НАВЧАЛЬНО-МЕТОДИЧНА ЛІТЕРАТУРА

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

Sketching by Non-parametric Drawing

STUDY GUIDES № 1

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

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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 non-parametric (sketch) drawings of geometric objects in the system T-FLEX CAD 3D and acquisition of practical skills for their implementation.

1. GOAL

T-FLEX CAD allows creating a drawing similar to most well-known CAD systems, using standard functionalities for creating various primitives, such as: line segments, arcs, circles, ellipses, splines. Sketcher functionalities, including object binding and dynamic tooltips, significantly simplify and speed up the process of drawing creation. With the help of a sketch, you can create a non-parametric drawing, a parametric drawing or a combination of the two specified types. Non-parametric drawings do not share the advantage of parametric drawings in the effective usage of modifiable dimensions. However, in certain cases, development of such drawings is faster, requires less resources for constructions calculation and can bring certain benefits, when large modifications are not expected.

Sketching can be used as an independent way of creating non-parametric drawings, and in combination with the classic for T-FLEX CAD parameterization based on the construction lines. Parameterization based on sketching with constraints makes it possible to eliminate dependence on the order of the constructions, giving the user greater freedom when working on projects. To set or remove constraints between elements, you can change the values of the model parameters at any time while designing. If you remove constraints, the parametric sketch becomes non-parametric. Conversely, after creating a nonparametric sketch on it, you can specify constraints by making parametric a drawing or its parts.

Quick creation of drawing lines (both non-parametric and parametric based on constraints) is performed using the "Sketch" command group.

Introduction to basic concepts and methods creating Sketch and non-parametric drawing, acquisition of practical skills two-dimensional design in a software product environment T-FLEX CAD.

2. THEORETICAL INFORMATION

We will use the same familiar drawing example of the plate with a conical hole. Let’s begin with constructing the main view. Thereafter, we will create two projections, the “**Left Side View**” and the “**Plan View**”, using an object snapping mechanism.

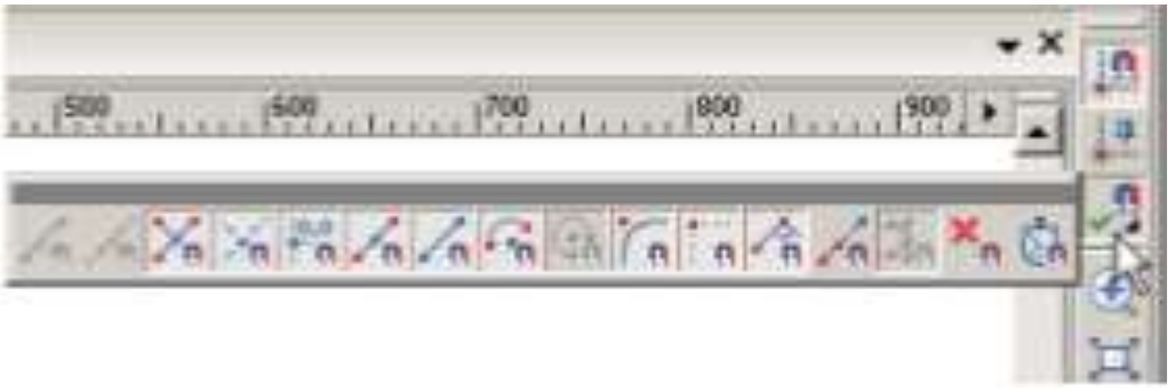
In this case, all construction is done using the command “**SK: Create Sketch**”. Call the command via:

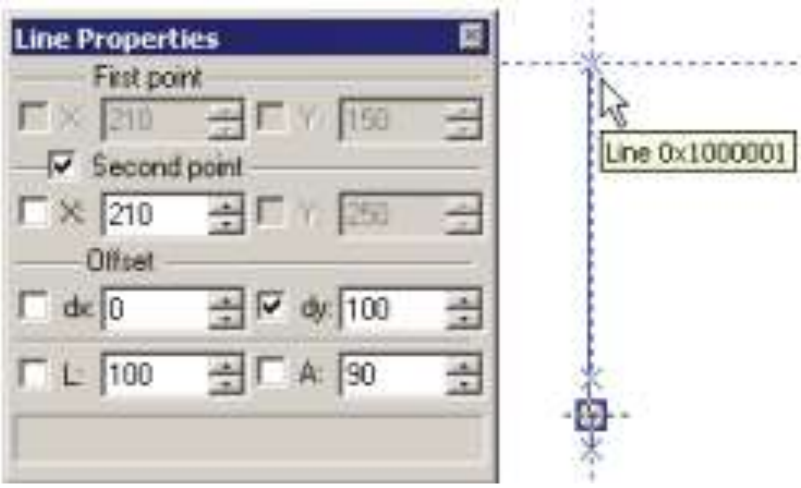
Keyboard	Textual Menu	Icon
<SK>	**«Draw|Sketch»**	

This command can be used to create either a sketch (non-parametric drawing) or a parametric drawing in the automatic parameterization mode. Since we are going to create specifically a sketch, please make sure that the automatic parameterization icon is switched off on the “**View**” toolbar:

	Automatic parameterization on/off

When creating a sketch, object snaps are widely used. The control over snaps is performed with the “**Snaps**” toolbar. To access this toolbar, press the icon  on the toolbar “**View**”.





All snaps turned on by the present moment correspond to the toolbar icons which have been pressed.

To turn off a pushed option, point at and click  on the respective icon. To turn off snapping completely, set the option:

	Clear all sketch Snaps

Unsetting this option sets all snaps on. In our exercise, the following snaps will be used:

	Line Midpoint
	Horizontal / Vertical
	Orthogonal
	Line Intersection
	Horizontal/Vertical tangent

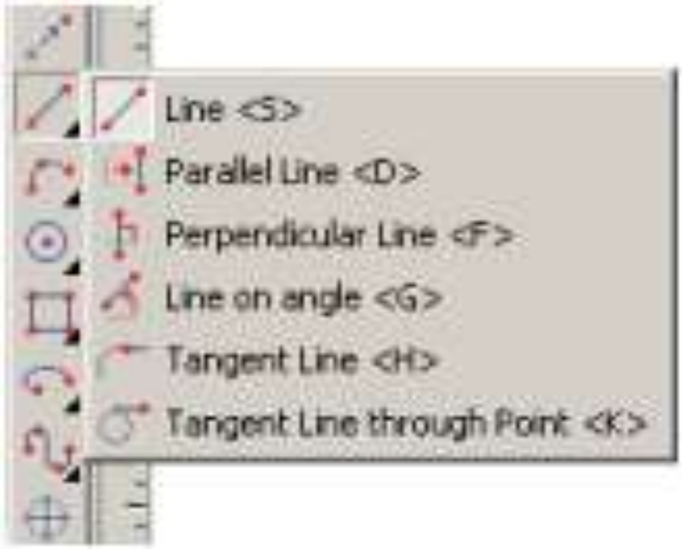
Push these icons on the “**Snaps**” toolbar. The object snaps can also be managed within the command “**SO: Set System Options**”, using “**Snaps**” tab.

When creating line segments, arcs and circles, the point coordinates can be defined by simply clicking in the drawing area. To specify exact node coordinates, one can use the property window. The two options are turned on automatically in the automenu while within the sketch creation command:

	<J>	Continuous creation
	<S>	Line

The first icon allows drawing continuously, so that the end of a just created line becomes the start of the new line. This mode will stay active until the user turns it off by pointing at the icon and clicking .

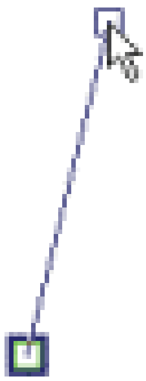
We recommend keeping this option on for speedy sketching. The other option sets the segment input mode. A black triangle in the lower-right corner of the icon marks availability of more options. To access these encapsulated options, press and hold a bit longer, and a menu of options will appear.



Attention: the automenu may display any of the encapsulated option icons in the given position. Usually this is the icon of the last used option among the encapsulated set.

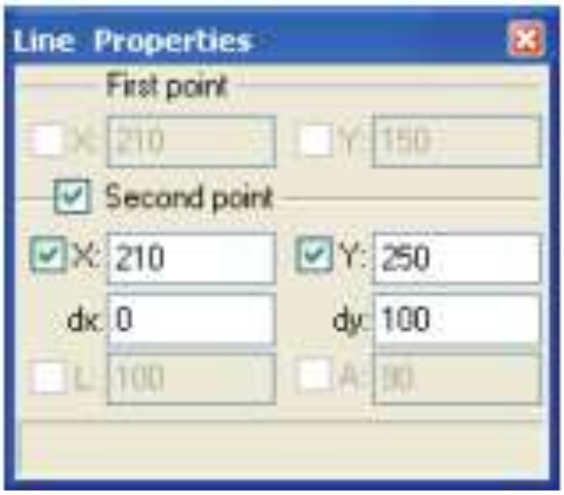
In “**Sketch**” automenu, each set of encapsulated options corresponds to a group of actions related to sametype element creation, such as creating segments, arcs, or circles.

The cursor on screen appears as a little square. Move the cursor to the intended location at the lower-right corner of the main view, near the middle of the screen, and press . This creates the first node of a line segment and starts rubberbanding the segment to be created. At the same time, the fixed coordinates of the first point are displayed in the property window.

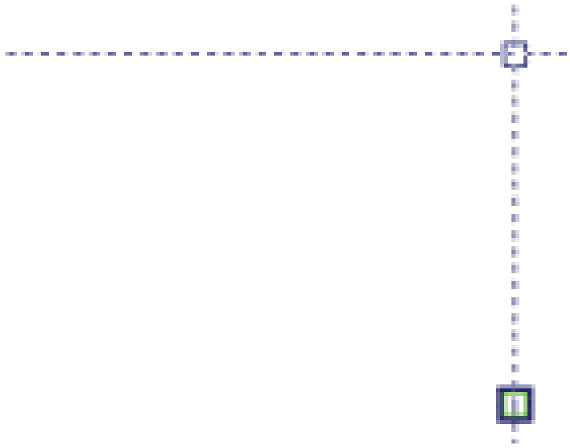


While sketching, consider leaving sufficient margins. This space will later be used for creating dimensions.

Move the cursor upward. Note that the coordinates of the cursor are dynamically updated in the property window, along with the vertical and horizontal shifts from the start node of the segment. We can use the property window for specifying exact location of the segment end. The end node can be defined in several ways. One way is to enter absolute Cartesian coordinates (X, Y) of the segment end node. Another way is specifying X and Y shifts of the end node from the segment start (dx, dy). Yet another way is to define the end of the segment in polar coordinates, or as a combination of the other ways.



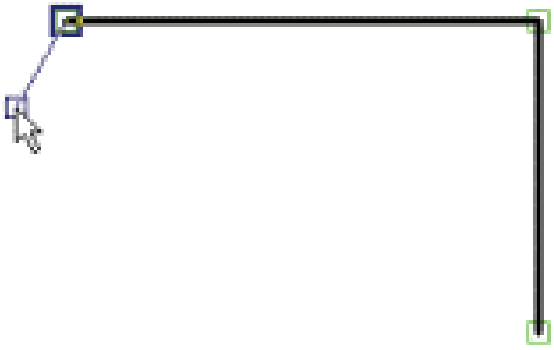
Let's create the end of the segment by specifying its shift from the start point. Make "dx" value equal to 0, and "dy" parameter equal to 100. The parameters "X" и "Y" will instantly update with the absolute coordinates of the segment end and get checkmarked. Checkmarking locks the value of the respective coordinate in spite of cursor movements. The end node of the segment will be displayed in the drawing window per the entered coordinate values.



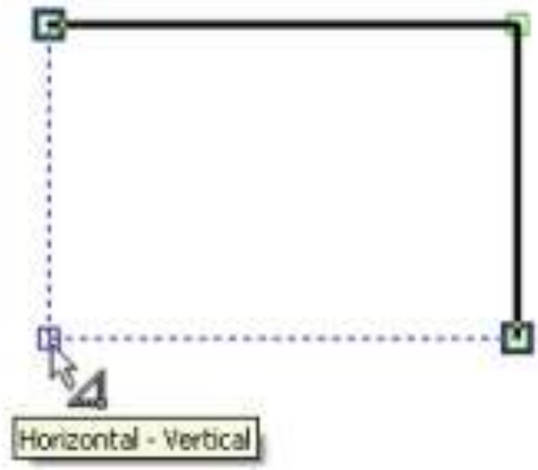
To complete the end node input, press [**Enter**] or click  in the drawing area. The first segment will be created. Move the cursor leftwards and place it so that it snaps to the horizontal constraint with respect to the last created node. The snap will be indicated by the specific glyph next to the cursor, and a pop-up help message saying “**Horizontal**”.

To lock this snap, press <**Space**> bar. Then, a horizontal helper line will be displayed passing through the node snapped to. The cursor will keep sliding along this helper line as an unattached node. The same effect can be achieved by setting the “dx” shift to 0 in the property window and locking the X coordinate with the checkmark.

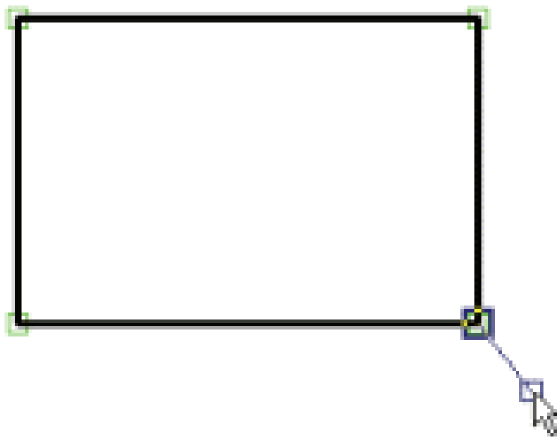
Place cursor on the side of the segment intended direction. Type the value of the “dx” shift for the end node of the segment being created in the property window. In our case, this value represents the width of the part and is equal to -150, while “dy” equals 0. The new segment will be created upon confirmation by pressing [**Enter**] or .



You are still within the segment creation mode. For further construction, you need to move the cursor downward to a point where snapping occurs to both horizontal and vertical constraints simultaneously. This will be indicated by a special glyph next to cursor and pop-up help message. Press . A new segment is created.

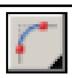


Move the cursor rightward to snap at the very first created node, as indicated by a glyph and the pop-up help, and press . We have thus completed the perimeter of the main view. We are still in line rubberbanding mode, with a new line stretching from the last created node to the cursor. Quit this line rubberbanding by clicking .



Now you are still within continuous segment input mode, with snapping active but no line rubberbanding after the cursor.

The next step is to round the corner of the plate. To do so, set the option:

	**<Ctrl+A>**	Fillet

This option belongs to an encapsulated set and may not necessarily be displayed in the automenu. Instead, it may be under the fillet/chamfer icon group (see explanation above).

Once the option is set, the property window changes appearance. Now it provides the input box for the fillet radius. Set the radius value equal to 31:

What is left now is to select two segments to be filleted. In our case, it is the top and the right-hand-side segments of the plate. Once the second segment is selected, the fillet is created, and the segments trimmed appropriately.

Now let’s draw the conical hole on the main view. To do so, let’s create two centerlines whose intersection will define the exact location of the center of the hole. Set the option:

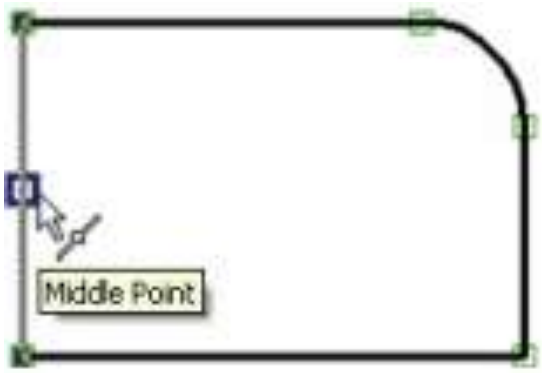
	<S>	Line

Once this is set, rubberbanding resumes with a line attached to the last created node. Reject this line by clicking . To create centerlines, set the appropriate line type.

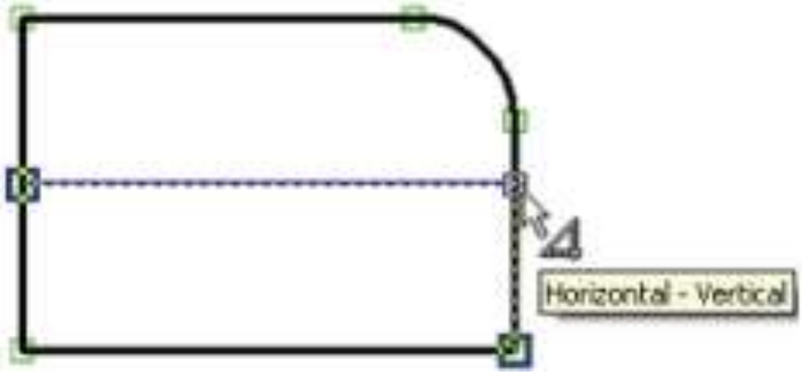
Set the line type to DASHDOT in the system toolbar or in the graphic line parameters of the dialog box. Call the dialog box by:

	<P>	Set Graphic line parameters

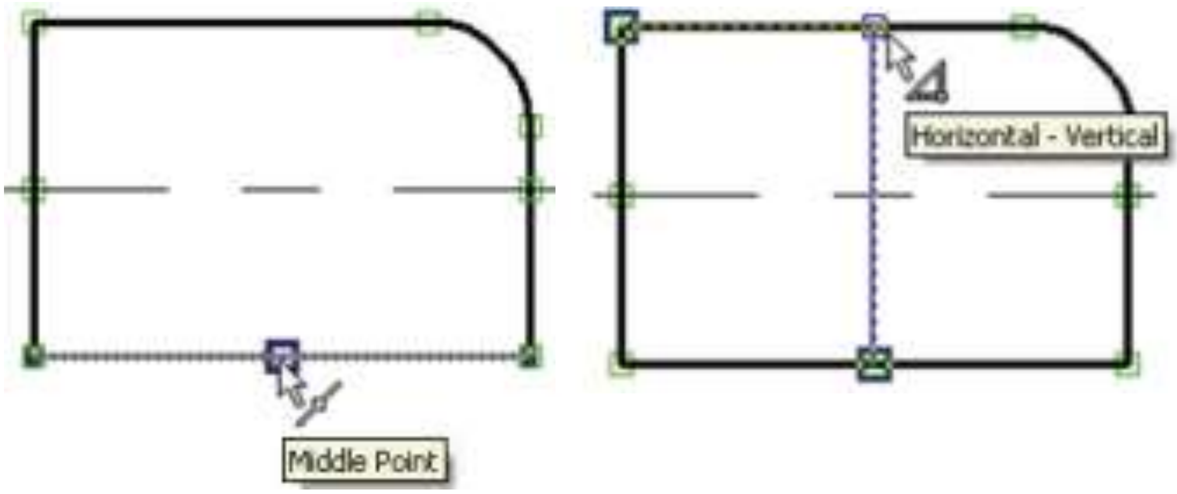
Then move the cursor to the left-hand-side graphic line segment to get vertical snapping to one of the segment nodes, and slide the cursor along the segment to its midpoint. When the cursor reaches the midpoint, the glyph beside the cursor will change to indicate this, along with the pop-up help.



Press here. A node will be created at the midpoint, and a line will start rubberbanding from this node. Move the cursor horizontally to the right-hand-side graphic segment and stop at the intersection of the horizontal and vertical projections of the two nodes as shown on the diagram.



Press , creating a centerline and a node. Rubberbanding resumes from the latest node. As we do not intend to construct another line through this node, quit rubberbanding with . Then follow the same steps to create the vertical centerline, beginning at the bottom segment.



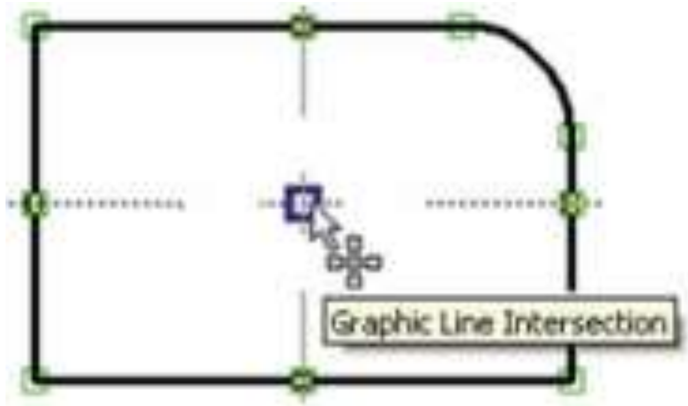
Now, let’s create circles. First, reset the line type to CONTINUOUS by selecting in the system toolbar or in the command parameters dialog box. Call the dialog by typing <**P**>.

Then pick the option:

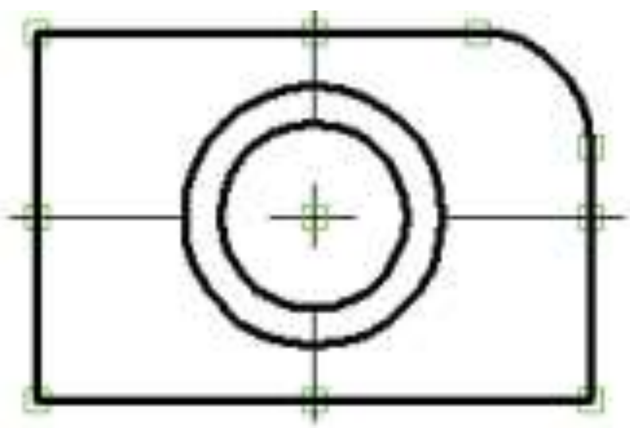
	<O>	Circle By Center And Radius

This is also an encapsulated option that may not be shown on the automenu, rather, be within a group of options.

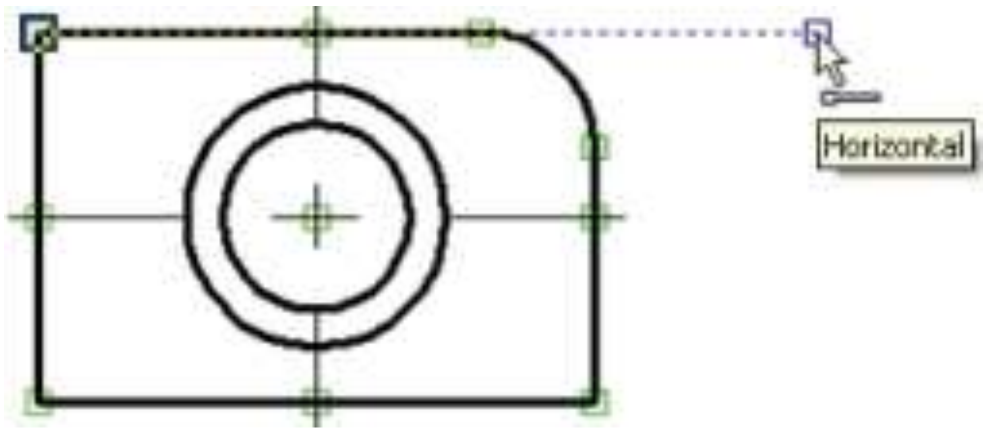
After activating the option, move the cursor to the intersection of the two centerlines. Both centerlines will get highlighted, and the cursor will gain a glyph and a pop-up help of the graphic line intersection snap. Press here. A circle starts rubberbanding on screen.



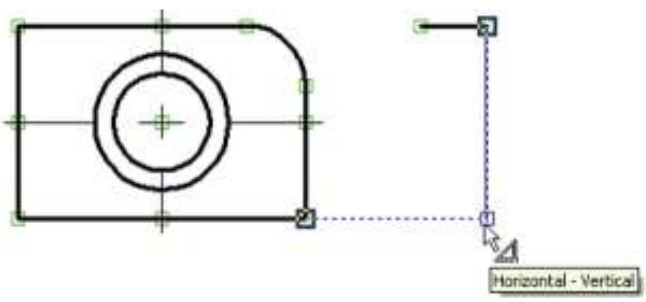
Enter the value of the radius of the smaller circle of the cone equal to 25 in the property window, and press [Enter] key. A full circle is now fixed on screen. Now you are still in the circle creation mode. Select the node at the two centerlines intersection and create a circle of a bigger radius, 35. This mostly completes creation of the main view of the part.

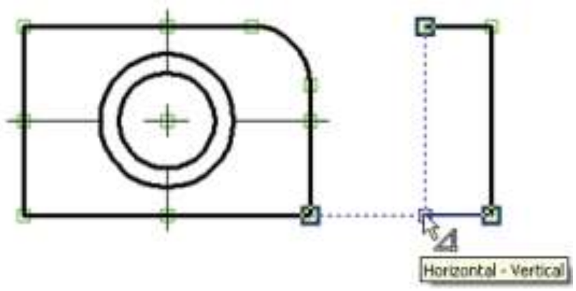


Now, let's construct the left side view. Again, set the line creation mode via the option. Line rubberbanding resumes on screen with the line attached to the end node of the last created segment. Since we are not making a line from this node, quit with . Move the cursor to the right of the drawing area and place it so to snap to the horizontal constraint with respect to a node of the top line of the main view.



Click here , and move the cursor horizontally rightwards. In the property window, set the end node shifts. Set the X equal to 35, Y to 0. Press [**Enter**] or . The new segment will be fixed on screen, and rubberbanding resume from the last created node. Next, move the cursor vertically downward maintaining the "vertical" snap, up to the point when snapping occurs with a node of the bottom line of the main view. Click there , then move the cursor leftwards to snap against the left-hand-side end of the top segment.

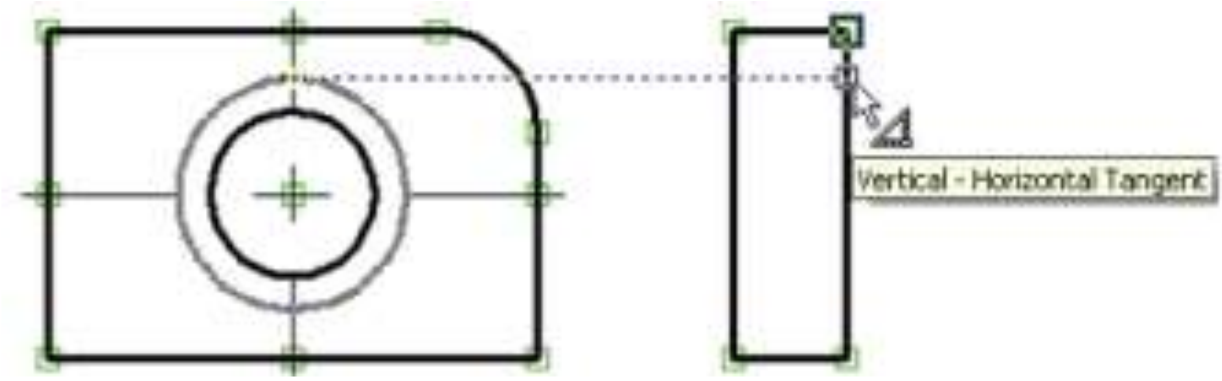




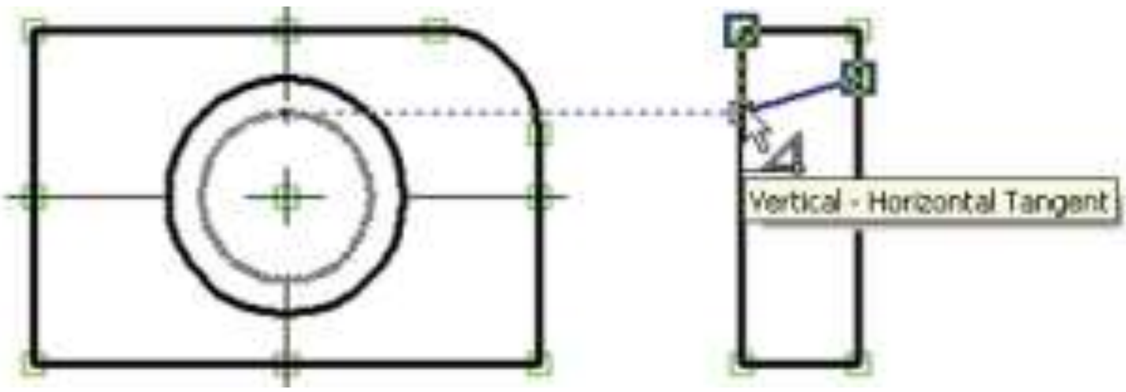
Click . Now close the perimeter of graphic lines by moving cursor to the first created node on this view, and clicking , and then .

> One-degree-of-freedom snaps can be locked by pressing <**Space**> bar.

Next, we need to create the image of the conical hole on the side view. Without quitting the current command, move the cursor to the right-hand-side segment of the side view, and move along the segment until it snaps to horizontal tangency against the top of the bigger circle.

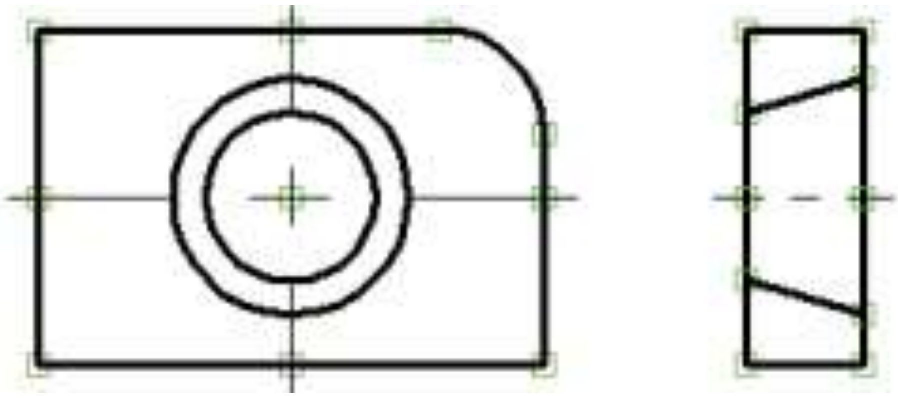


Click at this spot, then move the cursor to the left-hand-side segment of the side view and locate so to get it snapped against the smaller circle.



Click , and a segment will be created, with rubberbanding resuming from its end node. For now, quit rubberbanding by clicking .

Then construct the lower segment of the hole view in the same way. Next, using already familiar snapping constraints, construct the centerline, setting the line type to DASHDOT in the graphic line parameter dialog box (the key <**P**>) or in the system toolbar.

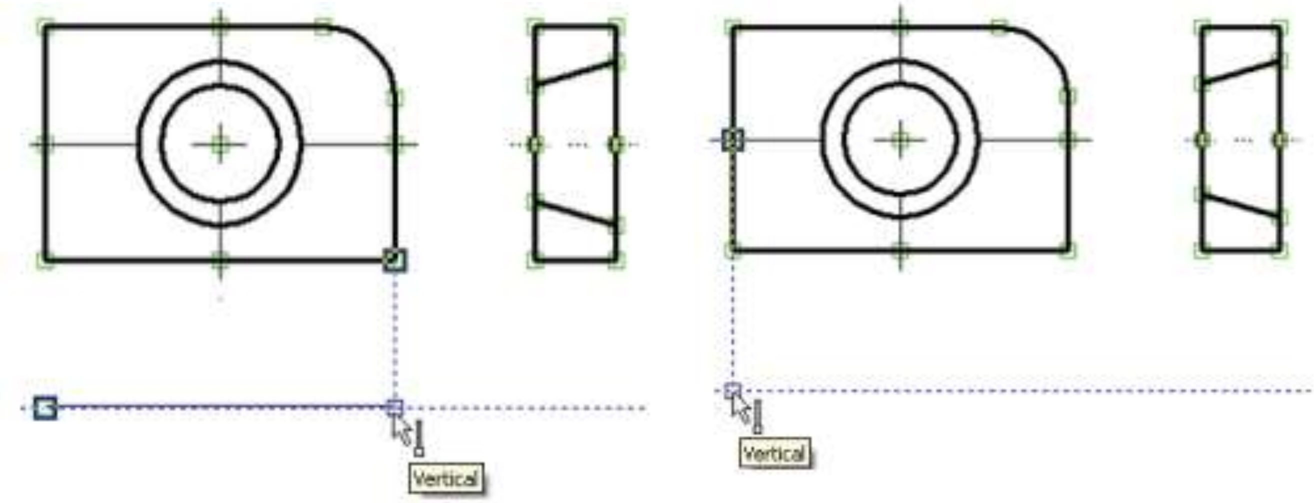


Proceed with the plan view. This view can be created in the same way as the side view. However, for deeper exploration of non-parametric drawing capabilities, we will follow a different approach. Set the option:

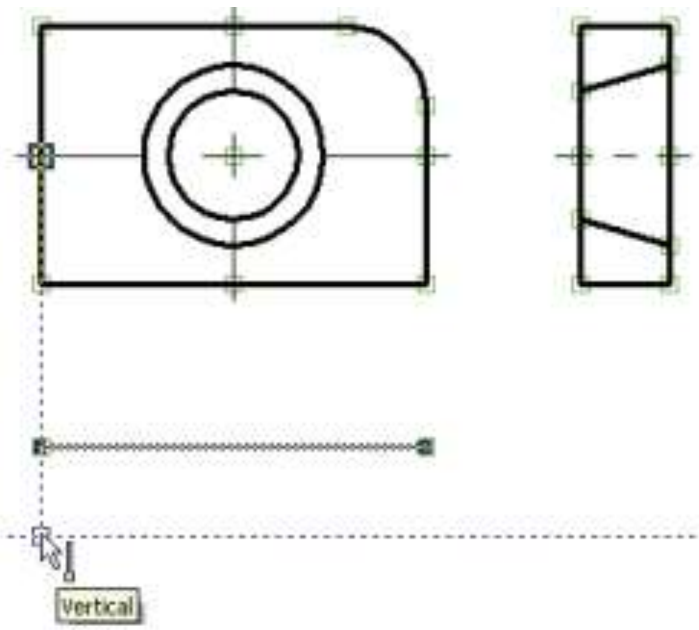
	<D>	Parallel Line

This is an encapsulated option in the segment creation group. If this icon is not displayed in the automenu, find it under one of the group icons marked with a black triangle (see explanation above).

Once this option is set, the cursor starts rubberbanding an auxiliary infinite line parallel to the last created segment. The reference segment is highlighted. The current reference suites us. Otherwise, we would reject the system-selected segment with and select an intended one to construct a parallel line. Make sure the line type is back to CONTINUOUS in the graphic line parameter dialog (the key <**P**>) or in the system toolbar. Move the cursor to get a snap against a node of the main view, and click . A node will be created at this point, and the auxiliary line will get fixed. Slide the cursor along the line up to the point of another vertical snapping, and again click .

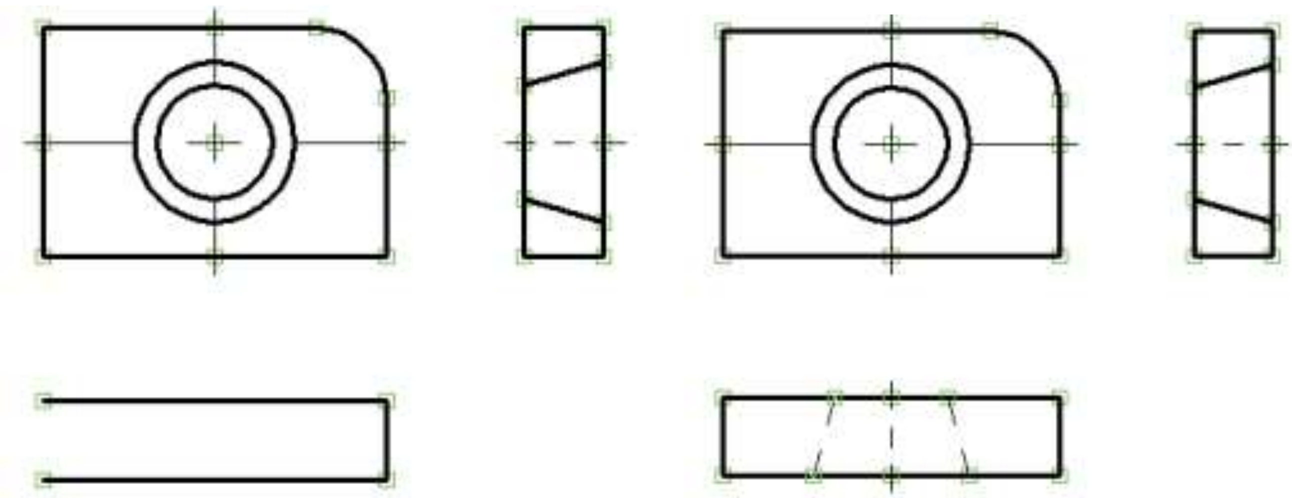


Thus, we have created the top segment of the plan view. A new auxiliary line starts rubberbanding after the cursor, parallel to the newly created graphic segment, as indicated by highlighting. Move the cursor down, and set the desired distance, equal to 35, in the property window, thus defining the thickness of the plate. This will fix the auxiliary line with respect to the reference segment. Slide the cursor along the line, locating as shown on the diagram.



Click , fixing the start node of the segment being created. Slide the cursor rightwards to get vertical snap against the end node of the reference segment, and again click . The bottom segment will be created.

At this moment, parallel line rubberbanding resumes. Now, set the option , thus switching to normal continuous line input mode. A line will start rubberbanding, attached to the last created node. Move the cursor upward to the top segment node, and click , and then . Next, connect the left-hand-side ends on the plan view with another segment. Create the centerline and the lines of the conical hole projection in the same way as on the side view. Doing so, maintain the appropriate line types.



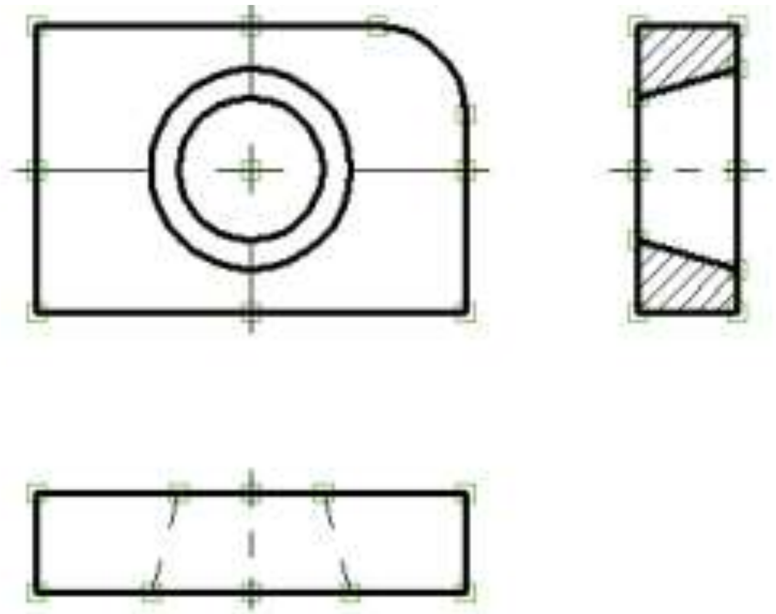
What is left now is to apply hatch on the side view. Call the command “**H: Create Hatch**”:

	<H>	Create Hatch

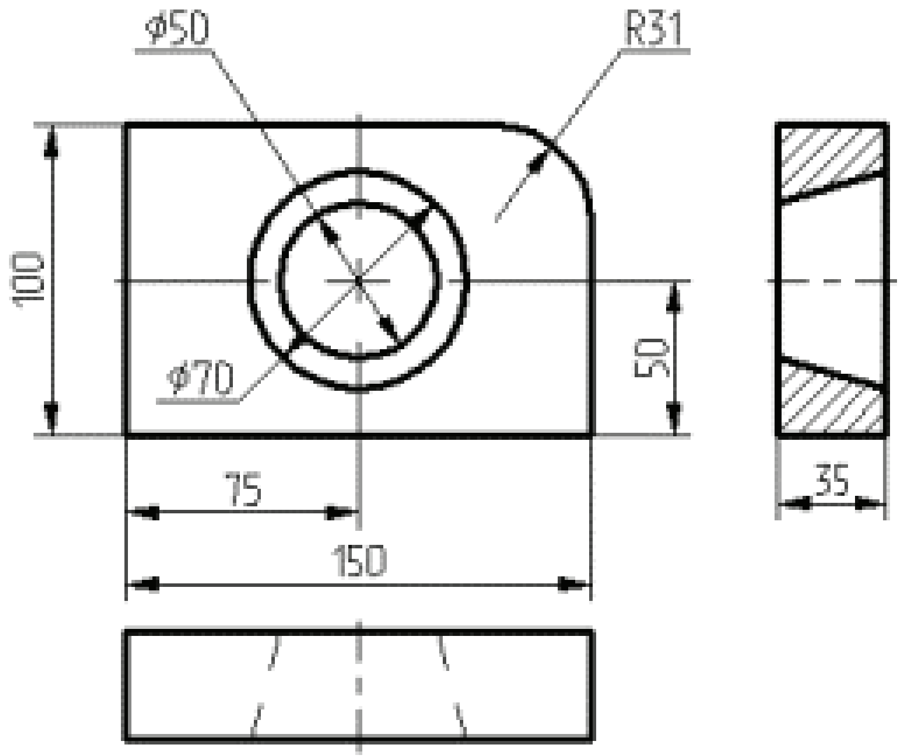
Set the option:

	<A>	Automatic Contour search mode

Then move the cursor to the top portion of the side view, place it in the middle of the area to be hatched, and click . The closed contour will be highlighted. Now move the cursor to the lower portion of the view, and similarly select the other contour to be hatched. Then pick the button.



Now, let’s create the necessary dimensions on the drawing. Dimensions are created on a sketch in the same way as on a parametric drawing. One can select graphic lines instead of construction lines in this case. Let’s skip the detailed description of this functionality, as it was described in depth as part of the main drawing technique.



This completes creation of the non-parametric drawing. Further modification of its elements will not affect the whole drawing. One would have to modify each view separately. The elements of such a drawing cannot be related by variables. All other functionalities such as use of visibility levels, layers, hide/show construction entities, etc. are fully supported.

3. THE ORDER OF LAB WORK

1. Get acquainted with the system and functions of T-FLEX CAD 3D.
2. Start the T-FLEX CAD 3D program.
3. Create a non-parametric drawing of the part according to the individual task (Tables 1,2).
4. Conclude lab work.
5. Complete a report.

4. REPORT STRUCTURE

1. Theme and purpose of lab work.
2. Brief theoretical information.
3. Non-parametric drawing of the part.
4. Conclusion.
5. Answer the questions from the checklist.

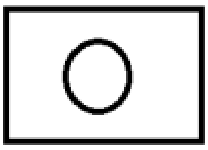
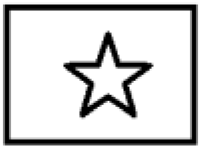
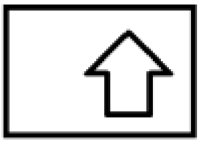
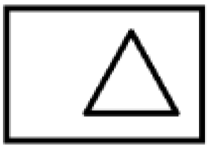
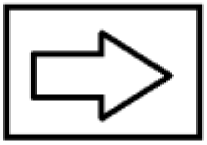
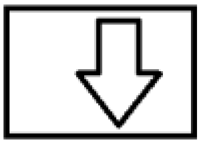
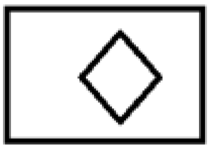
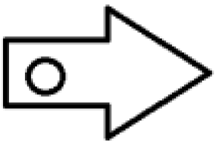
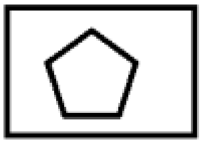
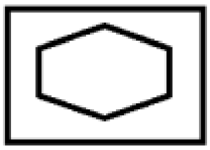
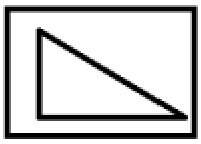
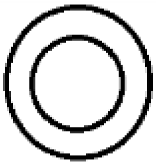
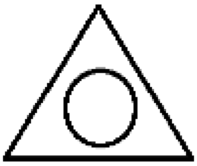
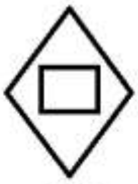
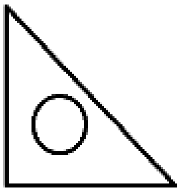
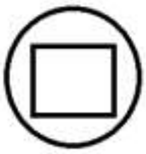
5. CHECKLIST

1. Purpose, functions and properties of T-FLEX CAD 3D.
2. Algorithm for creating a non-parametric drawing.
3. What is the basis of non-parameterization?
4. What is the difference between a parametric drawing and a non-parametric one?
5. Construction of sketches of machine parts in frontal, horizontal and profile planes of projections.
6. How to edit plot lines and image lines?
7. How are image lines applied to drawings?
8. How are construction lines applied to drawings?
9. Algorithm for entering text on the non-parametric drawing.
10. Algorithm for marking dimensions in the non-parametric drawing.

Task 1
for lab work № 1

Draw a drawing (sizes are arbitrary) using T-Flex CAD.

Table 1

№ var.	Drawing	№ var.	Drawing	№ var.	Drawing
1		8		16	
2		9		17	
3		10		18	
4		11		19	
5		12		20	
6		13		21	
7		14		22	
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Task 2

for lab work № 1

Create the parametric drawing using T-Flex CAD.

Table 2

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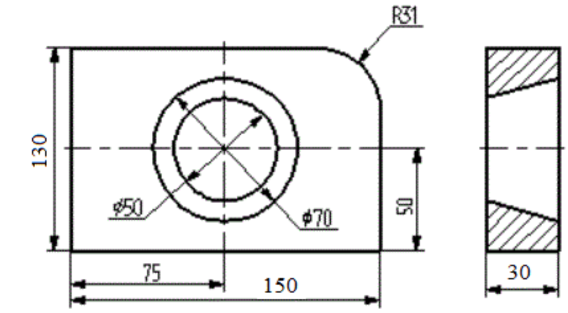
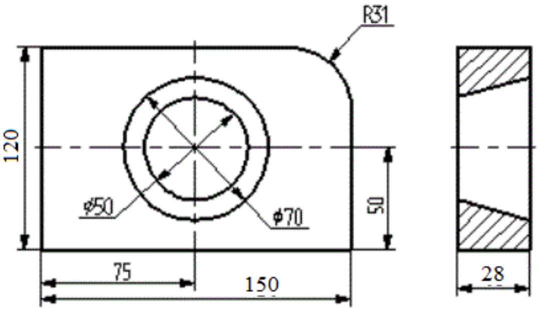
<tr><td>1</td><td>
</td><td>5</td><td>
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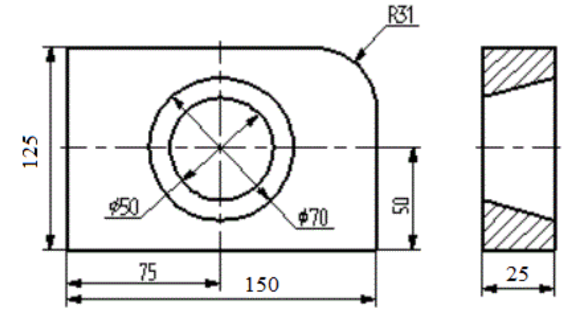
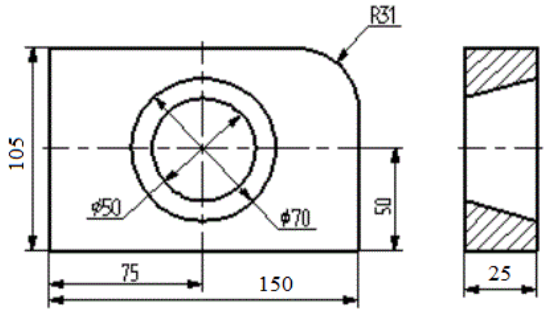
<tr><td>2</td><td>
</td><td>6</td><td>
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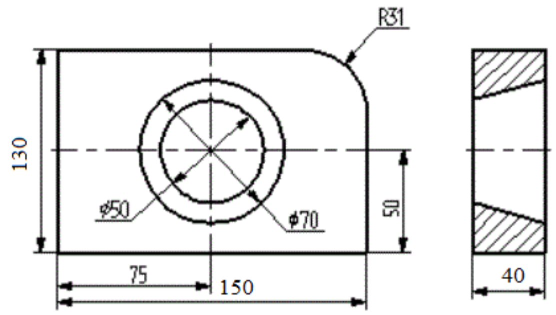
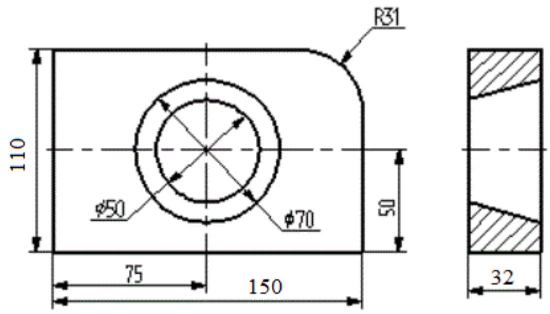
<tr><td>3</td><td>
</td><td>7</td><td>
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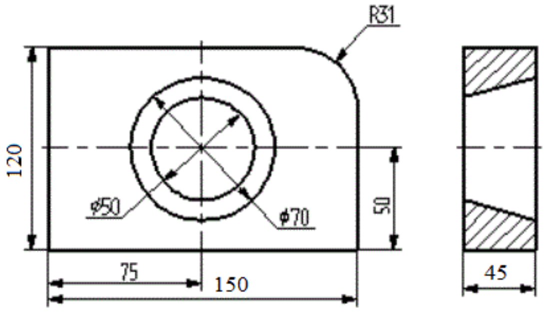
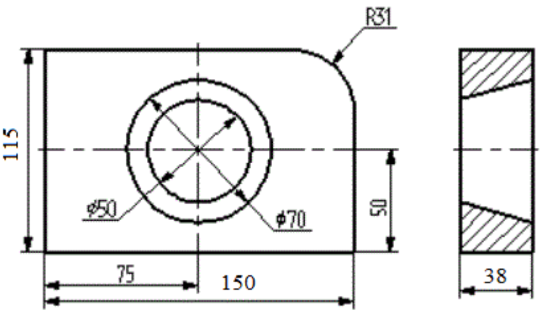
<tr><td>4</td><td>
</td><td>8</td><td>
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2. Lošonczi P., Kováčová L., Vacková M., Mesároš M., Nečas P., Case of the CAD program for creating 3d models, In: *Journal Of Security And Sustainability Issues*, Vol. 6, Slovakia, 2016. ISSN 2029-7017, P. 136-146.
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

CONTENTS