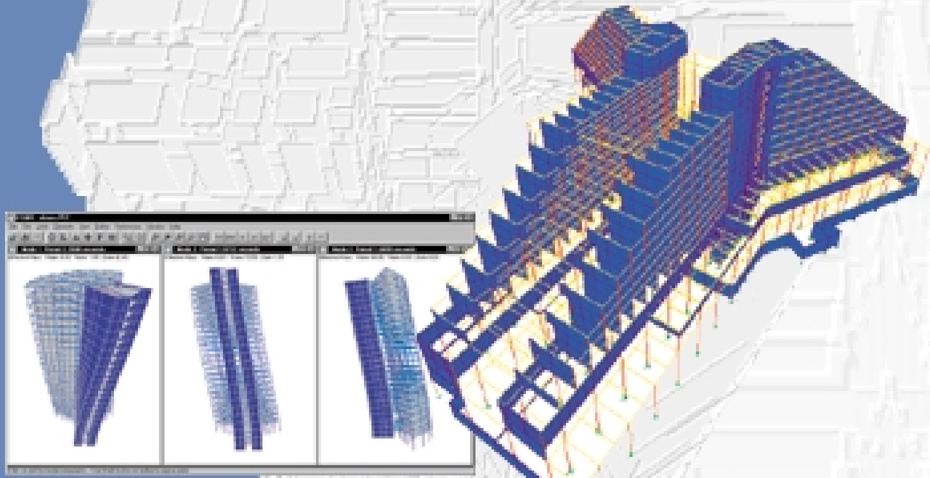


ETABS®

Section Designer Manual

COMPUTERS & STRUCTURES INC.®

STRUCTURAL AND EARTHQUAKE ENGINEERING SOFTWARE

ETABS®

Three Dimensional Analysis and Design of Building Systems

Section Designer Manual

Computers and Structures, Inc.
Berkeley, California, USA

First Edition
February 2000

Copyright

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA

Phone: (510) 845-2177
FAX: (510) 845-4096
e-mail: info@csiberkeley.com (for general questions)
e-mail: support@csiberkeley.com (for technical support questions)
web: www.csiberkeley.com

DISCLAIMER

Contents

Tip:

If you are just getting started with ETABS Version 7 we suggest that you read Chapters 1, 2, 3, 13 and 14 and then use the rest of the manual as a reference guide on an as-needed basis.

The Table of Contents for this manual consists of a chapter list followed by an expanded table of contents. The chapter list devotes one line to each chapter. It shows you the chapter number (if applicable), chapter title and the pages that the chapter covers.

Following the chapter list is the expanded table of contents. Here all section headers and subsection headers are listed along with their associated page numbers for each chapter in the manual.

When searching through the manual for a particular chapter, the highlighted tabs at the edge of each page may help you locate the chapter more quickly.

If you are new to ETABS we suggest that you read Chapters 1, 2, 3, 13 and 14 and then use the rest of the manual as a reference guide on an as-needed basis.

C

Section Designer Manual Chapter List

C

Section Designer Manual - Expanded Contents

Chapter 1

Introduction

Overview of Section Designer

Note:

Section Designer is fully integrated into the ETABS graphical user interface.

Section Designer is a powerful utility that is fully integrated into the ETABS graphical user interface. It allows you to graphically define frame and wall pier sections for use in analysis and design. The sections can be of any arbitrary shape and they can consist of one or more material properties. Reinforcing bars can be specified for concrete sections.

The section properties that you define in Section Designer are assigned to objects in ETABS. The ETABS analysis is based on the *gross* section properties of the section multiplied times any user-defined modification factor that you might choose to apply.

The modification factors for frame elements can be assigned in ETABS using the **Assign menu > Frame/Line > Frame Property Modifiers** command. The modification factors for shell elements that make up wall piers can be assigned in ETABS using the **Assign menu > Shell/Area > Shell Stiffness Modifiers** command. These commands are discussed in the ETABS User's Manual.

If you want your analysis of concrete members to consider cracked sections then you should specify appropriate modification factors. There is no built-in cracked section analysis for frame and wall pier sections in ETABS.

Note:

You can define sections with arbitrary shape in Section Designer. If the defined section is a concrete section you can also define arbitrary reinforcing steel in the section.

For design of concrete frame sections and wall piers that are defined using Section Designer, ETABS develops design interaction curves based on the section defined in Section Designer. Thus, using Section Designer you can define arbitrary-shaped concrete sections with arbitrary reinforcing and then use the ETABS Concrete Frame Design or Shear Wall Design postprocessors to design those sections.

For design of steel frame sections ETABS treats sections defined using Section Designer as General Sections. The consequence of this is that the Steel Frame Design postprocessor in ETABS assumes all General Sections are noncompact and the Composite Beam Design postprocessor does not design General Sections.

The reason that the Steel Frame Design postprocessor assumes all General Sections are noncompact is because it does not have enough information to check width to thickness ratios and determine whether the section is compact, noncompact or slender. Although the program has the complete section geometry, it has no way of identifying a web, flange, cover plate, etc., for use in checking width to thickness ratios.

Our main purpose in developing Section Designer is to allow the following:

1. Definition of unsymmetrical concrete sections with reinforcing and the development of the associated PMM interaction surface for use in the Concrete Frame Design postprocessor.

2. Definition of wall pier geometry and reinforcing and development of the associated PMM interaction surface for use in the Shear Wall Design postprocessor

Organization of Manual

Tip:

We recommend that you read Chapters 1, 2, 3, 13 and 14 of this manual and then use the rest of the manual as a reference guide on an as-needed basis.

This manual is organized as follows:

- **Chapters 1 through 3:** Introduction and important general information about Section Designer.
- **Chapters 4 through 10:** Documentation of the menu commands in Section Designer.
- **Chapters 11 and 12:** Discussion of how Section Designer develops interaction surfaces and moment curvature curves.
- **Chapters 13 and 14:** Frame section and wall pier examples.

Other Reference Information

Section Designer Help

You can access the Section Designer Help by clicking the **Help menu** > **Search for Help On** command in the Section Designer window. You can also access context-sensitive help by pressing the F1 function key on your keyboard when a dialog box is displayed.

Recommended Initial Reading

We recommend that you initially read Chapters 1, 2, 3, 13 and 14 of this manual. You can then use the remainder of this manual as a reference on an as-needed basis.

Chapter 2

Starting Section Designer

The Section Designer utility is started from within ETABS. The location where you start it depends on whether you are defining a frame section or a wall pier. Each of these possibilities is discussed in the sections below.

Starting Section Designer for Frame Sections

Note:

The location where you start it depends on whether you are defining a frame section or a wall pier.

Use the following steps to start Section Designer when you want to define a frame section.

1. Click the **Define menu > Frame Sections** command in ETABS to bring up the Define Frame Properties dialog box.

2. To add a new frame section defined in Section Designer click Add SD Section in the drop down box in the Click To area. To review or modify an existing frame section that was created using Section Designer, highlight that section in the Properties area of the dialog box and then click the **Modify/Show Property** button.

3. The SD Section Data dialog box appears. Select the appropriate options for your circumstances in this dialog box. See the subsection below titled "SD Section Data Dialog Box" for more information.

4. Click the **Section Designer** button in the SD Section Data dialog box to start Section Designer.

SD Section Data Dialog Box

The following bullet items describe each of the areas in the SD Section Data dialog box:

- **Section name:** Here you can specify or modify the name of the frame section.

Note:

The section properties are calculated by converting the area of each shape that makes up the section into an equivalent area of the base material.

- **Base Material:** Here you can specify or modify a defined material property. Depending on the design type of the material property selected, various options in the Section Type area may or may not be available.

  ✓ If the design type of the specified material property is Steel, then the General Steel Section and Other (not designed) options are available.

  ✓ If the design type of the specified material property is Concrete, then the Concrete Column and Other (not designed) options are available.

  ✓ If the design type of the specified material property is None, then only the Other (not designed) option is available.

  The base material serves two purposes. First, it determines what design postprocessing options are available for the section. Second, the section properties are calculated by converting the area of each shape that makes up the section into an equivalent area of the base material. Thus, while a section can be made up of various shapes with different material properties, the reported section properties are based on an equivalent area of the base material.

See the Section titled "Show Section Properties" in Chapter 9 for additional information.

- **Design Type:** Here you can specify the section type. If the section type is General Steel Section then any frame section assigned this property is designed by the Steel Frame Design postprocessor as a general section. If the section type is Concrete Column then any frame section assigned this property is designed by the Concrete Frame Design postprocessor. If the section type is No Check/Design (not designed) then any frame section assigned this property is not designed by any postprocessor.

- **Concrete Column Check/Design:** This area is only active if the Concrete Column option is selected in the Design Type area. Here you specify whether the concrete column is to have its specified reinforcing checked or new longitudinal reinforcing designed when it is run through the Concrete Frame Design postprocessor.

- **Define/Edit/Show Section:** Once you have appropriately specified items in the rest of the dialog box, click the **Section Designer** button in this area to go to the Section Designer utility and draw the section. When you exit the Section Designer utility you return to the SD Section Data dialog box. You can then click the **OK** button twice to complete the definition of the frame section property.

Starting Section Designer for Wall Piers

Use the following steps to start Section Designer when you want to define a wall pier.

1. Click the **Design menu > Shear Wall Design > Define Pier Sections for Checking** command in ETABS to bring up the Pier Sections dialog box.

2. To add a new wall pier section click the **Add Pier Section** button. To review or modify an existing wall pier section

highlight that section in the Sections area of the dialog box and then click the **Modify/Show Pier Section** button.

3. The Pier Section Data dialog box appears. Select the appropriate options for your circumstances in this dialog box. See the subsection below titled "Pier Section Data Dialog Box" for more information.

4. Click the **Section Designer** button in the Pier Section Data dialog box to start Section Designer.

Pier Section Data Dialog Box

The following bullet items describe each of the areas in the Pier Section Data dialog box:

- **Section Name:** Here you can specify or modify the name of the pier section.

- **Base Material:** This is the base material property used for the pier section. The base material serves two purposes. First, it determines what design postprocessing options are available for the section. Second, the section properties are calculated by converting the area of each shape that makes up the section into an equivalent area of the base material. Thus, while a section can be made up of various shapes with different material properties, the reported section properties are based on an equivalent area of the base material.

  See the Section titled "Show Section Properties" in Chapter 9 for additional information.

- **Add Pier:** The Add New Pier Section option allows you to start the pier section from scratch.

  The Start from Existing Wall Pier option allows you to start with the geometry of an existing wall pier. When you select this option you also specify a story and a wall pier label so ETABS knows which existing pier geometry to use. In cases where the top and bottom geometry of the pier is different ETABS uses the geometry at the *bottom* of the pier.

Tip:

It is usually easier and quicker to start from the analysis pier section geometry rather than starting from scratch.

When an existing pier is brought into Section Designer the pier is oriented in Section Designer such that its positive local 2-axis is parallel to the Section Designer positive X-axis. The pier is located in Section Designer such that the lower right hand corner of the bounding rectangular box around the section (whose sides are parallel to the Section Designer X and Y axes) is located at the Section Designer origin.

- **Check/Design:** Select the Reinforcement to be Checked option if you want to specify your own reinforcement (location and size) and have ETABS check it.

  Select the Reinforcement to be Designed if you want ETABS to determine the required amount of reinforcing for you. In this case you still lay out the reinforcing bars in Section Designer. ETABS will use that layout and report the required percentage of steel. In cases where you use the design option you should, at the end of the design process, always specify your actual final reinforcing and have ETABS check it.

- **Define/Edit/Show Section:** Once you have specified the data in the other areas of the dialog box click the **Section Designer** button to start the section designer utility. Here you can define the pier geometry and the reinforcing.

  When you are done with Section Designer close it and return to the Pier Section Data dialog box where you can click the **OK** button twice to complete the definition of the pier.

Chapter 3

Section Designer Basics

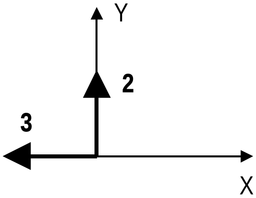
This chapter describes some basic information about Section Designer that you must understand to effectively use the program.

General Information

Section Designer Coordinate System

Section Designer has its own X and Y axes. On the screen the X-axis is always horizontal and the Y-axis is always vertical. There is no correspondence between the Section Designer X and Y axes and the ETABS global X and Y axes. In general, the Section Designer X and Y axes are simply a convenient set of reference axes.

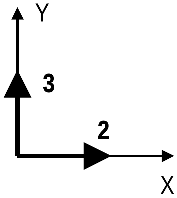
The section local axes are axes 1, 2 and 3. In Section Designer the local 1 axis is always pointing out of the screen toward you. Thus you are looking at a section in the local 2-3 plane. The local axes system is a right-handed system.



Default Local Axes for Frame Sections

The sketch to the left shows the default relationship between the Section Designer X and Y axes and the section local 2 and 3 axes *for frame sections*. By default, for frame sections (but not wall pier sections) the local 2-axis is parallel to the Section Designer positive Y axis and the local 3-axis is parallel to the Section Designer negative X-axis.

For frame sections (but not wall pier sections) you can rotate the section local axes. To do this click the Display menu > Show Section Properties command to bring up the Properties dialog box and then change the angle. See the section titled "Show Section Properties" in Chapter 9 for additional information.



Default Local Axes for Wall Pier Sections

The sketch to the left shows the default relationship between the Section Designer X and Y axes and the section local 2 and 3 axes *for wall pier sections*. By default, for wall pier sections (but not frame sections) the local 2-axis is parallel to the Section Designer positive X axis and the local 3-axis is parallel to the Section Designer positive Y-axis. Unlike frame sections, it is not possible to rotate the local axes of wall pier sections in Section Designer.

The local axes for wall piers are described in the ETABS Users Manual. *If you are drawing a wall pier from scratch rather than basing it on a previously defined pier in ETABS* ***then it is very important that you*** *understand the local axes definition so that you* ***orient the pier correctly in Section Designer****.*

Reinforcing Bar Sizes

The reinforcing bar sizes used in Section Designer are those specified in ETABS using the **Options menu** > **Preferences** > **Reinforcement Bar Sizes** command.

Sections and Shapes

Two important terms that you should understand when working with Section Designer are ***section*** and ***shape***. The term *section* refers to the entire section that you define in a single session of Section Designer. A *section* is made up of one or more *shapes*.

Shapes define the geometry and reinforcing for a *section*. Although there are many different types of *shapes* in Section Designer, they can be broken down into two main categories. Those categories are geometric shapes and additional reinforcing shapes.

Sections

Note:

In Section Designer a section is made up of one or more shapes.

As previously mentioned a section is the entire section that you define in a single session of Section Designer. A section is made up of one or more shapes.

The local axes of the section are designated 2 and 3. The local axes are designated by two orthogonal arrows labeled 2 and 3. The origin of these arrows occurs at the centroid of the section. As you add additional shapes to the section the origin of the local axes arrows shifts as the centroid of the section shifts.

Geometric Shapes

The main purpose of the geometric shapes is to define the geometry of the section. In addition, there is reinforcing associated with the geometric shapes. This reinforcing is separate and independent from the additional reinforcing shapes that are discussed in the subsection titled "Additional Reinforcing Shapes" later in this chapter. In most cases, the reinforcing associated with the geometric shape is all you will need. In general, additional reinforcing shapes are only required in special or unusual cases.

The geometric shapes can be drawn using one of the following three commands in Section Designer:

- **Draw menu > Draw Structural Shape command:** Eight different structural shapes can be drawn using this command. They are I/wide flange, channel, tee, angle, double angle, box/tube, pipe and plate shapes.

- **Draw menu > Draw Solid Shape command:** Four different solid shapes can be drawn using this command. They are rectangle, circle, circular segment and circular sector shapes.

- **Draw menu > Draw Poly Shape command:** This command allows you to draw an arbitrary polygon.

Each of these commands has an associated toolbar button located on the side toolbar. See Chapter 7 for additional information on these commands.

Tip:

We recommend that you use the structural and/or solid shapes as much as possible rather than the poly shapes. It is quicker and easier to build the section using the structural and solid shapes. Use the poly shapes when no structural or solid shape is adequate for the section you are defining.

All of the geometric shapes drawn using the above commands have associated properties. You can review and, if desired, modify these properties in the Object Properties dialog box that is accessed by right clicking on the geometric shape. The properties can be broken down into the following four categories:

- **Material:** This is the material property associated with the shape. The material properties are defined in ETABS (not Section Designer) using the ETABS **Define menu > Material Properties** command. Different shapes that are part of the same section can have different material properties if desired.

- **Dimensions and location:** The dimensions and location of structural and solid shapes are specified in the same way. The dimensions of these shapes are defined by various height and width properties. The location is defined by the coordinates of the insertion point, which is the center point of a bounding rectangle that encloses the shape, and a specified rotation. The rotation is measured counterclockwise from the Section Designer X-axis to the original horizontal axis of the shape.

  The dimensions and location of *poly shapes* are controlled by modifying the coordinates of the corner points of the shape. To do this:

  1. Click the **Draw menu > Reshape Mode** command.
  2. Left click once on the poly shape. Selection handles appear at all of the poly shape corners.
  3. Place the mouse pointer directly over a corner point (the mouse pointer will change shape when it is directly over a corner point). Click the *right* mouse button once to bring up the Change Coordinates dialog box.

4. Change the coordinates as desired and click the **OK** button.

- **Reinforcing:** If the material associated with the shape is concrete then the reinforcing property is available, otherwise it is not available (i.e., it is invisible). When the reinforcing property is available it can be specified as either Yes or No.

  If the reinforcing property is Yes then there is reinforcing associated with the geometric shape. This reinforcing is described in the subsection below titled "Reinforcing Associated with the Geometric Shape." This reinforcing is separate from and independent of any additional reinforcing shapes that may be defined within the bounds of the geometric shape. See the subsection below titled "Additional Reinforcing Shapes" for more information.

  If the reinforcing property is No then there is no reinforcing associated with the geometric shape. However, additional reinforcing shapes may still be defined within the bounds of the geometric shape.

- **Color:** This allows you to specify a color that is associated with the shape. Assigning different colors to different shapes may make it easier for you to distinguish between the various shapes in a section.

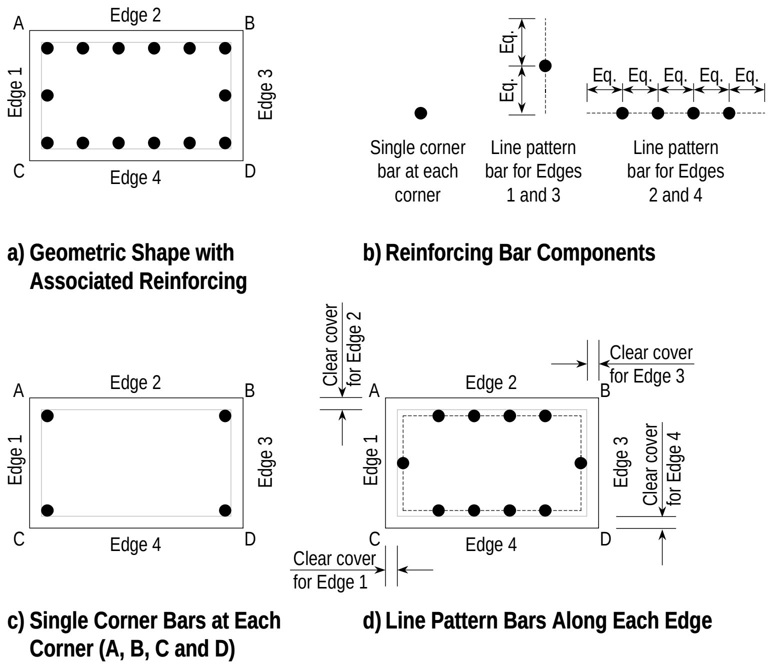
See the sections titled "Drawing Structural Shapes", "Drawing Solid Shapes" and "Drawing Poly Shapes" in Chapter 7 for more information about the properties of geometric shapes.

Reinforcing Associated with the Geometric Shape

When you specify that there is reinforcing associated with a geometric shape then Section Designer automatically inserts single corner bars at all corners of the shape and inserts line pattern edge bars along all edges of the shape.

3

Figure 3-1:
Reinforcing associated with a geometric shape



a) Geometric Shape with Associated Reinforcing

b) Reinforcing Bar Components

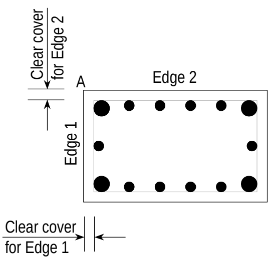
c) Single Corner Bars at Each Corner (A, B, C and D)

d) Line Pattern Bars Along Each Edge

Figure 3-1a shows an example of a rectangular geometric shape with the Reinforcing property set to Yes. The edges of the shape are identified as Edge 1 through Edge 4 and the corners are identified as A, B, C and D. Figure 3-1b shows the reinforcing bar components that are present in Figure 3-1a.

Corner Bars

There are single corner bars at each of the four corners labeled A, B, C and D as shown in Figure 3-1c. These four bars are independent of one another. They are defined by a bar size. You do not actually define a clear cover (i.e., location) for the corner bars. Instead their clear cover (location) is based on the clear cover that is specified for the edge reinforcing on each side of the corner bar.



As an illustration of how a corner bar is located, consider the corner bar at the corner labeled A in the sketch shown to the left. Notice how the corner bar is located such that it satisfies the clear cover specified for edges 1 and 2.

You can edit the size of any corner bar by right clicking on the bar. This brings up the Corner Point Reinforcing dialog box where you can specify the bar size by choosing from any defined bar size. There is also a check box in the dialog box that allows you to specify that the specified bar size applies to all corner bars associated with the geometric shape.

Edge Bars

There are line pattern bars along each of the four edges of the shape as shown in Figure 3-1d. The line pattern bars along any edge are independent of the bars along any other edge. The line pattern edge bars include all of the bars along an edge of the shape except for the corner bars.

The line pattern edge bars are specified by a bar size, spacing and clear cover. All of the bars in a line pattern have the same size, spacing and clear cover. The clear cover is measured from the associated edge of the geometric shape to the face (not center line) of the line pattern edge bars.

It is easiest to think of the line pattern edge bars as an imaginary line with equally spaced bars along the line as shown in Figure 3-1b. In the figure the imaginary line is shown dashed. No bars are included at the ends of the line because this is where the corner bars are located.

For line pattern edge bars associated with a geometric shape the imaginary line extends until it intersects the imaginary lines for the line pattern edge bars on adjacent edges of the geometric shape. This is illustrated in Figure 3-1d. In this figure the corner bars are not shown for clarity.

When you specify bar size, clear cover and spacing for line pattern edge bars Section Designer uses the following process to locate the edge bars:

1. Section Designer locates the imaginary line parallel to the associated edge at a distance from the edge equal to the specified clear cover plus one-half of the diameter of the specified bar size.

2. Section Designer extends the imaginary line until it intersects similar imaginary lines on adjacent edges of the shape. These intersection points define the starting and ending points of the imaginary line.

3. Section Designer calculates the required number of bars by dividing the length of the imaginary line by the specified bar spacing and adding one to the result. Usually this calculation leaves some fraction of a bar left over. If that fraction is greater than 0.1 then Section Designer rounds the number of bars up; otherwise it rounds the number of bars down. Once Section Designer knows the final number of bars to be used it calculates the final bar spacing by dividing the length of the imaginary line by the final number of bars minus one.

You can edit the size, spacing and clear cover for the bars along any of the edges of the of the geometric shape. To do this right click on any of the bars along the edge (except for a corner bar) to bring up the Edge Reinforcing dialog box. Actually you can bring up the Edge Reinforcing dialog box by clicking anywhere along the imaginary line associated with the line pattern edge bars. Do not click on the corner bars because this brings up the Corner Point Reinforcing dialog box that was discussed in the previous subsection.

In the Edge Reinforcing dialog box the Bar Size item allows you to choose from any defined bar size. The Bar Spacing item specifies the maximum spacing Section Designer allows along the edge. The actual spacing may be less since Section Designer is simply defining equal spaces along the edge as described above. The Bar Cover item specifies the clear cover for the edge bars.

Additional Reinforcing Shapes

The additional reinforcing shapes are separate from and independent of the reinforcing that is associated with the geometric shapes. In general, the additional reinforcing shapes are intended as a supplement to the reinforcing associated with the geometric shapes.

In most cases you will find that the reinforcing associated with the geometric shapes is sufficient to define your section and that it is not necessary to use any additional reinforcing shapes. The additional reinforcing shapes are provided for use in special or unusual cases. one example where an additional reinforcing shape is needed is to define a circular reinforcing pattern in a noncircular shape. Another example where it is needed is to define more than one layer of reinforcing along the edge of a section.

Note:

In general, the additional reinforcing shapes are intended as a supplement to the reinforcing associated with the geometric shapes.

Additional reinforcing shapes are drawn using the **Draw menu > Draw Reinforcing Shape** command or its associated toolbar button located on the side toolbar. Four different reinforcing shapes can be drawn using this command. They are single bar, line pattern, rectangular pattern and circular pattern. All reinforcing shapes are drawn with a single click at the center except for line pattern reinforcing which is drawn with two clicks, one at each end point.

All of the additional reinforcing shapes drawn using the above commands have associated properties. You can review and, if desired, modify these properties in the Shape Properties dialog box that is accessed by right clicking on the additional reinforcing shape. The properties can be broken down into the following three categories:

- **Material:** This is the material property associated with the shape. The material properties are defined in ETABS (not Section Designer) using the ETABS **Define menu > Material Properties** command. Section Designer picks up the yield stress for the reinforcing from this material property.

  The modulus of elasticity of the reinforcing steel, E_s, is not specified in the material property. Section Designer uses code-specified values for the modulus of elasticity of the reinforcing steel. If you are defining a frame section then Section Designer uses the value specified in the code currently designated for the Concrete Frame Design postprocessor. If you are defining a wall pier section then Section Designer uses the value specified in the code currently designated for the Shear Wall Design postprocessor.

For example, if you are specifying a wall pier and the design code is the 1997 UBC then Section Designer uses E_s of 29,000 ksi based on 1997 UBC Section 1908.5.2.

- **Location:** Various properties are defined to locate the rebar.
- **Bar Size:** The size of the rebar. It can be any defined bar size.

See the section titled "Drawing Reinforcing Shapes" in Chapter 7 for more information about the properties of additional reinforcing shapes.

Reference Lines

Note:

You can snap to reference lines and you can align items to reference lines.

Reference lines are drawn using the **Draw menu > Draw Reference Lines** command or its associated toolbar button located on the side toolbar. Two different types of reference lines can be drawn using this command. They are line and circle. The circle-type reference line is drawn with a single click at the center and the line-type reference line is drawn with two clicks, one at each end point.

Reference lines can assist you in drawing and editing your model. You can snap to reference lines and circles. See the section titled "Snap Options" in Chapter 7 for more information on the Section Designer snap features. You can also align shapes to reference lines and circles. See the section titled "Align" in Chapter 5 for more information on the Section Designer align feature.

Chapter 4

The Section Designer File Menu

This chapter describes the commands available on the Section Designer File menu.

Print Graphics

The **File menu > Print Graphics** command and its associated **Print Graphics** button, , located on the main (top) Section Designer toolbar, print a graphic representation (to scale) of the current section to a printer. You can use the **Options menu > Colors** command to control the colors of various items in the print out.

The **File menu > Print Graphics** command only prints the graphical representation of the section. It does not print the properties of the section. You can print the properties of the section from ETABS, or you can display the properties on the screen in Section Designer, perform a screen capture (as described below) and paste them into a word processing, spreadsheet or other type of graphics file.

The **File menu > Print Graphics** command also does not print information for the interaction surface or the moment curvature curves. In this case you can copy the tabular information into another file, such as a spreadsheet, and plot it yourself, or you can perform a screen capture of the graphics as described below.

Screen Capture

An alternative method of getting graphical output from Section Designer is to use the built-in Windows screen capture features. If you press the Print Screen key on your keyboard then Windows copies the entire screen as a picture to the clipboard. You can then paste this picture into a Windows graphics program such a word processor, spreadsheet, etc. For example, we often use this process to paste screenshots into Microsoft Word or Microsoft Powerpoint. If you press the Alt key and the Print Screen key simultaneously on your keyboard, then Windows copies the active window on the screen to the clipboard as a picture.

Some of the figures in the ETABS manuals were created using the following technique:

1. Set the ETABS display up as you want it to appear in the picture.
2. Press the Alt key and the Print Screen key simultaneously on your keyboard to copy the screen image to the clipboard as a picture.
3. Paste the picture from the clipboard into a graphics program. (We used Microsoft PowerPoint).
4. If necessary crop and/or resize the picture.
5. In some cases add annotations to the picture.
6. If desired, cut and paste the completed figure from the graphics program into a report as a picture.

Print Setup

The **File menu > Print Setup** command allows you to temporarily change your printer settings. These changes only apply while you are in the current session of Section Designer. They do not apply to the main ETABS program. You must use the ETABS (not Section Designer) **File menu > Print Setup** command to temporarily change the printer settings for printing from the main ETABS program.

The **File menu > Print Setup** command does not permanently change your default printer settings.

Return to ETABS

Note:

*To permanently save the section definition you must click the **OK** button twice to close all dialog boxes and save your ETABS model.*

The **File menu > Return to ETABS** command closes Section Designer and returns to ETABS in the SD Section Data dialog box (frame section) or Pier Section Data dialog box (wall pier). To permanently save the section definition you must click the **OK** button twice to close all dialog boxes and save your ETABS model. The Section Designer data is saved with your ETABS model.

If you click the **Cancel** button to close one of the dialog boxes then the Section Designer data is immediately lost.

Chapter 5

The Section Designer Edit Menu

This chapter describes the commands available on the Section Designer Edit menu.

Undo and Redo

Shortcut:

Use the Undo button on the main tool-bar to undo a previous op-eration.

Section Designer has a Undo feature for geometry changes that you make to a section. The Undo feature works for multiple steps back to when you started the current Section Designer session. For example, if you draw one or more objects as part of your section and then decide you didn't want them after all, then you can use the **Edit menu > Undo** command, or the associated **Undo** button, , on the main (top) Section Designer toolbar to get rid of them. If you then decide you really did want them, then the **Edit menu > Redo** command, or the associated **Redo** button, , on the main (top) Section Designer toolbar will bring them back. The Undo and Redo commands work sequentially. In other words, if you have just finished the sixteenth operation since you started the current Section Designer session you can use the Undo feature to undo the sixteenth then fifteenth and so on operation. You could not however decide that you just wanted to undo the seventh operation.

The Undo and Redo features do not work for changes made in dialog boxes. When an item is changed in a dialog box, the change is not actually implemented until the **OK** button is clicked. If the **Cancel** button is clicked the change is not made and all values in the dialog box automatically go back to their original values. If you are working in a sub-dialog box, that is, a dialog box that is called from another dialog box, the changes are not actually implemented until the **OK** button is clicked in the topmost dialog box, that is, until the last dialog box is closed by clicking the **OK** button.

Shortcut:

Use the Redo button on the main tool-bar to redo a previously un-done operation.

Suppose for example that you are in a series of sub-dialog boxes that go two levels deep. In order to have the changes made at the second level be accepted and implemented you must click the **OK** button at the second and topmost levels. Clicking the **Cancel** button at either of these levels would cancel any of the changes made at that level and at any lower levels. Thus if you click the **Cancel** button in the topmost level of dialog boxes no changes will be made at any level.

Delete

In general the **Edit menu** > **Delete** command in ETABS works like the standard Windows delete command. This command deletes the selected object(s) and all of its associated properties from the section definition. Alternatively you can select the objects and press the Delete key on your keyboard to accomplish the same thing.

Align

The **Edit menu > Align** command allows you to align two or more shapes in your section in a variety of different ways. Clicking the **Edit menu > Align** command brings up a submenu with six different alignment options. Those options are Left, Center, Right, Top, Middle and Bottom.

To understand how the align command works you should visualize each shape enclosed by an imaginary bounding rectangle. The sides of the bounding rectangle always remain parallel to the Section Designer X and Y axes regardless of the rotation angle specified for the shape. In most cases (except for circles), changing the rotation angle can change the size of the imaginary bounding box.

Special note: For segments and sectors the bounding box is taken around the circle that defines the segment or sector rather than just the segment or sector itself.

Next, the **Edit menu > Align** command always aligns the selected shapes to the first selected shape. Thus it works best if you select the first object separately rather than windowing to select all of the objects at once. If you window to select all of the objects at once you have no way of knowing what ETABS assumes to be the first selected object.

Note:

*The **Edit menu > Align** command always aligns the selected shapes to the first selected shape.*

Now we can describe the six different alignment options.

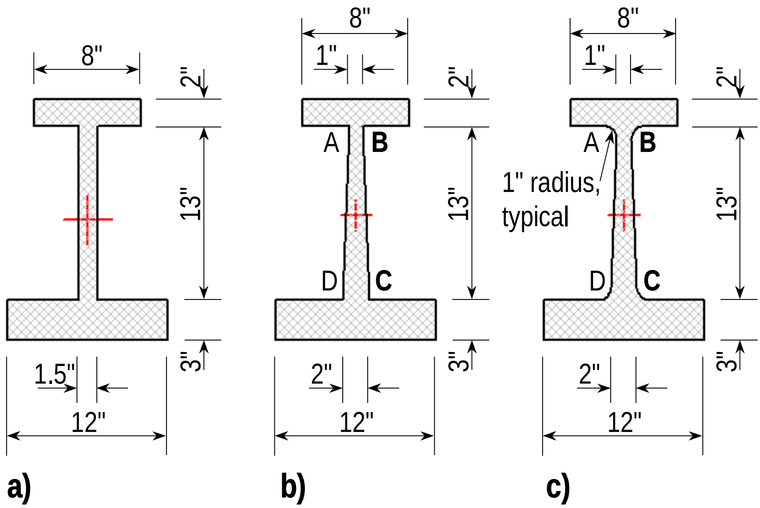
- **Left:** Horizontally aligns the left side of the imaginary bounding rectangle of each selected shape to the left side of the imaginary bounding rectangle of the first selected shape.

- **Center:** Horizontally aligns the center of the imaginary bounding rectangle of each selected shape to the center of the imaginary bounding rectangle of the first selected shape.

- **Right:** Horizontally aligns the right side of the imaginary bounding rectangle of each selected shape to the right side of the imaginary bounding rectangle of the first selected shape.

- **Top:** Vertically aligns the top of the imaginary bounding rectangle of each selected shape to the top of the imaginary bounding rectangle of the first selected shape.

Figure 5-1:
Example of a structural shape converted to a poly shape



- **Middle:** Vertically aligns the middle (center) of the imaginary bounding rectangle of each selected shape to the middle (center) of the imaginary bounding rectangle of the first selected shape.

- **Bottom:** Vertically aligns the bottom of the imaginary bounding rectangle of each selected shape to the bottom of the imaginary bounding rectangle of the first selected shape.

Change Shape to Poly

The **Edit menu > Change Shape to Poly** command applies to geometric shapes. It converts structural shapes and solid shapes to poly shapes.

Recall that the geometry of structural and solid shapes is defined by specified height and width items. The geometry of poly shapes is defined by the coordinates of the corner points of the poly shape.

Converting a structural or solid shape to a poly shape allows you to tweak the geometry of the shape in ways that you would not otherwise be able to do. As an example, consider the I-shape with a constant web thickness shown in Figure 5-1a. This section can be easily drawn using the predefined I/Wide flange shape in Section Designer.

Now suppose you want to draw the I-shape shown in Figure 5-1b where the web thickness varies from 1 inch at the top to 2 inches at the bottom. One way to create this section is:

1. Use the **Draw menu > Draw Structural Shape > I/Wide Flange** command to draw the structural shape.
2. Right click on the structural shape and define the properties as required to produce the shape shown in Figure 5-1a.
3. Use the **Edit menu > Change Shape to Poly** command to convert the structural shape into a poly shape.
4. Click the **Draw menu > Reshape Mode** command.
5. Left click once on the poly shape. Selection handles appear at all of the poly shape corner points.
6. Place the mouse pointer directly on the corner labeled A in Figure 5-1b. The mouse pointer will change shape when it is directly over the corner point. Click the *right* mouse button once to bring up the Change Coordinates dialog box.
7. Change the coordinates in the Change Coordinates dialog box as desired and then click the **OK** button. At point A you should modify the X-ordinate by 0.25 inches. For example, if the X-ordinate is originally -0.75 inches you should change it to -0.50 inches. Note that in the Change Coordinates dialog box you input the actual new coordinates, not the difference between the old and the new coordinates.
8. Repeat steps 5 and 6 at the other corner points of the poly shape that are labeled B, C and D.

Note:

*The **Edit menu > Change Shape to Poly** command converts structural shapes and solid shapes to poly shapes.*

An alternative method of creating the section shown in Figure 5-1b is to use the **Draw menu > Draw Poly Shape** command to draw it as a poly shape from the beginning.

Another reason you may want to change a structural or solid shape to a poly shape is to put a radius on the corner of the shape. You can only define corner radii in poly shapes, not in structural or solid shapes.

For example suppose you wanted to place fillets with a 1 inch radius at points A, B, C and D in the shape shown in Figure 5-1b. The section with the fillets is shown in Figure 5-1c. Use the following process to create the fillets:

1. If the shape is not already a poly shape then use the **Edit menu > Change Shape to Poly** command to convert it into a poly shape.

2. Click the **Draw menu > Reshape Mode** command.

3. Left click once on the poly shape. Selection handles appear at all of the poly shape corners.

4. Place the mouse pointer directly on the corner labeled A in Figure 5-1b. The mouse pointer will change shape when it is directly over the corner point. Click the *right* mouse button once to bring up the Change Coordinates dialog box.

5. Change the radius in the Change Coordinates dialog box from 0 to 1 and then click the **OK** button.

6. Repeat steps 3 and 4 at the other corner points of the poly shape that are labeled B, C and D.

Change Bar Shape to Single Bars

Note:

*The **Edit menu > Change Bar Shape to Single Bars** command only applies to poly shapes and additional reinforcing shapes.*

The **Edit menu > Change Bar Shape to Single Bars** command only applies to poly shapes and additional reinforcing shapes. It does not apply to structural and solid shapes. The command changes line pattern, rectangular pattern and circular pattern additional reinforcing shapes, and the reinforcing directly associated with poly shapes to a series of single bar reinforcing shapes.

This command does not change the reinforcing that is associated with solid or structural shapes. If you want to change the bars associated with a solid or structural shape to single bars then first change the structural or solid shape to a poly shape.

The Section Designer Nudge Feature

Section Designer includes a nudge feature that allows you to modify the geometry of your section by "nudging" shapes. This item is not on the Edit menu, but since it is an editing feature it is documented in this chapter. To use the nudge feature you simply select the shape(s) that you want to nudge and then press the Ctrl key and one of the arrow keys on your keyboard simultaneously. Note the following about the nudge feature:

- Pressing the Ctrl key plus the right arrow key nudges the object in the positive Section Designer X direction.

- Pressing the Ctrl key plus the left arrow key nudges the object in the negative Section Designer X direction.

- Pressing the Ctrl key plus the up arrow key nudges the object in the positive Section Designer Y direction.

- Pressing the Ctrl key plus the down arrow key nudges the object in the negative Section Designer Y direction.

- The distance that the object(s) are nudged (moved) when you press the Ctrl and arrow keys is specified by the Nudge Value item in the Section Designer Preferences. You can see this item by clicking the **Options menu > Preferences** command. The name of the item that controls the movement is Nudge Value.

Tip:

To nudge a selected shape press the Ctrl key and one of the arrow keys on your keyboard simultaneously.

Other Editing Features

Another editing feature not included on the Edit menu is the specification of the section local axis angle for frame sections. This item can be reviewed and/or modified using the **Display menu > Show Section Properties** command. See the section titled "Show Section Properties" in Chapter 9 for more information.

Chapter 6

The Section Designer View Menu

This chapter describes the commands available on the Section Designer View menu.

Zoom Commands

There are five zoom commands available in Section Designer. These commands allow you to zoom in or out on a section. Zooming in shows you a closer view of the section and zooming out shows you a farther away view of the section. All five zoom commands are available both on the View menu and on the top Section Designer toolbar. The zoom commands and their associated toolbar buttons are:

- **Rubber Band Zoom,**  **:** This command allows you to zoom in on the section by windowing. To use the command you depress and hold down the left button on your mouse. While keeping the left button depressed, drag your mouse to "rubber band" a window around the portion of the section that you want to zoom in on. The rubber band window that shows the extent you have

6

Note:

*Use the View menu > **Restore Full View** command to resize the display of the section such that it fits within the Section designer window.*

dragged the mouse appears as a dashed line on your screen. When you release the mouse left button the new view is displayed.

- **Restore Full View, :** This command returns you to a full view of the section. The view is sized such that the entire section is visible in the Section Designer window.

- **Previous Zoom, :** This command takes you back to your immediately previous zoom settings. If you use the **View menu > Previous Zoom** command repeatedly without using other commands to change the zoom in between then the effect is to toggle between two zoom settings. You can not use the **View menu > Previous Zoom** to go back more than one zoom setting.

  The **View menu > Previous Zoom** command has no effect in the following circumstances:

  ✓ Immediately after you first display a view in a window.

  ✓ Immediately after you use the **View menu > Restore Full View** command.

- **Zoom In One Step, :** This command zooms in on the section one step. The size of the step is controlled by the Auto Zoom Step item in the Preferences dialog box that you reach using the **Options menu > Preferences** command.

  The ETABS default value for the Auto Zoom Step is 10 percent. What this means is that when you use the **View menu > Zoom In One Step** command ETABS increases the magnification of all objects in the view by 10 percent.

- **Zoom Out One Step, :** This command zooms out on the section one step. The size of the step is controlled by the Auto Zoom Step item in the Preferences dialog box that you reach using the **Options menu > Preferences** command.

The ETABS default value for the Auto Zoom Step is 10 percent. What this means is that when you use the **View menu > Zoom Out One Step** command ETABS decreases the magnification of all objects in the view by 10 percent.

Pan Command

The **View menu > Pan** command allows you to move a view within the window such that you can see beyond the original edges of the view. The distance you can move beyond the original edge of the view is controlled by the Pan Margin item that is set in the preferences. The **Options menu > Preferences** command gives you access to the Pan Margin preference item. See the section titled "Preferences" in Chapter 10 for more information on the Pan Margin item.

Click the **View menu > Pan** command or the **Pan** button, , located on the main (top) toolbar to pan a view. Once you have clicked the menu command or toolbar button, click and hold down the left mouse button in the view and drag the mouse (while still holding down the left mouse button) to pan the view.

Show Guidelines

Note:

There are invisible gridlines between the guidelines. See the section titled "Preferences" in Chapter 10 for more information.

The **View menu > Show Guidelines** command toggles the display of the background guidelines on and off. Note the following about the guidelines.

- You can not snap to the guidelines unless they are visible.
- The spacing of the guidelines is controlled by the Background Guideline Spacing preference item. The **Options menu > Preferences** command gives you access to the Background Guideline Spacing preference item.
- The color of the guidelines item is controlled by the guidelines item that is accessed using the **Options menu > Colors** command.

Show Axes

The **View menu > Show Axes** command toggles the display of the section local axes and the Section Designer X and Y axes on and off. Note the following about these axes.

- The color of the local axes is controlled by the Local Axes item that is accessed using the **Options menu > Colors** command.

- The color of the Section Designer X and Y axes is controlled by the Text item that is accessed using the **Options menu > Colors** command.

Refresh Window

The **View menu > Refresh Window** command and the associated **Refresh Window** button, , on the top toolbar refreshes (redraws) the section designer window but does not rescale the view in any way. If you want to refresh the view and rescale it to fill the window then use the **View menu > Restore Full View** command or its associated toolbar button.

Chapter 7

The Section Designer Draw Menu

This chapter describes the commands that are available on the Section Designer Draw menu.

Select Mode

Tip:

Most of the tools available on the Draw menu are also available on the side toolbar.

The **Draw menu > Select Object** command is simply used to switch you from a drawing mode where mouse clicks draw shapes into a selection mode where mouse clicks select shapes. Alternative methods of switching from a drawing mode to a selection mode include pressing the Esc key on your keyboard, clicking the **Pointer** button, , on the side toolbar and executing one of the Select menu commands.

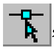
Reshape Mode

The **Draw menu > Reshape Object** command activates the reshaper tool.

Note:

The drawing constraints discussed in the section titled "Drawing Constraints in Section Designer" later in this chapter are available when you are in the reshape mode.

7

You can enter the reshape mode by either clicking on **the Reshaper** button, , on the side toolbar or by clicking **Draw menu** > **Reshape Mode**. Once you are in reshape mode you can do any of the following:

- Left click on a shape and while holding down the left mouse button drag it to a new location.
- For lines and poly shapes only, left click on the shape once and selection handles appear at the end or corner points of the shape. You can then do one of the following two things:
  - ✓ Left click and hold down the mouse left button on a selection handle at the end or corner point of the shape and drag that point to a new location.
  - ✓ Right click on a selection handle at the end or corner point of the shape to bring up the Change Coordinates dialog box. Type new coordinates for the point in this dialog box and then click the **OK** button.
- For shapes that are not lines and poly shapes, and are not rotated (i.e., Rotation = 0°), left click on the shape once and selection handles appear on a bounding box around the shape. You can then drag these selection handles to resize the bounding box and thus resize the shape.

When you are in reshape mode you remain in that mode until you do one of the following:

- Click the Pointer button, , on the side toolbar.
- Press the Esc key on your keyboard.
- Choose one of the drawing options from the Draw menu or the side toolbar.
- Click on one of the select items in the Select menu.
- Exit Section Designer.

Drawing Structural Shapes

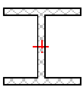
There are eight separate types of structural shapes in Section Designer. They are I/wide flange, channel, tee, angle, double angle, box/tube, pipe and plate. Typically the geometry of these shapes is defined by a center point, rotation and various width and thickness dimensions. The material associated with the shape is selected from any material property defined in ETABS.

Note that the center point of the shape is defined as the center of the rectangle that bounds the shape. The sides of the bounding rectangle are parallel to the Section Designer X and Y axes when the rotation of the shape is 0 degrees.

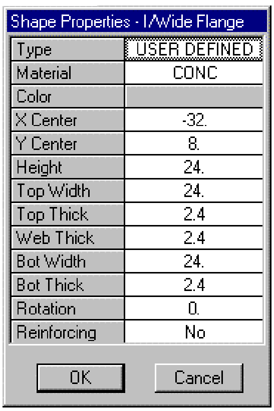
All structural shapes are initially drawn by selecting the appropriate **Draw menu > Draw Structural Shape** command or clicking the associated toolbar button, , on the side toolbar and then clicking the appropriate flyout toolbar button. You then left click once to draw the shape. The center of the shape is located at the location where you left click.

The structural shape initially comes into Section Designer with default dimensions and a default material property. You can then right click on the section to bring up the Shape Properties dialog box where you can modify its dimensions and properties.

The following subsections describe the shape properties associated with each of the eight structural shapes in Section Designer.



I/Wide Flange



You can draw an I/Wide Flange shape by clicking the **Draw menu > Draw Structural Shape > I/Wide Flange** command or by clicking the associated toolbar button, , that is located on the side toolbar. Following are the shape properties associated with the I/Wide Flange shape.

- **Type:** The shape type is either User Defined or any I/Wide Flange section that has been previously defined in ETABS, not including I/Wide Flange sections that were previously defined in Section Designer. You can change this item by right clicking in the cell that initially says User Defined and selecting any available section in the resulting drop-down box. If no additional I/Wide Flange sections are defined in ETABS then only the User Defined option is available in the drop-down box.

See the section titled "Important Note about Structural Shapes" later in this chapter for additional information.

- **Material:** The default material property for I/Wide Flange shapes is Steel. You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

  If you select a material property whose design type is Concrete then the Reinforcing property for the shape is visible. Otherwise it is not visible.

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.

- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Height:** The height of the shape measured from the bottom of the bottom flange to the top of the top flange.

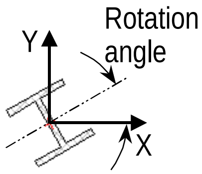
- **Top Width:** The width of the top flange of the shape.

- **Top Thick:** The thickness of the top flange of the shape.

- **Web Thick:** The thickness of the web of the shape.

- **Bot Width:** The width of the bottom flange of the shape.

- **Bot Thick:** The thickness of the bottom flange of the shape.



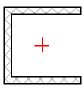
- **Rotation:** Angle (in degrees) measured from the Section Designer X-Axis to the original horizontal axis of the shape. See the sketch to the left.

  Note that the shape is rotated about the center of the bounding rectangle.

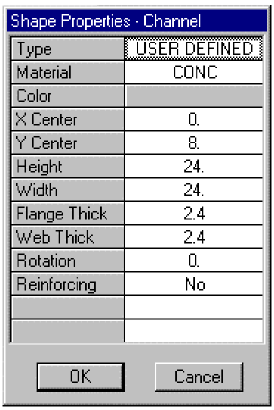
- **Reinforcing:** This item is only visible if the material property associated with the shape has a Concrete design type. The value of this item is either Yes or No. Setting it to Yes inserts edge reinforcing bars and corner bars in the shape. In other portions of this manual these bars are referred to as bars that are associated with the shape.

  If you set this item to No you can still use the **Draw menu > Draw Reinforcing Shape** command to place rebar in the shape.

  See the subsection titled "Reinforcing Associated with the Geometric Shape" in Chapter 3 for information about modifying the size, spacing and cover of the bars associated with the shape.



Channel



You can draw a Channel shape by clicking the **Draw menu > Draw Structural Shape > Channel** command or by clicking the associated toolbar button, , that is located on the side toolbar. Following are the shape properties associated with the Channel shape.

- **Type:** The shape type is either User Defined or any Channel section that has been previously defined in ETABS, not including Channel sections that were previously defined in Section Designer. You can change this item by right clicking in the cell that initially says User Defined and selecting any available section in the resulting drop-down box. If no additional Channel sections are defined in ETABS then only the User Defined option is available in the drop-down box.

  See the section titled "Important Note about Structural Shapes" later in this chapter for additional information.

- **Material:** The default material property for Channel shapes is Steel. You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

  If you select a material property whose design type is Concrete then the Reinforcing property for the shape is visible. Otherwise it is not visible.

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.

- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

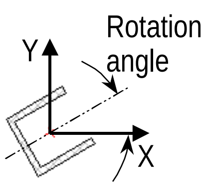
- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Height:** The height of the shape measured from the bottom of the bottom flange to the top of the top flange.

- **Width:** The width of the top and bottom flanges of the shape.

- **Flange Thick:** The thickness of the top and bottom flanges of the shape.

- **Web Thick:** The thickness of the web of the shape.



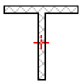
- **Rotation:** Angle (in degrees) measured from the Section Designer X-Axis to the original horizontal axis of the shape. See the sketch to the left.

  Note that the shape is rotated about the center of the bounding rectangle.

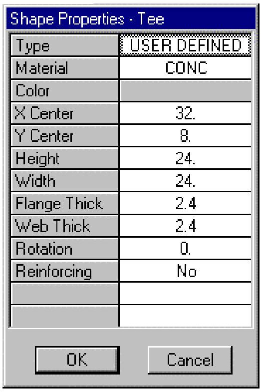
- **Reinforcing:** This item is only visible if the material property associated with the shape has a Concrete design type. The value of this item is either Yes or No. Setting it to Yes inserts edge reinforcing bars and corner bars in the shape. In other portions of this manual these bars are referred to as bars that are associated with the shape.

If you set this item to No you can still use the **Draw menu > Draw Reinforcing Shape** command to place rebar in the shape.

See the subsection titled "Reinforcing Associated with the Geometric Shape" in Chapter 3 for information about modifying the size, spacing and cover of the bars associated with the shape.



Tee



You can draw a Tee shape by clicking the **Draw menu > Draw Structural Shape > Tee** command or by clicking the associated toolbar button, , that is located on the side toolbar. Following are the shape properties associated with the Tee shape.

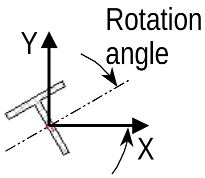
- **Type:** The shape type is either User Defined or any Tee section that has been previously defined in ETABS, not including Tee sections that were previously defined in Section Designer. You can change this item by right clicking in the cell that initially says User Defined and selecting any available section in the resulting drop-down box. If no additional Tee sections are defined in ETABS then only the User Defined option is available in the drop-down box.

  See the section titled "Important Note about Structural Shapes" later in this chapter for additional information.

- **Material:** The default material property for Tee shapes is Steel. You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

  If you select a material property whose design type is Concrete then the Reinforcing property for the shape is visible. Otherwise it is not visible.

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.
- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.
- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.
- **Height:** The height of the shape measured from the bottom of the web to the top of the flange.
- **Width:** The width of the flange of the shape.
- **Flange Thick:** The thickness of the flange of the shape.
- **Web Thick:** The thickness of the web of the shape.

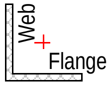


- **Rotation:** Angle (in degrees) measured from the Section Designer X-Axis to the original horizontal axis of the shape. See the sketch to the left.

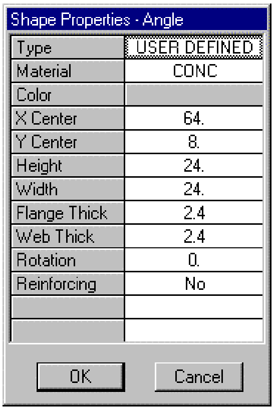
  Note that the shape is rotated about the center of the bounding rectangle.
- **Reinforcing:** This item is only visible if the material property associated with the shape has a Concrete design type. The value of this item is either Yes or No. Setting it to Yes inserts edge reinforcing bars and corner bars in the shape. In other portions of this manual these bars are referred to as bars that are associated with the shape.

  If you set this item to No you can still use the **Draw menu > Draw Reinforcing Shape** command to place rebar in the shape.

  See the subsection titled "Reinforcing Associated with the Geometric Shape" in Chapter 3 for information about modifying the size, spacing and cover of the bars associated with the shape.



Angle



You can draw an Angle shape by clicking the **Draw menu** > **Draw Structural Shape** > **Angle** command or by clicking the associated toolbar button, , that is located on the side toolbar.

When an Angle shape is initially drawn in Section Designer, one leg of the angle is drawn parallel to the Section Designer X-Axis. This leg is called the flange of the Angle shape. The other leg is drawn parallel to the Section Designer Y-Axis and it is called the web of the angle shape. Thus initially the flange is horizontal and the web is vertical. This orientation changes if you rotate the shape. For example, if after drawing the angle you rotate it 90 degrees then the flange is vertical and the web is horizontal.

Following are the shape properties associated with the Angle shape.

- **Type:** The shape type is either User Defined or any Angle section that has been previously defined in ETABS, not including Angle sections that were previously defined in Section Designer. You can change this item by right clicking in the cell that initially says User Defined and selecting any available section in the resulting drop-down box. If no additional Angle sections are defined in ETABS then only the User Defined option is available in the drop-down box.

  See the section titled "Important Note about Structural Shapes" later in this chapter for additional information.

- **Material:** The default material property for Angle shapes is Steel. You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

  If you select a material property whose design type is Concrete then the Reinforcing property for the shape is visible. Otherwise it is not visible.

7

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.

- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

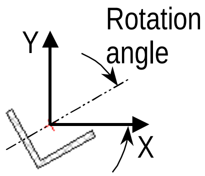
- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Height:** The height of the shape measured from the top of the web to the bottom of the flange.

- **Width:** The width of the flange of the shape.

- **Flange Thick:** The thickness of the flange of the shape.

- **Web Thick:** The thickness of the web of the shape.



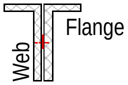
- **Rotation:** Angle (in degrees) measured from the Section Designer X-Axis to the original horizontal axis of the shape. See the sketch to the left.

  Note that the shape is rotated about the center of the bounding rectangle.

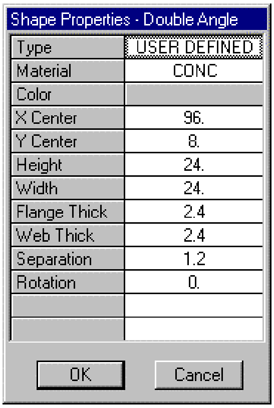
- **Reinforcing:** This item is only visible if the material property associated with the shape has a Concrete design type. The value of this item is either Yes or No. Setting it to Yes inserts edge reinforcing bars and corner bars in the shape. In other portions of this manual these bars are referred to as bars that are associated with the shape.

  If you set this item to No you can still use the **Draw menu > Draw Reinforcing Shape** command to place rebar in the shape.

  See the subsection titled "Reinforcing Associated with the Geometric Shape" in Chapter 3 for information about modifying the size, spacing and cover of the bars associated with the shape.



Double Angle



You can draw a Double Angle shape by clicking the **Draw menu > Draw Structural Shape > Double Angle** command or by clicking the associated toolbar button, , that is located on the side toolbar.

When a Double Angle shape is initially drawn, the angle legs parallel to the Section Designer X-axis are called the flange of the Double Angle shape. The angle legs parallel to the Section Designer Y-axis are called the web of the Double Angle shape. Thus initially the flange is horizontal and the web is vertical. This orientation changes if you rotate the shape. For example, if after drawing the Double Angle you rotate it 90 degrees then the flange is vertical and the web is horizontal.

Note that there is no reinforcing associated with a Double Angle shape. If you want to place reinforcing in a Double Angle shape then you must use the **Draw menu > Draw Reinforcing Shape** command, or its associated toolbar button, , to do it.

Following are the shape properties associated with the Double Angle shape.

- **Type:** The shape type is either User Defined or any Double Angle section that has been previously defined in ETABS, not including Double Angle sections that were previously defined in Section Designer. You can change this item by right clicking in the cell that initially says User Defined and selecting any available section in the resulting drop-down box. If no additional Double Angle sections are defined in ETABS then only the User Defined option is available in the drop-down box.

  See the section titled "Important Note about Structural Shapes" later in this chapter for additional information.

- **Material:** The default material property for Double Angle shapes is Steel. You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

7

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.

- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

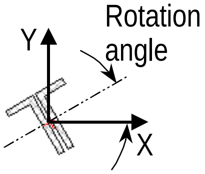
- **Height:** The height of the shape measured from the bottom of the web to the top of the flange.

- **Width:** The total width of the flange of the shape.

- **Flange Thick:** The thickness of the flange of the shape.

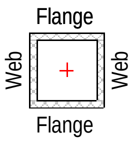
- **Web Thick:** The thickness of the web of *one of the angles* in the shape.

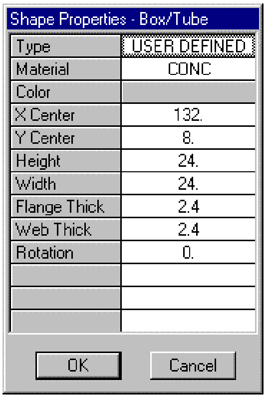
- **Separation:** The distance between the webs of the two angles.



- **Rotation:** Angle (in degrees) measured from the Section Designer X-Axis to the original horizontal axis of the shape. See the sketch to the left.

  Note that the shape is rotated about the center of the bounding rectangle.





Box/Tube

You can draw a Box/Tube shape by clicking the **Draw menu** > **Draw Structural Shape** > **Box/Tube** command or by clicking the associated toolbar button, , that is located on the side toolbar.

When a Box/Tube shape is initially drawn, the sides parallel to the Section Designer X-axis are called the flanges of the Box/Tube shape. The sides parallel to the Section Designer Y-axis are called the web of the Box/Tube shape. Thus initially the flanges are horizontal and the webs are vertical. This orientation changes if you rotate the shape. For example, if after drawing the Box/Tube you rotate it 90 degrees then the flanges are vertical and the webs are horizontal.

Note that there is no reinforcing associated with a Box/Tube shape. If you want to place reinforcing in a Box/Tube shape then you must use the **Draw menu** > **Draw Reinforcing Shape** command, or its associated toolbar button, , to do it.

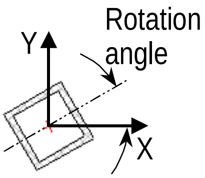
Following are the shape properties associated with the Box/Tube shape.

- **Type:** The shape type is either User Defined or any Box/Tube section that has been previously defined in ETABS, not including Box/Tube sections that were previously defined in Section Designer. You can change this item by right clicking in the cell that initially says User Defined and selecting any available section in the resulting drop-down box. If no additional Box/Tube sections are defined in ETABS then only the User Defined option is available in the drop-down box.

  See the section titled "Important Note about Structural Shapes" later in this chapter for additional information.

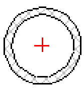
- **Material:** The default material property for Box/Tube shapes is Steel. You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.
- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.
- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.
- **Height:** The height of the shape measured from the bottom of the bottom flange to the top of the top flange.
- **Width:** The width of the flanges of the shape.
- **Flange Thick:** The thickness of the flanges of the shape.
- **Web Thick:** The thickness of the webs of the shape.

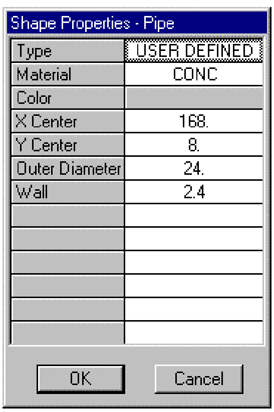


- **Rotation:** Angle (in degrees) measured from the Section Designer X-Axis to the original horizontal axis of the shape. See the sketch to the left.

  Note that the shape is rotated about the center of the bounding rectangle.

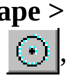


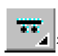
Pipe


Shape Properties - Pipe

Type	USER DEFINED
Material	CONC
Color	
X Center	168.
Y Center	8.
Outer Diameter	24.
Wall	2.4

OK Cancel

You can draw a Pipe shape by clicking the **Draw menu** > **Draw Structural Shape** > **Pipe** command or by clicking the associated toolbar button, , that is located on the side toolbar.

Note that there is no reinforcing associated with a Pipe shape. If you want to place reinforcing in a Pipe shape then you must use the **Draw menu** > **Draw Reinforcing Shape** command, or its associated toolbar button, , to do it.

Following are the shape properties associated with the Pipe shape.

- **Type:** The shape type is either User Defined or any Pipe section that has been previously defined in ETABS, not including Pipe sections that were previously defined in Section Designer. You can change this item by right clicking in the cell that initially says User Defined and selecting any available section in the resulting drop-down box. If no additional Pipe sections are defined in ETABS then only the User Defined option is available in the drop-down box.

- **Material:** The default material property for Pipe shapes is Steel. You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.

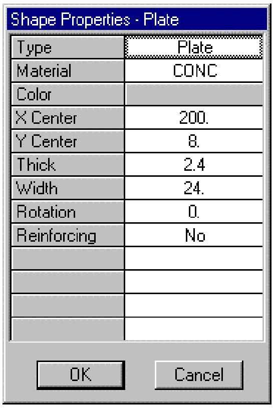
- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Outer Diameter:** The diameter of the pipe measured to the outside face of the pipe wall.

- **Wall:** The thickness of the pipe wall.

Plate



You can draw a Plate shape by clicking the **Draw menu > Draw Structural Shape > Plate** command or by clicking the associated toolbar button, , that is located on the side toolbar.

Following are the shape properties associated with the Plate shape.

- **Type:** This item is Plate to identify the type of structural shape. It is not editable by the user.

- **Material:** The default material property for Plate shapes is Steel. You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

  If you select a material property whose design type is Concrete then the Reinforcing property for the shape is visible. Otherwise it is not visible.

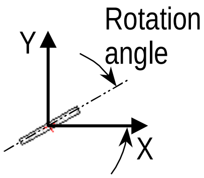
- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.

- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Thick:** The thickness of the shape. When the rotation angle is 0 degrees the thickness is measured parallel to the Section Designer Y-axis.

- **Width:** The width of the shape. When the rotation angle is 0 degrees the width is measured parallel to the Section Designer X-axis.



- **Rotation:** Angle (in degrees) measured from the Section Designer X-Axis to the original horizontal axis of the shape. See the sketch to the left.

  Note that the shape is rotated about the center of the bounding rectangle.

- **Reinforcing:** This item is only visible if the material property associated with the shape has a Concrete design type. The value of this item is either Yes or No. Setting it to Yes inserts edge reinforcing bars and corner bars in the shape. In other portions of this manual these bars are referred to as bars that are associated with the shape.

  If you set this item to No you can still use the **Draw menu > Draw Reinforcing Shape** command to place rebar in the shape.

  See the subsection titled "Reinforcing Associated with the Geometric Shape" in Chapter 3 for information about modifying the size, spacing and cover of the bars associated with the shape.

7

Important Note about Structural Shapes

If you specify that a structural shape in Section Designer has a Type other than User Defined, then Section designer does not actually calculate the properties for that shape. Instead it retrieves them from the ETABS database. Furthermore, Section Designer may not draw the shape 100% accurately. Instead it draws a representation of the non-User Defined shape as best it can based on the properties retrieved from the ETABS database.

For example, suppose that you change an I/Wide Flange shape from User Defined to W8X31. Section Designer picks up the properties for the W8X31 from the ETABS database. When Section Designer draws the W8X31 it draws everything correctly except that the fillets are not shown because Section Designer does not have enough information to draw them. Even though the fillets are not shown their effect is included in the properties that Section Designer considers for the shape because as previously mentioned those properties are taken directly from the ETABS database.

For I/Wide Flange shapes the types of things that may graphically appear simplified for non-User Defined shapes include:

- The fillets are not shown for W, M and HP shapes.
- For S shapes, the fillets, slope of the flanges and radius at the ends of the flanges are not shown.

For Channel shapes the fillets, slope of the flanges and radius at the ends of the flanges are not shown.

For Tee shapes the fillets are not shown.

For Angle and Double Angle shapes the fillets and radius at the ends of the webs and flanges are not shown.

For Box/Tubes the corner radii are not shown.

Drawing Solid Shapes

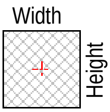
There are four separate types of solid shapes in Section Designer. They are rectangle, circle, segment and sector. Typically the geometry of these shapes is defined by a center point, rotation and various dimensions. The material associated with the shape is selected from any material property defined in ETABS.

Note that the center point of the shape is defined as the center of the rectangle that bounds the shape. The sides of the bounding rectangle are parallel to the Section Designer X and Y axes when the rotation of the shape is 0 degrees. The rectangles bounding the pie chord and pier arc shapes bound the entire circle that defines the shape.

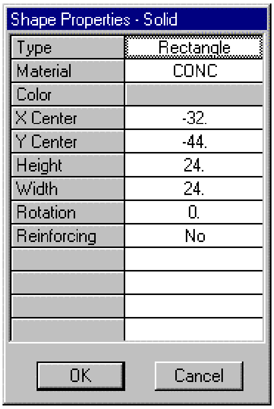
All solid shapes are initially drawn by selecting the appropriate **Draw menu > Draw Solid Shape** command or clicking the associated toolbar button, , on the side toolbar and then clicking the appropriate flyout toolbar button. You then left click once to draw the shape. The center of the shape is located at the location where you left click.

The solid shape initially comes into Section Designer with default dimensions and a default material property. You can then right click on the section to bring up the Shape Properties dialog box where you can modify its dimensions and properties.

The following subsections describe the shape properties associated with each of the four solid shapes in Section Designer.



Rectangle



Shape Properties - Solid

Type	Rectangle
Material	CONC
Color	
X Center	-32.
Y Center	-44.
Height	24.
Width	24.
Rotation	0.
Reinforcing	No

OK Cancel

You can draw a Rectangle shape by clicking the **Draw menu** > **Draw Solid Shape** > **Rectangle** command or by clicking the associated toolbar button, , that is located on the side toolbar.

When a Rectangle shape is initially drawn the height is measured parallel to the Section Designer Y-axis and the width is measured parallel to the Section Designer X-axis. Thus initially the height is vertical and the width is horizontal. This orientation changes if you rotate the shape. For example, if after drawing the Rectangle you rotate it 90 degrees then the height is horizontal and the width is vertical.

Following are the shape properties associated with the Rectangle shape.

- **Type:** This item is Rectangle to identify the type of solid shape. It is not editable by the user.

- **Material:** The default material property for Rectangle shapes is Concrete (CONC). You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.

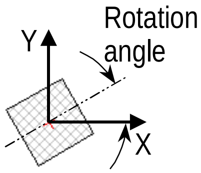
- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Height:** The height of the shape.

- **Width:** The width of the shape.

7



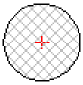
- **Rotation:** Angle (in degrees) measured from the Section Designer X-Axis to the original horizontal axis of the shape. See the sketch to the left.

  Note that the shape is rotated about the center of the bounding rectangle.

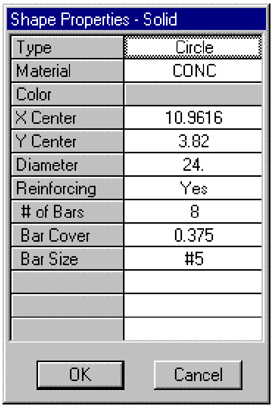
- **Reinforcing:** This item is only visible if the material property associated with the shape has a Concrete design type. The value of this item is either Yes or No. Setting it to Yes inserts edge reinforcing bars and corner bars in the shape. In other portions of this manual these bars are referred to as bars that are associated with the shape.

  If you set this item to No you can still use the **Draw menu > Draw Reinforcing Shape** command to place rebar in the shape.

  See the subsection titled "Reinforcing Associated with the Geometric Shape" in Chapter 3 for information about modifying the size, spacing and cover of the bars associated with the shape.



Circle



You can draw a Circle shape by clicking the **Draw menu** > **Draw Solid Shape** > **Circle** command or by clicking the associated toolbar button, , that is located on the side toolbar.

Following are the shape properties associated with the Circle shape.

- **Type:** This item is Circle to identify the type of solid shape. It is not editable by the user.

- **Material:** The default material property for Circle shapes is Concrete (CONC). You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.

- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Changing this coordinate relocates the shape.

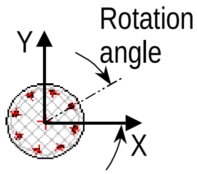
- **Diameter:** The diameter of the circle.

- **Reinforcing:** This item is only visible if the material property associated with the shape has a Concrete design type. The value of this item is either Yes or No. Setting it to Yes allows you to specify the number of equally spaced bars, the bar cover and the bar size.

  If you set this item to No you can still use the **Draw menu** > **Draw Reinforcing Shape** command to place rebar in the shape.

See the subsection titled "Reinforcing Associated with the Geometric Shape" in Chapter 3 for information about modifying the size, spacing and cover of the bars associated with the shape.

- **# of Bars:** This item is only visible if the Reinforcing item is set to Yes. It is the number of equally spaced bars for the circular reinforcing.

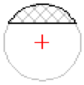


- **Rotation:** Angle (in degrees) measured from the Section Designer X-Axis to the first bar as illustrated in the sketch to the left. This item allows you to rotate the reinforcing steel associated with the shape to any angle.

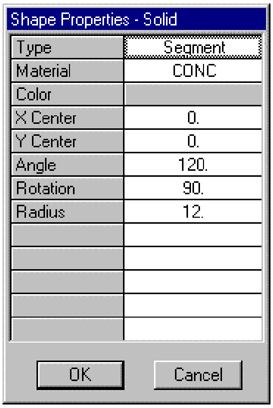
  Note that the shape is rotated about the center of the bounding rectangle.

- **Bar Cover:** This item is only visible if the Reinforcing item is set to Yes. It is the clear cover for the specified rebar.

- **Bar Size:** This item is only visible if the Reinforcing item is set to Yes. It is the size of the reinforcing bar. This can be any size that is currently defined in ETABS. Reinforcing bars are defined in ETABS using the **Options menu > Preferences > Reinforcement Bar Sizes** command.



Segment



You can draw a circular Segment shape by clicking the **Draw menu > Draw Solid Shape > Segment** command or by clicking the associated toolbar button, , that is located on the side toolbar.

Note that there is no reinforcing associated with a Segment shape. If you want to place reinforcing in a Segment shape then you must use the **Draw menu > Draw Reinforcing Shape** command, or its associated toolbar button, , to do it.

Following are the shape properties associated with the Segment shape.

- **Type:** This item is Segment to identify the type of solid shape. It is not editable by the user.

Note:

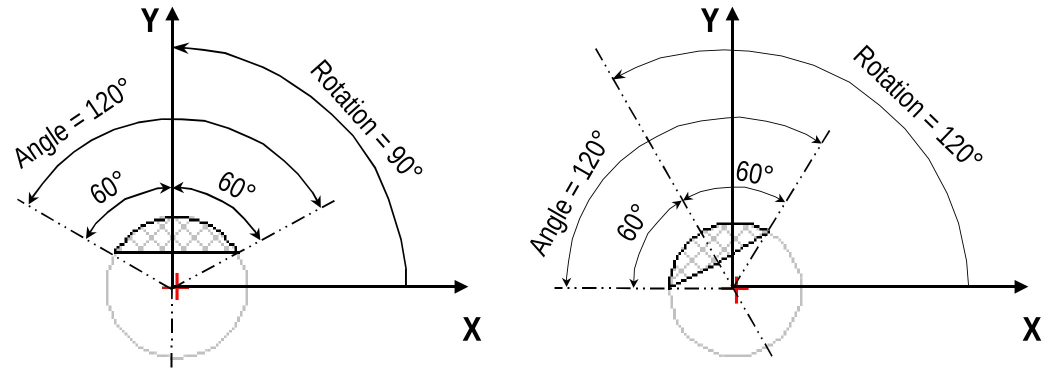
A circular segment is defined as the part of a circular area that is cut off by a chord.

- **Material:** The default material property for Segment shapes is Concrete (CONC). You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.

- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Note that for Segment shapes the bounding box bounds the circle that defines the circular segment. Changing this coordinate relocates the shape.

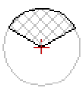
- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Note that for Segment shapes the bounding box bounds the circle that defines the circular segment. Changing this coordinate relocates the shape.



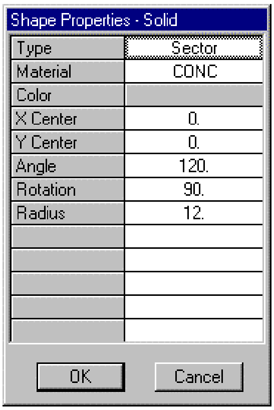
a) Rotation = 90°, Angle =120°   b) Rotation = 120°, Angle =120°

Figure 7-1: *Illustration of Angle and Rotation items for a Segment shape*

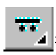
- **Angle:** The angle (in degrees) between lines drawn from the center of the circle to the end points of the chord that defines the segment. See Figure 7-1 for an example.
- **Rotation:** The angle (in degrees) measured from the Section Designer X-axis to a radial line that bisects the segment. See Figure 7-1 for an example.
- **Radius:** The radius of the circle defining the Segment.



Sector

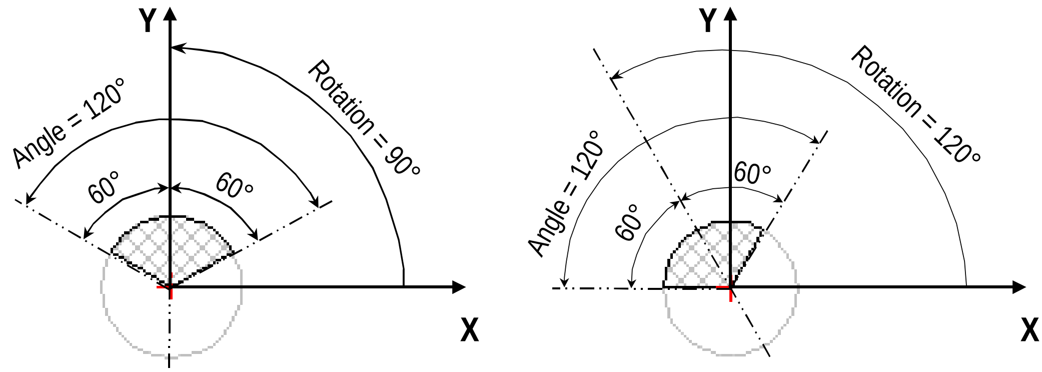


You can draw a circular Sector shape by clicking the **Draw menu** > **Draw Solid Shape** > **Sector** command or by clicking the associated toolbar button, , that is located on the side toolbar.

Note that there is no reinforcing associated with a Sector shape. If you want to place reinforcing in a Sector shape then you must use the **Draw menu** > **Draw Reinforcing Shape** command, or its associated toolbar button, , to do it.

Following are the shape properties associated with the Sector shape.

- **Type:** This item is Sector to identify the type of solid shape. It is not editable by the user.



a) Rotation = 90°, Angle =120°

b) Rotation = 120°, Angle =120°

Figure 7-2:
Illustration of Angle and Rotation items for a Sector shape

- **Material:** The default material property for Sector shapes is Concrete (CONC). You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

Note:

A circular sector is defined as a geometric figure bounded by two radii and the included arc of a circle.

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.

- **X Center:** The Section Designer X coordinate of the center of the bounding box for the shape. Note that for Sector shapes the bounding box bounds the circle that defines the circular sector. Changing this coordinate relocates the shape.

- **Y Center:** The Section Designer Y coordinate of the center of the bounding box for the shape. Note that for Sector shapes the bounding box bounds the circle that defines the circular sector. Changing this coordinate relocates the shape.

- **Angle:** The angle (in degrees) between the two radii that define the circular sector. See Figure 7-2 for an example.

- **Rotation:** The angle (in degrees) measured from the Section Designer X-axis to a radial line that bisects the Sector. See Figure 7-2 for an example.

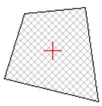
- **Radius:** The radius of the circle defining the Sector.

Drawing Poly Shapes

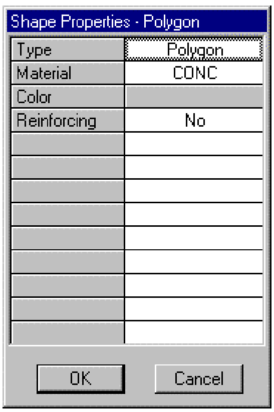
You can draw a Poly shape by clicking the **Draw menu > Draw Poly Shape** command or by clicking the associated toolbar button, , that is located on the side toolbar.

A Poly shape is defined by the coordinates of its corner points. To draw the Poly shape you simply click on the desired location of each corner point. You can complete the shape by double clicking on the last corner point, or by single clicking on the last point and then pressing the Enter key (or the Esc key) on your keyboard.

Note that the drawing constraints are available while you are drawing Poly shapes. See the section titled "Drawing Constraints in Section Designer" later in this chapter for more information.



You can use the Reshaping feature of Section Designer to modify the geometry of a Poly shape. See the section titled "Reshape Mode" earlier in this chapter for additional information.



Following are the shape properties associated with the Poly shape.

- **Type:** This item is Polygon to identify the type of shape. It is not editable by the user.

- **Material:** The default material property for Polygon shapes is Concrete (CONC). You can change this item to any material property that is defined in ETABS. To change this item right click in the cell and select any available material property in the resulting drop-down box.

- **Color:** This item controls the color of the fill for the shape. Left click on the cell to bring up the Color dialog box where you can set the color for the fill.

- **Reinforcing:** This item is only visible if the material property associated with the shape has a Concrete design type. The value of this item is either Yes or No. Setting it to Yes inserts edge reinforcing bars and corner bars in the shape. In other portions of this manual these bars are referred to as bars that are associated with the shape.

If you set this item to No you can still use the **Draw menu** > **Draw Reinforcing Shape** command to place rebar in the shape.

See the subsection titled "Reinforcing Associated with the Geometric Shape" in Chapter 3 for information about modifying the size, spacing and cover of the bars associated with the shape.

Drawing Reinforcing Shapes

There are four separate types of reinforcing shapes in Section Designer. They are single, line, rectangle and circle. Typically the geometry of these shapes is defined by a center point, rotation and various dimensions.

Note that the center point of the shape is defined as the center of the rectangle that bounds the shape. The sides of the bounding rectangle are parallel to the Section Designer X and Y axes when the rotation of the shape is 0 degrees.

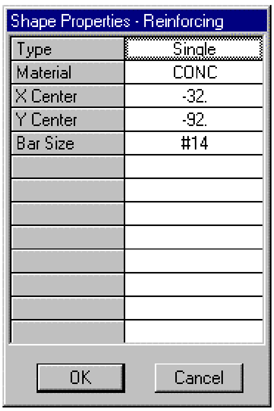
The material associated with the shape is selected from any material property defined in ETABS that has a Concrete design type. The yield stress of the reinforcing is determined from the material property. The modulus of elasticity of the reinforcing is always assumed to be 29000 ksi.

All reinforcing shapes are initially drawn by selecting the appropriate **Draw menu > Draw Reinforcing Shape** command or clicking the associated toolbar button, , on the side toolbar and selecting the appropriate flyout toolbar button. For all reinforcing shapes, except line pattern reinforcing, you then left click once to draw the shape. The center of the shape is located at the location where you left click. The shape initially comes into Section Designer with default dimensions and a default material property. You can then right click on the section to bring up the Shape Properties dialog box where you can modify its dimensions and properties.

To draw a line pattern reinforcing shape you left click once on the start point and once on the end point. The line reinforcing shape initially comes into Section Designer with default properties. You can then right click on the section to bring up the Shape Properties dialog box where you can modify its dimensions and properties.

The following subsections describe the shape properties associated with each of the four reinforcing shapes in Section Designer.

Single Bar



You can draw a single bar reinforcing shape by clicking the **Draw menu > Draw Reinforcing Shape > Single Bar** command or by clicking the associated toolbar button, , that is located on the side toolbar. Following are the shape properties associated with single bar reinforcing shapes.

- **Type:** This item is Single to identify the type of reinforcing shape. It is not editable by the user.

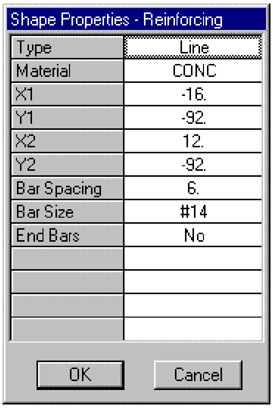
- **Material:** The material property associated with the reinforcing steel. This can be any material property with a Concrete design type. Section Designer uses the steel yield stress from the material property.

- **X Center:** The Section Designer X coordinate of the center of the bar.

- **Y Center:** The Section Designer Y coordinate of the center of the bar.

- **Bar Size:** The size of the reinforcing bar. This can be any size that is currently defined in ETABS. Reinforcing bars are defined in ETABS using the **Options menu > Preferences > Reinforcement Bar Sizes** command.

Line Pattern



You can draw a line pattern reinforcing shape by clicking the **Draw menu > Draw Reinforcing Shape > Line Pattern** command or by clicking the associated toolbar button, , that is located on the side toolbar. Following are the shape properties associated with line pattern reinforcing shapes.

- **Type:** This item is Line to identify the type of reinforcing shape. It is not editable by the user.

- **Material:** The material property associated with the reinforcing steel. This can be any material property with a Concrete design type. Section Designer uses the steel yield stress from the material property.

- **X1:** The Section Designer X coordinate of the first drawn end point of the line pattern reinforcing.

- **Y1:** The Section Designer Y coordinate of the first drawn end point of the line pattern reinforcing.

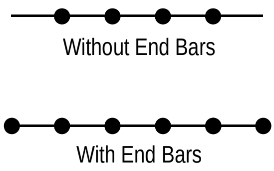
- **X2:** The Section Designer X coordinate of the second drawn end point of the line pattern reinforcing.

- **Y2:** The Section Designer Y coordinate of the second drawn end point of the line pattern reinforcing.

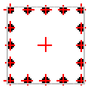
- **Bar Spacing:** The specified (not necessarily actual) center to center spacing of bars along the specified line.

  Section Designer calculates the required number of bars by dividing the length of the line by the specified bar spacing and adding one to the result. Usually this calculation leaves some fraction of a bar left over. If that fraction is greater than 0.1 then Section Designer rounds the number of bars up; otherwise it rounds the number of bars down. Once Section Designer knows the final number of bars to be used it calculates the final bar spacing by dividing the length of the line by the final number of bars minus one.

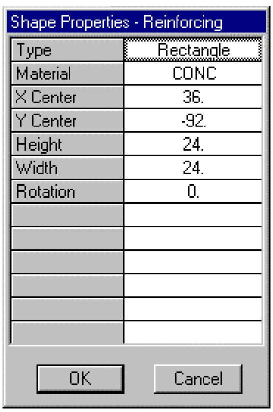
- **Bar Size:** The size of all of the reinforcing bars along the specified line. This can be any size that is currently defined in ETABS. Reinforcing bars are defined in ETABS using the **Options menu > Preferences > Reinforcement Bar Sizes** command.

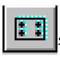


- **End Bars:** The value of this item is either Yes or No. Setting it to Yes means that there are bars at the end points of the specified line. If this item is set to No then the first bar starts one space in from the end point of the line. The sketch to the left illustrates line pattern reinforcing with and without end bars.



Rectangular Pattern



You can draw a rectangular pattern reinforcing shape by clicking the **Draw menu > Draw Reinforcing Shape > Rectangular Pattern** command or by clicking the associated toolbar button, , that is located on the side toolbar. Following are the shape properties associated with rectangular pattern reinforcing shapes.

- **Type:** This item is Rectangle to identify the type of reinforcing shape. It is not editable by the user.

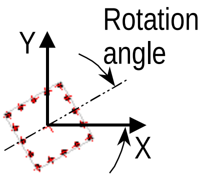
- **Material:** The material property associated with the reinforcing steel. This can be any material property with a Concrete design type. Section Designer uses the steel yield stress from the material property.

- **X Center:** The Section Designer X coordinate of the center of the rectangular bounding box for the reinforcing shape.

- **Y Center:** The Section Designer Y coordinate of the center of the rectangular bounding box for the reinforcing shape.

- **Height:** The height of the shape measured from the outside face of rebar at the bottom of the shape to the outside face of rebar at the top of the shape when the rotation angle is equal to 0 degrees.

- **Width:** The width of the shape measured from the outside face of rebar at the left side of the shape to the outside face of rebar at the right side of the shape when the rotation angle is equal to 0 degrees.



- **Rotation:** Angle (in degrees) measured from the Section Designer X-Axis to the original horizontal axis of the shape. See the sketch to the left.

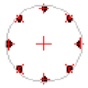
  Note that the shape is rotated about the center of the bounding rectangle.

You modify the rebar size and spacing in the rectangular pattern reinforcing by right clicking on it. The edge bars and the corner bars are edited separately.

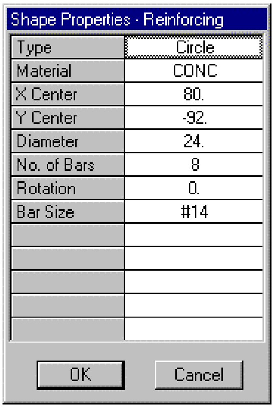
You can edit the size of any corner bar by right clicking on the bar. This brings up the Corner Point Reinforcing dialog box where you can specify the bar size by choosing from any defined bar size. There is also a check box in the dialog box that allows you to specify that the specified bar size applies to all corner bars associated with the reinforcing shape.

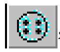
There are line pattern bars along each of the four edges of the rectangular reinforcing shape. The bars along any edge are independent of the bars along any other edge. The edge bars include all of the bars along an edge of the shape except for the corner bars.

The edge bars are specified by a bar size and spacing. All of the bars along an edge have the same size and spacing. See Item 3 in the subsection titled "Edge Bars" in Chapter 3 for information on how Section Designer calculates the exact number and spacing of edge bars.



Circular Pattern



You can draw a circular pattern reinforcing shape by clicking the **Draw menu > Draw Reinforcing Shape > Circular Pattern** command or by clicking the associated toolbar button, , that is located on the side toolbar. Following are the shape properties associated with circular pattern reinforcing shapes.

- **Type:** This item is Circle to identify the type of reinforcing shape. It is not editable by the user.

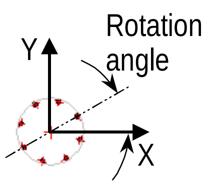
- **Material:** The material property associated with the reinforcing steel. This can be any material property with a Concrete design type. Section Designer uses the steel yield stress from the material property.

- **X Center:** The Section Designer X coordinate of the center of the rectangular bounding box for the reinforcing shape.

- **Y Center:** The Section Designer Y coordinate of the center of the rectangular bounding box for the reinforcing shape.

- **Diameter:** The diameter of the shape measured from the outside face of rebar to the outside face of rebar.

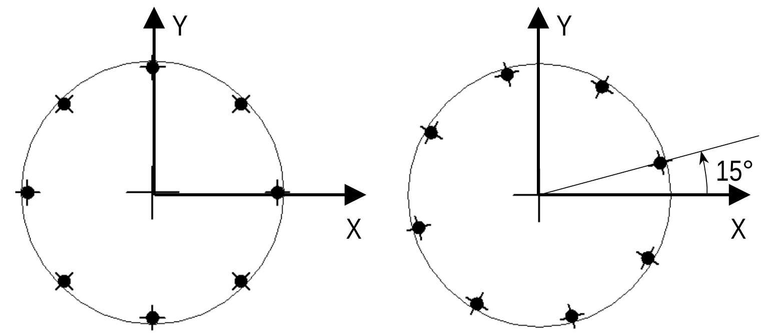
- **No. of Bars:** Number of equally spaced bars for the circular reinforcing pattern.



- **Rotation:** Angle (in degrees) measured from the Section Designer X-Axis to the first bar as illustrated in the sketch to the left. Figure 7-3 illustrates this angle in more detail.

  Note that the shape is rotated about the center of the bounding rectangle.

Figure 7-3:
Illustration of Rotation angle for a circular pattern reinforcing shape



a) Rotation Angle of 0°   b) Rotation Angle of 15°

Drawing Reference Lines

There are two separate types of reference lines in Section Designer. They are line and circle reference lines. Sometimes you may find it useful to snap to a reference line. Other times you may want to align items to a reference line.

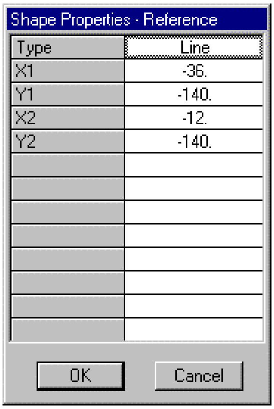
Reference lines and circles are initially drawn by selecting the appropriate **Draw menu > Draw Reference Lines** command or clicking the associated toolbar button, , on the side toolbar and selecting the appropriate flyout toolbar button.

The geometry of the reference line is defined by the coordinates of the end points. To draw a reference line you left click once on the start point and once on the end point. You can then, if desired, right click on the reference line to bring up the Shape Properties dialog box where you can modify its dimensions.

The geometry of the reference circle is defined by the coordinates of the center point and a diameter. To draw a reference circle you then left click once to draw the shape. The center of the shape is located at the location where you left click. The shape initially comes into Section Designer with default dimensions. You can then right click on the reference circle to bring up the Shape Properties dialog box where you can modify its dimensions.

The following subsections describe the shape properties associated with the two types of reference lines in Section Designer.

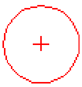
Line



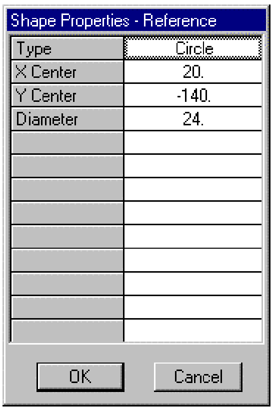
Shape Properties - Reference	
Type	Line
X1	-36.
Y1	-140.
X2	-12.
Y2	-140.

You can draw a reference line by clicking the **Draw menu > Draw Reference Lines > Draw Reference Line** command or by clicking the associated toolbar button, , that is located on the side toolbar. Following are the shape properties associated with reference lines.

- **Type:** This item is Line to identify the type of reference line. It is not editable by the user.
- **X1:** The Section Designer X coordinate of the first drawn end point of the line.
- **Y1:** The Section Designer Y coordinate of the first drawn end point of the line.
- **X2:** The Section Designer X coordinate of the second drawn end point of the line.
- **Y2:** The Section Designer Y coordinate of the second drawn end point of the line.



Circle



Shape Properties - Reference	
Type	Circle
X Center	20.
Y Center	-140.
Diameter	24.

You can draw a reference circle by clicking the **Draw menu > Draw Reference Lines > Draw Reference Circle** command or by clicking the associated toolbar button, , that is located on the side toolbar. Following are the shape properties associated with reference lines.

- **Type:** This item is Circle to identify the type of reference line. It is not editable by the user.
- **X Center:** The Section Designer X coordinate of the center of the circle.
- **Y Center:** The Section Designer Y coordinate of the center of the circle.
- **Diameter:** The diameter of the circle.

Snap Options

The Section Designer snap features allow you to snap to various items when you are drawing or editing lines or poly shapes. There are six separate snap features available in Section Designer. You can have these six features toggled on or off in any combination. You can toggle the snap features on and off using the **Draw menu > Snap to** command or by clicking one or more of the six snap feature toolbar buttons on the side toolbar. The six snap features and their associated toolbar buttons are:

Note:

The snap options are evaluated in the order they are listed here.

- **Guideline Intersections and Points, :** This feature snaps to guideline intersections and to the corner, end and center points of shapes.

- **Line Ends and Midpoints, :** This feature snaps to the ends and midpoints of lines and edges of shapes. Note that the end of an edge of a shape is a corner point of the shape.

- **Line Intersections, :** This feature snaps to the intersections of lines with other lines and with the edges of shapes. It does not snap to the intersection of the edge of one shape with the edge of another shape.

- **Perpendicular Projections, :** This feature works as follows. First draw the first point for a line or shape. Then, if this snap feature is active, place the mouse pointer over another line or edge of a shape and left click. A line object or edge of a shape is drawn from the first point perpendicular to the line object or edge of a shape that the mouse pointer was over when the second point was clicked.

- **Lines and Edges, :** This feature snaps to guidelines, lines and edges of shapes.

- **Snap to Fine Grid, :** This feature snaps to an invisible grid of points. The spacing of the points is controlled by the Fine Grids Between Guidelines item which is available under the **Options menu** > **Preferences** command.

Use the following procedure when using the snap commands:

- If the appropriate snap tool is not already activated then select it from the side toolbar or using the **Draw menu > Snap to** command.

- Move the mouse pointer in the graphics window.

- When a snap location is found close to the mouse pointer a dot appears at the snap location as well as a popup text field describing the snap location.

  Note the distance that the pointer must be from a snap location before it snaps to that location is controlled by the Screen Snap to Tolerance item which is available under the **Options menu > Preferences** command.

- When the desired snap location is found click the left mouse button to accept it.

- Modify the snap options if necessary and continue drawing or editing objects.

The snap options are evaluated in the order they are listed above. If more than one snap option is active and the mouse pointer is located such that it is within the screen snap to tolerance of two different snap features then it will snap to the snap feature that is first in the list above. This is true even if the item associated with the other snap feature is closer to the mouse pointer as long as both items are still within the screen snap to tolerance.

As an example, suppose Snap to Line Intersections and Snap to Fine Grids are both active. Assume that the mouse pointer is located such that it is within the screen snap to tolerance of both an intersection of two lines and one of the invisible grid points. The snap will be to the intersection of the two lines because this snap feature occurs first in the above list.

When two items from the same snap feature are within the screen snap to tolerance of the mouse pointer the snap occurs to the first drawn item which may or may not be the closest item.

Drawing Constraints in Section Designer

Drawing constraints provide the capability to constrain one of the axes when you are drawing or reshaping lines and poly shapes. Using the drawing constraints you can quickly draw a line or edge of a poly shape parallel to the Section Designer X or Y axes or at any arbitrary angle. The drawing constraint tools can be activated using the **Draw menu** > **Constrain Drawn Line to** command or they can be activated by pressing either the X, Y or A key on your keyboard.

7

Note:

Snaps can be used in conjunction with constraints. In this case only the unconstrained component of the selected snap point is used when a constraint is selected.

The drawing constraints include:

- **Constant X:** Locks the X component of the next point so that it is the same as the previous point.
- **Constant Y:** Locks the Y component of the next point so that it is the same as the previous point.
- **Constant Angle:** Allows you to specify an angle in degrees in the status bar at the bottom of the Section Designer window. Drawing is then constrained along this angle. The angles are measured from the Section Designer X-axis. Positive angles appear counterclockwise as you look down on the section.

  Note that when you use the keyboard command to specify a constant angle constraint, every time you press the A key on the keyboard the constant angle changes to be equal to the angle of the line that is within the screen selection tolerance (**Options menu** > **Preferences** > **Dimensions/Tolerances**) distance from the mouse pointer. You can, of course, always change this angle by editing it in the Section Designer status bar.
- **None:** Removes the current drawing constraint. Pressing the space bar on your keyboard also removes the current drawing constraint.

There are three steps to using the constraint tools:

- Locate the first point.

- Press one of the constraint keys on the keyboard (X, Y or A) or use the **Draw menu > Constrain Drawn Line to** command.

- Locate the next point. Section Designer only picks up the unconstrained component of the next point.

Drawing constraints are always removed as soon as you draw the next point.

Note that snaps can be used in conjunction with constraints. In this case only the unconstrained component of the selected snap point is used when a constraint is selected.

Chapter 8

The Section Designer Select Menu

This chapter describes the commands that are available on the Section Designer Select menu.

General

There are three basic methods of selecting objects in Section designer. They are:

Note:

The three basic selection methods are left click, window and intersecting line.

- **Left click:** Here you simply left click on a shape to select it. If there are multiple shapes one on top of the other then you can hold down the Ctrl key on your keyboard as you left click on the shapes. A dialog box will appear that allows you to specify which shape you want to select.

- **Window:** Here you draw a window around one or more shapes to select them. To draw a window around a shape first position your mouse pointer above and to the left of the shape(s) that you want to window. Then depress and hold down the left button on your mouse. While keeping the left button depressed drag your mouse to a position below and to the right of the shape(s) that you want to select. Finally release the left mouse button. Note the following about window selection:

✓ As you drag your mouse a "rubberband window" appears. The rubberband window is a dashed rectangle that changes shape as you drag the mouse. One corner of the rubberband window is at the point where you first depressed the left mouse button. The diagonally opposite corner of the rubberband window is at the current mouse pointer position. Any shape that is completely inside the rubberband window when you release the left mouse button is selected.

Note:

An entire shape must lie within the rubber band window for the shape to be selected.

✓ You do not necessarily have to start the window above and to the left of the shape(s) you are selecting. You could alternatively start the window above and to the right, below and to the left or below and to the right of the shapes(s) you want to select. In all cases you would then drag your mouse diagonally across the shapes(s) you want to select.

An *entire* shape must lie within the rubberband window for the shape to be selected.

- **Intersecting Line:** Here you draw a line through one or more shapes to select them. To use this selection method you first tell Section Designer that you want to use intersecting line selection either by clicking the **Select menu > Select > Intersecting Line** command or by clicking on the **Set Intersecting Line Select Mode** button, .

  Once you have told Section Designer to use the intersecting line selection method you then draw the intersecting line as follows. First position your mouse pointer to one side of the shapes(s) you want to select. Then depress and hold down the left button on your mouse. While keeping the left button depressed drag your mouse across the shapes(s) you want to select. Finally release the left mouse button. Note the following about the intersecting line selection method:

- ✓ As you drag your mouse a "rubberband line" appears. The rubber band line is a dashed line that changes length and orientation as you drag the mouse. It extends from the point where you first depressed the left mouse button to the current mouse pointer position. Any shape that is intersected (crossed) by the rubberband line when you release the left mouse button is selected.

- ✓ When you make a selection using the intersecting line method you do not then remain in an intersecting line mode such that you could immediately make another intersecting line selection. Instead you default back to a window selection mode. You must tell ETABS that you want to use the intersecting line selection method (either by clicking the **Select menu > Select > Intersecting Line** command or by clicking on the **Set Intersecting Line Select Mode** button, ) every single time you use the selection method even if you are doing several intersecting line selections in a row.

Select

The **Select menu > Select** command brings up a submenu with three choices. Those choices are:

- **Pointer/Window:** This command sets Section Designer in its default select mode where you can select shapes either by left clicking on them or windowing them. Anytime you are in select mode (not draw mode) this is the default mode of selection. Thus in most instances it is not necessary to click this command before making a selection. You can just simply make the selection by left clicking or windowing.

  The real purpose of this command is to switch you out of draw mode into select mode.

- **Intersecting Line:** This command sets Section Designer in the intersecting line select mode. See the section above titled "General" for more information.

- **All:** This command automatically selects all shapes in the section.

Deselect

You can deselect shapes one at a time by left clicking on the selected shapes. A more powerful way to deselect shapes is to use the **Select menu > Deselect** command. This gives you access to all of the above described selection methods except that now they are used to deselect rather than to select.

Get Previous Selection

The **Select menu > Get Previous Selection** command selects whatever shapes were last previously selected. For example suppose you select some shapes by clicking on them and then align their tops. You can then use the **Select menu > Get Previous Selection** command to select the shapes again and do something else to them such as change them to poly shapes. You can also use the **Restore Previous Selection** button, 

, on the side toolbar to execute this command.

Clear Selection

The **Select menu > Clear Selection** command clears the selection of all currently selected shapes. It is an all or nothing command. You can not selectively clear a portion of a selection using this command. If you want to selectively clear a selection you can either left click on the selected objects one at a time or you can use the Deselect command documented above. You can also use the **Clear Selection** button, 

, to clear the entire selection.

The Section Designer Display Menu

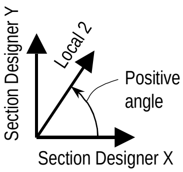
This chapter describes the commands that are available on the Section Designer Display menu.

Show Section Properties

The **Display menu > Show Section Properties** command brings up the Section Properties dialog box. This dialog box serves three functions:

1. It displays the section properties. See the subsection below titled "Section Properties" for additional information.

2. It allows you to view, but not edit, the section base material property. Do not confuse the section base material property with the material property of one of the shapes that makes up the section. See the subsection below titled "Section Properties" for additional information.



3. For frame sections only it allows you to modify the local axis angle for the section. This is the angle from the Section Designer X axis to the section local 2-axis. Positive angles appear counterclockwise as you look down on them as illustrated in the sketch to the left. You can not modify the local axis angle for wall pier sections; it is always 0 degrees.

Section Properties

The section properties are reported with respect to the section local axes (2-3), not the Section Designer X and Y axes. Furthermore the section properties are reported assuming that the entire section is transformed into an equivalent area of the specified base material. In other words, each infinitesimal area of the section, dA, is multiplied by the ratio E_{shape}/E_{base} when computing the section properties where E_{shape} and E_{base} are defined below. Using this transformation the following relationship holds true.

$$\sum_{shape=1}^{n} A_{shape} E_{shape} = A_{section} E_{base} \qquad \text{Eqn. 9-1}$$

where,

$A_{section}$ = Area reported for the section, length^2.

A_{shape} = Area of a geometric shape (not reinforcing shape) included in the section, length^2.

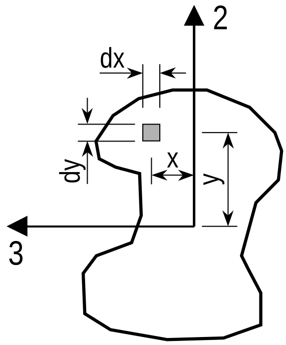
E_{base} = Modulus of elasticity of the base material, force/length^2.

E_{shape} = Modulus of elasticity of the material specified for the shape, force/length^2.

n = Number of geometric shapes included in the section, unitless.

<u>Important note:</u> The reinforcing steel is *not* considered when calculating the section properties. This includes the reinforcing steel defined as a reinforcing shape and the reinforcing steel that is associated with a geometric shape. The section properties are based on the gross area of all geometric shapes transformed to an equivalent area of the base material.

Figure 9-1:
Derivation of I_{23}



$$I_{22} = \int x^2 dxdy$$

$$I_{33} = \int y^2 dxdy$$

$$I_{23} = \int xy\, dxdy$$

The following section properties are reported:

- **A:** Area of the section, length^2.
- **J:** Torsional constant, length^4.
- **I33:** Moment of inertia about the 3-axis, length^4.
- **I22:** Moment of inertia about the 2-axis, length^4.
- **I23:** Moment of inertia, length^4. Figure 9-1 illustrates the derivation of I23, I22, and I33.

  The I23 moment of inertia is equal to zero when the local 2 and 3 axes are the principal axes of the section. If the local 2 and 3 axes are not the principal axes of the section then I23 is nonzero.

  Important note: The analysis algorithm in ETABS assumes that the local axes of frame sections are principal axes of the section. In other words it assumes that I23 is zero. Thus the I23 value reported in Section Designer is not used in the ETABS analysis. If the I23 value for a section is not close to zero then you may not be analyzing the section correctly in ETABS.
- **As2:** Shear area for shear parallel to the 2-axis, length^2.
- **As3:** Shear area for shear parallel to the 3-axis, length^2.

Note:

The section properties are based on the gross area of all geometric shapes transformed to an equivalent area of the base material. Reinforcing steel is not considered when calculating the section properties.

- **S33(+face):** Section modulus about the 3-axis at extreme fiber of the section in the *positive* 2-axis direction, length^3.

- **S22(+face):** Section modulus about the 2-axis at extreme fiber of the section in the *positive* 3-axis direction, length^3.

- **S33(-face):** Section modulus about the 3-axis at extreme fiber of the section in the *negative* 2-axis direction, length^3.

- **S22(-face):** Section modulus about the 2-axis at extreme fiber of the section in the *negative* 3-axis direction, length^3.

- **r33:** Radius of gyration about the 3-axis, length.

- **r22:** Radius of gyration about the 2-axis, length.

- **Xcg:** The Section Designer X-axis coordinate of the center of gravity of the section, length.

- **Ycg:** The Section Designer Y-axis coordinate of the center of gravity of the section, length.

Show Interaction Surface

Note:

*The **Display menu > Show Interaction Surface** command is only available for concrete sections that have reinforcing specified.*

The **Display menu > Show Interaction Surface** command is only available for concrete sections that have reinforcing specified. Clicking the **Display menu > Show Interaction Surface** command brings up the Interaction Surface dialog box.

In Section Designer the interaction surface is defined by a series of PM curves that are equally spaced around a 360 degree circle. For example, if 24 curves are specified (the default) then there is one curve every 360°/24 curves = 15°.

If you are specifying a frame section then the number of curves is taken from the ETABS Concrete Frame Design preferences. These are accessed in ETABS using the **Options menu > Preferences > Concrete Frame Design** command.

If you are specifying a wall pier section then the number of curves is taken from the ETABS Shear Wall Design preferences. These are accessed in ETABS using the **Options menu > Preferences > Shear Wall Design** command.

See Chapter 11 for discussion of how Section Designer computes the interaction surface. The following subsections discuss the various areas of the Interaction Surface dialog box including the table, charts, Options area, 3D View area and Edit menu.

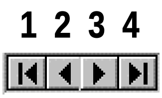
Table

<u>Important note:</u> Recall that in ETABS axial compression is negative and axial tension is positive. This holds true in the interaction surface tables.

If you are specifying a frame section then the number of points that are used to define an interaction curve is taken from the ETABS Concrete Frame Design preferences. These are accessed in ETABS using the **Options menu > Preferences > Concrete Frame Design** command.

If you are specifying a wall pier section then the number of points that are used to define an interaction curve is taken from the ETABS Shear Wall Design preferences. These are accessed in ETABS using the **Options menu > Preferences > Shear Wall Design** command.

Any point on one of the interaction curves is identified by a P, M2 and M3 coordinate. P is the axial load, M2 is the moment about the local 2-axis and M3 is the moment about the local 3-axis. The Interaction Surface dialog box displays the P, M2 and M3 coordinates of each of the curves that make up the interaction surface in a tabular format, one curve at a time.



You can use the arrow buttons below the table to scroll through the various PM curves. These arrow buttons are reproduced in the sketch to the left and labeled 1 through 4 for reference. Arrow button 1 jumps you up to the first PM curve (0°). Arrow button 2 takes you up one PM curve. Note that there is an arrow to the left of the table indicating the current line. Arrow button 3 takes you down one PM curve. Arrow button 4 jumps you down to the last PM curve. Note that the current curve number and its angle in degrees are reported below the table to the left of the arrows.

See the discussion of the Edit menu later for information on copying the data in the table to the Windows clipboard.

Charts

Important note: Recall that in ETABS and Section Designer axial compression is negative and axial tension is positive. This holds true in the interaction surface charts. *However, in the interaction surface charts the positive axial load (tension) axis points downward and the negative axial load (compression) axis points upward.* This is consistent with the way PM curves and PMM surfaces are usually plotted, with compression at the top of the chart and tension at the bottom.

Note:

In ETABS and Section Designer tension is positive.

Two charts are plotted in the Interaction Surface dialog box. The first is a 2D chart. This 2D chart is a cut through the interaction surface at the specified angle. The angle is shown below the table and can be changed using the arrow keys below the table. The origin of the 2D chart occurs at the intersection of the two red axes.

The other chart plotted is a 3D view of the interaction surface. You can use the controls in the 3D View area of the dialog box to rotate this view into any orientation. See the subsection below titled "3D View Area" for more information.

Options Area

The Options area provides three options for how the interaction surface is created. Those options are:

- **Phi:** The code specified strength reduction factors are considered when creating the interaction surface.

- **No phi:** The code specified strength reduction factors are *not* considered (i.e., set to 1.0) when creating the interaction surface.

- **No phi with fy increase:** The code specified strength reduction factors are *not* considered (i.e., set to 1.0) and the reinforcing steel yield stress is increased by a code-specified amount (1.25 for ACI and UBC) to estimate probable strength when creating the interaction surface.

3D View Area

The 3D View area provides controls for viewing the 3D chart of the interaction surface. You can define the view direction by specifying a plan angle and an elevation angle. All angles are specified in degrees. The view direction defines the location where you are standing as you view the interaction surface from the outside.

Figure 9-2a shows a three dimensional view of an interaction surface using the default view direction of plan angle = 315 degrees and elevation angle = 35 degrees. Figures 9-2b and c illustrate how the plan and elevation angles are defined. Following are explanations of the terms used in Figure 9-2.

- **Eye point:** This is the location from which you are viewing the interaction surface.

- **Target point:** This is the origin of the interaction surface.

- **View direction:** This is defined by a line drawn from the eye point to the target point.

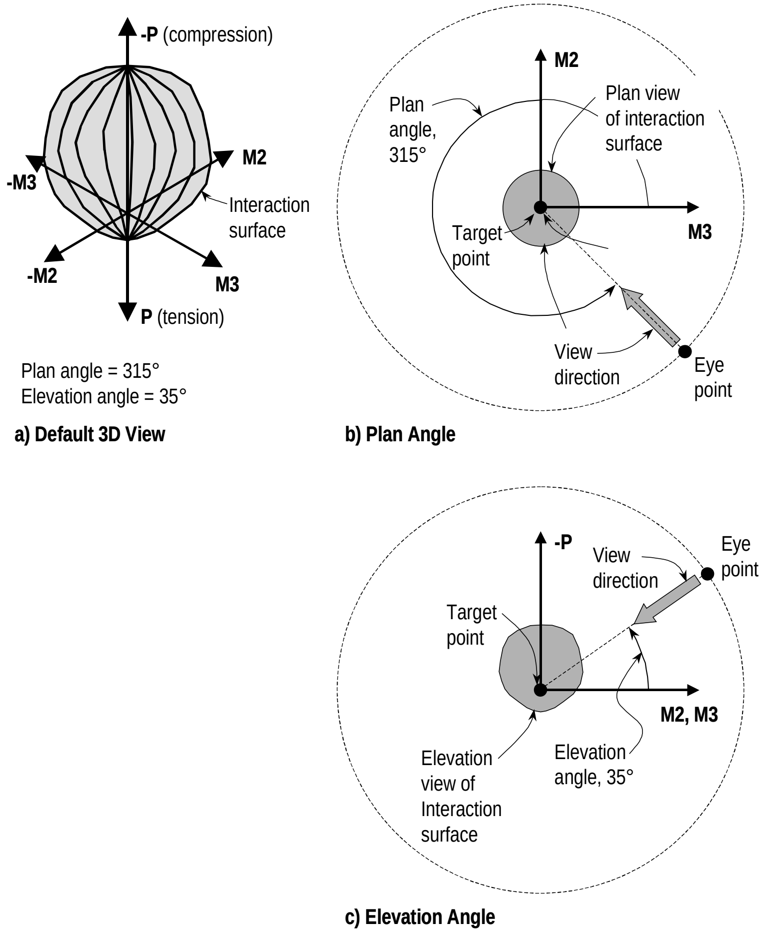
Note:

The plan and elevation angles together control the direction from the eye point to the target point.

- **Plan angle:** This is the angle (in degrees) from the positive M3-axis to the line defining the view direction measured in a horizontal plane. A positive angle appears counterclockwise as you look down on the interaction surface. Any value between -360 degrees and +360 degrees, inclusive, is allowed for the plan angle.

- **Elevation angle:** This is the angle (in degrees) from the M2-M3 plane to the line defining the view direction. A positive angle starts from the M2-M3 plane and proceeds toward the *negative* P-axis. A negative angle starts from the M2-M3 plane and proceeds toward the *positive* P-axis. (Recall that the negative P-axis points upward and the positive P-axis points downward). Any value be-

Figure 9-2: *Illustration of plan and elevation angle for 3D view of interaction surface*



9

tween -360 degrees and +360 degrees, inclusive, is allowed for the elevation angle.

The 3D View dialog box has four fast view buttons labeled 3d, MM, PM3 and PM2. The fast view buttons automatically set the plan and elevation angle to give you the specified 3D view. The fast view 3d view is as shown in Figure 9-2. The other fast views give you 3D perspective views of the specified planes.

Edit Menu

The Edit menu in the Interaction Surface dialog box has a Copy All command. This command copies the P, M2 and M3 values at each point for each interaction curve in the interaction surface to the Windows clipboard.

Show Moment Curvature Curve

Note:

The ***Display menu > Show Moment Curvature Curve*** *command is only available for concrete sections that have reinforcing specified.*

The **Display menu > Show Moment Curvature Curve** command is only available for concrete sections that have reinforcing specified. Clicking the **Show Moment Curvature Curve** command brings up the Moment Curvature Curve dialog box.

See Chapter 12 for discussion of how Section Designer computes the moment curvature curve. The following subsections discuss the various areas of the Moment Curvature Curve dialog box including the table, charts, edit boxes and Edit menu.

Table

The table lists the moment and curvature values at each of the considered points of the moment curvature curve. Note that the chart plots the values shown in the table connected by straight line segments.

If the section is capable of carrying the axial load specified in the Axial Load edit box up to the curvature specified in the Max Curvature edit box then the number of points shown in the table will equal the number specified in the No. of Points edit box. Otherwise there will be less points shown in the table than specified in the No. of Points edit box.

See the discussion of the Edit menu later for information on copying the data in the table to the Windows clipboard.

Chart

The chart plots the moment curvature curve with moment on the vertical axis and curvature on the horizontal axis. The chart is created by plotting the points shown in the table and connecting them with straight line segments.

You can run your mouse pointer over the chart and read the associated value of the moment curvature curve just below the chart. The curvature is listed first followed by the moment.

Edit Boxes

There are four edit boxes in the Moment Curvature Curve dialog box. They are discussed in the bullet items below.

Note:

The axial load input in the P (Tension Pos) edit box should be positive for tension and negative for compression.

- **P (Tension Pos):** The axial load for which the moment curvature curve is plotted. *Tension values are positive and compression values are negative in this edit box.*

- **Max Curvature:** The maximum curvature that Section Designer considers when plotting the moment curvature curve.

  Note that the section may not be capable of reaching the maximum curvature that you specify. In other words, the section may fail prior to reaching the curvature specified in the Max Curvature edit box. If this happens, the maximum curvature shown in the moment curvature chart or in the maximum curvature table may be less than the maximum curvature specified in the Max Curvature edit box.

  When the maximum curvature reported in the chart or table is less than the curvature you specified then you can conclude that the section is *not* capable of carrying the axial load specified in the Axial Load edit box at a curvature equal to that specified in the Max Curvature edit box.

- **Angle:** This angle defines the orientation (direction) of the neutral axis, but not its exact location. The angle is measured from the negative local 3-axis of the section. Positive angles appear counterclockwise as you look down on the section. The angle can vary from -360° to +360°.

Figure 9-3:
Illustration of angle specified in Angle edit box

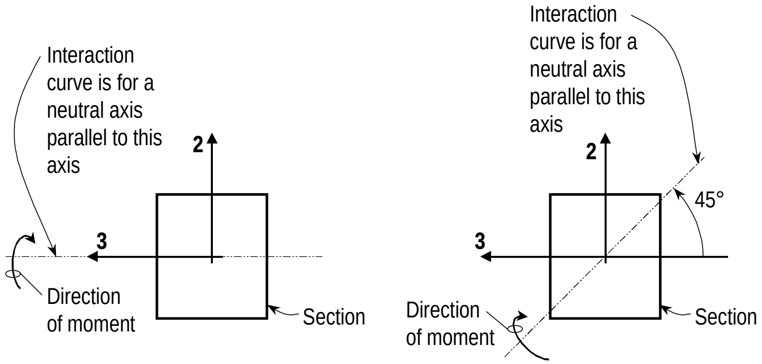
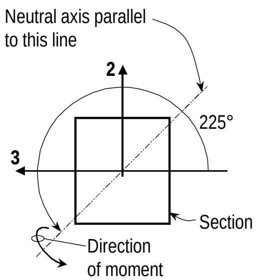


Figure 9-3 illustrates how the angle controls the orientation of the neutral axis. Figure 9-3a shows the angle at 0° and Figure 9-3b shows the angle at 45°.

The angle also dictates the direction of the moment considered. When the angle is 0 degrees the direction of moment is in the positive direction of the local 3-axis. You can use the right hand rule to get a sense of the direction of this moment. When the angle is 180 degrees the direction of moment is in the *negative* direction of the local 3-axis. Figure 9-3 illustrates the sense of the moment when the angle is 0 and 45 degrees.



Angle is 225 degrees

As an example, if the angle specified were 225 degrees, then the neutral axis would have the same orientation as that shown in Figure 9-3b but the direction of the moment would be reversed as shown in the sketch to the left.

- **No. of Points:** The number of points used to construct the moment curvature curve.

Edit Menu

The Edit menu in the Moment Curvature Curve dialog box has a Copy All command. This command copies the data in the table to the Windows clipboard.

Chapter 10

10

The Section Designer Options Menu

This chapter describes the commands that are available on the Section Designer Options menu.

Preferences

The **Options menu > Preference** command allows you to set various dimensional control items. Clicking this command brings up the Preferences dialog box. The following bullet items discuss the preference items included in this dialog box.

- **Background Guideline Spacing:** The spacing of the background guidelines in length units.

- **Fine Grids between Guidelines:** The number of equally spaced invisible grid lines between guidelines. As an example, specifying 3 means that there is a fine grid line at the one-quarter point, half-point and three-quarter point between guidelines.

- **Nudge value:** This is the distance that a nudged shape moves after you have pressed the appropriate key on the keyboard. This item is entered in length units. The ETABS default for this item is 48 inches in English units or 1 meter in metric units. See the section titled "The Section Designer Nudge Feature" in Chapter 5 for more information.

- **Screen selection tolerance:** When clicking on a shape to select it your mouse pointer must be within this distance of the shape to select it. This item is entered in pixels. The screen selection tolerance has no affect on selection by windowing. The Section Designer default for this item is 3 pixels.

Note:

A <u>pixel</u> is the smallest graphic unit (dot) that can be displayed on the screen. A typical screen resolution is 1024 pixels by 768 pixels.

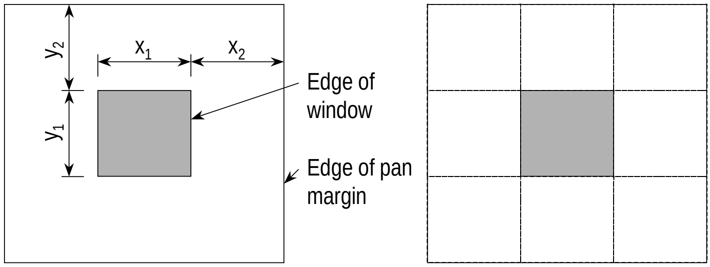
- **Screen snap to tolerance:** When using the snap features in ETABS your mouse pointer must be within this distance of a snap location to snap to it. This item is entered in pixels. The Section Designer default for this item is 12 pixels.

- **Screen line thickness:** This parameter controls the thickness of all lines on the screen. The thickness is entered in pixels. It also does not affect the aerial view. The Section Designer default for this item is 1 pixel.

- **Printer line thickness:** This parameter controls the thickness of lines and fonts that are output to the printer. The thickness is entered in pixels. The Section Designer default for this item is 4 pixels.

- **Pan margin:** This is the distance beyond the edge of a view that you can pan. It is entered as a percent of the window size. The Section Designer default for this item is 50%. This is a recommended value.

Figure 10-1:
Example of pan margin



a) Illustration showing a 100% pan margin

b) Illustration showing that a 100% pan margin covers nine times the area of the window

See Figure 10-1 showing an example of the pan margin. In the figure the window is shown shaded. Figure 10-1a shows an example of 100% pan margin. Note that the dimension x2 is equal to 100% of x1 and similarly y2 is equal to 100% of y1. Figure 10-1b illustrates that setting the pan margin to 100% allows you to potentially cover nine times more screen area than when the pan margin is set to 0%. If the pan margin is set to 0% you can not pan.

Note:

Do not set your pan margin too high. It could use up all of your computer's memory. The default value of 50% is typically sufficient.

Note that setting the pan margin to 100% also requires nine times more memory than when the pan margin is set to 0% because nine times more screen area must be saved in memory! Thus you need to be very careful with this control or you may use up all of your available memory and have a difficult time getting your ETABS model to run.

See the section titled "Pan Command" in Chapter 6 for additional information on panning.

- **Auto zoom step:** This is the size of the step used for the **View menu > Zoom In One Step** command and the **View menu > Zoom Out One Step** command as well as their associated toolbar buttons on the main (top) Section Designer toolbar. This parameter is entered in percent. The magnification of all objects in a view are increased or decreased by this percent. The ETABS default for this item is 10%.

See the section titled "Zoom Commands" in Chapter 6 for additional information.

Colors

The **Options menu** > **Colors** command allows you to set the colors of various items both for display on the screen and for output to a printer. Clicking this command brings up the Assign Display Colors dialog box. The following bullet items discuss the areas in this dialog box.

- **Click to Change Color:** In this area you can left click on any of the color boxes to change the display color for the associated item. Following is a list of items for which you can change the display color.

Note:

The colors used for the screen and for the printer are controlled separately.

✓ **Reinforcing:** This controls the color of the rebar. It applies to the rebar in reinforcing shapes and to the rebar associated with geometric shapes.

✓ **XY Axes:** This controls the color of the Section Designer X and Y axes.

✓ **Guide Lines:** This controls the color of the background guidelines. Note that you can also toggle these guidelines on and off using the **View menu** > **Show Guide Lines** command.

✓ **Local Axes:** This controls the color of the section local 2 and 3 axes.

✓ **Reference Lines:** This controls the color of reference lines and circles. It also controls the color of the line associated with line pattern reinforcing shapes and the color of the bounding line associated with rectangular and circular reinforcing shapes.

✓ **Background:** This is the background color of the Section Designer window. The outline of geometric shapes (I-sections, rectangles, etc.,) is always displayed in the color that is *opposite* of the background color.

The background item does not control the background color of the display windows for the interaction surface and the moment curvature curve. These background colors can not be changed.

- **Device Type:** Here you indicate whether the colors you are specifying are for screen display, output to a non-color printer or output to a color printer. Note that you can specify different display colors for each of these three device types.

- **Reset Defaults button:** This button resets the colors to the built-in ETABS default colors. The Reset Defaults button not only resets the colors for the currently chosen device type, it resets the colors for all three device types, regardless of which one is currently chosen.

The fill color for geometric shapes (I-sections, rectangles, etc.,) is set in the shape properties and is not controlled in any way by these options.

Interaction Surface

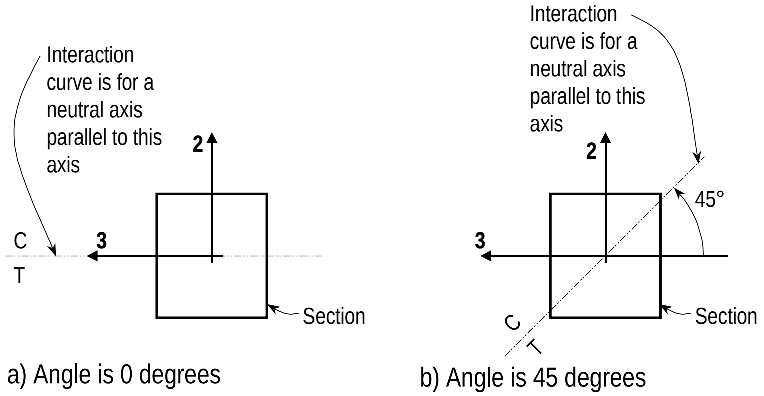
General

This chapter provides a general description of how Section Designer computes the interaction surface for a section. The exact details of how the interaction surface is calculated vary depending on the building code considered. However the general methodology is the same for all building codes.

For frame sections the interaction surface is based on the building code specified in the ETABS Concrete Frame Design preferences. These can be accessed in ETABS using the **Options menu > Preferences > Concrete Frame Design** command.

For wall pier sections the interaction surface is based on the building code specified in the ETABS Shear Wall Design preferences. These can be accessed in ETABS using the **Options menu > Preferences > Shear Wall Design** command.

Figure 11-1: *Orientation of neutral axis and direction of moment considered for various interaction curves*



11

Interaction Surface

In Section Designer a three-dimensional interaction surface is defined referenced to the P, M2 and M3 axes by a series of PMM interaction curves that are created by rotating the direction of the pier neutral axis in equally spaced increments around a 360 degree circle. For example, if 24 PMM curves are specified (the default) then there is one curve every 360°/24 curves = 15°.

Figure 11-1 illustrates the assumed orientation of the pier neutral axis and the associated sides of the neutral axis where the section is in tension (designated T in the figure) or compression (designated C in the figure) for 0 and 45 degree angles. Note that for a curve at 180 degrees the neutral axis is parallel to the curve at 0 degrees. Similarly, the neutral axis for a curve at 225 degrees is parallel to the curve at 45 degrees.

The following section describes how a typical interaction curve is created.

Interaction Curve

This section describes in general terms how an interaction curve is created in Section Designer. The exact details of how the curve is created depend on the building code considered.

Once the geometry and the vertical reinforcing distribution for a section is specified Section Designer can generate a strength interaction curve by plotting the design axial load strength, ϕP_n, against the corresponding design moment strengths, $\phi M2_n$ and $\phi M3_n$, assuming the neutral axis of the section to be oriented at any arbitrary angle in the Section Designer XY plane. Note that ϕ is a strength reduction factor or factor of safety depending on the building code considered.

Section Designer uses the requirements of force equilibrium and strain compatibility to determine the axial load and moment strength of the section. The axial force and moment coordinates of these points are determined by rotating a plane of linear strain on the section as described below in the section titled "Details of the Strain Compatibility Analysis."

The axial load at the balanced strain condition where the tension reinforcing reaches the strain corresponding to its specified yield strength, f_y, just as the concrete reaches its assumed ultimate strain is designated P_b.

The theoretical maximum compressive force the section can carry is designated P_{oc}. The theoretical maximum tension force the section can carry is designated P_{ot}. If the centroid of the concrete and the centroid of the reinforcing coincide then there are no wall moments associated with both P_{oc} and P_{ot}. Otherwise there are moments associated with both P_{oc} and P_{ot}.

By default 11 points are used to define an interaction curve, that is, to define an interaction curve from P_{oc} to P_{ot}. This number of points is specified, and can be modified, in the ETABS design preferences for the type of section you are defining. For frame sections, modify the Concrete Frame Design preferences. For wall piers, modify the Shear Wall Design preferences. If you input an even number for the number of points in the preferences it is incremented up to the next odd number by ETABS.

Section Designer includes the points at P_b, P_{oc} and P_{ot} on the interaction curve. Half of the remaining number of specified points on the interaction curve occur between P_b and P_{oc} at approximately equal spacing along the P axis. The other half of the remaining number of specified points on the interaction curve occur between P_b and P_{ot} at approximately equal spacing along the P axis.

Figure 11-2:
Typical interaction curve

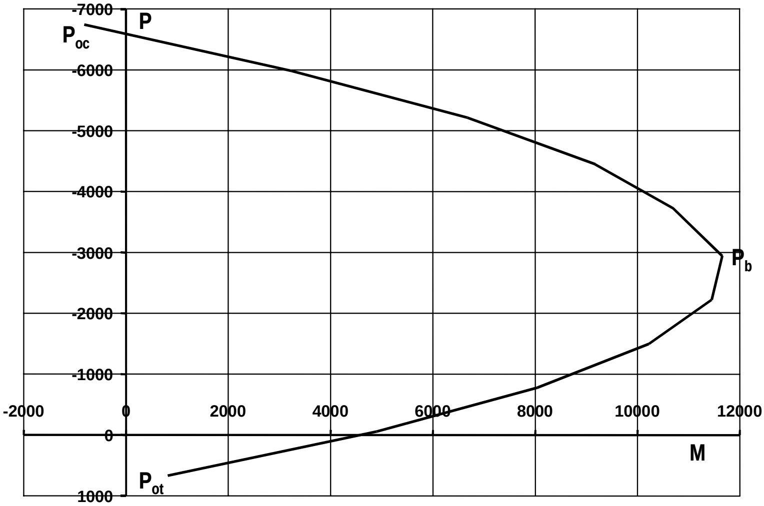


Figure 11-2 shows a typical interaction curve for a section. The points P_b, P_{oc} and P_{ot} are labeled. Note that since P_{oc} and P_{ot} do not fall on the P axis you can surmise that the centroid of the reinforcing does not coincide with the centroid of the concrete for the section considered.

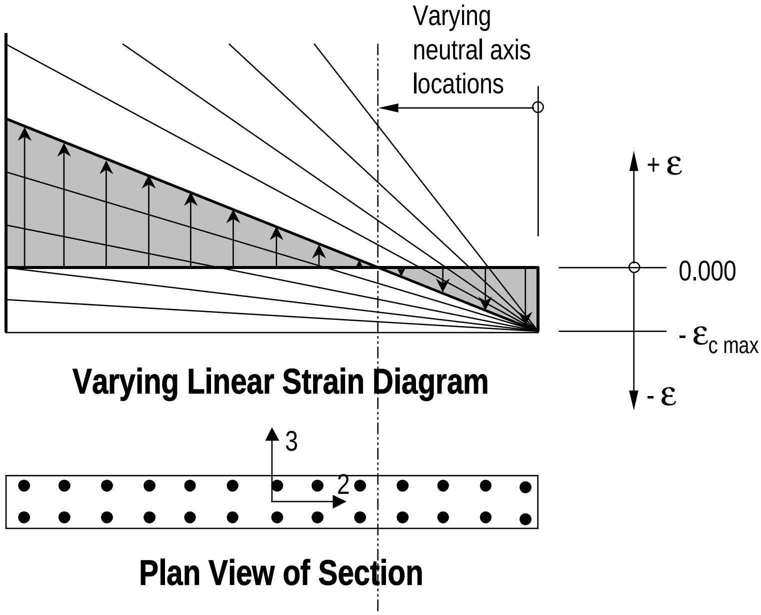
Details of the Strain Compatibility Analysis

Figure 11-3 illustrates varying planes of linear strain that Section Designer considers on a wall pier section when the angle between the 3-axis and the neutral axis is taken as 0 degrees. In these planes, the maximum concrete strain is always taken as $-\varepsilon_{c\ max}$ and the maximum steel strain is varied from $-\varepsilon_{c\ max}$ to plus infinity. Note that the value of $\varepsilon_{c\ max}$ is code-dependent.

When the steel strain is $-\varepsilon_{c\ max}$ the maximum compressive force in the wall pier, P_{oc}, is obtained from the strain compatibility analysis. When the steel strain is plus infinity the maximum tensile force in the wall pier, P_{ot}, is obtained. When the maximum steel strain is equal to the yield strain for the reinforcing (e.g., 0.00207 for f_y = 60 ksi) then P_b is obtained.

Figure 11-4 illustrates the concrete wall pier stress-strain relationship that is obtained from a strain compatibility analysis of a typical plane of linear strain shown in Figure 11-3.

Figure 11-3:
Varying planes of linear strain



In Figure 11-4 the compressive stress in the concrete, C_c, is calculated using Equation 11-1.

$$C_c = \text{Factor}\,(f'_c * A_c * b_1) \qquad \text{Eqn. 11-1}$$

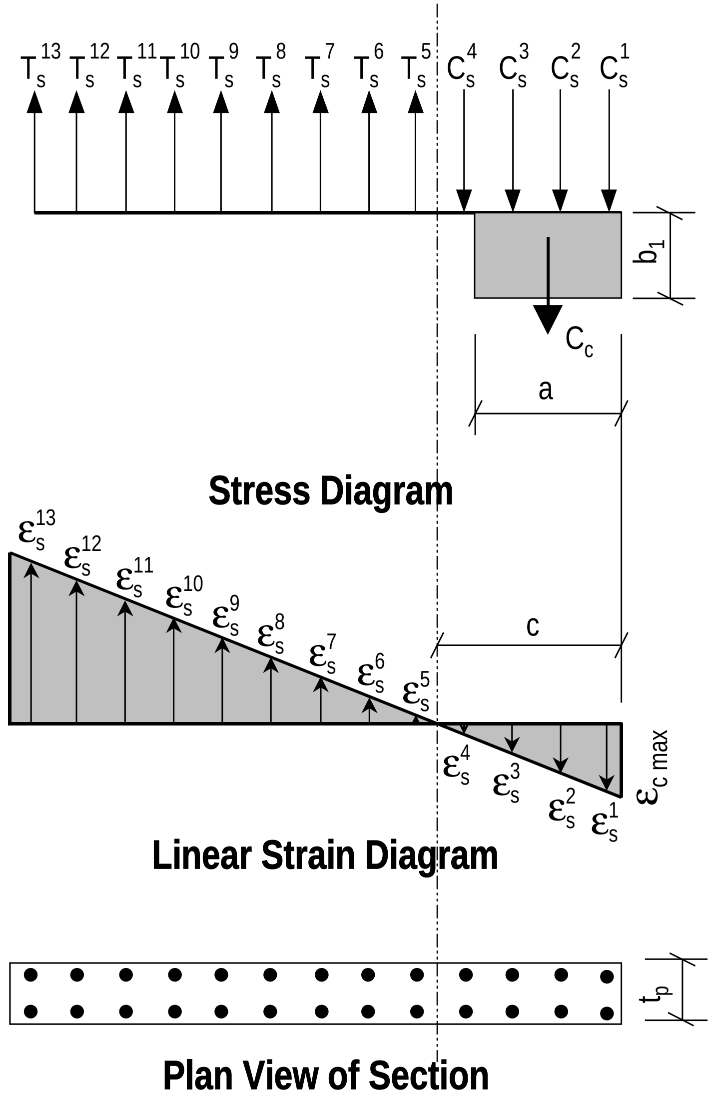
In Equation 11-1 the term "Factor" is a code-dependent term. The term A_c is the area of concrete that is effective in compression. For the example shown in Figure 11-4, $A_c = a * t_p$ where "a" is the length of the rectangular stress block assumed for the concrete.

In Figure 11-4 a value for maximum strain in the reinforcing steel is assumed. Then the strain in all other reinforcing steel is determined based on the assumed plane of linear strain. Next the stress in the reinforcing steel is calculated using Equation 11-2 where ε_s is the strain, E_s is the modulus of elasticity, σ_s is the stress and f_y is the yield stress of the reinforcing steel.

$$\sigma_s = \varepsilon_s E_s \le f_y \qquad \text{Eqn. 11-2}$$

The force in the reinforcing steel (T_s for tension or C_s for compression) is calculated using Equation 11-3.

Figure 11-4:
Wall pier stress-strain relationship



T_s or $C_s = \sigma_s A_s$ Eqn. 11-3

For the given distribution of strain the value of P is calculated using Equation 11-4.

$\phi P = \phi(\Sigma T_s - C_c - \Sigma C_s) \leq P_{max}$ Eqn. 11-4

In Equation 11-4 the tensile force T_s and the compressive forces C_c and C_s are all positive. If P is positive it is tension and if it is negative it is compression. The term P_{max} is a limiting value that is code dependent. The term ϕ is a strength reduction factor (factor of safety).

The value of $\phi M2_n$ is calculated by summing the moments due to all of the forces about the pier local 2-axis. Similarly, the value of $\phi M3_n$ is calculated by summing the moments due to all of the forces about the pier local 3-axis. The forces whose moments are summed to determine $\phi M2_n$ and $\phi M3_n$ are ϕP_n, ϕC_c, all of the ϕT_s forces and all of the ϕC_s forces.

The ϕP_n, $\phi M2_n$ and $\phi M3_n$ values calculated as described above make up one point on the wall pier interaction diagram. Additional points on the diagram are obtained by making different assumptions for the maximum steel stress, that is, considering a different plane of linear strain, and repeating the process.

When one interaction curve is complete the next orientation of the neutral axis is assumed and the points for the associated new interaction curve are calculated. This process continues until the points for all of the specified curves have been calculated.

Chapter 12

12

Moment Curvature Curve

General

This chapter provides a general description of how Section Designer computes the moment curvature curve for a section. The exact details of how the moment curvature curve is calculated vary depending on the building code considered. However the general methodology is the same for all building codes.

Three items are required for Section Designer to construct the moment curvature curve. They are a stress-strain diagram for concrete, a stress-strain diagram for reinforcing steel and an assumed building code. The stress-strain diagrams are built into Section Designer and are discussed in the section titled "Stress-Strain Diagrams" later in this chapter.

For frame sections the moment curvature curve is based on the building code specified in the ETABS Concrete Frame Design preferences. These can be accessed in ETABS using the **Options menu** > **Preferences** > **Concrete Frame Design** command.

For wall pier sections the moment curvature curve is based on the building code specified in the ETABS Shear Wall Design preferences. These can be accessed in ETABS using the **Options menu** > **Preferences** > **Shear Wall Design** command.

Stress-Strain Diagrams

Figure 12-1 illustrates the stress-strain diagrams used by Section Designer. These stress-strain diagrams are built directly into the program. Currently you can not modify them and you can not define your own stress-strain diagrams.

Note the following about the Section Designer stress-strain diagram for concrete shown in Figure 12-1a:

- For strains greater than ε_0 each segment of the stress-strain diagram is linear. For strains less than ε_0 the variation of the stress-strain diagram is parabolic.
- The value of ε_0 is given by Equation 12-1.

$$\varepsilon_0 = 2\frac{f'_c}{E_c} \qquad \text{Eqn. 12-1}$$

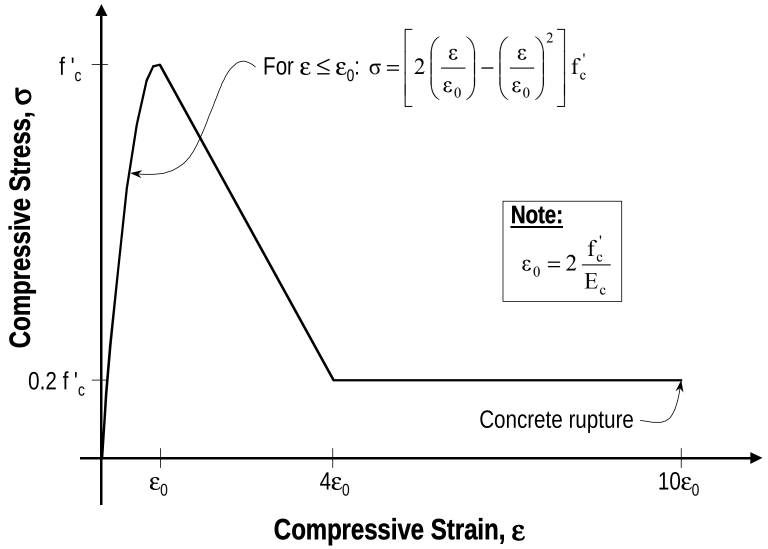
- For strains less than ε_0 the concrete stress is given by Equation 12-2.

$$\sigma = \left[2\left(\frac{\varepsilon}{\varepsilon_0}\right) - \left(\frac{\varepsilon}{\varepsilon_0}\right)^2\right] f'_c \qquad \text{Eqn. 12-2}$$

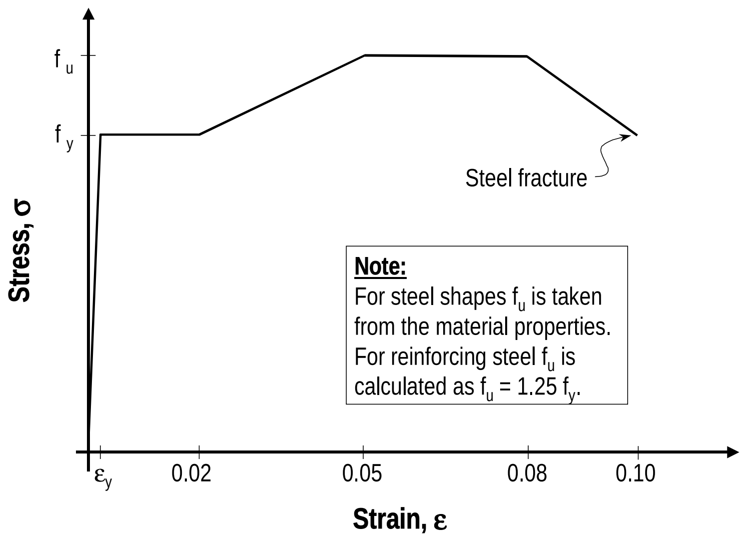
  Internally in the program the curve where $\varepsilon \le \varepsilon_0$ is idealized as a series of straight line segments by using Equation 12-2 to calculate σ for the following discrete values of ε: 0, $0.2\varepsilon_0$, $0.4\varepsilon_0$, $0.6\varepsilon_0$, $0.8\varepsilon_0$, $0.9\varepsilon_0$ and ε_0.

- Concrete rupture occurs at a strain equal to $10\varepsilon_0$.
- When there is tensile strain the stress is zero.

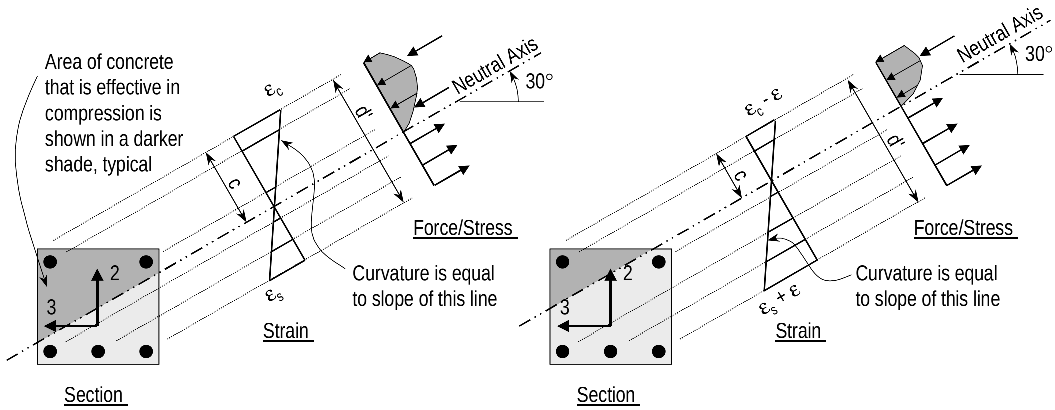
Figure 12-1: *Stress-strain diagrams used by Section Designer for concrete and steel*



a) Stress-Strain Relationship used for Concrete



b) Stress-Strain Relationship used for Steel Shapes and Reinforcing



12

a) b)

Figure 12-2: Stress and strain diagrams

Note the following about the Section Designer stress-strain diagram for steel shown in Figure 12-1b:

- The diagram applies to both steel shapes and steel reinforcing.
- The diagram applies for both tension and compression.
- The stress f_u is determined from the associated material property for steel shapes. For reinforcing steel f_u is calculated as $f_u = 1.25\ f_y$.
- The yield strain, ε_y, is calculated given by Equation 12-3.

$$\varepsilon_y = \frac{f_y}{E_s} \qquad \text{Eqn. 12-3}$$

- Steel fracture occurs at a strain equal to 0.10.

Background Information

Refer to Figure 12-2a that shows a rectangular section. Moments are considered about an axis rotated 30 degrees from the local 3-axis. A given axial load and a given moment cause the strain and force/stress diagrams shown. Note that the stresses are determined from stress-strain diagrams that are built into Section Designer. See the previous section titled “Stress-Strain Diagrams” for more information. For the given axial load and moment there is a unique location of the neutral axis. This location is designated by the distance c in the figure.

Note:

For moment curvature calculations the concrete stress is based on a concrete stress-strain diagram rather than a simplified rectangular stress block.

The maximum concrete strain in Figure 12-2a is denoted $-\varepsilon_c$ and the maximum reinforcing steel strain is denoted ε_s. The distance from the extreme compression fiber to the center of the extreme rebar in tension is labeled d'.

The curvature of the section for the given axial load and moment is equal to the slope of the strain diagram, that is, the curvature, κ_a, is given by Equation 12-4.

$$\kappa_a = \frac{\varepsilon_s + \varepsilon_c}{d'} \qquad \text{Eqn. 12-4}$$

The axial force associated with Figure 12-2a is equal to the sum of all the forces, considering signs, in the force diagram. The moment associated with Figure 12-2a is equal to the sum of all the forces in the force diagram times their respective distances to the section centroid, considering signs.

Now consider the section shown in Figure 12-2b. This section has the same geometry and reinforcing as that in Figure 12-2a. Moments are considered about an axis rotated 30 degrees from the local 3-axis, just like in Figure 12-2a. However, the axial load and moment on the section are different from that in Figure 12-2a. The axial load and moment in Figure 12-2b are specially chosen such that they give the section the same curvature as the axial load and moment in Figure 12-2a.

The curvature of the section in Figure 12-2b, κ_b, is given by Equation 12-5.

$$\kappa_b = \frac{\varepsilon_s + \varepsilon + \varepsilon_c - \varepsilon}{d'} = \frac{\varepsilon_s + \varepsilon_c}{d'} = \kappa_a \qquad \text{Eqn. 12-5}$$

In Figure 12-2b if you are given the curvature (slope of the strain diagram) as $\kappa_b = \kappa_a$, and you are given the axial load, then you can determine the location of the neutral axis such that the sum of the forces in the force diagram is equal to the specified axial load. This requires an iterative (trial and error) process where you assume a neutral axis location and check to see if the sum of the forces is equal to the axial load. Once the neutral axis is located you can calculate the associated moment.

The previous paragraph describes the basic process used by Section Designer to calculate a point on the moment curvature diagram. The axial load is given by the user. Section Designer selects a curvature value and for that curvature value it iteratively determines the neutral axis location. Then it calculates the associated moment. The curvature value and the moment value make up a point on the moment curvature curve.

Procedure to Create Moment Curvature Curve

Overview

The following five items are given when Section Designer begins to create the moment curvature curve.

1. The section geometry and properties.

2. An axial load to consider. The default value for this is zero.

3. The orientation (angle) of the neutral axis with respect to the section negative local 3-axis. Note that although the orientation of the neutral axis is given, its exact location is an unknown that must be determined by the program. The default value for this angle is 0 degrees.

4. The maximum number of points desired on the moment curvature curve. The default value for this item is 11.

5. The maximum curvature to consider. See the subsection below titled "Initial Default Value for the Maximum Curvature" for additional information.

Items 2 through 5 above are defined in the Moment Curvature Curve dialog box.

Based on the given items listed above, Section Designer calculates the following three items:

1. The specific curvature values to be considered at each of the points identified in item 4 above. See the subsection below titled "Specific Curvature Values Considered" for more information.

2. The exact location of the neutral axis at each of the curvature values considered. See the previous section titled "Background Information" for more information. Also see the subsection titled "Number of Points on the Moment Curvature Curve" later in this chapter.

3. The moment associated with each of the curvature values considered. See the previous section titled "Background Information" for more information. Also see the subsection titled "Number of Points on the Moment Curvature Curve" later in this chapter.

Initial Default Value for the Maximum Curvature

This subsection describes how the initial default value for the maximum curvature is determined by Section Designer when you first open the Moment Curvature Curve dialog box. Note that this default value is only calculated once when the dialog box is opened. The default value is not updated as you make changes in the dialog box.

To calculate the initial default value for the maximum curvature the following items are given:

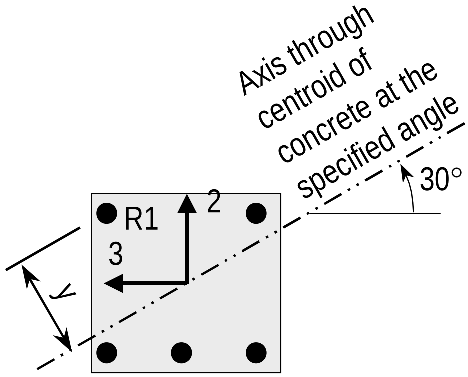
1. The section geometry and properties.

2. The orientation (angle) of the neutral axis with respect to the section negative local 3-axis. In this example it is specified as 30 degrees.

Section Designer uses the following process to determine the initial default value for the maximum curvature:

To determine the value to use for the initial default value for the maximum curvature Section Designer considers each steel element in the section separately and calculates a very approximate

Figure 12-3:
Illustration of how the initial default maximum curvature is calculated



value of the curvature required to fracture the bar using Equation 12-6. *Note that Equation 12-6 is a gross approximation of the curvature required to fracture the bar!*

$$\kappa_{fracture} = \frac{\varepsilon_{s\ fracture}}{y} \qquad \text{Eqn. 12-6}$$

where

y = The perpendicular distance from the center of the steel element to an axis through the centroid of the concrete in the direction specified. Figure 12-3 illustrates the dimension y for the rebar labeled R1.

$\varepsilon_{fracture}$ = The fracture strain associated with the steel element considered.

$\kappa_{fracture}$ = The fracture curvature associated with the steel element considered.

Section Designer checks Equation 12-6 for each steel element in the section and then uses the smallest $\kappa_{fracture}$ value calculated for any steel element in the section as the initial default value of κ_{max}. Again, the intent of this process is simply to get a ballpark estimate of the curvature at first rebar fracture based on the assumption that first rebar fracture is where you will want to stop the moment curvature curve.

Note that the example in this section refers to rebar. If the concrete section is reinforced with a steel shape instead of rebar (e.g., reinforced with a steel I-shaped section) then Section Designer automatically and internally meshes the steel shape into a series of small steel elements and treats each discrete steel element similar to a rebar.

Number of Points on the Moment Curvature Curve

Note:

The moment curvature curve is created by computing exact values at various points along the moment curvature curve and then connecting those points with straight line segments.

The moment curvature curve is created by computing exact values at various points along the moment curvature curve and then connecting those points with straight line segments. The maximum number of points considered, n, is input in the Moment Curvature Curve dialog box. Each of the points is located at a predetermined curvature value. The following section describes how those curvature values are determined.

Section Designer cuts off the moment curvature curve at the first (smallest) curvature value where one of the following two things happens.

1. The curvature considered has reached the specified maximum curvature, κ_{max}.

2. The section is *not* capable of carrying the specified axial load at some curvature less than the specified maximum curvature.

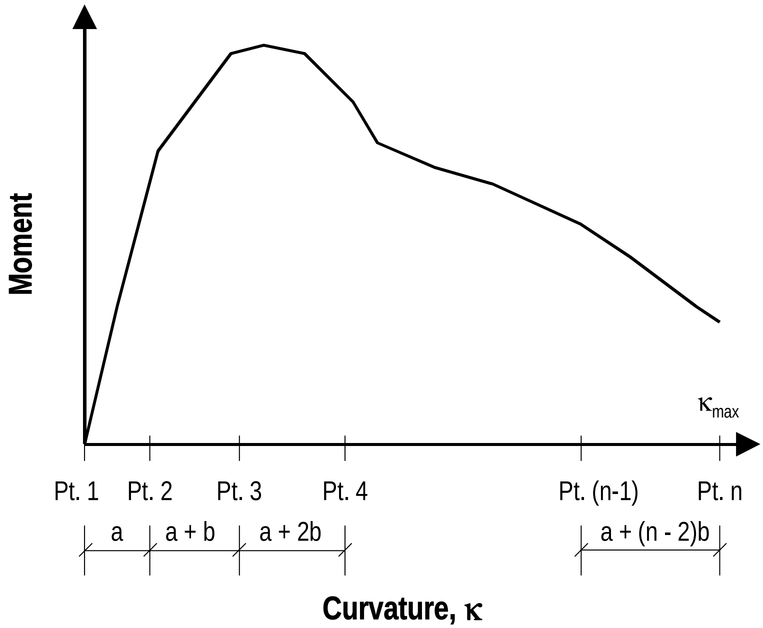
If the moment curvature curve is cut off because it reached the specified κ_{max} value then the curve will have the specified maximum number of points. If the moment curvature curve is cut off because it is not capable of carrying the axial load then it will have less than the specified number of points.

Specific Curvature Values Considered

Figure 12-4 shows the arithmetic progression of spacing assumed between the n specified points that occur between the curvature values of 0 and κ_{max}. The number of points, n, and the maximum curvature considered, κ_{max}, are both specified in the Moment Curvature Curve dialog box. Note that there are n-1 spaces between the n points.

12

Figure 12-4:
Arithmetic progression used by Section Designer to locate the specified points on the moment curvature curve



The required distance a in Figure 12-4 can be determined using Equation 12-7.

$$a = \frac{\kappa_{max}}{(n-1)\left[1 + \frac{n-2}{2}\left(\frac{b}{a}\right)\right]} \qquad \text{Eqn. 12-7}$$

Note that you need the ratio b/a in Equation 12-7. Section Designer assumes that this ratio is equal to 0.5.

Again note that the output for the moment curvature curve may have less than the specified number of points because the section is *not* capable of carrying the specified axial load at some curvature less than the specified maximum curvature. In this case the spacing of the points is still based on the specified κ_{max} value. The spacing is not based on the curvature that the section reached before it could no longer carry the axial load.

Chapter 13

13

Frame Section Example

This chapter presents a Section Designer example for a concrete frame section. Although there are many ways to create the example section, only one method is illustrated in this chapter.

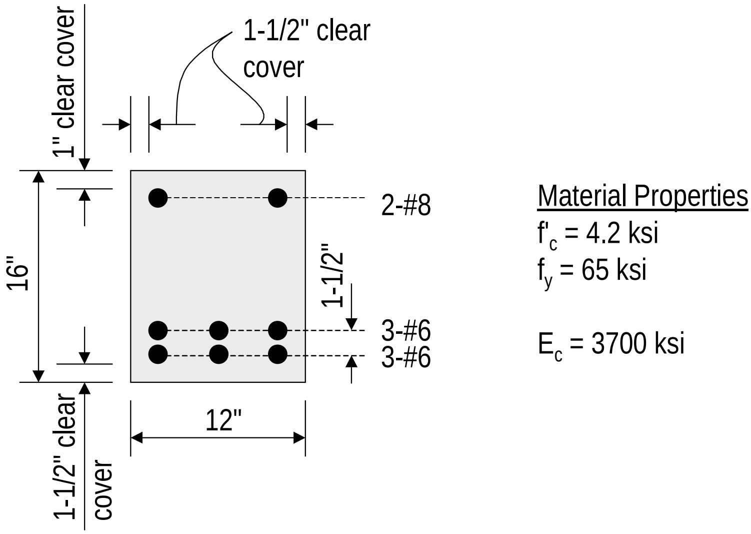
Problem Statement

The frame section is the concrete beam shown in Figure 13-1. The beam is 12" wide and 16" deep. There are two rows of three #6 bars (six bars total) at the bottom of the beam and 2 #8 bars at the top of the beam.

The clear cover is 1.5 inches on the sides and bottom of the beam and 1" on the top of the beam.

Relevant material properties assumed for the beam are f'_c = 4.2 ksi, f_y = 65 ksi and E_c = 3700 ksi. Also for this example assume that the building code specified for Concrete Frame Design is the ACI 318-99 for which Section Designer assumes that E_s = 29000 ksi.

Figure 13-1:
Frame section example problem



13

The remainder of this chapter steps through the process of defining this section.

Start ETABS and Set Preferences

Use the following process to start ETABS and set the design code preference for this example.

1. Start the ETABS program by clicking on the appropriate desktop shortcut or by selecting ETABS from your Windows Start menu.

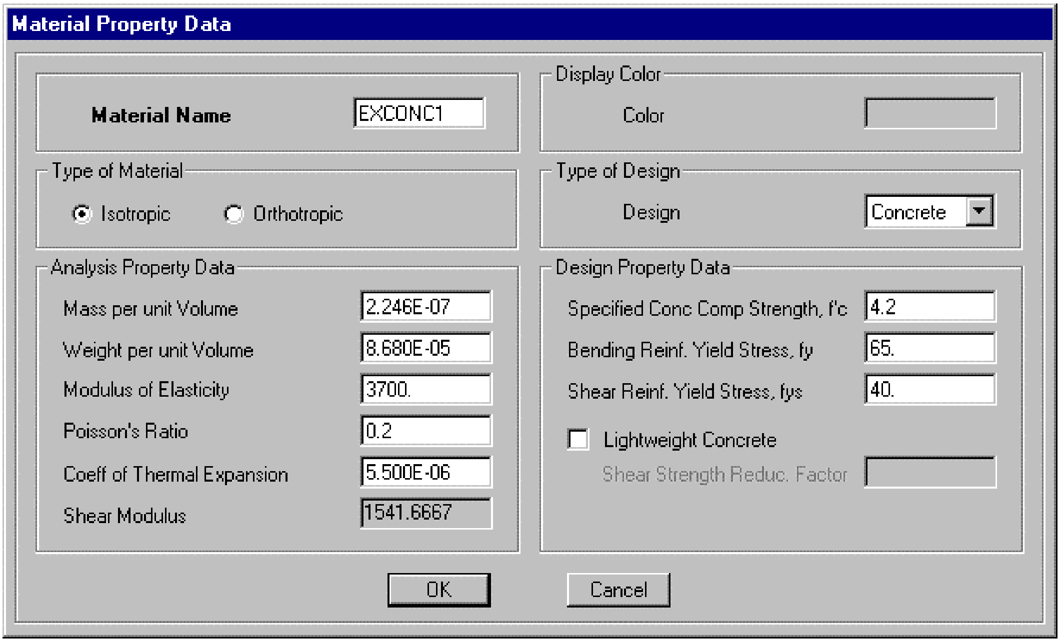
2. Set the units to kips and inches.

3. Click the **File menu > New Model** command.

4. Click the **No** button in the New Model Initialization form.

5. Click the **OK** button in the Building Plan Grid System and Story Data Definition dialog box.

6. Click the **Options menu > Preferences > Concrete Frame Design** command and confirm that the Design Code item is set to ACI 318-99.



Define Material Properties

(Above)
Figure 13-2:
Material properties defined for frame section example problem

The material properties used in Section Designer are defined in ETABS. Thus we will define those properties before entering Section Designer.

1. Click the **Define menu** > **Material Properties** command to bring up the Define Materials dialog box.

2. Click the **Add New Material** button to bring up the Material Property Data dialog box. In this dialog box:

   a. Set the Material Name to EXCONC1.

   b. Leave the Type of Material as Isotropic.

   c. Fill in the Analysis Property Data as shown in the dialog box in Figure 13-2. You can not edit the Shear Modulus item. It will change automatically when you after you click the **OK** button to close the dialog box.

   d. Change the Type of Design to Concrete.

e. Fill in the Design Property Data as shown in the dialog box in Figure 13-2.

f. Click the **OK** button twice to exit all dialog boxes.

Start Section Designer

To define frame sections start Section Designer from the Define menu.

1. Click the **Define menu > Frame Sections** command to bring up the Define Frame Properties dialog box.

2. Click on the drop-down box that says Add I/Wide Flange, scroll down to the bottom of the list and click the Add SD Section item in the drop-down box. The SD Section Data dialog box appears.

3. Do the following in the SD Section Data dialog box:

   a. Set the Section Name to BEAM1.

   b. Set the Base Material to EXCONC1.

   c. In the Design Type area leave the No Check/Design option selected.

   Note: If this were a column section, rather than a beam section, and we wanted the ETABS Concrete Frame Design postprocessor to check the reinforcing in the column then we would select the Concrete Column option here.

   d. Click the **Section Designer** button to start Section Designer.

Draw Section Geometry

This section describes two different methods to create the initial rectangular shape. Note that the dimensions of the rectangle are initially default dimensions that we will modify to the desired dimensions in the next section.

Method 1: Click the **Draw menu > Draw Solid Shape > Rectangle** command. Left click once anywhere on the Section Designer grid to draw the initial shape. We will resize and relocate it in the next section.

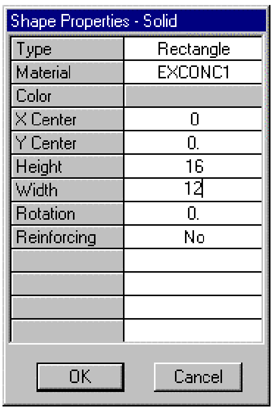
Method 2: Click the **Draw Solid Shape** button, , on the side tool bar and then click the Rectangle flyout button, . Left click once anywhere on the Section Designer grid to draw the initial shape. We will resize and relocate it in the next section.

Note that when you click to draw the rectangle the center of the rectangle is located at the point that you click on.

Define Section Location and Dimensions

In this section we will define the beam section location (in Section Designer) and dimensions. We will center the beam about the Section Designer origin. Note that the beam can be located anywhere in the Section Designer XY coordinate system. The location of the beam in the Section Designer XY coordinate system has no effect on the section properties. The relocation of the section is done here for the purpose of demonstrating how to do it.

4. Click the **Draw menu > Select Mode** command or click the **Select Pointer** button, , on the side toolbar to exit draw mode and enter select mode.



5. Right click on the section to bring up the Shape Properties dialog box. Fill in this dialog box as shown in the sketch to the left and described below.

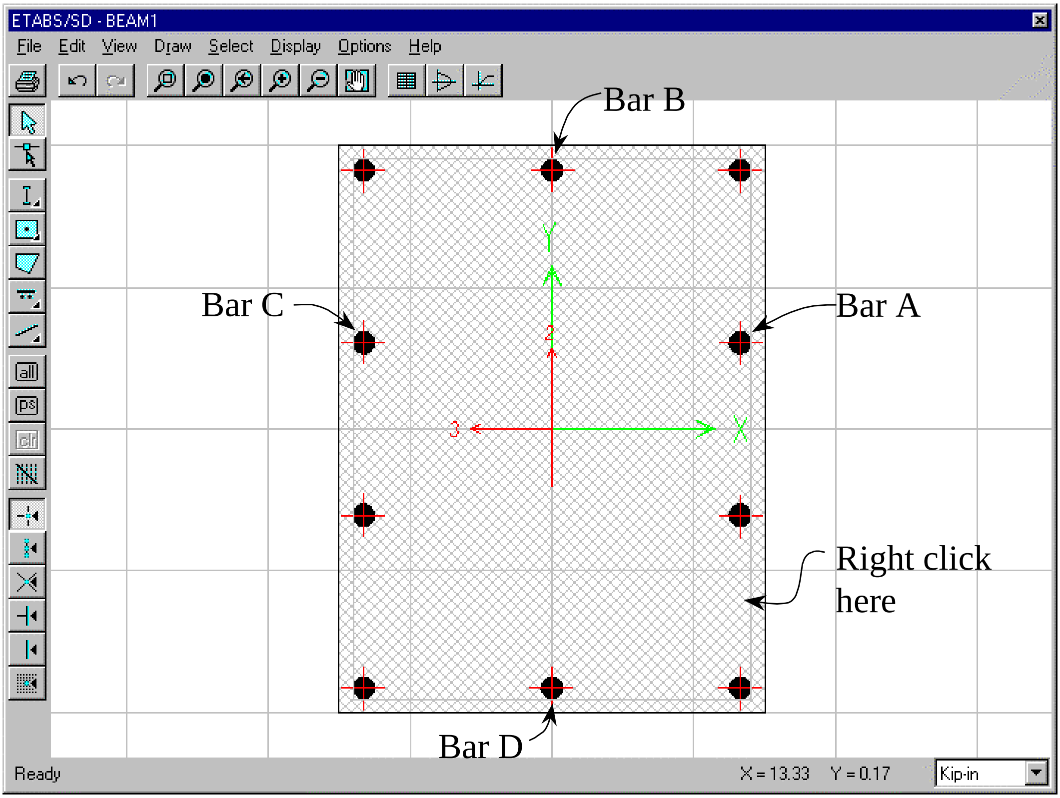
   a. Set the Material to EXCONC1.

   b. Set both the X Center and Y Center to 0.

   c. Set the Height to 16 and the Width to 12.

   d. Leave the Rotation at 0.

   e. Set the Reinforcing to Yes.



(Above)
Figure 13-3:
Beam after step 3 in the Define Section Location and Dimensions section

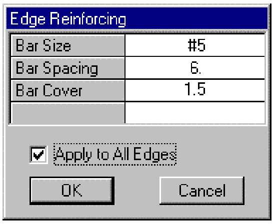
f. Click the **OK** button. The beam should now be centered about the Section Designer X and Y axes, it should have the correct dimensions (12" wide by 16" deep) and there should be default rebar in the section. We will modify this rebar in the next few sections.

6. Click the **View menu > Restore Full View** command, or the associated **Restore Full View** button, , on the top toolbar to resize the section to fill the full Section Designer window. The beam section should now appear as shown in Figure 13-3.

Define Section Edge Bars

For edge bars you can specify a bar size, spacing and cover. The corner bars are located based on the specified cover for edge bars on edges adjacent to the corner.

1. Right click anywhere around the edge of the beam along a line that connects the center of each rebar, except do not right click on a corner bar. For example click where it says "Right click here" or where it says "Bar A" in Figure 13-3. The Edge Reinforcing dialog box appears. In this dialog box:



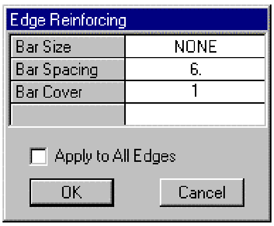
   a. Set the Bar Cover to 1.5 inches.

   b. Check the Apply to All Edges check boxes. We will adjust the cover for the top edge later.

   c. At this particular point the Bar Size and Bar Spacing are irrelevant so we will not change them.

   d. Click the **OK** button.

2. Right click on the bar labeled "Bar B" in Figure 13-3 to bring up the Edge Reinforcing dialog box. In this dialog box:



   a. Set the Bar Size to NONE.

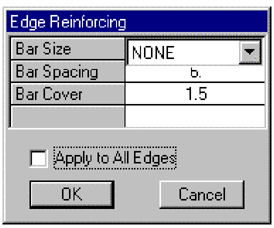
   b. In this instance the Bar Spacing is irrelevant so do not change it.

   c. Set the Bar Cover to 1 inch.

   d. Do *not* check the Apply to All Edges check box since the changes we are specifying here only apply to the top edge.

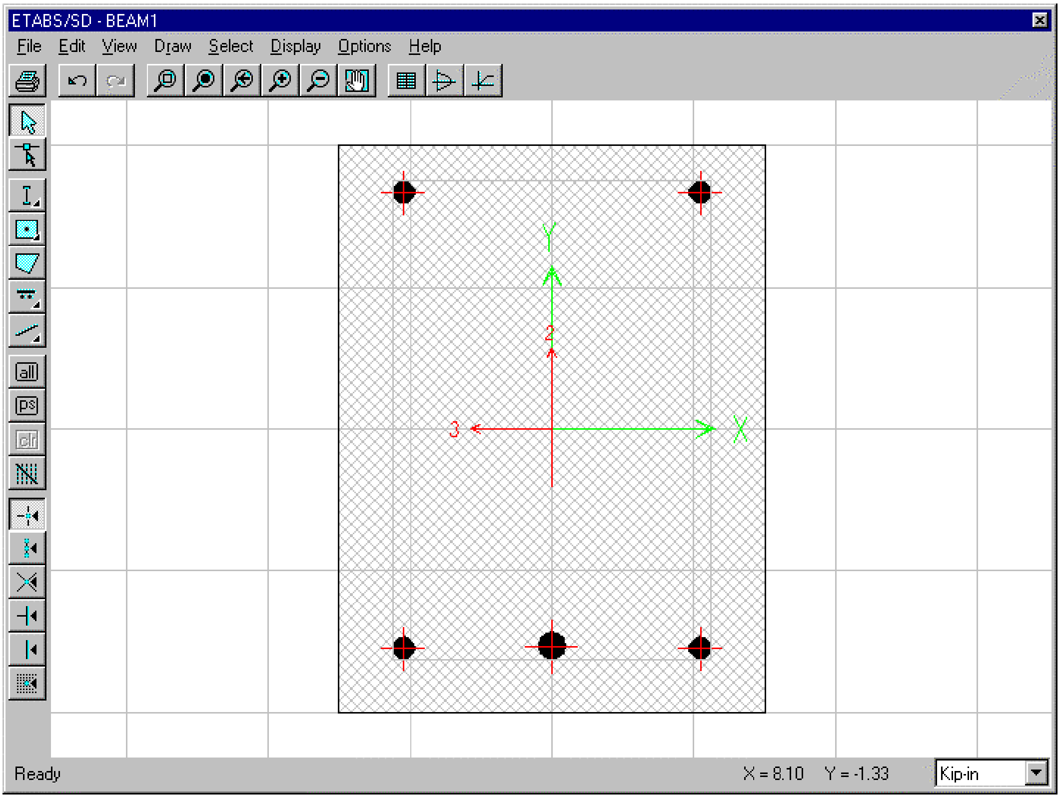
   e. Click the **OK** button.

3. Right click on the bar labeled "Bar A" in Figure 13-3 to bring up the Edge Reinforcing dialog box. In this dialog box:



   a. Set the Bar Size to NONE.

   b. Click the **OK** button.



(Above)
Figure 13-4:
Beam after step 5 in the Define Section Edge Bars section

4. Repeat step 3 for the bar labeled "Bar C" in Figure 13-3.

5. Right click on the bar labeled "Bar D" in Figure 13-3 to bring up the Edge Reinforcing dialog box. In this dialog box:

   a. Set the Bar Size to #6.

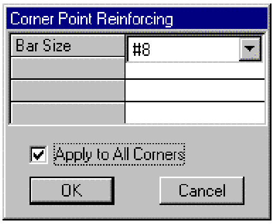
   b. Set the Bar Spacing to 4.2 inches.

   c. Click the **OK** button.

The section now appears as shown in Figure 13-4.

Define Section Corner Bars

For corner bars you can specify a bar size. The corner bars are located based on the specified cover for edge bars on edges adjacent to the corner.



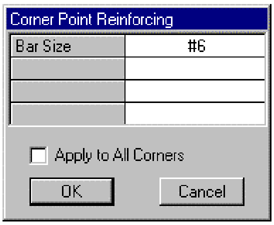
1. Right click on one of the corner bars to bring up the Corner Point Reinforcing dialog box. In this dialog box:

    a. Set the Bar Size to #8.

    b. Check the Apply to All Corners check box. We will fix the bottom corner bar size later.

    c. Click the **OK** button.

2. Right click on the bottom left corner bar to bring up the Corner Point Reinforcing dialog box. In this dialog box:



    a. Set the Bar Size to #6.

    b. Do *not* check the Apply to All Corners check box. We will fix the bottom corner bar size later.

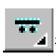
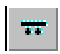
    c. Click the **OK** button.

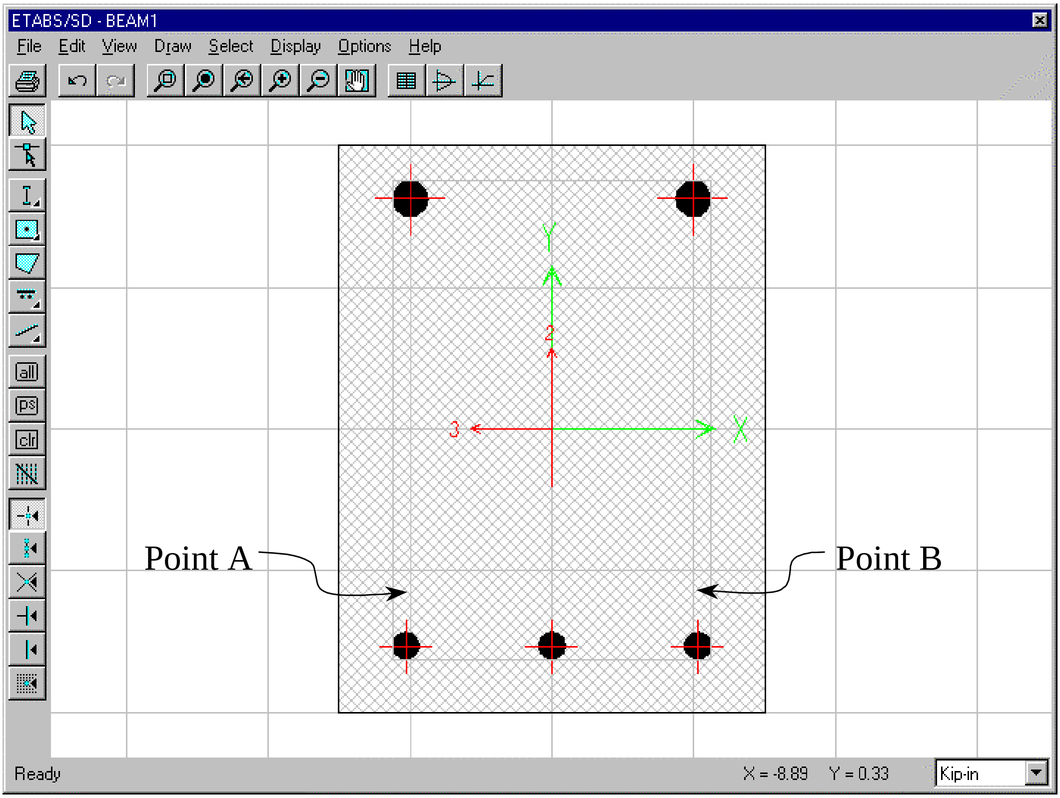
3. Repeat step 2 for the bottom right corner bar.

The section now appears as shown in Figure 13-5.

Draw Line Pattern Reinforcing

We will use additional line pattern reinforcing to draw the second layer of rebar at the bottom of the beam.

1. Click the **Draw menu > Draw Reinforcing Shape > Line Pattern** command or click the associated **Draw Reinforcing Shape** button, , on the side toolbar and then click the Line Pattern flyout button, .



(Above)
Figure 13-5:
Beam after step 3 in the Define Section Corner Bars section

2. Left click once near the point labeled "Point A" in Figure 13.5 to start the line pattern reinforcing. Left click again near the point labeled "Point B" in Figure 13.5 to complete the line pattern reinforcing. We will locate the rebar exactly in the next section.

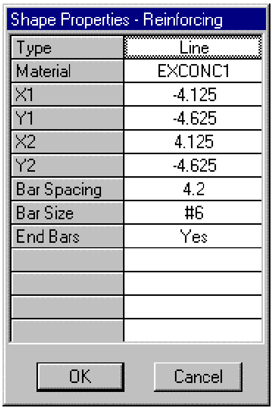
Define Line Pattern Reinforcing Properties

In this section we locate the line pattern reinforcing exactly and define its properties.

1. Click the **Draw menu** > **Select Mode** command or click the **Select Pointer** button, , on the side toolbar to exit draw mode and enter select mode.

2. Right click on the just drawn line pattern reinforcing to bring up the Shape Properties dialog box for the line reinforcing. The exact title of the dialog box is Shape Properties - Reinforcing and the Type item in the dialog box is Line. If this is not the dialog box that appeared you may have not clicked directly on the line pattern reinforcing and instead brought up the Shape Properties dialog box for the solid rectangular section.

3. In the Shape Properties - Reinforcing dialog box:



   a. Set the Material to EXCONC1.

   b. Set the X1 dimension to -4.125 inches.

   c. Set the Y1 dimension to -4.625 inches.

   d. Set the X2 dimension to 4.125 inches.

   e. Set the Y2 dimension to -4.625 inches.

   f. Set the Bar Spacing to 4.2 inches.

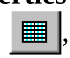
   g. Set the Bar Size to #6.

   h. Set the End Bars item to Yes.

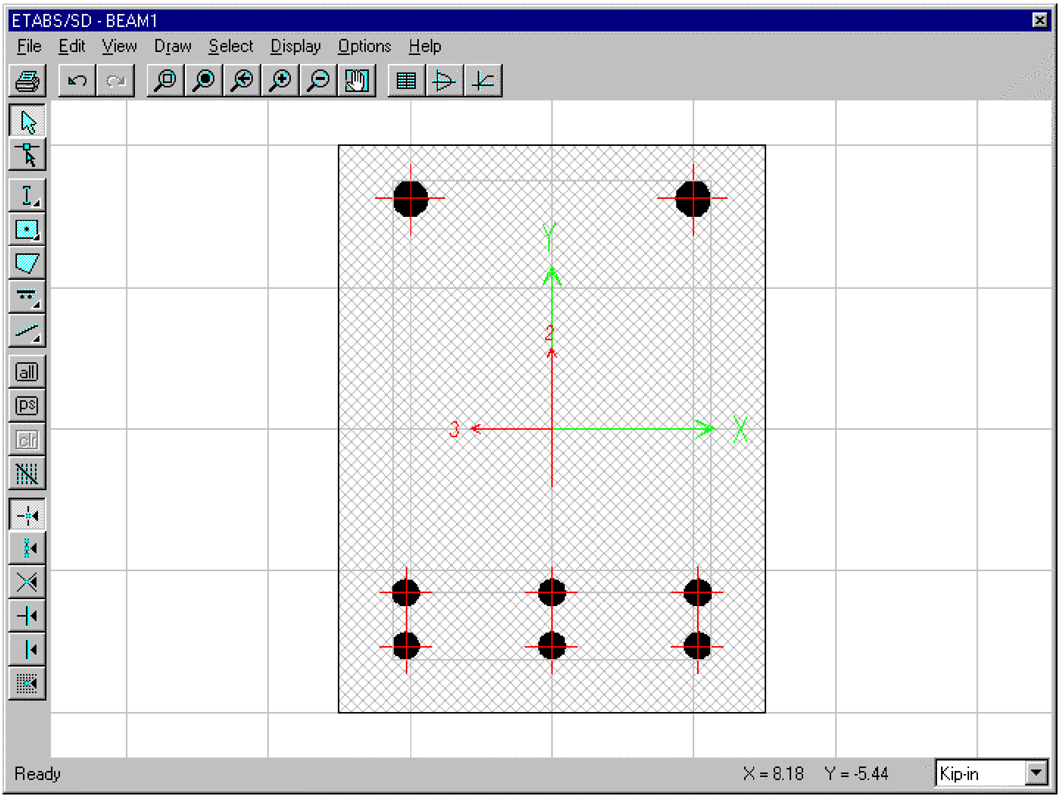
   i. Click the **OK** button.

The section is now complete and it appears as shown in Figure 13-6.

Display Section Properties

Now that the section is complete you can display the section properties if desired. To do this click the **Display menu > Show Section Properties** command or click the **Show Section Properties** button, , on the top toolbar. The Properties form is displayed. See the section titled "Show Section Properties" in Chapter 9 for information on this form.

All of the properties shown except for I_{23}, X_{cg} and Y_{cg} are used by ETABS for analysis and design when you use this section in a model.

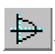


(Above)
Figure 13-6:
Completed section

The properties obtained for this example section are displayed in Figure 13-7. Note that the section properties are based on the gross concrete section. The reinforcing is not considered when computing these section properties. The reinforcing *is* considered when creating the interaction surface and the moment curvature curve.

Display Interaction Surface

In Section Designer you can display the interaction surface for concrete sections that have reinforcing steel defined. The beam created in this example meets this criteria.

To display the interaction surface click the **Display menu** > **Show Interaction Surface** command or click the **Show Interaction Surface** button, , on the top toolbar. The Interaction Surface dialog box is displayed. See the section titled "Show In-

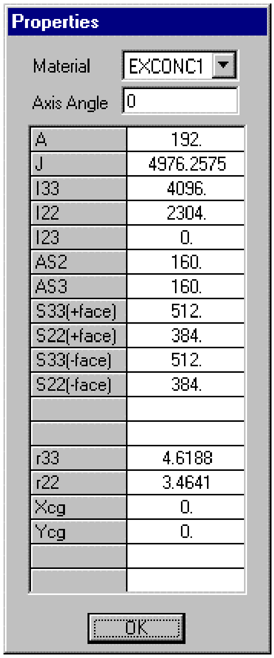


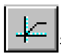
Figure 13-7:
Section properties

13

teraction Surface" in Chapter 9 for information on this dialog box. See Chapter 11 for background information on how the interaction surface is created by Section Designer.

Display Moment Curvature Curve

In Section Designer you can display a moment curvature curve for a concrete section that has reinforcing steel defined. The beam created in this example meets this criteria.

To display the moment curvature curve click the **Display menu > Show Moment Curvature Curve** command or click the **Show Moment Curvature Curve** button, , on the top toolbar. The Moment Curvature Curve dialog box is displayed. See the section titled "Show Moment Curvature Curve" in Chapter 9 for information on this dialog box. See Chapter 12 for background information on how the moment curvature curve is created by Section Designer.

Close Section Designer

To close Section Designer do the following:

1. Click the **File menu > Return to ETABS** command. This closes the Section Designer window and returns you to the SD Section Data dialog box.

2. Click the **OK** button to close the SD Section Data dialog box.

   Important note: If you instead click the **Cancel** button to close the SD Section Data dialog box then the new section just defined in Section Designer is lost. Similarly, if you were modifying an existing section in Section Designer rather than creating a new one, then the changes to the section are lost if you click the **Cancel** button.

3. At this point you can define another frame section, or if you are through defining frame sections you can click the **OK** button to close the Define Frame Properties dialog box.

   Important note: If you instead click the **Cancel** button to close the Define Frame Properties dialog box then the section just defined in Section Designer and any other sections just defined are lost. Similarly, if you were modifying an existing section in Section Designer or any other existing section, then the changes to the section are lost if you click the **Cancel** button.

   In summary, to successfully close Section Designer and save your changes you should Click the **File menu > Return to ETABS** command in Section Designer and then click the **OK** button twice to close all dialog boxes.

Chapter 14

14

Wall Pier Section Example

This chapter presents a Section Designer example for a concrete wall pier section. Although there are many ways to create the example section, only one method is illustrated in this chapter.

Problem Statement

The wall pier section is a two-dimensional wall pier that is 19'-4" long and 12" wide. It is typically reinforced with #6 bars at 12" on center on each face. There are 2-#8 edge bars at each end of the wall pier.

The clear cover at the ends of the wall pier is 2 inches. The clear cover along one side of the wall pier is 1 inch and it is 3/4" along the other side.

Relevant material properties assumed for the wall are f'_c = 4.2 ksi, f_y = 65 ksi and E_c = 3700 ksi. Also for this example assume that the building code specified for Shear Wall Design is the UBC97 for which Section Designer assumes that E_s = 29000 ksi.

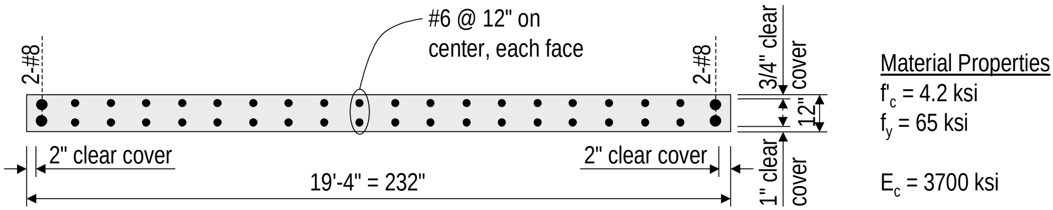


Figure 14-1:
Wall pier section example problem

The remainder of this chapter steps through the process of defining this section.

Start ETABS and Set Preferences

Use the following process to start ETABS and set the design code preference for this example.

1. Start the ETABS program by clicking on the appropriate desktop shortcut or by selecting ETABS from your Windows Start menu.

2. Set the units to kips and inches.

3. Click the **File menu > New Model** command.

4. Click the **No** button in the New Model Initialization form.

5. Click the **OK** button in the Building Plan Grid System and Story Data Definition dialog box.

6. Click the **Options menu > Preferences > Shear Wall Design** command and confirm that the Design Code item is set to UBC97.

Define a Shear Wall and Label Some Wall Piers

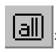
Use the following process to define a shear wall. Note that your model must have shear walls defined and it must have one or more wall pier labels assigned before you can use Section Designer to define wall pier sections. The **Design menu > Shear Wall Design > Define Pier Sections for Checking** command, which is used to start Section Designer is not available until you have assigned at least one wall pier label in your model.

1. Click the **Edit menu > Add to Model from Template > Add 2D Frame** command to bring up the 2D Frame dialog box.

2. In the 2D Frame dialog box click the **2D Wall** button to bring up the Shear Wall dialog box.

3. Click the **OK** button in the Shear Wall dialog box to define a shear wall.

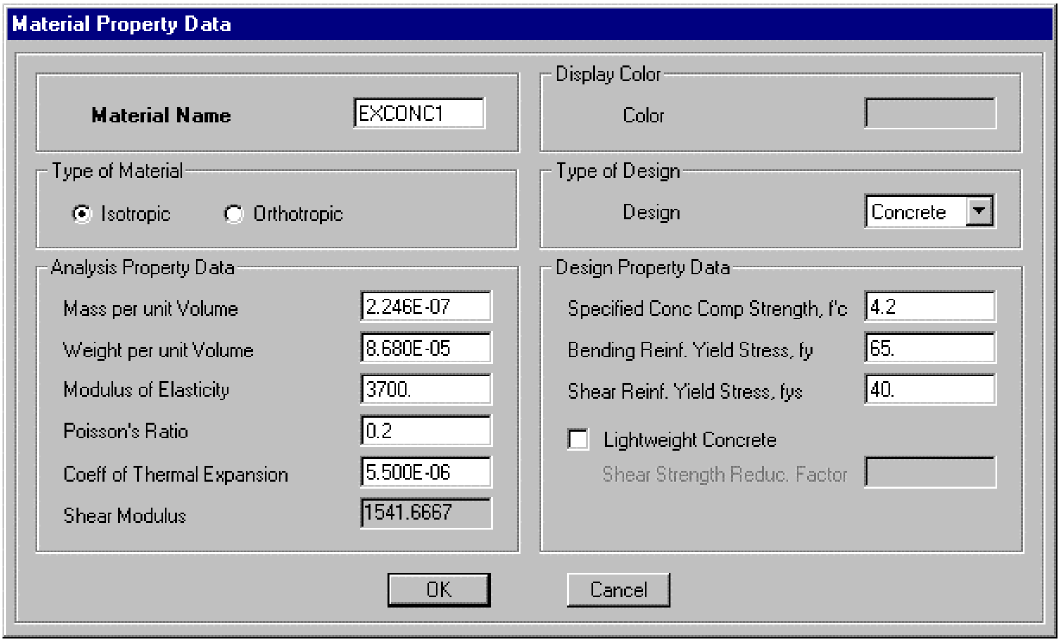
4. Click the **Select** button, , on the side toolbar or click the **Draw menu > Select Object** command to exit draw mode and enter select mode.

5. Click the **Select All** button, , on the side toolbar, or click the **Select menu > Select All** command. This selects all of the wall objects.

6. Click the **Assign menu > Shell/Area > Pier Label** command to open the Pier Names dialog box. In this dialog box:

   a. In the Wall Piers area left click once on the WP1 item to highlight it.

   b. Click the **OK** button to assign the WP1 label to all of the selected wall objects (piers).



Define Material Properties

(Above)
Figure 14-2:
Material properties defined for wall pier section example problem

The material properties used in Section Designer are defined in ETABS. Thus we will define those properties before entering Section Designer.

1. Click the **Define menu > Material Properties** command to bring up the Define Materials dialog box.

2. Click the **Add New Material** button to bring up the Material Property Data dialog box. In this dialog box:

    a. Set the Material Name to EXCONC1.

    b. Leave the Type of Material as Isotropic.

    c. Fill in the Analysis Property Data as shown in the dialog box in Figure 14-2.

    d. Change the Type of Design to Concrete.

e. Fill in the Design Property Data as shown in the dialog box in Figure 14-2.

f. Click the **OK** button twice to exit all dialog boxes.

Start Section Designer

For defining frame sections Section Designer is started from the Design menu.

1. Click the **Design menu > Shear Wall Design > Define Pier Sections for Checking** command to bring up the Pier Sections dialog box.

2. Click the **Add Pier Section** button to bring up the Pier Section Data dialog box.

3. Do the following in the Pier Section Data dialog box:

    a. Set the Section Name to PIER1.

    b. Set the Base Material to EXCONC1.

    c. In the Add Pier area select the Add New Pier Section option.

    d. In the Check/Design area select the Reinforcement to be Checked option.

    e. Click the **Section Designer** button to start Section Designer.

Draw Section Geometry

This section describes two different methods to create the initial rectangular shape. Note that the dimensions of the rectangle are initially default dimensions that we will modify to the desired dimensions in the next section.

Method 1: Click the **Draw menu > Draw Solid Shape > Rectangle** command. Left click once anywhere on the Section Designer grid to draw the initial shape. We will resize and relocate it in the next section.

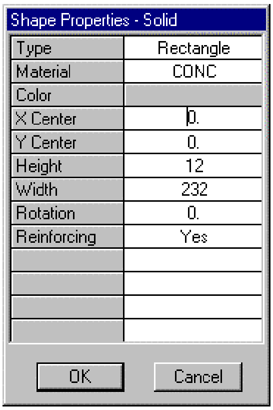
Method 2: Click the **Draw Solid Shape** button, , on the side tool bar and then click the Rectangle flyout button, . Left click once anywhere on the Section Designer grid to draw the initial shape. We will resize and relocate it in the next section.

Note that when you click to draw the rectangle the center of the rectangle is located at the point that you click on.

Define Section Location and Dimensions

In this section we will define the wall pier section location (in Section Designer) and dimensions. We will center the wall pier about the Section Designer origin. Note that the pier can be located anywhere in the Section Designer XY coordinate system. The location of the pier in the Section Designer XY coordinate system has no effect on the section properties. The relocation of the section is done here for the purpose of demonstrating how to do it.

1. Click the **Draw menu > Select Mode** command or click the **Select Pointer** button, , on the side toolbar to exit draw mode and enter select mode.



2. Right click on the section to bring up the Shape Properties dialog box. Fill in this dialog box as shown in the sketch to the left and described below.

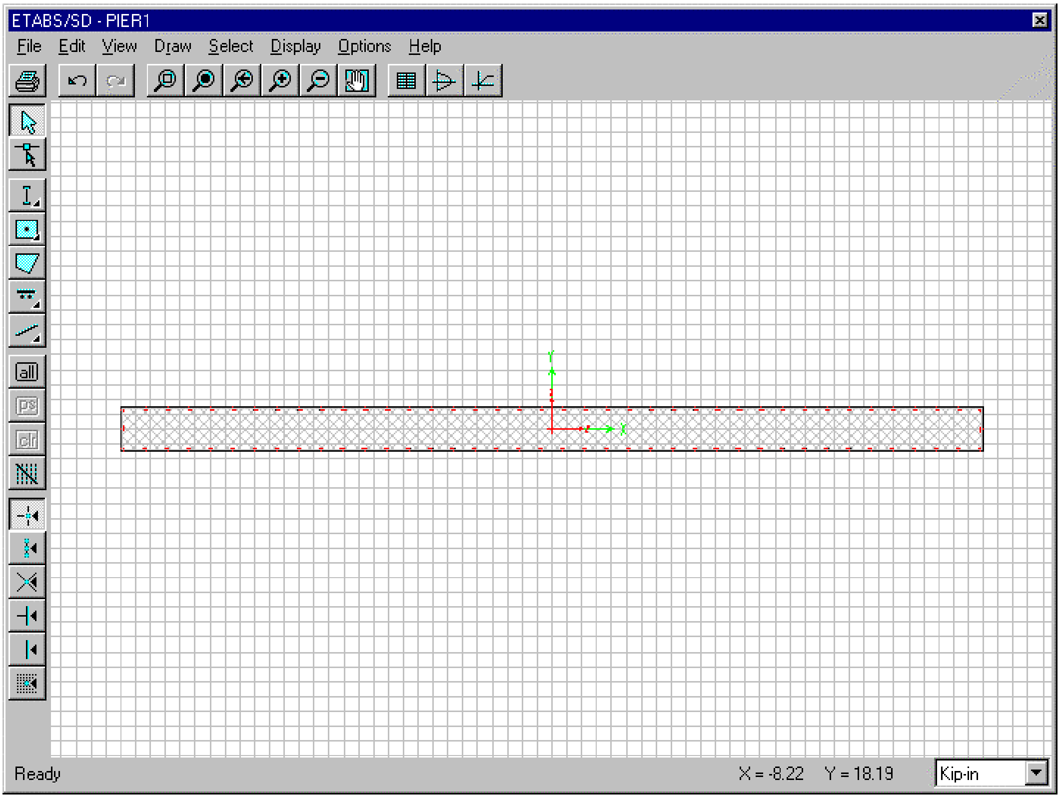
   a. Set the Material to EXCONC1.

   b. Set both the X Center and Y Center to 0.

   c. Set the Height to 12 and the Width to 232.

   d. Leave the Rotation at 0.

   e. Set the Reinforcing to Yes.



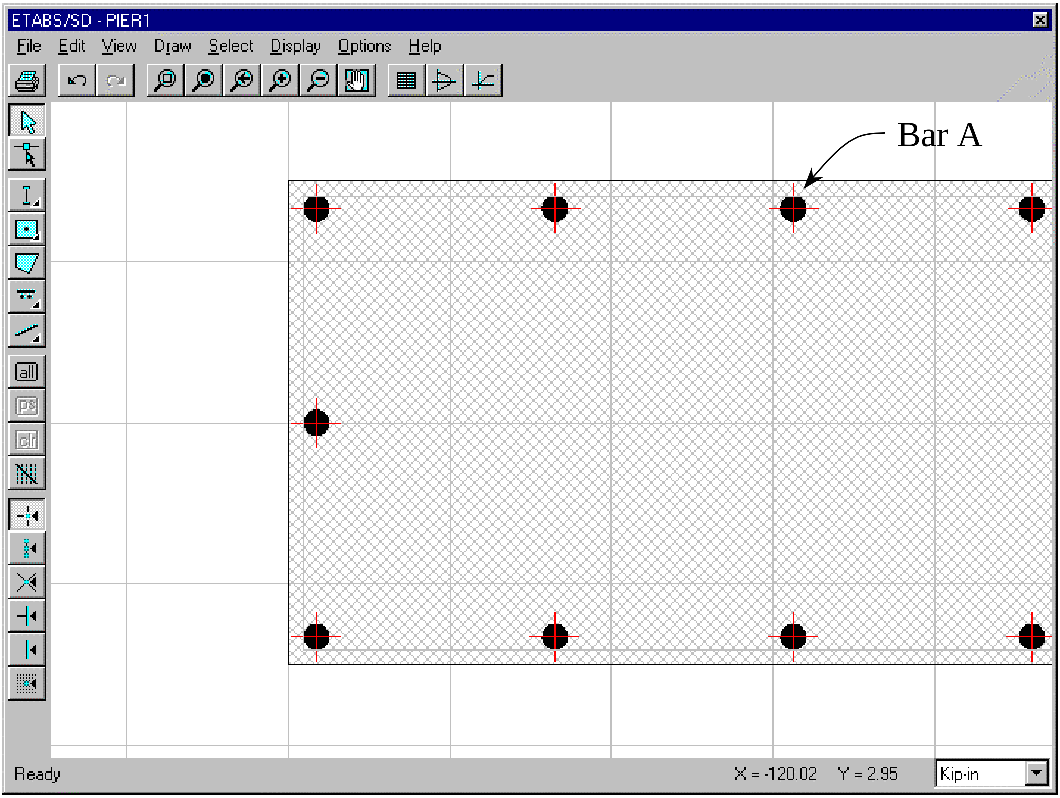
(Above)
Figure 14-3:
Wall pier after step 3 in the Define Section Location and Dimensions section

f. Click the **OK** button. The wall pier should now be centered about the Section Designer X and Y axes, it should have the correct dimensions (232" long by 12" thick) and there should be default rebar in the section. We will modify this rebar in the next few sections.

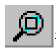
3. Click the **View menu > Restore Full View** command, or the associated **Restore Full View** button, , on the top toolbar to resize the section to fill the full Section Designer window. The wall pier section should now appear as shown in Figure 14-3.

Define Section Edge Bars

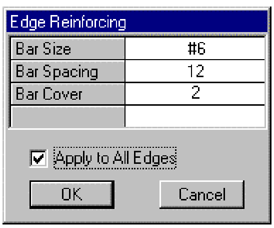
For edge bars you can specify a bar size, spacing and cover. The corner bars are located based on the specified cover for edge bars on edges adjacent to the corner.



(Above)
Figure 14-4:
Wall pier after step 1 in the Define Section Edge Bars section

1. Click the **View menu** > **Rubberband Zoom** command or click the **Rubberband Zoom** button, , on the top toolbar and zoom in on the left end of the wall as shown in Figure 14-4.

2. Right click on the bar labeled "Bar A" in Figure 14-4. The Edge Reinforcing dialog box appears. In this dialog box:

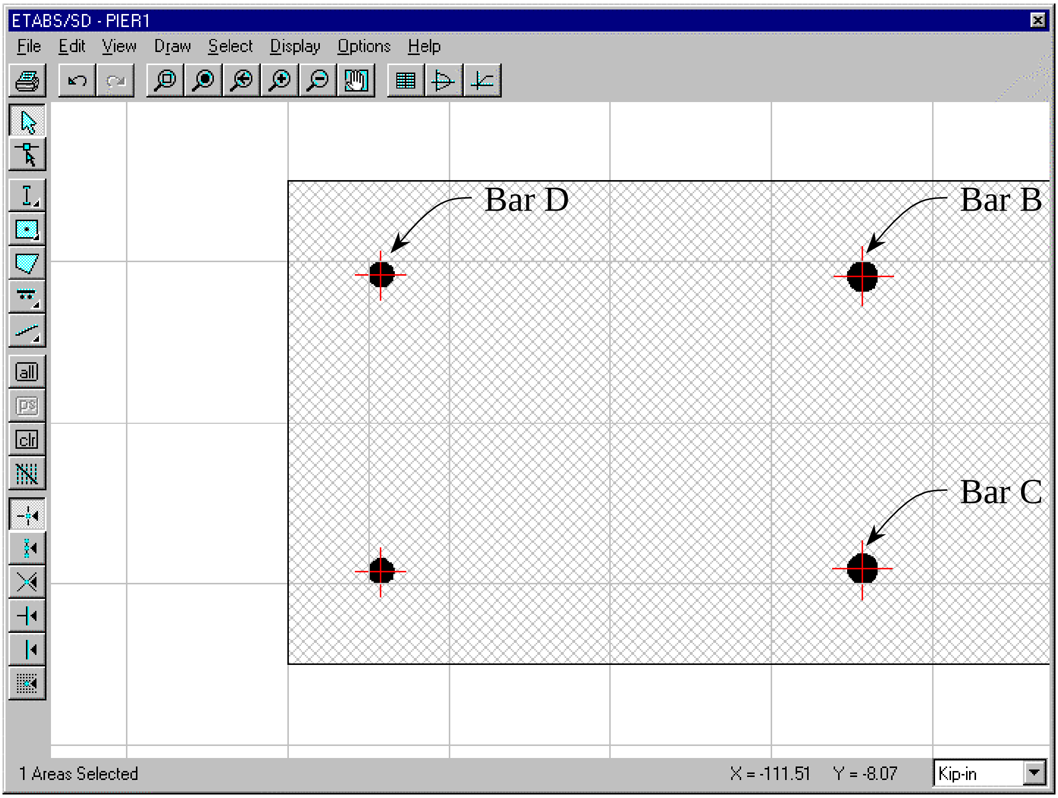


   a. Set the Bar Size to #6.

   b. Set the Bar Spacing to 12 inches

   c. Set the Bar Cover to 2 inches.

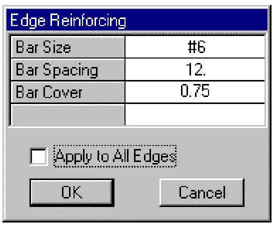
   d. Check the Apply to All Edges check boxes. We will adjust the cover for the top and bottom edges later.



(Above)
Figure 14-5:
Beam after step 2e in the Define Section Edge Bars section

e. Click the **OK** button. The section now appears as shown in Figure 14-5.

3. Right click on the bar labeled "Bar B" in Figure 14-5 to bring up the Edge Reinforcing dialog box. In this dialog box:



a. Set the Bar Cover to 0.75 inches.

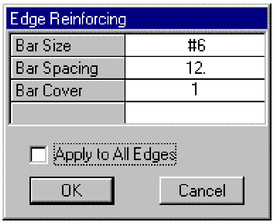
b. Leave the bar Spacing at 12 inches.

c. Leave the bar size at #6.

d. Do *not* check the Apply to All Edges check box since the changes we are specifying here only apply to the top edge.

e. Click the **OK** button.

4. Right click on the bar labeled "Bar C" in Figure 14-5 to bring up the Edge Reinforcing dialog box. In this dialog box:



   a. Set the Bar Cover to 1 inch.

   b. Leave the bar Spacing at 12 inches.

   c. Leave the bar size at #6.

   d. Do *not* check the Apply to All Edges check box since the changes we are specifying here only apply to the top edge.

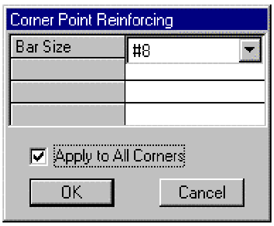
   e. Click the **OK** button.

14

Define Section Corner Bars

For corner bars you can specify a bar size. The corner bars are located based on the specified cover for edge bars on edges adjacent to the corner.

1. Right click on the corner bar labeled "Bar D" in Figure 14-5 to bring up the Corner Point Reinforcing dialog box. In this dialog box:

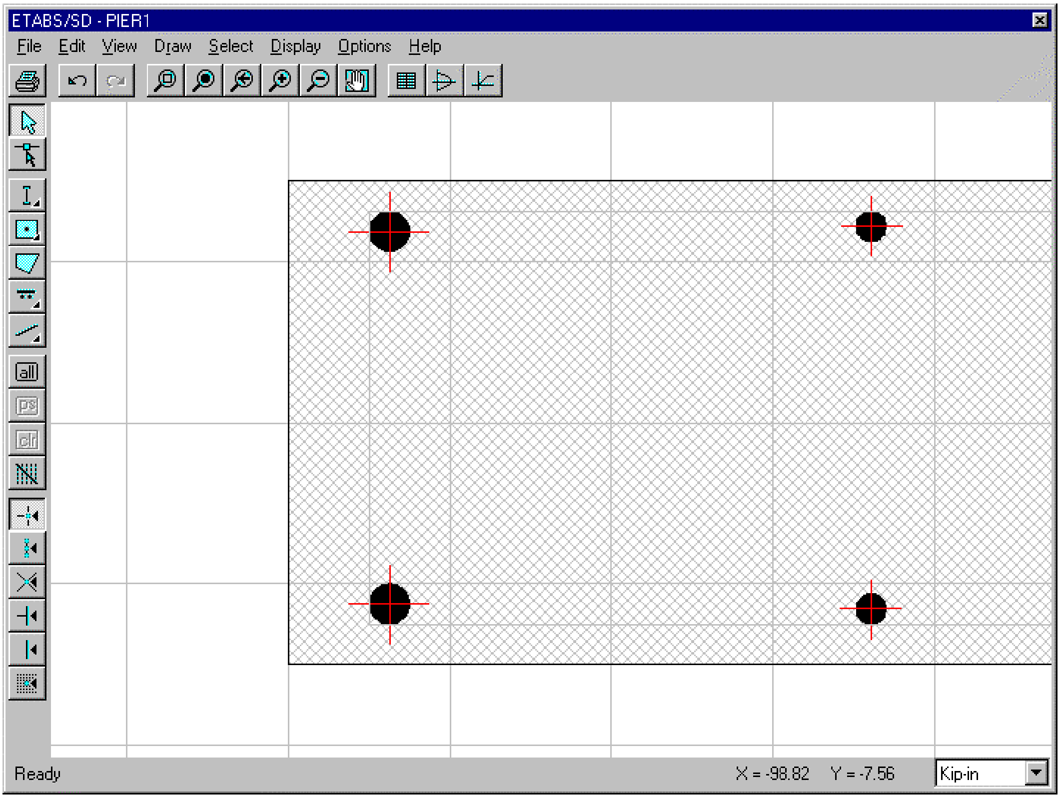


   a. Set the Bar Size to #8.

   b. Check the Apply to All Corners check box.

   c. Click the **OK** button.

The section is now complete and it appears as shown in Figure 14-6. Click the **View menu** > **Restore Full View** command, or the associated **Restore Full View** button, , on the top toolbar to resize the section to fill the full Section Designer window. The wall pier section should now appear as shown in Figure 14-7.

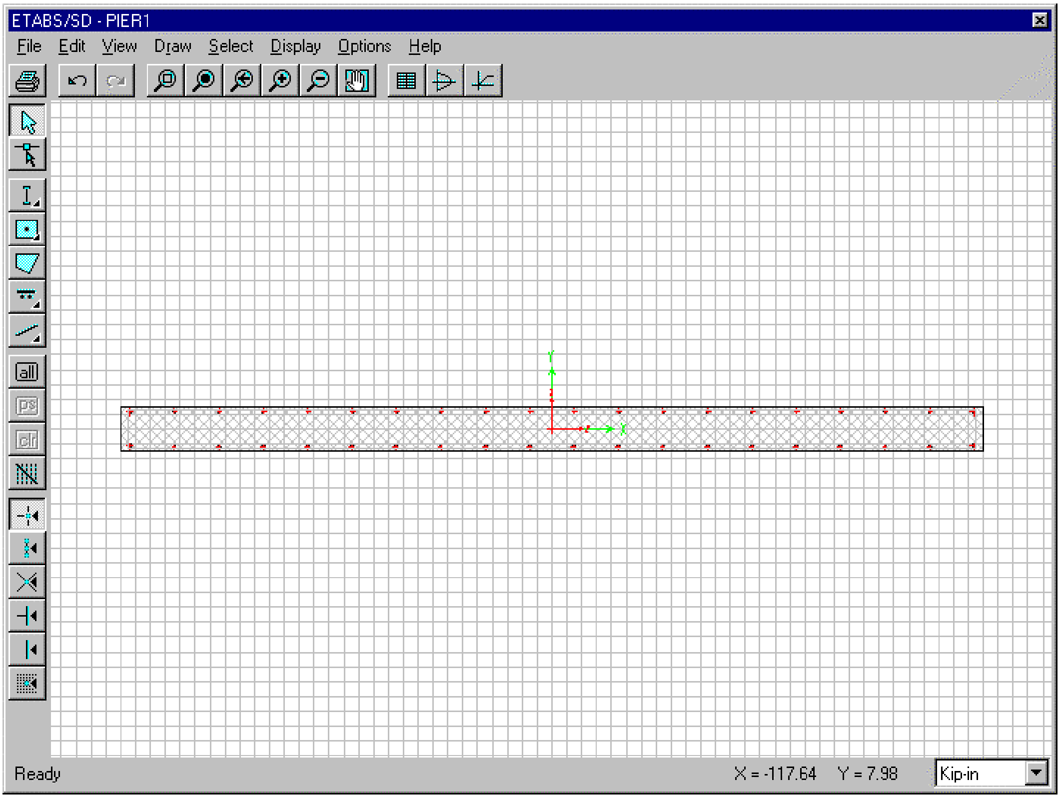


Display Section Properties

(Above) Figure 14-6:
Beam after step 1c in the Define Section Corner Bars section

Now that the section is complete you can display the section properties if desired. To do this click the **Display menu > Show Section Properties** command or click the **Show Section Properties** button, , on the top toolbar. The Properties form is displayed. See the section titled "Show Section Properties" in Chapter 9 for information on this form.

The properties obtained for this example section are displayed in Figure 14-8. Note that the section properties are based on the gross concrete section. The reinforcing is not considered when computing these section properties. The reinforcing *is* considered when creating the interaction surface and the moment curvature curve.



14

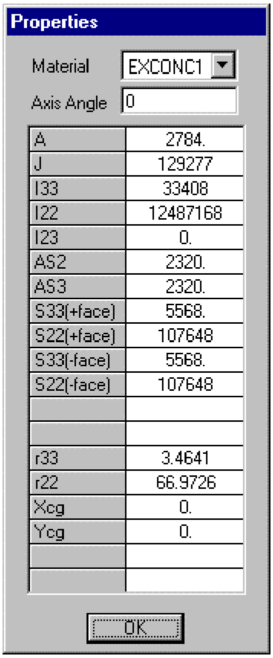
Display Interaction Surface

***(Above)* Figure 14-7:**
Completed section

In Section Designer you can display the interaction surface for concrete sections that have reinforcing steel defined. The wall created in this example meets this criteria.

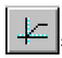
To display the interaction surface click the **Display menu** > **Show Interaction Surface** command or click the **Show Interaction Surface** button, , on the top toolbar. The Interaction Surface dialog box is displayed. See the section titled "Show Interaction Surface" in Chapter 9 for information on this dialog box. See Chapter 11 for background information on how the interaction surface is created by Section Designer.

Figure 14-8:
Section properties



Display Moment Curvature Curve

In Section Designer you can display a moment curvature curve for a concrete section that has reinforcing steel defined. The beam created in this example meets this criteria.

To display the moment curvature curve click the **Display menu > Show Moment Curvature Curve** command or click the **Show Moment Curvature Curve** button, , on the top toolbar. The Moment Curvature Curve dialog box is displayed. See the section titled "Show Moment Curvature Curve" in Chapter 9 for information on this dialog box. See Chapter 12 for background information on how the moment curvature curve is created by Section Designer.

Close Section Designer

To close Section Designer do the following:

1. Click the **File menu > Return to ETABS** command. This closes the Section Designer window and returns you to the Pier Section Data dialog box.

2. Click the **OK** button to close the Pier Section Data dialog box.

   Important note: If you instead click the **Cancel** button to close the Pier Section Data dialog box then the new section just defined in Section Designer is lost. Similarly, if you were modifying an existing section in Section Designer rather than creating a new one, then the changes to the section are lost if you click the **Cancel** button.

3. At this point you can define another wall pier section, or if you are through defining frame sections you can click the **OK** button to close the Pier Sections dialog box.

   Important note: If you instead click the **Cancel** button to close the Pier Sections dialog box then the section just defined in Section Designer and any other sections just defined are lost. Similarly, if you were modifying an existing section in Section Designer or any other existing section, then the changes to the section are lost if you click the **Cancel** button.

   In summary, to successfully close Section Designer and save your changes you should Click the **File menu > Return to ETABS** command in Section Designer and then click the **OK** button twice to close all dialog boxes.

Appendix 1

A1

The Section Designer Menu Structure

This appendix lays out the complete menu structure of Section Designer. The eight menus available in Section Designer are:

- File menu
- Edit menu
- View menu
- Draw menu
- Select menu
- Display menu
- Options menu
- Help menu

All of the commands available in each of these menus are listed in this appendix. Submenu items are indented in the list.

File Menu Commands

Print Setup
Print Graphics
Return to ETABS

Edit Menu Commands

Undo
Redo
Delete
Align
- Left
- Center
- Right
- Top
- Middle
- Bottom

Change Shape to Poly
Change Bar Shape to Single Bars

View Menu Commands

Rubberband Zoom
Restore Full View
Previous Zoom
Zoom In One Step
Zoom Out One Step
Pan
Show Guide Lines
Show Axes

Draw Menu Commands

- Select Mode
- Reshape Mode
- Draw Structural Shape
  - I/Wide Flange
  - Channel
  - Tee
  - Angle
  - Double Angle
  - Box/Tube
  - Pipe
  - Plate
- Draw Solid Shape
  - Rectangle
  - Circle
  - Segment
  - Sector
- Draw Poly Shape
- Draw Reinforcing Shape
  - Single Bar
  - Line Pattern
  - Rectangular Pattern
  - Circular Pattern
- Draw Reference Lines
  - Draw Reference Line
  - Draw Reference Circle
- Snap to
  - Guideline Intersections and Points
  - Line Ends and Midpoints
  - Line Intersections
  - Perpendicular Projections
  - Lines and Edges
  - Fine Grids
- Constrain Drawn Line to
  - None
  - Constant X
  - Constant Y
  - Constant Angle

Select Menu Commands

Select
- Pointer/in Window
- Intersecting Line
- All

Deselect
- Pointer/in Window
- Intersecting Line
- All

Get Previous Selection
Clear Selection

Display Menu Commands

Show Section Properties...
Show Interaction Surface...
Show Moment Curvature Curve...

Options Menu Commands

Preferences...
Colors...

Help Menu Commands

Search for Help on ...
About Section Designer...

Appendix 2

A2

The Section Designer Toolbar Buttons

This appendix describes the toolbar buttons that are available in Section Designer. Throughout the manual reference is made to the top toolbar and the side toolbar. These are identified in Figure A2-1.

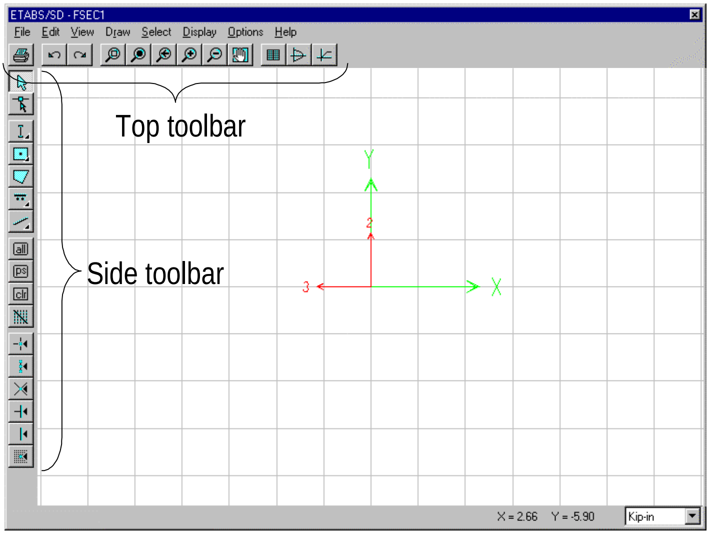
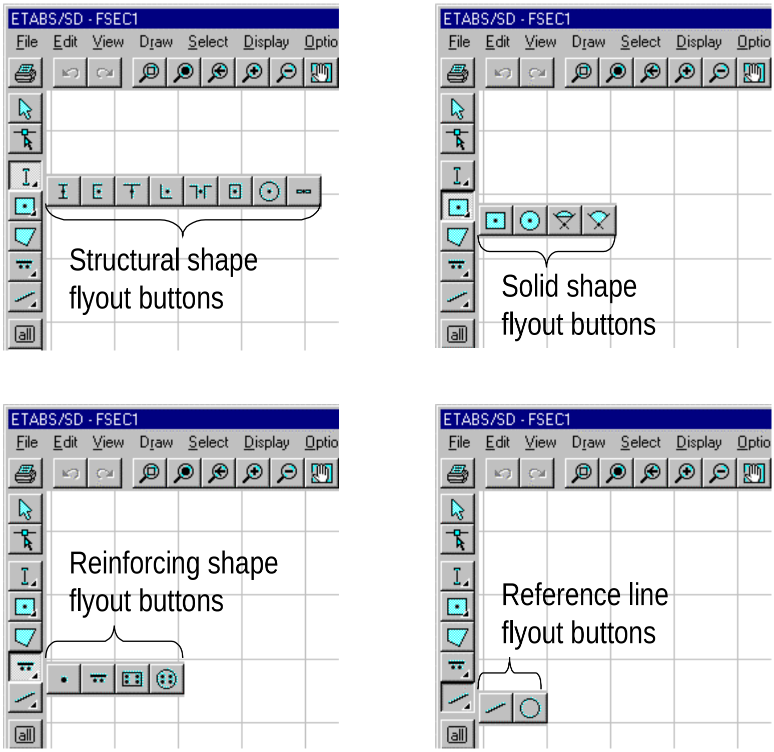


Figure A2-1: *Illustration of the top toolbar and the side toolbar*

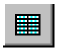
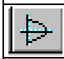
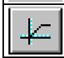
Figure A2-2:
Flyout toolbar buttons on the side toolbar

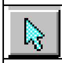
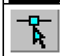
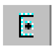
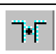
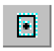
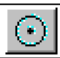


There are five separate Drawing-related buttons on the side toolbar. They are the Draw Structural Shape, Draw Solid Shape, Draw Polygon Shape, Draw Reinforcing Shape and Draw Reference Lines. Each of these drawing-related buttons has additional flyout buttons associated with it except for the Draw Polygon Shape button. The flyout buttons appear immediately after you click on the drawing-related button. Figure A2-2 illustrates the flyout buttons.

Table A2-1 lists the toolbar buttons available on each toolbar. Typically the buttons shown in the first column of the table are left justified. However, the flyout buttons on the side toolbar are shown indented in the first column of the table.

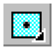
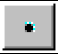
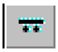
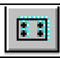
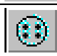
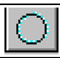
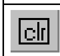
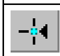
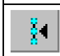
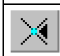
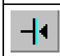
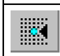
Table A2-1:
Toolbar buttons

Top Toolbar Buttons	
	Print Graphics
	Undo
	Redo
	Rubberband Zoom
	Restore Full View
	Restore Previous Zoom
	Zoom In One Step
	Zoom Out One Step
	Pan
	Show Section Properties
	Show Interaction Surface
	Show Moment Curvature Curve

Side Toolbar Buttons	
	Pointer (Select Mode)
	Reshaper (Reshape Mode)
	Draw Structural Shape (Displays Flyout Buttons)
	I/Wide Flange
	Channel
	Tee
	Angle
	Double Angle
	Box/Tube
	Pipe
	Plate

A2

Table A2-1, cont.:
Toolbar buttons

	Draw Solid Shape (Displays Flyout Buttons)
	Rectangle
	Circle
	Segment
	Sector
	Draw Poly Shape (Polygon)
	Draw Reinforcing Shape (Displays Flyout Buttons)
	Single Bar
	Line Pattern
	Rectangular Pattern
	Circular Pattern
	Draw Reference Lines (Displays Flyout Buttons)
	Reference Line
	Reference Circle
	Select All
	Restore Previous Selection
	Clear Selection
	Set Intersecting Line Select Mode
	Snap to Guide Intersections/Points
	Snap to Middle and Ends
	Snap to Intersections
	Snap to Perpendicular
	Snap to Lines and Edges
	Snap to Fine Grid

A2

Appendix 3

Section Designer Keyboard Commands

A3

This appendix describes the keyboard commands that are available in Section Designer. When referring to keyboard commands in this appendix a "+" sign means to press two keys simultaneously. For example, "Ctrl + C" means to press the Ctrl key and the C key on your keyboard at the same time.

The keyboard commands in Section Designer can be broken into three main categories. They are selection commands, constraint commands and editing commands.

The table starting on the next page lists the Section Designer keyboard commands.

Selection Commands

Key	Description
Ctrl	When two shapes are drawn one on top of the other, holding down the Ctrl key as you left click on the shapes brings up a dialog box where you can choose the shape to select. Similarly, holding down the Ctrl key as you right click on the shapes brings up a dialog box where you can select the shape whose properties you want to see.
Ctrl + A	Select all shapes.
Ctrl + C	Copies the entire table displayed in the Interaction Surface dialog box. Also copies the entire table displayed in the Moment Curvature Curve dialog box.

Constraint Commands

Key	Description
X	Drawing constraint that locks the X component of the next point so that it is the same as the previous point.
Y	Drawing constraint that locks the Y component of the next point so that it is the same as the previous point.
A	Allows you to specify an angle in degrees in the status bar. Drawing is then constrained along this angle
Spacebar	Remove the current drawing constraint.

Editing Commands

Key	Description
Del	Deletes the currently selected shape(s).
Ctrl + ← Ctrl + ↑ Ctrl + → Ctrl + ↓	Moves the selected shape in the direction specified by the arrow. You can press and hold down the Ctrl key first and then press the arrow key. Each time you press the arrow key the selected shape moves the amount specified by the Nudge Value item in the Section Designer preferences.

Interaction Surface Example Calculation

Note:

See the section titled "How to Create this Example Problem" at the end of this appendix for a description of how to define this section in Section Designer.

This appendix presents some example hand calculations for developing an interaction surface. Consider the section shown in Figure A4-1. The section is 24 inches wide by 36 inches deep with one #9 bar located at each corner. The cover *to the center* of the #9 bars is 1.5 inches.

Rather than showing calculations for developing the entire interaction surface of this section we will show calculations for two of the PMM interaction curves that make up the interaction surface, one curve at 0 degrees and the other at 30 degrees. See Chapter 11 for more information. The calculations in this example are based on the 1997 UBC.

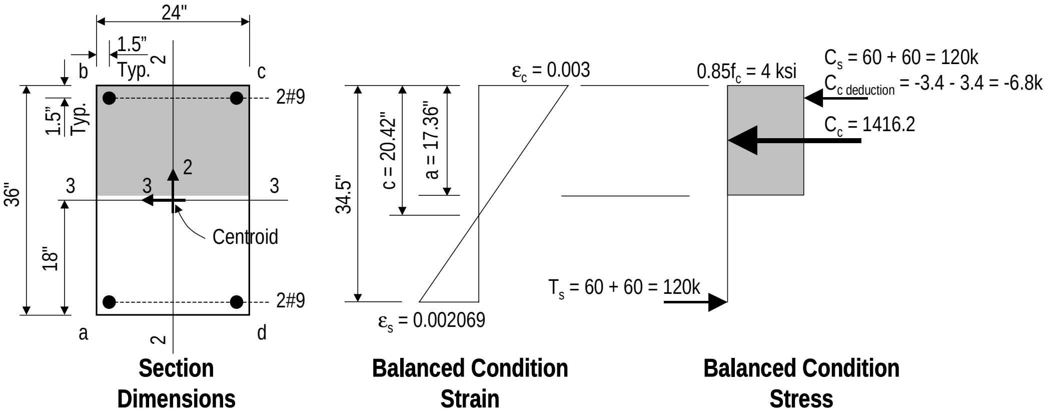
The following assumptions are made for this example:

f'_c = 4 ksi

f_y = 60 ksi

β = 0.85

E_s = 29000 ksi



(Above)
Figure A4-1:
Section for interaction surface example

$$\phi_b = 0.9$$

$$\phi_c = 0.7$$

A further assumption is that we will use eleven points (the ETABS default) to define an interaction curve.

A4

Interaction Curve at 0 Degrees

The first step in developing the interaction curve is to determine which eleven points will be used for the interaction curve. Recall that ETABS always considers an odd number of points for an interaction curve. In general, ETABS assumes constant compression for the first point, balanced conditions for the middle point (6th point in this example) and constant tension for the last point (11th point in this example).

Each of these three points has a depth of compression block, a, associated with it. ETABS assumes that the a dimensions associated with points 2 through 5 are equally spaced between the a dimensions obtained for points 1 and 6. Similarly, ETABS assumes that the a dimensions associated with points 7 through 10 are equally spaced between the a dimensions obtained for points 6 and 11.

For the first point ETABS assumes constant compression on the section. For this point, c equals ∞ and a equals the section depth, 36 inches.

For the middle (6th) point ETABS assumes balanced conditions and calculates the a value for those conditions. This is shown in Figure A4-1 where a_b = 17.36 in. Recall that a = 0.85c. Also note that the maximum steel strain at balanced conditions is equal to the steel yield strain, ε_y, where $\varepsilon_y = f_y/E_s$.

For the last (11th) point ETABS assumes constant tension on the section. For this point, c and a both equal zero.

Once ETABS knows the c and a dimensions for each point on the interaction curve, and it assumes that ε_c = 0.003 at each point, it has the necessary information to construct the strain diagram (using similar triangles) and thus derive the force diagram and ultimately determine the section axial force and moment. Table A4-1 summarizes the strain values and the c and a dimensions for each of the eleven points.

Table A4-1: *Concrete and steel strain, and c and a dimensions for each point*

Point	ε_c in/in	c in	Max ε_s in/in	a in
1	-0.003	Infinity	-0.00300	36.00
2	-0.003	37.97	-0.00027	32.27
3	-0.003	33.58	0.00008	28.54
4	-0.003	29.19	0.00055	24.81
5	-0.003	24.81	0.00117	21.08
6	-0.003	20.42	0.00207	17.36
7	-0.003	16.33	0.00334	13.88
8	-0.003	12.25	0.00545	10.41
9	-0.003	8.17	0.00967	6.94
10	-0.003	4.08	0.02234	3.47
11	-0.003	0.00	Infinity	0.00

Next we calculate the concrete compressive force, $C_c = 0.85f'_cab$, where b is the width of the section, 24 inches as shown in Table A4-2. Note that we will later deduct a small portion of this force to account for the area of concrete that does not exist at the reinforcing steel locations.

Table A4-2: *Concrete compressive force for each point*

Point	f'c ksi	a in	b in/in	Cc kips
1	4	36.00	24	-2937.6
2	4	32.27	24	-2633.3
3	4	28.54	24	-2329.0
4	4	24.81	24	-2024.8
5	4	21.08	24	-1720.5
6	4	17.36	24	-1416.2
7	4	13.88	24	-1133.0
8	4	10.41	24	-849.7
9	4	6.94	24	-566.5
10	4	3.47	24	-283.2
11	4	0.00	24	0.0

Now we turn our attention to the reinforcing steel. Note that in Figure A4-1 the four corners of the section are labeled a, b, c and d. Based on this we will refer to the bars in corners a, b, c and d bar a, bar b, bar c and bar d respectively. Knowing the concrete strain and the maximum steel strain for each of the 11 points, we can use similar triangles to calculate the strain of each bar. These strains are shown in Table A4-3.

Table A4-3: *Strain in each bar*

Point	Bar a ε in/in	Bar b ε in/in	Bar c ε in/in	Bar d ε in/in
1	-0.00300	-0.00300	-0.00300	-0.00300
2	-0.00027	-0.00288	-0.00288	-0.00027
3	0.00008	-0.00287	-0.00287	0.00008
4	0.00055	-0.00285	-0.00285	0.00055
5	0.00117	-0.00282	-0.00282	0.00117
6	0.00207	-0.00278	-0.00278	0.00207
7	0.00334	-0.00272	-0.00272	0.00334
8	0.00545	-0.00263	-0.00263	0.00545
9	0.00967	-0.00245	-0.00245	0.00967
10	0.02234	-0.00190	-0.00190	0.02234
11	Infinity	Infinity	Infinity	Infinity

Now the stress in each bar can be determined. If the strain in the bar is less than the yield strain ($\varepsilon_y = f_y E_s$) then the bar stress, σ, is calculated as $\sigma = \varepsilon E_s$. If the strain in the bar is greater than or equal to the yield strain then the bar stress is equal to the yield stress, f_y. Table A4-4 lists the bar stress for each bar at each point on the interaction surface.

Table A4-4:
Stress in each bar

Point	Bar a σ ksi	Bar b σ ksi	Bar c σ ksi	Bar d σ ksi
1	-60.00	-60.00	-60.00	-60.00
2	-7.94	-60.00	-60.00	-7.94
3	2.39	-60.00	-60.00	2.39
4	15.82	-60.00	-60.00	15.82
5	34.00	-60.00	-60.00	34.00
6	60.00	-60.00	-60.00	60.00
7	60.00	-60.00	-60.00	60.00
8	60.00	-60.00	-60.00	60.00
9	60.00	-60.00	-60.00	60.00
10	60.00	-55.04	-55.04	60.00
11	60.00	60.00	60.00	60.00

A4

The force in each bar is obtained by multiplying the stress times the bar area. The #9 bars in this example have an area of 1.00 in^2. Table A4-5 lists the bar force for each bar at each point on the interaction surface.

Table A4-5:
Force in each bar

Point	Bar a kips	Bar b kips	Bar c kips	Bar d kips
1	-60.00	-60.00	-60.00	-60.00
2	-7.94	-60.00	-60.00	-7.94
3	2.39	-60.00	-60.00	2.39
4	15.82	-60.00	-60.00	15.82
5	34.00	-60.00	-60.00	34.00
6	60.00	-60.00	-60.00	60.00
7	60.00	-60.00	-60.00	60.00
8	60.00	-60.00	-60.00	60.00
9	60.00	-60.00	-60.00	60.00
10	60.00	-55.04	-55.04	60.00
11	60.00	60.00	60.00	60.00

Next we determine the concrete compressive force that must be deducted because the assumed concrete area is actually taken up by reinforcing steel. The force is equal to $0.85f'_cA_{bar}$ for each reinforcing bar that falls in the a dimension of the section. Table A4-6 lists the concrete compressive forces that must be deducted at each of the eleven points on the interaction surface.

Table A4-6: *Concrete compressive forces that must be deducted*

Point	**Bar a** kips	**Bar b** kips	**Bar c** kips	**Bar d** kips
1	3.4	3.4	3.4	3.4
2	0	3.4	3.4	0
3	0	3.4	3.4	0
4	0	3.4	3.4	0
5	0	3.4	3.4	0
6	0	3.4	3.4	0
7	0	3.4	3.4	0
8	0	3.4	3.4	0
9	0	3.4	3.4	0
10	0	3.4	3.4	0
11	0	0	0	0

Now we can sum up all of the forces, and all of the moments about the section centroid for each point. The results of this are shown in Table A4-7.

Table A4-7: *Axial forces and moments*

Point	**Sum of Axial Force** kips	**Sum of Moments** kip-in
1	-3164.0	0
2	-2762.4	6515
3	-2437.5	10631
4	-2106.3	13715
5	-1765.7	15821
6	-1409.4	17050
7	-1126.2	16376
8	-842.9	14719
9	-559.7	12078
10	-266.5	8291
11	240.0	0

As an example of how the values in Table A4-7 are calculated refer to Table A4-8 which shows the details of the calculation for Point 6.

Table A4-8: *Details of how values in Table A4-7 are calculated for Point 6*

Item	**Force** kips	**Distance to Centroid** in	**Moment** kip-in
Concrete compression	-1416.2	-9.32	13202
Bar a tension	60	16.5	990
Bar b compression	-60	-16.5	990
Bar c compression	-60	-16.5	990
Bar d tension	60	16.5	990
Bar a correction	0	16.5	0
Bar b correction	3.4	-16.5	-56
Bar c correction	3.4	-16.5	-56
Bar d correction	0	16.5	0
Sum	**-1409.4**		**17050**

Note:

See the section titled "Sign Convention for Axial Load and Moments" later in this appendix for information on the sign of the moment.

Next we calculate the maximum compression force, P_{oc}, as:

$$P_{oc} = 0.85f'_c (A_g - A_s) + f_yA_s$$

$$P_{oc} = 0.85 * 4 * [(24 * 36) - (4 * 1.00)] + 60 * (4 * 1.00)$$

$$P_{oc} = 3164.0 \text{ kips}$$

The maximum compression force in the interaction surface can not exceed $0.8P_{oc}$.

$$0.8P_{oc} = 0.8 * 3164.0 = 2531.2 \text{ kips}$$

Any compression force in Table A4-7 that is larger than 2531.2 kips must be reduced to 2531.2 kips. Thus points 1 and 2 in Table A4-7 must have their axial force reduced. This reduction does not affect the moment.

The new values of axial force and moment, accounting for the required reduction in axial force at points 1 and 2, are shown in Table A4-9. These values correspond to those obtained in Section Designer with no phi.

Table A4-9: *Interaction curve values with no phi factor*

Point	Sum of Axial Force kips	Sum of Moments kip-in
1	-2531.2	0
2	-2531.2	6515
3	-2437.5	10631
4	-2106.3	13715
5	-1765.7	15821
6	-1409.4	17050
7	-1126.2	16376
8	-842.9	14719
9	-559.7	12078
10	-266.5	8291
11	240.0	0

Now we will consider the phi factors (strength reduction factors). To consider the phi factors first find the smaller of $0.1A_gf'_c$ and ϕP_b.

$0.1A_gf'_c = 0.1 * (24 * 36) * 4 = 345.6$ kips

$\phi P_b = 0.7 * 1409.4 = 986.6$ kips

The smaller value is $0.1A_gf'_c = 345.6$ kips. Thus the phi factor is increased linearly from 0.7 to 0.9 as the compression axial force goes from -345.6 kips to 0.

Table A4-10 shows the phi factor assumed for each of the eleven points. The phi factor at point 10 is calculated as follows:

$phi_{10} = 0.9 - [0.2P / \text{minimum of } (0.1A_gf'_c \text{ and } \phi P_b)]$

$phi_{10} = 0.9 - [0.2 * 266.5 / 345.6]$

$phi_{10} = 0.9 - [0.2 * 266.5 / 345.6]$

$phi_{10} = 0.75$

Note that there may not necessarily be points on the interaction curve where the transition of phi starts and ends. Thus the shape of the curve obtained may be slightly different from the exact theoretical solution. As you increase the number of points considered on the curve, the shape of the interaction curve will converge to the exact solution.

Table A4-10:
Phi factors

Point	**Sum of Axial Force** kips	**Sum of Moments** kip-in	**Phi Factor**
1	-2531.2	0	0.7
2	-2531.2	6515	0.7
3	-2437.5	10631	0.7
4	-2106.3	13715	0.7
5	-1765.7	15821	0.7
6	-1409.4	17050	0.7
7	-1126.2	16376	0.7
8	-842.9	14719	0.7
9	-559.7	12078	0.7
10	-266.5	8291	0.75
11	240.0	0	0.9

Finally, the axial force and moment are multiplied by the phi factor to get the final force and moment for each of the 11 points. This is demonstrated in Figure A4-11. These values correspond to those obtained in Section Designer with a phi factor.

Table A4-11:
Interaction curve values with a phi factor

Point	**Sum of Axial Force** kips	**Sum of Moments** kip-in
1	-1771.8	0
2	-1771.8	4561
3	-1706.2	7442
4	-1474.4	9600
5	-1236.0	11075
6	-986.6	11935
7	-788.3	11463
8	-590.1	10303
9	-391.8	8455
10	-198.8	6183
11	216.0	0

Finally, if you wanted to calculate the values for the Section Designer "no phi with fy increase" option, then you would repeat the process described here up through Table A4-9 with the f_y value increased appropriately. For the 1997 UBC the fy value is increased by a factor of 1.25. Thus you would repeat the process described here up through Table A4-9 with f_y equal to 1.25 * 60 = 75 ksi.

Interaction Curve at 30 Degrees

For the interaction curve at 30 degrees we will demonstrate the calculation to provide the axial force and moment at one of the eleven points of the interaction curve. We will do this for point 6, the point of balanced strain conditions.

Refer to Figure A4-2. We begin by determining the distances from the section centroid to the extreme compression fiber and to all of the reinforcing steel in the rotated (2'-3') coordinate system. Note that the 2'-3' coordinate system is rotated 30 degrees from the 2-3 coordinate system as shown in Figure A4-2a.

Given the 41.128" dimension from the extreme compression fiber to the extreme tension rebar, and given the $\varepsilon_c = 0.003$ and $\varepsilon_s = 0.002069$ the c and a dimensions in Figure A4-2b are determined using similar triangles. The area of concrete shown shaded in Figure A4-2a corresponds to the depth, a, from the extreme compression fiber to the bottom of the rectangular concrete stress block.

Note:

A rectangular concrete stress block is used in all calculations of the interaction surface.

The concrete compression force, Cc, illustrated in Figure A4-2c, is equal to $0.85f'_cA_c$, where A_c is the area of concrete in Figure A4-2a. The dimensions of this concrete area are shown in Figure A4-2e. The dashed lines in Figure A4-2e also indicate how the area can be broken up into a triangle and a rectangle for easier calculation.

$$A_{rectangle} = 24 * 10.034 = 240.82 \text{ in}^2$$

$$A_{triangle} = 0.5 * 24 * (23.891 - 10.034) = 166.28 \text{ in}^2$$

$$C_c = 0.85 * 4 * 240.82 + 0.85 * 4 * 166.28$$

$$C_c = 818.79 + 565.35 = 1384.14 \text{ kips}$$

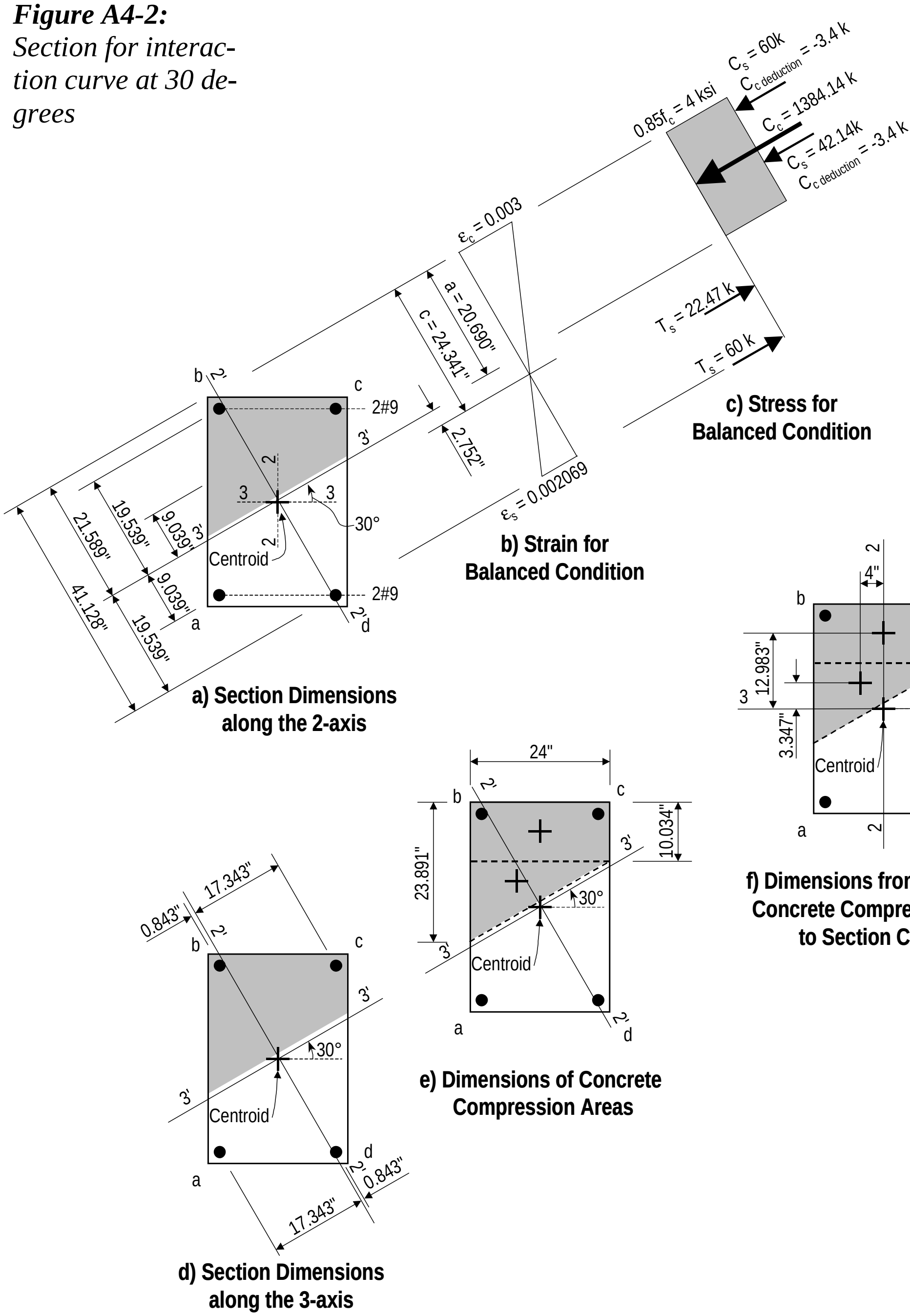


Figure A4-2:
Section for interaction curve at 30 degrees

A4

To determine the steel forces and the concrete deduction forces shown in Figure A4-2c we must first calculate the strain at each rebar location. These strains are calculated using similar triangles. Table A4-11 summarizes these strains.

Table A4-11: *Strain at each rebar location*

Rebar	Bar Strain, in/in
a	0.002069 * [(9.039-2.752) / (19.539-2.752)] = 0.000775
b	0.003 * [(19.539 + 2.752) / 24.341] = 0.002747
c	0.003 * [(9.039 + 2.752) / 24.341] = 0.001453
d	0.002069

Now the stress in each bar can be determined. If the strain in the bar is less than the yield strain ($\varepsilon_y = f_y E_s = 0.002069$) then the bar stress, σ, is calculated as $\sigma = \varepsilon E_s$. If the strain in the bar is greater than or equal to the yield strain then the bar stress is equal to the yield stress, f_y. Table A4-12 lists the bar stress for each bar. A negative strain and stress in this table indicates the bar is in compression.

Table A4-12: *Stress in each rebar*

Rebar	Strain	Bar Stress, ksi
a	0.000775	0.000775 * 29000 = 22.47 ksi
b	-0.002747	-60 ksi
c	-0.001453	-0.001453 * 29000 = -42.14 ksi
d	0.002069	60 ksi

The force in each bar is obtained by multiplying the stress times the bar area. The #9 bars in this example have an area of 1.00 in^2. Table A4-13 lists the bar force for each bar.

Table A4-12: *Force in each rebar*

Rebar	Bar Force, kips
a	22.47 k
b	-60 k
c	-42.14 k
d	60 k

Next we determine the concrete compressive force that must be deducted because the assumed concrete area is actually taken up by reinforcing steel. The force is equal to $0.85f'_cA_{bar}$ for each reinforcing bar that falls in the a dimension of the section. In this case a 3.4 kip force must be deducted at bar locations b and c.

Now we can sum up all of the forces, and all of the moments about the centroid, around the 2 and 3 axis. *Note that the moments are taken about the 2 and 3 axes, not the 2' and 3' axes.* The results of this are shown in Table A4-13.

The distances from the centroid of the concrete compression areas to the centroid of the section are shown in Figure A4-2f. Note that the M2 moment is equal to the force times the 3-dir distance to the centroid and the M3 moment is equal to the force times the 2-dir distance to the centroid.

(Below) Table A4-13: *Summing forces and moments*

Item	**Force** kips	**2-Dir Distance to Centroid** in	**3-Dir Distance to Centroid** in	**M3 Moment** kip-in	**M2 Moment** kip-in
Conc comp (rectangle)	-818.79	-12.983	0	10630.4	0.0
Conc comp (triangle)	-565.35	-3.347	-4	1892.2	2261.4
Bar a tension	22.47	16.5	-10.5	370.8	-235.9
Bar b compression	-60	-16.5	-10.5	990.0	630.0
Bar c compression	-42.14	-16.5	10.5	695.3	-442.5
Bar d tension	60	16.5	10.5	990.0	630.0
Bar b correction	3.4	-16.5	-10.5	-56.1	-35.7
Bar c correction	3.4	-16.5	10.5	-56.1	35.7
Sum	**-1397.0**			**15456**	**2843**

A4

Note:

See the section titled "Sign Convention for Axial Load and Moments" later in this appendix for information on the sign of the moments.

Next we calculate the maximum compression force, P_{oc}, exactly as we did for the 0 degree case.

$$P_{oc} = 0.85f'_c (A_g - A_s) + f_y A_s$$

$$P_{oc} = 0.85 * 4 * [(24 * 36) - (4 * 1.00)] + 60 * (4 * 1.00)$$

$$P_{oc} = 3164.0 \text{ kips}$$

The maximum compression force in the interaction surface can not exceed $0.8P_{oc}$.

$$0.8P_{oc} = 0.8 * 3164.0 = 2531.2 \text{ kips}$$

Since 1397.0 kips is less than 2531.2 kips we do not need to reduce the axial force. Thus the final axial force and moments when no phi factors are used are P = -1397.0 k, M3 = 15456 k-in and M2 = 2843 k-in.

Now we will consider the phi factors (strength reduction factors). To consider the phi factors first find the smaller of $0.1A_gf'_c$ and ϕP_b.

$$0.1A_gf'_c = 0.1 * (24 * 36) * 4 = 345.6 \text{ kips}$$

$$\phi P_b = 0.7 * 1397.0 = 977.9 \text{ kips}$$

The smaller value is $0.1A_gf'_c$ = 345.6 kips. Thus the phi factor is increased linearly from 0.7 to 0.9 as the compression axial force goes from -345.6 kips to 0. Since the compression force we are considering, -1397.0 kips is larger than -345.6 kips, we use a phi factor of 0.7. Thus the axial force and moments are all multiplied by 0.7 to get the final axial force and moments when phi factors are used. The final axial force and moments when phi factors are used are:

P = 0.7 * 1397.0 = 977.9 kips

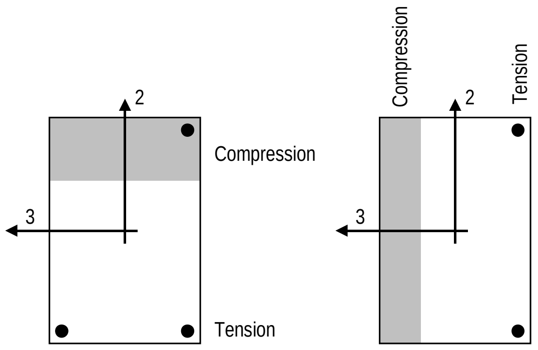
M3 = 0.7 * 15456 = 10819 kips

M2 = 0.7 * 2843 = 1990 kips

Sign Convention for Axial Load and Moments

This section defines the sign convention for axial loads and moments in the interaction surface. For axial loads tension is positive and compression is negative.

The sketch to the left defines positive M2 and M3 moments for the interaction surface.



a) Positive M3 Moment a) Positive M2 Moment

How to Create this Example Problem

This section provides step-by-step directions for creating this example problem in Section Designer.

1. In ETABS set the units to kips and inches.

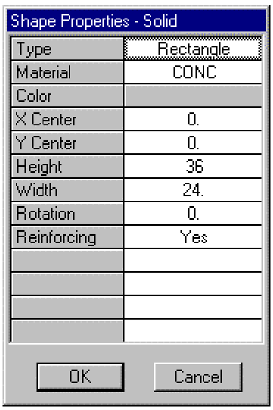
2. If you do not already have a model open then click the **File menu** > **New Model** command. Next click the **No** button on the New Model Initialization form and then click the **OK** button in the Building Plan Grid System and Story Data Definition dialog box.

3. In ETABS click the **Define menu** > **Material Properties** command and verify the material named CONC has E = 3600 ksi, f'c = 4 ksi, fy = 60.

4. In ETABS click the **Options menu** > **Preferences** > **reinforcement Bar Sizes** command. Verify that the #9 bar has a 1.00 in^2 bar area and a 1.128 in bar diameter.

5. In ETABS click the **Define menu** > **Frame Sections** command and then select the Add SD Section item from the drop-down box.

6. This opens the SD Section Data dialog box. Click the **Section Designer** button in the dialog box to start Section Designer.

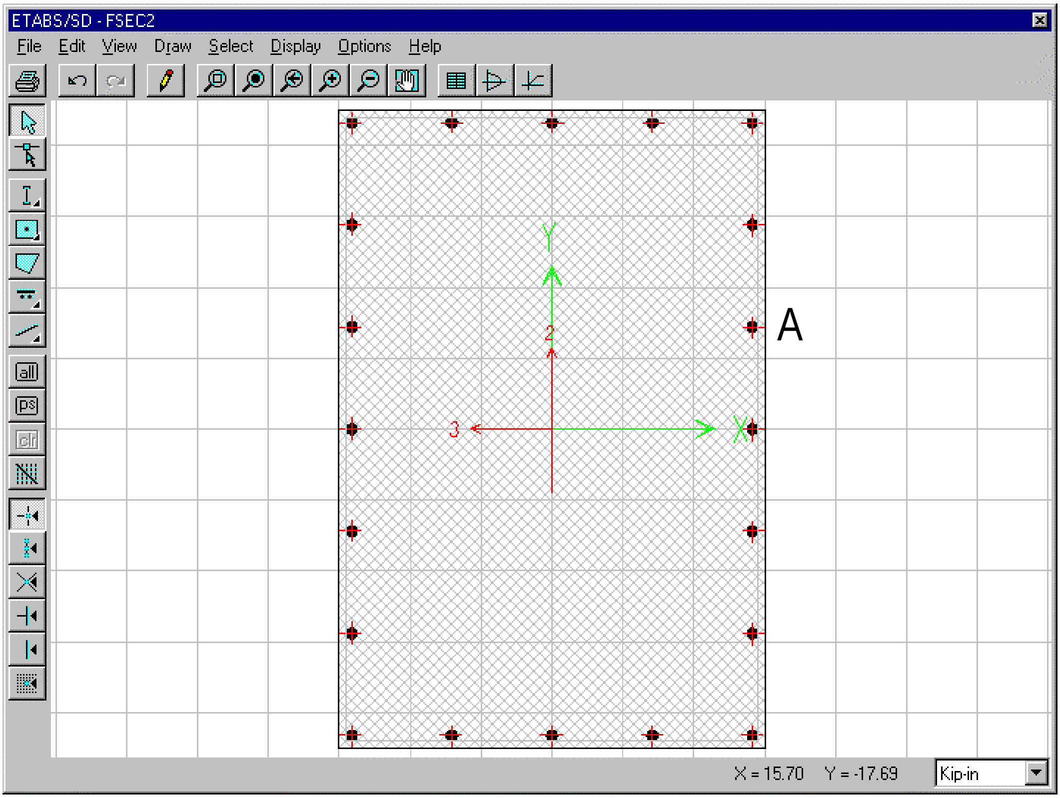


7. In Section Designer click the **Draw menu** > **Draw Solid Shape** > **Rectangle** command. Then click on the origin to draw a rectangle.

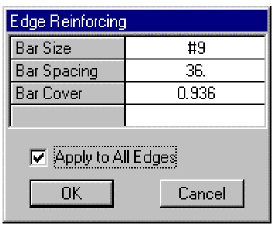
8. Click the **Draw menu** > **Select Mode** command.

9. Right click on the rectangle to bring up the Shape Properties dialog box. Fill in this dialog box as shown in the sketch to the left. Be sure to set the height to 36 inches and to set the Reinforcing item to Yes. Then click the **OK** button.

10. The section now appears as shown in Figure A4-3.



(Above)
Figure A4-3:
Section Designer at step 9



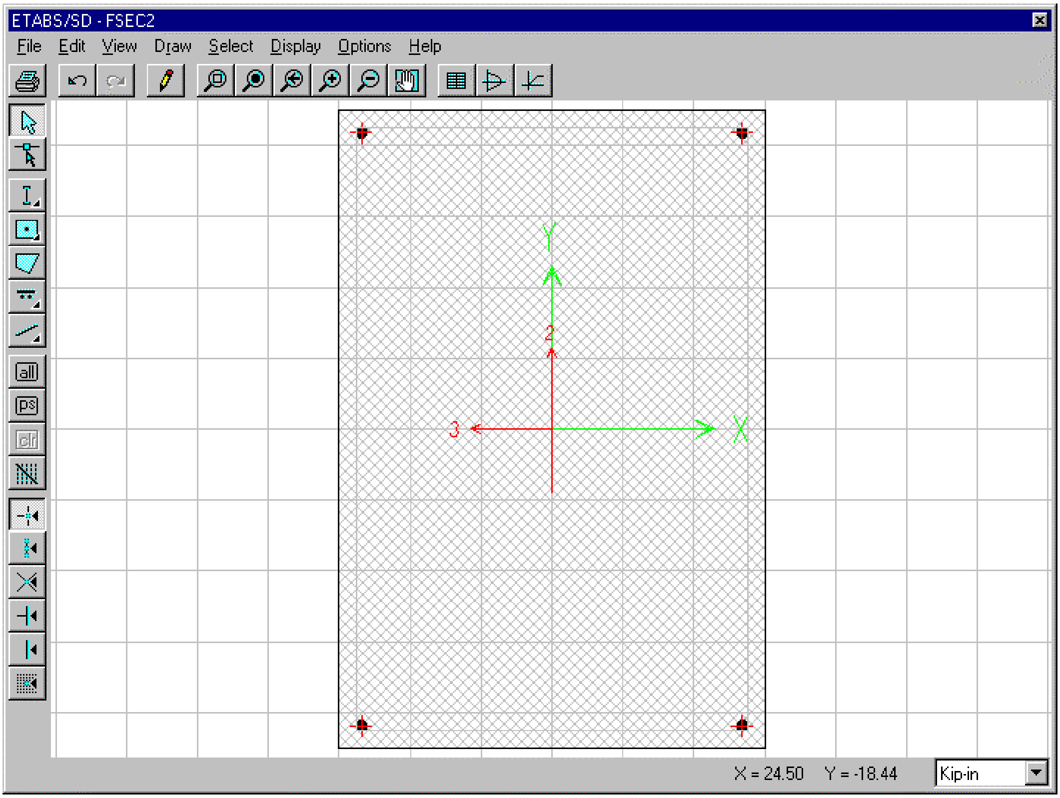
11. Right click on one of the edge rebar, for example the one labeled "A" in Figure A4-3 to bring up the Edge Reinforcing dialog box. Fill in this dialog box as shown in the sketch to the left. Be sure to check the Apply to All Edges check box. Then click the **OK** button.

12. The section now appears as shown in Figure A4-4.

13. Right click on one of the corner bars to bring up the Corner Point Reinforcing dialog box. Set the bar size to #9, check the Apply to All Corners check box and then click the **OK** button.

14. Click the **Display menu > Show Interaction Surface** command to display the interaction surface.

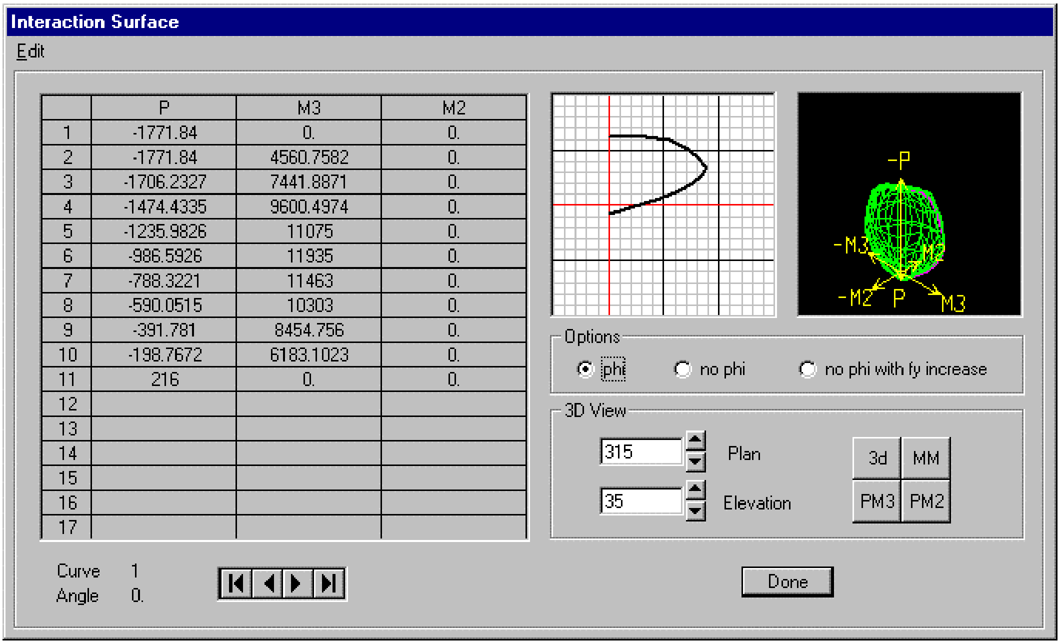
15. When the interaction surface is first displayed you can see the tabulated data for the interaction curve at 0 degrees.



(Above)
Figure A4-4:
Section Designer at step 11

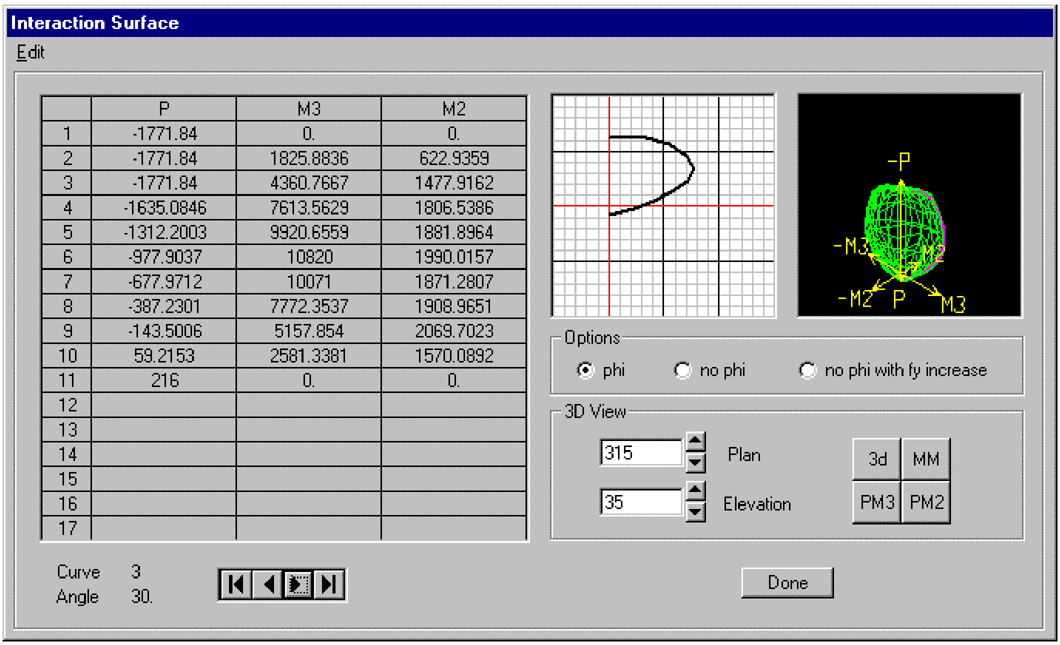
Click on the arrow buttons at the bottom of the interaction surface dialog box to change the angle.

The Section Designer results for the 0 degree and 30 degree angles are shown in Figure A4-5.



	P	M3	M2
1	-1771.84	0.	0.
2	-1771.84	4560.7582	0.
3	-1706.2327	7441.8871	0.
4	-1474.4335	9600.4974	0.
5	-1235.9826	11075	0.
6	-986.5926	11935	0.
7	-788.3221	11463	0.
8	-590.0515	10303	0.
9	-391.781	8454.756	0.
10	-198.7672	6183.1023	0.
11	216	0.	0.

A4



	P	M3	M2
1	-1771.84	0.	0.
2	-1771.84	1825.8836	622.9359
3	-1771.84	4360.7667	1477.9162
4	-1635.0846	7613.5629	1806.5386
5	-1312.2003	9920.6559	1881.8964
6	-977.9037	10820	1990.0157
7	-677.9712	10071	1871.2807
8	-387.2301	7772.3537	1908.9651
9	-143.5006	5157.854	2069.7023
10	59.2153	2581.3381	1570.0892
11	216	0.	0.

Figure A4-5:
Section Designer results at 0 degrees (top) and at 30 degrees (bottom)

Appendix 5

Moment Curvature Curve Example Calculation

Overview

Note:

See the section titled "How to Create this Example Problem" at the end of Appendix 4 for a description of how to define the example section in Section Designer.

This appendix presents some example hand calculations for developing a moment curvature curve. See Chapter 12 for more information on the moment curvature curves. The section considered in this example is discussed at the beginning of Appendix 4 and is illustrated in Figure A4-1. See the section titled "How to Create this Example Problem" at the end of Appendix 4 for a description of how to define the example section in Section Designer.

The following assumptions are made for this example:

- The axial load is 0 kips.
- The angle considered is 0 degrees.

- There are 11 points (maximum) on the moment curvature curve.

Default Maximum Curvature

We begin by calculating the default maximum curvature considered by ETABS. This maximum curvature is equal to the steel strain at fracture (0.1 in/in) divided by the largest perpendicular distance from the center of a rebar to an axis through the section centroid at a zero degree angle (horizontal). This distance is the same for all rebar in the section and is equal to 16.5 inches. Thus the default maximum curvature is 0.1 / 16.5 = 6.061E-3 in^{-1}.

Specific Curvature Values Considered

The specific curvature values considered are defined by Figure 12-4 and Equation 12-7 (see Chapter 12). From Equation 12-7:

$$a = \frac{\kappa_{max}}{(n - 1)\left[1 + \frac{n-2}{2}\left(\frac{b}{a}\right)\right]}$$

$$a = \frac{6.061E - 3}{(11 - 1)\left[1 + \frac{11-2}{2}(0.5)\right]}$$

a = 1.865E-4 in^{-1}

b = 0.5a = 9.324E-5 in^{-1}

Now the curvature values associated with each of the eleven points on the moment curvature diagram can be determined based on Figure 12-4. The spaces between each curvature point are reported in Table A5-1. The actual curvature values at each point are reported in Table A5-2.

Table A5-1:
Space between each curvature point

Points	Space in
1 to 2	a = 1.865E-04
2 to 3	a + b = 2.797E-04
3 to 4	a + 2b = 3.730E-04
4 to 5	a + 3b = 4.662E-04
5 to 6	a + 4b = 5.594E-04
6 to 7	a + 5b = 6.527E-04
7 to 8	a + 6b = 7.459E-04
8 to 9	a + 7b = 8.392E-04
9 to 10	a + 8b = 9.324E-04
10 to 11	a + 9b = 1.026E-03

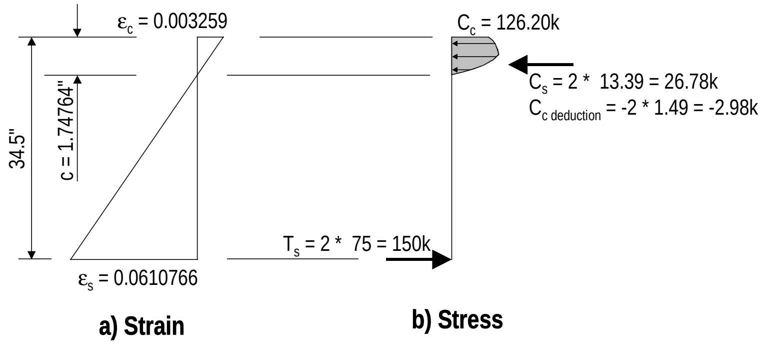
Table A5-2:
Curvature values used for each point on the moment curvature curve

Point	Curvature 1/in
1	0
2	1.865E-04
3	4.662E-04
4	8.392E-04
5	1.305E-03
6	1.865E-03
7	2.517E-03
8	3.263E-03
9	4.103E-03
10	5.035E-03
11	6.061E-03

For this example we will calculate the moment associated with point 6, that is, associated with a curvature of 1.865E-03 in^{-1}. Actually, if you take this curvature to a few more significant digits, it is 0.0018648 in/in. This is the curvature we will actually use in our calculations. Note that for a given strain the concrete stress and the steel stress are taken from the stress-strain diagrams shown in Figure 12-1.

The actual procedure of calculating the moment involves a trial and error procedure. In this procedure, given the curvature, you assume a concrete strain. Then you can calculate the associated

Figure A5-1:
Strain and stress diagrams



axial load and moment. If the axial load equals your desired axial load, 0 kips in this example, then the assumption for the concrete strain is correct and the iteration is done. Otherwise you try again with a new value for the concrete strain.

After several iterations of the above process you can arrive at a concrete strain of 0.003259 in/in. We will start there with our hand calculation.

Note: Some of the moment curvature calculations need to be carried out to a large number of significant digits in order to properly converge. Thus some of these hand calculations are done with an unusually large number of significant digits.

The first step is to develop the strain diagram as shown in Figure A5-1a. The concrete strain is given and the curvature is given. Using a rearranged version of Equation 12-4 we can solve for the maximum tensile strain, ε_s.

$$\varepsilon_s = \kappa_a d' - \varepsilon_c$$

$$\varepsilon_s = 0.0018648 \text{ in}^{-1} * 34.5 \text{ in} - 0.003259 \text{ in/in}$$

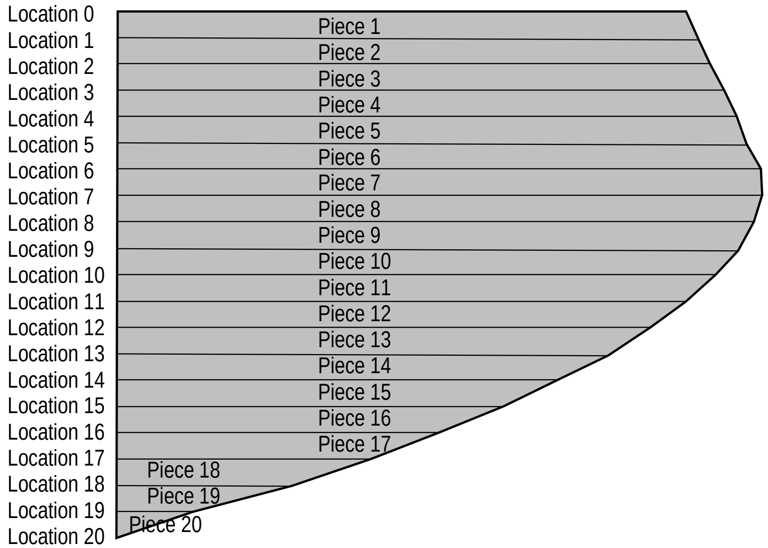
$$\varepsilon_s = 0.0610766 \text{ in/in}$$

The distance from the top of the section to the neutral axis, c, is calculated using similar triangles as 1.74764 in.

As previously mentioned, for a given strain the concrete stress and the steel stress are taken from the stress-strain diagrams shown in Figure 12-1. To consider the concrete compression force we will discretize the concrete compression area (top

A5

Figure A5-2: *Discretization of concrete compression block*



1.74764 in) of the section into 20 equal-depth pieces as shown in Figure A5-2.

The locations shown in Figure A5-2 are the locations of the top and bottom of each piece. For example, Location 1 is the top of Piece 2 and Location 2 is the bottom of Piece 2. Table A5-3 lists the distance from the top of the section to each (equally spaced) location. Also shown in the table is the concrete strain and the concrete stress at each location.

As an example, consider Location 7. The distance is calculated as:

$$\text{Distance} = \frac{7}{20} * 1.74764 = 0.61167 \text{ in}$$

The concrete strain is calculated from similar triangles as:

$$\text{Strain} = \left(1 - \frac{7}{20}\right) * 0.003259 = 0.002118 \text{ in/in}$$

A5

Table A5-3:
Stress and strain at each specified location in the concrete compression block

Location	Distance from Top of Section in	Strain in/in	Stress ksi
0	0.00000	0.003259	3.502
1	0.08738	0.003096	3.581
2	0.17476	0.002933	3.659
3	0.26214	0.002770	3.737
4	0.34952	0.002607	3.815
5	0.43691	0.002444	3.893
6	0.52429	0.002281	3.972
7	0.61167	0.002118	3.991
8	0.69905	0.001955	3.942
9	0.78643	0.001792	3.850
10	0.87381	0.001629	3.715
11	0.96119	0.001467	3.537
12	1.04857	0.001304	3.316
13	1.13596	0.001141	3.052
14	1.22334	0.000978	2.745
15	1.31072	0.000815	2.395
16	1.39810	0.000652	2.002
17	1.48548	0.000489	1.566
18	1.57286	0.000326	1.087
19	1.66024	0.000163	0.565
20	1.74762	0.000000	0.000

The concrete stress is taken from Figure 12-1:

$$\varepsilon_0 = 2\frac{f_c'}{E_c} = 2 * \frac{4}{3600} = 0.002222 \text{ in/in}$$

The strain, 0.002118 in/in is less than ε_0, therefore:

$$\sigma = \left[2\left(\frac{\varepsilon}{\varepsilon_0}\right) - \left(\frac{\varepsilon}{\varepsilon_0}\right)^2\right] f_c'$$

A5

$$\sigma = \left[2\left(\frac{0.002118}{0.002222}\right) - \left(\frac{0.002118}{0.002222}\right)^2 \right] * 4$$

$$\sigma = 3.991$$

Table A5-4 shows the area and compressive force associated with each of the 20 pieces of the compression block identified in Figure A5-2.

Table A5-4:
Force in each piece of the concrete compression block

Piece	Area in^2	Average Stress ksi	Force kips
1	2.0971	3.541	-7.43
2	2.0971	3.620	-7.59
3	2.0971	3.698	-7.76
4	2.0971	3.776	-7.92
5	2.0971	3.854	-8.08
6	2.0971	3.933	-8.25
7	2.0971	3.981	-8.35
8	2.0971	3.967	-8.32
9	2.0971	3.896	-8.17
10	2.0971	3.783	-7.93
11	2.0971	3.626	-7.61
12	2.0971	3.427	-7.19
13	2.0971	3.184	-6.68
14	2.0971	2.899	-6.08
15	2.0971	2.570	-5.39
16	2.0971	2.199	-4.61
17	2.0971	1.784	-3.74
18	2.0971	1.327	-2.78
19	2.0971	0.826	-1.73
20	2.0971	0.283	-0.59
		Sum	**-126.20**

Considering Piece 7 in Table A5-4, the area is calculated as:

$$\text{Area} = \left(\frac{1}{20} * 1.74764\right) * 24 = 2.0971\ \text{in}^2$$

where the term in parenthesis is the depth of the piece and 24 is the width of the section (piece).

The average stress is determined by averaging the stress at the top and the bottom of the piece.

Average Stress = 0.5 * (3.972 + 3.991) = 3.981 ksi

The force is obtained by multiplying the area times the average stress:

Force = 2.0971 * 3.981 = 8.35 kips

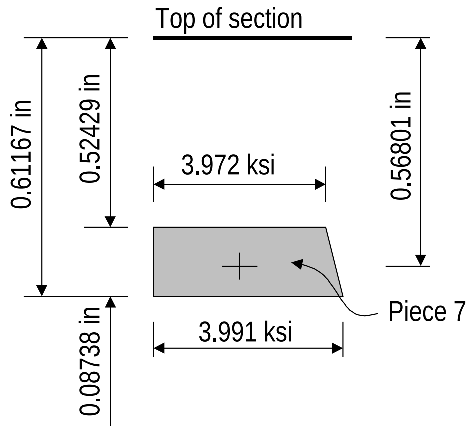
The negative sign in Table A5-4 indicates that the force is a compression force.

Table A5-5 shows how the contribution of the concrete compression to the section moment (about the section centroid) is calculated.

Table A5-5: *Section moment from the concrete compression block*

Piece	**Piece Centroid** in	**Distance to Section Centroid** ksi	**Moment** kip-in
1	0.04385	17.95615	133.4
2	0.13123	17.86877	135.6
3	0.21861	17.78139	137.9
4	0.30599	17.69401	140.1
5	0.39336	17.60664	142.3
6	0.48074	17.51926	144.5
7	0.56801	17.43199	145.6
8	0.65527	17.34473	144.3
9	0.74257	17.25743	141.0
10	0.82986	17.17014	136.2
11	0.91715	17.08285	129.9
12	1.00441	16.99559	122.1
13	1.09166	16.90834	112.9
14	1.17888	16.82112	102.3
15	1.26604	16.73396	90.2
16	1.35311	16.64689	76.8
17	1.44001	16.55999	62.0
18	1.52654	16.47346	45.8
19	1.61195	16.38805	28.4
20	1.68937	16.31063	9.7
		Sum	**2181.0**

Figure A5-3:
Distance from top of section to centroid of Piece 7



Considering Piece 7 in Table A5-5, the distance to the piece centroid measured from the top of the section is illustrated in Figure A5-3 and is calculated as:

Note:

See the section titled "Sign Convention for Axial Load and Moments" in Appendix 4 for discussion of the sign for the moments.

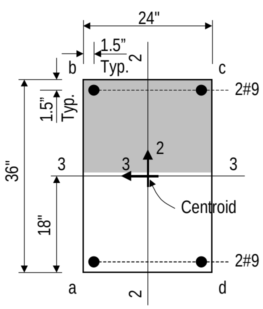
$$0.52429 + \frac{0.08738 * (2 * 3.991 + 3.972)}{3 * (3.991 + 3.972)}$$

$$= 0.56801 \text{ in}$$

The distance from the piece centroid to the section centroid is equal to 18 - 0.56801 = 17.43199 in.

The moment is equal to the force (from Table A5-4) times the distance to the section centroid.

Moment = 8.35 * 17.43199 = 145.6 k-in



That completes the consideration of the concrete compression block. Now we consider the contribution of the reinforcing steel.

The section is shown for reference in the sketch to the left. The corners of the section are labeled a, b, c and d. These labels are used to refer to each of the four bars in the section.

First we calculate the strain and the corresponding stress for each rebar. The strain is calculated using similar triangles and the stress is determined from Figure 12-1. Table A5-6 shows the strain and stress for each bar.

Table A5-6: *Strain and stress for each rebar*

Rebar	Distance from Top of Section in	Strain in/in	Stress ksi
a	34.5	0.061077	75.00
b	1.5	-0.000462	-13.39
c	1.5	-0.000462	-13.39
d	34.5	0.061077	75.00

Consider Bar b in Table A5-6. the strain is calculated as:

$$\text{Strain} = -0.003259 * \left(\frac{1.74764 - 1.5}{1.74764}\right)$$

$$\text{Strain} = -0.000462 \text{ in/in}$$

The negative sign is used to indicate that the bar is in compression.

The bar stress is taken from Figure 12-1. Noting that the yield strain, ε_y, is (60 ksi / 29000 ksi) = 0.00207 in/in, and the bar yield stress, f_y, is 60 ksi, the stress for Bar b is calculated as:

$$\text{Stress} = \frac{\varepsilon}{\varepsilon_y} f_y = \frac{-0.000462}{0.00207} * 60 = -13.39 \text{ ksi}$$

The bar force is calculated as the area times the stress. This is shown in Table A5-7.

Table A5-7: *Force in each rebar*

Rebar	Area in	Stress ksi	Force kips
a	1.00	75.00	75.00
b	1.00	-13.39	-13.39
c	1.00	-13.39	-13.39
d	1.00	75.00	75.00
		Sum	**123.22**

A5

The section moment due to the bar force is calculated by multiplying the bar force times the distance to the section centroid as shown in Table A5-8.

Table A5-8:
Moment caused by each rebar

Rebar	Distance to Section Centroid in	Force kips	Moment k-in
a	16.5	75.00	1237.5
b	-16.5	-13.39	220.9
c	-16.5	-13.39	220.9
d	16.5	75.00	1237.5
		Sum	**2916.8**

See the section titled "Sign Convention for Axial Load and Moments" in Appendix 4 for discussion of the sign for the moments.

Finally, we can deduct the force and moment for the concrete in compression that is displaced by the reinforcing steel in the compression area. The bars in the compression area are bars b and c.

Table A5-9 shows the concrete strain at Bars b and c. The stress shown in Table A5-9 is based on Figure 12-1.

Table A5-9:
Strain and stress for each location where concrete force must be deducted

Rebar	Distance from Top of Section in	Strain in/in	Stress ksi
b	1.5	0.000462	1.49
c	1.5	0.000462	1.49

Since the strain in Table A5-9, 0.000462 in/in, is less than ε_0 (0.002222 in/in), the stress is calculated as:

$$\sigma = \left[2\left(\frac{0.000462}{0.002222}\right) - \left(\frac{0.000462}{0.002222}\right)^2 \right] * 4$$

$$\sigma = 1.49 \text{ ksi}$$

The deducted force is calculated as the area times the stress. This is shown in Table A5-10.

Table A5-10:
Deducted force

Rebar	Area in	Stress ksi	Force kips
b	1.00	1.49	1.49
c	1.00	1.49	1.49
		Sum	**2.98**

The deducted moment is shown in Table A5-11.

Table A5-11:
Deducted moment

Rebar	Distance to Section Centroid in	Force kips	Moment k-in
b	-16.5	1.49	-24.6
c	-16.5	1.49	-24.6
		Sum	**-49.2**

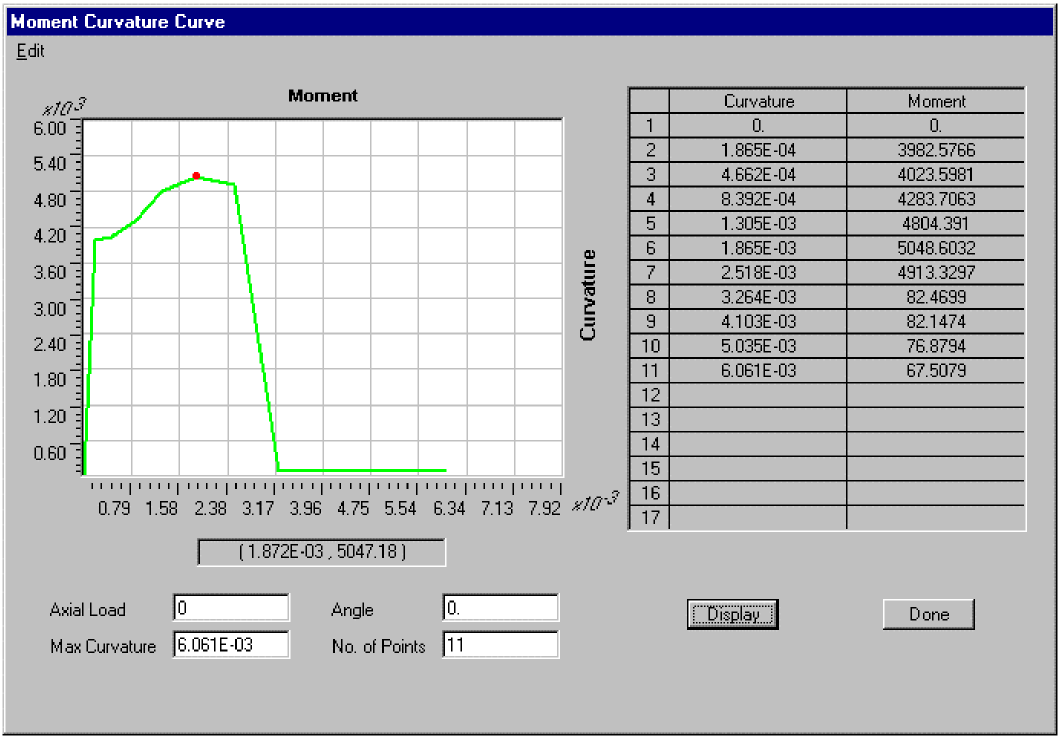
Table A5-12 sums up the various components of force and moment for the condition when the curvature is 0.0018648 in/in. and the concrete strain is 0.003259 in/in.

Table A5-12:
Sum of forces and moments

Item	Force kips	Moment k-in
Concrete compression	-126.20	2181.0
Bars a through d	123.22	2916.8
Concrete correction	2.98	-49.2
Sum	**0.00**	**5048.6**

Since the force is essentially equal to zero, our original target force, no further iteration is required. The final moment is 5048.6 k-in.

The section titled "How to Create this Example Problem" in Appendix 4 provides directions for creating this section in Section Designer. Once you have created the section you can click the **Display menu > Show Moment Curvature Curve** to display the moment curvature curve.



	Curvature	Moment
1	0.	0.
2	1.865E-04	3982.5766
3	4.662E-04	4023.5981
4	8.392E-04	4283.7063
5	1.305E-03	4804.391
6	1.865E-03	5048.6032
7	2.518E-03	4913.3297
8	3.264E-03	82.4699
9	4.103E-03	82.1474
10	5.035E-03	76.8794
11	6.061E-03	67.5079
12		
13		
14		
15		
16		
17		

Figure A5-4: *Moment curvature curve for example problem*

Figure A5-4 shows the moment curvature curve for this example problem.

A5

Index

I

E

F

G

I

K

L

M

N

O

P

Q

R

S

T

U

V

W

Z