MINISTRY OF EDUCATION AND SCIENCE OF UKRAINE

NATIONAL TECHNICAL UNIVERSITY
«KHARKIV POLYTECHNIC INSTITUTE»

METHODOLOGICAL INSTRUCTIONS

for performing the laboratory work

«Configurations of parts»

on the discipline «Fundamentals of Electronic Device Design»
for students of specialty 171 «Electronics»
for the first (bachelor) level of higher education of all forms of training

Kharkiv 2023

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INTRODUCTION

Methodical instructions for the laboratory work on the discipline «Fundamentals of Electronic Device Design » for students of all forms of education in the specialty 171 «Electronics» on the discipline «Fundamentals of Electronic Device Design», which is learned by the students of the «Industrial and Biomedical Electronics» department.

SOLIDWORKS is a computer-aided design (CAD) system that allows you to create electronic models of parts and assembly units with the ability to generate working drawings based on them.

In the process of preparing for the class, the student has to study questions about the features of using the SOLIDWORKS automated design system and master the skills of creating models in the program.

In the course of performing this laboratory work, students work out the possibilities of creating configurations of parts in various ways and have to:

– know the sequence of actions when building part models and the main methods of creating their configurations;

– be able to apply the acquired theoretical knowledge when creating details, as well as configurations of these details.

Methodological instructions include:

– the purpose of work;

– a step-by-step description of the sequence of work performance on an example;

– tasks for the laboratory work;

– control questions.

Methodological instructions provide methods for creating configurations of parts and consider an example of step-by-step configuration creation in several ways, which will help in effective and practical mastering the material on the discipline «Fundamentals of Electronic Device Design».

Performance and protection of laboratory work is carried out individually.

LABORATORY WORK
«CONFIGURATIONS OF PARTS»

Goal: to study the basic methods of building configurations of part models in the SOLIDWORKS automated design system.

1 THEORETICAL KNOWLEDGE

In these methodological instructions, the terminology according to DSTU 3321:2003, DSTU GOST 2.052:2006, DSTU 3278-95 and the terminology used in the SOLIDWORKS program will be used:

- **Part** – is a product that is made from material of the same brand, without performing assembly operations;
- **Assembly** – is a product, the component parts of which are connected together at the producer's enterprise**;**
- **Drawing** – is a graphic design document that contains an image of the product, defines its design and contains data according to which the product is developed, produced, controlled, installed, operated and repaired**;**
- **Drawing** – is a drawing containing images of details and other data, according to which it is produced and controlled;
- **Assembly drawing** – is a drawing containing an image of an assembled unit and other data, according to which it is assembled (produced) and controlled;
- **Model** consists of a geometric model that describes the geometric shape, dimensions and other properties of the product, which depend on its shape and dimensions and the free attributes of the model – dimensions, tolerances, text or symbols that are required to determine the geometry of the product or its characteristics and may include technical requirements;
- **Feature** – is an objective feature of a product that can be produced during its creation, operation or consumption. In SOLIDWORKS *Features* are separate functions that are used to construct a certain shape or certain properties of the model.

The process of creating electronic models in SOLIDWORKS is based on the principles of adding and removing material, similar to the methods of real technological processes.

Configuration is one of the options for the constructive implementation of a detail or assembly in the middle of one document (project). Options can have different sizes, features and properties. For example, a single part, such as a bolt, can

have different configurations in which the diameter, length, or shape of the bolt head can vary. In fact, it is a complex of formal description of the model, which is developed, for example, in order to create its own base of typical structures, when designing a group of products of the same nomenclature with different standard sizes (the so-called model range), to develop a simplified version of the project, etc.

Part configurations as a built-in SOLIDWORKS feature allows users to maintain multiple versions of a part, an assembly, or an assembled unit in a single file.

In the SOLIDWORKS software environment, there is a *ConfigurationManager* panel, hereinafter abbreviated as *Configurations*, which allows you to create new configurations of parts and assembled units based on previously built ones, as well as view, edit and delete the corresponding properties of structures.

ConfigurationManager  panel is located on the left side of the SOLIDWORKS program window (Fig. 1).

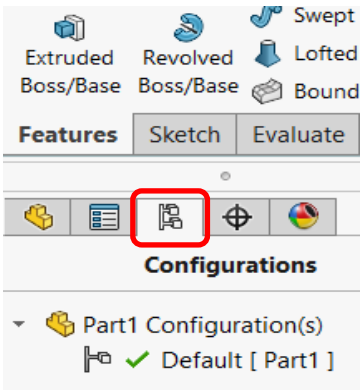


Figure 1 – Location of the *ConfigurationManager* panel

Creating configurations in SOLIDWORKS is possible in all types of program documents, such as *Part*, *Assembly*, *Drawing*.

In documents of the *Part* type it is possible to create a number of parts of different sizes, with different functions, but with outwardly similar features (Fig. 2). Thus, differences between configurations are established by changing the values of dimensions, unsuppres (suppress) features of the model, and changing other parameters (for example, color, material, etc.).

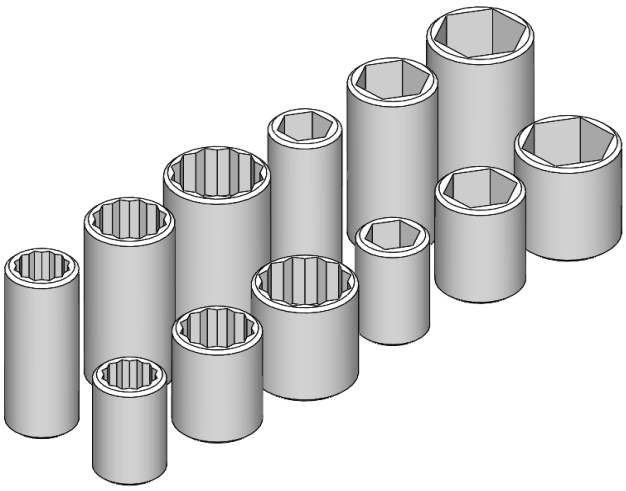


Figure 2 – Example of several bushing configurations

In the image (Fig. 2), all twelve bushing configurations are contained in one file of a part. By assigning different values to the outside diameter and depth of the socket, and choosing whether to use a 12-corner or a 6-corner cut, multiple bushing configurations can be easily created without having to go through the work of recreating all the sketches and features that make up the model. Thus, the configuration of the part allows you to easily change the design solution.

One approach to designing configurations is used – the features that are present in all modifications must be added to the product model, and the transition from one active state to another is carried out by changing the corresponding parameters of the parts. Thus, the transition between different configurations comes down to:

1) changes in the values of numerical parameters (for example, sizes, colors, materials);

2) changes in the state of unsuppres (suppress) features of the model, equations, relations, messages, etc.

In *Assembly* type documents, configurations allow you to create:

− project variants by overriding certain features applied to other project configurations;

− rows of assemblies with different features configurations, different options for assembly parts, sizes or user-defined properties;

− assemblies with parts scattered in space.

In documents of the *Drawing* type you can display the names of configurations created in documents of *Part* and *Assembly* types.

The following elements can be specified in the configuration of the ***part***:

− dimensions of the part;

- status of certain features (suppres/unsuppress);
- material;
- user-defined properties.

There are several methods of creating configurations. Configurations can be created manually, using a parameter table or a parameter calculation table, which can be used to create multiple configurations at the same time. The availability of several methods of creating configurations is due mainly to the number of variable parameters. If you need to create a couple of part options that differ in a small number of sizes or features used (for example, chamfer, rounding, etc.), then it is more convenient to do it manually. In the case where it is necessary to create, for example, all standard sizes of bearings, a parameter table should be used or a parameter equation table should be used.

Each configuration is displayed separately on the *Configurations* tab and has a specific type of icon. Icons mean:

 – the configuration is created manually;

 – the configuration is created using the parameter table;

 – the configuration is created manually and contains an exploded view or derived configuration;

 – the configuration is created using a parameter table and contains an exploded view or derived configuration.

2 CREATING A MODEL FOR CONFIGURATIONS

As an example of creating a configuration using different methods, the «Socket» detail was chosen (Fig. 3).

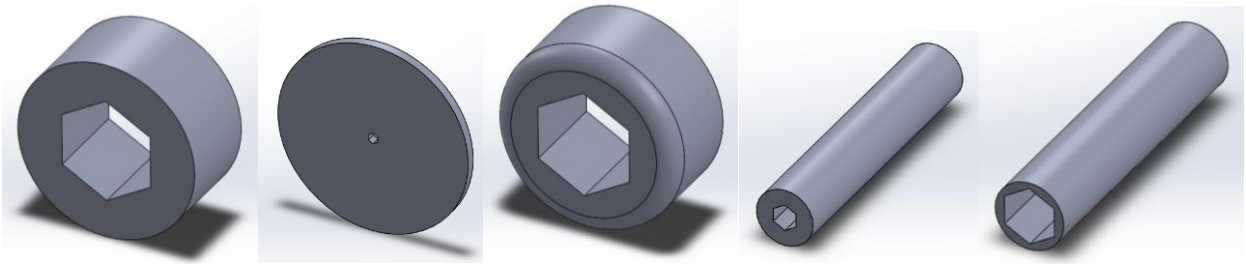


Figure 3 – An example of the configurations of the «Socket» part

Procedure for preparing the model for configuration.

1. Create a model of a part. For this you need:

– to draw a sketch on the projection plane (Fig. 4). The sketch consists of a circle and a hexagon with a common center. In the project, the dimensions can be chosen arbitrarily. In this case, using the *Smart Dimension* command  , you can set the dimensions of the diameter of the circle inscribed in the hexagon – 50 mm, the outer diameter of the bushing – 100 mm.

– to create a simple part by extruding the sketch to 50 mm, that is, using the *Boss-Extrude* element.

2. Display dimensions (sometimes required).

WARNING! The configuration of an arbitrary part can be defined by a large number of dimensions and other parameters. It is necessary to determine the dimensions that will change. To do this, SOLIDWORKS has the ability to display dimensions and their names, as well as change these names.

In this example, there are few dimensions (internal diameter of the hexagonal cutout, external diameter of the socket and thickness of the part). To display the dimensions, right-click on the *Annotations* label, which is located in the *FeatureManager Design Tree* tab, then abbreviated as *FeatureManager*, in the left part of the SOLIDWORKS window (Fig. 5), and select *Show Feature Dimensions*.

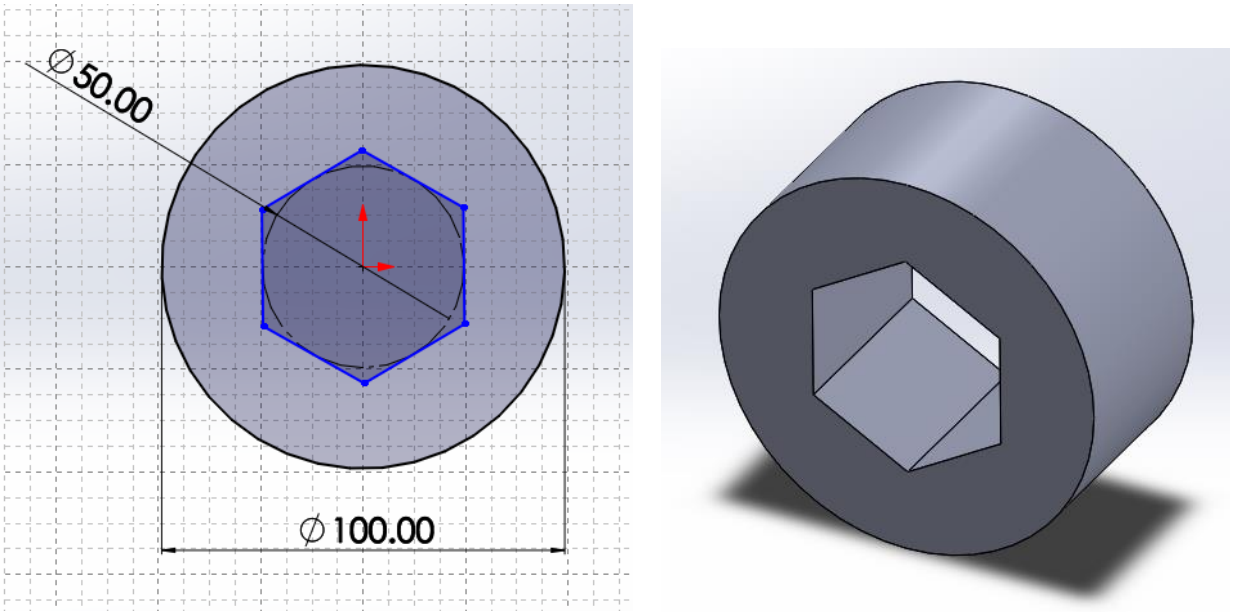


Figure 4 – Sketch and detail model

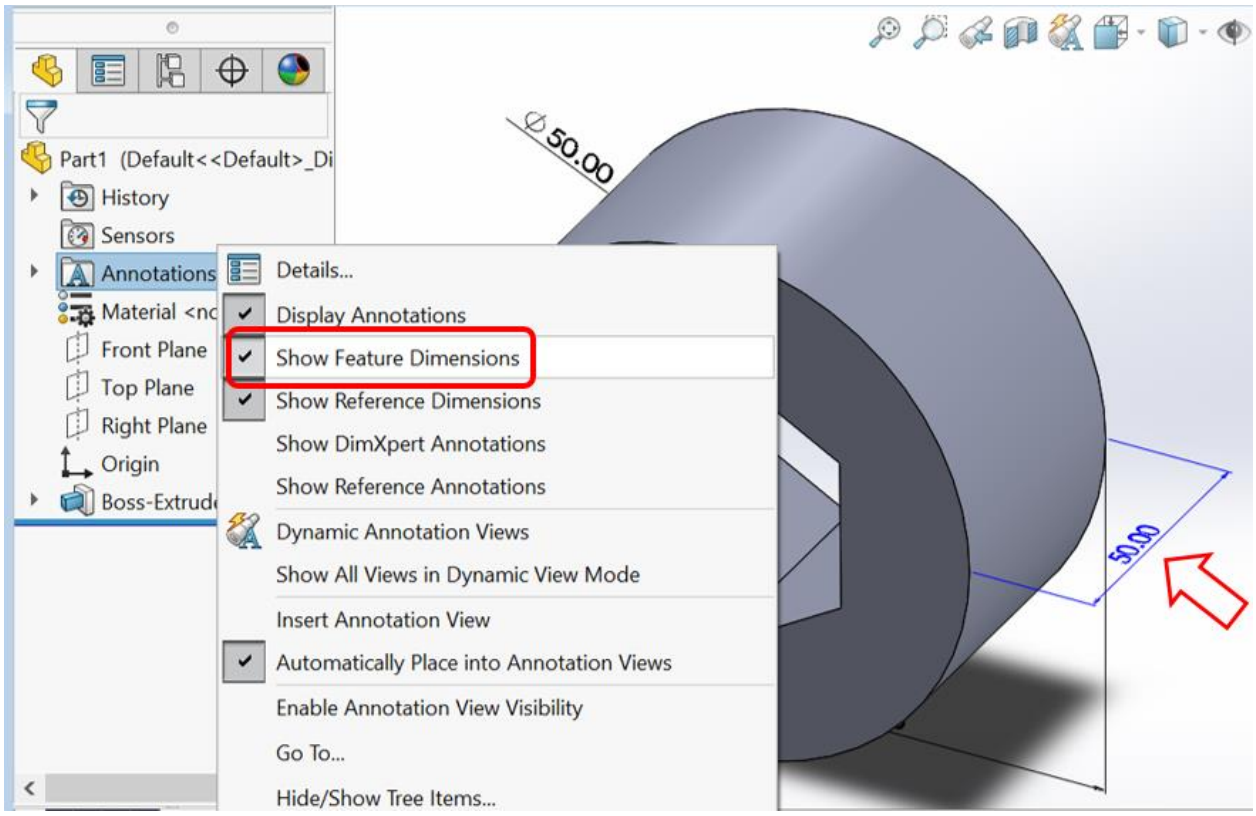


Figure 5 – Display of part dimensions in the design area

If necessary, you can display the name of each size in the design area, since the program assigns it by default.

Ways to display the names of dimensions (Fig. 6):

– in the *View* tab of the main menu, click *Hide/Show* and activate the *Dimension Names* function.

– you need to make them visible in the menu *Hide/Show Items* on the *Heads-up View Toolbar*.

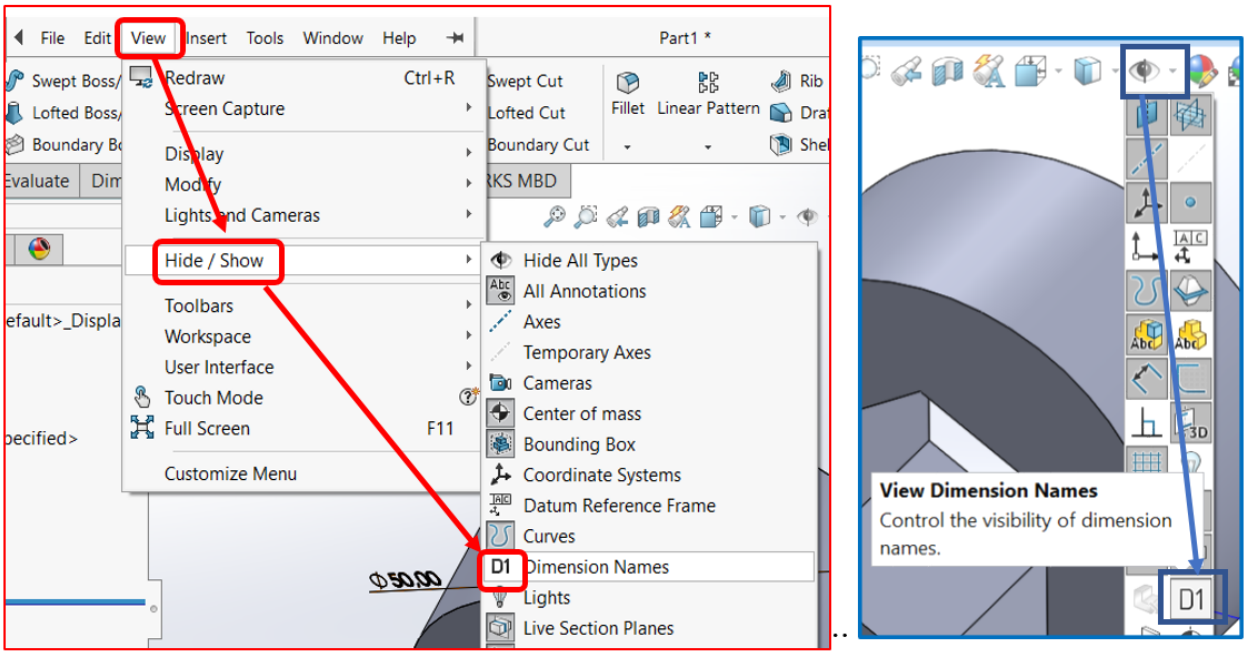


Figure 6 – Ways to display dimension names in the design area

3. Change of the names of the dimensions. Since customizing a model often involves changing dimension values and changing the behavior of a certain tool, it is a good idea to change the names of dimensions and features to simplify the process of creating a configuration. To change the size name, right-click on the numerical value of the size, after that the *Dimension PropertyManager* will appear on the left (Fig. 7).

In this example, two dimensions need to be renamed for the model to more clearly represent which specific dimensions change in the process of creating configurations.

The *Dimension* tab displays a name that refers to the name of the diameter of the circle inscribed in the hexagon – *D2@Sketch*1, where *D*2 is the name of the dimension, *Sketch*1 shows which sketch this dimension refers to. *D*2 is changed to the name *Inner* (Fig. 7), and the name of the outer diameter *D*1 is changed to the name *Outer* (similarly).

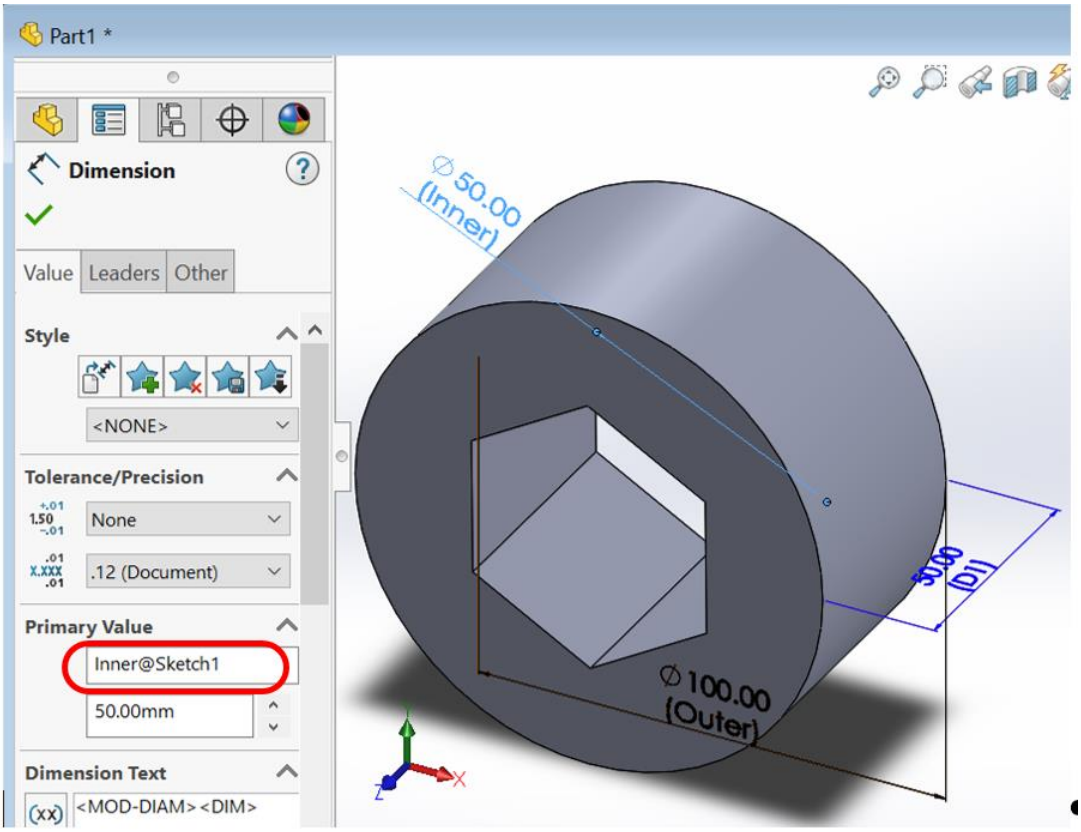


Figure 7 – Displaying the *Dimension PropertyManager* window for changing the dimension name

All elements of the created part «Socket» are displayed in the *FeatureManager* (Fig. 8). In the future, part elements that will be added to other configurations will not be applied to the base part.

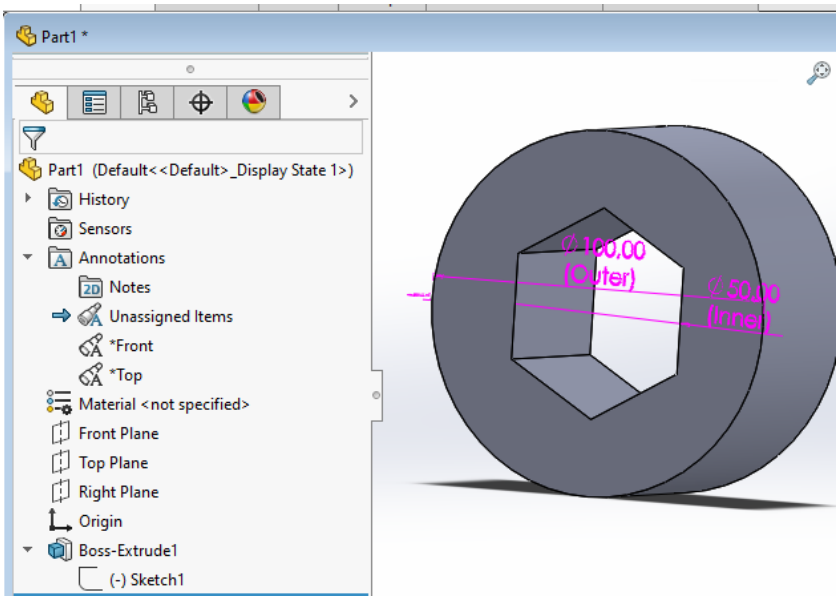


Figure 8 – Part «Socket»

3 WAYS OF CREATING PART CONFIGURATIONS MANUALLY

Creating part configurations manually is more often used in cases where you can quickly create and change part geometry by editing part parameters (dimensions), suppres (unsuppres) of model features. When a feature is suppres, it is temporarily disabled (but not removed) and disappears from a particular model configuration. At the same time, in the *FeatureManager*, this feature becomes inactive for this configuration.

3.1 The first method of creating part configurations manually

The sequence of creating the part configuration.

1. Open *Configurations.*
2. In the tab, you need to highlight the name of the part or node. To do this, right-click on the name of the part so that the context menu appears, and select *Add Configuration* (Fig. 9).

WARNING! A new configuration is always created based on the configuration that is in the active state.

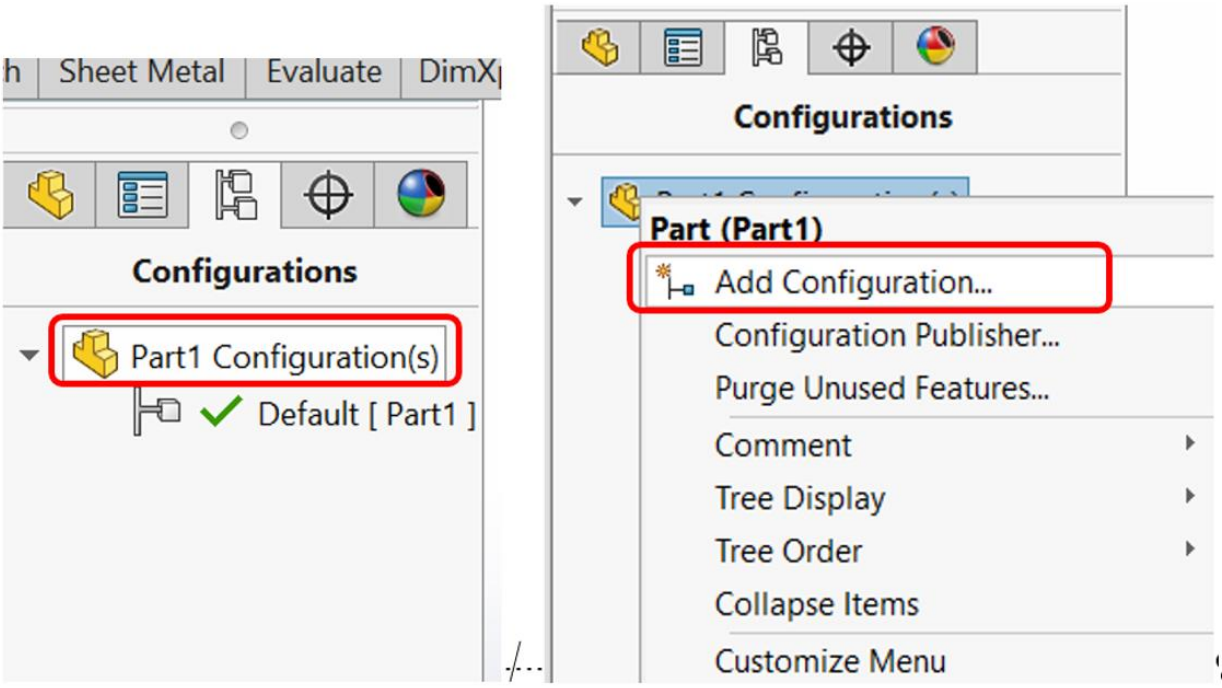


Figure 9 – Adding a new configuration manually

3. In the dialog box (Fig. 10), enter the name of the configuration and, if necessary, specify the features for the new configuration.

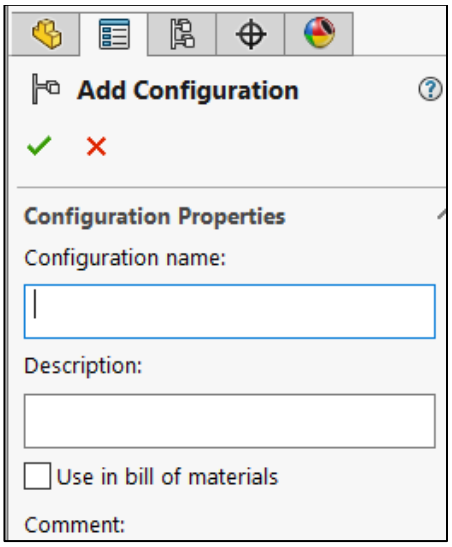


Figure 10 – Creating a new configuration

Tip: It is recommended to give a meaningful name to each configuration to avoid confusion in complex parts and assemblies and to help those who will use the model.

You can specify the color for the configuration in the *Advanced Options* section.

4. Click *OK* to confirm the creation of the new configuration.

In this case, both configurations are the same. Any new part configurations created using this method will retain the settings of the last used configuration. New characteristics for each configuration can be set in the next step.

3.2 Editing a manually created configuration

1. In order to activate the necessary configuration, double-click on its name or right-click on the name of the configuration and select the *Show command* in the context menu *Configuration* (Fig. 11).

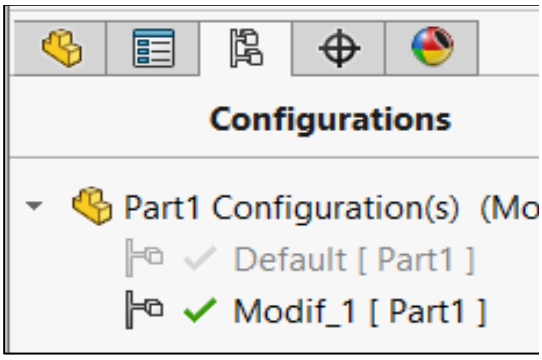


Figure 11 – Activating the configuration

After these actions, the necessary configuration becomes active, and its image appears in the graphic area of the program.

2. Go to *FeatureManager*.

3. Make changes to the model to create a variant of the design solution.

In this case, it is necessary to change:

– on the sketch of the base of the part, change the diameter of the circle inscribed in the hexagon (*Inner*), setting aside 5 mm;

– edit the *Boss-Extrude1*: change the extrusion distance to 5 mm (Fig. 12);

– specify for which configuration this size is used (Fig. 13).

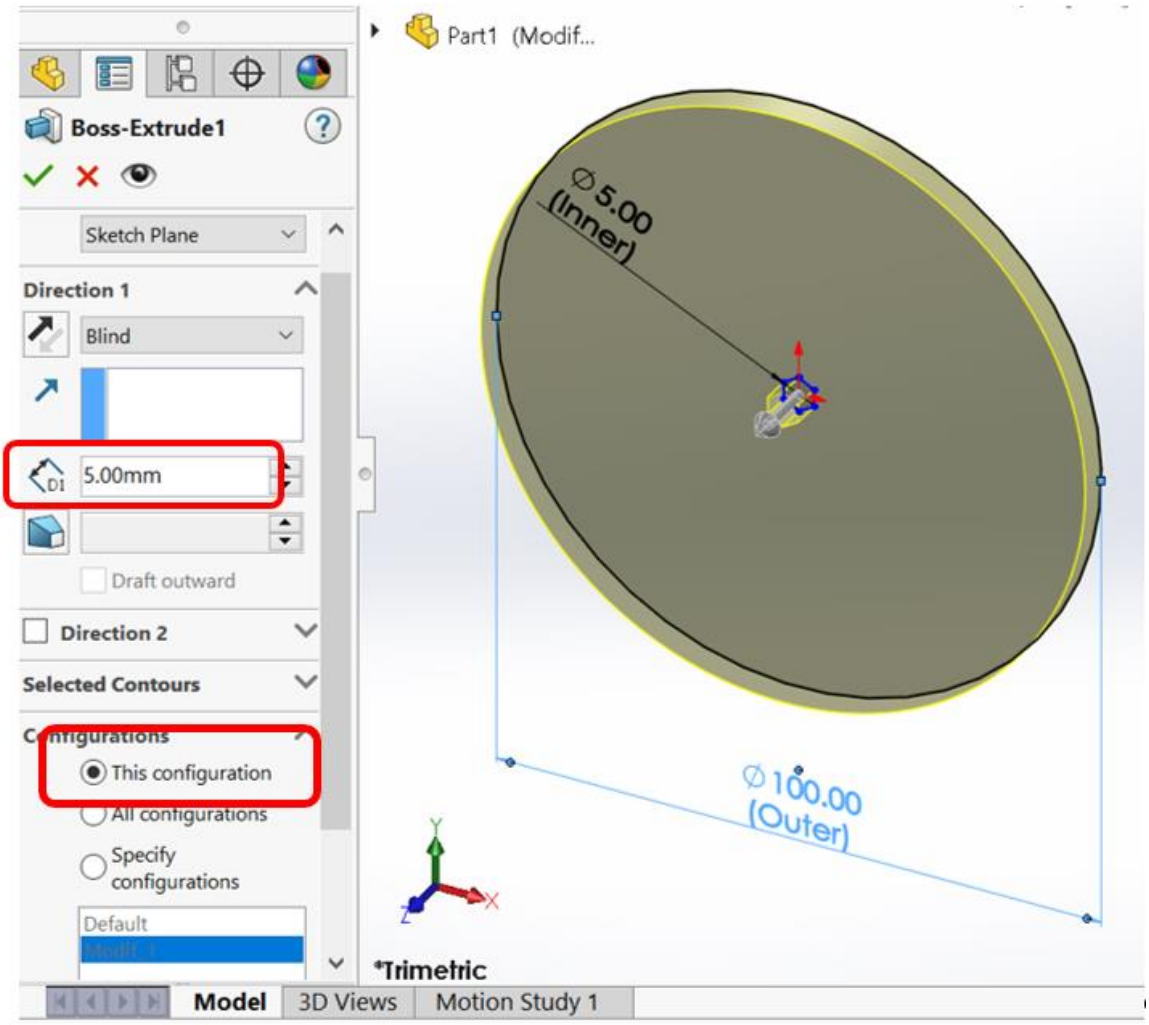
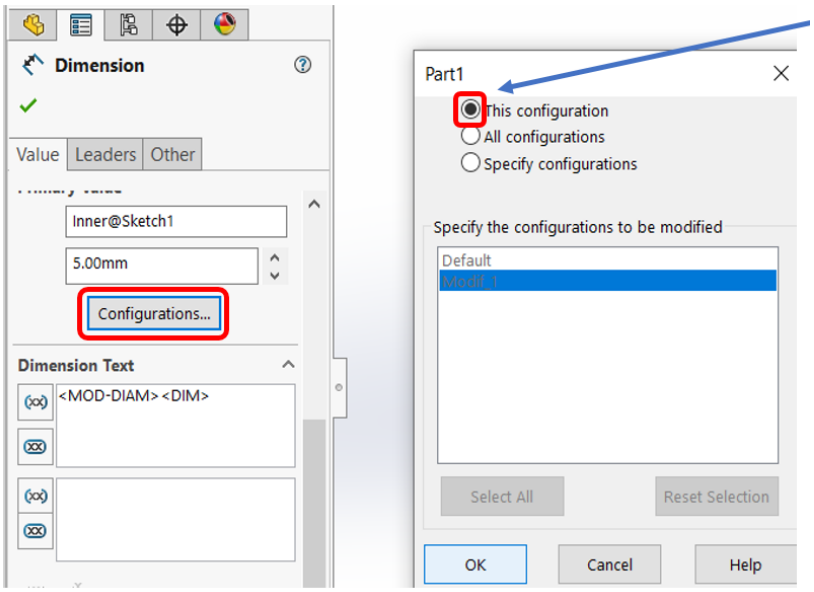


Figure 12 – Selection of parameter assignment of a part for a specific configuration or all configurations



WARNING! When making changes to the configuration, it is necessary to monitor and correctly choose the purpose of the program parameter.

Figure 13 – Selecting a parameter assignment in a sketch for a specific configuration or configurations

You can choose:

– *This configuration.*

– *All configurations.*

– *Specify configurations.*

Select the configuration in the window that appears and press the *OK* button.

4. Save the model.

Thus, on the basis of the basic part «Socket» a new part – configuration of the part was created (Fig. 14).

Initial configuration: ***Manually created configuration:***

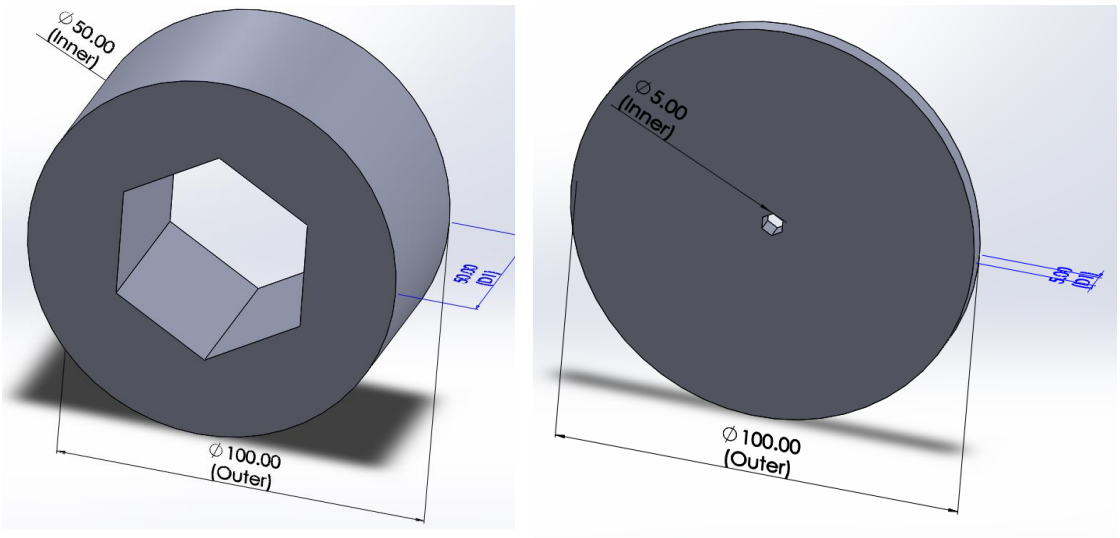


Figure 14 – Available configurations of the «Socket» part

3.3 The second way to create a configuration

In the *FeatureManager*, right-click on the feature of details (for example, *Boss-Extrude1*) and select the command *Configure Feature* (Fig. 15).

Figure 15 – *Configuration Feature* Location

As a result, the *Modify Configuration* dialog box opens on the screen in the form of a table. You can also **use this** *Modify Configuration* to edit previously created configurations:

−adding, removing and renaming configurations (Fig. 16);

−editing features and dimensions (Fig. 17);

− editing and suppressing features or certain properties that occur in defined configurations rations (Fig. 17);

The *Modify Configuration* dialog box is a table where the first column presents a list of available part configurations, and the other columns show configuration parameters (dimensions, features elements, their status, etc.).

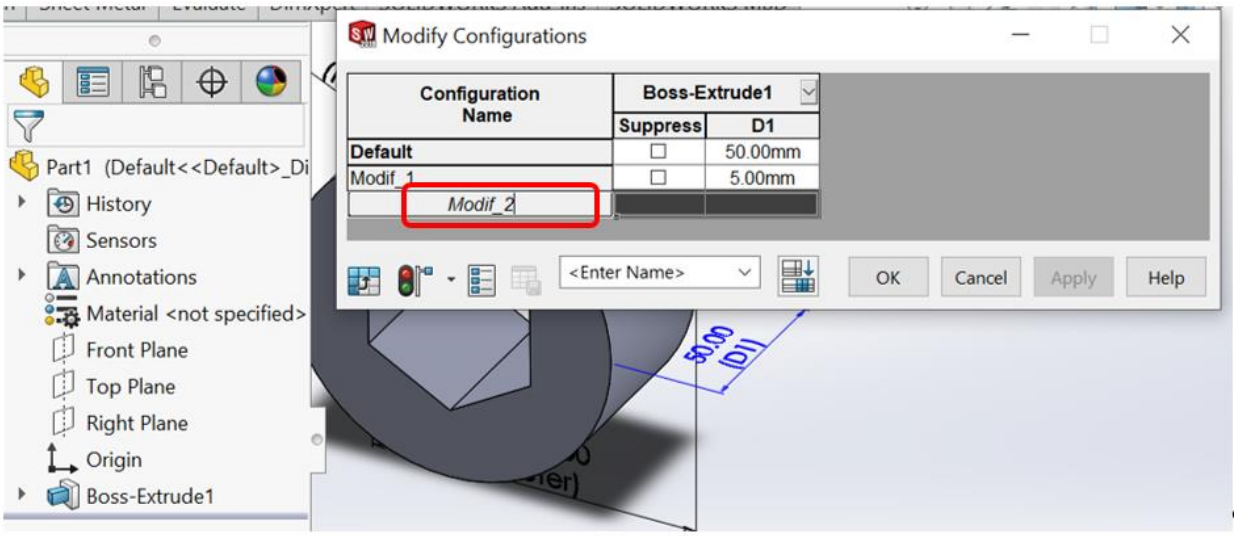


Figure 16 – Entering the name of the new configuration

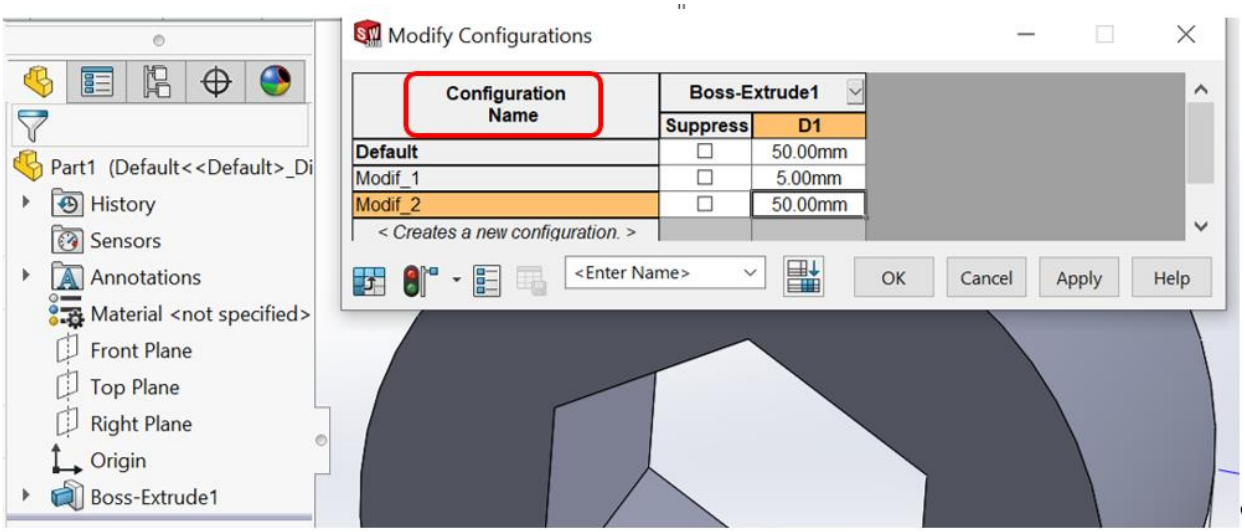


Figure 17 – Entering a numeric value for the new configuration

In the *Modify Configuration* dialog, the *Fillet* operation can only be shown for one configuration. To do this:

1. Create a feature *Fillet* (Fig. 18).
2. In the *FeatureManager*, right-click on this feature, select *Configure Feature* from the context menu.
3. In the *Modify Configuration* dialog box, check the configurations in which this feature will not be applied (Fig. 19).

Thus, using *Modify Configuration* you can *Suppress*, *Unsuppress* necessary features in the selected part configuration.

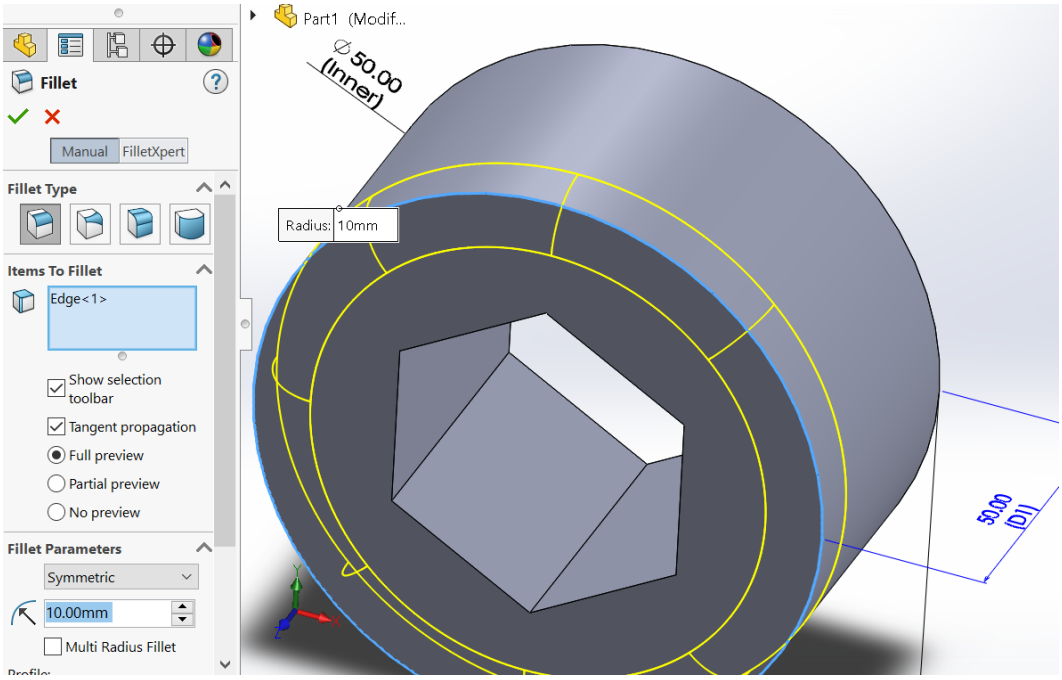


Figure 18 – Creating a rounding for a new configuration

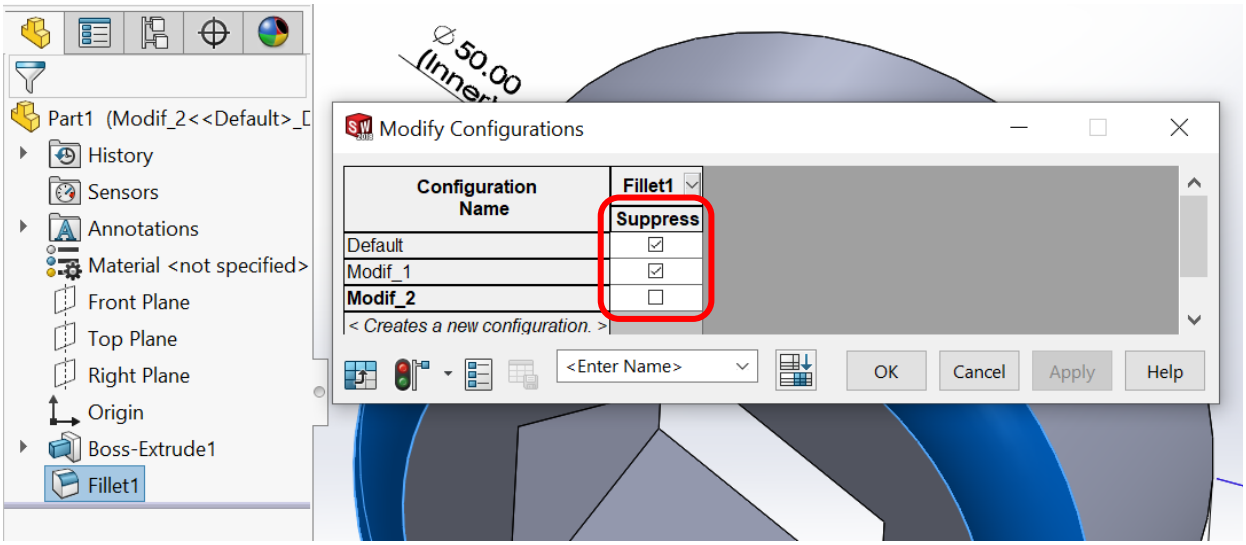


Figure 19 – Changing the status of the feature *Suppress/Unsuppress*

There is another way to apply features to configurations directly in the *FeatureManager*, namely by right-clicking on the feature and selecting *Suppress/Unsuppress* (Fig. 20).

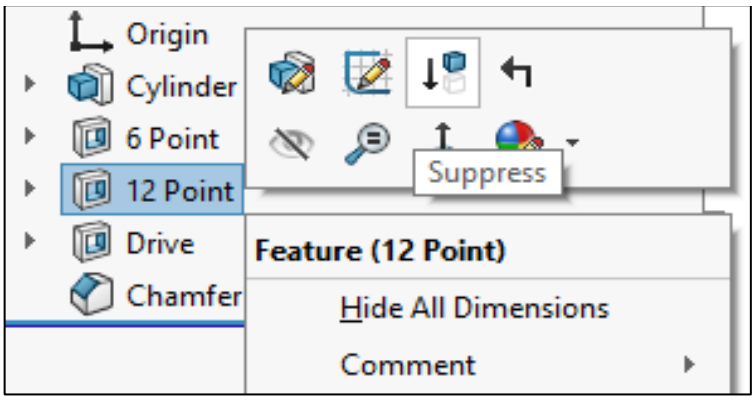


Figure 20 – Changing the status of feature in the *FeatureManager*

3.4 The third way to create a configuration

Configurations can be not only created, but also copied using the **Ctrl + C** and **Ctrl + V** keys.

You can create another configuration by selecting the name of the configuration and clicking *Edit*, *Copy* and *Edit*, *Paste* in the main menu of the program (Fig. 21)

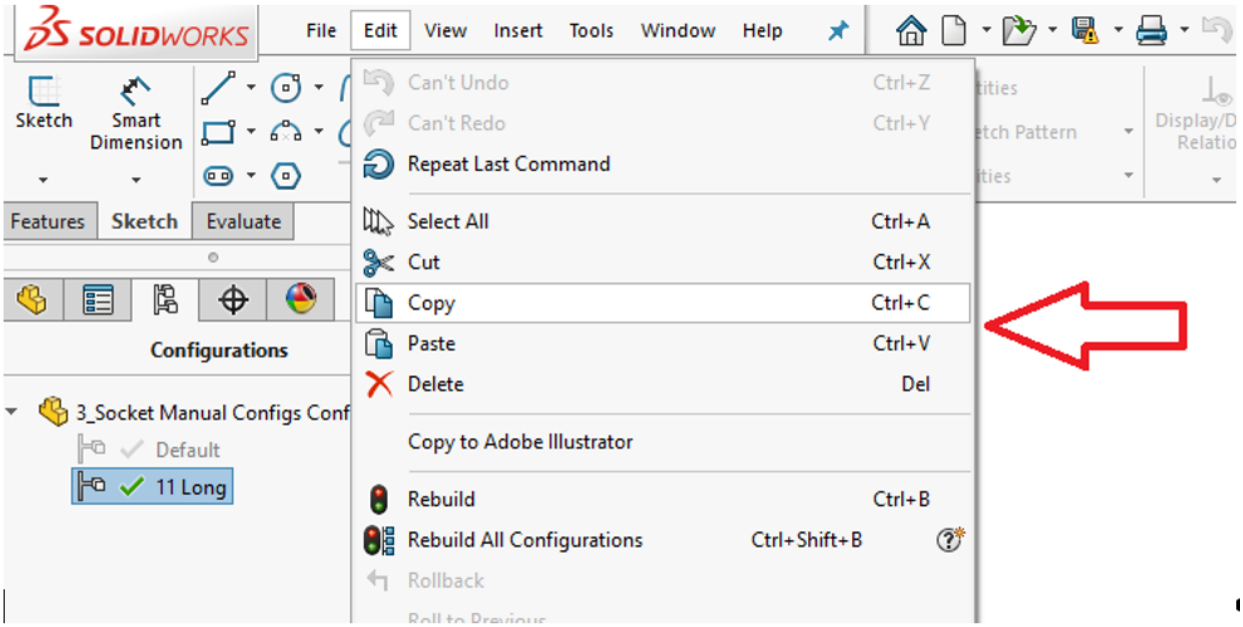


Figure 21 – Creating a configuration (the third method)

The newly inserted configuration appears in the *ConfigurationManager*, this part configuration can then be edited.

4 CREATION OF PART CONFIGURATIONS USING THE PARAMETERS TABLE

The parameter table allows you to create multiple part configurations simultaneously by specifying parameters in a *Microsoft Excel* spreadsheet and using functions, formulas, etc. when configuring parts and assemblies. (*Excel* 2010 or later must be installed on a PC to use *Design Table* in SOLIDWORKS*).*

The parameter table is stored in the model document, and can be created both with SOLIDWORKS tools and manually as a separate *Microsoft Excel* file.

There are two possible approaches to designing the initial model:

1 method. If the detail is simple, all the configuration features are created and their parameters are entered into the table at the same time, with the possibility of further changes.

2 method. For complex parts with a large number of controlled parameters, it is more convenient to first create one configuration and edit the values, then add structural elements of the second configuration, change the parameter table again, etc.

3 method. Inserting an external *Microsoft Excel* file as a parameter table.

Parts. You can control the following items in the part parameters table:

– dimensions and state of display of features, dimensions of holes for fastening;

– configuration parameters, including designations in the specification, derived configurations, equations, sketch relationships, notes.

Assembly. You can control the following options in the build options table:

– details: display state, associated configuration, or arbitrary or fixed location;

– parts and components of the assembly: dimensions, display status, dimensions of mounting holes;

– connection: dimensions for connection *Distance* and *Angel*, display state;

– configuration features: designation and display in the specification (when used as an assembly), configuration derivatives, equations, sketch relationships, notes, display states.

When using and creating a *Design Table* in the SOLIDWORKS program, it is necessary to correctly format them.

4.1 Methodology for creating a table of parameters

To create a *Design Table*, select the tab in the main menu, then *Tables*, then *Design Table* (note that the *Excel* logo is superimposed on the icon). In the *PropertyManager* tab *Source* appears (Fig. 22):

– *Blank*. Insertion of an empty table of parameters into which parameters must be entered: it is necessary to manually assign names to the created configurations, specify names of variable sizes and enter in the table of values of these dimensions.

Therefore, the *Design Table*, which is created automatically, is more often used.

– *Auto-Create* automatically creates a new parameter table and loads all configured parameters and their associated values from the part (Fig. 22).

– *From File.* Here you can specify whether to use an existing or automatically generated *Excel* file.

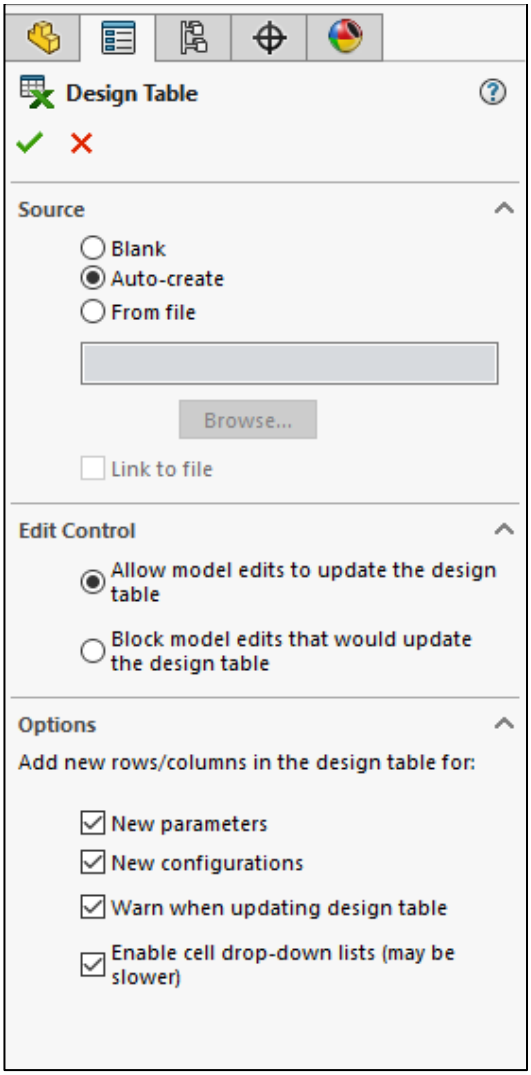


Figure 22 – Creating a *Design Table*

When automatically creating an *Avto-Create Excel* table in the *ConfigurationManage*, a built-in table that is *Design Table* (Fig. 23) will appear above the list of configurations. In this table, cell A2 is empty and marked *Family*. The next line already contains the name of the designed part of the first *Default* configuration and the value of its dimensions, and may also contain all configurations that were previously created manually (in this example in Fig. 23, these are configurations *Modif* _1 and *Modif* _2)

WARNING! Configuration names must not contain some special characters ("/", "@"). These configuration names are displayed in the *ConfigurationManager* window in alphabetical order, not in the order in which they appear in the parameter table.

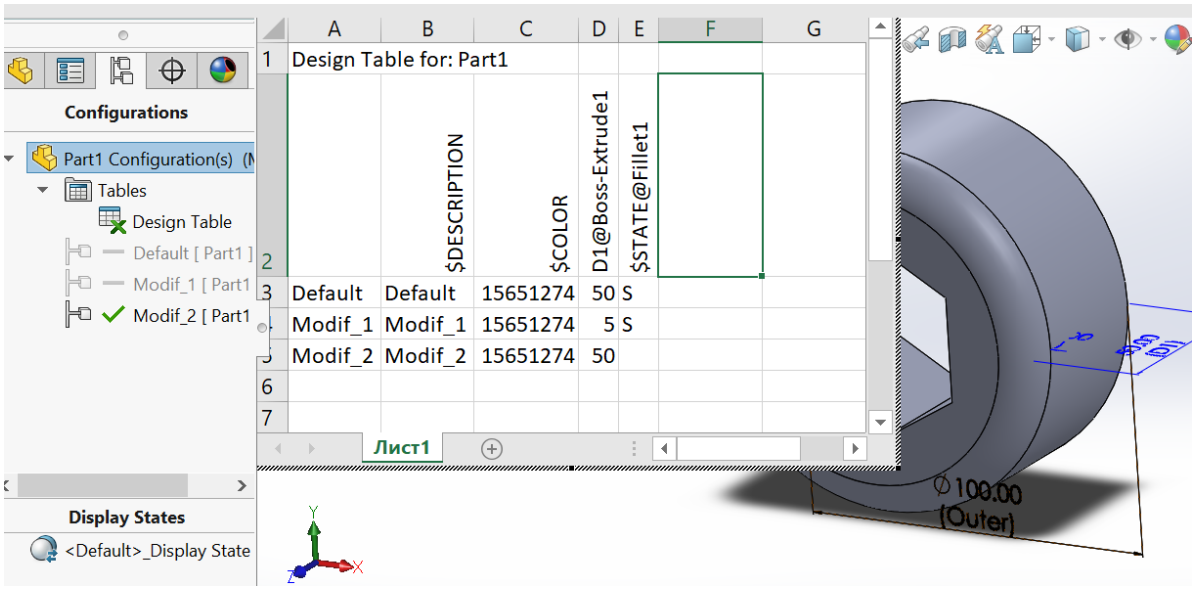


Figure 23 – *Design Table* window

To the right of *Family*, in the same line, there are the names of the parameters that control the model:

– managing thumbnail sizes;

– sizes of features of the model;

– in assemblies – dimensions characterizing the relative arrangement of parts.

You can edit the table: create new configurations, enter their names in the cells below the existing configurations, and edit the size values by simply changing the values contained in the cells (Fig. 24).

To create a new configuration, just copy the line with the standard configuration, change the name of the configuration and enter new parameter values.

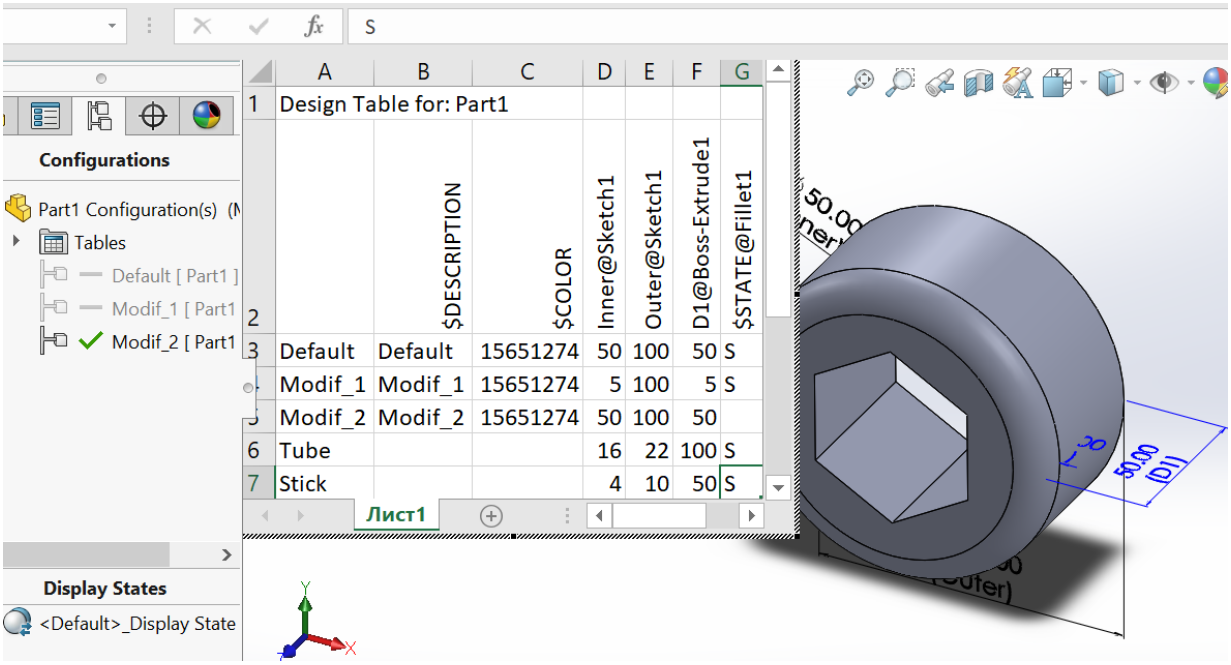


Figure 24 – *Design Table* editing

WARNING! In the parameters table, just like in the *Modify Configuration* dialog box table, you can both change the dimensions of the part and control the display (*suppress*) of its features (Fig. 24, column $*State@Fillet*1).

The *$* symbol represents the state of the *Suppress* feature, *U – Unsuppressed*.

Closing the table after filling is done by clicking outside the borders of the table, but in the working area. The program will exit *Microsoft Excel* mode, and the table will disappear from the screen. On the screen the display will open a window with a message about any new configurations (Fig. 25), one of which is currently active, that have been created using the *Design Table*.

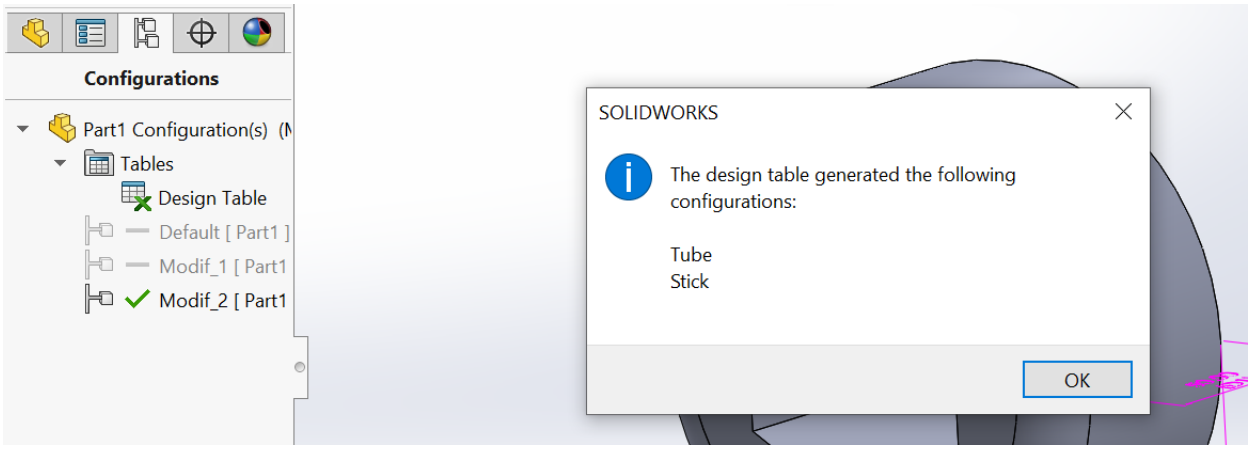


Figure 25 – Confirmation of creation of new configurations

The created configurations, whose parameters are recorded in table, will be added to the *FeatureManager*. In order to view the created configurations, it is necessary to activate them in the *ConfigurationManage* (Fig. 26) by double-clicking on the name of the configurations (icons).

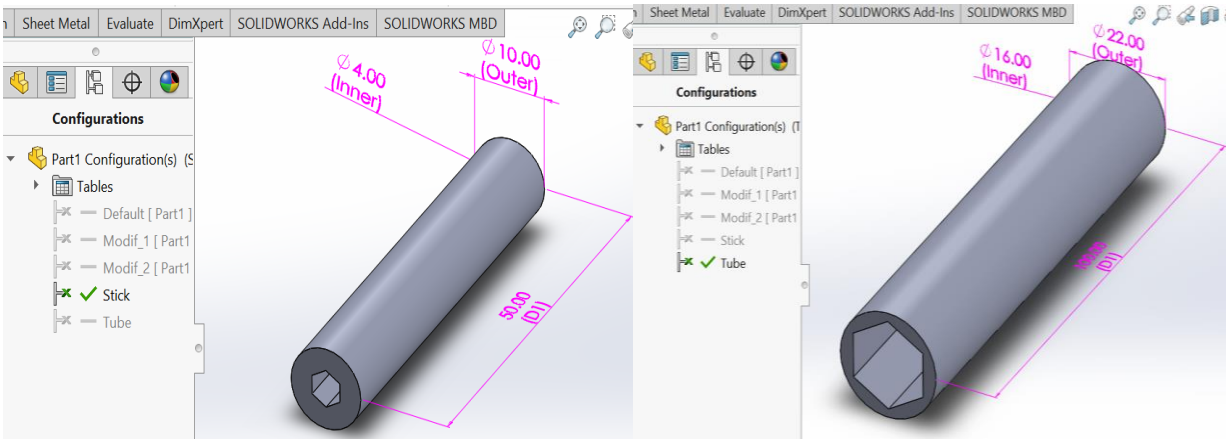


Figure 26 – Display of created configurations

Data in *the Design Table* can be edited using one of two methods:

– go to the *Edit* and select *Design Table* and then *Edit Table*. A table will be displayed on the screen, in which you can change certain configuration features;

– right-click the *Design Table* line in the *ConfigurationManager* and select *Edit Table*. In addition, you can remove any configuration by removing its name from the table parameters, or add a new configuration by entering its name and parameters (dimensions) in table of parameters.

4.2 Adding *Excel* spreadsheets

1. To add an *Excel* spreadsheet as a *Design Table*, select the *Insert* tab in the main menu, then *Tables*, then *Design Table*.

Next, select the *From File*, then click the *Browse* button to find the *Excel* file (Fig. 27). To link the parameter table with the model, select the *Link to file*. The associated parameter table calculates data from an external *Excel* file.

2. Set other settings. Click *OK*.

Excel toolbars will appear (Fig. 28).

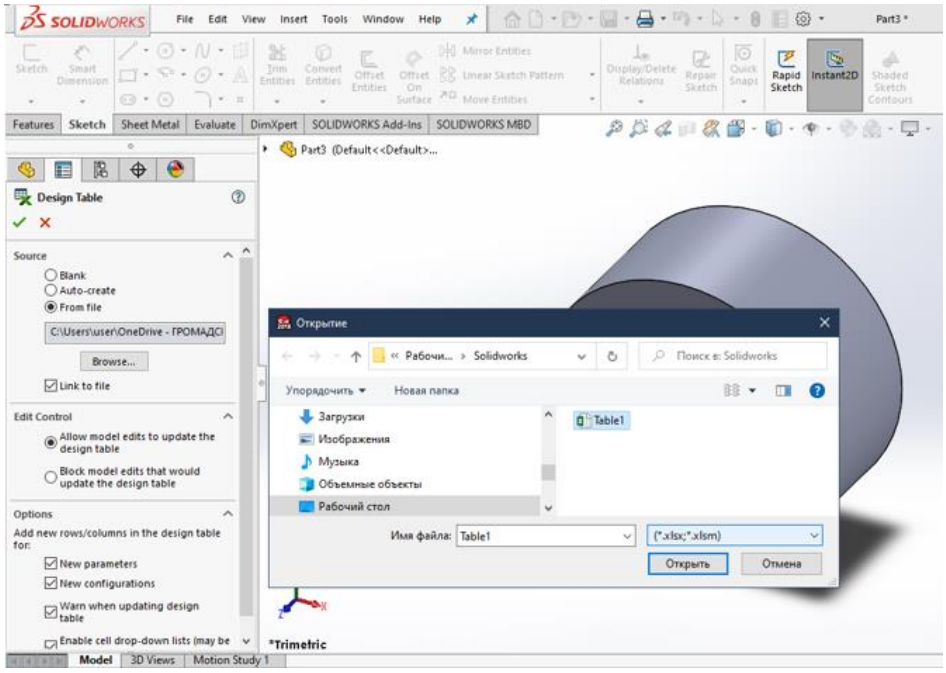


Figure 27 – Selecting an external file *Excel*

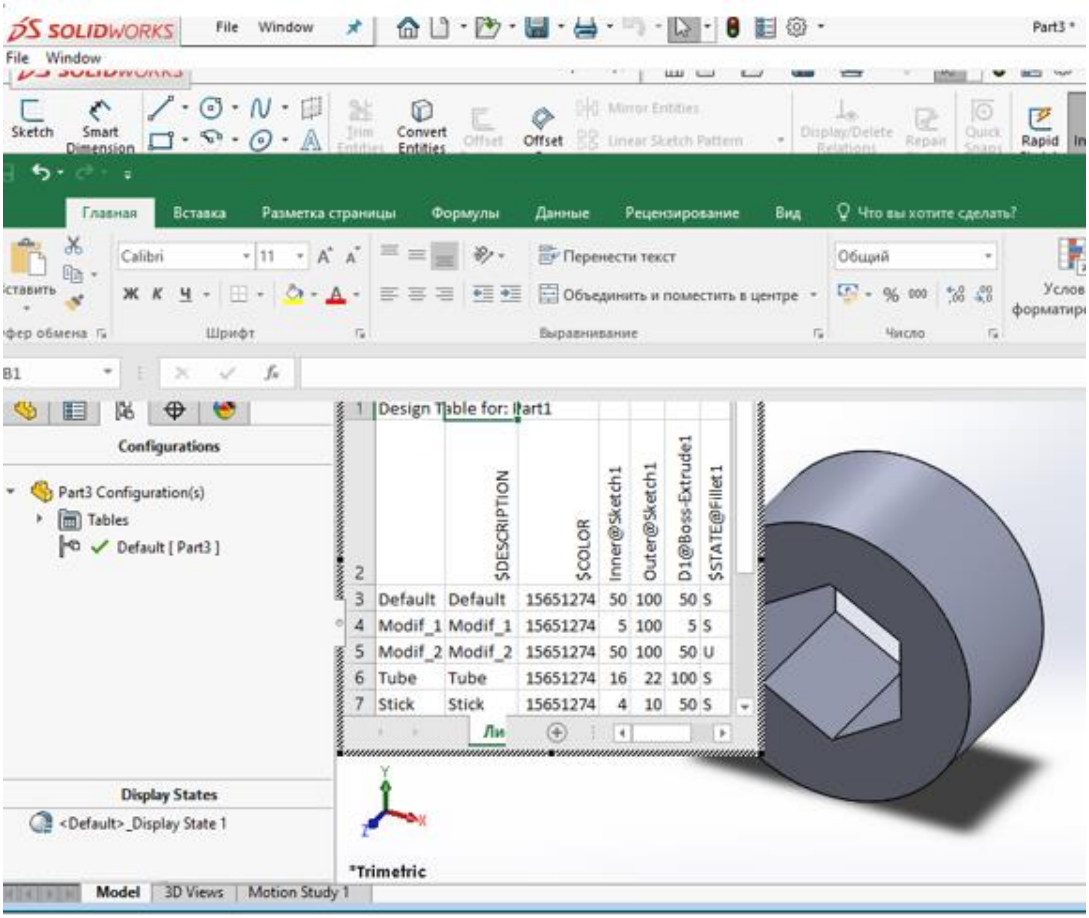


Figure 28 – Connection of the *Excel* parameter table with the model

After creation, it is possible to edit the table. To close the parameters of the table, click anywhere outside the table (not within the graphic area).

5 CREATION OF PART CONFIGURATIONS USING THE CALCULATION OF PARAMETERS TABLE

In SOLIDWORKS, the main parameterization tool is the *Equations* block in the *FeatureManager*.

Parameterization is the ability to use the part model an infinite number of times, each time adapting the dimensions and parameters of the arrays according to the design solution.

Parameterization is based on working with two main blocks: *Global Variables* and *Equations*. In the *Global Variables* section list of global variables is specified. In the *Equations* section, dependencies between dimensions and global variables are established.

Using global variables in parameterization is a method for creating several different numerical values for dimensions. Equations are used to create mathematical relationships between dimensions. For example, the sizes of several rounds can have different values. A global variable will be created and applied to all of them. This variable can be applied to any size, using it to calculate each size. When you create a global variable, it appears in the *Equations* tab of the *FeatureManager*.

SOLIDWORKS uses equations of the form ***Dependent = Independent***. This means that in the equation **A=B,** the system calculates the value of **A** if the value of **B is given**. You can edit the value of **B.** Once the equation is written, the value of **A** cannot be changed. Such mathematical functions, both trigonometric and logical, can be included in the equation (table 1).

Table 1 - Mathematical functions

sin( )	cotan( )	arccotan( )	int( )
cos( )	arcsin( )	abs( )	sgn()
tan( )	arccos( )	exp( )	if ( ) (for comparison you can use =, <= or =>)
sec( )	atin( )	log( )	
cosec( )	arcsec( )	sqr( )	

Thus, having defined all the necessary equations, it is possible to change the design solution by changing the values of the geometric dimensions using the parameterization of the part. The dimensions of the part are automatically updated when the equation table is closed.

Using the parameter calculation table helps to perform operations on the model several times faster, as it is enough to change one parameter value (length, width, height, etc.) and the dimensions associated with it will change automatically.

5.1 Creating a table of parameter equations

When creating a parameter calculation table on the example of the configuration of the «Socket» part, a global variable must be added.

1. To do this, select the ∑ *Equations* item on the *Tools* tab main menu (Fig. 29).

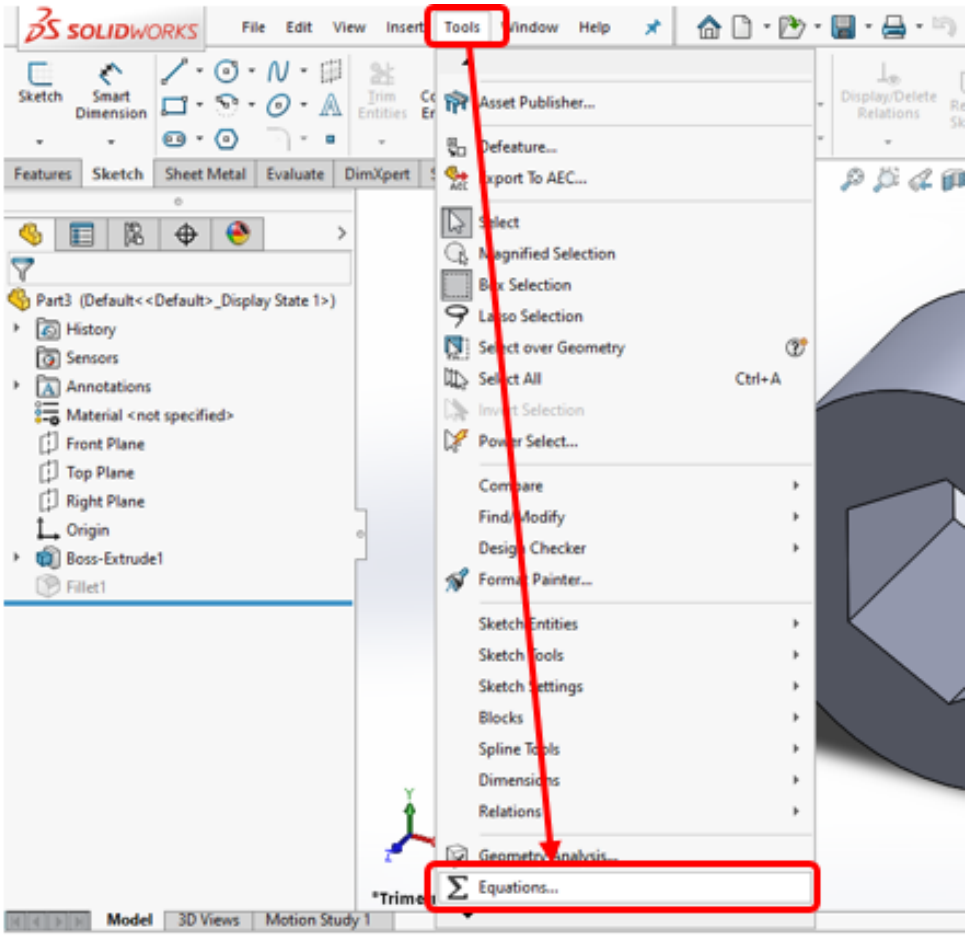


Figure 29 – Adding the *Equations*

2. Add a variable (eg '*a*') to the table in the *Global Variables* section and its value (Fig. 30). After you finish entering the global variable, click the button ✔.

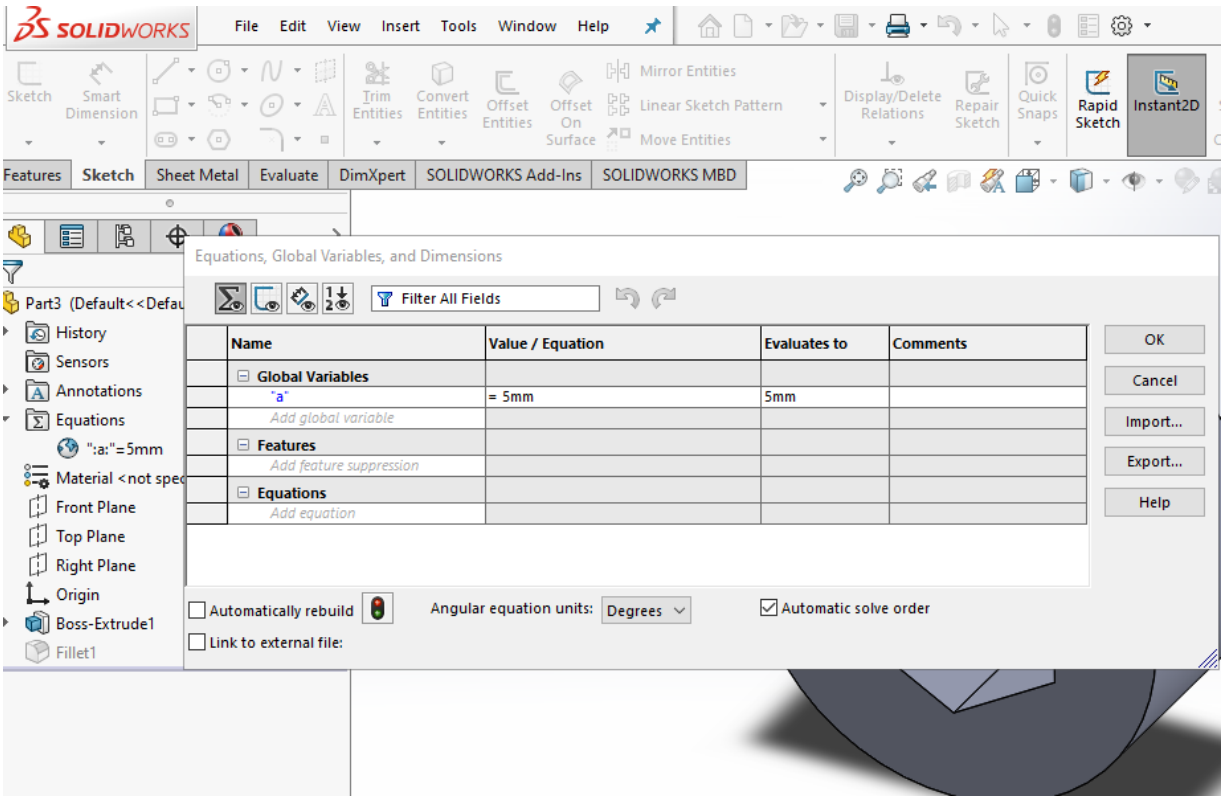


Figure 30 – Adding a global variable to the *Equations* table

When creating a global variable, it appears in the *Equations* tab in the *FeatureManager* (Fig. 31).

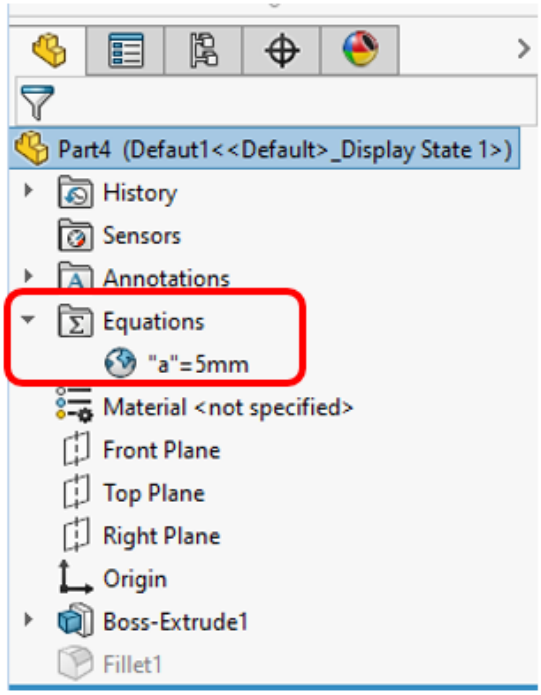


Figure 31 – Global variable in *FeatureManager*

3. Apply this variable to a dimension, that is, establish a relationship between the existing dimensions and the variables in the *Equations* section, namely:

– right-click on the *Equations* section in the *FeatureManager* and choose *Manage Equations* (Fig. 32);

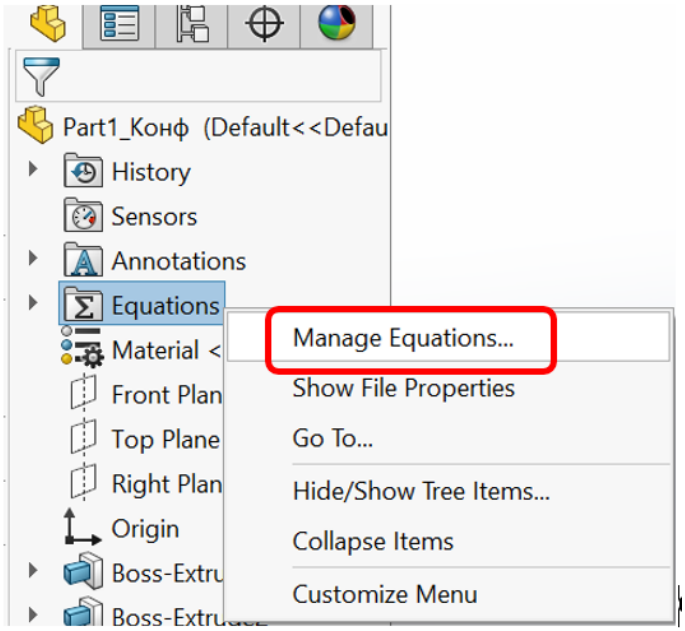


Figure 32 – Access to editing the parameter equation table

– place the cursor in the *Equations* section parameter equation tables;

– click on the required size of the model in the graphic area (pre-check the display of *Show Feature Dimensions*), after which the name of the size will appear in the corresponding section of the parameter calculation table (Fig. 33);

– in the second column of the table after the sign "=", indicate the global variable and the mathematical operation with it (Fig. 34) to get the final value of the size that depends on the global variable.

In this example, the dimension named *Outer@Sketch*1 will be defined by an expression '*a*'*20 mm.

The size determination appears in the column *Evaluates to*, and the cursor moves to the next column cell.

– Click *OK* to close the dialog box.

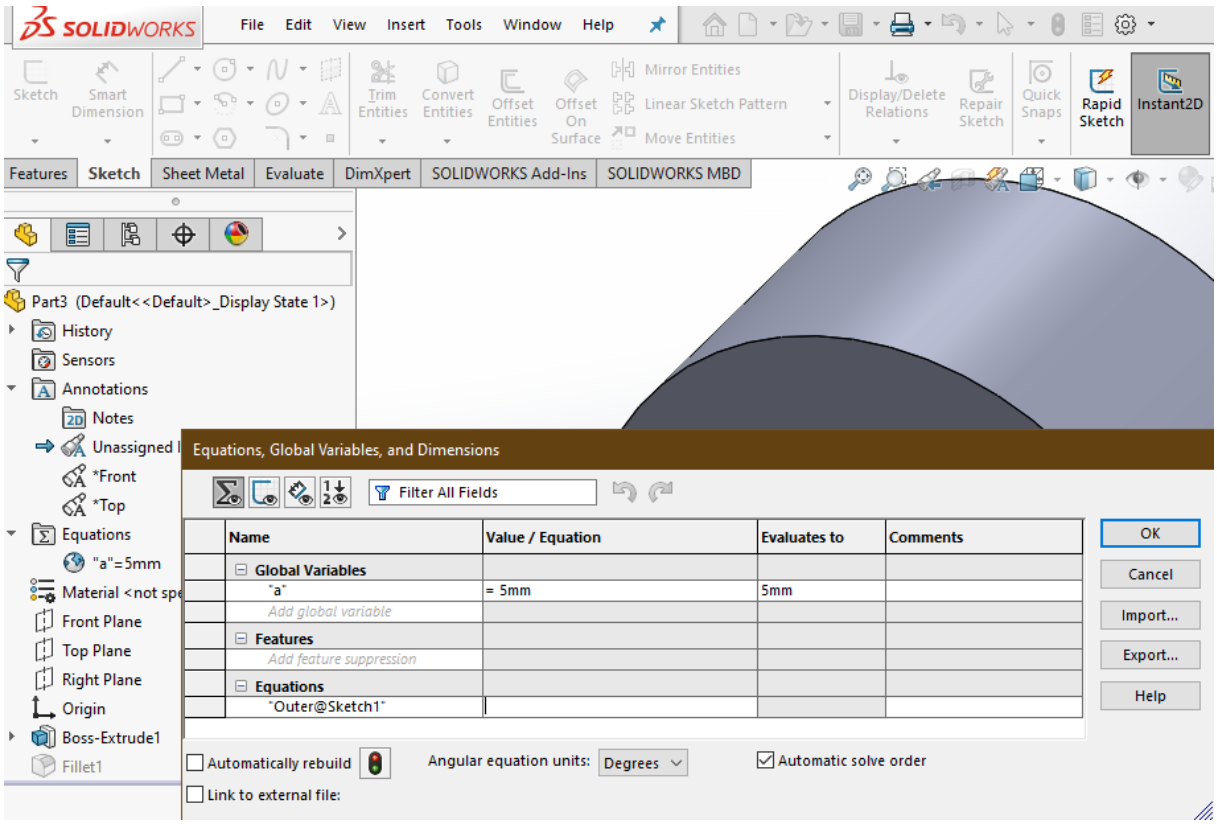


Figure 33 – Adding a dimension name

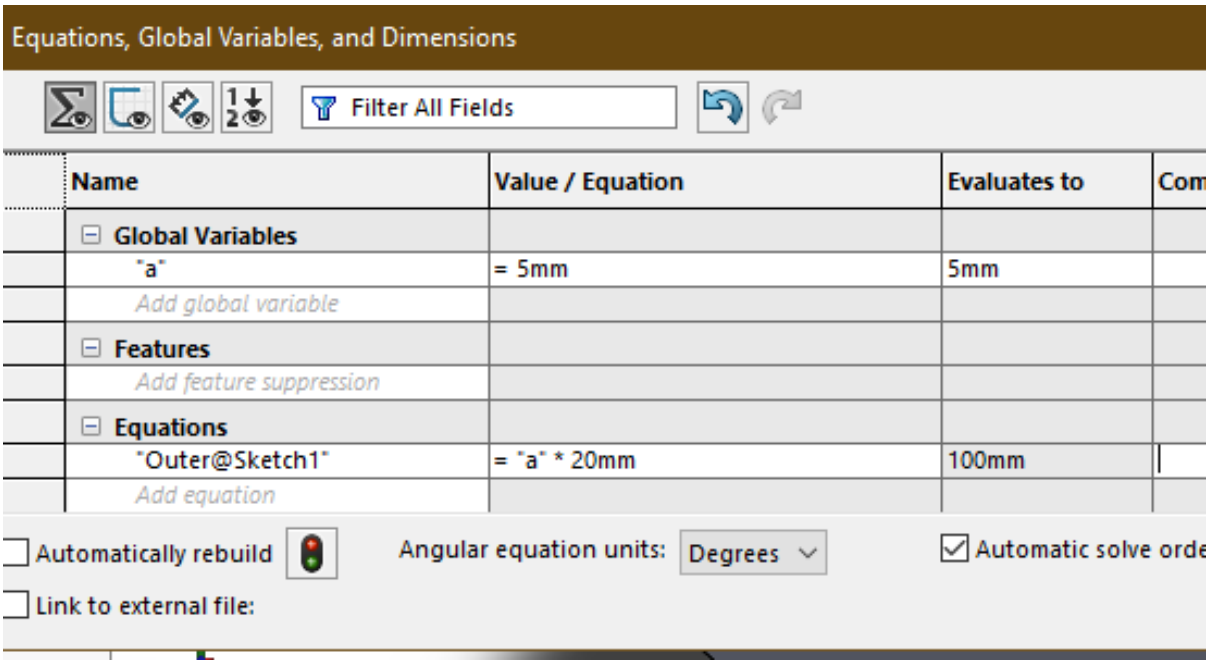


Figure 34 – Setting the size via a global variable

4. This variable can then be applied to any size to avoid entering it every time.

5. Similarly, you need to create 2 more global variables ('*b*' and '*c*') and apply them to other dimensions of the part (Fig. 35).

WARNING! It is necessary to carefully determine the dimensions that will be parameterized in order to avoid errors and obtain the required shape of the model.

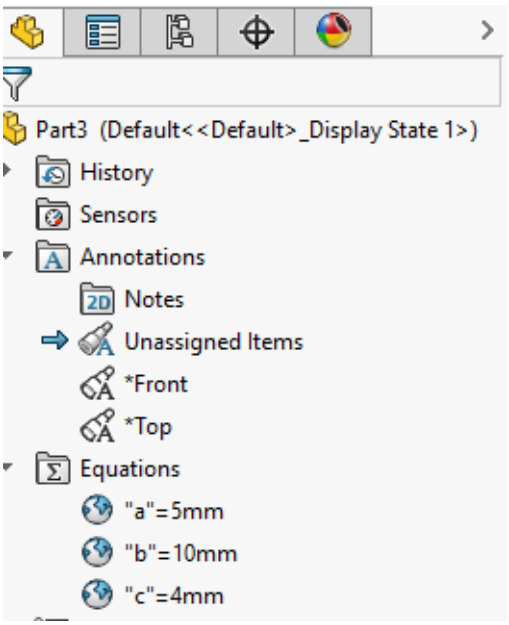


Figure 35 – Global variables in *FeatureManager*

As a result, the dimensions of the basic part «Socket» are specified using three global variables, which are indicated by the sign ∑ in the names of the dimensions (Fig. 36).

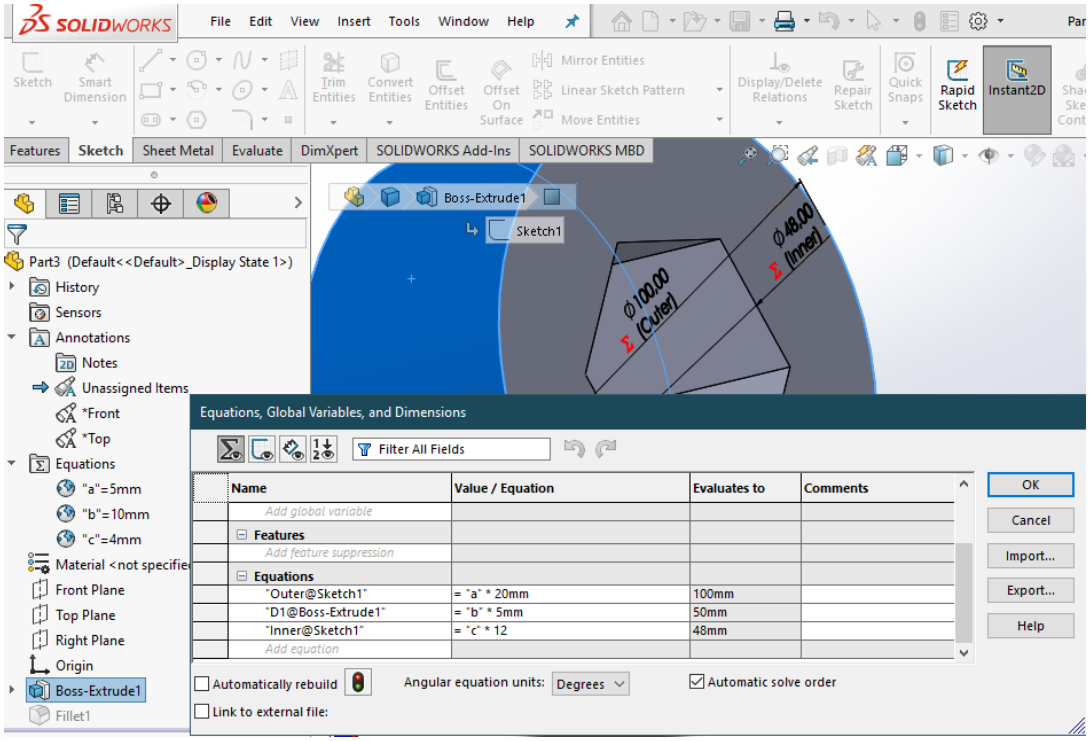


Figure 36 – Dimensions of the base part «Socket» using global variables

5.2 Creation of part configurations using a table of parameter equations

To create part configurations, it is necessary to repeat the sequence of actions when adding a *Design Table*, described in section 4. Editing the dimensions of each configuration is carried out according to the rules for editing *Excel* spreadsheets.

Set the required dimensions of the part by changing the coefficients in the expressions that contain global variables. Click *OK*. On the display screen a window will open with a message about the creation of all configurations (Fig. 37).

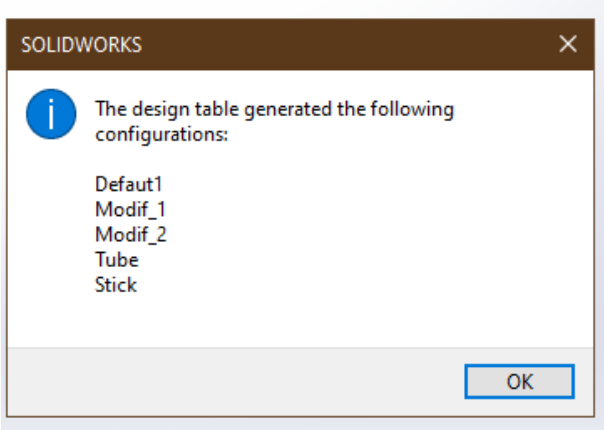


Figure 3 7 – Creation of configurations of the basic part «Socket»

The created *Equations* table with global variables is similar to the *Design Table* parameter table (Fig. 38).

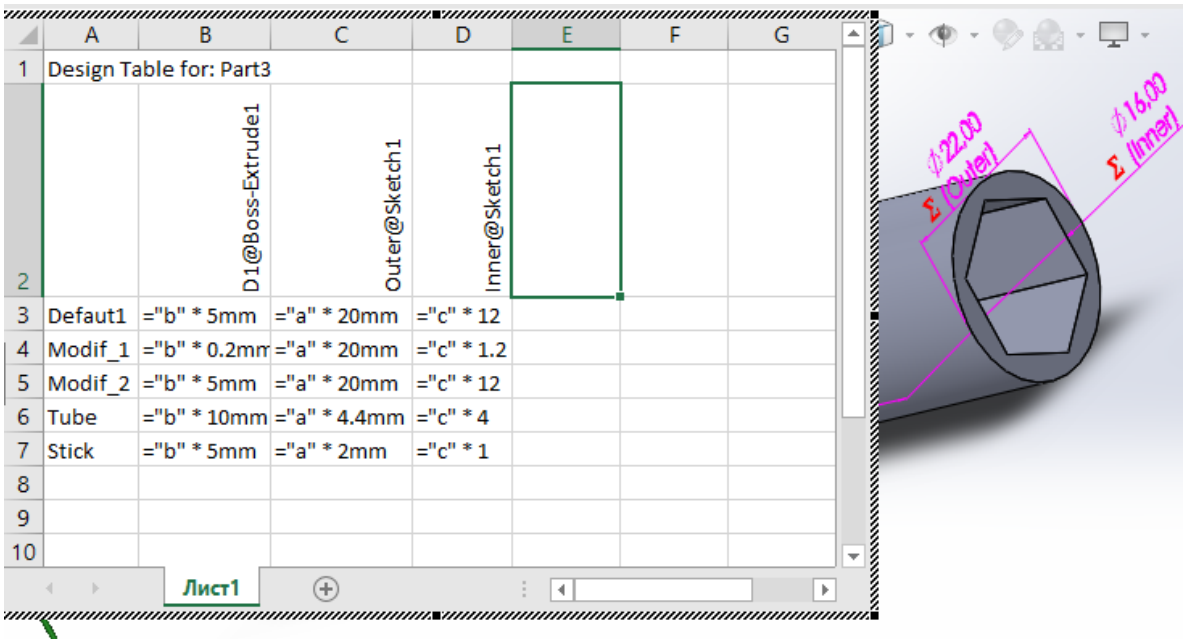
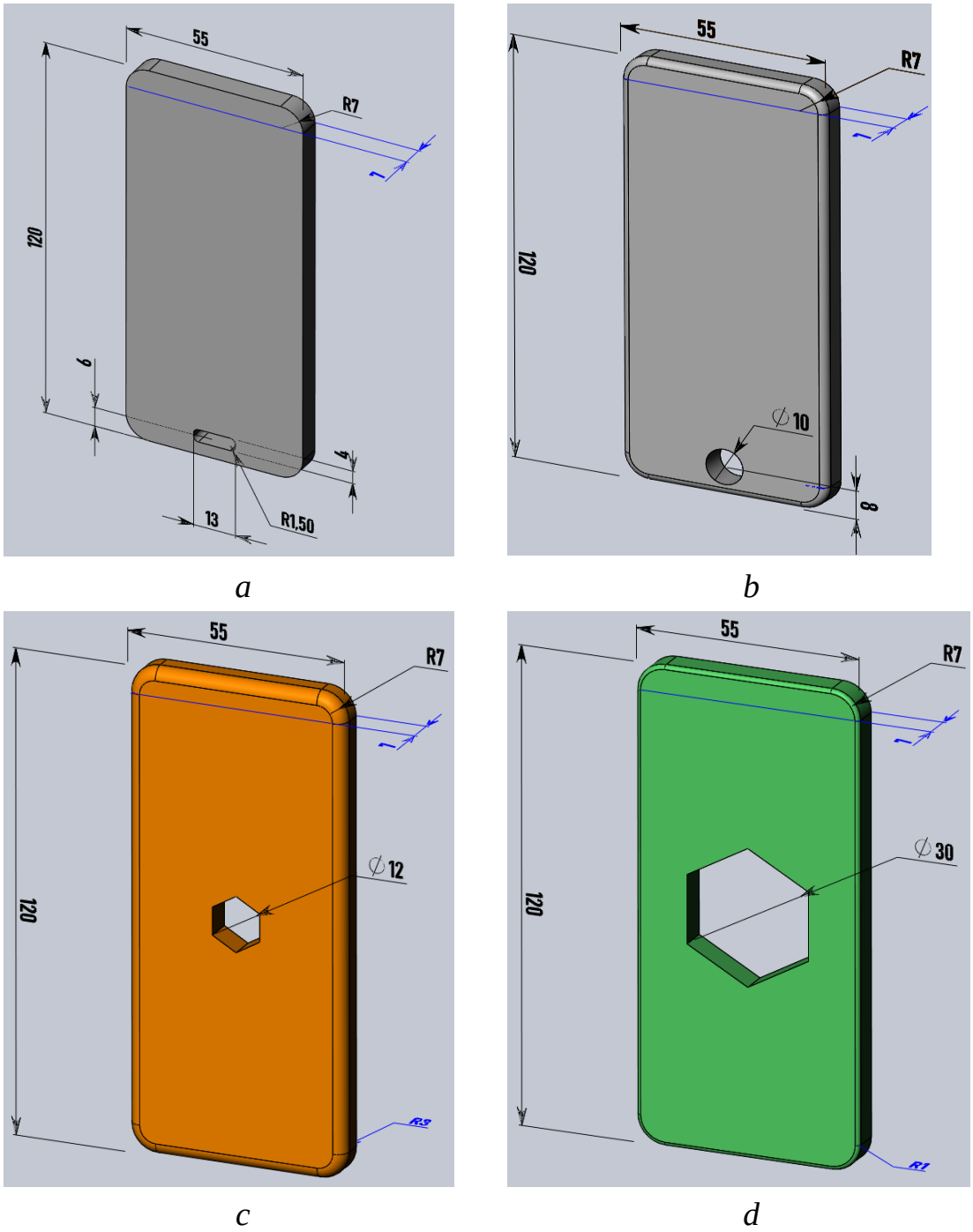


Figure 38 – *Design Table view* with global variables

6 TASKS FOR THE LABORATORY WORK

6.1 Task 1

Manually create 4 part configurations (Fig. 39 *a*-*d*). Specify the unspecified dimensions yourself. You can make a fifth configuration or change one of the configurations at your own will (creative approach). Optionally use color options for each configuration.



a *b*

c *d*

Figure 39 – Task 1

6.2 Task 2

Create 3 configurations of the part (Fig. 40) using the parameters table. The main parameters of the two configurations are known. Unspecified dimensions and parameters of the third configuration can be set independently. When creating configurations, you can use the *Pattern* or *Mirror* tools.

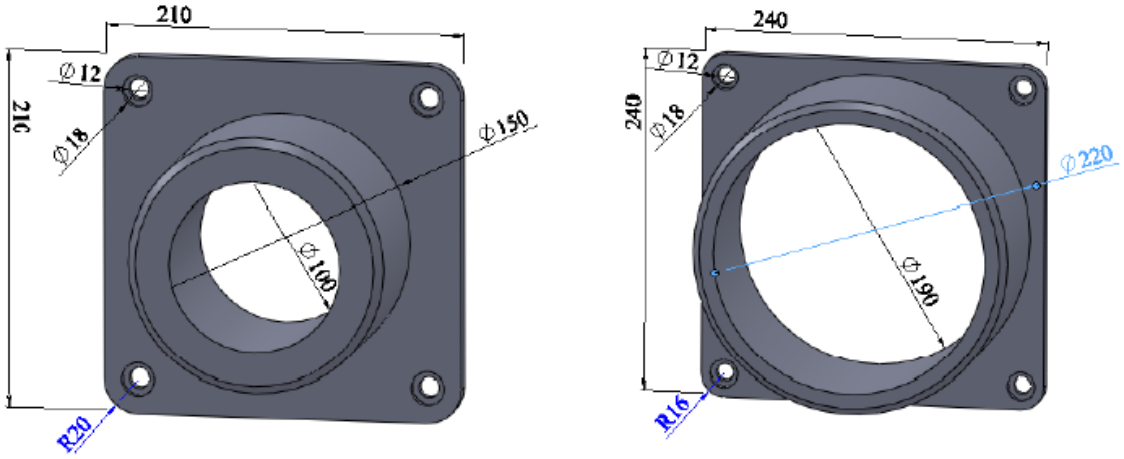


Figure 40 – Task 2

6.3 Task 3

Repeat the creation of 3 configurations of the part from task № 2 (Fig. 40), using the table of global variables. The number of global variables and the calculation of parameters using them – is chosen independently.

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CONTROL QUESTIONS

1. What is part configuration?
2. What are the methods of creating part configurations in SOLIDWORKS?
3. What are the ways to edit part configurations?
4. How can one create a part configuration using a parameter table?
5. When should you create configurations using parameter tables?
6. What are global variables used for in parameter tables?
7. How can one delete a part configuration?

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з дисципліни «Основи проєктування електронних пристроїв»
для студентів спеціальності 171 «Електроніка»
першого (бакалаврського) рівня вищої освіти всіх форм навчання

Укладачі: БУТОВА Ольга Анатоліївна
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