Creation of AutoCAD entities: constructing a stands plan and small architectural forms arranging.

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

Укладачі: Б. П. Бочаров,
І. Л. Яковицький,
М. Ю Воєводіна,
Ю.В. Левіков.

Рецензент: канд.фіз-матем.наук А.Б. Костенко

Рекомендовано кафедрою Прикладної математики і Інформаційних технологій,
протокол № 6 від 24 січня 2009 р.
Theme: Creation of AutoCAD entities: constructing a stands plan and small architectural forms arranging

Purpose: confirming skills of work with drawing and displaying procedures, learning how to draw ellipses, polygons and rings in AutoCAD

Work order:

1. Start AutoCAD and open the drawing *tsk1.dwg*.

2. Create a new layer.

3. Set SNAP and GRID equal to 2 m

4. Around the basketball court, draw stands in the form of five ellipses. The centers of all the ellipses coincide with the center of basketball court, and the axes are magnified by 2 m.

5. Create a new layer.

6. Near the basketball court, draw a booth of 9-side polygon described around a circle of 3 m diameter.

7. Create a new layer.

8. Near the basketball court, draw a 7-side booth inscribed into a circle of 4 m diameter.

9. Create a new layer.

10. Set SNAP and GRID equal to 0.1m
11. Near the basketball court, draw 3 flowerbeds as rings (using the DONUT command). Internal diameters are 2m, 2.5m, and 3m. The widths of the rings are 0.2m, 0.3m, 0.4m.

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

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

**Draw Ellipses**

The shape of an ellipse is determined by two axes that define its length and width. The longer axis is called the major axis, and the shorter one is the minor axis.

The illustrations below show two different ellipses created by specifying axis and distance. The third point specifies only a distance and does not necessarily designate the axis endpoint.

If you are drawing on isometric planes to simulate 3D, you can use ellipses to represent isometric circles viewed from an oblique angle. First you need to turn on Isometric Snap in the Drafting Settings dialog box.
Drawing with the Ellipse tool examples

Ellipses can be regarded as what is seen when a circle is viewed from directly in front of the circle and the circle rotated through an angle about its horizontal diameter. Ellipses are measured in terms of two axes - a major axis and a minor axis, the major axis being the diameter of the circle, the minor axis being the height of the ellipse after the circle has been rotated through an angle.

To call the Ellipse tool, click on its tool icon in the 2D Draw control panel or click on its name in the Draw drop-down menu. The abbreviation for calling the Ellipse tool is el.

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

Command: _ellipse
Specify axis endpoint of elliptical arc or [Center]: 30,190
Specify other endpoint of axis: 150,190
Specify distance to other axis or [Rotation]: 25

Command:

Second example

In this second example, the coordinates of the centre of the ellipse (the point where the two axes intersect) are entered, followed by entering coordinates for the end of the major axis, followed by entering the units for the end of the minor axis.

Command: right-click
ELLIPSE
Specify axis endpoint of elliptical arc or [Center]: c
Specify center of ellipse: 260,190
Specify endpoint of axis: 205,190
Specify distance to other axis or [Rotation]: 30
Command:

Third example

Command: right-click
ELLIPSE
Specify axis endpoint of elliptical arc or [Center]: 30,100
Specify other endpoint of axis: 120,100
Specify distance to other axis or [Rotation]: r (Rotation)
Specify rotation around major axis: 45

Command:

Draw Rectangles

Use RECTANG command to create closed polylines in a rectangular shape. You can specify the length, width, area, and rotation parameters. You can also control the type of corners on the rectangle—fillet, chamfer, or square.

Toolbar: Draw

Menu: Draw ➤ Rectangle

At the Command prompt, enter rectang.
Command entry: rectang or rectangle

Drawing with the Rectangle tool examples

Call the Rectangle tool - either, with a click on its tool icon in the 2D Draw control panel, or by entering rec or rectangle at the command line.

The tool can be called from the Draw drop-down menu.
The command line shows:

Command:_rectang
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
25,240
Specify other corner point or [Area/Dimensions/Rotation]:
160,160
Command:

![Rectangle with chamfers](image)

Command:_rectang
[prompts]: c (Chamfer)
Specify first chamfer distance for rectangles <0>: 15
Specify first chamfer distance for rectangles <15>: right-click
Specify first corner point: 200,240
Specify other corner point: 300,160
Command:

![Rectangle with chamfers](image)

Command:_rectang
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: w
Draw Regular Polygons

Use POLYGON command to create closed polylines with between 3 and 1,024 equal-length sides. The following illustrations show polygons created using three methods. In each case, two points are specified.
Drawing with the Polygon tool examples

Call the **Polygon** tool - either with a *click* on its tool icon in the **2D Draw** control panel, by *entering pol* or *polygon* at the command line.

![Polygon tool icon](image)

It can be called from the **Draw** drop-down menu.

![Draw drop-down menu](image)

The command line shows

**Command:** `_polygon Enter number of sides <4>: 6`

**Specify center of polygon or [Edge]:** 60,210

**Enter an option [Inscribed in circle/Circumscribed about circle] <I>:** *right-click*
(accept Inscribed)

**Specify radius of circle:** 60

**Command:**

In the same manner construct a **5**-sided polygon of centre 200,210 and of radius 60.

Then, construct an **8**-sided polygon of centre 330,210 and radius 60.
Repeat to construct a **9**-sided polygon circumscribed about a circle of radius 60 and centre 60,80.

Construct yet another polygon with **10** sides of radius 60 and of centre 200,80.
Finally another polygon circumscribing a circle of radius 60, of centre 330,80 and sides **12**.

---

**Draw Donuts**

Donuts are filled rings or solid-filled circles that actually are closed polylines with width.
To create a donut, you specify its inside and outside diameters and its center. You can continue creating multiple copies with the same diameter by specifying different center points. To create solid-filled circles, specify an inside diameter of 0.

To create a donut
1. Click Draw menu ➔ Donut. At the Command prompt, enter donut.
2. Specify the inside diameter (1).
3. Specify the outside diameter (2).
4. Specify the center of the donut (3).
5. Specify the center point for another donut, or press ENTER to complete the command.
Creation of AutoCAD entities: constructing a stands plan and small architectural forms arranging.

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

Укладачі: Борис Петрович Бочаров,
Ігор Леонідович Яковицький,
Марія Юріївна Воєводіна,
Юрій Володимирович Левіков