Content

Introduction

The level of acceleration of scientific and technological progress of society is determined by the rate of growth in labor productivity of developers of new products, reduction of design deadlines, and improvement of the quality of project development.

Computer-aided design (CAD) systems make it possible to develop new methods for designing complex systems and objects based on the latest achievements in fundamental sciences.

Currently, various CAD systems have been created and are in use. Some of them solve the task of ensuring the automation of various procedures and operations, such as: preparation of text documentation, integrated solutions for the automation of design and engineering and technological work, construction of graphic images, communication devices between a computer and a CNC machine, and automated calculation systems. Others represent entire systems that perform narrowly focused production management tasks - project management systems and electronic archives at enterprises, design documentation management systems for departments and work groups.

This course introduces the capabilities of the automated two-dimensional solid-state design modeling system in the form of a CAD drafting and design system. This system allows you to conveniently and quickly develop various drafting and design documentation, create specifications and other text-graphic documents. CAD has built-in interactive parameterization of drawings; is equipped with application libraries of standard structural elements with databases; supports the creation of your own libraries and frequently used elements; supports DWG, DXF, IGES, PDF 3 formats.

Methodological guidelines are used by students of specialty 133 "Industrial Mechanical Engineering" of the educational program "Transport and Technological Machines and Equipment" of all forms of study.

Purpose and procedure for implementation

Studying in this CAD course involves active student work in conducting laboratory work and completing an individual task by each student.

In laboratory work, students get acquainted with the work of the AutoCAD software product, study the purpose and functions of the program components, and consolidate the material considered by performing fragments of exercises.

When conducting laboratory work, the student, using the knowledge gained, performs working drawings of various parts. The complexity of the drawings performed increases as the computer-aided design system is studied, and in conclusion, the student performs a drawing of the part.

Requirements for work design

A report on the completion of laboratory work is prepared by each student separately. The report must contain:

- laboratory work assignments;
- the purpose of performing laboratory work;
- electronic version of the completed work.

When performing laboratory work, it is possible to replace the task with individual work.

The defense of laboratory work reports is carried out both during the work and during additional consultations. Each student receives control questions and a grade for the performance of laboratory work separately. The grade is entered in the journal of laboratory work performance and is credited to students when issuing the semester final grade for this discipline.

Laboratory work No. 1

Software product features. AutoCAD installation. Getting to know the program settings. Interface

Purpose of work– familiarize yourself with the characteristics, installation and configuration of the software product "AutoCAD Mechanical" from Autodesk.

Task:

- Get to know the characteristics;
- Software product installation;
- Get acquainted with the software product interface.

Introductory information

AutoCAD Mechanical is a specialized software product, part of the AutoCAD family, designed for creating and designing mechanical systems and components. It is aimed at engineers and designers working in the field of mechanical design and offers numerous features to automate and simplify design processes. Here are the main purposes of AutoCAD Mechanical:

1. **Design of mechanical parts and assemblies**: AutoCAD Mechanical includes powerful tools for creating accurate drawings of mechanical elements – from simple parts to complex assemblies.
2. **Standardization**: The program has built-in libraries of standard mechanical components (bolts, nuts, gears, connections, etc.), which significantly speeds up the design process and ensures compliance with standards.
3. **Drawing automation**: AutoCAD Mechanical automates numerous tasks, such as generating parts, creating sections, applying dimensions, and automatically updating drawings when changes are made to the model.
4. **Documentation processing and management**: The program supports the creation and editing of technical documentation, taking into account all the necessary norms and standards. It allows you to generate reports, bills of materials (BOM) and other related documents.

5. **Integration with other Autodesk products**: AutoCAD Mechanical can be used in conjunction with other Autodesk programs, which provides convenient collaboration and data exchange within a large project.
6. **3D modeling support**: Although the main purpose of the program is 2D drawing, it also supports 3D modeling of mechanical elements and helps in the visualization of complex mechanisms.

Software product installation

The process of registering on the Autodesk website and installing Autodesk AutoCAD software can be a bit complicated for beginners. Below is a detailed step-by-step guide to help you successfully complete the process.

Follow the procedure according to the points below.

1. Open the Autodesk website, enter the address Autodesk Education Community Resources | Autodesk in your web browser. Click the Sign in button or https://accounts.autodesk.com/Authentication/, located in the upper right corner of the page.

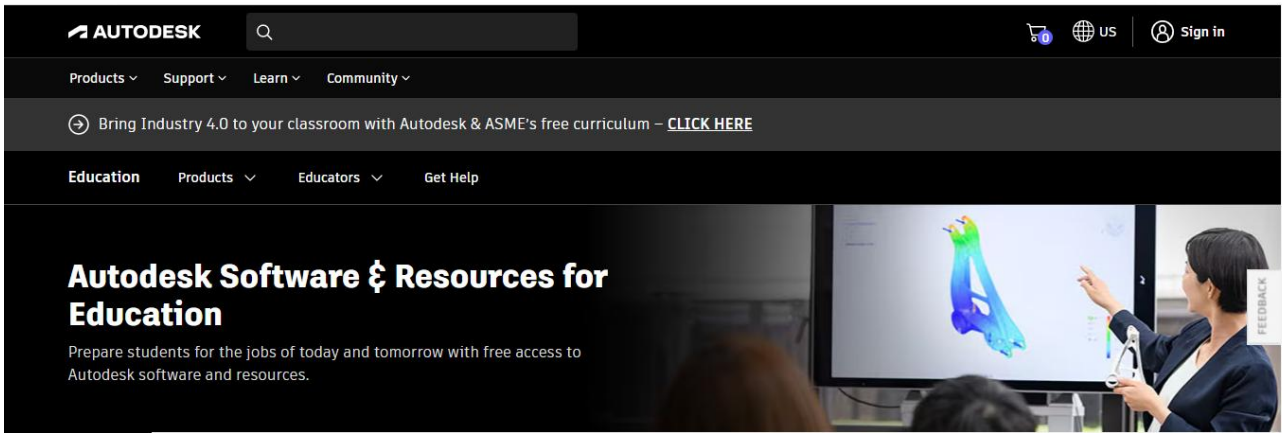


Figure 1.1 – Window of the official Autodesk website

2. Select the "Create Account" option that appears on the screen. After that, enter your personal data (first name, last name, email address and password). When creating a password, you must adhere to the requirements for length, complexity and security.

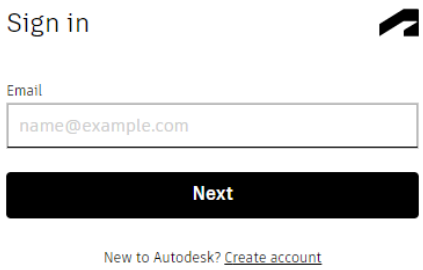


Figure 1.2 – Authorization window

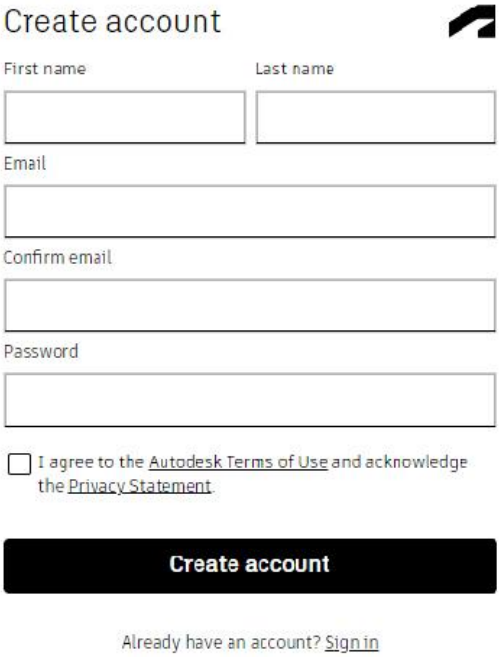


Figure 1.3 – Account creation window

3. Verify your account via the email sent by Autodesk. To do this, go to your email and open the email from Autodesk. Follow the instructions to verify your account.

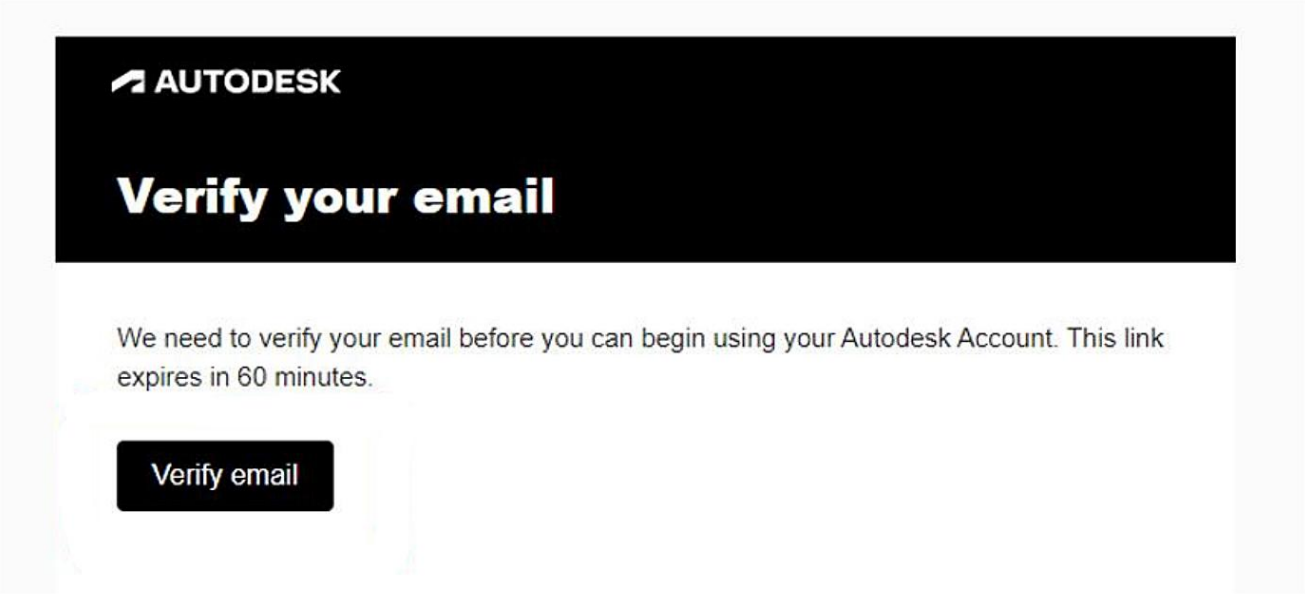


Figure 1.4 – Email confirmation window

4. After you have verified your account, return to the Autodesk website https://accounts.autodesk.com/Authentication/ and enter your login details.

5. On the Autodesk Education Community Get Products page, click the Get educational access button.

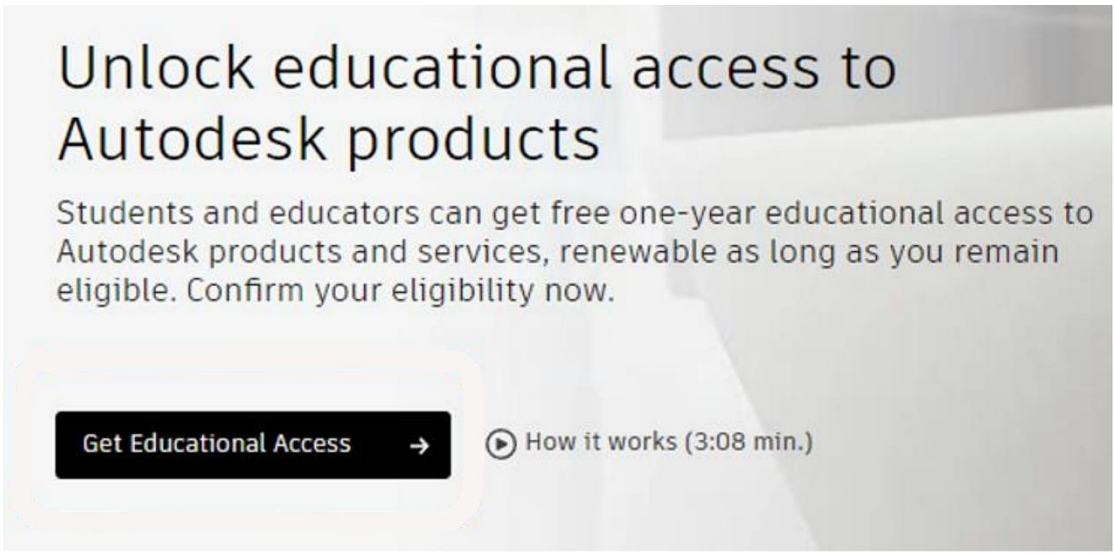


Figure 1.5 – Learning access window

6. Follow the on-screen instructions to enter basic information about your educational tab and yourself. When prompted to select an educational role, select one of the options below.

– Select “Student” to use the software on your personal device.

– Select “Teacher” if you plan to use this software with students.

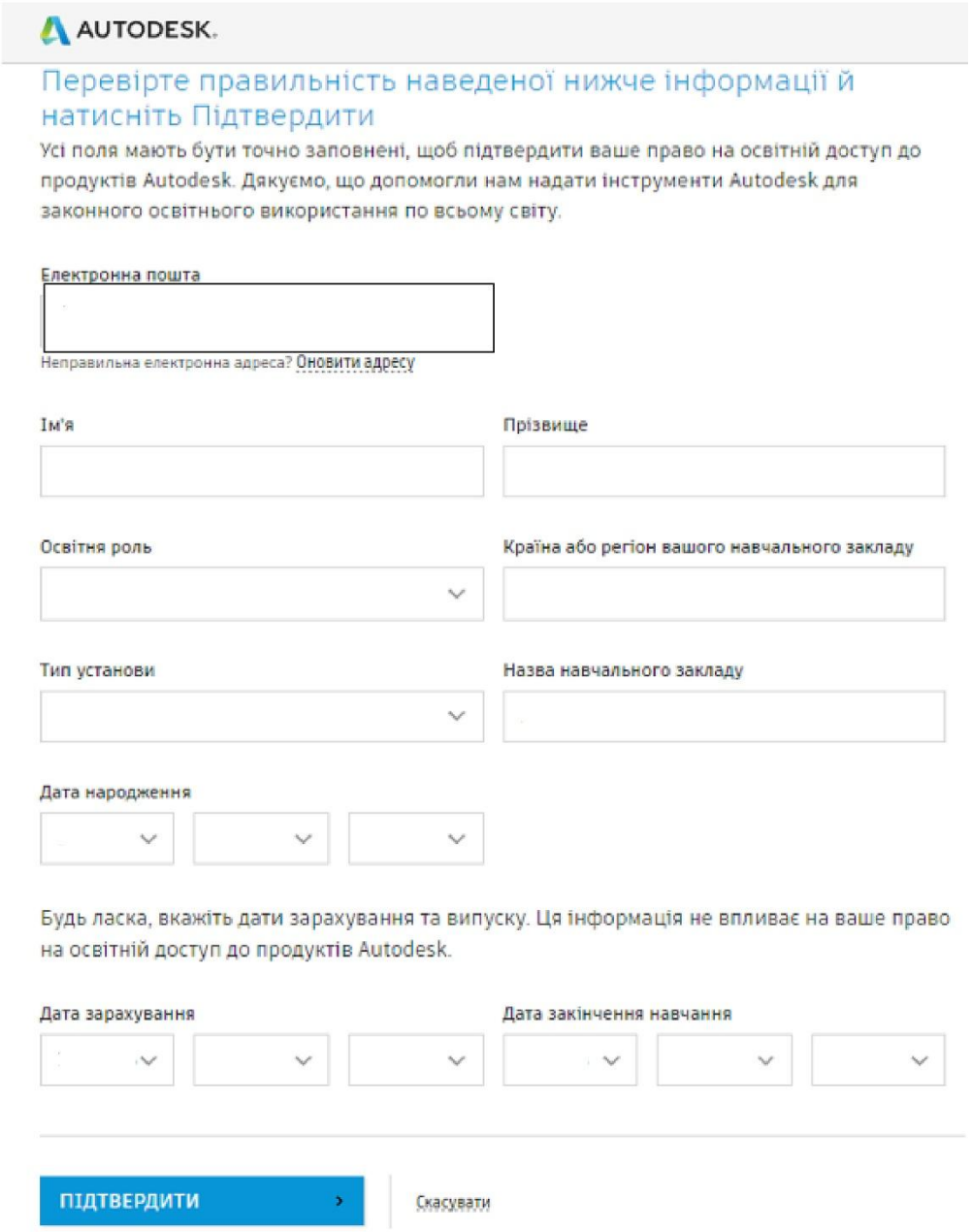


Figure 1.6 – Personal data entry window

– Upload a copy of documentation issued to you by your educational institution (e.g., tuition receipt or student ID, employee ID) as proof that you attend, teach, or work at a qualified educational institution.

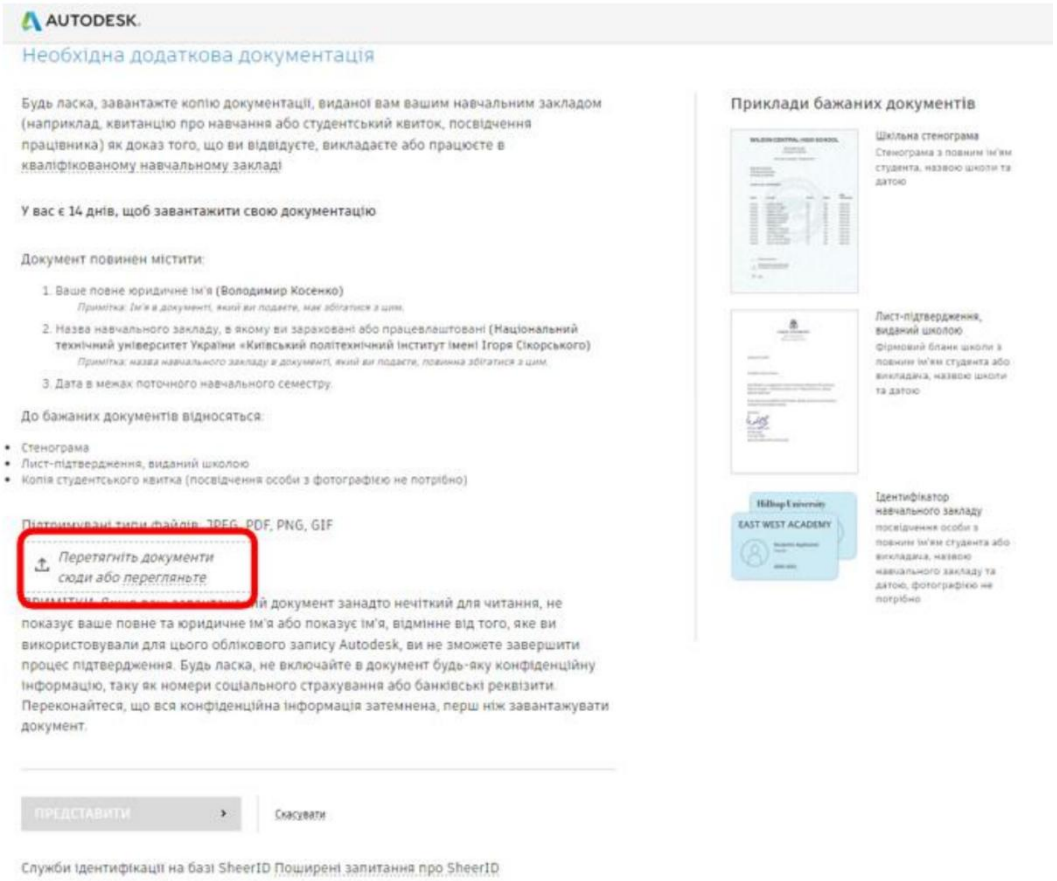
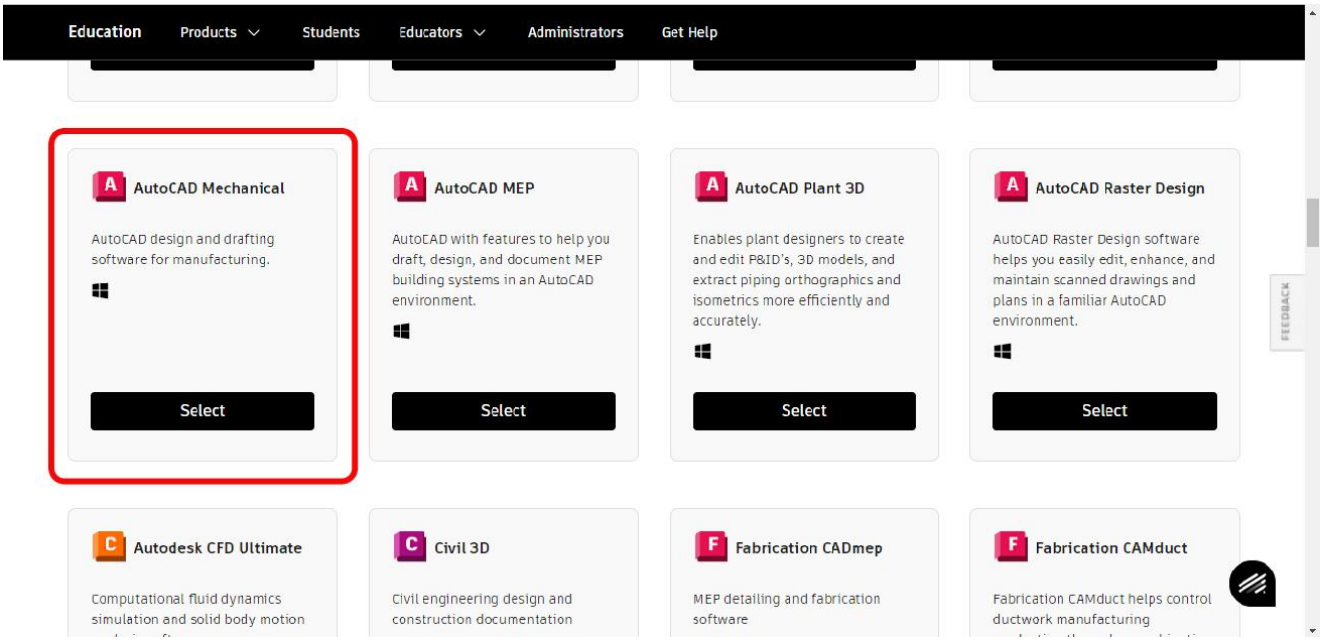


Figure 1.7 – Window for attaching documents confirming personal data

– Once your account details have been submitted and you receive confirmation of the validity of the document (this may take up to 2 weeks), go to the Autodesk Education & Student Access | Autodesk page (Figure 1.8) or click on the button in the top menu on the Autodesk website (Figure 1.9)

Support>Downloads>Students and educators, then select "AutoCAD Mechanical" from the product list. Click "Select" under the program (Figure 1.8).



– Click “Select” under “AutoCAD Mechanical” from the Students and educators page. To do this, click the button in the top menu on the Autodesk Support>Downloads>Students and educators website, and then select “AutoCAD Mechanical” from the product list. Go to the download page and click the “Download” button to begin downloading the application.

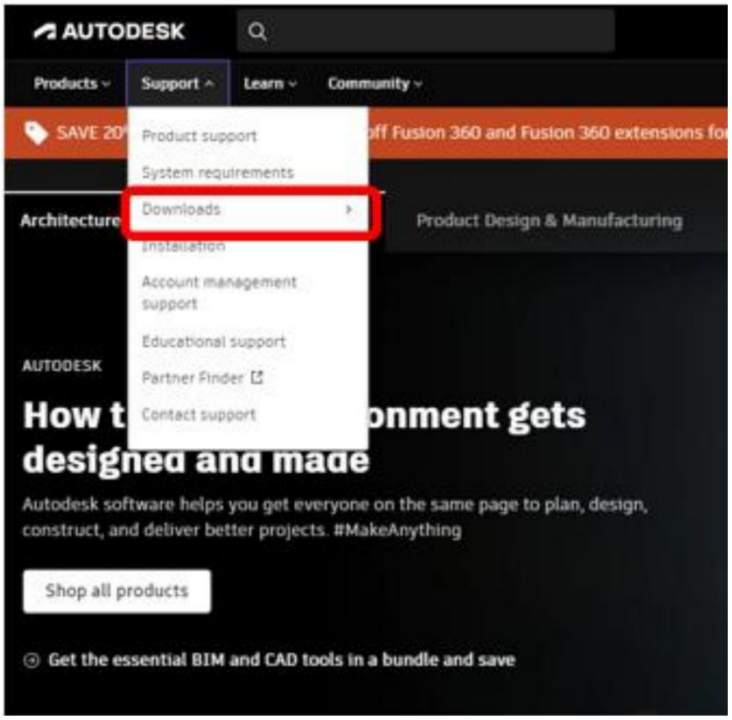


Figure 1.9 – Software product download window

–Select the appropriate version of the program for your operating system. For example, if you are running Windows, select the software version "2024", and the language.

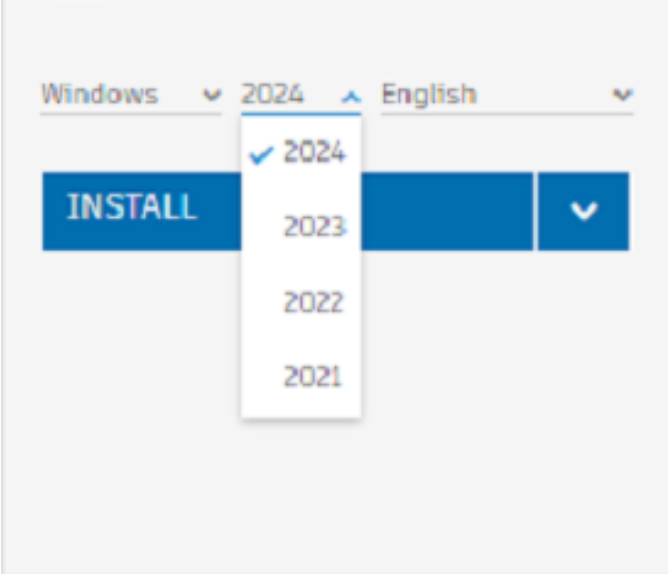


Figure 1.10 – Operating system, language and version selection window

– Wait for the program to download and start installing it.

– Run the program (Fig. 1.11)

Figure 1.11 – Program shortcut

Getting to know the software product interface

When you start AutoCAD Mechanical, you are presented with either a Getting Started dialog box or a blank drawing window. In the latter case, you begin working on a drawing using the default AutoCAD Mechanical settings and the units of measurement selected in the previous session.

After creating a new drawing, a graphical editor window will appear on the screen, the standard view of which is shown in Fig. 1.12. The location of most elements in the window and their shape can be changed.

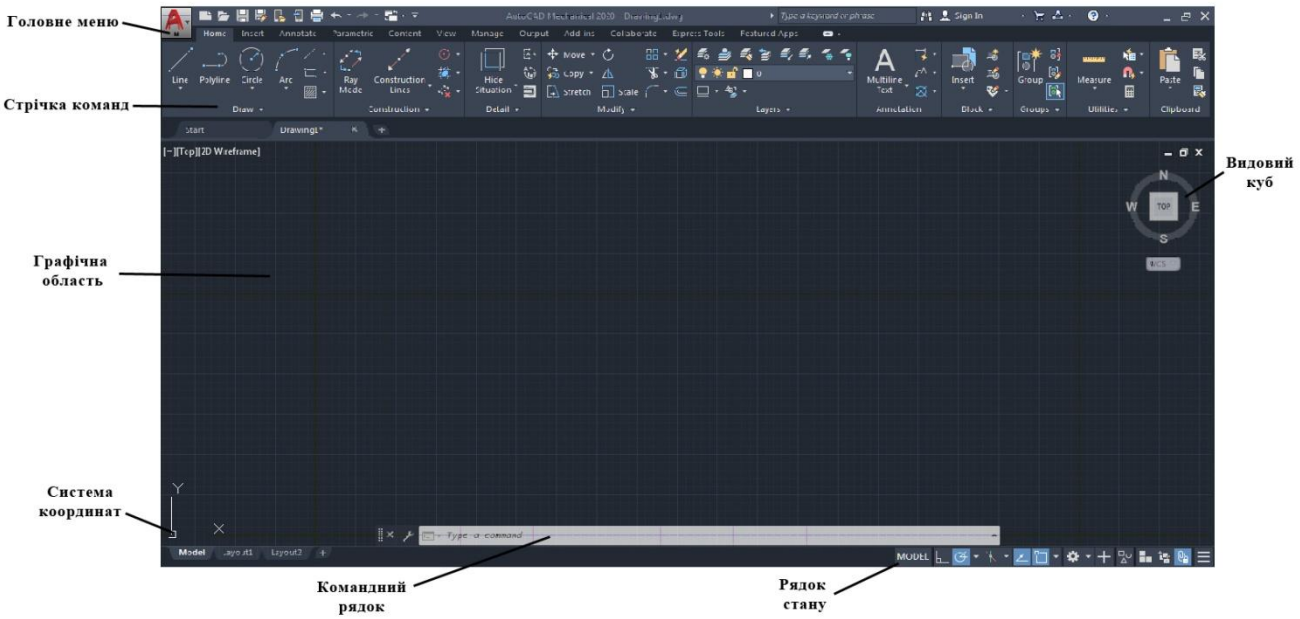


Figure 1.12 – AutoCAD Mechanical window

The central part of the window is called the graphics or work area. It displays the objects that make up the drawing. When the mouse cursor is in this area, it looks like a cross with a small square in the middle. The size of the cursor crosses can be changed in the settings or using a system variable. The coordinate system icon is located in the lower left corner of the graphics area. To ensure that the cursor is guaranteed to hit the desired point of the graphic object, there is a certain mechanism. So, when the cursor is in the area of the point marker, it seems to stick to it until the user moves the mouse to

the side. At this time, the characteristic point is highlighted with a green square, and in the status bar you can see the coordinates of this point.

Directly in the title bar is the main program menu and a quick command bar that can be customized by the user.

All the main AutoCAD Mechanical commands are grouped into several ribbons according to their purpose. Their display is contextual and depends on the current command or the type of selected objects and can be edited by the user. If you move the mouse cursor to any of the buttons on the command ribbon, after a while a tooltip with the name and description of the command will appear near the mouse cursor. Buttons with a triangle image display sub-menus containing additional commands related to a common functional purpose.

AutoCAD is a program that has been around for about 20 years. In this regard, many elements that were relevant in the past have now partially or completely lost their relevance, but have remained in the program interface. As an example, we can cite the command line, which was in demand in the days when the computer did not have a mouse. It is gradually losing its relevance, but it is still impossible to completely abandon it.

*Command line*is located at the bottom of the AutoCAD window. In this
The command line displays commands entered by the user. The command line is a means of dialogue between the user and the program. When the user invokes a command (from the main program menu or by clicking on the corresponding toolbar buttons), the command line automatically displays the name of the invoked command.

Almost every command has certain parameters or additional information to be entered (for example, coordinates of points). In this case, the user must read the question that appears on the command line and answer it. Until this moment, you cannot start executing a new command and exit the program, otherwise this command will be interrupted.

Full information about user actions and system messages for the current session can be obtained in the text window that opens when you press the "F2" key (pressing

this key again closes the window). Using the text window, you can repeat any command or sequence of commands that was executed previously.

The status bar is located at the bottom of the AutoCAD window. It displays the current coordinates of the graphics cursor and also contains

a row of buttons designed to set drawing modes. There are ten of these buttons on the panel by default, but any of them can be removed or added to the panel. They work like switches - the image of a pressed button corresponds to the on state of the corresponding mode, and the image of an unpressed button corresponds to the off state. The functional purpose of the buttons in the order of their location:

(MODEL/PAPER) – used to switch between model space and paper space.

(GRID) – enables or disables the display of the background grid on the screen.

auxiliary grid of points. This button corresponds to the function key "F7".

(SNAP) – sets the step snap parameters, i.e. controls the mode

snapping to grid points with a certain step - "F9".

(ORTHO) – turns on or off the orthogonal drawing mode, when

which the system draws lines only parallel to the coordinate axes - "F8".

(ISOORTHO) – turns on or off the isometric coordinate plane mode and provides switching between them - "F5".

(POLAR) – enables or disables the polar tracking mode, in which the system displays temporary auxiliary infinite straight lines (alignment lines) on the screen, directed at angles that are multiples of the angle specified by the user - "F10".

(OTRACK) – enables or disables object snap tracking mode, which allows you to accurately place new objects relative to

intermediate point specified using object snapping - "F11".

(OSNAP) – enables or disables the object snap mode, which allows the user to select characteristic points during object editing - F3.

(LWDISPLAY) – enables or disables the display of line weights (thickness)

on the monitor screen.

Next are buttons for managing annotative drawing objects, switching workspace settings profiles, and clearing the screen.

INAutoCAD makes extensive use of context menus, which are activated by clicking the right mouse button. The contents of the context menu depend on which command or dialog box is active at the time it is selected.

call.

Review questions

1. List the capabilities of the AutoCAD system.
2. Explain the importance of installing a licensed version of a software product
3. Features of the AutoCAD interface. Define the "Command Ribbon".
4. How to create a new file?
5. Define "Coordinate system".

Laboratory work No. 2

Introduction to AutoCAD

Purpose of work– learn how to perform basic drawing annotation in the AutoCAD Mechanical software product.

Task:

- Get acquainted with the AutoCAD Mechanical package shell;
- Be able to choose a drawing format and make initial settings;
- Draw the frame of the main inscription using the “Pline”, “Line”, “Offset” commands;
- Adjust layers, fonts, and sizes;
- Sign the main inscription;
- Save the file (file type - .dwg);
- Run the appropriate report.

General information

The rules for designing a specific design document are determined by its specifics and the provisions of the relevant standards. Let us first consider the general rules for designing drawings that apply to documentation in all industries.

Drawings and other design documents are executed in formats defined by GOST 2.301-68. The main formats and their symbols are given in Table 2.1.

Table 2.1 – Dimensions of main formats, mm

Позначення формату	Формат
A0	841x1189
A1	594x841
A2	420x594
A3	297x420
A4	210x297

It is allowed to use additional formats, which are formed by increasing the smaller sides of the main formats by a multiple of their sizes. The sizes of additional formats should be selected from Table 2.2.

The designation of additional formats consists of the designation of the main format and its multiplicity in accordance with Table 2.2. For example, the format 297x1261 is designated A4x6.

An inner frame is made on the format (Fig. 2.1, Frame.doc), and the main inscription is placed in the lower right corner (DSTU GOST 2.104:2006). On A4 sheets, according to GOST 2.301-68, the main inscriptions are placed only along the short side of the format.

Table 2.1 – Dimensions of additional formats, mm

Кратність	Формат				
	A0	A1	A2	A3	A4
2	1198x1682	-	-	-	-
3	1189x2523	841x1783	594x1261	420x891	297x630
4	-	841x2378	594x1682	420x1189	297x841
5	-	-	594x2102	420x1486	297x1051
6	-	-	-	420x1783	297x1261
7	-	-	-	420x2080	297x1471
8	-	-	-	-	297x1682
9	-	-	-	-	297x1892

Figure 2.1 shows the basic caption for drawings.

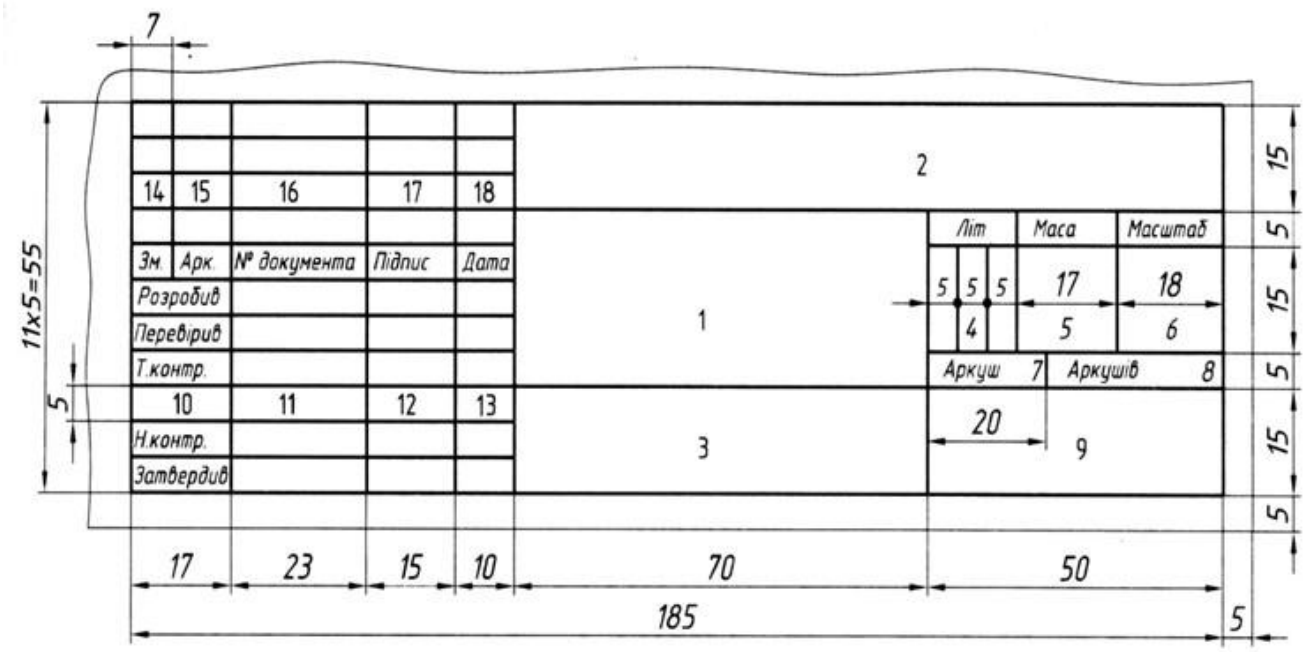


Figure 2.1 – Main inscription

The columns of the main inscription indicate:

1) pos. 1 – the name of the product, starting with a noun, and the name of the document, if it is assigned a code. For example, for a working drawing of a part - “Drive wheel”;

2) item 2 – document designation in accordance with GOST 2.201-80;

3) item 3 – designation of the part material (the column is filled in only on the part drawings);

4) item 4 – the letter assigned to this document in accordance with GOST 2.103-68;

5) item 5 – mass of the product in accordance with GOST 2.109-73. The column is not filled in on training drawings;

6) item 6 – scale (set in accordance with GOST 2.302-68 and GOST 2.109-73);

7) item 7 – serial number of the sheet (on documents consisting of one sheet, the column is not filled in);

8) item 8 – the total number of sheets of the document (the column is filled in only on the first sheet);

9) item 9 – name or code of the organization issuing the document (the column is not filled in if the code is contained in the document designation);

10) item 10 – the nature of the work performed by the person signing the document;

11) item 11 – surname of the person who signed the document;

12) item 12 – signature of the person whose surname is indicated in item 11;

13) item 13 – date of signing the document;

14) positions 14-18 record data on changes made to the document in accordance with the requirements of GOST 2.503-90.

According to DSTU GOST 2.104:2006, additional columns are used to the main inscription, in which details are entered regarding approval, copying of the document, etc. The location and size of some other additional columns can be set by the document developer.

Procedure for performing laboratory work

1) Create a folder with the group name (in English letters), inside the folder, for example MIT-1224a;

2. Download AutoCAD Mechanical.

3. Set the drawing format. To do this, enter the "Limits" command in the AutoCAD command line and set the coordinates for the A3 format: the lower left point is (0,0), the upper right point is (420, 297).

4. Bind the drawing format to the editing field using the "Zoom" command, selecting the "All" option.

5. In the bottom line of the editor, find the buttons: “Snap”, “Grid” and “Ortho” and activate them by clicking on them.

6. The “Snap” command sets the movement step – right mouse button, Setting option – set to “1”.

7. The “Grid” command displays a grid (millimeter scale) – right-click the Setting option – set “5”.

8. The "Ortho" command allows you to draw lines at right angles.

9. We proceed to drawing the outer frame (according to the dimensions given above). Select the “Polyline” button from the toolbar (keyboard equivalent – Pline).

10. Drawing of the main inscription (according to the dimensions given above).

The construction commands "Pline", "Line" and the editing commands "Copy", "Offset" are used.

11. In the "TextStyle" tab, set the style. In the Font Name list, select the font "ISOCPEUR", set its height to Height 2.5 - 7 mm.

12. Use the “MText” command to sign the graphs.

13. Saving the drawing – main menu items Save or (Save as).

Review questions

1. Explain the main titles of the main inscription.
2. Explain the meaning of the "Limits" command.
3. Explain the meaning of the "Snap", "Grid" and "Ortho" commands.
4. How to change the "line weight"?

Laboratory work No. 3

Geometric primitives

Purpose of work– learn to work with commands for constructing geometric primitives.

Task

- Be able to construct a line according to given parameters;
- Be able to construct a rectangle according to given parameters;
- Be able to construct an arc according to given parameters;
- Be able to construct a circle according to given parameters;
- Be able to construct an ellipse according to given parameters;
- Be able to construct a polygon according to given parameters;
- Be able to perform constructions using bindings;
- Construct a polyline according to the specified option.

General information:

Line Team – was considered in LR No. 2.

Rectangle command – builds a polyline in the form of a rectangle with sides parallel to the X and Y axes of the current coordinate system. After starting it, the system will display a request and a list of additional options: Specify first corner point or [Chamfer/ Elevation/ Fillet/ Thickness/ Width]:. By default, the construction is carried out according to two diagonally opposite vertices of the rectangle, which are indicated respectively in the requests: Specify first corner point: and Specify other corner point or [Dimensions]:. The Dimensions option, offered as an alternative to the opposite vertex, allows you to build a rectangle according to the lengths of its sides.

After selecting it, the system will sequentially prompt you for the length of the rectangle (Specify length for rectangles:), its width (Specify width for rectangles:), and then for entering a point (Specify other corner point:). Specifying a point in this case is only necessary to determine the orientation of the rectangle relative to its first point.

The Arc command implements 11 ways to construct an arc, depending on the selection and combination of options. Among them: by start point, center and end point; by start point, center and angle; by start and end points and radius, and others.

Options: 3 Points – by three points; Start – starting point of the arc; Center – the center point of the arc; End – the end point of the arc; Angle – the central angle of the arc; Length – the length of the chord; Direction – the direction of the tangent (indicated by a single point and coincides with the vector drawn to this point from the starting point); Radius – the radius of the arc.

The Circle command is designed to draw circles and provides the user with a number of options that provide different ways of drawing. After running the command, the query Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: is displayed, in response to which you need to specify the center point of the circle or select an option.

Options:

ZR – builds a circle from three points, which must be specified according to the requests: Specify first point on circle:, Specify second point on circle:, Specify third point on circle:

2P – determines the diameter of a circle based on two given points.

Ttr – constructs a circle of a given radius, tangent to two graphic elements (they can be lines, arcs or circles). First, you need to select the tangents (in the queries Specify point on object for first tangent of circle: and Specify point on object for second tangent of circle:),

and then specify the radius.

Center point – center point. After selecting it, the query Specify radius of circle or [Diameter]: is displayed, in response to which you must specify the value of the circle radius or select the option

Diameter to then specify the diameter value.

The Circle toolbar submenu contains the Tan, Tan, Tan option. This option corresponds to the 3P option when the Tangent snap mode is enabled. When selected, AutoCAD draws a circle tangent to three graphic elements.

If the command is selected from the toolbar, the following circle drawing options are offered:

Center, Radius – by center and radius;

Center, Diameter – by center and diameter;

2 Points – by two points;

3 Points – by three points;

Tan, Tan, Radius – by two tangents and radius;

Tan, Tan, Tan – by three tangents.

The Ellipse command is designed to construct ellipses and elliptical arcs. The construction is carried out along two axes or along the center and radius of an isometric circle, if the isometric snap mode is set. After starting the command, the query Specify axis endpoint of ellipse or [Arc/Center]: is displayed, in response to which you must specify the endpoint of the ellipse axis or select another option.

Options:

Axis endpoint – defines the first axis of the ellipse by its two endpoints. This option is used by default. After the endpoint of the ellipse axis is specified, AutoCAD will prompt for the second endpoint of this axis: Specify other endpoint of axis:. Next, in response to the Specify distance to other axis or [Rotation]: prompt, you need to specify the second axis of the ellipse. By default, the second axis is set to a distance that is half its length. The value of this distance can be entered from the keyboard or specified with the cursor.

Rotation – allows you to build an ellipse as a projection onto the drawing plane of a circle rotating around a diameter defined by the previously specified points. The range of permissible angles is from 0 to 89.4 degrees.

Center – the central point of the ellipse. After selecting it (in response to the Specify center of ellipse: prompt), the Specify endpoint of axis: prompt appears, where you need to specify the endpoint of the axis, and then the already known prompt: Specify distance to other axis or [Rotation]:

Arc – allows you to construct an elliptical arc. If you select this option, you will be prompted to specify the axis endpoint or center of the elliptical arc (Specify axis endpoint of elliptical arc or [Center]:), which requires the same information as when constructing a complete ellipse. After answering the sequence of prompts for a

complete ellipse, you will be prompted to specify the starting (Specify start angle or [Parameter]:) and ending (Specify end angle or [Parameter/Included angle]:) angles of the arc. Angle values can be entered from the keyboard or by specifying points. When entering values from the keyboard, keep in mind that the zero angle corresponds to from the center of the ellipse to the endpoint of its major axis. Selecting the Included angle option allows you to specify the central angle of the arc, measured relative to its starting angle, instead of the ending angle. Constructing an elliptical arc using the Parameter option requires the same input as constructing it using the starting and ending angles. However, the arc is constructed according to the following parametric equation:

$$p(u) = c + a \cdot \cos(u) + b \cdot \sin(u)$$

where c is the center of the ellipse, a and b are the lengths of the major and minor axes, respectively.

The Polygon command builds a regular polygon with sides from 3 to 1024. After entering it, the Enter number of sides: prompt appears, where you need to specify the number of sides of the polygon. The next prompt Specify center of polygon or [Edge]: requires specifying the center or selecting an option. If you specify the center point of the polygon in response to it, then options will be offered that determine the method of its construction, namely:

Inscribed in circle – a polygon is constructed as if inscribed in a certain circle;

Circumscribed about circle – the polygon is constructed as if circumscribed around a certain circle.

After determining the construction method, it is necessary to specify the radius of the circle in response to the query Specify radius of circle:. If the radius is specified by entering its numerical value, then the orientation of the lower side of the polygon coincides with the orientation of the grid for fixed cursor movement. Most often, this is 0. If the radius is specified using the cursor, then for an inscribed polygon, one of its vertices will coincide with the specified point, and for a circumscribed polygon, the midpoint of one of the sides will coincide.

The Edge option allows you to construct a polygon based on the position of one of its edges by specifying its starting (Specify first endpoint of edge:) and ending (Specify second endpoint of edge:) points. AutoCAD constructs the polygon by creating a circular array of the specified edges. The construction is done counterclockwise.

Join command – allows you to combine multiple segments, arcs, or polylines into one polyline.

After running the command, you need to select the lines that will be joined into a polyline. If the command is successful, the message N segments joined into 1 polyline (if there was already a polyline among the objects to be joined) or N objects converted to 1 polyline (if only segments and arcs were joined) will appear, where N is the number of objects that were selected. For the Join command to be successful, the ends of the lines must coincide. If there is a gap between the ends of the lines, the joining will not occur, and the message N objects discarded from the operation will appear in the command line. Therefore, it is necessary to use snaps or exact coordinate values when constructing lines to which the Join command will be applied in the future.

The Pline and Osnap commands are discussed in LR No. 2.

Procedure for performing laboratory work

Follow the procedure according to the points.

1. Download AutoCAD.

2. Download the file created on LR No. 2 and save it with a new name, for example, lr3_1.dwg.

3. Perform constructions according to the option given in Appendix B (see task for LR No. 3, part one).

4. Construct a segment from the center of the ellipse tangent to the circle using snaps.

5. Construct a segment from the middle of the segment constructed in point 3 to the intersection point

the segment constructed in point 4 and the ellipse, using the constraints.

6. Save the file.

7. Download the file created on LR No. 2 and save it with a new name, for example, lr3_2.dwg.

8. Perform construction with a polyline according to the given option in Appendix B (see task for LR No. 3, part two). The contour is performed with one polyline using auxiliary lines.

Review questions

1. How do you understand the concept of "Geometric primitives"?
2. Define the "polyline" command?
3. What does the “Edge” option allow you to build?
4. The main function of the "Circle" team?

Laboratory work No. 4

Conjugation

Purpose of work– learn how to perform object conjugation in AutoCAD software.

Task

- Be able to create layers and set object properties;
- Be able to use commands for constructing geometric primitives;
- Be able to use object editing commands;
- Be able to perform conjugations;
- Be able to set linear and radial dimensions.

General information

Creating drawing layers.Using layers is the best way to organize your drawings. Layers provide the user with powerful tools for grouping and selecting graphical objects in a drawing.

Layer command – opens the drawing layer properties window. Instead of the command, you can use the Layer Properties button from the toolbar. Layer properties are presented in the form of a table, the columns of which have the following values:

Status – if there is a “tick” in the column, then the layer is current, otherwise not. Only one layer can be current. When constructing graphic objects, they will be located on the current layer.

Name – layer name.

On/Off – enabled layers are visible by default. Disabled layers are invisible, but participate in the drawing regeneration process. If the layer is On, the “light bulb” is on, otherwise it is not. If the current layer is disabled, all subsequent constructions will not be displayed on the screen until the layer is enabled. For this reason, a warning window will be displayed before disabling the current layer.

Freeze – thawed layers are visible by default. Frozen – invisible and cannot be edited, and do not participate in the drawing regeneration process. If this option is a "snowflake", the layer is frozen, if it is a "sun", it is thawed.

Lock – unlocked layers are visible and editable by default. Locked layers are not editable, but they are visible. You can also use snaps to objects on a locked layer, but you cannot modify these objects. An unlocked layer has a "lock" open, otherwise it is locked.

Color – layer color. The color is set per layer for all its objects. To use the layer color in the properties of graphic objects, a value must be specified instead of color

ByLayer (default value). If you change the color in the object properties, the object color will remain the same, regardless of the layer color.

Linetype – line type for a layer for all its objects. When changing the line type, a window opens in which only one line type is initially available – Continuous. You need to load the desired line type using the Load button (an additional window will open with a list of all available line types), and then select the line type for the layer.

Lineweight – this option sets the line weight for all objects on the layer.

Transparency – this option sets the transparency of the image on the layer. Possible options are from 0 to 90 (the image is not visible).

Plot Style – this option sets the print style.

Plot – allows you to display the layer during printing.

New VP Freeze – this option is used for floating view screens.

Description – in this option you can describe the layer and make notes for the user.

To create a new layer, right-click in the Layer Properties Manager window, i.e. call the context menu, and select New. Then specify the properties of the objects on this layer, i.e. color, line type, and line thickness.

The context menu provides the following options for working with layers: set the current layer, delete a non-current layer, rename a layer, change the layer description, etc.

Object editing commandsMost of the editing command buttons are located in the Modify section of the toolbar.

Execution of editing commands begins with the request Select objects: – selection of objects. There are several ways to select objects. The simplest is to place the AutoCAD cursor on the object and press the left mouse button, thus selecting one object. The selected object is marked with a dotted line. AutoCAD will repeat the request Select objects:, so you can sequentially specify the desired objects. When all the objects planned for the operation are selected, press the Enter key or click the right mouse button.

In AutoCAD, there is a possibility to select a set of objects. After starting the edit command, the Select objects: prompt can be answered with one of 16 object selection options. The most popular are:

Window – selection of objects using a rectangular frame. To make a selection, you need to specify the first corner of the frame with the left mouse button, without holding down the left mouse button, move the cursor to the opposite corner (a frame appears on the screen), specify the opposite corner of the frame with the left mouse button. If during selection with the frame the cursor moved from left to right, then only those objects that were completely inside the frame will be selected. If the cursor moved from right to left, then all objects that were at least partially inside the frame will be selected. If you hold down the left mouse button while moving the cursor, then instead of a rectangular frame, the selection area will have the shape of a polygon, which is created along the trajectory of the cursor movement.

Last – at the Select objects prompt: you should specify the letter L, meaning the last created object will be selected.

Crossing – selects objects with a frame, similar to Window. Only those objects that intersect the frame will be selected.

Undo – cancel the selection of the last object without interrupting the execution of the current command.

All – selects all objects in the active drawing layers. If objects were selected before starting the command, the Select objects: prompt does not appear after starting the command.

Let's look at some of the editing commands.

Erase command – deletes objects. After running the command, the Select objects: prompt is displayed, in which you should select objects using one of the methods described above, press the Enter key or the right mouse button.

The Copy command copies objects. At the Select objects prompt: select objects using one of the methods described above, press the Enter key or the right mouse button. Next, at the Specify base point or [Displacement mOde] <Displacement> prompt: specify the base point of the objects being copied or the displacement, which is specified by relative coordinates or specified with the mouse button. If a base point was selected, the next prompt will be Specify second point or [Array / Exit / Undo] <Exit>:, i.e. specify the second point for copying the object or an array (if an array of objects is being created), or complete the copying, or cancel the last action. As a result of the command, a copy of the existing object is created, i.e. two objects remain on the AutoCAD workspace.

The Move command moves objects. The command works similarly to the Copy command, but as a result of the operation, only one object remains on the AutoCAD workspace, which has been moved to a new position.

The Mirror command is a mirror image of objects, mainly used to build symmetrical images. That is, you can build half or a quarter of the image, and then use the command to build the full image. After starting the command, at the Select objects prompt: select the objects using one of the methods described above, press the Enter key or the right mouse button. At the Specify first point of mirror line prompt: say the first point of the reflection axis. At the Specify second point of mirror line prompt: – the second point of the reflection axis. At the Erase source objects? [Yes / No]<N> prompt: press the Enter key or the right mouse button to confirm the No answer, that is,

the original image should be saved. If you want to destroy the original image, then the Yes answer is selected.

The Rotate command – rotates the specified objects by a specified angle. After starting the command, at the Select objects prompt: select the objects using one of the methods described above, press the Enter key or the right mouse button. At the Specify base point prompt: specify the base point around which the selected objects will be rotated. At the Specify rotation angle or [Copy Reference]<0> prompt: specify the rotation angle or reference.

The Copy Reference option allows you to define an angle by referencing another angle or another object. In this case, you should use snap mode.

Trim command – used to adjust the drawing, namely when you need to trim part of an arc, circle, ellipse, line, etc. The command works as follows – first, the so-called cutting edge is specified, and then those parts of the objects that should be trimmed. On request

Select objects or <select all>: specify the cutting edge and press the Enter key or the right mouse button. At the Fence Crossing prompt [Project / Edge / eRase / Undo]: specify the parts of the objects that should be cut. If you select the Project option, you are supposed to work with the projection. The Edge option is an edge, used to continue the cutting edge. The eRase option is used to delete the object. The Undo option is used to cancel the previous action.

The Break command is a break command. It is often easier to build one object and then break it into several parts than to build these individual parts. This command is used for this. At the Select objects prompt: specify the first point from which the break begins and press the Enter key or the right mouse button. The points are set counterclockwise. Then at the Specify second break point or [First point] prompt: specify the second point. For the command to work, the snap mode must be enabled. If you select the First point option, the first point is specified, then the request for the second point is made, i.e. the command continues.

The Fillet command is used to construct an arc that smoothly joins two segments (lines or arcs). Joining is performed in two stages. First, the join radius is specified. At the Specify first object or [Undo / Polyline / Radius / Trim / Multiple] prompt, select Radius and specify the join radius. Then, at the same Specify first object or [Undo / Polyline / Radius / Trim / Multiple] prompt, specify the first join object. At the Select second object prompt, specify the second join object. The program will create the join. Command options:

Undo – cancels the last action.

Polyline – the selected objects are converted to a polyline.

Trim – if the mode is enabled, then the objects that are being combined are trimmed at the same time, if disabled, then the combination is performed, but the objects are not trimmed.

Multiple – repeat the previous action with other objects.

Constructing conjugations in AutoCAD. Conjugation is a smooth transition from one line to another, performed using an intermediate line. The main elements of conjugation are the radius of the conjugation arc, the center of conjugation, and the points of conjugation. The common point at which a smooth transition occurs is called the point of conjugation. A necessary condition for a smooth transition is the existence of a common tangent at the point of conjugation.

There are several methods used to construct conjugations in AutoCAD. Let's consider them.

1. Conjugation of intersecting lines using an arc of a given radius. That is, conjugation of lines that intersect at a right, acute, and obtuse angle is performed using the Fillet command.

2. Constructing an external tangent to the arcs of a circle. This conjugation is performed using the Line command with the snap mode (Tangent) enabled.

3. The conjugation of a line and a circular arc with an arc of a given radius is performed by the Fillet command.

4. Joining two circles with an arc of a given radius. If the radius is internal, then use the Circle command with the Ttr option (two tangents and a radius). If the radius is external, then use the Fillet command.

Setting the application size.Dimensioning is an important step in the development of design documentation. In AutoCAD, dimensions can be drawn and edited in many ways. Using dimension styles allows you to quickly format dimensions, ensuring their compliance with government and industry standards.

AutoCAD supports three types of dimensions: linear, radial, and angular. Dimensions can be horizontal, vertical, parallel, rotated, or ordinate; chained and common base dimensioning is supported.

Dimensions can be associative, non-associative, and disjoint.

Associative dimensions automatically change their position, orientation, and magnitude values when you edit the geometric objects associated with them.

Non-associative dimensions need to be selected and edited along with the objects they belong to. Non-associative dimensions do not change automatically when objects are changed.

Dissected dimensions are not single objects (dimension blocks), but a set of objects: arrows, lines, text.

AutoCAD places dimensions on the current layer. Each dimension is drawn according to the current dimension style, which determines properties such as the shape and size of arrows, the placement of dimension text and tolerances, and so on.

A dimension style is a named group of dimension variable settings that define the appearance of a dimension. By creating different dimension styles, you can quickly and easily set the values of the required dimension variables and control the position and appearance of the dimensions that are applied. Dimension styles can be modified, renamed, deleted, and moved to another drawing.

There are two ways to control the dimension variables that define the dimension style: by directly entering the variable name on the command line or by using the Dimension Style dialog box.

Procedure for performing laboratory work

Follow the procedure according to the points.

1. Download AutoCAD.

2. Download the .dwg file from LR No. 2.

3. Analyze the given version of the task (see Appendix B, task to LR No. 4).

4. Create three layers:

4.1. Layer1: name – axis, color – red, line type – dash-dotted, line thickness – Default. For drawing axis lines.

4.2. Layer2: name – contour, color – black, line type – solid, line thickness – 0.3. For drawing a contour.

4.3. Layer3: name – dim, color – green, line type – solid, line thickness – Default. For setting dimensions.

5. Connect Snap, Grid, Ortho.

6. Go to the axis layer. Draw axis lines.

7. Go to the contour layer. Disable Ortho. Enable snaps. Construct the contour using the geometric primitives and editing commands, and perform conjugation.

8. Go to the dim layer. Put the dimensions.

9. Save the drawing.

Review questions

1. How do you understand the concept of "Drawing Layer".

2. Assignment of the "Layer" command.

3. Define the term "Conjugation"?

4. What types of dimensions does AutoCAD support?

5. The main purpose of the "Trim" command

Laboratory work No. 5

Massif

Purpose of work– Learn how to create object arrays in AutoCAD.

Task

- Be able to create user coordinate systems;
- Be able to create arrays in AutoCAD;
- Be able to use the Divide and Measure commands.

General information

User coordinate systems – defined by specifying the initial coordinates and orientation of the X, Y, and Z planes. There are no restrictions on the position of the user coordinate system in three-dimensional space. You can define, save, and update an unlimited number of user coordinate systems. All coordinates, both when input and when output, are defined relative to the current user coordinate system. For ease of construction when working in three-dimensional space, it makes sense to specify several coordinate systems with different starting points and orientations.

Defining a new user coordinate system can be done in several ways:

1. Specifying a new coordinate point, a new X, Y plane, and a new Z axis;
2. Alignment of the user's coordinate system with an existing object.
3. Aligning the user's coordinate system with the current direction of gaze.
4. Rotate the current user coordinate system around any of the axes.
5. Shifting the initial coordinates along the Z axis by a given distance.
6. Alignment of the X, Y plane with the face of a three-dimensional object.
7. User selection of a standard coordinate system from the AutoCAD list.

All work on creating, saving, and modifying a UCS is done using UCS commands. The number of options in this command is quite large, and not all options are regularly used in practice. Some of the UCS command options can be selected directly using the corresponding icons on the UCS toolbar.

UCS command – creation of a new coordinate system. The command allows you to create a local coordinate system, the direction of the axes of which and the location of the origin of the coordinates are selected by the user. After running the command, a prompt appears on the command line:

Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del /Apply/?/ World] <World>:

In response to which you should select the desired option. Available options:

New – allows you to create a new user coordinate system. Selecting this option initiates the request: Specify origin of new UCS or [zAxis/3point /oBject/Face/View/X/Y/Z] <0,0,0>:. You can specify the origin of the new UCS (the directions of the X, Y, and Z axes will remain unchanged) or select an option.

Options in the mode of creating a new user coordinate system: zAxis – defines a new CCM by the origin point and direction of the Z-axis. In response to subsequent requests Specify new origin point <0,0,0>: and Specify point on positive portion of Z-axis:, you must first specify the new origin point, and then the point on the new positive direction of the Z-axis, respectively.

3point – creates a new CCM based on the starting point and points in the positive directions of the X and Y axes. The dialogue with the system when selecting this option will be as follows:

Specify new origin point <0,0,0>: – specify point 1.

Specify point on positive portion of Z-axis: – specify point 2.

Specify point on positive Y portion of the UCS XY plane: – specify point 3 (Fig. 13.3). oBject – aligns the coordinate system by the object. For example, if in response to the following query: Select object to align UCS: (select an object to align the UCS) select a circle, then the XY plane of the coordinate system will coincide with the plane of the circle, and the X axis will pass through the center of the circle and the selection point.

Face – aligns the SSC along the face of a 3D object. View – sets a coordinate system in which the XY plane is perpendicular to the direction of view, i.e. parallel to the screen. The origin of the coordinates remains unchanged.

X/Y/Z – when these options are selected, the coordinate system rotates around the corresponding axis by a specified angle; the required angle value is specified in response to the request: Specify rotation angle about … axis<0>:. Other UCS command options:

Move – allows you to move the origin to a specified point without changing the orientation of the XY plane. In response to the request: Specify new origin point or [Zdepth]<0,0,0>: you need to specify a new origin point or select the proposed Zdepth option (moving along the Z axis). In the latter case, the coordinate system is shifted along the Z axis by the amount specified by the user.

orthoGraphic – defines one of the six coordinate systems available in AutoCAD, corresponding to the six main views in the drawing (top, bottom, front, back, left, right). At the prompt: Enter an option [Top/Bottom/Front/Back/Left /Right]: you need to select the appropriate system name.

Prev – restores the previous coordinate system. Restore – restores a previously saved coordinate system, i.e. makes it current. At the prompt: Enter name of UCS to restore or [?]: you need to specify the name of the coordinate system or enter ? from the keyboard to get a list of names of coordinate systems created in this drawing.

Save – saves the current CCM under the specified name.

? – displays a list of coordinate systems defined in this drawing. Del – deletes the user-specified CCMs from the list of saved systems. Apply – applies the settings of the current coordinate system to the specified view screen.

? – gives a list of the names of all available coordinate systems along with their origin coordinates measured relative to the current CCM.

World – restores the global coordinate system that existed in the drawing from the moment it was created.

Creating arrays of objects. Array command – creates an array of several objects of the same type. Objects can be arranged in the array in the form of a rectangular grid, along a circle, or along an arbitrary line. The finished array of objects is considered one drawing object, if you select it, you can change the parameters of the array – the number of objects, the distance between objects, etc. If you need to edit individual objects of the array, you must first decompose it into primitives using the Explode command.

After running the Array command, a Select objects: prompt appears – select the objects from which the array will be created. Then a prompt appears to select the array type Enter array type [Rectangular/PAth/POlar] <Polar>:, in which the following options are available:

POlar – creates a circular array, further operation of the command is similar to the Arraypolar command;

Rectangular – creates a rectangular array, further operation of the command is similar to the Arrayrect command;

PAth – creates an array along a line, further operation of the command is similar to the command

Arraypath.

Arraypolar command – creating a circular array. After selecting objects, a prompt appears to select the center of the array Specify center point of array or [Base point/Axis of rotation]:. The Base point option allows you to select the base point of the object, and the Axis of rotation option allows you to create an array in space relative to the axis of rotation. After selecting the center of the array, a prompt appears Select grip to edit array or [Associative/Base point/Items/Angle between/Fill angle /ROWs/Levels/ROTate items/eXit]<eXit>: and the following options become available:

Associative – allows you to enable or disable the creation of an array as a single object;

Base point – select the base point of the object; Items – the number of elements in the array; Angle between – the angle between the elements in the array; Fill angle –

the angle of the sector within which the copies of the object will be located; ROWs – create a circular array with several rows of objects; Levels – create a circular array with several levels of objects, the levels are located in space in parallel planes. ROTate items – allows you to enable or disable the rotation of objects when creating an array;

eXit – ends the command. Arrayrect command – creates a rectangular array. After selecting objects, the prompt Select grip to edit array or [Associative /Base point/COUnt/Spacing/COLumns/Rows/Levels/eXit] <eXit> appears: and the following options become available:

Associative – allows you to enable or disable the creation of an array as a single object; Base point – select the base point of the object; COUnt – simultaneous selection of the number of rows and columns of the array; Spacing – selection of the distance between the rows and columns of the array; COLumns – number of columns in the array; Rows – number of rows in the array Levels – number of levels in the array in space; eXit – termination of the command.

The Divide command – divides a line (segment, arc, polyline) along its length into equal parts. After starting the command and selecting a line to divide, a prompt appears: Enter the number of segments or [Block]: – select the number of parts into which the line will be divided. After executing the command, the boundaries of the line parts will be marked with points. Later, you can use Node binding to the points and change the display of the points on the screen using the DDPtype command. The Block option allows you to use blocks instead of points.

The Measure command divides a line (segment, arc, polyline) into equal parts along its length, but unlike Divide, you can specify not the number of sections, but their length.

After running the command and selecting a line to divide, the Specify length of segment or [Block] prompt appears: select the distance between the two points of dividing the line.

The DDPtype command changes the display of dots on the screen. After running the command, a dialog box opens that allows you to select the symbol for displaying dots and its size.

Explode command – decomposes a complex object (polyline, block, array) into primitives.

In agreement with the teacher, select a tractor or car transmission assembly, preferably based on the topic of the course project, or a subassembly that is part of the assembly of the main assembly.

Procedure for performing laboratory work

Follow the procedure according to the points.

1. Download AutoCAD.

2. Download the .dwg file with the frame from lab #2.

3. Analyze the given version of the task (see Appendix B, task to LR No. 5).

3.1. Create your own coordinate system (Origin or 3points options), start

The coordinate should be located in the center of the part.

3.2. Perform auxiliary constructions using the Divide and Measure commands, if necessary.

4. Use the Array command to draw the drawing.

5. Set the dimensions.

6. Save the drawing

Review questions

1. Purpose of the “UCS” command?

2. Purpose of the Explode command?

3. How do you understand the concept of "Array"?

4. What constructions can be performed using the Divide and Measure commands?

Laboratory work No. 6

Projection drawing in AutoCAD

Purpose of work– learn how to perform projection drawing in AutoCAD.

Task

- Make a projection drawing.
- Learn to use object tracking.
- Learn how to hatch in AutoCAD.
- Learn to set dimensions in a chain and from a common base.

General information

Object tracking is a mechanism that works in conjunction with the object snapping mechanism, and allows the user to temporarily use characteristic drawing points to track the direction of each of them from their current position until they are next moved to another object. To use this mechanism, you need to enable orthogonal object tracking, the icon at the bottom of the editing window is Object Snap Tracking (F11).

Hatch command – hatching. After starting the command, you need to select the Settings option, which opens the hatching settings window. The hatching parameters are available in this window:

Pattern – select the hatch type by name. The hatch type is also displayed in the preview field;

Angle – hatching angle;

Scale – hatch density.

After selecting the hatching parameters, you must click the Pick points button, which allows you to select the hatching points. The hatching is applied from the selected point within the closed contour (fill). You must select one or more points, and then complete the command by pressing the Esc button. The created hatching is considered one of the drawing objects. If you select an existing hatching, the toolbar

will change, allowing you to adjust the hatching parameters. The hatching is deleted using the Erase command.

Dimbaseline command – drawing dimensions from the baseline. After starting the command, the Select base dimension: prompt appears. If a linear dimension was drawn before executing the Dimbaseline command, the specified prompt does not appear. Next, you need to specify one or more points, each of which will be a point for the second extension line of the new dimension. In this case, the first extension line coincides with the first extension line of the base dimension, and the position of the dimension line is selected automatically.

Dimcontinue command – drawing dimensions from the baseline. After starting the command, the Select continued dimension: prompt appears – selecting the dimension to continue. If a linear dimension was drawn before executing the Dimcontinue command, the specified prompt does not appear. Next, you must specify one or more points, each of which will be a point for the second extension line of the new dimension. In this case, the first extension line coincides with the second extension line of the previous dimension, and the dimension lines of successively drawn dimensions are located on the same straight line.

Procedure for performing laboratory work

Follow the procedure according to the points.

1. Download AutoCAD.

2. Download the .dwg file with the frame from LR No. 2.

3. Analyze the given version of the task (see Appendix B, task to LR No. 6).

4. Create additional layers if necessary.

5. Given two projections, construct a third one using object tracking.

6. Make sections using object snaps (for symmetrical objects – half view / half section).

7. Use the Hatch command to perform hatching.

8. Set dimensions using the options - chain and from a common base.

9. Turn off the display of the guide line layer.

10. Save the drawing.

Review questions

1. Define the term "Object Tracking".
2. Appointment of the "Dimbaseline" team.
3. What is the main purpose of hatching?
4. What is the purpose of the “Dimcontinue” command?
5. What is the purpose of the “Hatch” command?

Laboratory work No. 7

Execution of the working drawing of the bushing

Purpose of work– Learn to make drawings in AutoCAD.

Task

- Get acquainted with the working drawing of the part;
- Prepare a working drawing of the part in accordance with the requirements of the standard;

Procedure for performing laboratory work

Follow the procedure according to the following points:

1. Download AutoCAD;

2. Download the file created on LR No. 2 and save it with a new name, for example, lr7.dwg;

3. Read Appendix B (see task for LR No. 7) with the working drawing of the part;

4. Perform constructions according to the option given in Appendix B, using the skills already acquired.

5. Draw the part using auxiliary lines;

6. Fill in the stamp;

7. Save the file.

Test questions

1. How to apply hatching if there is a break in the contour?
2. How to perform a two-sided equidistant contour?
3. How to copy an object along an arbitrary curve?
4. What command can be used to enlarge the image of part of an object?

Laboratory work No. 8

Making a working drawing of a gear

Purpose of work– familiarize yourself with the procedure for executing a working drawing of a gear-type part

Task

- Get acquainted with the working drawing of the part.
- Perform all necessary technological views of the part on a scale of 5:1.
- Draw a table of controlled gear parameters in a graphic document.
- Prepare a working drawing of the part in accordance with the requirements of the standard.

Procedure for performing laboratory work

Follow the procedure according to the following points:

1. Download AutoCAD;

2. Download the file created on LR No. 2 and save it with a new name, for example, lr8.dwg;

3. Read Appendix B (see task for LR No. 8) with the working drawing of the part;

4. Perform constructions according to the option given in Appendix B, using the skills already acquired.

5. Draw the part using auxiliary lines;

6. Fill in the stamp;

7. Save the file.

Test questions

1. How to enter an additional row in the table of controlled parameters?
2. How to split a table row horizontally?
3. How to transfer a separate view of a drawing element from one drawing to another?
4. How to copy a view while scaling it?

List of information sources

1. Autodesk [Electronic resource]. Resource access mode: https://www.autodesk.com/education/edu-software/overview.

2. Computer graphics workshop with autocad / V.M. Slidenko, M. P. Osadchuk, V. O. Polishchuk. – Kyiv: Igor Sikorsky Kyiv Polytechnic Institute, 2022 – 101 p.

3. Fundamentals of automated design: a textbook / S.M. Pavlovsky, A.V. Babkov. – Odesa: Helvetika, 2021. – 598 p.

4. Graphics systemAutoCAD. Fundamentals of mechanical engineering drawing, modeling and animation: laboratory practical / V.I. Topchiy, I.S. Aftanaziv, P.P. Voloshkevych. – Lviv: Lviv Polytechnic, 2019. – 388 p.

5. Computer graphics: AutoCAD: a textbook / M.M. Kozyar, Y.V. Feshchuk. – Odesa: Helvetika, 2020. – 304 p.

6. 3D modeling systems. Textbook. / Zinko R. V., Topilnytskyi V. G. – Lviv: Galician Publishing House, 2017. – 150 p.

7. Fundamentals of CAD in automotive engineering: textbook / O.M. Artyukh, O.V. Dudarenko, V.V. Kuzmin and others. Zaporizhzhia: National University "Zaporizhzhia Polytechnic", 2021. 168 p.

8. Topchiy V. I. AutoCAD graphics system. Fundamentals of mechanical engineering drawing, modeling and animation. Laboratory practical / V. I. Topchiy, I. S. Aftanaziv, P. P. Voloshkevych. Lviv: Lviv Polytechnic Publishing House, 2019. 388 p.

9. Tsvirkun L.I., Beshta L.V. Engineering and computer graphics. AutoCAD: teaching aids / Ministry of Education and Science of Ukraine, NTU “Dnipro Polytechnic”. – Dnipro: NTU “DP”, 2018. –209p.

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11. Vanin V.V., Bliok A.V., Gnitetskaya G.O. Design documentation. K. Karavella. 2012 – 200p.

Appendix A

Examples, results of tasks

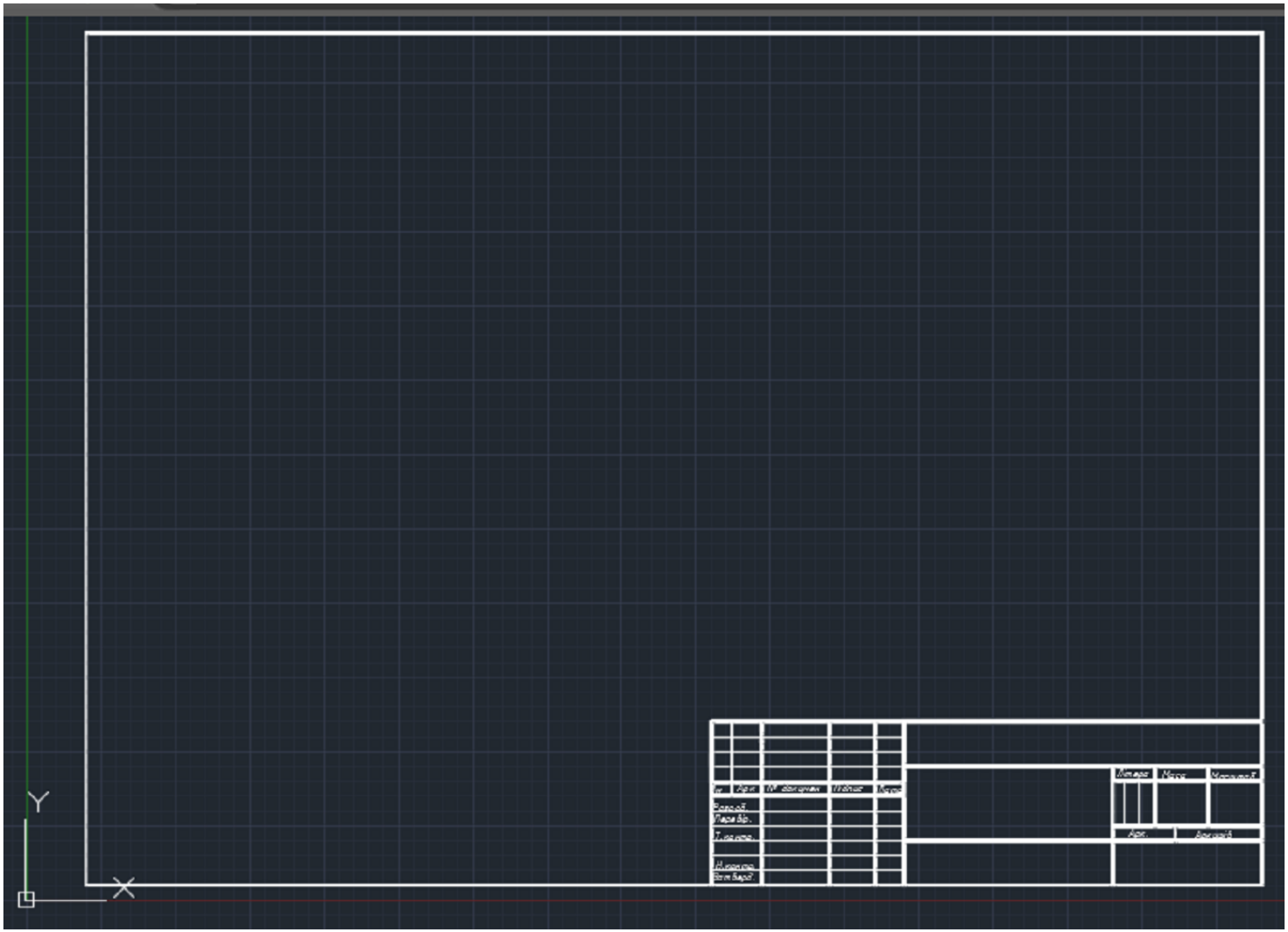


Figure A.1 – Result of completing the task for LR No. 2

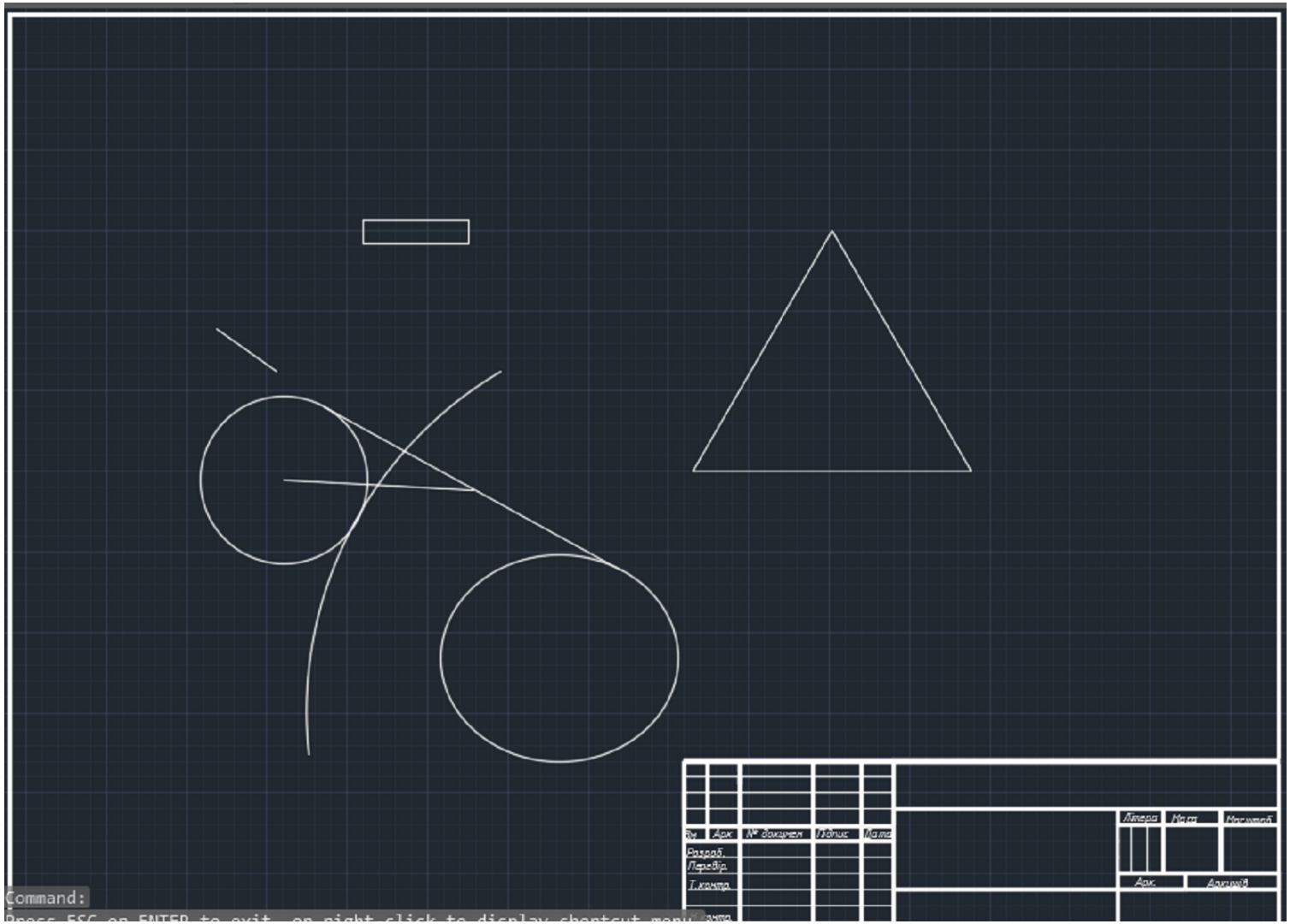


Figure A.2 – Result of completing the task for LR No. 3 (part one)

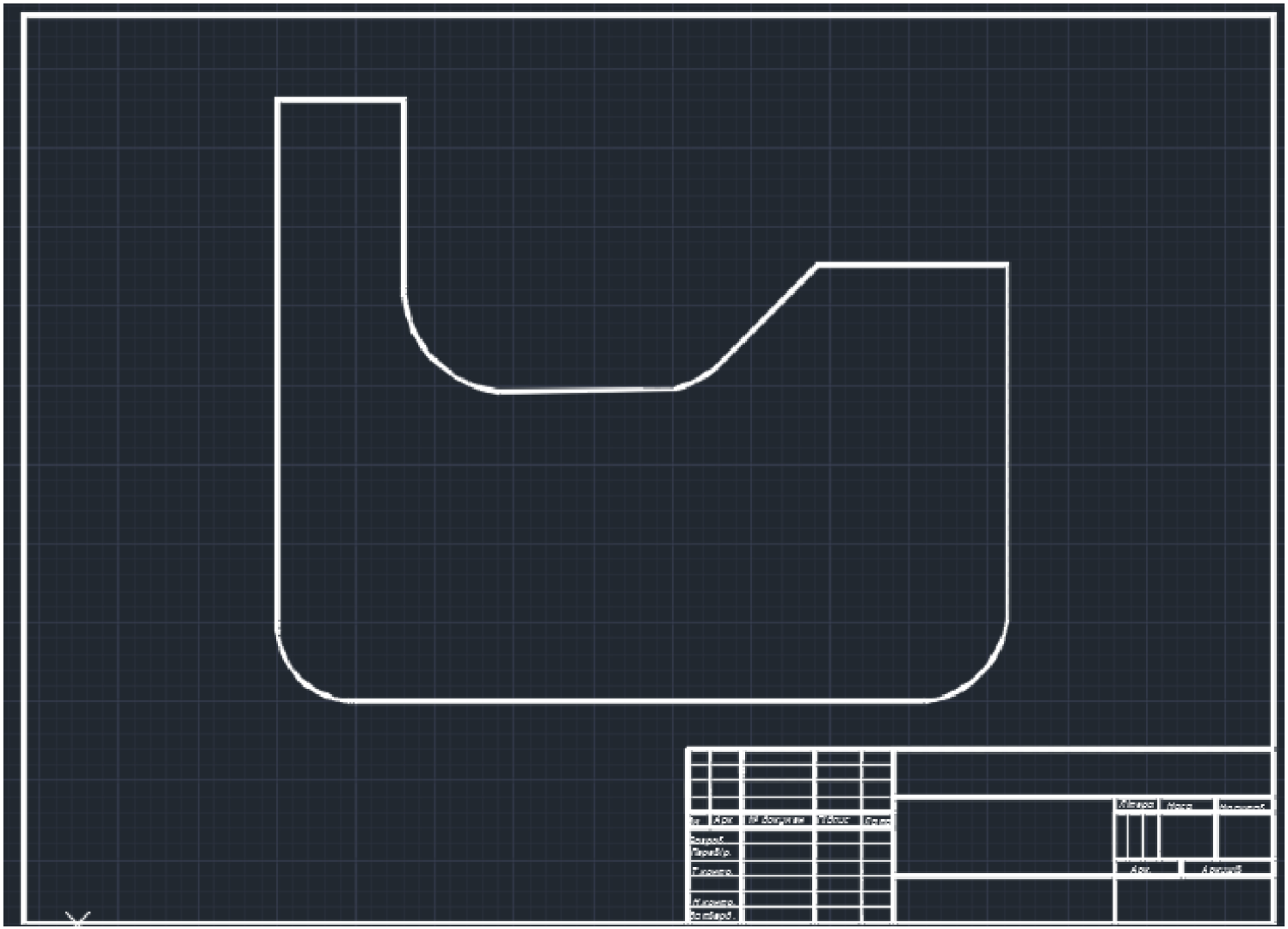


Figure A.3 – Result of completing the task for LR No. 3 (part two)

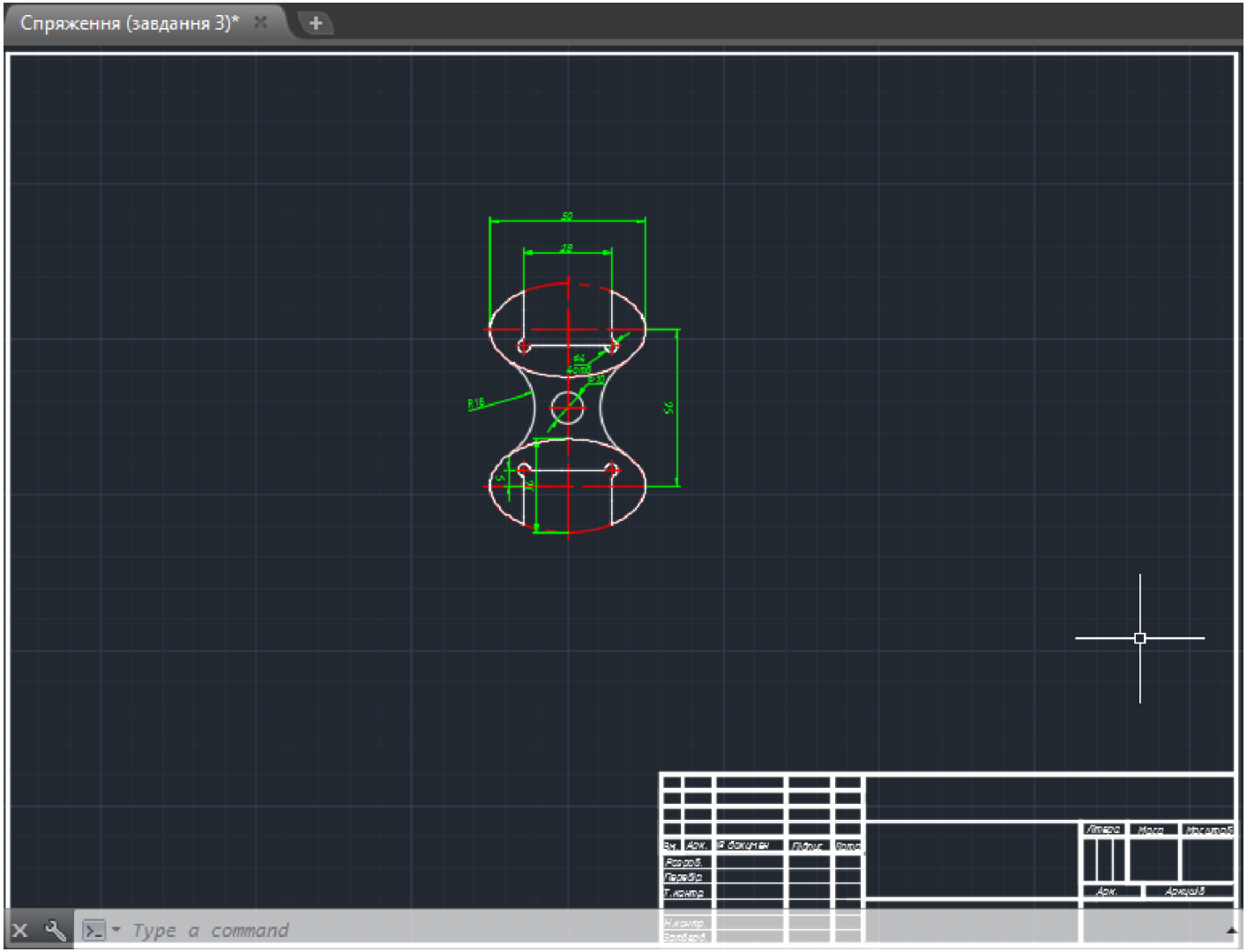


Figure A.4 – Result of completing the task for LR No. 4

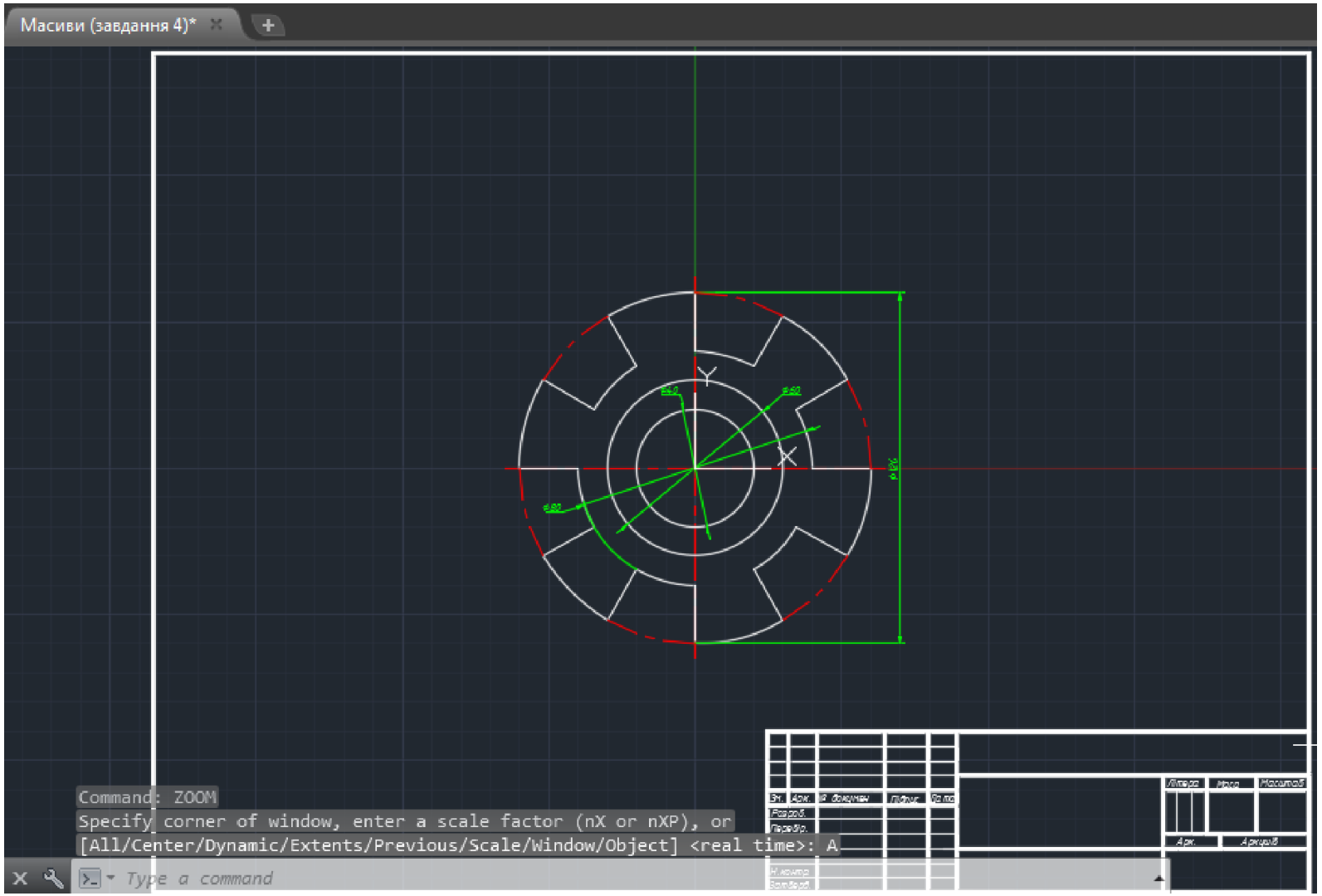


Figure A.5 – Result of completing the task for LR No. 5

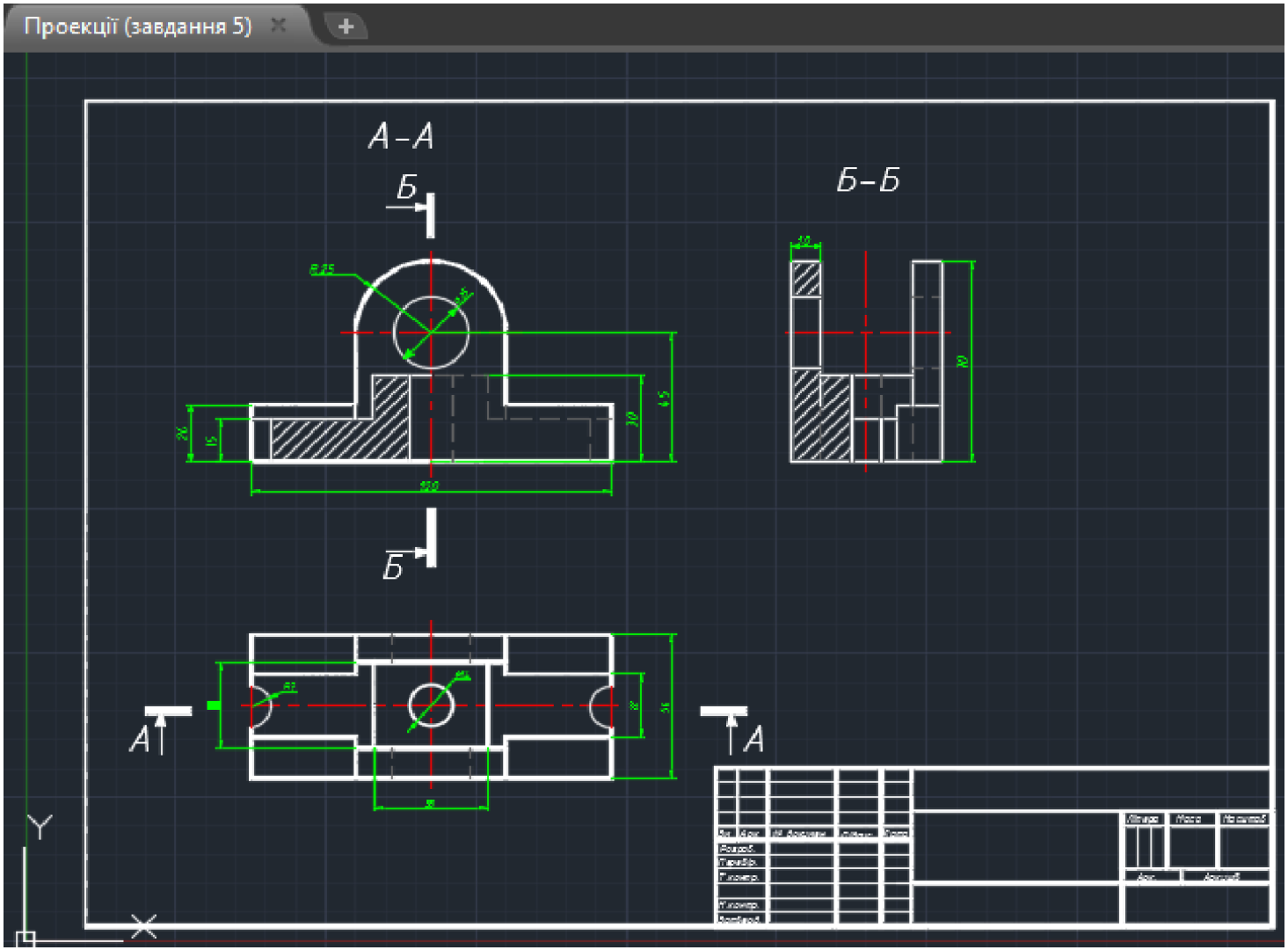


Figure A.6 – Result of completing the task for LR No. 6

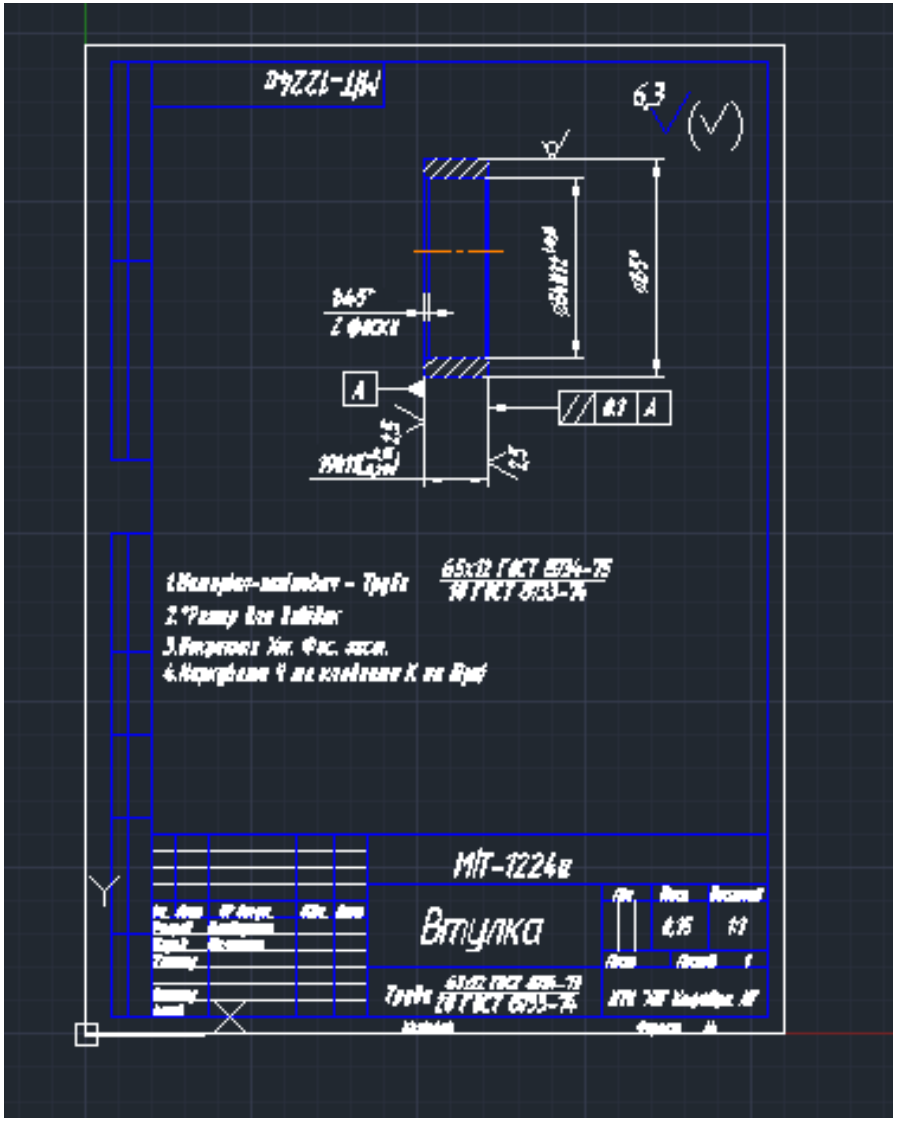


Figure A.7 – Result of completing the task for LR No. 7

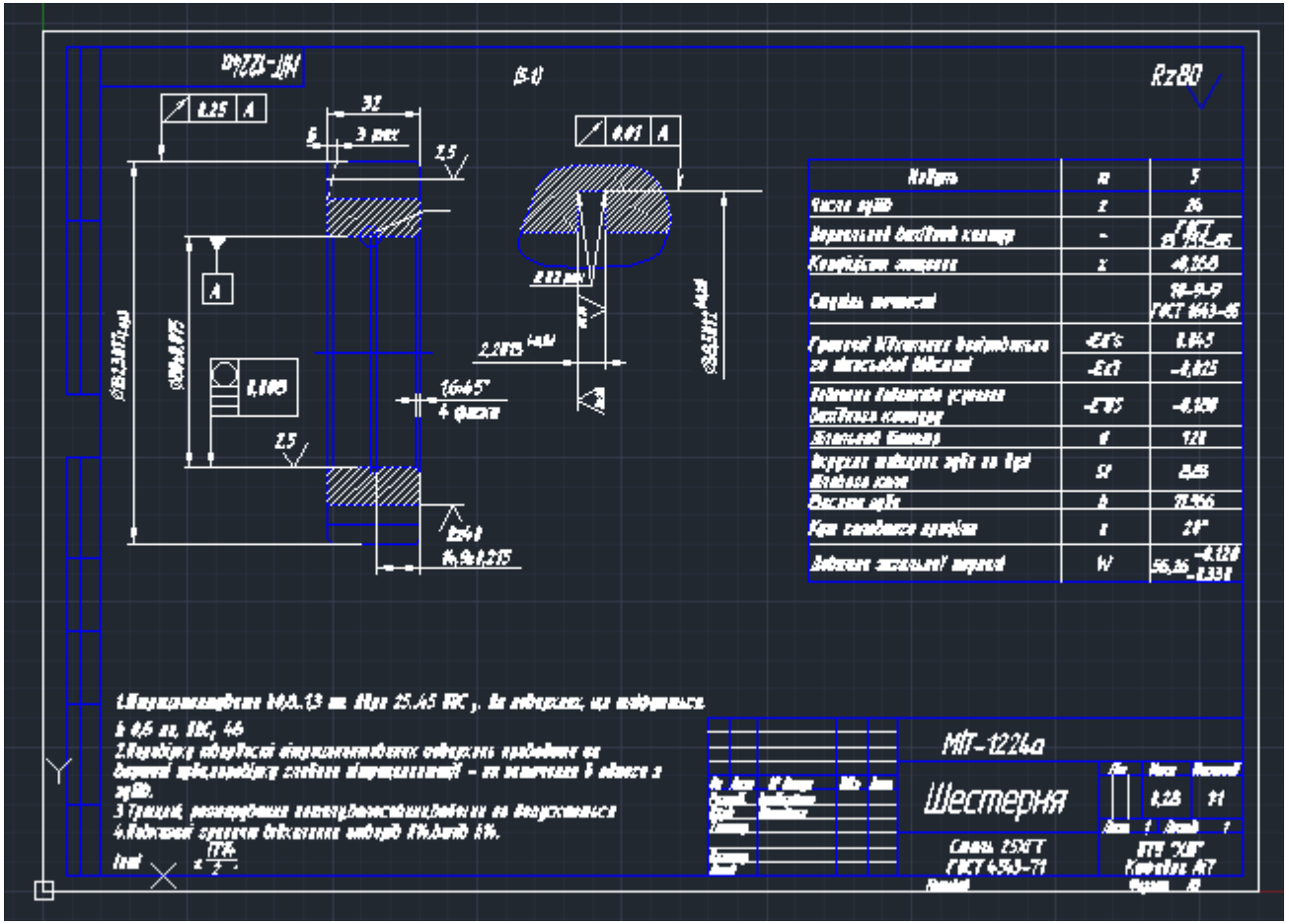


Figure A.7 – Result of completing the task for LR No. 8

Appendix B

Individual tasks on the LR

Початкова точка вибирається довільно													
Варіант	***Відрізок***		***Прямокутник***		***Дуга (S,E,R)***		***Коло (T,T,R)***		***Еліпс***		***Багатокутник***		
	довжина, мм	***кут нахилу, град***	***довжина, мм***	***висота, мм***	***кінцева точка***	***радіус, мм***	***радіус, мм***	***дотик до***	***довша вісь, мм***	***коротша вісь, мм***	***кількість вершин***	***радіус кола, мм***	***коло***
1	11	168	33	8	110,64	117	22	дуга	58	51	8	27	вписане
2	7	175	39	18	250,66	133	17	відрізок	64	30	8	25	описане
3	20	106	27	8	60,77	130	18	дуга	61	36	8	25	вписане
4	16	66	23	24	290,52	116	24	відрізок	64	48	6	20	описане
5	10	138	31	15	280,65	119	22	дуга	66	53	8	21	вписане
6	5	151	21	9	35,64	122	26	відрізок	61	30	5	16	описане
7	7	162	23	14	21,57	121	20	дуга	61	35	7	15	вписане
8	14	173	24	17	29,78	125	18	відрізок	59	54	10	17	описане
9	21	150	35	15	25,72	127	18	дуга	55	43	8	26	вписане
10	6	142	37	12	27,76	121	15	відрізок	61	52	10	14	описане
11	24	74	29	15	191,74	128	18	дуга	60	37	9	26	вписане
12	6	73	24	15	70,75	126	12	відрізок	69	59	8	10	описане
13	11	48	26	9	122,67	122	18	дуга	53	42	9	20	вписане
14	9	179	34	5	29,50	130	27	відрізок	65	34	9	27	описане
15	19	139	30	16	27,52	116	27	дуга	55	23	5	15	вписане
16	22	105	24	15	25,74	117	14	відрізок	61	41	10	27	описане

Figure B.1 – Options for implementing the first part of LR No. 3

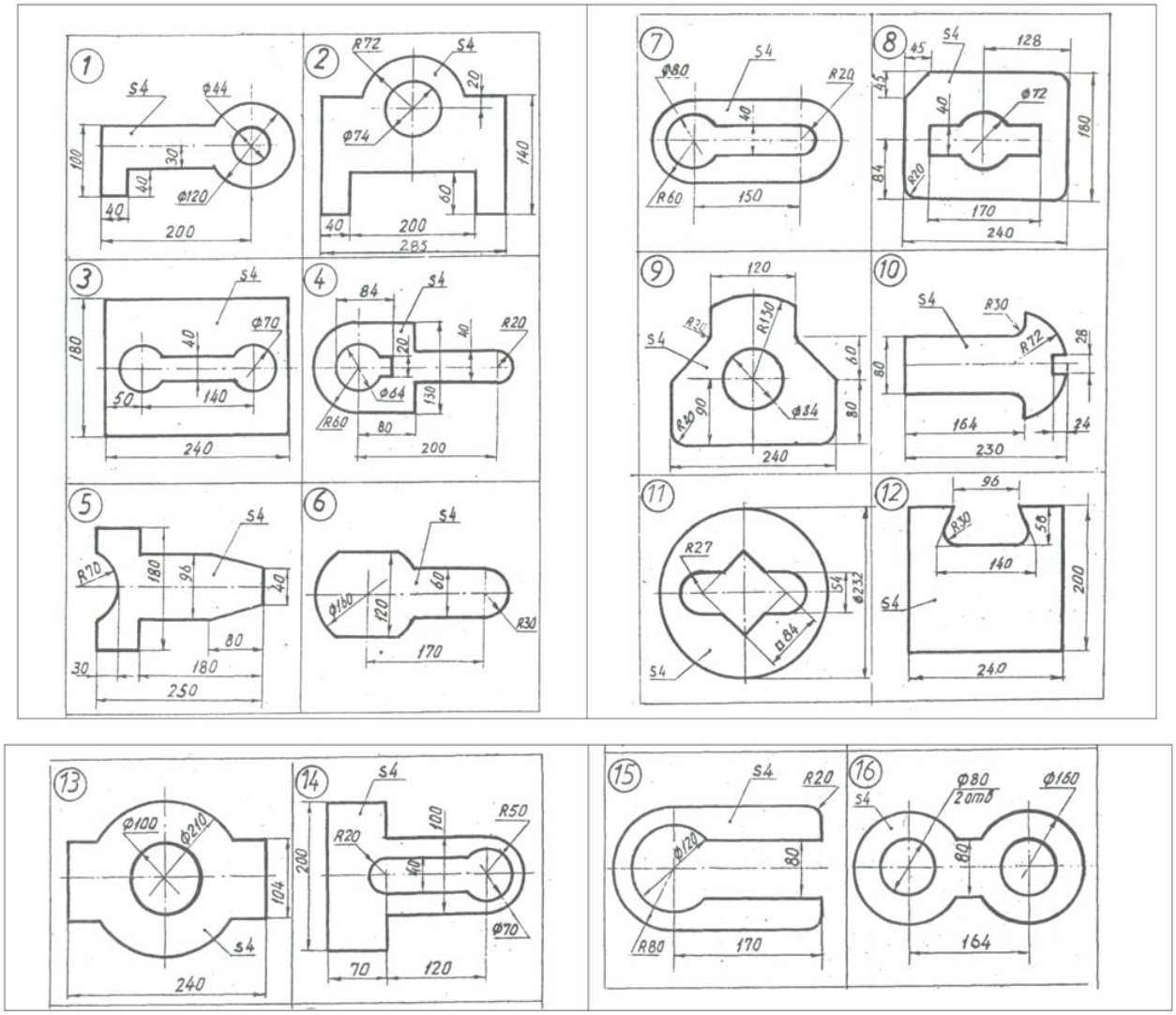
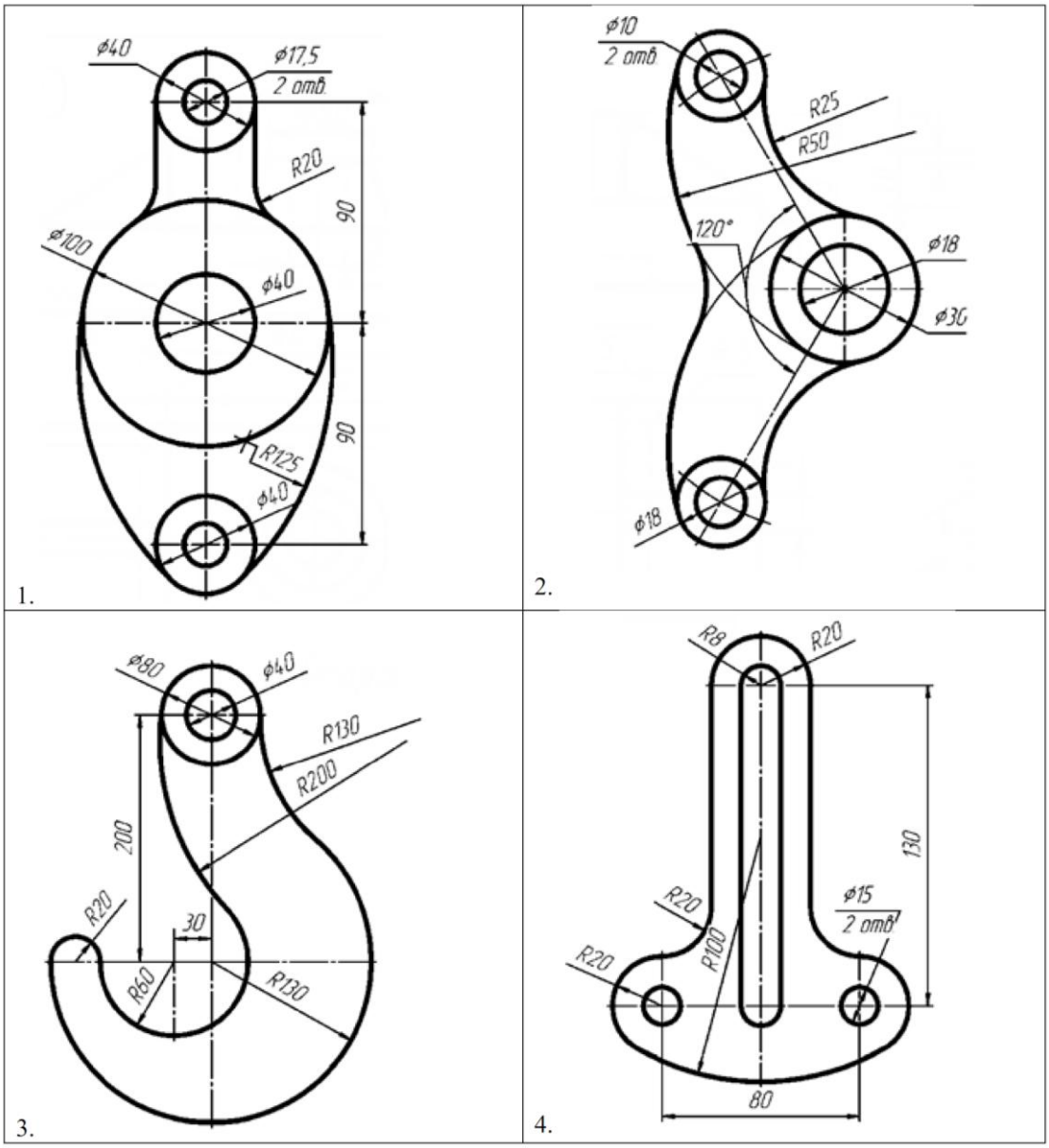
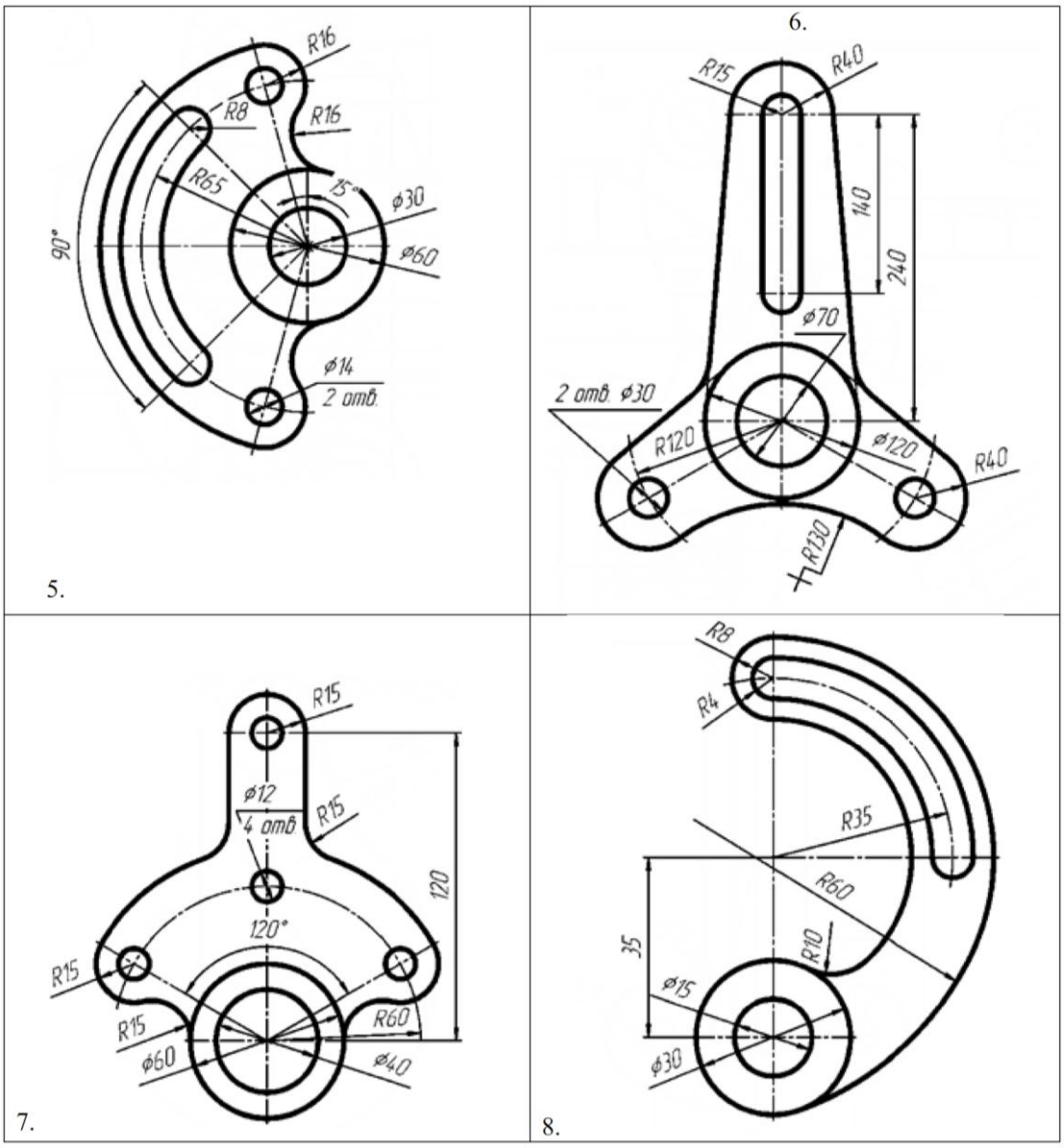
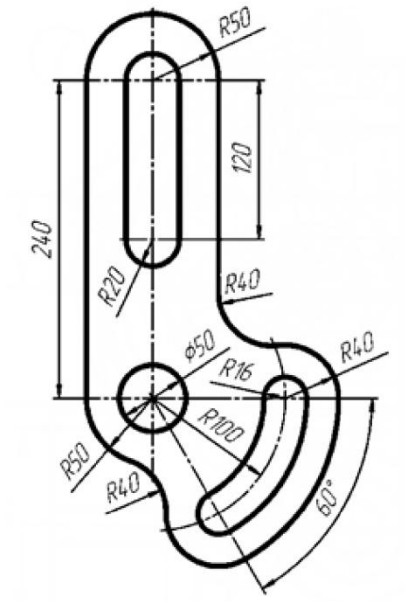


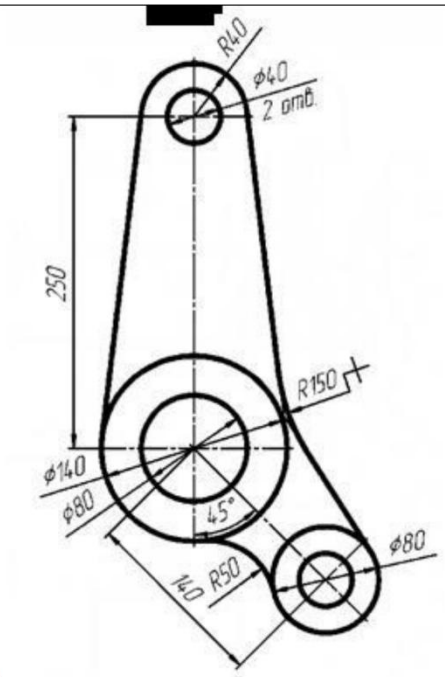
Figure B.2 – Options for implementing the second part of LR No. 3


ϕ40
ϕ17,5
2 отв
R20
90
ϕ100
ϕ40
90
R125
ϕ40
1.
ϕ10
2 отв
R25
R50
120°
ϕ18
ϕ36
ϕ18
2.
ϕ80
ϕ40
R130
R200
200
R20
30
R60
R130
3.
R8
R20
130
R20
ϕ15
2 отв
R20
R100
80
4.

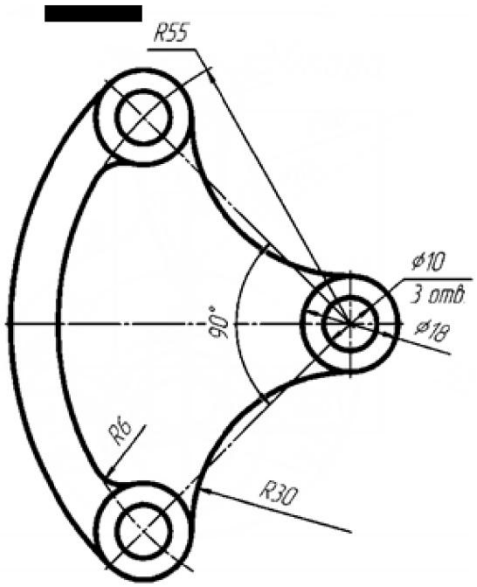

5.
R16
R8
R16
R65
15°
ϕ30
90°
ϕ60
ϕ14
2 отв.
6.
R15
R40
140
240
ϕ70
2 отв. ϕ30
R120
ϕ120
R40
R130
7.
R15
ϕ12
4 отв
R15
120
120°
R15
R15
R60
ϕ60
ϕ40
8.
R8
R4
R35
R60
35
R10
ϕ15
ϕ30


R50
120
240
R20
R40
ϕ50
R16
R40
R100
R50
R40
60°

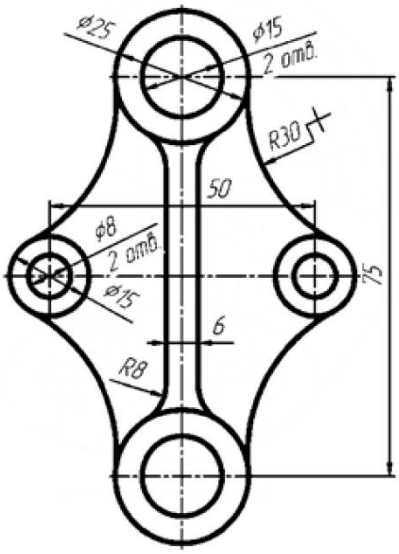
9.


R40
ϕ40
2 отв.
250
R150
ϕ140
ϕ80
45°
ϕ80
140
R50

10.


R55
ϕ10
3 отв
ϕ18
90°
R6
R30

11.


ϕ25
ϕ15
2 отв.
R30
50
ϕ8
2 отв.
ϕ15
6
R8
75

12.

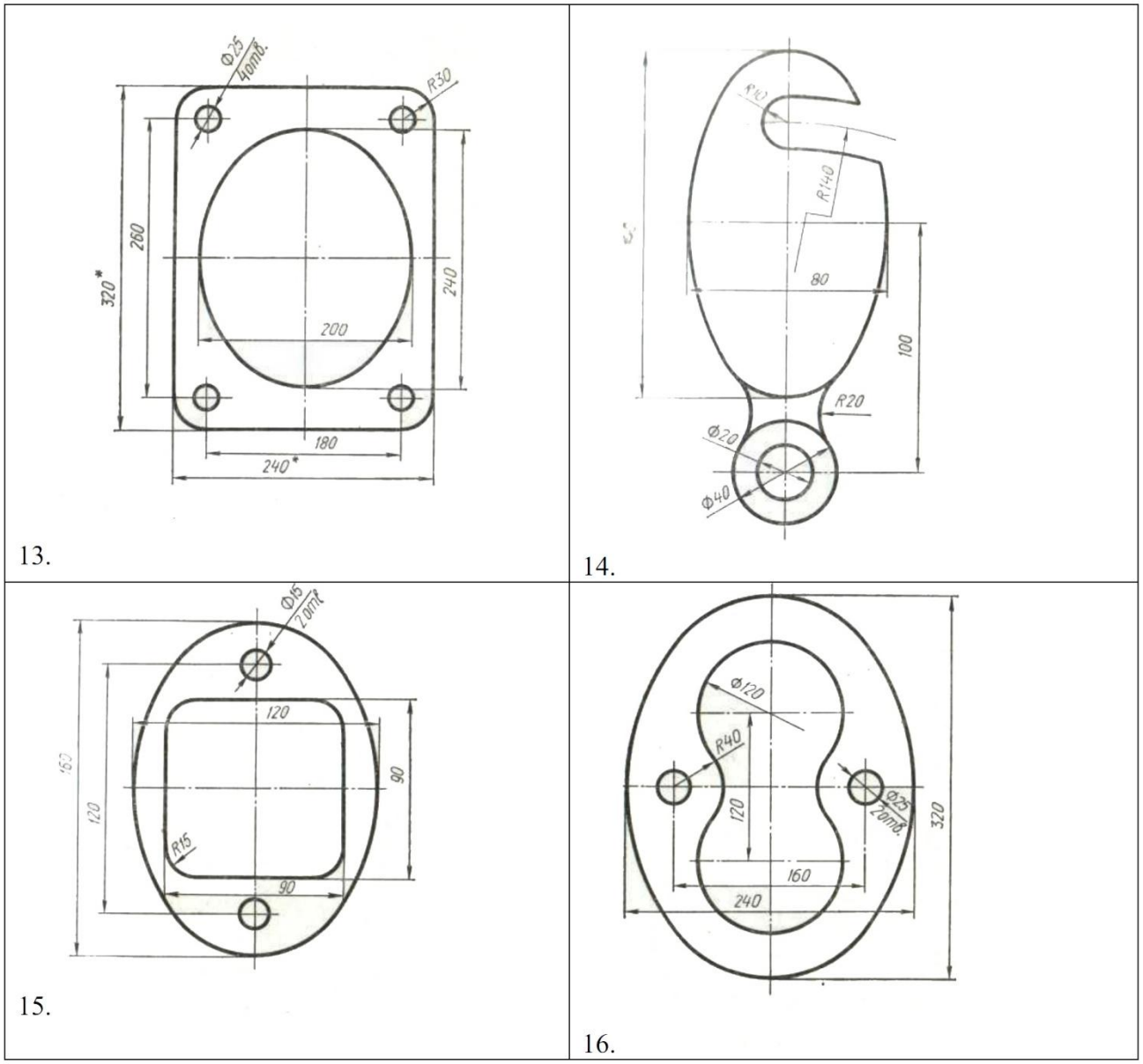
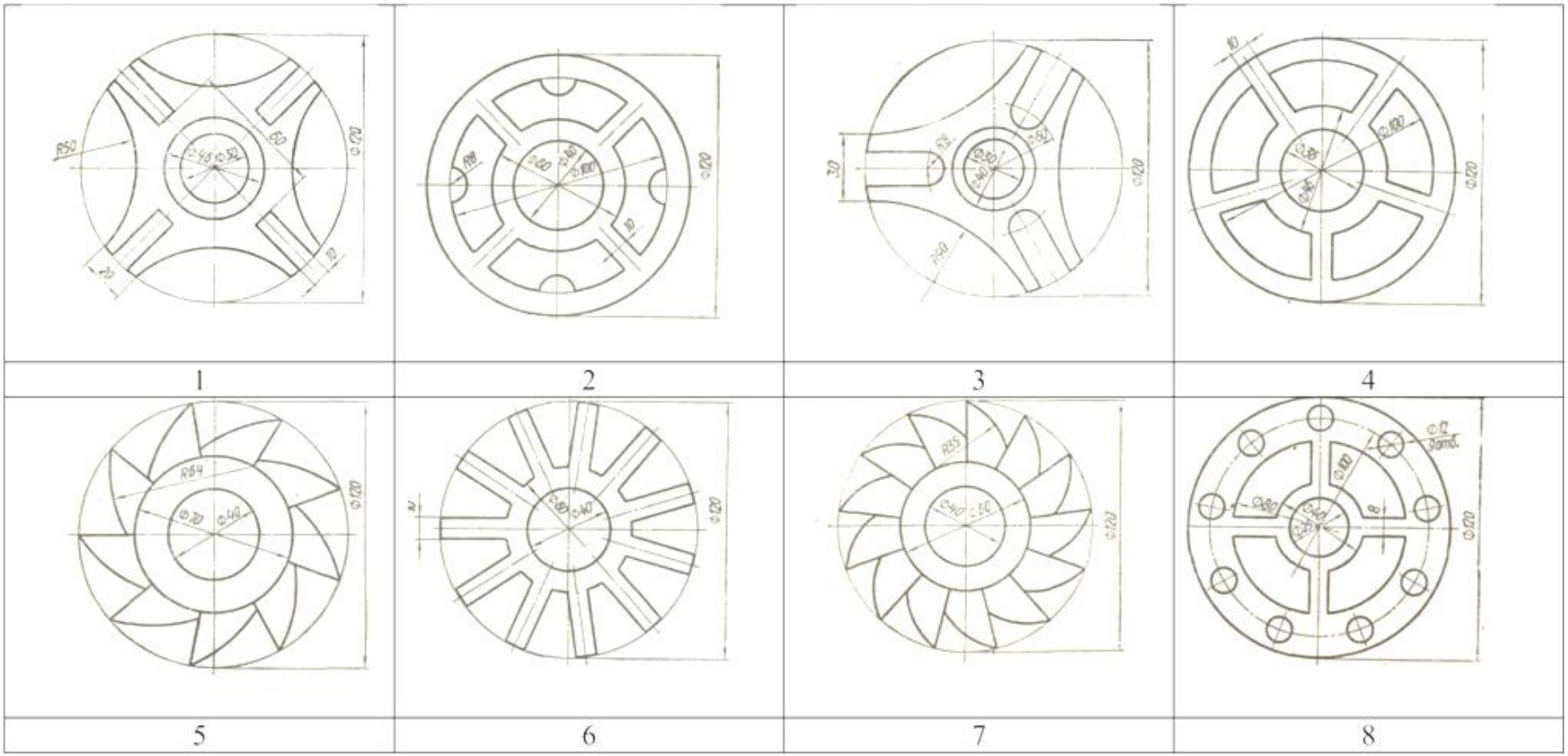


Figure B.3 – Options for implementing LR No. 4


1
2
3
4
5
6
7
8

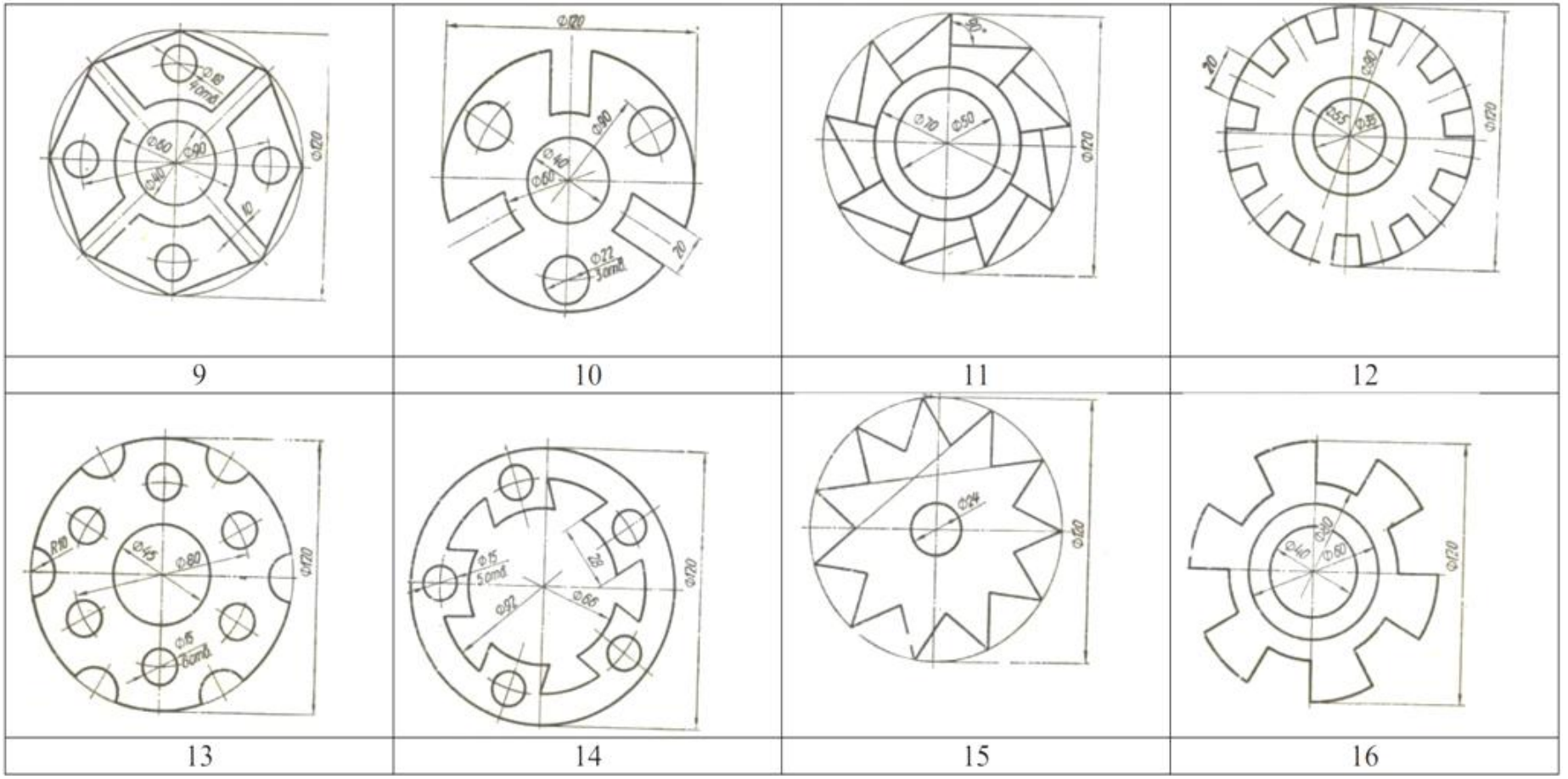
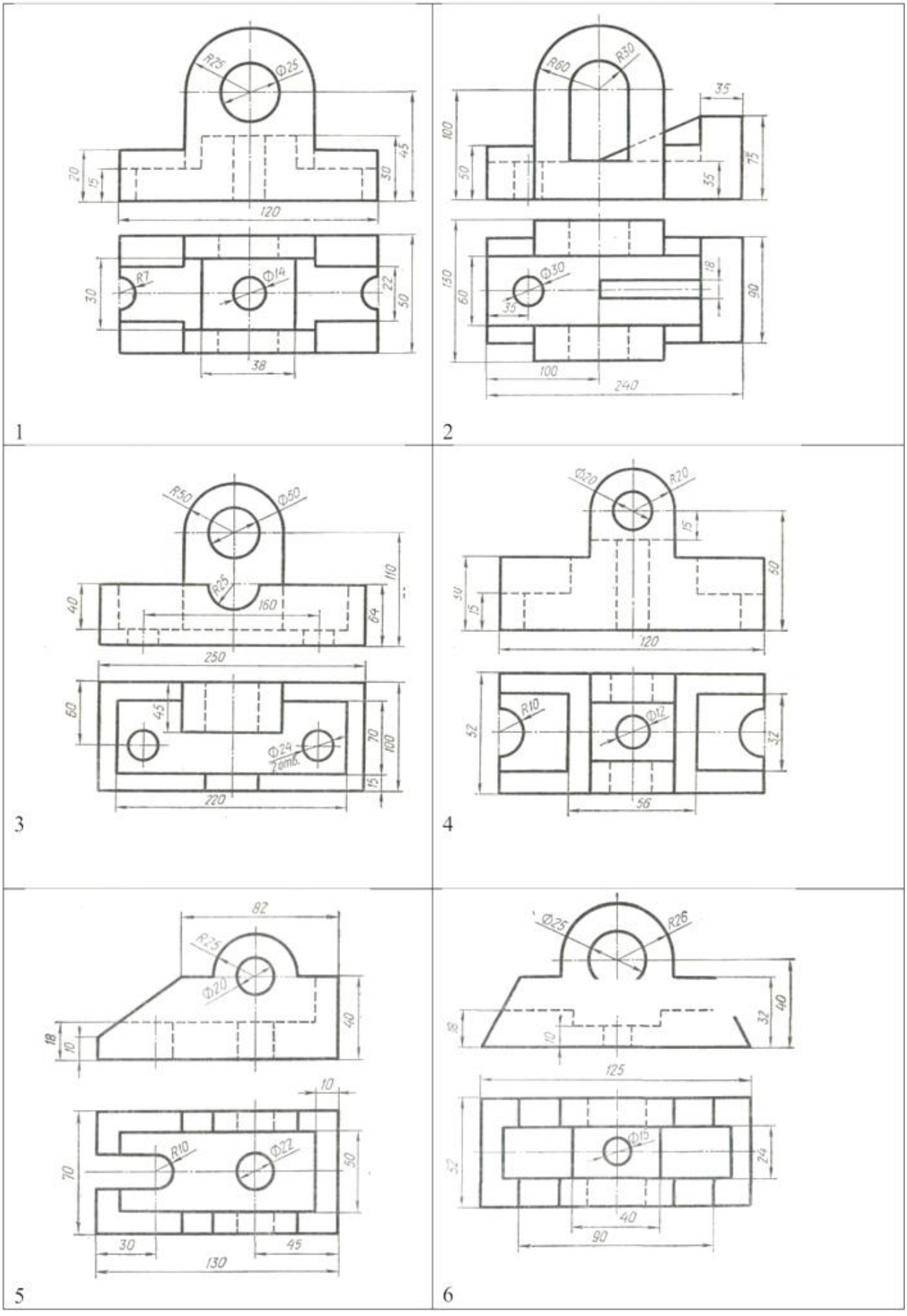


Figure B.4 – Options for implementing LR No. 5


R25
Ø25
20
15
30
45
120
R7
Ø14
30
22
50
38
1
R60
R30
35
100
50
35
75
130
60
Ø30
35
18
90
100
240
2
R50
Ø50
110
40
R25
160
64
250
60
45
70
100
Ø24
2отв.
15
220
3
Ø20
R20
15
50
54
15
120
R10
Ø12
52
32
56
4
82
R25
Ø20
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18
10
10
R10
Ø22
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18
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125
Ø15
52
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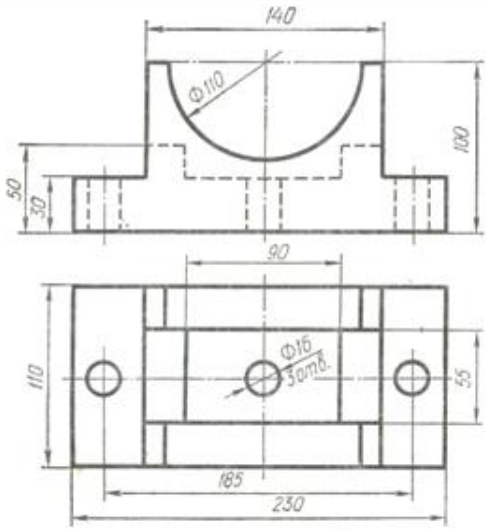
40
90
6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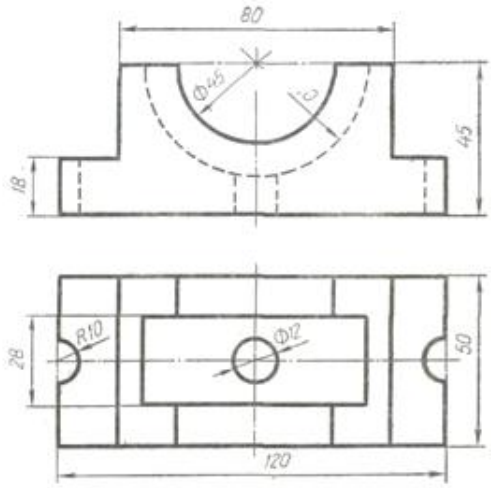
110
Ф16
3отв.
55
185
230

7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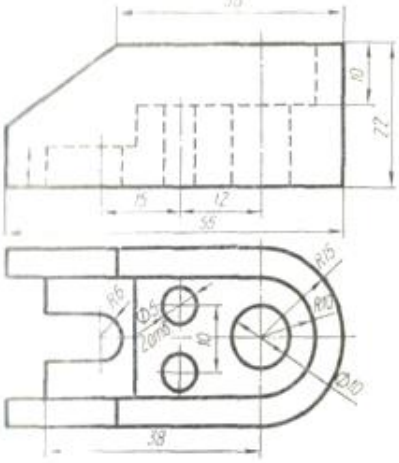
Ф45
45
18
R10
Ф12
28
50
120

8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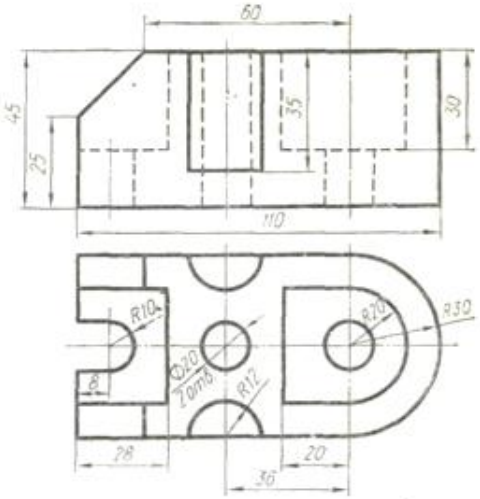
55
R15
R16
Ф5
2отв.
R8
40
Ф10
38

9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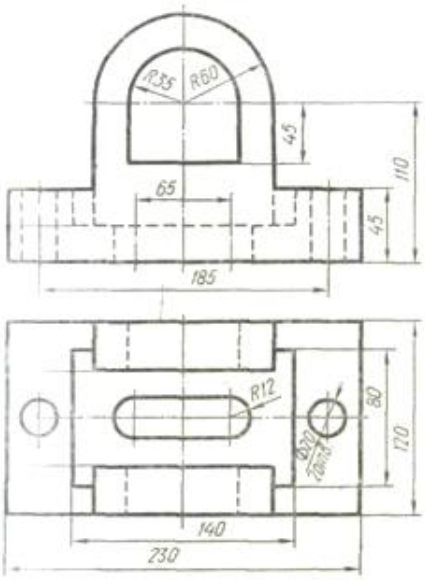
30
110
R10
R30
Ф30
2отв.
R12
8
28
20
36

10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R12
Ф20
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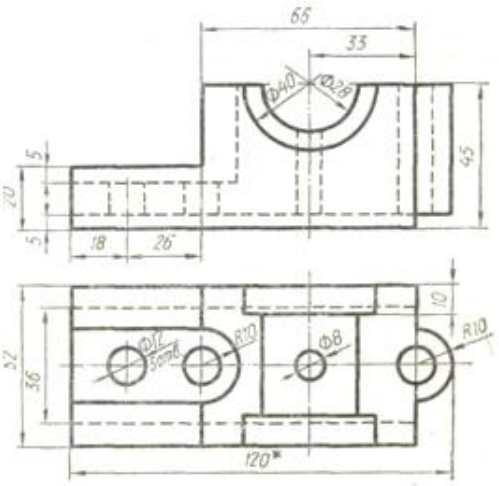
11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R10
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3отв.
Ф8
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32
36
120

12

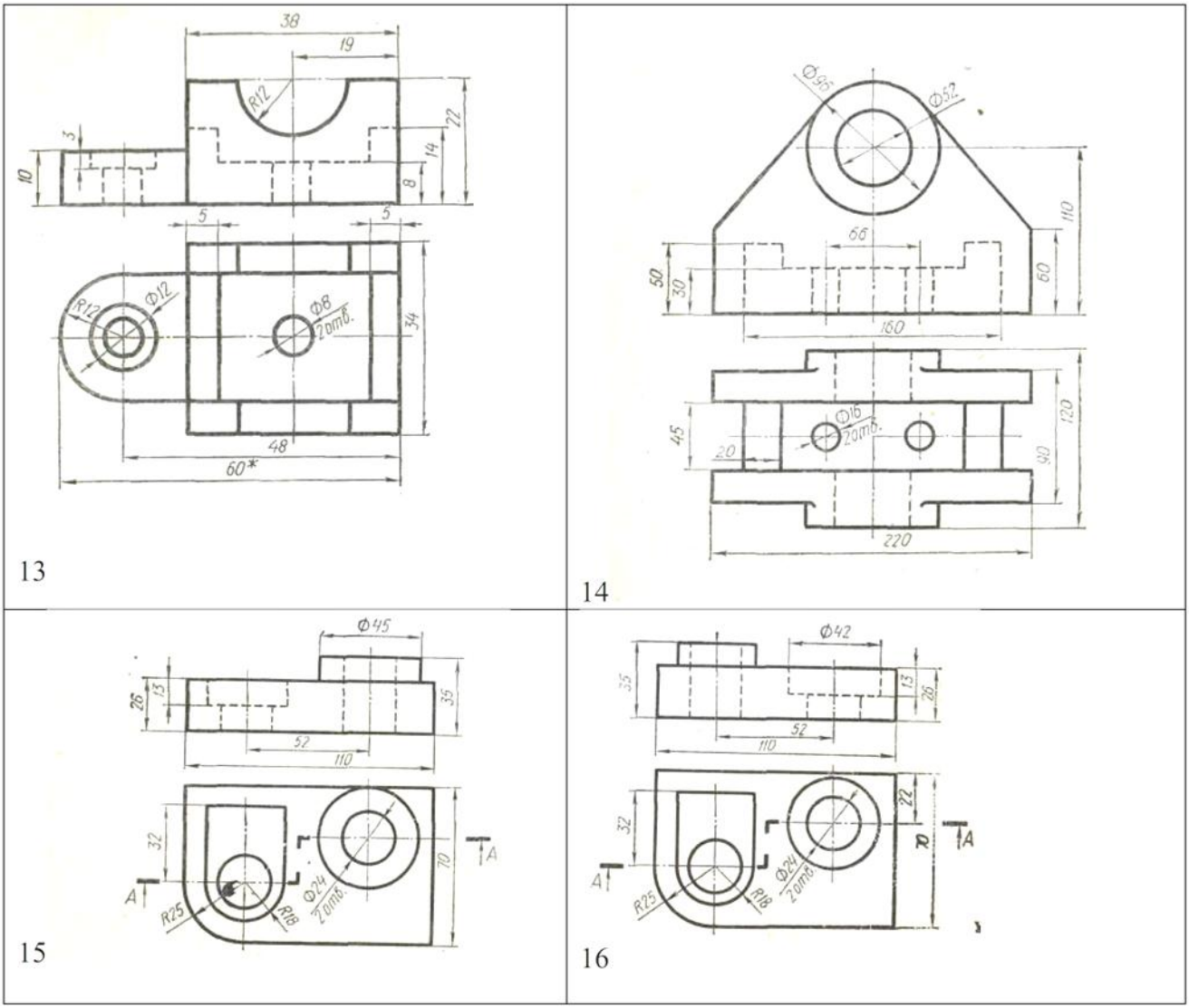


Figure B.5 – Options for implementing LR No. 6

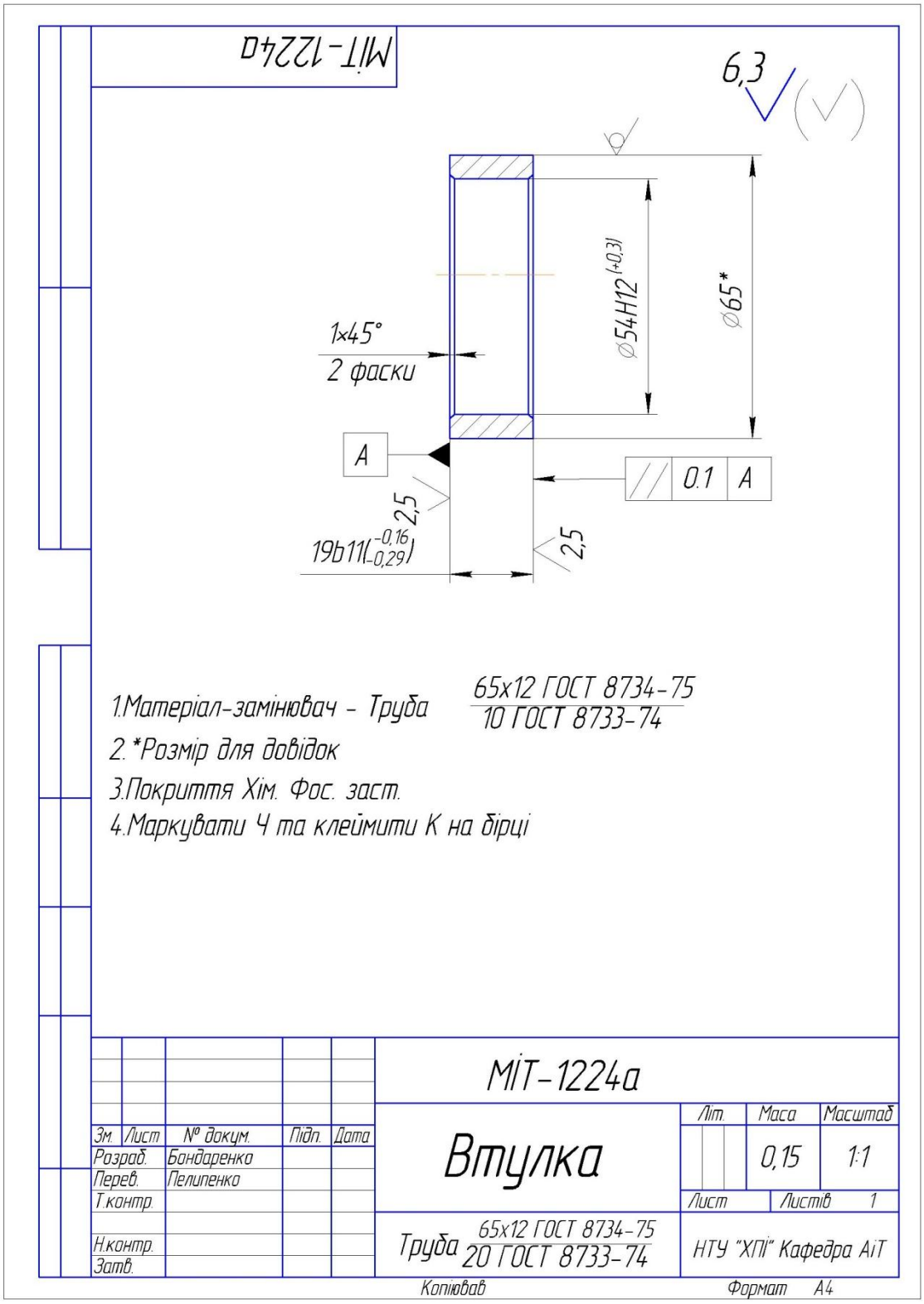
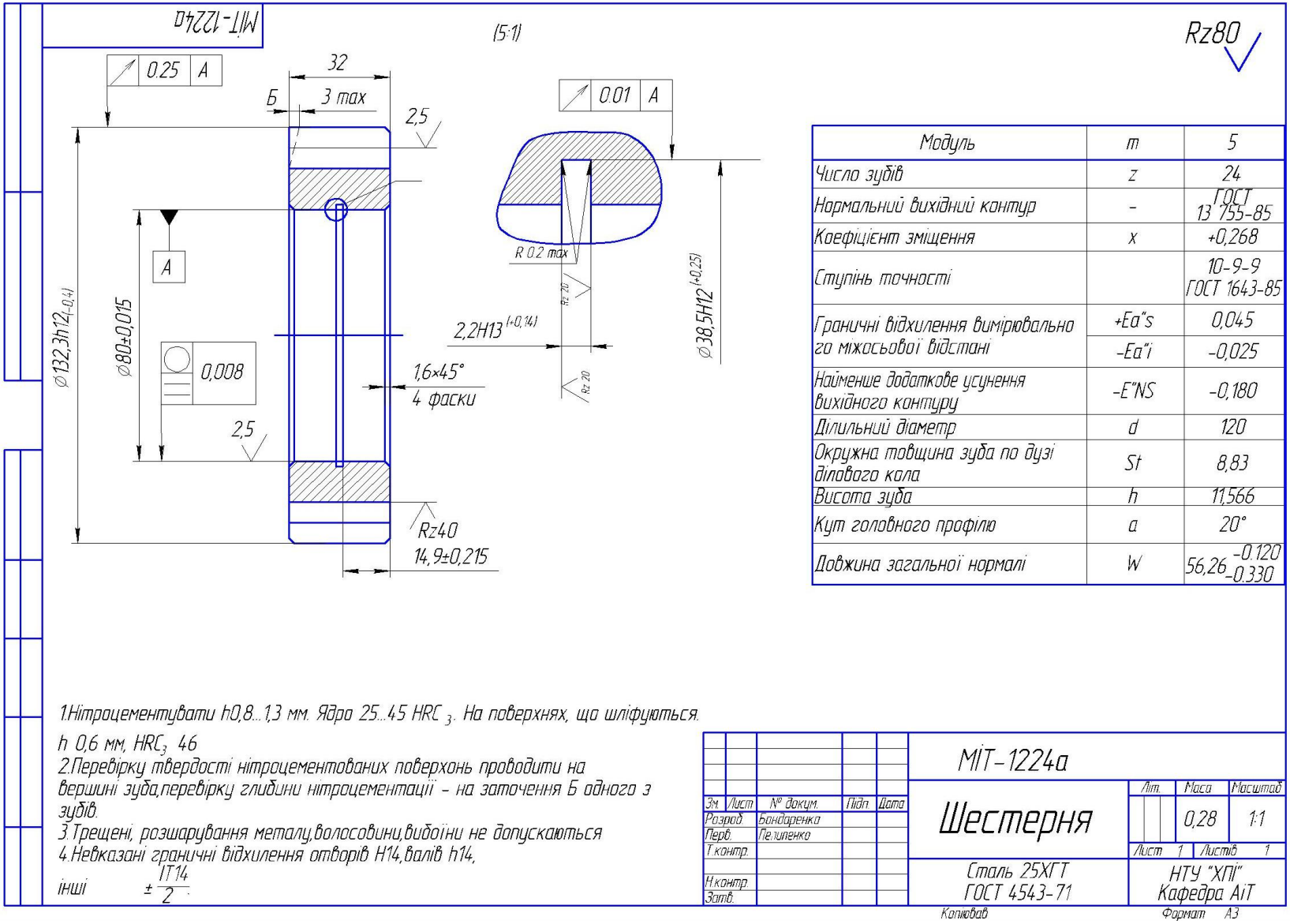


Figure B.6 – Task LR No. 7



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Число зубів	z	24
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Коефіцієнт зміщення	x	+0,268
Ступінь точності		10-9-9 ГОСТ 1643-85
Граничні відхилення вимірювального міжосьової відстані	+Ea"s	0,045
	-Ea"i	-0,025
Найменше додаткове усунення вихідного контуру	-E"NS	-0,180
Ділильний діаметр	d	120
Окружна товщина зуба по дузі ділового кола	St	8,83
Висота зуба	h	11,566
Кут головного профілю	a	20°
Довжина загальної нормалі	W	56,26 -0.120 -0.330

Figure B.7 – Task LR No. 8

Project

Educational edition

CAD Basics

METHODOLOGICAL INSTRUCTIONS

for laboratory work

according to the educational program "Transport and technological machines and equipment" specialty 133 "Industrial mechanical engineering"

Compilers:

PELIPENKO Yevhen Serhiyovych

Responsible for the release: Prof. O.Yu. Rebrov

The work was recommended for publication by Prof. V.M. Shevtsov.

In the author's edition

Plan, pos.

Ready for printing « » 20___ year. Format 60x84 1/16. Offset paper.

Riso-print. Font Times New Roman. Conditional. printing. sheets. ____

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