# Creating a parametric drawing in the automatic parameterization mode

## STUDY GUIDES № 3

to lab classes, independent work and distance learning
for foreign students in "CAM and CAE Systems
of Machine Building Production" subject

of First (Bachelor of Science) Level of Higher Education
Majoring in «Mechanical Engineering»

Ternopil 2021

ЛІТЕРАТУРА

ТНТУ

НАВЧАЛЬНО-МЕТОДИЧНА

**MINISTRY OF EDUCATION AND SCIENCE OF UKRAINE**

**TERNOPIL IVAN PUL'UJ NATIONAL TECHNICAL UNIVERSITY**

**Faculty of Engineering of Machines, Structures and Technologies**

Department of Mechanical
Engineering Technologies

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Study guides were developed according to the educational-professional program of applicant training of first (bachelor of science) level of higher education majoring in "Mechanical Engineering".

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Lab work objective: Mastering the basic rules and methods of parametric drawings of geometric objects in the automatic parameterization mode and acquisition of practical skills for their implementation.

# 1. GOAL

One of the main novelties of the latest versions of T-Flex CAD is the appearance of the automatic parameterization mode. The appearance of this mode greatly simplifies the creation of parametric models in many commands and, above all, in the sketch. The fact is that, while providing powerful parameterization tools, developers do not necessarily impose on the user the use of mechanisms for creating associative models - construction lines and variables. To create non-parametric drawings, the user can use the "sketch" command, which creates "free" graphic lines. Users appreciated the convenience of the sketch commands, but, often making subsequent changes to the drawing, they regretted the lack of parameterization. In the tenth version, the developers went to meet them and created a mode in which, when building ordinary sketch objects, the system itself creates a parametric model.

What innovations await sketch users when the automatic parameterization mode is enabled. First, all graphic lines will be drawn on their corresponding construction lines. That is, when creating a parallel segment, a construction line will be created parallel to the construction line corresponding to the initial segment, and an image line will already be built on it. Thus, when editing the initial segment, the construction lines associated with it will change, that is, the constructed model will be fully associative. Moreover, in different sketch commands, the number of additional constructions produced by the system will

be different.

The system will take over all the routine work, allowing the user to focus on the design. For example, when constructing a rectangle, the system will create a horizontal and vertical line, as well as construction lines parallel to them and graphic lines on them. And the number of construction lines created in the case of creating an equidistant line is not limited at all. Thus, when the "automatic parameterization" mode is turned on, the user has the opportunity to create dozens of fully parametric objects with two clicks.

For the successful application of automatic parameterization, it is not necessary that the original drawing be completely parametric - the system itself will create all the necessary construction lines. So, for example, when creating a tangent line to a non-parametric circle, the system itself converts the circle into a parametric one (it will build a construction line a circle with the same center and radius and transfer the image line to it). And after that, a straight line tangent to the construction line "circle" and an image line on it will be created.

When working in a sketch with the automatic parameterization mode turned on, the system itself creates a parametric model, depending on the parameters and bindings fixed in the dialog. When creating an equidistant line in the automatic parameterization mode, construction lines will be built - parallel straight lines for segments and concentric circles for arcs.

It is important to note that objects created in parametric sketch mode do not differ from other objects in the system when edited. Since when creating them, the same tools were used that are used when creating the rest of the parametric models. That is, when editing, the user gets all the opportunities to change the resulting associative links.

Dialogues have gained a new important role in the parametric sketching

mode. Now, with the help of the dialog, the user can not only set the parameters of the model, but also influence the type of construction lines being created. If the user, when creating the second point of the line, entered, for example, an angle and a distance, then a construction line will be created with a straight line at an angle. And the second node will be built at the intersection of the first line with the line perpendicular to it, the parameter of which will be the distance from the first node. That is, the angle and distance will be the parameters of the resulting segment.

A completely different parametric model will be created if the user sets the offset dx, dy relative to the first point in the dialog. The second end of the segment will be built at the intersection of construction lines parallel to the vertical and horizontal straight lines passing through the first point and lying at a distance dx and dy from them, respectively. In general, when constructing the second point, the user can fix in the dialog the absolute coordinates X, Y, relative coordinates dx, dy, angle and distance. In this case, in each of the possible cases of a combination of two values, the corresponding construction line types will be created.

Similar changes affected the whole variety of sketch commands and dialogs. Let's give another important example - creating a parallel line. If the user fixes the distance in the dialog, the system will create a straight line parallel to the given one with the distance parameter and a node on it.

If the coordinates of the first point are specified, the system will create a node at the intersection of the vertical and horizontal construction lines, and through it will draw a straight line parallel original. Thus, to create completely different types of parametric models, the user only needs to click a couple of times in the properties dialog box or fix the corresponding parameters using hot

keys.

Another major change in the sketch dialogs is the ability to enter variables, making sketch a complete parametric modeling tool. While the sketch was used to create non-parametric models, only numeric constants could be entered in the dialogs, but with the advent of the "automatic parameterization" option, it became possible to enter any line in the sketch dialogs. As in other T-FLEX dialogs, if a variable with this name exists in the system, it will be used as a construction line parameter. If such a variable does not exist, you will be prompted to create this variable after the construction is complete.

## 2. THEORETICAL INFORMATION

To explain the sequence of creating the drawing in the mode of automatic parameterization, we use as an example a similar drawing as in previous lab works. The stages of constructions will be the same as described there in the theoretical information.

When working in the automatic parameterization mode, we will be creating only graphic lines (as when constructing a sketch). Meanwhile, the system will be automatically "slipping" nodes and construction lines with parametric relations underneath those graphic lines. What constructions to create and what dependencies to use in relations is determined by the system based on the user-selected snaps and parameters defined in the command's property window when creating sketched lines.

Call the command **"SK: Create Sketch"**. Make sure that the following snaps are enabled:

| | |
|---|---|
|  | Construction |

| | |
|---|---|
|  | Line Midpoint |
|  | Horizontal/Vertical |
|  | Orthogonal |
|  | Graphic Line Intersection |
|  | Horizontal/Vertical tangent |

To create a parametric drawing, the automatic parameterization mode must be turned On in the system. This mode is enabled with the icon on the "View" bar:

| | |
|---|---|
|  | Automatic parameterization On/Off |

We will start with creating the main view of the plate. If necessary, enable the line segment creation option  in the command's automenu (if desired). Create the first point of the segment, corresponding to the lower-right corner of the plate's main view. Please note that it was not a free node that was created at the location of the click, rather, there are two crossed lines (vertical and horizontal), and a node at their intersection.

For the second node of the segment, define the Y-axis offset (100) in the properties window. The cursor will start moving along a horizontal auxiliary line. Move it to the vertical construction line going through the first segment node. When the snap to the latter line engages ("Line …"), click .
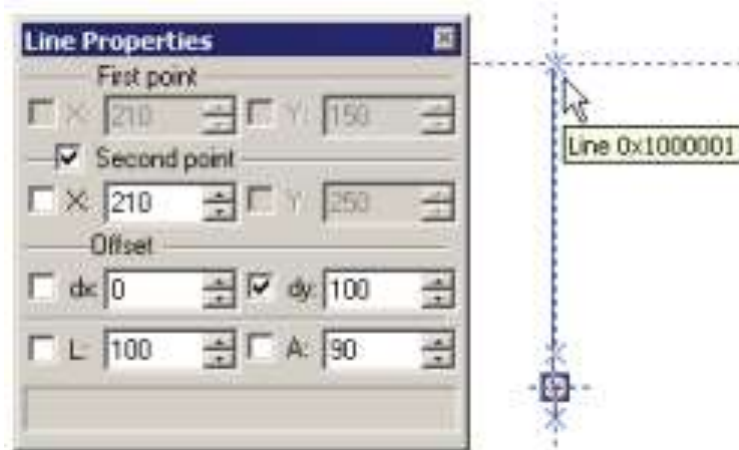
As a result, the second node of the created segment will also be constructed as snapped. It will lie on the intersection of the vertical line created at the time of constructing the first node, and a line parallel to the horizontal line of the first node.

Please note, that, when selecting a snap, the system may offer the vertical snap to the first segment node rather than snapping to the line (the order of displaying snaps is determined by the settings in the command **"SO: Set System Options"**). To select the desired snap, do the following: briefly rest the cursor at the location of the snap activation. After a brief while, the cursor will change its shape: the mark and a tooltip will appear next to it, showing the total number of object snaps found at this point. Using the mouse wheel, you can scroll through those snaps. In the ongoing construction, select the desired snap from the list of possible ones at a given point using the same method.
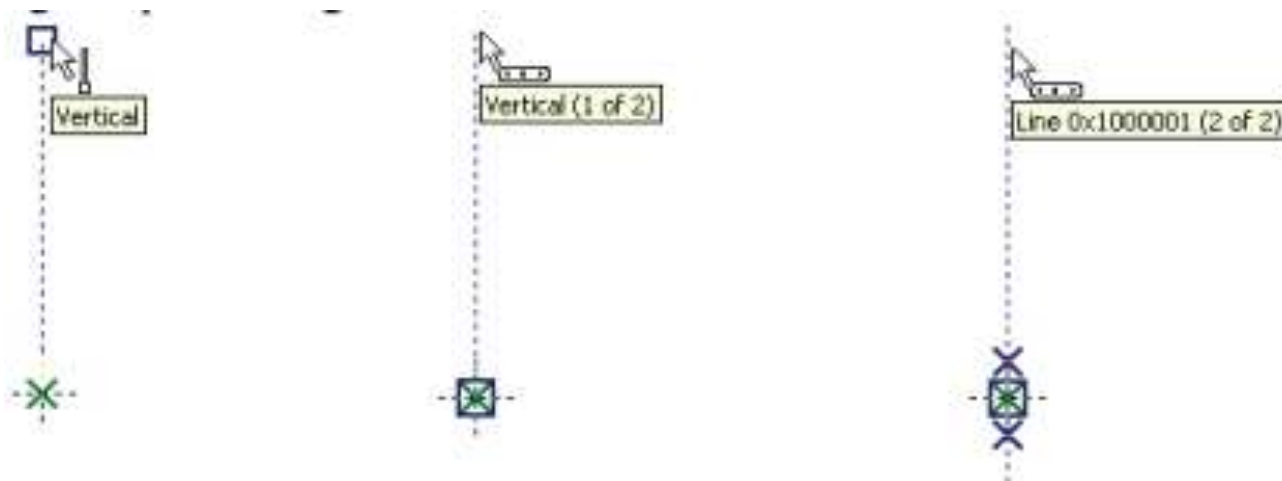
Create the second, horizontal, line segment with the length 150. When constructing it, specify the desired offset along the X-axis and use a snap to the line again.

If all was done correctly, then the created line segment will lie on the line created at the time of constructing the second node of the previous segment. Meanwhile, the second node of the current segment will be constructed as one lying on the intersection of the same line and a new line parallel to the very first vertical line.
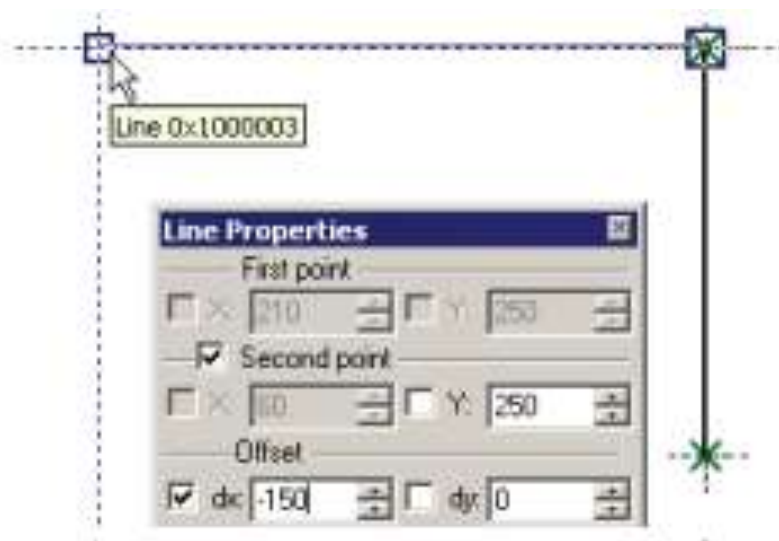


The third line segment, again vertical, is constructed by snapping to two existing lines at once.
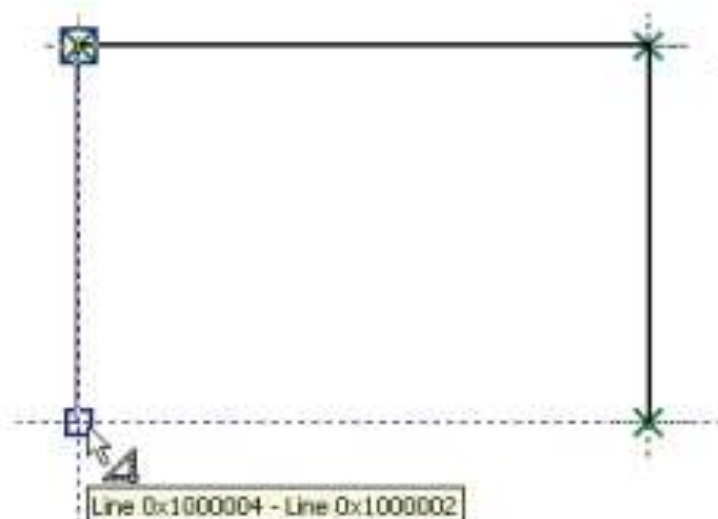
The fourth segment must be closing the described rectangle. After that, move the cursor rightward up to the first created node, as indicated by a special mark in the dynamic tooltip, and click 🖱. This completes the base for the part's main view.



Node 0x2000007
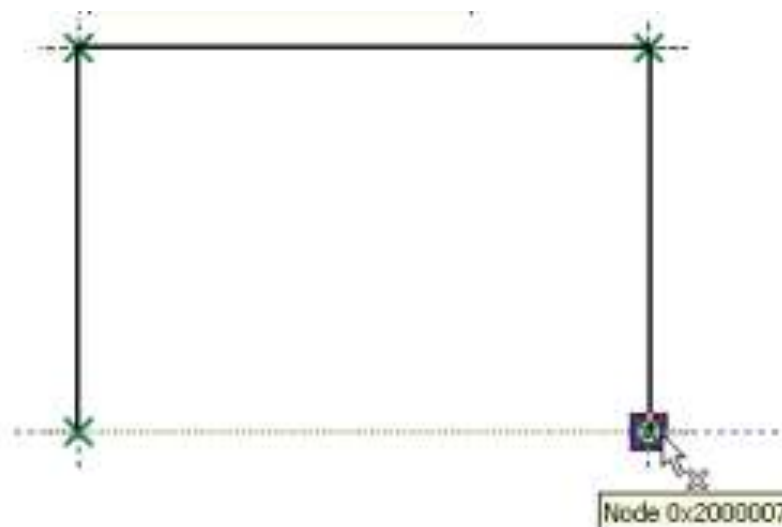
Please note, that the resulting drawing we obtained is the same as when constructing a parametric drawing by the conventional approach (as was described in the first section of this chapter). Just like when creating a sketch, to create a fillet one would need to quit the continuous line input mode (using 🖱).

To create the fillet, let's use the option ![icon]. After activating the option, set the fillet radius equal to 31 in the properties window. After that, select two segments, at whose intersection the fillet needs to be constructed (the top and the right-hand-side segments of the plate) or the node (the rectangle vertex) at their intersection.

This will result in constructing a graphic line – a circular arc with a "slipped underneath" construction line-circle. Just like when creating a common sketch, the extra pieces of the fillet segments will be automatically trimmed.
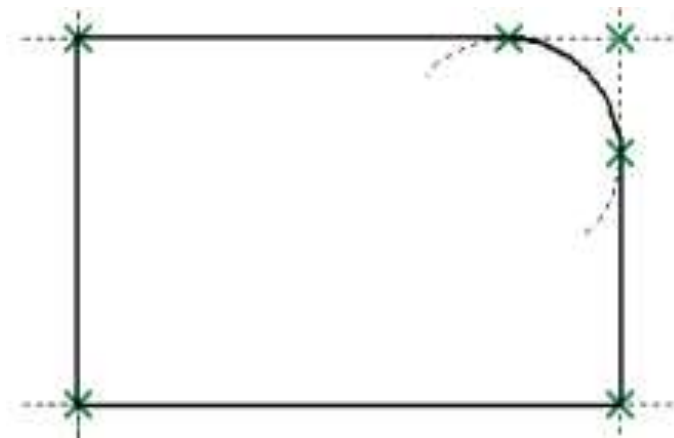


Next, we will create the image of the conical hole.

We will start with creating the axes. To create the axes, enable the ![icon] option again. Set the "Axis" line type in the system panel or in the graphic line parameters (the option ![icon]).

Move the cursor to the middle of the left-hand-side segment of the plate's main view image. Construct the first node of the axis using the line midpoint snap. Move the cursor horizontally to the right-hand-side segment of the image and stop it at the intersection of the two lines as shown on the figure. Click ![icon]. The created segment will lie on the line that divides the segment (the left-hand side of the main view) in the 0.5 ratio.
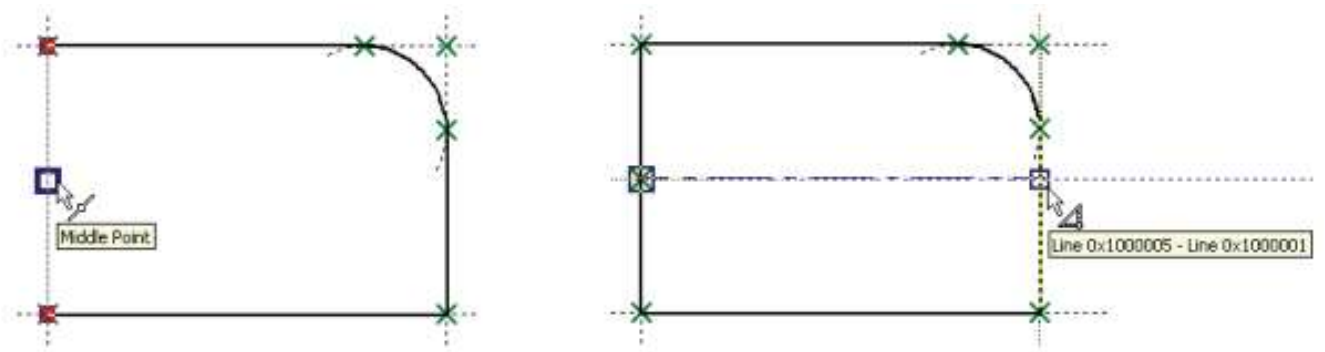
Similarly construct the vertical axis.
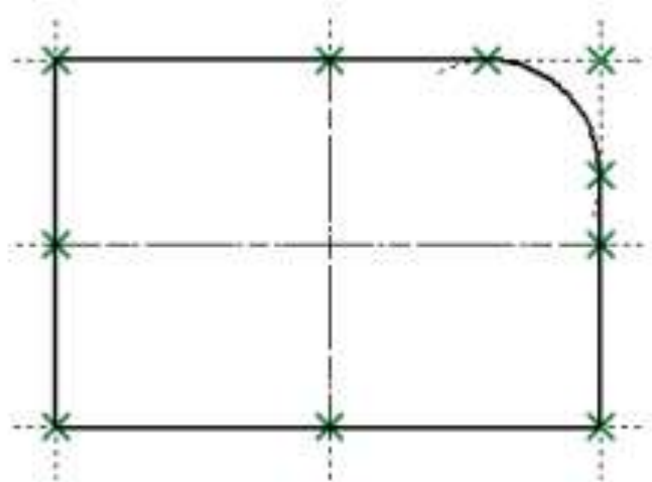


Now, let's create the circles. Set the normal graphic line type. After that, select the option . Move the cursor to the intersection of the axial lines, wait until the tooltip appears indicating the available snapping to the intersection lines. Click  right there. A rubberbanding circle will appear on the screen. Set the radius value equal to 25 for the smaller circle of the conical hole and click  or press the button **[Enter]**. Similarly construct the second circle with the radius equal to 35.
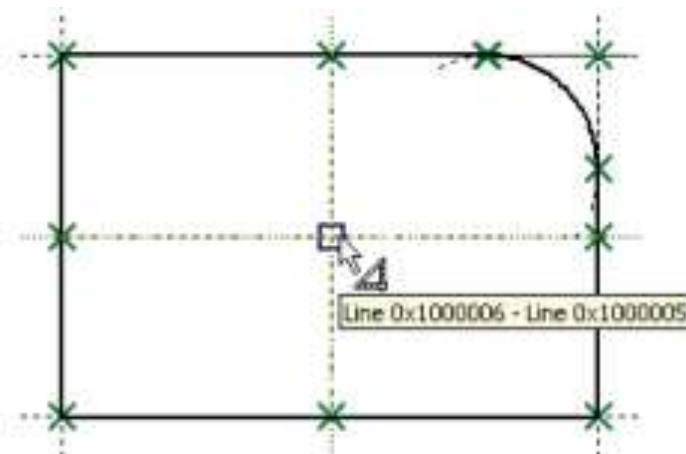
Note, that the construction result is not just free graphic lines representing the circles. The system constructed them as lying on the construction lines-circles.
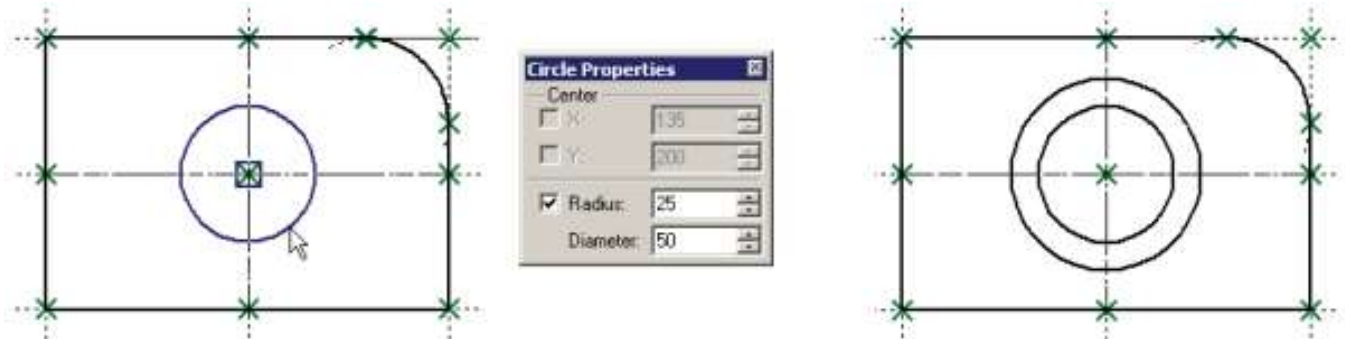


Now let's create the left view. To do that, enable the segment creation mode again (the option ![icon]). If the system offers creating a segment from the last created node, refuse that by right-clicking ![icon]. Move the cursor to the right-hand side of the drawing and set it so as to maintain the snap to the top line of the main drawing view.
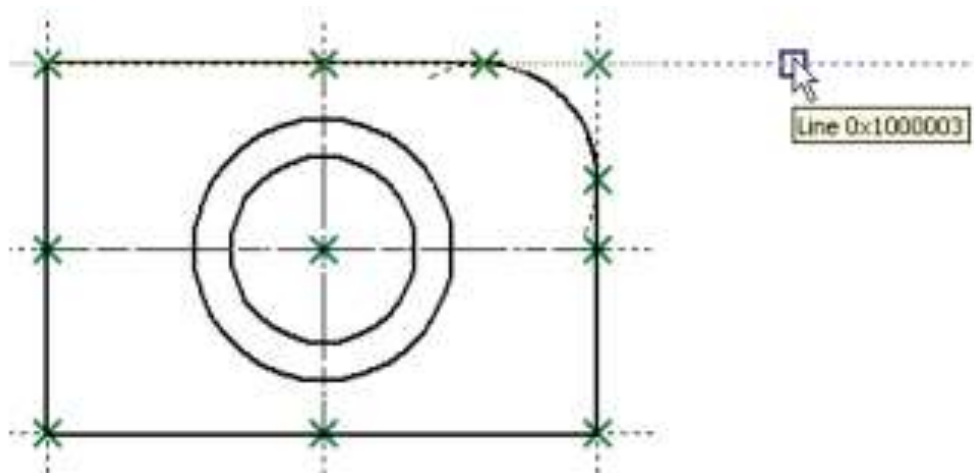


Click there ![icon]. The first node of the new segment will be constructed as lying on the selected construction line. Move the cursor horizontally rightward. In the properties window set the offset of the second point along the X-axis equal to 35. Then move the cursor so as to pick the snap to the top line of the main view. Click ![icon]. As a result, the second segment node will also lie on the top line of the main view at the distance 35 from the first node.
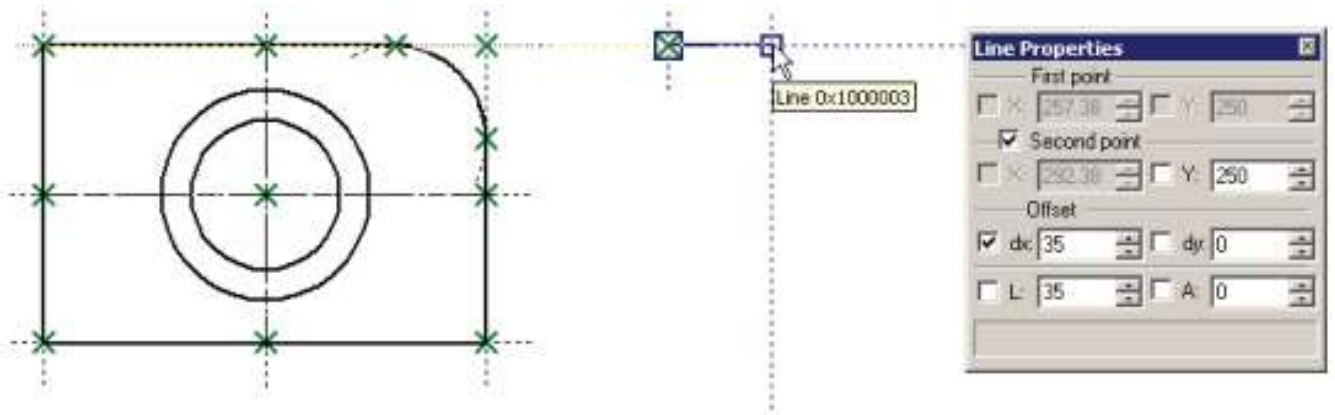
Then move the cursor downward vertically to the last created node up until a snap to two lines appears on the screen.
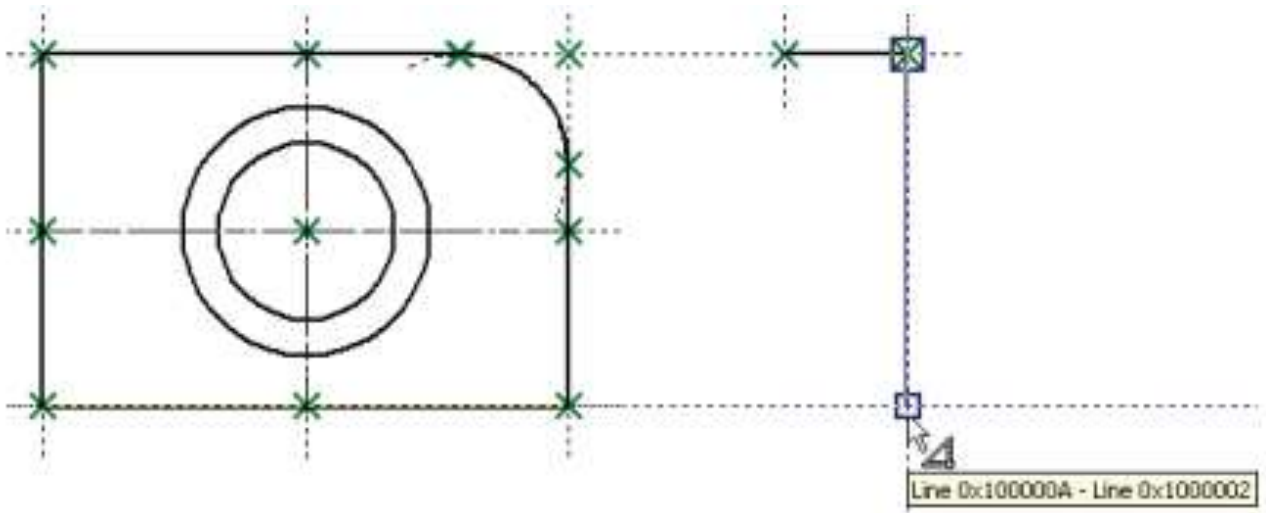


Click 🖱 and move the cursor leftwards till snapping to two other lines.
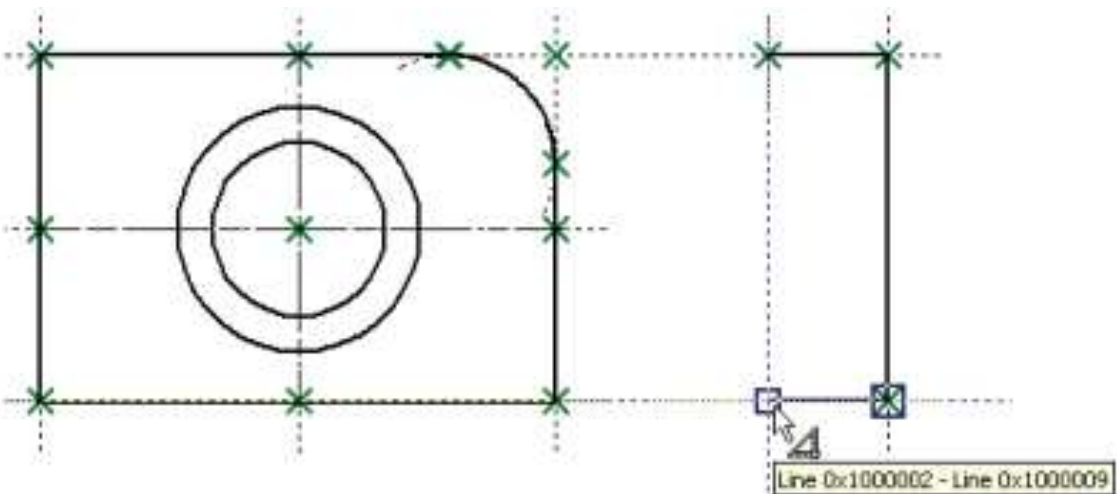
Click ⊟. Now, close the created graphic lines by moving the cursor to the first created node on this view, and click ⊟, then ⊟ (to cancel the mode of continuous line input).

After that, we will create the lines belonging to the conical hole, on the left view. Move the cursor to the right-hand-side segment of the left view, and then move it along that segment up until establishing the relation between the line underlying that segment and the greater circle. Click ⊟ right there.
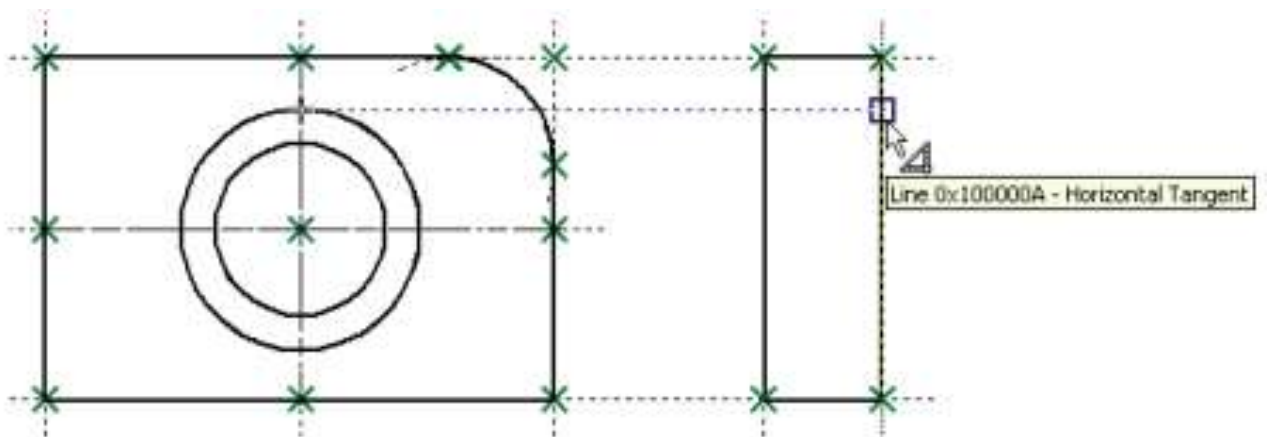


Move the cursor to the left-hand-side segment of the same view so as to establish the relation between the smaller circle and the line underlying that segment. Click ⊟.
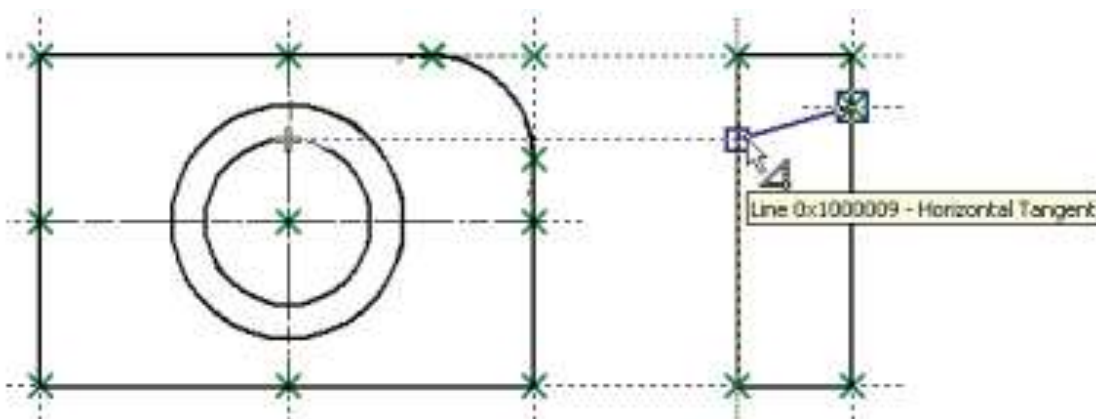


As a result, a segment will be created. No construction line will be underlying that segment. Nevertheless, each segment node will be constructed as a node at an intersection of the selected line and the line tangent to the circle.

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Next, use the same method to construct the lower line of the conical hole. Then create the centerline by snapping to the middles of the lateral sides of the left view. Do not forget to also set the dash-dot line type in the graphic line parameters (the option ![option]) or on the system panel.
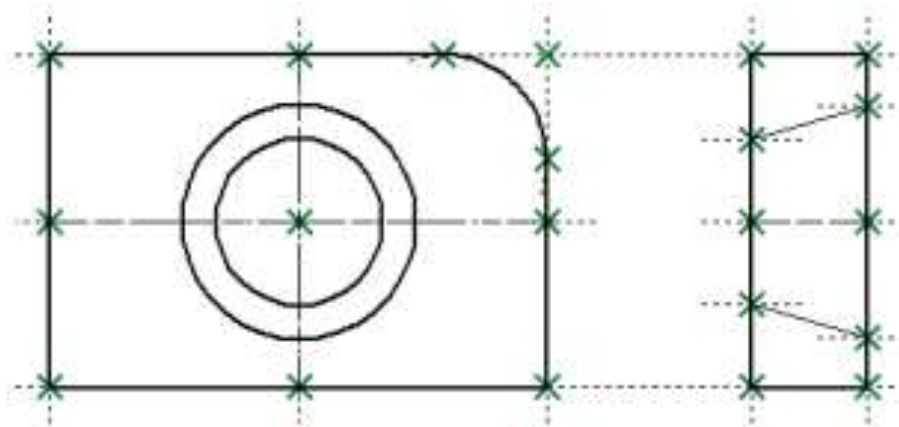


Let's proceed to creating the top (plan) view. We will create it a little bit different then when creating a simple sketch. We will not use the option of constructing a parallel segment. Now there is no practical necessity in defining that particular relation. When using the automatic parameterization mode, the use of that option will make the system try to create a construction line beneath the segment parallel to a line beneath another segment. As a result, relations would be created that we didn't need. Therefore, we will continue using the option ![option].

> Do not forget to reset the normal line type after finishing the creation of the centerline (in the graphic line parameters or on the system panel).

Move the cursor to the drawing area below the main view so as to invoke the desired relation with the line of the main view. Click ![mouse]. The first segment node will be created as lying on the intersection of the main view line and the horizontal line.
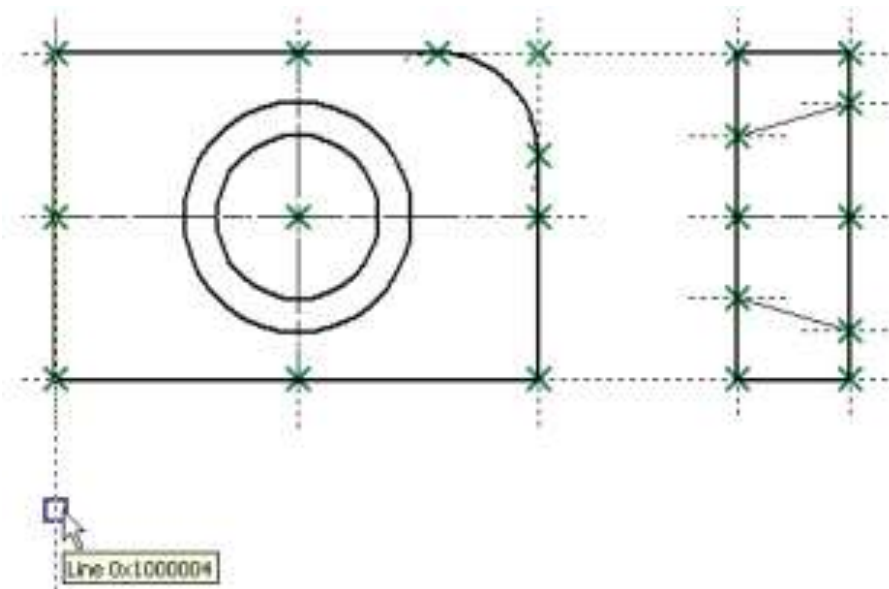
Next, move the cursor rightward up until hitting the snap to another line of the main view. Click ⊞ again. We have just created the upper segment of the top view.
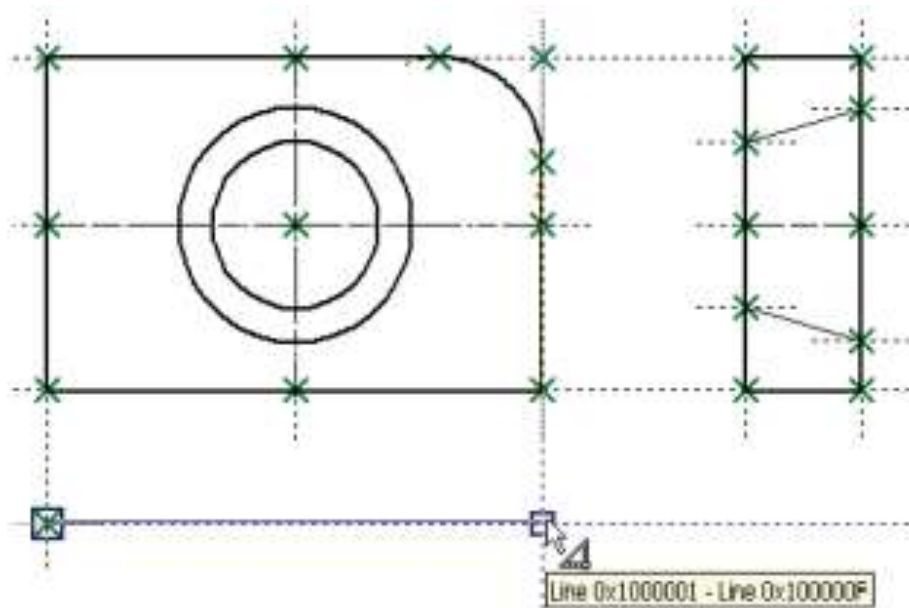


Next, we will have to temporarily quit the sketching command. The reason for that is that it is impossible to create a relation between the left view and the top view by the common means of automatic parameterization. Such relation can be achieved only by various workaround methods (for example, introducing variables as the parameters of the segments being created). But we will simply use the command **"L: Construct Line"** and create an auxiliary line at the angle

of 45° to the outer lines of the left view and top view (just as we did in the conventional creation of the parametric drawing).

So go ahead and call the command **"L: Construct Line"**. With the help of the option ![icon], construct a line – the symmetry axis between the left-hand-side vertical line of the left view and the horizontal line of the top view. Place the cursor at the intersection point of the created line and the right-hand-side vertical line of the left view, and then press the button **<Space>**.
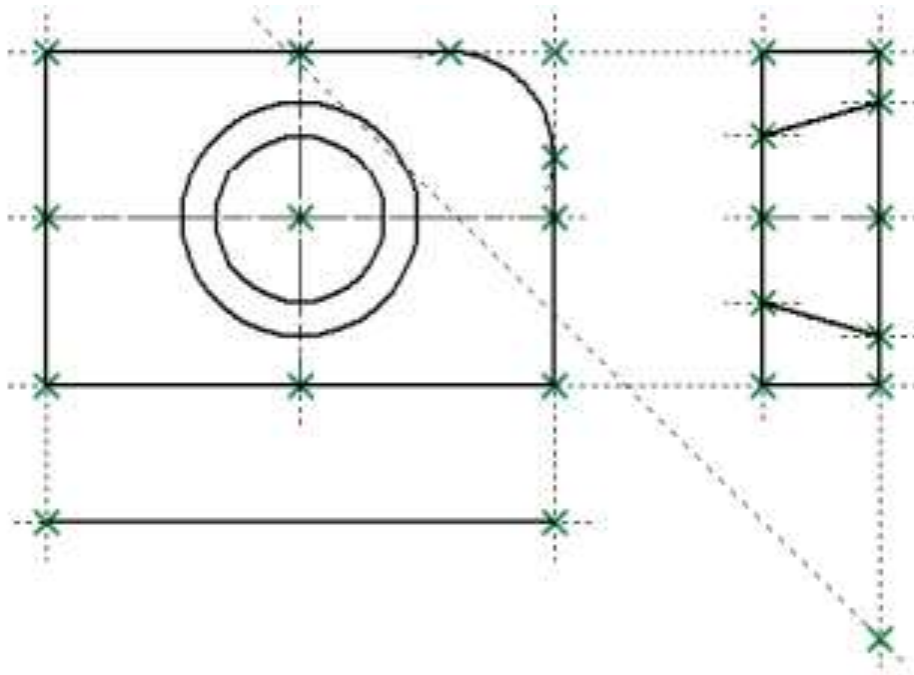


After that, call the command **"SK: Create Sketch"** again. We will create the next segment of the top view. Select the end node of the last created segment as the first node of the next segment. Then move the cursor up until reaching the intersection between the line and the horizontal through the node as shown on the figure. Fix the achieved point by clicking ![mouse icon].
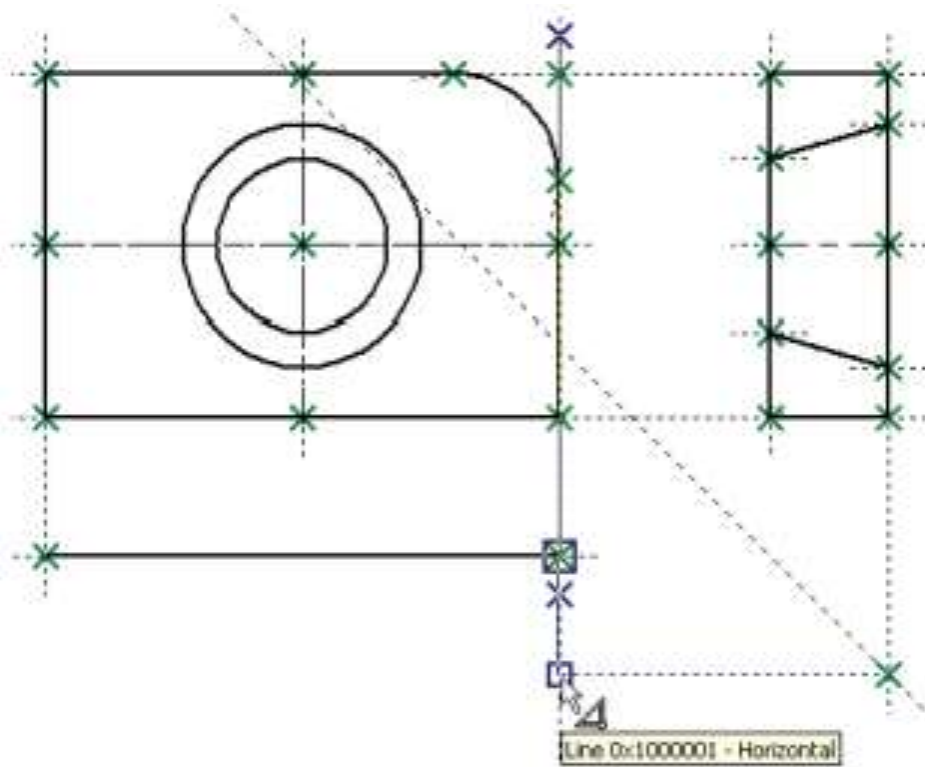
The next segment is constructed by snapping to two lines.
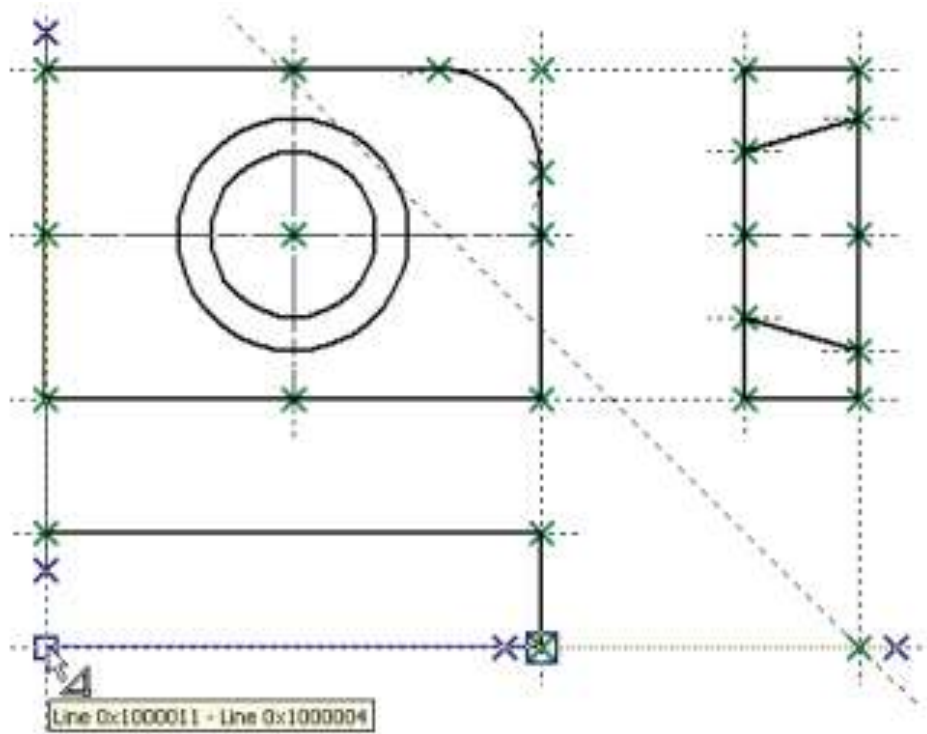


Next we need to close the created graphic lines of the top view by moving the cursor to the first created node of the view and clicking 🖱, followed by 🖱 (to cancel the continuous line input mode).

Create the centerline and the lines defining the conical hole (just like that on the left view).
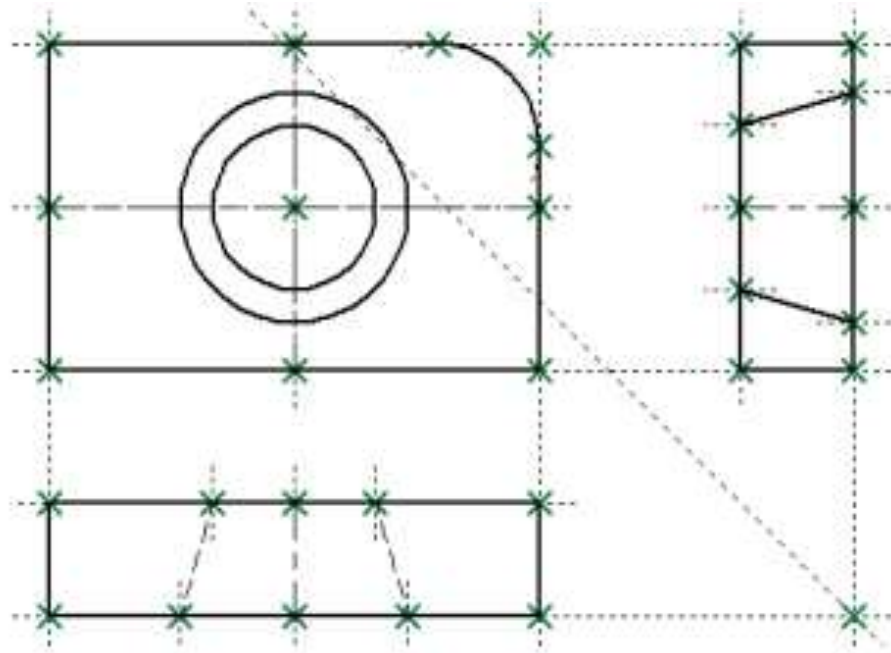


Create the hatch on the left view and the dimensions in the same way as in the previous cases.
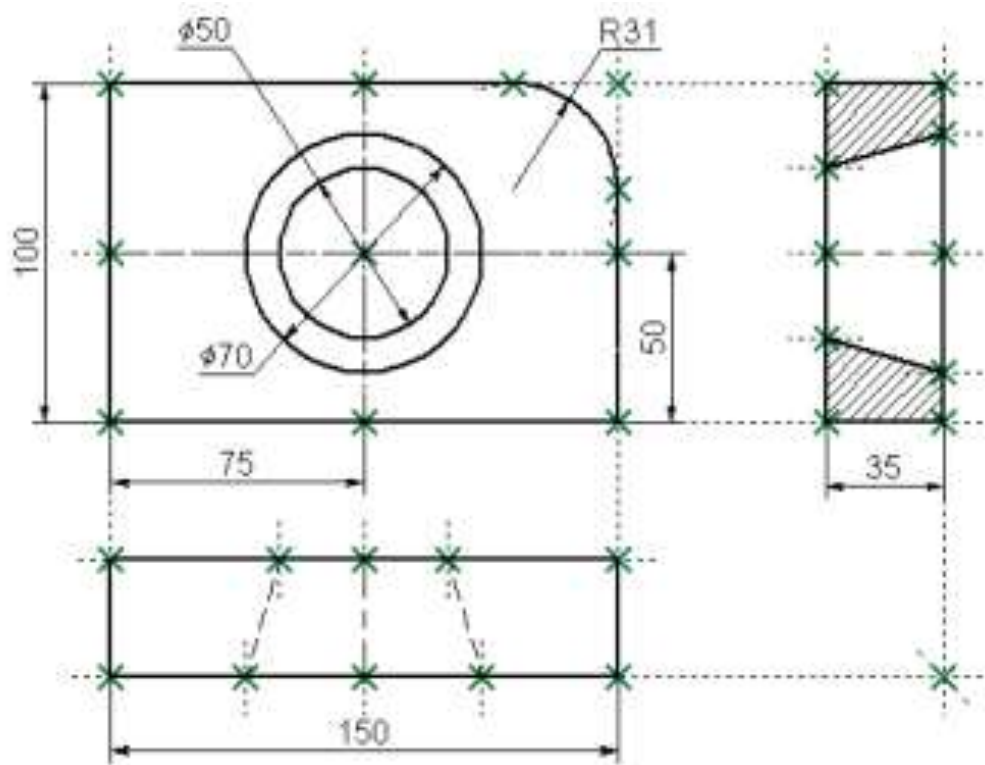
This completes the creation of a parametric drawing in the automatic parameterization mode. From now on, such drawing will behave just as a common parametric drawing.

To test, move the cursor to the segment that makes up the left border of the main view, and click ⊞. The command will be launched to edit the selected graphic line. If you click ⊞ on the line once again, the system will automatically go into the command of editing the construction line underlying this graphic line. Move the line around, define the new position by ⊞. The width of the plate main view shall automatically change. Besides that, the top view should change as well, since it was constructed by snapping to the lines of the front view.
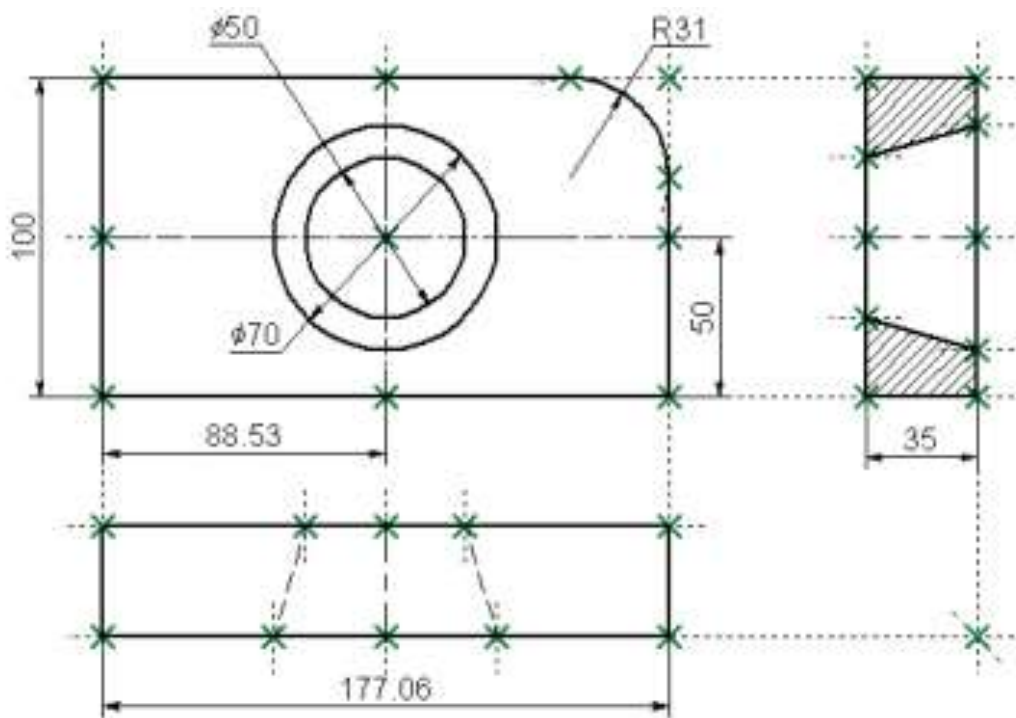


Similarly, try to edit the position of the right boundary of the main view. In this case, the entire plate drawing will move. Try the same with other drawing elements, including circles. As construction elements are moved, the shape and dimensions of the plate will be changing so as to maintain the relations defined by the construction.

## 3. THE ORDER OF LAB WORK

1. Get acquainted with the process of creating a parametric drawing of geometric objects in the automatic parameterization mode.

2. Start the automatic parameterization mode program.

3. Create a parametric drawing of the part in the automatic parameterization mode according to the individual task, where a=30 mm; b=60 mm (Table 1).

4. Conclude lab work.

5. Complete a report.

## 4. REPORT STRUCTURE

1. Theme and purpose of lab work.

2. Brief theoretical information.

3. Parametric drawing of the part in the automatic parameterization mode.

4. Conclusion.

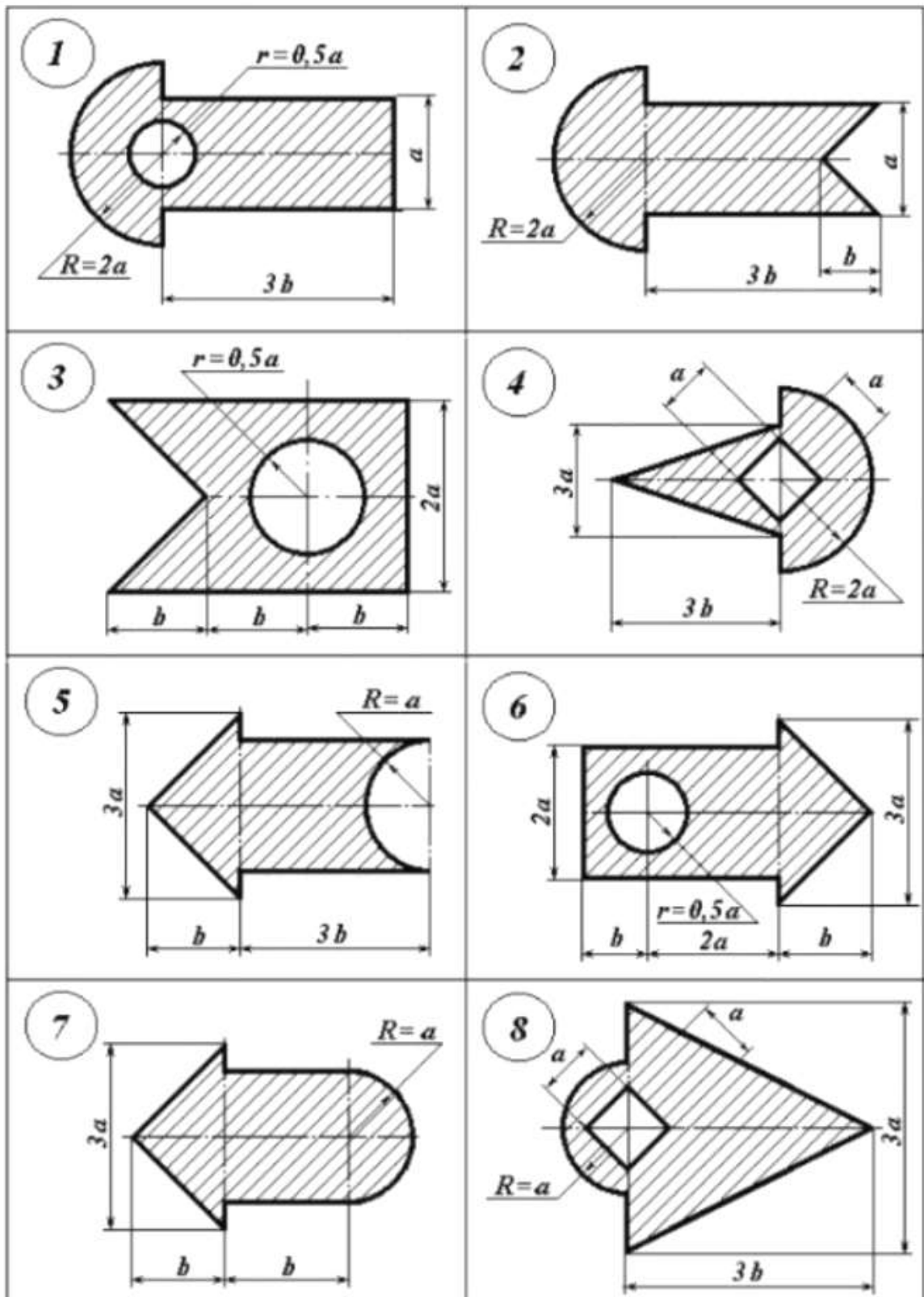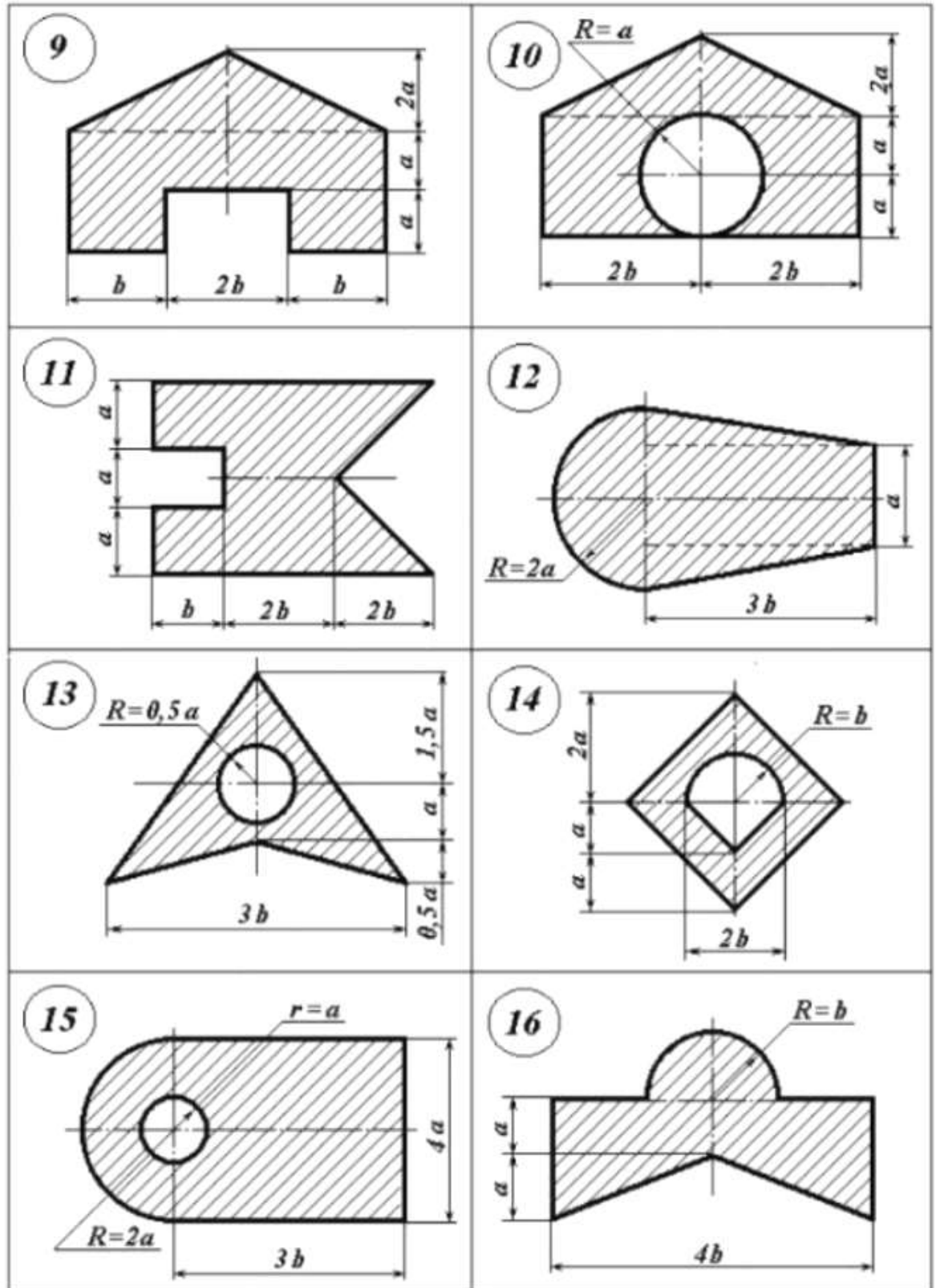5. Answer the questions from the checklist.

# 5. CHECKLIST

1. Algorithm for creating a parametric drawing in the automatic parameterization mode.

2. What are the advantages of parametric drawing in the automatic parameterization mode?

3. What are the difference between the parametric drawing and the drawing one, created in the automatic parameterization mode?

4. Preparing the sketches for drawing in the automatic parameterization mode.

5. Automatic parameterization of a previously created and prepared drawing.

6. Automatic parameterization in the process of creating a drawing.

7. Algorithm for entering text on the parametric drawings in the utomatic parameterization mode.

8. Algorithm for marking dimensions on the parametric drawing in the utomatic parameterization mode.

9. Basic commands for creating drawings in the automatic parametrization.

10. The main principles of creating drawings in the automatic parametrization.

# Task
## for lab work № 3

Create a parametric drawing of the plate in the utomatic parameterization mode.

Table 1

**9**

b | 2b | b
a
a
2a

**10** R=a

2b | 2b
2a
a
a

**11**

a
a
a
b | 2b | 2b

**12**

R=2a
3b
a

**13** R=0,5a

3b
1,5a
a
0,5a

**14**

2a
a
a
R=b
2b

**15** r=a

R=2a
3b
4a

**16** R=b

a
a
4b

**25** — 3b, 2b, 3a, R=a

**26** — R=2a, 3b, 2b

**27** — a, a, 3b, R=a

**28** — r=0,5a, 3b, R=a

**29** — 0,5a, 0,5a, R=2b, a

**30** — a, R=2b, R=b

**31** — 3b, a, 2b, R=a

**32** — 3b, b, R=a

28
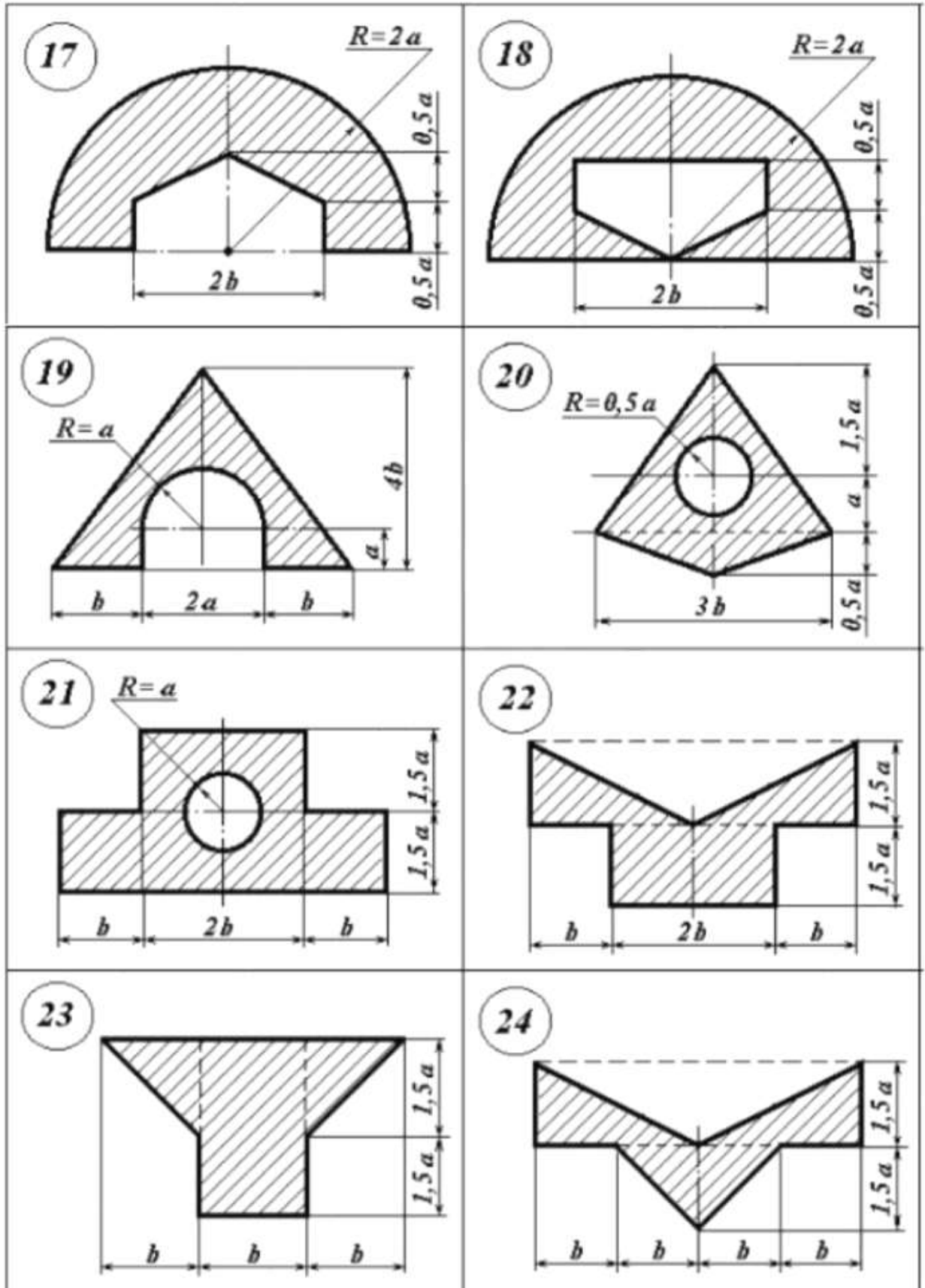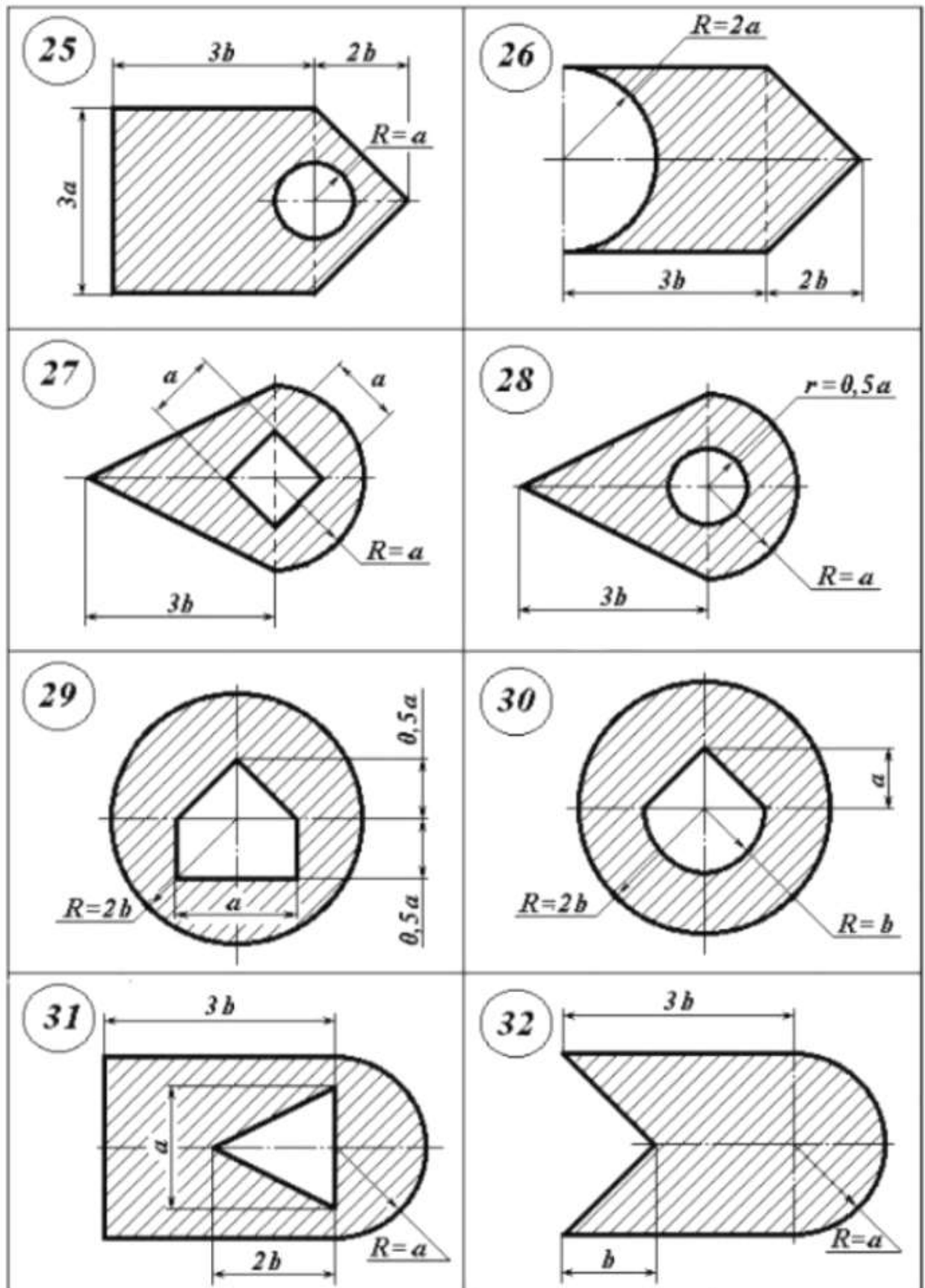
# REFERENCES

1. Carlota V., Top 10 Best CAD Software for All Levels, Frankfurt, Germany, 2016. – 183 p.

2. Lošonczi P., Kováčová L., Vacková M., Mesároš M., Nečas P., Case of the CAD program for creating 3d models, In: *Journal Of Security And Sustainability Issues*, Vol. 6, Slovakia, 2016. ISSN 2029-7017, P. 136-146.

3. Sachidanand J., T-FLEX CAD Exercises: 200 3D Practice Drawings For T-FLEX CAD and Other Feature-Based 3D Modeling. [Print Replica] Kindle Edition, 2019. - 111 p.

4. T-FLEX CAD Brief Introductory Course, M: *Top Systems*. 2008. – 210 p.

5. T-FLEX Parametric CAD. Three-Dimensional Modeling. User Manual. *Siemens Industry Software*, 2019. – 1371 p.

# VIDEO RESOURCES

1. T-Flex CAD 15 2D - Exercise 2 https://www.youtube.com/watch?v=705UCQnzlik T-Flex CAD 15 2D - Exercise 1 https://www.youtube.com/watch?v=ZUCMJ8otKzo

2. T-Flex CAD 15 Portugues - Ligao 1/99 - Funcionalidades e Aplicagoes https://www. youtube.com/watch?v=mNRZpUgR65E T-Flex CAD 15 English - Lesson 2/99 - Interface https://www.youtube.com/watch?v=gm6k80RERIs

3. T-Flex CAD 15 English - Lesson 3/99 - Drawing Techniques https://www. youtube.com/watch?v=qMnZk-HmdvE

4. T-Flex CAD 15 English - Lesson 4/99 - 2D Parametric Example https ://www. youtu be. com/watc h ?v=zZj B b N AY790

5. T-Flex CAD 15 English - Lesson 5/99 - Main Concepts of Operation https://www. youtube. com/watch?v=Aks16KKELB8

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