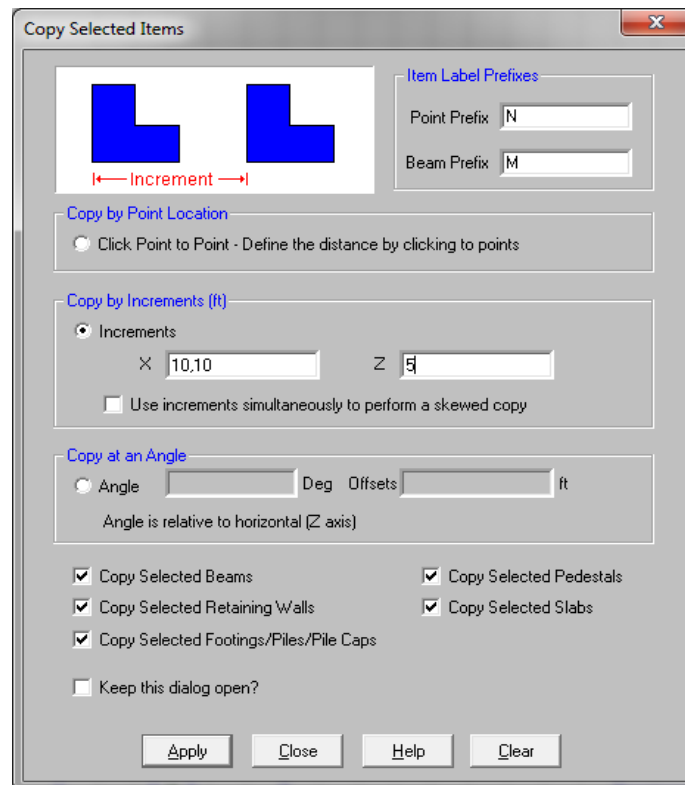
Select the **Use increments simultaneously to perform a skewed copy** checkbox to combine the orthogonal increments into a single resultant increment vector to copy the elements in a direction other than the three global orthogonal directions.

There are many **Copy Selected** checkboxes at the bottom of the dialog to define which elements in the model you wish to copy.

Copy by Angle

This allows you to copy selected items a certain distance at an input angle. Simply enter the angle (relative to the global Z axis) and the increment and your selected items will be copied in that direction.
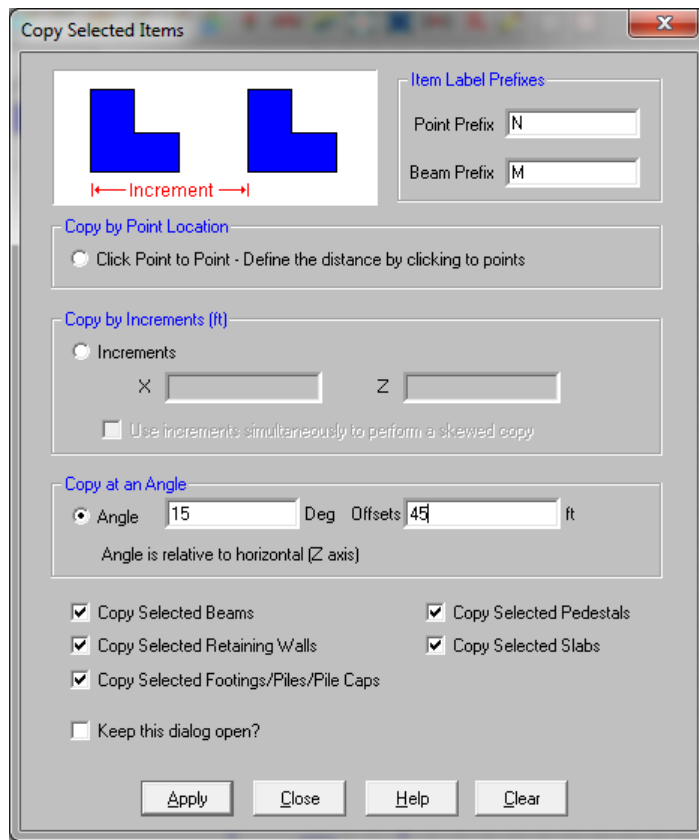
For example, if you enter an angle of 45 degrees and an increment of 15 feet in the model below, it will copy the selected items as shown.
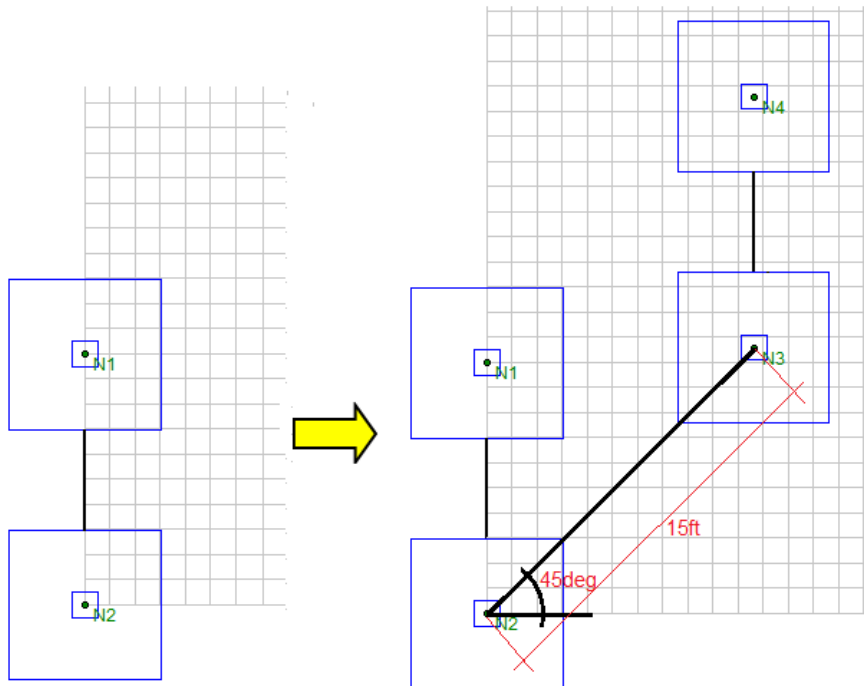
Moving and Rotating Model Elements

To Move or Rotate Selected Items

1. If there is not a model view already open then click ⊞ on the **RISA Toolbar** to open a new view and click ✎ to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to move.
3. To move the selection linearly click the **Move Selected Parts of the Model** ⬚ button and specify the offset distances.

Note:

- You may undo any mistakes by clicking the Undo ↺ button.
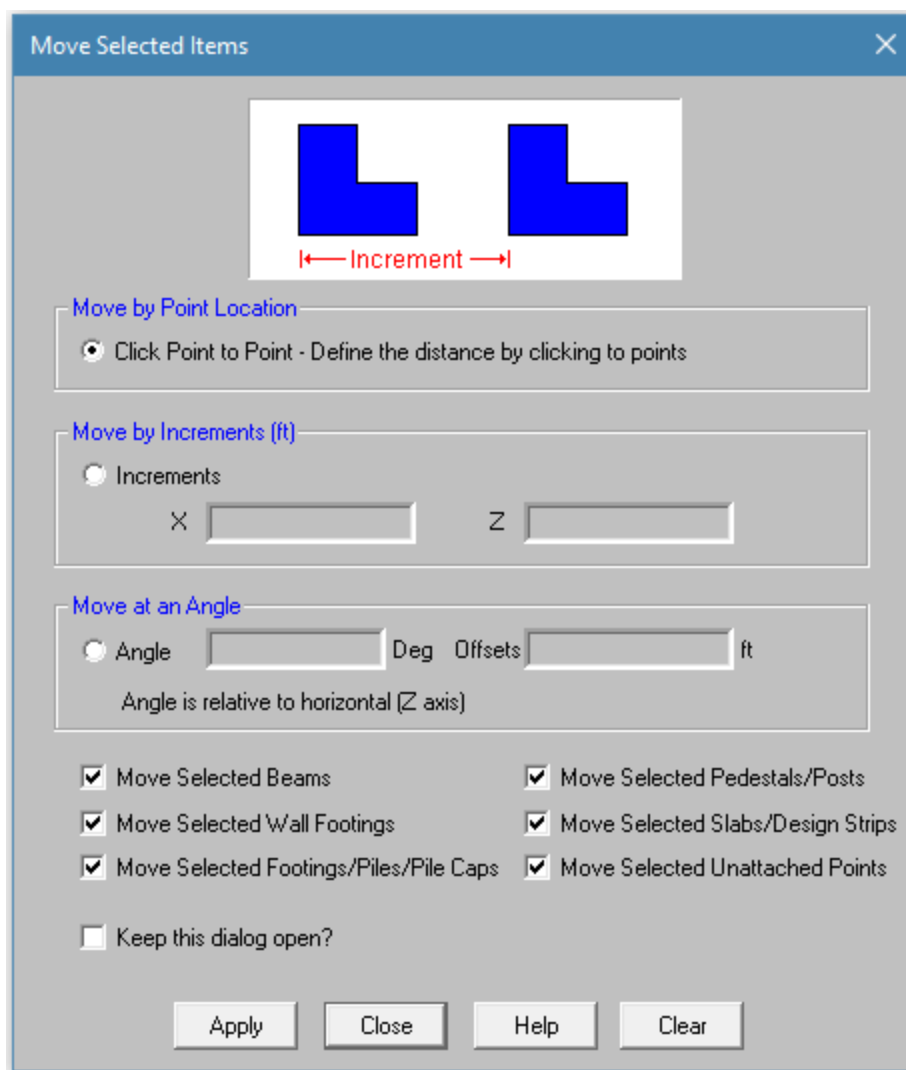
To Move Selected Items

1. If there is not a model view already open then click ⊞ on the **RISA Toolbar** to open a new view and click ✎ to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to move.
3. To move the selection items *linearly* click the **Move** ⬚ button and select on of the following options:
   - **Move by Point Location:** This allows you to move the selected items by clicking on any two points. The selected items will move the distance between the two points in the direction of the first to second click.

- **Move Distance:** This allows you to move the selected items by entering in an increment of the global directions. If you enter both X and Z directions, the selected items will move diagonally.
- **Move at an Angle:** This allows you to move selected items a certain distance at an input angle. Simply enter the angle (relative to the global Z axis) and the increment and your selected items will be moved in that direction.

Note

- You may undo any mistakes by clicking the Undo button.
- The nodes and loads that come from an integrated RISA-3D and RISAFloor into RISAFoundation are protected and cannot be moved using the Move tool.

Merging Model Elements

As you build your model, you may find that you will need to perform a model merge from time to time. In fact, if you count on doing this, you can generally build your models faster and let the model merge feature do a lot of the work for you. See Merge Tolerance to learn about inputting the merge tolerance.

To perform a model merge, select **Merge...** on the **Modify Menu** and the following dialog will appear.
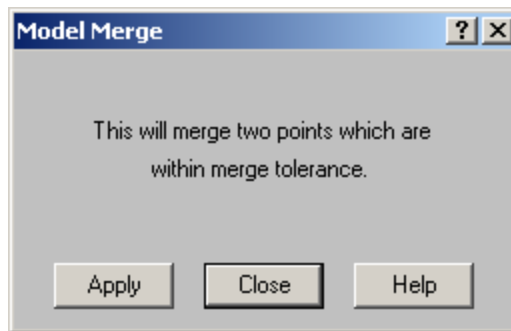
Click **Apply** to continue with the model merge or click **Close** to cancel.

Deleting Elements

You can delete parts of the model based on the current selection state, or you can click on the items you wish to delete individually. If you accidentally delete something you didn't want deleted, you can use the Undo feature to repair the damage.

If you wish to delete based on the current selection state, you must use the check boxes to define the criteria the program will use to perform the deletion. Only items that are selected and that have their check boxes "checked" will be deleted. The choices let you delete points, springs, beams, slabs, soil regions, and/or loads.

Keep in mind that if you delete points, any beams, slabs, footings, etc. attached to the deleted points MUST also be deleted, regardless of whether those elements are selected or not.
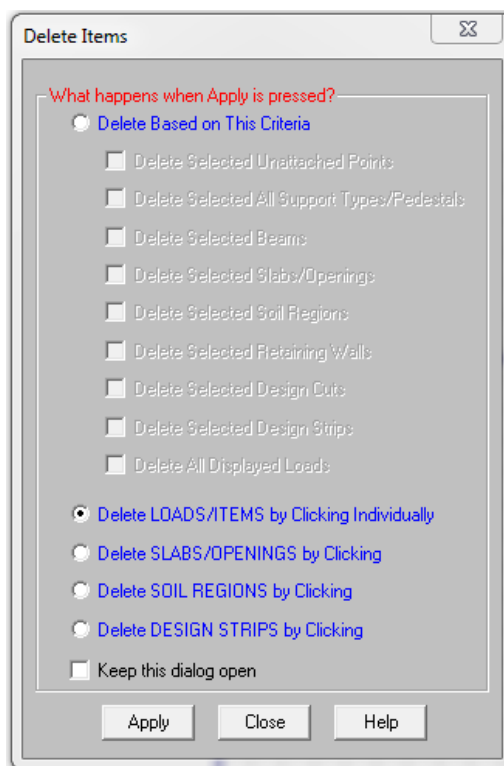
A noteworthy feature is the ability to delete unattached points. Sometimes in the process of modeling you accidentally create unwanted unattached points. This parameter gives you a convenient way to remove these unwanted points.

If you request deletion of displayed loads, you'll get exactly that. Any load currently displayed will be deleted. By controlling what loads are displayed via the **Loads** tab in the **Model Display Options Dialog**, you can easily delete specific types of loads for particular load categories or load combinations.

Alternatively, you can choose to delete items by clicking on them individually. Select the radial button next to the item type that you would like to delete and use the mouse to click on any individual item you wish deleted.

Remember to click the **Apply** button to make your choices active.

To Delete Selected Items

1. If there is not a model view already open then click the ![button] button on the **RISA Toolbar** to open a new view and click ![pencil] to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to delete.
3. To delete the items click the **Delete** ![X] and specify the types of items.

Note

- You may undo any mistakes by clicking the Undo ![undo] button.

Graphic Selection

You may graphically select items in order to view or modify part of the model or to print part of the results. When used in conjunction with the graphic editing features it allows you to quickly model and make modifications to the model. If used with results it allows you to view and print only the input or results that you want.
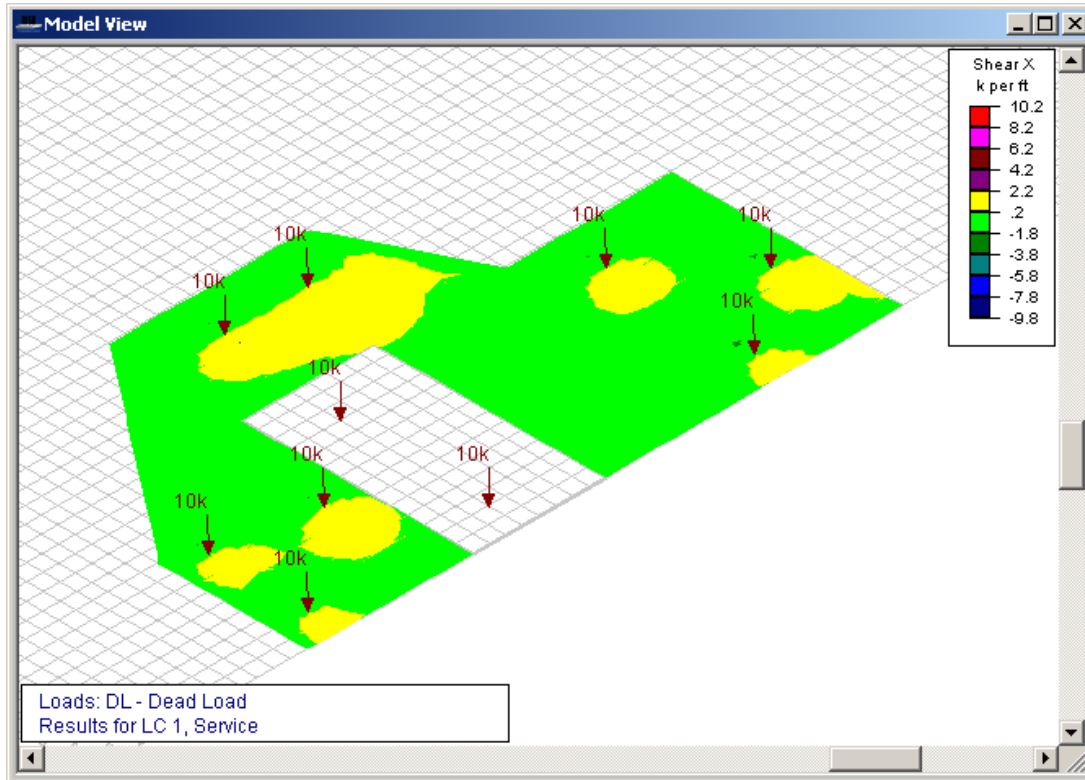
The elements that you see in the model views have two possible states; selected and unselected. By default, all items are selected, and therefore fully displayed. If you unselect any items, they are drawn in a gray or "ghosted" shade. To select or unselect an item simply click on it with the left mouse button. You may also use one of the selection tools below to select multiple items.

Note

- As an alternative to using the selection tools, some operations offer a **Click to Apply** option, which allows you to modify items by clicking or boxing them with the mouse. This is useful when working with a few items.
- If you choose not to display items by turning them off in the **Model Display Options Dialog**, they remain in the current selected state and can still be selected/unselected.

Selection Modes

The **Selection Toolbar** is the vertical toolbar located on the left side of the screen. This toolbar is for selecting points, beams, footings, and slabs in model views. This toolbar will not be available when a model view is not active. You may also access the selection tools from the **View Menu**.

Some of the tools are for one-time applications such as **Select All**. Other tools, such as **Box Select**, place you in a selection mode that remains active until you cancel it. The current mode is indicated by the mouse pointer and by the state of the button. While in a selection mode the button will stay depressed until you click it again, choose another button, or press the ESC key. You may have more than one model view open and may be in different modes in each view.

Note

- There are other types of graphic modes such as **editing** and **viewing** (zooming) modes. The viewing mode overrides the selection mode, which overrides the editing mode. This is so that while you are editing the model you can alter the selected state. You will be returned to the current editing mode when you terminate a selection mode.
- To cancel a selection mode re-click the same button so that it is in the "up" position, or press the ESC key, or click the right mouse button.
- You can cancel a box, line, or polygon selection that is underway by dragging the mouse off the model view while still holding the mouse button down.

Select All and Unselect All

Clicking **Select All** and **Unselect All** tools will select or unselect all of the active points, beams, footings, and platesslabs in the model.

Box Select and Unselect Modes

The **Box Select** and **Box Unselect** tools allow you to draw a box around the items that you wish to select or unselect. Beams and platesslabs must be entirely within the box in order for them to be affected.

Polygon Select and Unselect

The **Polygon Select** and **Polygon Unselect** tools allow you to draw a polygon around the items that you wish to select or unselect. Beams and platesslabs must be entirely within the polygon in order for them to be affected.

Line Select and Unselect

The **Line Select** and **Line Unselect** tools allow you to draw a line through the items that you wish to select or unselect. Any element the line crosses will be affected. This is useful when choosing items between other items.

Inverting Selections

The **Invert Selection** button is used to invert the selected state of the model. When clicked, all selected items are made unselected and all previously unselected items are made selected. For example, this can be very useful when just a few items are to be selected in a large model. Simply click on the desired items to unselect them, then click the **Invert Selection** button to make them selected and everything else unselected.

Criteria Selections

The **Criteria Selection** button allows you to select items based on a wide range of criteria such as coordinates, labels and many other conditions. When the **Criteria Select** button is clicked, you will be presented with the Criteria Selection Dialog with options grouped by tabs across the top. Each tab represents groups of criteria that you may use to refine your selection.

The options are numerous making it easy to quickly achieve complicated selections. This is a powerful tool so it is worth taking the time to experiment with the options so that you will know how to use it to your advantage. The various tabs in the dialog are described below.

Supports/Beams

You may **Select** or **Unselect** Points as follows.

Point Range - You may specify a range of point labels. Specifying only one label selects just that one point.

Point Properties - You may specify a boundary, footing, pile, or pile cap type to select.

Beam Range - You may specify a range of beam labels. Specifying only one label selects just that one beam.

Beam Properties - You may select beams based on material, shape, or design rule

Note

- You must also make sure that the **Use?** checkbox is turned on if you wish to use that criteria.
- If no criteria is specified, all the Points/Beams will be selected/unselected.

Pedestals

Pedestal Range - You may specify a range of pedestal labels. Specifying only one pedestal selects just that pedestal.

Pedestal Properties - You may select pedestals based on a range of angles or heights, and/or select them by material, shape, or design rule.

Wall Range - You may specify a range of retaining wall labels. Specifying only one retaining wall selects just that retaining wall.

Wall Properties - You may select retaining walls based on material or retaining wall definition as well.

Slabs/Soil Regions/Design Strips

Slab/Soil Region/Design Strip Range - You can specify a range of objects. Specifying only one selects just that object.

Slab Properties - You can select slabs based on a range of thickness, and/or by material.

Soil Region Properties - You can select soil regions based on a range of modulus and/or bearing capacity.

Design Strip Properties - You can select design strips based on a range of angles within the horizontal plane, and/or by design rule.

Note

- You must also make sure that the **Use?** checkbox is turned on if you wish to use that criteria.

Coordinates
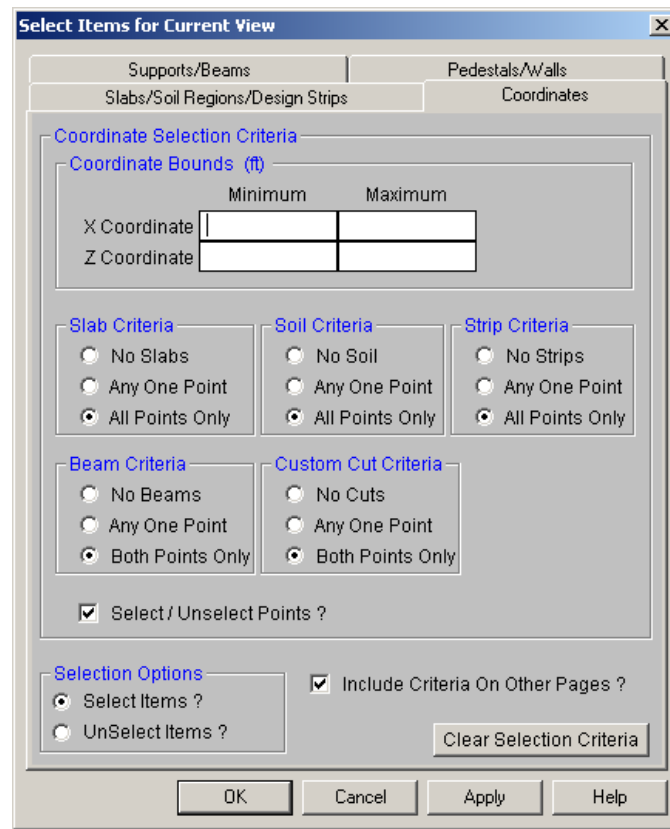
You may select or unselect elements based on coordinate criteria and combine this with the criteria on the other tabs.

Coordinate Bounds – You may specify minimum and maximum bounds in the global directions. Items within these bounds AND meeting the criteria in the other groups will be selected/unselected.

Select/Unselect Points – You may specify that joints in the coordinate range be selected or not.

Include Criteria On Other Pages – This option allows you to include selection criteria from the other pages (Beams, Walls, Slabs, Strips, etc) and combine it with the coordinate range specified on this page.

Note

- If the "Include Criteria On Other Pages" option is selected, all criteria will be applied together so that the affected items will meet all of the criteria. For example if you specify a range of joint labels and coordinates, only the joints within the coordinate bounds and within the label range will be selected/unselected.

Lock & Dim Lock

Click the **Lock Unselected** button to cause all currently unselected items to stay unselected and be visually removed from the current model view. This is useful when you are editing or printing a portion of a model and need to clear the model view of all items not involved.

To Unlock the unselected members in the model view, click the **Unlock Unselected** button and the unselected items that were previously "removed" from the model view will be returned to the view in the unselected state.

Saving Selections

You can save and recall various selection states for a model. If the model is altered after a selection state has been saved, the saved selection state will also be altered. Any new items (points, beams...) will be set to "selected" in any selection states saved prior to the creation of the new item.

To save a selection state, click the **Save Selection State** button on the **Selection Toolbar**.
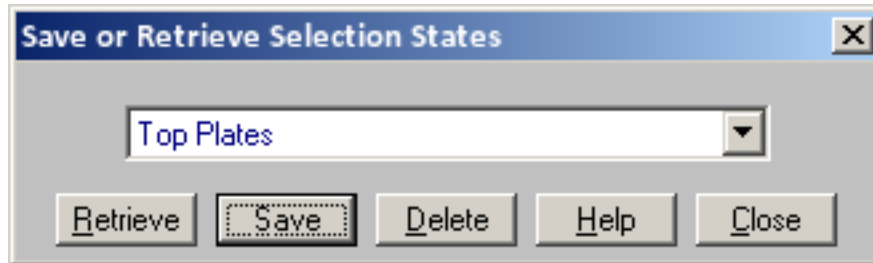
Click the **Save** button and provide a name for the saved selection. You can have up to 15 different saved selections in a model. To retrieve a saved selection, choose the selection state from the drop down list and click the **Retrieve** button. To delete a saved selection, choose the selection state from the drop down list and click the **Delete** button.

Help Options

RISA Tech, Inc. has, and will, put a great deal of effort into assisting you in getting your work done as quickly as possible. This includes providing many ways that you can get help in understanding the software.

Electronic Help File

The **Help File** is designed to help you get your work done as quickly as possible and is intended to provide:

- Procedures that lead users through the steps of completing tasks
- Context-sensitive Help topics that provide users with quick descriptions of items on their screens
- Troubleshooting topics that guide users through solutions to common problems
- Extensive discussions for a thorough understanding of engineering and modeling topics
- Easy access to related topics

The electronic help file can be accessed by clicking the **Help File** button on the **RISA Toolbar**. A new window containing a Table of Contents will be opened. Click on any item in the Table of Contents for extensive information on the topic.

Context Sensitive Help

Context Sensitive Help is help that a user can access in context while working in a program. It provides you with the information you need where and when you want it.

You may get detailed help when working in a dialog, spreadsheet or graphic view by pressing **F1**. This will launch a Help File window displaying the topic that is related to the window in which you are working. The topic will be explained and links to related topics may also be provided.

RISA Tech, Inc. Online

Our website, www.risa.com, provides various support information and documents.

Visit RISA Tech, Inc. on the web for:

- Download program Manuals (General Reference or Tutorial)
- Check our website for the latest updates - When a bug is discovered it is posted on the web site along with possible work-around procedures and/or service releases to update your software.

Tool-tips

Are you uncertain what a toolbar button is for? Simply hold your mouse pointer over that button without clicking. **Tool-tips** are displayed that will explain what the button will do should you decide to press it.

Tutorials

The comprehensive **Tutorials** guide you through using most features. It is a real-world example of building and solving a model, making changes, and optimizing. This is the best way to quickly get up and running. The Tutorials are designed to be read in two ways. If you are already familiar with structural modeling in general you can skip the supporting text and read only the underlined action items to quickly move through the tutorial. If you want more thorough explanations of the modeling process you may read all or some of the supporting text as you see fit.

Hot Rolled Steel - Databases

Shapes are organized by **Shape Type** and **Database / Manufacturer**. Common shapes are supported such as wide flanges, tubes, pipes, channels, etc. You may type in the names directly, select shapes from these databases or add your own shapes.

RISA currently supports the following common Hot Rolled steel databases: AISC (United States), Australian, British, Canadian, Chilean, Chinese, European, Indian, and Mexican.

Note:

- Older AISC shapes which are no longer part of the AISC database will be automatically moved / exported to a new AISC_backup database during installation or update.
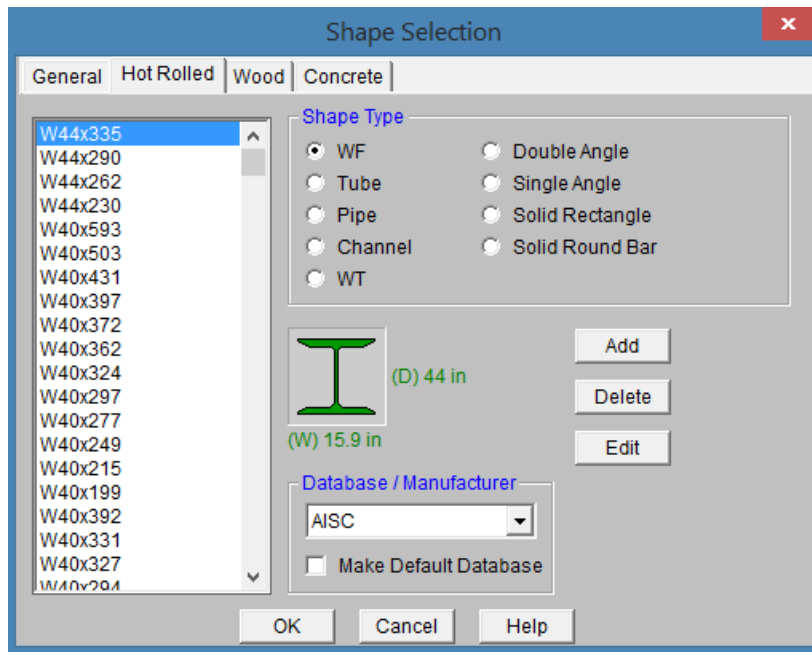
To Select a Hot Rolled Database Shape

1. From the **Hot Rolled** tab on the **Section Sets Spreadsheet**, or the **Primary** tab of the **Members Spreadsheet**, move the cursor to the **Shape** field and click ▶.
2. Specify the database and shape type you wish to use and then select from the list of available shapes by clicking on ▾.

To Add a Database Shape

1. On the **RISA Toolbar** click the **Edit Shape Database** ⊥ button.
2. Select the Hot Rolled tab, then specify the database and shape type you wish to add and click the **Add** button.
3. Specify a name for the shape and fill in the **Basic Properties.**
4. Click **Calc Props** to determine the shape properties.
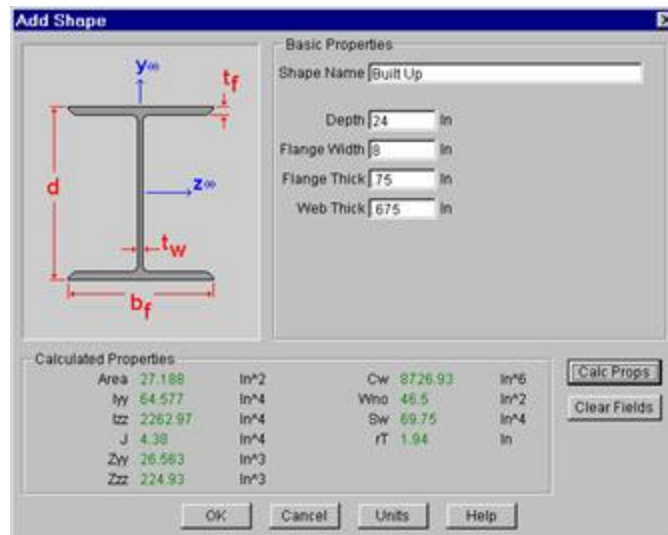
Note

- Alterations to the shape database are not permanent unless you agree to save them. Changes that are not saved only remain valid for the current session and will not be present the next time you start RISA.
- New shapes are added to the bottom of the database.
- To delete a shape, specify the database and shape type you wish to delete and then click the **Delete** button.
- To edit a shape, click the **Edit** button and edit the shape properties. Values can only be manually edited here, nothing will be recalculated. If you wish to have all the values for a shape recalculated, you will need to delete the shape and then add it again with the new properties.

Wide Flange Database Shapes

For the AISC database, wide flange shapes are called out by the designation given them in the steel manuals. For example, if you wanted to use a W10x33 you would enter W10X33 as the shape name in the database shape field. M, S and HP shapes are also available. Trade Arbed shapes are called out similar to AISC shapes but with a "_ARB" suffix. I.e. to call a Trade Arbed W12X96 would enter W12X96_ARB as the shape name in the database shape field. Canadian and British shapes use the same format as the AISC shapes, but their values are metric. The depth is called out in millimeters and the mass per length is kg/meter.

Tube Database Shapes(Hollow Rectangular shapes)

The HSS tube properties are also available in the AISC database. The prefix for these tube shapes is "HSS". The syntax is "HSSdepthXwidthXthick", where "depth" is the tube depth, "width" is the tube width and "thick" is the tube thickness in number of 1/16ths. The nominal wall thickness is always used to call out a HSS tube, even though the design wall thickness will vary based on the manufacturing process for the tube. Tubes manufactured using the ERW process will use .93 times the nominal wall thickness as their design thickness. Tubes manufactured using the SAW process will use the full nominal thickness as their design thickness. For example, an HSS12X10X8 would be a 12" deep, 10" wide tube, and a have a design wall thickness of .465" = .93*1/2" (8/16ths) . A HSS32X24X10 would be 32" deep by 24" wide, and have a design wall thickness of 5/8"(10/16ths)

For the Canadian database, tubes also have a "HSS" prefix and the dimensions are all called out in millimeters. British shapes use the prefix "SHS" for square tubes and "RHS" for rectangular tubes.

Note:

- The prefix for older AISC tube shapes is "TU". These shapes reflect the properties that were published with the older AISC 9th edition. As such, these shapes may only exist in the AISC_backup database.
- Tubes using the TU prefix will have a design wall thickness of the nominal wall thickness. The syntax is "TUdepthXwidthXthick", where "depth" is the tube depth, "width" is the tube width and "thick" is the tube thickness in number of 1/16ths. For example, TU16X12X8 would be a 16" deep, 12" wide tube with a thickness of 1/2" (8/16ths).

Pipe Database Shapes

Pipe shapes, which are hollow circular shapes, are entered as on-line shapes. The syntax for these shapes is "PIdiaXthick", where "dia" is the pipe outside diameter and "thick" is the pipe thickness (in inches or centimeters). For example (assuming US Standard units), PI10X.5 would be a 10" diameter pipe with a wall thickness of 1/2".

Channel Database Shapes

Channel shapes are entered with the "C" or "MC" prefix. For example C15X50 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meter. Double channels are entered with the "2C" or "2MC" prefix.

Tee Database Shapes

The Tee shapes are entered with the "WT", "MT" or "ST" prefix. For example WT15X74 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meters.

Double Angle Database Shapes

These shapes are entered with the prefix "LL". The syntax is "LLbackXflangeXthickXspace" where "back" is the back to back leg length, "flange" is the single angle flange leg length, "thick" is the angle thickness in number of 1/16ths and "space" is the space between the angles in 1/8ths. For example, **LL6X3.5X5X3** would be L6X3.5 angles 5/16" thick, long legs back to back with a spacing of 3/8". For the Canadian and British shapes, all the dimensions are called out in millimeters.

Single Angle Database Shapes

Angles are entered with an "L" prefix. The syntax is "LlongXshortXthick", where "long" is the long leg length, "short" is the short leg length, and "thick" is the thickness, in number of 1/16ths. For example, L9X4X8 is a 9" by 4" angle 1/2" (8/16ths) thick. The thickness is entered as 8, because the number of 1/16ths in 1/2 is 8. For the Canadian and British shapes, all the dimensions are called out in millimeters.

Note:

- **rZ** is the Radius of Gyration along the rotated (Principal) vertical axis (local y). The Canadian CSA code calls this variable ry' but RISA calls it rZ based on the naming convention of the AISC 360.

Per CSA S16-16 (11th edition):

-
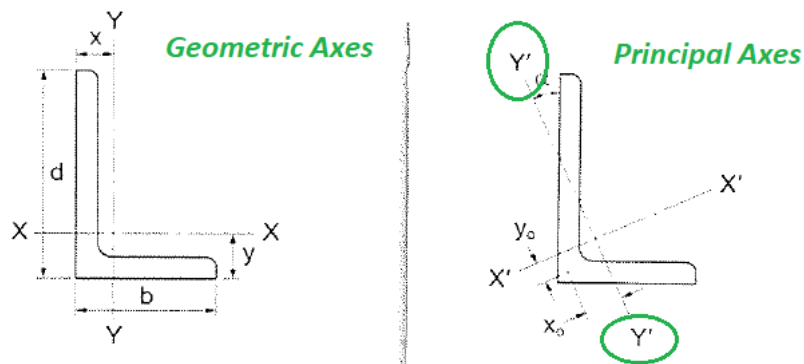
Per AISC 360-16 (15th edition):

Solid Rectangular Shapes

These shapes can be defined as on-line shapes. The syntax is "REhtXbase", where "ht" is the rectangle height and "base" is the rectangle base (in inches or cm). For example, RE10X4 would be a 10" deep, 4" width rectangular shape (assuming US Standard units). These shapes can also be defined in the Shape Editor. When defined in the Shape Editor the depth of the solid rectangular section must always be greater than or equal to the width.

Solid Circular Shapes

These shapes are defined as on-line shapes. The syntax is "BARdia", where "dia" is the circle diameter. For example (assuming metric units), BAR2 would be a circular bar with a diameter of 2 cm.

Buckling-Restrained Brace Database Shapes

Buckling-restrained brace (BRB) shapes are only available in RISAFloor-RISA3D integration models. They can be selected by choosing the BRB shape type and one of the BRB-related databases. Currently, only CoreBrace database is supported.

When CoreBrace database is chosen, the syntax for CoreBrace BRB shapes is "CoreBRB_area", where "CoreBRB" is the prefix for the manufacturer and shape, and "area" is the steel core area. For example, CoreBRB_0.50 refers to a BRB shape manufactured by CoreBrace with a 0.50 in^2 steel core area. Other properties such as the casing size of the brace are based on the data provided by the manufacturers.

BRBs are not prefabricated with specific core sizes, casing sizes, or brace lengths. Instead, they are custom-fabricated for each project. Therefore the BRB properties in RISA-3D may not be the final and shall be carefully reviewed by BRB manufacturers.
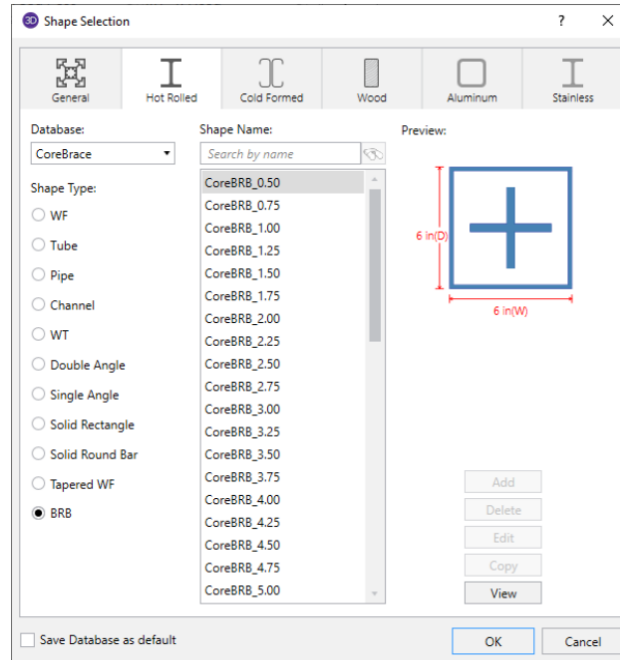
Note:

- The BRB shape database is only available for RISAFloor/RISA-3D integration models.

Loads

Loads can be defined and applied to the model graphically or from within the spreadsheets. There are a number of different **Load Types** that may used when loading the model. For more information about the application of each Load Type (Area Load, Line Load, and Point Load), see Loads - Load Types.

As you define the loads in the model, you will notice that each individual instance of a load type must be assigned to a particular **Load Category**. You will then use these load categories to define the various **Load Combinations** that are to be solved when a solution is performed. See Loads - Load Combinations for more information.

You may view the loads on your model for each load category or load combination. This is an excellent way to verify the loads applied to the model. See Model Display Options – Loads to learn more about displaying loads in model views.

Self Weight (Gravity Load)
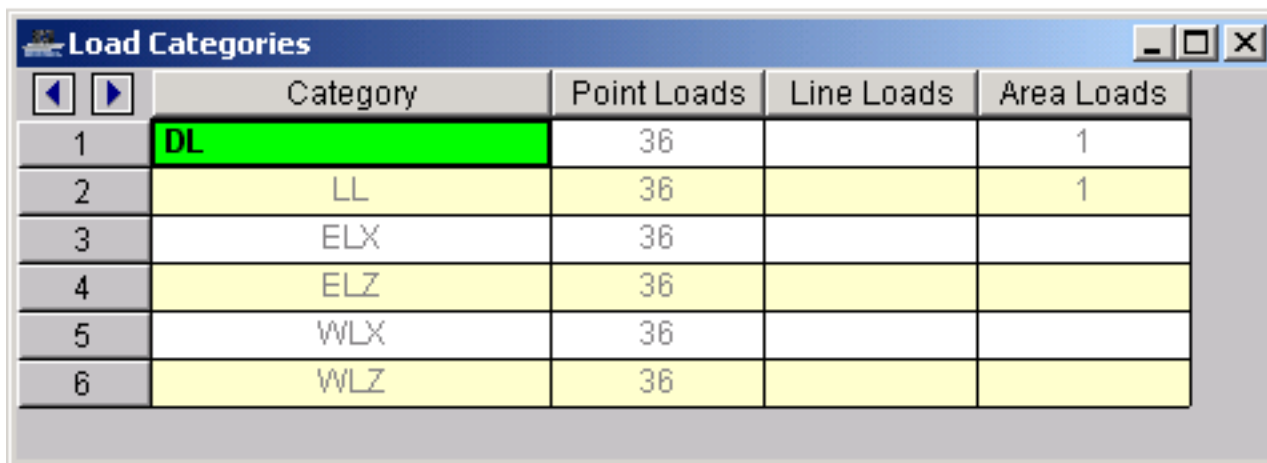
The **Self Weight** of slabs and beams is automatically calculated and included as part of the Dead Load Category (DL).

The 'Self Weight' is applied as a uniform distributed load along each beam and as an area load for the slab. The magnitude of the beam loads is the cross sectional area times the material weight density.

Note

- If you do not want the program to apply a member's self weight, create that member using a weightless material. See Material Properties for more information.

Load Categories

When loads are defined in RISAFoundation they are automatically grouped based on their load categories. These are the basic building blocks of the final load combinations applied to the structure.

Information about Basic Load Cases (BLC's) is recorded on the **Load CategoriesSpreadsheet** and the loads themselves are recorded in the load spreadsheets for each load type. The **Load Categories** spreadsheet may be accessed by selecting **Load Categories** from the **Spreadsheets** menu.

The first column in this spreadsheet merely displays the Category for which the rest of the information in the row relates to. The remaining columns display the quantity of each type of load that is contained in that category. You may not edit these values but you may click on the quantities to open the spreadsheet and view the loads that it represents. For example, the previous figure has 36 point loads and one area load part of the DL (Dead Load)

category. Clicking on the number 36 with the mouse will open the Point Loads spreadsheet and display these loads whereas clicking on the 1 will open up the Area Loads spreadsheet and display that load.

At solution time, the model will be solved against **Load Combinations** that are built from the following **Load Categories**. The load categories and descriptions below are based on the definitions and load combinations listed in various building codes.

Load Category	Description
DL	Dead Load
LL	Live Load
EL	Earthquake Load
WL	Wind Load
SL	Snow Load
RLL	Roof Live Load
LLS	Live Load Special (public assembly, garage, storage, etc.)
TL	Long Term Load (creep, shrinkage, settlement, thermal, etc.)
SLN	Snow Load Non-shedding
HL	Hydrostatic Load
FL	Fluid Pressure Load
RL	Rain Load
PL	Ponding Load
EPL	Earth Pressure Load
IL	Impact Load
OL1 - OL100	Other Load 1 - 100 (generic)
ELX, ELY, ELZ	Earthquake Load along global X-axis, Y-axis, Z-axis
WLX, WLY, WLZ	Wind Load along global X-axis, Y-axis, Z-axis
WL+X, WL+Y, WL+Z	Wind Load along positive global X-axis, Y-axis, Z-axis
WL-X, WL-Y, WL-Z	Wind Load along negative global X-axis, Y-axis, Z-axis
WLXP1, WLYP1, WLZP1	Partial Wind Load 1 along global X-axis, Y-axis, Z-axis
WLXP2, WLYP2, WLZP2	Partial Wind Load 2 along global X-axis, Y-axis, Z-axis
ELX+Z, ELX+Y	Eccentric Earthquake Load along global X-axis shifted along positive global Z-axis, Y-axis
ELX-Z, ELX-Y	Eccentric Earthquake Load along global X-axis shifted along negative global Z-axis, Y-axis
ELZ+X, ELZ+Y	Eccentric Earthquake Load along global Z-axis shifted along positive global X-axis, Y-axis
ELZ-X, ELZ-Y	Eccentric Earthquake Load along global Z-axis shifted along negative global X-axis, Y-axis
ELY+X, ELY+Z	Eccentric Earthquake Load along global Y-axis shifted along positive global X-axis, Z-axis

Load Category	Description
ELY-X, ELY-Z	Eccentric Earthquake Load along global Y-axis shifted along negative global X-axis, Z-axis
SX, SY, SZ	Response Spectra Results (from RISA-3D)
NLX, NLY, NLZ	Notional Load along global X-axis, Y-axis, Z-axis

Copying Load Categories

You may copy the loads from one Category into another Category. This can be useful when one load is similar to another and can be entered quickly by copying a load category and then making changes to the copy with features such as block math.

To do this, open the **Load Categories** spreadsheet and then click [icon]. Specify what category to copy loads from and which category the loads are to be copied into. You may further specify which types of loads are to be copied. For example, if you check **Point Loads** and uncheck all the other load types, only the joint loads will be copied.

Any loads copied into a Category will be added to any loads that may already be in that category.

Deleting Load Categories

You may automatically clear all the loads in a Category. To do this, open the **Load Categories** spreadsheet and then click the **Delete Category** [icon] button on the **Window** toolbar. Select the Category you wish to delete.

Modifying Loads

Modifying point loads, line loads, and area loads can be done from within their respective spreadsheets. Typically, the modification of load geometry and magnitude may be performed in this way. However, area load geometry may only be modified graphically (deleted and redrawn). Select **Loads** from the **Spreadsheets Menu** and then choose the spreadsheet that you wish to modify. You may then move through different Floors using the drop down list in the spreadsheet.You may use spreadsheet operations to help you modify the loads. See Spreadsheet Operations for more information.

Deleting Loads

Deleting loads may be done manually within the spreadsheets or graphically using the **Delete** [icon] button on the **Drawing Toolbar**. Select **Loads** from the **Spreadsheets Menu** and then choose the spreadsheet that you wish to modify or delete. You may use spreadsheet operations to help you modify or delete the loads. See Spreadsheet Operations for more information.

Loads - Load Types

This section discusses the various **Load Types** that are available for applying loads to a model.

Area Loads

Area Loads are loads that are spread out over the surface of a slab or a portion of a slab. RISAFoundation allows area loads to be oriented along a vertical axis perpendicular to the XZ plane, with positive loads being downward and negative loads being upward. Loads may be input manually in the spreadsheets or drawn graphically in a model view. The area loads can be uniform or tapered.

Drawing Area Loads

To apply area loads click on the **Drawing Toolbar** to open the **Draw Area Loads Dialog**. Select the Uniform or Tapered tab, select Load Category, and input the magnitude. Apply the area load by clicking from point to point to define the perimeter of the area load. Uniform area loads can be defined as any polygon. Tapered area loads must be either rectangular or triangular. Area loads must be drawn within the boundaries of a slab.
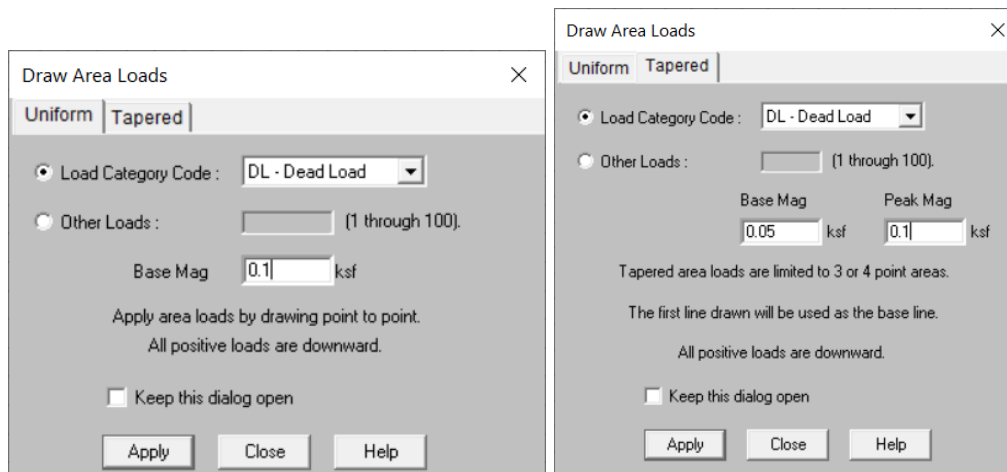
To Apply Area Loads
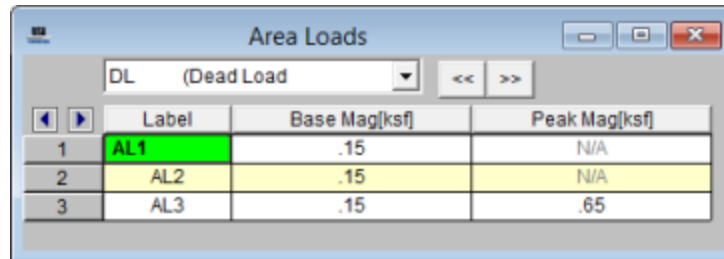
1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Area Load** button on the **Drawing Toolbar** and define the load in the **Draw Area Loads Dialog**.
3. Click **Apply**.
4. Draw the load by clicking point to point to define the perimeter of the area load.
5. For Tapered Area Loads, click on four grid or point locations to define the area. For three sided areas click on the last point twice.
6. To stop drawing altogether, click the right mouse button or press the Esc key.

Note:

- To apply more loads with different parameters, press CTRL-D to recall the **Draw Area Loads Dialog** .
- You may undo any mistakes by clicking the Undo button.

Area Loads Spreadsheet

The **Area Loads Spreadsheet** defines the uniform area load magnitudes and may be accessed by selecting **Loads ▸ Area Loads** from the **Spreadsheets Menu**. While the load magnitudes are defined in this spreadsheet, they can only be assigned graphically.

The first column is the area load **Label**. This label is a description for your own use.

The **Type** pulldown entry is used to select the load category for the area load. For a list of the available load categories, see [Loads - Load Types](#).

The **Base Mag** and **Peak Mag** columns are used to view/update the magnitude of the area load. The units are listed in the column header.

Note:

- If you draw a surface load as a Uniform load you can not later make it a tapered load. You will have to delete and re-draw it as a tapered surface load.

Line Loads

Line Loads are linear loads that can be placed anywhere on a slab or beam and can be of uniform, stepped, or varying magnitude such as triangular or trapezoidal. RISAFoundation allows line loads and moments which act along/about the Global Axes. For Y-direction loads a positive magnitude represents a downward load. Line loads may be assigned manually in the **Line Loads Spreadsheet** or drawn graphically in a model view by clicking the **Draw Line Loads** � button on the **Drawing Toolbar**.

Drawing Line Loads

To draw line loads graphically, click the 🚛 button on the **Drawing Toolbar** to open the **Draw Line Loads Dialog**. Enter the start magnitude and the end magnitude and select the category you wish to use. Apply the line load by clicking on grid or point locations in the model view. You may apply a line load anywhere on a slab or beam.
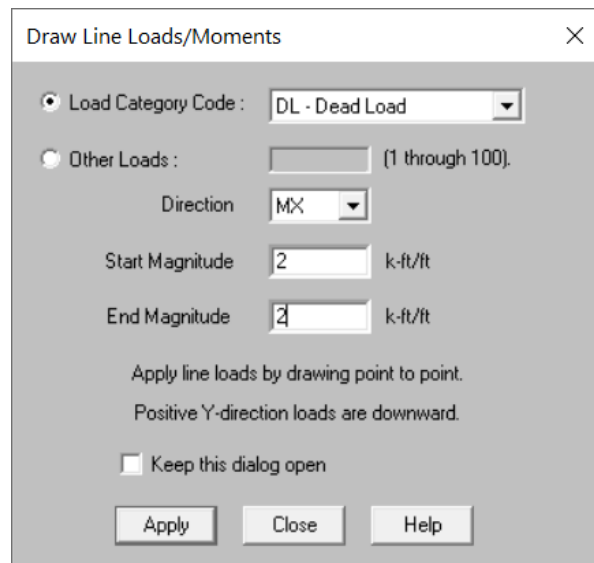
To Apply Line Loads

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Line Loads** button on the **Drawing Toolbar** and define the load in the **Draw Line Loads Dialog**.
3. Click **Apply** and draw the line load by clicking on consecutive drawing grid or point locations.
4. To stop drawing altogether, click the right mouse button, or press the Esc key.

Note

- To apply more loads with different parameters, press CTRL-D to recall the **Draw Line Loads Dialog**.
- You may also specify or edit line loads in the **Line Loads Spreadsheet.**
- You may undo any mistakes by clicking the **Undo** button.

Line Loads Spreadsheet

The **Line Loads Spreadsheet** records the line loads on all slabs and beams and may be accessed by selecting **Loads ▸ Line Loads** on the **Spreadsheets Menu**.

The first column contains the **Start Point** of the load and the second column contains the **End Point** of the load.

The **Type** entry is used to select the load category for each line load. For a list of the available load categories, see Loads.

The **Direction** entry is used to select the direction of the load, relative to the Global Axes.

The **Start Magnitude** and **End Magnitude** columns are used to enter the magnitudes of the line load. If the start and end magnitudes of a line load are NOT the same, a tapered line load will be created. The units are listed in the column header.

Point Loads and Moments

Point Loads and **Point Moments** are concentrated loads and may be placed anywhere on a slab or beam. Point loads may be assigned manually in the **Point Loads and Moments Spreadsheet** or drawn graphically in a model view by clicking the **Draw Point Loads/Moments** button on the **Drawing Toolbar**.

RISAFoundation allows point loads to be oriented along any of the global axes or applied as a point moment about those axes. Point loads with a positive magnitude are applied in the direction of positive global axis they are assigned to and negative loads are applied in the direction of the negative global axis. The exception to this is that loads applied along the global Y-axis are reversed (i.e. positive loads are applied in the direction of the negative global Y-axis and negative loads are applied in the direction of the positive global Y-axis). All point moments are applied about the global axes according to the right hand rule, including MY moments.

Drawing Point Loads/Moments

To draw point loads graphically, click the button on the **Drawing Toolbar** to open the **Draw Point Loads/Moments Dialog**. Enter the magnitude of the point load and select the category you wish to use. Select the direction of the point load from the choices in the Direction drop down list. Apply the point load to all selected points or by clicking or boxing grid or point locations in the model view. You may apply a point load anywhere on a slab or beam.
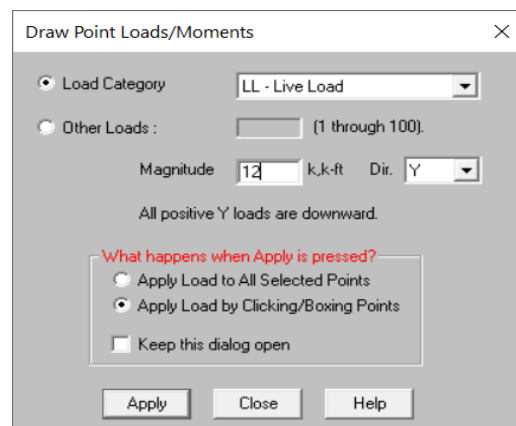
To Apply Point Loads or Moments

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Point Loads/Moments** button on the **Drawing Toolbar** and define the load in the **Draw Point Loads/Moments Dialog**.
3. You can apply the point load to all selected points and click **Apply**.

   You can apply the point load by clicking **Apply** and clicking or boxing grid or point locations in the model view.

4. To stop drawing altogether click the right mouse button, or press the Esc key.

Note

- To apply more loads with different parameters, press CTRL-D to recall the **Draw Point Loads Dialog**.
- You may also specify or edit point loads in the **Point Loads and Moments Spreadsheet.**
- You may undo any mistakes by clicking the **Undo** button.

Point Loads and Moments Spreadsheet

The **Point Loads and Moments Spreadsheet** records the point loads on all slabs and beams and may be accessed by selecting **Loads ▸ Point Loads** on the **Spreadsheets Menu**.

The first column is the **Label** of the point to which the point load is applied.

The **Type** entry is used to select the load category for each point load. For a list of the available load categories, see [Loads](#).

The **Direction** column defines the direction of the point load or moment. The available direction options are listed in a drop down list within each cell.

The **Magnitude** column is used to enter the magnitude of the point load or point moment. The units are listed in the column header.

Loads - Load Combinations

During solution the model is loaded with a combination of factored Load Categories. These combinations, load factors, and other parameters are defined on the **Load Combinations Spreadsheet**. Most standard load combinations are included in the program. See [Solution](#) to learn how to solve load combinations.

To Add Load Combinations Manually

1. From the **Spreadsheets Menu** select **Load Combinations**.
2. Enter load combinations by pairing loads in the **Category** fields with factors in the **Factor** fields.

To Add Auto-Generated Building Code Combinations

1. From the **Spreadsheets Menu** select **Load Combinations**.
2. Select the **LC Generator** button from the **Window Toolbar**.
3. Select the Load Combination **Region** and **Code** from the drop down lists provided in the Gravity tab.
4. Modify the generated combinations and options in the spreadsheet as necessary.

Note

- The generated building code combinations are made up of Load Categories and Factors. Loads that are not assigned to these categories will not be included in the combinations upon solution.
- All generated combinations are added and marked for the envelope solution. You may remove combinations from the envelope after adding them.
- Verify the Footing ABIF (allowable bearing increase factor) settings for combinations after you add them.
- You may save any preferred load combinations as the default by clicking the **Save As Defaults** 💾 button on the **Window Toolbar**.

To Solve Load Combinations

- To solve load combinations, first mark the combinations to be solved by checking the **Solve** box on the **Load Combinations** spreadsheet. Click the **Solve** = button on the **RISA Toolbar**.

Load Combinations Spreadsheet

The **Load Combinations Spreadsheet** records the combinations of loads for solution and may be accessed by selecting **Load Combinations** on the **Spreadsheets Menu**.

	Label	Solve	Service	ABIF	SF	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor
1	IBC 16-1	☑	☐			DL	1.4	HL	1.4						
2	IBC 16-2 (a)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	RLL	.5	HL	1.6
3	IBC 16-3 (a)	☐	☐			DL	1.2	RLL	1.6	LL	.5	LLS	1		
4	IBC 16-8	☑	☑		1.5	DL	1								
5	IBC 16-9	☑	☑		1.5	DL	1	LL	1	LLS	1				
6	IBC 16-10 (a)	☑	☑		1.5	DL	1	RLL	1	HL	1				
7	IBC 16-10 (b)	☐	☑		1.5	DL	1	SL	1	HL	1				
8	IBC 16-10 (c)	☐	☑		1.5	DL	1	RL	1	HL	1				
9	IBC 16-11 (a)	☐	☑		1.5	DL	1	LL	.75	LLS	.75	RLL	.75	HL	1

The first field, the **Label**, is strictly for the user's reference. Enter any descriptive label you wish and it may be displayed with the results when the combination is solved.

The **Solve** box is used to indicate which combinations should be included the solution. See [Solution](#) for more information.

Check the **Service** box if the load combination should be used for the calculation of service level soil bearing checks, footing stability checks, and service load deflections of beams and slabs. Load combinations for which the **Service** box is NOT checked will be used for the strength design of slab, beam, and footing reinforcement.

The **Service** checkbox is also used for masonry wall footings, depending on which masonry code is selected. If the ASD code is selected then only combinations with the checkbox checked will be used for the masonry wall design. If the Strength code is selected then only combinations with the checkbox unchecked will be used for masonry wall design.

The **ABIF** column is used to set the allowable bearing increase factor used for the soil bearing check in slab, footing, and wall design. If this field is left blank, a default of '1.0' will be used. See RISAFoundation Design Combinations for more information.

The **SF** column is used as the value with which overturning and sliding of spread footings and retaining walls are checked against. This value is also used in the slab overturning check. This value is only used in service load cases. If the results do not meet the safety factor, they will be denoted as failing with red text.

The next eight pairs of fields (**Category**, **Factor**) are for defining what load categories are to be part of the combination, along with factors for each. Select load categories from the drop down lists in the **Category** columns of the spreadsheet. The following are the entries you can use in the **Category** field:

Entry	Description
Category	Enter a category code such as DL, LL, etc to include all loads in that category
Lnn	Enter "Lnn" to nest the loads from another combination, where "nn" is the number of the other combination, i.e. "L3" means include all loads from load combination 3.
Sn	Includes response spectra results for the global direction "n", I.e. enter "SX" to include response spectra results calculated in the global X direction, "SY" for global Y , etc.

There are also a number of Category Prefixes that may be included in your load combinations. You can have these generated by the LC Generator or you may click on the red arrow button in the Category cell to select them manually. These are defined in the Seismic tab of the Model Settings. They are explained below:

Prefix	Description
SF	Scaling Factor - applied to RSA results
ρ (Rho)	Redundancy factor - applied to EL loads for seismic load combinations (or RSA results) Note: If ELX and ELZ used, the ρ from the X and Z direction will be used directionally. If EL is used, the ρ from the X direction will be used and the Z direction will be disregarded. If ELY is used, the ρ from the Z direction will be used and the X direction will be disregarded.
SDS	Spectral Response Acceleration Parameter for Short Periods - applied to DL for seismic load combinations
Ω (Om)	Overstrength Factor - applied to EL loads for seismic load combinations (or RSA results) Note: If ELX and ELZ used, the Ω from the X and Z direction will be used directionally. If EL is used, the Ω from the X direction will be used and the Z direction will be disregarded. If ELY is used, the Ω from the Z direction will be used and the X direction will be disregarded.

Enter in the **Factor** field a multiplier to be applied to the load category being included.

You may also use the red button ▶ in the Category cell to help you specify the loads. Choose the load category, seismic options or you can quickly specify an Other Load category by specifying the Other Loads option and specify the number of the Other Load (i.e. 3 for OL3 or 16 for OL16).
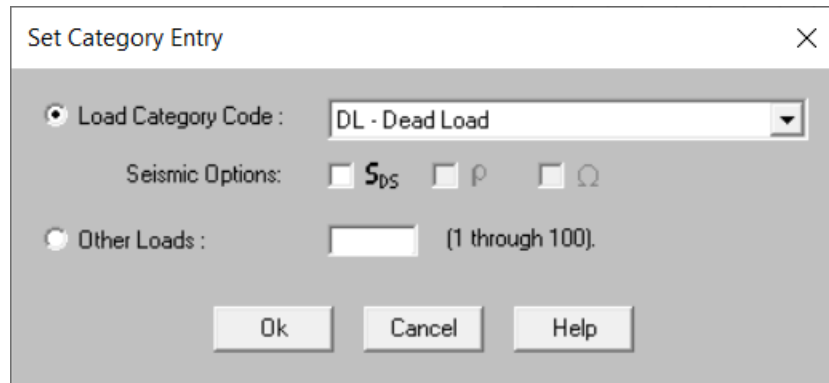
When transferring from RISA-3D with RSA results you will have additional RSA options as shown below.
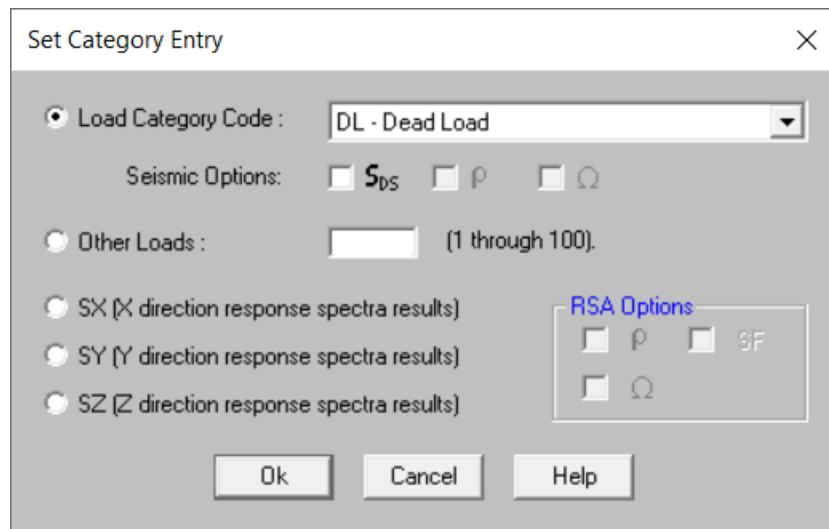
Load Combinations with RSA Results

The results from response spectra analyses in the direction may also be included in the load combinations. Remember, when you perform a response spectra analysis (RSA) in RISA-3D, you specify in which global direction the spectrum is applied. When transferring RSA results to RISAFoundation, the resulting structure reactions from each RSA solution (for each direction) will be transferred with a factor of 1.

To include the RSA results for a particular direction in a load combination, enter "**Sn**" in the BLC field, where n is the global direction. Suppose you wanted to include X direction RSA results, you would enter **SX** in the BLC field. You would enter **SY** for Y direction RSA results. Also be sure to put the RSA Scaling Factor for the RSA results in the Factor field. You can have more than one RSA entry in a load combination.

Nesting Load Combinations

You are allowed only 10 Categories per load combination, which may not be enough. For this reason you can define "combinations of load combinations". This means if you need more than 10 categories in a single combination, you

can define the needed categories over several load combinations and then pull these load combinations together into a single load combination.

To use a nested load combination, within a specific load combination simply choose a load combination to reference from the category dropdown list. Other existing load combinations will be available at the bottom of the category dropdown list and will be designated with the letter L followed by the load combination line number (i.e. L1, L2, L3, etc). For example, say Load Combination 4 has "L3" entered for one of its Categories. This specifies to include all the categories (with their factors) entered in Load Combination 3 as part of Load Combination 4. The flags for Load Combination 3 (i.e. Solve, Service, ABIF and SF entries) apply only to Load Combination 3 and will not be used when Load Combination 4 is solved.

Also, the factor we enter with L# entry will be applied to the Category factors entered for the L#. Thus, if we enter "L1" with a factor of 0.9, we're including 90% of the Category and corresponding factor in Load Combination 1.

Note

- These "combinations of load combinations" can only be nested to one level; i.e. the load combs referenced with the **L#** entries may not themselves have **L#** entries.
- Nest load combinations cannot contain duplicate categories within a single load combination.
- In RISAFoundation, the available nested load combinations will be located at the bottom of the load category dropdown. To access these load combinations quicker, you can repeated hit the 'L' key while highlighting that category

RISAFoundation Design Combinations

RISAFoundation recognizes two types of combinations: Service and Design. Service combinations are used to determine if the soil bearing capacity and overturning resistance are adequate. Design combinations are used to design the slabs, beams, and footings and pedestals for flexure and shear per the chosen code.

The **ABIF** column stands for "Allowable Bearing Increase Factor". For transient loads, such as wind or seismic loads, you may want to specify an allowable bearing increase factor (ABIF) to increase the allowable soil bearing pressure for that combination. The increase factor is typically 1.333 (a 1/3 increase).

Service Combinations

The service combinations are used to calculate actual soil bearing for comparison with allowable soil bearing defined in the **Model Settings Dialog** and the **Soil Regions Spreadsheet**.

An exact biaxial analysis is performed to calculate the soil bearing stress distribution on the footings. The service combinations are also used to calculate the overturning moment safety factor.

The service combinations are also used to check both single piles as well as pile caps. They are also used to provide soil bearing, overturning and sliding checks for retaining walls.

Design Combinations

The design combinations are used to calculate required reinforcing steel and also to check both shear and bearing on the concrete footings and slabs.

The design combinations are also used to design pile caps, retaining walls and grade beams for reinforcement design and shear checks.

Generating Building Code Combinations

Major portions of the load combinations that are specified by building codes are included and may be applied to the model for solution. These combinations may be inserted by selecting the **LC Generator** button from the **Window Toolbar** once the **Load Combinations Spreadsheet** is open and active. This will activate the **Load Combination Generator Dialog** shown below:

The **LC Region** refers to the various regions supported by the program (U.S., Canada, India, British, et cetera).

The **LC Code** refers to the actual code used to build the load combinations. For the United States, there are a number of different codes that could be used to build load combinations. If the only option is *Sample*, that means that no load combinations have been input for that region. See Customizing the Load Combination Generator for more information on how to add or edit combinations for that region.

Wind Load Options

The **Wind Load Options** specify how detailed the generated wind load combinations should be.

When **None** is selected as the wind load option, the program will not generate any Load Combinations that include wind load categories.

The **2D Only** option generates only the most basic wind load category (WL).

The **X and Z** option generates separate wind load combinations for each horizontal direction (WLX and WLZ when Y is set at the vertical axis).

The **X and Z w/ Ecc** option generates all possible wind load combinations that include partial / eccentric wind loading (WLX, WLXP1, WLXP2, et cetera) per Case 2 from Figure 27.3-8 in the ASCE 7-16.

The **X and Z w/ Ecc, Quart** option generates all possible wind load combinations per Cases 2, 3 and 4 from Figure 27.3-8 in the ASCE 7-16. For additional advice on this topic, please see the RISA Tips & Tricks website: www.risa.com/post/support. Type in Search keyword: **Quartering**.

The **Reversible** option generates two combinations for each wind load, one with positive load factors and one with negative load factors.

The **Generate Semi-Rigid Diaphragm Loads** option generates wind loads for BLC's using windward and leeward direction loads prescribed by the ASCE 7-16 (WLX+Z, WLX-Z, WLZ+X, WLZ-X).

Seismic Load Options

The **Seismic Load Options** specify how complex the generated seismic load combinations should be.

When **None** is selected as the seismic load option, the program will not generate any Load Combinations that include the EL load category.

The **2D Only** option is used to indicate that only the most basic seismic load category (EL) will be used.

The **X and Z** option is used to indicate that the program should generate separate seismic load combinations for each horizontal direction (ELX and ELZ).

The **X and Z w/ Eccentric** option is used to indicate that the program should generate all possible seismic load combinations that include eccentricities (ELX, ELX+Z, ELX-Z, et cetera).

The **X and Z RSA** option will use the spectral seismic loads (SX, SY, SZ) instead of the regular earthquake loading (ELX, ELY, ELZ).

When the **Reversible** box is checked, the program will generate every seismic load combination twice. Once with a positive load factor, once with a negative load factor.

The **Save as Defaults** button may be used to establish the current load combination generator settings as the defaults for future use. Clicking the **Generate** button will generate load combinations in the **Load Combinations Spreadsheet** based on the selected options.

General Notes on the Load Combination Generator

The following loads are not generally included in the standard combinations but may be added by editing the combinations in the spreadsheet or by modifying the source document itself. For more information, see Customizing the Load Combination Generator below:

Category Code	Description
TL	Long Term Load
HL	Hydrostatic Load
FL	Fluid Pressure Load

Category Code	Description
PL	Ponding Load Category
EPL	Earth Pressure Load
OL#	Other Load Categories

Note

- The standard combinations are made up of Load Categories and Factors. Loads that are not assigned to these categories will not be included in the combinations upon solution.
- Some load categories do not occur in all of the design codes. Loads placed in categories that are not part of the standard combinations will not be included in the solution of these combinations.
- See Editing Spreadsheets - Inserting, Deleting and Clearing Cells for quick ways to edit or add combinations.
- When a new installation is performed the program will back-up any existing XML files that contain load combination data rather than overwriting them. The back up copies will be given the extension BAK.

Customizing the Load Combination Generator

The Load Combinations for each region are contained in an XML file which can be opened and edited using a standard spreadsheet program. An example of on of these XML files is shown below:

The name of the file itself becomes the name of the Load Combination Region that appears in the Load Combination Generator. Each XML file has a series of "worksheets" and the name of each worksheet will be the name of the Load Combination Code that appears in the Load Combination Generator. The user may add, edit or modify these documents to completely customize the available load combinations.

Note

- When the program is updated for new versions, the existing XML files will be "backed up" and saved with a '.bak' extension while the new XML files in the update will replace them. If any customizations were made to the XML files, they can be retrieved from the back ups.

The first row of each sheet is reserved for the column headers. The recognized column headers are as follows: Label, Solve, ,Service, and ABIF. In addition, there will be multiple pairs of Category & Factor headers.

Other than Label and the Category / Factor pairs, the user may omit columns. If a column is omitted, the default values will be used for those entries. The order of the column labels is optional except for pairs of Category and Factor. For the program to correspond the Factor with correct Category, the user should always provide Category label PRIOR to the corresponding Factor. The user may insert blank columns or columns with other labels than described above. In those cases, the program omits those undefined columns.

Note

- As shown in the above example, the wind and seismic loads should be entered as WL and EL in order for them to be "expanded" using the Wind Load Options and Seismic Load Options.
- The program will read these files from the directory specified in Tool - Application Settings - File Locations. See Customizing RISA for more information.

- SX, SY, and SZ are not valid BLC that can be used within the Load Combination Generator spreadsheet. A response spectra analysis must be done first within RISA-3D.

RSA Scaling Factor

When RSA reactions are transferred from RISA-3D, you will have the ability to access the scaling factor dialog. To access this dialog, first open the load combinations spreadsheet and then click the "SF" button in the upper toolbar.
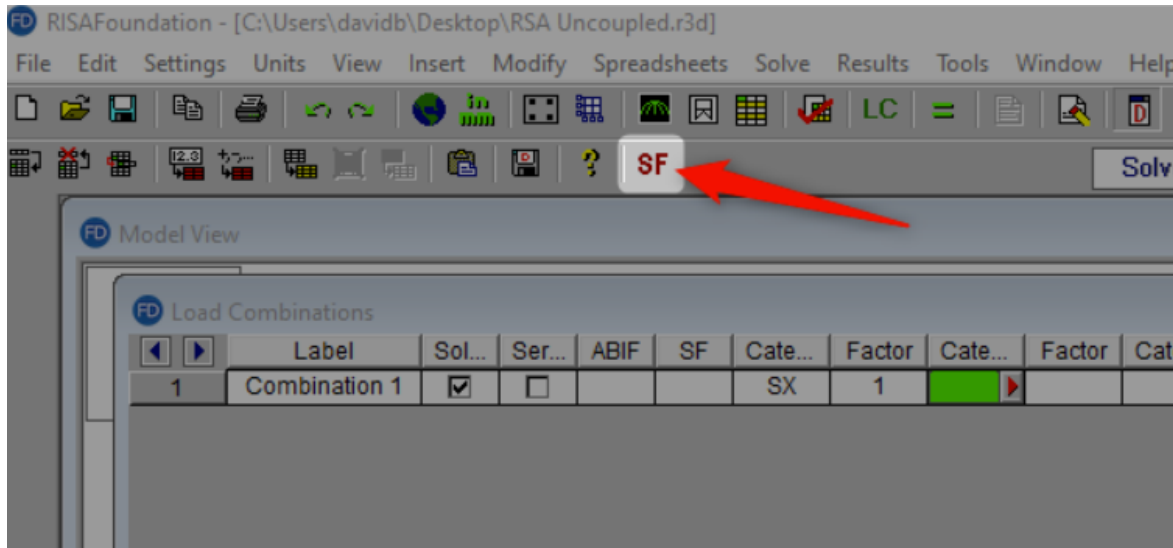
In this dialog you will see the scaling factor along with corresponding unscaled base shear and static base shear that was calculated in RISA-3D.
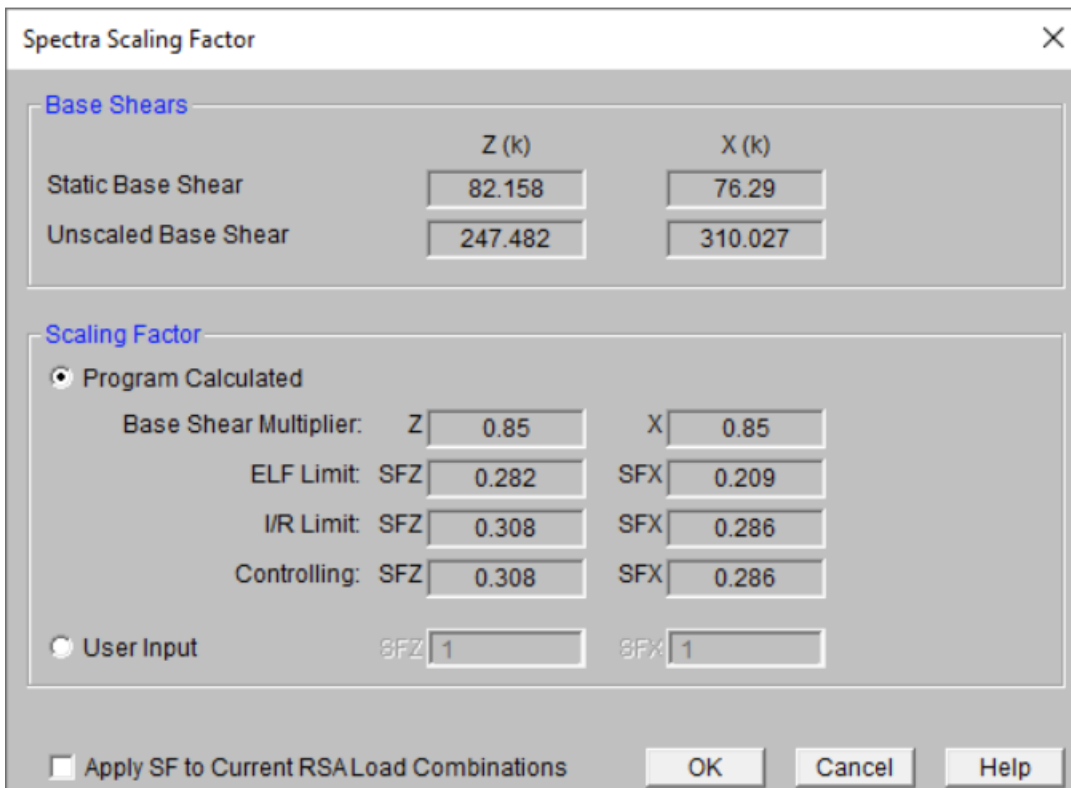
Base Shears

The Base Shears section of the Spectra Scaling Factor dialog box reports to you the static base shear and the un-scaled RSA base shear for each direction. These values will be transferred automatically from RISA-3D. These cannot be modified within RISAFoundation.

Scaling Factor

Program Calculated

This section compares the ELF (Equivalent Lateral Force) Limit, as reduced by the Base Shear Multiplier, and the I/R Limit. These values are explained in detail below.

- The **Base Shear Multiplier** is used in the calculation of the ELF Limit. This multiplier is used to reduce the calculated (unscaled) base shear below the base shear value obtained by the Equivalent Lateral Force Method. For the 1997 UBC, this is covered in section 1631.5.4. For the 2000 IBC, this is covered in section 1618.7. For the 2015 NBC, it is discussed in clause 4.1.8.12 (8). In the ASCE 7-16, it is discussed in section 12.9.1.4.1.

- The **ELF Limit** is the limiting scaling factor calculated per the Equivalent Lateral Force procedure:

$$ELF \; Limit = \frac{(Base \; Shear \; Multiplier * Static \; Base \; Shear)}{Unscaled \; Base \; Shear}$$

- The **I/R Limit** simply takes the elastic dynamic base shear and reduces it to an inelastic base shear. This value is calculated per the I (Importance Factor) and R (Response Modification Coefficient). You may edit these values in the Seismic tab of Model Settings.

- The Controlling Limit is the governing (larger) value of the ELF limit and I/R limit.

It is important to note that these values are transferred automatically from the RISA-3D analysis only when RSA results are transferred. The program calculated scaling factor cannot be modified from within RISAFoundation. If you would like to update this value you will need to transfer back to RISA-3D to do so.

User Input

If you would like to use a different scaling factor you can select the User Input option and manually enter a custom value to override the program calculated value.

Apply Checkbox
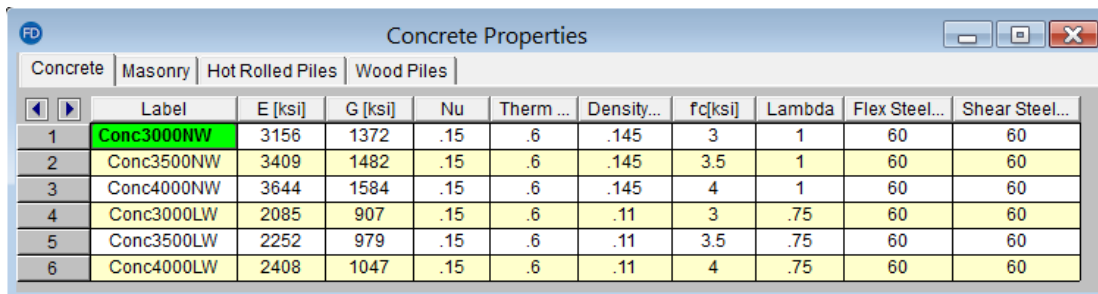
The Apply SF to all RSA Load Combinations checkbox will apply the SF scale factors to all load combinations that currently reference response spectra results.

Material Properties

Material properties are defined on the **Material Properties Spreadsheet** and then are referred to as you build section sets, members, and plates beams and slabs. You may perform analysis using any type of concrete material; simply define the properties for the material here. You may use up to 500 materials in a single model although most models will only have one or two.

Material Properties Spreadsheet

The **Material Properties Spreadsheet** records the material properties to be used in the model and may be accessed by selecting **Materials** on the **Spreadsheets Menu**. The entries are explained below.

The materials shown are the program defaults. However, you can change these and also add as many other materials as you need. You may then save your preferred materials as the default materials by clicking the **Save as Defaults** button on the **Window Toolbar**.

Label is the material label you wish to use to describe the entered material properties. This label is how you will reference the set of properties later when defining beams and slabs.

E is Young's modulus that describes the material stiffness.

G is the shear modulus and may be left blank if you would like it calculated automatically. The equation for "G" is:

$$G = \frac{E}{2.0 * (1.0 + Poisson's\ Ratio)}$$

Note

- If you enter a value for Shear Modulus that does not match the value calculated using the above equation you will be given a warning.
- The default Plywood material has a much lower G value than the formula above would suggest. This is because plywood is NOT an isotropic material. Also, the generic value (from the NDS or APA's Plywood Design Specification) has been reduced somewhat to account for assumed nail slip.
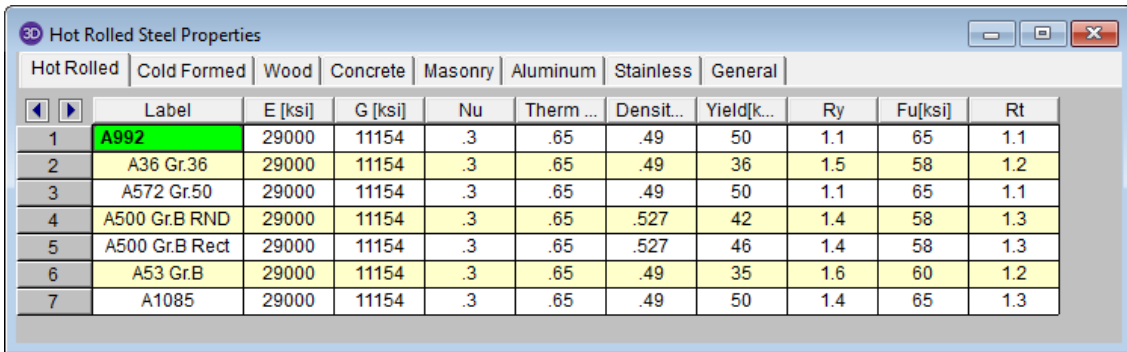
Nu is Poisson's ratio. Besides being used for the "G" calculation, this value is also used for shear deformation calculations. The value of Poisson's ratio may not exceed '0.5'.

Therm is the coefficient of thermal expansion and is entered per 10^5 (100,000) degrees.

Density is the material density and is used in the calculation of the member and slab self weight. This density times the area gives the self weight per unit length for the members; density times slab volume gives the total weight of a given slab.

Hot Rolled Specific Material Data

The **Hot Rolled** tab records a number of hot rolled steel specific material properties that do not exist for the other materials. These entries are described below:

	Label	E [ksi]	G [ksi]	Nu	Therm ...	Densit...	Yield[k...	Ry	Fu[ksi]	Rt
1	A992	29000	11154	.3	.65	.49	50	1.1	65	1.1
2	A36 Gr.36	29000	11154	.3	.65	.49	36	1.5	58	1.2
3	A572 Gr.50	29000	11154	.3	.65	.49	50	1.1	65	1.1
4	A500 Gr.B RND	29000	11154	.3	.65	.527	42	1.4	58	1.3
5	A500 Gr.B Rect	29000	11154	.3	.65	.527	46	1.4	58	1.3
6	A53 Gr.B	29000	11154	.3	.65	.49	35	1.6	60	1.2
7	A1085	29000	11154	.3	.65	.49	50	1.4	65	1.3
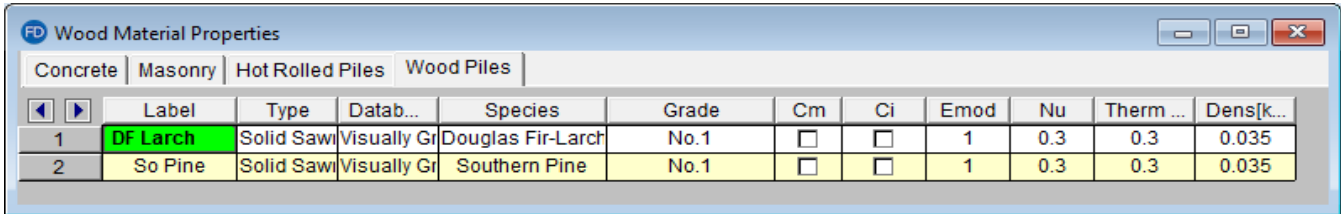
Ry is the ratio of the expected yield stress to the specified minimum yield stress. This value is used in determining the required strength of an element for the seismic detailing checks.

Fu is the specified minimum tensile strength. This value is used per *AISC 358* to calculate C_{pr} which is in turn used to calculate the probable maximum moment at a plastic hinge in the seismic detailing calculations.

Rt is currently not used by the program. In a future version of the program this value will be used to calculate the expected tensile strength for the seismic detailing checks.

Wood Specific Material Data

The **Wood** tab records a number of wood specific material properties that do not exist for the other materials. These entries are described below:

	Label	Type	Datab...	Species	Grade	Cm	Ci	Emod	Nu	Therm ...	Dens[k...
1	DF Larch	Solid Saw	Visually Gr	Douglas Fir-Larch	No.1	☐	☐	1	0.3	0.3	0.035
2	So Pine	Solid Saw	Visually Gr	Southern Pine	No.1	☐	☐	1	0.3	0.3	0.035

Type - Currently there are four wood types available: Solid Sawn, Glulam, SCL (this includes LVLs), and Custom.

Database - Based on your **Type** selection, the **Database** menu will filter to the appropriate databases.

Species - Based on your **Database** selection, the **Species** menu will filter to the appropriate databases. This is the wood species designation from your selected design code.

Grade is the wood grade designation from your selected design code. This has a drop down list where you can select the appropriate grade.

The **Cm** check-box determines if the wet service / moisture content factor should be applied. If you put a check in the **Cm** field, the appropriate factors will be applied to the allowable stresses and Young's Modulus (E).

The **Ci** check-box determines if the incision factor should be applied. If you put a check in the **Ci** field, the appropriate factors will be applied to the allowable stresses and Young's Modulus (E).

Emod is a factor that is applied to the Young's modulus modifier to reflect the NDS Appendix F criteria. This is not applicable to the CSA O86 design code.

Note:
Commercial Species Groups:
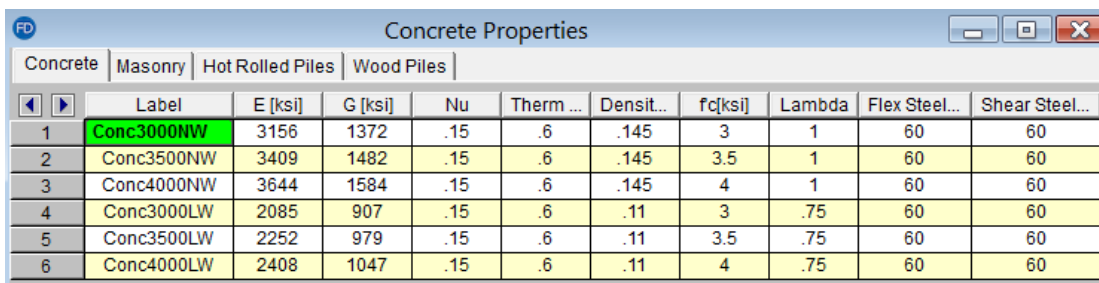
When either the NDS 05/08, 12, 15, or 2018 wood design code is selected, the Solid Sawn Species menu lists two new species listed that are not specifically shown in the NDS code. These are the *Commercial Species Group I - DF/SP* and *Commercial Species Group II - HF/SPFe Fir*. These are meant to be simplified groupings of the most commonly used wood species. It is meant to simplify the selection procession for wood member design in the United States.

The design values for Group I take the NDS allowable stress values for two of the most widely used species (Doug Fir-Larch and Southern Pine) and uses the lower bound allowable stress value for each size and grade.

Similarly, the design values for Group II take the allowable stress values for Hem Fir and Spruce-Pine-Fir and use the lower bound allowable stress values for each size and grade.

Concrete Specific Material Data

The **Concrete** tab records a few concrete specific material properties that do not exist for the other materials. These entries are described below:

	Label	E [ksi]	G [ksi]	Nu	Therm ...	Densit...	fc[ksi]	Lambda	Flex Steel...	Shear Steel...
1	Conc3000NW	3156	1372	.15	.6	.145	3	1	60	60
2	Conc3500NW	3409	1482	.15	.6	.145	3.5	1	60	60
3	Conc4000NW	3644	1584	.15	.6	.145	4	1	60	60
4	Conc3000LW	2085	907	.15	.6	.11	3	.75	60	60
5	Conc3500LW	2252	979	.15	.6	.11	3.5	.75	60	60
6	Conc4000LW	2408	1047	.15	.6	.11	4	.75	60	60
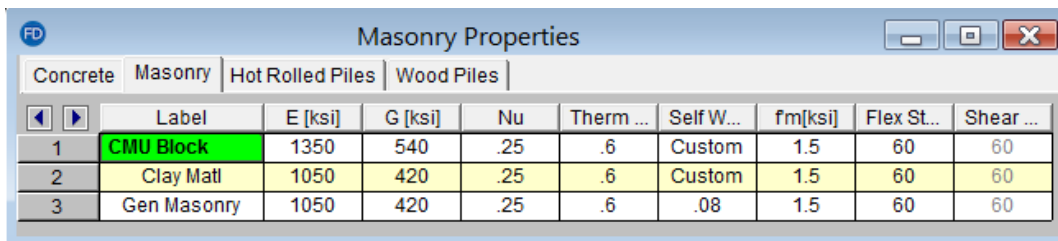
f'c is the concrete compressive strength used for concrete design.

Lambda, λ is the lightweight concrete modification factor. This factor only applies to the *ACI 318-14*, *ACI 318-11*, *ACI 318-08* and *CSA A23.3-04* codes. For all other codes the Density value of the material determines any strength reduction. The program will automatically calculate the correct value if it is left blank. Only values between 0.75 and 1.0 will be considered.

Flex Steel is the reinforcement yield strength for flexural bars in members and vertical bars in walls. **Shear Steel** is the reinforcement yield strength for shear bars in members and horizontal bars in walls.

Masonry Specific Material Data

The **Masonry** tab records masonry specific material properties. Here we will describe items that do not exist for other materials.

	Label	E [ksi]	G [ksi]	Nu	Therm ...	Self W...	fm[ksi]	Flex St...	Shear ...
1	CMU Block	1350	540	.25	.6	Custom	1.5	60	60
2	Clay Matl	1050	420	.25	.6	Custom	1.5	60	60
3	Gen Masonry	1050	420	.25	.6	.08	1.5	60	60

Masonry **Self Weight** can be accounted for by two different methods. Entering a number for the density will result in masonry walls which have a self weight equal to that density multiplied by the wall cross-sectional area. Otherwise, click within the cell to launch the masonry self weight dialog:
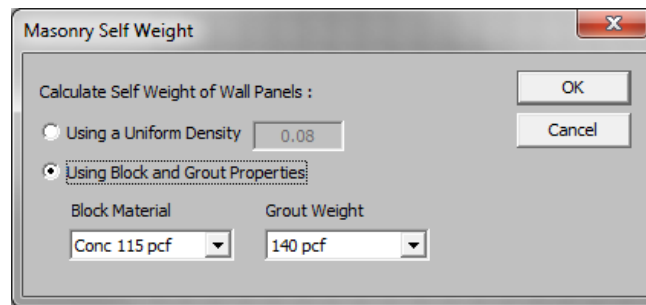
Set the Using **Block and Grout Properties** option to have the program automatically calculate the self weight of the wall using the weights from tableB3 of the Reinforced Masonry Engineering Handbook (RMEH). The self weight will be listed as **Custom**.

f'm is the masonry compressive strength used for masonry design.

Flex and **Shear Steel** are the rebar yield strengths for flexural and shear bars used to reinforce the masonry.

Note:

- Masonry Shear Steel is automatically set to the Flex steel strength.
- Masonry materials are used only for walls in wall footings. See the Wall Footing Definitions spreadsheet for more information.

Material Take-Off

Access the **Material Take-Off** spreadsheet by selecting it from the **Results** menu.

	Material	Quantity	Volume[yds^3]	Weight[K]
1	**Piles**			
2	Pile 1	49	N/A	N/A
3	Pile 2	28	N/A	N/A
4	Total Piles	77	N/A	N/A
5				
6	Slabs			
7	Conc4000NW	2	1235.8	4838
8				
9	Beams			
10	Conc4000NW	6	16.3	63.7
11				
12	Footings			
13	Conc4000NW	2	3	11.6
14				
15	Pedestals			
16	Conc4000NW	12	4	15.7
17	Conc3000NW	11	4.1	15.9
18				
19	Pile Caps			
20	Conc3000NW	2	20.6	80.8
21				
22	Retaining Walls			
23	Conc3000NW	2	119.5	467.9
24				
25	Totals			
26	Conc3000NW		144.2	564.7
27	Conc4000NW		1259	4928.9
28	Total Concrete		1403.2	5493.6

This spreadsheet shows material takeoff information for each material and element type in the model. For each material the element types are listed with the total quantity, volume and weight.

The weight is the sum of the self-weight for all elements

Note:

- This material takeoff report is independent of the loads applied to the model, i.e. the applied loads do not influence this report.
- Beams that frame into a footing or pile cap will show the beam stopping at the location of the element. However, for material take-off concerns the entire length of beam (all the way to the footing/cap pedestal) is considered.

Modeling Tips - Foundations

Here are some specialty topics that can help model more complex situations.

Soils Springs for Eccentric Footings

Review the section in the Soils Topic for detailed information about modeling of combined footings with eccentric pedestals.

Modeling Complex Slabs

Review the section in the Slabs Topic for detailed information about modeling of complicate slab geometries or slabs with varying thickness.

Pedestals/Posts & Stem Walls

Aside from applying loads directly to the slab as a point or line load, loading can be applied to the slab with a Pedestal/Post or a Stem Wall. Pedestals consist of cast-in-place concrete, and are designed (complete with code check and rebar) similar to concrete columns. Posts allow you to define the outline of a post or column baseplate which bears directly on the slab. No design is performed for the post or baseplate itself, but the slab is checked for punching shear based on the post's outline. Stem walls are wall entities whose only purpose is as a load transfer mechanism to a slab. There is no design of the stem wall itself, but it is used for self-weight.

Pedestal/Post and Stem Wall data may be viewed and edited in two ways, through the Pedestals/Stem Walls Spreadsheet and through the **Draw/Modify Pedestals or Stem Walls** tool.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Pedestals**.

Draw Pedestals/Posts
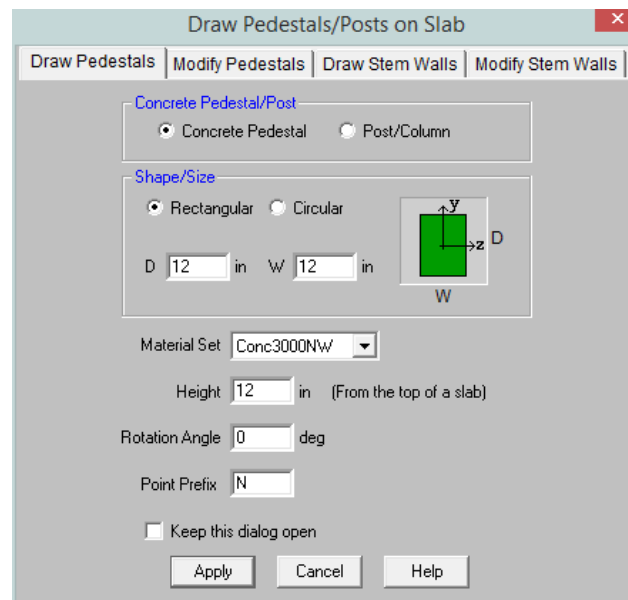
To draw new Pedestals/Posts:

1. If there is not a model view already open then click [icon] in the **RISA Toolbar** to open a new view and click [icon] to turn on the **Drawing Toolbar** if it is not already displayed.

2. If you are not drawing on existing points, you will need to create a drawing grid or define points on the **Point Coordinates** spreadsheet.

3. Click the **Draw/Modify Pedestals or Stem Walls** [icon] button and select the **Draw Pedestals** tab. Then set the pedestal/post properties. Click **OK** to start drawing pedestals/posts by clicking on the points or grid points with the left mouse button

4. To stop drawing altogether click the [icon] button, right click or press the **Esc** key.

Note

- To draw more pedestals/posts with different properties, press CTRL-D to recall the **Pedestals/Posts** settings.

- You may also specify or edit pedestals in the Pedestals/Stem Walls Spreadsheet.
- You may undo any mistakes by clicking the Undo button.

Modify Pedestals/Posts

There are a couple of ways to modify pedestals/posts. You may view and edit the pedestal/post data in the **Pedestals/Stem Walls Spreadsheet** or you can use the **Modify Pedestals/Posts** tool to graphically modify a possibly large selection of pedestals/posts.

The graphical **Pedestal/Post Modify** tool discussed here lets you modify the properties of pedestals/posts that already exist in your model. To use this, you will typically specify the properties you want to change, then select the pedestals/posts that you want to modify. You can modify pedestals one at a time by selecting the **Click to Apply** option and then click on the pedestals/posts you wish to modify. You may also modify entire selections of pedestals/posts by selecting the pedestals/posts first and then use the **Apply to Selected** option. See the Graphic Selection topic for more on selecting.

The parameters shown are the same as those used to define new pedestals/posts.
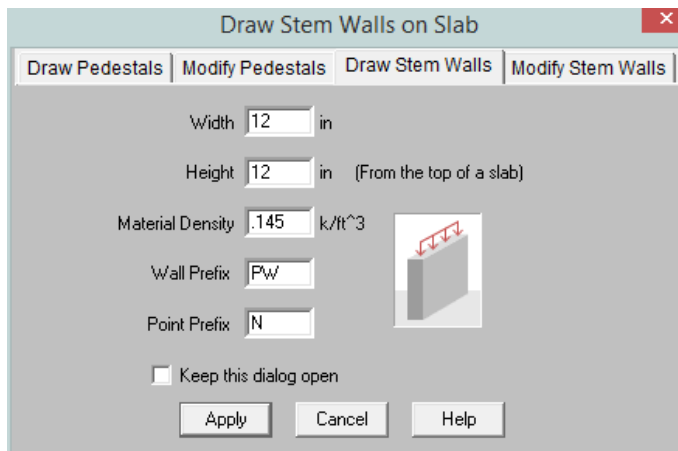
The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected pedestals. If the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on pedestals without affecting all the rest of the properties. Note that if no value is entered in a field (i.e. the field is blank) and the corresponding check box is checked, clicking "Apply" will have the effect of clearing the data for these fields.

To Modify Pedestals/Posts

1. If there is not a model view already open then click ⊞ on the **RISA Toolbar** to open a new view and click ✎ to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Draw/Modify Pedestals or Stem Walls** ✎ button and select the **Modify Pedestals** tab. Then set the parameters for the pedestals/posts. Check the **Use?** Box for the items to apply.
3. You may choose to modify a single pedestal/post at a time or to an entire selection of pedestals/posts.
   - To modify a selection of pedestals, choose **Apply Entries to All Selected Pedestals/Posts** and click **Apply.**
   - To modify a few pedestals choose **Apply Entries by Clicking Pedestals/Posts Individually** and click **Apply**. Click on the pedestals/posts with the left mouse button.

Note:

- You cannot use the graphic modify tool to change a Pedestal to be a Post or vice versa. This modification can only be done in the Pedestals/Stem Walls Spreadsheet.
- To modify more pedestals/posts with different parameters, press CTRL-D to recall the **Modify Pedestals/Posts** settings.
- You may also modify pedestals/posts in the Pedestals/Stem Walls Spreadsheet.
- You may undo any mistakes by clicking the **Undo** ↶ button.

Draw Stem Walls

To Draw New Stem Walls:

1. If there is not a model view already open then click ⊞ in the **RISA Toolbar** to open a new view and click ✎ to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you are not drawing on existing points, you will need to create a drawing grid or define points on the **Point Coordinates** spreadsheet.
3. Click the **Draw/Modify Pedestals or Stem Walls** ✎ button and select the **Draw Stem Walls** tab. Then set the stem wall properties. Click **OK** to start drawing stem walls by clicking two points or grid points with the left mouse button
4. To stop drawing altogether click the ✎ button, right click or press the **Esc** key.

Note

- To draw more pedestals/posts with different properties, press CTRL-D to recall the **Stem Walls** settings.
- You may also specify or edit pedestals in the <u>Pedestals/Stem Walls Spreadsheet</u>.
- You may undo any mistakes by clicking the Undo button.

Modify Stem Walls

There are a couple of ways to modify stem walls. You may view and edit the stem wall data in the **Pedestals/Stem Walls Spreadsheet** or you can use the **Modify Stem Walls** tool to graphically modify a possibly large selection of stem walls.

The graphical **Stem Wall Modify** tool discussed here lets you modify the properties of stem walls that already exist in your model. To use this, you will typically specify the properties you want to change, then select the stem walls that you want to modify. You can modify stem walls one at a time by selecting the **Click to Apply** option and then click on the stem walls you wish to modify. You may also modify entire selections of stem walls by selecting the stem walls first and then use the **Apply to Selected** option. See the <u>Graphic Selection</u> topic for more on selecting.

The parameters shown are the same as those used to define new stem walls.
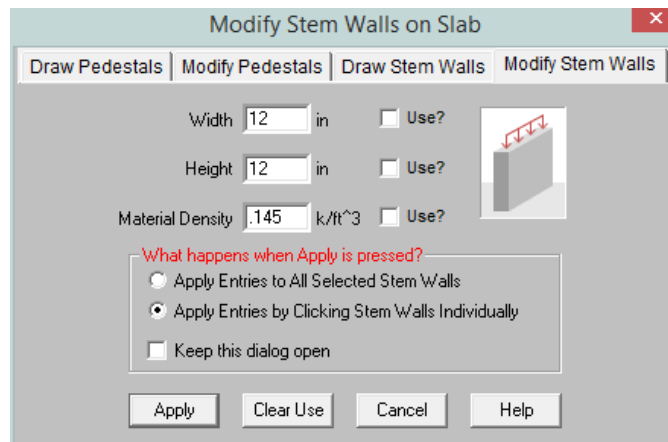
The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected pedestals. If the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on stem walls without affecting all the rest of the properties. Note that if no value is entered in a field (i.e. the field is blank) and the corresponding check box is checked, clicking "Apply" will have the effect of clearing the data for these fields.

To Modify Stem Walls

1. If there is not a model view already open then click ▦ on the **RISA Toolbar** to open a new view and click ▨ to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Draw/Modify Pedestals/Stem Walls** ✎ button and select the **Modify Stem Walls** tab. Then set the parameters for the stem wall. Check the **Use?** Box for the items to apply.
3. You may choose to modify a single stem wall at a time or to an entire selection of stem walls.
   - To modify a selection of stem wall, choose **Apply Entries to All Selected Stem Walls** and click **Apply.**
   - To modify a few stem walls choose **Apply Entries by Clicking Stem Walls Individually** and click **Apply**. Click on the stem walls with the left mouse button.

Note:

- To modify more stem walls with different parameters, press CTRL-D to recall the **Modify Stem Walls** settings.
- You may also modify stem walls in the [Pedestals/Stem Walls Spreadsheet](#).
- You may undo any mistakes by clicking the **Undo** button.

Pedestals/Posts Spreadsheet

Another way of adding or editing pedestals/posts is through the **Pedestals/Posts Spreadsheet** which is accessible in the **Data Entry** toolbar. This spreadsheet includes the primary data for pedestals/posts, including shape, material, design rule, angle and height.

	Label	Type	Shape	Material	Design Rules	Angle(deg)	Height[in]	Pedestal Layout	Shear Layout
10	N16	Pedestal	CRND24	Conc4000NW	Typical	0	12	Use Design Rule	Use Design Rule
11	N17	Pedestal	CRND24	Conc4000NW	Typical	0	12	Use Design Rule	Use Design Rule
12	N18	Pedestal	CRND24	Conc4000NW	Typical	0	12	Use Design Rule	Use Design Rule
13	N19	Pedestal	CRECT30X24	Conc4000NW	Typical	0	12	Use Design Rule	Use Design Rule
14	N20	Pedestal	CRECT30X24	Conc4000NW	Typical	0	12	Use Design Rule	Use Design Rule
15	N21	Pedestal	CRECT30X24	Conc4000NW	Typical	0	12	Use Design Rule	Use Design Rule

The following data columns hold the primary data for the pedestals/posts:

Label

You may assign a unique label to any or all of the pedestals. You can then refer to that pedestal by its label. Each label has to be unique, so if you try to enter the same label more than once you will get an error message.

Type

You may specify whether this is a Pedestal or a Post. Posts do not have a material or design rule, and their height is always set to zero since Posts are only used for punching shear checks.

Shape

You may assign the size of your pedestals/posts in two ways:

1. You can use the red arrow within the green selection window to open the **Set Pedestal Size** window.
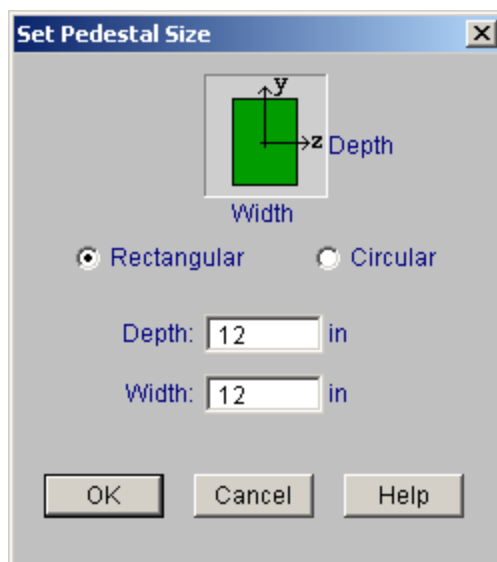
This window allows you to input shape information. Set your parameters, choose **OK** and you will return to the **Pedestals/Posts Spreadsheet** with that shape/size selected.

2. If you already know the nomenclature, you can enter the size by simply typing it in. Use CRNDXX (where XX is the diameter of the pedestal) if you want a circular pedestal, or use CRECTYYxZZ (where YY is the depth and ZZ is the width of your pedestal. if you want a rectangular pedestal.

Material

This column allows you to define the material for the pedestal for use in analysis and design.

Design Rule

This allows you to choose the design rule specific to this pedestal. The Design Rule defines the pedestal reinforcing and cover dimensions.

Angle

This column allows you to enter an angle (in degrees) if you want your rectangular pedestal/post rotated about its local y axis.

Height

This entry allows you to set the height of your pedestal from the top of slab.

Pedestal Layout
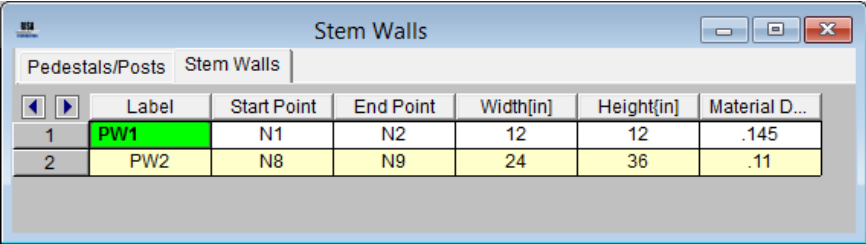
This column allows you to select a custom flexural rebar layout. See the Pedestal Rebar Layouts topic for more information on how to create a custom layout. If you do not select a custom layout, then the program will design the reinforcement for you, based on the input in your Design Rules.

Shear Layout

This column allows you to select a custom shear rebar layout. See the Shear Rebar Layouts topic for more information on how to create a custom layout. If you do not select a custom layout, then the program will design the reinforcement for you, based on the input in your Design Rules.

Stem Walls Spreadsheet

Another way of adding or editing stem walls is through the **Stem Walls Spreadsheet** which is accessible by pressing the **Pedestals/Stem Walls** button on the **Data Entry** toolbar and clicking the **Stem Walls** tab. This spreadsheet includes the primary data for stem walls, including end points, width, height and material density.

	Label	Start Point	End Point	Width[in]	Height[in]	Material D...
1	PW1	N1	N2	12	12	.145
2	PW2	N8	N9	24	36	.11

The following data columns hold the primary data for the stem walls:

Label

You may assign a unique label to any or all of the stem walls. and then refer to it by that label. Each label has to be unique, so if you try to enter the same label more than once you will get an error message.

Start/End Point

This shows which two points define the stem wall. These points can be edited in this spreadsheet.

Width/Height

These fields allow you to define the geometry of the stem wall. The height may be used for overturning calculations. The volume will be used for self-weight calculations.

Material Density

This column allows you to define the material density for the stem wall for self-weight considerations.

Pedestal/Post - Design and Punching Shear Results

The Pedestals/Posts Design spreadsheets reports pedestal/post design information as well as the two way/punching shear code checks for the defined Pedestal/Post as described in the following sections.

Note:

- This spreadsheet does not apply to beam design or individual spread footing/pedestal design.
- For detailed information on punching shear calculations see the Punching Shear - Design topic.
- There are no results for stem wall design or punching shear in the program. A stem wall is only an analysis tool.
- For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Pedestals**.

Pedestal Design Values

This spreadsheet reports pedestal code checks and the reinforcing for the pedestals

Note:

- Only reinforcement bars selected by the program are listed in this spreadsheet. If a custom rebar layout is used for a particular beam, the reinforcement section entries will report "CUSTOM" as a reminder that the user applied a custom rebar layout.

UC (Unity Check) and Governing LC

The **UC** value is the governing code check for the pedestal combined axial and bending using the PCA Load Contour method. The **Gov LC** value is the load combination that resulted in this maximum code check.

Shear UC (Unity Check), Governing LC, and Direction

The **Shear UC** value is the governing code check for the pedestal beam shear check. The **Gov LC** value is the load combination that results in the governing code check, based on the highest shear force. The **Direction** value is the local axes direction of this maximum shear force in the pedestal.

The shear check is an envelope check. Therefore the program will find the largest shear force (Vu) in each lateral direction and then keep track of the associated axial force (Pu) in the governing load combination(s). This means that there can be different axial forces for the check in each direction since the governing Vux and Vuz may come from different load combinations.

ACI 318-19 requires the consideration of shear interaction in y and z directions when both $Vuy/\varphi Vny$ and $Vuz/\varphi Vnz$ are larger than 0.5. In this case, the program recalculates the shear UC based on Equation 22.5.1.11. If the UC considering shear interaction happens to govern, this UC will be reported herein.

When ACI 318-19 is selected, the shear strength of concrete (Vc) uses equations in Table 22.5.5.1. Note that for members meeting the minimum shear reinforcement requirement (Av≥Av,min), Vc is taken as the larger of the

results calculated by the equations (a) and (b) in the table. Note that ACI 318-19 code suggests ρw may be taken as the sum of the areas of longitudinal bars located more than two-thirds of the overall member depth away from the extreme compression fiber. Therefore, RISA calculates ρw as the sum of the areas of longitudinal bars on the tension face.

When ACI 318-14 and code codes were selected, Vc will be taken as zero if the load combination includes a tensile axial force. Since the shear check is an envelope check, any load combination resulting in a tensile axial force will cause Vc to be taken as zero. The program will not attempt to determine if the tensile load is "significant". In ACI 318-14 Section 22.5.7.1 the language was modified but RISA still considers Vc = 0 if there is tension present.

Aside from determining if the pedestal/post is in tension or not, the shear check does not consider any axial load in the pedestal.

Phi Used

This value is based on the strain in the tension steel in the pedestal and code provisions in ACI 318-14 Section 21.2 (ACI 318-11 Section 9.3.2).

Pedestal Reinforcement

The **Vertical Reinf** and **Shear Reinf** values are reinforcement requirements for the pedestal considering spacing and reinforcement requirements per the ACI code and the Design Rules-Pedestal. If these sections report "CUSTOM" it means that the user has applied a custom rebar layout.

Punching Shear Values

This section reports the basic / summary data for the punching shear code check.

UC (Unity Check) and Governing LC

The **UC** value is the governing code check (Total Stress /phi*Vnx) for the pedestal punching shear. The **Gov LC** value is the Load combination that resulted in this maximum code check.

Location

The pedestal and slab geometry is investigated and checked for all three types of punching failure (Interior, Edge and Corner). This field represents the type of failure method that controlled the code check calculation for the given pedestal. For more information on how this works, see the Punching Shear - Design topic.

Vuy

This value represents the factored load transferred from the pedestal into the slab for the governing load combination. This corresponds to the axial force in the Pedestal/Post, and subtracting the counteracting bearing pressure force that occurs within the punching shear perimeter.

Muz and Mux

These values represent the factored moments about the local axes of the Pedestal/Post.

Total Stress

The Total Stress is the direct shear stress plus the shear stress due to moment. For more information, see the Punching Shear - Design topic.

Phi*Vny

This represents the shear *stress* capacity of the slab calculated based on the three ACI equations for two way or punching shear capacity of the slab. Refer to ACI 318-14 Section 22.6.5 (ACI 318-11 Section 11.11.7.2) for more information. When ACI 318-19 is selected, the shear strength of concrete (Vc) uses equations in Table 22.6.5.2. Note that an additional size effect factor λs is added in the equation, λs is a function of effective depth of the slab d, and is calculated per Equation (22.5.5.1.3).

Note:

- This value does NOT include any additional capacity due to shear reinforcing in the slab.
- For the CSA A23.3-04 code provision 13.3.4.3 is used.
- For the Eurocode2 : BS EN 1992-1-1:2004+A1:2014 code section 6.4.4 is used.

Minimum Slab Reinforcement at Punching Shear Critical Section

ACI 318-19 requires a minimum slab reinforcement at punching shear critical section if $vuv > \varphi 2\lambda \varphi \sqrt{f'c}$ per Section 8.6.1.2. The As,min is calculated and presented in the punching shear result of columns under this condition. Please note that the reinforcement design in elevated slabs does not consider this requirement, users need to adjust the slab reinforcement at punching shear critical section to account for this code requirement. RISA provides the calculations of As,min per Equation (8.6.1.2) in the Detail Report - Punching Shear Results. Please note that the As,min calculated is based on area of steel per foot, i.e., bslab was taken as 12". The calculated As,min shall be multiplied by bslab (specified in 8.4.2.2.3) to obtain the total area of steel required.

Punching Shear Geometries
Effective depth 4 in

L1 along zz	10 in	Polar Moment Ixx	2773.333 in^4	Gamma xx	0.4
L2 along xx	10 in	Polar Moment Izz	2773.333 in^4	Gamma zz	0.4

Minimum Slab Reinforcement Requirements (per foot) provided over the effective slab width (bslab) per ACI318-19 Sec8.6.1.2

v_uv	0.36 ksi	alpha	20
lambda_s	1	bo	40 in
As_min	0.959 in^2/ft		

Punching Shear Geometries

Pedestal Punching Shear Geometries

Pedestal Design Values | Punching Shear Values | Punching Shear Geometries

	Label	Shape	Location	Effective Depth[in]	L1[in]	L2[in]	Polar Moment-xx[in^4]	Polar Mom...	Gamma-x	Gamma-z
1	Drum-Anchor	CRECT12X12	INTERIOR	10	22	22	74653.333	74653.333	.4	.4
2	Drum-Slide	CRECT12X12	EDGE	10	23	22	43793.578	66366.667	.405	.395
3	Exchr-Anchor	CRECT12X12	CORNER	10	23	23	27264.583	27264.583	.4	.4
4	Exchr-Slide	CRECT12X12	INTERIOR	10	22	22	74653.333	74653.333	.4	.4
5	Pipe Support	CRECT12X12	INTERIOR	10	22	22	74653.333	74653.333	.4	.4

Note:

- For detailed information on many of these values see the Punching Shear - Design topic.

Location

The pedestal and slab geometry is investigated and checked for all three types of punching failure (Interior, Edge and Corner). This field represents the type of failure method that controlled the code check calculation for the given pedestal. For more information on how this works, see the Punching Shear - Design topic.

Effective Depth

The effective depth of the slab is used to define the punching shear perimeter at a distance of d/2 beyond the edge of the pedestal.

L1 and L2

Length of the critical section along the local axes.

Note:

- For a skewed Edge condition L1 or L2 may have two values. See the Punching Shear - Design topic for more information.

Polar Moment - xx and Polar Moment - zz

This is a property (Jc) of the critical section which is analogous to polar moment of inertia about the longitudinal column/pedestal axes.

Gamma x and Gamma z

This is calculated from the dimensions of the assumed punching shear perimeter. It represents the fraction of the unbalanced moment for each axis that is transferred by eccentricity of shear.

Pedestal/Post Detail Report

The detail reports for pedestals/posts show the overall geometry, materials, codes and analysis/design values used in coming up with the reinforcing and code check values. The report is split into three sections: input echo/graphic view, design results (for pedestals only) and punching shear check results.

Note:

- - This button will allow you to take a snapshot of the current detail report you are viewing so that it can be added to a report. View the Printing topic for more information.

Input Echo/Graphic View

Pedestal: **Exchr-Anchor**
Shape: **CRECT12X12**   Tension Bar Fy: **60 ksi**   Stress Block: **Parabolic**
Material: **Conc3000NW**   Shear Bar Fy: **60 ksi**   Design Rule: **Typical**
Height: **12 in**   Pedestal Cover: **1.5 in**   Design Code: **ACI 318-05**
Location: **(19 , 19) ft**   Rebar Set: **ASTM A615**
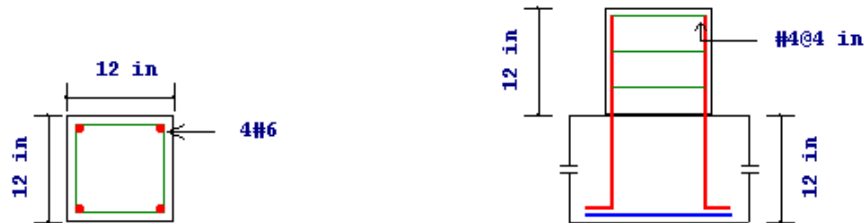
The input echo lists off the point label of the pedestal/post, material properties, reinforcement properties and design requirements as applicable. The graphic view gives a plan and section view of the geometry of the pedestal/post along with the actual pedestal reinforcement designed by the program.

Note:

- The program bases the number and spacing of bars on requirements per the applicable code regarding spacing and reinforcement limits.

Design Results

Design Results
Shear Check Results (Envelope)

Along xx Vcx	367.442 k	Vsx	64.018 k	Vux	36.16 k	Vux/∅ Vnx	.112	Gov LC	132
Along zz Vcz	378.242 k	Vsz	109.833 k	Vuz	57.95 k	Vuz/∅ Vnz	.158	Gov LC	146
Shear Ties	#4 @ 12 in								

Bending Check Results

Unity Check	.12	Phi	.9	Parme Beta	.65
Pu	0 k	Mux	-409.212 k-ft	Muz	-157.463 k-ft
Pn	9583.905 k	Mnx	454.681 k-ft	Mnz	174.958 k-ft
Gov LC	169	Mnox	2185.075 k-ft	Mnoz	1288.733 k-ft
Long. Bars	36 #6	% Steel	.541		

Compression Development Length for Longitudinal Bars

Lrequired	11.25 in	Lprovided	38.5 in	Lreq./Lpro.	.292

The **Shear Check Results** summarize the shear capacities and demand, along with the shear ties used. The **Vc** values are the shear capacities of concrete only in each of the horizontal directions. The **Vs** values are the shear reinforcement capacities in each of the horizontal directions. The **Vu** values are the demand requirements in each of the horizontal directions. The code check value comes from the equation below:

$$\text{Interaction} := \frac{V_u}{\phi \cdot \left(V_c + V_s\right)}$$

The **Gov LC** value is the load combination that results in the governing code check, based on the highest shear force. The shear check is an envelope check. Therefore the program will find the largest shear force (Vu) in each lateral direction and then keeps track of the associated axial force (Pu) in the governing load combination(s). This means that there can be different axial forces for the check in each direction since the governing Vux and Vuz may come from different load combinations.

When ACI 318-19 is selected, the shear strength of concrete (Vc) uses equations in Table 22.5.5.1. Note that for members meeting the minimum shear reinforcement requirement (Av≥Av,min), Vc is taken as the larger of the results calculated by the equations (a) and (b) in the table. Note that ACI 318-19 code suggests ρw may be taken as the sum of the areas of longitudinal bars located more than two-thirds of the overall member depth away from the extreme compression fiber. Therefore, RISA calculates ρw as the sum of the areas of longitudinal bars on the tension face.

When other ACI 318 editions are selected, the shear strength of the concrete alone is limited to the standard $2*\lambda*\sqrt{(f'c)}$ equation from ACI 318-14 Section 22.5.5.1 (ACI 318-11 Section 11.2.1.1) and does not use the more detailed calculations of ACI 318-14 Table 22.5.5.1 (ACI 318-11 Section 11.2.2). Per ACI 318-11 section 11.2.1.3, Vc will be taken as zero if the load combination includes a tensile axial force. Since the shear check is an envelope check, any load combination resulting in a tensile axial force will cause Vc to be taken as zero. The program will not attempt to determine if the tensile load is "significant". In ACI 318-14 Section 22.5.7.1 the language was modified but RISA still considers Vc = 0 if there is tension present.

Note:

- ACI 318-19 requires the consideration of shear interaction in y and z directions when both Vuy/φVny and Vuz/φVnz are larger than 0.5. In this case, the program recalculates the shear UC based on Equation 22.5.1.11. If the UC considering shear interaction happens to govern, a new section is added in the Detail Report to present this check.

Design Results
Shear Check Results (Envelope)

Along xx Vcx	13.803 k	Vsx 55.21 k	Vux 50 k	Vux/∅ Vnx 0.966	Gov LC 2
Along zz Vcz	13.803 k	Vsz 55.21 k	Vuz 40 k	Vuz/∅ Vnz 0.773	Gov LC 2
Shear Ties	#4 @ 2 in				

Shear Check Results (Interaction for biaxial shear)
Gov LC 2

Along xx Vcx	13.803 k	Vsx 55.21 k	Vux 50 k	Vux/∅ Vnx 0.966	> 0.5
Along zz Vcz	13.803 k	Vsz 55.21 k	Vuz 40 k	Vuz/∅ Vnz 0.773	> 0.5
Shear Ties	#4 @ 2 in	UC=(Vux/Vnx+Vuz/Vnz)/1.5	1.159		

Aside from determining if the pedestal/post is in tension or not, the shear check does not consider any axial load in the pedestal.

The **Bending Check Results** summarize the moment capacities and demand, along with the longitudinal reinforcing used. The **Pu** and **Mu** values are the axial and moment demand requirements. The **Pn** and **Mn** values are the axial and moment capacities based on the PCA Load Contour method. The **Mnox** and **Mnoz** are the uniaxial moment capacities. For more information on this method, see the <u>Notes on the ACI 318 Building Code Requirements for Structural Concrete</u>, a document put out by the PCA. The **Parme Beta** is a factor that defaults to a value of 0.65. The **Gov LC** is the load combination that produced the maximum unity check. The **Long. Bars** and the **% Steel** give the actual amount of steel and the number of bars. The **Unity Check** is a combined axial and bending unity check based on the interaction diagram for the pedestal.

The **Compression Development Length** section gives the values for longitudinal bar compression development length checks. Lrequired is the comp. dev. length required per the ACI 318 code. Lprovided is the actual maximum value considering the thickness of slab and the bottom cover requirements. The equation is:

Lprovided = footing thickness – cover – 2*footing bar diameters – dowel bar diameter

The program uses a default value of the diameter of a #5 bar as the footing bar diameter for this calculation (0.625 in^2).

Lreq/Lpro gives a code check ratio of these values.

Note:

- The compression development length check may not be the most essential check for current building codes. But it is a criteria that may be necessary to engineers in specific conditions.

Punching Shear Check Results

Punching Shear Check Results
Punching Shear Values

Unity Check	.417	Vuy	19.637 k	Total Stress	.069 ksi
Location	CORNER	Mux	0 k-ft	ϕvny	.164 ksi
Gov LC	2	Muz	-8.4 k-ft		

Punching Shear Geometries
Effective depth **10 in**

L1 along xx	**23 in**	Polar Moment Ixx	**27264.583 in^4**	Gamma xx	.4
L2 along zz	**23 in**	Polar Moment Izz	**27264.583 in^4**	Gamma zz	.4

The **Unity Check** gives a ratio of demand to capacity, while the **Gov LC** gives the load combination that produced this maximum unity check.

For an explanation of the rest of these values see both the spreadsheet result explanations above, as well as the Punching Shear - Design topic.

Piles and Pile Caps

Piles and **pile caps** can be defined in RISAFoundation. Singular **piles** are used *when individual pile locations need to be defined to support slab elements*. These are best utilized for mat foundations where the mats are continuously supported by piles.

A **pile cap** is an element consisting of *multiple piles, a cap and a pedestal* that are defined at a single node in the model. A pile cap is *defined at a single point location*, typically where a column reaction is located.
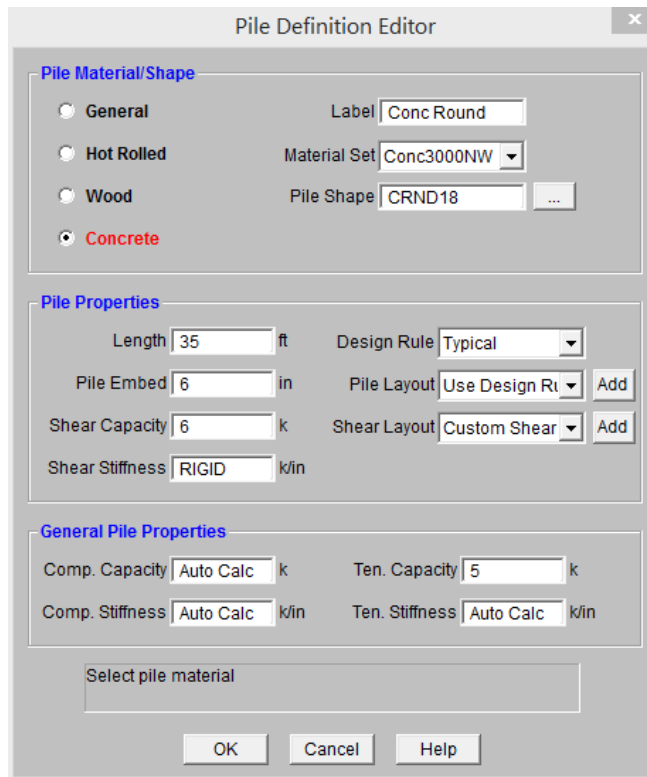
Here we will explain the pile and pile cap interface and modeling techniques.

To see individual pile results see the Pile Results topic. To see pile cap results see the Pile Cap Results topic.

Defining Singular Piles

Piles can be defined at the time of drawing by the [....] button from within the **Assign Support Type at Points** dialog to open the Pile Definition Editor. They can also be defined ahead of time from the **Pile Definitions** spreadsheet, accessible from the Data Entry toolbar or Spreadsheets drop-down list.

Pile Definition Editor

This dialog provides a quick input selection screen to use to build Pile Definitions. Any updates made from within this dialog will automatically update the **Pile Definitions** spreadsheet. Each of the items here are explained below.

General Tab

Label

This label provides a unique name for the pile definition.

Shape

For **general** piles, two shape options are provided: **Round** and **Square**. For other pile shapes you can approximate them to one of these shapes (H-piles would just be added as a square pile with the side dimension equal to either the depth or width). This shape will affect punching shear calculations. A general pile material is consistent with the analysis and pile checks prior to RISAFoundation version 9.0.

The **hot rolled steel** shape database is provided when a hot rolled material is selected. Although the full HR Steel shape database is provided, steel piles are typically pipe and H-section shapes.

The **wood** shapes are available when a wood material is selected. The program assumes a constant cross section along the pile's length in the soil.

The **concrete** shapes are available when a concrete material is selected. The program assumes a constant cross section along the pile's length in the soil.

Material

The pile materials available are hot rolled steel, concrete, and wood. The material properties are defined in the Materials spreadsheet under the respective material tab. The program will use the material properties to calculate the pile tensile and compressive capacities and stiffnesses.

Note:

- Selecting a general pile material and shape is consistent with the analysis and pile checks prior to RISAFoundation version 9.0.

Length

The total length of the pile in soil. The pile length is used for HR steel, Concrete, and Wood pile design calculations.

Embedment Depth

This is depth that the pile is embedded in the pile cap. This affects the "d" for pile cap design and also punching shear calculations.

Note:

- The **Embedment Depth** field is only used to calculate "d" with Pile Cap entities. If you are using slabs with pile supports then this embedment depth is not considered. Thus, for pile cap entities d = t_{slab} - pile embed - bot cover - $d_{bot\ bar}$. For slab entities d = t_{slab} - bot cover - $d_{bot\ bar}$. Thus, the bottom cover must include the embedment depth for slabs supported on piles but not for pile cap entities.

Pile Compressive and Tensile Capacities

For HR Steel, Concrete, and wood piles, the program will calculate these values based on the material properties and display *Auto Calc*. For general piles, the user will specify the pile allowable capacities. The program will use these capacities to do a design check of your general piles.

Pile Shear Capacity

For all pile shapes and materials, the user will specify the pile allowable shear capacity. The program will use the user input value of the shear capacity to do a design check.

Note:

- The program currently calculates the static bearing capacity however it does not consider lateral pile design at this time.

Pile Compressive and Tensile Stiffness Capacities

For HR Steel, Concrete, and wood piles, the program will calculate the stiffness values based on AE/L and displays *Auto Calc*. The *Auto Calc* stiffness entry may be overridden by a user entered value.

For general piles, this entry allows you to define the stiffnesses for your piles. By default the compression entry is set to completely RIGID. The tension entry is set to 0, meaning there is no resistance to tension.

Note:

- The default entries for general piles are typical for most pile cap designs. To define the piles as a spring support simply enter the pile stiffnesses.
- To get an approximate vertical stiffness of an individual pile, simply take the pile capacity and divide that by the maximum possible settlement.
- When Auto Calc is displayed for the tensile stiffness of concrete piles, the program assumes only the steel reinforcement will contribute to the tensile stiffness. However because the reinforcement is designed after the analysis, the program makes an assumption for the analysis that the tensile stiffness is 5% of the compressive capacity.

Pile Shear Stiffness

For all pile shapes and materials, this entry allows you to define the shear stiffness for your piles. By default the shear stiffness entry is set to completely RIGID.

Note:

- To get an approximate lateral stiffness of an individual pile, simply take the pile capacity and divide that by the maximum possible lateral deflection.

Concrete Pile Tab

The concrete reinforcement can also be specified in the *Concrete Pile* tab of the Pile Definitions spreadsheet. The user can either use the reinforcement specified in the *Design Rules* spreadsheet or create a custom rebar layout to apply to the concrete pile.

ACI 318-19 provides explicit equations in Table 13.4.2.1 on allowable compressive strength for piles depending on the pile types. When ACI 318-19 code is selected, the pile type can be modified in Concrete Pile tab, currently RISAFoundation supports pile types (a)-(d) in this table.

Defining Pile Caps

Pile caps can be defined at the time of drawing by the [....] button from within the **Assign Support Type at Points** dialog. They can also be defined ahead of time from the **Pile Cap Definitions** spreadsheet, accessible from the Data Entry toolbar or Spreadsheets drop-down list.

Pile Cap Definition Editor
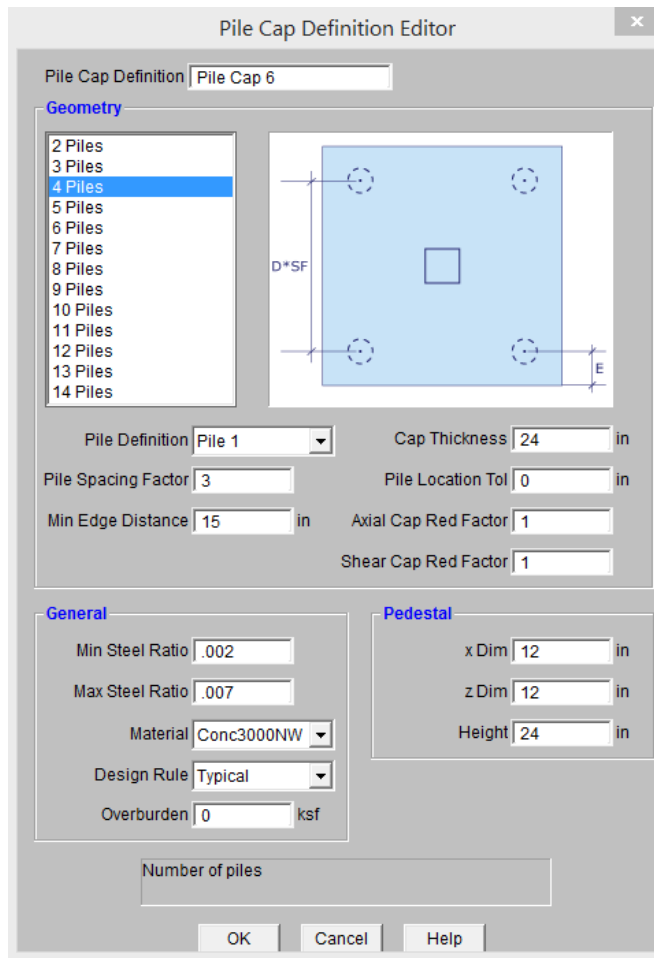
This dialog provides a quick input selection screen to use to build Pile Cap Definitions. Any updates made from within this dialog will automatically update the **Pile Cap Definitions** spreadsheet. Each of the items here is explained below.

General Tab

Label

This label provides a unique name for the pile cap definition. When clicking in this column a red arrow will become available. This will open the **Pile Cap Definition Editor** and allow you to update all spreadsheet entries via a single dialog.

Reinforcement Ratios

When ACI 318-19 is selected, ACI 318-19 Section 24.4.3 stipulates that the ratio of deformed shrinkage and temperature reinforcement area to gross concrete area shall be greater than or equal to 0.0018.

When other ACI 318 editions are selected, acceptable steel ratios are controlled by ACI 318-14 Section 24.4.3 (ACI 318-11 section 7.12.2.1) which stipulates that the minimum steel ratio for Grade 40 or 50 steel is .0020. For Grade 60 or greater the ratio is 0.0018.

Note:

- When you are only viewing three decimal places, the 0.0018 appears as 0.002, though we are really using 0.0018. To change the number of decimal places, go to Tools-Application Settings-Data Entry tab

In addition, the codes limit the maximum steel that can be used. ACI 318-14, for example, limits the minimum strain on the reinforcing steel to 0.004.

Note:

- If the Max Steel Ratio you define does not meet minimum requirements or strength requirements we will override your ratio and give you a message in the detail report.

Material

This allows you to define the pile cap material from the Materials spreadsheet.
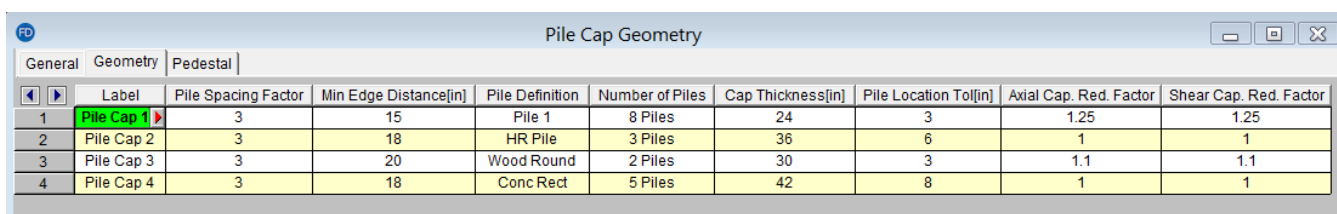
Design Rules

This allows you to define the design rule from the Design Rules spreadsheet. For pile caps we take information from the **General**, **Footing/Pile Cap**, and the **Pedestal** tabs of the design rules.

Overburden

This allows you to define a soil overburden on your pile cap. This overburden weight is added to the weight of the slab for pile demand calculations. These pile forces affect flexural and shear design for the cap.

Geometry Tab

Label	Pile Spacing Factor	Min Edge Distance[in]	Pile Definition	Number of Piles	Cap Thickness[in]	Pile Location Tol[in]	Axial Cap. Red. Factor	Shear Cap. Red. Factor
1	Pile Cap 1	3	15	Pile 1	8 Piles	24	3	1.25	1.25
2	Pile Cap 2	3	18	HR Pile	3 Piles	36	6	1	1
3	Pile Cap 3	3	20	Wood Round	2 Piles	30	3	1.1	1.1
4	Pile Cap 4	3	18	Conc Rect	5 Piles	42	8	1	1

Pile Spacing Factor

This value defines the center to center spacing of piles. This factor multiplied by the diameter/side dimension of the pile defines this spacing. Typically this factor is used to define capacity reduction factors for closely spaced piles.

Min Edge Distance

This value defines the minimum distance from the centerline of pile to edge of pile cap.

Pile Definition

This allows you to choose a pile type for your pile cap from the Pile Definitions spreadsheet.

Number of Piles

This selection defines the number and layout of piles. These pile layouts were derived from the CRSI manual on pile caps. Here are the standard layouts from the CRSI manual.
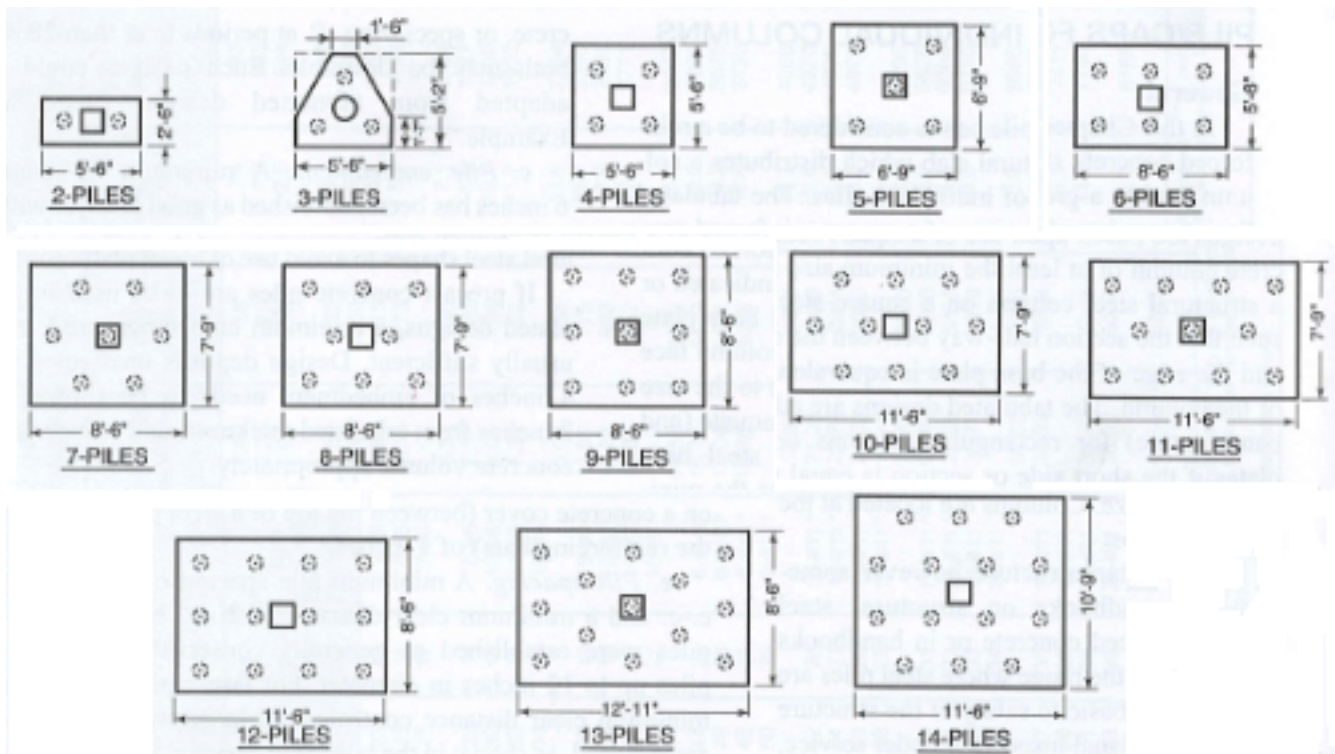
Figure 13-2 from CRSI Design Handbook 2008

Note:

- RISAFoundation uses the pile layouts from CRSI, but allows you to change the spacing, edge dist and other properties to customize your pile cap design.
- Currently, the program only includes pile cap design up to 14 piles. For custom pile caps and pile caps that have greater than 14 piles you can use a slab with singular piles to consider this.
- Once you have defined your pile caps and applied them to your model, solve the model and look at the Pile Cap Detail Report for exact geometry and locations of piles.

Cap Thickness

This value defines the total thickness of your pile cap.

Pile Location Tolerance

This allows you to define a field placement tolerance that is acceptable for your pile caps. This affects two checks in the program: the moment demand calculation and the one way shear capacity calculation.

Axial and Shear Capacity Reduction Factors

These factors will reduce the pile capacities from the Pile Definitions spreadsheet. The value in the Pile Definitions spreadsheet will be divided by this value to produce a reduced capacity based on closely spaced pile groups.

Pedestal Tab

The pedestal dimensions are entered in these fields. The x dim and z dim values are the pedestal dimensions parallel to the local x and z axes of the pile cap, respectively. The height is the distance from the upper surface of the footing to the top of the pedestal.

Note:

- Pedestals are always located at the centroid of the pile group.

Pile and Pile Cap Modeling Considerations

Modeling with Piles, Slabs and Pedestals vs Pile Caps

As discussed previously, the general functionality of the program is that piles would be used in scenarios where a mat slab was drawn and piles are located at specific locations under the slab. Pile caps, on the other hand, would be used at a specific point location to support a reaction coming down.

The Pile Cap feature has some limitations and can not be applied to all situations. If there is a limitation in the pile cap, however, using the singular piles, slabs and pedestals within the program will almost always allow you to overcome them. The piles, slabs and pedestal option allows you to create whatever custom geometries you have.

The pile cap pile layouts are configured based on the CRSI design manual layouts. Thus, these configurations can not be manipulated. However, it is possible to simply create a pile cap for a concentrated force using singular piles, a slab and a pedestal. Simply draw your pile cap as a slab element, add the pedestal using the draw pedestals utility, and locate/draw your piles according to your specific pile configuration. Below is an example where this was done.
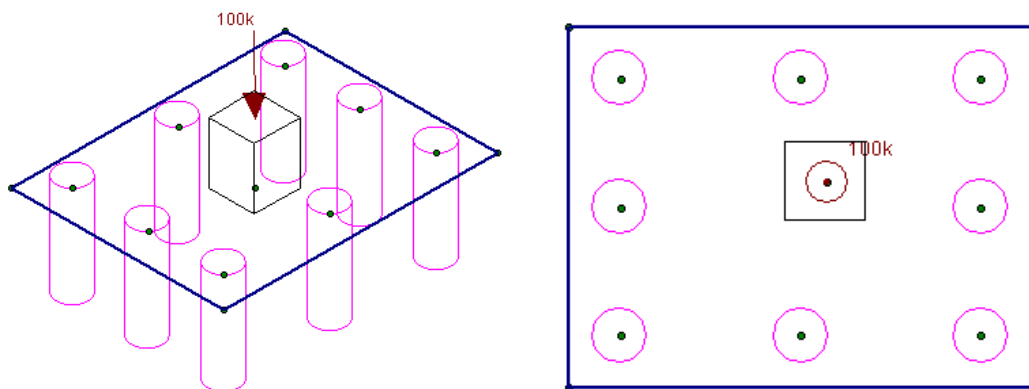
Compare this with CRSI layout for 8 piles:
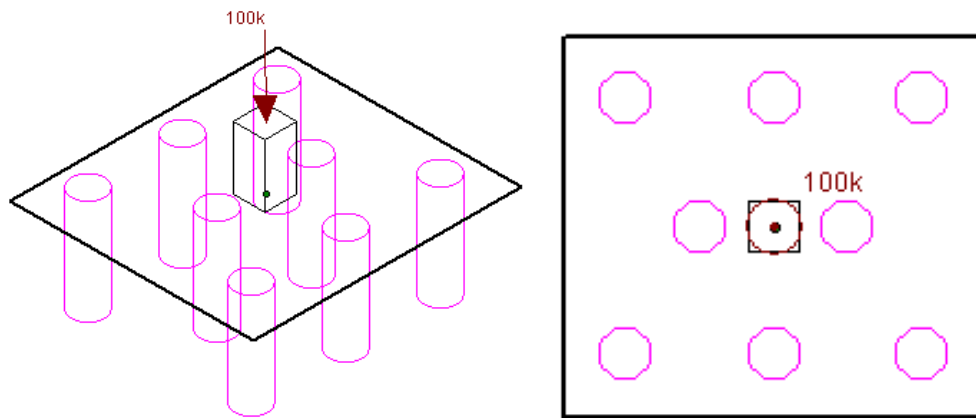
From this example we can also see that locating a pedestal at an eccentric location to the pile group is also possible.

It is also possible to model identical configurations both with the piles, slabs and pedestals and the pile cap. When doing this considering identical loading you may get nearly identical results.
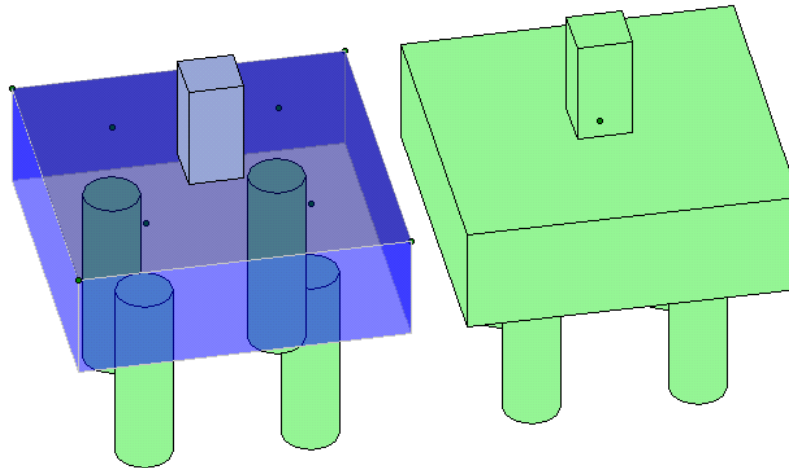
Rigidity Assumptions

Keep in mind that pile cap entities assume the cap is fully rigid. Slabs supported with piles assume a semi-rigid slab. Thus, the slab entity will place load in the piles based on the rigidity of the cap and this can alter the force distribution into the piles.

Also, the default stiffness for general piles is completely rigid. Thus, you can get very different distributions of force to these two seemingly identical entities. However, if you lower the pile stiffness to a non-completely rigid value then your pile force distribution should be much more similar.

If the slab is relatively thick, such that the slab pile cap behaves in a fairly rigid manner, and the pile stiffnesses are not set to completely RIGID then likely you will get similar pile force distributions between the two pile cap entities.

Pile Groups and Thickened Portions of a Slab within a Mat

Many times there are regions within a large mat foundation where large loads are coming down and a pile group is required in this location. The program does not allow you to define a pile cap within a slab, but you can simply draw in individual piles at this close spacing. See the example below, where large column loads are coming down on a mat foundation.
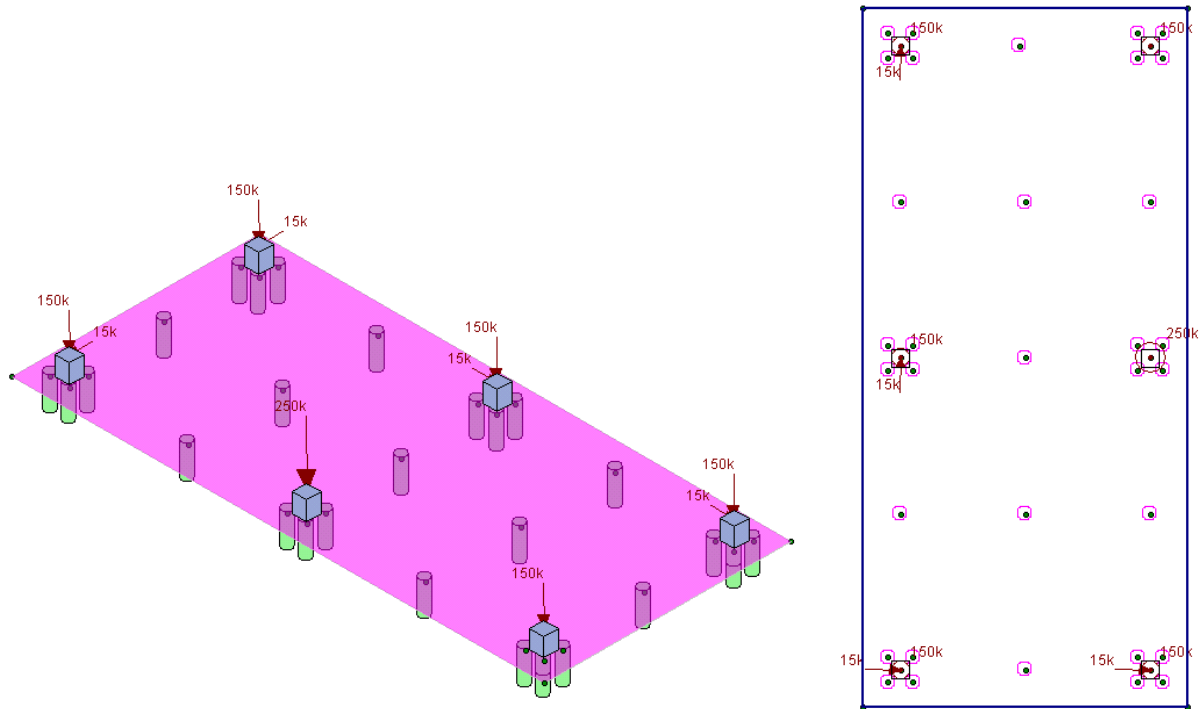
In this scenario the slab is one continuous thickness over the entire mat. If the slab was thickened at the pile groups and around the exterior, we would simply need to draw multiple slab elements adjacent to each other. If two slab elements are adjacent to each other, they will have full fixity to each other. If we thicken the previous model it may look something like this:
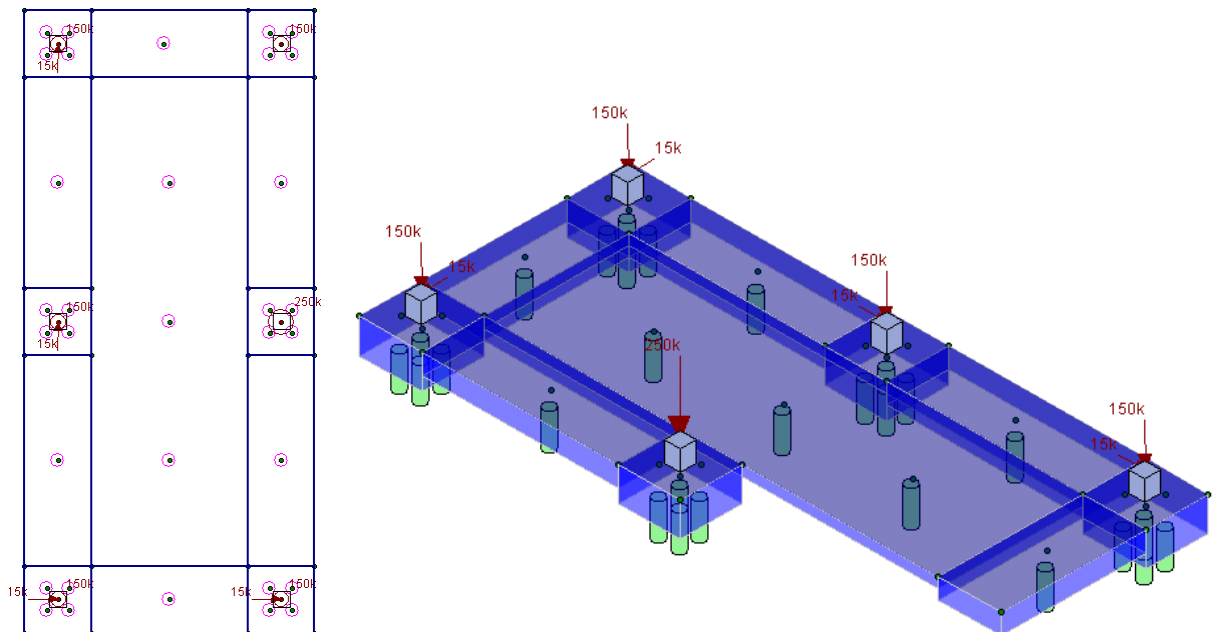
Here we can see that this mat slab was drawn in multiple pieces of different thicknesses. The image on the right shows a rendered view with part of the slab cut away to see the differences in thickness.

Pile and Pile Cap Limitations

- The program does not consider battered piles.

Pile Caps - Design

In this section we will talk specifically about the calculations for pile caps. These design results are NOT available when using piles and a slab to create your pile cap. The only portions of this section that are applicable to individual piles is the Pile Punching Shear section. For the slab/pile cap design, see the Slabs topic

Pile Cap Design Philosophy

Pile cap design within RISAFoundation considers the beam philosophy as opposed to the truss philosophy. Both are acceptable by code (ACI 318-14 Section 13.4.2/ACI 318-11 Section 15.5.3).

Pile Capacity Checks

These checks allow the engineer to see if any of the piles in the model are exceeding the allowable forces that were entered.

Allowable Pile Loads

The allowable pile loads are defined by the user for general piles in the Pile Definitions spreadsheet. For hot rolled steel, concrete, and wood piles, the program calculates the axial capacity based on the pile and material properties. If the piles are part of a pile cap element then, these allowable loads are reduced by the Reduction Factors on the Pile Cap Definitions - Geometry tab.

Loads Developed in Piles in Pile Cap Applications

The loads developed in the piles within the pile cap elements are found using the equations below. Some things to note on the computation of these forces:

- Loads are only applied to the cap at the pedestal location, which is placed at the centroid of the cap.
- The axial pile stiffness (given in the Pile Definitions spreadsheet) is considered the same for both compression and tension and is taken from the **Comp. Stiffness**.
- The pile cap is assumed rigid.
- The centroid location for force calculations of the piles is from the mid-height of the pile cap.

$$P_{pile} = \left(\frac{P}{N} \right) + \left[\frac{\left[M_x \cdot z_p + V_z \cdot \left(h_{ped} + \frac{d}{2} \right) \cdot z_p \right]}{\Sigma z_p^{\,2}} \right] + \left[\frac{\left[M_z \cdot x_p + V_x \cdot \left(h_{ped} + \frac{d}{2} \right) \cdot x_p \right]}{\Sigma x_p^{\,2}} \right]$$

$$Vpile_x := \frac{V_x}{N} + \frac{T \cdot \sqrt{x_p^{\,2} + y_p^{\,2}}}{\Sigma x_p^{\,2} + \Sigma y_p^{\,2}} \cdot \sin(\alpha) \qquad Vpile_z := \frac{V_z}{N} + \frac{T \cdot \sqrt{x_p^{\,2} + y_p^{\,2}}}{\Sigma x_p^{\,2} + \Sigma y_p^{\,2}} \cdot \cos(\alpha)$$

$$V_{pile} := \sqrt{Vpile_x^{\,2} + Vpile_z^{\,2}}$$

where:

P_{pile} = Total axial load on an individual pile

P = Column load and self-weight of the pile cap, pedestal and overburden

N = total number of piles in pile cap

$M_{x,z}$ = Applied moment to pedestal

x_p, y_p = Distance from centroid of pile cap to centroid of individual pile parallel to the respective axis.

$V_{x,z}$ = Applied shear to pedestal.

h_{ped} = Height of pedestal from the top of the pile cap

d = Thickness of pile cap

T = Applied torque to pedestal

α = Angle between the centroid of the column and the centroid of the pile location, taken from the negative local x direction counter-clockwise

Vpile$_x$: Shear in pile in local x direction

Vpile$_z$: Shear in pile in local z direction

V_{pile}: Total demand shear in pile (in any direction)

Note:

- The program does not consider the self-weight of the pile as part of the axial force in the pile. See the "Pile Caps - Design" section for information on how to account for pile self-weight.

Pile Cap Flexural Design

The flexural design of pile caps is based on the moment at the face of the pedestals in each direction. The image below explains how the Mu moment is evaulated.
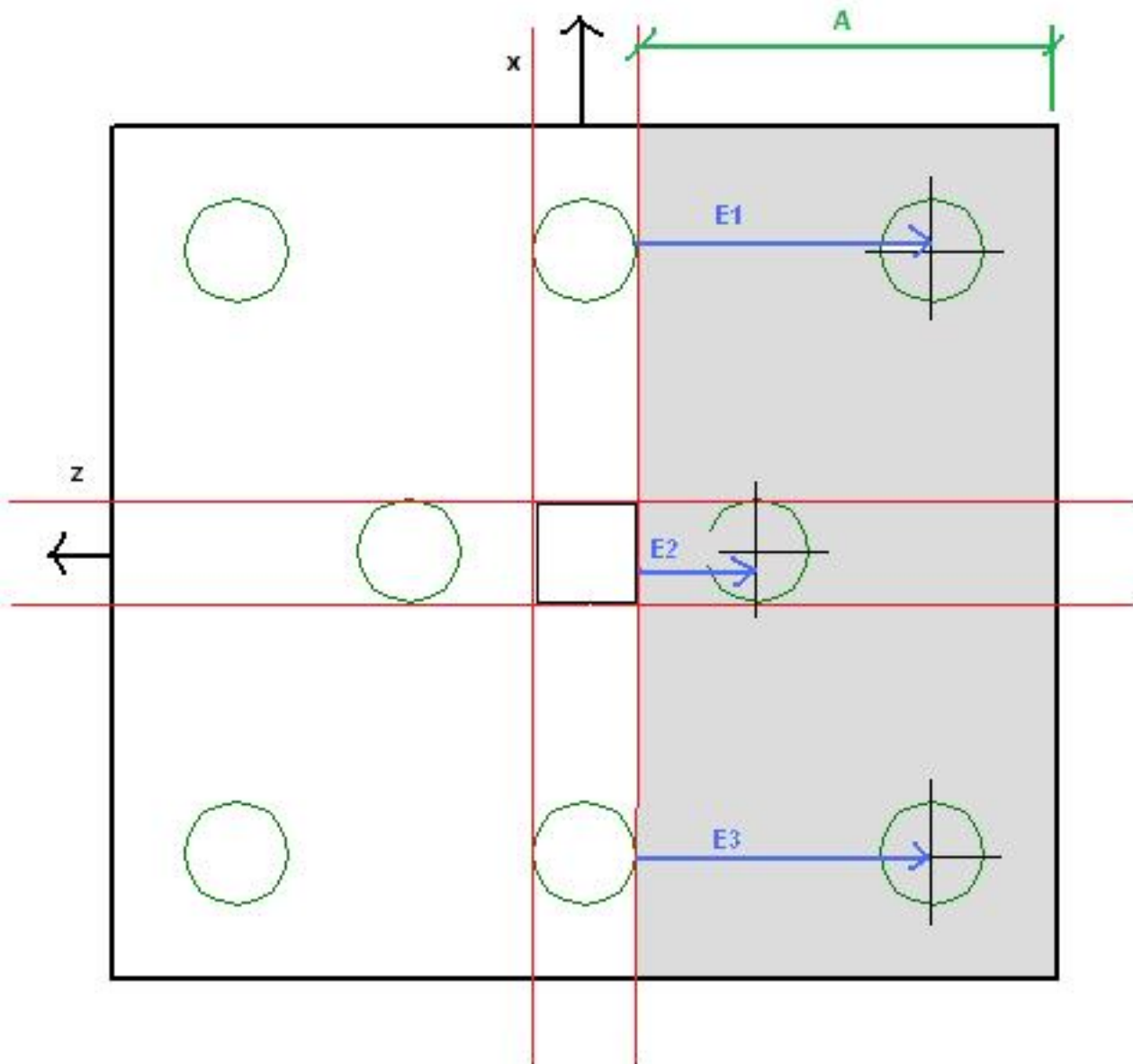
The red lines in the image show the locations where we compute the moments. The calculation of the moment at the face of column is as follows:

$$M_u := P_{u1} \cdot E_1 + P_{u2} \cdot E_2 + P_{u3} \cdot E_3 - \left(\frac{A}{2}\right) \cdot (A \cdot \text{Width}) \cdot \left(\rho_{slab} \cdot t_{slab} + \text{Over}\right)$$

Thus, the program uses the factored pile forces multiplied by their horizontal distance to the face of column and subtracts the self-weight of the pile cap and overburden in this area multiplied by its moment arm. In the example this would be the Muxx moment at the right face of the pedestal. This calculation would then be done for the left face of the pedestal and the enveloped maximum positive and negative moments are given.

This calculation is then done identically in the Muzz direction.

Flexural Reinforcement Design

The design of reinforcement runs through a design routine to come up with the area of steel required in both directions. This routine checks the area of steel based on flexure and then checks minimum requirements and considers the short side amplification factor as well.

$A_{s \, req'd}$ = area of steel required for the design moment, using typical bending equations.

1. If $A_{stop \, req'd}$ > .0033bd use $A_{stop \, req'd}$. This is exactly the same for A_{sbot}.
2. If $A_{stop \, req'd}$ < .0033bd < 4/3A_s, use 0.0033bd. This is exactly the same for A_{sbot}.
3. If 4/3$A_{s \, top \, req'd}$ < .0033bd, use4/3A_s. This is exactly the same for A_{sbot}.
4. Check for amplification of A_s in the short direction considering the β factor. See below for more information.
5. After these steps compare:
    1. ACI 318-14: $A_{s \, top}$ to 0.0018bh and $A_{s \, bot}$ to 0.0018bh. If both are greater than 0.0018bh then you are done. If less than 0.0018bh, then steel is added to meet this requirement for each face.
    2. ACI 318-11: $A_{s \, top}$ + $A_{s \, bot}$ to 0.0018bh. If greater than 0.0018bh then you are done. If less than 0.0018bh, then we simply take the amount that yet needs to be added and add half of the steel to the top and half to the bottom.
6. The final check would be to compare the steel ratios to the min and max ratios from the Pile Cap Definitions-General tab.

Notes:

- If the user-defined steel ratios do not meet code requirements we will use the code requirements and provide a note in the detail report.

- **β Factor consideration**
    - β = ratio of long dimension of pile cap to short dimension of pile cap.
    - $A_{s\beta} = A_{s \, req'd}$*(2β)/(1+β)
    - For pile caps with clipped corners we are taking the full dimension for the β calculation.
- If no flexural reinforcement required in a given direction, all of the shrinkage and temperature steel in that direction will be added to the bottom of the slab.
- If top reinforcement is required in one direction and bottom reinforcement is required in both directions, then all provisions may be satisfied without giving reinforcement in the top in the direction where reinforcement is not required. We currently do not reinforce this opposite direction, though it is recommended to do so.

d Calculation

For bottom bar reinforcement design:

d = t_{slab} - pile embed - bot cover - $d_{bot \, bar}$

For top bar reinforcement design:

d = t_{slab} - top cover - $d_{top \, bar}$.

Note:

- Because we do not know exactly how the reinforcement will be placed, we take the interface of the two directions of reinforcement as the location for the d calculation.
- The **Embedment Depth** field is only used to calculate "d" with Pile Cap entities. If you are using slabs with pile supports then this embedment depth is not considered. Thus, for pile cap entities d = t_{slab} - pile embed - bot cover - $d_{bot \, bar}$. For slab entities d = t_{slab} - bot cover - $d_{bot \, bar}$. Thus, the bottom cover must include the embedment depth for slabs supported on piles, but not for pile cap entities.

Pile Location Tolerance Consideration

If there is a pile location tolerance defined, we simply modify Mu equation above, adding the tolerance to the moment arm for each individual pile. This will nearly always increase the moment demand.

$$M_u := P_{u1} \cdot (E_1 + Pile_{Tol}) + P_{u2} \cdot (E_2 + Pile_{Tol}) + P_{u3} \cdot (E_3 + Pile_{Tol}) - \left(\frac{A}{2}\right) \cdot (A \cdot Width) \cdot (\rho_{slab} \cdot t_{slab} + Over)$$

One Way (Deep Beam) Shear Check

This check considers the entire pile cap as a beam element and checks the overall shear across the pile either at a distance **d** from the face of the pedestal, or at the face of the pedestal. Where this check occurs depends on the distance, w.

w = the distance along the local axes from the face of pedestal to the centerline of the nearest pile in that direction.

If **d < w**, the critical location where shear is checked is taken at a distance d from the face of the pedestal. In this case the equation for the shear stress capacity is found by:

$$v_c = 1.9\sqrt{f_c'} + 2500\rho_w\left(\frac{V_u d}{M_u}\right) \geq 2\sqrt{f_c'} \quad \text{(ACI Eq. 11-5)}$$

ACI 318-14 Table 22.5.5.1/ACI 318-11 Equation 11-5

If **d > w**, then the critical location where shear is checked is taken at the face of support.

$$v_c = \left(\frac{d}{w}\right)\left[3.5 - 2.5\left(\frac{M_u}{V_u d}\right)\right]\left[1.9\sqrt{f_c'} + 2500\rho_w\left(\frac{V_u d}{M_u}\right)\right]$$
$$\leq 10\sqrt{f_c'} \qquad ; 1.0 > M_u/V_u d > 0; \infty > V_u d/M_u \geq 1.0$$

CRSI Design Handbook 2008 Eqn 13-2 on P.13-26

Note:

- When ACI 318-19 is selected, the Vc equations were changed per Table 22.5.5.1. Since pile cap has no shear reinforcement, the Vc shall be calculated using Equation (c) in the table. Ignoring axial loads in pile cap, the vc term $1.9\sqrt{fc}+2500*\rho w*(Vu*d/Mu)$ in the above two equations is changed into the equation below. Note that size effect factor λs is a function of effective depth of the slab d, and is calculated per Equation (22.5.5.1.3).

$$v_c = 8\lambda_s\lambda(\rho_w)^{1/3}\sqrt{f_c'}$$
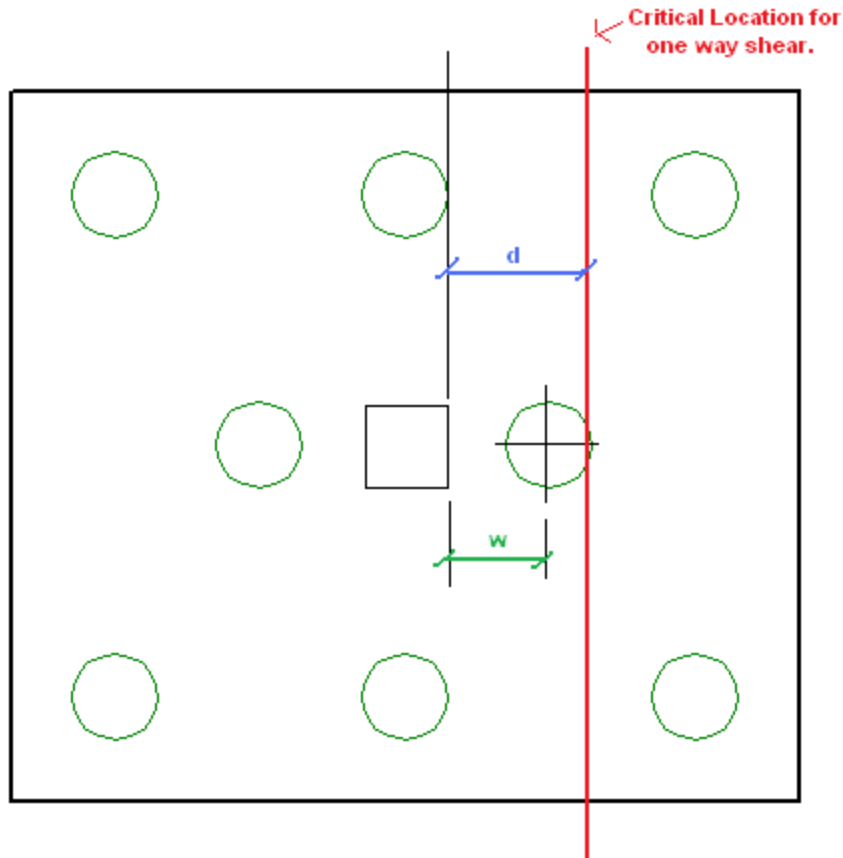
The equation to calculate the shear capacity is:

$$\phi V_n := \phi \cdot v_c \cdot b \cdot d$$

$$\phi V_{nmax} := \phi \cdot 10 \cdot \sqrt{f_c} \cdot b \cdot d$$

(φVnmax only applies for eqn 13-2)

where:

M_u = enveloped moment at critical location.

V_u = enveloped shear at critical location

$\rho_w = A_{sprovided}/bd$

$d = t_{slab}$ - pile embed - cover - d_{bar}

We do these calculations for both sides of the pedestal in both directions and envelope the worst code check in each direction.

Considerations:

- Mu and Vu are the enveloped moment and shear values. The maximum moment and shear occur at the same location for the majority of cases.

- We are NOT considering the one way beam shear failures that can occur in other locations within the pile cap. An example of this type of failure would be a shear force that might shear off a corner of the pile cap.
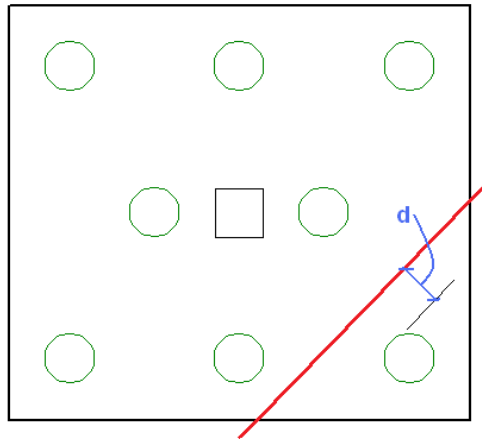
- For lightweight concrete, the lambda reduction factor specified in the Materials spreadsheet is applied to the capacity per ACI 318-14 Section 22.5.6.1 (ACI 318-11 Section 11.2.1.2).

Pile Location Tolerance Considerations

When a pile location tolerance is defined that tolerance is simply added to the w term in each direction. An increased w will decrease the one way shear capacity of the pile cap.

Code Considerations

For the Saudi Code the ACI 318-05 is used. This is noted in the detail report. The Mexican and Canadian code are simply using their standard beam and slab one way bending equations.

Pile Punching Shear Checks

The punching shear in a pile is simply the axial compression force in a pile considering strength load combinations.

The punching shear capacity is obtained via the ACI code equations, seen below. The code requires this value to be the minimum of these three values.

$$V_c = \left(2 + \frac{4}{\beta}\right)\sqrt{f_c'}\, b_o d \qquad (11\text{-}33)$$

$$V_c = \left(\frac{\alpha_s d}{b_o} + 2\right)\sqrt{f_c'}\, b_o d \qquad (11\text{-}34)$$

$$V_c = 4\sqrt{f_c'}\, b_o d \qquad (11\text{-}35)$$

Note:

- When ACI 318-19 is selected, an additional size effect factor λs is added in the above equations, λs is a function of effective depth of the slab d, and is calculated per Equation (22.5.5.1.3).

- For lightweight concrete, the lambda reduction factor specified in the Materials spreadsheet is applied to the capacity per ACI 318-14 Section 22.5.6.1 (ACI 318-11 Section 11.2.1.2).

b_o Calculations

The calculation of b_o is done considering an interior, edge and corner condition. The program calculates a b_o value for each of these conditions for each pile and then uses the minimum value for design and reports that condition.

When there is no reinforcement at the top face of the pile cap (in either direction)

If there is no reinforcement in the top of the pile cap, then for the consideration of pile punching the slab is considered unreinforced. Because of this we use the provisions from ACI 318-14 Chapter 14 (ACI 318-11 Chapter 22). From ACI 318-14 Section 21.2.1 (ACI 318-11 Section 9.3.5, the φ factor for shear is 0.55 (as compared to 0.75 for reinforced concrete). The punching shear equation for structural plain concrete (ACI 318-14 Table 14.5.5.1/ACI 318-11 Equation 22-10) is the reinforced punching shear equations taken with a 2/3 factor. Thus, the ratio of using structural plain concrete to reinforced concrete is : 2/3*(0.55/0.75) = .488. RISAFoundation rounds this to 0.5 and takes a **50% reduction in capacity** for punching shear if there is no reinforcement in the top slab.

Note:

- If there is reinforcement in ONE direction we do not use this reduction.

When the Pile's Local Axes do not correspond to the slab geometry

The punching shear perimeters are always investigated with respect to the pile local axes. As shown below, there may be cases where these local axes do not correspond to the cap edge or corner geometry adjacent to that pile. In cases like these it is possible for this to result in an artificially large assumed punching shear perimeter.

Using Equivalent Square Piles

When calculating the punching shear perimeter for a round pile, the program always converts the pile into an equivalent square pile of equal perimeter (similar to super-imposed square in the image below). This greatly simplifies the calculation of punching perimeters when close to an edge or corner.
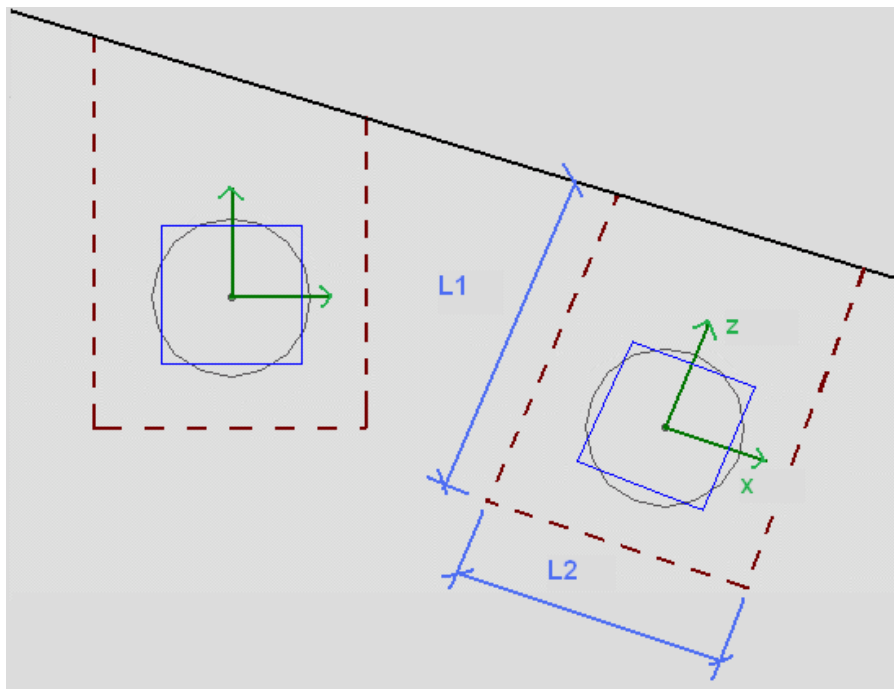
d Calculation

d = t$_{slab}$ - pile embed - top cover - d$_{top\ bar}$

Group Punching for Piles

We do not currently calculate a group punching check. At solution we will check to see if closely spaced piles have shear perimeters that overlap. If so we will give a warning in the warning log noting this. We will still give the capacities assuming the total shear perimeter for the pile not considering group action.

Beams/Pedestal intersecting the shear perimeter of a Pile

- When a beam is co-linear with the pedestal joint, the program will not provide a punching shear check on the pedestal. In these cases a code check of NC or will be listed.
- When the pedestal for the pile cap is located in the piles shear perimeter the program does not check for this. Thus, the punching shear capacity will be calculated as if there is no pedestal above.

Pedestal Punching Shear Checks

Pedestal punching shear checks are handled in much the same way that pile punching shear checks are considered. A major difference is that pedestals can transfer moment into the pile cap. Thus, the pile cap punching shear capacity is reported as a stress and the maximum stress induced due to axial load and moment is checked. See the Pile Cap Results for more on this.

To calculate the shear demand in the pile, simply sum the factored axial forces in the piles outside of the pedestal shear perimeter and add to that the factored self-weight of the pedestal and a ratio of the factored self-weight of the pile cap.

$$V_{punch} := \Sigma P_{uOutsideShearPerim} + SelfWeight_{Pedestal} + SelfWeight_{Slab} \cdot \left(\frac{No_{PilesOutsidePerim}}{TotalNo_{Piles}} \right)$$

Note:

- For lightweight concrete, the lambda reduction factor specified in the Materials spreadsheet is applied to the capacity per ACI 318-14 Section 22.5.6.1 (ACI 318-11 Section 11.2.1.2).

d Calculation:

d = t$_{slab}$ - pile embed - bot cover - d$_{bot\ bar}$

When Piles intersect the shear perimeter of the Pedestal

At solution the program checks to see whether any of the piles in the pile cap have their **centerlines** located within the shear perimeter of the pedestal. If so, then these piles do not contribute to the pedestal punching shear demand in the cap. If this centerline falls outside of the shear perimeter than the entire force of the pile adds to the pedestal punching shear demand.
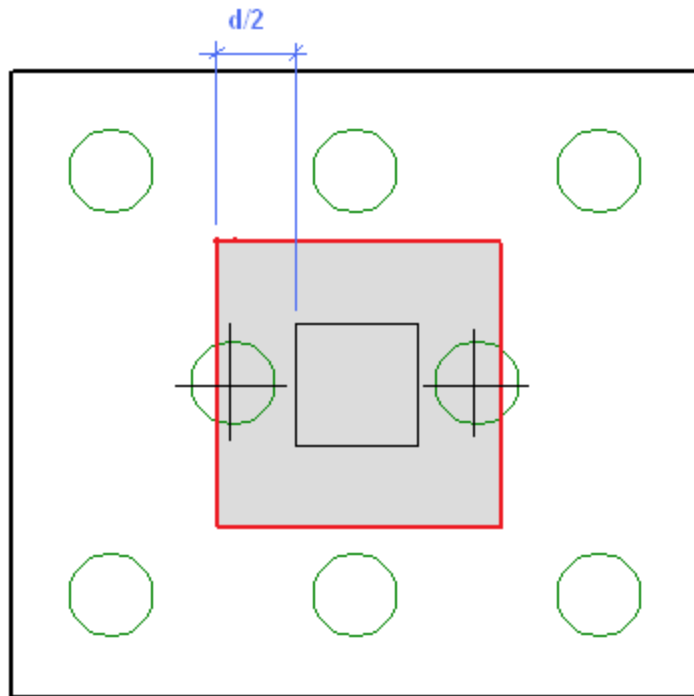
Note:

- ACI 318-14 Section 13.4.2.5 (ACI 318-11 Section 15.5.4) discusses the cases where piles are very near the shear perimeter location. They recommend proportioning load according to how much of the pile is within the pedestal shear perimeter. RISAFoundation only considers the centerline of the pile.

Pedestal Design

Pedestal design for pile caps is identical to what is done for pedestals on slabs. See the Pedestals-Design Results topic for detailed information.

Pile Design

Pile design for pile caps is identical to what is done for individual piles. See the Pile Results topic for detailed information.

Pile Results

Results for individual piles can be found in the **Pile Results** spreadsheet.

Pile Design Checks

Pile UC Checks

The Shear, Compression and Tension UC values give the ratio of demand forces to allowable forces.

- For general piles, the allowable forces are pulled directly from the [Pile Definitions](#) spreadsheet. These are given for the load combination that produces the worst case UC.
- For hot rolled steel, wood, and concrete piles, the compressive tensile capacities are calculated by the program based on the material, cross sectional area, and design code selected in the Model Settings.
- For concrete piles, only the reinforcement is considered in the tensile capacity.

Forces

The shear, compression and tension forces are given for the load combination that produces the worst case UC.

Note:

- The program does not consider the self-weight of the piles as part of the Pile's compression/tension forces. Therefore, user should account for pile self-weight by manually applying a force to account for the pile's self-weight. For piles part of a pile cap system, the user should apply a manual force on the pile cap which then would distribute onto the supporting piles. For piles supporting slabs, the pile self-weight can be applied as a point load directly on top of each respective pile.

- For shear this value is directionless. The program takes:

$$V = \sqrt{V_x{}^2 + V_z{}^2}$$

Gov LCs

These are the governing load combinations for each type of force which produce the highest unity check.

Soil UC Checks

The soil unity checks are the static pile capacities for tension and compression.

The **Soil Comp UC** is the ratio of the required compressive force to the resisting force. The compressive capacity is equivalent to the sum of the skin friction at each layer and the tip bearing capacity at the layer of soil which the pile terminates in. The capacity is divided by the pile factor of safety in the Model Settings. For the soil compressive

capacity, the program uses the equation below (Equations reference: Joseph E. Bowles, *Foundation Analysis and Design, 5th Ed.*):

$$Q_u = \frac{Q_p + \Sigma Q_s}{FS}$$

- Qu : Ultimate pile capacity in compression
- Qp : Bearing capacity at each soil layer, $Q_p = A_p \times q_p$
    - Ap : Area of pile including the soil plug
    - qp : Unit tip-bearing
- Σ : Summation process over i soil layers making up the soil profile over the length of the pile shaft
- Qs : Skin Friction at each soil layer, $Q_s = p \times \Delta L \times q_s$
    - p : Perimeter of pile cross section including soil plug
    - ΔL: Length of pile in soil layer
    - qs : Unit skin friction
- FS : Pile Factor of Safety input in the Model Settings

Note:

- The program does not consider the self-weight of the pile as part of the axial force in the pile. See the "Forces" section for information on how to account for pile self-weight.

The **Soil Tens UC** is the ratio of the required tensile force to the resisting tensile force. The tensile capacity is equivalent to the weight of the pile plus the sum of the skin friction at each soil layer. The Soil Tensile UC is divided by the pile factor of safety specified in the Model Settings. For the soil tensile capacity, the program uses the equation below (Equations reference: Joseph E. Bowles, *Foundation Analysis and Design, 5th Ed.*):

$$Q_u = \frac{W + \Sigma Q_s}{FS}$$

- Qu : Ultimate pile capacity in tension
- W : Weight of the pile, $W = \gamma \times A_p \times L$
    - γ : unit weight of pile
    - Ap : Area of pile
    - L : Length of pile
- Σ : Summation process over i soil layers making up the soil profile over the length of the pile shaft.
- Qs : Skin Friction at each soil layer, $Q_s = p \times \Delta L \times q_s$
- p : Perimeter of pile cross section
- ΔL: Length of pile in soil layer
- qs : Unit skin friction
- FS : Pile Factor of Safety set in Model Settings

Punching Shear Values

UC Check

This is the ratio of demand punching shear to punching shear capacity for the worst case load combination.

Gov LC

This is the load combination that produces the worst case punching shear for each individual pile.

Location

This helps to define the geometry for the shear perimeter. The program calculates a shear perimeter assuming a corner failure, an edge failure and an interior failure. The smallest perimeter is the one that governs and is given here.

Vuy

This is the factored punching shear demand.

Phi*Vny

This is the punching shear capacity of the slab/pile cap at this pile location. For more information on this calculation see the Pile Caps - Design topic.

Punching Shear Geometries

Effective Depth

The effective depth, d, of the slab/pile cap is used to define the punching shear perimeter at a distance of d/2 beyond the edge of the pile.

$d = t_{slab}$ - pile embed - top cover - $d_{top\ bar}/2$

The effective depth is based on the smallest depth (to top reinforcing) for the Design Strips that encompass the pile. In cases where no design strips are defined at the pile location, the program will assume the bar size and cover specified in the first rule defined in the Design Rules spreadsheet.

L1 and L2

Length of the critical section along the local axes. This value is used to define the punching shear perimeter. See the image in the Pile Caps - Design topic for more information.

Note:

- For further discussion refer to ACI 318-14 Sections 22.6.4.1 and 8.4.2.3.1 (ACI 318-11 Sections 11.11.1.2 and 13.5.3).

Concrete Pile Rebar

Phi Used

This value is based on the strain in the tension steel in the pile and code provisions in ACI 318-14 Section 21.2 (ACI 318-11 Section 9.3.2).

Pile Reinforcement

The **Vertical Reinf** and **Shear Reinf** values are reinforcement requirements for the pile considering spacing and minimum reinforcement requirements per the ACI code and the Design Rules - Pedestal/ Pile tab. If these sections report "CUSTOM" it means that the user has applied a custom rebar layout.

Pile Detail Report

To see more detailed information of the pile results, press the ⬚ button on the left side of the screen (when in a graphical view) and click on the node that defines the pile. The pile detail report shows the overall geometry, materials, codes, and analysis/design values used in coming up with the reinforcing and code check values. The report is split into three sections: input echo/graphic view, design results, punching shear checks, and a static soil capacity diagram.

Input Echo/Graphic View

Pile: N6
Shape: CRECT12X12   Tension Bar Fy: 60 ksi   Stress Block: Rectangular
Material: Conc3000NW   Shear Bar Fy: 60 ksi   Design Rule: Typical
Length: 30 ft   Pedestal Cover: 1.5 in   Design Code: ACI 318-14
Pile Embed: 4 in   Rebar Set: ASTM A615
Soil Definition: Default
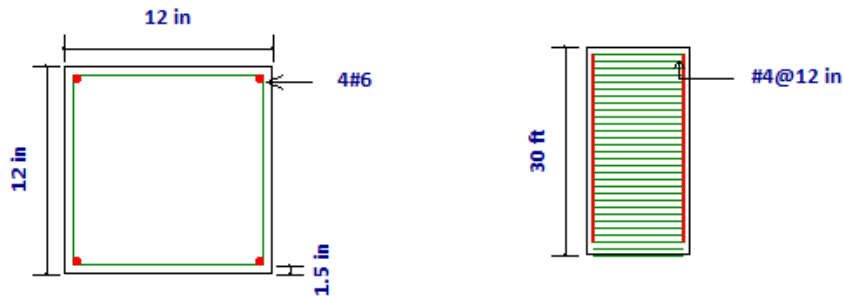
The input echo lists off the point label of the pile, material properties, reinforcement properties and design requirements as applicable. The graphic view gives a plan and section view of the geometry of the pile along with the pile reinforcement designed by the program.

Note:

- The program bases the number and spacing of bars on the requirements per the applicable code regarding spacing and reinforcement limits.
- If a custom rebar layout is used, the program will display the reinforcement as input by the user.

Design Results

Design Results

Pile Results
Pile Comp UC   .974   Pile Tens UC   0
Pile Comp Force   58.44 k   Pile Tens Force   0 k
Gov LC   1   Gov LC   NC
Pile Comp Capacity   60 k   Pile Tens Capacity   0 k

Soil Results
Soil Comp UC   .371   Soil Tens UC   0
Soil Comp Capacity   157.667 k   Soil Tens Capacity   150.783 k
Gov LC   1   Gov LC   NC
Factor of Safety   3
Total Skin Friction   149.333 k
Tip Bearing at Bottom Layer 8.333 k

Shear Check Results
Shear UC   .333
Shear Force   1 k (Z: 0 k, X: -1 k)
Gov LC   1
Shear Capacity   3 k

The **Pile Results** summarizes the code check for the pile axial capacity based on the pile shape and material properties. Compression and Tension UC values give the ratio of demand forces to allowable forces.

The **The Soil Results** show the static pile capacities for compression and tension. This section also gives more detail about the total skin friction and tip bearing values used to calculate the capacities. The factor of safety input by the user in the model settings is also shown in the report.

The **Shear Check Results** section shows the total shear demand and its force components in the Z and X directions. The shear capacity is a user defined value in the Pile Definitions spreadsheet and is compared against the demand for the unity check.

Punching Shear Check Results

Punching Shear Check Results

Punching Shear Values

Unity Check	.392	Vuy	70.128 k
Location	CORNER	$\emptyset$vny	179.067 k
Gov LC	2		

Punching Shear Geometries
Effective depth 18.125 in
L1 along zz   30.063 in
L2 along xx   30.063 in

The **Punching Shear Values** section displays the unity check for the ratio of the shear demand to capacity. The governing LC is the load combination which produces the maximum unity check.

The **Punching Shear Geometries**are used to calculate the punching shear capacity.

For an in depth explanation of the these values, please see the spreadsheet result explanations above at the top of the Pile Results topic.

Axial Capacity Diagram
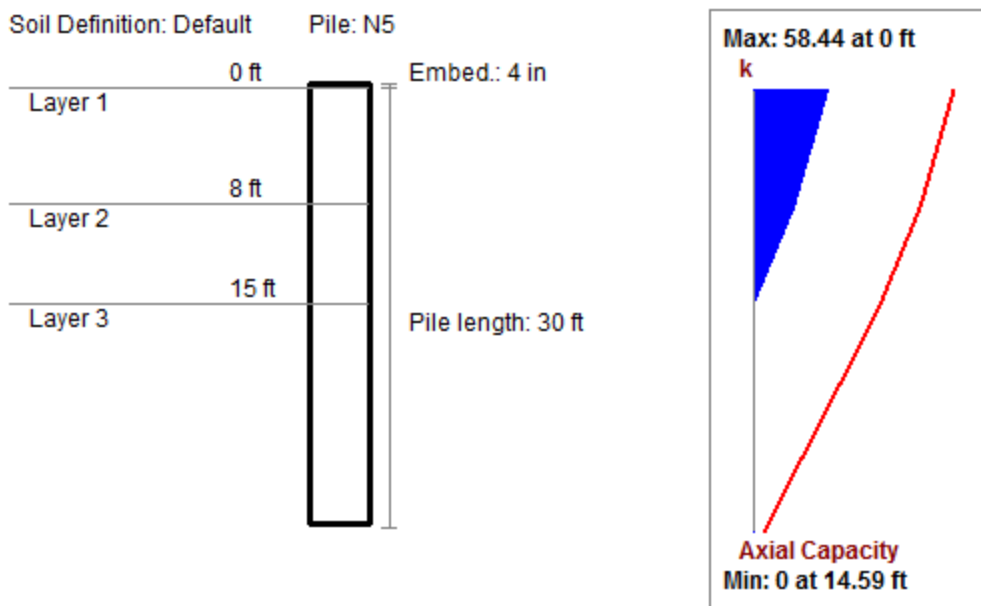
The axial capacity diagram shows the demand force in blue versus the static pile capacity line in red. The capacity line represents the skin friction as a function over the length of the pile. At the bottom of the pile, the tip bearing capacity is reflected by an offset. The factor of safety is considered in both the skin friction and tip bearing for the capacity line.

Pile Cap Results
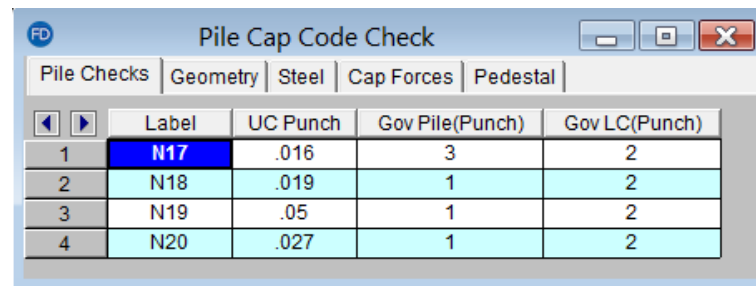
The locations for output for pile caps are the Pile Cap Results browsers and the Pile Cap detail report.

Pile Cap Results Browser

To view the pile cap results browser, click the **Pile Cap Results** from the Results toolbar, or go to Results-Pile Cap Results. These output browsers are meant to summarize the results for the pile cap. To see more detailed information view the detail report by click the Detail Report for Current Item button near the top of the screen when you are in the Pile Cap Results browser. Alternatively, press the Detail button on the left side of the screen (when in a graphical view) and click on the node that defines the pile cap.

Pile Design Checks
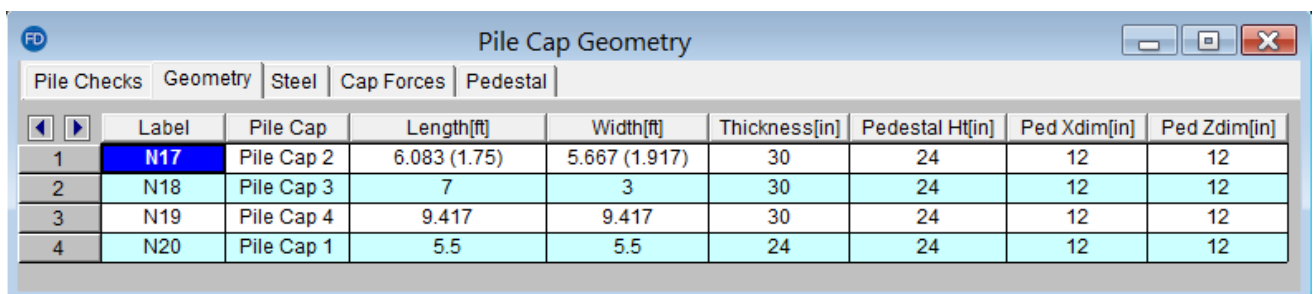
The Pile Design Checks spreadsheet gives the worst case pile for the worst case load combination for the punching shear code check.

The rest of the pile design checks including shear, compression, tension, and punching shear are in the Pile Results spreadsheet. For more information and details on each of these design checks, see the detail report.

Pile Cap Geometry

The Pile Cap Geometry spreadsheet gives dimensions for the pile cap and pedestal. To get a detailed sketch of this information see the detail report.

Note:

- The values in parentheses for the length and width are the dimensions for non-rectangular sides of the cap. See the 3 pile layout for an example of this.

Pile Cap Steel Reinforcement

	Label	Pile Cap	Bot z Dir. Steel[in^2]	Bot x Dir. Steel[in^2]	Top z Dir. Steel[in^2]	Top x Dir. Steel[in^2]	Ped Long	Ped Shear
1	N16	Pile Cap 1	3.093	3.093	0	0	4#6	#4@12 in
2	N17	Pile Cap 2	4.418	4.418	0	0	4#6	#4@12 in
3	N18	Pile Cap 3	3.976	3.976	0	0	4#6	#4@12 in
4	N19	Pile Cap 4	4.86	4.86	0	0	4#6	#4@12 in
5	N20	Pile Cap 5	3.976	6.185	0	0	4#6	#4@12 in
6	N21	Pile Cap 5	3.976	6.185	0	0	4#6	#4@12 in
7	N22	Pile Cap 7	6.185	6.627	0	0	4#6	#4@12 in
8	N23	Pile Cap 8	4.418	4.418	0	0	4#6	#4@12 in

The Pile Cap Steel Reinforcement gives required steel for both the pile cap and pedestal. To get more information on the design of the pile cap reinforcement, see the Pile Cap - Design topic and the Pile Cap Flexural Design portion of the detail report.

Pile Cap Design Forces

	Label	Pile Cap	Muxx[k-ft]	Muxx LC	Muzz[k-ft]	Muzz LC	Vu-p[k]	Vu-p LC	Vu-x[k]	Vu-x LC	Vu-z[k]	Vu-z LC
1	N16	Pile Cap 1	1.952	2	2.102	2	2.277	2	1.131	2	1.251	2
2	N17	Pile Cap 2	3.521	2	3.521	2	3.49	2	1.699	2	1.699	2
3	N18	Pile Cap 3	1.849	2	2.011	2	3.49	2	1.479	2	1.609	2
4	N19	Pile Cap 4	1.746	2	1.918	2	0	2	.815	2	.905	2
5	N20	Pile Cap 5	2.597	2	4.393	2	2.013	2	.025	2	6.43	2
6	N21	Pile Cap 5	2.597	2	4.393	2	2.013	2	.025	2	6.43	2
7	N22	Pile Cap 7	1.076	2	9.505	2	2.013	2	.971	2	1.702	2
8	N23	Pile Cap 8	4.263	2	4.403	2	2.013	2	4.285	2	4.365	2

The Pile Cap Design Forces spreadsheet gives a summary of the forces in the pile cap for the worst case load combinations. The **Muxx** and **Muzz** moments are the worst case moments from the positive and negative faces of the pedestal in both directions. They are based on the local axis of the pile cap, which can be viewed in the Model Display Options dialog. For more information on these design values see the Pile Cap - Design topic and the Pile Cap Flexural Design portion of the detail report. The **Vu-p** is the pedestal punching shear demand. See the Pile Cap - Design topic and the Pedestal Punching Shear portion of the detail report. The **Vu-x** and **Vu-z** shears are the worst case shears from the positive and negative faces of the pedestal in both directions. See the Pile Cap - Design topic and the One Way Shear portion of the detail report.

Pedestal Design Checks

	Label	Pile Cap	UC Bend	UC Bend LC	UC Shear	UC Shear LC	UC Punch	UC Punch LC
1	N16	Pile Cap 1	.007	2	.005	2	.012	2
2	N17	Pile Cap 2	.005	2	.005	2	.012	2
3	N18	Pile Cap 3	.007	2	.005	2	.011	2
4	N19	Pile Cap 4	.007	2	.005	2	.011	2
5	N20	Pile Cap 5	.007	2	.005	2	.014	2
6	N21	Pile Cap 5	.007	2	.005	2	.014	2
7	N22	Pile Cap 7	.007	2	.005	2	.014	2
8	N23	Pile Cap 8	.007	2	.005	2	.014	2

The Pedestal Design Checks section gives a summary of the design checks for the pedestal for the worst case load combination. For more detailed information see the Pedestal - Design and Punching topic and the Pedestal Design portion of the detail report.

Detail Report Information

The Pile Cap Detail Report gives detailed information about geometry, materials, criteria, design checks and sketches associated with this pile cap. Here we will walk through each section of the report.
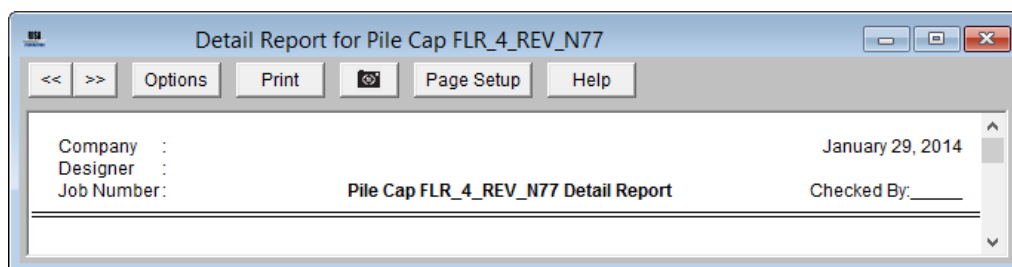
Header

The header of the detail report is useful for navigating and viewing information in the pile cap reports.

- The **double arrows** allow you to click through detail reports for the different pile caps in the model.
- The **Options** button opens up the report options dialog. This allows you to customize exactly what you see in the detail report. There are also options for viewing the Expanded View for some sections of reports. We will talk about what this means for these checks.
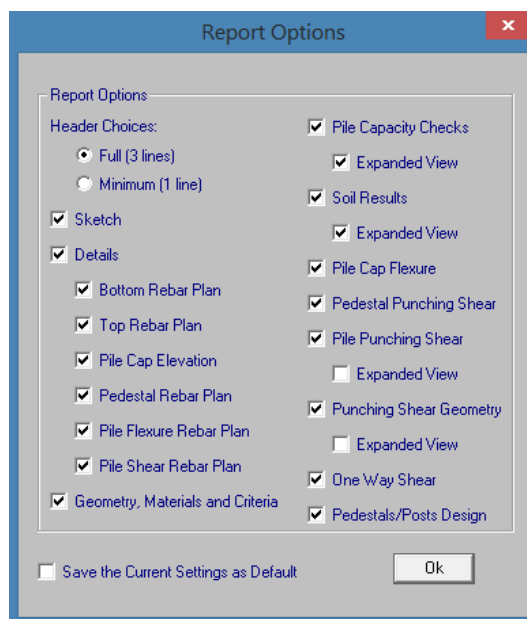
- The **Print** button opens you to the print dialog to print the report.
- - This button will allow you to take a snapshot of the current detail report you are viewing so that it can be added to a report. View the Printing topic for more information.
- The **Page Setup** button allows you to modify page margins.
- The **Help** button opens you to the detail report section in the Pile Cap Results topic.

Sketch

Sketch
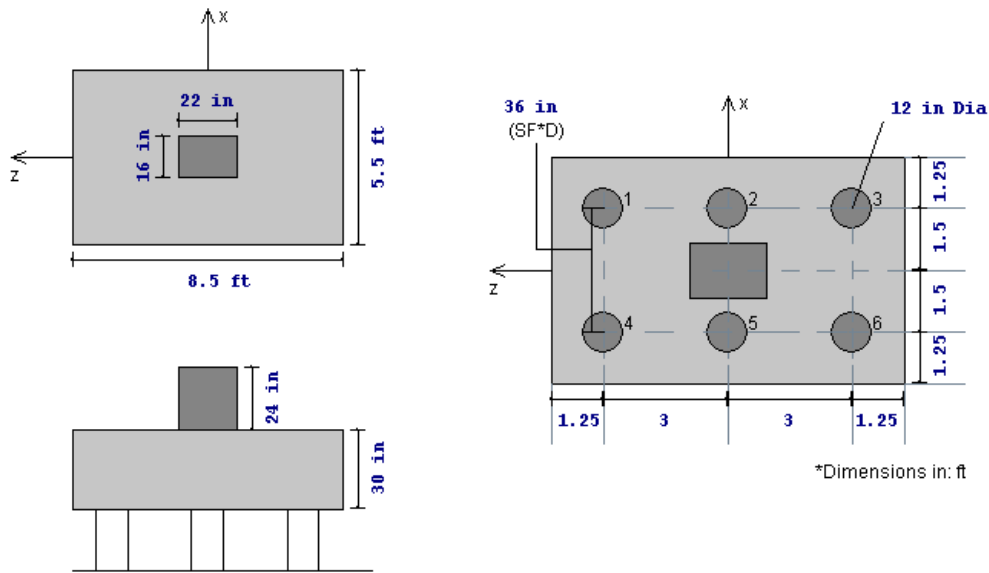
The sketch section gives a full sketch of pedestal and pile locations. The dimensions were found using the CRSI pile layouts, edge distance, pile spacing factor and the pile diameter/side dimension. The minimum distance between piles is defined by the Spacing Factor x Pile Diameter/Side Dimension and is shown on the sketch as well (SF*D).

Reinforcement Details

Details

Pile Flexural Rebar Plan   Pile Shear Rebar Plan
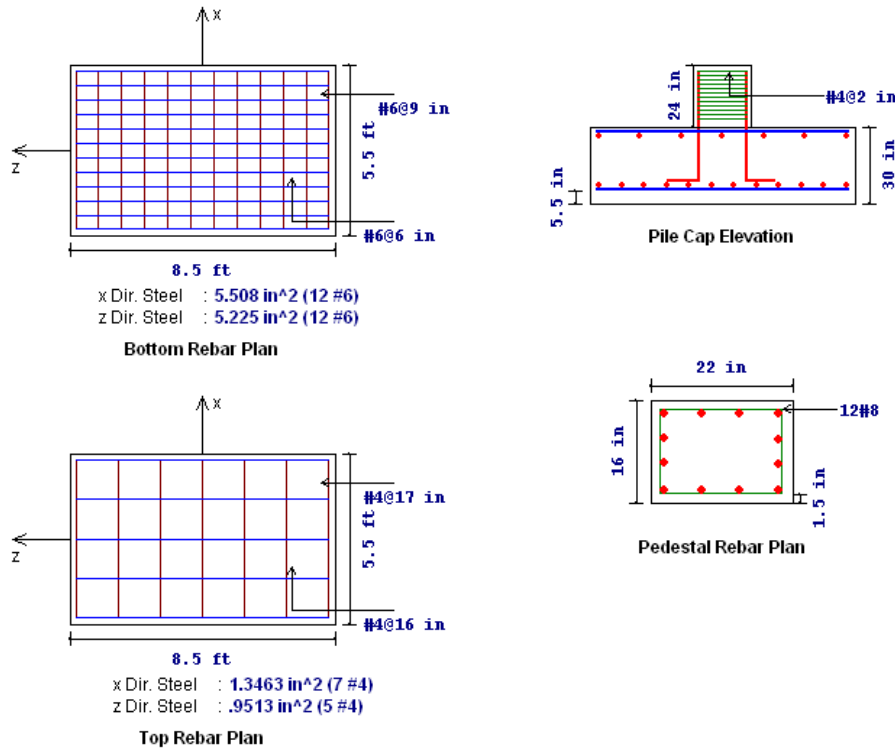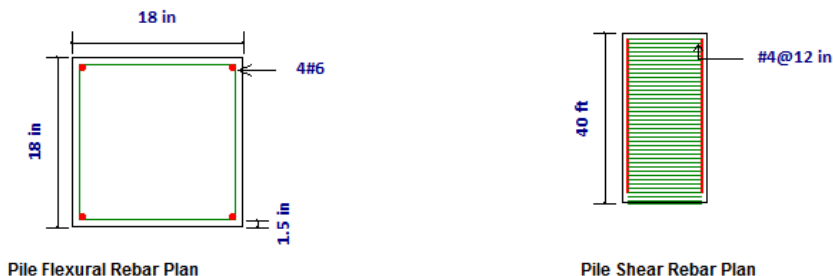
The reinforcement details section lists bar size, spacing and total reinforcement in each direction of the pile cap. If welded wire bars are designed, the output result will be preceded by a "D" instead of "#" to indicate wire bar size. This gives you a to scale visual of the reinforcement in your pile cap, pedestal, and concrete piles. For more information on the required steel calculation see the Pile Cap - Design topic and the Pile Cap Flexural Design portion of the detail report.

Geometry, Materials and Criteria

Geometry, Materials and Criteria

Point	: N5	Concrete Weight	: .145 k/ft^3	Steel fy	: 60 ksi
Length	: 8.5 ft	Concrete f'c	: 3 ksi	Minimum Steel	: .0018
Width	: 5.5 ft	Design Code	: ACI 318-05	Maximum Steel	: .0075
Thickness	: 30 in	Pile Definition	: Pile 1		
Ped. Height	: 24 in				
Ped. X	: 16 in				
Ped. Z	: 22 in				

Cap Top Bar Cover	: 1.5 in	Allow Shear	: 5 k	Rotation Angle	: 0 deg
Cap Bot Bar Cover	: 1.5 in	Allow Axial Comp	: 100 k	Group Red Fact (V)	: 1
Ped Long Bar Cover	: 1.5 in	Allow Axial Tens	: 0 k	Group Red Fact (P)	: 1
Pile Embedment	: 4 in	Phi for Flexure	: .9		
Pile Dia/Side Dim	: 12 in	Phi for Shear	: .75		
Pile Spacing Factor	: 3				
Pile Min Spacing	: 36 in				
Pile Min Edge Dist	: 15 in				

The geometry, materials and criteria section gives input information from the Design Rules, Pile Definitions, Pile Cap Definitions and All Support Types spreadsheets.

Pile Capacity Checks

Pile Capacity Checks

Pile Allow Tens : 0 k   Pile Allow Shear : 5 k
Pile Allow Comp : 100 k

LC	Pile	Shear Force (k)	Shear UC	Axial Comp Force (k)	Axial Comp UC	Axial Tens Force (k)	Axial Tens UC
1	1	5	1	63.776	.6378	0	0
	2	5	1	63.776	.6378	0	0
	3	5	1	63.776	.6378	0	0
	4	5	1	42.1093	.4211	0	0
	5	5	1	42.1093	.4211	0	0
	6	5	1	42.1093	.4211	0	0
2	1	3	.6	40.0312	.4003	0	0
	2	3	.6	40.0312	.4003	0	0
	3	3	.6	40.0312	.4003	0	0
	4	3	.6	27.0312	.2703	0	0
	5	3	.6	27.0312	.2703	0	0
	6	3	.6	27.0312	.2703	0	0

Note: Pile capacity checks are only given for service LCs

Pile Capacity Checks

Pile Allow Tens : 0 k   Pile Allow Shear : 5 k
Pile Allow Comp : 100 k

Shear UC Max	Gov Pile for Shear	Gov Shear LC	Axial Comp UC Max	Gov Pile for Axial Comp	Gov Axial Comp LC	Axial Tens UC Max	Gov Pile for Axial Tens	Gov Axial Tens LC
1	1	1	.6378	1	1	0	NC	NC

Note: See Expanded View for information for each individual pile for each load combination
         Pile capacity checks are only given for Service LC's

The pile capacity checks section has two options: the expanded view (above) and the governing view (below). The expanded view gives forces and code checks in each pile in the pile layout for each service load combination run. The governing view simply gives the maximum code check for the worst case pile for the worst case LC. To see more about how these forces are calculated and how the allowable forces are modified, see the Pile Caps - Design topic.

Soil Results

Soil Results

LC	Pile	Soil Comp Force (k)	Soil Comp UC	Soil Tens Force (k)	Soil Tens UC	Total Skin Friction (k)	Tip Bearing (k)
1	1	24.823	.258	18.86	0	43.53	30.938
	2	24.823	.258	18.86	0	43.53	30.938
	3	24.823	.258	18.86	0	43.53	30.938
	4	24.823	.258	18.86	0	43.53	30.938
	5	24.823	.258	18.86	0	43.53	30.938
3	1	24.823	.258	18.86	0	43.53	30.938
	2	24.823	.258	18.86	0	43.53	30.938
	3	24.823	.258	18.86	0	43.53	30.938
	4	24.823	.258	18.86	0	43.53	30.938
	5	24.823	.258	18.86	0	43.53	30.938

Note: Pile capacity checks are only given for service LCs

Soil Results

Soil Comp UC Max	Gov Pile for Soil Comp	Gov Soil Comp LC	Soil Tens UC Max	Gov Pile for Soil Tens	Gov Soil Tens LC	Total Skin Friction (k)	Tip Bearing (k)
.258	1	1	.258	NC	NC	43.53	30.938

Note: See Expanded View for information for each individual pile for each load combination
Pile capacity checks are only given for Service LC's

The pile soil results section has two options: the expanded view (above) and the governing view (below). The expanded view gives forces and code checks in each pile in the pile layout for each service load combination run. The governing view simply gives the maximum code check for the worst case pile for the worst case LC.

Pile Cap Flexural Design

Pile Cap Flexural Design

As-min x-dir (Top Flexure): **9.52 in^2**   As-min x-dir (T & S): **5.508 in^2**
As-min z-dir (Top Flexure): **6.16 in^2**   As-min z-dir (T & S): **3.564 in^2**
As-min x-dir (Bot Flexure): **8.075 in^2**
As-min z-dir (Bot Flexure): **5.225 in^2**

Bottom Bar Reinforcement

Mu-xx UC Max	Gov Mu-xx LC	Mu-xx (k-ft)	z-Dir As Required (in^2)	z-Dir As Provided (in^2)	Mu-zz UC Max	Gov Mu-zz LC	Mu-zz (k-ft)	x-Dir As Required (in^2)	x-Dir As Provided (in^2)
.7554	3	293.0881	2.8012	3.735	.373	3	213.7205	2.0195	5.508

Top Bar Reinforcement

Mu-xx UC Max	Gov Mu-xx LC	Mu-xx (k-ft)	z-Dir As Required (in^2)	z-Dir As Provided (in^2)	Mu-zz UC Max	Gov Mu-zz LC	Mu-zz (k-ft)	x-Dir As Required (in^2)	x-Dir As Provided (in^2)
0	NC	0	0	0	0	NC	0	0	0

The pile cap flexural design section starts by giving minimum steel requirements for both flexure and shrinkage and temperature. Then there is a bottom and top bar reinforcement section where the max. moments at the face of the pedestal is given in both orthogonal directions. The Muxx moment (moment causing bending about the x-axis) is the maximum moment computed considering all strength load combinations at both the positive and negative z faces of the pedestal. Top reinforcement is only added if there is negative bending in the pile cap. For more information on reinforcement calculations see the Pile Caps - Design topic.

Pedestal Punching Shear Check

Pedestal Punching Shear Check

Phi : .75

Gov LC	UC	Punching Shear Location	Punching Shear (k)	Muz (k-ft)	Mux (k-ft)	Total Stress (ksi)	Phi * Vny (ksi)
3	.482	INTERIOR	279.2819	136.5	0	.0792	.1643

The pedestal punching shear section gives the demand punching shear and moments at the base of the pedestal. These forces are then converted into an equivalent total stress and this stress is reported.

Phi*Vny represents the shear stress capacity of the slab calculated based on the three ACI equations for two way or punching shear capacity of the slab. This value does NOT include any additional capacity due to shear reinforcing in the slab. For information on the calculation of the punching shear demand, see the Pile Caps - Design topic. For information regarding geometry see the Punching Shear Geometry section of the detail report.

Pile Punching Shear Check

Pile Punching Shear Check

Phi : 0.55

LC	Pile	Punching Shear UC	Punching Shear Location	Punching Shear Cap (k)	Punching Shear (k)
3	1	.6347	CORNER	139.755	88.6978
	2	.4117	EDGE	215.4265	88.6978
	3	.6347	CORNER	139.755	88.6978
	4	.4176	CORNER	139.755	58.3645
	5	.2709	EDGE	215.4265	58.3645
	6	.4176	CORNER	139.755	58.3645
4	1	.3342	CORNER	139.755	46.703
	2	.2168	EDGE	215.4265	46.703
	3	.3342	CORNER	139.755	46.703
	4	.2257	CORNER	139.755	31.5363
	5	.1464	EDGE	215.4265	31.5363
	6	.2257	CORNER	139.755	31.5363

Note: Punching shear checks are only given for strength LCs

Pile Punching Shear Check

Phi : 0.55

Gov LC	Gov Pile	Punching Shear Location	Punching Shear Cap (k)	Punching Shear (k)	Punching Shear UC
3	1	CORNER	139.755	88.6978	.6347

Note: See Expanded View for information for each individual pile for each load combination
Punching shear checks are only given for Strength LC's

The pile punching shear checks section has two options: the expanded view (above) and the governing view (below). The expanded view gives forces and code checks in each pile in the pile layout for each strength load combination run. The governing view simply gives the maximum code check for the worst case pile for the worst case LC. To see more about how these forces are calculated and how the allowable forces are modified, see the Pile Caps - Design topic.

Punching Shear Geometry

Punching Shear Geometry

Pedestal Shape : CRECT16X22   Pile Shape   : Round/12

Pile/Pedestal	Location	Effective Depth (in)	L1 (in)	L2 (in)	Polar Moment-xx (in^4)	Polar Moment-zz (in^4)	Gamma-x	Gamma-z
Pedestal	INTERIOR	23.75	45.75	39.75	1.469e+6	1.196e+6	.417	.3833
1	CORNER	26	32.7124	32.7124	0	0	0	0
2	EDGE	26	35.4248	32.7124	0	0	0	0
3	CORNER	26	32.7124	32.7124	0	0	0	0
4	CORNER	26	32.7124	32.7124	0	0	0	0
5	EDGE	26	35.4248	32.7124	0	0	0	0
6	CORNER	26	32.7124	32.7124	0	0	0	0

Note: Punching shear checks are for the pedestal or for individual piles only.
Group punching shear checks are not considered at this time.

Punching Shear Geometry

Pedestal Shape : CRECT16X22   Pile Shape   : Round/12

Gov Pile/Pedestal	Location	Effective Depth (in)	L1 (in)	L2 (in)	Polar Moment-xx (in^4)	Polar Moment-zz (in^4)	Gamma-x	Gamma-z
Pedestal	INTERIOR	23.75	45.75	39.75	1.469e+6	1.196e+6	.417	.3833
1	CORNER	26	32.7124	32.7124	0	0	0	0

Note: See Expanded View for information for each individual pile for each load combination
Punching shear checks are for the pedestal or for individual piles only.
Group punching shear checks are not considered at this time.

The punching shear geometry section has two options: the expanded view (above) and the governing view (below). The expanded view gives geometry information for every pile. The governing view simply gives the geometry information for the worst case pile.

Location

The pedestal and slab geometry is investigated and checked for all three types of punching failure (Interior, Edge and Corner). This field represents the type of failure method that controlled the code check calculation for the given pedestal.

L1 and L2

Length of the critical section along the local axes. This value is used to define the punching shear perimeter as well as to define how much of the moment is transferred by eccentricity of shear (as opposed to the portion transferred by flexure).

Note:

- For further discussion refer to ACI 318-18 sections 11.12.1.2 and 13.5.3

Polar Moment - xx and Polar Moment - zz

This is a property (Jc) of the critical section which is analogous to polar moment of inertia about the local xx axes. This value is used to determine the shear stress in the critical section which is due to unbalanced moment.

Note:

- For further discussion refer to ACI 318-14 Sections 22.6.4.1 and 8.4.2.3.1 (ACI 318-11 Sections 11.11.1.2 and 13.5.3).

Gamma x and Gamma z

This is calculated from the dimensions of the assumed punching shear perimeter. It represents the fraction of the unbalanced moment for each axis that is transferred by eccentricity of shear.

Note:

- For further discussion refer to ACI 318-14 Sections 8.4.4.2 and 8.4.2.3.1 (ACI Sections 11.11.7.1 and 13.5.3).

To see more about how geometry is considered, see the Pile Caps - Design topic.

One Way Shear Check

The one way shear check section gives a results summary for beam shear across the pile cap. The **x-Dir Cut Vu** displays the worst case shear force at the positive and negative x sides of the pedestal for all strength LCs. The **Vc (x-Dir Cut)** and **Vc (z-Dir Cut)** gives the shear capacity about both local axis directions. The w, d, Vu and Mu-xx and Mu-zz values are given to verify the calculation of shear capacity. For more information on these calculations, see the Pile Cap - Design topic.

Pedestal Design Checks

The pedestal design section gives design check results for both shear and combined bending and axial forces. See the Pedestals-Design Results for more information on these checks.

Points

Points are used to define the basic geometry of the structure or model. This includes slabs, beams, footings, piles and pile caps. Points are also used to define load locations such as point loads, line loads, and area loads. Each Point is a point in space defined by coordinates in the horizontal and vertical directions or axes.

Points are created automatically as you draw, but may be created manually from within the **Point Coordinates Spreadsheet**. Once created, either automatically or manually, the points may be edited within this spreadsheet or by double-clicking the point.

To Define Points

- Select the **Point Coordinates Spreadsheet** from the **Spreadsheets Menu** and define the point coordinates.

Note

- You may use cut and paste, block fill, and block math to enter and edit points.
- You may choose the prefix that is used to label the points.

Point Coordinates Spreadsheet

The **Point Coordinates Spreadsheet** records the labels and coordinates for the points and may be accessed by selecting **Points** on the **Spreadsheets Menu**.

	Label	Z [ft]	X [ft]
1	N1	0	0
2	N2	0	10
3	N3	19.333	10
4	N4	19.333	0
5	N5	.333	4.583
6	N6	0.414663	5.412134
7	N7	0.656512	6.209405
8	N8	1.049254	6.944173
9	N9	1.577796	7.588204
10	N10	2.221827	8.116746
11	N11	2.956595	8.509488
12	N12	3.753866	8.751337

Use the first column to assign a **Label** for each point. You can then refer to each point by its label. Each label has to be unique. If you try to enter the same label more than once you will receive an error message. As you create new lines, the program will automatically create a new, unique label for each point.

The next two columns contain the coordinates of the point in the **Z** and **X** global directions. These represent the offsets of the points from the origin of the coordinate system (0, 0). The appropriate units are listed at the top of each column.

Point Information Dialog

You may double-click any point to view it's properties. All of the same information that is stored in the **Points** spreadsheet is displayed for the point you choose, and may be edited. This is a quick way to view and change point properties. For large selections of points however the spreadsheet may be the faster solution.

Label - You can view and edit the point's label.

Coordinates - You can view and edit the point coordinates.

Boundary Conditions - Here you can view and edit the point boundary conditions. The edit boxes to the right of the boundary condition list boxes are where you would enter additional data for a boundary condition such as a spring stiffness, a Story number, or a Primary joint label for a Tethered joint.
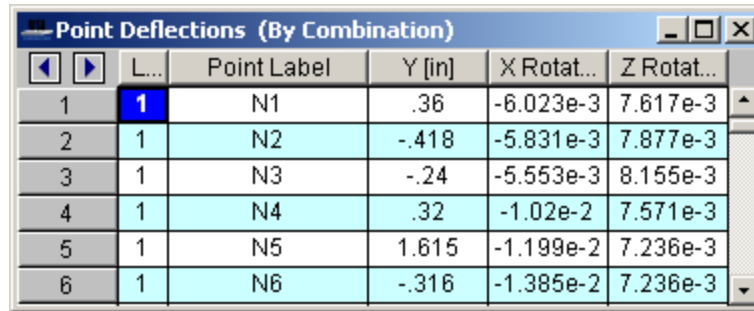
Support Type - This allows you to define a footing, pile, or pile cap at this point.

Points - Results

When the model is solved, there are several groups of results specifically for the points.

Point Deflections Results

Access the **Point Deflections Spreadsheet** by selecting it from the **Results Menu**.

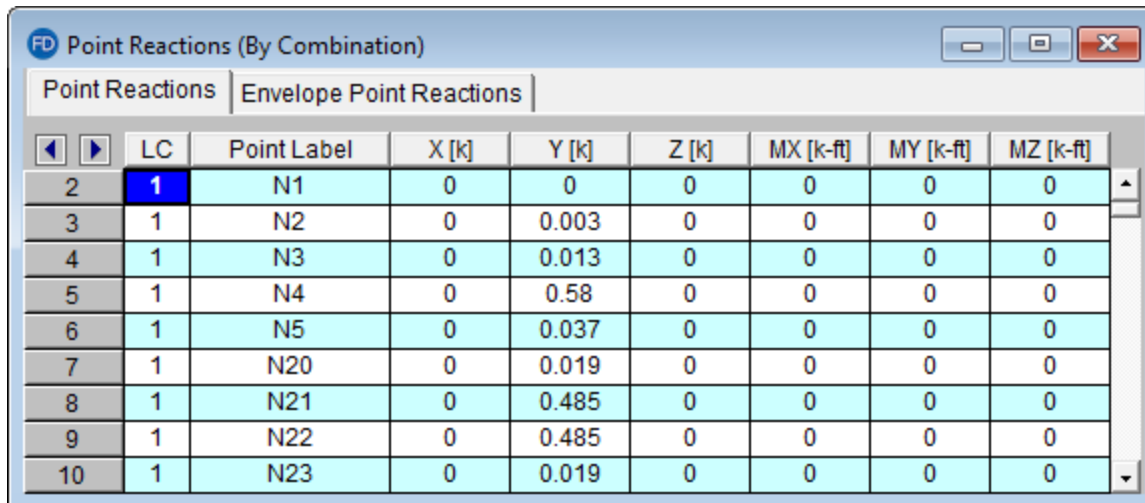|	L...	Point Label	Y [in]	X Rotat...	Z Rotat...
1	1	N1	.36	-6.023e-3	7.617e-3
2	1	N2	-.418	-5.831e-3	7.877e-3
3	1	N3	-.24	-5.553e-3	8.155e-3
4	1	N4	.32	-1.02e-2	7.571e-3
5	1	N5	1.615	-1.199e-2	7.236e-3
6	1	N6	-.316	-1.385e-2	7.236e-3

These are the deflections, or displacements, for every point in the structure. Each point has 3 values calculated, 1 for each of the 3 global degrees of freedom. Those being 1 translations and 2 rotations. The units for the displacements are shown at the top of each column. The rotations are shown in units of radians (1 radian = 57.3 degrees, approximately).

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort**, and other options.
- See Model Display Options - Points/Footings/Retaining Walls to learn how to plot point results.

Point Reaction Results

Access the **Point Reactions Spreadsheet** by selecting it from the **Results Menu**.

Point Reactions | Envelope Point Reactions

	LC	Point Label	X [k]	Y [k]	Z [k]	MX [k-ft]	MY [k-ft]	MZ [k-ft]
2	1	N1	0	0	0	0	0	0
3	1	N2	0	0.003	0	0	0	0
4	1	N3	0	0.013	0	0	0	0
5	1	N4	0	0.58	0	0	0	0
6	1	N5	0	0.037	0	0	0	0
7	1	N20	0	0.019	0	0	0	0
8	1	N21	0	0.485	0	0	0	0
9	1	N22	0	0.485	0	0	0	0
10	1	N23	0	0.019	0	0	0	0

The Point Reactions are grouped into two tabs, you can view the results by load combination or by envelope. These are the reactive forces **applied to the foundation** at its points of support. A positive reaction is applied in the upward vertical direction and a negative reaction is applied in the downward vertical direction. A positive moment is given according to the right hand rule assuming the thumb is pointing in the positive global axis direction. Assuming a reaction has been calculated at all points of support, the total of the reactive forces in each direction should equal the total applied force in each direction. The units for the reactions are listed at the top of each column, and a total reaction for each direction is summed at the bottom of each translation column.

Retaining walls will also show up in this dialog. The total self-weight of the retaining wall and the total of any externally applied loads to the wall will be calculated here. Keep in mind that there are soil loadings implicit in the design of the wall as well, but those are not shown here.

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort**, and other options.
- See Model Display Options - Points/Supports/Subgrades to learn how to plot point results.

Printing

You may print graphics as they appear on the screen or print all (or part) of the tabulated model information. If a spreadsheet is currently active and you click the print 🖨 button the following options are presented. If a model view is currently active, and you select the print button, [Graphic Print Options](#) are presented.
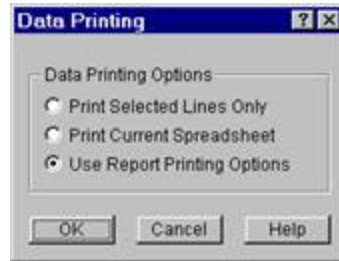
If you have lines selected in the spreadsheet then you may print only those lines by selecting the first option. If you wish to print the entire spreadsheet you may choose the second option. The last option allows you to print multiple spreadsheets in a custom report that is discussed in the next section. You may combine the last two options with the Exclude feature to hide some data and print only the data you want.

To Print a Report or a Spreadsheet

- While in a spreadsheet click on the **Print** 🖨 button and choose to print the current spreadsheet, a selected portion of the spreadsheet, or multiple sections by printing a report. If you are in a model view the Graphic Print Options are opened so click 'Print a Report Instead'.

To Print Graphics

- While in a model view click on the **Print** 🖨 button and choose from the options.

Note

- If you are not in a model view the Data Printing settings are opened instead. Close this and click on the model view that you want to print before clicking the **Print** 🖨 button.

Printing Reports
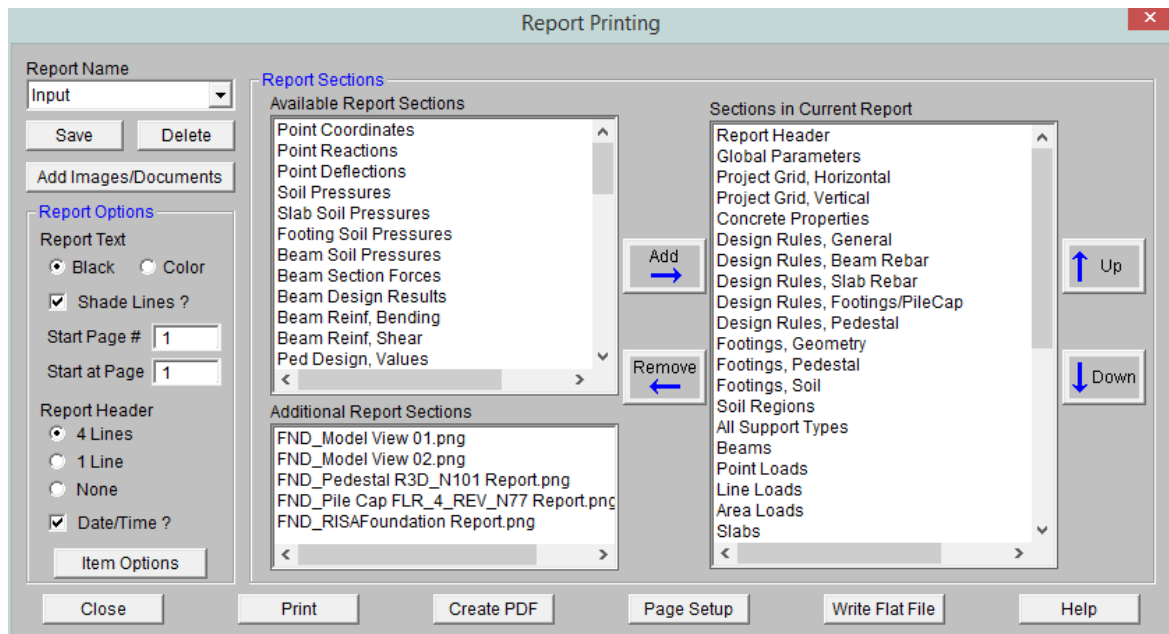
The **Report Printing Dialog** options help you build your reports. There are standard reports for you to choose from and you may also name and save any report format you custom build. To choose a standard report, simply pick it from the **Report Name** drop down list.

Building a Report

To build your own custom report you may double-click on report sections in order to move them from the available sections on the left, to the report defined on the right. You may use the mouse and the SHIFT or CTRL keys to pick multiple sections and then move them with the **Add** button. You can re-order the report by dragging and dropping the sections in your report you would like to movewith the **Up** and **Down** buttons and remove sections with the **Remove** button.

The **Sections in Current Report** portion of the dialog shows which sections and the order the printed report will be built. Report sections can come from either the **Available Report Sections** or the **Additional Report Sections**. The **Available Report Sections** contains every input and output spreadsheet in the program. The **Additional Report Sections** contains all images that were added to the project. Images can be added either using the "Snapshot" option or by adding external images using the **Add Images/Documents** button.

Note:

- Any saved reports from the Report Names drop-down will currently only save spreadsheet information. Any snapshots or images will manually need to be added to any saved reports.
- The program will flag solution spreadsheets as to whether there is information available in them. If that type of solution has not been run you will see a prefix of **NA** ahead of that spreadsheet in the **Available Report Sections**.
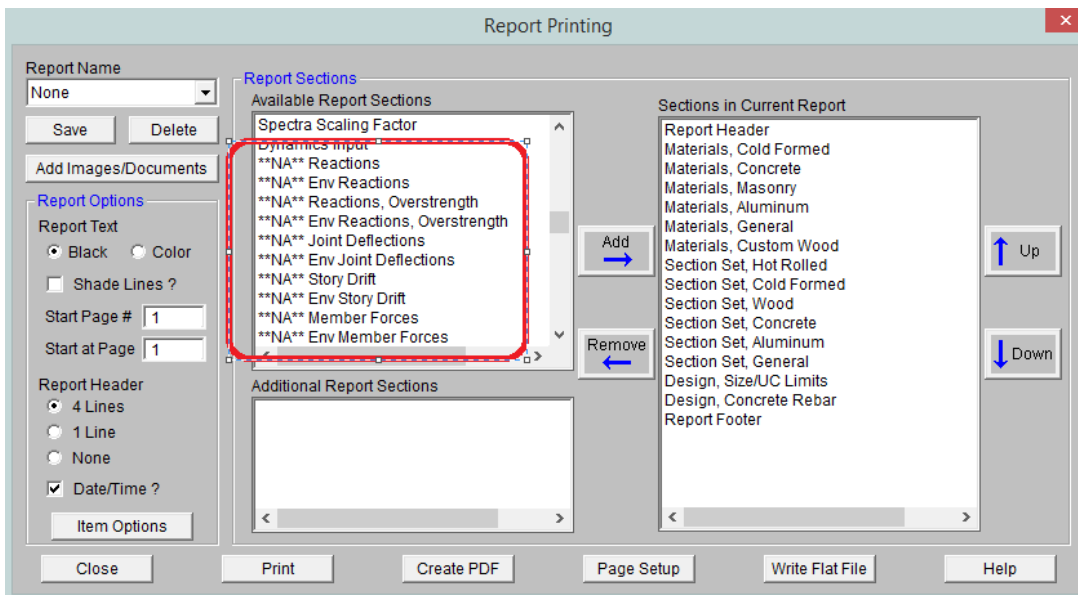
Snapshot Images

A snapshot image button is on the main graphic toolbar and most of the detail report windows.
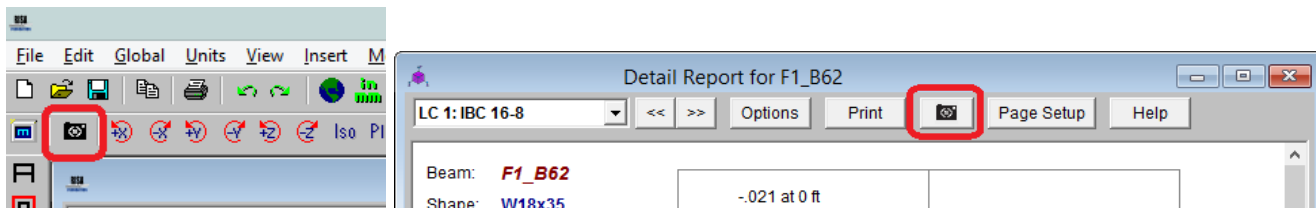
When this button is pressed a folder will be automatically created in the same location where the model file is located. This button will then give some options regarding the print location and file name. Once you define this information press Save and the program will create this image.

Note:

- If the model has yet to be solved the image will be created in the **Model Data Files** location from **Tools - Application Settings - File Locations**.
- The relative path of these image files will be saved with the model. If the model file is moved, move the associated **Images** folder as well and then all images will still be available.

This snapshot is a great way to get images of model views and detail reports that you want to add to your report. Once a snapshot is taken it will show up in the **Additional Report Sections** in the **Report Printing** dialog and can then be added to the **Sections in CurrentReport**.
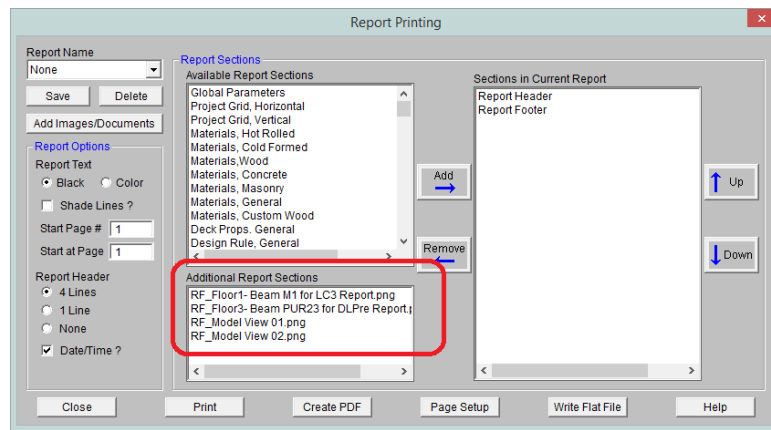
Adding Images/Documents

Any additional images, documents that you wish to add to your report that are not from RISA, can be added using the **Add Images/Documents** button. Press this button and navigate to any PNG, PDF, or JPEG file and you can select it. Once that happens it will then show up in the **Additional Report Sections** and can then be added to the **Sections in CurrentReport**.

Note:

- The complete path of these additional images are saved with the file. If the model file is moved it will still find these additional images.

Report Options

Formatting options allow you to specify many options:

Report Text - Allows you to define the text color as black or blue.

Shade Lines? - Allows you to shade every other line to enhance readability.

Start Page # - Allows you select the starting page number. The number shown will be the next page number in the current sequence, but you can override this for occasions where you need to insert your calculation pages into an existing report and you need the page numbering to match.

Start at Page - Allows you to start your numbering at some page other than page one. This can be helpful if you have a title page or table of contents that you don't want to number.

Report Header - All reports have a footer with version information, the file name and path, and the page number. The single line header option will include the Model Title specified in the **Model Settings** along with the date. The triple line header adds company, designer, job number, and a place to initial any checking to the header.

Date/Time? - Allows you to turn off the Date/Time stamp so that this will not appear on your printed reports.

Item Options - Allows you to select beam related options. You may specify that you want the beam results to be listed for each beam section (specified in the **Model Settings**) or just for the beam ends.

Note:

- See the Customizing RISAFoundation topic for more information about printing, including the ability to print out a RISA or custom company logo.

Printing to a File

A flat file is a file without column headings, print formatting, or graphical elements and is useful for importing and parsing into spreadsheets, database tables, or as post processor input data.
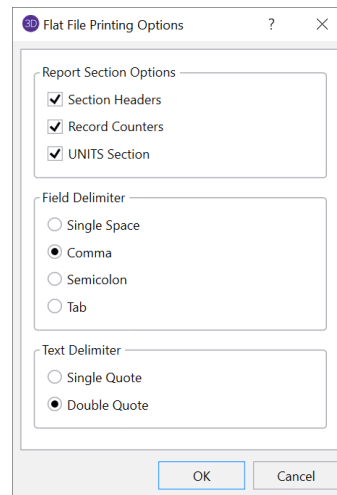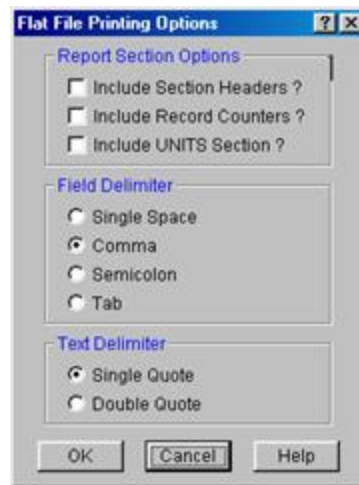
There are several options available to make the flat file easier to parse. Note that printing and then looking at a sample output file with all the options selected will make it easier to understand what the options do.

You have an option to include the 'Section Headers' which will print a text description of each block of data. For example, the Point Coordinates data would be preceded by a [POINT COORDINATES] header on it's own line before the data. You may also include a 'Record Count' which is useful if you're writing looping code to read in the number of records within each data block. The number of records prints on it's own line. You can also have a special 'UNITS Section' be printed out which will give you the units used for all the values in the program.

The field delimiter options let you choose what character will be used to separate each field of data within record. For example, each coordinate value in the Point Coordinates data would be separated by the character selected. Programs that you might be importing the file into like MS Excel or MS Access often have options to select what the field delimiter will be for records of data.

The text delimiter works like the field delimiter, except that it's used to set apart text labels. All text in the flat file will be enclosed at the beginning and the end by the selected text delimiter character. This is very useful when trying to read in label strings that contain embedded spaces. As an example, the Point labels in the Point Coordinates data would each be enclosed by a single or double quote.

The current flat file options are saved each time the program is closed.

To Print to a Flat File

- While in the a spreadsheet click on the **Print** 🖨 button, Choose **Use Report Printing Options** and click **OK**, then choose **Write Flat File**.

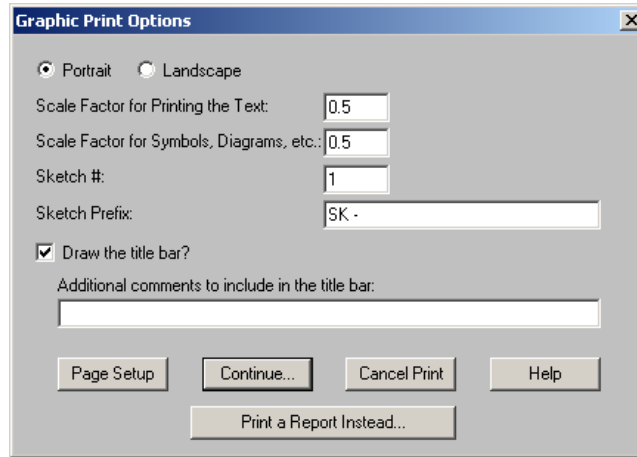# Graphics Printing

Many of the same options that exist in **Report Printing** are present in **Graphic Printing**, but some unique options available are **Scale Factors** and **Title Block Info**.

These options control the graphics printing. You may choose to print in **Portrait** or **Landscape** mode.

Next, two **Scale Factors** are defined. These scale factors are used to make the text and symbols displayed as part of the graphic, larger or smaller. A higher scale factor makes the text or symbol bigger. Since the resolution of the printer is probably much greater than the resolution of your screen, you can probably make the text and symbols smaller (by using a scale factor of less than 1.) than they appear on screen and they will still be easily readable. This makes for a cleaner looking graphic print. As far as what scale factors you should use, the only way to be sure is to experiment a little.

You may include the **Title Bar** to display the Model Title, Designer, Company Name, Date and Time (All of which are defined in your **Project Info**). You can also enter a comment that will be included in the Title Bar. **Page Setup** allows you to set the margins.

You can specify a **Sketch #** and a **Sketch Prefix** to label your graphic prints as well.

Once you have everything the way you want it choose **Continue**. This will bring you to the print settings that are specific to your printer. The choices are different for each printer but generally allow you to choose the printer, orientation, quality and quantity.

The bottom button **Print a Report Instead** is provided so you can get to the report printing dialog directly from a graphic view.

Punching Shear - Design

Punching shear calculations are provided for columns/pedestals that land on concrete slabs. Here is some information on those calculations.

Punching Shear Basics

Punching shear is a phenomenon where a concentrated force on a slab causes a shear failure cone that "punches" through.
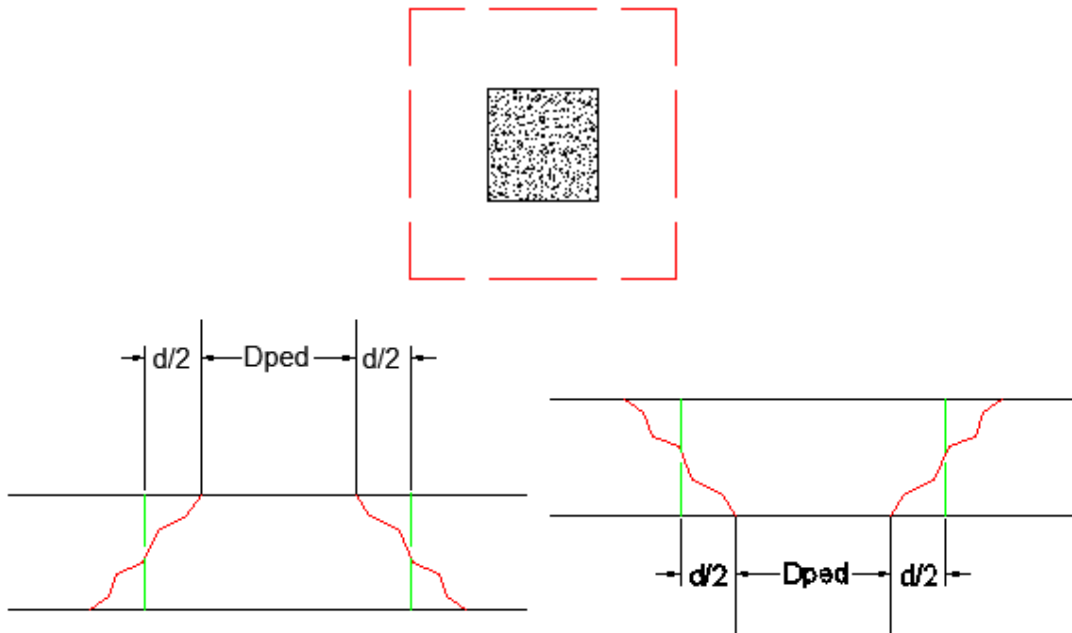
Punching shear calculations require a demand value and a capacity value. Many hand calculations will consider a punching shear force against a punching shear capacity force. This works if only shear force is present, but is insufficient if a moment is occurring on the column/pedestal. Thus, a stress demand/stress approach is taken. The applied shears and moments, along with the shape of the punching shear failure cone are used to calculate a maximum punching shear stress.

Punching Shear Calculation Values

Effective Depth, d

The effective depth of the slab is used to define the punching shear perimeter at a distance of d/2 beyond the edge of the column/pedestal.

The effective depth is based on the smallest depth to centroid of reinforcing for the Design Strips that encompass the pedestal. In cases where no design strips are defined at the pedestal location, the program will assume the bar size and cover specified in the first rule defined in the Design Rules spreadsheet.

L1 and L2

Length of the critical section along the local axes. This value is used to define the punching shear perimeter as well as to define how much of the moment is transferred by eccentricity of shear (as opposed to the portion transferred by flexure).
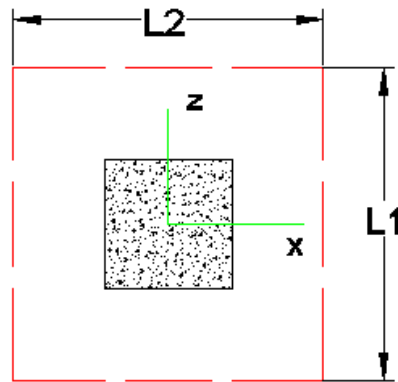
Note:

- For further discussion refer to ACI section 11.12.1.2 and 13.5.3

Polar Moment Ixx and Polar Moment Izz

This is a property (Jc) of the critical section which is analogous to polar moment of inertia about the local yy axes. This value is used to determine the shear stress in the critical location due to unbalanced moment.

Note:

- This calculation is very different depending on interior, edge or corner punching shear cases.
- For further discussion and equations refer ACI 318-14 Section 48.4.4.2.3 (ACI 318-11 Section R11.11.7.2) and Chapter 13-8 of the Reinforced Concrete Mechanics and Design textbook by MacGregor & Wight.

Centroid Location

These give the distance from the shear perimeter edge to the centroid of the shear perimeter. For an interior punching shear perimeter this will always be L1/2 or L2/2. For edge and corner cases this value is calculated as the moment of area of the shear perimeter/area of the sides.
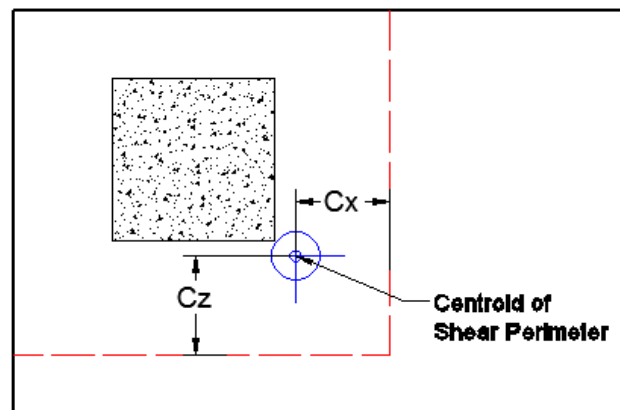
Note:

- For further discussion and equations refer to Chapter 13-8 of the Reinforced Concrete Mechanics and Design textbook by MacGregor & Wight.

Gamma xx and Gamma zz

This is calculated from the dimensions of the assumed punching shear perimeter. It represents the fraction of the unbalanced moment for each axis that is transferred by eccentricity of shear.

Note:

- For further discussion refer to ACI 318-14 Sections 8.4.4.2.2 and 8.4.2.3 (ACI 318-11 sections 11.11.7.1 and 13.5.3).

Total Stress

The Total Stress is the direct shear stress plus the shear stress due to moment. This calculation uses the geometry of the punching shear failure cone along with the induced shear and moment forces to come up with a maximum stress. This will always occur at one of the failure cone corner locations. For an example calculation see the ACI 318-14 Section R8.4.4.2.3 (ACI 318-11 Section R11.11.7.2).

Note:

- See the Punching Shear Loading section for more information.

*Phi*Vny*

This represents the shear ***stress*** capacity of the slab calculated based on the three ACI equations for two way or punching shear capacity of the slab. Refer to ACI 318-14 Section 22.6 (ACI 318-11 Section 11.11.7.2) for more information. When ACI 318-19 is selected, the shear strength of concrete (Vc) uses equations in Table 22.6.5.2. Note that an additional size effect factor λs is added in the equation, λs is a function of effective depth of the slab d, and is calculated per Equation (22.5.5.1.3).

Note:

- This value does NOT include any additional capacity due to shear reinforcing in the slab.

Minimum Slab Reinforcement at Punching Shear Critical Section

ACI 318-19 requires a minimum slab reinforcement at punching shear critical section if $vuv > \varphi 2\lambda\varphi\sqrt{f'c}$ per Section 8.6.1.2. The As,min is calculated and presented in the punching shear result of columns under this condition. Please note that the reinforcement design in elevated slabs does NOT consider this requirement, users need to consider separately the required additional slab reinforcement at punching shear critical section to account for this code requirement. RISA provides the calculations of As,min per Equation (8.6.1.2) in the Detail Report. Please refer to Elevated Slabs - Punching Shear Results for additional information.

Punching Shear Loading

Punching shear loading includes a direct shear component, as well as biaxial moments. It is these combined forces that are used to determine the punching shear stress and code check.

Direct Punching Shear

The direct punching shear demand is not exactly just the axial force in the Pedestal/Post. The area within the punching shear perimeter loading is assumed to pass directly into the support, so that force must be discounted.

This includes soil pressure resistance that occurs within the punching shear perimeter.
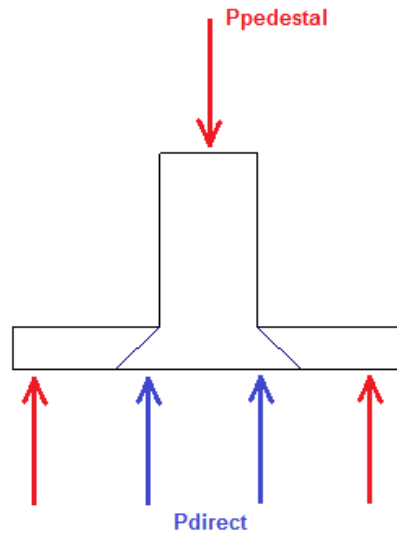
For the image above the Vu demand equals $P_{pedestal}$ - P_{direct}

Moment Consideration

The moments used are also not necessarily just the moments directly from the Pedestal/Post. If the centroid of punching shear area is not the same as the centerline of the Pedestal/Post, then there will be an eccentricity that also adds/subtracts moment.

Take the edge punching shear situation shown below:
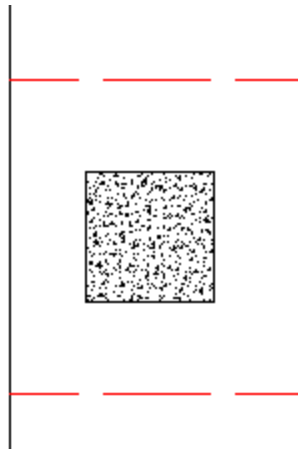
The punching shear perimeter is to the right of the center of Pedestal/Post location. This causes a moment in the punching shear calculation even if a moment is not explicitly applied on the Pedestal/Post. This eccentricity is automatically accounted for in the punching shear stress calculation.
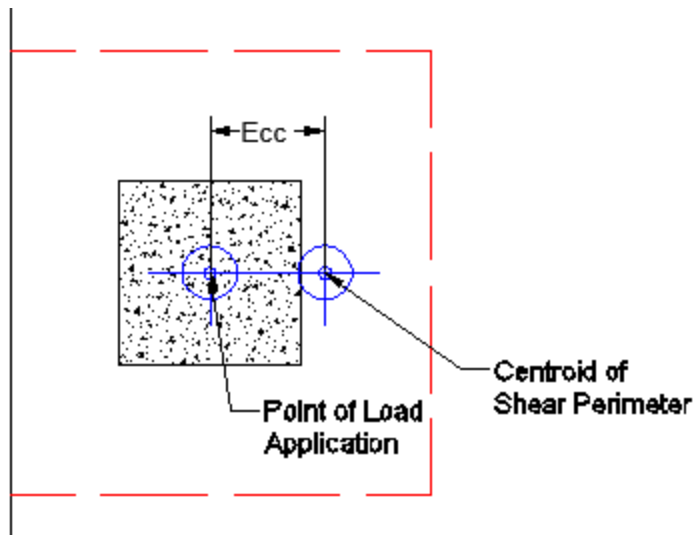
This same inherent eccentricity occurs in corner punching shear conditions. In this case, however, you get an induced moment in both local axes directions because of the centroid of shear perimeter location. This, similar to the edge condition, is automatically considered in the calculation of the maximum shear stress.
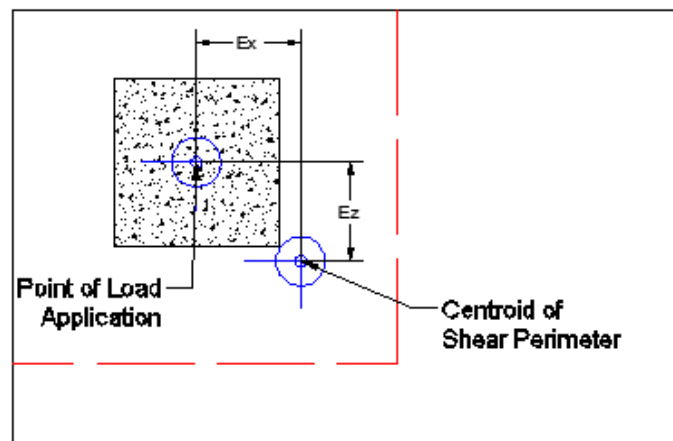
Note:

- The moments due to eccentricity are not accounted for in the detail report moment output. The moments shown there are taken directly from the Pedestal/Post.

Finding the Controlling Case (Interior, Edge or Corner)

The program considers as many as nine different punching shear perimeters for each column in the model. different scenarios. We look at the interior, four edges, and four corner cases and determine which configuration produces the largest code check and present that in the results.

For example, let's consider a Pedestal/Post that has an interior punching shear perimeter close to a corner.
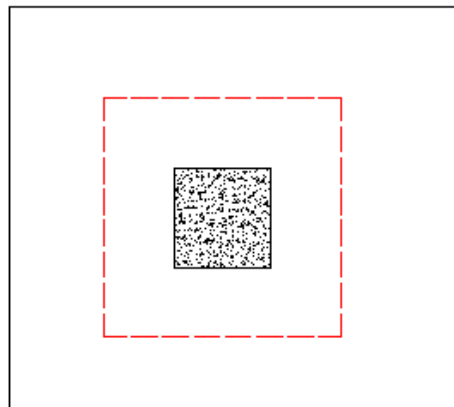
Here the program will also check out any EDGE punching shear perimeter considerations. The two below would be the most likely ones for this given geometry.
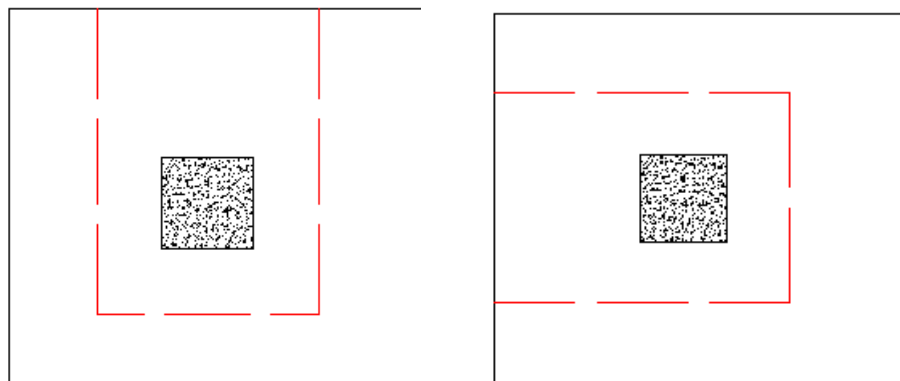
However, these other two edge conditions will also be checked:
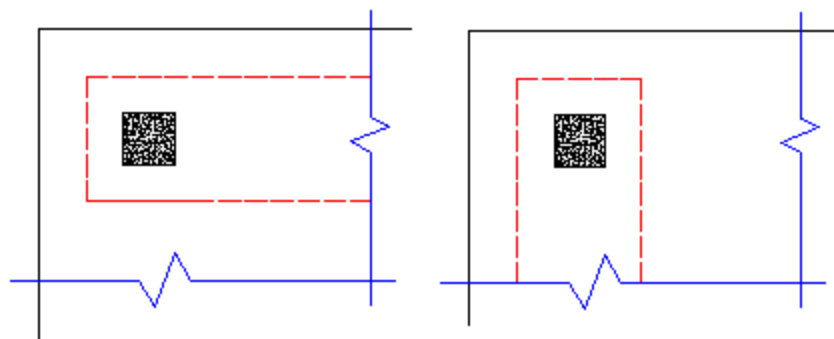
In most cases these odd situations will not govern. However, large moments in a given direction can cause punching shear perimeters with larger areas to govern in specific cases.

The program will also check out any CORNER punching shear considerations. The one below would be the most likely for this given geometry.
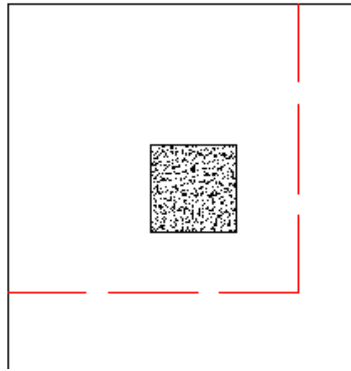
However, these other three corner conditions will also be checked:
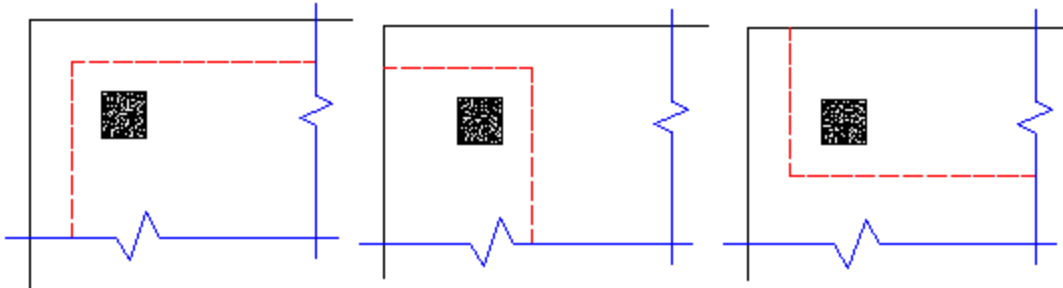
In most cases these odd situations will not govern. However, large moments in a given direction can cause punching shear perimeters with larger areas to govern in specific cases.

Whichever configuration produces a maximum code check will be reported in the punching shear results.

Note:

- The punching shear perimeter lengths shown pictorially above will be shown as the L1 and L2 value for punching shear calculations.
- To limit the total punching shear checks for a given column we have added two caveats to the above behavior. For any of the four Edge or four Corner cases, the program will NOT perform a punching shear check if:
    1. The punching shear perimeter of any punching shear case is more than 50% larger than the minimum Edge/Corner punching shear perimeter.
    2. The punching shear perimeter of a punching shear case encompasses a column within it.

Corner vs Edge Controlling Cases

When the program tests all different cases to determine whether a corner or edge situation governs, it bases the controlling case on the maximum code check produced. There are cases where an edge case is governing that by inspection you might think it would be a corner case. See the images below:

Corner Case   Edge Case 1   Edge Case 2
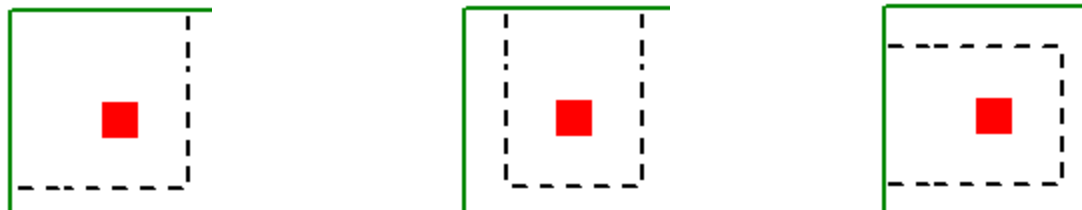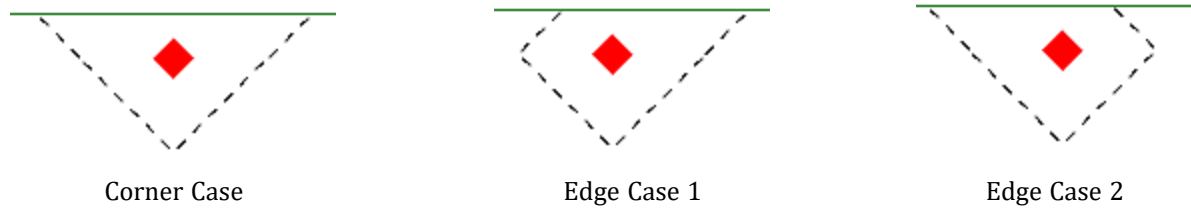
In these two punching shear scenarios shown above, by inspection you would likely expect the corner condition to control over the edge ones. Punching shear capacity is based on a minimum of three equations in the ACI code. One of these equations has an alpha term in it, where alpha is smallest for a corner condition. If this equation governs, this would guarantee the corner condition would have the smallest capacity. However, if this equation does not govern then these capacities may not be as different as you might expect.

Also, the demand portion of the equation will be different based on the geometry of the shear perimeter. It is possible that the edge cases produce a larger maximum stress than the corner cases.

In summary, our testing has shown that these non-intuitive scenarios where an edge controls over a corner have occurred in some instances.

Other Punching Shear Considerations/Limitations

Punching Shear Perimeters That Cross Different Slab Thicknesses

It is possible to have different slab thicknesses within a single column/pedestal's punching shear perimeter. If there is a different thickness at any point along the punching shear perimeter, the program will use the thinnest thickness for the entire calculation. When there is a different thickness, there will be a warning message in the Detail Report. The L1 and L2 values, for use in calculating the b_o, will use the depth at the location of the pedestal/column.

Pedestal/Post Groups

It may be possible for multiple, closely spaced Pedestal/Post to create a punching shear perimeter around the GROUP of piers at a lower force level than would be observed by investigating any one of the individual Pedestal/Post. This is NOT currently considered in the punching shear code checks.

Individual Punching Shear Perimeter   Group Punching Shear Perimeter
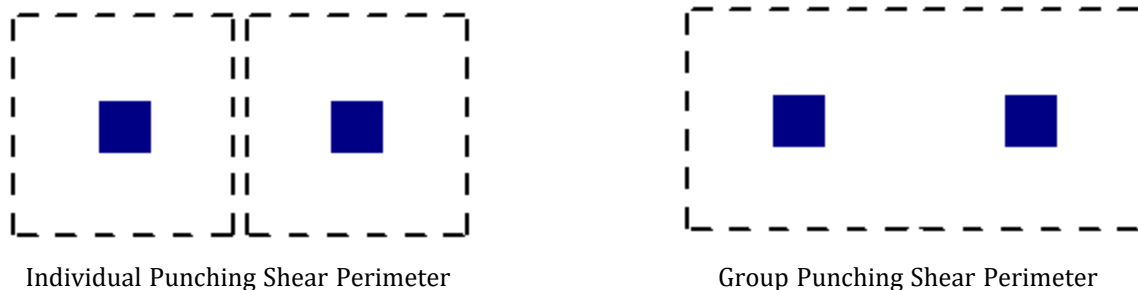
Beams, Columns and Walls intersecting the influence area of a Pedestal/Post

When a beam or wall panel is co-linear with the Pedestal/Post joint, the program will not provide a punching shear check on the Pedestal/Post. In these cases a code check of NC or "no calc" will be listed.
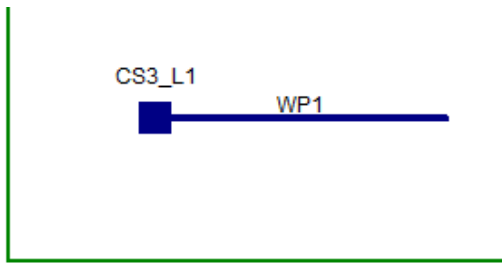
Wall Co-Linear with a Column   Beam Co-Linear with a Column
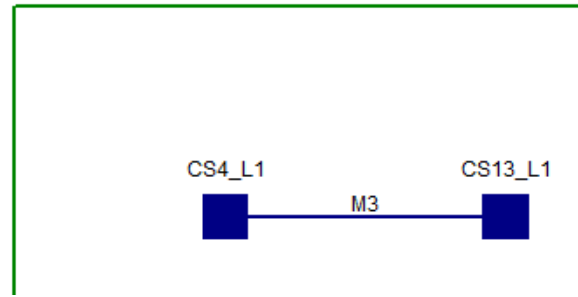
Note:

- This would also refer to situations where a column from a floor above lands on either a beam or a wall. In that case punching shear checks would be omitted as well.

When a beam or column isn't co-linear with the column but is located within the punching shear perimeter then punching shear checks will still be performed. However, the calculations will not be influenced by the presence of that column or beam.
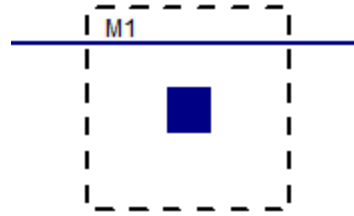
When the Pedestal/Post Local Axes Do Not Correspond to the Slab Geometry

The punching shear perimeters are always investigated with respect to the Pedestal/Post local axes. As shown below, there may be cases where these local axes do not correspond to the slab edge or corner geometry adjacent to that Pedestal/Post. In cases like these it is possible for this to result in an artificially large assumed punching shear perimeter.

Edge Conditions

If either the slab edge is sloping or a column local axis is rotated it is possible to get edge shear perimeters as shown below:
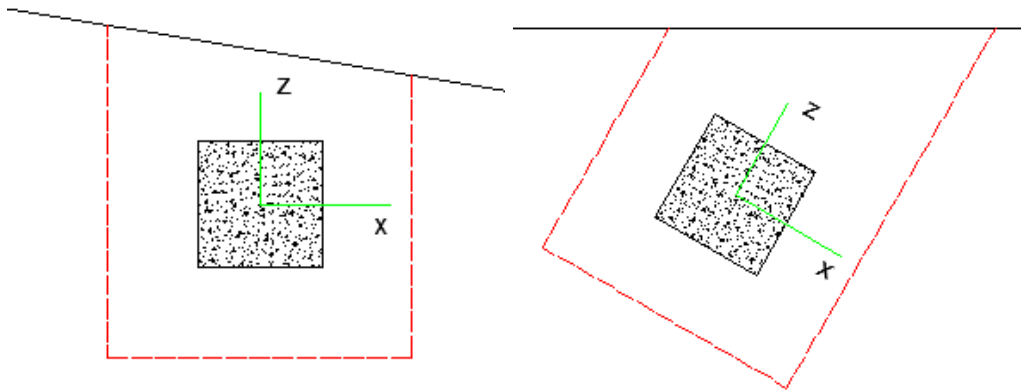
The program calculates this perimeter by drawing a line along the local axis direction along the centerline of the Pedestal/Post as shown below:
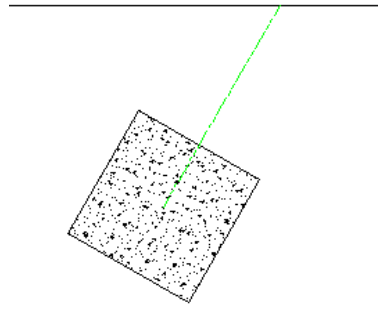
From here the program offsets this line on either side of the Pedestal/Post at a distance d/2 from the face. This produces a shear perimeter that looks like this:
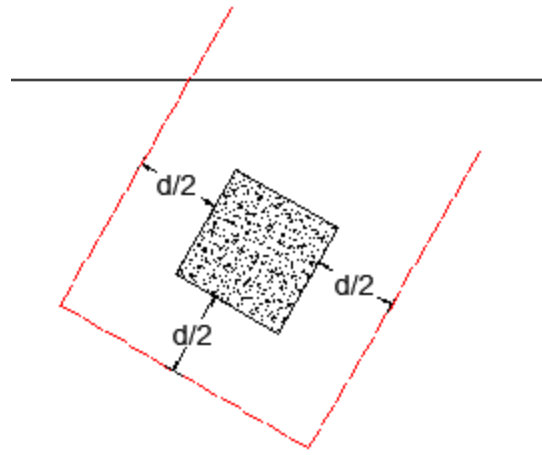
Though this shear perimeter is not technically correct it typically produces a nearly identical shear perimeter to the actual shear perimeter shown above.

Corner Conditions

If either the slab edge is sloping or a column local axis is rotated it is possible to get a corner shear perimeter. The program will start by trying to draw the full shear perimeter. When this happens in this case we find that the shear perimeter has two parts of it that run into a slab edge. When this occurs the program considers this a corner. In this case there are two extremes for what the punching shear perimeter might look like.
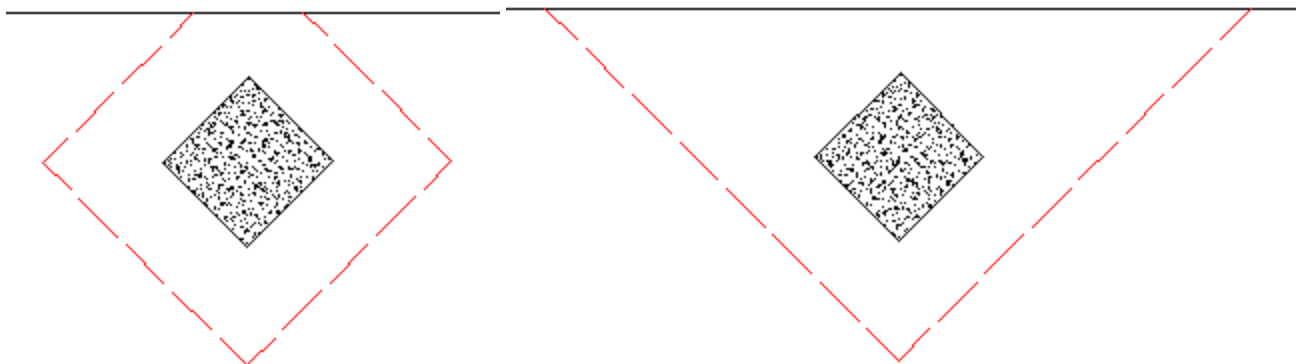
In reality the punching shear is likely somewhere between these two extremes. What the program does here is similar to what is done for the edge case, now just done for both axes on one side only. Lines are drawn along the local axes of the Pedestal/Post until they reach the edge:
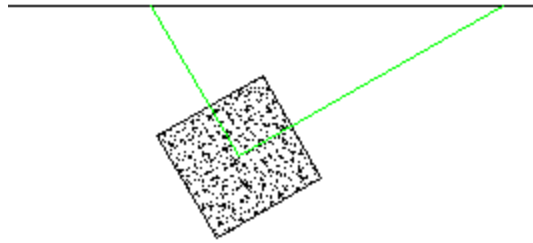
From here the program offsets this line on the side away from the edge at a distance d/2 from the face. This produces a shear perimeter that looks like this:
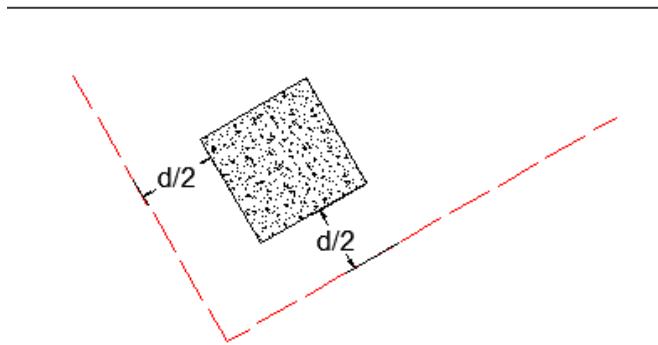
The red line represents the L1 and L2 values the program uses to calculate punching shear perimeter. This perimeter doesn't actually hit the edge of the slab in our calculation, but this puts us in a situation where we are between the two extremes shown above and is a conservative approximation.

Note:

- Pedestal/Posts are defined as a single node in the model. If this node lands inside of the slab then a punching shear check will be performed. If the single node does NOT land on the slab then the Pedestal/Post will not

be considered attached to the slab and no punching shear check will be done.
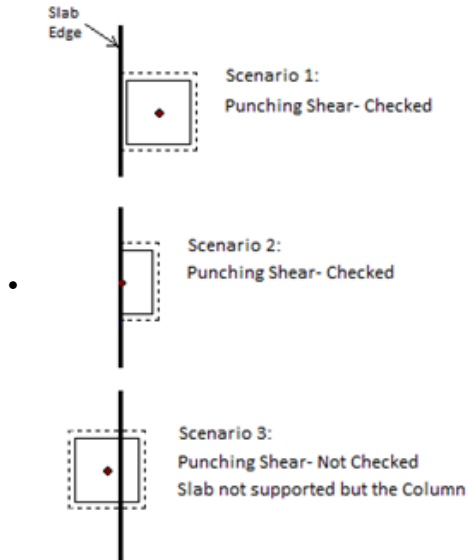
Using Equivalent Square Pedestal/Post

When calculating the punching shear perimeter for a round Pedestal/Post, the program always converts the dimension into an equivalent square of equal perimeter. This greatly simplifies the calculation of punching perimeters when close to an edge or corner.

For an example of this let's use a 12" diameter. The perimeter = Pi*Diameter = 37.7 in.

An equivalent square would have a side dimension = 37.7 in/4 = 9.424"

Thus, a 12" diameter round Pedestal/Post would be considered a 9.424" x 9.424" square for use in punching shear checks.
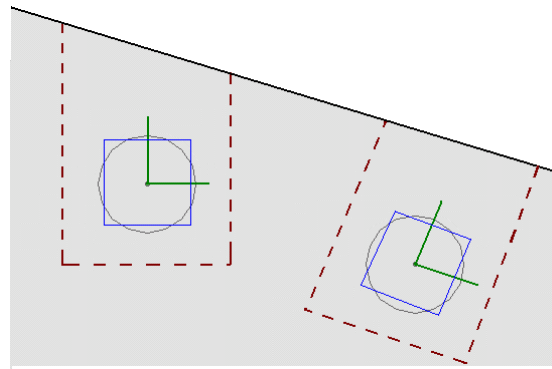
Punching Shear Reinforcement

The program currently does not allow for shear reinforcement. The only capacity considered for punching shear is the concrete itself. ACI 318-14 sections 22.6.6 (shear reinforcement), 22.6.9 (shearheads) and 22.6.8 (headed shear stud reinforcement) are not supported. ACI 318-11 sections 11.11.3 (shear reinforcement), 11.11.4 (shearheads) and 11.11.5 (headed shear stud reinforcement) are not supported.

Punching Shear Checks for Walls, Point Loads, and non-Concrete Columns

It is possible that punching shear failures can occur around walls (especially walls with a short length). It is also possible that punching shear failures can occur around point loads or non-concrete columns bearing on the slab.

RISAFloor ES does not currently check for punching shear for these situations.

Results

You may work with the results of a solution by viewing and sorting data in the spreadsheets, graphically plotting them with the model or by viewing detailed beam reports. You may also print the results in any of these forms. To learn about printing results, see Printing.

Upon the completion of a static solution, RISA opens the **Results Toolbar** and the **Point Reactions Spreadsheet**. If any warnings have been written to the **Warning Log**, it is displayed as well. You may then proceed to view any results and make any changes for further analysis.

If you make any changes to the model that would void the results, such as moving points or adding beams, the results will be purged and another solution will be required before the results can be viewed again.

Each of the result types is described in it's own section:

Results Spreadsheets

- Point Deflection Results
- Point Reaction Results
- Soil Pressure Results
- Beam Force Results
- Beam Design Results
- Beam Bending Reinforcement Results
- Beam Shear Reinforcement Results
- Design Cut Results
- Strip Reinforcing Results
- Plate Force Results
- Plate Corner Force Results
- Footing Results
- Safety Factors

Detail Reports

- Concrete Beam Detail Reports

Saving Results

When you save a file that has been solved you may also save the results. The next time that the file is opened the saved results will be opened as well. You may use the **Application Settings** on the **Tools Menu** to change the way that you are prompted to save results.

If changes are made to the model, any saved results are deleted. Saved results for models that no longer exist in the same directory are also deleted.

Results Spreadsheets

You may access the result spreadsheets by selecting them from the **Results Menu**. You may use the **Find**, **Sort**, and **Exclude** features to find the results you are interested in. For example you might sort the member stresses from high to low, bringing all of the highly stressed members to the top. You might then exclude members that do not have significant axial stresses so that they do not distract you or so that they are not printed.

Finding Results

- To go to a certain item while in a spreadsheet click the **Find** button, and type in the desired beam or point label.

<u>**Sorting Results**</u>

- To sort the results click on the column of results you wish to sort and then click the **Sort** button to specify sorting settings. You can sort based on maximum, minimum, absolute maximum or input order.

Excluding Results

There are ways to exclude results so that you can work with the results that are important to you.

Excluding Results After the Solution

You may run the solution and then graphically select the foundation elements that are of interest. By clicking the button, the spreadsheets and printed reports will be updated so that they will only have results for the selected items. All of the spreadsheets and reports will be controlled by this selection and you may adjust this selection at any time.

Excluding Results While in Spreadsheets

You may also select the results of interest on each spreadsheet. While viewing spreadsheet results you may select the last line of interest and exclude the rest. Exclusions are applied independently for each results spreadsheet. Any exclusion applied to the Member Forces spreadsheet will not affect the Member Stresses spreadsheet, etc. Excluding items graphically will reset **all** of the spreadsheets to match the graphic selection. If you wish to combine these two features to fine tune the results perform the graphic selection first.

To exclude results click on the last line of results you wish to keep and then right click and select click the **Exclude After**. You can bring the results back by right clicking in the spreadsheet and selecting **Unexclude**.

Note

- It might be best to first sort the results before excluding.

Graphic Results

Most of the analysis results may be viewed graphically as well as in the spreadsheets. For the points you may plot the reactions. For the beams you may plot force diagrams. Slab stress contours and plate corner forces may also be viewed graphically. Deflected shapes may be viewed and animated. See Model Display Options for more information.

<u>**To Plot Results Graphically**</u>
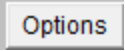
1. If you do not already have the model view open click the **New View** button on the **RISA Toolbar.**
2. Click the **Model Display Options** button on the **Window Toolbar**.
3. Select the options you would like to view and click **Apply**.

Clearing Results

You will be provided with a warning if changes are made to the model that might invalidate the current results. Should you decide to proceed, the results will automatically be cleared and you will have to re-solve the model to get results once you are finished making changes. This warning may be disabled in the **Application Settings** on the **Tools Menu**.

To manually clear results, click the **Clear Results** button.

Opening a model with a solution file that was saved in a different version will also prompt you to delete the solution file in order to open the model.

- To control whether the displayed diagrams and values consider LL Reduction, click on the [Options] button.

Beam Detail Reports

The detail reports for concrete Beams are discussed in greater detail in the <u>Concrete - Design Results</u> section.

RISAFoundation Interaction

RISAFoundation has the ability to transfer load information from RISA-3D and RISAFloor and use this information to design foundations. The following entry will describe how these loads come across the design modules so you can accurately use RISAFoundation for your foundation design.

RISAFoundation Interaction with RISA-3D

RISAFoundation transfers joint reactions from RISA-3D into RISAFoundation by way of load categories. This is not the same as load combinations. Load categories are defined in the Basic Load Cases spreadsheet of RISA-3D.

To get your reactions to transfer from RISA-3D to RISAFoundation:

1. Loads must besplit into specific load categories (DL, LL, etc.) under the Basic Load Cases spreadsheet.
2. The model must be solved.
3. The **Director** button is used to then transfer to RISAFoundation.
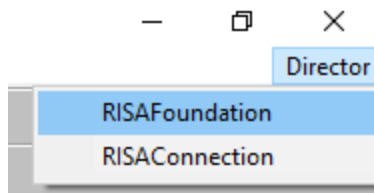
Note:

- If you do not define a load category for a basic load case, then the loads from that case will not be transferred into RISAFoundation.
- You must run at least a single load combination.
- When you are working between RISA-3D and RISAFoundation Model Settings will only be brought over to RISAFoundation the first time you transfer from the RISA-3D module. After this first transfer, any time you change something in RISA-3D from Model Settings that would affect RISAFoundation, then you would also have to go into RISAFoundation and make the change.
- Boundary conditions defined as **Reaction** or **Spring** will be transferred but **Fixed** boundary conditions will not.
- Wall reactions will be brought into RISAFoundation as a series of point loads along the base of the wall. The point reactions are taken from the location of the base of the wall mesh from the RISA-3D model.

Example of Interaction

A simple example would be the braced frame structure seen below on the left. This has been created in RISA-3D. The load applied to this structure came from Load Category WL (wind load). Note that this structure had other loads applied as well. But the wind loads are used as an example.
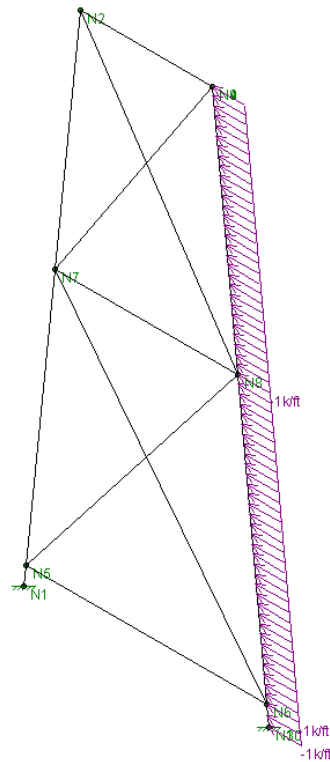
RISA-3D Wind Loads Graphical Representation
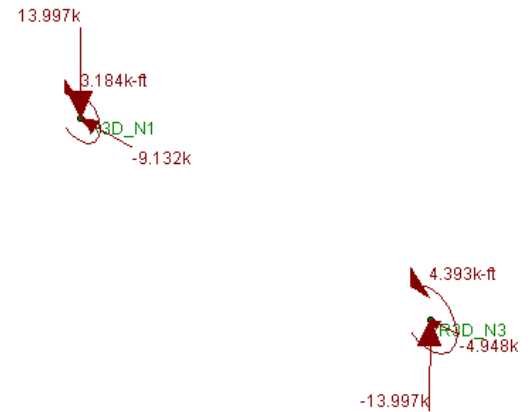
RISAFoundation Wind Loads Graphical Representation

First a load combination is run (note that this does not have to have wind in the load combination), then RISAFoundation is chosen from the Director in RISA-3D. The [Director](#) is the means by which navigation to different modules in the program is possible. When RISAFoundation is first entered dead loads applied to points in a plan view of your structure are displayed. Clicking the [Iso] button will then get an orientation similar to the one shown above. Next, choose WL-Wind Load from the drop down list. [WL-Wind Load ▼] The loads shown above on the right will be displayed. Note that these loads appear consistent for the type of loading present.

Note:

- The points are relabeled in RISAFoundation, though labeled in a way that can be easily compared to the original RISA-3D model joints. You can scroll through the different types of loads and compare them to RISA-3D for accuracy verification.
- Load combinations do not transfer over from RISA-3D, thus they will need to be re-created in RISAFoundation. An easy way to do this would be to just copy the spreadsheet over from RISA-3D.

Foundations at Different Elevations

If, when moving into the RISAFoundation from RISA-3D, there are foundations that are at different elevations, a dialog box will pop-up that looks like this:
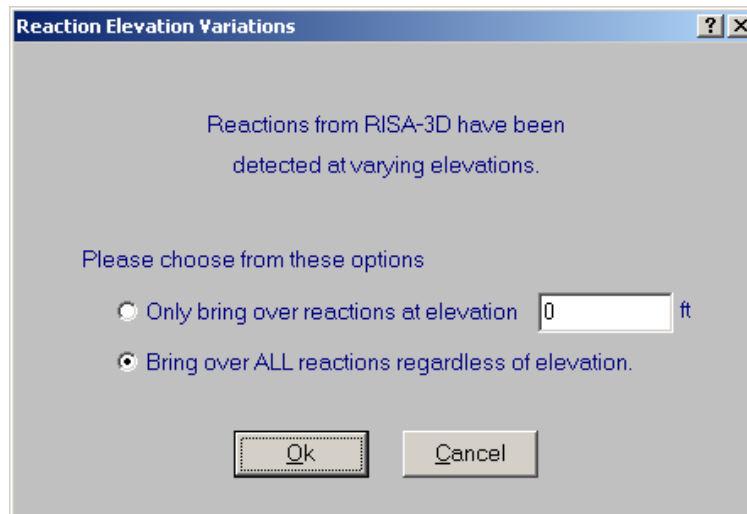
You may choose to bring over reactions at a single elevation of your choice. This may be helpful if there are some reactions that are due to an adjacent structure or support that is not a foundation.

You may choose to bring over all reactions regardless of elevation. This is a good option to have if there is a partial basement level or any condition where there truly are foundations at different levels.

Note:

- All of the differing elevations will be brought in at a single elevation.

RISAFoundation Interaction with RISAFloor

This interaction is a little more involved because it involves three programs (RISA-3D, RISAFloor and RISAFoundation). For RISAFoundation, the bases of columns and the bottom of walls will be considered boundary conditions. Thus, loads are applied at the bases of these elements for foundation design in RISAFoundation.

To transfer from RISAFloor to RISAFoundation you must pass through RISA-3D. Please see the RISA-3D to RISAFoundation procedure

Gravity Elements in RISAFloor

Elements (columns, walls) defined as gravity only are the only elements which transfer load directly from RISAFloor into RISAFoundation. Wall reactions come in as line loads and column reactions come in as point loads.

Wall line load reactions are brought in according the overall moment, M, and axial force, P in the overall wall. The wall moment is taken as a summation of force*distance about one end of the wall. Once P and M are calculated, the magnitudes at either end of the wall are calculated as:

$$I_{end} := \frac{4 \cdot P}{L_{wall}} - \frac{6 \cdot M}{L_{wall}^2} \qquad J_{end} := \frac{6 \cdot M}{L_{wall}^2} - \frac{2 \cdot P}{L_{wall}^2}$$

Lateral Elements in RISAFloor

When applying loads to lateral members within RISAFloor, those loads are simply attributed to the members brought over into RISA-3D. Thus, RISAFoundation does not bring any information in from RISAFloor concerning lateral members. All of this information is taken directly from RISA-3D. From this point we are just working as if we only had RISA-3D and RISAFoundation working together. See above.

Limitations

RISAFoundation can not recognize moving loads

Moving Loads, not described as a load category, and their reactions can't be read as loading within RISAFoundation, thus loads will need to be applied manually to the foundations to account for these.

RISAFoundation requires your Y-axis to be vertical

Models coming from RISA-3D need to have the Y-axis to be vertical to be brought into RISAFoundation. If the Y-axis is not the vertical axis, then a dialog box will pop up when you try to link to RISAFoundation with this message.

RISAFoundation will not consider non-linear load effects

In a true Non-Linear analysis (with Tension Only or Compression Only members), the individual reactions would vary depending on which LC was used to create it because different members may be active for different LC's. In that manner the Load Category reactions that make their way into RISAFoundation are imperfect because a DL or LL reaction would technically be different for the WLX and WLZ cases.
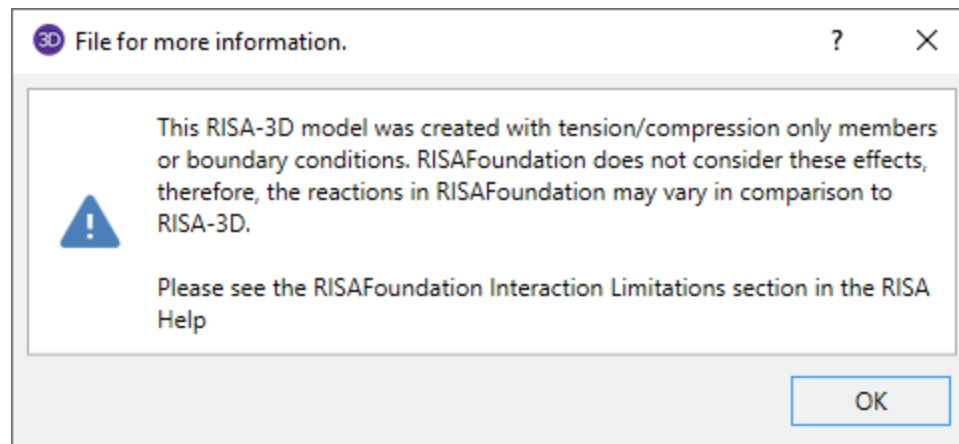
RISAFoundation will not consider second order effects

In a true second order analysis, the P-Delta effect would vary depending on which LC was used to create it because of the non-linearity of the effect when multiple loads are combined together. In that manner the Load Category reactions that make their way into RISAFoundation are imperfect because the P-Delta effect on a DL or LL reaction would technically be different depending on whether the final Load Combination included WLX or WLZ cases.

Shape Databases

For each material, there are several databases of common structural shapes such as Hot Rolled Steel Wide Flanges, Cold Formed Shapes, Wood, Concrete Tees, etc. You may also choose from shapes created in RISASection. You may type in the names directly, select shapes from these databases or add your own shapes.

Database Shape Types

There are different types of shapes for each material type including General shapes. Names for each shape type follow a syntax so that they may be typed directly into the **Shape** field on the **Primary** tab of the **s** spreadsheet. Alternately you may click the ▶ button to look up a shape and select it.

Hot Rolled Shapes

AISC, Canadian, Trade Arbed and custom Hot Rolled shapes are accessed by clicking the **Edit Shape Database** ⬛ button from the **RISA Toolbar**, and then clicking the **Hot Rolled** tab from within the **Shape Database Editor** dialog. The hot rolled shapes and databases are more fully described in the Hot Rolled Steel Design section. See Hot Rolled Steel Databases for more information.

Virtual Joists and Virtual Joist Girders

The Steel Joist Institute (SJI) has put together Virtual Joist and Virtual Joist Girder tables which convert common joist and joist girder sizes into equivalent wide flange beams. These shapes are available by selecting **Virtual Joist Girder** or **Virtual Joist** from the **Database/Manufacturer** menu. For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Virtual Joist Girders**

Concrete Shapes

Concrete shapes do not have a predefined database like hot rolled and cold formed steel. Instead, they are defined using a parametric shape code that may be assigned any depth or width. There are two types of shapes currently supported: Rectangular and Round. See Concrete Database for more information.

Wood Shapes

The available wood shapes are based on the dimension lumber and post and timber shapes given in the NDS or CSA O86, depending on what code you have chosen as your Wood design code. You may also design for multiple plies of these shapes. Note that the NDS dimension lumber shapes are all *nominal* sizes. CSA O86 shapes are *actual* sizes.

Allowable stress values for each shape are based on the species and grade information given in the selected design code. See Wood Database for more information.

General Shapes

Arbitrary Shapes

Arbitrary Shapes are a special, catch-all shape. This arbitrary shape type is provided so that any shape can be added to the shape database.

AISC code checks are not calculated for arbitrary shapes since their place in the specification is unknown. Everything else will be calculated for them (forces, deflections, stresses). The max thickness (Max thick) value for the cross section is used to determine the pure torsional shear stress for the shape. "J" is the torsional constant. The "d" values (the distances to the extreme fibers) allow the program to calculate stresses at the extreme fibers.

Note:

- These shapes will generally be rendered using a greenish cruciform shape. The center of the cruciform will reflect the centroid of the section with the tips of the cruciform representing the distances from the neutral axis to the extreme fiber.
- Refer to Member Shear Deformations and Member Shear Stresses for more information on the shear area factors (As-zz Def, As-yy Def, .As-zz Stress, & As-yy Stress) shown in the figure below.
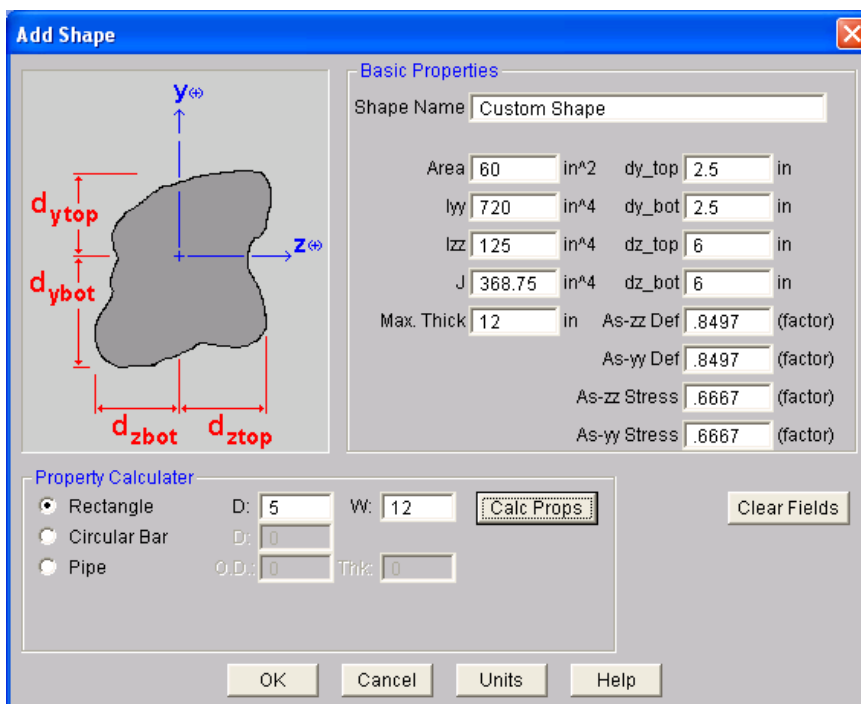
To Add an Arbitrary Shape to the Database

- To enter an arbitrary shape in the database, click the **Edit Shape Database** button from the **RISA Toolbar**, and then clicking the **General** tab from within the Shape Database Editor dialog. Set the shape type to Arbitrary, click **Add** and then enter the shape name and properties.

Pipe Database Shapes

Pipe shapes, which are hollow circular shapes, are entered as on-line shapes. The syntax for these shapes is "PIdiaXthick", where "dia" is the pipe outside diameter and "thick" is the pipe thickness (in inches or centimeters). For example (assuming US Standard units), PI10X.5 would be a 10" diameter pipe with a wall thickness of 1/2".

Solid Rectangular Shapes

These shapes can be defined as on-line shapes. The syntax is "REhtXbase", where "ht" is the rectangle height and "base" is the rectangle base (in inches or cm). For example, RE10X4 would be a 10" deep, 4" width rectangular shape (assuming US Standard units). These shapes can also be defined in the Shape Editor. When defined in the Shape Editor the depth of the solid rectangular section must always be greater than or equal to the width.

Solid Circular Shapes

These shapes are defined as on-line shapes. The syntax is "BARdia", where "dia" is the circle diameter. For example (assuming metric units), BAR2 would be a circular bar with a diameter of 2 cm.

Database Files

The shape databases are stored in the database files (*.FIL). These files may only be edited through the program. The path to these files is set in the **File Locations** tab of **Application Settings**, which is found in the **Tools** menu.

After adding a new shape in your model, you will be prompted with this Changes to Shape Database dialog. Click 'Yes' if you want to add the shape to your Shape Database to be available for all future models. Click 'No' if you do not want the new shape to be available for other models, this will not save the shape to your Shape Database.
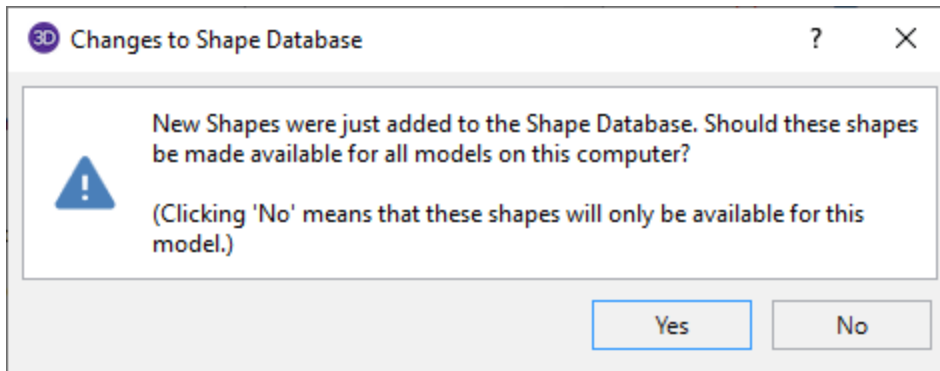
You can also make edits to the geometric and section properties of the shapes in your Shape Database. If you modify geometric properties, you can also use the Re-Calc button for the program to automatically calculate the corresponding section properties.
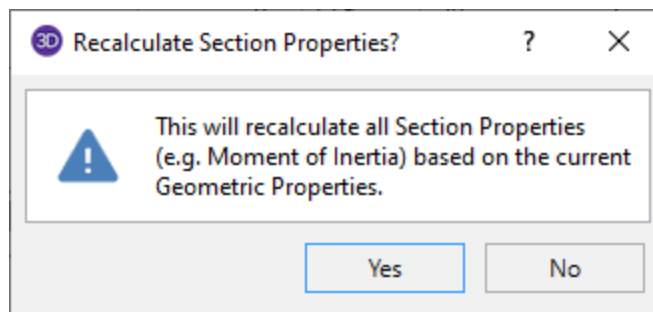
After editing the geometric or section properties for a shape in your model, you will be prompted with this Changes to Shape Database dialog. Click 'Yes' if you want to add the shape modifications to your Shape Database to be available for all future models. Click 'No' if you do not want the changes to apply to other models, this will not save the changes in the Shape Database for other models.
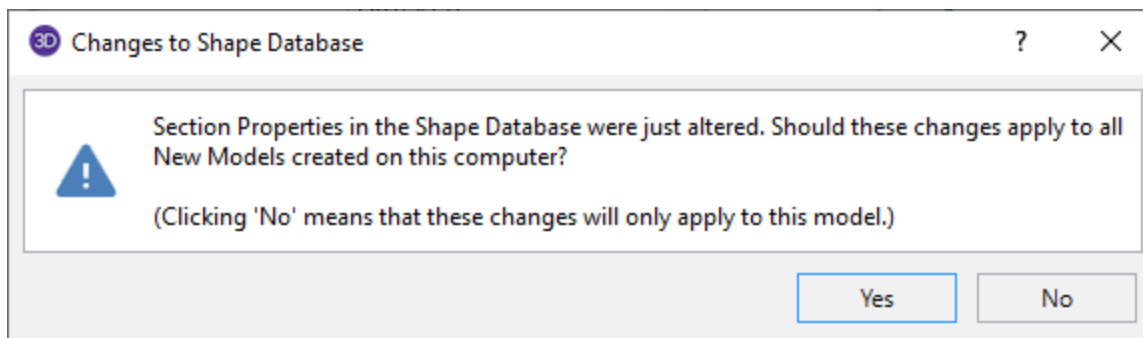
Note

- Alterations to the shape databases are not permanent unless you agree to save them. Changes that are not saved only remain valid for the current session and will not be present the next time you start RISA.

- New shapes are always added to the bottom of the database.
- To delete a shape specify the database and shape type you wish to delete and then click the **Delete** button.
- To edit a shape click the **Edit** button and edit the shape properties. Values can only be manually edited here, nothing will be recalculated. If you wish to have all the values for a shape recalculated, you will need to delete the shape and then add it again with the new properties. For Tapered WF shapes only, you can click **Calc Props** to recalculate the shape properties based on the basic dimensions. Tapered WF shapes are stored parametrically, so there are no database values to edit since they are calculated on the fly whenever an analysis is performed.

Slabs

In this section we will describe how to draw a slab and how RISAFoundation handles the automatic generation of finite element plates. For information about reinforcement design for slabs see the Design Strips and Design Cuts topics.

Draw Slabs

To create a new slab, click the ![icon] button on the **Drawing Toolbar** to open the Draw Slabs dialog. Choose what type of Concrete to use for the material, specify the thickness of the slab and a label prefix for the Slab and Points.

You may define the slab by drawing a polygon that defines the entire slab. Or, you may draw multiple polygons that each define a smaller portion of the slab. If the border of your slab region aligns with an existing slab, the two slabs will be fully connected as if they were a single slab. See the Automatic Slab Meshing section for a discussion on how the slabs are meshed.
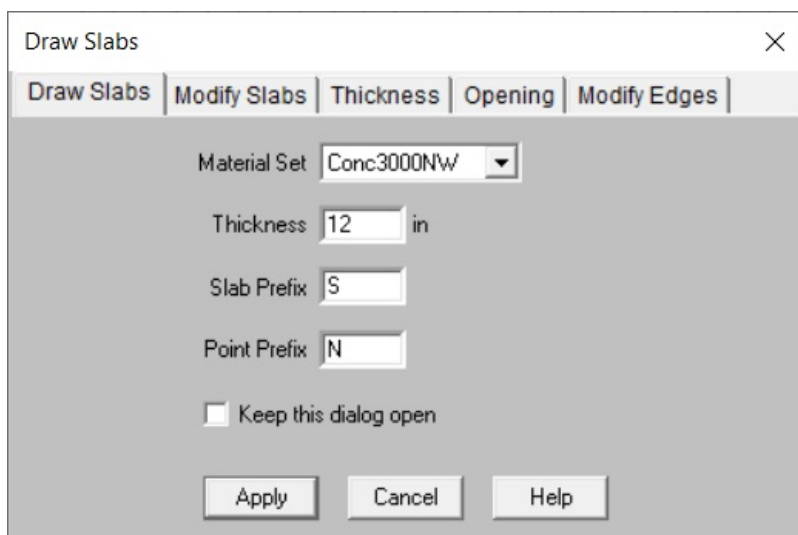
To Create a Slab

1. If there is not a model view already open then click ![icon] on the **RISA Toolbar** to open a new view and click ![icon] to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you do not already have beams or walls supporting the area you will need to create them first.
3. Click the ![icon] button and set the slab properties and click **Apply**.
4. Start drawing a polygon to define slab region. To indicate that you have finished drawing the region double-click on the last node of the polygon.
5. To cancel the and redraw it, click the right mouse button. To cancel drawing altogether, right click the mouse button again, or just press the Esc key.

Modify Slabs

To modify a slab, click the ![icon] button on the **Drawing Toolbar** and click the **Modify Slabs** tab. Here you can adjust the material, thickness, local axis angle, analysis offset, passive pressure, and soil overburden and Icr factor.

Note:

- Only items with the **Use?** checkbox checked will be updated in the model.
- For more information on the Local Axis Angle please see the Local Axis Angle section.
- For more information on the Passive Pressure, please see the Passive Pressure section.
- For more information on soil overburden, please see the Soil Overburden section.

Modify Slab Thickness and Thickness Region Edges

To Modify Slab Thickness

1. If there is not a model view already open then click the ![icon] button on the **RISA Toolbar** to open a new view and click ![icon] to turn on the **Drawing Toolbar** if it is not already displayed.
2. click the ![icon] button on the **Drawing Toolbar** and click the **Thickness** tab.
3. Select the method to Modify Slab Thickness:
4. • **Polygon**
   • **Rectangular Opening: Draw by Clicking Points**
   • **Click Center Point**

To Modify Slab Thickness Region Edges

1. Select the method to Modify Slab Thickness Region Edges:
2. • **Move Corner**
   • **Merge Points**
   • **Delete the Entire Perimeter**

Note

- The edges of the slab cannot be modified. Therefore, in order to create a thickened edge, you'll want to use the thicker dimension on the entire slab and use this tool on the interior of the slab to reduce the thickness.

Polygon: Point to Point Draw

You can create an opening or modify the thickness of any shape by selecting points to form a polygon shape. Click twice on the last point to indicate that the polygon is complete. Below shows an opening with a polygon shape.
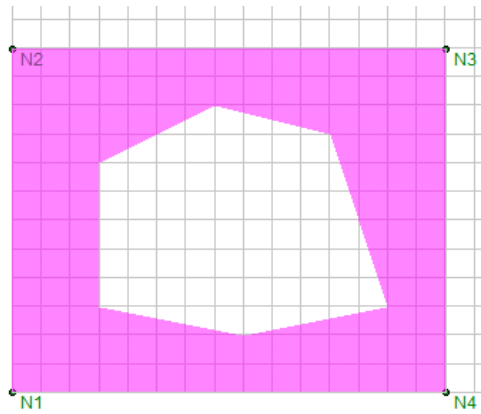
Rectangular: Draw by Clicking Points

You can create a rectangular opening or a thickened slab of any size by selecting two points to form the rectangle. The first click is the one corner, then stretch the mouse to create the desired size and the second click is the diagonal opposite corner. Below shows a rectangular opening.
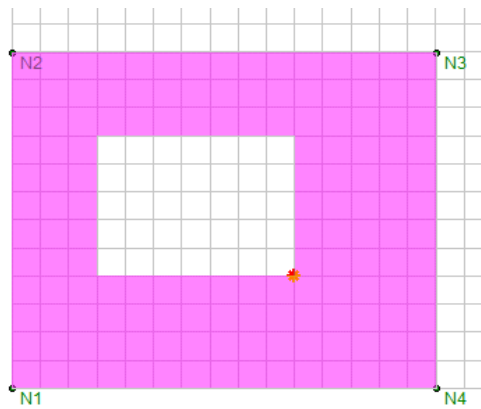
Rectangular: Click Center Point

You can create a rectangular opening or thickness change of a desired size by clicking in the center of the opening. The dimensions entered are the overall dimensions of the opening. Below shows the center point (red dot) with Z axis dimension of 24 in and X axis dimension of 24 in.
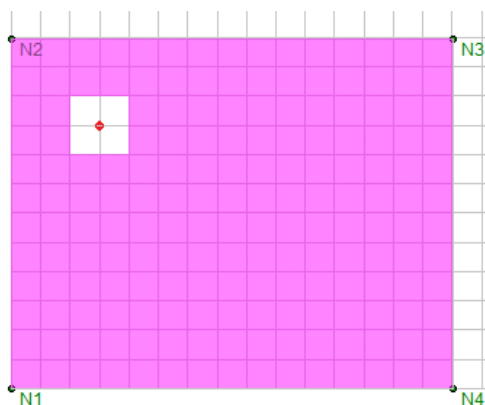
Create and Modify Slab Openings
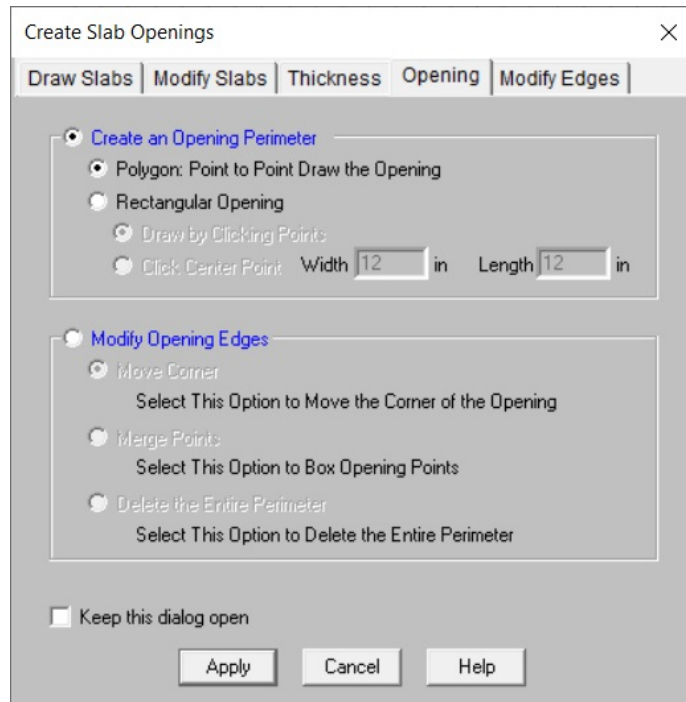
To Create Slab Openings

1. If there is not a model view already open then click the ▣ button on the **RISA Toolbar** to open a new view and click ⬜ to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click on the Create Slab Openings icon ⊠.
3. Select the method to Create an Opening Perimeter:
4. • **Polygon**
   • **Rectangular Opening: Draw by Clicking Points**
   • **Click Center Point**

To Modify Slab Openings

1. Select the method to Modify an Opening Perimeter:
2. • **Move Corner**
   • **Merge Points**
   • **Delete the Entire Perimeter**

Modify Edges

This tool ▣ allows you to modify the edges of a slab. One option is to enter a distance offset and clicking on the slab to be offset. You can input a positive number to enlarge the slab or a negative number to minimize the slab.
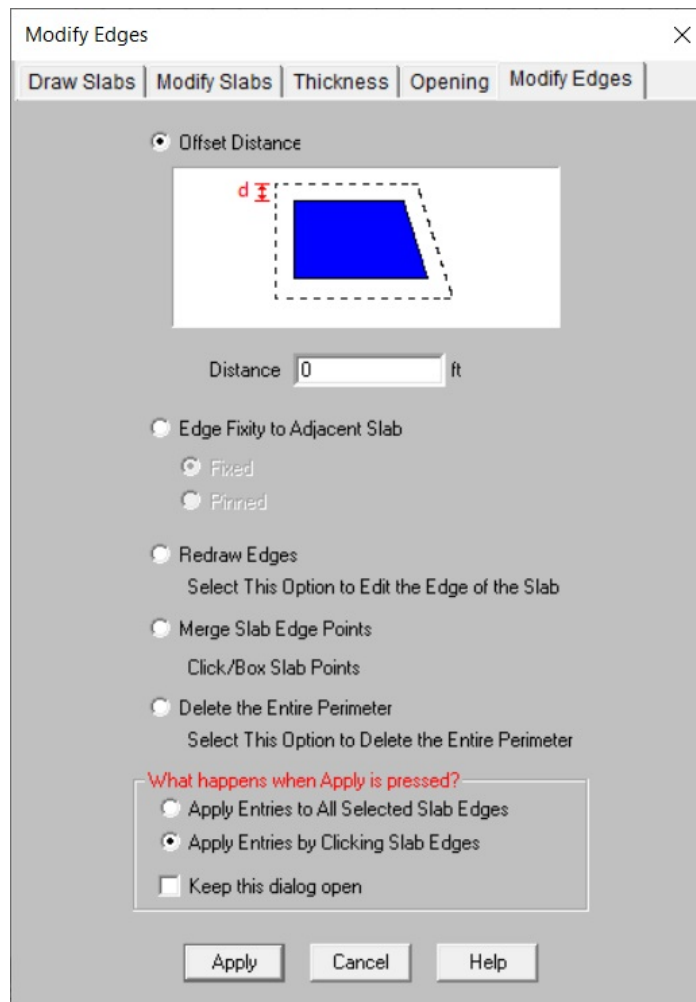
To Modify Slab Edges

1. Select the method to Modify Slab Edges:
2.
   - **Edge Fixity to Adjacent Slab**
   - **Redraw Edges**
   - **Merge Slab Edge Points**
   - **Delete the Entire Perimeter**

For example, if you indicate a 3 ft offset distance the slab edges will be offset linearly by 3 ft on all sides.
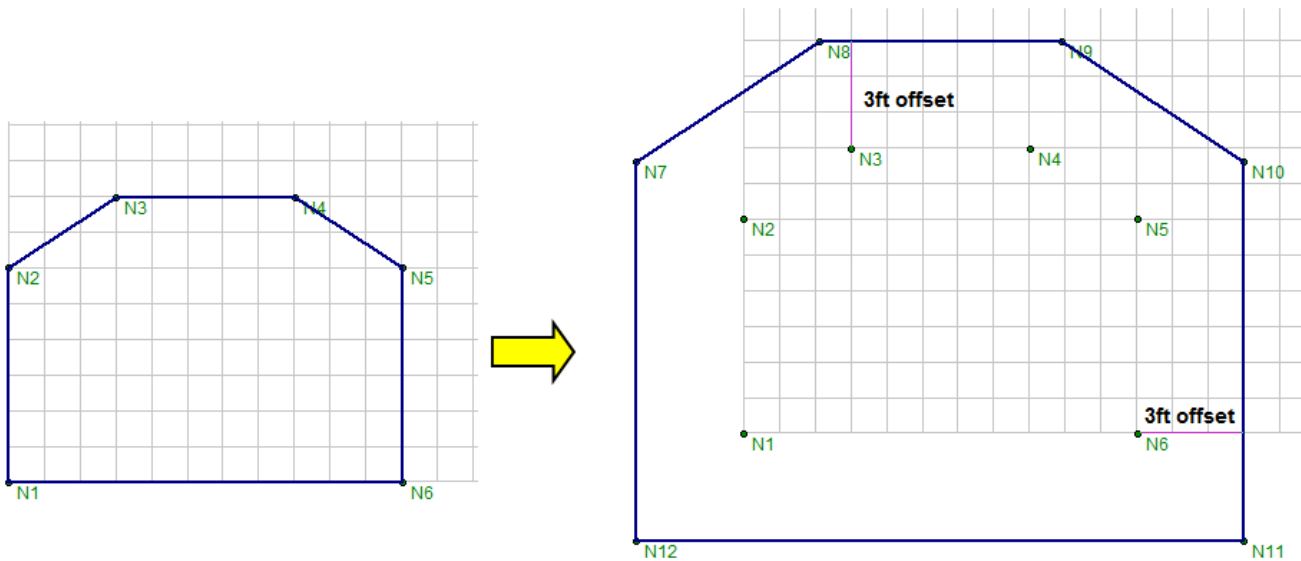
The other option is to specify a pinned edge of a slab. This is useful for slabs which are not monolithically placed, where a shear key or dowels are used to transfer forces out-of-plane to the slab. By default, slab edges are fixed to each other when two adjacent slabs touch each other. Pinned slab edges are graphically indicated with a salmon color, as opposed to the typical blue slab edge. In the example below slab S2 is pinned along both edges that it shares with slab S1:
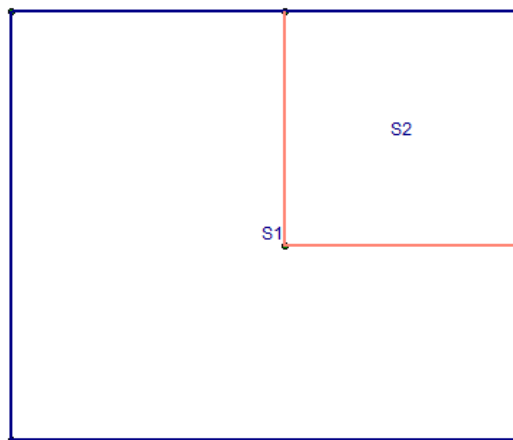
Slabs Spreadsheet

The **Slabs Spreadsheet** records the properties of each slab and may be accessed by selecting **Slabs** on the **Spreadsheets Menu**.

Label	Thickness [in]	Material	Local Axis Angle [deg]	Analysis Offset [in]	Passive Pressure [ksf]	Soil Overburden [ksf]	Icr Factor
S1	12	Conc3000NW	0	0	0	0	0.25

The entries are explained below:

Label

You may assign a unique label to each Slab. Each label has to be unique, so if you try to enter the same label more than once you will get an error message.

If you click the triangle button next to a slab Label, there is a separate spreadsheet to show all the slab regions for the current slab. You can modify the Label, Thickness, and Analysis Offset for each slab region.

Thickness

This entry designates the thickness of the slab.

Note

- Slabs with varying thickness may be created by defining adjoining slabs that share a border but which have a different thickness as shown in the image below. The thickened areas underneath each column are modeled as separate slabs that share edges with the main slab.
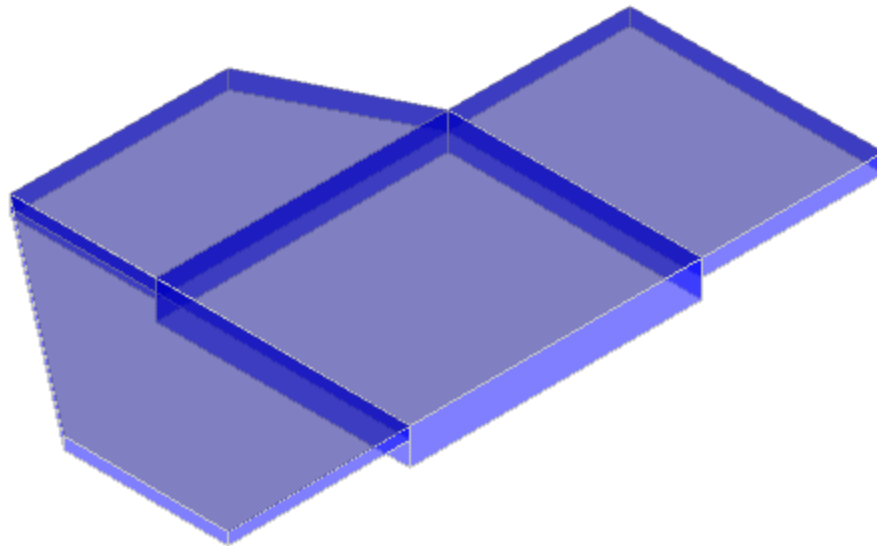
Material

This entry defines the concrete material that will be used for the properties of the slab.

Local Axis Angle

This angle defines the orientation for sliding and overturning checks for slabs. By default this value is set to 0 and this will provide sliding and overturning checks about the global axes.
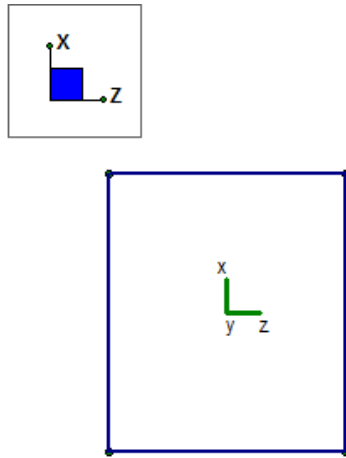
However, if you have a slab that is skewed from the global axes then you may wish to have the overturning and sliding checks taken about those different axes.
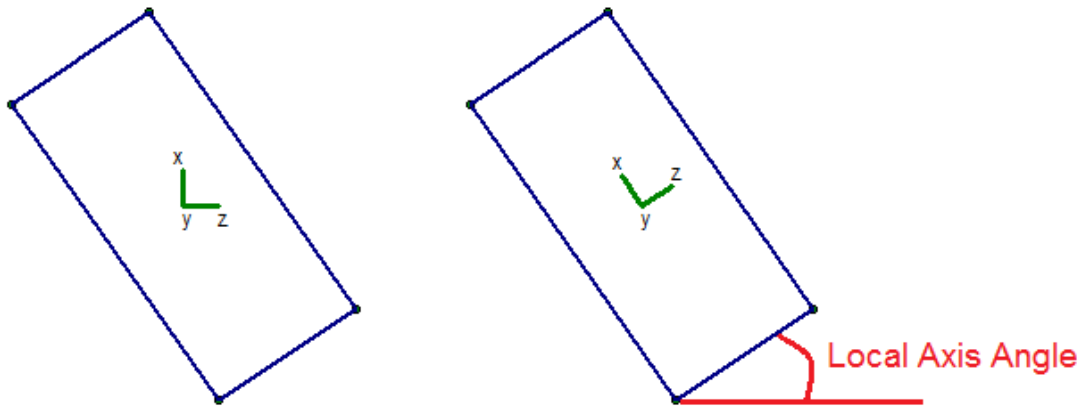
The results for sliding and overturning for slabs are then based on these local axes coordinates. For example, Mo-xx is the demand overturning moment about a slab's local x axis.

Note:

- The Local Axis Angle must be between 0 and 90.
- See the Slabs - Sliding & Overturning topic for output information.

Analysis Offset

This is a positive value that will move the slab downward in relation to any other element in the model. This not just a graphics change. The program is internally adjusting the location of the slab and connecting it to surrounding elements with rigid links. You can expect to see different distribution of forces due to the offset.

Example 1: This is an example of two slabs with no Analysis offset. The slabs are connected at their centerlines.

Example 2: This is an example of the slab offset in order to align the tops of each slab.

Example 3: This is an example of the slab offset in order to align the bottom of each slab.
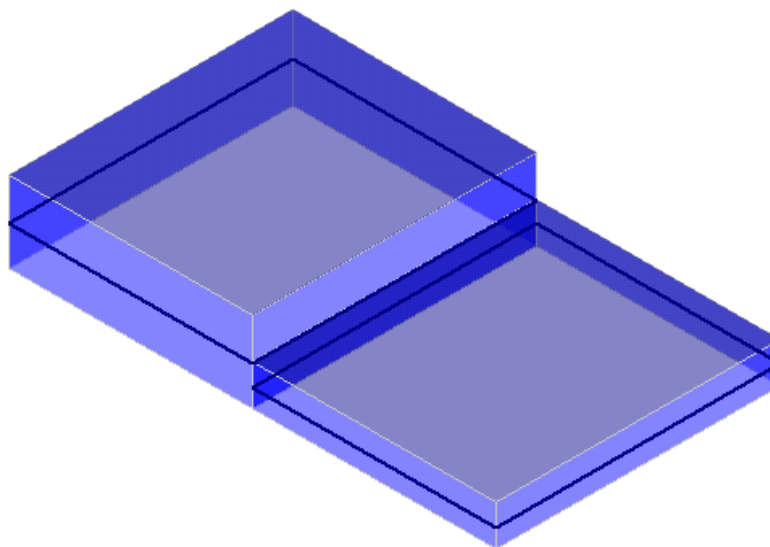
Passive Pressure

The passive pressure represents a soil pressure used to calculate sliding resistance. The passive pressure is multiplied by the vertical area of the slab in the slab's local z and local x directions. This total force is added to the

sliding resistance from friction in the **Safety Factors** results spreadsheet.

Note

- The passive resistance only applies to sliding resistance and has no effect on overturning.
- The passive resistance applies to each individual slab and the presence of adjacent slabs has no effect.

Soil Overburden

The Soil Overburden represents a soil pressure on top of the slab used to calculate resistance to overturning and sliding. The soil overburden pressure is multiplied by the area of the slab and is scaled to reduce the magnitude by the areas of and pedestals, stem walls, columns and post. This total force is added to the overturning resistance in the **Safety Factors** results spreadsheet.

Note

- The soil overburden magnitude is scaled to account for a reduction from pedestals and stem walls, however the pressure is applied uniformly over the slab. This is because the slab plate mesh constrains to only a pedestal center node and stem wall center line and not the perimeter. This assumption is typically valid when the areas of the pedestals and stem walls are small relative to the entire area of the mat slab. Although, if the pedestals or stem walls are relatively large when compared to the slab area, this assumption may be unconservative and the soil burden load should be applied manually.

Icr Factor (Cracked Moment of Inertia Factor)

The **Icr Factor** is used to reduce the bending stiffness of slabs. If this entry is left blank, the Icr factor will default to a value of 0.25.

Note

- The **Icr Factor** will be ignored if the **"Use Cracked Sections"** box is not checked on the Solution tab of the **(Global) Model Settings** dialog.

Service Level Stiffness

Due to cracking and material non-linearity, modeling the stiffness of concrete members is more complex than it is for steel or wood members.

For typical applications, ACI 318-14 Section 6.6.3.1 (ACI 318-11 Section 10.10.4) requires that member stiffness be reduced to account for the cracking that occurs when a member is subjected to ultimate level loads. As described in the previous section, RISA uses the **Icr Factor** to account for this stiffness reduction. However, for service level analysis, the level of cracking will be significantly less. Therefore, the stiffness used in your analysis should be representative of the reduced loading and reduced cracking. Per ACI 318-14 Section R6.6.3.1.1 (ACI 318-11 Section R10.10.4.1), the program will account for this increased stiffness by applying a factor of 1.43 to the cracked section properties for any load combination that has the **"Service"** flag checked in the **Load Combinations Spreadsheet.**

Automatic Slab Meshing

RISAFoundation uses a Finite Element based approach to solving a Slab foundation model. As such each slab will be automatically sub-meshed into a series of plate elements. This plate element mesh is very important for determining the quality of the structural model.

The **Mesh Size** set on the solution tab of the Model Settings is the most important entry for controlling the size of your model. This is the starting size of plate element grid. This entry can be adjusted larger or smaller depending on the size of your slab. Generally speaking a very large mat would need a larger mesh size whereas a smaller mat would require a smaller mesh size.

In RISAFoundation version 3.1 a more robust mesher was added for slabs. This new mesher decreases the amount of mesher errors and increases the speed of the mesher. This will also create a slightly different mesh than in

previous versions. The effect will be very minimal but forces/stresses/deflections may differ by 1-2% in the new version from the old.

Control Points

In addition to the mesh size, any points on the boundary of the slab and any support points within the slab will be considered **control points**. These **Control Points** are "fixed" points within the mesh and will dictate the layout of individual plates surrounding them. Any control points that are closer than the mesh distance specified on the Model Settings will cause an error message to occur during solution.

Note

- Points which are used to apply loads are NOT automatically considered part of the plate mesh. These loads will attributed to the surrounding nodes based on the local element stiffness and the location of the applied loading.
- Points which are not automatically meshed into the slab will have their soil pressure and deflection reported as a linearly interpolated value between the nearest meshed slab plate corner values.

Meshing Adjacent Slabs

When two slabs share a common border, each slab will initially be meshed individually. However, the mesh will then be modified so that all the plate nodes will line up at the slab borders. This is done to ensure perfect force transfer across the slab connection.

Modeling Complex Slabs

The automatic meshing routine can produce odd meshes for complex geometries. Therefore, it can be beneficial to model the slab as multiple adjacent slabs instead. This will force a more rectangular meshing pattern at borders between slabs. The image below shows what kind of an effect modeling a octagon foundation out of multiple slabs can have on the final plate mesh.
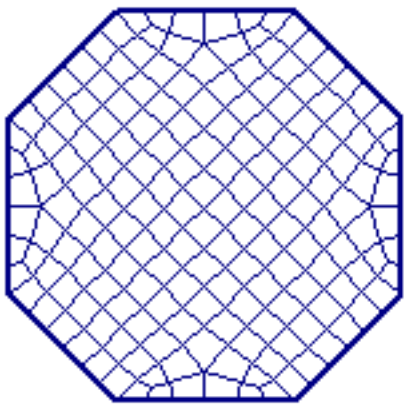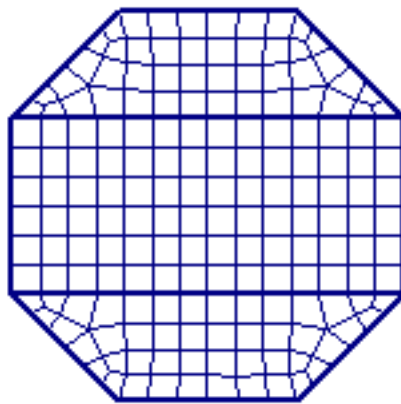
One Slab   Three Slabs   Seven Slabs
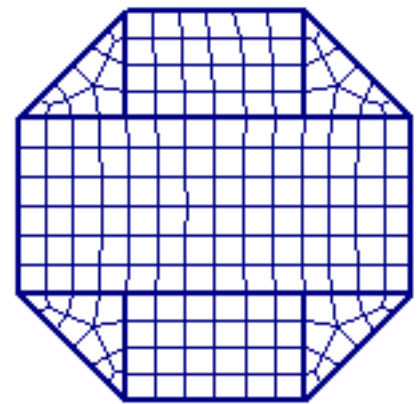
It should be stressed that ALL of the meshes shown above are good meshes that will produce accurate results. However, the image does clearly show how modeling adjacent slabs can force a more uniform and symmetric mesh.

Limitations

Foundation slabs do not consider any stiffness effects from concrete cracking.

Foundation slabs do not consider any long term deflections effects from creep.

Slabs - Design Cuts

Design cuts are used to look at areas of interest or concern within a slab, or to investigate specific critical locations that may be difficult to define with a design strip. They can also be used to double check design strip results as design strip results typically show the envelope of the worst case moments in the whole strip. A design cut is defined by drawing it from a start point to an end point. The total moment about the cut is computed from the underlying finite element results. This moment then is used to calculate the required area of steel and the bars required to satisfy that area of steel. The bars used to resist the moment will be perpendicular to the design cut. Care should be taken when drawing a design cut to only include the area of interest.

How to Create Design Cuts

Design cuts may be defined either before or after a solution is performed.

To create a Design Cut:
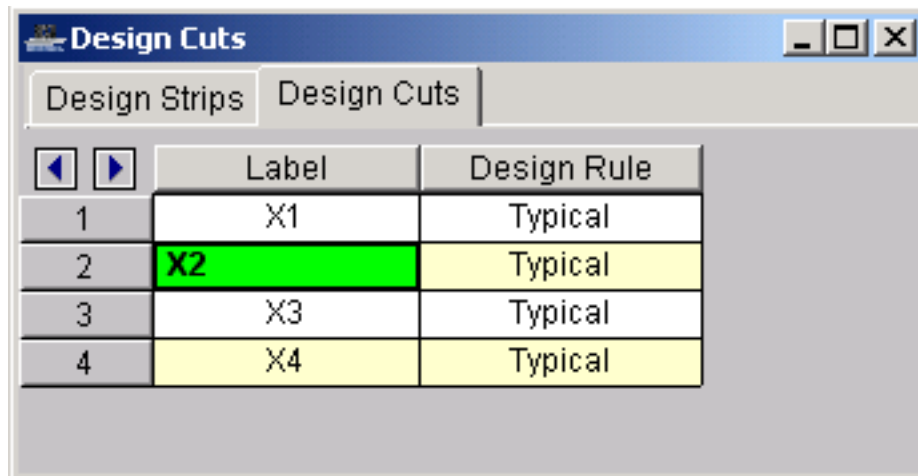
1. Click on the [button icon] button which is located on the lower left side, below the Selection Toolbar. Note that a Model view window must be the active window to see the toolbar buttons on the left side of the screen.

2. Once you have clicked the Design Cuts button, your cursor will change to look like a saw [cursor icon] .. Click two points to define the start and end of your design cut.

3. A design cut will be shown on the screen as a green dashed line, and you will see a new Design Cut definition show up in the Design Cut spreadsheet.

Note

- A partial arrow is also shown in the Design Cut to indicate the bar direction (which is always perpendicular to the cut).

Design Cuts Spreadsheet

As you draw Design Cuts, they are automatically entered into the Design Cut Spreadsheet.

	Label	Design Rule
1	X1	Typical
2	X2	Typical
3	X3	Typical
4	X4	Typical

As shown in the figure, you can modify the Label, the Design Rule used compute the reinforcing bars, and whether or not to show the Design Cut graphically. You cannot modify the Start and End points. If you need to move a Design Cut, you'll need to delete it and redraw it in the new location. The labels can be shown on the model view and it's useful to change them to something meaningful. The design rule used for each Design Cut defines the parameters used to calculate the required area of steel and the reinforcing bars (also the same way Design Rules

are used for beam design.) The option to Show the Design Cut or not is useful if you have a large number of design cuts, or if you are trying a couple of different alternate designs at the same location.

Design Cut Results Spreadsheet

The summary results for all your Design Cuts can be seen in the Design Cut Results Browser.

	Label	UC Top	LC	UC Bot	LC	Shear UC	LC	Phi*Mnz T...	Phi*Mnz B...	Phi*Vny[k]	Top Flex B...	Bot/Mid Fl...
1	X1	0	N/A	.908	2	.721	2	0	359.878	125.702		13 #8
2	X2	0	N/A	.982	7	.772	7	0	332.506	117.322		12 #8

The browser results let you quickly see the Unity Check and Shear Checks (measure of the demand over the capacity) for your Design Cuts. You can also review the moment capacity, as well as the bars required on the top or bottom of the slab.

Design Cut Detail Report

Additional results information can be obtained by viewing the Detail Report for a Design Cut. To see this report, you would click the Detail Report button and then click the Design Cut you want to review. The detail report gives additional information about the Design Cut properties, and also gives many of the supplementary values to help check the results.

General Info

Cut:	X1	Max Top bar Spac.:	12 in	Stress Block:	Parabolic
Material:	Conc3000NW	Min Top bar Spac.:	10 in	Slab Design Option:	Construction
Start:	(28 , 63) ft	Max Bot bar Spac.:	12 in	Rebar Spacing Inc:	2 in
End:	(52 , 75) ft	Min Bot bar Spac.:	8 in	Design Rule:	Typical

The first section of the detail report merely echoes back basic input data such as the starting and ending locations of the design cut, the basic Design Rule parameter, the type of stress block used, and the material type for the slab.

Code Check Data

ACI 318-19 Code Check

Top Bending Check	0.006	Bot Bending Check	0.105	1 Way Shear Check	0.328
Gov Mu Top	2.093 k-ft	Gov Mu Bot	-47.353 k-ft	Gov Vu	19.991 k
phi*Mn Top	369.707 k-ft	phi*Mn Bot	451.285 k-ft	phi*Vn	60.868 k
Governing LC	1	Governing LC	2	Governing LC	2

Tension Bar Fy	60 ksi	Concrete Weight	0.145 k/ft^3	Top Cover	1.5 in
Shear Bar Fy	60 ksi	λ	1	Bottom Cover	3 in
F'c	3 ksi	E_Concrete	3156 ksi	Side Cover	0 in
Flex. Rebar Set	ASTM A615				

Top As Reqd:		Bot As Reqd:			
Analysis	0.017 in^2	Analysis	0.399 in^2		
4/3	NA	4/3	NA	As Reqd(T/S)	2.59200
Min Flex	2.592 in^2	Min Flex	2.592 in^2	Rho Reqd(T/S)	0.00180
Top As Prvd	3.007 in^2 (5 #7)	Bot As Prvd	3.927 in^2 (5 #8)	Rho Prvd(Gross)	0.00481

The information in this section is intended to give the basic data used in the code checking of the concrete section. This included the demand moment (Mu) and capacity (Mn) for when the top bars are in tension or when the bottom bars are in tension. This section also reports the one way shear demand (Vu) and capacity (Vn).

Note

- The program only performs a one-way shear check on the design strip. The two way / punching shear code check is discussed in Pedestals - Punching Shear.

Area of Steel Required

The program calculates the area steel required based on the steps listed below. The green text indicates the number that governed for that Design Cut.

Step 1 (**Analysis**): Calculate the area steel required by analysis. Select a spacing and number of bars that fit the design region.

Step 2 (**Min Flex**): Check that the minimum reinforcement for flexure is being met per ACI 318-14 Section 8.6.1 (ACI 318-11 Section 10.5, CSA A23.3-14 Section 10.5.1.2).

Step 3 (**4/3**): Check the 4/3*As required by analysis per ACI 318-11 Section 10.5.3, CSA A23.3-14 Section 10.5.1.3.

> **Note:** ACI 2014, and Eurocode 2 do not use the 4/3 As for slabs, the results will show NA.

In the example shown below for ACI 2011, the green text indicates that the Top Area of Steel Provided met the Analysis requirements and was increased to meet the Minimum Flexural requirements. Since the 4/3 was greater than Min Flex, it was not used.

ACI 318-11 Code Check

Top Bending Check	0.770	Bot Bending Check	0.000	1 Way Shear Check	0.454
Gov Mu Top	359.001 k-ft	Gov Mu Bot	0 k-ft	Gov Vu	89.838 k
phi*Mn Top	466.405 k-ft	phi*Mn Bot	323.423 k-ft	phi*Vn	198.087 k
Tension Bar Fy	60 ksi	Concrete Weight	.145 k/ft^3	Top Cover	.75 in
F'c	3 ksi	λ	1	Bottom Cover	.75 in
Flex. Rebar Set	ASTM A615	E_Concrete	3156 ksi		

Top As Reqd:		Bot As Reqd:			
Analysis	6.396 in^2	Analysis	NA		
4/3	8.528 in^2	4/3	NA	As Reqd(T/S)	4.71908
Min Flex	8.037 in^2	Min Flex	NA	Rho Reqd(T/S)	0.00180
Top As Prvd	8.394 in^2	Bot As Prvd	5.743 in^2	Rho Prvd(Gross)	0.00539

Below shows an example where there is only negative moment and the bottom bending the As required is being governed by Temperature and Shrinkage.

ACI 318-11 Code Check

Top Bending Check	0.000	Bot Bending Check	0.333	1 Way Shear Check	0.203
Gov Mu Top	0 k-ft	Gov Mu Bot	-107.782 k-ft	Gov Vu	40.117 k
phi*Mn Top	0 k-ft	phi*Mn Bot	323.423 k-ft	phi*Vn	198.087 k
Tension Bar Fy	60 ksi	Concrete Weight	.145 k/ft^3	Top Cover	.75 in
F'c	3 ksi	λ	1	Bottom Cover	.75 in
Flex. Rebar Set	ASTM A615	E_Concrete	3156 ksi		

Top As Reqd:		Bot As Reqd:			
Analysis	NA	Analysis	1.878 in^2		
4/3	NA	4/3	2.503 in^2	As Reqd(T/S)	4.71908
Min Flex	NA	Min Flex	8.037 in^2	Rho Reqd(T/S)	0.00180
Top As Prvd	NA	Bot As Prvd	5.743 in^2	Rho Prvd(Gross)	0.00219

Reinforcement Design Routine

Because a design strip has multiple cuts we need to first consider each cut and then take the enveloped value for the cuts for the overall strip.

For each design cut:

1. Calculate As-Req'd to satisfy all provisions (Strength, T&S, Min_Flex, 4/3*Strength).
2. Calculate an integer number of bars (or an equivalent number of bars if Theoretical Method is selected in Model Setting) that meets or exceeds As-Req'd from Step 1.
3. Calculate a number of spaces (equals Step 2 value minus 1).
4. Spacing = Divide cut width by Number of spaces (from Step 3). This will almost always be a decimal value.
5. Calculate Min_Spacing as the greater of ACI min spacing requirements (2*db, etc) and Min Spacing from Design Rules.
6. Calculate Max_Spacing as the lesser of ACI min spacing requirements (6*db, etc) and Max Spacing from Design Rules.
7. If Min_Spacing < Spacing < Max_Spacing.
   1. Round Spacing down to closest value divisible by Increment (Increment comes from Design Rules). Report this as your Spacing value. Do not adjust As

8. If Spacing > Max_Spacing.
   1. Calculate the lowest integer number of bars which satisfies Max spacing.
   2. Calculate a number of spaces (equals number from Step 8.1 minus 1).
   3. Spacing = Divide cut width by number of spaces from Step 8.2.
   4. Round Spacing down to closest value divisible by Increment (Increment comes from Design Rules). Report this as your Spacing value. Report As from Step 1.
9. If Spacing < Min_Spacing.
   1. Calculate the lowest integer number of bars which satisfies Min spacing.
   2. Calculate a number of spaces (equals number from Step 9.1 minus 1).
   3. Spacing = Divide cut width by number of spaces from Step 9.2.
   4. Report Decimal Spacing. The cut will fail anyway because Req'd spacing was below what's allowed. Report As from Step 1.

Once this is complete we have a rebar spacing and As for each cut.

For each design strip then r eport the smallest value of Spacing from all of the Design Cuts.

Single Layer Rebar Design

The single layer rebar option allows the user to specify one layer of rebar at any depth in the slab. The location of the rebar in the slab is controlled by the top cover input in the design rules. This option especially benefits the design of thinner and lightly loaded slabs on grade where two layers of reinforcement may be too complex and/ or unnecessary. The single layer rebar uses the design routine above to determine how much steel is required, with the addition of the following assumptions:

1. If the single layer is placed <u>at or above mid-depth</u>, the one layer of rebar is assumed to resist both the positive and negative bending.
2. If the single layer is placed <u>below mid-depth</u>, the layer of rebar is only assumed to resist the bottom moment. Any top moment will be compared against the plain concrete capacity according to ACI 318-14 Equation 14.5.2.1a (ACI 318-11 equation 22-2).

Cross Section Detailing

This section provides a visual verification of the bar arrangement, slab thickness and cover. Depending on the Slab Design Option users selected in Global Model Setting - Concrete tab, the Cross Section Detailing can be different.

If the Construction Method is selected, the cross section will show the actual rebar amount:

Cross Section Detailing(All Bars Equally Spaced, Units: in)
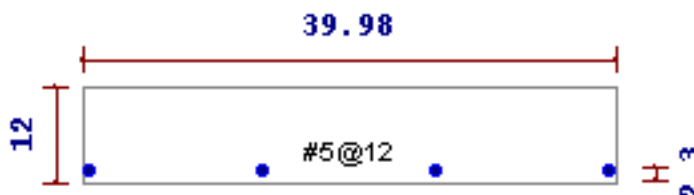
If the Theoretical Method is selected, the cross section will show a generic view and the rebar displayed are for reference only. We do not show the rebar amount because the rebar calculation is based on average rebar area and the rebar amount may not be an integer:

Cross Section Detailing(All Bars Equally Spaced, Amount of Bars for Reference Only, Units: in)


```
#7@10
#8@12
```

30   48   1.9   3.5
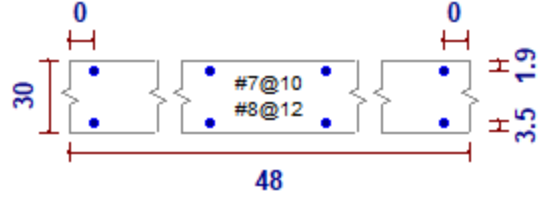
Slabs - Design Strips

Design Strips are used to create regions within a slab where design cuts will automatically be created and flexural reinforcing bars will designed. If the moment varies along the length of a Design Strip then the individual design cuts will show a varying amount of moment demand at each section. The results for the entire Design Strip will be governed by the maximum moment demand at an individual design cut. The same number and size of reinforcing bars will be used for the entire Design Strip, so this should be considered when defining the boundary for a particular strip.

The moment demand at an individual design cut is defined as the total moment over the width of the design cut. No attempt is made to design for local stress or force risers within the width of the design cut. Therefore, it is critical that good engineering judgment is used to determine an appropriate width for setting the design strips / design cuts. You can also see a detail report for each of these design cuts.

Slabs may be analyzed and designed using traditional reinforcement bars or welded deformed wire reinforcement (WWR). WWR can be enabled by going to the Global Model Settings - Design and checking the "Include Welded Wire Reinf." option.

Welded Wire Reinforcement Limitations:

- WWR design only available for Mat Slab, Footing, and Pile Caps
- Only ASTM A1064 deformed bars sizes are available
- Development length is conservatively calculated in the similar fashion as traditional reinforcement bars. Ψw factor is conservatively ignored which reduce the development length due to contribution of "welded intersections".

How to Create Design Strips

Design strips may be defined either before or after a solution is performed.

To create a Design Strip:

1. Click on the DesignStrip button which is located on the lower left side, below the Selection Toolbar. Note that a Model view window must be the active window to see the toolbar buttons on the left side of the screen.
2. Once you have clicked the Design Strip button, you will see the following dialog box so you can make some choices about the design properties before you begin drawing.
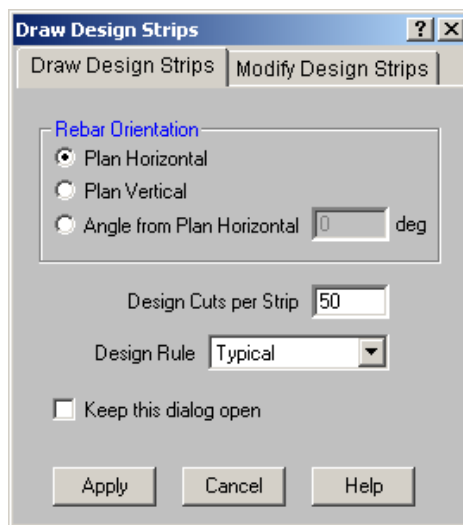
3. You can modify the direction of the reinforcing bars within the strip. You may choose to increase or decrease the number of design cuts that will be created automatically within the design strip, and you can choose which design rule to apply to all the design cuts within the strip. The design rule sets the various parameters needs to calculate the required area of steel and the reinforcing bar size and number. If creating more than one strip with different properties, it is often convenient to check the "Keep this dialog open?" checkbox so the dialog stays open as you draw your design strips.

4. After you press the "Apply" button on the Create Design Strips dialog, your cursor will change to look this

   , and you can then begin to click points to define the boundary of your design strip. You will need to double click the last point to indicate that the current design strip is finished.

5. A design strip will be shown graphically with a thick orange boundary line. Design cuts that are automatically created will be shown as orange dashed lines (as opposed to green if you draw a design cut manually.)

Setting the Width of Design Strips

One of the most important design considerations is how wide to set the Design Strip. If the width is set too large, then the program will average out the moments and shears over too wide of a region. This would result in unconservative design moments and shears. Similarly, if the strip is set too small, then the effect of stress risers in the FEM analysis will be over estimated and the design will be over conservative.

The setting of the design strip widths is truly a matter of engineering judgment. RISA Tech, Inc. makes no endorsement on what methods would be most appropriate.

ACI Definition of Strips (ACI 318-14 Section 8.4.1/ACI 318-11 Section 13.2)

This section of the ACI code is really intended for elevated slabs. But, the concepts can be extended into mat foundations as well. The requirement for "column strips" is that the width on each side should be set to 25% of the span length or width whichever is smaller. Then the "middle strip" is defined to span between the edges of the column strips.

This method requires engineering judgment for column grids that are not perfectly aligned and rectangular. In addition, when the column strip becomes very small then the middle strip may become very wide so that the entire slab is included in either a column strip or a middle strip.

The ACI strip method listed above is based on essentially 1/2 of the mid-span tributary lines. The hand calculation methods would have you design for the full tributary moments over this smaller width which should be conservative. Computer methods (like RISAFoundation) will design for the average moment over the assumed design width which should result in a more efficient design.

Zero Shear Transfer Method

The Zero Shear Transfer method used the shear force contours perpendicular to the span of the slab to set the design width. This should provide a result very similar to using the mid-span tributary lines, but is a bit more theoretically derived for non-rectangular column layouts. This method is described in greater detail in the PTI publication Design Fundamentals of Post-Tensioned Concrete Floors. Ideally, this method should give design strips of similar width to the ACI strip method. However, it is more rationally derived and should work better for cases where uneven column spacing makes the strip method difficult to apply.

Zero Moment Method

In a similar fashion to the zero shear transfer method, the Zero Moment method uses the moment contours to identify where the moment changes sign. This can be used to set the design strip width approximately equal to the distance between zero moments.

Shear Perimeter Method

Another basis would be to set the design width equal to the pedestal width plus a distance 'd' or 'd/2' on each side. This will end up being a more conservative assumption for flexure than the other methods listed. As such, it would be more appropriate for situations where shear or punching shear failures are a primary concern. Examples would also include cases where the pedestal is very large such as for a vertical vessel or grain silo. This is similar, though not identical, to a method given in the NEHRP document GCR 12-917-22 (Seismic Design of Reinforced Concrete Mat Foundations).

Hybrid Method / Engineering Judgment

A variation on these methods would be to start off setting the column strip using the ACI strip method. Then, if necessary, the width could be modified based on considering the other methods. This is especially true for situations where the column grid is not aligned or rectangular.

In addition, when the middle strip widths get too large, they could be set to values closer to the column strip width. The middle strip would normally be centered on the area with the highest mid-span moments. This would neglect lower moment regions between the column and middle strips. Hence the strips would designed for a higher moment per unit width. This reinforcement could then be extended into the lower moment regions between strips. Or the user could set up another design strip for these lower moment regions.

How to Modify Design Strips
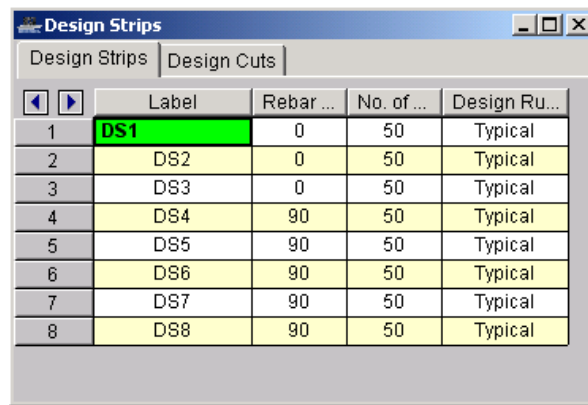
To edit an existing design strip click on the and select the Modify Design Strips tab.This will pull up the following dialog. You can modify the direction of the reinforcing bars within the strip, the number of cuts or the Design Rules associated with the strip.
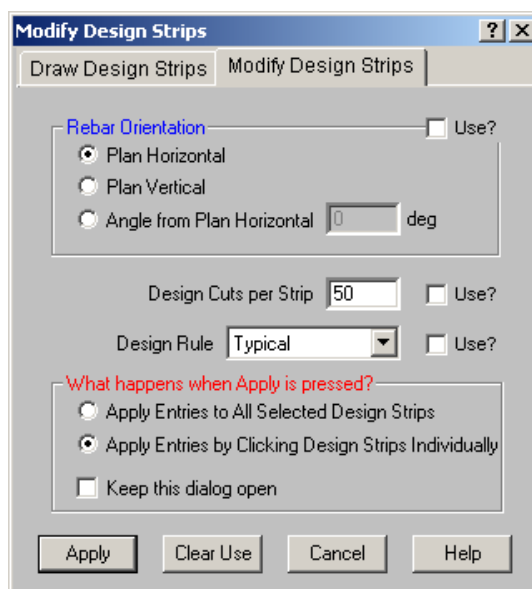
The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected beams, if the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on beams without affecting all the rest of the properties.

Design Strips Spreadsheet

As you create Design Strips, they are automatically entered into the Design Strips Spreadsheet.
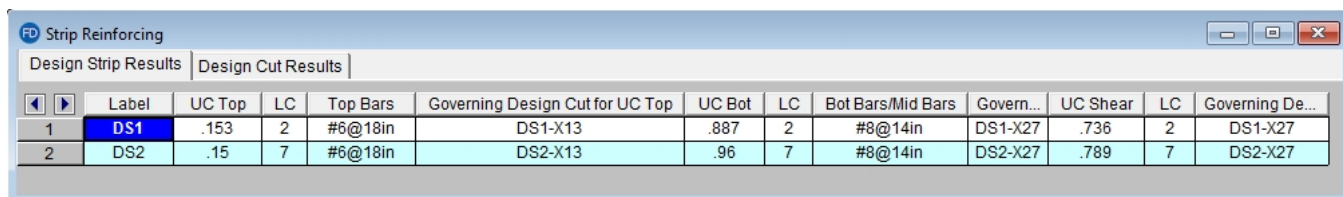
As shown in the figure, you can modify the Label, the angle of the reinforcing bars, the number of automatically generated design cuts, and the Design Rule used compute the reinforcing bars. You cannot modify the design strip boundary. If you need to modify the boundary of a design strip, you'll need to delete it and redraw it with the new modification.

The labels can be shown on the model view and it's useful to change them to something more descriptive. The rebar angle lets you handle curved and other types of custom layouts. The number of design cuts can be increased for larger design strips to make sure your capturing the representative loads within the design strip. The design rule is used for each design cut to define the parameters used to calculate the required area of steel and the reinforcing bars (the same way Design Rules are used for beam design.)

Design Strip Results

The results for all your Design Strips can be seen in the Design Strip Reinforcing Results Browser.

The browser results let you quickly see required top and bottom reinforcing bars for each design strip and the code check produced for that strip. If welded wire bars are designed, the output result will be preceded by a "D" instead of "#" to indicate wire bar size. Additionally, the governing design cut for the top and bottom reinforcing bars are shown so you can investigate the way the reinforcing bars were calculated. Also given is the shear code check and which design cut governed for that shear check. The results for each strip are the envelope of all the design cuts within the design strip. Top and bottom results are enveloped independently.

Design Strip Results

The browser results let you quickly see required top and bottom reinforcing bars for each design strip. Additionally, the governing design cut for the top and bottom reinforcing bars are shown so you can investigate the way the reinforcing bars were calculated. Also given is the shear code check and which design cut governed for that shear check. The results for each strip are the envelope of all the design cuts within the design strip. Top and bottom results are enveloped independently.

Design Strip Detail Report

Additional results information can be obtained by viewing the Detail Report for a Design Strip. To see this report, you would click the Detail Report button [Detail] and then click the Design strip you want to review. The detail

report gives additional information about the Design Cut properties, and also gives many of the supplementary values to help check the results.

Controls
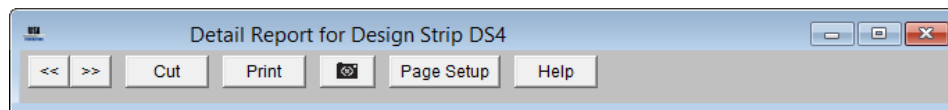
The detail report will always show the **Envelope** force diagrams.

The arrow buttons allow the user to quickly browse through numerous Design Strip detail reports. Whereas, the **Cut** button will switch between the detail reports for individual design cuts and the detail reports for the overall design strip. The button will allow you to take a snapshot of the current detail report you are viewing so that it can be added to a report. View the Printing topic for more information.

General Info

Strip:	DS2	Max Top bar Spac.:	12 in	Stress Block:	Rectan
Material:	Conc3000NW	Min Top bar Spac.:	12 in	Rebar Orientation:	0
Strip Width:	12 in	Max Bot bar Spac.:	12 in	Rebar Spacing Inc:	2 in
Total Cuts:	50	Min Bot bar Spac.:	12 in	Design Rule:	Typical

The first section of the detail report merely echoes back basic input data such as the strip width, number of cuts, basic Design Rule parameters, the type of stress block used, and the material type for the slab.

Envelope Shear and Moment Diagrams
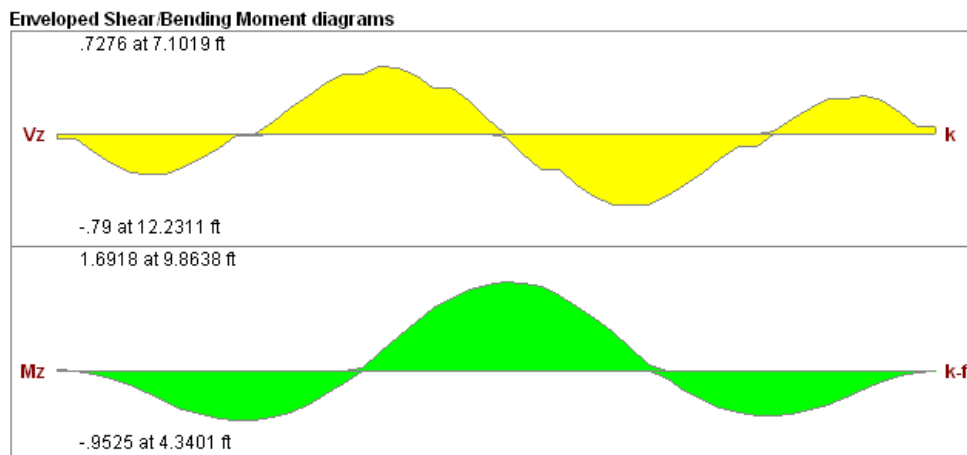
This section shows the shear and bending moment envelope diagrams for every cut along the length of the design strip.

Note:

- Since the width of a design strip can vary along its length, the controlling code check location (which will normally be based on the maximum force per unit width) does not necessarily correspond to maximum shear and moment locations reported in the diagrams.

Code Check Data

ACI 318-14 Code Check

Top Bending Check	0.153	Bot Bending Check	0.887	1 Way Shear Check	0.736
Gov Mu Top	29.864 k-ft	Gov Mu Bot	-319.129 k-ft	Gov Vu	92.566 k
phi*Mn Top	195.549 k-ft	phi*Mn Bot	359.878 k-ft	phi*Vn	125.702 k
Governing Cut	DS1-X13	Governing Cut	DS1-X27	Governing Cut	DS1-X27
Governing LC	2	Governing LC	2	Governing LC	2
Tension Bar Fy	60 ksi	Concrete Weight	.145 k/ft^3	Top Cover	1.5 in
Shear Bar Fy	60 ksi	λ	1	Bottom Cover	3 in
F'c	3 ksi	E_Concrete	3156 ksi		
Flex. Rebar Set	ASTM A615	Rho Bot Prvd	0.00667	Rho Top Prvd	0.00242
		Prvd Bot Bar Spac.	#8@14in	Prvd Top Bar Spac.	#6@18in

Bending Steel Reqd/Prvd, Units: in^2)

Cut Label	Top As Reqd	Top As Prvd	Bot As Reqd	Bot As Prvd	Rho Reqd(T/S)	Rho Reqd(Flex)	Rho Prvd(Gross)
DS1-X13	.658	4.418	NA	10.21	0.00180	0.00180	0.00677
DS1-X27	NA	4.418	8.961	10.21	0.00180	0.00180	0.00677

The information in this section is intended to give the basic data used in the code checking of the concrete section and the cut locations which governed those code checks. This includes the demand moment (Mu) and capacity (Mn) for when the top bars are in tension or when the bottom bars are in tension. This section also reports the one way shear demand (Vu) and capacity (Vn).

The Bending Steel details displays the amount of steel "required by analysis" compared to the amount provided. In addition, the temperature and shrinkage (T/S) requirements are compared to the gross reinforcement ratio provided (which includes both the top and bottom reinforcement).

Note

- The program only performs a one-way shear check on the design strip. Note that when ACI 318-19 is selected, the Vc is calculated based on Equation (c) in Table 22.5.5.1. The size effect factor λs is not considered for foundation slab as allowed in Section 13.2.6.2. The two way / punching shear code check is discussed in Pedestals - Punching Shear.
- If the design moments are too large or the Design Rules are too constrained it is possible to fail the reinforcement checks for the design strip. These checks include temperature and shrinkage reinforcement minimums, flexural minimums, maximum spacing requirements and strength requirements. If any of these checks fail a warning log message will be given.

Slabs - Results

When the model is solved, slabs are sub-meshed into plate elements and there are a couple of results spreadsheets specifically for these plates.

Plate Force Results

Access the **Plate Forces Spreadsheet** by selecting the **Results Menu** and then selecting **Plates** ‣ **Forces**.

The **Plate Forces** are listed for each plate. Interpretation of output results is perhaps the most challenging aspect in using the plate/shell element. The results for the plates are shown for the geometric center of the plate.
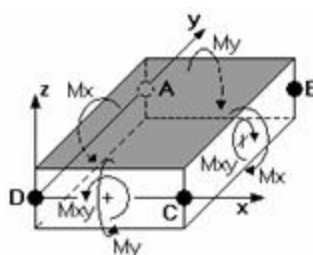
Plate Moments
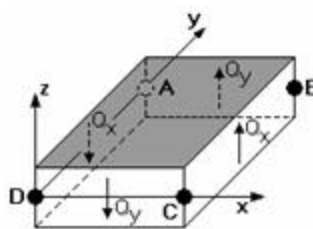
Transverse Shears
(Out of Plane)

The forces (Q_x and Q_y) are the out-of-plane (also called "transverse") shears that occur through the thickness of the element. The Q_x shear occurs on the element faces that are perpendicular to the local x-axis, and the Q_y shear occurs on the element faces that are perpendicular to the local y-axis. Q_x is positive in the z-direction on the element face whose normal vector is in the positive x-direction. Q_y is positive in the z-direction on the element face whose normal vector is in the positive y-direction. The total transverse shear on an element face is found by multiplying the given force by the width of the element face.

The plate bending moments (M_x, M_y and M_{xy}) are the plate forces that induce linearly varying bending stresses through the thickness of the element. M_x is the moment that causes stresses in the positive x-direction on the top of the element. Likewise, M_y is the moment that causes stresses in the positive y-direction on the top of the element. M_x can then be thought of as occurring on element faces that are perpendicular to the local x-axis, and the M_y

moment occurs on faces that are perpendicular to the local y axis. To calculate the total M_x or M_y on the face of an element, multiply the given value by the length of the element that is parallel to the axis of the moment. For example, looking at the 'Plate Moments' figure above, the total M_x moment could be obtained by multiplying the given M_x force by the length of side BC (the distance from point B to point C). The total M_y force can be calculated in the same way by instead using the length of side DC.

The M_{xy} moment is the out-of-plane twist or warp in the element. This moment can be added to the M_x or M_y moment to obtain the 'total' M_x or M_y moment in the element for design purposes. This direct addition is valid since on either the top or bottom surface, the bending stresses from M_{xy} will be going in the same direction as the M_x and M_y moments.

Note

- For the placement of concrete reinforcement, it is helpful to realize that laying reinforcement parallel to the local x-axis will resist the M_x moment.
- A positive M_x or M_y moment will put the top fiber of the plate in tension.

Note that the plate bending (Q_x, Q_y, M_x, M_y, M_{xy}) results are forces per unit length. For example, a rectangular element with a B to C length of 10 feet showing a M_x force of 20K-ft would have a total force on the B-C face of the element of 20K-ft (per foot) times 10 feet, or 200K-ft.

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Model Display Options - Slabs to learn how to plot point results.

Plate Corner Force Results

Access the **Plate Corner Forces Spreadsheet** by selecting the **Results Menu** and then selecting **Plates** ▸ **Corner Forces**.

L...	Plate Label	Point	Y[k]	MX[k-ft]	MZ[k-ft]
1	P1	N138	.674	-.305	-1.056
2		N139	.759	-.607	-.139
3		N140	.268	2	.207
4		N141	-1.702	2.554	-1.295
5 1	P2	N225	-1.16	1.701	2.249
6		N226	.497	1.338	.184
7		N227	.838	-.248	.2
8		N228	-.176	-.271	2.035
9 1	P3	N229	.57	-.102	1.443

The plate corner forces are the global forces at the corner of each plate.

These are the forces and moments calculated at the corners of the plates, in the GLOBAL directions. These values are obtained by multiplying the plate's corner displacements with the global stiffness matrix. Unlike the local forces, which are very accurate approximations, these corner forces represent EXACT results based on linear elastic theory. Also, the local forces are listed on a 'per unit length' basis, whereas these global direction corner forces represent the total force on the plate at the corner in the given direction, in the same way that joint reactions are reported. At any given point, the corner forces for all plates connected to that point should sum to zero (a requirement of equilibrium), assuming no beams or supports are also present at the point.

These are enveloped results and the maximum and minimum value is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

Note

- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort**, **Label Marked Lines in Current ViewShow Selected Lines in Current View** and other options.
- See [Model Display Options - Slabs](#) to learn how to plot point results.

Slab Contour Display Tools

Contour Display Details

This tool may be activated by clicking the Σ icon below the Selection Toolbar or by using the Right-Click menu from within a model view and selecting **Make a Cut Over Displayed Contour**. The tool is only available from within a model view that is currently plotting a slab contour.
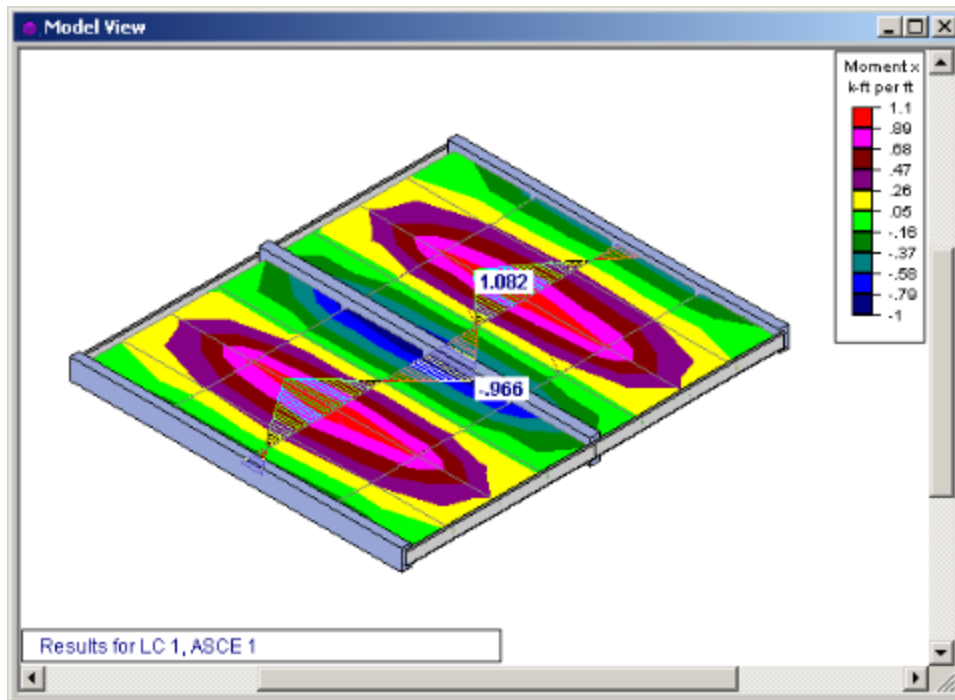
The integral of the force diagram is displayed in the Status Bar at the bottom of the screen. This may be useful for summing up shear forces in a wall or lintel.

The Contour Display Detail is used to turn a visual plate contour into a more traditional shear or moment diagram. The tool merely transforms the visual contour display into actual numerical that can be viewed by the user. In the image below, the contour display shows the numerical values associated with the contour at the cutting line.

When a Contour Display Detail is shown, the right click menu gives access to a Detailed Diagram of the contour cut as shown below:
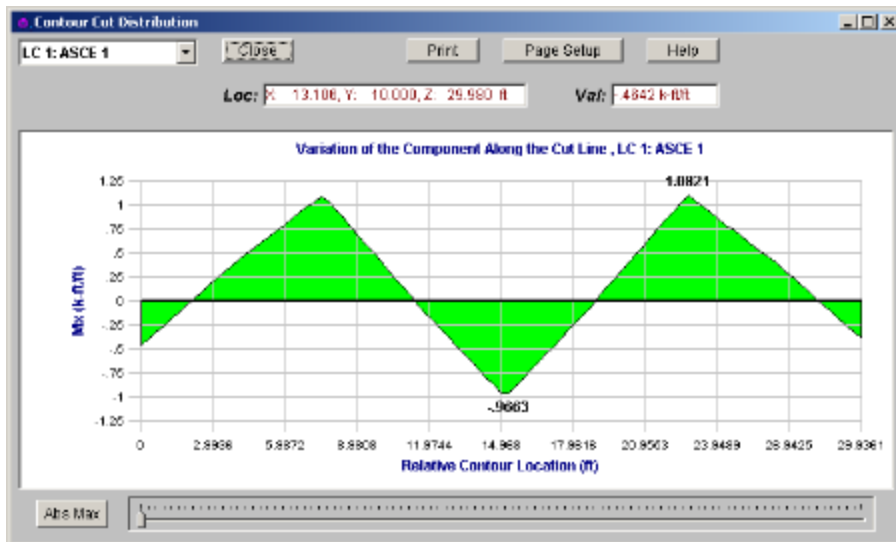
This detailed diagram will show you all the results for every section along the length of the diagram. this information may even be copied to the clipboard for use in a spreadsheet program. This can be done by selecting **Copy Data** from the right click menu.

Note

- Because this display is just a currently displayed plate contours, the units for the detailed diagrams are always the same as for the displayed contours.

Slabs - Sliding & Overturning

Slab sliding and overturning checks are done automatically for all service load combinations in RISAFoundation. These checks are very similar to the sliding and overturning checks for footing elements.

Note:

- You can access these spreadsheets by clicking the **Safety Factors** button on the **Results** toolbar.

Demand vs Resisting Forces

Forces (overturning or sliding) are considered based on load category, not on the overall load combination loads. Please see the conversation on that in the Footing Stability and Overturning topic for a conversation on this subject. That topic specifically talks about footings, but the same considerations hold for slabs as well.

Let's show an example. Below, we have a slab that may be a support for a four leg tower. The tower has both dead loads and wind loads applied. If we view a load combination with wind and dead load on it it may look something like this:
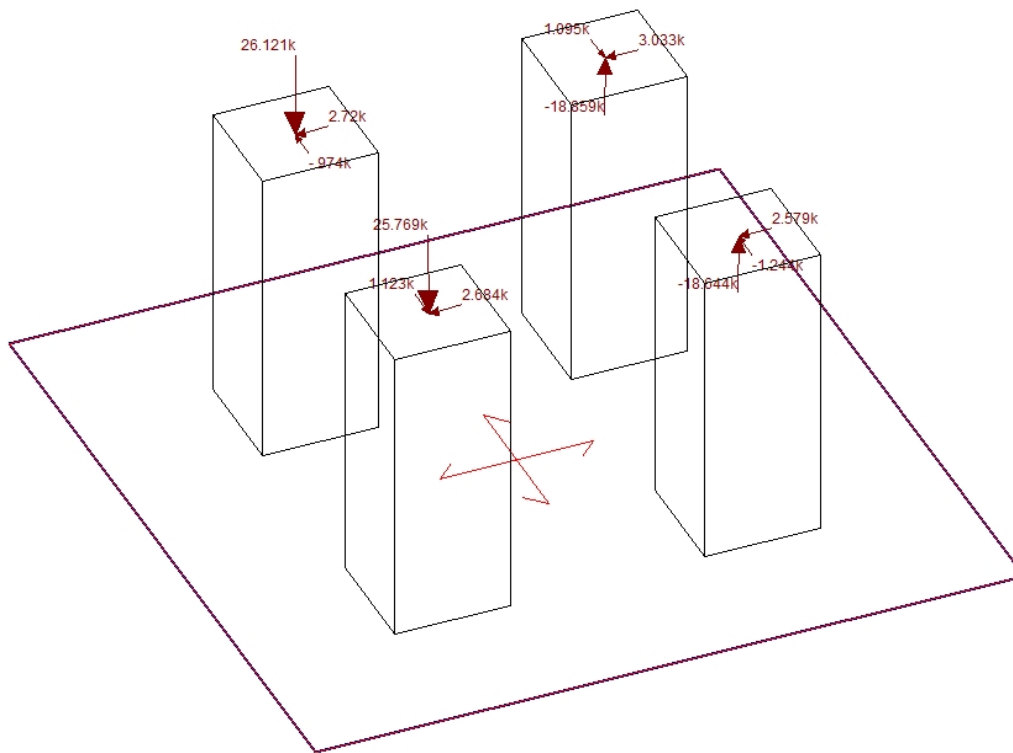
If we next look at the model by load category we'll see something like this:

DL Category   WL Category
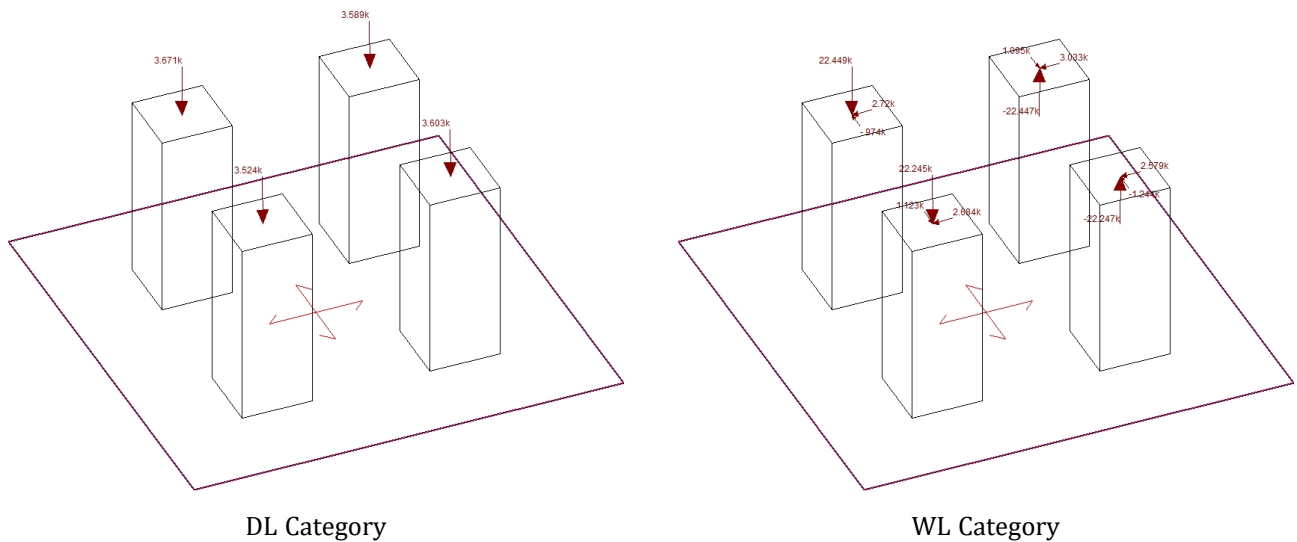
When coming up with overturning and sliding demand/resistance, take the contribution of *EACH category separately*. This is even more important when you have more load categories. Here, if attempting to verify the program, you would take all of the loads in the DL category and sum their effects for both overturning and sliding. From this you can see whether there is a net overturning/sliding force or a resistance force. This would then determine whether this value ends up in the demand or resistance column. You would do the same process for all load categories in a given load combination. Once you've done that you sum up each column (demand vs resistance) and these are the values you'll find in the program.

Overturning

The slab overturning checks are presented in the **Overturning Safety Factors** spreadsheet.

This spreadsheet lists the overturning checks for each slab under each service load combination.

Angle

This is the **Local Axis Angle** from the Slabs spreadsheet. This just defines the local axis of the slab, which defines the orientation of the sliding and overturning checks.

Overturning Moments

The **Mo-xx** and **Mo-zz** columns represent the overturning moments about the slab x and z local axes, respectively. The program will consider overturning about all edges of the slab (please see image below); the worst case in each direction is reported in the spreadsheet.
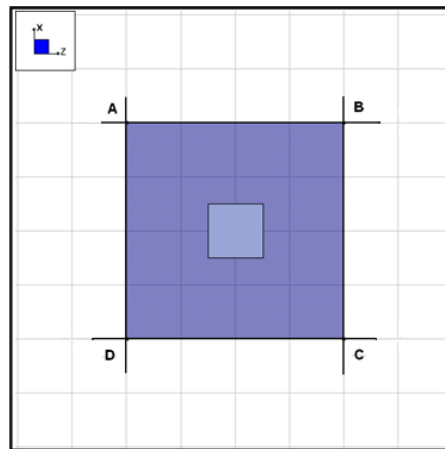
RISAFoundation calculates the moment about each of the four orthogonal lines set at the extreme points of the slab (AB, BC, CD, AD). **Mo-xx** reported in the spreadsheet is the largest moment taken about either AD or BC. **Mo-zz** reported in the spreadsheet is the largest moment taken about either AB or CD.
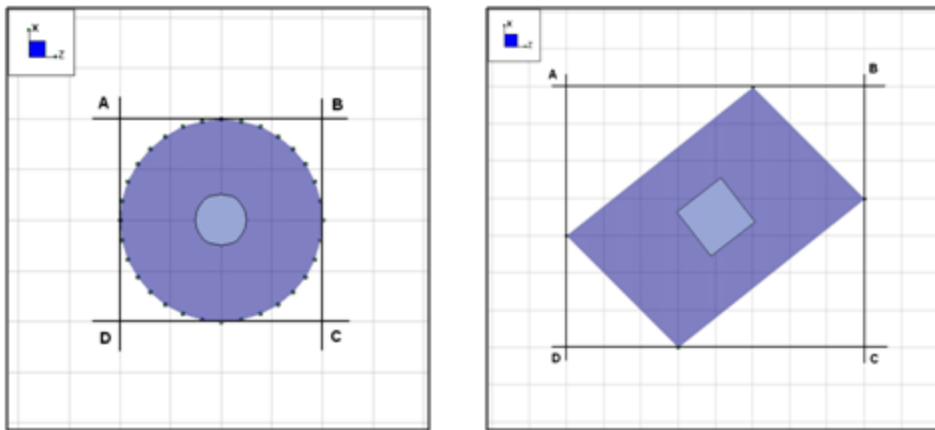
For more complex slabs, RISAFoundation sets four orthogonal lines at the four extreme points of the slab along the xx and zz axes. The program then calculates the moment about each of the four lines (AB, BC, CD, AD). **Mo-xx** reported in the spreadsheet is the largest moment taken about either AD or BC. **Mo-zz** reported in the spreadsheet is the largest moment taken about either AB or CD. These moments are calculated about the bottom face of the slab.

Note:

- For slabs in RISAFoundation the loading is applied to the mid-height of the slab for finite element calculations. However, for slab overturning checks the program will assume the loading is applied at the top of the slab. This affects the overturning when column/wall shear loads are applied to the slab.
- If the **Local Axis Angle** is non-zero the images above would be skewed at that angle.

Resisting Moments

The **Ms-xx** and **Ms-zz** columns represent the resisting moments about the slab local x and z axes, respectively. The program will consider all point loads, line loads, and area loads in addition to the self weight in the calculation of the moments. The soil overburden pressure in the Slabs spreadsheet is also considered to resist overturning. These moments are calculated about the bottom face of the slab.

Note:

- If there are piles defined on the slab, they will not affect these overturning calculations.

Ratio
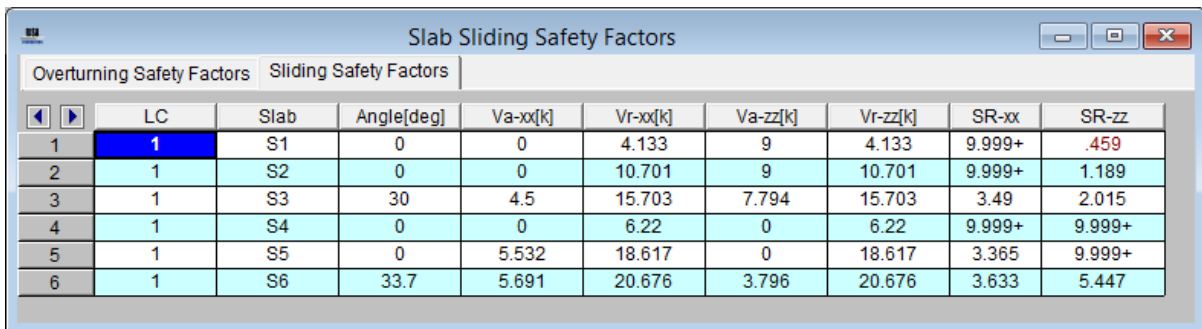
The **Ms-xx/(Mo-xx)** and **Ms-zz/(Mo-zz)** columns represent the ratio of resisting moment to overturning moment. This is then compared against the Overturning Safety Factor, **OSF**, value set in the Load Combinations spreadsheet. If this value is lower than the OSF then the value in the spreadsheet will be in red and your slab fails the overturning check.

Limitations

- Currently the program does not consider whether an individual slab is connected to other elements in the model. The program simply calculates the overturning and resisting forces it is seeing on that slab to determine the safety factor. The slab may, however, be connected to grade beams, pile caps, footings or other slabs that would prevent overturning of the slab from actually occurring.
- The overturning is reported for each individual slab element. If a system of slabs is connected to one another each overturning check will occur individually and the overturning of the overall group is not considered.

Sliding

The sliding check is done along both the slab local x and z axes and the results are presented in the **Sliding Safety Factors spreadsheet**.

| LC	Slab	Angle[deg]	Va-xx[k]	Vr-xx[k]	Va-zz[k]	Vr-zz[k]	SR-xx	SR-zz
1	S1	0	0	4.133	9	4.133	9.999+	.459
1	S2	0	0	10.701	9	10.701	9.999+	1.189
1	S3	30	4.5	15.703	7.794	15.703	3.49	2.015
1	S4	0	0	6.22	0	6.22	9.999+	9.999+
1	S5	0	5.532	18.617	0	18.617	3.365	9.999+
1	S6	33.7	5.691	20.676	3.796	20.676	3.633	5.447

This spreadsheet lists the sliding checks for each slab under each service load combination.

Angle

This is the **Local Axis Angle** from the Slabs spreadsheet. This just defines the local axis of the slab, which defines the orientation of the sliding and overturning checks.

Applied Forces

The **Va-xx** and **Va-zz** columns represent the applied lateral force that causes sliding along the slab local x and z axes, respectively.

Resisting Forces

The **Vr-xx** and **Vr-zz** columns represent the resisting lateral forces that counteract sliding along the x and z axes, respectively. These loads include applied normal loads and self weight, and is multiplied by the **Coefficient of Friction** set in the Solution tab of **Model Settings**. The passive pressure is also considered as a sliding resisting force in both the slab's local z and x directions.

Note:

- The active and resisting sliding forces will be shown as an absolute value.

Sliding Ratio (SR)

The **SR-xx** and **SR-zz** columns represent the ratio of resisting forces to applied forces. If this value is lower than 1.0 then your slab is subject to sliding.

Limitations

- Currently the program does not consider whether an individual slab is connected to other elements in the model. The program simply calculates the sliding and resisting forces it is seeing on that slab to determine the safety factor. The slab may, however, be connected to grade beams, pile caps, footings or other slabs that would prevent sliding of the slab from actually occurring.
- The sliding is reported for each individual slab element. If a system of slabs is connected to one another each sliding check will occur individually and the sliding of the overall group is not considered.

Overturn/ Resist by Category

The Overturn/ Resist by Category tab of the Safety Factors spreadsheet shows a breakdown of the overturning and resisting categories for each slab and load combination. Each category's X, Y and Z components are considered to act as either an overturning or resisting value. This category tab shows how RISA comes up with the net category value for each load combination in the Overturning Safety Factors tab.

In the first tab for the **Overturning Safety Factors** , RISAFoundation takes the net sum of the overturning and resisting values for each category per the True Safety Factor Method. Each category will either net a resisting or an overturning value in the slab's local xx and zz directions. The net overturn or resisting value for each category is combined with all load categories in a given load combination to come up with the Safety Factor result in each direction. It is important to assign the categories properly for your model to get correct safety factor results in the program.

Note:

- The results in the overturning/ resist by category tab gives the user the ability to take the overturning and resisting components of each category and calculate the results outside of the program using a traditional/ simplistic method if they choose.

Sliding/ Resist by Category

The Sliding/ Resist by Category tab of the Safety Factors spreadsheet shows a breakdown of the sliding and resisting categories for each slab and load combination. Each category's X, Y and Z components are considered to act as either a sliding or resisting value. This category tab shows how RISA comes up with the *net* category value for each load combination in the Sliding Safety Factors tab.

In the second tab for the **Sliding Safety Factors** , RISAFoundation takes the net sum of the sliding and resisting values for each category per the True Safety Factor Method. Each category will either net a resisting or an sliding value in the slab's local xx and zz directions. The net sliding or resiting values for each category is combined with all

load categories in a given load combination to come up with the Safety Factor result. It is important to assign the categories properly for your model to get correct safety factor results in the program.

	LC	Slab	Category Description	Va-xx[kN]	Vr-xx[kN]	Va-zz[kN]	Vr-zz[kN]
1	1	S1	DL	0	564.4367	0	564.4367
2	1	S1	OL4	-50.4	3.3	57.9	3.3
3	1	S1	WL+Z	0	0	49.0016	0

Slab Sliding by Category (By Combination) — tabs: Overturning Safety Factors | Sliding Safety Factors | Overturn/Resist by Category | Sliding/Resist by Category

Note:

- The results in the sliding/ resist by category tab give the user the ability to take the sliding and resisting components of each category and calculate the results outside of the program using a traditional/ simplistic method if they choose.
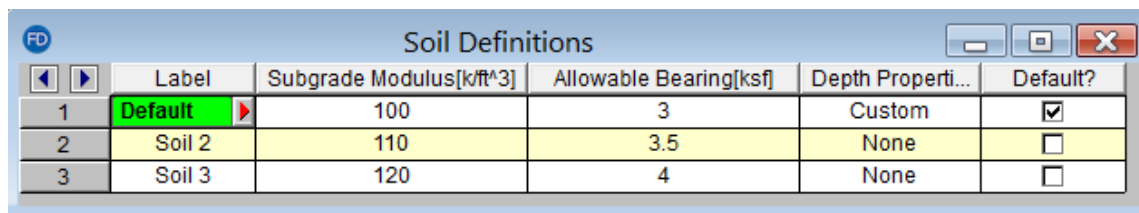
Soils

Assigning Soil Properties

In previous versions, the default **Subgrade Modulus** and **Allowable Bearing** were entered in the Model Settings. In version 10.0 and higher, the default subgrade modulus and allowable bearing pressure are entered in the **Soil Definitions** spreadsheet. There is a check box in the Soil Definitions spreadsheet to determine which soil definition will be used as the default. The default soil definition will be used for the soil properties in the whole model except where another soil region is drawn in the model. It is not necessary to draw a soil region for the default soil definition.

Soil Definitions Spreadsheet

The **Soil Definitions Spreadsheet** records the properties of each region and may be accessed by selecting **Soil Definitions** on the **Spreadsheets** menu.

	Label	Subgrade Modulus[k/ft^3]	Allowable Bearing[ksf]	Depth Properti...	Default?
1	Default	100	3	Custom	☑
2	Soil 2	110	3.5	None	☐
3	Soil 3	120	4	None	☐

The entries are explained below:

Label

You may assign a unique label to each Soil Definition. Each label has to be unique, so if you try to enter the same label more than once you will get an error message. The soil definition with the **default** box checked will be used for the soil properties in the model except where another soil region using a different soil definition is drawn.

Subgrade Modulus

The subgrade modulus is used to establish the elastic stiffness of the soil and foundation. Refer to the section on automatically generating soil springs for more information.

Allowable Bearing Pressure

Enter the gross allowable soil bearing pressure for this soil definition.

Depth Properties

The soil depth properties are used for static pile design of concrete, hot rolled, and wood piles. Clicking on the red arrow in a cell in this column will allow you to open the Soil Depth Properties spreadsheet for that soil definition. The Soil Depth Properties spreadsheet will allow the user to enter the soil layer properties. Each soil definition is associated with its own soil depth properties. Refer to the section on soil depth properties for more information.

Draw Soil Regions

To create a new soil region click ![icon] on the **Drawing Toolbar** to open the Draw Soil Regions dialog. Choose an existing Soil Definition or create a new soil definition by clicking on the [...] button.

You may apply the soil properties by drawing a polygon that defines the area where these properties exist. If two regions overlap, the properties that were drawn most recently will control for the overlapping area.
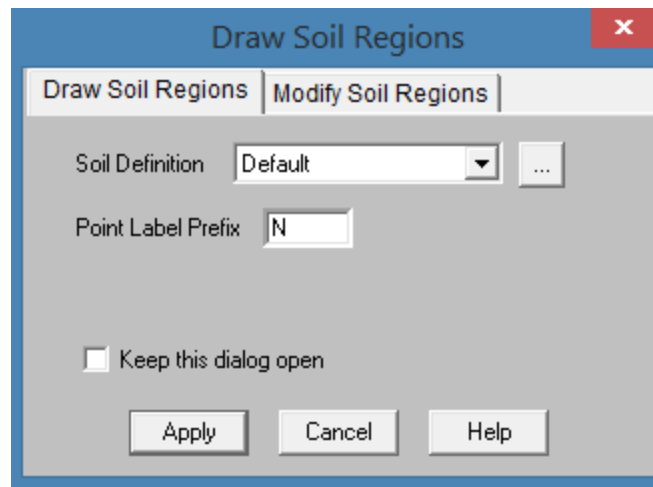
To Assign Soil Properties

1. If there is not a model view already open then click ⊞ on the **RISA Toolbar** to open a new view and click ✏️ to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the 📙 button and set the soil definition and click **Apply**.
3. Start drawing a polygon to define the region that will be controlled by these soil properties. To indicate that you have finished drawing the region double-click on the last node of the polygon.
4. To cancel the and redraw it, click the right mouse button. To cancel drawing altogether, right click the mouse button again, or just press the Esc key.
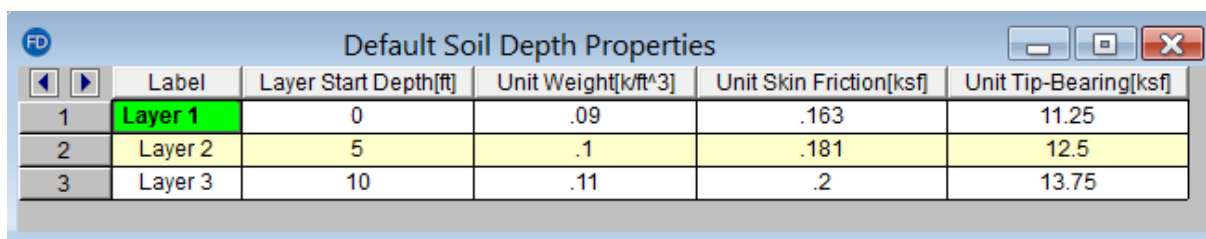
To create a new soil definition from the Draw Soil Regions dialog box, click the ⌶ button. This will open up the **Soil Definition Editor** dialog box where you can create a new soil definition. This dialog box can also be accessed to view existing soil definitions from the Soil Definitions spreadsheet. In that spreadsheet, simply click on a soil label and the red arrow in that cell and the Soil Definition Editor will open showing the inputs for that definition.
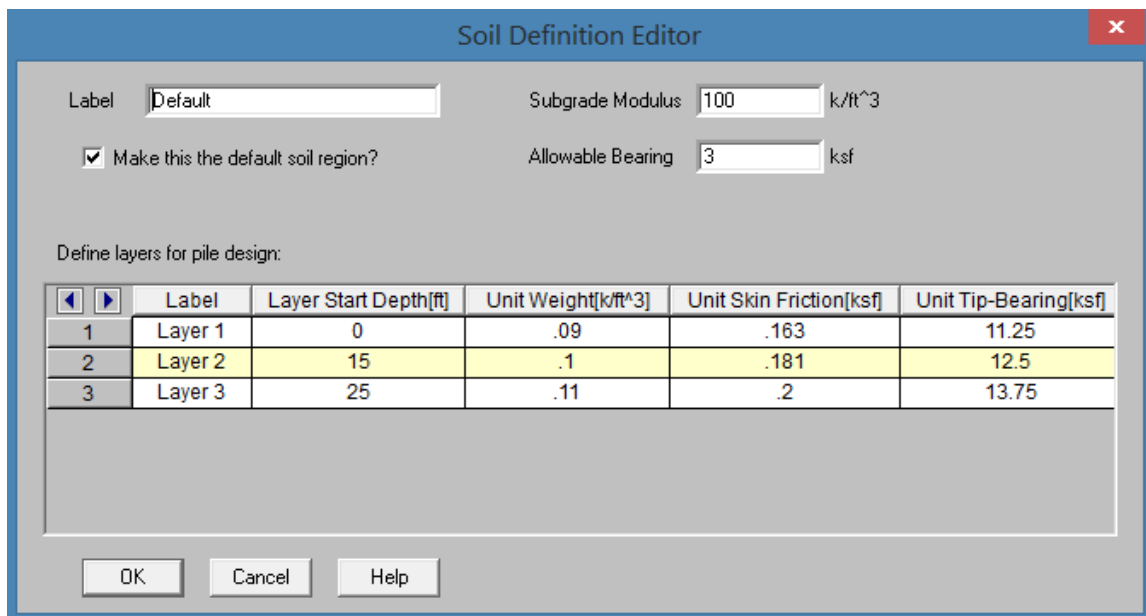
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Varying Soil**.
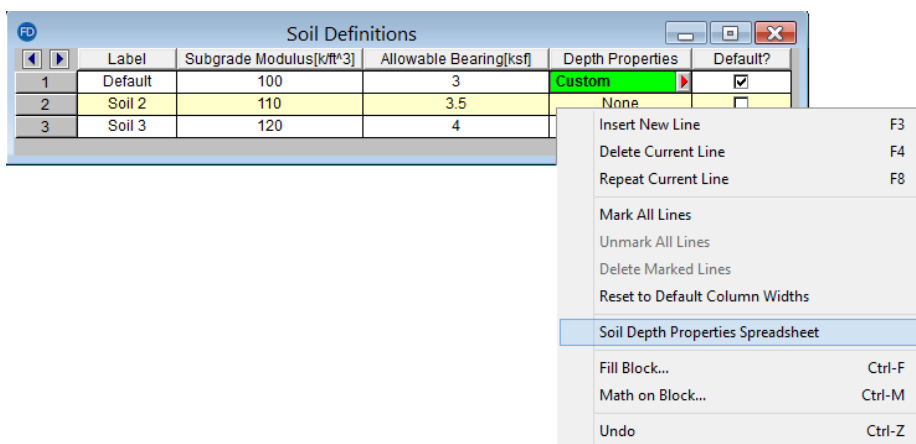
Soil Depth Properties Spreadsheet

The Soil Depth Properties spreadsheet is used to define soil layers for static pile design. It is only necessary to input soil layer data when using concrete, hot rolled, and wood piles.

Open the **Soil Definitions** spreadsheet to access the **Soil Depth Properties** spreadsheet. Click on a cell in the row of the soil definition which you wish to create or edit the soil depth properties. Then right click on the Soil Definitions spreadsheet and select Soil Depth Properties Spreadsheet from the right-click menu. Each soil definition is associated with its own soil depth properties spreadsheet. The soil definition label will be titled next to the soil depth properties spreadsheet which the user is currently editing.

Note:

- The soil depth properties spreadsheet is associated with an individual soil definition. To access the soil depth properties spreadsheet for a particular soil definition, highlight a cell in the row of that soil definition.
- You can also access the Soil Depth Properties spreadsheet by clicking on a cell in the Depth Properties column, then lick on the red arrow in that cell.

The entries in the Soil Depth Properties spreadsheet are explained below:

Label

You may assign a unique label to each layer of soil. Each label has to be unique, so if you try to enter the same label more than once you will get an error message.

Layer Start Depth

The layer start depth is the elevation of the top of the soil layer. The first layer of soil typically has a start depth of zero. Each layer depth of soil is assumed to exist until it reaches the next layer's start depth.

Unit Weight

The unit weight represents the effective unit weight of the soil layer in lb/ft^3 or kN/m^3. The unit weight, γ of a soil layer is typically determined from soil tests measuring the weight and volume of a sample. Typical values of the

unit weight for various soil types could be(from Lindeberg, *Civil Engineering Reference Manual, 11th ed.*) :

Soil Type	γ (lb/ft^3)	γ sat (lb/ft^3)
Sand, loose and uniform	90	118
Sand, dense and uniform	109	130
Sand, loose and well graded	99	124
Sand, dense and well graded	116	135
Glacial Clay, soft	76	110
Glacial Clay, stiff	106	125

Unit Skin Friction

The unit skin friction at each layer is the theoretical unit friction resistance for that layer which can be measured in lb/ft^2 or kN/m^2. There are several methods to calculate this value depending on the engineer's judgment , type of soil, and assumptions for the applicable factors. A few of these equations are listed below (Equations reference: Joseph E. Bowles, *Foundation Analysis and Design, 5th Ed.*):

1. The α Method: $q_s = \alpha c$

- α : adhesion factor
- c : average cohesion

2. The λ Method: $q_s = \lambda(\bar{q} + 2s_u)$

- λ : coefficient dependent on pile depth
- q : effective overburden pressure
- su : undrained shear strength

3. The β Method: $q_s = \beta \bar{q}$

- β : K tan δ
  - K : Earth pressure coefficient
  - δ : undrained friction angle
- q : effective overburden pressure

Unit Tip-Bearing

The unit tip-bearing is the unit tip or point bearing resistance for each layer of soil which can be measured in lb/ft^2 or kN/m^2. The unit tip-bearing resistance will be used to calculate the ultimate bearing capacity for a pile when the length of the bottom of the pile terminates in that layer. There are several methods to calculate this value depending on the engineer's judgment, type of soil, and assumptions for the bearing capacity factors. A few equations are listed below to calculate the unit tip bearing capacity (Equations reference: Joseph E. Bowles, *Foundation Analysis and Design, 5th Ed.*) :

1. Unit Point Resistance: $q_p = cN'_c d_c s_c + \eta \bar{q} N'_q d_q s_q + \frac{1}{2}\gamma' B_p N_\gamma s_\gamma$

- c : Cohesion of soil beneath pile point
- Bp : width of diameter of pile point
- N'c : bearing capacity factor for cohesion
- N'q: bearing capacity factor

- $N'\gamma$: bearing capacity factor for base width

2. Meyerhof's Equation: $q_p = (40N)\dfrac{L_b}{B}$

- N : statistical average standard penetration number near pile point
- Lb : pile penetration depth into point bearing stratum
- B : width of diameter of pile point

Soil Springs Calculation (Automatic)

The program uses what is commonly referred to as a Winkler foundation model. Essentially, the soil effect is concentrated at the node locations and the soil springs are based on a simple multiplication of Tributary Area of the node and the user entered subgrade modulus.

Adjusting the Subgrade Modulus

Typically, the geotechnical engineer will obtain the subgrade modulus for a 1' by 1' or .3m. x .3m sample plate. Typical values for various soil types could be:

Soil Type	k_s (kcf)
Loose Sand	30 ~ 200
Medium Dense Sand	60 ~ 500
Dense Sand	220 ~ 850
Clayey Medium Dense Sand	200 ~ 500
Silty Medium Dense Sand	150 ~ 300
Clayey Soil	80 ~ 300

For settlement calculations, this value would need to be modified to account for our actual foundation size (e.g. 10 ft by 14 ft). For this example we will use equations from Principles of Foundation Engineering, 3rd edition, by Braja Das, pgs. 263 - 264. We will assume a 1ft x 1ft sample plate with a base ks value of 500 kcf.

$$K_{10x10} = k_1 \left(\frac{B+1}{2B} \right)^2 = 500 \left(\frac{10+1}{2*10} \right)^2 = 151 kcf$$

$$K_{10x14} = K_{10x10} * \left(\frac{1+0.5*B/L}{1.5} \right) = 151 * \left(\frac{1+0.5*10/14}{1.5} \right) = 136.6 \, kcf = 0.07906 \, kip/in^3$$

These are the values that the user must enter for the subgrade modulus. The program does not automatically adjust the modulus according to the size of the foundations.

Once the user has entered in these values for the modulus, the program will automatically calculate the spring stiffness for all the joints in the model based on the tributary area of each node.

Single and Combined Footings

Since RISA optimizes footing dimensions, the final size of the spread footing is not known at the time of the FEM solution. Therefore, the program approximates the size as the average width and length in the user entered range of values.

Unlike large mat foundations, each spread footing is modeled with a single joint. Therefore, that joint needs to have a rotation spring to accurately distribute the loading for a true combined footing. RISAFoundation uses this same

assumptions regarding the modulus of subgrade reaction and the dimensions of the footing when approximating these rotational springs.

Note

- The nodes that define the Footings currently reside at the center of the footing PEDESTAL. This is fine in most cases, but can lead to problems when you have Eccentric footing that are part of a combined footing as shown on the left in the image below. In these cases it is preferable to model these footings with a small mat foundation as shown on the right in the image below.
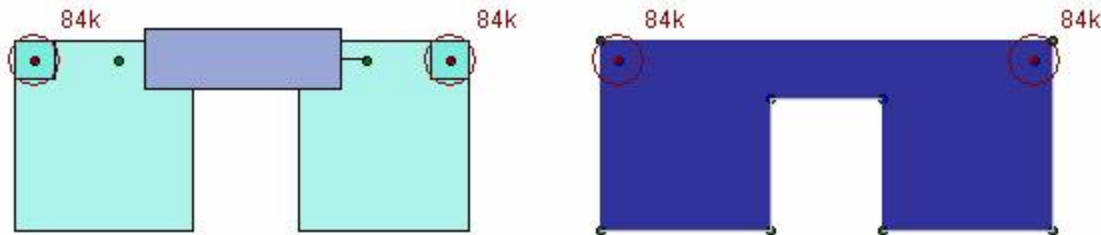
Grade Beams

Grade beams are also looked at as having compression only soil springs along their length. These springs will be equal to the sub-grade modulus times the width of the beam times the tributary length of the beam node.

If it is not desirable to have spring supports along the length of the beam, then a soil region with a very low sub-grade modulus can be locally defined for this region.

Modeling Mat Foundations

Although a little out of date, the ACI publication 336.2R-88, "Suggested Analysis and Design Procedures for Combined Footing and Mats", is a good reference for the modeling of mat foundations. For more information on these suggestion refer to the ACI document directly.

The program uses a "Winkler Foundation" approach to the solution. As such, it does NOT explicitly take into account node coupling effects between soil springs. Therefore, it may be desired to create a "dishing" effect for the mat foundation by adjusting the subgrade modulus for different "zones" of the mat as shown below. This procedure is described in more detail in the ACI report mentioned above.

Soil Pressure Results

RISAFoundation will automatically calculate the soil pressure profile that is induced by each one of your service level load combinations. For spread footings, these results are discussed in detail in the Footing Results section.

There is an **Enveloped** and a **Batch** version of this spreadsheet. The **Enveloped** version will open by pressing the **Soil Pressures** button on the **Results** toolbar.

To access the **Batch** version select it from the **Results** dropdown:
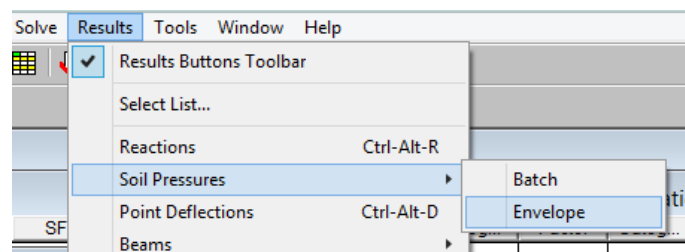
Envelope Soil Pressure Results Spreadsheet

The Envelope Soil Pressures results is a tabulated results spreadsheet which displays summaries of the soil pressure results.

The **Soil Pressures** tab displays the results for every node in the model. It also displays the allowable bearing pressure and the unity check for the load combination that produced the maximum unity check. If the load combination has an ABIF (allowable bearing increase factor) due to the presence of transient loads then this will be reflected in the Allowable Bearing value listed in this spreadsheet, NOT the U/C value. Therefore, a U/C value greater than 1.0 would always indicate a soil failure. A calculated Soil Pressure of NC indicates that a soil pressure was NOT calculated for that joint. This is most likely because that node was a loading node that was not used as part for the FEM solution of the mat foundation.

The **Slab**, **Footing** and **Beam Soil Pressure** tabs display summaries of each of the foundation elements in the model along with the point or location which controlled the soil bearing code check for that element.

Plotting the Soil Pressure Results

The **Soil Pressure** results may be plotted using the setting under Model Display Options - Slabs. An example of this plot is shown below. This plot shows a color contoured view of the soil pressures as they exist under the slab.
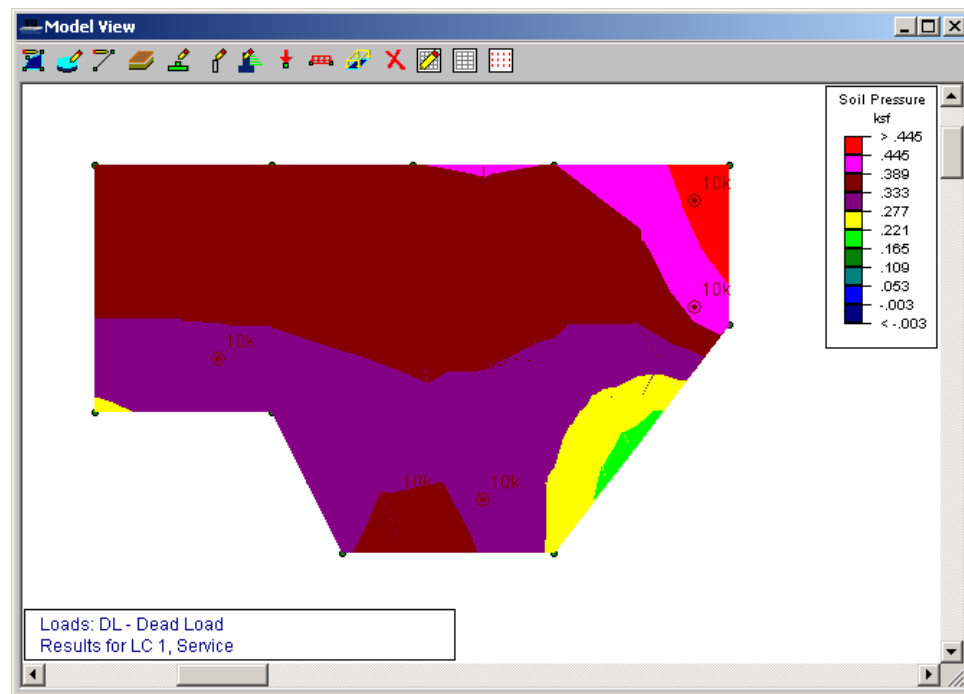
This view does note show if the soil is exceeding the allowable soil pressure. If this is desired, the user can specify a user defined Range of values to plot. Any values above the range would display in red and would indicate a failure. These specific Model Display Options are discussed in more detail in the Model Display Options section.

Note

- While it is the Slab Contour controls that are used to display the soil pressure results, the program will also display the soil pressure contours that exist underneath grade beams and spread footings.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Soil Bearing**.

Solution

To solve the model select **Solve** on the **Main Menu** or click the ☰ button on the **RISA Toolbar**. A progress bar displays information as the solution proceeds. In general, loads are attributed to the slabs, beams, and footings followed by an optimization of the slab reinforcement, as well as beam and footing sizes and reinforcement. Any warnings are listed in the **Warning Log** which can be opened from the **Spreadsheets Menu**. See Design Optimization for more information on how slabs and beams are optimized. See Footings - Results for more information on footing optimization.

Solution Method

The solution is based on load categories and load combinations using the widely accepted linear elastic stiffness method for solution of the model. The stiffness of each element of the structure is calculated independently. These element stiffnesses are then combined to produce the model's overall (global) stiffness matrix. This global matrix is then solved versus the applied loads to calculate point deflections. These point deflections are then used to calculate the individual element stresses. The primary reference for the procedures used is **Finite Element Procedures**, by K. J. Bathe (Prentice-Hall, 1996).

When a solution has been performed and the results are available the "S" icon on the status bar in the lower left corner will change from ⑤ to ⑤. If any changes are made to the model that invalidate the results then the results will be cleared and another solution will be necessary.

The specific type of solution performed by RISAFoundation is **Batch Solution**. This means that a solution may be performed on multiple combinations and the results retained for **each** solution. You may group the results by item or by load combination in the results spreadsheets by choosing from the **Results Presentation** options on the **Results Menu**. For example, you can have all the combination results for beam M1 grouped together or you can have all the beam results for Load Combination "DL+LL" grouped together. See Load Combinations to learn how to mark combinations to be solved in the batch solution.

When you save a file that has been solved, you may also save your results. The next time that file is opened the saved results will be opened as well. You may use the Application Settings on the Tools Menu to change the way that you are prompted to save results. See Application Settings for more information.

Static Solutions / Stiffness Matrix

Static solutions are based on load combinations and may be performed on any defined load combinations. When a static solution has been performed and the results are available the "S" icon on the status bar in the lower left corner will change from ⑤ to ⑤. If any changes are made to the model that invalidate the results then the results will be cleared and another solution will be necessary.

The solution is based on the widely accepted linear elastic stiffness method for solution of the model. The stiffness of each element of the structure is calculated independently. These element stiffnesses are then combined to produce the model's overall (global) stiffness matrix. This global matrix is then solved versus the applied loads to calculate joint deflections. These joint deflections are then used to calculate the individual element stresses. The primary reference for the procedures used is **Finite Element Procedures**, by K. J. Bathe (Prentice-Hall, 1996).

Skyline Solver

This solution method is also sometimes referred to as an Active Column solution method. In finite element analysis, the nonzero terms of the stiffness matrix are always clustered around the main diagonal of the stiffness matrix. Therefore, the Skyline or Active Column solutions take advantage of this by condensing the stiffness matrix to exclude any zero stiffness terms that exist beyond the last non-zero term in that column of the matrix.

Since the majority of terms in a stiffness matrix are zero stiffness terms, this method greatly reduces the storage requirements needed to store the full stiffness matrix. However, for large models (+10,000 joints) , the memory requirements even for a skyline solution can be problematic.

This solution method has been used successfully in RISA for more than 20 years, and has proven its accuracy continuously during that time.

Sparse Solver

The skyline solver described above is moderately efficient because it only stores and performs operations on the terms within the "skyline" of the stiffness matrix. However, that solution still contains a great number of zero stiffness terms within the skyline of the matrix. A Sparse Solver will reduce the matrix size to an absolute minimum by eliminating the storage of ALL zero stiffness terms.

A sparse solver is the most efficient solution methodology possible because it stores and performs operation only on the non-zero terms of the stiffness matrix. For this reason the sparse solution is preferred from both a solution speed and memory requirement standpoint.

Note

- The skyline solver is retained mostly for comparison and verification purposes.
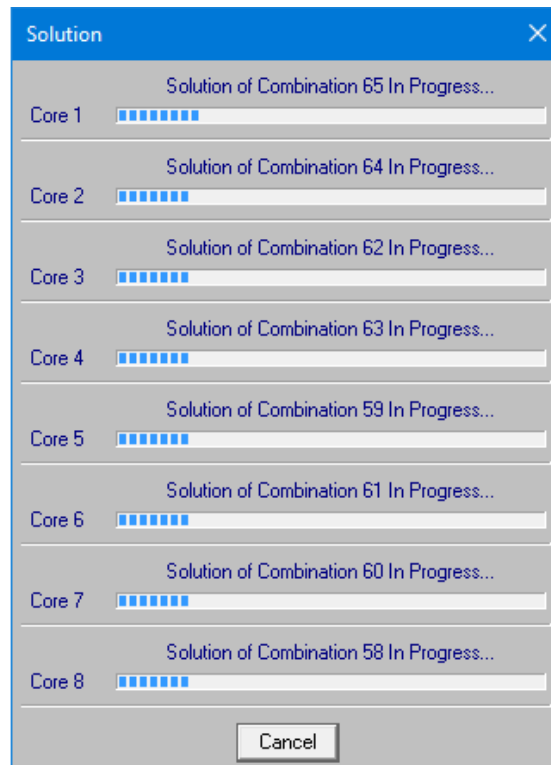
₀ Spreadsheet Operations

Powerful spreadsheets may be used to view, sort and edit the input data and results. The spreadsheets and model views are synchronized, unless you turn this feature off in the **Application Settings** on the **Tools Menu**. As you edit a model graphically the spreadsheets are automatically updated and as you make changes in the spreadsheets the model views reflect these changes immediately.

The input data may be accessed from the**Spreadsheets Menu**. You may edit the data or you can add new data. You can also paste data from another application directly into the spreadsheet via the Windows clipboard. Any changes made to the input spreadsheets may also be viewed graphically.

After solving the model, results are recorded in spreadsheets for browsing. These spreadsheets may be accessed from the **Results Menu**. You may sort the results in order to find maximums and exclude data that is not important. You may also copy this data to the Windows clipboard and use it in another application.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Advanced Spreadsheet**.

Moving and Scrolling

To move between cells in a spreadsheet, click any cell or use the keyboard arrow keys. When you move to a cell it becomes the active cell. To see a different area of the spreadsheet use the vertical scroll bars on the right side of the spreadsheet.

To Scroll	Do This
One row up or down	Click the arrows on the vertical scroll bar.
One column left or right	Click the arrows on the horizontal scroll bar.
One page up or down	Click above or below the scroll box in the vertical scroll bar.
One page left or right	Click to the left or right of the scroll box in the horizontal scroll bar.
A large distance	Drag the scroll box to the approximate relative position.

Note:

- The size of a scroll box indicates the proportional amount of the spreadsheet that is visible. The position of a scroll box indicates the current location relative to the spreadsheet.
- The mouse wheel is also an excellent tool for scrolling up and down in spreadsheets or graphics. Roll the wheel or click and drag to move around.

Spreadsheet Keyboard Commands

The following keyboard commands are available:

Key	Function
Arrow Keys	Move the active cell one location

Key	Function
TAB	Move right one cell
ENTER	Move to the first column of the next line. Adds new line if necessary.
PAGE UP	Move the active cell one full page up.
PAGE DOWN	Move the active cell one full page down.
HOME	Move to the first line of the spreadsheet.
END	Move to the last line of the spreadsheet.
F3	Insert new line below current line.
F4	Delete current line.
F8	Insert new line below current line and repeat the current values in the new line.

Selecting Spreadsheet Cells

Before you can carry out commands or tasks in a spreadsheet, you must select the cells that you want to work with.

To Select	Do This
A single cell	Click the cell, or press the arrow keys to move to the cell.
A range of cells	Click the first cell of the range, and then drag to the last cell.
An entire row	Click the row heading.
An entire column	Click the column heading.
Adjacent rows or columns	Drag across the row or column headings

Note:

- To cancel a selection of cells, click any single cell in the spreadsheet.

Undoing Operations

RISAFoundation provides you with virtually unlimited **Undo** capability so that you may easily correct mistakes or just back up to try different possibilities. Simply click the Undo button ↶ as many times as you wish to go back a step. The model view and the spreadsheets will visually display the "undoing". Remember that graphic edits are undone as well.

Redoing Operations

RISAFoundation provides you with virtually unlimited **Redo** capability so that you may reapply actions that were previously undone. Simply click the **Redo** button ↷ as many times as the Undo button was used just prior. The model view and the spreadsheets will visually display the "redoing". Remember that graphic edits are redone as well.

Editing Spreadsheets

The spreadsheets have been specifically developed for the input and editing of structural models. There are many ways to edit the spreadsheets allowing you to quickly build your model. You may copy and move data from other locations or other files. You may also fill large blocks of cells automatically and perform math on these cells.

To Fill Selected Cells

1. Select the cells you wish to fill.
2. Click the **Block Fill** 🔲 button.
3. Type the value you wish to apply.
4. Click **OK**.

To Perform Math on Selected Cells

1. Select the desired cells.
2. Click the **Block Math** 🔲 button.

3. Specify the value and the operation you wish to apply.

4. Click **OK**.
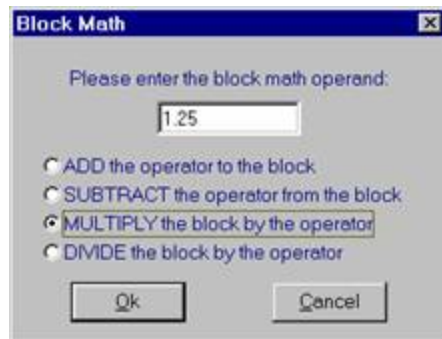
To Cancel or Undo Editing

- To cancel an entry before you press ENTER, press **ESC**.
- To undo a completed entry, click the **Undo** 🔄 button.

To Edit Cell Contents

1. Click the cell that contains the data you want to edit.
2. Make any changes to the cell contents.
3. To enter your changes, press **ENTER**.
4. To cancel your changes, press **ESC**.

Note

- To replace the contents of a cell, click on the cell and type the new entry.
- To edit the contents of a cell, double-click on the cell and use the arrow keys to locate the cursor and perform the edit.

Inserting, Deleting and Clearing Cells

To insert cells you must insert an entire row. When you delete cells you must delete an entire row. The row is removed from the worksheet and shifts the lower cells to fill the space. When you clear cells, you remove the cell contents but leave the blank cells on the spreadsheet.

To Insert Rows

1. Click a cell in the row immediately above where you want the new row.

2. To insert a blank line, click the **Insert New Line** ![icon] button.

3. To have the values of the current line copied in the new line, click the **Repeat Current Line** ![icon] button.

To Clear Cell Contents

1. Select the cells, rows, or columns you want to clear and press the **DELETE** key.

To Delete Rows

1. Select the rows you want to delete.

2. Click the **Delete Marked Lines** ![icon] button and the lower cells will shift up to fill the space.

Moving and Copying Cell Contents

Standard Windows copy and paste functions are fully supported. By using the clipboard you may move or copy any spreadsheet data to another location in the same spreadsheet, or to another spreadsheet (provided it fits). You may also copy data to or from another model or any other application that supports copy and paste such as a word processor or spreadsheet.

To Move or Copy Cells

1. Select the cells you want to move or copy.
2. To copy the cells select **Copy** on the **Edit Menu**.

3. Click on the cell you wish to place the data in and click the **Paste** ![icon] button.

To Move or Copy Cells between Existing Cells

1. Select the cells you want to move or copy.
2. To copy the cells select**Copy** on the **Edit Menu**.

3. Select a cell on the row above where you want to place the data.

4. Click the **Insert New Line** ![icon] button for each new line that is needed.

5. Click on the upper left cell you wish to place the data in and click the **Paste** 📋 button.

To Move or Copy Cells to Another File

1. Select the cells you want to move or copy.
2. To copy the cells select**Copy** on the **Edit Menu**.

3. Open the file you wish to copy the data to.

4. Click on the cell you wish to place the data in and click the **Paste** 📋 button.

Note:

- Your data stays in the clipboard until you copy new data. You may repeat step 4 to move or copy data to multiple locations.
- When you copy information a dialog will come up asking if you want to copy the spreadsheet headers as well. This is helpful if you are copying information to a spreadsheet-type program. See the Customizing RISA topic for more information on this.
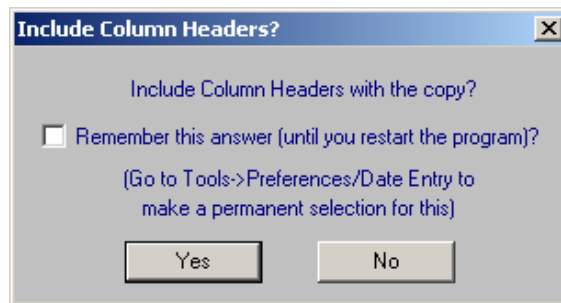
Sorting and Finding in Spreadsheets

You may sort spreadsheets by the values in most any column. Simply click in the column you wish to sort, click on the **Sort** button and choose the sorting method.

You may relabel the points after sorting them by using the options in the **Right Click Menu**.

To locate or find a specific point, beam, or slab while in a spreadsheet, click the **Find** button on the **Window Toolbar**.

Default Spreadsheet Data

Many of the spreadsheets provide the option to save the current data as the default and every new file subsequent to the save will already have that data. This way the office standards that you might use in most of your models are already entered and available in new models. This feature is available in the following spreadsheets: **Materials**, **Design Rules**, **Footing Definitions**, and **Load Combinations**.

To save default data simply click the **Save as Defaults** [icon] button when you are ready and the current data in the spreadsheet will be used in each new file that you create. The data is saved for the current, active, spreadsheet only and affects no other open or closed spreadsheets.

Export to Excel Spreadsheet

For all Results spreadsheet, the program is capable of exporting them into Excel Spreadsheet.

The process is as follows:

1. Right Click and click Export to Excel option.
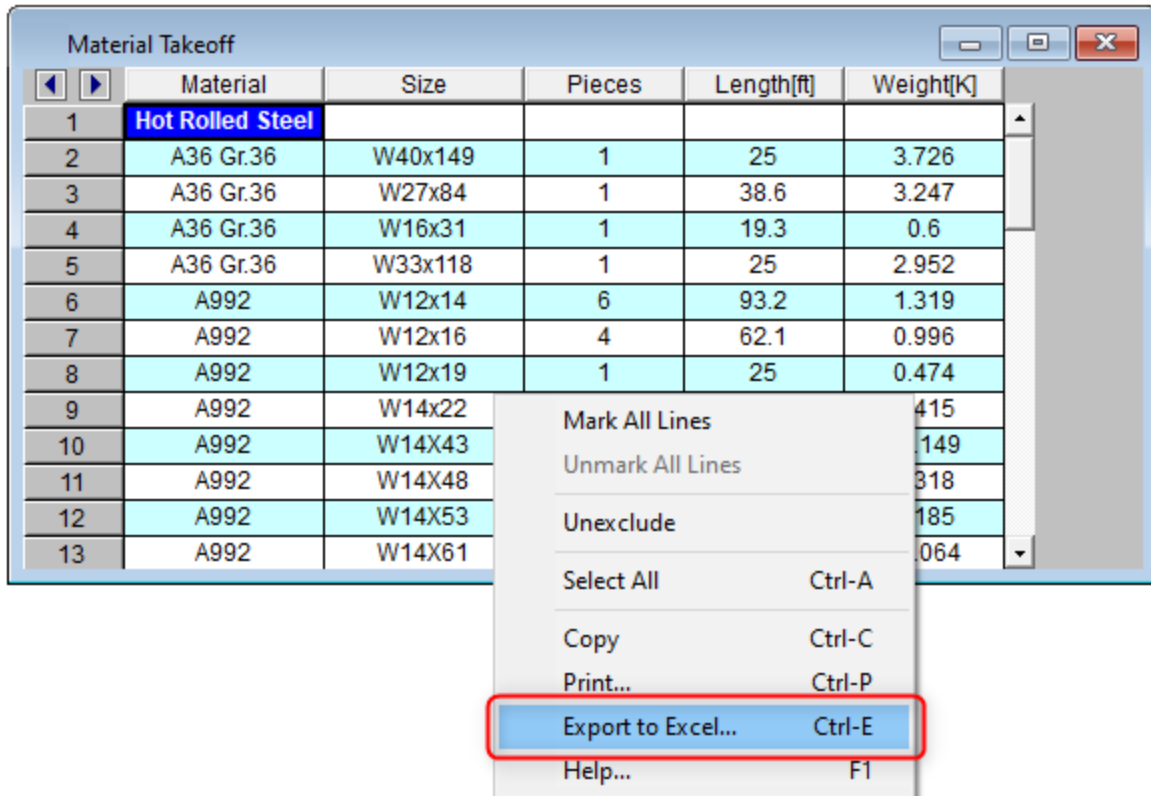
2. Choose if you want to export the current spreadsheet tab or all tabs.
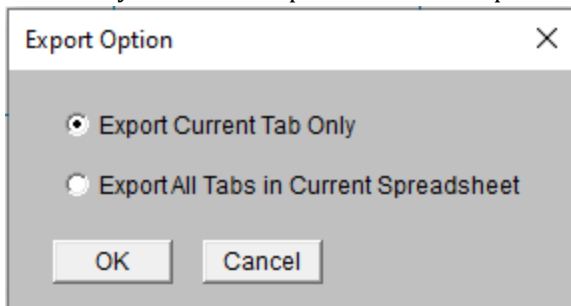
3. Select Excel file name and file save location and export.

Note: The Export to Excel can also be used by clicking the button on ribbon [Export to Excel].

Special Spreadsheet Functions

There are special functions that help you with a particular spreadsheet.

Load Combinations Spreadsheet

You may click [LC Generator] button to have the program generate load combinations based on a variety of Building Codes. See Generating Building Code Combinations for more information.
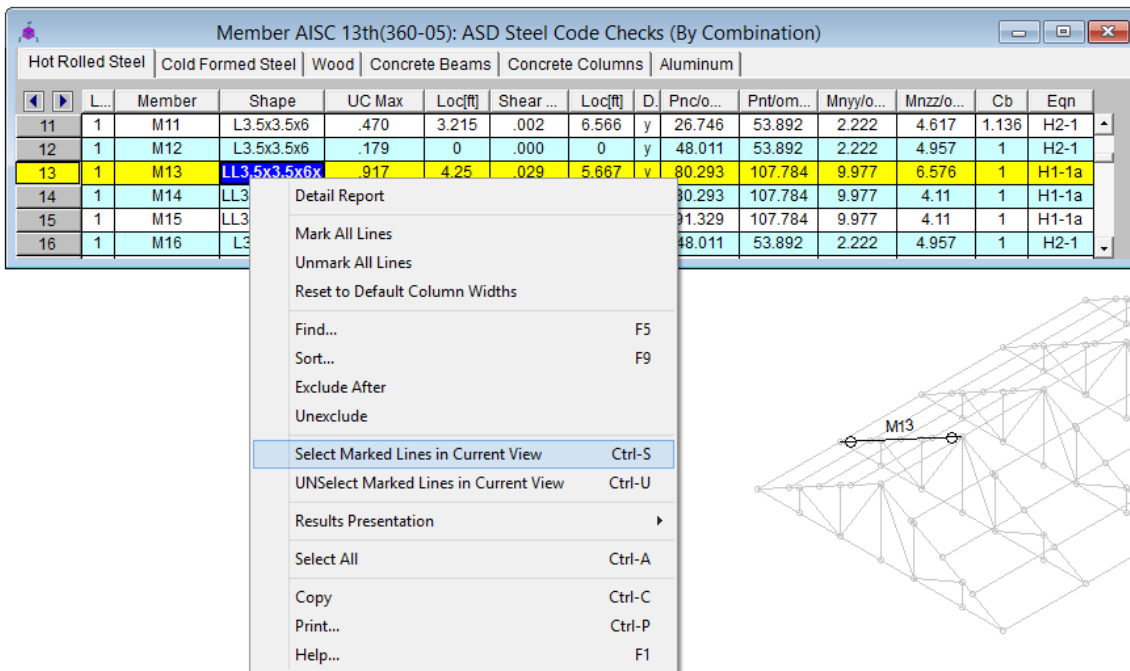
Selecting Items in the Model View

In certain results spreadsheets there is the capability to have elements selected graphically.

The process is as follows:

1. Unselect the entire model.
2. Open the desired results spreadsheet.
3. Highlight in Yellow the lines you wish to view graphically, by clicking on the number of those lines on the far left.
4. Click the **Select Marked Lines in Current View** icon [icon].
5. Only those elements highlighted will be selected graphically.

Note: The inverse process can be used to Unselect Marked Lines using the [icon].

Supports

Supports define how the model is externally constrained. The soil regions you define create a grid of compression-only springs to support your slabs/footings. You may define additional points of support as completely restrained or as partially restrained with a **Spring**. You can also define a spring support that has stiffness in only one direction with tension-only or compression-only springs.

Points of support may also be defined as **Footings, Piles or Pile Caps**. This section will deal primarily with non-footing points of support. See Footings - Design for more information on footings and Piles and Pile Caps for more information on these supports.

Creating and Modifying Supports

There are a number of ways to create or modify supports. You may view and edit the data in the **All Support Types Spreadsheet** or you can use the **Assign Support Type at Points** tool to graphically assign or modify a possibly large selection of supports.

Assign Support Type at Points

The graphical **Assign Support Type at Points** tool discussed here lets you specify and modify supports graphically. To use this, you will typically specify the new support, then select the points that you want to assign or modify.

You can modify or assign points one at a time by selecting the **Apply by Clicking/Boxing** option and then click on the points you wish to modify. You may also modify or assign entire selections of points by selecting the points first and then use the **Apply to All Selected** option.

The parameters shown below are the same as those on the **All Support Types Spreadsheet** and are described in Support Options. Use the arrow buttons to select the support type.

To Assign Support Types

1. If there is not a model view already open, click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.

2. Click the **Boundary Conditions** button on the **Home** tab and define the support.

3. You may apply the support by choosing points on the fly or apply it to a selection of points.

   To choose points on the fly, choose **Apply Entries by Clicking/Boxing Points** and click **Apply**. Click/Box the joints with the left mouse button.

   To apply the support to a selection of points, choose **Apply Entries to All Selected Points** and click **Apply**.

Note

- To apply more supports with different conditions, press CTRL-D to recall the **Assign Support Types Dialog**.
- You may also specify or edit supports in the **Assign Support Types Spreadsheet**.
- You may undo any mistakes by clicking the **Undo** button.
- If drawing a pile cap you can click the button to add a new Pile Cap Definition. See the [Piles and Pile Caps](#) topic for more information.

Auto-Generated Soil Springs

When a solution is performed, RISAFoundation auto-meshes each slab into a mesh of plate elements. In the process, points are automatically created to define the corners of each plate. A **Soil Spring**, or compression-only spring, is auto-generated at every plate corner point, in each slab. User-defined supports previously applied to any point within a slab will override any soil springs generated by the program. In other words, a soil spring will NOT be applied at those particular points in the slab.

RISAFoundation calculates the tributary area for each plate corner point individually and multiplies that area by the **Subgrade Modulus** defined in each particular **Soil Region** to determine the spring stiffness for the soil spring at that point. If no soil regions are defined, the subgrade modulus defined on the **Solution** tab of the **Model Settings Dialog** will be used in this calculation. See [Soil Springs Calculation](#) for more information.

All Support Types Spreadsheet

The **All Support Types Spreadsheet** records the supports for the points and may be accessed by selecting **All Support Types** on the **Spreadsheets Menu**.

	Point Label	Support[k/in]	Footings/Piles/Pile Caps	Support Angle(deg)
1	N3	Reaction		N/A
2	N4	Reaction		N/A
3	N5	Pile 1	Pile 1	N/A
4	N6	Pile 1	Pile 1	N/A
5	N7	Pile 1	Pile 1	N/A
6	N8	Pile 1	Pile 1	N/A
7	N9	Pile Cap 2	Pile Cap 2	60
8	N10	Pile Cap 2	Pile Cap 2	0

The **Point Label** column contains the label of the point that is restrained.

The **Support** column records the support conditions that apply to the point. See the **Support Options** listed in the next section for more information on the available choices.

Note:

- You can only manipulate supports in this column if they are simple point supports. Any footings, piles or pile caps are manipulated in the other columns.

The **Footings/Piles/Pile Caps** column records the support label when a support is assigned to a particular point. The available choices are listed from the **Footing Definitions**, **Pile Definitions** and **Pile Cap Definitions**

Spreadsheets, respectively. If no support is desired at the point, select 'No Support' from the drop down list and the cell will be left blank in the spreadsheet. If a footing label is selected, the Support entry will default to 'Calc'ed'.

The **Support Angle** column records the angle of rotation of each footing, pile or pile cap support about a plane defined by the vertical and Z direction. Supports can be rotated between -180 and 180 degrees. A positive angle will rotate clockwise and a negative angle counterclockwise.

Support Options

Free points have no restraint in any of the degrees of freedom and need not be listed on the **All Support Types Spreadsheet**. The following are the valid support options that may be used.
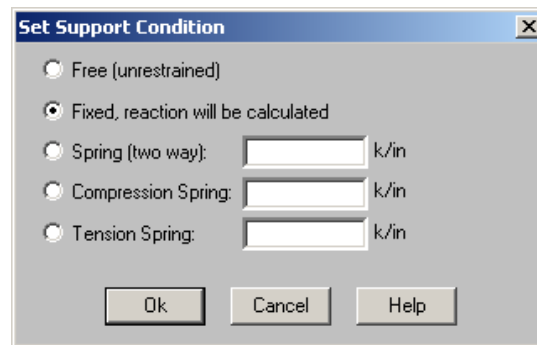
Note

- These options are only available for points not defined by a footing, pile or pile cap.
- Models that contain compression-only or tension-only springs must be iterated until the solution converges. Convergence is achieved when no more load reversals are detected in the springs. During the iteration process, each spring is checked, and if any springs are turned off (or back on), the stiffness matrix is rebuilt and model is resolved.
- With this iteration procedure it is possible to run into a phenomenon known as "clapping". This occurs when a compression-only spring keeps going between a tension force present (where the spring is removed) to a compression force present (where the spring is re-added). This can be somewhat common in non-linear solutions. If you run into this phenomenon, you may see the program give Error 1162. The fix is to very slightly adjust either the loading or the spring/element stiffness to get the offending spring out of the range where it can bounce back and forth. Another option would be to move the location of the soil spring slightly so it's not right at an inflection boundary.
- You can enter the first letter of the option ("R" for Reaction, "S" for Spring, etc.) rather than typing out the entire code. RISAFoundation fills in the rest automatically. After Tether, the primaryjoint needs to be entered (for example, Tether N2)

Reaction Support

The "R" code, for **Reaction**, specifies full restraint for the vertical direction. No movement will be allowed in the vertical direction, perpendicular to the XZ plane, for this point. Furthermore, the reaction will be calculated at this point.

Spring Support

The "Snnn" code, for **Spring**, models a spring attached to the point in the vertical direction. The "nnn" portion of the code is the numerical magnitude of the springs' stiffness. The appropriate units are shown at the top of the column.

For example, if a spring of stiffness 1000 Kips per Inch were desired in the vertical direction at a particular point, for that point you would enter 'S1000' for the support.

Compression-Only Springs

The "CSnnn" code, for **Compression-Only Springs**, models a one way "compression-only" spring attached to the point in the vertical direction. This spring has stiffness for negative displacements (downward) and NO stiffness for positive displacements (uplift). The "nnn" portion of the code is the numerical magnitude of the springs' stiffness. The spring stiffness units are the same as those for a normal spring. Compression-only springs are used by RISAFoundation when auto-generating soil springs for slabs.

For example, if a compression-only (CS) spring with a stiffness of 500k/in were desired in the vertical direction at a certain point, you would enter 'CS500' for the support.

This means that all displacements at this point in the downward direction will be resisted with a stiffness of 500k/in. However, the point is free to move in the upward direction.

- When a model contains T/C only springs, the program must iterate the solution until it converges. Convergence is achieved when no more load reversals are detected in the T/C only springs. During the iteration process, each T/C only support is checked. If any springs are turned off (or turned back on), the stiffness matrix is rebuilt and model is resolved.

Tension-Only Springs

The "TSnnn" code, for **Tension-Only Springs**, models a one way "tension-only" spring attached to the point in the vertical direction. This spring has stiffness for positive displacements (uplift) and NO stiffness for negative displacements (downward). The "nnn" portion of the code is the numerical magnitude of the springs' stiffness. The spring stiffness units are the same as for a normal spring.

For example, if a tension-only (TS) spring with a stiffness of 500k/in. were desired in the vertical direction at a certain point, you would enter 'TS500' for the support.

This means that all displacements at this point in the upward direction will be resisted with a stiffness of 500k/in. However the point is free to move in the downward direction.

- When a model contains T/C only springs, the program must iterate the solution until it converges. Convergence is achieved when no more load reversals are detected in the T/C only springs. During the iteration process, each T/C only support is checked. If any springs are turned off (or turned back on), the stiffness matrix is rebuilt and model is resolved.

Units

You can work with imperial (kips, inches, etc.) or metric (KN, meters, etc.) units, or any combination of the two. The current units appropriate for each data item are shown at the tops of the data columns in the spreadsheets and with the plot of values in the model view.

You may save any of the units as the default setting so that when you start a new model that information is already there. To do this, simply enter the information that you want to save and check the **Save as Defaults** box and click **OK**.
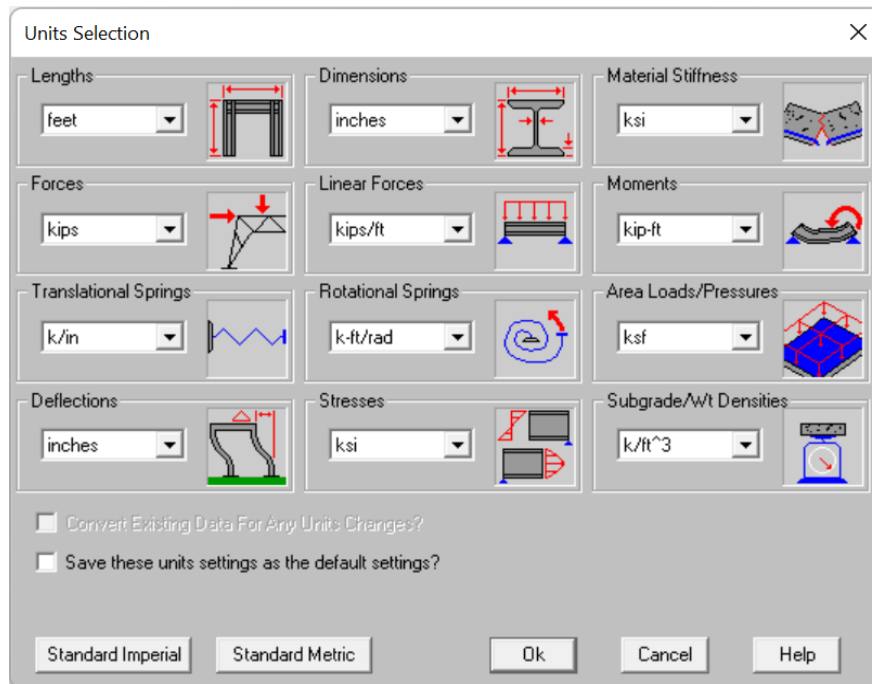
To Change Units

Standard units systems are preset and may be specified by clicking the **Standard Imperial** and **Standard Metric** buttons.

3. If you do not wish to convert values already entered then clear the check box for **Converting Existing Data**.

Standard Imperial Units

This is the units system currently prevalent in the United States. 'Feet' are used for location entries such as point coordinates and load locations, and 'Inches' are used for section property entries such as area and moment of inertia. Force and weight units are in 'kips', where 1 kip = 1000 pounds. Stress units are in 'ksi' (kips per square inch).

Standard Metric Units

This units system uses 'Meters' for location entries and 'Centimeters' for section property entries. Force units are in 'kN' (kiloNewtons), where 1 KN = 1000 Newtons. Stress units are in 'MegaPascals' (MPa), where a MegaPascal is 1,000,000 Newtons per square meter. Weight units are in 'Kilograms' and thermal units are in 'Degrees centigrade.

Units Specifications

The following are the unit specifications and their applications:

Measurement	Usage
Lengths	Coordinates, Load Locations
Dimensions	Shape Properties, PlateSlab Thickness
Material Stiffness	E, G
Weight Densities	Material Density
Forces	Loads, Forces
Linear Forces	Line Loads
Moments	Loads, Forces
Area Loads	Area Loads
Translational Springs	Supports
Rotational Springs	X Rot, Y Rot, Z Rot Boundary Conditions
Temperatures	Thermal Coefficient, Temperatures
Deflections	Deflections, Displacements
Stresses	Fy, Allowable and Actual Stresses

Wall Footings - Modeling

Here we will talk about the steps in modeling a wall footing and get into advanced wall footing modeling considerations. We will also talk about which type of wall footing would be appropriate for your application.

- See the Wall Footing Definitions topic for more information on defining wall parameters.
- See the Wall Footings - Design topic for more information on calculation details.
- See the Wall Footing Results topic for more information on the output.

Wall Footing Types

There are three types of wall footings possible in RISAFoundation: a retaining wall, a strip footing and generated slab/stem wall combination. See the Draw Wall Footings section for how you select each option.

Retaining Wall

This is the traditional retaining wall. This type of wall will perform all stability checks (sliding, overturning and soil bearing) due to hydrostatic pressures and any superimposed load on the wall. This wall type is a single entity in that the wall and footing are considered together and defined in the Wall Footing Definitions spreadsheet.
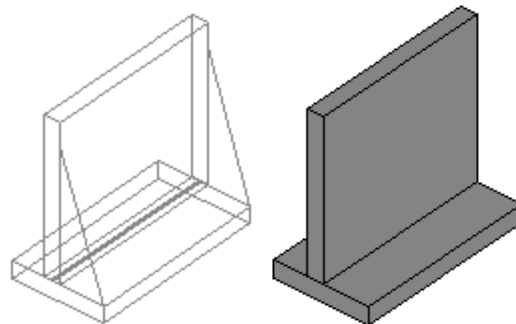
Strip Footing (Gravity)

This is a simplified version of the retaining wall and would be used in cases where you have a simple strip footing layout. Hydrostatic pressures on the wall are not a concern here. This type of wall footing pares down the input and output in the program, removing a lot of the complexity necessary for retaining walls, but not required for a simple strip footing. This wall type would generally be used for simple gravity loads. This wall type is also a single entity in that the wall and footing are considered together and defined in the Wall Footing Definitions spreadsheet.
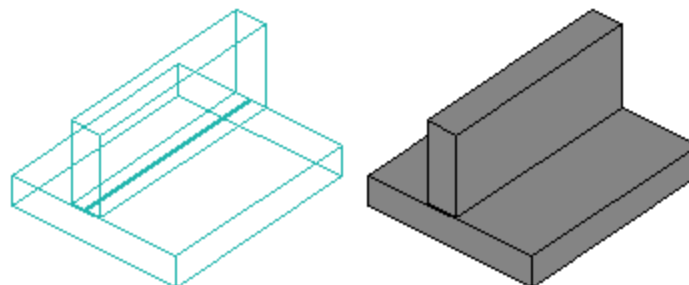
Strip Footing (Lateral/Custom)

This is a wall type that is really a generation utility that will create a slab, slab design strip(s), and a stem wall to represent your wall footing. This type of footing would not be controlled by the Wall Footing Definitions, but rather by each individual entity (slab, slab design strip and stem wall) separately. This option generates these elements one time and then allows you to edit/modify these elements as necessary.

The main purpose of this type of wall is to allow for any customization and to allow the program to perform a finite element analysis of the footing itself. The other two wall types do not perform an FEA on the footing itself.
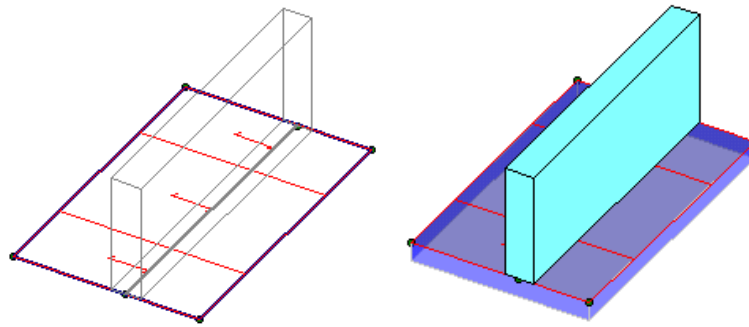
Draw Wall Footings
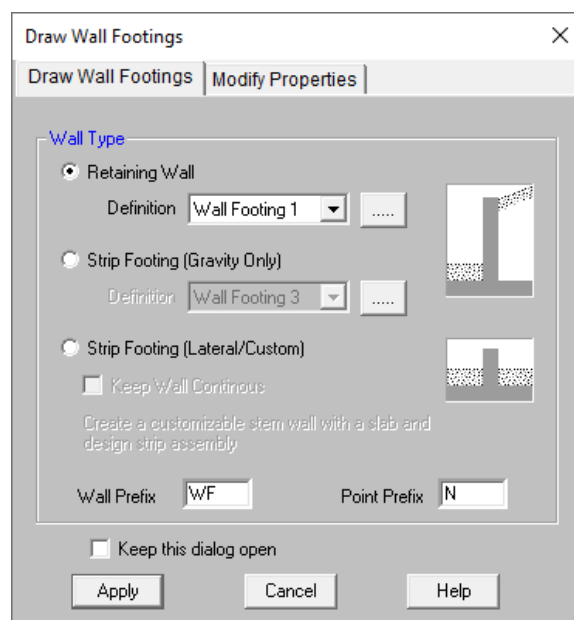
Once you've decided which wall type you want (Wall Footing Types) you can select the appropriate option here. To draw a wall footing click the **Draw or Modify Wall Footings** button and select the **Draw Wall Footings** tab. Choose the wall type, including the **Definition** if applicable.

If you are modeling a **Retaining Wall** or **Strip Footing (Gravity Only)**, click **OK** to immediately start drawing wall footings by clicking the start and end location of the wall footing in the graphic view.

If you are modeling a **Strip Footing (Lateral/Custom)**, you have the option to make the wall footing continuous by checking **Keep Wall Continuous**, then click **OK** and you will be brought to a **Strip Footing Generator** dialog where you can input the properties of your composite slab, slab design strip and stem wall entity. This dialog is similar to the **Wall Footing Definition Editor**.
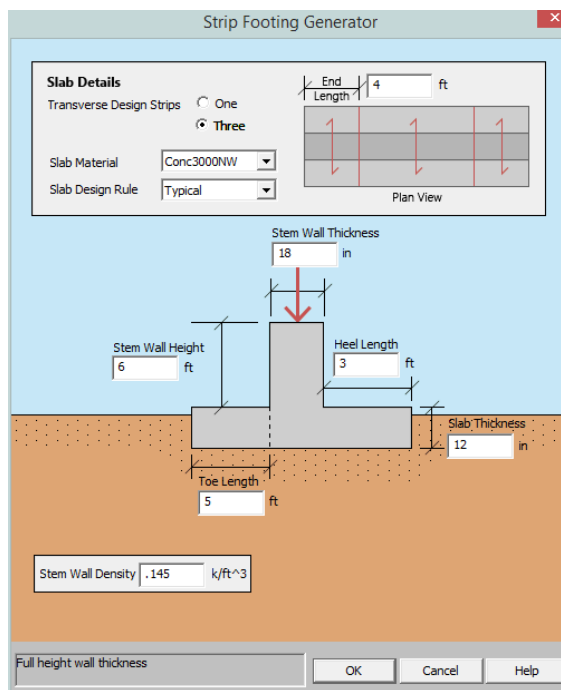
After saying **OK** in this dialog you can then start drawing wall footings by clicking the start and end location of the wall footing in the graphic view.

Note

- If you have not created Wall Footing Definitions ahead of time, you can create a new definition by clicking the [.....] button. This will open you up to the [Wall Footing Definition Editor](#) dialog.

- You may undo any mistakes by clicking the Undo button.

Modify Wall Footings

How you modify **Retaining Walls** and **Strip Footings (Gravity Only)** can be done from the information below. **Modifying Strip Footings (Lateral/Custom)** requires you to edit the individual entities (slab, slab design strip(s) and stem wall) that make up that wall footing.

For non-custom wall footings you may view and edit the wall footing data in the [Wall Footings Spreadsheet](#) or you can use the **Modify Wall Footings** tool to graphically modify multiple wall footings at once.

The graphical **Modify Wall Footings** tool discussed here lets you modify the properties of wall footings that already exist in your model. To use this, specify the properties you want to change, then select the wall footings you want to modify.
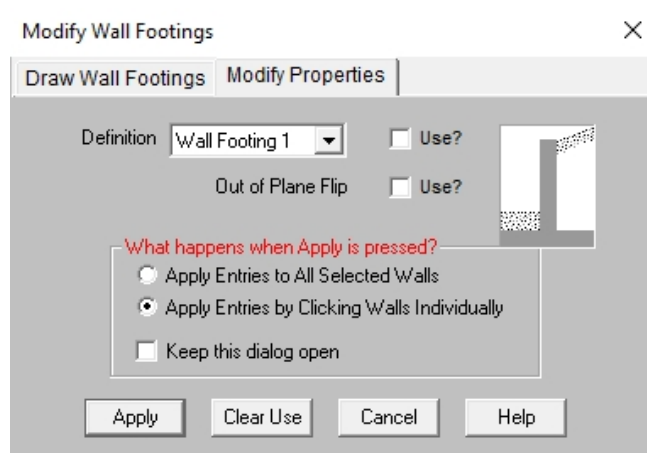
The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected retaining walls. If the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on wall footings without affecting the rest of the properties.

To modify a wall footing click the **Draw or Modify Wall Footings** button and select the **Modify Properties** tab. Then set the parameters that you want to change. Check the **Use?** box for the items to apply.

Note

- If you have drawn your wall footing so the soil is on the wrong face of the wall you can use the **Out of Plane Flip** option. This will flip the start and end nodes of the wall so that the wall footing is flipped to the correct orientation.

Wall Footings Spreadsheet

To access this spreadsheet click the **Wall Footings** button on the **Data Entry** toolbar or choose **Spreadsheets - Wall Footings** from the drop-down menu.

Label

This gives a unique label to each wall footing in the model.

Start Point/End Point

This defines the start and end location points for the wall footing and can be modified here.

Definition

This is the Wall Footing Definition to be used. This setting gives all of the other information required for design of the wall footing.

Wall Footing Modeling Tips

Here are some tips to think about when modeling wall footings.

Wall Footing Orientation

When drawing wall footings, click on two points to define the length of the wall footing. The wall (with footing) will be drawn so that the centerline of the wall lines up along the two nodes defining the wall. These elements, though drawn in a 3D rendering, are really just line elements.

Figure 1: Plan View of Wall Footing

When drawing a wall footing the orientation of the local axes for the wall will always have the local x-direction parallel to the line that defines the two nodes drawn.
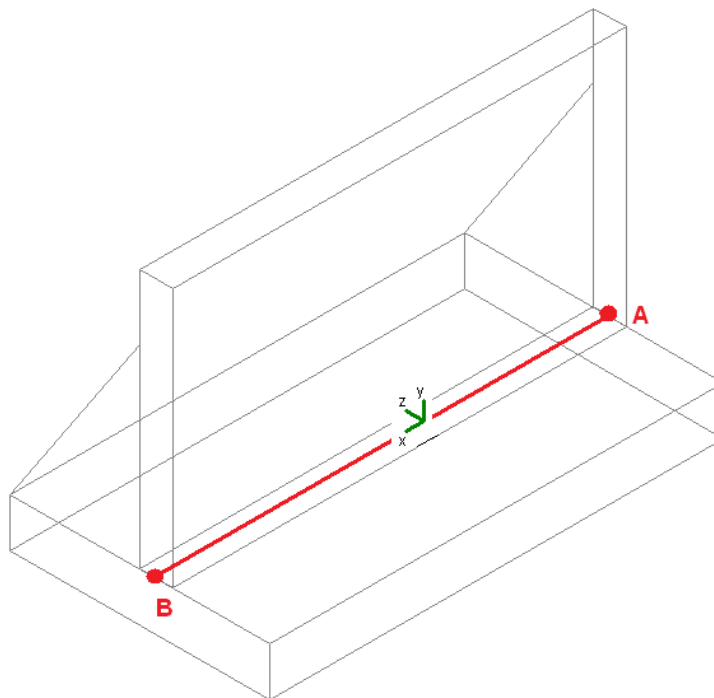
Figure 2: Isometric View of Wall Footing

Here we can also see the orientation of the positive z-axis and that the soil side of the wall footing is oriented on the positive z-axis. So you must keep this in mind when drawing your walls.

If you are drawing basement walls for a structure then make sure to draw in a counter-clockwise fashion. This will ensure that the soil is on the exterior side of the walls. Note that this is only applicable for **Retaining Walls**.
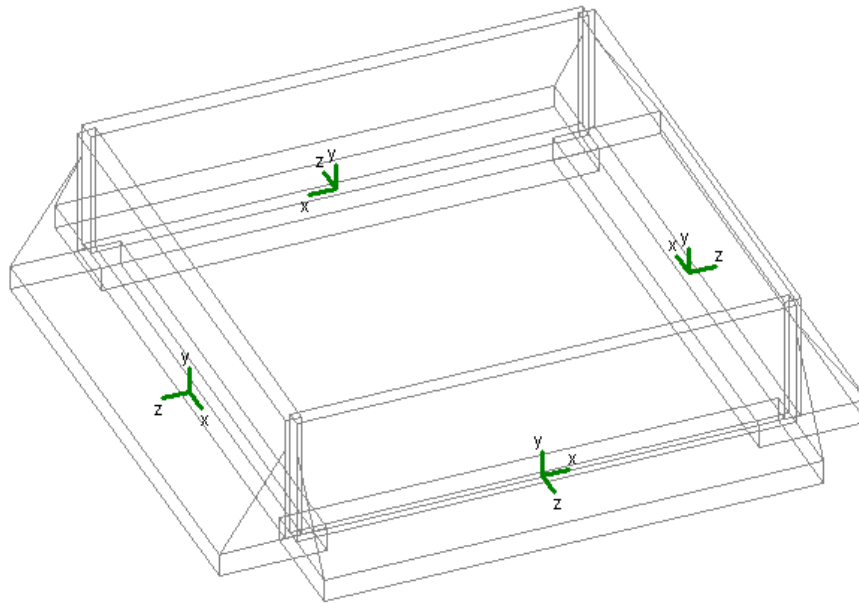
Figure 3: Basement Walls Drawn Counter-Clockwise

Note

- If you have drawn your wall panel so the soil is on the wrong face of the wall you can use the Out of Plane Flip option on the **Modify Wall Footings** dialog. This will flip the start and end nodes of the wall so that the retaining wall is flipped to the correct orientation.

Load Categories

Once you have defined all of the required properties in the Wall Footing Definitions spreadsheet, the program calculates the weights, hydrostatic loads, soil pressures, etc, and applies them to the wall for analysis and design purposes. The program also places these loads in load categories for you automatically.

Dead Loads (DL): weight of concrete and weight of soil

Live Loads (LL): surcharge loads (applicable only to retaining walls)

Hydrostatic Loads (HL): Any lateral pressure/hydrostatic loads (applicable only to retaining walls).

Note:

- Any loads applied to the top of the wall footing will be considered as the load category they are applied as.
- If you have a sloped backfill on your retaining wall, then a portion of the hydrostatic loads will be applied in the vertical direction of your wall. The cosine of these forces will be applied horizontally as a pressure. The sine of these forces will be applied as a vertical force that is conservatively lumped at the top of the wall.
- Water pressure (if there is a water table present) will always be assumed to act horizontally, even in the presence of a sloping backfill.
- It is possible for the horizontal component of hydrostatic force to act ABOVE the height of the wall. All loading that occurs above this height will be lumped as a point load right at the top of the wall.
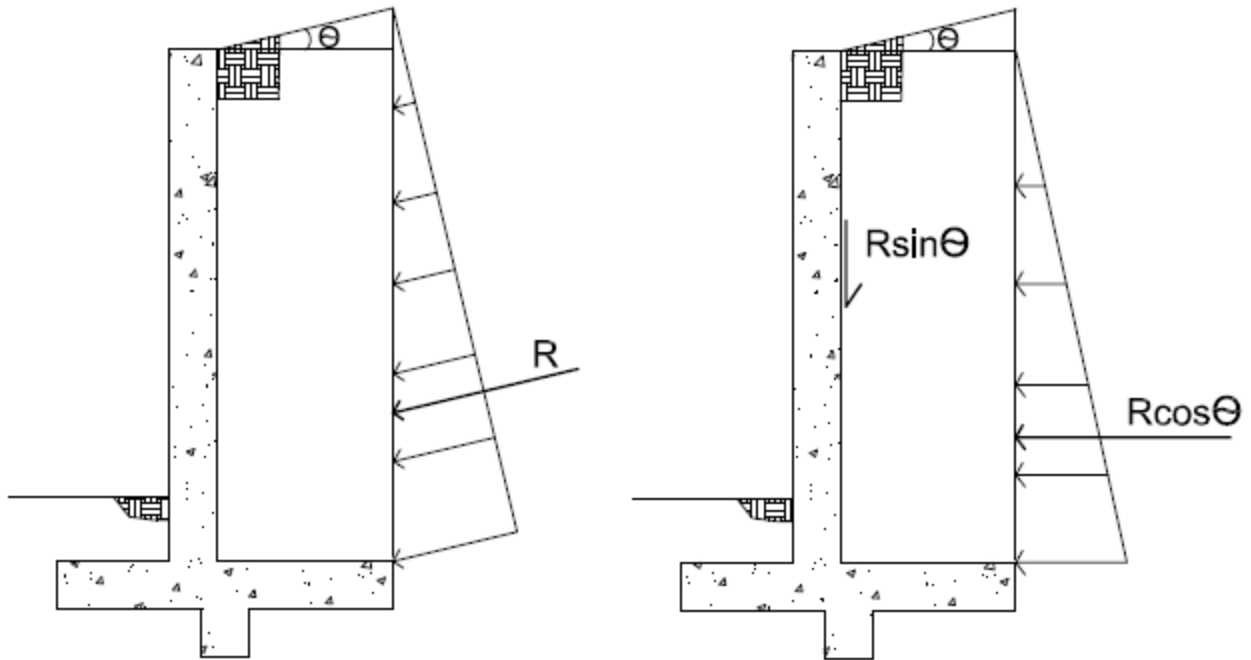
Figure 4: Hydrostatic Loads with Sloped Backfill

Loading of Wall Footings with Externally Applied Loads

Wall footings will accept externally applied loads to them, but only loads applied to the top of the wall. To apply loads to a wall footing you must apply the loads to the centerline of the wall at the base of the wall.
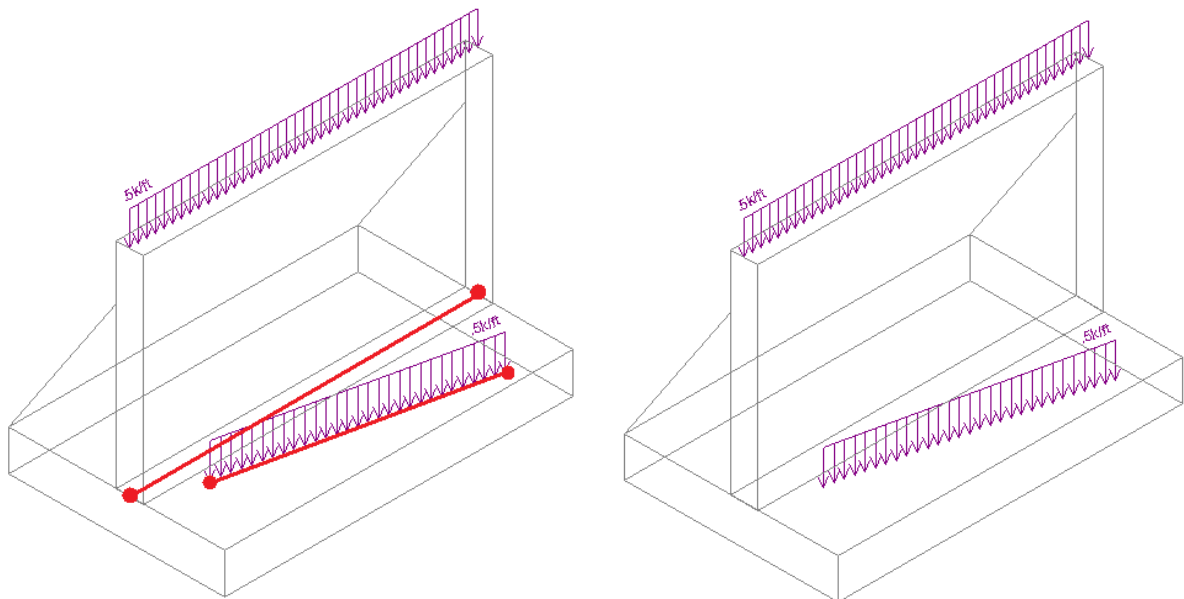
Figures 5A & 5B: Wall Footing with External Loads

In the figure shown above, we can see (from the red lines) how the loads were applied and how they ended up on the wall footing. The distributed load applied along the centerline of the wall ended up applied to the top of the wall and will be analyzed/designed as such. The other load did not line up with the centerline of the wall, so it will not be considered at all in the wall footing design.

Partial length distributed loads (even trapezoidal) and point loads work in the same fashion.
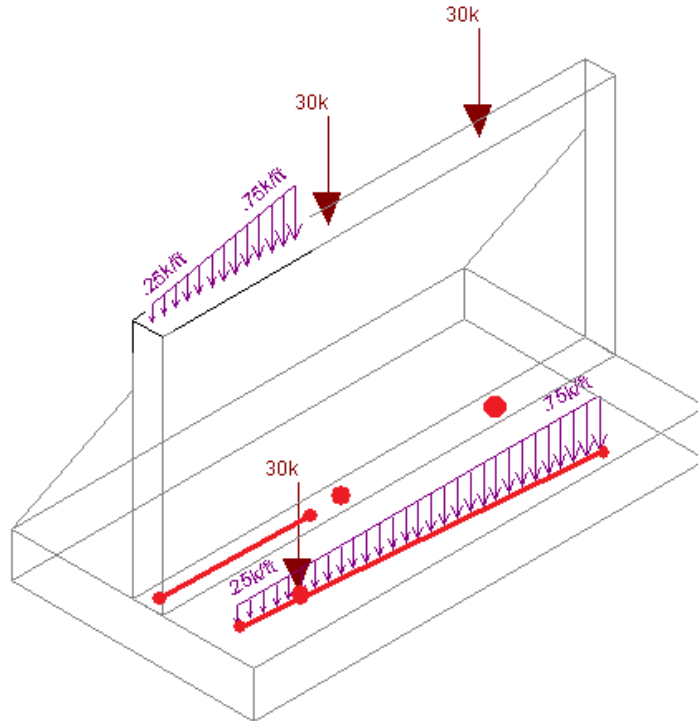
Figure 6: Wall Footings with Trapezoidal Dist. Loads and Point Loads

Looking at the model in a Plan View while drawing can make modeling these loads much easier.
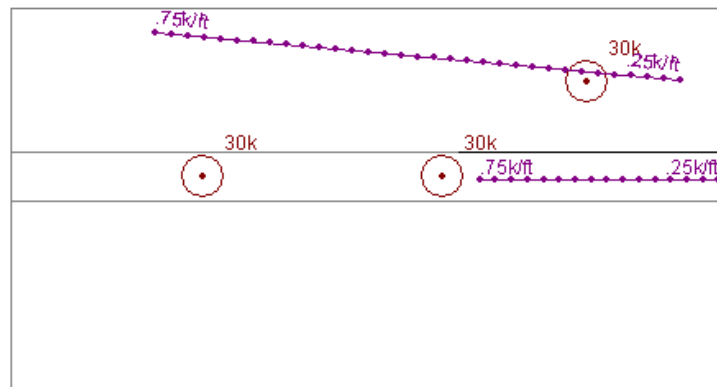
Figure 7: Wall Footing Loading shown in a Plan View

Note:

- If you wish to apply loads adjacent to the wall to work as surcharge loading, apply the loads in the Wall Footing Definitions - Soil tab in the **Heel Surcharge** entry.

If you have two wall footings adjacent (defined by the same node), then applying the load to model will always place that load on the wall footing that comes first in the Wall Footings spreadsheet.
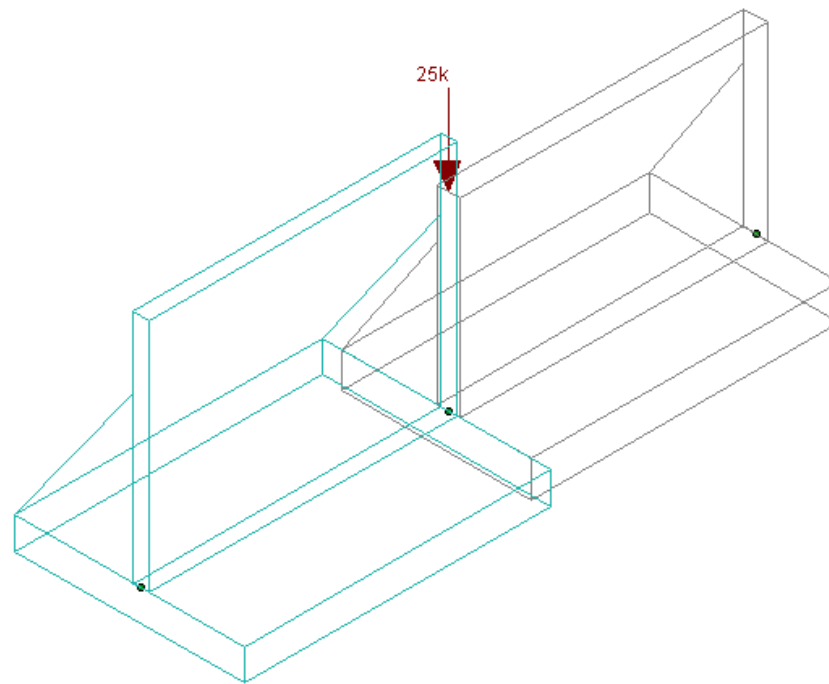
Figure 8: Two Adjacent Wall Footings with a Point Load

Connectivity with Other Elements in the Model

Non-custom **Wall Footings** in RISAFoundation have no connectivity to any other elements in the model, not even adjacent retaining walls. So while it is possible to connect beams and place slabs adjacent to the wall footing, no forces will be transferred between the two. Here are some examples where wall footings and other elements may seem to come into contact with each other.

- **Beams and Wall Footings**: Here it appears that the wall footing and a beam are in the same space. However, if any load was applied along the line where the beam and the wall footing overlap, the load would only be taken by the beam and none of the load would be seen by the wall footing.
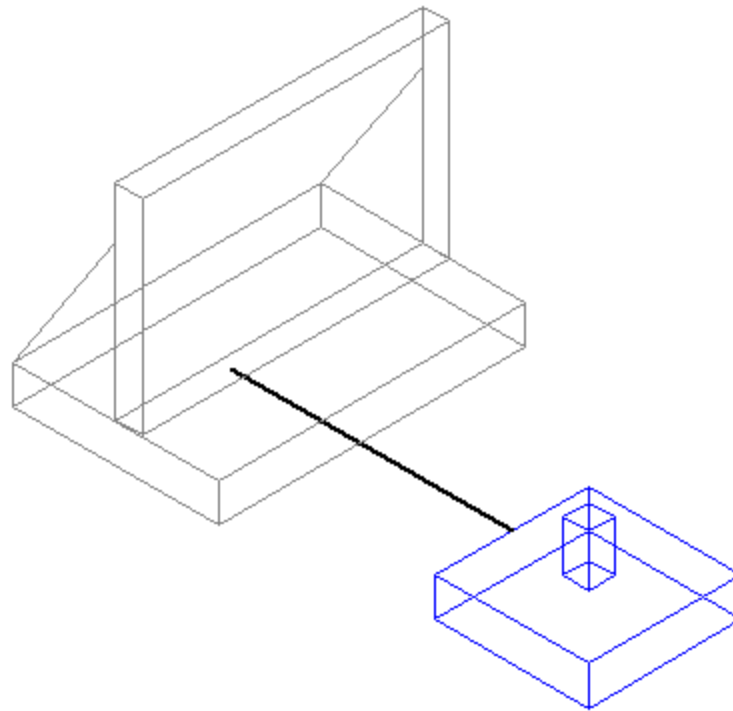
Figure 9: A Beam Overlapping with a Wall Footing

- **Beams and Slabs**: Slabs can be drawn so that they are adjacent with wall footings. Any load that is defined on the area of overlap of a slab and a wall footing will only be taken by the slab. The wall (and footing) will have no interaction with the slab.
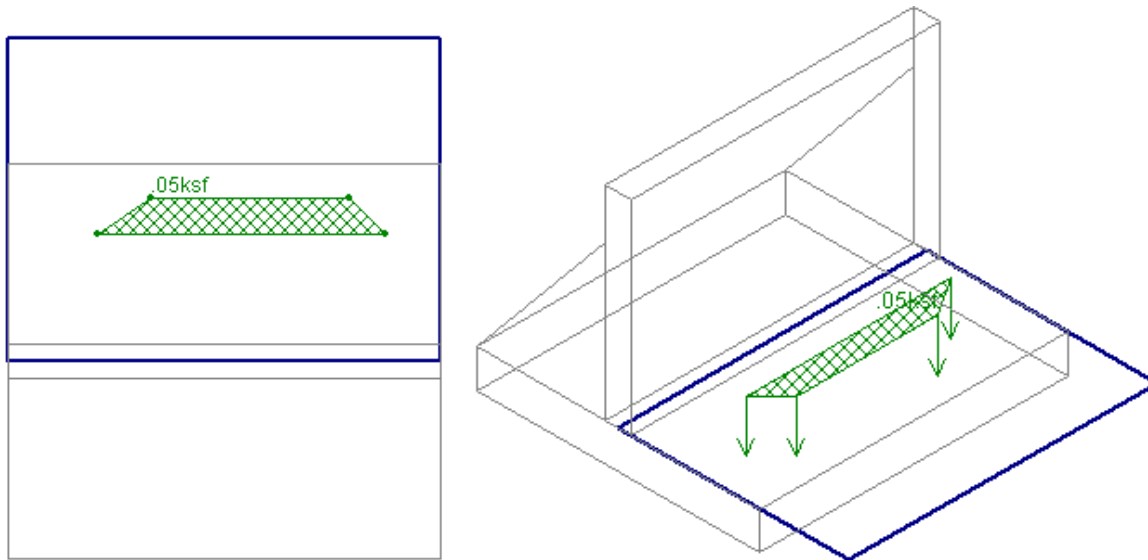
Figures 10A & 10B: A Slab Overlapping with a Wall Footing

- **Other Items that Overlap**: Many other overlapping situations in the program are not handled and an error will occur at solution.
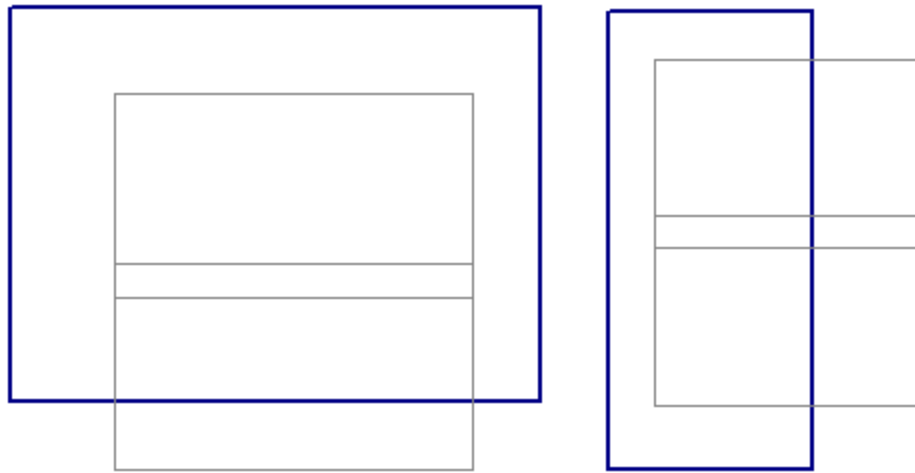
Figure 11: A Slab Drawn Overlapping the Wall Portion
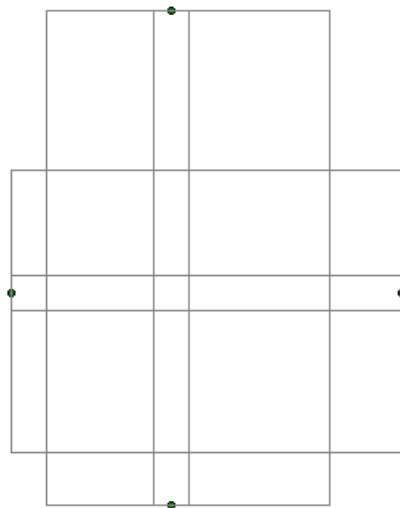
Figure 12: Two Wall Footings Where the Walls Cross
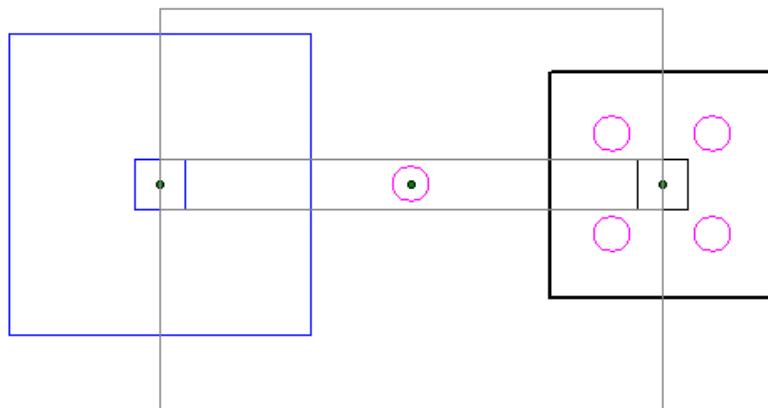
Figure 13: A Pile, Pile Cap, or Footing, that Intersects the Wall Footing Centerline

Wall Footing Definitions

Wall footings consider soil loading and exterior loads to design both the wall stem and footing for concrete design, and consider overturning and sliding serviceability considerations.

- See the Wall Footings - Modeling topic for more information on modeling requirements.
- See the Wall Footings - Design topic for more information on calculation details.
- See the Wall Footing Results topic for more information on the output.

Here we will explain the **Wall Footing Definitions** editor and spreadsheets, found on the Data Entry toolbar or the Spreadsheets drop-down list. Note that you can have multiple wall footings in a given project that may require multiple definitions.

The **Wall Footing Definitions** spreadsheet and editor are always synchronized. Making changes from one will update the other.
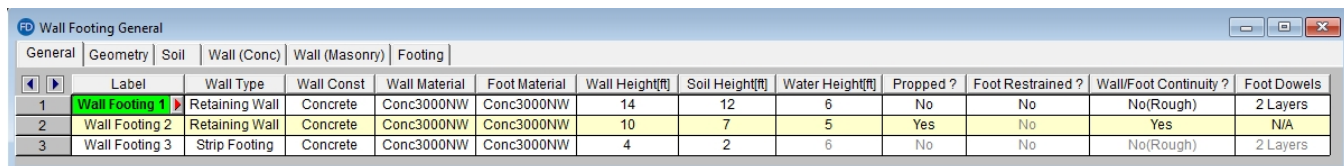
Here we will explain what each of these options mean.

Note:

- See the Wall Footing Definition Editor information for how this information can be created from a graphic dialog.
- Items with an * are not applicable for strip footings.

General Tab

Here are some general considerations for wall footings:

Label

This label provides a unique name for each wall footing definition. Note that there is a red arrow that is available to click on. Clicking this arrow will open the Wall Footing Definition Editor. See below for more information on this dialog.

Wall Type

This allows you to choose between a retaining wall element and a strip footing element. See the Wall Footings - Modeling topic for more information on this.

Wall Const

This allows you to choose between a concrete or a masonry retaining wall.

Wall Material

This is the wall material for the wall. If a concrete wall is selected the concrete materials will be available. Similar with masonry.

Foot Material

This is the concrete material for the footing.

Wall Height

This is the wall height from the top of footing.

Soil Height

This is the height of the backfill behind the wall taken from the top of the footing for a retaining wall. This is the soil height on both sides of the wall above the footing for a wall footing.

Water Height*

This is the height of water behind the wall taken from the base of the footing. If this is input as zero (fully drained condition), then the water table will be assumed to occur below the bottom of footing or shear key.

Propped?*

This defines whether the top of the wall is propped. If the wall is propped it is assumed the base is also restrained laterally and sliding and overturning checks for the wall are omitted.

Foot Restrained?*

This defines whether the bottom of the footing is restrained laterally (i.e. a basement slab is poured adjacent to the wall). If so then the sliding check will be omitted.

Wall/Foot Continuity?*
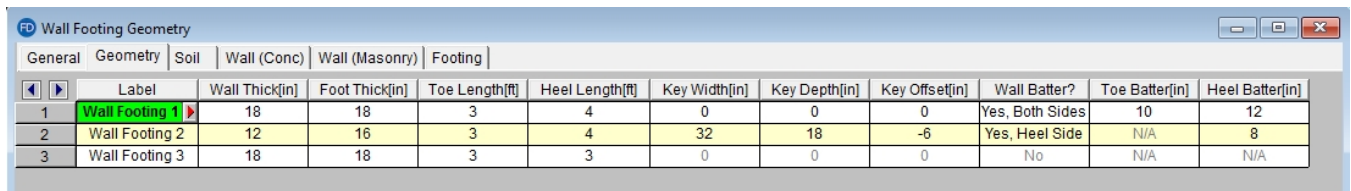
This defines whether the wall and footing are assumed to be monolithic. If the wall and footing are monolithic (the **Yes** option) then we will not do a shear friction check for the dowels from the footing to the wall. If the wall and footing are not monolithic, then we will do a shear friction check. Smooth versus rough defines whether the footing at the location where the wall is poured has been intentionally roughened or not. This affects the shear friction calculation. For more information see the [shear friction check](#) section in the **Wall Footings - Design** topic.

Foot Dowels*

The number of layers defines whether there are dowels in both faces of the wall, or only the interior face (soil face). This input is only required if the footing and wall are NOT monolithic. If that is the case then the shear friction check at the base of the wall requires an area of steel for the dowel bars. The program assumes that the dowels from the footing into the wall will match the wall reinforcement and calculates A_{vf} accordingly.

Geometry Tab

Here are some geometry inputs.

Label	Wall Thick[in]	Foot Thick[in]	Toe Length[ft]	Heel Length[ft]	Key Width[in]	Key Depth[in]	Key Offset[in]	Wall Batter?	Toe Batter[in]	Heel Batter[in]
Wall Footing 1	18	18	3	4	0	0	0	Yes, Both Sides	10	12
Wall Footing 2	12	16	3	4	32	18	-6	Yes, Heel Side	N/A	8
Wall Footing 3	18	18	3	3	0	0	0	No	N/A	N/A

Wall Thickness

For concrete walls it is possible to batter the wall. This thickness excludes the battered portion and would be the thickness at the top of the wall.

For masonry this field is the block nominal thickness and must be set to 4, 6, 8, 10 or 12 inches. The program will subtract 3/8" from this value to get the actual thickness. This value is used, along with the value of grout / bar spacing, to determine the effective thickness of the wall. The effective thickness is based on table B3 of the Reinforced Masonry Engineering Handbook, by Amrhein, Copyright 1998.

Note:

- Even if metric units are used for a masonry wall the nominal inch dimensions are required. Thus, the program will use these values in their converted metric form.

Foot Thickness

This is the footing thickness.

Toe/Heel Length

This is the length of the footing from the outer/inner face of the wall.

Note:

- If the stem wall is battered then these lengths are from the full battered width of the wall.

Key Width/Depth*

These are the dimensions of the shear key if there is one. If either of these dimensions is zero then no key will be considered.

Key Offset*

This is the offset of the key from the inside (soil) face of the wall. A positive dimension will move the key in the heel direction. A negative dimension will move the key in the toe direction.

Wall Batter?

This allows you to slope the toe face of the wall, the heel face, or both.

Note:

- Wall batter can only be used on retaining walls.

Toe/Heel Batter

If you've chosen to batter the wall from the **Wall Batter?** section then this is where you can define the batter dimension. This value is an additional thickness from the **Wall Thick** entry, at the base of the wall.

Soil Tab

Here are the soil parameters.

	Label	Method	Gamma Heel...	Phi Heel(Deg)	K Lat Heel	Gamma Toe[...	Phi Toe(Deg)	K Lat Toe	Delta Wall(D...	Backfill Angle...	Heel Surch...	Toe Depth[ft]	EQ Max[...	Kh	Gamma Sat ...	Phi Sat Heel(...	K Lat Sat Heel
1	Wall Footing 1	Coulomb	.111	30		.111	30		15	10	.5	2		.2	.156	30	
2	Wall Footing 2	Rankine	.111	25	.33	.111	25	3	N/A	15	0	0	N/A	N/A	.164	25	.33
3	Wall Footing 3	Rankine	.111	30		.111	30		N/A	0	.5	2	N/A	N/A	.156	30	

Method (Rankine or Coulomb)*

Here you have the choice of how you want lateral earth pressures calculated. Note that if you want seismic forces applied to your retaining wall Coulomb must be selected.

Note:

- Calculations for soil pressures for both methods are defined in the Wall Footings - Design topic.
- Seismic forces are described in the Wall Footings - Design topic.

Gamma Heel/Gamma Toe

These are the soil densities on both the heel and toe sides of the wall for a retaining wall. For a strip footing the Gamma Heel value is used for both the heel and toe.

Phi Heel/Phi Toe*

This is the internal angle of friction of the soil on both the heel and the toe sides of the wall. This is used, along with the backfill angle, in determining the lateral force coefficient (K). Note that there is a K input field. If this field is input then we will override the calculation and simply use the input K value. For more information on this see **Soil Considerations** in the Wall Footings - Design topic.

K Lat Heel/K Lat Toe*

This allows you to directly input the soil lateral force coefficient, K, for the soil on the heel and toe sides. You can leave this blank and the program will calculate K for you, using the **Phi** and **Backfill Angle** values. For more information on this see **Soil Considerations** in the Wall Footings - Design topic.

Delta Wall*

This is the wall friction angle. It is used only if **Coulomb** is set for the Lateral Pressure Method. For more information on how this is used see **Soil Considerations** in the Wall Footings - Design topic.

Note:

- If **Rankine** is selected then this input is ignored.

Backfill Angle*

This is the slope angle of the backfill on the heel side from horizontal. A negative slope is not allowed.
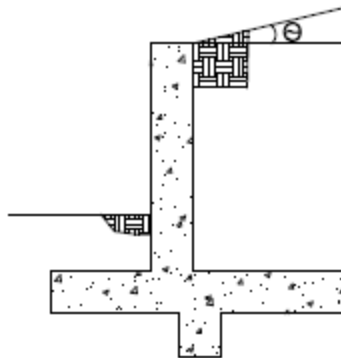
Heel Surcharge*

This is a vertical surcharge pressure on the soil adjacent to the heel side of the wall and is always considered in the **LL load category**.

Toe Depth*

This is the soil depth above the top of footing on the toe side of the wall. If this is set to zero, then the soil level will be assumed at the bottom of footing elevation. Thus, if there is a shear key, you can consider the passive pressure only on the key.

EQ Max*

This is used for seismic loading and, if input, would define the maximum earlhquake pressure at the top of an inverted earthquake pressure diagram. If this value is blank the program will use **Kh** and calculate the seismic forces per the Mononobe-Okabe formulation. Seismic forces are described in the Wall Footings - Design topic.
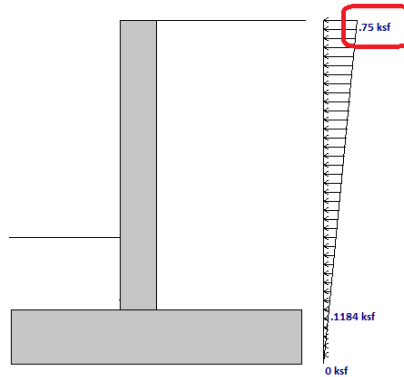
Note:

- If **Rankine** is selected then this input is ignored.

Kh*

This is the design horizontal ground acceleration (in units of g). This is an input into the Mononobe-Okabe seismic force calculation method. For more information on this procedure see the Wall Footings - Design topic.

Note:

- If **Rankine** is selected then this input is ignored.

Gamma Sat/Phi Sat Heel/K Lat Sat Heel*

These inputs are only considered if the water height is greater than zero. These are the soil parameters in the saturated portion of soil. If **K Lat Sat Heel** is left blank, the program will use the **Phi Sat Heel** and the **Backfill Angle** to calculate K. For more information on this see **Soil Considerations** in the Wall Footings - Design topic.

Wall Tab (Conc)

This tab is only used if your wall is made of concrete to define reinforcement parameters.

Label	Location	Int Bar	Min Int Bar S...	Max Int Bar S...	Ext Bar	Min Ext Bar S...	Max Ext Bar S...	Bar Inc[in]	Int Cover[in]	Ext Cover[in]	Hor Bar Size	Hor Bar Spa[in]	Outer Bars
1	Wall Footi... Each Face	#7	6	18	#8	6	18	1	3	1.5	#4	18	Vertical
2	Wall Footing Single Layer	#6	6	12	N/A	N/A	N/A	2	3	N/A	#5	18	N/A
3	Wall Footing Each Face	#5	6	18	#5	6	18	1	3	1.5	#5	18	Vertical

Location

This defines whether you have a single layer of reinforcement or bars each face.

Int Bar and Ext Bar

This defines the vertical bar on the interior (soil) and exterior faces of the wall. If you have defined a single layer of reinforcement then the Ext Bar field will not be used.

Min/Max Int and Ext Bar Space

The program will design the vertical reinforcement spacing based on these guidelines. If you want the reinforcement to be at an exact spacing, enter that spacing as both the min and max in order to force this spacing.

Note:

- If you have a cantilevered wall, the exterior bar spacing design will be designed solely to meet minimum requirements.
- If you have defined a single layer of reinforcement then the exterior min and max spacing will not be used.

Bar Inc

This is the spacing change increment that the program will use for design. If the maximum spacing does not work, the spacing will drop by this increment and be checked again. The program will work its way down until it reaches a spacing that meets all reinforcement requirements.

Int and Ext Cover

These are cover dimensions for the interior (soil) and exterior faces of the wall. If you have defined a single layer of reinforcement then the exterior cover will not be used.

Hor Bar Size and Hor Bar Spacing

This defines the bar size and spacing of the horizontal bars in the wall. The program will check that these bars meet minimum requirements.

Note:

- The program does not consider two way bending, so this reinforcement is only for non-strength requirements.

Outer Bars

This defines whether the vertical or horizontal bars in the wall are nearest the faces of the wall. This affects the "d" calculation for the wall. If you have defined a single layer of reinforcement then this field will not be used and it will be assumed that the vertical bar is nearest the inside (soil) face.

The images below are a section cutting horizontally through the wall.

<u>**Horizontal**</u>
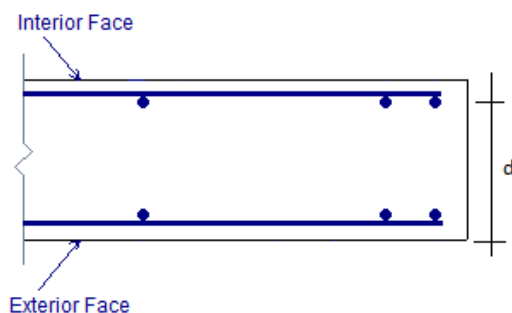
Vertical
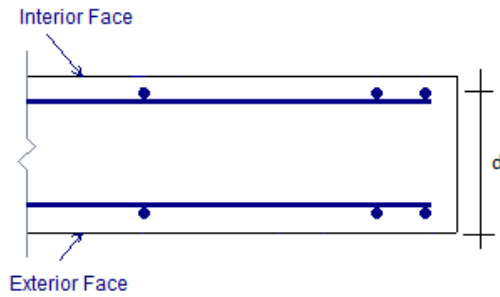
Wall Tab (Masonry)

This tab is only used if your wall is made of masonry to define reinforcement and design parameters.

Block Grouting

This option defines how the wall is grouted. If "Partially Grouted" is chosen, then the spacing of grout will be based on the Bar Spacing Min/Max..

Reinforced

This defines whether the wall is reinforced or not. Note that if the wall is unreinforced the program will also consider the wall ungrouted.

Wall Area Method

This option defines where the wall area is taken from. The **NCMA** option pulls the A_n value from the NCMA TEK 141B document. The **RMEH** option pulls the "Equivalent Solid Thickness" value from Table B-3a and B-3b from the *Reinforced Masonry Engineering Handbook*, James Amrhein, 5th edition copyright 1998.

In doing research on these two methods of calculating the area for a masonry wall, the two methods produce very different results. The **NCMA** values assume face-shell mortar bedding and web bedding around grout-filled cells. The **RMEH** values assume full mortar bedding (both face-shells and all webs). The Amrhein values also appear to average in the area of horizontal bond beams as well. This would make the area conservative for a self-weight calculation, but unconservative for stress calculations. With these considerations in mind we are defaulting the behavior to use **NCMA**.

See the Masonry Wall - Design topic for the specific calculations regarding NCMA or RMEH.

Note:

- For self-weight the program will ALWAYS use **RMEH**, as the presence of bond beams in the wall is realistic and this is a conservative value.

Bar Size

This is the main vertical bar size that will be used for design.

Bar Space Min/Max

This allows you to give a maximum and minimum bar/grout spacing. If you give a range between the max and min, then the program will optimize the reinforcement spacing according to code check requirements.

Bar Placement

This defines how the reinforcement is placed in the wall.

Center will put a single bar centered in a given cell.

Non-Center will put a single bar non-centered in a given cell.

Each Face will put reinforcement on both faces of a given cell.

Staggered alternates the bars on either face along the length of the region.

Note:

- When using the staggered option you are selecting to space the bars at each face at double the bar/grout spacing defined above. For example, if you have a staggered spacing at 24" oc you have a bar on the outside face at 48" oc and a bar on the inside face at 48" oc. These bars are staggered, thus you have grout filled cores at 24" oc.

Cover

Allows you to specify the distance from the exterior of the masonry block to the extreme fiber of the reinforcement. The default cover value is defined as **Min**. The **Min** input will use whichever governs between these two sections of the specification:

- Section 6.1.3.5 states a minimum distance between the inside of the face shell and the reinforcement . RISA is conservative and always assumes this to be 1/2". Thus, the cover calculated in this fashion = $t_{faceshell}$ + 1/2"
- Section 6.1.4 requires a cover dimension = 1-1/2" or 2" depending on bar size and whether the masonry is exposed to weather. In the program this is always assumed to be 1-1/2".

Thus, the **Min** term will create a cover equal to the maximum of $t_{faceshell}$ + 1/2" OR 1-1/2".

For any other cover you wish to impose simply overwrite the **Min** value with your value. If using the **Non-Center** option then the cover will always be from the *+z local axis face of the wall*.

For example, let's assume the wall shown below is using the "Non-Centered" reinforcement and the cover is set to 2.5". The image below shows exactly where this bar is then located in the wall (d = 7.625" - 2.5" - 3/8" = 4.75").
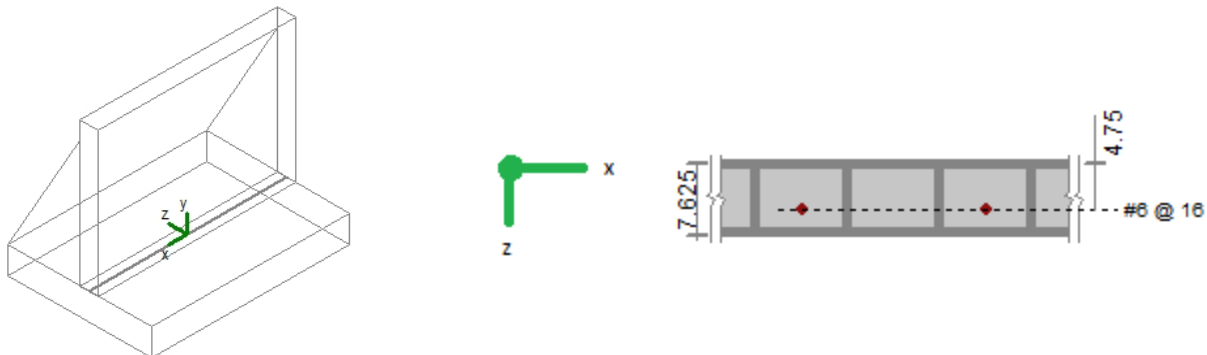
The actual "d" used in design will depend on the governing direction of loading. If load is applied in both out-of-plane directions to the wall then it is possible that a lower level of loading can produce a higher code check because the "d" is smaller in one direction than the other.

If a non-sensical value is defined (i.e. one where the bar does not fall inside of the block core then the program will give an error in the results and place the reinforcement just inside the faceshell of the block.

Note:
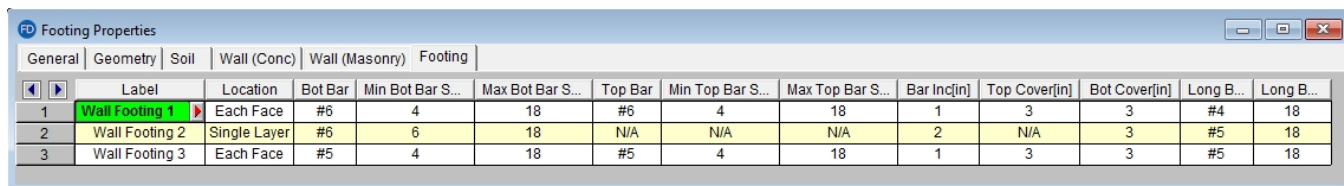
- For the **Center** bar placement this field is ignored.

Mortar/Cement Types

Allows you to specify the type of mortar/cement in the wall. This affects the modulus of rupture (flexural tensile stresses) from Tables 8.2.4.2 and 9.1.9.2.

Footing Tab

Here are footing reinforcement parameters.

	Label	Location	Bot Bar	Min Bot Bar S...	Max Bot Bar S...	Top Bar	Min Top Bar S...	Max Top Bar S...	Bar Inc[in]	Top Cover[in]	Bot Cover[in]	Long B...	Long B...
1	Wall Footing 1	Each Face	#6	4	18	#6	4	18	1	3	3	#4	18
2	Wall Footing 2	Single Layer	#6	6	18	N/A	N/A	N/A	2	N/A	3	#5	18
3	Wall Footing 3	Each Face	#5	4	18	#5	4	18	1	3	3	#5	18

Location

This defines whether you have a single layer of reinforcement or bars both top and bottom.

Bot Bar and Top Bar

This defines the bottom and top bar sizes in the footing. If you have defined a single layer of reinforcement then the Top Bar field will not be used.

Min/Max Bot and Top Bar Space

The program will design the main footing reinforcement spacing based on these guidelines. If you want the reinforcement to be at an exact spacing, simply enter that spacing as both the min and max in order to force this spacing. If you have defined a single layer of reinforcement then the top min and max spacing will not be used.

Bar Inc

This is the spacing change increment that the program will use for design. If the maximum spacing does not work, the spacing will drop by this increment and be checked again. The program will work its way down until it reaches a spacing that meets all reinforcement requirements.

Top Cover and Bot Cover

These are cover dimensions for the footing. If you have defined a single layer of reinforcement then the top cover dimension will not be used.

Long Bar Size and Long Bar Spacing

This defines the bar size and spacing of the longitudinal bars in the footing. The program will check that these bars meet minimum requirements.

Note:

- The program does not consider two way bending, so this reinforcement is only for non-strength require-ments.

- Items with an * are not applicable for strip footings.

Wall Footing Definition Editor

The Wall Footing Definition Editor provides a way to create or modify wall footing definitions in a graphical way. This editor automatically updates any information input into the **Wall Footing Definitions** spreadsheet.

Note:

- Keep in mind that this dialog has multiple tabs. Make sure to fill out the information on all relevant tabs before clicking OK.

- See the information in the **Status Bar** for more information on each of these entries.
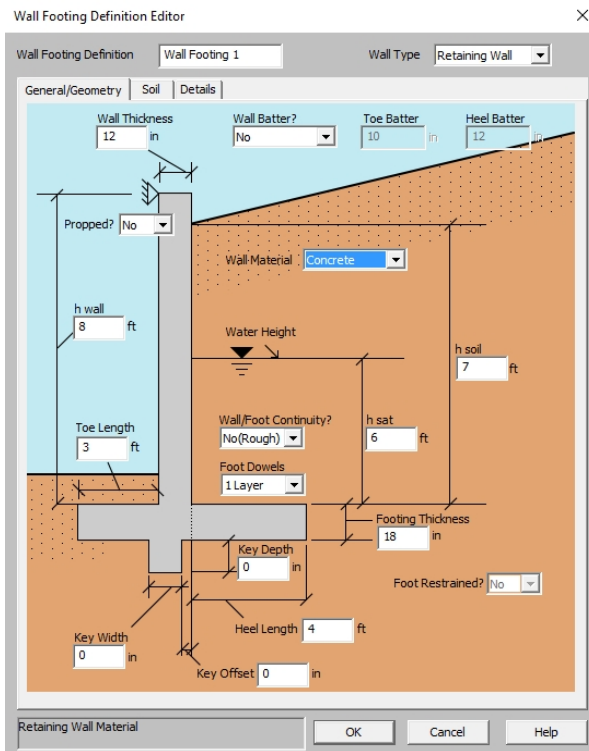
General/Geometry tab for Retaining Walls
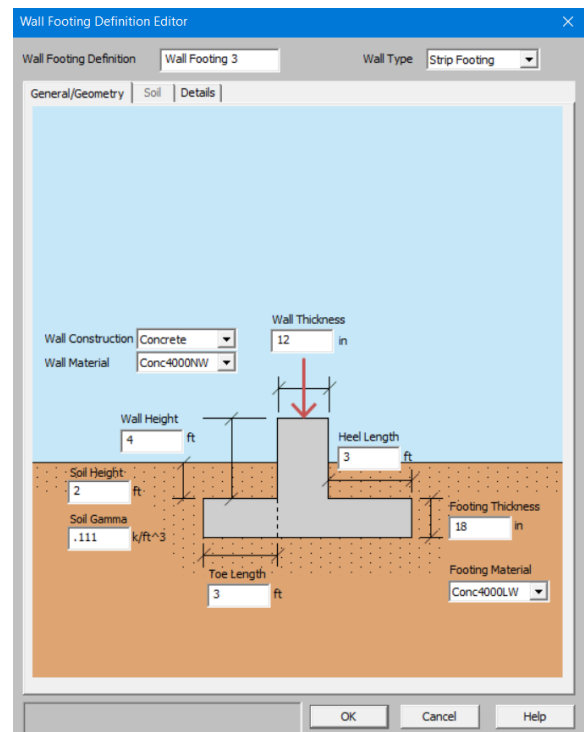
General/Geometry tab for Strip Footings
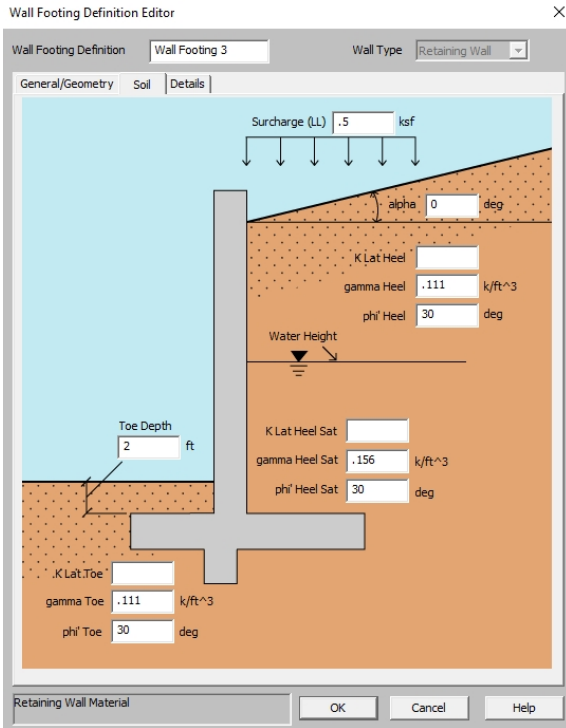
Soil tab not applicable for Strip Footings
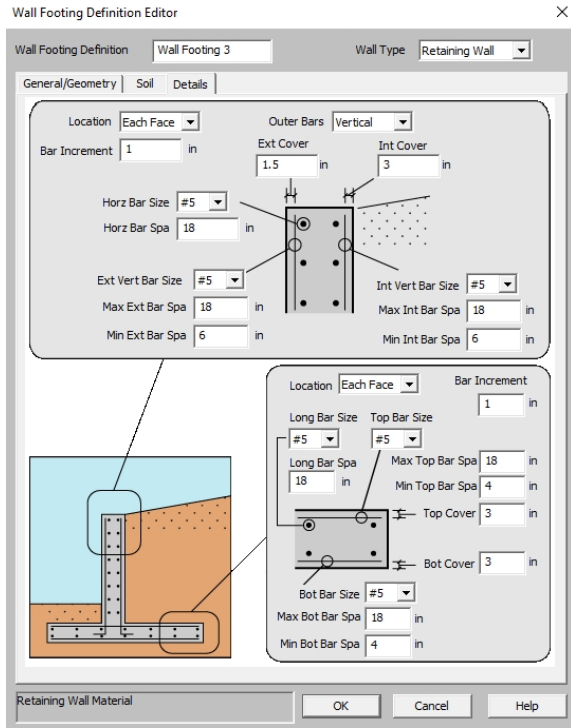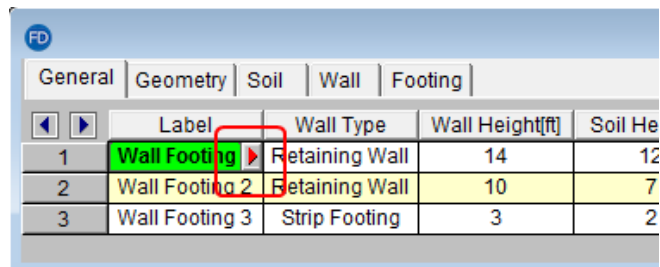
Details tab applicable for Concrete Retaining Walls
and Strip Footings
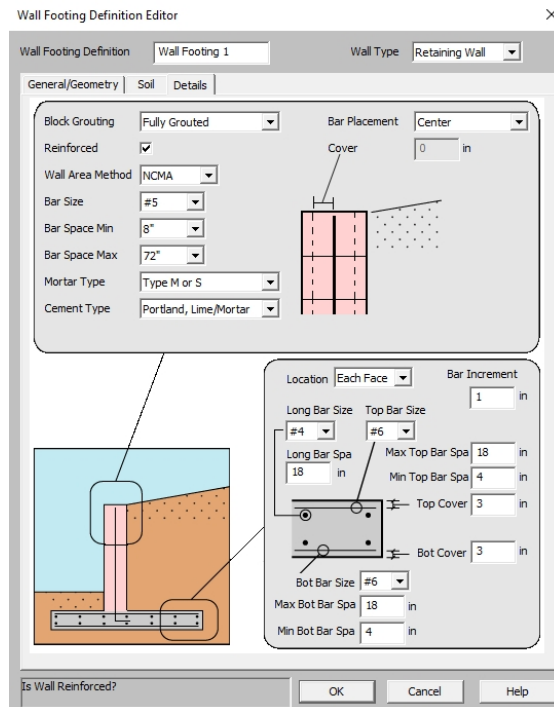
Details tab applicable for Masonry Retaining Walls
and Strip Footings

Accessing the Wall Footing Editor

To access this editor you can click the red arrow on any tab in the **Wall Footing Definitions** spreadsheet (click in the Label column to make the red arrow appear).

This dialog can also be accessed from the **Draw Wall Footings** dialog as well by clicking either of these buttons.
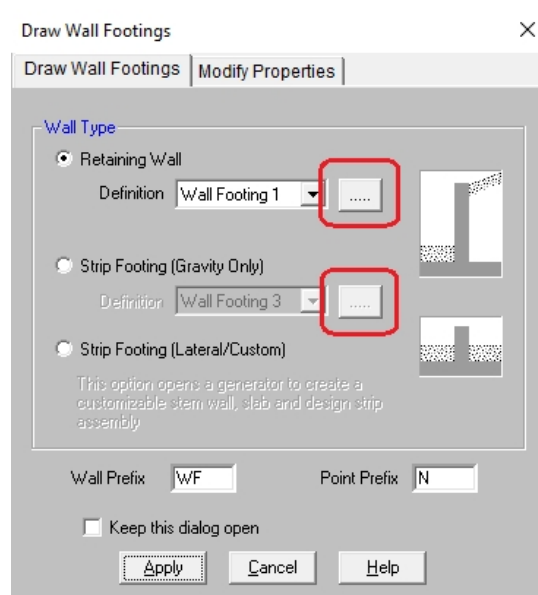
Wall Footings - Design

In this section we will talk specifically about the calculations for wall footings.

- See the Wall Footing Definitions topic for more information on defining wall parameters.
- See the Wall Footings - Modeling topic for more information on modeling requirements.
- See the Wall Footing Results topic for more information on the output.

RISAFoundation currently designs wall footings for all of the **ACI** codes, as well as the **CSA A23.3-04/14 Canadian** code. In this topic we will make specific reference to provisions for some of these codes. Note that the only differences in behavior with the differing codes is with the reinforcement design. All soil and serviceability checks are identical across codes.

Soil Considerations

Here we will discuss some of the details in considering the computation of soil pressures.

Note:

- This section is only applicable for retaining walls. Strip footings use soil solely for vertical load considerations.

Lateral Force Coefficient (K) Calculations

From the Wall Footing Definition Editor (or the Wall Footing Definitions spreadsheet) you can define either **Rankine** or **Coulomb** for the Lateral Earth Pressure Methodology

Rankine

From the Wall Footing Definition Editor (or the Wall Footing Definitions spreadsheet) you can either define the coefficient of internal friction (φ) and backfill angle (θ) and allow the program to calculate K. Or you can directly input the K factors. If the program is used to calculate K, these are the equations used:

$$K_o := 1 - \sin(\phi)$$

$$K_a := \cos(\theta) \cdot \left[\frac{\cos(\theta) - \sqrt{(\cos(\theta))^2 - (\cos(\phi))^2}}{\cos(\theta) + \sqrt{(\cos(\theta))^2 - (\cos(\phi))^2}} \right]$$

$$K_p := \cos(\theta) \cdot \left[\frac{\cos(\theta) + \sqrt{(\cos(\theta))^2 - (\cos(\phi))^2}}{\cos(\theta) - \sqrt{(\cos(\theta))^2 - (\cos(\phi))^2}} \right]$$

Note:

- The calculation of K_o assumes normally consolidated granular soil.
- K_a and K_p are calculated considering Rankine theory.
- Battered retaining walls behave identically to uniform thickness walls for hydrostatic loading because of the use of Rankine theory.

Coulomb

With the Coulomb method you can also directly input K and the program will ignore the other values. If the program is used to calculate K, these are the equations used.

$$K_0 := 1 - \sin(\phi)$$

$$K_a := \frac{(\sin(\alpha + \phi))^2}{(\sin(\alpha))^2 \cdot \sin(\alpha - \delta) \cdot \left(1 + \sqrt{\dfrac{\sin(\delta + \phi) \cdot \sin(\phi - \theta)}{\sin(\alpha - \delta) \cdot \sin(\alpha + \theta)}}\right)^2}$$

$$K_p := \frac{(\sin(\alpha - \phi))^2}{(\sin(\alpha))^2 \cdot \sin(\alpha + \delta) \cdot \left(1 - \sqrt{\dfrac{\sin(\delta + \phi) \cdot \sin(\phi + \theta)}{\sin(\alpha + \delta) \cdot \sin(\alpha + \theta)}}\right)^2}$$

where:

- θ = Backfill angle
- φ = Soil friction angle
- δ = Wall friction angle
- α = Wall slope angle measured from horizontal (90 degrees for a vertical wall face)

Both Methods

There is a potential to have three K factors for a single retaining wall: K Heel, K Sat Heel and K Toe.
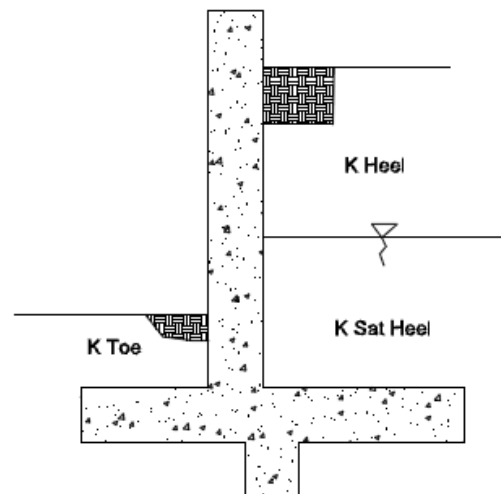
Figure 1: Three Possible K factors

Note:

- If the wall is a **cantilever**, the program will use the K_a equation for the heel side K factors and K_p equation for the toe side K factors.
- If the wall is a **propped cantilever**, the program will use the K_0 equation for both the heel and toe K factors.

- RISAFoundation performs a calculation to compare the total passive forces to the active forces. If the passive forces exceeds the active force, the wall is assumed to be in an at rest condition and the program will use the K_o equation for both the heel and toe K factors. When this occurs, the wall footing detail report will show (At Rest) after both the heel and toe K factors.

Lateral Soil Summation

The program will perform a calculation comparing the total active force from the heel side and the total passive force from the toe side. If the passive force exceeds the active force you then have an irrational solution. In this case, the program will give a message in the detail report and not design your retaining wall. The message states "The hydrostatic passive forces (toe side) exceed the hydrostatic active forces (heel side). Consider manually entering K factors for this wall. In this case the Rankine active vs passive pressures will not work, so instead you may need to manually input your K factors in the Wall Footing Definitions spreadsheet.

Loading Considerations

Regarding the loading of retaining walls, all hydrostatic, soil and surcharge loads are taken into account on a per foot basis. For more information on this, see the Wall Footings - Modeling topic.For externally applied loads to either retaining walls or strip footings, these loads are summed over the entire length of the wall and then divided out on a per foot basis. Because of this stress risers and areas of the wall that require tighter reinforcement are not taken into account. To get more information on applying external loads to the model, see the Wall Footings - Modeling topic.

Note:

- For highly loaded areas of wall footings where you want the reinforcement to be modified, break the wall footing into multiple pieces. Each piece will have its reinforcement optimized separately.

Seismic Loading Considerations

RISAFoundation will consider an inverted triangular seismic loading diagram if:

- An EL category (EL, ELX, etc) is used in the load combination
- The Coulomb method is used for the Lateral Earth Pressure Methodology
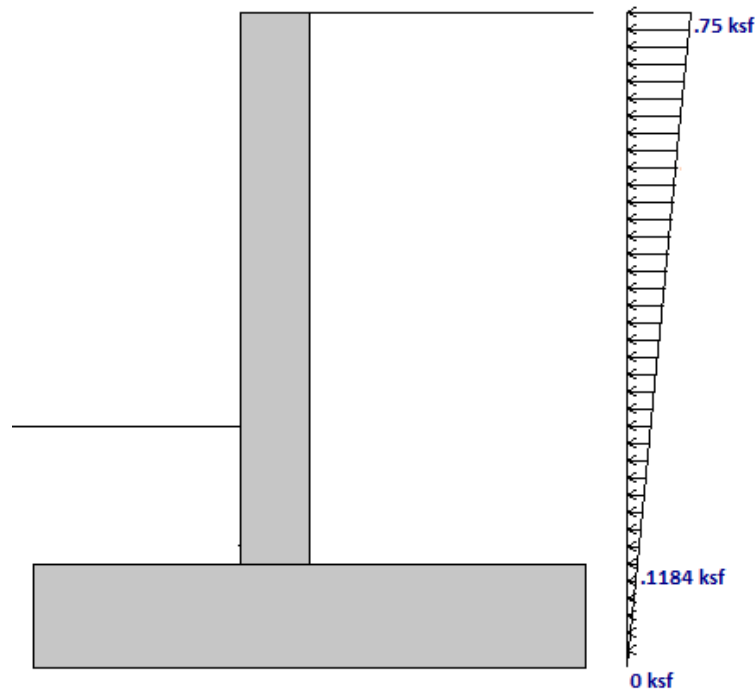
There are two paths to applying your seismic loads:

User Define the Magnitude of the Maximum Value of the Diagram

In the [Wall Footing Definitions - Soil](#) tab there is an input for EQ Max. If there is a value in here this is used as the maximum.

Allowing the Program to Calculate this Magnitude Using the Mononobe-Okabe/Seed-Whitman Formulation

In the Soil tab of the Wall Footing Definitions spreadsheet, there is an input for EQ Max and Kh. For the program to calculate the magnitude of the maximum value of the diagram, EQ Max should be left blank and a value for Kh should be input.

The Mononobe-Okabe/Seed-Whitman equation is an expression for the total lateral earth force, seismic plus active. Thus, to calculate the portion due solely to seismic we must remove the active pressure portion. The M-O pressure coefficient is calculated as follows:

$$K_{a\theta} := \frac{\left(\sin\left(\alpha + \phi - \kappa\right)\right)^2}{\cos\left(\kappa\right) \cdot \left(\sin\left(\alpha\right)\right)^2 \cdot \sin\left(\alpha - \delta - \kappa\right) \cdot \left(1 + \sqrt{\frac{\sin\left(\delta + \phi\right) \cdot \sin\left(\phi - \theta - \kappa\right)}{\sin\left(\alpha - \delta - \kappa\right) \cdot \sin\left(\alpha + \theta\right)}}\right)^2}$$

where:

- θ = Backfill angle
- φ = Soil friction angle
- δ = Wall friction angle
- κ = Seismic Factor = tan-1(Kh)
    - Kh = Input on Soils tab
- α = Wall slope angle measured from horizontal (90 degrees for a vertical wall face)

The new term here is κ. This value uses the input **Kh** value for the Design Horizontal Ground Acceleration (in g). **Kh** is generally defined as **Sds/2.5**, but the actual value is up to the user. The vertical component of earthquake acceleration is not considered in the program.

The total seismic lateral force (per foot) is then defined as:

$$P_{Seismic} := \frac{1}{2} \cdot \gamma_{soil} \cdot H^2 \cdot \left(K_{AE} - K_a \right)$$

Per the Seed-Whitman variation this seismic force is assumed to act at 0.6*Hwall. Thus, to calculate the magnitude at the top of the wall this equation is used:

$$EQ_{max} := \frac{P_{Seismic}}{0.3 \cdot H^2}$$

Notes:

- Seismic calculations only apply to the Coulomb Lateral Earth Pressure Methodology.
- Seismic is ignored if K factors are hard-coded and EQmax is left blank in the Soils tab.
- Seismic is ignored if the wall is propped at the top.

Serviceability Considerations

Overturning and sliding checks are only provided for **Service** load combinations (LC's with the Service checkbox checked).

Overturning

Overturning is checked at the base of the footing at the extreme edge of the toe. Each portion of soil weight, concrete weight, surcharge loading, lateral soil pressure and lateral water pressure is taken into account. Each piece is either part of the resisting moment or the overturning moment.
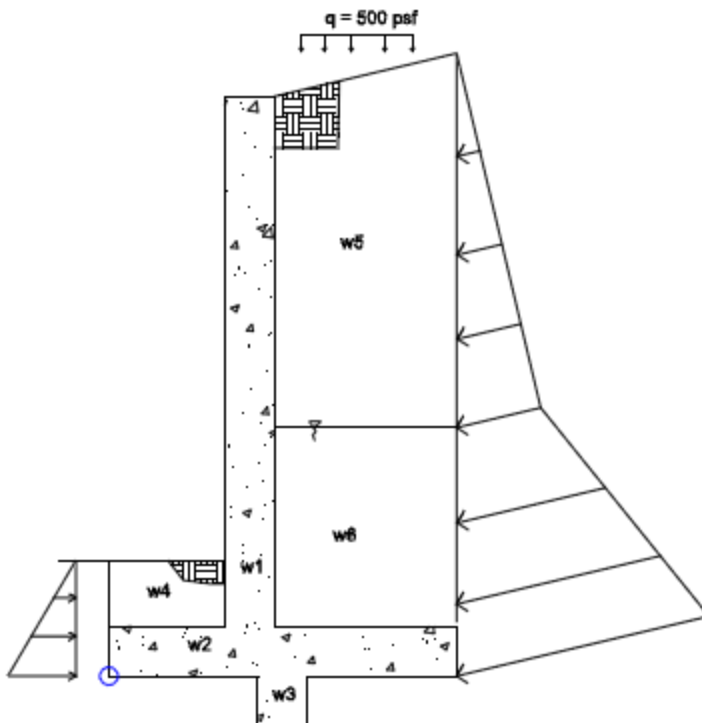
Figure 2: Overturning Diagram

From here, all the resisting moments are summed and all of the overturning moments are summed. The ratio of resisting moments to overturning moments is checked against the **SF** given in that individual load combination.

$$OSF := \frac{M_{Resisting}}{M_{Overturning}}$$

$$OT_{UC} := \frac{SF}{OSF}$$

Note:

- If the wall is a strip footing then there is no consideration for lateral hydrostatic forces. Only vertical soil loads, self-weight and external forces are considered.
- If the wall is a propped cantilever then overturning checks will be omitted.
- The program considers the passive soil resistance from the toe in overturning considerations.
- For sloped backfills, the vertical component of the hydrostatic load will be included in the resisting moment.
- The presence of a key has no bearing on overturning except that the weight of the key will be accounted for.

Sliding

Sliding is checked at the extreme bottom face of the footing. If there is no shear key, then this occurs at the bottom of footing. If there is a shear key this occurs at the bottom of the key.
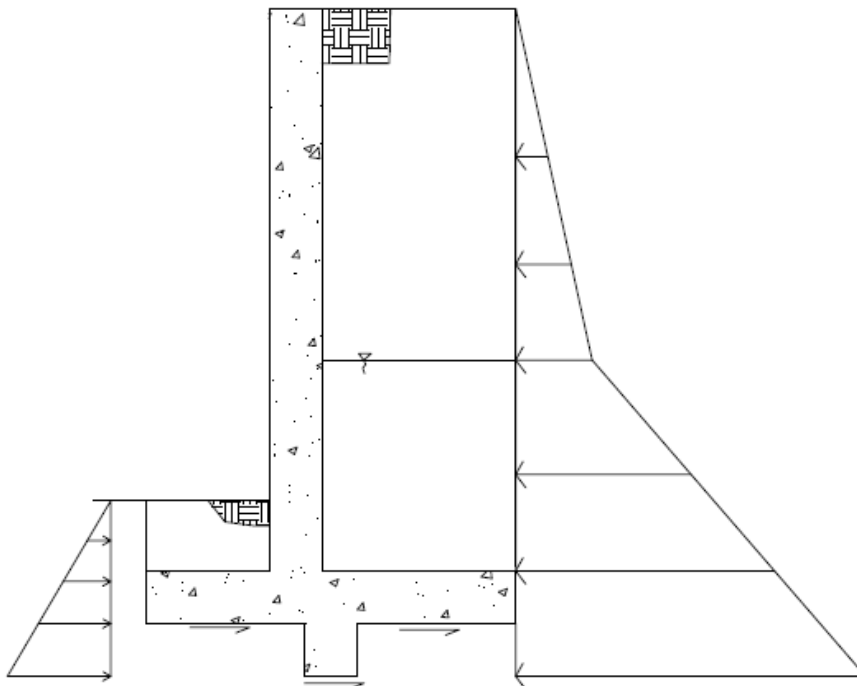
Figure 3: Sliding Forces Diagram

Both the passive soil pressure and friction resistance is accounted for.

$$Friction_{Resistance} := Total_{Weight} \cdot \mu \qquad PassiveSoil_{Resistance} := \frac{1}{2}K_p \cdot \gamma_{toe} \cdot D_{toesoil}^{2}$$

$$Sliding_{Resistance} := Friction_{Resistance} + PassiveSoil_{Resistance}$$

where:

- Total Weight = Total downward force acting on the footing due to soil, concrete and surcharge forces
- μ = Soil coefficient of friction defined in the [Model Settings - Solution](#) tab.

The sliding resistance is then compared against the sliding force caused by the hydrostatic pressure.The ratio of resisting forces to sliding forces is checked against the **SF** given in that individual load combination.

$$SSF := \frac{Sliding_{Resistance}}{Sliding_{Force}}$$

$$Sliding_{UC} := \frac{SF}{SSF}$$

Note:

- Deflection calculations for the top of the wall are not given.
- Sliding is not checked for retaining walls where the footing is restrained or if the top of the wall is propped.
- For strip footings the hydrostatic lateral forces from either the heel or the toe are not considered. Sliding is only resisted by frictional forces.

Stem Wall Design Considerations

Stem walls can be made of concrete or masonry materials. See the applicable sections for design details.

General Considerations

The stem wall design is analyzed along its height to come up with axial, shear and moment diagrams. The wall is designed for the worst case forces at the critical section. The program checks 20 locations over the height of the wall to determine the critical section. For a cantilevered uniform stem wall the critical section is always at wall/top of footing location. For a propped retaining wall and/or one with a batter this will be somewhere along the length of the wall.The program will take any externally applied loads in combination with hydrostatic pressures, surcharges, etc. The program will then factor the loads according to load combinations to come up with the axial, shear and moment diagrams.
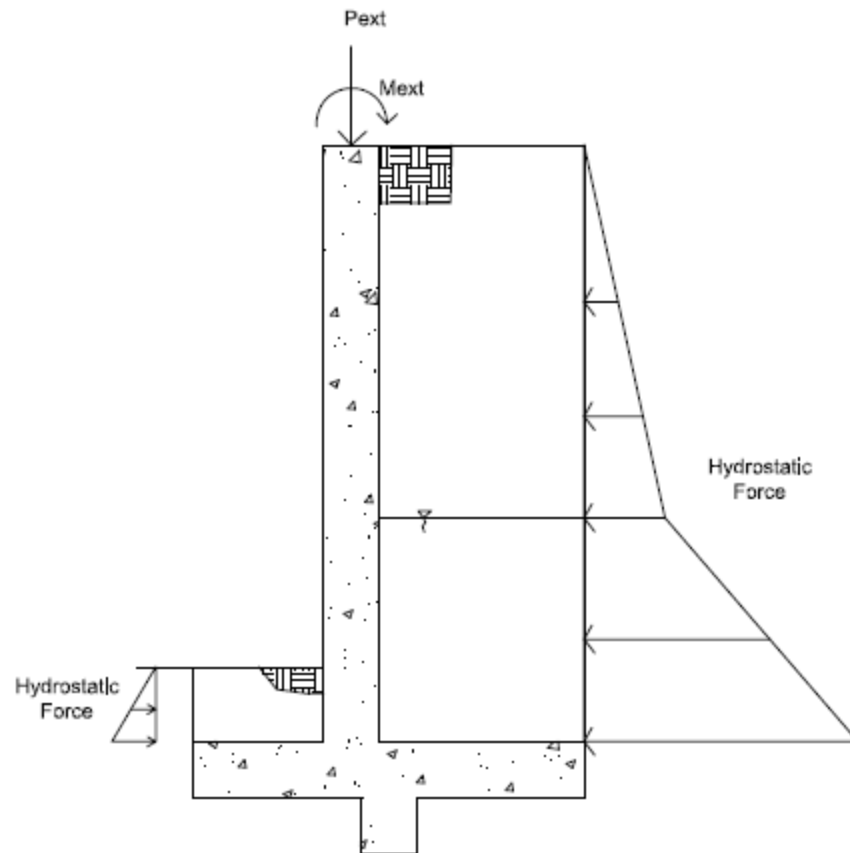
Note:

- Wall deflections are not considered in the analysis, because the combined effects of the flexural deflection of the stem and the rotational deflection of the overall wall/footing is generally small and hard to quantify.
- Strip footing stem walls require the soil height to be equal on both sides. Thus, hydrostatic forces are ignored. Only applied forces will induce any bending into a strip footing stem wall.
- If there is a sloping backfill, then the vertical portion of that load will be assumed to act at the top of the wall and be included in the axial force in the wall.
- **P-Delta** (big and little) is not considered for stem walls. Typically axial forces are small in retaining walls and P-Delta will have little effect.

Concrete Wall Considerations

For concrete walls the stem wall design is only considered for non-service load combinations. See Masonry Wall Considerations for masonry stem walls.

Axial/Bending Design

The program, given the wall thickness, reinforcement sizes, and max and min spacing and spacing increment, will design the wall reinforcement spacing to meet strength and minimum steel requirements.

The program will start with the maximum spacing. If that does not work for design, the spacing will drop by the spacing increment and be checked again. The program will work its way down until it reaches a spacing that meets all reinforcement requirements.The wall is checked considering the entire wall as a column. Thus, interaction and compression reinforcement is considered in the code check.Results are presented on a per foot basis, so the entire wall demand and capacity is calculated and then divided out on a per foot basis.

Note:

- The exact location and number of bars in the wall is used in capacity calculations. Use the **As Provided** in the detail report when doing any hand calculation verification.
- Reinforcement checks include the minimum reinforcement requirements of ACI 318-14 Section 11.7, ACI 318 Section 14.3 and CSA A23.3-04 Clause 14.
- For battered retaining walls the controlled section will be reported, though this may not be the location of maximum moment.
- For battered retaining walls the reinforcement on the battered face of the wall will be parallel to the face.

Moment and Axial Force Thresholds

The program will ignore axial forces and moments that are below a certain threshold. If the moment or axial force is deemed to be inconsequential to the code check then the program will not include the interaction of that force. There are two thresholds that are considered:

- Axial Force Threshold: If **Pu < 0.01*f'c*Ag** for that LC, then the axial force in the wall will be ignored for code checks for that LC.
- Moment Threshold: If **Mu < 0.01*d*Pu** for that LC, then the moment in the wall will be ignored for code checks for that LC.

These two thresholds allow the concrete solver to work much more efficiently while having little to no effect on code check values.

Shear Check (Concrete)

The program will do a concrete shear check. This check is taken at a distance "d" from the top of footing for a **cantilevered** wall. It is taken at a distance "d" from both the top of footing and from the top of the wall for a **propped cantilevered** wall.
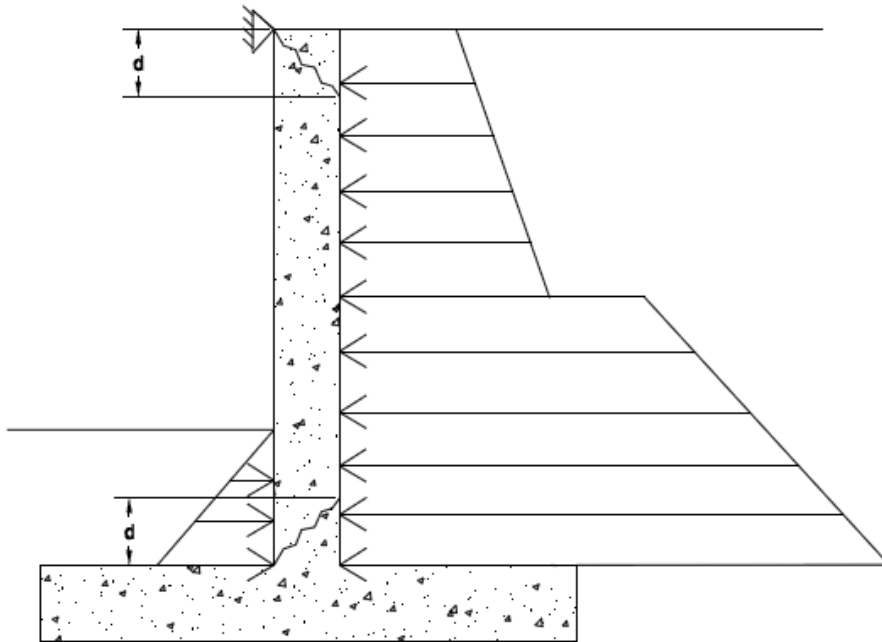
Note:

- For a battered retaining wall this check will occur at 20 sections along the height of the wall and the worst case code check will be given in the results.
- The shear check does not consider axial force. The program check is using the ACI 318 equation (and a similar equation for the **CSA A23.3-04**):

$$V_c := 2 \cdot \lambda \cdot \sqrt{f_c} \cdot b_w \cdot d$$

Masonry Wall Considerations

For masonry walls, per the [Model Settings](#) you can use either ASD or Strength code checks. stem wall design is only considered for non-service load combinations. Here we will break down both ASD and Strength aspects. For concrete walls the stem wall design is only considered for non-service load combinations. See [Concrete Wall Considerations](#) for concrete stem walls. Codes currently supported for masonry:

- The 2016 TMS 402 ASD and Strength codes.
- The 2013 ACI 530 ASD and Strength codes.

Note:

- All code references in this section refer to the **TMS 402-16** specification.

ASD Design Considerations

Axial Stress, fa

The axial stress in a wall due to axial forces, fa, is calculated as:

$$f_a := \frac{P}{A_n}$$

P = Total axial force in wall at the reported location

$$A_{nRMEH} := Eq_{solidthickness} \cdot L_{wall}$$

$$A_{nvNCMA} := A_{nvNCMAperfoot} \cdot L_{wall}$$

Note:

- The program does not design masonry for net tension forces. fa will equal 0 ksi in these cases.

Axial Capacity, Fa

The calculation of Fa is per either Equation 8-13 or 8-14, depending on the h/r ratio. These equations match Equations 8-18 and 8-19 if you assume Ast = 0. RISA conservatively uses only the masonry in calculating the compression capacity. The equations are as follows:

if h/r <= 99

$$F_a := \frac{1}{4} \cdot f_m \cdot \left(1 - \left(\frac{h}{140 \cdot r}\right)^2\right)$$

if h/r > 99

$$F_a := \frac{1}{4} \cdot f_m \cdot \left(\frac{70 \cdot r}{h}\right)^2$$

where:

- r is taken from the UBC-97 Table 21-H-1 (concrete masonry units) and Table 21-H-2 (clay masonry units).
- h = Total height of wall

Note:

- The program does not design masonry for net tension forces. If there is net tension, Fa will still equal the compressive capacity of masonry.

Stresses fb and fs

The masonry bending stresses are referenced in UBC Section 2107.2.15 and are as follows:

$$f_b := \frac{2 \cdot M}{j \cdot k \cdot b \cdot d^2} \qquad f_s := \frac{M}{A_s \cdot j \cdot d}$$

where:

$$\rho := \frac{A_s}{A_n} \qquad n := \frac{E_s}{E_m}$$

$$k := \sqrt{2 \cdot \rho \cdot n + (\rho \cdot n)^2} - \rho \cdot n$$

$$\rho := \frac{A_s}{A_n} \qquad n := \frac{E_s}{E_m}$$

$$j := 1 - \frac{k}{3}$$

b = spacing of reinforcement

M = moment over a width b

However, if you are using a partially grouted wall where the neutral axis passes through the webs of your masonry, then RISA will do a T-section analysis to define the section properties. We use a similar analysis as if you were doing a t-beam analysis for a concrete tee section. For more information on this, see "Design of Reinforced Masonry Structures" by Narendra Taly, copyright 2001, example 6.3, P6.61.

T-SECTION ANALYSIS
kd   : **1.638**   **in**
Flange thickness: **1.375**   **in**
Web width   : **8.5**   **in**
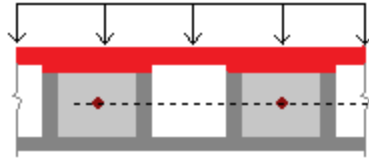
The area in red is shown as the compression block in the image above.

Bending Capacity, Fb

For unreinforced masonry, the Equation 8-15 is:

$$F_b := \frac{1}{3} \cdot f_m$$

For reinforced masonry, Section 8.3.4.2.2 states:

$$fa + fb <= 0.45*fm$$

Because of this provision, RISA defines:

$$F_b := 0.45 \cdot f_m$$

Steel Capacity, Fs

Section 8.3.3.1 defines the allowable steel stress, Fs.

Shear Stress, fv

This stress is calculated from Equation 8-21:

$$f_V := \frac{V}{A_{nv}}$$

$$V = \text{Shear out of plane, per foot}$$

$$A_{nvRMEH} := Eq_{solidthickness} \cdot 1 \cdot ft$$

$$A_{nvNCMA} := A_{nvNCMAperfoot}$$

Shear Capacity, Fv

The program calculates the capacity, Fv, from Equation 8-22 shown above, except that only the Fvm term is considered. There is no way to add shear reinforcing steel.The program also checks to verify we do not exceed the **Fv max** value from Equations 8-23 and 8-24 (or interpolation between them) that is reported in the detail report.

Strength Design Considerations

Axial Strength, Pn at max Mom

Here there are two different possible calculations.If there is out-of-plane moment on the walls (from a retaining wall for example) this is calculated from Section 9.3.5.4.2 as follows:

- If h/t < 30, **Pn at max Mom** = 0.20*f'm*An

- If h/t > 30, **Pn at max Mom** = 0.05*f'm*An

If there is no out-of-plane moment (from a strip footing for example) this is calculated from Equations 9-11 and 9-12:

if h/r <= 99:

$$P_n := 0.8 \cdot \left(0.8 \cdot A_{n1} \cdot f_m \cdot \left(1 - \left(\frac{h}{140 \cdot r} \right)^2 \right) \right)$$

if h/r > 99:

$$P_n := 0.8 \cdot \left(0.8 \cdot A_n \cdot f_m \cdot \left(\frac{70 \cdot r}{h} \right)^2 \right)$$

Note:

- t from the above equations is taken as the Block Nominal Thickness.
- h = Height of wall region

Moment Strength, Mn

This calculation comes from the Commentary of Section 9.3.5.2 as follows:

$$M_n := \left(\frac{P_u}{\phi} + A_s \cdot f_y \right) \cdot \left(\frac{t_{sp} - a}{2} \right) + A_s \cdot f_y \cdot \left(d - \frac{t_{sp}}{2} \right)$$

where:

$$a := \frac{A_s \cdot f_y + \dfrac{P_u}{\phi}}{0.80 \cdot f_m \cdot b}$$

b = reinforcement spacing

tsp = actual block thickness

The program is actually using an interaction diagram for out of plane bending as well. However, the code places a limit on axial force from Section 9.3.5.4.2. This limit essentially means that only the lower portion of the interaction diagram will be used. In this lower portion of the interaction diagram, the bending capacity changes in a linear fashion with respect to axial force.
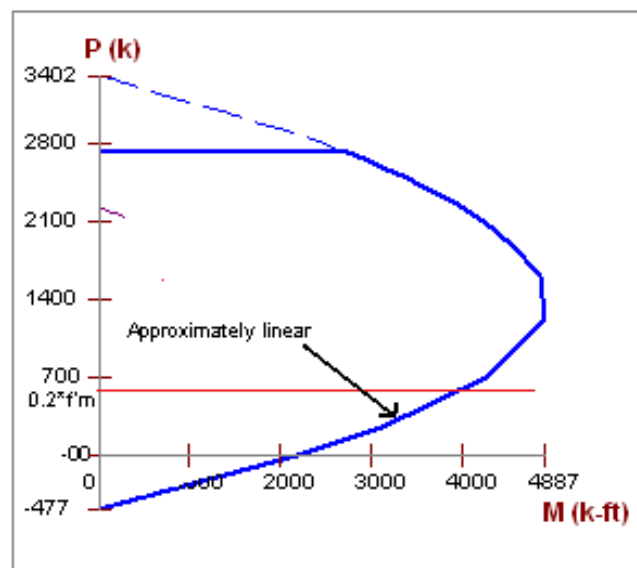
Thus, the equation above is nearly identical to the value that the program's interaction diagram will produce.

Note:

- In the case of high axial tension, it is possible that moment capacity will near zero.
- In partially grouted walls an effective width is calculated for b from Section 5.1.2

ASD Masonry Design For Unreinforced Walls

Compression Check (per Equation 8-11)

$$Unity := \frac{f_a}{F_a} + \frac{f_b}{F_b}$$

where:

- f_a and F_a are computed the same way as for [reinforced masonry](#).
- f_b = M/S. S = S_x value from NCMA TEK 141B (regardless of Wall Area Method designation).
- F_b = 1/3*f'm per Equation 8-15

Tension Check (per Section 8.2.4.2)

$$Unity := \frac{(f_b - f_a)}{F_{bt}}$$

where:

- Fbt = Allowable flexural tensile stress per Table 8.2.4.2

Slenderness Check (per Equation 8-12)

$$Unity := \frac{P}{0.25 \cdot P_e}$$

where:

$$P_\epsilon := \frac{\pi^2 \cdot E_m \cdot I}{h^2} \cdot \left(1 - 0.577 \, \frac{e}{r}\right)^3$$

- Per Equation 8-16.
- I comes directly from NCMA TEK 141B for both Wall Area Methods, NCMA and RMEH.
- e is always assumed equal to 0.
- h = Height of wall

Note:

- If using an **Unreinforced** wall the program will ALWAYS use an ungrouted wall for design, regardless of what is set in the masonry wall design rules.

Shear Design (per Equation 8-21)

$$f_V := \frac{V}{A_{nv}}$$

V = Shear out of plane, per foot

$$A_{nvRMEH} := Eq_{solidthickness} \cdot 1 \cdot ft$$

$$A_{nvNCMA} := A_{nvNCMAperfoot}$$

Note:

- We do not use Equation 8-17 to calculate shear stress.

The program calculates the capacity, Fv, from Equation 8-22 shown above. The program also checks to verify we do not exceed the **Fv max** value from Equations 8-23 and 8-24 (or interpolation between them) that is reported in the detail report.

Note:

- Unreinforced strength design is not available.

Footing Design Considerations

Here we will discuss some of the details of footing and dowel design (shear friction). Note that footing design is only considered for non-service load combinations. Shear friction checks are strength level checks.

Shear Friction Check (for dowels between the wall and footing)

If the stem wall is not poured monolithically with the footing then a shear friction check is required at the base of the wall. This check considers the dowels from the footing to the wall and the program assumes that these dowels match the vertical reinforcement (either on the inside face or both faces; see the Wall Footing Definitions topic for more information).
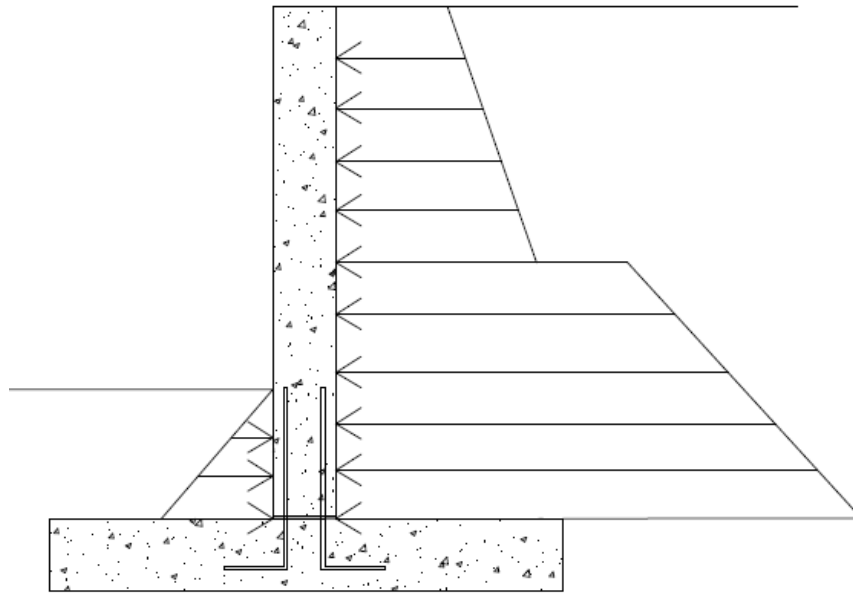
ACI 318 Codes

This check is done using ACI 318-14 Equation 22.9.4.2 (ACI 318-11 Equation 11-25) for shear friction:

$$V_n := A_{vf} \cdot f_y \cdot \mu$$

where:

- A_{vf} = the reinforcement that crosses the shear plane
- μ = 1.0*λ for intentionally roughened surface; 0.6*λ for a smooth surface

CSA A23.3-04 Code

This check is done using the CSA A23.3-04 shear friction capacity equation (11-24):

$$\sigma := \rho_v \cdot f_y \cdot \sin\left(\alpha_f\right) + \frac{N}{A_{wall}}$$

$$v_r := \min\left(\lambda \cdot \phi_c \cdot \left(c + \mu \cdot \sigma\right), 0.25 \cdot \phi_c \cdot f_c\right) + \phi_s \cdot \rho_v \cdot f_y \cdot \cos\left(\alpha_f\right)$$

$$Vn := A_{wall} \cdot v_r$$

where:

- N = unfactored dead load acting on the wall (1.0*DL)
- α_f = angle between shear plane and reinforcement (always taken as 90 degrees)
- ρ_v = ratio of shear friction r/f to total wall area (from Eq 11-27)
- λ = factor for lightweight concrete
- c and μ = taken from Clause 11.5.2
- A_{vf} = the reinforcement that crosses the shear plane

ACI 530-13 Masonry Codes (ASD & Strength)

This code uses the ACI 318 procedure, so see above for that.

TMS 402-16 Masonry ASD

This check is done using Equations 8-28 and 8-29, depending on the M/Vd ratio:

$$F_{f0.5} := \frac{\mu \cdot (A_{sp} \cdot F_s + P)}{A_{nv}}$$

$$F_{f1.0} := \frac{0.65 \cdot (0.6 \cdot A_{sp} \cdot F_s + P)}{A_{nv}}$$

where:

- μ = coefficient of friction, 1.0 rough, 0.7 smooth
- A_{sp} = area of reinforcement within net shear area
- A_{nv} = net shear area
- F_s = Allow reinforcement stress in tension/compression
- P = axial load

adfasd

Note:

- The program will interpolate M/Vd between 0.5 and 1.0

TMS 402-16 Masonry Strength

This check is done using Equations 9-33 and 9-34, depending on the M/Vd ratio:

$$V_{nf0.5} := 0.42 \cdot f_m \cdot A_{nc}$$

$$V_{nf1.0} := \mu \cdot (A_{sp} \cdot f_y + P)$$

where:

- μ = coefficient of friction, 1.0 rough, 0.7 smooth
- A_{sp} = area of reinforcement within net shear area
- f_y = yield stress in steel
- A_{nc} = net cross-sectional area between neutral axis of bending and fiber of max compressive strain, = b_{eff}*c.
- F_s = Allow reinforcement stress in tension/compression
- P = axial load

Note:

- The program will interpolate M/Vd between 0.5 and 1.0

Overall Shear Friction Notes:

- The coefficient in the μ factor is defined by the **Wall/Foot Continuity?** entry in the Wall Footing Definitions spreadsheet.
- The shear friction check is omitted for strip footings, as loads are generally vertical as opposed to lateral.

Soil Pressure Considerations

Computing the soil pressures under the footing is the first step here. To calculate the pressure under a footing requires both the moment (M) and the axial force (P) on the centerline of the footing at the base.An alternative hand calculation approach is to take the sum of moments (factored) from the overturning check and divide that out by the total axial force (factored). This gives you the resultant force location from the toe end.

$$x := \frac{M}{P}$$

Note:

- For propped cantilevers the moment, M, used in this calculation is equal to the moment at the base of the footing. Hence, we assume a fixed (bottom of footing) - pinned (top of wall) condition and calculate the moment at the base using the hydrostatic forces.

From the x-value you can calculate the eccentricity from the footing centerline as:

$$e := \frac{L_{foot}}{2} - x$$

From here, if the resultant is in the middle third of the footing the force distribution is:

$$q_{max} := \frac{P}{L_{foot}} + \frac{6 \cdot (P \cdot e)}{L_{foot}^2} \qquad q_{min} := \frac{P}{L_{foot}} - \frac{6 \cdot (P \cdot e)}{L_{foot}^2}$$
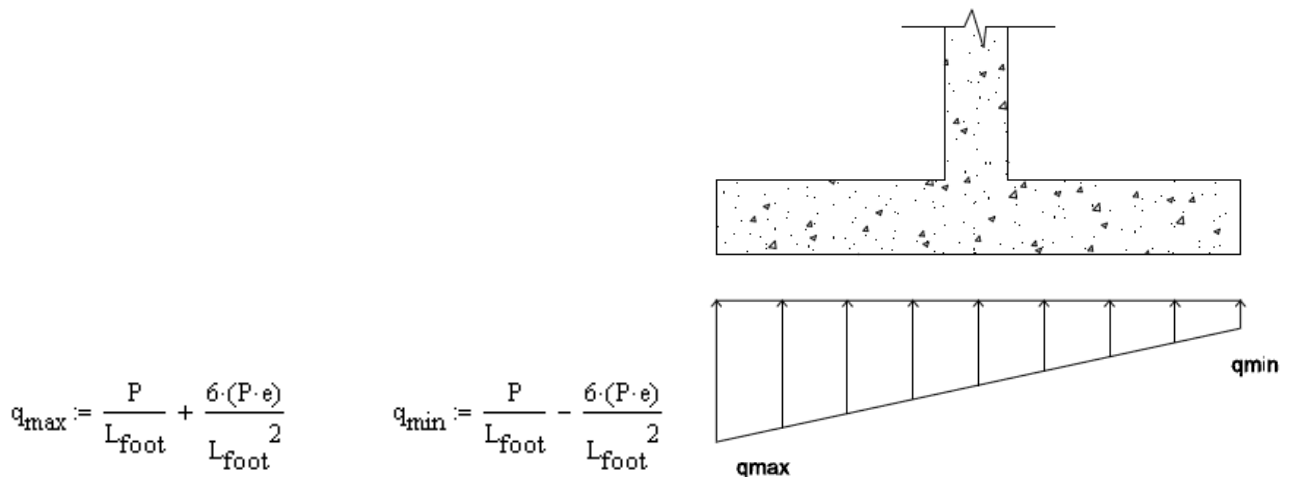
If the resultant is outside the middle of the footing the force distribution is:

$$q_{max} := \frac{4 \cdot P}{3 \cdot (L_{foot} - 2 \cdot e)} \qquad L_{soil} := 3 \cdot x$$
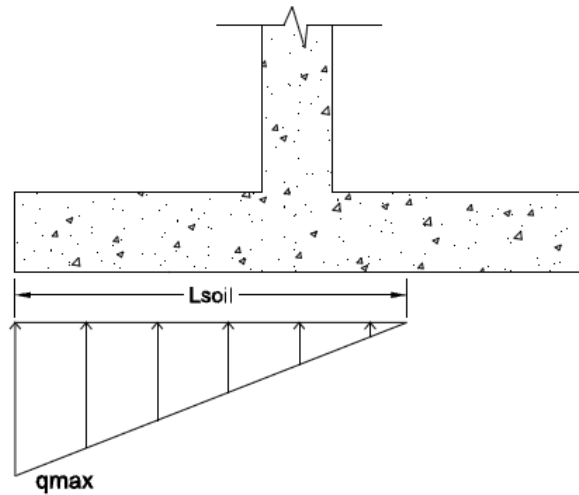
The soil (and surcharge) bearing down on the soil is also considered:
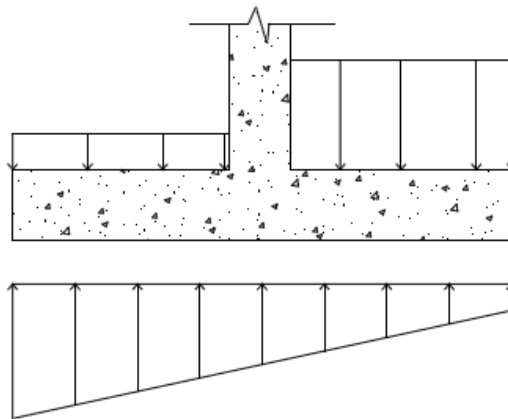
Note:

- Many design examples may disregard certain portions of load for conservatism or for ease of calculation. However, the program uses the exact loading conditions defined.
- Per ACI 318-14 Section 13.3.1.2 (ACI 318-11 Section 15.7) the depth of footing above bottom reinforcement shall not be less than 6".

Flexural Design

Flexural design for wall footings is considered assuming that the heel and toe portions of the footing are designed as cantilevered beams from the face of the retaining wall. There are generally two deflected shapes that will occur for footings depending on the loading.
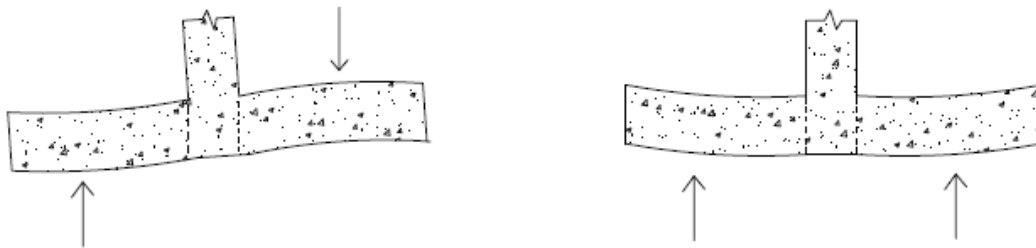
The left image is likely the most common for retaining walls. The right image will generally occur for strip footings.In either case, the program considers the moment at the face of both the heel and toe and designs

reinforcement for the worst case load combination. Both the flexural reinforcement (parallel to the footing) and the shrinkage and temperature reinforcement (perpendicular to the footing) are designed for minimum steel requirements as well.

Note:

- If the "d" distance for the footing is very small the program may not be able to properly calculate an As Req'd. In these cases a "-1" will be displayed and an area of 0 will be used. The program will, however, still meet minimum reinforcement requirements and base the capacity on those values. So in many cases your footing may still work.

Shear Checks

Shear design of footings also considers both the toe and heel sections as cantilevered beams. The difference here is where the shear failure will occur. The left image below shows the standard shear failure planes typically loaded retaining walls. The right image below shows the standard shear failure planes typically loaded strip footings. The program will check the loading of the footing and check the shear at the appropriate location.
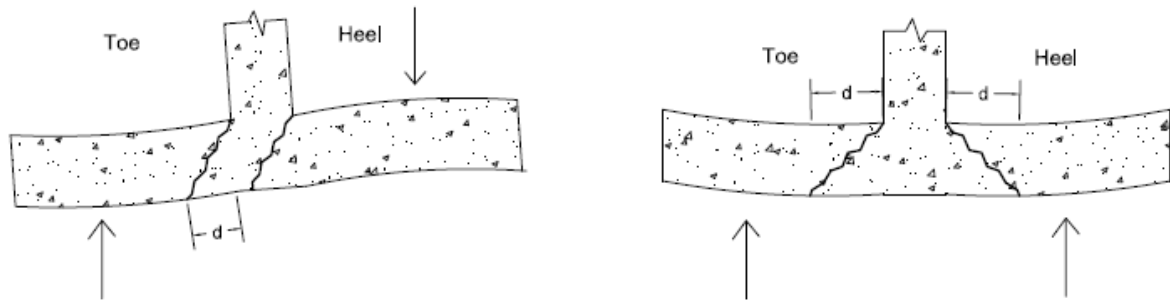
From the left image you see that the shear check would occur at a distance "d" from the face of the toe and right at the face of the heel.From the right image you can see that both the heel and the toe would have their shear check occur at a distance "d" from the face of wall.

Note:

- The loading (top and bottom of footing) will always be taken from the location where the code check is taken from.

Wall Footing Results

The locations for output of wall footings are the Wall Footing Results browsers and the wall footing detail report.

- See the Wall Footing Definitions topic for more information on defining wall parameters.
- See the Wall Footings - Modeling topic for more information on modeling requirements.
- See the Wall Footings - Design topic for more information on calculation details.

Wall Footing Results Browser

To view the wall footing results browser, click the **Wall Footing Results** from the Results toolbar, or go to **Results - Wall Footing Results**. These output browsers are meant to summarize the results for wall footings. To see more detailed information view the detail report by clicking the [Detail Report for Current Item] button near the top of the screen when you are in the **Wall Footing Results** browser. Alternatively, press the [Detail] button on the left side of the screen (when in a graphical view) and click on the line that defines the wall footing.

Footing Code Checks

The Wall Footing Code Check spreadsheet gives the flexural and shear checks and capacities for both the footing toe and heel design. To get more information on the design of these portions, see the **Wall Footings - Design** topic in the Footing Design Considerations sections and also see the detail report.

Concrete Wall Results

The Wall (Concrete) spreadsheet gives bending and shear code checks, as well as capacity values, for the wall stem. To get a detailed sketch of this information see the detail report.

Masonry Wall Results

Service

	Wall Panel	Combined UC	LC	Shear UC	LC	UC Max Dowel	Fa[ksi]	Fb[ksi]	Fv[ksi]
1	WF1	.825	1	.438	1	.1	.363	.675	.046
2	WF5	.958	1	.844	1	.2	.363	.675	.044
3	WF6	.958	1	.844	1	.2	.363	.675	.044

Masonry Wall Code Check (ACI 530-13: ASD). Tabs: Footing | Wall (Concrete) | Wall (Masonry) | Geometry | Steel | Stability

Strength

	Wall Panel	Axial UC	LC	Bending UC	LC	UC Max Dowel	Pn*phi[k-ft]	Mn*phi[k-ft]
1	WF1	.316	-1-1	.707	N/A	.9	41.85	6.449
2	WF5	.172	-1-11981376	.996	N/A	.6	41.85	4.582
3	WF6	.172	-1-15613907	.996	N/A	.6	41.85	4.582

Masonry Wall Code Check (ACI 530-08: Strength). Tabs: Footing | Wall (Concrete) | Wall (Masonry) | Geometry | Steel | Stability

The Wall (Masonry) spreadsheet gives bending and shear code checks for the wall stem. To get a detailed sketch of this information see the detail report.

Geometry

	Label	Wall Height[ft]	Wall Thick[in]	Toe Length[ft]	Heel Length[ft]	Foot Thick[in]	Key Depth[in]	Key Width[in]
1	WF1	7	12	3	4	18	0	0
2	WF2	10	12	3	4	16	18	32
3	WF3	4	12	3	3	18	NC	NC
4	WF4	4	12	3	3	18	NC	NC

Wall Footing Geometry. Tabs: Footing | Wall (Concrete) | Wall (Masonry) | Geometry | Steel | Stability

The Geometry spreadsheet gives a summary of the overall geometry of the wall and footing. This is a reiteration of input parameters.

Steel

	Label	Int Wall Verts	Ext Wall Verts	Wall Horz Bars	Foot Top Bars	Foot Bot Bars	Foot Long Bars	Dowel Bars
1	WF1	#5@72"in			#6@18in	#6@18in	#4@18in	#5@72in
2	WF2	#6@12in		#5@18in		#6@14in	#5@18in	#6@12in
3	WF3	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in
4	WF4	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in

Wall Footing Steel. Tabs: Footing | Wall (Concrete) | Wall (Masonry) | Geometry | Steel | Stability

The Steel spreadsheet gives all of the steel requirements for the wall footing, including wall, footing and dowel reinforcement.

Stability

The Wall Footing Stability spreadsheet gives code check information for soil bearing, overturning and sliding checks for wall footings.

For the Soil Bearing Checks the **Bearing UC** column is meant to act as the code check value. Anything exceeding 1.0 here means you are failing this check.

Note:

- The **Allowable Bearing Pressure** value is defined for each soil region in the model. If you are using the default value this can be found in the Model Settings - Solution tab. If your wall footing lands on multiple soil regions, the smallest allowable bearing pressure will be used.
- If you fail the soil bearing checks we will not give a full solution to your model. Since you are failing the soil this is not necessary. We will, however, give you the soil pressure diagrams so you can decide how you want to reconcile this failure.

For more information on the specifics of the overturning and sliding checks, see the **Wall Footings - Design** topic in the Serviceability Considerations section and also the detail report for more information.

Graphic Results

The reactions for the wall footing can viewed in the Model Display Options dialog. For wall footings, only the vertical reactions are given. The total vertical load is calculated and the display divides that value by two and places reactions at the ends of the walls.
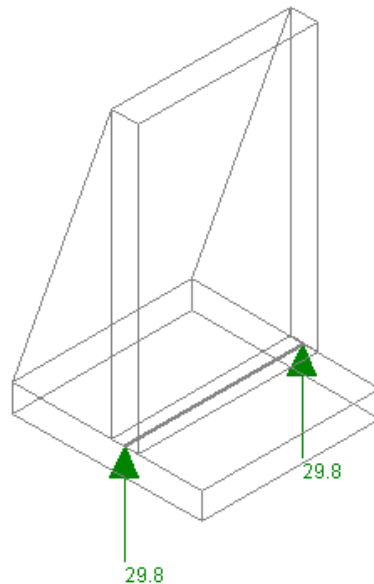
Note:

- Because there may be the presence of a key and because the program doesn't know which reactions you are looking for laterally, the program does not give these values. To find these values look at the detail report for that wall footing.

Wall Footing Detail Report Information

The Retaining Wall Detail Report gives detailed information about geometry, materials, criteria, design and stability checks and sketches associated with this retaining wall. Here we will walk through each section of the report.
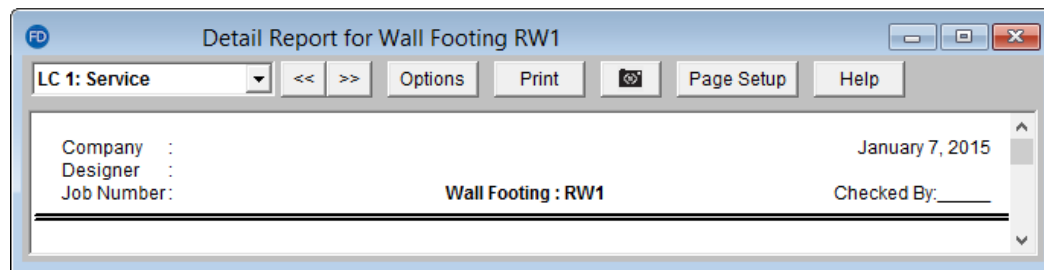
Header

The header of the detail report is useful for navigating and viewing information in the wall footing reports.

- The **LC dropdown** allows you to view the overall loading diagram and the axial, shear and moment diagrams for the stem by individual load combination. Every other item in the detail report is enveloped or results are given for each LC individually.
- The **double arrows** allow you to click through detail reports for the different wall footings in the model.
- The **Options** button opens up the report options dialog. This allows you to customize exactly what you see in the detail report.
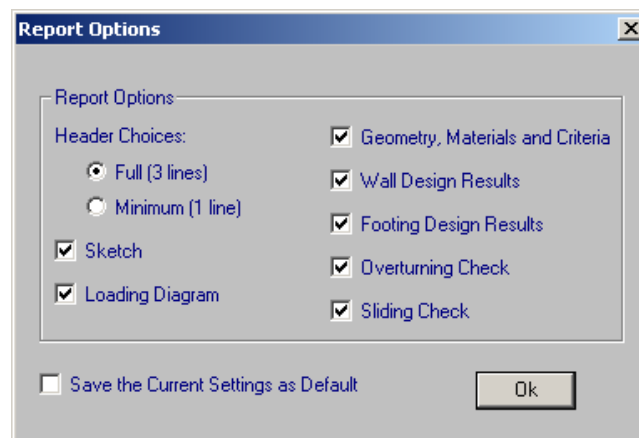
- The **Print** button opens you to the print dialog to print the report.
- - This button will allow you to take a snapshot of the current detail report you are viewing so that it can be added to a report. View the Printing topic for more information.
- The **Page Setup** button allows you to modify page margins.
- The **Help** button opens you to the detail report section in the Wall Footing Results topic.

Sketch
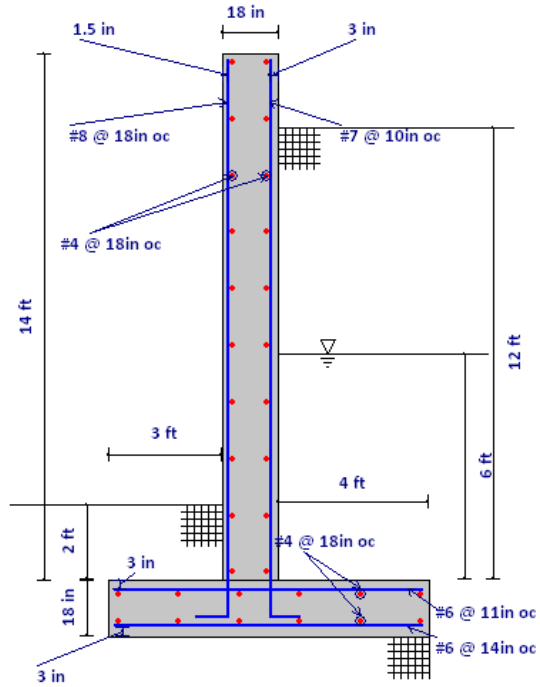
The sketch section gives a full sketch of wall footing dimensions, reinforcement size and spacing, and cover dimensions.

Loading Diagram
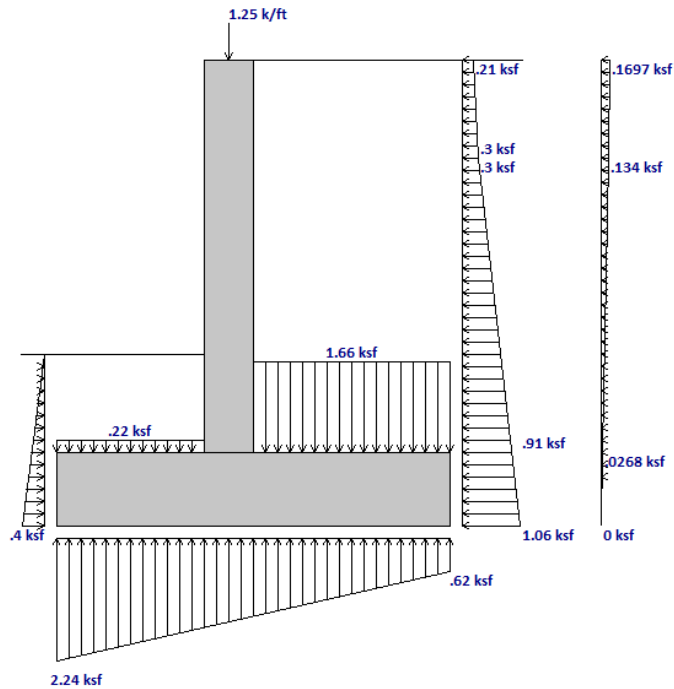
The loading diagram section gives a visual of all the calculated pressures and loading on the wall. This loading information is what is used to provide the design checks for the wall footing. Loads shown on the top of the wall footing are any externally applied loads to the wall, divided out on a per foot basis. For more information on the calculation of these pressures see the Footing Soil Pressure Considerations and Wall Pressure Considerations information in the **Wall Footings - Design** topic.

Note:

- This diagram is based on the load combination shown in the dropdown in the upper left-hand corner of the report.
- If you have a sloped backfill on your retaining wall, then a portion of the hydrostatic loads will be applied in the vertical direction of your wall. The cosine of these forces will be applied horizontally as a pressure. The sine of these forces will be applied as a vertical force that is shown along the inside face of the wall and conservatively applied at the top of the wall.
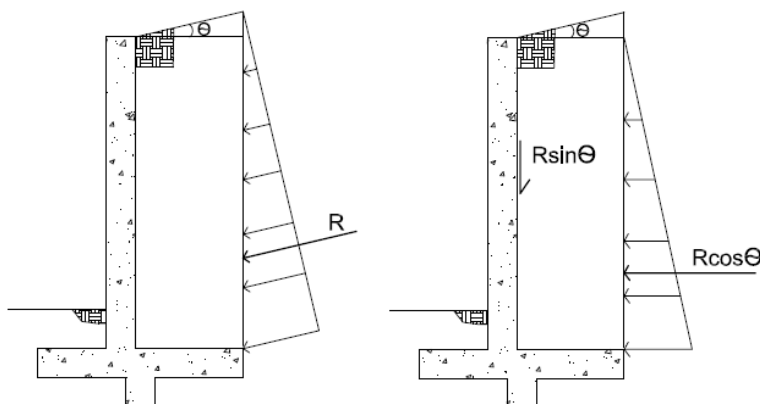
Geometry, Materials and Criteria

<u>Concrete Wall</u>

Geometry, Materials and Criteria

|					
Start Point	: N1	Design Code	: ACI 318-14	Wall Ext Bar	: #6@6 in
End Point	: N2	Wall Material	: Concrete	Wall Int Bar	: #6@12 in
				Wall Horz Bar	: #6@12 in
Wall Length	: 10 ft	Wall Weight	: .145 k/ft^3	Wall Bar Location	: Each Face
Wall Height	: 8 ft	Wall Conc fc	: 4 ksi	Wall Outer Bar	: Horizontal
Wall Thickness	: 14 in	Wall Steel fy	: 60 ksi		
Wall Int Cover	: 2 in	Foot Weight	: .145 k/ft^3	Foot Top Bar	: #6@12 in
Wall Ext Cover	: 2 in	Foot Conc fc	: 4 ksi	Foot Bot Bar	: #6@12 in
		Foot Steel fy	: 60 ksi	Foot Long Bar	: #6@12 in
Foot Toe Len.	: 2 ft	Heel Soil Height	: 8 ft	Foot bar Location	: Each Face
Foot Heel Len.	: 5 ft	Toe Soil Height	: 3 ft		
Foot Thickness	: 18 in	Backfill Angle(θ)	: 0 deg	γ_{Toe}	: .1 k/ft^3
Foot Top Cover	: 2 in	Water Height	: 2 ft	Φ_{Toe}	: 32 deg
Foot Bot Cover	: 3 in	Heel Surcharge	: .25 ksf	$K_{Lat\ Toe}$	: .47 (At Rest)
Wall Batter ?	: No	γ_{Heel}	: .125 k/ft^3		
		Φ_{Heel}	: 32 deg		
Shear Key ?	: NO	$K_{Lat\ Heel}$	: .47 (At Rest)		
Wall Propped ?	: NO	$\gamma_{Heel\ Sat}$	: .125 k/ft^3		
		$\Phi_{Heel\ Sat}$	: 32 deg		
Soil Method	: Coulomb	$K_{Lat\ Heel\ Sat}$	: .47 (At Rest)		

Masonry Wall

Geometry, Materials and Criteria

General/Geometry

Start Point	: **N1**
End Point	: **N2**
Wall Length	: **15 ft**
Wall Height	: **8 ft**
Wall Propped ?	: **NO**

Footing Criteria

Concrete Code	: **ACI 318-14**
Foot Toe Len.	: **3 ft**
Foot Heel Len.	: **4 ft**
Foot Thickness	: **18 in**
Foot Top Cover	: **3 in**
Foot Bot Cover	: **3 in**
Foot Top Bar	: **#6@18 in**
Foot Bot Bar	: **#6@18 in**
Foot Long Bar	: **#4@18 in**
Foot bar Location	: **Each Face**
Shear Key ?	: **YES**
Key Width	: **12 in**
Key Depth	: **12 in**
Key Offset	: **0 in**

Wall Materials & Criteria

Design Code	: **ACI 530-13: ASD**
Material	: **Masonry**
Reinforced	: **Yes**
Blk Nom Width	: **12"**
Blk Grouting	: **Fully Grouted**
Wall Vert Bar	: **#5**
Wall Bar Location	: **Center**
Wall Grt/Bar Spacing	: **8"**
Cover	: **5.5 in**
Eq Sld Thickness	**11.625 in**
An c/c of reinf	: **93 in2**
Masonry f'm	: **1.5 ksi**
Masonry Em	: **1350 ksi**
Steel fy	: **60 ksi**
Steel E	: **29000 ksi**
Blk Material	: **Conc 115 pcf**
Grt Weight	: **140 pcf**
Wall Dead Wt	: **.124 ksf**
Mortar Type	: **Type M or S**
Cement Type	: **Portland, Lime/Mortar**
Wall Area	: **NCMA**

Soil Criteria

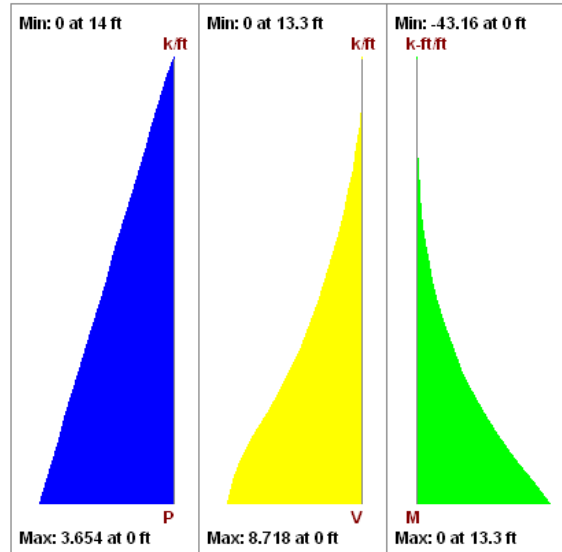|	
Heel Soil Height	: **8 ft**
Toe Soil Height	: **2 ft**
Backfill Angle(α)	: **0 deg**
Water Height	: **6 ft**
Heel Surcharge	: **.5 ksf**
γ_{Heel}	: **.111 k/ft^3**
Φ_{Heel}	: **30 deg**
$K_{Lat\ Heel}$	: **.416 (Active)**
$\gamma_{Heel\ Sat}$	: **.156 k/ft^3**
$\Phi_{Heel\ Sat}$	: **30 deg**
$K_{Lat\ Heel\ Sat}$	: **.416 (Active)**
γ_{Toe}	: **.111 k/ft^3**
Φ_{Toe}	: **30 deg**
$K_{Lat\ Toe}$	: **1.035 (Passive)**
δ_{wall}	: **15 deg**
Kh	: **.5 g**
K_{AE}	: **1.334 (Active)**
$P_{Aseismic}$	: **3.259 k/ft**

The geometry, materials and criteria section gives input information from the Wall Footing Definitions and design information such as the reinforcement spacing and K calculations (see the Soil Considerations topic).

Wall Design

Concrete Wall


```
Wall Design
LC 2 DIAGRAMS
```

Min: 0 at 14 ft   **Min: 0 at 13.3 ft**   **Min: -43.16 at 0 ft**

    k/ft   k/ft   k-ft/ft

P   **V**   **M**

Max: 3.654 at 0 ft   **Max: 8.718 at 0 ft**   **Max: 0 at 13.3 ft**

ACI 318-08 Code Check

AXIAL/BENDING DETAILS

UC Max Int(+z)	: .956
Location	: 0 ft
Gov Pu Int(+z)	: 0 k/ft
phi *Pn Int(+z)	: 0 k/ft
Gov Mu Int(+z)	: -43.165 k-ft/ft
phi *Mn Int(+z)	: 45.155 k-ft/ft
phi eff. Int(+z)	: .9
Gov LC Int(+z)	: 2
UC Max Ext(-z)	: .001
Location	: 14 ft
Gov Pu Ext(-z)	: .365 k/ft
phi *Pn Ext(-z)	: 323.693 k/ft
Gov Mu Ext(-z)	: 0 k-ft/ft
phi *Mn Ext(-z)	: 0 k-ft/ft
phi eff. Ext(-z)	: .65
Gov LC Ext(-z)	: 2

SHEAR DETAILS

UC Max	: .769
Location	: 12.786 ft
Gov Vu	: 5.07 k/ft
phi*Vn	: 14.357 k/ft
Gov LC	: 2
UC Max Dowel	: .04
μ	: 1
A_{vf}	: 1.245 in^2/ft
Dowel Bars	: #7@10in Int #8@18in Ext

RESULTS FOR FULL WALL

As Provided(H)	: 3.927 in^2 (20 #4)
Rho Provided(H)	: .0013
As min(H)	: 6.048 in^2
Rho min(H)	: .002
Int As Provided(V)	: 72.158 in^2 (120 #7)
Int rho Provided(V)	: .0033
Ext As Provided(V)	: 52.622 in^2 (67 #8)
Ext rho Provided(V)	: .0024
As min(V)	: 32.4 in^2
rho min(V)	: .0015

ENVELOPED WALL FORCES (NOT USED FOR DESIGN)

M in Plane   : **0 k-ft (LC: 1)**   V in Plane   : **0 k (LC: 1)**

As provided(H) does not meet code required minimum

The **Axial, Shear** and **Moment Diagrams** are based on each load combination from the drop-down list.

The **Axial/Bending Checks** are given for both the interior and exterior faces of the wall. Note that only a propped cantilevered wall will have any bending in the "Gov Mu Ext", unless there is a large externally applied load in the direction opposite the soil loading. The code checks here are based on a concrete wall interaction diagram and bending capacity checks also consider compression reinforcement.

The **Shear Details** give both the concrete shear check (at a distance "d" from the base of the wall) and the dowel shear checks (at the base of the wall). Note that the dowel checks are only done in the case of the retaining wall and footing not being defined as monolithic. See the Wall Footing Definitions - General topic for more information.
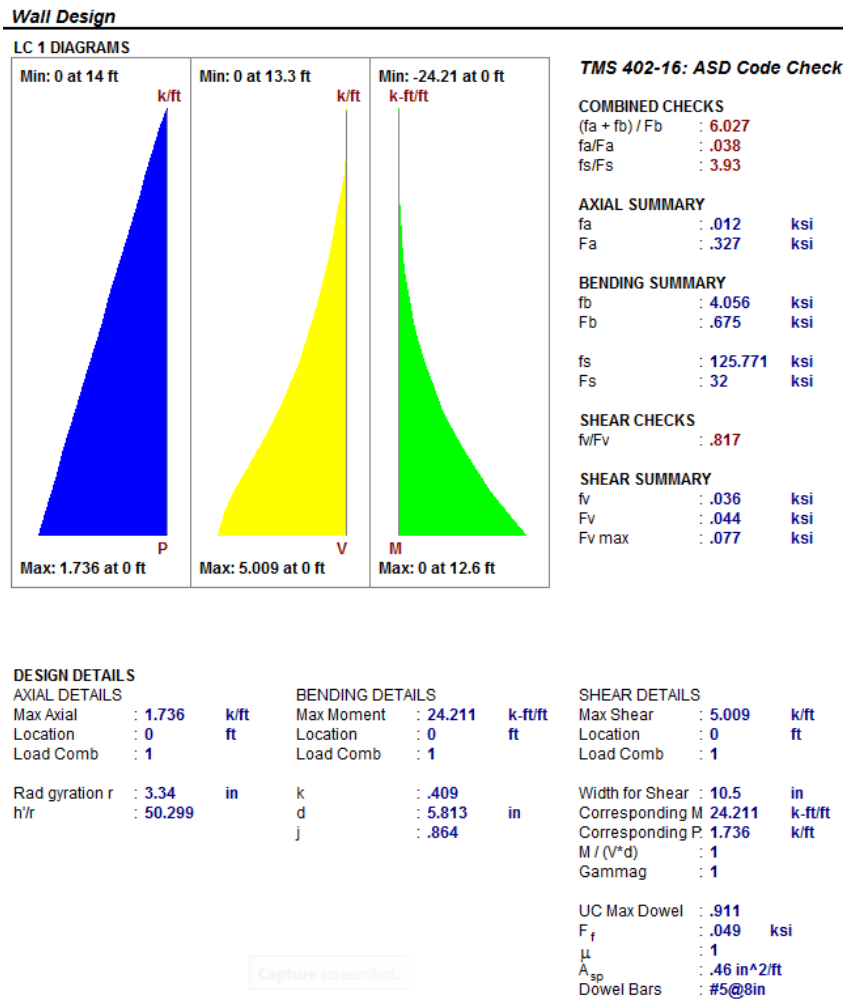
The **Results For Full Wall** section provides the minimum reinforcement requirements and the reinforcement provided for the wall.

The **Enveloped Wall Forces** section gives other forces found in the wall that the program did not explicitly use. For example, the wall footing will not be designed for in plane shears, but those forces will be reported here.

Note:

- If the minimum spacing requirements defined in the **Wall Footing Definition** does not meet minimum code requirements you will receive warning messages in the detail report.

Masonry Wall (ASD)

Wall Design

LC 1 DIAGRAMS

Min: 0 at 14 ft   Min: 0 at 13.3 ft   Min: -24.21 at 0 ft

k/ft   k/ft   k-ft/ft

P   V   M
Max: 1.736 at 0 ft   Max: 5.009 at 0 ft   Max: 0 at 12.6 ft

TMS 402-16: ASD Code Check

COMBINED CHECKS
(fa + fb) / Fb	: 6.027
fa/Fa	: .038
fs/Fs	: 3.93

AXIAL SUMMARY
fa	: .012	ksi
Fa	: .327	ksi

BENDING SUMMARY
fb	: 4.056	ksi
Fb	: .675	ksi
fs	: 125.771	ksi
Fs	: 32	ksi

SHEAR CHECKS
fv/Fv	: .817

SHEAR SUMMARY
fv	: .036	ksi
Fv	: .044	ksi
Fv max	: .077	ksi

DESIGN DETAILS

AXIAL DETAILS			BENDING DETAILS			SHEAR DETAILS		
Max Axial	: 1.736	k/ft	Max Moment	: 24.211	k-ft/ft	Max Shear	: 5.009	k/ft
Location	: 0	ft	Location	: 0	ft	Location	: 0	ft
Load Comb	: 1		Load Comb	: 1		Load Comb	: 1	
Rad gyration r	: 3.34	in	k	: .409		Width for Shear	: 10.5	in
h'/r	: 50.299		d	: 5.813	in	Corresponding M	24.211	k-ft/ft
			j	: .864		Corresponding P.	1.736	k/ft
						M / (V*d)	: 1	
						Gammag	: 1	
						UC Max Dowel	: .911	
						F_f	: .049	ksi
						μ	: 1	
						A_{sp}	: .46 in^2/ft	
						Dowel Bars	: #5@8in	

Capture screenshot.

ENVELOPED WALL FORCES (NOT USED FOR DESIGN)
M in Plane   : 0 k-ft (LC : 1)   V in Plane   : 0 k (LC : 1)

The **Combined Check Summary** gives maximum overall code checks considering the effects of both bending and axial stresses.

The **Axial Summary** and **Bending Summary** give the values used in the axial and flexural code checks at the location which controls the combined check (fa/Fa)+(fb/Fb).

For ASD design, the maximum bending stress in the flexural reinforcement is reported as **fs**, and the allowable steel bending stress as **Fs**. These values are given for the Load Combination and section that produce the maximum code check (fs/Fs).

The **Shear Summary** reports the maximum shear demand on the beam as **fv** or **Vu**. The allowable shear stresses are then reported as **Fv**

Design Details

- **Radius of Gyration (r)**: The out of plane radius of gyration for the wall.
- **h/r**: Slenderness ratio of the wall.

- **k**: This value multiplied by d gives the depth to the neutral axis.
- **d**: The depth between the extreme face of masonry in compression and the center of the tension reinf.
- **j**: This value multiplied by d gives the moment arm between the compression and tension resultants.
- **Width for Shear**: This is the value used to calculate the shear capacity. For fully-grouted walls, this is simply the center to center spacing of vertical reinforcement. For partially-grouted walls, this value is conservatively taken as the width of the grouted cell plus the thickness of the web and end wall on either side. An example of this can be seen in Example 6.3 (p6.65) of "Design of Reinforced Masonry Structures" by Taly, copyright 2001.
- **Corresponding M & P**: These are the moments and axial forces used to calculate the shear capacity. These are both conservatively taken as the maximum M & P in the entire wall, rather than the M & P at the location of maximum shear.
- **M/(V*d)**: This is the M/Vd ratio used to calculate the shear capacity.

See the Wall Footings - Design topic for information on a T-section analysis.

```
T-SECTION ANALYSIS
kd                 : 1.609    in
Flange thickness:  1.375    in
Web width          : 8.5      in
```

Masonry Wall (Strength)
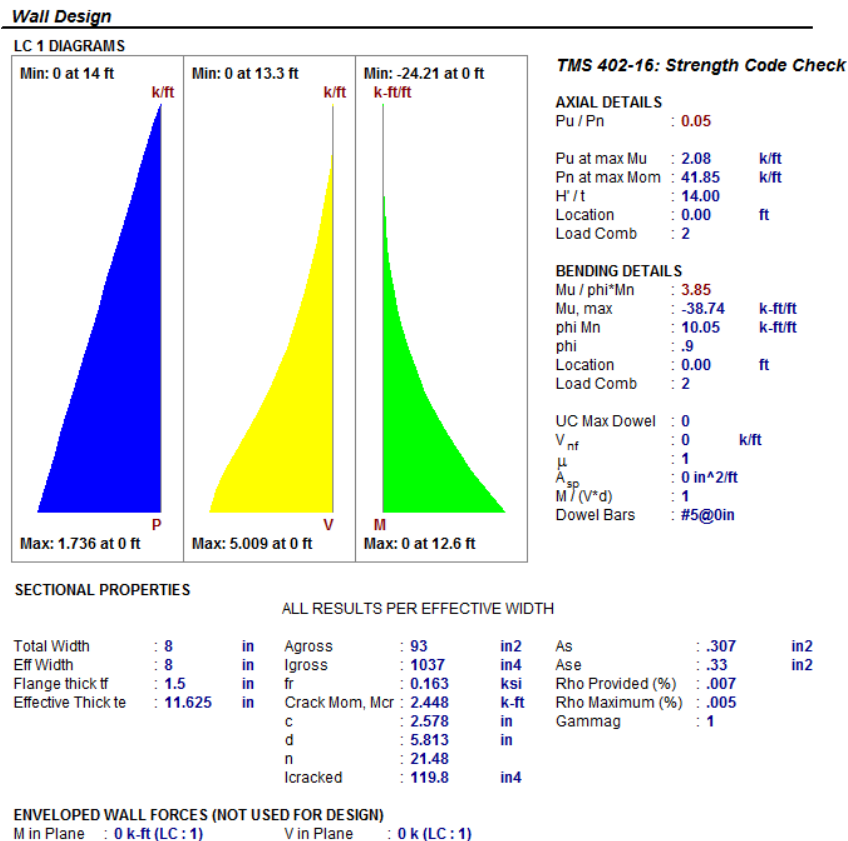
The **Axial Details** provide the axial code check for pure compression forces. The **Pn at max Mom** value reported is the maximum allowable axial force based on either Section 9.3.5.4.2 or Equations 9-11 and 9-12

The **Bending Details** are explained below:

- **Mu/ phi*Mn** represents the ratio of applied moment to moment capacity of the wall
- **Mu, max** is the out-of-plane bending moment at the controlling location of the wall.
- **phi Mn** is the out-of-plane moment capacity of the wall at the controlling location. It is calculated per Section 9.3.5.2.
- **phi** is the strength reduction factor specified in Section 9.1.4.
- **Location** is the elevation of the wall which resulted in the highest (Mu / phi*Mn) ratio. It is the location at which Mu and Pu are used to calculate phi Mn.
- **Load Combination** is the load combination which resulted in an Mu and Pu which yielded the highest (Mu / phi*Mn) ratio.

Note:

- If there is an over-reinforced section this can cause a failure in the masonry wall solver. If this occurs the program will produce an **Over-reinforced** message. To reconcile this you will need to either increase the thickness of masonry block, decrease the grout spacing or decrease the bar size or number of bars in the wall.

The **Sectional Properties** give design values based on the center to center spacing of reinforcement and are explained below:

The center to center distance between reinforcing **Total Width** and the **Eff Width** (which accounts for partially grouted walls) are both reported in this section. The **Flange Thick tf** refers to the thickness of the face shell whereas the **Effective Thick te** refers to the overall effective depth of the block.

The gross area and moment of inertia are reported along with the modulus of rupture, cracking moment, neutral axis, modular ratio between steel and masonry, and the cracked moment of inertia.

The area of steel **As** and the effective area of steel **Ase** are reported, with the effective area based upon As + Pu/fy.

The gross steel ratio (rho gross) is calculated as the area of steel divided by the total area of the section.

The **Rho Provided (%)** is calculated as the area of steel divided by the total width times the effective depth (As/bd). This is limited to a **Rho Maximum (%)** value of 0.5 * rho balanced per the UBC-97 Section 2108.2.3.7. This provision was implemented in future codes as well.

The Sectional Properties gives a number of the properties used during the iterative slender wall design procedure. All of these properties are reported based on a section of wall equal to the center to center spacing of the reinforcement.

Footing Design

Footing Design

As-min z-dir (Top Flexure): .623 in^2/ft	z-dir (T & S) : .389 in^2/ft
As-min z-dir (Bot Flexure) : .583 in^2/ft	x-dir Long. (T & S) : 4.082 in^2

Bottom Bar Design (Toe) : Shear (Toe)

Mu UC Max	Gov Mu LC	Max Mu (k-ft/ft)	Phi*Mn (k-ft/ft)	As Req'd (in^2/ft)	As Prov'd (in^2/ft)	Vu UC Max	Gov Vu LC	Vu (k/ft)	Phi*Vn (k/ft)
.695	2	39.213	56.416	.618	.902	.887	2	14.702	16.578

Top Bar Design (Heel) : Shear (Heel)

Mu UC Max	Gov Mu LC	Max Mu (k-ft/ft)	Phi*Mn (k-ft/ft)	As Req'd (in^2/ft)	As Prov'd (in^2/ft)	Vu UC Max	Gov Vu LC	Vu (k/ft)	Phi*Vn (k/ft)
.813	2	49.139	60.475	.727	.902	.931	2	16.498	17.717

This section provides an enveloped report of the footing design for both the top and bottom bars. The top of this section provides the flexural and temperature & shrinkage minimums. For the toe and heel we provide shear and moment checks for the worst case load combination. For more information on footing design considerations see the Wall Footings - Design topic.

Note:

- If the bottom bar is in tension in any solved combination for both the toe and the heel, the program will envelope the worst case reinforcement for both the toe and the heel.
- If the minimum spacing requirements defined in the Retaining Wall Definition does not meet minimum code requirements you will receive a warning message in the detail report.

Overturning Check (Service)

Overturning Check (Service)

Description	Categories and Factors	Moverturn (k-ft/ft)	Mresist (k-ft/ft)	SF	SF Min	SF Min/SF
IBC 16-8	1DL	.775	67.336	86.885	1	.012
IBC 16-9	1DL+1LL+1LLS	.775	80.336	103.659	1	.01
IBC 16-10 (a)	1DL+1RLL+1HL	38.942	70.337	1.806	1	.554

This section gives overturning results for all service load combinations that are solved. The **SF Min/SF** column is meant to act as the code check value. Anything exceeding 1.0 here means you are failing this check. See the Serviceability Considerations section of the Wall Footings - Design topic for more information.

Note:

- The **SF Min** column is the *minimum safety factor*. This value is defined for each load combination separately in the Load Combinations spreadsheet.

Sliding Check (Service)

Sliding Check (Service)

Description	Categories and Factors	Vsliding (k/ft)	Vresist (k/ft)	SF	SF Min	SF Min/SF
IBC 16-8	1DL	.049	9.054	184.768	1.5	.008
IBC 16-9	1DL+1LL+1LLS	.049	10.454	213.339	1.5	.007
IBC 16-10 (a)	1DL+1RLL+1HL	7.967	11.626	1.459	1.5	1.028

This section gives sliding results for all service load combinations that are solved. The **SF Min/SF** column is meant to act as the code check value. Anything exceeding 1.0 here means you are failing this check. See the Serviceability Considerations section of the Wall Footings - Design topic for more information.

Note:

- The **SF Min** column is the *minimum safety factor*. This value is defined for each load combination separately in the Load Combinations spreadsheet.

Soil Bearing Check (Service)

Soil Bearing Check (Service)

Description	Categories and Factors	Max Bearing (ksf)	Bearing Allowed (ksf)	Bearing UC
IBC 16-8	1DL	.049	1.5	.049
IBC 16-9	1DL+1LL+1LLS	.049	1.5	.049
IBC 16-10 (a)	1DL+1RLL+1HL	1.806	1.5	1.028

This section gives soil bearing check results for all service load combinations that are solved. The **Bearing UC** column is meant to act as the code check value. Anything exceeding 1.0 here means you are failing this check.

Note:

- The **Bearing Allowed** column is the *allowable bearing pressure*. This value is defined for each soil region in the model. If you are using the default value this can be found in the Model Settings - Solution tab.
- If your wall footing lands on multiple soil regions, the smallest allowable bearing pressure will be used.

- If you fail the soil bearing checks we will not give a full solution to your model. Since you are failing the soil this is not necessary. We will, however, give you the soil pressure diagrams so you can decide how you want to reconcile this failure.

Warning Log

The **Warning Log Spreadsheet** provides you with a record of any warnings or errors that occurred during the solution of your model. The log should be reviewed for warnings or errors that would affect the design of your structure. The error log reports back the item label for which the error occurred.

You can view the warning log by clicking on the Spreadsheets menu item and then clicking on the Warning Log selection.

Pressing the **?** hyperlink will open the help file to give further explanation on this item.

*Please refer to the **Help File** in RISAFoundation for detailed information about each error message. The Help File may be accessed by selecting **Help Topics** from the **Help Menu**. Choose the **Warning Log** topic*

Wood - Database

The Wood Database may be accessed by clicking the **Shape Database** button ⊥ on the **RISA** toolbar and then clicking the **Wood** tab of the **Shape Database** dialog.
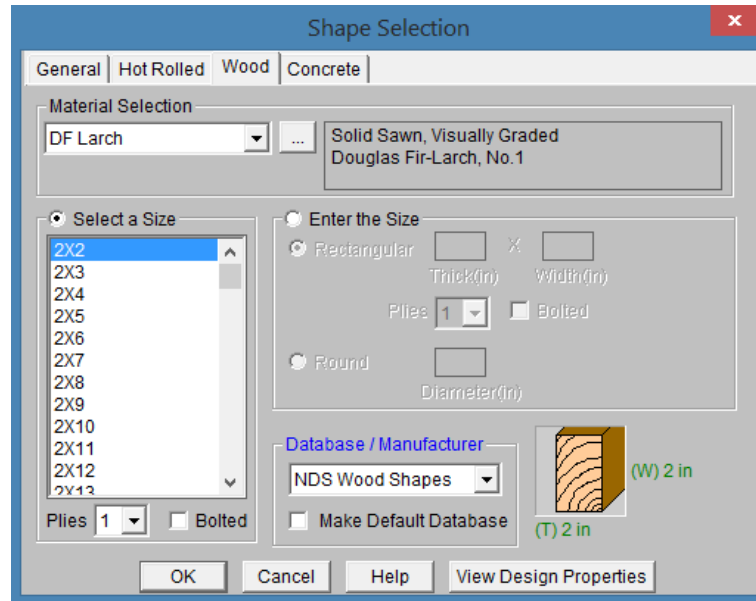
Wood Material/Shape Selection

Unlike any other material, the design properties for wood members depends both on the material selection as well as the shape selection. Therefore, the **Wood** tab of the **Shape Selection** database contains shape and material selections.

Material Selection

The **Material Selection** menu reads in material options that are currently listed on the **Wood** tab of your [Materials](#) spreadsheet. You may select one of these existing materials, or click on the **Add Material** button to open the **Create Wood Material** dialog which will allow you to add in a new material without having to go to the **Materials** spreadsheet.

Create Wood Material dialog

The **Create Wood Material** dialog allows you to create or edit a material that will then be automatically added to your [Materials](#) spreadsheet. This allows you to enter/select this information directly from the **Draw Members** dialog without having to first manually enter the information in the **Materials** spreadsheet.

- Click on the **Select Existing Material** button to select a material using the drop down menus to select a solid sawn, glulam, SCL, or already created custom material from the material databases.
- Click on the **Create a Custom Material** button to manually enter the design values for a new custom wood material. This information will then be automatically saved to the <u>Custom Wood Species</u> spreadsheet.

Size Selection

Currently there are two ways to define a member size, **Select a Size** menu or by using the **Enter the Size** option (typically referred to as Full Sawn).

- The **Select a Size** menu is a list of nominal sized shapes for the NDS design code and a list of net sized shapes for the CSA O86 design code. When the NDS design code is chosen, the program will automatically reduce the shape properties to their net sizes. If the CSA O86 design code is selected, a list of *actual* metric sizes will be shown.
- The **Enter the Size** option will allow you to manually enter any *dressed* size that you like.
  - *Rectangular imperial* full sawn (as opposed to nominal dimensions) shapes will be designated with a "FS" suffix. You may also use this nomenclature to type the size directly into the spreadsheet. For example, a 7.25"x12" net rectangular member can be typed in as "7.25x12FS".
  - *Rectangular metric* full sawn (as opposed to nominal dimensions) rectangular shapes will be designated with a "MFS" suffix. You may also use this nomenclature to type the size directly into the spreadsheet. For example, a 184mmx305mm net rectangular member can be typed in as "184x305MFS".
  - *Round imperial* shapes will be designated with a "RND" suffix. You many also use this nomenclature to type the size directly into the spreadsheet. For example, a 10" net round member can be typed as "10RND".

○ *Round metric* shapes will be designated with a "MRND" suffix. You many also use this nomenclature to type the size directly into the spreadsheet. For example, a 254mm net round member can be typed as "254MRND".

Note

- Multiple-ply lumber is assumed to be fastened sufficiently to achieve full composite behavior between plies. Therefore a two-ply member will have eight times greater minor-axis flexural stiffness than a single ply member (as opposed to double the stiffness).
- If using multiple plies there is a **Bolted** check-box. If this box is unchecked it is assumed that the plies are nailed together. If bolted is checked then it is assumed the plies are bolted together. This affects the Kf value from section 15.3.2 of the NDS 2005/2008, 2012, 2015, and 2018. If you define the multi-ply section as bolted a "B" (or "MB" for metric sizes) will appear after the shape name.

View Design Properties

Once you have selected the appropriate member size and material, you can click on the **View Design** properties button to view the un-adjusted wood design values (Fb, Fv, Ft, etc.) for the selected combination. The final adjusted values (Fb', Fv',Ft', etc.) will then be displayed after solution in the [Member Detail Reports](#).

Wood Database Limitations:

Round Shapes

Per section 3.7.3 of the NDS, the design of a round cross section shall be based on the design calculations for a square shape with the same cross-sectional area. Therefore the moment of inertia values will be calculated per the equivalent square shape, not the round cross section. However, the section modulus value, S is calculated per the actual round shape.

Glulam Databases

The program includes five tables of available glulam materials: NDS Tables 5A - 5D and CSA O86 Table 6.3. The values in NDS tables 5B & 5D depend on the number of laminations in the glulam. Therefore you will see three listings (2 Laminations, 3 Laminations, or 4 Laminations) for each material. If you have a custom material that is not available from these lists, you can always enter your appropriate design values as a [Custom Wood Material](#) for use in your model.

Note:

- The design values used per NDS tables 5B & 5D take into account footnote #1 to reduce the reference shear design value Fvy depending on the number of lamintations.
- None of the other footnotes in either of these tables is taken into account.

Mechanically Graded Lumber

Mechanically graded lumber grades (MSR & MEL) are available for use with the *ANSI/AWC NDS-2015* and the *CSA O86-2014* design codes. Older versions of the design codes will not support design of mechanically graded materials from the program databases. However, you can always enter the properties as a Custom Wood Material to get the design calculations for these materials in older code versions.
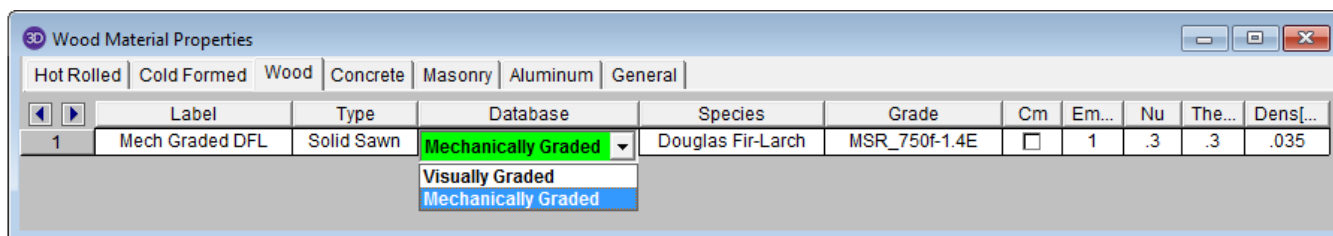
In order to assign a mechanically graded lumber material, you must select both the species and then the mechanical (MEL or MSR) grade. The species selection determines the shear capacity and compression perpendicular to grain capacity design values.

See *Footnote #2* from *NDS 2015 Table 4C* which says that all species from *NDS 2015 Table 4A* are applicable. See *clause 6.3.2* from *CSA O86-14* which says that specified shear strengths shall be taken from CSA O86 Table 6.3.1A.

The other design values are defined per the mechanical grade. See *NDS 2015 Table 4C* and *CSA O86-14 Tables 6.3.2 and 6.3.3*.

Structural Composite Lumber (SCL)

Each SCL manufacturer has documentation of their proprietary material design values. Because these are updated so often, RISA-3D relies instead on the design data from approved product reports (APA and ICC-ES) to build the material databases:

Manufacturer	Product Report	Notes
Ainsworth Engineered Wood*	APA PR-L260	*In 2015, Ainsworth Engineered was merged with Norbord Inc.
APA PRL-501	APA PR-L501	-
Boise Cascade	ICC-ES ESR-1040	-
Georgia Pacific**	APA PR-272	** Georgia Pacific sold to Boise Cascade in 2016
	APA PR-L257	
	APA PR-L266	
Global LVL	APA PR-L301	-
Louisiana Pacific	APA PR-L280	-
Murphy Engineered Wood	APA PR-L283	-
Pacific Woodtech	APA PR-L233	-
Roseburg Forest Products	APA PR-L289	-
	APA PR-L270	
TrusJoist	ICC-ES ESR-1387	-

Footnote Limitations:
Each of the above reports includes tables with design values which were used to create the RISA-3D and RISAFloor SCL design property material databases. However, due to the difficulty of design variation, the following limitations are present:

- The Fb design values are not factored in accordance with the depth of the member. Currently RISA-3D always uses the single tabular Fb value regardless of member size.
- The Ft design values are not factored in accordance with the length or thickness of the member. Currently RISA-3D always uses the single tabular Ft value regardless of member size.
- The Fb design value will be increased by 4% if the member is designated as a repetitive member using the Cr check-box on the Wood tab of the Members spreadsheet.

Alternatively, you can always enter the appropriate design values, per the applicable footnotes, as a [Custom Wood Material.](#)

Design Values

The design properties may be verified by clicking on the View Design Properties button on the Properties tab of the Member Information dialogs.
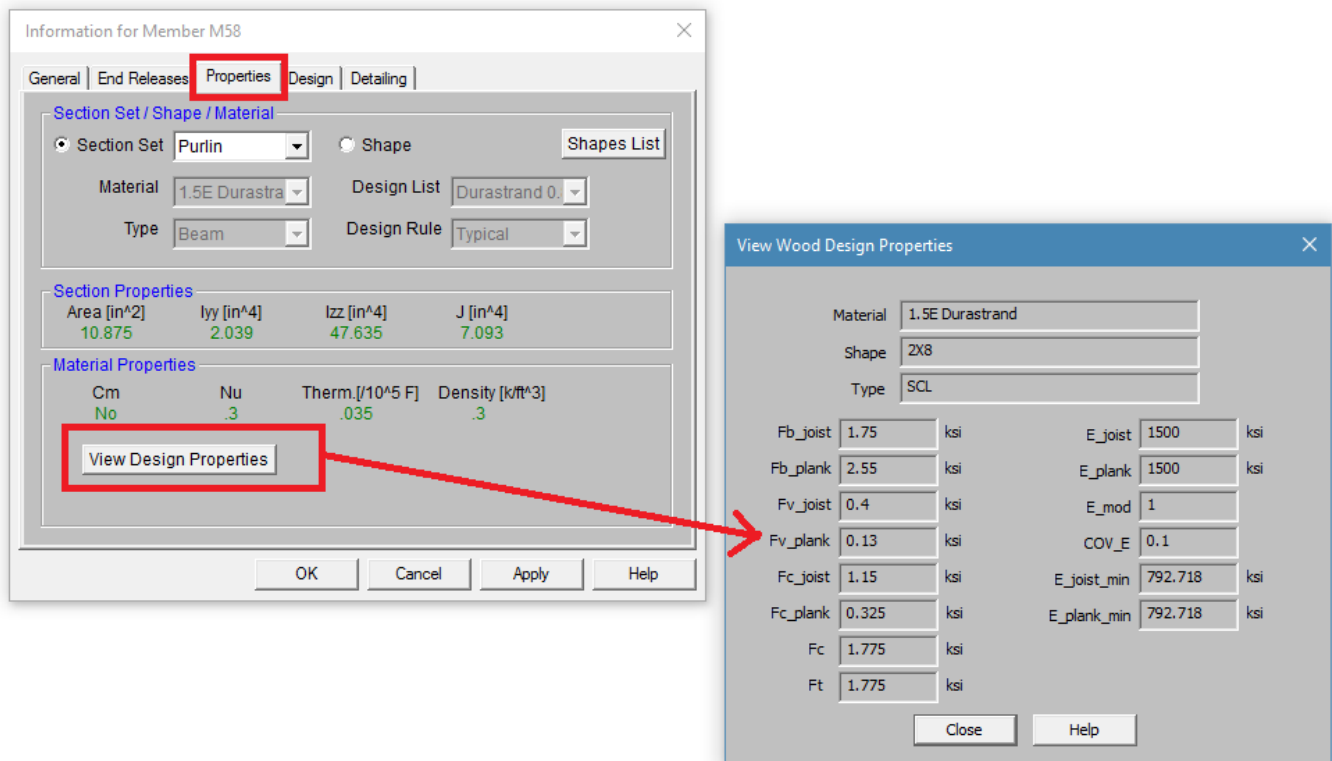
The values are as follows:

- **Fb_joist:** Bending capacity due to joist/beam edge loading
- **Fb_plank**: Bending capacity due to plank/flat edge loading
- **Fv_joist:** Shear capacity due to joist/beam edge loading
- **Fv_plank:** Shear capacity due to plank/flat edge loading
- **Fc_joist:** Compressive (perpendicular to grain) axial capacity due to joist/beam edge loading
- **Fc_plank:** Compressive (perpendicular to grain) axial capacity due to plank/flat edge loading
- **Fc:** Compressive (parallel to grain) axial capacity, regardless of load direction
- **Ft:** Tensile axial capacity, regardless of load direction
- **E_joist:** Modulus of elasticity due to joist/beam edge loading
- **E_plank:** Modulus of elasticity due to plank/flat edge loading
- **E_mod:** Young's Modulus modifier (always set as 1.0)
- **COV_E:** Coefficient of Variation in Modulus of Elasticity per NDS Table F1. This value is used in the Emin calculation. For SCL materials COV_E always equals 0.10.
- **E_joist_min:** Emin (as calculated per NDS Appendix D equation D-4) due to joist/beam loading
- **E_plank_min:** Emin (as calculated per NDS Appendix D equation D-4) due to plank/flat edge loading

Currently the program only designs using the *Joist* properties of the SCL materials. This assumes that the loading is always on the small dimension side of the member. In a future version we will be taking the Plank values into consideration as well. In the meantime you can always enter the appropriate design values from these tables as a [Custom Wood Material](#) for use in your model.

Note:

- Canadian wood member design only includes SCL design for the CSA O86-14 code.

Custom Wood Sizes:

If you would like to enter explicit dimensions of a member or if the member is "Not Dressed", the member must be designated as a "Full Sawn" member by checking the **Use Full Sawn Size** check box and entering the exact dimensions of the member in the boxes below. This applies to regular wood species, custom wood species, Structural Composite Lumber (SCL), and Glu-Lam members. The member thickness should be entered in the box on the left and the member width should be entered in the box on the right.

Wood - Design

Full code checking can be performed on Dimension Lumber and Post and Timber size wood shapes based on the following codes:

- The 2018 edition of the NDS (with 2015 and 2021 SDPWS)
- The 2015 edition of the NDS
- The 2012 edition of the NDS
- The 2005/08 edition of the NDS
- The 2001 edition of the NDS
- The 1991 / 1997 editions of the NDS
- The 2014 edition of the CSA O86 Canadian wood design code
- The 2009 edition of the CSA O86 Canadian wood design code

Note

- When the 1991 / 1997 NDS is selected, the 1991 NDS specification will be used with the 1997 stress tables. This is consistent with the requirements of the 1997 UBC.

Glu-Lams

Glu-Lams are treated as any other wood species and may be selected from the list of species on the **Wood** tab of the **Materials** spreadsheet.

Glulam Material Properties

Available Glulam Materials are per Tables 5A and 5C of the NDS Supplement and Table 6.3 of the CSA O86. When a Glu-Lam is selected, the grade will be listed as "na" or not applicable.

If you prefer to use a material that is not listed in the design code glulam tables, please enter the material type as a [Custom Wood Species](#).

Note

- Glu-Lams from Table5A are always assumed to have the special tension laminations. Therefore, the Fbx value is not reduced.
- RISA is NOT applying any of the footnotes to Table 5A and 5C at this time except for the following:
    - Footnote #1 from Table 5A - For balanced materials Fbx- shall equal Fbx+ for the stress class.
    - Footnote #3 from Table 5A - Fvx and Fvy are increased for Southern Pine Glulam materials.
- Commentary to CSA O86 Clause 6.2 notes that the Hem Fir glulam species listed in Table 6.3 are not common and therefore they are not included by default in RISA. If you prefer to use one of these materials, please enter it as a Custom Wood Species.

Glulam Dimensions

All Glu-Lam members should be dimensioned as "Full Sawn" using the format *wXdFS* (or *wXdMFS* for metric sizes), where "w" and "d" are the actual width and depth dimensions. If the size is entered as *wXd* without the *FS* designation, then the size will be assumed to be regular dimensional lumber.

Glulam Redesign Lists/ Optimization

RISA includes two redesign lists for Glu-Lams: Glu-Lam_Western for Western Species and Hardwoods (HW), and Glu-Lam_SouthernPine for Southern Pine (SP/SP).

Glulam Limitations
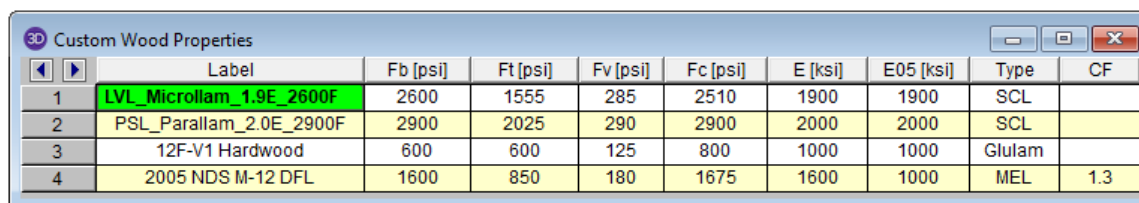
Please note that glulam design is not supported for the 91/97 NDS design code.

Custom Wood Materials

To use a custom wood material that is not part of the standard NDS or CSA O86 databases, you will need to define the custom design properties. This can be done through the **Custom Wood Species** spreadsheet or directly through the **Materials** spreadsheet.

Custom Wood Properties Spreadsheet

To access the **Custom Wood Properties** spreadsheet, select **Custom Wood Species** from the **Spreadsheets** menu.

	Label	Fb [psi]	Ft [psi]	Fv [psi]	Fc [psi]	E [ksi]	E05 [ksi]	Type	CF
1	LVL_Microllam_1.9E_2600F	2600	1555	285	2510	1900	1900	SCL	
2	PSL_Parallam_2.0E_2900F	2900	2025	290	2900	2000	2000	SCL	
3	12F-V1 Hardwood	600	600	125	800	1000	1000	Glulam	
4	2005 NDS M-12 DFL	1600	850	180	1675	1600	1000	MEL	1.3

- **Fb** - Bending Stress Limit
- **Ft** - Tension Stress Limit
- **Fv** - Shear Stress Limit
- **Fc** - Compression (Parallel to Grain) Stress Limit
- **E** - Modulus of Elasticity
- **E05** - Fifth Percentile Modulus of Elasticity (for Canadian CSA O86 design only, ignored for NDS design)
- **Type** - Wood Type, used for determination of design variables (COV_E, E_{min}, c, etc.) and design calculations.
- **CF/KZ** - Wood Size Factor. The program will default this to 1.0 unless manually entered by the user.

Notes:

- Most wood materials (sawn lumber visually & mechanically graded, glulam, SCL, and LVL) are available in the current wood materials database. Please see the **Wood** tab of the Materials spreadsheet for more information.
- Due to the limited entries in the Custom Wood spreadsheet, Glulam members are always assumed to be balanced, uniform material:
  - Fb is taken as the Fbx+, Fbx-, and Fby values for glulam design.
  - Fv is taken as the Fvx and Fvy values for glulam design.
  - E is taken as the Ex and Ey values for glulam design.
  - Glulams will always use the CV volume factor equation per NDS Table 5C (this is the same equation as Tables 5A,5B, and 5D assuming that the material is not Southern Pine).

Create and Apply a Custom Wood Material

Create and apply your custom material by doing the following:

1. First, open the **Materials** spreadsheet to the **Wood** tab.
2. Select "**Custom**" as the material **Type**.
3. Click within the **Species** cell and then click on the red arrow. This will open the **Select Custom Material** dialog box.
4. Either select an **Existing Material** (this is read in from the **Custom Wood Properties** spreadsheet, see above), or **Create a Custom Material** by typing the properties directly into this dialog.

Note

- The **Type** menu is used to designate whether the entry is a **Visually Graded** sawn lumber, a **MSR** mechanically graded lumber, a **MEL** mechanically graded lumber, a **Glulam** material, or a **SCL** (structural composite lumber). This selection effects the design factors and the applicable provisions of the selected design code.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Custom Wood Species**.

Appendix B – Error Messages

Extensive error checking is performed during data entry and model solution in order to prevent you from having to scan input files for offending data. If an error or questionable entry is detected, an error or warning message will be displayed. Errors are divided into two groups such that data entry error numbers are between 1000-1999 and solution errors are between 2000-2999.

*Please refer to the **Help File** in RISAFoundation for detailed information about each error message. The Help File may be accessed by selecting **Help Topics** from the **Help Menu**. Choose **Appendices : Appendix B - Error Messages**.*

Appendix D – File Format

RISAFoundation uses a plain text, keyword driven input file. The input data is delimited by sets of keyword labels that divide the input into a number of segments containing semantically related data.

Each input segment may include one or more input records. The input records are text strings that define sets of properties for single input items, e.g., label, coordinates, etc. for nodes. Each record is terminated with a semicolon (;), except for a few project description records (between the [.PROJECT_DESCRIPTION] and [.END_PROJECT_ DESCRIPTION] labels). Those records are delimited by their corresponding keyword labels.

Multiple records are used to describe multiple instances of input items. The number of records immediately follows the keyword label and is given in the <> brackets.

Lines starting with a // are ignored by the program and may be used for user comments.

The reading of Label fields (such as member names, or shape names) uses a combination of fixed length fields and field delimiter. The field lengths are set in the Label Length Data. Care should be taken to maintain the proper length for all label fields. If the length of the field is not entered correct, the program will attempt to use the field delimiters (") to read the data. A combination of incorrect field length and the use of (") marks within a shape label itself (e.g. a shape label of: 3/4" rod bracing) will result in the program producing an error during the reading of the file.

*Please refer to the **Help File** in RISAFoundation for detailed information about the file format. The Help File may be accessed by selecting **Help Topics** from the **Help Menu**.*

Technical Support

Technical support is an important part of the RISAFoundation package. There is no charge for technical support for all licensed owners of, or subscribers to, the **current** version of RISAFoundation. Technical support is very important to the staff at RISA Tech, Inc.

Hours: 6AM to 5PM Pacific Time, Monday through Friday (excluding holidays)

Before contacting technical support, you should typically do the following:

1. **Please search the Help File or General Reference Manual**. The Help File and General Reference are meant to cover most common questions asked about RISAFoundation. The Table of Contents or Search can be used to find specific topics. This is a good first step in finding the answer to your question. It is also good practice to go through all of the <u>Tutorials</u> when you first get the program. This will help you get a feel for the program and likely increase your modeling confidence.
2. Ensure that you are running the latest version of the software. Under the *Help* menu click **Check for Updates**
3. Visit our website at **risa.com**.
4. Make sure you understand the problem, and make sure your question is related to the program or structural modeling.
5. Take a few minutes to experiment with the problem to try to understand and solve it.

Email: support@risa.com. If your question is related to a specific model, please send us that model. That makes it much easier for us to debug a problem or answer questions about specific issues. The model file has a .FND extension. Make sure you tell us your name, company name, serial number, Key ID, or Cloud ID, phone number, and give a thorough problem description. If you have multiple beams, slabs, or load combinations, make sure you specify which ones to look at.

Phone Support: (949) 951-5815. Feel free to call, especially if you need a quick answer and your question is not model specific and therefore doesn't require us to look at your file.

Index

E

F

G

H

I