AutoCAD® 2022 FOR INTERIOR DESIGN AND SPACE PLANNING

THE DESIGN APPROACH

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Sample pages

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3 chapterthree

Drawing with AutoCAD: Conference and Lecture Rooms

CHAPTER OBJECTIVES

- Correctly use the following commands and settings:

BREAK	FILLET
CHAMFER	From
COPY	Grips
Distance	HATCH
DIVIDE	ID Point
Drawing Template	MEASURE
EXPLODEEXTEND	MIRROR
OFFSET	Polyline Edit
OSNAP	Rectangle
POINT	RECTANGULAR ARRAY
POLAR ARRAY	ROTATE
POLYGON	Tracking
Polyline	TRIM

- Draw using polar tracking.
- Use Point Style to set the appearance of points.

EXERCISE 3-1
Drawing a Rectangular Conference Room, Including Furniture

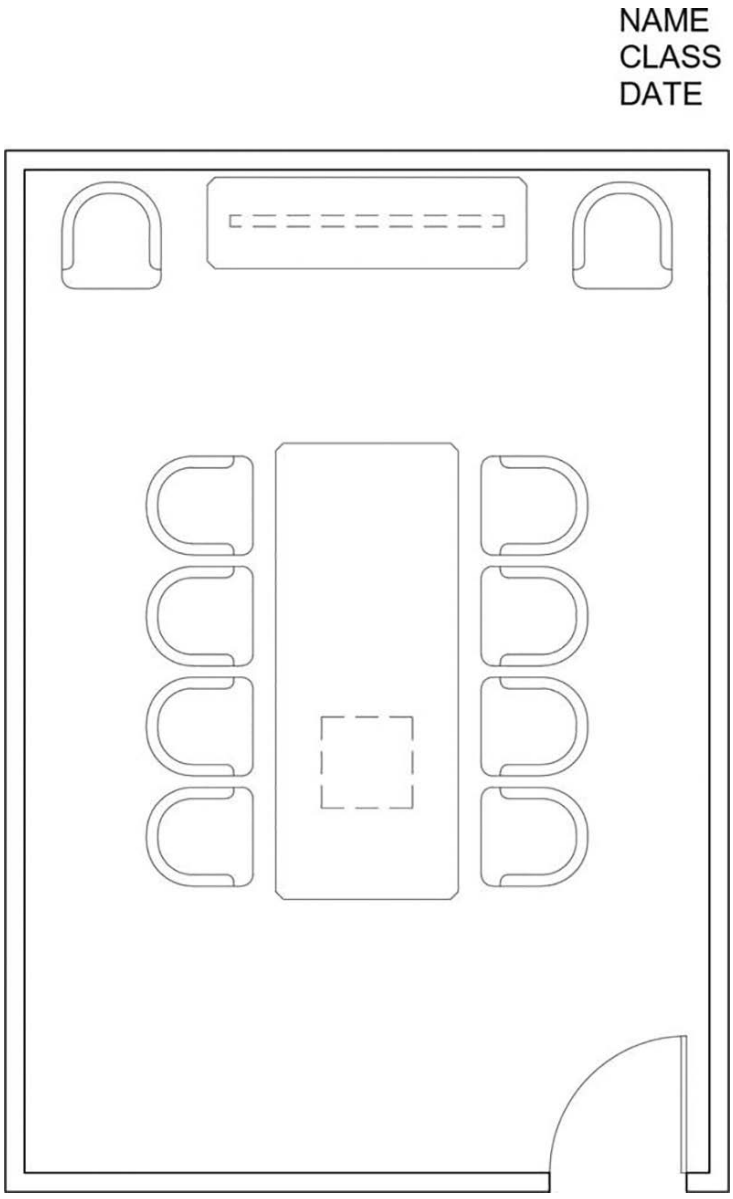
In Exercise 3-1, you learn to draw a conference room, including walls and furnishings. When you have completed Exercise 3-1, your drawing will look similar to Figure 3-1.

Sample pages

Step 1. Use your workspace to make the following settings:

1. Use **Save As...** to save the drawing with the name **CH3 EXERCISE1**
2. Set drawing units: **Architectural**

Figure 3-1
Exercise 3-1: Drawing a rectangular conference room, including furniture (scale: 1/4″ = 1′-0″)



3. Set drawing limits: **25′,35′** (Don't forget the foot marks.)
4. Set **GRIDDISPLAY**: **0**
5. Set grid: **12″**
6. Set snap: **6″**
7. Create the following layers:

Layer name	Color	Linetype	Lineweight
a-anno-text	green	continuous	.006[dp] (.15 mm)
a-door	red	continuous	.004[dp] (.09 mm)
a-wall-intr	blue	continuous	.010[dp] (.25 mm)
i-eqpm-ovhd	red	hidden	.004[dp] (.09 mm)
i-furn	cyan	continuous	.004[dp] (.09 mm)

Sample pages

TIP

You can create the layers by clicking the **New layer** icon and then typing the layer names separated by a comma. When you type the comma, the **Name** list moves to the next layer, and you can type the next layer name.

8. Set layer **a-wall-intr** current.
9. Use **Zoom-All** to view the limits of the drawing.
10. Turn **SNAP**, **GRID**, and **LWDISPLAY** on. The remaining buttons in the status bar are off.

Making a Drawing Template

drawing template: A drawing used to ensure consistency by providing standard styles and settings.

You will be able to use these settings for the remaining tutorials in this chapter. Making a ***drawing template*** of the settings will save you the time of setting up Exercises 3-2, 3-3, and 3-4.

Step 2. Save the drawing as a template on the drive and/or folder in which you want to save (Figures 3-2 and 3-3), as described next:

Prompt	**Response**
Type a command:	**Save As...**
The **Save Drawing As** dialog box appears:	Click the down arrow in the **Files of type:** input box and click **AutoCAD Drawing Template (*.dwt)** Type **Ch3-conference-rm-setup** (in the **File name:** input box so the **Save Drawing As** dialog box appears as shown in Figure 3-2). Notice the text in the **Save in:** input box has changed to **Template** Click the down arrow in the **Save in:** input box and highlight the drive and folder in which you want to save Click **Save**
The **Template Options** dialog box appears (Figure 3-3):	Type **Setup for Ch3 conference rooms** (as shown in Figure 3-3) Click **OK**

NOTE

Remember to save often to avoid losing your work. Backing up your work by saving on two drives is always a good idea.

Sample pages

Figure 3-2
Save the drawing as a template

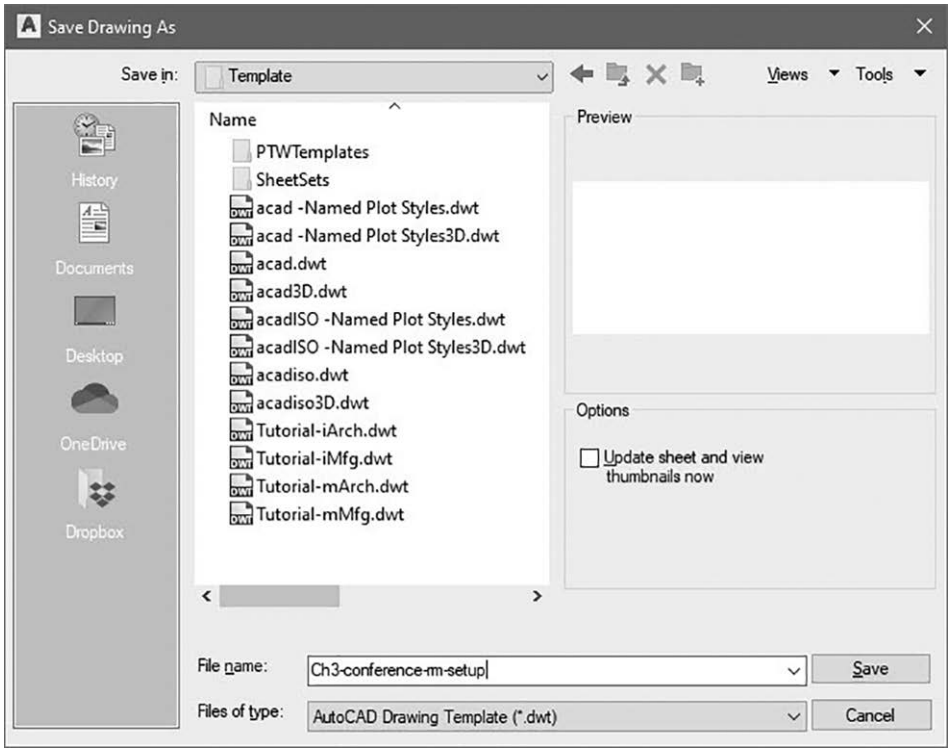
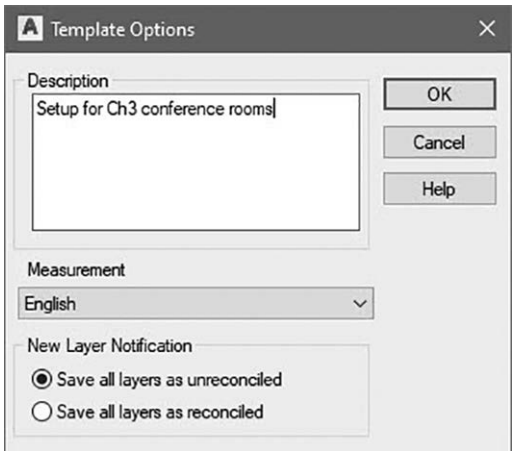


Figure 3-3
Template Options dialog box



Step 3. The drawing remains as a template in the **Template** folder, so you must save it again as a drawing file. Save the drawing as a drawing file on the drive and/or folder in which you want to save, as described next:

Prompt	**Response**
Type a command:	**Save As...**
The **Save Drawing As** dialog box appears:	Click the down arrow in the **Files of type:** input box and click **AutoCAD 2018 Drawing (*.dwg)**. (Remember, **AutoCAD 2018 Drawing** is the drawing file format used by AutoCAD 2022.)

Sample pages

Prompt	Response
	Click the down arrow in the **Save in:** input box and highlight the drive and folder in which you want to save
	Click **CH3-EXERCISE1** (to appear in the **File name:** input box)
	Click **Save**
The **Save Drawing As** dialog box appears saying *The drawing already exists. Do you want to replace it?*	
	Click **Yes**

Polyline

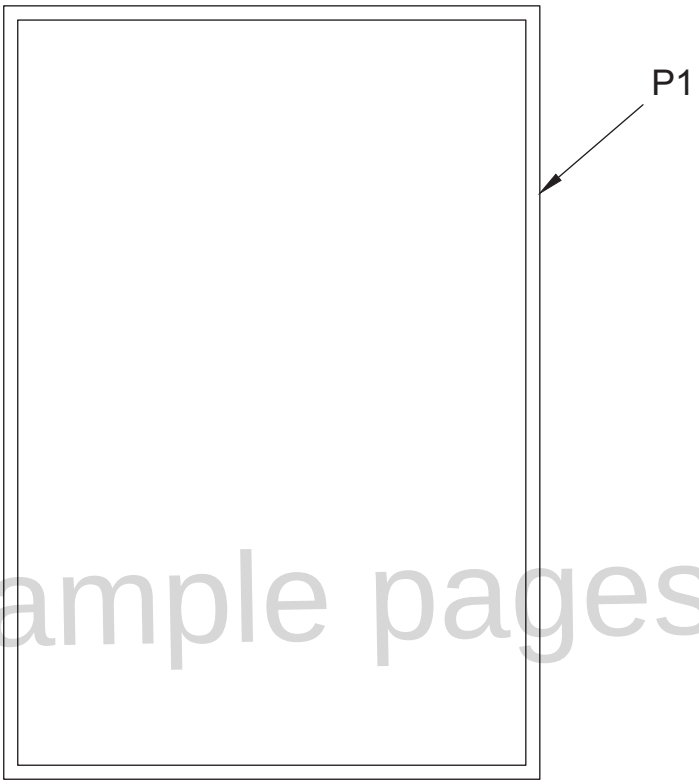
polyline: A continuous line or arc composed of one or more segments, the width of which can be changed.

POLYLINE	
Ribbon/Panel	Home/ Draw
Draw Toolbar:	
Menu Bar:	Draw/ Polyline
Type a Command:	PLINE
Command Alias:	PL

Begin by drawing the conference room walls using the **Polyline** command. A ***polyline*** is different from a regular line in that regardless of the number of segments that make up a polyline, AutoCAD treats a polyline drawn with one operation of the **Polyline** command as a single entity. This is especially helpful when you are drawing walls, because after you draw the outline of a single room or entire building, you can offset the entire polyline to show the thickness of the walls.

Step 4. Use **Polyline** to draw the inside lines of the conference room walls (Figure 3-4), as described next:

Prompt	Response
Type a command:	**Polyline** (or type **PL<Enter>**)
Specify start point:	Type **5′,5′ <Enter>**



Sample pages

Figure 3-4
Draw the conference room walls

Prompt	Response
	(You have just entered absolute coordinates; the polyline starts 5′ to the right on the x-axis and 5′ up on the y-axis.)
	Set **ORTHO** on (press **<F8>** or click **ORTHO**)
Current line-width is 0′-0″. Specify next point or [Arc Halfwidth Length Undo Width]:	Move your mouse to the right and type **15′ <Enter>** (direct distance entry)
Specify next point or [Arc Close Halfwidth Length Undo Width]:	Move your mouse up and type **22′ <Enter>**
Specify next point or [Arc Close Halfwidth Length Undo Width]:	Move your mouse to the left and type **15′ <Enter>**
Specify next point or [Arc Close Halfwidth Length Undo Width]:	Type **C <Enter>**

Undo

The **Polyline Undo** option is similar to the **LINE** command. If you do not like the last polyline segment drawn, use the **Undo** option to erase it and continue with the *Specify next point or [Arc Close Halfwidth Length Undo Width]:* prompt.

You can enter any of the capitalized options in the **Polyline** prompt by typing the letters in either upper- or lowercase, or you can simply click the option in the command-line window. The remaining options in the **Polyline** prompt are described later in this chapter.

OFFSET

Because the polyline is treated as a single entity, when you click one point on the polyline, you are able to offset the entire outline of the conference room at once. If the outline of the room had been drawn with the **LINE** command, using the **OFFSET** command would offset each line segment individually, and the corners would not meet.

Step 5. Use the **OFFSET** command to draw the outside line (showing depth) of the conference room walls (Figure 3-4), as described next:

OFFSET	
Ribbon/Panel	Home/ Modify
Draw Toolbar:	
Menu Bar:	Modify/ Offset
Type a Command:	OFFSET
Command Alias:	O

Prompt	Response
Type a command:	**Offset** (or type **O <Enter>**)
Specify offset distance or [Through Erase Layer] <Through>:	Type **5 <Enter>**
Select object to offset or [Exit Undo] <Exit>:	Click anyplace on the polyline
Specify point on side to offset or [Exit Multiple Undo] <Exit>:	**P1→** (outside the rectangle, Figure 3-4)
Select object to offset or [Exit Undo] <Exit>:	**<Enter>**

Sample pages

The four options in the **Offset** prompt are **offset distance**, **Through**, **Erase**, and **Layer**. To complete the conference room wall, 5″ was set as the offset distance. To use any of the other options, type and enter the capital letter shown for the option in the command line or press **<Enter>** to start the **<Through>** default option.

Through

When you start the **Through** option and select the object to be offset, AutoCAD prompts: *Specify through point or [Exit Multiple Undo] <Exit>:*. You respond by clicking a point on the drawing through which you want the object to be offset.

Erase

When you start the **Erase** option, AutoCAD prompts: *Erase source object after offsetting? [Yes No] <No>:*. You can then respond with **Yes** or **No,** and AutoCAD continues by asking you to specify the offset distance, object to offset, and point on side to offset.

Layer

When you start the **Layer** option, AutoCAD prompts: *Enter layer option for offset objects [Current Source] <Source>:*. You can then respond with the selection of current or source layer, and AutoCAD continues by asking you to specify the offset distance, object to offset, and point on side to offset.

EXPLODE

EXPLODE	
Ribbon/ Panel	Home/ Modify
Draw Toolbar:	
Menu Bar:	Modify/ Explode
Type a Command:	EXPLODE
Command Alias:	X

Because the polyline is treated as a single entity, it must be "exploded" before individual line segments can be edited. The **EXPLODE** command splits the solid polyline into separate line segments. After the polyline is exploded into separate line segments, you will be able to add the conference room door.

Step 6. Use the **EXPLODE** command to split the two polylines that make the conference room walls, as described next:

Prompt	**Response**
Type a command:	**Explode** (or type **X <Enter>**)
Select objects:	Click anyplace on the outside polyline
Select objects:	Click anyplace on the inside polyline
Select objects:	**<Enter>**

After you use the **EXPLODE** command, the walls do not look different, but each line segment is now a separate entity.

ID POINT	
Ribbon/ Panel	Home/ Utilities
Draw Toolbar:	
Menu Bar:	Tools/ Inquiry/ ID Point
Type a Command:	ID

ID Point

Sample pages

A useful command, **ID Point** (located under the expanded **Utilities** panel of the **Home** tab on the ribbon) allows you to locate a point on a drawing and have the position of the point displayed in coordinates. AutoCAD remembers the coordinate location of the point. You can initiate a command, such as **LINE,** *immediately* after the **ID Point** command has located a point on the drawing. You can enter the start point of the **LINE** command by using

relative or polar coordinates, or you may also use direct distance entry to specify a distance from the established ID point location. Alternatively, you can use the **From** option of the **Osnap** menu (shown later in Figure 3-16 and used in Step 39) to define a reference point and then define the x- and y-offset from that point. The upcoming steps explain this further. Let's continue with the exercise using **ID Point**.

Step 7. Use **Zoom-Window** to magnify the lower-right corner of the conference room where the door will be located.

Step 8. Use **ID Point** to locate a point on the drawing. Use **LINE** to draw the right side of the door opening (Figure 3-5), as described next:

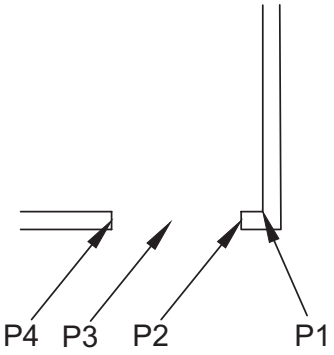


Figure 3-5
Draw the door opening

Prompt	Response
Type a command:	**ID Point** (or type **ID <Enter>**)
Specify point:	**P1→** (with **SNAP** on, snap to the inside lower-right corner of the conference room, Figure 3-5)
Point: X = 20′-0″ Y = 5′-0″ Z = 0′-0″	
Type a command:	Type **L <Enter>**
Specify first point:	Type **@6<180 <Enter>** (you have just entered polar coordinates; move your mouse so you can see where the line is attached)
Specify next point or [Undo]:	Type **@5<–90 <Enter>** (using polar coordinates; the line 5″) is extended downward
Specify next point or [Undo]:	**<Enter>**

TIP

Instead of typing **@5<–90 <Enter>,** type **PER <Enter>** and draw the line down until it intersects at a 90° angle with the outside line of the wall. This is an **Object Snap** mode (**Perpendicular**), which is described more fully later in this exercise.

Step 9. Offset the line 3′ to the left to form the door opening, as described next:

Prompt	Response
Type a command:	**Offset** (or type **O <Enter>**)
Specify offset distance or [Through Erase Layer] <0′-5″>:	Type **3′ <Enter>**
Select object to offset or [Exit Undo]<Exit>:	**P2→** (the 5″ line you just drew; Figure 3-5)
Specify point on side to offset or [Exit Multiple Undo]<Exit>:	**P3→** (pick to the left)
Select object to offset or [Exit Undo]<Exit>:	**<Enter>**

Sample pages

TRIM

TRIM	
Ribbon/ Panel	Home/ Modify
Draw Toolbar:	
Menu Bar:	Modify/ Trim
Type a Command:	TRIM
Command Alias:	TR

Watch the **Trim** prompts carefully. You cannot pick the objects to trim until you have selected all cutting edges (the edge to which the object is trimmed) and pressed the **<Enter>** key, so that the prompt *Select object to trim or shift-select to extend or [Fence Crossing Project Edge eRase Undo]:* appears. If you are unable to trim an object because it does not intersect a cutting edge, and you have selected **all** as the cutting edges, hold the **<Shift>** key down and click on the entity to extend while still in the **TRIM** command.

NOTE

Press **<Enter>** at the **Trim** prompt *Select objects or <select all>:* to select all objects as cutting edges.

Step 10. Use the **TRIM** command to trim the horizontal wall lines between the two 5″ vertical lines that represent the door opening (Figure 3-5), as described next:

Prompt	Response
Type a command:	**Trim** (or type **TR <Enter>**)
Current settings: Projection = UCS Edge = None, Mode = Quick	
Select object to trim or Shift-select to extend or [cuTting edges Crossing mOde Project eRase]	Click **CuTting edges** in the command-line window, or type **T**
Select objects or <select all>:	**P2→** (the 5″ vertical line; Figure 3-5)
Select objects: 1 found	
Select objects:	**P4→** (the second 5″ vertical line)
Select objects: 1 found, 2 total	
Select objects:	**<Enter>**
Select object to trim or Shift-select to extend or [Fence Crossing Project Edge eRase Undo]:	Click the two horizontal wall lines between **P2→** and **P4→** (Figure 3-5)
	<Enter> (to complete the command)

Step 11. Set layer **a-door** current.

Rectangle

RECTANGLE	
Ribbon/Panel	Home/ Draw
Draw Toolbar:	
Menu Bar:	Draw/ Rectangle
Type a Command:	RECTANG
Command Alias:	REC

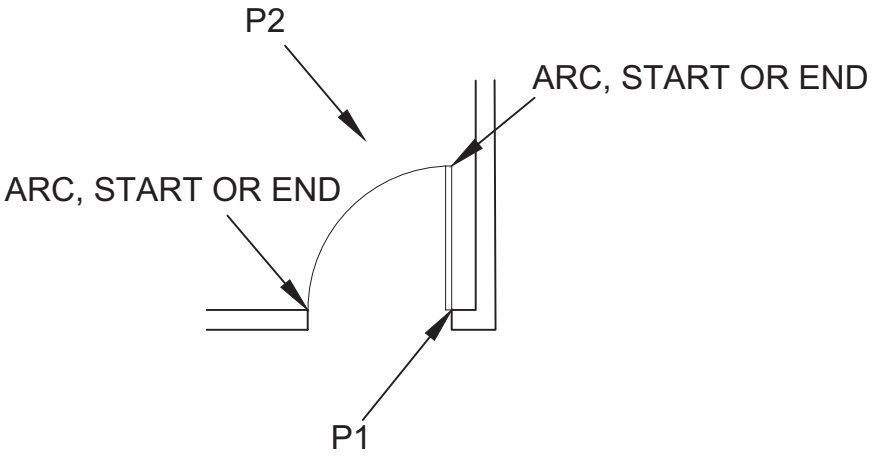
Use the Rectangle command to create a door for the floor plan.

Step 12. Draw a 1-1/2″-long by 3′-wide rectangle to represent the door (Figure 3-6):

Prompt	Response
Type a command:	**Rectangle** (or type **REC <Enter>**)
Specify first corner point or [Chamfer Elevation Fillet Thickness Width]:	**P1→** (be sure **SNAP** is on); snap to the upper-right corner of the door opening to begin the rectangle

Sample pages

Figure 3-6
Draw the door using **Rectangle** and **Arc-Start, End, Direction** commands



Prompt	Response
Specify other corner point or [Area Dimensions Rotation]:	Type **D <Enter>**
Specify length for rectangle <0′-1-1/2″>:	Type **1-1/2 <Enter>**
Specify width for rectangle <3′-0″>:	Type **3′ <Enter>**
Specify other corner point or [Area Dimensions Rotation]:	**P2→** (pick any point to the left of the door symbol so the rectangle appears as shown in Figure 3-6)

Step 13. Use the **Arc-Start, End, Direction** method to draw the door swing arc. Be sure **SNAP** and **ORTHO** are on. The arc can be drawn clockwise or counterclockwise. Move your mouse so the direction of the arc appears, as shown in Figure 3-6.

TIP

The default setting for the **Rectangle** command when the **Dimension** option is selected is:

Default Rotation setting of 0:
Length is the x direction value.
Width is the y direction value.

When you change the Rotation setting to 90:
Length is the y direction value.
Width is the x direction value.

When the rectangle is visible, and the prompt *Specify other corner point:* appears, you change the position of the rectangle by moving your mouse right or left, up or down.

Step 14. Set layer **i-furn** current. Use **Zoom-Extents.**

Sample pages

Step 15. Use the **Polyline** command to draw a credenza (84″ long by 24″ deep) centered on the 15′ rear wall of the conference room, 2″ away from the wall. Locate an ID point by snapping to the inside upper-left corner of the conference room. Start the polyline **@48,-2** (relative coordinates) away from the point. Finish drawing the credenza by using direct distance entry. You can use feet or inches. Remember, AutoCAD defaults to inches in architectural units, so use the foot (′) symbol if you are using feet. Be sure to draw the credenza using one operation of **Polyline** so it is one continuous polyline. Use the **Close** option for the last segment of the polyline (Figure 3-7).

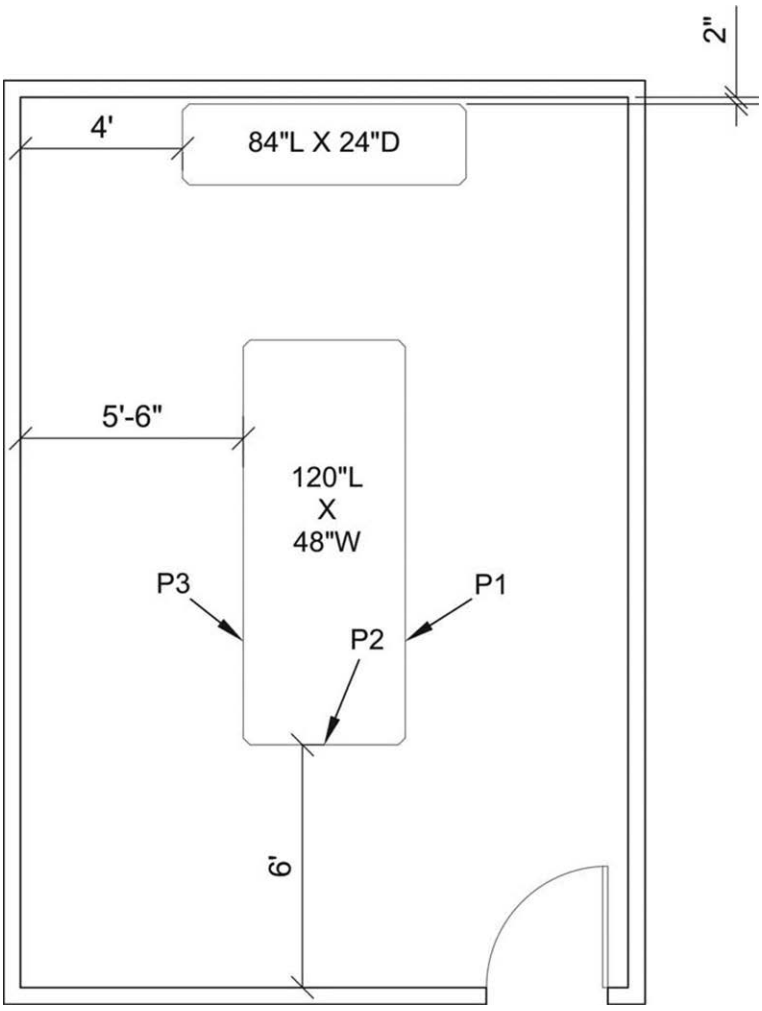


Figure 3-7
Draw a credenza and a conference table; chamfer the corners

Step 16. Draw a conference table 120″ long by 48″ wide using the **LINE** command. You can determine the location of the first point by using **ID Point** or by using grid and snap increments. Use direct distance entry to complete the table. Refer to Figure 3-7 for the location of the table in the room.

Step 17. Zoom in on the table.

Sample pages

CHAMFER

CHAMFER	
Ribbon/ Panel	Home/ Modify (Fillet drop-down)
Draw Toolbar:	
Menu Bar:	Modify/ Chamfer
Type a Command:	CHAMFER
Command Alias:	CHA

A **chamfer** is an angle (usually 45°) formed at a corner. The following steps will use the **CHAMFER** command to make the beveled corners of the conference table and credenza.

Step 18. Use the **CHAMFER** command to bevel the corners of the table (Figure 3-7), as described next:

Prompt	Response
Type a command:	**Chamfer** (or type **CHA <Enter>**)
(TRIM mode) Current chamfer Dist1 = 0′-0″ Dist2 = 0′-0″	
Select first line or [Undo Polyline Distance Angle Trim mEthod Multiple]:	Type **D <Enter>**
Specify first chamfer distance <0′-0″>:	Type **2 <Enter>**
Specify second chamfer distance <0′-2″>:	**<Enter>**
Select first line or [Undo Polyline Distance Angle Trim mEthod Multiple]:	**P1→** (Figure 3-7)
Select second line or Shift-select to apply corner or [Distance Angle mEthod]:	**P2→**
Type a command:	**<Enter>** (repeat **CHAMFER**)
(TRIM mode) Current chamfer Dist1 = 0′-2″, Dist2 = 0′-2″	
Select first line or [Undo Polyline Distance Angle Trim mEthod Multiple]:	**P2→**
Select second line or Shift-select to apply corner:	**P3→**

NOTE

Type **M <Enter>** (for **Multiple**) at the **Chamfer** prompt so you do not have to repeat the **CHAMFER** command.

Step 19. Chamfer the other corners of the table (Figure 3-7).

Step 20. Zoom in on the credenza.

Polyline

Because you drew the credenza using one operation of the **Polyline** command and used the **Close** option to complete the credenza rectangle, it is treated as a single entity. The **CHAMFER** command **Polyline** option chamfers all corners of a continuous polyline with one click.

Sample pages

Undo

Undo allows you to undo the previous chamfer.

Angle

The Angle option of the **CHAMFER** command allows you to specify an angle and a distance to create a chamfer.

Trim

The Trim option of both the **CHAMFER** and **FILLET** commands allows you to specify that the part of the original line removed by the chamfer or fillet remains as it was. To do this, type **T <Enter>** at the **Chamfer** prompt and **N <Enter>** at the *Trim/No trim <Trim>:* prompt. Test this option on a corner of the drawing so you know how it works. Be sure to return it to the **Trim** option.

mEthod

The **mEthod** option of the **CHAMFER** command allows you to specify whether you want to use the **Distance** or the **Angle** method to specify how the chamfer is to be drawn. The default is the **Distance** method.

Multiple

Multiple allows you you to chamfer multiple corners without repeating the **CHAMFER** command.

Step 21. Use chamfer distance 2″ to chamfer the corners of the credenza (Figure 3-7), as described next:

Prompt	**Response**
Type a command:	**Chamfer**
(TRIM mode) Current chamfer Dist1 = 0′-2″, Dist2 = 0′-2″	
Select first line or [Undo Polyline Distance Angle Trim mEthod Multiple]:	Type **P <Enter>** (accept 2″ distances as previously set)
Select 2D polyline or [Distance Angle mEthod]:	Click anyplace on the credenza
Four lines were chamfered	

TIP

If the last corner of the credenza does not chamfer, this is because the **Close** option of the **Polyline** command was not used to complete the polyline rectangle. Explode the credenza and use the **CHAMFER** command to complete the chamfered corner.

NOTE

While in the **CHAMFER** command, hold down the **<Shift>** key to select any two lines that do not meet, and you can make 90° corners of those two lines. This is the same as a 0 chamfer distance but will work regardless of the chamfer distance set.

Sample pages

When setting the chamfer distance, you can set a different distance for the first and second chamfers. The first distance applies to the first line

clicked, and the second distance applies to the second line clicked. You can also set the distance by clicking two points on the drawing.

You can set a chamfer distance of zero and use it to remove the chamfered corners from the table. Using a distance of zero will make 90° corners on the table. Then you can erase the old chamfer lines. This will change the table but not the credenza because it does not work with a polyline. If you have two lines that do not meet to form an exact corner or that overlap, use the **CHAMFER** command with 0 distance to form an exact corner. The **CHAMFER** command will chamfer two lines that do not intersect. It automatically extends the two lines until they intersect, trims the two lines according to the distance entered, and connects the two trimmed ends with the chamfer line.

NOTE

Remember to turn **SNAP** off and on as needed. Turn **SNAP** off when it interferes with selecting an entity. Turn it back on as needed.

Step 22. Zoom in on a portion of the grid outside the conference room walls.

Step 23. Draw a rectangle 26″ wide by 28″ deep using the **POLYLINE** command (Figure 3-8). Be sure to have **SNAP** on when you draw the rectangle. Next, you will edit this rectangle using the **FILLET** command to create the shape of a chair.

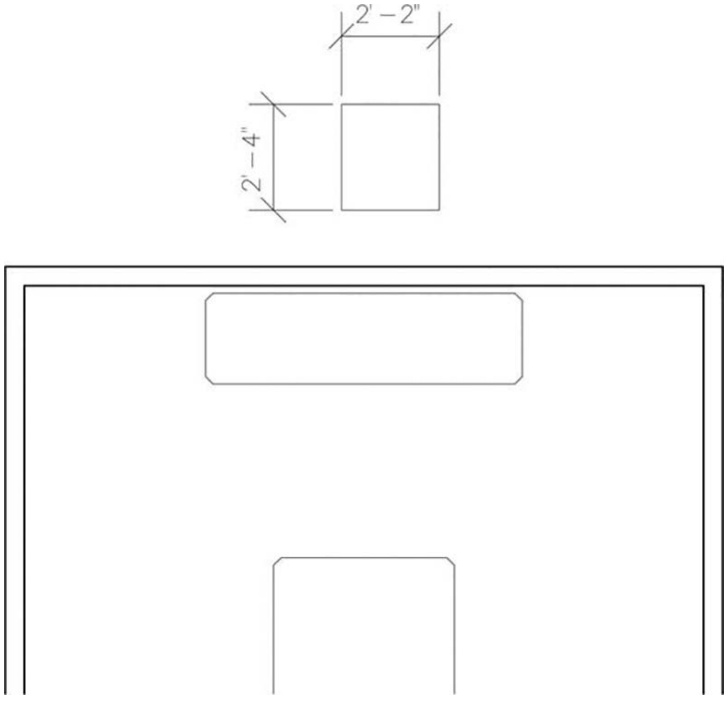


Figure 3-8
Draw a rectangle 26″ wide × 28′ deep using the **POLYLINE** command

FILLET	
Ribbon/Panel	Home/ Modify Fillet
Draw Toolbar:	
Menu Bar:	Modify/ Fillet
Type a Command:	FILLET
Command Alias:	F

Sample pages

FILLET

The **FILLET** command is similar to **CHAMFER**, except the **FILLET** command creates a round instead of an angle.

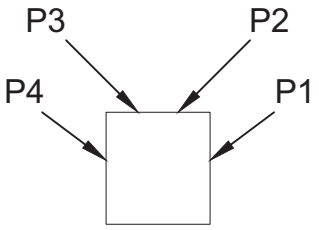


Figure 3-9
Use **FILLET** to create the chair symbol

Step 24. Use the **FILLET** command to edit the back of the rectangle to create the symbol of a chair (Figure 3-9), as described next:

Prompt	Response
Type a command:	**Fillet** (or type **F <Enter>**)
Current settings: Mode = TRIM, Radius = 0′-0″	
Select first object or [Undo Polyline Radius Trim Multiple]:	Type **R <Enter>**
Specify fillet radius <0′-0″>:	Type **12 <Enter>**
Select first object or [Undo Polyline Radius Trim Multiple]:	Type **T <Enter>**
Enter Trim mode option [Trim No trim]<Trim>:	Type **T <Enter>** (verify **Trim** option)
Select first object or [Undo Polyline Radius Trim Multiple]:	**P1→** (Figure 3-9)
Select second object or shift-select to apply corner or [Radius]:	**P2→**
Type a command:	**<Enter>** (repeat **Fillet**)
Current settings: Mode = TRIM, Radius = 1′-0″	
Select first object or [Undo Polyline Radius Trim Multiple]:	**P3→**
Select second object or Shift-select to apply corner or [Radius]:	**P4→**

The **Polyline** option of **Fillet** automatically fillets an entire continuous polyline with one click. Remember to set the fillet radius first.

Fillet will also fillet two circles, two arcs, a line and a circle, a line and an arc, or a circle and an arc.

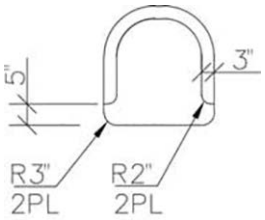


Figure 3-10
Use **OFFSET**, **TRIM**, **EXTEND**, and **FILLET** commands to complete the shape of the chair

Step 25. Use the commands **OFFSET**, **TRIM**, **EXTEND**, and **FILLET** to complete the shape of the chair, as shown in Figure 3-10.

NOTE

When using TRIM, you can invoke the **EXTEND** command by holding down the space bar as you select objects. For more on **EXTEND**, see Chapter 6.

Osnap: An abbreviation of *object snap,* which specifies a snap point at an exact location on an object.

COPY	
Ribbon/Panel	Home/ Modify
Draw Toolbar:	
Menu Bar:	Modify/ Copy
Type a Command:	COPY
Command Alias:	CO or CP

COPY and Osnap-Midpoint

The **COPY** command allows you to copy any part of a drawing either once or multiple times. Object snap modes (***Osnap***), when combined with other commands, help you to draw very accurately. As you become more familiar with the object snap modes, you will use them constantly to draw with extreme accuracy. The following introduces the **Osnap-Midpoint** mode, which helps you snap to the midpoint of a line or arc.

Sample pages

NOTE

Save your drawing often so you do not lose your work.

Step 26. Use the **COPY** command, combined with **Osnap-Midpoint**, to copy the chair you have just drawn (Figure 3-11), as described next:

Prompt	Response
Type a command:	**Copy** (or type **CP <Enter>**)
Select objects:	Click the first corner of a window that will include the chair
Specify opposite corner:	Click the other corner of the window to include the chair
Select objects:	**<Enter>**
Specify base point or [Displacement mOde] <Displacement>:	Type **MID <Enter>**
mid of	**P1→** (Figure 3-11) (Turn **SNAP** off as needed)
Specify second point or [Array] <use first point as displacement>:	**P2→** (be sure **SNAP** is on, and leave enough room to rotate the chair, Figure 3-12)
Specify second point or [Array Exit Undo]<Exit>:	**<Enter>**

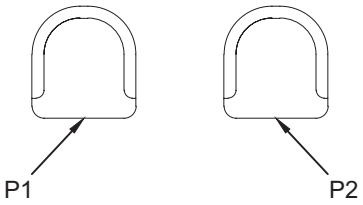


Figure 3-11
Copy the chair using **Osnap-Midpoint**

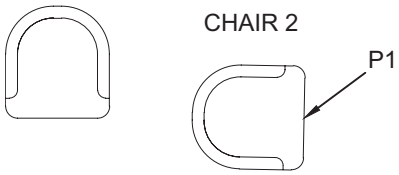
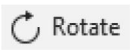


Figure 3-12
The rotated chair

The **Osnap-Midpoint** mode helped you snap very accurately to the midpoint of the line; you used the midpoint of the line that defines the front of the chair as the base point. When using the **COPY** command, carefully choose the base point so that it helps you easily locate the copies.

ROTATE	
Ribbon/Panel	Home/ Modify 
Draw Toolbar:	
Menu Bar:	Modify/ Rotate
Type a Command:	ROTATE
Command Alias:	RO

ROTATE

The **ROTATE** command rotates a selected drawing entity in the counter-clockwise direction; 90° is to the left, and 270° (or –90°) is to the right. You select a base point of the entity to be rotated, and the entity rotates about that base point.

TIP

The AutoCAD system variable **ANGDIR** sets the direction of positive angles. If the variable is set to 1, the direction is clockwise and is the same as selecting the **Clockwise** check box on the **Drawing Units** dialog box. When **ANGDIR** is set to 0, the direction is counter-clockwise, and the **Clockwise** check box of the **Drawing Units** dialog box is not selected.

Sample Pages

Step 27. Use the **ROTATE** command to rotate CHAIR 2 (Figure 3-12), as described next:

Prompt	Response
Type a command:	**Rotate** (or type **RO <Enter>**)
Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0	
Select objects:	Start the window to include CHAIR 2
Specify opposite corner:	Complete the window to include CHAIR 2
Select objects:	**<Enter>**
Specify base point:	Type **MID <Enter>**
mid of	**P1→** (Figure 3-12)
Specify rotation angle or [Copy Reference]:	Type **90 <Enter>**

NOTE

If part of the entity that is to be rotated lies on the specified base point, that part of the entity remains on the base point while the entity's orientation is changed.

Reference

If you don't know the specific rotation angle, the **Reference** option of the **Rotate** prompt is sometimes easier to use. It allows you to select the object to be rotated and click the base point. Type **R <Enter>** for **Reference.** Then you can enter the *Reference angle:* (current angle) of the object by typing it and pressing **<Enter>**. If you don't know the current angle, you can show AutoCAD the *Reference angle:* by picking the two endpoints of the line to be rotated. You can specify the *New angle:* by typing it and pressing **<Enter>**. If you don't know the new angle, you can show AutoCAD the *New angle:* by picking a point on the drawing.

POINT	
Ribbon/ Panel	Home/Draw (slideout)
Draw Toolbar:	
Menu Bar:	Draw/Point
Type a Command:	POINT
Command Alias:	PO

POINT

The **POINT** command allows you to draw points on your drawing. **Object Snap** recognizes these points as nodes. You use the **Osnap** mode **Node** to snap to points.

You can choose from many different styles of points. The appearance of these points is determined by the **PDMODE** (point definition mode) and **PDSIZE** (point definition size) options within the **POINT** command.

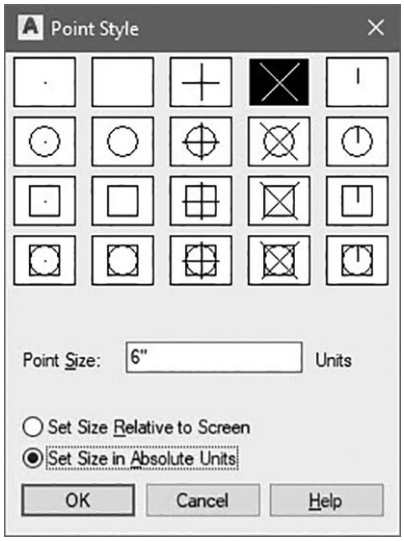
POINT STYLE	
Ribbon/ Panel	Home/ Utilities (slideout) Point Style...
Menu Bar:	Format/Point Style
Type a Command:	PTYPE

Step 28. Use the **Point Style...** command to set the appearance of points, as described next:

Prompt	Response
Type a command:	**Point Style...** (or type **PTYPE <Enter>**)
The **Point Style** dialog box appears (Figure 3-13):	Click the **X** box Type **6″** in the **Point Size:** input box Click **OK**

Sample pages

Figure 3-13
Point Style dialog box



You have just set the points to appear as an X, and they will be 6″ high. The **Point Style** dialog box shows the different types of points available. You may set the size of the point in a size relative to the screen or in absolute units.

Step 29. Use the **OFFSET** command to offset the line that defines the long left side of the conference table. The chairs will be placed 6″ from the edge of the table, so set 6″ as the offset distance. Offset the line outside the table, as shown in Figure 3-14. You will use this line as a construction line to help locate the chairs.

DIVIDE

DIVIDE	
Ribbon/ Panel	Home/Draw (slideout)
Menu Bar:	Draw/Point/ Divide
Type a Command:	DIVIDE
Command Alias:	DIV

The **DIVIDE** command indicates the divisions of an entity in equal parts and places point markers along the entity at the dividing points. The **PDMODE** variable has been set to 3 (an X point), so an X will appear as the point marker when you use **DIVIDE.**

Step 30. Use **DIVIDE** to divide the offset line into eight equal segments (Figure 3-14), as described next:

Prompt	**Response**
Type a command:	**Divide** (or type **DIV <Enter>**)
Select object to divide:	Click anyplace on the offset line
Enter the number of segments or [Block]:	Type **8 <Enter>** (the X points divide the line into eight equal segments)

Sample pages

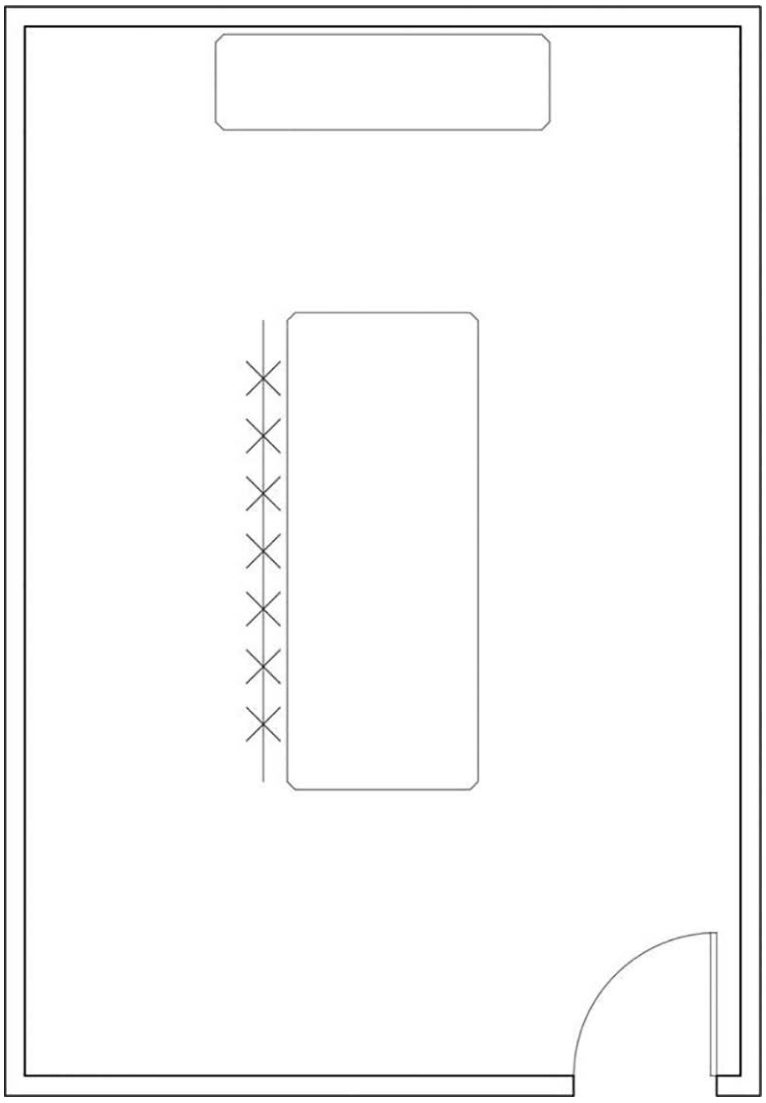
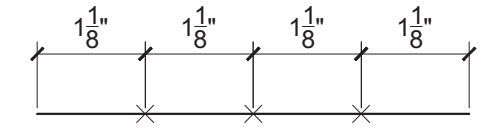
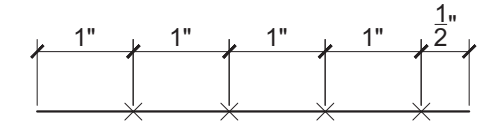


Figure 3-14
Offset the lines defining the long left side of the conference table and use the **DIVIDE** command to show eight equal segments along the line object



DIVIDE: Four equal parts of a 4-1/2″ line



MEASURE: 1″ lengths of a 4-1/2″ line

Figure 3-15
Examples of the **DIVIDE** and **MEASURE** commands

MEASURE

MEASURE	
Ribbon/ Panel	Home/Draw (slideout)
Menu Bar:	Draw/Point/ Measure
Type a Command:	MEASURE

The **MEASURE** command is similar to the **DIVIDE** command (Figure 3-15) except that with **MEASURE,** you specify the distance. **DIVIDE** calculates the interval to divide an entity into a specified number of equal segments. The **MEASURE** command places point markers at a specified distance along an entity.

The measurement and division of a circle start at the angle from the center that follows the current snap rotation. The measurement and division of a closed polyline start at the first vertex drawn. The **MEASURE** command also draws a specified block at each mark between the divided segments.

Sample pages

OSNAP

It is important that you become familiar with and use object snap modes in combination with **DRAW, MODIFY,** and other AutoCAD commands. When an existing drawing object is not located on a snap point, connecting a line or other drawing entity exactly to it is impossible. You may try, and you may think that the two points are connected, but a close examination **(Zoom-Window)** will reveal that they are not. Object snap modes are used in combination with other commands to connect exactly to points of existing objects in a drawing. You need to use object snap modes constantly for complete accuracy.

Activating Osnap

You can activate **Osnap** mode in the following ways:

- Type the **Osnap** abbreviation (first three letters of the object snap mode).
- Press **<Shift>** and right-click in the drawing area, then choose an object snap mode from the **Object Snap** menu that appears (Figure 3-16).
- Right-click **OSNAP** on the status bar, and then click Object Snap **Settings...** (Figure 3-17) to access the **Drafting Settings** dialog box (Figure 3-18). Select the desired **Osnap** mode or modes check boxes.

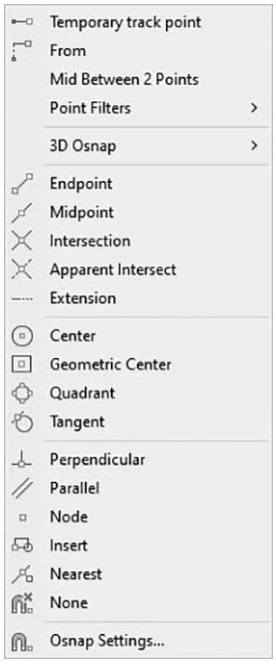


Figure 3-16
Activate the **Osnap** menu by pressing **<Shift>** and right-click in the drawing area

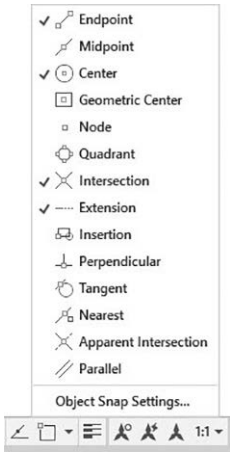
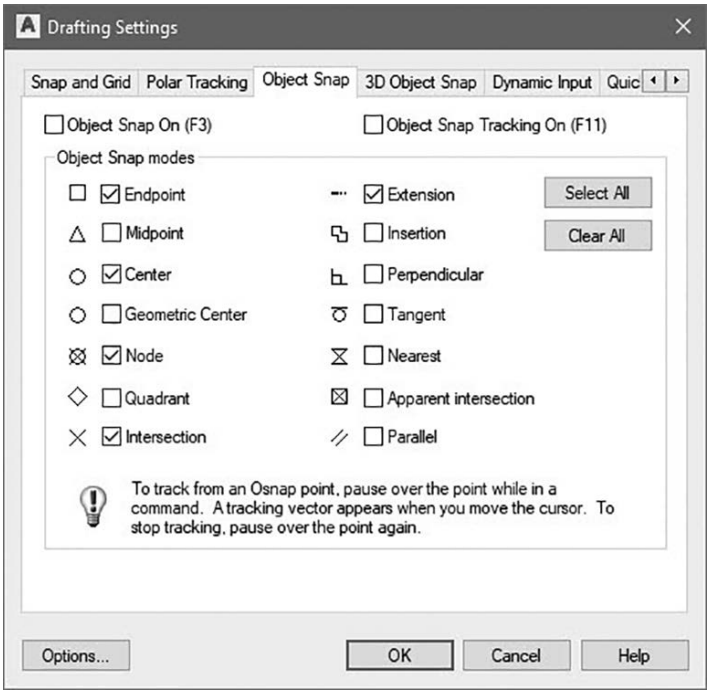


Figure 3-17
Activate **OSNAP** by right-clicking **Object Snap** on the status bar, then clicking **Settings...** to access the **Drafting Settings** dialog box

Sample pages

Figure 3-18
Drafting Settings dialog box with **Node** selected



Copy, Osnap-Midpoint, Osnap-Node

Next, you copy the chair several times using different object snap modes.

Step 31. Right-click **Snap cursor to 2D reference points** on the status bar, click **Object Snap Settings...**, and set a running **Osnap** mode of **Node** (Figures 3-17 and 3-18).

Step 32. Make sure **ORTHO** and **SNAP** are off and **OSNAP** is on in the status bar.

Step 33. Use the **COPY** command (combined with **Osnap-Midpoint** and **Osnap-Node**) to copy CHAIR 2 four times on the left side of the conference table (Figure 3-19), as described next:

Prompt	**Response**
Type a command:	**Copy** (or type **CP <Enter>**)
Select objects:	Click below and to the left of CHAIR 2
Specify opposite corner:	Window CHAIR 2
Select objects:	**<Enter>**
Specify base point or [Displacement mOde] <Displacement>:	Type **MID <Enter>**
mid of	**P1→** (anyplace on the straight line that forms the front of the chair symbol)
Specify second point or [Array] <use first point as displacement>:	**P2→, P3→, P4→, P5→ <Enter>** (Figure 3-19)

The points act as nodes (snapping exactly on the center of the X) when a running **Object Snap** is set.

Sample pages

Step 34. Type **PDMODE <Enter>** at the command prompt. Set the **PDMODE** to **1**, and the drawing is regenerated. The Xs will disappear. You have set the **PDMODE** (point definition mode) to be invisible.