Міністерство освіти і науки України

Харківська національна академія міського господарства

Creation of AutoCAD entities: drawing a basketball court plan.

Guidance for laboratory works for "Informatics and computer modeling basics" course /for foreign students of 2nd year studying by orientation 6.060102 «Architecture»/

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Створення примітивів AutoCAD: побудова плану баскетбольного майданчика: Методичні вказівки до виконання лабораторних робіт з курсу "Інформатика і основи комп'ютерного моделювання" (англ..) /для студентів 2 курсу напряму підготовки 6.060102 «Архітектура»/ Укл.: Бочаров Б.П., Яковицький І. Л., Воєводіна М.Ю., Левіков Ю.В.– Х.: ХНАМГ, 2009. – 31с.

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Theme: **Creation of AutoCAD entities: drawing a basketball court plan**

Purpose: **to confirm skills of work with drawings and layers, to become acquainted with the procedures of drawing and displaying of drawings, to learn how to draw lines segments, circles, arcs and polylines in AutoCAD**

Work order:

1. Create a new drawing in AutoCAD and save it in the personal folder under the name *tsk1.dwg*.

2. Set the units of measurement to *meters.*

3. Set the limits of the drawing to -500,-500 and 10000,100000.

4. Set SNAP and GRID equal to 0.5 m.

5. Create a new layer.

6. Using the LINE, CIRCLE and ARC commands, draw a basketball court plan (length 24 m, width 14 m, the lower left corner on point 0,0).

7. Create a new layer.

8. Draw all the lines of the basketball court plan created in No.6 over with polylines (width 0.1 m)

9. Save the drawing in the personal folder and compress it (Zip).

10. Save this archive in the Academy distance learning system.

Draw Lines

In a simple *line* with connected segments, each segment is a separate line object.

With LINE, you can create a series of contiguous line segments.

Each single line segment can be edited separately from the other line segments in a series. You can close a sequence of line segments so that the first and last segments are joined.

You specify the locations that define the endpoints of each line with precision. You can

- Enter the coordinate values for an endpoint, using either absolute or relative coordinates
- Specify an object snap relative to an existing object. For example, you can specify the center of a circle as one endpoint of the line
- Turn grid snap on and snap to a location

Use *polyline* objects instead of line objects if you want the segments to be connected as a single object.

Using LINE command

Toolbar: Draw

Menu: Draw ➤ Line

Command entry: line

Specify first point: Specify a point or press ENTER to continue from the last drawn line or arc
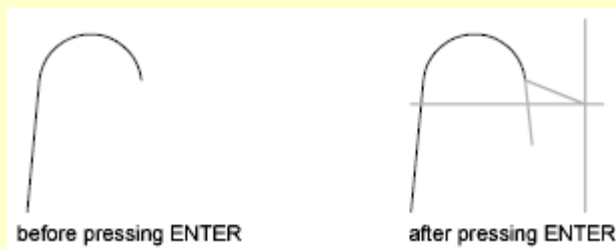
Specify next point or [Close/Undo]:

Continue

Continues a line from the endpoint of the most recently drawn line.

before pressing ENTER   after pressing ENTER

If the most recently drawn line is an arc, its endpoint defines the starting point of the line, and the line is drawn tangent to the arc.
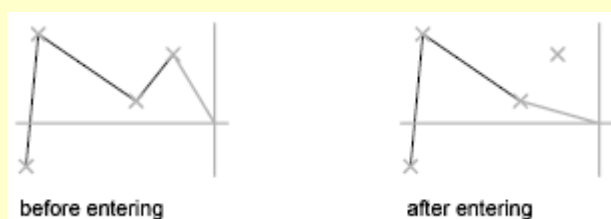
before pressing ENTER   after pressing ENTER

Close

Ends the last line segment at the beginning of the first line segment, which forms a closed loop of line segments. You can use Close after you have drawn a series of two or more segments.

before entering   after entering

Undo

Erases the most recent segment of a line sequence.

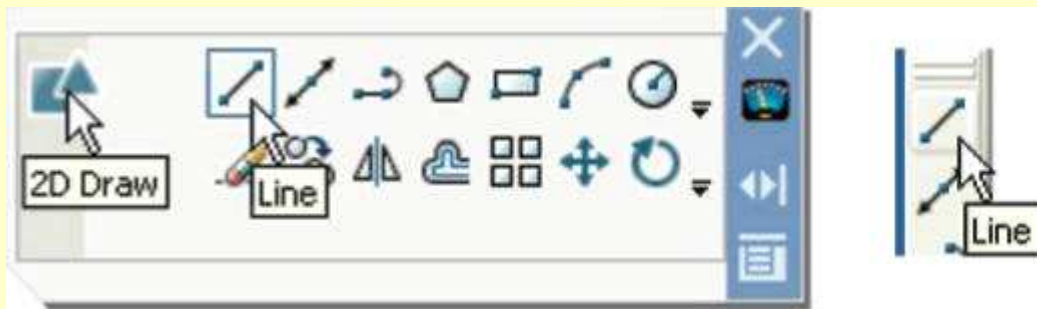before entering   after entering

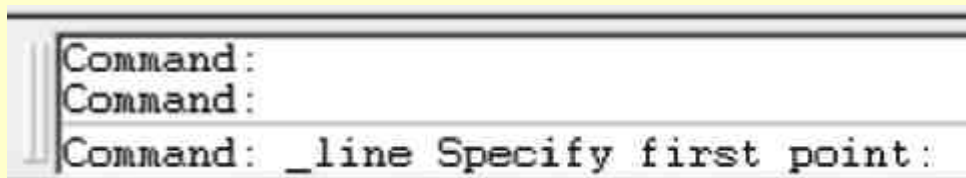Entering **u** more than once backtracks through line segments in the order you created them.

Drawing with the Line tool examples

Open AutoCAD. The drawing area will show the settings of the **acadiso.dwt** template - **Limits** set to **420,297, Grid** set to **10, Snap** set to **5** and **Units** set to **0**.

Left-click on the **Line** tool in the **2D Draw** control panel

The prompt **Command: _line Specify first point:** which appears in the command window at the command line

Make sure **Snap** is on by either pressing the **F9** key or the **SNAP** button in the status bar. **<Snap on>** will show in the command palette.

Move the mouse around the drawing area. The cursor's pick box will jump from point to point at 5 unit intervals. The position of the pick box will show as coordinate numbers in the status bar (left-hand end).

Move the mouse until the coordinate numbers show **60,240,0** and press the **Pick** button of the mouse *(left-click).*
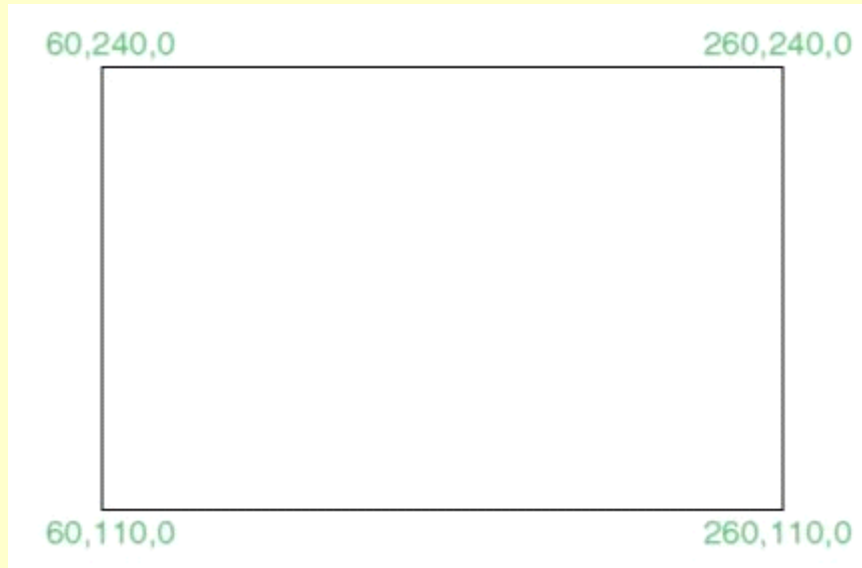
Move the mouse until the coordinate numbers show **260,240,0** and *left-click.*

Move the mouse until the coordinate numbers show **260,110,0** and *left-click.*

Move the mouse until the coordinate numbers show **60,110,0** and *left-click.*

Move the mouse until the coordinate numbers show **60,240,0** and *left-click.*

Then press the **Return** button of the mouse *(right-click).*

Second example

Close the drawing and open a new drawing

Left-click on the **Line** tool icon and *enter* figures as follows at each prompt of the command line sequence:

Command:_line Specify first point: *enter* **80,235** *right-click*

Specify next point or [Undo]: *enter* **275,235** *right-click*

Specify next point or [Undo]: *enter* **295,210** *right-click*

Specify next point or [Close/Undo]: *enter* **295,100** *right-click*

Specify next point or [Close/Undo]: *enter* **230,100** *right-click*

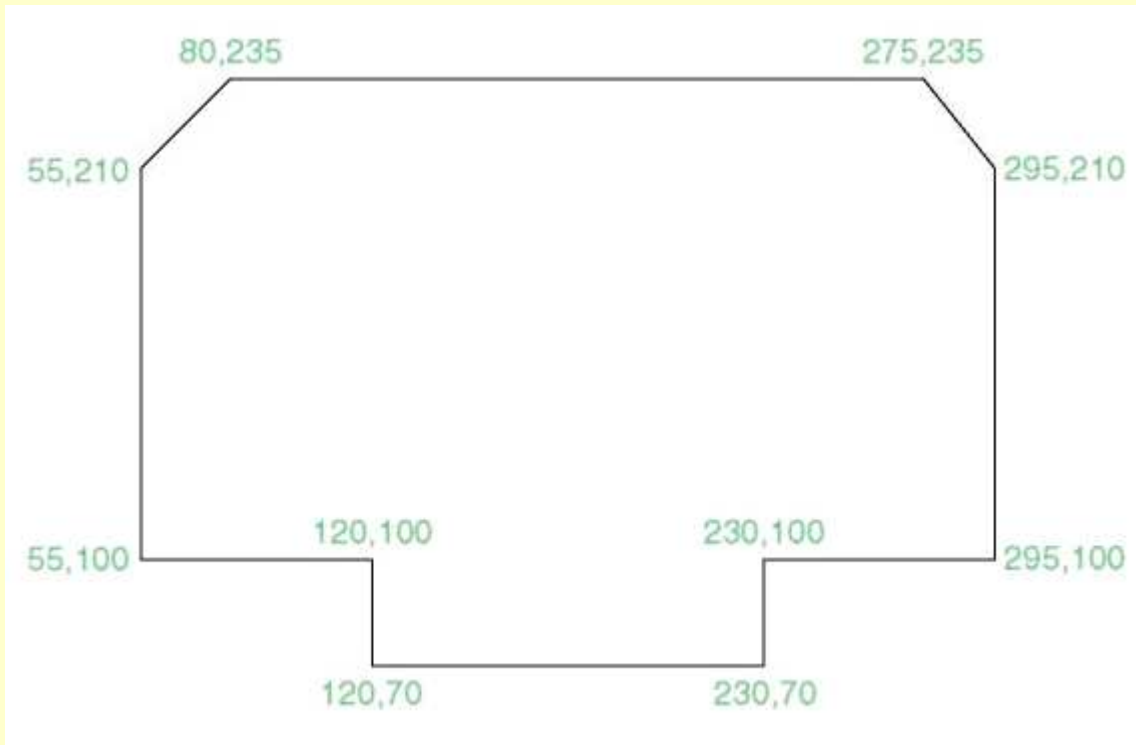**Specify next point or [Close/Undo]:** *enter* **230,70** *right-click*

Specify next point or [Close/Undo]: *enter* **120,70** *right-click*

Specify next point or [Close/Undo]: *enter* **120,100** *right-click*

Specify next point or [Close/Undo]: *enter* **55,100** *right-click*

Specify next point or [Close/Undo]: *enter* **55,210** *right-click*

Specify next point or [Close/Undo]: *enter* **c** (Close) *right-click*

Third example

Close the drawing and open a new **acadiso.dwt** window.

Left-click on the **Line** tool icon and *enter* figures as follows at each prompt of the command line sequence:

Command:_line Specify first point: *enter* **60,210** *right-click*

Specify next point or [Undo]: *enter* **@50,0** *right-click*

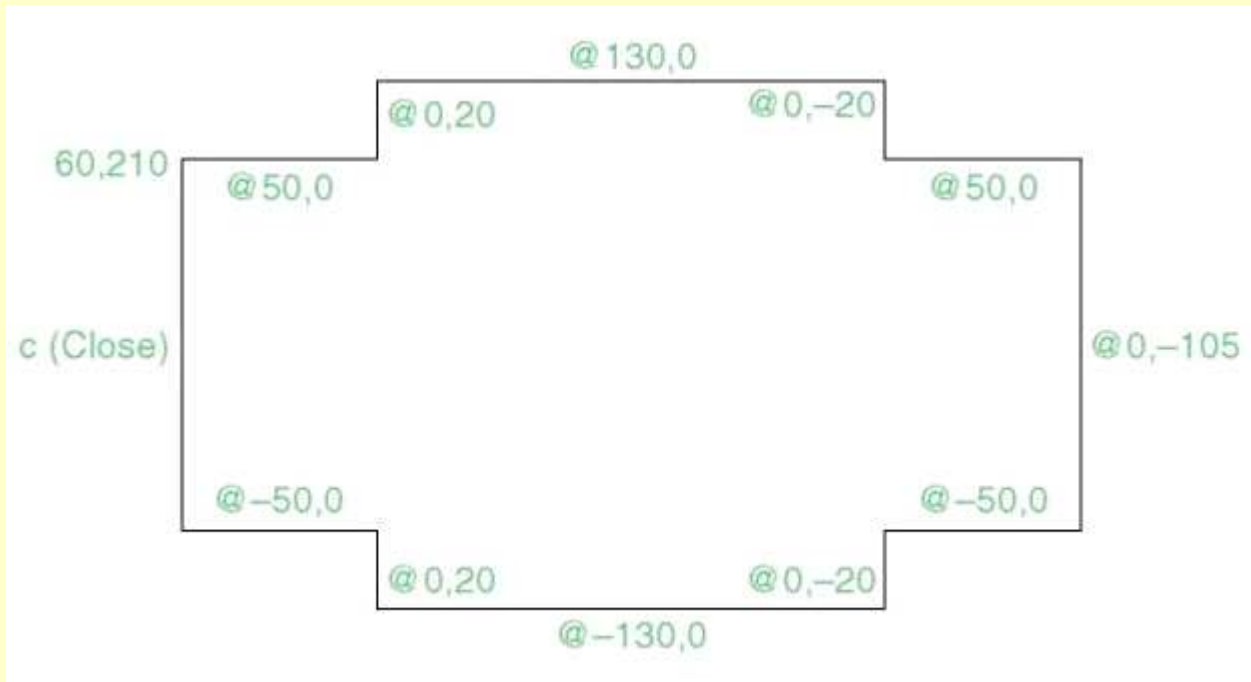**Specify next point or [Undo]:** *enter* **@0,20** *right-click*

Specify next point or [Close/Undo]: *enter* **@130,0** *right-click*

Specify next point or [Close/Undo]: *enter* **@0,-20** *right-click*

Specify next point or [Close/Undo]: *enter* **@50,0** *right-click*

Specify next point or [Close/Undo]: *enter* **@0,—105** *right-click*

Specify next point or [Close/Undo]: *enter* **@—50,0** *right-click*

Specify next point or [Close/Undo]: *enter* **@0,-20** *right-click*

Specify next point or [Close/Undo]: *enter* **@—130,0** *right-click*

Specify next point or [Close/Undo]: *enter* **@0,20** *right-click*

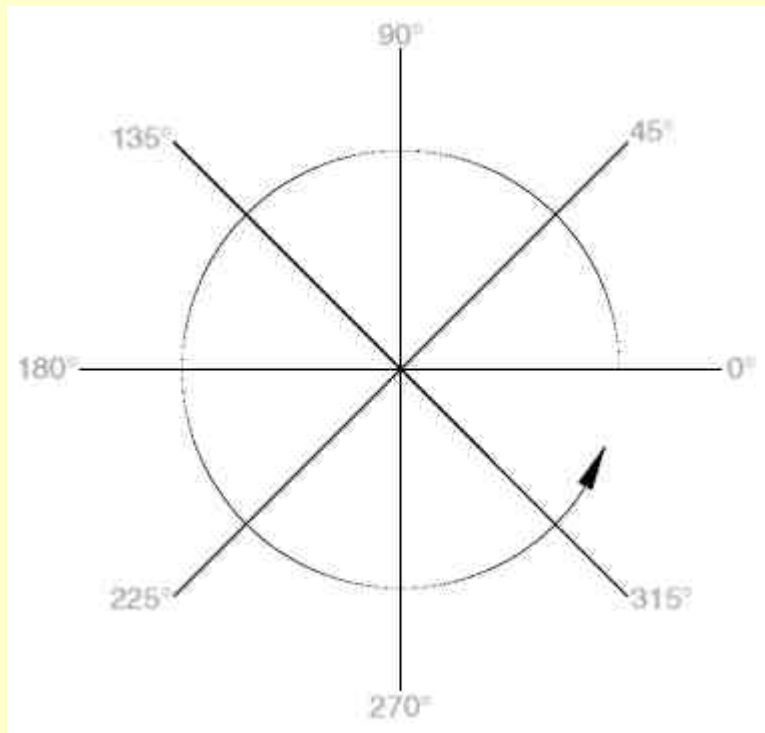**Specify next point or [Close/Undo]:** *enter* **@—50,0** *right-click*

Specify next point or [Close/Undo]: *enter* **c** (Close) *right-click*

Notes

- The figures typed at the keyboard determining the corners of the outlines in the above examples are two-dimensional (2D) **x, y** coordinate points. When working in 2D, coordinates are expressed in terms of two numbers separated by a comma.
- Coordinate points can be shown as positive or negative numbers.
- The method of constructing an outline as shown in the first two examples is known as the **absolute coordinate entry** method, where the **x, y** coordinates of each corner of the outlines are *entered* at the command line as required.
- The method of constructing an outline as in the third example is known as the **relative coordinate entry** method - coordinate points are *entered* relative to the previous entry. In relative coordinate entry, the @ symbol is *entered before* each set of coordinates with the following rules in mind:

    The next example (the fourth) shows how lines at angles can be drawn taking advantage of the relative coordinate entry method. Angles in AutoCAD are measured in 360 degrees in a counter-clockwise (anticlockwise) direction. The < symbol precedes the angle.

Fourth example

Left-click on the **Line** tool icon and *enter* figures as follows at each prompt of the command line sequence:

Command:_line Specify first point: 70,230

Specify next point: @220,0

Specify next point: @0,—70

Specify next point or [Undo]: @115 < 225

Specify next point or [Undo]: @ —60,0

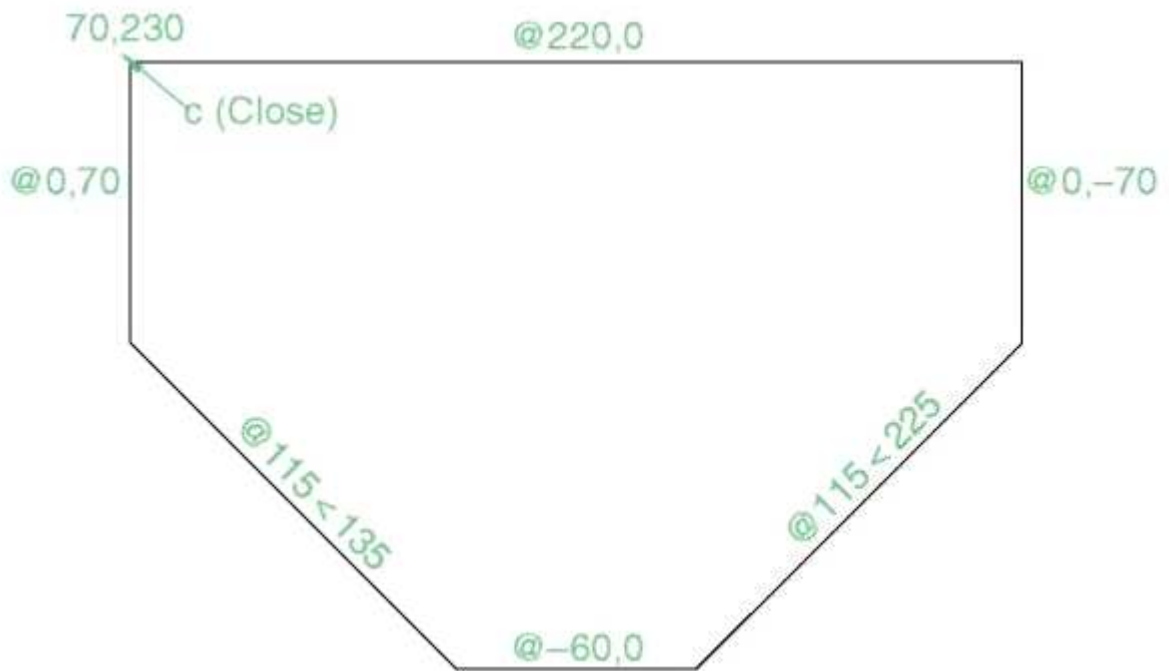**Specify next point or [Close/Undo]:** @115 < 135

Specify next point or [Close/Undo]: @0,70
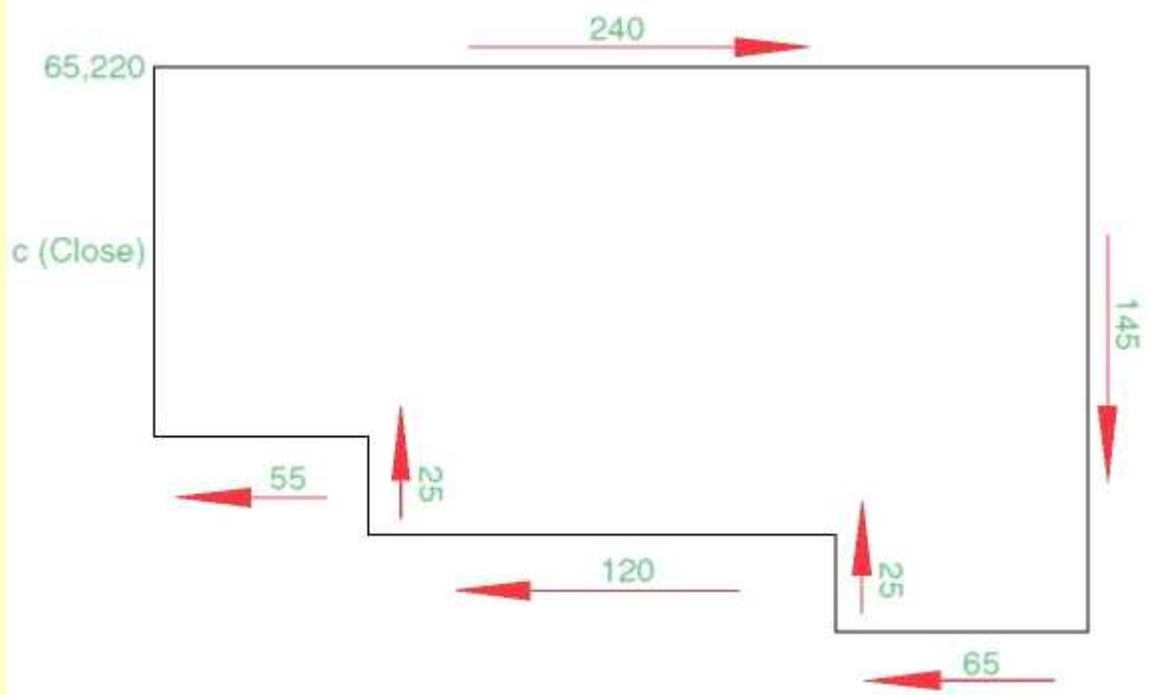
Specify next point or [Close/Undo]: c (Close)

Fifth example

Another method of constructing accurate drawings is by using a method known as **tracking**. When **Line** is in use, as each **Specify next point:** appears at the command line, a *rubber-banded* line appears from the last point *entered*. *Drag* the rubber-band line in any direction and *enter* a number at the keyboard, followed by a *right-click*. The

line is drawn in the *dragged* direction of a length in units equal to the *entered* number.

In this example because all lines are drawn in either the vertical or the horizontal direction, either press the **F8** key or *click* the **ORTHO** button in the status bar.

Left-click on the **Line** tool icon and *enter* figures as follows at each prompt of the command line sequence:

Command:_line Specify first point: *enter* **65,220** *right-click*

Specify next point: *drag* to right *enter* **240** *right-click*

Specify next point: *drag* down *enter* **145** *right-click*

Specify next point or [Undo]: *drag* left *enter* **65** *right-click*

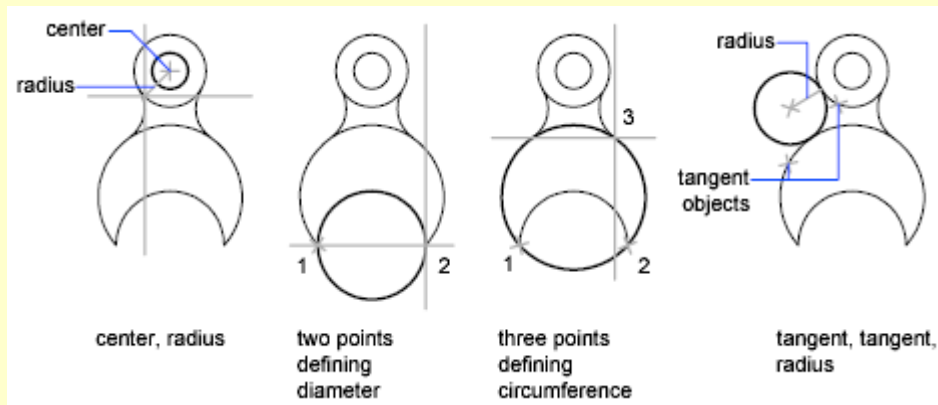**Specify next point or [Undo]:** *drag* upwards *enter* **25** *right-click*

Specify next point or [Close/Undo]: *drag* left *enter* **120** *right-click*

Specify next point or [Close/Undo]: *drag* upwards *enter* **25** *right-click*

Specify next point or [Close/Undo]: *drag* left *enter* **55** *right-click*

Specify next point or [Close/Undo]: c (Close) *right-click*
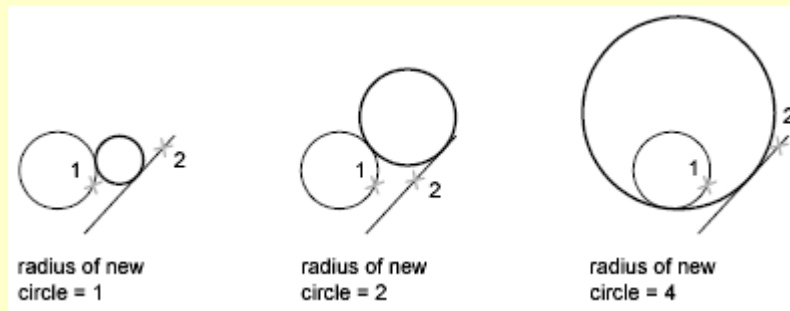

Draw Circles

To create circles, you can specify various combinations of center, radius, diameter, points on the circumference, and points on other objects.

You can create circles in several ways. The default method is to specify the center and the radius. Three other ways to draw a circle are shown in the illustration.

center, radius   two points defining diameter   three points defining circumference   tangent, tangent, radius

Draw a Circle Tangent to Other Objects

The tangent point is a point where an object touches another object without intersecting it. To create a circle that is tangent to other objects, select the objects and then specify the radius of the circle. In the illustrations below, the bold circle is the one being drawn, and points 1 and 2 select the objects to which it is tangent.

radius of new
circle = 1

radius of new
circle = 2

radius of new
circle = 4

To create a circle tangent at three points, set running object snaps (OSNAP) to Tangent and use the three-point method to create the circle.

<u>Using CIRCLE command</u>

Toolbar: Draw

Menu: Draw ➤ Circle

Command entry: *circle*

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: Specify a point or enter an option
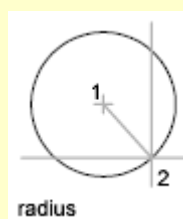
Center Point

Draws a circle based on a center point and a diameter or a radius.

Specify radius of circle or [Diameter]: Specify a point, enter a value, enter d, or press ENTER
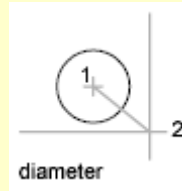
Radius

Defines the radius of the circle. Enter a value, or specify a point (2). The distance between this point and the center point determines the radius of the circle.

radius

Diameter

Draws a circle using the center point and a specified distance for the diameter.

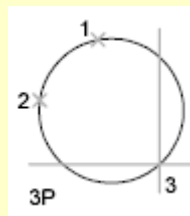*Specify diameter of circle <current>:* Specify a point (2), enter a value, or press ENTER

3P *(Three Points)*

Draws a circle based on three points on the circumference.

Specify first point on circle: Specify a point (1)

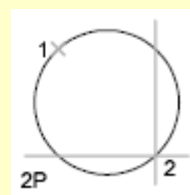*Specify second point on circle:* Specify a point (2)

Specify third point on circle: Specify a point (3)

2P *(Two Points)*

Draws a circle based on two endpoints of the diameter.

Specify first endpoint of circle's diameter: Specify a point (1)

Specify second endpoint of circle's diameter: Specify a point (2)
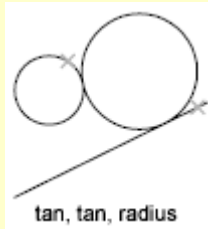

TTR *(Tangent, Tangent, Radius)*

Draws a circle with a specified radius tangent to two objects.

Specify point on object for first tangent of circle: Select a circle, arc, or line

Specify point on object for second tangent of circle: Select a circle, arc, or line

Specify radius of circle <current>:
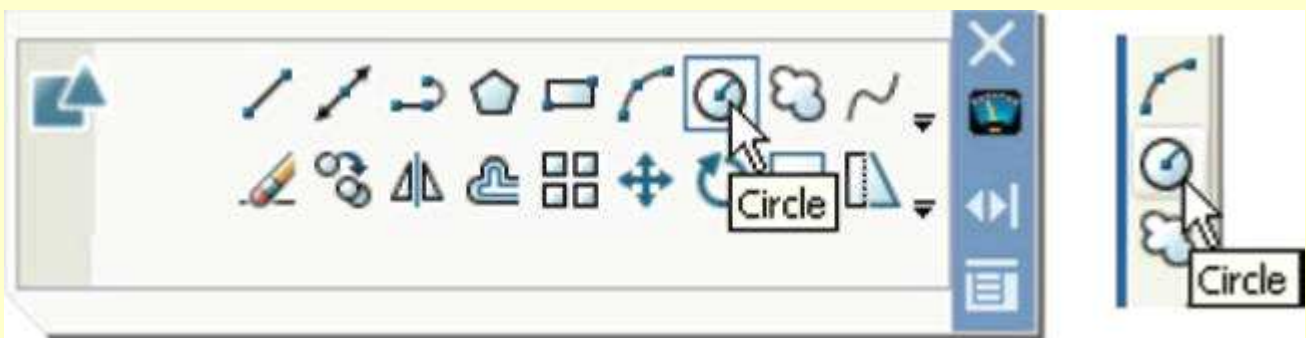

tan, tan, radius

Sometimes more than one circle matches the specified criteria. The program draws the circle of the specified radius whose tangent points are closest to the selected points.

Drawing with the Circle tool examples

Left-click on the **Circle** tool icon in the **2D Draw** control panel
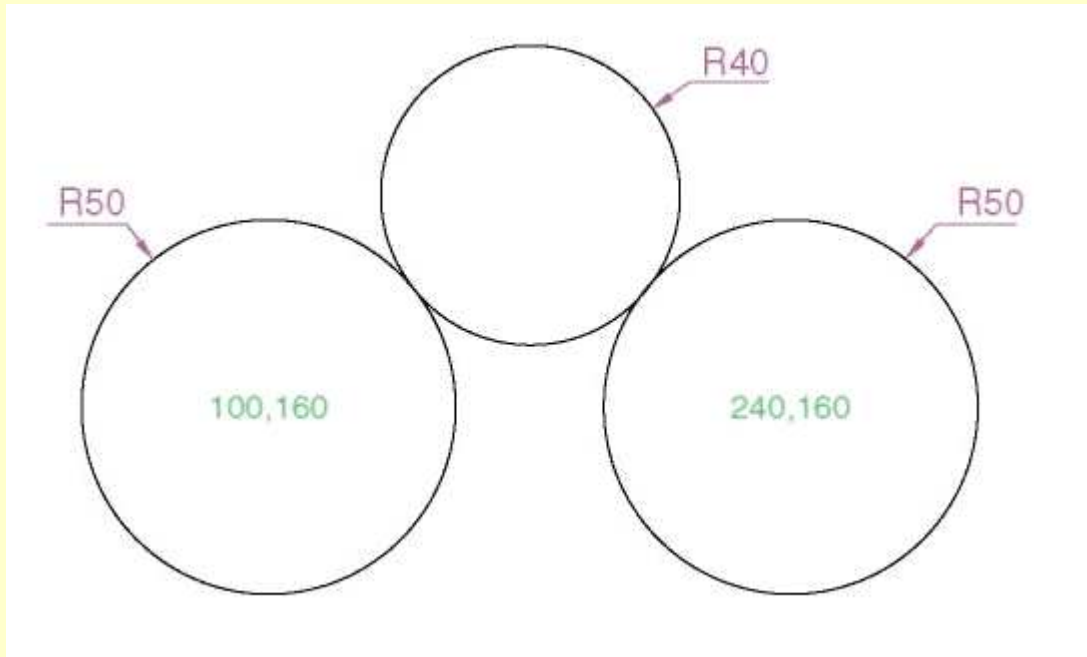

Enter numbers against the prompts appearing in the command window as shown

```
Command: _circle Specify center point for circle or [3P/2P/Ttr (tan tan
radius)]: 180,160
Specify radius of circle or [Diameter]: 55
Command:
```

The circle appears on screen

Second example

Left-click on the **Circle** tool icon and construct two circles (radius 50) as shown in the drawing

Click the **Circle** tool again and against the first prompt *enter t* (the abbreviation for the prompt **tan tan radius**), followed by a *right-click.*

Command: _circle Specify center point for circle or [3P/2P/Ttr (tan tan radius]: *enter t right-click*

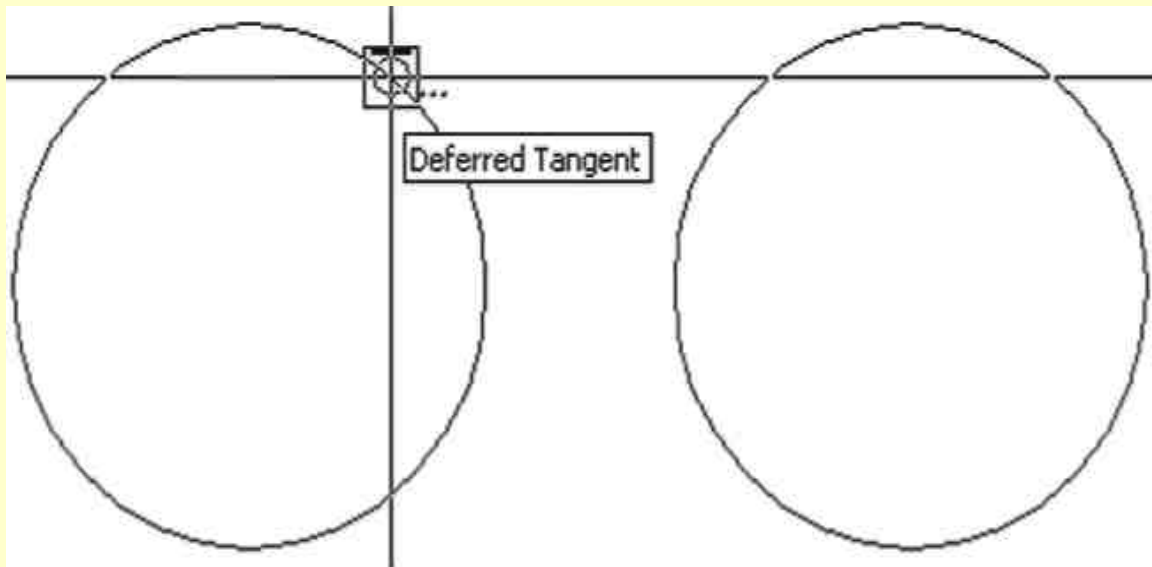**Specify point on object for first tangent of circle:** *pick*

Specify point on object for second tangent of circle: *pick*

Specify radius of circle (50): *enter* **40** *right-click*

When a point on either circle is picked the **Deferred Tangent** tip appears. This tip will only appear when the **OSNAP** button is set on with a *click* on its button in the status bar, or by pressing the **F3** key of the keyboard.

Circles can be drawn through 3 points or 2 points *entered* at the command line in response to prompts brought to the command line by using **3P** and **2P** in answer to the circle command line prompts
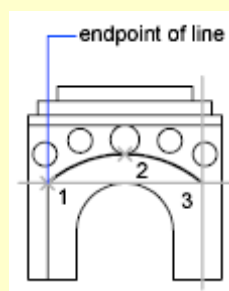
Draw Arcs

To create an arc, you can specify various combinations of center, endpoint, start point, radius, angle, chord length, and direction values.

You can create arcs in several ways. With the exception of the first method, arcs are drawn counterclockwise from the start point to the endpoint.

<u>Draw Arcs by Specifying Three Points</u>

You can draw an arc by specifying three points. In the following example, the start point of the arc snaps to the endpoint of a line. The second point of the arc snaps to the middle circle in the illustration.
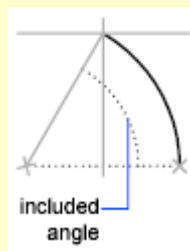

<u>Draw Arcs by Specifying Start, Center, End</u>

When you know the start point, center point, and endpoint, you can draw an arc by specifying either the start point or the center point first. The center point is the center of a circle that the arc is part of.
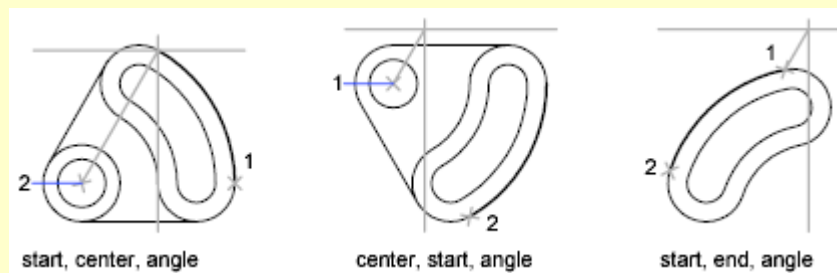
start(1), center(2),
end(3)

center(1), start(2),
end(3)

Draw Arcs by Specifying Start, Center, Angle

When you have a start point and a center point you can snap to and you know the included angle, use the Start, Center, Angle or the Center, Start, Angle option.
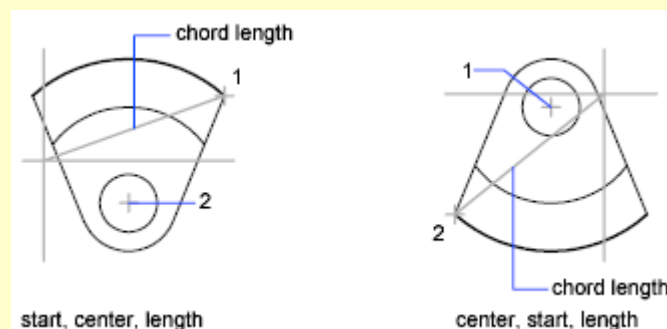

included angle

The included angle determines the endpoint of the arc. Use the Start, End, Angle method when you know both endpoints but cannot snap to a center point.

start, center, angle   center, start, angle   start, end, angle

Draw Arcs by Specifying Start, Center, Length

When you have a start point and a center point you can snap to; when you know the chord length, use the Start, Center, Length or the Center, Start, Length option.
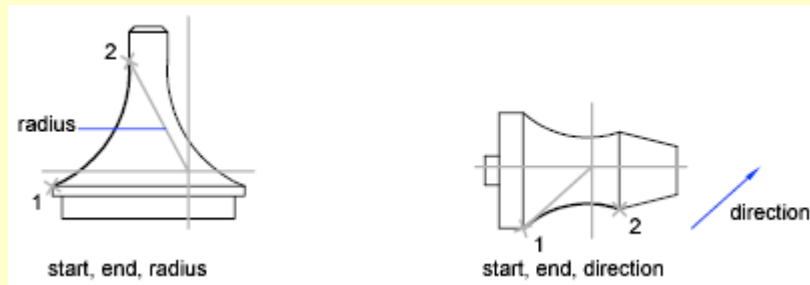

chord length

start, center, length   center, start, length

The length of the chord of the arc determines the included angle.

Draw Arcs by Specifying Start, End, Direction/Radius

When you have a start point and an endpoint, use the Start, End, Direction or the Start, End, Radius option.

The illustration on the left shows an arc drawn by specifying a start point, endpoint, and radius. You can specify the radius by entering a length or by moving the pointing device clockwise or counterclockwise and clicking to specify a distance.

The illustration on the right shows an arc drawn with the pointing device by specifying a start point and an endpoint and a direction. Moving the cursor up from the start point and endpoint draws the arc concave to the object, as shown here. Moving the cursor down draws the arc convex to the object.

Draw Contiguous Arcs and Lines

Immediately after you complete an arc, you can start a line tangent to the arc at an endpoint by starting the LINE command and pressing ENTER at the Specify First Point prompt. You need to specify only the line length.

Conversely, after you complete a line, you can start an arc tangent to the line at an endpoint by starting the ARC command and pressing ENTER at the Specify Start Point prompt. You need to specify only the endpoint of the arc.

You can connect sequentially drawn arcs in the same way. To create connected arcs using a menu, click Draw menu ➤ Arc ➤ Continue. In both cases, the resulting
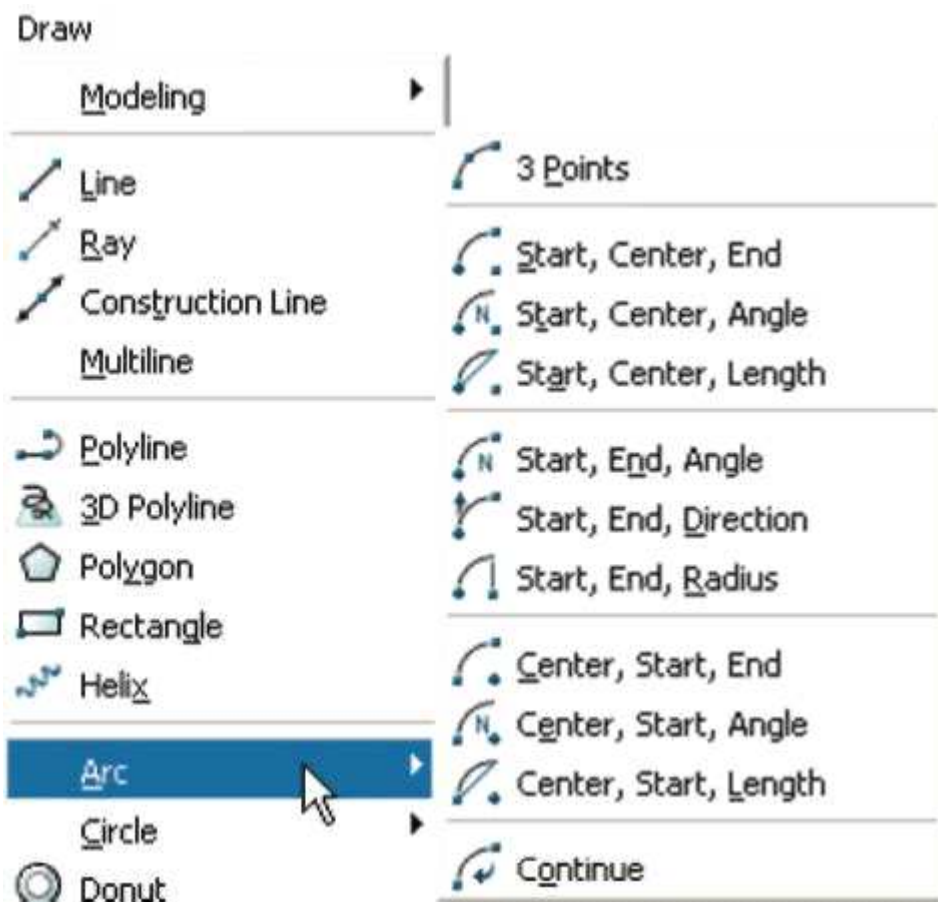
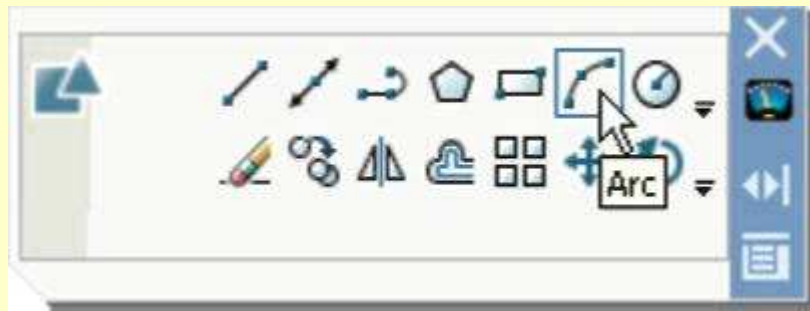object is tangent to the previous one. You can use the shortcut menu to repeat the Continue option.

Drawing with the Arc tool examples

In AutoCAD 2008, arcs can be constructed using any three of the following characteristics of an arc: its **Start** point; a point on the arc (**Second** point); its **Center**; its **End**; its **Radius; Length** of the arc; **Direction** in which the arc is to be constructed; **Angle** between lines of the arc.

In the examples which follow, *entering* initials for these characteristics in response to prompts at the command line when the **Arc** tool is called allows arcs to be constructed in a variety of ways.

To call the **Arc** tool *click* on its tool icon in the **2D Draw** control panel or *click* on **Arc** in the **Draw** drop-down menu. A sub-menu shows the possible methods of constructing arcs. The abbreviation for calling the **Arc** tool is **a**.
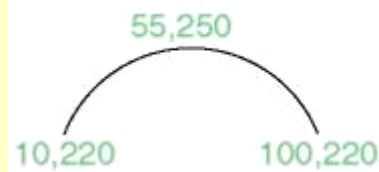
Left-click the **Arc** tool icon. The command line shows:

Command:_arc Specify start point of arc or [Center]: 100,220

Specify second point of arc or [Center/End]: 55,250

Specify end point of arc: 10,220

Command:

Second example

Command: *right-click* brings back the **Arc** sequence
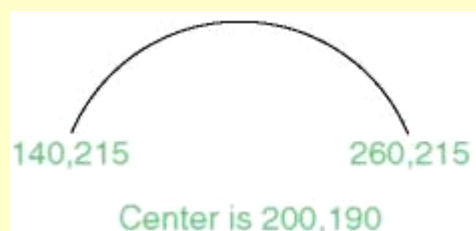
ARC Specify start point of arc or [Center]: c (Center)

Specify center point of arc: 200,190

Specify start point of arc: 260,215

Specify end point of arc or [Angle/chord Length]: 140,215

Command:

Third example

Command: *right-click* brings back the **Arc** sequence

ARC Specify start point of arc or [Center]: 420,210
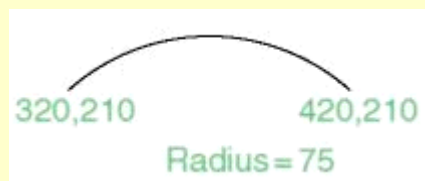
Specify second point of arc or [Center/End]: e (End)

Specify end point of arc: 320,210

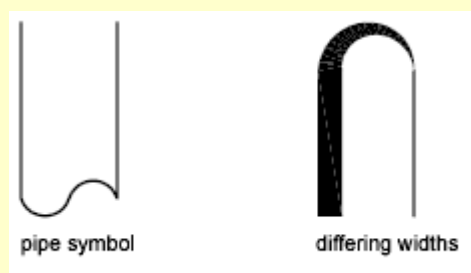**Specify center point of arc or [Angle/Direction/Radius]:** r (Radius)

Specify radius of arc: 75

Command:

320,210   420,210
Radius=75

Draw Polylines

A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.

pipe symbol   differing widths

Multisegmented lines provide editing capabilities unavailable for single lines. For example, you can adjust their width and curvature. After you've created a polyline, you can edit it with PEDIT or use EXPLODE to convert it to individual line and arc segments. You can

- Convert a spline-fit polyline into a true spline with SPLINE
- Use closed polylines to create a polygon

- Create a polyline from the boundaries of overlapping objects
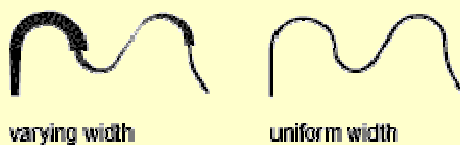
Create Arc Polylines

When you draw arc segments in a polyline, the first point of the arc is the endpoint of the previous segment. You can specify the angle, center point, direction, or radius of the arc. You can also complete the arc by specifying a second point and an endpoint.

Create Closed Polylines

You can draw a closed polyline to create a polygon. To close a polyline, specify the starting point of the last side of the object, enter c (Close), and press ENTER.

Create Wide Polylines

You can draw polylines of various widths by using the Width and Halfwidth options. You can set the width of individual segments and make them taper gradually from one width to another. These options become available after you specify a starting point for the polyline.

varying width   uniform width

The Width and Halfwidth options set the width of the next polyline segments you draw. Zero (0) width produces a thin line. Widths greater than zero produce wide lines, which are filled if Fill mode is on and outlined if Fill mode is off. The Halfwidth option sets width by specifying the distance from the center of the wide polyline to an outside edge.
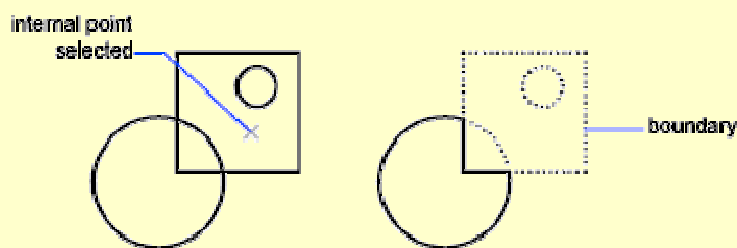
Taper

When you use the Width option, you are prompted for both a starting and an

ending width. By entering different values, you can taper the polyline. The starting and ending points of wide polyline segments are in the center of the line. Intersections of adjacent wide segments are usually beveled. However, nontangent arc segments, acute angles, or segments that use a dash-dot linetype are not beveled.

Create Polylines from the Boundaries of Objects

You can create a polyline from the boundaries of overlapping objects that form a closed area. A polyline created using the boundary method is a separate object, distinct from the objects used to create it. You can edit it using the same methods used to edit other polylines.

To expedite the boundary selection process in large or complex drawings, you can specify a group of boundary candidates, called a boundary set. You create this set by selecting the objects you want to use define the boundary.
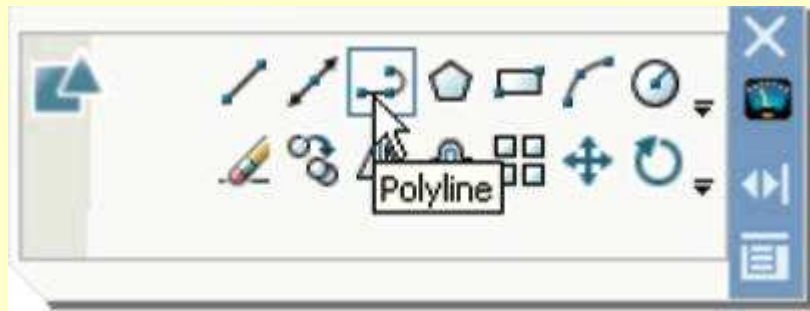

Drawing with the Polyline tool examples

When drawing lines with the **Line** tool, each line drawn is an object in its own right. A rectangle drawn with the **Line** tool is four objects. A rectangle drawn with the **Polyline** tool is a single object. Lines of different thickness, arcs, arrows and circles can all be drawn using this tool as will be shown in the examples describing constructions using the **Polyline** tool. Constructions resulting from using the tool are known as **polylines** or **plines**.

The **Polyline** tool can be called from the **2D Draw** control panel or from the **Draw** toolbar

Left-click the **Polyline** tool. The command line shows (In this example *enter* and *right-click* have not been included)

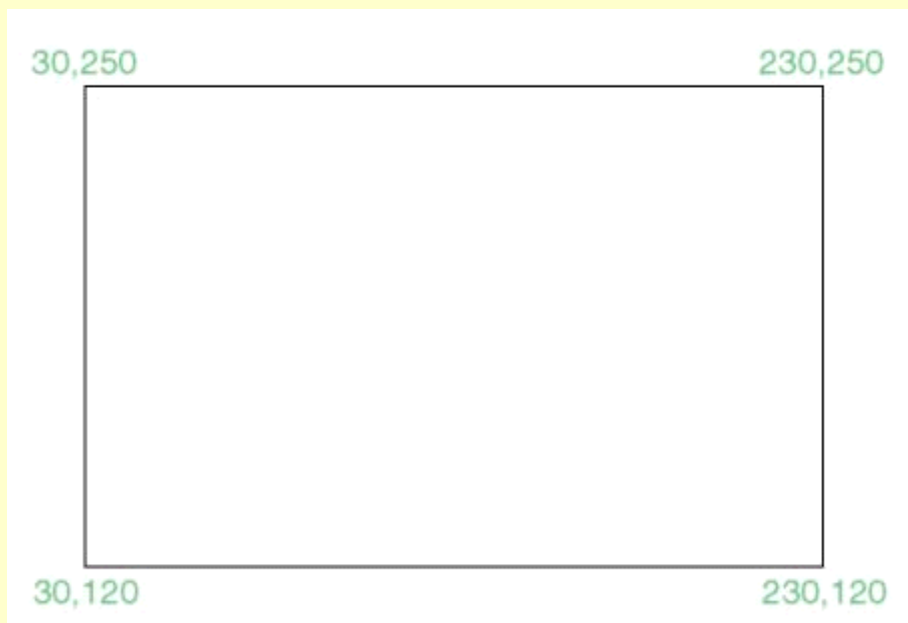**Command: _pline Specify start point:** 30,250

Current line width is 0

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: 230,250

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 230,120

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 30,120

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: c (Close)

Note the prompts – **Arc** for constructing pline arcs; **Close** to close an outline; **Halfwidth** to halve the width of a wide pline; **Length** to *enter* the required length of a pline; **Undo** to undo the last pline constructed; **Close** to close an outline.

Only the capital letter(s) of a prompt needs to be *entered* in upper or lower case to make that prompt effective.

Only the capital letter(s) of a prompt needs to be *entered* in upper or lower case to make that prompt effective.

Other prompts will appear when the **Polyline** tool is in use as will be shown in later examples.

Second example

This will be a long sequence, but it is typical of a reasonably complex drawing using the **Polyline** tool. In the following sequences, when aprompt line is to be repeated, the prompts in square brackets ([ ]) will be replaced by **[prompts]**.

Left-click the **Polyline** tool icon. The command line shows:

Command:_pline Specify start point: 40,250

Current line width is 0

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: w (Width)

Specify starting width <0>: 5

Specify ending width <5>: *right-click*

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 160,250

Specify next point or [prompts]: h (Halfwidth)

Specify starting half-width <2.5>: 1

Specify ending half-width <1>: *right-click*

Specify next point or [prompts]: 260,250

Specify next point or [prompts]: 260,180

Specify next point or [prompts]: w (Width)

Specify starting width <1>: 10

Specify ending width <10>: *right-click*

Specify next point or [prompts]: 260,120

Specify next point or [prompts]: h (Halfwidth)

Specify starting half-width <5>: 2

Specify ending half-width <2>: *right-click*

Specify next point or [prompts]: 160,120

Specify next point or [prompts]: w (Width)

Specify starting width <4>: 20

Specify ending width <20>: *right-click*

Specify next point or [prompts]: 40,120

Specify starting width <20>: 5

Specify ending width <5>: *right-click*

Specify next point or [prompts]: c (Close)

Third example

Left-click the **Polyline** tool icon. The command line shows:

Command:_pline Specify start point: 50,220

Current line width is 0

[prompts]: w (Width)

Specify starting width <0>: 0.5

Specify ending width <0.5>: *right-click*

Specify next point or [prompts]: 120,220

Specify next point or [prompts]: a (Arc)

Specify endpoint of arc or [prompts]: s (second pt)

Specify second point on arc: 150,200

Specify end point of arc: 180,220

Specify end point of arc or [prompts]: l (Line)

Specify next point or [prompts]: 250,220

Specify next point or [prompts]: 250,190

Specify next point or [prompts]: a (Arc)

Specify endpoint of arc or [prompts]: s (second pt)

Specify second point on arc: 240,170

Specify end point of arc: 250,150

Specify end point of arc or [prompts]: l (Line)

Specify next point or [prompts]: 250,150

Specify next point or [prompts]: 250,120

And so on until the outline in the picture is completed.

Fourth example

Left-click the **Polyline** tool icon. The command line shows:

Command:_pline Specify start point: 80,170

Current line width is 0

Specify next point or [prompts]: w (Width)

Specify starting width <0>: 1

Specify ending width <1>: *right-click*

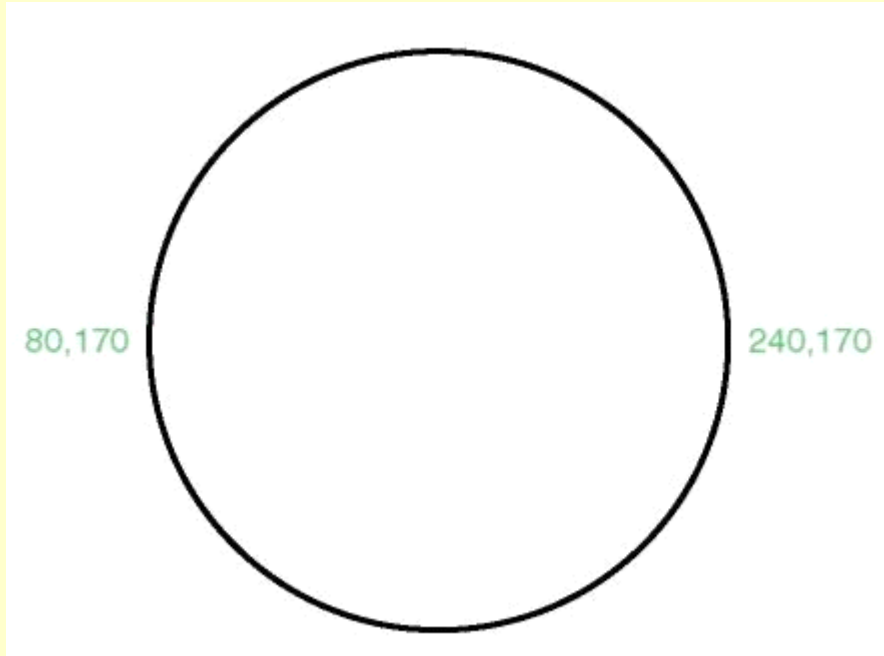**Specify next point or [prompts]:** a (Arc)

Specify endpoint of arc or [prompts]: s (second pt)

Specify second point on arc: 160,250

Specify end point of arc: 240,170

Specify end point of arc or [prompts]: cl (CLose)

The circle in the picture is formed

Fifth example

Left-click the **Polyline** tool icon. The command line shows:

Command:_pline Specify start point: 60,180

Current line width is 0

Specify next point or [prompts]: w (Width)

Specify starting width <0>: 1

Specify ending width <1>: *right-click*

Specify next point or [prompts]: 190,180

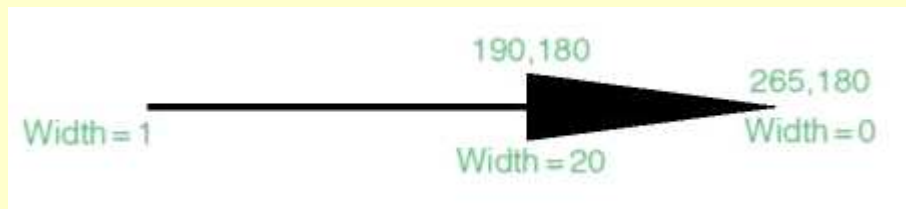**Specify next point or [prompts]:** w (Width)

Specify starting width <1>: 20

Specify ending width <20>: 0

Specify next point or [prompts]: 265,180

Specify next point or [prompts]: *right-click*

The arrow in the picture is formed.

Навчальне видання

Creation of AutoCAD entities: drawing a basketball court plan.

Guidance for laboratory works for "Informatics and computer modeling basics" course /for foreign students of 2nd year studying by orientation 6.060102 «Architecture»/

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