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**AUTOMATED DESIGN SYSTEMS.
WORKSHOP IN FREECAD**

*Workshop on discipline for students of specialties
163 "Biomedical Engineering" and
152 "Metrology and information-measuring technique"*

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Educational Tutorial

AUTOMATED DESIGN SYSTEMS. WORKSHOP IN FREECAD

Workshop on discipline for students of specialties 163 "Biomedical Engineering" and 152 "Metrology and information-measuring technique"

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AUTOMATED DESIGN SYSTEMS. WORKSHOP IN FREECAD

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У навчальному посібнику розглянуто особливості роботи та основні теоретичні основи проектування механічних вузлів медичних приладів та систем, а також основні компоненти інтерфейсу у середовищі моделювання FreeCAD. Теоретичні та практичні результати комп'ютерного практикуму у FreeCAD можуть бути використані при проектуванні та експлуатації медичних пристроїв. Практична частина спрямована на науково-аналітичний огляд, проектування та конструювання медичних приладів і систем, комп'ютерне моделювання медичної діагностичної та лікувальної техніки.

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INTRODUCTION

In the tutorial manual discusses the general features of practical work and the main theoretical foundations of the medical devices and systems mechanical components design, as well as the main components of the interface in the FreeCAD modeling environment. The theoretical and practical results of a computer workshop in FreeCAD can be used for the design and operation of medical devices. The practical part is aimed to scientific and analytical review, design and construction of medical devices, computer modeling of medical diagnostic and medical equipment.

Despite of the FreeCAD system was primarily created for parametric 3D modeling in the field of mechanical engineering, but this is not the only direction of its application. The FreeCAD workshop developed for supports the application of the system in the biomedical engineering field in cases where you want to model three-dimensional objects with a given accuracy and control.

To master the developed workshop, it is recommended to study the FreeCAD manual in advance. The design experience in FreeCAD involves the acquisition of parametric 3D modeling skills that allow you to create and process objects.

Applying 3D models to components of medical devices and systems provides an advantage that is easy to change your projects during modeling. The use of open-source principles in the FreeCAD environment allows free use of existing 3D models. In this tutorial, which consists of computer workshops on FreeCAD, you will learn the basic commands and tools for visualizing your ideas and their subsequent implementation.

SECTION 1. SOFTWARE FOR PARAMETRIC MODELING

1.1. Parametric modeling

FreeCAD software is designed for parametric 3D modeling, like other CAD, designed for the creation and processing of solid objects. Its advantage is the ease of modifying your projects during modeling, as well as the principles of open source and, of course, the possibility of free use.

One of the most powerful features of FreeCAD is the scripting environment. Almost all parts of FreeCAD can be accessed from the built-in Python console (or from any other external Python script): creating and modifying geometry, changing the display of objects on the screen, and accessing and modifying the interface. Scripts can be used in macros, which let us easy to create your own commands.

In this FreeCAD workshop, you will learn the basic commands and tools for visualizing your ideas and further implementing them using 3D printing "for some tasks and applications that are not relevant at the moment" [1]. We will create simple objects step by step. Finally, you will be able to create the first 3D CAD model by yourself.

In the FreeCAD workshop, you will get the basics knowledge of the working with this system. To do this, you need to download and install FreeCAD first. You can download it from the FreeCAD website. The platforms available are Windows (32-bit, 64-bit), Mac (64-bit), and Linux (64-bit AppImage). Installing and configuring FreeCAD on the Windows platform is shown in Figure 1.

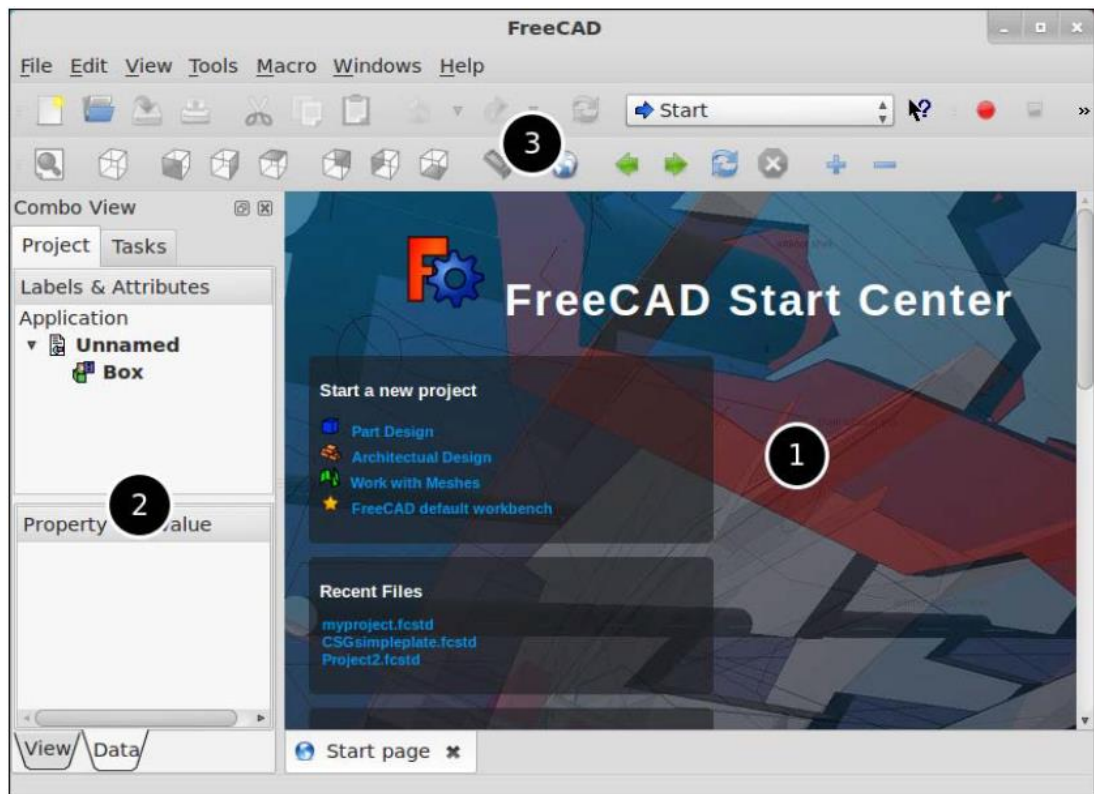


Figure 1 – FreeCAD loading window

The basic concept of the FreeCAD interface is to divide it into workbenches. Workbenches are a set of tools tailored for a particular task, such as working with grids, or drawing 2D objects, or sketches with guides. The current workbench can be switched using the workbench selector. It is possible to customize the tools of each Workbench, add them from another workbench. **PartDesign** and **Part** workbenches are widely used as starting points.

The FreeCAD system has two types of extensions:

1. Macros are small chunks of Python code that provide additional tools or functionality. Macros are intended to simplify or automate the task of drawing or editing individual objects. If many of these macros are gathered in a directory, the entire directory could be like a new workbench.

2. Third Party Workbenches - A collection of tools, programmed in Python or C ++, that significantly extend FreeCAD. If a workbench is

successfully designed and well documented, it can be turned on by one of FreeCAD's workbenches.

Each interface tool has the most used tools to execute a specific task:

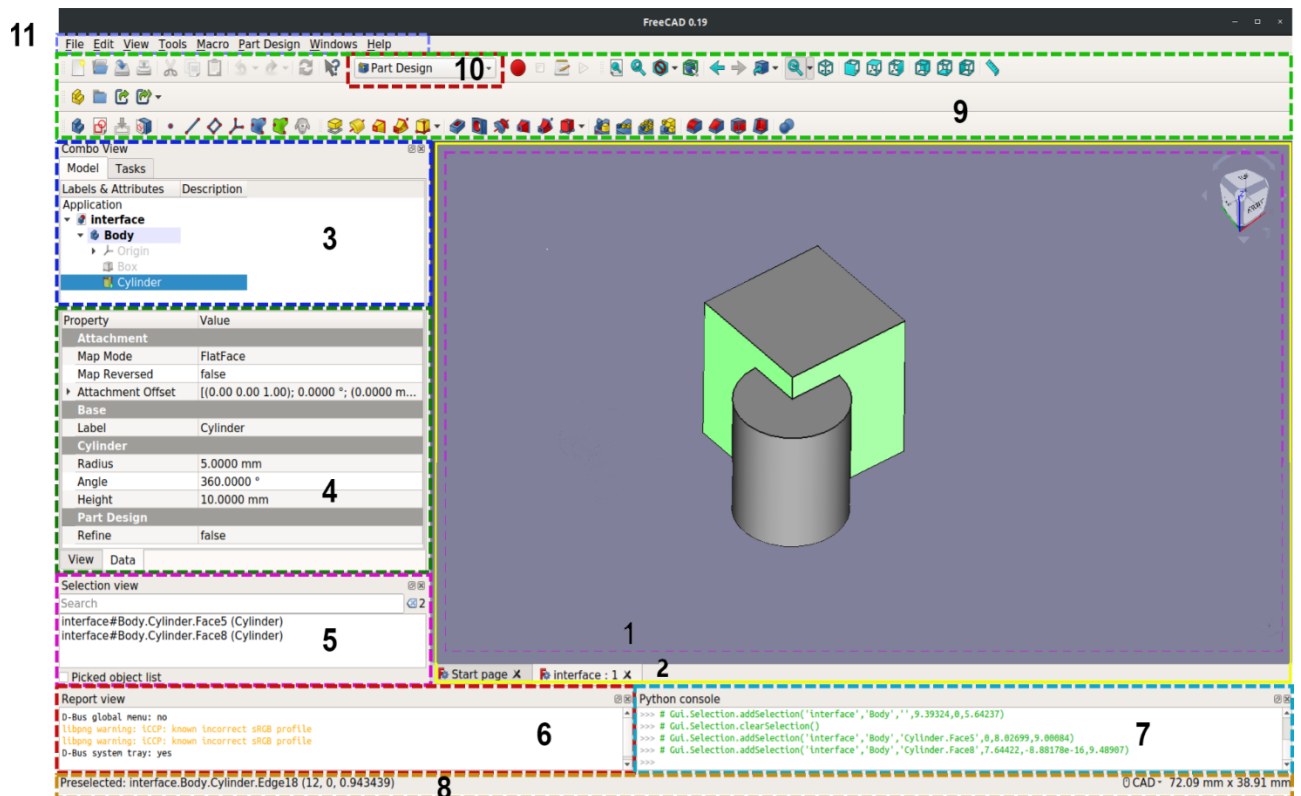


Figure 2 – Interface overview

The main elements of the interface are:

1. Main view - may contain various tabbed windows, in this case the 3D view tab is open.
2. Tabs of the main view - a list of all open objects and documents in the form of tabs.
3. Project tree (part of the combo panel) - showing the hierarchy and history of the design of objects in the document, can also display the taskbar of active commands.
4. Properties editor (part of the combo panel) - allows you to view and change the properties of the selected objects.

5. Selection view - displays objects or elements of selected objects (vertices, edges, faces).
6. Report (or Python output window) - this panel displays messages, warnings, and errors in the work in the FreeCAD system.
7. Python Console - a console for entering Python code and viewing the results of its execution.
8. Status bar - displays some messages and tips.
9. Toolbars section - the area where toolbars are located.
10. Workbench Switch - Allows you to switch the workbench.
11. Standard menu - contains basic operations.

1.2. Units

Before you start building your project, you need to decide which units you are going to use. We will work using the metric system.

When you open FreeCAD, you should see the start window (Figure 3):

In the Units tab you can specify the following:

User system: selection of a unit system that should be used for all parts of FreeCAD

Number of decimals: the number of decimals that should be shown for numbers and dimensions in FreeCAD

Minimum fractional inch: Minimum fractional inch that should be displayed. This setting is only available if the unit system Building US (ft-in/sqft/cuft) is used.

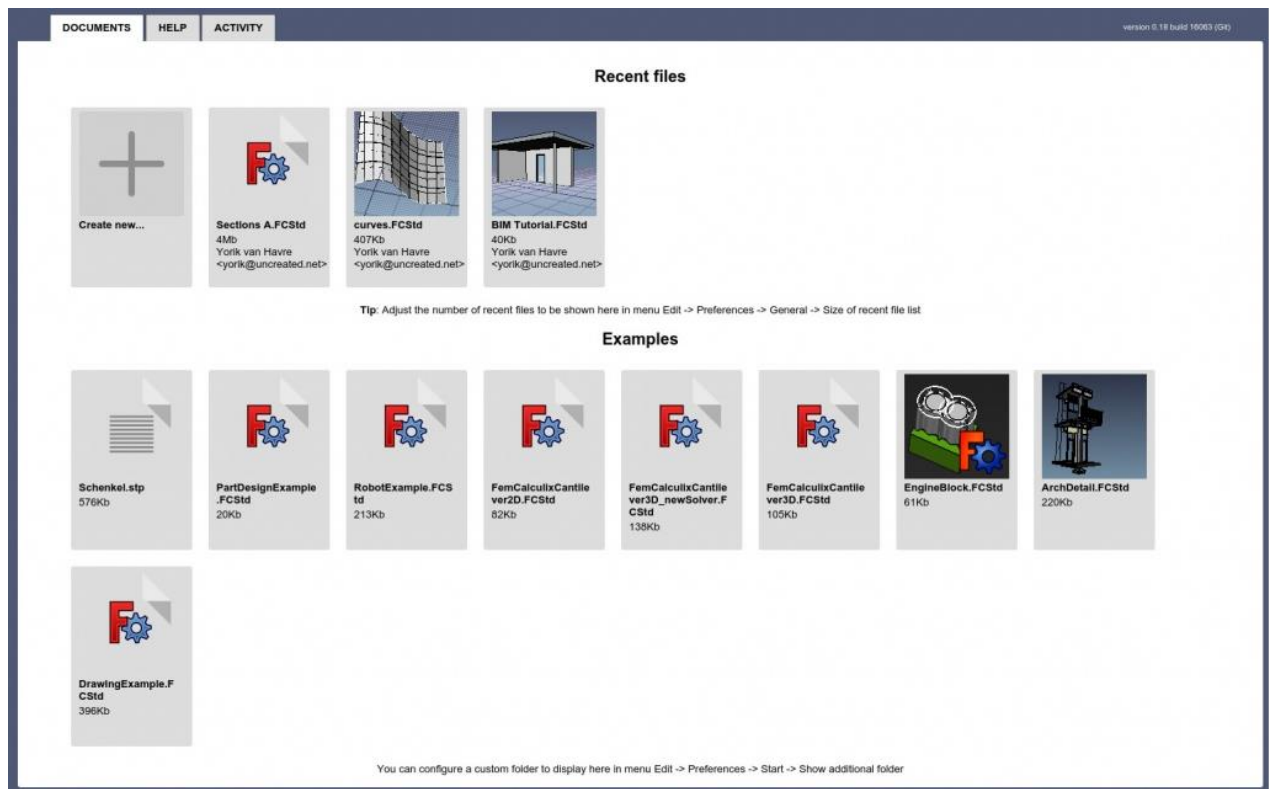


Figure 3 – Getting Started Window

FreeCAD allows you to work with the major OSE formats and tools that focus on the using of an open-source drawings for global replication.

SECTION 2. CREATION AND CONFIGURATION OF THE PROJECT

FreeCAD uses the concept of workbenches, where each workbench is responsible for its specific tasks and functions. The workbench can have its own settings section. Initially, there are four sections available in the Edit->Preferences (even if no workbench has been loaded). These are the sections: "General", "Display", "Import/Export" "Start", "Part Design", "Sketcher", "FEM" and "Draft". Workbenches that can be loaded in the process have their own settings tab and will be discussed further.

2.1. Creating a new project

Click “**Create new**” .

You can now set your preferred navigation style. For this we will use **Blender**. **Rightclick > Navigation styles > Blender**> (Right Click > Navigation Styles> Blender) (Figure 4).

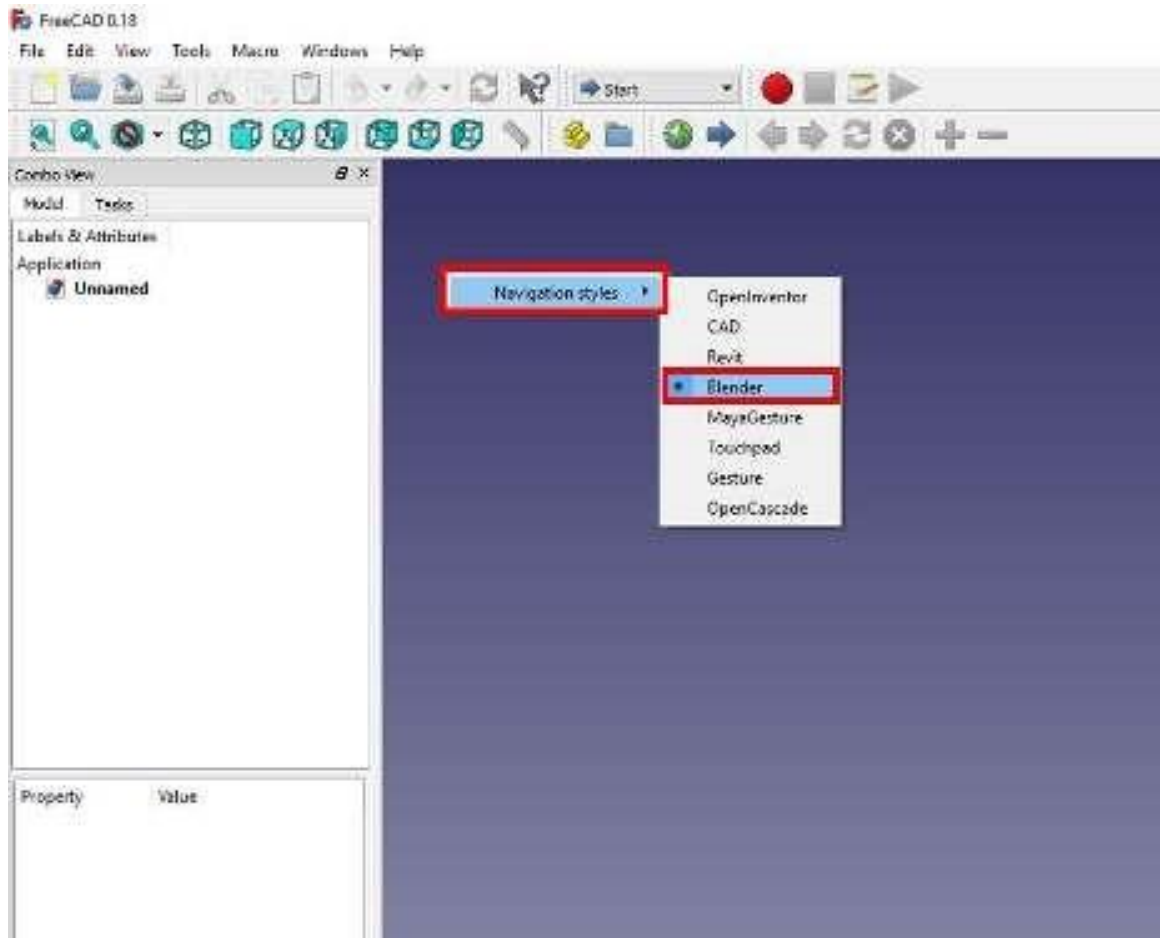


Figure 4 – Navigation styles window

Changing the units of measurement in the FreeCAD system:

1. Select and click the **Edit> Preferences** menu (Figure 5).

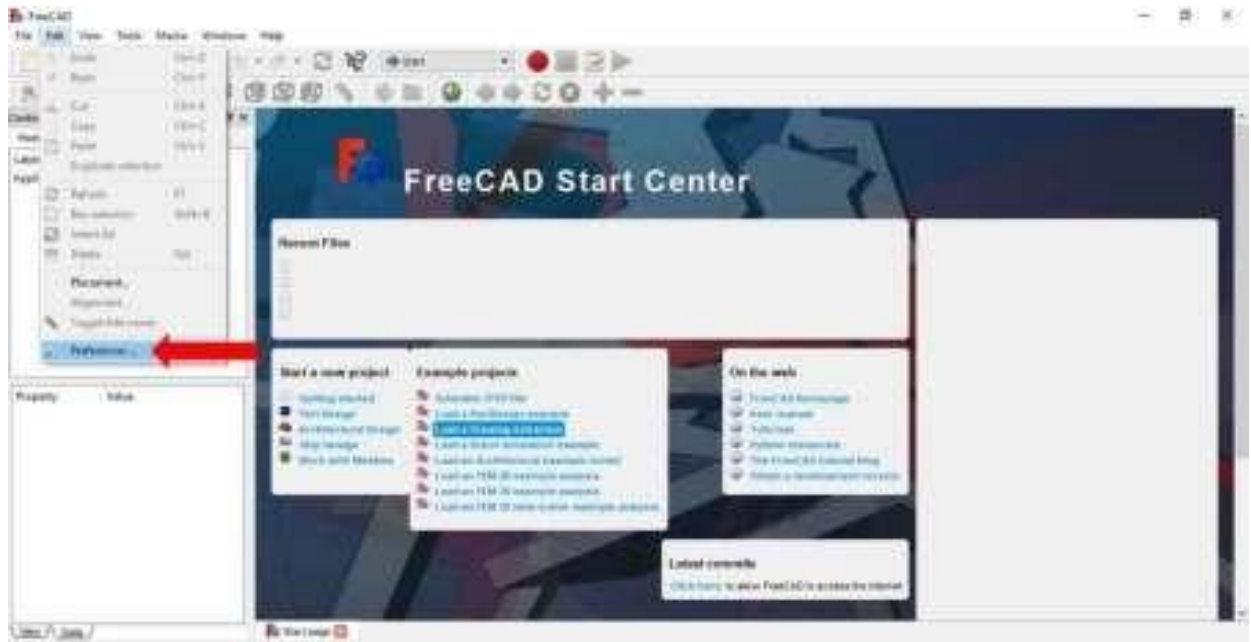


Figure 5 – Preferences window

2. Select **General** in the appeared window (Figure 6):

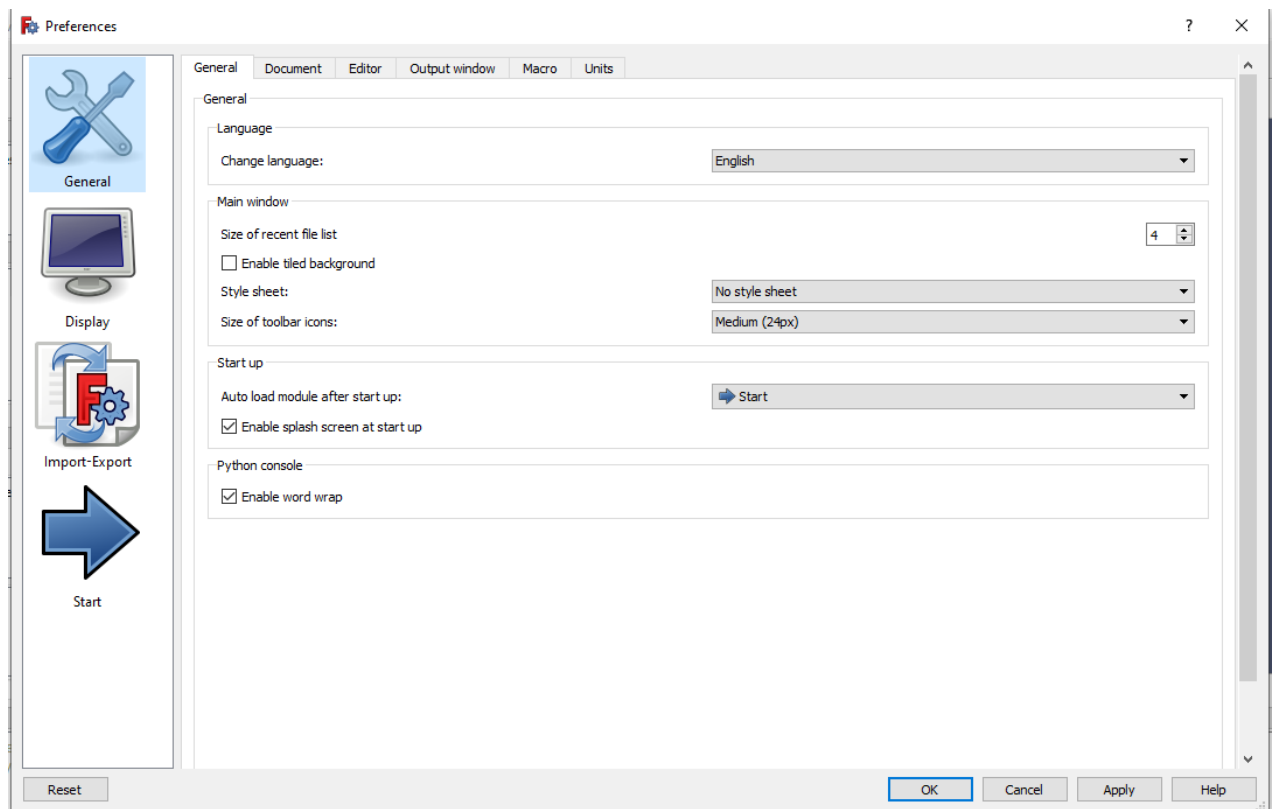


Figure 6 – Working with an option **General**

3. Go to the "**Units**" section, change the user system to "**Standart**" and

confirm by clicking "OK" (Figure 7).

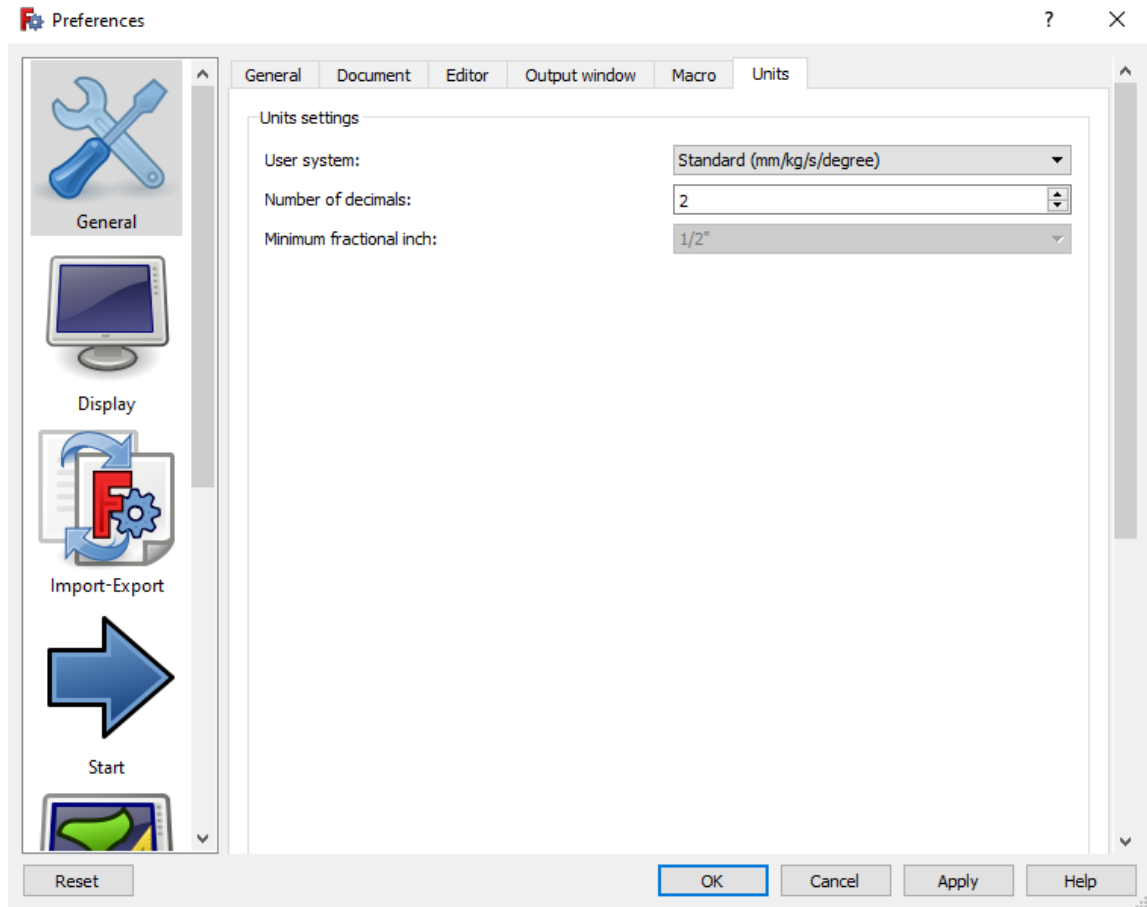


Figure 7 – Working with an option **Units**

2.2. View

Now start a new document by pressing **CTRL + N**. You will see a new, unnamed tab opening at the bottom left of the window (Figure 8).

If the **Combo View** on the left is already open, you can skip the next step.

Open the **Combo View** by clicking **View > Panels > Combo View** (Figure 9).

This window is very useful for quickly executing commands. It also contains a view **Tree** and **Property**, that allows you to become more familiar with the structure of your project.

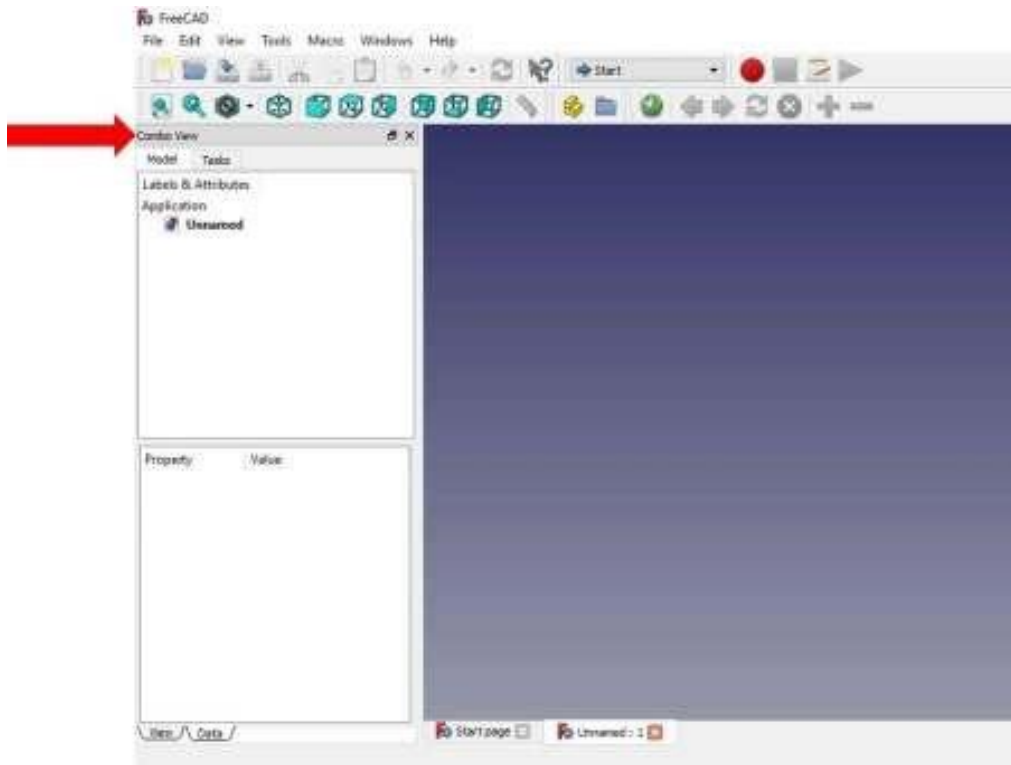


Figure 8 – Create a new document

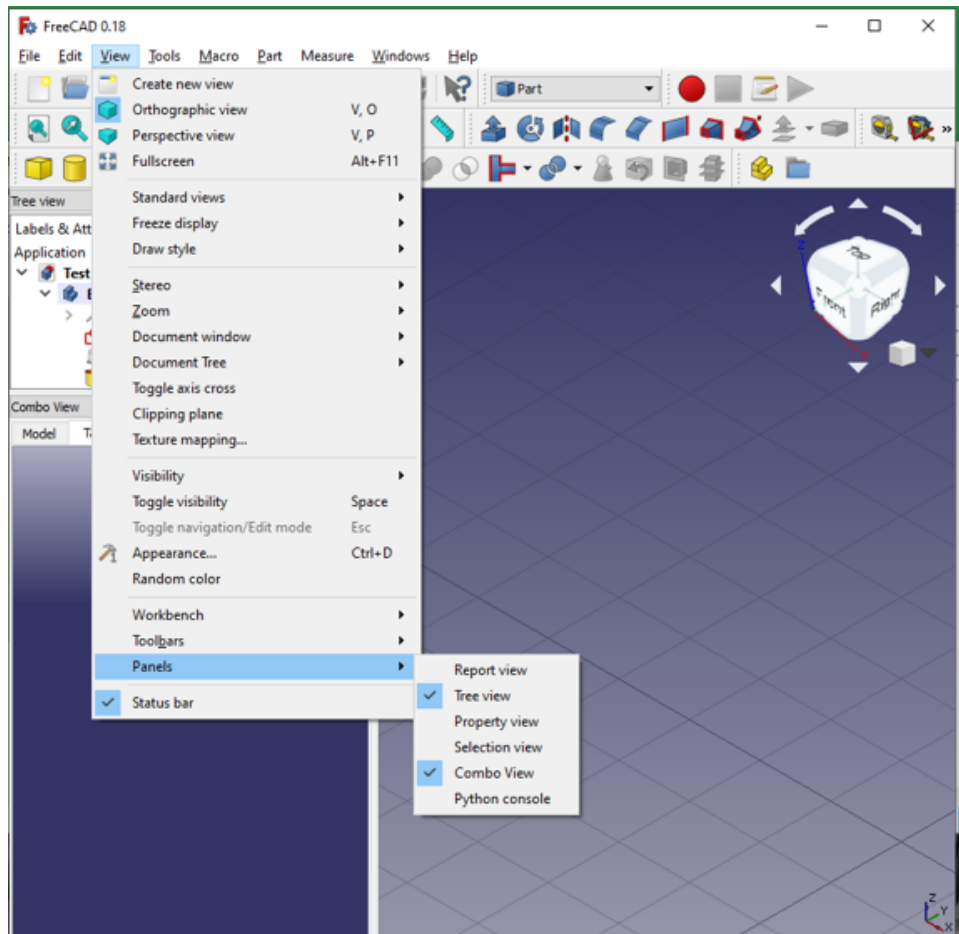


Figure 9 – Working with the Combo View

2.3. Sketch

To build a 3D object, you first need to sketch the base:

1. Change the **Workbox** from **Start** to **Part Design**. This will activate the **Create Sketch** option in the **Task Slide** in the **Combo View** window (Figure 10).

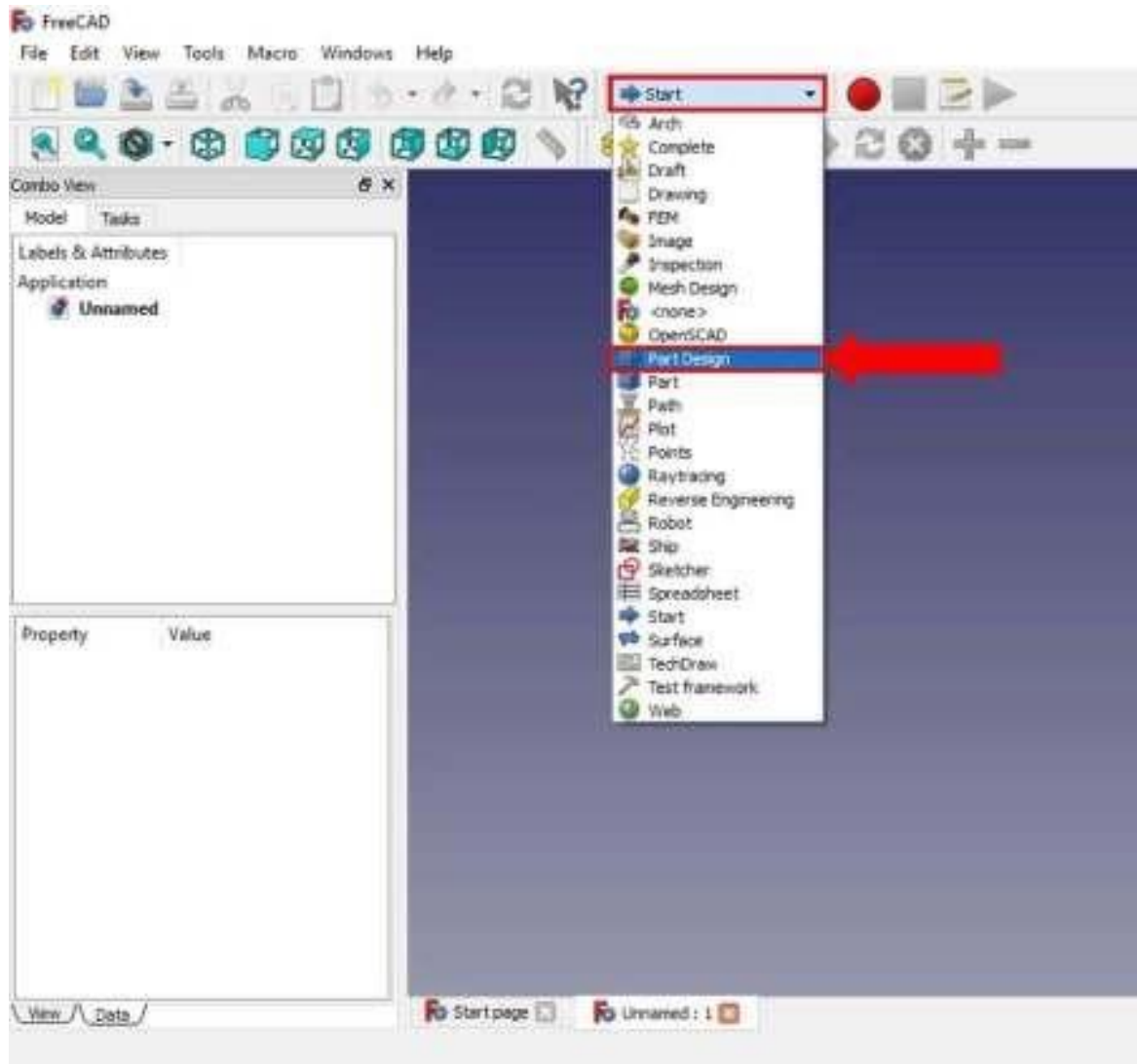


Figure 10 – Work with an option **Part Design**

2. Make sure you are under the taskbar in Combo View. Click Create body and then Create sketch. A new window will open for selecting the orientation of the sketch (Figure 11 and Figure 12).

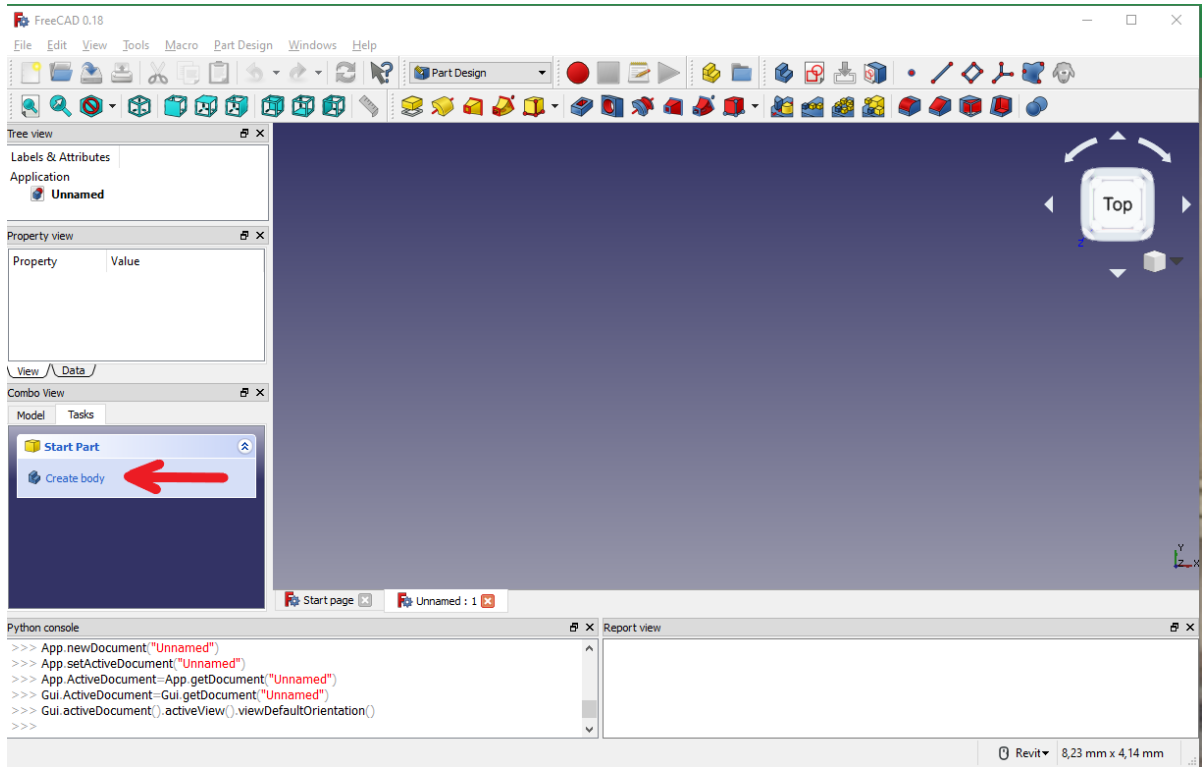


Figure 11 – Window with body creation

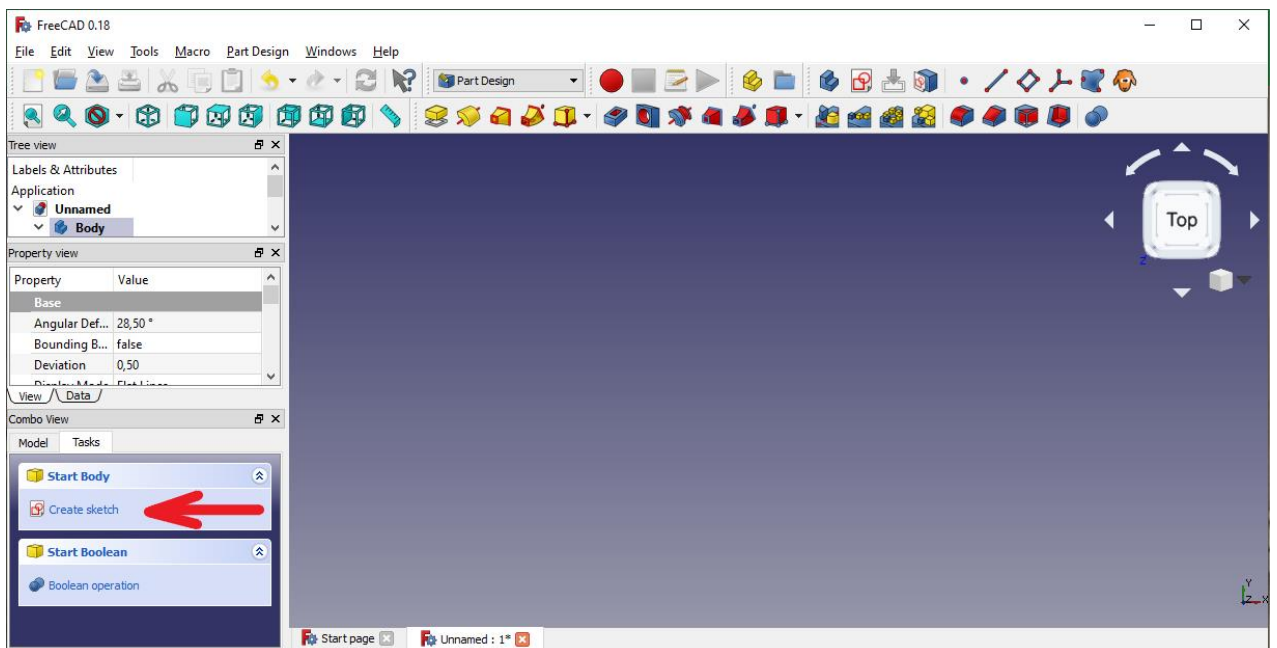


Figure 12 – Working with sketch creation

3. Choose a tool **XY-Plane**. In the latest edition **FreeCAD** you can continue by simply clicking (Figures 13 and 14).

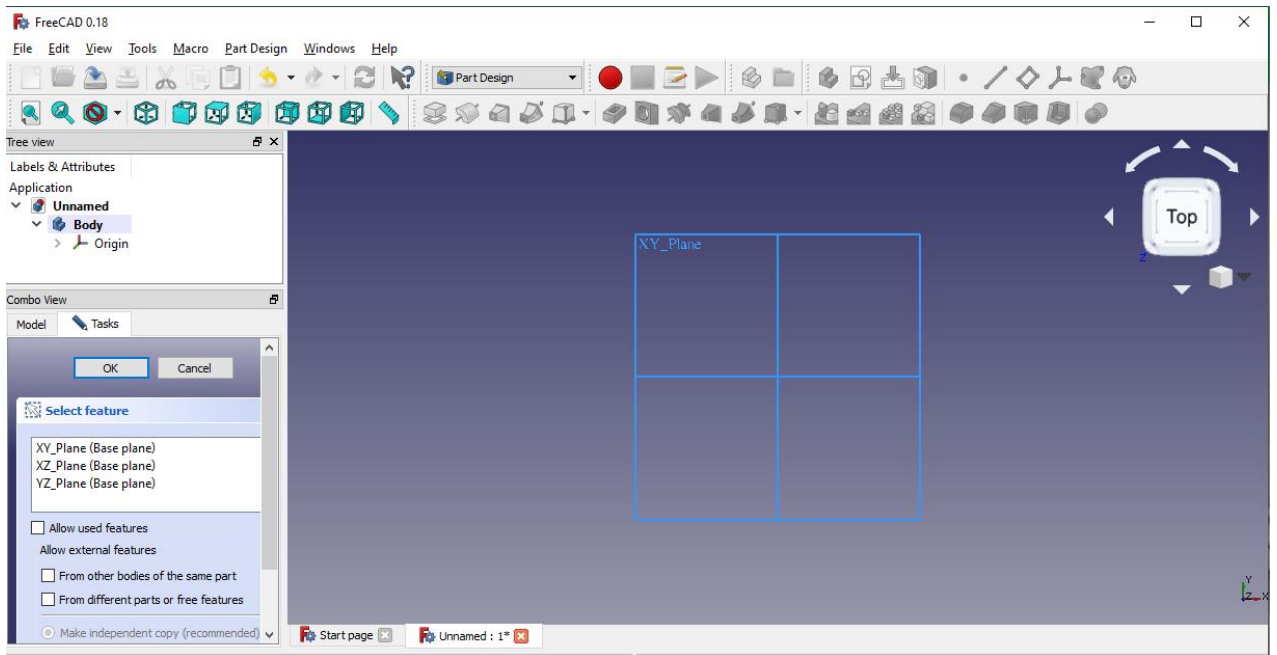


Figure 13 – Working with a cube **XY-Plane**

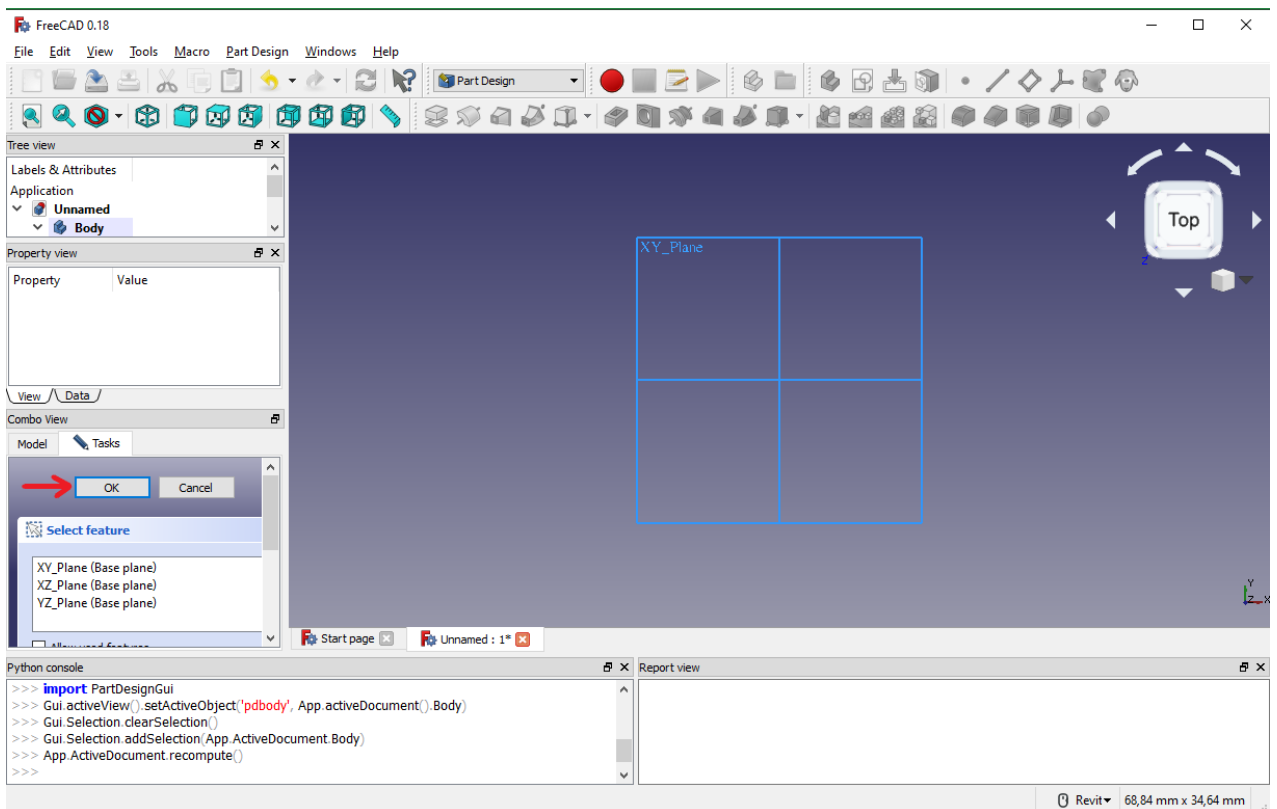


Figure 14 – Working with a **XY-Plane**

If you have an older version, uncheck the reverse direction and zero the offset. In this window, the Cube will show you the selected plane on the right. The

reverse direction option will switch to the opposite plane. With the **Offset** option, you can change the height of the plane. Click **OK** to continue (Figure 15).



Figure 15 – Offset option of the plane window

4. New options have appeared under the taskbar. Here we can change the properties of the grid. Check all the grid parameters and set the grid size to 10 mm. Unchecking the “**Show grid**” checkbox will hide\open the grid, and the “**Grid size**” will change its spacing. Selecting “**Grid Snap**” automatically snaps the point to the nearest intersection (Figure 16).

5. Select the **Create a Rectangle** sketch tool. When you hover your cursor over a coordinate system, you can see the current coordinates right next to your cursor. Click on the **IV quadrant** of the coordinate system (lower right quadrant) at point **40.0, -40.0** (Figure 17, 18).

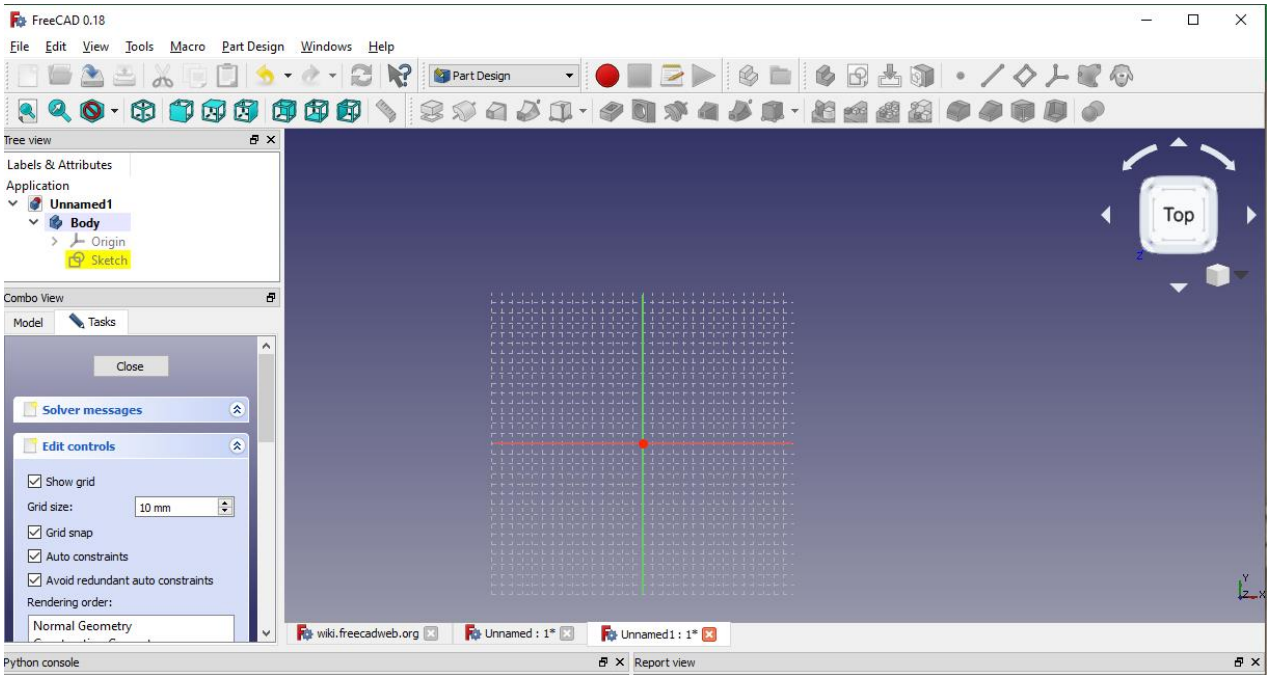


Figure 16 – Working with a **Grid** option

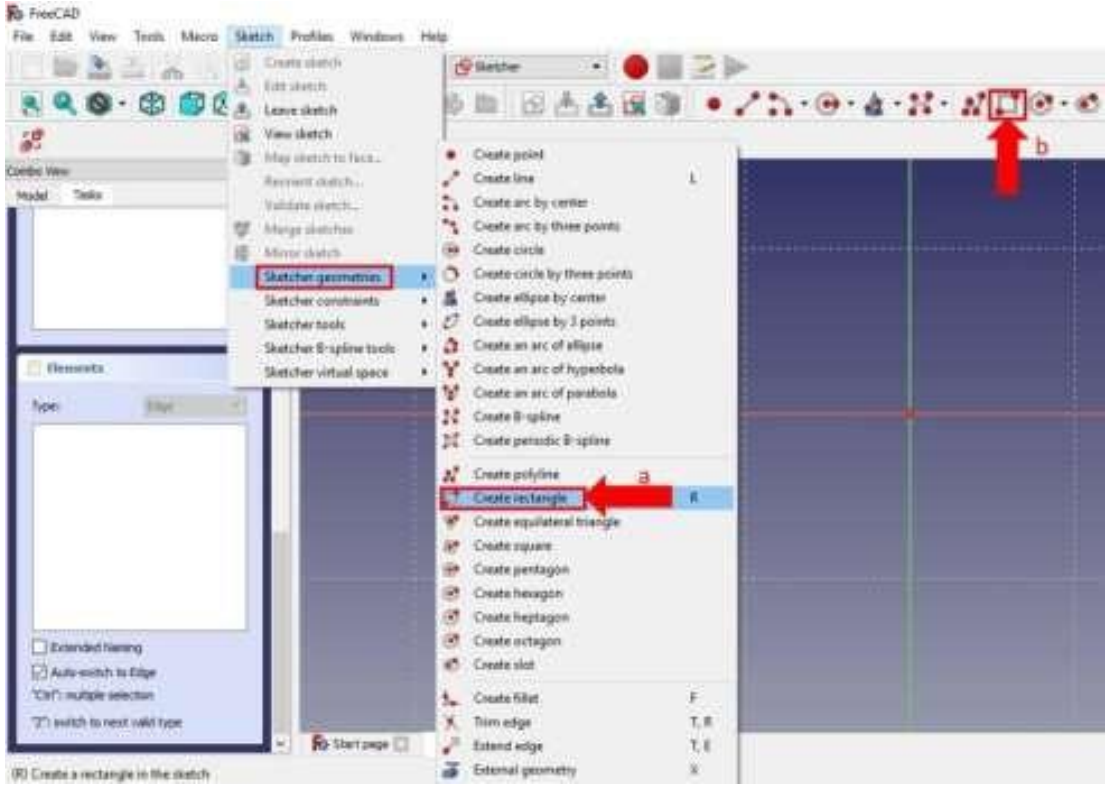


Figure 17 – Create a Rectangle

6. You may also notice how the points snap to the grid to make your work more accuracy. Make your rectangle **80.0 x 80.0** (drag the rectangle to - **40.0, 40.0**).

Set the grid options and select **draw rectangle** (Figure 18).

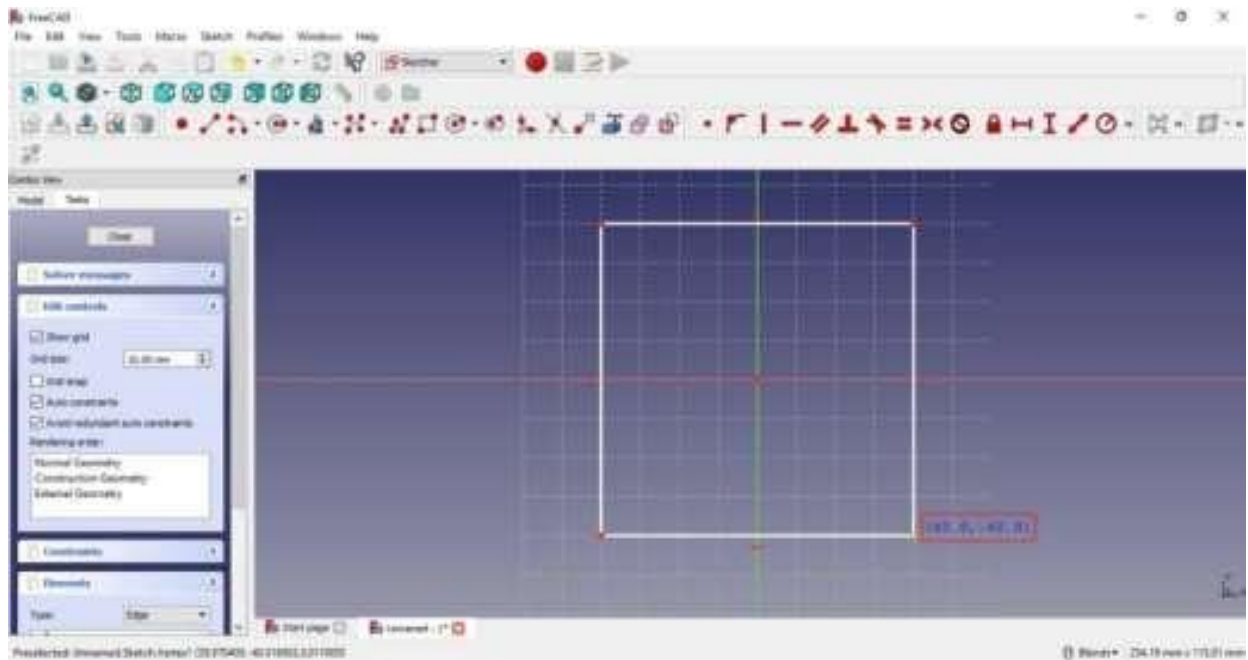


Figure 18 –Grid options and draw rectangle

FreeCAD automatically sets constraints on your sketch, as you can see in the “**Constraints**” window on the left. Click “**Close**” to exit the sketch. You can now extrude (**Pad** option) the sketch into a 3D object.

2.4. Setting new points

1. Select the **Pad** command in the **Task** section of the Combo View (Figure 19).

Note: If you don't see these options, go back to the model, and select **Sketch**. Then go to the **Task** tab.

2. Set the length to **60 mm** in the **Dimension** tab and uncheck **Symmetric to plane** and **Reversed**. The length will determine how far your object is extruded. The **Reversed** option will stretch the sketch in the opposite direction (Figure 20).

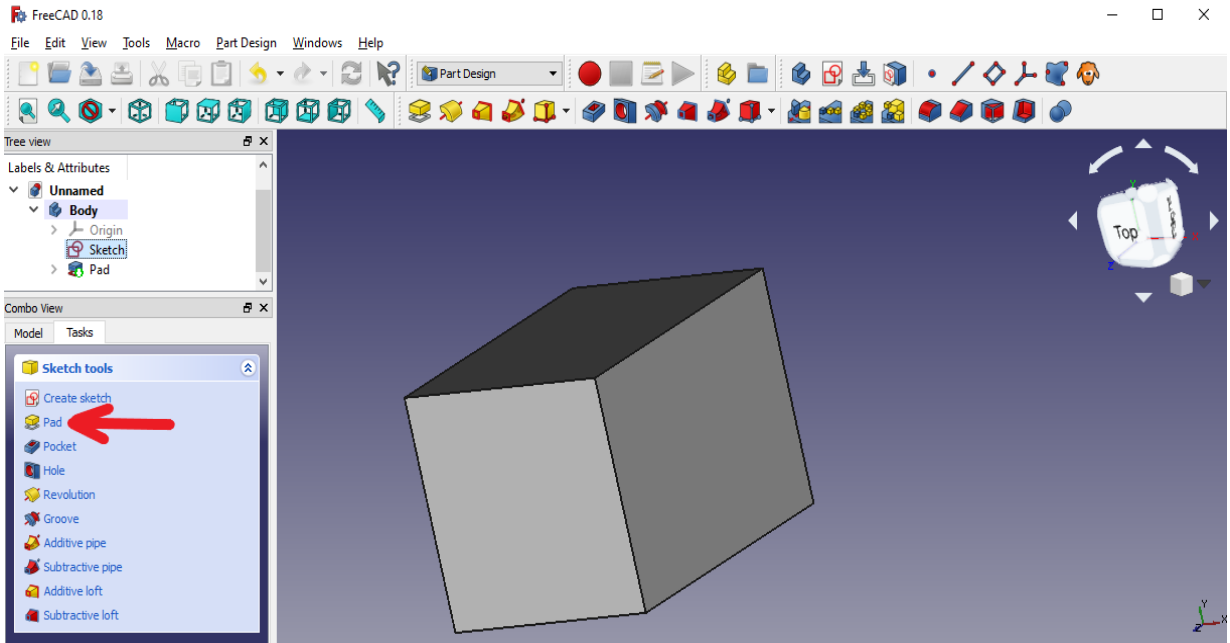


Figure 19 – Working with the **Pad** command

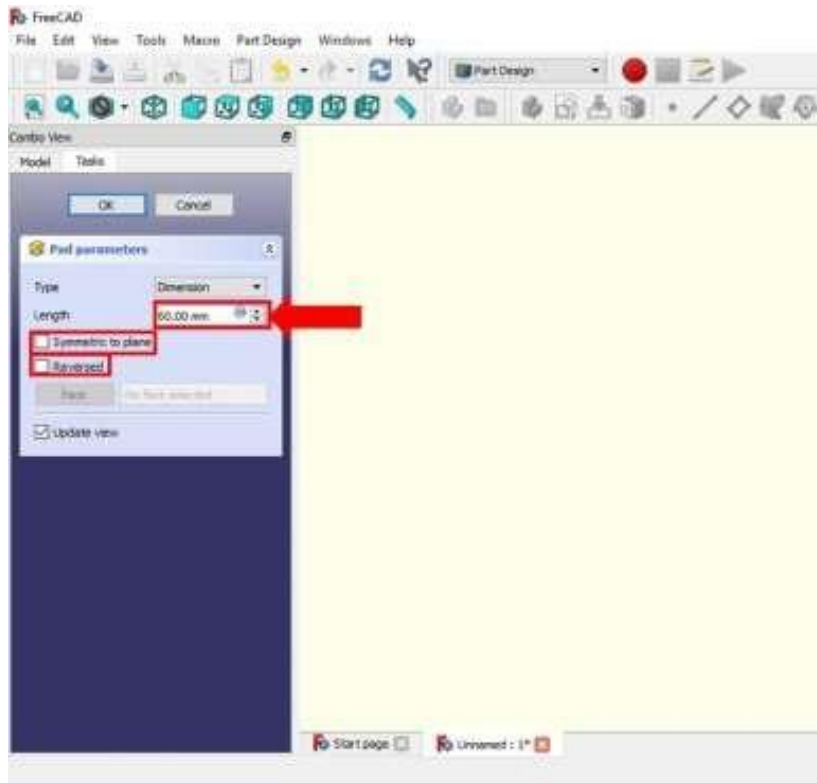


Figure 20 – Working with the **Reversed** option

Length 2 will be included by switching the “**Two dimensions**” tab and will extrude the object from both sides (Figure 21).

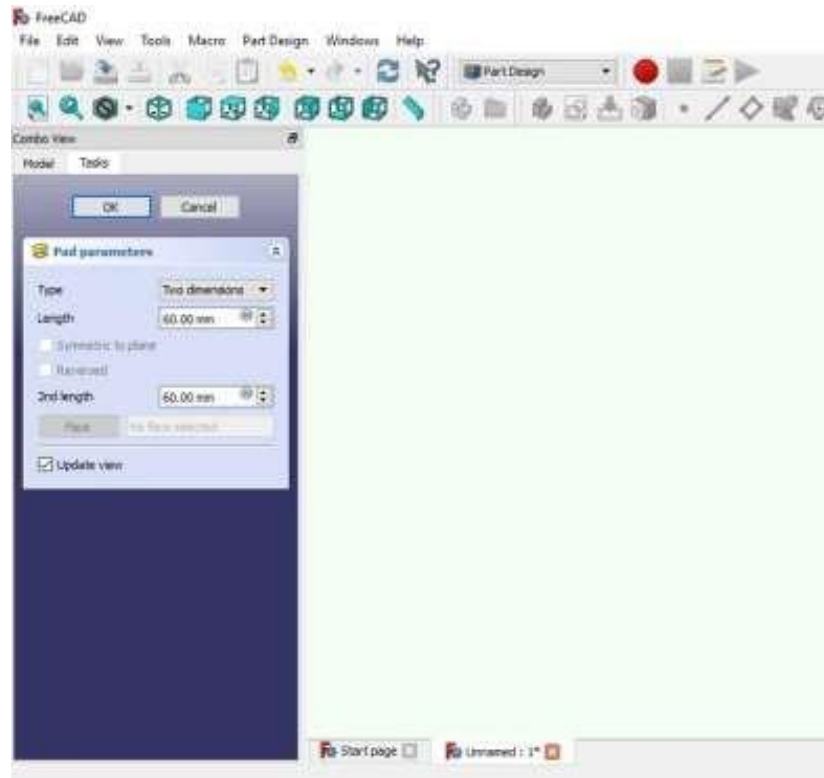


Figure 21 – Working with dimensions

We will not use this function for our model. Go back to the 1st dimension and click **OK**.

2.5. Moving objects

Moving objects has different points of view: 1 - isometric projection, 2 - front view of the object.

You can also move the object and look at it from different sides.

1. By holding down the middle mouse button and moving it, you will rotate the object. Note that this feature is available after right-clicking and selecting **Blender** under **Navigation Styles**.
2. You can also choose different viewing angles. The different views can be found in the center line of the FreeCAD toolbar.

Axiometric View projects an object in 3D. You can also choose one

specific side.

3. If you have lost track of your subject, click the zoom to the left of the viewer toolbar. This will scale your entire project.

Self-study Task:

1. Create a drawing of typical part 1 (Figure 22):

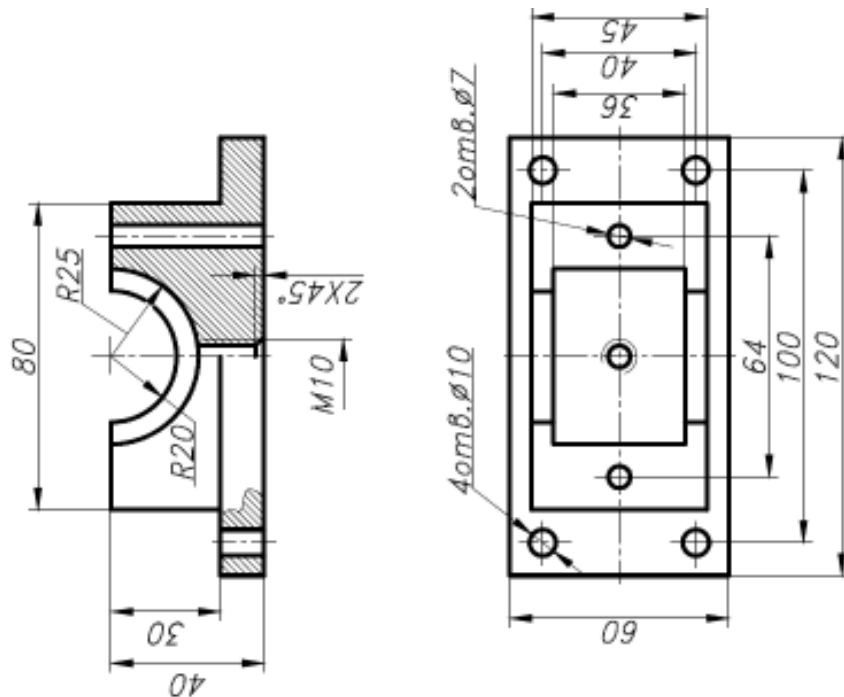


Figure 22 – Typical part 1

Control questions:

1. What is FreeCAD system?
2. What kind of modeling is it intended for?
3. What units of measurement does the system operate with?

SECTION 3. A SIMPLE PART CREATION

PartDesign and **Part** workbenches are widely used as starting points for working with three-dimensional objects. The **PartDesign** workbench is designed to create complex objects, starting with simple shapes, to which features are added and removed until the final object is obtained. All elements that are added during modeling are saved in a separate view, called the project tree, which also contains other objects in your document. The **PartDesign** object can be thought of as a series of operations applied to the results of the previous ones, creating one large chain.



Draft and **Arch** workbenches operate slightly differently, although they follow the general rules for FreeCAD. **Sketcher** and **PartDesign** workbenches are designed primarily for designing unified parts, while **Draft** and **Arch** workbenches for working with a few simple objects. The **Draft** workbench provides 2D tools. Most of the **Draft** workbench tools work not only on a plane, but also in volume, and benefit from special assistance systems such as plane selection and snapping to objects. The **Arch** Workbench adds a BIM tool to FreeCAD that allows you to create architectural models through parametric objects. The **Arch** workbench depends on other modules like **Draft** and **Sketcher**. All **Draft** tools are also available in the Arch workbench, and most **Arch** tools use the **Draft** workbench support systems.

In this lesson, you will use the part construction tools to create the 3D solid model of the part shown below in the **Drawing** module. You will start by creating a basic shape from a base sketch, and then create a part adding the necessary elements. These elements will either add material to the solid or remove it from the solid using additional sketches and appropriate operations. This lesson will not use all the functions and tools that are available for developing parts, but they will be enough to provide a foundation for creating a simple part on which you can develop your knowledge and skills.

3.1. Workbench Tools (Part Design)

First, make sure you are on the Part Design workbench menu. Once you've done this, create a new document if you haven't already. It's a good habit to save your work often, so first, save the new document with any name you want.

Part Design begins by creating the body of the object. We will then build the base of the solid, starting with a sketch.

1. Click on  "Create New Body" to create and activate a new body container. When sketching, if no existing body is found, a new body will be automatically created and activated.
2. Click on  "Create New Sketch". This will create a sketch in the body you just created.
3. We need to define where the sketch will be attached. We will attach it to the plane of the **Origin** body.

On the "**Tasks**" tab in the combo panel, select "**YZ_Plane**" (Figure 23) in the list and click OK.

Note: If the OK button is not visible, you can make the sidebar wider by dragging the right border. Put mouse cursor over the border; when the pointer changes to a double-headed arrow, press and hold the left mouse button and drag.

When you click OK, FreeCAD will automatically switch to the Sketcher Workbench and open the sketch for editing (Figure 24):

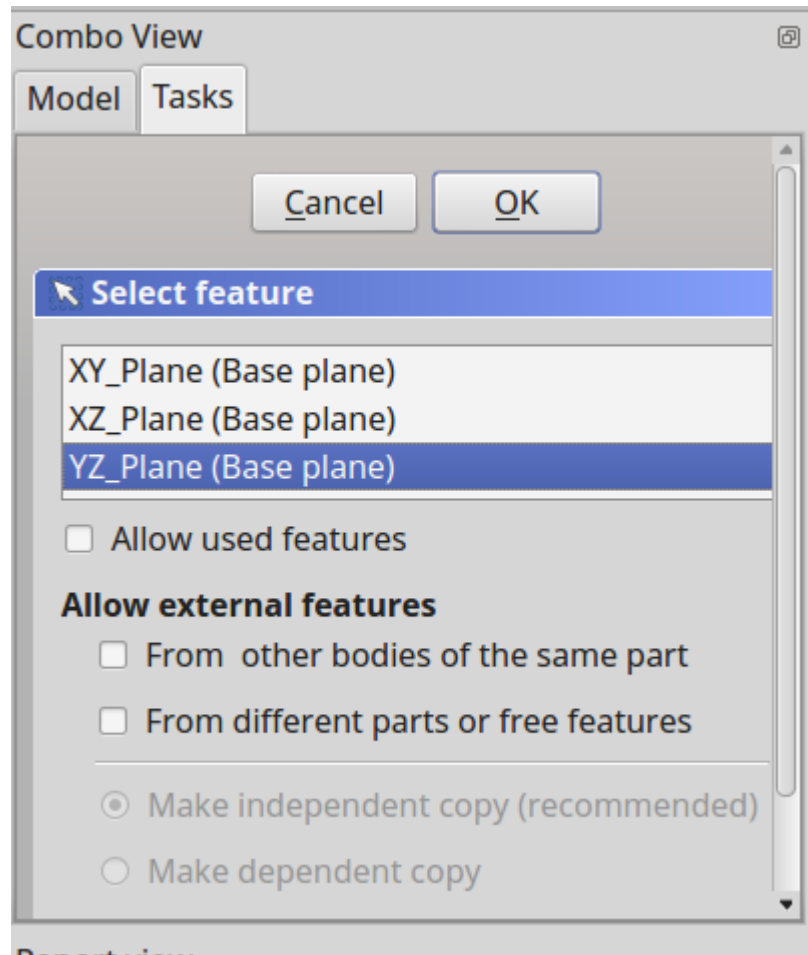


Figure 23 – Combo View Window Tabs

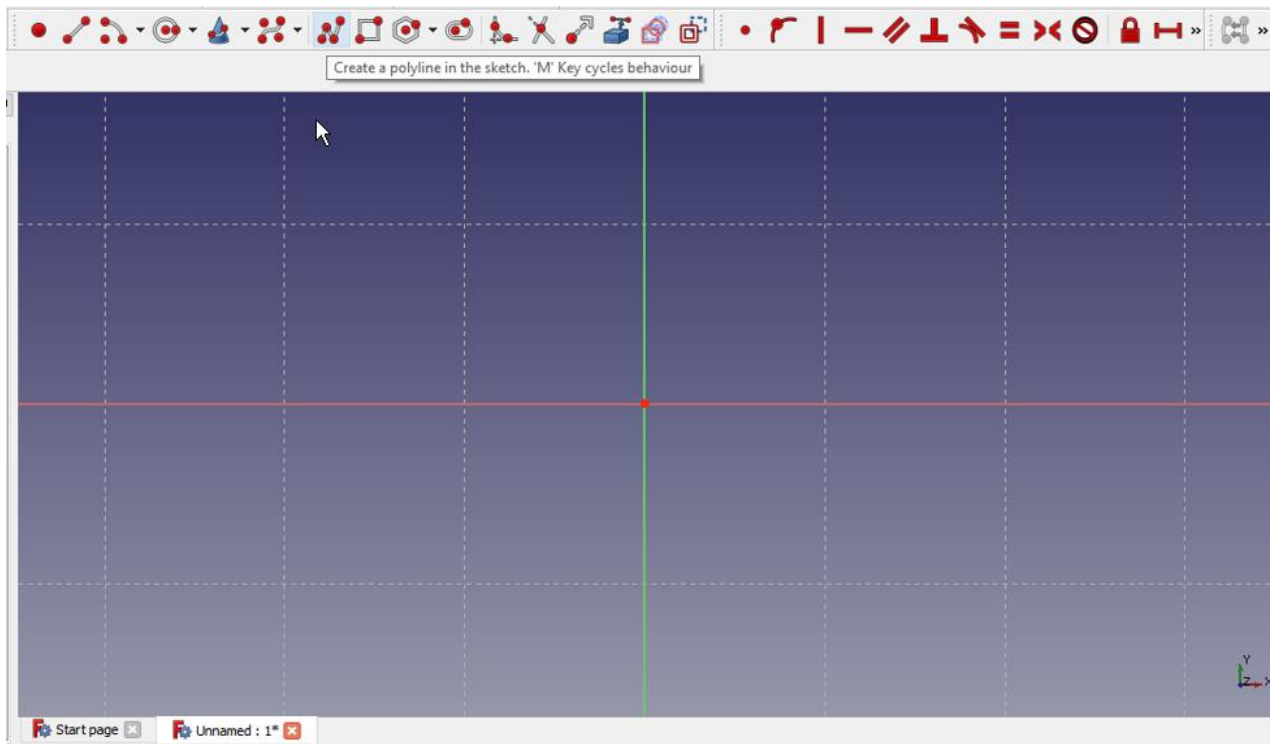





Figure 24 - Sketcher Workbench


Next, you can use the **Polyline** tool  and create a new shape. It doesn't have to be perfect, as the final shape will be constrained. Once you get the basic shape, we'll start applying constraints. If you have the Auto Constraints option enabled, some of the constraints will be applied automatically, if not, follow these steps. But first make sure you exit the polyline construction tool by double-clicking the right mouse button or pressing ESC; the mouse cursor should return from the crosshair to the standard arrow cursor.


Note: Do not press ESC a third time, otherwise you will exit sketch editing mode; if this happens, click on the "Model" tab, then in the tree of elements, double-click on the Sketch, or right-click and select "**Edit Sketch**" from the context menu.



To edit a sketch, follow these steps:

1. Select with the mouse two horizontal lines by clicking on them, and after selection click on  the horizontal constraints

2. Select the vertical line on the right and then click on  the vertical constraint.


3. Select the start and end points of your polyline and click on  "Coincident" selected.

4. Select the bottom horizontal line and the right vertical line and set  the "Equal" between the two lines.

5. Select a horizontal or vertical line and set the appropriate  horizontal or  vertical distance by setting the value to 26 mm.

6. Select the top horizontal line and set the horizontal dimension by setting

it to 5 mm.

7. Select the bottom-right point of the horizontal Start line, and then the center point of the grid, and apply  a merge of points to fix your shape.

At this point, you should have a fully constrained sketch, as indicated by its color change and the message displayed in the combo panel.

The sketch should now look like Figure 25.

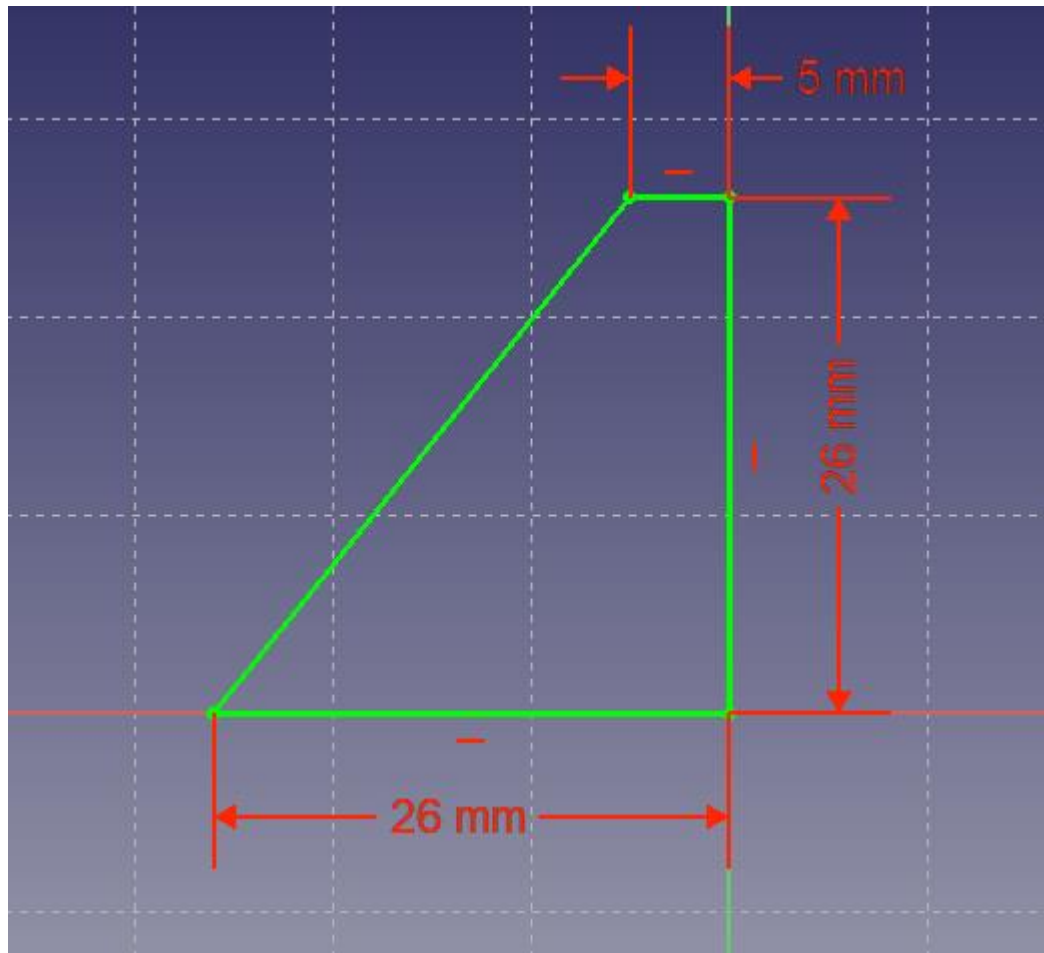



Figure 25 - Sketch of a figure

Next, in the editor's combo view, click the Close button, and to exit the sketch

editing mode, select  **Pad** from the toolbar or from the Part Design menu.

This will open the **Pad** dialog box in the combo panel. Using this dialog, first use the Type drop-down menu to select two dimensions: the part length is 53 mm.

You will do this by extruding (**Pad** command) our sketch to both sides of the center plane to get this distance, i.e., extrude symmetrically to the location of the sketch plane.

Now, if you want the length to be 53 mm, enter 26.5 for the length and 26.5 for the second length. Alternatively, you can specify a single length of 53 mm and select the “**Symmetrical Plane**” checkbox. Once this is done, you will have a base on which to add additional elements to build your part.

3.2. Applying the Pocket and External Geometry functions

Using the mouse or the view button, you can rotate the model to see its rear side. Once the rear face of the part is visible, select the face by clicking on it, as shown in the following Figure 26.

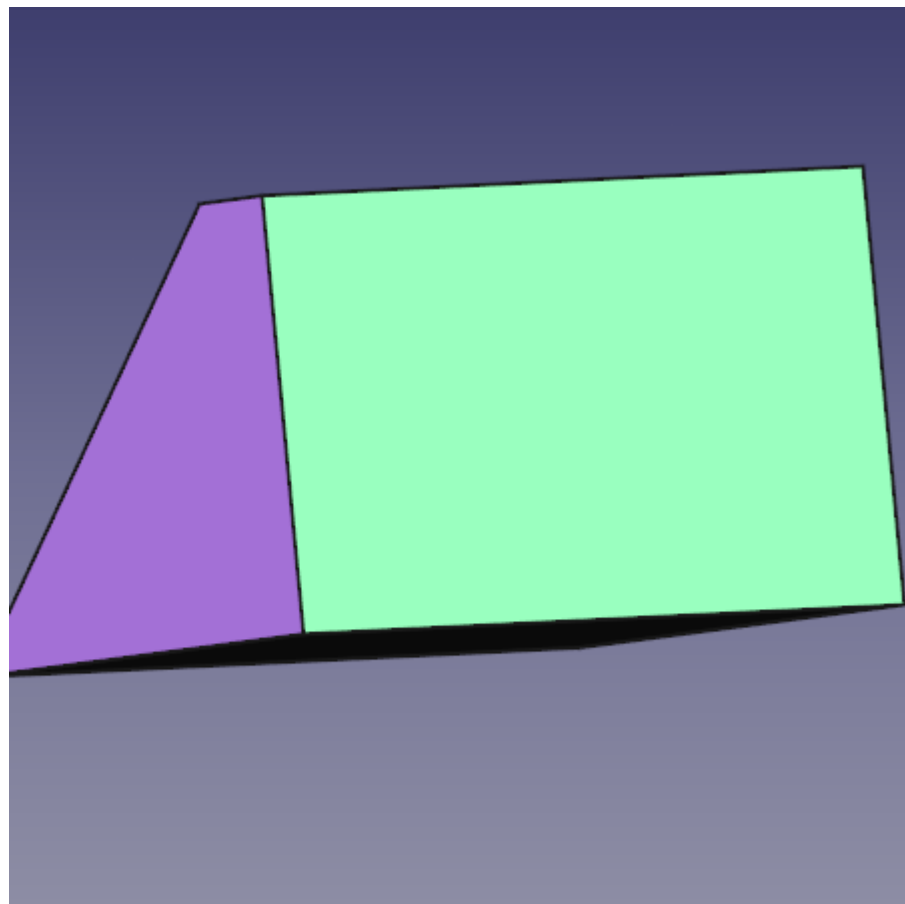


Figure 26 - Rear face of the part

After selecting a face, click on the “New Sketch” icon in the toolbar or in the “Part Design” menu and this will place your next sketch on the rear face of the part.

Now select the next  “Rectangle” tool and place the rectangle on the rear face in the same way as shown below.

Follow the steps listed to constrain the sketch:

1. Select one of the horizontal lines, set the horizontal dimension and set the value to 5 mm.

2. Select one of the vertical lines and set its vertical dimension to 11 mm.

3. Select  the “External Geometry” tool.

4. Select the top right point of the face and click on it to get a point from the external geometry to link our sketch to it (Figure 27).

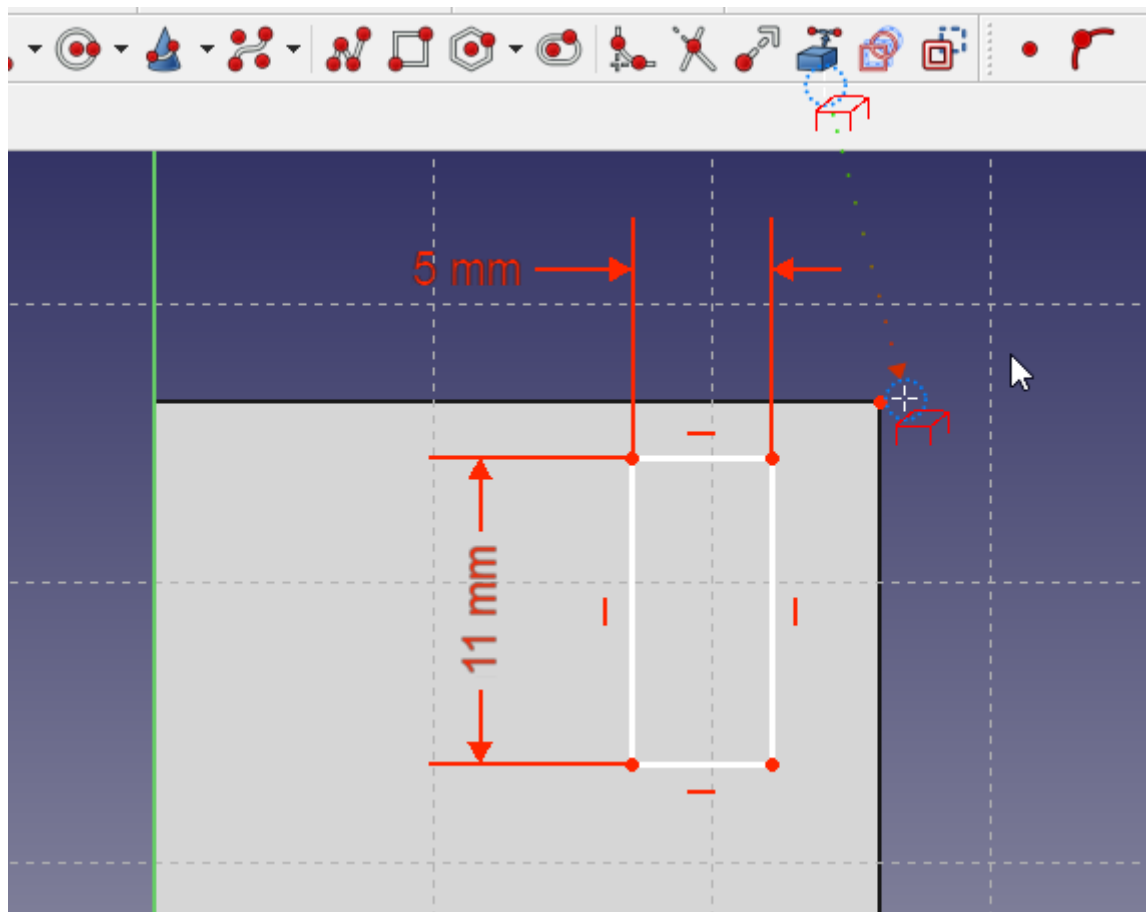



Figure 27 - Application of the tool "External geometry"

5. Click the right mouse button to exit the External Geometry mode.
6. Select the point you just made available with the External Geometry tool, then select the top-right point of the rectangle and merge the selected points.

At this stage, the sketch should be completely constrained and look like the one shown in Figure 28.



Figure 28 - Sketch after "constraint"

Once this is done, click the “**Close**” button at the top of the “**Tasks**” tab in the combo window, then select the “**Pocket**”  tool from the toolbar or from the “**Part Design**” menu. The action of this tool is the opposite of the “**Pad**” tool. The “**Pad**” tool adds material to the part, the “**Pocket**” tool removes material from the part. Both operations are functions. In this cut operation, we need to select “**Through**” from the Type drop-down menu and then click OK.

For the next step, make sure the “Pocket” tool is selected in the Model

Tree View, and only then click the  “**Mirrored Feature**” tool from the toolbar or from the “**Part Design**” menu. In the “**Mirrored Transform**” dialog in the combo panel, select “**Horizontal Sketch Axis**” from the Plane drop-down menu. Then click OK.

“Mirrored” works this way because the base element of your model was extruded (**Pad** command) to either side of the horizontal plane in the first base sketch operation. If everything went well, you should now have a detail like Figure 29 after you rotate it to face first.

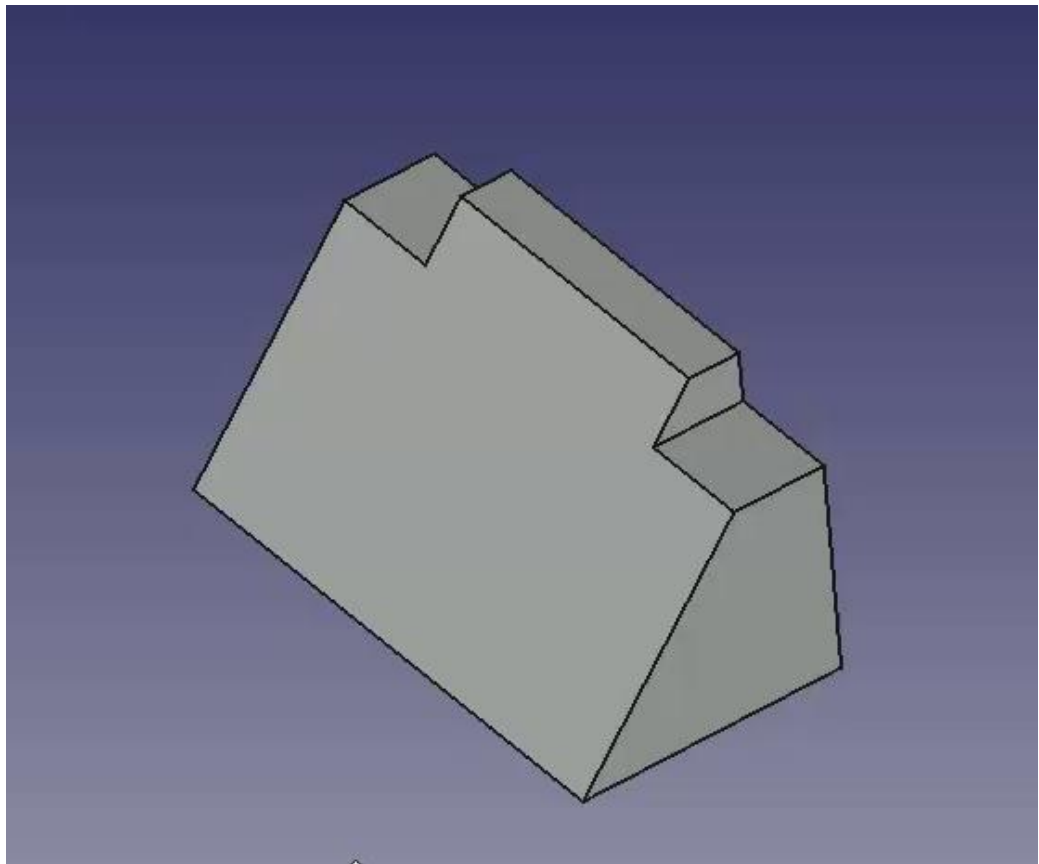


Figure 29 – Using the " Mirrored" function

3.3. Pad and External Geometry applying function

After inspecting, flip the part back and select the Rear Face of the part again to place the next sketch on it.

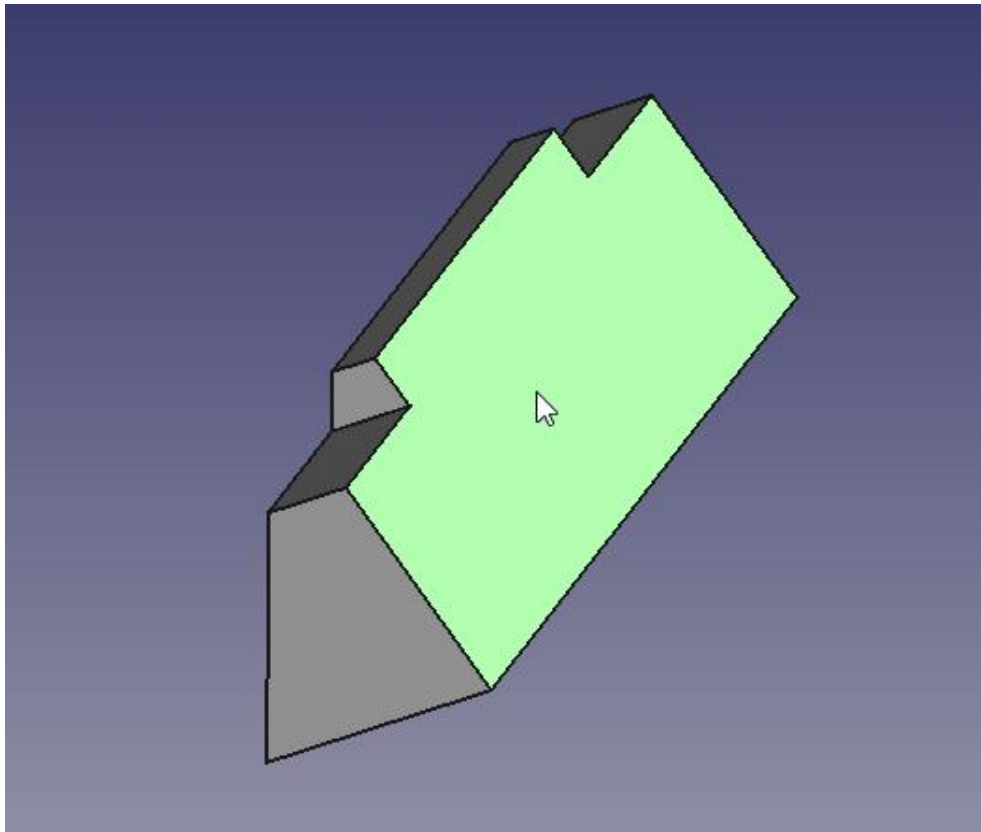


Figure 30 - Rear face of the part

Select New Sketch and create a new rectangle in the same manner as shown in the image below. Then set dimensions of the rectangle.

1. Select the horizontal line and set the horizontal dimension by setting the value to 16.7 mm.
2. Select the vertical line and set the vertical dimension by setting the value to 7 mm.
3. Using the “External Geometry” tool, select the top-left point of the part face.

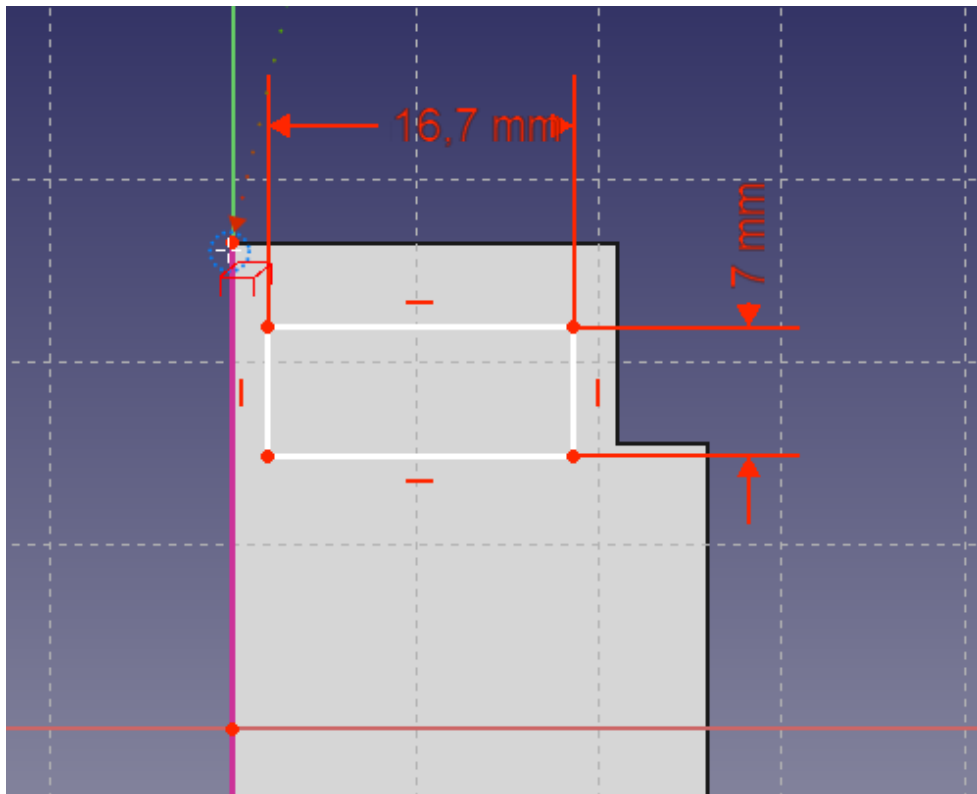


Figure 31 – Using the External Geometry tool

Now with the top-left point of the rectangle and the external geometric point selected, merge (“Coincident”) the selected points to completely constrain the sketch.

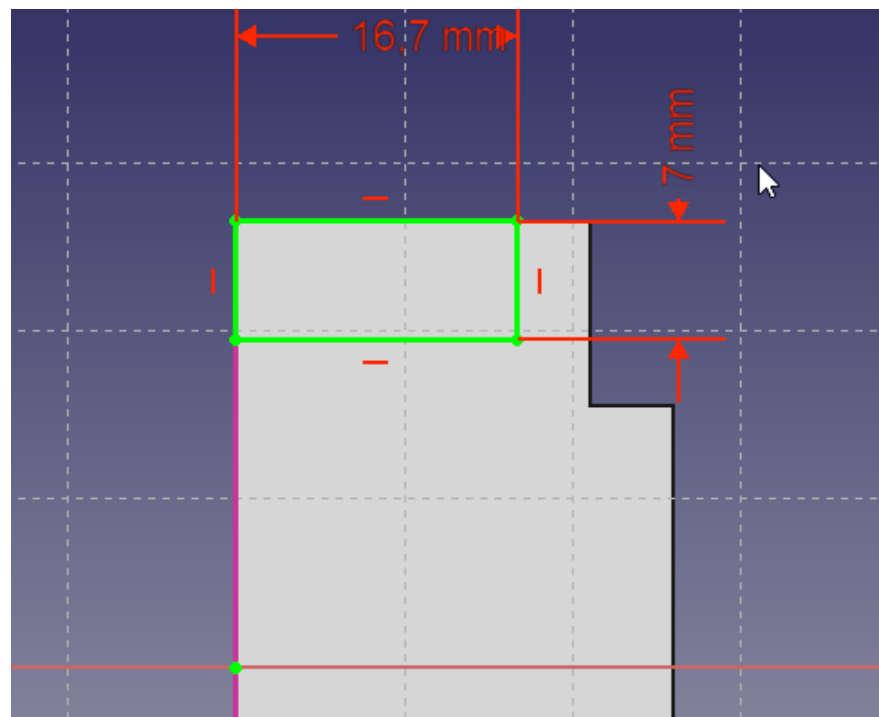


Figure 32 – Coinciding selected points

Next, click on the **"Pad"** function and in the **Pad** dialog box in the ComboView panel, set the length to 26 mm, setting the Type field to dimension, and then check the "B" checkbox for the reverse side. Using the B flag on the reverse side will cause the extrusion to go into the part, not away from it. This operation will produce the following result:

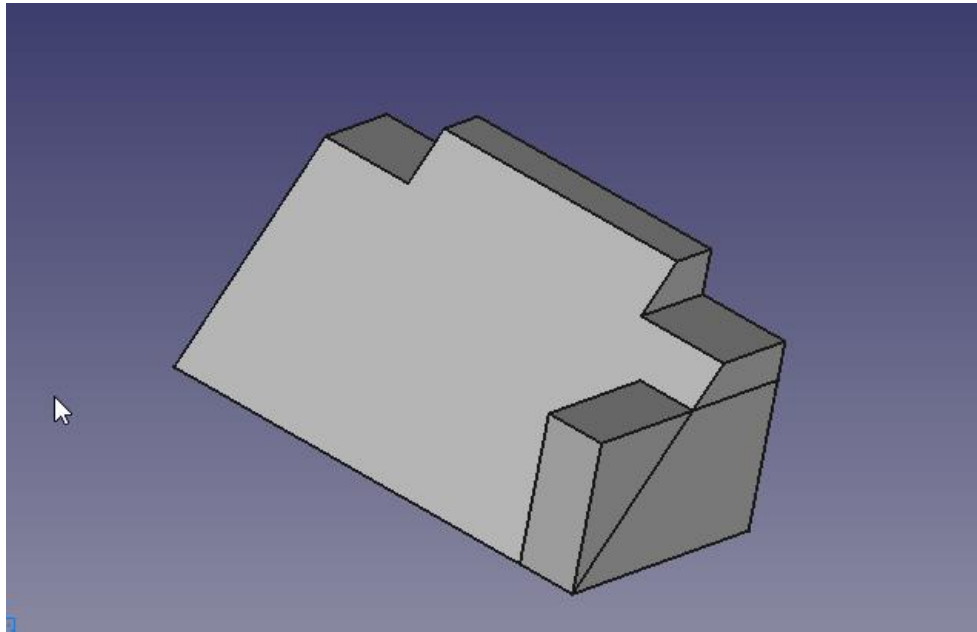


Figure 33 – Using the "Pad" function

Use the “Mirrored Feature” again to create a second extrusion (**Pad** command). First, make sure the extrusion you created is selected in the Model Tree, then click “Mirrored Feature” in the toolbar or select it from the “Part Design” menu. Repeat the operation you used for the cut above and select “Horizontal Sketch Axis” from the “Plane” field drop-down menu.

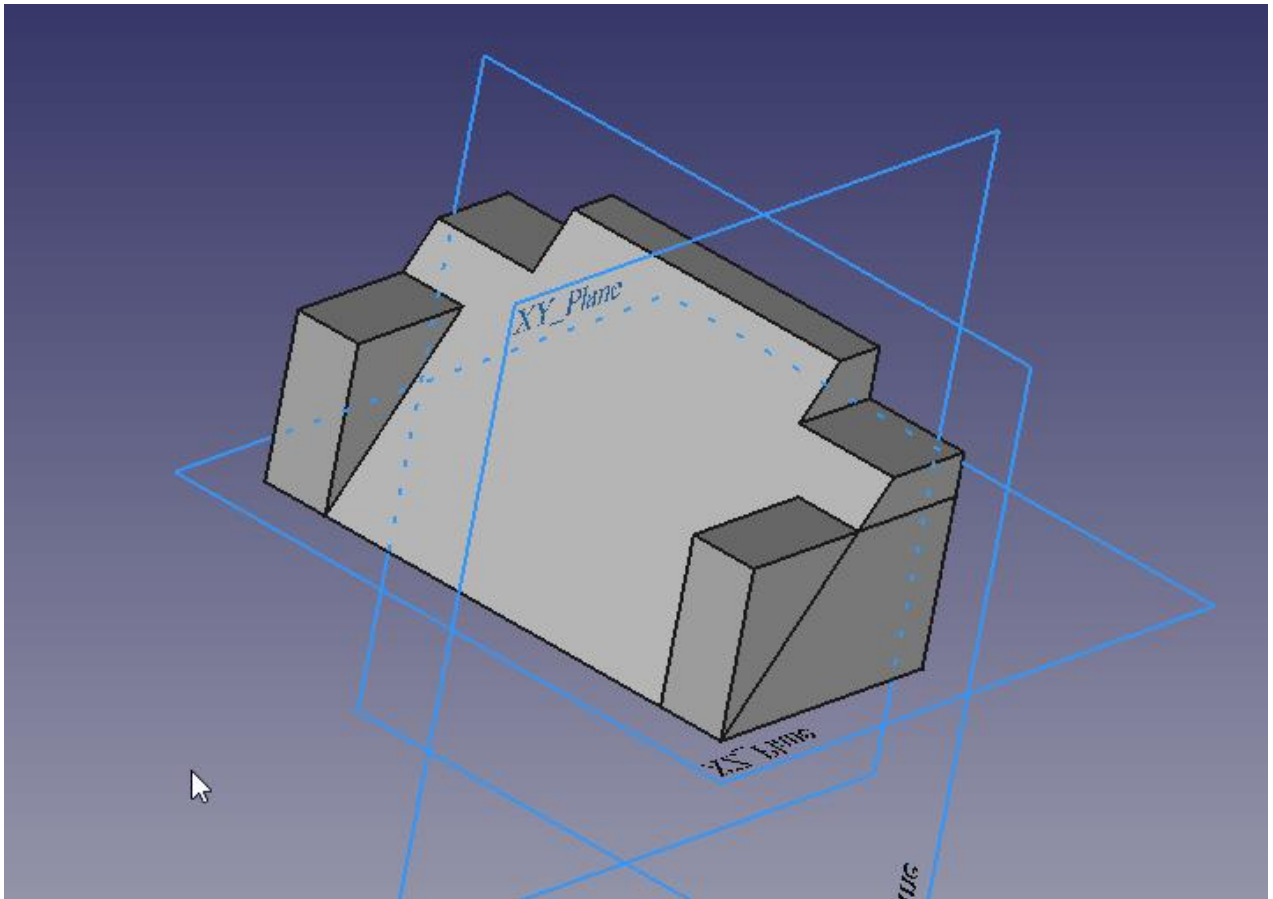


Figure 34 – Using the Element of “Mirrored feature”

3.4. Constraint and external geometry function

Sketcher's "constraint" functions allow you to define exact geometric dimensions for shapes by defining lengths, angles, and ratios (horizontal, vertical, perpendicular, etc.). The “constraint” solver interactively calculates the degrees of freedom constraints of the sketch geometry.

Most CAD models are based on a plane 2D sketch, since a 2D sketch can be "extruded" (“Pad” option) to create 3D shapes; 2D sketches can be used to create other features, such as cutouts, ridges, or "add-ons" over previously drawn volumetric shapes.

At this point, as you rotate the part around the face, you can see that it starts to look like the part in the dimensional drawing as at the beginning of this tutorial. Once you see the front of the part, click on the angled face to select it for the next sketch.

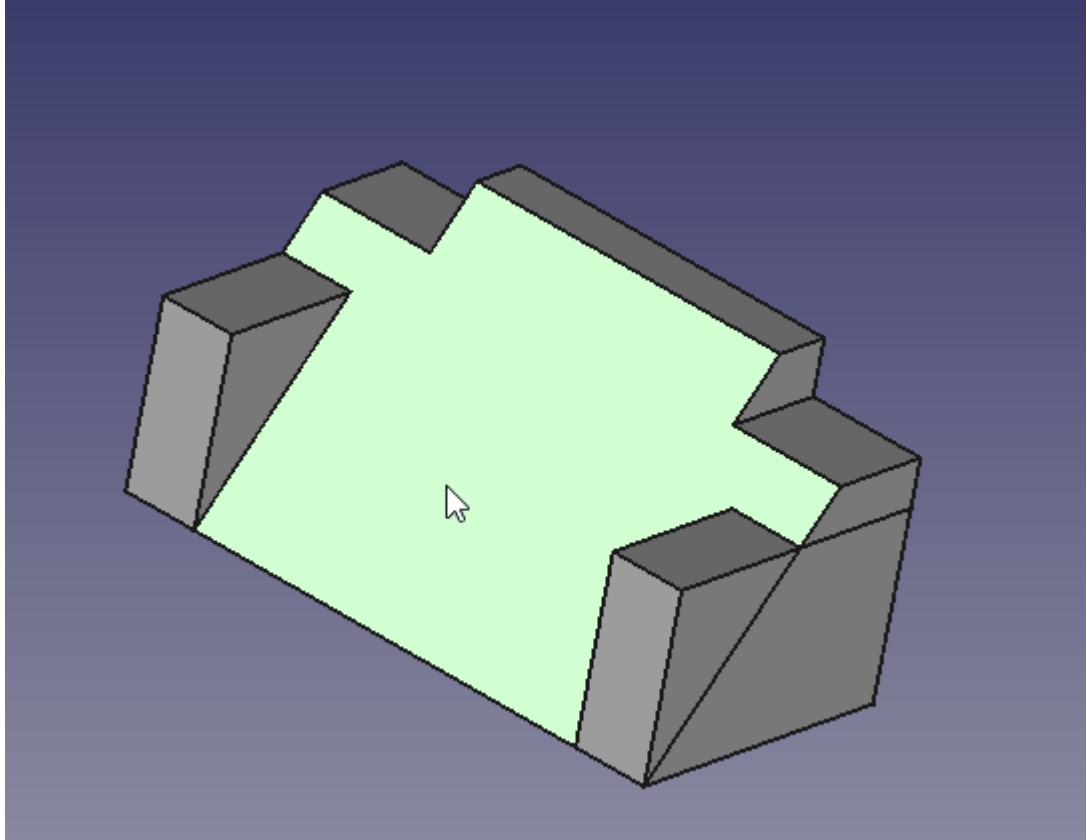


Figure 35 - Workpiece part

Here you will use the “Rectangle” Tool and place a rectangle in your sketch, and then apply the following constraints:

1. Select the horizontal line and vertical line, and after both are selected, set "Equal".
2. Select a horizontal or vertical line and set the appropriate horizontal or vertical dimension with a value of 17 mm.
3. Using the External Geometry tool, select the top right point as shown in the Figure 36.

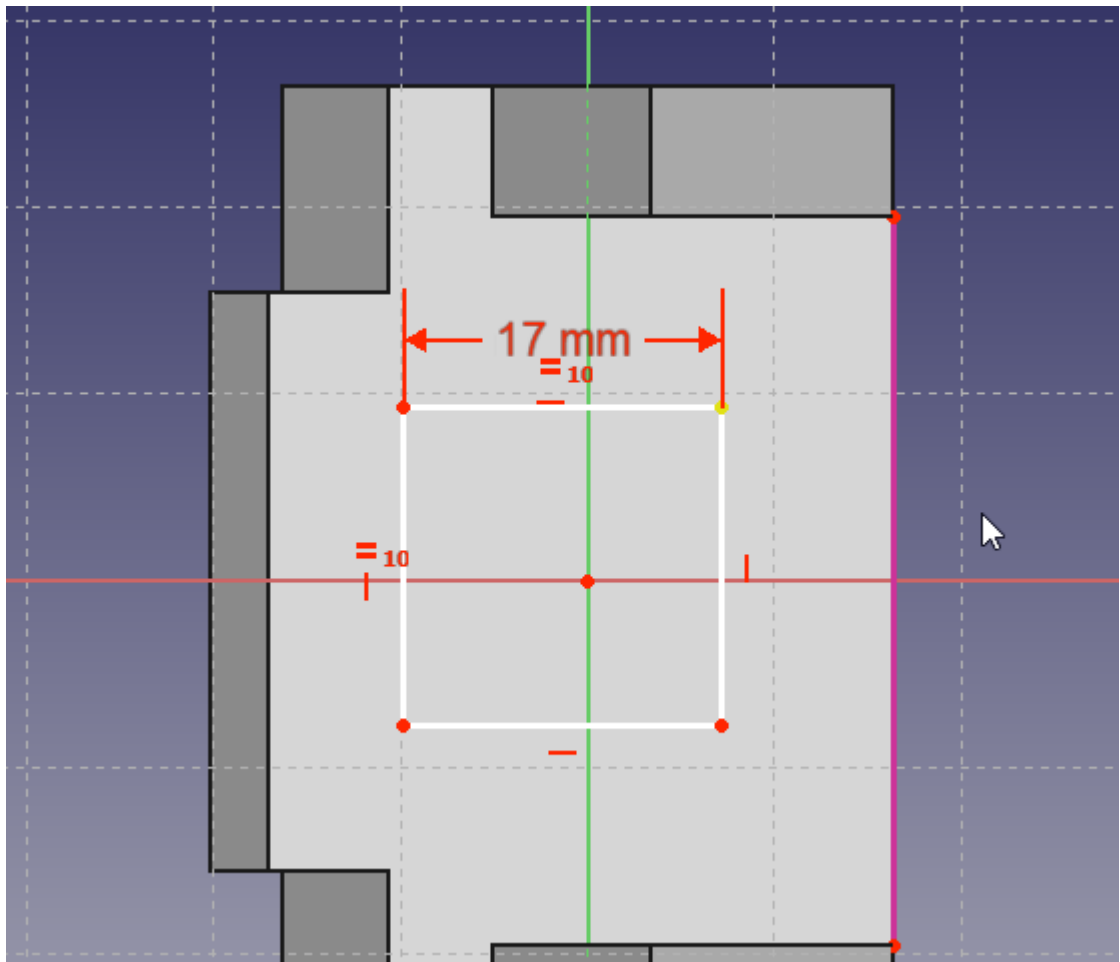


Figure 36 – Using the External Geometry tool

Now, using the dimensions from the drawing, apply the following constraints:

1. Select the external geometry point and the top-right point of the square and set the horizontal dimension to 7 mm.
2. Select the external geometry point and the top-right point of the square and set the vertical dimension to 11 mm.

The result should be as follows (Figure 37):

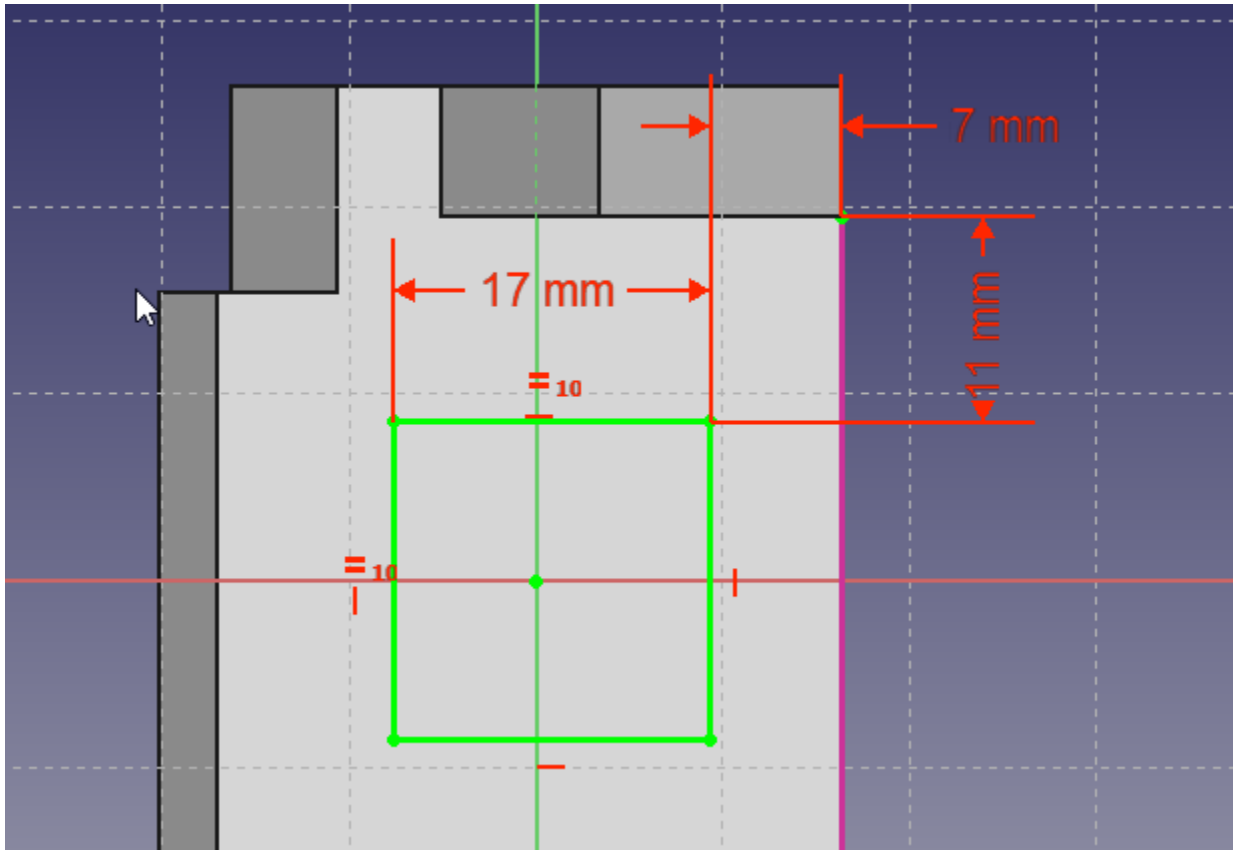


Figure 37 – Setting the dimensions of the part

For now, if you just cut out this sketch, the resulting hole will be perpendicular to the sloped face it was drawn on, and this is not what you want.

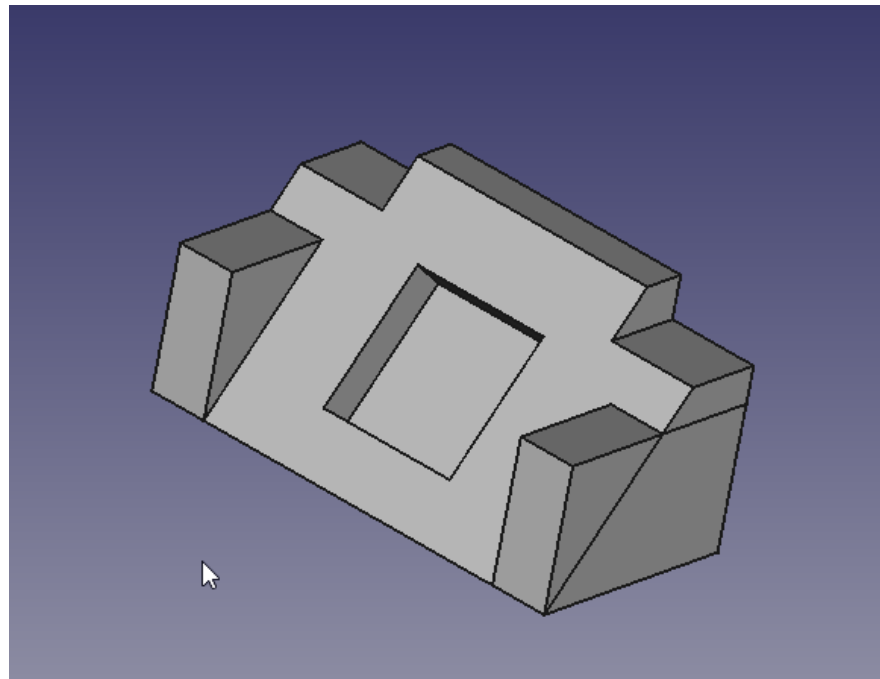


Figure 38 – Pockets in the workpiece part

You need the created pocket to be perpendicular to the rear face, but its projected dimensions do not match the 17 x 17 mm shown in the drawing. You can calculate the required size, or you can use the tools provided by FreeCAD to make these calculations.

To create a slanted rectangle cutout at the exit, you need to draw a new rectangle on the rear face using the slanted rectangle's projection as an outer reference. Rotate the solid to see the rear face of the part again and select it to place the final sketch.

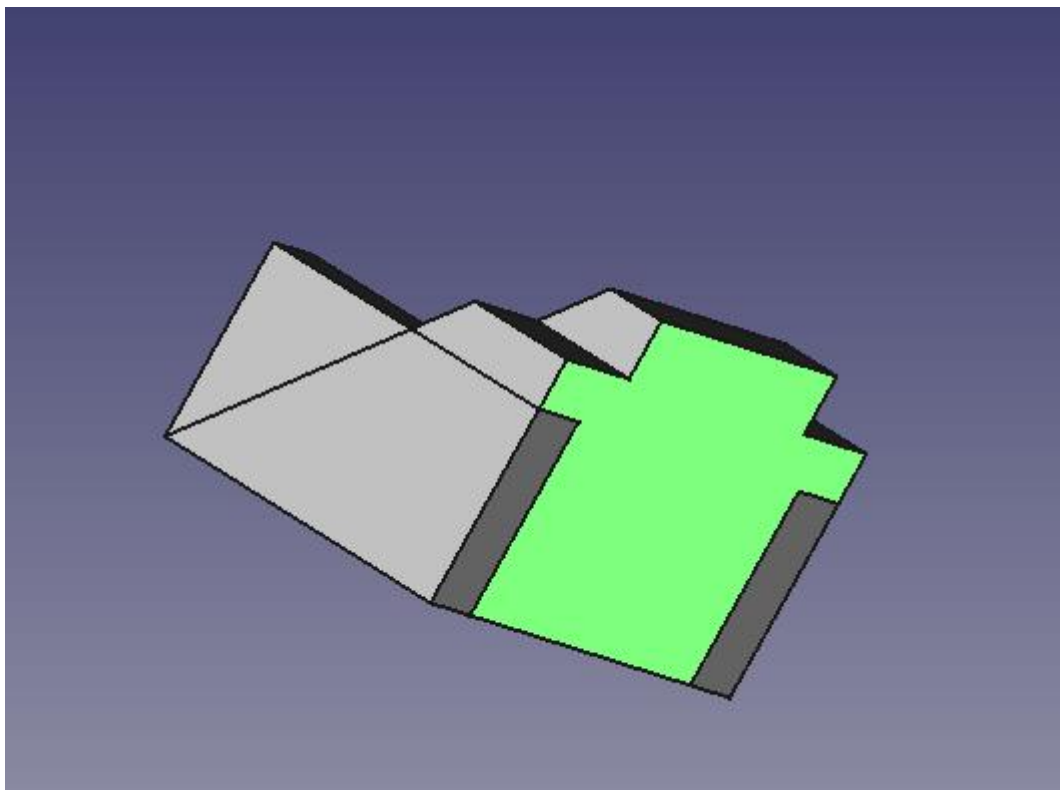


Figure 39 – Final sketch

Select “New Sketch” from the toolbar or the “Part Design” menu. Now, in sketch edit mode, you do not see the drawn rectangle on the slope. To be able to select it, switch the combo panel to the model tab and select the last sketch made on the oblique face. Then, using the spacebar, make it visible. Next, select the element “Symmetries” and again, using the space bar, hide it. You should now be able to see a slanted rectangle inside the 3D view. You can continue working with the “Model”

tab visible, or return to the “Tasks” tab. Using the “External Geometry” tool, select the top and bottom edges of the slanted rectangle. Then add a new rectangle to the sketch using the “Rectangle” tool:

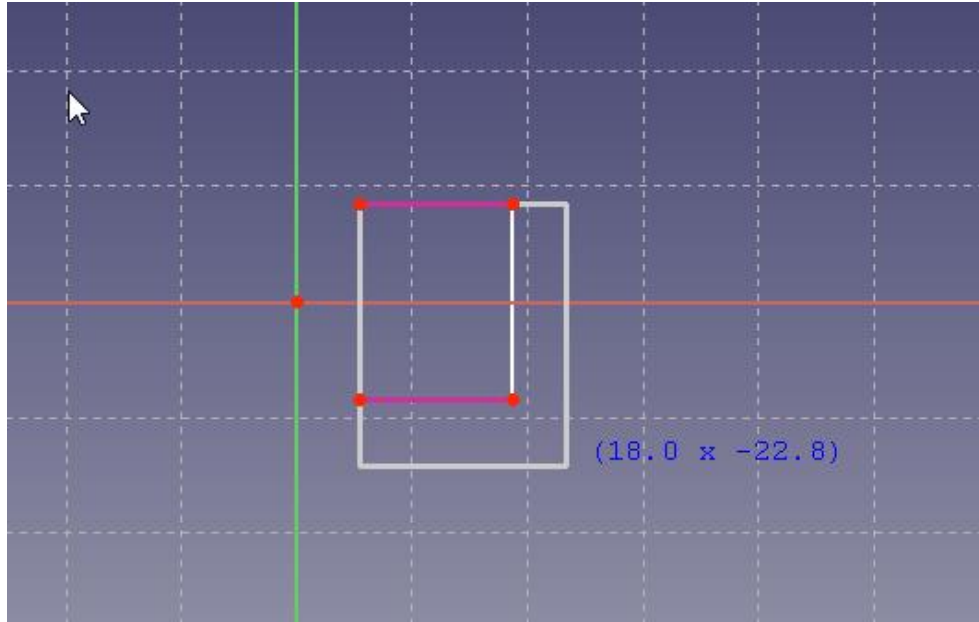


Figure 40 – Using the "Rectangle" tool

1. Select the top-left point of the new rectangle and the top-left point of the external geometry and merge them.
2. Click on the bottom-right point of the new rectangle and the bottom-right point of the external geometry and merge them.

As a result, we get a closed loop| contour (Figure 41):

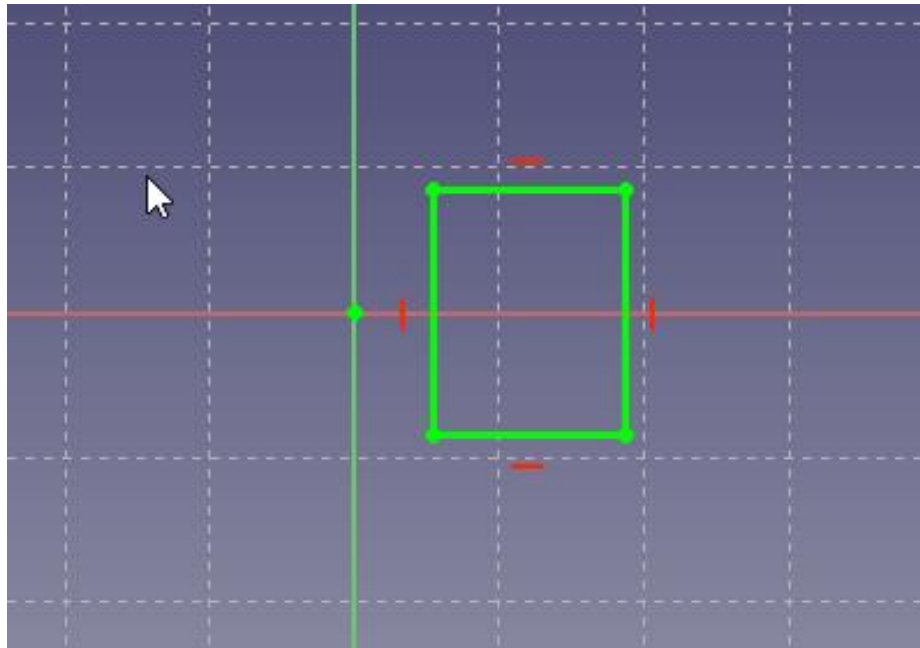


Figure 41 – Closed loop

Next, close the sketch using the close button or complete editing from the **Sketch** context menu, and then select the **“Pocket”** tool from the toolbar or from the **“Part Design”** context menu. In the drop-down menu of the **“Type”** field, select **“Through all”** and click OK.

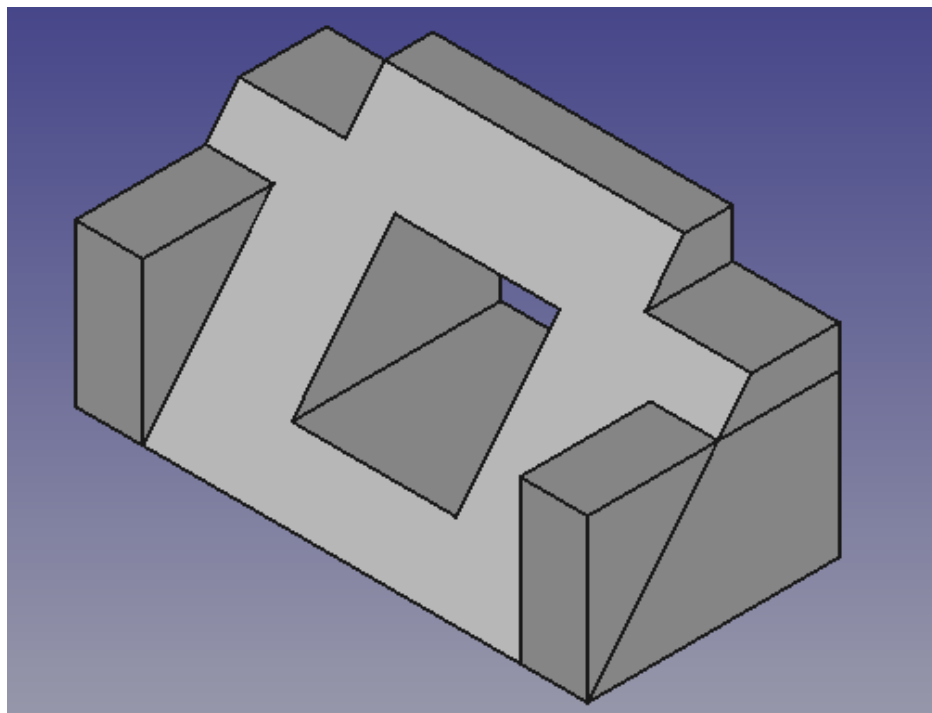


Figure 42 – Using the "Pocket" tool

At this point, you can see the lines that appear due to the crossing elements. The side block intersects with the base profile, forming a triangular block above the profile (these are the additional lines visible in the picture above on the right side of the model).

To remove these lines, you can turn on "**Shape Refinement**" in the **Part Design** preferences. Set the **Refine Property** to TRUE for the **Pocket** properties to do the cleanup:

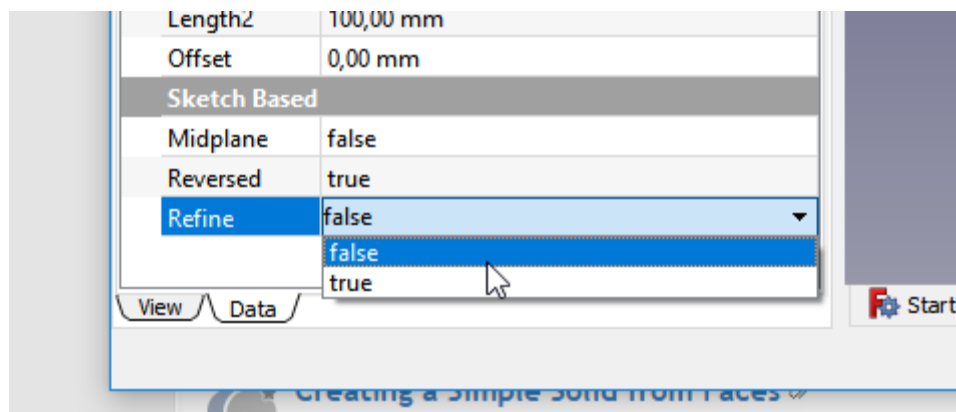


Figure 43 – Turning on the “Shape Refinement” of the part

To save time in the speed of processing the sketch of the part and save these lines for design, it is possible to individually include them for each function that creates such lines.

You can configure object property values on the Data tab in the Model combo panel menu.

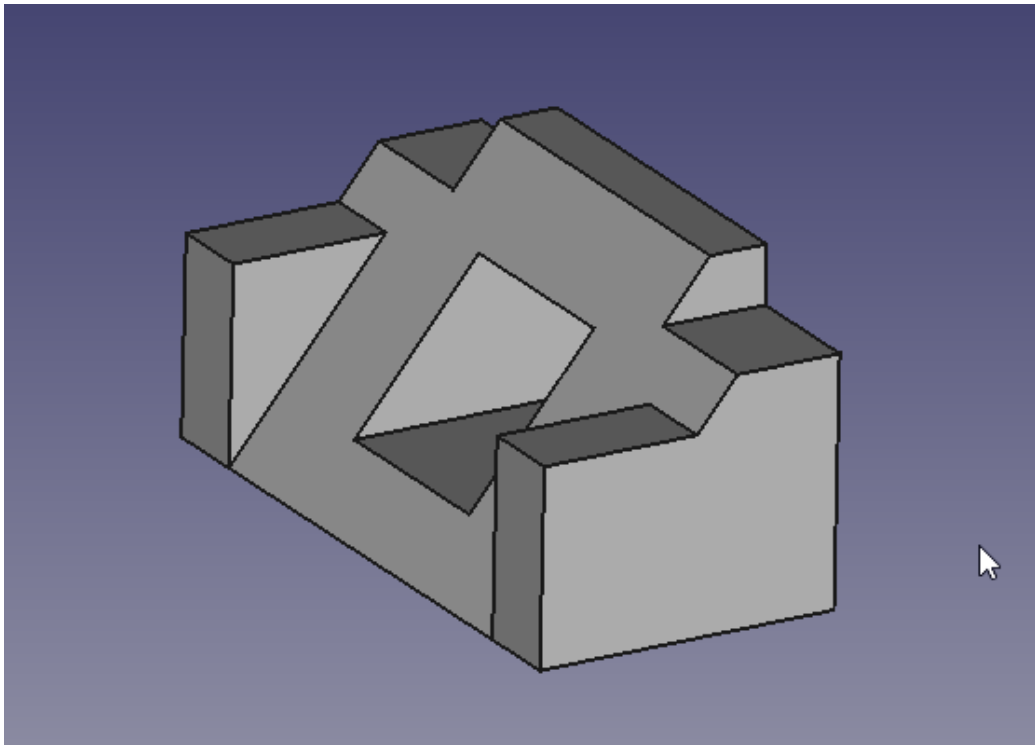


Figure 44 – The final model of the part

Self-study Task:

1. Create the final model of the part (Figure 44);
1. Create a drawing of typical part 2 (Figure 45)

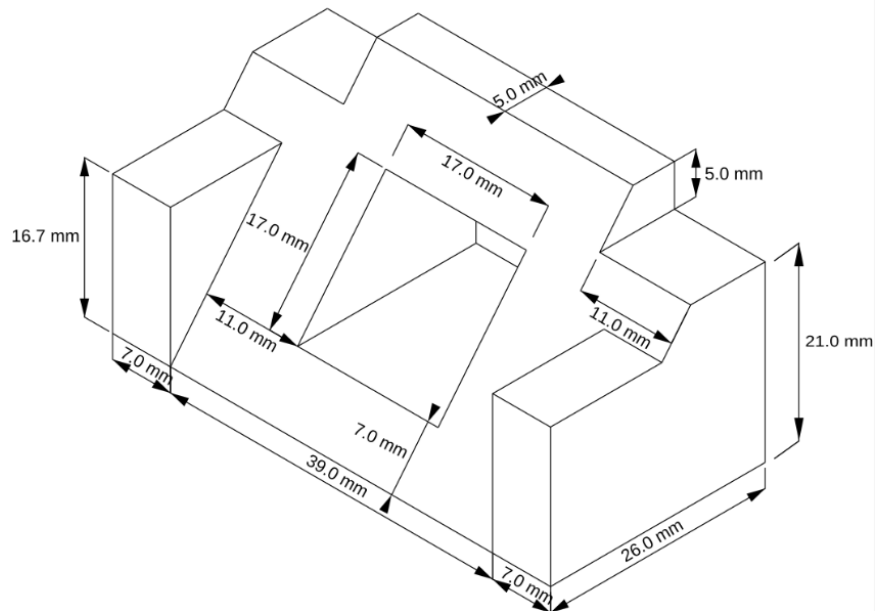


Figure 45 - Drawing of a typical part 2

Control questions:

1. What **Part Design** tools are used for design?
2. Does the **Pad** function apply to external geometry?
3. Does the **Pocket** function apply to the external geometry?

SECTION 4. WORK WITH 3D-OBJECTS

4.1. Toolbars and navigation

Let's start the editor and configure it to work. First, select the desktop:

View> Workspace> Part (Figure 46).

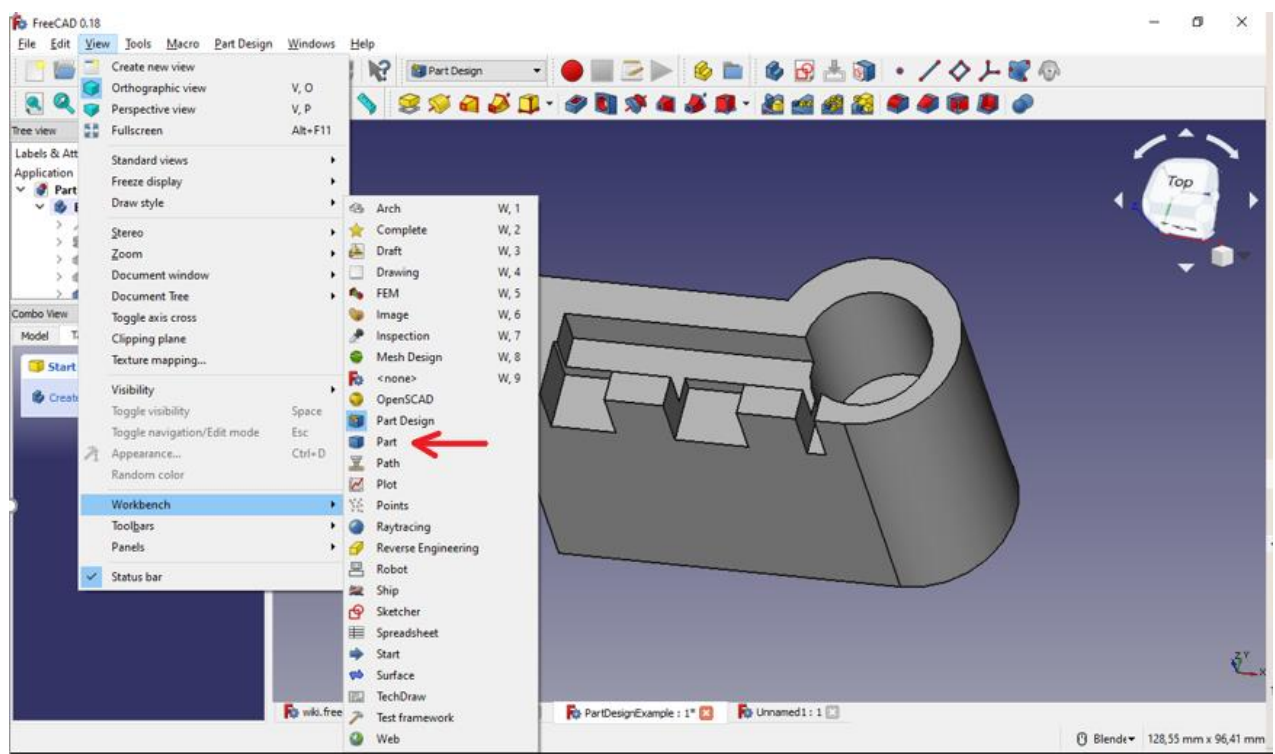


Figure 46 – Desktop setup

Let's create a new file, on the left we will see the windows of the document tree, the properties view and the **Combo View**. Close by clicking the mouse in the upper right corner of the document tree window and view of properties, leaving only the combo panel. Then click **File-> Save As**, save the project under

the name "Test".

The desktop is ready for use (Figure 47).

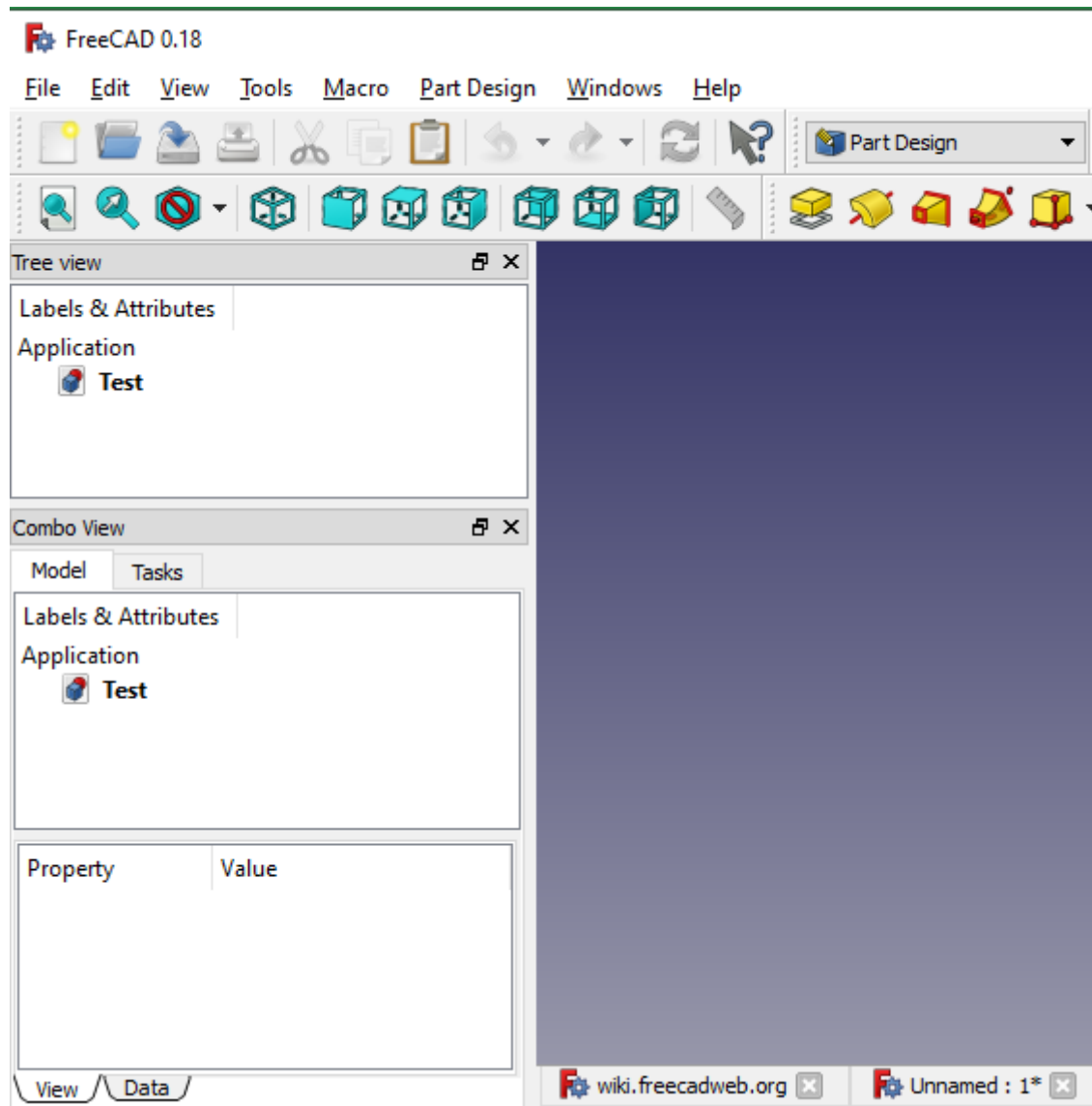


Figure 47 –New file creating

Now let's look at the tools we will use.

They are located at the top of the window.

The first group of tools is the standard views (Figure 48).

With their help, we will be able to switch between views and projections.



Figure 48 – Standard views

From left to right:

Axonometric View, Front View, Top View, and so on. You can also switch between standard views using the **Numpad** keys: 0 - axonometry, 1 - front view, 2 - top view, etc.

The second group of tools is the panel of basic shapes (primitives) (Figure 49).



Figure 49 – Basic shapes

As you can see, there are three-dimensional shapes: **Cube, Cylinder, Sphere, Cone, Torus**, as well as the button for creating primitives. With these tools, you can create the corresponding objects. Click on the "Cylinder" button (Figure 50).

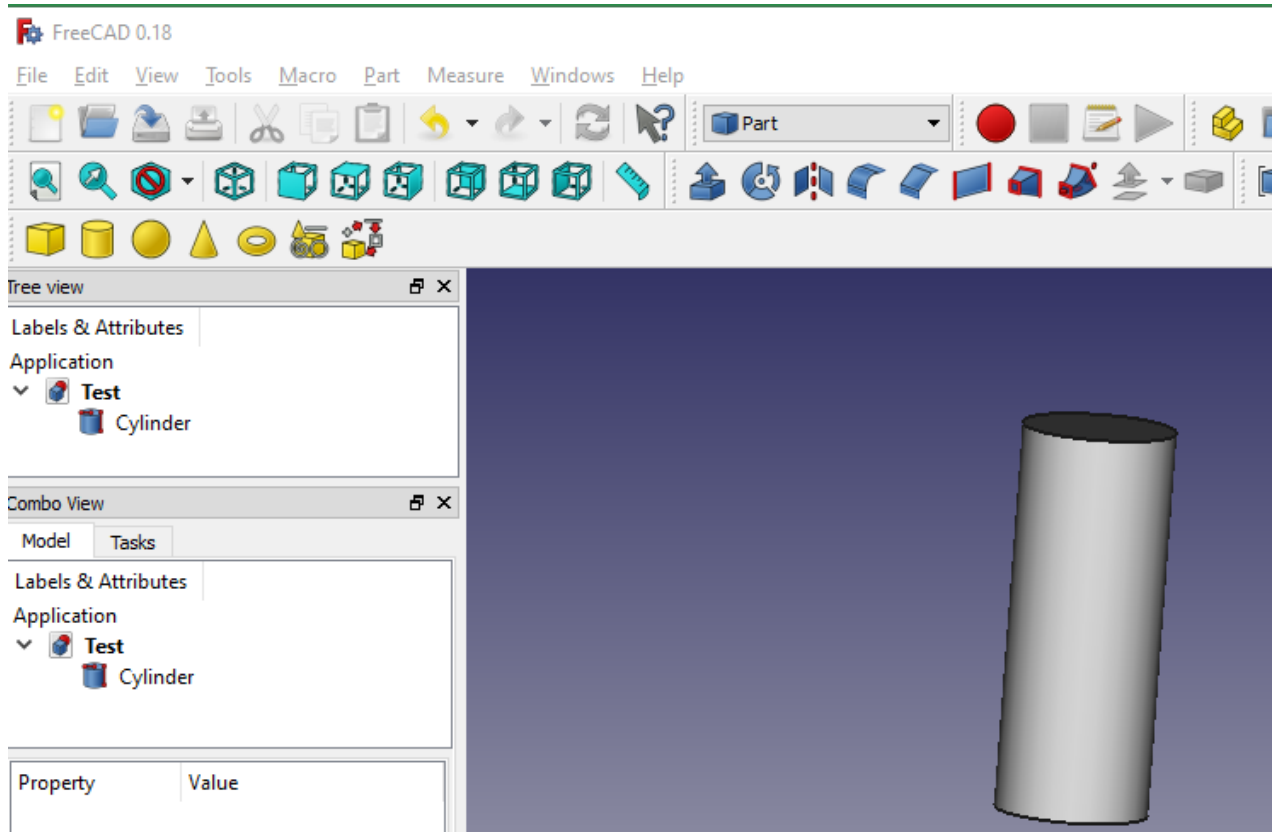


Figure 50 – Element **Cylinder**

A cylinder appears on the working area, and a tree appears in the "model" window on the left, at the top is the name of the model, which includes the cylinder.

Let's practice switching model views, top, front, return the axonometric view. Now let's look at how navigation is carried out using the mouse (Figure 51).

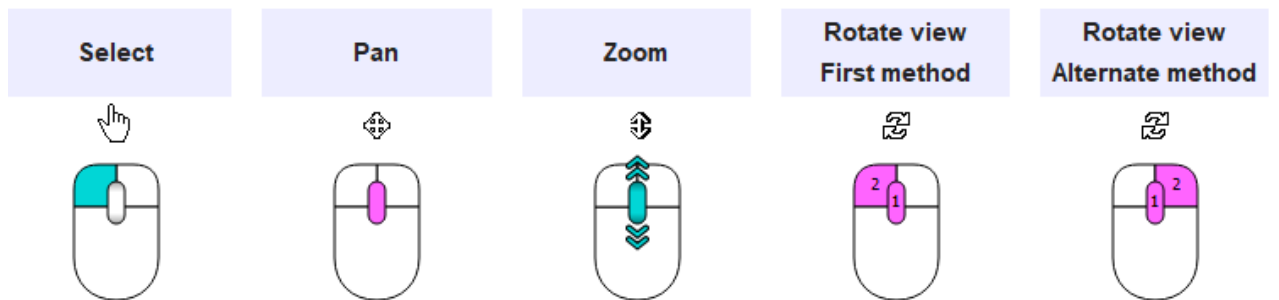


Figure 51 – Navigation with the mouse

Select - Press the left mouse button over an object you want to select. Holding down Ctrl allows the selection of multiple objects.

Pan - Hold the middle mouse button, then move the pointer.

Zoom - Use the mouse wheel to zoom in and out. Clicking the middle mouse button re-centers, the view on the location of the cursor.

Rotate view. First method - Hold the middle mouse button, then press and hold the left mouse button, then move the pointer. The cursor location when the middle mouse button is pressed determines the center of rotation. Rotation works like spinning a ball which rotates around its center. If the buttons are released before you stop the mouse motion, the view continues spinning, if this is enabled. A double click with the middle mouse button sets a new center of rotation.

Rotate view. Alternate method - Hold the middle mouse button, then press and hold the right mouse button, then move the pointer. With this method the middle mouse button may be released after the right mouse button is held pressed. Users who use the mouse with their right hand may find this method easier than the first method.

Let's practice selecting an object, rotating, moving, changing the scale. To return the object to the standard view, press the "**Axonometric View**" button.

To implement geometric primitives, use the "**Tasks**" function in the "**Create Sketch**" menu. As you can see, you can create several different shapes in a single object. For example, select a prism from the list and click "**Create Sketch**" and "**Close**" (Figure 52).

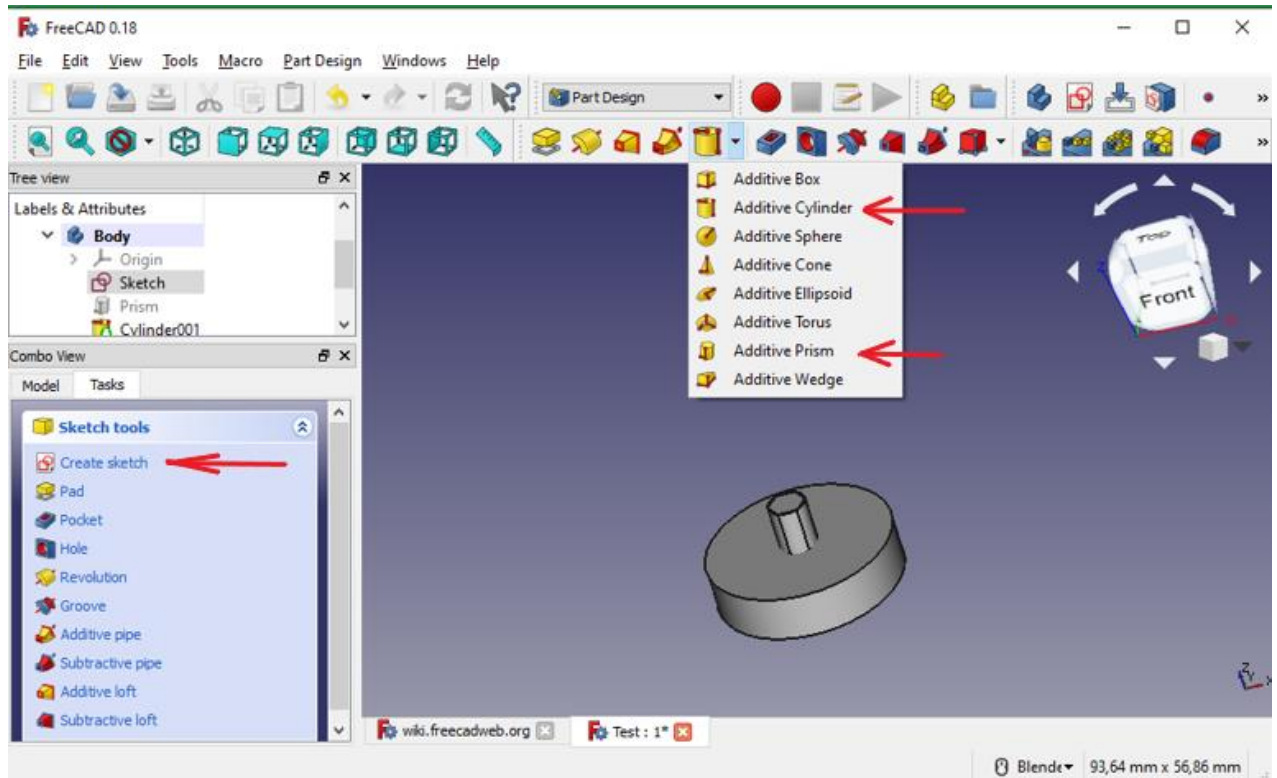


Figure 52 – Geometric primitives

As we can see, a prism appeared inside the cylinder. Now let's change the parameters of the objects. In the upper window of the **Combo** panel, select the Cylinder with the left mouse button, in the lower window, select the **Data** tab (Figure 53).

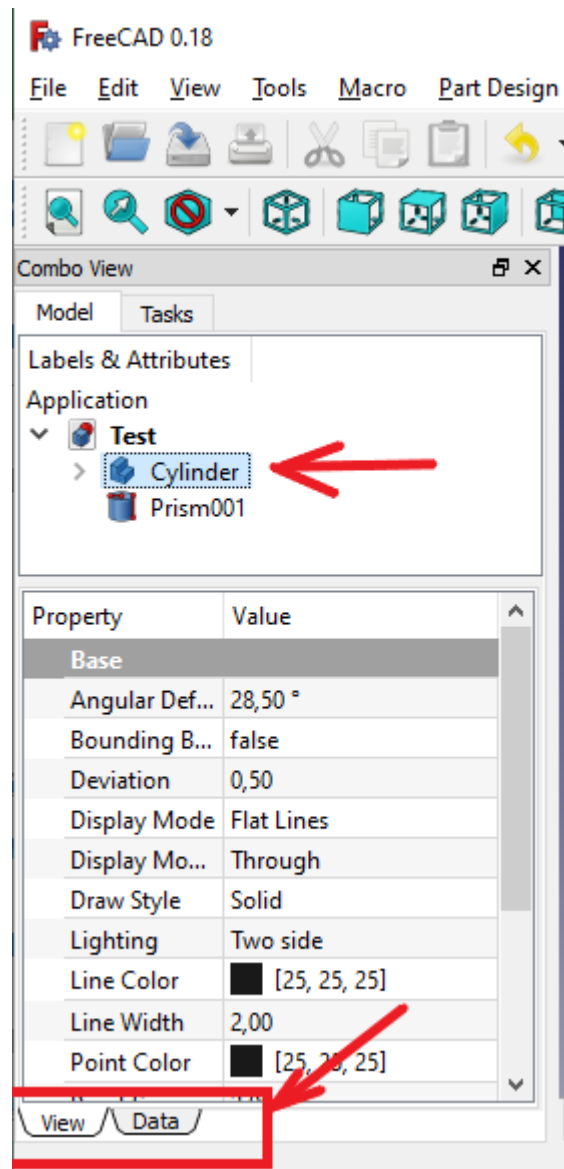


Figure 53 – The "Data" tab in the **Combo** panel

Now in the data window, left click on the value of the **Radius** parameter and type the value of the radius of the cylinder 4.5 mm on the keyboard. Left click on the value of the **Height** parameter and enter the value of 5.5 mm. As we can see, the dimensions of the cylinder on the working field have changed (Figure 54).

Now select the prism at the top of the **Combo** panel.

Important: it is necessary to remember to select in this window the object with which it is supposed to work because all parameters, the sizes, a location will change in a data window for the selected object.

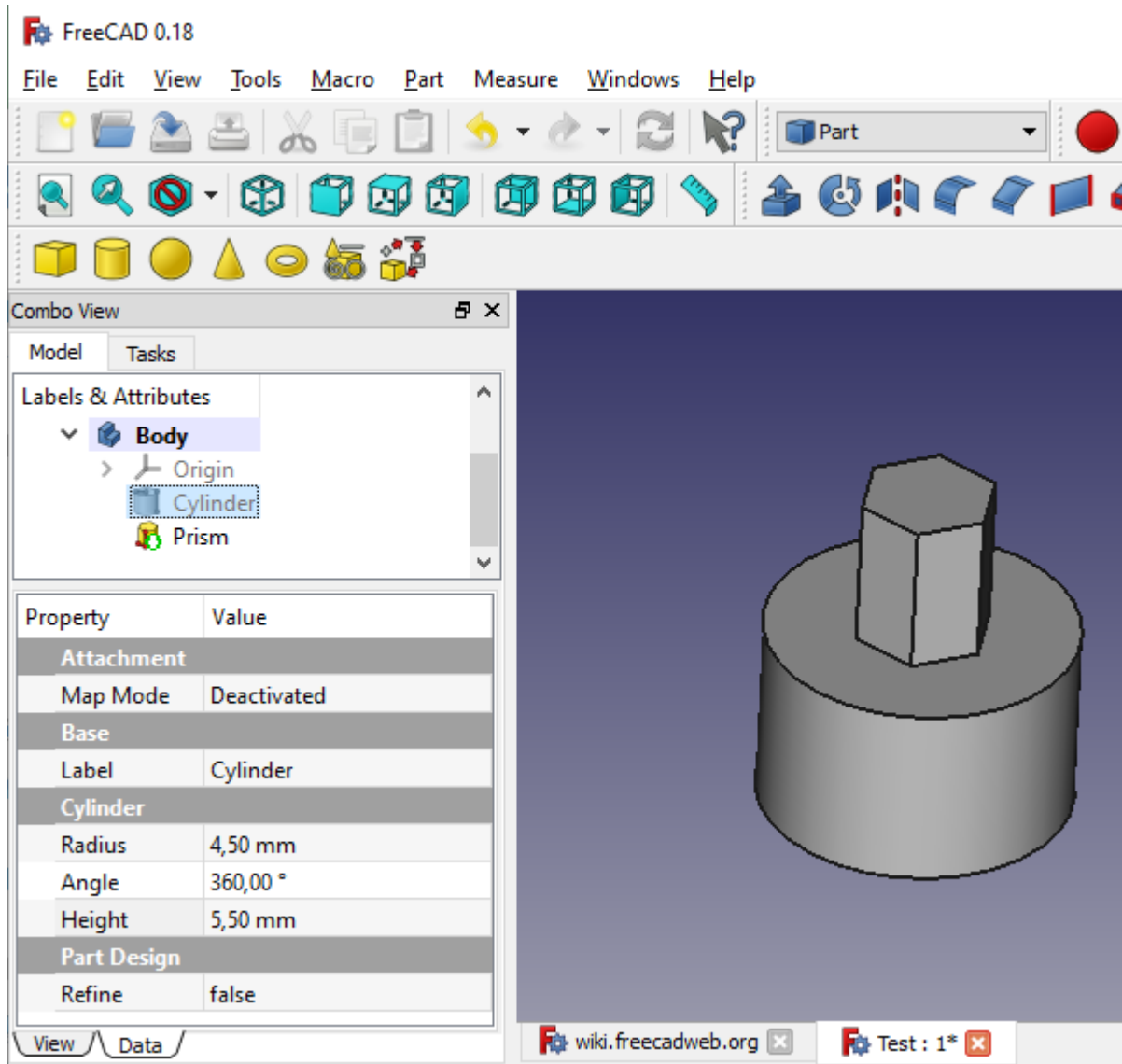


Figure 54 – Resize the Cylinder

For a prism, first change the number of faces. To do this, left click on the value of the **Polygon** parameter and enter the value **5**.

Next, change the position of the prism. To do this, at the bottom of the panel, left click on the value of **Placement** and click on the small button that appears on the right side of the window (Figure 55).

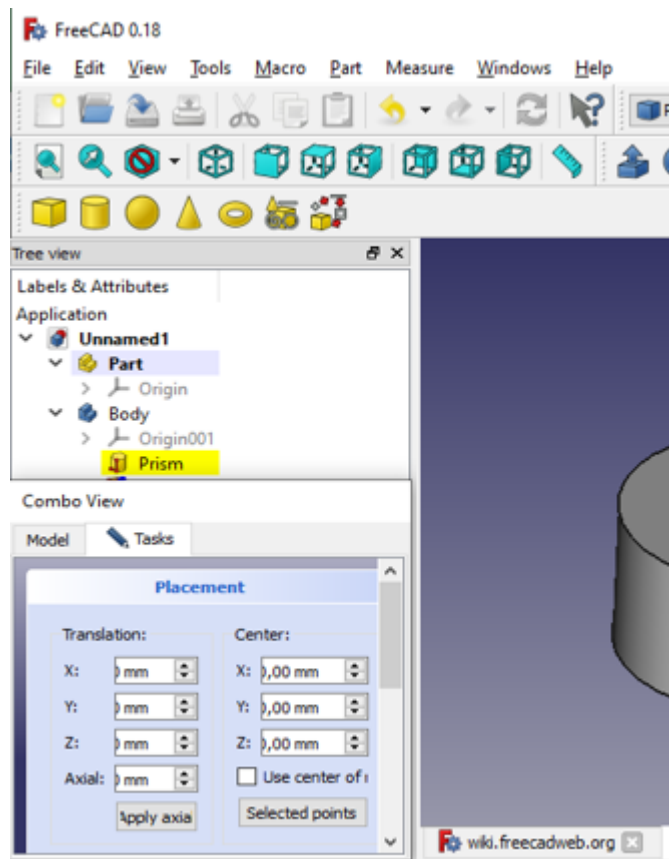


Figure 55 – Change the objects placement

The window for changing the position of the object has opened. In the **X** coordinate window, we enter **-2**, in the **Y** coordinate window we enter **-2.5**, which leads to the displacement of the prism along the **X** and **Y** axes by the corresponding number of millimeters.

Then select the **Y-axis** parameter from the Axis drop-down list, enter an angle of **17** degrees. Click on the "**Apply**" and "**Yes**" buttons (Figure 56).

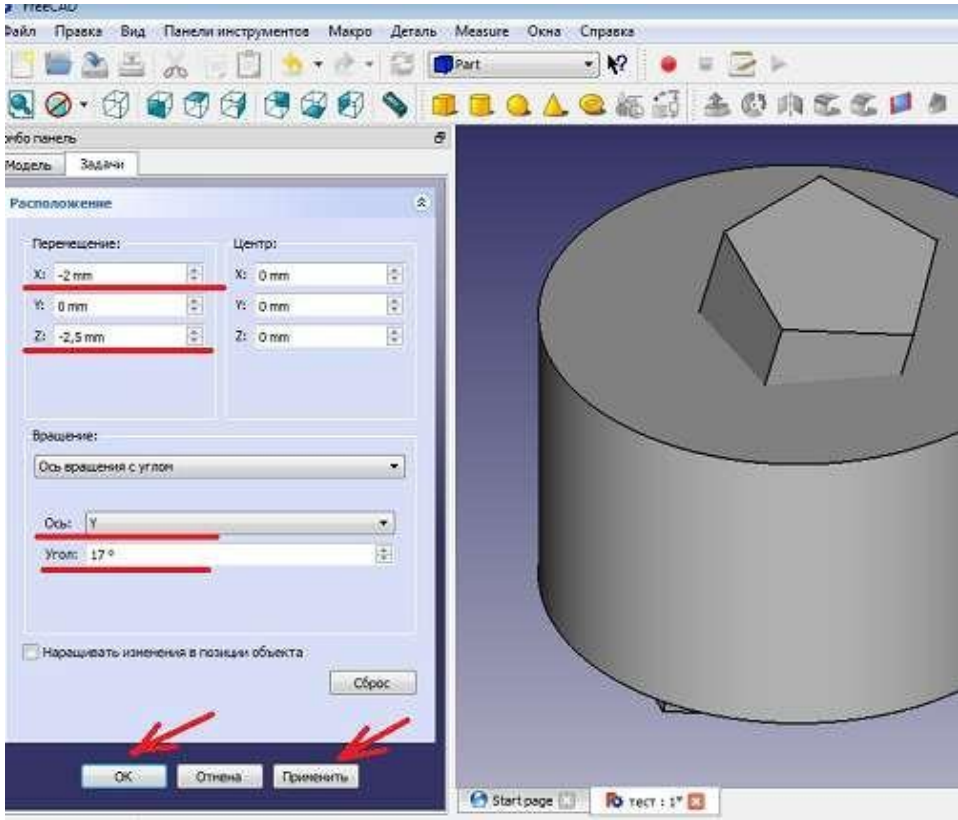



Figure 56 – Changing the object position

Finally, consider the third and final group of tools, of which we will use – a group of logical operations with objects (Figure 57).



Figure 57 – Logical operations with objects

From left to right: Boolean operations, Difference, Union, Intersection, etc. Let's look at the first 4 operations and start with **Union** . In the upper left window of the **Combo** panel, select both shapes, the prism and the cylinder, with the **shift** key pressed. Click on the “**Union**” button. As a result, we got one object – **Fusion** and the result of adding objects (Figure 58).

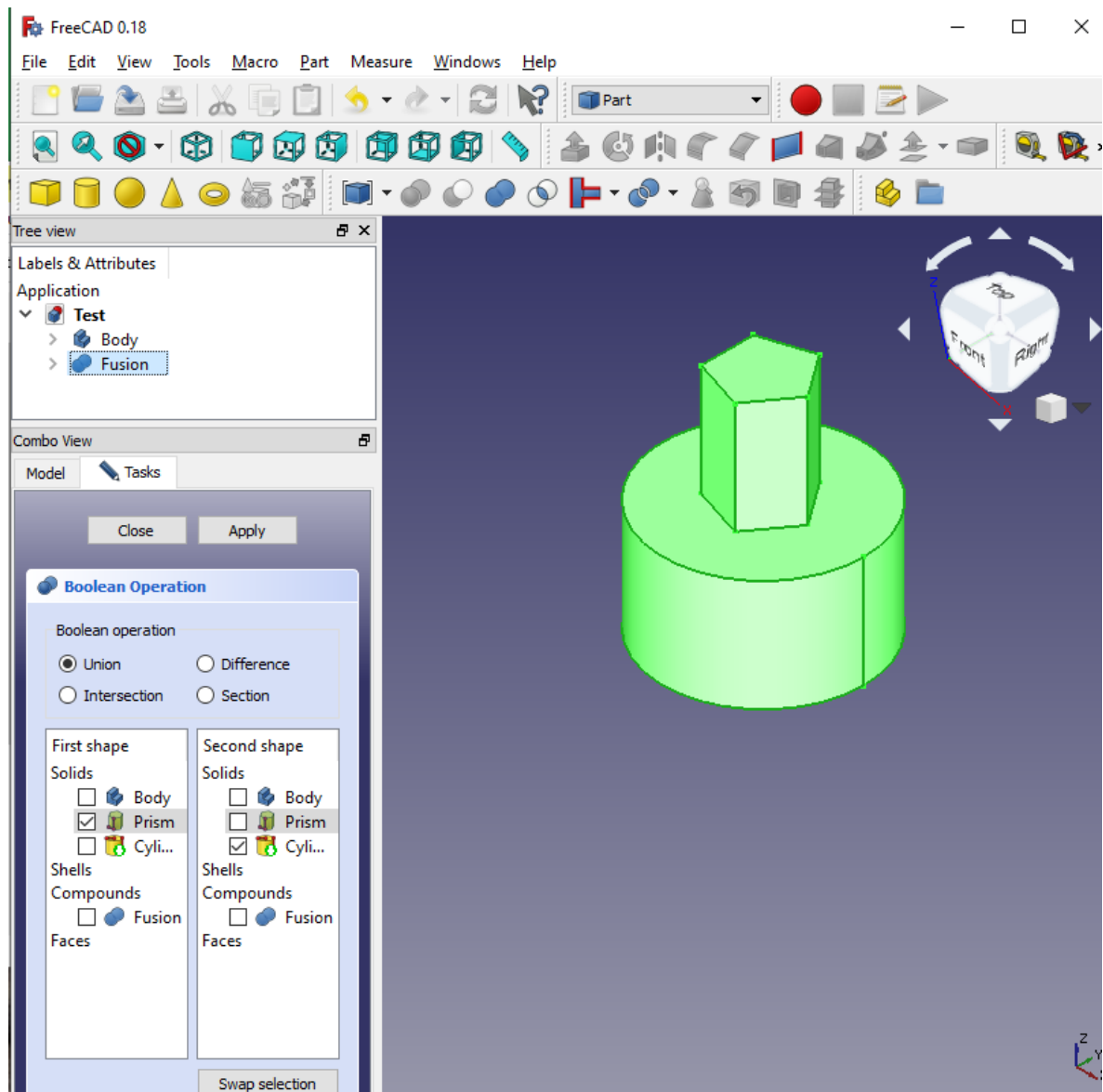


Figure 58 – Union objects

Now we will undo the last operation by pressing **Ctrl + Z**. The object disappeared from the screen. Figures **Prism** and **Cylinder** have appeared in the combo panel, but the labels are grayed out. They have become invisible. To restore visibility to the object, you need to select the shape in the combo panel and press the "**space**". Do this for both figures. To make an object invisible, you need to do the same - select and press the "**space**". Now, when the shapes become visible again, we will apply the operation "**Difference**". To do this, first select in the **Combo** panel the figure from which we will subtract - **Cylinder**, then hold down **Ctrl**, select the figure to be differenced -

Prism, click on the subtraction button ("cut two figures") (Figure 59).

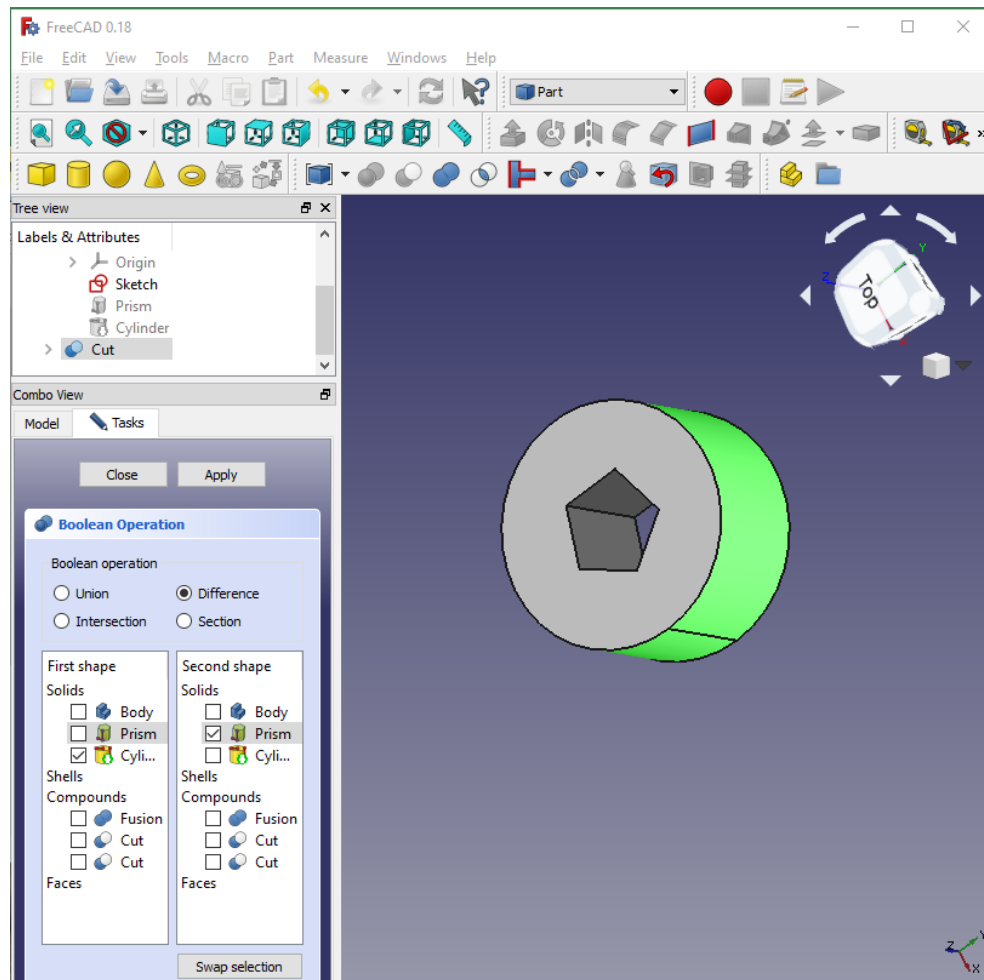


Figure 59 – Difference two shapes

Rotate the resulting detail with the mouse. As you can see in the combo panel window, the result of the **Difference** two shapes is the **cut** object. Now we will undo the operation again by pressing **Ctrl+Z**, make the shapes visible, and try the "**Intersection**" operation. To do this, select both shapes again and click on the "**Intersection of two shapes**" button. The result of the operation became the **Common** object. Admire the beveled prism by rotating it with the mouse. Undo the operation - **Ctrl+Z**, making both objects visible - select + space. Now let's look at the "**Boolean Operations**". Without selecting any objects, press the button "**Perform a boolean operation with two selected forms**". As you can see on the figure,

you can also perform various operations with this tool. For example, in the upper part we select the operation - the **difference**, below in the windows we select the first shape by putting a “**tick**” in front of the **prism**, in the second window we select the second shape - the **cylinder** (Figure 60).

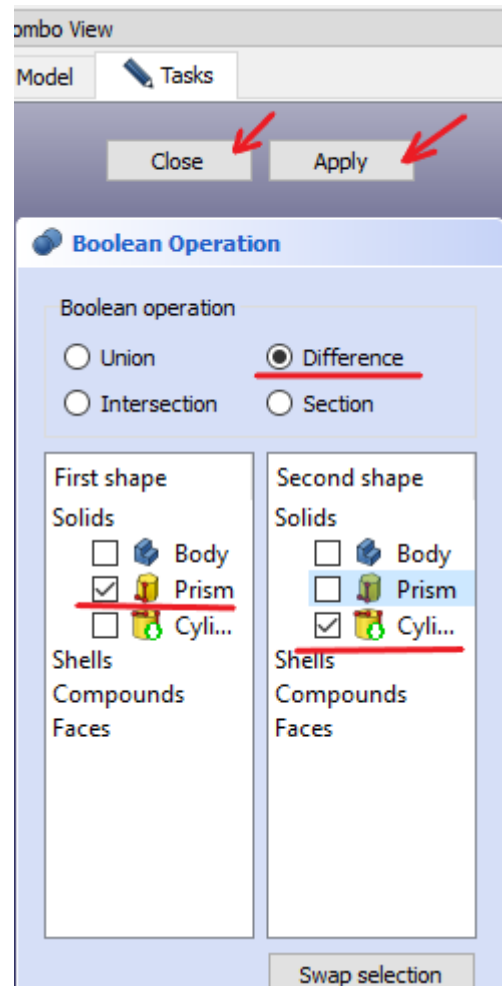


Figure 60 – Prism with cut

Click on the "**Apply**" and "**Close**" buttons. As a result, we got a prism from which the middle is cut - this is our **Cut** object. Now let's try deleting shapes. Left click on the **Cut** object, select "delete". The **Cut** object disappeared, and the **Prism** and **Pyramid** reappeared. Let's delete them too, save the project and close the program.

4.2. Basic cut function

Set the dimensions of the **Pocket** function found in the combo view:

1. Select the top plane by switching to the top view in the view menu and clicking on the visible face of your rectangle.
2. Create a new sketch of the part under Task in the Combo view (Figure 61).

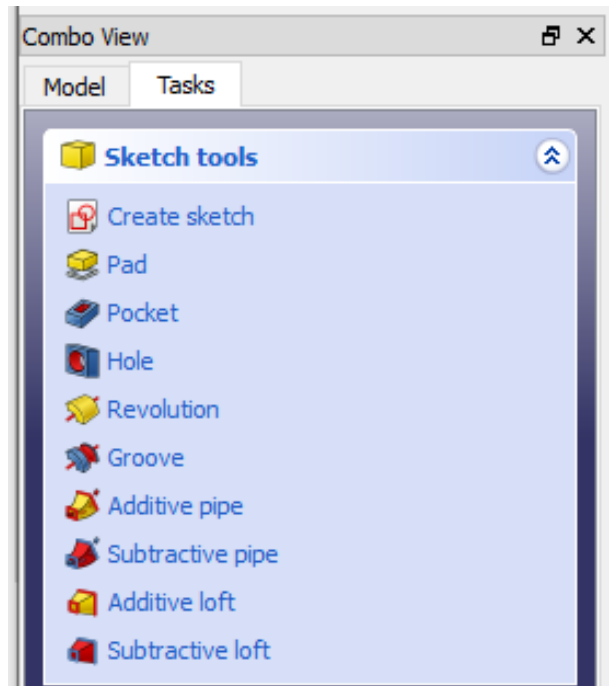


Figure 61 – Combo View “Tasks”

3. Set the grid to 5mm and select the Grid Snap. Now draw a rectangle starting at -35.0, 35.0 and make it big 70.0 x -70.0. It should offset 5mm from the edges of your box. Confirm your changes to the rectangle and close the sketch.

4. Click on the Pocket function just below the Pad function in the Combo View to continue. This function will cut everything inside your last drawn rectangle. You can change the direction and cut limit in the type selection.

5. Set Type to Size and Length to 55.0 mm for a 5 mm floor thickness.

Also rename the created pocket to "Hole" by pressing F2. When you switch to isometric view, you can see a hole in the box.

4.3. Offset

To shift the work plane using the Offset tool, you need to:

1. To hold the working plane of the future part, you need to create a rim around the top of the object. Switch to axonometric view and select the top plane of the window (don't select the base or sides of your box). Resize the grid to 5mm and turn on grid snapping. Now create a rectangle aligned to the top edges of your block, starting at -40, 40.0.

2. Close the sketch of the part when finished and rename the finished sketch to "Rim". To access the **Offset** tool, switch your desktop from **Part Design** to **Draft**.

3. Redefine the work plane by clicking **Set Work Plane** and then at the top plane of the window. The grid should become visible. To show or hide the grid, select **Draft > Utilities -> Toggle Grid**. The height of the work plane will determine the drawing level. Creating drawings at different levels will allow you to extrude the drawing into a specific object.

4. Select the "Rim" sketch in the **Model section** and the **Task** select the **Offset** tool. Select the **Copy** check box to leave "Rim" after the offset.

5. When using the **Offset** tool, it is very important to move the cursor and keep it in the modeling window. You will notice a rectangle following your cursor. Now don't move your mouse, enter a value of -2.5 and press enter. This will offset your sketch 2.5mm inside.

6. Turn off the **Draft grid** and return the work pane back to the part design.

4.4. Alternative method to trim or subtract from an object

To trim and subtract objects, you need to:

1. Double-click the "Rim" sketch to edit it. Set the grid to 2.5mm and check the grid snap.
2. Draw a second rectangle just above the "Draft" starting at (-37.5, 37.5) with a size of (70.70). Close the sketch. You can now also delete the "Draft" by clicking on it and press the delete button.
3. Select "Rim" again and trim it using the **Pocket** function. Set the type for **size** and length up to 5mm. This will reduce the space inside the rectangles.

Self-study Task:

1. Create a drawing of part 3 (Figure 62 a);
2. Create a sketch of part 3 (Figure 62 b):

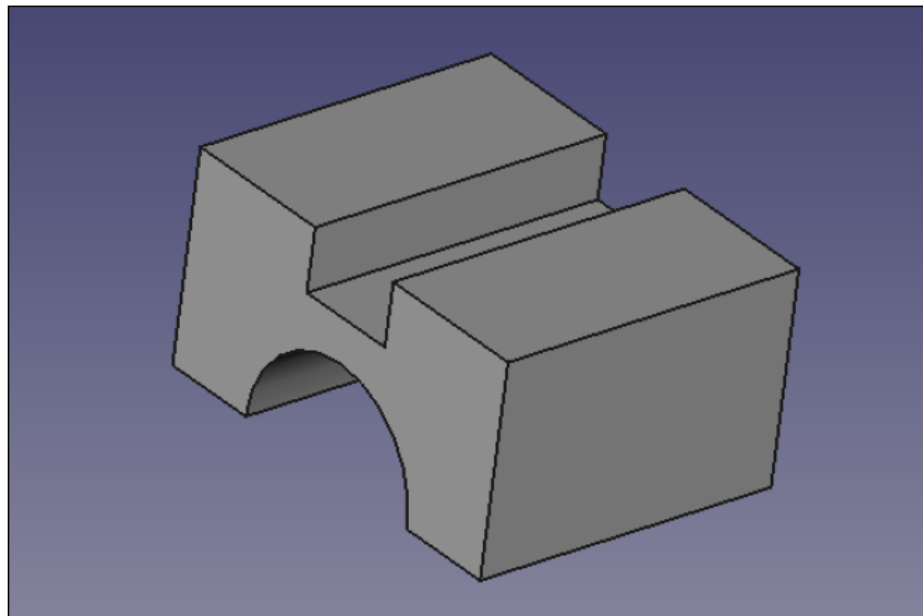


Figure 62 (a) – Sketch of part 3

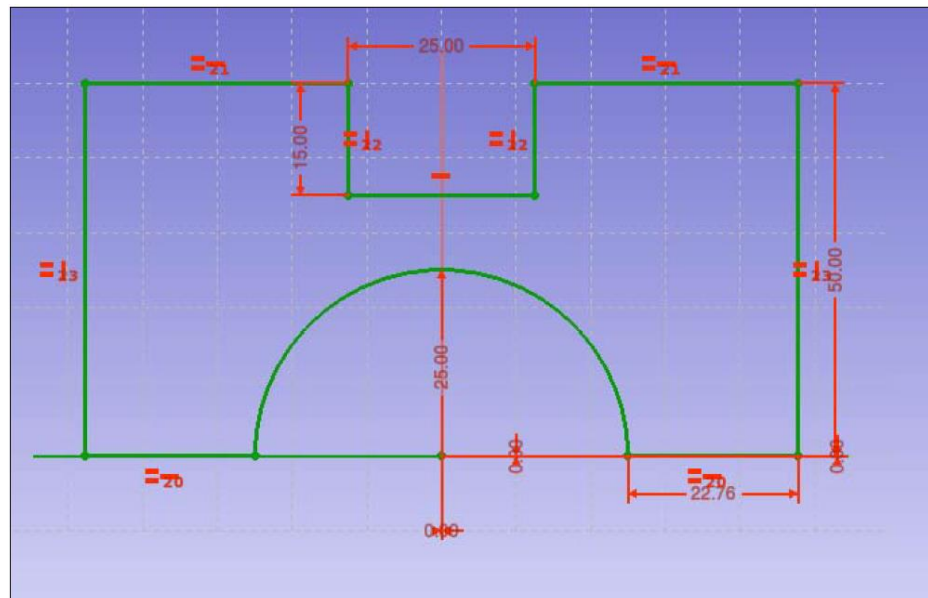


Figure 62 (6) – Part Drawing 3

Test questions:

1. List the design toolbars.
2. List the ways to use the mouse to design
3. What is the purpose of the Pocket function?

What is the purpose of the Offset function?

SECTION 5. FILLET AND PATTERNS


This tool creates a fillet for the selected face of an object. The dialog box allows you to select an object and faces to process.

5.1. Features of the Pattern

1. The **Pattern** function distributes the object equally. Therefore, before you can work with an object, you need to create it. In this step, you must add small additions to the bottom of the box. Switch your view to bottom view and select a visible plane. You should see the bottom of your box. Now create a new sketch on this plane and set the grid size to 2.5mm, also check the grid snapping.

Select **Create a Circle** and draw a circle in the upper right corner of the box with a value of -30.0, 30.0 with a radius of 5 mm. Now close the sketch.


2. Select the sketch drawn in step 1 and pull it out using the **Pad**. Set the length to **5mm** and uncheck **Symmetric to Plane** and **Reverse**. Click **OK** to finish and rename **Pad** to **Nub** by pressing **F2**.

3. Now press **Nub** again and select the **Polar Pattern**  function. Set the **Angle** to **360.00**, **Occurrences** to **4**, and uncheck the box in the **Reverse direction**. This tool creates a circular model of a part with a specific angle and occurrences. The angle determines the size of your drawing. For example, at a **360°** angle, objects will be positioned in a full circle, at a **90°** angle only within a quarter of a circle. With the reverse direction check, your drawing will rotate clockwise.

5.2. Fillet function

Create a fillet from circular overlays with a 2.4mm radius:

1. Switch to **bottom** view and select all **Nubs**. To select many parts, hold down the CTRL key and click on different parts. To deselect a part, simply click on it again.


2. Now select the fillet function . The adjustable radius determines the size of your fillet. The radius should be smaller than the object you are creating, so set the radius to 4.9mm. Close the function and rename the Fillet to " **Fill** "

5.3. Fillet the corners of a part

First, create a rectangle on the existing rim. Reduce this by 2.5mm and then select the outside edge of your box to create the fillet:

1. Switch to the top view and create a new sketch on the “Rim” that you created in paragraph 4.3 and paragraph 4.4.

2. Draw a rectangle on the right side starting at 35.0, 37.5 and make it large 2.5 x -75.0. Exit the sketch and go to the next step. Create a new pocket with a sketch **Start Part**, set the length to 2.5 mm and uncheck the **Reversed** box. Also rename pocket to “**SmallRim**”.

3. Using the tool "**Fillet the selected edges of a shape**"  in the workbench **Part** round off the edges of the created part. Select the outer edge of the “SmallRim” part and click on the **Fillet** element Set the fillet radius to 2.4 mm.

As a result, we get a fillet with a radius of 2.4 mm (Figure 63).

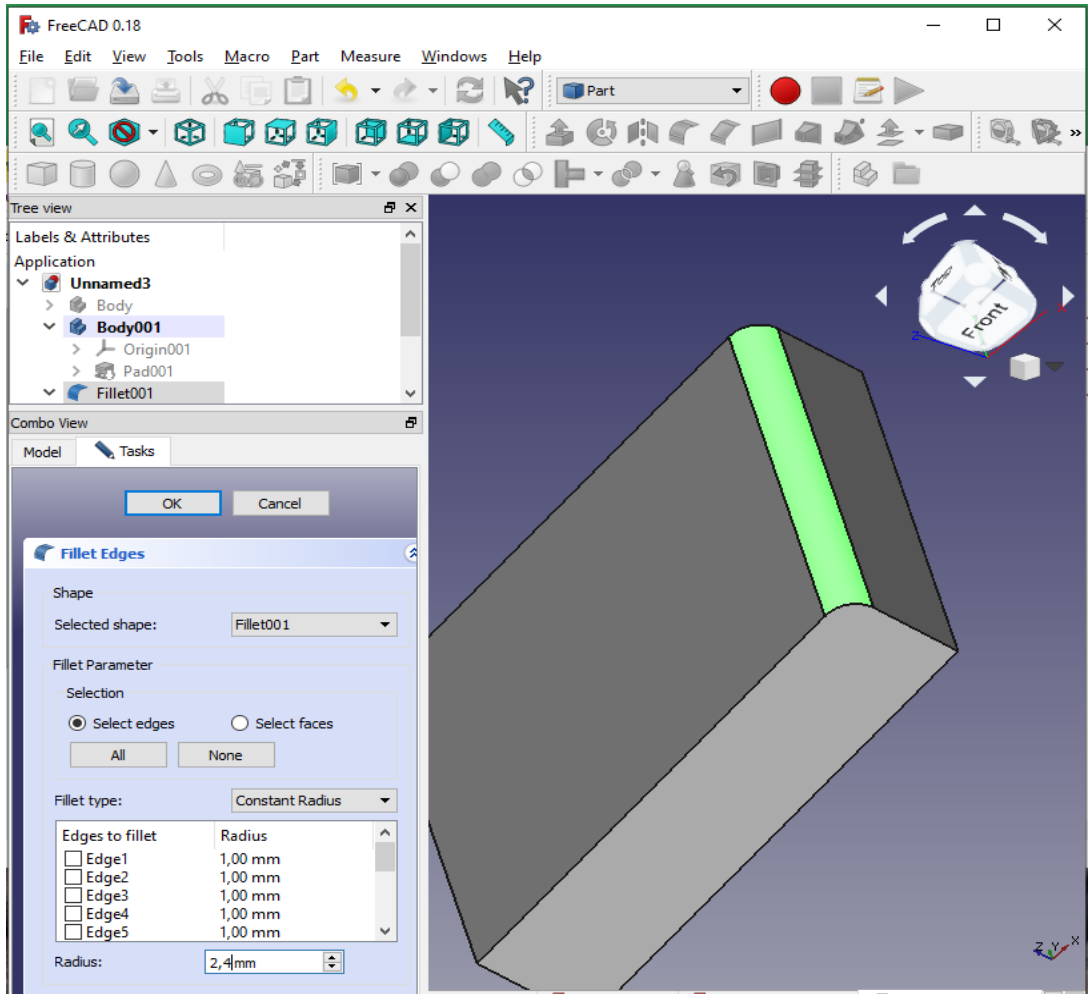


Figure 63 – Fillet the selected edges of a shape

Self-study Task:

1. Fillet the part in the necessary, in your opinion, places for part 4 (Figure 64):

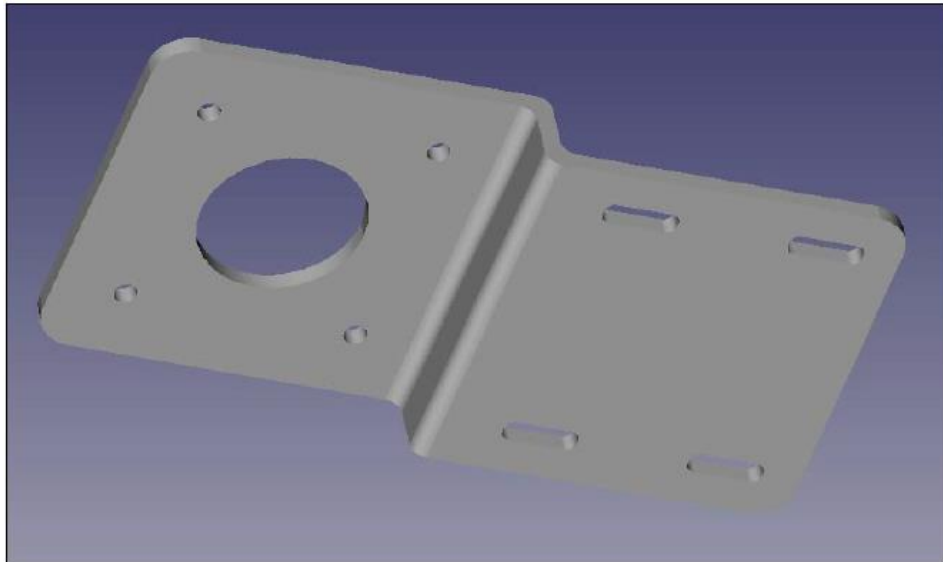


Figure 64 – Part 4 with fillet

Control questions:

1. List the design toolbars.
2. List ways to use the mouse for design
3. What is the purpose of the **Fillet** function?
4. What is the purpose of the **Offset** function?

SECTION 6. BENDING, EXTRUSION, CUTTING

Open a new document and save it as "Box Cover". Then switch your Workbox to Part Design.

6.1. Bending and Extruding

1. Create a new sketch on the **XY** plane, draw a rectangle starting at -40.00, 40.00 and make it 80.00 x -80.00. Rename the sketch to "**Base**". This will be a new, independent object.

2. Now extrude the sketch 5 mm high and rename it to "**Base**". When done, start a new sketch on top of the extruded box. Draw another rectangle starting at -40.00, 40.00 and make it 80.00 x -40.00big. Rename it to "**Base of Revolution**".

3. To create a **Revolution**, first click on "**Base of Revolution**" and then on the **Sketch feature**. Set the **Axis** to **Horizontal Sketch Axis** and the **Angle** to **180°**. Uncheck **Symmetric to Plane** and **Reversed**. A symmetrical plane will distribute the element evenly on each side equally on the plane.

4. Hide the **Revolution** you created in step, by clicking on it in the model view and pressing the spacebar. Now create a new sketch on top of the **Base** extruder. Set the grid to 2.5mm and draw a rectangle at (-37.50, 37.50) and make it large at 75.00 x -37.50. Create a new function **Revolution**. Set **Axis** to horizontal and angle to **180°**, also uncheck **Symmetric to Plane** and **Reversed**.

You can use the model of this object to create a drawing of a part, make the object larger, and reduce its placement during 3D printing.

6.2. Cut with two objects

1. Switch your **Workbox** to a **Part** and select **Run** in **Boolean operation**. Select **Difference** in the Boolean Operations window. It will take away **Second Shape** from **First Shape**. **Union** will combine two objects, **Intersection** will extract the common part of the two objects, and **Section** will extract the common part of the two objects as non-solid.

2. Choose **First Shape** for the first **Revolution**. This must be the biggest **revolution** of the two. **Second shape** will be smaller revolution. Click **Apply** to confirm the subtraction.

3. To finish, we create a new sketch at the bottom of the "**Base Extrude**". Draw a rectangle starting at -37.50, 37.50 and make it large at 75.50 x -75.50. Cut the sketch to 5 mm using the **Slice apart** function from the **Shape splitting** tools.

Self-study Task:

1. Create an extruded object - Part 4 (Figure 64).
2. Create an object with a cutout - Part 5 (Figure 65).

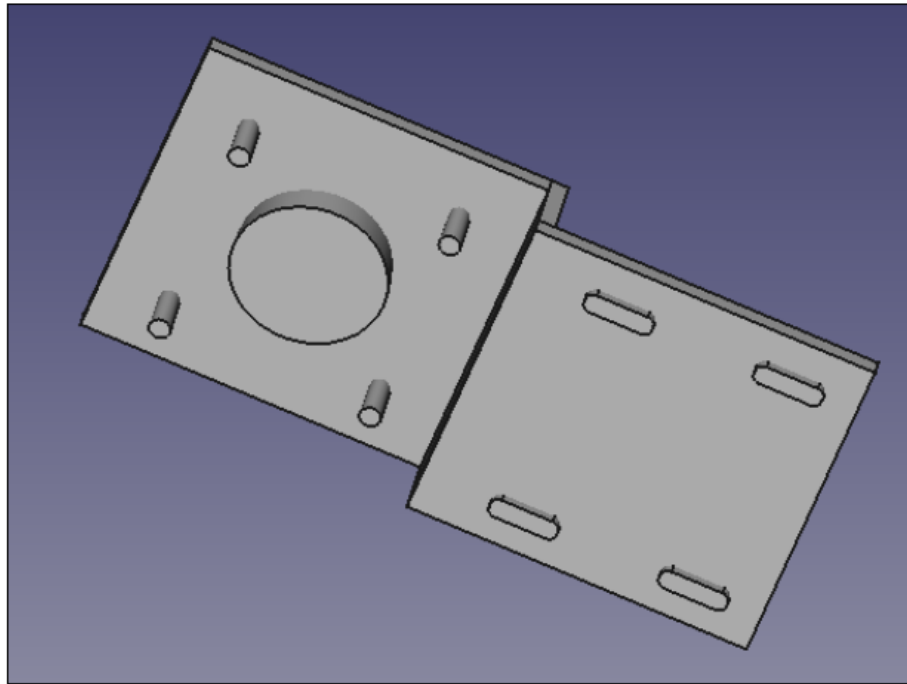


Figure 65 – Part 5 with an extrusion

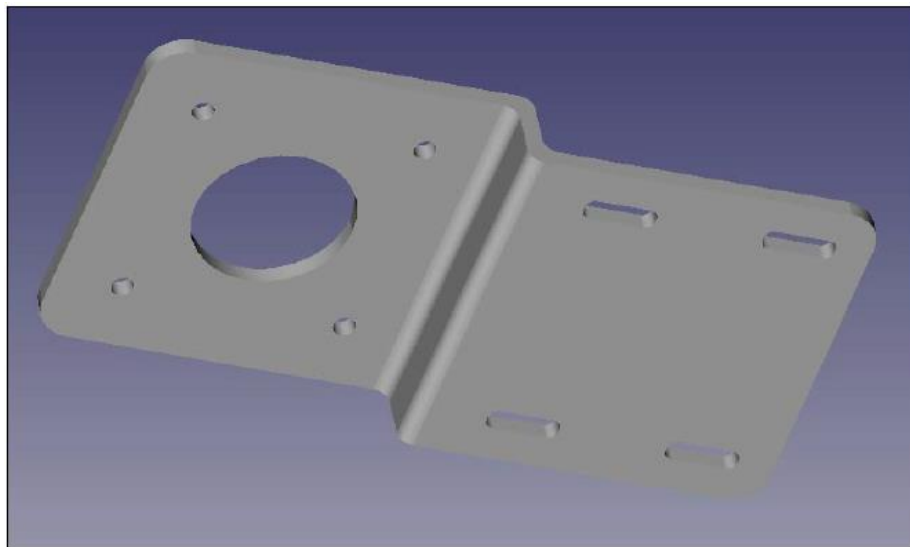


Figure 66 – Part 6 with cutout

Control questions:

1. What is the purpose of the Revolution function?
2. Why is a cutout with two objects made?
3. How to cutout with two objects?

SECTION 7. CREATING AND EDITING OF A MACRO

Macros are combined commands that are a series of commands or keystrokes and can perform multiple recursive tasks. You can use macro writing to automate tasks, which does not guarantee full-scale script programming.

7.1. Creating a macro

In FreeCAD, macros are a step-by-step representation of individual commands and are stored as a **Python** script with the * **.FCMacro** file extension.

Open the Macro toolbar or find the Macro menu item (Figure 67).

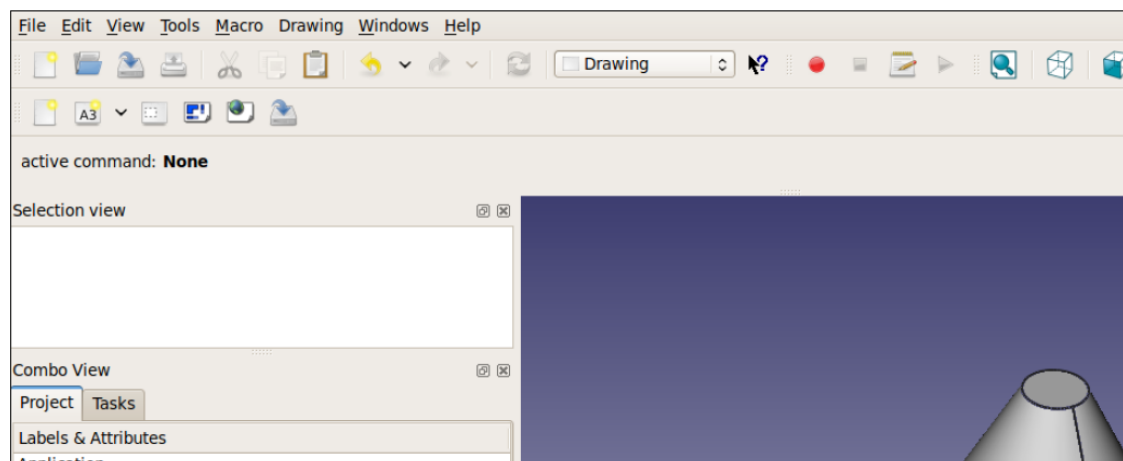


Figure 67 – Macro recording menu

To record a macro, follow these steps:

1. Press the red button to record the original macro:



2. The Macro Recording dialog box appears. Give the macro a name and save it:

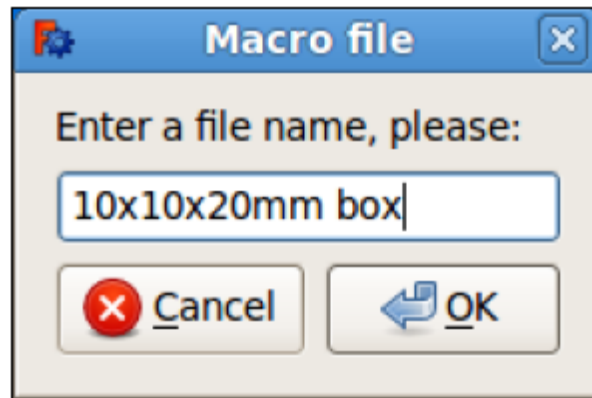


Figure 68 – Macro recording window

3. Write down the macro for the sketch of part 3 (Figure 62) and change its width to 20 mm.

4 When the task for the script is finished, press the green button to stop recording:



5. Execute the macro by pressing a button that looks like a keyboard thumbnail:



6. Select the name of the macro you created and click on **Execute** (Figure 69):

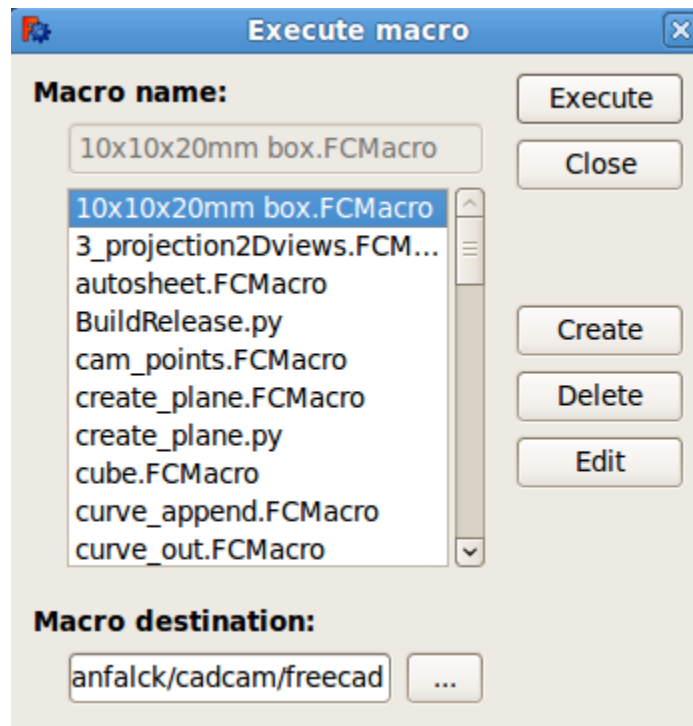


Figure 69 – Macro execution window

To change where macros are stored, open General Settings by clicking **Edit-> Preferences-> General**, and then clicking the **Macro** tab. From the **Macro recording settings** menu, select **Macro path** and specify the valid path to the directory of files with them, or click on the button to the right of the **Macro path** and correct it to the valid directory name.

7.2. Using and editing macros

Since macros are really a list of **Python** commands, you can edit them and create very complex scripts. You just need to record your actions on the object as you do them, and then save and replay them when necessary.

We could read and edit macro entries that contain **Python** instructions. To edit, open the created macro with the **Edit** button in the macro dialog. This feature opens our macro in the built-in FreeCAD macro editor.

The following is a Python macro code:

```
# Macro Begin: /home/freecad/10x10x20mm box.FCMacro ++
```

```

import FreeCAD
import Part

#Gui.activateWorkbench("PartWorkbench")
App.ActiveDocument.addObject("Part::Box","Box")
#App.ActiveDocument.recompute()
#Gui.SendMsgToActiveView("ViewFit")
FreeCAD.getDocument("Unnamed").getObject("Box").Width = 20.00

# Macro End: /home/freecad/10x10x20mm box.FCMacro ++

```

Entries beginning with # are not executed in FreeCAD; they are comments in Python. Comments help us understand what is happening in the code.

The executable lines of code that run are as follows:

```

import FreeCAD
import Part
App.ActiveDocument.addObject("Part::Box","Box")
FreeCAD.getDocument("Unnamed").getObject("Box").Width = 20.00

```

Operators that start with **import** open the basic modules that are needed to execute our macro. The **Import Part** feature allows us to create geometric objects such as boxes, cylinders, spheres, and many other 3D objects.

For example, the boxing feature looks like this:

```

App.ActiveDocument.addObject("Part::Box","Box")

```

You could edit macros to program what they didn't do originally. Open the macro in the built-in FreeCAD editor by clicking a button that looks like a text editing pad. Select the macro, and then click **Edit**.

You can also change parameter values for different objects. In our created macro, you can change the following two functions to get a different name and object length:

```
App.ActiveDocument.addObject("Part::Box","newbox")
```

```
FreeCAD.getDocument("Unnamed").getObject("newbox").Width=20.5
```

You can save changes after editing by selecting **File-> Save** from the top menu. You can then press the green play button to run the modified macro:



So, when performing tasks, each macro corresponds to Python commands that run in the background. The FreeCAD macro just captures this sequence of commands and saves them to a file that you can reuse.

Control questions:

1. Purpose of macros?
2. How is a macro created?
3. How is a macro edited and executed?

CONCLUSION

Thanks to the latest advances in both CAD and medical technology, CAD is increasingly used in medical applications and devices. In particular, CAD and 3D printing technologies are used in biomedical engineering, clinical medicine, individual medical implants, tissue engineering, dentistry, artificial joints and robotic surgery.

Medical technology is often complex, with many individual components interconnected to achieve a specific function. In addition, such machines must be designed and constructed very precisely, as design errors can lead to the failure of the functionality that these machines have to offer.

The fundamental advantages of CAD have made it an ideal solution for the revolution in the design of medical devices, such as microarrays, MRI machines and more. An experienced professional can create models of medical equipment that meet the specific measurements and requirements specified in the specifications.

By the time the product passes the modeling phase and is built, the result must be extremely accurate, as described in the initial specifications. Therefore, research and study in this area provides an opportunity to become a master in the field of accurate digital modeling, which creates medical machines that save lives.

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