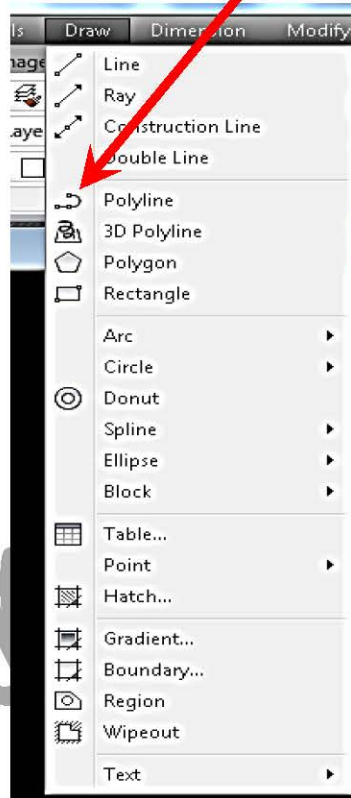## 2-3 POLYLINE COMMAND

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 act as one segment and its can be activated by one of these methods:

1. From tools bar menu, choose Draw >> Polyline.

2. Click the Polyline icon from the ribbon, ⌐Ɔ .

3. Type PLINE in the command bar or PL, as shown below:

Command: Pline or Pl, and press ENTER

Command: pl PLINE

Specify start point: (point)

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

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: A

Specify endpoint of arc or

[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: (point)

Specify endpoint of arc or

[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: L

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

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: A

Specify endpoint of arc or

[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: (point)

Press ESC or ENTER

- Command options appear on the command line. The capitalized letter(s) represents the letter(s) you enter to use that option. You are not required to enter the letter(s) as a capital letter.

- Options for the command appear within [...] brackets. If there is a default option for the command, it appears within <...> brackets. To use the default option, press ENTER.

Width

Used in order to give the thickness of the beginning of the line to be different from the thickness at the end, and its abbreviated character W. After you choose this command and press Enter, you receive the following message:

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

Specify starting width <0.0000>:
```

Enter the value of the width of the line to be 15 and then press Enter, the following message appears:

```
Specify starting width <0.0000>: 15

Specify ending width <15.0000>:
```

Which asks to set the width value to the end of the line, let be 25, and then press Enter, and if we continue draw line, we get the following line shown in Figure (2-4).
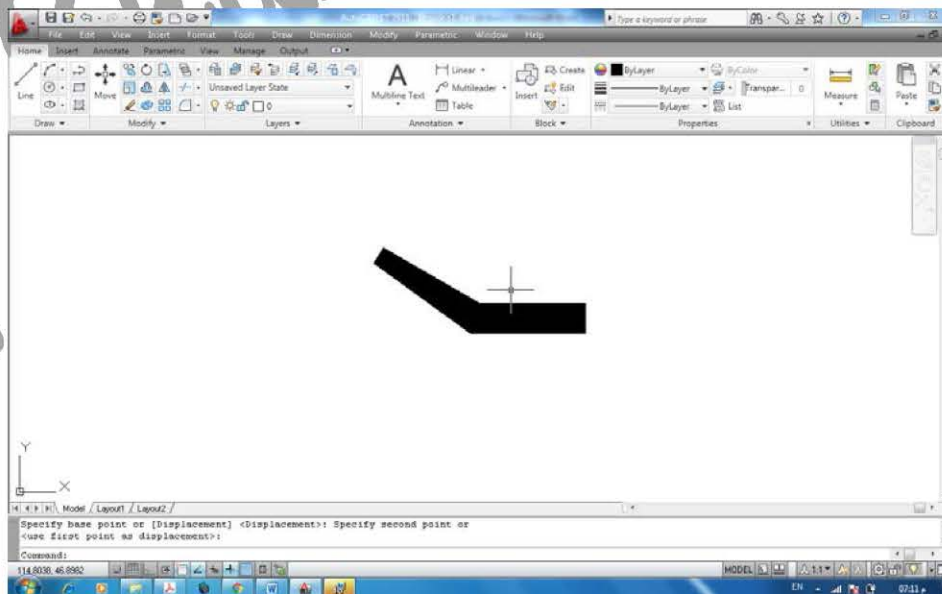
Fig. (2-4): Drawing using polyline command with width option.

➤ Note that the thickness at the beginning of the line differs from the thickness at the end.

➤ Note that Polyline line is drawn by the last value of the thickness entered.

Undo

This command option is used to undo the last drawn line, and its abbreviated character U.

Length

This command is used to increase the length of the free straight segment, the previous command that was drawn by a specific value being entered, and the same straightness, and its abbreviated character L.

Half width

This command is used to give half of the thickness value for the beginning of the line, half the thickness value for its end, and its abbreviated character H.

That is, if we enter the thickness values of the beginning of the line by 25, it will appear on the drawing screen by 50, as well as the thickness value of the end of the line. After you choose this command and pressing Enter, the following message appears:

```
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: h

Specify starting half-width <0.0000>:
```

Which asks for half of the thickness value of the beginning of the line to be 25 and then press Enter. This message appears next:

```
Specify starting half-width <0.0000>: 25

Specify ending half-width <25.0000>:
```

Which requires half the value of the thickness to the end of the line and be 100 and then Enter.

➢ Note that the thickness value at the beginning of the line is 50 and its value at the end is 200.

➢ You can switch between drawing line and arc by choosing arc command and line command.

Close

This command is used to close the polygon, and its abbreviated character **C**.

Arc

This command is used to draw arc, and its abbreviated character **A**. When you choose this command, you receive the following message that contains sub-commands options for the arc command:

```
Specify endpoint of arc or

[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:
```

We will now explain these orders separately:

➢ Halfwidth, Undo, and width command previously explained.

Second pt

This command is used to specify a second point that passes, and its abbreviated character **S**. After you choose this command, you receive the

following message:

```
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: s
Specify second point on arc:
```

Which requests the coordinates of the second point of the arc.

Radius

This command is used to give a radius value to the circle in which the arc will be cut off, and its abbreviated character **R**. After you choose this command, you receive the following message:

```
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: r
Specify radius of arc:
```

Which requires the insertion of the radius of the circle from which the arc will be cut. After you specifying radius, you receive the following message:

```
Specify radius of arc: 10
Specify endpoint of arc or [Angle]:
```

Which requires end point coordinate or the central angle of the arc.

Line

This command is used to return to the straight-line drawing for Polyline command, and its abbreviated character **L**.

Direction

Use this command to define the direction of the arc and his concavity, and its abbreviated character **D**.

CLose

This command is used to close the shape, and its abbreviated the two characters **CL**.

<u>CEnter</u>

This command is used to determine the center of the arc and its radius, and its abbreviation is **CE**. After you choose this command, you receive the following message:

```
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: ce
Specify center point of arc:
```

Which asks to select the center of the arc, and then the following message appears:

```
Specify center point of arc:
Specify endpoint of arc or [Angle/Length]:
```

Which asks to specify the end point of the arc, or determine the central angle of the arc through the command (Angle) or select the length of the arc through the command (Length).

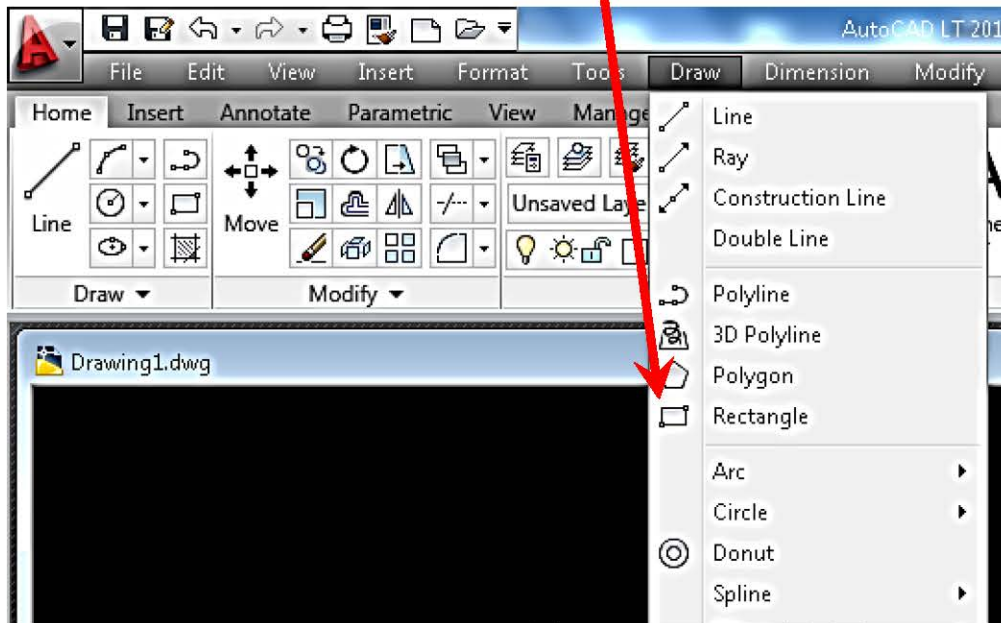*H.W. 2:* Write the command-statements for drawing the figure shown below using Polyline command starting from point (0,0) (the axis of symmetry A) and in the direction of the arrow shown in the figure.
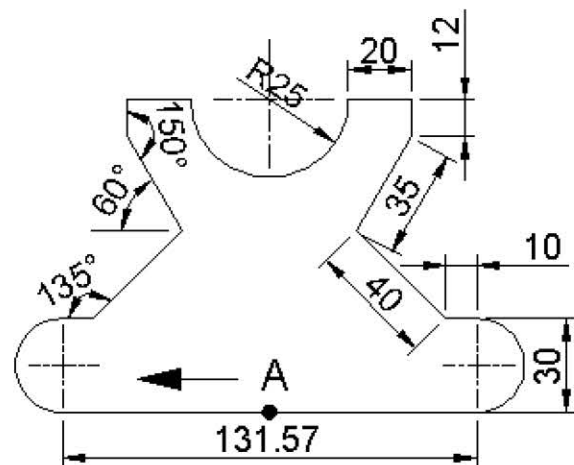
2-4 RECTANGLE

This command is used to draw rectangle. There are three methods to activate it:

1. From tools bar menu, choose Draw >> Rectangle.

2. Click the Rectangle icon from the ribbon, .

3. Type RECTANGLE in the command bar or REC, as shown below:

Command: Rectangle or rec, and press ENTER. Then the following message appears:

```
Command: rec RECTANG
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
```

Which asks to specify the first corner point of the rectangle, or determine the (Chamfer, Elevation, Fillet, Thickness, and Width).

The rectangle is drawn by entering the coordinates of the two diameter points. Enter the coordinates of the first point in one of the ways to enter the coordinates, after which the following message appears:

```
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
Specify other corner point or [Area/Dimensions/Rotation]:
```

then enter the other corner point coordinate, such as: @250,300 then press Enter. In this case, we have introduced the coordinates of the

second point of the diameter relative to the first point of the diameter that is the length and width of the rectangle. We will now explain the secondary commands of the command RECTANGLE.

Chamfer

The abbreviation of this command is character **C**. This command is used to create a cut for the rectangle corners, whose length is determined by the first axis length and the second axis length counterclockwise. After you execute this command, you receive the following message:

```
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: c
Specify first chamfer distance for rectangles <0.0000>:
```

Which asks for the length of the cut on the first leg to be 15 and then press Enter, the following message appears:

```
Specify first chamfer distance for rectangles <0.0000>: 15
Specify second chamfer distance for rectangles <15.0000>:
```

Which asks to determine the length of the cut on the second side to be 25 and then press Enter. Then we draw the rectangle as mentioned earlier, and note how to cut the corners of the rectangle.

Elevation

The abbreviation of this command is character **E**. This command is used to determine the level of this rectangle (ie the height of the rectangle on Z axis).

Fillet

The abbreviation of this command is character **F**. This command is used to convert the corners of the rectangle into circular arc, after selecting this command the following message appears:

```
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: f
Specify fillet radius for rectangles <0.0000>:
```

which asks to enter the arc radius, let it be 25, then press Enter, Then we draw rectangle. We notice how the corners of the rectangle were converted into circular arches.

Thickness

The abbreviation of this command is character **T**. This command is used to give thickness to the rectangle on the axis Z.

Width

The abbreviation of this command is character **W**. This command is used to specify the thickness of the rectangle lines, after selecting this command this message appears:

```
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: w
Specify line width for rectangles <0.0000>:
```

Which asks to determine the thickness of the lines of the rectangle, let it be 20, then press Enter. Then we draw the rectangle.

After you choose a rectangle drawing command and determine the coordinates of the first diameter point, you receive the following message:

```
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
Specify other corner point or [Area/Dimensions/Rotation]:
```

Area

The abbreviation of this command is character **A**. After you choose this command, you receive the following message:

```
Specify other corner point or [Area/Dimensions/Rotation]: a
Enter area of rectangle in current units <100.0000>:
```

Which asks to enter the value of the area of the rectangle on which the rectangle will be drawn, let it 200 and then press Enter. The following message appears:

```
Enter area of rectangle in current units <100.0000>: 200
Calculate rectangle dimensions based on [Length/Width] <Length>:
```

Which asks to choose the length or width of the rectangle. Choose length L then presses Enter. The following message appears:

```
Calculate rectangle dimensions based on [Length/Width] <Length>: l
Enter rectangle length <10.0000>: |
```

Enter the value of the length of the rectangle and set the value to 50 a rectangle of length 50 and area 200 is drawn.

Dimensions

The abbreviation of this command is character **D**. After you choose this command, you receive the following message:

```
Specify other corner point or [Area/Dimensions/Rotation]: d
Specify length for rectangles <10.0000>:
```

Which asks you to enter the value of the length of the rectangle, set the value to 80, then press Enter. The following message appears:

```
Specify length for rectangles <10.0000>: 80
Specify width for rectangles <20.0000>:
```

Which asks you to enter the width value of the rectangle, set the value to 40, then press Enter. Rectangle is drawn.

Rotation

The abbreviation of this command is character R. This command is used to rotate the rectangle at a certain angle from the horizon. After you choose this command, you receive the following message:

```
Specify other corner point or [Area/Dimensions/Rotation]: r
Specify rotation angle or [Pick points] <0>:
```

Which asks for the angle of rotation, set the angle to be 45, then press Enter. Rectangle is drawn.