



# Chapter 17

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## Working with Advanced Drawing Options

### Learning Objectives

**After completing this chapter, you will be able to:**

- Create double lines using the **DLINE** command.
- Create revision clouds using the **REVLOUD** command.
- Draw **NURBS** using the **SPLINE** command.
- Edit **NURBS** using the **SPLINEDIT** command.

## CREATING DOUBLE LINES

**Menu:** Draw > Double Line  
**Command:** DLINE

The AutoCAD LT Double Line feature allows you to create double lines that consist of two parallel lines or arcs. Double lines are drawn like lines. You can also draw double connected lines. AutoCAD LT treats each line segment forming a double line as an individual object. The prompt sequence that will follow when you choose the **Draw > Double Line** from the menu bar (Figure 17-1) to create a simple double line is given next.

Specify start point or [Break/Caps/Dragline/Snap/Width]: *Specify the start point of the double line.*  
 Specify next point or [Arc/Break/Caps/CLOSE/ Dragline/Snap/Undo/Width]: *Specify the next point or select an option.*

The options of the **DLINE** command are discussed next.

### Start Point

This is the default option and requires you to specify a point by entering its coordinates or by picking the point on the screen with the pointing device (example mouse).

### Break

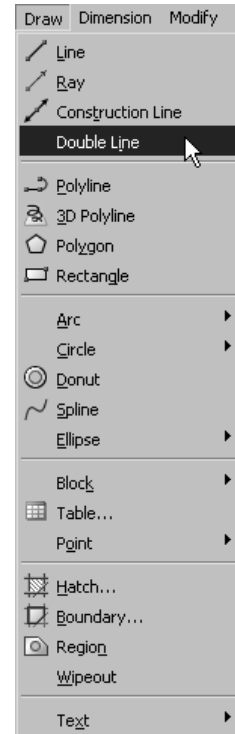
If you require that a line should be broken when it intersects a double line, as shown in Figure 17-2, set **Break** to ON. The default option is ON. Set **Break** OFF if you do not want to have breaks at double line intersections. The prompt sequence to set **Break** to ON is as follows.

Command : **DLINE**   
 Specify start point or [Break/Caps/Dragline/Snap/Width]: **B**  
 Break Dlines at start and end points? [OFF/ON] <ON>:

### Caps

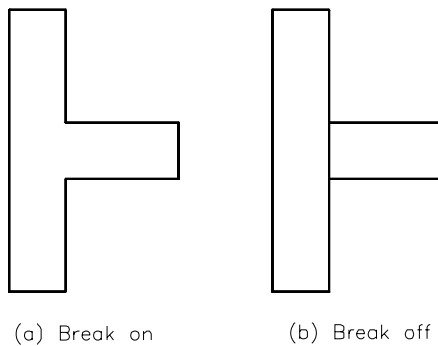
With this option, you can close the ends of a double line and obtain clean corners as shown in the Figure 17-3.

Command: **DLINE**   
 Specify start point or [Break/Caps/Dragline/Snap/Width]: **C**   
 Enter option for drawing endcaps [Both/End/None/Start/Auto] <Auto>:

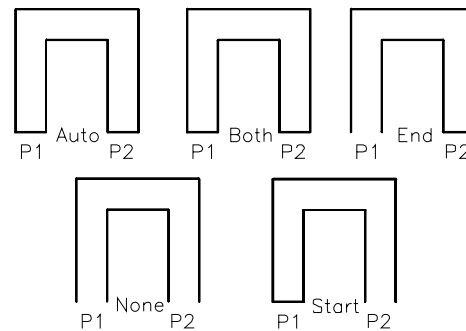


**Figure 17-1** Invoking the **DLINE** command from the menu bar

At this prompt, you can enter any of these options. If you press ENTER, that is if you select **Auto**, AutoCAD LT caps all the ends of the double line except those that are snapped to an object. With the rest of the options, AutoCAD LT caps the ends specified even if they are snapped to an object. Select **Both** if you want both ends of a double line to be capped. To specify the end of the double line which you want capped, select **End** or **Start**. **Start** caps the double line at its beginning while **End** caps the end you pick last. If you do not want any ends to be capped, select **None**.



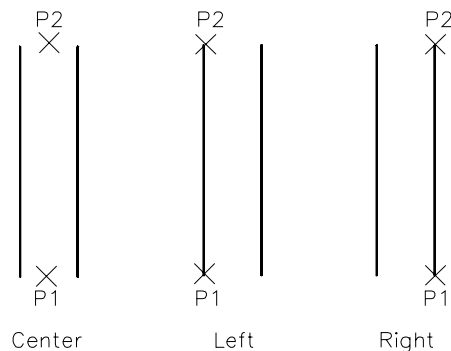
**Figure 17-2** Using the **Break** option



**Figure 17-3** Using the **Caps** option

## Dragline

This option is used to set the location of the pick point with respect to the double line. You can set the pick point to be located at the center, or on the left or right leg of the double line as shown in Figure 17-4. To ascertain the left and right sides of the double line, imagine looking towards the end of the double line from its start point. The default option is set to center of the double line. The prompt sequence to set the pick point to be on the right leg of the double line is given next.



**Figure 17-4** Using the **Dragline** option

Command: **DLINE**

Specify start point or [Break/Caps/Dragline/Snap/Width]: **D**

Enter offset from center or dragline position option [Left/Center/Right]

<Offset from center = 0.0000>: **R**

Another way of setting the location of the pick point is by setting the offset distance from the center. You can enter any negative value to set the pick point to the left of the center of the double line. Any positive value sets the pick point to the right of the center of the double line.

## Snap

If you want a double line to start or end at an existing entity, set the **snap** option of the **DLINE** command. AutoCAD LT prompts you as follows.

Set snap size or snap On/Off. [Size/OFF/ON] <ON>:

You can set the snap on or off or set a snap size. To set a snap size enter S at the above prompt.

New snap size (1 - 10) <3>: *Select a number between 1 and 10.*

This size is in pixels and refers to the search area set up by AutoCAD LT to find an object to snap to. The larger the number you can enter, the farther the cursor can be made from the object, that is, AutoCAD LT sets up a larger area to search for the object. AutoCAD LT breaks up the object if the **Break** option is on.

## Undo

If you draw one or more double lines by mistake, you can undo them in the **DLINE** command itself. With the **Undo** option, you can erase the last drawn double line segment. Thus, you can undo a series of double line segments while you are still in the **DLINE** command.

## Width

You can assign any width to the double line using the **Width** option of the **DLINE** command. The prompt sequence to set the width of 1 for the double line is given next.

Command: **DLINE**

Specify start point or [Break/Caps/Dragline/Snap/Width]: **W**

Specify width of dline <0.0500>: **1**

## Arc

You can draw double line arcs with the **DLINE** command. The prompt sequence is given next.

Command: **DLINE**

Specify start point or [Break/Caps/Dragline/Snap/Width]: *Specify the start point of the double line.*

Specify next point or [Arc/Break/Caps/Close/Dragline/Snap/Undo/Width]: **A**

Specify second point or


[Break/Caps/Center/Close/Dragline/Endpoint/Line/Snap/Undo/Width]:

Here, you pick the next point of the double line arc. The prompt sequence specifying its third point is given next.

Specify end point of arc: *Select the endpoint of the arc.*

The prompt sequence to draw an arc by specifying its center point is given next.


Specify second point or

[Break/Caps/CEnter/CLose/Dragline/Endpoint/Line/Snap/Undo/Width]: **CE** 

Specify center point of arc: *Specify the center point of the arc.*


Specify end point of arc or [Angle/chord Length]: *Specify the endpoint of the arc.*

You can also specify the included angle by entering **A** or chord length by entering **L** at the **Specify end point of arc or [Angle/chord Length]** prompt. The prompt sequence for specifying included angle is given next.

Specify end point of arc or [Angle/chord Length]: **A** 

Specify included angle: *Specify the included angle of the arc.*


Another method of drawing a double line arc is by specifying the endpoint first. The prompt sequence to draw an arc by this method is given next.

Command: **DLINE** 

Specify start point or [Break/Caps/Dragline/Snap/Width]: *Specify the start point of the double line.*

Specify next point or [Arc/Break/Caps/CLose/Dragline/Snap/Undo/Width]: **A** 

Specify second point or

[Break/Caps/CEnter/CLose/Dragline/Endpoint/Line/Snap/Undo/Width]: **E** 

Specify end point of arc: *Specify the endpoint of the arc.*

Specify center point of arc or [Angle/Direction/Radius]: *Specify the included angle of the arc.*

## Line

The **Line** option allows you to change from double line arcs to drawing double lines. AutoCAD LT continues to make double line arcs once you select the **Arc** option. To resume drawing double lines enter **L** at the **Specify second point or [Break/Caps/CEnter/CLose/Dragline/Endpoint/Line/Snap/Undo/Width]** prompt.

## Close

You can close a double line with this option of the **DLINE** command. When you select this option, AutoCAD LT closes the double line with an arc or a line depending on the mode which is active. The **Close** option works only if the double line consists of two or more line segments or one or more arcs. After closing the double line, AutoCAD LT exits the **DLINE** command. The prompt sequence for closing an arc segment is:

Command: **DLINE** 

Specify start point or [Break/Caps/Dragline/Snap/Width]: *Specify the start point.*

Specify next point or [Arc/Break/Caps/CLose/Dragline/Snap/Undo/Width]: **A** 


Specify second point or

[Break/Caps/CEnter/CLose/Dragline/Endpoint/Line/Snap/Undo/Width]: *Specify the second point of the arc.*

Specify end point of arc: *Specify the endpoint of the arc.*

Specify second point or

[Break/Caps/CENter/Close/Dragline/Endpoint/Line/Snap/Undo/Width]:

**CL**  (An arc is drawn in continuation of the arc drawn earlier such that a complete circle is displayed on the screen.)

## CREATING REVISION CLOUDS

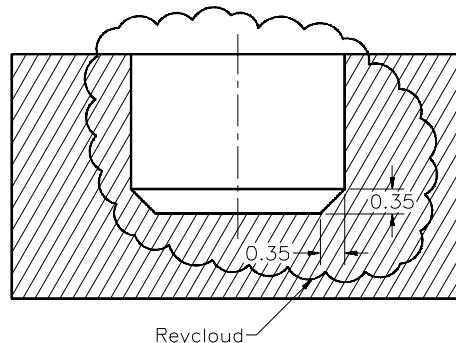
**Toolbar:** Draw > Revision Cloud  
**Command:** REVCLOUD



The **REVCLOUD** command is used to create a cloud shaped polyline.

Figure 17-5 shows the use of the **REVCLOUD** command. This command can be used to highlight the details of a drawing. The prompt sequence that will follow when you choose the **Revcloud** button from the **Draw** toolbar 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...

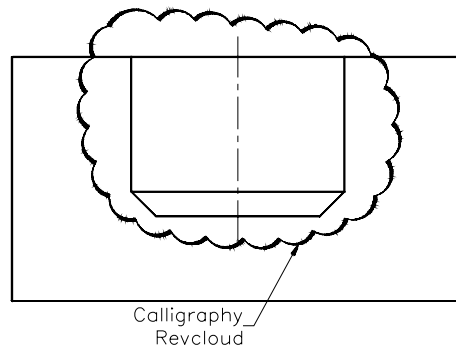


**Figure 17-5** Creating revclouds

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

### Revision cloud finished.

Using the **Arc length** option, you can define the length of the arcs to be drawn. Also, all the arcs drawn are of constant length. Using the **Object** option, you can convert a closed loop into a revision cloud. 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** that creates a revision cloud similar to the one shown in Figure 17-5. You can change the style to **Calligraphy**, and this creates a revision cloud similar to the one shown in Figure 17-6.



**Figure 17-6** Calligraphy revclouds

**Note**

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

## CREATING WIPEOUTS

<b>Menu:</b>	Draw > Wipeout
<b>Command:</b>	Wipeout

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

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

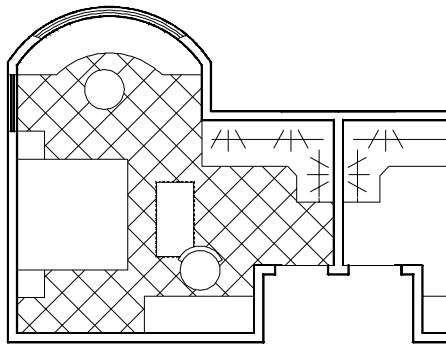
Specify next point: *Specify the next point of the wipeout.*

Specify next point or [Undo]: *Specify the next point of the wipeout.*

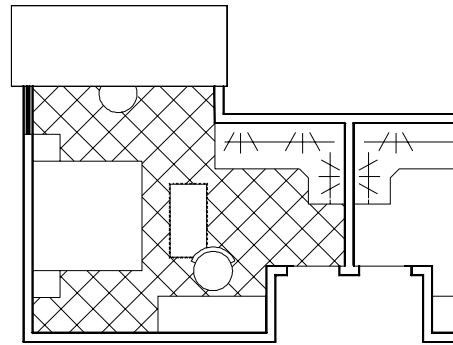
Specify next point or [Close/Undo]: *Specify the next point of the wipeout.*

Figure 17-7 shows a drawing before creating wipeout and Figure 17-8 shows a drawing after creating 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 the existing wipeouts will be turned off. Also, the display of frames of all the new wipeouts will be turned off.



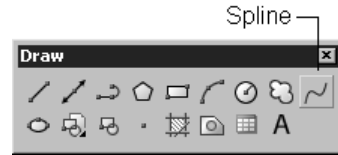
**Figure 17-7** Drawing before creating a wipeout



**Figure 17-8** Drawing after creating a wipeout

## CREATING NURBS

<b>Toolbar:</b>	Draw > Spline
<b>Menu:</b>	Draw > Spline
<b>Command:</b>	SPLINE



**Figure 17-9** Invoking the **SPLINE** command from the **Draw** toolbar

The NURBS is an acronym for **NonUniform Rational Bezier-Spline**. These splines are considered true splines. In AutoCAD LT, you can create NURBS using the **SPLINE** command. The spline created with the **SPLINE**

command (Figure 17-9) is different from the spline created using the **PLINE** command. 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 following is the prompt sequence for creating the spline shown in Figure 17-10.

Specify first point or [Object]: *Select point P1.*

Specify next point: *Select point P2.*

Specify next point or [Close/Fit tolerance]

<start tangent>: *Select point P3.*

Specify next point or [Close/Fit tolerance]

<start tangent>: *Select point P4.*

Specify next point or [Close/Fit tolerance]

<start tangent>: *Select point P5.*

Specify next point or [Close/Fit tolerance]

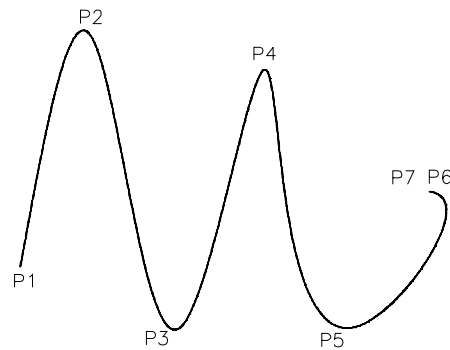
<start tangent>: *Select point P6.*

Specify next point or [Close/Fit tolerance]

<start tangent>: *Press ENTER to end the process of point specification.*

Specify start tangent: *Press ENTER to accept the default start tangent.*

Specify end tangent: *Select point P7.*



**Figure 17-10** Creating the spline

You can specify the start and end tangents to change the direction, in which the spline curve starts and ends at the **Specify start tangent:** and **Specify end tangent:** prompts. The **SPLINE** command options are discussed next.

## Options for Creating Splines

The options under this command for creating the splines are as follows.

### Object

This option allows you to change a 2D or 3D splined polyline into a NURBS. The original splined polyline is deleted if the system variable **DELOBJ** is set to 1, which is the default



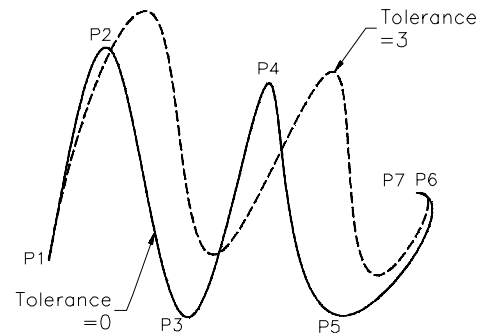
value of the variable. You can change a polyline into a splined polyline using the **Spline** option of the **PEDIT** command.

### 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.

### Fit Tolerance

This option allows you to control the fit of the spline between specified points. By default, this value is zero and so the spline passes through the points through which it is created. Using this option, you can specify some tolerance value that will govern the spline creation, see Figure 17-11. 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.



**Figure 17-11** Creating a spline with a **Fit Tolerance** of 0 and 3

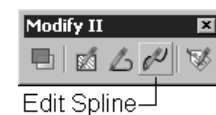
### Start and End Tangents

This allows you to control the tangency of the spline at the start point and endpoint of the spline. If you press ENTER at these prompts, AutoCAD LT will use the default value. By default, the tangency is determined by the slope of the spline at the specified point.

## EDITING SPLINES

<b>Toolbar:</b>	Modify II > Edit Spline
<b>Menu:</b>	Modify > Object > Spline
<b>Command:</b>	SPLINEDIT

The NURBS can be edited using the **SPLINEDIT** command (Figure 17-12). With this command, you can fit data in the selected spline, close or open the spline, move vertex points, and refine or reverse a spline. Apart from the ways mentioned in the preceding command box, you can also invoke the **SPLINEDIT** command by choosing **Spline Edit** from the shortcut menu that is displayed when you select a spline and right-click. The prompt sequence that will follow when you choose the **Edit Spline** button is given next.



**Figure 17-12** Choosing the **Edit Spline** button from the **Modify II** toolbar

Select spline: *Select the spline that is to be edited if not selected already, using the above-mentioned shortcut menu.*

Enter an option [Fit data/Close/Move vertex/Refine/rEverse/Undo]: *Select any one of the options.*

## Options for Editing Splines

The options under this command for editing the splines are described next.

### Fit Data

When you draw a spline, the spline is fit to 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, then select the **Tangents** option. The prompt sequence that will follow when you invoke this option is given next.

Select spline: *Select the spline that is to be edited.*

Enter an option [Fit data/Close/Move vertex/Refine/rEverse/Undo]: **F**

Enter a fit data option

[Add/Close/Delete/Move/Purge/Tangents/toLerance/eXit] <eXit>:

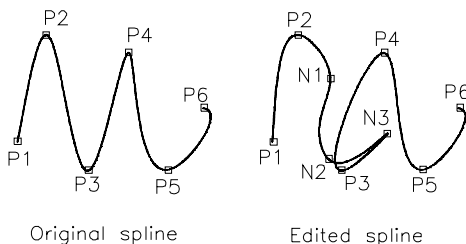
The start and end tangent points can be selected or their coordinates can be entered. The options available within the **Fit data** option are as follows.

**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 control point. This control point should be one of the existing control points on the spline. After selecting the existing control 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 17-13. If you select the start point or endpoints of the spline, only those points are 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.

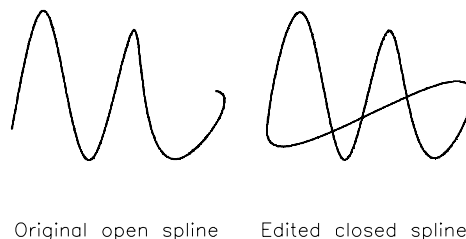
**Close/Open.** This option allows you to close an open spline or open a closed spline, see Figure 17-14. If the spline is open, the **Close** option is available and if the spline is closed, the **Open** option is displayed.

P = original control points

N = new points added



**Figure 17-13** Original spline and the edited spline after adding the data points



**Figure 17-14** An open spline and a closed spline

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

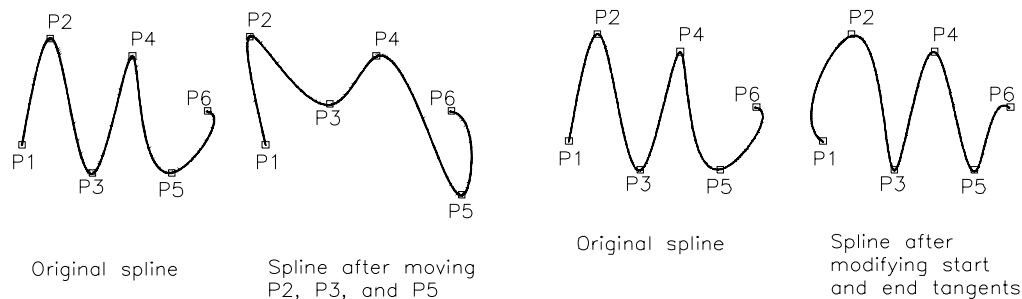
**Move.** You can move fit points by using this option. When you invoke this option, the start point of the spline is highlighted and the prompt sequence is as follows.

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 can exit the command. Figure 17-15 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 allows you 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. But, once the fit point data has been removed from a spline, editing a spline gets difficult. Also, the **Fit Data** option is not displayed when you again use the **SPLINEDIT** command on the edited spline.

**Tangents.** This option allows you to modify the tangents of the start and endpoints of the selected spline, see Figure 17-16. When you invoke this option, you will be prompted to specify the start and end tangents for the spline. You can specify the start and end tangents or use the systems default tangents.



**Figure 17-15** Original spline and the edited spline after moving the data points

**Figure 17-16** Original spline and the spline after modifying the start and end tangents

**tolerance.** This option allows you 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 allows you to exit the **Fit data** option of editing splines.

### Close/Open

This option allows you to close an open spline or open a closed spline. When you select the **Close** option, AutoCAD LT lets you open, move the vertex, or refine or reverse the spline.

### Move vertex

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. To display this frame with the spline, set the value of the **SPLFRAME** system variable to 1. The default value of this system variable is 0 and the frames are not displayed with the spline. The **Move vertex** option is similar to the **Move** option of the **Fit data** option of editing the splines. The prompt sequence that follows when you invoke this option is given next.

Enter an option [Fit data/Close/Move vertex/Refine/rEverse/Undo]: **M**  
Specify new location or [Next/Previous/Select point/eXit] <N>:

### Refine

This option allows you 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 [Fit data/Close/Move vertex/Refine/rEverse/Undo]: **R**  
Enter a refine option [Add control point/Elevate order/Weight/eXit] <eXit>:

**Add control point.** This option allows you 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.

**Elevate order.** This option allows you to increase the order of a spline curve. An order of the 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.

**Weight.** You can also add weight to any of the vertices of the spline by using this option. When weight is added to a particular vertex, the spline gets pulled more toward it. Similarly, a lower value of weight of a particular spline vertex will result in the spline getting pulled less toward 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 the distance through which the spline will remain tangent to it. By default, the spline gets pulled equally toward 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 a refine option [Add control point/Elevate order/Weight/eXit] <eXit>: **W**  
Enter new weight (current = 1.0000) or [Next/Previous/Select point/eXit] <N>:

### **rEverse**

This option allows you to reverse the spline direction. This implies that when you apply this option to a spline, its start point becomes its endpoint and vice versa.

### **Undo**

This option will undo the previous editing operation applied to a spline within the current session of the **SPLINEDIT** command. 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 your answers to those given at the end of this chapter:

1. You can specify whether or not the double line will be open or closed at the point where it intersects another double line. (T/F)
2. The **Dragline** option of the **DLINE** command can be used to specify the location of the pick point with respect to the double line. (T/F)
3. All the double line segments drawn using single **DLINE** command becomes a single entity. (T/F)
4. You can also draw an arc from within the **DLINE** command. (T/F)
5. The \_\_\_\_\_ command can be used to draw a revision cloud.
6. The \_\_\_\_\_ option of the **DLINE** command is used to specify whether or not the double line will be open or closed at the start and endpoints.
7. The revision cloud drawn using the **REVLOUD** command is a \_\_\_\_\_ entity.
8. The \_\_\_\_\_ option of the **Fit data** option of the **SPLINEDIT** command reduces the file size by removing the fit data of the selected splines.
9. NURBS is an acronym for \_\_\_\_\_.

10. The \_\_\_\_\_ option of the **SPLINE** command is used to convert a splined polyline into an actual spline.

### Review Questions

Answer the following questions:

1. The revision cloud is closed automatically when you take the cursor close to the start point of the revision cloud. (T/F)
2. Once the **Purge** option of the **Fit Data** option of the **SPLINEDIT** command is used on a spline, editing it gets difficult and the **Fit Data** option of the **SPLINEDIT** command is no longer available for the particular spline. (T/F)
3. You cannot specify the included angle for the arc created using the **DLINE** command. (T/F)
4. If the **Break** option is set to **ON**, the double line will be opened at the point where it intersects another double line. (T/F)
5. By default, what is the value of the **Dragline** option of the **DLINE** command?
  - (a) Center
  - (b) Left
  - (c) Right
  - (d) Top
6. Which option of the **DLINE** command is used to modify the distance between the two lines that comprise the double line?
  - (a) Caps
  - (b) Offset
  - (c) Width
  - (d) Snap
7. Which sub-option of the **Caps** option of the **DLINE** command caps all the ends of the double line except the ends that are snapped to an object?
  - (a) Both
  - (b) Left
  - (c) Right
  - (d) Auto
8. Using which option of the **REVCLOUD** command can you convert an existing closed loop into a revision cloud?
  - (a) **Object**
  - (b) **Select**
  - (c) **Length**
  - (d) **None**

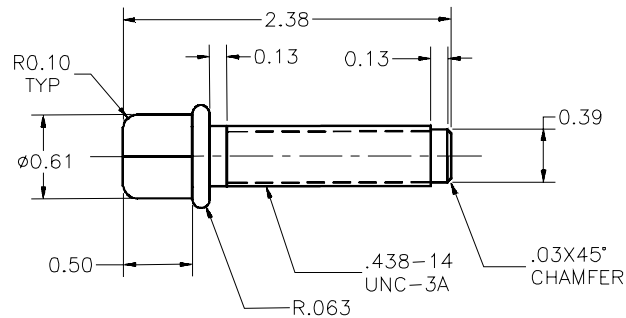
9. Which option can be used to reverse the direction of spline creation while editing the splines?
- |                     |                   |
|---------------------|-------------------|
| (a) <b>Fit data</b> | (b) <b>Refine</b> |
| (c) <b>Reverse</b>  | (d) <b>Weight</b> |
10. The \_\_\_\_\_ option is used to edit the data points of the spline.
11. The value of the \_\_\_\_\_ variable is set to 1 to display the frames of the splines.
12. If the Fit tolerance value of a spline is \_\_\_\_\_, the spline passes exactly through its fit points.
13. The \_\_\_\_\_ option is used to pull the spline more towards the vertex.
14. The \_\_\_\_\_ sub-option of the **Refine** option of the **SPLINEDIT** command is used to increase the order of the spline curve.
15. The \_\_\_\_\_ sub-option of the **Fit Data** option is used to remove the fit points from the spline.

## Exercises

### Exercise 1

*Mechanical*

Create the drawing shown in Figure 17-17. Assume the missing dimensions.



*Figure 17-17 Drawing for Exercise 1*

### Exercise 2

*Architectural*

Create the drawing shown in Figure 17-18. Use the **DLINE** command to draw the walls. The wall thickness is 12 inches. Assume the missing dimensions.



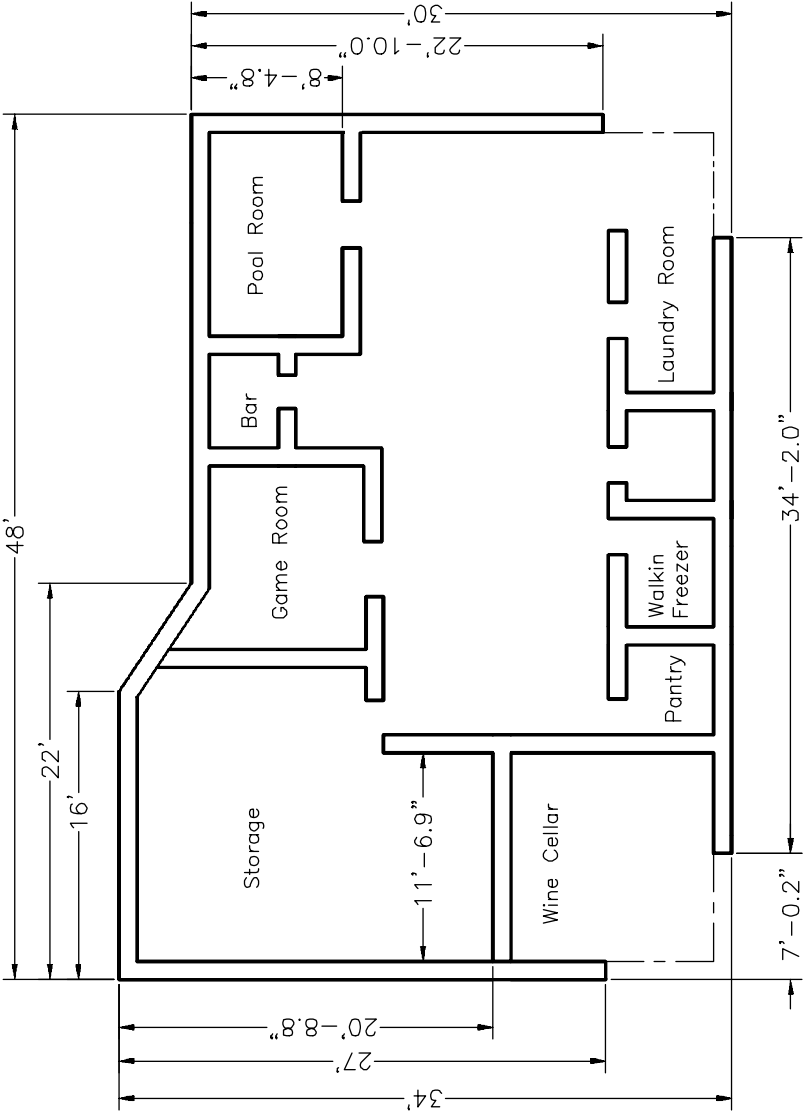
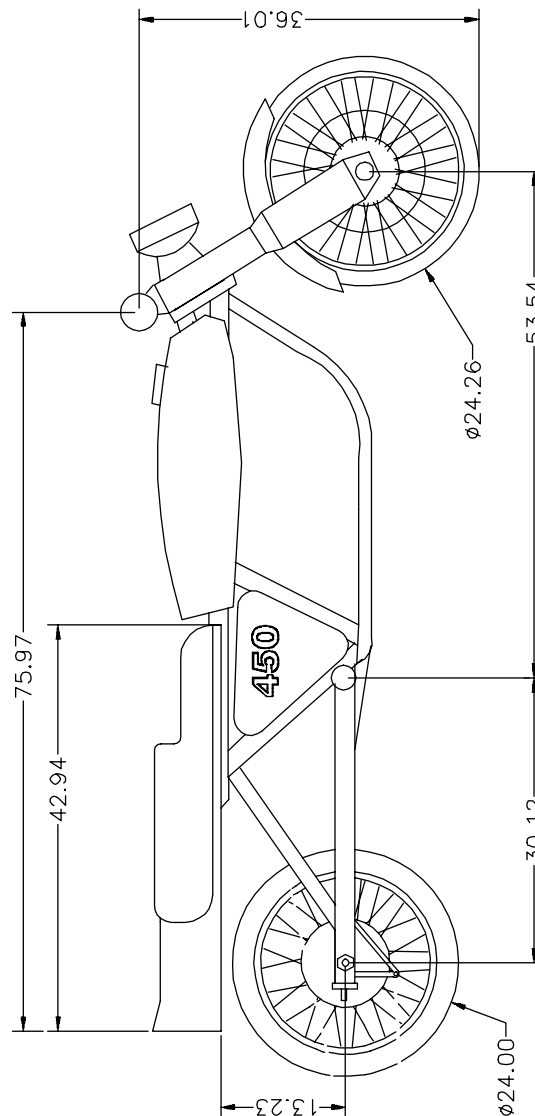


Figure 17-18 Drawing for Exercise 2

**Problem Solving Exercise 1***General*

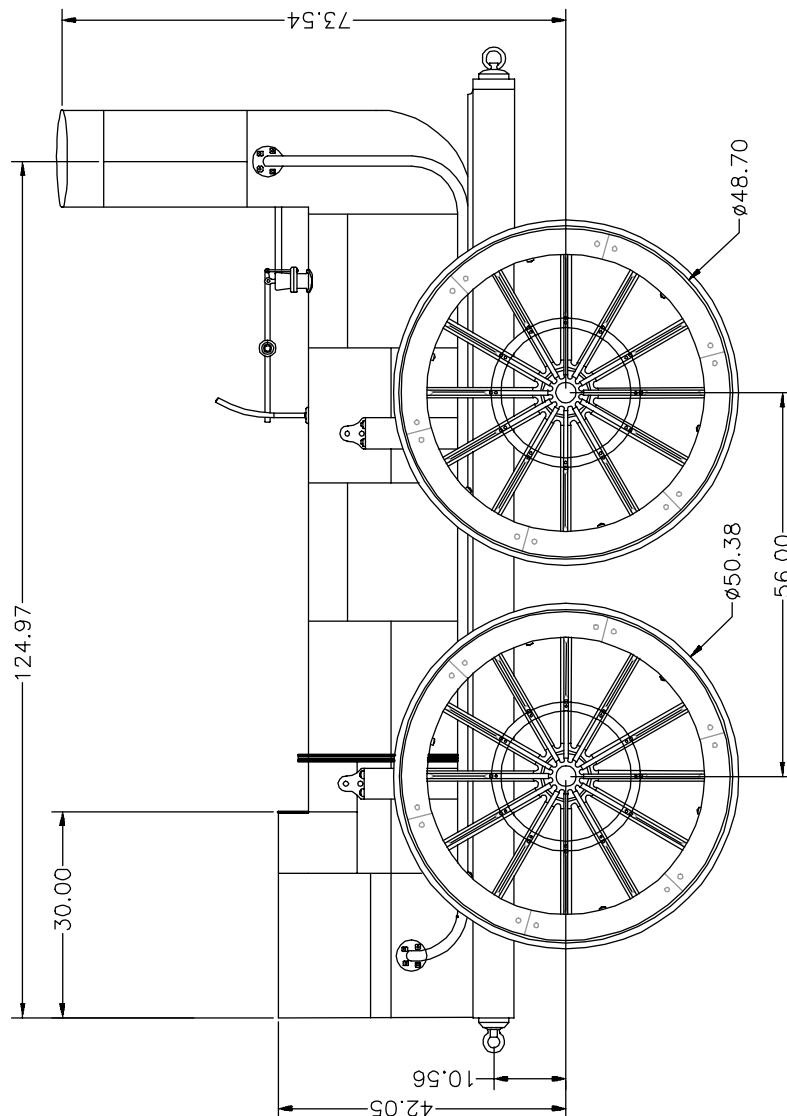
Create the drawing shown in Figure 17-19. Some of the reference dimensions are given in the drawing. Assume the missing dimensions so that the drawing looks similar to the one given below.



**Figure 17-19** Drawing for Problem Solving Exercise 1

**Problem Solving Exercise 2***General*

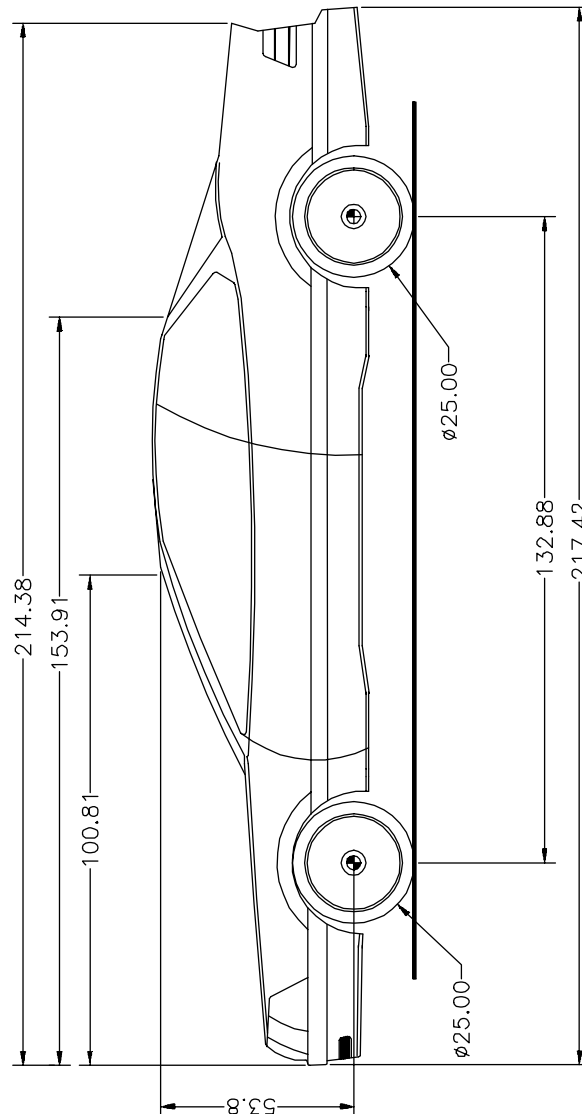
Create the drawing shown in Figure 17-20. Some of the reference dimensions are given in the drawing. Assume the missing dimensions so that the drawing looks similar to the one given below.



*Figure 17-20 Drawing for Problem Solving Exercise 2*

**Problem Solving Exercise 3***General*

Create the drawing shown in Figure 17-21. Some of the reference dimensions are given in the drawing. Assume the missing dimensions so that the drawing looks similar to the one given below.



**Figure 17-21** Drawing for Problem Solving Exercise 3

**Problem Solving Exercise 4***Architecture*

Create the drawing shown in Figure 17-22. Some of the reference dimensions are given in the drawing. Assume the missing dimensions so that the drawing looks similar to the one given below.

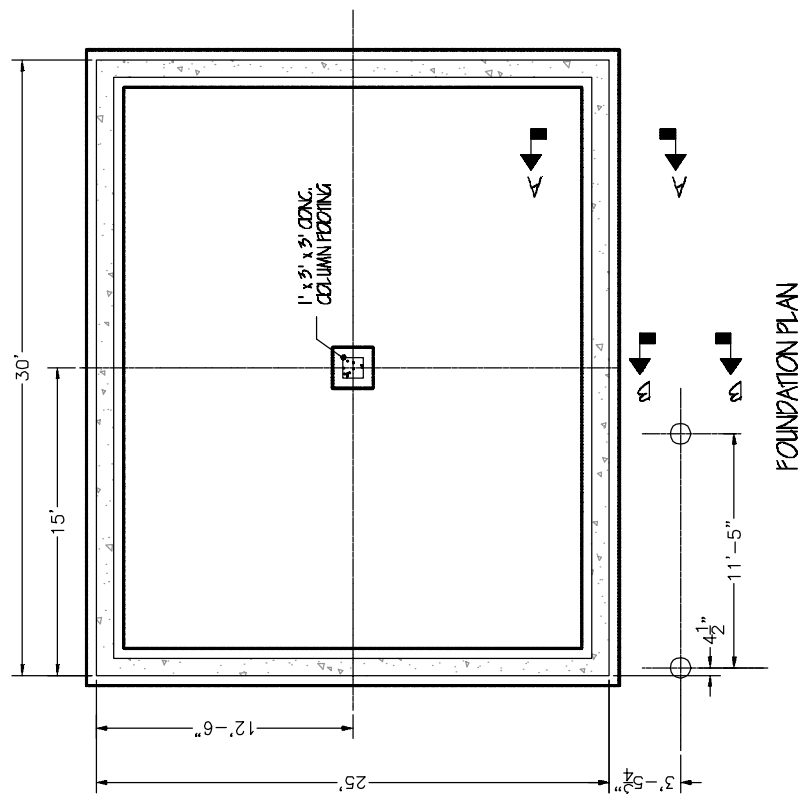


Figure 17-22 Drawing for Problem Solving Exercise 4

## Problem Solving Exercise 5

Architecture

Create the drawing shown in Figure 17-23. Some of the reference dimensions are given in the drawing. Assume the missing dimensions so that the drawing looks similar to the one given below.

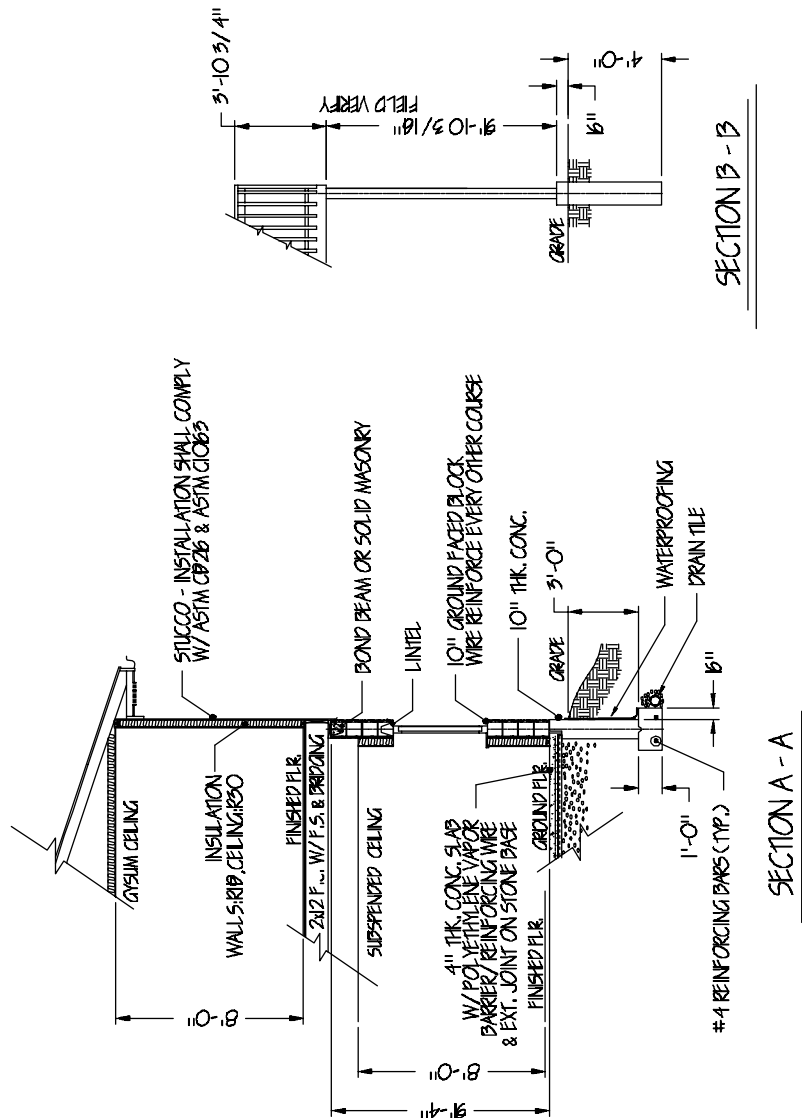


Figure 17-23 Drawing for Problem Solving Exercise 5

**Problem Solving Exercise 6***Architecture*

Create the drawing shown in Figure 17-24. Some of the reference dimensions are given in the drawing. Assume the missing dimensions so that the drawing looks similar to the one given below.

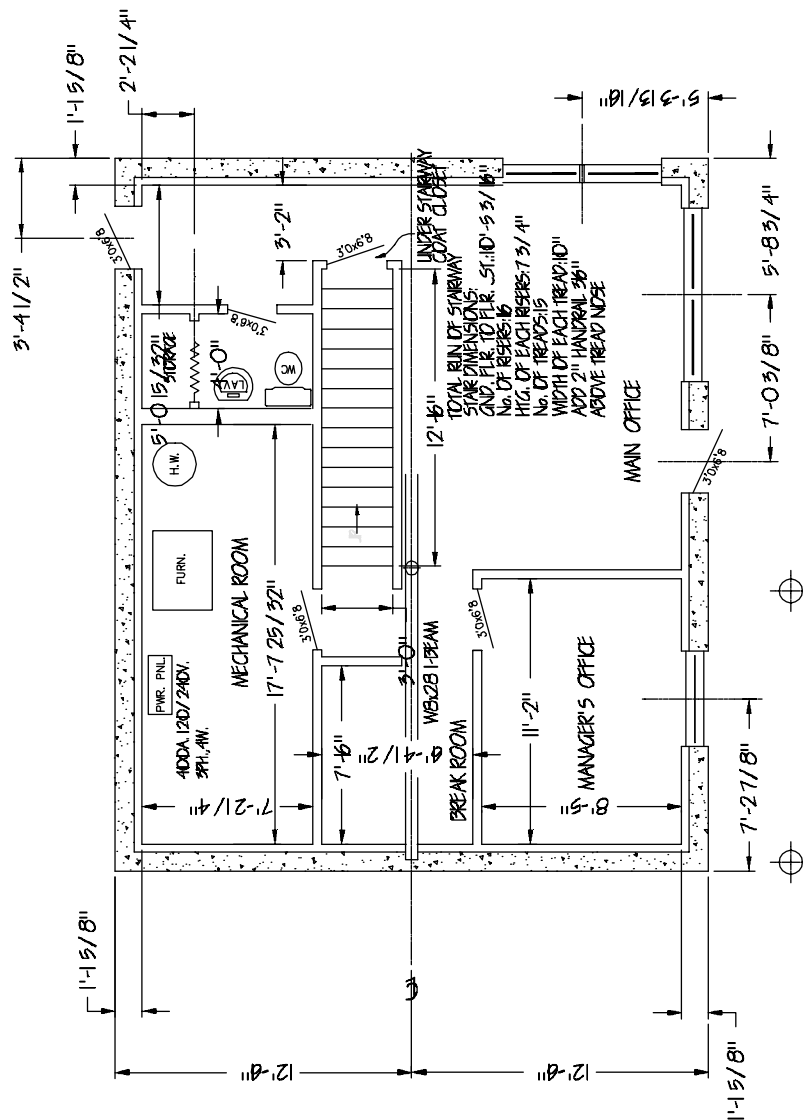
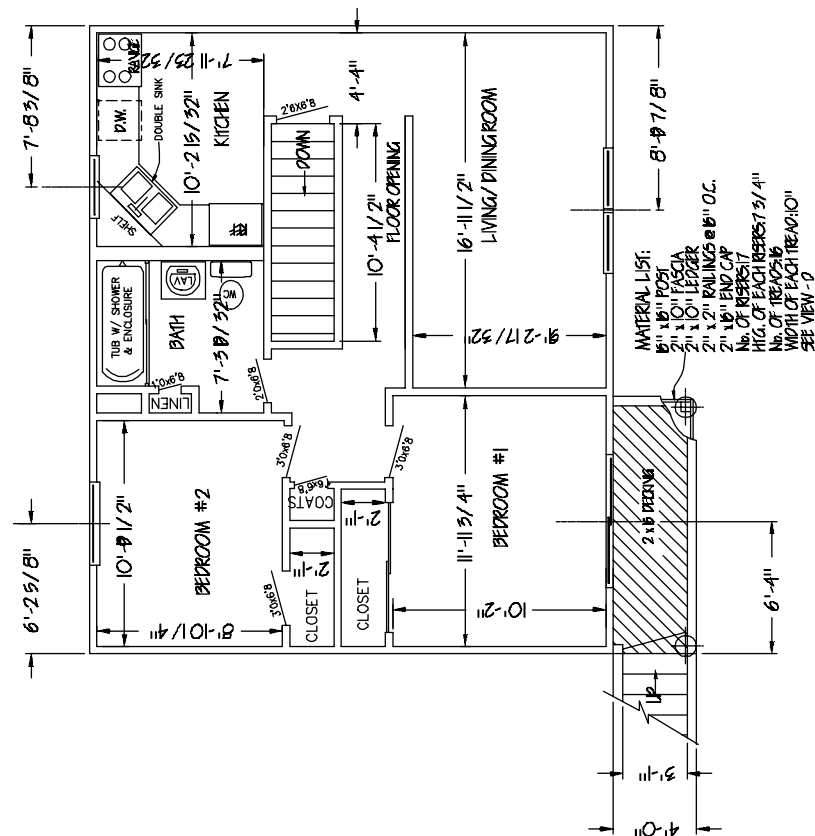


Figure 17-24 Drawing for Problem Solving Exercise 6

Create the drawing shown in Figure 17-25. Some of the reference dimensions are given in the drawing. Assume the missing dimensions so that the drawing looks similar to the one given below.

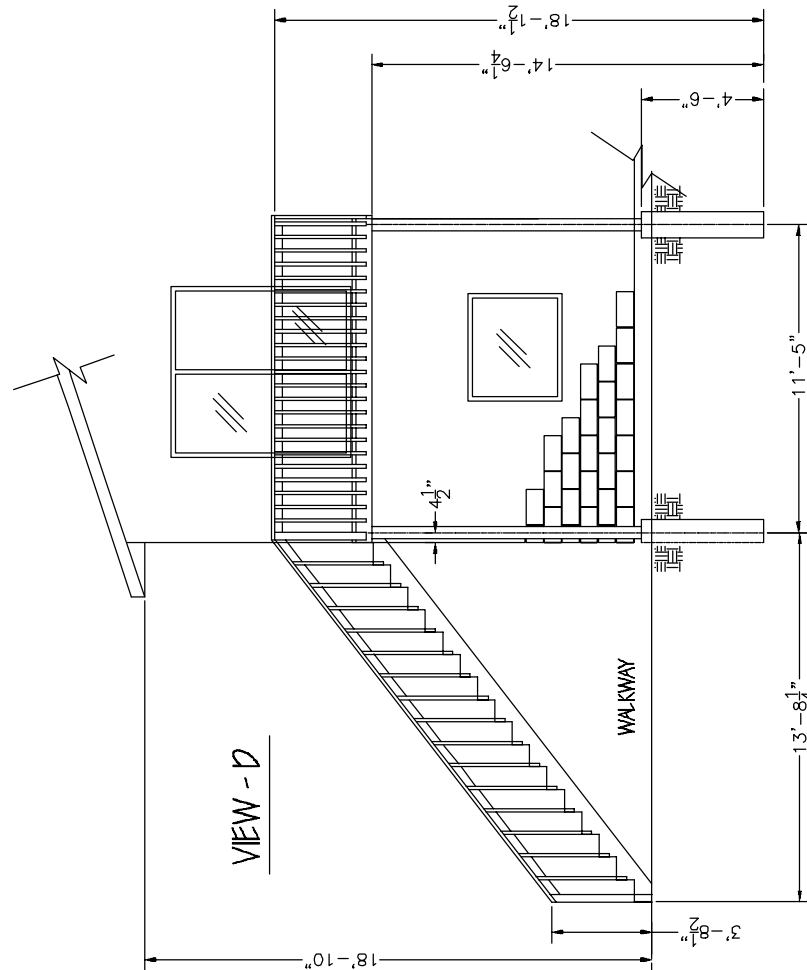


*Figure 17-25 Drawing for Problem Solving Exercise 7*



**Problem Solving Exercise 8***Architecture*

Create the drawing shown in Figure 17-26. Some of the reference dimensions are given in the drawing. Assume the missing dimensions so that the drawing looks similar to the one given below.



**Figure 17-26** Drawing for Problem Solving Exercise 8

**Answers to Self-Evaluation Test**

**1** - T, **2** - T, **3** - F, **4** - T, **5** - REVCLLOUD, **6** - Caps, **7** - single, **8** - **Purge**, **9** - Non Uniform Rational Bezier-Spline, **10** - **Object**

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