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Electronics and Robotics
department

TRAINING INTERNSHIP. DESIGN IN AUTOCAD / SOLIDWORKS

Guidelines for the implementation of practical work on educational practice
for a bachelor of all forms of training in the specialty 6B07109 – Instrument making

Almaty 2021

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Methodical instructions are devoted to the study of all well-known graphic platforms AutoCad, SolidWorks, which are the main elements for creating graphic drawings and technical documentation.

In detail, step by step, the steps for completing the assignments given to students are reviewed and explained. Starting from building a drawing frame and filling it with text, ending with building a 3D model in space. Also, the topic of cuts, sections is touched upon.

In addition, rational techniques for constructing and editing drawings are explained.

The methodological instructions are drawn up in order to obtain practical skills in working in graphic editors and are intended for students of the specialty 5B071600 - "Instrumentation".

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Reviewer: associated professor M.Khizirova

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Introduction

AutoCad, SolidWorks programs are one of the most common computer-aided design systems (CAD). This is due to their capabilities and unique functionality, as well as the versatility of the programs. They are easy to learn and quite effective for making electronic drawings, part models.

Laboratory workshop on the discipline "Design in AutoCad / SolidWorks" consists of 8 works, ranging from simple to more complex.

The methodological instructions contain work assignments for eight practical works on educational practice in the AutoCad / SolidWorks environment. A technique is given for creating drawings, models, applying dimensions, and so on.

Classes are conducted on personal computers using AutoCad and SolidWorks programs.

№1 practical work. Downloading and installing AutoCad software

Objective: acquaintance with the graphic space of the Autocad environment, acquiring practical skills in working with the program and setting its parameters.

1.1 Short theoretical information

AutoCAD is the most powerful computer-aided design (CAD) system for personal computers. From building designs and bridges to fancy suit designs, everything is within the scope of AutoCAD.

Before installing AutoCAD, you must make sure that the computer on which you are installing the product meets all system requirements. For example, to install the latest version, your computer must meet the following requirements.

Operating system: Microsoft Windows 2000 (SP4), 32/64-bit version of Windows XP Home / Professional (SP2) or Windows Vista.

Quite high requirements are also imposed on the hardware: the processor must be at least Pentium IV (or other compatible), the amount of RAM must be at least 512 MB.

There must be 750 MB free on the hard disk for installing the program files and the same amount for temporary files created by the system during operation. Of course, there should be some space on the disk for saving the files that will be created.

The computer must also have Microsoft Internet Explorer 6.0 SP1 or higher installed.

1.2 Work order

1.2.1 Go to the official AutoDesk website and choose the "Student version" section, after which a window will open (figure 1.1).

Unlock educational access to Autodesk products

Students and educators can get free one-year educational access to Autodesk products and services, renewable as long as you remain eligible. Confirm your eligibility now.

GET STARTED



▶ [How it works \(3:08 min.\)](#)

Already have educational access? [Sign in](#)



Figure 1.1 - Window of the AutoDesk site for obtaining the student version

1.2.2 Register on this site by clicking on the “Sing” button, where in the e-mail section you will need to indicate your corporate mail address (... @ aues.kz)

1.2.3 After successful registration on the site, the site will ask you to re-register. Produce it. And then, a window will appear (figure 1.2). Having filled in all the data, the system will ask you to upload documents confirming your status as a student. Upload your document and wait for a response from AutoDesk Community.

Tell us about your educational institution and how you plan to use your free software.

Name of educational institution

Area of study

☐ Architecture, Engineering & Construction

☐ Media & Entertainment

☐ Product Design & Manufacturing

☐ Others

Enrolled from

Month Year

Graduate in

Month Year

NEXT

Figure 1.2 - Process for obtaining a student license

1.2.4 Further, after the system approves your status, you need to go to the site, register on it, download the AutoCad 2018 installation disk (AutoCad 2019, AutoCad 2020, AutoCad 2021 - depending on the characteristics of your PC) (figure 1.3).

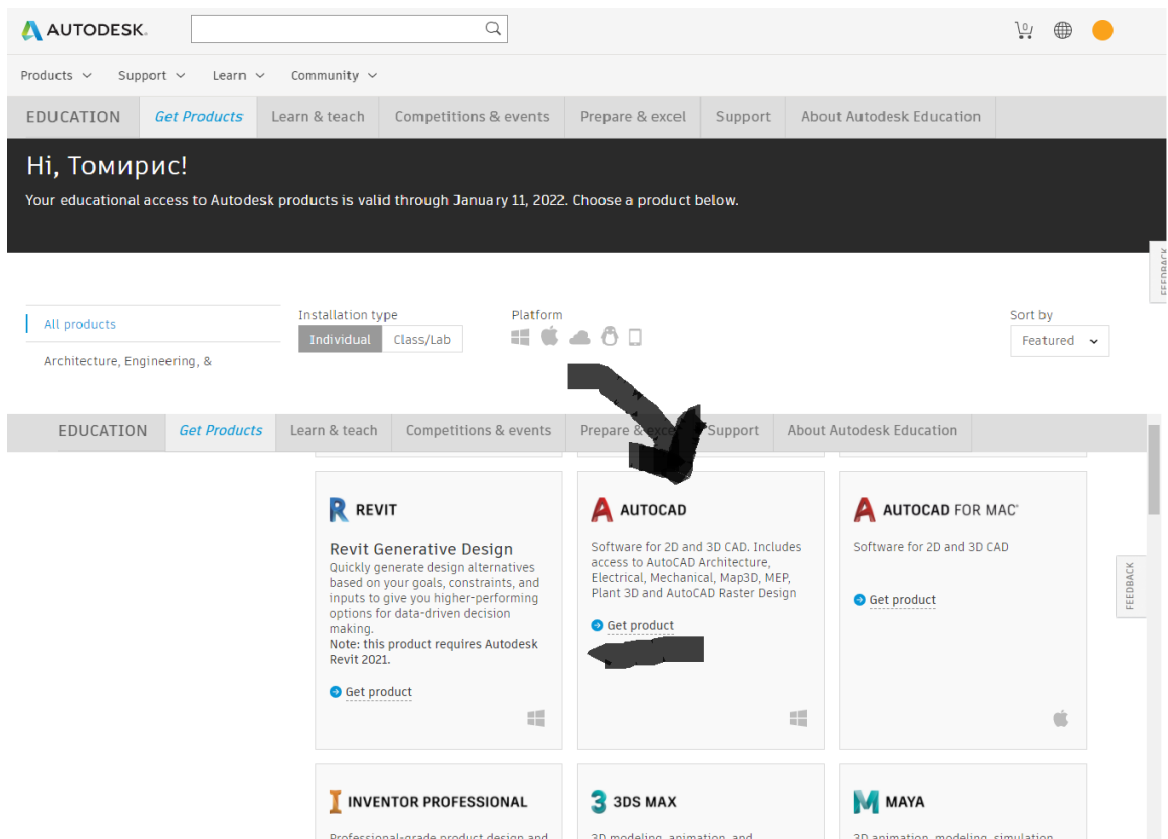


Figure 1.3 - Downloading the installation file

1.2.5 Download the installation file.

1.2.6 Wait until installation is over. Get familiar with the working environment and tools.

1.3 Content of the report

- goal of the work;
- a description of the installation of the program with screenshots and photos;
- conclusion.

1.4 Defence questions

- 1) What is AutoCad?
- 2) What graphics programs are there and describe their difference between AutoCad?
- 3) Tell us about your AutoCad model?


№2 practical work. Drawing frame construction

Objective: acquaintance with the program, its functionality and building the main frame with text blocks.

2.1 Short theoretical information

The ability of a specialist to work correctly with technical documentation is a prerequisite for the design, manufacture and operation of devices, apparatus, machines, and others. All the necessary information for the assembly of the product, production must be clearly and fully shown in the drawing. When designing instrumentation products, strict adherence to the rules of the standards collected in the "Unified System for Design Documentation" (USDD) is required.

2.2 Work order

2.2.1 To create a new drawing, move the mouse over the icon located in the upper left corner of the program . We select the acadiso template from the window that opens, since the correct metric values are selected in this template. A new drawing window will appear on the screen. By default, the Models field will open.

2.2.2 Before starting to create a drawing, enable the following properties of the model as: Mesh, Ortho, Snap, Weight (figure 2.1).

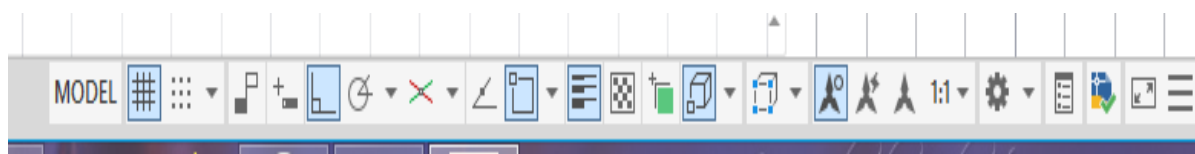



Figure 2.1 – Command line status

2.2.3 The process of creating a sheet A3, with sizes as 297×420 mm, is as follows (according to the GOST 2-301-68 standard). From the Upper panel located above the graphics space, point to the Rectangle tool (Draw panel \rightarrow Rectangle element), click on the icon .

It is necessary to enter the coordinates of the lower left corner (0,0) into the command line into the command menu, press "Enter" and thereby confirm the entered value. Then, at the command line, the program will ask you to determine the second point or specify the area, dimensions. We select the second item - dimensions and enter the dimensions of the rectangle as: length 420 mm, width 297 mm. After you have specified the width, you must confirm your choice with the "Enter" button. We do the same with other parameters (figure 2.2). Then, when you see a rectangle in graphics space, left-click anywhere. A3 sheet format is ready.

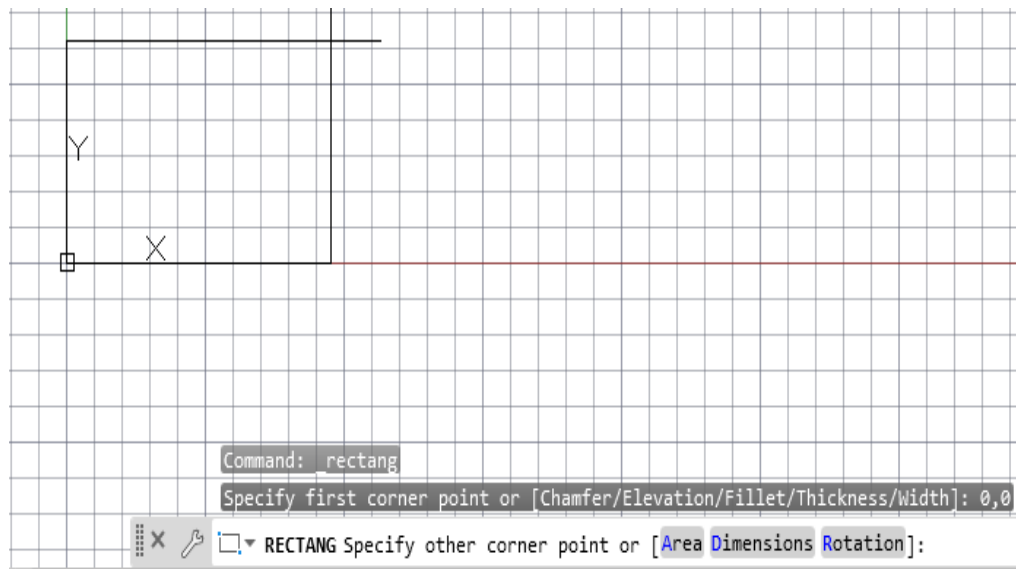


Figure 2.2 – Constructing a rectangle with the specified parameters

2.2.4 Similar to how the sheet drawing was produced, we will draw the frame and border of the title block of the drawing. To do this, inside the sheet drawing, select the Rectangle element. Coordinate of the shambles (20.5), length 395 mm, width 287 mm. Then, similarly, we will build a table to fill in the main inscriptions of the drawing (figure 2.3). We will also press the thickness of the A3 sheet and the inner frame equal to 0.5 mm (figure 2.4).



Figure 2.3 – Inside frame

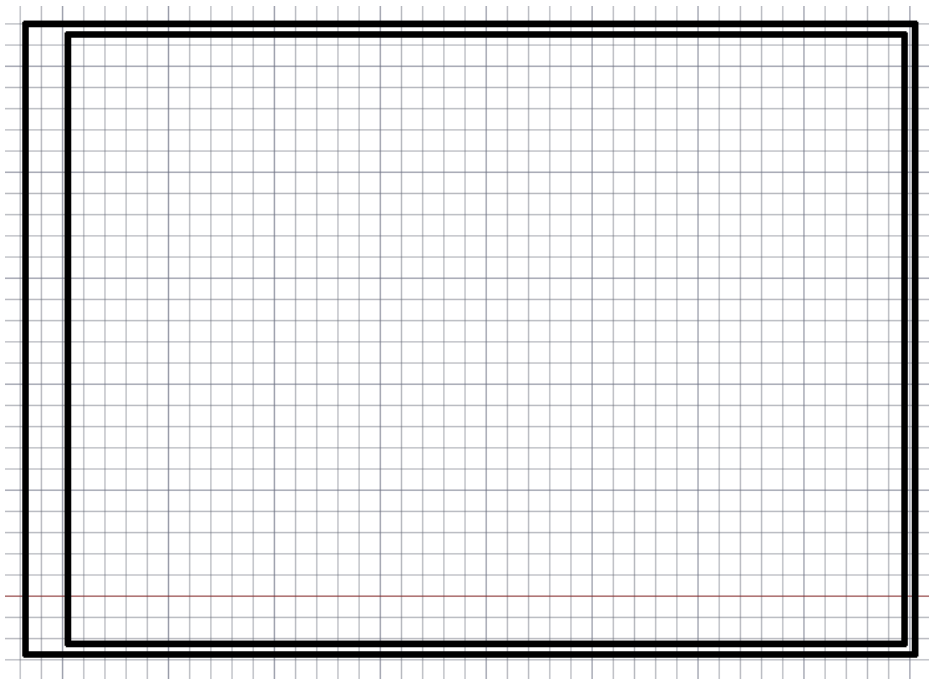


Figure 2.4 – Final version of the sheet and drawing frame

2.2.5 There should be a title block table at the bottom left of the drawing. The title block table includes segments, for construction of which we use the Line tool from the Draw panel. Before that, in the state of the command line, enable the Ortho mode. After selecting this tool, the program in the lower command line will ask you to specify the coordinate of the starting point. Enter the value (415.5) and press the "Enter" key. Point the mouse vertically up, enter a value of 55 and confirm "Enter". Then, from the same point, give the direction to the mouse - horizontally to the left and set the length value equal to 185, "Enter". We continue to build the missing segments further. The dimensions are shown below (figure 2.5).

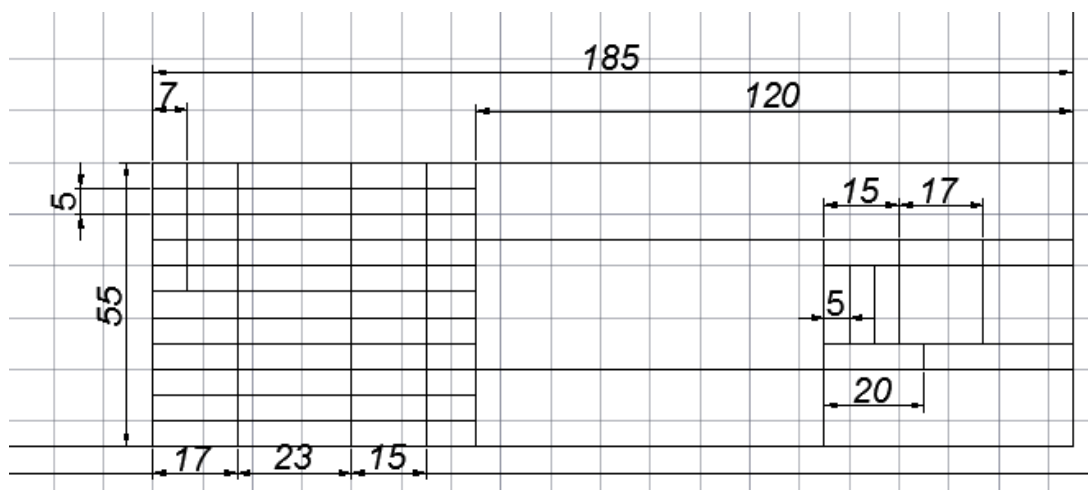


Figure 2.5 – Drawing line segments

2.2.6 After constructing all the segments, it is necessary to arrange the line

thickness in accordance with the standards of GOST 2-303-68. The thickness of the main lines according to the standard is taken in the range of 0.5 - 1.4 mm, let us take it equal to 0.5 mm, and the auxiliary lines from 0.16 to 0.7 mm, let us take it equal to 0.25 mm. To set the lineweight, select the lines, go to the "Properties" panel and set the lineweights from the drop-down menu (figure 2.6).

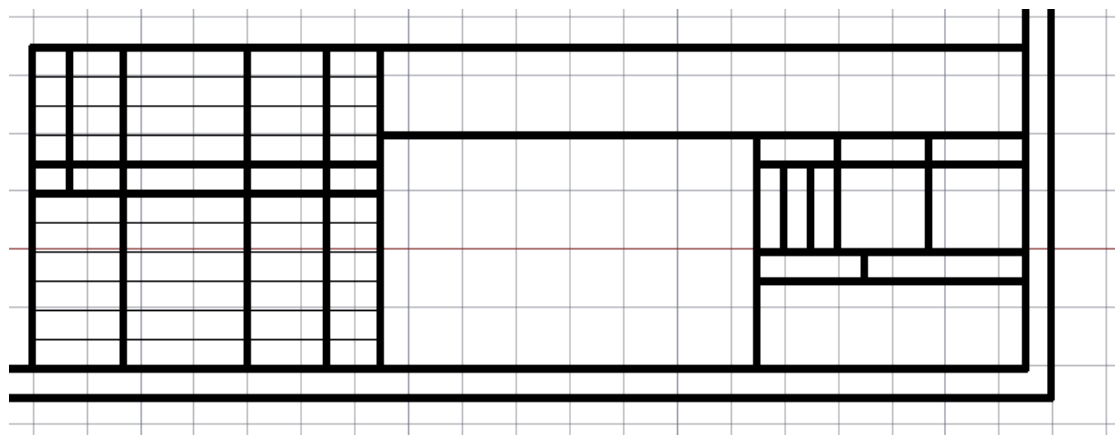


Figure 2.6 – Result of the executed main table

2.3 Content of the report

- goal of the work;
- a description of the installation of the program with screenshots and photos;
- conclusion.

2.4 Defence questions

- 1) What is USDD?
- 2) How thick can the main and auxiliary lines be shown in the drawing?
- 3) According to what standards are the line thickness and the table of main captions on the drawing established?

№3 practical works. Adding text to a drawing

Objective: learn how to insert text, fill in the main captions table and drawing.

3.1 Short theoretical information

The drawing should always consist of explanatory notes, with the help of which the engineer supplements his drawing with the necessary information. Therefore, all inscriptions must be executed correctly and clearly. The drawing font is selected in accordance with GOST 2.304.81, which establishes the following dimensions (height) of the font: 2.5; 3.5; 7; ten; 20; 28. Height is defined perpendicular to the base of the row.

The table of the main inscriptions is placed in the lower right corner of the drawing, its standard is GOST 2.104-2006. This standard specifies the forms of title blocks in drawings.

3.2 Work order

3.2.1 Opening the drawing file created in the second practical lesson.

3.2.2 After opening the drawing, select the "Multiline text" element in the Drawing panel (figure 3.1). By clicking on this icon, a white dynamic window will appear, which must be moved to the place where the text will be placed (figure 3.2).

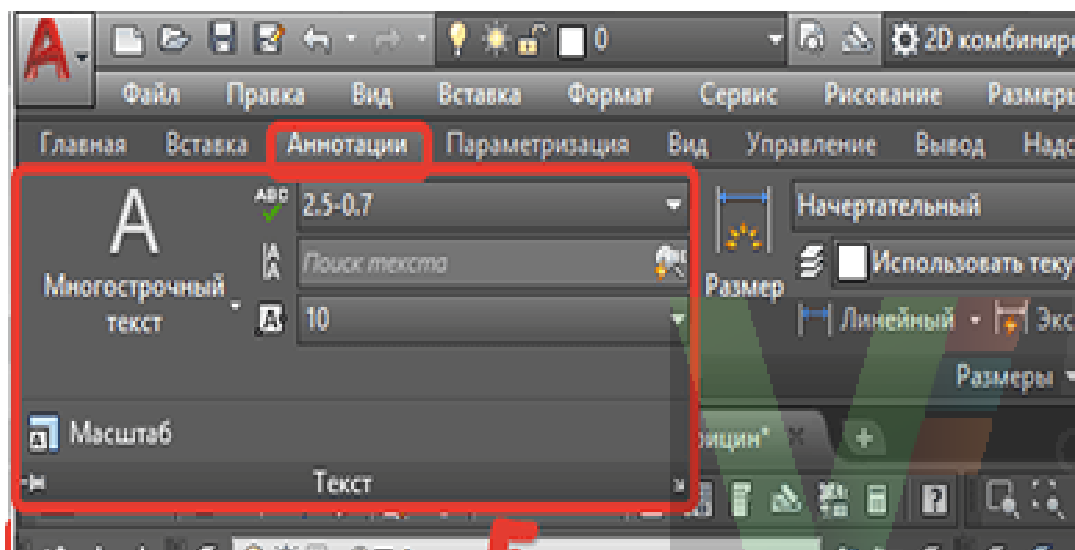


Figure 3.1– Configuration interface

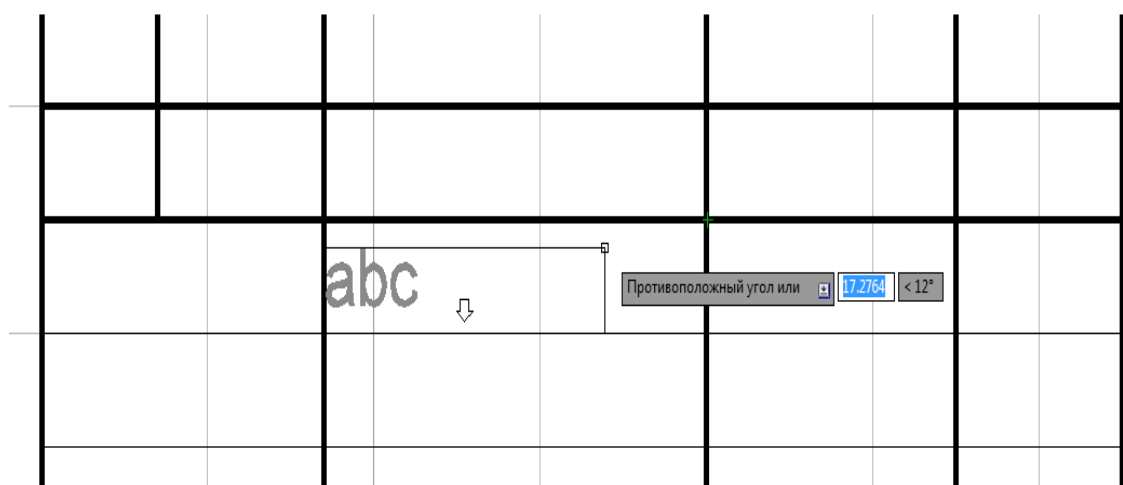


Figure 3.2 – Dynamic frame of the "Text" element

3.2.3 Further, it is necessary to maintain the text, according to Figure 3.3. In order to retreat from the cell line, you must press the space bar. In order to change the

height of the text, the angle of inclination, compression ratio, font and so on, you need to select all the text and change its parameters through the tab "Text" - "Formatting" on the panel "Drawing" (figure 3.3).

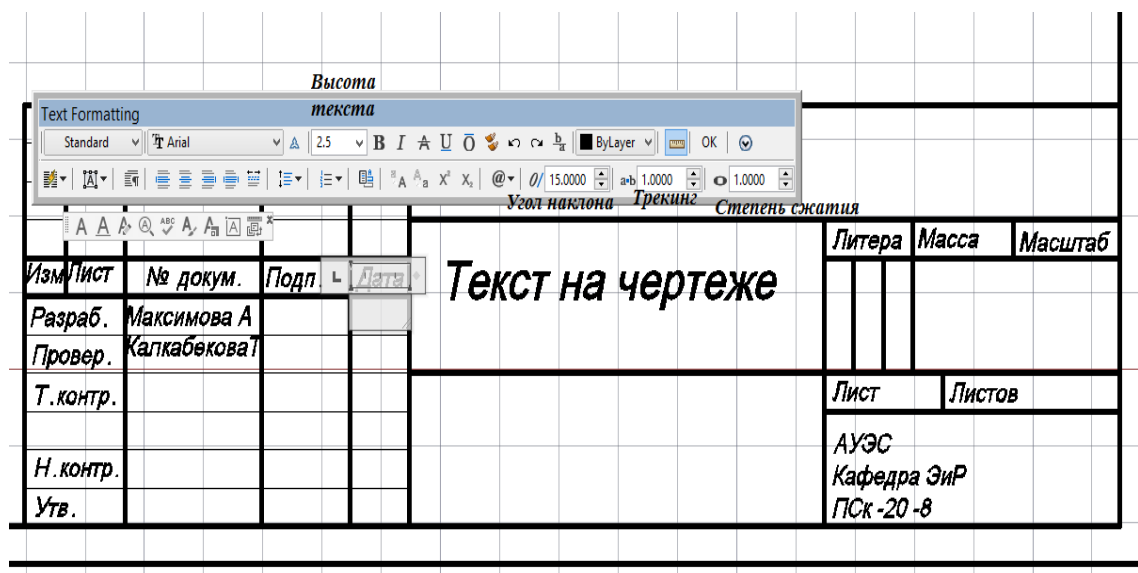


Figure 3.3 – Completed title block table

3.2.4 Now, in case of moving the text to another place, click on the blank space outside the text to exit edit mode. Then, left click on the entered text, it will light up in blue, clamping a square on the symbol, it is possible to transfer the word to any place in the graphics space.

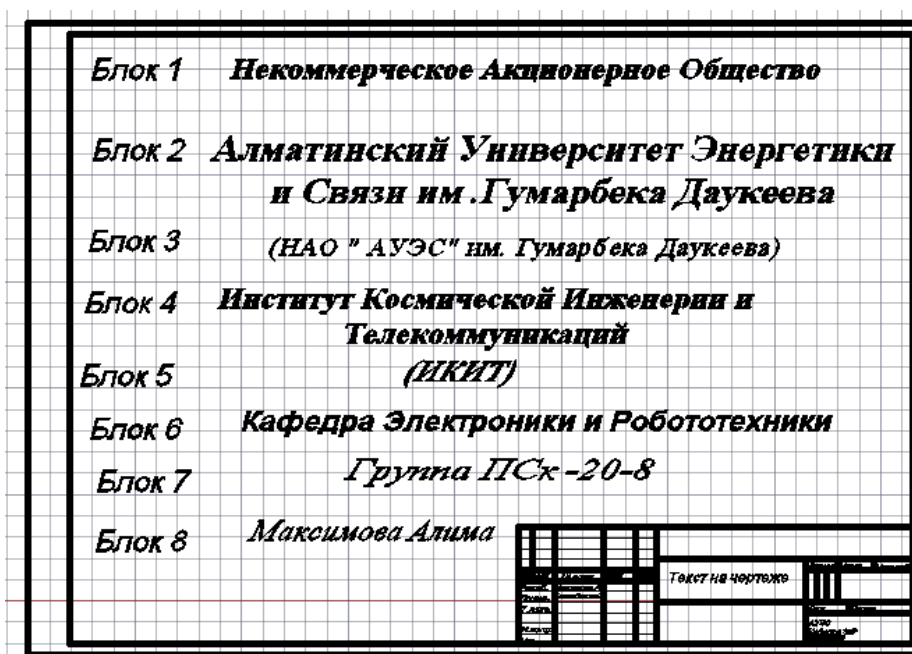
3.2.5 Then, after filling out the table of title blocks, we will fill the drawing itself with the following text, the parameters of which are specified in tables 1.1, 1.2, 1.3, and an example is shown in figure 3.4.

Table 1.1 - Parameters of text elements (the grade book number ends with an even number)

№ block	Text height	Orientation	Tilt angle	Tracking	Compression ratio	Italics	Bolt
1	10	Centered	15	1,5	1,5	No	No
2	14	On the left side	15	1	1	No	Yes
3	10	Centered	1	1	1,2	Yes	No
4	14	On the right side	15	1	1	No	Yes
5	10	Centered	20	2	1,5	No	No
6	10	On the left side	1	1	1	No	Yes

No block	Text height	Orientation	Tilt angle	Tracking	Compres sion ratio	Italic s	Bolt
7	10	Centered	15	1	0,7	No	No
8	10	On the right side	0	2,5	1	No	No

№ block	Text height	Orientation	Tilt angle	Tracking	Compression ratio	Italics	Bolt
1	14	On the right side	15	0,7	1	Yes	No
2	7	Centered	15	1,5	1,5	No	Yes
3	10	On the left side	25	1	1	Да	No
4	14	Centered	15	2	0,5	No	Yes
5	10	On the right side	20	1	2,5	No	No
6	14	Centered	10	0,5	1	No	Yes
7	10	On the right side	15	1	0,7	No	No
8	14	On the left side	0	2,5	1	No	Yes



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3.3 Content of the report

- цель работы;
- отчет о проделанных командах со скриншотами;
- выводы и особенности данной ЛР.

3.4 Defence questions

- 1) Tell us about the GOST 2.304 - 81 standard.
- 2) Explain all the required columns for filling in the title block table.
- 3) Show how you can format the text.

№4 practical work. Linking

Objective: performing linkings for created elements, learning how to use an array, mirroring elements.

4.1 Short theoretical information

A touch is a smooth transition from one line to another. The point at which the touch occurs is called the touch point. Conjugation, in turn, is a smooth transition from one line to another using an auxiliary line (most often, a circular arc).

The drawing is always supplemented with the necessary dimensions, when applying which they are guided by the main sections of GOST 2.307-2011. When drawing dimensions, their actual value is set regardless of the scale of the drawing.

In the drawing, the dimensions are indicated by extension thin solid lines (thickness 0.25 mm). When drawing the size of the line, the extension line is drawn parallel to this segment. When specifying the size of the corner, the dimension line looks like an arc in the center of which the value of the angle is indicated.

When drawing dimension lines, specify dimensions outside of the image thumbnail. It is forbidden to use as dimension lines - center lines, contour lines, as well as auxiliary lines.

4.2 Work order

4.2.1 Create a new drawing, including the frame and the title block table, enter the "Linkings" theme in the title block.

4.2.2 To begin with, let's draw auxiliary lines using the "Line" command on the "Draw" panel. We will build auxiliary and horizontal lines of any length at any point in the graphics space. For the lines to be strictly vertical or horizontal, turn on the "Ortho" mode in the status bar (figure 4.1).

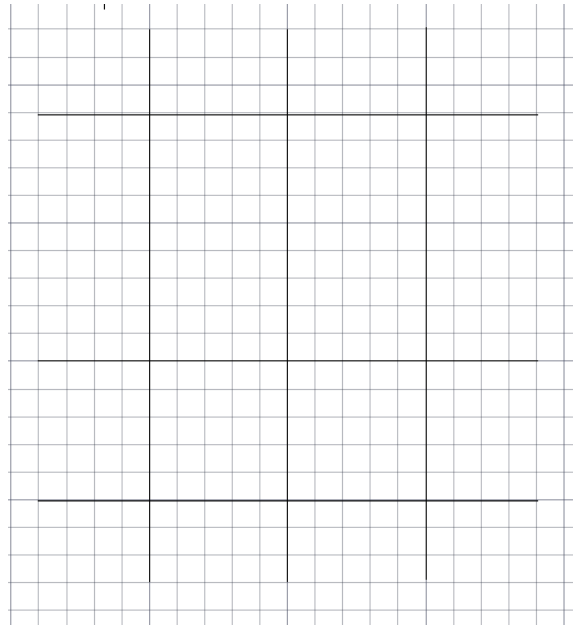


Figure 4.1 – Construction lines in the drawing

4.2.3 We draw circles. To do this, select the "Circle" command from the "Draw" panel. Then, on the command line, the program will ask you to specify the center (center is the point of intersection of the auxiliary lines, in the center of the drawing), and will also ask you to specify the diameter. Enter the value of the circle diameter equal to 20 mm into the command line. The remaining circles have diameters of 50mm, 80mm, 40mm, 26mm, 20mm and 34mm. We build them according to the same principle, setting the centre's at the intersection points of the auxiliary lines (figure 4.2).

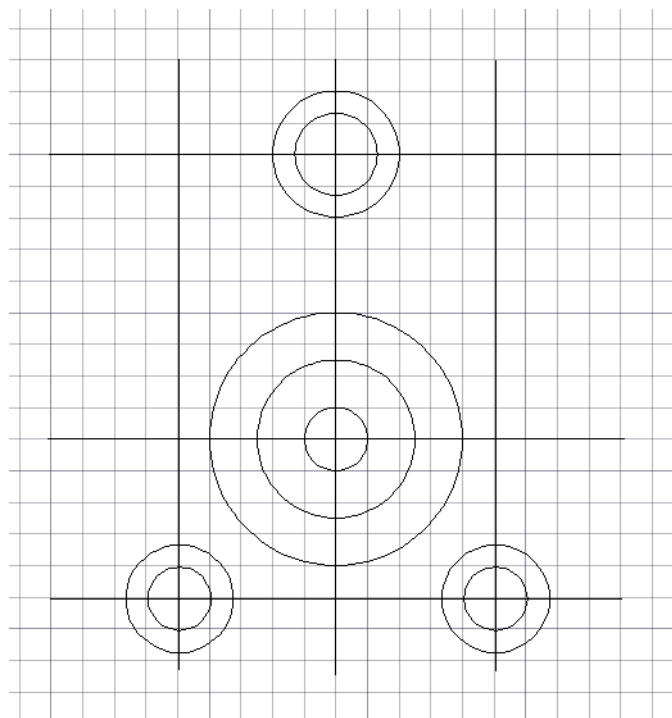



Figure 4.2 – Circles drawing

4.2.4 Now let's move on to creating mates. To do this, select the "Fillet" element in the "Editing" panel or click on the icon . The program in the command line will ask for the fillet radius, enter the value 50 mm. Click on a circle with a diameter of 40 mm and on a circle with a diameter of 80 mm, press "Enter" (figure 4.3).

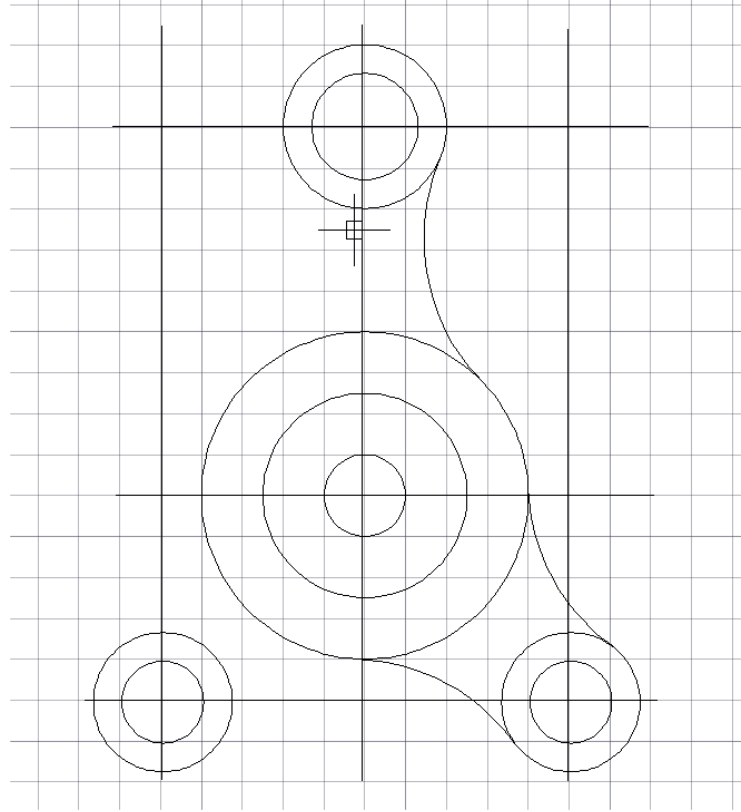


Figure 4.3 – Linking circles

4.2.5 Then, we will make a mirror image of the part about the construction line. For this we select the "Mirror" element from the "Editing" panel. After that, you need to select 3 constructed arcs. Next, you need to select two points on the axis of symmetry (on the auxiliary line in the center) and press "Enter", having answered "No" to the request to delete objects in the command line (figure 4.4).

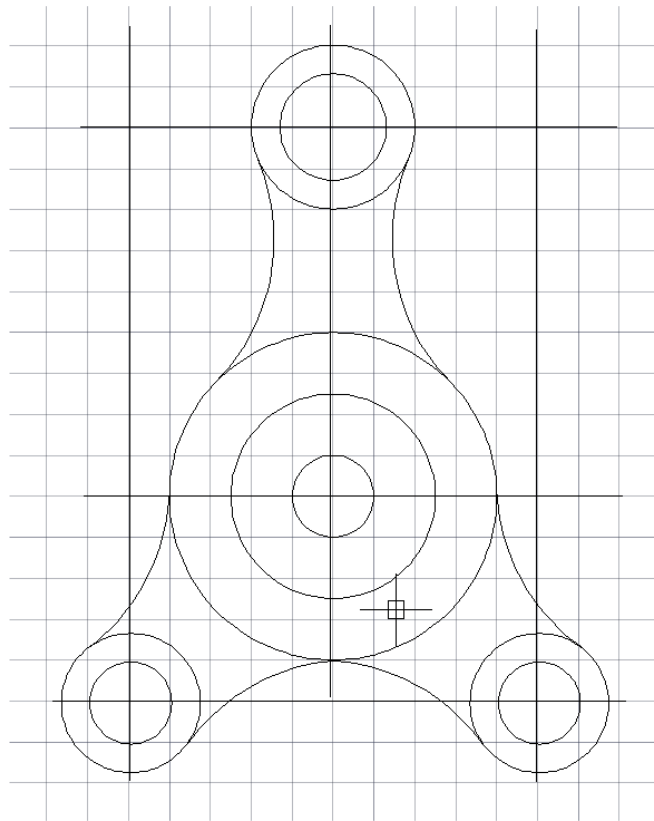


Figure 4.4 – Mirror reflection

4.2.6 Next, let's take a look at arrays. Before that, set the thickness of the main lines equal to 0.5 mm, and the auxiliary ones - 0.25 mm. Selecting the element "Circle", draw a circle with a diameter of 12 mm in the upper part of the large circle, designate the thickness of the lines equal to 0.5 mm. Click in the "Editing" panel on the "Array" element, select the circular one. Click on the constructed circle and on the center point of the array, press "Enter" (figure 4.5).

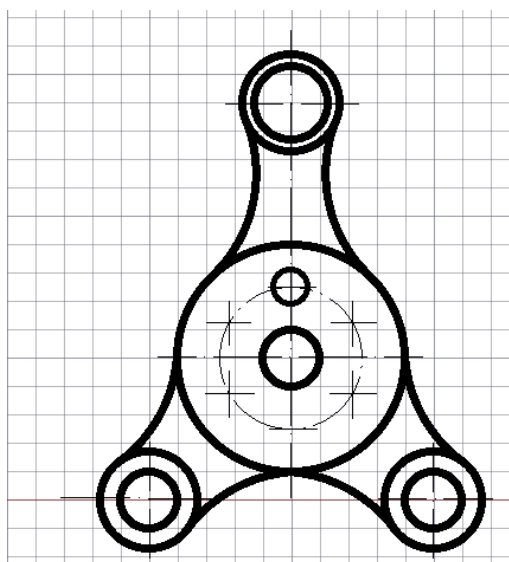


Figure 4.5 – Using a circular array

4.2.7 Delete construction lines using the Erase command from the Modify panel. And then draw the centerlines, having previously loaded them into the "Line Palette". You will need to select the centerline of the ACAD_ISO04W100 type. Set the weight of the main lines to 0.5 mm and the centerline to 0.25 mm. Then, in the "Annotation" panel, select the "Size" element and put down all the necessary dimensions in the drawing (figure 4.6).

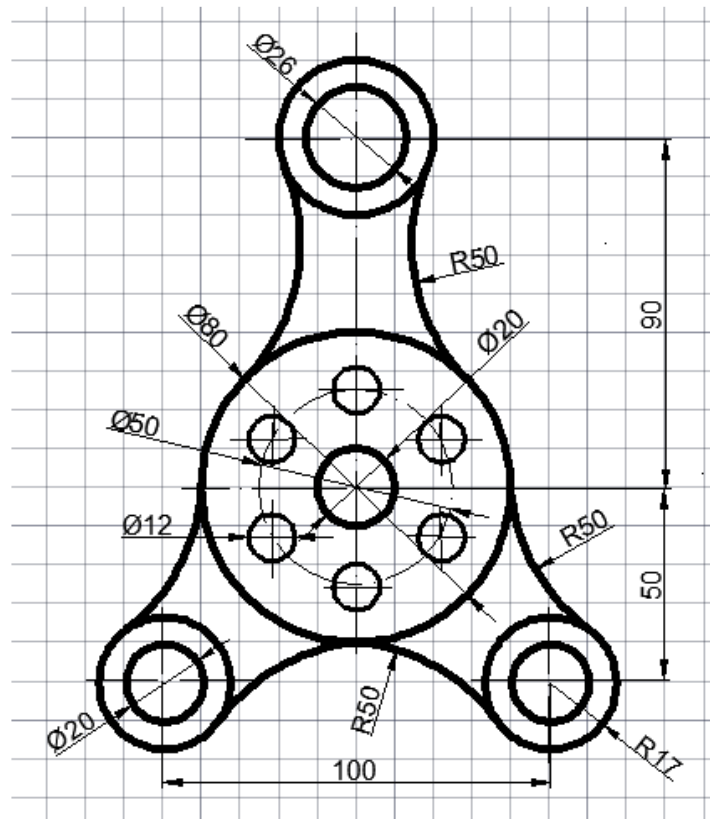


Figure 4.6 – Inserted dimensions of the part

4.2.8 Transfer the created part with fillets to the drawing frame, fill in the title block table.

4.2.9 Next to the completed drawing, complete the drawing of the part yourself according to the option according to the list of the magazine. Options for tasks are given in Appendix A.

4.3 Content of the report

- goal of the work;
- progress report with screenshots;
- conclusions and advantages of management through a graphical environment.

4.4 Defense questions

- 1) What is linkings?

- 2) According to which standard are dimension lines applied?
- 3) Tell us about the features of the dimension lines.

№5 practical work. Creation of 3 types of parts

Objective: to learn how to carry out a drawing of a part from three angles: front view, top view and left view. Set line thickness and dimension.

5.1 Short theoretical information

View - an image that shows the visible part of the surface of the part facing the observer. During the construction of drawings, it is necessary to build a complex drawing of a part, consisting of two or more interconnected orthogonal projections. GOST 2.305-68 establishes the name of the main types as: front view, top view, left view. The front view is the main view, which gives a clear, most complete description of the shape and dimensions of the drawing. If you look at the object from above, perpendicular to the horizontal plane of the projection, you can get a top view. The left view can be obtained by looking at the image at right angles to the profile plane of the projection.

The main view is always arranged in a certain order: the left view is to the right of the main view, and the top view is under the main view.

5.2 Work order

5.2.1 Let's create a new drawing in AutoCad, transfer the drawing frame and the table of title blocks to the working graphics space. Let us edit the title block table, changing the theme to "3 types of part".

5.2.2 In the course of the drawing, we use such AutoCad commands as "Circle", "Rectangle", "Line" and others, which are located on the "Drawing" panel. Also, if the part is symmetrical, start by drawing centerlines. For example, take the following detail (figure 5.1).

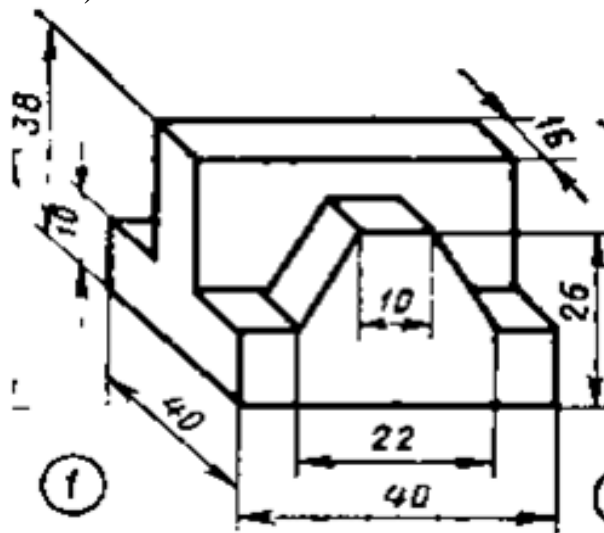


Figure 5.1 – Isometric part model

5.2.3 First, let's divide the graphic space into 4 squares by drawing 2 segments with a thickness of 0.5 mm.

We will start by building the main view, hence starting the drawing from the top left square. It is noticeable that the part is completely symmetrical, therefore, we will start by building the centerlines. Create a new layer and set it in the item "Linetype"- "ACAD_ISO04W100", line weight 0.25 mm. We draw the center line, setting the starting point of the segment at the point (0,0) (figure 5.2).

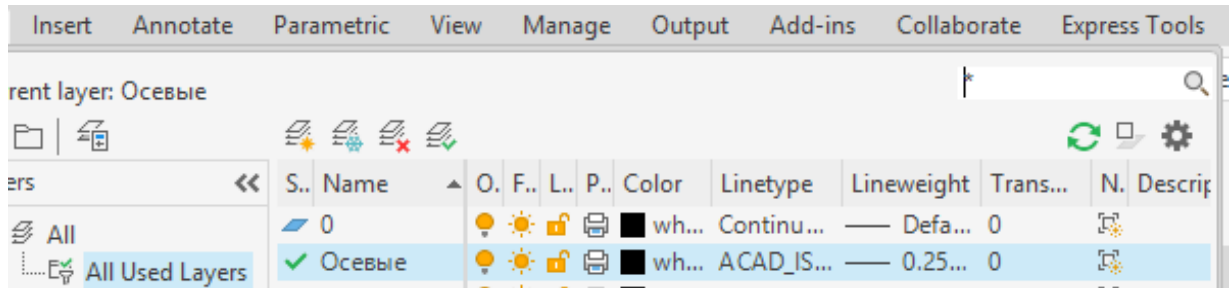


Figure 5.2 – Create a layer for creating centerlines

5.2.4 Next, go back to layer "0". In the zero layer, you need to change the thickness of the main lines from "Default" to 0.5 mm. and draw the left half of the main view using the Line or Polyline tool. Let's pay attention to the isometry of the part for sizing. Then, with the help of the "Mirror reflection" tool, complete the entire frontal view of the part (figure 5.3).

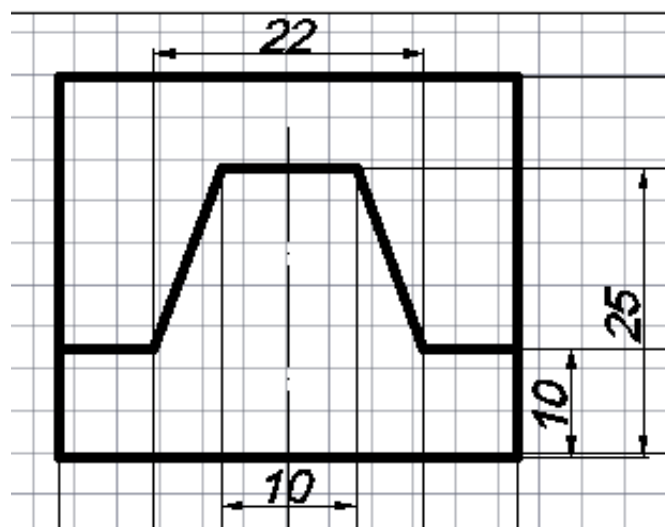


Figure 5.3 – Drawing the main view

5.2.5 Next let's build a top view. To do this, to begin with, draw all the auxiliary lines from the main points of the top view. Let us define which lines are auxiliary and which are main. Let's put down the appropriate dimensions (figure 5.4).

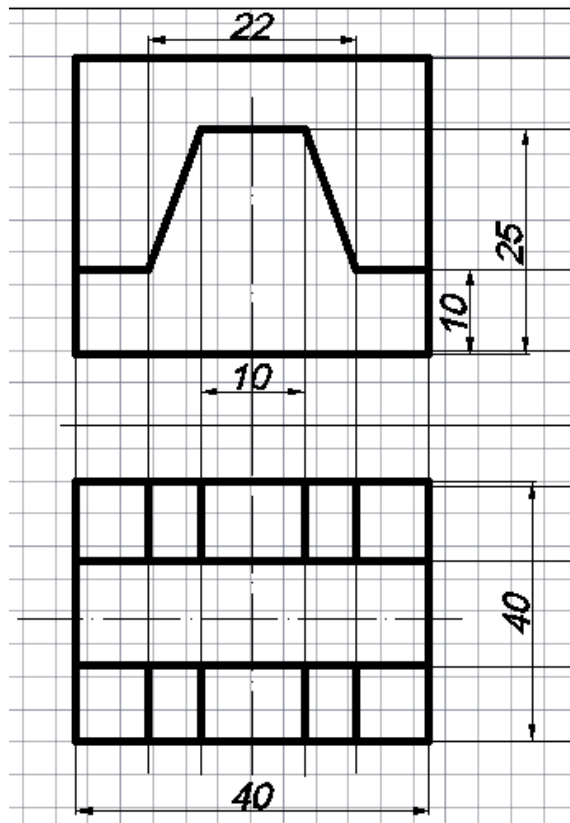


Figure 5.4 –Drawing a top view

5.3.5 Then, delete all the construction lines that were drawn to create the top view. To build a view on the left, you need to draw an inclined line at an angle of 45° from the point of intersection of the straight lines dividing the space. Further, from all the anchor points of the top view, draw auxiliary lines to create a view on the left. Change the thickness of the main lines and set the dimensions (figure 5.5).

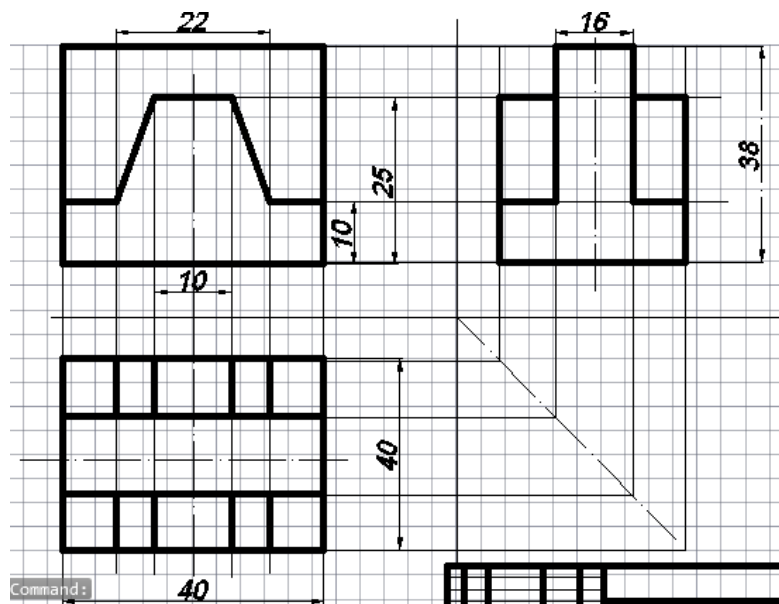


Figure 5.5 – Details 3 views

5.3.6 Delete all construction lines, except for the center lines, fill in the title block table and the drawing is ready. Perform a similar task according to the option (Appendix B).

5.3 Content of the report

- goal of the work;
- report on the work done with the corresponding screenshots;
- an explanation of the drawing creation process.

5.4 Defence questions

- 1) What is a drawing view?
- 2) What are the types of parts in the drawing?
- 3) Tell us about local and additional species.

№6 practical work. Modeling a 3D object

Objective: to learn how to work in the AutoCad 3D modeling environment, be able to use the basic functions of this mode and create a three-dimensional object according to the option.

6.1 Short theoretical information

A product is any item or set of production items to be manufactured in an enterprise.

Part - a product made of a material of the same name and brand without the use of assembly operations, for example: a roller made of one piece of metal, a cast body, a printed circuit board. Parts also include box-type products (glued, welded, welded) from one piece of sheet material (cardboard, sheet steel).

Parts of a part that have a specific purpose are called its elements, for example: chamfer, groove, rib, etc.

The types and completeness of design documents for products are established by GOST 2.102-68. One of the most common is a drawing of a part - a document containing an image of a part and other data necessary for its manufacture and control.

6.2 Work order

6.2.1 Before starting work, you need to enable such modes as "Ortho", "Object Snap", "Line Thickness". And also in the tab "Switching workspace" or "Workspace Switching" select the mode "3D Basics" or "3D Basics" (figure 6.1).

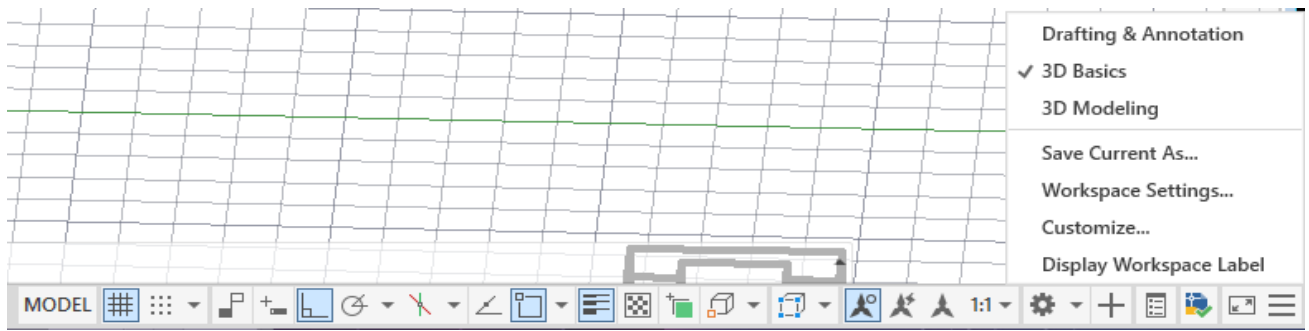


Figure 6.1 – Switch to 3D Basics

6.2.2 Go to the tab "Home" => "Layers and View" and set the view "SW isometric", and then select the tool "Box" or "Box" (figure 6.2).

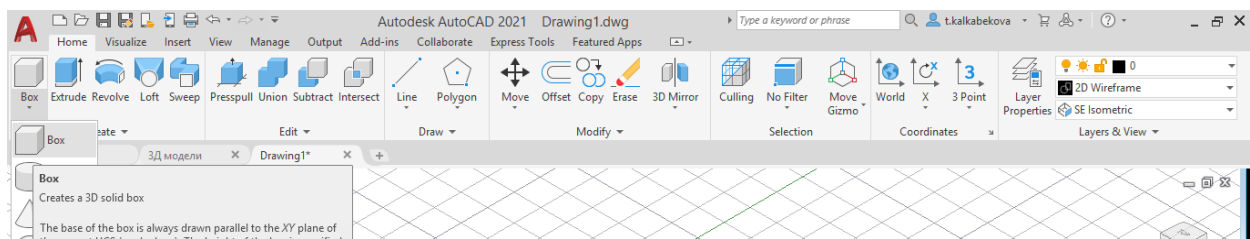


Figure 6.2 – Using the Drawer tool

6.2.3 With the Drawer tool selected, perform the following actions at the command line:

- 1) Show the first corner in an arbitrary place;
- 2) To define a different angle, click on the word "Length", then use the cursor to show the direction and enter the value 125 in the command line, press "Enter";
- 4) Next, in the command line, the program will ask for the width value, enter the value 78 and press "Enter";
- 5) At the request of the program, the height value, enter the value 24 and press "Enter".

As a result, we get the following box (figure 6.3).

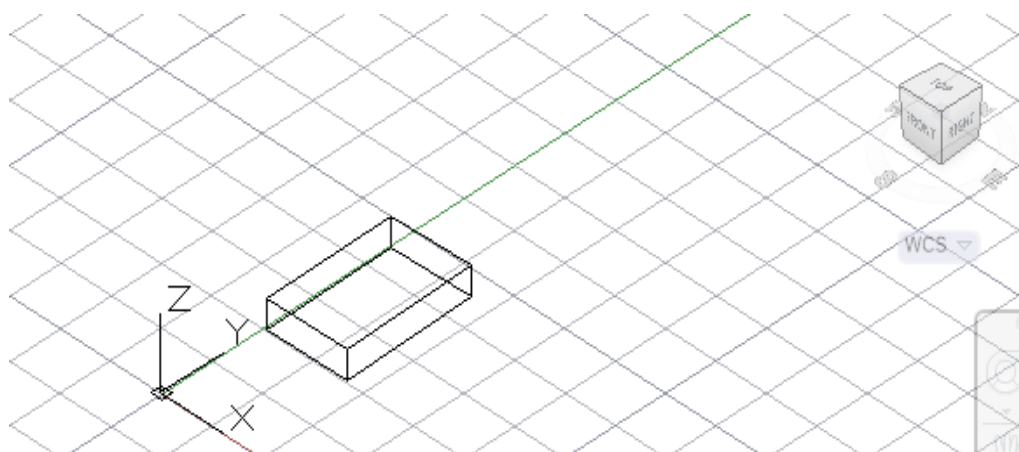


Figure 6.3 – Prismatic base

6.2.4 Then you need to make fillets based on the part, on one side, using the Fillet command. Select the tool in the "Ribbon" - "Fillet" panel and perform the following actions:

- 1) Choose any vertical edge of the prism;
- 2) At the request of the system in the command line, enter the value of the radius of conjugation equal to 15 and press "Enter";
- 3) Then, the program will ask you to select an edge or [Chain / Radius], click on the Chain option;
- 4) With the cursor, click on the second vertical edge and press "Enter".

As a result, we get the following image of a box with rounded corners (figure 6.4).

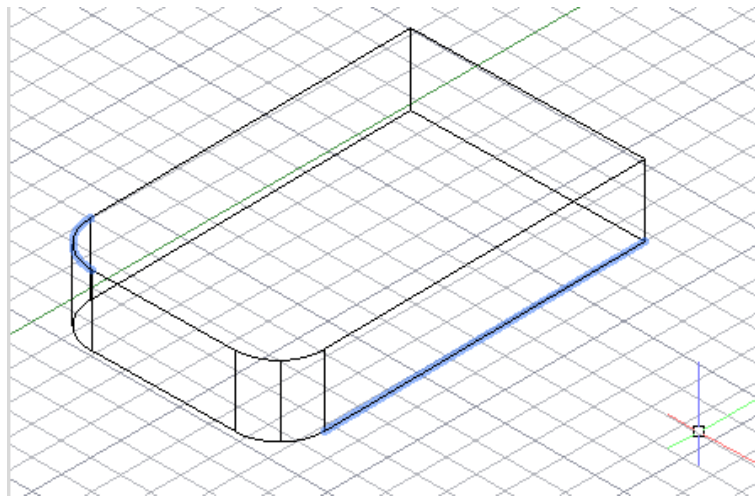


Figure 6.4 – Rounding the corners of a prismatic base

6.2.5 Then we use such tools as "Drawer", "Transfer" and "Subtraction".

To begin with, let's build, similarly to paragraph 6.2.3, a box with dimensions: length - 95, width - 30, height -8. Then we rotate the main model so that the base of the prismatic hole is visible. Next, use the Move tool and move the parallelepiped created at this point to the base of the object. To do this, we will perform such actions as:

- 1) Select the "Move" tool, click on the box created in this paragraph, and press "Enter";
- 2) Using object snaps, "Middle", pick the first point on the base of the box;
- 3) The second point must also be selected in the middle of the base of the part.

The result is shown in figure 6.5.

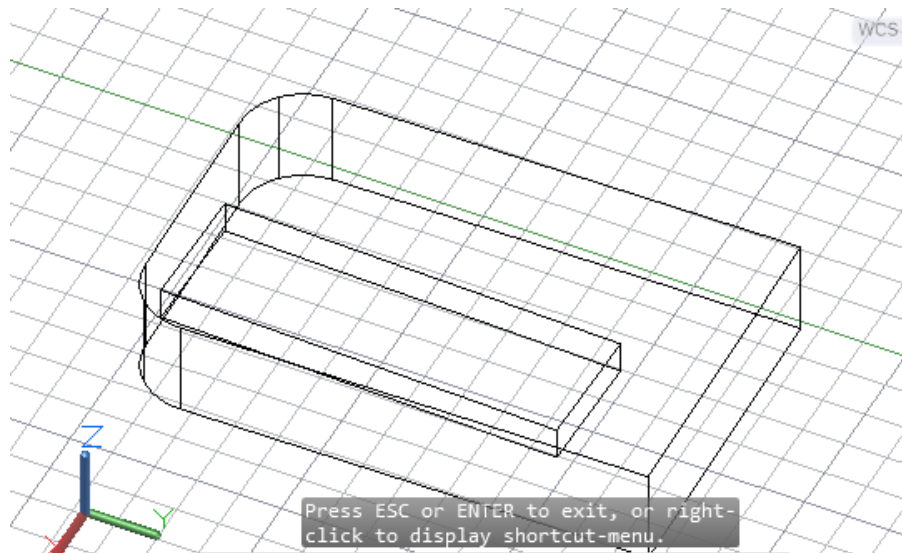


Figure 6.5 - Using the "Move" command

Then, click on the "Subtract" command. After that, you need to click first on the main part and press "Enter". For the command to work, you need to select the element that is subtracted. In our case, this is the box delivered in clause 6.2.5 and press "Enter". The result is shown in figure 6.6.

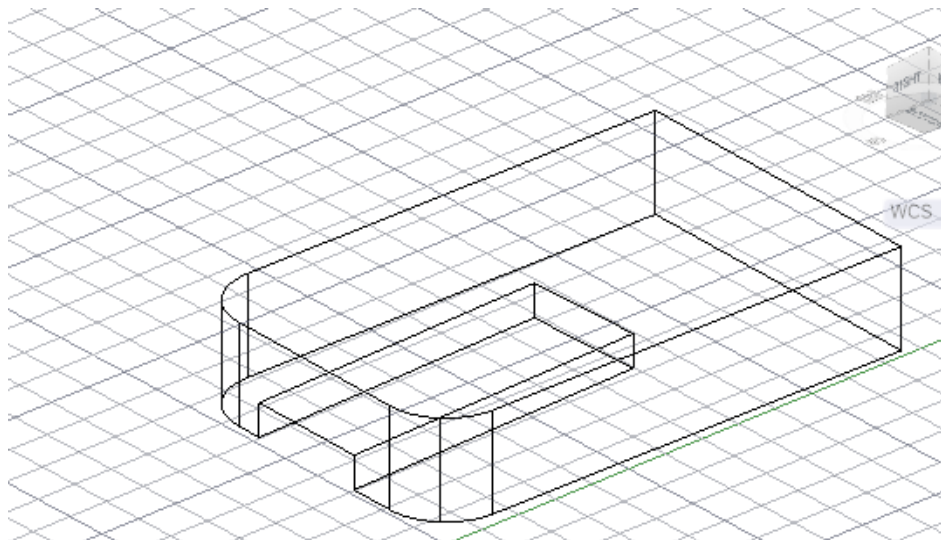


Figure 6.6 – Cut groove in the base

6.2.6 The next step is to draw a cone on the upper plane of the object. For this, select the "Cone" element in the "Ribbon". Using the "Middle" object snap and the "Object tracking" mode, you need to step back a little from the middle of the left horizontal edge of the prism, click on the found point. Next, in the command line, enter the value of the radius of the lower base equal to 28 mm. Click in the command line on the "Radius of the upper base" and set the value -15mm, the height of the cone is 16 (figure 6.7).

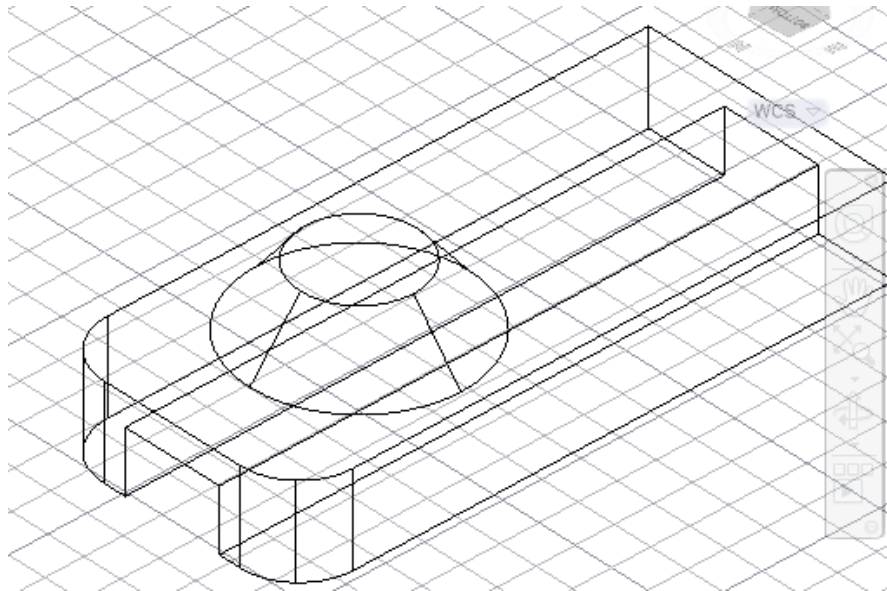


Figure 6.7 – Constructing a conoid

6.2.7 The next step is to build a through prismatic hole. For this, commands such as "Polygon", "Extrude" and "Subtraction" are used.

The first step is to build a polygon at the bottom of the cone. To do this, click on the "Polygon" tool in the "Draw" panel, indicate the number of sides equal to 4, set the center to the center of the upper circle of the cone. Select the option "Described around a circle", the radius of the circle is 8.

Next, you need to extrude the previously created polygon to the base of the object. Select the Extrude command from the Modeling panel, click on the polygon, set the extrusion height to 32 and the direction down to the base.

Then, apply the Subtract command and subtract the previously created quadrangle from the base of the part and the frustum surface. To make it convenient to visualize the object, click in the "Layers and View" tab on the "Home" panel, select the "Conceptual" rendering style (figure 6.8).

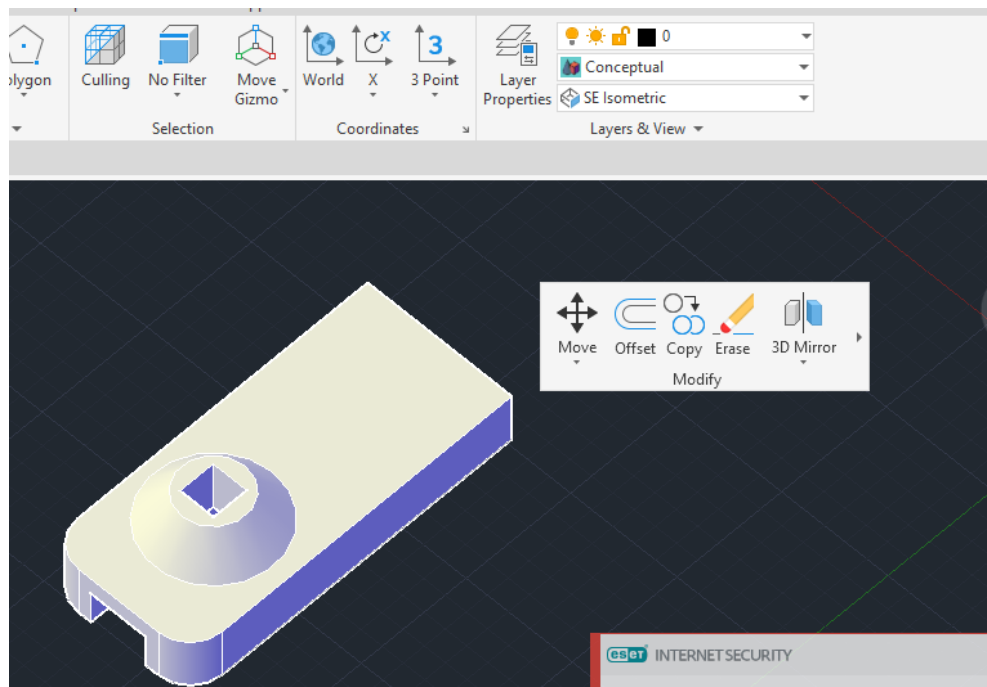


Figure 6.8 - Quadrangular hole in object

6.2.8 The final step is to add a wall to the object on the opposite side of the conoid.

First, let us change the SW isometry to a top view. Draw on the bottom of the model a rectangle with dimensions: length 78, width 20. Then, return to the SO isometry, click on the Extrude command and raise the Box to a height of 30 mm (figure 6.9).

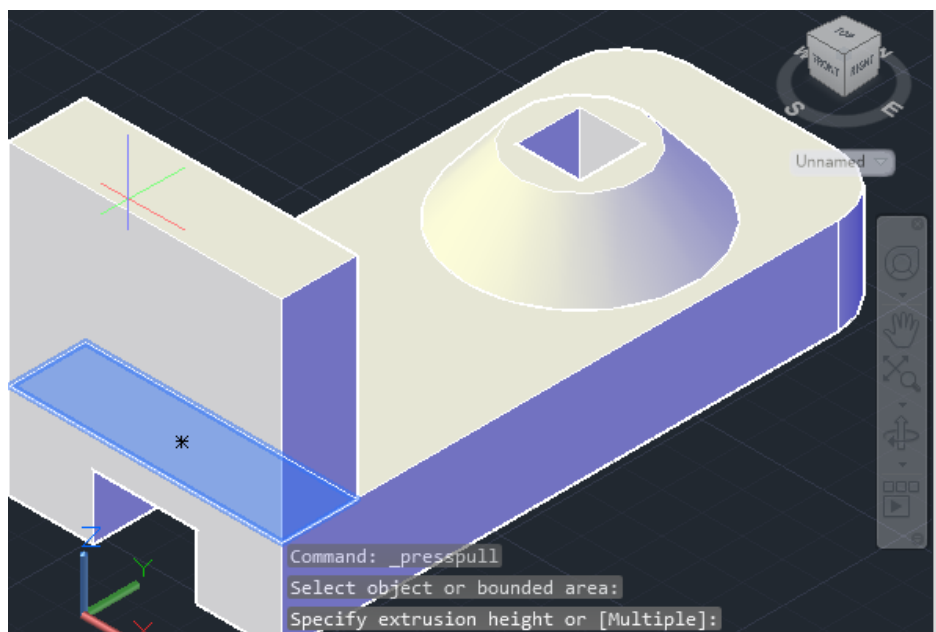


Figure 6.9 – Extruded object wall

At the end, you need to go back to the top view, draw a rectangle in the center of the extreme wall, squeeze it inward by 15 mm and subtract it from the wall (figure 6.10).

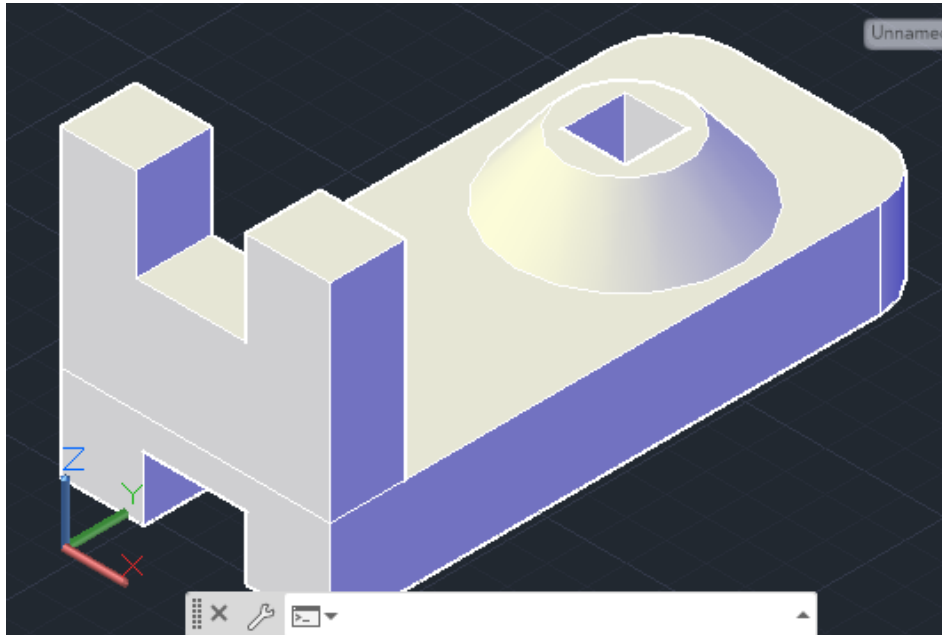


Figure 6.10 – Final model

6.2.9 Execute, according to the option according to the list, a drawing of a three-dimensional model taken from Appendix B.

6.3 Content of the report

- goal of the work;
- a report on the work done with the corresponding screenshots and explanations for each;

6.4 Defence questions

- 1) What is a part?
- 2) Give a definition to the concept of a product.
- 3) What standard establishes the types and completeness of design documents for products?

№7 practical work. Building a solid model in the SolidWorks environment

Objective: acquiring practical skills in constructing such objects as: rectangle, circle, segment; learn to use the program tools such as Extruded boss, Extruded cut.

7.1 Short theoretical information

SolidWorks is a 3D design system using a Windows graphical interface. The advantage of the system is its complete Russification. All teaching aids and SolidWorks user manual, as well as menus and user interface are implemented in Russian in all versions.

Product design is the process of creating a set of technical documents required to manufacture a product. The end result of the design is the creation of a set of design documentation intended for the manufacture of a product.

A solid model is a three-dimensional image of a spatial object that fully imitates its physical and mechanical properties.

7.2 Work order

7.2.1 Get familiar with the part model (figure 7.1).

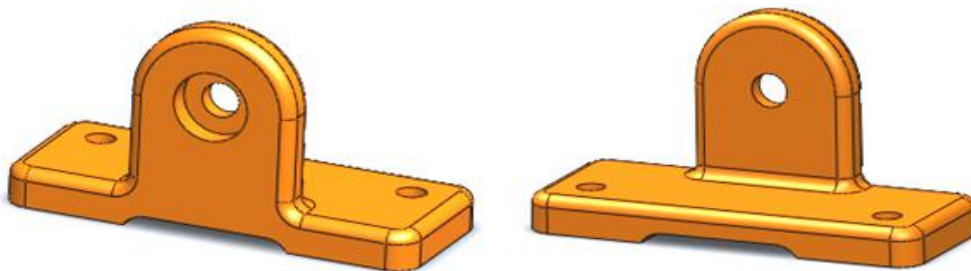



Figure 7.1 – Part model

7.2.2 After analyzing the model, it is noticeable that the part consists of 2 parallelepipeds, one of which has a round hole, and the edges of which are rounded.

7.2.3 To create this model, click on the "New SolidWorks Document" button, select the "Detail" element and click on the "OK" button.

7.2.4 Let's start the construction process with a horizontal parallelepiped. On the left side of the program there is a "Design tree", where we select the "Top" view. By clicking on the "Above" view once, a floating panel will open in front of you. Click on the first button "Sketch"  (figure 7.2).

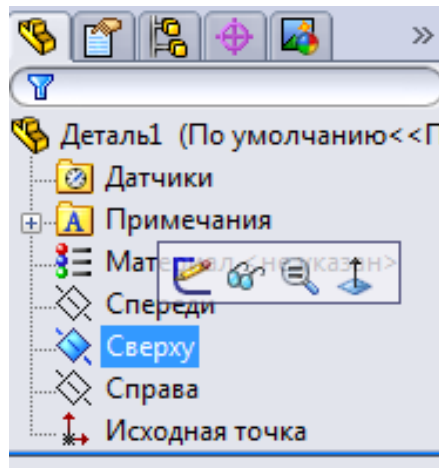





Figure 7.2 – Sketching a top view

7.2.5 In the upper part of the panel, click on the "Rectangle" tool , fix the first point in the center of the origin, draw a rectangle. The dimensions of the rectangle are as follows: length 100 mm, width 40 mm. To set these values, click on the "Automatic dimensioning" icon , we put down the dimensions. Upon completion, click on the "OK" icon . Then we exit the sketch by clicking on the arrow button in the upper right corner of the drawing (figure 7.3).

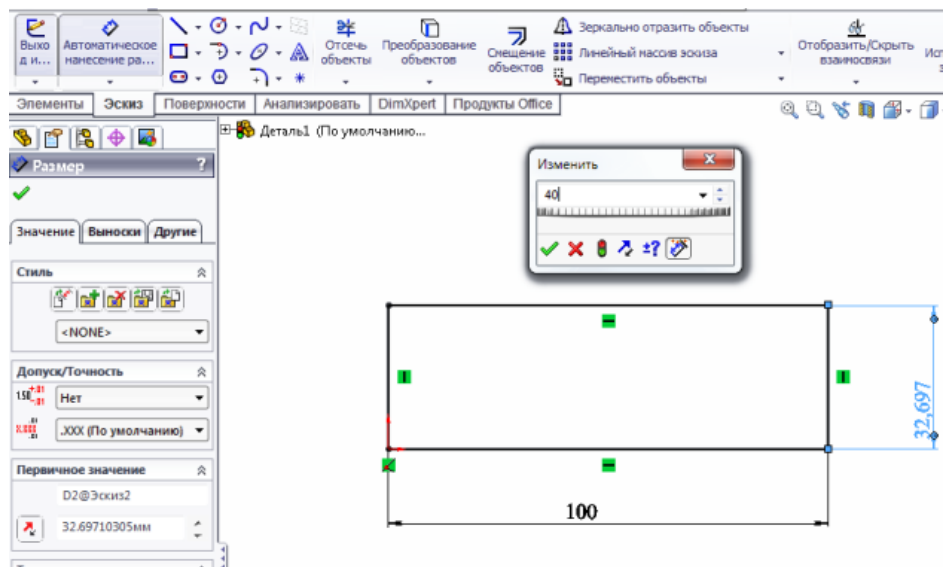



Figure 7.3 – Constructing a rectangle

7.2.5 After the sketch mode is closed, you need to extrude the object. To do this, in the ribbon panel, in the "Elements" section, select the "Extruded boss" tool . The direction of pulling, as well as the parameter by how much the part is pulled, are shown in the figure below.

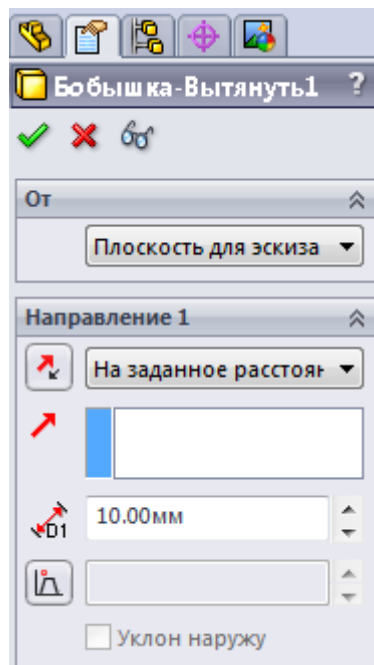



Figure 7.4 – Pulling the boss

7.2.6 Next, as a sketch plane, click on the side face of the part, perpendicular to which the second parallelepiped will be located. You need to dimension and add all the necessary relationships using the Add Relationship tool . Exit sketch. Extrude the created feature with the Extruded Boss tool 10 mm. The result is shown in the figure below.

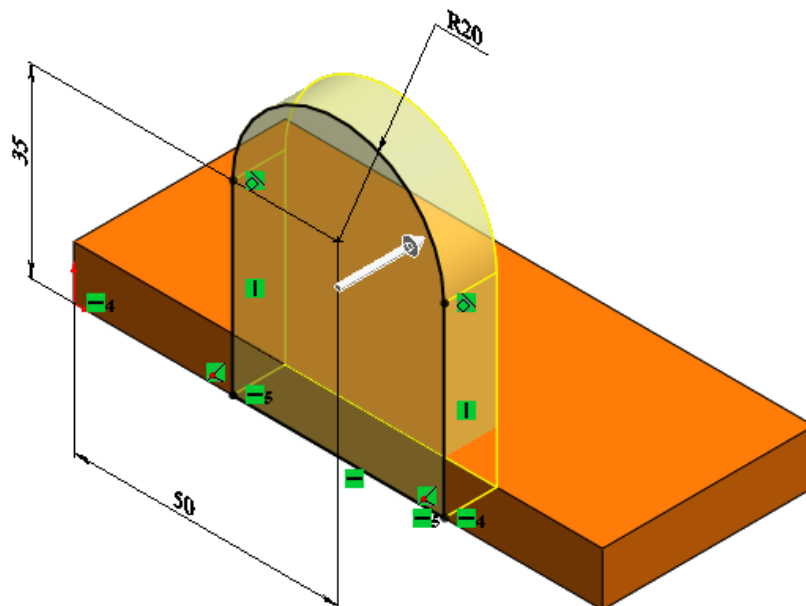



Figure 7.5 – Extruding the sidewall of a part

7.2.7 A horizontally positioned object has a base cutout. To create it, you first need to select a sketch plane. In our case, we create a sketch on the "Front" plane. We draw a rectangle with a width of 3 mm, a length of -50 mm. Establish a "Collinear"

 relationship between the vertical sides of the rectangle and the vertical edges of the vertical parallelepiped. We put down the dimensions of the rectangle (figure 7.6).

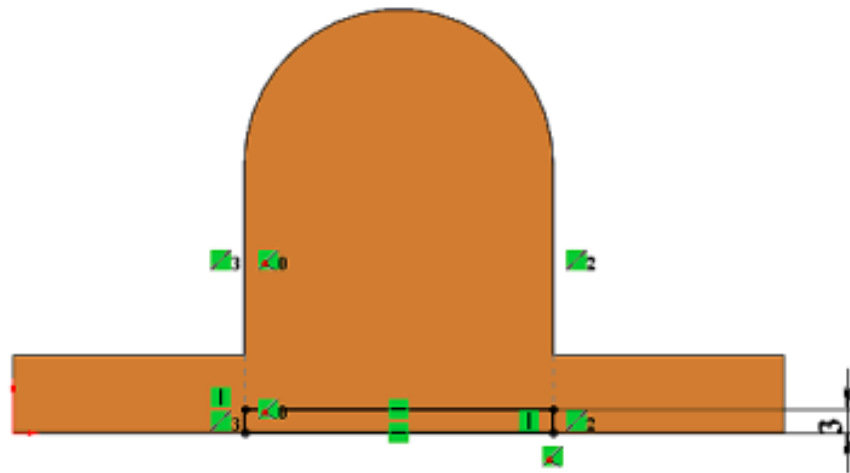



Figure 7.6 – Creating a cut along the base of a part

7.2.8 Click on the Extruded Cut tool to create a through hole . The result is shown in the figure below.

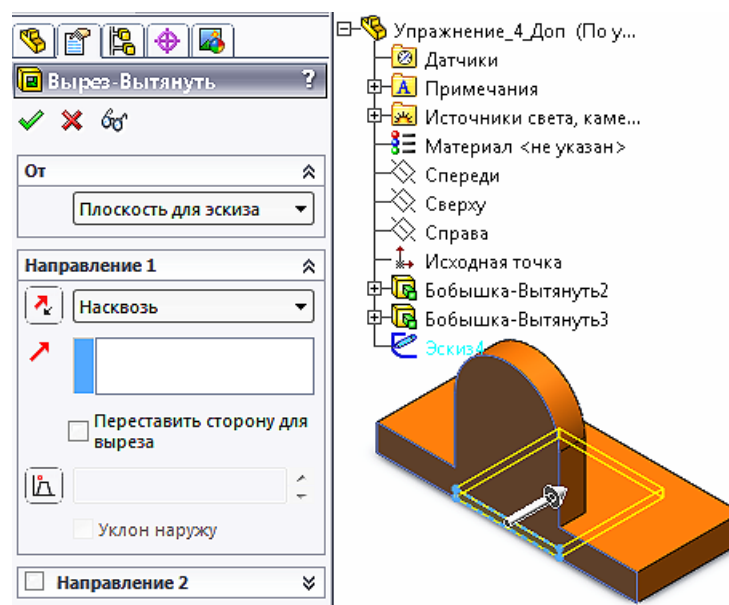



Figure 7.7 – Extruded cut out at the base of the model

7.2.9 Create a hole in the front face of the vertical box. Why find the command "Insert" => "Elements" => "Hole for fasteners" in the main menu . Then, click on the desired face, on which the hole will be located, set its parameters according to figure 7.8, "End condition" - "Through and through".

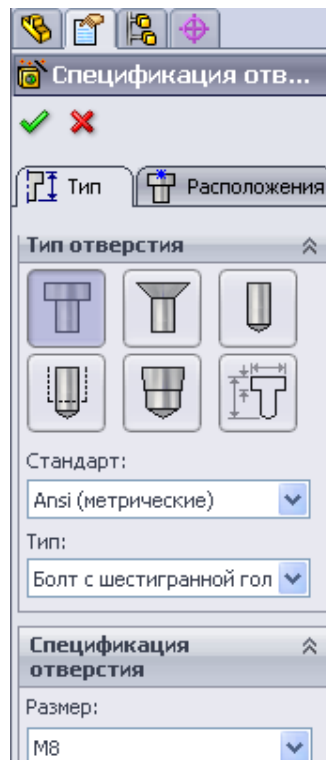



Figure 7.8 – Hole parameters

7.2.10 The final step is to round the edges. You must select the Fillet tool from the Elements toolbar . Pick edges by clicking on them in the model and pick a radius of 10 mm.,

7.3 Content of the report

- goal of the work;
- a description of the actions taken;
- screenshots of command execution results;
- conclusion.

7.4 Defence questions

- 1) What is SolidWorks?
- 2) Describe the solid model, definition and application, and its properties.
- 3) Show how to extrude a boss by a specified distance and create a cut on a part.

№8 practical work. Modelling a needle radiator in SolidWorks

Objective: to study the basics of hardware modelling using the SolidWorks environment.

8.1 Short theoretical questions

Temperature is one of the main criteria for the performance of semiconductor devices. The devices heat up as they work. To prevent overheating, the device must be cooled using an active or passive cooling method. In order to select the appropriate cooling method and its technical implementation, methods of modelling heat and mass transfer or modelling of heat removal are used. For this purpose, the Flow Simulation SolidWorks package is an assistant.

8.2 Work order

8.2.1 Create a new "Part" and name it "Needle Radiator". After that, in the "Design Tree", click on the "Above" plane and switch to the "Sketch" mode.

8.2.2 Then, on the "Top" plane, create a sketch of a rectangle that will be the base of the radiator. We set its dimensions as shown in figure 8.1.

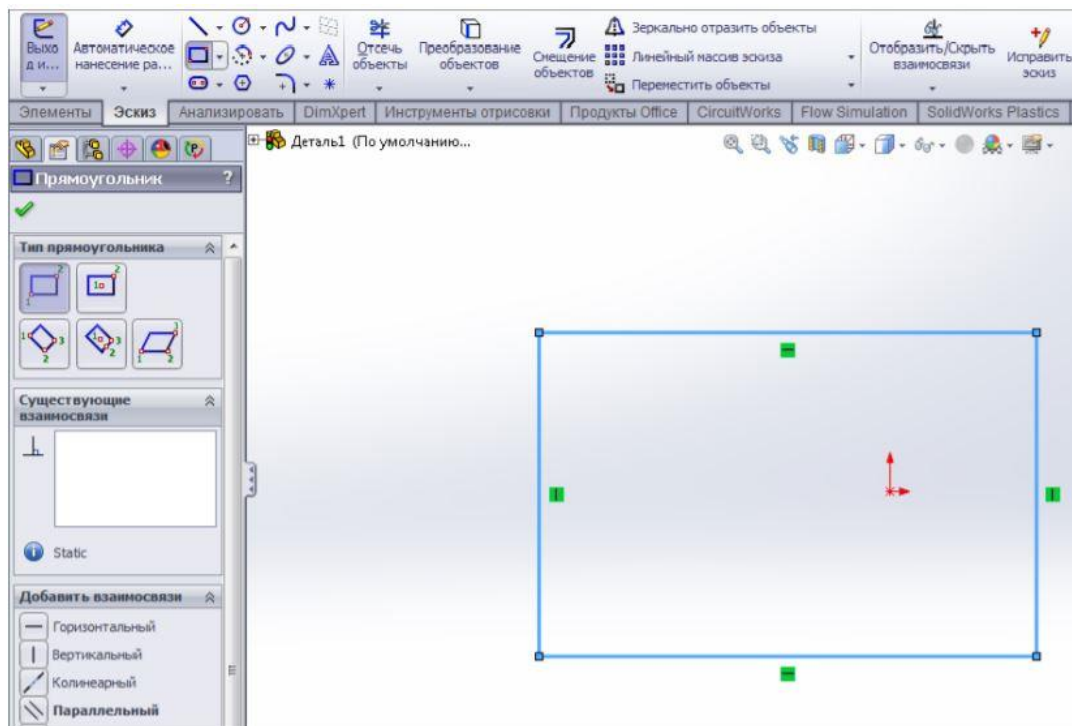


Figure 8.1 – Rectangle sketch

8.2.3 Next, using the Extruded boss tool, pull out the radiator base by 5 mm (figure 8.2).

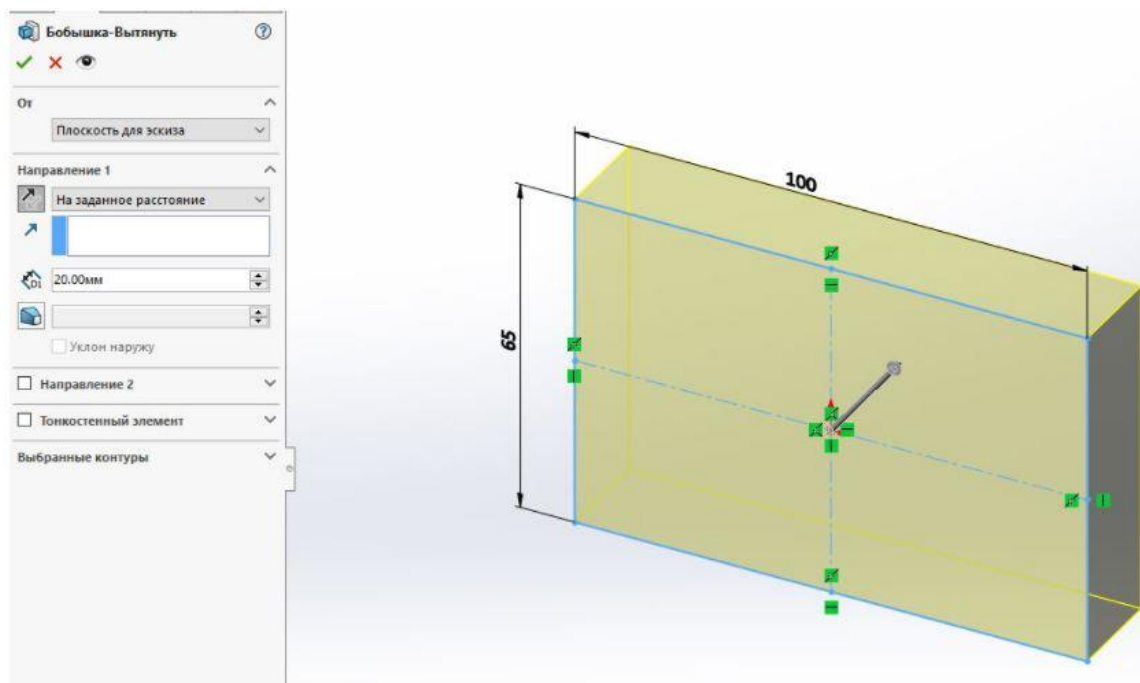


Figure 8.2 – Extruding a rectangle

8.2.4 After that, click on the upper plane of the radiator and create a sketch of a circle with a diameter of 2.5 mm, with indents of 3 mm on each side (figure 8.3).

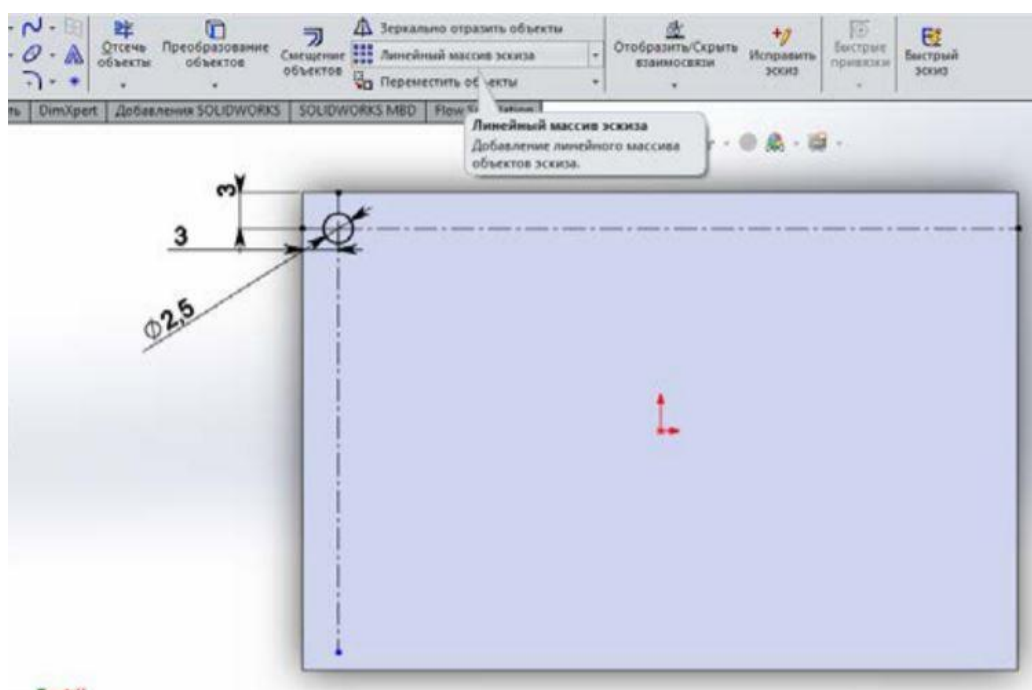



Figure 8.3 – Creating a circle on the radiator surface

8.2.5 In order to duplicate the circle, we will use the Linear Sketch Pattern tool located in the Sketch tab . Let's set the following parameters of the array: direction 1 - X-axis, distance between elements - 4.12 mm, number 14, checkmark “Show

number of copies"; direction 2 - Y-axis, distance between elements - 4.12 mm, quantity 9, checkbox on "Show number of copies". As a result, we get the following sketch, presented below.

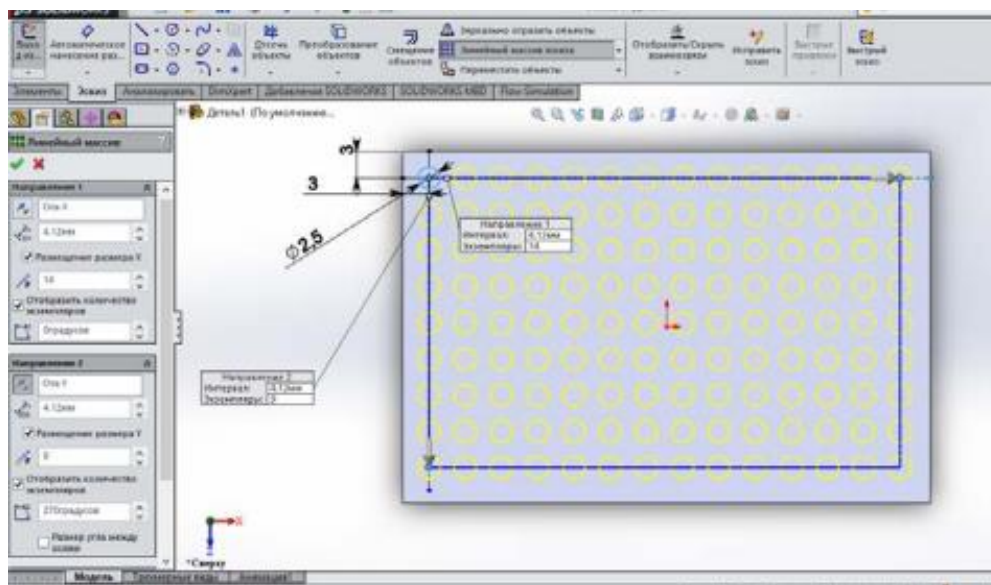


Figure 8.4 – Line array application

8.2.6 Next, select the "Pull boss" command, click on the array, pull the radiator fins up 25 mm (figure 8.5).

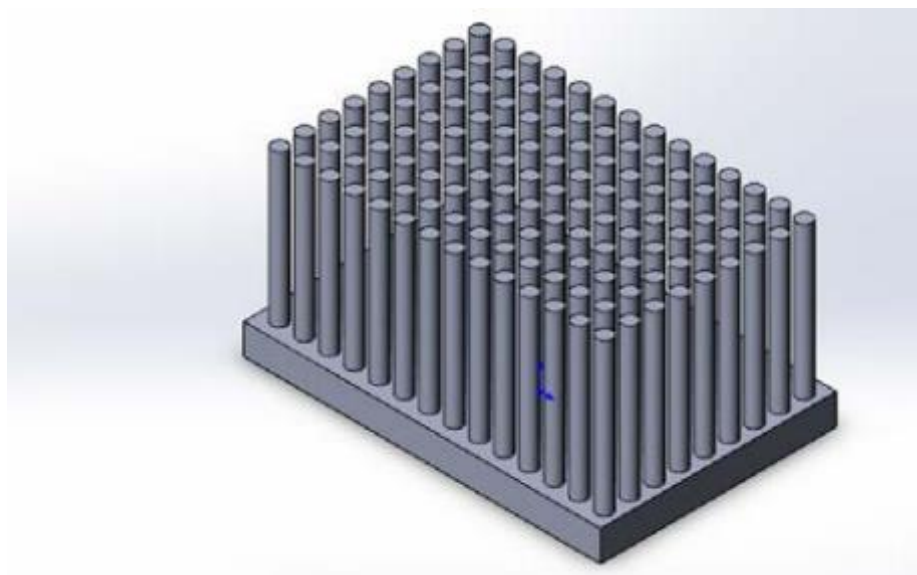


Figure 8.5 – Created radiator model

8.2.7 To simulate the heat transfer process, click on the "SolidWorks Additions" tab, select the "SolidWorks Flow Simulation" tool. We start the new project wizard. Then a window of the project wizard will open, in which we enter the name of the project as "Needle copper" (figure 8.6).

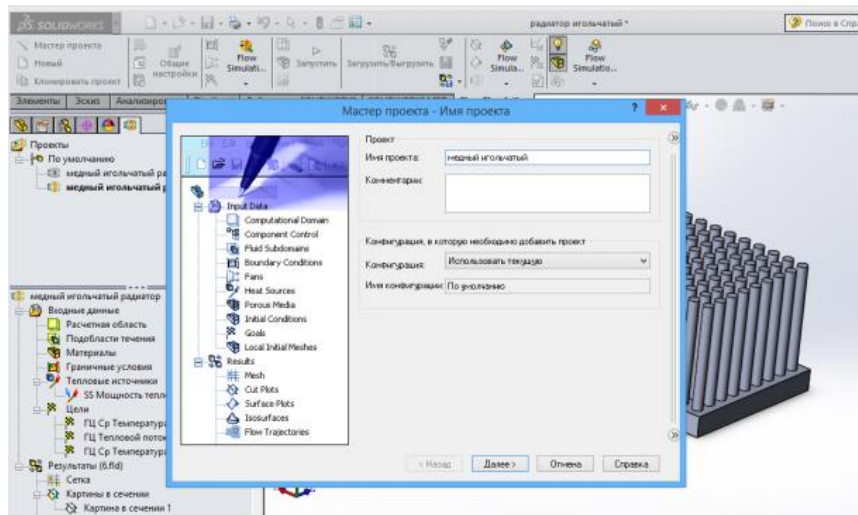


Figure 8.6 - Tool "SolidWorks Flow Simulation"

8.2.8 It is necessary to set the system of units of measurement. In the "Temperature" parameter, select "Celsius" from the list of suggested units. Set the problem type as "External", put a check mark next to "Thermal conductivity in solids" and "Gravity". Select "Air" for the type of fluid. The flow type is laminar and turbulent. Heatsink material - "Copper" (copper). Then set the initial external conditions as shown in the figure below.

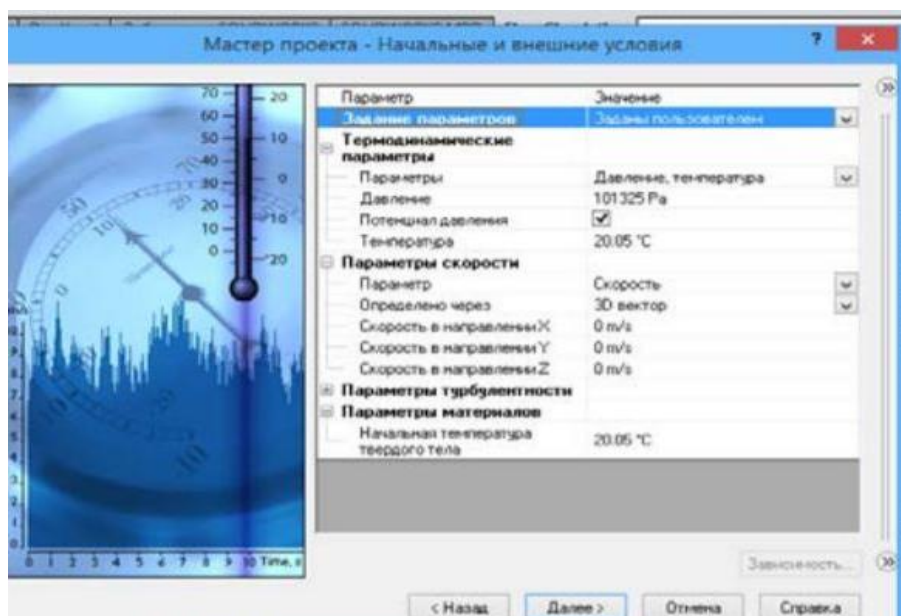


Figure 8.7 – External conditions

8.2.9 Set the resolution level to 5. Check the boxes next to "Improve resolution of narrow channels" and "Optimize resolution of thin walls". By clicking on the arrow in the Ribbon under "Flow Simulation", select "Surface heat source" from the proposed list. Then the program will ask you to indicate the edge on which the heat

source acts and the power of its impact. You need to select the base of the radiator, power - 80 W (figure 8.8).

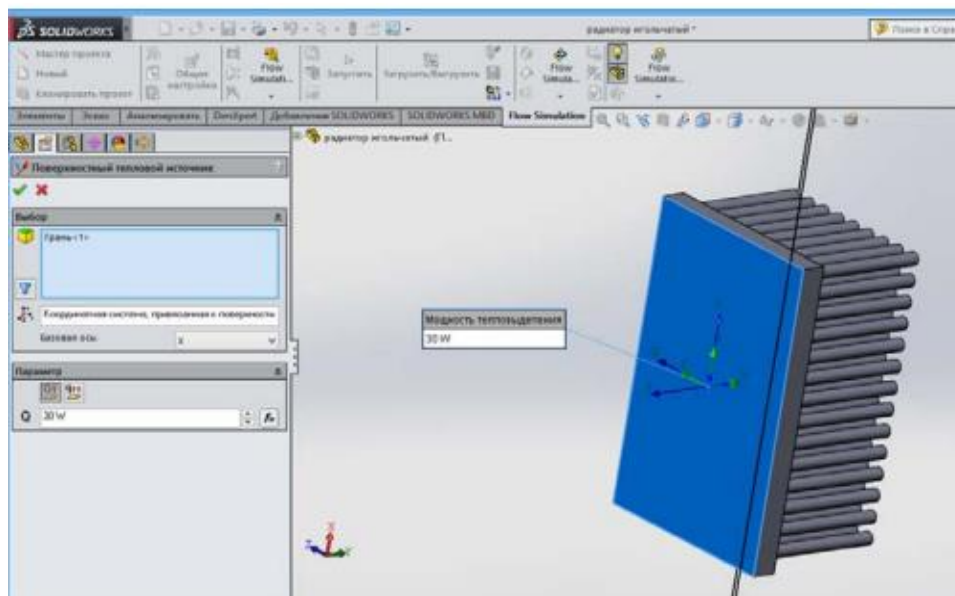


Figure 8.8 - "Surface heat source»

8.2.10 We start the simulation by clicking on the "Run" command in the "Ribbon" of the program. The result of the heat transfer process is shown below (figure 8.9).

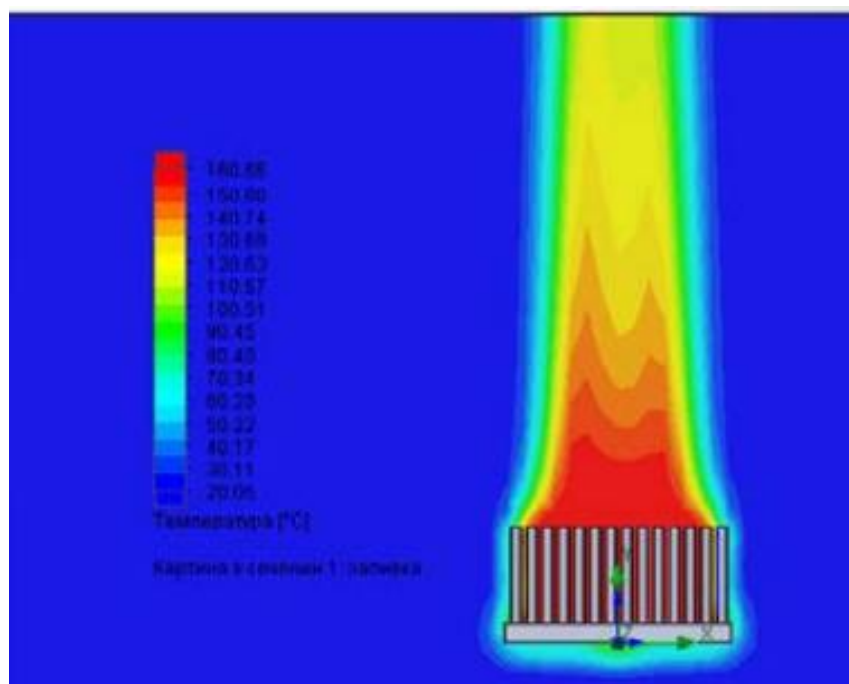


Figure 8.9 – Heat transfer simulation result

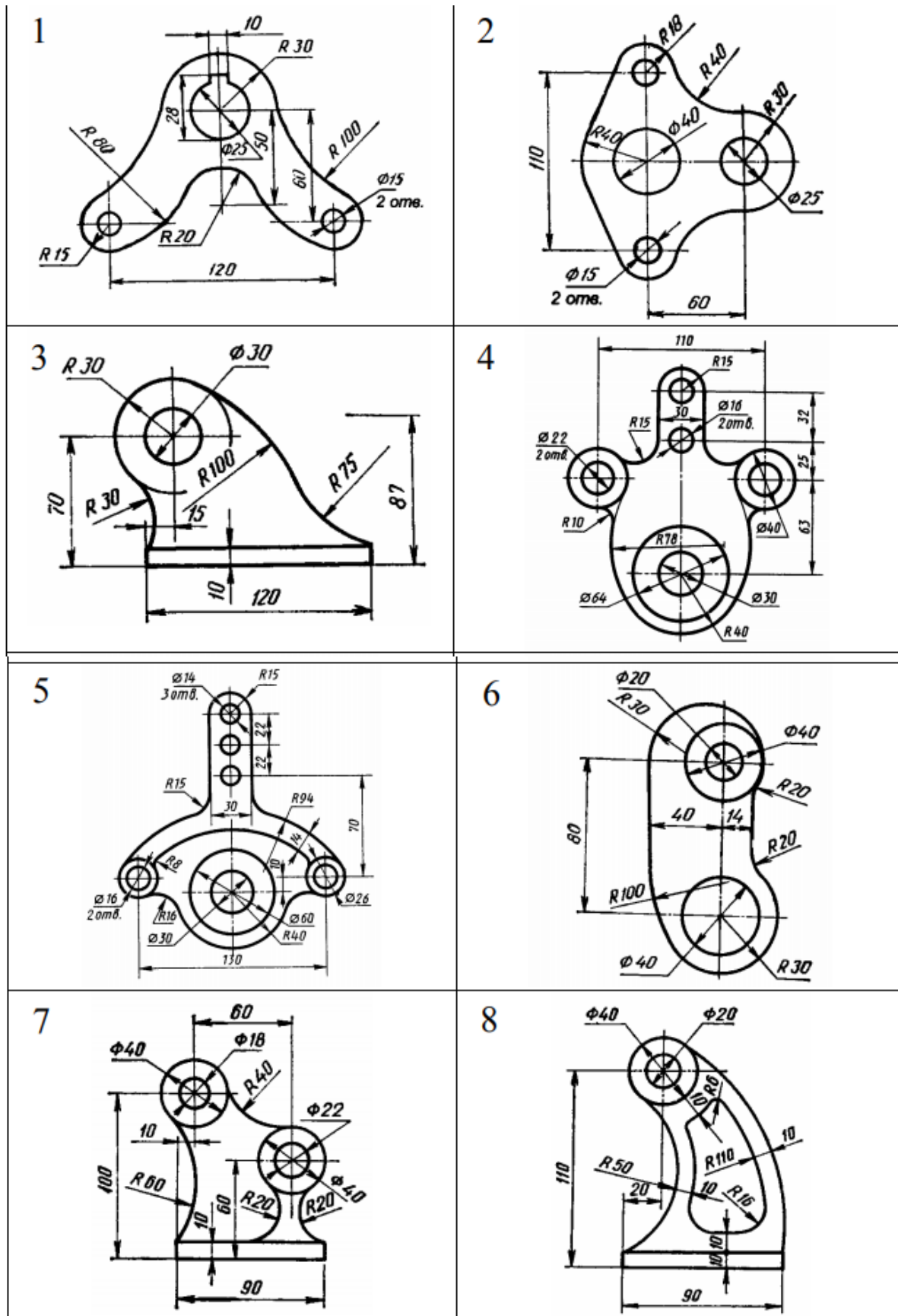
8.3 Content of work

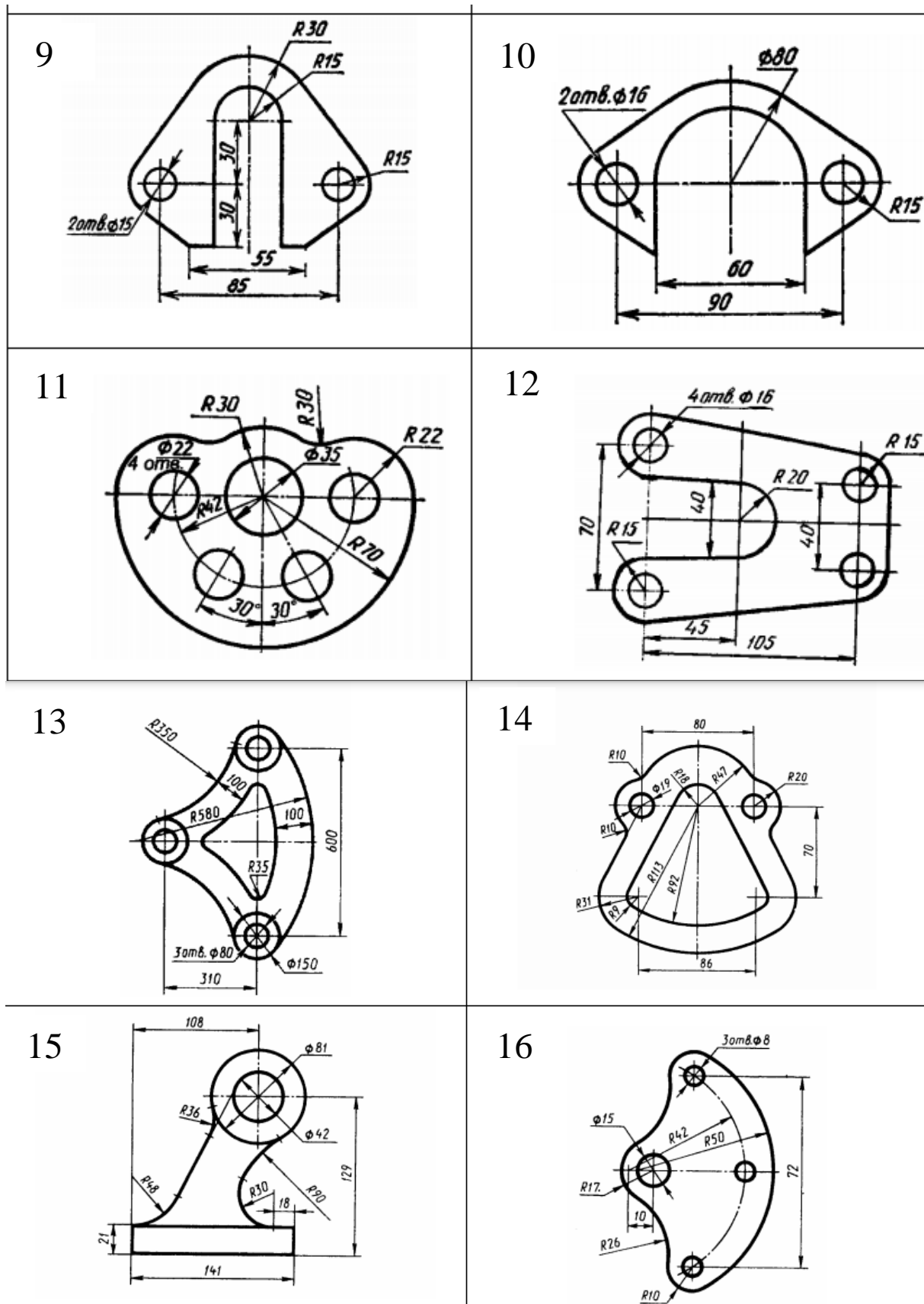
- goal of the work;
- progress report and screenshots of the radiator creation;
- to simulate the heat transfer process.

8.4 Defense questions

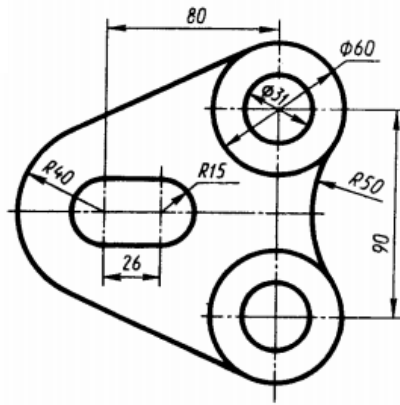
- 1) Tell us about the Flow Simulation SolidWorks package.
- 2) Show how to use the Linear Array tool.
- 3) Make conclusions on the performed heat transfer modelling.

Appendix A

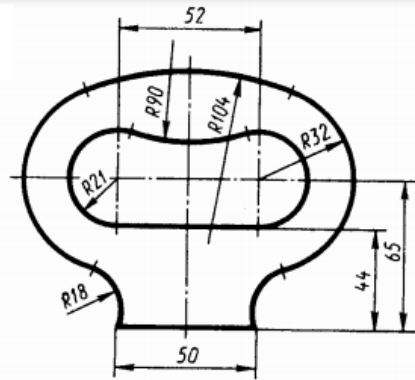




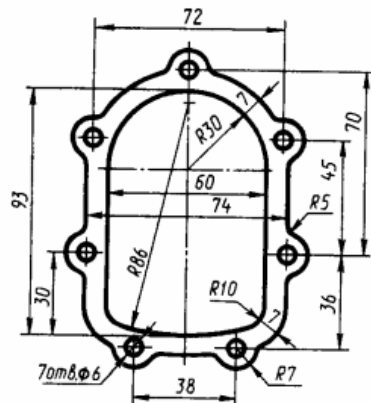
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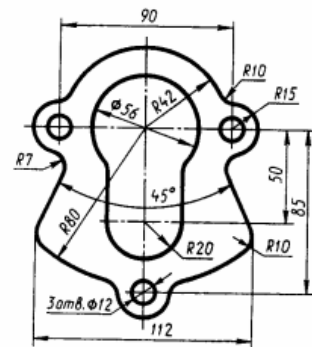
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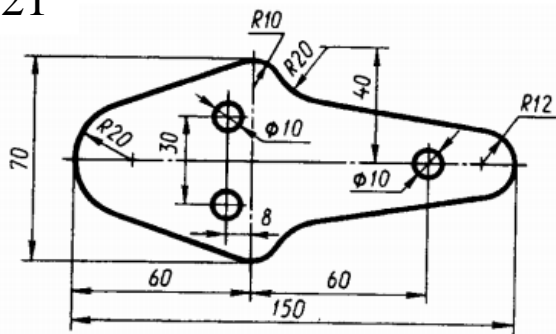
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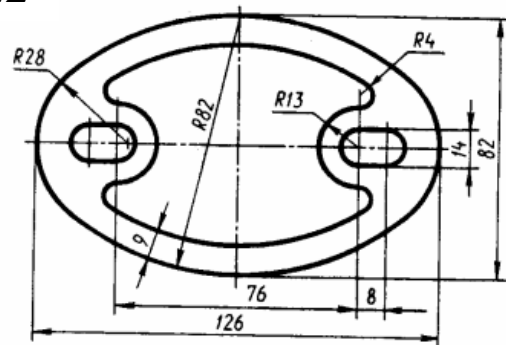
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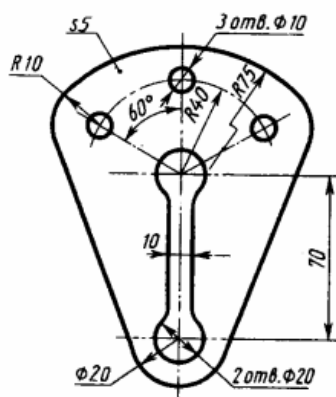
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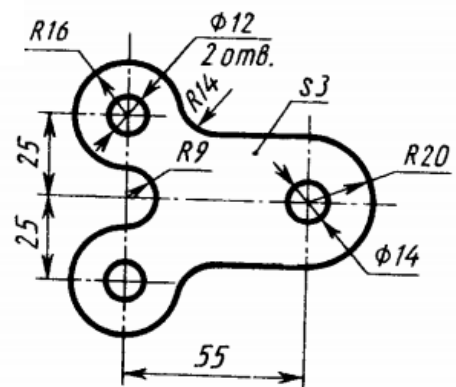
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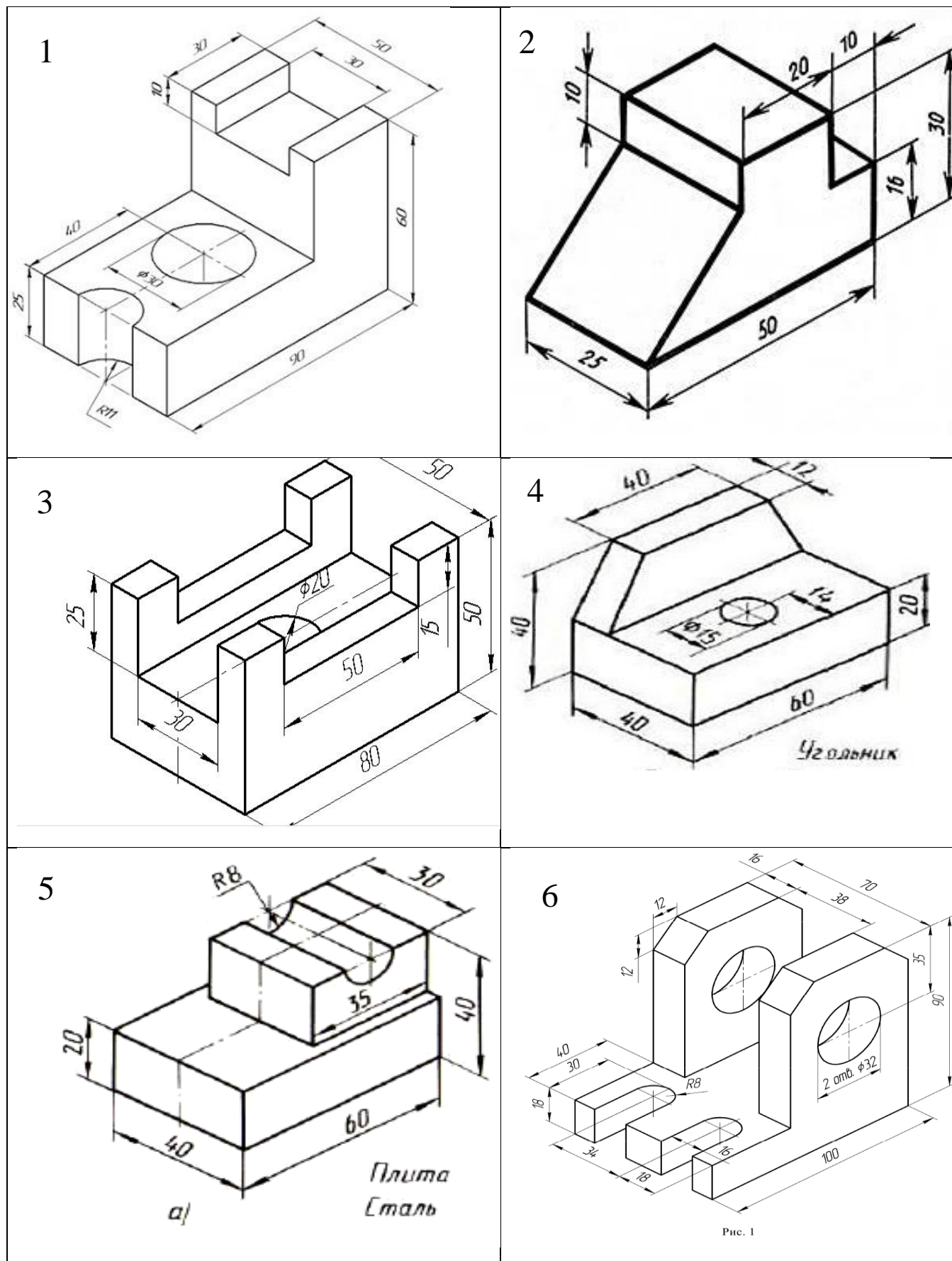
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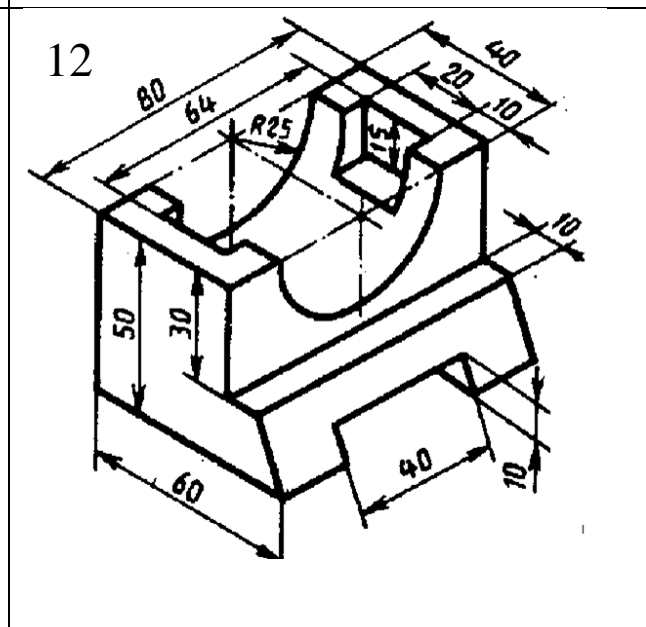
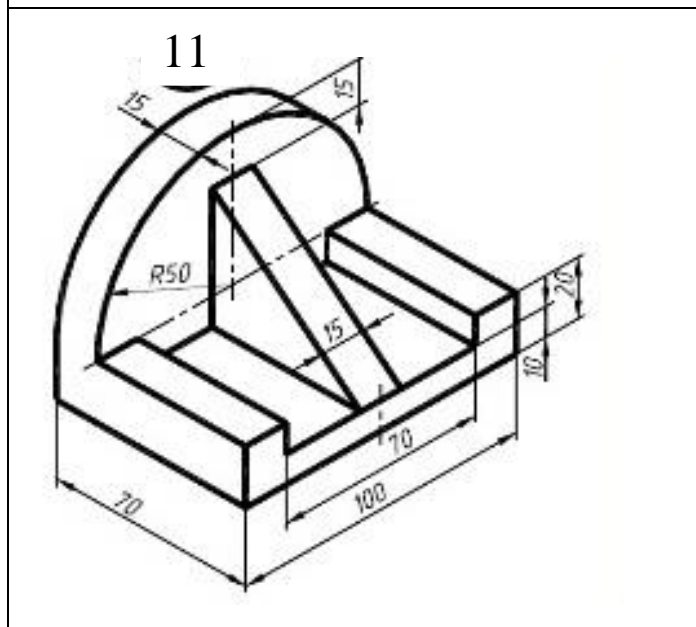
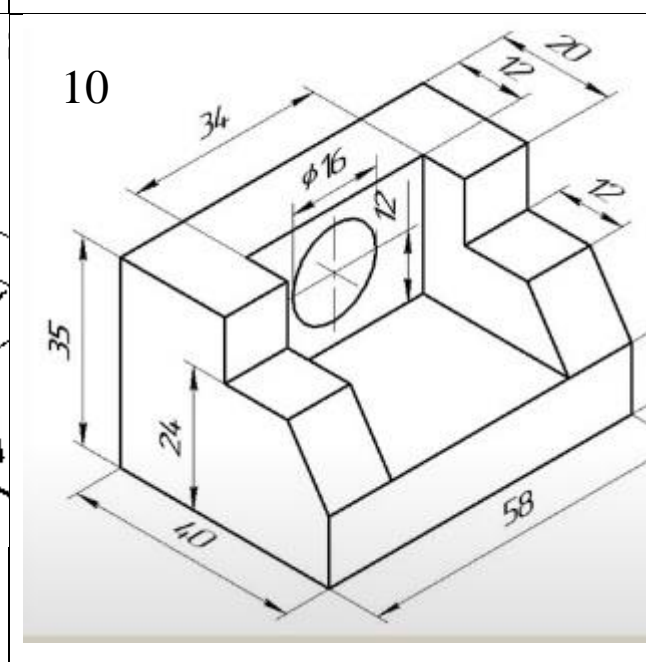
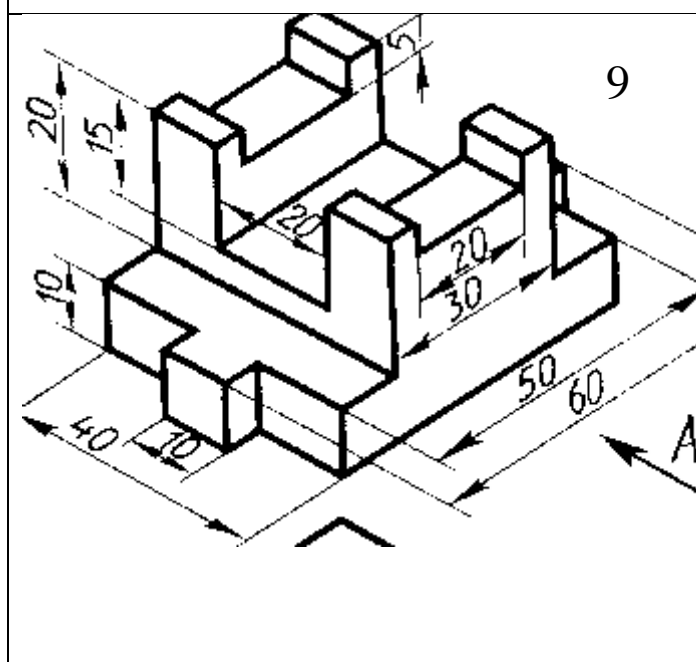
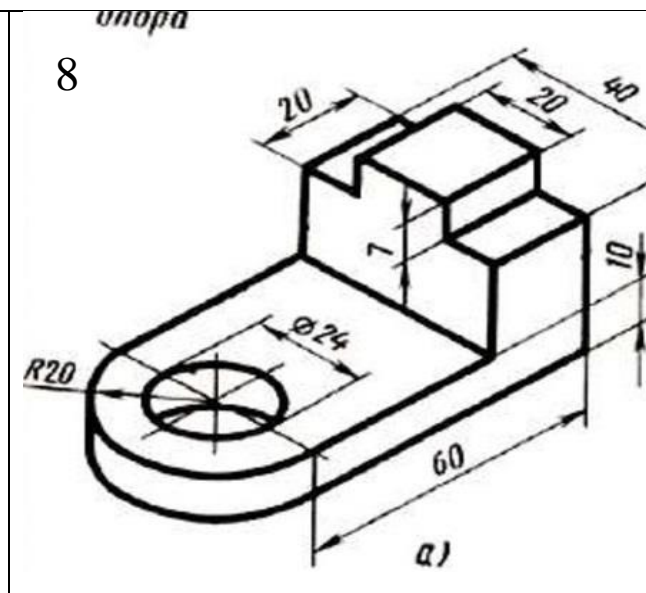
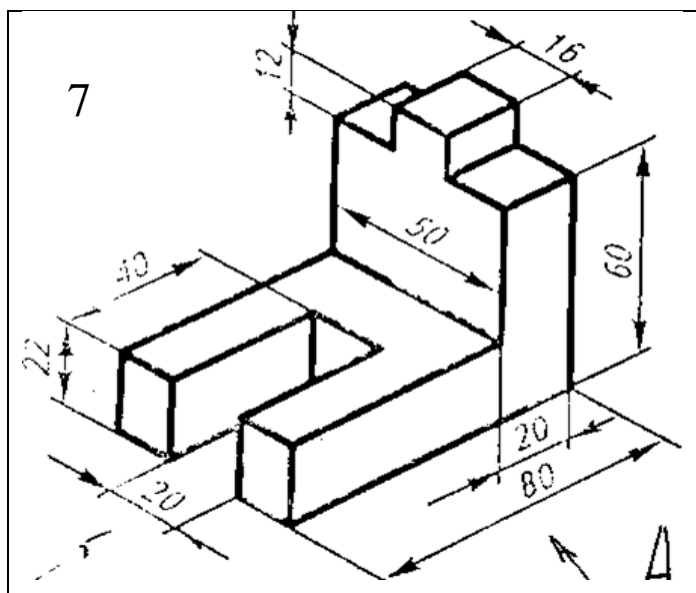


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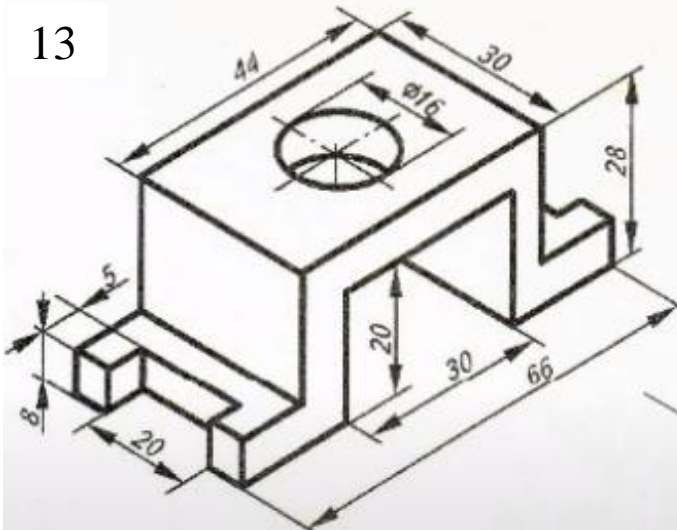


Appendix B

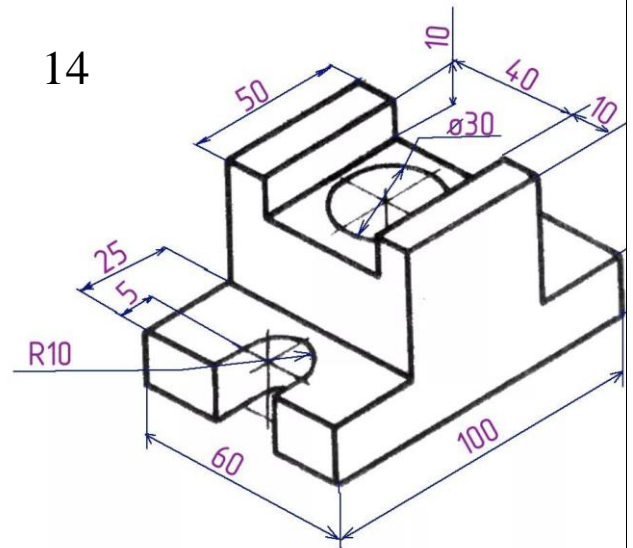




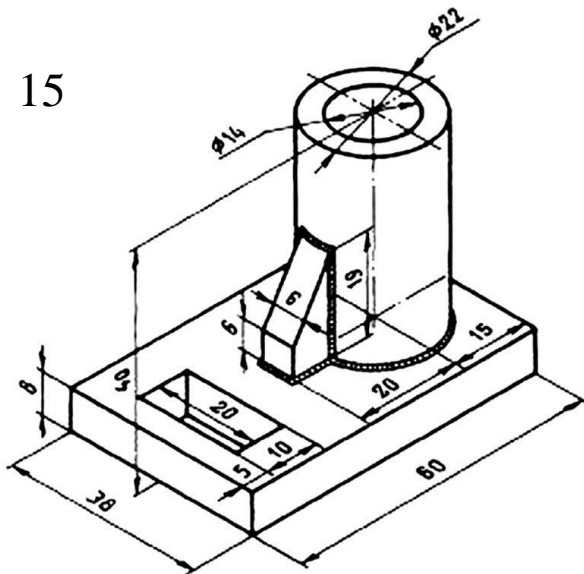
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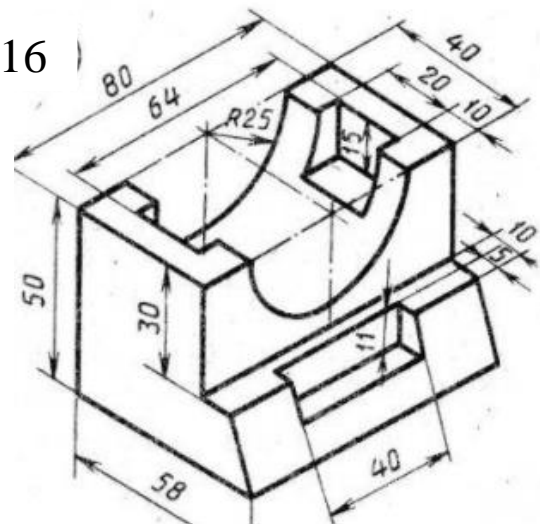
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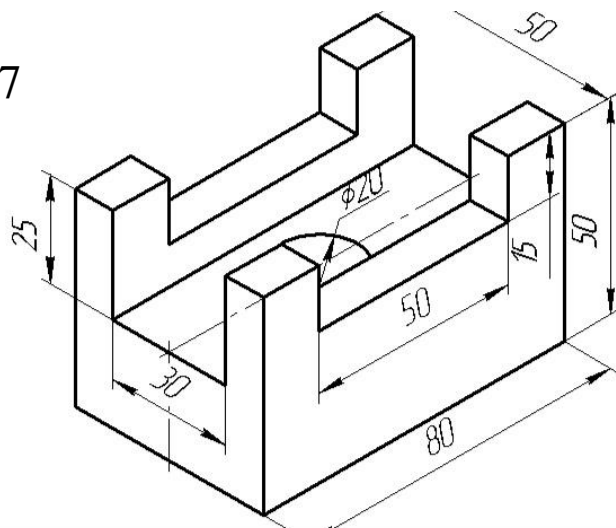
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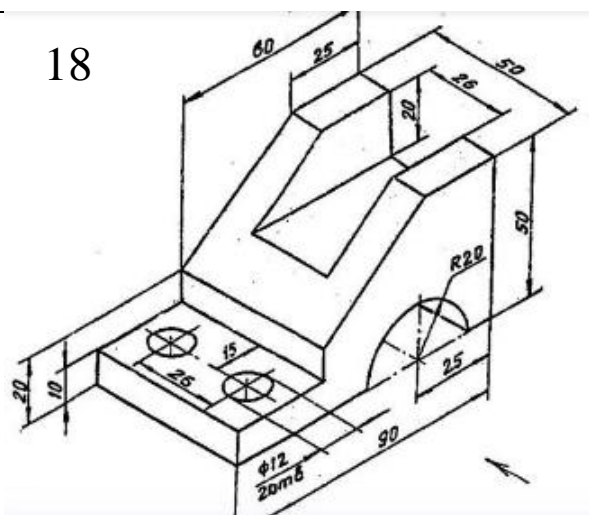
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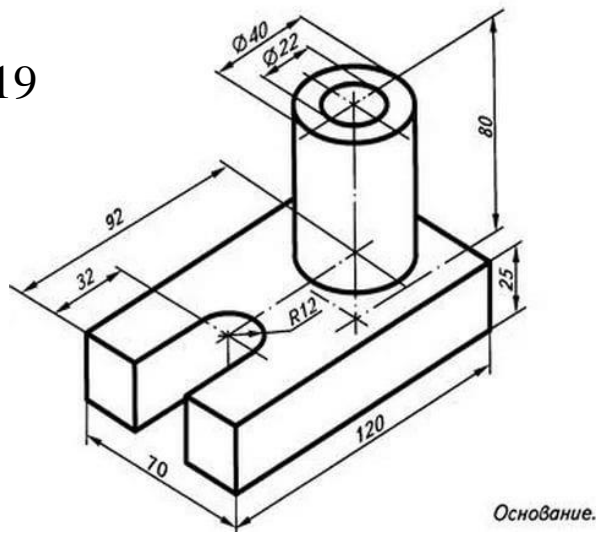
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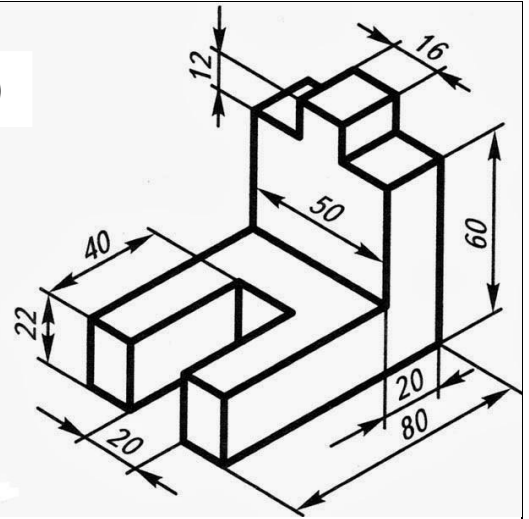
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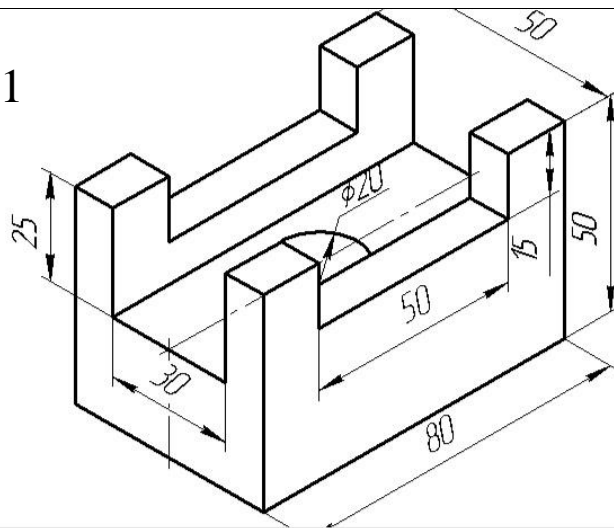
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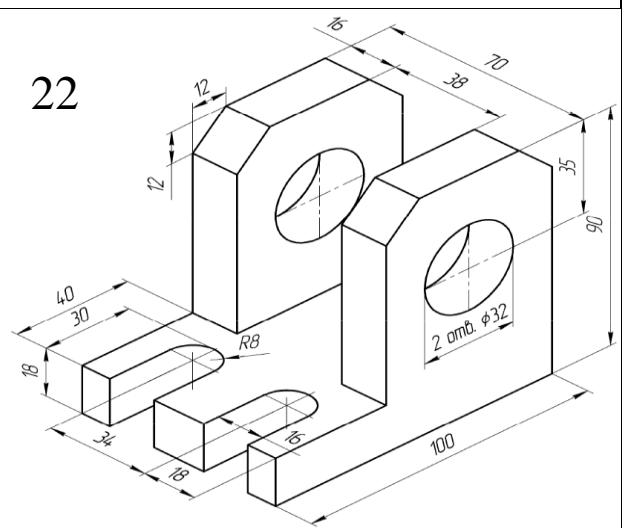
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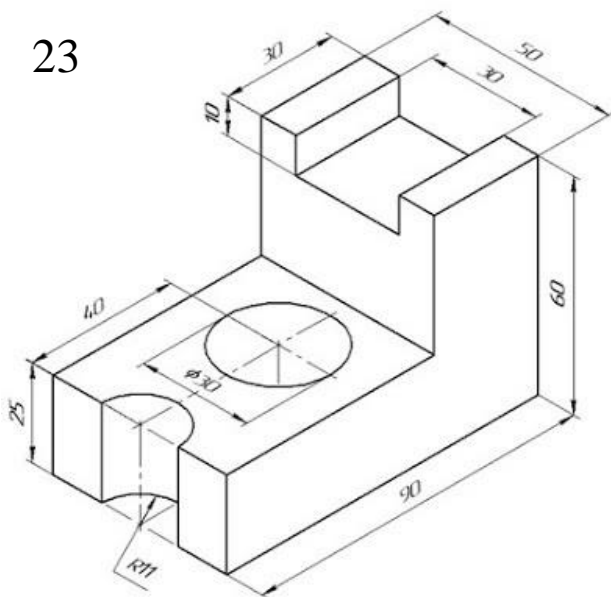
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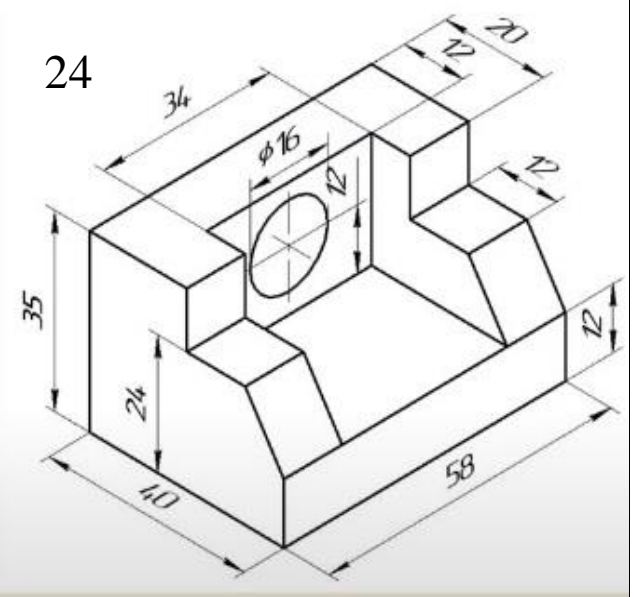
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Kalkabekova Tomiris Zhenisbekkyzy

TRAINING INTERNSHIP. DESIGN IN AUTOCAD / SOLIDWORKS

Guidelines for the implementation of practical work on educational practice
for a bachelor of all forms of training in the specialty 6B07109 – Instrument making

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