

ЛІТЕРАТУРА



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 a 3D model mode

STUDY GUIDES № 4

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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Lab work objective: Mastering the basic commands and principles in creating three-dimensional models using simple examples, steps in the modeling process and acquisition of practical skills for their implementation.

1. GOALS

Almost all machinery parts are now designed with the obligatory application of the achievements of modern computer technologies, including such of them as 3D graphics. Technology such as 3D modeling makes it possible to create images of almost any part in such a way that they are as realistic as possible. 3D modeling makes it possible to visualize those parts that do not yet exist in reality and that are still in development.

In order to visually create three-dimensional models of various parts, based on which various mathematical calculations are made and a visual assessment is made, specialized software packages are used. Their creators were able to ensure that any change in at least one of the three coordinates of the 3D model would automatically lead to corresponding changes in the rest of the design and engineering documentation.

As practice shows, now three-dimensional modeling is successfully used in many areas of modern technology, but it gives the best results when the layout of various plant equipment is designed, high-complexity drawings are created, and various complex structures

are designed. In addition, 3D modeling is almost indispensable when it is required to ensure the most accurate (that is, with minimal gaps) mating of various parts and components, and it is also necessary to ensure that the most stringent technical requirements are met.

Modern computer technologies have at their disposal such tools for three-dimensional modeling, which allow simulating the functioning of robots and generating tool paths. Thanks to this, 3D modeling has already become one of the integral parts of modern production.

It should be noted that the overwhelming majority of modern three-dimensional modeling systems have tools for automatic analysis of such physical characteristics as moments of inertia and weight, they have a variety of tools that allow solving geometric problems that arise when creating complex mates, as well as with various interpretations.

3D modeling is especially useful in cases where you need to repeatedly edit a three-dimensional object at almost all stages of its design. The fact is that in three-dimensional modeling systems there is an automatic connection between those data that have different geometric types of images.

Three-dimensional models are divided into three types according to their functional purpose: wireframe, surface and solid.

The simplest of these are wireframes, which are essentially sets of lines and arc segments interconnected in three-dimensional space. There is no relationship between them, they describe only the edges of objects and their use is very limited.

Surface models are sets of surfaces that are connected in space in such a way that they form a three-dimensional object. Unlike wireframes, they carry information about the silhouette and contours of surfaces. Surface models are widely used in computerized manufacturing systems.

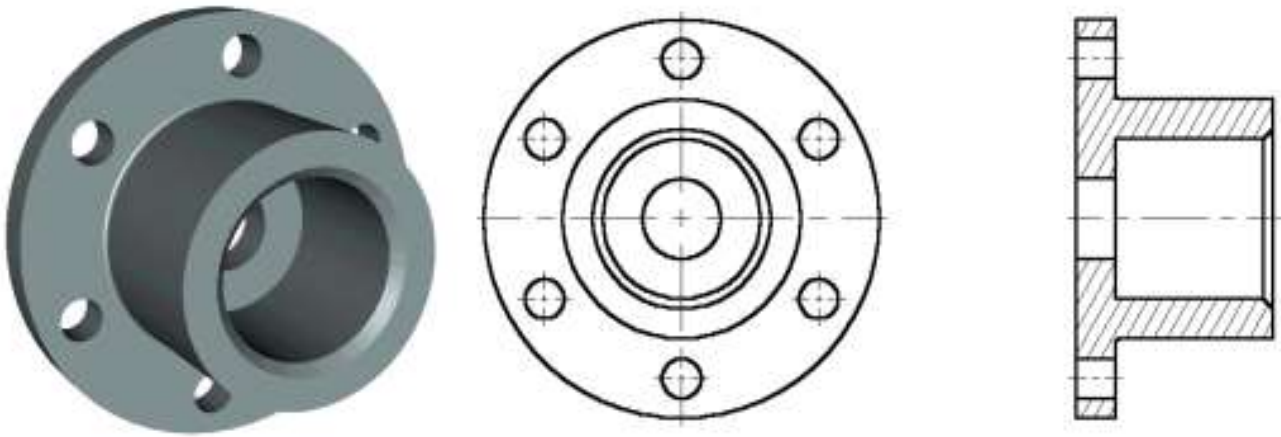
Solid models are the most advanced, since they carry information not only about edges and planes, but also about the internal volume. They can be used not only for visualization, but also for carrying out various calculations.

2. THEORETICAL INFORMATION

T-FLEX CAD system allows various approaches to creating a 3D model. The main method is creating most of constructions directly in the 3D window. In another approach, a 3D model is created based on already prepared 2D drawings or auxiliary 2D constructions. This construction may further be used for generating drawings by projecting the model in all necessary views and making cuts and sections. These 2D objects can further be furnished with necessary dimensions and other drawing attributes.

2.1. Main Approach to 3D Model Creation

This approach makes possible creating a three-dimensional model without using the 2D window. The diagram below shows a sample part that we will be modeling. We will create the three-dimensional model first, and then automatically generate views and a section.



The model file is located in the folder “...\Examples\Documentation examples. Introductory course\Detail with drawing detailing”.

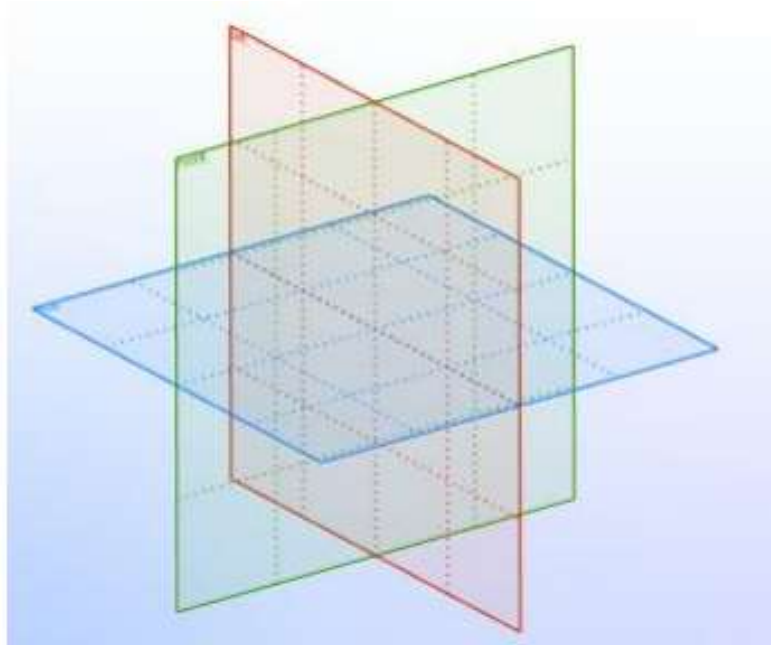
The model creation will be done in several steps. First, one has to create initial auxiliary elements. Based on those, you will be able to create a first draft of the considered part – without the holes and chamfer. To do this, use the “Rotation” operation. At the next step, add six holes in the part’s body. Holes can be created by different means. We will discuss several methods here to give you a more complete idea about 3D model creation methods. Then, to get the final version of the part, you will just need to apply a chamfer via the “Blend” command.




2.2. Creating Auxiliary Elements

Let’s start from scratch. Create new document.


To begin modeling, one can use an appropriate prototype file among available in the T-FLEX CAD system (File > New 3D Model).

You have just created a new document that already has three standard workplanes – Front, Right and Bottom. You can see that the 3D window with these elements opens automatically within the system window.



For convenience of modeling in 3D window, spinning, panning and zooming of the scene view are available at all times, even when commands are active. Note: the view manipulations while in certain commands require a specific option to be set. These commands will be described below. To spin the scene press and hold  and move the cursor as desired. One can also use the two arrow key pairs and the pair <Page Up> and <Page Down> for spinning around the three respective axes. Zooming can be done at any moment by scrolling the mouse wheel button (as in IntelliMouse design) or via the provided commands on the “View” toolbar located along the right border of the window. Panning and zooming of the view can also be controlled via <Ctrl>+  and <Shift>+  combinations respectively.

As we move the cursor close to a workplane, the latter gets highlighted. T-FLEX CAD provides pre-highlighting on cursor over for all elements in the 3D window depending on the selection filter settings.

The selection filter icons are located on the system toolbar. To pick an element among the icons, simply press .




Let's select the "Right" workplane. One can see now that 2D drawing commands become accessible. We will further use these commands for creating auxiliary elements in the 3D window.

What auxiliary elements do we need? The first body to be created is a body of revolution. To create it, we will need the contour and the axis of rotation for this contour.


Drawing in 3D window can be performed using any of the tools available for 2D drawing. Thus, one can use the sketching environment for rapid creation of non-parametric models. Accordingly, parametric modeling approach requires creation of construction lines first, then application of graphic lines. The system automatically creates 3D profiles based on drawn graphic lines, to be later used in 3D operations.


To start drawing, call the command, Icon Ribbon

| Icon | Ribbon |
|---|-------------------------|
|  | Draw → Construct → Line |
| Keyboard | Textual Menu |
| <L> | Construct > Line |

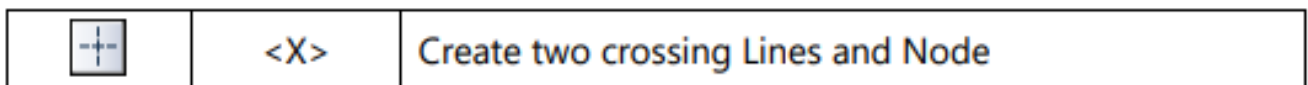
After activating the workplane by any 2D drawing command, the main toolbar is switched to the mode "Workplane". In this mode a set of icons for working on the active workplane is displayed on the toolbar.




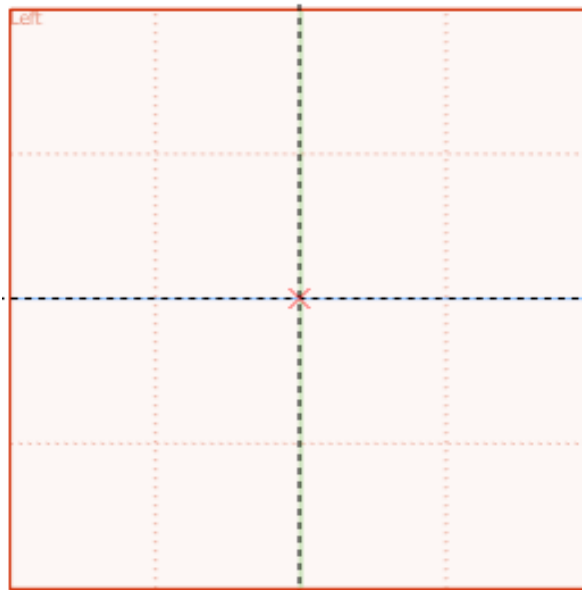
For convenience, the active workplane is reoriented parallel to the screen. However, it can be spun in any desired way, after setting the option .

It is also possible to open the 2D window and continue drawing in 2D mode. Once the 2D window is closed, all changes will be reflected on the 3D scene. The 2D window can be opened or closed using the  icon.

Let's construct two base lines, a vertical and a horizontal one. To do so, push the following button in the automenu:






Point cursor at the center of the workplane and click .




Thus, we have created two orthogonal lines and a node. These will be used as a reference for all further construction. Press to quit the last active command. We are now in the parallel line creation mode. (This mode is set by default in the command “**L: Construct Line**”).

Just like in 2D drawing, we first need to create a framework of thin lines, and then apply graphic lines along the intended segments.

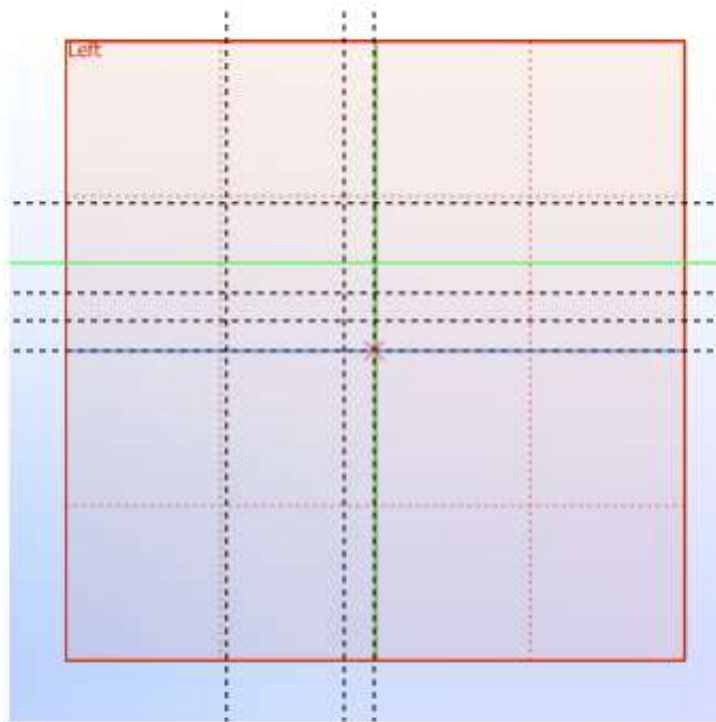
To construct a parallel line, select a reference line first, that will be used as the “parallel” reference for the new one. Similarly to 2D operation, the 3D drawing mode also supports object snapping. Thus, to select a line, move the cursor to the vertical line. The cursor will gain a glyph like this, . Click  and move the cursor leftwards. One can see the cursor changing appearance again – it is now rubberbanding a line parallel to the selected.

To fix the line at an arbitrary location, just click . However, we would like to specify the distance of 100 mm, by typing it in the property window. (In future, this parameter value can be changed at any time as desired.) Press **<Enter>** to complete the line creation.





We have just created a line parallel to the selected, at the distance of 100 mm from it. As one can see, rubberbanding resumes at this point. The system is still in the mode of parallel line creation with respect to the original selection. Type a new value of 20 mm right away, followed by **<Enter>**, thus creating another line. That’s all with the parallel line creation for now. Click  or **<Esc>** key for quitting the parallel line creation mode.


Next, create four more lines in the similar way, parallel to the horizontal base line. Use the distances equal to 20, 40, 60 and 100 mm respectively. The result should look like shown on the following diagram.

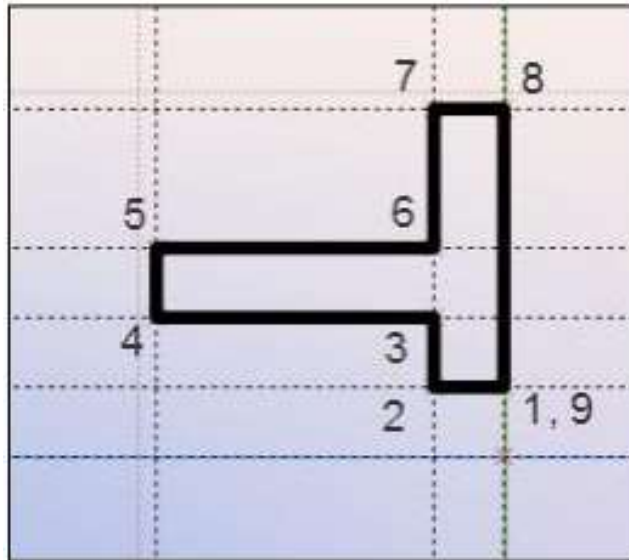



Now, apply graphic lines along the construction lines appropriately. Call the graphic line creation command:


| Icon | Ribbon |
|---|----------------------------|
|  | Draw → Draw → Graphic Line |
| Keyboard | Textual Menu |
| <G> | Draw > Graphic Line |

Graphic lines snap to the construction entities – lines, circles, nodes, etc., as well as to construction line intersections. In the latter case, a 2D node is automatically created at intersections snapped to by graphic lines. Straight graphic line segments maintain snapping at their end points. To have an end of a graphic line snapped to some entity, move the cursor to the intended location. Let the cursor snap as indicated by an appropriate glyph, and click .

In this example, we use snapping to construction line intersections and to 2D nodes. Draw the profile as shown on the following diagram. To do so, click with  at the appropriate points in the order marked on the diagram.

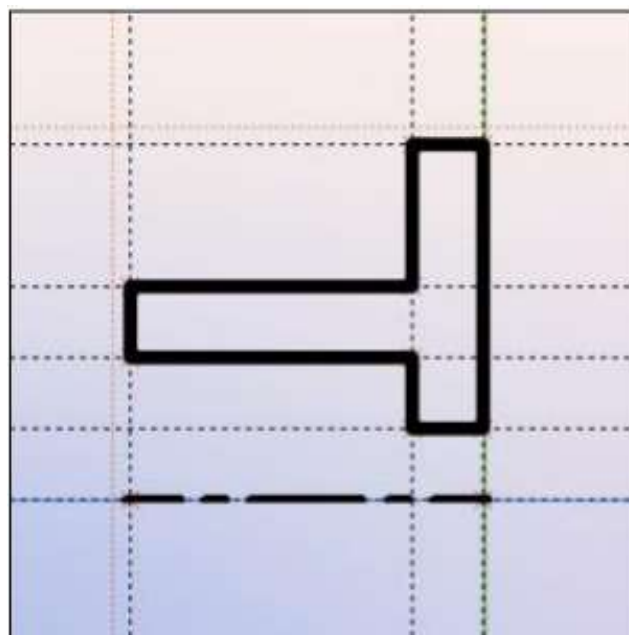


To quit continuous line creation mode, click . Note that the graphic line creation command is still active.

Change the line type in order to draw the axis. To do so, pick the icon  on the system toolbar and select CENTER type from the pull-down list.




Draw the centerline as shown on the following diagram.




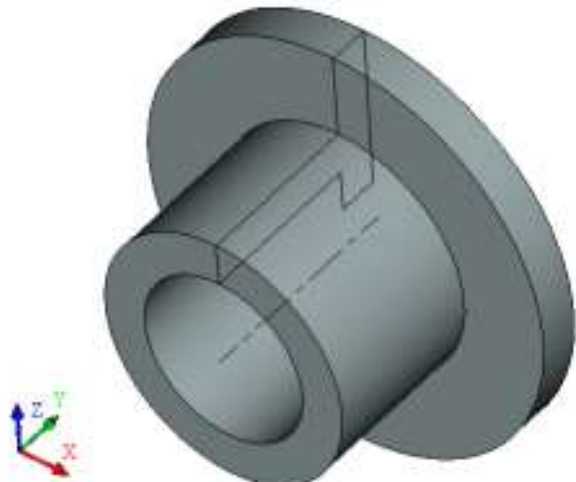
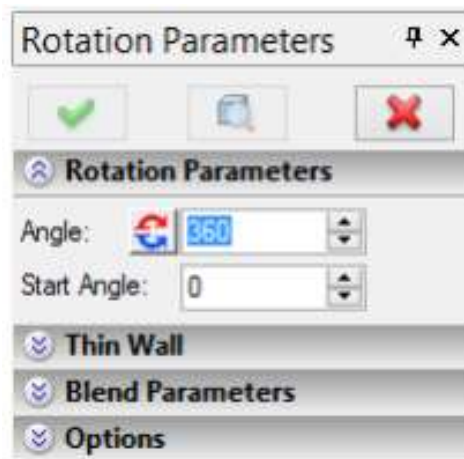
2.3. Creating Rotation Operation

One need not quit drawing explicitly in order to create a body of revolution. Simply call the “Rotation” command as follows:

| Icon | Ribbon |
|---|------------------------------|
|  | 3D Model → Create → Rotation |
| Keyboard | Textual Menu |
| <3RO> | Operation > Rotation |

The system automatically determines the rotation axis and the contour among the created lines to be used as a 3D profile. You can preview the result as a wireframe model. In our case, the rotation angle is 360 degrees. Note that the value of 360 degrees for the rotation angle is set by default in the property window.

Just confirm the operation by pushing the check button , and the body of revolution is created.



2.4. Creating Holes

Next, we need to cut six holes in the part.


Holes can be created by different methods. The fastest and simplest one is using the dedicated “**Hole**” operation. The latter allows creating holes of standard shapes in solids, using templates provided in the T-FLEX CAD utility library. In this way, the user only needs to specify the position of the would-be hole within the solid and specify its type and dimensions.

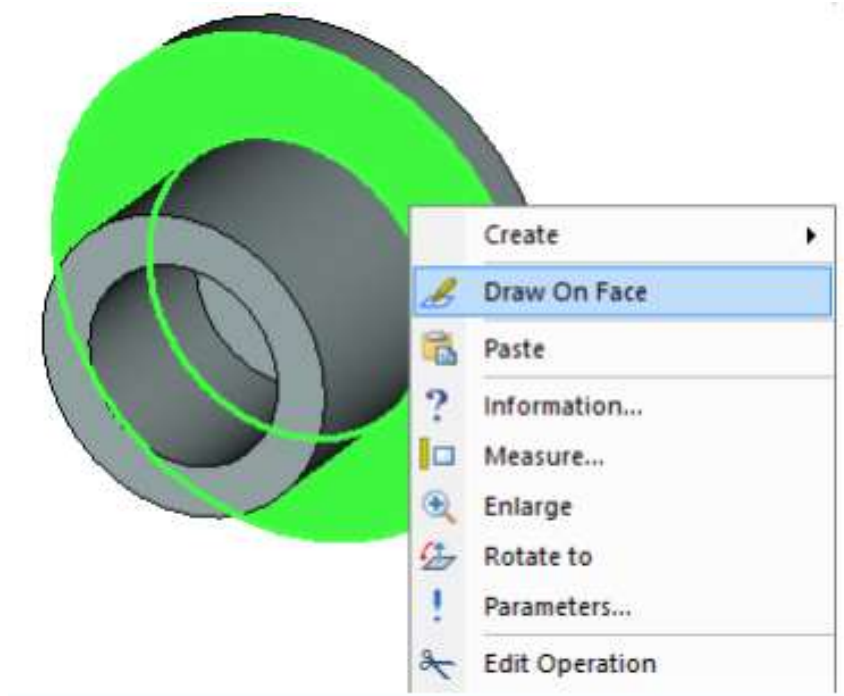
Non-standard holes and slots can also be created without using the dedicated operation. To do this, the user should create an additional solid representing the internal volume of the hole, and then **“subtract”** it from the main body’s volume by a Boolean operation.

Let us consider both techniques.

2.5. Creating Holes Using a Dedicated Command

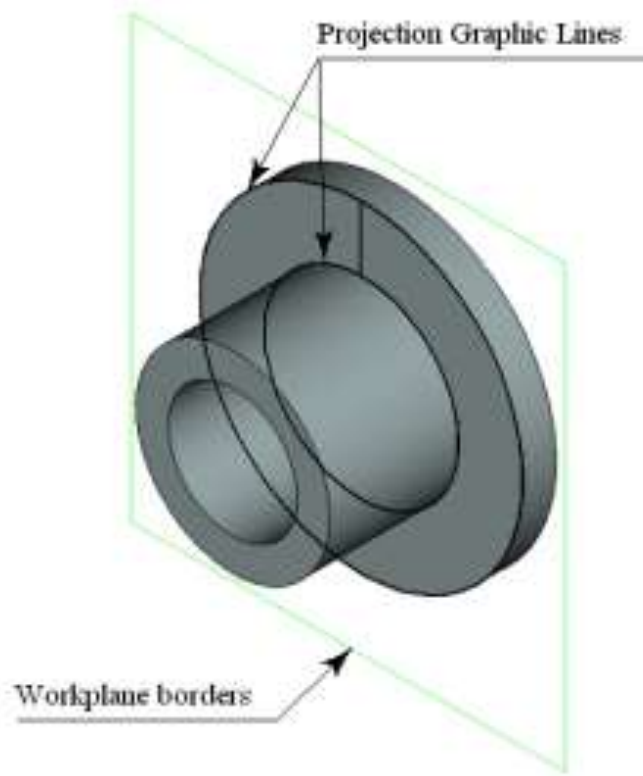
First of all, we will create 3D nodes corresponding to the centers of the would-be holes, on one of the part’s faces. To create them, we will again use the 2D drawing mode for creating auxiliary 2D nodes on the respective face of the part.

To select a face, move the cursor to the desired element of the model so that it pre-highlights. At this point, press  to invoke the context menu and select **“Draw On Face”** item in it (see the diagram).

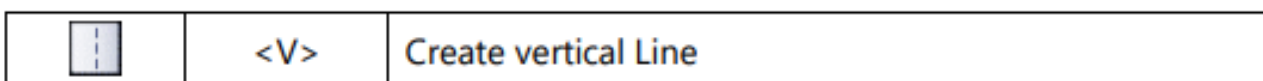



If the desired element is not getting pre-highlighted, make sure the selector is set up for this element type. To change selector’s settings, you can use selector filter icons on the system panel or select a combination of types from the list.

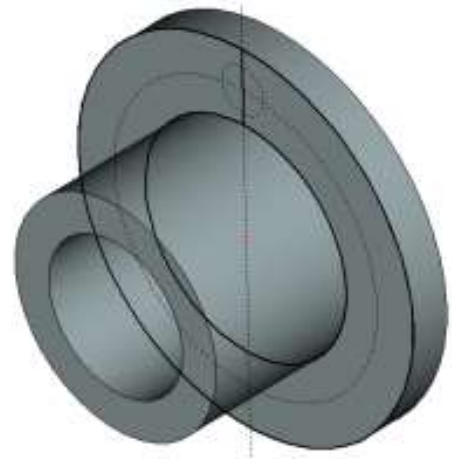
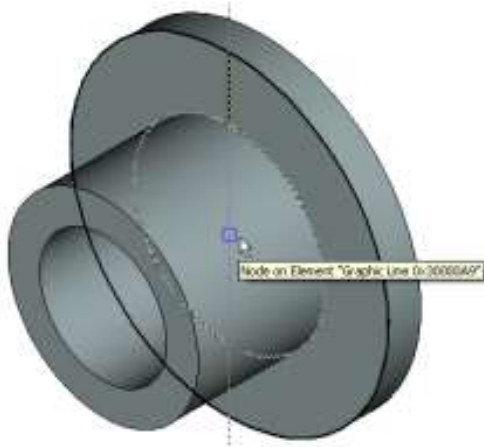
The “**Draw On Face**” command creates a new workplane based on the selected flat face. The selected face is automatically projected on this plane, and the mode of drawing in the 3D window activates. Further construction can use snapping to elements of the face projection.




Call the command “**L: Construct Line**” again. Select the following option in the automenu,



Move the pointer to the center of the circle - the circle center will be pre-highlighted. It is available for snapping to the vertical line. Click . The constructed line will be snapped to the 2D node which was automatically created at the circle's center.




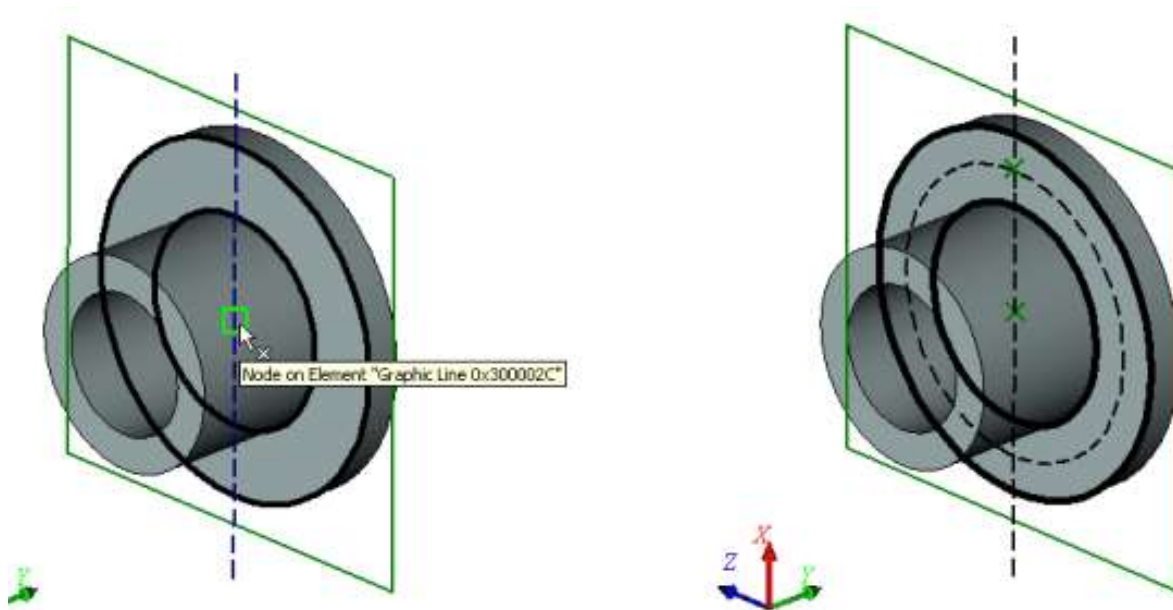
Call the command:

| Icon | Ribbon |
|---|---------------------------|
|  | Draw → Construct → Circle |
| Keyboard | Textual Menu |
| <C> | Construct > Circle |

Point the cursor at the center node to select it as the center of the new circle. Specify the radius of the circle equal to 80 mm by typing in the property window.

Create a 2D node at the intersection of the vertical line and the new circle using the command:

| Icon | Ribbon |
|---|-------------------------|
|  | Draw → Construct → Node |
| Keyboard | Textual Menu |
| <N> | Construct > Node |



Now, one can construct a 3D node based on the created 2D node. To do this, without leaving the drawing on face mode, call the command:

| Icon | Ribbon |
|----------|----------------------------|
| | Draw → Construct → 3D Node |
| Keyboard | Textual Menu |
| <3N> | Construct > 3D Node |


When the 3D node creation command starts, move the pointer to the created 2D node and click . The 2D node will be highlighted, and the option will become accessible in the automenu. Click , and the 3D node will be created. It will be located in the plane of the selected face, while the 2D node will be its projection on that face.

The created 3D node will define the center of one of the six holes. There are two ways to define centers of other holes:


1. Construct five more 2D nodes on the same workplane and create the required 3D nodes based on the 2D ones (similar to the first 3D node creation);

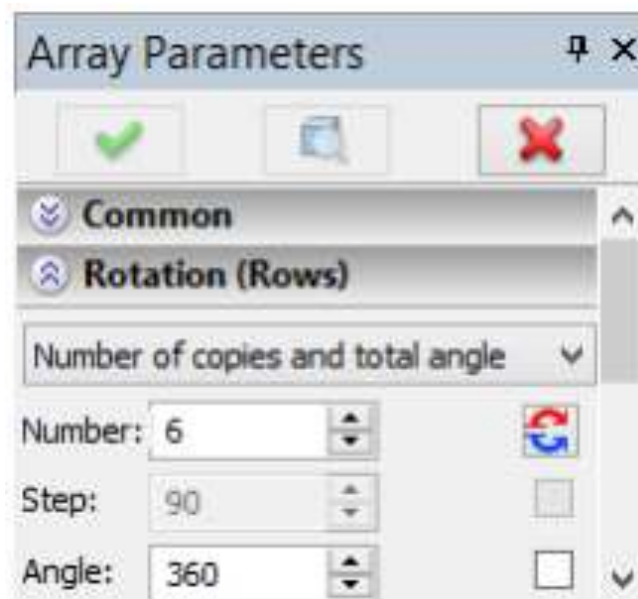
2. Create the required 3D nodes by means of a 3D array based on the first 3D node.

The second method is faster, so we will use it in this case. Call the command **3AR: Create Circular Array**:

| Icon | Ribbon |
|---|------------------------------------|
|  | 3D Model → Create → Circular array |
| Keyboard | Textual Menu |
| <3AR> | Operation > Circular array |

You do not have to explicitly finish the draw on face mode - it will terminate automatically upon launching the 3D array creation command.

In the command's properties window, set the array type as “Array of Construction”. Then, in the 3D window, move the pointer to the created 3D node (the latter shall highlight) and click  to select it. If all was done right, the name of the 3D node selected for copying shall appear in the properties window.



After that, you need to specify the rotation axis of this circular array. To do this, you can use a pair of 3D nodes automatically created when defining the axis for the rotation operation. Select these two 3D nodes one by one.

Next, let's specify what array parameters will be used (“Number of copies and total angle”), in the “**Rotation (Rows)**” section of the properties window, and enter the required values of the parameters (the number of copies – 6, the total angle – 3600).

To complete creation of this circular array of 3D nodes, simply click

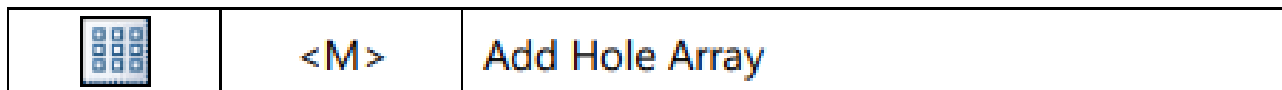



| | | | |
|---------------------------|---|--|---|
| | | | |
| Selecting 3D node to Copy | Selecting first 3D node of the array's axis of rotation | Selecting second 3D node of the array's axis of rotation | Result of creating an array of 3D nodes |

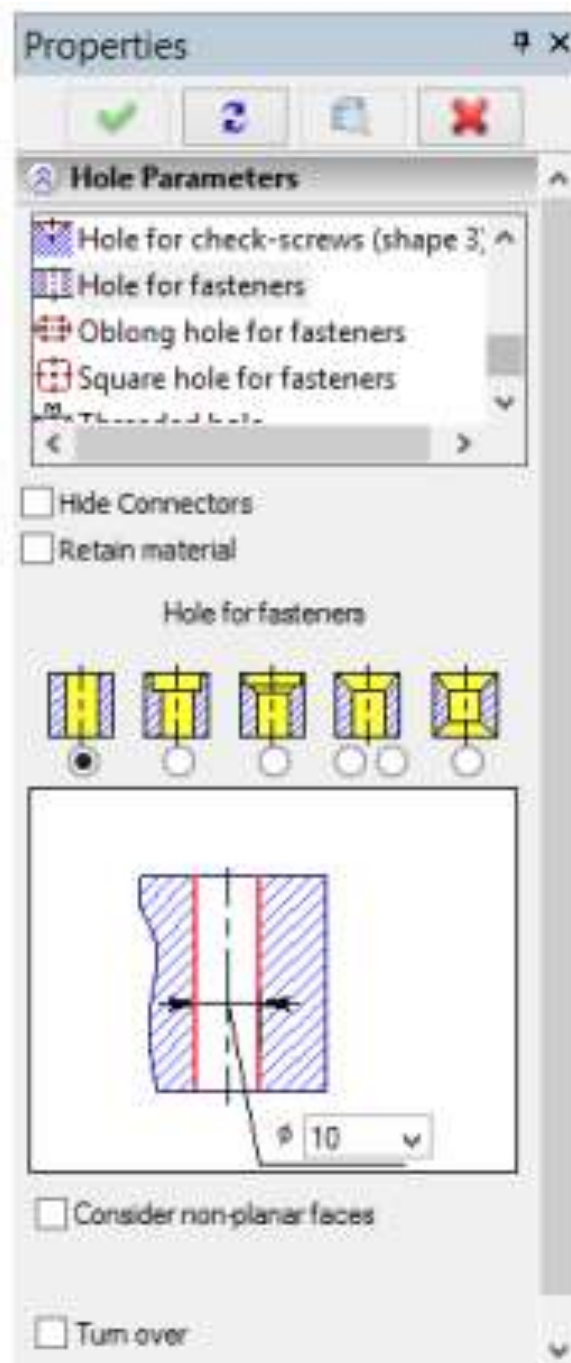
After creating the array of 3D nodes, you can call the command **3H: Create Hole:**

| Icon | Ribbon |
|----------|--------------------------|
| | 3D Model → Modify → Hole |
| Keyboard | Textual Menu |
| <3H> | Operation >Hole |

Upon entering the command, you need to activate the option:




After that, move the pointer to one of the 3D nodes of the circular array and click . In the command's properties window, select the hole type – “Hole for fasteners”. The preview image of the holes being created will appear in the 3D window.



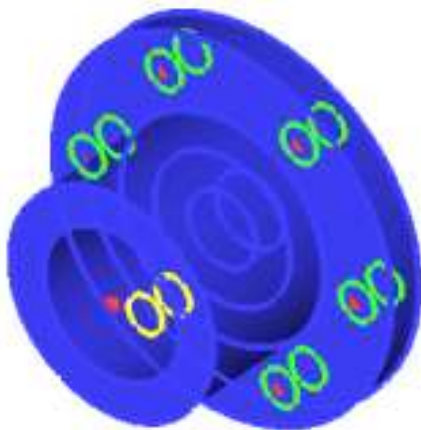
At the bottom of the properties window there is a pane with a schematic image of the hole of the selected type and fields for entering the hole parameters. Set the hole diameter to be 20 mm.

The following option will be automatically activated in the command's automenu for the holes of this type:

| | | |
|---|-----|-------------|
|  | <F> | Through all |
|---|-----|-------------|


When this option is set, the depth of the holes will be defined automatically by the part's thickness.

To complete creation of holes, simply click in the properties window or in the command's automenu.




2.6. Creating Holes without Using a Special Command


In this approach, initial constructions of the holes creation will be same as in the previous method.


You select a part's face and call the context menu command «Draw On Face». Call the command “**L: Construct Line**”. Select the automenu option . Construct a vertical line passing through the center of the circle (of the face's projection). Then, call the command “**C: Construct Circle**”.

Create the circle with the radius equal to 80 and the center at the central node. Similarly, create another circle with the radius equal to 10 mm at the intersection of the previous circle's circumferential and the vertical line.

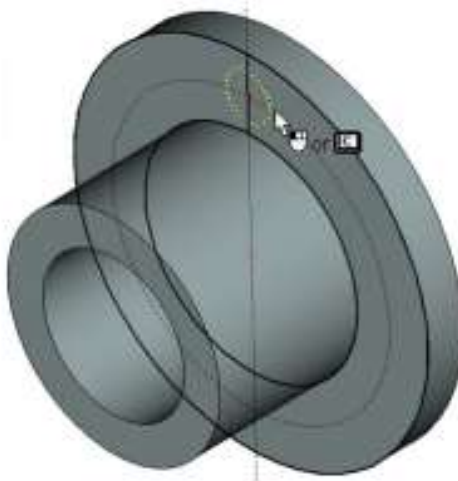
Then draw graphic line over the newly created circle. To do so, call the graphic line creation command:

| Icon | Ribbon |
|---|----------------------------|
|  | Draw → Draw → Graphic Line |
| Keyboard | Textual Menu |
| <G> | Draw > Graphic Line |

Pick the  icon on the system toolbar, and select CONTINUOUS type from the pull-down list.

Move the cursor to the just drawn circle. The circle will get highlighted, and the cursor gain a circle glyph. Once so, click , that applies graphics over the whole circle. On a crowded drawing with multiple elements near the cursor, snapping may occur to an unintended element.


In such cases, the circle can be explicitly selected by typing <C> key. This will select the circle nearest to the cursor.



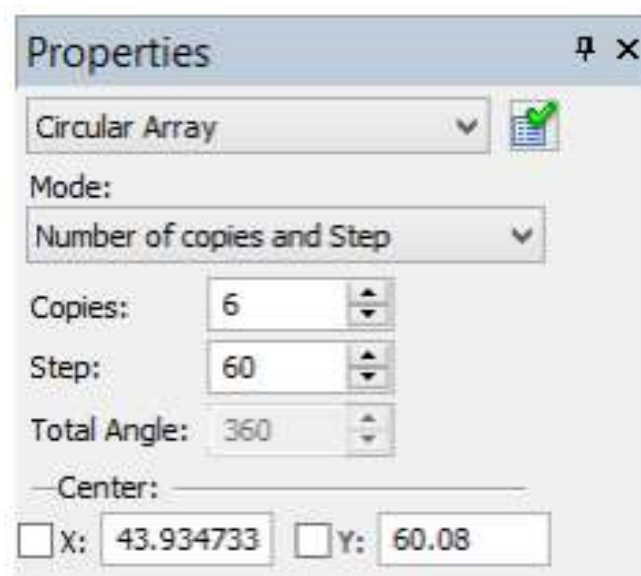
Two possible approaches are available for further construction. In one of them, you could draw five more circles, to extrude and subtract them all from the body of the part. Alternatively, extrude just a single circle, and then “replicate” the resulting hole by means of the circular 3D array.

The first approach by steps:


A handy tool for creating the required number of copies of the circle is the “Create Circular Array” command:

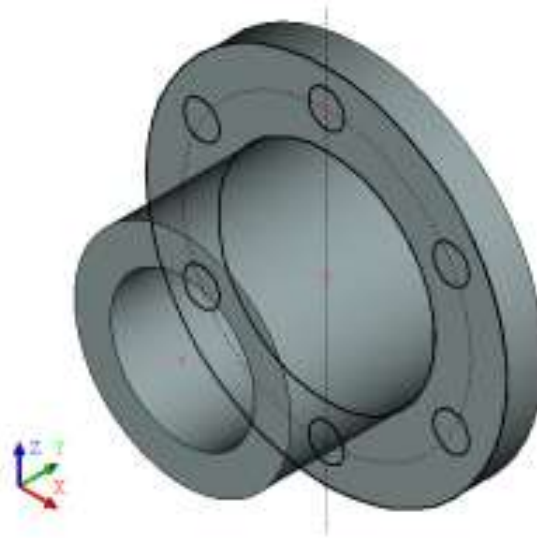
| Icon | Ribbon |
|---|------------------------------------|
|  | Draw → Additional → Circular array |
| Keyboard | Textual Menu |
| <XR> | Operation > Array > Circular |

Once in the command, the system waits for selection of a graphic entity. Move the cursor to the image of the hole and click to select the graphic line. That’s all to be selected, therefore, press in the automenu.




Now the system expects selection of the array center, which must be a 2D node. Note that the default setting for the number of copies of the circular array is 4, and we need six. Therefore, specify the correct parameters in the property window, as shown on the diagram at right.

Now you can select the node. Move the cursor to the center node and click . The result will look as on the following diagram.



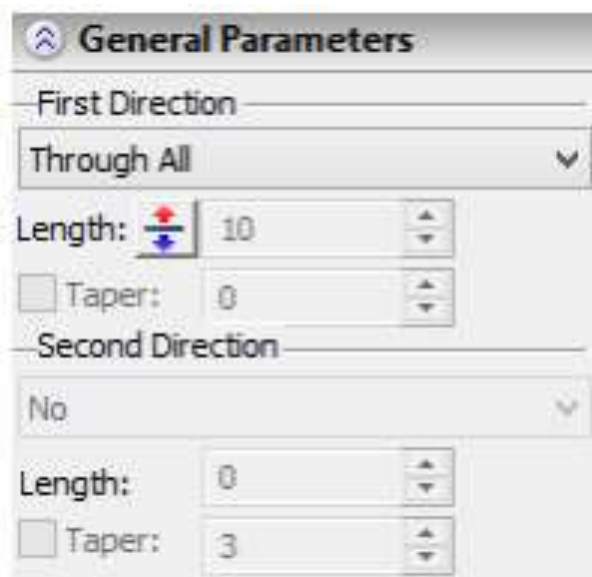
Next, we will need to call the profile extruding command,

| Icon | Ribbon |
|---|-------------------------------|
|  | 3D Model → Create → Extrusion |
| Keyboard | Textual Menu |
| <3X > | Operation > Extrusion |

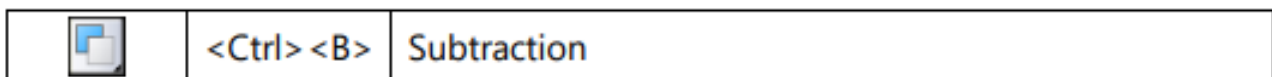
The system automatically makes a 3D profile from the drawn lines and sets the extrusion vector normal to the profile.

The amount of extruding, that is, the depth of the hole, can be defined in various ways (by a numerical value, by the length of the direction vector, etc.). In this particular case, we need to get a through hole, that is the one penetrating the entire width of the part.

In the properties window, set the “First direction” parameter to the value “Through all”, by selecting from the drop-down list. In this definition, the extrusion amount is determined by the thickness of the auxiliary bounding body. Additionally, the created extrusion is automatically subtracted from that body. This method of specifying the extrusion distance was devised specifically for quick creation of through holes in various bodies.

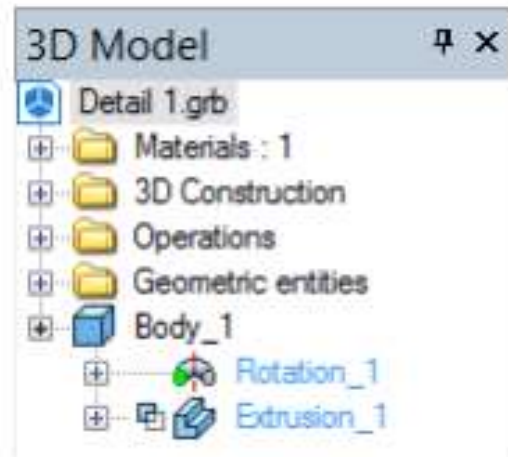


The option of creating a Boolean operation will automatically activate in the automenu:




Since only one body is present in the scene at the time of creating the extrusion, it will be automatically selected as the bounding body and the first operand of the Boolean subtraction.

Upon clicking , two operations will be created at once – an extrusion and a Boolean operation.




The second approach by steps:

Call the profile extruding command . Specify extrusion length parameter equal to 20 in the reverse direction, and confirm the extrusion operation.




Next, let's call the circular array creation command,

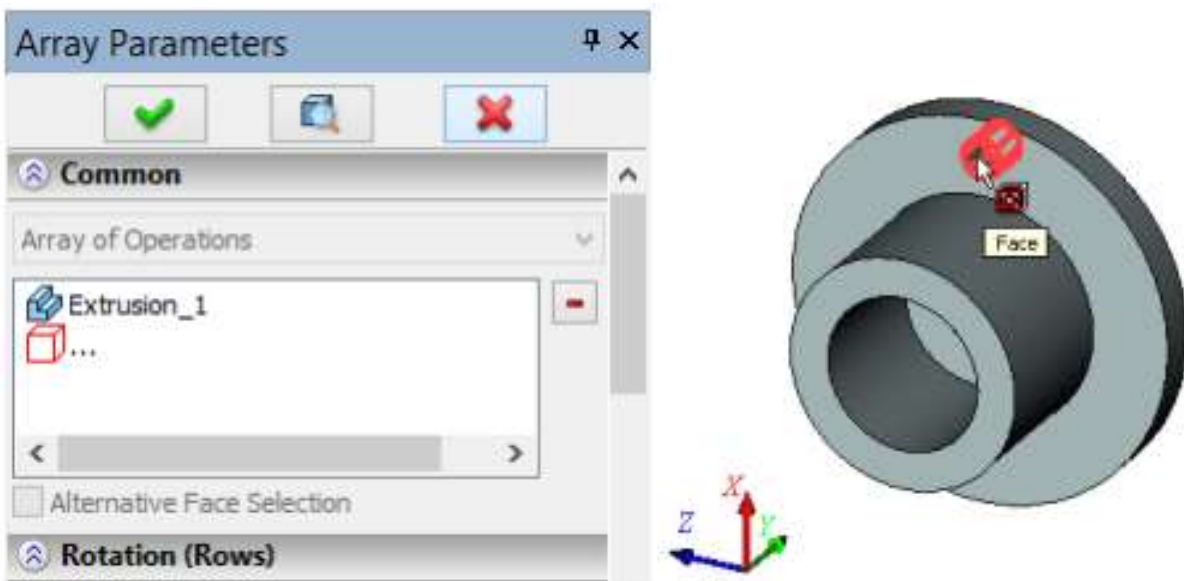
| Icon | Ribbon |
|---|------------------------------------|
|  | 3D Model → Create → Circular array |
| Keyboard | Textual Menu |
| <3AR> | Operation > Circular array |

In the command's properties window, select the array type – “Array of Faces”.


The following option will be automatically activated in the automenu:

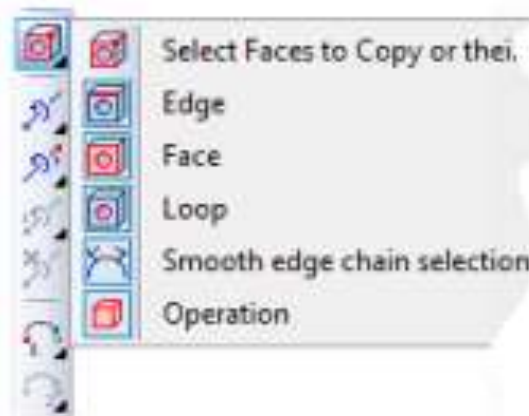
| | | |
|---|-----|---|
|  | <N> | Select Faces to Copy or their boundary Edges or Loops |
|---|-----|---|





Aim the pointer at the cylindrical face of the hole and click .

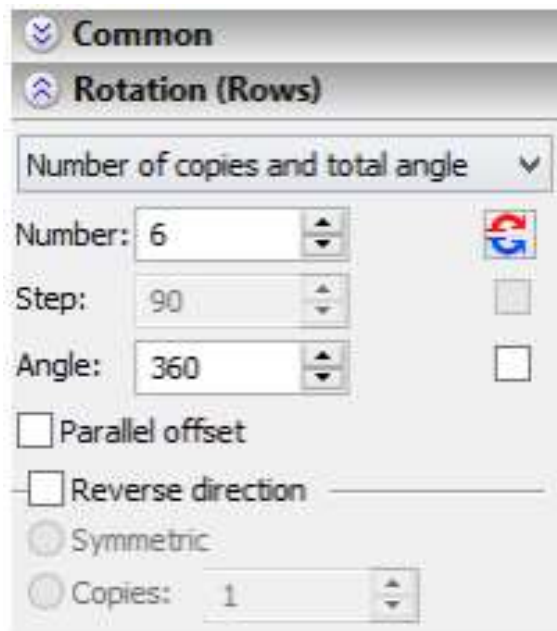


If edges or loops highlight instead of the face, then you can do any of the following:

– rest the pointer over the face until the multiple selection widget  appears next to it. Then scroll through the list of elements offered for selection by rotating the mouse wheel until you get the desired face;

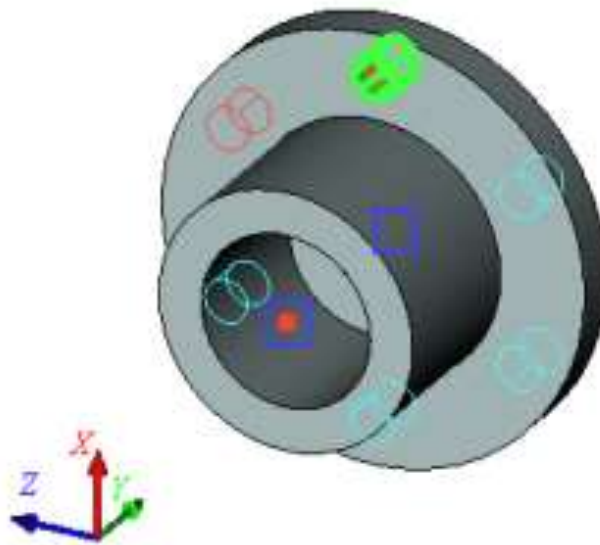


– before selecting the face, set up the list of 3D objects available for selection (banning from selection anything but faces). To do this, double-click  the icon of the faces filter . The list of selectable objects can be also set up by the drop-down menu of the option : point at the option, depress  and hold the mouse button for a second. In the appearing menu, disable all flags except “Faces”.



Once the intended operation is selected, the system expects input of the axis of revolution for the array. The axis can be assign by the same two 3D nodes as those used in the first method of creating holes. Select these two 3D nodes one by one.

You can define the array parameters in the properties window – number of copies and total angle.




Press the button in the automenu to complete the operation. As a result, five new faces will be “embedded” in the original body, making up the additional holes.

2.7. Creating a Blend

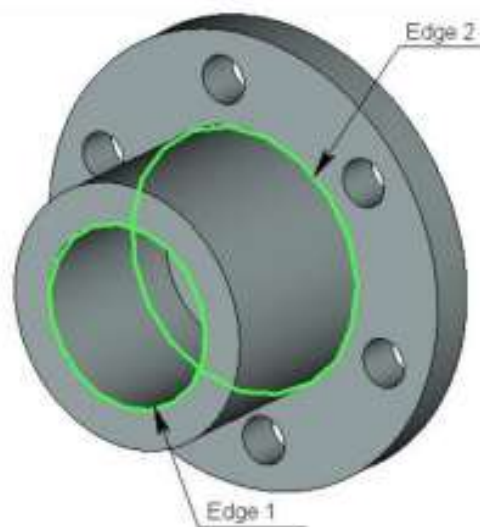
The next step is finalizing 3D model creation. This includes creation of a chamfer and a rounding. Call the command:

| Icon | Ribbon |
|---|----------------------------------|
|  | 3D Model → Create → Blend → Edge |
| Keyboard | Textual Menu |
| <3DE> | Operation > Blend > Edge |

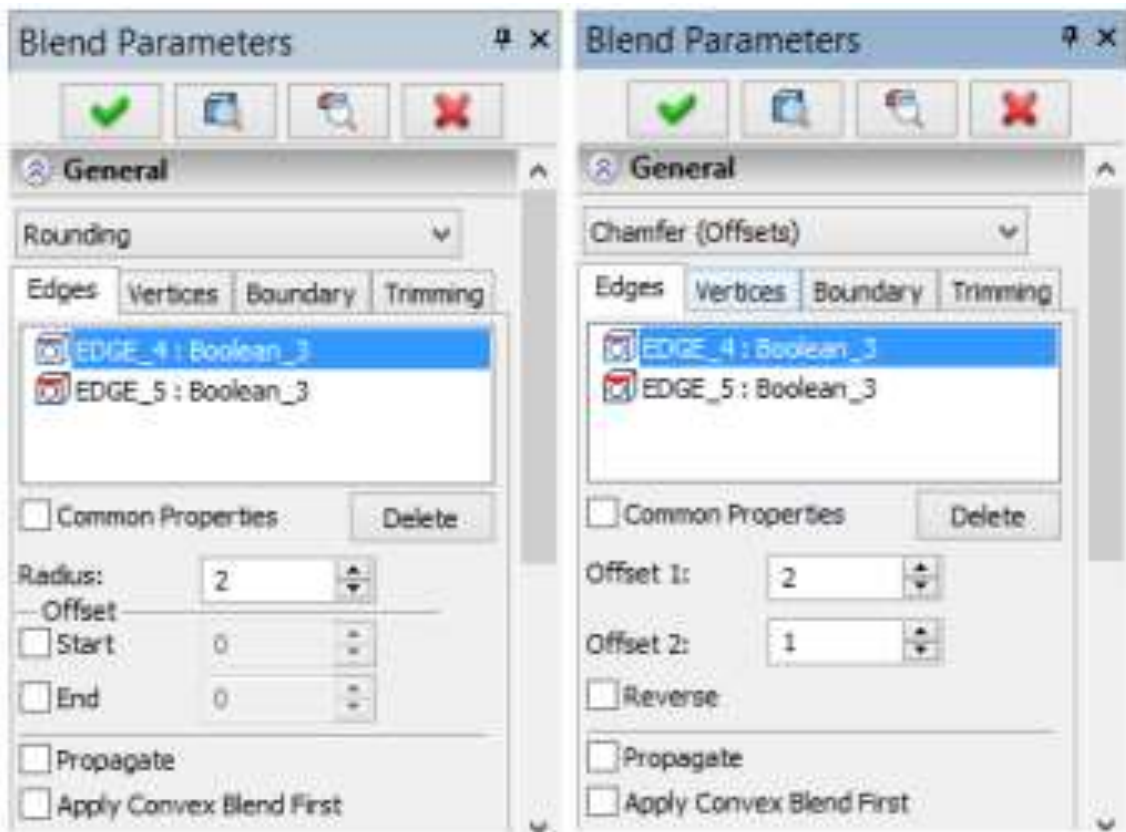
Pick the following option in the automenu:

| | | |
|---|-----|-------------|
|  | <E> | Select Edge |
|---|-----|-------------|

The system is waiting for an edge selection. Select two edges in the order shown on the following diagram.



Next, specify the operation parameters. Each edge can be assigned its specific set of parameters in the property window by unchecking the “Common Properties” option.



Select the first edge in the list and specify the blending type as Rounding with Radius equal to 2 mm. For the second edge, define the blending type as Chamfer (Offsets), with Offset1 and Offset 2 both equal to 5 mm. Confirm the operation by pressing . The result appears as shown on the following diagram.



This completes creation of the given 3D model.

3. THE ORDER OF LAB WORK

1. Come to know the principles and steps of the modeling process.
2. Get acquainted with the basic commands in creating three-dimensional models of geometric objects.
3. Create three-dimensional model (elementary three-dimensional figure with a hole) according to the individual tasks (Table 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. Three-dimensional model of the part.
4. Conclusion.
5. Answer the questions from the checklist.

5. CHECKLIST

1. The possibilities of 3D modeling technology in the design of machine parts.
2. What are the advantages of 3D modeling?
3. Main approach to 3D model creation.
4. What is the basis of 3D modeling?
5. The principles and steps of the modeling process.
6. The sequence of creating a 3D model of the part.
7. What are the basic commands in creating three-dimensional models of geometric objects?
8. Classification of three-dimensional models by functional purpose and their difference.
9. The functions of the dashboard, toggle bar and work bar of the 3D modeling program.
10. Creating auxiliary elements in 3D modeling.
11. Creating rotation operation by means 3D modeling.
12. Creating holes of the part by means 3D modeling.
13. Creating holes in the 3D modeling program using a dedicated command.
14. Creating holes in the 3D modeling program without using a special command.
15. Creating a blend of the part by means 3D modeling.
16. How to specify the operation parameters in the 3D modeling program?

Task 1

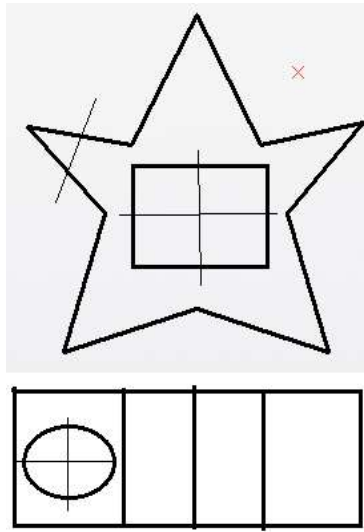
for lab work № 4

Create three-dimensional model (elementary three-dimensional figure with a hole).

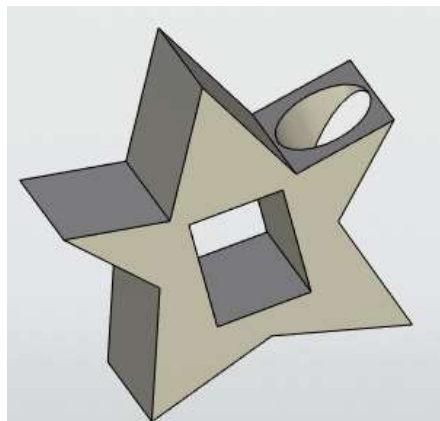
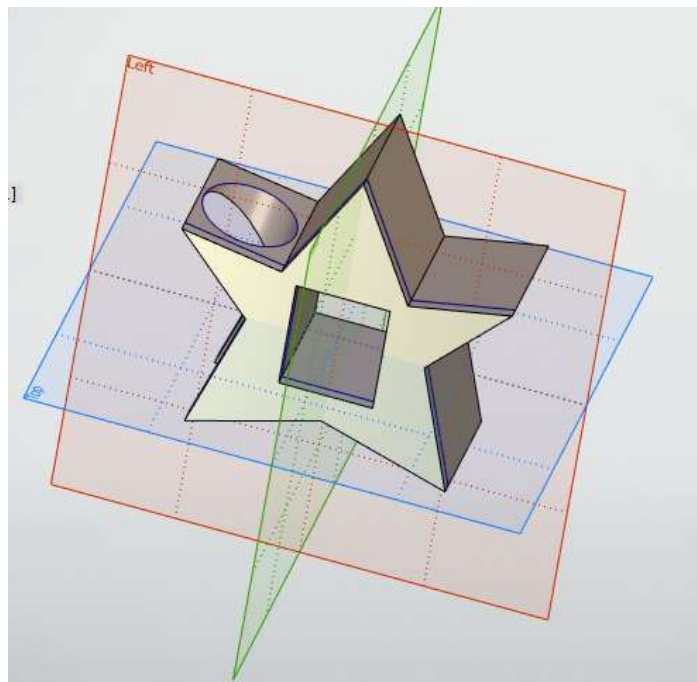
Table 1

| | | | | | |
|---|--|----|--|----|--|
| 1 | | 8 | | 16 | |
| 2 | | 9 | | 17 | |
| 3 | | 10 | | 18 | |
| 4 | | 11 | | 19 | |
| 5 | | 12 | | 20 | |
| 6 | | 13 | | 21 | |
| 7 | | 14 | | 22 | |
| | | 15 | | 23 | |
| | | | | 24 | |

An example



Three-dimensional model

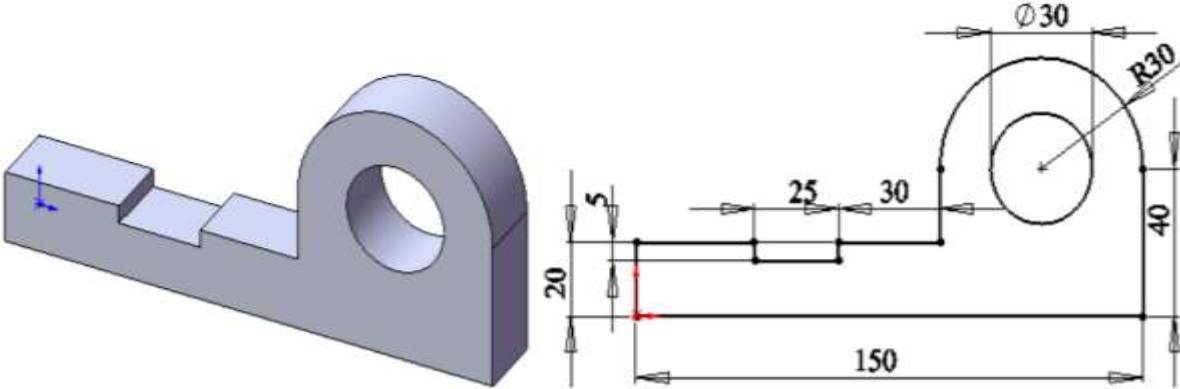
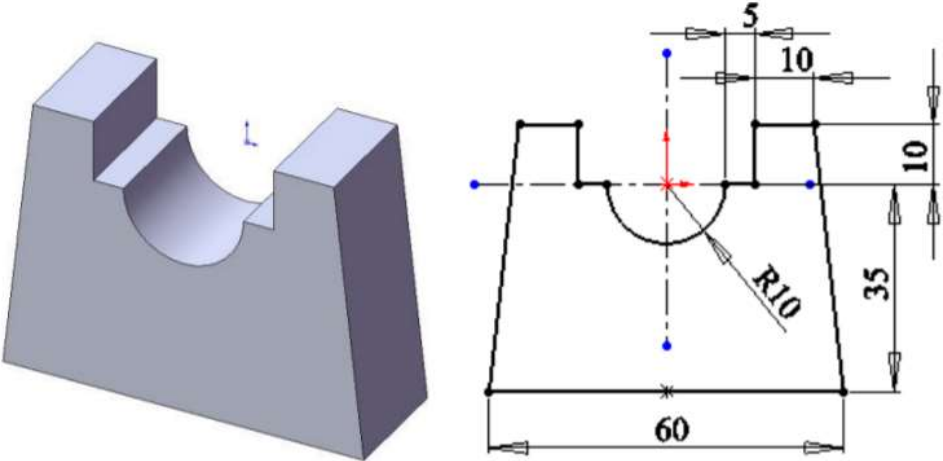
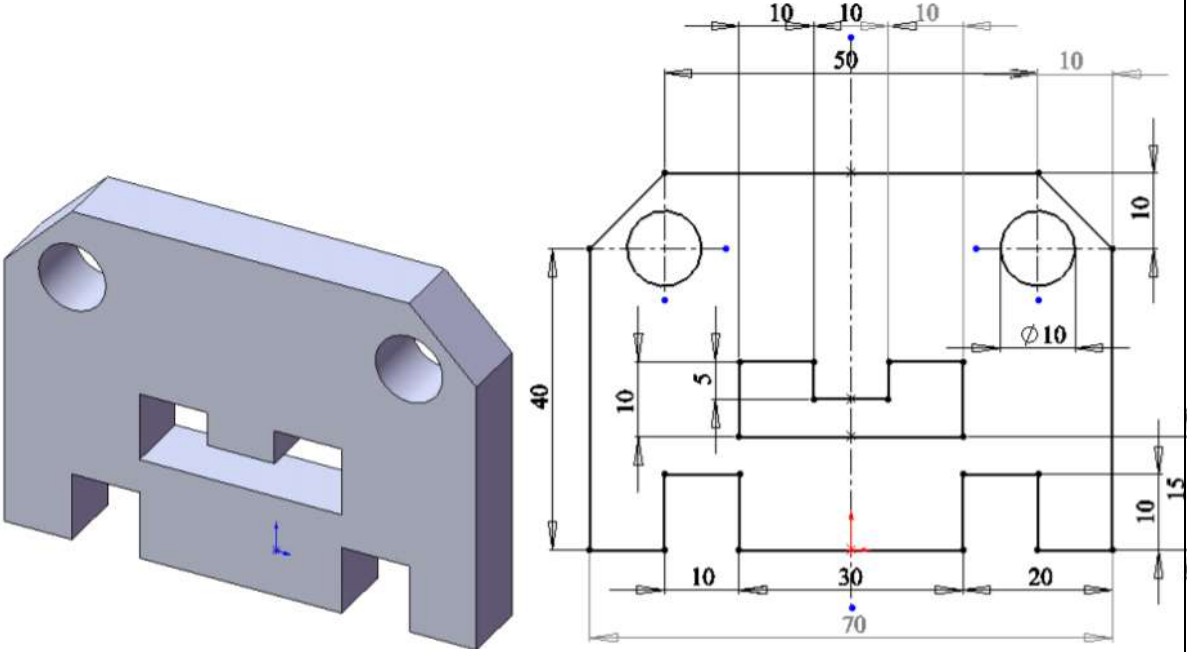


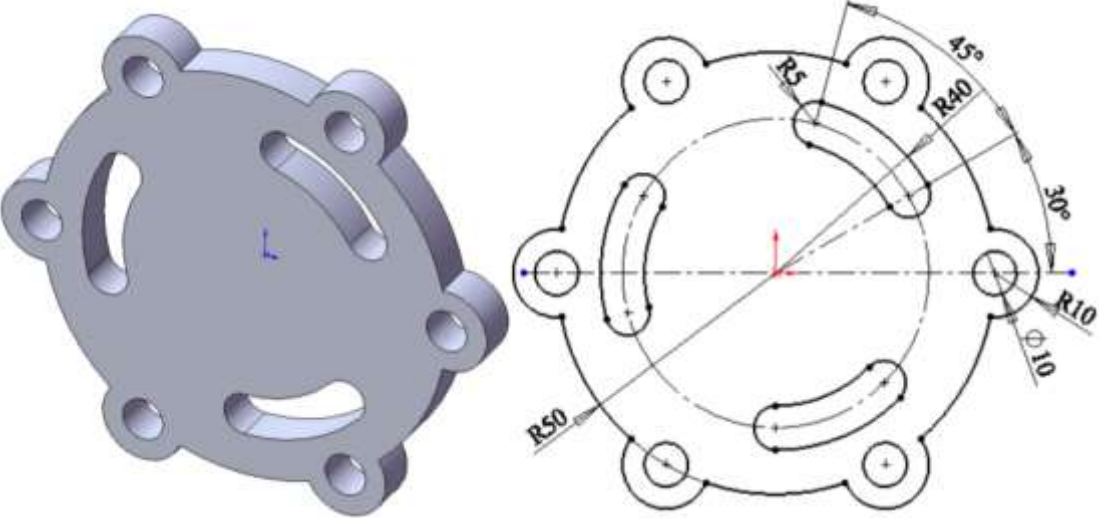
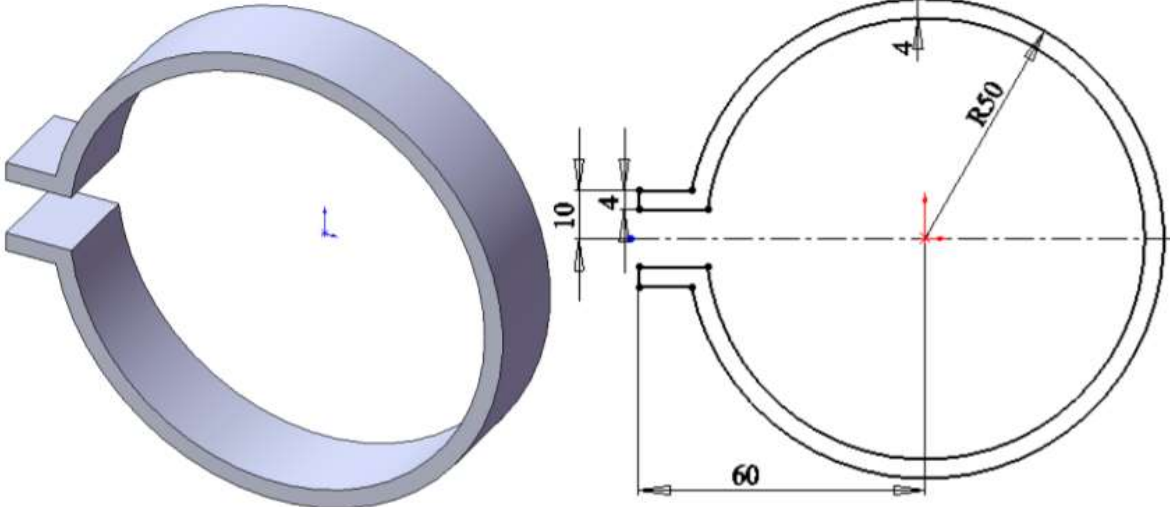
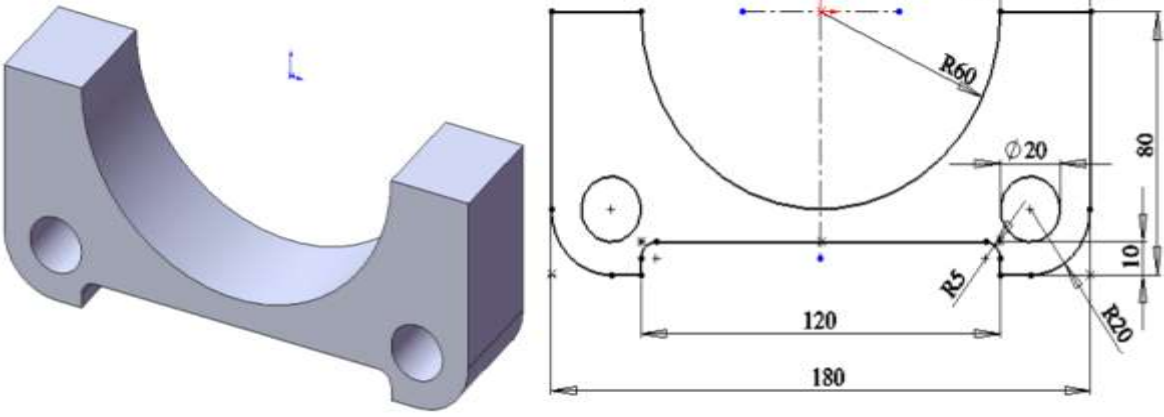
Task 2

Create three-dimensional model of the part.

Table 2

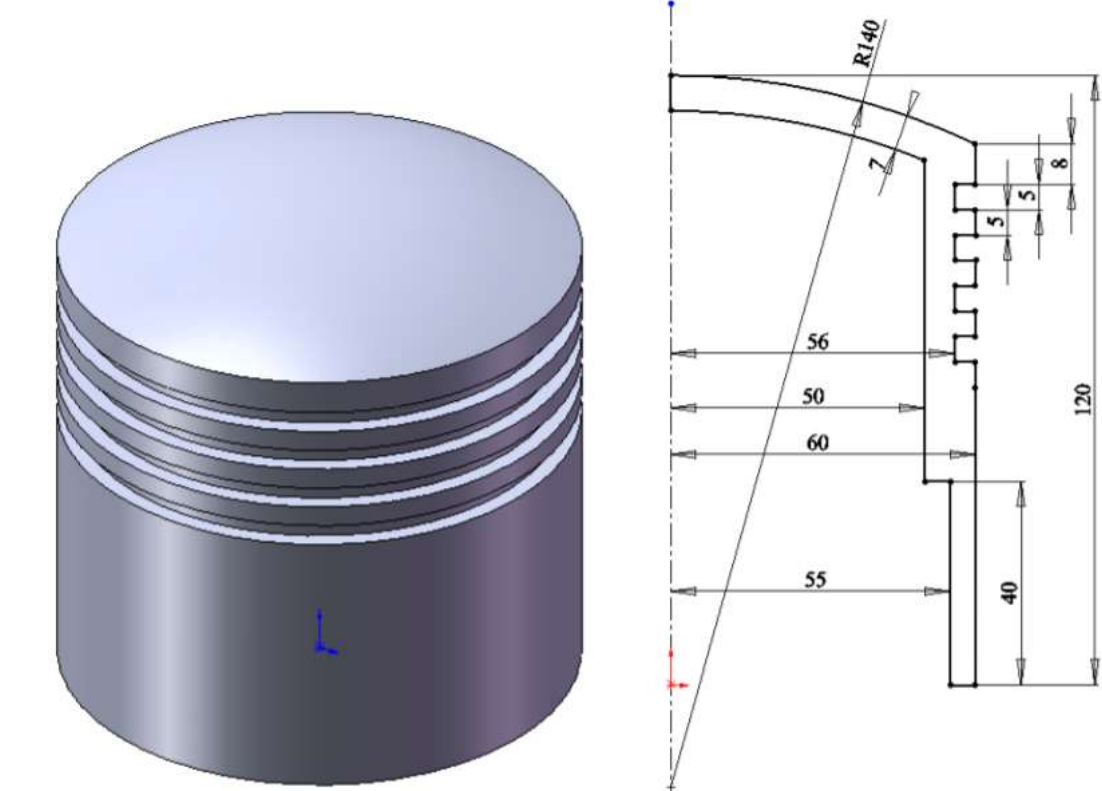
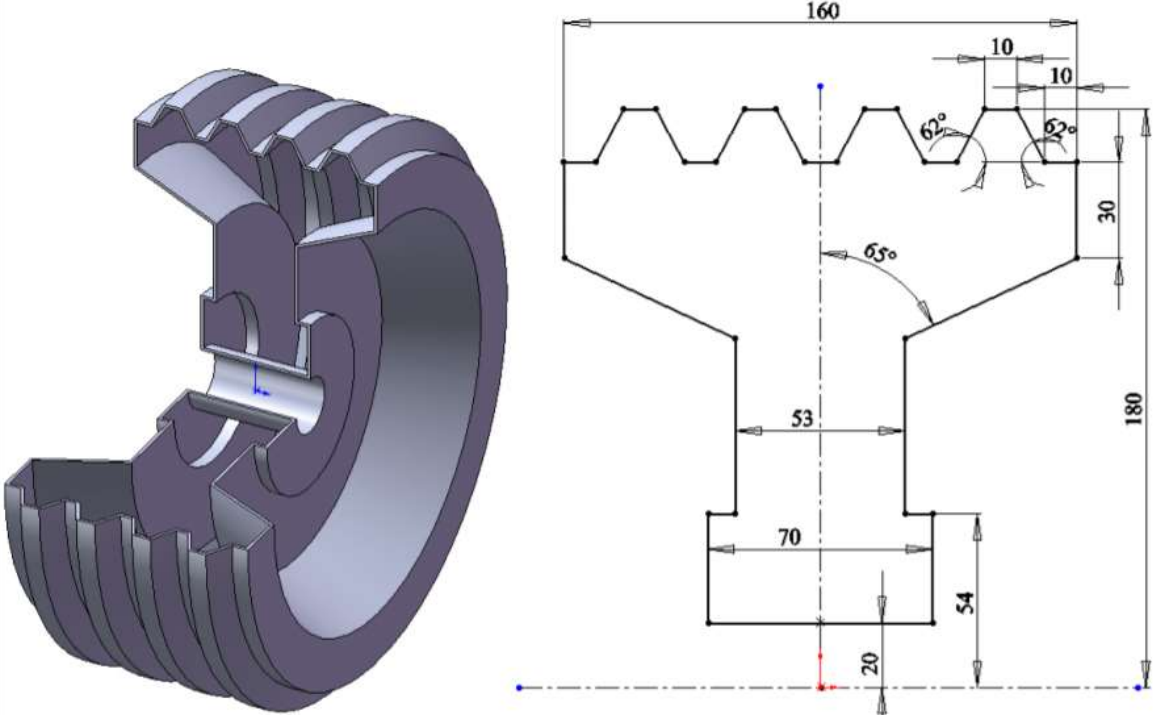
| № var. | Drawing of the part and its dimensions |
|--------|--|
| 1 | 2 |
| 1 | |
| 2 | |
| 3 | |

| | |
|---|--|
| 1 | 2 |
| 4 |  <p>3D model and 2D drawing of a mechanical part. The 3D model shows a rectangular base with a semi-circular top and a circular hole. The 2D drawing shows a total length of 150, a height of 40, and a hole diameter of $\varnothing 30$. The hole is centered 30 units from the right edge. The top edge has a radius of $R30$. The base has a thickness of 5 units. The distance from the left edge to the start of the hole is 25 units.</p> |
| 5 |  <p>3D model and 2D drawing of a mechanical part. The 3D model shows a trapezoidal base with a semi-circular cutout. The 2D drawing shows a total width of 60 and a height of 35. The cutout has a radius of $R10$. The top edge has a thickness of 5 units. The distance from the center of the cutout to the right edge is 10 units.</p> |
| 6 |  <p>3D model and 2D drawing of a complex mechanical part. The 3D model shows a rectangular base with a central slot and two circular holes. The 2D drawing shows a total width of 70 and a height of 40. The central slot has a width of 30 and a depth of 5. The two circular holes have a diameter of $\varnothing 10$ and are spaced 10 units from the left and right edges. The top edge has a thickness of 10 units. The base has a thickness of 15 units.</p> |

| 1 | 2 |
|---|--|
| 7 |  <p>The 3D model shows a circular flange with six holes around its perimeter and a central slot. The 2D drawing includes the following dimensions: a central slot with a radius of $R5$, a hole with a radius of $R40$, a hole with a radius of $R10$, a hole diameter of $\phi 10$, and angles of 45° and 30°. A dashed circle with a radius of $R50$ is also shown.</p> |
| 8 |  <p>The 3D model shows a circular ring with a central hole and two protrusions. The 2D drawing includes the following dimensions: a central hole with a radius of $R50$, a hole diameter of $\phi 60$, and a thickness of 4. The protrusions have a height of 10 and a width of 4.</p> |
| 9 |  <p>The 3D model shows a C-shaped bracket with two holes. The 2D drawing includes the following dimensions: a total width of 180, a total height of 80, a hole diameter of $\phi 20$, a hole offset of 120 from the left edge, a hole offset of 30 from the right edge, a hole offset of 10 from the bottom edge, and radii of $R60$ and $R5$.</p> |

| | |
|----|---|
| 1 | 2 |
| 10 | |
| 11 | |
| 12 | |

| 1 | 2 |
|----|---|
| 13 | |
| 14 | |
| 15 | |

| 1 | 2 |
|-----------|---|
| <p>16</p> |  <p>The image shows a 3D perspective view of a cylindrical stepped shaft on the left and its corresponding technical drawing on the right. The shaft has a total length of 120 units. The top end is chamfered with a radius of R140 and a chamfer angle of 7 degrees. The shaft features several steps of varying diameters. From the left end, the diameters are 56, 50, 60, and 55 units. The 55 unit diameter section has a length of 40 units. The 50 unit diameter section has a length of 5 units. The 60 unit diameter section has a length of 5 units. The 56 unit diameter section has a length of 5 units. The final 8 units of the shaft have a diameter of 56 units. The technical drawing includes a vertical centerline and a red arrow indicating the start of the chamfer.</p> |
| <p>17</p> |  <p>The image shows a 3D perspective view of a complex stepped shaft on the left and its corresponding technical drawing on the right. The shaft has a total length of 180 units and a maximum diameter of 160 units. It features a central hole with a diameter of 20 units. The shaft has several steps and chamfered ends. The technical drawing includes a vertical centerline and a red arrow indicating the start of the chamfer. The diameters of the shaft sections are 70, 53, 54, 30, and 10 units. The 10 unit diameter section has a length of 10 units. The 30 unit diameter section has a length of 10 units. The 54 unit diameter section has a length of 54 units. The 53 unit diameter section has a length of 53 units. The 70 unit diameter section has a length of 70 units. The chamfered ends have a chamfer angle of 65 degrees. The technical drawing includes a vertical centerline and a red arrow indicating the start of the chamfer.</p> |

CONTENTS

1. Goals
 2. Theoretical Information
 3. The Order of Lab Work
 4. Report Structure
 5. Checklist
- Task 1
- Task 2