

Ministry of Education and Science of Ukraine
Chernihiv Polytechnic National University
Educational-scientific institute of mechanic engineering, technology
and transport

DELICAM software in mechanical engineering

Methodological instructions

for laboratory classes from the mandatory discipline

"CAD of machines and tools"

for students majoring in 133 "Industrial mechanical engineering"
full-time studies for the second (master's) level of higher education

APPROVED
at the meeting department of
Automobile Transport and Sectoral
Machine Building
Protocol № 12, 16.06.2023

Chernihiv, 2023

DELTCAM software in mechanical engineering. Methodological instructions for laboratory classes from the mandatory discipline "CAD of machines and tools" for students majoring in 133 "Industrial mechanical engineering for the second (master's) level of higher education full-time studies / compiled by V. Kalchenko, A. Kolohoida., N. Sira, H. Pasov - Chernihiv: Chernihiv Polytechnic NU, 2023. – 108 c.

Compilers:

Kalchenko Volodymyr, Doctor of Technical Sciences, Professor Department of Automobile Transport and Sectoral Machine Building

Kolohoida Antonina, PhD, Department of Automobile Transport and Sectoral Machine Building

Pasov Gennadiy, PhD, Department of Automobile Transport and Sectoral Machine Building

Sira Nataliia, PhD, Department of Automobile Transport and Sectoral Machine Building

Responsible for release:

Kolohoida Antonina, PhD, Department of Automobile Transport and Sectoral Machine Building

Reviewer:

Volodymyr Venzheha, PhD, Department of Automobile Transport and Sectoral Machine Building

INTRODUCTION

Methodical instructions for laboratory classes from the course "CAD of machines and tools" are intended for students of technical specialties of higher educational institutions who are studying in the specialty "Industrial mechanical engineering".

The discipline "CAD of machines and tools" belongs to the cycle of professional training, it is based on previous knowledge of fundamental and professionally oriented disciplines, in particular, such as "Metalworking equipment", "Machine parts", "Higher mathematics", " Fundamentals of CAD", etc.

The task of the discipline is to acquaint students of the specialty with automated design systems (CAD) for their further use in solving design tasks, for example, in the design of machine tools and tools. Students should understand the general principles of construction and operation of CAD, as well as know the tools necessary for designing machines and tools. To acquire basic knowledge of the general principles of work and trends in the development of modern systems of automated design with the help of computer technology; to be able to use the most common domestic and foreign CAD systems.

History about the Delcam software

Delcam was founded in 1977 on the Cambridge University site. In 1997, the company became a joint-stock company, and its constant development is due to the demand for software. Currently, the company is the largest developer of software for modeling and production of products. Delcam software is used by more than 45,000 organizations in more than 80 countries around the world.

The Delcam company is one of the world's leading manufacturers of progressive CAD/CAM solutions for the manufacturing industry. Delcam's software covers all areas of production, from modeling to precision control of the finished product, providing fully automated CAD/CAM solutions to support and ensure concept-to-reality.

One of the main representatives of the Delcam software is the PowerShape package - a modern modeling package with solid and surface modeling. Surface modeling is the strongest point of the PowerShape package. The package automatically suggests the most appropriate way to build the surface. At the same time, the user at any stage has the opportunity to independently set the construction method and parameters he needs.

Delcam PowerMill is a package for preparing high-performance control programs for CNC milling machines. It allows you to increase the productivity of the machines and, at the same time, achieve the highest quality in the manufacture of parts and equipment. Delcam PowerMill is traditionally considered the best system for preparing control programs for milling of products of complex shape.

Delcam PowerInspect is an accuracy control system using 3-coordinate control and measuring machines and manipulators. Combining the advantages of a wide range of capabilities and ease of use, it allows you to control complex details by comparing them with the original computer models created in the CAD system. Delcam PowerInspect allows you to control both complex free-form surfaces (showing the deviation of any measured points from the computer model)

and geometric elements of the correct shape, control equipment, scan prototypes to create computer models.

Delcam ArtCAM is a software package for spatial modeling and mechanical processing, which allows you to automatically generate spatial models from a flat drawing and obtain products based on them on CNC machines. ArtCAM offers a powerful, easy-to-use set of modeling tools that gives the designer the freedom to create complex spatial reliefs.

A powerful representative of the Delcam family of software products is the FeatureCAM system. This is a system for preparing control programs based on automatic recognition of typical elements. The technology of automatic recognition of processing elements has become extremely popular among customers of small and medium-sized businesses, allowing to make the development of programs for CNC machines as fast and simple as possible.

The Delcam software line includes a specialized solution for the shoe industry - Delcam Crispin, which allows you to solve the design, decoration and manufacturing tasks of all types of shoes. Delcam also offers a specialized solution for dentists. This solution is based on an open system consisting of two modules. Delcam DentCAD is a specialized system for modeling dental bridges and crowns. Delcam DentMill is a specialized system for manufacturing dental bridges and crowns on CNC machines.

FeatureCAM comprises several modules, which enable you to perform different functions. This help file documents the entire product family, so, to differentiate between the features specific to individual products, the titles of help topic titles are followed by product codes in parentheses. The codes are:

(25D) – requires a minimum of the 2.5D Milling product.

(3D LITE) – Requires a minimum of the 3D Lite product. The functionality is also available in the 3D MX and 3D HSM products. 3D Lite is an entry-level 3D product, which enables you to mill only one surface per feature, but you can create multiple features. The strategies available in 3D Lite are Z-level rough, Parallel rough, Parallel finish, Isoline, and 2D spiral.

(3D MX) – Requires a minimum of the 3D MX product. The functionality is also available in the 3D HSM product. 3D MX is a mid-level 3D product that enables you mill multiple surfaces. It has the same strategies as 3D Lite, plus Z-level finish, Radial finish, Flowline finish, Four-axis finish, Horizontal + vertical, Between curves, and Swarf milling.

(3D HSM) – Requires a minimum of the 3D HSM product. 3D HSM is a high speed machining product with the same strategies as 3D MX, plus Pencil, Plunge roughing, Remachine, Steep and Shallow, and Spiral 3D.

(REC) – Requires the FeatureRECOGNITION product.

(MSIM) – Requires the Machine Simulation product.

(5AP) – Requires the 5-axis Positioning product.

(5AS) – Requires the 5-axis Simultaneous product.

(TURN) – Requires the Turning product.

(TURNMILL) – Requires the Turn/Mill product.

(WIRE) – Requires the Wire EDM product.

(TOMB) – Requires the Tombstone product.

(SOLID) – Requires the Solid Modeling product.

(MTT) – Requires the Advanced Turn/Mill (MTT) (Multi-turret turning) product.

(SND) – Requires the shared Network Database product.

LABORATORY LESSON №1

TURNING IN DELCAM FEATURECAM

1.1 Goal

Familiarize yourself with the capabilities of the DELCAM FeatureCAM system. Learn to create control programs for turning parts.

1.2 Brief theoretical information

1.2.1 The general interface of the DELCAM FeatureCAM program

The FeatureCAM interface contains a number of standard Windows elements, such as toolbars, dialogs, context menus, and wizards.

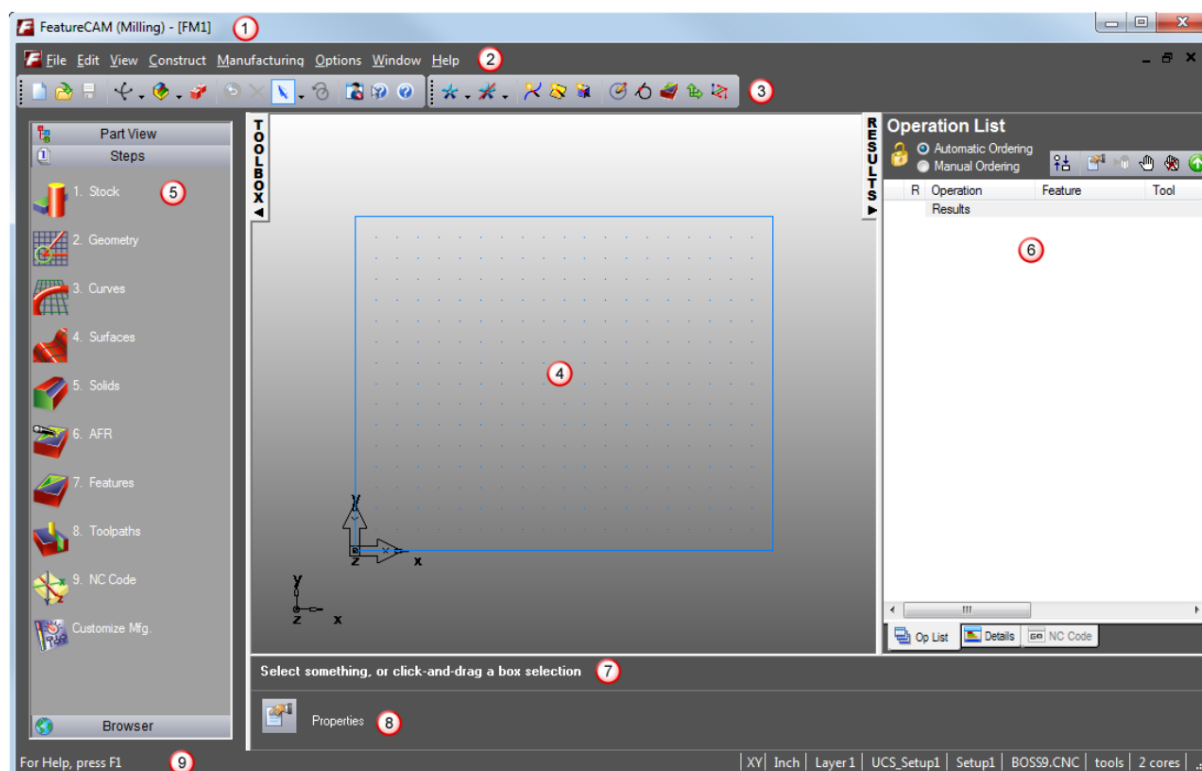


Рисунок 1.1. – Загальний вигляд робочого вікна FeatureCAM

1 – Title bar displays the type of part setup in round brackets, in this case (Milling), and the name of your part file in square brackets, in this case [FM1]. When you have any unsaved changes in your part file, an asterisk (*) is displayed next to its name;

2 – Menu bar provides access to a number of menus. Selecting a menu, such as View, opens a list of associated commands and sub-menus. Sub-menus are indicated by a small arrow to the right of the text. For example, selecting View > Principal Views displays a list of commonly used views;

3 – Toolbars provide quick access to the most commonly used commands in FeatureCAM;

4 – The graphics window is the main working area;

5 – Toolbox window with the Steps panel, Part View panel, and Browser. The Steps panel contains an ordered list of steps for creating part programs. Each step is a wizard that presents a series of dialogs for each process. They are listed in the order in which you should use them during the process of creating a part program. The Part View panel provides a hierarchical view of the part. The Browser contains information on the latest features available in FeatureCAM, including example files that you can load straight into FeatureCAM;

6 – Results window contains the automatically generated documentation including tooling lists, setup sheets, and the NC part programs. Selecting one of the tabs at the bottom of the window changes the content of this window;

7 – Assistance bar displays help for the current command;

8 – Feature/Geometry Edit bar lets you select and edit a feature, or enter the point locations and parameters for geometry creation;

9 – Status bar shows your current drawing units, tool crib, and post processor settings, as well as your keyboard status and information about the simulation when you run one.

As in other Windows programs, in FeatureCAM you can call the necessary command in several ways:

- select the button on the toolbar;
- select an option from the menu;
- select an option from the context menu;
- press the quick call key.

The right mouse button opens a context menu. The menu changes depending on where you are in the program. Common commands and functions that are applicable in this area may also be available.

1.2.1. Panel "Part view"

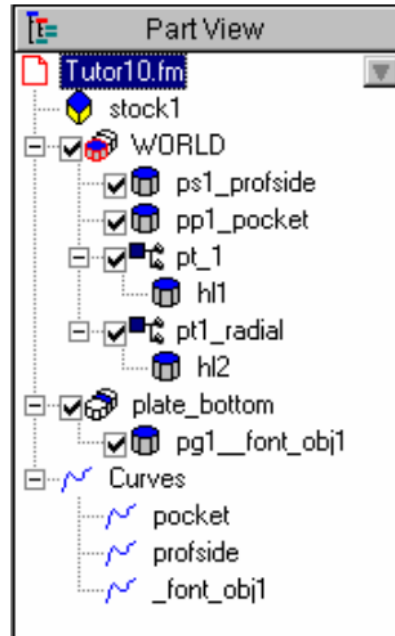



Figure 1.2 – Panel "Part view"

The Part View toolbox provides a hierarchical view of the part. Each setup is listed with each of the features of the setup listed underneath. From this view you can select, show, hide or modify features. To navigate the part view:

1. If an object has a "+" in its left margin, click it to expand the view to reveal objects that are subordinate. Click on the "-" to collapse the view and hide the subordinate objects. Double clicking on the object also performs the same function.

2. Clicking on object in the tree view will select the object in the graphics window. This selection method is good for distinguishing between overlapping objects in the graphics window. Make sure you click on the name and not the check box. See item 4 below.

3. Clicking on an object also reveals the  button for that feature. Clicking on this button brings up menu of actions you can perform on the object. The

actions available are dependent on the object type. Right clicking on the object in the tree view also reveals this menu.


We will get acquainted with the main capabilities of the system and the features of using commands using the example of creating a typical part.

1.3 An example of creating a typical turning part

This tutorial shows you how to create a simple turning part, generate toolpaths and output the toolpaths used to machine the part.

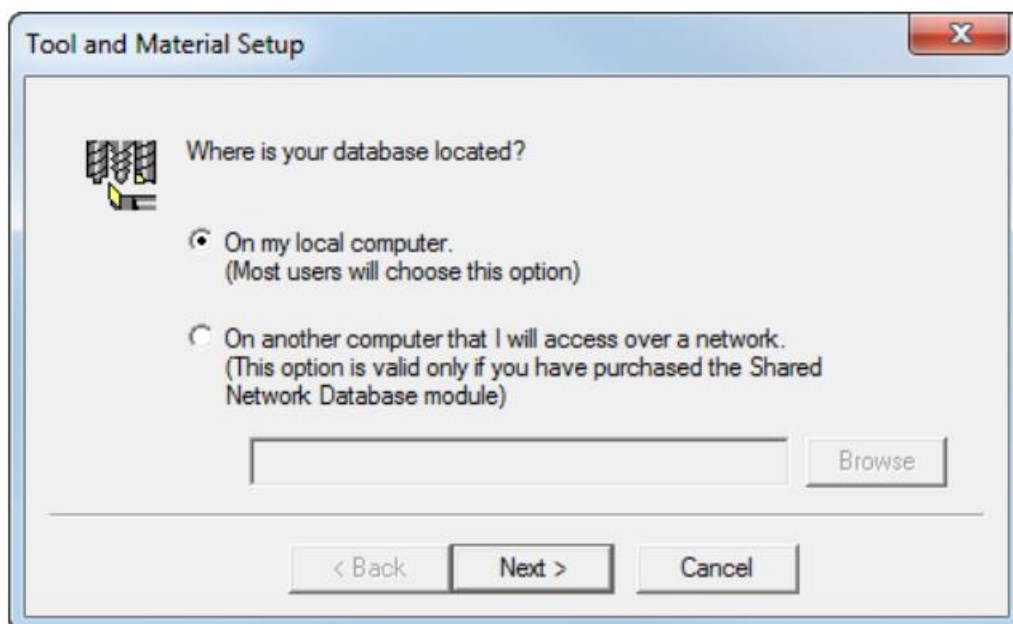
1.3.1. Starting FeatureCAM for the first time

1. From the Start menu, select All Programs > FeatureCAM > FeatureCAM.

✓ You can also start the program by double-clicking the FeatureCAM  icon on your desktop.

The first time you start FeatureCAM, it runs a program to create the tools and materials database.

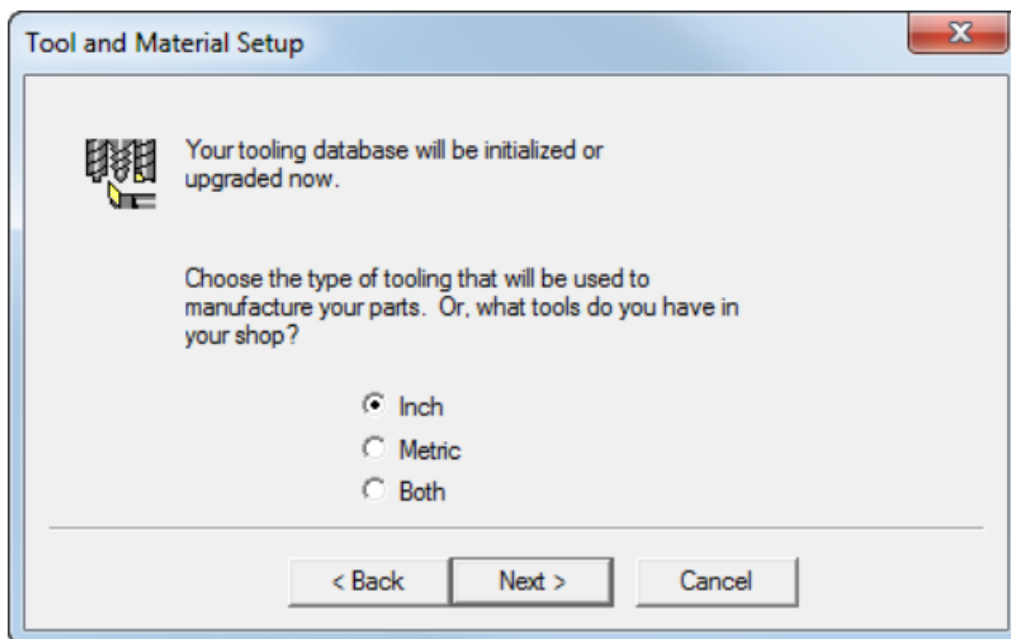
2. Click OK to begin the configuration. This displays the Tool and Material Setup dialog.



3. To create a local database, select On my local computer.

If you want multiple computers to share the same tool and material information:

- Select On another computer that I will access over a network.
 - Click the Browse button, and use the Database Location dialog to select the folder where the database is located.
- ✓ You need to create a database folder on your network first, and then copy an empty MDB database from the FeatureCAM CD-ROM to this location. The default database is created by MS Access, and should be accessed using the MS Jet database driver. You may use a different database type, such as MS SQL Server. For more information, refer to the online help.
- ✓ You need to have the Shared Network Database module to use this option.
4. Click Next.
 5. Choose the tools to load:



Inch - loads only the inch tools

Metric - loads only the metric tools.

Both - loads both inch and metric tools..

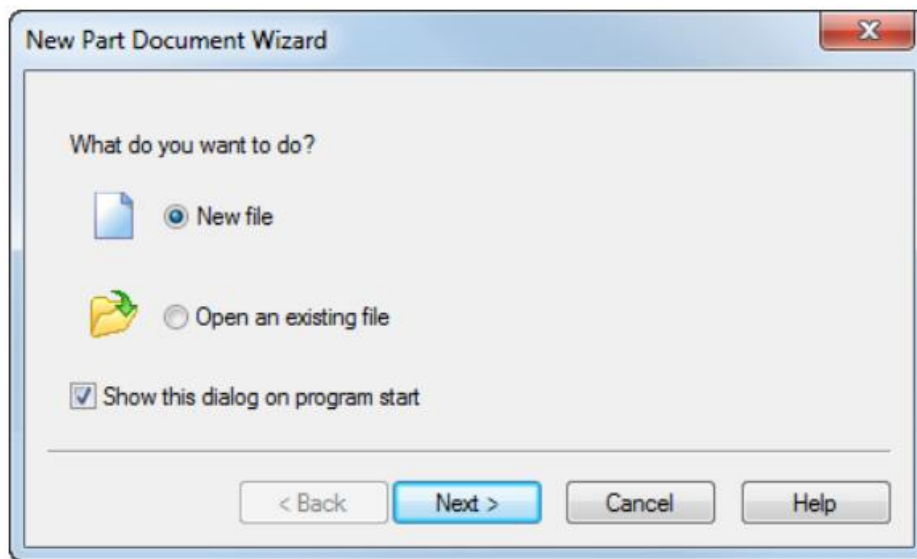
6. Click Next.

7. If you chose to load both tool types, you are asked which tool type you use more often. Select Inch or Metric, and click Next.

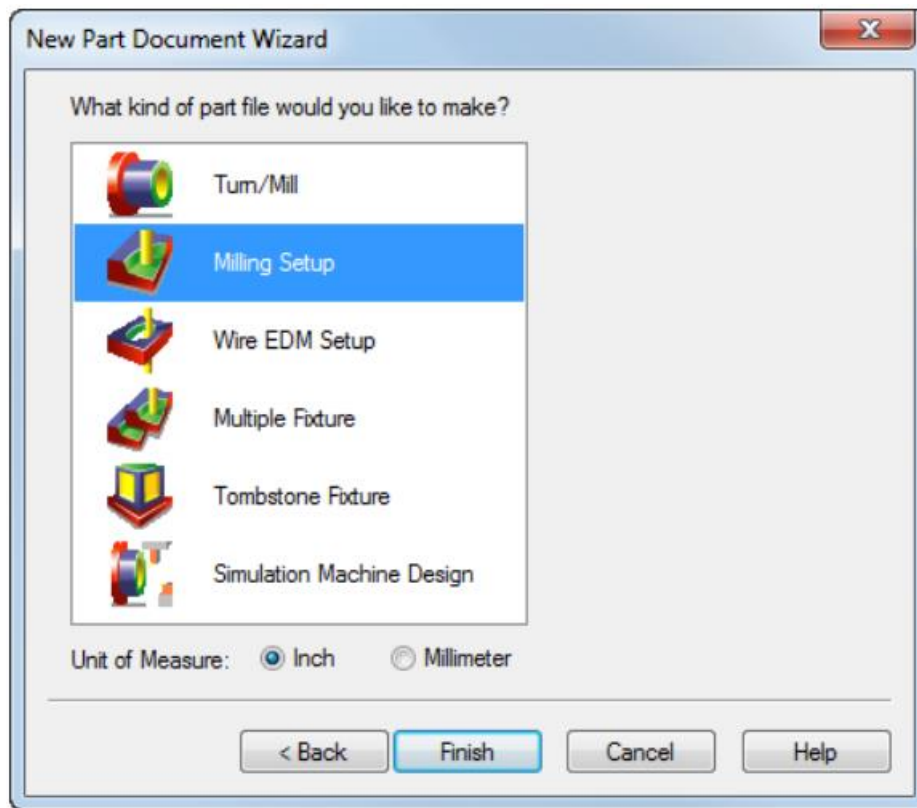
8. Click Finish to initialize the database.
- ✓ The tools database specifies the set of tools used by FeatureCAM to perform manufacturing operations. For best results, use the Tool Manager (available from the Manufacturing menu) to customize the database to reflect the tools in your shop.

1.3.2. Creating a new file

1. Starting FeatureCAM displays the New Part Document wizard.



1. If the New Part Document wizard is not displayed, select the File > New menu option.
2. Select New file in the New Part Document Wizard, and click Next. Select a Milling Setup.



3. Select the Unit of Measure (Inch or Millimeter).

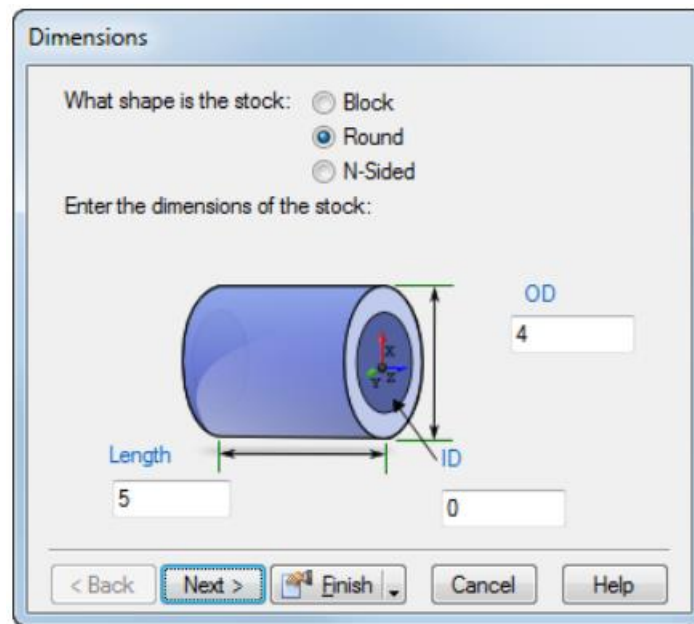
- ✓ You can change the default dimension units later, by selecting Options > File Options from the menu.

4. Click Finish.

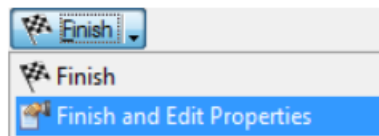
1.3.3. Defining the stock

The stock is the initial material from which you cut your part. By default, the Stock wizard (Dimensions page) opens on the screen as soon as you create a new part. It enables you to set the shape and dimensions for the stock, the stock material, part program zero, and the coordinate system for modeling.

1. On the Dimensions page of the Stock wizard:

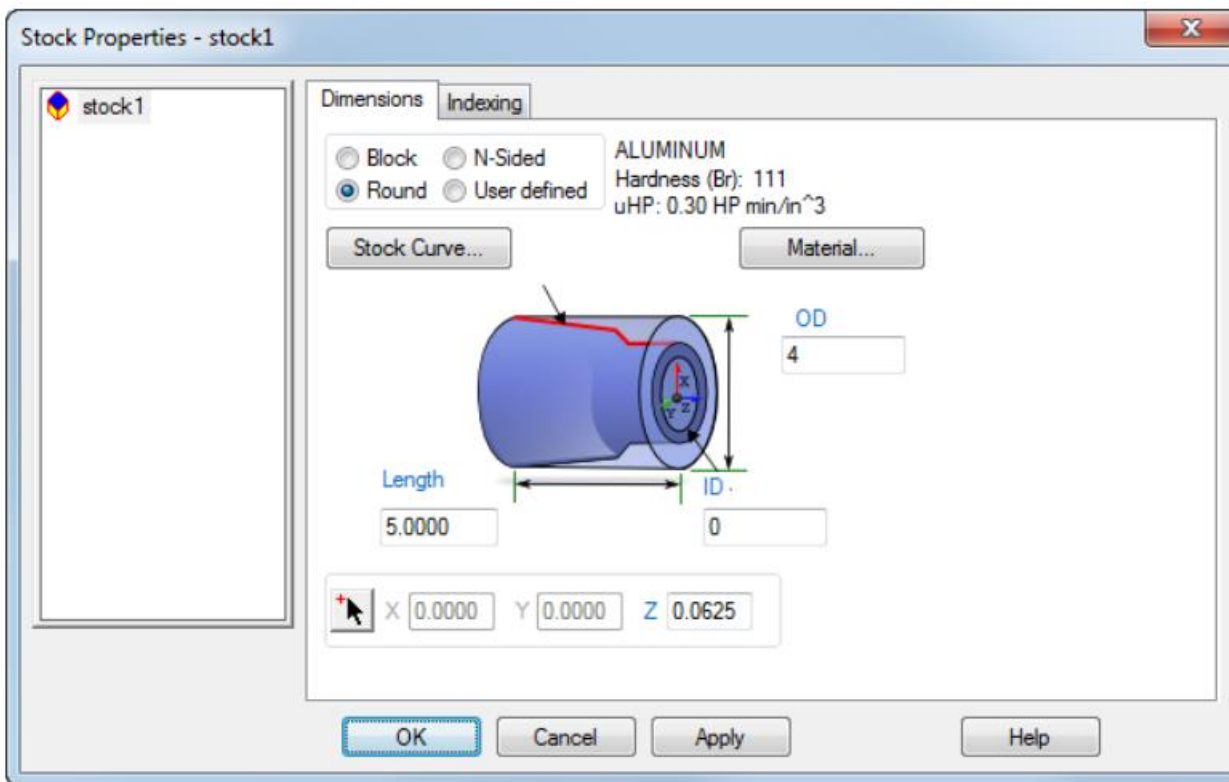


- Enter an OD (outside diameter) of 100 mm.
- Enter a Length of 125 mm.
- Enter an ID (inside diameter) of 0 mm.
- From the Finish menu button select the Finish and Edit Properties option.




This displays the Stock Properties dialog.

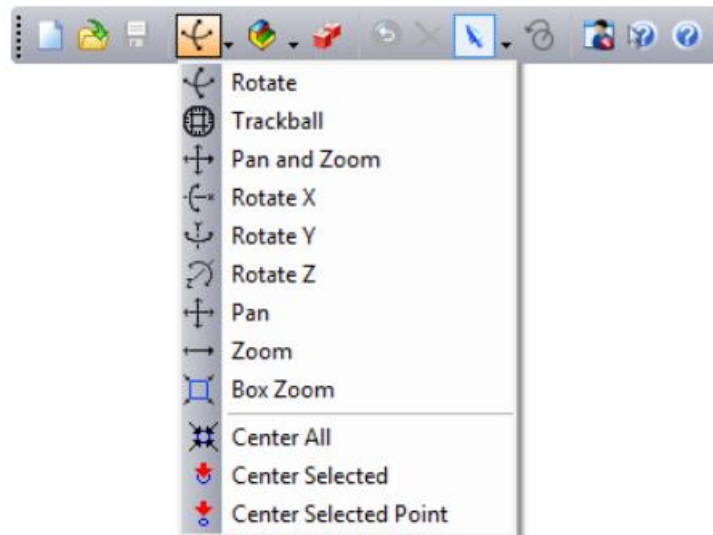
2. In the Stock Properties dialog enter a Z of 0.0625 (1.5 mm), and click OK.



1.3.4. Preparatory steps

The preparatory steps determine the coordinate system and tool crib.

1. Select Options > Turning Input Modes > 3D (XYZ) from the menu to enable you to enter coordinates as X, Y, and Z values.
2. Select Manufacturing > Set Tool Crib from the menu to display the Select Active Tool Crib dialog.
3. Select the tools option from the Crib List, and click OK.
4. To display the complete part:
 - Click the Rotate View , menu button to display the View menu:

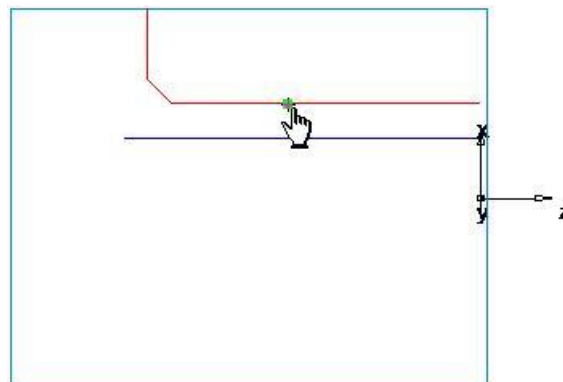


- Click Center All .



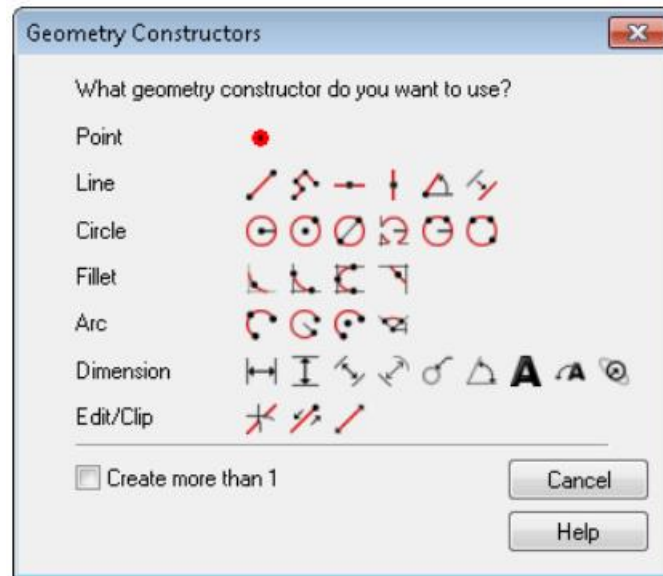
1.3.5 Defining the geometry


This shows you how to design your part.



1. Draw two lines:

- Click the Geometry  step in the Steps panel. This displays the Geometry Constructors dialog.



- Select the Create more than 1 option, and click the Line from two points  button. This displays the Feature/Geometry Edit bar.

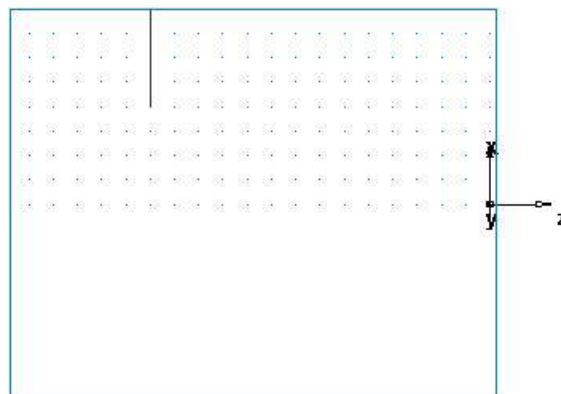


- Create two lines that define the outer profile, in the Feature/Geometry Edit bar:

For point 1, enter an XYZ 1 of X 2 (50 mm), Y 0, Z -3.5 (-88 mm).

For point 2, enter an XYZ 2 of X 1 (25 mm), Y 0, Z -3.5 (-88 mm).

Press Enter. This draws a line in the graphics window.

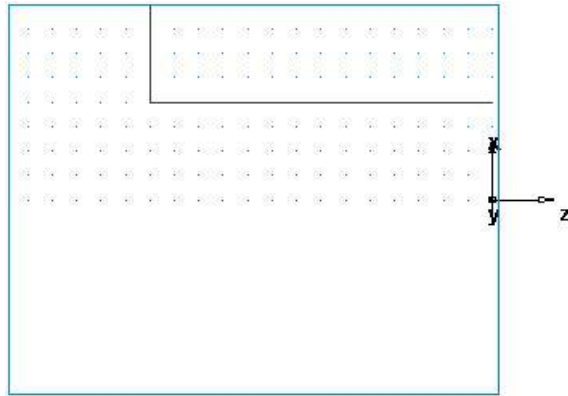


- Create a second line:


For point 1 enter an XYZ 1 of X 1 (25 mm), Y 0, Z -3.5 (-88 mm).


For point 2 enter an XYZ 2 of X 1 (25 mm), Y 0, Z 0.

Press Enter to create a second line.



2. Create a chamfer to trim your lines.

Click the Geometry  step in the Steps panel.

– In the Geometry Constructors dialog, in the list of Fillet options click the Chamfer  button.

– In the Feature/Geometry Edit bar, enter:

– A width of 0.25 (6 mm).

– A height of 0.25 (6 mm).


– Position your mouse pointer close to the chamfer location. The chamfer snaps into place.



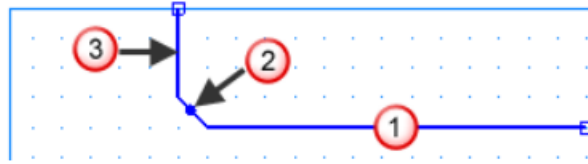
– Click to place the chamfer on your drawing. The chamfer automatically trims your lines.

3. To turn the part you need to convert these three individual lines into a single curve (chain the curve).

– Select the Curves  step from the Steps panel.


– In the Curves Creation dialog, select the Pick Curve Pieces  button.


– In the graphics window, click locations 1, 2 and 3. Each line segment changes color when selected.



– In the Feature/Geometry Edit bar, name the curve turn, and press Enter.

4. Create third line which you will use to create a Bore feature.

– Click the Geometry  step in the Steps panel.

– In the Geometry Constructors dialog, click the Line from two points  button.

– In the Feature/Geometry Edit bar:


– For point 1 enter an XYZ 1 of X 0.625 (16 mm), Y 0, Z 0.

– For point 2 enter an XYZ 2 of X 0.625 (16 mm), Y 0, Z -3.75 (-94 mm).

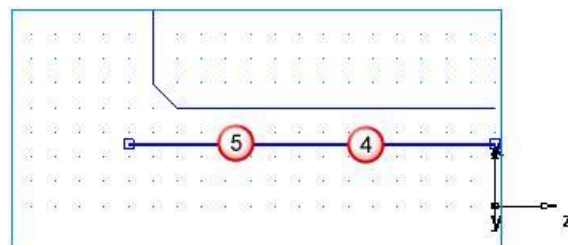
– Press Enter.

5. To chain the bore curve:

– Select the Curves  step from the Steps panel.

– In the Curves Creation dialog, select the Pick Curve Pieces  button.

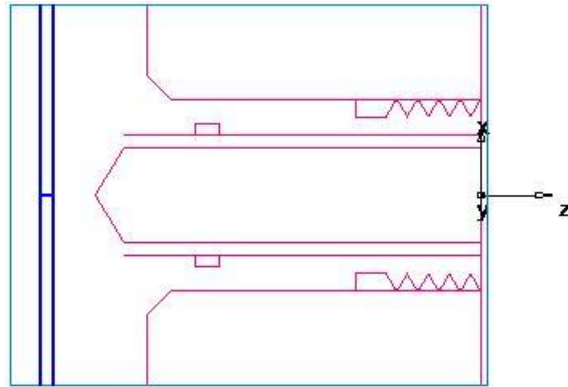
– In the graphics window, click locations 4 and 5 (you select the same line twice).




In the Feature/Geometry Edit bar, name the curve bore, and press Enter.

1.3.6. Creating the features

This shows you how to create the turning features.




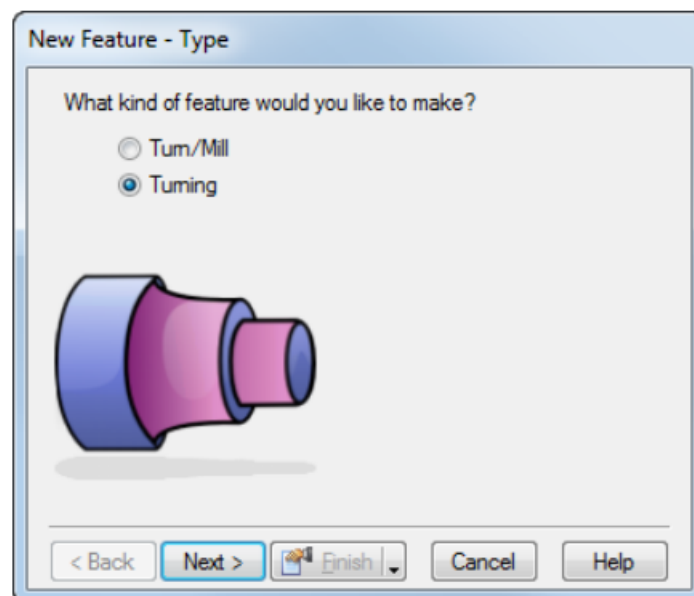
1. Select the 2D Turned Profiles  button, on the Display Mode toolbar, to switch to a simplified 2D representation of the part.




✓ To open the Display Mode toolbar, select the View > Toolbars menu option, select the Display Mode option, then click OK.

2. Create a Turn feature.

- Click the Features  step in the Steps panel.
- If you have the Turn/Mill module, the New Feature wizard asks you which type of feature you want to create. Select the Turning option, and click Next.



- Select Turn in the From Curve section, and click Next.
- In the Curve field select turn from the list.

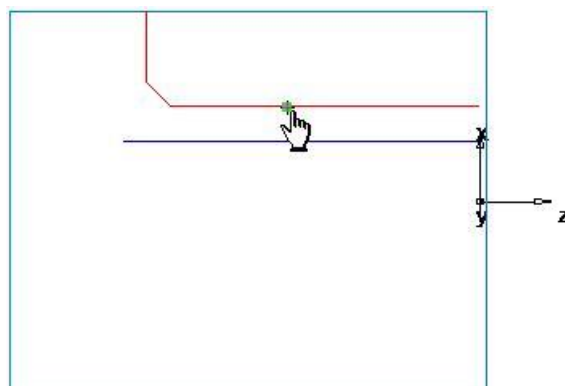
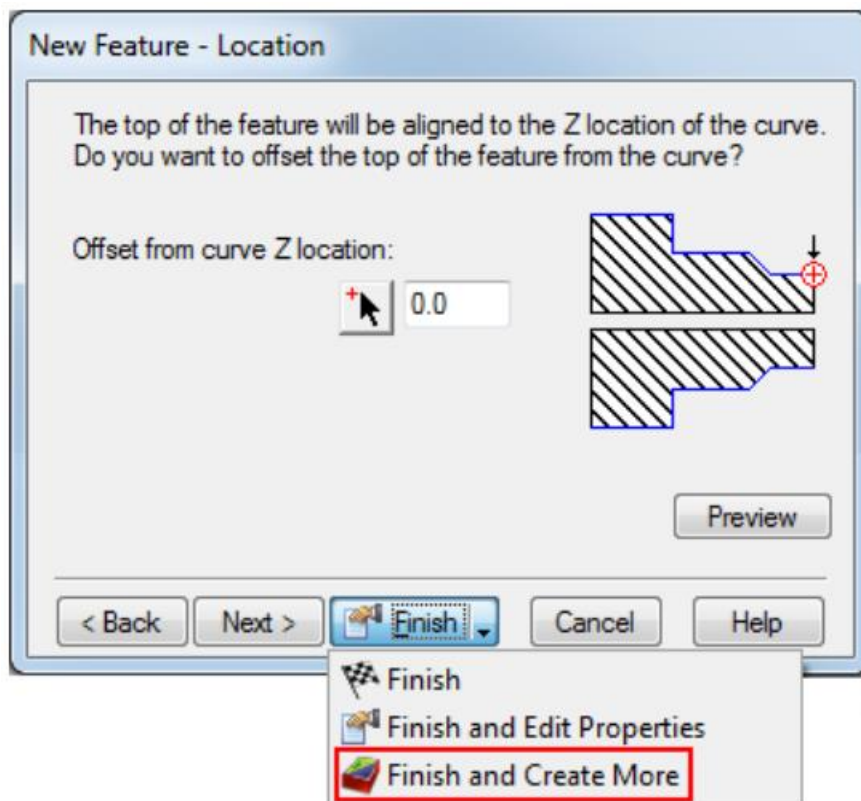
Click the Pick Curve  button to select the curve graphically. The dialog minimizes to reveal the graphics window beneath.

Click the curve you named turn earlier.

In this particular case, two objects are available for selection: a line and a curve. Whenever your selection needs to be clarified, FeatureCAM opens the Select dialog.

In the Select dialog, select turn, and click OK.

– From the Finish menu button, select the Finish and Create More option to continue creating features.



3. Create a Face feature.

- In the New Feature wizard, select the Turning option, and click Next.
- In the From Dimensions frame, select Face, and click Next.
- On the Dimensions page:

Enter a Thickness of 0.0625 (1.5 mm).

Enter an Outer Diameter of 4 (100 mm).

Enter an Inner Diameter of 0.

Click Next.

- Click Finish and Create More.

4. Create a Hole feature.

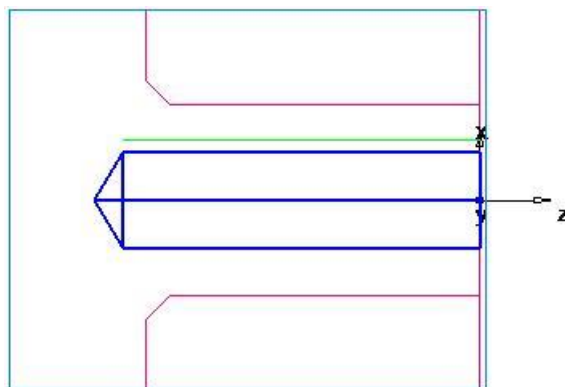
- In the New Feature wizard, select the Turning option, and click Next.
- In the From Dimensions frame, select Hole, and click Next.
- On the Dimensions page:

Enter a Depth of 3.75 (94 mm).

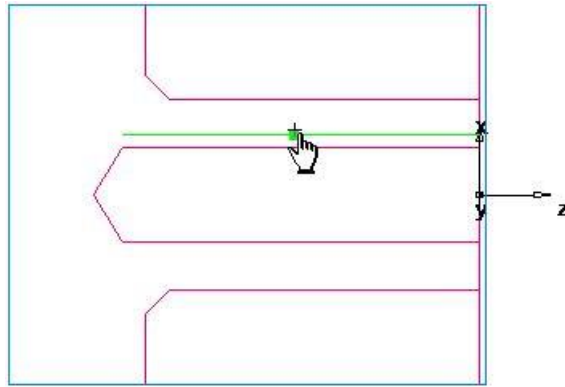
Enter a Diameter of 1.0 (24 mm).

Click Next.

- On the Location page enter a Z of 0.
- Click Finish and Create More.



5. Create a Bore feature by using the same process you used to create the Turn feature. Use the curve named Bore.



6. Create a Groove feature.

- In the New Feature wizard, select the Turning option, and click Next.
- In the From Dimensions frame select Groove, and click Next.
- On the Dimensions page:

Select a Location of ID.

Select an Orientation of X axis.

Enter a Diameter of 1.25 (31 mm).

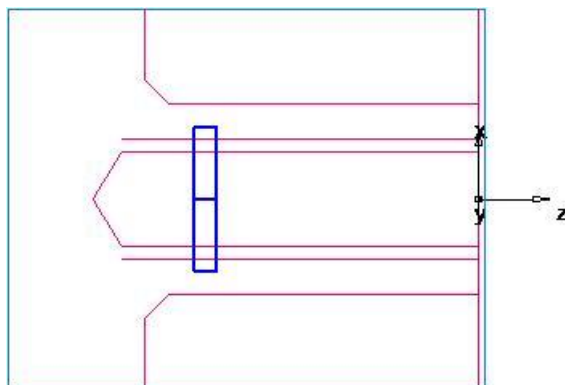
Enter a Depth of 0.125 (3 mm).

Enter a Width of 0.25 (6 mm).

Leave the other settings at 0.

Click Next.

- On the Location page enter a Z of -3 (-75 mm).
- Click Finish and Create More.



7. Create a Thread feature.

- In the New Feature wizard, select the Turning option, and click Next.
- In the From Dimensions frame, select Thread, and click Next.
- On the Dimension page:

Select Get the thread dimensions from a standard thread. Select OD.

In the Designation field select the 2.0000- 4.5 UNC (M50-15 for metric).

Click Next.

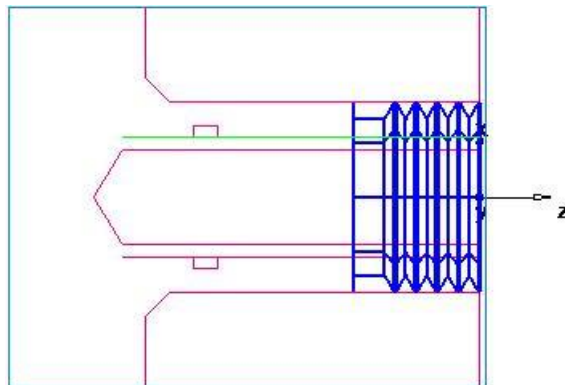
– On the Dimensions page:

Select a Thread of Right hand.

Enter a Thread Length of 1.0 (24 mm).

Click Next.

– Click Finish and Create More.



8. Create a Cutoff feature.

– In the New Feature wizard, select the Turning option, and click Next.

– In the From Dimensions frame select Cutoff, and click Next.

On the Dimensions page:

Enter a Diameter of 4 (100 mm).

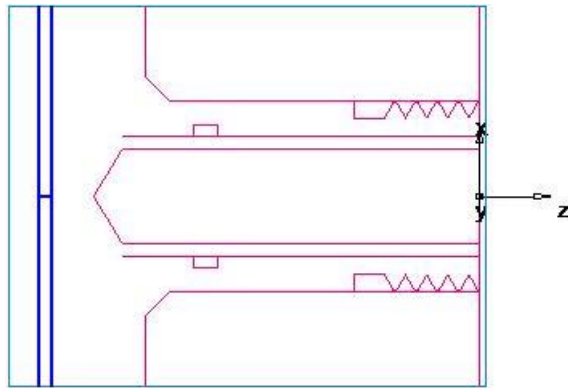
Enter an Inner Diameter of 0.

Enter a Width of 0.122 (3 mm).

Click Next.

– On the Location page enter a Z of -4.5 (-112 mm).


– Click Finish.



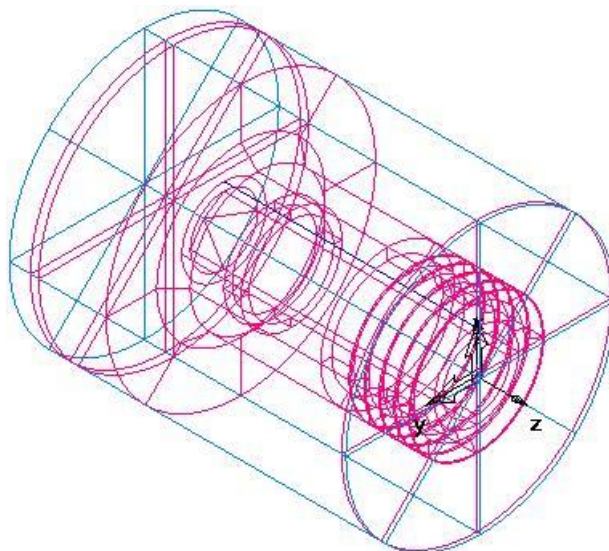
1.3.7. Viewing the part

To look at the part in a different orientation you can select one of the standard predefined views. These options are available from the Standard toolbar:




1. To return to a 3D view of the model, click the 2D Turned Profiles  button, on the Display Mode toolbar.

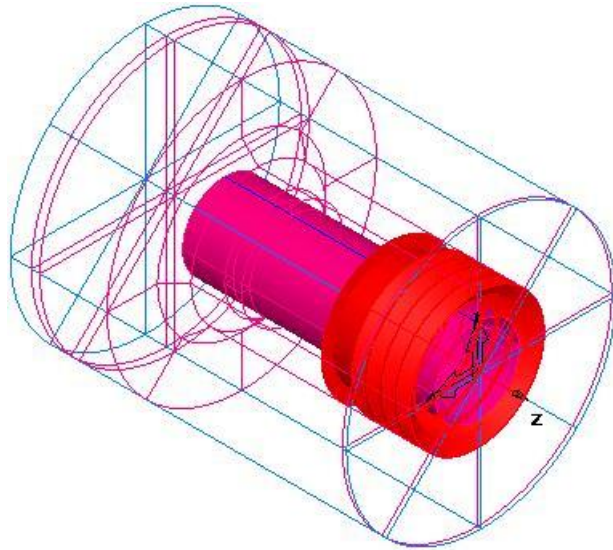
2. Click the Isometric View  button on the Standard toolbar.




3. Shade the part.

- Open the Part View panel, and select bore1 under the Setup1 node.
- Click the Shade Selected  button on the Display Mode toolbar.
- Select thread1 in the Part View panel.

- Click the Shade Selected  button again.




4. Click the Unshade All  button on the Display Mode toolbar to return to the wireframe view.

1.3.8. Simulating the toolpaths



Now you have created the features, FeatureCAM automatically:

- Selects the most appropriate tools and operations;
- Recommends machining strategies;
- Calculates speeds and feeds;
- Generates toolpaths and creates the NC code.

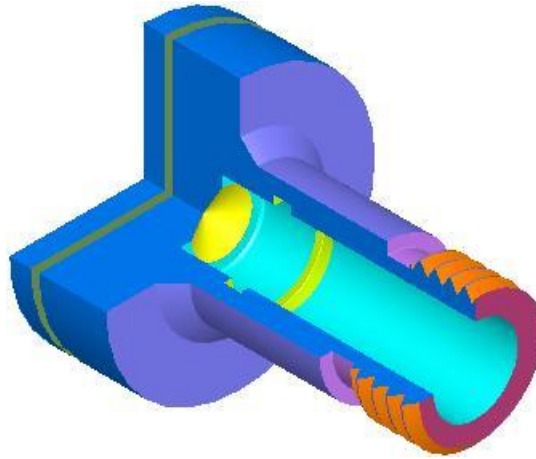
To view the simulated toolpath:


1. Click the Toolpaths  step in the Steps panel to display the Simulation toolbar.



2. On the Simulation toolbar, select the 3D Simulation  option, and then click Play  to start the simulation. If the Automatic Ordering Options dialog displays, click OK to close it. This accepts the default ordering options.


This displays a solid 3D rendering of the cutting process. By default, the 3/4 view is shown when cutting or drilling the ID of the part.



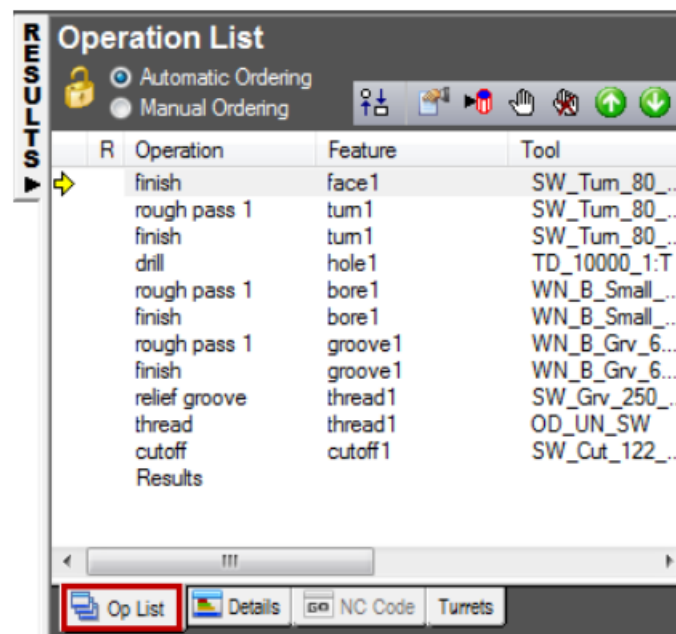
✓ If the 3/4 view is not displayed, select Options > Simulation > Round Stock from the menu, select the 3/4 view with lathe ID work option, then click OK to close the dialog. Click the Play  button on the Simulation toolbar to see the changes.

3. Click the Play to Next Operation  button. This displays the face operation.

4. Repeat step 3 to view each operation until the whole part is cut.

5. Click Eject . This removes the simulation.

1.3.9. Order of manufacturing operations



The Op List tab in the Results window shows all of the operations needed to machine the features. A yellow warning sign next to an operation indicates a

potential problem with that operation. In this case, if you see any warnings ignore them.


You can control the automatic ordering of operations by using either rules or operation templates. We will consider this question in more detail in laboratory work №3.

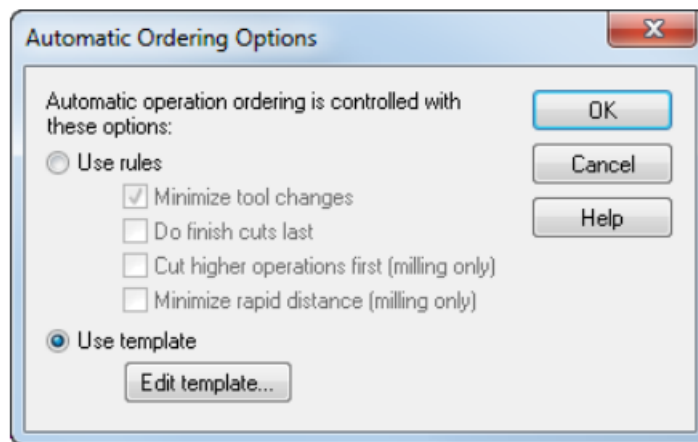
This section changes the automatic ordering by modifying the Turn Operation template.

To modify the template

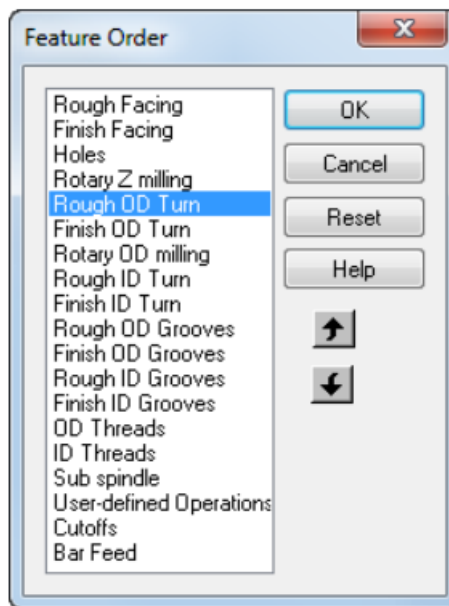
1. Select the Automatic Ordering option on the Op List tab. This ensures the automatic ordering rules are applied to the operations.


2. Change the automatic ordering to group together the operations which use the same tool.

- Click the Ordering Options  button.
- In the Automatic Ordering Options dialog, select Use template.





- Click Edit template.
3. Click Edit template.




- Select Rough OD Turn.
 - Click  until Rough OD Turn is below Finish ID Turn.
 - Click OK to close the Feature Order dialog.
4. Click OK to close the Automatic Ordering Options dialog.
 5. Simulate the part.

- Select the Toolpaths  step from the Steps panel. This displays the Simulation toolbar.

- Click the 3D Simulation  button, and then click the Play  button to start the simulation.

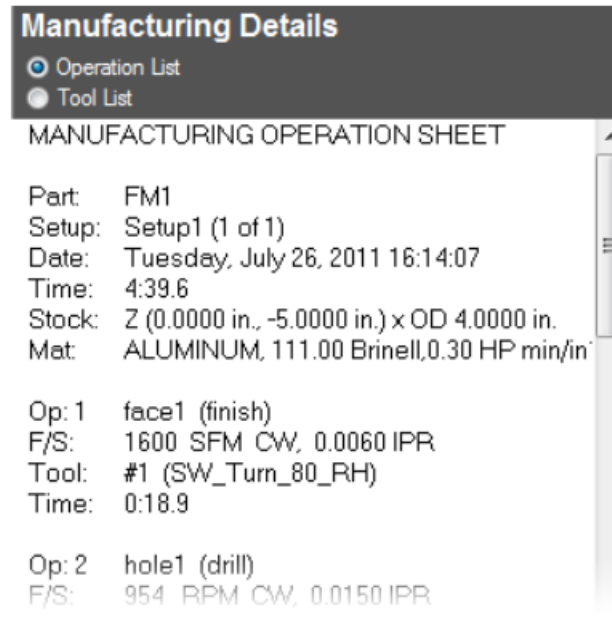
Notice that the OD roughing and finishing now happen after the hole is drilled.

- Click the Stop  button when simulation is complete to exit simulation mode.

1.3.10. Part documentation (Turning)

As well as simulating the manufacturing of the part, the simulation also generates complete tool and operations lists. The tools selected are based on your tool database. You can print all of this information for use as an operator's checklist.

1. Click the Details tab in the Results window to display the Manufacturing Operations sheet.




You can review this sheet using the scroll bars.

2. Select the Tool List option at the top of the Details tab to show the Manufacturing Tool Detail sheet. It contains all of the tools used to create the part based on the tool crib you selected.

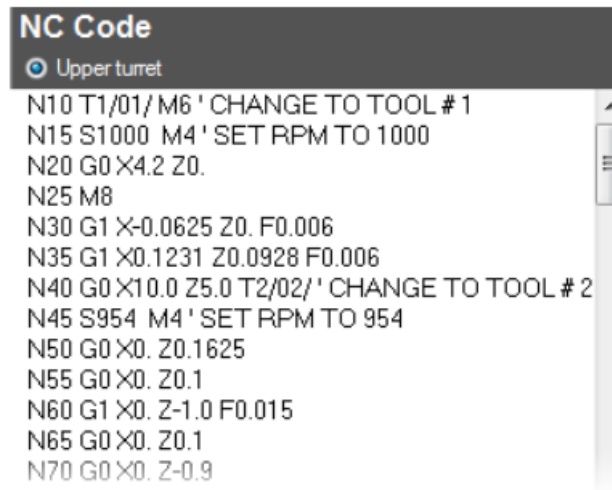
✓ You can print this documentation from the File > Print menu option.

1.3.11. Generating NC code (Turning)

FeatureCAM generates the NC code to manufacture parts on a CNC machine. You can generate NC code after you have simulated the part, and therefore calculated the toolpaths.

1. Select the NC Code  step from the Steps panel. This displays the NC Code dialog.

2. Click the Display the NC Code  button to generate the NC code.



```

NC Code
Upper turret
N10 T1/01/ M6 ' CHANGE TO TOOL # 1
N15 S1000 M4 ' SET RPM TO 1000
N20 G0 X4.2 Z0.
N25 M8
N30 G1 X-0.0625 Z0. F0.006
N35 G1 X0.1231 Z0.0928 F0.006
N40 G0 X10.0 Z5.0 T2/02/' CHANGE TO TOOL # 2
N45 S954 M4 ' SET RPM TO 954
N50 G0 X0. Z0.1625
N55 G0 X0. Z0.1
N60 G1 X0. Z-1.0 F0.015
N65 G0 X0. Z0.1
N70 G0 X0. Z-0.9

```

1.3.12. Changing the post processor

To change the post processor:

1. Choose Processing > Postprocessor from the menu. The Post-processor dialog will open.

2. Select Manufacturing > Post Process from the menu. This displays the Post Options dialog.

Click Browse to view available post processors. The default folder for posts is C:\Program Files\Delcam\FeatureCAM\Examples\Posts.

3. Select your post processor and click Open.

The new post processor is displayed in the CNC File field.


4. Click OK to exit the Post Options dialog and use the new post processor; click Cancel to exit the dialog and keep the original post processor.

5. Select the Toolpaths  step from the Steps panel.

6. Run a simulation of the part to regenerate the NC code.

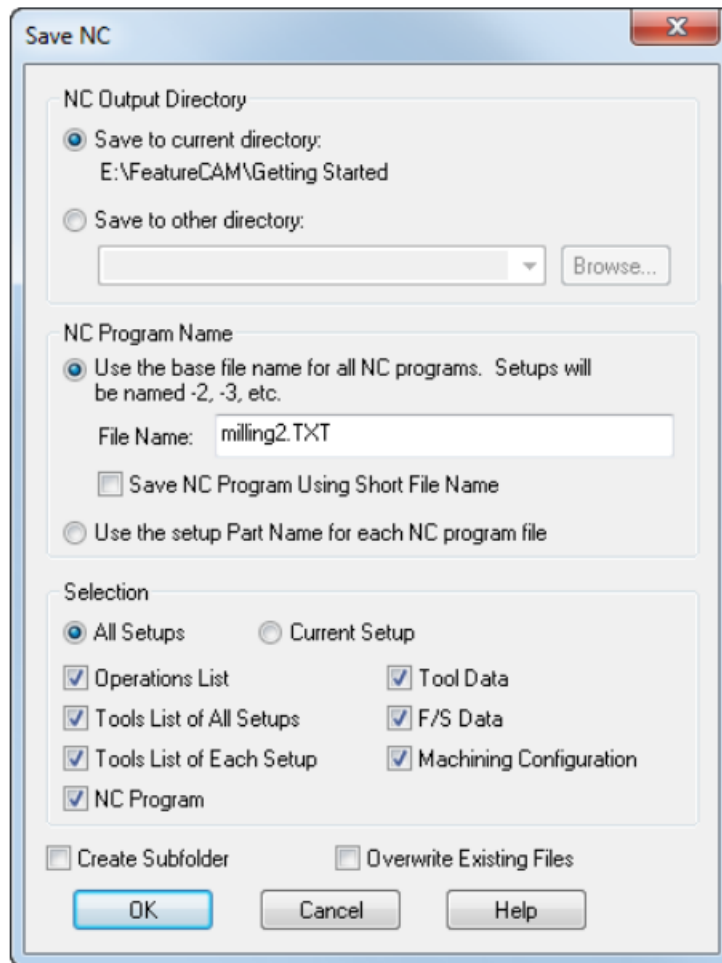
1.3.13. Saving the NC code

To save an NC program:

1. Select the NC Code  step from the Steps panel. This displays the NC Code dialog.

2. Click the Save NC  button in the NC Code dialog.

3. In the Save NC dialog, accept the default filename and folder, and click OK.



1.4 Tasks for the laboratory session

According to the individual option, in the Delcam FeatureCAM system, create the necessary turning elements, determine the list and sequence of operations, and generate the control program code for the CNC machine tool.

The main goal of this task is to study the main commands of the "Geometry", "Curves", "Elements" and "Trajectories" toolbar. In the process of performing the work, the student must learn to create elementary contours and consider all possible turning elements and methods of their determination. Be able to set up simulation of part processing and generate control program code.

Variants of tasks are given in Appendix A, p. 90.

1.5 Requirements for a report on the performance of a laboratory session

The report should contain the following sections:

1. Topic and purpose.
2. Individual task.
3. Development of the control program for turning:
 - appearance and dimensions of the workpiece;
 - elements and contours for generating the control program;
 - operations and their main characteristics;
 - control program code;
 - part model.
4. Conclusions from the laboratory session.

1.6 Control questions

1. Describe the main elements of the DELCAM FeatureCAM interface.
2. Tool database. Methods of creating and using a database at one or more workplaces.
3. Determine the methods of entering coordinates when constructing geometric primitives, pay attention to the peculiarities of using each of the methods.
4. Describe the commands available on the Geometry toolbar and how to use them.
5. Options for creating path elements, list the possible elements for a turning operation.
6. Postprocessor - tasks and choices.

LABORATORY LESSON №2

TURNING AND MILLING IN DELCAM FEATURECAM

2.1 Goal

Explore the capabilities of the DELCAM FeatureCAM system when combining turning and milling operations. Learn to create control programs for turning and milling single parts in one facility.

2.2 An example of creating a typical turning-milling part

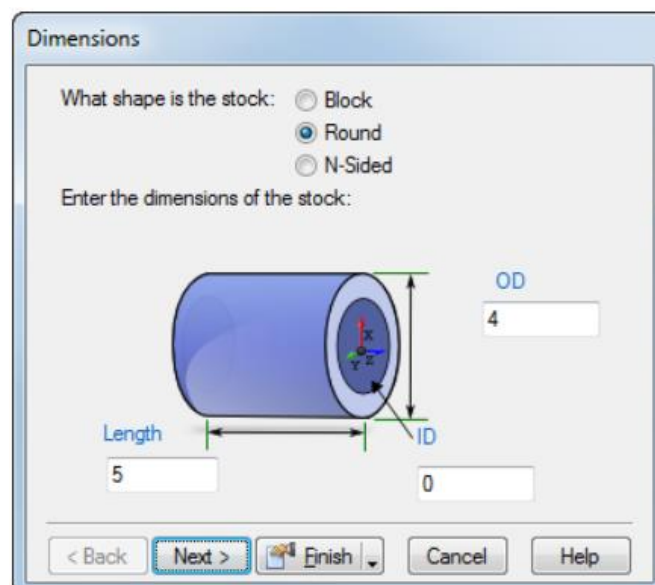
This example introduces students to the creation of parts for lathes with the possibility of milling. Methods of combining turning and milling operations and creating milling elements on the end of the part and the outer diameter.

Entering the system and setting the initial parameters is carried out according to the method given in laboratory work No. 1, however, as the type of processing, you must select Turning/Milling, the units of measurement are inches.

2.2.1. Preparatory steps

The preparatory steps define the stock and determine the coordinate system and view.

1. On the Dimensions page of the Stock wizard:



Select a shape of Round.

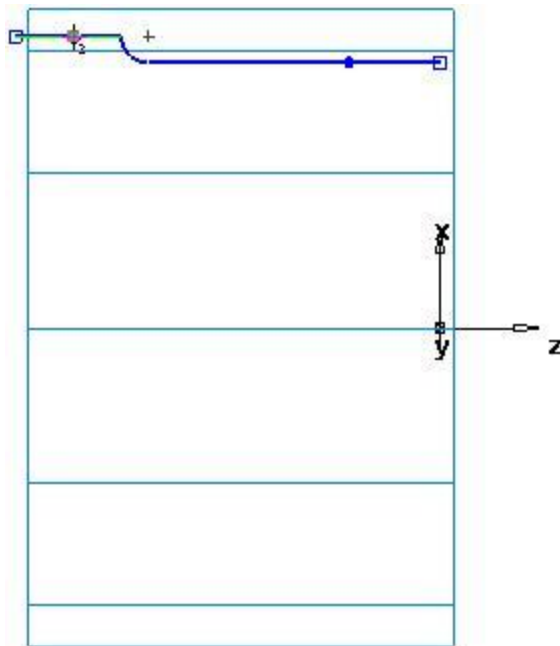
- Enter an OD (outside diameter) of 3.
- Enter a Length of 2.
- Enter an ID (inside diameter) of 0.
- From the Finish menu, select the Finish button.

2. From the View  menu select Center All .

3. Select Options > Turning Input Modes > Diameter (DZ) from the menu to enter coordinates as Diameter and Z values.

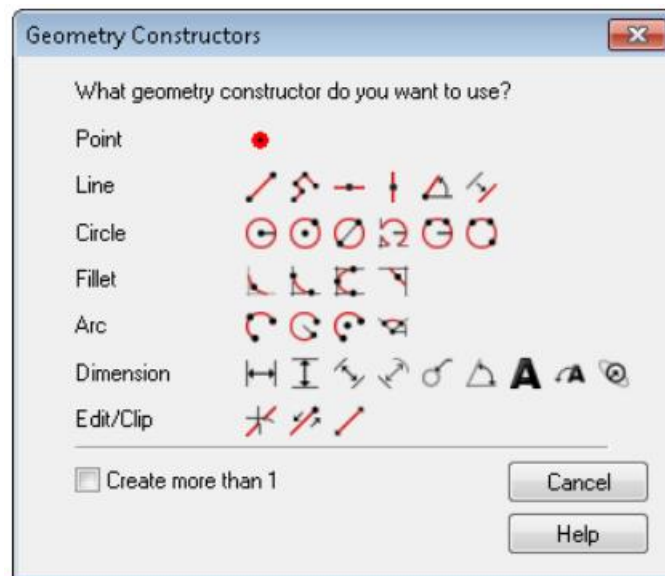
2.2.2. Defining the geometry


This shows you how to design your part.



1. Draw three lines:

- Select the Geometry  step from the Steps panel. This displays the Geometry Constructors dialog.



Select the Create more than 1 option, and click the Connected Lines  button. This displays the Feature/Geometry Edit bar.

– To create two lines that define the outer profile, in the Feature/Geometry Edit bar:

For point 1 enter a D/Z 1 of D 2.5, Z 0.

For point 2 enter a D/Z 2 of D 2.5, Z -1.5.

Press Enter to create a line.

– Create a second line with the values:

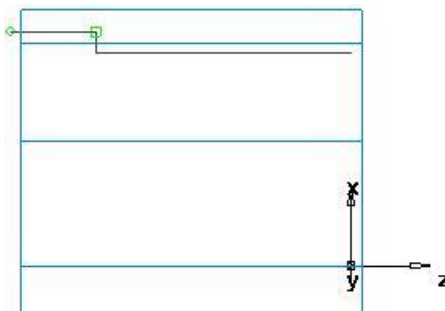
For point 2 enter a D/Z 2 of D 2.75, Z -1.5.

Press Enter to create a second line.



– Create a third line with the values:

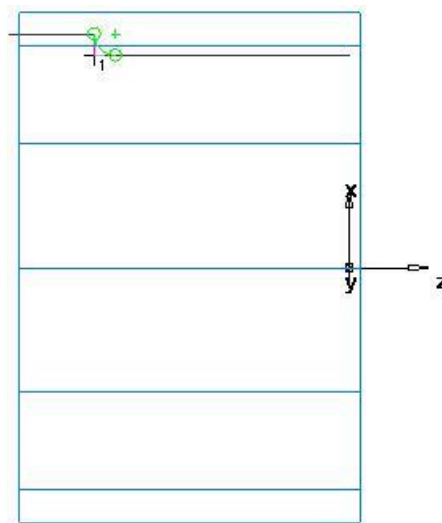
For point 2 enter a D/Z 2 of D 2.75, Z -2.

Press Enter to create a third line.





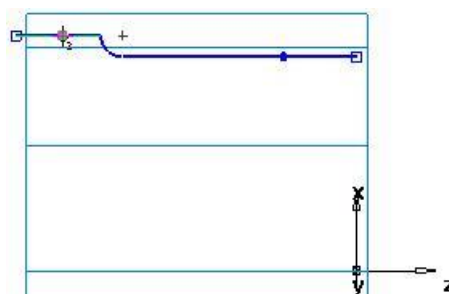
2. Create a Fillet to trim your lines.

- Select the Geometry  step from the Steps panel.
- In the Geometry Constructors dialog, in the list of Fillet options click the Corner Fillet  button.
- In the Feature/Geometry Edit bar, enter a radius (R) of 0.125.
- Position your mouse pointer in the corner between the first and second lines, and click to create the fillet. The fillet automatically trims your lines.



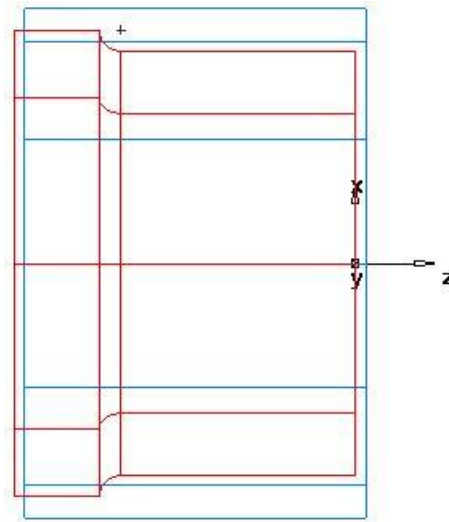
3. To turn the part you need to chain the curves.


- Select the Curves  step from the Steps panel.
- In the Curves Creation dialog, select the Pick Curve Pieces  button.
- In the graphics window, click the first line and then the third line.
- In the Feature/Geometry Edit bar, name the curve Turn, and press Enter.



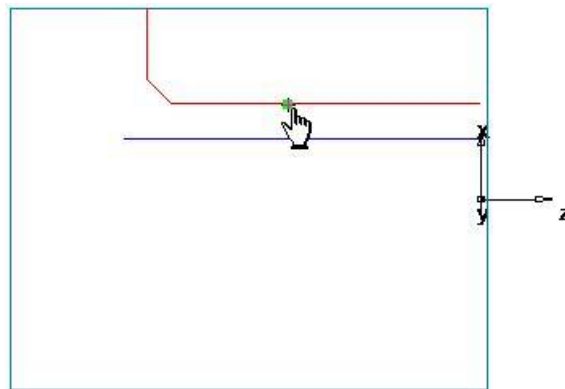
2.2.3. Creating the features


This example shows you how to create the turning features.




1. Click the 2D Turned Profiles  button, on the Display Mode toolbar, to switch to a 3D representation of the part.

2. Create a Turn feature.



- Click the Features  step in the Steps panel.
- In the New Feature wizard, select the Turning option, and click Next.
- Select Turn in the From Curve section, and click Next.
- In the Curve field select turn from the list.


Click the Pick Curve  button to select the curve graphically. The dialog minimizes to reveal the graphics window beneath.

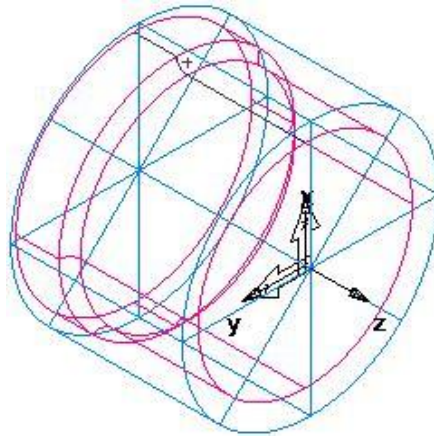
Click the curve you named Turn earlier.


In the Select dialog, select turn, and click OK.

- Click Finish.

2.2.4. Viewing the part

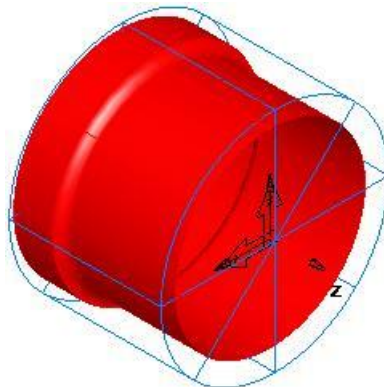
1. On the Standard toolbar select the Isometric View_  button.





- ✓ If this displays a 2D representation of the part, click the 2D Turned Profiles  button, on the Display Mode toolbar.

2. Shade the part:

- Open the Part View panel, and select turn1 under the Setup1 node.
- Click the Shade Selected  button on the Display Mode toolbar.

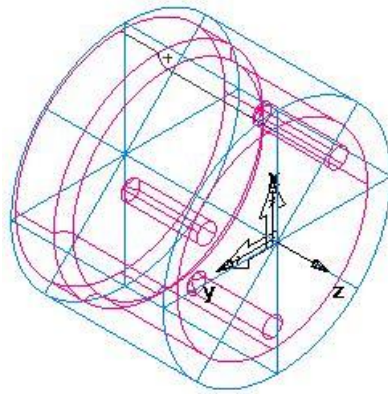


- Click the Unshade All  button on the Display Mode toolbar to return to the wireframe view.

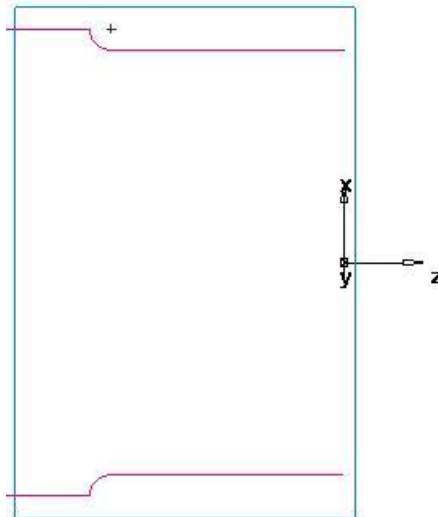
3. To change the view to a top view; from the Principal View  menu button, click the Top  button.

2.2.5. Creating three radial holes on the face


This shows you how to add three Holes to the part.



1. To return to a 2D view of the model, click the 2D Turned Profiles button, on the Display Mode toolbar.



2. Create a Hole.

- Click the Features  step in the Steps panel.
- In the New Feature wizard, select the Turn/Mill option, and click Next.
- In the From Dimensions field, select Hole and click Next.
- In the Dimensions dialog:
 Enter a Chamfer of 0.0.
 Enter a Depth of 1.0.
 Enter a Diameter of 0.25.
- Click Finish and Create More.

3. Create a Pattern from feature:

In the New Feature wizard, select the Turn/Mill option, and click Next.

- In the From Feature field, select Pattern, and click Next.
- Select the hole you just created and click Next.
- Select Radial in the setup XY plane, and click Next.
- On the Pattern - Dimensions page:

Enter a Number of 3.0.



Enter a Diameter of 2.0.

Enter a Spacing Angle of 120.

Enter an Angle of 60.

- Click Finish.
- Click Cancel.

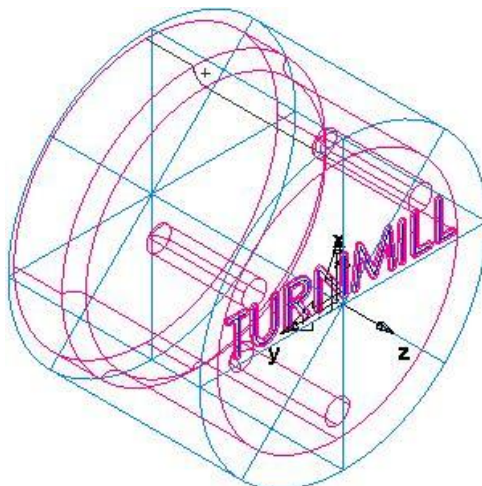
4. View the 3D wireframe representation of the part:

- Click the 2D Turned Profiles  button, on the Display Mode toolbar, to switch to a 3D representation of the part.
- Click the Isometric View  button on the Standard toolbar.



2.2.6. Engraving the face

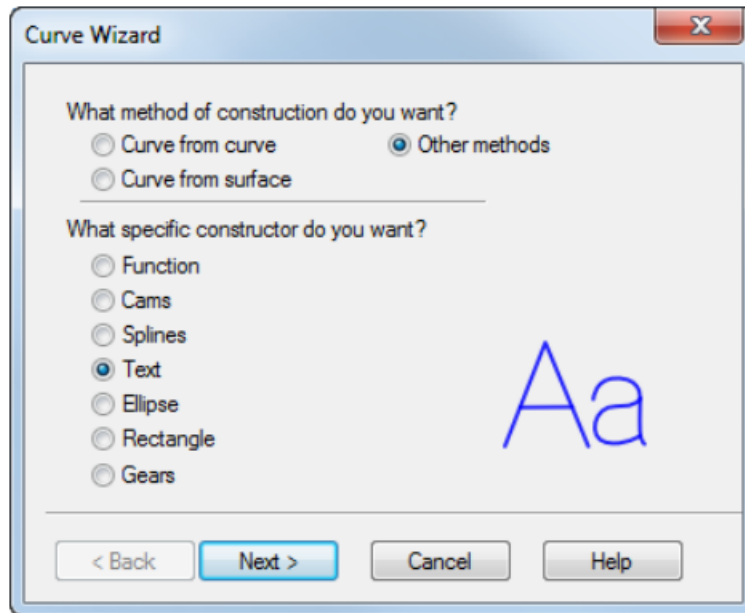
This shows you how to engrave the part by:

- Creating the engraving text.
- Creating a Groove feature.



1. Create a curve.

- Select the Curves  step from the Steps panel.
- In the Curves Creation dialog, select the Curve Wizard  button.
- In the Curve wizard:

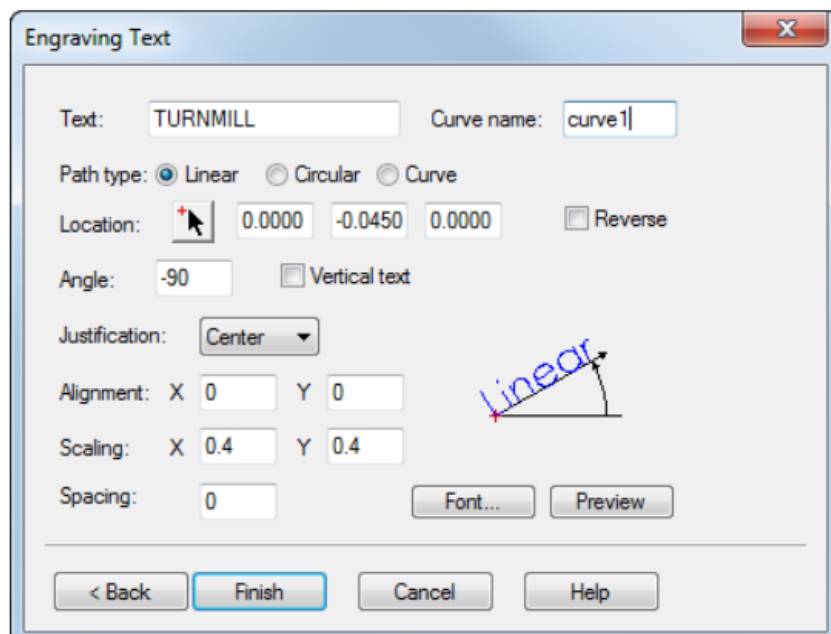


Select Other methods as the construction method.

Select Text as the constructor.

Click Next.

- On the Engraving Text page, configure the text properties.



Enter a Text of TURNMILL.

Select a Path type of Linear.

Enter a location of X 0.0, Y -0.045, Z 0.0.

Enter an Angle of -90.

From the Justification list, select Center.

Enter a Scaling of X 0.4, Y 0.4.

Click the Font button to display the Font dialog.

From the Font list, select Machine Tool Gothic.

Enter a Size of 72.

Click OK to close the dialog.

Click Finish to close the wizard.

2. Create a Groove feature.

– Select the TURNMILL text (curve1) in the graphics window.

– Click the Features  step in the Steps panel.

– In the New Feature wizard, select the Turn/Mill option, and click

Next.

– In the From Curve field, select Groove, and click Next.

– On the Curve page, click Next (as you have selected the text).

– On the Location page, click Next.

– On the Dimensions page:

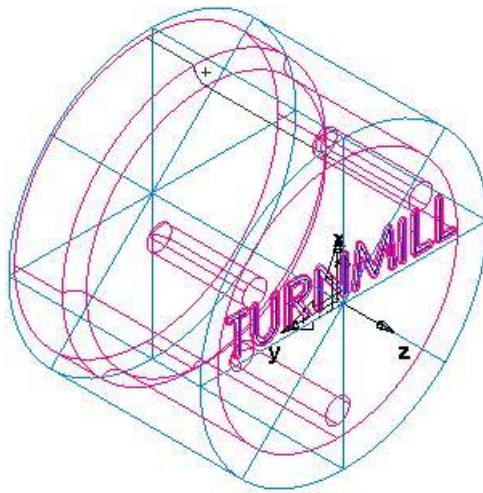
Enter a Width of 0.0625.

Enter a Depth of 0.02.

Select Face.

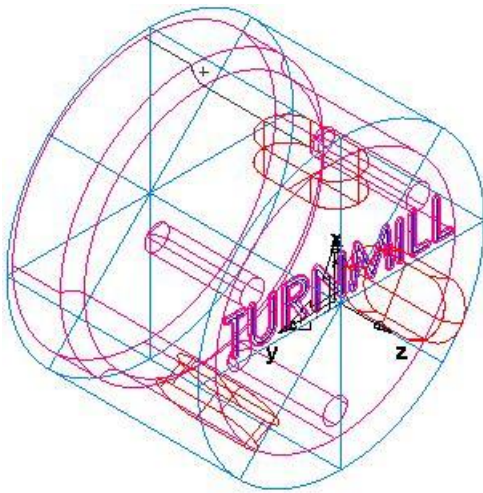
Select Simple (Engrave).

– Click Finish.




2.2.7. Creating three slots

This shows you how to add three milled slots to the part.



1. Create a Slot feature:

- Click the Features  step in the Steps panel.
- In the New Feature wizard, select the Turn/Mill option, and click Next.
- In the From Dimensions section, select Slot.
- Select Make a pattern from this feature, and click Next.
- On the Dimensions page:
 - Enter a Length of 1.0.
 - Enter a Width of 0.5.
 - Enter a Depth of 0.25.

Click Next.

– On the Patterns page select Radial around index axis, and click Next.

– On the Location page:

Enter a B Angle of 90.

Enter a Radius of 1.25.

Enter a Z to 0.25.

Click Next.

– On the Dimension page:


Enter a Number of 3.


Enter a Spacing Angle of 120.

– Click Finish.

2.8. Simulating the toolpaths

To view the simulated toolpath:


1. Click the Toolpaths  step in the Steps panel to display the Simulation toolbar..

2. Click the 3D Simulation  button, and then click the Play  button to start the simulation. If the Automatic Ordering Options dialog displays, click OK to close it. This accepts the default ordering options.



The toolpaths are accurately simulated including the part rotations.

- ✓ To simulate turning and milling paths, you must select a CNC file for a machine that supports turning with a driven tool, otherwise an error message will appear.

3. Click Eject . This removes the Simulation toolbar.

2.3 Tasks for the laboratory session

According to the individual option, in the Delcam FeatureCAM system, create the necessary elements of turning and milling processing, determine the list and sequence of the operation, and generate the control program code for the workbench.

The main goal of this task is to study additional commands of the "Geometry", "Curves", "Elements" and "Trajectories" toolbar. In the process of performing the work, the student must learn to more freely use the set of commands of the "Geometry" toolbar, create contours of various configurations with the help of the wizard. Be able to create milling elements and arrays based on them.

Variants of tasks are given in Appendix B, p. 93.

2.4 Requirements for a report on the performance of a laboratory session

The report should contain the following sections:

1. Topic and purpose.
2. Individual task.
3. Development of the turning and milling control program:
 - appearance and dimensions of the workpiece;
 - elements and contours for generating the control program;
 - operations and their main characteristics;
 - control program code;
 - part model.
4. Conclusions from the laboratory session.

2.5 Control questions

1. Methods of creating roundings and chamfers on the contour of the part.
2. What milling elements can be created without creating a contour.
3. Techniques for creating and arranging arrays of milling elements.
4. Define the concept of contour. Justify the necessity of their creation.

Types of contours, methods of their creation.

LABORATORY LESSON №3

2.5D MILLING IN DELCAM FEATURECAM

3.1 Goal

Explore the possibilities of DELCAM FeatureCAM for creating elements of parts that are processed in the 2.5D-milling mode. Learn to create control programs.

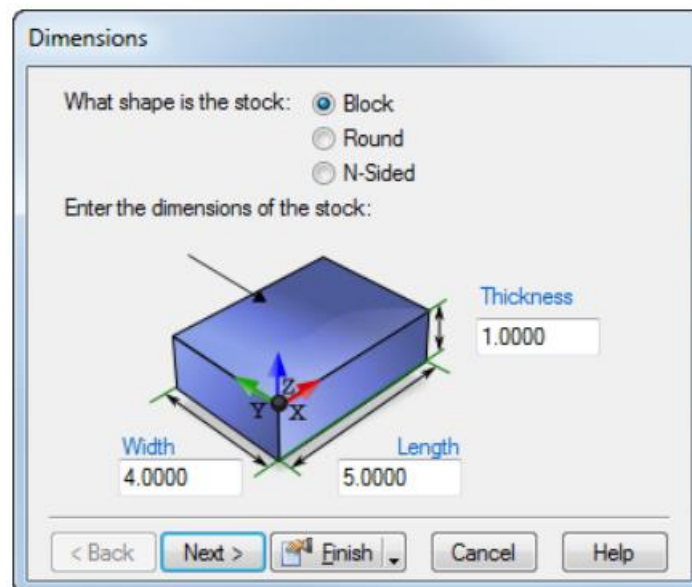
3.2 An example of creating a typical 2.5D milling part

This example shows how to create several simple elements, generate the trajectories used to process the part and display them. And also explains the main principles and methods of managing the order of operations.

Entering the system and setting the initial parameters is carried out according to the method given in laboratory work No. 1, however, as the type of processing, you must select Milling operation, the units of measurement are millimeters.

3.2.1. Defining the Stock

On the Dimensions page of the Stock wizard:

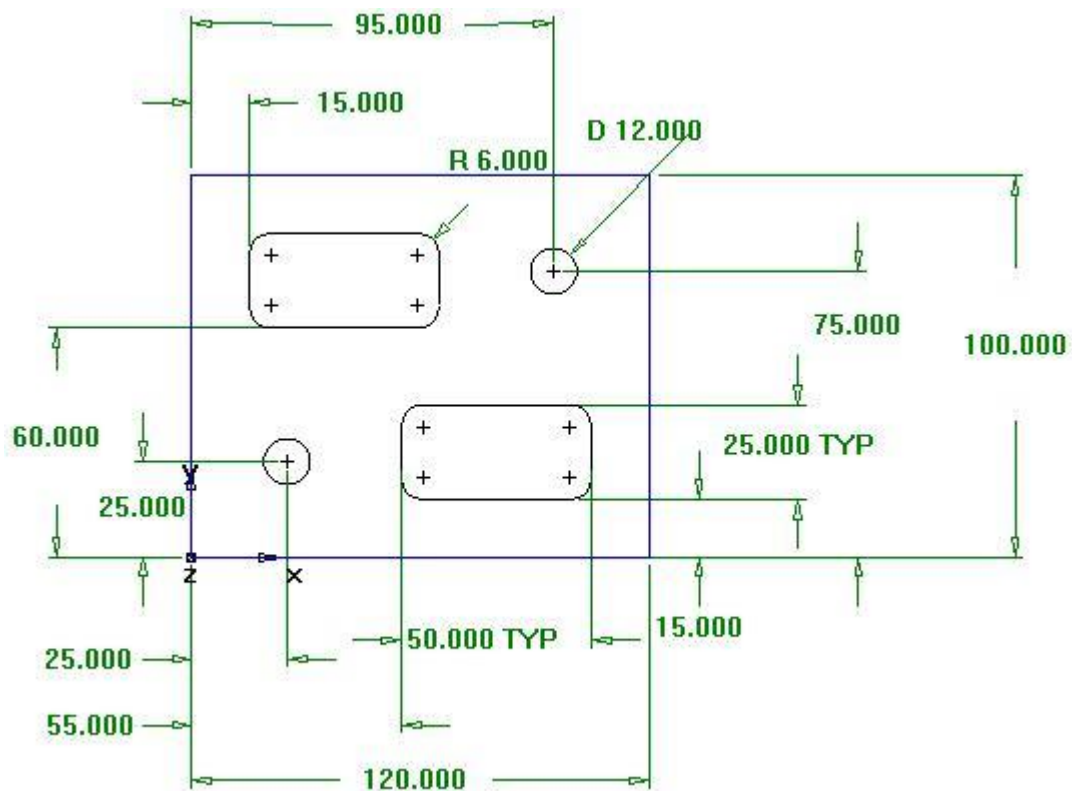


- Enter a Thickness of 1 (25 mm).
- Enter a Width of 4 (100 mm).


- Enter a Length of 5 (120 mm).
 - Click Finish.
2. Click OK to accept the default values of the Stock wizard.

3.2.2. Creating the features

This step shows how to create Hole and Rectangular Pocket features.



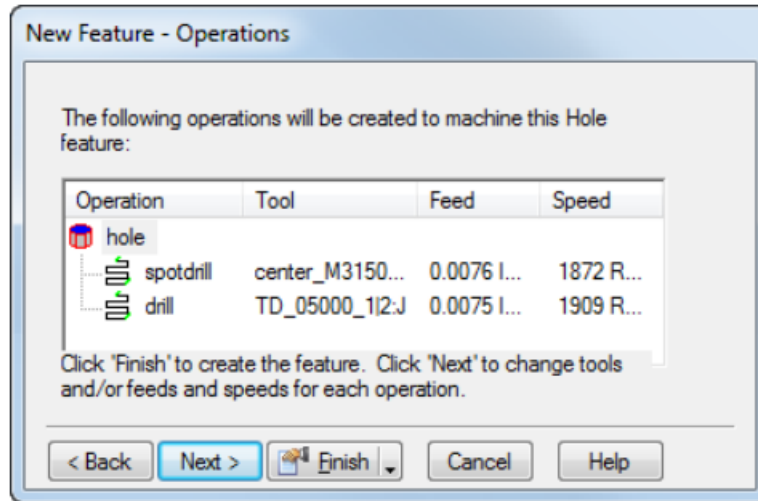
1. Create a Hole feature.

- Click the Features  step in the Steps panel.
- In the New Feature wizard, select Hole in the From Dimensions section, and click Next.
- Enter a Diameter of 0.5 (12 mm), and click Next.
- Enter a hole center location of X 1.0 (25 mm) and Y 1.0 (25 mm), and click Next.

This displays the Strategies page. This page controls the types of operations used to cut the feature. The default operations for a Hole feature are to spot drill and then drill the hole. If the Hole has a chamfer, the default is to cut the chamfer with the spot drill operation.

- Accept the default strategy settings by clicking Next.


The Operations page shows a summary of the operations to cut the feature, the automatically selected tools, and the feeds and speeds.

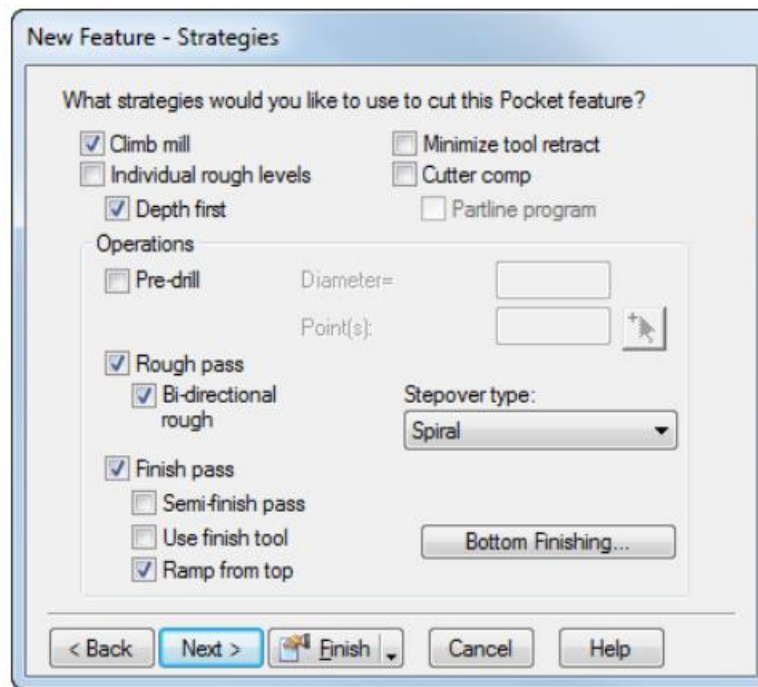


From the Finish menu button select the Finish  option.





2. Create a Rectangular Pocket feature.

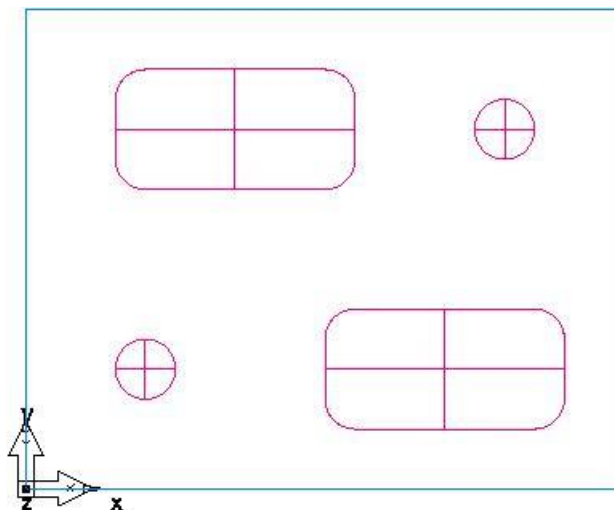
- Click the Features  step in the Steps panel.
- In the New Feature wizard, in the From Dimensions section, select Rectangular Pocket, and click Next.
- Accept the default dimensions by clicking Next.
- Enter a pocket location of X 0.75 (15 mm), and Y 2.5 (60 mm), and Z 0 (0 mm), and click Next.
- Enter a pocket location of X 0.75 (15 mm), and Y 2.5 (60 mm), and Z 0 (0 mm), and click Next.



– Click the Finish button.

3. Use the Features  step to create a second Hole with a diameter of 0.5 (12 mm), located at X=4 (95 mm) and Y=3 (75 mm).

4. Use the Features , step to create another Rectangular Pocket the same dimensions as the first, but positioned at X=2.5 (55 mm), Y=0.5 (15 mm).




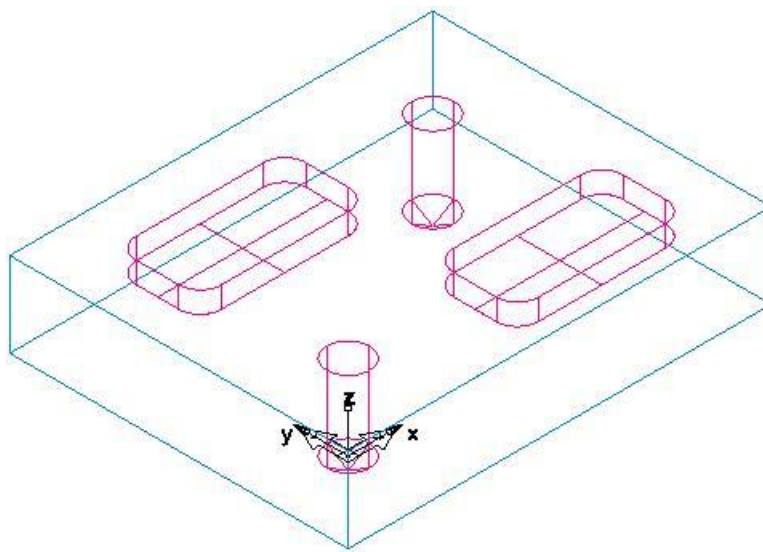
5. Select File > Save, and save the part as milling.fm.



3.2.3. Viewing the part

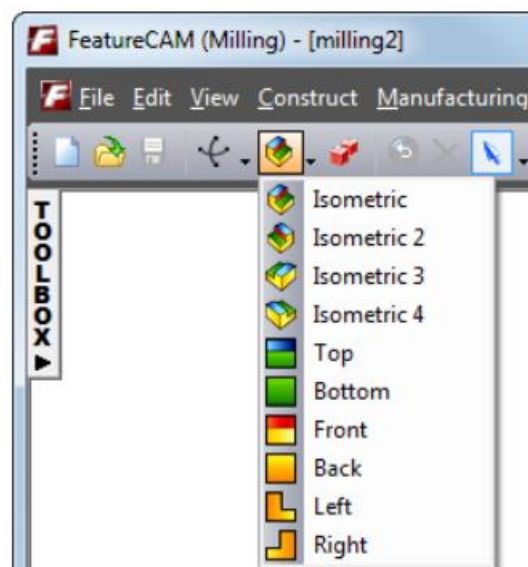
To look at the part in a different orientation you can select one of the standard predefined views. These options are available from the Standard toolbar:

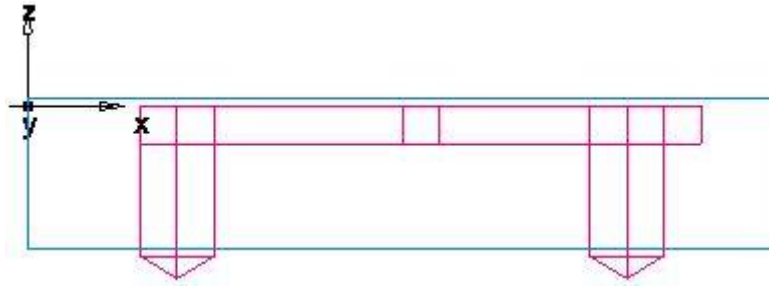



To change the view to an isometric view, click the Isometric  button on the Standard toolbar.



2. To change the view to a front view, from the Principle View  menu button, click the Front  button.






3 Click the Isometric  button to return to the isometric view.



3.2.4. Simulating the toolpaths

Now you have created the features, FeatureCAM automatically. Selects the most appropriate tools and operations; Recommends machining strategies; Calculates speeds and feeds; Generates toolpaths and creates the NC code.

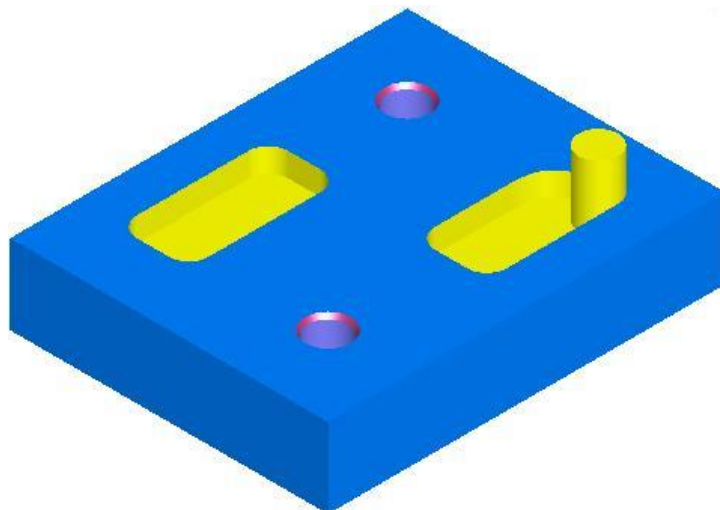
To view the simulated toolpath:


1. Click the Toolpaths  step in the Steps panel to display the Simulation toolbar.



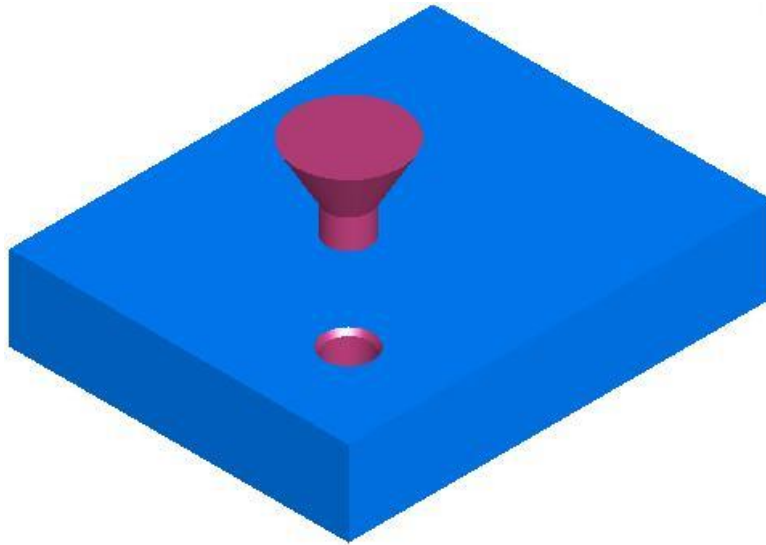
2. On the Simulation toolbar, select the 3D Simulation  option, and then click Play  to start the simulation. If the Automatic Ordering Options dialog displays, click OK to close it. This accepts the default ordering options.

This displays a solid 3D rendering of the cutting process.



- ✓ If all tools are displayed in gray in the simulation, select Options > Simulation > General from the menu, and select the Tool Colors option, then click OK to close the dialog. This displays tools in different colors so you can see which features are machined by each tool. Click the Play  button on the Simulation toolbar to see the changes.

3. Click the Play to Next Operation  button. This displays the spot drill operation.

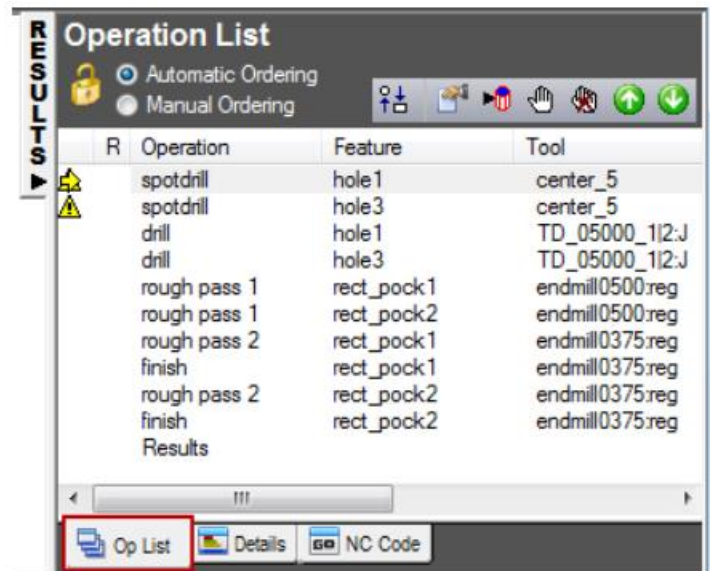


4. Repeat step 3 to view each subsequent operation until you complete the simulation.

5. Click Eject . This removes the Simulation toolbar.

3.2.5. Order of manufacturing operations

The Op List tab in the Results window shows all of the operations needed to machine the features. A yellow warning sign next to an operation indicates a potential problem with that operation. In this case, if you see any warnings ignore them.




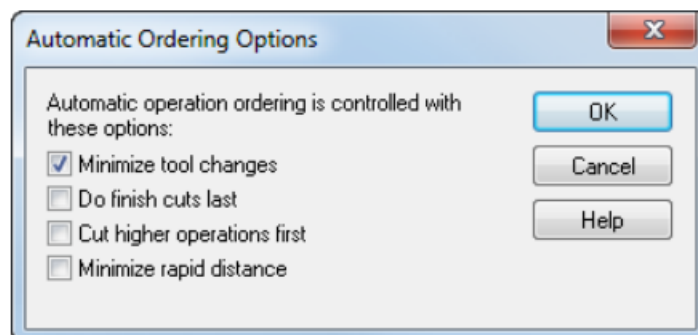
3.2.5.1 Automatic ordering operations

You can control the automatic ordering of operations by using either rules or operation templates. The turning tutorial looks at operation templates.

1. Select the Automatic Ordering option on the Op List tab. This ensures the automatic ordering rules are applied to the operations.



2. Change the automatic ordering to group together the operations which use the same tool.

- Click the Ordering Options  button.
- In the Automatic Ordering Options dialog, select Minimize tool changes, deselect everything else, and click OK.



3. Simulate the part.


- Select the Toolpaths  step from the Steps panel. This displays the Simulation toolbar.

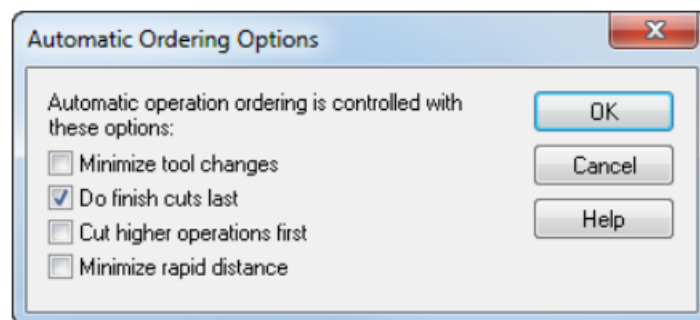
- Click the 3D Simulation  button.
- Click the Play  button.

If the Automatic Ordering Options dialog displays, click OK to close it. Notice that the simulation first performs all the spot-drills, then the drills, and then the rough and finish milling for the pockets.

- Click the Stop button when simulation is complete to exit the simulation mode.


4. Change the automatic ordering to move the finish operations to the end of the list.

- Click the Ordering Options  button.
- In the Automatic Ordering Options dialog, select Do finish cuts last, deselect everything else, and click OK.





This changes the order of operations in the Operation List.

5. Simulate the part.

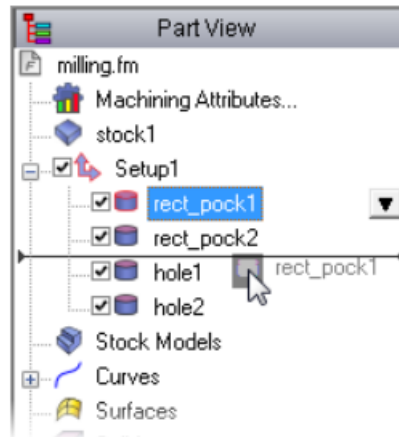
- In the Simulation toolbar, click Play . The finish cuts for the two pockets are now cut last.

- Click Stop  when simulation is complete.



6. Change the automatic ordering to match the order of the features in the Part View panel.

- Click the Ordering Options  button.
- Deselect everything, and click OK.
- Open the Part View panel by clicking on  Part View. The tree view contains all the setups and features you have created.

- Click the rect_pock2 item in the Setup1 node, and drag it up above hole2.



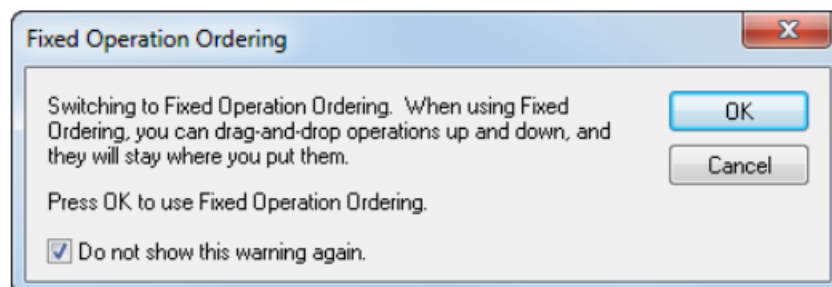
7. Simulate the part.

- In the Simulation toolbar, click Play . The second pocket is cut as the second feature.
- Click Stop .

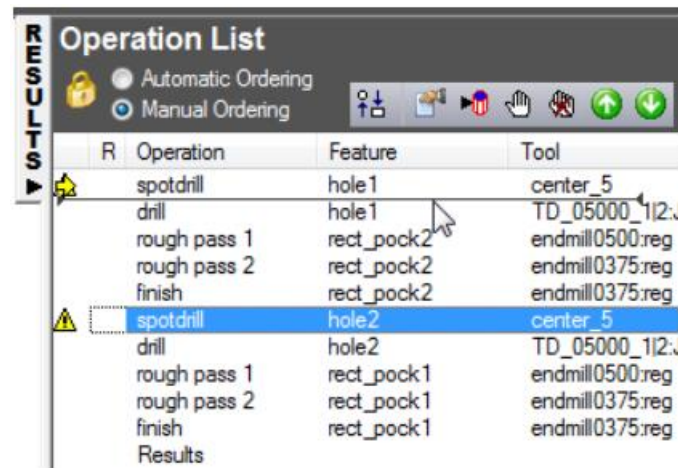
3.2.5.2 Manual ordering options

The automatic ordering of operations determined the order by a set of rules. You can also specify an exact ordering of operations manually.

1. Select the Manual Ordering option on the Op List tab.
2. In the Fixed Operation Ordering dialog, select Do Not Show This Warning Again, and click OK.



3. Select the spotdrill operation for hole2 from the list, and drag it up ahead of the drill operation for hole1.




4. Simulate the part.

- In the Simulation toolbar, click Play . The simulation performs the operations in the new order.

- Click Stop , when simulation is complete.

5. Selecting Automatic Ordering to return to automatic ordering.

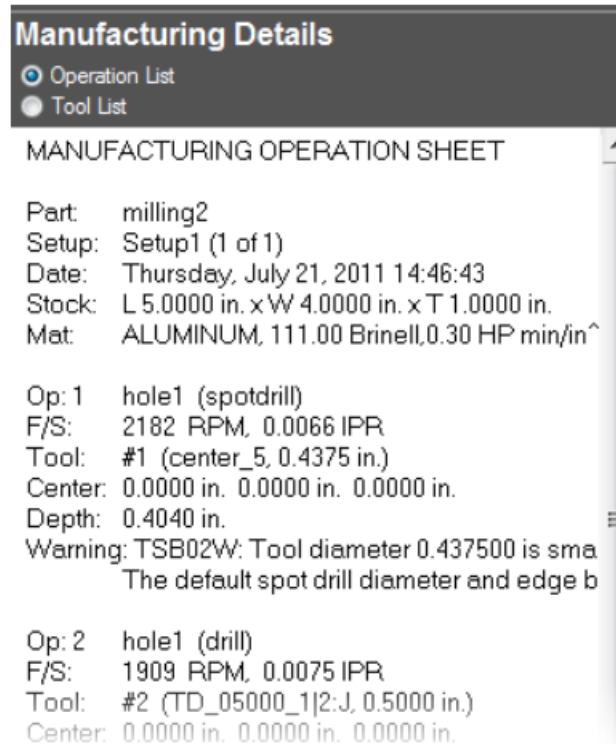
6. Click OK to close the Automatic Operation Ordering dialog.

- ✓ If you want to erase the simulation and remove the Simulation toolbar, click Eject .

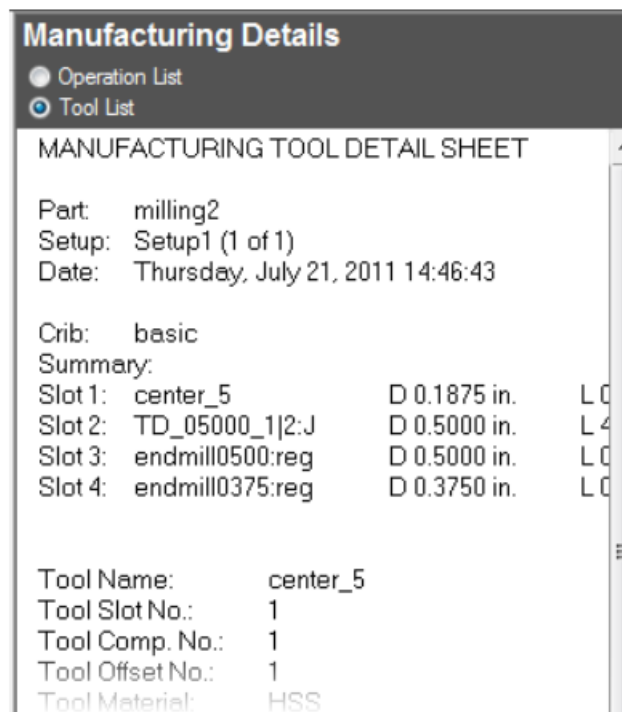
3.2.6. Part documentation

As well as simulating the part manufacture, the simulation generates tool and operation lists. The tools selected are based on your tool database. You can print this information to use as an operator's checklist, using the File > Print menu option.

Click the Details tab in the Results window to display the Manufacturing Operations sheet.



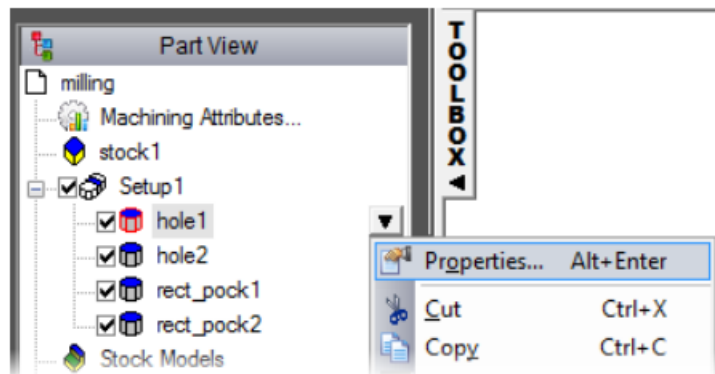
2. Select the Tool List option at the top of the Details tab to show the Manufacturing Tool Detail sheet. It contains all of the tools used to create the part based on the tool crib you selected.



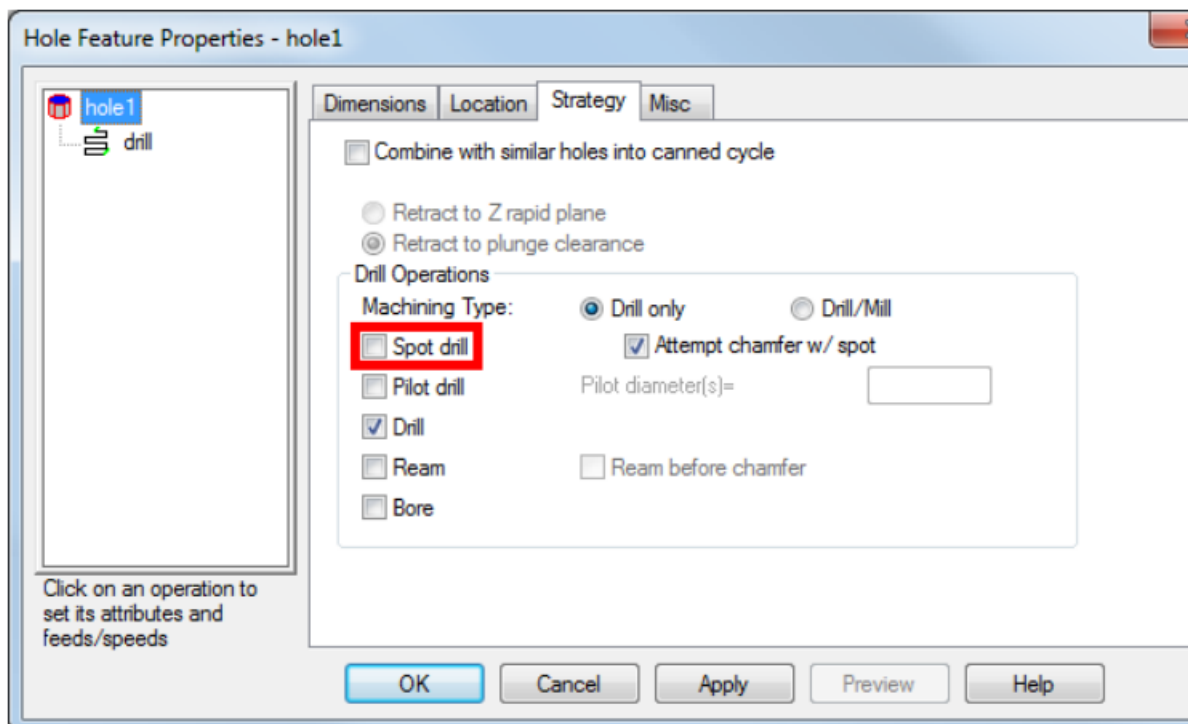
3.2.7. Controlling the strategies

You can control the strategies used to manufacture the part from the properties dialog.

1. Open the Part View panel.
2. Right-click on hole1 under the Setup1 node, and select the Properties option.





3. In the Properties dialog:



- Select the Strategy tab
- Deselect the Spot Drill option
- Click OK.

4. Select the Toolpaths  step from the Steps panel.


5. In the Simulation toolbar, click the 3D Simulation  button, and then click Play  to start the simulation.

- ✓ There is no spot drilling for the first hole. If you look through the operations list, there is only one spotdrill operation listed. FeatureCAM optimizes the part manufacturing process, but you control the level of automatic optimization.

6. Click Eject . This removes the Simulation toolbar.

3.2.8. Generating NC code

FeatureCAM generates the NC code to manufacture parts on a CNC machine. You can generate NC code after you have simulated the part, and therefore calculated the toolpaths.

1. Select the NC Code  step from the Steps panel. This displays the NC Code dialog.

2. Click the NC Program  button to generate the code.



```

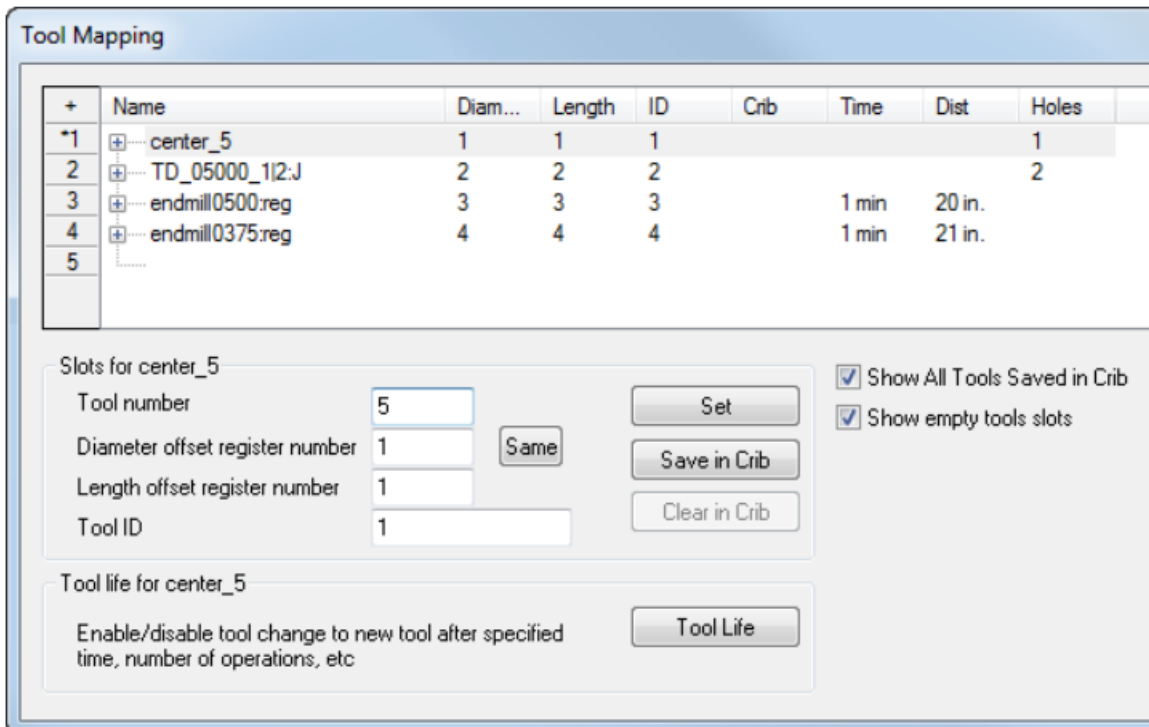
NC Code
.N10G70G94G75G90
'milling2 7-21-2011'
'HOLE1'
'TOOL NUMBER:1'
'SPINDLE RPM:2182'
N35G0X0.Y0.T1M6
N40S2182
N45X0.Y0.
N50Z0.1M8
.N55G81Z0.504F14.3
N60X0.
.N65G80
N70Z1.0
'HOLE1'
'TOOL NUMBER:2'
'SPINDLE RPM:1909'
N90G0X0.Y0.T2M6
N95G94F14.3S1909
N100X0.Y0.
N105Z0.1
.N110G83Z1.2502Z0.6Z0.25F14.3
N115X0.
.N120G80
N125Z1.0
N130X4.0Y3.0Z0.1
.N135G83Z1.2502Z0.6Z0.25F14.3

```

3.9. Tool mapping

To change the location of the tools in the tool changer:

1. Select the NC Code  step from the Steps panel. This displays the NC Code dialog.
2. Click the Tool Mapping  button. This displays the Tool Mapping showing the current tool order.



+	Name	Diam...	Length	ID	Crib	Time	Dist	Holes
*1	center_5	1	1	1				1
2	TD_05000_1 2:J	2	2	2				2
3	endmill0500.reg	3	3	3		1 min	20 in.	
4	endmill0375.reg	4	4	4		1 min	21 in.	
5								

Slots for center_5

Tool number:

Diameter offset register number:

Length offset register number:

Tool ID:

Show All Tools Saved in Crib

Show empty tools slots



Tool life for center_5

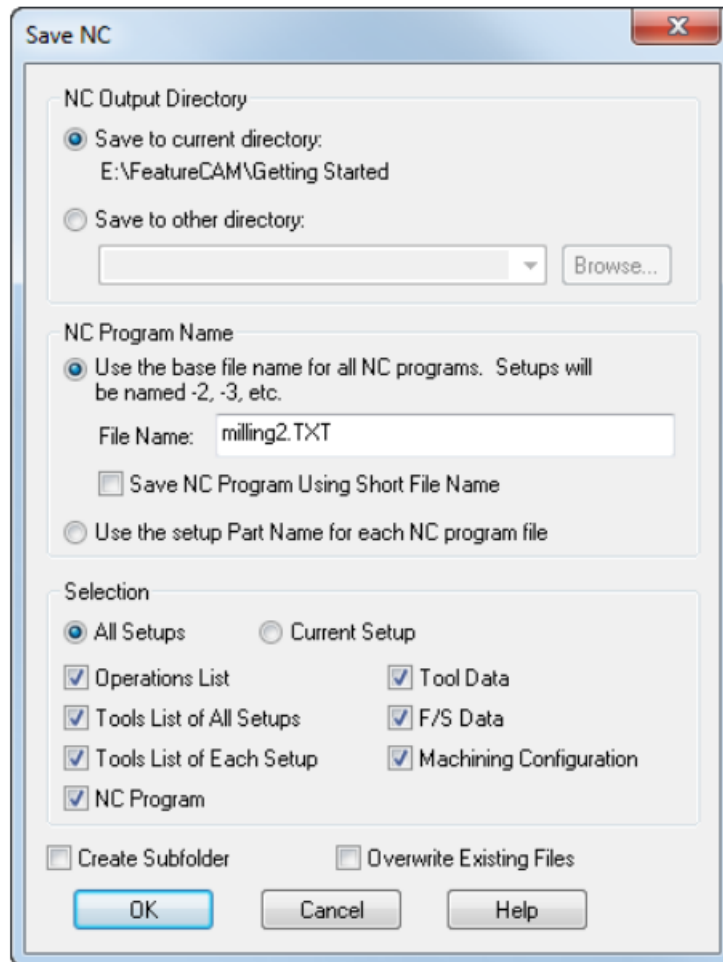
Enable/disable tool change to new tool after specified time, number of operations, etc

3. To move the center drill to the 5th position in the tool changer:
 - Select Center_5 in the table.
 - Enter a Tool Number of 5 in the Slots frame.
 - Click Set.
- ✓ You cannot change the number in the table.
4. Click OK to save the changes, and close the Tool Mapping dialog.

3.2.10. Saving the NC code

Для збереження керуючої програми:

1. Select the NC Code  step from the Steps panel. This displays the NC Code dialog.
2. Click the Save NC  button in the NC Code dialog.
3. In the Save NC dialog, accept the default filename and folder, and click OK.



3.3 Tasks for the laboratory session

According to the individual option, in the Delcam FeatureCAM system, create the necessary elements of milling processing, determine the list and sequence of the operation, and generate the control program code for the CNC machine.

The main goal of this task is to study the commands of the "Elements" tool panel for milling operations. In the process of performing the work, the student must learn to manage the strategy of processing certain elements, change

the order of operations and the location of the tool, manage the auto-arrangement options. Know and be able to create milling elements and arrays based on them.

Variants of tasks are given in Appendix C, p. 96.

3.4 Requirements for a report on the performance of a laboratory session

The report should contain the following sections:

1. Topic and purpose.
2. Individual task.
3. Development of a control program for milling processing:
 - appearance and dimensions of the workpiece;
 - elements and contours for generating the control program;
 - operations and their main characteristics;
 - control program code;
 - part model.
4. Conclusions from the laboratory session.

3.5 Control questions

1. What milling elements can be created without first creating contours.
2. Purpose and features of using elements: wall, ledge, groove, pocket.
3. Describe the methods of changing the order of operations.
4. Auto setting options.
5. Management of processing strategies.
6. Justify the necessity and method of changing the postprocessor.
7. How to get information about the project.
8. Methods of saving the control program code.

LABORATORY LESSON №4

3.5D MILLING IN DELCAM FEATURECAM

4.1 Goal

Familiarize yourself with the technique of creating different types of surfaces in the DELCAM FeatureCAM system. Learn to create control programs for processing complex surfaces.

4.2 An example of creating a typical 3.5D milling part

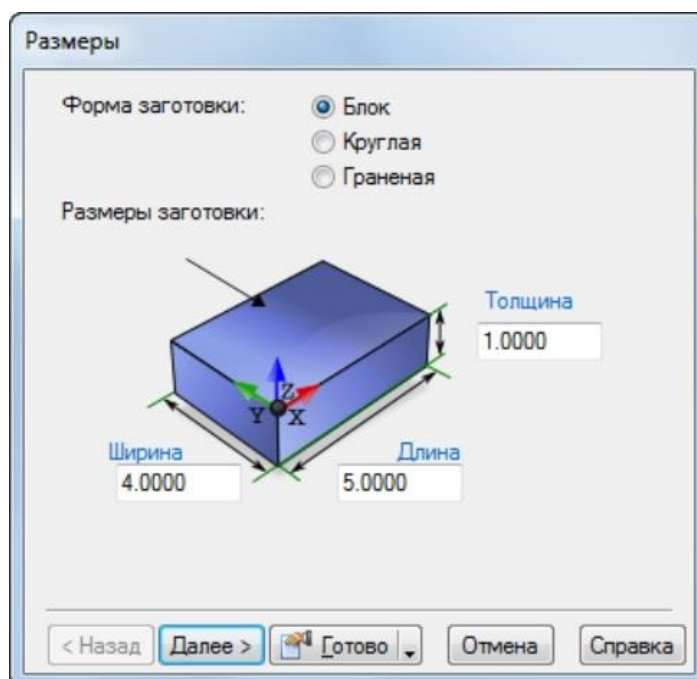
This example introduces students to the method of 3D surface modeling and their processing using surface milling elements. Definition of processing operations and their attributes.

Entering the system and setting the initial parameters is carried out according to the methodology given in laboratory work No. 1, however, as the type of processing, you must select Milling, the units of measurement are inches.

4.2.1. Defining the Stock

The stock is the initial material from which you cut your part.

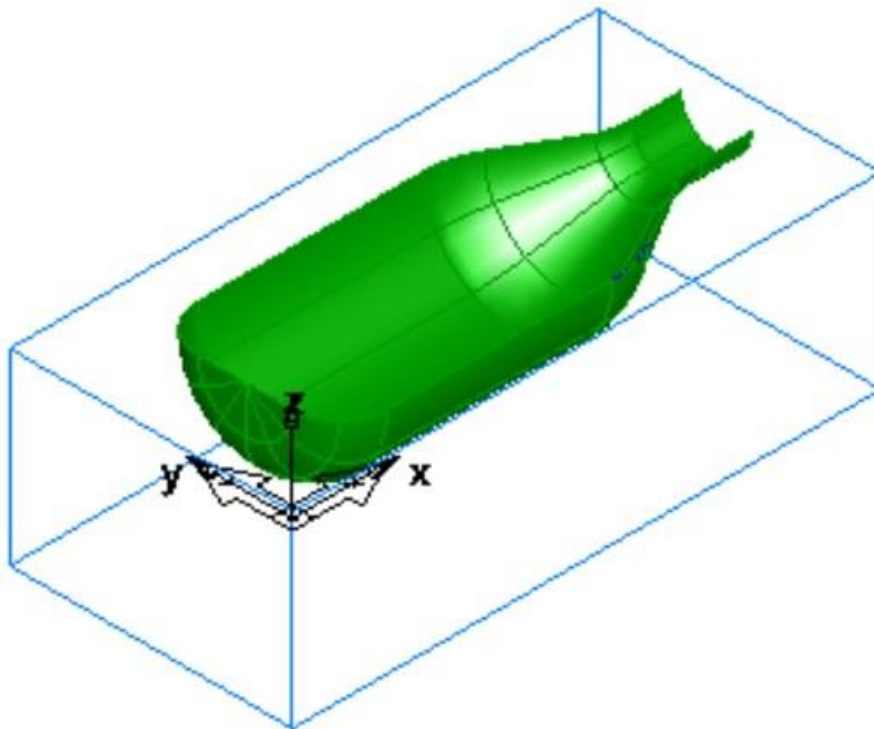
1. On the Dimensions page of the Stock wizard:



- Enter a Thickness of 2.
 - Enter a Width of 3.
 - Enter a Length of 6.25.
 - Click Finish.
2. Click OK to accept the default values of the Stock Properties dialog.

4.2.2. Defining the geometry

This shows you how to design your part.



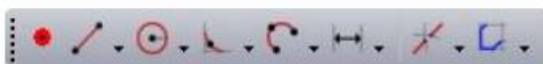
1. Select View > Toolbars from the menu, in the Toolbars frame:

- Select Advanced.
- Select Geometry.
- Click OK.

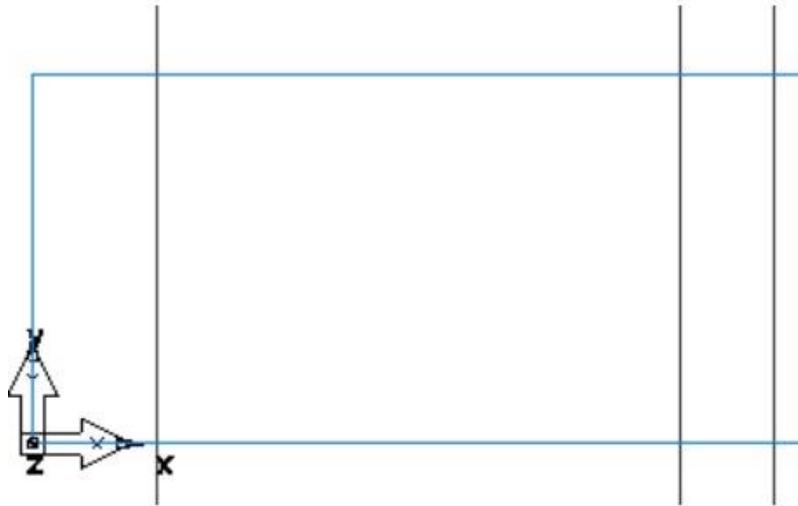
2. Create three vertical lines

- On the Geometry toolbar, select Vertical  from the Line  menu.



menu.

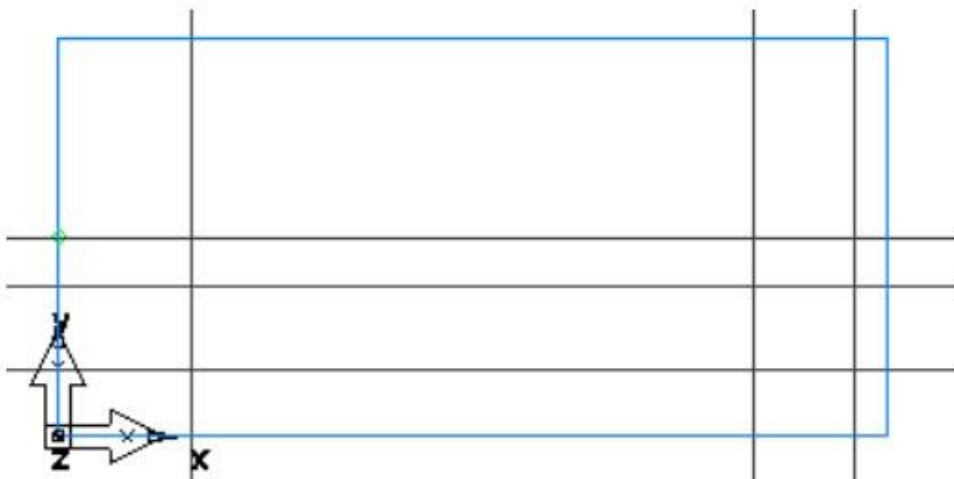


- In the Feature/Geometry Edit bar, enter an XYZ of X 1, Z 0, and press Enter.
- Create a second line by entering an XYZ of X 5.25, Z 0, and press Enter.
- Create a third line by entering an XYZ of X 6, Z 0, and press Enter.





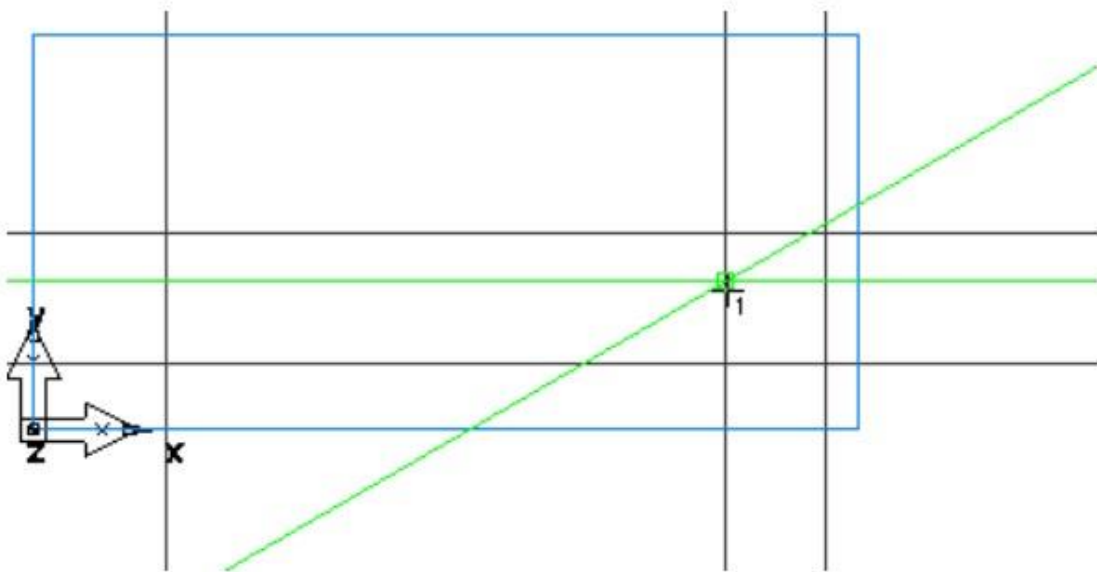
3. Create three horizontal lines:

- On the Geometry toolbar, select Horizontal  from the Line  menu.
- Enter an XYZ of Y 0.5, Z 0, and press Enter.
- Create a second line by entering an XYZ of Y 1.125, Z 0, and press Enter.
- Create a third line by entering an XYZ of Y 1.5, Z 0, and press Enter.





4. Create a through line:

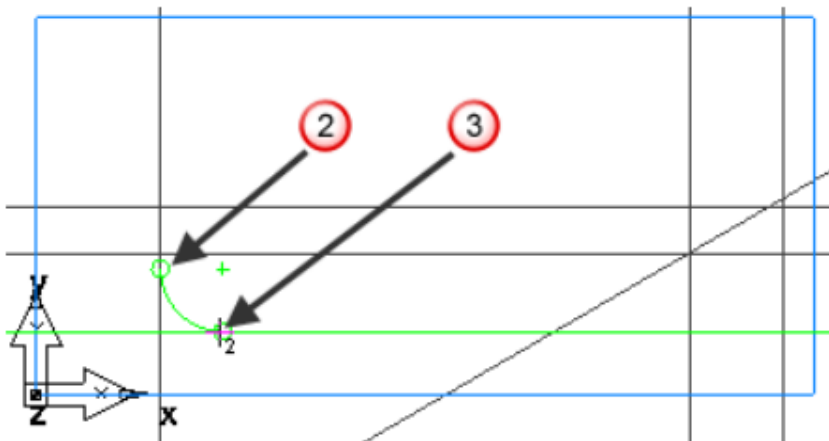
- On the Geometry toolbar, select Point, Angle  from the Line  menu.
- In the Feature/Geometry Edit bar, enter an angle A of 30.
- In the graphics window, click at the intersection between the second horizontal and second vertical lines, at point 1, to create a through line.



5. Create arcs.

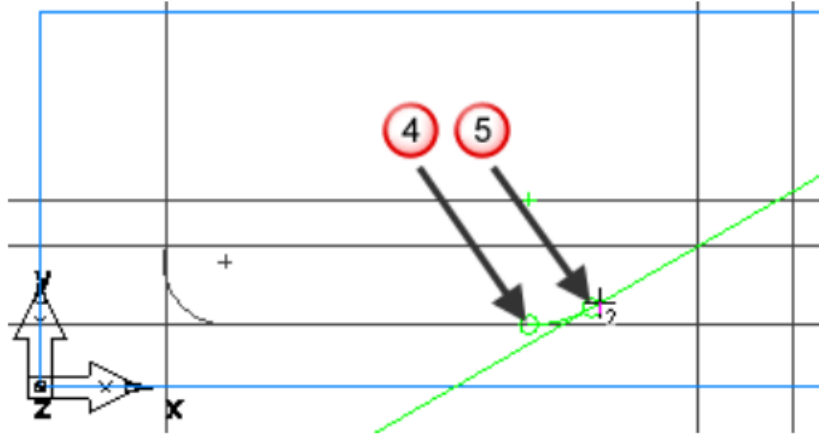
On the Geometry toolbar, select 2 Pts, Radius  from the Arc  menu.

- In the Feature/Geometry Edit bar, enter a radius R of 0.5, and click the vertical line around point 2 and the horizontal line around point 3.

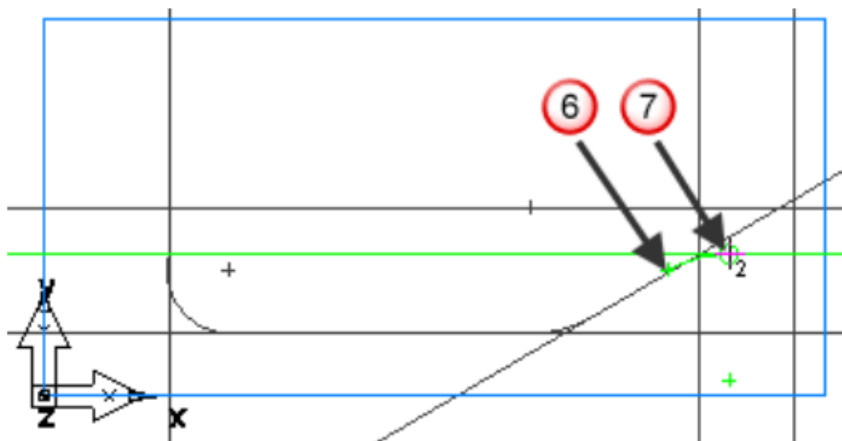


- Create the second arc:



In the Feature/Geometry Edit bar, enter a radius R of 1.0, and click the horizontal line around point 4 and the through line around point 5.

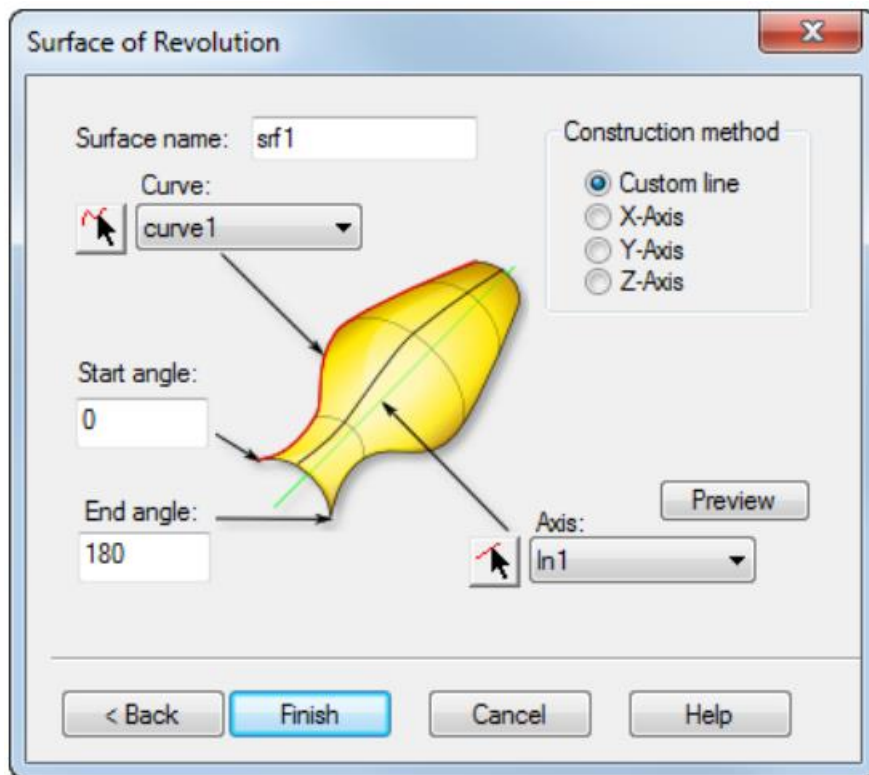



- Create the third arc by clicking the through line around point 6 and the horizontal line around point 7.

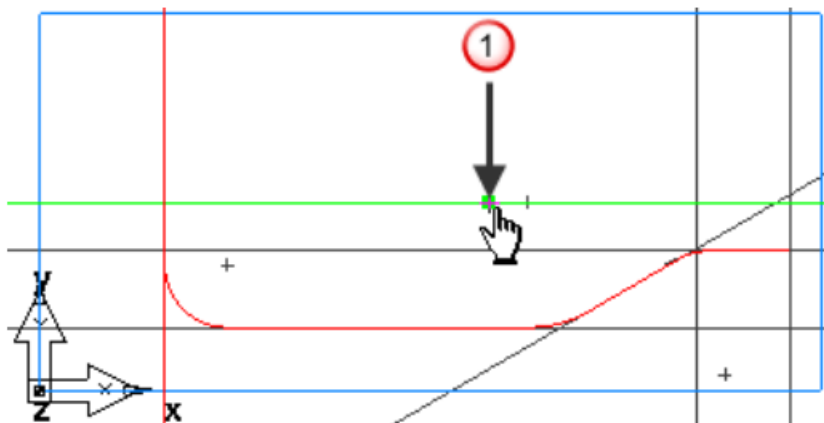


6. To mill the part you need to chain the curves.

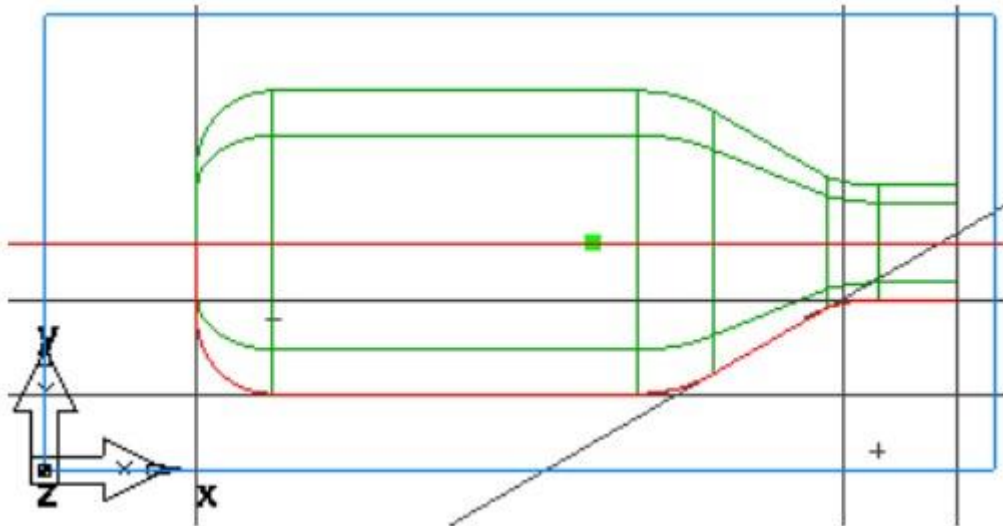
- Select the Curves  step from the Steps panel.
- In the Curves Creation dialog, select the Pick Curve Pieces  button.
- Click at the intersection of the vertical and horizontal line at point 8 and at the intersection of the vertical and horizontal line at point 9.




- Enter a Start Angle of 0.
- Enter an End Angle of 180.
- FeatureCAM automatically selects your chained curve in the Curve field.
- In the Axis field, click the Pick line  button, and select the horizontal line around point 1.

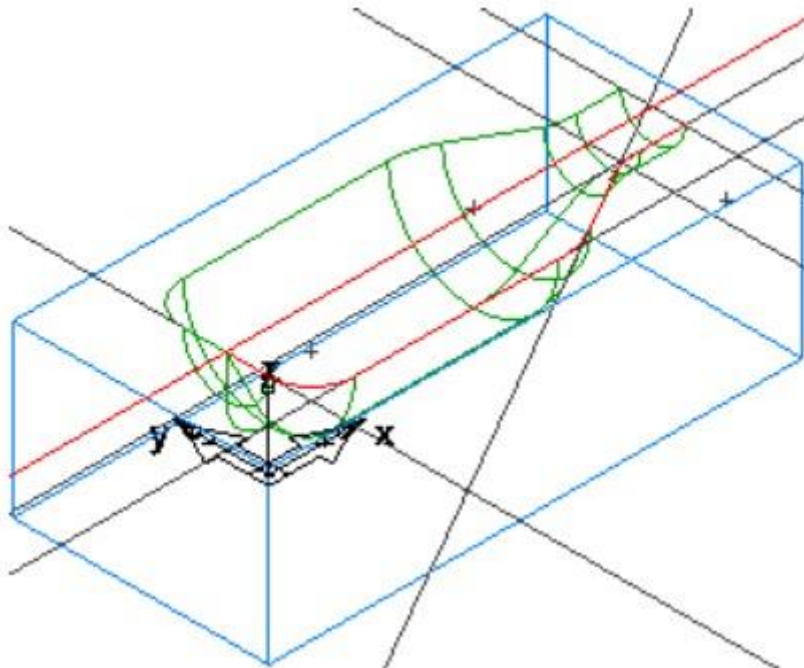


Click Finish.



4.2.4. Viewing the part

1. To change the view to an isometric view, click the Isometric  button on the Standard toolbar.



2. Control how the part is displayed using the Viewing Options.
 - Select Options > Viewing from the menu. This displays the Viewing Options dialog.
 - Select the Show surface boundaries only option, and click Apply.

This displays the surfaces as only their outer boundaries and trimmed loops. No additional lines are drawn in the interior of the surface. This makes the display of larger models much faster.



- Deselect the Show surface boundaries only option, and click Apply.



This displays the surfaces with lines in the interior of the surface. This aids visualization, but for large models, it makes the display of the part slower.


- Enter a Surface fineness Wireframe of 20, and click Apply.

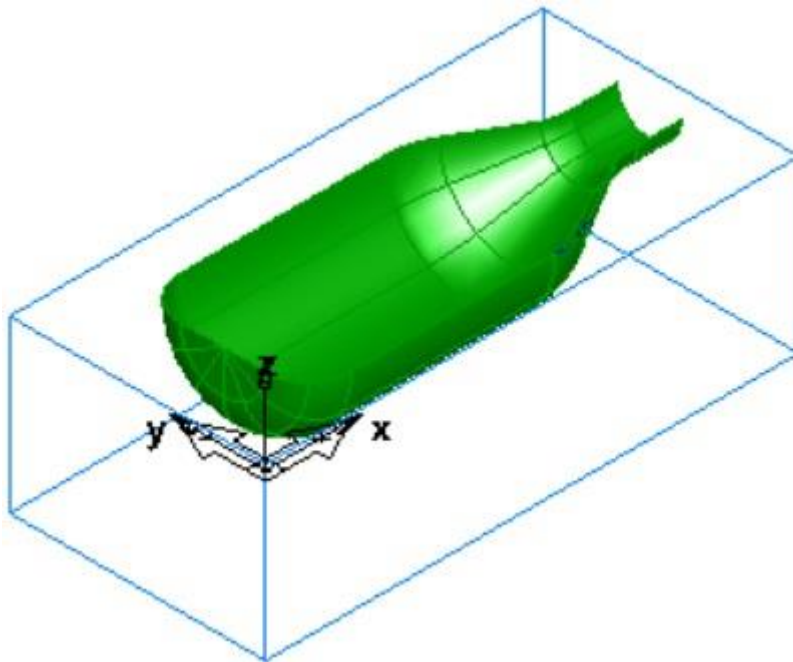
This displays the surfaces with more lines. Decreasing the value of Surface Fineness improves the display quality but slows down the graphics.

- Click OK to close the dialog

3. From the Hide  menu on the Advanced toolbar, click the Hide All Geometry  button.


4. From the Show  menu on the Advanced toolbar, click the Show all surfaces  button.

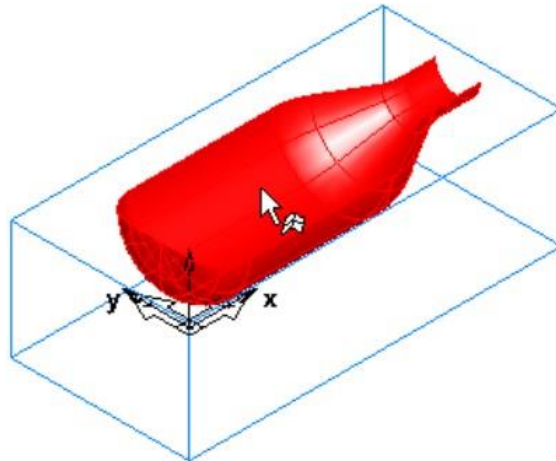
5. Click the Shade  button, on the Standard toolbar, to shade the part.




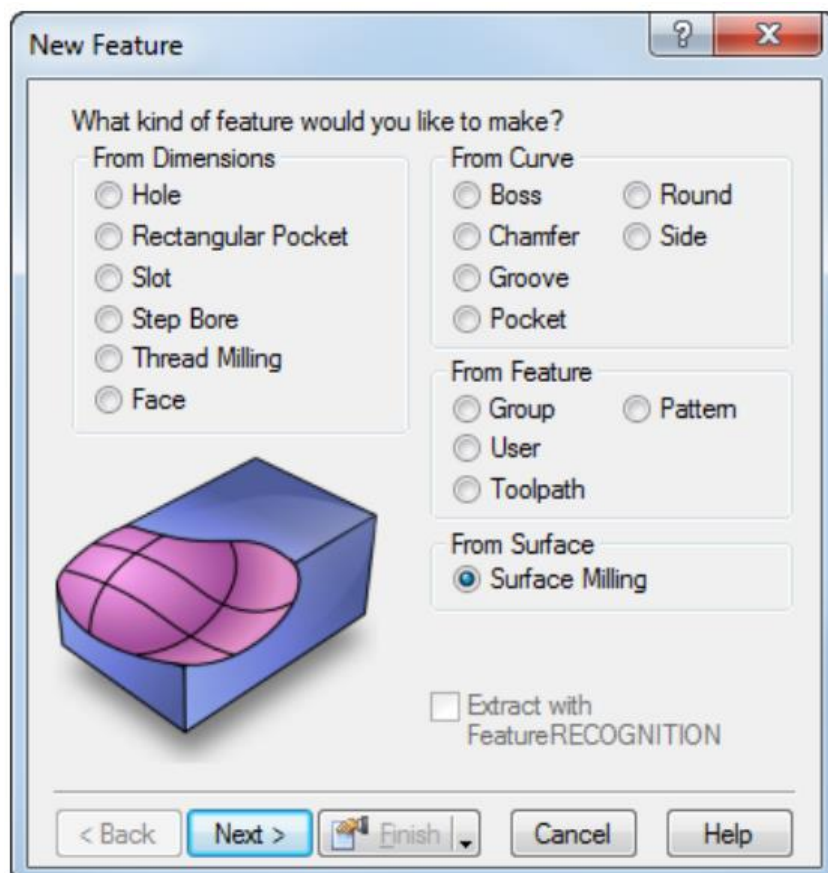
4.2.5. Creating a surface milling feature

This shows you how to create the surface features and select the toolpath strategies.

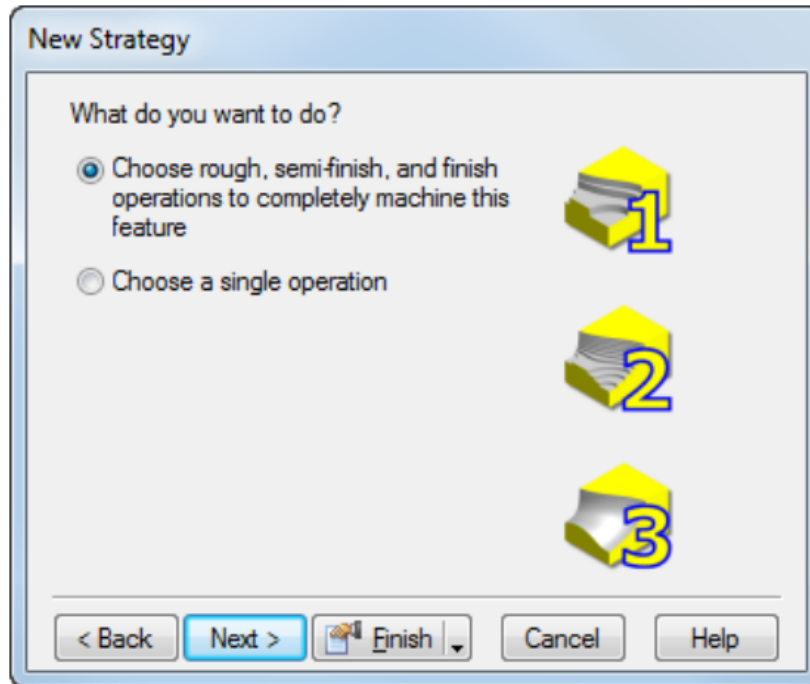
1. On the Standard toolbar, click the Select  button, and select the surface (srf1). On selection it turns red.



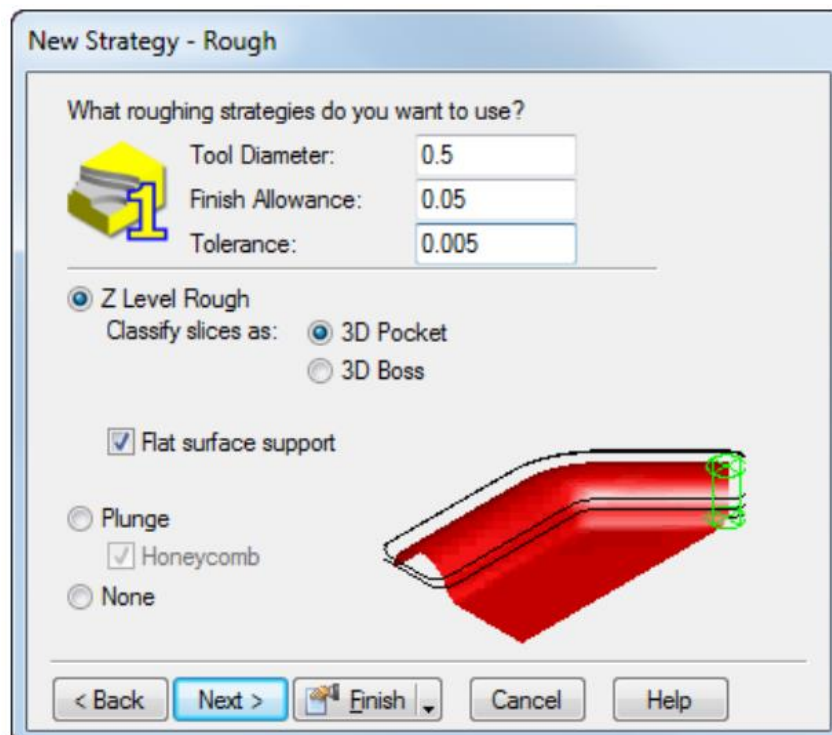
2. Select the Features  step from the Steps panel.



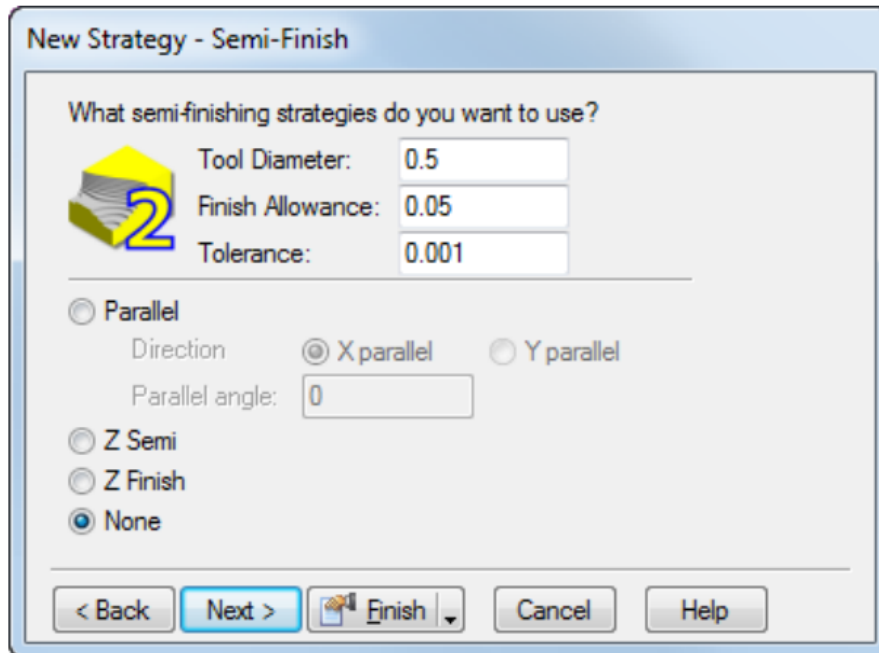
3. In the New Feature wizard, in the From Surface frame, select the Surface Milling option, and click next.
4. On the Part Surface page click Next.
5. On the New Strategy page, select the Choose Rough, Semi Finish, and Finish... option, and click Next.



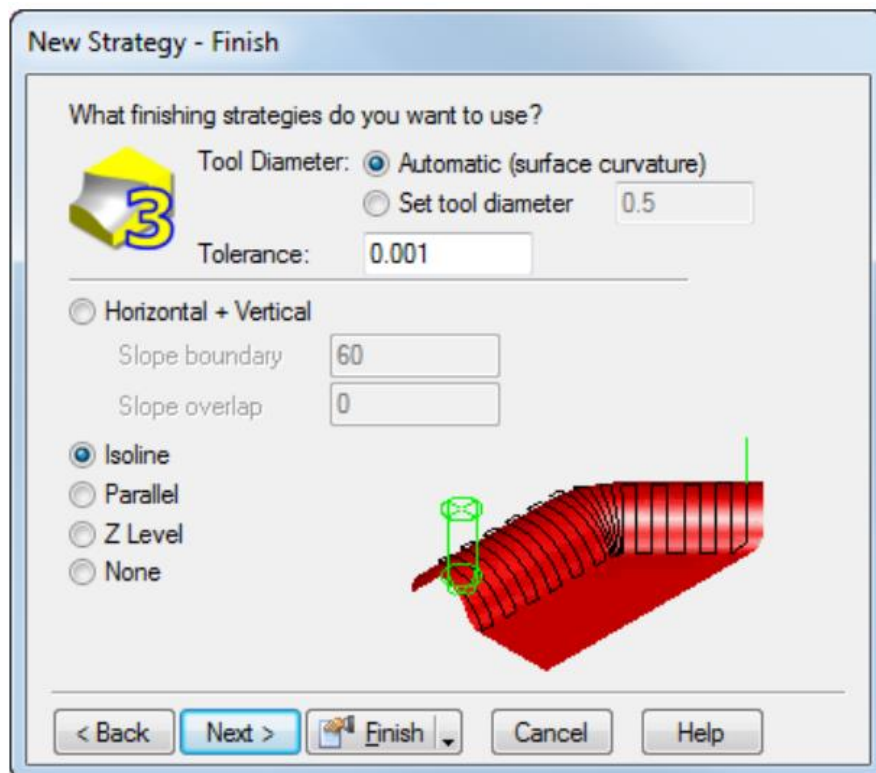
6. On the Rough page:



- Select the Z Level Rough option.
 - Select Classify slices as 3D Pocket.
 - Click Next.
7. On the Semi-Finish page, select None, and click Next.



8. On the Finish page, select Isoline.





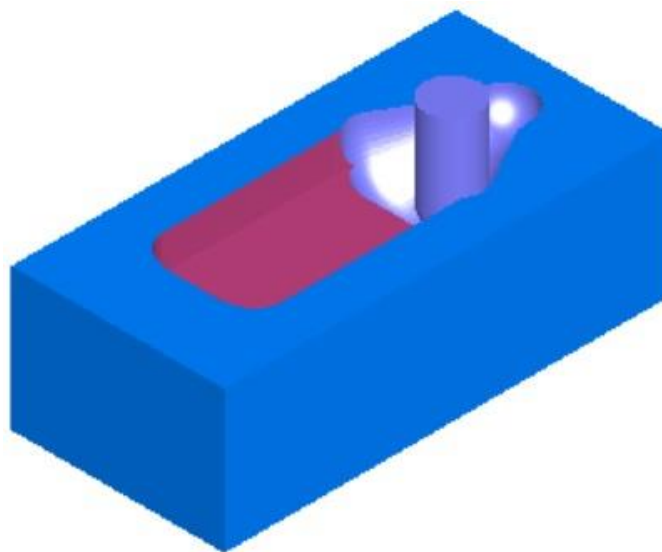
9. Click the Finish button.

4.2.6. Simulating the toolpaths

To view the simulated toolpath:

1. Click the Toolpaths  step in the Steps panel to display the Simulation toolbar.

2. Click the 3D Simulation , button, and then click the Play  button to start the simulation. If the Automatic Ordering Options dialog displays, click OK to close it. This accepts the default ordering options.



Note how the toolpaths are accurately simulated including the part rotations.

3. Click Eject . This removes the Simulation toolbar.

4.3 Tasks for the laboratory session

According to the individual option, in the Delcam FeatureCAM system, create the necessary elements of milling processing, determine the list and sequence of operations, and generate the control program code for the CNC machine.

The main goal of this task is to study the auxiliary possibilities of creating geometry elements and methods of creating surfaces. In the process of perform-

ing the work, the student must learn to create elements of surface treatment. Be able to adjust processing modes to achieve the required quality of the part.

Variants of tasks are given in Appendix D, Art. 99.

4.4 Requirements for a report on the performance of a laboratory session

The report should contain the following sections:

1. Topic and purpose.
2. Individual task.
3. Development of a control program for milling 3.5D processing:
 - appearance and dimensions of the workpiece;
 - elements and contours for generating the control program;
 - operations and their main characteristics;
 - control program code;
 - part model.
4. Conclusions from the laboratory session.

4.5 Control questions

1. Features of creating vertical and horizontal lines. Geometry editing commands. Creating an ellipse and other complex contours.
2. Describe methods for creating DELCAM FeatureCAM surfaces.
3. Describe the method of creating a surface treatment element.
4. Explain the features of use and the difference in surface treatment in the 3D pocket and 3D protrusion modes.
5. Explain the difference when using isoline, raster, and Z-level contour processing mode.

LABORATORY LESSON №5

EDM IN DELCAM FEATURECAM

5.1 Goal

Explore the possibilities of the DELCAM FeatureCAM system when working in the EDM mode. Learn to create control programs for EDM.

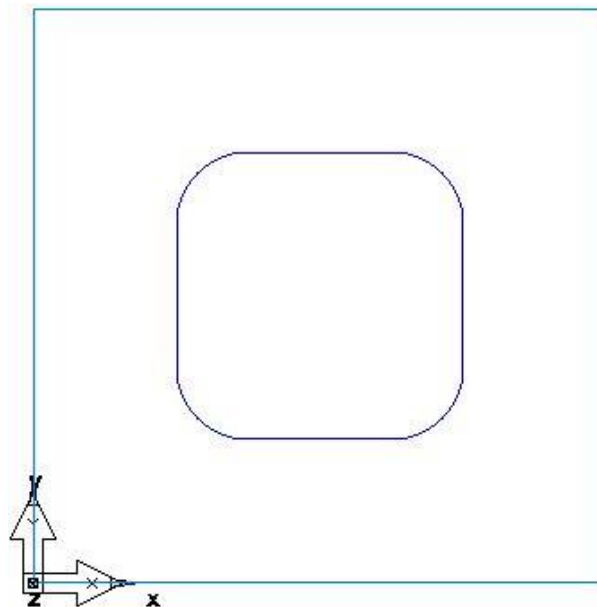
5.2 An example of creating a typical EDM part

This example introduces students to the method of 3D surface modeling and their processing using surface milling elements. Definition of processing operations and their attributes.


Entering the system and setting the initial parameters is carried out according to the method given in laboratory work #1. Electroerosion operation should be selected as the type of processing, the units of measurement are inches. As a blank, choose a block with a thickness of 0.5, a width of 4, and a length of

4.5.2.1. Creating the profile

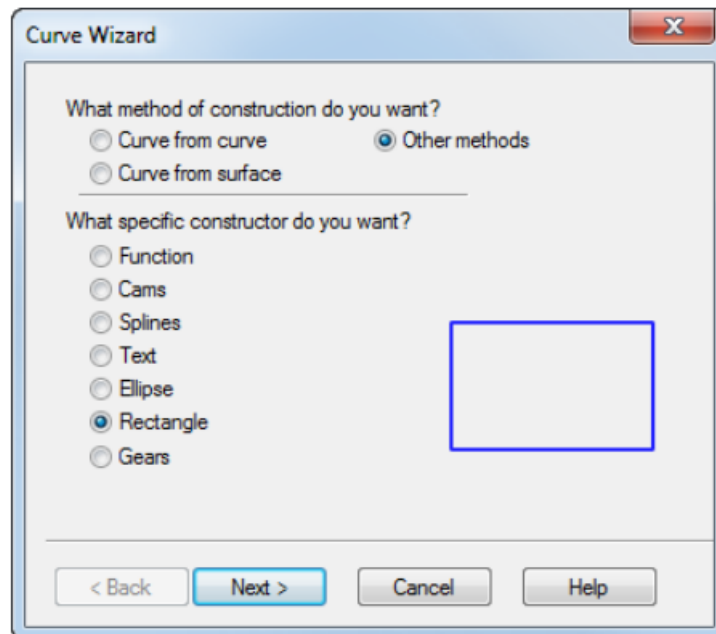
This step defines the profile.



1. Select the Curves  step from the Steps panel.

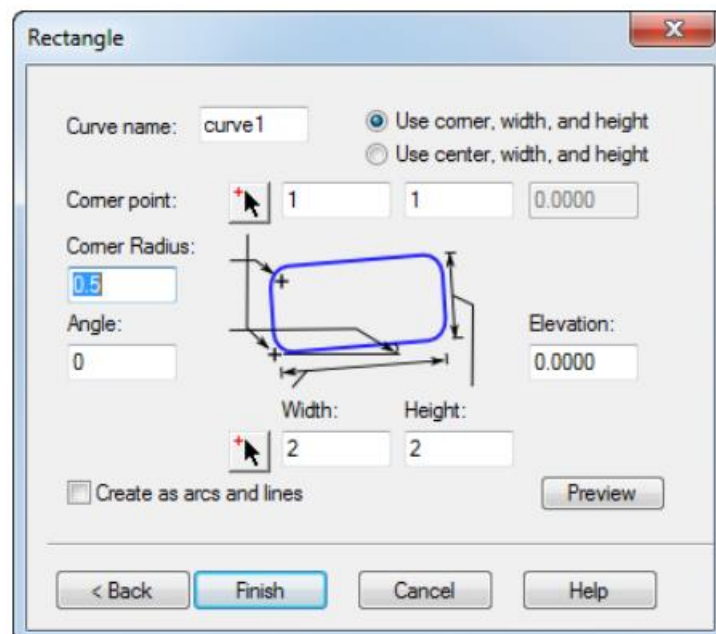
2. In the Curves Creation dialog, select the Curve Wizard  button.

3. In the Curve Wizard:



- Select a construction method of Other methods.
- Select a constructor of Rectangle.
- Click Next.

4. On the Rectangle page:



- Select Use corner, width, and height.
- Enter corner point of 1, 1, 0.

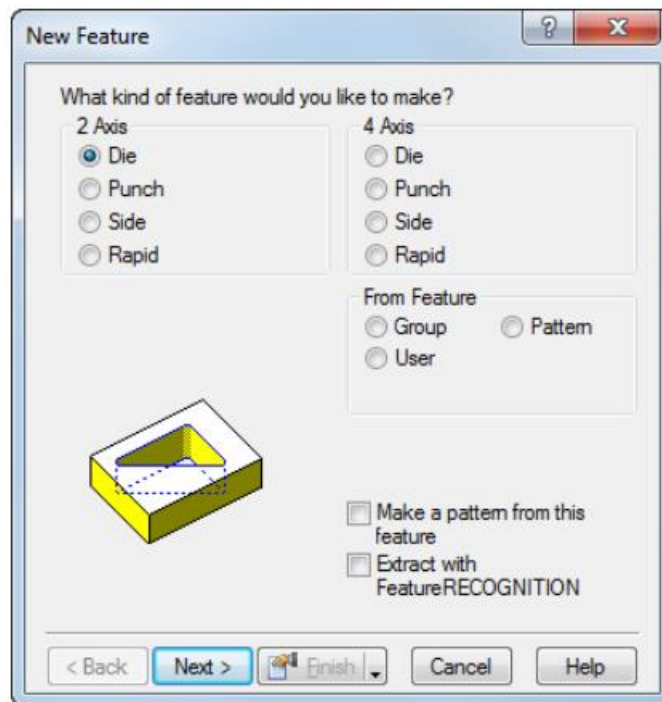
- Enter a corner radius of 0.5.
- Enter a Width of 2.0.
- Enter a Height of 2.0.
- Click Finish.


5.2.2 Creating a wire EDM feature

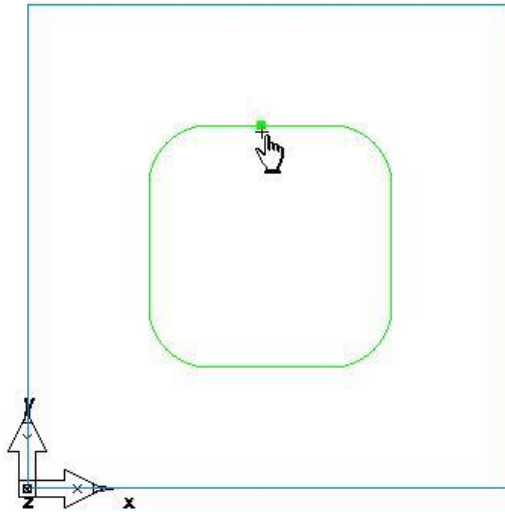
This shows you how to create a wire EDM feature.

1. Click the Features  step in the Steps panel.

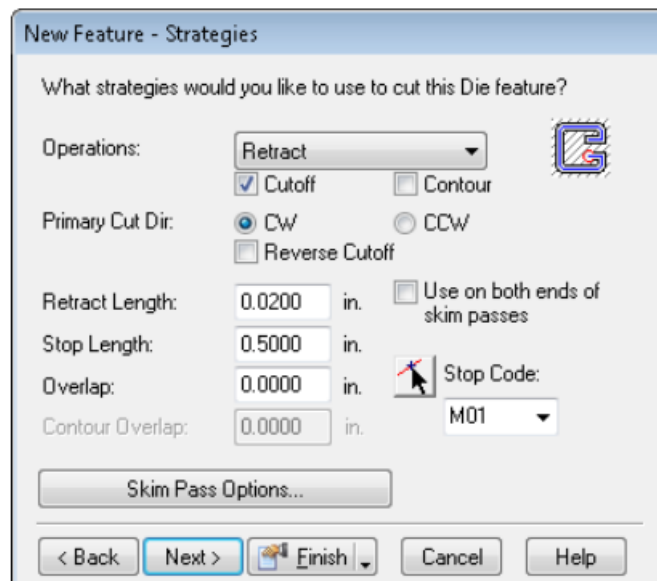
2. In the New Feature wizard, select the Die option in the 2 Axis frame, and click Next.



3. On the Curves page, click the Pick curve or geometry  button, select the curve you created, and click Next.



4. On the Location page, click Next.
5. On the Dimensions page, enter a Thickness of 0.5 and click Next.
6. On the Start page, click Next.
7. On the Strategies page:



- In the Operations field select Retract.
- Select the Cutoff option.
- Select the Contour option.
- Click Finish.


5.2.3. Simulating the wire EDM toolpath

Now you have created the features, FeatureCAM automatically:






- Selects the most appropriate tools and operations;

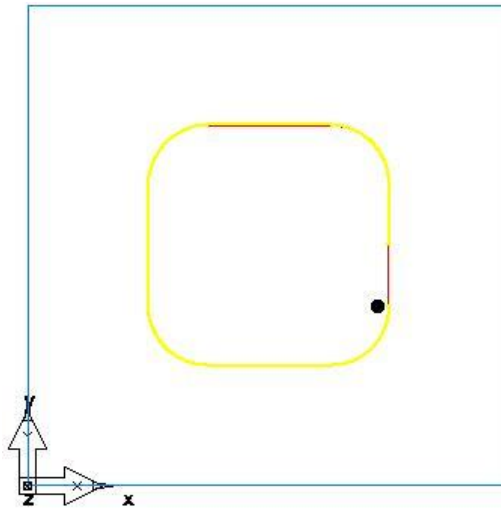
- Recommends machining strategies;
- Calculates speeds and feeds;
- Generates toolpaths and creates the NC code.

To view the simulated toolpath:

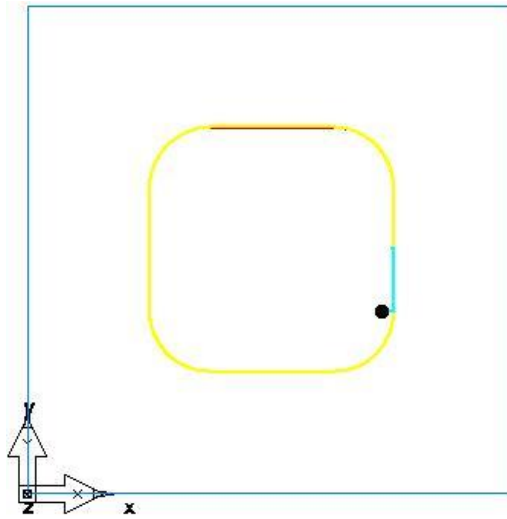
1. Click the Toolpaths  step in the Steps panel to display the Simulation toolbar.



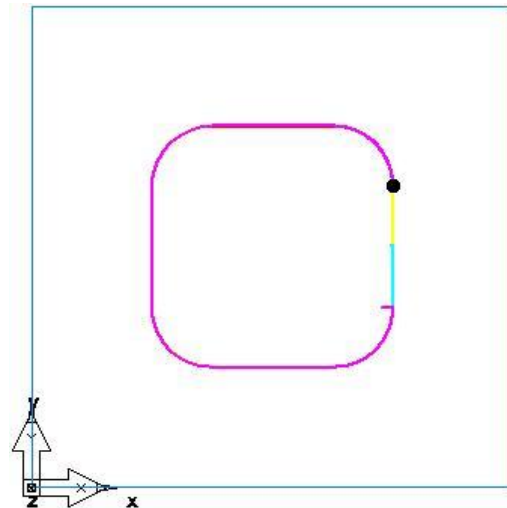
2. Click the 2D Simulation  button on the Simulation toolbar.
 3. Center the Simulation Speed  slider to specify the simulation rate.
 4. From the Simulation Next  menu button, select the Play to Next Operation  button to see the retract operation. If the Automatic Ordering Options dialog displays, click OK to close it.
- ✓ To slow down the simulation, drag the Simulation Speed  slider to the left..



5. Click the Play to Next Operation , button again to see the cutoff operation.




6. Click the Play to Next Operation , button again to see the final contour operation.



7. Click Eject .

5.2.4. Generating NC code

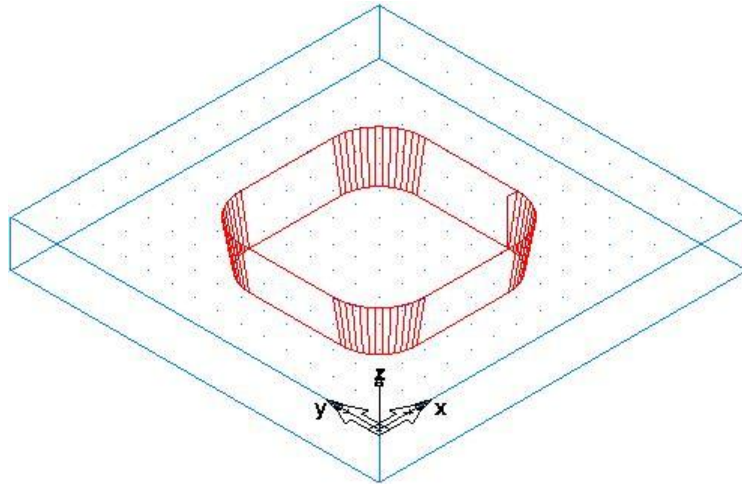
FeatureCAM generates the NC code to manufacture parts on a CNC machine. You can generate NC code after you have simulated the part, and therefore calculated the toolpaths.


1. Select the NC Code  step from the Steps panel. This displays the NC Code dialog.

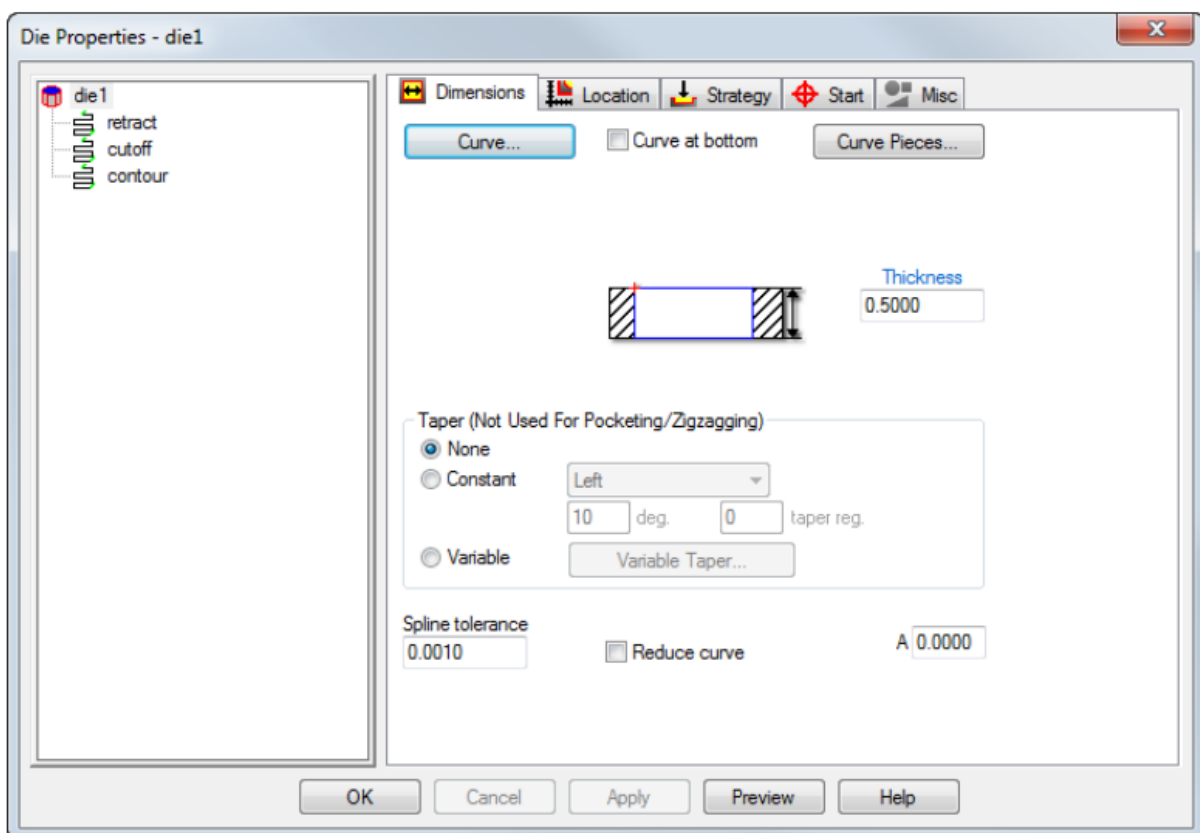
2. Click the Display the NC Code  button to generate the NC code.

5.2.5. Adding a taper angle

This example shows you how to add a draft angle to a wire EDM part.





1. Open the Part View panel, select the die1 feature from the Setup1 node, and click the Properties  button on the Feature/Geometry Edit toolbar.
2. In the Properties dialog for die1:



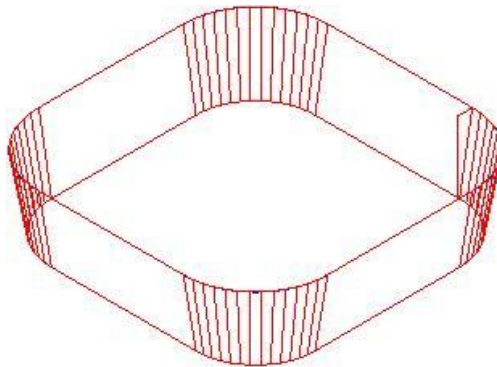
- Select Constant.
- Select a taper type of Left.

- Enter a deg. of 10 as the taper angle.
- Click Apply.

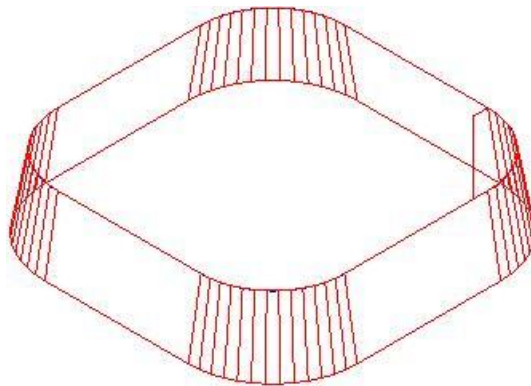
3. Click the Hide Stock  button from the Hide  menu on the Advanced toolbar.



4. Click the Isometric View  button on the Standard toolbar.



5. Return back to the Properties dialog, set the taper type to Right, and click Apply.

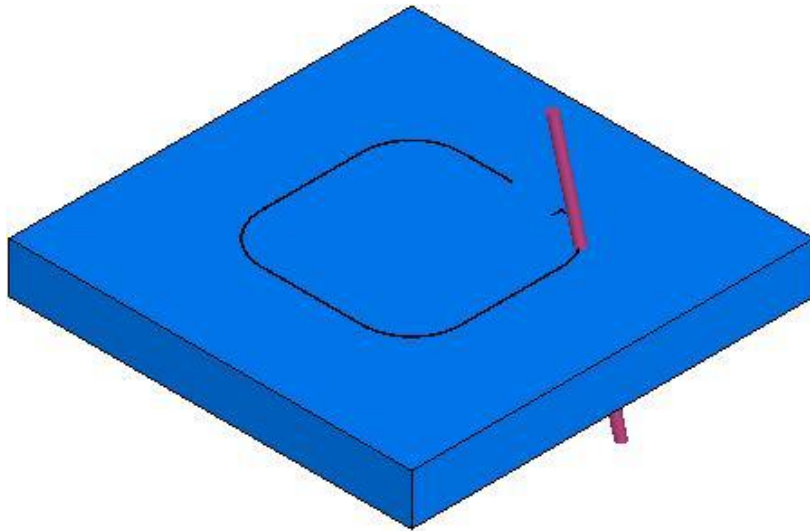


6. Change the taper type back to Left, and click OK to close the Properties dialog.

7. Select the Toolpaths  step from the Steps panel.

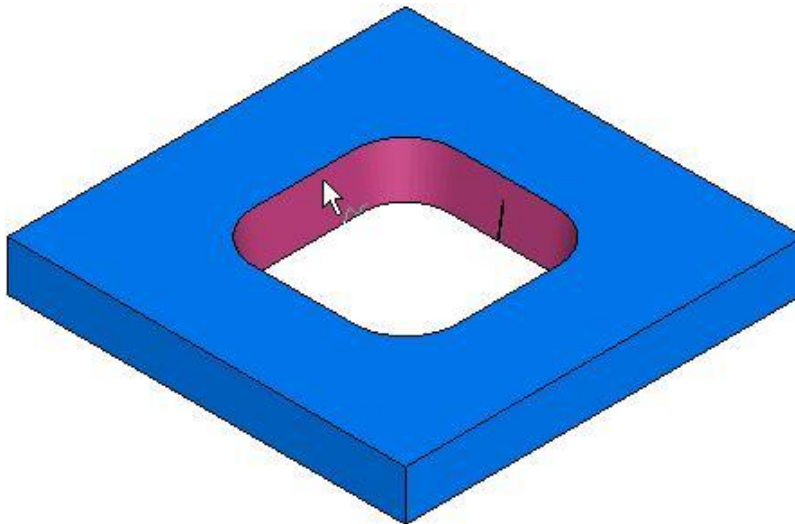
8. Click the 3D Simulation , button, and then click the Play  button.

Since this is a matrix element, it is implied that the region in the middle of the curve will be removed.





9. Click the Select  button on the Standard toolbar.

10. Click inside the curve. FeatureCAM deletes that part of the stock.



11. Click Eject .

12. From the Show  menu on the Advanced toolbar, click the Show Stock  button.

5.3 Tasks for the laboratory session

According to the individual option, in the Delcam FeatureCAM system, create the contour of the EDM part.

The main goal of this task is to study various methods of creating contours, including standard curves. To study the peculiarities of creating EEO elements, to determine the main differences when creating elements of the "Pan-

son" and "Matrix" type. Be able to set up simulation of part processing and generate control program code, as well as remove redundant elements after processing.

Variants of tasks are given in Appendix E, Art. 103

5.4 Requirements for a report on the performance of a laboratory session

The report should contain the following sections:

1. Topic and purpose.
2. Individual task.
3. Development of a control program for EDM:
 - appearance and dimensions of the workpiece;
 - elements and contours for generating the control program;
 - operations and their main characteristics;
 - control program code;
 - part model.
4. Conclusions from the laboratory session.

5.5 Control questions

1. Describe the possible options for creating DELCAM FeatureCAM contours.
2. Explain the features of creating EEO elements of the "Punch" and "Matrix" type.
3. Determine the need to create slopes for details, explain the specifics of the team's work.
4. Determine the methods of detail display, describe the sequence of hiding unnecessary elements.

RECOMMENDED BOOKS

1. FeatureCAM 2020 R2. Reference Help 2359 p.
2. FeatureCAM 2021 R3. Getting Started (Feature-based CAM software for mills, multi-tasking lathes and wire EDM) 86 p.
3. Доля В.М. Програмування, введення та відпрацювання управляючих програм для верстатів з ЧПУ та РТК: Навчальний посібник. – Харків: НТУ „ХПІ”, 2004. – 169 с.
4. Конспект лекцій з дисципліни «Технологічні основи програмування для верстатів з ЧПК» призначений для студентів Машинобудівного коледжу ДДМА за напрямом підготовки 050503 «Машинобудування», спеціальністю 5.05050302 Технологія обробки матеріалів на верстатах та автоматичних лініях (133 Галузеве машинобудування). Краматорськ 2016.
5. <https://www.autodesk.com/products/powershape/overview>
6. Yakushenko, S.O., Nosov, P.S. (2013). Proektni rozrakhunky ta 3D modeliuvannya dvyhunik vnutrishnoho zghoriannia u SAPR Delcam PowerShape. [Design calculations and 3D modeling of internal combustion engines in CAD Delcam PowerShape] Inform. tekhnolohii v osviti, nauksi ta vyr-vi. 4 (5). 165-179 <http://dspace.opu.ua/jspui/handle/123456789/2687>
7. Kuang-Hua Chang (2013) Product Performance Evaluation using CAD/CAE.
8. Kuang-Hua Chang (2016). e-Design.
9. Kuang-Hua Chang (2014). Product Design Modeling using CAD/CAE.

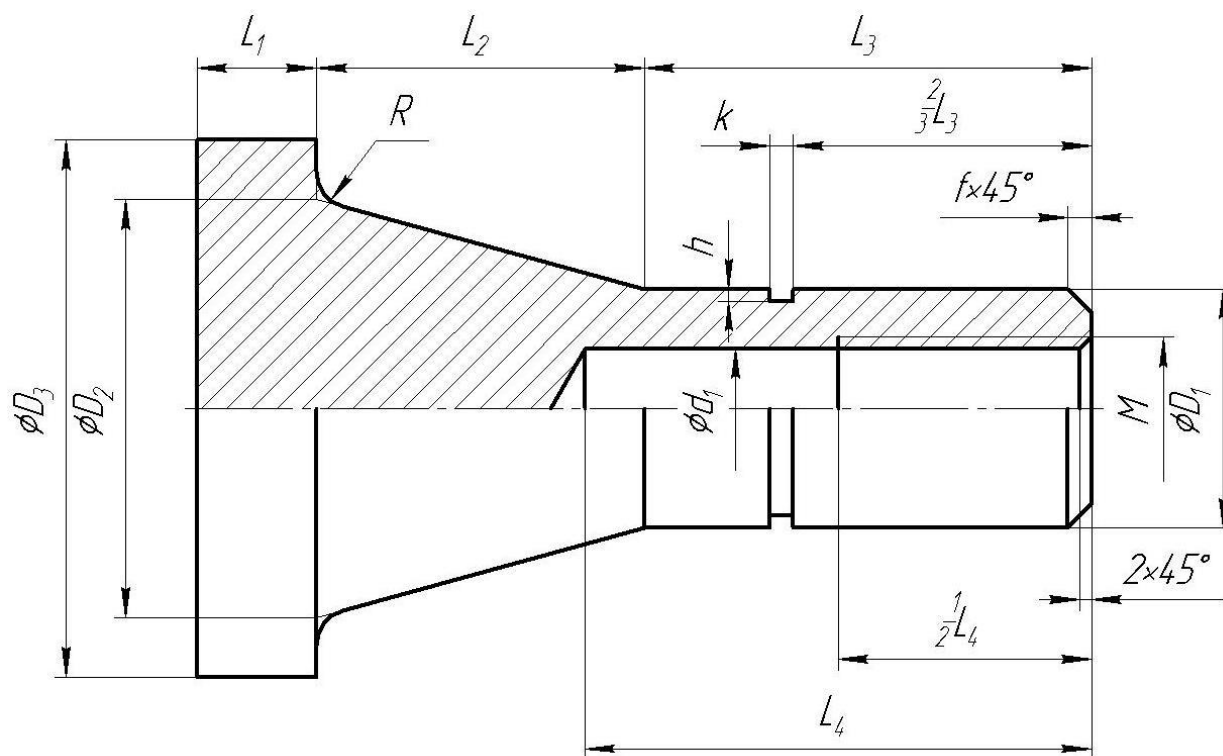
ADDITION A

Variants of tasks for laboratory work №1

Develop a turning program for a CNC machine tool.

Workpiece: cylinder 180 mm long, 90 mm in diameter.

Processing on the outer diameter of the largest step should not be provided, the end processing should be carried out with an allowance of 1.5 mm.

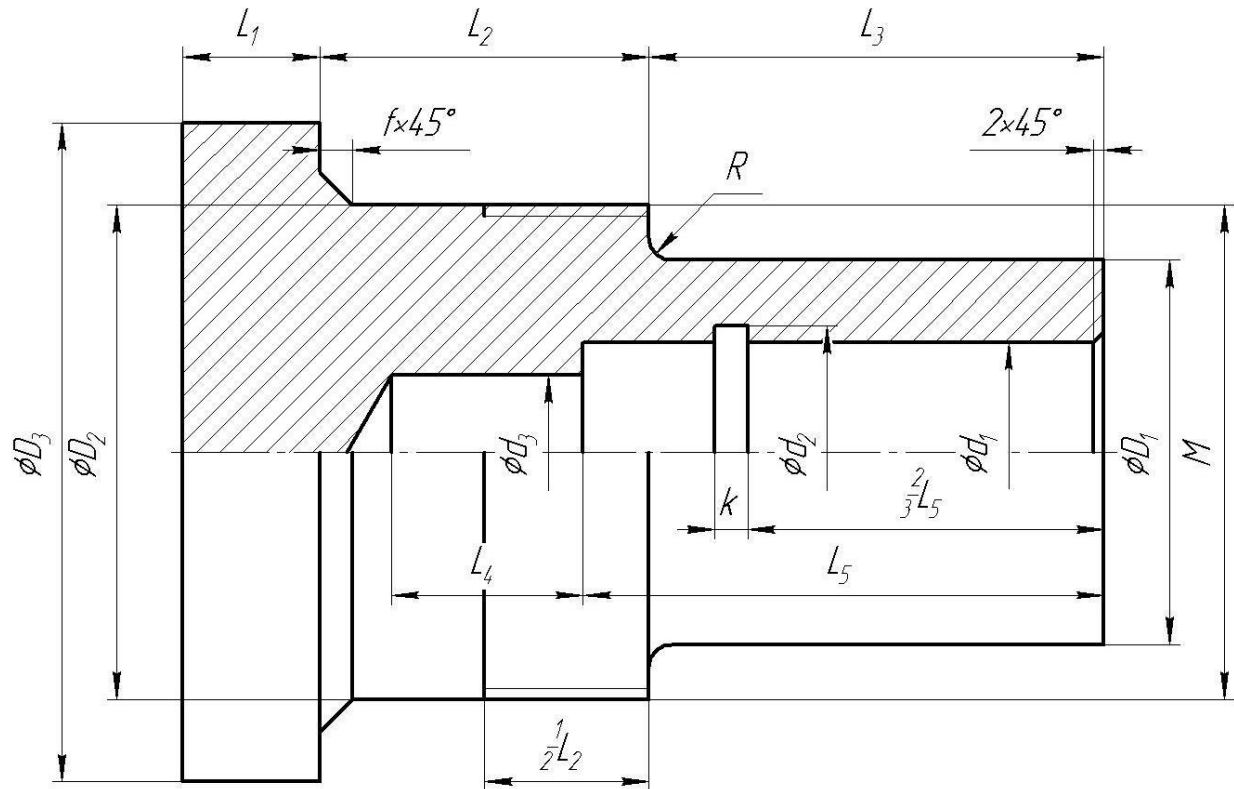


№ var.	L_1 , mm	L_2 , mm	L_3 , mm	L_4 , mm	M , mm	D_1 , mm	D_2 , mm	D_3 , mm	d_1 , mm	h , mm	k , mm	f , mm	R , mm
1	20	55	75	85	22	40	70	90	20	2	4	4	6
4	25	45	80	90	24	50	75	90	22	2	3	3	5
7	20	50	70	80	26	60	80	90	24	3	4	5	2
10	15	60	75	80	24	45	75	90	22	3	3	3	4
13	25	50	70	70	22	55	80	90	20	4	4	4	3

Develop a turning program for a CNC machine tool.

Workpiece: cylinder 200 mm long, 100 mm in diameter.

Processing on the outer diameter of the largest step should not be provided, the end processing should be carried out with an allowance of 2 mm.

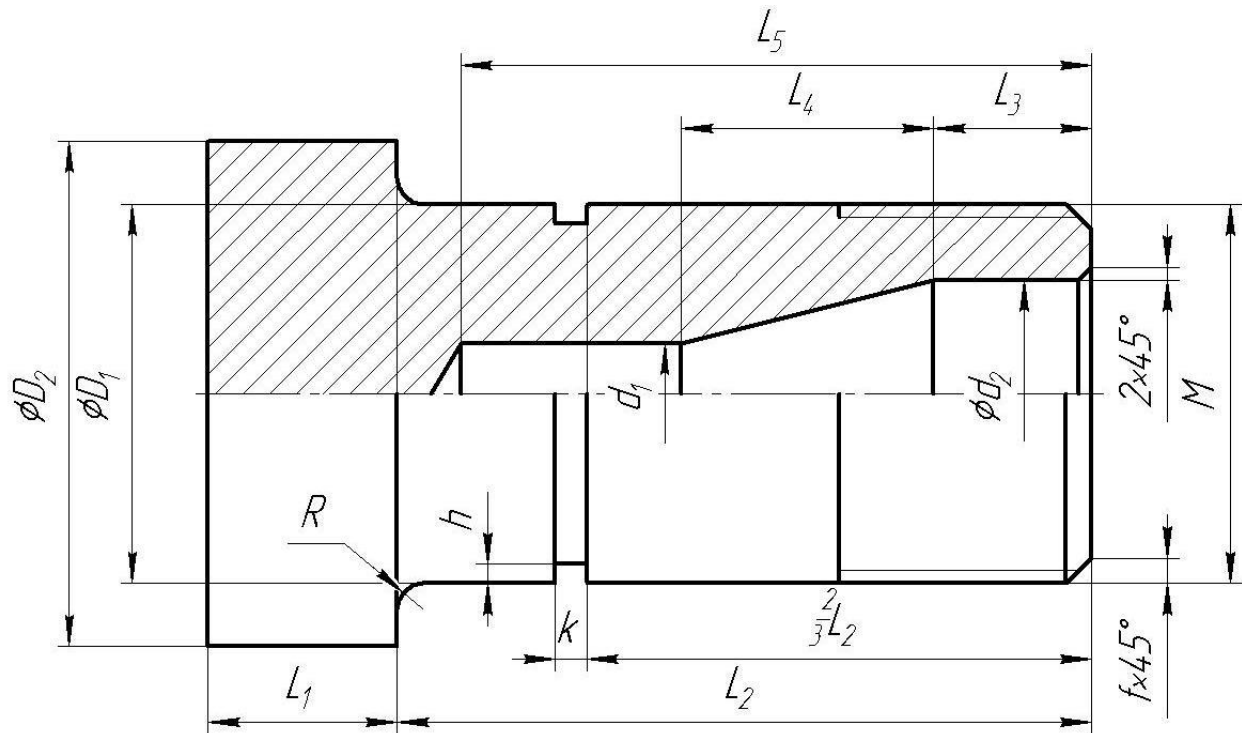


N ^o var.	L_1 , mm	L_2 , mm	L_3 , mm	L_4 , mm	L_5 , mm	M , mm	D_1 , mm	D_2 , mm	D_3 , mm	d_1 , mm	d_2 , mm	d_3 , mm	R , mm	f , mm	k , mm
2	25	60	80	35	96	80	70	80	100	40	46	18	4	4	3
5	25	70	70	30	72	75	60	75	100	38	44	35	2	6	3
8	35	80	50	40	66	85	65	85	100	44	48	40	3	5	4
11	30	65	70	45	69	72	60	72	100	50	54	38	2	6	4
14	30	55	80	50	78	80	60	80	100	35	39	24	4	4	2

Develop a turning program for a CNC machine tool.

Workpiece: cylinder 180 mm long, 80 mm in diameter.

Processing on the outer diameter of the largest step should not be provided, the end processing should be carried out with an allowance of 1 mm.



N ^o var.	L ₁ , mm	L ₂ , mm	L ₃ , mm	L ₄ , mm	L ₅ , mm	M, mm	D ₁ , mm	D ₂ , mm	d ₁ , mm	d ₂ , mm	R, mm	f, mm	h, mm	k, mm
3	30	110	25	40	100	60	60	80	16	32	4	4	3	5
6	35	115	30	20	95	50	50	80	14	25	6	3	3	4
9	25	120	35	30	110	55	55	80	12	25	8	2	2	3
12	30	120	20	40	115	60	60	80	18	30	5	5	2	3
15	35	110	15	30	90	55	55	80	20	28	6	2	2	4

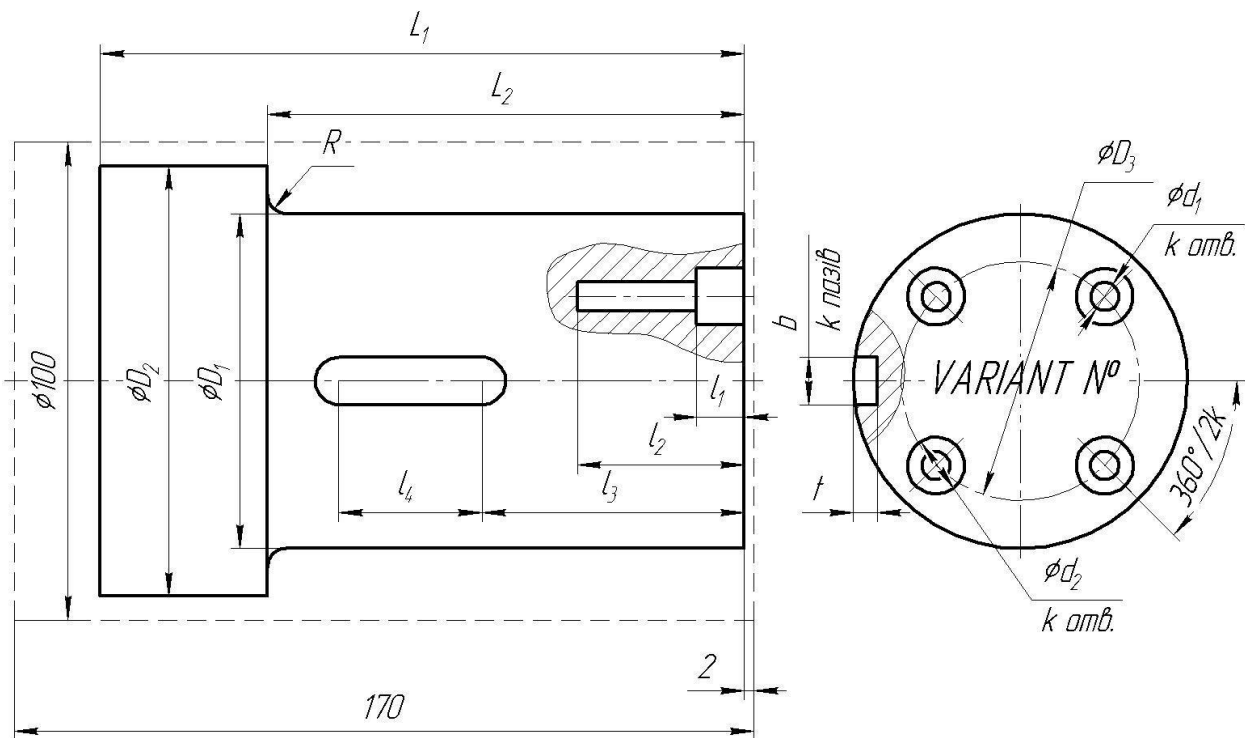
ADDITION B

Variants of tasks for laboratory work №2

Develop a turning and milling program for a CNC machine tool.

The dimensions of the workpiece and the size of the end allowance are shown in the figure with a dashed line.

The format and size of the font, as well as the angle of inclination of the text line, must be chosen arbitrarily, provided that the end surface of the part is most rationally filled. That is, when creating a test line, it is necessary to avoid the intersection of the text with holes, as well as to choose the font size and its scale in such a way as to obtain the largest text font.

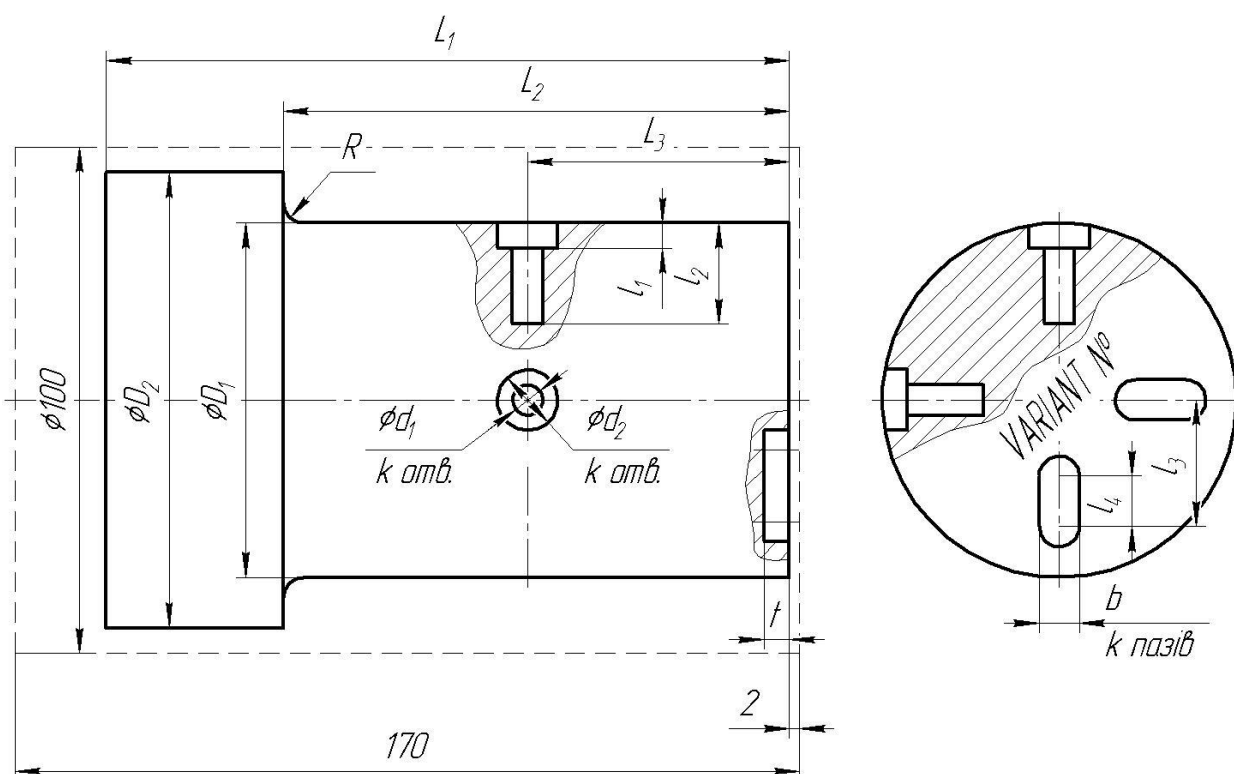


№ var.	L_1 , mm	L_2 , mm	l_1 , mm	l_2 , mm	l_3 , mm	l_4 , mm	D_1 , mm	D_2 , mm	D_3 , mm	d_1 , mm	d_2 , mm	b , mm	t , mm	k , mm	R , mm
1	135	100	10	35	55	30	70	90	50	6	12	10	5	4	4
4	150	120	8	30	40	25	75	95	55	8	16	10	6	3	6
7	140	115	12	25	35	40	80	95	48	10	14	12	7	5	5
10	130	110	8	20	40	20	65	80	40	4	12	14	8	3	4
13	125	105	5	25	45	25	70	95	50	8	14	10	5	6	7

Develop a turning and milling program for a CNC machine tool.

The dimensions of the workpiece and the size of the end allowance are shown in the figure with a dashed line.

The format and size of the font, as well as the angle of inclination of the text line, must be chosen arbitrarily, provided that the end surface of the part is most rationally filled. That is, when creating a test line, it is necessary to avoid the intersection of the text with holes, as well as to choose the font size and its scale in such a way as to obtain the largest text font.

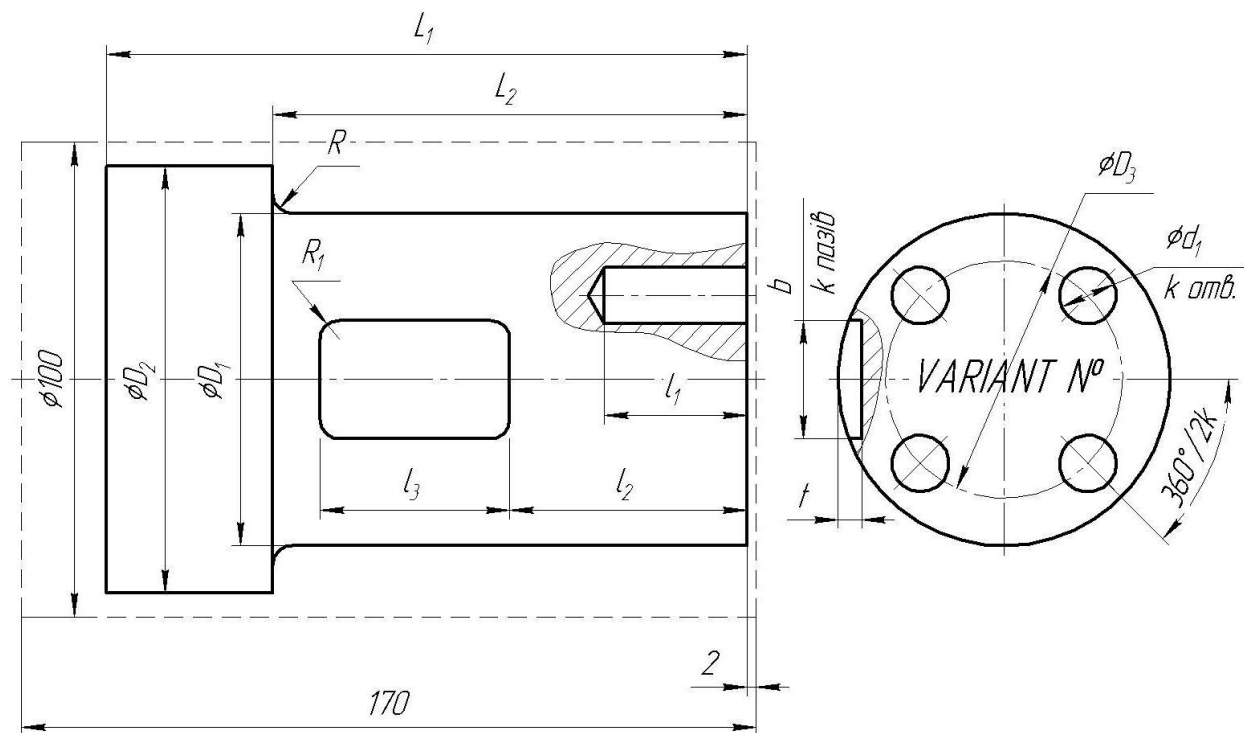


No var.	L ₁ , mm	L ₂ , mm	L ₃ , mm	l ₁ , mm	l ₂ , mm	l ₃ , mm	l ₄ , mm	D ₁ , mm	D ₂ , mm	d ₁ , mm	d ₂ , mm	b, mm	t, mm	k, mm	R, mm
2	135	100	55	10	30	25	10	70	90	6	12	8	5	4	4
5	150	120	40	8	30	30	15	75	95	8	16	10	6	3	6
8	140	115	55	12	25	32	10	80	95	10	14	12	7	5	5
11	130	110	40	8	20	25	10	65	80	4	12	10	5	3	4
14	125	105	45	5	25	30	12	70	95	8	14	6	4	6	7

Develop a turning and milling program for a CNC machine tool.

The dimensions of the workpiece and the size of the end allowance are shown in the figure with a dashed line.

The format and size of the font, as well as the angle of inclination of the text line, must be chosen arbitrarily, provided that the end surface of the part is most rationally filled. That is, when creating a test line, it is necessary to avoid the intersection of the text with holes, as well as to choose the font size and its scale in such a way as to obtain the largest text font.



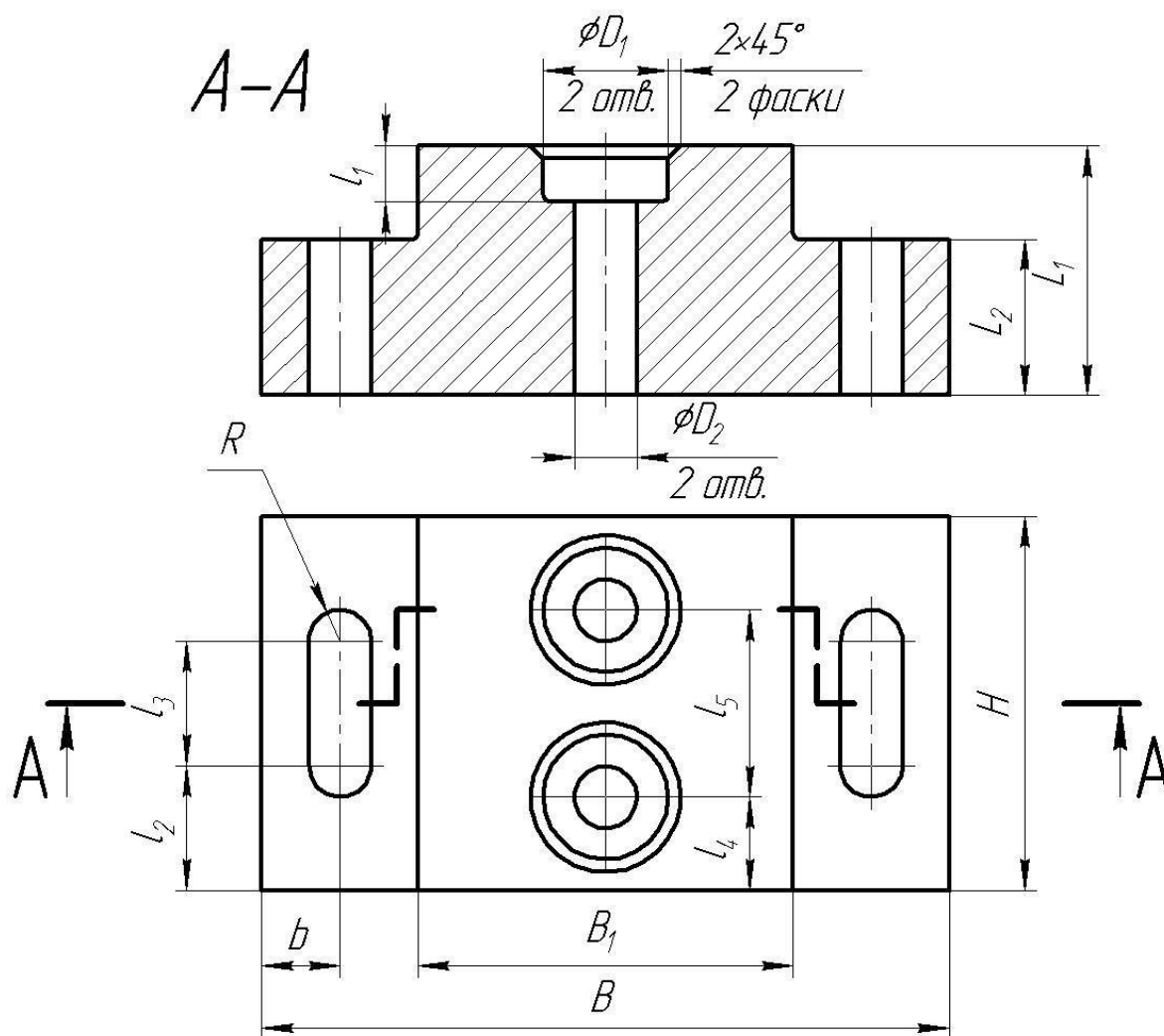
№ var.	L_1 , mm	L_2 , mm	l_1 , mm	l_2 , mm	l_3 , mm	D_1 , mm	D_2 , mm	D_3 , mm	d_1 , mm	b , mm	t , mm	k , mm	R , mm	R_1 , mm
3	135	100	35	50	35	70	90	50	12	20	5	4	4	4
6	150	120	30	40	50	75	95	55	8	25	6	3	6	6
9	140	115	25	35	40	80	95	48	10	18	7	5	5	4
12	130	110	20	40	45	65	80	40	14	22	8	3	4	5
15	125	105	25	45	30	70	95	50	8	15	5	6	7	3

ADDITION C

Variants of tasks for laboratory work №3

Develop a milling program for a CNC machine tool.

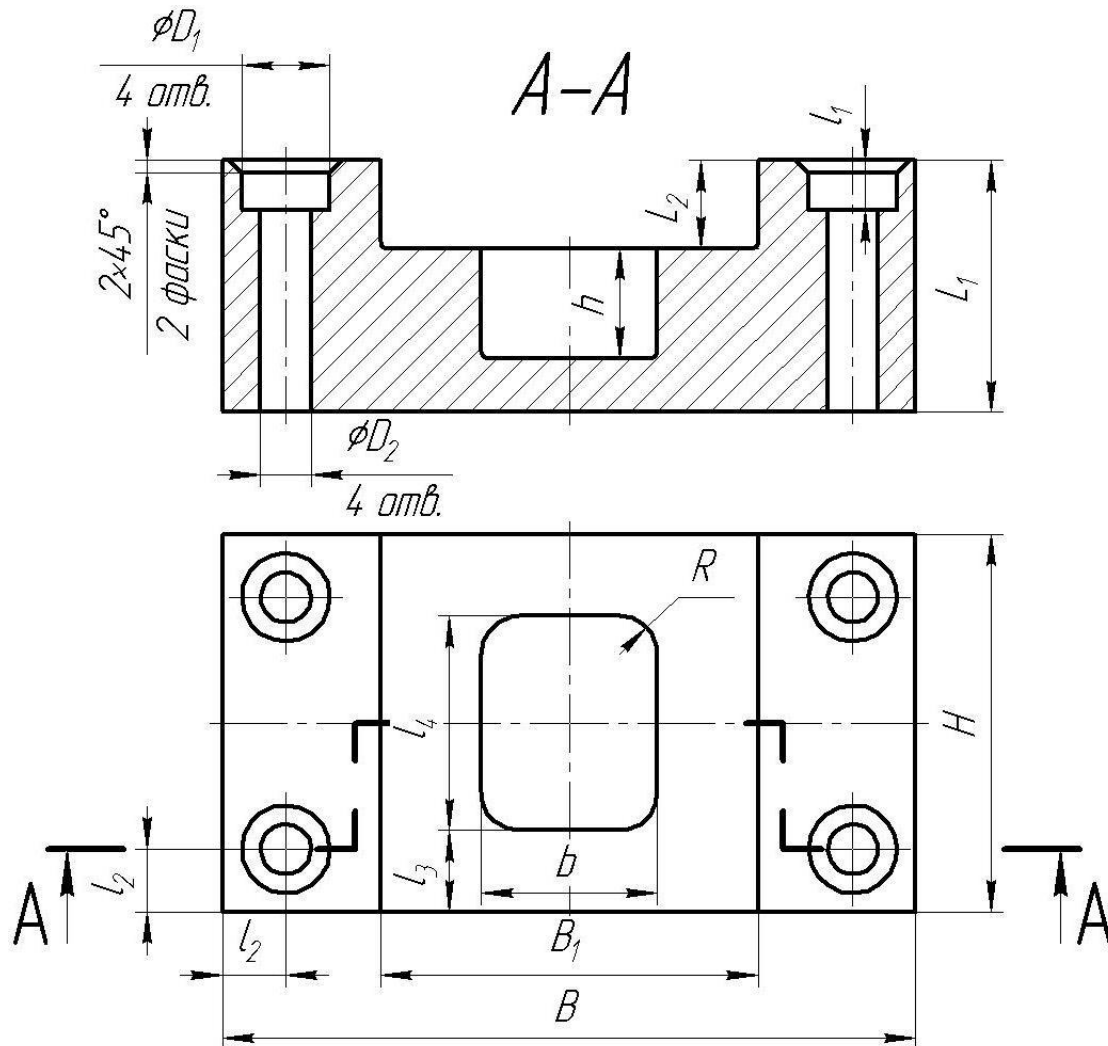
The dimensions and shape of the workpiece should be chosen from the condition of ensuring the necessary parameters of the part. Processing along the outer contour is not expected. Processing of the upper flat surface should be carried out with an allowance of 2 mm.



№ var.	H, mm	B, mm	B ₁ , mm	L ₁ , mm	L ₂ , mm	l ₁ , mm	l ₂ , mm	l ₃ , mm	l ₄ , mm	l ₅ , mm	D ₁ , mm	D ₂ , mm	b, mm	R, mm
1	60	110	60	40	25	9	15	30	15	30	20	10	12,5	5
4	70	120	65	60	30	12	15	40	17,5	35	24	12	15	7
7	75	130	70	60	35	15	15	40	17,5	40	26	14	15	8
10	80	140	60	70	30	10	30	35	15	50	24	12	20	10
13	85	150	50	75	40	14	20	30	20	45	22	10	25	12

Develop a milling program for a CNC machine tool.

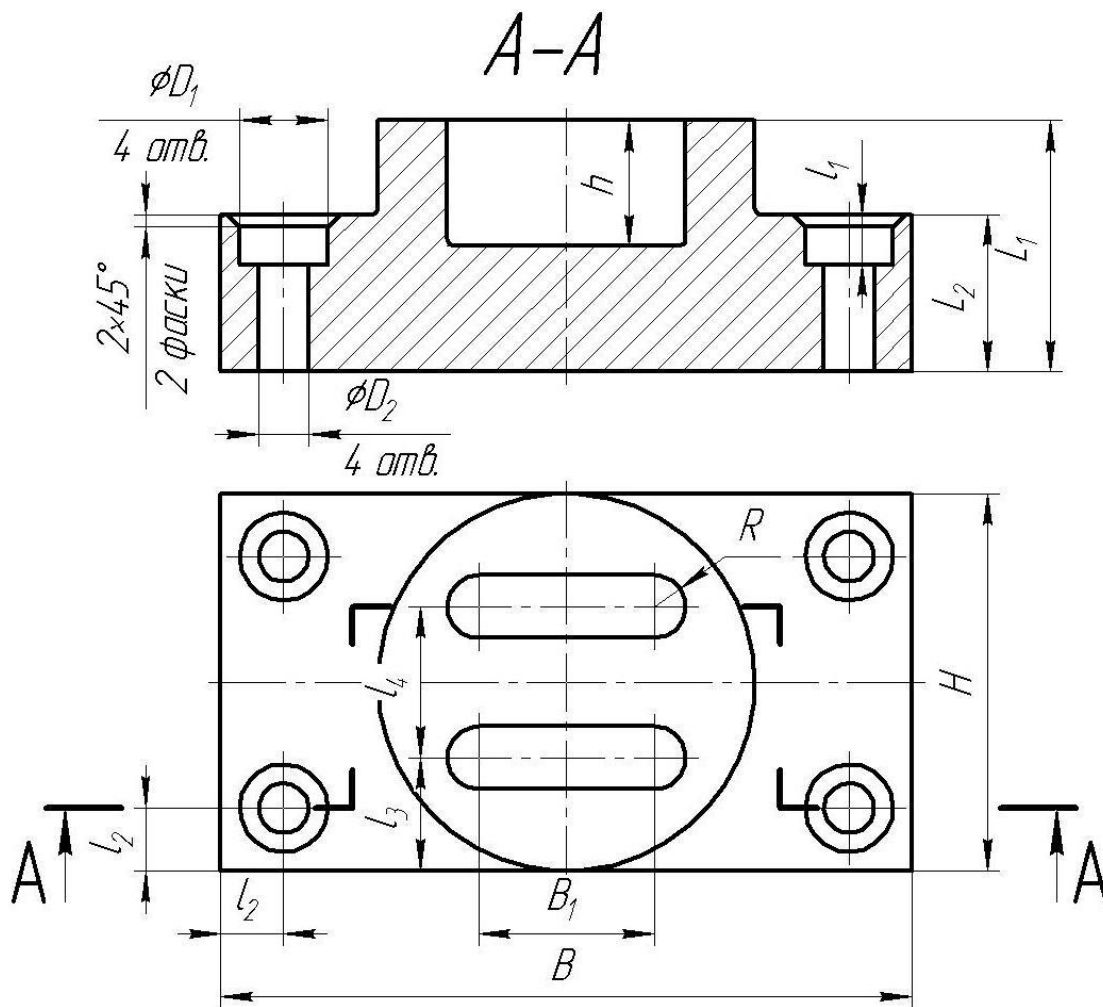
The dimensions and shape of the workpiece should be chosen from the condition of ensuring the necessary parameters of the part. Processing along the outer contour is not expected. Processing of the upper flat surface should be carried out with an allowance of 2 mm.



№ var.	H, mm	B, mm	B ₁ , mm	L ₁ , mm	L ₂ , mm	l ₁ , mm	l ₂ , mm	l ₃ , mm	l ₄ , mm	D ₁ , mm	D ₂ , mm	b, mm	h, mm	R, mm
2	60	110	60	40	15	8	10	18	24	14	8	30	18	6
5	70	120	65	60	20	15	12	15	40	20	12	25	15	5
8	75	130	70	60	25	10	15	17,5	40	22	14	35	18	8
11	80	140	60	70	25	20	17	15	50	24	12	20	20	4
14	85	150	75	75	30	20	17	22,5	40	22	10	40	25	7

Develop a milling program for a CNC machine tool.

The dimensions and shape of the workpiece should be chosen from the condition of ensuring the necessary parameters of the part. Processing along the outer contour is not expected. Processing of the upper flat surface should be carried out with an allowance of 2 mm.

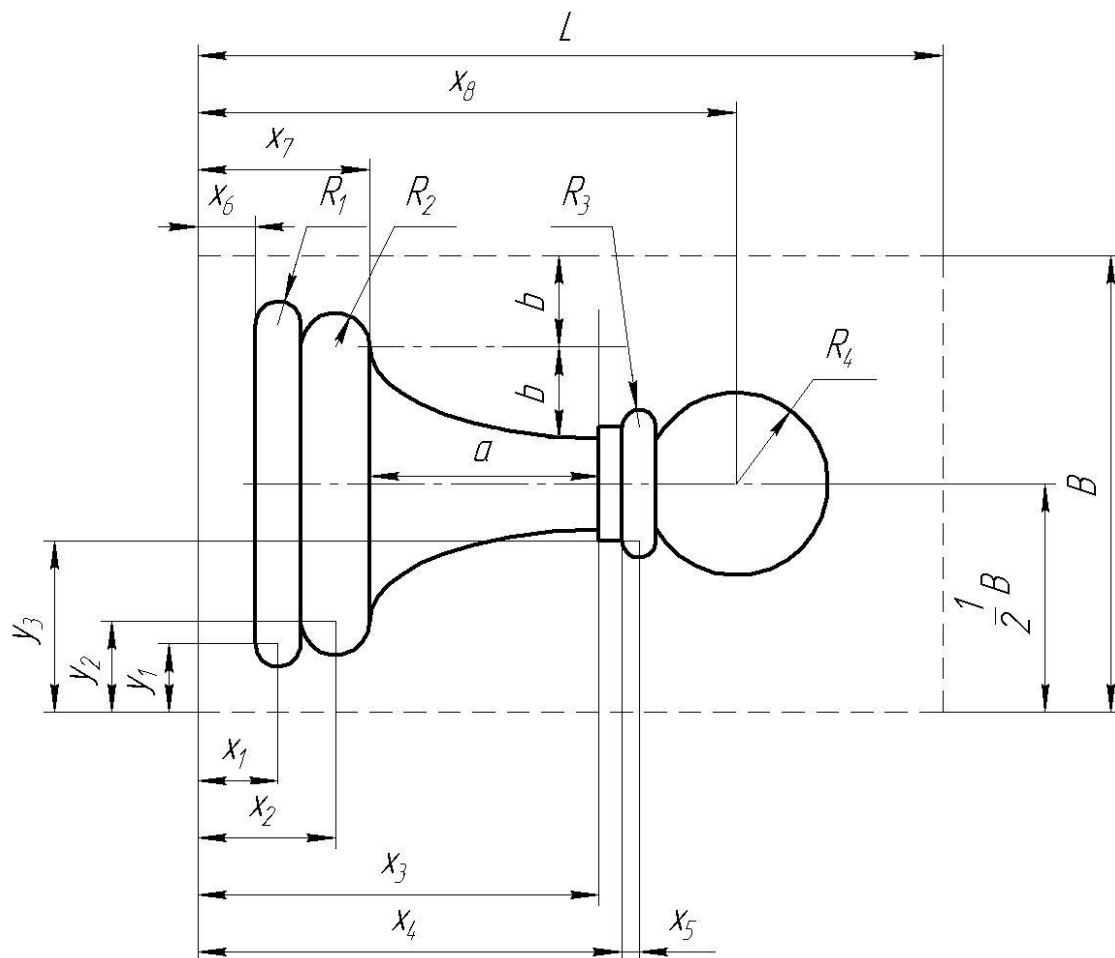


№ var.	H, mm	B, mm	B ₁ , mm	L ₁ , mm	L ₂ , mm	l ₁ , mm	l ₂ , mm	l ₃ , mm	l ₄ , mm	D ₁ , mm	D ₂ , mm	h, mm	R, mm
3	60	110	28	40	25	9	12	18	24	14	8	20	5
6	70	120	30	60	20	12	15	15	40	20	12	15	7
9	75	130	32	60	25	15	15	17,5	40	22	14	18	8
12	80	140	20	70	25	10	17	15	50	24	12	20	10
15	85	150	25	75	30	14	17	22,5	40	22	10	25	12

ADDITION D

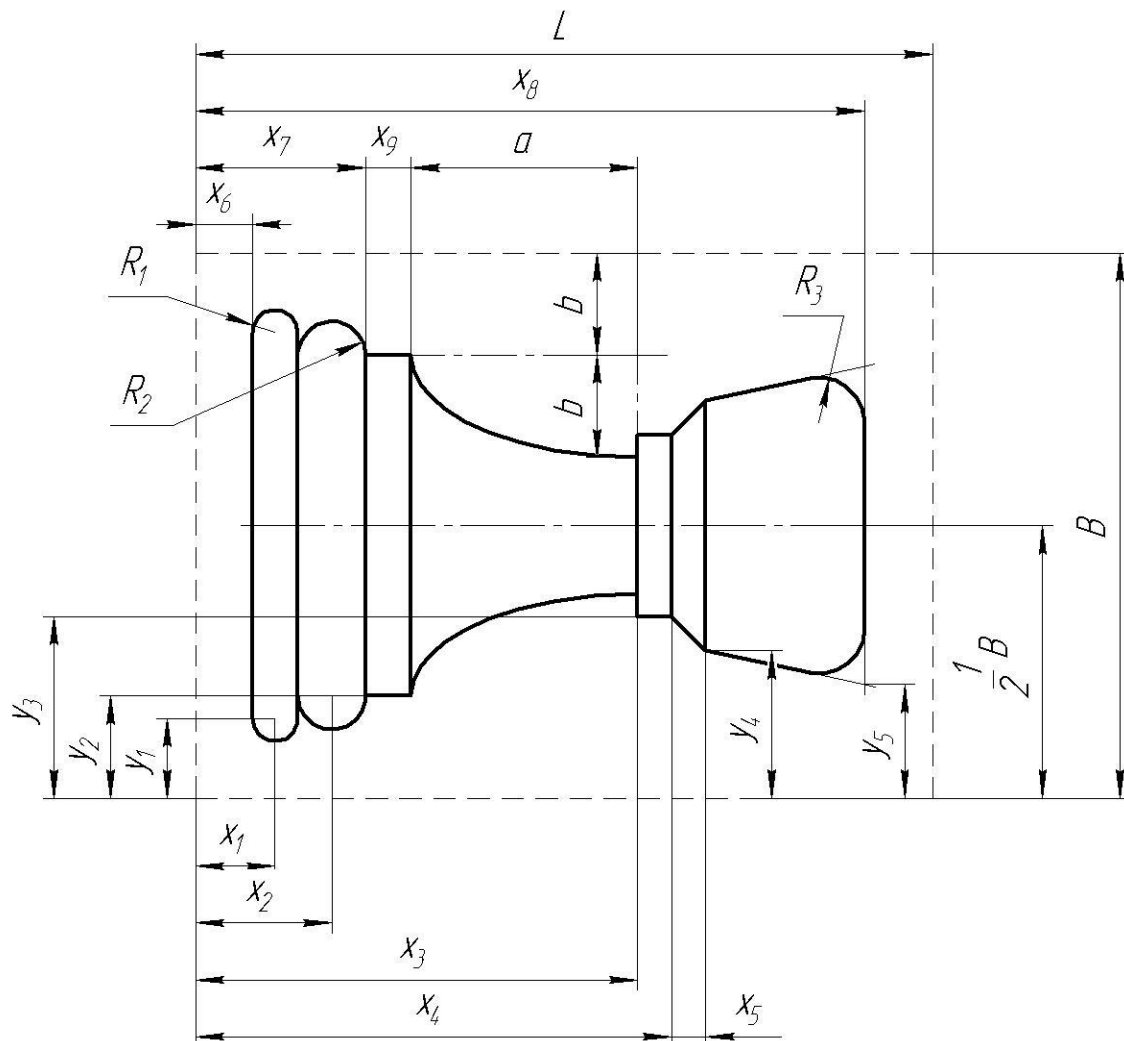
Variants of tasks for laboratory work №4

Create a foundry mold for the production of a three-dimensional chess figure. The length and width of the workpiece are indicated in the figure, the height should be chosen equal to half the width.



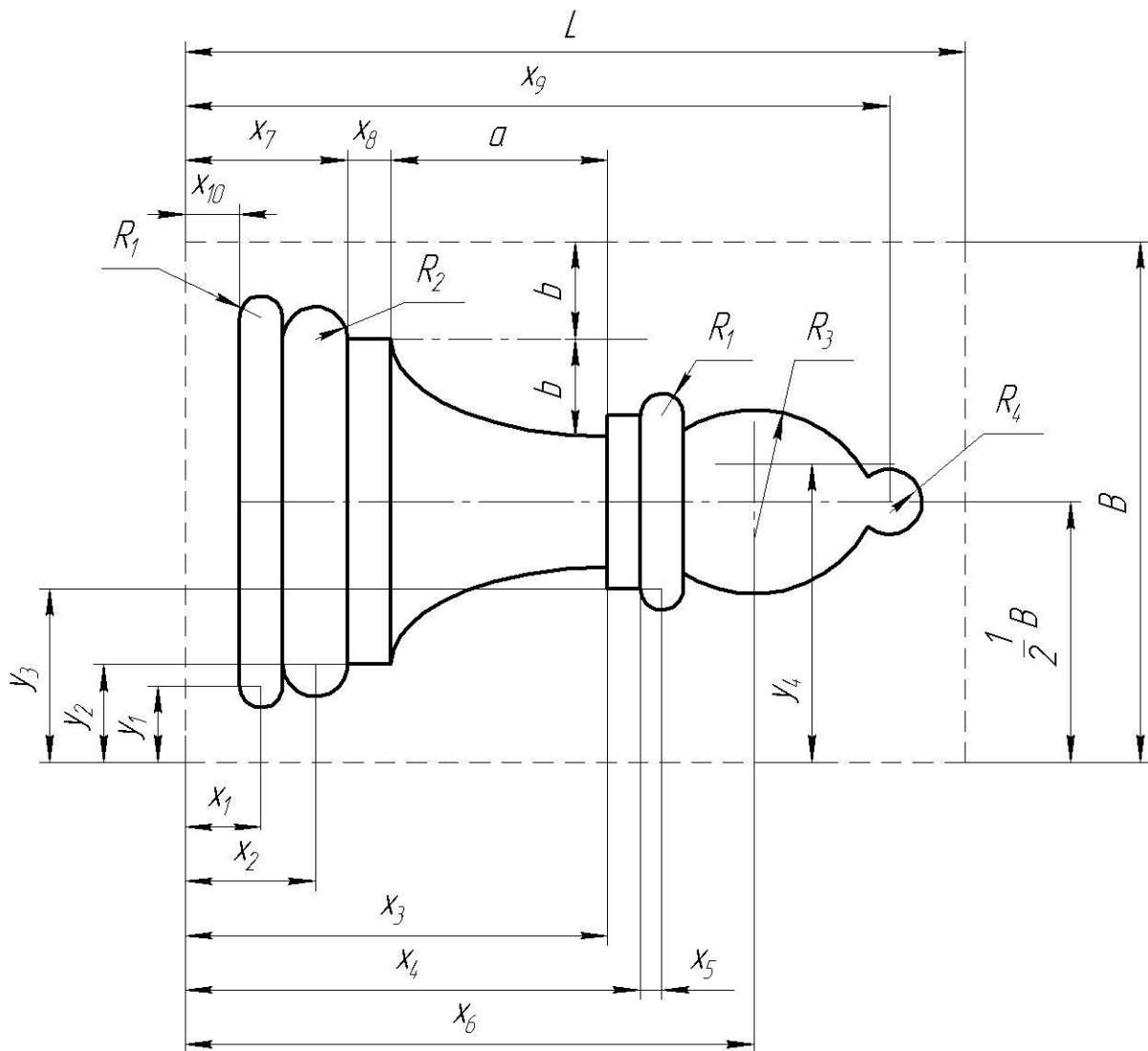
№ var.	B, mm	L, mm	Dimen. of the ellipse, mm		x ₁ , mm	x ₂ , mm	x ₃ , mm	x ₄ , mm	x ₅ , mm	x ₆ , mm	x ₇ , mm	x ₈ , mm	y ₁ , mm	y ₂ , mm	y ₃ , mm	R ₁ , mm	R ₂ , mm	R ₃ , mm	R ₄ , mm
			a	b															
1	100	165	50	20	17,5	30	87,5	92,5	3,75	12,5	37,5	117,5	15	20	37,5	5	7,5	3,75	20
5	80	130	40	16	14	24	70	74	3	10	30	94	12	16	30	4	6	3	16
9	60	100	30	12	10,5	18	52,5	55,5	2,25	7,5	22,5	70,5	9	12	22,5	3	4,5	2,25	12
13	40	65	20	8	7	12	35	37	1,5	5	15	47	6	8	15	2	3	1,5	8

Create a foundry mold for the production of a three-dimensional chess figure. The length and width of the workpiece are indicated in the figure, the height should be chosen equal to half the width.



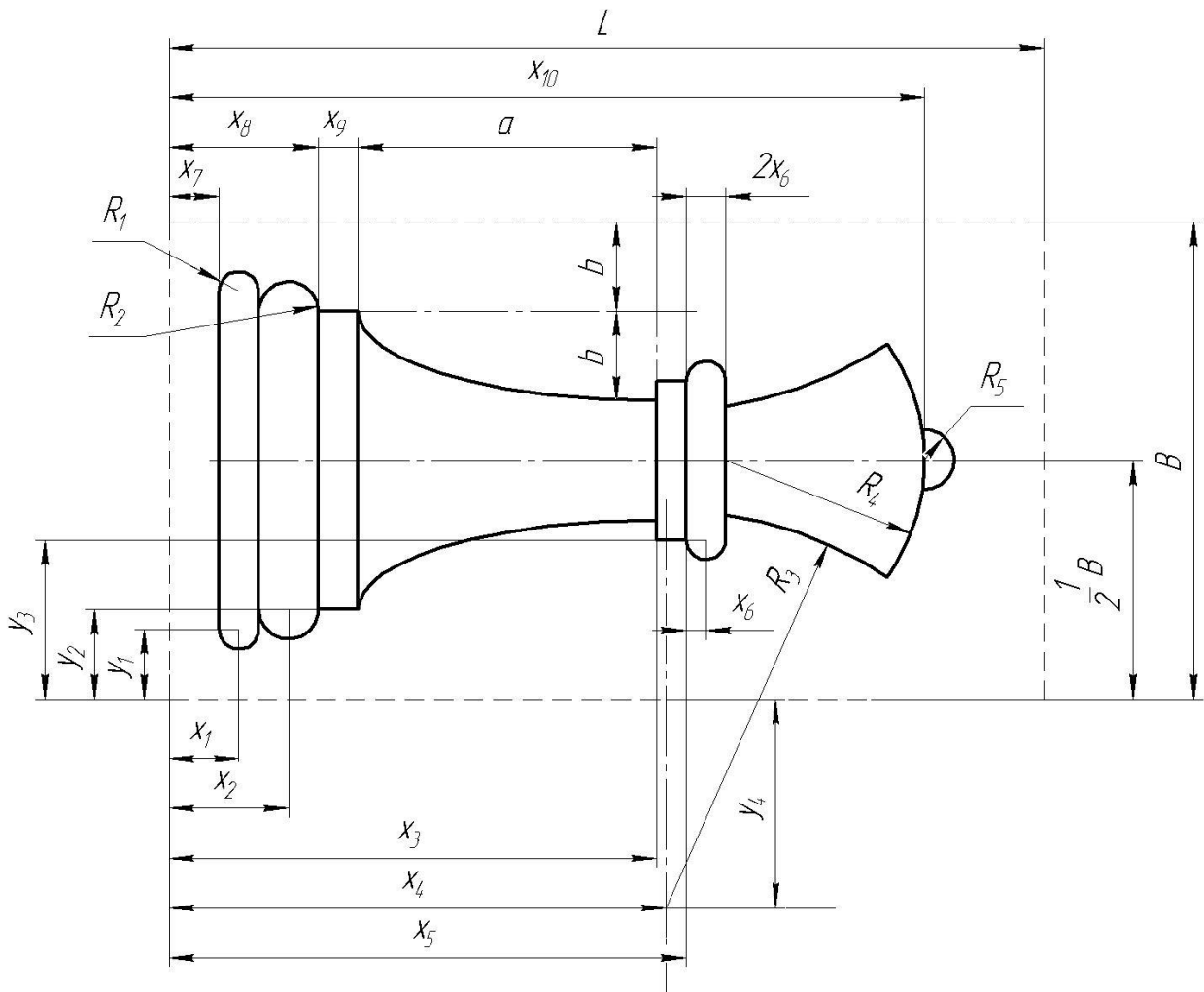
№ var.	B, mm	L, mm	Dimen. of the ellipse, mm		x ₁ , mm	x ₂ , mm	x ₃ , mm	x ₄ , mm	x ₅ , mm	x ₆ , mm	x ₇ , mm	x ₈ , mm	x ₉ , mm	y ₁ , mm	y ₂ , mm	y ₃ , mm	y ₄ , mm	y ₅ , mm	R ₁ , mm	R ₂ , mm	R ₃ , mm
			a	b																	
2	120	165	50	22,5	17,5	30	97,5	105	7,5	12,5	37,5	147,5	10	17,5	22,5	40	32,5	25	5	7,5	10
6	96	130	40	18	14	24	78	84	6	10	30	118	8	14	18	32	26	20	4	6	8
10	72	100	30	13,5	10,5	18	58,5	63	4,5	7,5	22,5	88,5	6	10,5	13,5	24	19,5	15	3	4,5	6
14	48	65	20	9	7	12	39	42	3	5	15	59	4	7	9	16	13	10	2	3	4

Create a foundry mold for the production of a three-dimensional chess figure. The length and width of the workpiece are indicated in the figure, the height should be chosen equal to half the width.



№ var.	B, mm	L, mm	Dimen. of the ellipse, mm		x ₁ , mm	x ₂ , mm	x ₃ , mm	x ₄ , mm	x ₅ , mm	x ₆ , mm	x ₇ , mm	x ₈ , mm	x ₉ , mm	x ₁₀ , mm	y ₁ , mm	y ₂ , mm	y ₃ , mm	y ₄ , mm	R ₁ , mm	R ₂ , mm	R ₃ , mm	R ₄ , mm
			a	b																		
3	120	180	50	22,5	17,5	30	97,5	105	5	131,25	37,5	10	162,5	12,5	17,5	22,5	40	68,75	5	7,5	30	7,5
7	96	144	40	18	14	24	78	84	4	105	30	8	130	10	14	18	32	55	4	6	24	6
11	72	108	30	13,5	10,5	18	58,5	63	3	78,75	22,5	6	97,5	7,5	10,5	13,5	24	41,25	3	4,5	18	4,5
15	48	72	20	9	7	12	39	42	2	52,5	15	4	65	5	7	9	16	27,5	2	3	12	3

Create a foundry mold for the production of a three-dimensional chess figure. The length and width of the workpiece are indicated in the figure, the height should be chosen equal to half the width.



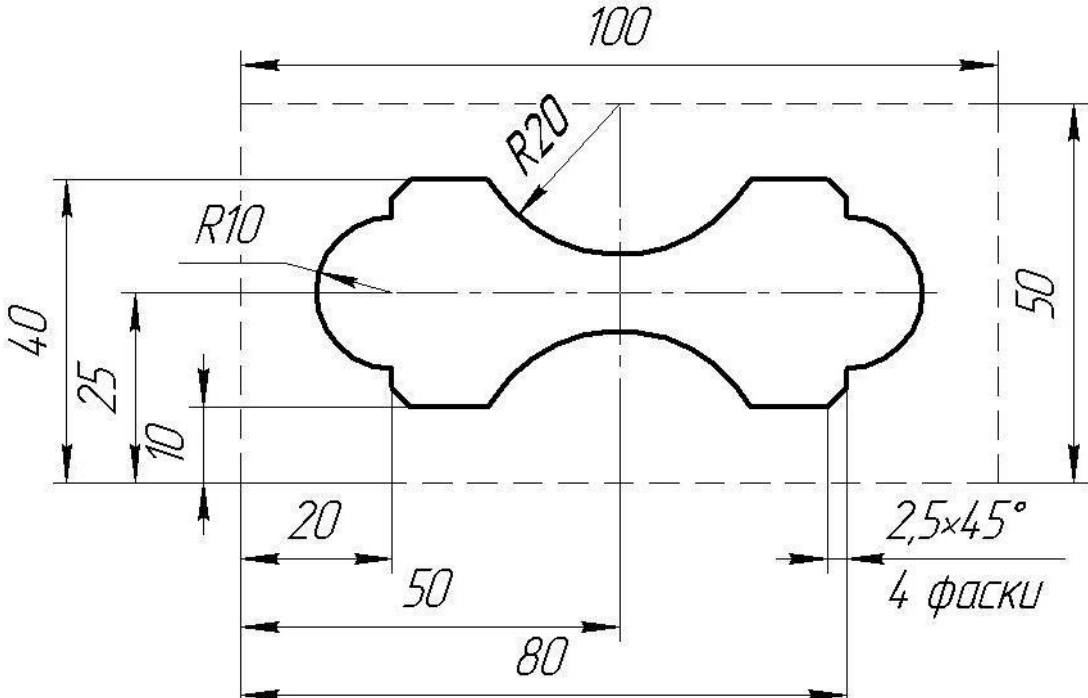
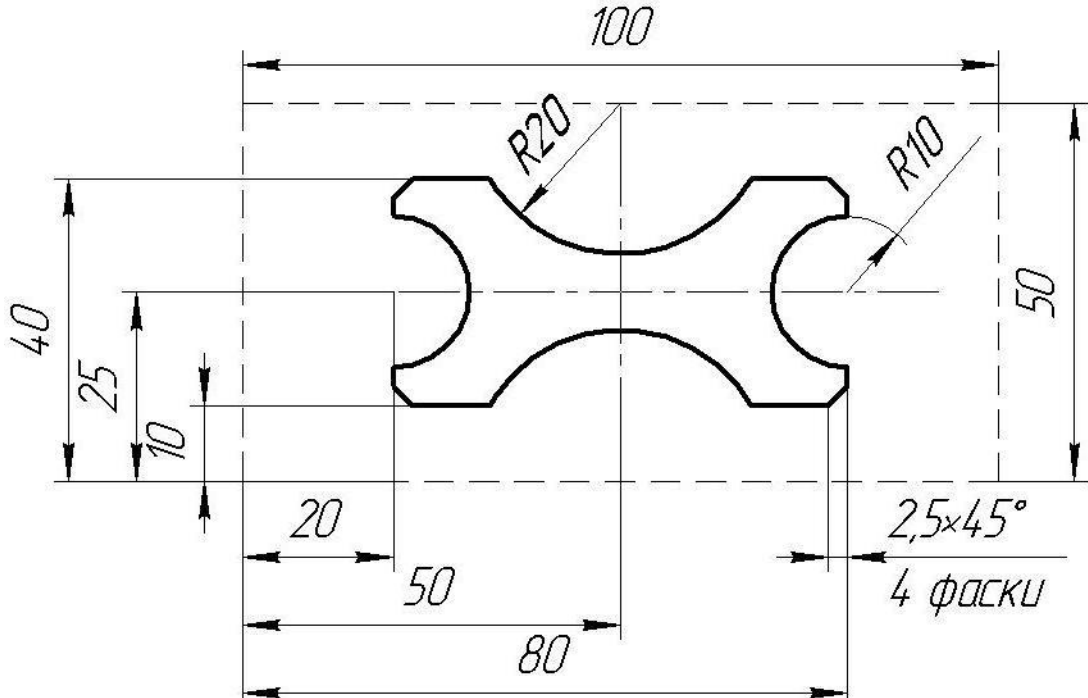
№ var.	B, mm	L, mm	Dimen. of the ellipse, mm		x ₁ , mm	x ₂ , mm	x ₃ , mm	x ₄ , mm	x ₅ , mm	x ₆ , mm	x ₇ , mm	x ₈ , mm	x ₉ , mm	x ₁₀ , mm	y ₁ , mm	y ₂ , mm	y ₃ , mm	y ₄ , mm	R ₁ , mm	R ₂ , mm	R ₃ , mm	R ₄ , mm	R ₅ , mm
			a	b																			
3	120	220	75	22,5	17,5	30	122,5	125	130	5	12,5	37,8	10	190	17,5	22,5	40	52,5	5	7,5	100	50	7,5
7	96	176	60	18	14	24	98	100	104	4	10	30	8	152	14	18	32	42	4	6	80	40	6
11	72	132	45	13,5	10,5	18	73,5	75	78	3	7,5	22,5	6	114	10,5	13,5	24	31,5	3	4,5	60	30	4,5
16	48	88	30	9	7	12	49	50	52	2	5	15	4	76	7	9	16	21	2	3	40	20	3

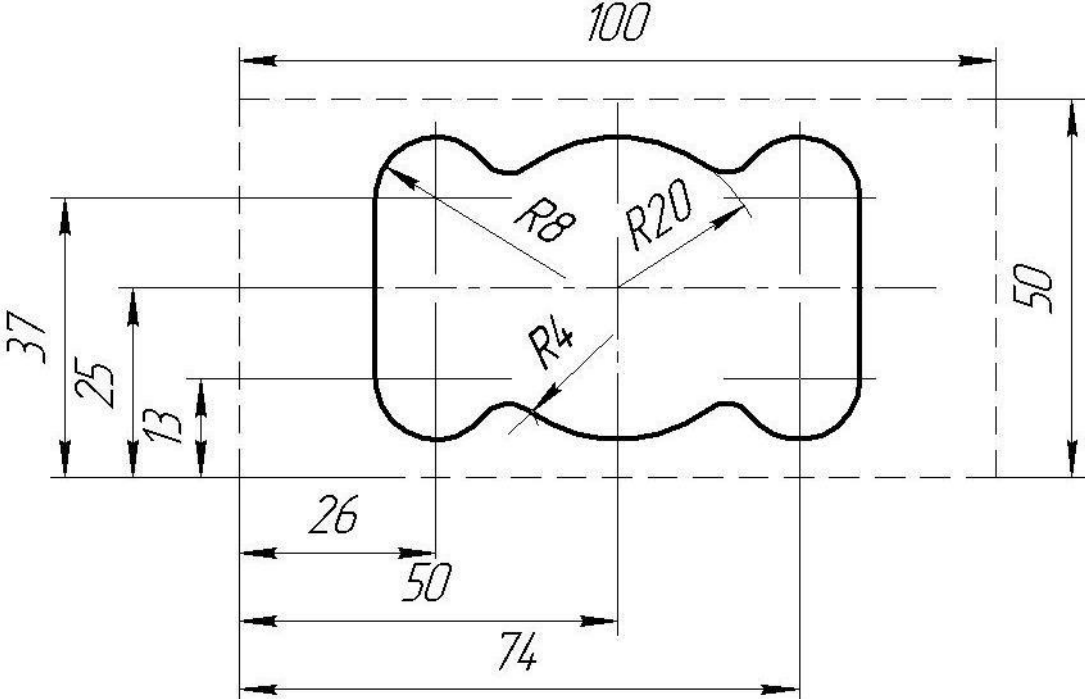
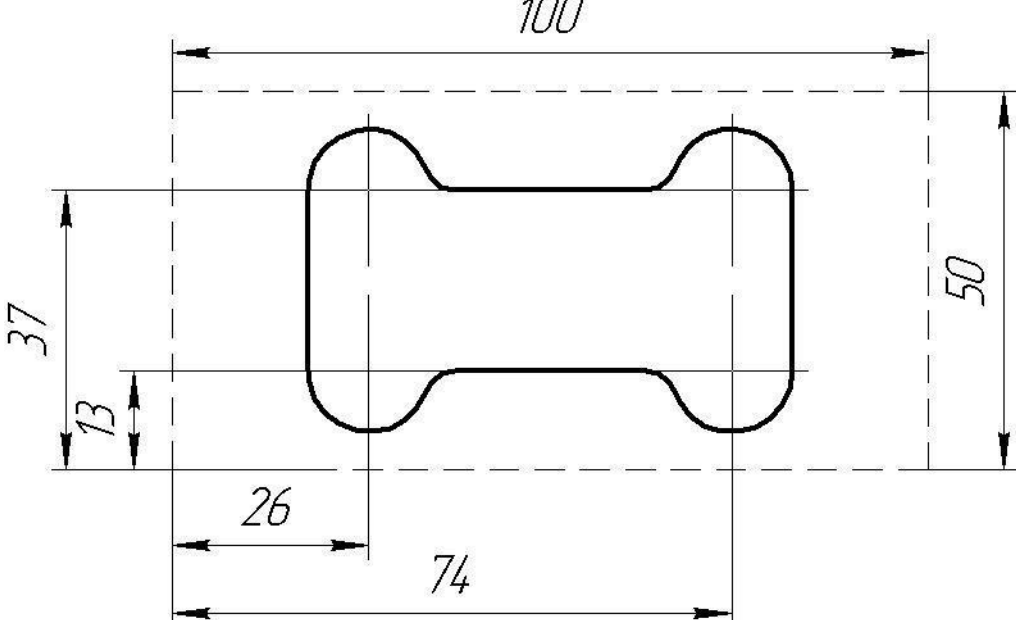
ADDITION E

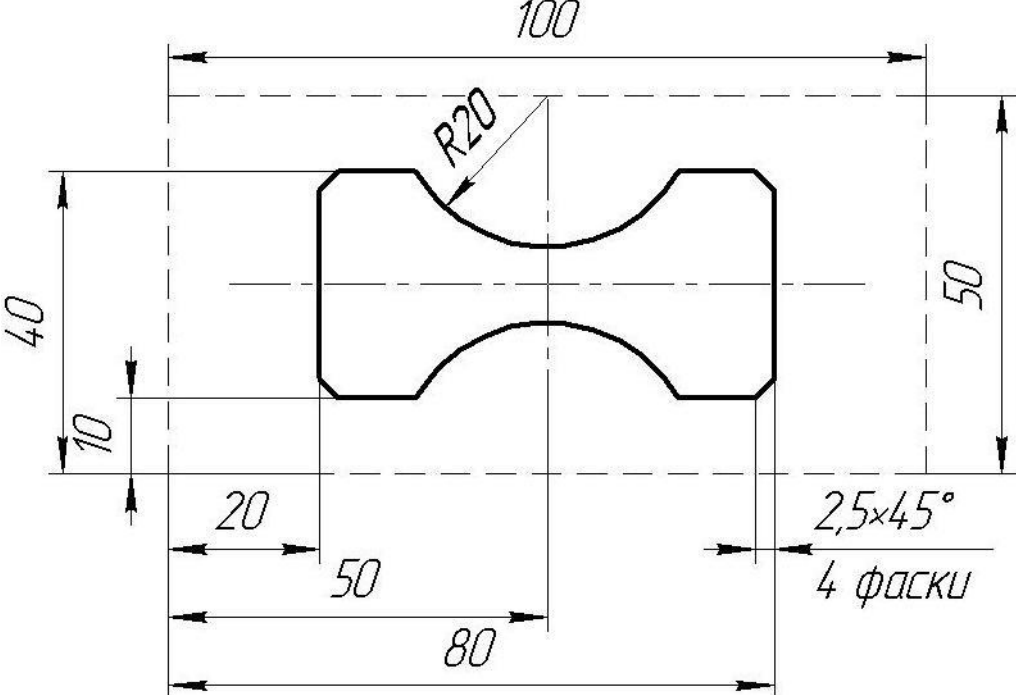
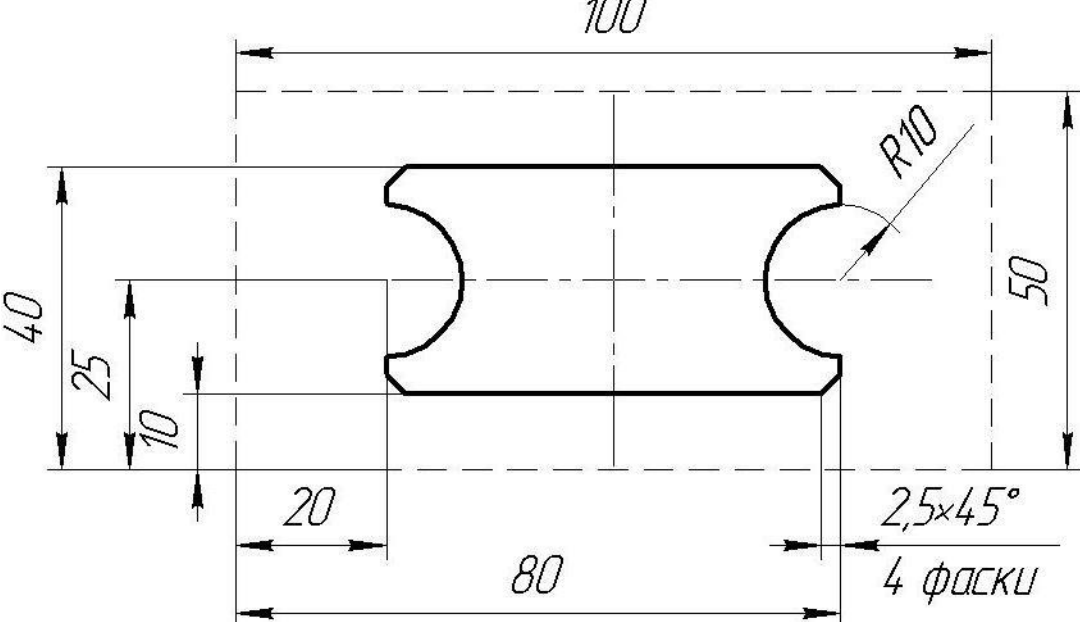
Variants of tasks for laboratory work №5

The length and width of the workpiece are indicated by a dashed line, the thickness is individual for each variant.

<p>1 Create a matrix of the depicted form. The thickness of the workpiece 5 mm. Angle: left, 5°.</p>	<p>2 Create a punch of the shown shape. The thickness of the workpiece 5 mm. Angle: right, 5°.</p>
<p>3 Create a matrix of the depicted form. The thickness of the workpiece 7 mm. Angle: left, 7°.</p>	<p>4 Create a punch of the shown shape. The thickness of the workpiece 7 mm. Angle: right, 7°.</p>

	
<p>5 Create a matrix of the depicted form. The thickness of the work-piece 10 mm. Angle: left, 5°.</p>	<p>6 Create a punch of the shown shape. The thickness of the workpiece 10 mm. Angle: right, 5°.</p>
	
<p>7 Create a matrix of the depicted form. The thickness of the work-piece 10 mm. Angle: left, 10°.</p>	<p>8 Create a punch of the shown shape. The thickness of the workpiece 10 mm. Angle: right, 10°.</p>

	
<p>9 Create a matrix of the depicted form. The thickness of the workpiece 5 mm. Angle: left, 10°.</p>	<p>10 Create a punch of the shown shape. The thickness of the workpiece 5 mm. Angle: right, 10°.</p>
	
<p>11 Create a matrix of the depicted form. The thickness of the workpiece 7 mm. Angle: left, 10°.</p>	<p>12 Create a punch of the shown shape. The thickness of the workpiece 7 mm. Angle: right, 10°.</p>

	
<p>13 Create a matrix of the depicted form. The thickness of the work-piece 10 mm. Angle: left, 7°.</p>	<p>14 Create a punch of the shown shape. The thickness of the workpiece 10 mm. Angle: right, 7°.</p>
	
<p>15 Create a matrix of the depicted form. The thickness of the work-piece 8 mm. Angle: left, 10°.</p>	<p>16 Create a punch of the shown shape. The thickness of the workpiece 8 mm. Angle: right, 10°.</p>

CONTENT

INTRODUCTION.....	3
History about the Delcam software.....	4
LABORATORY LESSON №1 TURNING IN DELCAM FEATURECAM.....	7
1.1 Goal	7
1.2 Brief theoretical information.....	7
1.3 An example of creating a typical turning part	10
1.4 Tasks for the laboratory session.....	32
1.5 Requirements for a report on the performance of a laboratory session	33
1.6 Control questions	33
LABORATORY LESSON №2 TURNING AND MILLING IN DELCAM FEATURECAM.....	34
2.1 Goal	34
2.2 An example of creating a typical turning-milling part	34
2.3 Tasks for the laboratory session.....	46
2.4 Requirements for a report on the performance of a laboratory session	46
2.5 Control questions	47
LABORATORY LESSON №3 2.5D MILLING IN DELCAM FEATURECAM	48
3.1 Goal	48
3.2 An example of creating a typical 2.5D milling part	48
3.3 Tasks for the laboratory session.....	63
3.4 Requirements for a report on the performance of a laboratory session	64
3.5 Control questions	64
LABORATORY LESSON №4 3.5D MILLING IN DELCAM FEATURECAM	65
4.1 Goal	65
4.2 An example of creating a typical 3.5D milling part	65
4.3 Tasks for the laboratory session.....	77
4.4 Requirements for a report on the performance of a laboratory session	78
4.5 Control questions	78
LABORATORY LESSON №5 EDM IN DELCAM FEATURECAM.....	79

5.1 Goal	79
5.2 An example of creating a typical EDM part	79
5.3 Tasks for the laboratory session.....	87
5.4 Requirements for a report on the performance of a laboratory session	88
5.5 Control questions	88
RECOMMENDED BOOKS	89
ADDITION A Variants of tasks for laboratory work №1	90
ADDITION B Variants of tasks for laboratory work №2	93
ADDITION C Variants of tasks for laboratory work №3	96
ADDITION D Variants of tasks for laboratory work №4	99
ADDITION E Variants of tasks for laboratory work №5.....	103