Chapter

3

Working with Elements
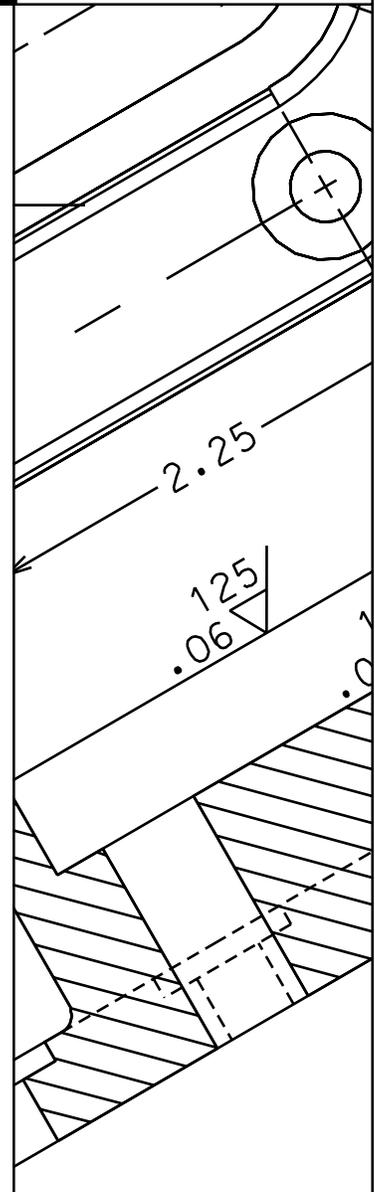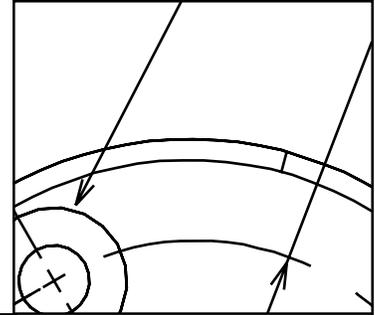
Visit the following websites to learn more about this book:

SDC Publications

amazon.com

BARNES&NOBLE

Introduction

The basic component of MicroStation's drawing structure is the drawing element. Elements are geometric shapes that can be assigned certain characteristics that distinquish it from other elements in the design file. Some characteristics are visual and others are less readily apparent.

Drawing Elements

Elements are single components of a drawing (i.e., line, arc, circle, etc.) and are the smallest item in a drawing organization that can be acted upon by the system (i.e. delete, scale, move, etc.). Some CAD systems refer to these as primitives. The elements include:
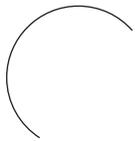
ARC
A continuous portion of a circle.
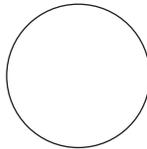
CIRCLE
A closed curve an equal distance from a center.
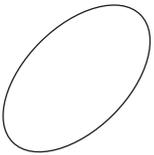
ELLIPSE
A closed curve defined by a center point and the radii of two axes. The radius of the first axis is called the primary and the other is referred to as the secondary.
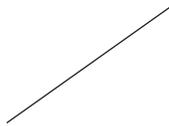
LINE
A line is a element defined by two endpoints. Although a "string" of lines can be drawn, each segment is an individual element.

POINT
An element that represents a single location in two- or three-dimensional space.
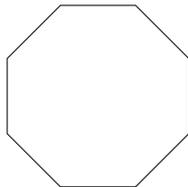
POLYGON (also called BLOCK and SHAPE) and COMPLEX SHAPES
A closed continuous series of interconnected lines. Although comprised of several segments, a polygon is a single element.

LINE STRINGS and COMPLEX CHAINS
Line strings and complex chains are an open continuous series of interconnected segments defined by more than two vertices or endpoints. Although it resembles a series of elements, the segments of a line string or complex chains represent a single element.
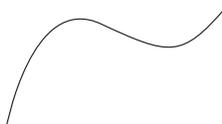
SPLINE
MicroStation offers several B-spline curve options. Although the actual methods are different, this element is a smooth curve defined by a set of control points.

Complex Chains and Shapes

Although a single line, arc, and point is rather easily identified, certain other elements are more difficult to assess. MicroStation uses line strings, complex chains, shapes, and complex shapes to define elements that appear to made up of multiple primitives, but are actually a single element or act like a single element. For example, a rectangular shape can be created in several different ways. One method is to draw four lines bounding the rectangular shape using the PLACE LINE tool. In this case, the rectangle is comprised of four line elements. Another approach is to draw the rectangle as a shape using the PLACE BLOCK tool. With a shape, the four sides of the rectangle are comprised of a single element rather than the four elements that result from using lines. Another approach is to draw four lines to form a rectangle and then combine those lines into a complex shape. A complex shape, much like a shape, treats the four lines as if they are a single element. Although they are treated as a single element, MicroStation refers to each component as a "segment." A rectangular complex shape would be a single element comprised of four line segments.

Shapes and *complex shapes* define elements that are closed. A shape is a single element. A complex shape is a single element comprised of multiple segments. *Line strings* and *complex chains* refer to geometry in which the segments form a single element, but are not closed. A line string forms a single element.

Line string and complex chain   Shape and complex shape
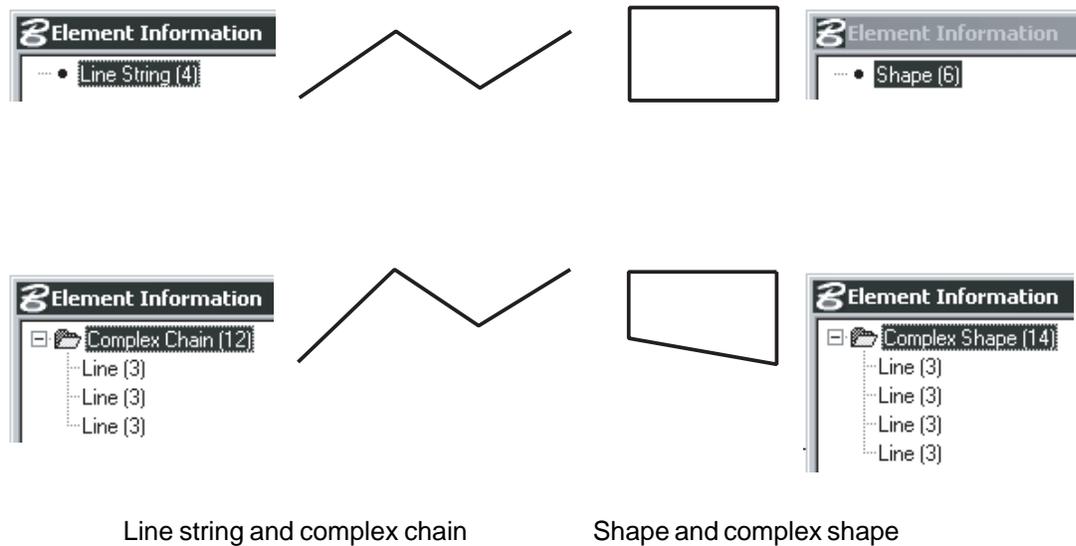
Figure 3-1. An illustration of string, complex chain, shape, and complex shape.

How objects are created, such as the rectangle, is related to how they later can be manipulated. For example, the rectangular shape created as four line elements can have any side of the rectangle selectively deleted. The rectangle, drawn as a single element shape or one that has been converted to a complex shape, cannot be partially deleted without first "dropping" it from a shape or complex shape to its individual elements. Remember, an element is the smallest item that can be acted upon by the system.

MicroStation does provide tools for taking individual elements and combining them into complex chains and shapes. They also provide tools for taking complex chains and shapes and making each segment an individual element.

Drop element   Create complex chain   Create complex shape   Create region
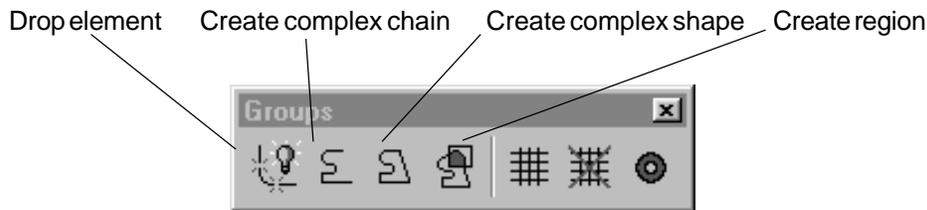
Figure 3-2. Groups tool box.

The DROP ELEMENT tool in the main tool frame is used to "break" elements into simpler components. The CREATE COMPLEX CHAINS tool is used to combine contiguous elements into a single, non-closed element. The CREATE COMPLEX SHAPE tool combines contiguous elements that form a closed shape into a single-element complex shape.

1.   Use **File > Open** to load the file *complex.dgn* (downloaded from web site).

2.   After the file is loaded, select the FIT VIEW [ ] tool from the view border to make sure the entire drawing is displayed. Cancel the command by clicking the right mouse button. Note the tool remains active until you click cancel (right mouse button). The design file should appear similar to the following illustration.
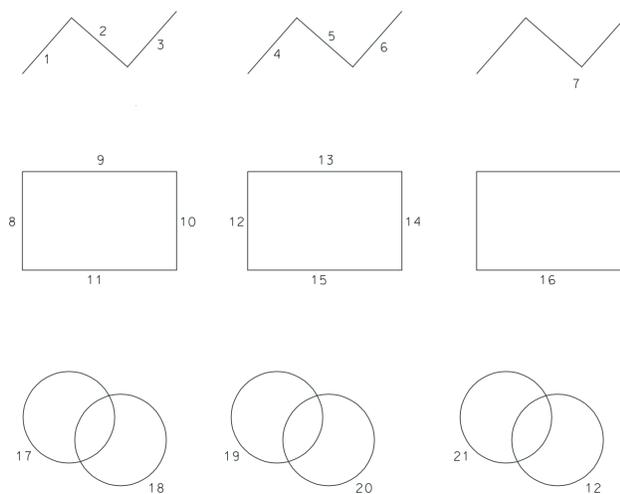
Figure 3-3. *Complex.dgn* design file.

The design file contains elements, complex chains, and complex shapes.

3.   Use the ELEMENT SELECTION [ ] tool and pick each numbered element to determine which are segments and which are independent elements. Each selected entity is highlighted and the status bar indicates the selection.

4.   Next, select the **Element > Information** menu option and pick item 7 and then item 14. Note the information that is displayed in the GENERAL tab of the displayed dialog. You can also select the ELEMENT INFORMATION [ ] tool from the primary toolbar.

Creating Chains and Shapes
Select the CREATE COMPLEX CHAIN tool from the MAIN toolbar.
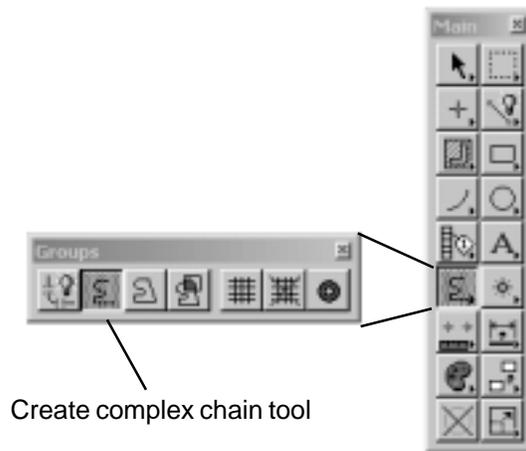
Figure 3-4. Create complex chain tool on the groups tool box.

If SINGLE CLICK in the TOOLS section of **Wor**k**space > **P**references** is set
to *single shot*, the tool is deselected when the command is completed. If
SINGLE CLICK is set to *locked*, the tool remains active until another tool has
been selected. If set to *single shot*, the tool can still be locked by double
clicking rather than single clicking the tool.

A tool settings box is displayed.

Figure 3-5. Create complex chain tool settings box.

5.   Set the METHOD in the create complex tool settings box to *Manual*.

Create Complex Chain > Identify Element

Data point (click left mouse button) on line segments 1, 2, and 3. Each element is
highlighted when selected. If you select the wrong element, press RESET (right mouse
button).

Create Complex Chain > Accept/Reject (select next input)

MicroStation needs to know when you are done selecting. To continue to select use the
DATA POINT button (left mouse button). To end the selection, click in an area of the
window where there are no elements to accept the current selection set. You would
click the RESET button (right mouse button) to reset the command.

6. After picking element 3, click a data point away from any element. Next, click a RESET to reset the command.

After completing the operation, this should result in a complex chain. You can use the **Element > Information** menu option to verify by selecting segment 1. The information display should show a complex chain.

7. Select the CREATE COMPLEX CHAIN tool again from the MAIN toolbar.

Figure 3-6. Using the simplify geometry option with create complex chain tool.

This time, change the METHOD to *Automatic* and check *Simplify geometry* to enable the option.

Create Complex Chain > Identify Element

Click (left mouse button) on line segment 4. Click any point and MicroStation automatically selects elements 5 and 6 since they form a coincident chain to line 4.

Create Complex Chain > Accept/Reject (select next input)

After the three elements are selected, click a DATA POINT away from an element or RESET to finish the selection and complete the complex chain.

With the *Simplify geometry* option enabled, the resulting geometry is a line string rather than a complex chain. Again, you can use the **Element > Information** menu option to check by selecting segment 4. This time, the information display should show a line string.

8. Select the CREATE COMPLEX SHAPE tool from the MAIN toolbar. Set the METHOD to *Manual* and disable the *Simplify geometry* option.

Create Complex Shape > Identify Element

Click (left mouse button) on line segments 8, 9, 10, and 11.

Create Complex Shape > Accept/Reject (select next input)

Click a DATA POINT away from any element and MicroStation creates a complex shape from the four line elements.

After completing the operation, you can use the **Element > Information** menu option to check by selecting segment 8. The information display should show a complex shape.

9. Repeat the complex shape command setting the METHOD to *Automatic* and enabling the *Simplify geometry* option. Select line 12 and then click a data point away from any element. This should automatically create a shape from line segments 12, 13, 14, and 15. You can check the results using the **Element > Information** menu option.

Drop Elements

10. Select the DROP ELEMENT tool from the MAIN toolbar.
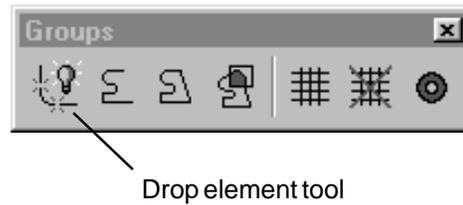
Drop element tool

Figure 3-7. Breaking element into components.

A tool settings box is displayed.
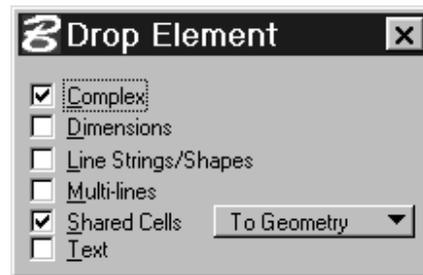
Figure 3-8. Drop element settings box.

The options on the settings box determines which items are broken into their component parts.

Item	*If enabled*
Complex	Cell, complex chain, complex shape, text node, surface, and solid are dropped to their component parts.
Dimensions	Breaks dimensions into lines, line strings, ellipses, arcs, and text.
Line Strings Shapes	Breaks line strings and shapes into individual line elements.
Multi-lines	Breaks multi-line elements into line strings, lines, and arcs.
Shared Cells	Shared cells are dropped to their components if set *to geometry*. Shared cells are dropped to unshared (normal) cells if set *to normal cells*.
Solids (3D)	Solids are dropped to either *wireframe* or *surfaces*.
Text	Text characters are converted to the line, line string, arc, ellipse, and shape elements used to create the character.

11. Check *Line Strings/Shapes* and then pick element 7. This drops the element from a line string to 3 line elements. Repeat the command and select element 16. This converts the shape to 4 line elements.

Working with Regions

Another useful tool is the CREATE REGION ⊞ tool. This tool allows you to manipulate two intersecting complex shapes. There are four options available.

Option	Action
Difference	Subtract one complex shape or shape from another.
Union	Combine two complex shapes or shapes into a single complex shape comprised of the total area of both.
Intersection	Combine two complex shapes or shapes into a single complex shape comprised of only the area common to both.
Flood	A variation on intersection that creates a single complex shape from elements that bound the area indicated.

　　Originals　　　　Difference　　　　Union　　　　Intersection
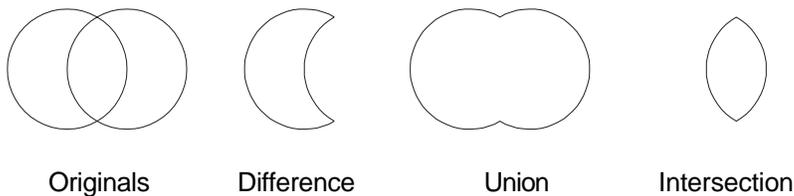
Figure 3-9. Create region options.

12. Select the *Difference* method.

> **Create Region From Element Difference > Identify element**

Select element 17.

> **Create Region From Element Difference > Accept/Reject (select next input)**

Select element 18. Click a data point away from any element and MicroStation subtracts element 18 from element 17. You may need to select the UPDATE VIEW tool in the border tools if the shape is not completely clear.

13. Choose the *Union* and *Intersection* methods and apply one to elements 19 and 20, and the other option to elements 21 and 22. The procedure is the same as was done for the *Difference* method.

Linear Elements

There are several tools for creating lines of varying appearance and use. These tools can be found on the LINEAR ELEMENTS toolbar.

SmartLine　　Line　　　Multi-line
Tool　　　　Tool　　　Tool
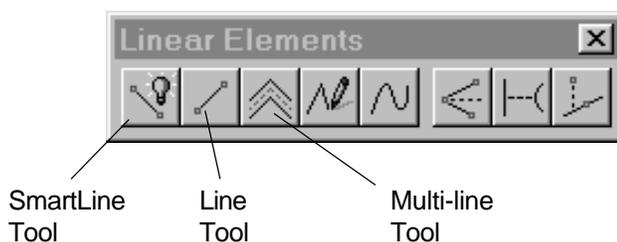
Figure 3-10. Line tools in the linear elements toolbar.

The PLACE LINE [icon] tool is used to place individual line elements. If the tool is in the locked mode, it places several contiguous lines with each segment being a separate line element. The line tool settings box provides options to constrain the line length and angle in which the line is placed.
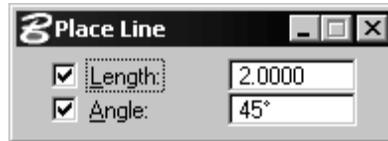
Figure 3-11. Place line dialog with length and angle constrained.

In the above example, the line would be 2 units in length and positioned at a 45 degree angle from the data point signifying the start of the line.

Line Style

MicroStation allows you to draw in one of several standard line styles or by defining a custom line style. The standard line styles and several predefined custom line styles are selected from the ACTIVE LINE STYLE list box in the ATTRIBUTES toolbar.
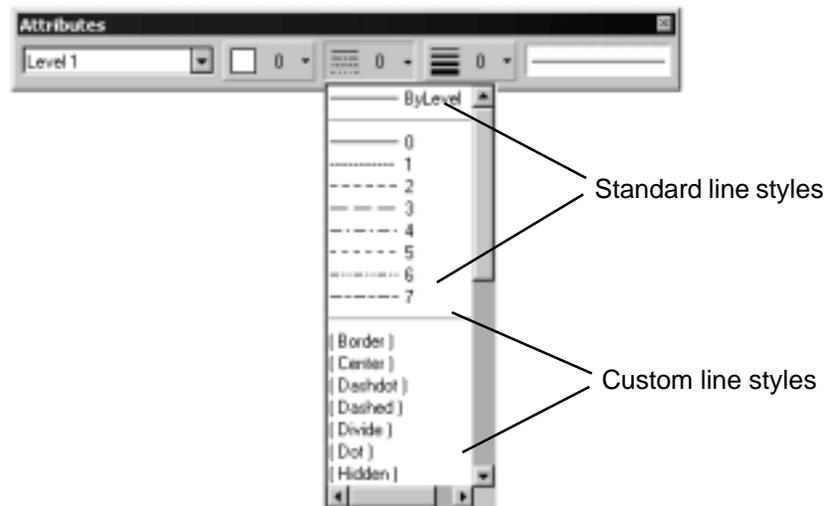
Figure 3-12. Selecting the active line style from the attributes toolbar.

There are eight standard line styles numbered 0-7. The active line style is applied to any elements added to the file. It is also applied to any elements that are selected with the active line style is changed.

The following illustrates some of the custom line styles provided with MicroStation.
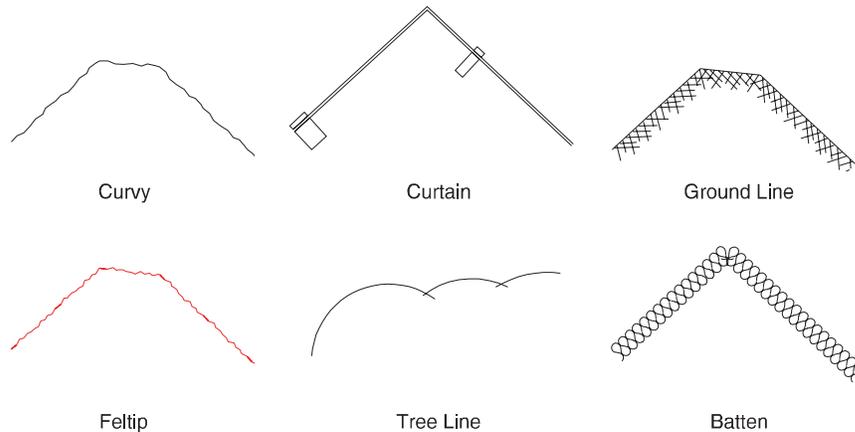
Figure 3-13. Sample custom line styles available with MicroStation.

Line styles are stored as a *line style definition* comprised of *stroke patterns* and *point symbols* that can be customized. Line styles can also contain *line style modifiers* which allow you to adjust the placement of the line style without creating a new line style definition. The modifiers allow you to change the width of the start and end of a line, shift the pattern relative to the start point, and scale the line style pattern by a factor.

A custom line style is selected from the ACTIVE LINE STYLE list box in the ATTRIBUTES toolbar or by selecting the **Element > Line Style > Custom** option.

Figure 3-14. Selecting a custom line style from attributes toolbar and line style dialog.

The Line Style list box remembers the four most used line styles and lists those styles above the custom line styles.

The **Element > Line Style > Custom** option displays a dialog with the line style modifiers. You can change the modifiers and the results are previewed in the dialog box when you click on the line style preview to activate the change.
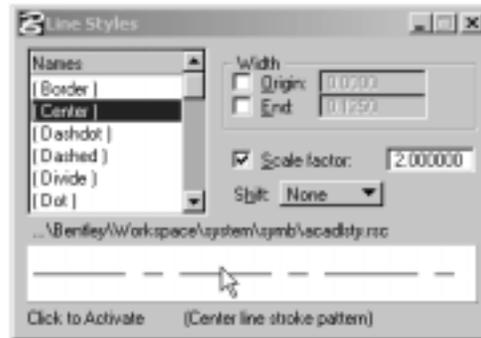
Figure 3-15. Activating modifiers for a custom line style.

In the above example, the *Center* custom line style pattern would be scaled by a factor of 2. If you had an element selected when the change was made, it would be applied to the element. Note that the change may not be visible on the displayed element at the current view scale. Once you have explored several line styles and modifiers, return the active setting to the standard solid line style (style number 0).

Line styles are stored in a library resource file. You can view the resource file or create a new resource file by selecting the **Element > Line Style > Edit** option. You then select **File > Open**. The default line style library is *lstyle.rsc*.

Even though the style may be elaborate, the line style is applied to a line or line string and is treated by the system as a line element or complex chain.

> Note that the standard line styles (those number 0-7) are hardware dependent; therefore, their appearance is dependent on the output device. In other words, what you see on the display may be significantly different than the printed output. Custom line styles on the other hand operate in a more WYSIWYG (what-you-see-is-what-you-get) mode.

SmartLine

The PLACE SMARTLINE tool is used in a manner similar to PLACE LINE, but each segment is either an independent element or it is part of a complex chain (open) or complex shape (closed). In addition to placing lines, SMARTLINE also allows you to place arcs. In fact, you can switch between arcs and lines as you create the elements, the complex chain, or the complex shape.

Figure 3-16. SmartLine tool.

There are several options available when using the SmartLine tool:

Line segments
Sharp vertex

Line segments
Rounded vertex

Line segments
Chamfered vertex

Arc segments
Sharp vertex

Arc segments
Rounded vertex
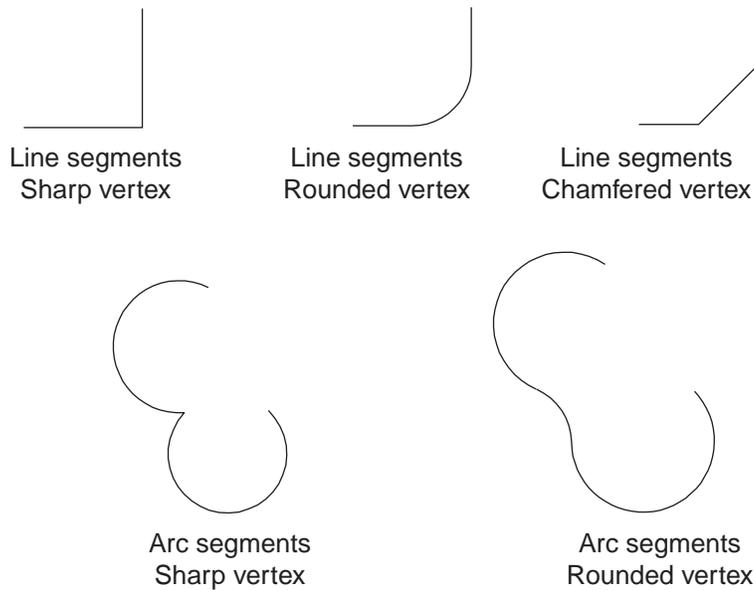
Figure 3-17. SmartLine tool options.

The following describes the options:

Segment Type	*Lines*	Used to draw line segments.
	Arcs	Used to draw arc segments.
Vector Type	*Sharp*	A single vertex point is formed between adjacent segments.
	Rounded	A rounded arc is placed between adjacent segments equal to the *rounding radius.*
	Chamfered	A chamfer is placed between adjacent line segments with an offset on each segment equal to the *chamfer offset.*
Rounding Radius		Arc radius when using rounded vector type.
Chamfer Offset		The distance along each segment that is offset when chamfering a vertex.
Join elements		If enabled, the segments form a complex chain or complex shape.

A drawing can be used to illustrate the use of the Smartline tool.
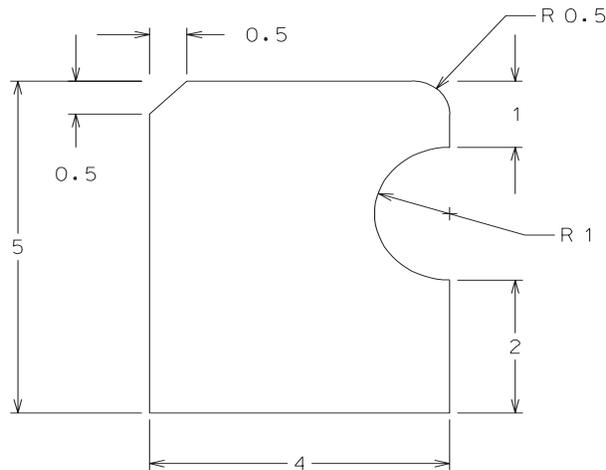
Figure 3-18. SmartLine drawing.

1. Start a new drawing by selecting **File > New**. Name the file *lines*. When prompted to save changes to *complex.dgn*, select NO. Make sure AccuDraw is active and is displaying a rectangular rather than a circular compass. You toggle between the two compasses by pressing the SPACE BAR.

2. Select the SMARTLINE tool and set the following:

Segment Type	**Lines**
Vertex Type	**Sharp**
Join Elements	(do not enable)

    Click in the lower left of the window to begin the line.

    Place SmartLine > Enter next vertex or reset to complete

    Move to the right along the x-axis and enter a value of **4**. Make sure AccuDraw remains on the x-axis (horizontal). Click a data point (left mouse button) to accept that point.

3. Next, move the cursor directly up along the y-axis (vertical). Enter a value of **2** and click a data point to accept.

    At this point, switch to a SEGMENT TYPE of *Arc*. Again, move the cursor directly up. Since the AccuDraw compass has rotated relative to the last line segment entered, this direction (vertical) is relative to the x-axis in the AccuDraw dialog. Enter a value of **.5** which is the radius of the arc. Click a data point to accept that value.

    Move the cursor in a counter clockwise direction to establish the arc on the inside of the previous line. Continue to move until the AccuDraw angle displays **180**. Notice that AccuDraw automatically changes from a rectangular to a circular compass when specifying an arc. Click a data point to accept that angle.

4. Change the SEGMENT TYPE back to *Line*. Press the SPACE BAR to change to a rectangular compass. Again, move directly up. This time it is along the y-axis. Enter a value of **1** and click a data point to accept that point.

If at any point a portion of the drawing is outside the view area, click the FIT

VIEW tool in the view border. After the drawing is zoomed, cancel the Fit

View tool by pressing reset once. Make sure you press the right mouse button
only once to cancel Fit View. If you click reset a second time you cancel the
Place SmartLine command.

---

5. Keep the SEGMENT TYPE set to *Line* and change the VERTEX TYPE to *Rounded.* Enter
a ROUNDING RADIUS of **.5**. Once you have entered the ROUNDING RADIUS, move the
cursor directly to the left. Because AccuDraw rotated the compass relative to the last
entry, moving to the left is along the y-axis. Enter a value of **3** and click a data point to
accept that position. Notice the previous vertex is rounded to a radius of .5.

Note that you can use **Edit > Undo**, CTRL -Z, or the UNDO tool in the
STANDARD toolbar if you make an error entering a point. UNDO acts upon the
last point if selected during the execution of a command.

---

6. Keep the SEGMENT TYPE set to *Line* and change the VERTEX TYPE to *Chamfered.*
Enter a CHAMFER OFFSET of **.5**. Once you have entered the CHAMFER OFFSET,
tentative to the starting point of the shape. Change the VERTEX TYPE to *Sharp* and
click a data point to accept the position. Notice the previous vertex is chamfered with
an offset of .5 on each line segment and the end of the shape has a sharp vertex.

Multi-lines

The MULTI-LINE tool creates a line which appears to be comprised of several ele-
ments, but remains a single line element. The multi-line tool allows you to place walls or
similar features by drawing a single line. The multi-line can be comprised of several compo-
nents having different styles, weights, and colors. The appearance of the multi-line is set
using the **Element > Multi-lines** command.
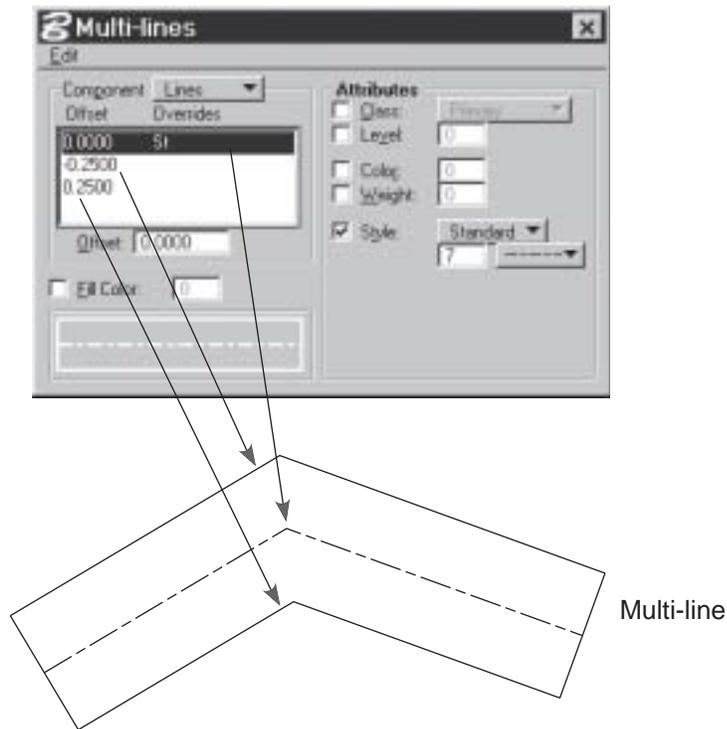
Figure 3-19. The multi-line definition dialog box.

The multi-line is comprised of a series of components - line segments, start cap, end cap, and joints.
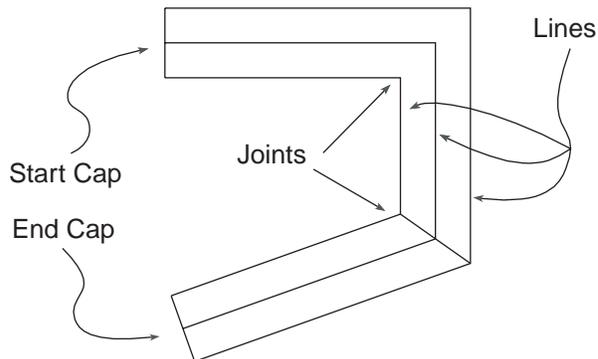
Figure 3-20. Multi-line caps and joints.

Line segments are offset by a prescribed distance. Each line segment can be assigned attributes independent of the other line segments. The COMPONENT pull-down list provides options to modify each of the components.

Using Multi-line Tools

Creating a drawing best illustrates how to use the multi-line tools.

1. Load MicroStation and make sure the MicroStation Manager is displayed. If it is not displayed, select **File > New** from the menu bar.

Finished Drawing

The example below illustrates the finished drawing.
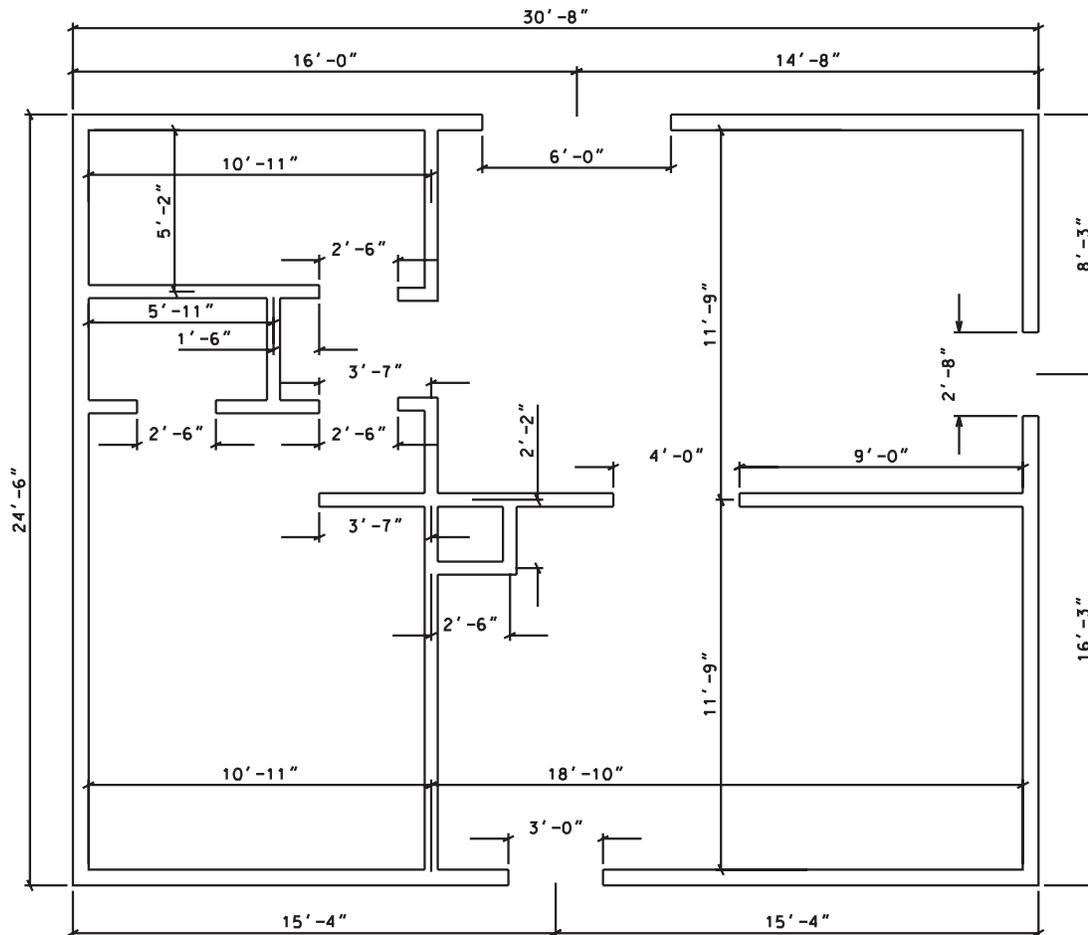
Figure 3-21. Finished multi-line drawing.

Create a New Drawing

2. Determine the location of the new file by specifying the *Drives* and *Directories* parameters. Name the new file **cape.dgn** and use the *seed2d.dgn* seed file. The file extension .DGN is automatically appended. If the *seed2d.dgn* is not the current seed file, click on SELECT and choose that file. Remember, this is a 2D drawing, so it is important to select a 2D seed file.

3. Once you have specified all the necessary information, select **OK**.

4. At this point, notice that the file has been created but not loaded. To load the file, you must have it highlighted in MicroStation Manager and then select **OK**.

5. Close *Window 2* using the **Window > View** command.

Basic Drawing Setup

CAD systems generally draw in real world units. In other words, geometry is drawn at its actual size. The first step is to establish the units of entry that you want to use with the drawing. Naturally, the unit of entry must match the nature and extent of the item being drawn.

6. Establish the unit of measurement for this drawing by selecting **Settings > Design File**.

7. Select the *Working units* category and set the following:

> **Master Units**   **feet**
> **Sub Units**   **inches**

Do not select OK at this point.

---

8. While the Design File Settings are displayed, select the *Grid* category. This is used to establish the spacing for the drawing grid. A grid is a drawing aid that does not print. Set the following:

> **Grid lock**   (do not check)
> **Grid Master**   1
> **Grid Reference**   10
> **Grid Config**   Ortho
> **Grid Aspect**   1

The grid master sets the points on the grid to be spaced at 1 foot (master units) intervals. The grid master sets a larger marker every 10 units of the grid; in this case every 10 feet. The grid configuration is set to a standard orthochromatic (parallel to view) grid. And the aspect sets the grid spacing equal both horizontally and vertically. The grid is displayed to assist in drawing, but is not output when printing or plotting.

9. Select **OK**.

10. Click the ZOOM OUT ▬ tool in the view border twice. Click a reset to cancel the tool. While drawing, you may need to use the ZOOM OUT tool, ZOOM IN ✚ tool, and the FIT VIEW ▲ tool to adjust the view of the design file. These tools can be selected at anytime while completing a command. Once selected, they must be cancelled before continuing with the previous command. Carefully watch the prompting displayed on the status bar to determine the required operation.

11. Select **File > Save Settings** to save the modifications made to the settings. Those changes are then saved with the design file. You can use the *Operation* CATEGORY in **Workspace > Preferences** to set MicroStation to automatically save settings when exiting.

Setup of Multi-line Tool

Before using the Multi-line tool it is necessary to set it up according to the type of line you plan to draw and the current working units.

12. Select the **Element > Multi-lines** option. The multi-lines settings box is displayed.

From the settings box, you determine the appearance of the line, start cap, end cap, and joints components of the multi-line.

The appearance of the multi-line for this activity is comprised of three lines segments. It includes both a start and end cap. The joints are not displayed.
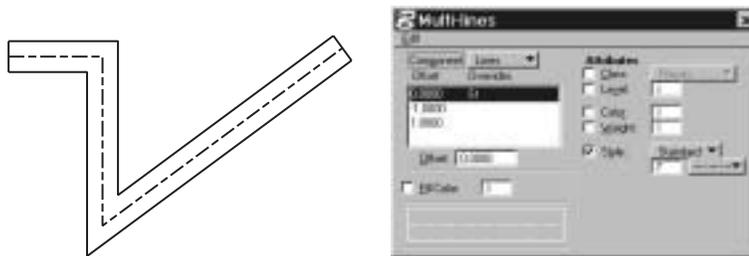
Figure 3-22. Appearance of multi-line settings.

13. In the COMPONENT pull-down, select *Line*.

The appearance of the line is displayed at the bottom of the settings box. The multi-line for this activity is comprised of three lines segments. If there are fewer than three lines, highlight one of the lines and select **Edit > Delete** in the settings box. If there are more than three lines, select **Edit > Insert** to add lines.

Each line is positioned relative to a zero (0) offset line. The outer walls of the floor plan are 6" in width. Set up three lines with the center line at 0, the inside of the line at -3" and the outside line at 3". If expressed in master units, the distances are .25' and -.25' respectively.
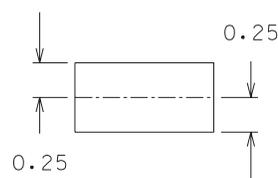
Figure 3-23. Appearance of lines for multi-line.

14. Select **Settings > View Attributes** and check to make sure CONSTRUCTION is enabled. Click the ALL button. Since the center line is a construction class element, it will not display unless the option is enabled in view attributes.

15. Pick each line and set the OFFSET in the multi-line settings box. If you enter inches, type a colon to indicated the measurement is in sub units rather than master units. You can either enter **.25** or **:3** for the outside of the wall and **-.25** or **-:3** for the inside.

16. Pick the center line (offset is 0) and change the attributes to:

Class	**Construction**
Style	**Custom**

Pick the SELECT LINE STYLE and pick the CENTER custom line style. The construction class allows the center line to display, but not print or plot.
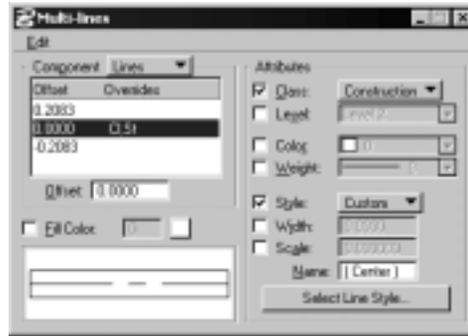
Figure 3-24. Setting lines for multi-line.

You can close the multi-line settings box.

17. Select the PLACE MULTI-LINE tool. Use this tool to draw the outside walls of the floor plan.

    Set the PLACE BY to *Maximum*. There are four options: *work line*, *center*, *maximum*, and *minimum*. In this case, the center and workline are the same. The maximum represents the outside of the wall and the minimum would be the inside.

18 Make sure AccuDraw is active. If it is not, select the ACCUDRAW tool. Also, make

    sure the AccuSnap is enabled. To check, click the ACTIVE SNAP MODE tool on the
    status bar. You can also access the settings using the **Settings > Snap > AccuSnap**
    menu option. In the GENERAL tab, enable the *Enable AccuSnap, Show Tentative Hint,*
    Identify Elements Automatically, and *Pop-up Info Automatic* functions. Disable the
    Display Icon function. If enabled, this function displays an icon indicating the type of
    currently active snap mode.
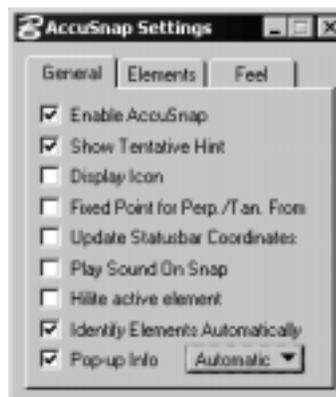
Figure 3-25. AccuSnap settings dialog.

AccuSnap provides a drawing aid that automatically inputs tentative snap points. Snap points are used to precisely locate a position relative to existing elements. For example, a position at the end of a line or the center of circle.

When AccuSnap is enabled, a snap point symbol is displayed when you move the pointer over a snap point. The symbol indicates you are near enough to the desired snap point to enter a data point. If the data point is entered while the symbol is displayed, it automatically locates the position on the snap point.

Pointer

Snap point symbol
at endpoint of line

Snap point symbol
at center of circle
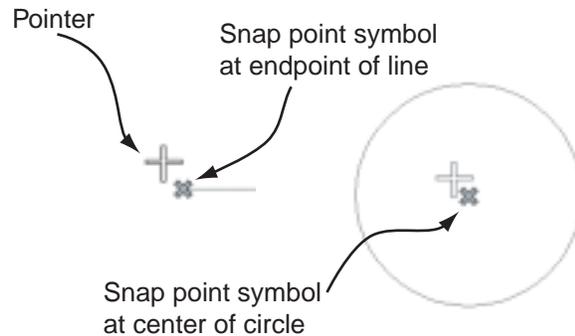
Figure 3-26. AccuSnap display of snap points.

How close the cursor needs to be to a snap point is controlled by two settings. The first is the *Snap tolerance* setting found in the FEEL tab of the AccuSnap settings dialog which adjusts how close the pointer must be to an element in order to snap a tentative point to it. The second uses the **Wor**k**space > **P**reference** menu option to access the *Locate Tolerance* user preference found in the INPUT category. The tolerance value is set in screen resolution (pixels) and determines the size of the location zone surrounding the pointer..

Once you have made the settings, close the AccuSnap Settings box.

19. Click in the lower left corner of the window to begin the drawing. Move the cursor up along the vertical axis and enter **24:6**. Click to accept that point. Note that the colon indicates you entered 24 feet and 6 inches since the colon indicates the following value is in sub units. The entry format is master units:sub units. You may need to select the ZOOM OUT ▬ tool several times to see the line. If you selected the ZOOM OUT tool, you need to click a reset to cancel the tool.

20. Move the cursor the right along the horizontal axis. Make sure AccuDraw indexes to that axis and type **30:8**. Click to accept that point.

21. Move the cursor down directly along the vertical axis and type **24:6**. Click to accept that point.

22. Close the shape by clicking CLOSE ELEMENT in the settings box. Click the FIT VIEW ▲ tool to see the entire drawing. Cancel the fit view tool.

Figure 3-27. Outer wall of floor plan.

Before drawing the inside walls, you can change the wall thickness from 6" to 5".

23. Select **Element > Multi-lines**. The multi-lines settings box is displayed.

24. First, pick the outside line indicated as .25 (feet). Change the OFFSET in the multi-line settings box for this line to **:2.5** (inches). Next, pick the inside line indicated as -.25 and change the OFFSET of that line to **-:2.5**. Make sure you enter a negative value and the colon to indicated inches (sub units). The settings box displays the values in master units (.2083 and -.2083).

You can close the multi-line settings box.

25. Select the PLACE MULTI-LINE tool and set the PLACE BY to *Center*.

26. Make sure AccuDraw is active. Tentative to the inside corner of the wall in the upper left side of the drawing. The AccuSnap symbol should display at the location when the pointer is positioned near the corner.
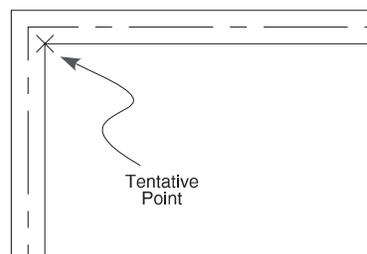
Figure 3-28. Tentative point on outer wall of floor plan.

With AccuSnap enabled, you can either do a tentative on the location and then a data point or you may data point directly without a tentative when AccuSnap provides an indication that you are at the desired location.

27. If you successfully located the tentative point, enter the letter **O** to set the AccuDraw origin at the tentative position. Move the cursor down along the vertical axis and enter **5:2**. Click to accept that point. You may need to select the ZOOM IN [+] tool to make it easier to locate the tentative point. If you selected the ZOOM IN tool, you need to click a reset to cancel the tool. Make sure you do not click reset twice or you will cancel the multi-line command.

28. Move the cursor right along the horizontal axis. Make sure AccuDraw indexes to that axis and type **10:11**. Click to accept the point.

29. Move the cursor up along the vertical axis. Make sure AccuDraw indexes to that axis and type **5:2**. Click to accept the point
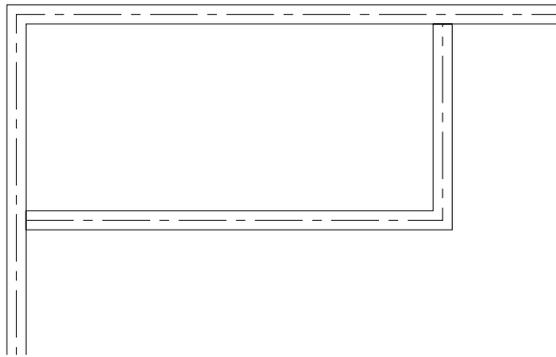
Figure 3-29. Interior wall of floor plan.

30. Click a reset to cancel the multi-line tool.

31. Select the PLACE MULTI-LINE [icon] tool and set the PLACE BY to *Center*.

32. Tentative to the far left center of the line you just completed.
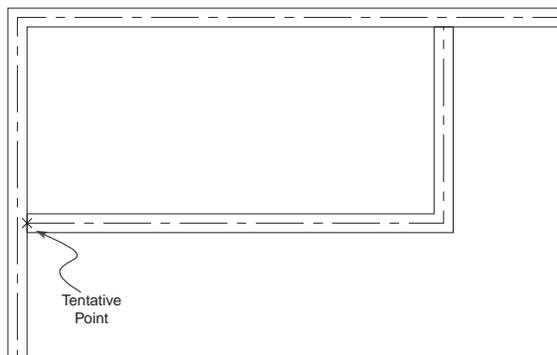
Tentative
Point

Figure 3-30. Tentative point on outer wall of floor plan.

33. If you successfully located the tentative point, enter the letter **O** to set the AccuDraw origin at this position. Move the cursor down along the vertical axis and enter **3:8**. Click to accept the point.

34. Move the cursor right along the horizontal axis. Make sure AccuDraw indexes to that axis and type **10:11**. Click to accept the point.

35. Move the cursor down along the vertical axis. Make sure AccuDraw indexes to that axis and type **14:9**. Click to accept the point
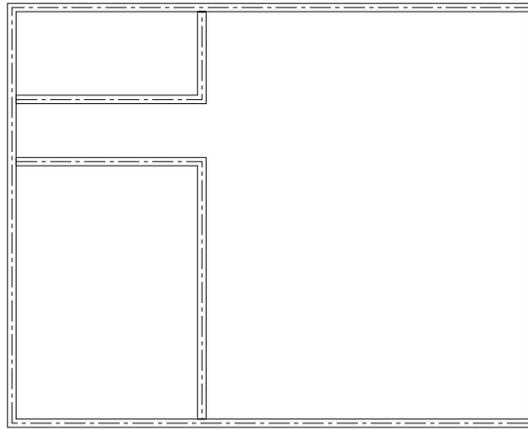
Figure 3-31. Position of second interior wall of floor plan.

36. Click a reset to cancel the multi-line tool.

37. Select the PLACE MULTI-LINE [icon] tool and set the PLACE BY to *Center*.

38. Tentative to the inside corner at the lower left of the outside wall.
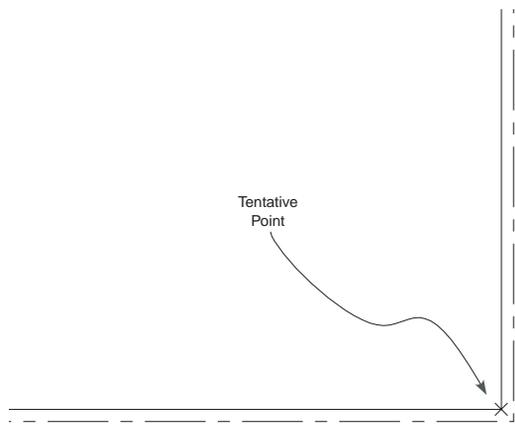
Tentative
Point

Figure 3-32. Tentative point at lower left corner of outside wall.

39. If you successfully located the tentative point, enter the letter **O** to set the AccuDraw origin at this position. Move the cursor up the vertical axis and enter **11:9**. Click to accept that point.

40. Move the cursor to the left along the horizontal axis. Make sure AccuDraw indexes to that axis and type **22:10**. Click to accept the point.

41. Click a reset to cancel the multi-line tool.
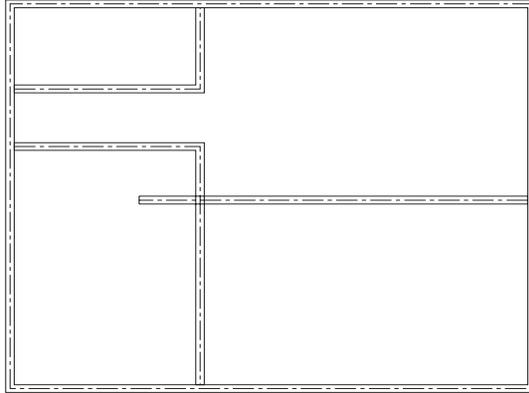
Figure 3.33. Appearance of interior walls of floor plan.

42. Select the PLACE MULTI-LINE ![icon] tool and set the PLACE BY to *Center*.

43. Tentative to the position indicated in the following illustration.
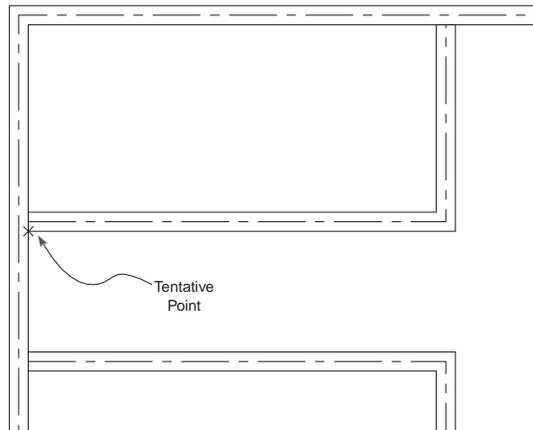
Figure 3-34. Tentative point for interior wall of floor plan.

44. If you successfully located the tentative point, enter the letter **O** to set the AccuDraw origin at this position. Move the cursor to the right along the horizontal axis and enter **5:11**. Click to accept the point.

45. Move the cursor down along the vertical axis. Make sure AccuDraw indexes to that axis and type **3:3**. Click to accept the point
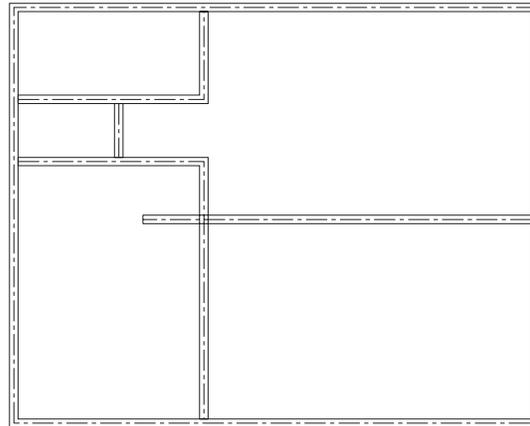
Figure 3-35. Appearance of interior walls of floor plan.

46. Click the ACTIVE SNAP MODE [icon] tool on the status bar and access ACCUSNAP. In the GENERAL tab, check to enable the *Display Icon* function. This function displays an icon indicating the type of currently active snap mode. Close the AccuSnap settings box.

47. Select the PLACE MULTI-LINE [icon] tool and set the PLACE BY to *Center*.

48. Select the INTERSECTION snap from the ACTIVE SNAP MODE [icon] tool on the status bar. Move the pointer to the intersection of the two center lines indicated in the illustration below. Once the intersection is located, tentative to that point. Notice that an icon is displayed next to the pointer indicating the current snap mode is set to intersection [icon].
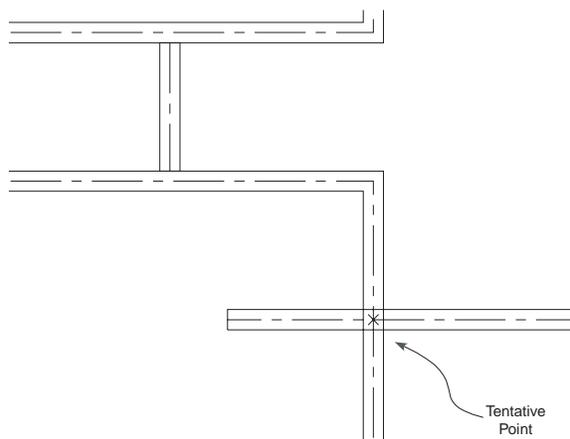
Figure 3-36. Tentative point for interior wall of floor plan.

49. If you successfully located the tentative point, enter the letter **O** to set the AccuDraw origin at this position. Move the cursor to down along the vertical axis and enter **2:2**. Click to accept the point.

50. Move the cursor right along the horizontal axis and enter **2:6**. Click to accept the point.

51. Move the cursor directly up along the vertical axis. Make sure AccuDraw indexes to that axis and type **2**. Click to accept that point
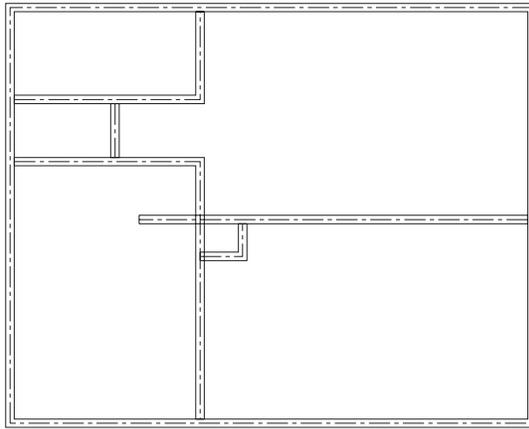
Figure 3-37. Appearance of interior and exterior walls.

Multi-Line Joints

The multi-line joint tools allow you to construct a variety of joints and openings in lines created with the multi-line tool.

Construct Closed Cross Joint

Construct Open Cross Joint

Construct Merged Cross Joint

Cut Single Component Line

Cut All Component Lines

Construct Closed Tee Joint

Construct Open Tee Joint

Construct Merged Tee Joint

Construct Corner Joint

Uncut Component Lines

Multi-line Partial Delete

Move Multi-line Profile

Edit Multi-line Cap
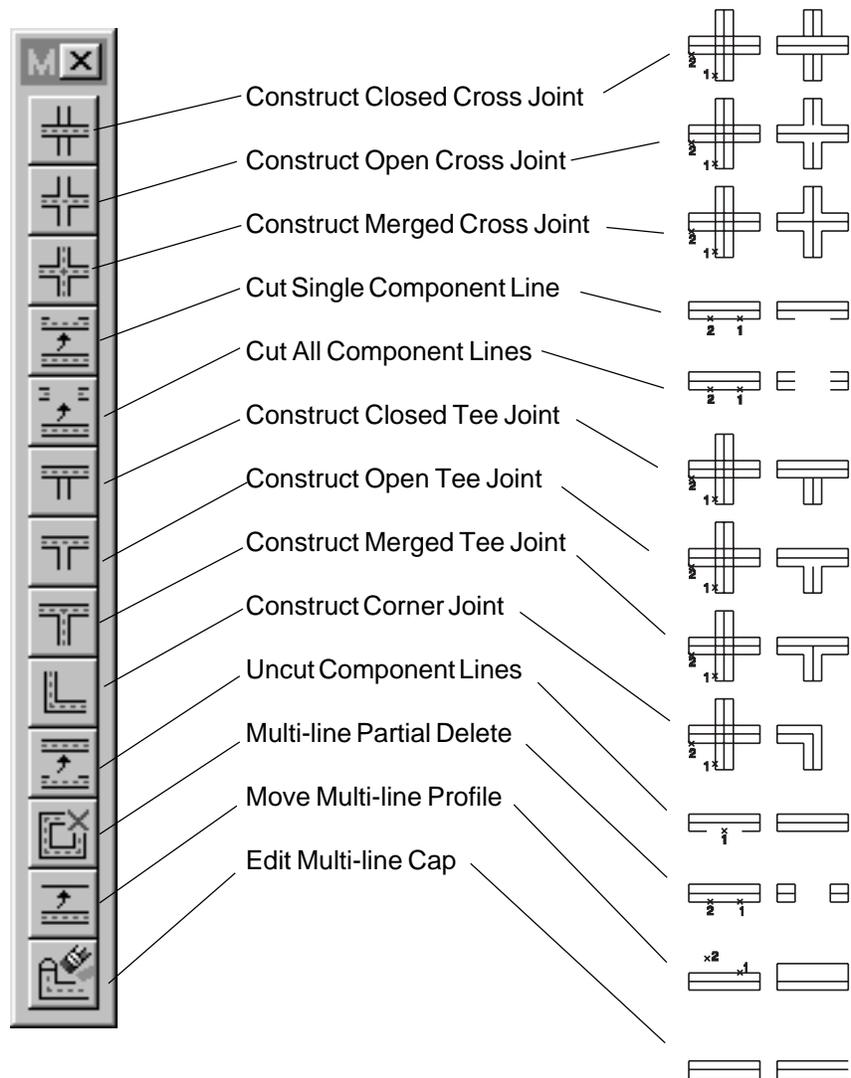
Figure 3-38. Multi-line joint tools.

1.   Select the **Tools > Multi-line Joints** option.

2.   Select the CONSTRUCT MERGED TEE JOINT [icon] tool from the MULTI-LINE JOINTS tool box. You may want to view the tool description on the status bar since the multi-line joint tools look very similar.

Once you have selected the tool, click the multi-lines indicated in the following illustration to change the joints formed between the various multi-lines.
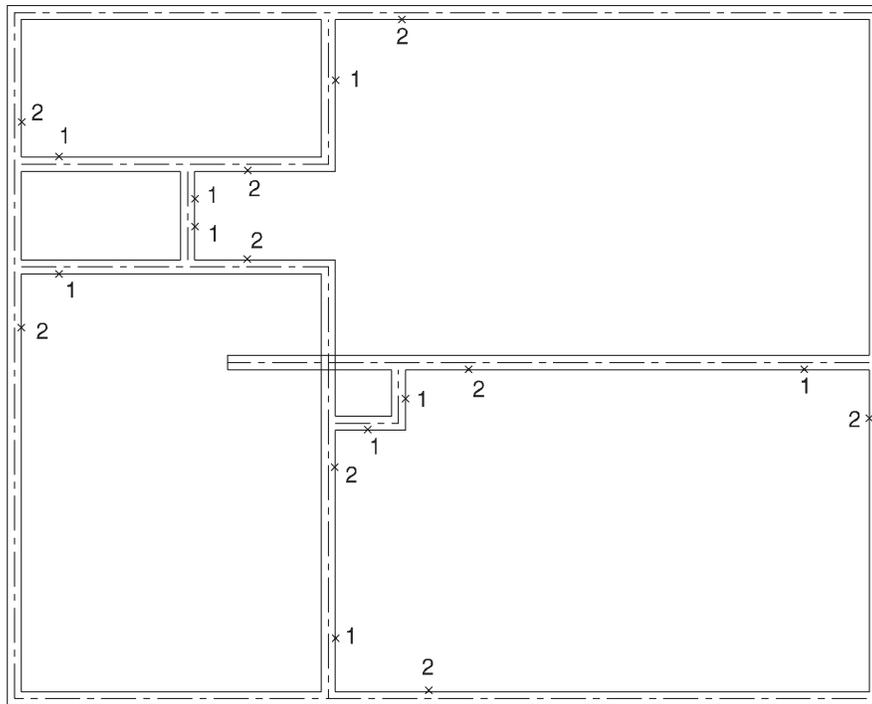
Figure 3-39. Multi-line tee joints.

3.   Next, select the CONSTRUCT MERGED CROSS JOINT ⊕ tool from the MULTI-LINE JOINTS tool box. Click the crossing lines indicated to form a cross joint.
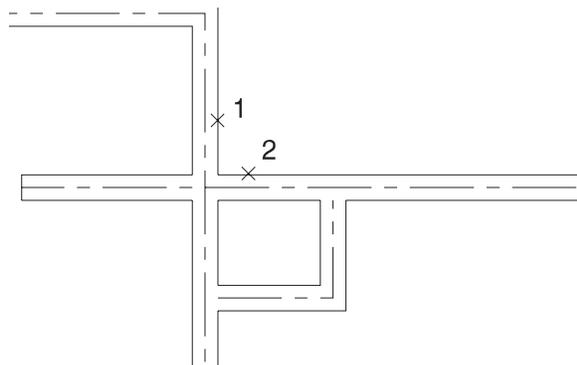
Figure 3-40. Multi-line cross joint.

This completes the joints needed to merge the multi-lines. The next step is to create openings for doors and windows. In this example, only simple door openings are created.

4.   Select the MULTI-LINE PARTIAL DELETE 🔲 tool from the MULTI-LINE JOINTS tool box. A tool settings box is displayed.

5. Set the CAP MODE to *Active*. The active mode is the one currently set by the **Element > Multi-lines** command. Use that command to check to make sure the *Start Cap* and *End Cap* COMPONENTS are both set to draw a line.

<div align="center">

Multi-line Partial Delete > Identify multi-line at start of delete

</div>

The prompt indicates that you are to identify the multi-line element at the point the partial delete is to begin.

6. On the far right side of the drawing, tentative to the point indicated in the illustration below.
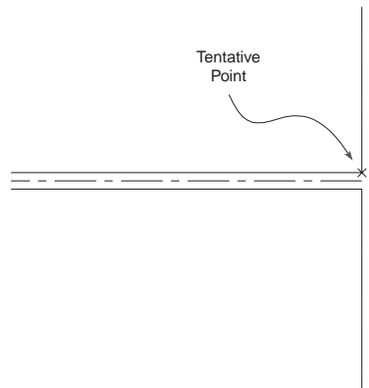
Figure 3-41. Tentative point for beginning the partial delete

7. If you successfully located the tentative point, enter the letter **O** to set the AccuDraw origin at this position. Move the cursor left along the horizontal axis and enter **9**. Click to accept the point. This represents the start of the delete.

8. Move the cursor left along the horizontal axis and enter **4**. Click to accept the point. This represents the end of the cut created by the partial delete. An end cap should appear on the multi-line.

9. Again, select the MULTI-LINE PARTIAL DELETE [icon] tool.

10. Tentative to the midpoint of the line indicated.
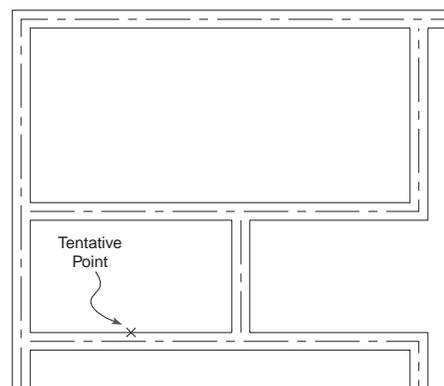
Figure 3-42. Tentative point at the midpoint of the wall.

11. If you successfully located the tentative point, enter the letter **O** to set the AccuDraw origin at this position. Move the cursor left along the horizontal axis and enter **1:3** (half the width of the door opening). Click to accept the point.

12. Move the cursor left along the horizontal axis and enter **2:6**. Click to accept the point. This represents the other end of the door opening.
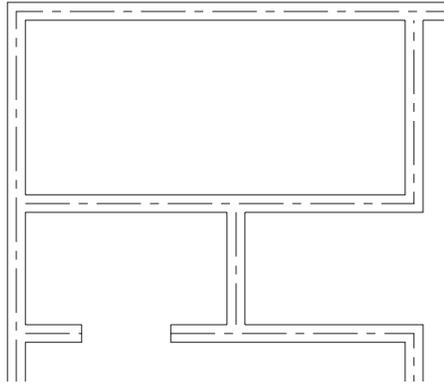
Figure 3.43. Door opening centered on wall section.

13. Complete the remaining door openings where indicated using the MULTI-LINE PARTIAL DELETE 🔲 tool. If the position of the opening is not indicated, assume it is centered on the wall section. The complete drawing is shown in Figure 3-21.

After you have completed the openings in the wall, the visibility of the center line can be disabled by selecting **Settings > View Attributes** and removing the check from CONSTRUCTIONS. If you remember, the multi-line definition defined the center line as a construction element rather than a primary element. The view attributes allow you to disable the display of construction elements.

The center lines are valuable when dimensioning wall sections that typically dimension from the center of the wall. Once the dimensions lines have been placed, the construction lines used to help position the dimension lines can be disabled.

Selection Tool

Before you begin to look at selection tools, load the drawing *selection.dgn*.
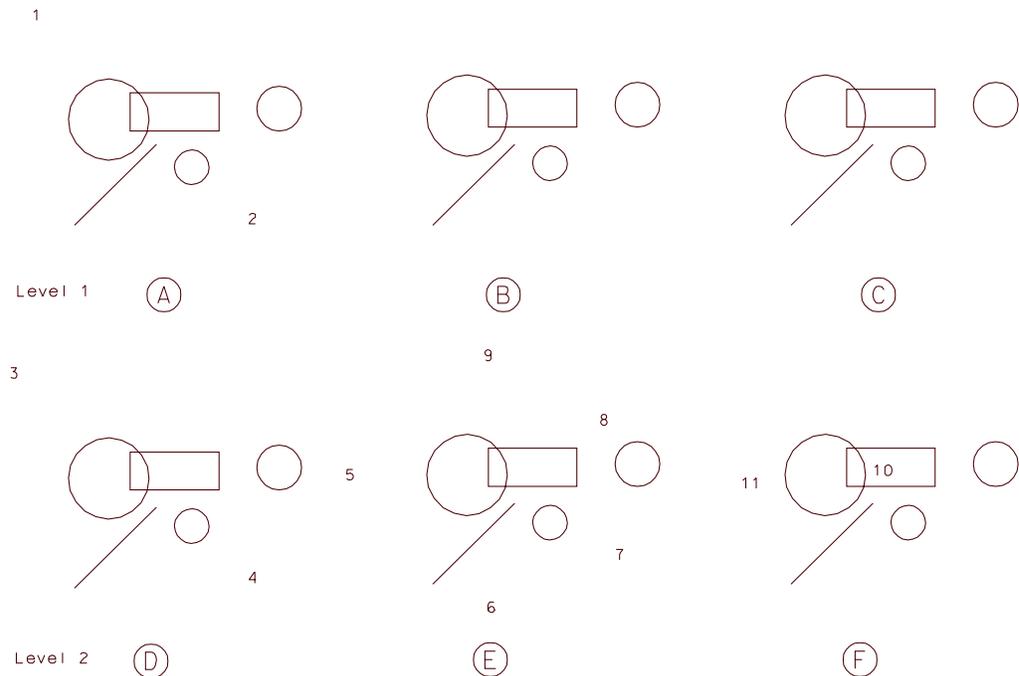
Figure 3-44. *Selection.dgn* design file.

Many commands require that you select one or more elements. There are several methods for selecting a single element or multiple elements. The primary tool for selecting elements is the ELEMENT SELECTION tool .

1. Pick the ELEMENT SELECTION tool and select one of the large circles.

The selected element is indicated with handle points and highlighted. You will also notice that elements are highlighted when the pointer is over the element.
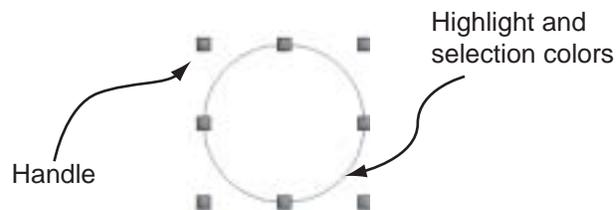
Figure 3-45. Handles and color are used to display selected element.

How a selection is displayed is based on user preference settings. Select **Workspace > Preferences** and then select the OPERATIONS category. The *Disable Edit Handles* parameter determines if a selected element is displayed with "handles." Colors associated with selection are established in the COLOR category accessed with the **Settings > Design File** menu option. Remember, if want changes in settings to be saved with the design file, select **File > Save Settings** or use the **Workspace > Preferences** to automatically save settings.
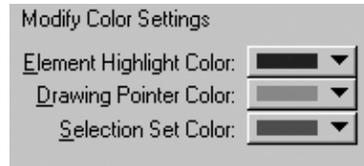
Figure 3-46. Color settings related to selecting elements

Parameter	*Description*
Element Highlight Color	While in the element selection mode, this parameter determines the color used to highlight when the pointer encounters an element.
Drawing Pointer Color	While the ELEMENT SELECTION tool is selected, this parameter determines the color used for the pointer.
Selection Set Color	After an element is selected, it becomes part of the selection set. This parameter determines the color used to highlight the selection set.

The ELEMENT SELECTION tool can be used to select elements in several different ways. The resulting selected elements comprise the selection set. The number of elements in the selection set are displayed on the status bar.

Element Selection Tool

Action	*Results*
Single Click	Selects a single element.
Single Click while holding CTRL key	Selects multiple elements.
Click and drag	Selects all elements that are entirely within enclosure.

2. To deselect the currently highlighted element, select the ELEMENT SELECTION tool and click the mouse pointer in an area that is not near any element.

When you use the selection tool, the mouse pointer includes a circle, often called an aperture or target.

Figure 3-47. The selection tool cursor showing selection target area.

If no element is inside the target, nothing is selected. To select an element, a portion of that element must reside within the target. The size of the target is controlled by the **Workspace > Preference** command. In the *Input* category, you can set the *Location Tolerance* to a smaller or larger value. The value represents the diameter of the target in pixels.

3. Pick the ELEMENT SELECTION tool. Next, select the **Workspace > Preference** command. In the *Input* category, set the *Location Tolerance* to a value of **20**. The value represents the diameter of the target in pixels. Select **OK** and notice the target area has increased in size. The change appears similar to the example below.

Figure 3-48. Location tolerance size change for the selection tool cursor.

A larger location tolerance makes it easier to select elements, but also increases the likelihood of selecting the wrong one when the selection includes several elements in close proximity.

4. Select the **Workspace > Preference** command. In the *Operations* category, set the *Location Tolerance* back to a value of **10**.

5. Use the ELEMENT SELECTION [icon] tool to pick items in *Selection.dgn*. Click several elements. Next, hold the CTRL key when clicking elements. Click away from any elements to deselect the currently selected element. Place the cursor near, but not on the number "1" in the "A" section of the design file. Click (left mouse button) and drag (keep the mouse button depressed) it to the number "2." You should notice a rectangular "marquee" defined by the diagonal points. Release the mouse button when located near position "2."

If you had clicked directly on the number "1" when doing this operation, you will physically move it rather than create a marquee. You must start the marquee in an area without an element.

Notice the items highlighted within the area created by clicking and dragging the selection tool from position "1" to position "2." Only those elements completely within the rectangular "marquee" are part of the selection set.

6. Click away from any elements to deselect any currently selected elements.

PowerSelector Tool

An alternative to using the Element Selection tool is the POWERSELECTOR tool. This tool incorporates several methods for selecting multiple elements. The PowerSelector tool is located on the same tool box as the Element Selection tool.

1. Select the POWERSELECTOR [icon] tool. The tool settings box is displayed.
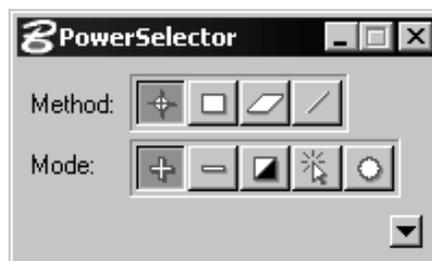
Figure 3-49. PowerSelector tool.

PowerSelector offers several options for selecting elements. The *Method* determines the selection process. The *Mode* determines if items are selected or deselected. Clicking the buttons toggles through the available options.

	Method	*Result*
	Individual	Selection is performed on each element. If you enter a data point but miss an element, the Block method activates.

Block selection includes two options. Options are changed by selecting the block tool when active.

	Method	*Result*
	Block Inside	All elements inside the defined block are selected.
	Block Overlap	All elements inside and overlapping the defined block are selected.

Shape selection includes two options. Options are changed by selecting the shape tool when active.

	Method	*Result*
	Shape Inside	All elements inside the defined shape are selected.
	Shape Overlap	All elements inside and overlapping the defined shape are selected.
	Line	All elements intersecting a drawn line are selected.

	Mode	*Result*
	Add	Select elements.
	Subtract	Deselect elements.
	Invert	Toggle the selection status of an element. If selected, it becomes deselected.
	New	Clears the current selection set and starts a new set.
	Select All	Select all elements if no elements are currently selected.
	Clear	Deselect all currently selected elements if element(s) selected.

Figure 3-49. PowerSelector tool methods and modes.

2. Click the SHOW SELECTION INFORMATION ▼ button on the PowerSelector settings box.

Clicking the SHOW SELECTION INFORMATION button expands the tool settings window to provide additional criteria for making selections with the PowerSelector. The top entry box allows you to select an attribute (level, color, style, weight, type, and class). The bottom entry box contains the values for the attribute selected.

Figure 3-50. PowerSelector selection attributes.

The six tabs allow you to select elements based on the assigned attributes (level, color, line style, line weight, element type, and element class). The outcome is dependent on the currently active method and mode.

3. Click NEW [icon] mode in PowerSelector. Click in the view window away from any elements to begin a new selection set.

4. Experiment with the various methods and modes for selecting elements. Make sure you use the COLOR category accessed with the **Settings > Design File** menu option to pick a unique color for selected elements.

5. After you have experimented with the various methods and modes, click NEW [icon] mode to begin a new selection set.

6. Using PowerSelector, set the method to INDIVIDUAL [icon] and the mode to ADD [icon].

   Click the SHOW SELECTION INFORMATION [icon] button and pick the CO (color) attribute tab. Next, pick the *green* (2) and *red (0)* colors. Note the elements selected.

   After selecting based on color, pick the LV (level) attribute tab. Note that levels 1 and 2 are highlighted indicating elements on those two layers have been selected. Click layer 1 to remove it from the selection set and note which elements remain selected. The criteria now are all green and red elements located on layer 2.

7. Change the MODE to SUBTRACT [icon] and select large circle in the D section. This should remove the circle from the selection set.

8. Change the MODE to ADD ⊕ . On the color attribute tab, select *blue* (4). MicroStation scans the design file for all elements that are the color 4 (blue). Note that elements on both levels are selected. If you select the level tab, you will notice that both level 1 and level 2 are in the selection set.

9. Click NEW ⚡ mode in PowerSelector. You can also use the CLEAR ⚡ mode in PowerSelector or the menu option **Edit > Select None** to deselect all elements.

10. Pick the PowerSelector tool and change the METHOD to LINE ╱ and the MODE to ADD ⊕ . In the *Selection.dgn* drawing, click on the number 1 and then click on the number 7. Notice that all elements that intersect the line formed between the two points are selected.

11. Select **Edit > Select None** to deselect all elements and close the PowerSelector dialog.

Fence

Another technique used to select items based on location is the use of a *fence*. A *fence* describes a technique for selecting elements that uses a variety of names (window, marquee, lasso, etc.). It allows the user to draw a fence or border around the desired geometry for the purpose of selection. Many software packages have added functions that mask the selection based on the desired outcome. For example, the software may display options for selecting everything inside the fence, everything outside the fence, or items that overlap the fence.

1. Select the PLACE FENCE ▦ tool. The tool settings show several options for use of the tool.

Figure 3-51. Place fence tool settings.

There are two parameters that control the fence: type and mode.

The *Fence Type* option determine the shape of the fence used to select elements. The options are:

Type	Description
Block	The fence is a rectangular block.
Shape	The fence is an irregular shape.
Circle	The fence is in the shape of a circle.
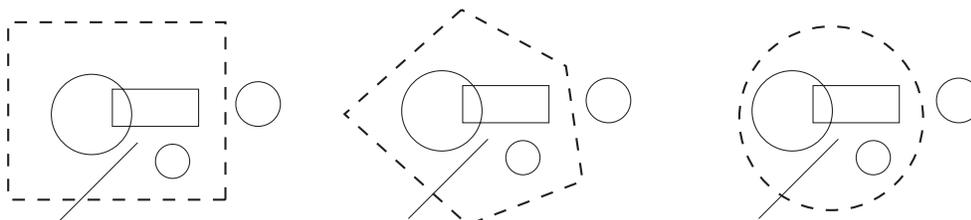
Figure 3-52. Block, shape, and circle fence type.

Type	*Description*
Element	The fence uses the boundaries of an existing element. If you select this type, you are prompted for a enclosed element.

Create Fence From Element > Select element

From View	The fence is derived from the extents of a selected window. If you select this type, you are prompted for a view.

Create Fence From View > Select view

This option can be used with a single window or when displaying multiple view windows.

From Design File	The fence is placed at the extents of the active design file, an attached reference file, or combination of both. If you select this type, the *Design* parameter appears in the settings box.

Figure 3-53. From Design File parameter added to place fence.

Active	The fence is placed at the extents of the currently active design file.
All	The fence is placed at the extents of the currently active design file and any attached reference files.
Choose	The fence is placed at the extents of the file (active or reference) that you identify. You are prompted to identify an element in the desired file.

The *Fence Mode* determines which elements are included in the selection. The options are:

Mode	*Description*
Inside	Elements selected are completely within the fence.
Overlap	Elements selected are completely or partially within fence.
Clip	Only items completely in fence or portions inside fence are acted upon. Elements that are partially inside fence are "broken" into multiple elements.
Void	Elements selected are completely outside the fence.
Void-Overlap	Elements selected are completely or partially outside fence.
Void-Clip	Only items completely outside fence or portions outside fence are acted upon. Elements that are partially outside of fence are "broken" into multiple elements.
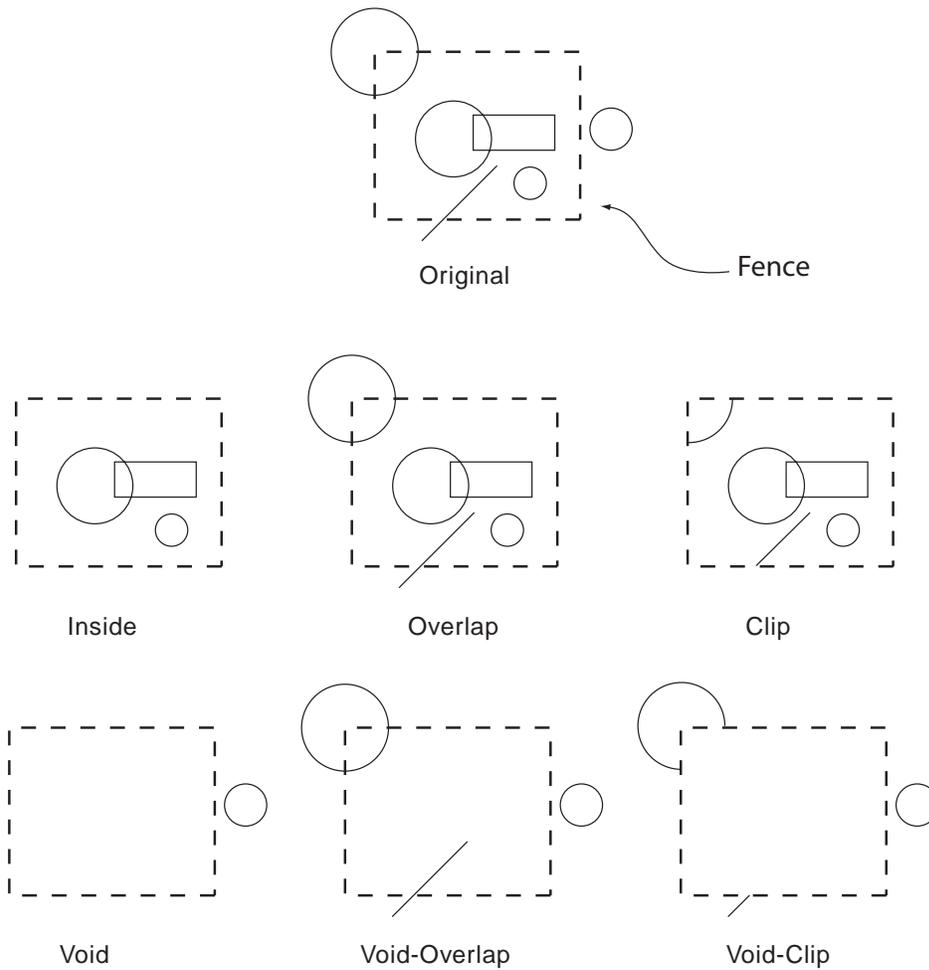
Figure 3-54. Illustration of selection with different fence modes.

The *Fence* tool does not take action directly on an element; it is only for the purpose of selection. When you select other tools, they often have an option to "use fence" for the selection of elements. For example, if you select the copy element tool, a USE FENCE option is available. In essence, the fence works similar to a temporary group by only selecting the items that have been fenced.
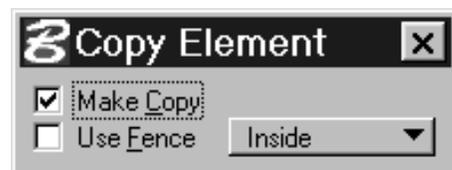
Figure 3-55. Fence option display in copy settings box.

In the above example, if USE FENCE was enabled, the copy would apply to all elements totally within the fence. To remove the fence, select the FENCE tool.

2.   Select **Edit > Select None** to deselect all elements.

3. If the fence dialog is not displayed, select the FENCE [icon] tool. Select the *Block* for TYPE and *Inside* for MODE. In example D of the *Selection.dgn* design file, tentative to the small point located next to the number 3. Accept this location with a data point. Next, tentative to the point next to the number 4 and accept with a data point. This places a rectangular fence around several elements. The illustration below shows the fence with a broken line.
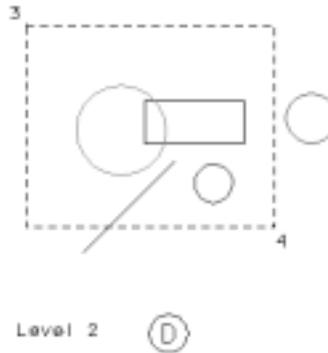
Figure 3-56. Illustration of a block fence.

4. Select the CHANGE ELEMENTS ATTRIBUTE [icon] tool. Note that the USE FENCE option is available for selection when a fence has been established. Enable the FENCE option and set MODE to *Inside*. Click on COLOR and select *green* (2). Accept the change by clicking a data point in the view window.

5. Select **Edit > Undo change element attributes** to change the elements back to their previous color.

6. Next, select the *Shape* for TYPE and *Inside* for MODE. In example E of *Selection.dgn*, first tentative to the point next to number 5, accept, and then tentative and accept on each of the points next to numbers 6 through 9. Complete the shape by selecting CLOSE ELEMENT. This places a shape fence around several elements.
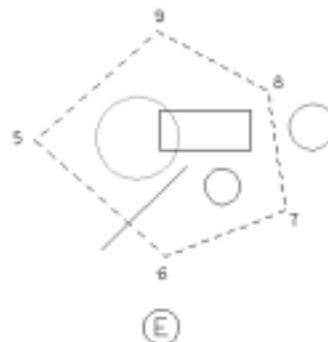
Figure 3-57. Illustration of a shape fence.

7. Select the CHANGE ELEMENTS ATTRIBUTE [icon] tool. With the FENCE option enabled, set MODE to *Inside*. Click on COLOR and select *red* (3). Accept the change by clicking a data point in the view window.

8. Select **Edit > Undo change element attributes** to change the elements back to their previous color.

9. Next, select the *Circle* for TYPE and *Inside* for MODE. In example F of *selection.dgn*, tentative and accept to the point next to number 10 and then tentative and accept to the point next to number 11. This places a circular fence around several elements.

10. Again, use the CHANGE ELEMENTS ATTRIBUTE [icon] tool and the **Edit > Undo change element attributes** to modify the fenced elements.

11. Experiment with using several of the fence types and modes. Remember, when a fence has been established, the fence option is available with several of the available tools.

Selection Options

In many cases, it is desirable to select elements based on specified criteria to increase speed and efficiency. The process is often referred to as masking and can be used to include or exclude geometry from selection. Some of these options are similar to those used with the PowerSelector tool. MicroStation offers three menu options that define the selection set.

<div align="center">

Edit > Select All
Edit > Select None
Edit > Select By Attributes

</div>

The **Edit > Select All** and **Edit > Select None** are used to select all elements or deselect all elements respectively. They work the same as the SELECT ALL [icon] and CLEAR [icon] modes available with PowerSelector.

The **Edit > Select By Attributes** is used to selectively include or exclude elements based on specified characteristics or conditions.

Figure 3-58. Select by attributes dialog box.

1. Select the **Edit > Select By Attributes** menu option.

When selected, you have several options that determine what is selected, how it is selected, and if the MODE is to include (*Inclusion*) or exclude (*Exclusion*) elements. These options are categorized in terms of *level*, *types*, *symbology*, and *mode*. You can also use *properties* and *tags* to determine the selection set.

The following is an overview of the options:

LEVELS
The selection set is based on the selected level(s). The highlighted levels indicate those that are part of the selection set. Each level can be selected independently or you can drag along the levels to toggle the setting.

TYPES
The selection set is based on the type of geometry (circles, lines, arcs, etc.). The highlighted types of geometry indicate those that are part of the selection set.

SYMBOLOGY
There are three different options.

> *Color*   If enabled, the selection set is based on color (red, blue, etc.).
>
> *Style*   If enabled, the selection set is based on line type (solid, dashed, etc.).
>
> *Weight*   If enabled, the selection set is based on line weight.

MODE
The MODE determines how criteria are used when the EXECUTE button is clicked.

> *Inclusion* Determines if selected items are included from the selection set.
> *Exclusion* Determines if selected items are excluded from the selection set.
>
> *Selection* Elements are selected based on Select By criteria.
> *Location* Filters geometry based on Select By criteria. Only those elements that meet the criteria are eligible for further selection. The elements are not selected when the EXECUTE button is pressed, only a filter is applied that limits further selection.
> *Display*   Filters display of elements based on Select By criteria. Only the elements in the selection set that meet the criteria are displayed. When the selection set is cancelled and the view is updated, the elements are again displayed.
>
> *On*   If set to On (the default), the Select By criteria is applied when the EXECUTE button is pressed.
> *Off*   If set to Off, Select By criteria is ignored.

2. In the SELECT BY ATTRIBUTES dialog, select *Level 2* in LEVELS and *Ellipse* in TYPES. Click EXECUTE and all circles and ellipses on level 2 are selected. The EXECUTE button creates the selection set based on the criteria specified.

3. Close the SELECT BY ATTRIBUTES dialog. A dialog is displayed.
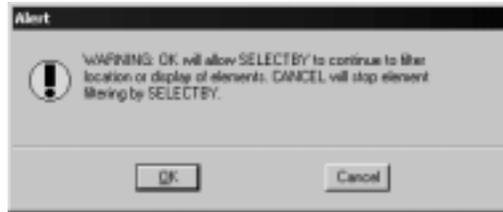
Figure 3-59. Select by attributes alert.

Your response to the alert message determines if the selection set remains active. If you select OK, the criteria specified by SELECT BY ATTRIBUTES remains in effect. To remove the selection criteria, you must select CANCEL. Selecting CANCEL allows you to select any element. Select CANCEL.

Review Questions

1. Define a drawing element.

2. Differentiate an element, complex chain, and complex shape.

3. How do you create a complex shape?

4. What happens when you "drop" an element?

5. Describe the three operations you can do to regions. These operations are often referred to as Boolean operations.

6. Differentiate the line, SmartLine, and multi-line tools.

7. What are multi-line joints?

8. Describe the different methods available for selecting multiple elements.

9. What is PowerSelector?

10. Describe the types and modes available for use with the fence tool.

11. The **Edit > Select By Attributes** command provides several options for determining the selection set. List and describe the available options.

Activities

1. Create a new drawing and experiment with each of the drawing elements. Some tools for placing elements may not be displayed and can be found in the **Tools** menu.

2. Experiment with placing lines using the line, SmartLine, and multi-line tools. While using the multi-line tool, select **Element > Multi-lines** to modify the appearance of the multi-line.

3. In the ATTRIBUTES tool box, select the ACTIVE LINE STYLE pull-down and view the custom line styles. Select **Element > Line Style > Custom** and experiment with modifying the width, scale factor, and shift. After making each modification, use the PLACE LINE tool to draw lines using the modified line styles.

4. Complete the non-dimensioned drawings below. Merely draw the items without regard
    to precise size.

    A. Create this drawing using the PLACE SMARTLINE tool.
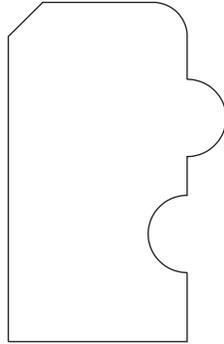
    B. Use the PLACE CIRCLE tool and the CREATE REGION tool to complete
    the following shapes.
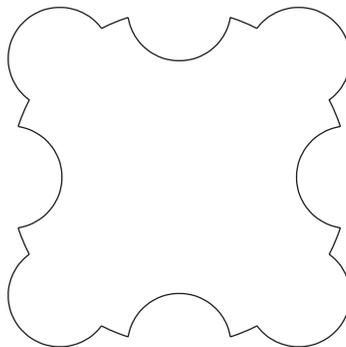
    C. Create a new drawing and set the master working units to feet and the sub-units to
    inches.

        Draw the item using the PLACE MULTI-LINE tool and the multi-line joint tools.