

MINISTRY OF EDUCATION AND SCIENCE OF UKRAINE

**O. M. BEKETOV NATIONAL UNIVERSITY
OF URBAN ECONOMY in KHARKIV**

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GUIDELINES

for practical works on the course

«COMPUTER GRAPHICS»

(in english)

*(for the 2nd year full-time students for the Bachelor
by specialty 122 – Computer Science and Information Technology)*

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Practical work #1.

Launching AutoCAD. AutoCAD files and folders

Purpose: acquiring skills of work with disks, folders and files (documents) in AutoCAD

Work order:

1. Create a folder with the same name as the name of your group in “My documents” folder.
2. Create a personal folder, the folder name is your surname (use the English alphabet), in this folder.
3. Open AutoCAD.
4. Create 4 drawings in AutoCAD and save them in your personal folder (the surname is written in the English alphabet):
 - *surname_ris1.dwg*
 - *surname_ris2.dwg*
 - *surname_ris3.dwg*
 - *surname_ris4.dwg*
5. Exit AutoCAD.
6. Open AutoCAD.
7. Open all these 4 drawings created before and locate them with the “Arrange Icons” in the Window menu.
8. In every drawing, draw a picture using the lines, circles and arcs drawing commands.
9. Save all the drawings.
10. Compress all the drawings in the same archive folder.
11. Save this archive in the Academy distance learning system.

Basic theoretical information.

Operations with AutoCAD Drawings

A quick way to begin a new drawing is to start from scratch, which starts a drawing that uses settings from a default drawing template file.

You can start from scratch from the Create New Drawing dialog box, from the Select Template dialog box, or without using a dialog box at all. In each case, you use either the NEW command or the QNEW command.

Create New Drawing Dialog Box

The dialog box defines the settings for a new drawing. Start from Scratch creates a new drawing using either imperial or metric default settings. Use a Template creates a new drawing using the settings defined in a drawing template you select. Use a Wizard creates a new drawing using the settings you specify in either

the Quick or Advanced wizard. (The first option, Open a Drawing, is not available from the NEW command. To open an existing drawing, use OPEN.)

Standard File Selection Dialog Boxes

Several commands display standard file selection dialog boxes, in which you can navigate through local and network drives and through FTP sites and Web folders to select files. While each dialog box may vary slightly, following is a list of the possible options.

Partial Open Dialog Box

The dialog box displays the drawing views and layers available for specifying what geometry to load into the selected drawing. When working with large drawing files, you can select the minimal amount of geometry you need to load when opening a drawing.

When a drawing is partially open, all named objects, as well as the specified geometry, are loaded into the file. Named objects include blocks, dimension styles, layers, layouts, linetypes, text styles, UCSs, views, and viewport configurations.

Save AutoCAD Drawings

You save drawing files for later use just as you do with other Microsoft Windows applications. You can also set up automatic saving and backup files and save only selected objects.

When you work on a drawing, you should save it frequently. Saving protects you from losing work in the event of a power failure or other unexpected event. If you want to create a new version of a drawing without affecting the original drawing, you can save it under another name.

The file extension for drawing files is .dwg, and unless you change the default file format in which drawings are saved, drawings are saved in the latest drawing file format. This format is optimized for file compression and for use on a network.

The character limit for a DWG file name (including its path) is 256 characters.

If FILEDIA is set to 1 the *Save Drawing As* standard file selection dialog box is displayed. When FILEDIA is set to 0, SAVEAS displays command prompts.

Note. AutoCAD 2004 is the drawing file format used by the AutoCAD 2004, AutoCAD 2005, and AutoCAD 2006 releases.

In the *Save Drawing As* dialog box, Tools Options displays the Saveas Options dialog box, which controls various DWG and DXF settings.

DWG Options Tab (Saveas Options Dialog Box) controls the drawing index and display of custom objects and specifies the default file format for saving drawings.

Practical work #2.

AutoCAD interface. Commands input methods, the use of menus and tool bars. Work with layers. Points coordinates input methods.

Purpose: acquiring skills of work with layers, learning how to enter commands and points coordinates in AutoCAD

Work order:

1. Create a new AutoCAD drawing and save it in the personal folder.
2. Create 10 layers (with names *layer1*, *layer 2* ..., *layer 10*) with different colors.
3. For every layer:
 - make it current,
 - draw a circle with the identical center coordinates and diameters for all the layers.
4. Turn off all the layers, except for *layer1*.
5. Remove the circle drawn in the layer.
6. Turn on all the layers.
7. Create *layer11* and draw a rectangle with sides 14 and 24, the left lower corner coordinates are $0,0$. Enter the coordinates of the points using the mouse.
8. Create *layer12*. Draw a rectangle with sides 14 and 24 with the coordinates of the left lower corner are $0,0$. Enter the absolute coordinates of the points with the keyboard.
9. Create *layer13*. Draw a rectangle with sides 14 and 24, the coordinates of the left lower corner are $0,0$. Enter the relative coordinates of the points with the keyboard.
10. Create *layer14*. Draw a rectangle with sides 14 and 24, the coordinates of the left lower corner are $0,0$. Enter the absolute polar coordinates of the points with the keyboard.
11. Create *layer15*. Draw a rectangle with sides 14 and 24, the coordinates of the left lower corner are $0,0$. Enter the relative polar coordinates of the points with the keyboard.
12. Save the drawing in the personal folder and compress it (**ZIP**).
13. Save this archive in the Academy distance learning system.

Basic theoretical information.

The User Interface

You can use several menus, shortcut menus, toolbars, and the dashboard to access frequently used commands, settings, and modes.

Toolbars

Use buttons on toolbars to start commands, display flyout toolbars, and display tooltips. You can display or hide, dock, and resize toolbars.

Toolbars contain buttons that start commands. When you move your mouse or pointing device over a toolbar button, the tooltip displays the name of the button. Buttons with a small black triangle in the lower-right corner are flyout toolbars that contain related commands. With the cursor over the icon, hold down the left button on your mouse until the flyout toolbar is displayed.

The Standard toolbar at the top of the drawing area is displayed by default. This toolbar is similar to those found in Microsoft Office programs. It contains frequently used AutoCAD commands such as PROPERTIES, PAN, and ZOOM, as well as Microsoft Office standard commands such as New, Open, and Save.

Status Bars

The application and drawing status bars provide useful information, buttons for turning on and off drawing tools.

The application status bar displays the coordinate values of your cursor along with several buttons for turning on and off drawing tools and may display tools for scaling annotations. You can choose which buttons you want to display in the status bar.



The drawing status bar displays several tools for scaling annotations.

Different tools display for model space and paper space.

When the drawing status bar is turned on it displays at the bottom of the drawing area. When the drawing status bar is turned off, the tools found on the drawing status bar are moved to the application status bar.

When the drawing status bar is turned on, you can use the Infobar menu to choose which tools display in status bar.

The Menu Bar

Display pull-down menus from the menu bar using one of several methods. You can also specify alternate menus.

Menus are available from the menu bar at the top of the AutoCAD drawing area.

To use a menu choose one of the following methods:

- On the menu bar, click a menu name to display a list of options. On the menu, either click an option or use the DOWN ARROW to move down the list, and then press ENTER.
- Press ALT and press the key for the underlined letter in the menu name; then, press the underlined letter in the option name. For example, to open a new drawing, press ALT and press F to open the File menu; then, press N for New.

You can specify menus to display in the program by customizing a CUI file and loading it into the program.

Shortcut Menus

Display a shortcut menu for quick access to commands that are relevant to your current activity.

You can display different shortcut menus when you right-click different areas of the screen, including

- Within the drawing area with or without any objects selected
- Within the drawing area during a command
- Within the text and command windows
- Within areas and on icons in DesignCenter
- Within areas and on text in the In-Place Text Editor
- On a toolbar or tool palette
- On the Model or layout tabs
- On the status bar or the status bar buttons
- In certain dialog boxes
- Shortcut menus typically include options to
 - Repeat the last command entered
 - Cancel the current command
 - Display a list of recent user input
 - Cut, copy, and paste from the Clipboard
 - Select a different command option
 - Display a dialog box, such as Options or Customize
 - Undo the last command entered

You can customize right-click behavior to be time-sensitive, so that a quick right-click acts the same as pressing ENTER, and a longer right-click displays a shortcut menu.

Shortcut menus can be customized using a customization (CUI) file. The main CUI file is called acad.cui by default.

The Dashboard

The dashboard is a special palette that displays buttons and controls that are associated with a task-based workspace.

The dashboard provides a single interface element for operations that are relevant to the current workspace. The dashboard eliminates the need to display a lot of toolbars and reduces the clutter in the application window. Thus, you can maximize the area available for work, and use a single interface for speed and convenience.

The dashboard opens automatically by default when you use either the 2D Drafting & Annotation workspace or the 3D Modeling workspace. You can open the dashboard manually:

- Click Tools menu ► Palettes ► Dashboard.
- At the Command prompt, enter *dashboard*.

The dashboard is organized into a series of control panels. Each control panel includes related tools and controls that are similar to the tools in toolbars and controls in dialog boxes.

The large icons displayed at the left side of the dashboard are called control panel icons. Each control panel icon identifies the purpose of the control panel. On some control panels, if you click the icon, a slide-out panel opens that contains additional tools and controls. An open slide-out panel closes automatically when you click another control panel icon. Only one slide-out panel is displayed at a time.

A tool palette group can be associated with each control panel. To display the associated tool palette group, click a tool or open a slide-out panel.

You can customize the display of the dashboard (display or hide, dock, and resize the dashboard) in the following ways:

- Size the dashboard horizontally. If not enough space is available to display all the tools in a row, a black down-arrow called an overflow control is displayed.
- Use the shortcut menu to control the auto-hide, docking, and anchoring behavior. With anchoring, the dashboard results is displayed as a narrow space-saving strip along the left or right edge of the drawing area.

You can customize the dashboard in the following ways:

- You can create and modify dashboard panels using the Customize User Interface dialog box.
- You can specify which control panels are displayed by right-clicking the dashboard and, on the shortcut menu, clicking or clearing the names of control panels.
- You can associate a customizable tool palette group with each control panel on the dashboard. Right-click the control panel to display a list of available tool palette groups.

Enter Commands on the Command Line

You can enter a command by using the keyboard. Some commands also have abbreviated names called command aliases.

To enter a command by using the keyboard, type the full command name on the command line and press ENTER or SPACEBAR.

Note. When Dynamic Input is on and is set to display dynamic prompts, you can enter many commands in tooltips near the cursor.

Some commands also have abbreviated names. For example, instead of entering line to start the LINE command, you can enter l. Abbreviated command names are called command aliases and are defined in the acad.pgp file.

To find a command, you can type a letter on the command line and press TAB to cycle through all the commands that begin with that letter. Press ENTER or SPACEBAR. Restart a recently used command by right-clicking on the command line.

When you enter commands on the command line, you see either a set of options or a dialog box. For example, when you enter circle at the Command prompt, the following prompt is displayed:

Specify center point for circle or [3P/2P/Ttr (tan, tan, radius)]:

You can specify the center point either by entering X,Y coordinate values or by using the pointing device to click a point on the screen.

To choose a different option, enter the letters capitalized in one of the options in the brackets. You can enter uppercase or lowercase letters. For example, to choose the three-point option (3P), enter 3p.

To execute commands, press SPACEBAR or ENTER, or right-click your pointing device after entering command names or responses to prompts. The instructions in Help assume this step and do not specifically instruct you to press ENTER after each entry.

If you want to repeat a command that you have just used, press ENTER or SPACEBAR, or right-click your pointing device at the Command prompt.

You also can repeat a command by entering multiple, a space, and the command name, as shown in the following example:

Command: multiple circle

To cancel a command in progress, press ESC.

Many commands can be used transparently: that is, they can be entered on the command line while you use another command. Transparent commands frequently change drawing settings or display options, for example, GRID or ZOOM. In the Command Reference, transparent commands are designated by an apostrophe in front of the command name.

Enter Coordinates

Absolute and relative 2D Cartesian and polar coordinates determine precise locations of objects in a drawing.

You can use absolute or relative Cartesian (rectangular) coordinates to locate points when creating objects.

To use *Cartesian* coordinates to specify a point, enter an X value and a Y value separated by a comma (X,Y). The X value is the positive or negative distance, in units, along the horizontal axis. The Y value is the positive or negative distance, in units, along the vertical axis.

Absolute coordinates are based on the UCS origin (0,0), which is the intersection of the X and Y axes. Use absolute coordinates when you know the precise X and Y values of the point.

With Dynamic Input, you can specify absolute coordinates with the # prefix. If you enter coordinates on the command line instead of in the tooltip, the # prefix is not used. For example, entering #3,4 specifies a point 3 units along the X axis and 4 units along the Y axis from the UCS origin. For more information about Dynamic Input, see *Use Dynamic Input*.

Use Dynamic Input

Dynamic Input provides a command interface near the cursor to help you keep your focus in the drafting area.

When Dynamic Input is on, tooltips display information near the cursor that is dynamically updated as the cursor moves. When a command is active, the tooltips provide a place for user entry.

After you type a value in an input field and press TAB, the field then displays a lock icon, and the cursor is constrained by the value that you entered. You can then

enter a value for the second input field. Alternately, if you type a value and press ENTER, the second input field is ignored and the value is interpreted as direct distance entry.

The actions required to complete a command or to use grips are similar to those for the command prompt. The difference is that your attention can stay near the cursor.

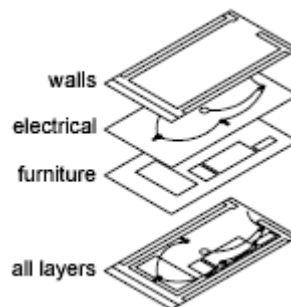
Dynamic Input is not designed to replace the command window. You can hide the command window to add screen area for drawing, but you will need to display it for some operations. Press F2 to hide and display command prompts and error messages as needed. Alternately, you can undock the command window and use Auto-hide to roll open or roll up the window.

Work with Layers

Layers are like transparent overlays on which you organize and group objects in a drawing.

Layers are used to group information in a drawing by function and to enforce linetype, color, and other standards.

Layers are the equivalent of the overlays used in paper-based drafting. Layers are the primary organizational tool used in drawing. You use layers to group information by function and to enforce linetype, color, and other standards.

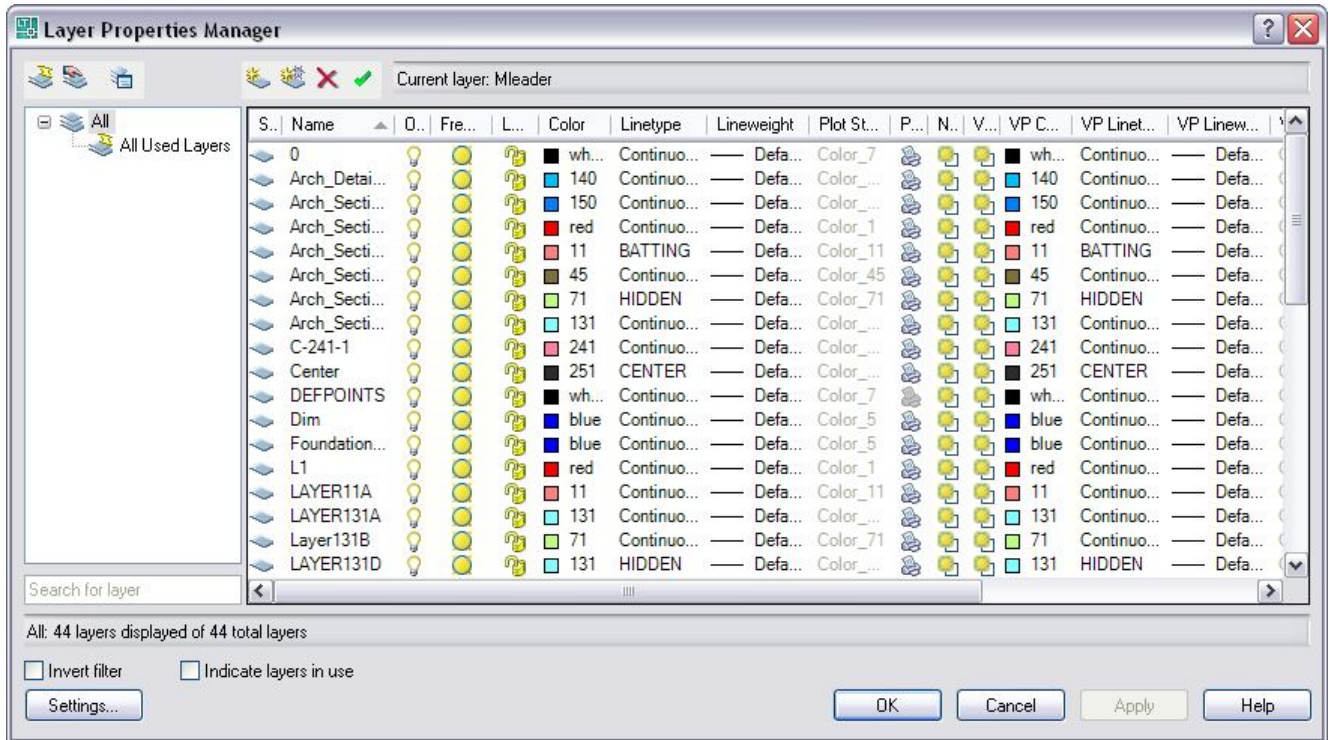


By creating layers, you can associate similar types of objects by assigning them to the same layer. For example, you can put construction lines, text, dimensions, and title blocks on separate layers. You can then control the following:

- Whether objects on a layer are visible or dimmed in any viewports
- Whether and how objects are plotted
- What color is assigned to all objects on a layer
- What default linetype and lineweight are assigned to all objects on a layer
- Whether objects on a layer can be modified
- Whether objects display with different layer properties in individual layout viewports
- Every drawing includes a layer named 0. Layer 0 cannot be deleted or renamed. It has two purposes:
 - Ensure that every drawing includes at least one layer
 - Provide a special layer that relates to controlling colors in blocks

Layer Properties Manager

It displays a list of the layers in the drawing and their properties. You can add, delete, and rename layers, change their properties, set property overrides for layout viewports, or add descriptions. Layer filters control which layers are displayed in the list and can also be used to make changes to more than one layer at a time.



Practical work #3.

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.

Basic theoretical information.

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

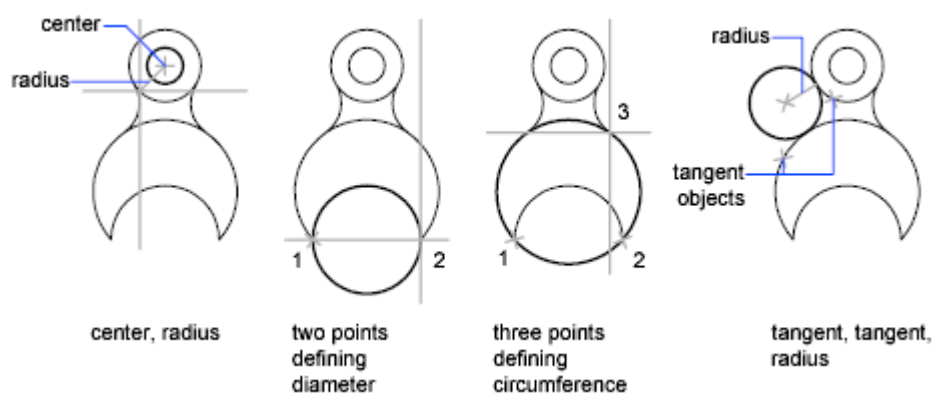
- Enter the coordinate values for an endpoint, using either absolute or relative coordinates
- Specify an object snap relative to an existing object. For example, you can specify the center of a circle as one endpoint of the line
- Turn grid snap on and snap to a location

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

Draw Circles

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

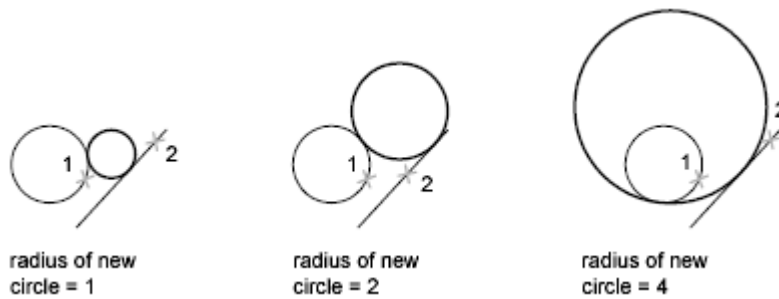
You can create circles in several ways. The default method is to specify the center and the radius. Three other ways to draw a circle are shown in the illustration.



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.



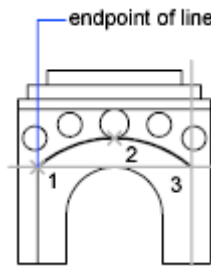
To create a circle tangent at three points, set running object snaps (OSNAP) to Tangent and use the three-point method to create the circle.

Draw Arcs

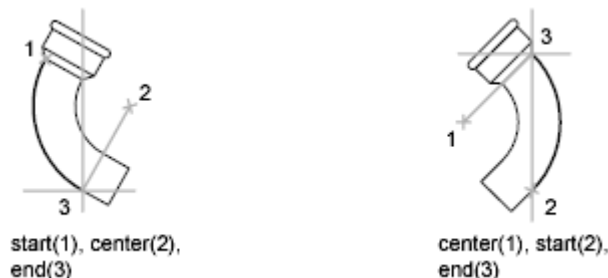
To create an arc, you can specify various combinations of center, endpoint, start point, radius, angle, chord length, and direction values.

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.

Draw Arcs by Specifying Three Points. 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.



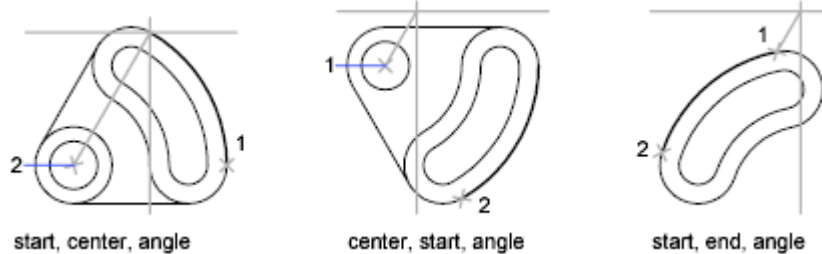
Draw Arcs by Specifying Start, Center, End. When you know the start point, center point, and endpoint, you can draw an arc by specifying either the start point or the center point first. The center point is the center of a circle that the arc is part of.



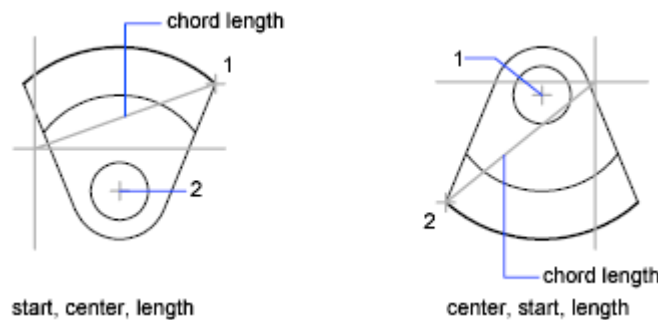
Draw Arcs by Specifying Start, Center, Angle. When you have a start point and a center point you can snap to and you know the included angle, use the Start, Center, Angle or the Center, Start, Angle option.



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.



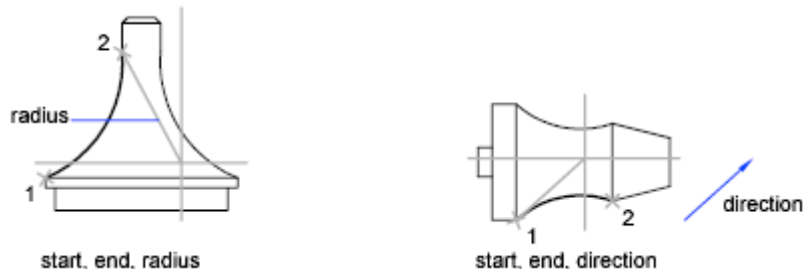
Draw Arcs by Specifying Start, Center, Length. When you have a start point and a center point you can snap to; when you know the chord length, use the Start, Center, Length or the Center, Start, Length option.



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.

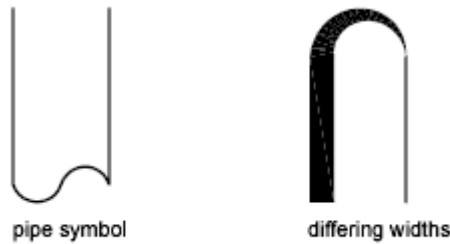
The illustration on the left shows an arc drawn by specifying a start point, endpoint, and radius. You can specify the radius by entering a length or by moving the pointing device clockwise or counterclockwise and clicking to specify a distance.



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 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.



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.



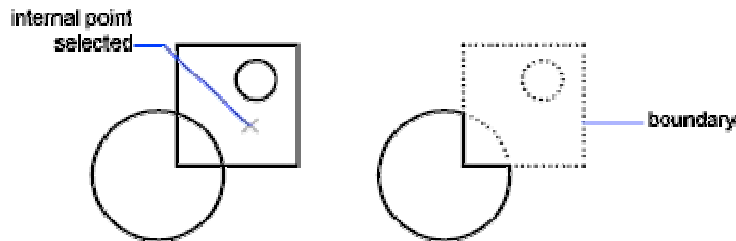
The Width and Halfwidth options set the width of the next polyline segments you draw. Zero (0) width produces a thin line. Widths greater than zero produce wide lines, which are filled if Fill mode is on and outlined if Fill mode is off. The Halfwidth option sets width by specifying the distance from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



Practical work #4.

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

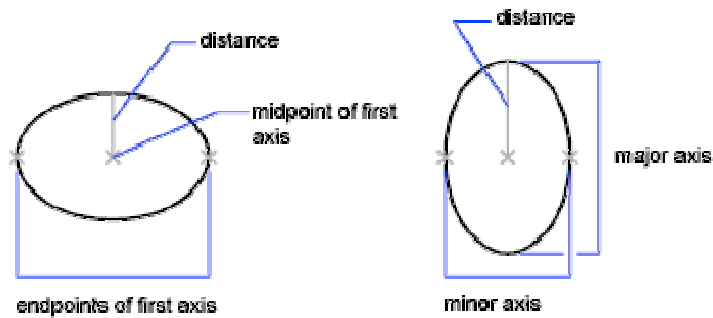
Work order:

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.

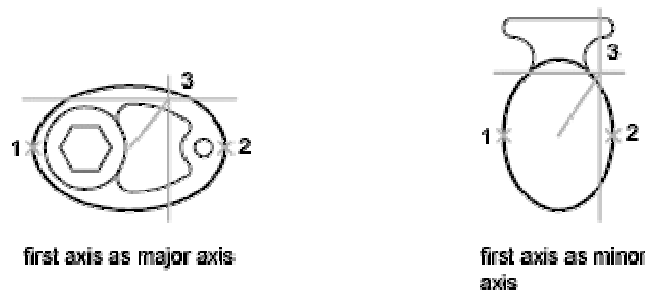
Basic theoretical information.

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.

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.

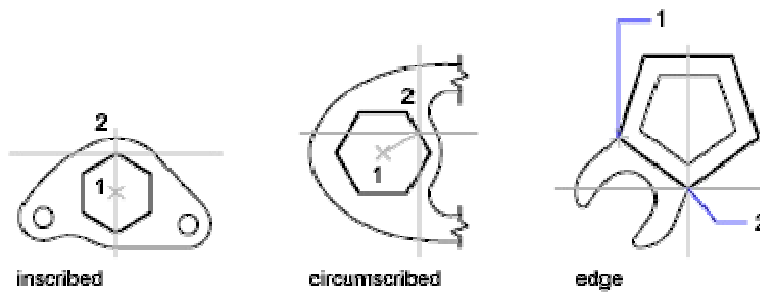
Toolbar: Draw 

Menu: Draw ► RectangleAt the Command prompt, enter `rectang`.

Command entry: `rectang` or `rectangle`

Draw Regular Polygons

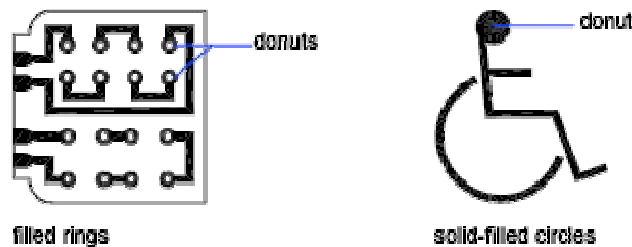
Use POLYGON command to create closed polylines with between 3 and 1,024 equal-length sides. The following illustrations show polygons created using three methods. In each case, two points are specified.



Draw Donuts

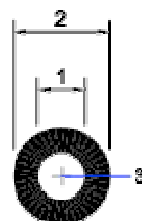
Donuts are filled rings or solid-filled circles that actually are closed polylines with width.

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.



Practical work #5.

Editing the AutoCAD entities: constructing a summer theater and circus plan

Purpose of work: entities creation skills developing, working with layers, learning how to copy and remove objects, create objects arrays

Work order:

1. Start AutoCAD and open the drawing *tsk1.dwg*.
2. Draw a summer theater plan.
 - 2.1. Create a new layer.
 - 2.2. Using the PLINE command (width 0), draw seats in form of a square with a side of 1 m.
 - 2.3. Using the ARRAY command (rectangular array), create a hall for an audience: 15 rows of seats with 10 ones in each row. The distance between the rows is 1 m, the distance between the seats is 0.1 m.
 - 2.4. Create a new layer.
 - 2.5. Using the PLINE command (width 0), draw a stage in a rectangular form.
3. Draw a summer circus plan.
 - 3.1. Create a new layer for the seats of the circus.
 - 3.2. Copy one of the seats into a free place.
 - 3.3. Using the CHANGE command, carry a chair into the created layer.
 - 3.4. Using the ARRAY (polar array) command, create a hall for an audience of four identical sectors. In every sector there are 5 circular rows of seats. Passage-ways are placed between the sectors. Pick the distance between the seats in a row and between the rows yourself.
 - 3.5. Create a new layer.
 - 3.6. In the center of the circus draw an arena in a form of a circle.
4. Save the drawing in the personal folder and compress it (Zip).
5. Save this archive in the Academy distance learning system.

Basic theoretical information.

Modify Objects

You can easily modify the size, shape, and location of objects. You can

- Enter a command first, and then select the objects to modify.
- Select the objects first, and then enter a command to modify them.
- Select and right-click an object to display a shortcut menu with relevant options.
- Double-click an object to display the Properties palette or, in some cases, a dialog box or editor that is specific to that type of object. (You can specify the double click action for each object type by customizing a CUI file and loading it into the program.)

Select Objects

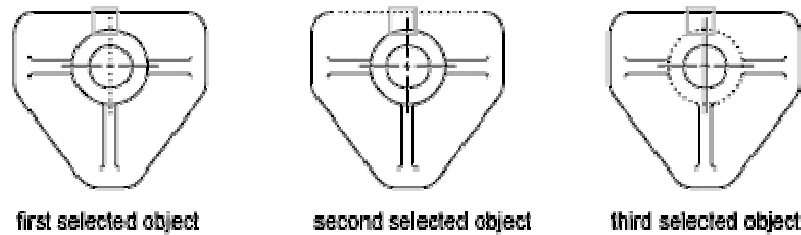
You have a wide range of options when you need to select objects for editing operations.

Select Objects Individually. At the Select Objects prompt, you can select one or more objects individually.

Use the Pickbox Cursor. When the square pickbox cursor is in position to select an object, the object is highlighted. Click to select the object.

You can control the size of the pickbox in the Options dialog box, Selection tab.

Select Objects Close Together. It is difficult to select objects that are close together or lie directly on top of one another. The example shows two lines and a circle that lie within the pickbox.



If selection preview is turned on, you can cycle through the objects by rolling over the object on top to highlight it, and pressing and holding SHIFT and then pressing SPACEBAR continuously. When the required object is highlighted, left-click to select it.

If selection preview is turned off, hold down SHIFT + SPACEBAR and click to cycle through these objects, one after the other, until the one you want is selected. Press ESC to turn off cycling.

Remove Selection from Objects

Remove objects from the current selection set by holding down SHIFT and selecting them again.

Correct Mistakes

You can backtrack your recent actions using one of several methods.

Undo a Single Action. The simplest method of backtracking is to use Undo on the Standard toolbar or the U command to undo a single action. Many commands include their own U (undo) option so that you can correct mistakes without leaving the command. When you are creating lines and polylines, for example, enter u to undo the last segment. By default, the UNDO command is set to combine consecutive pan and zoom commands into a single operation when you undo or redo. However, pan and zoom commands that are started from the menu are not combined, and always remain separate actions.

Undo Several Actions at Once. Use the Mark option of UNDO to mark an action as you work. You can then use the Back option of UNDO to undo all actions that occurred after the marked action. Use the Begin and End options of UNDO to define a set of actions to be treated as a group.

You can also undo several actions at once with the Undo list on the Standard toolbar.

Reverse the Effect of Undo. You can reverse the effect of a single U or UNDO command by using REDO immediately after using U or UNDO.

You can also redo several actions at once with the Redo list on the Standard toolbar.

Erase Objects. You can erase any object that you draw. If you accidentally erase the wrong object, you can use the UNDO command or the OOPS command to restore it.

Cancel a Command. You can cancel a command without completing it by pressing ESC.

Erase Objects

You can erase objects from your drawing using several methods.

1. Erase them with ERASE.
2. Select them and then cut them to the Clipboard with CTRL+X.
3. Select them and press DELETE.

You can restore accidentally erased objects using UNDO. The OOPS command restores all objects deleted by the most recent use of ERASE.

Clean Up the Display

You can remove the plus-shaped markers called blips and stray pixels that may be left over from some editing operations from the display area.

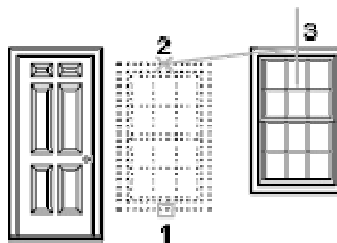
- To remove blips, use REDRAW.
- To remove stray pixels, use REGEN.

Move Objects

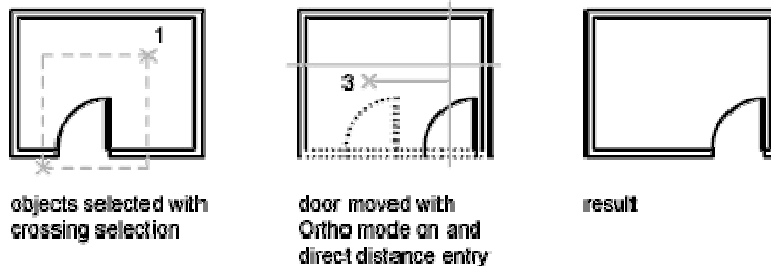
You can move objects at a specified distance and direction from the originals.

Use coordinates, grid snap, object snaps, and other tools to move objects with precision.

Move an object using the distance and direction specified by a base point followed by a second point. In this example, you move the block representing a window. Click Edit menu ► Move. Then select the object to be moved (1). Specify the base point for the move (2) followed by a second point (3). The object is moved the distance and direction of point 2 to point 3.



A practical example is moving a door in a wall. The door in the illustration is entirely within a crossing selection, while the wall lines are only partly within the crossing selection area.



The result is that only the endpoints that lie within the crossing selection move.

Rotate Objects

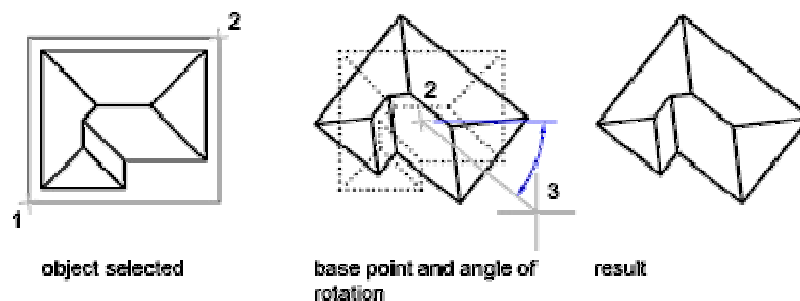
You can rotate objects in your drawing around a specified base point.

To determine the angle of rotation, you can enter an angle value, drag using the cursor, or specify a reference angle to align to an absolute angle.

Rotate an Object by a Specified Angle. Enter a rotation angle value from 0 to 360 degrees. You can also enter values in radians, grads, or surveyor bearings. Entering a positive angle value rotates the objects counterclockwise or clockwise, depending on the Direction Control setting in the Drawing Units dialog box.

Rotate an Object by Dragging. Drag the object around the base point and specify a second point. Use Ortho mode, polar tracking, or object snaps for greater precision.

For example, you can rotate the plan view of a house by selecting the objects (1), specifying a base point (2), and specifying an angle of rotation by dragging to another point (3).

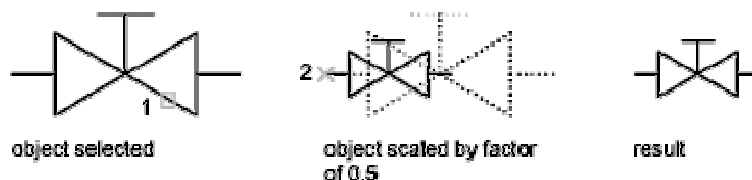


Scale Objects

With SCALE, you can make an object uniformly larger or smaller. To scale an object, you specify a base point and a scale factor. Alternatively, you can specify a length to be used as a scale factor based on the current drawing units.

A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object.

Scaling changes the size of all dimensions of the selected object. A scale factor greater than 1 enlarges the object. A scale factor less than 1 shrinks the object.



Note. When you use the SCALE command with annotative objects, the position or location of the object is scaled relative to the base point of the scale operation, but the size of the object is not changed.

You can also scale by reference. Scaling by reference uses an existing distance as a basis for the new size. To scale by reference, specify the current distance and then the new desired size. For example, if one side of an object is 4.8 units long and you want to expand it to 7.5 units, use 4.8 as the reference length.

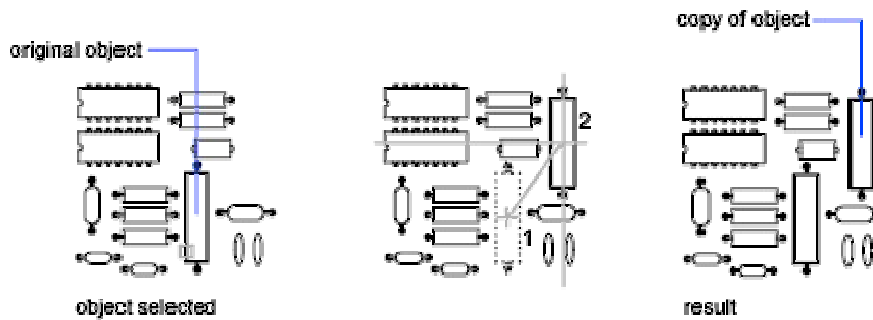
You can use the Reference option to scale an entire drawing. For example, use this option when the original drawing units need to be changed. Select all objects in the drawing. Then use Reference to select two points and specify the intended distance. All the objects in the drawing are scaled accordingly.

Copy Objects

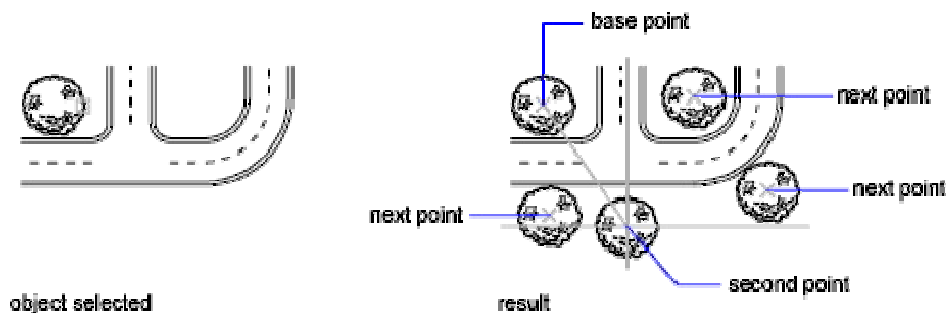
You can create duplicates of objects at a specified distance and direction from the originals.

Use coordinates, grid snap, object snaps, and other tools to copy objects with precision.

Copy an object using the distance and direction specified by a base point followed by a second point. In this example, you copy the block representing an electronic component. Click Edit menu > Copy. At the Command prompt, enter copyclip. Then select the original object to be copied. Specify the base point for the move (1) followed by a second point (2). The object is copied the distance and direction of point 1 to point 2.



The COPY command repeats automatically by default. To exit the command, press ENTER. To change the default, use the COPYMODE system variable.

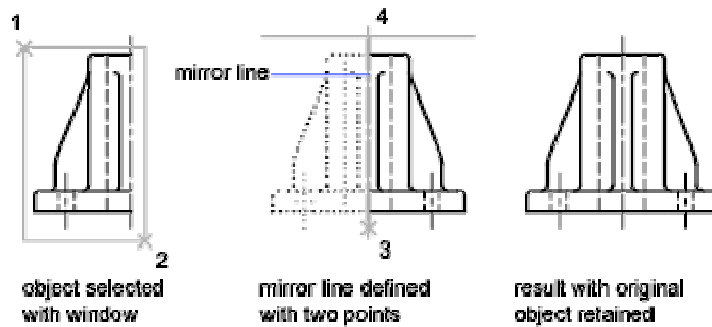


Mirror Objects

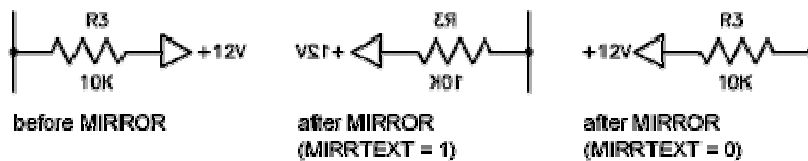
You can flip objects about a specified axis to create a symmetrical mirror image.

Mirroring is useful for creating symmetrical objects because you can quickly draw half the object and then mirror it instead of drawing the entire object.

You flip objects about an axis called a mirror line to create a mirror image. To specify this temporary mirror line, you enter two points. You can choose whether to erase or retain the original objects.



By default, when you mirror text, attributes, and attribute definitions, they are not reversed or turned upside down in the mirror image. The text has the same alignment and justification as before the object was mirrored. If you do want text to be reversed, set the `MIRRTXT` system variable to 1.

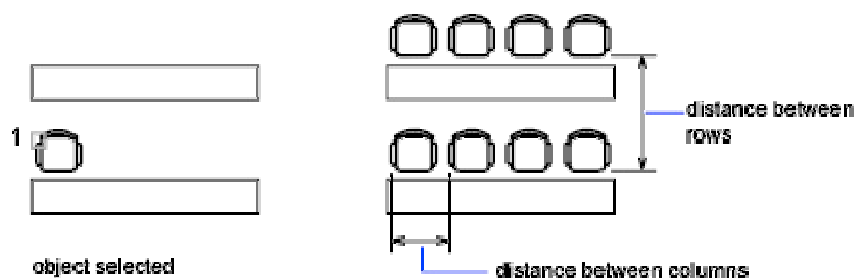


`MIRRTXT` affects text that is created with the `TEXT`, `ATTDEF`, or `MTEXT` commands; attribute definitions; and variable attributes. Text and constant attributes that are part of an inserted block are reversed when the block is mirrored regardless of the `MIRRTXT` setting.

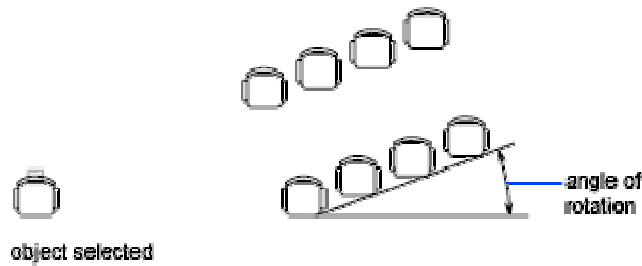
Create an Array of Objects

You can create copies of objects in a rectangular or polar (circular) pattern called an array.

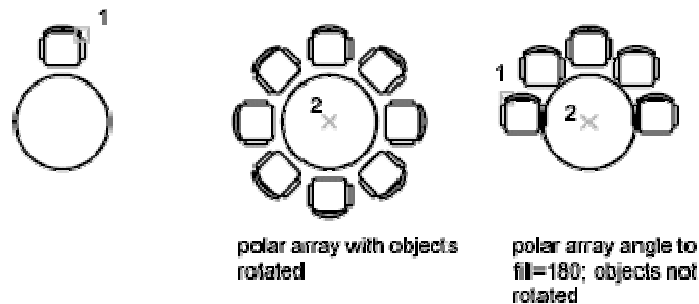
For rectangular arrays, you control the number of rows and columns and the distance between each.



A rectangular array is built along a baseline defined by the current snap rotation angle. This angle is zero by default, so the rows and columns of a rectangular array are orthogonal with respect to the X and Y axes. The default angle 0 direction setting can be changed in `UNITS`.



For polar arrays, you control the number of copies of the object and whether the copies are rotated. To create many regularly spaced objects, arraying is faster than copying.



Practical work #6.

Changing the properties of the AutoCAD entities: constructing park plan variants

Purpose: skills of entities creating and editing developing, working with layers, learning to change the properties of objects

Work order:

1. Start AutoCAD and open the drawing tsk1.dwg.
2. Create a new layer PLAN1.
3. Copy all the drawing objects into a free area.
4. Remove the copy of all the drawing objects into the layer PLAN1.
5. On every object in the layer PLAN1, change the COLOR property (this property of all the objects must have the initial value BYLAYER). The color of every object must correspond to the color of the layer in which this object has been created in.
6. Think over the layout of the objects in the park.
7. Using the MOVE command, put the stadium, the theater and the circus into proper places.
8. Place a few booths and flowerbeds into the park.
9. Using polylines of different widths, draw avenues, roads and paths.
10. Create a new layer PLAN2.

11. Copy all the objects of the layer PLAN1 into a free area.
 12. Remove the objects copies into the layer PLAN2.
 13. Create a new variant of the park planning (pp. 6-9).
 14. Save the drawing in the personal folder and compress it (Zip).
- Save this archive in the Academy distance learning system.

Basic theoretical information.

Object Properties

Every object you draw has properties. Some properties are general and apply to most objects; for example, layer, color, linetype, and plot style. Other properties are object-specific; for example, the properties of a circle include radius and area, and the properties of a line include length and angle.

Most general properties can be assigned to an object by layer or can be assigned to an object directly.

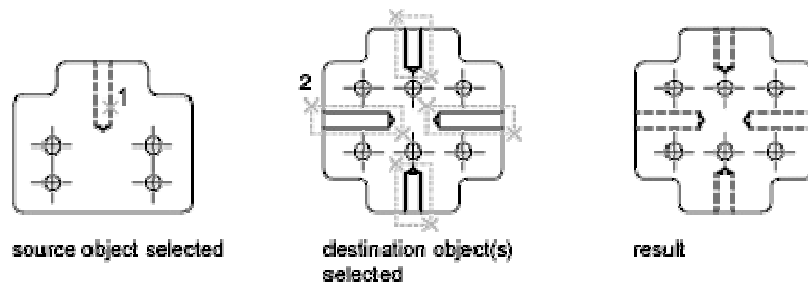
- When a property is set to the value BYLAYER, the object is assigned the same value as the layer on which it is drawn. For example, if a line drawn on Layer0 is assigned the color BYLAYER, and Layer0 is assigned the color Red, the line is red.
- When a property is set to a specific value, that value overrides the value set for the layer. For example, if a line drawn on Layer0 is assigned the color Blue, and Layer0 is assigned the color Red, the line is blue.

Copy Properties Between Objects

You can copy some or all properties of one object to other objects using Match Properties.

The types of properties that can be copied include, but are not limited to, color, layer, linetype, linetype scale, linewidth, plot style, viewport property overrides, and 3D thickness.

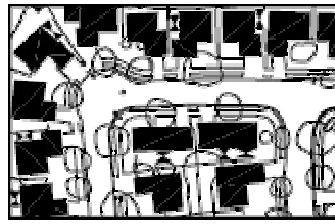
By default, all applicable properties are automatically copied from the first object you selected to the other objects. If you don't want a specific property or properties to be copied, use the Settings option to suppress the copying of that property. You can choose the Settings option at any time during the command.



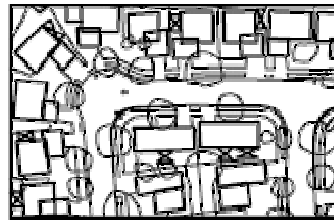
Control the Display Properties of Certain Objects

When you turn off Fill mode, wide polylines, solid-filled polygons, gradient fill, and hatches are displayed in outline form. Except for patterned hatches and

gradient fills, solid fill is automatically turned off for hidden view and nonplan views in three dimensions.



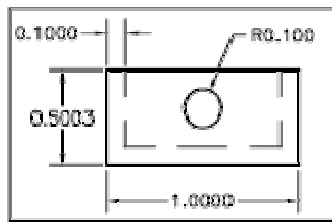
Fill mode on



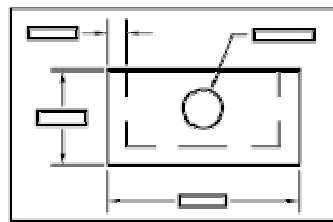
Fill mode off

When you turn on Quick Text mode in drawings that contain a lot of text using complex fonts, only a rectangular frame defining the text is displayed or plotted.

Any lineweight width that is represented by more than one pixel may slow down performance. If you want to improve display performance, turn lineweights off. You can turn lineweights on and off by choosing the LWT button on the status bar or by using the Lineweight Settings dialog box. Lineweights are always plotted at their real-world value whether their display is turned on or off.



Quick Text mode off



Quick Text mode on

New objects automatically use the current settings for displays of solid fill and text. Except for lineweights, to update the display of existing objects using these settings, you must use REGEN.

Practical work #7.

Geometrical constructing with the use of Object Snap.

Purpose: acquiring skills of geometrical constructing with the use of Object Snap

Work order:

1. Create a new drawing - tsk2.dwg and save it in the personal folder.
2. Create a new layer.
3. Disable the snap mode.
4. Draw 10 circles of different diameters.
5. Join their centers with a closed polyline (none-zero width).
6. Create a new layer.

7. Disable the snap mode.
8. Draw 8 segments and 5 arcs of arbitrary sizes.
9. Join their centers with a closed polyline (none-zero width).
10. Join their extreme points with a closed polyline (none-zero width).
11. Create a new layer.
12. Disable the snap mode.
13. Draw a circle, 5 segments, intersecting the circle fully, and 7 arcs, intersecting the circle fully.
14. Join together the points of the circle and segments, the circle and arcs intersections with a closed polyline (none-zero width).
15. Create a new layer.
16. Copy all the entities created in pp. 2-14.
17. Remove all these entities into the new layer (using the CHANGE command).
18. Rotate all the entities (around the point located to the left of the drawing), consistently, on 43, 178, 251, -36 degrees.
19. Create a new layer.
20. Copy all the entities created in pp. 2-14.
21. Transfer all these entities into a new layer (using the CHANGE command).
22. Change the sizes of all the entities (a base point is found approximately in the center of the drawing), consistently, in 3.45, 0.79, 35.78/67.45 times.
23. Save the drawing in the personal folder and compress it (Zip).
24. Save this archive in the Academy distance learning system.

Basic theoretical information.

Use Precision Tools

You can use a variety of precision drawing tools to help you produce accurate drawings quickly and without performing tedious calculations.

You can use Drafting Settings Dialog Box that specifies drafting settings organized for drawing aids in four categories: Snap and Grid, Polar Tracking, Object Snap, and Dynamic Input.

Snap and Grid

To enhance drawing speed and efficiency, you can display and snap to a rectangular grid. You can also control its spacing, angle, and alignment.

The grid is a rectangular pattern of dots or lines that extends over the area you specify as the grid limits. Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps you align objects and visualize the distances between them. The grid is not plotted.

Snap mode restricts the movement of the crosshairs to intervals that you define. When Snap mode is on, the cursor seems to adhere, or "snap," to an invisible rectangular grid. Snap is useful for specifying precise points with the arrow keys or the pointing device.

Grid mode and Snap mode are independent but are often turned on at the same time.

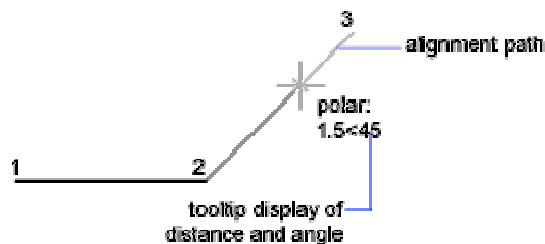
Polar Tracking

Polar tracking restricts cursor movement to specified angles. PolarSnap restricts cursor movement to specified increments along a polar angle.

When you are creating or modifying objects, you can use polar tracking to display temporary alignment paths defined by the polar angles you specify. In 3D views, polar tracking additionally provides an alignment path in the up and down directions. In that case, the tooltip displays a +Z or -Z for the angle.

Polar angles are relative to the orientation of the current user coordinate system (UCS) and the setting for the base angle convention in a drawing. The angle base direction is set in the Drawing Units dialog box.

Use PolarSnap to snap to specified distances along the alignment path. For example, in the following illustration you draw a two-unit line from point 1 to point 2, and then draw a two-unit line to point 3 at a 45-degree angle to the line. If you turn on the 45-degree polar angle increment, an alignment path and tooltip are displayed when your cursor crosses the 0 or 45-degree angle. The alignment path and tooltip disappear when you move the cursor away from the angle.



As you move your cursor, alignment paths and tooltips are displayed when you move the cursor near polar angles. The default angle measurement is 90 degrees. Use the alignment path and tooltip to draw your object. You can use polar tracking with Intersection and Apparent Intersection object snaps to find where a polar alignment path intersects another object.

Note. Ortho mode and polar tracking cannot be on at the same time. Turning on polar tracking turns off Ortho mode. Similarly, PolarSnap and grid snap cannot be on at the same time. Turning on PolarSnap turns off grid snap.

Object Snap

Use object snaps to specify precise locations on objects. For example, you can use an object snap to draw a line to the center of a circle or to the midpoint of a polyline segment.

You can specify an object snap whenever you are prompted for a point. By default, a marker and a tooltip are displayed when you move the cursor over an object snap location on an object. This feature, called AutoSnap™, provides a visual clue that indicates which object snaps are in effect.

To specify an object snap at a prompt for a point, you can

- Press SHIFT and right-click to display the Object Snap shortcut menu.
- Click an object snap button on the Object Snap toolbar.
- Enter the name of an object snap at the command prompt.

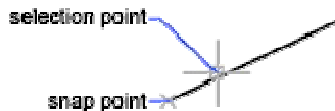
When you specify an object snap at a prompt for a point, the object snap stays in effect only for the next point that you specify.

Note. Object snaps work only when you are prompted for a point. If you try to use an object snap at the Command prompt, an error message is displayed.

Object Snap Modes

Lists object snaps that you can turn on as running object snaps.

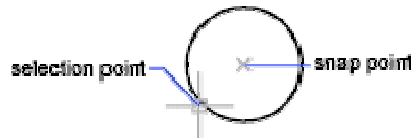
Endpoint snaps to the closest endpoint of an arc, elliptical arc, line, multiline, polyline segment, spline, region, or ray, or to the closest corner of a trace, solid, or 3D face.



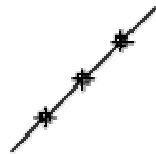
Midpoint snaps to the midpoint of an arc, ellipse, elliptical arc, line, multiline, polyline segment, region, solid, spline, or xline.



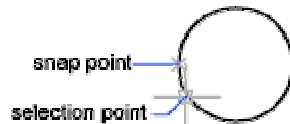
Center snaps to the center of an arc, circle, ellipse, or elliptical arc.



Node snaps to a point object, dimension definition point, or dimension text origin.



Quadrant snaps to a quadrant point of an arc, circle, ellipse, or elliptical arc.



Intersection snaps to the intersection of an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, region, spline, or xline. Extended Intersection is not available as a running object snap.

Intersection and Extended Intersection do not work with edges or corners of 3D solids.



Note. You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time.

Extension causes a temporary extension line or arc to be displayed when you pass the cursor over the endpoint of objects, so you can specify points on the extension.

Note. When working in perspective view, you cannot track along the extension line of an arc or elliptical arc.

Insertion snaps to the insertion point of an attribute, a block, a shape, or text.

Perpendicular snaps to a point perpendicular to an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, region, solid, spline, or xline. Deferred Perpendicular snap mode is automatically turned on when the object you are drawing requires that you complete more than one perpendicular snap. You can use a line, arc, circle, polyline, ray, xline, multiline, or 3D solid edge as an object from which to draw a perpendicular line. You can use Deferred Perpendicular to draw perpendicular lines between such objects. When the aperture box passes over a Deferred Perpendicular snap point, an AutoSnap tooltip and marker are displayed.



Tangent snaps to the tangent of an arc, circle, ellipse, elliptical arc, or spline. Deferred Tangent snap mode is automatically turned on when the object you are drawing requires that you complete more than one tangent snap. You can use Deferred Tangent to draw a line or xline that is tangent to arcs, polyline arcs, or circles. When the aperture box passes over a Deferred Tangent snap point, a marker and an AutoSnap tooltip are displayed.



Note. When you use the From option in conjunction with the Tangent snap mode to draw objects other than lines from arcs or circles, the first point drawn is tangent to the arc or circle in relation to the last point selected in the drawing area.

Nearest snaps to the nearest point on an arc, circle, ellipse, elliptical arc, line, multiline, point, polyline, ray, spline, or xline.

Apparent Intersection snaps to the visual intersection of two objects that are not in the same plane but may appear to intersect in the current view. Extended Apparent Intersection is not available as a running object snap. Apparent and Extended Apparent Intersection do not work with edges or corners of 3D solids.

Note. You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time.

Parallel constrains a line segment, polyline segment, ray or xline to be parallel to another linear object. After you specify the first point of a linear object, specify the parallel object snap. Unlike other object snap modes, you move the cursor and hover over another linear object until the angle is acquired. Then, move the cursor back

toward the object that you are creating. When the path of the object is parallel to the previous linear object, an alignment path is displayed, which you can use to create the parallel object.

Note. Turn off ORTHO mode before using the parallel object snap. Object snap tracking and polar snap are turned off automatically during a parallel object snap operation. You must specify the first point of a linear object before using the parallel object snap.

Dynamic Input

Dynamic Input provides a command interface near the cursor to help you keep your focus in the drafting area.

When Dynamic Input is on, tooltips display information near the cursor that is dynamically updated as the cursor moves. When a command is active, the tooltips provide a place for user entry.

After you type a value in an input field and press TAB, the field then displays a lock icon, and the cursor is constrained by the value that you entered. You can then enter a value for the second input field. Alternately, if you type a value and press ENTER, the second input field is ignored and the value is interpreted as direct distance entry.

The actions required to complete a command or to use grips are similar to those for the command prompt. The difference is that your attention can stay near the cursor.

Dynamic Input is not designed to replace the command window. You can hide the command window to add screen area for drawing, but you will need to display it for some operations. Press F2 to hide and display command prompts and error messages as needed. Alternately, you can undock the command window and use Auto-hide to roll open or roll up the window.

Pointer Input

When pointer input is on and a command is active, the location of the crosshairs is displayed as coordinates in a tooltip near the cursor. You can enter coordinate values in the tooltip instead of on the command line.

The default for second and subsequent points is relative polar coordinates (relative Cartesian for RECTANG). There is no need to type the at sign (@). If you want to use absolute coordinates, use the pound sign (#) prefix. For example, to move an object to the origin, for the second point prompt, enter #0,0.

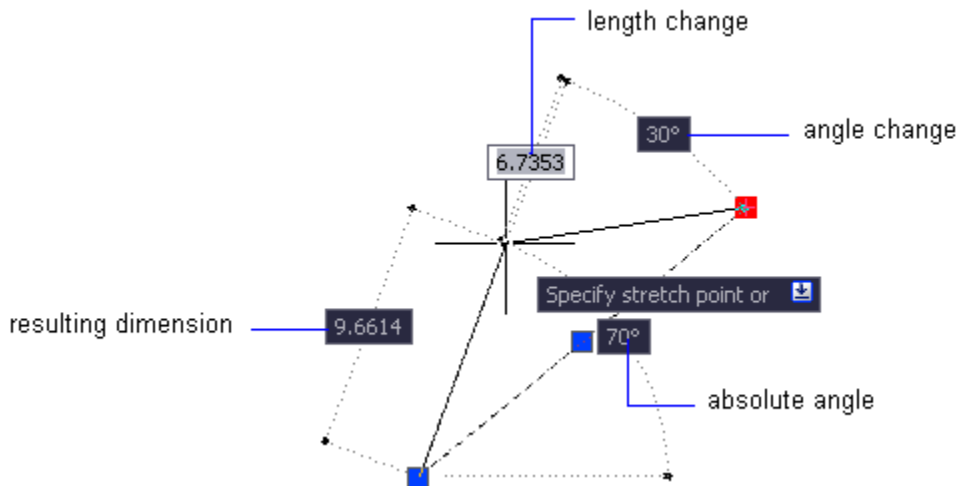
Use the pointer input settings to change the default format for coordinates and to control when pointer input tooltips are displayed.

Dimensional Input

When dimensional input is on, the tooltips display distance and angle values when a command prompts for a second point. The values in the dimensional tooltips change as you move the cursor. Press TAB to move to the value you want to change. Dimensional input is available for ARC, CIRCLE, ELLIPSE, LINE, and PLINE.

When you use grips to edit an object, the dimensional input tooltips can display the following information:

- Original length
- A length that updates as you move the grip
- The change in the length
- Angle
- The change in the angle as you move the grip
- The radius of an arc



Use the dimensional input settings to display only the information you want to see.

When you use grips to stretch objects or when you create new objects, dimensional input displays only acute angles, that is, all angles are displayed as 180 degrees or less. Thus, an angle of 270 degrees is displayed as 90 degrees regardless of the ANGDIR system variable setting (set in the Drawing Units dialog box). Angles specified when creating new objects rely on the cursor location to determine the positive angle direction.

Dynamic Prompts

When dynamic prompts are on, prompts are displayed in a tooltip near the cursor. You can enter a response in the tooltip instead of on the command line. Press the DOWN ARROW key to view and select options. Press the UP ARROW key to display recent input.



Note. To use PASTECLIP in a dynamic prompt tooltip, type a letter and then backspace to delete it before you paste the entry. Otherwise, the entry is pasted into the drawing as text.

Practical work #8.

Working with a text in the AutoCAD program

Purpose: **acquiring skills of a text input and editing**

Work order:

1. Open the drawing *tsk2.dwg*.
2. Create a new layer.
3. Using the TEXT command, write a short autobiography (you must use the command TEXT options Start point, Align, Fit, Center, Mid point).
4. Create a new layer.
5. Copy the text created in p. 3 into a free place.
6. Transfer the text into the new layer (using the CHANGE command).
7. Mirror the text (using the MIRROR command) in relation to a line lying to the left of the text (allow AutoCAD to change the text orientation).
8. Mirror the source text in relation to a line lying to the right of text (do not allow AutoCAD to change the text orientation).
9. Mirror the text (the MIRROR command) in relation to a line lying below the text (allow AutoCAD to change the text orientation).
10. Mirror the source text in relation to a line lying above the text (deny AutoCAD changing the text orientation).
11. Create a new layer.
12. Draw a none-zero width closed polyline consisting of 5 straight lines and 5 arc segments.
13. Use the BREAK command in each of the segments to «place windows», that is, to delete a part of every segment.
14. Save the drawing in the personal folder and compress it (Zip).
15. Save this archive in the Academy distance learning system.

Basic theoretical information.

Creating Text

The text you add to your drawings conveys a variety of information. It may be a complex specification, title block information, a label, or even part of the drawing.

For short entries that do not require multiple fonts or lines, create single-line text. Single-line text is most convenient for labels.

For long, complex entries, create multiline, or paragraph text. Multiline text consists of any number of text lines or paragraphs that fit within a width you specify; it can extend vertically to an indefinite length.

Regardless of the number of lines, each set of paragraphs created in a single editing session forms a single object, which you can move, rotate, erase, copy, mirror, or scale.

There are more editing options for multiline text than there are for single-line text. For example, you can apply underlining, fonts, color, and text height changes to individual characters, words, or phrases within a paragraph.

Use annotative text for notes and labels in your drawing. You create annotative text by using an annotative text style, which sets the height of the text on the paper.

For more information about creating and working with an annotative text, see *Create Annotative Text*.

Create Single-Line Text

You can use single-line text to create one or more lines of text, where each text line is an independent object that you can relocate, reformat, or otherwise modify.

Use single-line text (TEXT) to create one or more lines of text, ending each line when you press ENTER. Each text line is an independent object that you can relocate, reformat, or otherwise modify.

When you create single-line text, you assign a text style and set alignment. The text style sets the default characteristics of the text object. The alignment determines what part of the text character aligns with the insertion point. Use the TEXT command to enter the text in-place, or enter -text on the command line to enter the text on the command line.

You can insert a field in single-line text. A field is text that is set up to display data that might change. When the field is updated, the latest value of the field is displayed.

The text styles used for single-line text are the same as those used for multiline text. When you create text, you assign an existing style by entering its name at the Style prompt. If you need to apply formatting to individual words and characters, use multiline text instead of single-line text.

You can also compress single-line text to fit between points that you specify. This option stretches or squeezes the text to fill the designated space.

The DTEXTED system variable specifies the user interface displayed for editing single-line text.

Create Multiline Text

You can create one or more paragraphs of multiline text (mtext) in the In-Place Text Editor (or an alternative text editor) or use prompts on the command line. You can also insert text from a file saved in ASCII or RTF format.

Before entering or importing text, you specify opposite corners of a text bounding box that defines the width of the paragraphs in the multiline text object. The length of the multiline text object depends on the amount of text, not the length of the bounding box. You can use grips to move or rotate a multiline text object.

The In-Place Text Editor displays the bounding box with a ruler at the top and the Text Formatting toolbar. The editor is transparent so that, as you create text, you can see whether the text overlaps other objects. To turn off transparency while you

work, select Opaque Background on the Options menu. You can also make the background of the finished multiline text object opaque and set its color.

You can also insert fields in multiline text. A field is text that is set up to display data that might change. When the field is updated, the latest value of the field is displayed.

Text Style

Most characteristics of the text are controlled by the text style, which sets the default font and other options, such as line spacing, justification, and color. You can use the current text style or select a new one. The STANDARD text style is the default.

Within the multiline text object, you can override the current text style by applying formatting such as underlining, boldface, and different fonts to individual characters. You can also create stacked text, such as fractions or geometric tolerances and insert special characters, including Unicode characters, for TrueType fonts.

Note. Not all SHX and TrueType text fonts support Unicode characters.

Text Properties

In the Properties palette, you can view and change the object properties of a multiline text object, including properties that apply specifically to text.

- Justification determines where text is inserted with respect to the bounding box and sets the direction of text flow as text is entered.
- Line space options control the amount of space between lines of text.
- Width defines the width of the bounding box and therefore controls where the text wraps to a new line.
- Background inserts an opaque background so that objects under the text are masked.

In-Place Text Editor

The MTEXT command creates paragraphs of text as a single multiline text (mtext) object

After you specify the point for the opposite corner, the in-place text editor is displayed.

If you specify one of the other options, or if you enter -mtext at the command prompt, MTEXT bypasses the In-Place Text editor and displays additional command prompts.

In-Place Text Editor Creates or modifies single or multiline text objects. You can import or paste text from other files to use in multiline text, set tabs, adjust paragraph and line spacing and alignment, and create and modify columns.

The In-Place Text Editor includes a Text Formatting Toolbar, a Paragraph Dialog Box, a Columns Menu, and an Display Options Menu. When a table cell is selected for editing, the In-Place Text Editor displays column letters and row numbers.

Practical work #9.

Building floor plans and facades computer-aided design.

Purpose: to learn how to use AutoCAD for a real architectural project automated implementation.

To develop skills of entities creating and editing, working with layers, geometrical constructing, using drawing and displaying procedures, text information input and editing. To acquire skills of design drawing constructing for the projects executed in the AutoCAD program. To learn how to plot the drawings on paper.

This laboratory work is basic for the process of two-dimensional graphics studying.

It is necessary to build plans of a cottage (not less than two floors). Apartments plans and specifications development for two floors will be described later on. However it is very useful to add something to the project (for example, a basement).

Work order:

1. Preliminary work (without a computer).
 - 1.1. Think the project over and draw sketches.
 - 1.2. Define approximately which information will be plotted on A4 format paper. Minimum set of leaves (for a two-storied cottage): the ground and the 1st floor plans and apartment specifications.
 - 1.3. Taking into account the objects sizes, define the scale to be used on the A4 format paper. You must enter all the objects sizes in natural scale units (1 unit of drawing equals to 1 meter). However to define basic data of project is necessary just at this stage.
2. Create a new drawing.
3. Create a new layer to draw a boundary and a stamp for the A4 format. Enter the boundary and stamp sizes in natural scale units (1 unit of drawing equals to 1 millimeter).
4. Draw the plan of the ground floor.
 - 4.1. In a new layer, draw a sketch of the ground floor using thin lines. Windows and doors are not to be indicated at this stage. Enter the objects sizes in natural scale plan units (1 unit of the drawing is equal to 1 millimeter).
 - 4.2. Create doorways and window openings.
 - 4.3. In a new layer, outline the walls with wide polylines.
 - 4.4. In a new layer insert windows and doors.
 - 4.5. In a new layer insert the numbers of apartments.
5. Draw the ground floor apartments specification.
 - 5.1. Create a new layer.
 - 5.2. Copy and paste the boundary and the stamp in this layer.
 - 5.3. In the boundary, create a list of the apartments. All the sizes related to the text enter in millimeters.

6. Copy and paste the information for the 1st floor in new layers. Edit the information.
 - 6.1. Create necessary new layers.
 - 6.2. Copy and paste the information for the ground floor in new layers.
 - 6.3. Edit the information and get a plan and apartments specification for the 1st floor.
7. Place the floors plans within the boundaries.
 - 7.1. Create a new layer.
 - 7.2. Copy and paste the boundary and the stamp on this layer.
 - 7.3. Match the boundary and plan sizes of the ground floor. Do not change the plan, but change the sizes of the boundary in accordance with the scale chosen.
 - 7.4. Transfer the plan to a proper place within the boundary.
 - 7.5. Repeat actions of pp. 7.1-7.4 for the 1st floor.
8. Print out the information, using **Window** output region determination option and **Fit** output scale determination option.
9. Save the drawing in the personal folder and compress it (Zip).
10. Save this archive in the Academy distance learning system.

Basic theoretical information.

Create Single-View Drawings (Model Space)

If you are going to create a two dimensional drawing that has one view, you can create the drawing and its annotation entirely in model space. This is the traditional method for creating drawings with AutoCAD.

Model Space Drafting

The process of creating and plotting a drawing file in model space is very different from the process used in manual drafting.

In AutoCAD, there are two distinct working environments that are represented by the Model and layout tabs. These tabs are located near the bottom of the drawing area.

If you are going to create a two-dimensional drawing that has one view, you can create both the model and its annotation entirely in model space, not using a layout tab. This is the traditional method for creating drawings with AutoCAD. This method is simple but has several limitations, including

- It is suitable for 2D drawings only.
- It does not support multiple views and view-dependent layer settings.
- Scaling the annotation and title block requires computation unless you use annotative objects.

With this method, you always draw geometric objects at full scale (1:1) and text, dimensions, and other annotation at a scale

Draw, Scale, and Annotate in Model Space

If you draw and plot from model space, you must determine and apply a scale factor to annotate objects before you plot.

You can draw and plot entirely from model space. This method is useful primarily for two-dimensional drawings that have a single view.

Create Multiple-View Drawing Layouts (Paper Space)

Paper space is a sheet layout environment where you can specify the size of your sheet, add a title block, display multiple views of your model, and create dimensions and notes for your drawing.

There are two distinct working environments, or “spaces,” in which you can create objects in a drawing. These are represented by the Model and layout tabs.

Typically, a model composed of geometric objects is created in a three-dimensional space called model space. A final layout of specific views and annotations of this model is created in a two-dimensional space called paper space. These spaces are accessible on two or more tabs near the bottom of the drawing area: the Model tab and one or more layout tabs.

Note. These tabs can be hidden, appearing instead as buttons on the status bar at the bottom-center of the application window.

Working on the Model tab, you draw a model of your subject at 1:1 scale. Working on a layout tab, you can create one or more layout viewports, dimensions, notes, and a title block to represent a drawing sheet.

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Методичні вказівки

до виконання практичних робіт з дисципліни

“КОМП’ЮТЕРНА ГРАФІКА”

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