NATIONAL UNIVERSITY OF CIVIL DEFENCE OF UKRAINE

DEPARTMENT OF ENGINEERING AND EMERGENCY RESCUE EQUIPMENT

Lecture Course on the Discipline

«Engineering and Computer Graphics»

SECTION: «COMPUTER GRAPHICS»

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TABLE OF CONTENTS

INTRODUCTION 4

Lecture №1 5

SOLIDWORKS PROGRAM INTERFACE 5

Lecture №2 24

2D DRAWING, SKETCH CREATION IN SOLIDWORKS ENVIRONMENT 24

Lecture №3 41

FUNDAMENTALS OF PART MODELING IN SOLIDWORKS ENVIRONMENT 41

Lecture № 4 54

CREATING DRAWINGS BASED ON GENERATED THREE-DIMENSIONAL MODELS IN SOLIDWORKS 54

Lecture №5 70

ASSEMBLY MODELING IN SOLIDWORKS 70

Lecture №6 80

ADDITIONAL TECHNIQUES IN SOLIDWORKS ENVIRONMENT 80

INTRODUCTION

Currently, three-dimensional models of mechanisms and machines are used not only for generating design and technological documentation but also for conducting engineering analysis through specialized software. The quality of geometric models directly influences the feasibility and accuracy of such analyses. Thus, obtaining simple geometric models of technical objects, free from unnecessary elements, is a relevant task in the design process.

SolidWorks, developed by SolidWorks Corp. (USA), is a powerful design tool that comprehensively addresses the challenges faced by everyday engineering practitioners.

The SolidWorks software is the most widely used tool for Computer-Aided Design (CAD) and 3D modeling. SolidWorks is a parametric solid modeling tool, allowing the creation of details suitable for future 3D printing. This protects the designer from errors that may inevitably occur during the manual drafting of product projections.

The SolidWorks program employs a familiar and user-friendly Windows interface, making it accessible to a wide range of users. Users can create fully associative three-dimensional solid models with or without constraints, utilizing automatic or user-defined relationships, thus realizing the project's concept.

SolidWorks design involves the creation of volumetric models for parts and assemblies, with the capability to generate working drawings based on these models.

SolidWorks fully complies with the standards of the Unified System of Design Documentation (ESKD) regarding the presentation of design documentation.

Mastering the modeling process in SolidWorks will enable students to enhance their repertoire of graphic knowledge and skills, develop creative thinking, and expand spatial imagination.

Lecture №1

SOLIDWORKS PROGRAM INTERFACE

Lecture Objective: To explore the purpose of the program, become familiar with the SolidWorks interface, and understand the basics of operation.

Launching SolidWorks

Launching SolidWorks on the Windows operating system is done by running the solidworks.exe file from the SolidWorks folder, activating the program logo, or opening any of the SolidWorks graphic files. Using the opened program window, you can either create a new SolidWorks file by clicking the button or open a previously created one by clicking the button .

In SolidWorks, three types of documents can be created: part, assembly, drawing (Fig. 1.1).

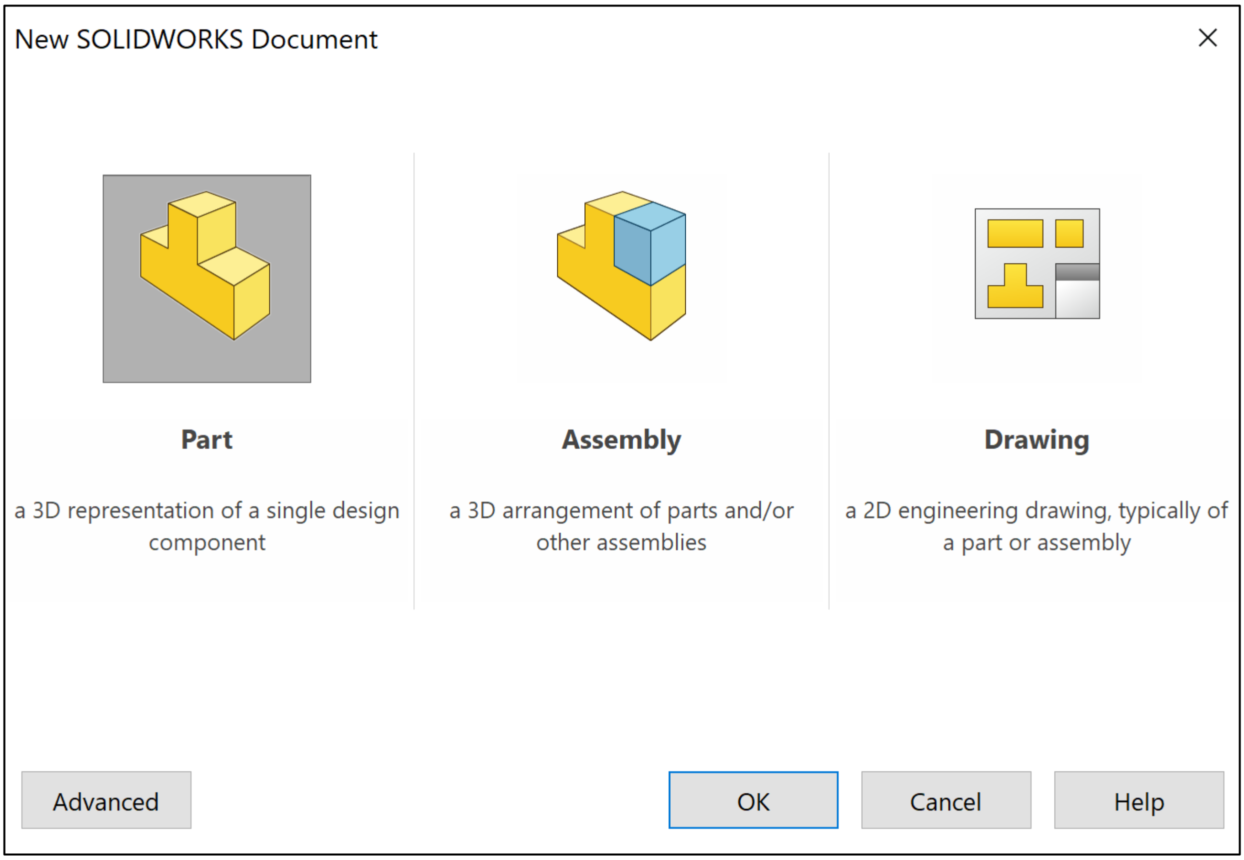


Fig. 1.1 – Creating a new document in SolidWorks

When saving documents, files will be assigned an extension depending on their type (table 1.1).

Table 1.1 – File extensions depending on the type of document

Document type	File extension
Part	.SLDPRT
Assembly	.SLDASM
Drawing	.SLDDRW

Робоче вікно SolidWorks

The SolidWorks workspace is illustrated in figure 1.2 and consists of the SolidWorks shell, which includes the menu, toolbars, and the status bar, and the document window, which is divided into two parts: the graphics area and the information-management area. Multiple document windows can be open simultaneously, and their layout and sizes are configured through the "Window" section of the main menu. If multiple documents are open, you can use the Ctrl+Tab key combination to switch between them.

Access all SolidWorks commands using menus. SolidWorks menus use Windows conventions, including submenus and checkmarks to indicate that an item is active. Can also use context-sensitive shortcut menus by clicking the right mouse button.

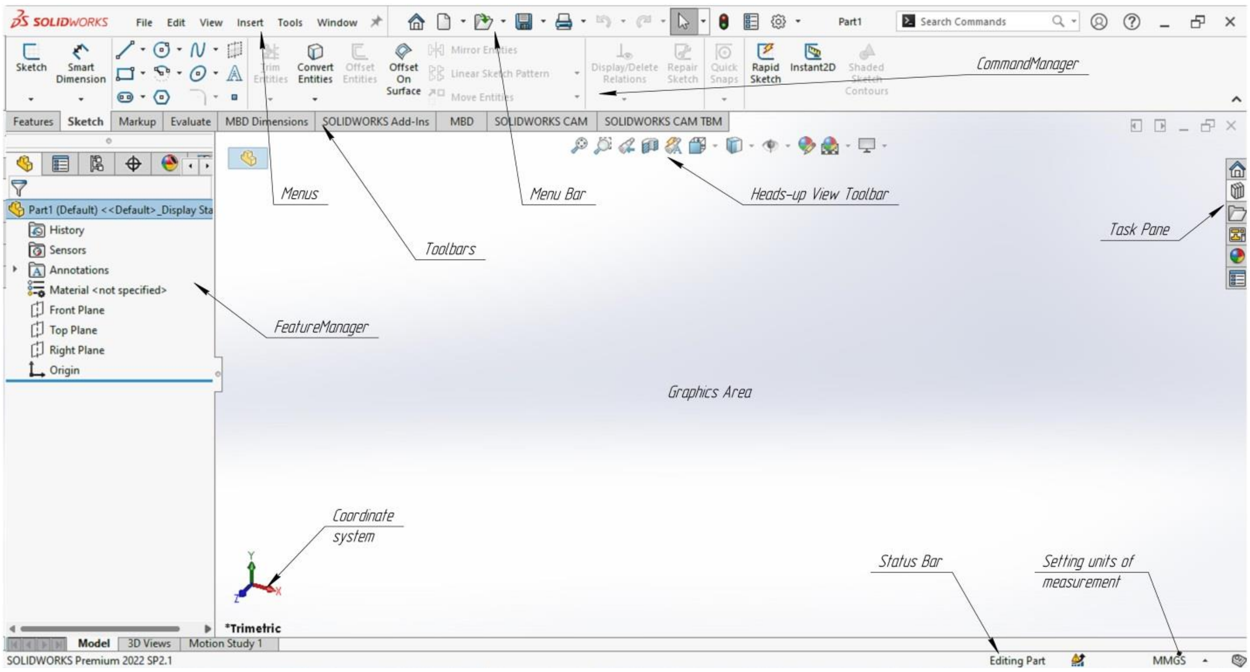


Fig. 1.2 – Interface Elements SolidWorks

Menu Bar

The menu bar contains the SolidWorks menus, a set of Quick Access Tools, the SolidWorks Search, and the Help options.

The menus appear when you mouse over or click the SolidWorks logo. You can pin the menus to keep them visible. The toolbar moves to the right when the menus are pinned (fig. 1.3).

Fig. 1.3 – SolidWorks menus

1. **File** – performing operations with files.
2. **Edit** – performing operations with objects.
3. **View** – changing the way objects are displayed.
4. **Insert** – inserting objects from other programs.
5. **Tools** – system configuration, tools for construction.
6. **Window** – switching between document windows.

Each Quick Access Tools button has a downward-pointing arrow that opens a flyout menu containing with additional options (fig. 1.4).

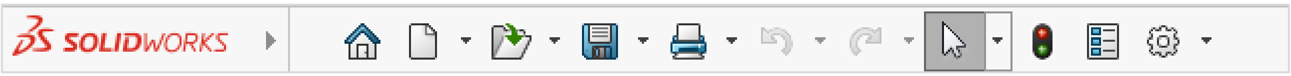
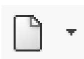


Fig. 1.4 – Quick Access Tools

You can customize the tools shown in Quick Access Tools in the same way that you customize other toolbars in the Customize dialog box. You cannot turn off Quick Access Tools but you can position it to appear in the Menu Bar or in the Command Manager. Click **Tools** > **Customize** and on the Toolbars tab, under **Quick Access Tools**, select **Show in the Menu Bar** or **Show in the CommandManager**.

 **New**: Helps create a new part/assembly file. Keyboard shortcut: «Ctrl + N».

 **Open**: Use this button if you want to open the last or previously saved part/assembly file. Keyboard shortcut: «Ctrl + O».

 **Save**: Press this button to save active part or assembly files on the PC's hard drive. It also includes options such as «Save, Save as, Save all, and «Publish to eDrawings». Keyboard shortcut: «Ctrl + S» (fig. 1.5).

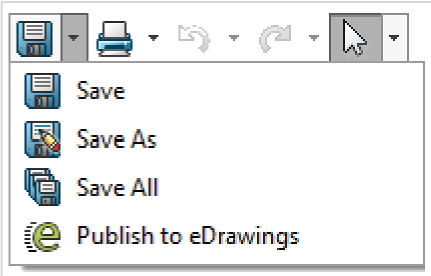


Fig. 1.5 – Save file

 **Print**: Use this button to print the active document of the part file. Keyboard shortcut: «Ctrl + P».

 **Undo**: By using the drop-down tree, you can choose which step you want to undo. Keyboard shortcut: «Ctrl + Z».

 **Select**: Used for selecting parts of the model or sketch, such as edges, components, vertices, etc. The context menu includes other selection options like extended selection, select hidden, select all, toolbar, internal components, etc. (fig. 1.6)

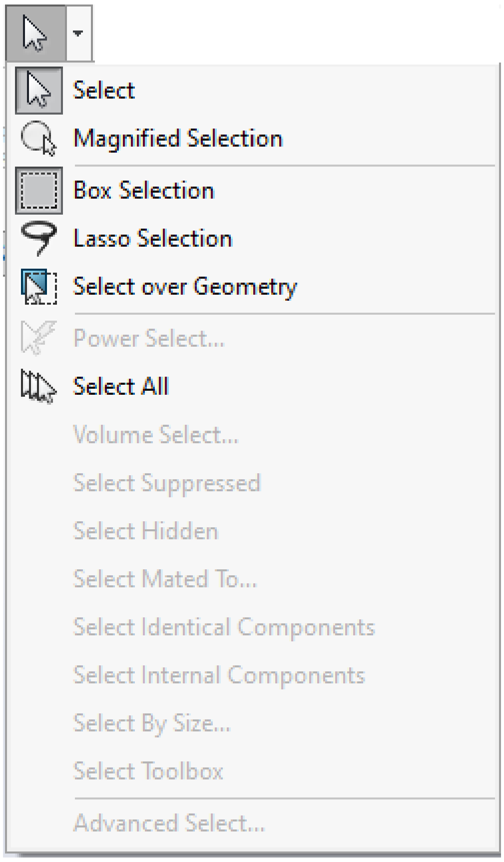


Fig. 1.6 – Selekt: expanded menu

 **Rebuild**: Rebuilds the part/assembly/drawing file. Keyboard shortcut: «Ctrl + B».

 **File Properties**: Here, you can see complete information about the active part/assembly document file.

 **Options**: In this section, you can modify SolidWorks settings, including both system properties and document properties. The expanded menu contains options for "settings" and managing add-ins (fig. 1.7).

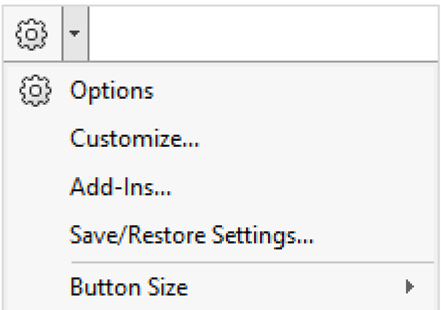


Fig. 1.7 – Options

The information-control area is located on the left side of the workspace and contains several tabs: Feature Manager, Property Manager, Configuration Manager.

FeatureManager Design Tree

To visually represent the design process in SolidWorks, there is the *FeatureManager Design Tree*. It is implemented in the style of the traditional Windows Explorer, typically located on the left side of the SolidWorks workspace, and it is a sequence of construction elements forming a part, as well as additional construction elements (axes and planes).

The *FeatureManager Design Tree* contains comprehensive information about the three-dimensional object and is dynamically linked to the graphics area. This means that the selection of sketch elements and auxiliary geometry can be done not only in the drawing area but also in the *FeatureManager*. In assembly mode, the *FeatureManager* displays a list of components included in the assembly, as well as the necessary connections between components and the assembly.

The main functions of the *FeatureManager Design Tree* include:

– select items in the model by name (left mouse button click).
– identify and change the order in which the features are created. You can drag items in the FeatureManager design tree list to reorder them. Dragging an item changes the order in which features regenerate when the model rebuilds.
– defining and changing the sequence in which elements are created.
– display the dimensions of a feature by double-clicking the feature’s name displaying.
– rename items by slowly clicking two times on a name to select it and entering a new name
– suppress and unsuppress part features and assembly components.
– locate errors and warnings associated with the model or a feature and described in tooltips and in What's Wrong?

– add or modify a material applied to a part by right-clicking Material.

When constructing a new three-dimensional part model, the *FeatureManager Tree*, by default, includes the following graphic elements (fig. 1.8):

– origin point with zero initial coordinates.

– three mutually perpendicular planes: Front Plane, Top Plane, Right Plane.

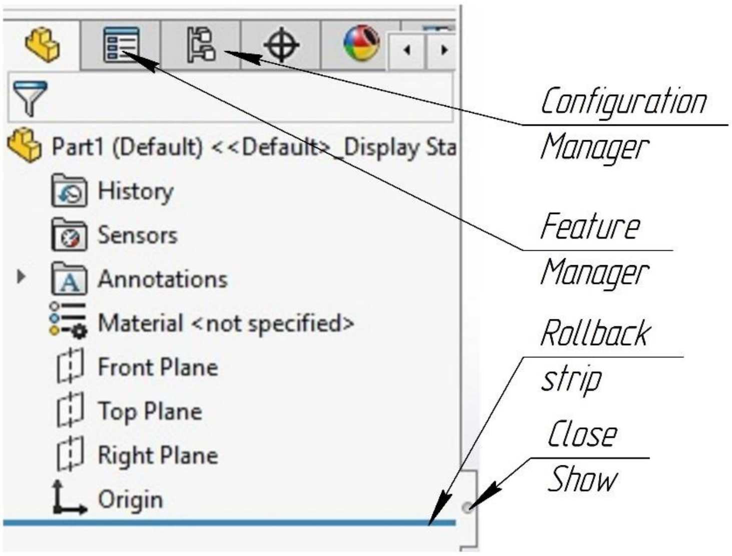


Fig. 1.8 – FeatureManager Design Tree

The FeatureManager design tree uses the following conventions:

The symbol ▸ to the left of the feature name, it indicates the presence of nested and associated elements (for example, sketches). To expand the feature and display its contents, simply click on the symbol ▸. Upon clicking ▸, the symbol changes to ▾, which can be clicked to return the tree to its initial view (fig. 1.9).

Note. Double-clicking on the name of a sketch displays all dimensions associated with that sketch on the screen. Dimensions created by the designer are displayed in black, while dimensions created by the software are shown in blue.

The symbol «+» to the left of the sketch name indicates its over defined, meaning there are redundant dimensions or dependencies (fig. 1.9).

The symbol «–» to the left of the sketch name indicates its under defined, meaning there is a lack of dimensions or dependencies (fig. 1.9).

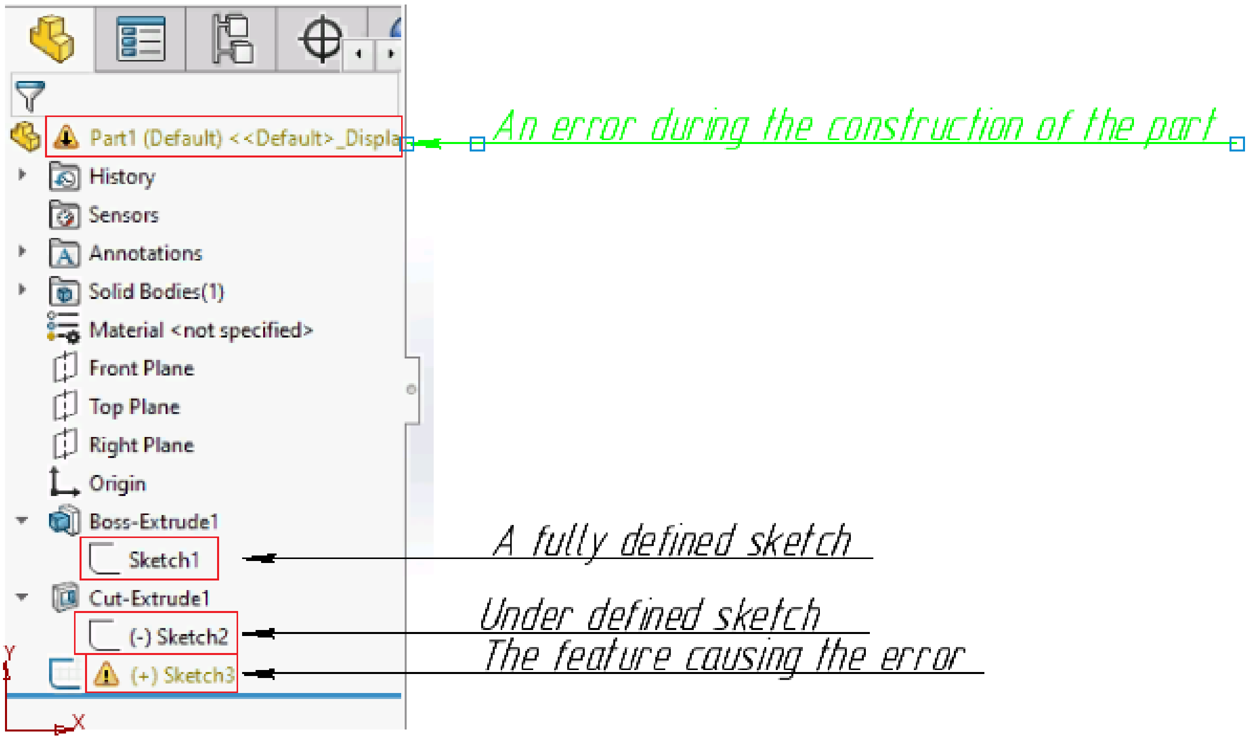


Fig. 1.9 – Conventions in the FeatureManager

The symbol «?» to the left of the name of the sketch indicates that it cannot be solved.

The symbol ⚠ to the left of the name of a part or assembly at the top of the FeatureManager indicates the presence of errors that need to be corrected. (fig. 1.9).

The symbol ⚠ indicates the feature causing the error (fig. 1.9).

The symbol Part2 to the left of the feature name indicates the need for model regeneration, which is performed using the Toolbars .

The row of documents and folders in the FeatureManager are closed by default. Through commands **Options** – **System Options** – **Feature Manager** to manage their visibility, you can choose one of the three commands: **Automatic**, **Hide**, **Show** (fig. 1.10).

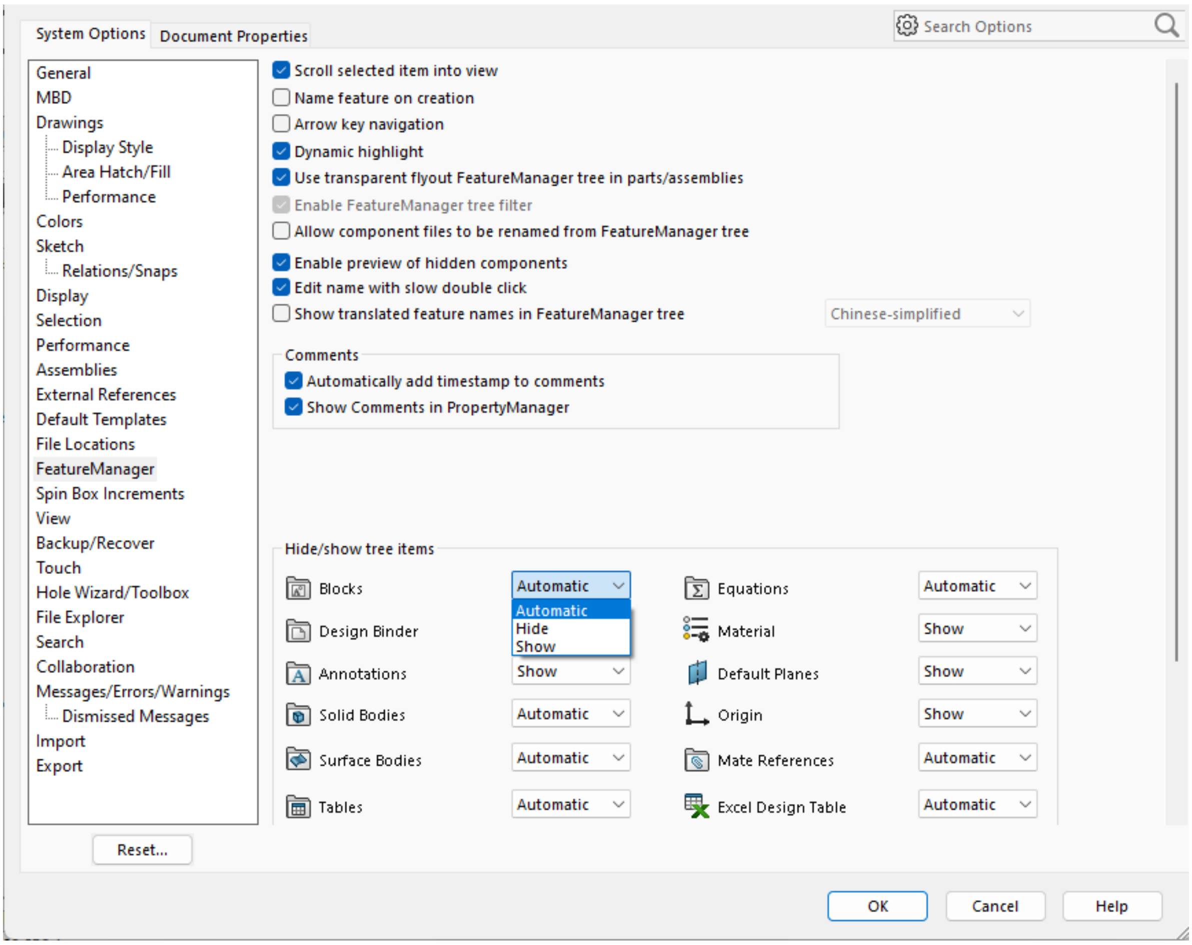


Fig. 1.10 – Setting the visibility of elements in the FeatureManager

PropertyManager

A series of commands in SolidWorks are executed using ***PropertyManager***. The PropertyManager menu occupies the same position as the FeatureManager (fig. 1.11) and replaces it during project execution.

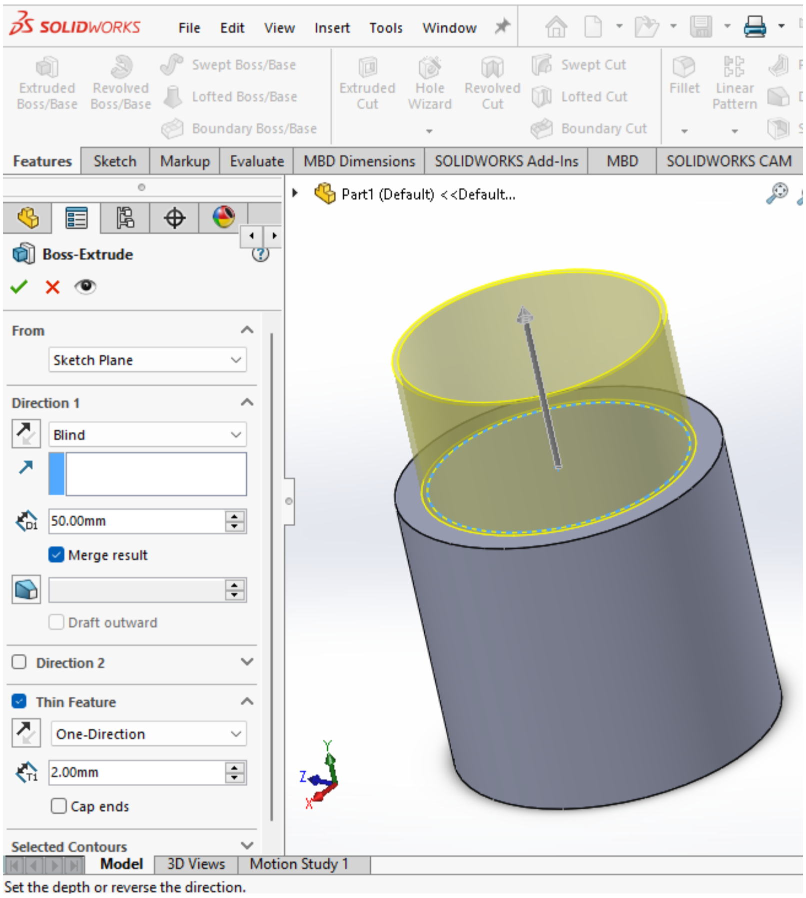


Fig. 1.11 – Property Manager

Shortcut Bars

Customizable shortcut bars let you create your own sets of commands for part, assembly, drawing, and sketch mode. To access the bars, you press a user-defined keyboard shortcut, by default, the **S** key.

CommandManager – the CommandManager is a context-sensitive toolbar that dynamically updates based on the active document type. During part construction, the **CommandManager**, by default, includes the toolbars: **Features** and **Sketch**, in assembly mode, it includes **Assembly** and **Sketch** toolbars. For example, if you click the **Sketch** button, the **CommandManager** will display sketch tools (fig. 1.12).

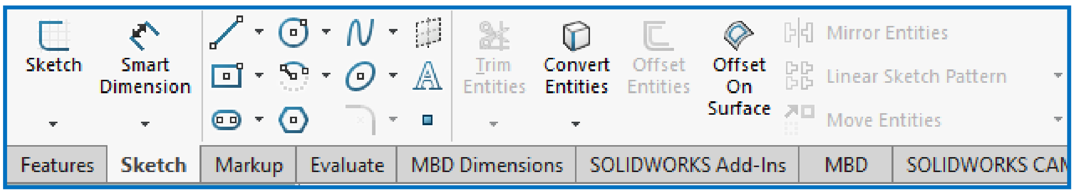


Fig. 1.12 – Sketch tools in the CommandManager

If you click the Features button, the CommandManager will display feature tools (fig. 1.13).

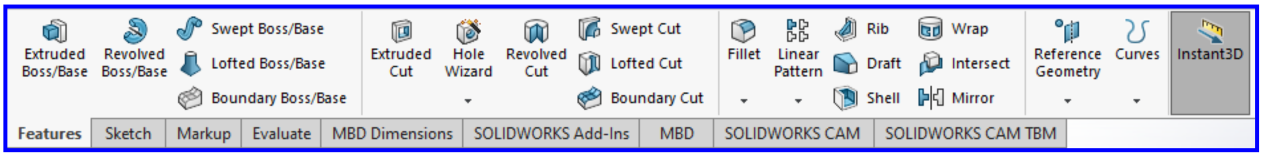


Fig. 1.13 – Features tools in the CommandManager

You can also add or delete tools to customize the CommandManager. Tooltips display when you hover over each icon (fig. 1.14).

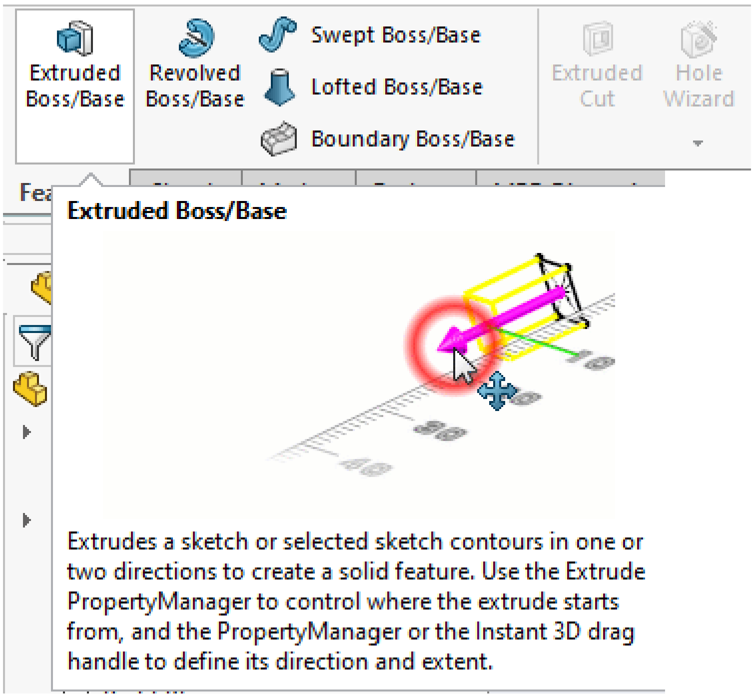


Fig. 1.14 – Tooltips display

Context Toolbars

Context toolbars appear when you select items in the graphics area (fig. 1.15) or FeatureManager design tree (fig. 1.16). They provide access to frequently performed actions for that context. Context toolbars are available for parts, assemblies, and sketches.

Fig. 1.15 – Context toolbars in the graphics area

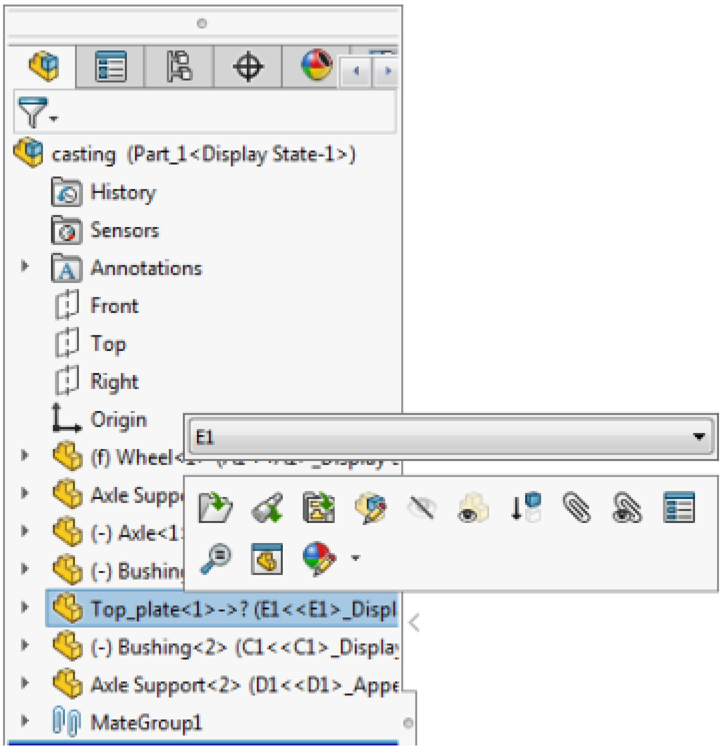


Fig. 1.16 – Context toolbars in the FeatureManager design tree

The mouse manipulator in SolidWorks corresponds to the standard functions of the Microsoft Windows operating system family.

The computer mouse buttons work as follows:

– **left** – selects menu items, entities in the graphics area, and objects in the FeatureManager design tree.;

– **right** – displays the context-sensitive shortcut menus;

– **middle** – rotates, pans, and zooms a part or an assembly, and pans in a drawing.

When working or designing models in SolidWorks, the view of the model is crucial. For this, SolidWorks provides users with a very convenient solution through the Heads-up View Toolbar for interactive viewing. It is located in the upper central part of the SolidWorks graphics area and contains quick access buttons to various tools.

Heads-up View Toolbar includes numerous tools for manipulating views. (fig. 1.17).

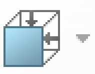
Fig. 1.17 – Heads-up View Toolbar

 **Zoom to Fit**. The tool's name itself speaks to its function. If you are zooming in or out on a specific detail and want to quickly fit the detail completely into the graphics area, simply hover the cursor over this tool  and click on it.

 **Zoom to Area**. Zoom to Area is a very useful tool when you want to inspect a specific region of your model's design. You can view small portions of your detail in an enlarged view. To do this, click and drag a bounding box diagonally to enclose the area to enlarge.

 **Previous.** After moving your model to one or more views, you can return your model or drawing to a previous view. You can undo the last 10 view changes.

 **Section View**. Section View helps easily visualize the section of 3D models. This feature includes a PropertyManager that controls various types of cross-sections according to your needs. You can also use double-section features through this PropertyManager. Section View provides a better overview of the internal features of your design.

 **View Orientation**. View Orientation helps change the current orientation of the model you have developed or the viewing point. It includes various types of view orientations, such as top view, isometric, trimetric, dimetric, left, front, right, back, and bottom views (fig. 1.18).

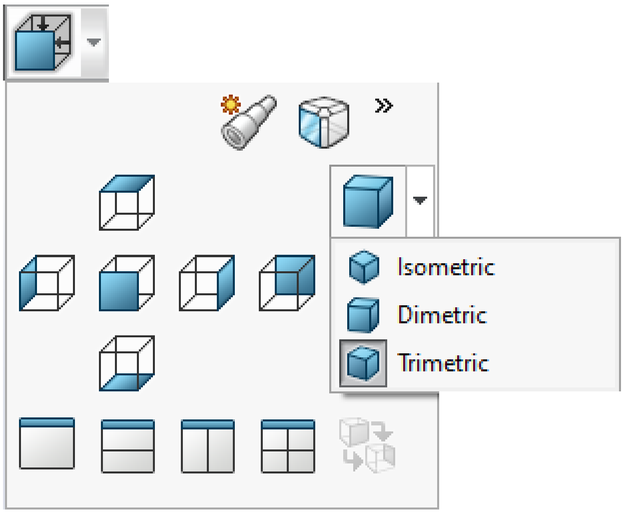
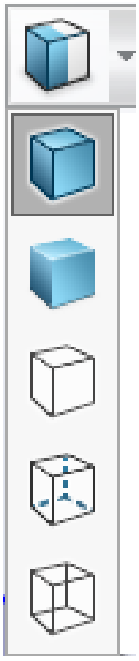


Fig. 1.18 – View Orientation

 **Display style**. It changes the display style of your model, such as wireframe, shaded with edges, shaded, hidden lines removed, and hidden lines visible. You see your model in a different display style after its design (fig. 1.19).



– **Shaded with Edges:** Displays a shaded view of the model with edges visible.

– **Shaded:** Displays a shaded view of the model.

– **Hidden Lines Removed:** Displays the model with all edges that are not visible from the current view angle removed.

– **Hidden Lines Visible:** Displays the model with all edges that are not visible from the current view angle in gray.

– **Wireframe:** Displays all the edges of the model.

Fig. 1.19 – Display style

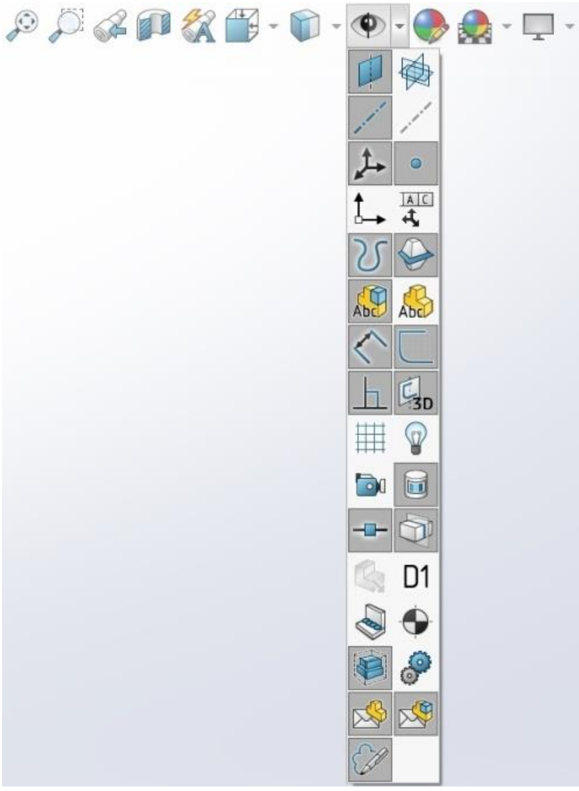
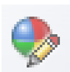


Fig. 1.20 – Show all types of View Tolbar

 **Edit Appearance**. This tool allows you to edit the appearance of the model. It has a color property manager interface that helps select surfaces/planes/parts/elements/bodies. You can change the appearance of a model created using this CAD software by applying different colors to them. Different combinations of colors can be assigned to each part. It also has an advanced section for external appearance in the property manager, allowing you to edit model lighting, surface quality, color/image, and display settings (fig. 1.21).

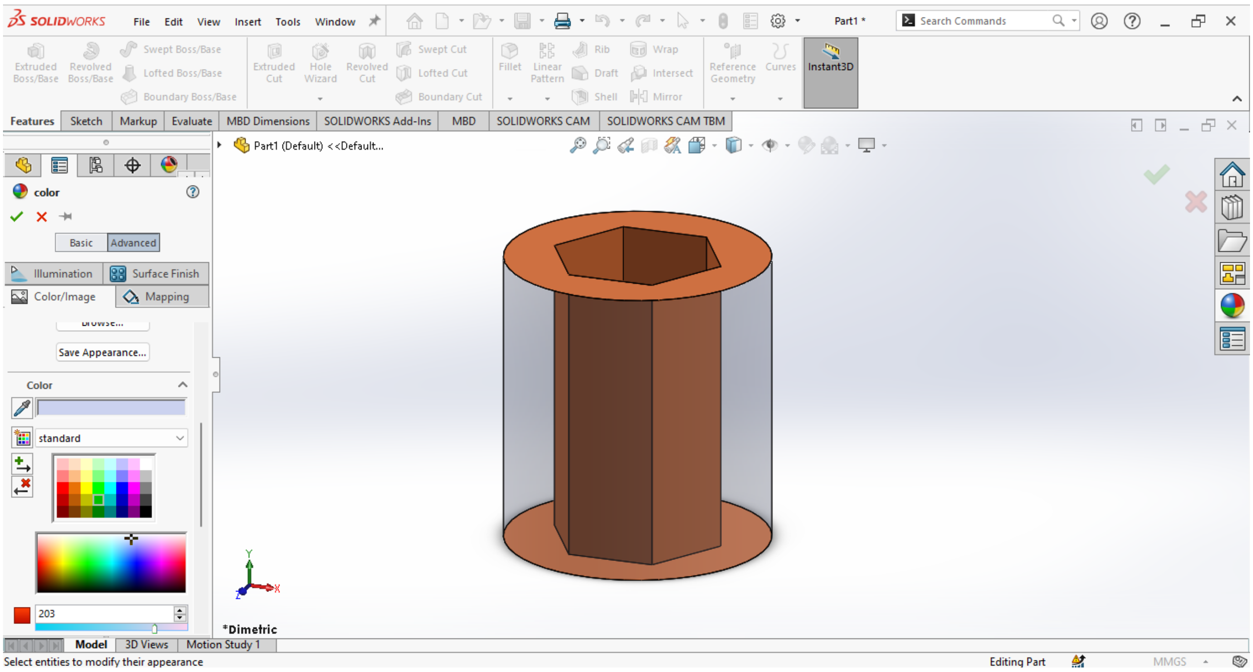
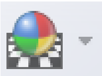


Fig. 1.21 – Model Appearance Editing Tool

 **Apply Scence**. Using this tool, you can add various scenes to your model. When you use the scene view tool, it automatically changes the graphics area scenes accordingly. This feature includes many scenes, and you can add them as you see fit. This only enhances the appearance of your model by changing various backgrounds (fig. 1.22).

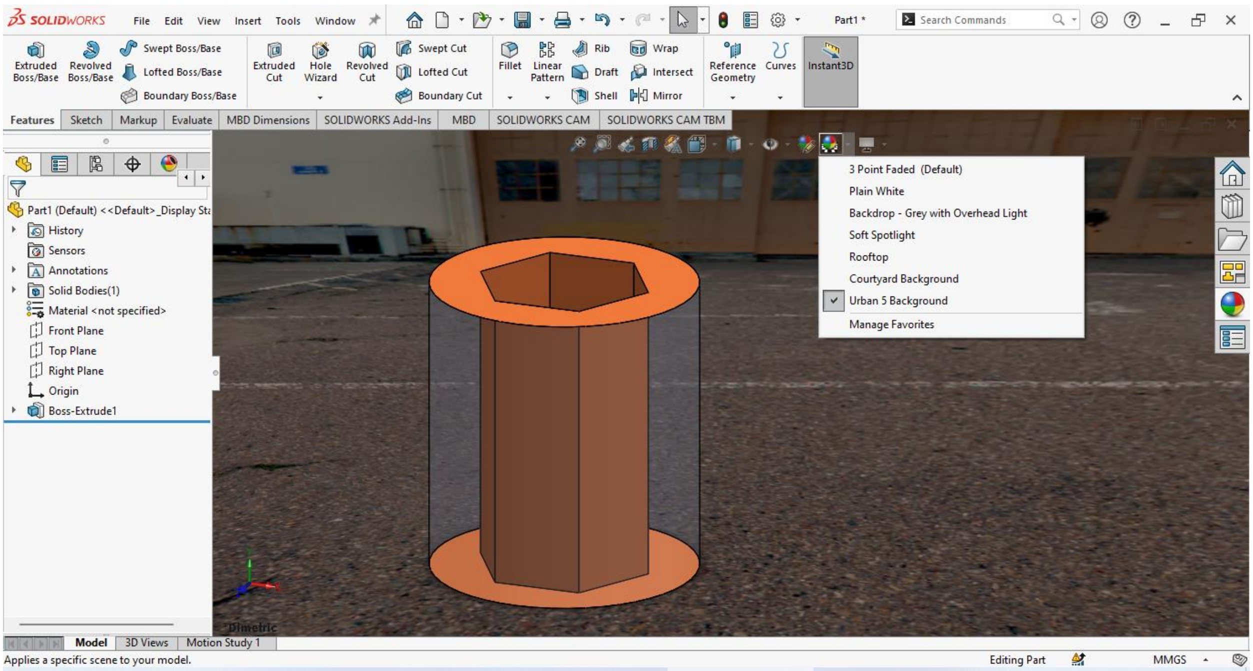


Fig. 1.22 – Apply Scene Tool

View Settings. View Settings mainly contains four viewing parameters such as «Shadow in Shaded Mode», «Ambient Occlusion», «Cartoon», and «Perspective» (fig. 1.23).

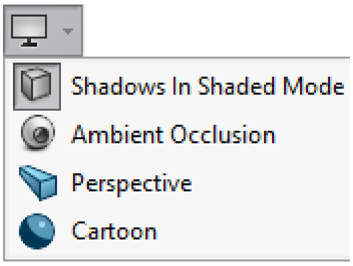


Fig. 1.23 – View Settings Tool

Task Pane

The Task Pane lets you access SolidWorks resources, libraries of reusable design elements, views to drag onto drawing sheets, and other useful items and information.

The Task Pane appears when you open SolidWorks. It contains the following tabs:

– **SolidWorks Resources:** Groups of commands for SolidWorks Tools, Online Resources, and Subscription Services, plus a link to the Welcome dialog box.

– **Design Library**: Access the Design Library, Toolbox, and SolidWorks Content for a wide variety of standard parts, library features, and other reusable content.

– **File Explorer**. Duplicate of File Explorer on your computer, plus an Open in SolidWorks folder.

– **View Palette.** Images of standard views, annotation views, section views, and flat patterns (sheet metal parts) to drag onto a drawing sheet.

– **Appearance, Scence, Decals**. Library of appearances, scenes, and decals.

–  **Custom Properties**. In a customized interface that you create using the Property Tab Builder, enter custom properties in SolidWorks files.

The Task Pane is displayed by default on the right side of the screen and can be opened or closed. In figure 1.24, the Task Pane is shown open with the Design Library tab selected.

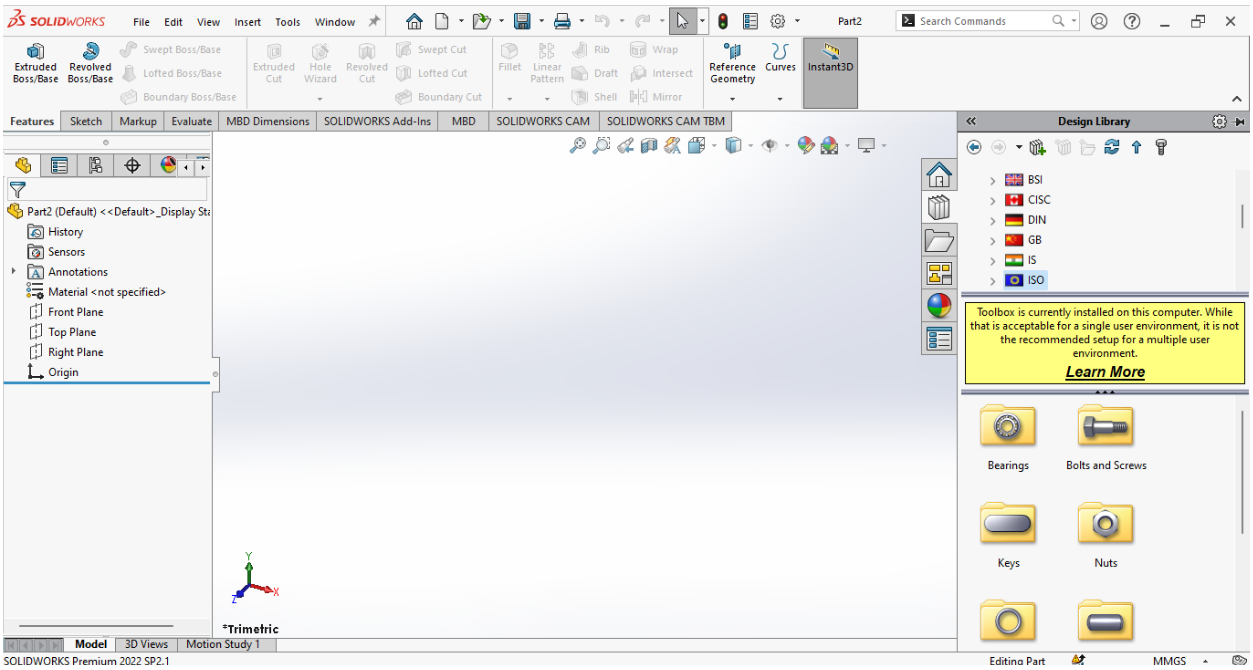


Fig. 1.24 – Task Pane

Status Bar

The status bar at the bottom of the SOLIDWORKS window provides information related to the function that you are performing.

Fig. 1.25 – Status Bar

SolidWorks Settings

After the initial launch of the program, users typically proceed to customize it according to their individual preferences. Most default settings are suitable for many users. However, some settings may need to be adjusted to personal preferences. To access the SolidWorks settings, you need to click on the gear icon  in the menu (fig. 1.26).

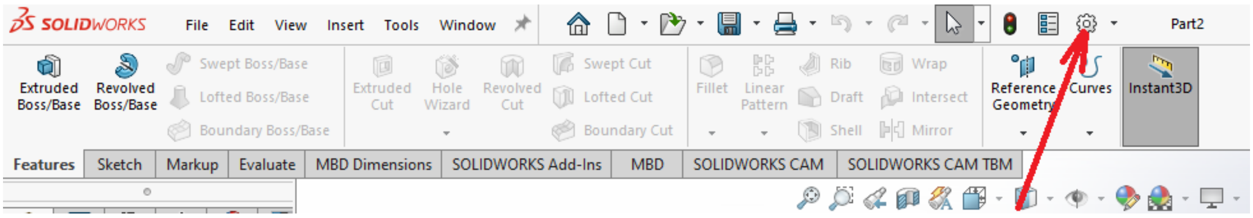


Fig. 1.26 – SolidWorks Settings

After clicking, a dialog box with numerous settings will appear in front of you (fig. 1.27).

It is recommended to configure ***Backup/Recover***. To do this, go to the «*Backup/Recover*» section and make the necessary configurations.

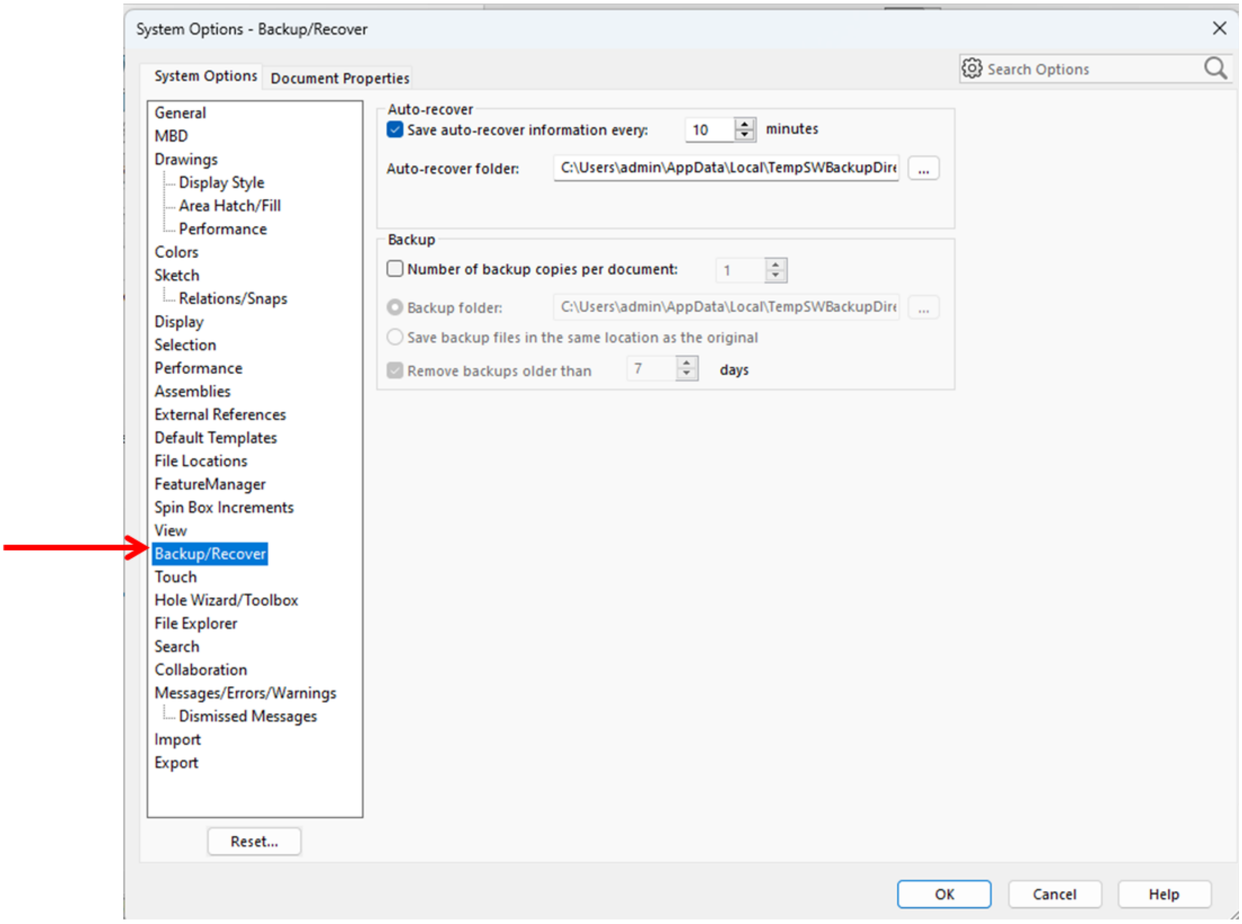


Fig. 1.27 – Settings window

Lecture №2

2D DRAWING, SKETCH CREATION IN SOLIDWORKS ENVIRONMENT

Lecture Objective: Introduction to methods of constructing simple geometric objects based on a sketch, sketch requirements, sequence of sketch execution, methods of dimensioning, and defining relationships between objects in the SolidWorks computer-aided design system.

Sketch

Sketches are the foundation for creating three-dimensional solid models of parts. Therefore, the creation of any part in SolidWorks, no matter how simple or complex, begins with a sketch.

A sketch is a 2D profile or cross section.The main elements of a sketch include points, lines, rectangles, polygons, arcs, and others.

In SolidWorks, sketches are drawn on a plane. By default, when creating a new part, three mutually orthogonal planes passing through the origin are provided. Additionally, you can add any number of planes with the required orientation in space. However, in some cases, it is convenient to use a three-dimensional sketch (in space) when you need to construct a long part, such as a pipe changing its direction in space, or a welded structure consisting of profiles with a specific cross-section.

All sketches, both two-dimensional and three-dimensional, are constructed on three mutually orthogonal planes (fig. 2.1). To create a sketch, you need to choose the plane on which it will be built: **Front**, **Top**, or **Right**. The initial selection of one plane or another is not crucial. A sketch can also be created on a planar surface or on the face of a solid object.

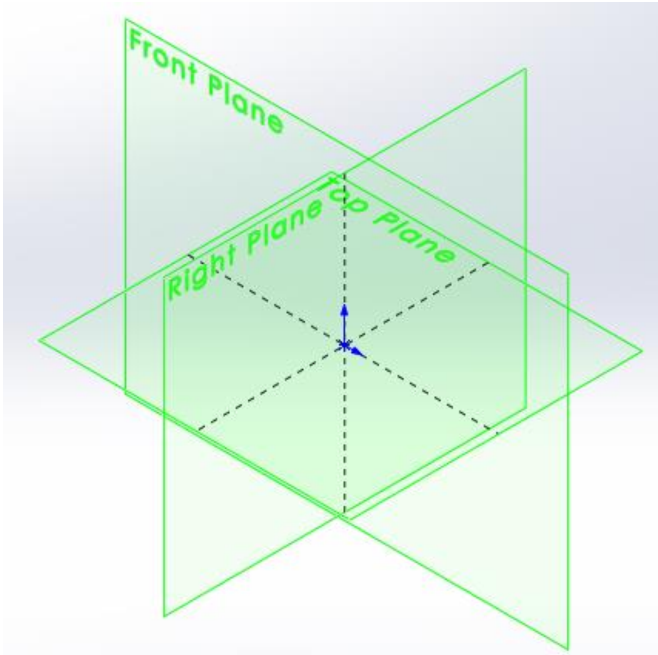
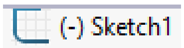


Fig. 2.1 – Orthogonal planes

Switch to sketch mode

After selecting the working plane using the button , you enter the two-dimensional sketch mode, where you create the outline of the object.

The sketch mode can be identified by several signs:

1. In the upper right corner of the graphics area, the symbol appears ;
2. In the FeatureManager a new object is listed: 

;
3. The CommandManager with the Sketch toolbar active;
4. In the graphics area, the sketch origin  is displayed in red in an open sketch. Use the sketch origin to help you understand the coordinates of the sketch.

Every sketch in the part has its own origin, so there usually are multiple sketch origins in a part. When a sketch is open, you cannot turn off the display of its origin. It is recommended to start drawing any sketch from this point so that sketch elements are automatically constrained to it, and no additional relationships are needed to fully define the sketches.

For orientation of the sketch plane in space, the triad symbol (coordinate system) is always present on the screen in the graphics area.

Creating a Sketch

The first created three-dimensional element of any part in SolidWorks is called the "Base" and serves as the blank canvas. In the construction process, new elements are sequentially added to it: protrusions, cuts, fillets, chamfers, bends, stiffening ribs, etc. Material can be removed from the blank or added to create new features.

At the initial stage of creating the base, its sketch is drawn – a two-dimensional representation of a characteristic cross-section from which the volume will be formed later.

A sketch is created using the buttons on the Sketch Tools toolbar (fig. 2.2).

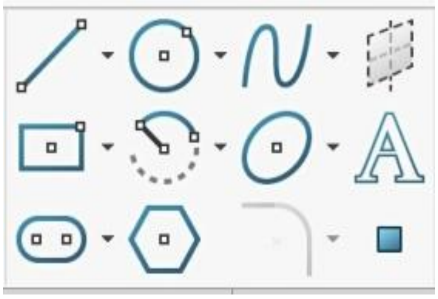
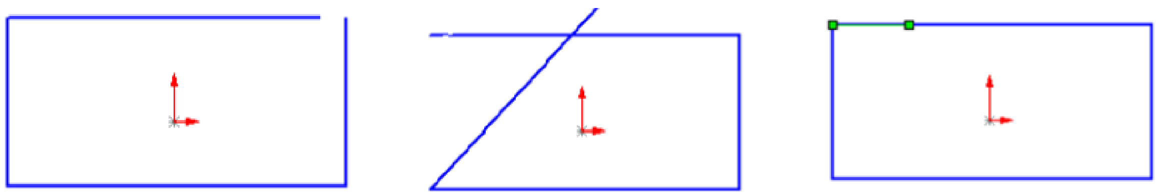


Fig. 2.2 – The buttons on the Sketch Tools toolbar

You should not attempt to create a complex sketch right away, as it will complicate the editing process. It's more rational to add or remove additional elements while working with the volumetric representation.

Sketch requirements

1. When constructing a sketch, ensure that the sketch has a closed contour (fig. 2.3, a), there are no intersections between sketch elements (fig. 2.3, b), and there are no overlapping elements (such as surfaces drawn over others) (fig. 2.3, c).



a b c

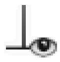
Fig. 2.3. Sketch requirements

2. SolidWorks allows for the presence of multiple closed contours in a single sketch, forming a multi-body part. In this case, when extruding the sketch, the program will prompt you to specify the material location within the contours.
3. If the sketch does not consist of a closed contour, then during extrusion, the program will interpret the sketch as a thin-walled feature and prompt you to specify its thickness.
4. When creating sketches, you can cut and paste or copy and paste one or more sketch entities, both between sketches and within a single sketch.
5. During the sketching process, inferencing lines appear, working in conjunction with pointers, constraints, and relations to graphically represent how sketch entities influence each other. Inferencing lines are dotted lines that appear as you sketch. When your pointer approaches highlighted cues such as midpoints, the inferencing lines guide you relative to existing sketch entities.
6. Two-dimensional sketches can only be created in planes or on existing faces of a part, while three-dimensional sketches can be created in three-dimensional space.

The next stage in the sketching process is specifying dimensions and constraints.

Dimensions are added using the universal button  in the «Sketch» panel for all dimensions.

Sketching is simplified and accelerated by applying location and shape constraints to elements. Clicking the button  in the «Sketch Relations» panel opens the «Add Geometric Relations» dialog box.

In SolidWorks, it is not mandatory for a sketch to be fully defined using dimensions and constraints. The sketch can be defined in the subsequent work process. However, this may lead to errors in later stages. Adding dimensions and constraints to the sketch is not obligatory but is a useful practice. It is essential to consider that dimensions and constraints should not duplicate each other, as it will result in numerous errors during further reconstruction. The button  is used to display and edit existing constraints.

A fully defined sketch helps avoid errors in work, especially when significant changes need to be made to the model later on.

In SolidWorks, the degree of sketch definition is typically indicated by color-coding, with the values provided in table 2.1.

Sketches can be in one of the following five states:

Table 2.1 Conventional notations for the state of the sketch

Sketch color	Value
Black	**A fully defined sketch**. All lines and curves in the sketch, as well as their placements, are described using dimensions and relations. This is an optimal state for the sketch. It means that all dimensions and relations are specified correctly and in sufficient quantity.
Red	**Over Defined** (error). There are conflicting or duplicate dimensions or relations.
Blue	**Under Defined**. Some dimensions and/or relations are not defined in the sketch, and they can be modified.
Pink	**Unsolved sketch.** A solution has not been found. It is impossible to create the feature. Displayed geometry, relations, and dimensions interfere with sketch calculation (for example, elements to which there are references have been deleted).
Yellow	**Resolved sketch.** Indicates sketch entities that are invalid, creating a sketch without resolution in its current state. Requires deleting some relations or dimensions, or returning the sketch entity to its prior state.

It is not recommended to reconfigure the standard sketch colors.

The sketch state is displayed in the <Status Bar> when the sketch is in edit mode.

Sketch Elements

Forming Elements

When constructing flat sketch objects (lines, arcs, polygons, etc.), so-called **Forming Elements** are used: inferencing lines, pointers, sketch snaps, and relations. Forming elements dynamically illustrate how sketch elements influence each other.

Inferencing lines are dotted lines that appear as you sketch, displaying relations between the pointer and existing sketch entities (or model geometry). When your pointer approaches highlighted cues such as midpoints, the inferencing lines guide you relative to existing sketch entities. (fig. 2.4).

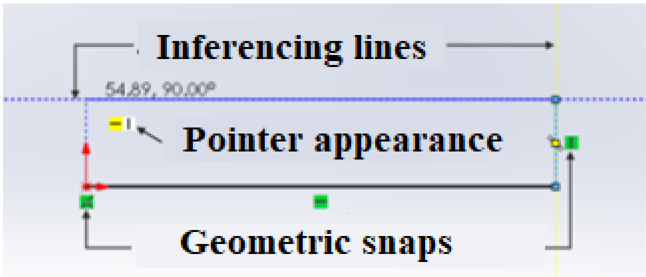
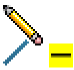


Fig. 2.4 – Elements of forming sketch objects

When constructing objects, the pointer appearance changes depending on the selected drawing tool (arc, circle, line), as well as when the pointer is on a geometric relation (intersection, point) or a dimension. If the pointer displays a relation (such as  for a horizontal relation), and you click to accept the sketch entity while the relation is displayed, the relation is added automatically to the entity.

Sketch Snaps are on by default. As you sketch, Sketch Snaps icons are displayed.

In addition to **Sketch Snaps**, you can display icons that represent **Relations** between sketch entities. As you sketch, the entities display icons that represent

Sketch Snaps; once you click to indicate a sketch entity is done, relations are displayed.

Objects for constructing a flat sketch

Objects for constructing a sketch are located on the Sketch toolbar or can be activated through the top menu by selecting Tools >> Sketch Entities. All properties of objects are divided into three groups: type (line, arc, circle, ellipse), relations (horizontal, vertical), and geometric parameters (coordinates, length, angle, diameter). Properties are displayed in the PropertyManager when constructing objects.

Among the fundamental flat objects used in SolidWorks sketches are:

– lines;

– rectangles;

– circles;

– arcs;

– polygons;

– complex curves and shapes (ellipses, parabolas, splines, etc.).

There are two modes for sketching in 2D: click-drag or click-click.:

1. Mode «Click-drag» – the design of the object begins by clicking on the first point and dragging it further without releasing the mouse button, and it is completed when the button is releaseddrag.

2. Mode «click-click» – the design an object that begins and ends with a mouse click, and the object is designed by moving between these two clicks.

The most versatile and commonly used element for designing flat objects in SolidWorks is the ***Line***. When using the ***Line*** object in the "click-click" mode, a chain of segments is created – a polyline. To terminate a sketch chain, do one of the following:

- Double-click to terminate the chain of entities and leave the tool active.
- Right-click and select ***End chain***, which is the same as double-clicking.
- Press ***Esc*** to terminate the chain and release the tool.

Additionally, in SolidWorks, there is the capability to transition from sketching a line to sketching a tangent arc , and vice versa, without selecting the arc tool. To achieve this, while drawing a new segment of the polyline from the endpoint of the previous segment, move the mouse cursor away and then return to the endpoint. During further construction, a dynamic tangent arc is formed (fig. 2.5). The automatic transition from Line to Tangent Arc is also performed by pressing **the letter A** on the keyboard.

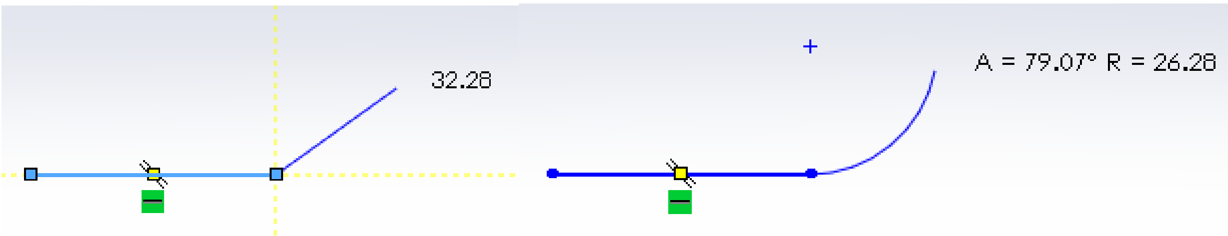
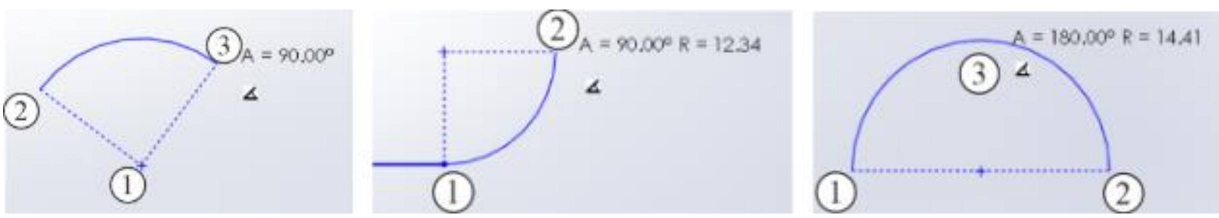


Fig. 2.5 − Automatic transition from a Line to a Tangent Arc

To construct an arc in SolidWorks, one of the three tools can be used:

1. **Centerpoint Arc:** Specify the coordinates of the center and one of the end points, and then fix the arc angle (the third point) (fig. 2.6, a).
2. **Tangent Arc:** This command can be applied to the endpoint of an existing object (fig. 2.6, b). After its execution, a Tangent relation is automatically established between the objects.
3. **Three Point Arc:** Requires specifying two end points of the arc, and by moving the third point, the radius value is set (fig. 2.6, c).



a b c

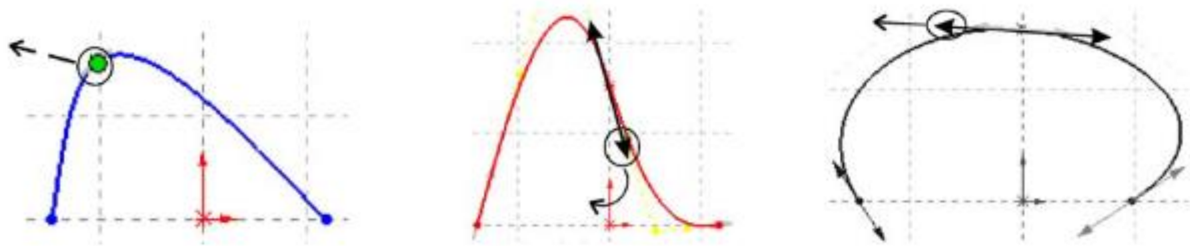
Fig. 2.6 – Methods of constructing arcs

For a large number of practical tasks, it is necessary to construct a smooth curve passing through specified points. For these purposes, splines are used.

Splines are the primary tool in SolidWorks for constructing complex geometry of sketches and are applied in the development of design projects for original casings. Splines can also be used as an "approximating curve" in engineering tasks where the trajectory of geometry change is defined by a certain mathematical law.

In SolidWorks, a Spline is used, and its curvature is controlled by the distribution of control points. The shape of the spline can be manipulated in three ways:

- By moving a node (fig. 2.7, a);
- By changing the **Radial Tangent Direction** – the angle of inclination to the coordinate axis (fig. 2.7, b);
- By changing the **Tangent Magnitude** – the radius of curvature at a specific point (fig. 2.7, c).



a b c

Fig. 2.7. Methods of controlling the shape of a spline

The construction of circles, rectangles, and polygons in sketches corresponds to most Computer-Aided Design (CAD) systems for working with two-dimensional vector graphics. A detailed description of the rules for construction and working with sketch objects is provided in [1], as well as in the SolidWorks help system.

Dimensioning a 2D Sketch

Geometric objects constructed in a sketch must be defined in space. In the end, the coordinates of points for respective objects (lines, arcs, circles) must be specified. In **Sketch** mode, the position of objects is described mathematically by applying dimensional constraints or by imposing constraints on the location of objects.

In SolidWorks, each dimensional constraint corresponds to a separate variable. After defining the necessary set of parameters for a sketch object, all constructed elements (lines, arcs, circles, splines) can be represented as a system of equations. The program automatically rebuilds the object according to the specified dimensional constraint value (all changes are reflected in the graphics area).

Using the **Smart Dimension** tool  on the Sketch toolbar, you can apply dimensions to objects in the sketch. To execute the command, you need to first select one or two objects (lines, points, arcs, circles) and specify the position of the dimension line (fig. 2.8). If the lines are not parallel, the system will automatically determine the angular dimension. Similarly, diametrical and radial dimensions are determined.

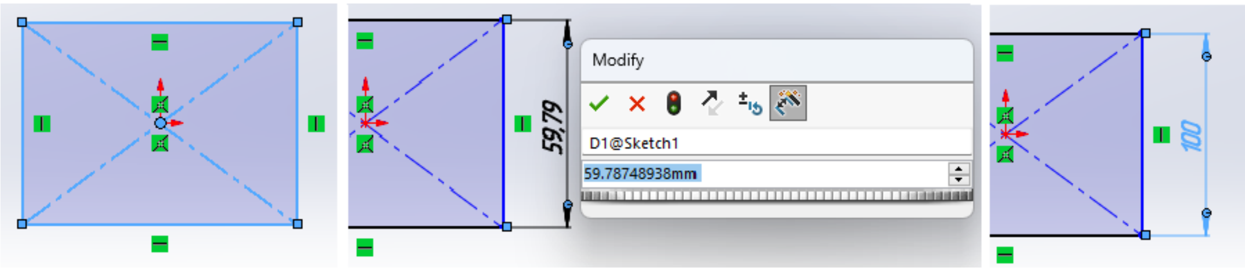


Fig. 2.8. Dimensioning objects on a flat sketch

Dimensions can be set either relative to existing objects within the sketch, including the Origin Point and auxiliary axes, or with respect to already constructed three-dimensional elements and their sketches.

Once the objects are selected and the dimension line's location is determined, the Modify dialog window will appear. By entering a new value in this dialog window, the dimension can be changed (fig. 2.8).

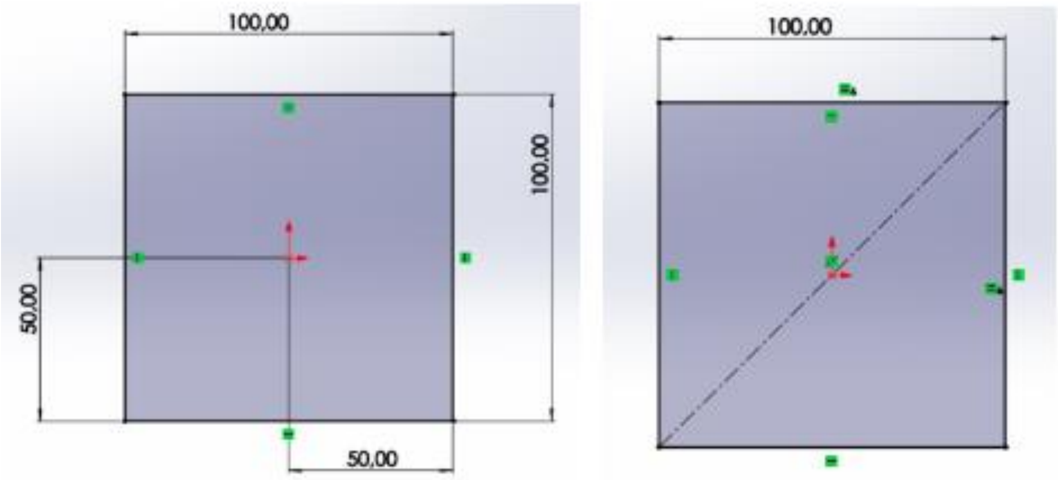
It should be noted that dimension lines of control dimensions placed on sketches are not mandatory dimension lines for future drawings, although they may be transferred there automatically. Dimensioning in Sketch constitutes the parametrization of geometric objects, whereas dimensions on drawings are established according to the requirements of the standards of engineering documentation (EDS).

Interrelations

Interrelations – are constraints on the placement of flat objects in a sketch. The main purpose of adding interrelations is to reduce the number of controlling dimensions. Figure 2.9 shows two options for defining a square in the sketch, the center of which coincides with the **Origin** of the sketch.

In the first case (fig. 2.8, a), the construction of the **Rectangle** object was performed with subsequent assignment of the necessary dimensions, including the dimensions from the sides to the **Origin** (a total of four dimensions).

In the second case (fig. 2.8, b), besides the rectangle, its diagonal was constructed using an auxiliary **Centerline**. Two interrelations were added: **Equal** of adjacent sides of the rectangle and **Midpoint** for the diagonal of the rectangle and the **Origin.** To fully define the square, it is enough to specify one dimension - the length of the side.



a b

Fig. 2.9. Ways of defining a square in a sketch

To add a relation, you should activate the "**Add Relations**" command on the **Sketch** toolbar or directly select the necessary object or objects in the sketch (selecting multiple objects is done by holding down the ***Ctrl*** key on the keyboard).

The system autonomously determines permissible relations for the selected objects and suggests selecting one of them. In the ***Property Manager*** window, you should click on the corresponding icon (**Coincident, Horizontal, Fixed**, etc.).

The specified relative positions of objects cannot be changed until the relations are removed. To delete relations, you need to select the object (line or point), and in the list of **Existing Relations** in the **Property Manager** (fig. 2.10), delete the corresponding relation by pressing the *Delete* key on the keyboard.

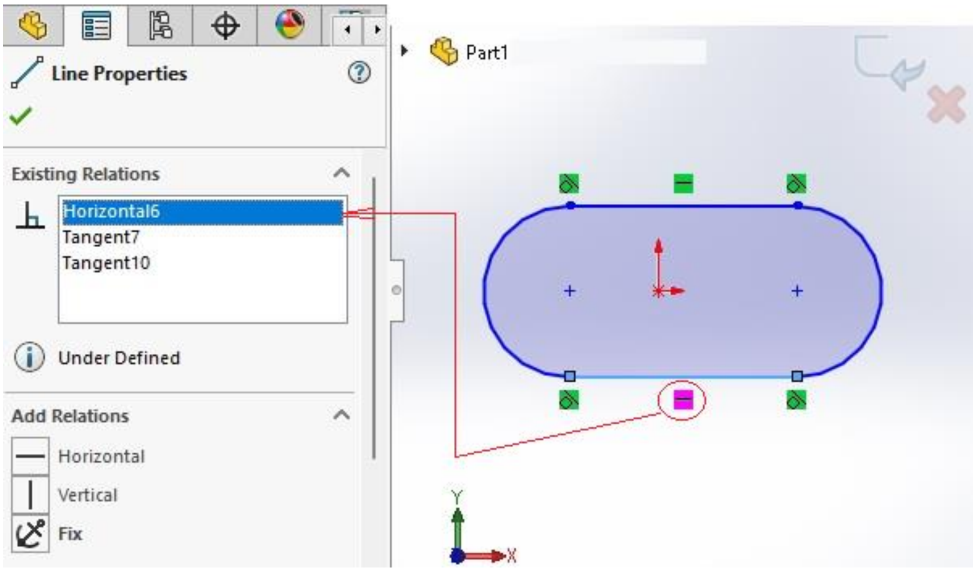


Fig. 2.10. Displaying existing relations of a sketch object

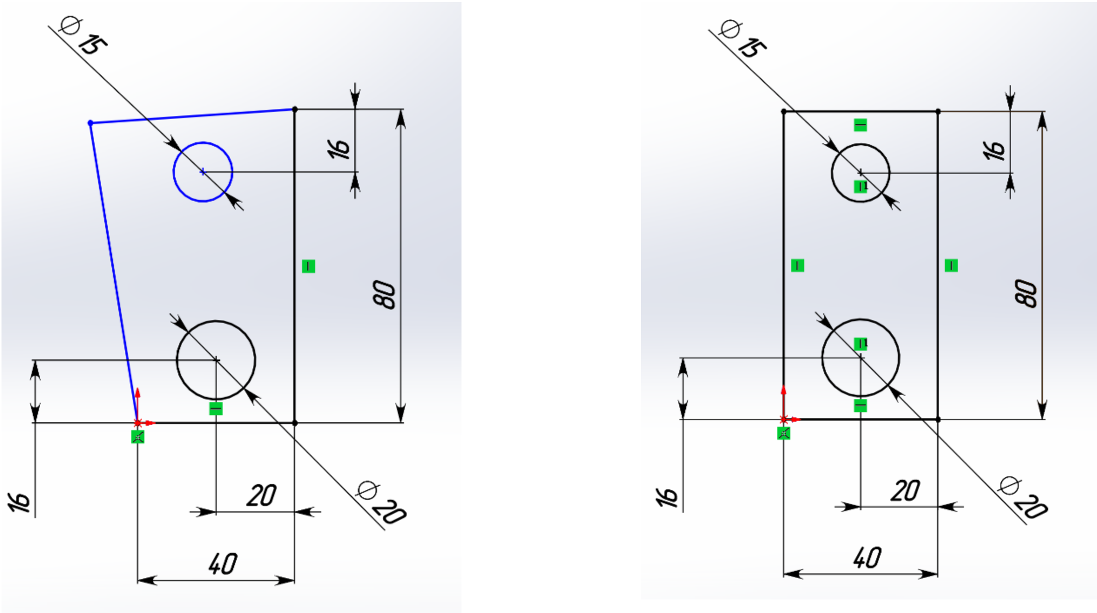
Table 2.2 Geometric Relations in Sketches

Interrelation	Selected objects	Resulting relations
Horizontal Vertical	One or multiple lines, or two or several points	Lines become horizontal or vertical (as determined by the current coordinate system of the sketch). Points are aligned horizontally or vertically.
Collinearity	Two or more line	The elements lie on the same infinite line.
Coaxiality	Two or more arcs	The elements use the same center and radius.

Interrelation	Selected objects	Resulting relations
Perpendicularity	Two lines	The two elements are perpendicular to each other.
Parallelism	Two or more lines. Line and plane (or flat face) in a three-dimensional sketch.	The elements are parallel to each other.
Parallel YZ	Line and plane (or flat face) in a three-dimensional sketch.	The line is parallel to the YZ plane with respect to the selected plane.
Parallel ZX	Line and plane (or flat face) in a three-dimensional sketch.	The line is parallel to the ZX plane with respect to the selected plane.
Along the Z-axis	Line and plane (or flat face) in a three-dimensional sketch.	The line is perpendicular to the face of the selected plane.
Tangency	Arc, ellipse, or spline, and a line or arc.	The two elements remain tangent to each other.
Concentricity	Two or more arcs and a line or arcs	The same center is used for the arcs.
Midpoint	Two lines or a point and a line	The point remains at the center of the line.
Intersection	Two lines or a point and a line.	The point remains at the intersection of two lines..
Coincidence	Point and line, arc or ellipse.	The point lies on the line, arc, or ellipse.
Equality	Two or more lines, or two or more arcs.	The lengths of lines or radii remain equal
Symmetry	Centerline and two	The elements remain at an equal

Interrelation	Selected objects	Resulting relations
	points, lines, arcs, or two ellipses.	distance from the centerline, on a line perpendicular to it.
Fixation	Any object.	The dimensions and position of the object are fixed.
Point of intersection	Sketch point and any axis, edge, line, or spline.	The sketch point coincides with the location where the axis, edge, or curve intersects the sketch plane. The "Point of Intersection" relation is used in trajectory elements with guide curves.
Merge points	Two sketch points or endpoints.	Two sketch points merge into one point.

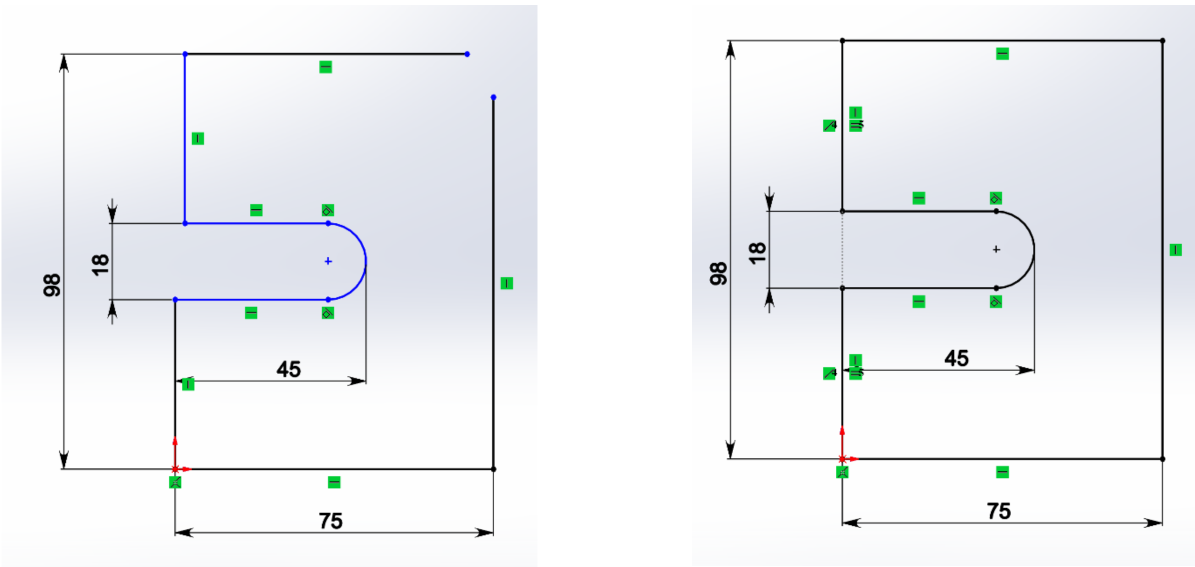
Example of adding the «Horizontal» and «Vertical» relations (fig 2.11). To apply a relation, you need to select the object (in this case, a line). Note that the program highlights the relation in bold, indicating its priority. Thus, for one line, shown in blue, we add the "Horizontal" relation, and for the other line, we add the «Vertical» relation. To fully define the upper circle, we need to add the «Vertical» relation, but before that, we need to select the centers of both circles by holding down the ***Ctrl*** key. As we can see, the «Horizontal» and «Vertical» relations can be added not only for lines but also for multiple points.



a b

Fig. 2.11. Adding the «Horizontal» and «Vertical» relations

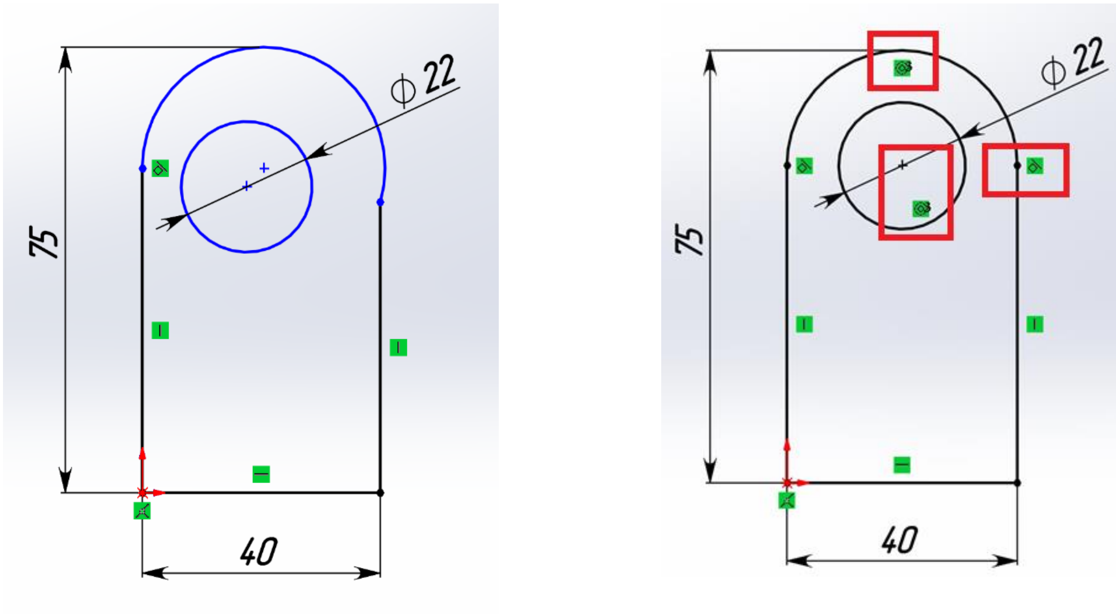
Example of adding the «Merge» , «Collinearity» , and «Equality» relations (fig. 2.12). Looking at the next sketch, we see that the sketch is open in the top right corner. To close it, we select two points and add the «Merge» relation. To determine the position of the top edge, we select it, hold down *Ctrl*, select the bottom line, and add the «Collinearity» relation. To position the hole in the middle of the height, we add the condition of equality of two lines «Equality» .



a б

Fig. 2.12 Adding the «Merge», «Collinearity», and «Equality» relations.

Example of adding the «Tangency» and «Concentricity» relations (fig. 2.13).



а б

Fig. 2.13. Adding the «Tangency» and «Concentricity» relations.

A large number of relations significantly complicates the error correction process, as it requires removing unnecessary relations. To avoid cluttering the drawn objects with relation icons, it is recommended to disable them (top menu **View** >> **Sketch Relations**).

Useful tips for creating sketches

1. Try to use only fully defined sketches.
2. If the sketch is symmetrical, position it symmetrically about the **Sketch Origin**.
3. When creating sketches for rotational parts, use diameter dimensions.
4. When creating part models from existing drawings, add dimensions to the sketches as specified in the drawing.
5. Avoid using sketches containing self-intersecting profiles, as they complicate further model modifications and understanding by other developers.

Completing work with the sketch

Upon completing work in sketching mode, press the «**Exit Sketch**» button  in the **Sketch** toolbar or use the **Exit Angle** icon located in the top right corner of the construction area. To accept all changes and exit the sketch, click the checkmark icon  in **the Exit Angle**. To discard changes and exit, click the cross icon . Tooltips will appear when hovering over these icons for clarification.

Lecture №3

FUNDAMENTALS OF PART MODELING IN SOLIDWORKS ENVIRONMENT

Lecture Objective: To study the basic methods of constructing three-dimensional models of parts in the SolidWorks computer-aided design system.

In SolidWorks, a part refers to a three-dimensional object composed of a certain number of elements. Elements are individual geometric shapes that, when combined, form the part. The primary formative elements – extrudes and cuts – are built based on 2D sketches. Other elements – such as shells, fillets, and chamfer – modify an existing 3D model.

Creating a Part Model

The process of creating a part model consists of five main steps that are repeated regularly until the necessary model is fully created:

1. Selecting the work plane for sketch creation.
2. Switching to sketch mode – button  .
3. Drawing the sketch using the tools on the corresponding panel.
4. Adding dimensions and/or constraints – buttons  and  .
5. Creating a three-dimensional feature using the tools on the «**Features**» panel.

Creating a Three-Dimensional Feature

The methodology of sketch creation was discussed in the previous lecture.

After completing the sketch, you proceed to form the volume of the model: obtaining the base on the first stage and creating a **Boss** or **Cut** on subsequent stages. For working with the volume of the model, there are tools available on the «**Features**» panel.

There are four methods for obtaining a solid model from a sketch.

Elements can be created by:

- Extrusion **(Extruded Boss/Base**) – button .
- Revolution around an axis (**Revolved Boss/Base**) – button .
- Moving the created sketch along a trajectory (**Swept Boss/Base**) – button .
- Transition between cross-sections of different shapes located in space (**Lofted Boss/Base**) – button .

During further work, you can add material to the created blank, perform various holes, cuts, fillets, etc. The program also allows for various design and technical calculations of the constructed models.

Moreover, arrays of elements - linear and circular, as well as mirrored copies of elements, can be used for creating solid geometry elements.

Additionally, in SolidWorks, operations for transforming three-dimensional geometry of a part are implemented: Deformation, Scaling, Bending, and others. These operations are performed on individual elements and replace the process of creating complex geometry.

Another additional type of operations are Boolean operations. They are performed with two or more elements and are necessary for combining elements into a single object using logical operations or solid body compositions.

Solid models in SolidWorks can be created using any of the mentioned methods. The final result will not depend on the chosen method; however, for better understanding and ease of editing, it is recommended to create the model in a way that mirrors the manufacturing process. For example, if the shaft processing is expected to be done using turning operations, then the model should be created using contour revolution methods.

Methods of constructing three-dimensional details:

1. **Extrusion** is the simplest method of forming a solid body, based on pulling a flat sketch in one or simultaneously in two directions. This command is activated by the «Extrude Boss/Base» button .

The function of the command is to fill the volume described by the contour of the sketch with virtual material of the solid body during its parallel linear displacement. Extrusion of a sketch can also be done at an angle, forming a solid body in the shape of a cone or pyramid. When extruding, it is also possible to create a thin-walled detail by specifying this when executing the command and setting the wall thickness.

As the initial condition for creating an element, the following can be specified (fig. 3.1):

– Sketch plane;

– Surface, edge, plane, or vertex of a 3D model;

– Offset.

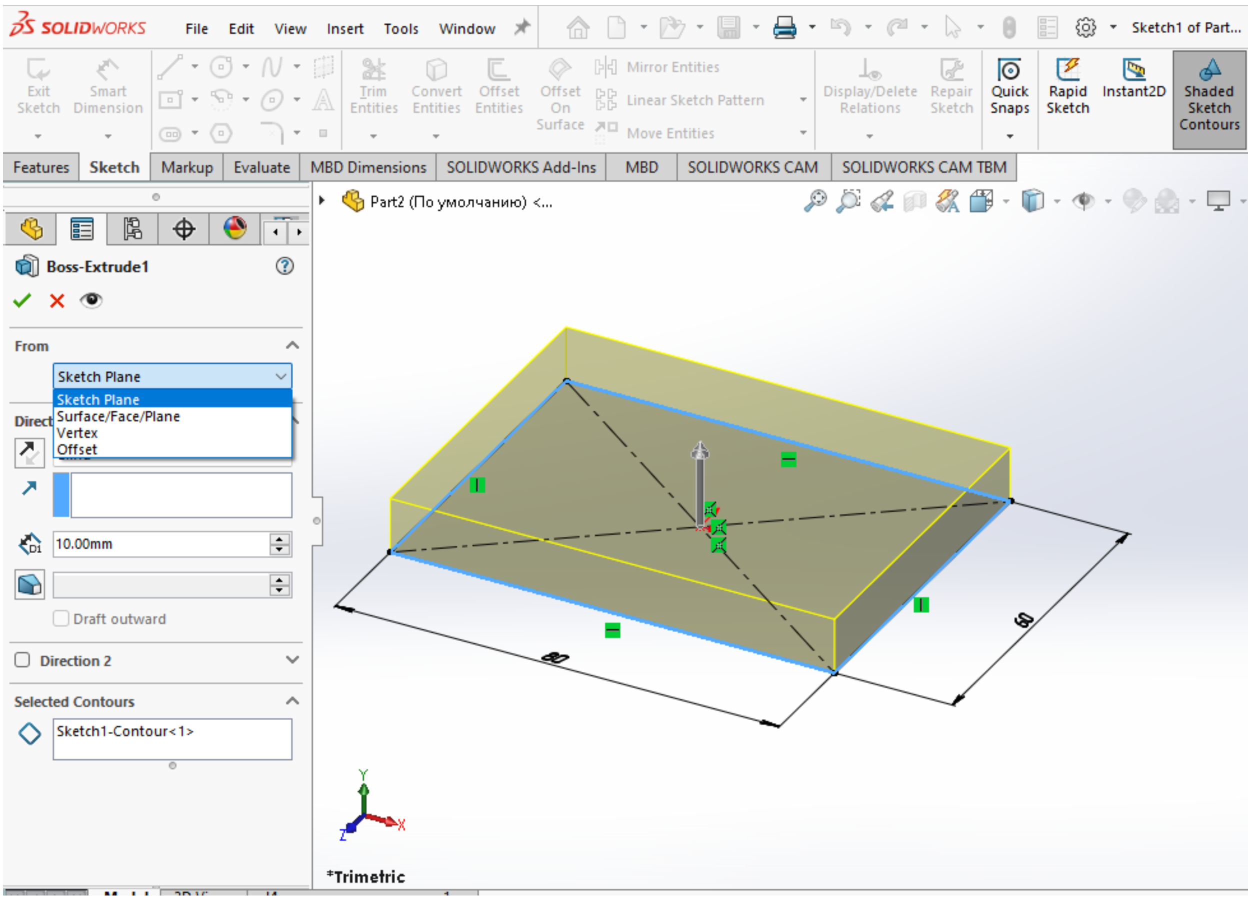


Fig. 3.1. Setting parameters to create an element

In the first case, the solid will be constructed from the plane on which the sketch is located; in the second, from the selected geometric element; and in the third, from a reference surface offset parallel to the sketch plane by a specified distance. When selecting surfaces, faces, or planes as the initial condition, the contour of the **Extrude feature** must be fully contained within their boundaries.

Boundary conditions are used to define the limits of the extruded element.

If we imagine that the extrusion operation is performed by moving the sketch along a directed line segment, then its initial conditions will play the role of its first point, and the boundary conditions – the role of its second point. There are a total of six conditions that should accept either numerical dimension values or geometric objects as input:

1. To a specified distance (**Blind**) – defines the limit of the extruded element by explicitly specifying the extrusion depth (the value can be specified numerically (fig. 3.2) or by directly dragging the arrow with the mouse (fig. 3.3).

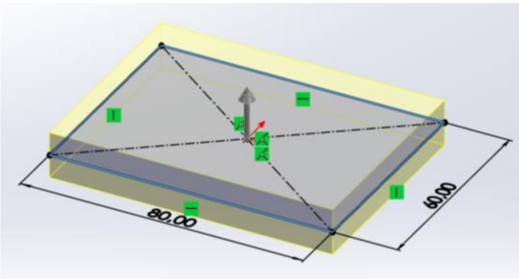


Fig. 3.2

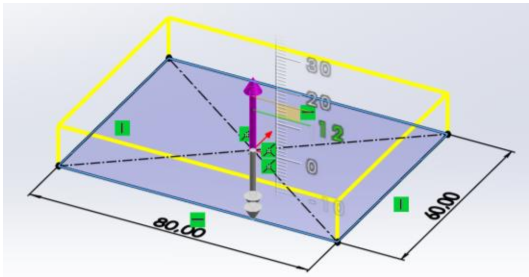


Fig. 3.3

2. **Up to Vertex** – the sketch is extruded up to a plane passing through the specified vertex.
3. **Up to Surface** – the element fills the space from the sketch plane to the selected surface.
4. **Offset from Surface** – the element fills the space from the sketch plane to a surface that is equidistant from the selected one.
5. **Up to Body** – the element is constructed from the sketch plane to the specified body (used in multi-body parts, assemblies, molds).

6. **Mid Plane** – the element is created by extruding the sketch to an equal depth in both directions from the sketch plane (fig. 3.2).

To change the direction of the extrusion to the opposite, you can click on the «**Reverse Direction**» button , located to the left of the list.

If there is a draft angle, the resulting element will have a tapering or widening effect (instead of a cylinder, you will get a cone, instead of a cuboid, you will get a pyramid).

To perform the modification, simply press the «**Enable/Disable Draft**» button, specify the angle and direction of tapering (inside/outside).

Thin Feature. A thin feature can be created based on both closed and open sketches. This operation requires specifying the direction of the sketch offset (inside or outside) to create a cavity inside the feature, as well as the magnitude of the offset in each direction. The method for determining the thickness is set in the Type dropdown menu: one-directional (used to add thickness from one side of the sketch), mid-plane (equal thickness in both directions), or two-directional (different thicknesses on both sides of the sketch).

2. **Revolve** is another widely used method for constructing solid bodies, executed by clicking the «**Revolved Boss/Base**» button 

When this command is executed, the sketch rotates around the specified axis, and the space described by the sketch contour is filled with material to form the solid body. In this process, the sketch of the detail formed by rotation must consist of the detail contour and the rotation axis. The rotation of the contour around the axis can be performed at any desired angle up to 360°.

When creating revolved bodies, there are several constraints:

– The sketch must contain at least one line with the «**Reference Geometry**» property assigned to it, representing the axis of rotation (fig. 3.4).

– The contour cannot intersect the axis line or touch it at an isolated point.

– The contour must be closed; otherwise, a thin feature will be created.

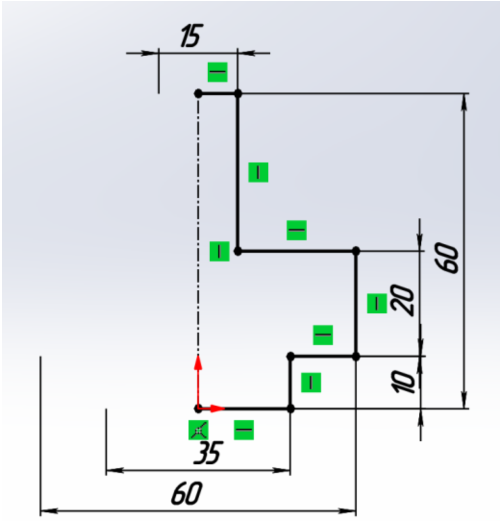


Fig. 3.4.

The «**Revolved Boss/Base**» tool offers three possible ways to construct a model: Revolve, Thin Feature, and a feature built based on closed Selected Contours from the sketch (fig. 3.5). The sketch of the revolved feature can consist of one or multiple closed contours and rotation axes constructed with auxiliary lines. If there are multiple axes in the sketch, the axis around which the contour will be revolved must be specified during the construction of the solid element (by left-clicking). When constructing the element, the direction and angle of rotation must be specified (fig. 3.5).

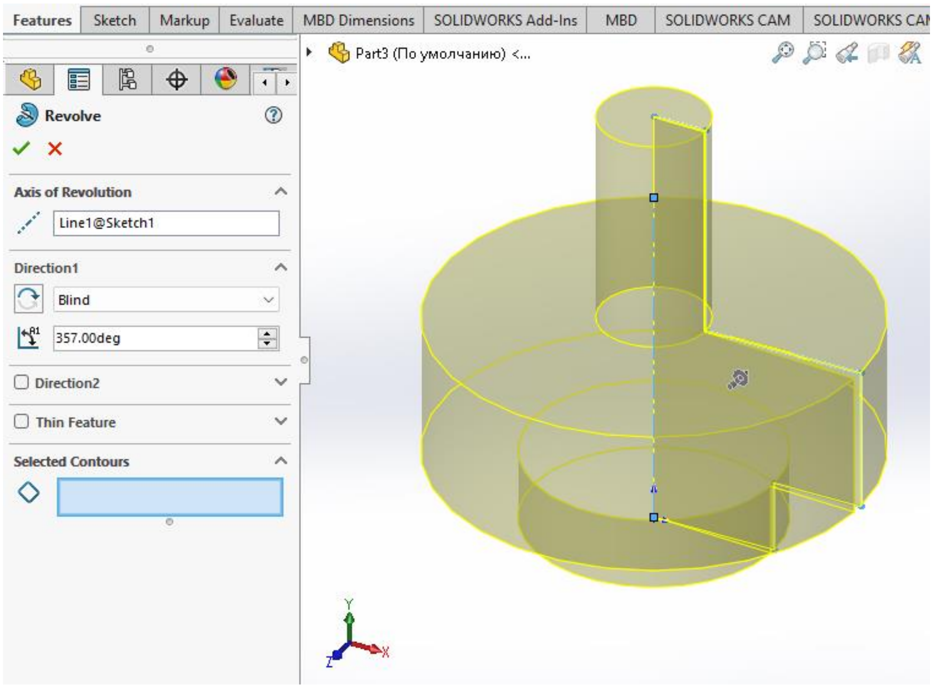


Fig. 3.5

The thin revolved feature is primarily used for creating shell-like forms. For the thin feature, additional input is required for specifying the direction and numerical value of the thickness (the presence of a closed contour is not mandatory for this option).

When choosing the revolve method for constructing a solid body, it's essential to consider the complexity of the sketch profile. In general, the more complex the sketch, the fewer constructive elements will be needed to build the part, and computer resources will be used more efficiently. However, it is easier for the developer to control the model construction process if the sketches are simplified to the maximum extent possible (without including minor structural elements such as fillets and chamfers).

3. ***Extruding along a Path***.

The essence of this method lies in forming a solid body by filling the volume created by moving a profile along a certain trajectory with virtual material. The command for sweeping a feature along a trajectory is initiated by pressing the **«Swept Boss/Base»** button .

Unlike the «Extruded Boss/Base»" and «Revolved Boss/Base» elements, designing parts along a path requires creating at least two sketches – a profile sketch and a path sketch. The profile typically consists of a closed contour sketch, while the path can be either closed or open (fig. 3.6). During the movement, the profile can remain parallel to itself or maintain the original angle with the trajectory. The starting point of the trajectory direction should lie on the profile plane. The profile and direction sketches are constructed on mutually perpendicular planes - Right and Front, respectively.

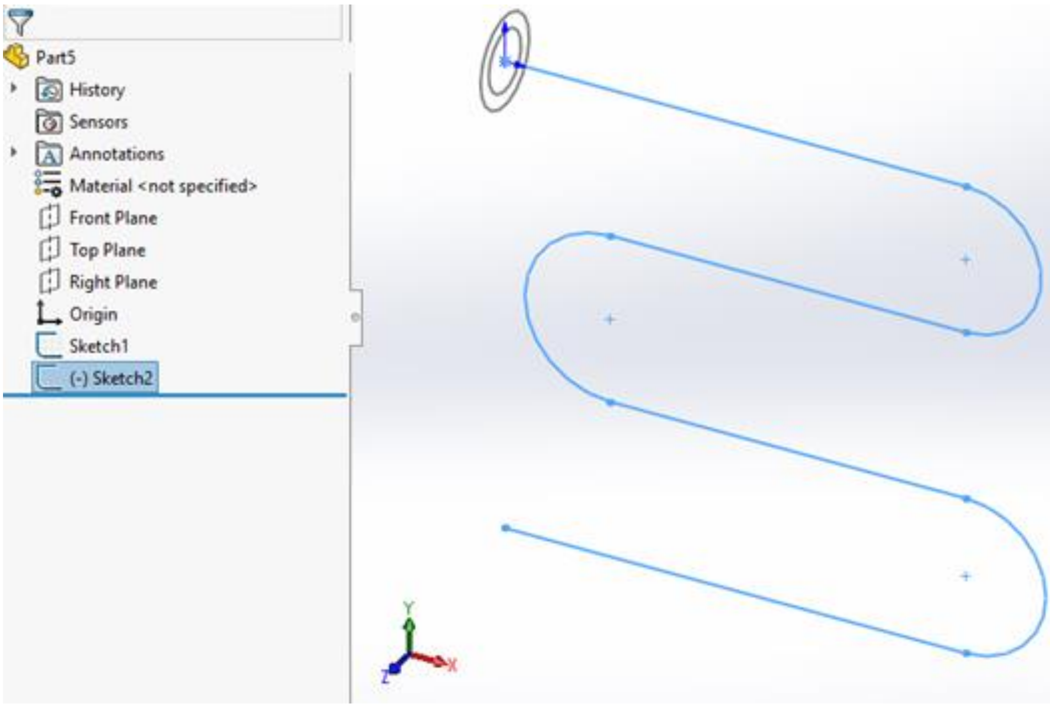


Fig. 3.6.

Fig. 3.7 illustrates an example of constructing a three-dimensional model of a pipe for manufacturing a coil using the «**Swept Boss/Base**» method.

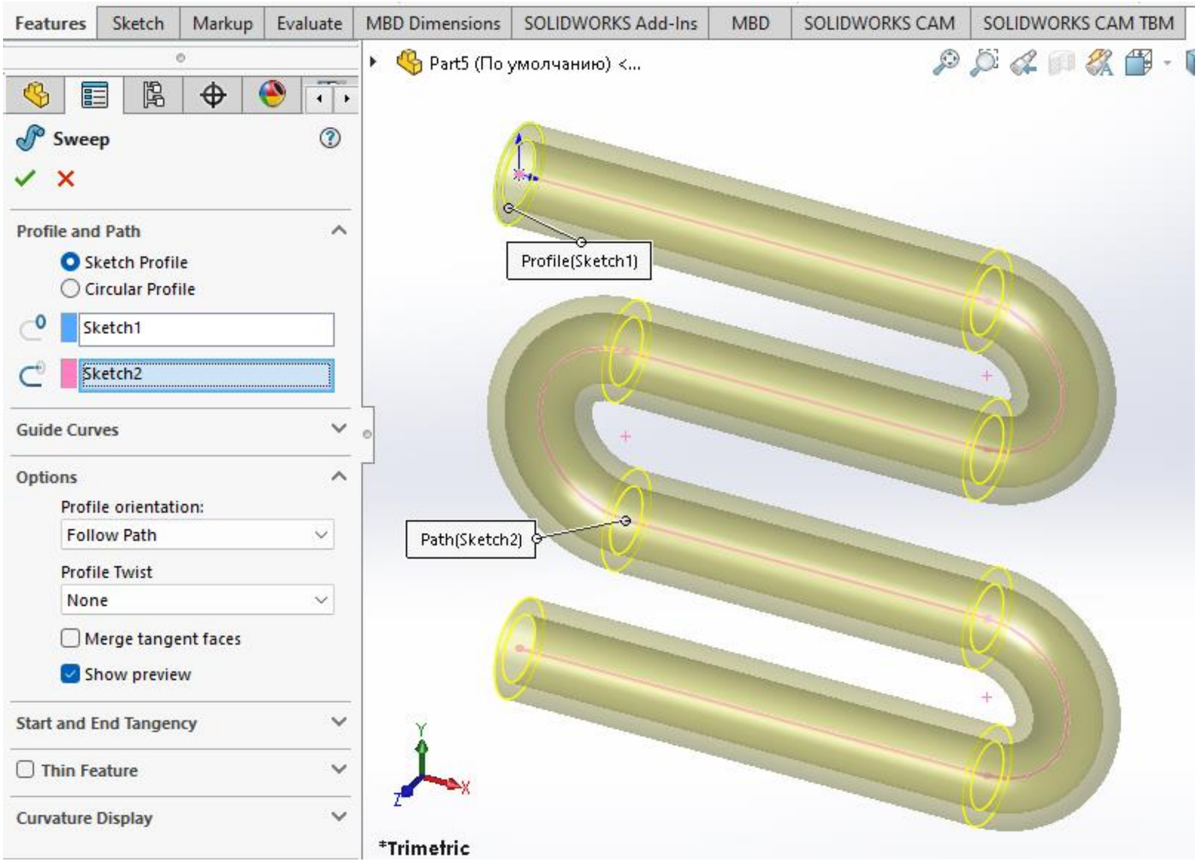


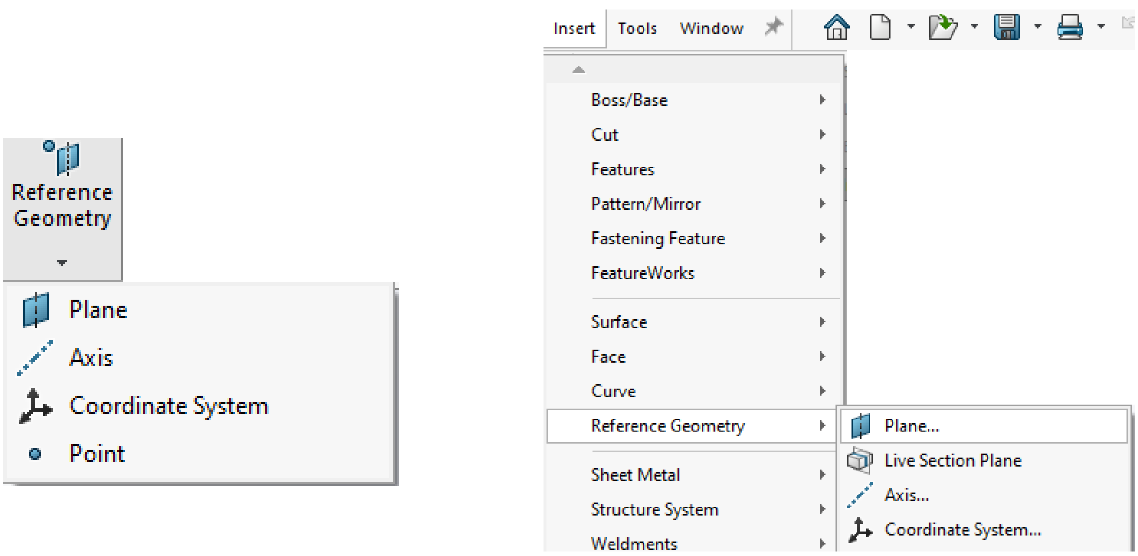
Fig. 3.7.

4. Extrusion Along Paths.

The command for extruding solid bodies based on intersections is activated by the button «**Lofted Boss/Base**»  on the «Features» toolbar. It allows designing details by creating smooth transitions between profiles (cross-sections). Profiles are sketches located on different planes. These planes can be either parallel to each other or at an angle. To construct such an element, at least two cross-sections are required.

Let's consider this command using the example of constructing a sectioned hexagonal pyramid.

Sketch 1 is constructed in the *Top plane*. The next *Sketch 2* needs to be built in another plane. To do this, we need to create an auxiliary plane parallel to the *Top plane* at a distance equal to the height of the pyramid. To create a new auxiliary plane, press the Plane button on the ***Reference Geometry*** toolbar (fig. 3.8 a) or use the menu command: *Insert / Reference Geometry / Plane* (fig. 3.8 b). After that, the Plane window will appear on the screen. In this window, you need to specify the main parameters of the reference plane, providing reference data in the Selection section, and select the method of its construction (fig. 3.9):



a b

Fig. 3.8

1. **Offset Distance**: By specifying a distance, you can construct a reference plane parallel to the original plane. In this example, we build a plane parallel to the Top plane, selecting it in the FeatureManager tree and setting the distance to 80 mm (fig. 3.9).

2. **Parallel to Plane**: You can construct a plane parallel to any existing plane or face passing through a specified point.

3. **Perpendicular**: Allows you to create a reference plane perpendicular to the original one.

4. **At an Angle**: Enables the construction of a plane at an angle to any plane or face through a line, edge, or axis.

5. **Coincident**: You can create a plane parallel to the original one and coincident with any element (face, edge).

If the distance between planes is consistent, you can specify the Number of planes parameter.

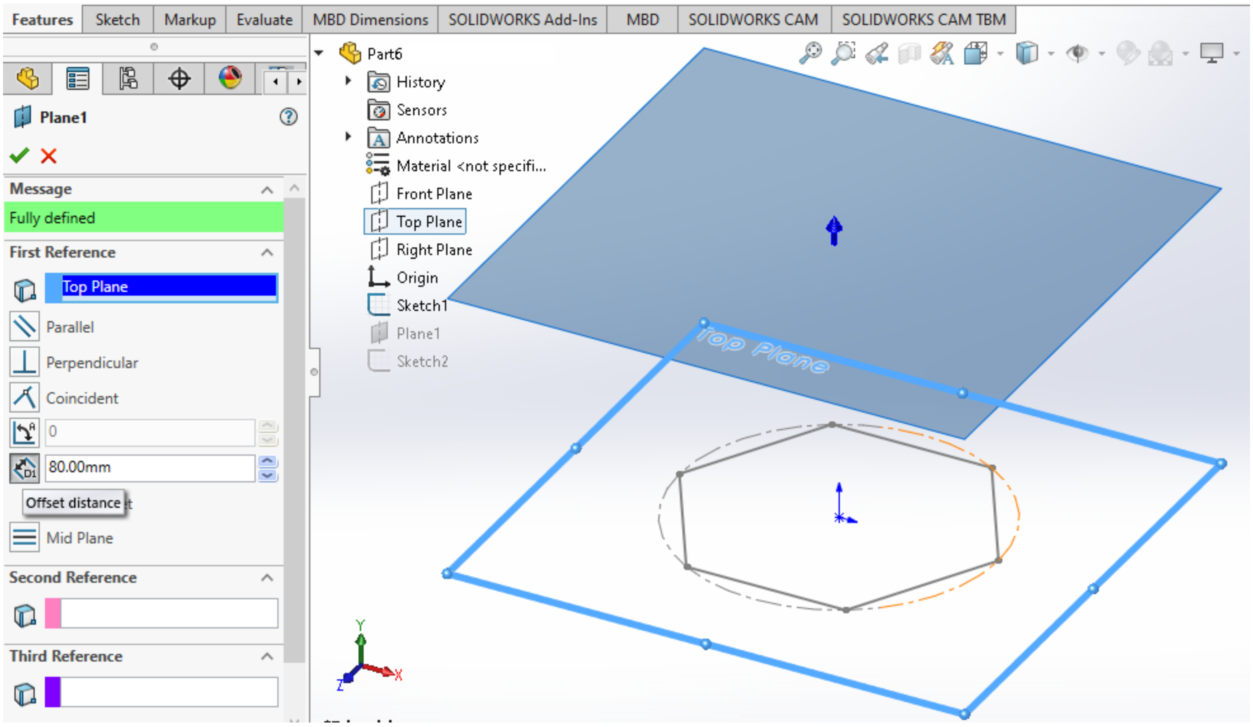


Fig. 3.9

Next, we construct Sketch 2 on *Plane 1* (fig. 3.10).

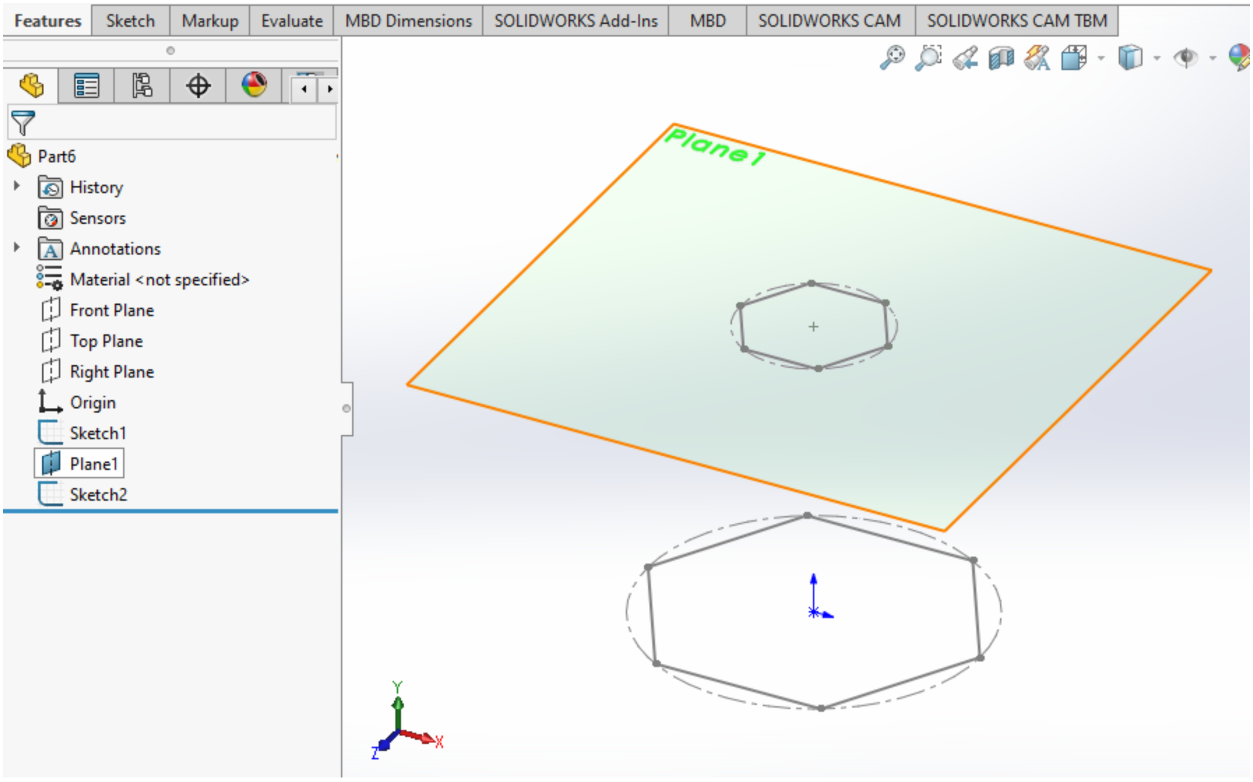


Fig. 3.10

On the **Features** toolbar, activate the «**Lofted Boss/Base**» button. The «**Lofted Boss/Base**» window will appear on the screen. In the *Selections* section, choose the profiles. To do this, click on *Sketch 1* and *Sketch 2* in the *FeatureManager* (fig. 3.11). Then, click OK. The pyramid is now constructed.

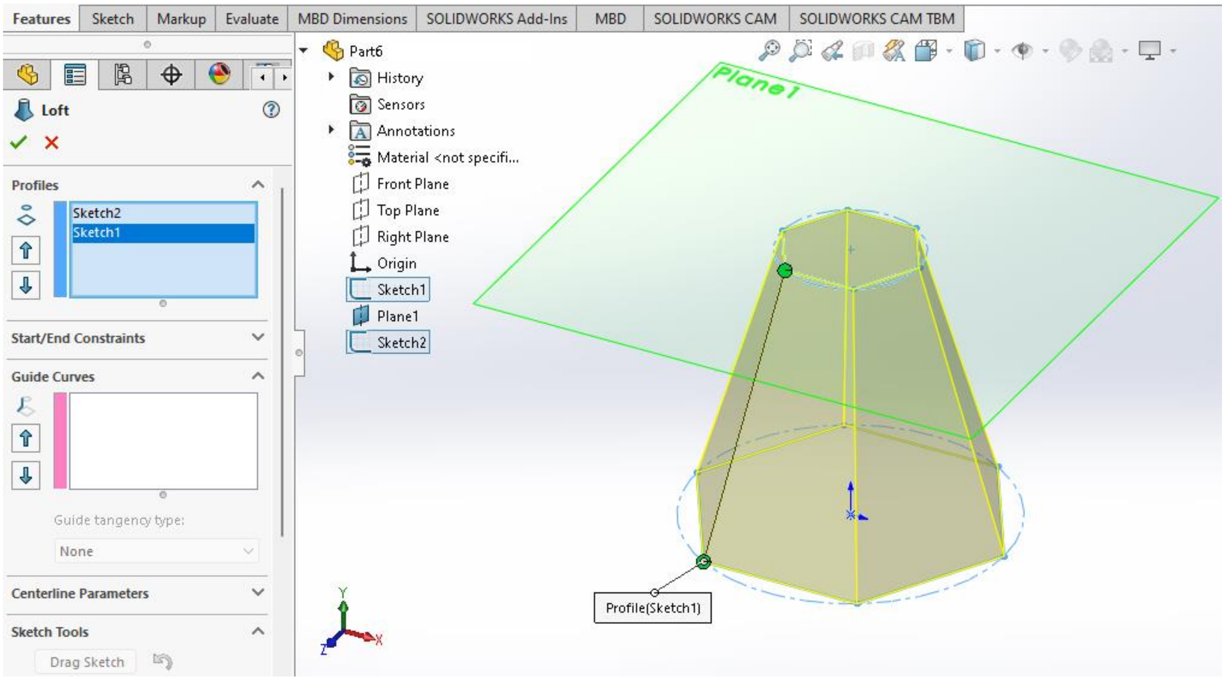
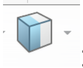
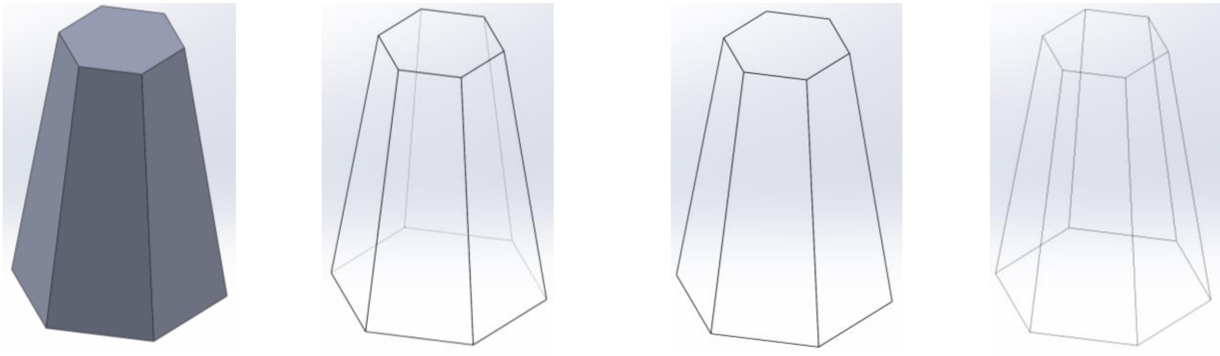


Fig. 3.11

SolidWorks allows representing the obtained solid model in several ways (fig. 3.12) in the Display Style tab  :

– Shaded representation (fig. 3.12 a);
– Shaded with edges;
– With the display of hidden lines (fig. 3.12 b);
– Without the display of hidden lines (fig. 3.12 c);
– Wireframe representation of the model (fig. 3.12 d).



a b c d

Fig. 3.12

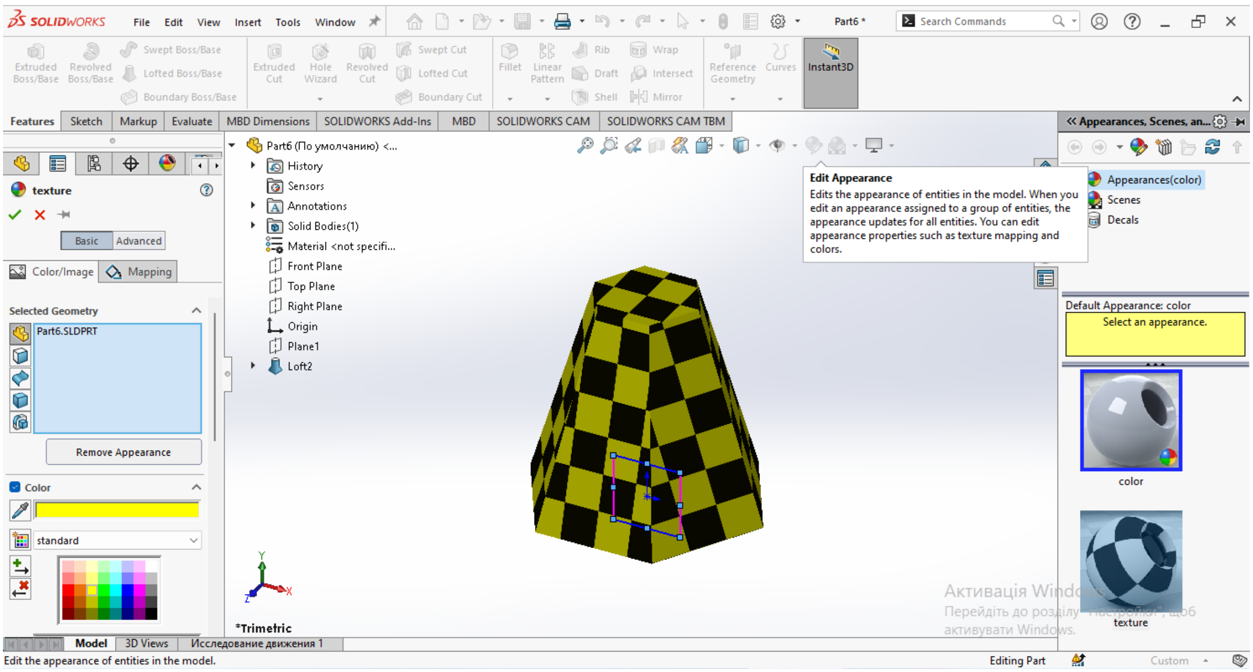


Fig. 3.13

Useful tips for creating detail models

1. Position the solid model so that its symmetry planes align with the main planes.

2. In the process of creating the model, try to reflect the sequence of technological operations needed to manufacture the part.

3. Models of parts obtained by turning should be formed using the «**Revolved Boss/Base»** feature.

4. If the part has a threaded surface, it should be indicated using the «*Insert* / *Annotations* / *Thread Annotation»* command.

5. Use the «*Hole Wizard»* feature exclusively for creating holes for fastening.

6. When creating linear or circular patterns of simple shapes, it's better to use the corresponding sketch tool rather than the feature.

7. When creating a «*Shel»*" feature, all fillets should be added to the model beforehand.

8. Specify the material for each part to obtain the appropriate part mass.

Lecture № 4

CREATING DRAWINGS BASED ON GENERATED THREE-DIMENSIONAL MODELS IN SOLIDWORKS

Lecture Objective: To master techniques for constructing associative detail drawings; to study methods for automated generation of sections and cuts on associative detail drawings and axonometry.

In SolidWorks, there is an option for automatic creation of associative drawings, which are created and stored in memory based on three-dimensional models.

All types of such drawings are linked to the model: changes made to the model result in automatic updates to the associated views.

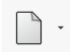
There are two methods for creating a drawing based on a created three-dimensional model:

1. **Automatic drawing creation**: This method involves automatically generating a drawing based on the three-dimensional model without the need to manually place views and annotations. SolidWorks automatically generates the drawing according to user-defined settings, such as sheet templates and view configurations.

2. **Manual drawing creation**: This approach allows the user to manually create drawings based on the three-dimensional model. The user can select views, place them on the sheet, and add annotations according to their preferences. This method provides greater flexibility to the user but requires more attention and effort to create the drawing.

Let's consider the methods of creating a drawing based on the created three-dimensional model:

1. In the open document of the constructed 3D model, we select to create a drawing of the part from the **File** menu. Alternatively, we can click on the

Create button  on the toolbar. This action will generate a drawing based on the current document (fig. 4.1).

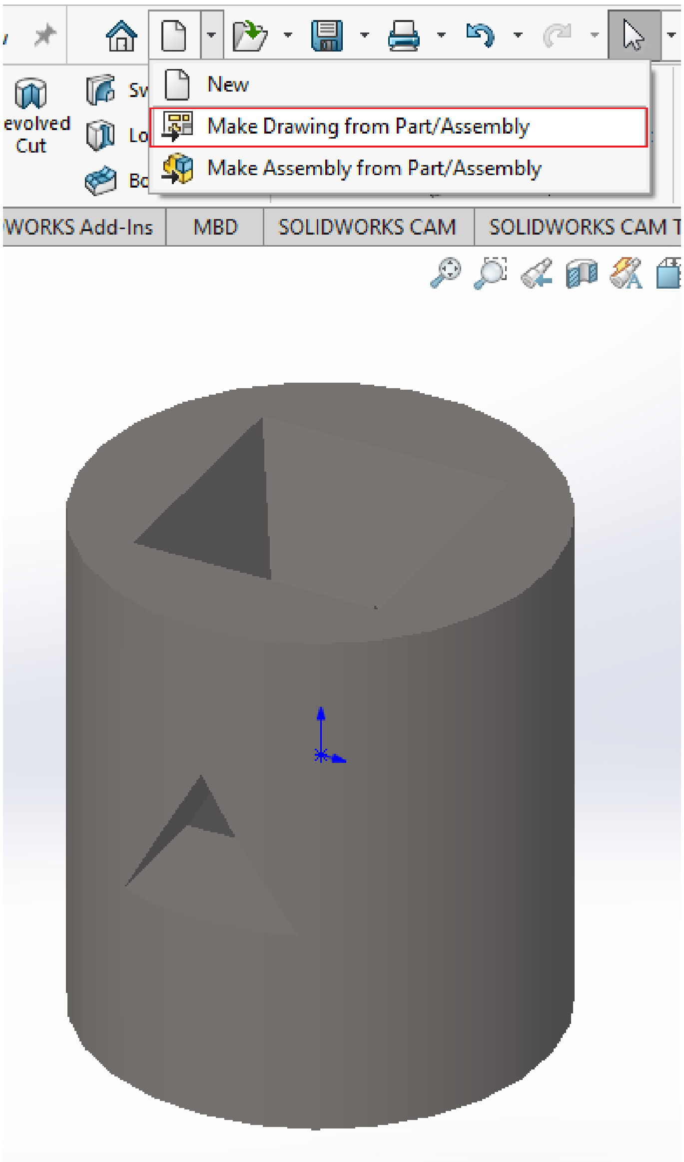


Fig. 4.1.

2. Select ***File / New / Drawing***. If there are open documents, SolidWorks will prompt you to choose which one to base the drawing on. If the necessary documents are not open, select them by clicking the ***Browse*** button in the selection window.

When creating the drawing, the user will be presented with the standard window for selecting the sheet format for the drawing (fig. 4.2).

You need to choose the appropriate format depending on the complexity and size of the part.

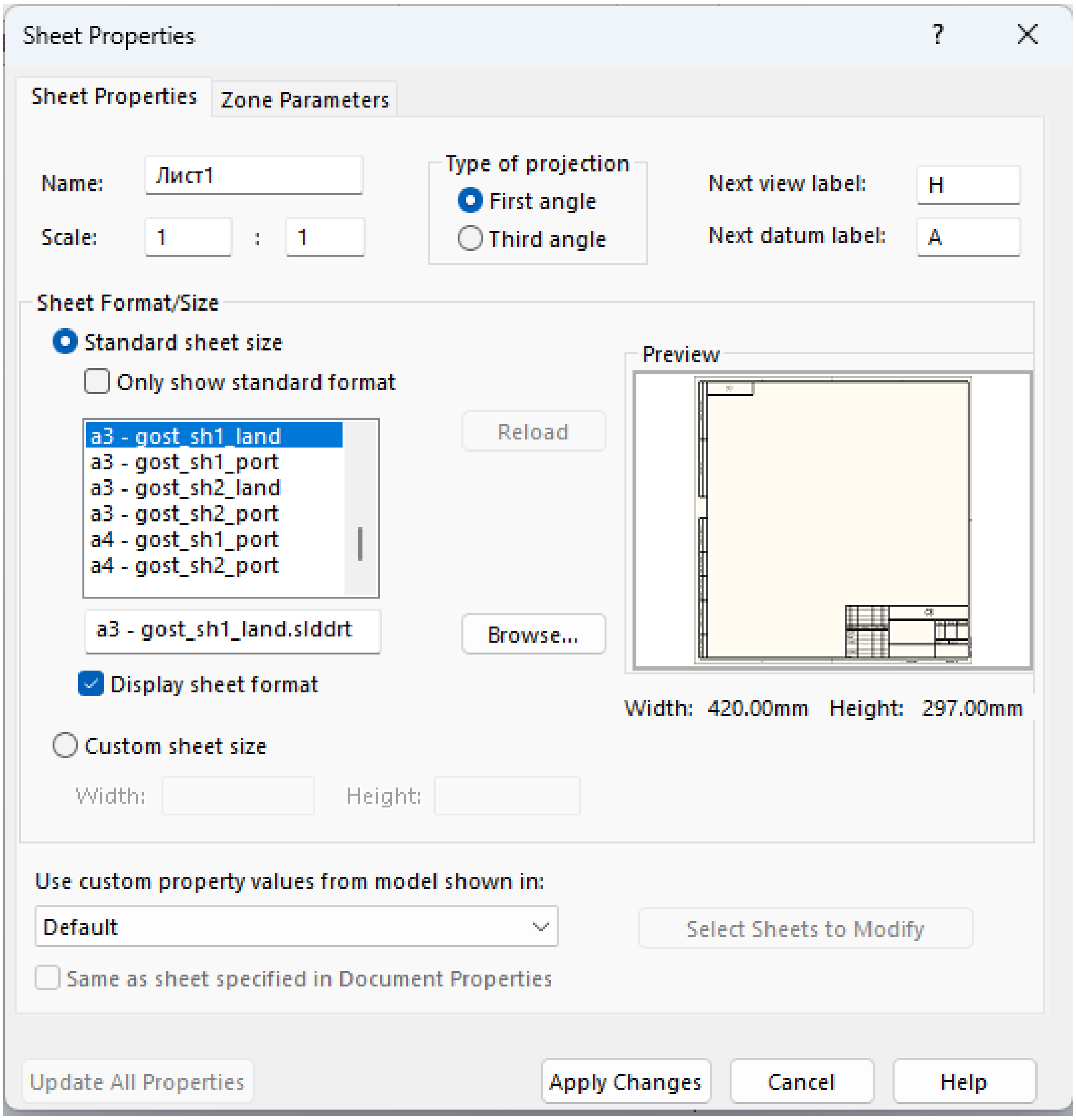


Fig. 4.2.

On the **Drawing** toolbar, select «**Model View**». The ***PropertyManager*** will appear on the left. Click on «**Browse**» and load the part file into the drawing (fig. 4.3).

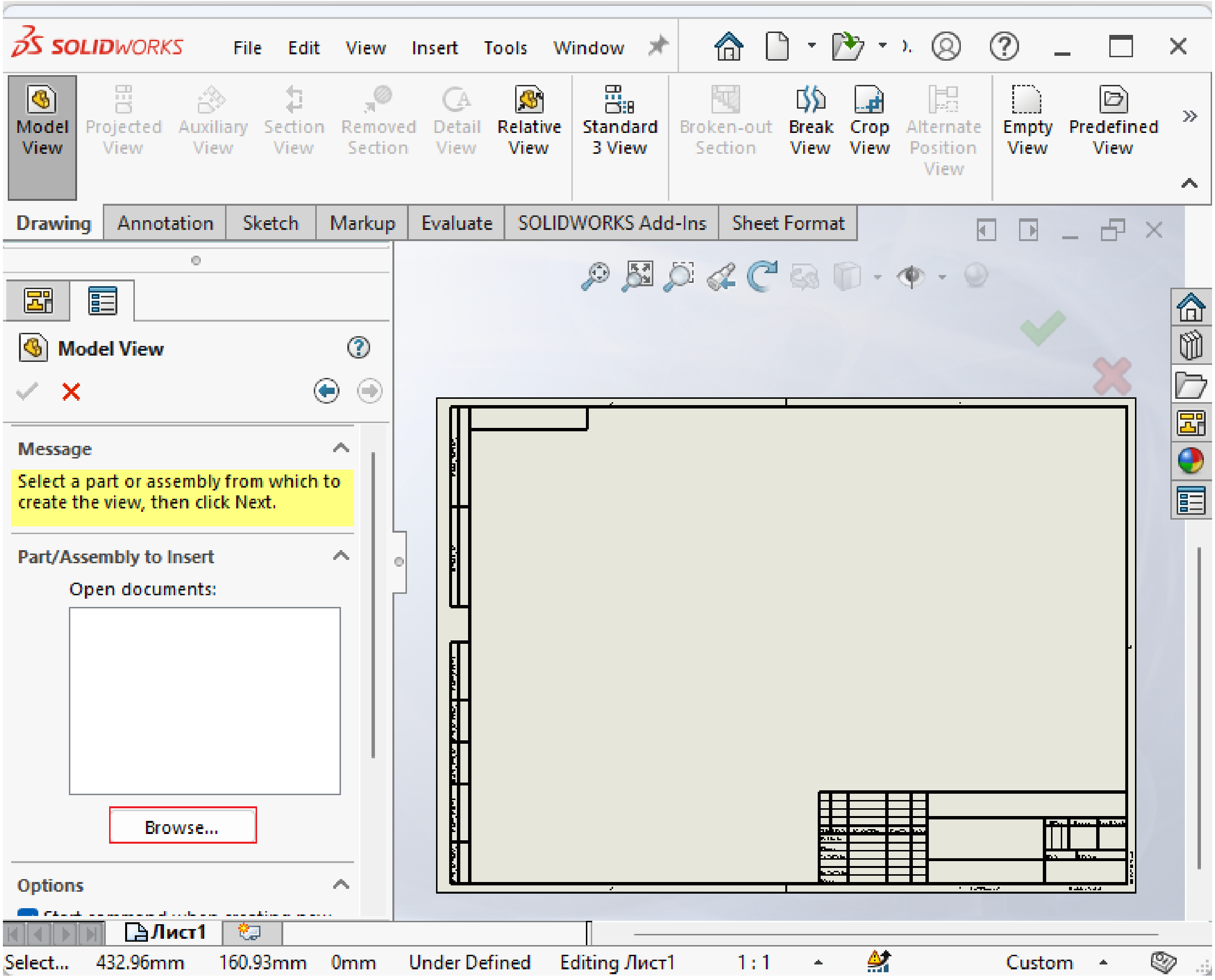


Fig. 4.3.

After loading the part file, the system will immediately proceed to create projected views. The ***PropertyManager*** window will change, and options for configuring the views will appear.

This window consists of several areas.

– In the «Orientation» section, you will find a panel for selecting the orientation of the detail in the projected view you are about to create. This panel is essential for ensuring that the projected views accurately represent the three-dimensional object in the desired manner. When you select the option to "Create multiple views," you open up the possibility of choosing from six standard orthogonal projection views: front, back, left, right, top, and bottom. These views provide a comprehensive representation of the object from various angles, offering a thorough understanding of its shape and features (fig. 4.4).

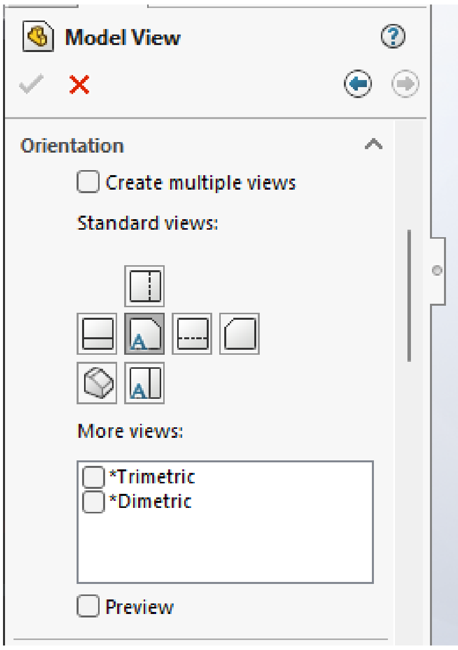


Fig. 4.4.

By selecting one or more of these standard views, you dictate how the detail will be projected onto the drawing sheet. For example, choosing the "Front" view will generate a projected view showing the object as it would appear when viewed directly from the front. Similarly, selecting the "Top" view will project the detail as seen from above. Each view offers a unique perspective, allowing for a comprehensive depiction of the object's dimensions and contours.

Additionally, you have the option to select axonometric views, which provide a more three-dimensional representation by showing the object from angles that are not strictly aligned with the standard axes. These views offer versatility in visualizing the detail and can be useful for conveying depth and spatial relationships within the drawing.

It's important to note that these views are created relative to the principal planes of the loaded project. This means that the orientation of the projected views will be based on the orientation of the original three-dimensional model. This ensures consistency and accuracy in the representation of the detail across different views.

– **Options** (fig. 4.5) – Auto-launch of projected views. Selecting this option allows for the creation of linked projected views immediately after creating the first view.

– **Display Quality** –The method of displaying the model in the created view.

– **Scale Selection:** Choosing the scale of the created view. If the «*Use Sheet Scale*» option is selected, the system will automatically select the scale that best fits the placement of the required projected views on the sheet. Additionally, the user can manually select the desired scale using the «*Use Custom Scale*» option. The scale selected for the view can always be interactively changed by selecting the desired view on the sheet.

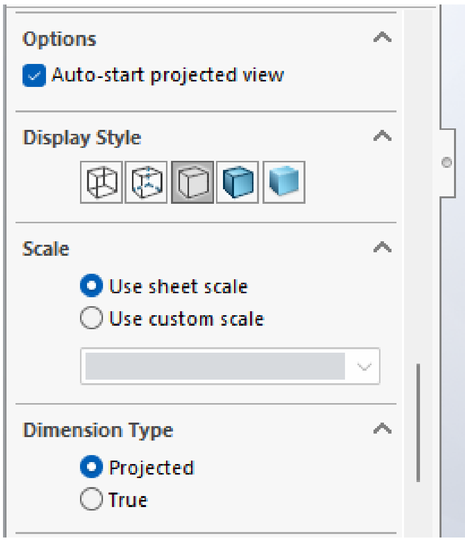


Fig. 4.5.

– **Dimension type** – selection of the model dimension display type:

True – only the dimensions of the actual elements of the model will be displayed;

Projected – the size of the model geometry according to its projection.

Select the *Front* view, scale – *Use Custom Scale*, and click in the upper left corner of the sheet (fig. 4.6).

After creating the first view, the *Projection View* tool will start automatically. By moving the mouse left, right, down, or up, the user can see a new view that is projection-linked to the previous one. To break the projection link, press and hold the **CTRL** key. If you move the mouse diagonally, the system will suggest the corresponding axonometric view.

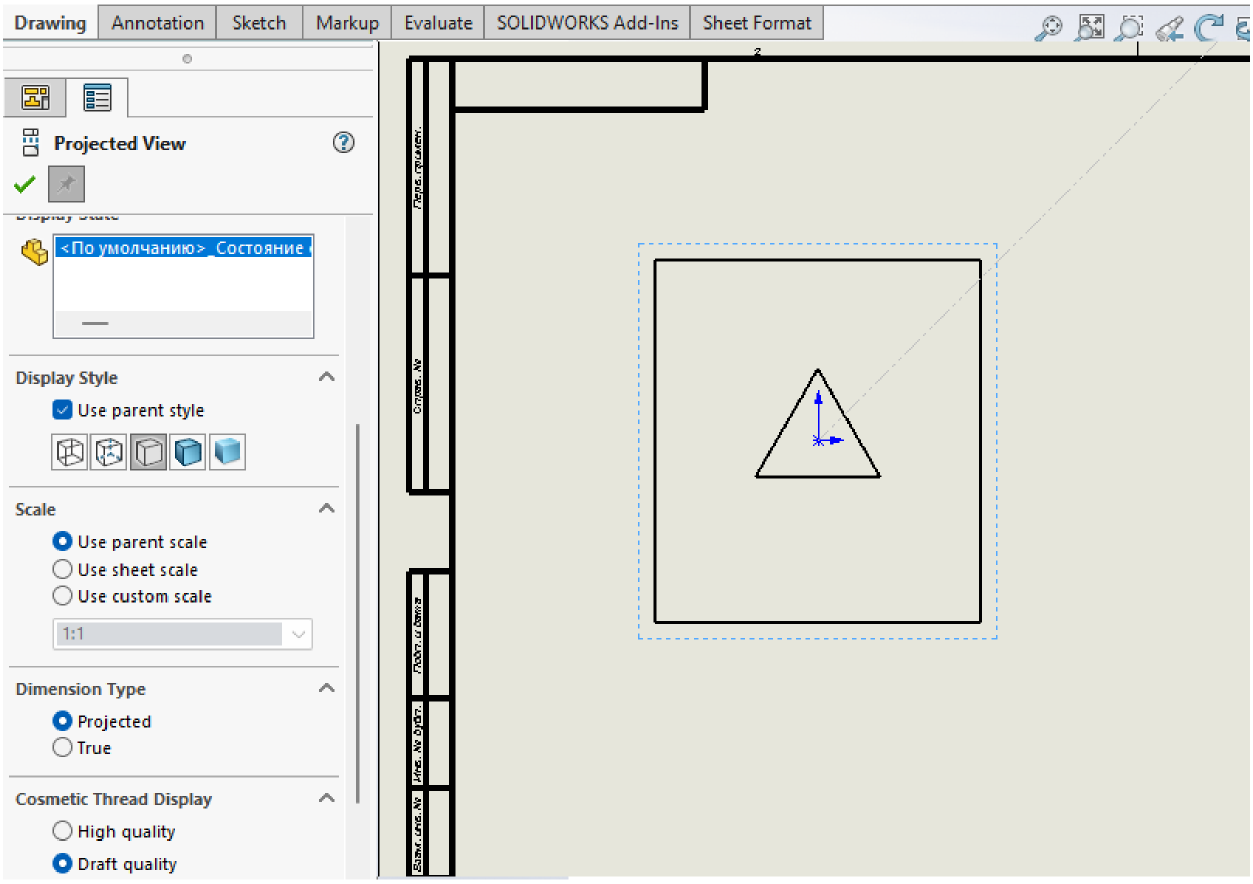


Fig. 4.6.

We move the mouse downwards and press the left button. A new projection view is created (the top view) (fig. 4.7). Similarly, we'll create a left-side view by moving the mouse to the right from the front view.

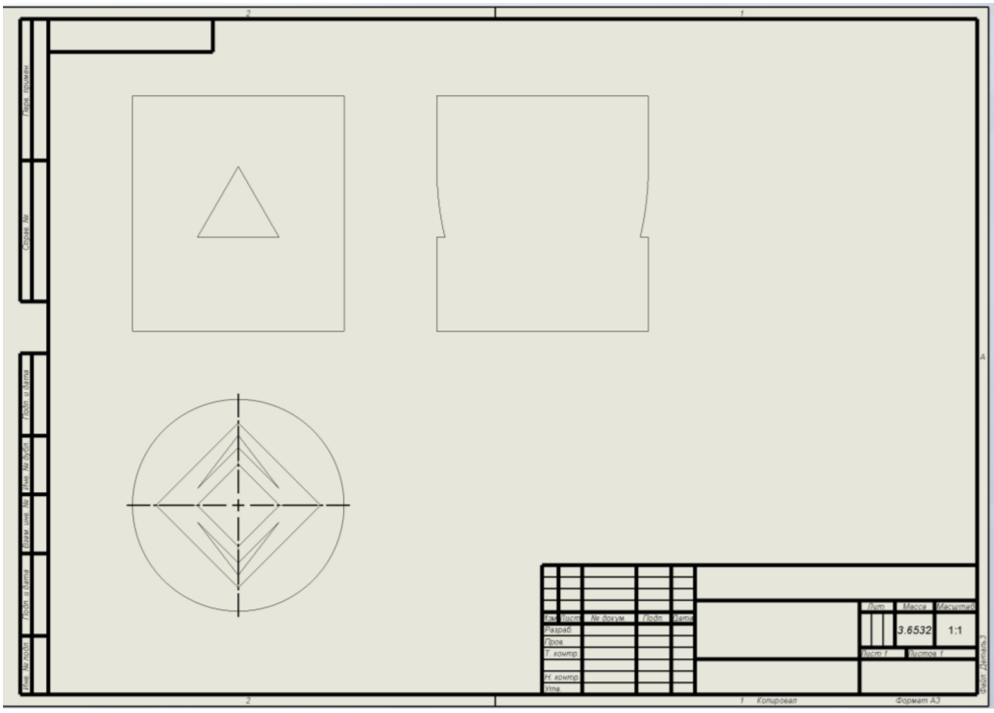


Fig. 4.7.

We complete everything by creating an isometric view (fig. 4.8).

To create an isometric view, you need to choose the projection that provides the most illustrative isometric representation. To create an isometric image, move the mouse from the front view upwards and to the left. Interrupt the action by pressing the **ESC** key on the keyboard. Also, it is necessary to show the hidden contour lines on the isometric image. To do this, click on this image with the left mouse button, which activates the **PropertyManager**, and options for configuring the isometric image appear. Set the *Display styie* to show hidden lines (fig. 4.8). Since there is a hole inside the detail in the form of a four-sided pyramid, change the isometric view to dimetric.

You can change the position of the drawing views by dragging them across the drawing sheet. In the default arrangement of views, the Front View or the Main View serves as the primary view. Therefore, when moving the Front View, all other views follow it.

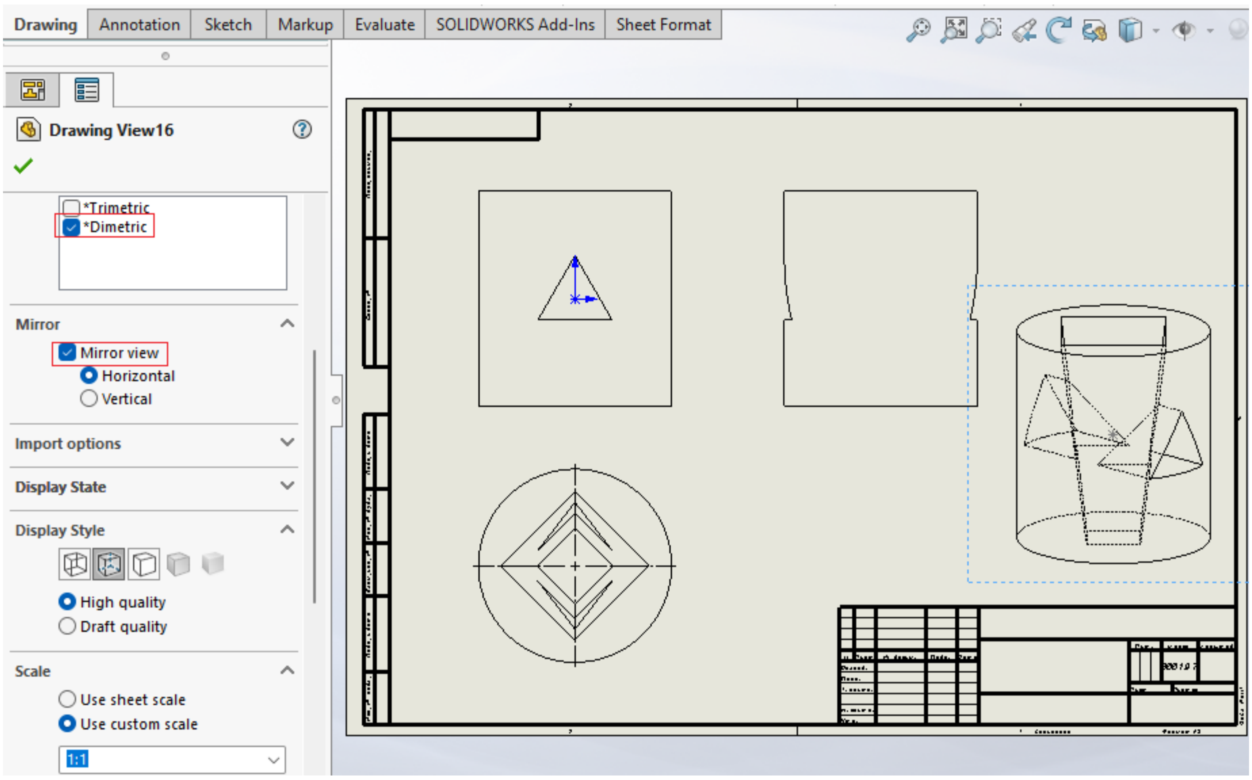


Fig. 4.8.

To add centerlines, you need to switch to the «**Annotation**» tab and select «**Centerline**»  (as shown in fig. 4.9). For automatic insertion of centerlines in all views, you need to choose the auto-insert option and select the drawing views where you want to place the symmetry axes.

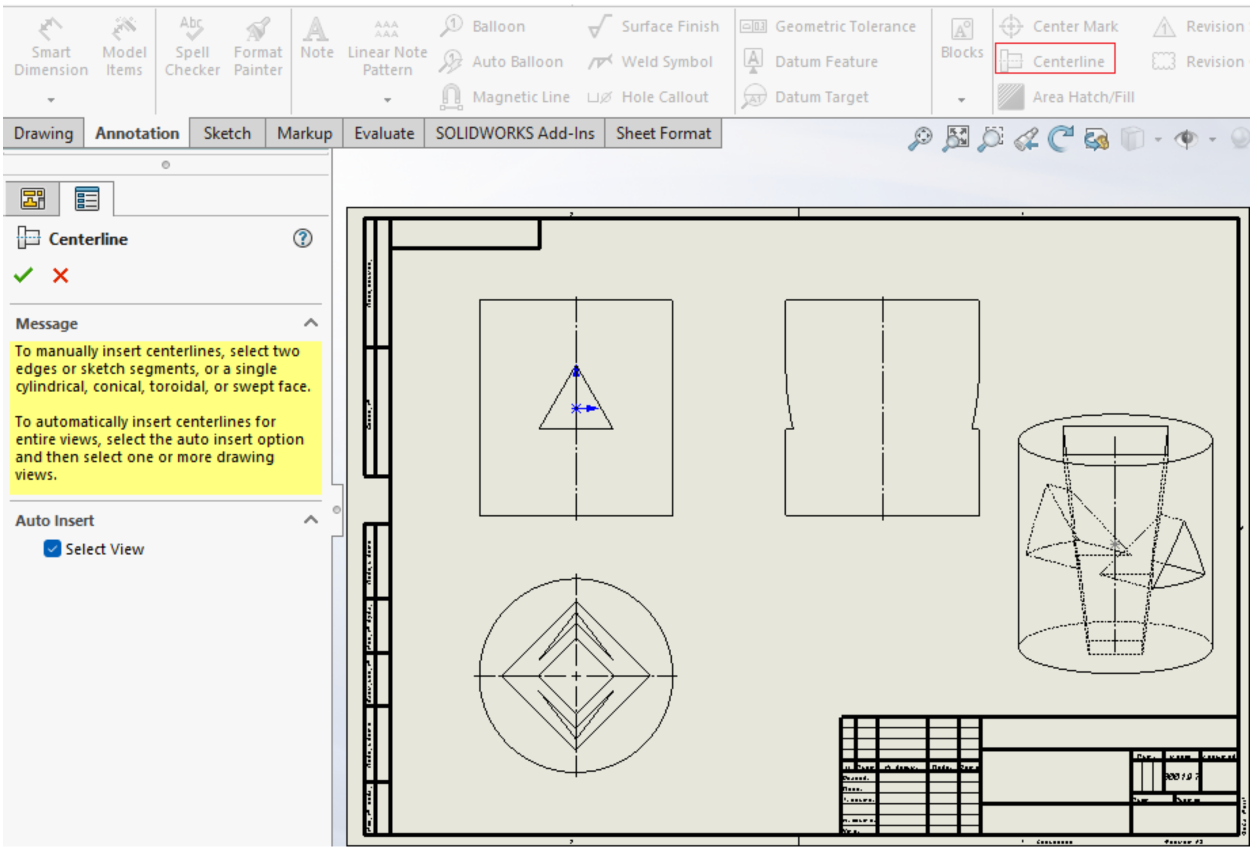


Fig. 4.9.

Building Cross Sections

Let's consider the sequence for creating a drawing of a cylinder with an inclined cross section A-A:

1. Make the front view active by double-clicking the left mouse button on the view, then draw the cutting line using the **Section view** (fig. 4.10).

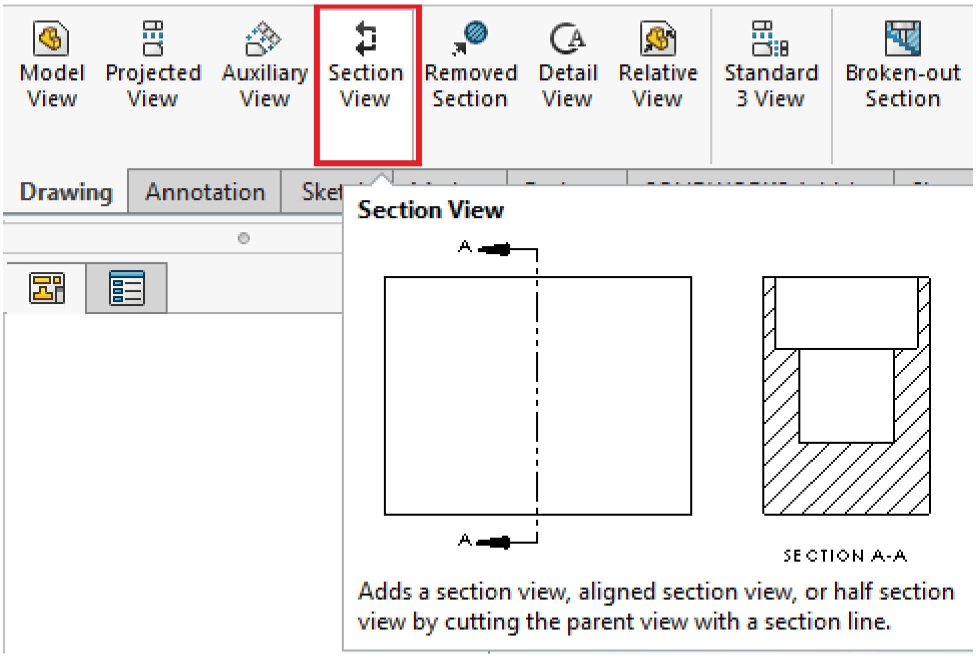


Fig. 4.10.

After placing the cutting line on the drawing, a section view automatically appears. To change it to a cross-section, it is necessary to select **Slice section** on the PropertyManager panel (fig. 4.11). Next, place the cross-section in an open area of the drawing. Since the cross-section is created in a projected relationship, this relationship can be disabled by holding down the **Ctr**l key.

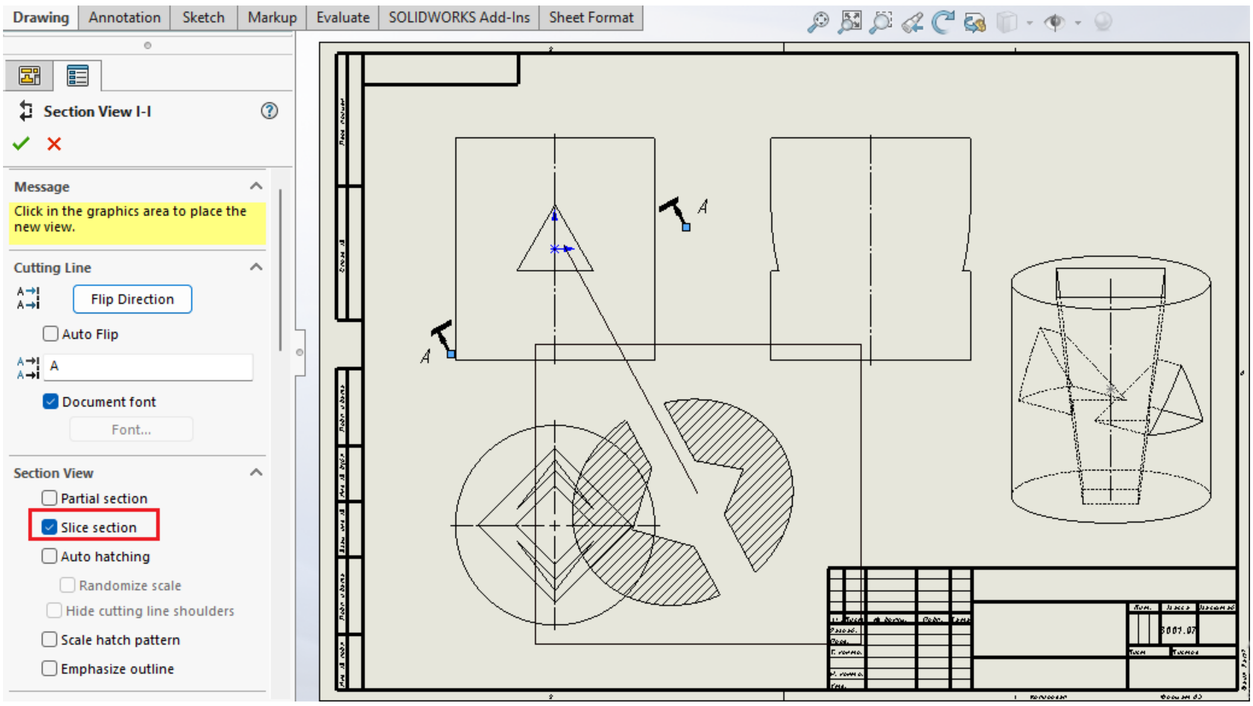


Fig. 4.11.

Creating Section Views

If a part has complex internal voids, cross-sections are used to illustrate them more clearly on the drawing. When the view and the cross-section are symmetric shapes, it is possible to connect half of the view with half of the cross-section, dividing them with a thin dashed line, which represents the axis of symmetry. Typically, the cross-section part is positioned to the right of the axis of symmetry that divides the view part from the cross-section part, or below the axis of symmetry. Invisible contour lines on the view part that connect to the cross-section part are usually not shown. If any line projection, such as an edge of a face of the figure, coincides with the axial line that divides the view and the cross-section, they are separated by a solid wavy line drawn to the left of the axis of symmetry (fig. 4.12 a) if the edge lies on the inner surface, or to the right (fig. 4.12 b) if the edge is external.

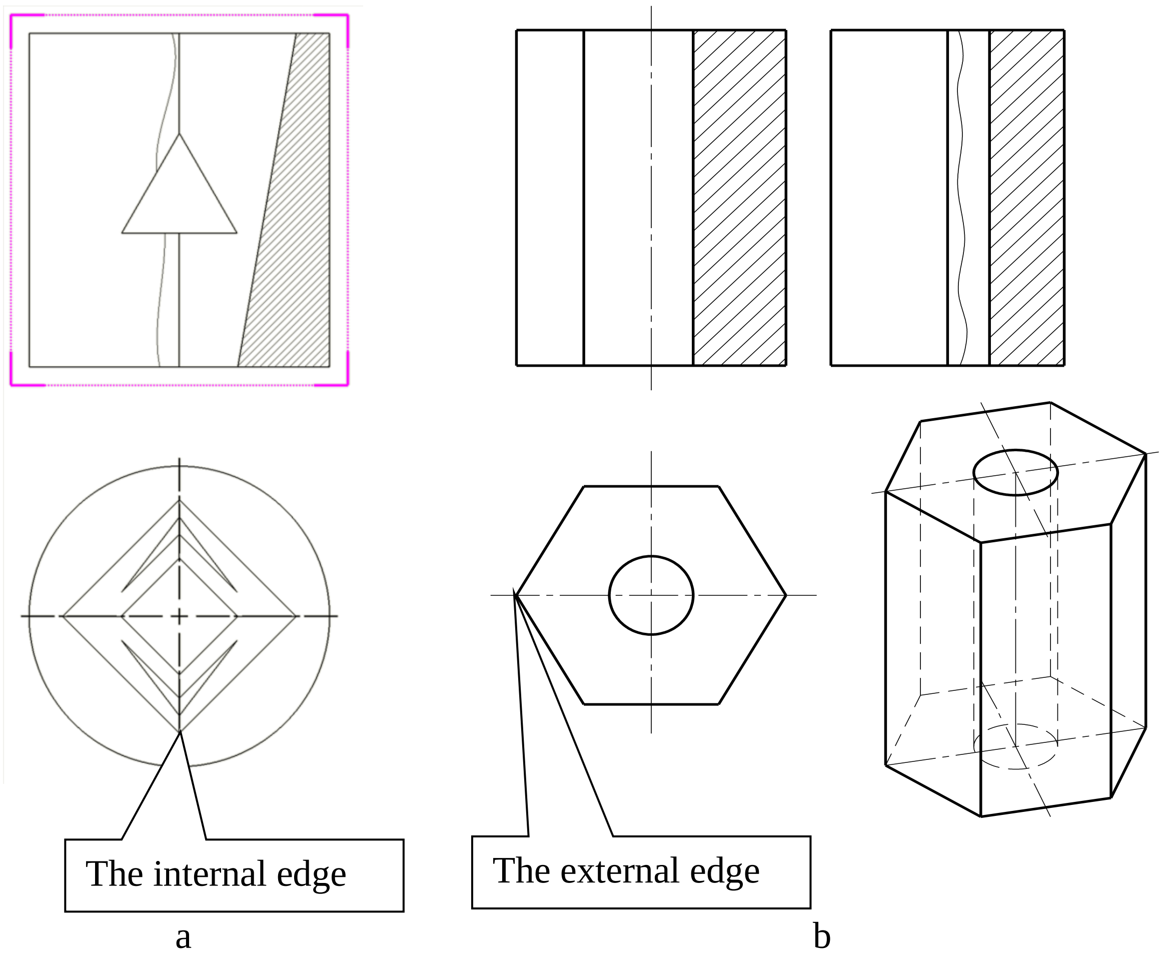
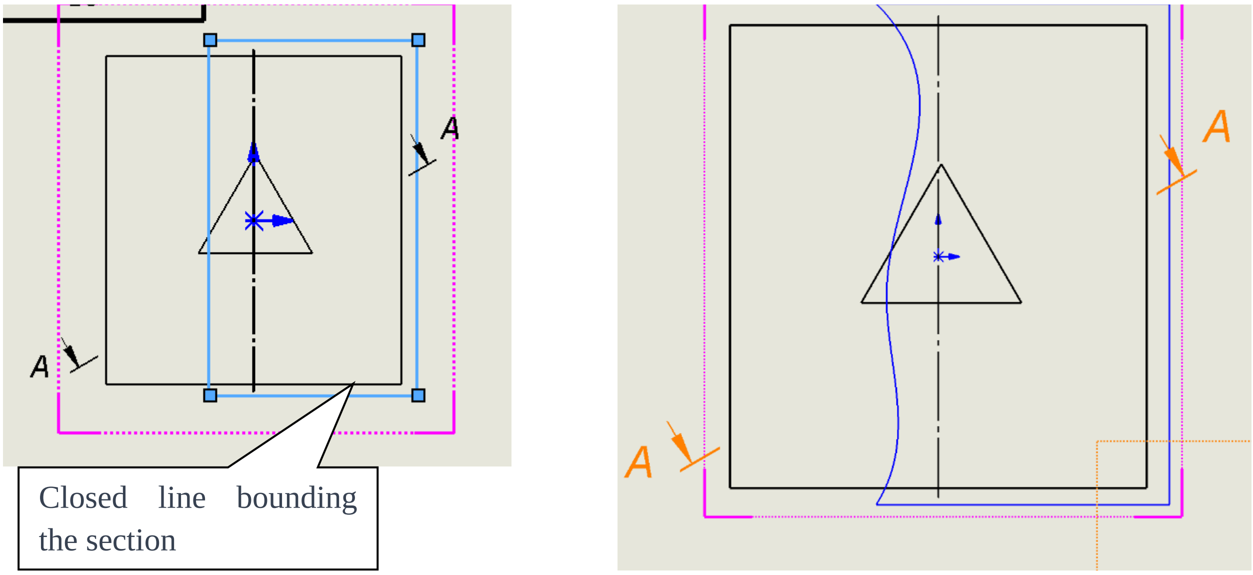


Fig. 4.12.

The part under consideration is symmetrical about the vertical axis and has internal voids. Taking this into account, to construct a frontal section, it is necessary to remove the right front part in accordance with the rule of aligning views and sections.

To do this:

- press the **Rectangle** button on the **Sketch** toolbar and draw a rectangle to encompass the part of the view where the section is needed instead (fig. 4.13 a). Since there is an internal edge at the boundary between the view and the section, the section needs to be enlarged and divided by a wavy line (fig. 4.13 b);



a b

Fig. 4.13.

- On the **Drawing** toolbar, select the **Broken-out Section** button and specify the depth of the section plane. Then, click OK. The system will generate the front section (fig. 4.14).

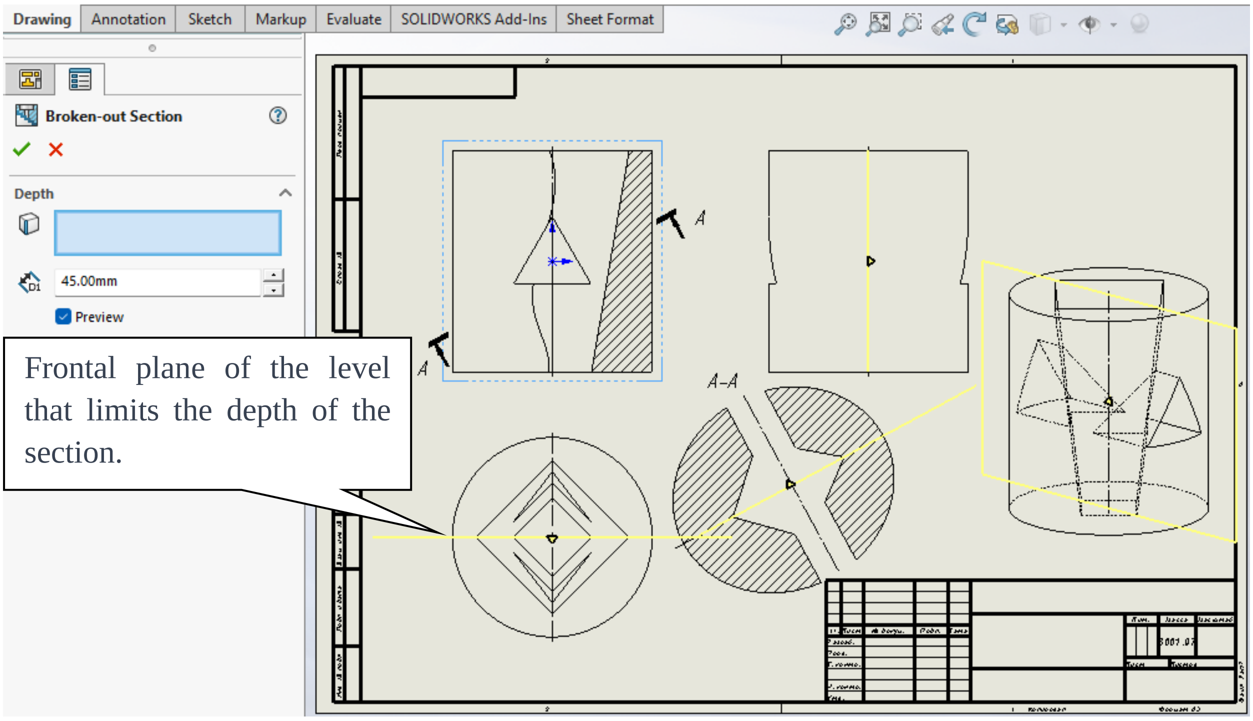


Fig. 4.14.

Next, we will construct a horizontal section. It is essential to make the top view active. Since the part is symmetrical, we will also align the view with the section. On the top view, we draw a rectangle to the right of the vertical axis of symmetry (fig. 4.15). Then, on the **Drawing** toolbar, we select the **Broken-out Section** button , specify the depth of the section plane, and enable the preview. The yellow color indicates the position of the section plane. Click OK . The system will generate the horizontal section (fig. 4.16).

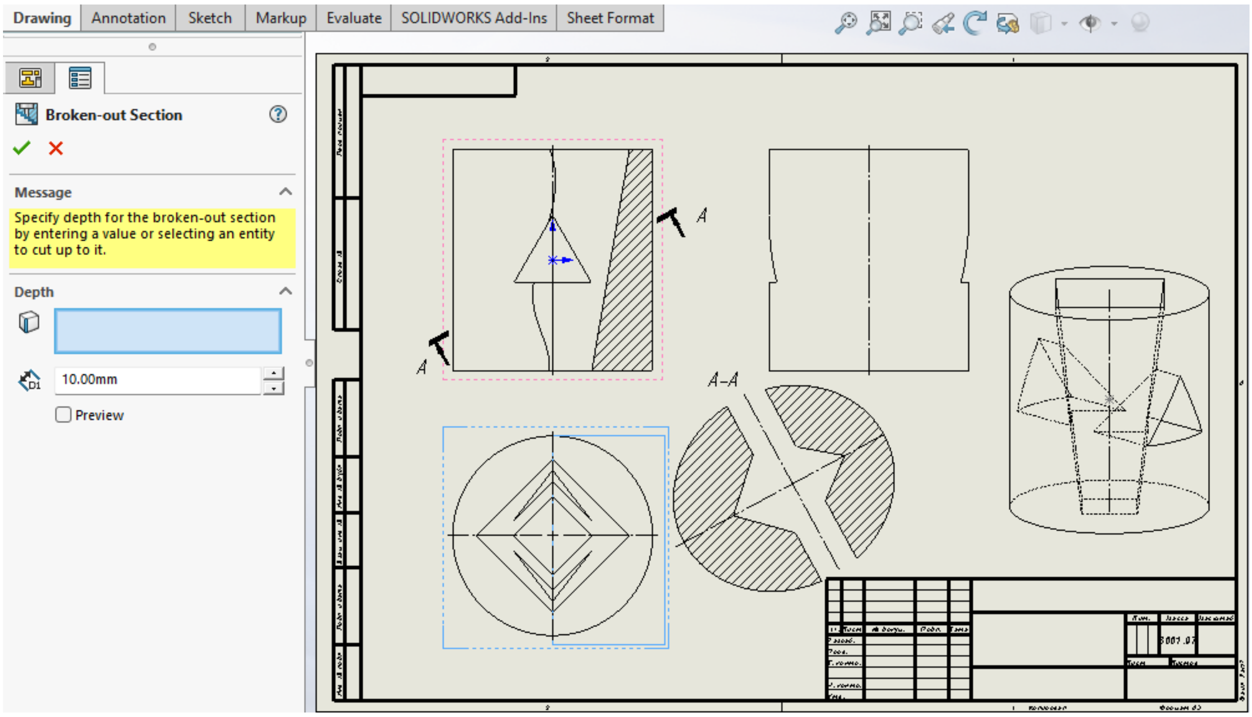


Fig. 4.15.

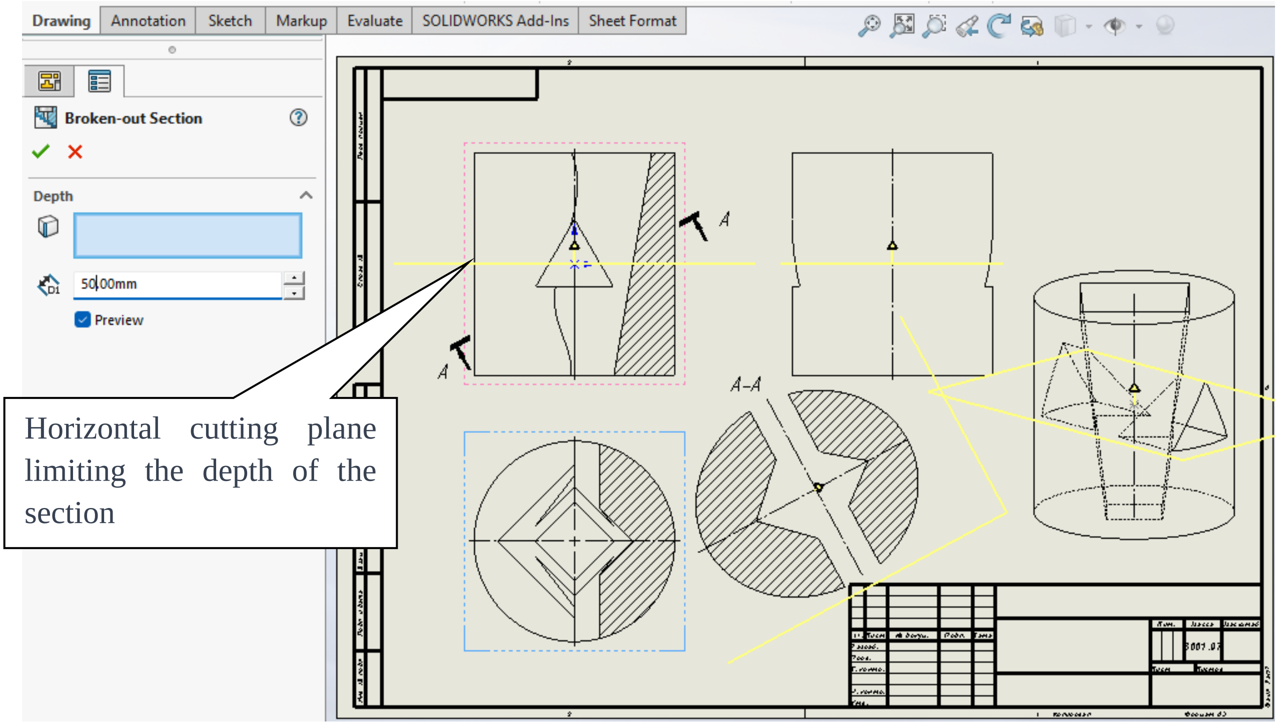


Fig. 4.16.

In this case, the horizontal section needs to be annotated (fig. 4.17). This is because depending on the height of the section line, the representation of the section on the top view will change. Annotation is done using the **Section view** command on the **Sketch** panel.

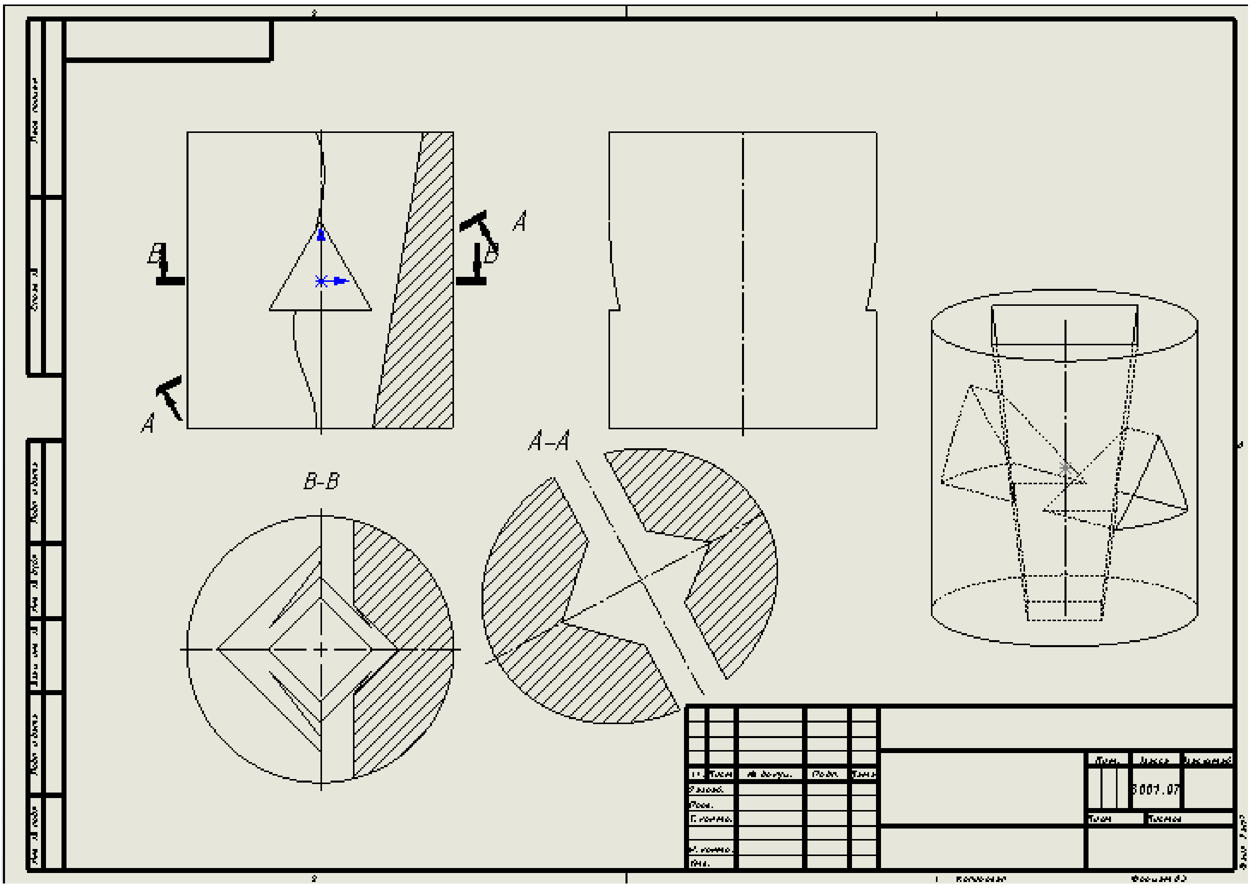


Fig. 4.17.

We need to construct a profile section by aligning a portion of the view with a portion of the section. The profile section is performed on the left-hand view. Double-clicking with the left mouse button activates this view. According to the standard, the section should be placed to the right of the symmetry axis. Since the internal edge falls on the boundary between the view and the section, we need to increase the section. Therefore, we place the wavy dividing line between the view and the section to the left of the symmetry axis (fig. 4.18).

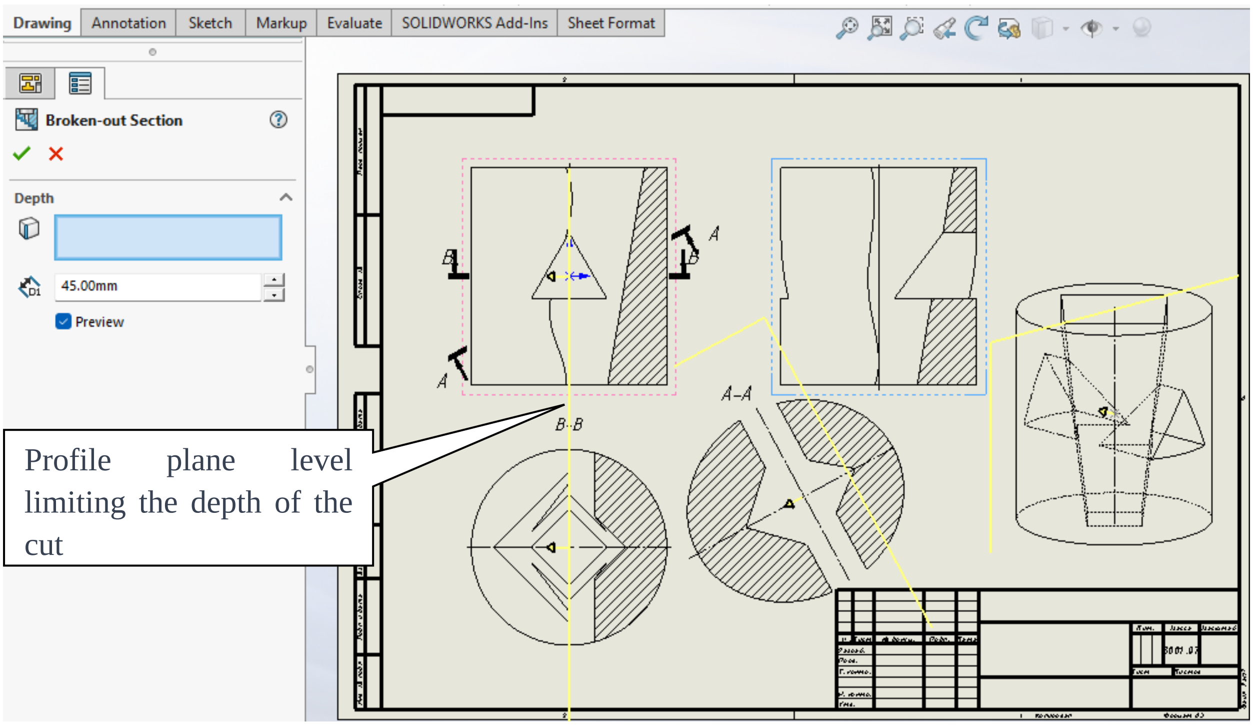


Fig. 4.18.

We only need to apply dimensions. For automatic dimensioning, click the «**Model Items**» button  on the **Annotation** toolbar. In the **PropertyManager**, activate «**Import items into all views**» and disable «**Eliminate Duplicates**» (fig. 4.19).

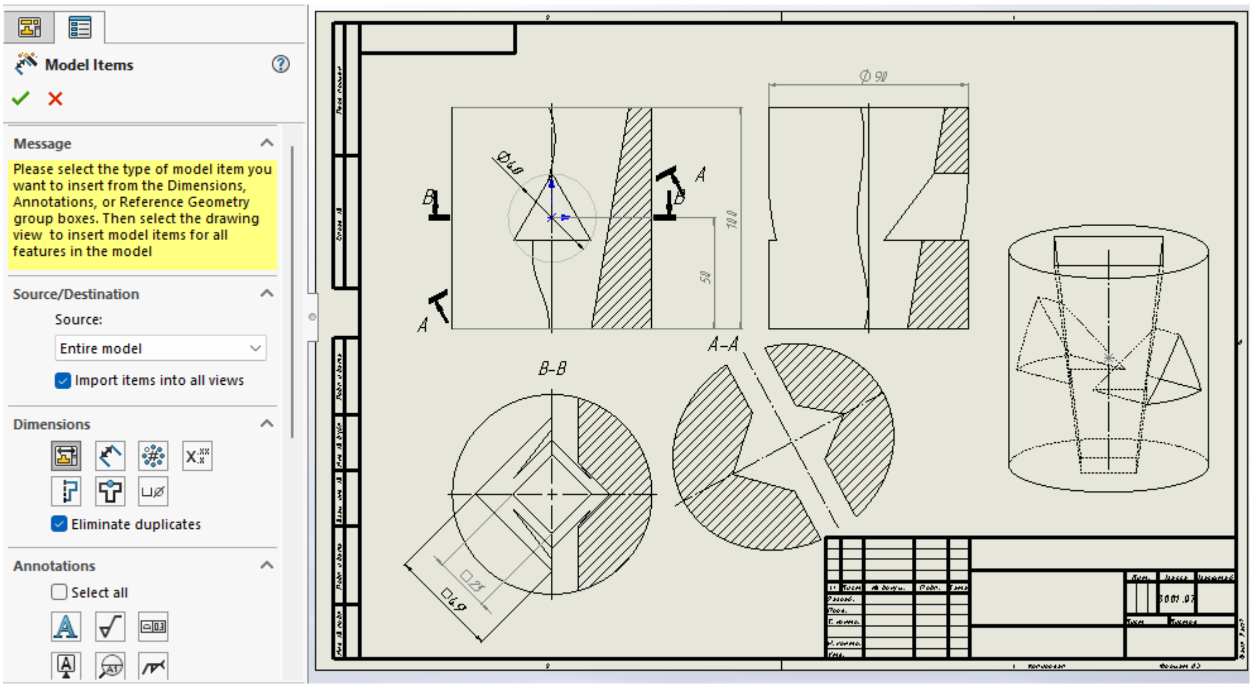


Fig. 4.19

Lecture №5

ASSEMBLY MODELING IN SOLIDWORKS

Lecture Objective: To learn the basic tools and techniques for working in assembly mode in SolidWorks.

An assembly drawing is created by adding and modifying existing assembly components. Adding components is reflected by inserting new components.

An assembly is a node composed of two or more parts, called components. Assembly files have the extension **.sldasm**. The location and orientation of components in the assembly are defined by mates, establishing relationships between components.

In SolidWorks, an assembly allows combining regular parts and structures created in a "part" file. The number of these parts is limited only by the computer's capability. An assembly contains elements interconnected into a single node. This not only allows visually perceiving the mechanism but also conducting additional research and obtaining graphical and digital characteristics of the studied object.

Assembly design can be carried out in two ways: bottom-up or top-down.

Bottom-up assembly involves assembling a structure from pre-existing parts. To build such an assembly, the parts must be previously designed and saved in separate files. The assembly or node is assembled from these parts similar to real-world assembly. During assembly, the parts need to be placed in the three-dimensional assembly space and conditions for their connection with each other specified.

In top-down assembly design, a layout sketch of the assembly is created first, and individual parts are then built based on it. These parts are immediately integrated into the overall assembly. This type of assembly is convenient because changes to the layout sketch automatically affect the dimensions and configurations of the parts that make up the assembly.

Let's examine the assembly modeling process using the example of constructing a connecting rod (fig. 5.1).

After launching the program, we open a new document window by clicking on the corresponding icon at the top of the screen or using the associated shortcut. Then, we select the **Assembly** icon (fig. 5.2).

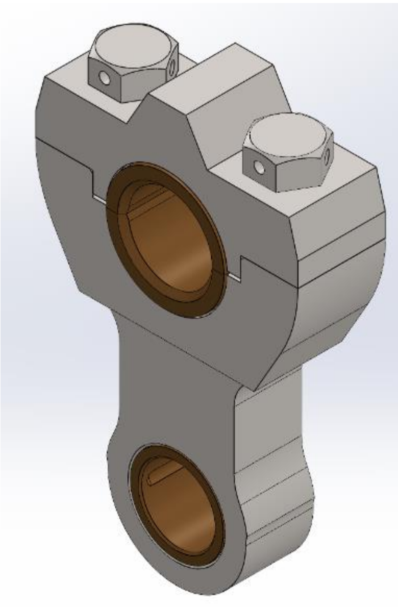


Fig. 5.1

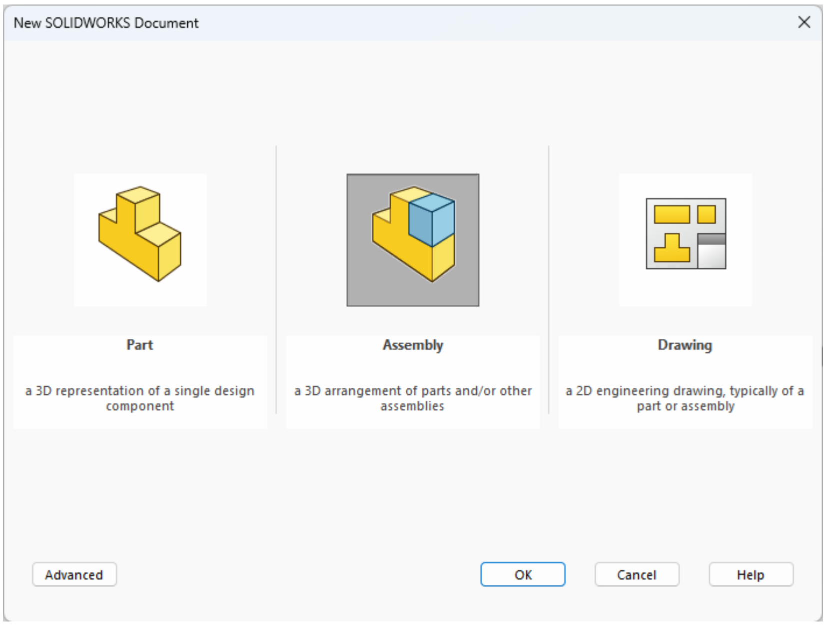


Fig. 5.2.

Creating an assembly starts with adding components to the workspace. When creating an assembly in SolidWorks, the software automatically launches the component addition tool. If some files are already open, they will be displayed in the property panel. Before selecting a component, click on the folder icon . To choose a component from the list, click on it and confirm . Alternatively, you can select from the list and drag it onto the workspace.

If components have not been loaded yet, on the left side in the **Property Manager**, the program will prompt you to choose a file to insert. This can be either a single part or an entire assembled unit.

To select components for use in creating an assembly, go to the «**Begin Assembly**» menu and click on «**Browse**». Then, navigate to the necessary files in the «**Open**» window. Confirm your selection by clicking the «**Open**»" button (fig. 5.3).

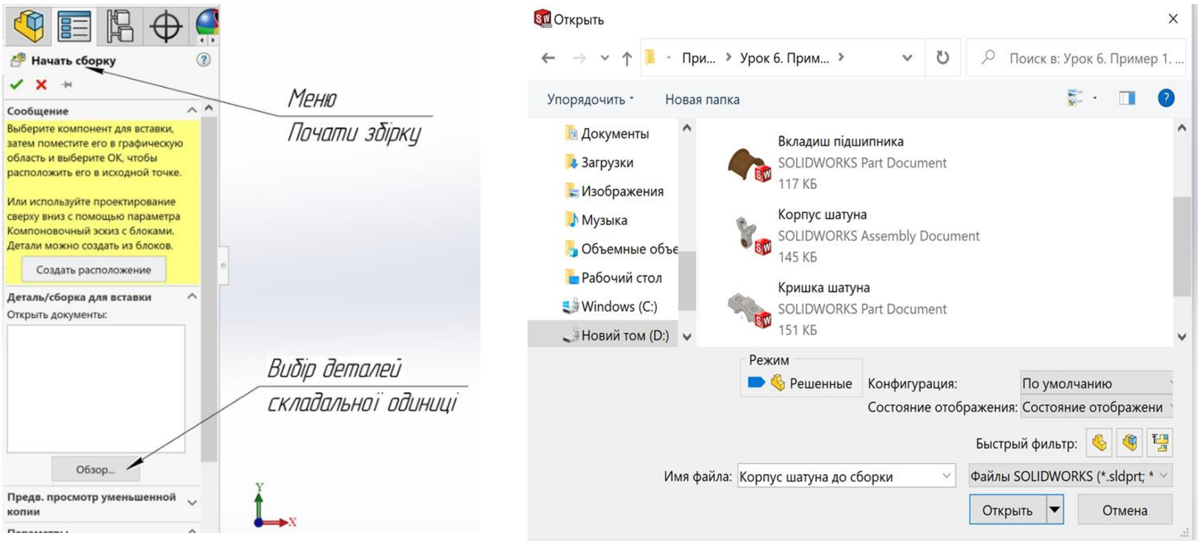


Fig. 5.3

After adding the necessary components, click on the checkmark icon Після того як перші необхідні компоненти додано натискаємо (fig. 5.4).

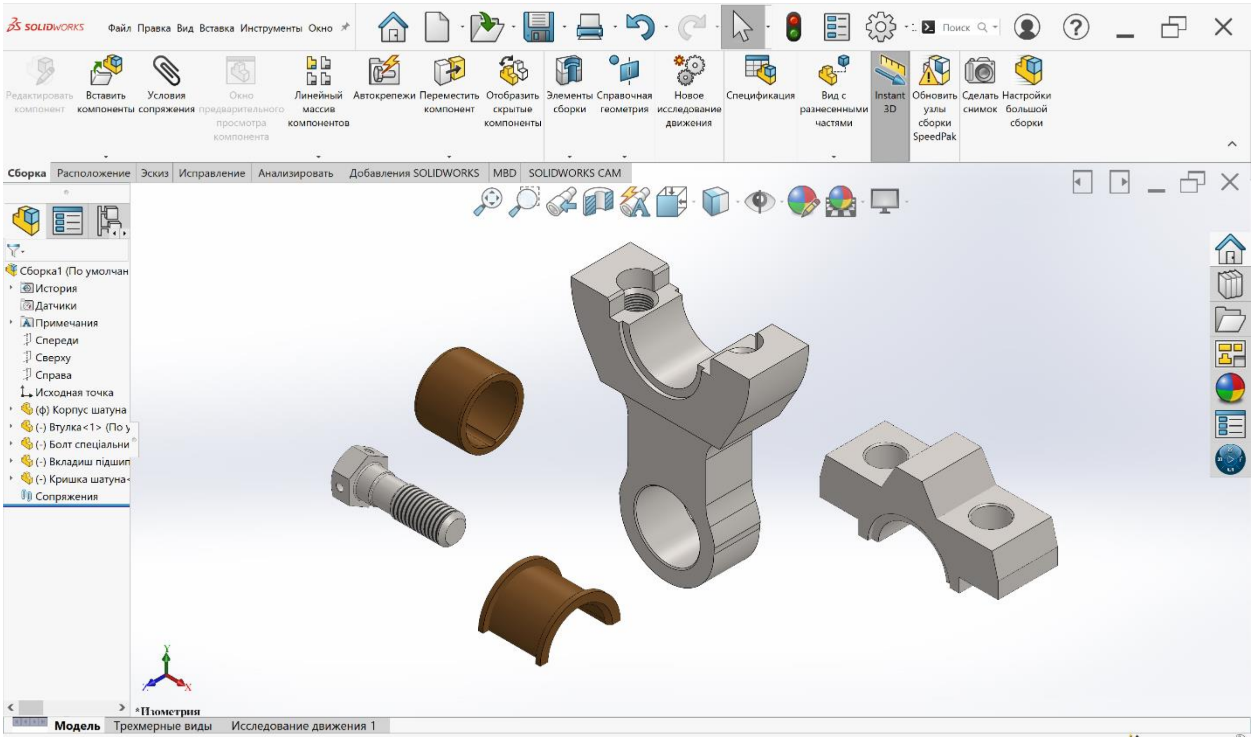


Fig. 5.4

Please note the changes in the interface. Two new tabs, "**Assembly**" and "**Layout**," have appeared in the **Command Manager** (fig. 5.5). These tabs contain commands for working with the assembly.

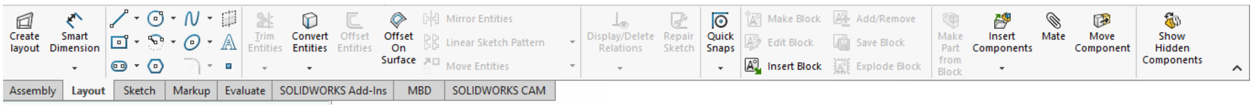


Fig. 5.5.

FeatureManager Design Tree also looks different. Instead of displaying model elements, it now shows assembly components (fig. 5.6). To the left of each component, there is an arrow. Clicking on this arrow opens the FeatureManager for that component. Further down is the representation of the component: details are displayed in yellow, assemblies in blue.

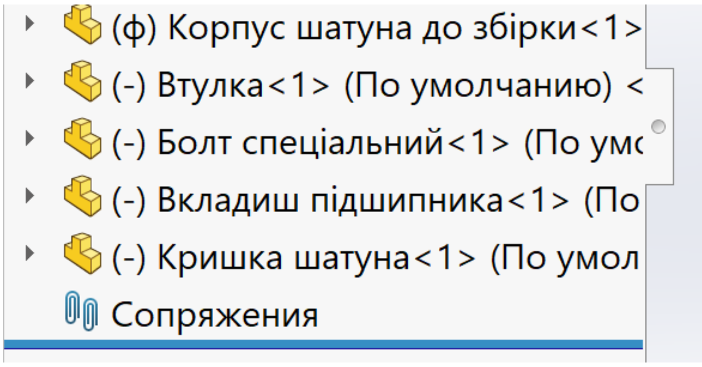


Fig. 5.6.

Then follows information on component definition: (f) means that the component is fixed and its position in space is determined (cannot be moved); (-) means that the part is not fixed and its position in space is not determined; (+) – overridden; (?) – unresolved. The absence of the (-) prefix means that the component's position is fully defined (fig. 5.7).

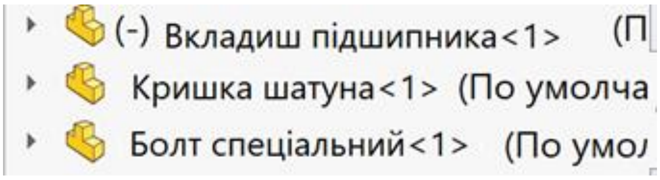


Fig. 5.7.

The first assembly component is automatically fixed. To fix or release a component, you can right-click on the component name in the **FeatureManager tree**.

Next is the component name and its serial number. This is necessary to distinguish identical components. For this assembly, you need to add another special bolt and a second bearing insert. Adding identical components can be done using the Insert Component command on the Assembly Tasks toolbar, or by copying. To do this, press and hold the Ctrl key on the keyboard and drag the component onto the workspace. You can also drag the component from the FeatureManager tree (fig. 5.8).

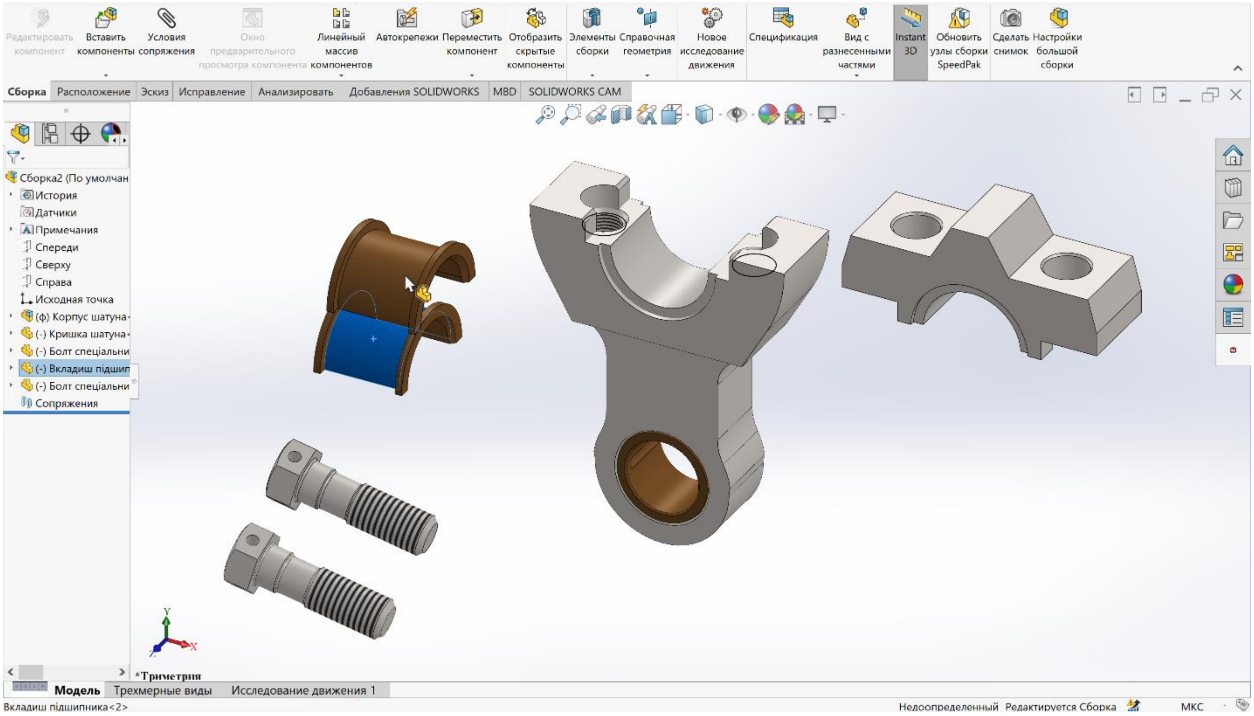


Fig. 5.8.

Unfixed components in the assembly space can be easily moved by holding down the left mouse button.

To rotate unfixed components, you can hold down the right mouse button. Components in the assembly will move or rotate only within the degrees of freedom defined by the mates (fixed and fully defined objects cannot change their position).

To assemble the components into a single structure, you need to specify mate conditions - geometric relationships between assembly components. When adding mates, you need to define allowable directions of linear or rotational movement of the components. The sequence in which mates are added to a group does not matter; all pairs are solved simultaneously. To do this, activate the Mate Conditions button on the Assembly toolbar .

У загальному випадку для створення збірок можна використовувати наступні види сполучень, які розташовуються в розділі **Стандартні сполучення** (рис. 5.9).

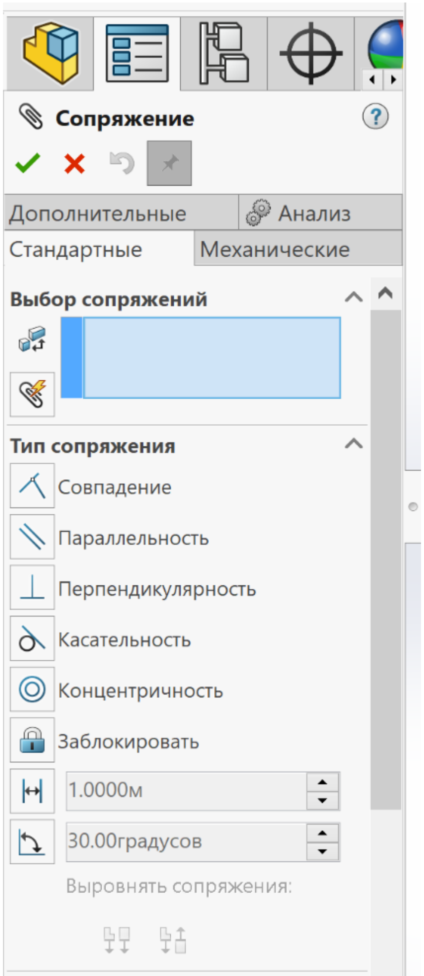


Fig. 5.9.

 **Збіг** – елементи деталей (осі, кромки, поверхні, грані) збігаються на нескінченності;

 **Паралельність** – вказує на паралельне розташування граней, поверхонь, кромок або осей деталей;

 **Перпендикулярність** – обрані елементи розташовуються під кутом 90°;

 **Дотичність** – вказує на дотичність відзначених поверхонь, при цьому хоча б одна поверхня повинна бути неплоскою (сферична, циліндрична, конічна);

 **Концентричність** – забезпечує концентричне розташування циліндричних, конічних, сферичних поверхонь і кромок;

 **Заблокувати** – дозволяє прив'язати два компоненти збірки друг до друга, зберігаючи їх взаємне розташування й орієнтацію;

 **Відстань** – виділені поверхні, осі, кромки розташовуються на зазначеній відстані;

 **Кут** – виділені елементи розташовуються під

деяким заданим кутом.

Standard mates are applied only for standard surfaces (plane, cylinder, cone, etc.), while for more complex ones, additional mates need to be used.

Умови сполучення компонентів

Умови сполучення визначають взаємне розташування компонентів у збірці відносно один одного. Для додавання сполучення в **Диспетчері команд** перемикаємось на вкладку *Збірка, Умови сполучення* . Елементами сполучення можуть бути грані, кромки, вершини компонентів. Більшість сполучень виконуються між парою об’єктів. *Збіг* і *Концентричність* – два найбільш часто використовуваних типи сполучення.

Розглянемо додавання необхідних сполучень при побудові збірки шатуна. Почнемо з умови співпадіння кришки та корпусу шатуна. Вказуємо грані цих деталей та обираємо тип сполучення *Збіг* (рис. 5.10). Обрані грані будуть розташовані в одній площині.

Наступним кроком додаємо концентричність поверхонь. Для цього треба вказати на отвір в кришці шатуна й на відповідний отвір в корпусі шатуна (рис. 5.11).

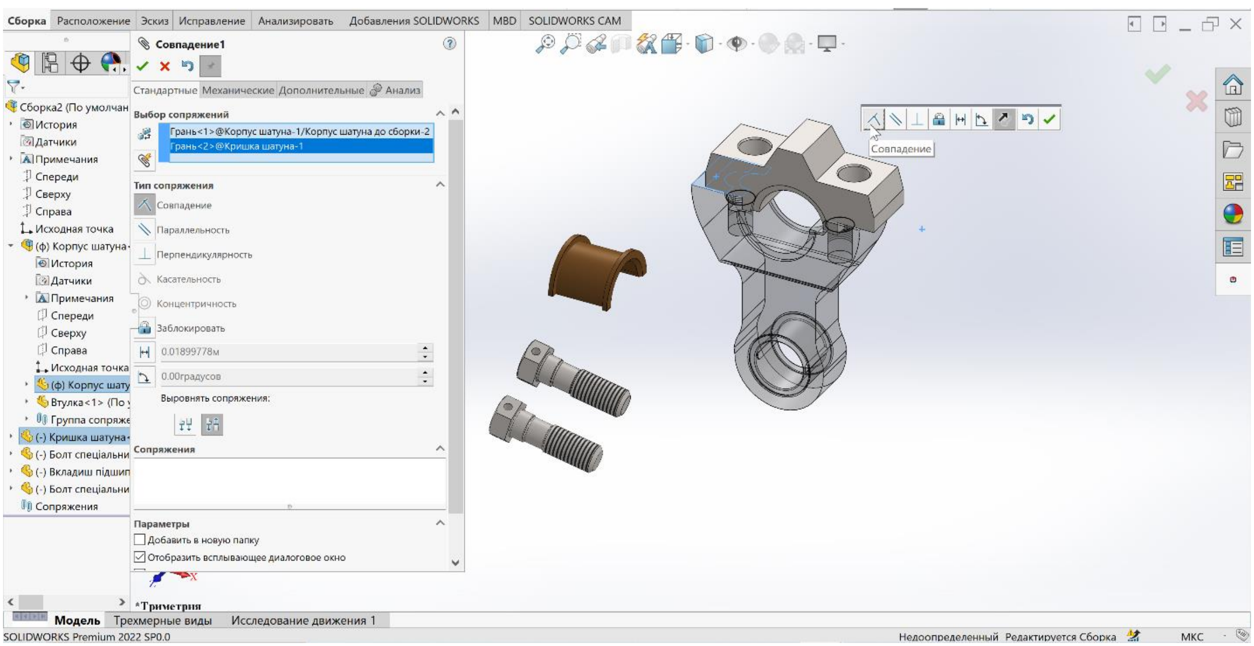


Fig. 5.10.

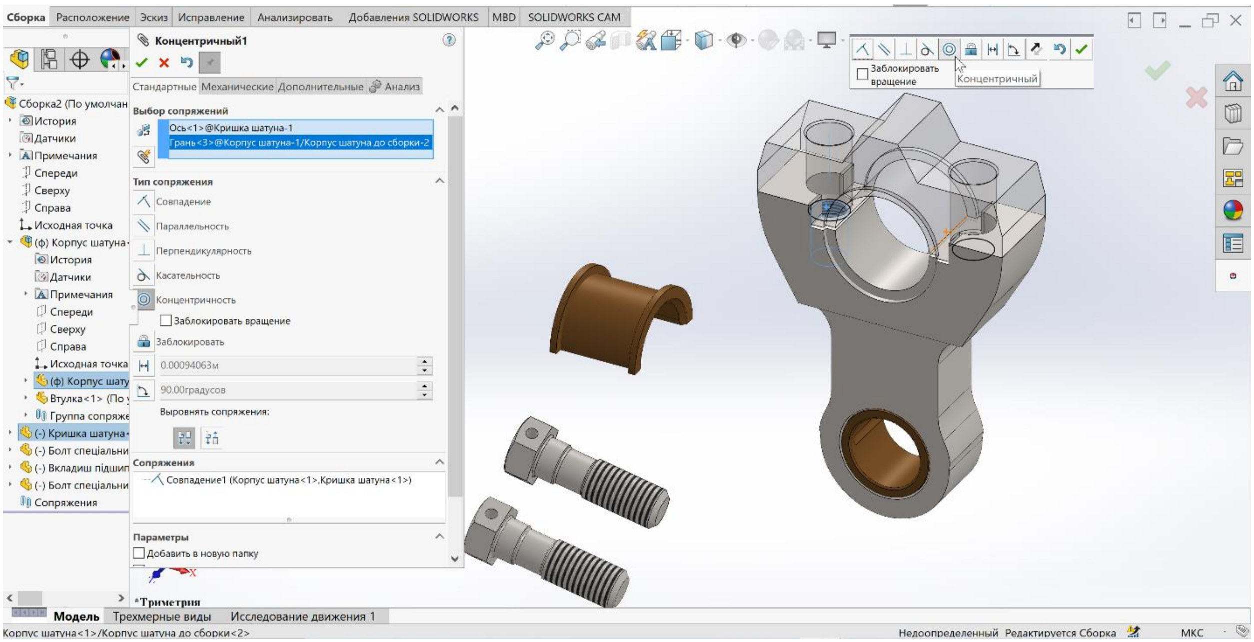
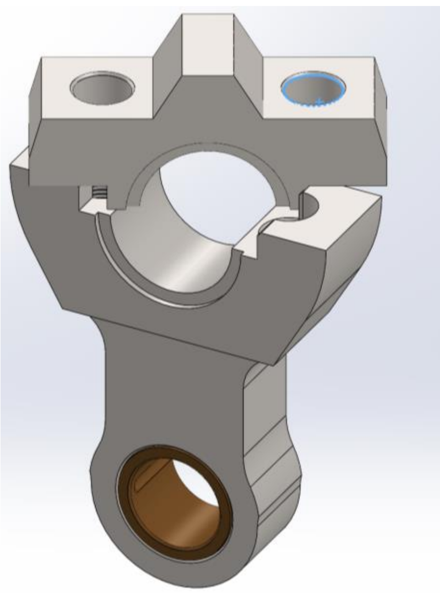


Fig. 5.11.

We temporarily close the mates window and try to move the piston cover. We see that the component still has one degree of freedom - rotation around the axis of the hole (figю 5.12). To fully constrain it, we need to add parallelism between the end faces of the connecting rod body and the cover (figю 5.13).



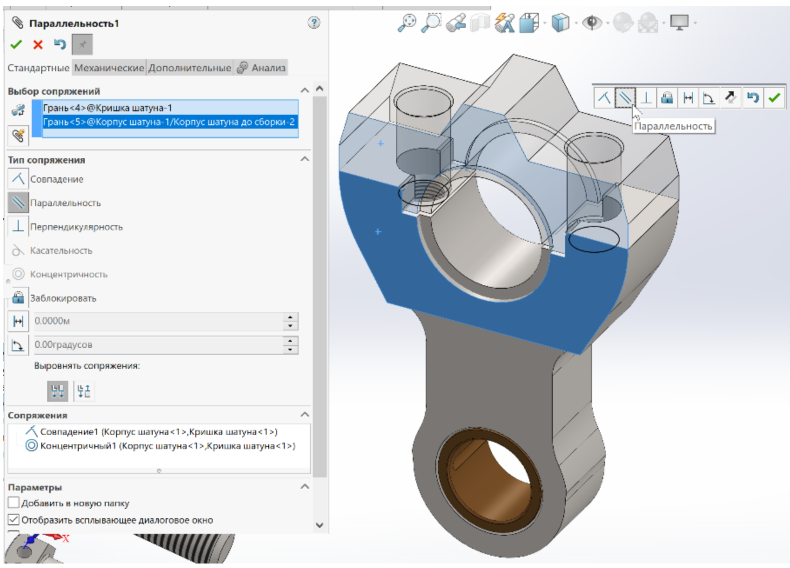
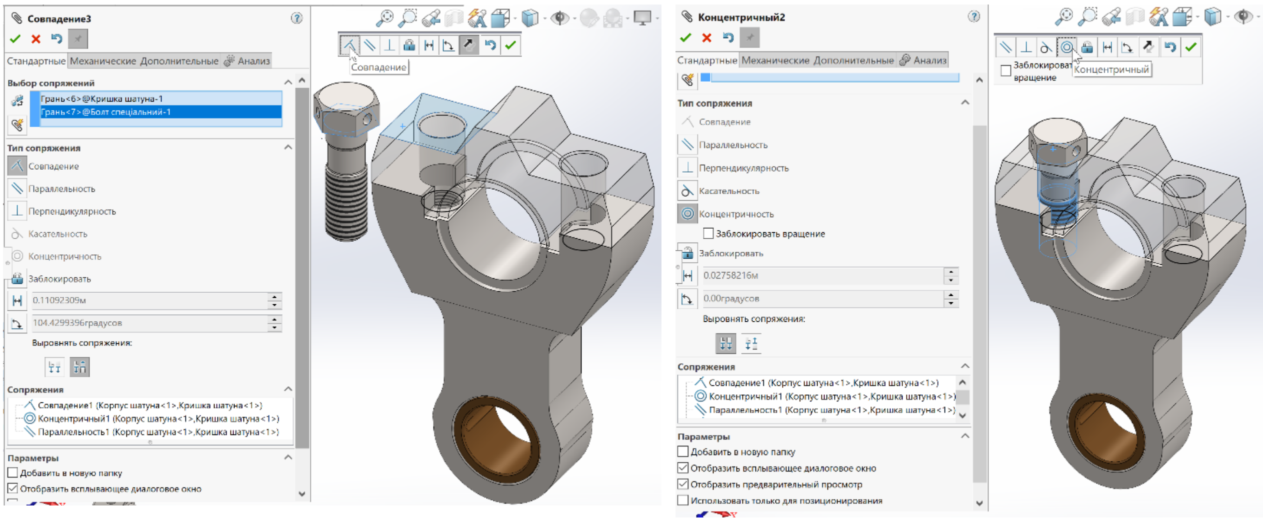


Fig. 5.12. Fig. 5.13.

The next step is to connect the cover to the body using special bolts. We apply mate conditions such as *Coincident* (see figure 5.14 a), *Concentricity* (see figure 5.14 b), and *Parallelism*, as the bolt is not fixed and can rotate. To fully constrain it, we need to add parallelism between the end faces of the cover and the bolt head (see figure 5.15).



a b

Fig. 5.14.

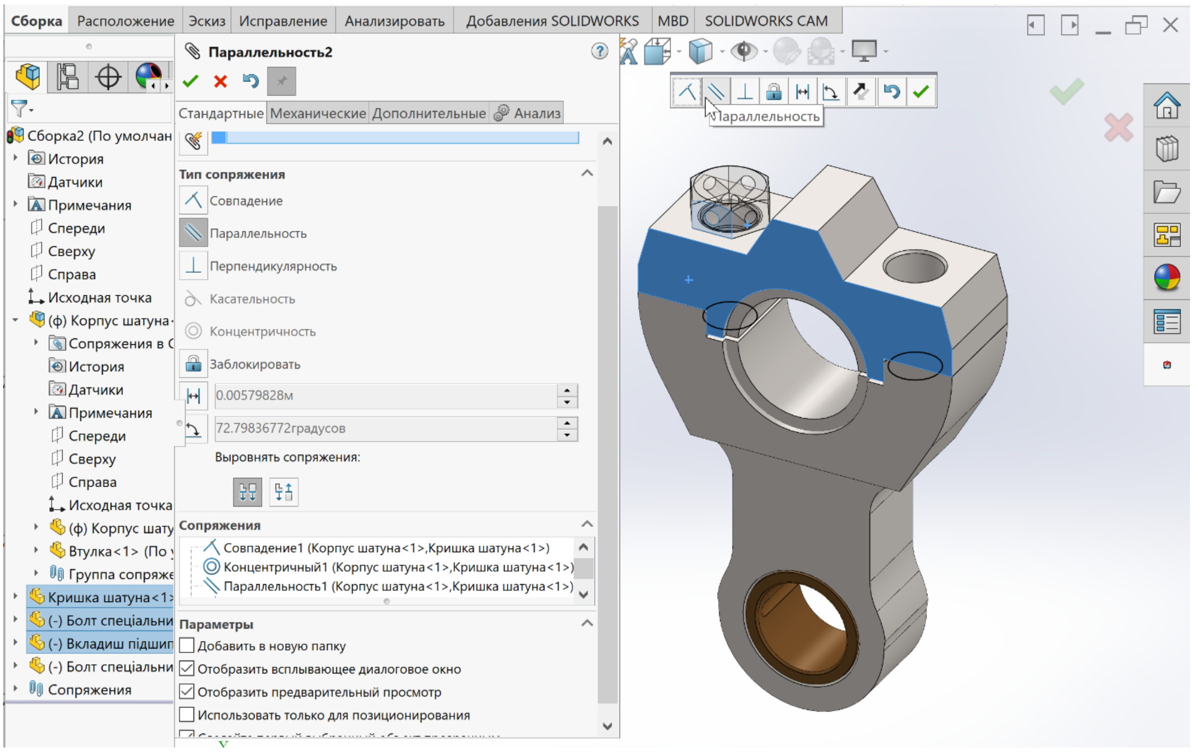


Fig. 5.15.

Similarly, we add the second bolt to the assembly.

Next, we mate the bearing insert with the body. The mate conditions required for these components are *Concentricity*, *Coincidence*, and *Parallelism*.

Once the assembly is completed (see figure 5.16), its correctness is verified by its determinacy. Depending on the state of the components, the assembly may be determined, underdetermined, or overdefined.

The assembly is considered **determined** if all components are fully defined and there are no conflicts.

It is **underdetermined** if at least one component is not fully defined.

It is **overdefined** if at least one component is overdefined.

The state of the **Assembly**, like the state of the part, is displayed in the status bar (see figure 5.16).

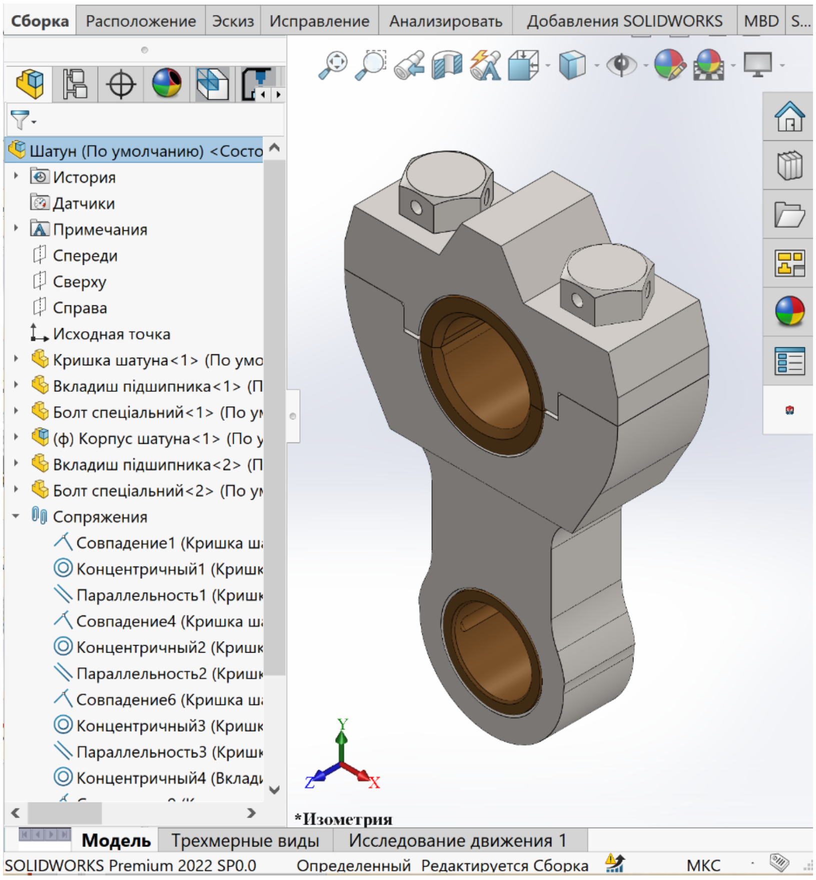


Fig. 5.16

Lecture №6

ADDITIONAL TECHNIQUES IN SOLIDWORKS ENVIRONMENT

Lecture Objective: to explore additional techniques in SolidWorks: mouse gestures, hotkeys, instant 3D, and model preparation for 3D printing**.**

Mouse Gestures

Let's consider the mouse gesture tool, which will be useful in construction and accelerates the work process in SolidWorks.

Mouse gestures are a circular menu (Figure 6.1) that appears when you right-click in the workspace and drag the mouse in the direction of the desired tool. Control elements will differ depending on the type of document: "Part", "Assembly", "Drawing", "Sketch".

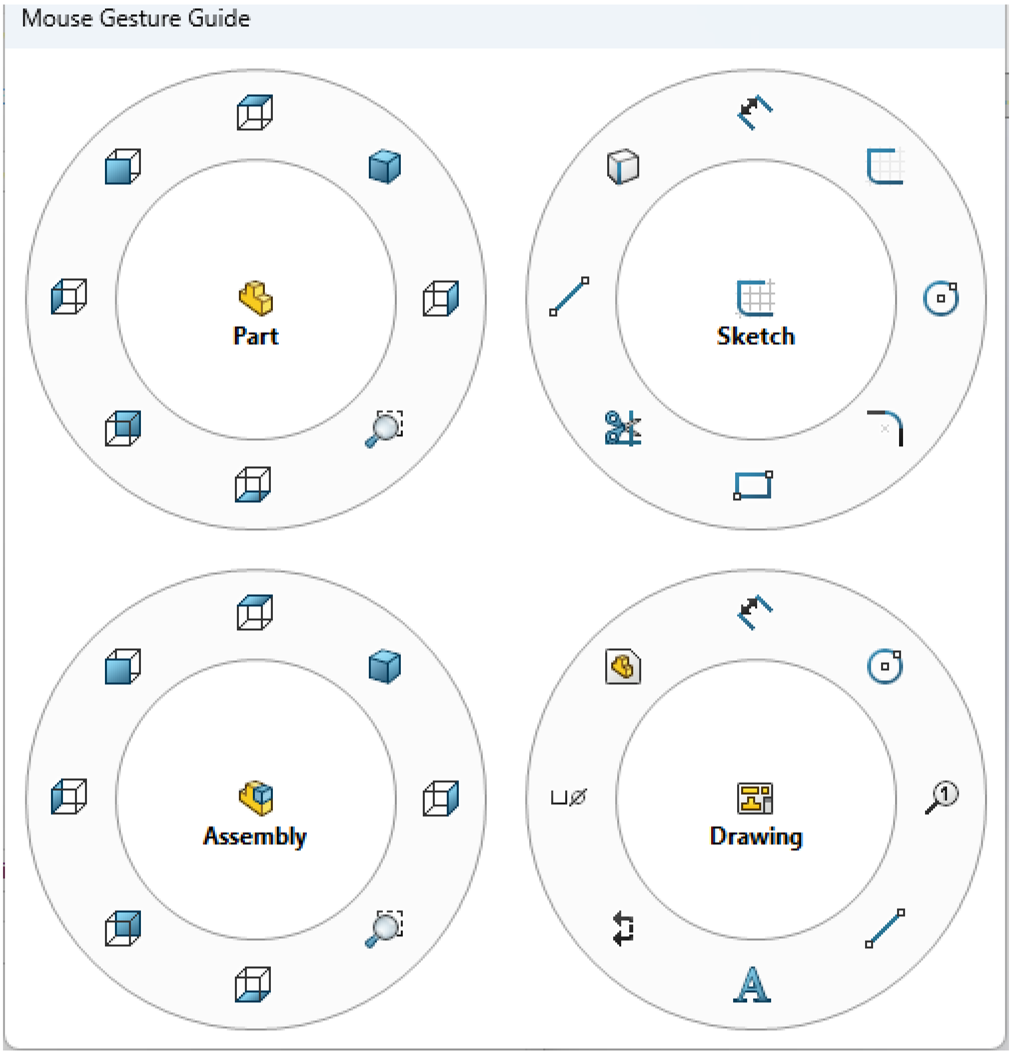


Fig. 6.1.

You can independently assign tools that will be active for a particular type of document. To do this, go to the *Parameters – Settings – Mouse Gestures* dialog box (Figure 6.2).

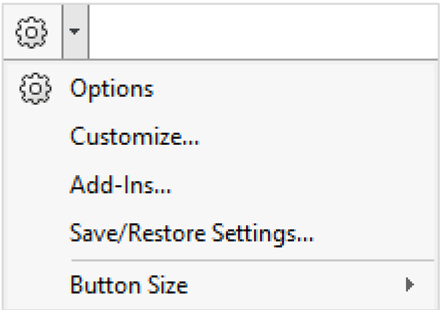


Fig. 6.2.

In this window, you can enable or disable this tool (Figure 6.3). You can also choose the number of mouse gestures: 2-3-4-8-12. By default, 4 gestures are selected. Figure 6.3 shows an example with 8 gestures. Also in this menu, you can configure which tools will be on the compass in a particular document by simply dragging them from the list of commands to the corresponding compass.

The following actions will be performed:

- If you drag a tool to an empty position, the tool is added to the mouse gesture list.
- If you drag a tool to an occupied position, it replaces the tool in the mouse gesture list.
- If you drag a tool while holding down the **Ctrl** key from one position in the mouse gesture list to another position, it is copied to the second position.

This function allows you to bring frequently used commands and tools to quick access.

After finishing the configuration, click OK.

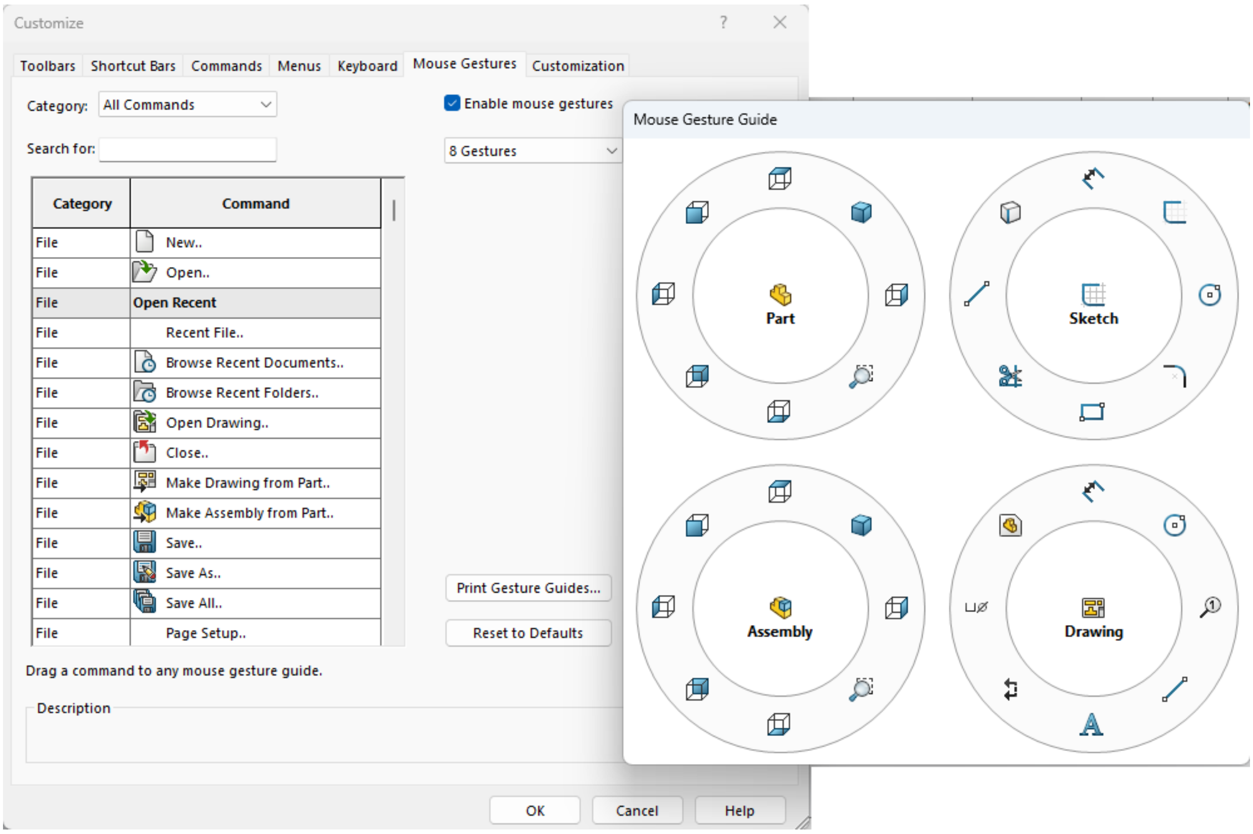


Fig. 6.3.

Hotkeys in SolidWorks

Hotkeys in SolidWorks are keys or key combinations for quick access. You can familiarize yourself with the list of hotkeys in the Keyboard dialog box (Figure 6.4). You can also create your own hotkeys.

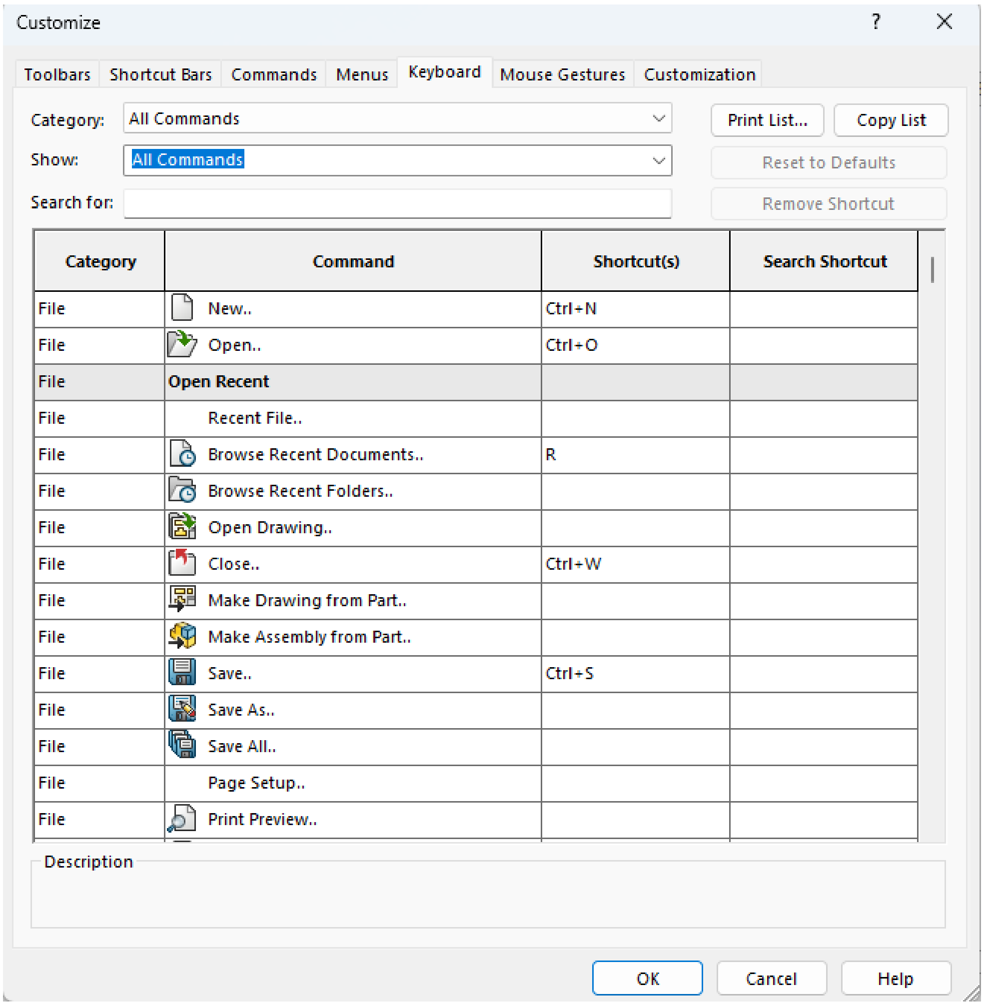


Fig. 6.4

Instant 3D

Let's consider another useful tool called **Instant 3D** 

. It allows you to quickly create a 3D model and modify its geometry using model markers.

To activate this mode, go to the **Features** tab and enable the **Instant 3D** command. Then select the sketch of the part. In our example, it's a circle. You will see an arrow appear next to the mouse pointer (Figure 6.5).

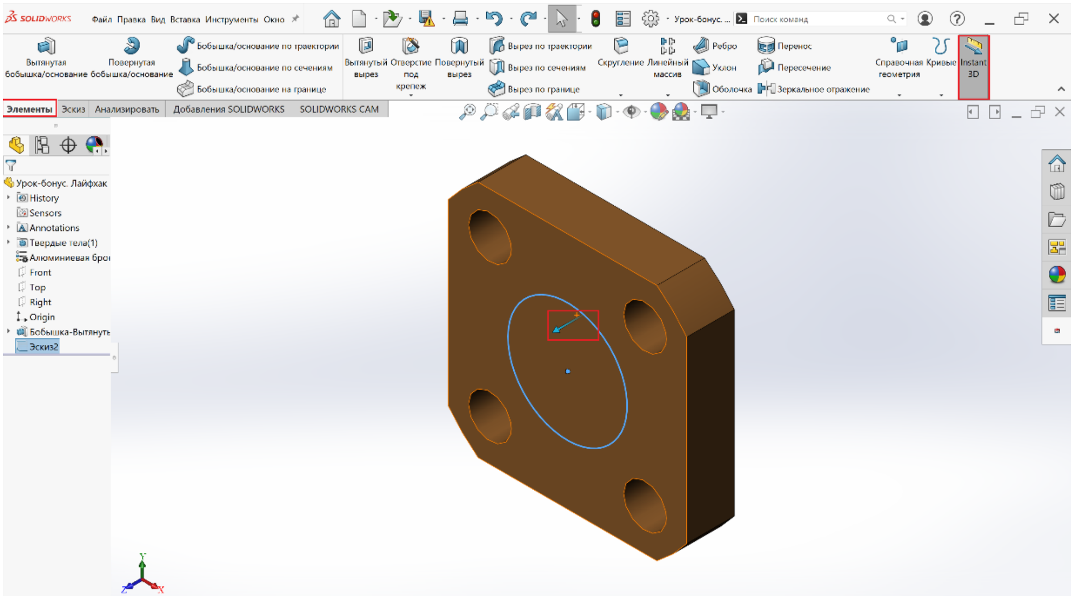
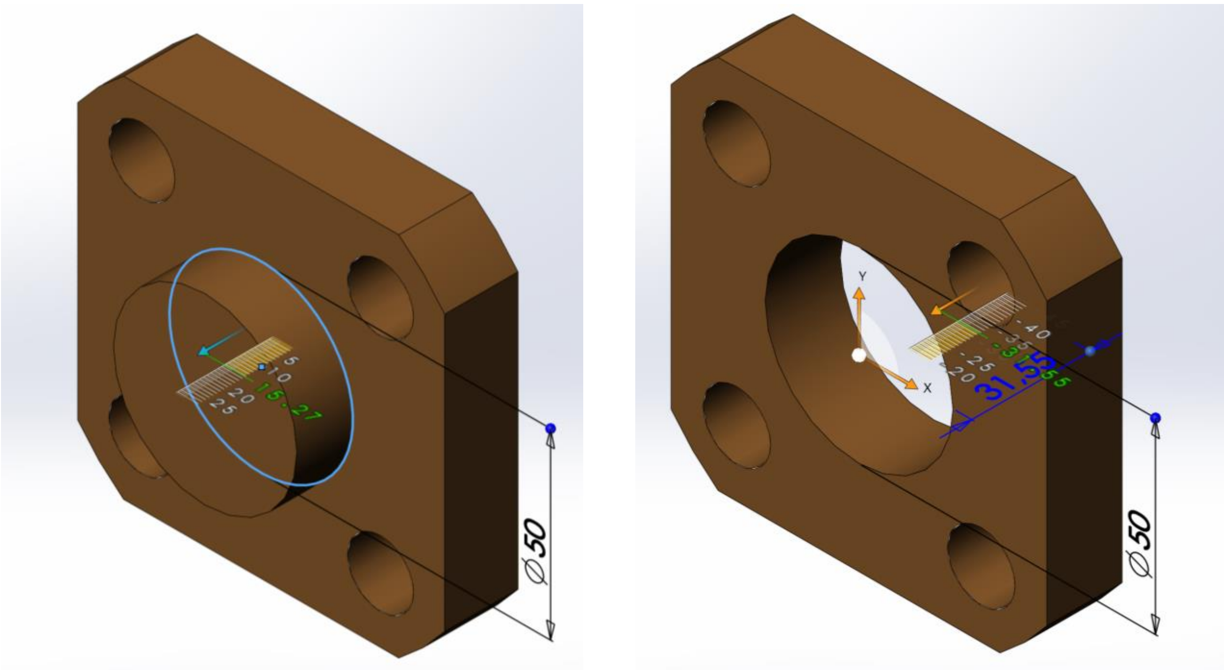


Fig. 6.5

Dragging it, you can create an extruded boss (Figure 6.6 a) or an extruded cut (Figure 6.6 b) depending on the direction of the drag. Note that when creating a cut, the scale bar shows negative values.



a b

Fig. 6.6

This way, you can adjust not only the depth of the feature but also the dimensions of the sketch itself. To do this, drag the point next to one of the dimension arrows (Figure 6.7). You will see a scale bar next to it (Figure 6.8).

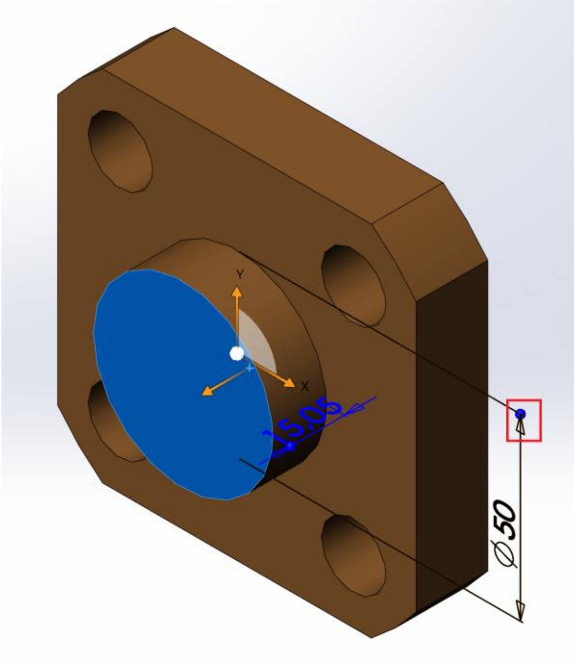


Fig. 6.7

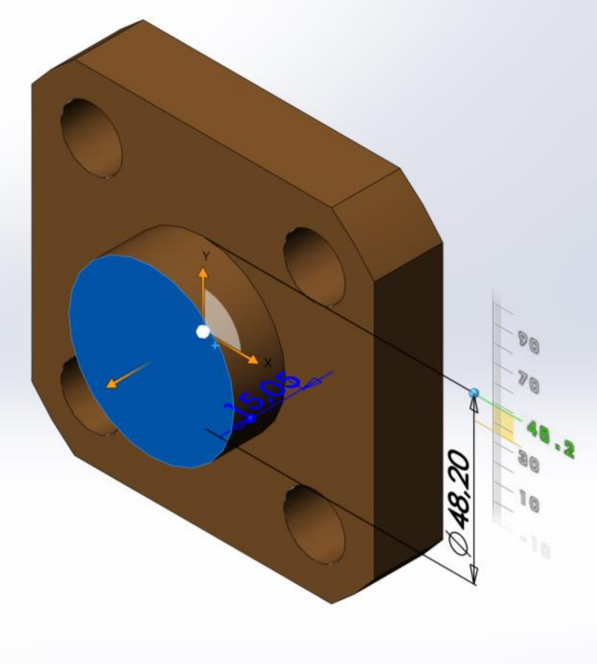


Fig. 6.8

To verify an element in a sketch

The "**Check Sketch**" tool verifies sketches for errors in the contour that may hinder the creation of the element. This tool checks for common errors across all types of contours, and if you specify the element type, it also checks for the type of contour required for that specific element type. When errors are detected, problematic geometry is highlighted.

For example, if a sketch has been created and the "**Extruded Boss/Base**" command is selected, but SolidWorks detects an error and reports that the sketch contains multiple unclosed contours (see figure 6.9).

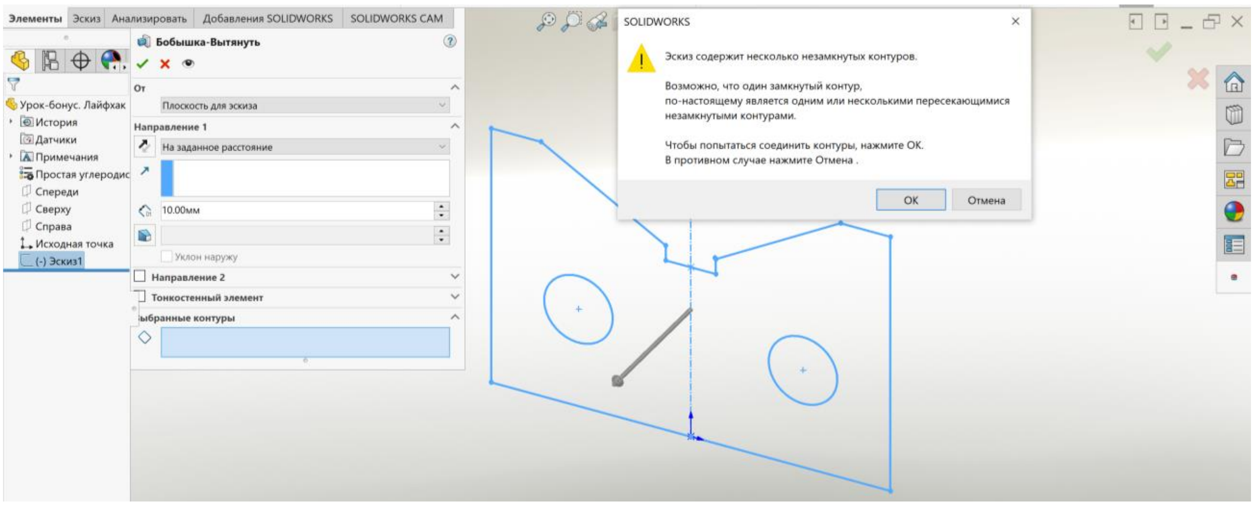


Fig. 6.9

Often, when quickly creating a sketch, short segments, known as artifacts, are created which are not noticeable at normal detail scales. To avoid wasting time searching for errors, we return to sketch editing mode. Navigate to the main menu toolbar *Tools* -> *Sketch Tools* -> -- for Feature (Figure 6.10). In the window that opens, select *the Feature to Apply* - *Extruded Boss/Base* (Figure 6.11 a). Press Check. Then, a diagnostic message appears on the screen: the sketch cannot be used to create the feature because the endpoint is split by multiple elements (Figure 6.11 b). Press OK, and the error will be displayed on the screen: the extra segment will be highlighted in blue (or green) depending on the SolidWorks version and circled (Figure 6.12). The problem will also be described: three or more contour segments meet at this point.

Delete the highlighted line and run the check again. Now there are no errors. The system reports that no errors were found. You can exit the sketch and execute the Extruded Boss/Base feature.

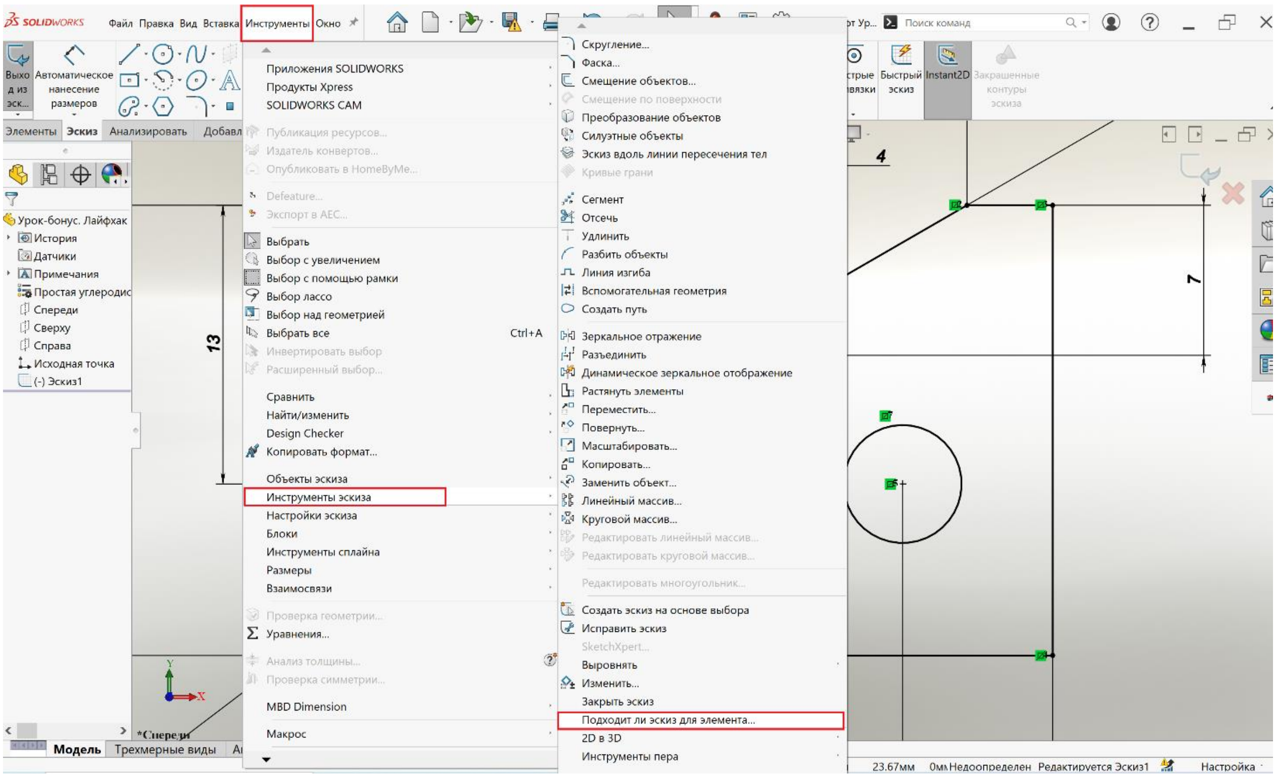
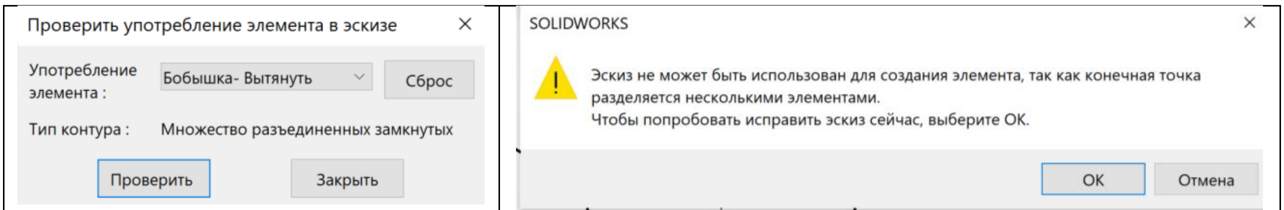


Fig. 6.10



а	б

Fig. 6.11

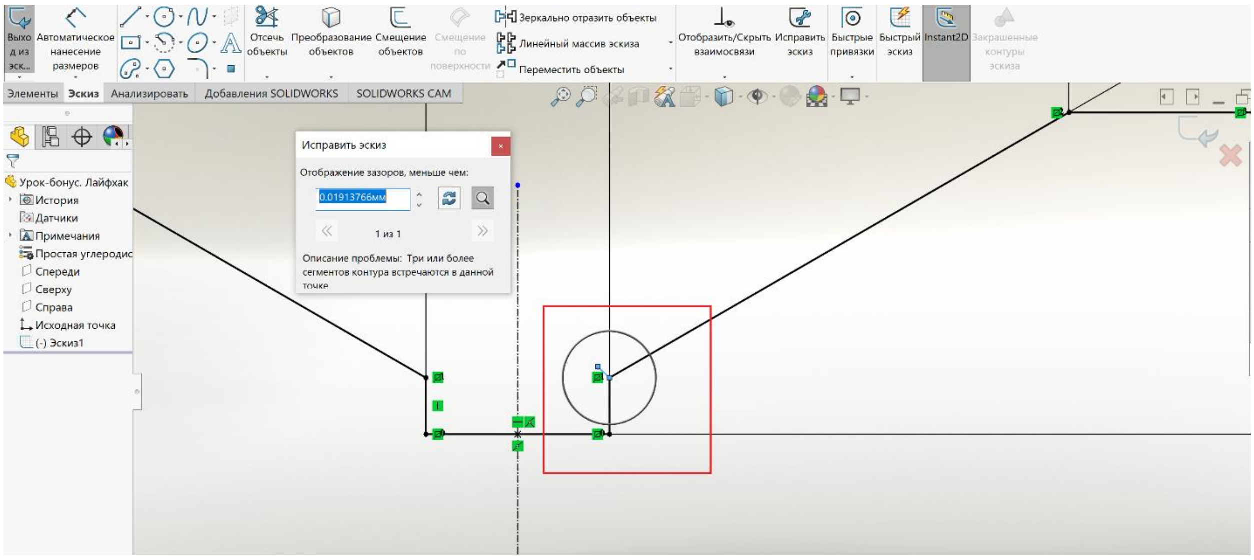

Автоматическое нанесение размеров
Отсечь объекты
Преобразование объектов
Смещение объектов
Смещение по поверхности
Зеркально отразить объекты
Линейный массив эскиза
Переместить объекты
Отобразить/Скрыть взаимосвязи
Исправить эскиз
Быстрые привязки
Быстрый эскиз
Instant2D
Закрашенные контуры эскиза
Элементы
Эскиз
Анализировать
Добавления SOLIDWORKS
SOLIDWORKS CAM
Урок-бонус. Лайфхак
История
Датчики
Примечания
Простая углеродис
Спереди
Сверху
Справа
Исходная точка
(-) Эскиз1
Исправить эскиз
Отображение зазоров, меньше чем:
0.01913766мм
1 из 1
Описание проблемы: Три или более сегментов контура встречаются в данной точке

Fig. 6.12

Literature

1. Kompiuterna hrafika: SolidWorks: Navchalnyi posibnyk. M.M. Koziar, Yu.V. Feshchuk, O.V. Parfeniuk. Kherson: Oldi-plius, 2018. 252 s.
2. Inzhenerna hrafika v SolidWorks: Navchalnyi posibnyk. S.I. Pustiulha, V.R. Samostian, Yu.V. Klak. Lutsk: Vezha, 2018. 172 s.

Information resources

1. https://help.solidworks.com/