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ИНЖЕНЕРНАЯ КОМПЬЮТЕРНАЯ ГРАФИКА В СРЕДЕ AUTOCAD. ПРАКТИКУМ

ENGINEERING COMPUTER GRAPHICS IN THE SPHERE OF AUTOCAD. PRACTICE

Рекомендовано УМО по образованию в области информатики и радиоэлектроники в качестве пособия для специальностей 1-39 02 03 «Медицинская электроника», 1-39 03 02 «Программируемые мобильные системы», 1-45 01 01 «Инфокоммуникационные технологии (по направлениям)», 1-58 01 01 «Инженерно-психологическое обеспечение информационных технологий», 1-98 01 02 «Защита информации в телекоммуникациях»

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Содержит материал для изучения интерфейса графической системы AutoCAD, описание технологии создания и редактирования чертежей, примеры практических заданий с пошаговым построением, индивидуальные задания.

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INTRODUCTION

Computer-aided design (CAD) is the design and drawing of drawings using a computer. Design is the creation of a real product from an idea. Drawing is the production of drawings that are used to document a project. CAD can be used to create 2D or 3D computer models.

AutoCAD is a two-and three-dimensional computer-aided design and drafting system developed by Autodesk. AutoCAD was first introduced in 1982. According to research, by 2000, there were more than 4 million AutoCAD users worldwide.

The proposed tutorial discusses methods and techniques for creating drawings in the AutoCAD 2020 graphical environment, such as: setting up a rational user working environment for performing graphical tasks; setting up the properties of drawing objects and changing them; drawing the simplest elements that make up the image of any complex mechanical drawing, editing images; rational sequence of graphic constructions.; creating text fragments. The presentation of the basics of the discipline «Engineering computer graphics» alternates with a step-by-step solution of problems of practical implementation of drawing elements.

Each section of the tutorial is based on the skills and information that were learned in the previous one. For better orientation, the exercises are shown in numbered steps.

This textbook is recommended as the main educational material for mastering the course of engineering computer graphics in the AutoCAD environment, for foreign students of technical areas of all forms of education, as it contains a sufficient number of options for tasks and exercises on each topic presented.

CHAPTER 1. STARTING AUTOCAD. EXPLORING THE INTERFACE

1.1. Starting AutoCAD

To Start AutoCAD, select the Start button → All Programs → Autodesk → AutoCAD 2020.

The AutoCAD Start tab appears, offering options to open existing drawings or to create a new drawing. Click the Start Drawing thumbnail in the left side of the view under «Get Started» (fig. 1.1).

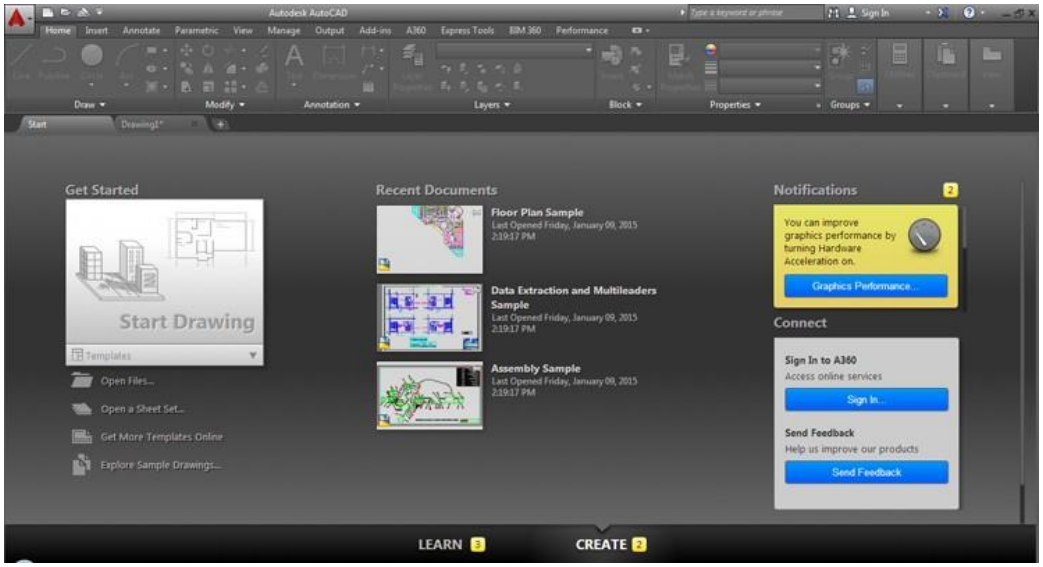


Fig. 1.1. Starting Window

Starts a drawing based on a drawing template file.

Template drawings store all the settings for a drawing and may also include predefined layers, dimension styles, and views.

Template drawings are distinguished from other drawing files by the .dwt file extension. They are normally kept in the template directory (fig. 1.2).

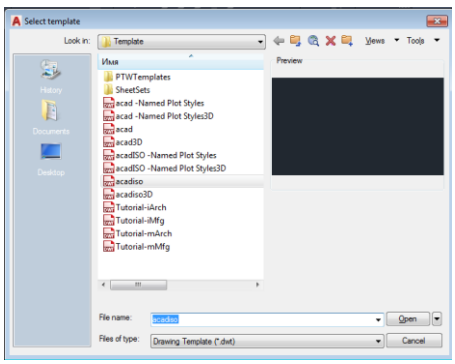


Fig. 1.2. The template directory

This tutorial does not cover the configuration of drawing parameters, but uses a ready-made template with all the settings for creating drawings.

1.2. The AutoCAD Window

The AutoCAD program window is divided into multiple parts:

- Application menu;
- Quick Access toolbar;
- Command window;
- Status bar;
- Ribbon;
- UCS icon (User Coordinate System icon);
- Drawing tabs;
- Drawing area;
- Viewport Controls;
- Navigation bar;
- ViewCube.

A typical AutoCAD window layout is shown in fig. 1.3. You can save and call up the workspace at any time using the Workspace Toggle tool in the Quick Access toolbar. The default workspace in fig. 1.3 is called the Drawing and Annotation Workspace, and is one of several workspaces built into AutoCAD.

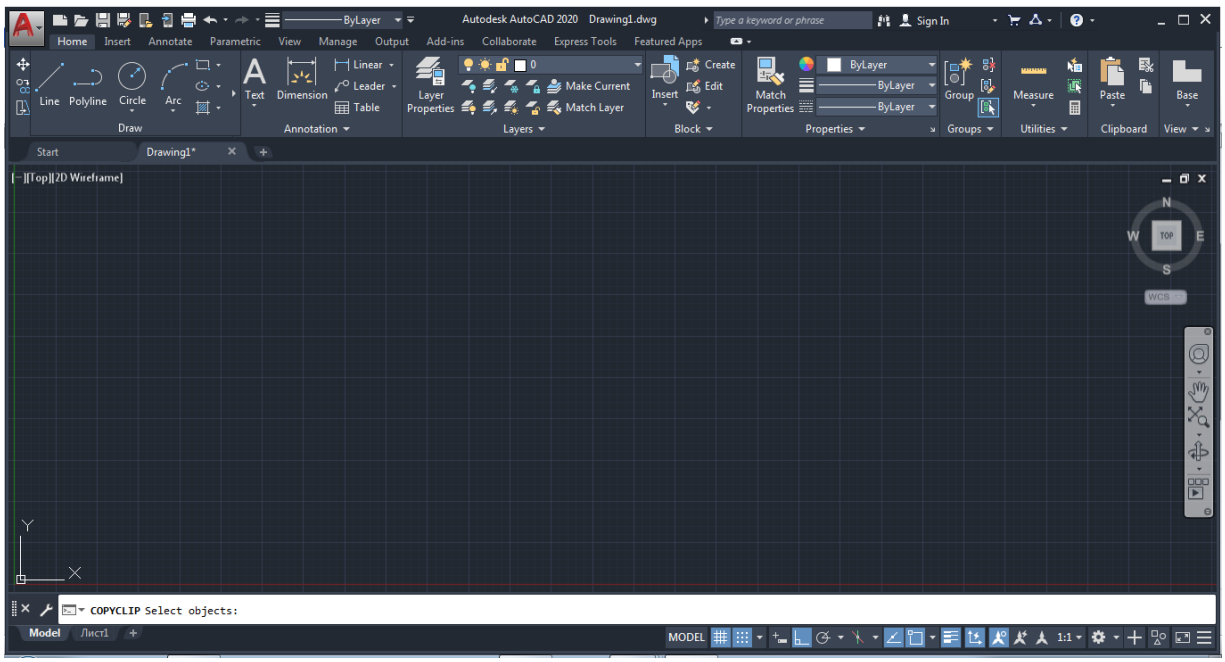


Fig. 1.3. Location of elements in the AutoCAD window

In the upper-left corner of the AutoCAD program window, the red AutoCAD icon displays the application menu, offering a set of options that are not related to creating drawings.

The Quick Access toolbar at the top of the drawing area includes the basic file-handling functions, which you find in nearly all Windows programs (fig. 1.4).

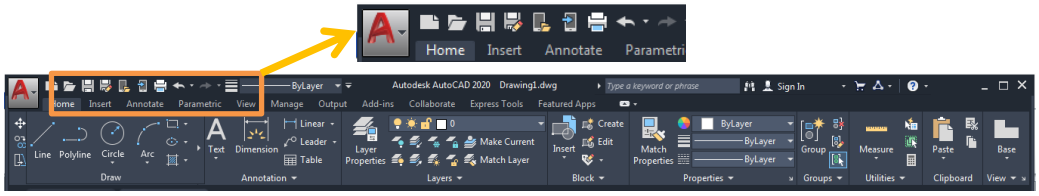


Fig. 1.4. The Quick access toolbar

The drawing area takes up most of the screen. Everything you draw is placed in this area. When you move the mouse, crosshairs appear within the drawing area. This is the drawing cursor, which allows you to point to locations in the drawing area.

Above the drawing area are the drawing tabs, which allow you to create new drawings or switch between open drawings in a similar way. Note the «x» on the current tab, which allows you to close the current drawing, and the «+» icon to the right of the tab, which allows you to create a new drawing or open an existing one. When you click on the «+» icon, a new drawing tab appears, which you can save as a new file. Right-click the «+» icon and a context menu will open, prompting you to open an existing drawing or create a new one as an additional tab.

The drawing area has several elements on the sides and in the corners. The UCS icon will appear in the lower-left corner. In the upper-right corner, you can see the view cube. ViewCube is designed primarily for 3D modeling.

The navigation bar is located along the right edge of the AutoCAD window. This panel offers tools that you can use to get around your drawing. Basic tools like zooming and panning can be found here, as well as some advanced tools for viewing 3D models.

Below the drawing area in the lower-left corner are the Model and Layout tabs. These tabs enable you to switch quickly between different types of views called the model and layout.

The Command window, located just below the drawing area, gives you feedback about the AutoCAD commands as you use them. By default, the Command window is in its docked position, as shown in fig. 1.5 [1].

Fig. 1.5. The Command widow

Status bar

Below the command window is the status bar (fig. 1.6). The status bar displays a summary of the drawing status and current settings. The tools in the status bar help you complete the drawing creation process.

Fig. 1.6. Status bar

These settings can be turned on and off by clicking on corresponding buttons (Snap, Grid, Ortho, etc.) or by pressing the function keys, F1, F2, etc. See the main button descriptions below.

 GRIDMODE (F7) – The grid (dots) is merely a visual «drawing aid». The default spacing is 1 unit.

 SNAPMODE (F9) – Increment Snap controls the move of the cursor. If it is off, the cursor will move smoothly. If it is ON, the cursor will jump in an incremental movement.

 ORTHO (F8) – When Ortho is ON, cursor movement is restricted to horizontal or vertical. When Ortho is OFF, the cursor moves freely.

 POLAR (F10) – Polar Tracking creates «Alignment Paths» at specified angles.

 OSNAP (F3) – Specific Object Snaps allow you to specify exact locations on objects when the corresponding queries for points within the team are displayed.

 OTRACK (F11) – Creates «Alignment Paths» at precise positions using object snap locations.

A feature that helps you visualize and specify coordinate vales angular values when drawing lines, arcs, circles, etc. DYN may display absolute Cartesian coordinates (X and Y values) or relative polar coordinates (distance and angle) depending on the current command prompt and the setting

Ribbon

The ribbon contains a set of panels that represent groups of tools for drawing, editing, or performing other functions.

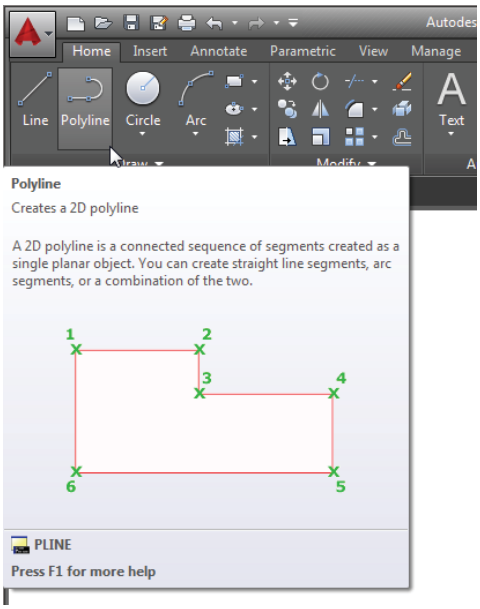


Fig. 1.7. The Ribbon panels

The name of each ribbon panel is located in the title bar at the bottom of the panel. The tabs that appear above the ribbon panels further organize them. Each tool on the ribbon contains a tooltip with a brief description of the tool icon (fig. 1.7).

Move the arrow cursor onto one of the Ribbon panel tools and leave it there for a moment; you'll see a tool tip appear just below the cursor. Hold the cursor there a bit longer and the tool tip changes to give you even more information about the tool.

Dynamic Input Display

AutoCAD performs all the actions that you assign to it. You interact with AutoCAD by using the tools and menu options that invoke the AutoCAD commands. A command is a monosyllabic instruction that you give to AutoCAD, telling it to do something, such as draw a Circle (the Circle tool in the Draw Ribbon panel) or move the object (the move tool in the Edit Ribbon panel). The entire dialog is displayed on the Command Line.

Every time you invoke a command, either by typing it or selecting a parameter or tool, AutoCAD responds by presenting you with messages on the command line and a dynamic input display, or by displaying a dialog window. The messages in the Command Line, or in the dynamic Input display, often tell you what to do next, or they may display a list of available options. A single command often presents a series of messages that you answer to complete the command.

In addition to messages, the command window records your activity in AutoCAD. You can use the scroll bar to the right of the command prompt window to view previous messages. You can also enlarge the window for a better view.

If you right-click to display a set of parameters in the context menu, you can see the set of parameters listed on the command line, as well as some additional parameters.

The dynamic input display allows you to enter the dimensions of objects while drawing them. In the command line, the dynamic input display shows all the dimensions, point coordinates, and angles of the objects you are drawing and editing (fig. 1.8).

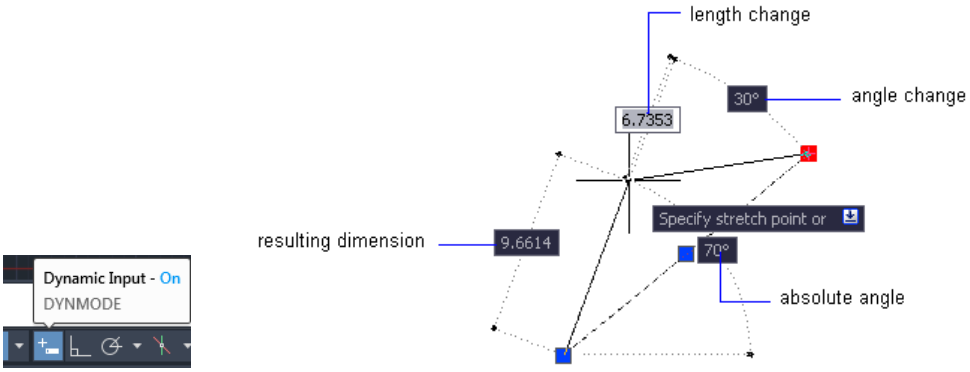


Fig. 1.8. The dynamic input display

When you enter coordinate values or angles using the keyboard, they are displayed on the dynamic input display. You can also enable or disable the display of dynamic input by clicking the Dynamic input tool in the status bar. When the dynamic input display is turned off, keyboard input is displayed only on the command line.

Expanding Hidden Panels

Besides to the visible tools, some buttons are hidden. You can expand many of the ribbon panels to select more. An arrow appears to the right of the panel title bar, which you can click to expand the panel (fig. 1.9).

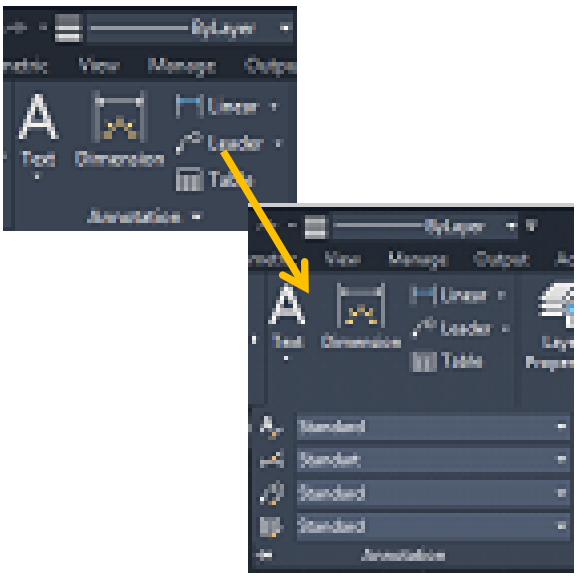


Fig. 1.9. Expanded panel

The location of the additional tools is called the extended panel.

If the screen has a lower resolution, then these ribbon panels on the right will only display their name (fig. 1.10).

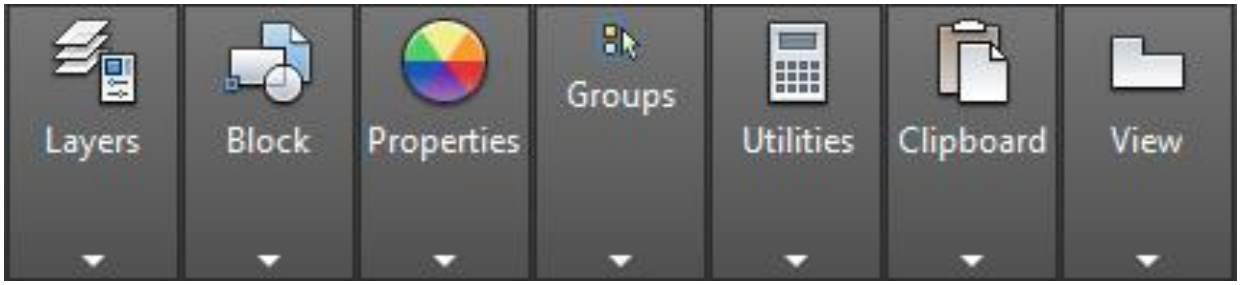


Fig. 1.10. Name of the ribbon panels

Hover your mouse over the panel to see the tools (fig. 1.11).

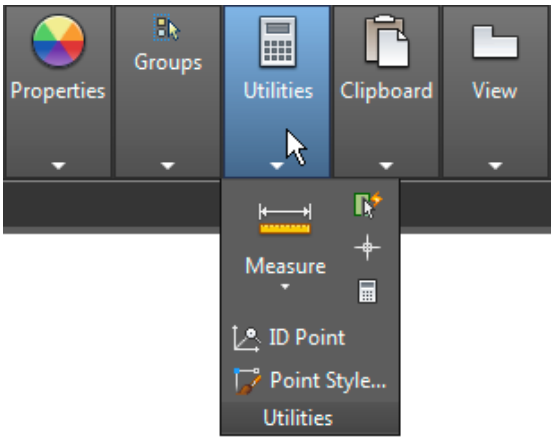


Fig. 1.11. Panel tools

Flyouts

Flyouts are similar to extended panels, however, unlike a whole panel, pop-ups give you access to different methods of using a particular tool. For example, AutoCAD allows you to draw circles in several ways, so it offers a flyouts for the Circle tool in the drawing panel of the Home tab. If you click on the arrow under the Circle icon in the Drawing panel, you will see additional tools for drawing circles (fig. 1.12).

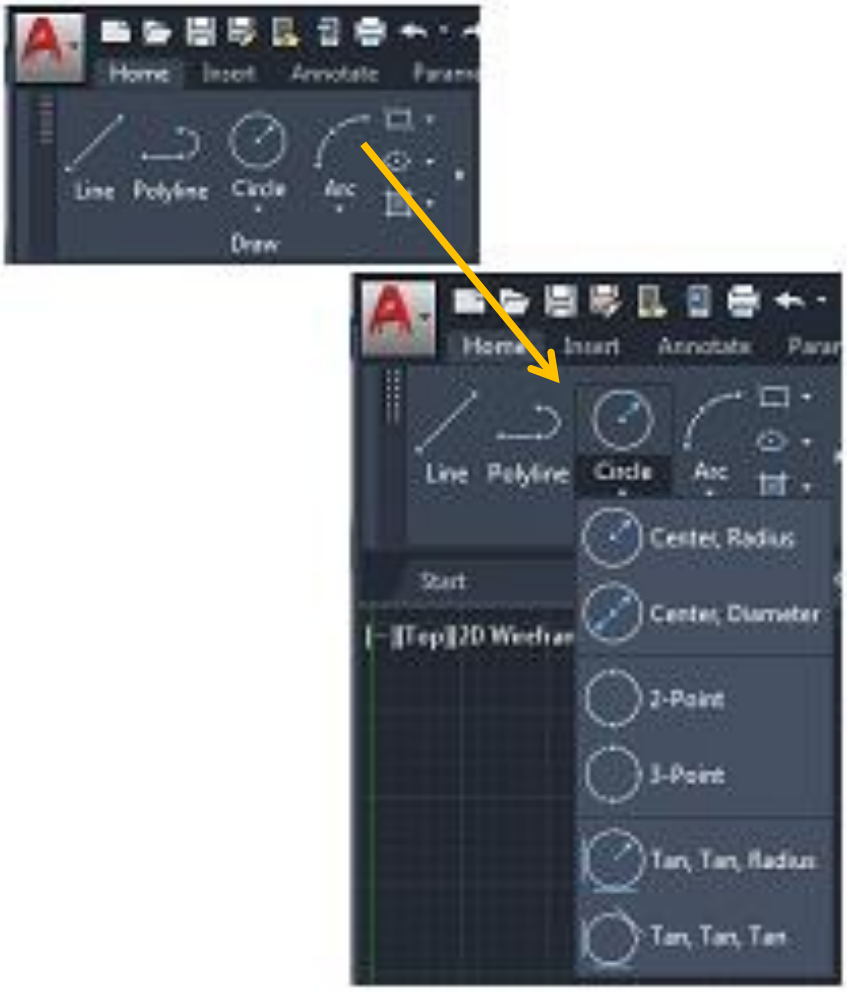


Fig. 1.12. Flyouts

If you select a tool option from, this option becomes the default tool for the selected icon. For example, if you hover over the Circle icon in the Drawing panel, the tool tip shows **Center, Radius** for the tool name. And if you click on the arrow under the **Center**, **Radius** tool and select **2-Point**, then **2-Point** will become the default tool, and you will see **2-Point** for the tool name in the tool tip [1].

Tools vs. the Keyboard

You can enter commands directly from the keyboard.

Many tools and commands have aliases. Aliases are one-, two-, or three-letter abbreviations of the command name.

Type the enter a letter _L on the Command Line and select a Command Line from the list or type the Line in press the enter key (fig. 1.13).

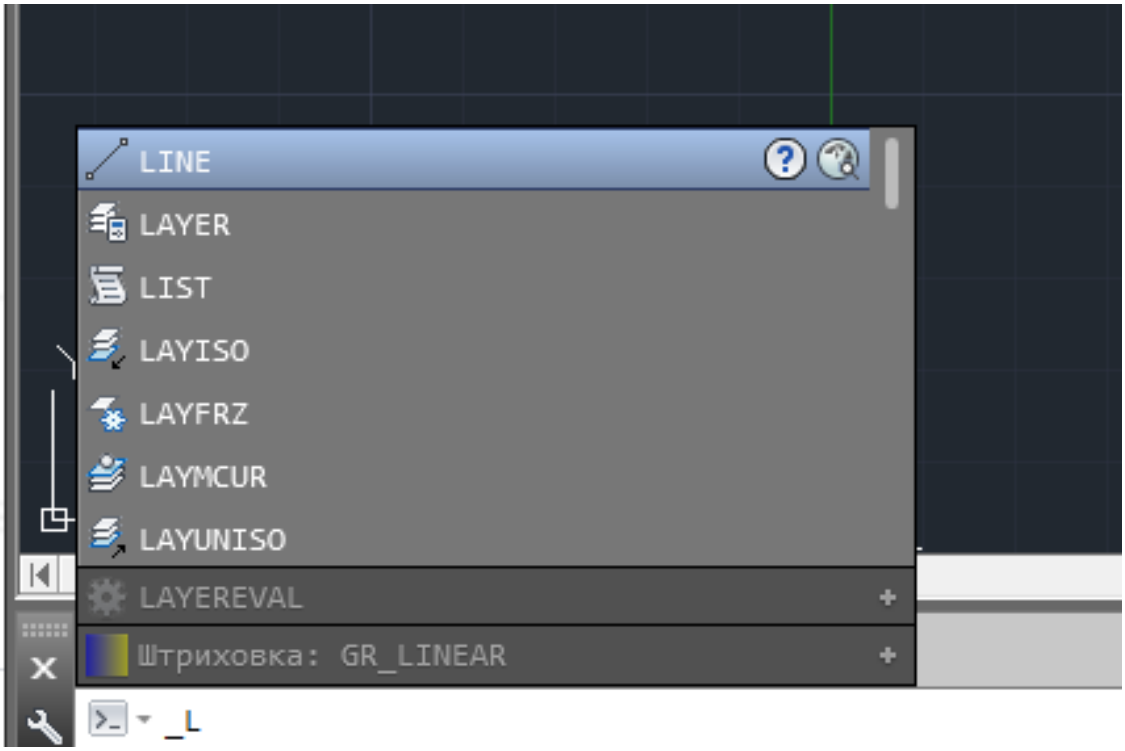


Fig. 1.13. The Command Line

The UCS Icon

In the lower-left corner of the drawing area, you see an **L**-shaped line.

This is the User Coordinate System (UCS) icon, which tells you your orientation in the drawing. This icon becomes helpful as you start to work with complex 2D drawings and 3D-models. The X and Y indicate the X- and Y-axes of your drawing. You can use it as a reference to tell you the direction of the axes. The UCS icon can be turned ON and OFF [1].

1.3. Mouse Options

The mouse functions are clearly shown in the fig. 1.14.

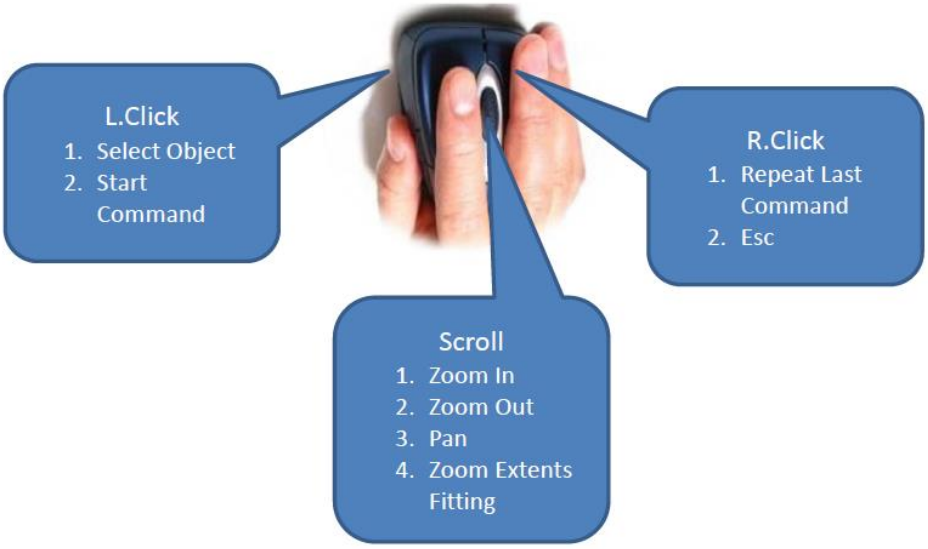


Fig. 1.14. The mouse functions

1.4. Using the Keys on the Keyboard

The fig. 1.15 shows the functions of the keys for entering, completing the repetition of commands in AutoCAD.

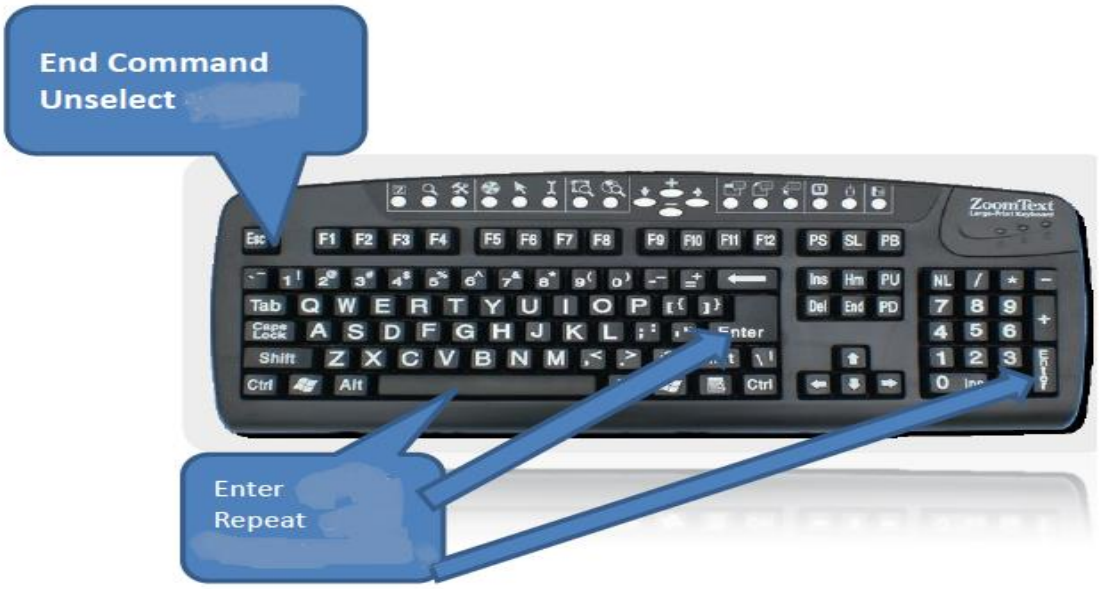


Fig. 1.15. The functions of the keys

CHAPTER 2. BASICS OF CREATING A DRAWING

2.1. Enter Commands

Let's create a sketch of a five-pointed star «by hand» using the **Line** command. This exercise will help you master the AutoCAD user interface (fig. 2.1).

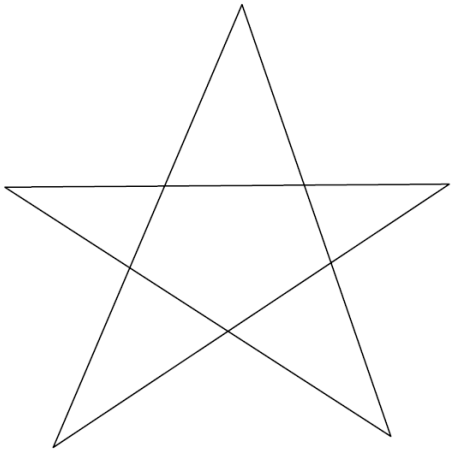


Fig. 2.1. The sketch of a five-pointed star

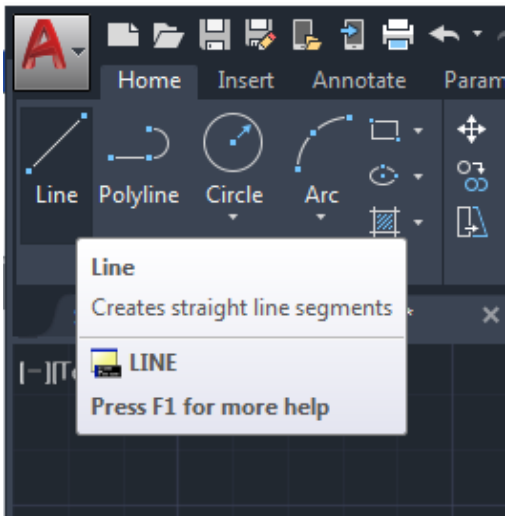


Fig. 2.2. The Draw panel

A **Line** can be one segment or a series of connected segments. Each segment is a individual object.

Move the graphics cursor to the **Line** icon in the **Draw** panel (fig. 2.2) (or type the enter a letter **_L** on the command line and select a **Line** from the list or type the **Line** in press the **Enter** key).

Select the icon by clicking once with the **left-mouse-button**, which will activate the **Line** command.

Lines are drawn by specifying the locations for the endpoints.

In the command prompt area the message: _line Specify first point: is displayed.

AutoCAD expects us to identify the starting location of a straight line. Move the graphics cursor inside the graphics window.

The graphic window can be viewed as a piece of paper, and the graphic cursor can be viewed as a pencil for drawing.

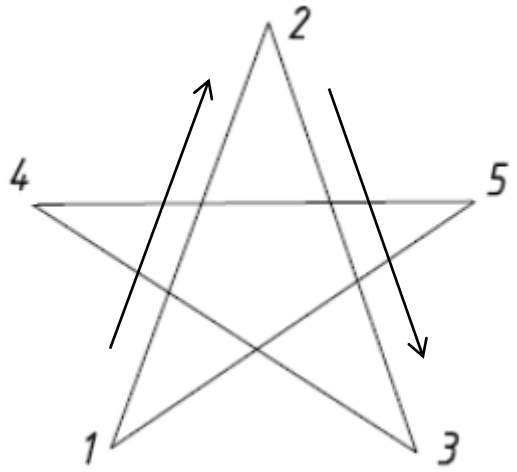


Fig. 2.3. The direction of the cursor movement

Move the cursor to the location of the «first» endpoint 1 then press the left mouse button. Move the cursor again to the «next» endpoint 2 and press the left mouse button (fig. 2.3).

Repeat the above steps and complete the freehand sketch by adding three more lines (from point 2 to point 3, point 3 to point 4, point 4 to point 5 and then connect to point 5 back to point 1).

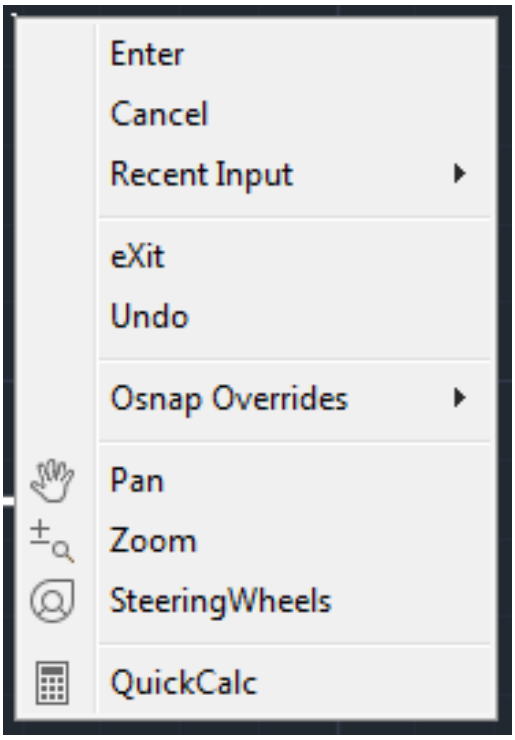


Fig. 2.4. The context menu

Notice that the **Line** command remains activated even after we connected the last segment of the line to the starting point (point 1) of our sketch. Inside the graphics window, click once with the right-mouse-button and a context menu appears on the screen (fig. 2.4).

Select «**Enter**» with the left-mouse-button to end the **Line** command. (This is equivalent to hitting the **Enter** end **Space** keys on the keyboard).

Repeat this exercise and use the other two ways to **Stop** drawing a line.

2.2. Erase Command

There are 2 methods to **erase** (delete) objects from the drawing. You decide which one you prefer to use.

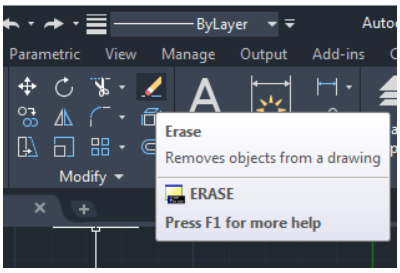


Fig. 2.5.The Icon Erase

Metod 1. Pick icon Erase in the Modify toolbar (fig. 2.5). The message «Select objects» is displayed in the command prompt area and AutoCAD awaits us to select the objects to erase.

Select any two lines on the screen; the selected lines are highlighted as shown in the fig. 2.6.

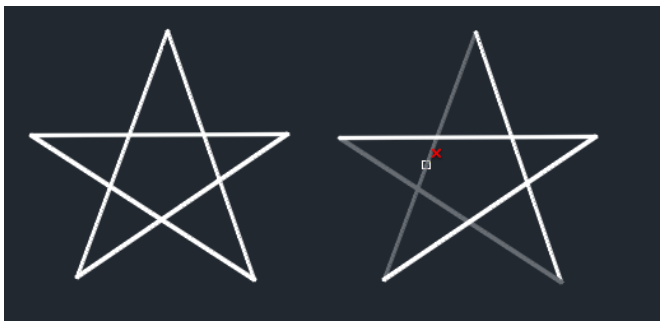


Fig. 2.6. Select lines

To deselect an object from the selection set, hold down the **Shift** key and select the object again.

To deselect an object from the selection set, hold down the **Shift** key and select the object again.

Right-mouse-click once to accept the selections (or press the **Enter** key). The selected two lines are erased.

Metod 2. First, select the objects, and then press the **Delete** key.

Repeat the last command

There are 3 methods to repeat the last command.

Metod 1. Inside the graphics window, click once with the right-mouse-button to bring up the context option menu.

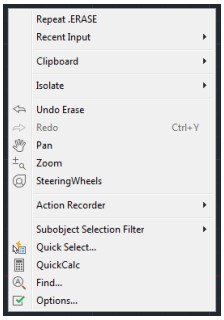


Fig. 2.7. The context menu

Pick **Repeat Erase**, with the left-mouse-button, in the popup menu to repeat the last command. Notice the other options available in the context menu (fig. 2.7).

Metod 2. Press the **Enter** key.

Metod 3. Press the **Space bar** key.

2.3. Select Object

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.

At the Select Objects prompt, you can select many objects at the same time.

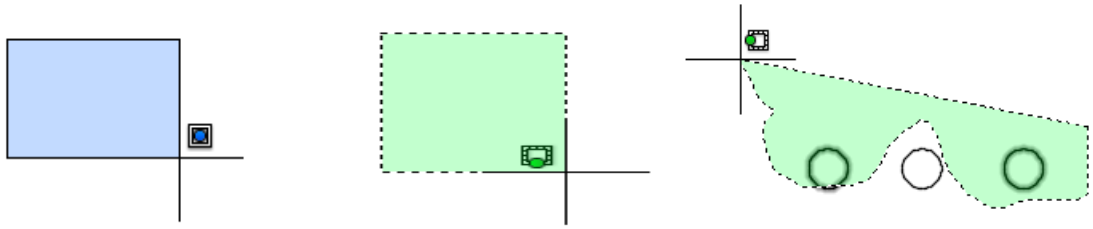
Specify a Rectangular Selection Area. Specify opposite corners to define a rectangular area. The background inside the area changes color and becomes transparent. The direction that you drag your cursor from the first point to the opposite corner determines which objects are selected.

Window selection. Drag your cursor from left to right to select only objects that are entirely enclosed by the rectangular area (fig. 2.8, *a*). With a window selection, usually the entire object must be contained in the rectangular selection area.

Crossing selection. Drag your cursor from right to left to select objects that the rectangular window encloses or crosses (fig. 2.8, *b*). If an object with a noncontinuous (dashed) linetype is only partially visible in the viewport and all the visible vectors of the linetype can be enclosed within the selection window, the entire object is selected.

To select objects using the lasso, click the mouse button, drag the cursor, and then and then release the button (fig. 2.8, *c*).

The fig. 2.8. shows different ways to select objects.



a *b* c

Fig. 2.8. Different ways to select objects

You can also remove objects from the current selection set by holding down **Shift** and selecting them again, or by holding down **Shift** and then clicking and dragging window or crossing selections. You can add and remove objects repeatedly from the selection set.

2.4. 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 **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.

Cancel a Command

You can cancel a command without completing it by pressing **Esc**.

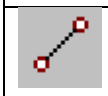
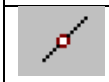
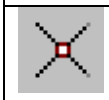
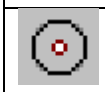
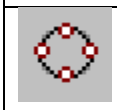
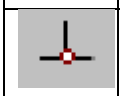
2.5. Object Snap

Object Snap enables you to snap to «objects» in very specific and accurate locations on the objects. For example: the end point of the line, the center of the circle, or the tangent line to the circle. Object Snap are made available during the execution of a command when you request AutoCAD to specify a point.

Tab. 2.1 shows the **Object Snap** options, that are most often used.

Table 2.1

Object Snap options

Icon	Short	Descript
	Endpoint	Snaps to the closest endpoint of a **Line**, **Arc** or **Polygon** segment. Place the cursor on the object close to the end
	Midpoint	Snaps to the middle of a **Line**, **Arc** or **Polygon** segment. Place the cursor anywhere on the object
	Intersection	Snaps to the intersections of any two objects. Place the Pick box directly on top of the intersection or select one object and then the other and AutoCAD will locate the intersection
	Center	Snaps to the center of an **Arc**, **Circle** or **Donut**. Place the cursor on the object, or the approximate center location
	Quadrant	Snaps to a 12:00, 3:00, 6:00 or 9:00 o'clock location on a circle. Place the cursor on the circle near the desired quadrant location
	Perpendicular	Snaps to a point perpendicular to the object selected. Place the cursor anywhere on the object
	Parallel	Constrains a new line segment, polyline segment, ray or line to be parallel to an existing linear object that you identify by hovering your cursor
	Tangent	Calculates the tangent point of an Arc or Circle. Place the cursor on the object as near as possible to the expected tangent point
	Nearest	Nearest is the snap option that works of this simple principle, you want your cursor to snap to the nearest point of the object to your cursor

Specify an Object Snap

To specify an Object Snap at a request for a point, you can use one of the following methods:

- press **Shift** and right-click to display the Object Snap shortcut menu;
- click an **Object Snap** button on the status bar (fig. 2.9).

Fig. 2.9. **Object Snap** button

After activating the binding sign, the bindings window appears, in which the necessary bindings are indicated with labels (fig. 2.10).

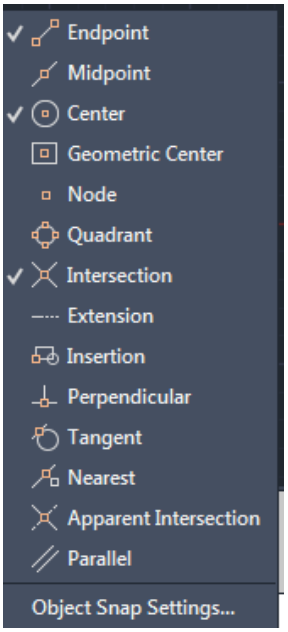


Fig. 2.10. Window Object Snap

Below is suggested abbreviated list of Object Snap modes required to execute a training drawing.

In the fig. 2.11, several different types of Object Snap are represented as markers [6].

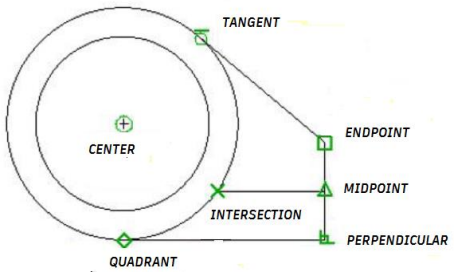


Fig. 2.11 Markers of Object Snaps

CHAPTER 3. CREATING IMAGES

The following exercises will help you master the main drawing commands.

3.1. Line Command

Exercise 1. Draw a rectangle by setting the points in absolute coordinates (fig. 3.1).

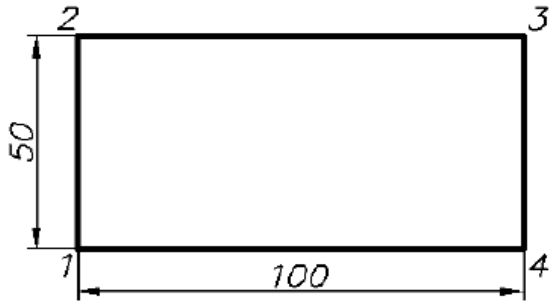


Fig. 3.1. Initial data for the Exercise 1

Command: LINE _line Specify first point: enter the coordinates of the first point 50,50 **Enter**.

The following prompts on the Command Line.

Next point or [Cancel] (Specify next point or [Undo]): 50,100 **Enter**.

Next point or [Cancel] (Specify next point or [Undo]): enter the coordinates of the next point 150,100 **Enter**.

Next point or [Close/ Undo] (Specify next point or [Close/ Undo]): enter the coordinates of the next point 150,50 **Enter**.

Next point or [Close/ Undo] (Specify next point or [Close/ Undo]): enter the Close point option **Enter**.

Exercise 2. Draw a triangle by setting the vertices in relative coordinates (fig. 3.2).

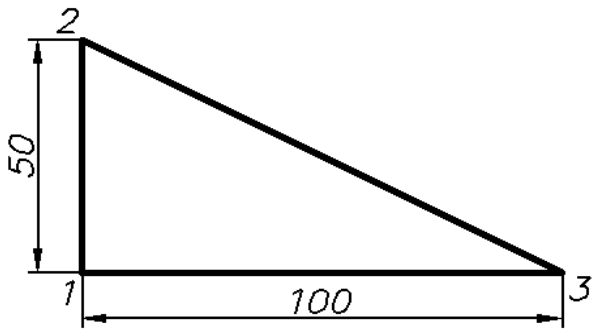


Fig. 3.2. Initial data for the Exercise 2

The system requests are similar to the ones listed above.

Command: LINE _line Specify first point: 50,120

Next point or [Cancel]: @0.50

Next point or [Cancel]: @100, -50

Next point or [Close/ Cancel]: C

Exercise 3. Draw an equilateral triangle by setting the vertices in relative polar coordinates (fig. 3.3).

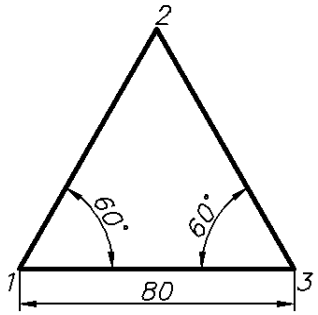


Fig. 3.3. Initial data for the Exercise 3

Command: LINE _line Specify first point: 180,50

Next point or [Cancel]: @80 < 60

Next point or [Cancel]: @80 < - 60

Next point or [Close/ Cancel]: C

Exercise 4. By yourself draw the outline shown in fig. 3.4.

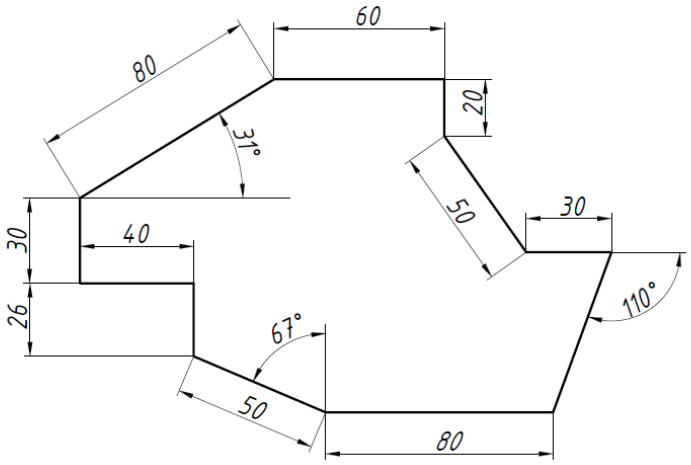


Fig. 3.4. Initial data for the Exercise 4

The student chooses the method of specifying coordinates independently, depending on the convenience of choosing in a particular situation [3].

3.2. Circle Command

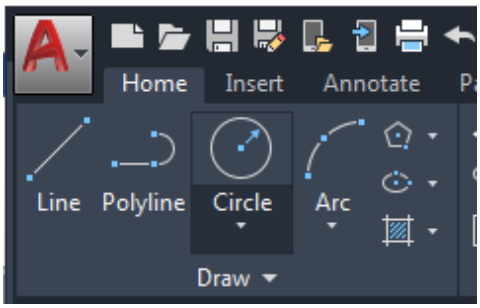


Fig. 3.5. The Circle Icon

In the **Draw** toolbar, click on the little triangle below the Circle Icon (fig. 3.5). Note that the little triangle indicates additional options are available (fig. 3.6).

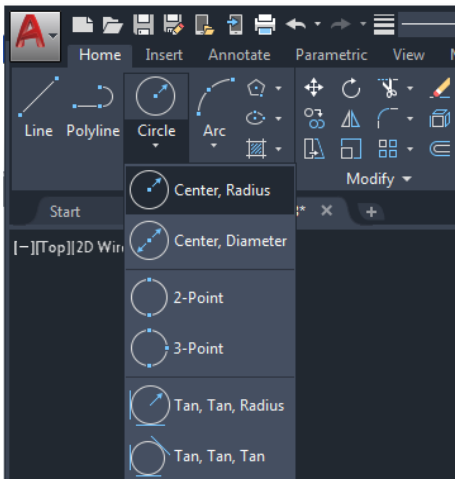


Fig. 3.6. The option list

Notice the different options available under the circle submenu. There are six ways to build a circle in AutoCAD:

1. **Center, Radius**: Draws a circle based on a center point and a radius.
2. **Center, Diameter**: Draws a circle based on a center point and a diameter.
3. **2 Points**: Draws a circle based on two endpoints of the diameter.
4. **3 Points**: Draws a circle based on three points on the circumference.
5. **TTR–Tangent, Tangent, Radius**: Draws a circle with a specified radius tangent to two objects.
6. **TTT–Tangent, Tangent, Tangent**: Draws a circle tangent to three objects.

After selecting the appropriate option, in the command prompt area, the message:

Specify center point for circle or[3P/2P/Ttr (tan tan radius)]: is displayed. AutoCAD expects us to identify the location of a point or enter an option (fig. 3.7).

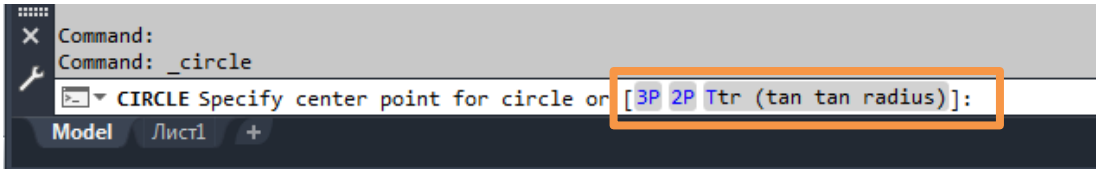


Fig. 3.7. Command options

After you specify the center of the circle with the cursor in the drawing area, enter the circle diameter parameter.

In the command prompt area, the message:

Specify diameter of circle: 20 **Enter** (as an example).

Exercise 5. Draw 6 circles in different ways, as shown in the fig. 3.8.

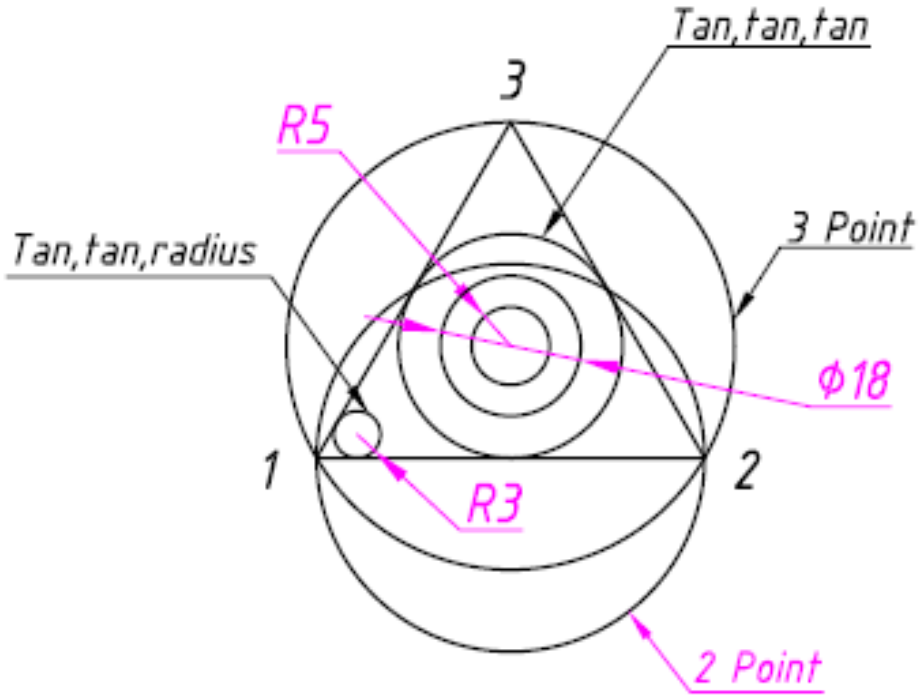


Fig. 3.8. Initial data for the Exercise 5

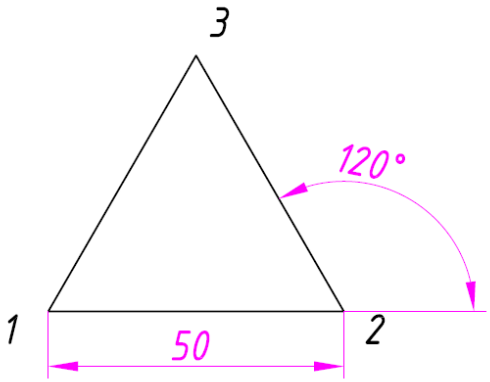


Fig. 3.9. The triangle

At the beginning you need to draw a triangle (fig. 3.9).

The construction of a triangle was considered earlier (see exercise 3).

This exercise will use Object Snaps (Object Snaps were discussed in section 2.5).

The sequence of drawing this drawing is suggested below (fig. 3.10).

The figure shows the sequence of building circles.

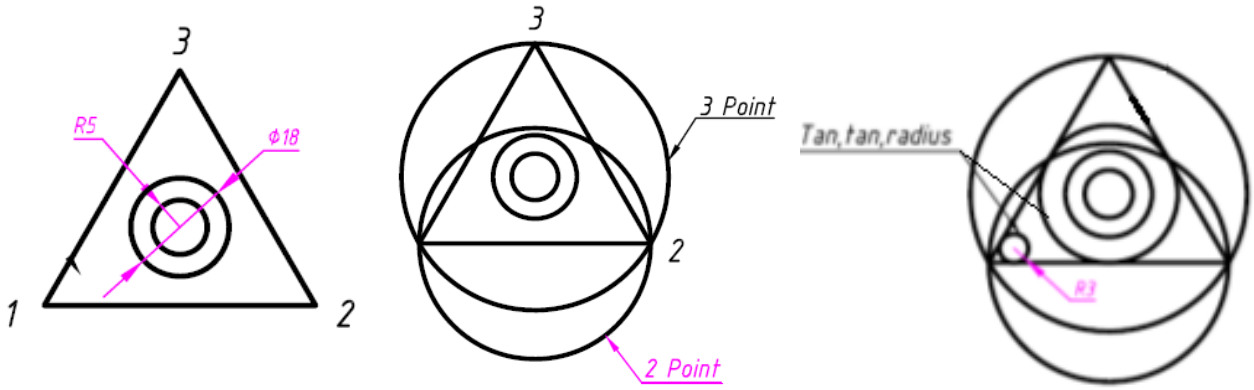


Fig. 3.10. The sequence of building circles

3.3. Creating and Assigning Layers

The layers resemble transparent sheets of tracing paper lying on top of each other (fig. 3.11). They are the main means of organizing objects in the drawing.

Layers allow you to group objects of the same type. For example, you can place objects such as auxiliary lines, texts, dimensions, and main labels on separate layers.

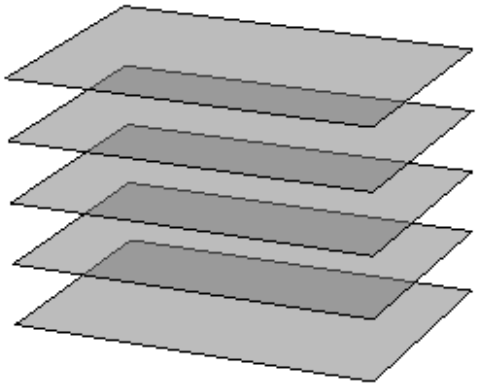


Fig. 3.11. Layers

Layers allow you to structure the drawing, which makes it easier to manage drawing data and various properties, such as line types, colors, etc.

AutoCAD is configured to work on two layers: layer 0 (system) and the Defpoints layer. In this case, the drawing is made with a thin line on layer 0, and then the thickness and type of lines are changed as necessary. The Defpoints layer is used for auxiliary constructions that will not be reflected on the paper when the drawing is printed.

When working with layers, you first need to analyze how many layers you need to create and what components of the drawing will be executed on them.

For each associated group of drawing elements, you can create a new layer, name it, and assign specific properties to each layer, such as color, thickness, and line type.

Layers can be **Disabled** or **Frozen**. In this case, the graphic information in these layers becomes invisible. Layers can be **Locked**. Locking allows you to create objects, but you can't edit these objects [7].

The number of layers that can be created in the drawing, and the number of objects that can be created on each of the layers, is unlimited.

Layers can be deleted, except for system layer 0.

There are two layers in the template used.

First layer – «Layer 0» layer contains lines with thickness 0,6 mm. These lines uses for drawing of all visible outlines (edges) of detail. In this layer contains the frame of engineering drawing and a table in right lower corner of drawing, this table named Basic Inscription. Second layer – Defpoints.

To perform the following practical tasks, you need to create the following layers:

1. **Layer 1** will be used to draw the center lines (a thin dash-dotted line).

Layer name – **Axis**, line type – Dash-dotted line, line thickness – Default, line color – red. The default line thickness in AutoCAD is 0,25 mm.

2. **Layer 2** will be used for hatch execution (a thin continuous line).

Layer name – **Hatch**, line type – Continuous, line thickness – Default, line color – turquoise.

3. **Layer 3** will be used to apply the dimensions (a thin continuous line).

Layer name – **Dimensions**, line type – Continuous, line thickness – Default, line color – green.

4. **Layer 4** will be used to execute the text.

Layer name – **Text**, line type – Continuous, line thickness – Default, line color – purple.

To display the Layer Properties Manager, click the Layer Properties tool in the Home tab's Layers panel, or type **LA**↵ to use the keyboard shortcut (fig. 3.12).

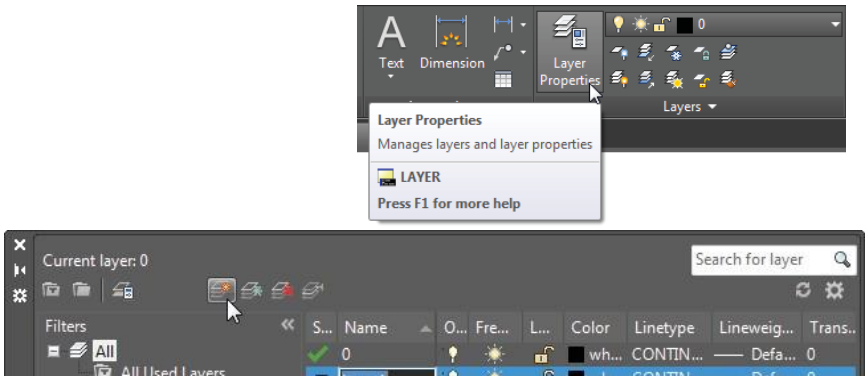


Fig. 3.12. Display the Layer Properties Manager

New windows of Layer Properties Manager will appear.

Click the New Layer button at the top of the palette. The button has an icon that looks like a star next to a sheet (fig. 3.13).

Fig. 3.13. The New Layer button

You have added a new layer in drawing. Then we will change the default properties of layer. First property – Name of layer. Click twice on name «Layer 1» with a small delay between clicks. Change name on «Axis».

With the «Axis» layer name highlighted, click the Color icon in the «Axis» layer listing to dis- play a dialog box in which you can assign a color to the Axis layer. You will find the Color icon under the Color column; it currently shows White as its value.

The icon is just to the left of the word white, which doesn't appear in its entirety in the following image.

Then change a Color of layer. Left click mouse button on «White» and in window «Select Color» select Color Red and click OK (fig. 3.14).

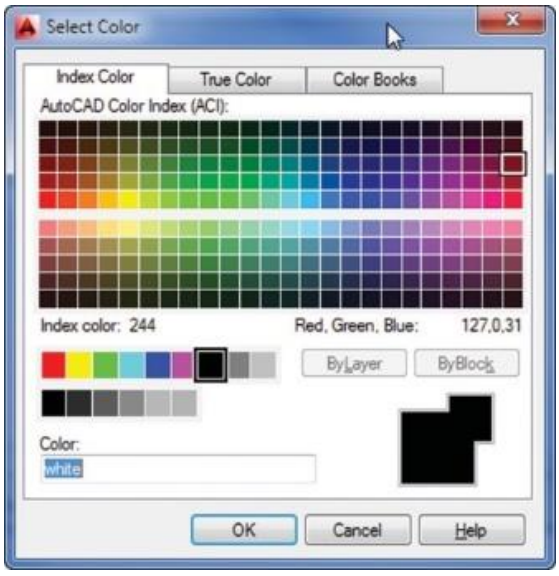


Fig. 3.14. The select Color dialog box

To change **Linetype** click on «Continues».

In window «Select Linetype» click button Load (fig. 3.15).

Fig. 3.15. Change Linetype

The last step in changing of layer properties is to change **Lineweight** (fig. 3.16). In window «Lineweight» select demanded Lineweight equal 0,25 mm (fig. 3.17) and click OK.

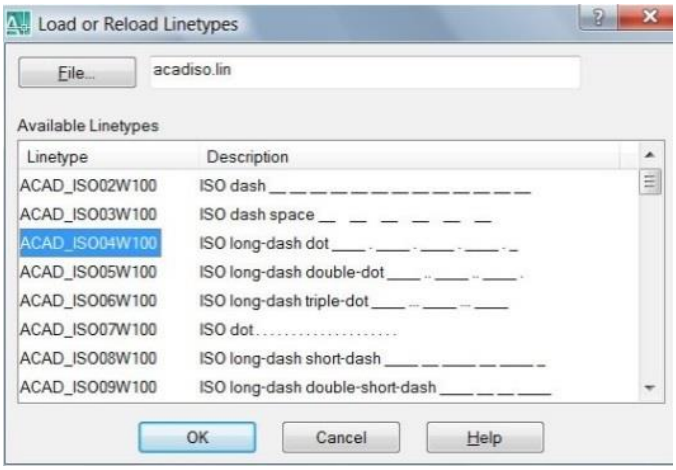


Fig. 3.16. Change Lineweight

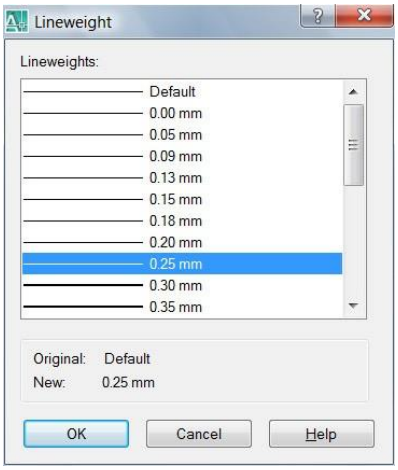


Fig. 3.17. Window «Lineweight

For the remaining layers, you must perform the same steps, taking into account the requirements set out above.

Compare a result of all changes with figure 3.18. Click buttons Apply then OK.

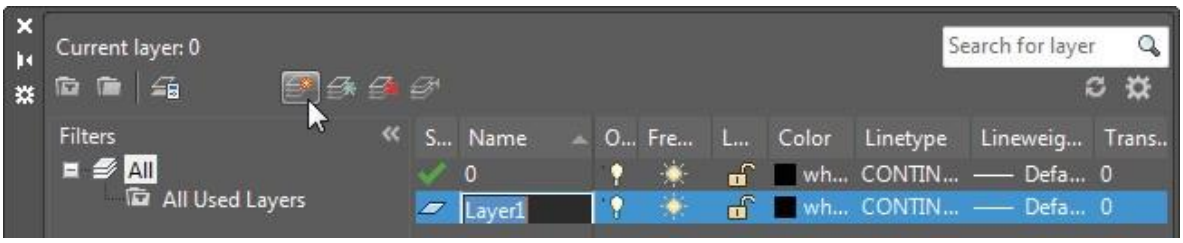


Fig. 3.18. Compare a result

CHAPTER 4. EDITING THE AUTOCAD ENTITIES

4.1. Modify Commands

Editing is changing an object that has already been created. Modify commands are called from the toolbar menu using the icon (fig. 4.1).

The principle of using Modify commands is the same as Draw commands. You need to constantly follow the requests on the command line.

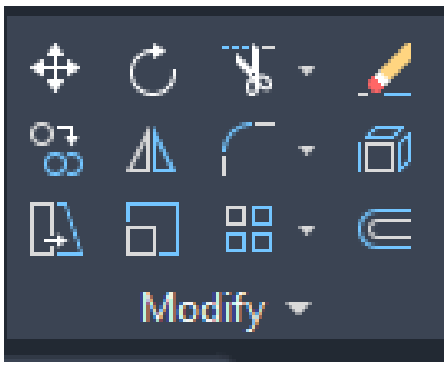


Fig. 4.1. Modify panel

This tutorial covers the basic editing commands of the Modify panel.

Move

The command allows you to move one or more selected objects to a new position without changing their orientation or size (fig. 4.2).

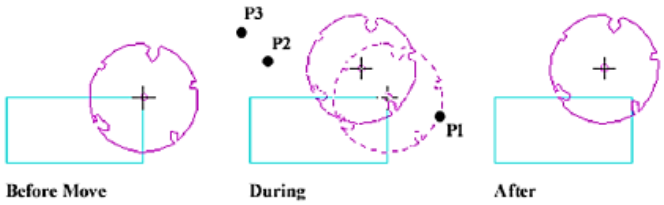


Fig. 4.2. Using the Move Command

Rotate

The command is used to rotate the selected objects relative to the base point by a specified angle.

Command execution algorithm: select objects, press the key **Enter** to confirm the end of the selection of objects, specify the base point (the point that remains stationary during rotation), enter the angle of rotation of the object in degrees.

Copy

This command allows you to create multiple copies of objects in a single call.

Command options:

1) Displacement – allows you to specify the displacement (the coordinates of the point that will serve as the values of the movement of the selected objects along the X and Y axes);

2) Mode – sets the copying modes: Multiple (Multiple) – multiple copying; Single (Single) – one-time copying.

The default query sequence is: select objects, press the **Enter** key to confirm that the selection of objects is complete, specify the base point (the point relative to which the group of selected objects is viewed), and specify the new position of the base point along with the group of selected objects (fig. 4.3).

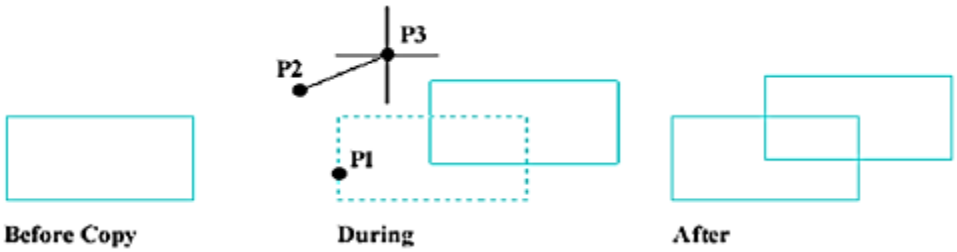


Fig. 4.3. Using the Copy Command

Mirror

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.

Offset

The command provides the construction of a primitive similar to an existing primitive at a given offset or passing through a given point while preserving the orientation of the original.

The algorithm for executing the command: set the offset value, select an object for the offset, specify a point (with the mouse) that determines the side of the offset (fig. 4.4).

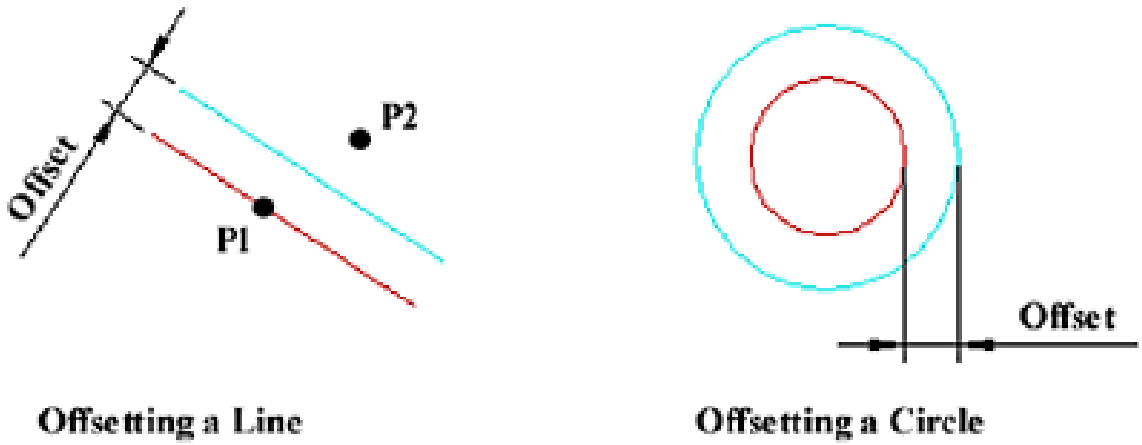


Fig. 4.4. Using the Offset Command

4.2. Grips

If you click on an object without calling editing commands, small blue squares (Grips) appear at the characteristic points of the object, for example, at the ends and in the middle of the segment (fig. 4.5).

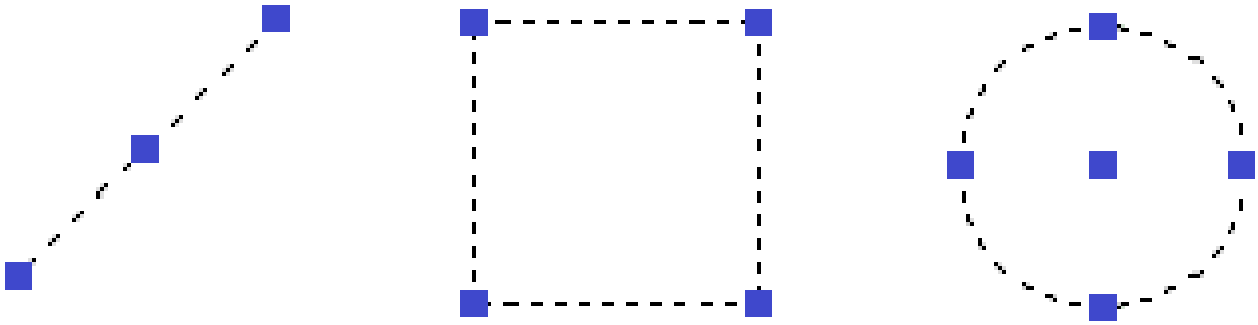


Fig. 4.5. Grips display

Grips can be dragged using the mouse in order to stretch, move, mirror, rotate or scale entities.

These Grips help you quick edit the drawing in the absence of a command.

4.3. Dimensions on Drawing

AutoCAD dimensioning is semi-automatic, when you invoke a command to create a linear dimension, AutoCAD only requires that you Pick an object or specify the extension line origins, and pick the location of the dimension line. AutoCAD then measures the feature and draws the extension line, arrow heads and dimension text.

Menu commands in the control panel **Dimensions** (fig. 4.6).

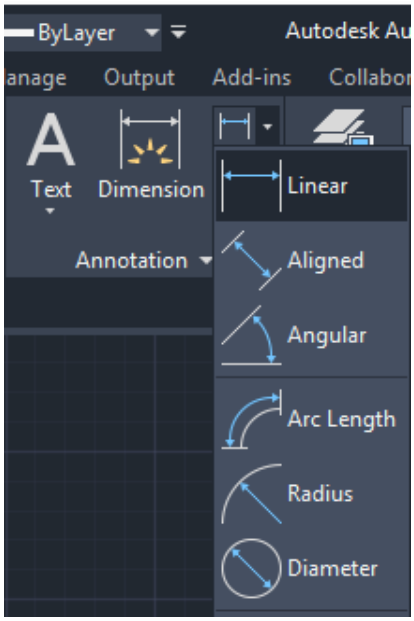


Fig. 4.6. The control panel Dimensions

 **Linear**

The command provides a linear dimension setting.

Command options: Mtext – allows you to edit multiline multi-dimensional text; Text (Text) – allows you to edit the dimensional text; Angle (Angel) – allows you to tilt the dimensional text; Horizontal – defines the orientation of the dimension line as horizontal; Vertical – defines the orientation of the dimension line as vertical; Rotated – allows you to set the angle of inclination of the extension lines.

 **Angular**

The command allows you to set the angular dimension between segments, such as the center corners of arcs, arc segments of polylines, or parts of a circle.

 **Radius**

The command sets the radius of a circle or arc.

Command execution algorithm: call the command to execute, specify an arc or circle, edit the text if necessary, and fix the position of the dimension line.

 **Diameter**

The command provides the setting of the diameter of the circle or arc.

4.4. Text on Drawing

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-linetext. 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.

4.5. Basic Inscription of the Drawing

Basic Inscription it’s a table in the right lower corner of the drawing. Basic Inscription should be filled with text always.

Select Text layer in layers list. Fill table cells as shown on fig. 4.7 [4].

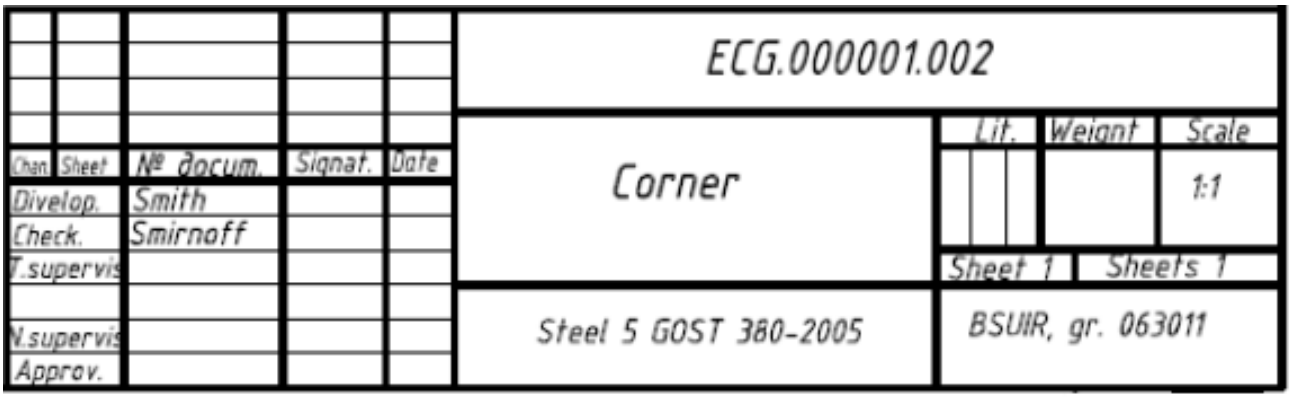


Fig. 4.7. Basic Inscription

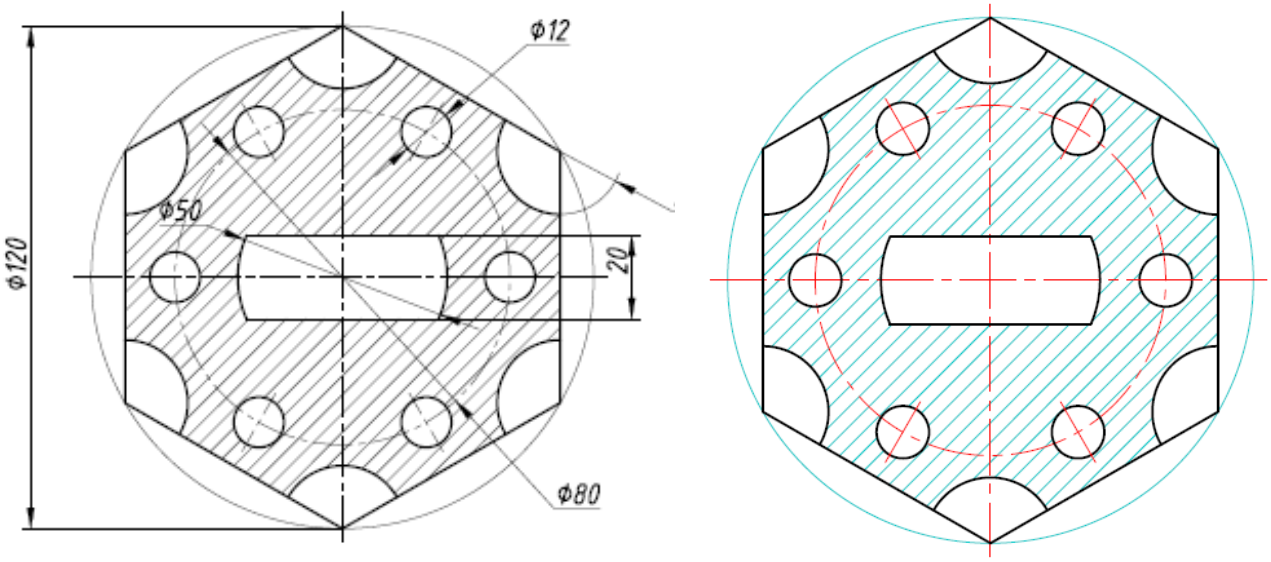
The following font height settings are offered:

1. Smith – developer's surname of drawing (write your own surname here). Height of text 3,5 mm.
2. Smirnoff – surname who has checked up drawing (surname of professor). Height of text 3,5 mm.
3. DGEG 000001.001 – designation of drawing. Height of text 7 mm.
4. Corner – detail's name. Height of text 7 mm.
5. Steel 5 GOST 1050-88 – it's designation of detail's material. Height of text 5 mm.
6. 1:1 – scale of images. Height of text 5 mm.
7. Sheets 1 – quantity of paper's sheets of drawing. Height of text 3,5 mm.
8. BSUIR, gr. 963011 – name of University and number of student group. Height of text 5 mm.

Practical Work 1

Draw the complex flat contour as shown in the fig. 4.8, *a*.

When creating this drawing, use the following Draw commands: Line, Polygon, Circle, Hetch and Modify Commands: Rotate, Trim, Aray, Mirror.



a *b*

Fig. 4.8. The original drawing for Practical work 1:
a –initial data; *b* – using layers

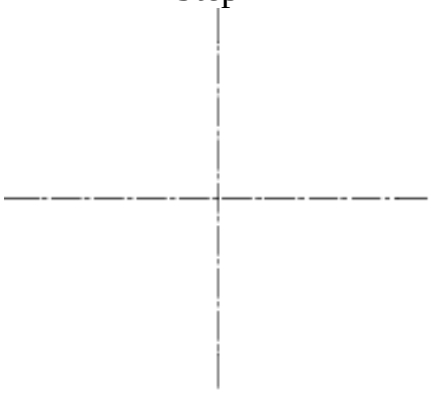
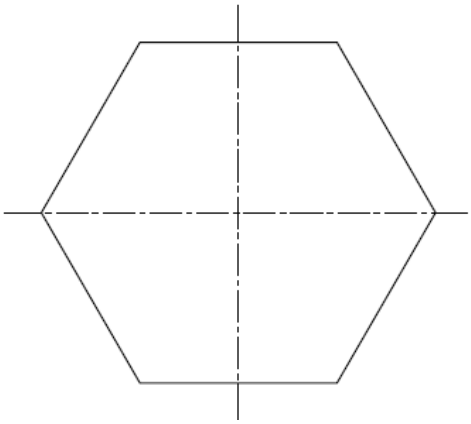
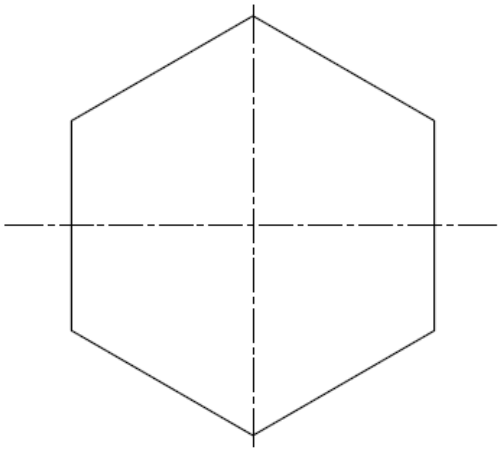
This task is performed using layers (see paragraph 3.3) on the template.

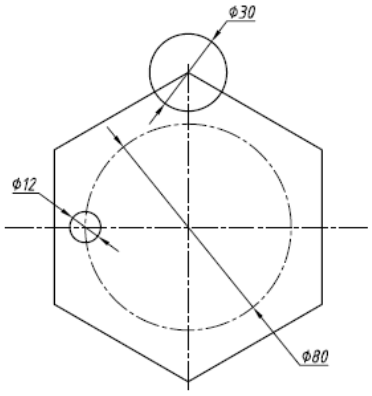
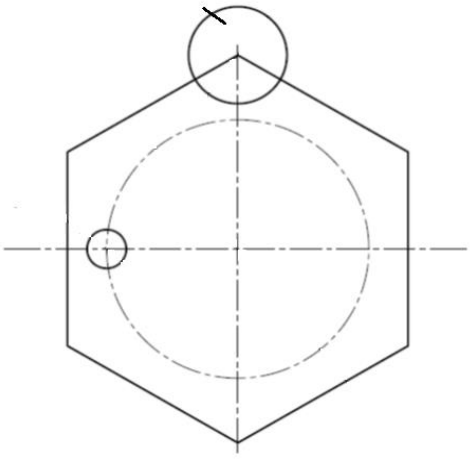
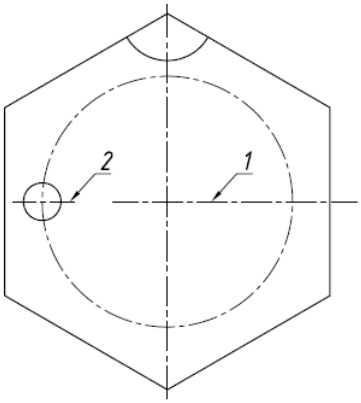
Ask the teacher for the «A4 template ENGLISH.dwt» template file.

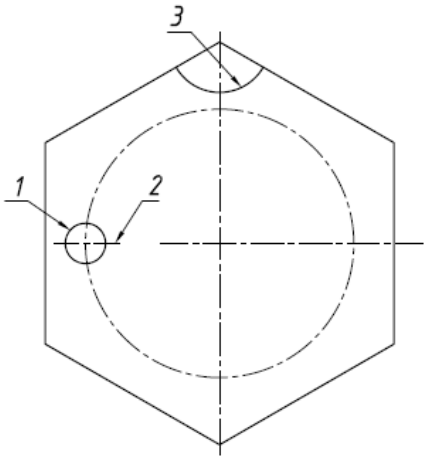
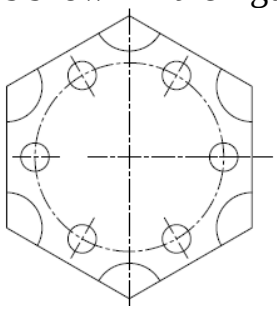
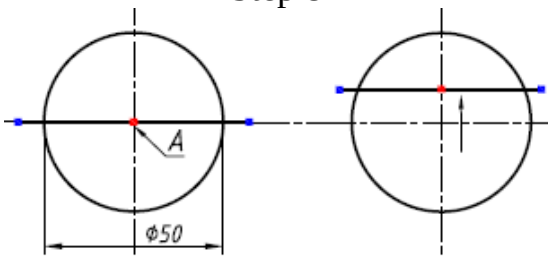
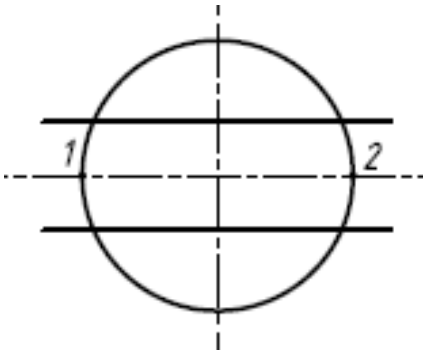
The tab. 4.1 shows the step-by-step construction of this drawing.

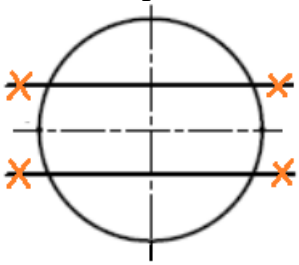
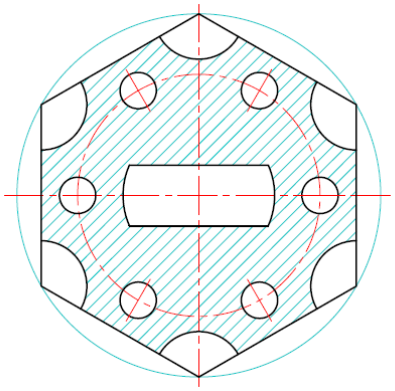
Table 4.1

The step-by-step construction of Practical Work 1

Drawing construction steps	Description of drawing construction
1	2
Step 1 	1. Set the current layer to Axial. 2. Use the **Line** Command to draw horizontal and vertical lines (at random), Ortho ON
Step 2 	1. Set the current layer to **Contour**. 2.  Find the **Polygon** icon in the **Draw** panel and the following prompts on the Command Line: _polygon Enter number of sides <4>: 6 **Enter**; Specify center of polygon or [Edge]: set the location of the center of the polygon at the intersection of the axes; Enter an option [Inscribed in circle/Circumscribed about circle] <I>: your choice – **Inscribed** in circle; Specify radius of circle: 60 **Enter**
Step 3 	 Find the **Rotate** icon in the **Modify** panel and the following prompts on the command line: Select objects: hexagon **Enter**; Specify base point: the intersection point of the axes; Specify rotation angle or [Reference]: 90 **Enter**

1	2
Step 4 	1. Set the current layer to **Axial** and draw a circle with a diameter of 80 mm, the center of the circle is the intersection of the axes (use Object Snap – Intersection). 2. Set the current layer to **Contour** and draw a circle with a diameter of 12 mm. The location the center of the circle at the intersection of the center lines. 3. Draw a circle with a diameter of 30 mm. The location the center of the at the top of the hexagon
Step 5 	Use **Trim** Command.  Find the **Trim** icon in the **Modify** panel. When using the **Trim** command to delete a part of a line, after selecting the command, immediately press the space bar, and then select the object. According to the drawing, you need to remove a part of the circle. Use the cursor to indicate the part of the line that you want to delete. Pressing the **Enter** key completes the process
Step 6 	1. Select line **1** and hover over the left Grip at the end of it and make the line shorter as shown in the figure. 2. Set the current layer to **Axial** and draw horizontal short line **2** (Object Snap – **Center**)

1	2
Step 7 	Use **Array** Command.  Find the **Array** icon in the **Modify** panel. Select the option – **POLAR** array – evenly distributes copies of the object in a circular pattern around a center point. The following prompts on the Command Line: Select objects: Specifies the object to be arrayed: cursor selection circle 1, line 2 and arc 3. After completing the requests, you should get the image as shown in the figure 
Step 8 	Performing the central element: 1. Draw a circle with a diameter of 50 mm. 2. Draw a horizontal line over the centerline. 3. Select the built line and use the central Grip for a parallel upward shift of 10 mm
Step 9 	Use **Mirror** Command.  Find the **Mirror** icon in the **Modify** panel. The following prompts on the Command Line: Select objects: the horizontal axial line **Enter**; Specify first point of mirror line: select point 1; Specify second point of mirror line: point 2; Erase source objects? [Yes/No] <No>: **Enter**. After completing the requests, you should get the image as shown in the figure

1	2
Step 10 	Use **Trim** Command to trim part of the lines as shown in the figure. Using the command was discussed earlier, step 4
Step 11 	Use **Hatch** Command.  Find the **Hatch** icon in the **Draw** panel Set the current layer to **Hatch**. In the select hatch pattern panel ANSI31. This pattern is used for hatching parts made of metal. The following prompts on the Command Line: Pick internal point or [Select objects/Undo/seTtings]: move the cursor one by one and select the areas to hatch. Pressing the **Enter** key completes the process

And in conclusion, you need to add the dimensions (see fig. 4.8, *a*).

Fig. 4.9 show the exercises that need to be performed using the **Draw** and **Modify** commands [5].

At the beginning, you need to make an analysis of draws and determine which drawing and editing commands will be used when performing the tasks.

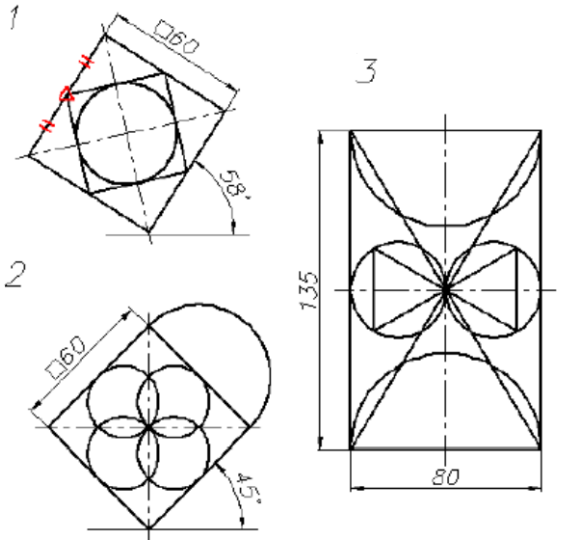


Fig. 4.9. Exercises for mastering panels Draw and Modify

It is important to perform the construction more efficiently.

Practical Work 2

Draw the curved contour as shown in the fig. 4.10.

Use the **Modify** commands similarity **Offset**, **Trim** and **Fillet**.

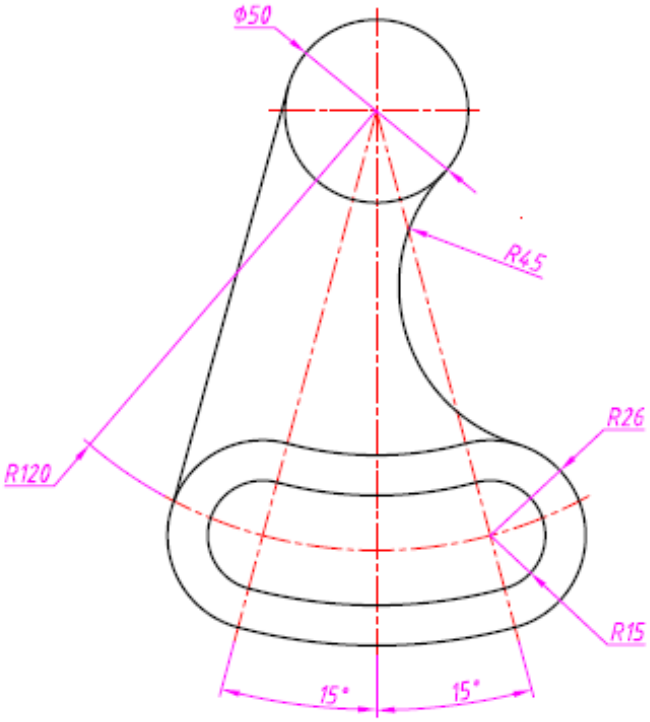


Fig. 4.10. The original drawing for Practical Work 2 with using layers

At the very beginning, you need to analyze the drawing and determine the sequence of its execution.

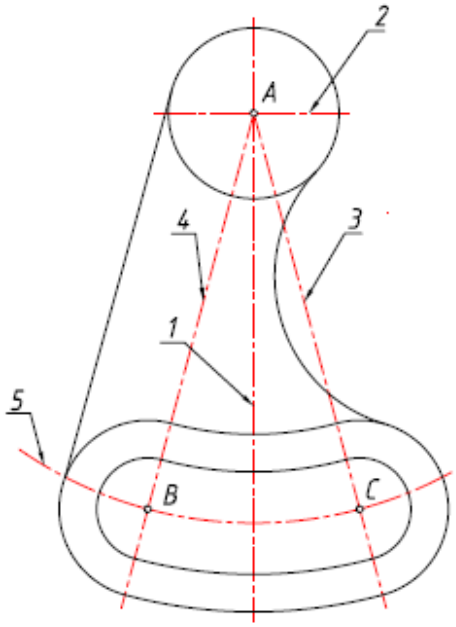


Fig. 4.11. Key points and lines

The intersection points of the center lines A, B, and C are the centers of the corresponding circles (fig. 4.11).

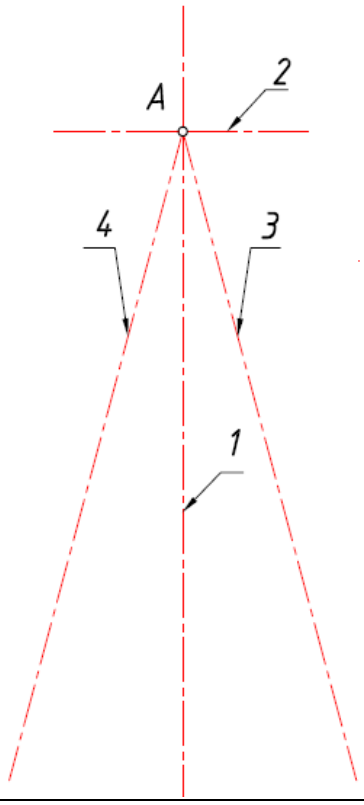
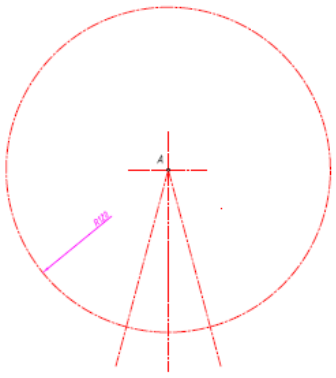
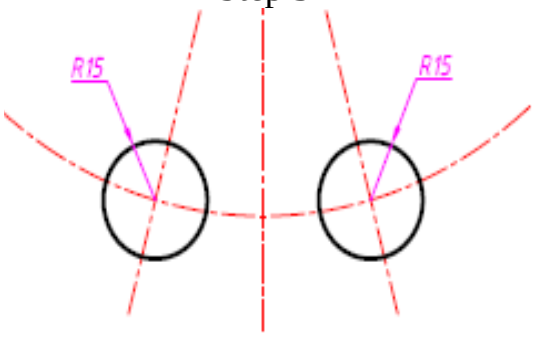
Therefore, the beginning of the construction will be associated with setting the location of these points.

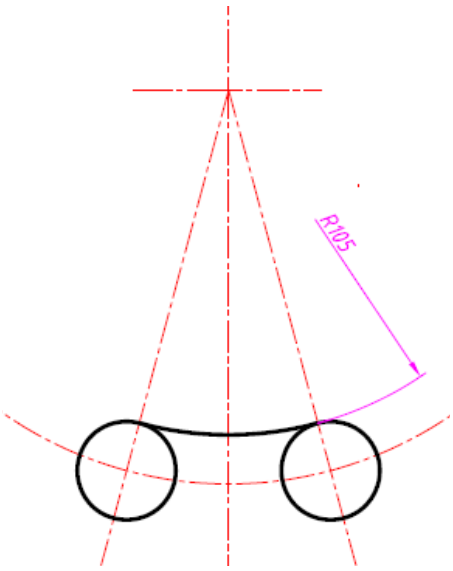
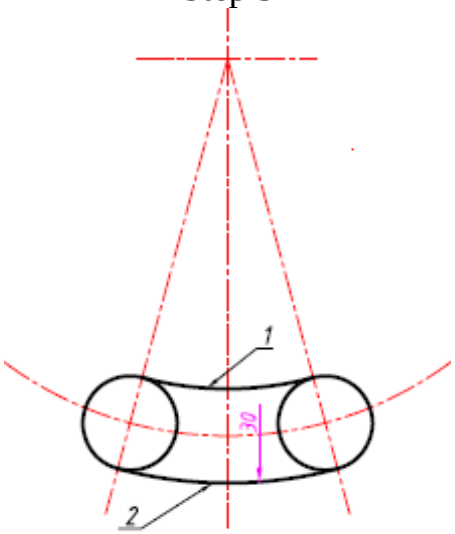
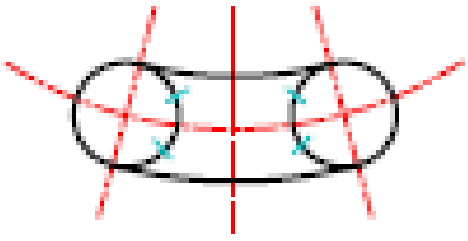
To do this, draw all the centerlines 1, 2, 3, 4, and 5.

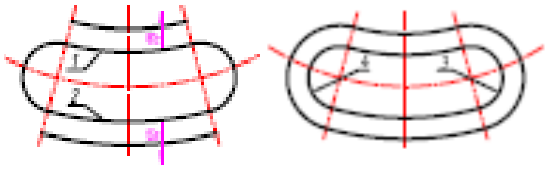
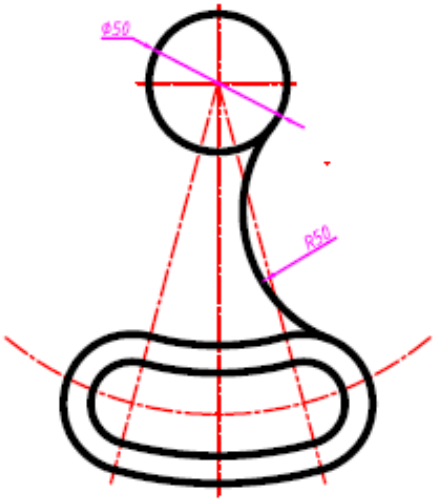
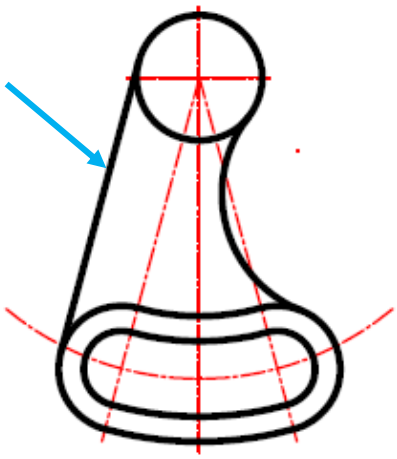
The tab. 4.2 shows the step-by-step construction of this drawing.

Table 4.2

The step-by-step construction of Practical Work 2

Drawing construction steps	Description of drawing construction
1	2
Step 1 	1. Set the current layer to **Axial**. 2. Ortho ON. Use the **Line** command to draw vertical 1 and horizontal 2 lines (at random). 3. Draw slanted lines 3 и 4. Ortho OFF, Dynamic Input is ON. Use the **Line** command to draw line 3: Specify first point: – the point **A** (use Object Snap **Intersection**). Specify next point or [Undo]: – Moving the cursor to the position where the value of the angular size 75 appears in the **Dynamic Input** area (90 – 15 = 75). The line length is approximately 150 mm. 4. To draw line 4. We perform the same actions as for drawing line 3. The only difference is the angular size of 105 (90 + 15)
Step 2 	Use the **Circle** command (Center, Radius) to draw the centerline 5. Circle radius – 120 мм. Center of the circle – the point **A** (Object Snap – **Intersection**)
Step 3 	1. Set the current layer to **Contour**. 2. Draw 2 circles with a radius of 15 mm. The centers of the circles are the intersection points of the center lines (Object Snap – **Intersection**)

1	2
Step 4 	Use **Fillet** Command.  Find the **Fillet** icon in the **Draw** panel and follow the prompts on the Command Line: Select first object or [Undo/Polyline/**Radius**/Trim/Multiple]: **R** (we point the cursor to the **Radius** option in the command line or type **R** on the keyboard). Specify fillet radius <0>: **105 Enter**. Select first object or [Undo/Polyline/Radius/Trim/Multiple]: (**choosing the first circle**). Select second object or shift-select to apply corner or [Radius]: (**choosing the second circle**)
Step 5 	Use **Offset** Command. The following prompts on the Command Line: Specify offset distance or [Through/Erase/Layer] <10.0>: **30** (R15 + R15) **Enter**. Select object to offset or [Exit/Undo] <Exit>: **arc 1**, show the offset direction with the cursor, and when arc 2 appears, fix it (<Clic>)
Step 6 	Use **Trim** Command. According to the drawing, you need to remove the parts of the circles. Using the command was discussed earlier, step 4 of Practical work 1

1	2
Step 7 	Use **Offset** Command. All actions are similar to step **5**, offset distance – 10 мм for all arcs 1, 2, 3 and 4
Step 8 	1. Draw circle with a diameter of 50 mm. The centers of the circles are the intersection points of the center lines (Object Snap – **Intersection**). 2. Use **Fillet** Command. All actions are similar to step 4. Fillet radius – 50 mm
Step 9 	1. Drawing the tangent line indicated by the arrow to the two arcs (All Object Snap **Off**, Object Snap **Tan**gent **On**). 2. Use the **Line** command. Specify first point: move the cursor to circle 1 and when the label «Different Tangent» appears, fix the point. Specify next point or [Undo]: perform the same actions with the second arc 2

And in conclusion, you need to add the dimensions (see fig. 4.10).

CHAPTER 5. INDIVIDUAL GRAPHIC TASKS

This section does not provide detailed step-by-step instructions for drawing drawings, but only indicates the main key stages of construction. During the exercises and practical tasks, basic skills of building and editing images in AutoCAD were obtained.

Individual Graphic Tasks (IGT) are performed on A4 and A3 formats using the appropriate templates. In order not to create layers every time, you need to open the previously created drawing with the corresponding ready-made settings (for example, Practical Work 1), delete all the images and save the drawing under a new name, for example IGT 1.

5.1. Individual Graphic Task 1

Draw two curved contours according to an individual option (fig. 5.1), see appendix 1 (Use A3 template).

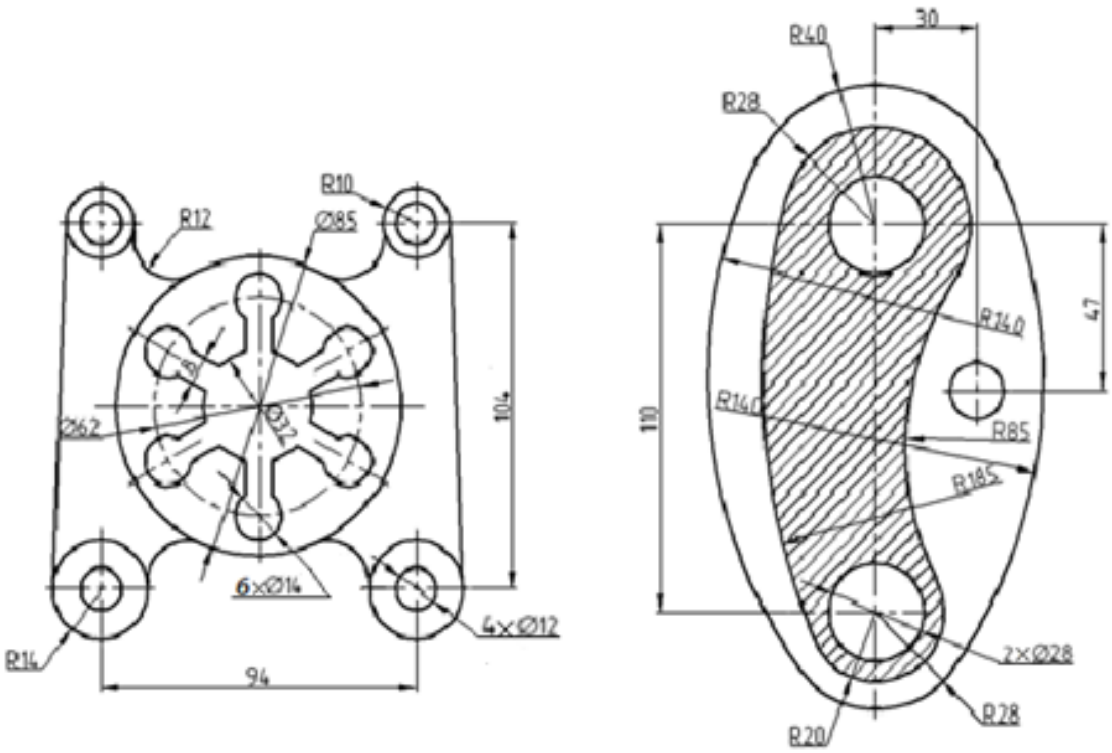


Fig. 5.1. Initial data for the Individual Graphic Task 1

When drawing IGT 1, it is necessary to use the acquired skills when performing Practical Work 1 and Practical Work 2. The completed task must be drawn up in accordance with the fig. 5.2.

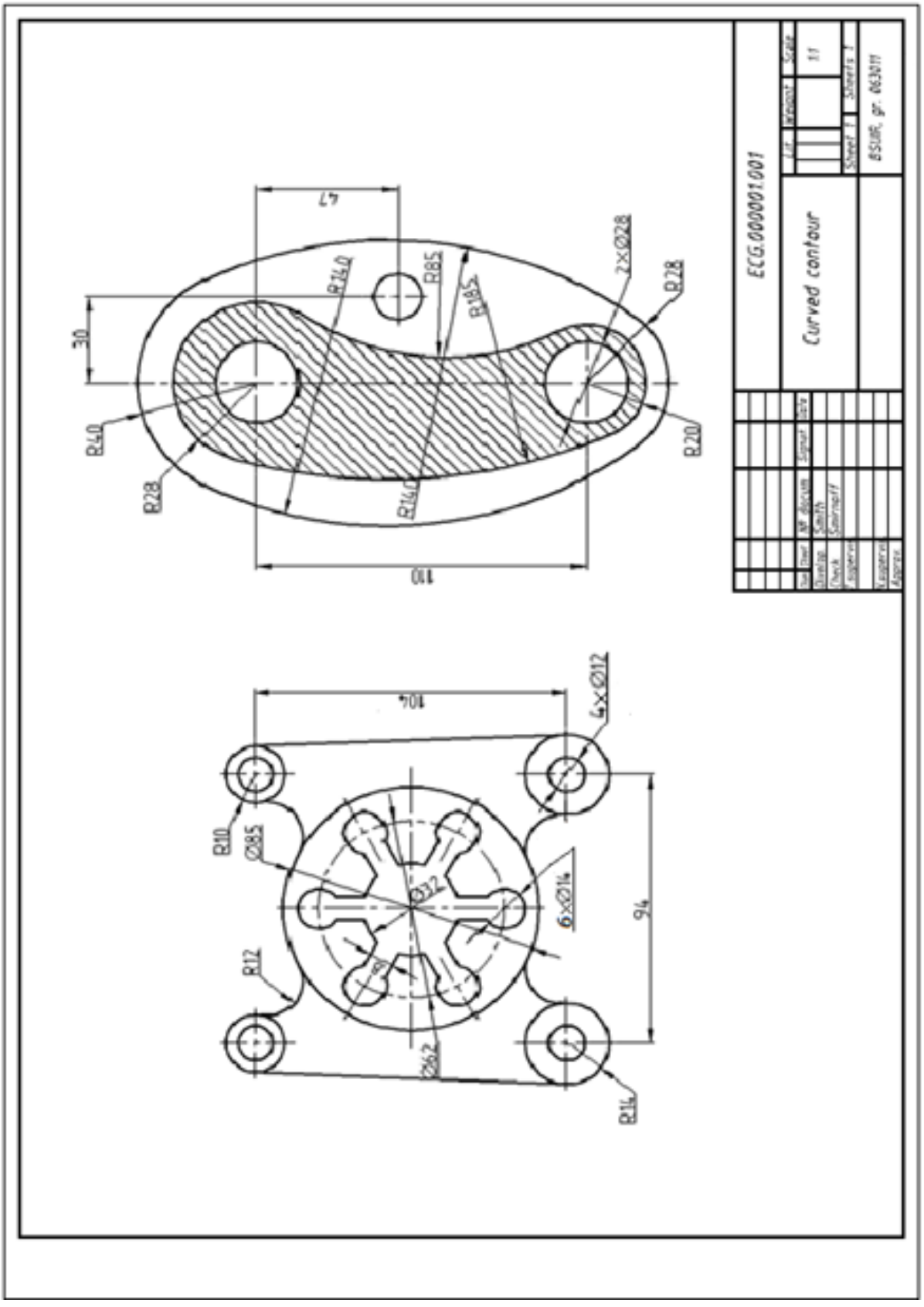


Fig. 5.2. Sample Individual Graphic Task 1

5.2. Individual Graphic Task 2

According to the Isometric drawing (fig. 5.3, *b*), draw three views: a Front view, a Top view and a Left said view (fig. 5.3, *a*).

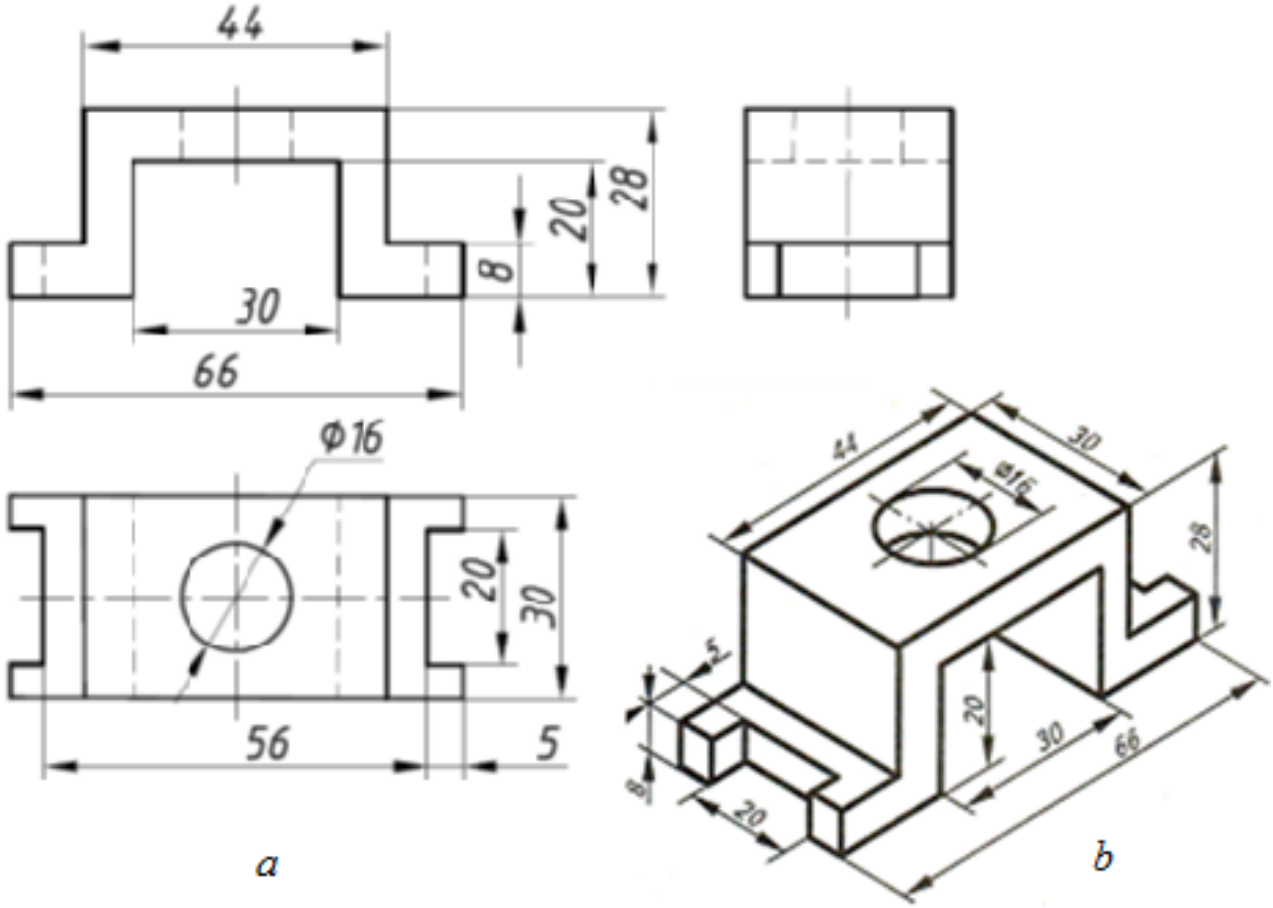


Fig. 5.3. The original drawing for Individual Graphic Task 2:
a – three views; *b* – the isometric drawing

As you can see from the drawing, an invisible contour line is used in the images. Therefore, you need to create another layer for drawing invisible lines – **Layer 5**.

Layer name – Invisible line, line type – dashed (Linetype – Hidden), line thickness – Default, line color – orange.

When performing the task, it is necessary to use auxiliary constructions on the Defpoints layer.

Brief instructions for drawing images:

1. Draw a vertical centerline 1-2 of arbitrary length.
2. Draw a Front view (fig. 5.4), the line 1-2 is the axis of symmetry of the image.

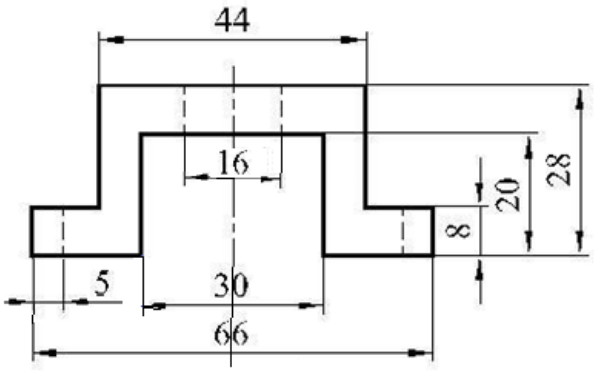


Fig. 5.4. The Front view

3. Draw auxiliary vertical lines on the Defpoints layer and the Top view using line 3-4 as the axis of symmetry (fig. 5.5).

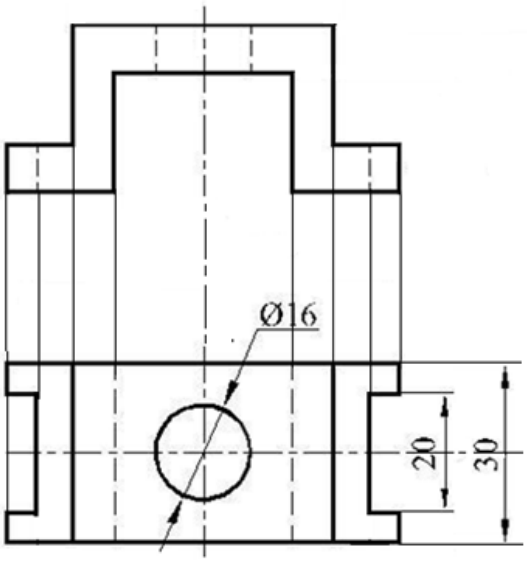


Fig. 5.5. The Top view

Use the method shown in the fig. 5.6 to draw the a Left said View.

When performing this task, you can first draw all the images on the Defpoint layer (with auxiliary lines), and then change the line type in accordance with the content.

Use the method shown in the fig. 5.6 to draw the Left said view (The **Modify** Comands – Move, Rotate).

The completed task must be drawn up in accordance with the fig. 5.7.

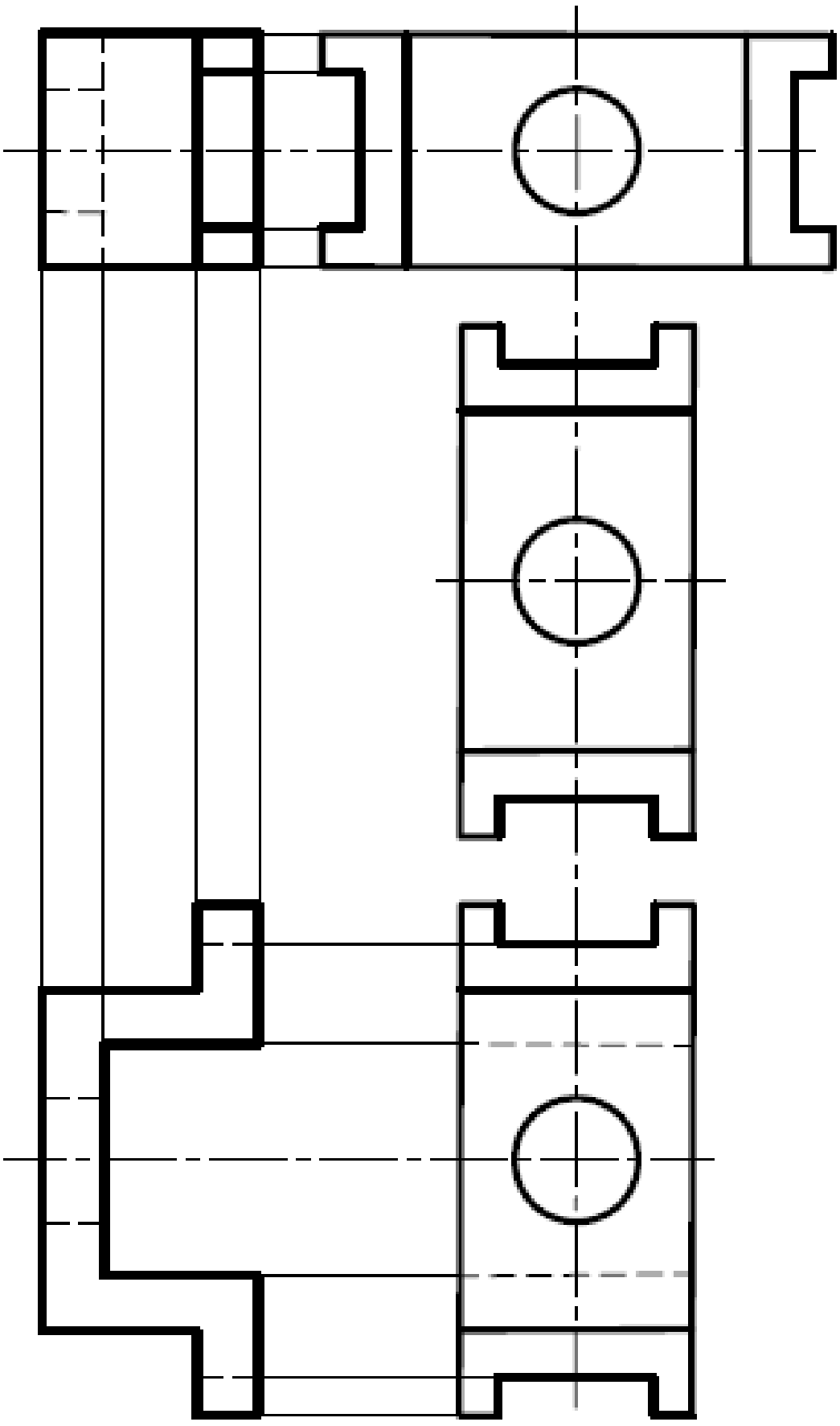


Fig. 5.6. Creating the Left said view

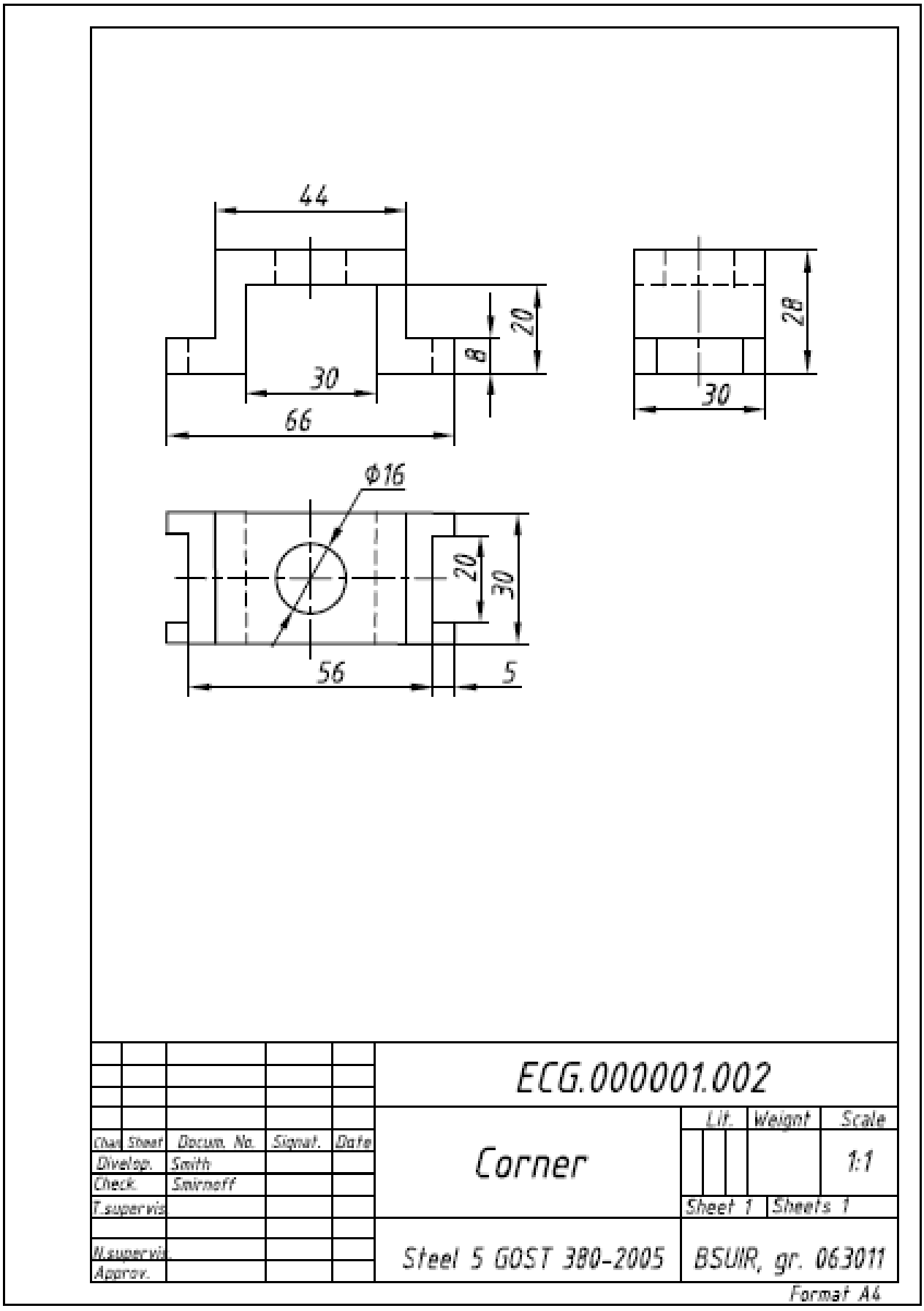
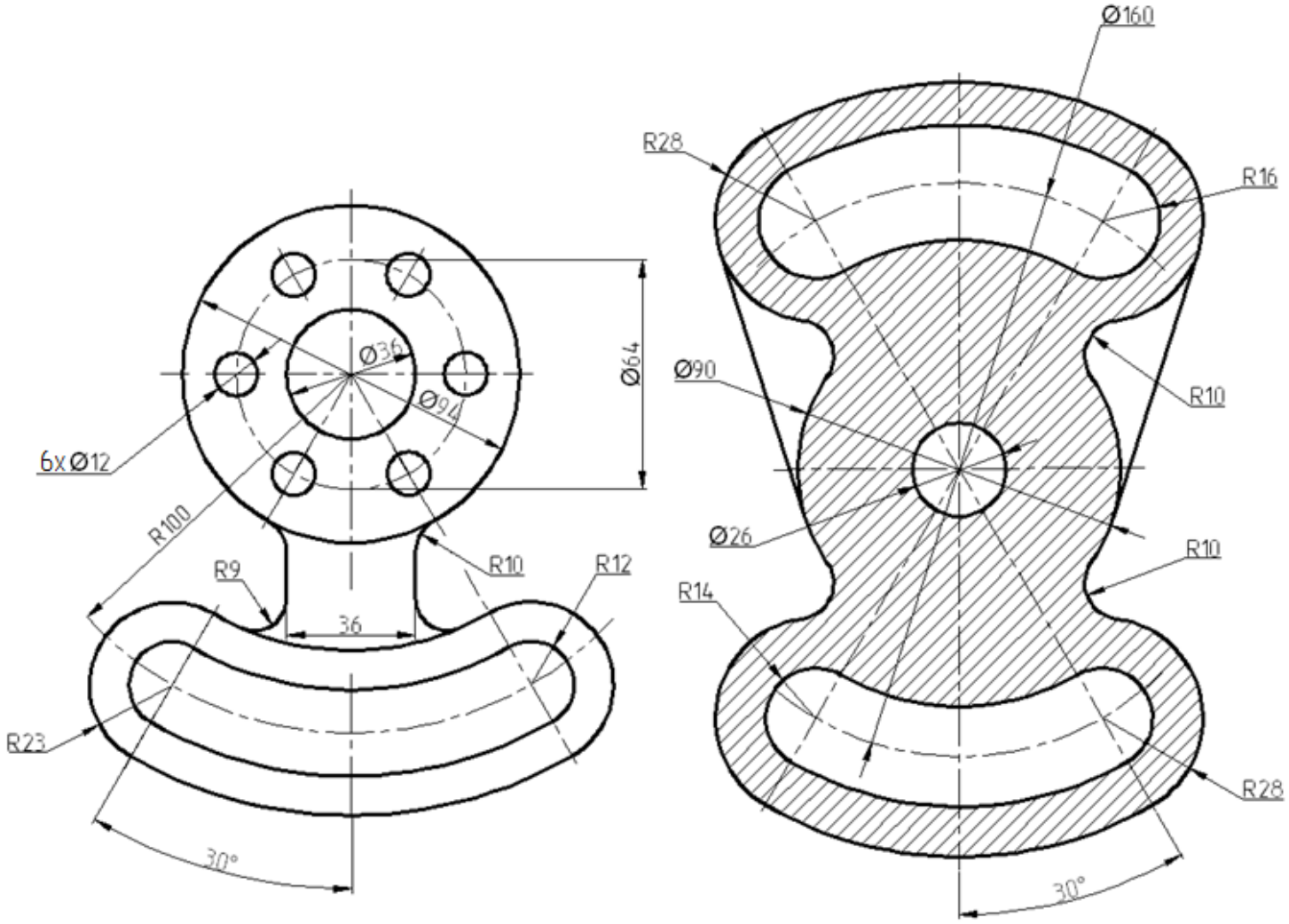


Fig. 5.7. Sample Individual Graphic Task 2

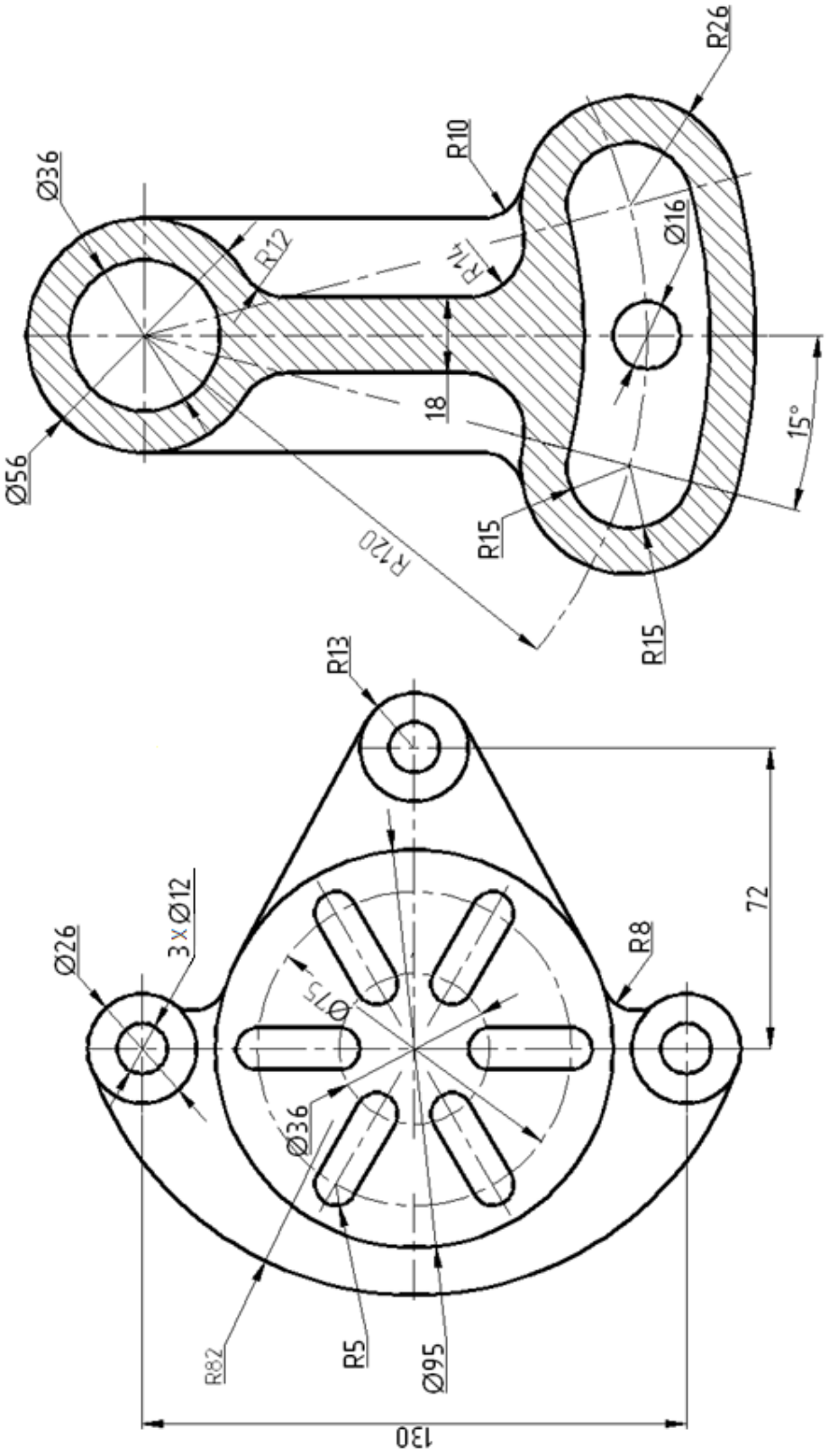
APENDIX 1

Options for Individual Graphic Task 1

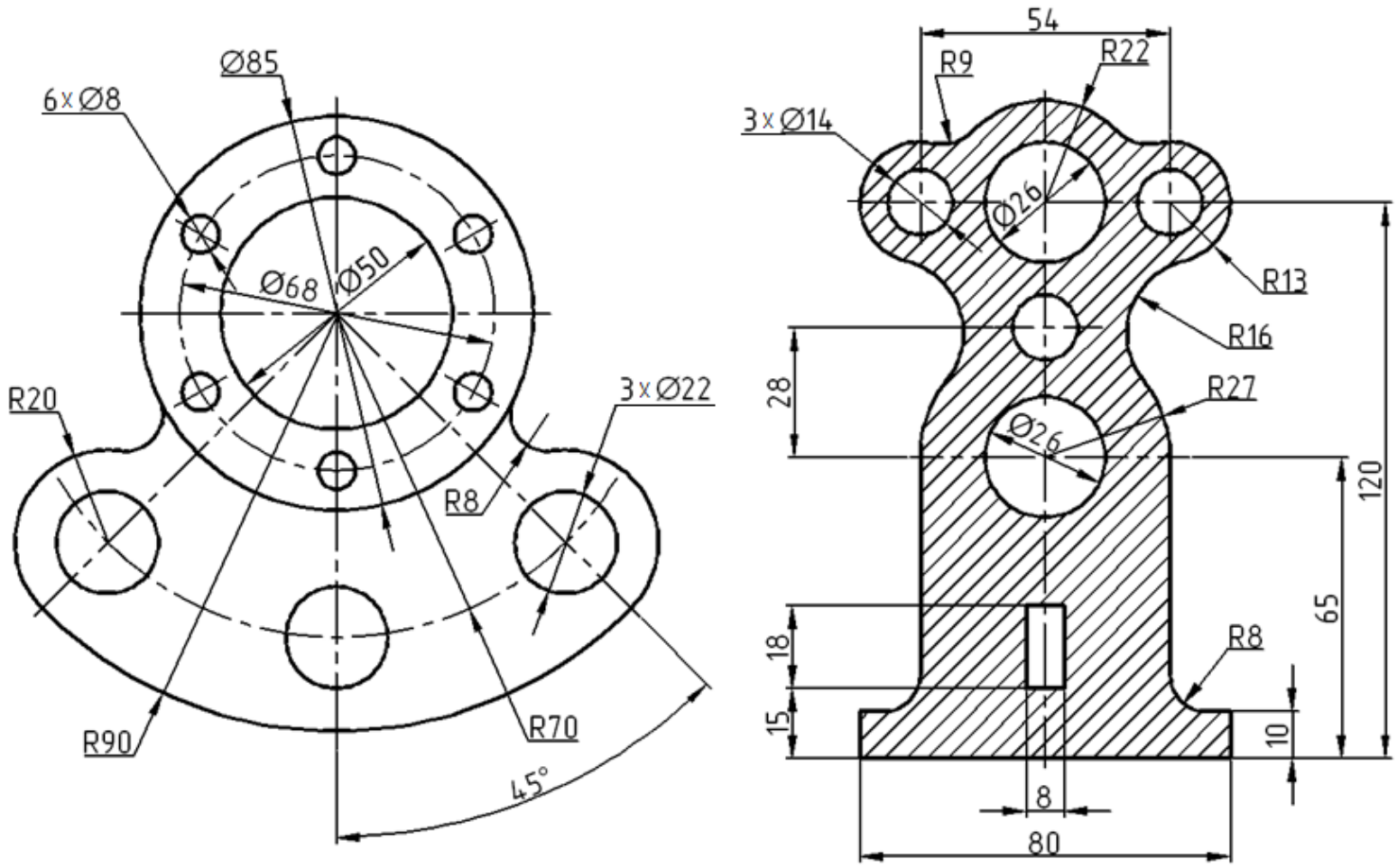
Option 1



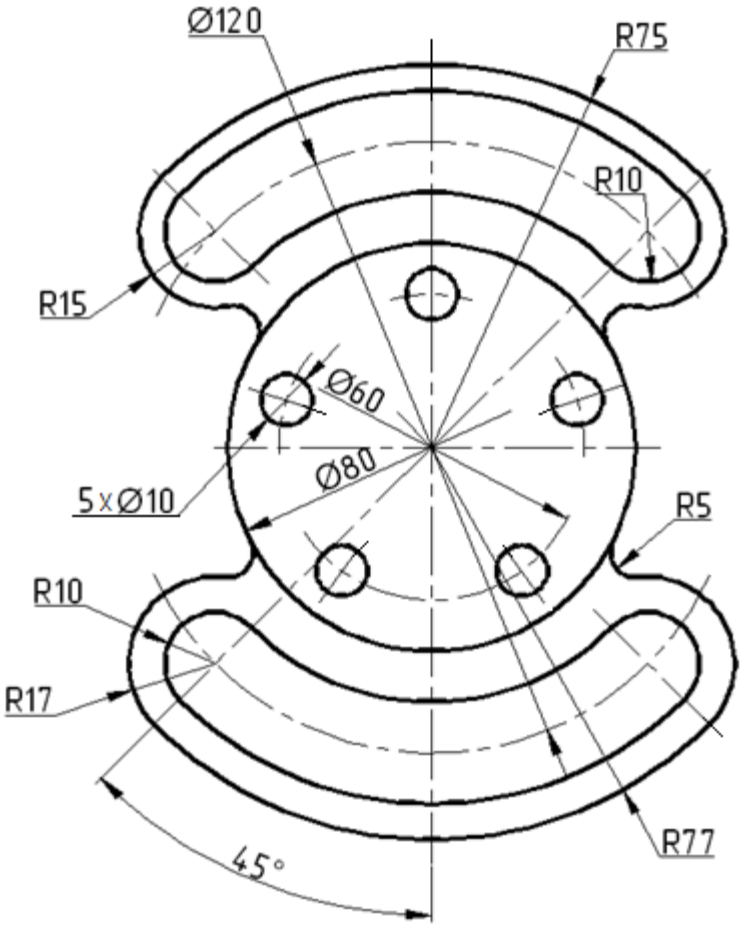
Option 2

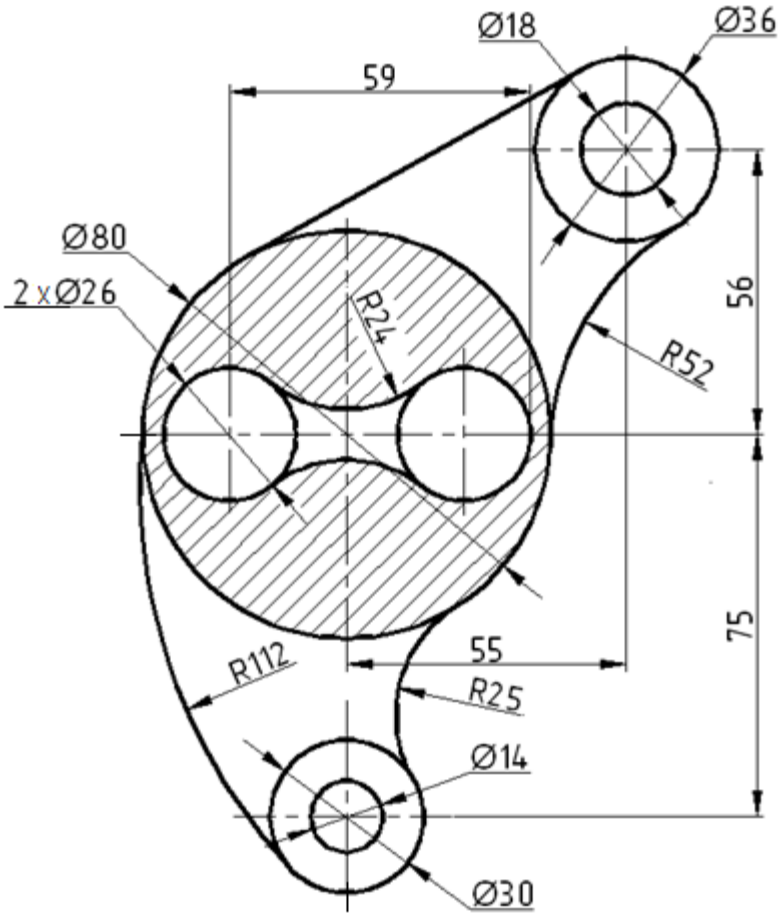

R26
R10
Ø36
R12
R17
Ø16
18
15°
Ø56
R120
R15
R15
R13
72
3 x Ø12
Ø26
R8
Ø75
Ø36
R82
R5
Ø95
130

Option 3

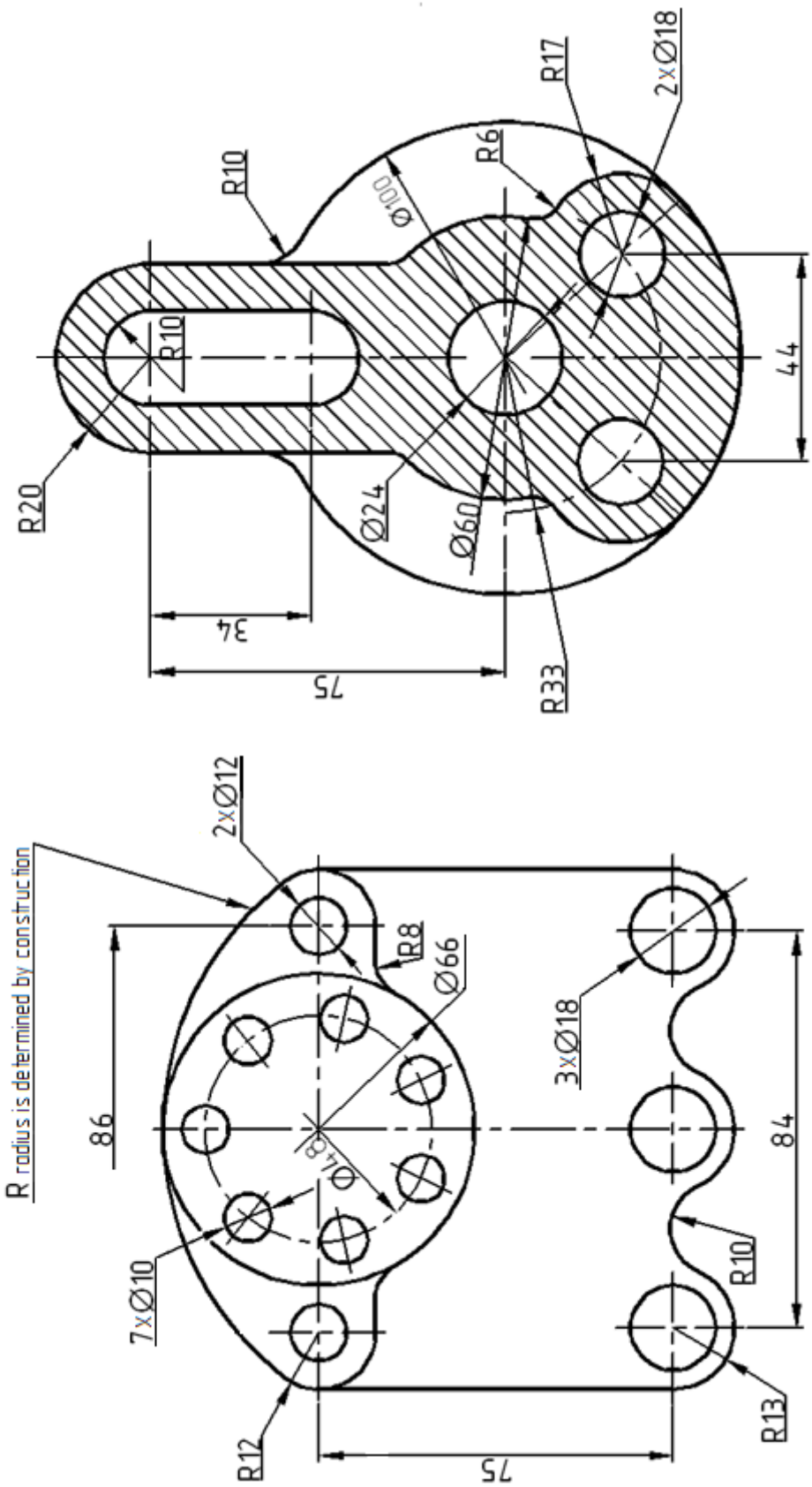

Ø85
6x Ø8
Ø68
Ø50
R20
3x Ø22
R8
R90
R70
45°
54
R9
R22
3x Ø14
Ø26
R13
R16
28
R27
Ø26
120
65
18
R8
15
10
8
80

Option 4

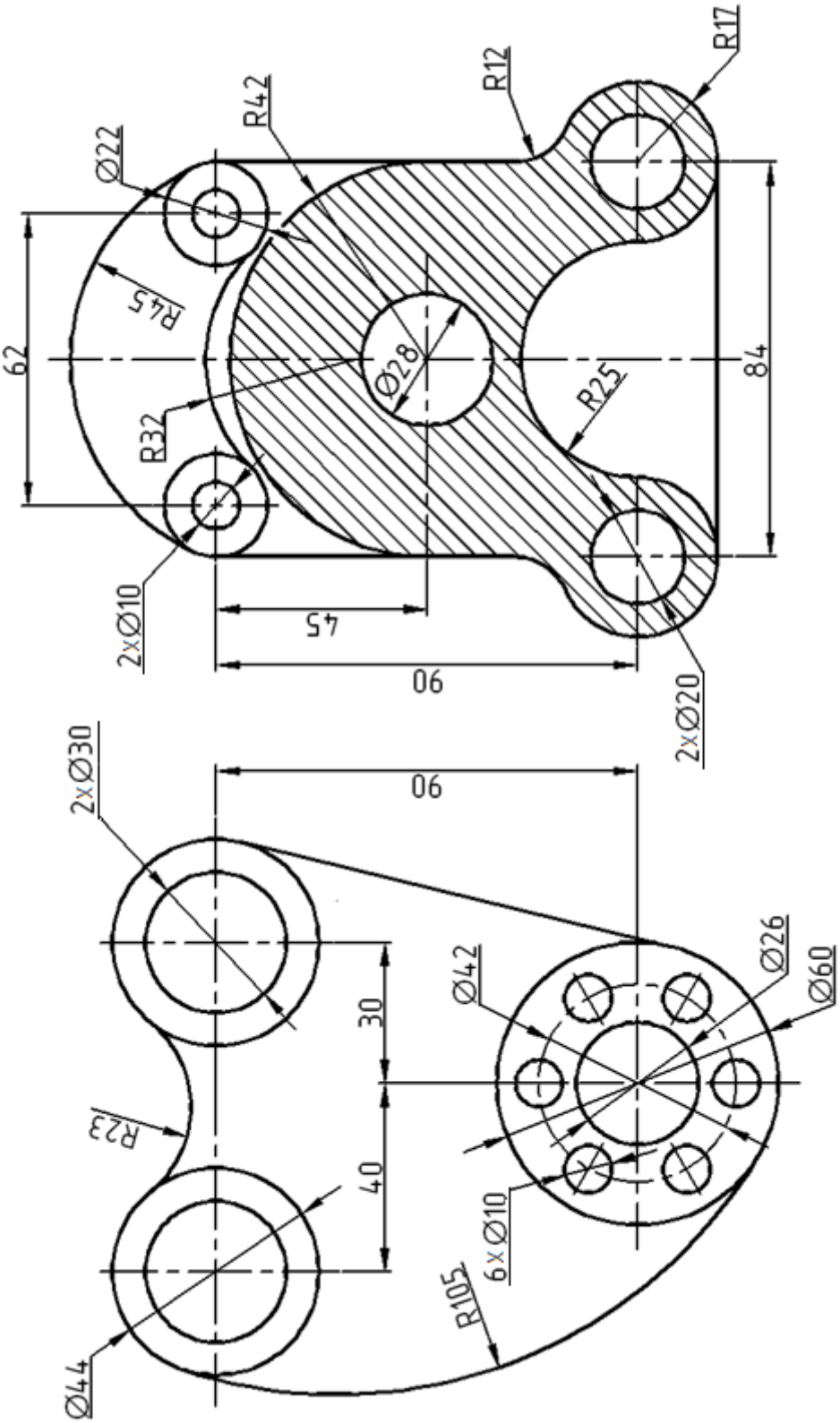

Ø120
R75
R10
R15
Ø60
Ø80
5 x Ø10
R5
R10
R17
45°
R77


Ø18
Ø36
59
Ø80
2 x Ø26
R24
R52
56
75
55
R112
R25
Ø14
Ø30

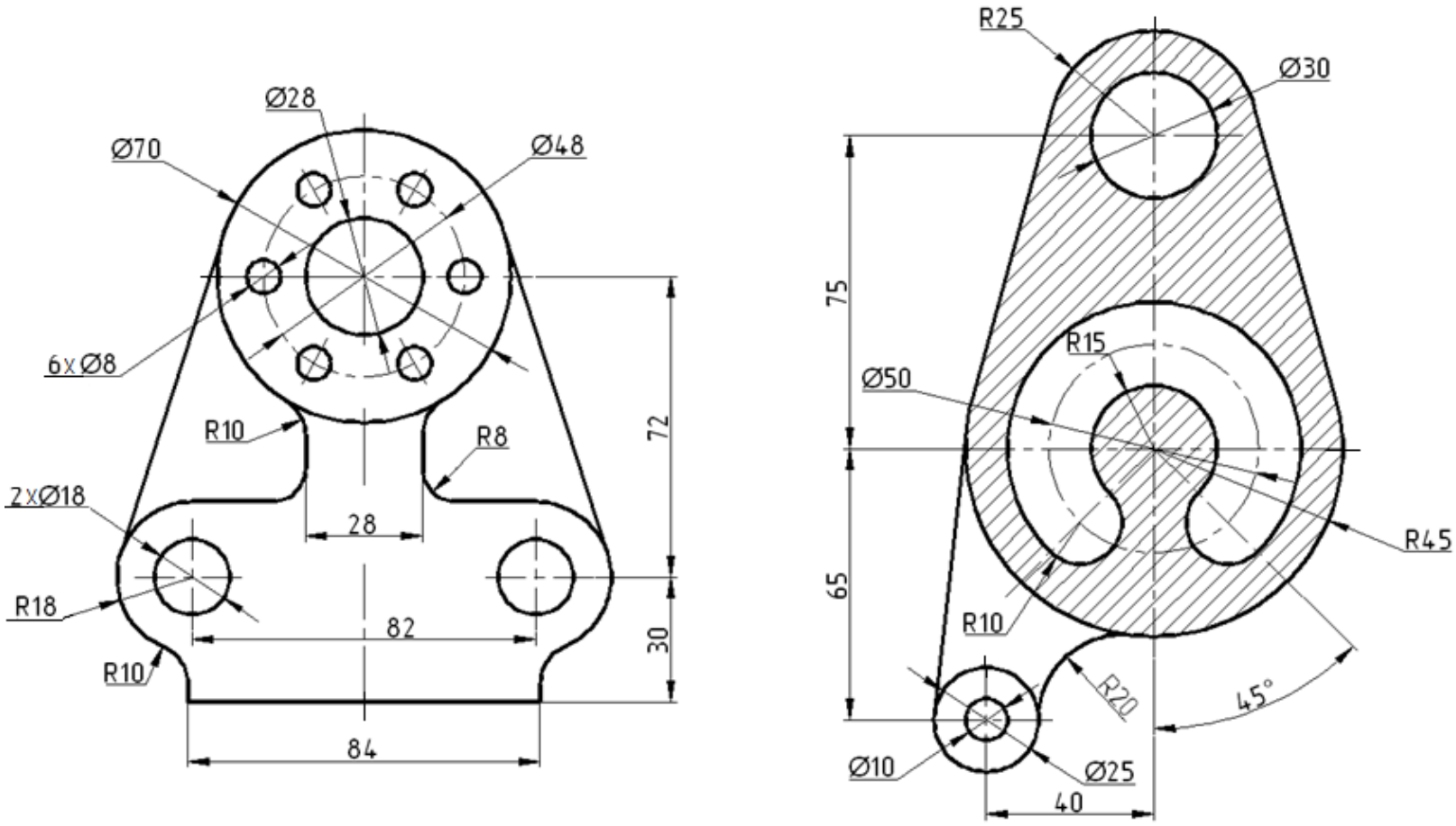
Option 5


R radius is determined by construction
2xØ12
R8
Ø66
86
Ø108
7xØ10
R12
75
3xØ18
84
R10
R13
R20
R10
R10
Ø100
R6
R17
2xØ18
44
Ø24
Ø60
R33
34
75

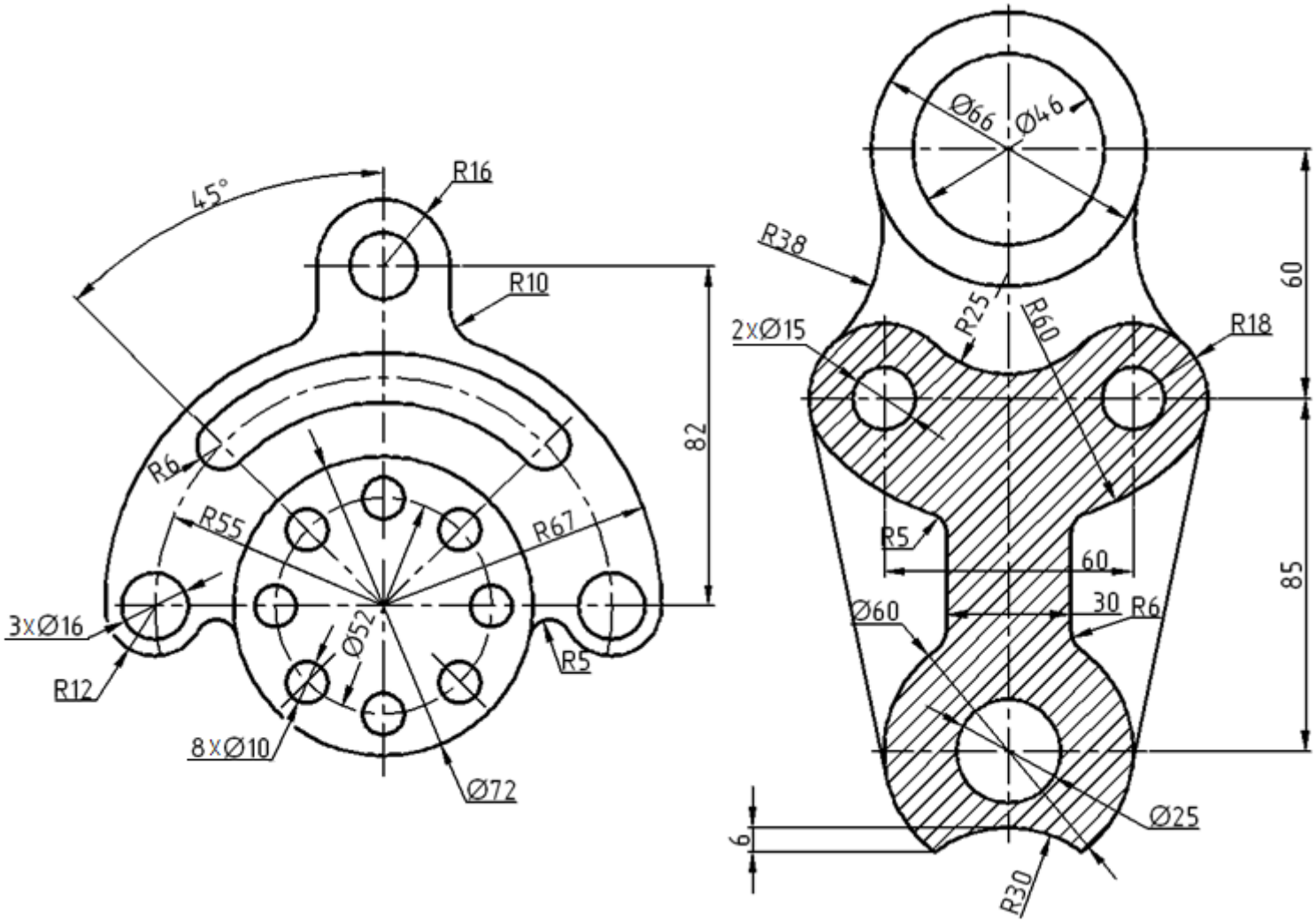
Option 6


R17
R12
R42
Ø22
R45
62
Ø28
R25
84
R32
2xØ10
45
90
2xØ20
2xØ30
90
Ø26
Ø42
Ø60
30
R23
40
6xØ10
R105
Ø44

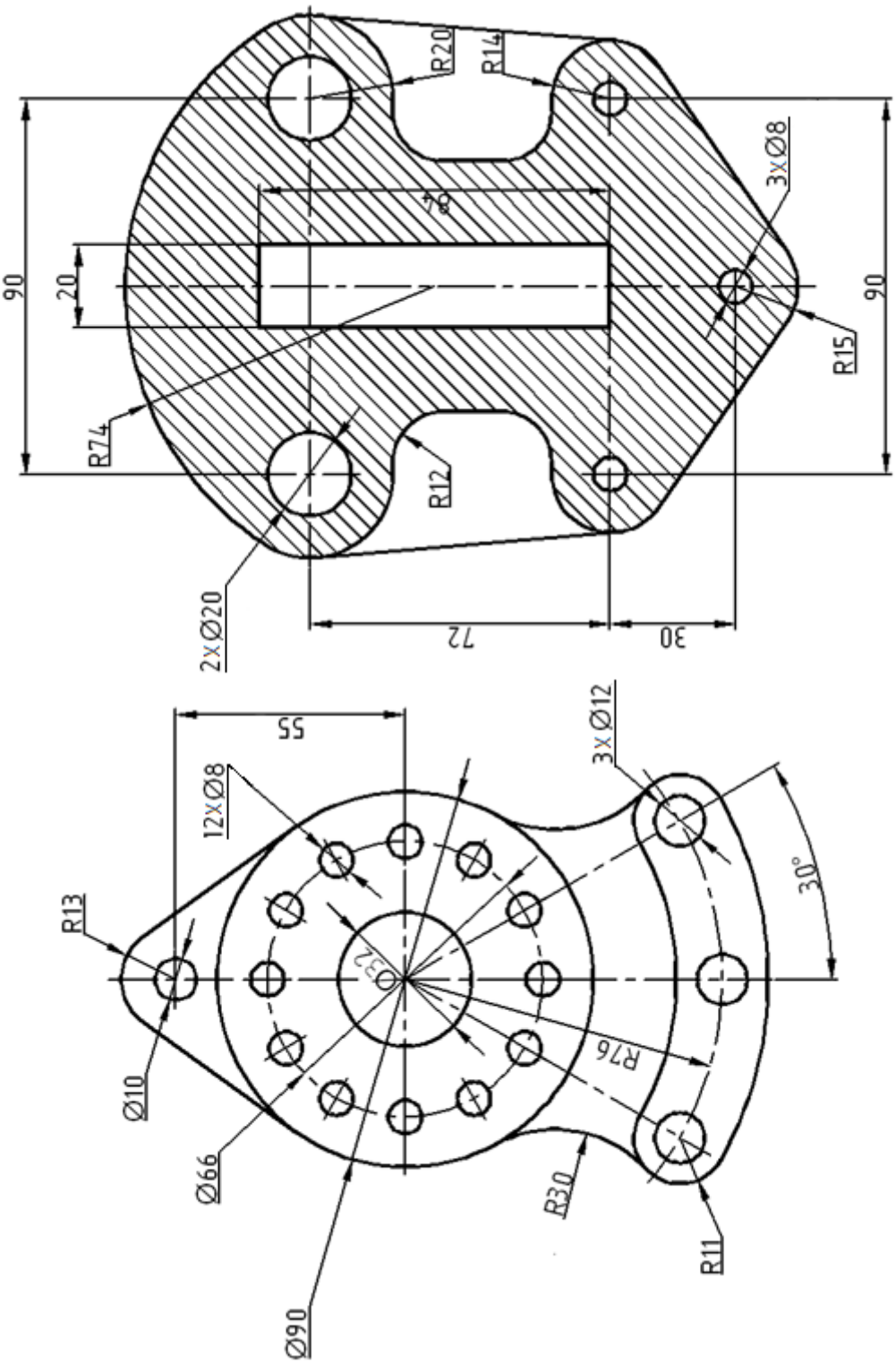
Option 7


Ø28
Ø70
Ø48
6xØ8
R10
R8
2xØ18
28
72
R18
82
30
R10
84
R25
Ø30
75
R15
Ø50
R45
65
R10
R20
45°
Ø10
Ø25
40

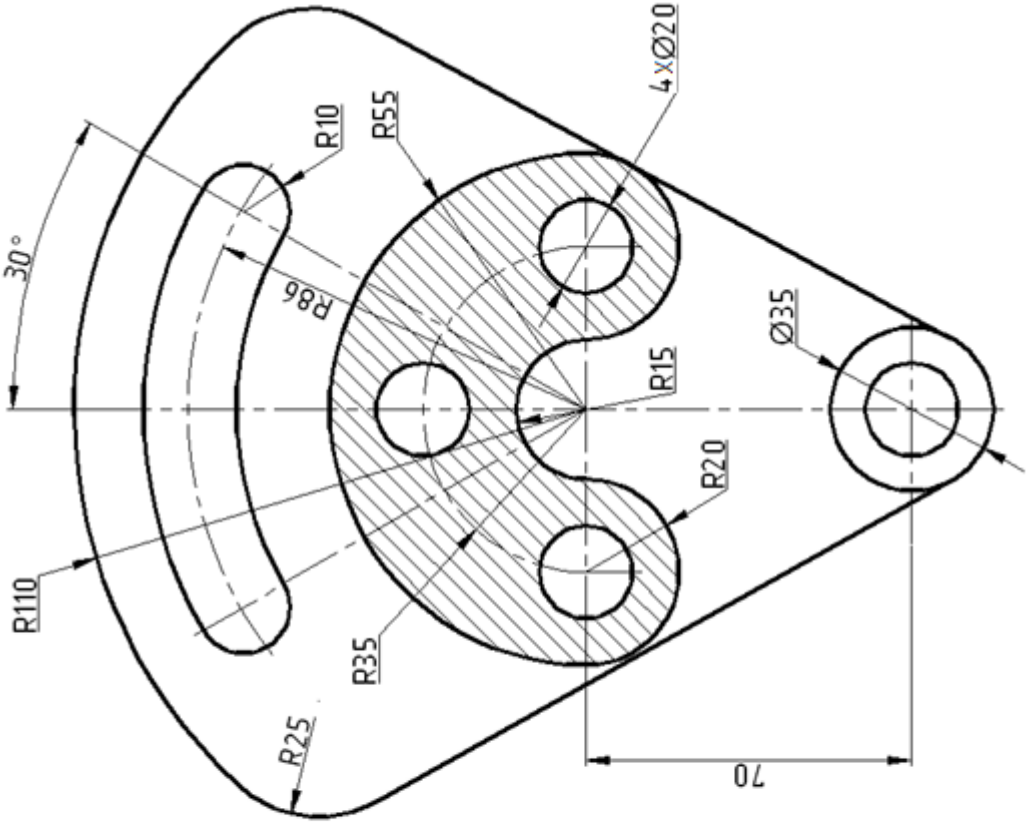
Option 8

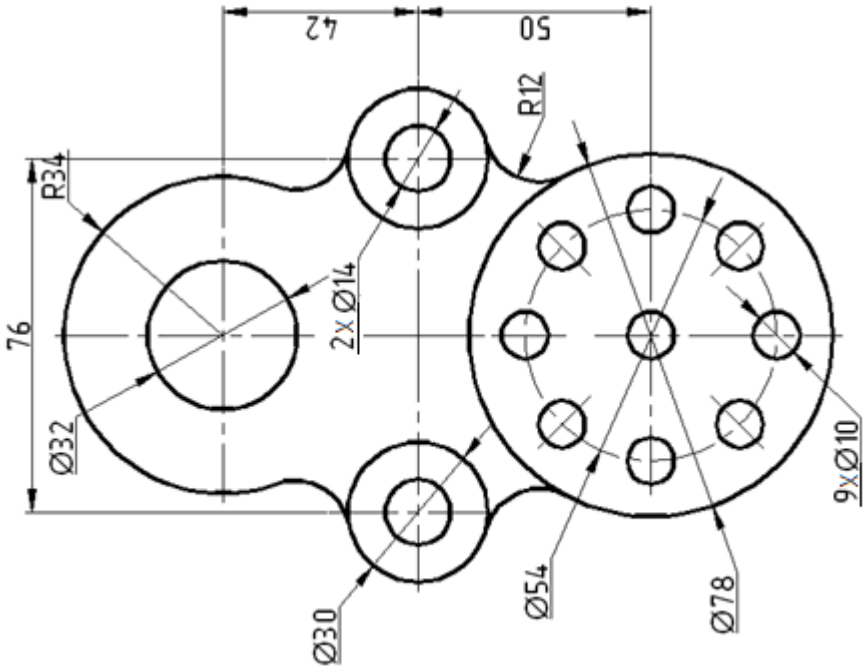

45°
R16
R10
82
R6
R55
R67
3xØ16
R12
Ø52
8xØ10
Ø72
R5
Ø66
Ø46
R38
R25
R60
2xØ15
R18
60
R5
60
85
Ø60
30
R6
Ø25
6
R30

Option 9

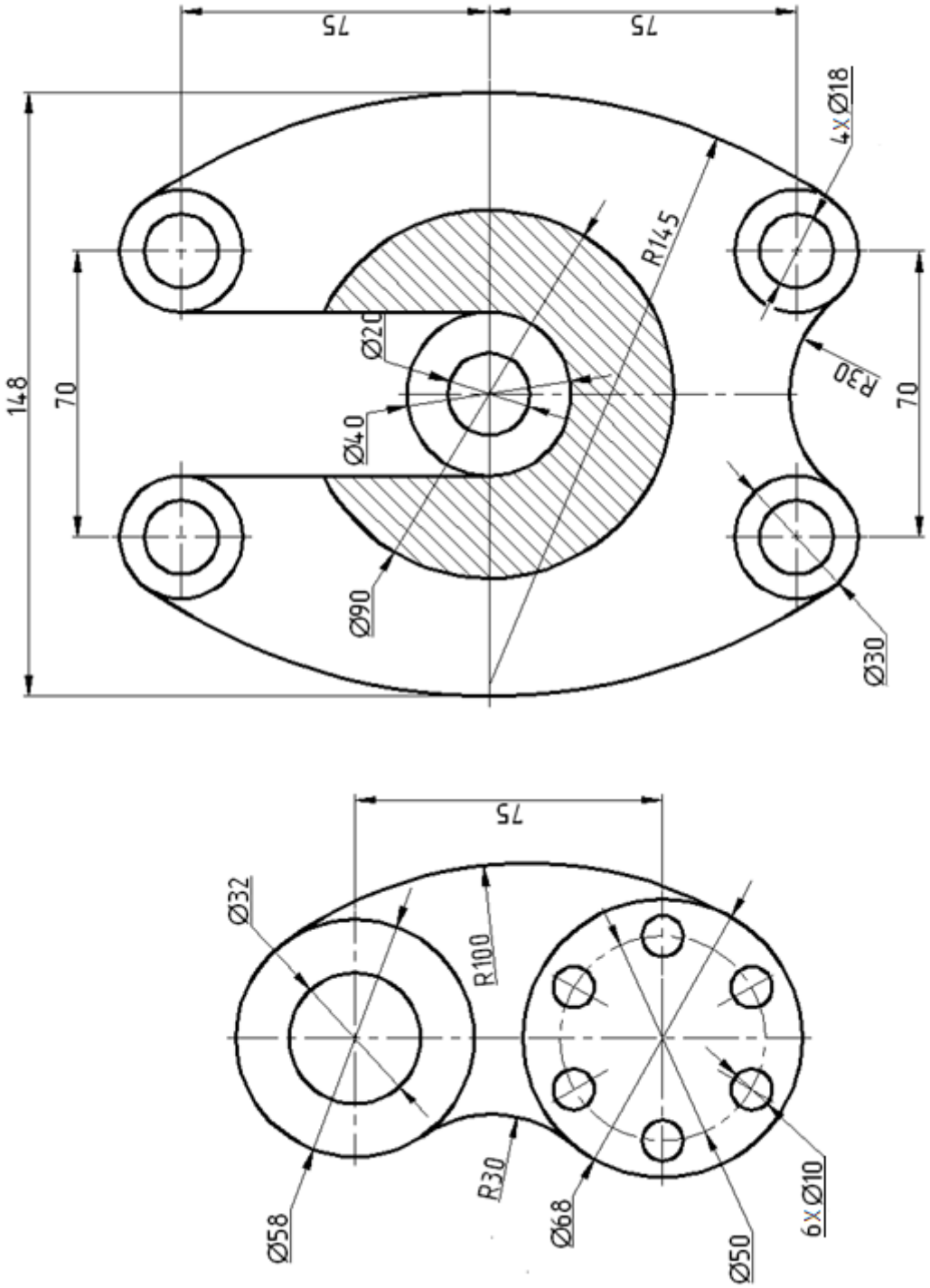

R20
R14
3XØ8
90
20
79
90
R15
R74
R12
2XØ20
72
30
55
12XØ8
3XØ12
30°
R13
Ø32
Ø10
R76
Ø66
R30
R11
Ø90

Option 10

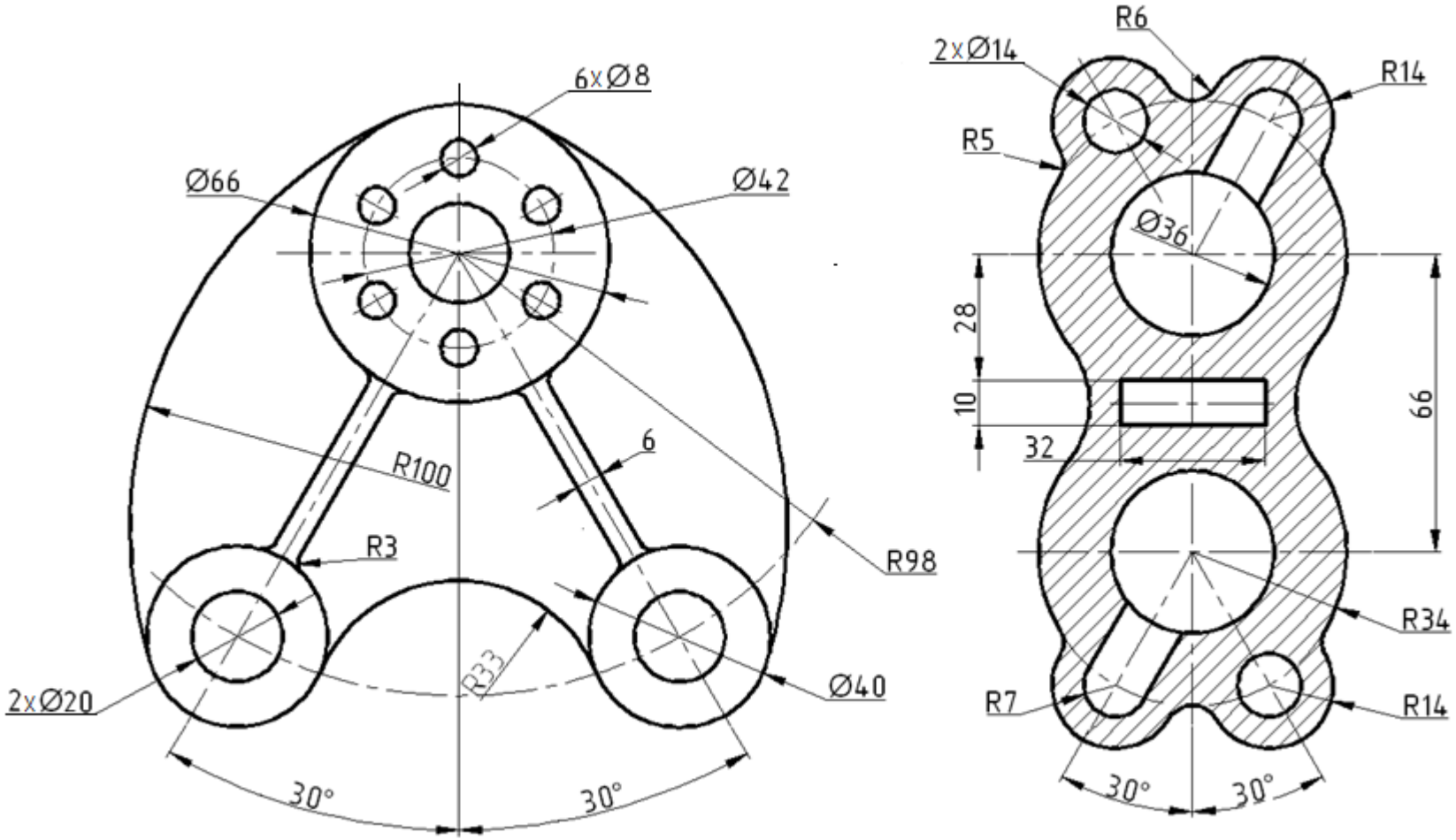

4XØ20
R10
R55
30°
R86
Ø35
R15
R20
R110
R35
R25
70


42
50
R12
R34
2XØ14
76
Ø32
9XØ10
Ø30
Ø54
Ø78

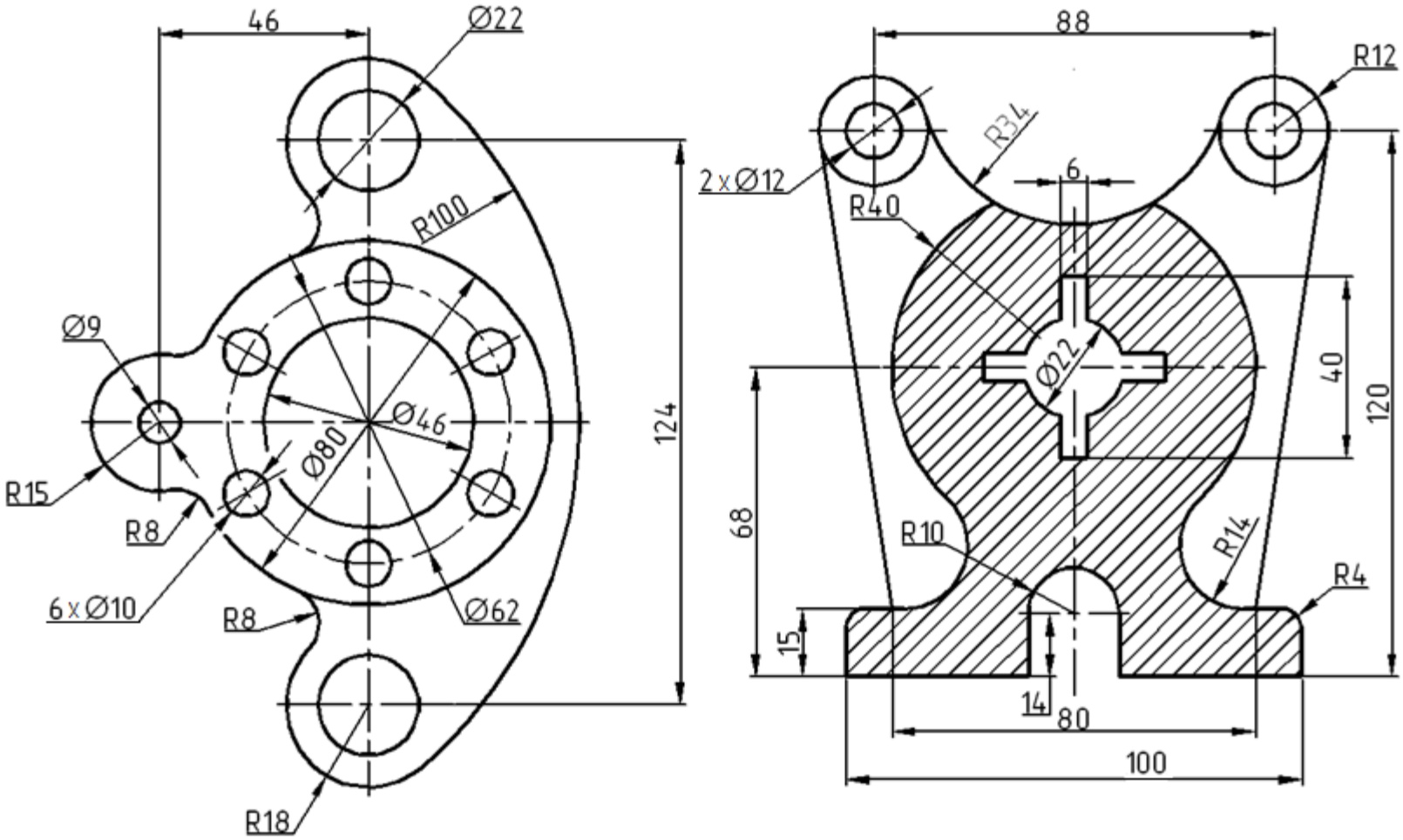
Option 11


75
75
4XØ18
R145
Ø20
Ø40
Ø90
148
70
R30
70
Ø30
75
Ø32
R100
R30
Ø58
Ø68
Ø50
6XØ10

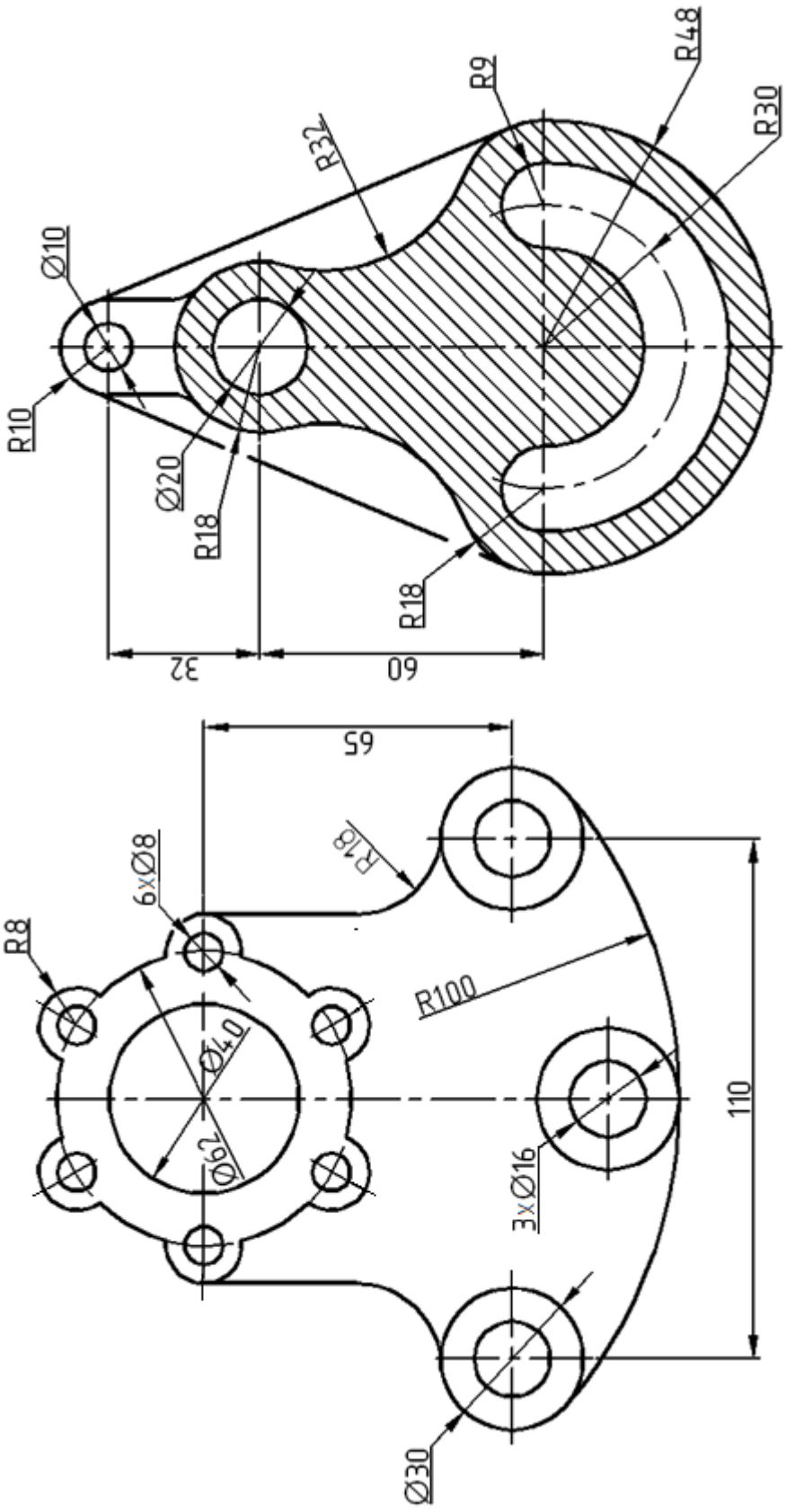
Option 12


6xØ8
Ø66
Ø42
R100
6
R3
R98
R33
2xØ20
Ø40
30°
30°
R6
2xØ14
R14
R5
Ø36
28
10
32
66
R34
R7
R14
30°
30°

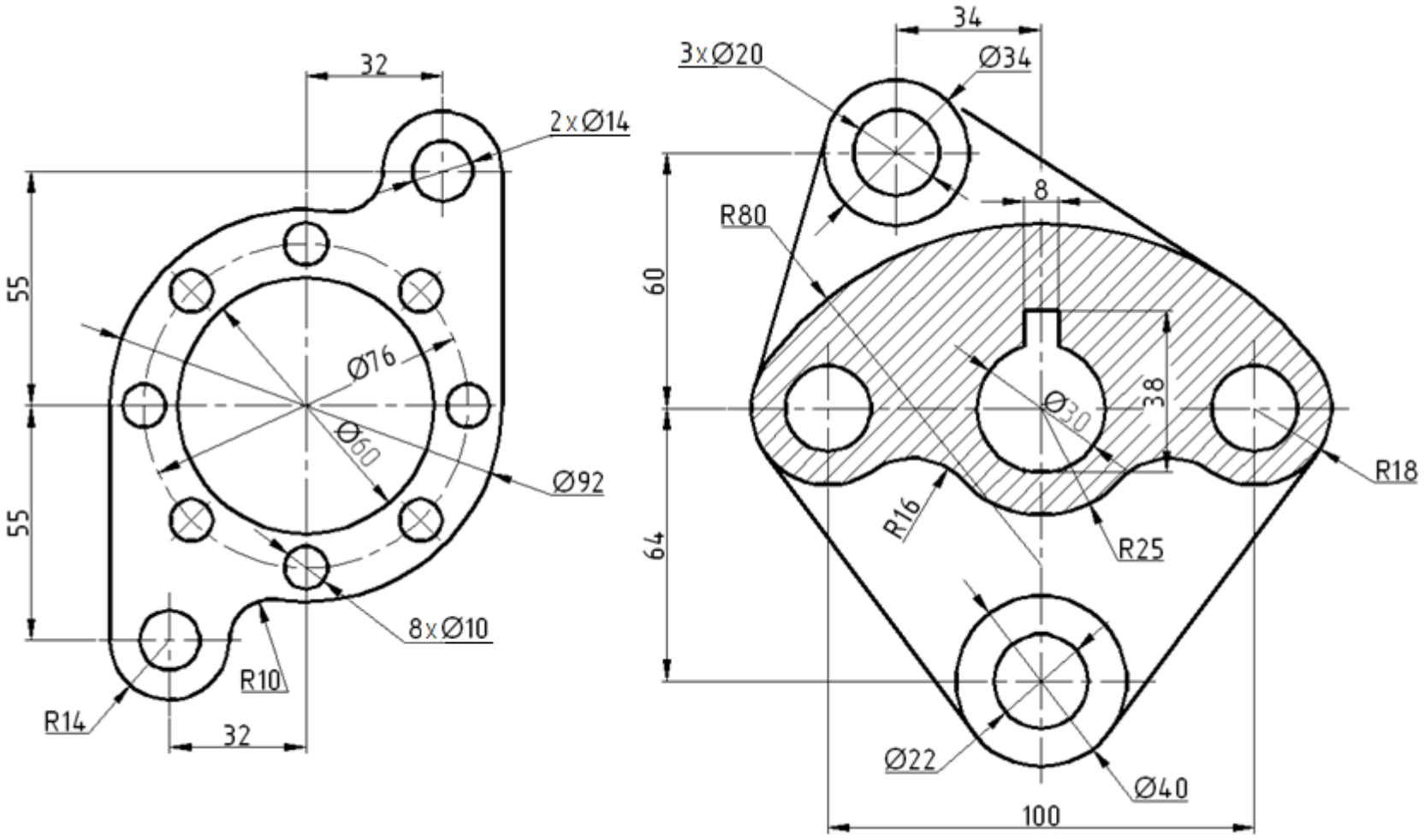
Option 13


46
Ø22
R100
Ø9
Ø46
Ø80
124
R15
R8
6 x Ø10
R8
Ø62
R18
88
R12
R34
2 x Ø12
6
R40
Ø22
40
120
68
R10
R14
R4
15
14
80
100

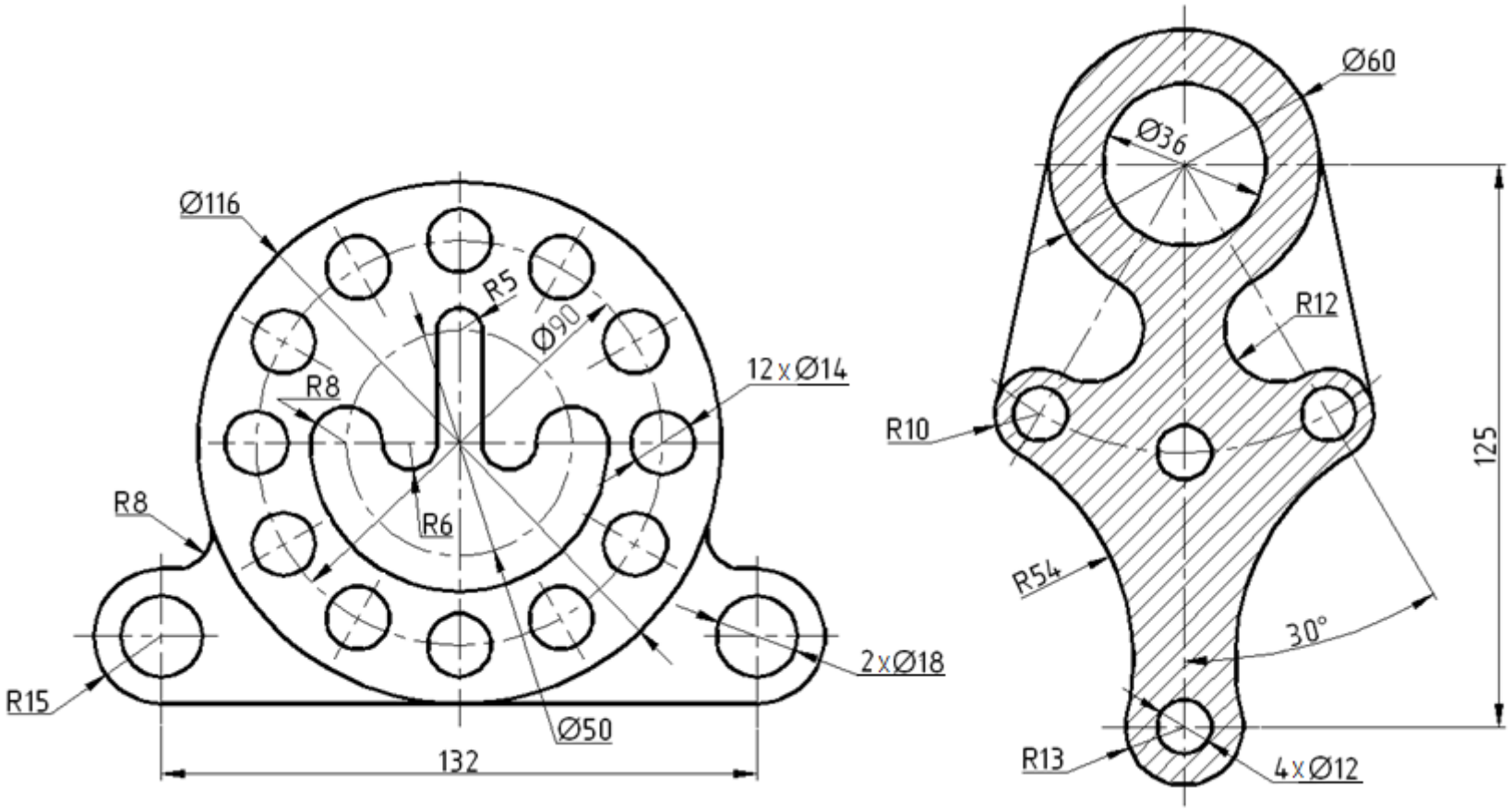
Option 14


R48
R9
R30
R32
Ø10
R10
Ø20
R18
R18
32
60
65
R18
6xØ8
R8
R100
Ø40
Ø62
110
3xØ16
Ø30

Option 15


32
2xØ14
55
55
Ø76
Ø60
Ø92
8xØ10
R10
R14
32
34
3xØ20
Ø34
8
R80
60
64
Ø30
38
R18
R16
R25
Ø22
Ø40
100

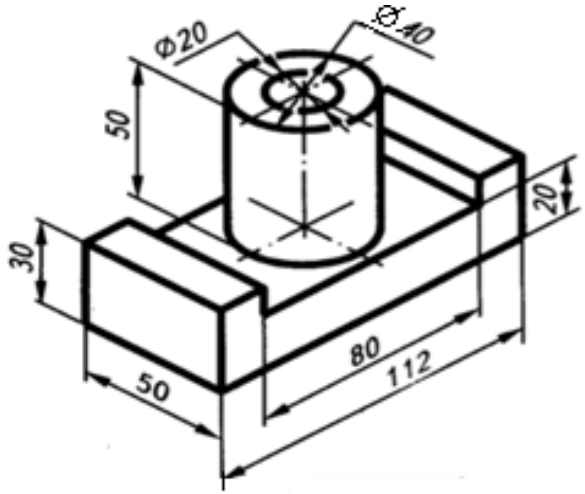
Option 16


Ø116
R5
Ø90
12xØ14
R8
R8
R6
R15
2xØ18
Ø50
132
Ø60
Ø36
R12
R10
125
R54
30°
R13
4xØ12

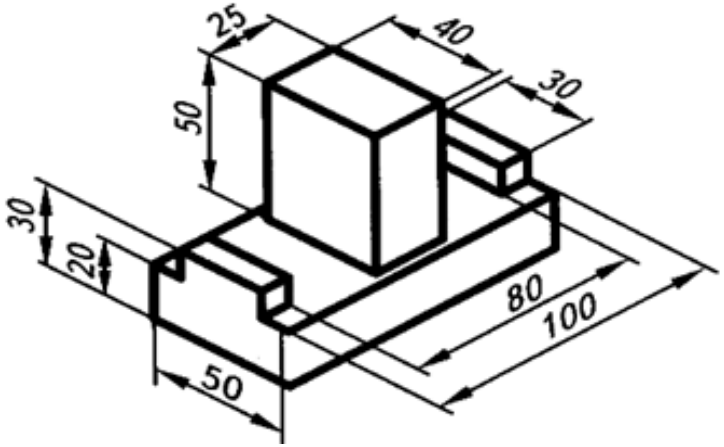
APENDIX 2

Options for Individual Graphic Task 2

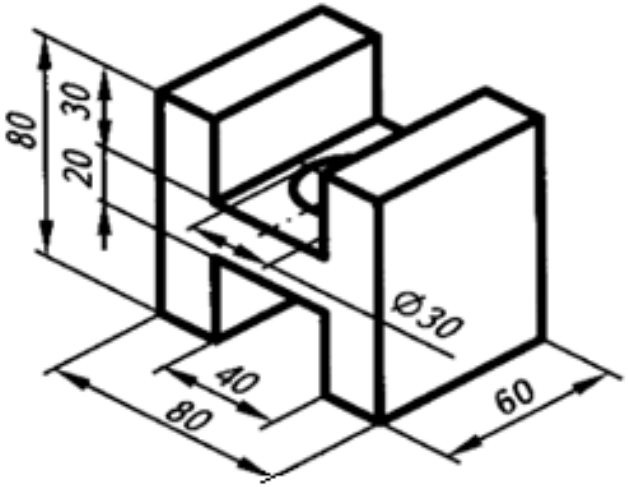
Option 1



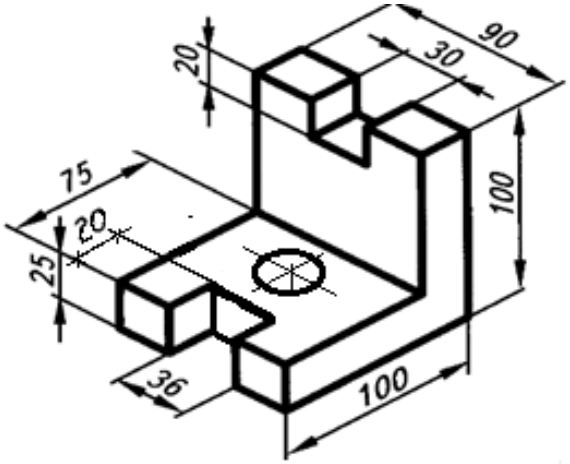
Option 2



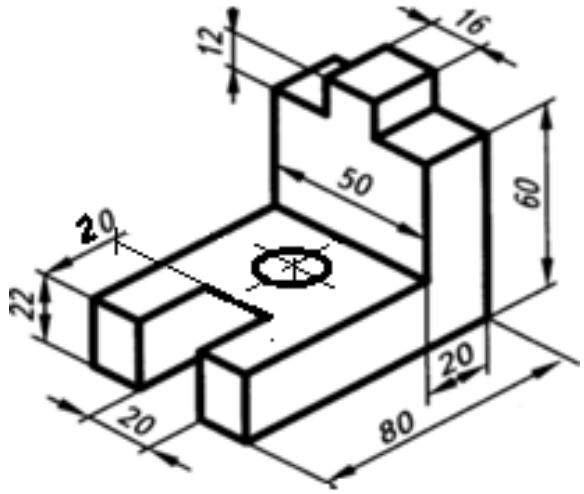
Option 3



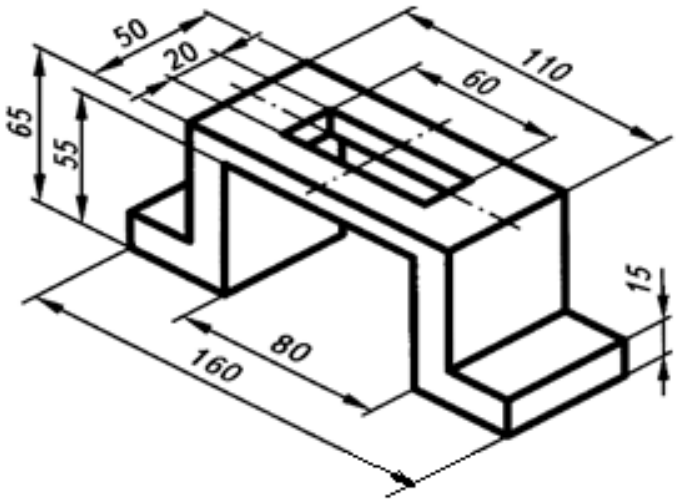
Option 4



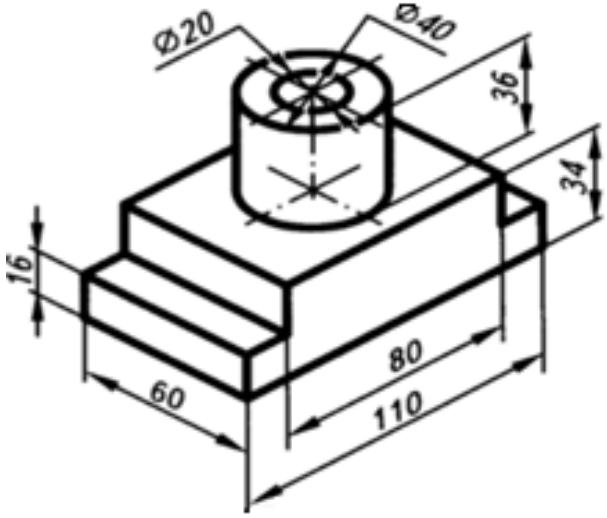
Option 5



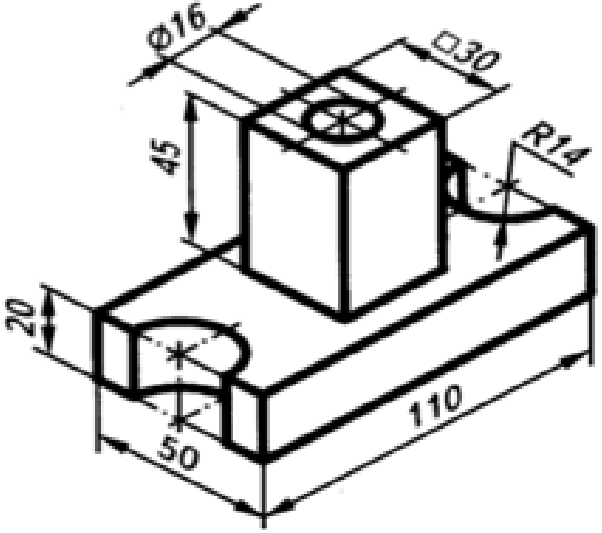
Option 6



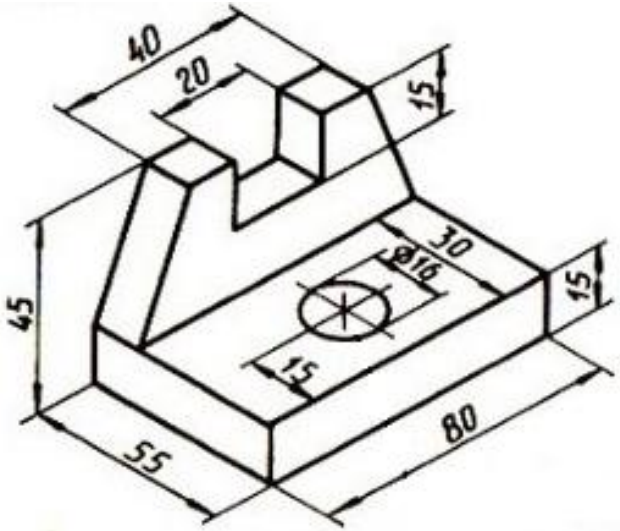
Option 7



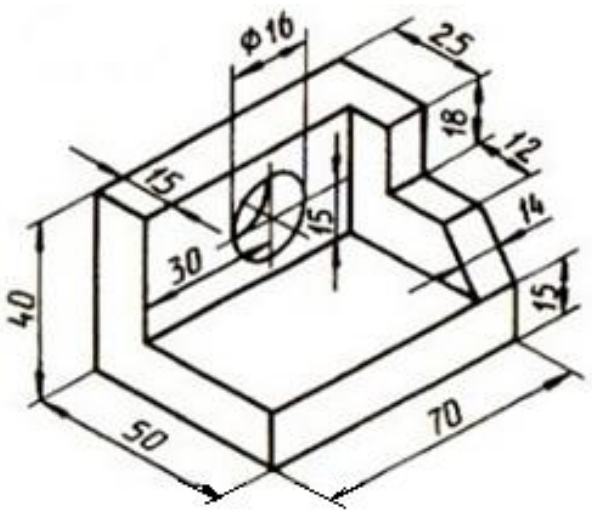
Option 8



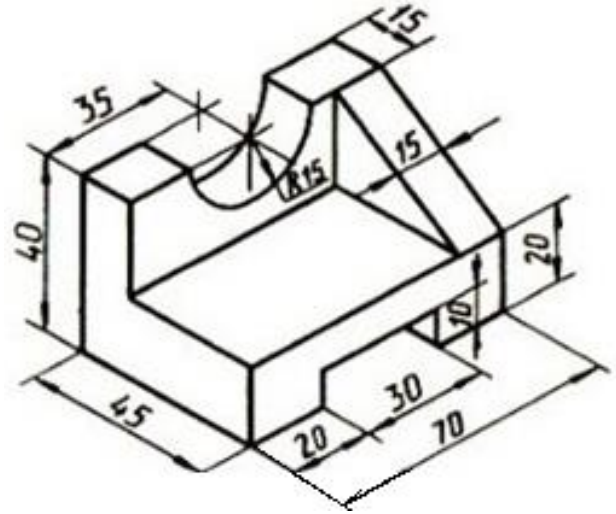
Option 9



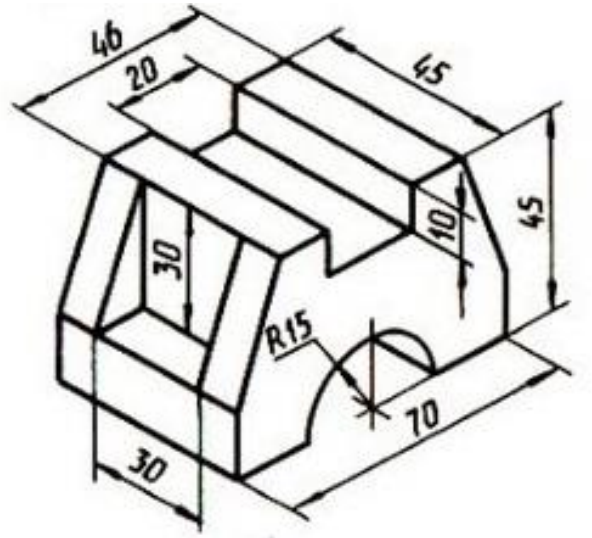
Option 10



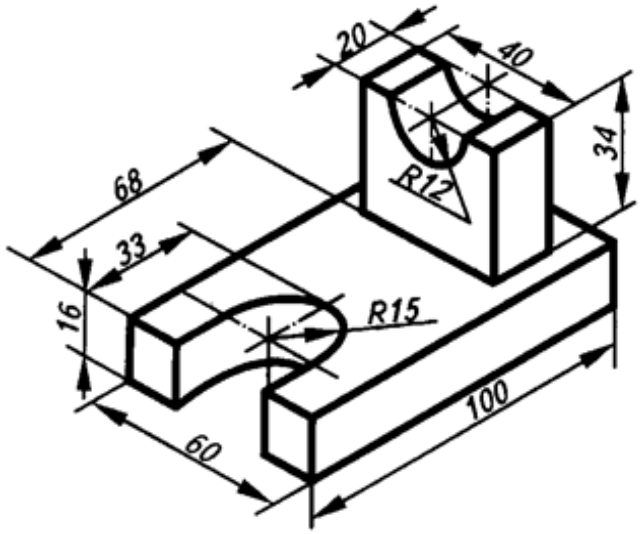
Option 11



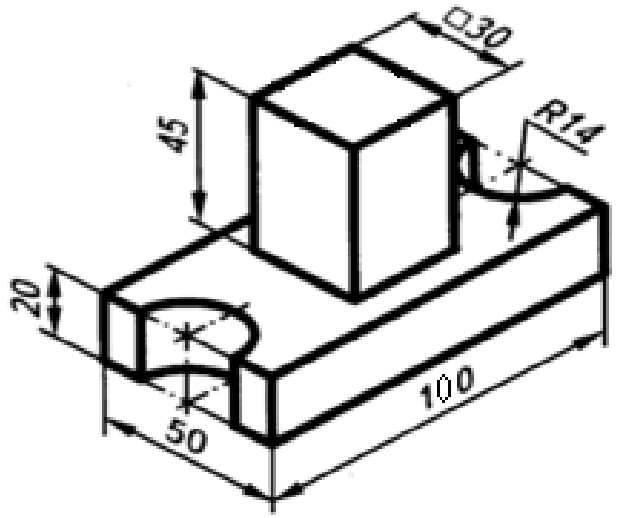
Option 12



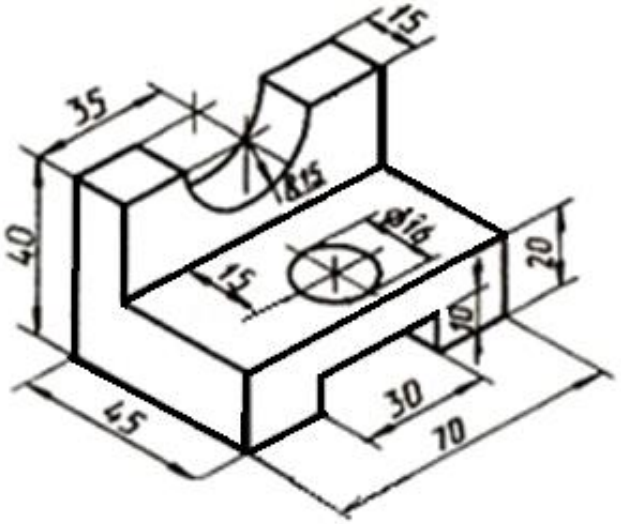
Option 13



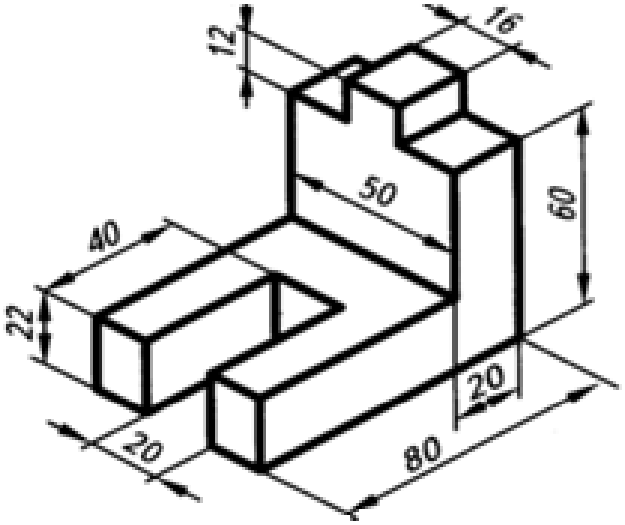
Option 14



Option 15



Option 16



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ENGINEERING 0COMPUTER GRAPHICS IN THE SPHERE OF AUTOCAD. PRACTICE

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