Chapter 18

Working with Advanced Drawing Options

Learning Objectives

After completing this chapter, you will be able to:

- *Draw NURBS splines using the Spline tool*
- *Edit NURBS splines using the Edit Spline tool*

Key Terms

- *Revision Cloud*
- *Wipeout*
- *Spline*
- *Fit Points*
- *CV*
- *Knots*
- *Kink*
- *Weight*

UNDERSTANDING THE USE OF REVISION CLOUDS

The revision cloud is a polyline of sequential arcs that forms a cloud-shaped object. It is used to highlight your drawing and helps you increase your productivity.

You can create a revision cloud from scratch, or you can convert objects such as a circle, ellipse, polyline, or spline into a revision cloud. A style is available to make the cloud look as though it was drawn with a calligraphy pen.

CREATING REVISION CLOUDS

Ribbon: Home > Draw > Revision Cloud or Annotate > MarkUp > Revision Cloud
Toolbar: Draw > Revision Cloud **Command:** REVCLOUD

The **Revision Cloud** tool in the **Draw** panel is used to create a cloud-shaped polyline, as shown in Figure 18-1. This tool can be used to highlight the details of a drawing. The prompt sequence that will follow when you choose this tool from the **Draw** panel is as follows:

Minimum arc length: 0.5000 Maximum arc length: 0.5000 Style: Normal
Specify start point or [Arc length/Object/Style] <Object>: *Specify the start point of the revision cloud.*
Guide crosshairs along cloud path...

As you move the cursor, different arcs of the cloud with varied radii are drawn. When the start point and endpoint meet, the revision cloud is completed and you get a message.

Revision cloud finished.

You can define the length of the arcs to be drawn using the **Arc length** option. Also, all the arcs drawn are of constant length. You can convert a closed loop into a revision cloud using the **Object** option. Note that the selected closed loop should be a single entity such as an ellipse, a circle, a rectangle, a polyline, and so on. The **Style** option is used to define the arc style for the revision cloud. The default style is **Normal** which when used creates a revision cloud similar to the one shown in Figure 18-1. You can change the style to **Calligraphy** to create a revision cloud similar to the one shown in Figure 18-2.

Note

*The **REVCLOUD** stores the last used arc length in the system registry. This value is multiplied by the **DIMSCALE** value to provide consistency when the program is used with drawings that have different scale factors.*

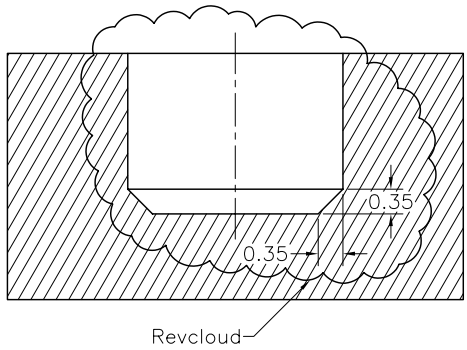

Figure 18-1 *Creating revcloud*

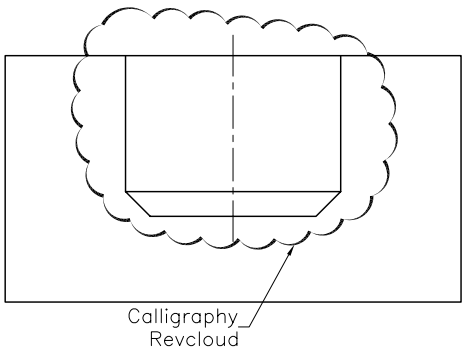

Figure 18-2 *Calligraphy revcloud*

CREATING WIPEOUTS

Ribbon: Home > Draw > Wipeout or Annotate > Markup > Wipeout
Command: WIPEOUT

The **Wipeout** tool is used to create a polygonal area to cover the existing objects with the current background color. The area defined by this tool is governed by the wipeout frame. The frame can be turned on and off for editing and plotting the drawings, respectively. This tool can be used to add notes and details to a drawing. The prompt sequence that will follow when you choose the **Wipeout** tool from the **Draw** panel is as follows:

Specify first point or [Frames/Polyline] <Polyline>: *Specify the start point of the wipeout.*
Specify next point: *Specify the next point of the wipeout.*
Specify next point or [Undo]: *Specify the next point of the wipeout.*
Specify next point or [Close/Undo]: *Specify the next point of the wipeout.*

Figure 18-3 shows a drawing before creating a wipeout and Figure 18-4 shows a drawing after creating the wipeout.

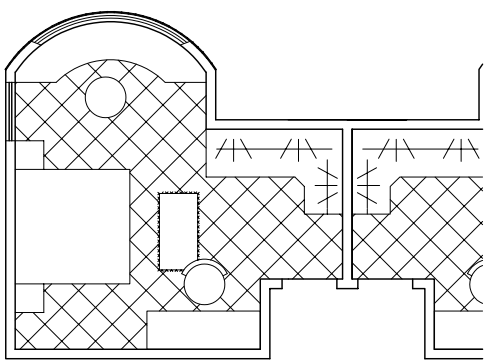

Figure 18-3 *Drawing before creating a wipeout*

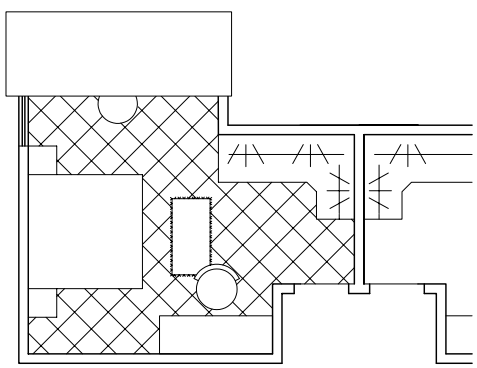

Figure 18-4 *Drawing after creating a wipeout*

If you do not want the frame of the wipeout to be displayed, enter **F** at the **Specify first point or [Frames/Polyline] <Polyline>** prompt and turn the frame off. The display of frames of all existing wipeouts will be turned off. Also, the display of frames of all new wipeouts will be turned off.

CREATING NURBS

Ribbon: Home > Draw > Spline Fit **Menu Bar:** Draw > Spline > Fit Points
Tool Palettes: Draw > Spline **Toolbar:** Draw > Spline **Command:** SPLINE

The NURBS is an acronym for Non-Uniform Rational Bezier-Spline. These splines are considered true splines. In AutoCAD LT, you can create NURBS using the **Spline** tool. The spline created with the **Spline** tool is different from the spline created using the **Polyline** tool. The nonuniform aspect of the spline enables the spline to have sharp corners because the spacing between the spline elements that constitute a spline can be irregular. Rational means that irregular geometry such as arcs, circles, and ellipses can be combined with free-form curves. The Bezier-spline (B-spline) is the core that enables accurate fitting of curves to input data with Bezier's curve-fitting interface. Not only are spline curves more accurate compared to smooth polyline curves, but they also use less disk space.

The prompt sequence to draw a spline with six fit points is given below.

*Choose the **Spline** tool.*
Current settings: Method=Fit Knots=Chord
Specify first point or [Method/Knots/Object]: *Select point P1*
Enter next point or [start Tangency/toLerance]: *Select point P2.*
Enter next point or [end Tangency/toLerance/Undo/Close]: *Select point P3.*
Enter next point or [end Tangency/toLerance/Undo/Close]: *Select point P4*
Enter next point or [end Tangency/toLerance/Undo/Close]: *Select point P5.*
Enter next point or [end Tangency/toLerance/Undo/Close]: *Select point P6 and press ENTER to end the process of point specification.*

Options for Creating Splines

The options of this tool for creating splines are as follows:

Object

This option allows you to change a 2D or 3D spline fitted polyline into a NURBS feature. The original splined polyline is deleted if the system variable **DELOBJ** is set to 1, which is the default value of the variable. You can change a polyline into a splined polyline using the **Spline** option of the **Edit Polyline** tool.

Method

In AutoCAD LT, you can create a spline by specifying fit points or control vertices. By default, a spline is created by specifying the fit points. Fit points are the points through which the spline will pass. Control vertices determine the shape of the curve. To set the method to create a spline, enter **M** at the **Specify first point or [Method/Knots/Object]** prompt. Then, you need to specify the method at the **Enter spline creation method [Fit/CV]** prompt. Figure 18-5

shows the spline created by using the **Fit** method. Figure 18-6 shows the spline created by using the **CV** method.

Note

On selecting a spline, a grip will be displayed at the start point. Click on the grip; a shortcut menu will be displayed. Now, you can choose an appropriate option to display the fit points or the control vertices, as shown in Figure 18-7.

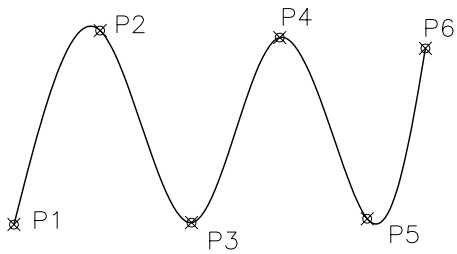

Figure 18-5 *Spline drawn by using the* ***Fit*** *method*

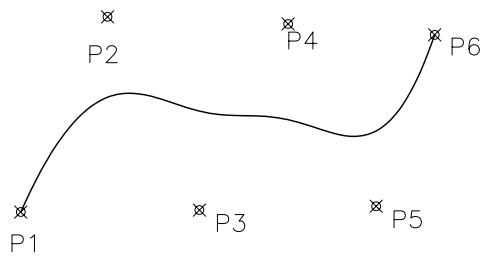

Figure 18-6 *Spline drawn by using the* ***CV*** *method*

Knots

If you have selected the **Fit** method, the **Knots** option will be displayed at the Command prompt, as given below.

Enter spline creation method [Fit/CV] <CV>: **FIT**
Current settings: Method=Fit Knots=Uniform
Specify first point or [Method/Knots/Object]:

You can specify three types of knots: Chord, Square root, and Uniform. Figure 18-8 shows the splines formed by using three different knot options.

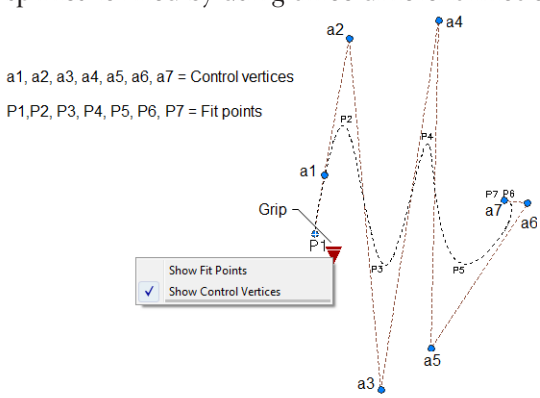

Figure 18-7 *Control vertices and fit points of a spline*

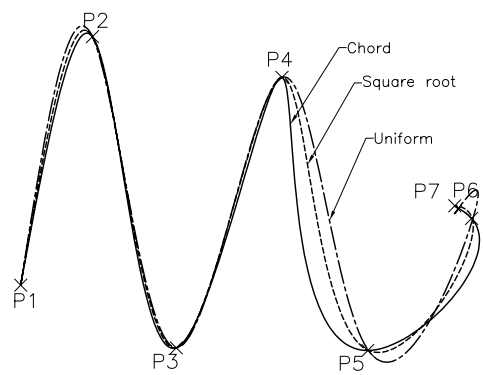

Figure 18-8 *Splines with different knot options*

Degree

If you have selected the **CV** method, then the **Specify first point or [Method/Degree/Object]** prompt will be displayed. Enter **D** at the Command prompt and specify the degree.

Close

This option allows you to create closed NURBS. When you use this option, AutoCAD LT will automatically join the endpoint of the spline with the start point, and you will be prompted to define the start tangent only.

Tolerance

This option is used to control the fit of the spline between specified points. By default, this value is zero; as a result, the spline passes through the points using which it has been created. Using this option, you can specify the tolerance values that will govern the spline creation, refer to Figure 18-9. The splines will be offset from the specified point by a distance equal to the tolerance value. The smaller the value of the tolerance, the closer the spline will be to the specified points.

Start and End Tangency

This option is used to control the tangency of a spline at its start point and the endpoint. On entering **T** at the **Enter next point or [start Tangency/toLerance]** or **Enter next point or [end Tangency/toLerance/Undo/Close]** prompt, you will be prompted to specify the tangent. Specify a point; the tangency will be determined by the slope of the spline at the specified point. Remember that after specifying the start point of the spline, you need to specify the start tangency. If you specify the second point, the default value will be used for the tangency. Similarly, you need to specify the end tangency after specifying the last point of the spline. Figure 18-10 shows a spline with start and end tangencies. The virtual tangent line is for reference.

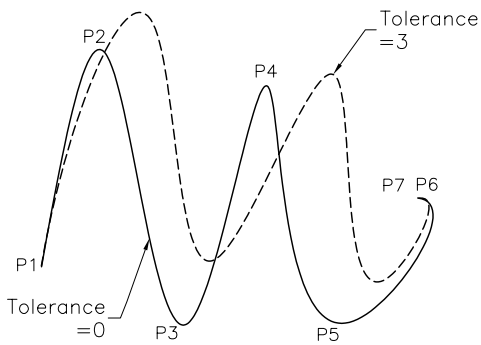

Figure 18-9 *Creating a spline with the* ***Tolerance*** *of 0 and 3*

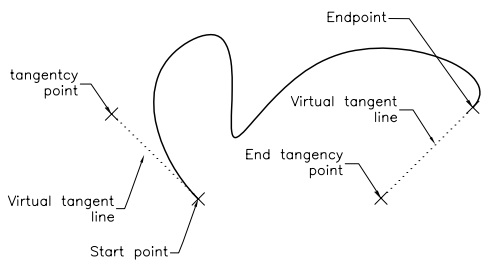

Figure 18-10 *Spline with start and end tangencies*

EDITING SPLINES

Ribbon: Home > Modify > Edit Spline **Menu Bar:** Modify > Object > Spline
Tool Palettes: Modify > Edit spline **Toolbar:** Modify II > Edit Spline
Command: SPLINEDIT

The NURBS can be edited using the **Edit Spline** tool. Using this tool, you can fit data in a selected spline, close or open a spline, move vertex points, and refine or reverse a spline. Apart from the ways mentioned in the preceding command box, you can use the options available in the **Spline** cascading menu from the shortcut menu that will be displayed when you select a spline and right-click. The prompt sequence that will follow when you choose the **Edit Spline** tool in the **Modify** panel is given next.

Select spline: *Select the spline to be edited if not selected already, using the above-mentioned shortcut menu.*
Enter an option [Close/Join/Fit data/Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit>: *Select any one of the options.*

Options for Editing Splines

The options of this tool for editing the splines are described next.

Fit data

When you draw a spline, the spline fits as per the specified points (data points). The **Fit data** option allows you to edit these points. You can add, delete, or move the data points. These data points or control points are also referred to as fit points. For example, if you want to redefine the start and end tangents of a spline, select the **Fit data** option and then select the **Tangents** option. The prompt sequence that will follow when you invoke this option is given next.

Select spline: *Select the spline to be edited.*
Enter an option [Close/Join/Fit data/Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit>: **F** Enter
Enter a fit data option [Add/Close/Delete/Kink/Move/Purge/Tangents/ toLerance/eXit] <eXit>: **T** Enter

The start and end tangent points can be selected or their coordinates can be entered. The options available in the **Fit data** option are as follows:

Add. You can use this option to add new fit points to the spline. When you invoke this option, you will be prompted to specify the existing fit point on the spline. After selecting the existing fit point, you will be prompted to specify the location of the new control points. The fit point you select and the next fit point appear as selected grips. You can now add a fit point between these two selected fit points, as shown in Figure 18-11. If you select the start point or endpoint of the spline, only they get highlighted. When you select the start point of the spline, you are prompted to specify whether you want to add the new fit point before or after the start point of the spline. AutoCAD LT will continue prompting for the location of new control points until you press ENTER at the **Specify new point <exit>** prompt.

Close/Open. This option is used to close an open spline or open a closed spline, refer to Figure 18-12. If the spline is open, the **Close** option is available and if the spline is closed, the **Open** option is displayed.

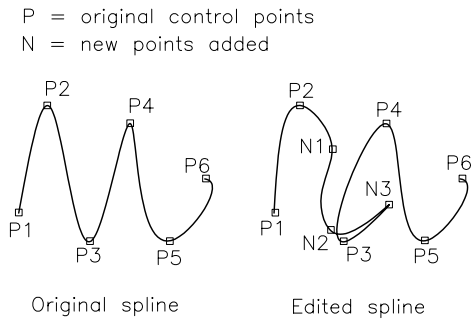

Figure 18-11 The original spline and the edited spline after adding fit points

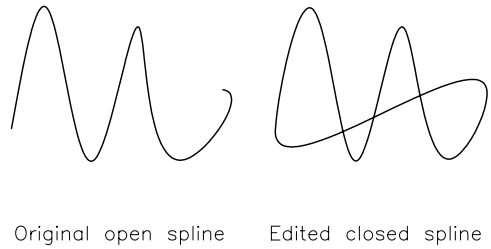

Figure 18-12 An open spline and a closed spline

Delete. This option is used to delete a selected fit point from a spline. You can continue deleting fit points from a spline until only two fit points are left in the spline.

Kink. This option is used to add a knot and a fit point to the selected point. On selecting this option, you will be prompted to specify a point on the spline. Select a point; a kink will be added.

Move. You can move fit points by using this option. When you select this option, the start point of the spline is highlighted and the following prompt sequence appears:

> Specify new location or [Next/Previous/Select point/eXit] <N>: *Select a new location for the start point using the mouse pick button or enter any one of these options.*

You can enter **N** if you want to select the next point, **P** if you want to select a previous point, or **S** if you want to select any other point. If you enter **X** at the preceding prompt, you will exit the tool. Figure 18-13 shows the movement of data points in a spline. Remember that this option is used to move only the data points on the spline and not the control points on the Bezier control frame.

Purge. This option is used to remove fit point data from a spline. This reduces the file size, which is useful when a drawing, for example, a landscape contour map, contains large number of splines. Purging simplifies the spline definition and the drawing file size.

Tangents. This option is used to modify the tangents of the start and end points of a selected spline, refer to Figure 18-14. When you select this option, you will be prompted to specify the start and end tangents of the spline. You can specify the start and end tangents or use the systems default tangents.

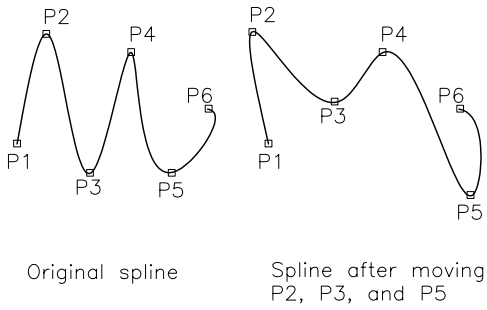

Figure 18-13 *The original spline and the edited spline after moving the data points*

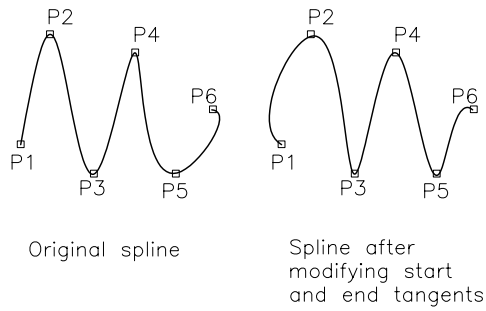

Figure 18-14 *The original spline and the spline after modifying the start and end tangents*

toLerance. This option is used to modify the fit tolerance values of a selected spline. As discussed, while creating the splines, different tolerance values produce different splines. A smaller tolerance value creates a spline that passes very closely through the definition points of a spline. When you invoke this option, you will be prompted to specify the tolerance value for the spline.

eXit. This option is used to exit the **Fit data** option of editing splines.

Close/Open

This option is used to close an open spline or open a closed spline. When you select the **Close** option, AutoCAD LT lets you open, edit the vertex, covert spline to polyline, or reverse the spline.

Edit Vertex

This option is used to refine a spline by adding more control points in it, elevating the order, or adding weight to vertex points. The prompt sequence that will follow when you invoke this option is given next.

Enter an option [Open/Fit data/Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit>: ***E*** Enter
Enter a vertex editing option [Add/Delete/Elevate order/Move/Weight/eXit] <eXit>:

The options in this prompt are discussed next.

Add. This option is used to add more control points on the spline. When you invoke this option, you will be prompted to specify the location of the new point on the spline. You can directly specify the location of the new point using the left mouse button. A new control point will be added at the specified location.

Delete. This option is used to delete a selected fit point from a spline.

Elevate order. This option is used to increase the order of a spline curve. The order of a curve can be defined as the highest power of the algebraic expression that defines the spline plus 1. For example, the order of a cubic spline will be 3 + 1 = 4. Using the **Elevate order** option, you can increase this order for a selected spline. This results in more control points on the curve and a greater possibility of controlling the spline. The value of the spline order varies from 4 to 26. You can only increase the order and not decrease it. For example, if a spline order is 18, you can elevate its order to any value greater than 18, but not less than 18.

Move. When you draw a spline, it is associated with the Bezier control frame. The **Move vertex** option allows you to move the vertices of the control frame. The **Move** option is similar to the **Move** option of **Fit data** for editing the splines. The prompt sequence that follows when you invoke this option is given next.

Enter an option [Close/Join/Fit data/Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit>: **E** Enter
Enter a vertex editing option [Add/Delete/Elevate order/add Kink/Move/Weight/eXit] <eXit>: **M** Enter
Specify new location or [Next/Previous/Select point/eXit] <N>:

Weight. You can also add weight to any of the control vertices of a spline by using this option. When weight is added to a particular vertex, the spline gets pulled more towards it. Similarly, a lower value of weight of a particular spline vertex will result in the spline getting pulled less towards that particular vertex. In other words, adding weight to a particular point will force the selected point to maintain its tangency with the point. The more weight added to the point, the more is the distance through which the spline will remain tangent to the point. By default, the spline gets pulled equally towards the vertices of the spline. The default value of weight provided to each control point is 1.0 and can have only positive values. Once you have added the weight to a point, you can proceed to the next point. You can also directly select the point to which the weight has to be added. The prompt sequence for using this option is given next.

Enter an option [Close/Join/Fit data/Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit>: **E** Enter
Enter a vertex editing option [Add/Delete/Elevate order/Move/Weight/eXit] <eXit>: **W** Enter
Spline is not rational. Will make it so.
Enter new weight (current = 1.0000) or [Next/Previous/Select point/eXit] <N>: **S** Enter
Specify existing fit point on spline <exit>: *Select a fit point*
Enter new weight (current = 1.0000) or [Next/Previous/Select point/eXit] <N>: **3** Enter
Enter new weight (current = 1.0000) or [Next/Previous/Select point/eXit] <N>: **Exit** Enter

Join

If a spline, line, or an arc is joined to the spline that is being edited, then you can use this option to join those entities to the spline. On choosing this option, you will be prompted to select an open curve. Select the curves to be joined and press ENTER; all curves will be joined

to the spline, and knot and fit points will be added at the joining point. The prompt sequence for using this option is given next.

Enter an option [Close/Join/Fit data/Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit>: **J**
Select any open curves to join to source: *Select an object. 1 found*
Select any open curves to join to source: *Select an object. 1 found, 2 total*
Select any open curves to join to source:
2 objects joined to source

Reverse

This option is used to reverse the direction of spline creation. This implies that when you apply this option to a spline, the start point of the spline becomes the end point of the spline and vice versa.

convert to Polyline

This option is used to convert a spline into a polyline. On specifying this option, you will be prompted to specify a precision value. Specify a value between 0 and 99, and then press ENTER; the spline will be converted into a polyline. The precision value determines the accuracy of the resulting polyline.

Undo

This option will undo the previous editing operation applied to a spline within the current session of the **Edit Spline** tool. You can continue to use this option till you reach the spline as it was when you started to edit it.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. The **Revision Cloud** tool in the **Draw** panel is used to create a cloud-shaped polyline. (T/F)

2. The **Wipeout** tool is used to create a polygonal area to cover the existing objects with the current background color. (T/F)

3. The _____ can be edited using the **Edit Spline** tool.

4. The _____ option of the **Fit data** option of the **Edit Spline** tool is used to reduce the file size by removing the fit data of the selected splines.

Review Questions

Answer the following questions:

1. Once the **Purge** option of **Fit data** is used on a spline, the **Fit data** option is no longer available for the particular spline. (T/F)

2. Which of the following options can be used to reverse the direction of spline creation while editing a spline?

   (a) **Fit data** (b) **Refine**
   (c) **Reverse** (d) **Weight**

3. If the value of **Fit tolerance** of a spline is _____, the spline passes exactly through the fit points of the spline.

4. You need to set the degree of a spline, if you are drawing the spline by using the _____ method.

Exercise 1

Create the drawing shown in Figure 18-15. Assume the missing dimensions.

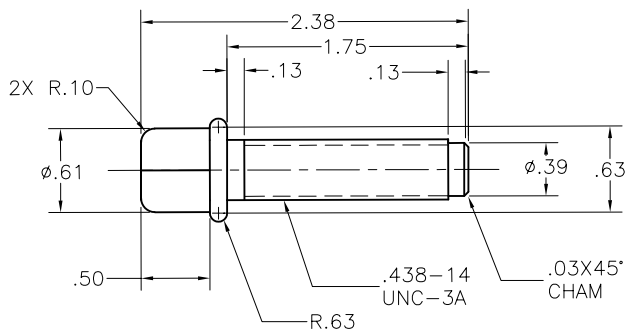

Figure 18-15 *Drawing for Exercise 1*

Problem-Solving Exercise 1

Create the drawing shown in Figure 18-16. Some of the reference dimensions are given in the drawing. Assume the missing dimensions.

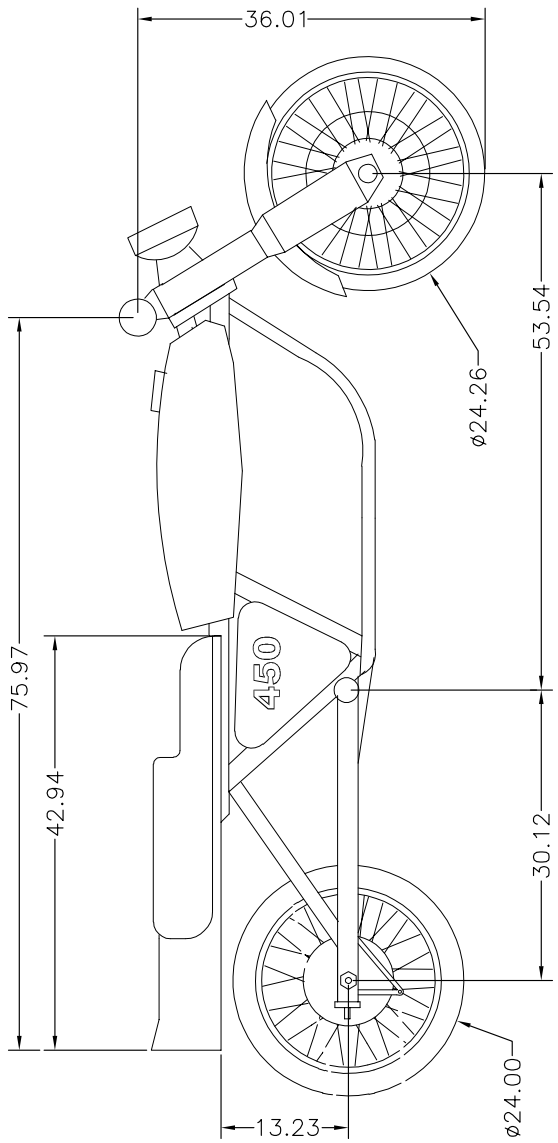

Figure 18-16 *Drawing for Problem-Solving Exercise 1*

Problem-Solving Exercise 2

Create the drawing shown in Figure 18-17. Some of the reference dimensions are given in the drawing. Assume the missing dimensions.

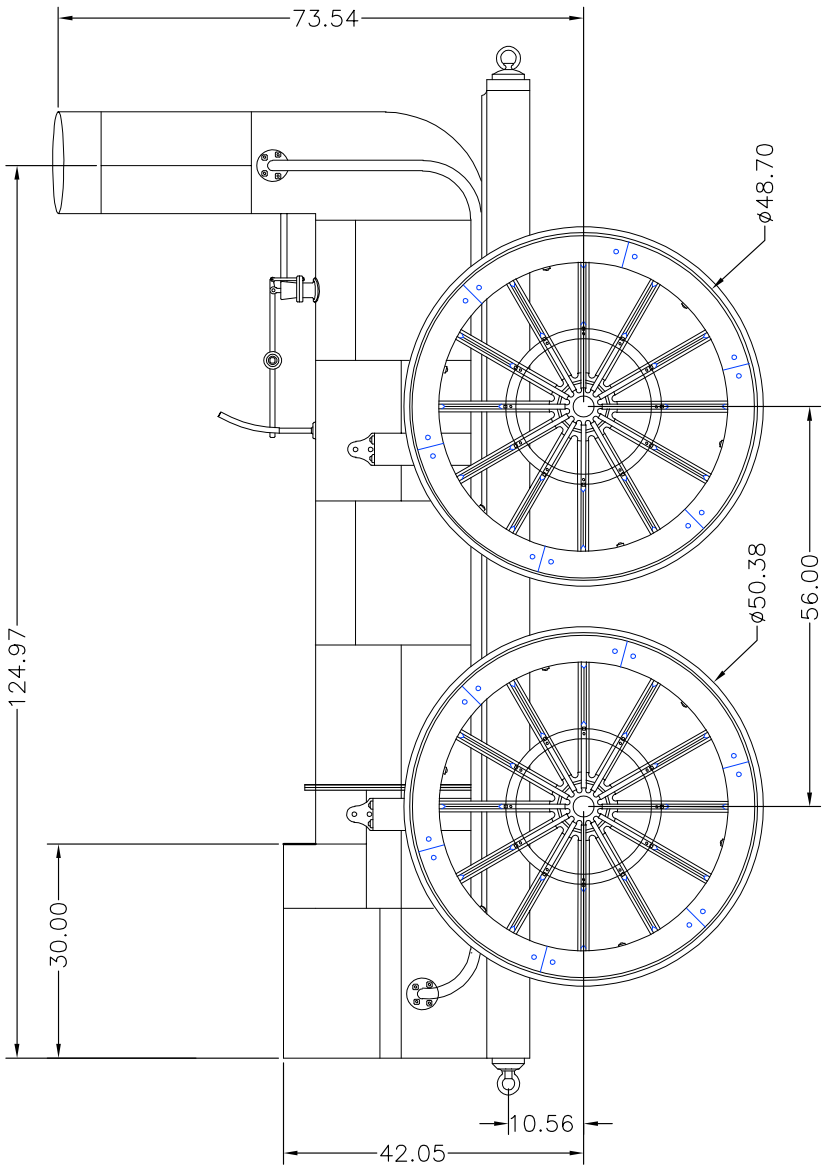

Figure 18-17 *Drawing for Problem-Solving Exercise 2*

Problem-Solving Exercise 3

Create the drawing shown in Figure 18-18. Some of the reference dimensions are given in the drawing. Assume the missing dimensions.

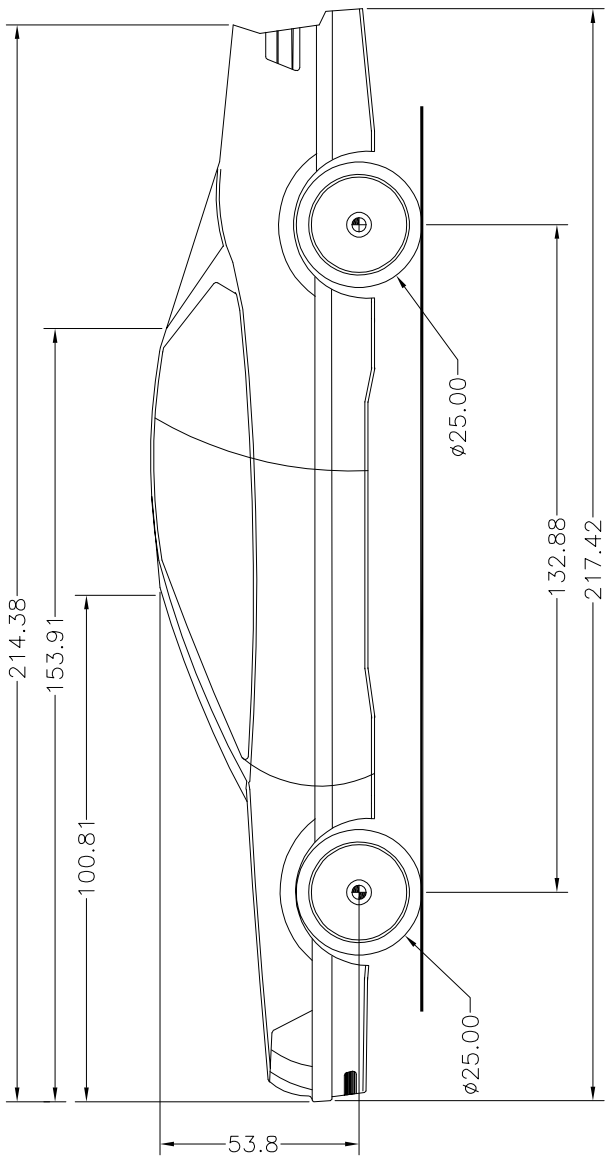

***Figure 18-18** Drawing for Problem-Solving Exercise 3*

Problem-Solving Exercise 4

Create the drawing shown in Figure 18-19. Some of the reference dimensions are given in the drawing. Assume the missing dimensions.

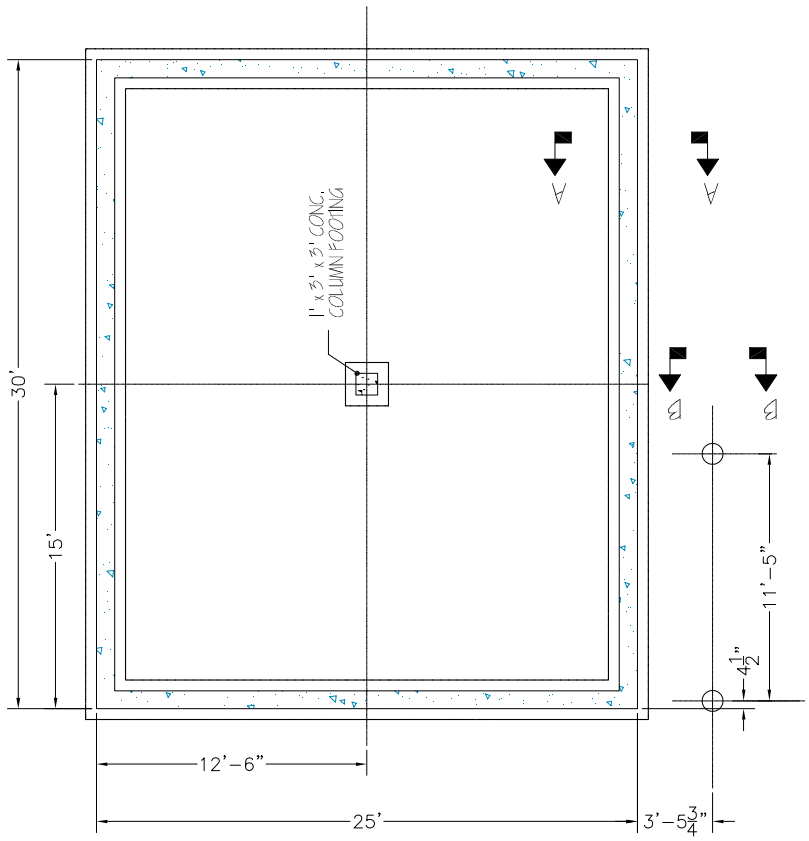

***Figure 18-19** Drawing for Problem-Solving Exercise 4*

Problem-Solving Exercise 5

Create the drawing shown in Figure 18-20. Some of the reference dimensions are given in the drawing. Assume the missing dimensions.

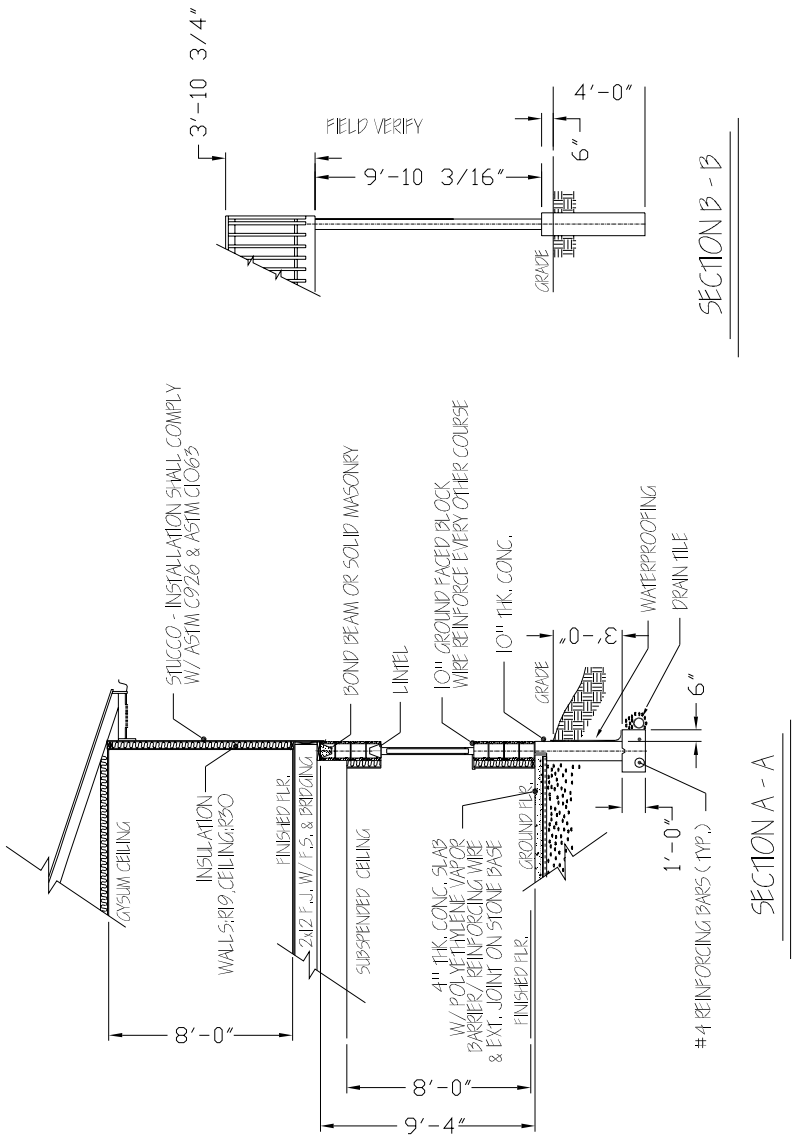

***Figure 18-20** Drawing for Problem-Solving Exercise 5*

Problem-Solving Exercise 6

Create the drawing shown in Figure 18-21. Some of the reference dimensions are given in the drawing. Assume the missing dimensions.

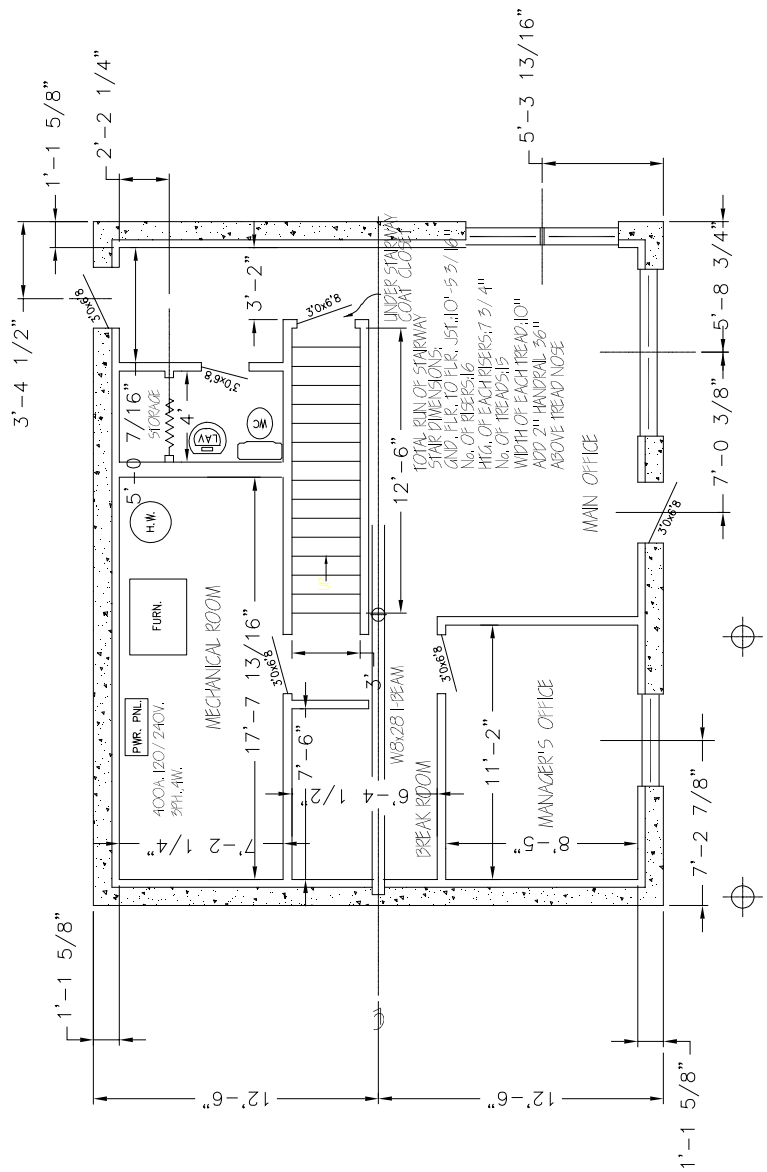

***Figure 18-21** Drawing for Problem-Solving Exercise 6*

Problem-Solving Exercise 7

Create the drawing shown in Figure 18-22. Some of the reference dimensions are given in the drawing. Assume the missing dimensions.

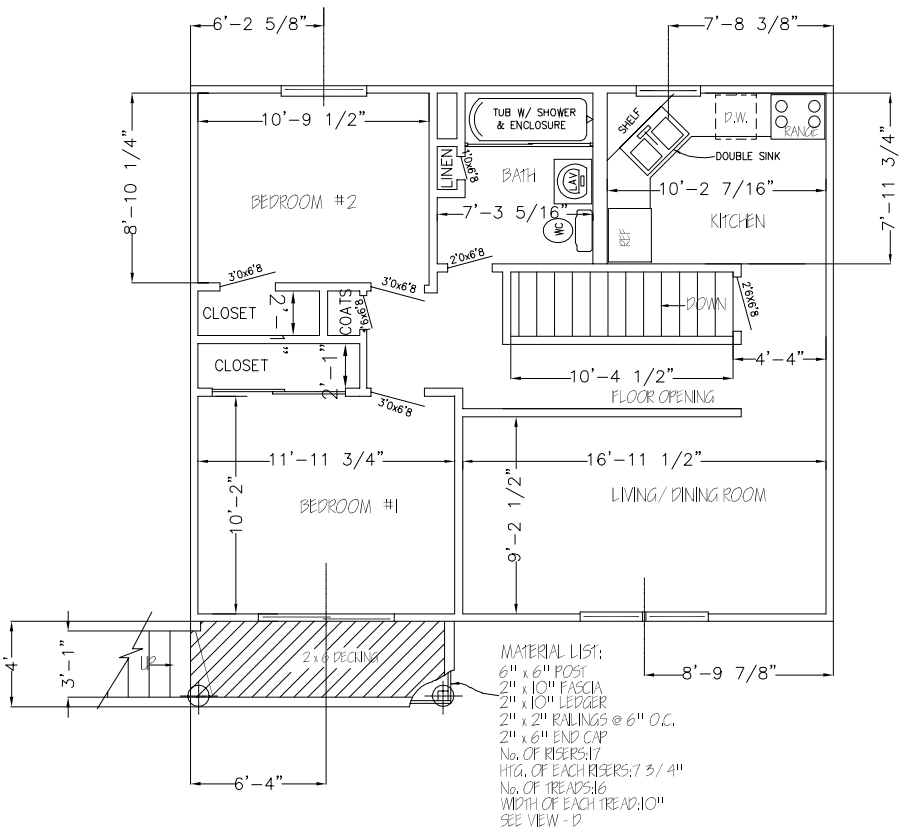

Figure 18-22 *Drawing for Problem-Solving Exercise 7*

Answers to Self-Evaluation Test

1. T, **2**. T, **3**. NURBS, **4**. purge