

УНІВЕРСИТЕТ



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

## Creating Parametric Drawing

**STUDY GUIDES № 2**

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

## **1. GOAL**

The parametric drawing is a method of making geometries with geometrical or dimensional constraints or restrictions. These constraints are applied to the geometry with respect to certain references or other geometries. Parametric drawings update itself if any one parameter is changed, this also helps in maintaining design intent.

Geometry in parametric drawing is constrained with a set of mathematical and geometrical rules with respect to other geometries of the drawing. A parametric approach to design offers a big list of advantages over classical non-parametric design approach. Consider a scenario where you have completed designing a part having complex geometry. Suddenly you realize that something needs to be changed in the drawing. Making that change will affect your complete design, and with each change you need to update its dependent component geometry manually.

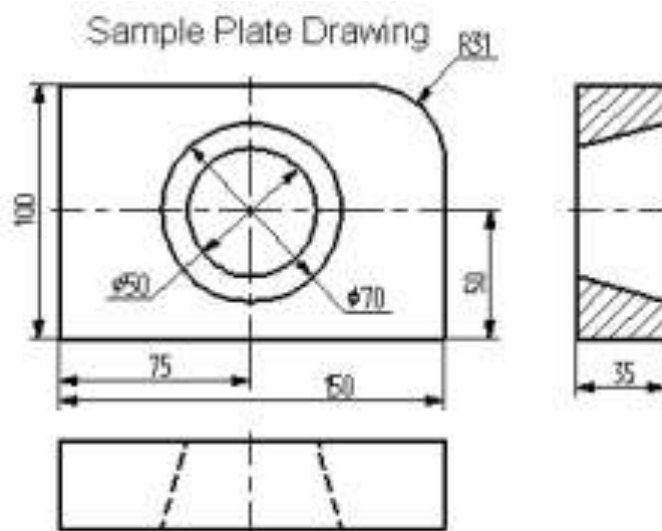
That's a tedious enough job by itself, but what if the same issue happens with an assembly drawing? You get the point; it will be a nightmare to update every part and then the overall assembly to ensure changes are properly reflected everywhere! It's also clearly apparent that this task will be error prone and it will cost you extra design hours. Now, if the design is made with a parametric approach, then a design change will automatically update any dependent geometry, eliminating the chances for human error and, best of all, saving you lots of design editing time.

## 2. THEORETICAL INFORMATION

The following figure shows a drawing to be created. It is a plate with a through hole of conical shape. The drawing will be defined parametrically so that any modifications will automatically reflect on all projections.




Let's begin with the main (elevation) view of the plate. First, create the necessary “thin” construction lines, and then draw the graphic lines on top. Next, create the other two views using the construction lines of the main view. This creates a dependency between the views so that the two views automatically adjust to the main view modifications. Finally, apply text and dimensions.

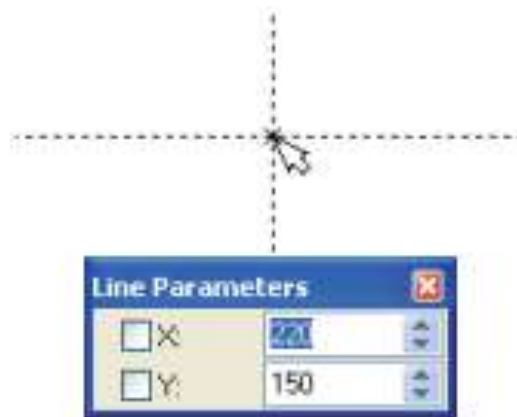
As was mentioned, any command can be called by a number of ways. It can be typed on the keyboard, selected from the textual menu, or picked on a toolbar.



Let's begin with the command “**L: Construct Line**”. To invoke the command, use:


Keyboard	Textual Menu	Icon
<L>	«Construct Line»	

Pick the icon  at the top of the automenu. A crosshair appears that follows cursor dynamically. The current coordinates of the crosshair crossing point are displayed in the status bar. There are several ways to define the crossing point. One is to simply place the cursor near the center of the drawing window and press . To define the crossing point more precisely, specify its coordinates in the property window. The coordinates can also be specified via a parameter dialog invoked by typing <P> key or picking the icon  in the automenu.





As a result, two crossing construction lines will be created. Besides, a node is created at the intersection point. These lines make the basis of the view being created. The line parameters represent the absolute coordinates. The view can be moved around on the drawing by moving the base lines.


Do not use more than two base lines on the main (independent) view, and more than one base line on the views defined by projections. This will insure freedom in placing the drawings.




A T-FLEX CAD command stays active up until it is quit or another command is called. Quitting the crosshair mode (as by pressing  once) cancels the crosshair rubberbanding, but the line creation command stays

active. After canceling the crosshair mode, move the cursor close to the vertical line. The line will get highlighted, and a pop-up help will appear next to cursor displaying the name of the highlighted entity. This is object snapping in action. This behavior relieves the user from typing on keyboard or using the automenu buttons.

The object snapping is on by default when starting the application. To set or unset this mode manually, use the button  on the “View” toolbar.


Pressing  now starts rubberbanding of a line that follows the cursor while staying parallel to the selected one. We are now creating a line parallel to a vertical line. Such a relationship between the two construction lines, established at the creation time, is an example of an important feature of T-FLEX CAD system. This defines behavior of a set of construction entities under parametric modifications.


Place the new line at the left of the highlighted vertical line by pressing . The exact value of the distance can be specified in the property window or parameter dialog box. The newly created line will become the left-hand side of the part.

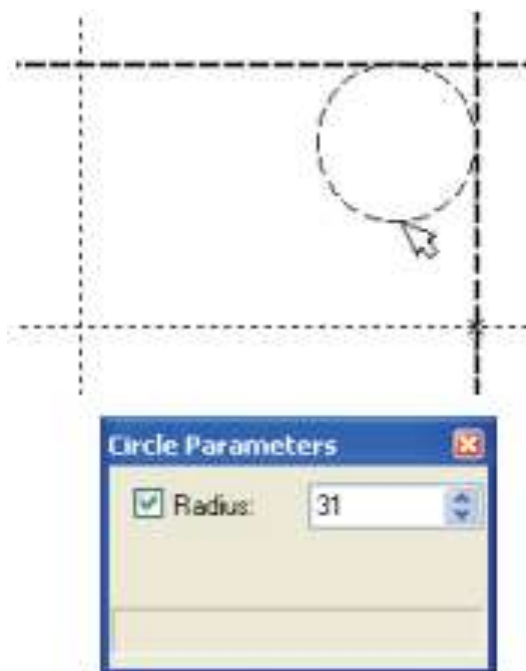
Pressing  once cancels the parallel line creation mode, yet the line creation command stays active. (Otherwise, call it again.) Next, move the cursor toward the horizontal line and press . The line is selected as a reference for a parallel line to be created. Move the cursor up, specify an property window, and press  fixing the top side of the part.





The next step is to round a corner of the plate with a fillet. For this purpose, let's use the command **“C: Construct Circle”**. Call the command via:

Keyboard	Textual Menu	Icon
<C>	«Construct Circle»	


To draw the fillet at the upper-right corner of the plate, construct a circle tangent to the top and the right-hand-side lines. Move the cursor to the top line and press  or <L>. This starts rubberbanding of a circle whose radius adjusts as the circle follows the cursor while the line tangency stays intact. This means a circle is being constructed that is tangent to the top line. Any future modifications of the top line location will not break the circle tangency condition.






Next, move the cursor to the right-hand-side line and again press  or <L>. Now, the circle becomes “tied” to the two construction lines and keeps the tangencies while being rubberbanded. Pressing  fixes the current circle radius. The exact value of the radius can be specified in the property window.

If the resulting construction does not match the illustration, use “**UN: Undo Changes**” command,


Keyboard	Textual Menu	Icon
<UN>, <Ctrl><Z>, <Alt><BackSpace>	«Edit Undo»	

Each call to this command brings the system one step back. If this command was called mistakenly, its action can be reversed with the command “**RED: Redo Changes**”,


Keyboard	Textual Menu	Icon
<RED>, <Ctrl><BackSpace>	«Edit Redo»	


This command restores the action that was mistakenly undone.

One can remove all construction lines and start creating a drawing from the beginning with the command “**PU: Delete Unused Construction**”:


Keyboard	Textual Menu	Icon
<PU>	«Edit Purge»	

This command will delete all construction entities and allow to start drawing anew. A specific construction entity can be deleted using command “**EC: Edit Construction**”:


Keyboard	Textual Menu	Icon
<EC>	«Edit Construction 2D Construction»	

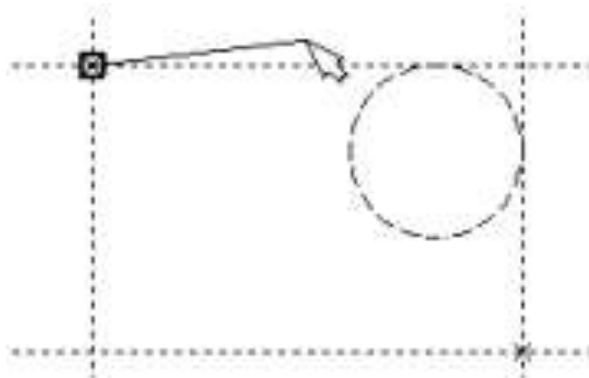
Once the command is called, select the entity and delete it by pressing <Delete> key on the keyboard or by picking the icon  in the automenu.


Now, draw the graphic lines on top of the completed construction portion of the drawing. To do so, let's create graphic lines by calling “**G: Create Graphic Line**”. Call the command via


Keyboard	Textual Menu	Icon
<G>	«Draw Graphic Line»	

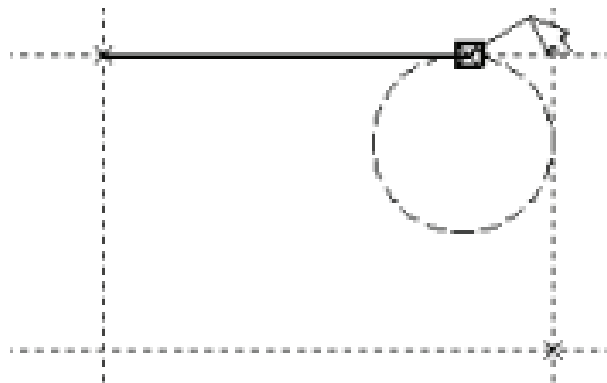
Note that the previous command is automatically terminated when calling another command via the toolbar icon button or the textual menu (no need to cancel the previous one explicitly).

Start drawing solid lines from the upper-left corner of the plate. The graphic lines snap automatically to a closest intersection of the construction lines. Simply move cursor to an intersection and press . The line will be rubberbanded after the cursor. Just keep selecting nodes or construction line intersections.



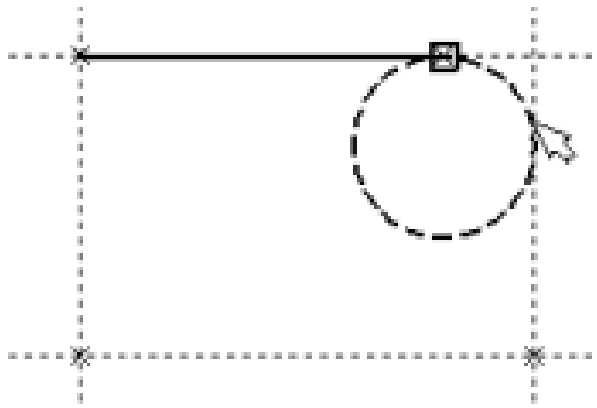
It is not recommended to select multiple (more than two) line intersections neither by pressing **<Enter>** nor by . In this case, we recommend creating nodes at such intersections first. The graphics can then be applied using the **<N>** key. When using the **<Enter>** key in “free drawing” mode, a “loose” node will be created that is not constrained to any construction line. Following these tips insures correct parametric function of the drawing under modifications.


Move cursor to the tangency point between the top line and the circle, and press . What you see on screen now should be similar to the illustration at right. Note that T-FLEX system automatically adds nodes to the end points of the graphic lines, unless already created.

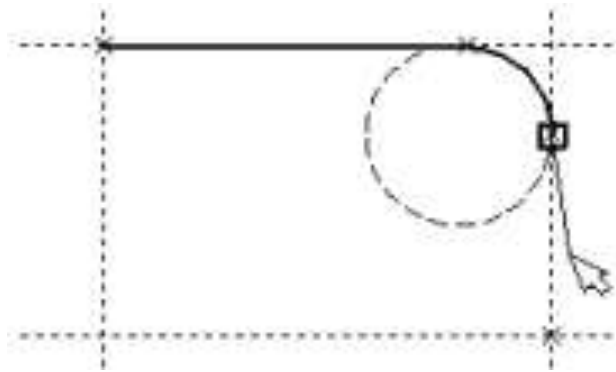




Now let's draw a graphic line along the circle to construct an arc between the two tangency points. To do so, move the cursor to the circle and press **<C>** key. The circle will then get highlighted. The direction of arc creation depends on the position of the cursor when selecting the circle. To change the arc direction, press the **<Tab>** key.

Place the cursor just above and to the left of the second tangency point as shown.

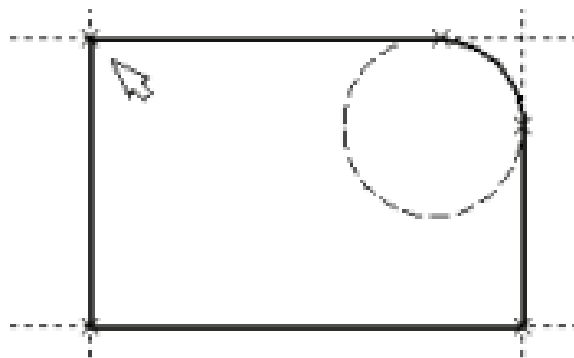


Then press , and the graphic arc will be created in the clockwise (CW) direction, spanning to the second tangency point. The result should look like on the diagram.







Continue drawing. Select with  the lower-right corner of the plate, then the lower-left one, and finish the construction in the upper-left corner where the drawing started. To complete the command press .

The drawing should look as shown here.




If applying graphic lines did not come out as desired, edit the graphics using the command **“EG: Edit Graphic Line”**. Call as follows,

Keyboard	Textual Menu	Icon
<EG>	«Edit Draw Graphics»	

Move the cursor to one of the lines to be edited, and press . This selects the line. It can then be deleted by pressing <Delete> key or picking the icon  in the automenu. Repeat for each line to be edited. If a whole area is to be edited, use box selection. To select by box, press  where one of the box corners should be, hold and drag to the desired location of the opposite corner, then release the button. As you drag the cursor, it rubberbands a rectangle of the selection box. The elements will be selected that are fully within the box. All these elements can be deleted at once.


To apply graphic lines again, call the command “**G: Create Graphic Line**”. To redraw the screen at any moment use the <F7> key, in case not all lines are displayed properly after editing.



Once the desired image is obtained, proceed to the next step of drawing creation. The drawing can be saved preliminary with the help of “**SA: Save Model**” command:

Keyboard	Textual Menu	Icon
<SA>	«File Save»	

Congratulations! You have accomplished your first drawing in T-FLEX CAD. Now let us briefly describe the system editing capabilities.

The current drawing uses five construction entities that define the shape and size of the part. These are the left-hand side, the right-hand side, the top, the bottom and the fillet radius. To modify construction entities call the command “**EC: Edit Construction**”:




Keyboard	Textual Menu	Icon
<EC>	«Edit Construction 2D Construction»	

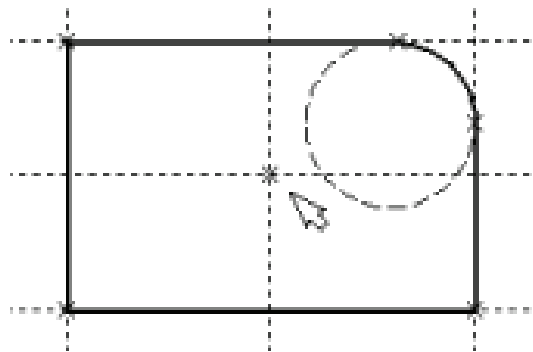
Move the cursor to the left-hand-side vertical line and press . The line gets highlighted. As you move the cursor left to right, the line will move along. Specify the new position of the line by pressing . The width of the plate will change. Note that modifying locations of construction entities causes instant update of their respective “snapped” graphic lines. If you try to move the right-hand side of the plate then the whole plate will move. This is because the left-hand side was created as a dependent of the right-hand one, and the dependency stays as the right-hand side is modified.

However, the left-hand side can move independently of the right. Try such manipulations with other construction entities, including the circle. As the construction entities move the size and shape of the plate will be changing while maintaining the dependencies defined at the construction time.


After testing modification capabilities of the system please bring the drawing back into an approximately original configuration as shown on a diagram above. Let's proceed with the next element of the drawing, which is the conical hole in the middle of the plate.



First, let's define the center of the circle to be constructed. To do so, let's do auxiliary construction to define the center point of the plate. T-FLEX CAD provides a handy command to create a line in the middle of two others. For two parallel lines, this command creates a parallel line in between at equal distances to the two. For intersecting ones, the resulting line passes through the intersection at equal angles to the two original lines. Thus, the new line appears as the symmetry line for the two.


Call the command **“L: Construct Line”** and choose the icon  in the automenu. Move the cursor to the right-hand side of the plate and select the vertical line by . A parallel line appears rubberbanding after the cursor. Move the cursor to the left-hand side of the plate without fixing the rubberbanded line. Now, select the left-hand-side vertical line with . This creates a new vertical line on the drawing that is the symmetry line for the two selected ones.





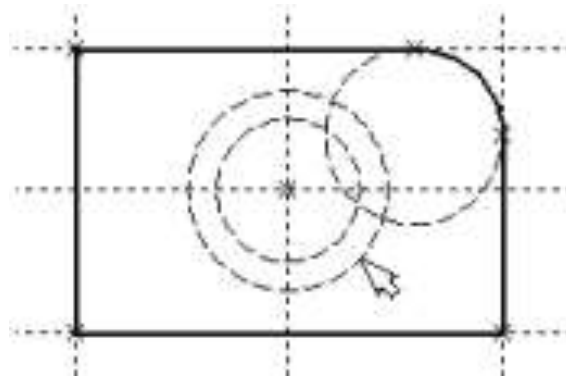
Follow same way to create a horizontal line as the symmetry line for the top and bottom sides of the plate. The intersection point of the two new lines will be the center of the hole to be constructed.



Next, call the circle creation command, move cursor to the intersection of the symmetry lines, and press . This starts rubberbanding of a circle with the fixed center, with the radius adjusting to the cursor position.

The circle center snaps to the node created automatically at the intersection of the symmetry lines. Fix the circle with . Just like line-to-line distances, the circle radius (diameter) can be defined approximately by mouse operation, and exactly in the property window. Note that after pressing  the command **“C: Construct Circle”** stays active.

The second circle of the conical hole can be constructed as concentric to the first one. To do so, pick the icon  in the automenu or


type <O>. Then select the existing circle with . The new circle starts rubberbanding after the cursor. Place the cursor so that the rubberbanded circle is slightly bigger than the original one, and fix with . The exact radius difference can be managed via the property window.




Proceed with the command “**G: Create Graphic Line**”, move cursor to the bigger of the two circles, and press  or <C>. The circle gets drawn in solid. Then, move the cursor to the smaller circle, and again press  or <C>. Now both circles are drawn solid. From this point, we can proceed with the two other views of the plate.

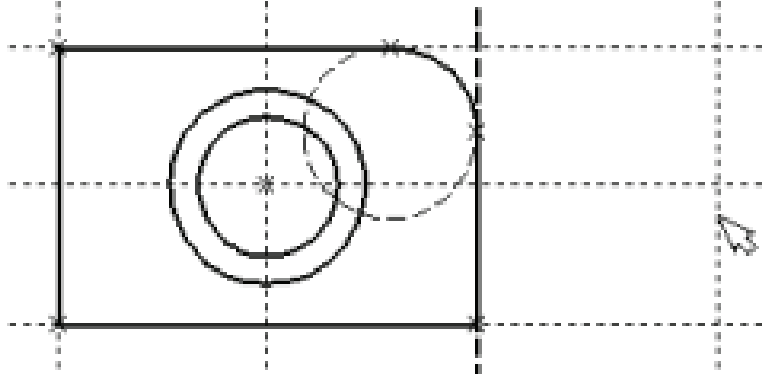
The two other views are not required for constructing a parametric drawing in T-FLEX CAD. In this example, creating the side and the plan views simply help demonstrating additional advantages of parametric modeling using T-FLEX CAD system.

Since the straight lines are considered infinite, one can see that the other views (side and plan) are already partially created. To finalize the drawing, we will need to establish additional dependencies between the construction lines. These additional steps are described next.

Activate the line creation command and move the cursor to the construction line defining the right-hand side of the plate. Select it with . This highlights the vertical construction line and starts rubberbanding of a new line parallel to the selected one. This line will be the right-hand side






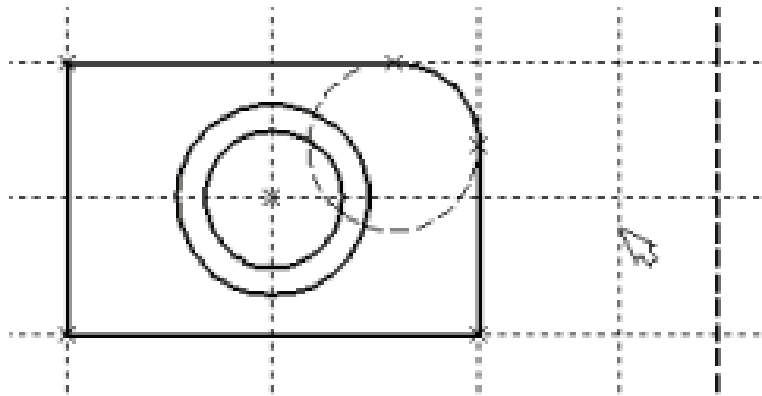
of the plate side view. Fix it at a desired location by pressing . As before, the exact distance from the selected line can be specified in the property window.



The new line is created relative to the right-hand side of the plate on the main view. Therefore, when the right-hand side of the plate is moved, the new line will follow, staying at the same distance. To place the new line at a different distance, use the command for editing construction lines. After that, moving the right-hand side of the plate will again preserve the new distance. The dependencies between construction entities stay valid until redefined in the construction line editing command.



The next step is creating the line of the left-hand-side edge of the part on the side view. After completing one line, a new line rubberbanding began automatically.

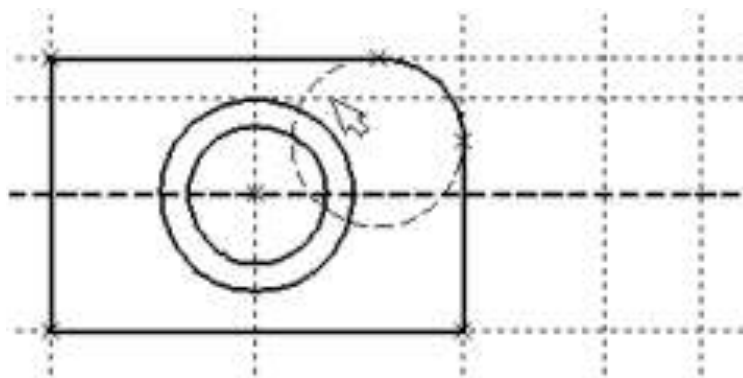
Note that the currently rubberbanded second line is also a dependent of the plate right-hand-side line as the latter is still highlighted. This is not our intention; therefore, press  to start line creation anew. Select the last created line – the one marking the right-hand side of the side view – by clicking  on it. Rubberband the new line up to the approximate location, and fix with , or enter exact value in the property window.



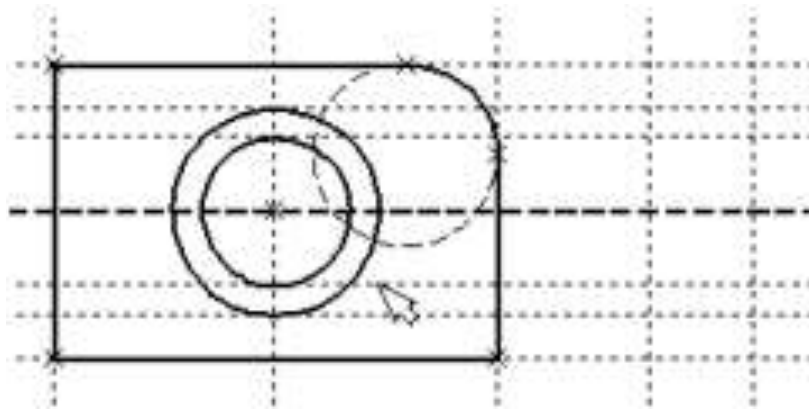
We recommend using specifically the right-hand side of the part main view as the base line, and construct all the rest vertical lines with respect to it. In this way, the line-to-line distances will be positive which is preferable in some situations.

Now let's proceed with constructing the projection of the conical hole. First, let's create horizontal lines tangent to the top and bottom of the inner and outer circles of the hole. These lines will be used as guides for the side view of the hole.



Press  once to restart line creation, move the cursor to the horizontal symmetry line, and select it by pressing  or typing <L>. The line gets highlighted. Rubberband the new line by moving cursor to the outer circle and type <C>. The line is created parallel to the horizontal symmetry line and tangent to the circle.

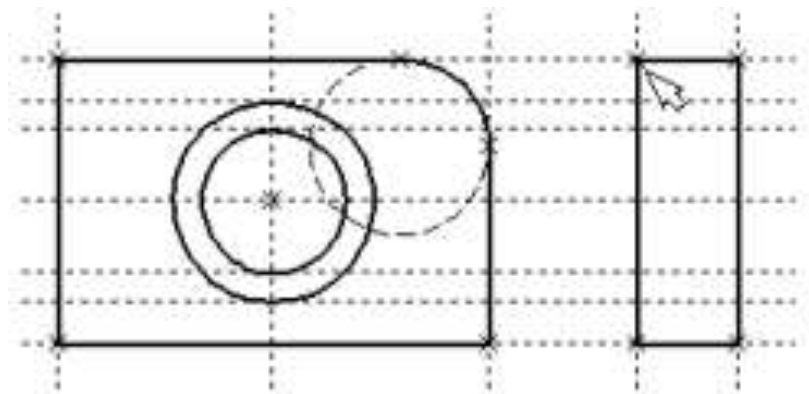


Repeat the same sequence of actions three more times: for the top tangency with the inner circle, and the two bottom tangencies.

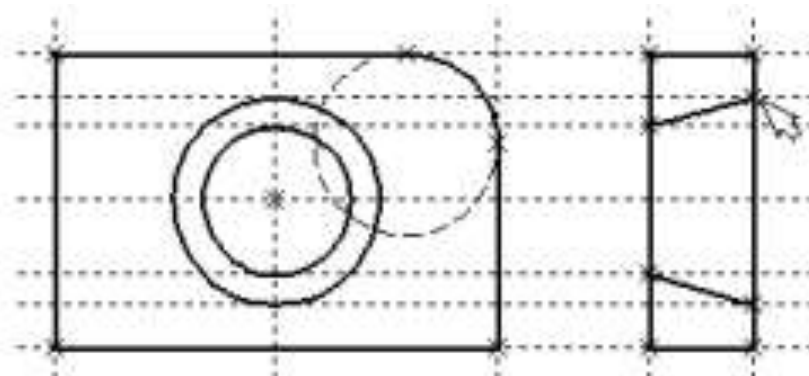


Now we have the necessary guides for applying graphics on the side view.


Call **“G: Create Graphic Line”** command and apply solid lines between the four corners of the plate side view. Simply move the cursor from corner to corner clicking  on each corner node, and then quit with .



Next, apply the two lines representing the conical hole. The view is now almost complete, with only the hatch yet to be created.




The hatch is created by “**H: Create Hatch**” command. Use

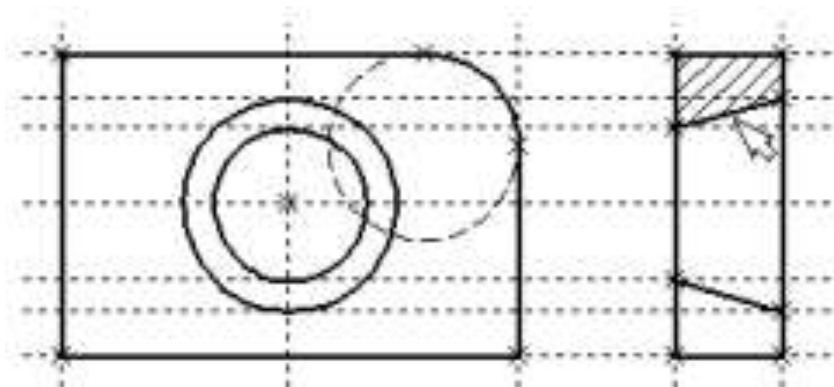
Keyboard	Textual Menu	Icon
<H>	«Draw Hatch»	


Set the following option, unless on by default:

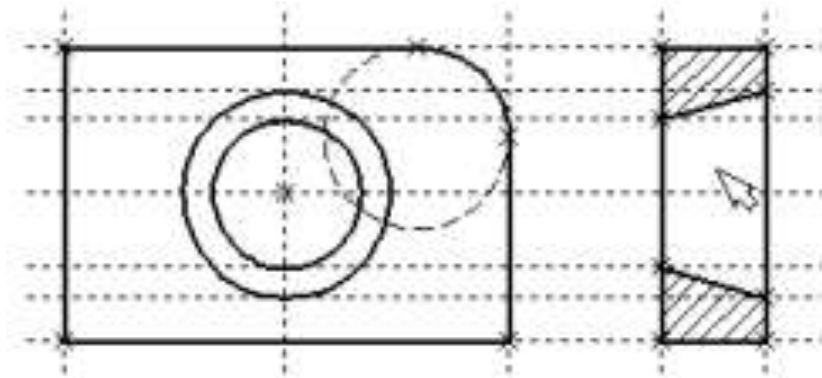
	<A>	Automatic Contour search mode
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Then move the cursor to the top portion of the side view, place within the area to be hatched, and press . The top contour gets highlighted. Next, type <P> to invoke the hatch area parameters dialog. Specify the type and scale factor of the hatch.

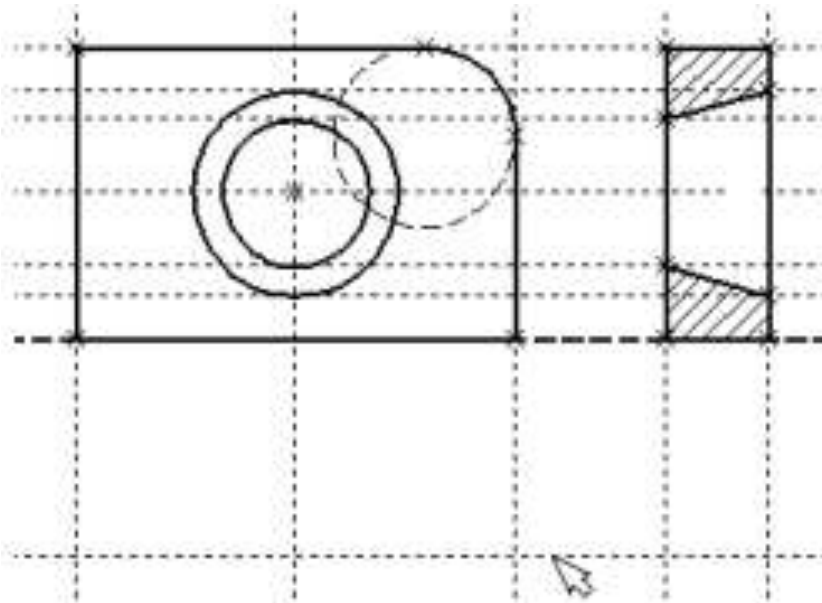
Pressing  in the automenu completes hatching of the selected area.



Repeat the same actions to hatch the bottom portion of the plate. It is also possible to create a single hatch consisting of two contours, instead of creating two separate hatches. To do so, one could select the second contour right after the first one, and then press <End> key or pick  icon in the automenu.





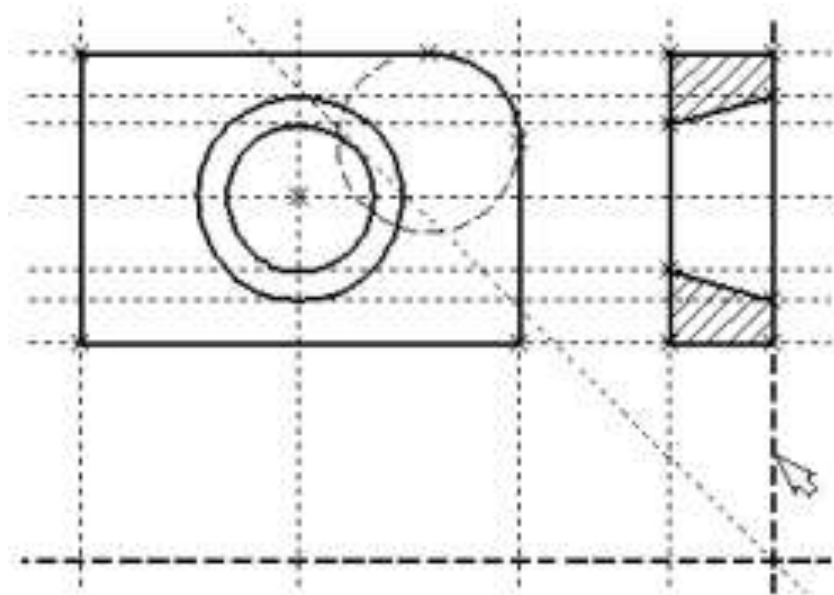
Once the hatch is created, proceed with the plan view. Call the line creation command **“L: Construct Line”**. Select the bottom line of the main view in order to create dependency of the plan view on the main view. Rubberband the new line to a location below the main view and fix with . Then quit next parallel line creation with .



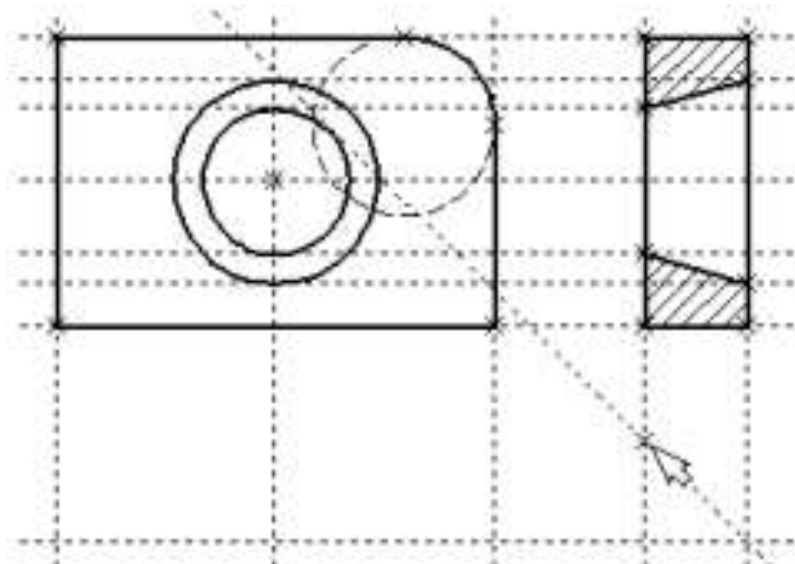
Let's try creating the plan view in such a way that modifications of other views propagated on the plan view via the established dependencies. The simplest way to create a dependency in projective drawing is creating a slanted line at 45-degree angle to the side lines of the side and plan view. The rest of auxiliary construction is done with respect to this slanted line.

Let's again use the symmetry line creation functionality, this time with a slanted symmetry line in mind. Since the lines of the side and the plan views are orthogonal, the resulting symmetry line will pass at the


intended 45-degree angle. Call the option . Point at the right-hand-side line of the side view and select with  or <L>. The line will get highlighted. Next, select the bottom line of the plan view by same means. A new line will be created passing through the intersection of the two selected lines at 45 degree to each.



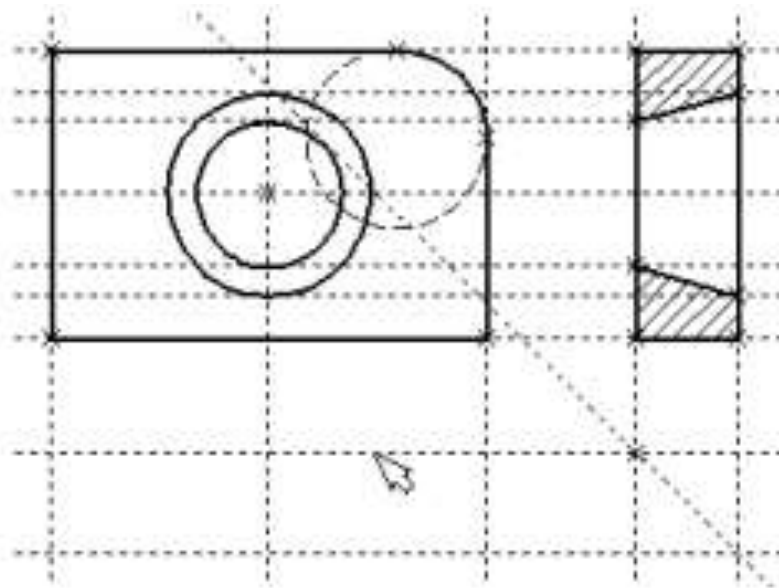
Let's create all necessary nodes at intersections while within the line creation command. The relevant intersection points are those on the right-hand-side line of the side view and the newly created slanted line. To create a node, place the cursor at an intersection and press the <Space> bar.



Another way of creating nodes is using command “N: Construct Node” via:

Keyboard	Textual Menu	Icon
<N>	«Construct Node»	

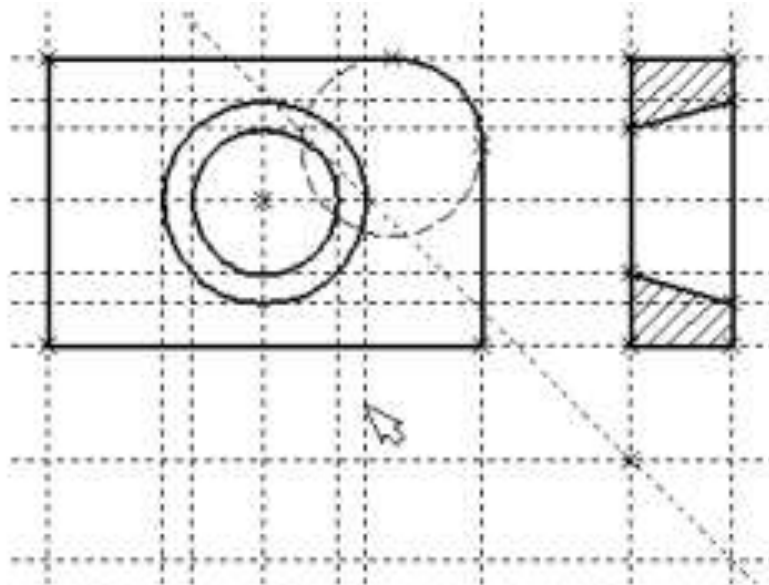
You should still be within the command “L: Construct Line”. Point the cursor at and select the bottom line of the plan view. This way we can create a line parallel to the bottom-side one. Now, move the cursor to the newly created node and type <N>. This creates a line parallel to the selected one and passing through the specified node. Thus, the top and the side view become parametrically related.



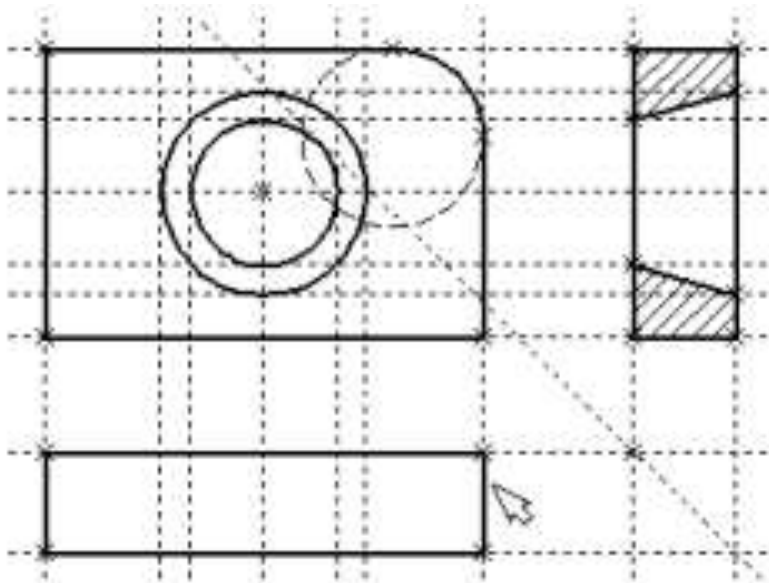
To witness this, call the construction line editing command “EC: Edit Construction”. Try changing location of the left-hand-side line of the side view. To do so, select it, move and fix in the new position. Note now that the corresponding line on the plan view moves accordingly.

Construction of the conical hole on the plan view follows the same steps as on the side view. Select a vertical line while in the construction line creation command, and create four parallel lines tangent to the two circles.

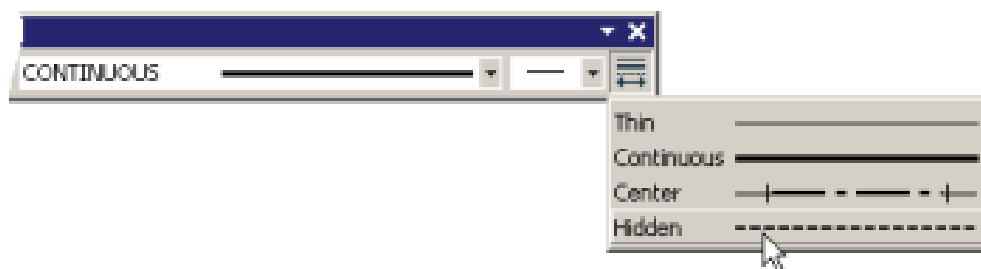




Now one can draw all graphic lines on the plan view. Use the command “**G: Create Graphic Line**” to draw the perimeter of the plan view.

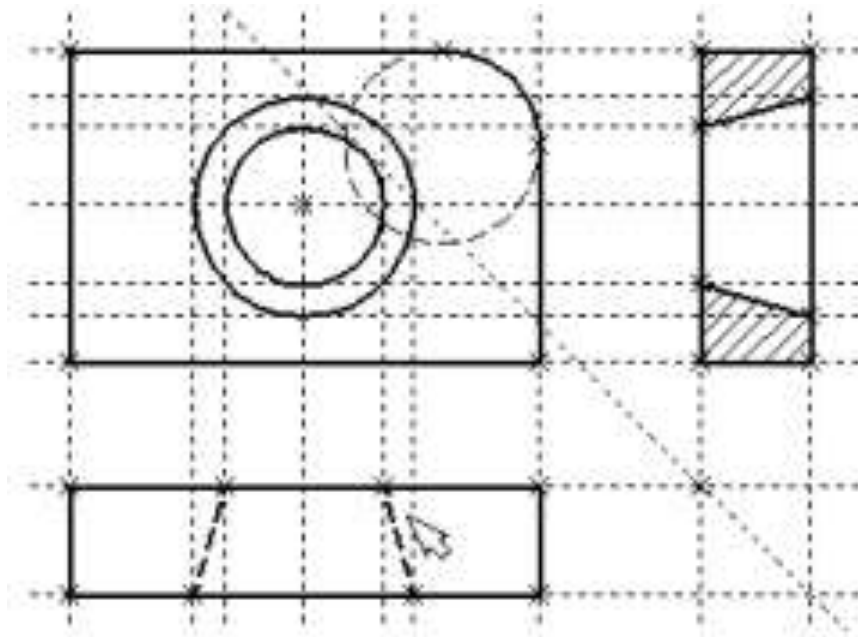


Next step is to apply the two dashed lines corresponding to the conical hole. Set the “**HIDDEN**” line type in the system toolbar.







Then create the two dashed lines representing the conical hole.





Now, let's create centerlines. Call “**AX: Create Axis**” command:

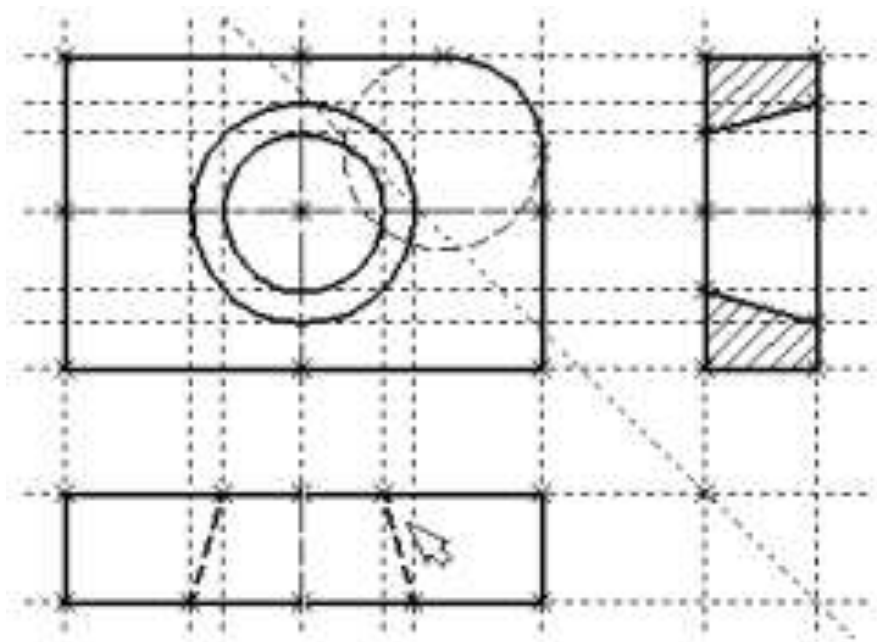
Keyboard	Textual Menu	Icon
<b>&lt;AX&gt;</b>	<b>«Draw Axis»</b>	

Set the automenu option:



	<b>&lt;I&gt;</b>	Create Axis of two Graphics lines
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Use  to select the left and then the right-hand side of the elevation view. Push the  automenu button.

This creates a centerline on the elevation view. Similarly, create a horizontal centerline on the elevation view and those on the side and plan views.



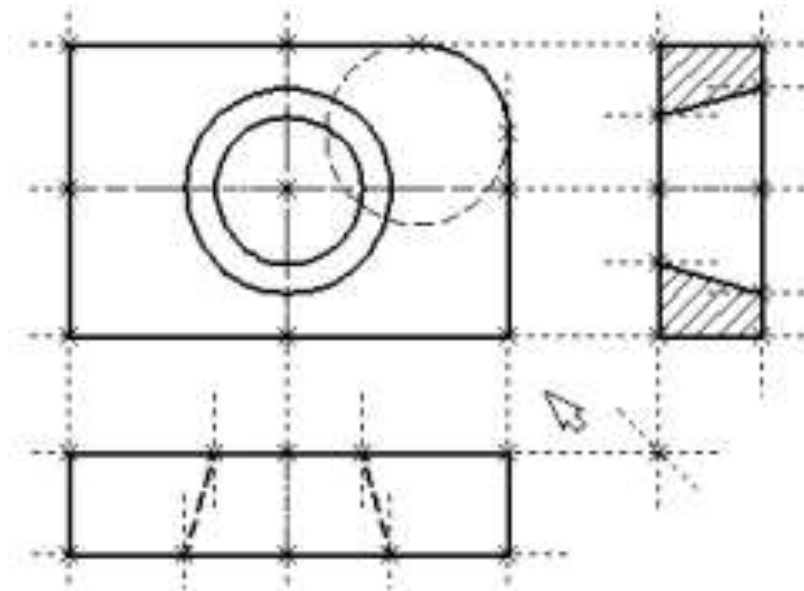
One could notice that all construction lines created so far were infinite. For convenience, an option is provided in the command **“EC: Edit Construction”** for trimming construction lines at outermost nodes. This works as follows:

1. Call **“EC: Edit Construction”** command.
2. Selecting one particular line and typing **<T>** or pushing  trims this selected line only.
3. Using option  trims all lines.
4. If you want to revert to the infinite line setting, call the command **“ST: Set Model Status”**:

Keyboard	Textual Menu	Icon
<b>&lt;ST&gt;</b>	<b>«Customize Status...»</b>	


Go to the parameter **«Screen|Construction Lines|Length»** and set it to “Default infinite”. Alternatively, enter the command **“EC: Edit Construction”**, select desired lines, type **<P>** and specify an appropriate setting.




The diagram shows a drawing with construction lines trimmed. It appears less crowded, although all necessary construction entities are present. By default, construction lines are not output to the printer or plotter, regardless of their length.



Next, let's create the necessary dimensions on the drawing as follows:





1. First, let's create linear dimensions. Call the command **“D: Create Dimension”**:

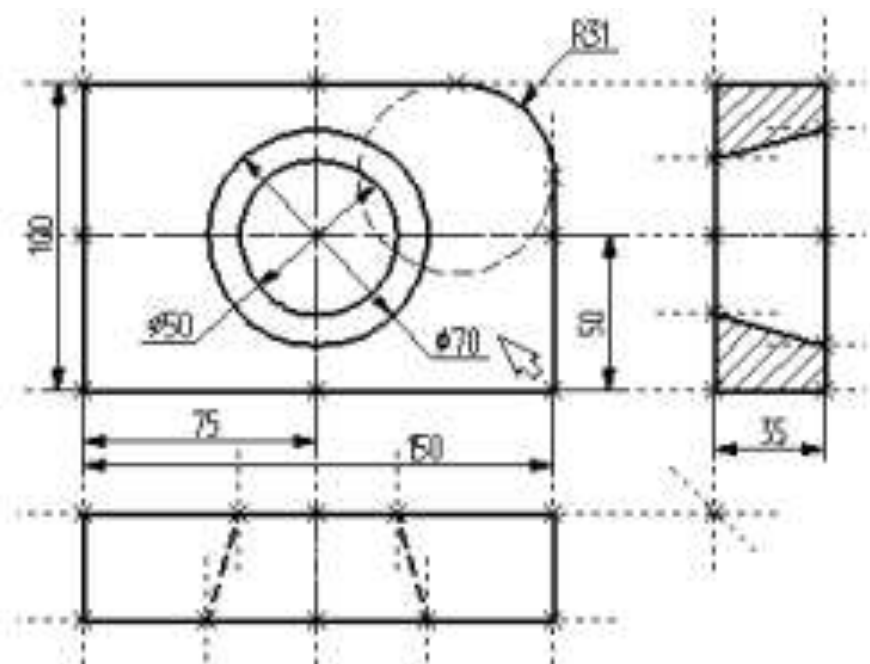
Keyboard	Textual Menu	Icon
<b>&lt;D&gt;</b>	<b>«Draw Dimension»</b>	

One can select any pair of construction lines or graphic lines to create a linear or angular dimension. Select the two outermost lines on the main view by . This starts rubberbanding of a dimension following the cursor motion. To change any dimension parameter, type **<P>** or push the  button in the automenu. The dimension parameters dialog box will appear on screen. Specify the desired parameters, close the dialog, and fix dimension placement with . To change the size of the dimension string


font, use the command **“ST: Set Model Status”**, the tab **“Font”**. The font parameters can be specified on this tab for the elements that did not have such parameters set originally.

2. Repeat the steps of the item 1 for the rest of the linear dimensions.

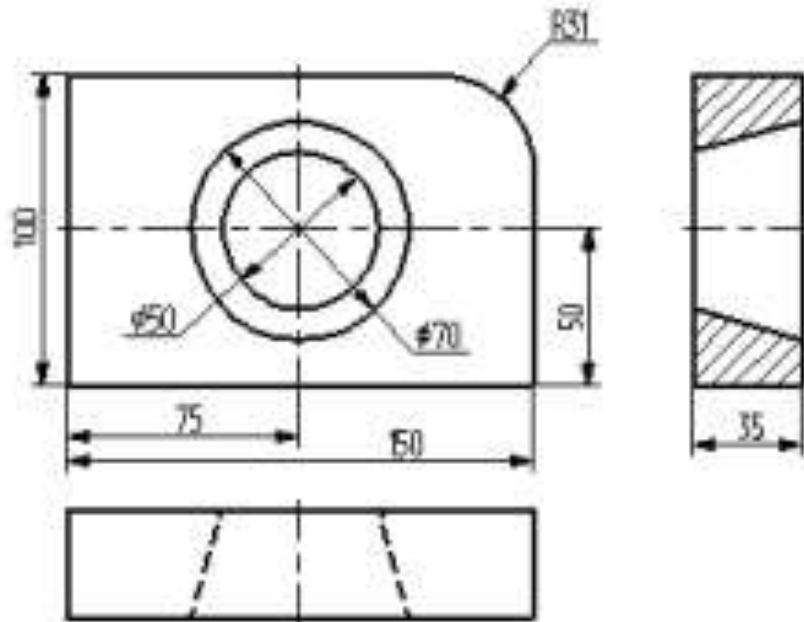
3. Diameter and radius dimension creation is also straightforward. While in **“D: Create Dimension”** command, move the cursor to a circle to be dimensioned, and type **<C>** or click . The circle gets selected, and a dimension begins rubberbanding after the cursor. Switch between the radius and diameter dimension types by typing **<R>** and **<D>** or picking  and  buttons of the automenu as appropriate. Typing **<M>** loops through the possible witness/leader line configurations for the entity to be dimensioned. The **<Tab>** key handles the direction of the dimension leader line jog. Point the cursor at the desired location and press . The newly created dimension will be displayed on the screen. Repeat this procedure to dimension all circles.



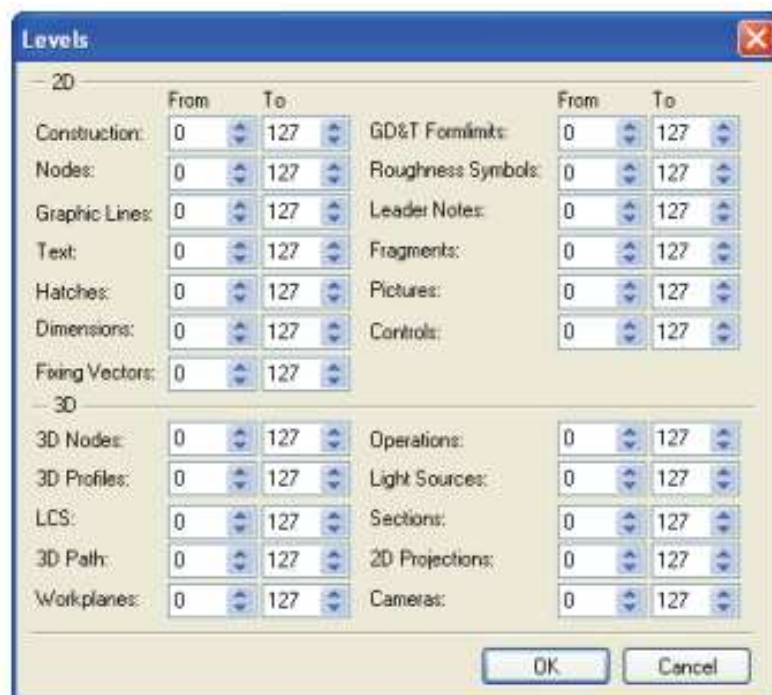
4. After finishing construction of all major elements, one can hide all construction entities using the command **“SH: Set Levels”**:

Keyboard	Textual Menu	Icon
<SH>	«Customize Levels...»	

This command controls visibility of various elements. An element visibility depends on the “level” at which it is residing.



After calling the command, a dialog box appears on screen that allows setting a range of visible levels for each element type of the model.




Think of levels as transparent films with images drawn on them. The complete drawing consists of all of them overlapped. The system permits making one or more levels invisible, displaying only intended ones. A drawing may consist from up to 255 levels enumerated from -126 to 127. All elements in T-FLEX CAD are automatically created on the level “0”. One can re-assign any element to another level at any time. In our example, we did not change levels of any element; therefore, all created elements fell in the level “0”.

As appears on the diagram above, all elements are visible by default whose level is in the range from 0 to 127. Setting the low limit of the visible range to 1 for construction lines and nodes hides the construction lines and nodes, because they reside on the level 0 which is not within the new range.


A simpler way to hide construction lines and nodes is to use a dedicated command. This command hides or shows all construction entities in the current window. It is preferable in the situations when hiding construction should not affect the document data, rather, the current window only.

It is thus possible to open the same document in several windows, and have construction entities displayed in some windows, and hidden in others.


Call the command via:

Keyboard	Textual Menu	Icon
<Ctrl><Shift><C>	«View Hide Construction»	


5. Let's make a line of text containing the name of the drawing using command “**TE: Create Text**”. Call the command via:



Keyboard	Textual Menu	Icon
<TE>	«Draw Text»	

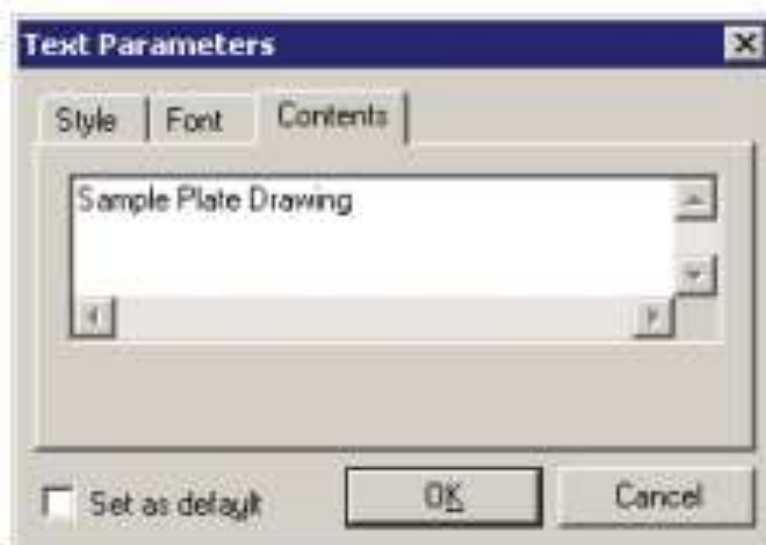
In the automenu of the command, turn on the option:

	<D>	Create string text
---	-----	--------------------

A text can be “snapped” to any construction entity on the drawing in order to have it move together with the drawing elements being modified.


Move cursor to the intersection of the vertical centerline and the top line on the main view. Type <N> in order to snap the text to the node at the intersection. Move the cursor to the text placement point and press . The text editor window appears on screen. Type a line of text “**Sample Plate Drawing**” and push [OK] button.

Should the text not be placed as intended, this can be corrected easily. Quit the text creation command. Point and click  at the text. This automatically starts the editing command “**ET: Edit Text**”. The selected text starts moving after the cursor. Locate it as desired and click .













To explicitly call the text editing command, use:

Keyboard	Textual Menu	Icon
<b>&lt;ET&gt;</b>	<b>«Edit Draw Text»</b>	

In this case, select the text to be edited after launching the command.

There is another way of creating a text, which is typing it directly in the drawing area. To do so, enter the **“TE: Create Text”** command and set the option **<T>** – “Create paragraph text” (icon ). Move the cursor to the intended location of the text and press . A rectangle starts rubberbanding on screen that defines the text box. Define the intended area and click , then push the  icon. A blinking cursor will appear in the box. Make sure of the correct input locale and enter the intended text. Then push the  icon or **<F5>** key.

The drawing is now finished. One can try moving construction lines using construction editing command. When editing, fix line new placement by either using  or specifying exact line location in the property window or parameters dialog (the latter accessible via the  pick). Note that the whole drawing, including dimensions, adequately responds to modifications. Changing diameters of the conical hole instantly reflects on the two other views. Hatches also adjust to their defining contours. Now one can easily witness the powerful capabilities brought in by the parametric technology.

From now, we will assign variables and expressions to the various drawing elements. Select the left-hand-side line on the main view by clicking .



The line will get highlighted, along with the one it is dependent on by construction. Line editing command will automatically activate as well. The two parameters are displayed in the property window. The first one is the original distance, and the second is the current value according to the cursor position.



Since the line was originally created as parallel to the left-hand side of the plate, the displayed distance is the distance between the right and the left-hand side of the plate. Instead of a specific value, one can input a variable. Type a variable name “W” instead of the value and press **<Enter>** or **[OK]**.




A new dialog window will appear requesting a confirmation for the new variable to be created.


Please note that the variable naming is case-sensitive. A variable “W” is not the same as “w”.


The created variable «w» and the value assigned for this variable can be seen in the window “Variables” located, by default, under the properties window. Point with a cursor at the number in the column “Expression”, press to enter the edit mode and specify the value for the variable, for example, «170». The line will move to a different location corresponding to the new value of the plate width.




The same operations can be carried out in the dialog window of the command “**V: Edit Variables**”:

Keyboard	Textual Menu	Icon
<b>&lt;V&gt;</b>	<b>«Parameters Variables»</b>	

Similarly, define a variable “H” as the distance from the base line to the top side of the main view. Select the line on the drawing by clicking  and enter the variable name in the property window. Now there will be already two variables in the window “Variables”, and you can, by modifying their values, observe the change in the drawing.

Try making an expression. In the window “Variables” place the cursor in the field “Expression” of the variable «H» and press  to enter the edit mode. Specify the expression «W/2» instead of the numerical value. This means that the value of “H” will be equal to the half of “W”. From now on, changing just the value of “W” will automatically reflect on the value of “H”.

Next, let's assign an "R" variable to the radius of the circle defining the fillet at the upper-right corner of the plate. Select the circle on the drawing by . In the property window specify the radius as "R" variable. After confirming its creation, in the window "Variables" set the variable to the following expression:  $W < 100 ? 0 : 6$

This expression means that "R" equals 0 when "W" is less than 100, and equals 6 otherwise.

Let's briefly explain the syntax of the expression. Its members are described as follows.

$<$  - is the "less than" sign

$?$  - means "then", "in such a case"

$:$  - means "else", "otherwise"

The complete expression is written as

$R = W < 100 ? 0 : 6$

The value of "R" equals 0, if " $W < 100$ ", and equals 6 for any other value of "W". Therefore, there are only two possible values of "R" - either "0" or "6".

Check this on your drawing. Try setting "W" values greater or less than 100, and watch what's happening.

Note that when the radius of the fillet equals 0, then the radial dimension automatically disappears. The system does it for the user.

Therefore, one can create a variety of relations between variables, including quite complex ones, using just a few basic terms. You will get to know all capabilities of the variables functionality in later lab classes.

### **3. THE ORDER OF LAB WORK**

1. Get acquainted with the process of creating a parametric drawing.
2. Consistently perform all the steps for parametric drawing.
3. Create a parametric drawing of the part according to the individual task (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.
4. Conclusion.
5. Answer the questions from the checklist.

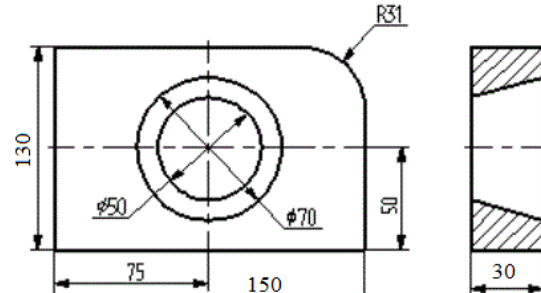
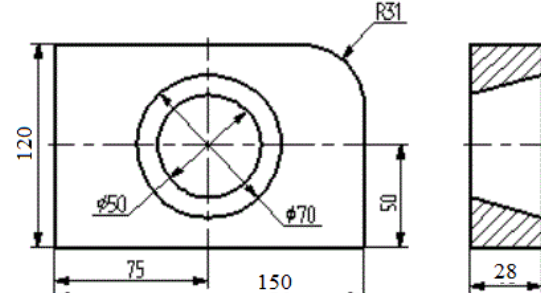
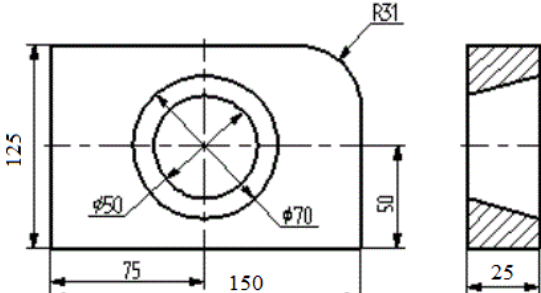
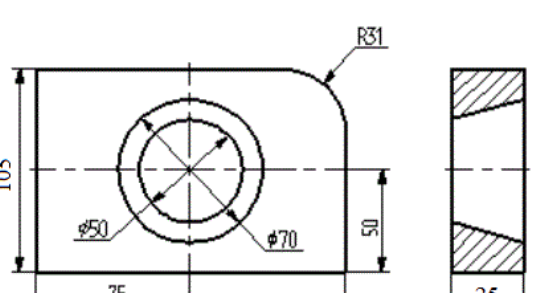
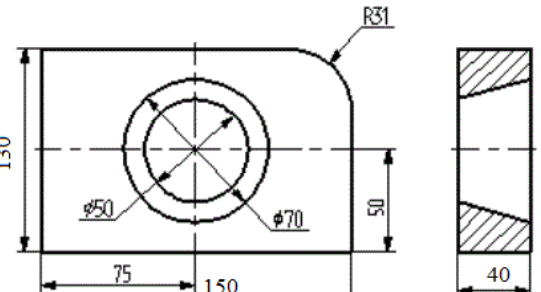
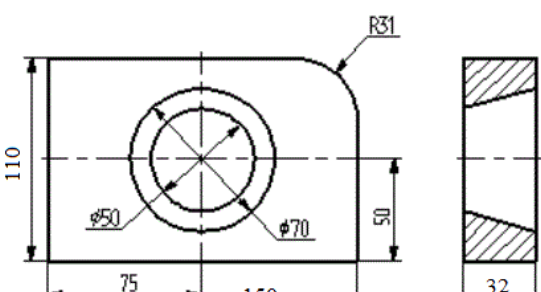
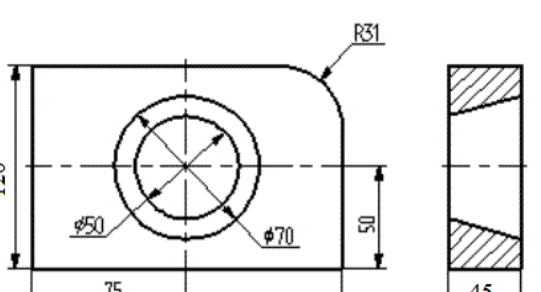
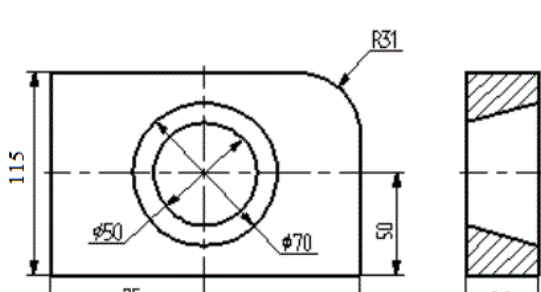
### **5. CHECKLIST**

1. Algorithm for creating a parametric drawing.
2. What is the basis of parameterization?
3. What are the advantages of parametric drawing?
4. Parametric design Software and its application.
5. How to create the main (elevation) view of the part?
6. How to edit plot lines and image lines in the parametric drawing?
7. How are image lines applied to parametric drawings?
8. How are construction lines applied to drawings?
9. Algorithm for entering text on the parametric drawings.
10. Algorithm for marking dimensions on the parametric drawing.

# Task for lab work № 2

Create a parametric drawing.

Table 1

№ var.	Drawing	№ var.	Drawing
1		5	
2		6	
3		7	
4		8	

## 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 Aplicacoes <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.youtube.com/watch?v=zZjBbNAY790>

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

## **CONTENTS**

1. Goal
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  4. Report Structure
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