MINISTRY OF EDUCATION AND SCIENCE OF UKRAINE

NATIONAL TECHNICAL UNIVERSITY
“KHARKIV POLYTECHNIC INSTITUTE”

METHODICAL INSTRUCTIONS

to perform laboratory work

in the course
“CAD of hydroturbines, reversible hydraulic machines, small,
mini and micro hydropower plants”
for full-time and part-time students
in the specialty “Industry Engineering”,
educational program “Industry Engineering”

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Reviewer: Assoc. Prof. Irina Tynyanova

Department of Hydraulic Machines named after G.F. Proskura

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Introduction

This set of laboratory works is aimed at providing students with systematic practical training in modern computer-aided design (CAD) using the SolidWorks software package. The course is focused on the development of skills in three-dimensional modeling of parts, preparation of technical drawings, and implementation of engineering analysis with integrated SolidWorks modules. Such competencies are essential for future engineers in the fields of mechanical engineering, hydropower, and related industries.

The laboratory course includes four works that gradually guide students from the basics of 3D part modeling to advanced analysis tasks.

In the first laboratory work, students learn to build solid models of parts in SolidWorks based on traditional engineering drawings.

The second work introduces parametric modeling principles through the construction of a membrane model, emphasizing the use of sketching tools and geometric relations.

The third laboratory work focuses on creating a 3D model of a shaft and preparing its complete technical drawing, consolidating skills in building assemblies and applying standards of engineering graphics.

Finally, the fourth laboratory work is devoted to strength analysis of a part using the SolidWorks Simulation module, where students perform static calculations and assess the stress-strain state and safety factor of the modeled component.

By completing these laboratory works, students will strengthen their theoretical knowledge of engineering graphics and computer modeling, while gaining practical experience in CAD tools that are widely used in design offices and industrial enterprises.

Mastery of SolidWorks functions, from modeling to engineering analysis, will significantly enhance their professional competencies and prepare them for solving real engineering problems in their future careers.

Lab Work №1: Building solid models of parts in the SolidWorks environment based on drawings

1. General provisions

Laboratory work is carried out within the framework of studying the discipline dedicated to computer modeling of parts in modern computer-aided design (CAD) systems. The main goal of this work is to develop practical skills in students to create three-dimensional solid-state models of parts according to the provided drawings using the SolidWorks software package.

SolidWorks is one of the leading computer-aided design systems that allows you to model both individual parts and assembly units, create design and construction documentation, perform engineering strength calculations, flow analysis, and other technological tasks.

As part of this laboratory work, each student receives an individual version of the task. The versions include a set of drawings of parts of varying geometric complexity. The student's task is to create a solid model of the part in the SolidWorks environment according to the provided dimensions and drawing requirements.

2. Theoretical part

2.1. General information about solid modeling

Solid modeling is the process of creating a computer model of an object that has fully defined geometry, mass, material properties, and precise dimensions. Unlike wireframe or surface modeling, a solid model is a complete virtual analogue of a real part.

SolidWorks provides a wide range of capabilities for creating solid models from 2D sketches and applying a variety of solid modeling operations to them. These operations include:

- **Extrude Boss / Base** (Extrude Body) – Create a solid body by extruding the sketch in a perpendicular direction.

- **Revolve** – creates a body of revolution based on a sketch, rotated around a specified axis.
- **Cut-** Extrude – creating through or blind holes and slots.
- **Fillet** and **Chamfer** – processing corners and edges of parts.

2.2. Basics of working in SolidWorks

The process of creating a solid model in SolidWorks includes several main steps:

1. Creating a new part – opening a new file in *.SLDPRT format.
2. Select the base plane (Front plane, Top plane, Right plane).
3. Sketching – creating a 2D profile of the future body.
4. Applying dimensions and relationships in a sketch.
5. Using solid modeling operations.
6. Adding auxiliary elements – cutouts, holes, chamfers, roundings.
7. Checking the geometry against the drawing.
8. Saving the model in *.SLDPRT and .STEP formats* .

3. Procedure for performing the work

1. Get acquainted with the individual version of the task, which includes a drawing of the part with all the necessary dimensions.
2. Identify the main operations that will be used to create the model.
3. Open a new part file in SolidWorks.
4. Select the base plane for the first sketch (Front Plane, Top Plane or Right Plane).
5. Build a sketch according to the drawing - the profile of the future part.
6. Apply the required dimensions (Smart Dimension) and relationships (Relations) to make the sketch fully defined.
7. Perform an extrusion or rotation operation to obtain the basic shape of the part.

8. Create additional elements (holes, grooves, cutouts) using additional sketches.

9. Add fillets or chamfers on the corresponding faces.

10. Check that the resulting model matches the drawing.

11. Save the part in SLDPRT and STEP format.

12. Prepare a report with attached screenshots of the main stages of modeling.

4. Requirements for preparing a report

The report must contain:

- Title page (discipline name, option number, student's last name and group).
- The purpose of the work.
- Description of the execution sequence.
- List of tools and commands used.
- Screenshots of all key stages of model building.
- Images of the finished 3D model in three standard views.
- Conclusions on the work performed.

Conclusion

This laboratory work allows students to acquire skills in creating three-dimensional models of parts from drawings, which is a mandatory component of the professional training of future mechanical engineers and specialists in the field of hydropower. Mastery of SolidWorks tools and the ability to analyze drawings are important competencies for work in modern design offices and production departments.

Lab Work №2: Creating a 3D model of a flat membrane using SolidWorks

Introduction

In this laboratory work, students will get acquainted with the basics of parametric modeling in SolidWorks – a modern CAD system that is widely used to create three-dimensional models in mechanical engineering, design and other industries. The work involves the development of a fully defined sketch of a flat membrane, which is the basis for creating a 3D model using the Extrude operation. In addition, students will gain practical skills in working with tools such as Circular Sketch Pattern (Circular sketch array), Mirror (Mirror image), Straight Slot, Polygon, Trim Entities (Trimming) and others. It is also intended to use the Fillet command to smooth the edges of the model.

The development of this part is aimed at consolidating practical skills in working with SolidWorks, since the correct construction of sketches is a key stage in the creation of any three-dimensional element. The sketch is the foundation on which the entire subsequent model is based, so it is important to be able to use the entire set of SolidWorks functionality to obtain fully defined geometry.

1. Purpose of the work

The purpose of this laboratory work is to develop practical skills in working with the SolidWorks program, in particular:

- Mastering the principles of building fully defined sketches using dimensions (Smart Dimension) and geometric dependencies (Relations), which are the basis of parametric modeling.
- Gaining the skills to create simple three-dimensional bodies based on 2D sketches using the Extrude operation.
- Using additional tools to optimize sketch construction, such as Circular Sketch Pattern (Circular sketch array), Mirror (Mirror image), Straight Slot, Polygon, and Trim Entities (Trimming), which allows you to quickly create repeating and symmetrical elements.

- Developing skills in creating fillets on the edges of a 3D model, which improves the functionality of the part and its aesthetic appearance.
- Developing skills in drawing up a simple drawing of a part, which is important for further production documentation.
- Preparation of a report in which the student will describe in detail all the stages of constructing a sketch, creating a 3D model and drawing, as well as analyze the results obtained and draw conclusions regarding the quality of the task.

2. Theoretical foundations of working in SolidWorks

SolidWorks is a parametric modeling system that allows you to create three-dimensional models by building 2D sketches with specified parameters. The main advantages of SolidWorks:

- **Parametric modeling.** All changes to the model are made by editing the sketch parameters. This allows you to easily modify the design by changing only the numerical values of dimensions or the parameters of dependencies.
- **Intuitive interface.** SolidWorks has a convenient command bar (CommandManager) and a construction tree (FeatureManager Design Tree), allowing you to quickly find the necessary tools.
- **Modular structure.** The program consists of separate modules for working with individual parts, assemblies, and drawings, which allows you to systematize the project and facilitate the development process.
- **Build automation.** Using tools like Circular Sketch Pattern, Mirror, Straight Slot, Polygon and Trim Entities, significantly speeds up the creation of complex sketches and ensures their accuracy.

3. Preparing the SolidWorks working environment

3.1 Creating a new document

1. Start SolidWorks and select the New → **Part command** (Figure 1).

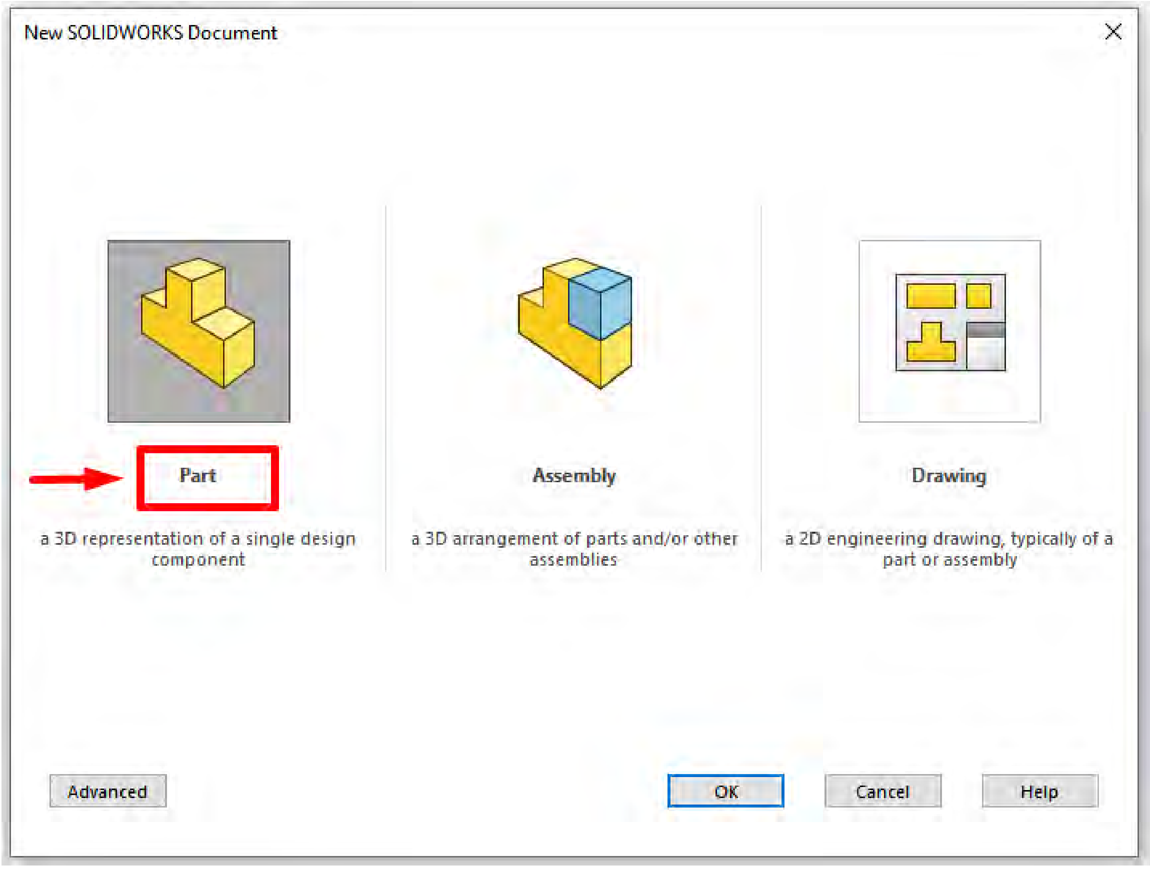
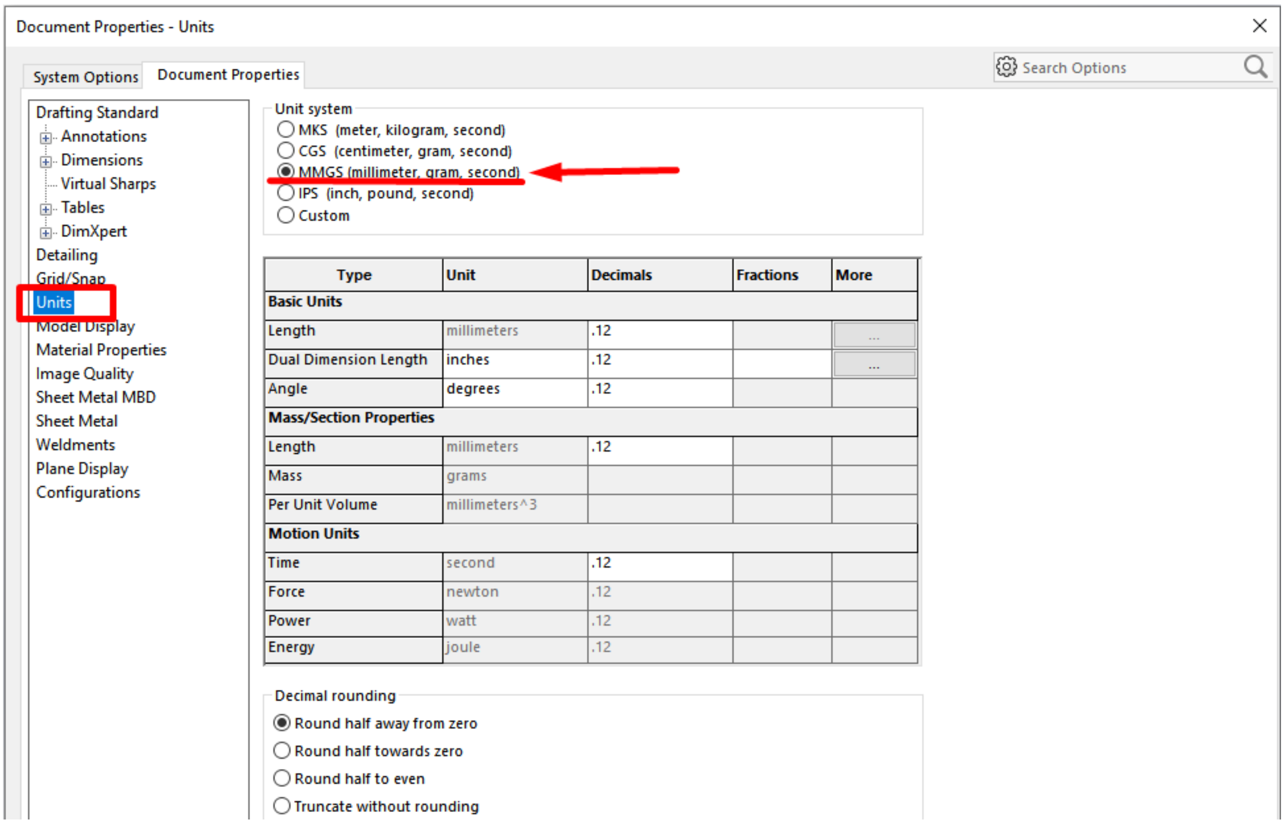


Fig. 1

2. Go to **the Options** → **Document menu. Properties** → **Units** (Options → Document Properties → Units) and set to millimeters (Fig. 2).
3. Select a plane to create a sketch on, for example, **Front Plane** (Front plane) (Fig. 3).

3.2 Setting up the workspace

- Make sure that the feature tree (FeatureManager) correctly displays the sequence of operations performed.
- Adjust the view orientation (View Orientation) for ease of working on sketches (isometric or front view).


Document Properties - Units
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Document Properties
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Annotations
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Tables
DimXpert
Detailing
Grid/Snap
Units
Model Display
Material Properties
Image Quality
Sheet Metal MBD
Sheet Metal
Weldments
Plane Display
Configurations
Unit system
MKS (meter, kilogram, second)
CGS (centimeter, gram, second)
MMGS (millimeter, gram, second)
IPS (inch, pound, second)
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Decimals
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Basic Units
Length
millimeters
.12
Dual Dimension Length
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Angle
degrees
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Mass/Section Properties
Length
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.12
Mass
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Force
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.12
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Energy
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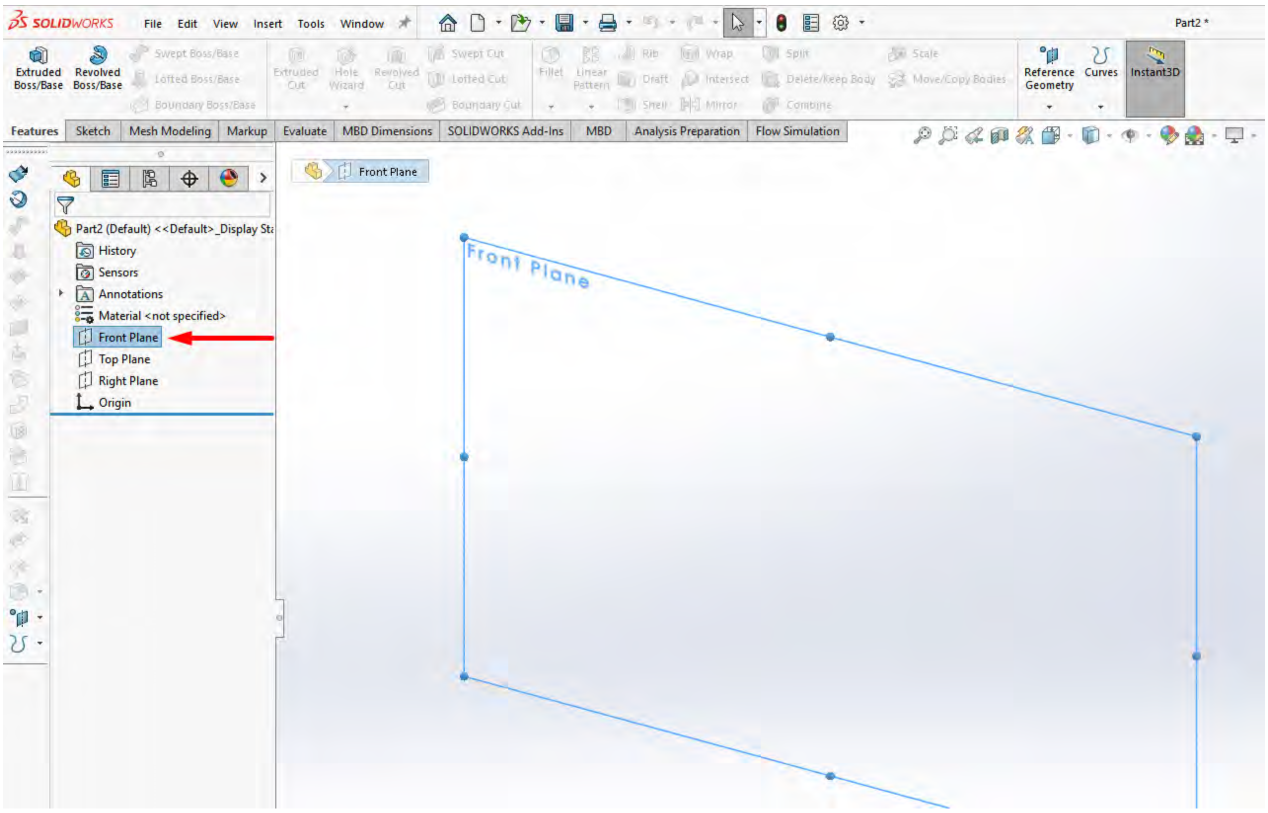
Decimal rounding
Round half away from zero
Round half towards zero
Round half to even
Truncate without rounding

Fig. 2


SOLIDWORKS
File
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Part2 *
Extruded Boss/Base
Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base
Boundary Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Swept Cut
Lofted Cut
Boundary Cut
Fillet
Linear Pattern
Rib
Draft
Shell
Wrap
Intersect
Mirror
Split
Delete/Keep Body
Combine
Scale
Move/Copy Bodies
Reference Geometry
Curves
Instant3D
Features
Sketch
Mesh Modeling
Markup
Evaluate
MBD Dimensions
SOLIDWORKS Add-Ins
MBD
Analysis Preparation
Flow Simulation
Front Plane
Part2 (Default) <<Default>_Display St
History
Sensors
Annotations
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Front Plane

Fig. 3

4. Creating an orthogonal projection of a flat membrane (2D sketch)

4.1 Developing a basic sketch

1. **Starting a sketch:**

On the selected plane (for example, Front Plane) click **Sketch**. The origin is used as the base point for constructing symmetrical elements (Fig. 4).

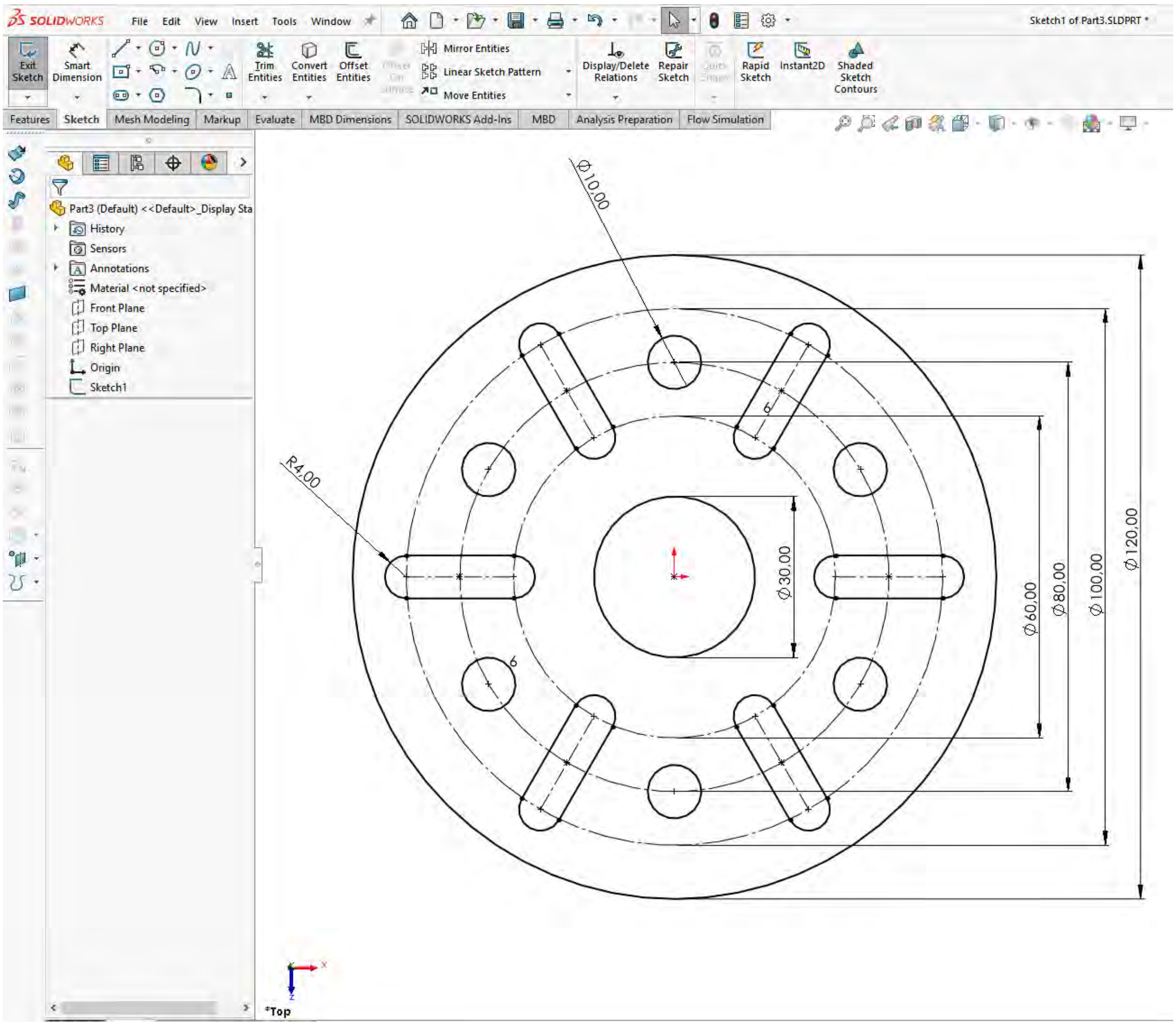


Fig. 4

2. **Construction of concentric circles:**
   - Use the **Circle tool** to create concentric circles.
   - Through **Smart Dimension** (Dimensions) set the radii of the circles. The centers of all circles must coincide with the Origin (using the Coincident property).

3. **Creating a cutting profile:**
   - Using tools such as Line, Arc**,** Circle**,** Polygon, Slot**,** etc., construct the contour of the membrane cut (Fig. 4)**.**
   - Set dimensions via **Smart Dimension** and set geometric dependencies (parallelism, perpendicularity, symmetry, coincidence, etc.) so that the sketch is fully defined (Fig. 4).
   - Use **Trim Entities** (Trimming) to remove unnecessary lines and curves that are not part of the main contour (Fig. 5).

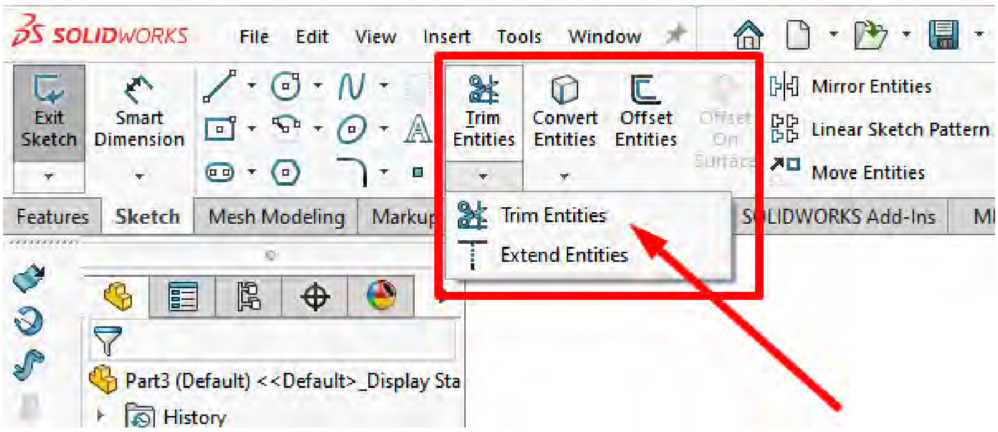


Fig. 5

4. **Using tools for repeating elements:**
   - **Circular Sketch Pattern (Circular Sketch Array):** Use to reproduce elements that repeat around a center. Specify the number of instances (Figure 6).

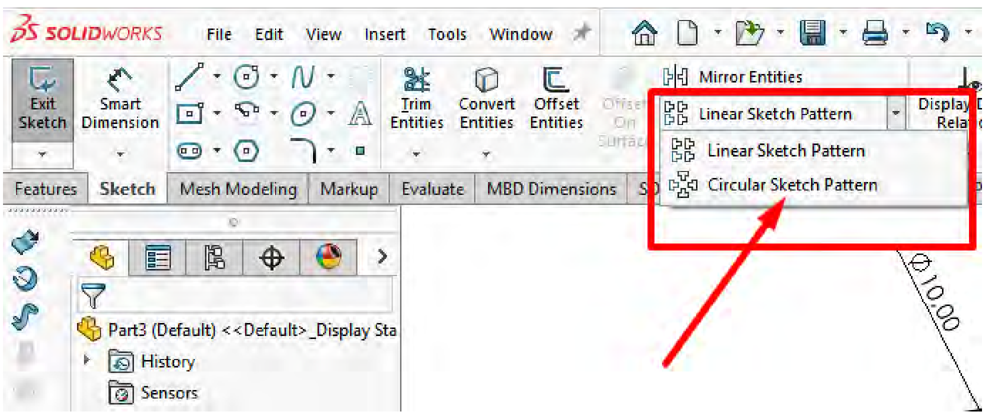


Fig. 6

- **Mirror:** Use to create symmetrical elements by specifying the centerline through Origin.
- **Straight Slot:** Use to create straight slots with precise positioning and dimensions (Fig. 7).
- **Polygon:** Use to construct elements with many sides if the task requires it (Fig. 8).

5. **Finishing the sketch:** Make sure all sketch elements have their dimensions and dependencies set, and click Exit Sketch.

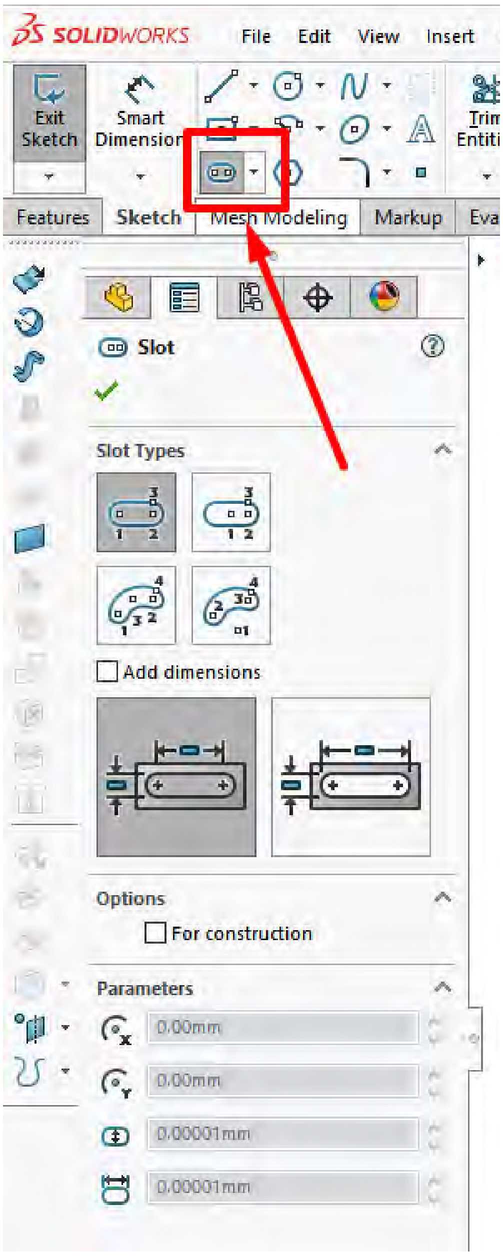


Fig. 7

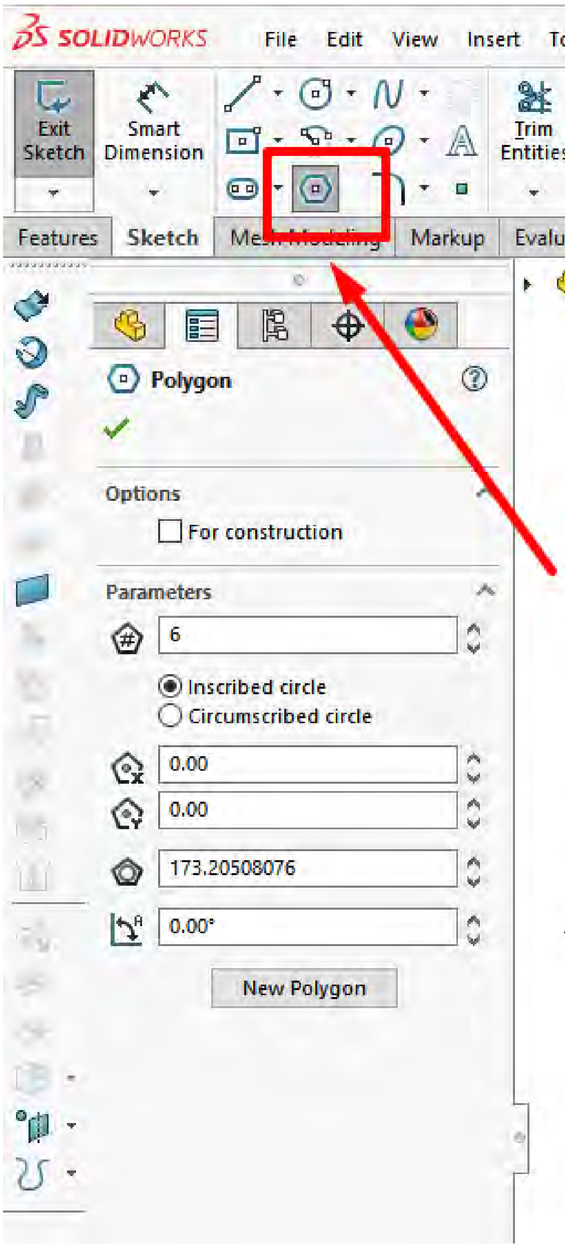


Fig. 8

5. Creating a 3D model of a flat membrane

5.1 Extrude Boss / Base (Extrusion)

1. **Starting the extrusion operation:**
   - Go to the **Features tab** and select **Extruded Boss / Base**.
   - In Property Manager (Properties Panel) in **the Depth field**, enter a value of **10 mm** and select the **Blind option** (Fig. 9).

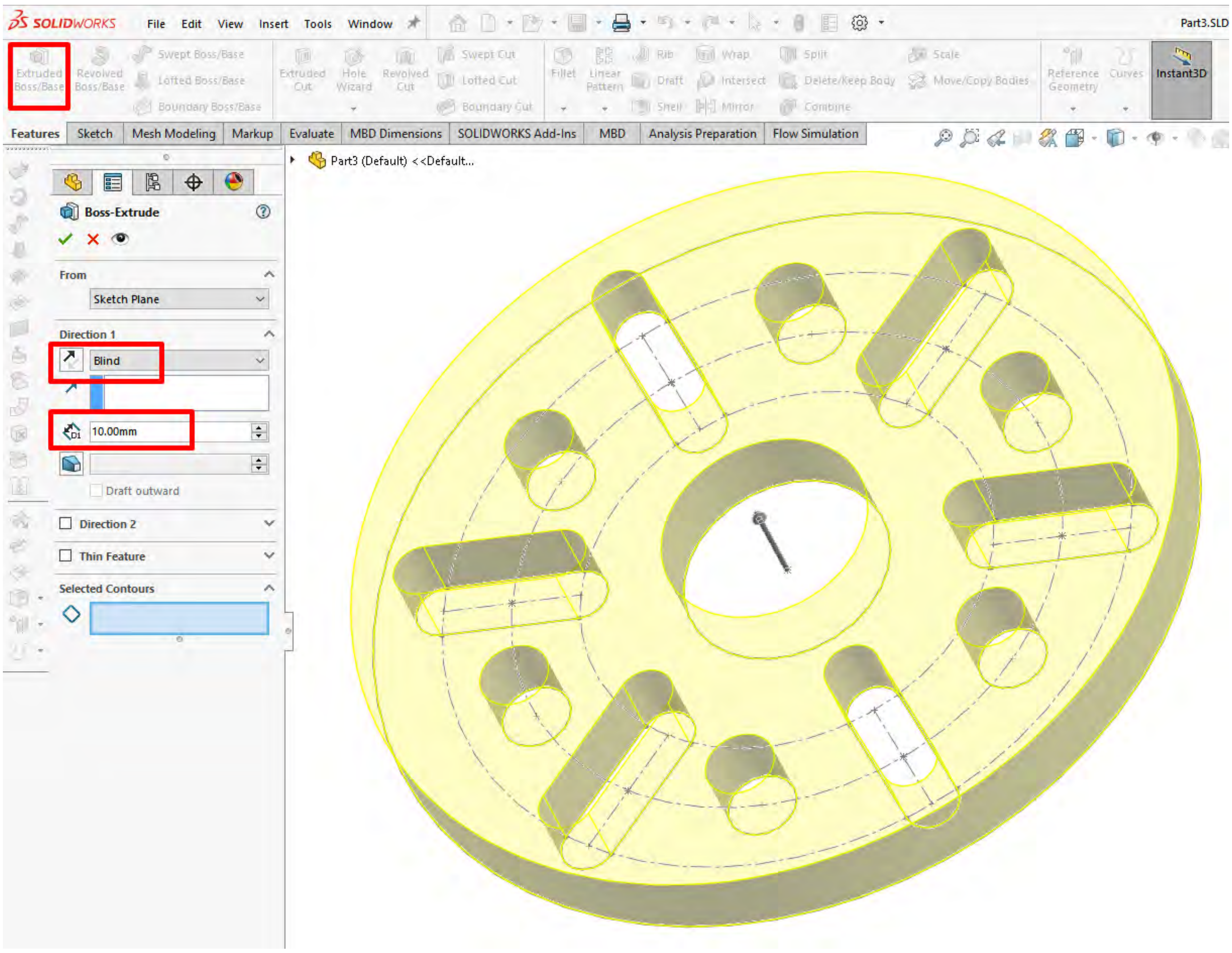


Fig. 9

2. **Converting a sketch into a 3D model:**
   - Click **OK** to complete the operation. A flat membrane model will be created with a thickness of 10 mm (Fig. 10).

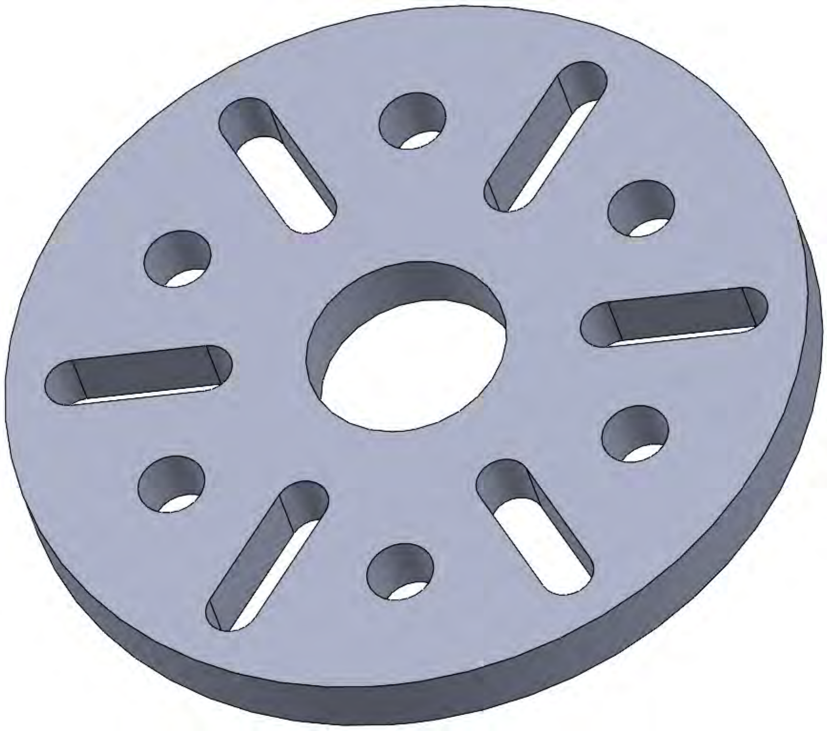


Fig. 10

3. **Model control:**

View the model in different views (Front, Top, Isometric) to verify the extrusion is correct. Make adjustments, if necessary, by returning to the sketch.

5.2 Adding Fillets

1. **Starting the rounding operation:**
   - In the **Features tab**, select **Fillet** (Figure 11).
2. **Setting rounding options:**
   - In the properties panel, enter a value of **2 mm** for **the Radius parameter** (Figure 11).
3. **Applying rounding:**
   - Select the top and bottom edges of the 3D model where you want to create a rounding (Fig. 11).
   - Click **OK** to complete the operation, making sure that the rounding is applied correctly.

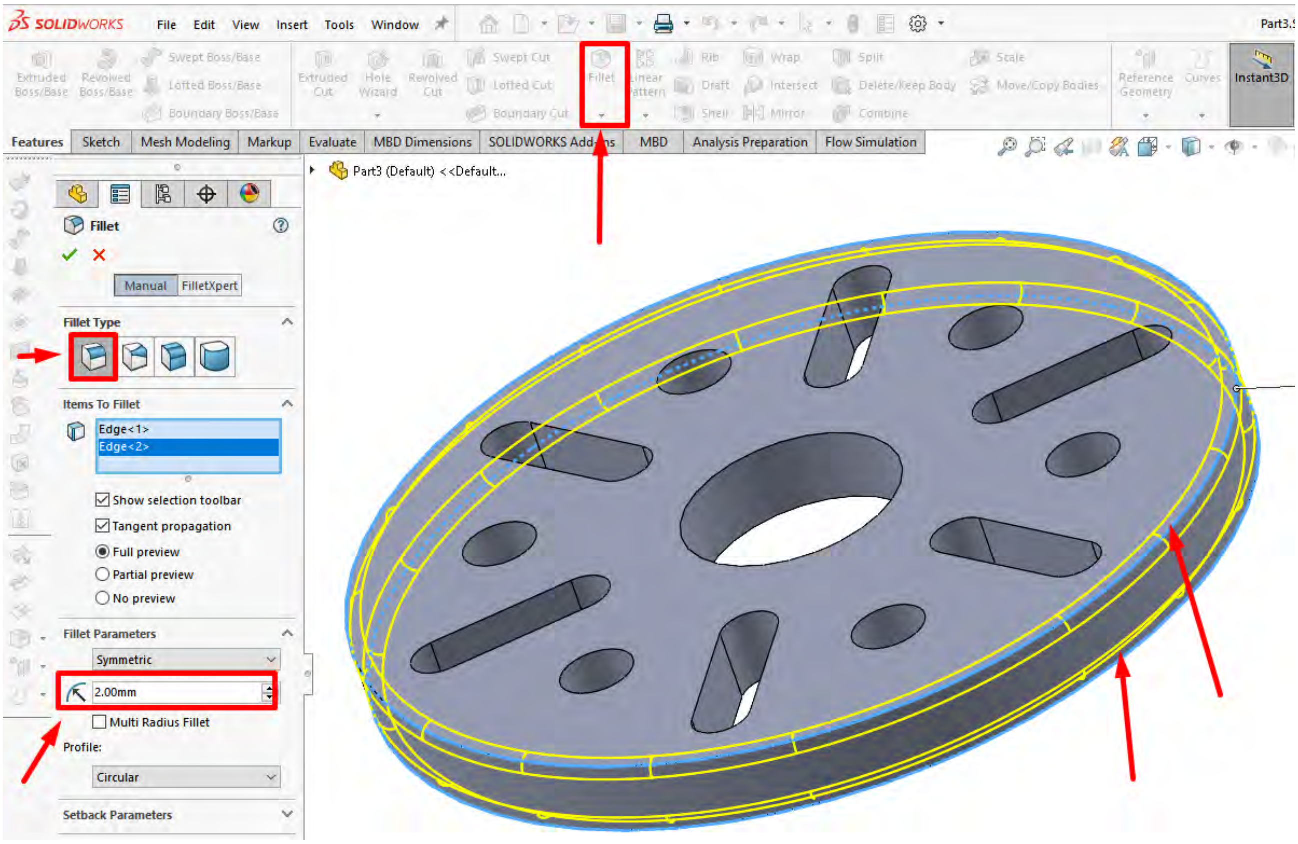


Fig. 11

6. Creating a drawing of a membrane part

6.1 Drawing formation

1. **Starting the drawing creation procedure:**
   - Use **the File → Make command Drawing from Part** (File → Create drawing from part).
   - Select a standard sheet (for example, A4_ISO format) and set the drawing scale (1:1 or 1:2 - depending on requirements).

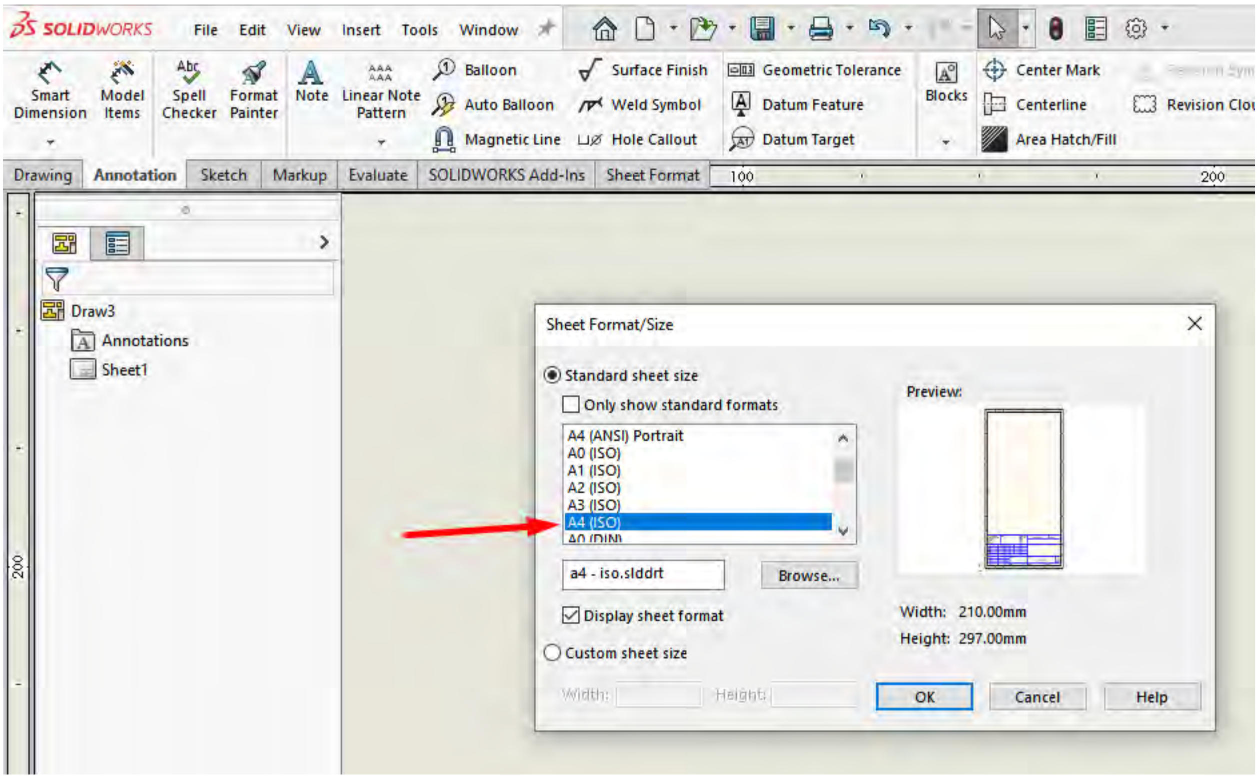
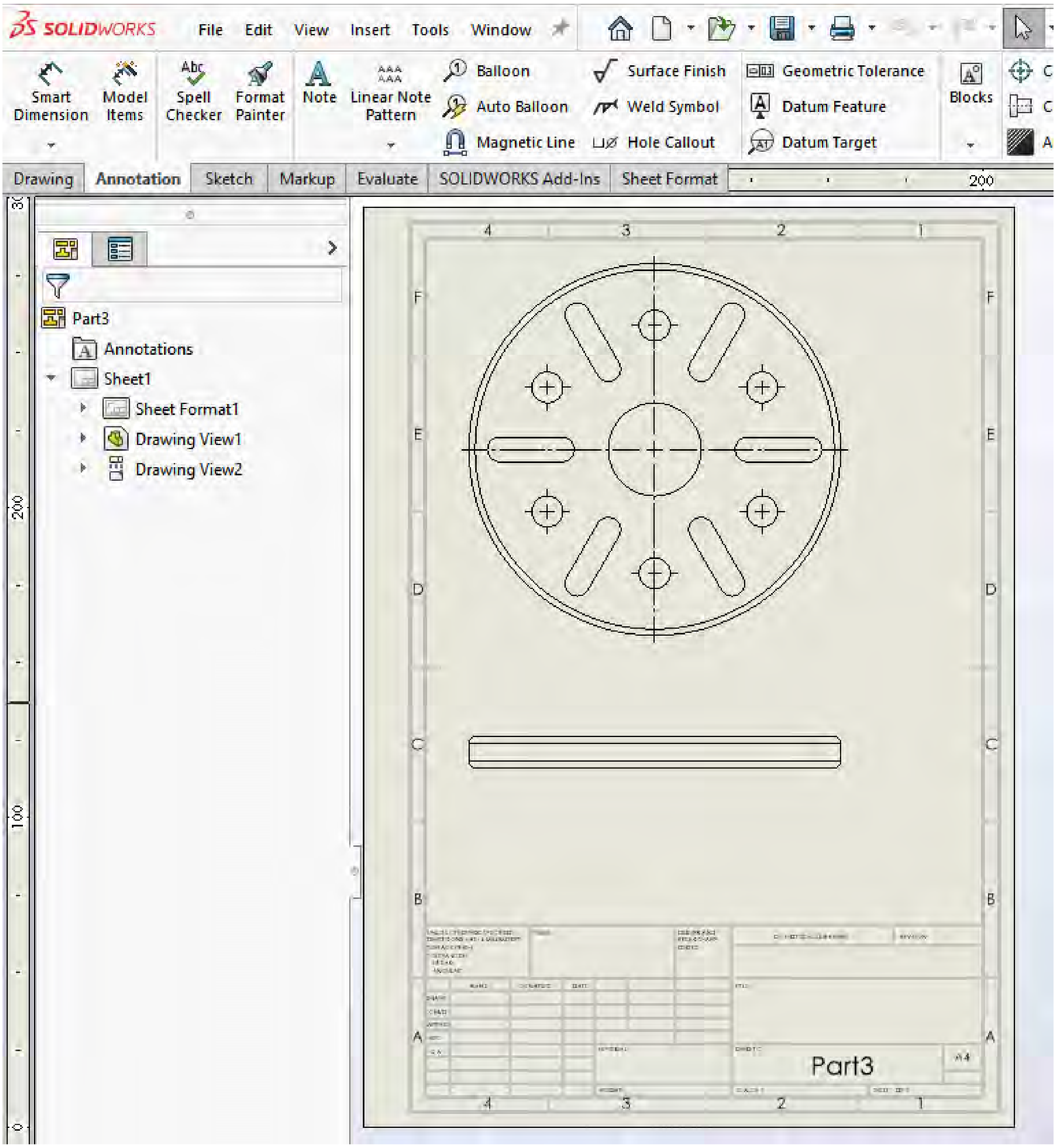


Fig. 12

2. **Model view placement:**
   - Using **Model View** (Model View) insert the main views of the flat membrane.
   - Arrange the main views so that all structural elements are clearly visible.
   - For example, you can create two views – Top View and Front View (Fig. 13).

6.2 Dimensioning

1. **Dimensioning:**
   - Using **Smart Dimension** manually applies dimensions corresponding to the main characteristics of the membrane (circle diameters, recess sizes, slot parameters, polygons).
2. **Alternative option:**
   - Use the **Import Annotation – Design Annotation function** to automatically transfer dimensions from the model to the drawing (Figure 14).


SOLIDWORKS
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Spell Checker
Format Painter
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Balloon
Auto Balloon
Magnetic Line
Surface Finish
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Drawing View2
Part3

Fig. 13

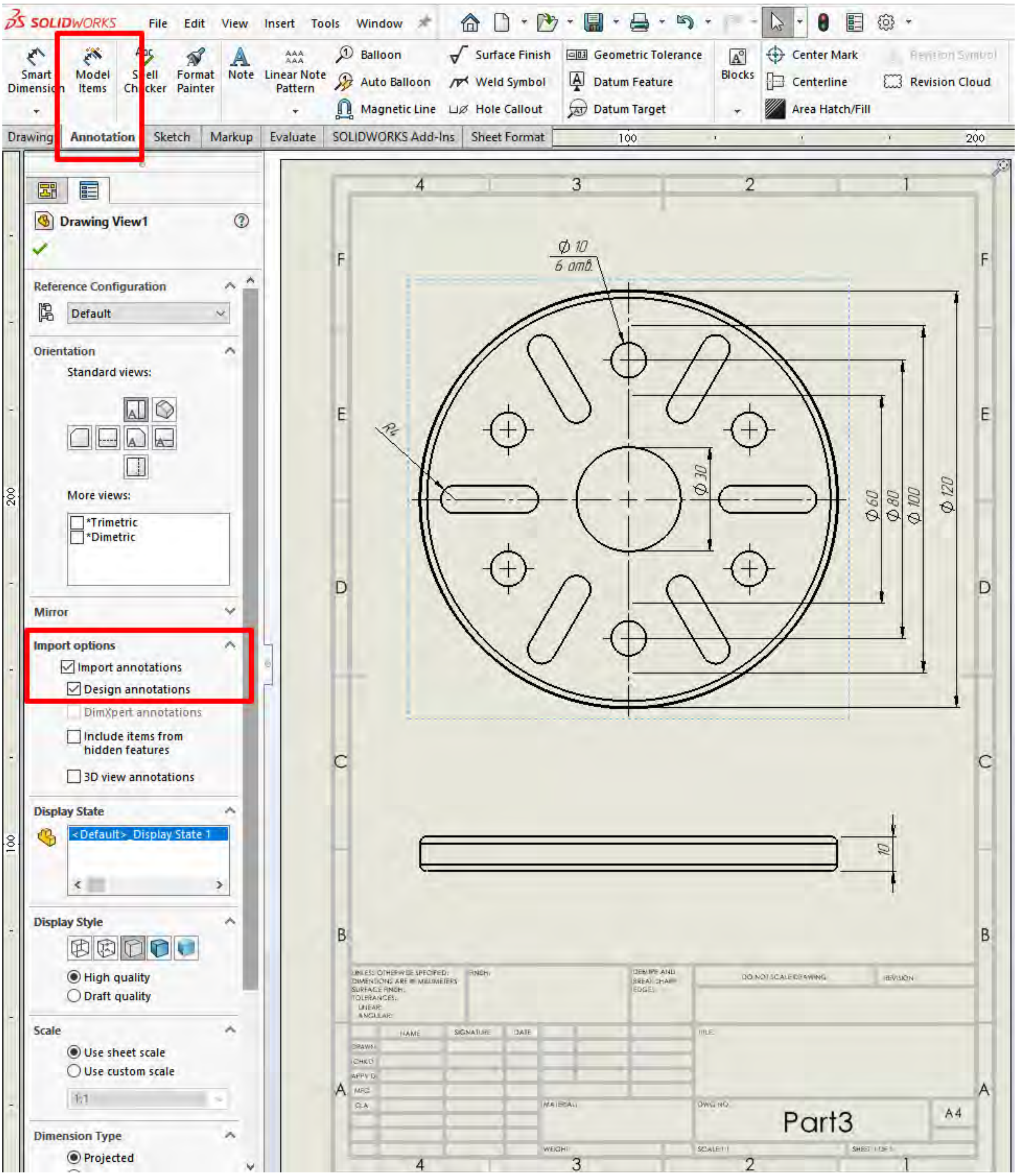
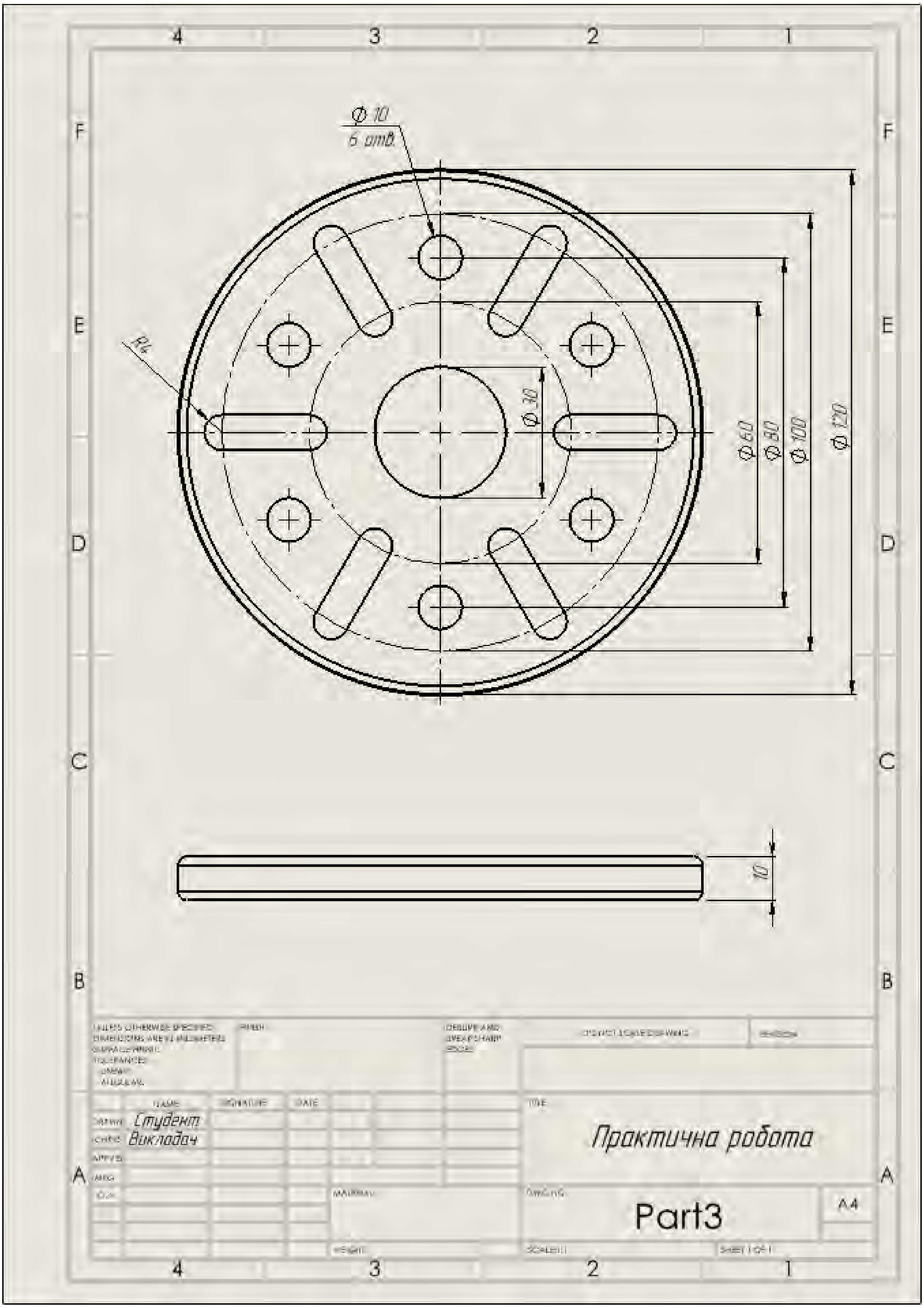


Fig. 14

3. **Drawing check:**

Make sure that the drawing contains all the necessary dimensions, text symbols, center lines and save it in SolidWorks Drawing (SLDDRW) format (Fig. 15).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Студент
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Fig. 15

7. Report preparation

7.1 Report structure

The report should contain the following sections:

- **Introduction:**

A brief description of the purpose of the work, the tasks and the importance of correctly constructing sketches as the basis of parametric modeling.

- **Description of the construction stages:**
  - Detailed description of creating a sketch of a flat membrane with an indication of the tools used (Circle, Line, Arc, Polygon, Circular Sketch Pattern, Mirror, Straight Slot, Trim Entities).
  - Description of converting a sketch into a 3D model using Extrude with parameters: Depth = 10 mm, Blind.
  - Description of the use of 2 mm radius fillets on the top and bottom edges.
- **Drawing design:**
  - Procedure for creating a drawing using Make Drawing from Part and dimensioning (via Smart Dimension or Import Annotation – Design Annotation).

9. Recommendations and tips

1. **Full certainty of the sketch:**

Make sure that each sketch element has set dimensions and geometric dependencies - this will avoid errors when applying the Extrude command.

2. **Optimization of the construction of repeating elements:**

Use Circular Sketch Pattern and Mirror to create symmetrical parts, which will reduce modeling time.

3. **Correct use of Extrude:**

When setting the extrusion parameters, enter a value of 10 mm in the Depth field in the Property Manager and select the Blind option.

4. **Drawing design:**

To create a drawing, use the Make command. Drawing from Part and apply dimensions using Smart Dimension or automatic annotation transfer via Import Annotation.

5. **Regular check:**

After each stage, check the model in different view modes (Front, Top, Isometric) to detect possible errors in a timely manner.

10. Conclusion

The implementation of this laboratory work contributes to the acquisition of practical skills in SolidWorks, in particular:

- Developing skills in constructing fully defined sketches using dimensions and geometric relationships.
- Gaining the skills to create simple three-dimensional bodies through the Extrude operation.
- Mastering methods for applying fillets.
- Creating a simple drawing of a part with accurate dimensions.
- Developing a report with a detailed description of the stages of work, analysis of the results obtained, and recommendations for further improvement of modeling skills.

Following these methodological guidelines will help students consolidate basic practical skills in working with SolidWorks, which is the basis for successfully completing more complex tasks in the field of computer modeling.

Lab Work №3: Construction of a three-dimensional model and shaft drawing in SolidWorks

Introduction

The purpose of the laboratory work is to master the process of modeling a three-dimensional model of a shaft and creating its drawing in SolidWorks. A shaft is a key part in mechanical engineering that transmits torque and supports rotating elements.

During the work, students will learn:

- Create three-dimensional models of shafts with various structural elements.
- Use SolidWorks tools to build sketches and solid models.
- Generate drawings with views, sections, and annotations.

To complete the task, basic knowledge of the SolidWorks interface and the basics of engineering graphics is required. Each student works with his own version of the shaft, the parameters of which are specified in the individual task.

1. General requirements and preparation of the working environment

1.1 Starting the program and creating a part

– Start SolidWorks and create a new file in **Part mode** via **the File > New > Part menu**.

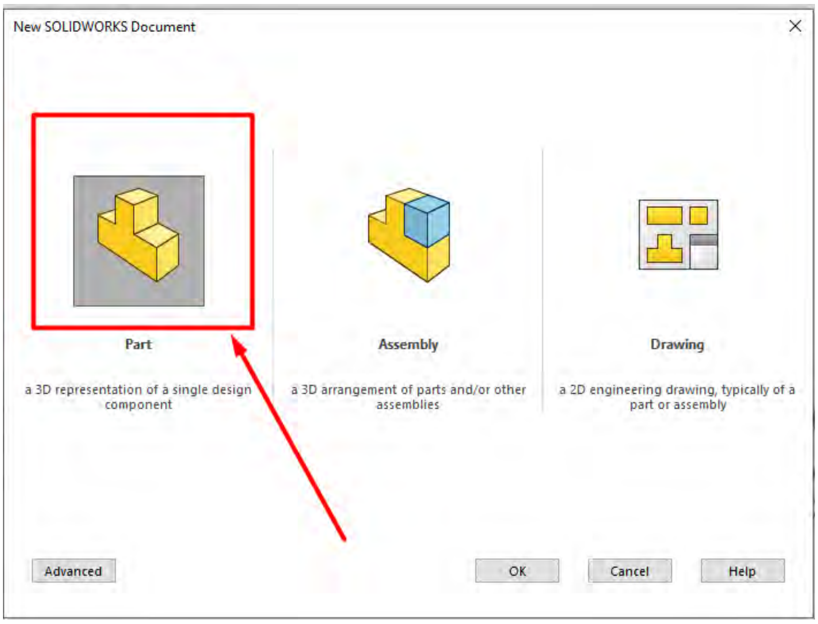


Figure 1 – Creating a new file in Part mode

– Check the units' settings via **Options > Document Properties > Units** and make sure the correct units are used for your task (usually millimeters).

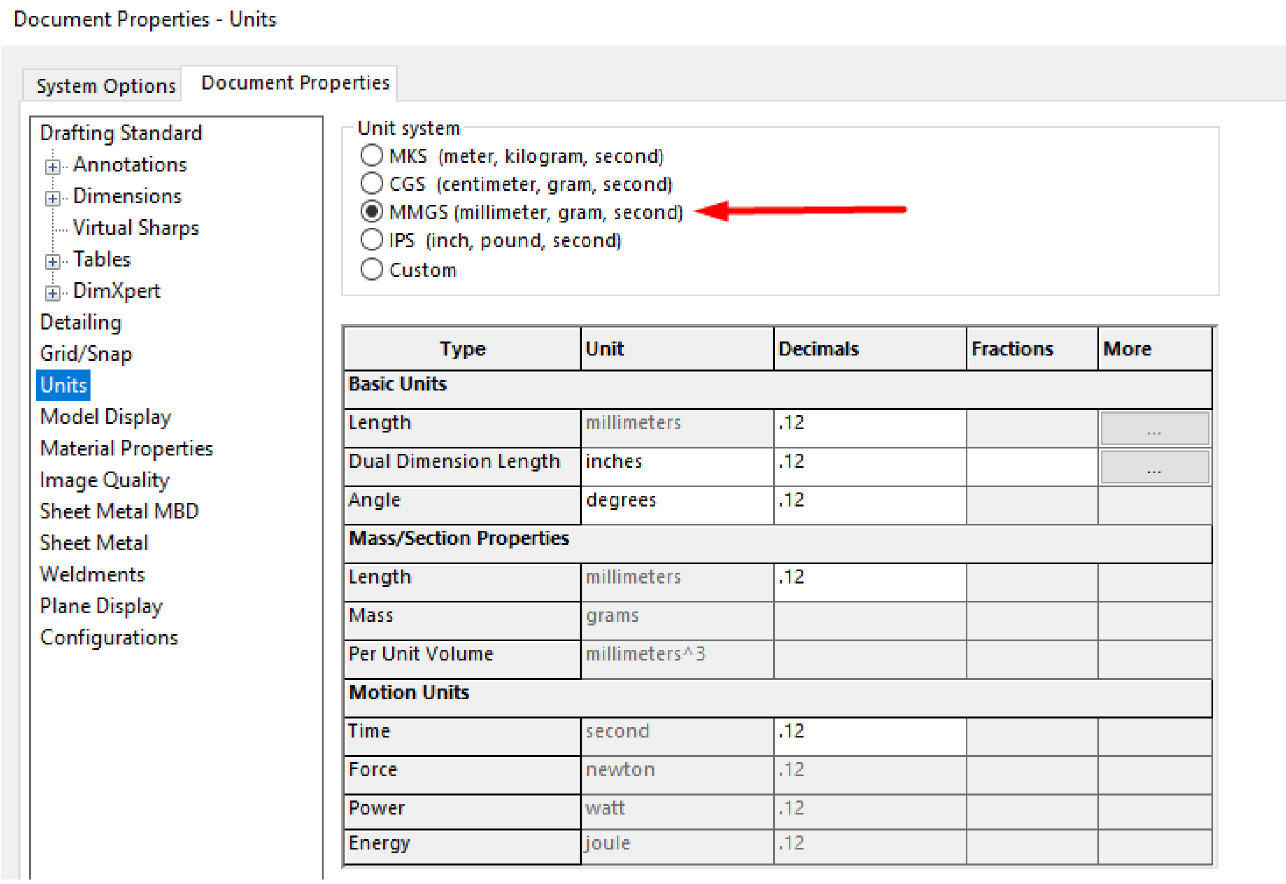


Figure 2 – Setting units of measurement

– Organize your workspace, create a separate folder to store project files, and regularly save your work using the save function (Ctrl+S).

1.2 Preparing for modeling

Before starting the construction, pay attention to the geometry of the shaft, the shapes of the segments, grooves, holes, the location of the keyway, as well as the requirements for chamfers and fillets (an example of shaft geometry is shown in Figure 3). This will allow you to correctly plan the construction sequence and choose the appropriate modeling method.

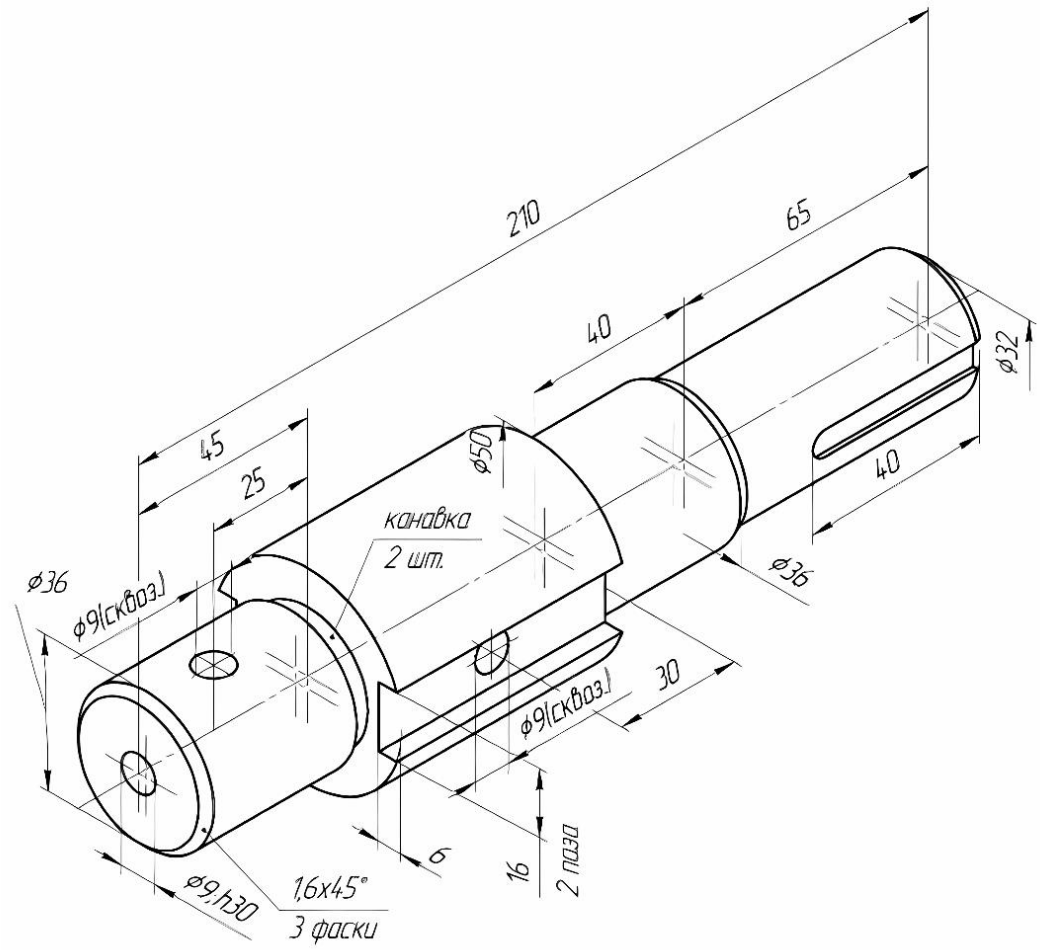


Figure 3 – Shaft geometry with required dimensions for modeling

2. Building a three-dimensional model of the shaft

There are two main methods for constructing a shaft model that a student can use depending on the design features of the part and their own preferences. A detailed description of each method is provided below.

2.1. Method 1: Constructing individual shaft segments using the Extruded Boss/Base command

This approach is based on creating a separate sketch for each shaft segment. The sketch directly reflects the desired shape – it can be not only a circle, but any closed contour (rectangle, truncated circle, other geometric shape) that corresponds to the drawing of the corresponding task variant. Thanks to this method, there is no need to perform additional trimming operations if the sketch already forms a segment with the desired geometry.

2.1.1. Creating individual segments

Selecting a plane for a sketch

Select one of the standard planes (for example, Front Plane) and create a new sketch using the **Sketch command**.

Building a segment sketch

Using construction tools (e.g., **Line, Circle**, **Rectangle**, **Arc**, etc.), create a closed path that matches the shape of the shaft segment according to the drawing (Figure 4).

Set the necessary dimensions and relationships to make the sketch fully defined. Make the sketch fully defined by adding the necessary dimensions and relationships (Figure 4).

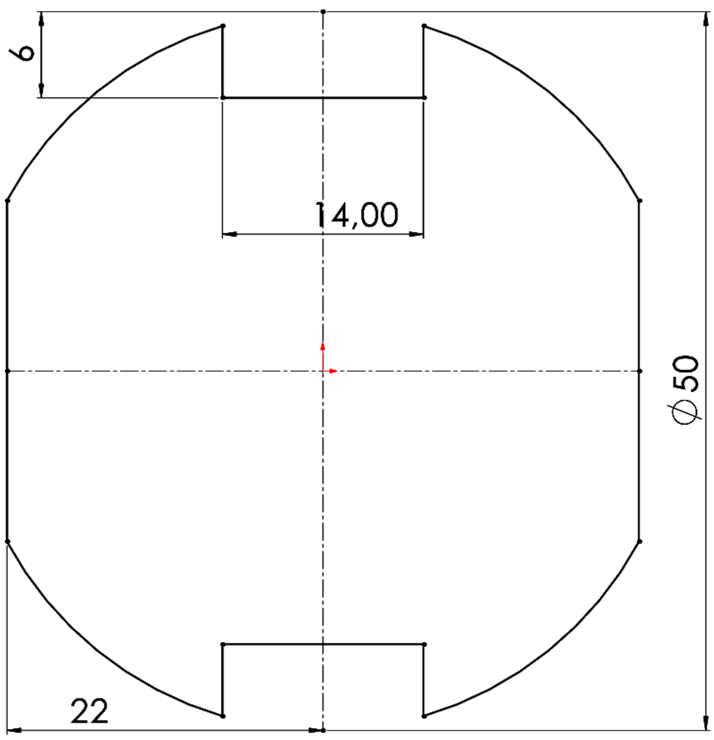


Figure 4 – Sketch of a shaft segment

Creating a 3D model of a shaft segment

After completing the sketch, click **Exit Sketch** and use the **Extruded Boss/Base command** to extrude the sketch to a specified length (Figure 5).

Be sure to check the " **Merge results** " box, which will allow you to merge the created volumes into one solid body (Fig. 5).

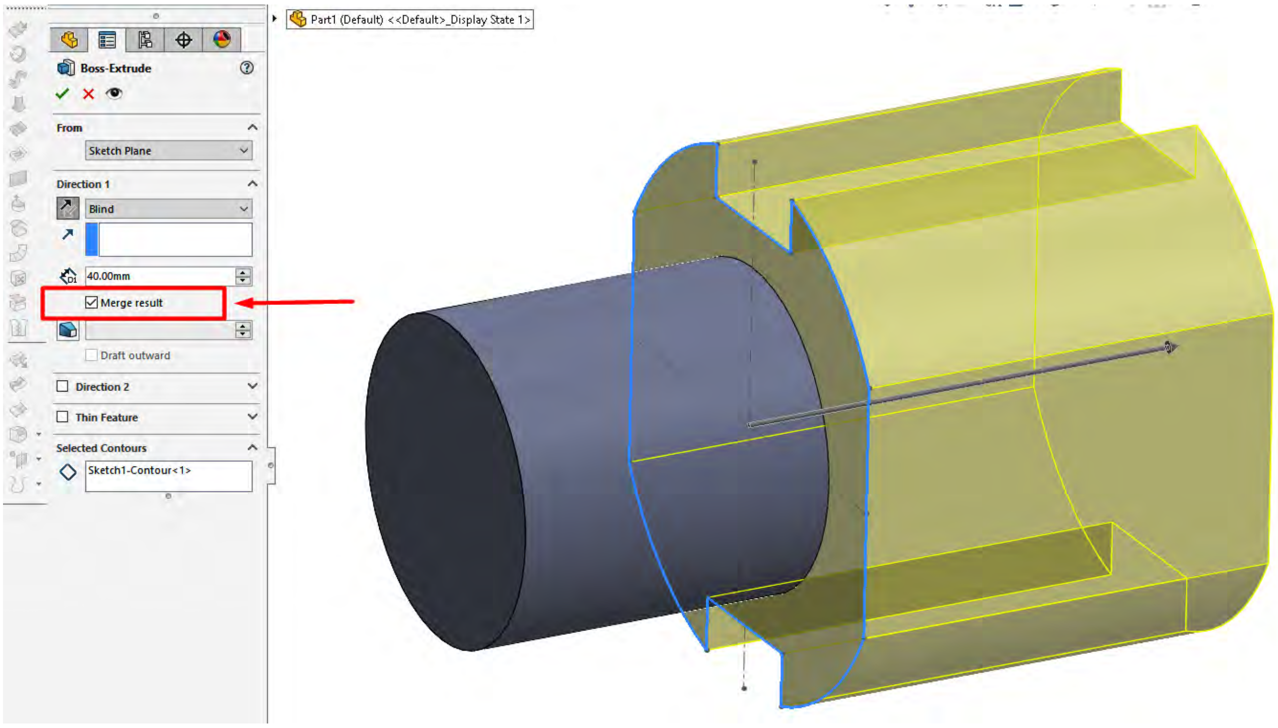


Figure 5 – Creating a 3D model of a shaft segment

2.1.2. Operations for forming structural elements

The main advantage of this method is that the sketch of each individual segment defines the required geometry without the need for additional trimming operations to achieve the desired shape. However, if the drawing requires a keyway, grooves, or other features, these must be created separately (see Chapter 3).

2.2. Method 2: Constructing a shaft by rotating a sketch using the Revolved Boss/Base command

This approach is more appropriate and rational for building shaft-type parts. It consists in creating a single shaft profile consisting mainly of straight lines, as this results in cylindrical segments. This profile is then rotated around a given axis using the **Revolved Boss/Base command**.

The resulting body contains all the shaft segments, however, in cases where the segment is not completely cylindrical, additional trimming operations must be applied.

2 .2.1. Creating a shaft sketch

Building an axis and a sketch

On the selected plane (for example, Front Plane), create a new sketch using the **Line tool** to draw a centerline. Use dimensions and relationships to precisely position this line.
Then, draw a closed path that describes the outer contour of the shaft, which consists mainly of straight lines (except for the grooves, where fillets must be applied). This will allow you to obtain cylindrical segments when rotating the profile (Figure 6).

Full definition of a sketch

Make the sketch fully defined by adding the necessary dimensions and relationships (Figure 6). This will ensure the accuracy of the resulting geometry during subsequent rotation.

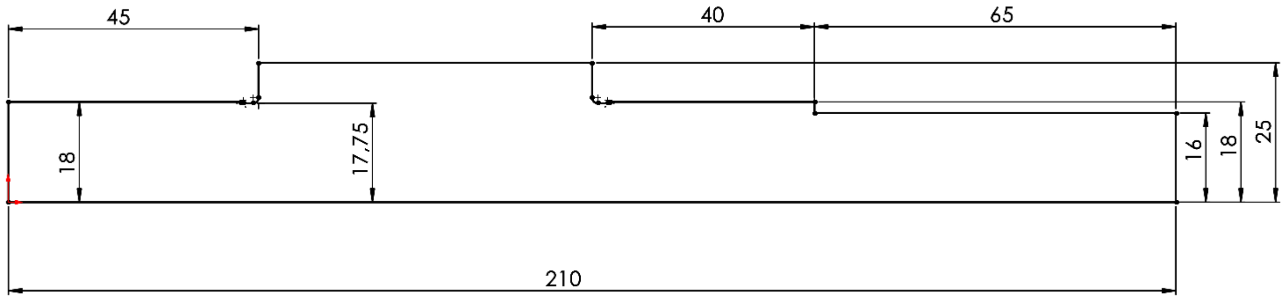


Figure 6 – Shaft sketch

2.2.2. Rotation operation and shaft shape correction

Revolved Boss/Base Application

Once the sketch is complete, use the **Revolved Boss/Base command** to rotate the profile around the axis. Set the rotation to 360° to obtain a full shaft body (Figure 7).

Shaft shape correction

In cases where the shaft segment, according to the drawing, is not cylindrical in shape, it is necessary to apply additional cutting operations using the **Extruded Cut command** to remove excess material (Fig. 8, 9).

This operation allows you to precisely adjust the shape of each segment according to the requirements of the drawing.

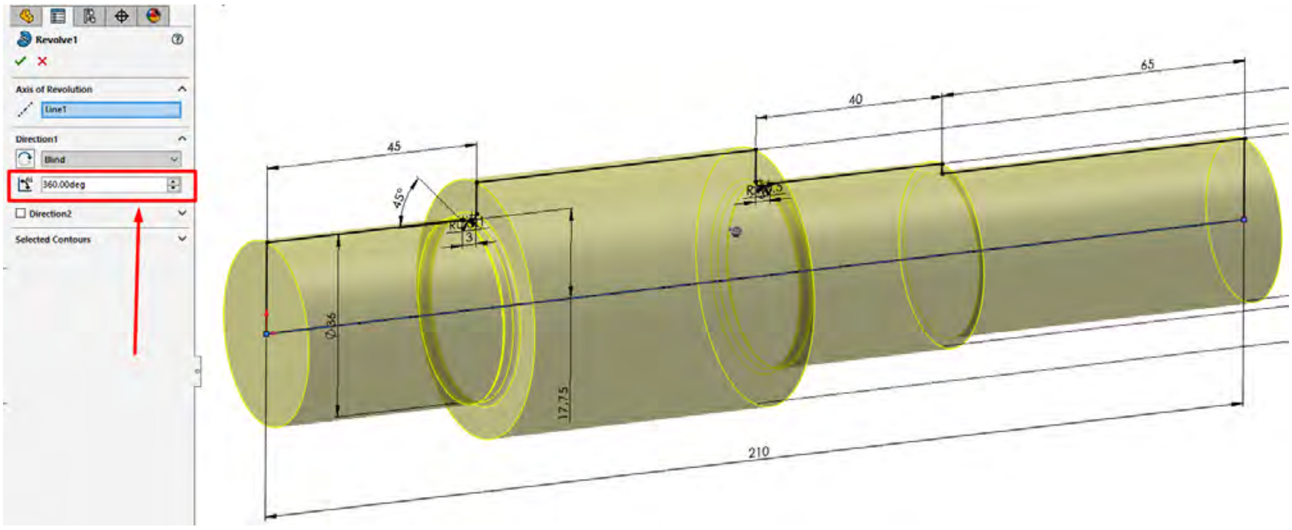


Figure 7 – Using Revolved Boss/Base to obtain a shaft model

For example, if the drawing involves a cut contour on a certain segment, after rotating the profile, it is necessary to apply the **Extruded Cut command** to remove part of the shaft body and obtain the specified geometry (Fig. 8, 9).

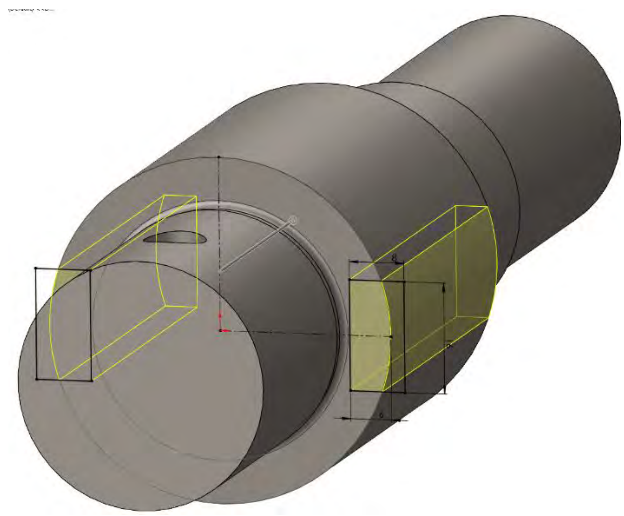


Figure 8 – Creating a sketch for cutting

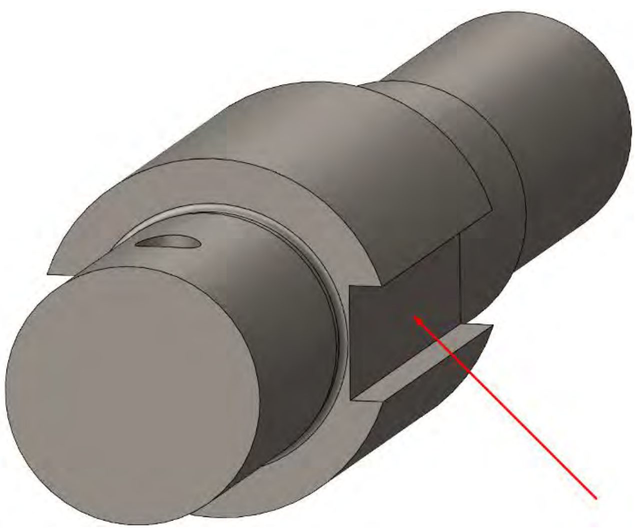


Figure 9 – Final view of the shaft after cutting

3. Construction of keyways, grooves, holes, chamfers and roundings

This section contains general recommendations for forming structural elements that apply regardless of the chosen method of constructing the shaft model.

3.1. Construction of a keyway groove

There are two methods you can use to create a keyway in SolidWorks:

1. Using an auxiliary plane tangent to the cylindrical surface of the shaft segment.
2. Create a sketch in the Front Plane using the **Offset command** by a distance equal to the radius of the corresponding segment.

3.1. 1 Method 1: Creating an auxiliary plane and building a sketch on it

Creating an auxiliary plane

– Use the **Reference Geometry > Plane command** to create a new plane.

– Position the plane so that it is tangent to the outer cylindrical surface of the shaft segment where the keyway is to be located.

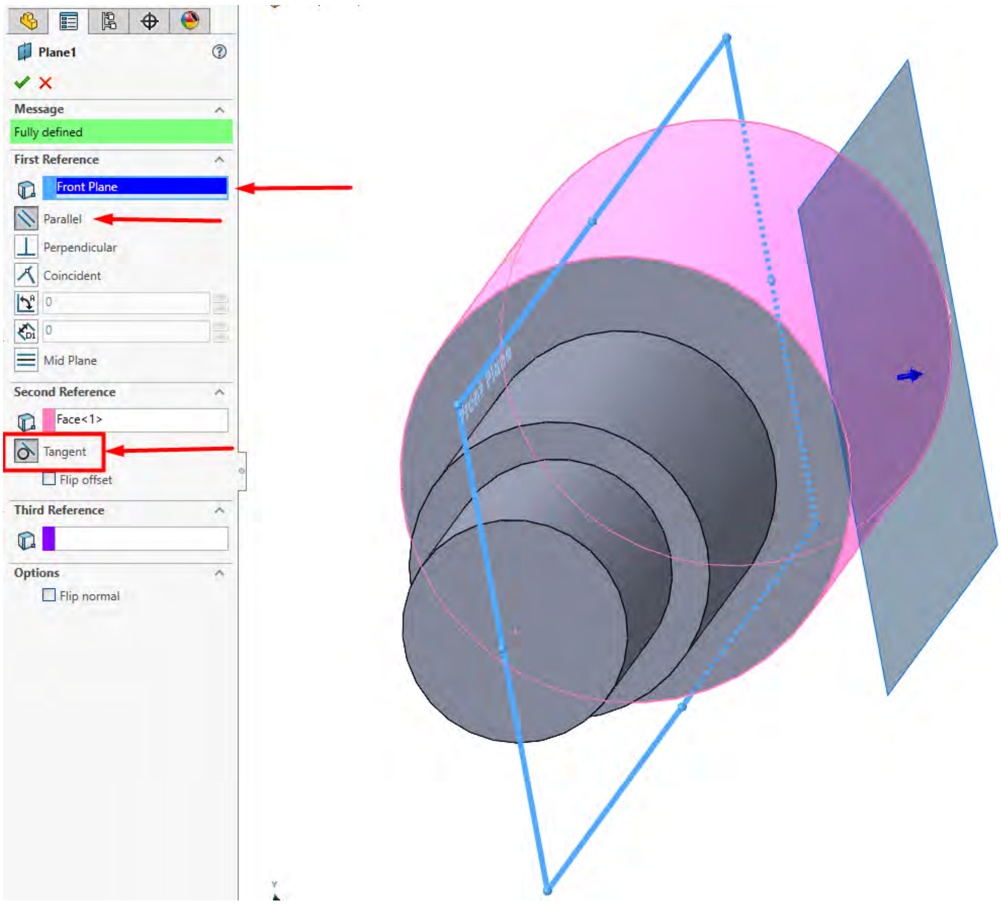


Figure 10 – Creating an auxiliary plane

Building a groove sketch

– On the created reference plane, sketch the keyway groove (Fig. 11).

– Use the **Straight Slot command**, which allows you to directly create the desired slot shape.

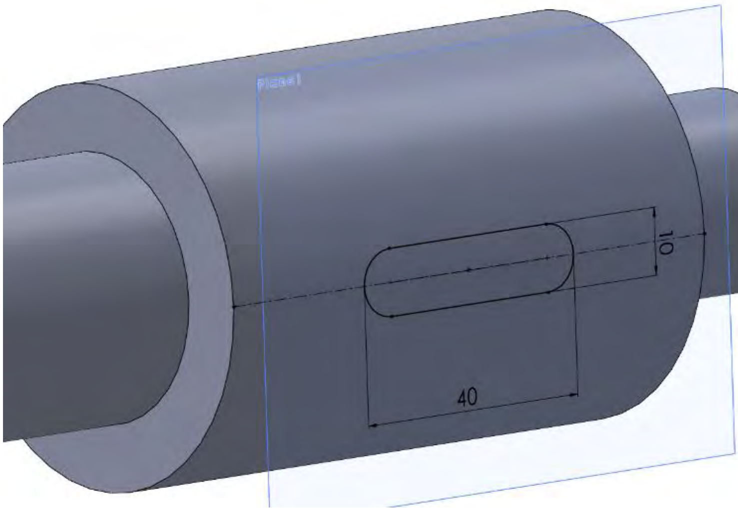


Figure 11 – Construction of a sketch of a keyway

– Alternatively, you can create a rectangular contour and then use the **Fillet command** to round off the required areas. However, it is most convenient to use **Straight Slot**, as it immediately forms the required profile.

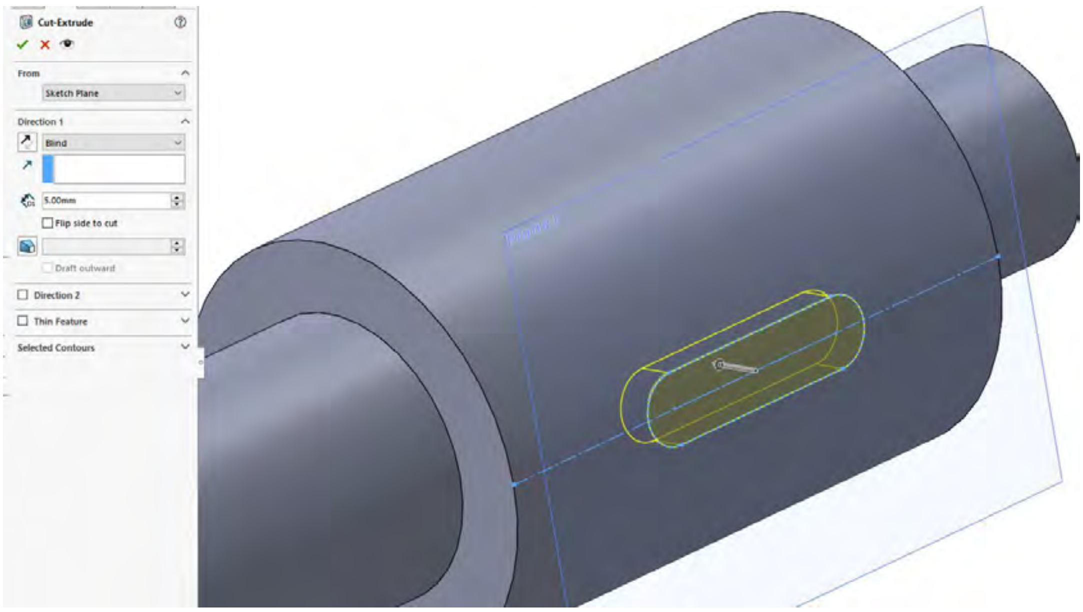


Figure 12 – Cutting a groove using Extruded Cut

Forming a groove using Extruded Cut

– After completing the sketch, use the **Extruded Cut command** to cut the shaft body to the specified keyway depth (Figure 12).

– Make sure that the direction of the cut coincides with the direction of the shaft axis to ensure correct geometry.

3.1. 2 Method 2: Using the Offset command to position the sketch

Creating a sketch in the front plane

– Select **the Front Plane** and create a new sketch on it.

– Construct the contour of the keyway using the **Straight Slot command** or a rectangle with subsequent rounding using **Fillet** (Fig. 13).

Sketch offset relative to shaft axis

– Use the **Offset command** to shift the sketch by a distance equal to the radius of the corresponding shaft segment (Figure 13).

This will allow you to correctly position the groove without the need to create a reference plane.

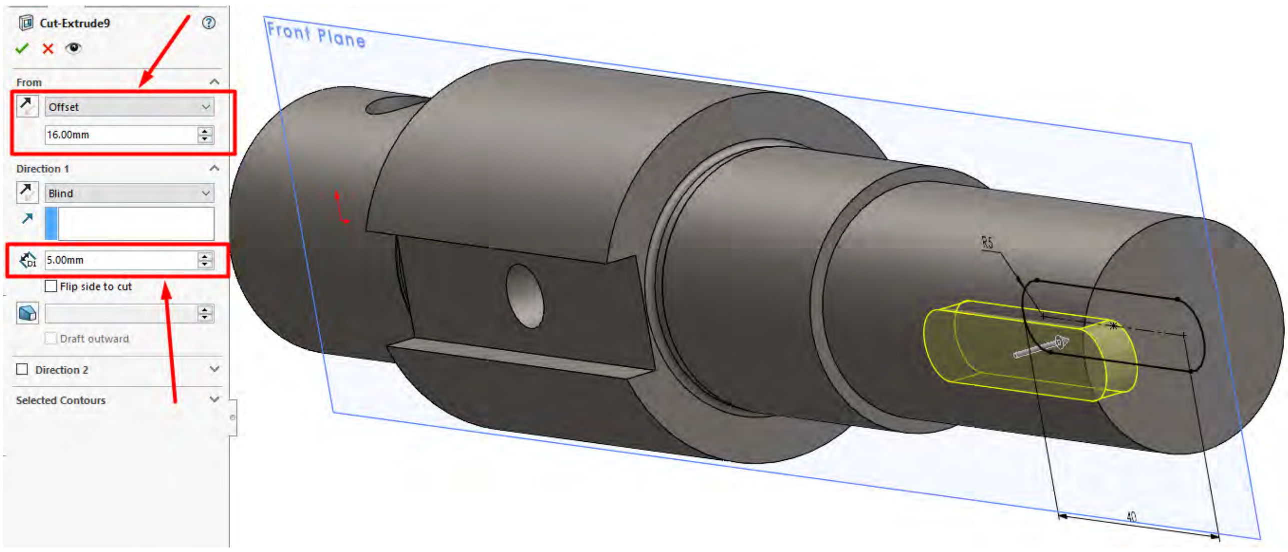


Figure 13 – Using the Offset command to create a groove

Forming a groove using Extruded Cut

– Apply the **Extruded Cut command** and set the depth of the cut according to the drawing (Fig. 13).

– Make sure that the operation is performed to the required depth according to the technical requirements.

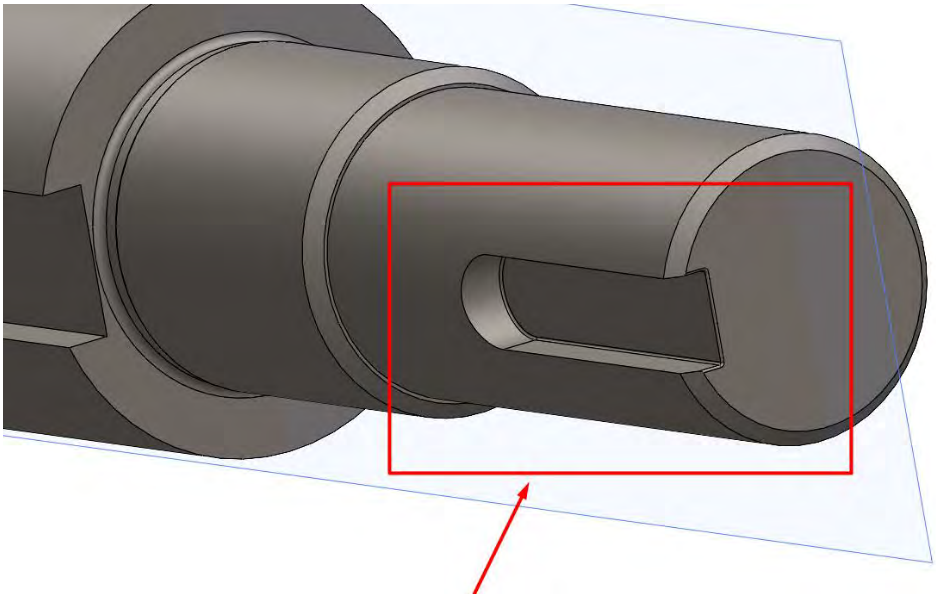


Figure 14 – Final view of the keyway

Both methods ensure the correct formation of the keyway, and the choice depends on the convenience and specifics of the construction of a particular shaft.

3.1.3 Selecting keyway sizes

To correctly create keyways in this laboratory work, students must use the standard dimensions given in Table 1 according to DSTU GOST 24069:2005. The table shows the recommended parameters of the grooves for different shaft diameters.

Table 1 – Shape and dimensions of prismatic keys (DSTU GOST 24069:2005)

Option number	Shaft diameter d	Keyway cross-section b×h	Keyway		Chamfer $S_1 \times 45°$ or radius r	
			shaft t_1	bushings t_2	max.	minimum
1	22...30	8 × 7	4.0	3.3	0.25	0.16

Option number	Shaft diameter d	Keyway cross-section b×h	Keyway		Chamfer S_1 ×45° or radius r	
			shaft t_1	bushings t_2	max.	minimum
2	30...38	10 × 8	5.0	3.5	0.25	0.16
3	38...44	12 × 8	5.0	3.3	0.4	0.25
4	44...50	14 × 9	5.5	3.8	0.4	0.25
5	50...58	16 × 10	6.0	4.3	0.4	0.25

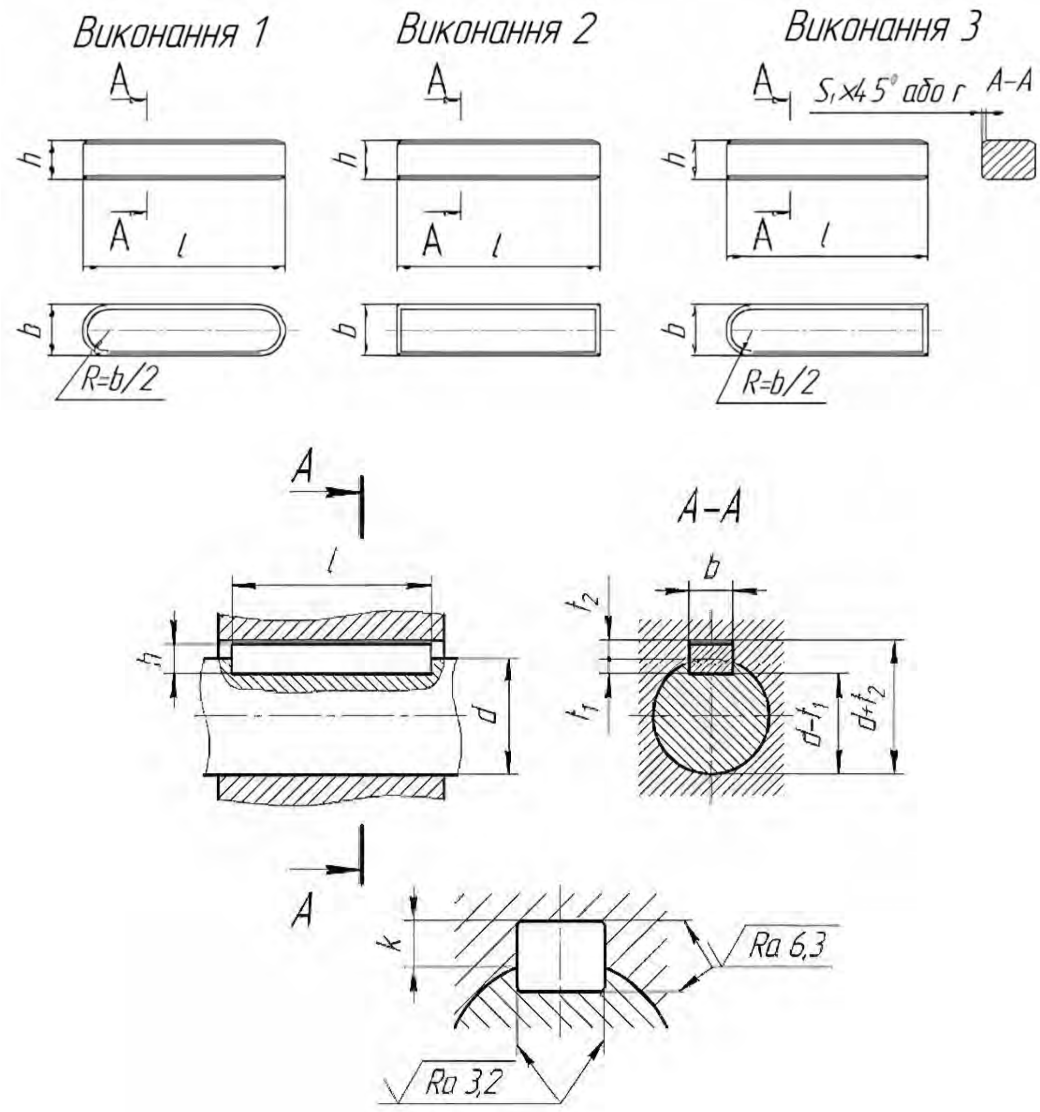


Figure 15 – Drawings with designations of main dimensions (to Table 1)

Keyway parameters

For each shaft diameter, the following are given (see Fig. 15 and Table 1):

- *Key cross-section (b×h)* – the width and height of the prismatic key.
- *The depth of the groove in the shaft (t_1)* is a recommended value.
- *The depth of the groove in the sleeve (t_2)* is a recommended value.
- *Chamfer (S_1 ×45°) or radius (r)* – parameters for processing the edges of the groove (recommended maximum and minimum values).

Determining the dimensions of the groove

– Before creating a groove, the student must determine the diameter of the corresponding shaft segment.

– Select the appropriate values from table 1 for the key cross-section and groove depth.

Creating a groove

– create a sketch of the groove according to the specified dimensions, and cut to the appropriate depth using one of the methods described above.

Chamfering and rounding

– If the drawing provides for chamfers on the edges of the groove, use the **Chamfer command** with the parameter S_1 ×45° specified in Table 1.

– If rounding is necessary, use the **Fillet command**, specifying the appropriate radius r, the values of which are also given in Table 1.

3.2 Construction of grooves

Grooves on shafts are commonly used to accommodate retaining rings, O-rings, or other components that require precise positioning. Creating them in a 3D model is essential to meet technical requirements.

When creating grooves on a shaft, students should follow the standard dimensions given in Table 2.

This data contains the parameters of the width, depth and radius of the grooves in accordance with DSTU ISO 286-2:2002.

The presented parameters ensure correct execution of the grooves according to technical requirements and contribute to the accurate reproduction of the shaft geometry in SolidWorks.

Table 2 – Grinding tolerance (DSTU ISO 286-2:2002)

<table>
<tr><th colspan="2">b to execute</th><th rowspan="2">External grinding d_1</th><th rowspan="2">Internal grinding d_2</th><th rowspan="2">h</th><th rowspan="2">r</th><th rowspan="2">r_1</th><th rowspan="2">d ≈</th></tr>
<tr><th>1; 2</th><th>3</th></tr>
<tr><td>1</td><td>—</td><td rowspan="2">d−0.3</td><td rowspan="2">d + 0.3</td><td rowspan="2">0.2</td><td>0.3</td><td>0.2</td><td rowspan="3">≤10</td></tr>
<tr><td>1.6</td><td>—</td><td rowspan="2">0.5</td><td rowspan="2">0.3</td></tr>
<tr><td>2</td><td>—</td><td rowspan="2">d−0.5</td><td rowspan="2">d + 0.5</td><td rowspan="2">0.3</td></tr>
<tr><td>3</td><td>1.5</td><td>1</td><td rowspan="2">0.5</td><td>> 10–50</td></tr>
<tr><td>5</td><td>2.25</td><td rowspan="3">d−1</td><td rowspan="3">d + 1</td><td rowspan="3">0.5</td><td>1.6</td><td>> 50–100</td></tr>
<tr><td>8</td><td>2.8</td><td>2</td><td rowspan="2">1</td><td rowspan="2">> 100</td></tr>
<tr><td>10</td><td>5.0</td><td>3</td></tr>
</table>

Note: When grinding several different diameters on the same part, it is recommended to use grooves of the same size.

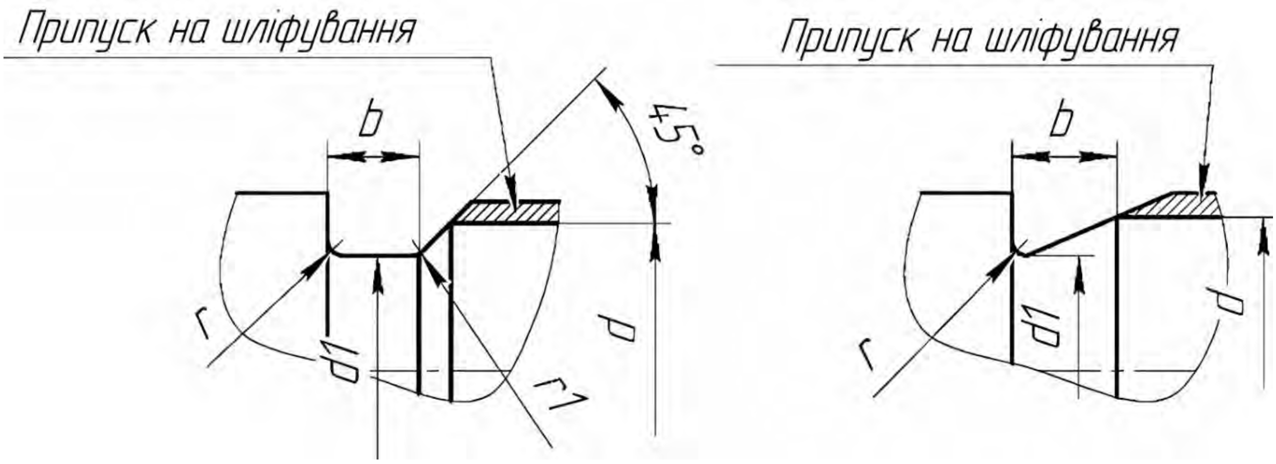


Figure 16 – Drawings with designations of the main dimensions of the grooves (to Table 2)

Methods for creating grooves in SolidWorks

There are two ways to create grooves that can be used depending on the method of shaft construction:

1. **Adding grooves to a general shaft sketch followed by using Revolved Cut**
   - If the shaft is created using the **Revolved Boss/Base method** , then the grooves can be immediately provided in the overall shaft sketch.
   - To do this, it is necessary to add notches to the sketch profile according to the groove sizes given in the table (Fig. 17).

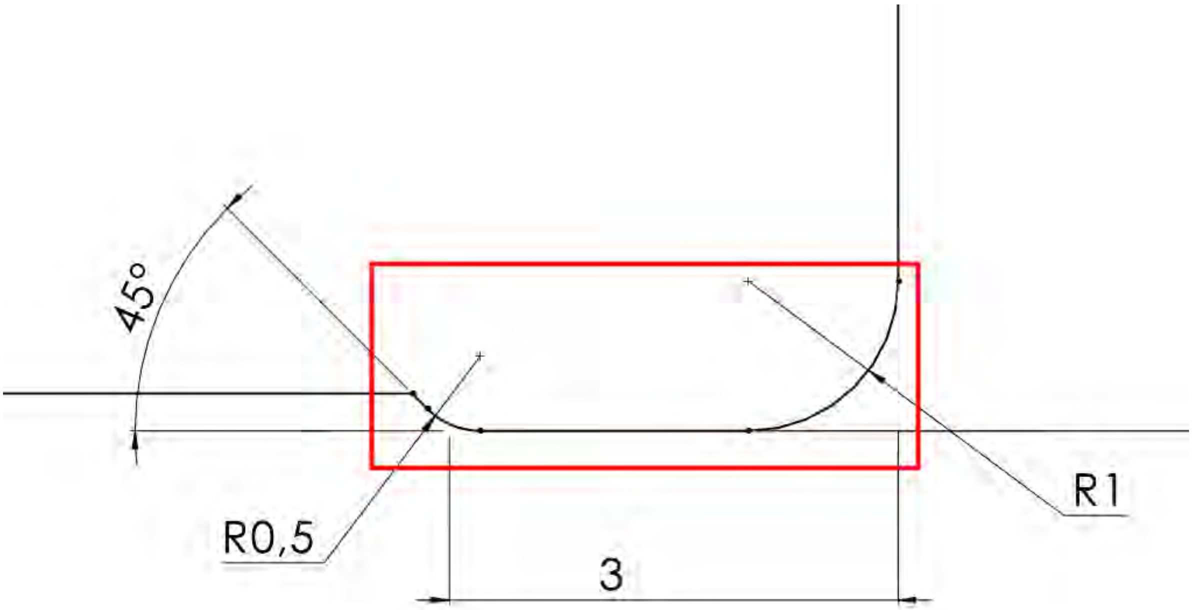


Figure 17 – Creating a groove profile in a general shaft sketch

   - Once the sketch is complete, **the Revolved Cut command is applied**, which cuts the grooves simultaneously with the formation of the cylindrical shaft segments.

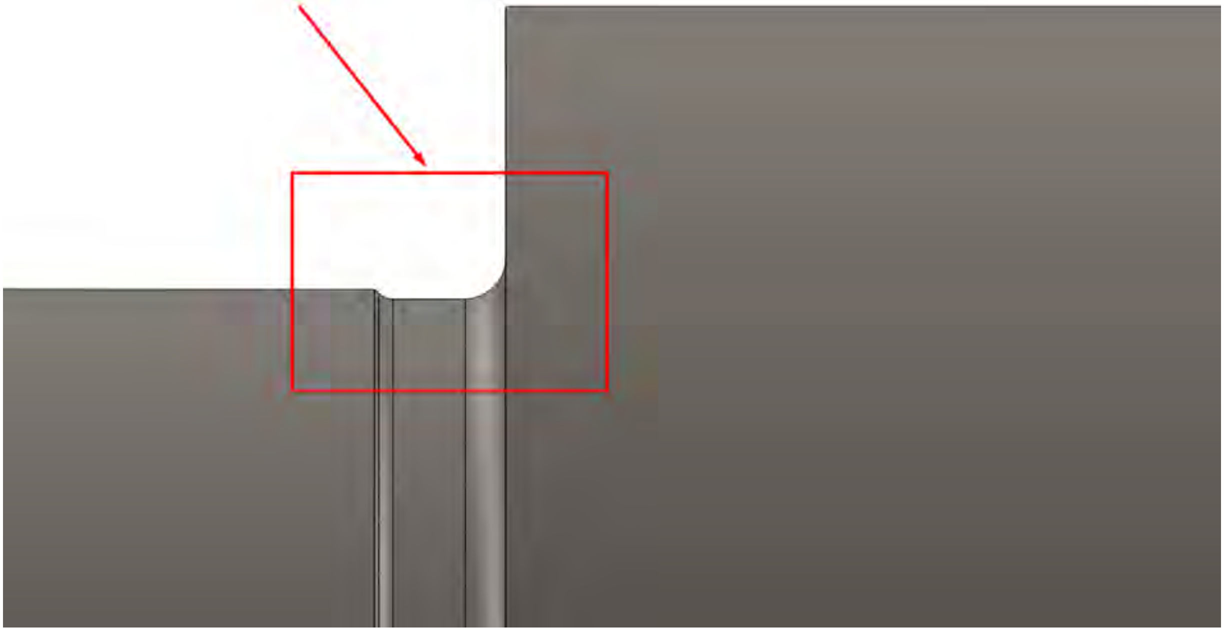


Figure 18 – View of the groove in the three-dimensional shaft model

2. **Creating grooves as a separate element**
   - If the shaft has already been created without grooves, they can be added separately using a new sketch.
   - Select a plane that passes through the shaft axis (for example, **Front Plane**) and sketch the groove profile.
   - Use the standard groove width, depth, and fillet dimensions listed in Table 2.
   - Once the sketch is complete, apply the **Revolved Cut command**, rotating the profile around the shaft axis to create the groove.

Recommendations for constructing grooves

- Before creating a groove, determine the appropriate shaft segment diameter and select the groove parameters from Table 2.
- If the drawing contains roundings, they should be made according to the radii r and r_1 (see Fig. 16, 17 and Table 2).
- It is important to verify that after applying **Revolved Cut,** all elements remain correct and meet the requirements of the drawing.

Using standard sizes ensures compliance with technical standards and simplifies the modeling process.

3.3 Creating cylindrical holes in a shaft model

If the shaft drawing contains cylindrical holes or cutouts (for example, through, countersunk, threaded, or shaped), they must be accurately reproduced in the model, adhering to the specified diameter and depth dimensions. There are three main commands you can use to create such elements in SolidWorks:

1. **Extruded Cut** – for simple through or blind holes of cylindrical shape.
2. **Revolved Cut** – for shaped holes that have a variable diameter or rounding.
3. **Hole Wizard** – for standard threaded, countersunk, countersunk and shaped holes.

Cutouts using Extruded Cut

This method is suitable for creating holes of simple geometry that have the same diameter throughout the entire depth.

Construction process:

- Select the plane on which the hole will be located (for example, the end surface of a shaft segment or the plane of symmetry).
- Sketch a circle that defines the diameter of the hole.
- Use the **Extruded Cut command**, specifying the required hole depth:
  - For through holes, select Through **All**.
  - For blind holes, set the depth manually according to the drawing.
- Make sure the hole size meets the drawing requirements.

Cutouts using Revolved Cut

If the drawing involves a shaped hole (for example, countersunk or stepped), it is advisable to use the **Revolved Cut command**, which allows you to obtain a cut by rotating the sketch around an axis.

Construction process:

- Select a plane that passes through the shaft axis (for example, **Front Plane**).

- Create a sketch of the hole profile, accurately specifying diameters, depths, and corner chamfers according to the drawing.
- Use the **Revolved Cut command**, specifying the axis of rotation.
- Make sure that the resulting geometry matches the drawing.

Holes using Hole Wizard

Hole Wizard command allows you to quickly create standard holes, including threaded, countersunk, or blind holes for fasteners.

Construction process:

- Select the shaft surface on which you want to make a hole.
- Open the **Hole Wizard menu** and select the hole type (threaded, countersunk, countersunk for screw, etc.).
- Specify the standard and hole size according to the drawing (e.g., metric thread M8 × 1.25).
- Position the hole using dimensions or snaps.
- Apply the operation and verify the correctness of the resulting geometry.

The correct application of these methods guarantees that the 3D model corresponds to the drawing and facilitates the subsequent preparation of technical documentation.

3.4 Creating chamfers

Chamfers on shafts perform several important functions in mechanical engineering and technology. The main purposes of chamfers are:

1. *Easier assembly* – chamfers help guide parts during assembly, especially when installing bearings, keys, couplings, etc.
2. *Reduce stress concentration* – sharp transitions between cylindrical and flat surfaces create areas of increased stress. Chamfers reduce the risk of cracking and failure.

3. *Improved processing processability* – when threading, grinding and turning, chamfers make processing easier and reduce the risk of burrs.
4. *Edge protection of parts* – sharp edges can be easily damaged during transportation or operation; chamfers help avoid such damage.
5. *Improved lubrication* – in some cases, chamfers contribute to better penetration of lubricant into the joint.

Methods for creating chamfers in SolidWorks

To add chamfers in SolidWorks, you use the **Chamfer command**. There are two main approaches to creating them:

1. **Applying the Chamfer command to a finished shaft model** is the most convenient and fastest way.
2. **Creating chamfers directly in the shaft sketch** is an alternative option that allows you to form a chamfer even at the stage of building the base geometry.

Creating chamfers on a finished shaft model

This method is the most convenient because it allows you to quickly add chamfers after completing all the basic model construction.

Construction process:

- From the **Features menu,** select the **Chamfer command.**
- Select the edges or corners where you want to create a chamfer (Fig. 19).
- Specify the chamfer parameters according to the shaft drawing.
- Confirm the operation by clicking **OK.**

Chamfers created in this way can be easily modified or removed if necessary.

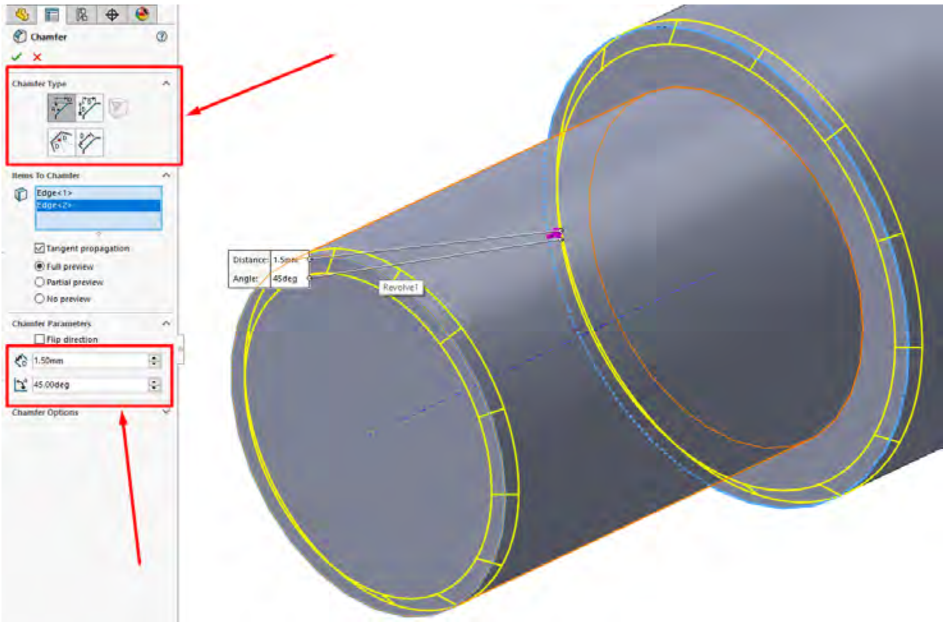


Figure 19 – Specifying chamfer parameters using the Chamfer command

Creating chamfers in a shaft sketch

If the shaft is built using the **Revolved Boss/Base method,** chamfers can be provided at the sketch stage using **the Sketch Chamfer command** .

Construction process:

- When sketching the main shaft profile, add chamfers in the appropriate places using the **Sketch Chamfer command** (Fig. 20).

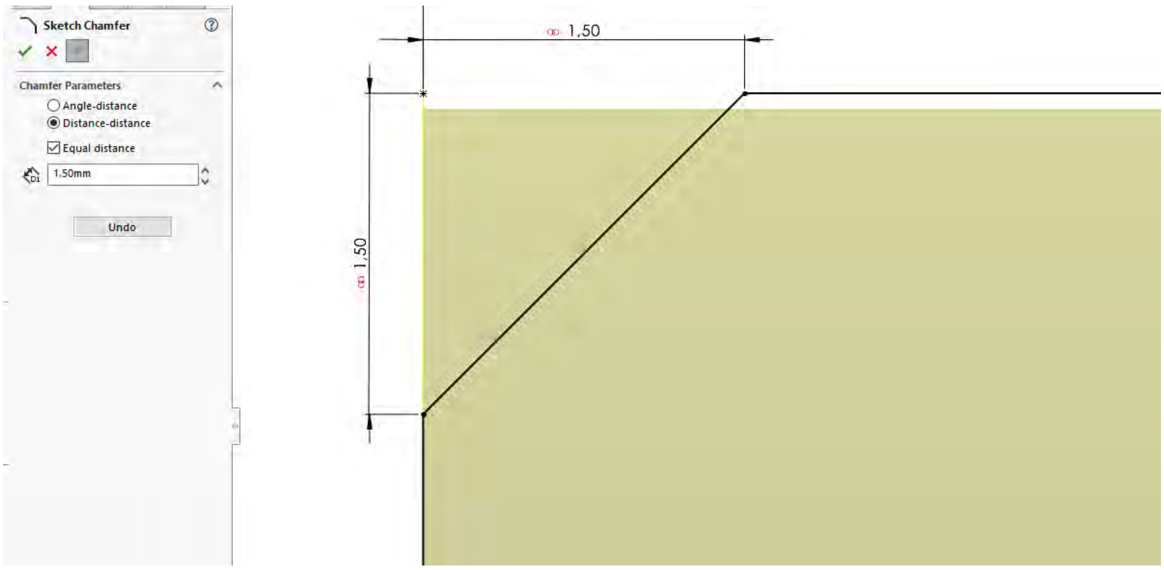


Figure 20 – Adding a chamfer to the main shaft profile sketch

- In the **Property Manager panel**, specify the chamfer parameters according to the shaft drawing (Fig. 20):
  - **Angle-Distance** – specifies the chamfer length and angle (for example, 1 × 45°).
  - **Linear method (Distance-Distance)** – two chamfer lengths are determined.
- Complete the sketch and use the **Revolved Boss/Base command** to create a solid model of the shaft.

This method provides automatic construction of chamfers along with the main geometry, but they will be more difficult to modify if necessary.

3.5 Final shaft model

After completing all stages of construction, we obtain the final three-dimensional model of the shaft, which contains all the necessary structural elements according to the drawing (Fig. 21). In the process of work, the operations of creating the basic geometry, adding grooves, slots, holes, as well as edge processing by adding chamfers and roundings were implemented. All dimensions, shapes and parameters of the elements are determined according to the task drawing, which ensures that the model meets the technical requirements.

The resulting model is fully completed and can be used for further drawing design, geometry compliance analysis, or other engineering calculations. Visual representation of the model allows you to check the correctness of the constructions and evaluate the overall appearance of the part before creating technical documentation.

Figure 21 – Three-dimensional shaft model

4. Creating a shaft drawing

After building a 3D model of the shaft, a technical drawing should be drawn up, which contains all the necessary views, sections, and dimensions.

4.1 Preparing the drawing

Creating a drawing based on a model

After saving the 3D model, use **the File > Make Drawing from Part option** to create a drawing that will automatically load the shaft model.

Select the desired sheet format (for example, A 3) and set the appropriate drawing design standard.

Location of species

Place the main shaft views using **the View Layout menu**. Add auxiliary views, **Section Views**, to show internal structural features such as keyways, chamfers, and fillets.

When placing views, consider ease of reading the drawing, clarity of lines, and compliance with the geometry of the model.

4.2 Dimensioning

Dimensioning of shaft drawing

To apply dimensions, use the **Smart Dimension command**, which allows you to manually specify all the necessary shaft geometry parameters, including diameters, segment lengths, slot width, chamfer and fillet sizes.

Alternatively, you can use the **Import Annotation – Design Annotation function**, which will allow you to automatically transfer dimensions from the model to the drawing, provided that the elements are properly parametrically defined.

Correct placement of dimensions

Arrange dimensions in a way that they do not interfere with each other and are clearly understood. All dimensions must correspond to the parameters specified in the task drawing.

4.3 Hatching of sections

Hatching settings

To display sections, use the standard hatch type used in SolidWorks (for example, Hatch ANSI 31). or another appropriate type set in the drawing template).

Make sure the rotation angle and hatch scale settings meet the requirements of the drafter.

5. Recommendations for documentation

File organization

Save your 3D model and drawings in appropriate folders, using clear file names that indicate the task variant. This will help maintain the sequence of work and make it easier to find files later.

Documenting the process

It is recommended to include screenshots of key stages of construction in the documentation – creation of sketches, command dialogs (**Extruded Boss/Base, Revolved Boss/Base, Extruded Cut, Straight Slot, Chamfer, Fillet**), as well as the final appearance of the model and drawing. Such documentation will contribute to a better understanding of the process and will help in further self-assessment of the work.

Compliance with standards

The design of the drawing must comply with established state standards. Pay attention to the correct arrangement of views, correct design and application of fonts, scales, and hatching.

Figure 22 shows an example of a shaft drawing design.

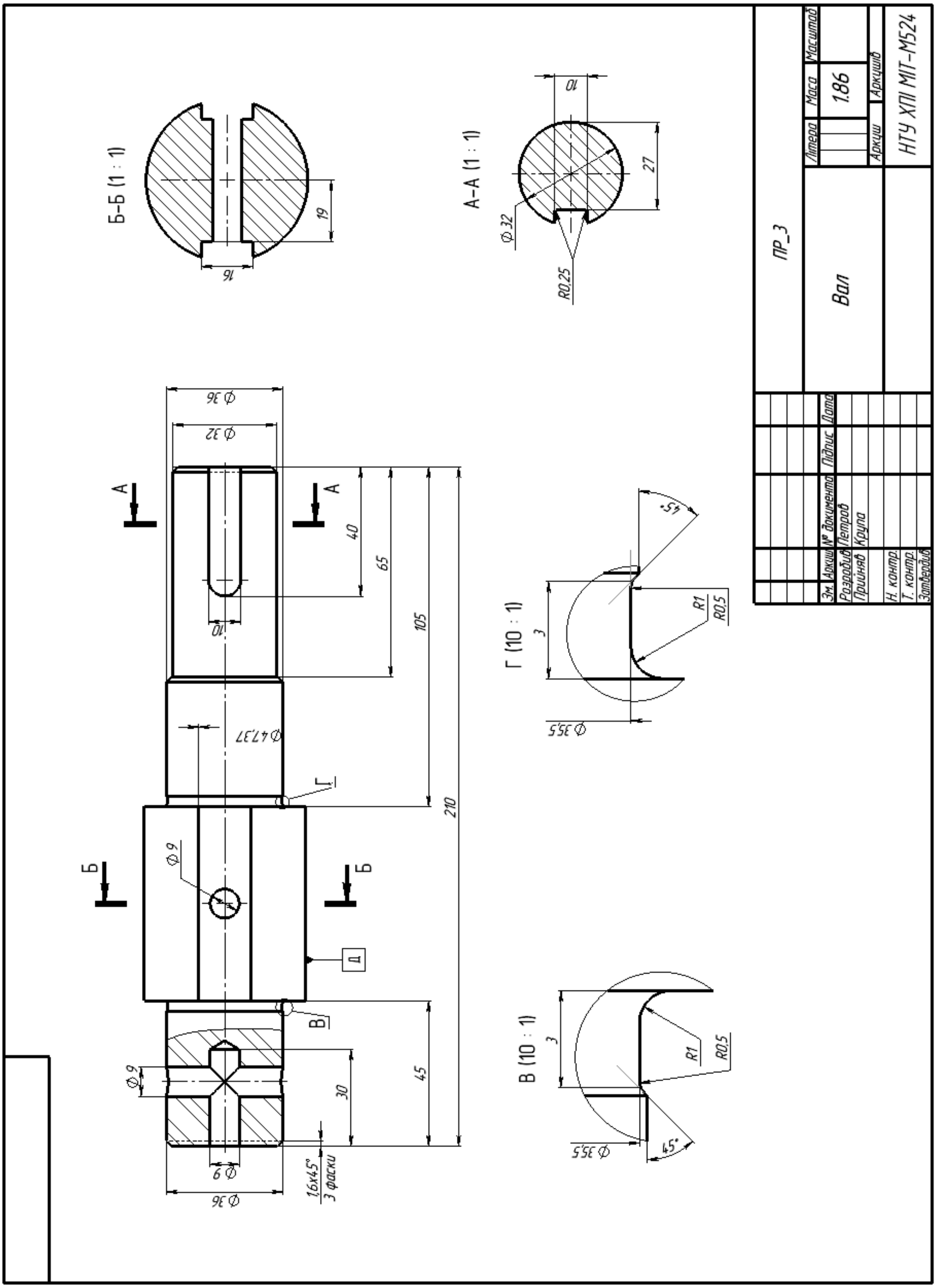


Figure 22 – Example of shaft drawing design

Conclusion

While performing laboratory work on building a three-dimensional model and creating a shaft drawing in SolidWorks, the student must:

• Create a three-dimensional model of the shaft with all components (grooves, cutouts, keyways, chamfers, roundings, etc.);

• Create a technical drawing. Arrange the main views and auxiliary views, including sections, to show the internal geometry of the shaft and structural elements.

Following these methodological guidelines will allow students to obtain a high-quality three-dimensional model of the shaft and a corresponding drawing that meets the requirements of the task.

Laboratory work will help develop parametric modeling skills, clearly define sketches using dimensions and relationships, and correctly design technical documentation in SolidWorks.

The knowledge gained will be useful for further work in CAD systems and design activities.

Lab Work №4: Strength analysis of a part in SolidWorks Simulation module

1. Introduction

Modern engineering design is impossible without the use of computer-aided design (CAD) and computer-aided engineering (CAE) systems. The integration of these systems allows not only to create accurate three-dimensional models of products, but also to conduct virtual tests of their behavior under the influence of various loads at the design stage. This significantly reduces development time, reduces the cost of manufacturing physical prototypes and increases the reliability of the final product.

The SolidWorks software package contains a powerful built-in SolidWorks Simulation module designed to perform various engineering calculations, including structural strength analysis using the finite element method (FEM).

This laboratory work is aimed at students acquiring practical skills in creating three-dimensional models of parts according to drawings and conducting their static analysis for strength using SolidWorks and SolidWorks Simulation tools.

The purpose of the work is to acquire practical skills in creating three-dimensional models of parts in the SolidWorks environment and conducting static analysis of structural strength using the SolidWorks Simulation module to assess the stress-strain state and safety factor.

Job objectives:

1. To get acquainted with the basic principles of performing strength calculations using the finite element method in CAE systems.
2. Create a solid three-dimensional model of a part in SolidWorks according to an individual task (drawing).
3. Perform a static strength calculation in the SolidWorks Simulation module.
4. Analyze the results obtained: stress diagrams (according to Mises), displacements, and safety factor.
5. Make a report on the work performed.

1 Brief theoretical information

1.1 Basics of strength calculations using CAD/CAE systems

Engineering analysis (CAE – Computer-Aided Engineering) is an integral part of the product development process. The use of CAE systems integrated with CAD systems allows you to model the behavior of structures under the influence of operational loads, analyze thermal processes, flows of liquids and gases, and optimize structures according to various criteria.

Strength analysis is one of the most important types of engineering analysis. Its main purpose is to determine whether a structure can withstand the loads applied to it without failure or unacceptable deformation.

The basis of most modern CAE systems, including SolidWorks Simulation, is the finite element method (FEM – Finite Element Method). Element Method). The essence of the method is that a complex geometric model of a part (a solid body) is divided into a large number of simple elements (finite elements - triangles, tetrahedra, etc.), which are interconnected at nodes.

For each finite element, equations are written that describe its behavior under load (based on the theory of elasticity, Hooke's law, etc.). These equations are then combined into a single system of equations for the entire model. Solving this system allows us to determine the main parameters of the stress-strain state at each node and element of the mesh:

- ***Stress:*** Internal forces that arise in a material under the action of external loads (measured in Pa or psi). The most common analysis is the Mises equivalent stress, which is compared to the yield strength of the material.
- ***Displacement:*** change in the position of points in a structure under the action of loads.
- ***Deformations:*** relative changes in the dimensions of structural elements.
- ***Safety factor*** (FOS – Factor of Safety): the ratio of the allowable stress (usually the yield strength or tensile strength of the material) to the maximum design stress in the structure. It shows how many times the applied loads are less than those that can cause failure or plastic deformation. The structure is considered

strong if the minimum SSR is greater than unity (in practice, a standard value is chosen, for example, 1.5; 2; 3 or more, depending on the reliability requirements).

1.2 Stages of conducting engineering analysis in SolidWorks Simulation

The process of performing a strength calculation in SolidWorks Simulation includes the following main stages:

1. ***Preprocessing (Preprocessing):***
   - Create or import a geometric model of a part (CAD).
   - Create a new study (Study) and select an analysis type (for example, Static).
   - Assigning physical and mechanical properties of the part material (Young's modulus, Poisson's ratio, yield strength, etc.). SolidWorks has a large library of standard materials.
   - Setting boundary conditions: fixing certain faces, edges, or vertices of the model to simulate its fixation (Fixtures).
   - Application of external loads: forces, pressures, moments, gravity, etc. (External Loads).
   - Meshing: Breaking the model into finite elements. The quality of the mesh significantly affects the accuracy of the results.
2. ***Solving:***
   - Run the calculation (Run). The program automatically generates and solves the system of equations of the MFE.
3. ***Postprocessing (Postprocessing):***
   - Analysis and visualization of results. SolidWorks Simulation allows you to display results in the form of color plots (graphs) of the distribution of stresses, displacements, strains, and safety factor over the volume or surface of the model.
   - Obtaining numerical values at specific points or areas.
   - Creating reports.

2 Procedure for performing the work

2.1 Creating a three-dimensional model of a part

1. Receive an individual task in the form of a detail drawing.
2. Carefully study the drawings, identify the basic elements and sequence of modeling operations.
3. Launch the SolidWorks application.
4. Create a new document of type Part: File - > New -> Part - > OK.
5. Using the tools in the Features and Sketch panels, sequentially create a three - dimensional model of the part (Fig. 1).

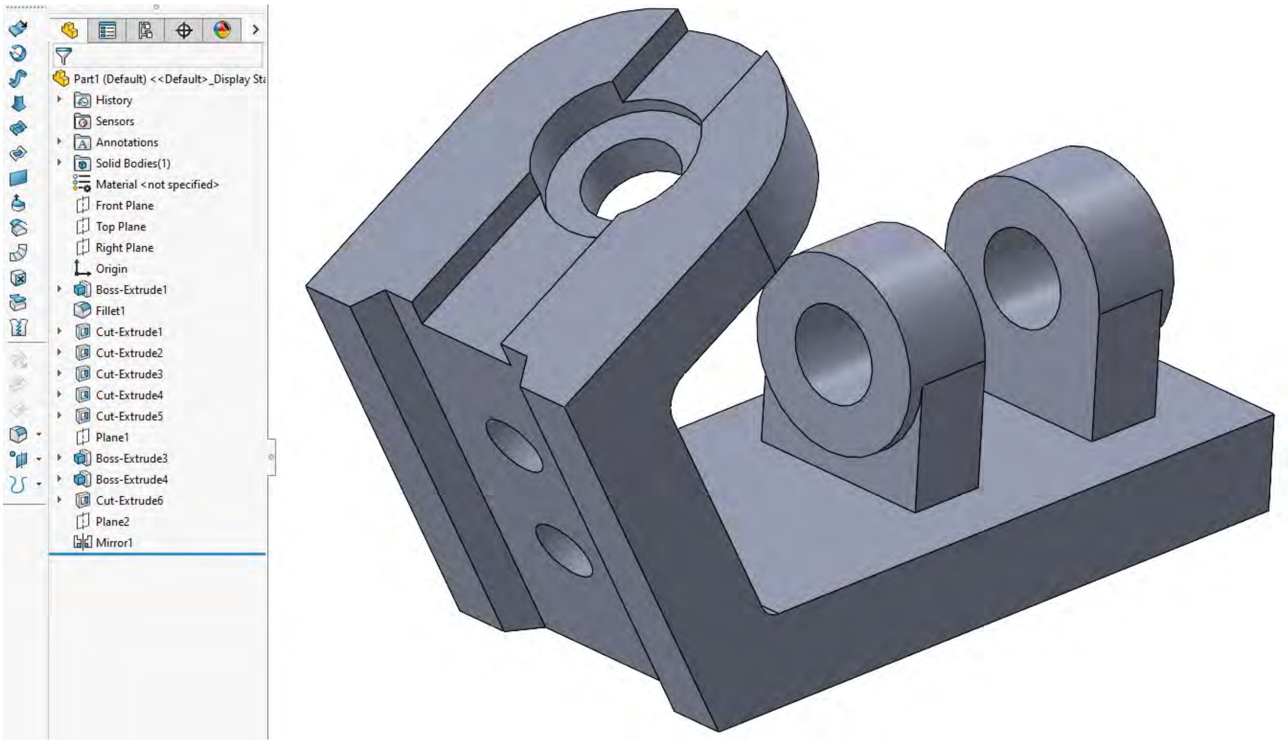


Figure 1 – Three-dimensional model of the part

Basic tools you may need:

- Extruded Body/Base Boss / Base): creating volume by extruding a sketch.
- Revolved Body/Base Boss / Base): creating bodies of revolution.
- Extruded Cutout Cut): Remove material by extruding the sketch.
- Revolved Cutout Cut): removes material by rotating the sketch around an axis.

- Mounting hole (Hole Wizard): creating standard holes (threaded, bolt holes, etc.).
- Fillet: creating fillets on edges.
- Chamfer: creating bevels on edges.
- Mirror: creating symmetrical elements.
- Arrays Linear Pattern, Circular Pattern: creating arrays of elements.

6. Make sure that the geometric dimensions of the model exactly match the drawing. Use the Measure tool on the Evaluate tab to control the dimensions.
7. Save the created part model in your working folder: File - > Save As ... As ...). Give the file a meaningful name (for example, Detail_Variant_XX.SLDPRT).

2.2 Activating the SolidWorks Simulation module

Before starting the calculation, you must make sure that the SolidWorks Simulation application is activated.

1. Go to the Tools - > Add -Ins ... menu.
2. In the window that appears, find SolidWorks Simulation in the list (Fig. 2).
3. Check the boxes next to SolidWorks Simulation in the left column (to activate in the current session) and, optionally, in the right column (to load automatically when SolidWorks starts) (Fig. 2).
4. Click OK.
5. the Simulation tab will appear in the CommandManager (toolbar).

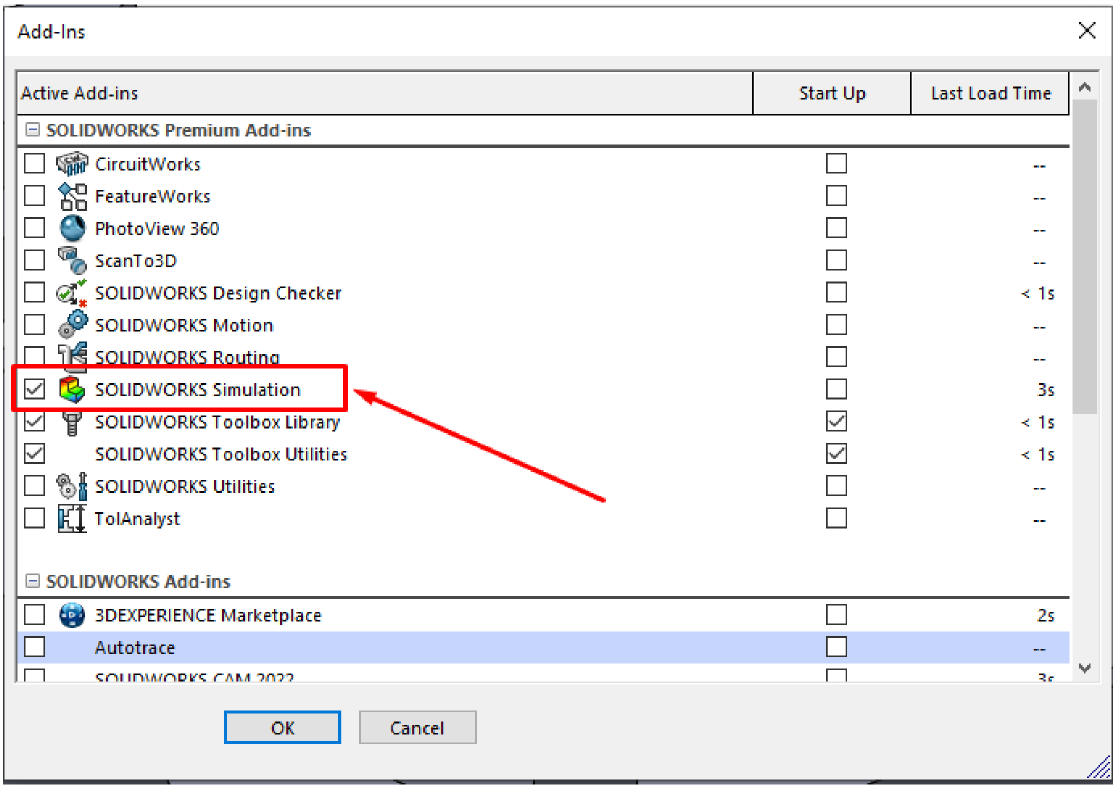


Figure 2 – Activating the SolidWorks Simulation module

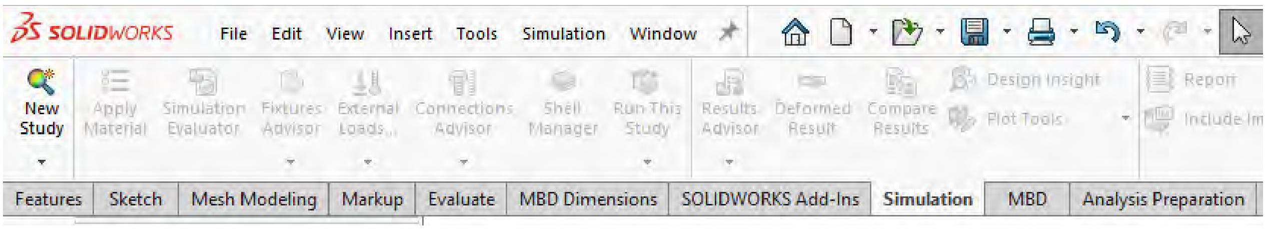


Figure 3 – Simulation tab in CommandManager

2.3 Creating a new static analysis study

1. Go to the Simulation tab in the CommandManager (Figure 3).
2. Click the New Study button (New Study) (Fig. 3).
3. In the PropertyManager (property pane on the left), under General Modeling (General Simulation) select the Static study type (Fig. 4).
4. Optionally, change the study name (default is " Static 1") to more informative.
5. Click the green checkmark (OK) to confirm the creation of the study.
6. in the Simulation study tree (located below the FeatureManager design tree) (see Figure 5).

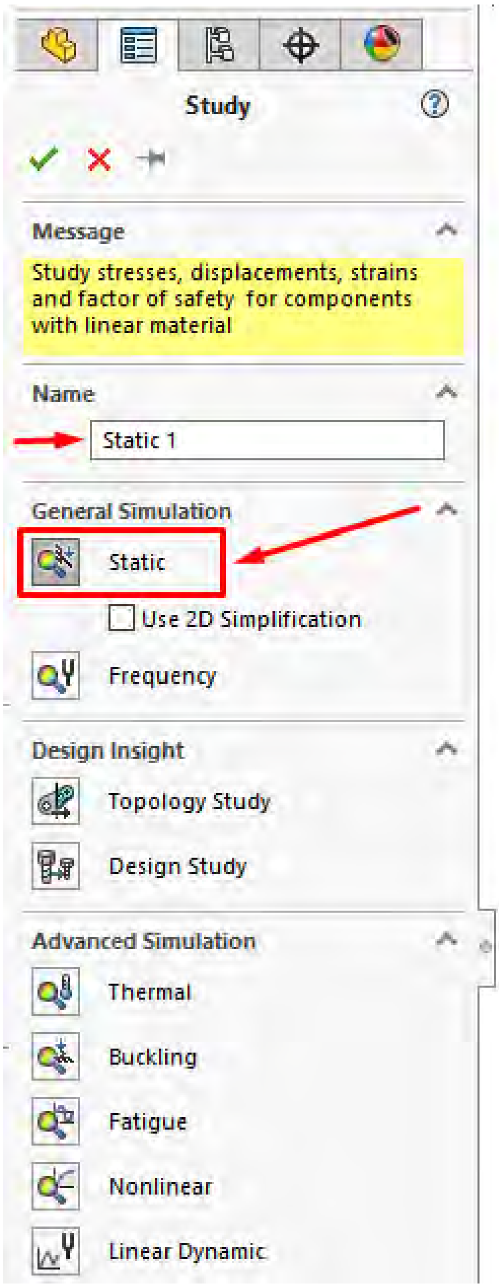


Figure 4 – Creating a new study

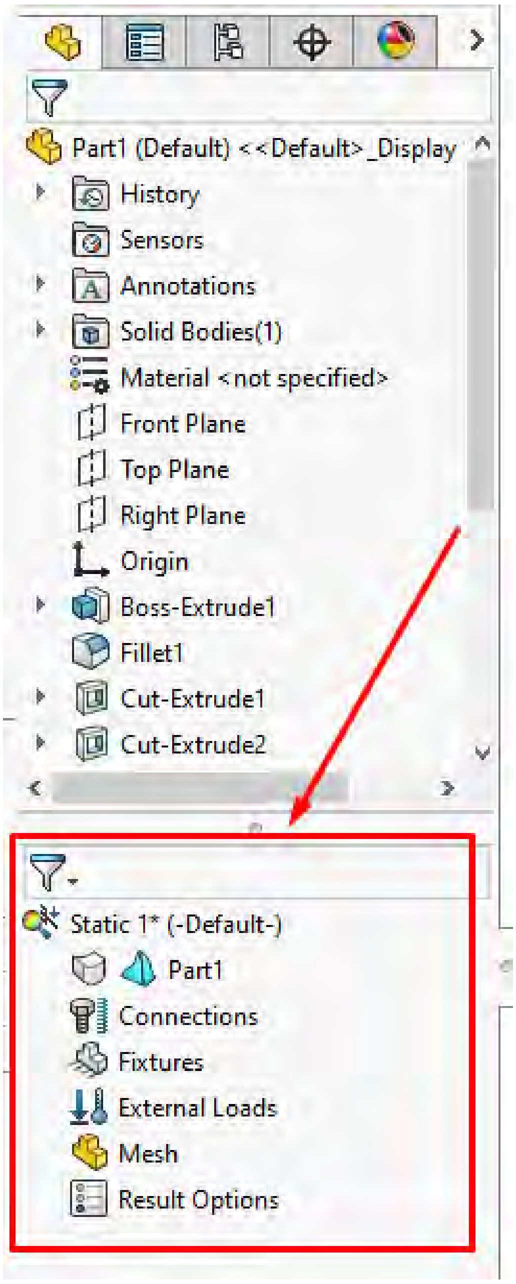


Figure 5 – Simulation research tree

2.4 Applying material to the model

To perform the calculation, it is necessary to specify the physical and mechanical properties of the material from which the part is made.

1. In the Simulation study tree, right-click the part icon (usually has the same name as the part file) and select Apply / Edit Material. Material) (Fig. 6).

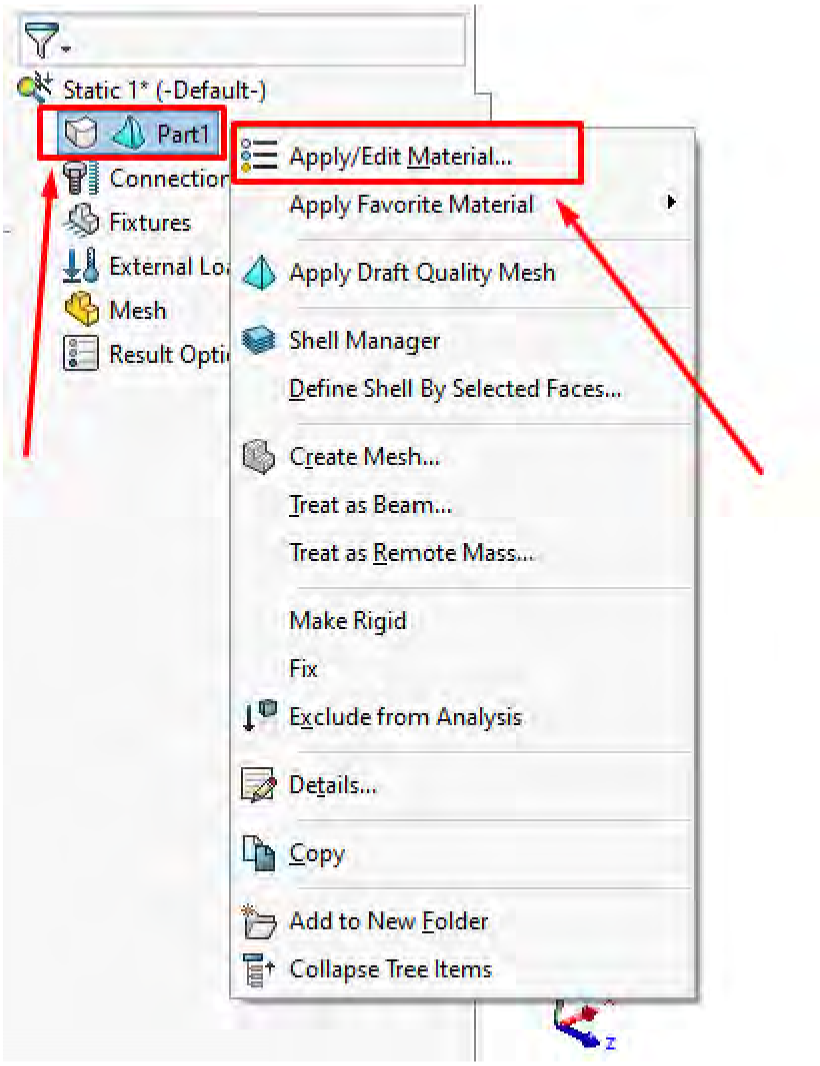


Figure 6 – Applying material to the model

2. The SolidWorks Material Library window opens.
3. Choose the material according to the task or as directed by the teacher. Frequently used materials:
   - Steel: Plain carbon steel Carbon Steel), Alloy steel (Alloy Steel)
   - Aluminum alloys (Aluminum Alloys): 1060 Alloy (1060 Alloy)
   - Others (copper, titanium, plastics, etc.).

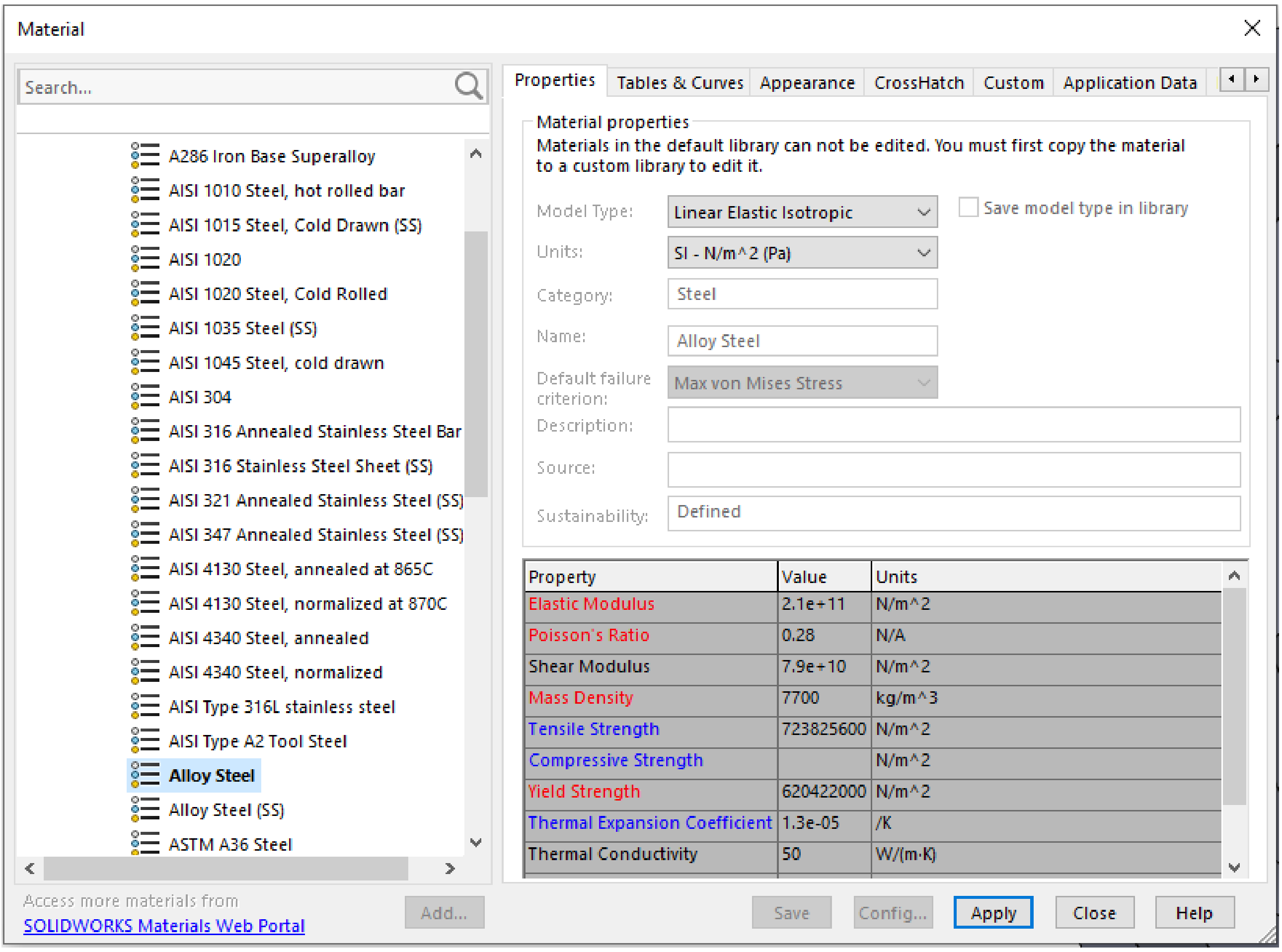


Figure 7 – SolidWorks Material Library Window

4. After selecting a material, its properties (Modulus of Elasticity, Poisson's Ratio, Yield Strength, Density, etc.) will be displayed on the right side of the window (Fig. 7). Make sure that the main properties required for static analysis (in particular, Yield Strength - Yield strength), given.
5. Click Apply, and then Close.
6. In the research tree, a green check mark and the name of the assigned material will appear next to the part name.

2.5 Setting boundary conditions (fixes)

Boundary conditions determine how a part interacts with its environment, i.e., which parts of it are stationary or have limited freedom of movement.

1. In the Simulation study tree, right-click the Fixtures folder and select Fixed Geometry or another fixture type that meets the requirements of the problem

(for example, Roller/Slider to allow movement along a plane, Fixed Hinge to fix cylindrical faces) (Fig. 8).

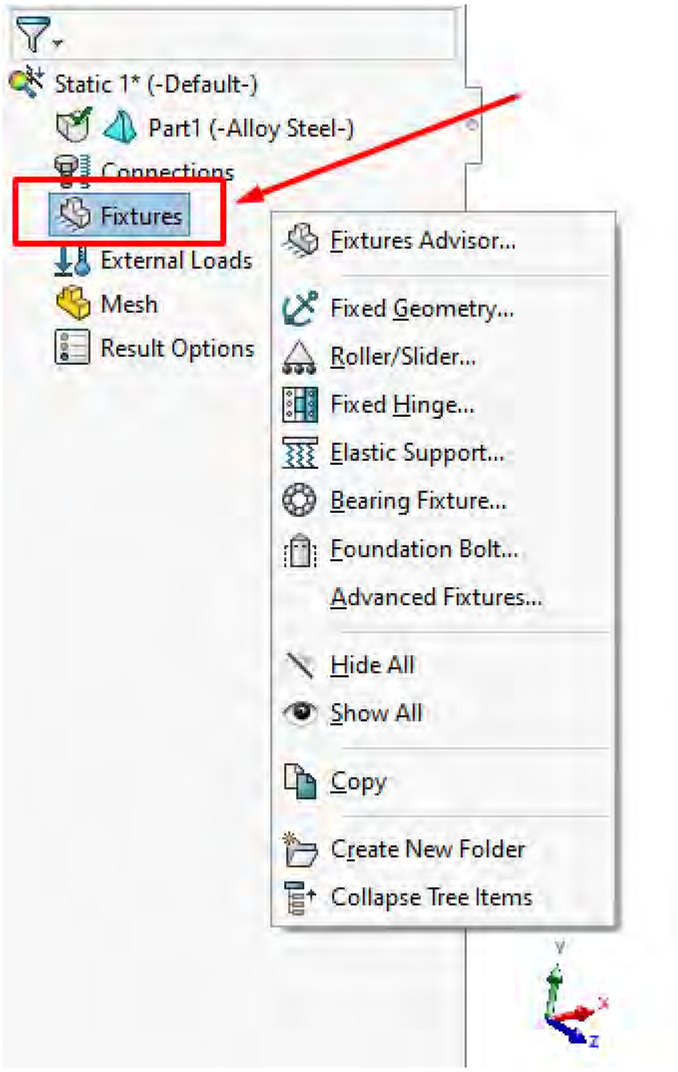


Figure 8 – Setting the part fastening condition

2. Fixed Geometry: This type of fixture completely prohibits any movement or rotation for the selected faces, edges, or vertices. In the PropertyManager:

- In the blue box, Faces, Edges, Vertices for Fastening (Faces, Edges, Vertices for Fixture) select those faces (or other objects) of the model that should be fixed. The selected objects will be highlighted in the graphics window and added to the list (Fig. 9).

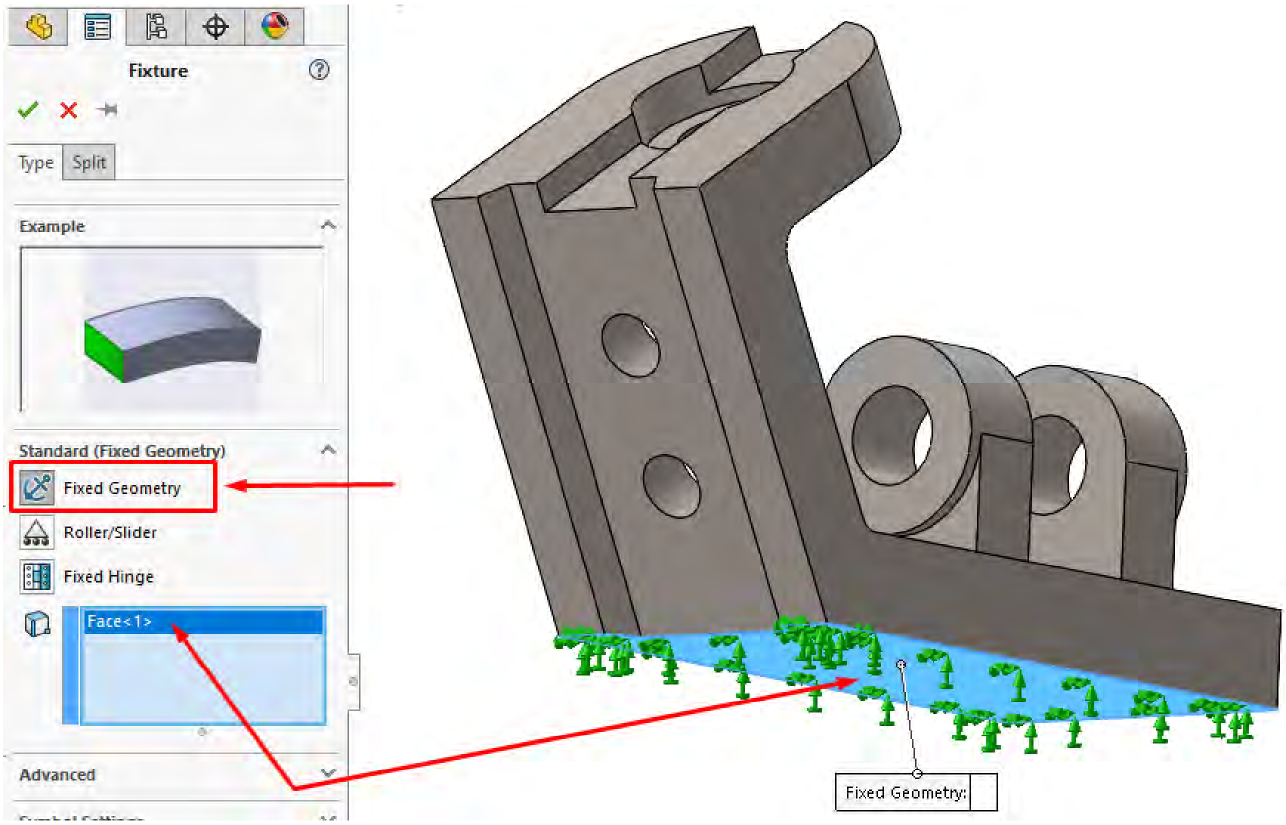


Figure 9 – Selecting a face for fixing a part

- Review the fastener symbols that will appear on the model (Fig. 9).
- Click the green check mark (OK).

3. Specify all necessary fasteners according to the part fastening scheme in the real structure or as instructed by the instructor.

2.6 Load specification

Loads are external influences acting on a part (forces, pressures, moments, temperature, gravity, etc.).

1. In the Simulation study tree, right-click the External Loads folder. Loads) (Fig. 10).
2. Select the load type that best suits your application. The most common ones for static analysis are:
   - Force: Applying a concentrated or distributed force to faces, edges, or vertices.
   - Pressure: Applies pressure evenly distributed across the selected face(s).

- Gravity: Takes into account the effect of the part 's own weight (you must specify the direction of gravity).
- Torque: Applying torque to cylindrical faces.

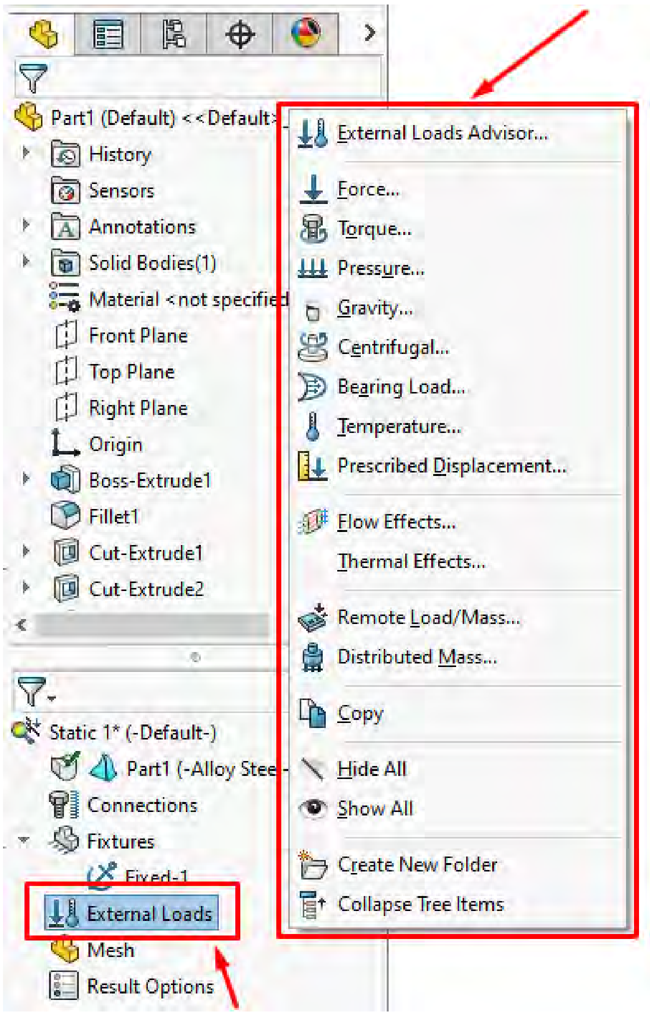


Figure 10 – Load assignment

3. Example of setting Force (Force)
   - Select Force (Fig. 11).
   - In the PropertyManager:
     - In the Selection section, select the faces, edges, or vertices to which the force is applied (Fig. 11).
     - Choose the direction of the force. You can do this:
       - By selecting the Normal option (Normal to) and specifying the face (the force will be perpendicular to it).

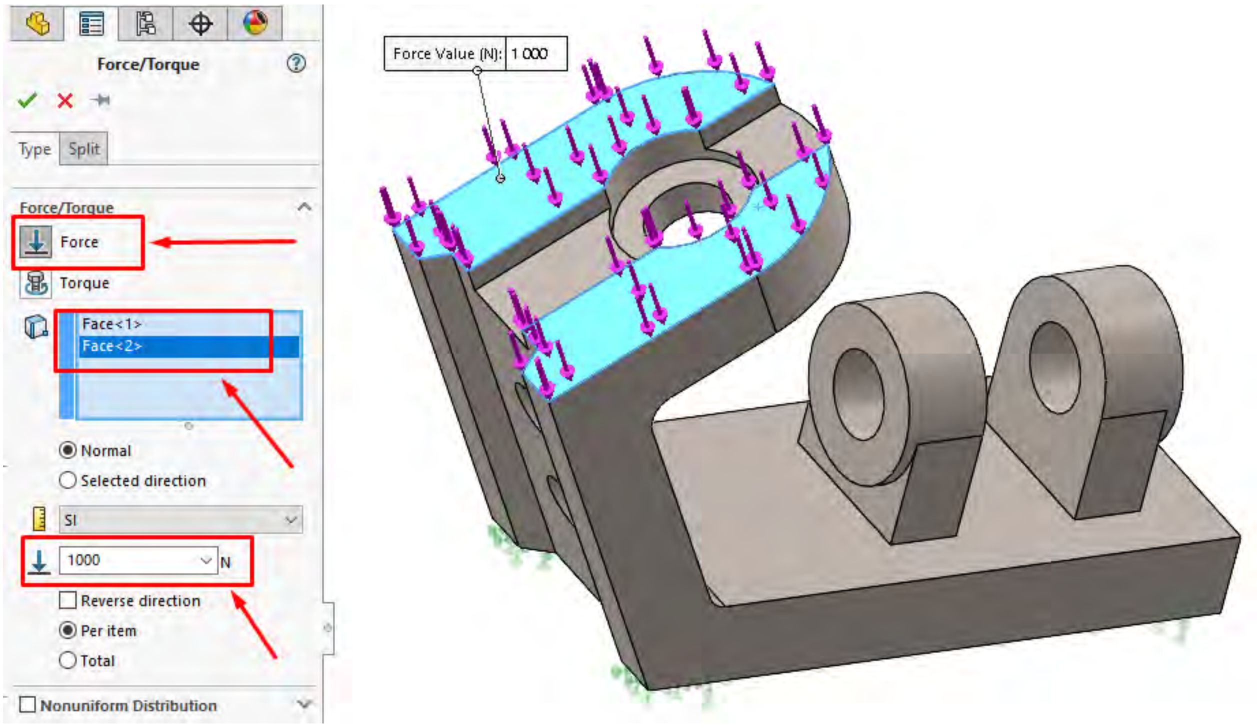


Figure 11 – Selecting a face for applying force and setting its value

- Selected direction option direction) and specifying a plane, axis, or edge to define the direction of the force. You may need to specify the force components along the X, Y, Z axes.
- Activate the Reverse option. direction) if needed.

- In the Force section, enter the numerical value of the force in the appropriate units (N, kgf, lbf).

Make sure the correct unit system is selected (SI, English, etc.) (Fig. 11).

- Click the green check mark (OK).

4. Assign all necessary loads according to the individual task or the teacher's instructions. Pay attention to the magnitude, direction, and location of the loads.

2.7 Creating a finite element mesh

Dividing the model into finite elements (tetrahedra for three-dimensional models) is an important step in MFE.

1. In the Simulation study tree, right-click the Mesh folder and select Create Mesh. Mesh).

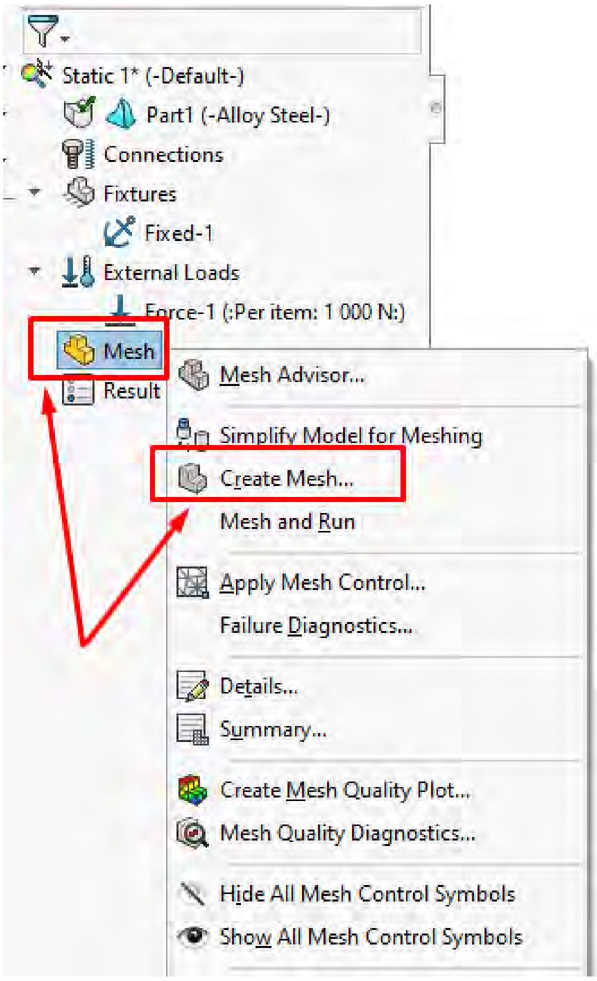


Figure 12 – Creating a calculation grid

2. The Grid Settings PropertyManager opens.
3. For the first calculation, you can leave the default settings:
   - Mesh density (Mesh Density: The slider is usually in the middle, providing a balance between accuracy and calculation time. Moving the slider to the right (Fine) creates a finer (more accurate, but more resource-intensive) mesh, to the left (Coarse) creates a coarser one.
   - Mesh parameters (Mesh Parameters: You can leave Standard mesh or Curvature based mesh. based mesh) (recommended for models with complex geometry).

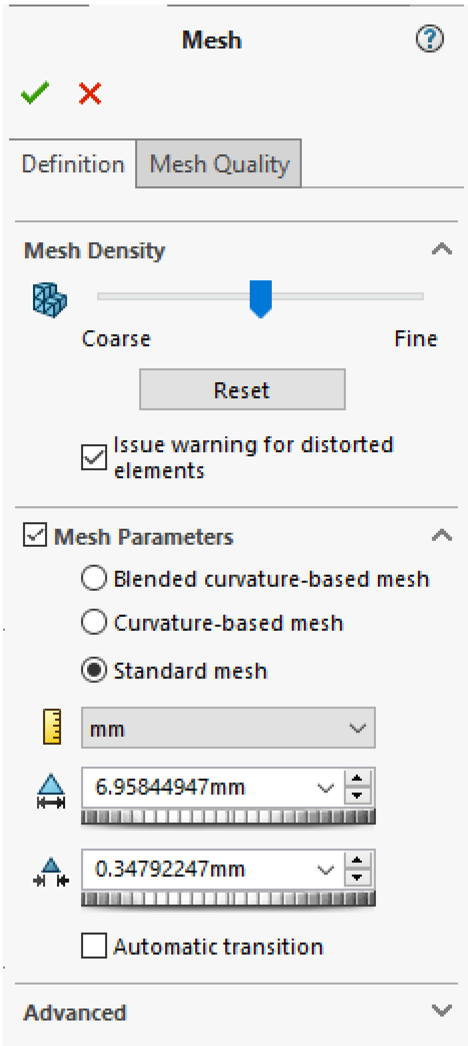


Figure 13 – Setting the calculation grid parameters

4. Click the green checkmark (OK). The program will automatically build the mesh. The build time depends on the complexity of the model and the selected mesh density.
5. After construction, the mesh will be displayed on the model (see Fig. 14). Visually assess the quality of the mesh. It should be sufficiently fine in places of stress concentration (holes, fillets, places of force application), but can be coarser in less loaded areas.

Note: If the calculation results are inaccurate or convergence problems occur, it may be necessary to refine the mesh (reduce the size of the elements globally or locally using the Mesh control tools). Control).

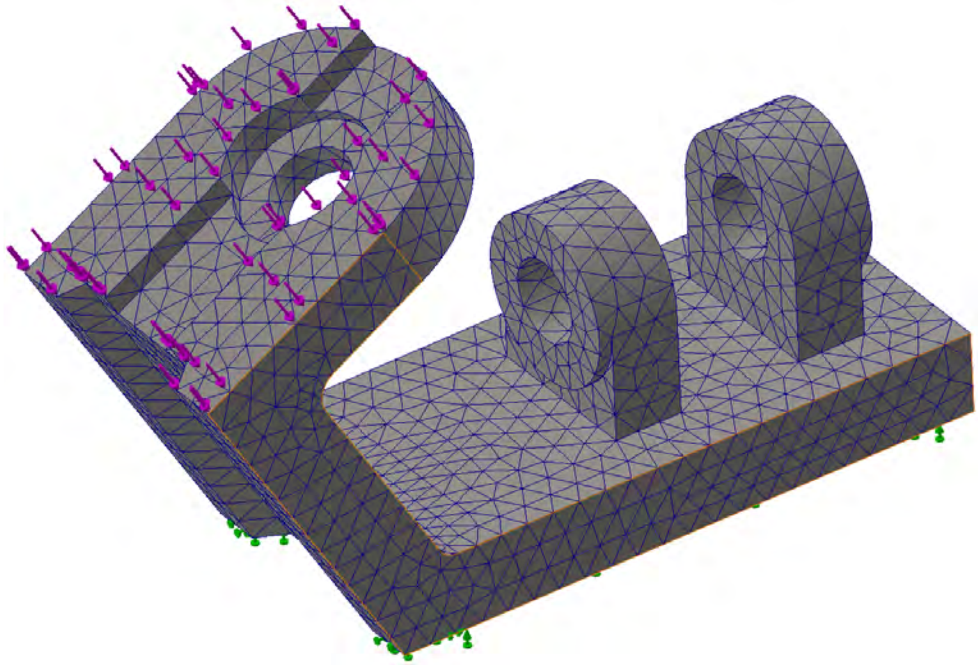


Figure 14 – Model of a part with a constructed finite element mesh

2.8 Starting the calculation

After the material, fasteners, loads have been specified and the mesh has been created, you can start the calculation.

1. In the Simulation study tree, right-click on the study name (for example, " Static 1") (Figure 15).
2. In the context menu, select Run (Fig. 15).
3. solver status window will display the progress. The calculation time depends on the number of mesh elements, the complexity of the problem, and the power of the computer (Figure 16).

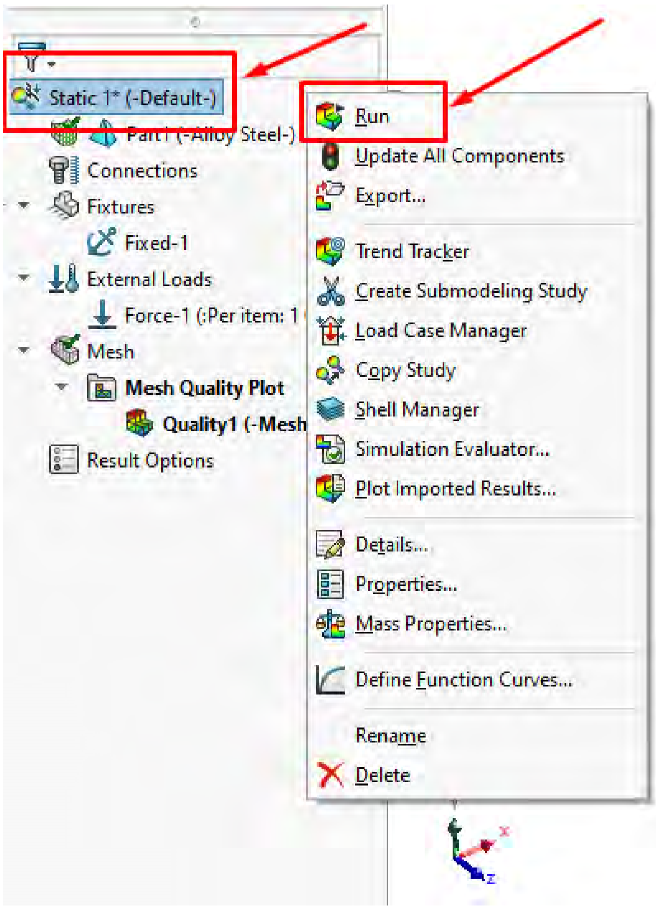


Figure 15 – Starting the calculation

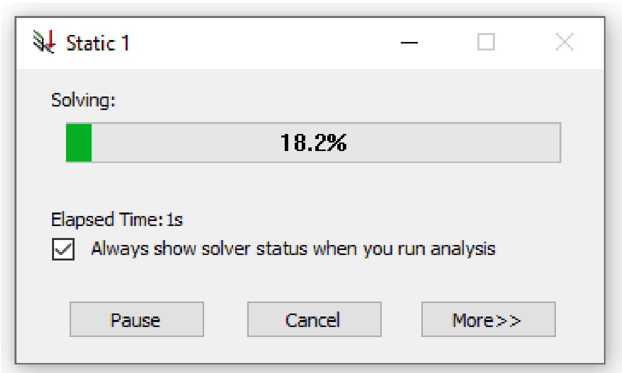


Figure 16 – Solver Status Window

4. After the calculation is successfully completed, the status window will close, and the Results folder with typical plots will appear in the study tree.

2.9 Analysis of results

, SolidWorks Simulation automatically creates several standard results plots that appear in the Results folder:

- Stress1: Usually shows the von Mises equivalent stresses (von Mises Stress).
- Displacement1: Shows the resulting displacements (Resultant Displacement).
- Strain1: Shows equivalent strains.

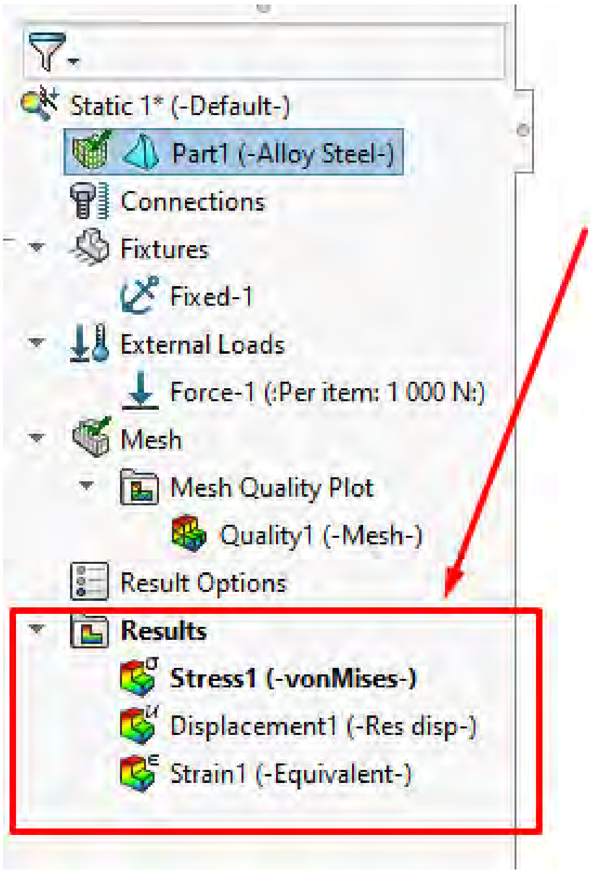


Figure 17 – Results in the research tree

To view any of these graphs, double-click on it in the study tree. The model will be colored according to the parameter values, and a scale (legend) with the corresponding numerical values will appear next to it.

2.9.1 Stress analysis

1. Activate the Stress1 (- vonMises -) graph.
2. Examine the stress distribution across the model. Pay attention to the areas with maximum values (usually marked in red on the default scale). These are the areas of stress concentration that are most dangerous from a strength perspective.

3. Find the maximum stress value on the scale. Compare this value with the yield strength (Yield strength) of the material, which is indicated on the scale by a red arrow (see Fig. 18).
    - If the maximum stress is less than the yield stress, the part will not undergo plastic (irreversible) deformation under a given load.
    - If the maximum stress exceeds the yield strength, plastic deformation will occur in this zone, which may be unacceptable.

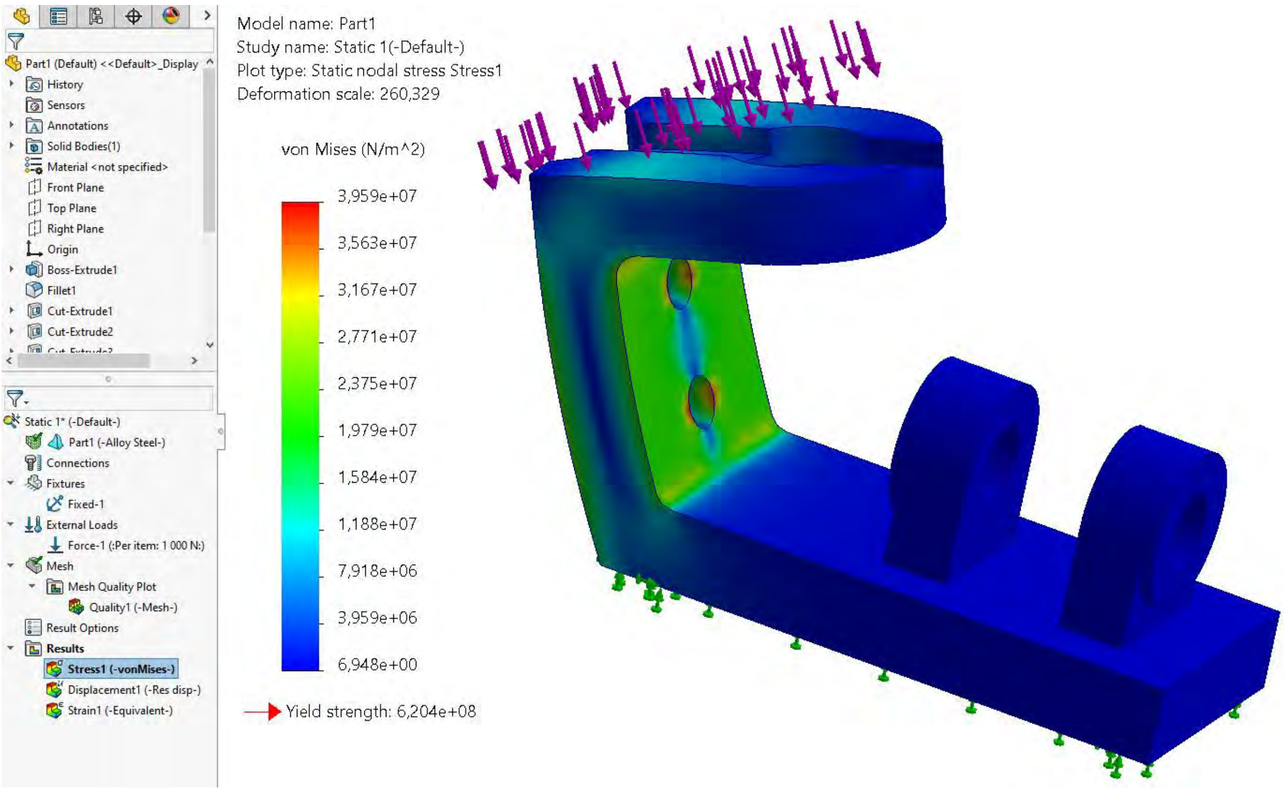


Figure 18 – Mises stress diagram

Analysis tools:

- Probe: Allows you to find out the exact value of the result at a selected point or grid node. Right-click on the results graph -> Probe.
- Chart settings (Chart Options): Allows you to change the display of the scale, units of measurement, etc. (Right mouse button on the graph -> Graph settings).

- Definition parameters: Allows you to change the type of displayed stresses (e.g., principal stresses, normal stresses along the axes), units of measurement (Right mouse button on the graph -> Edit definition (Edit Definition)).

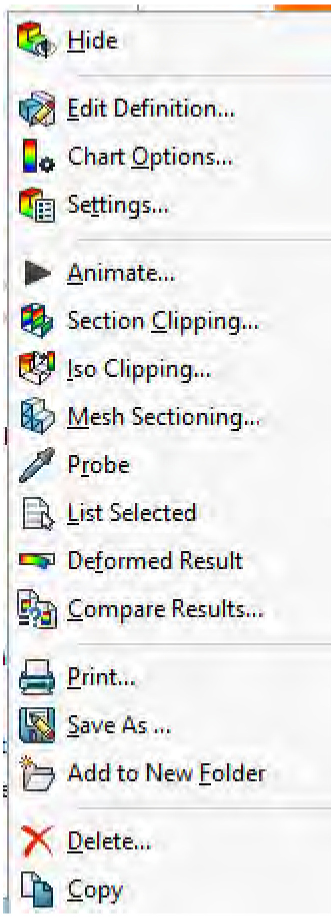


Figure 19 – Analysis tools

2.9.2 Movement analysis

1. Activate the Displacement1 graph (- Res disp -).
2. Study how the part deforms under load. The areas of maximum displacement are marked in red.
3. Find the maximum displacement on the scale. Assess whether this displacement is acceptable for normal operation of the structure.
4. To see a deformed view of the model (usually the displacements are visually magnified for clarity), right-click on the graph -> Edit Definition (Edit Definition -> Definition Options - > Select Deformed Shape Shape). You can adjust the scale of the deformation.

5. , right-click on the graph - > Animate.

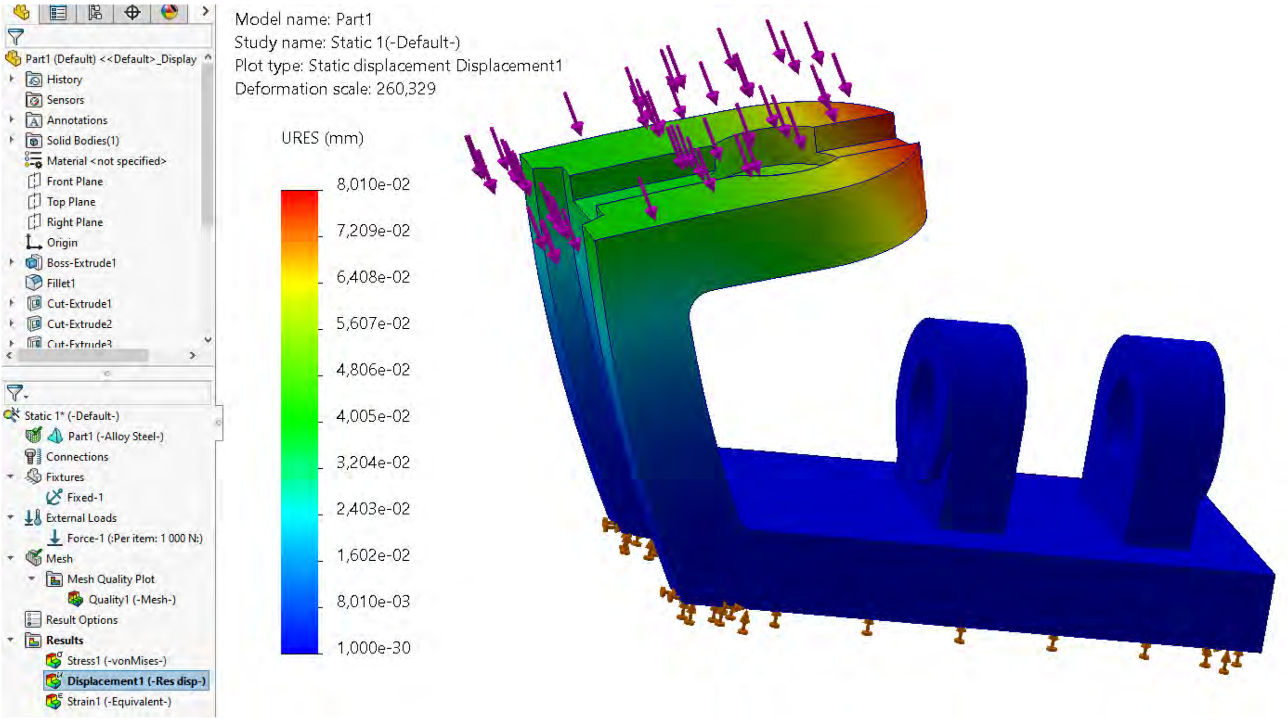


Figure 20 – Example of a displacement diagram

2.9.3 Safety Factor Analysis (SFRA)

Safety factor (Factor of Safety - FOS) is a key indicator for assessing the reliability of the structure.

1. In the study tree, right-click the Results folder and select Define Safety Factor Plot. Factor of Safety Plot) (Fig. 21).
2. The PropertyManager opens. In Step 1 of 3: Component Selection (Step 1 of 3: Component selection):
   - Select All to analyze the entire part.
   - Click Next.
3. In the Step 2 of 3: Criteria section:
   - Select the failure criterion. For ductile materials (steel, aluminum) the Max. von Mises stress (Max from Mises Stress).
   - The program will automatically substitute the value of the material's yield strength in the Yield strength field (Yield strength). Make sure the units of measurement are correct.

- Click Next.

4. In Step 3 of 3: Factor Results (Step 3 of 3: Factor of safety results):
   - Select the Factor Distribution option. of safety distribution).
   - Click the green check mark (OK).

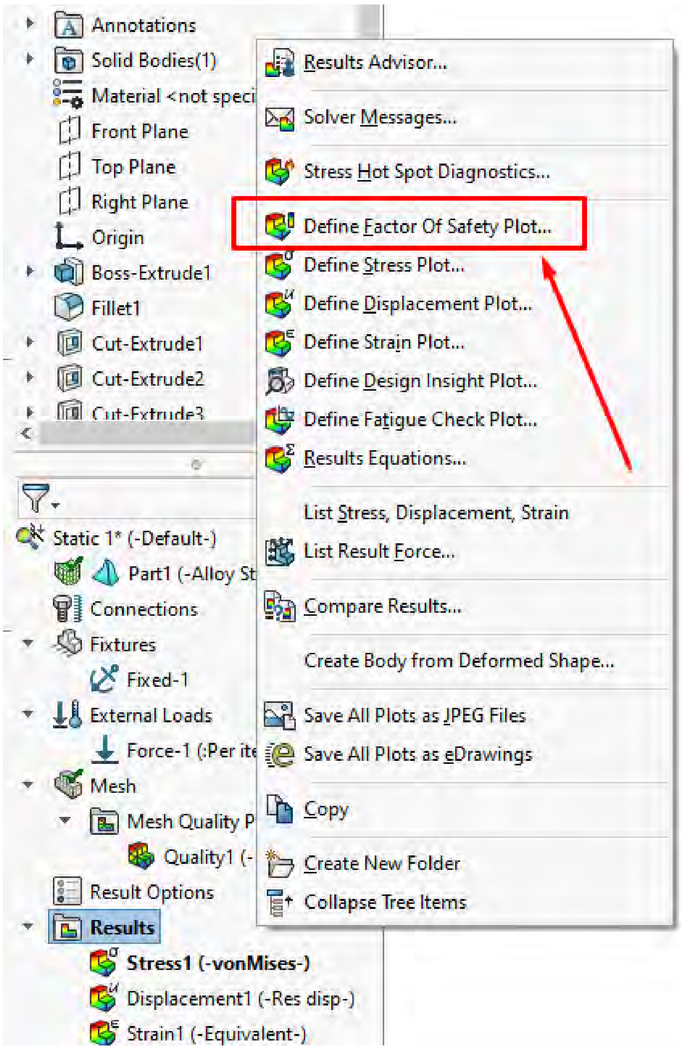


Figure 21 – Selection of the safety factor graph in the research tree

5. A graph of the distribution of CPM according to the model will be created (see Fig. 22). The areas with the lowest CPM (most dangerous) will be marked in red (by default).

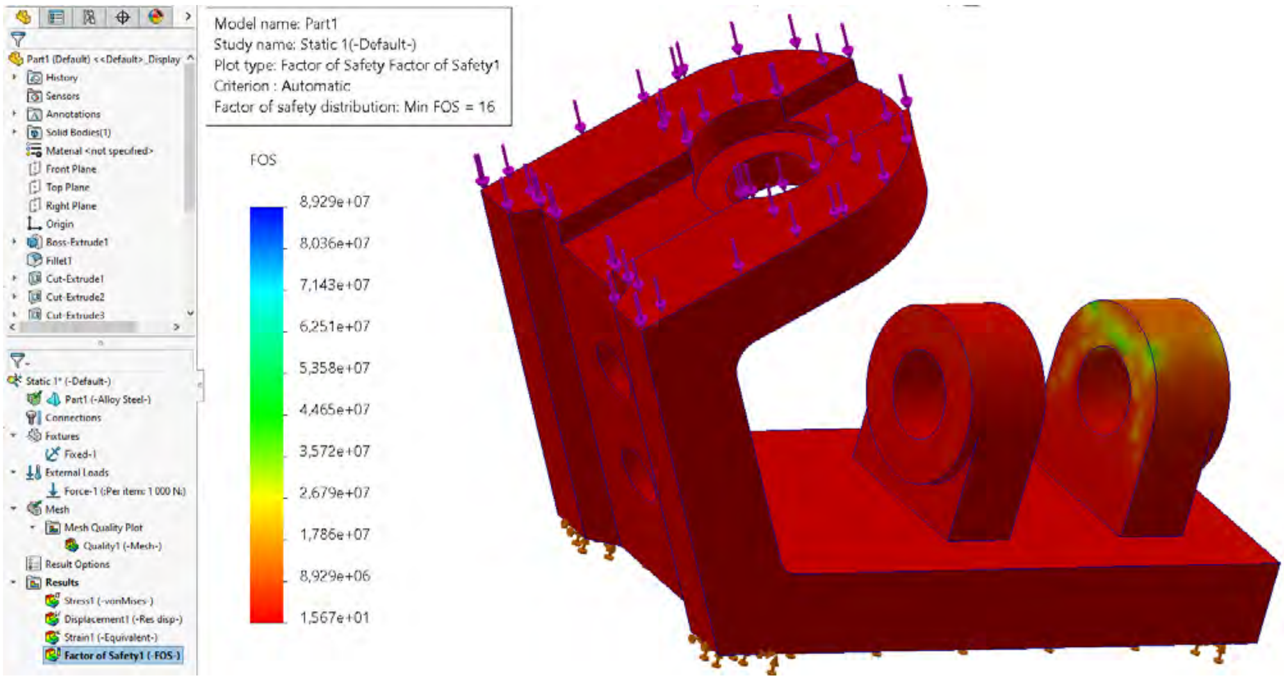


Figure 22 – Distribution of safety factor by model

6. Find the minimum CPM value on the scale or using the Probe tool.
7. Compare the minimum CBM with the standard (permissible) value for this type of structure (usually > 1, often 1.5...3 or more).
   - If the minimum CBM is greater than the allowable one, the structure is considered strong.
   - If the minimum CBM is less than the permissible one, the structure does not meet the strength requirements and requires refinement (change in geometry, material, reduction of loads).

2.10 Creating a report using SolidWorks Simulation

After running a calculation and analyzing the main results, SolidWorks Simulation automatically generates a report that includes key information about the model, study settings, and results. This can significantly speed up the documentation process.

1. **Running the report creation command:**
   - Go to the Simulation tab in the CommandManager.
   - Click the Report button.

Figure 23 – Report creation command in Simulation CommandManager

2. **Configuring report content:**
   - Report window will open. options, where you can customize the content of the future report.
   - You can usually choose which sections to include in the report:
     - Model information (Model information)
     - Study properties (Study properties)
     - Units of measurement (Units)
     - Material properties (Material properties)
     - Loads and fastenings and Fixtures
     - Contact information (if any)
     - Mesh information (Mesh information)
     - Assumptions
     - Results plots) - you can choose which plots to include.
     - Conclusion
     - Appendix
   - You can add your own sections, comments, images (for example, additional screenshots of specific settings or model views).
   - You can specify information about the author, company, logo, etc.

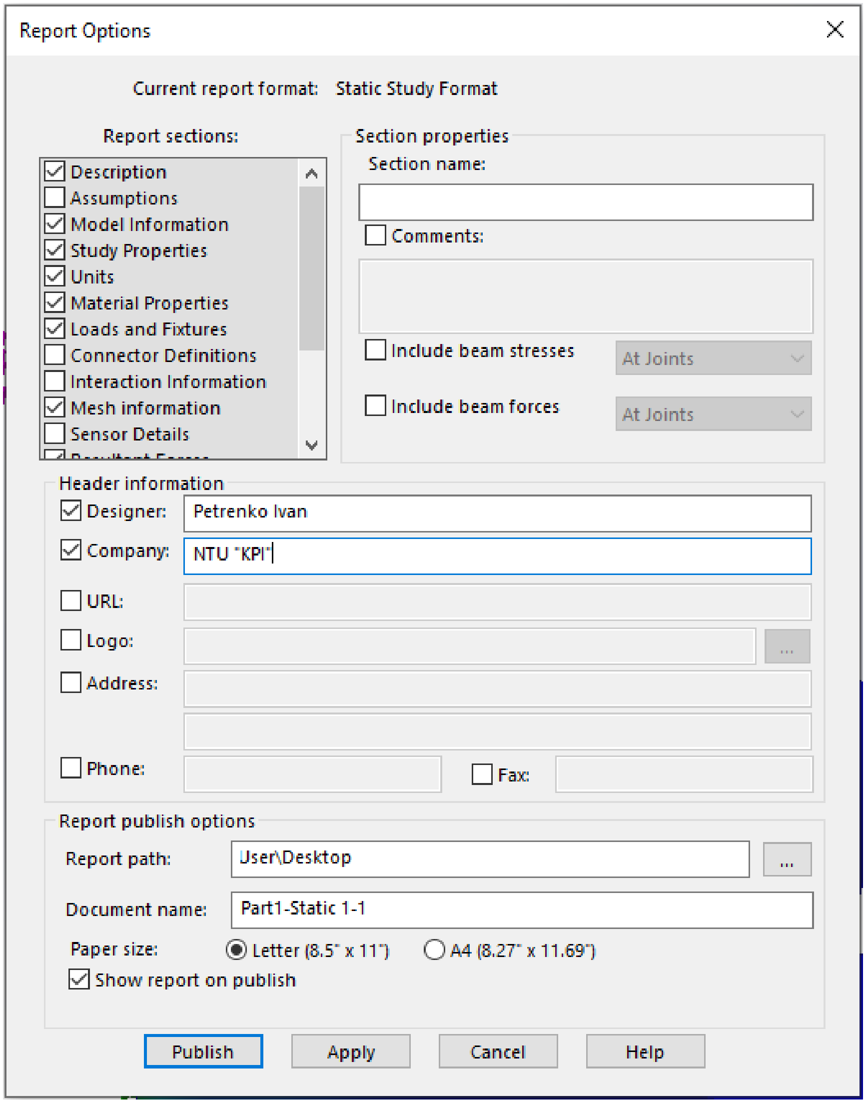


Figure 24 – Configuring report content

3. **Format selection and publication:**
    - Select the desired report format (usually Microsoft Word Document *. docx).
    - Specify the save path and file name for the report.
    - Click the Publish button. SolidWorks Simulation will automatically collect the data and generate a report document (Figure 25).
4. **Using the generated report:**
    - The automatically generated report contains structured information and basic results plots (Fig. 26).
    - It can be used as the basis for the final report required under Section 6 of these guidelines.

Simulation of Part1

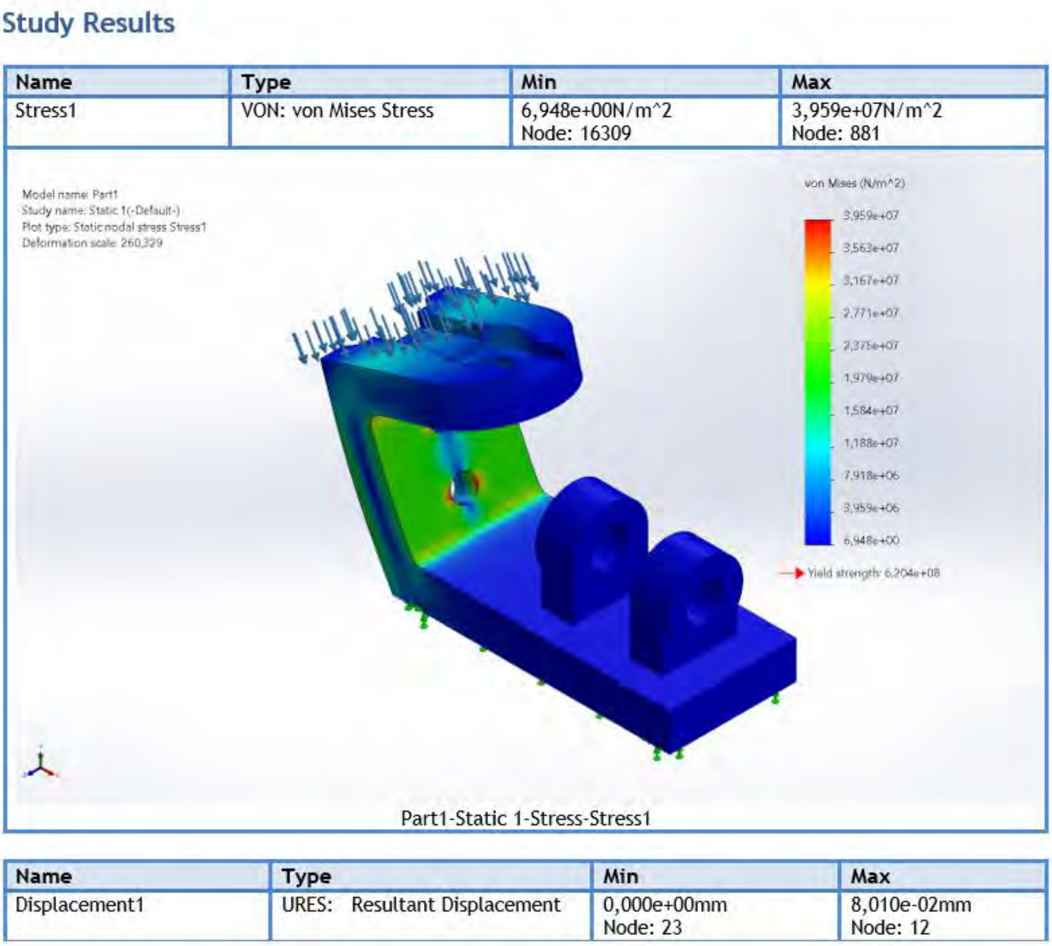
Date: 5 квітня 2025 р.
Designer: Petrenko Ivan
Study name: Static 1
Analysis type: Static

Table of Contents

Description

Strength calculation of a part

Figure 25 – First page of the report (Report)



Study Results

Name	Type	Min	Max
Stress1	VON: von Mises Stress	6,948e+00N/m^2 Node: 16309	3,959e+07N/m^2 Node: 881

Part1-Static 1-Stress-Stress1

Name	Type	Min	Max
Displacement1	URES: Resultant Displacement	0,000e+00mm Node: 23	8,010e-02mm Node: 12

Figure 26 – Example of presenting results in a generated report

Important: An automated report typically requires editing, formatting, and adding your own detailed analysis of the results, conclusions, and other information required by the task and the "Report Requirements" section. It does not replace the need for independent analysis and interpretation of the data received.

3 Description of task options

Each student receives an option number from the teacher.

The task includes:

1. A drawing or sketch of the part with the main dimensions. Missing dimensions (for example, chamfers, roundings, if not specified) can be chosen by the student constructively or in agreement with the teacher.
2. Basic conditions for calculation:
   - Material: If the material is not explicitly stated, select a typical structural material (e.g., Plain Carbon Steel). Carbon Steel) or Alloy steel (Alloy Steel)).
   - Fixing: Determine independently, based on the design of the part and its likely use, or as directed by the instructor (e.g., fix one of the end surfaces, mounting holes, etc.).
   - Load: Determine the type, magnitude, and location of load application as instructed by the instructor or based on an analysis of possible operating conditions of the part (for example, applying force to a specific surface, pressure on an internal cavity, etc.).

The student must:

1. Create a three-dimensional model of the part of your variant in SolidWorks.
2. Perform a static strength calculation in SolidWorks Simulation, specifying the selected material, specified fastening conditions, and loading.
3. Analyze the results (stress, displacement, CBM).

4 Report requirements

The report on the performance of laboratory work is drawn up on A4 sheets and must contain:

1. Title page (according to the established form).
2. The purpose of the work and the individual task (variant number, image (sketch) of the part).
3. A brief description of the sequence of creating a 3D model of a part (main tools and operations used).
4. Description of study settings in SolidWorks Simulation:
    - Study type (Static).
    - Selected material and its main properties (yield strength).
    - Schematic representation of the model with indicated anchoring zones (boundary conditions).
    - Schematic representation of the model with indicated applied loads (type, magnitude, direction, location of application).
    - Image of a finite element mesh (general view).
5. Calculation results in the form of screenshots of diagrams (graphs) with corresponding scales:
    - Mises equivalent stress diagram (with maximum value and yield stress indicated).
    - Plot of total displacements (with maximum value indicated).
    - Safety factor diagram (with minimum value indicated).
6. Analysis of results:
    - Estimation of maximum stresses (comparison with yield strength).
    - Assessment of maximum displacements (whether they are permissible).
    - Estimate the minimum safety factor (comparison with the permissible value, for example, [KZM] = 1.5 or other as indicated).
7. Conclusions regarding the strength and stiffness of the part under given conditions. Does the design meet the strength requirements? If not, possible ways to improve it.

Check questions

1. What are CAE systems and what are their advantages when used together with CAD?
2. What is the essence of the finite element method (FEM)?
3. What are the main stages of the strength calculation process in SolidWorks? Simulation (preprocessing, solution, postprocessing)?
4. How do I activate the SolidWorks Simulation module?
5. How do I create a new static analysis study?
6. How do I assign a material to a part in Simulation and what material properties are key for static strength analysis?
7. What are boundary conditions (fixes) and how to specify them? Give examples of types of fixes.
8. What types of loads can be specified in static analysis?
9. What is a finite element mesh and why is it needed? How to create a mesh?
10. How to run a strength calculation?
11. What are the main results of static strength analysis?
12. What does the von Mises stress diagram show? How to interpret it?
13. What does the displacement diagram show?
14. What is a safety factor (SFF)? How to calculate it and create a corresponding graph?
15. What minimum value of the CBM is usually considered acceptable to ensure structural strength?
16. What to do if the calculation shows that the maximum stresses exceed the yield strength or the CSM is insufficient?

LIST OF INFORMATION SOURCES

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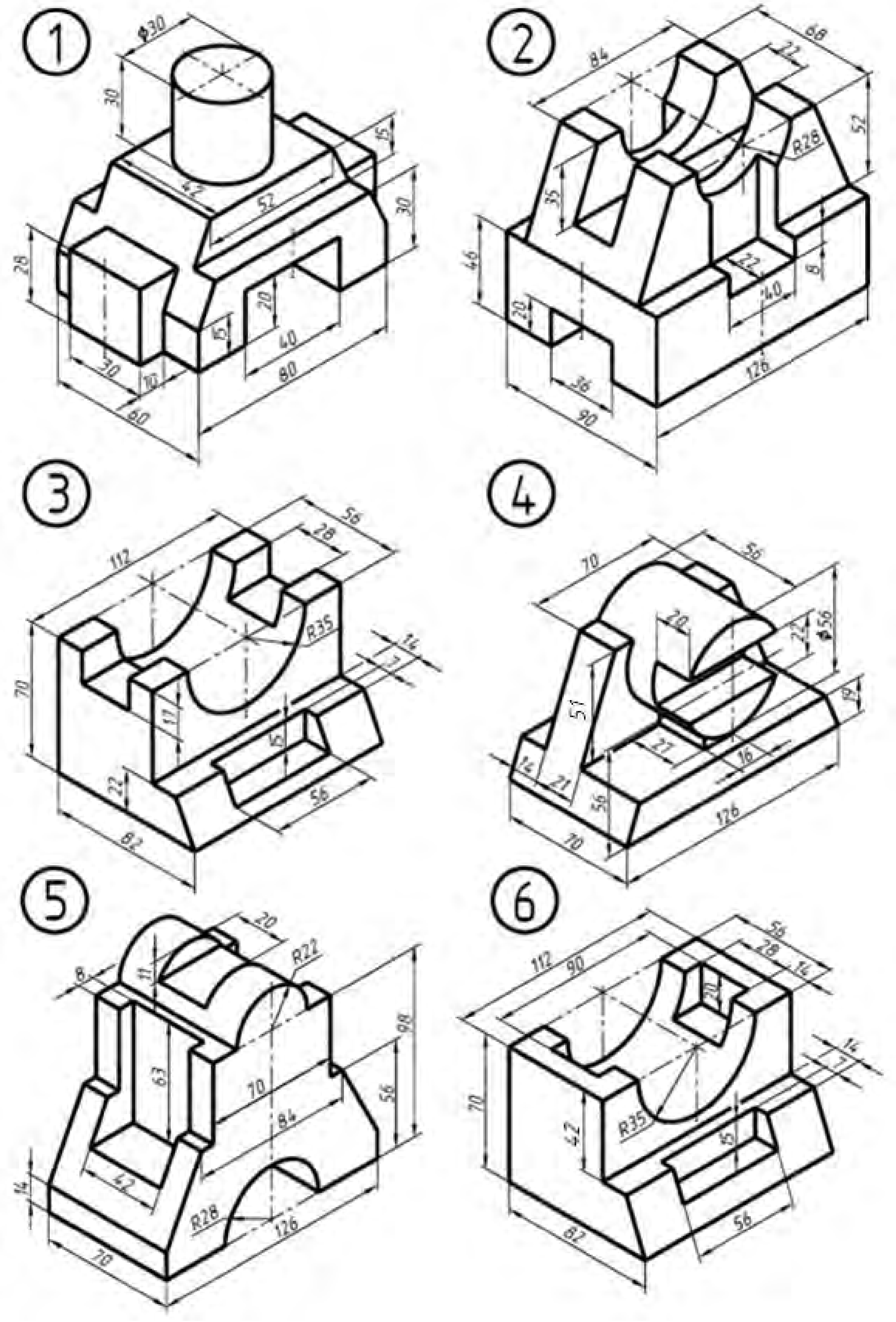
Laboratory Work Assignments

Assignments for Laboratory Work No. 1



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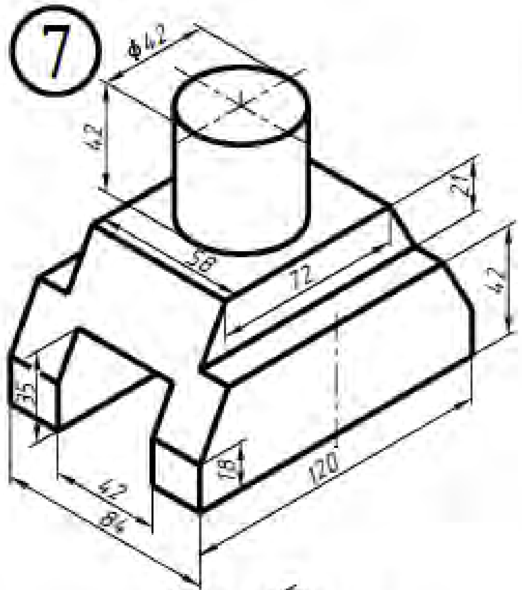
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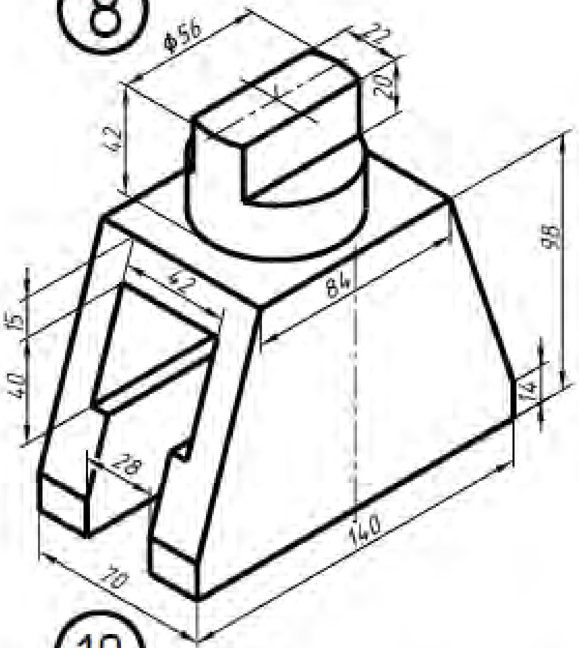
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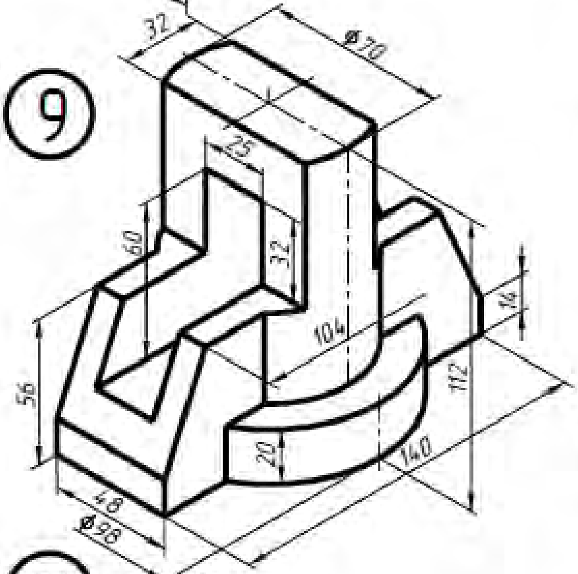
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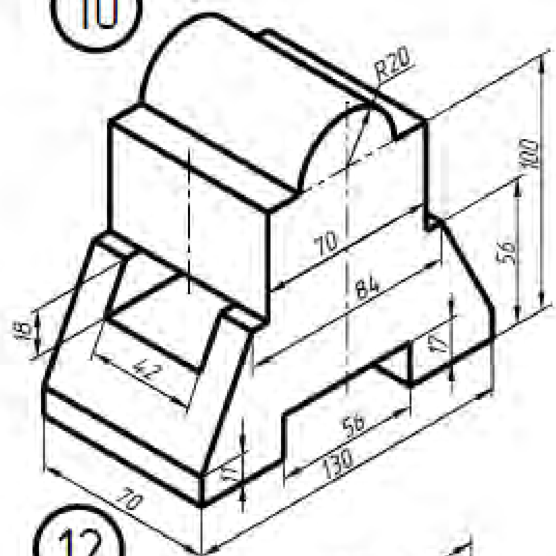
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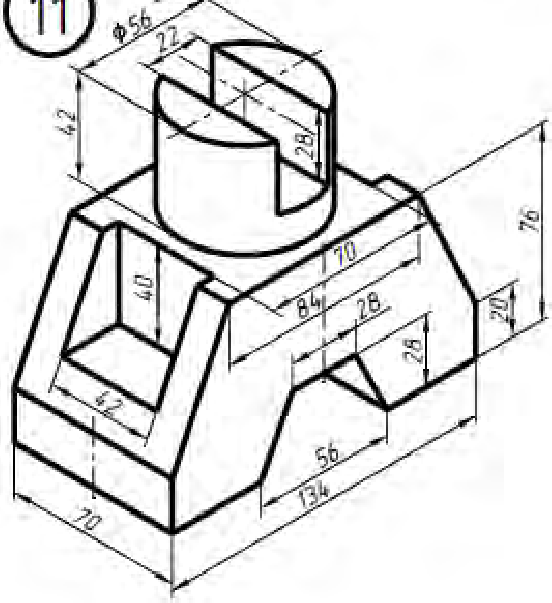
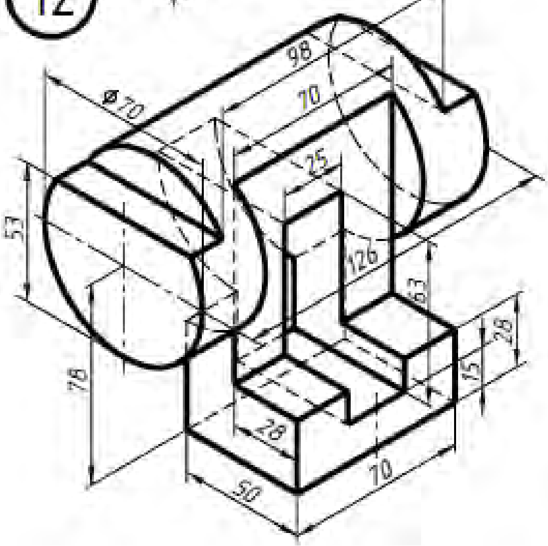
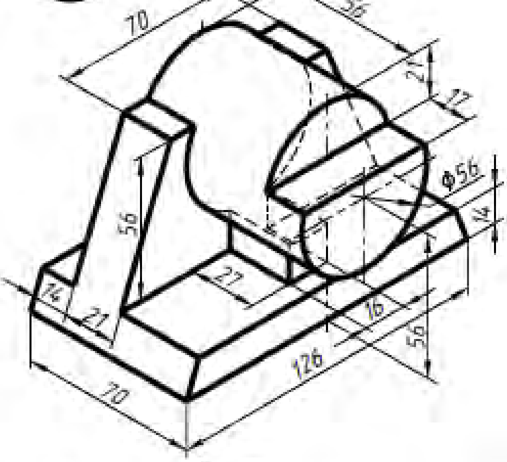

ϕ56
22
42
28
76
40
70
84
28
20
28
42
56
70
134

12


98
ϕ70
70
25
53
126
63
28
15
78
28
50
70

13



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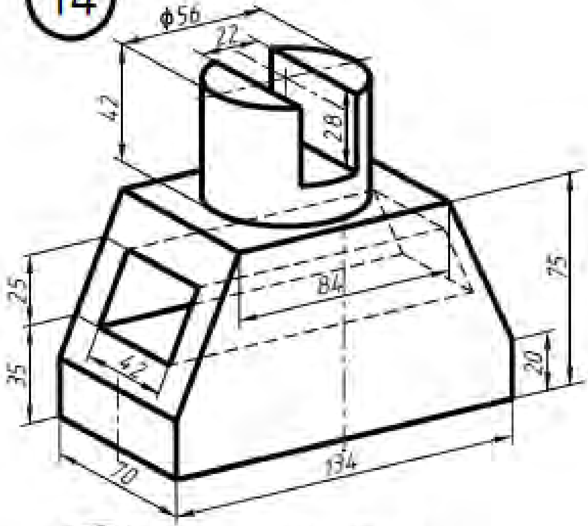

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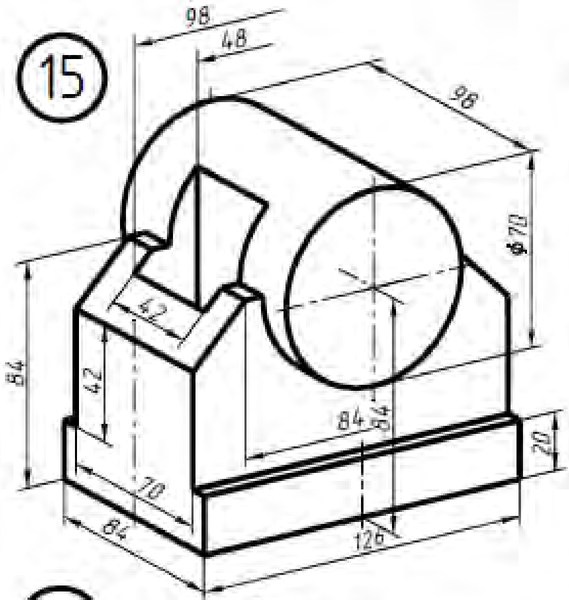

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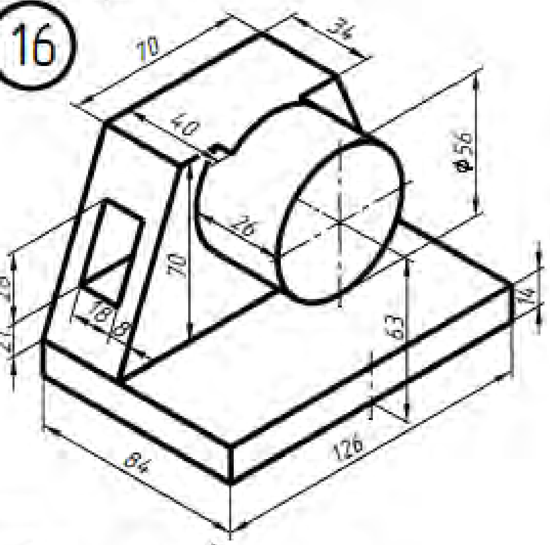
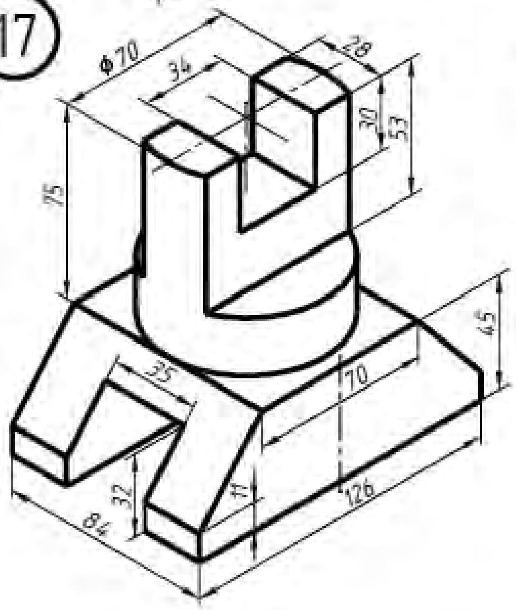


17



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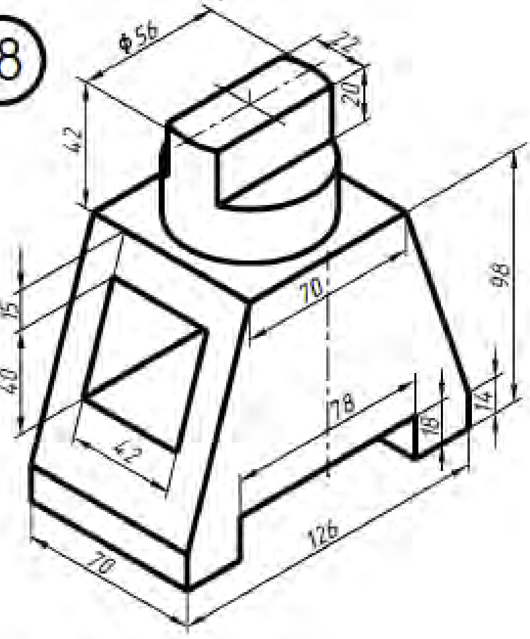

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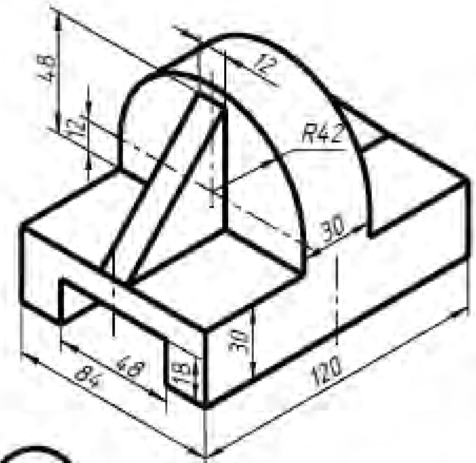

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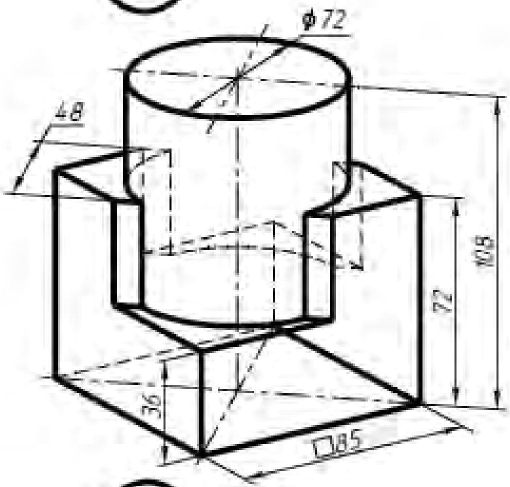

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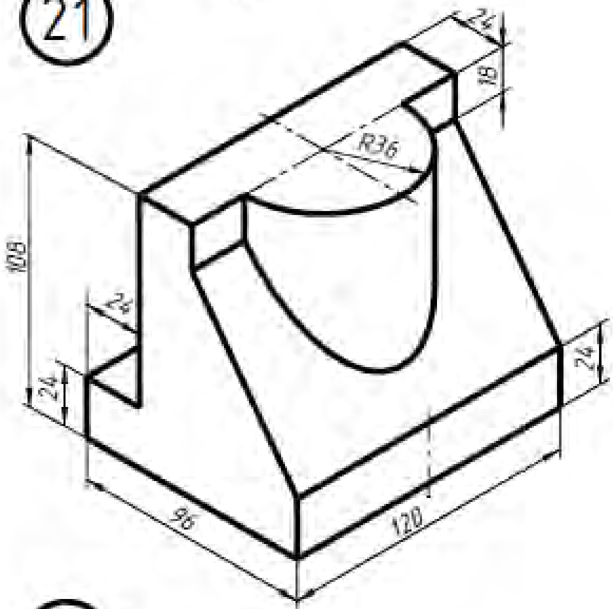

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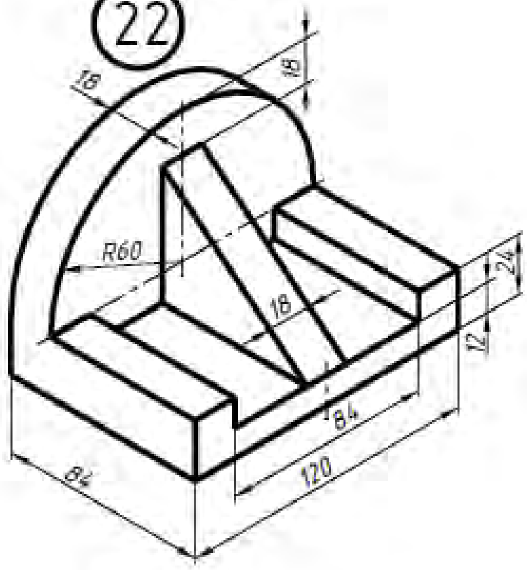
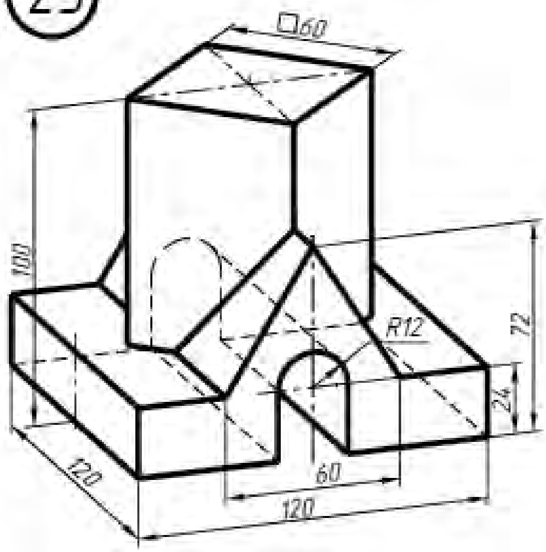


23



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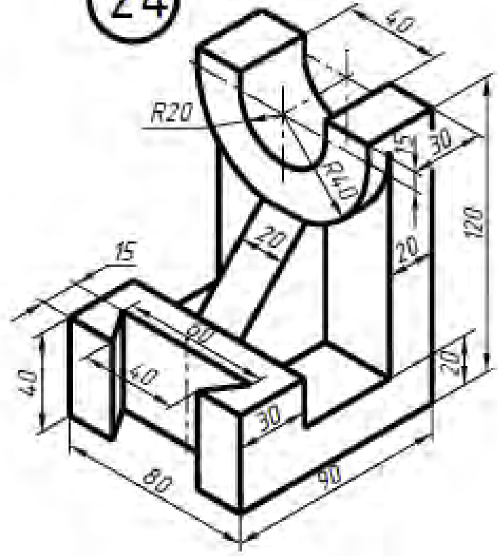

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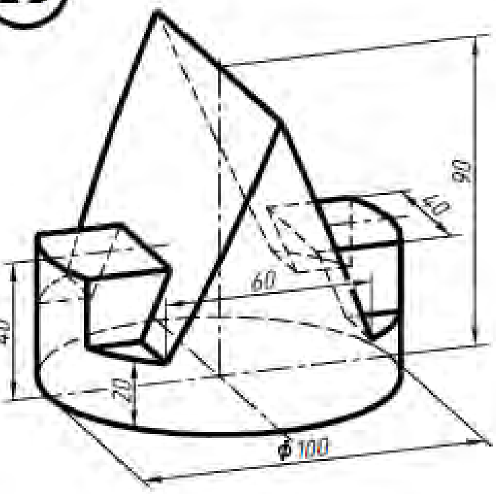

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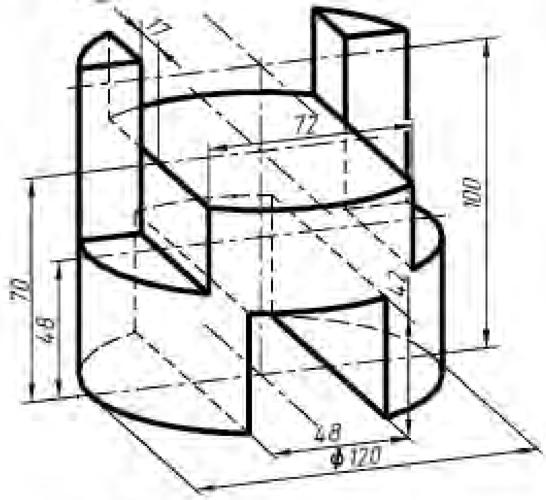

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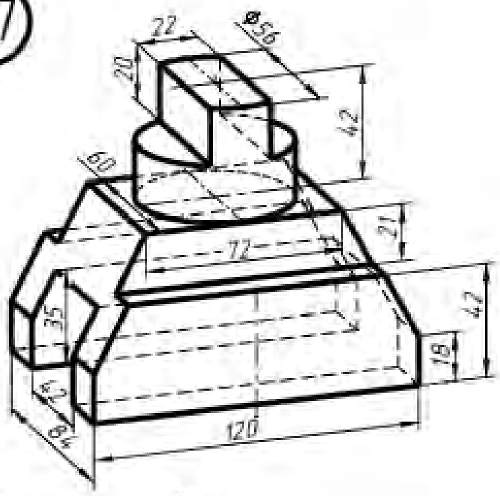

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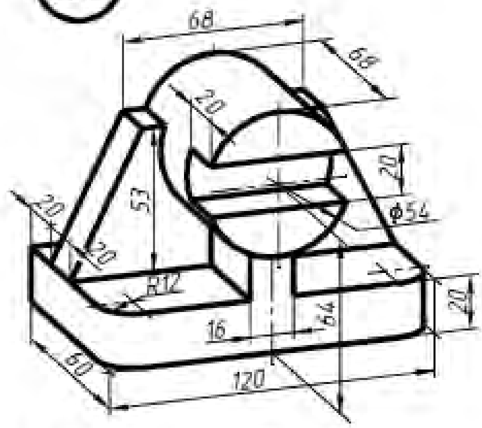

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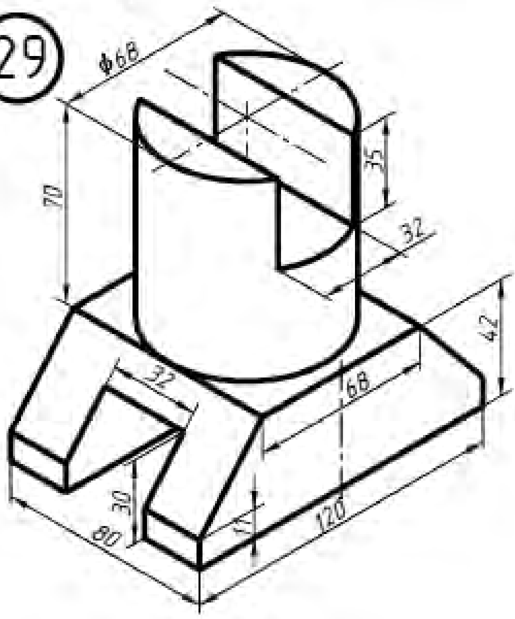
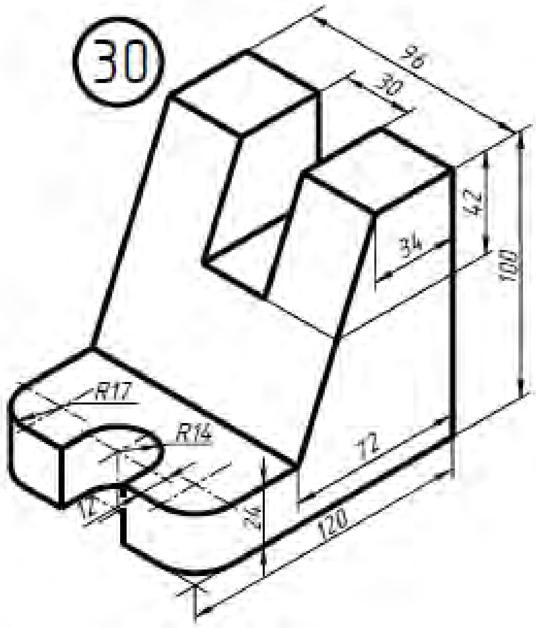
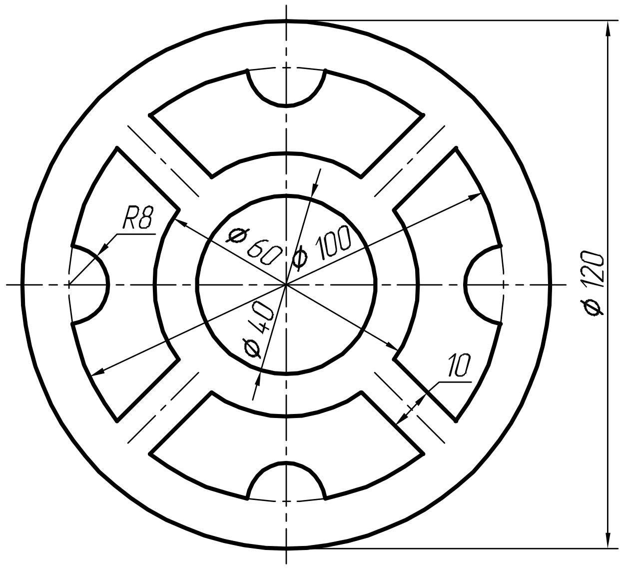


30

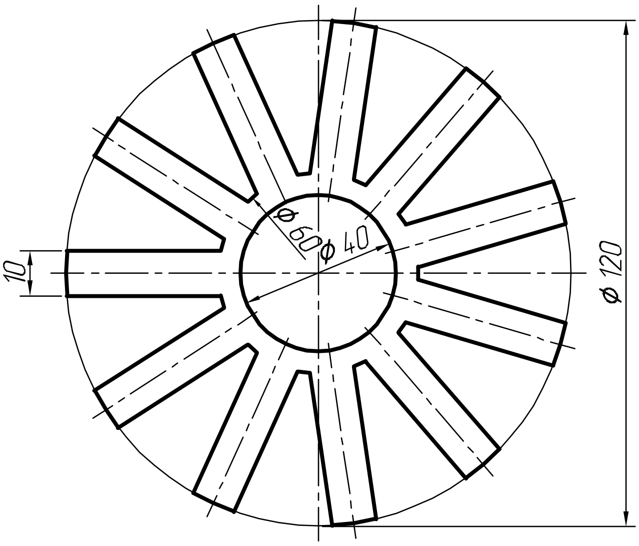


Assignments for Laboratory Work No. 2

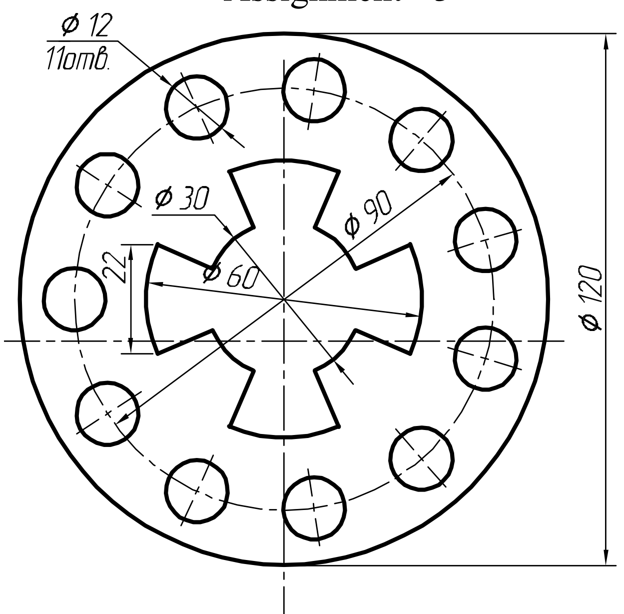
Assignment - 1



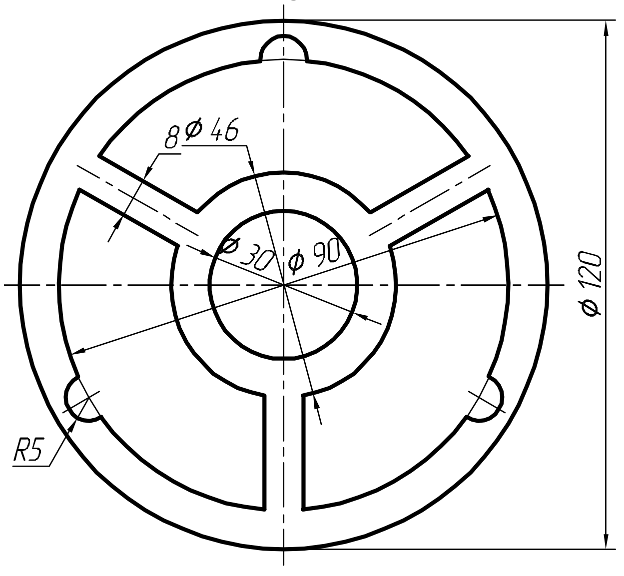
Assignment - 2



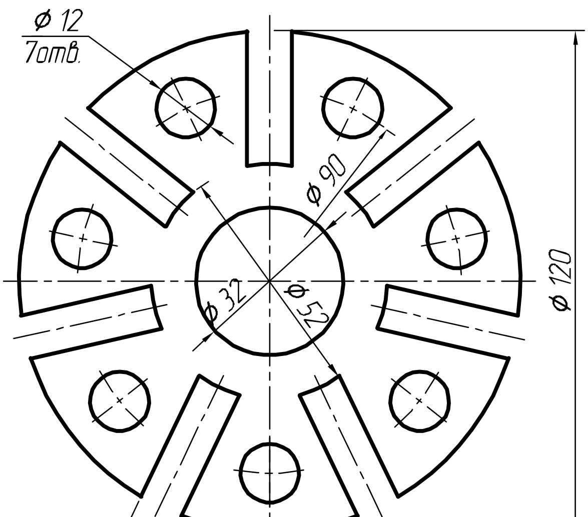
Assignment - 3



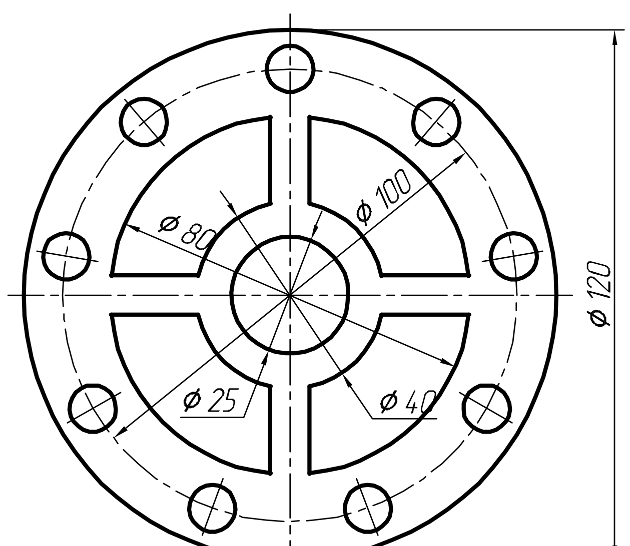
Assignment - 4



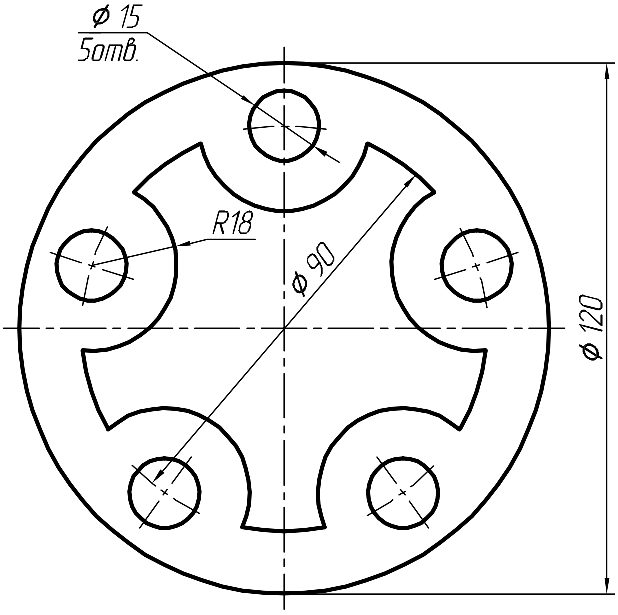
Assignment - 5



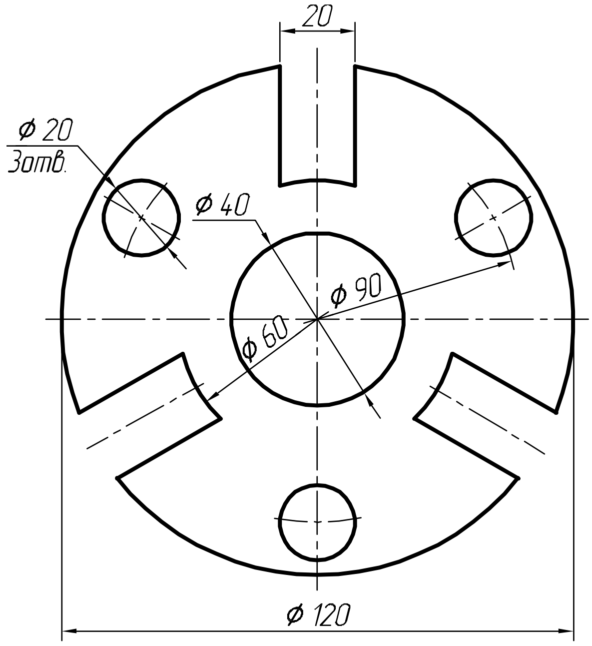
Assignment - 6



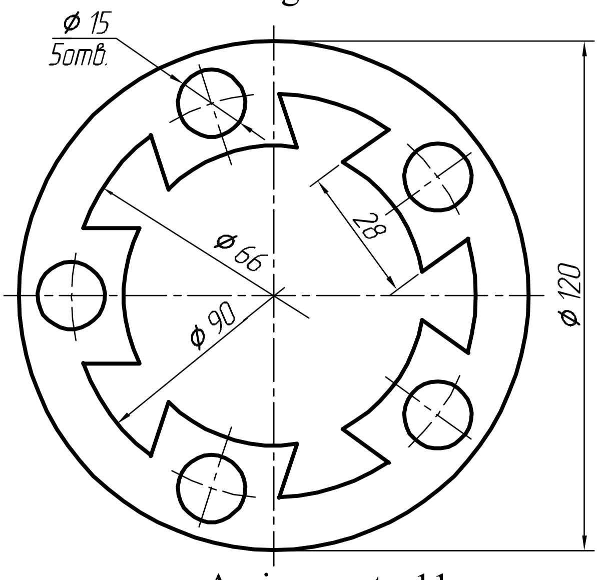
Assignment - 7



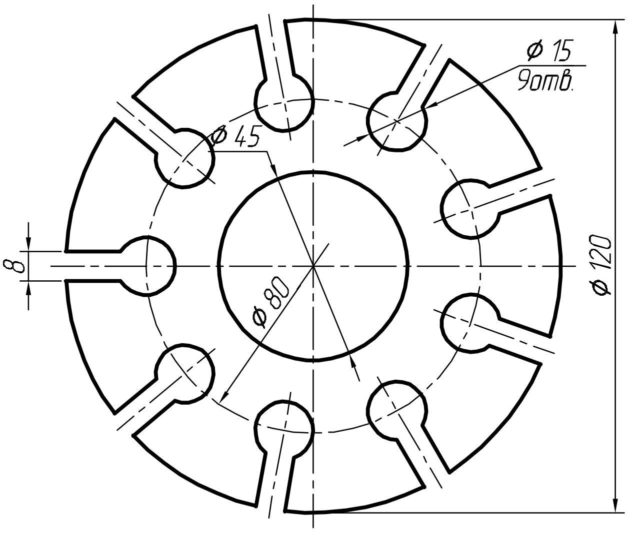
Assignment - 8



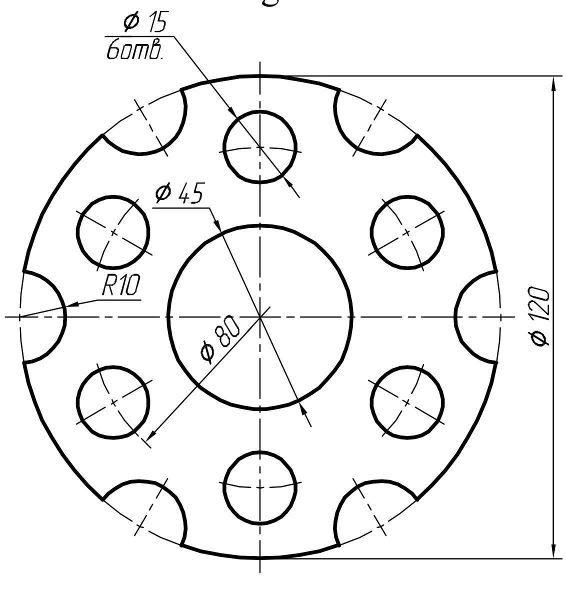
Assignment - 9



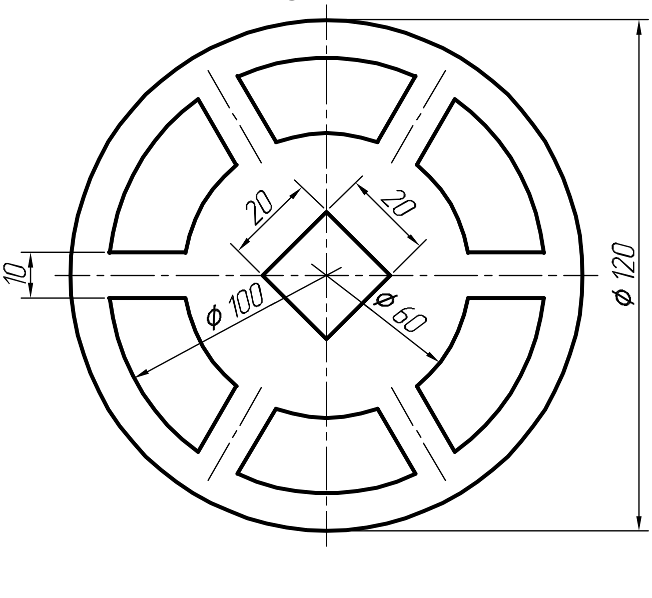
Assignment - 10



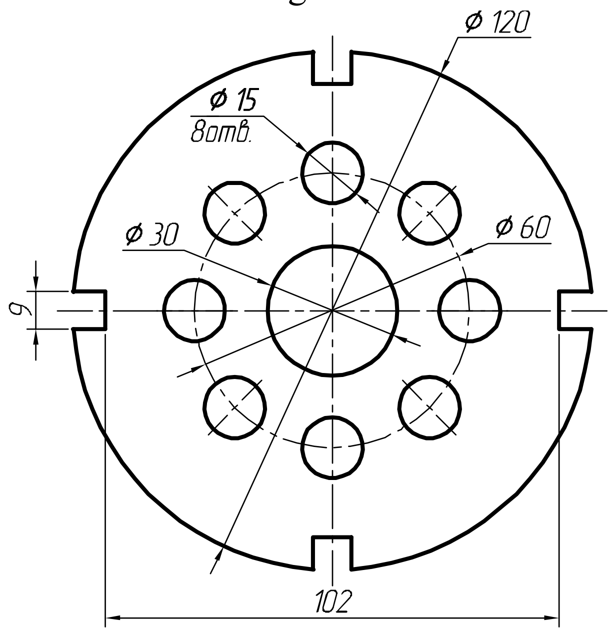
Assignment - 11



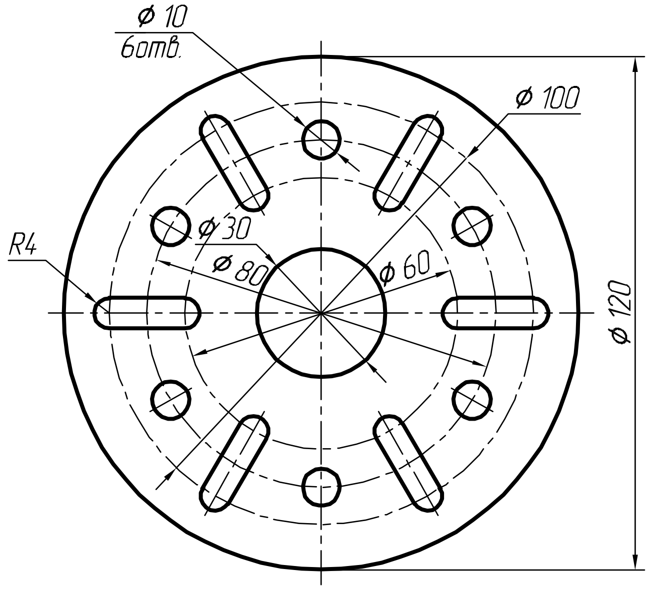
Assignment - 12



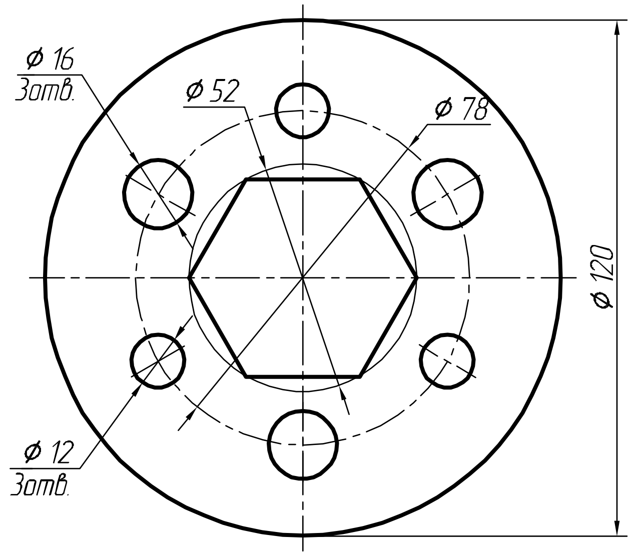
Assignment - 13



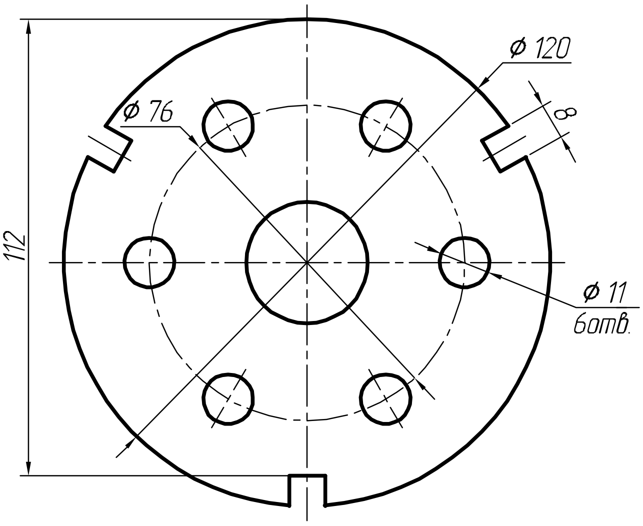
Assignment - 14



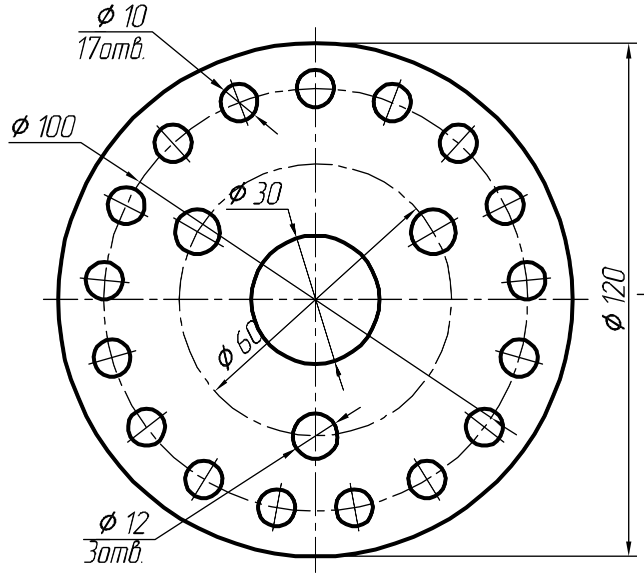
Assignment - 15



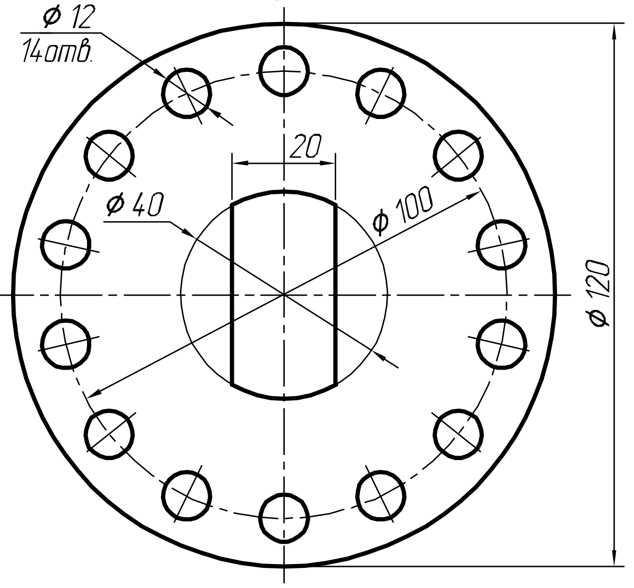
Assignment - 16



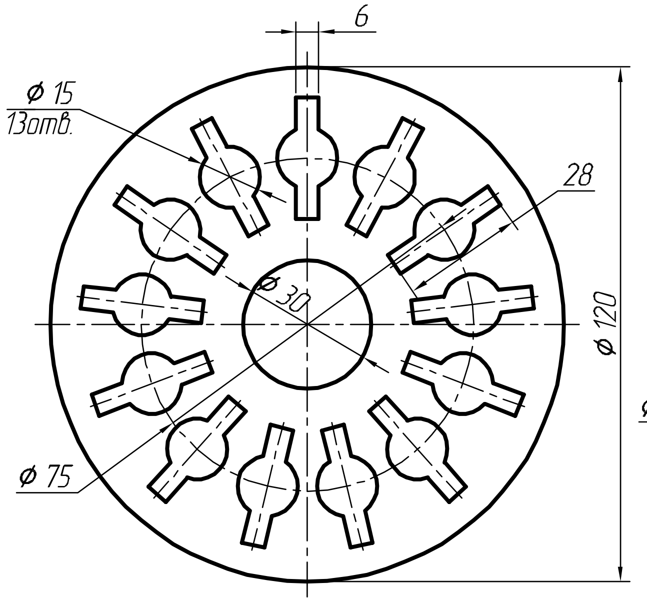
Assignment - 17



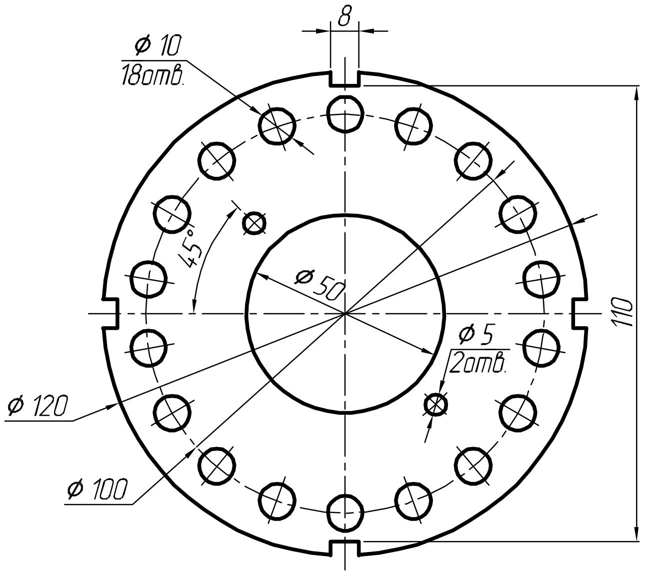
Assignment - 18



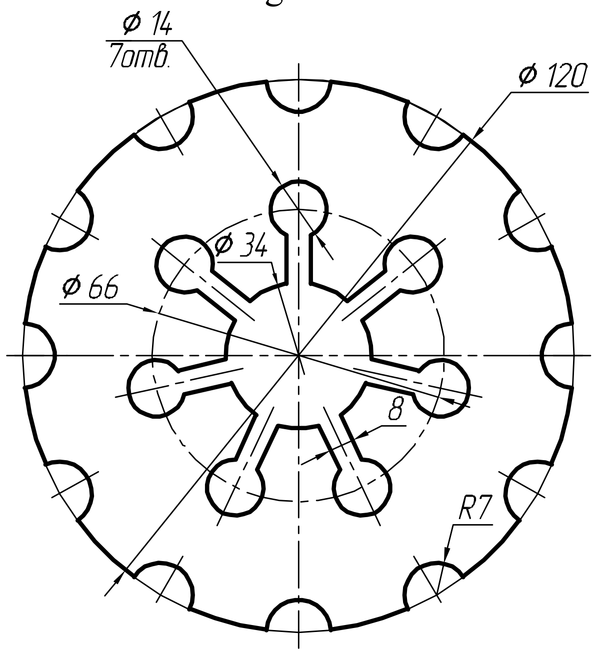
Assignment - 19



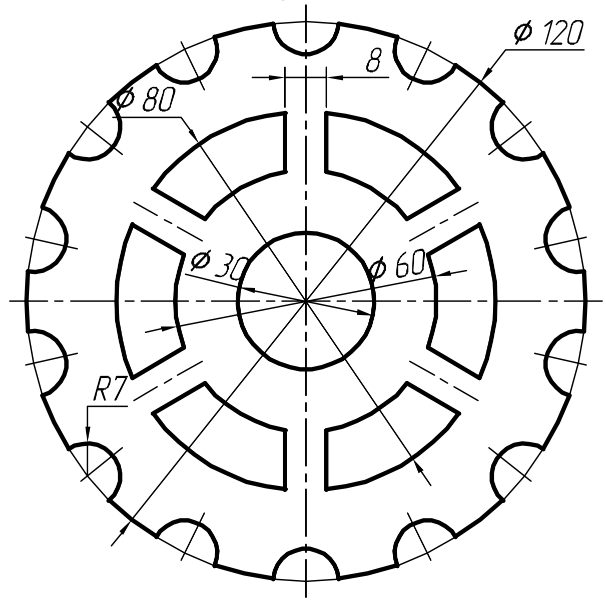
Assignment - 20



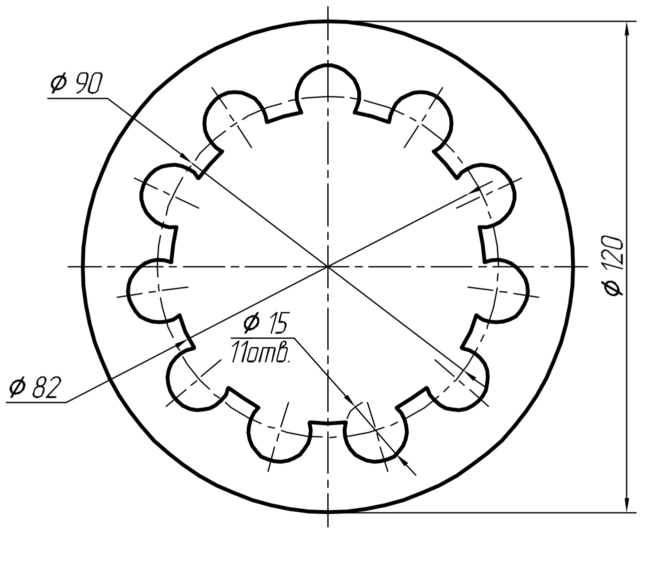
Assignment - 21



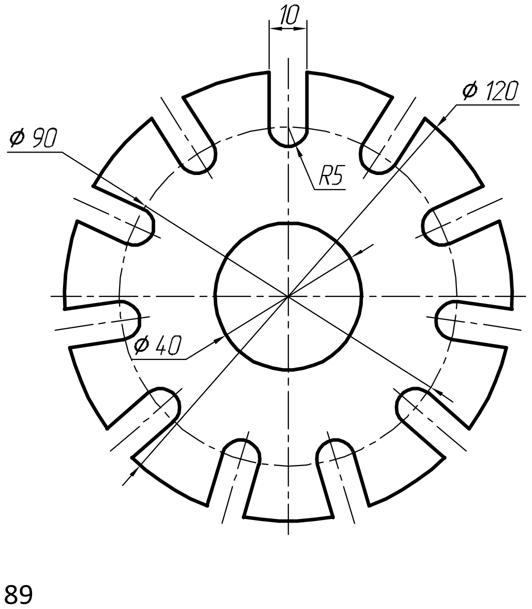
Assignment - 22



Assignment - 23

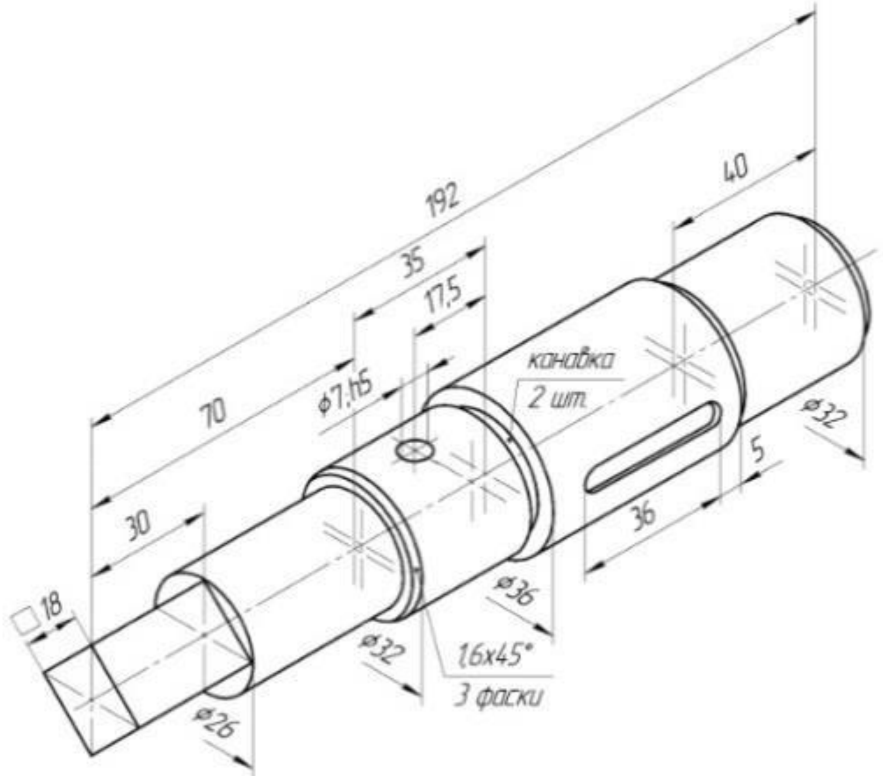


Assignment - 24

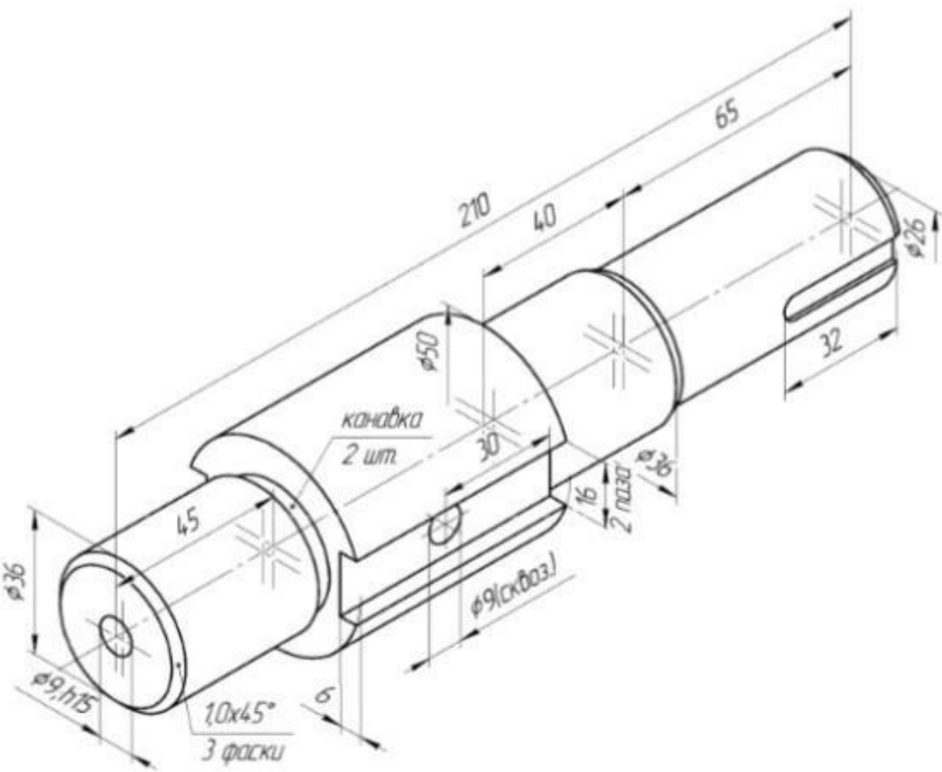


Assignments for Laboratory Work No. 3

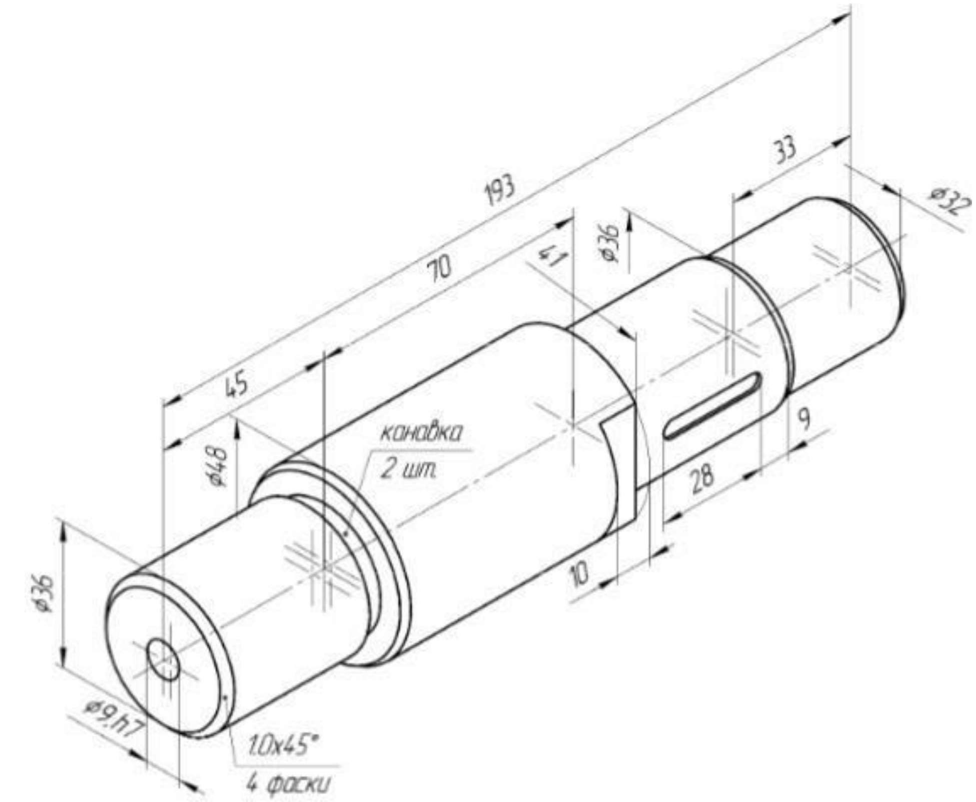
B-1



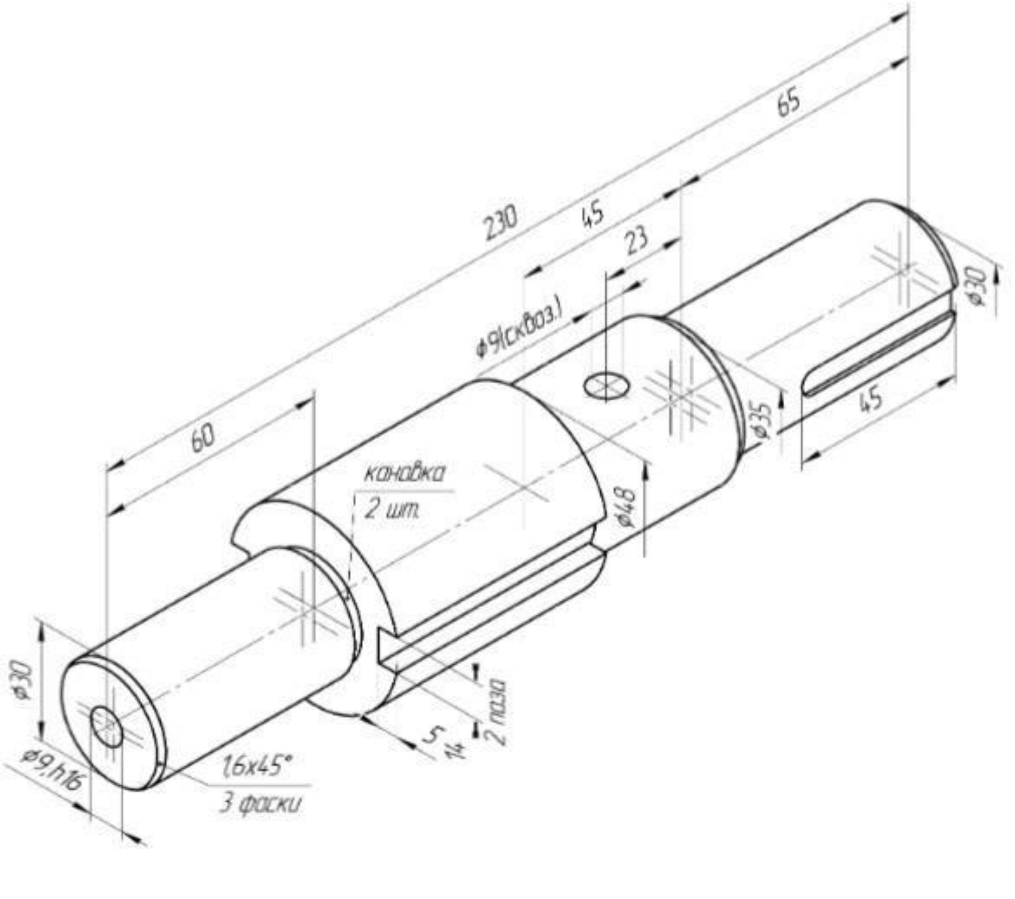
B-2



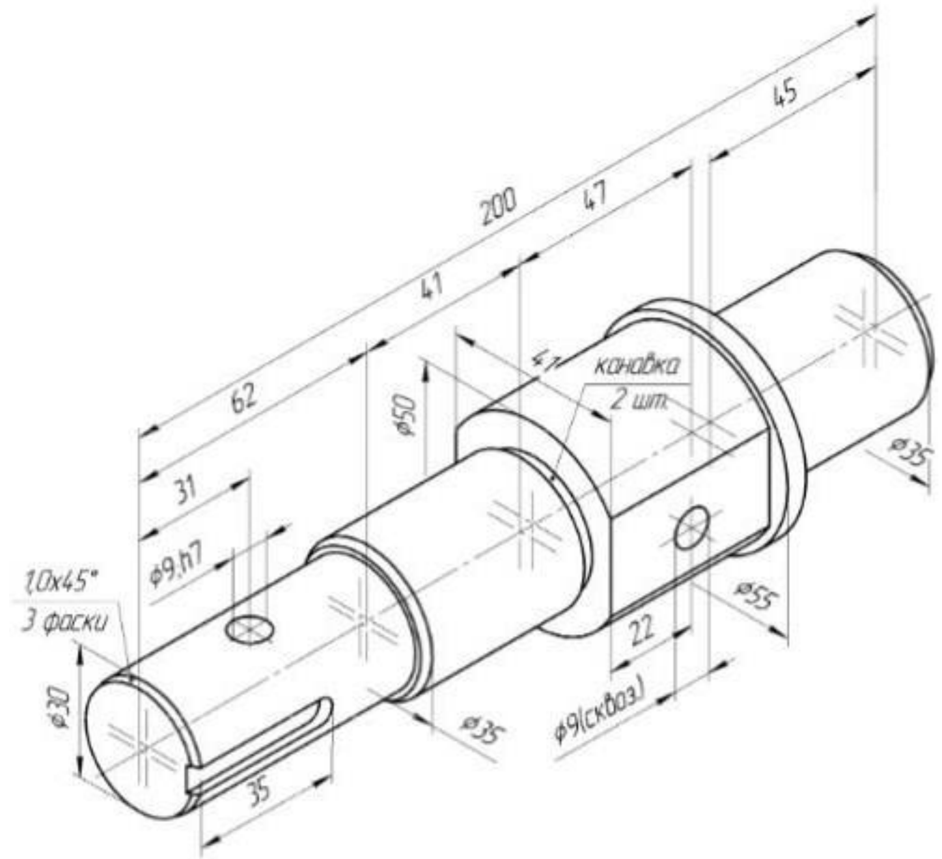
В-3


193
33
ø32
ø36
70
45
канавка
2 шт.
ø48
9
28
10
ø36
ø9,h7
1,0x45°
4 фаски

В-4

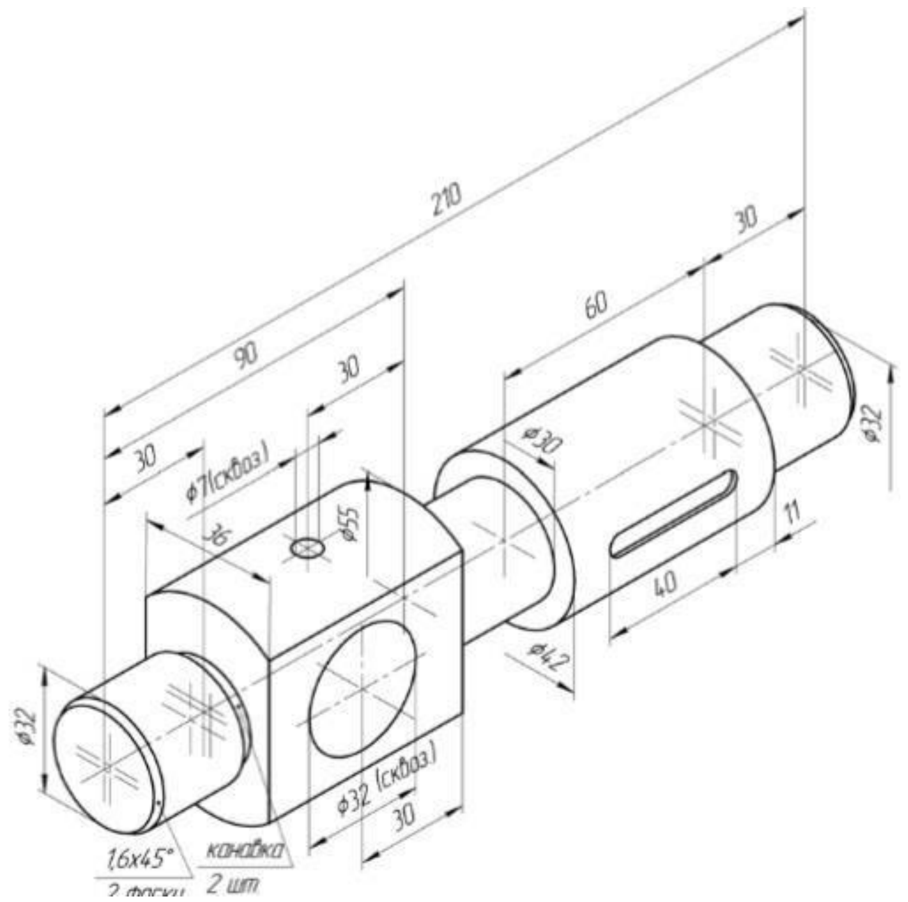

230
65
45
23
ø9(сквоз.)
ø30
45
ø35
60
канавка
2 шт.
ø48
ø30
5
14
2 паза
ø9,h16
1,6x45°
3 фаски

B-5



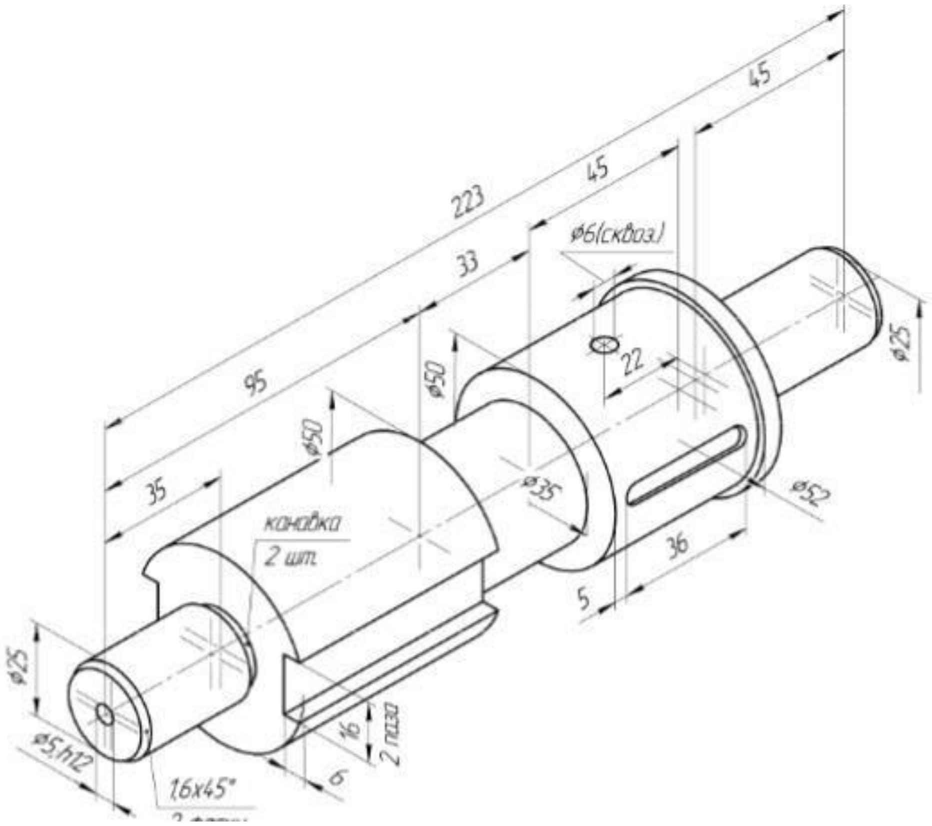
200
45
47
41
62
31
4
канавка
2 шт.
ф50
ф9,h7
1,0х45°
3 фаски
ф30
35
ф35
22
ф9(сквоз)
ф55
ф35

B-6

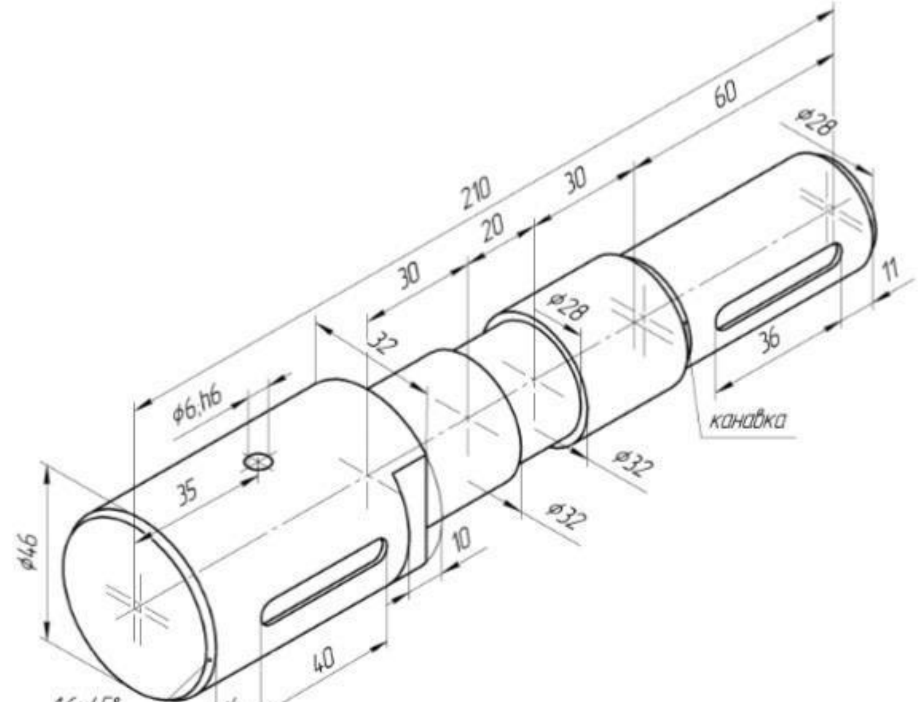


210
30
60
90
30
30
ф7(сквоз)
36
ф55
ф30
ф32
11
40
ф42
ф32
ф32 (сквоз)
30
1,6х45°
2 фаски
канавка
2 шт

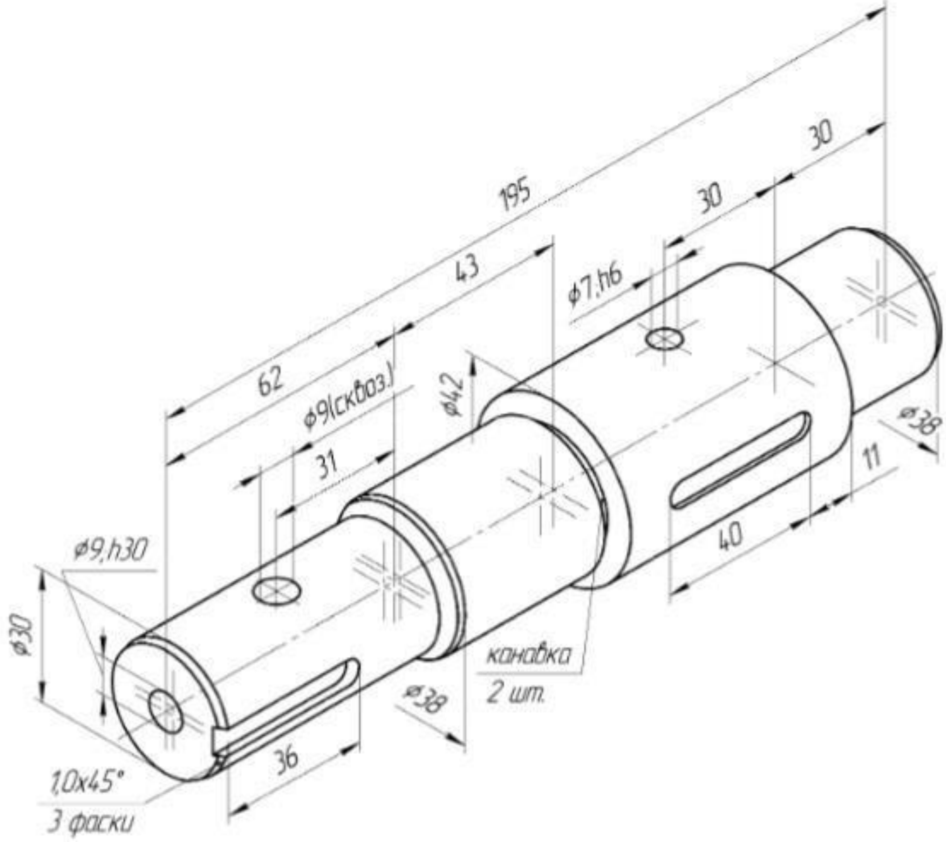
В-7


223
45
45
33
95
35
ø6(сквоз.)
22
ø50
ø50
ø35
ø52
ø25
36
5
канавка
2 шт.
ø25
ø5,h12
1,6x45°
16
6
2 паза

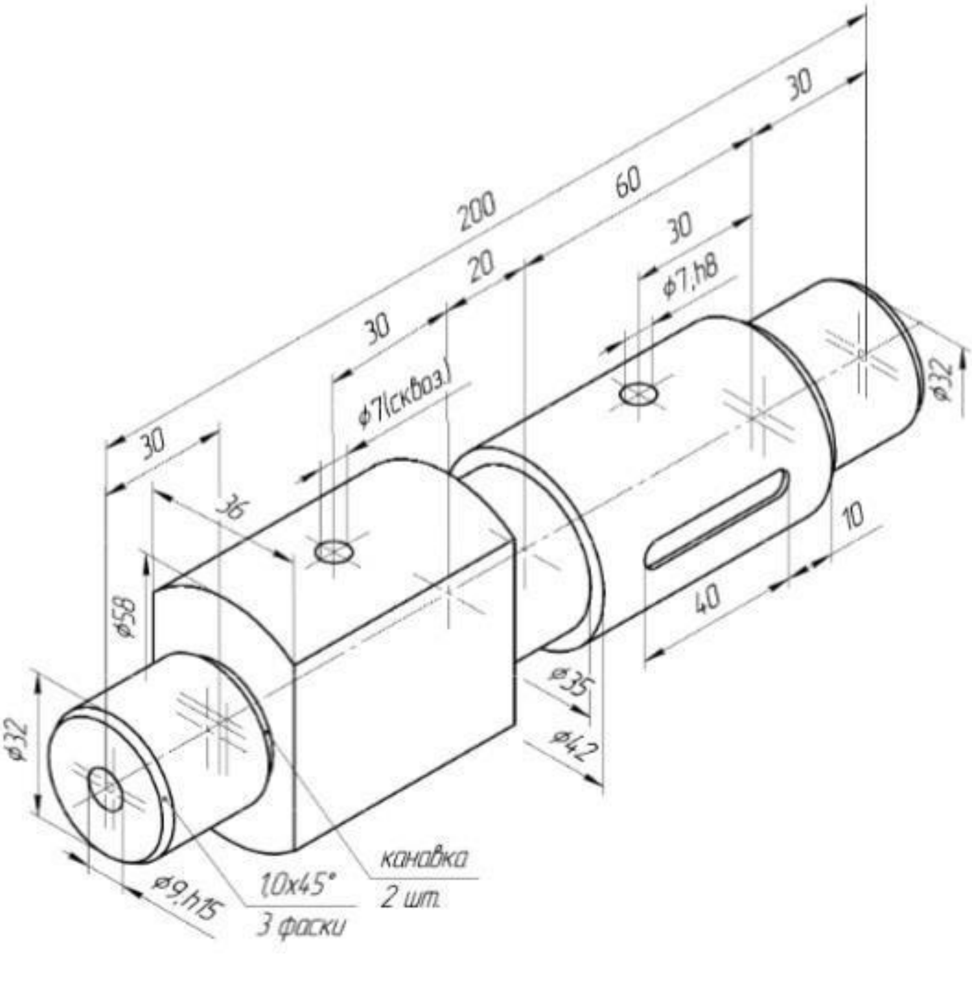
В-8


210
60
30
20
30
ø28
ø28
11
36
канавка
ø32
ø32
32
ø6,h6
35
ø46
10
40
16
1,6x45°
2 фаски

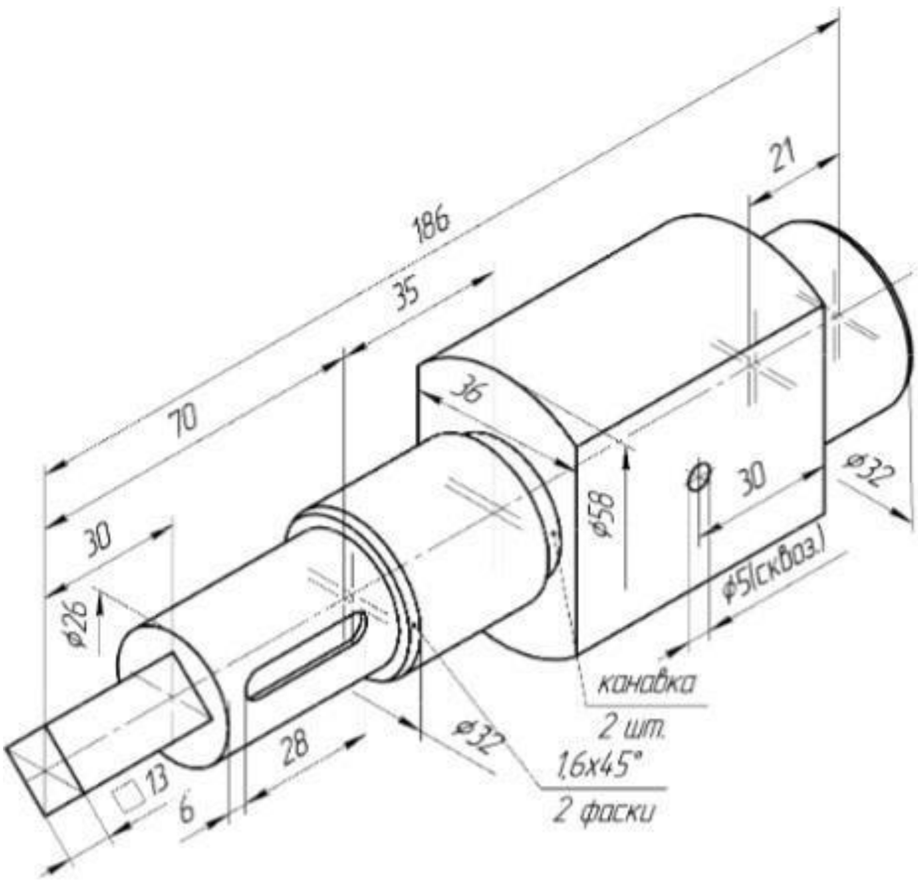
В-9


195
30
30
43
ϕ7,h6
62
ϕ9(скво3.)
ϕ42
31
ϕ38
11
40
ϕ9,h30
ϕ30
канавка
2 шт.
ϕ38
36
1,0x45°
3 фаски

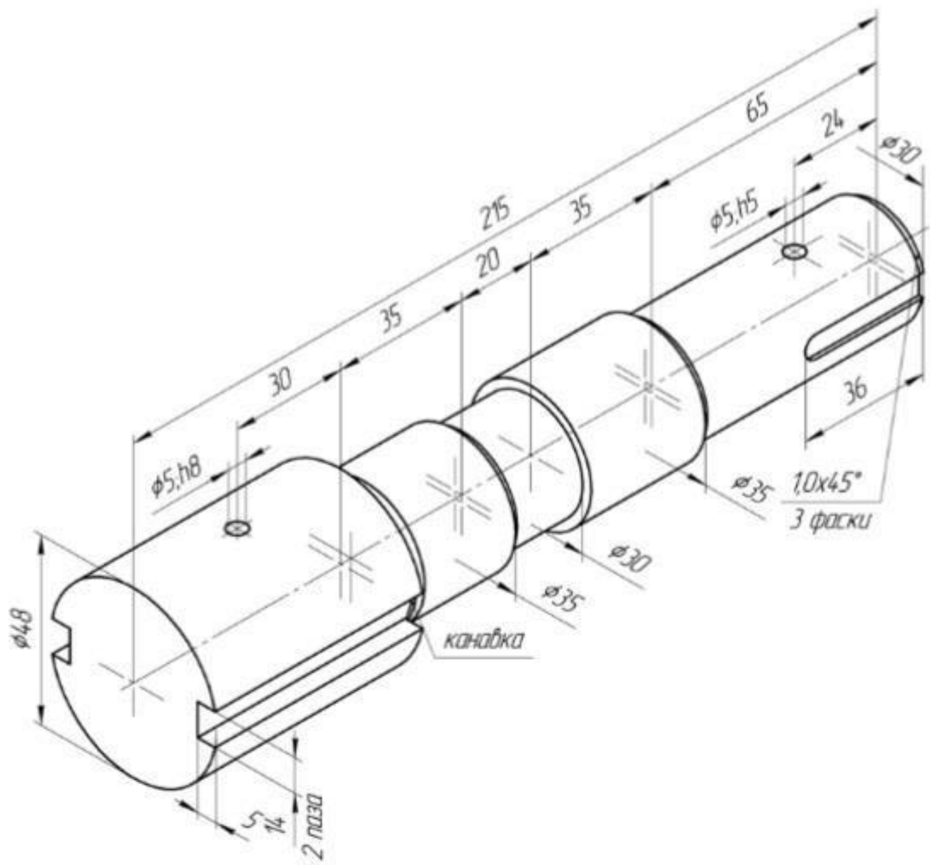
В-10


30
200
60
30
20
ϕ7,h8
30
ϕ7(скво3.)
ϕ32
30
36
10
40
ϕ58
ϕ35
ϕ32
ϕ42
канавка
2 шт.
1,0x45°
3 фаски
ϕ9,h15

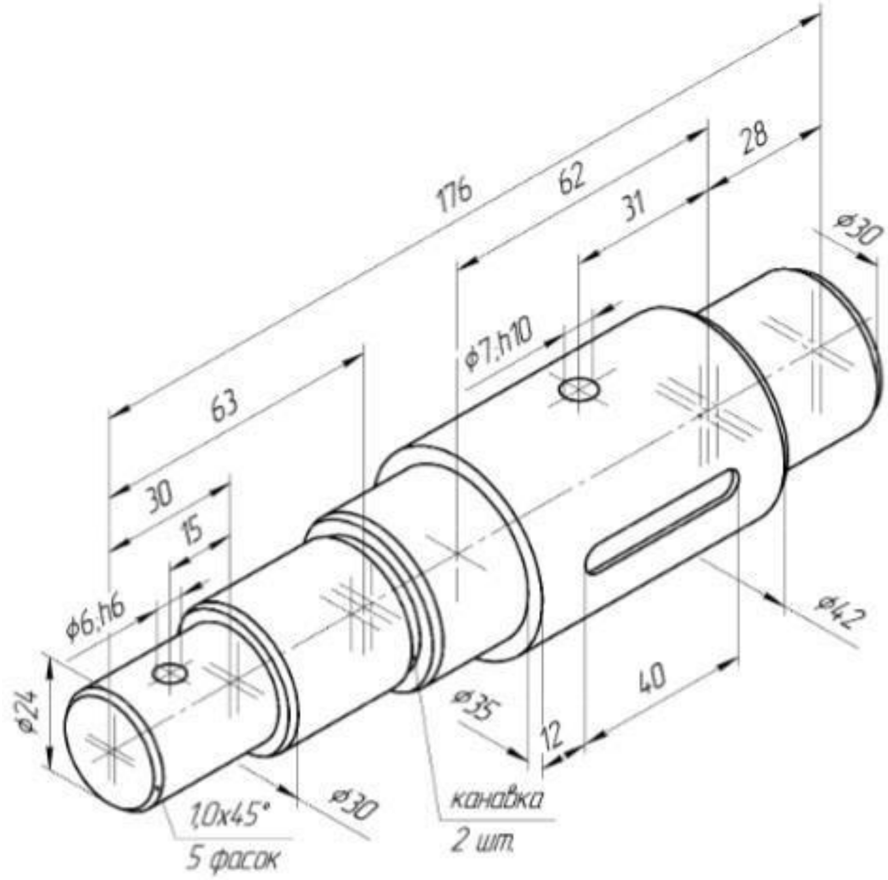
B-11


186
21
35
36
70
30
Ø58
30
Ø32
Ø26
Ø5(скв.оз.)
канавка
2 шт.
Ø32
1,6x45°
2 фаски
28
□13
6

B-12

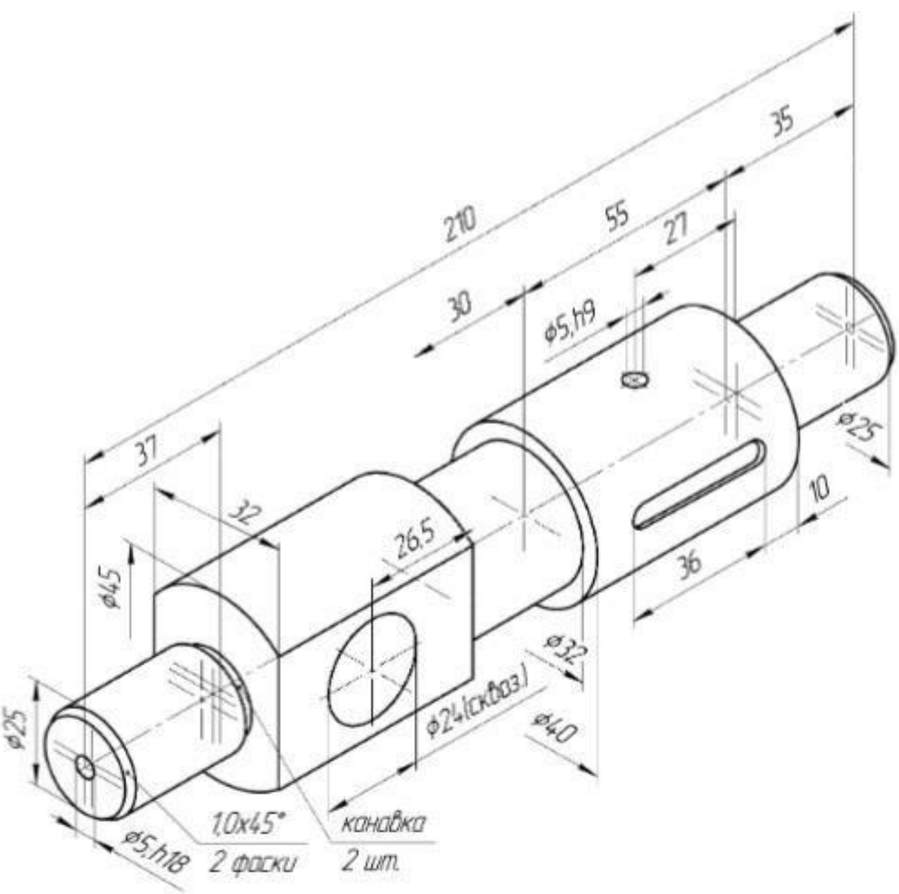

65
24
Ø30
215
35
Ø5,h5
20
35
30
36
Ø5,h8
Ø35
1,0x45°
3 фаски
Ø30
Ø35
Ø48
канавка
5
14
2 паза

B-13



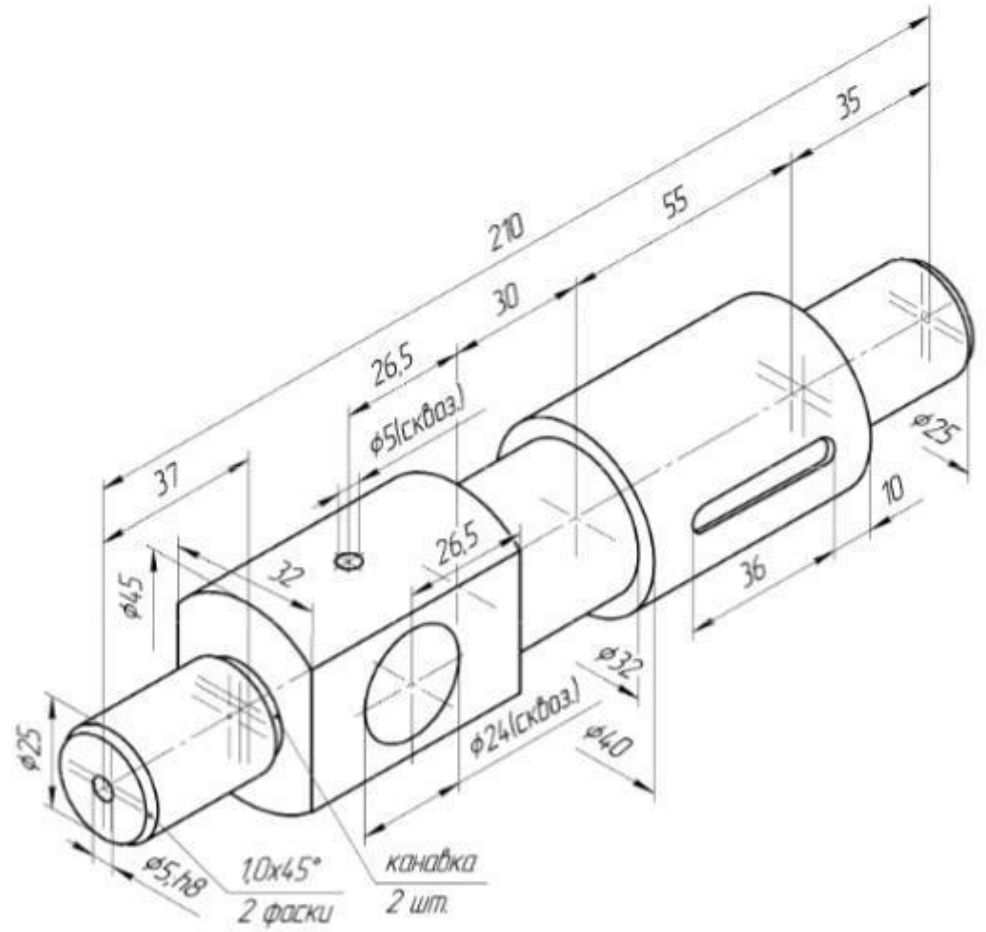
176
62
28
31
ø30
ø7,h10
63
30
15
ø6,h6
ø24
ø42
40
ø35
12
1,0x45°
5 фасок
ø30
канавка
2 шт.

B-14

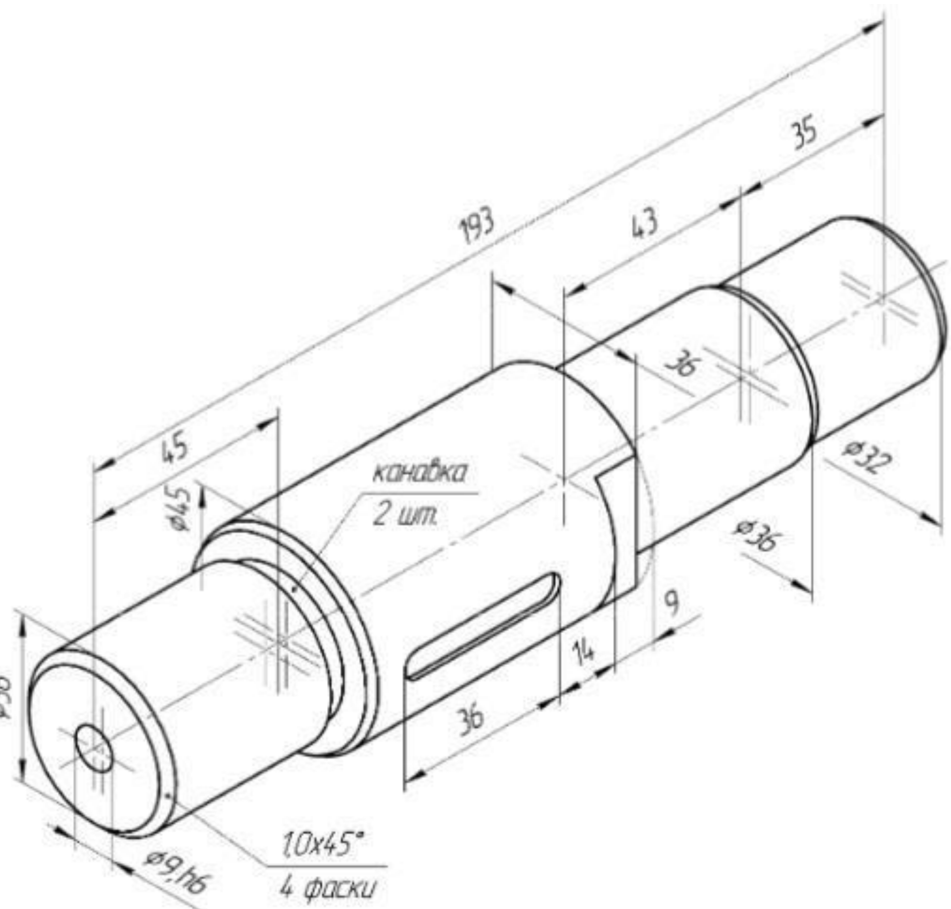


210
35
55
27
30
ø5,h9
37
32
26,5
ø45
ø25
10
36
ø32
ø24(сквоз.)
ø40
ø25
1,0x45°
2 фаски
ø5,h18
канавка
2 шт.

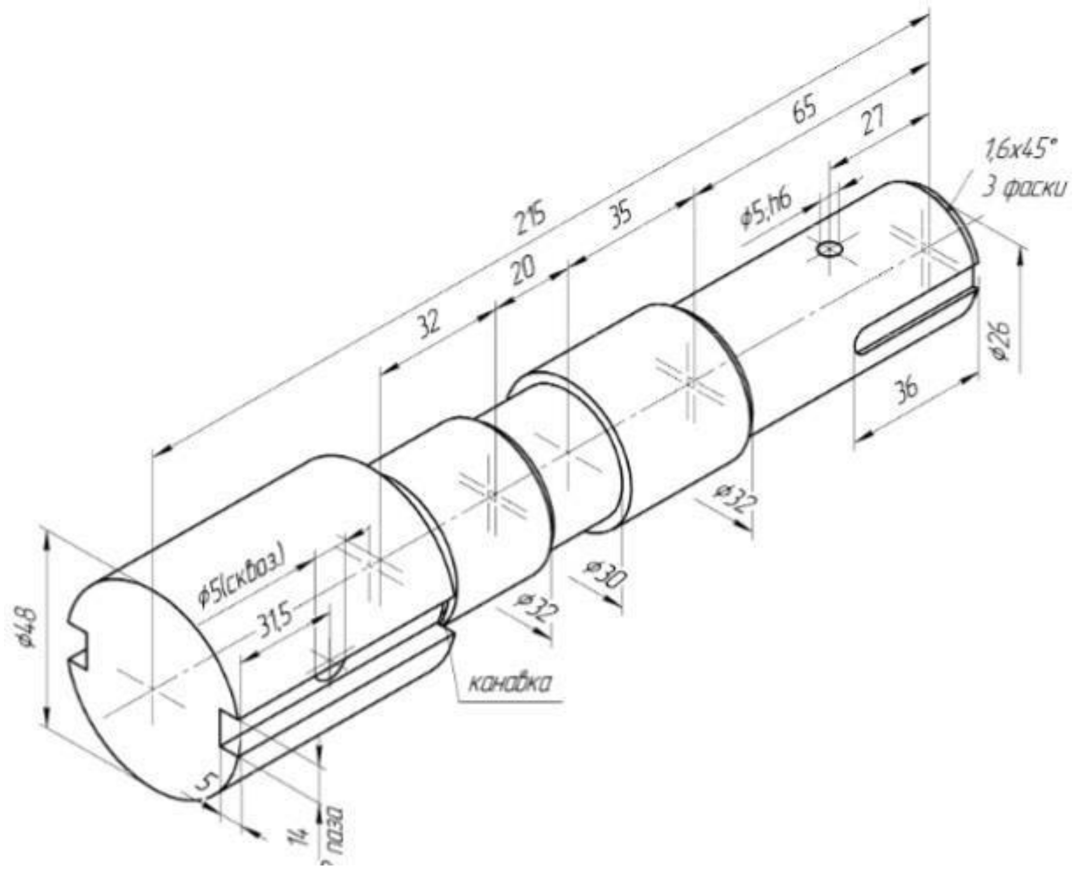
В-15


210
35
55
30
26,5
ϕ5(сквоз.)
37
26,5
32
ϕ45
ϕ25
10
36
ϕ32
ϕ24(сквоз.)
ϕ40
ϕ25
ϕ5,h8
1,0x45°
2 фаски
канавка
2 шт.

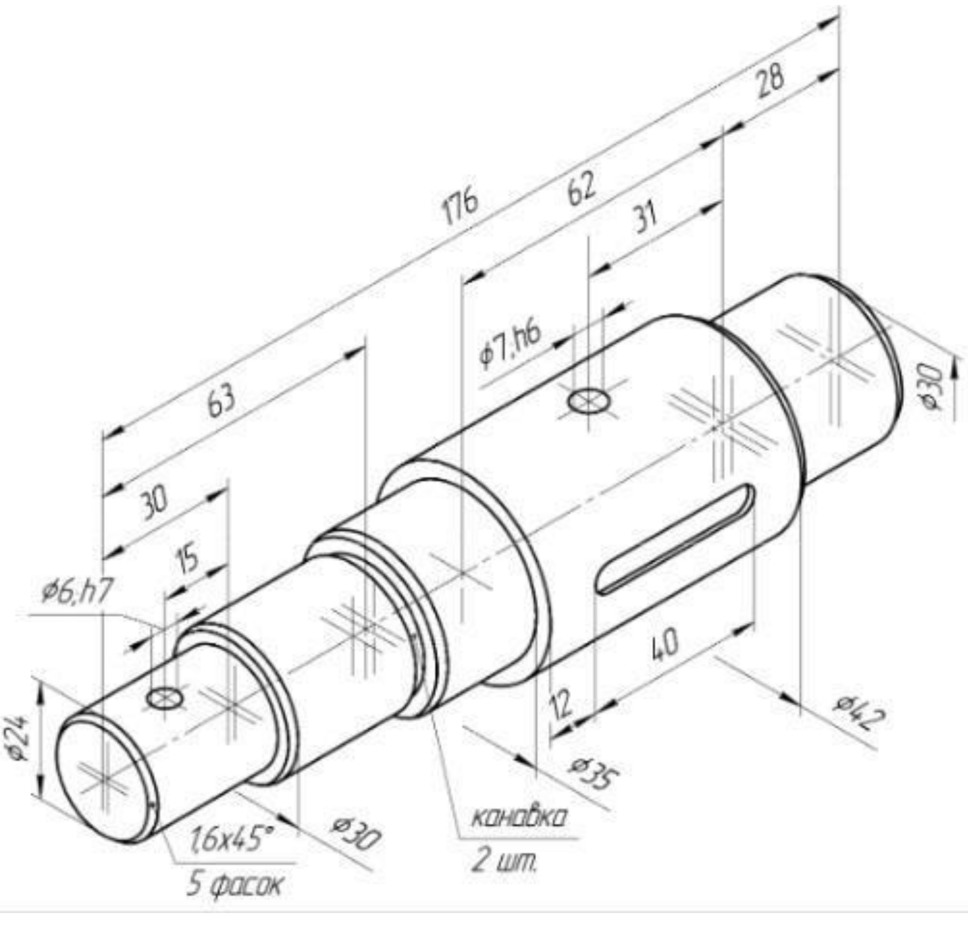
В-16


193
35
43
36
45
ϕ45
канавка
2 шт.
ϕ32
ϕ36
9
14
36
ϕ36
ϕ9,h6
1,0x45°
4 фаски

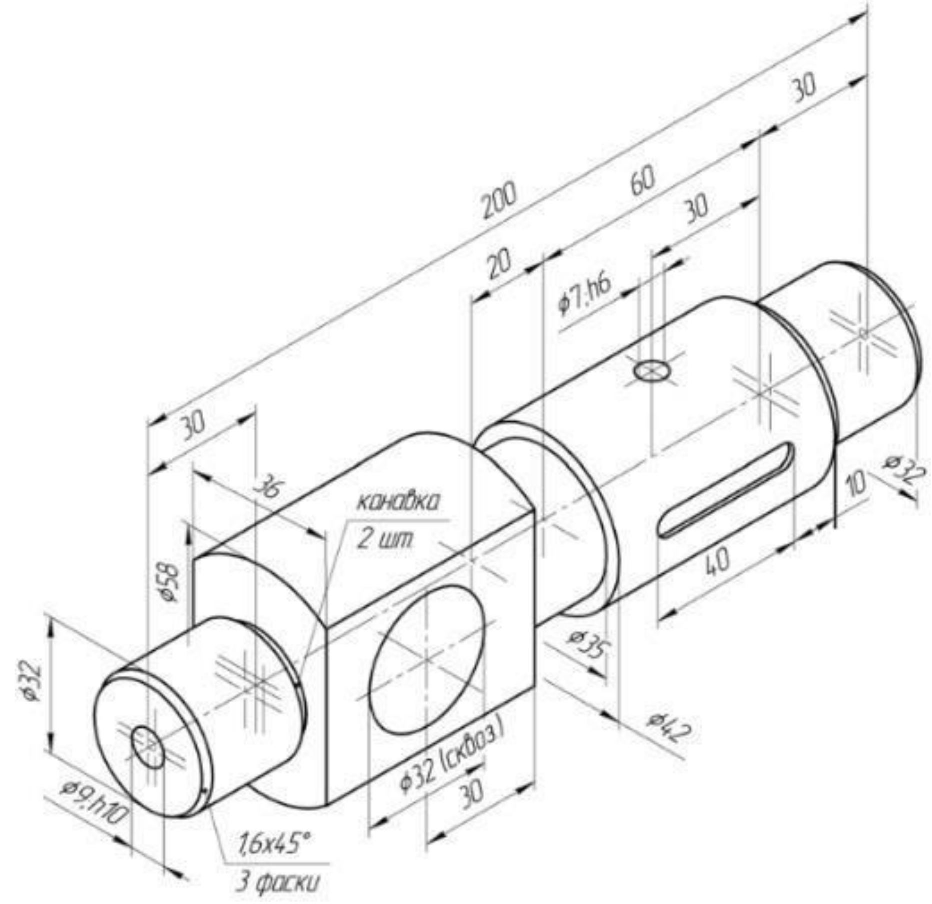
В-17


215
65
27
1,6x45°
3 фаски
35
ф5,h6
20
32
ф26
36
ф32
ф30
ф32
ф48
ф5(скв.)
31,5
канавка
5
14
паза

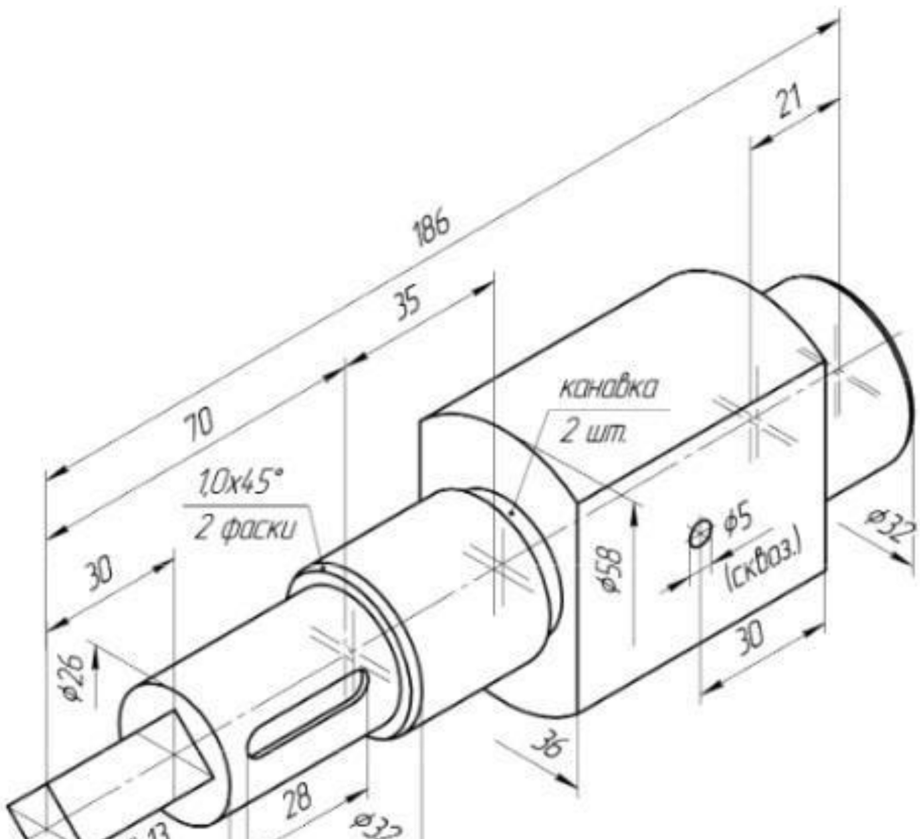
В-18


176
28
62
31
ф7,h6
63
ф30
30
15
ф6,h7
40
12
ф42
ф24
ф35
1,6x45°
5 фасок
ф30
канавка
2 шт.

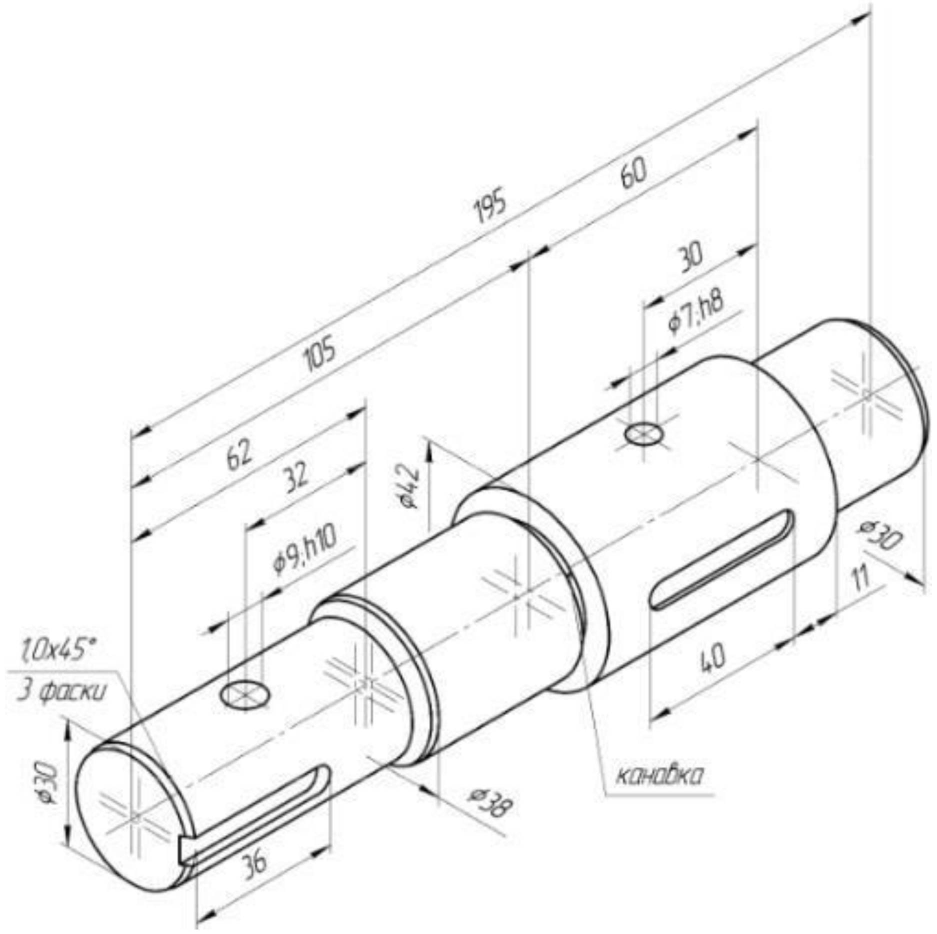
В-19


200
60
30
30
20
ф7,h6
30
36
канавка
2 шт.
ф58
ф32
ф32
10
40
ф35
ф42
ф32 (сквоз.)
30
ф9,h10
1,6х45°
3 фаски

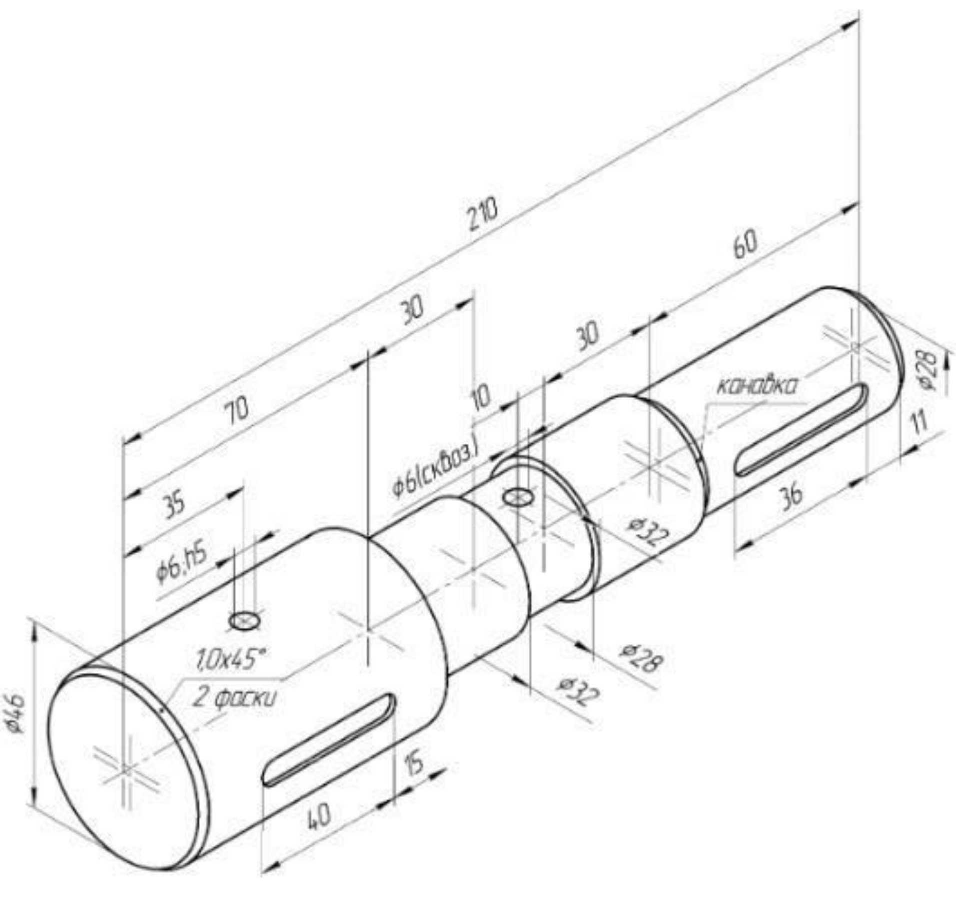
В-20


186
21
35
70
1,0х45°
2 фаски
канавка
2 шт.
ф5
(сквоз.)
ф58
ф32
30
30
ф26
36
28
ф32
□13
6

В-21

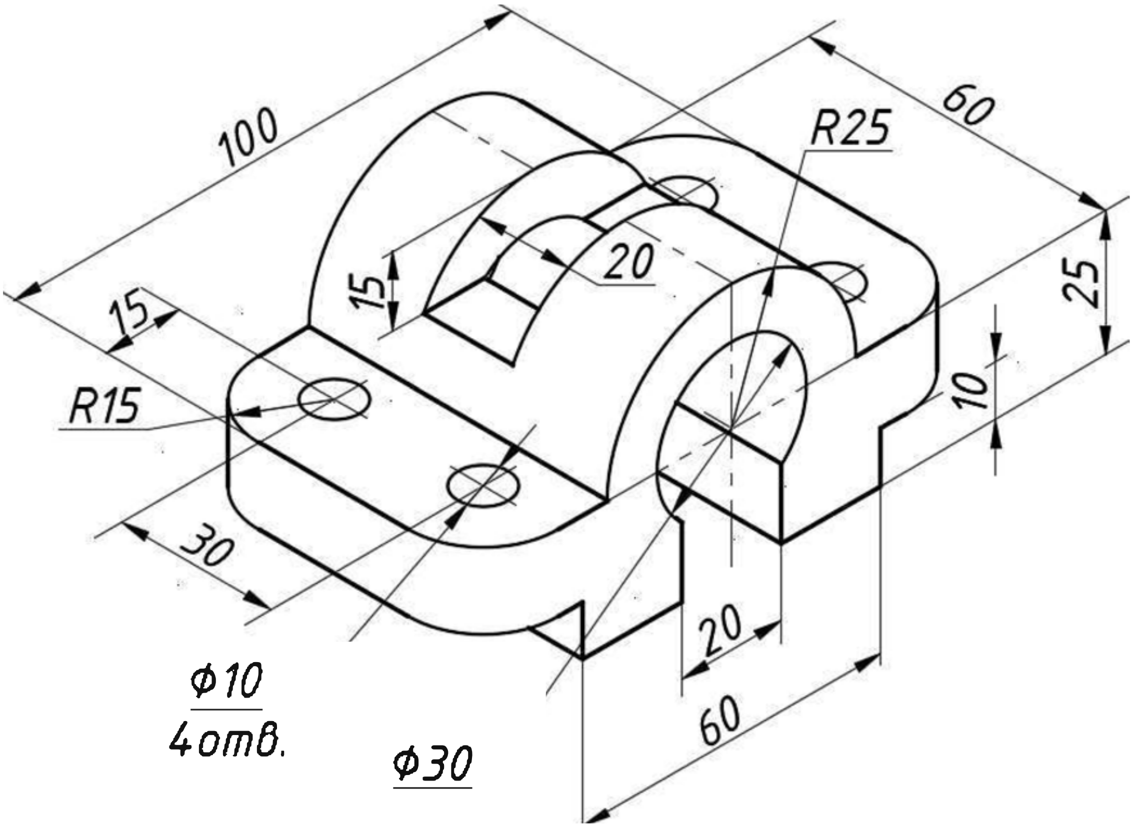

195
60
105
30
ф7,h8
62
32
ф42
ф9,h10
ф30
11
1,0x45°
3 фаски
40
ф30
канавка
ф38
36

В-22


210
60
30
30
70
10
канавка
ф28
11
ф6(сквоз.)
35
ф32
36
ф6,h5
1,0x45°
2 фаски
ф28
ф32
ф46
15
40

Assignments for Laboratory Work No. 4

Assignment 1

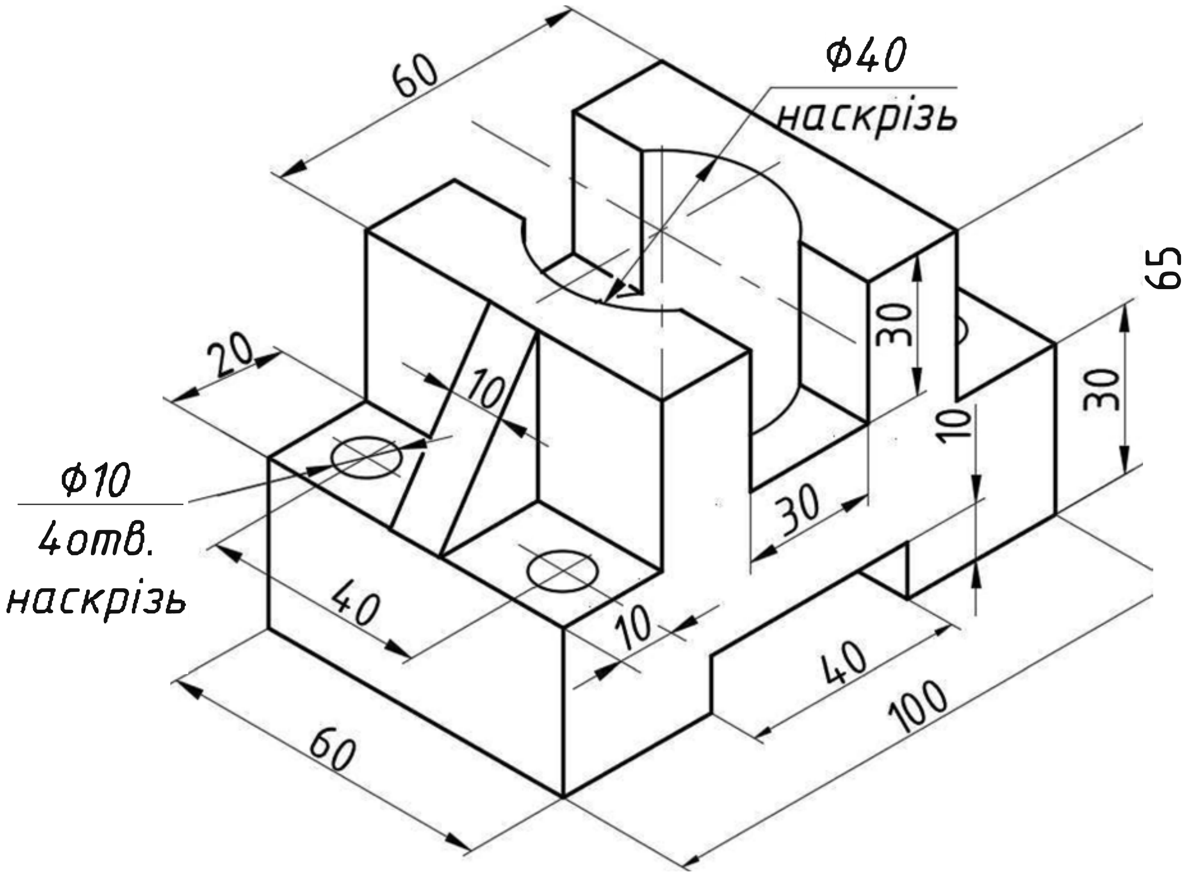


Static

1023 Carbon Steel Sheet (SS)

1000 H

Assignment 2

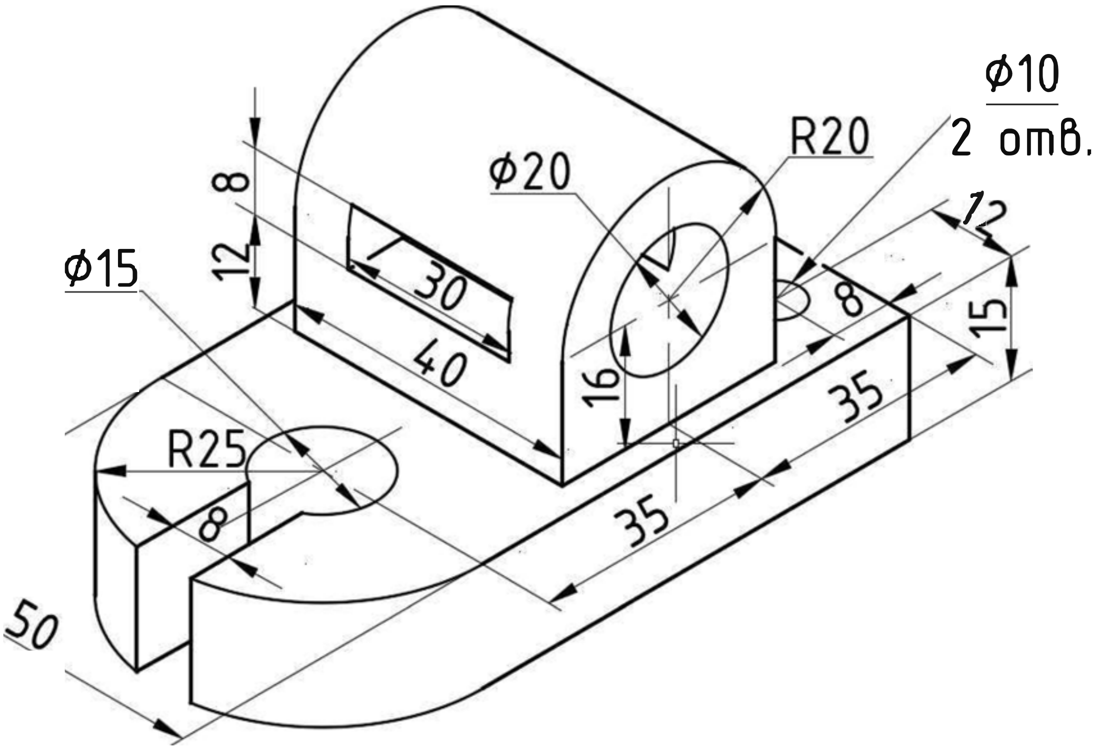


Static

AISI 1020

1200 H

Assignment 3

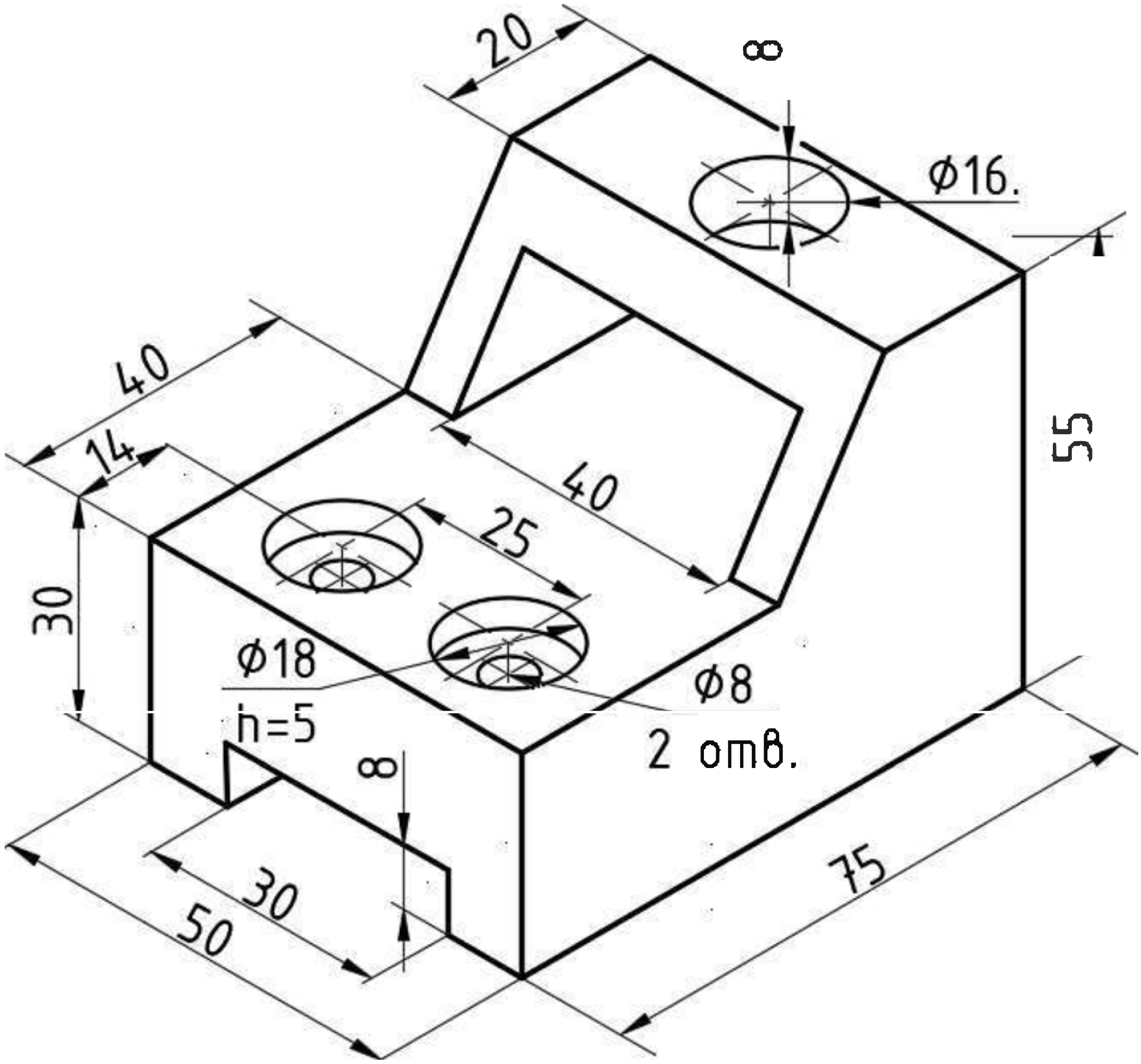


Static

AISI 1035 Steel (SS)

900 H

Assignment 4

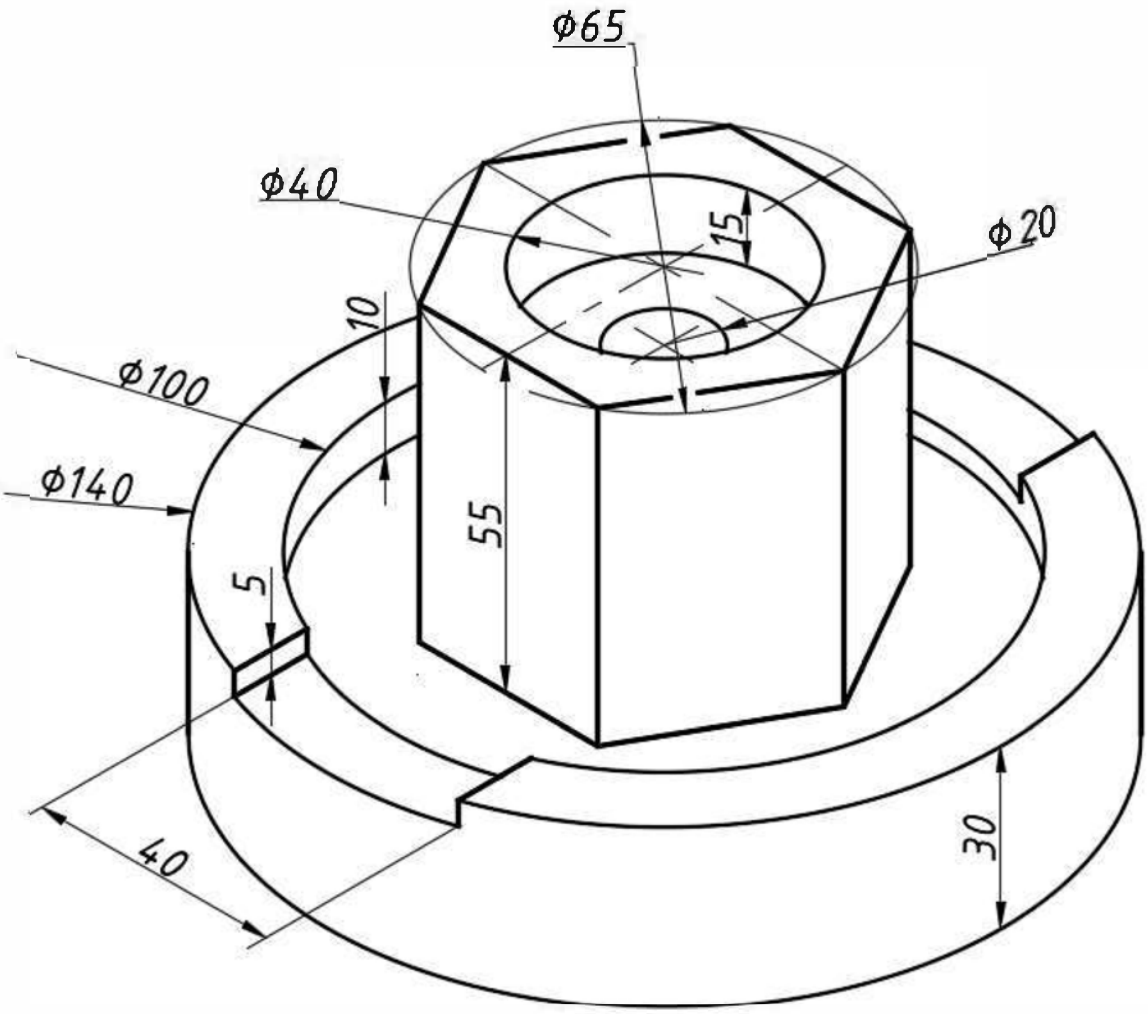


Static

AISI 304

500 H

Assignment 5

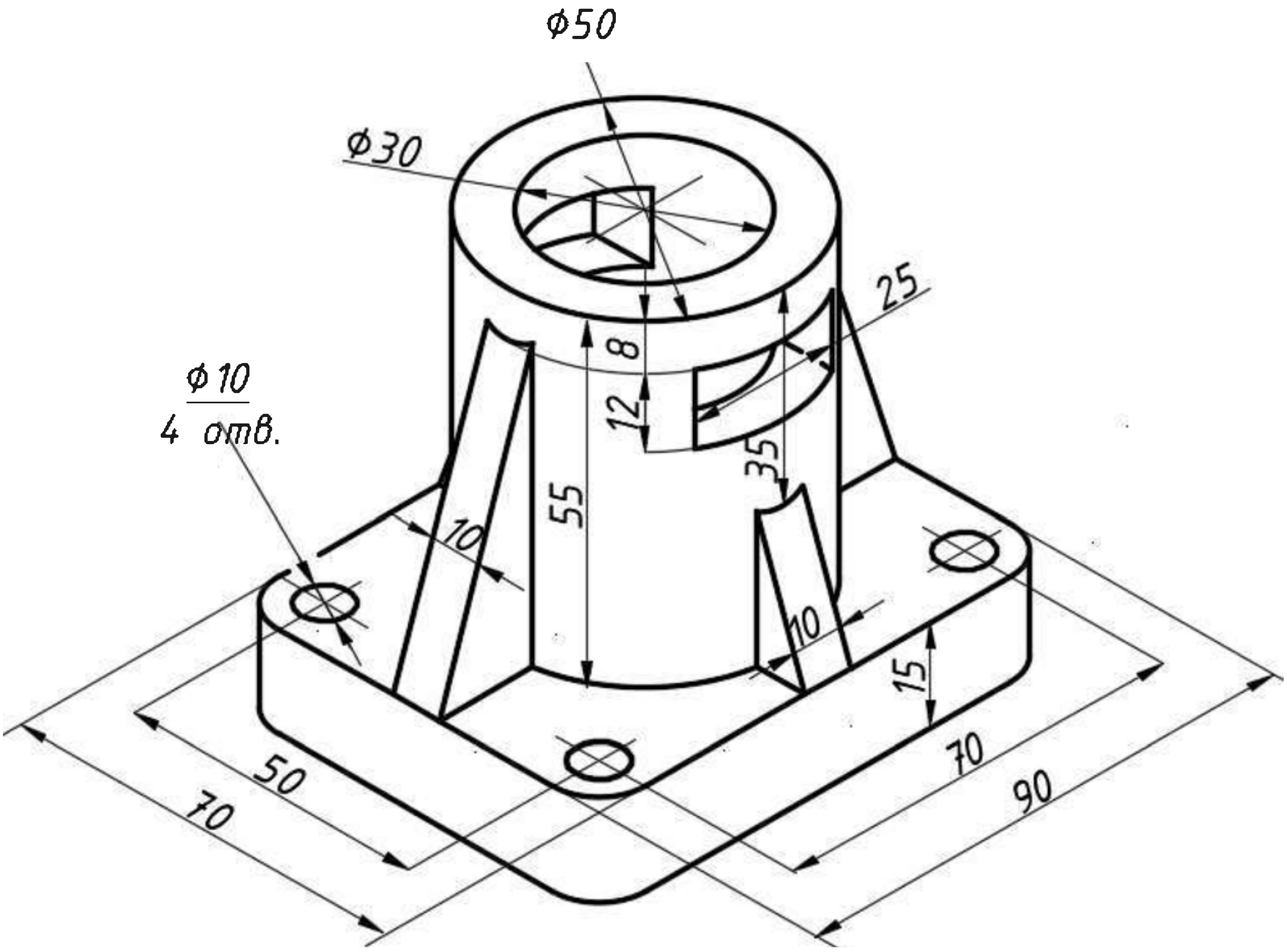


Static

Alloy Steel

1300 H

Assignment 6

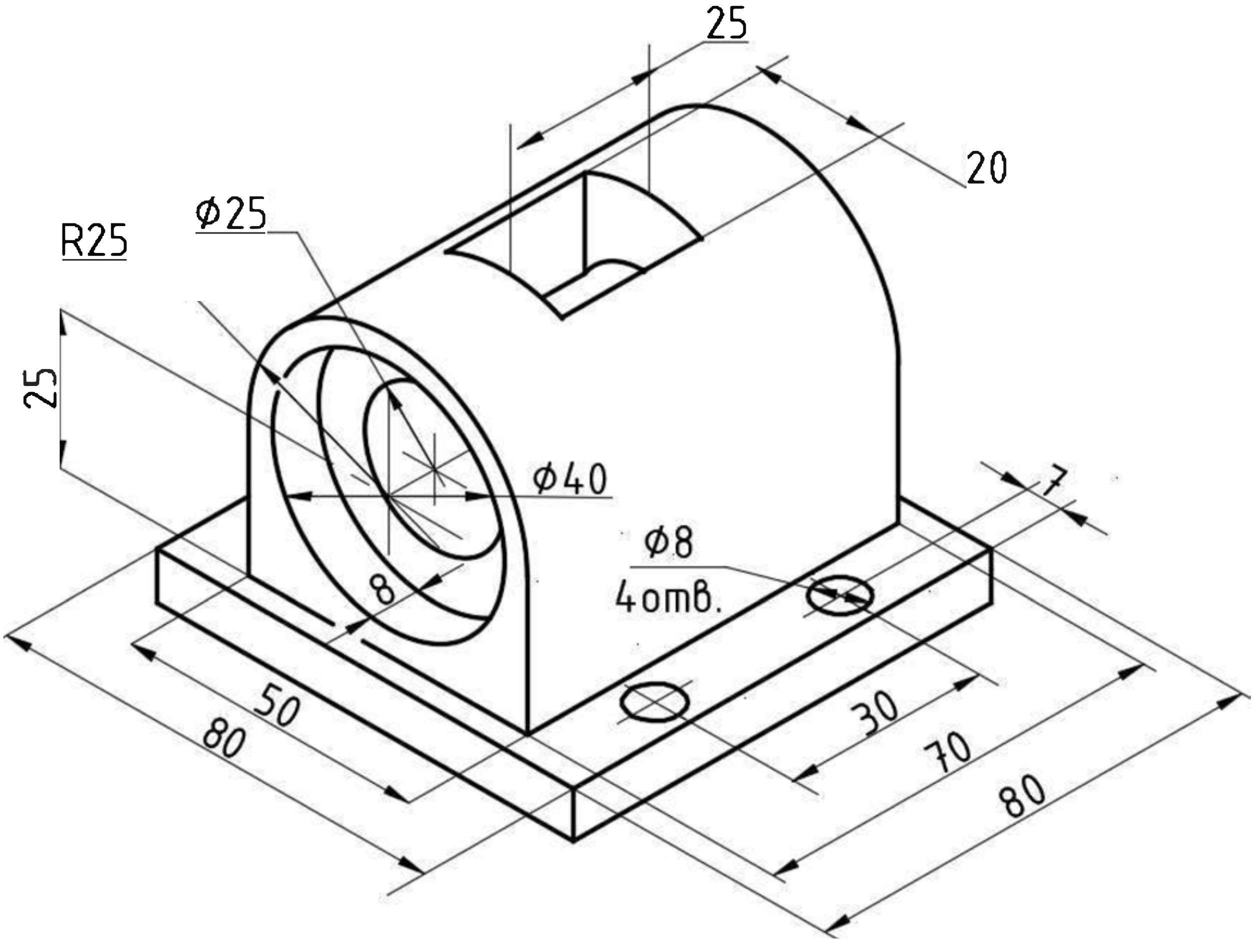


Static

Alloy Steel (SS)

360 H

Assignment 7

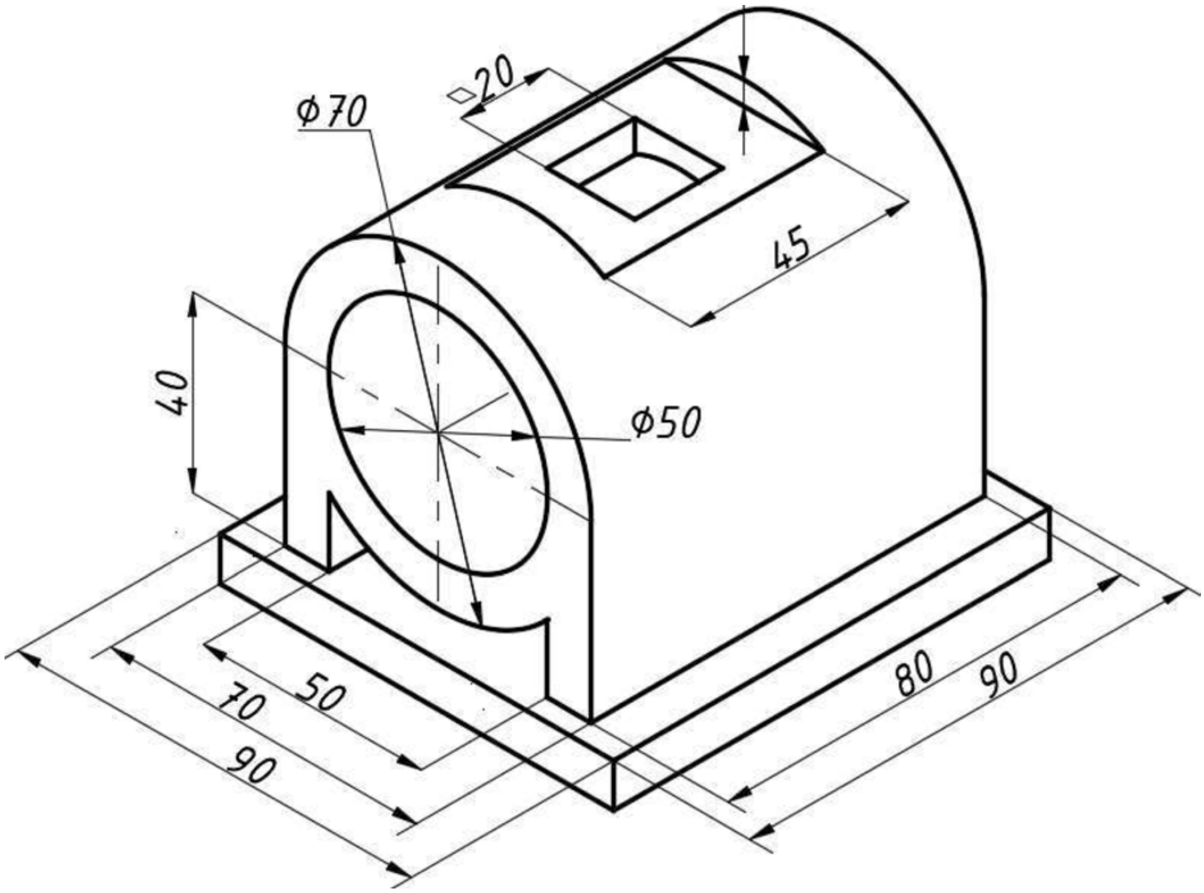


Static

Galvanized Steel

700 H

Assignment 8

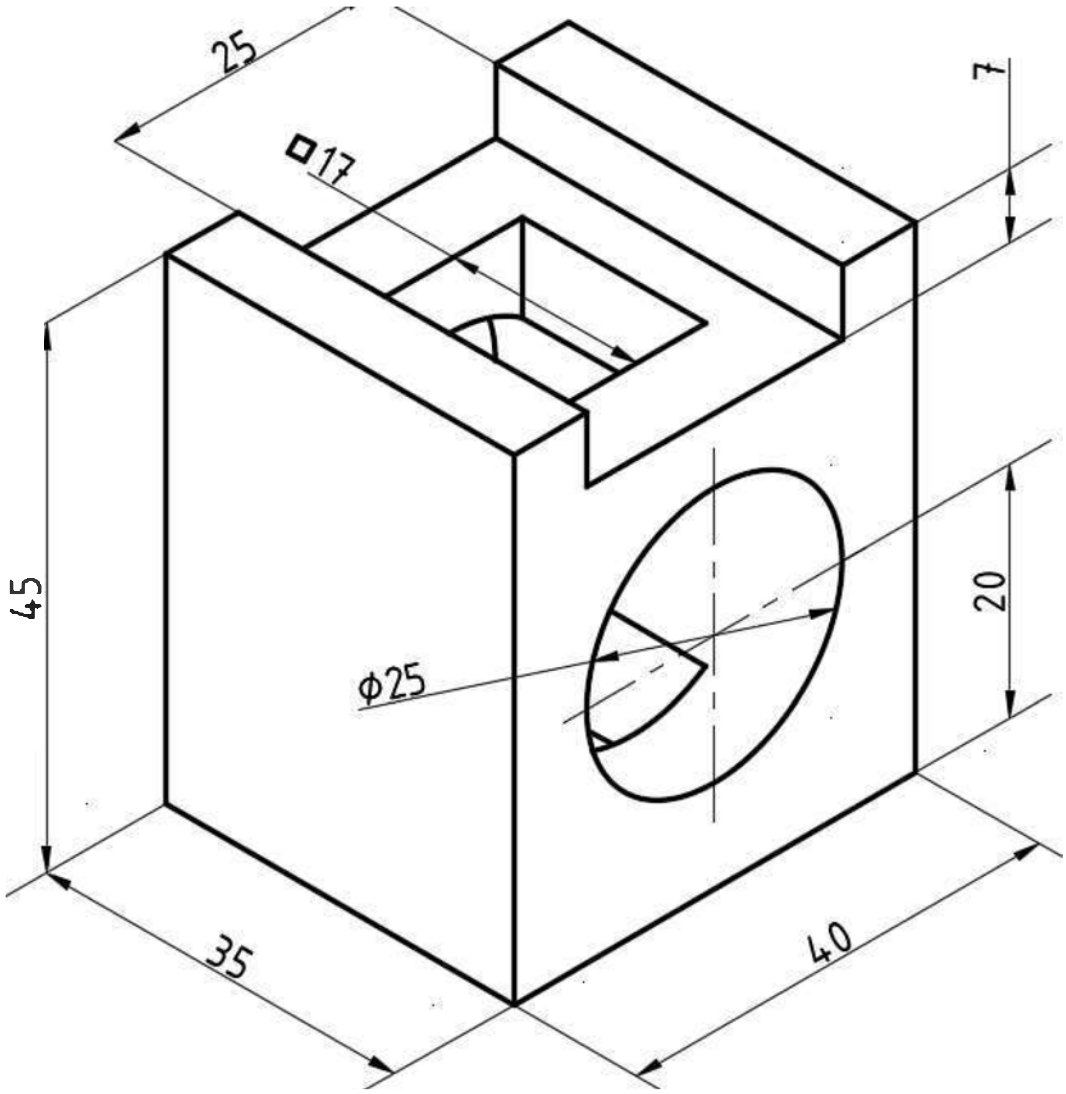


Static

Plain Carbon Steel

480 H

Assignment 9

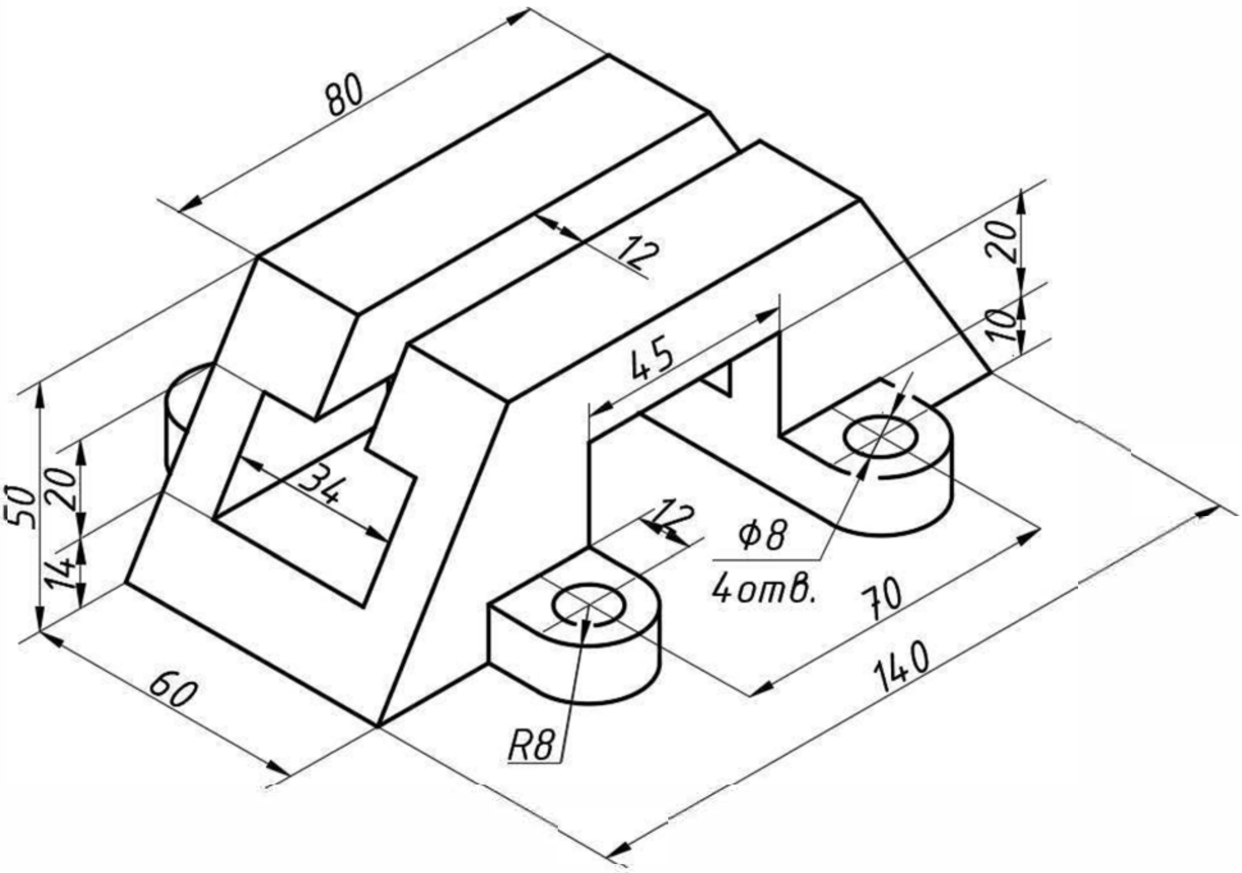


Static

Stainless Steel (ferritic)

300 H

Assignment 10

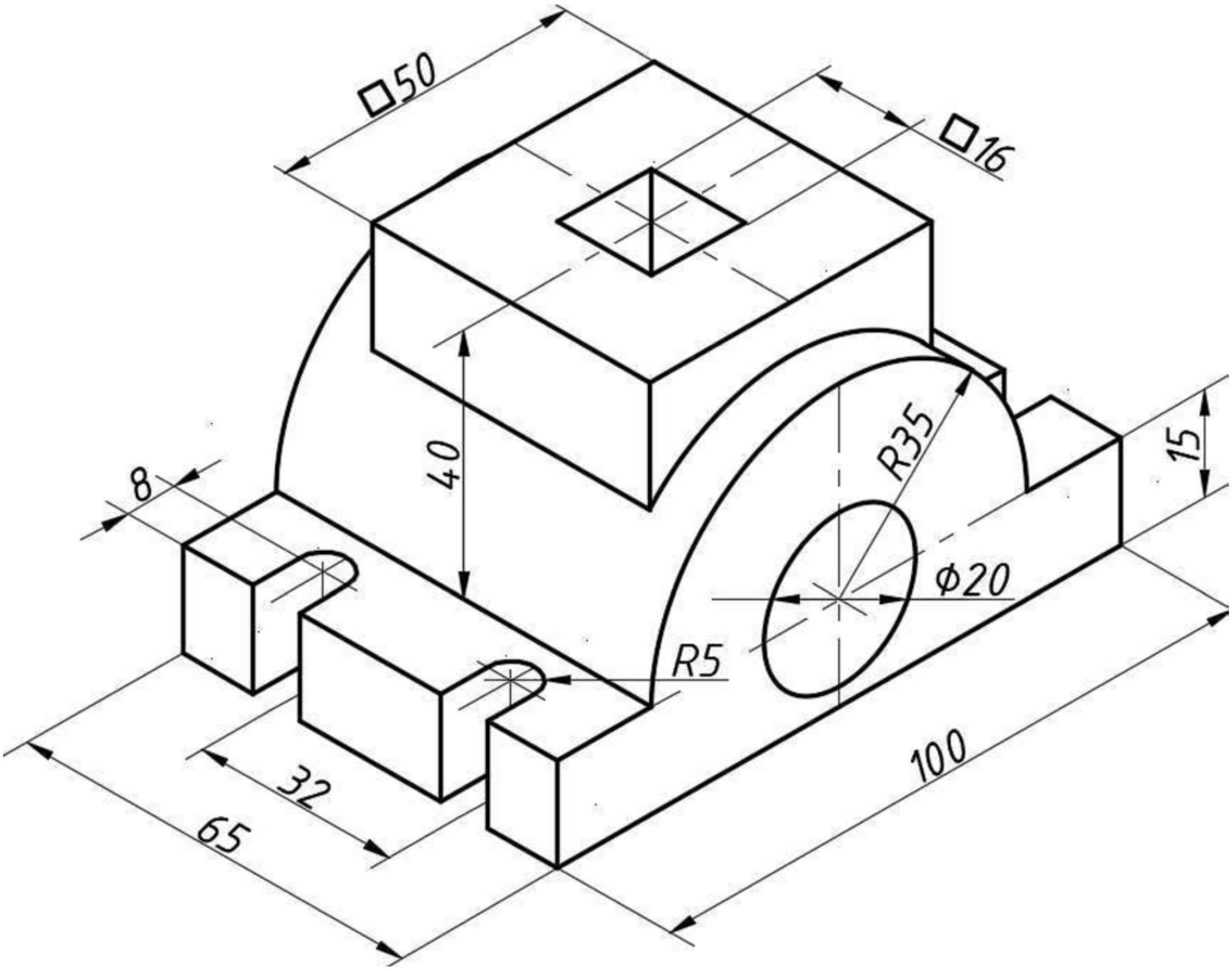


Static

Cast Alloy Steel

600 H

Assignment 11

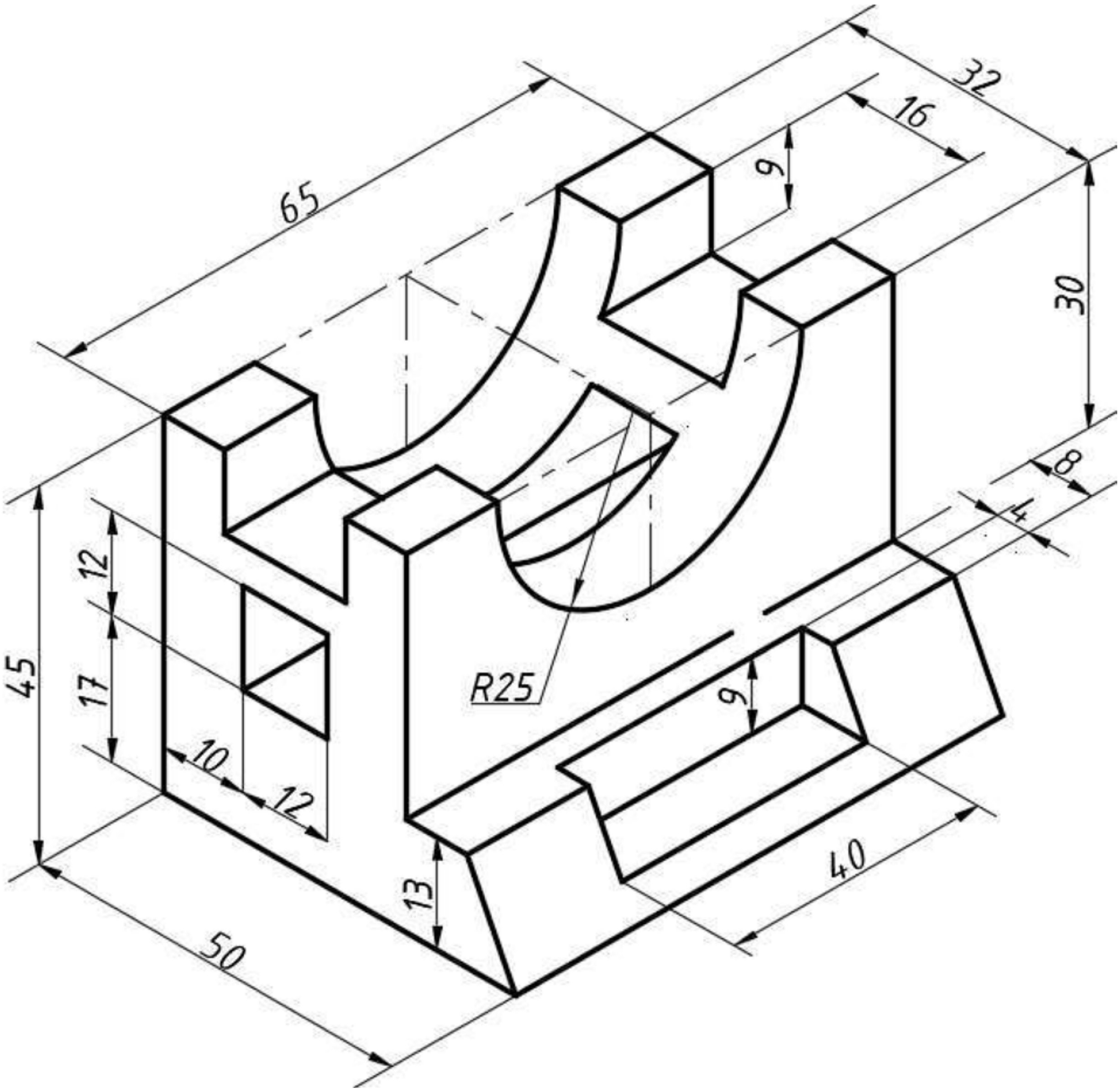


Static

Cast Carbon Steel

1000 H

Assignment 12

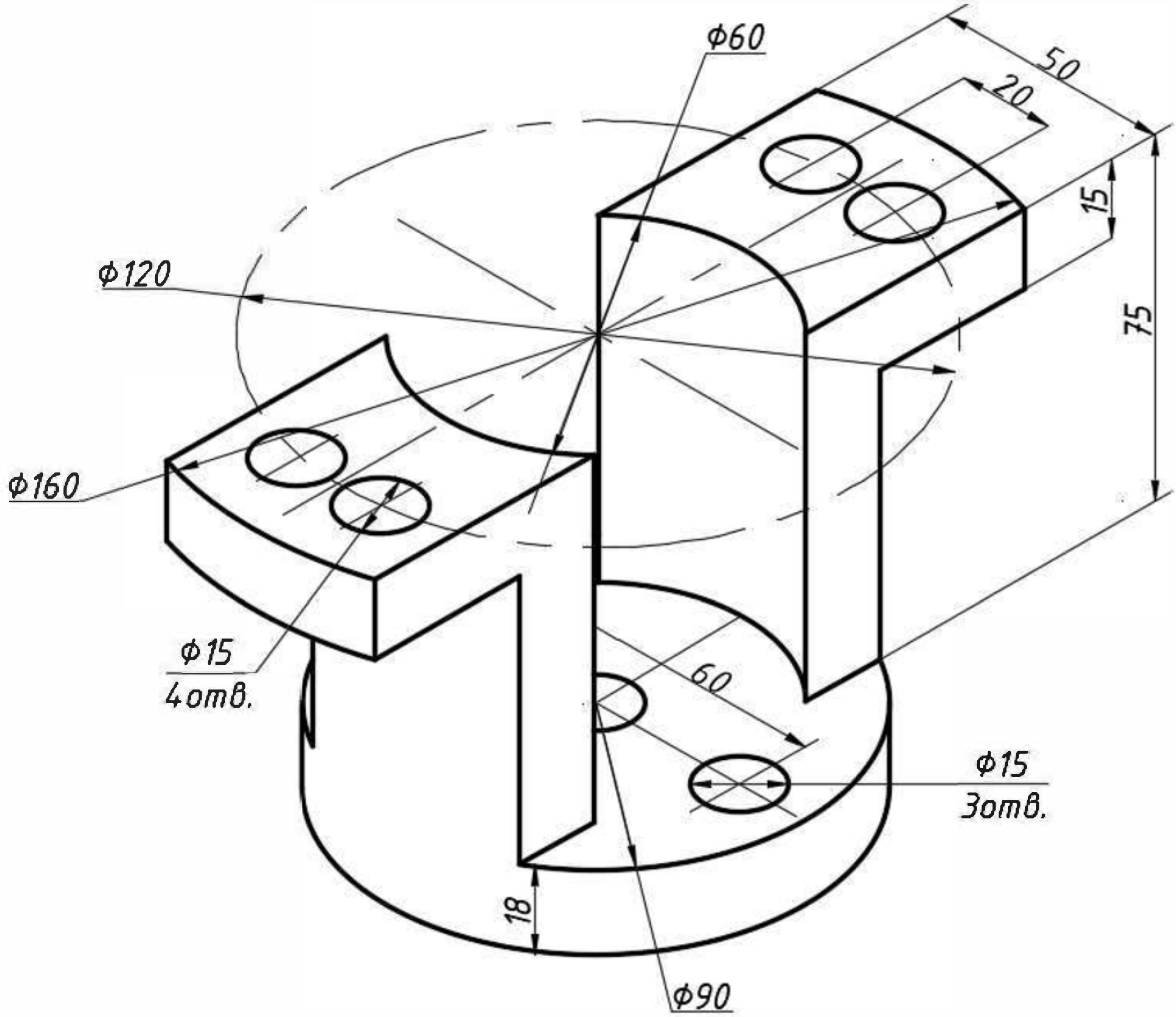


Static

Ductile Iron

1200 H

Assignment 13

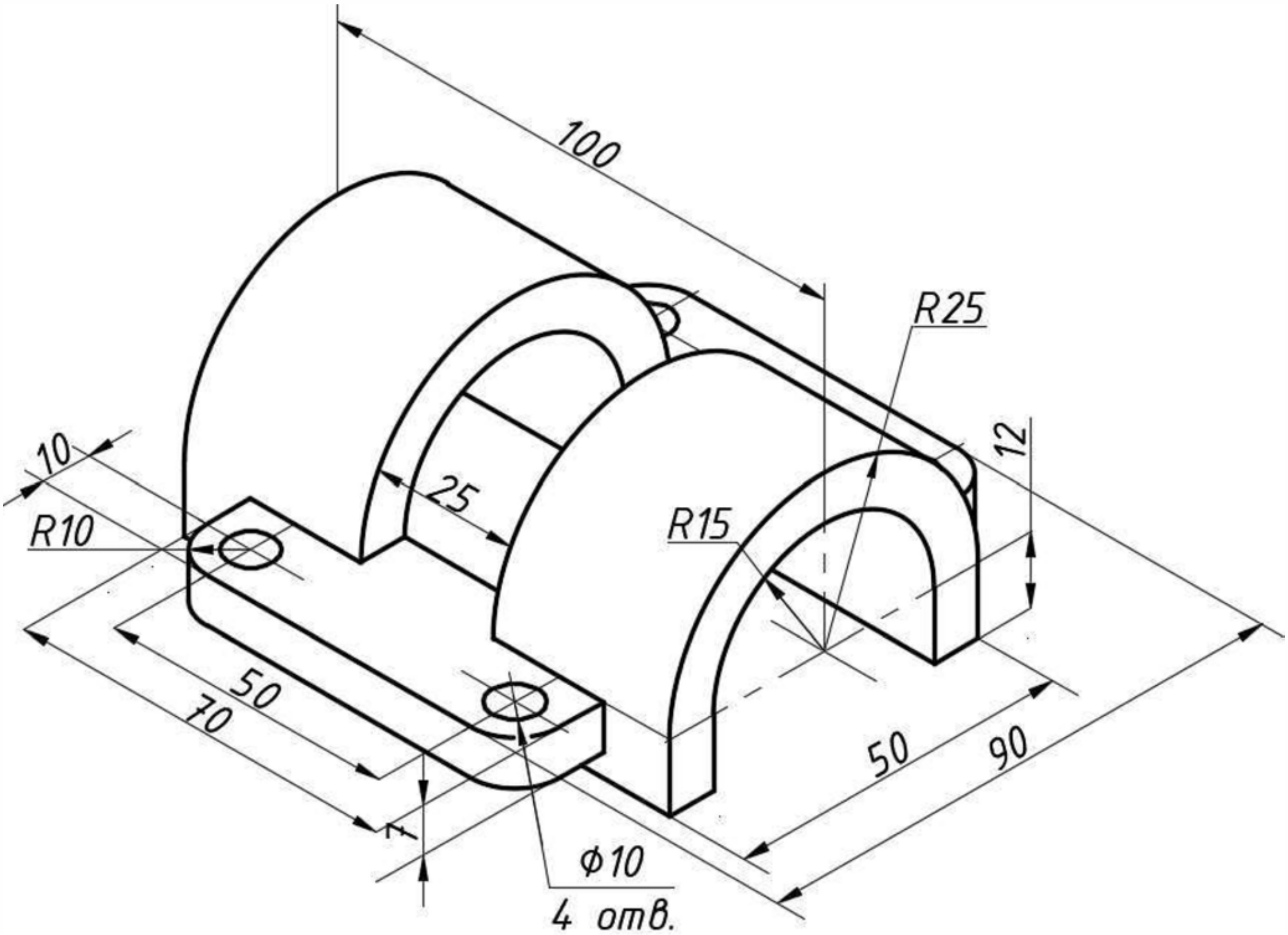


Static

1060 Alloy

200 H

Assignment 14

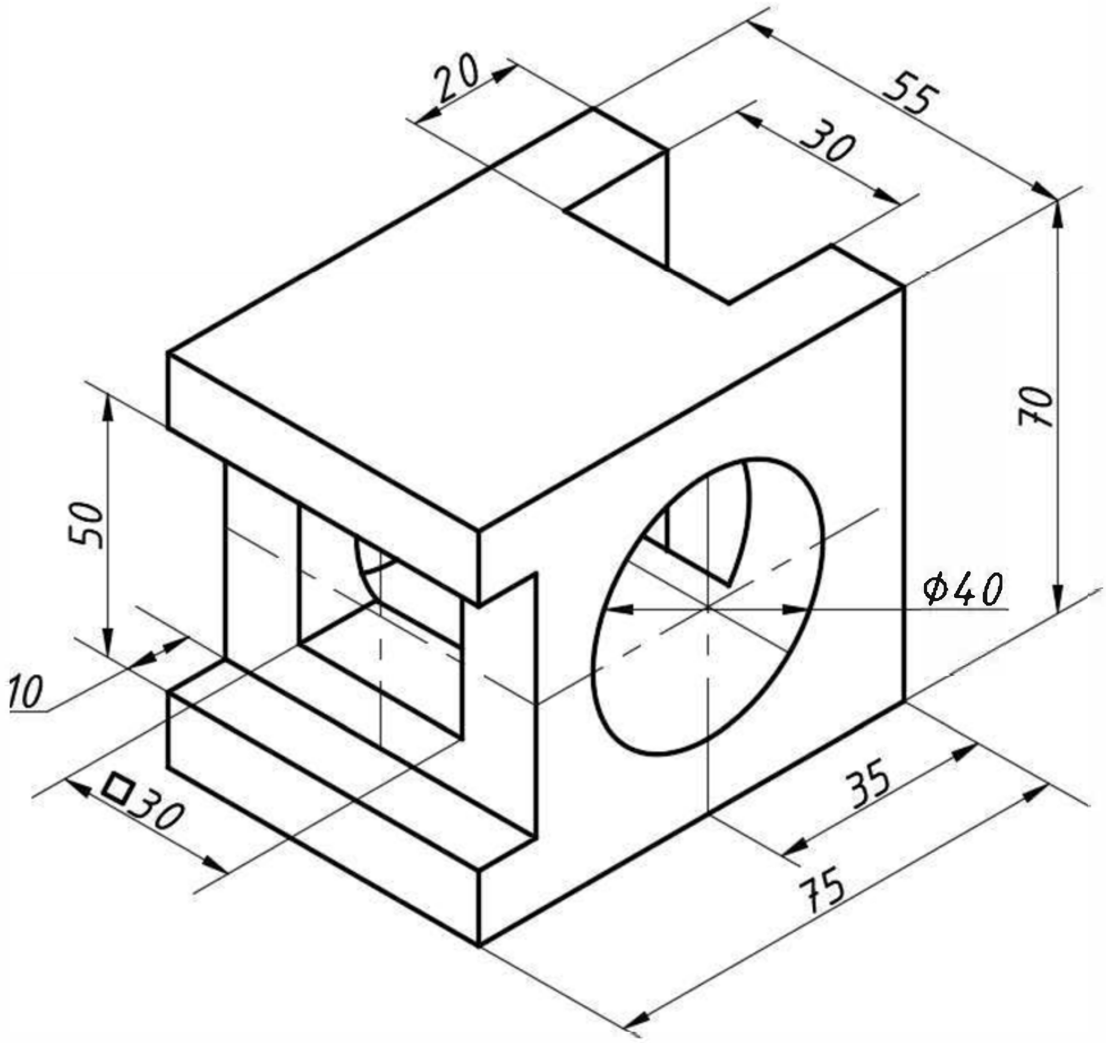


Static

1060-H12

50 H

Assignment 15

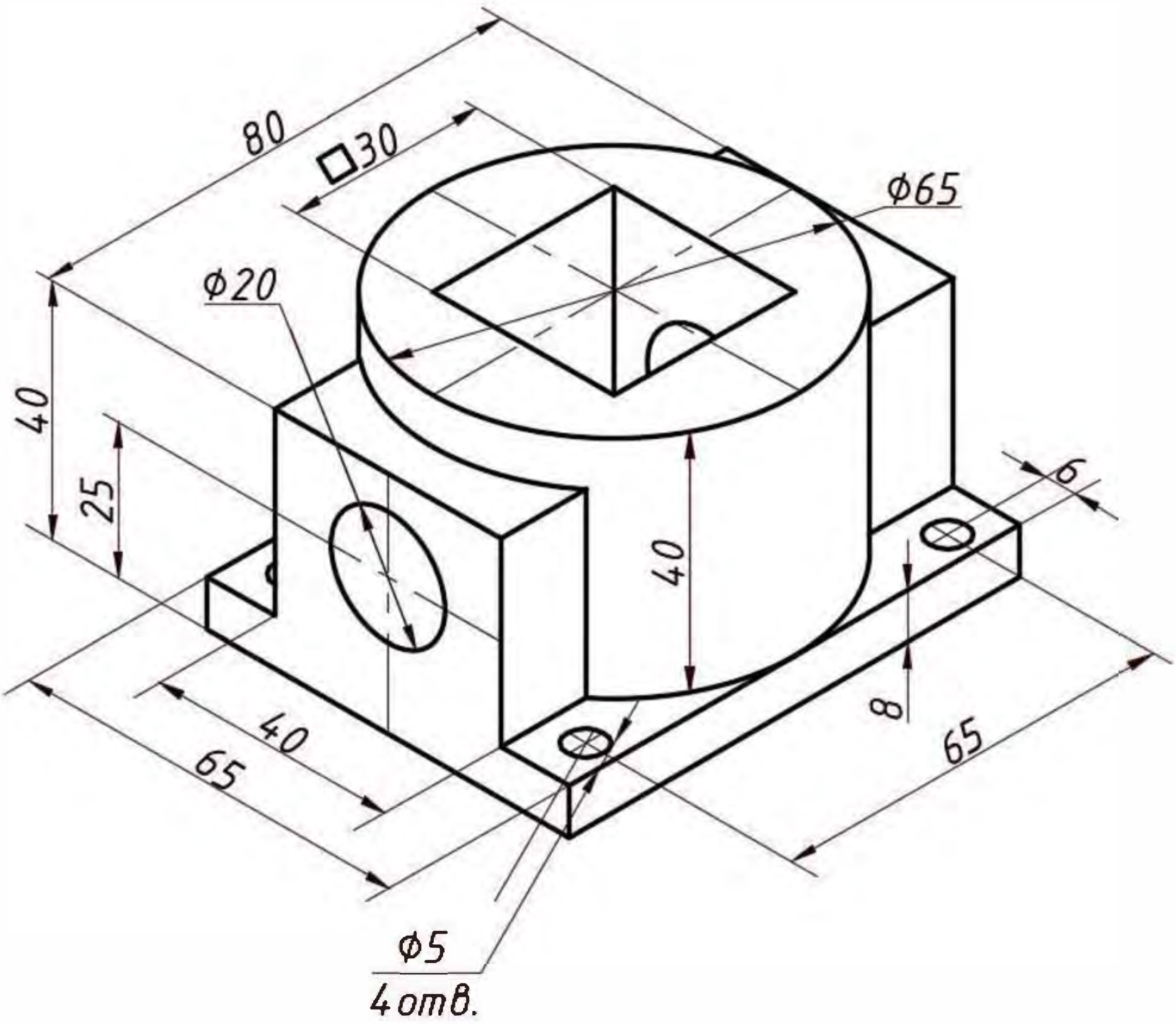


Static

1060-H14

150 H

Assignment 16

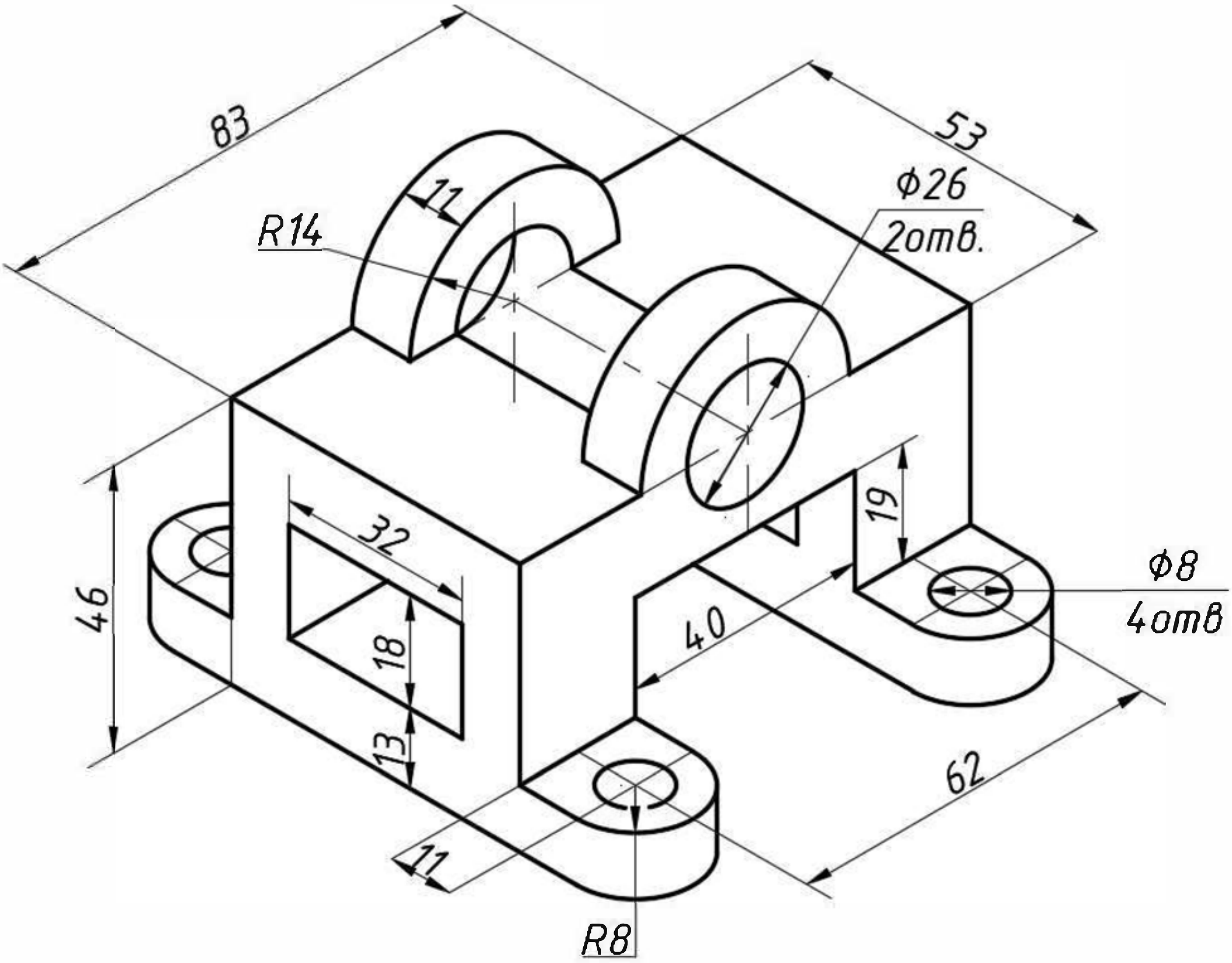


Static

1060-H16

380 H

Assignment 17

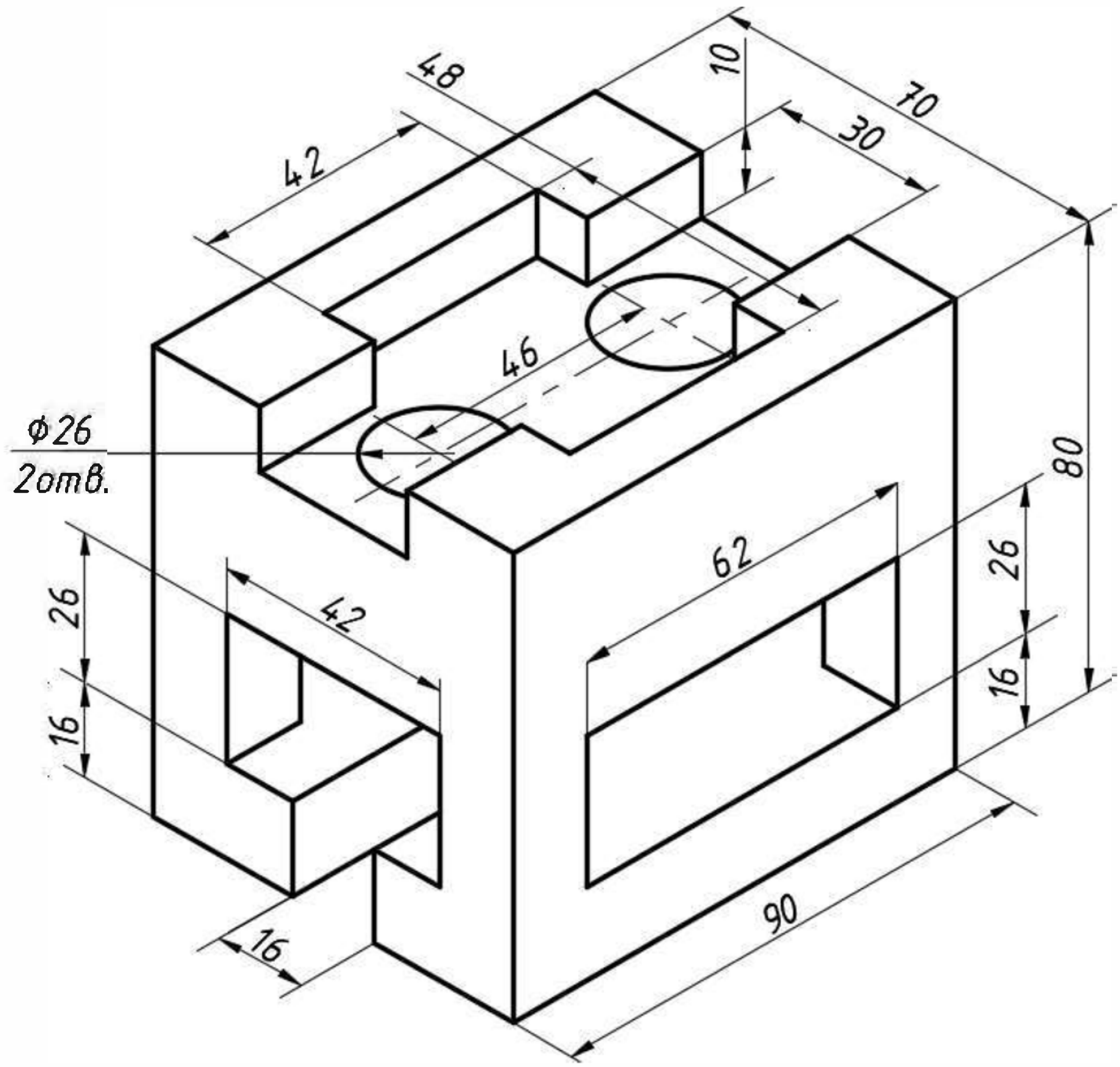


Static

1060-H18

500 H

Assignment 18

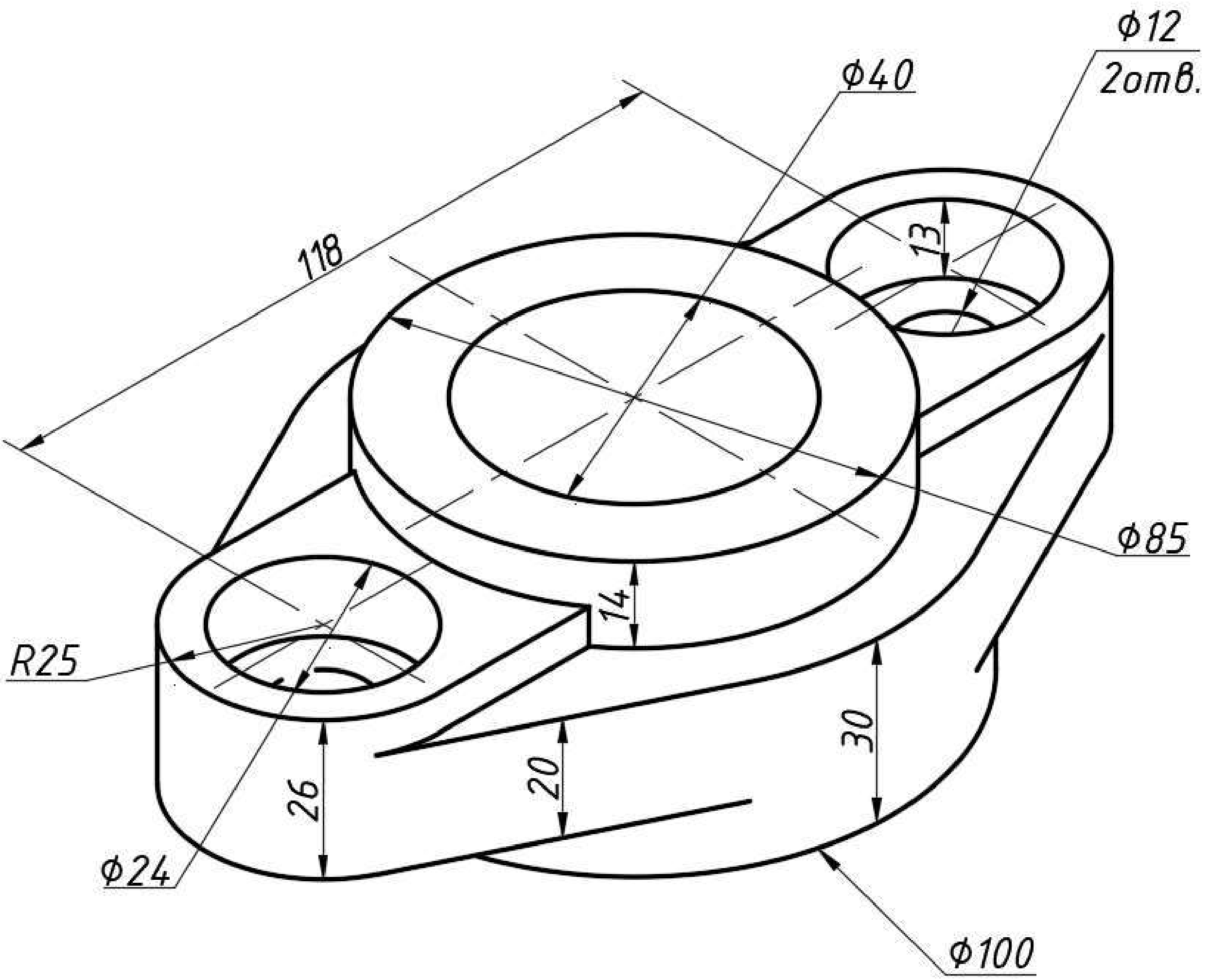


Static

1345 Alloy

270 H

Assignment 19

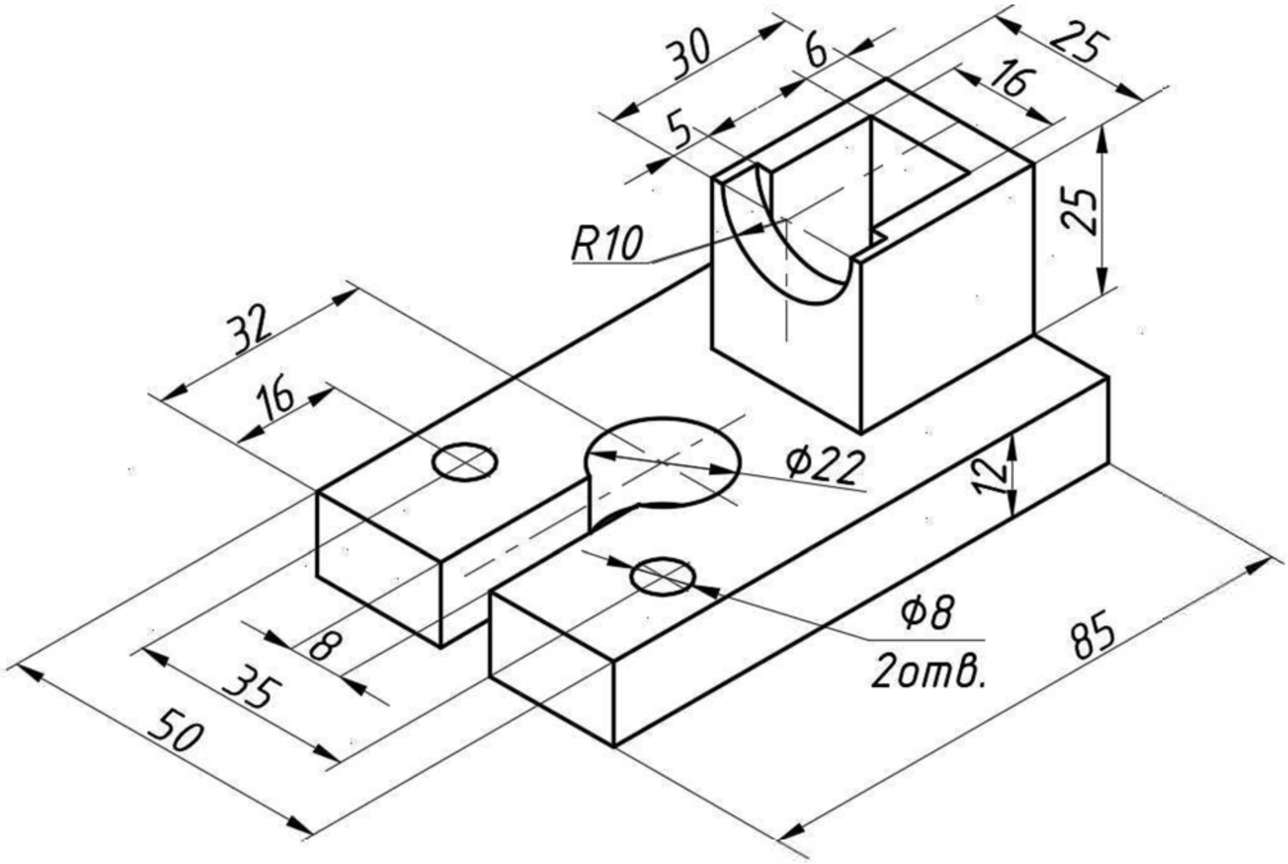


StaticM

1350 Alloy

700 H

Assignment 20



Static

Aluminum Bronze

800 H

FOR NOTES

Навчальне видання

Методичні вказівки до лабораторних робот з навчальної дисципліни «САПР гідротурбін, оборотних гідромашин, малих, міні- та мікро ГЕС(англійською мовою)»
для студентів денної та заочної форми навчання
за спеціальністю «Галузеве машинобудування»
освітня програма «Галузеве машинобудування»

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Роботу до видання рекомендував проф. Роговий А. С.

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