elektor books

FreeCAD for Electronics Applications

Practical Introduction to 3D Modeling from Enclosure to Front Panel

Dr. Thomas Duden

elektor
design > share > earn

FreeCAD for Electronics Applications

Practical Introduction to 3D Modeling from Enclosure to Front Panel

Dr. Thomas Duden

elektor
design > share > earn

This is an Elektor Publication. Elektor is the media brand of
Elektor International Media B.V.
PO Box 11, NL-6114-ZG Susteren, The Netherlands
Phone: +31 46 4389444

Declaration

The author, editor, and publisher have used their best efforts in ensuring the correctness of the information contained in this book. They do not assume, and hereby disclaim, any liability to any party for any loss or damage caused by errors or omissions in this book, whether such errors or omissions result from negligence, accident or any other cause. All the programs given in the book are Copyright of the Author and Elektor International Media. These programs may only be used for educational purposes. Written permission from the Author or Elektor must be obtained before any of these programs can be used for commercial purposes.

British Library Cataloguing in Publication Data
A catalogue record for this book is available from the British Library

ISBN 978-3-89576-574-2 Print
ISBN 978-3-89576-575-9 eBook

Editors: Alina Neacsu; Jan Buiting MA
Prepress Production: D-Vision, Julian van den Berg
Print: Ipskamp Printing, Enschede (NL)

Contents

Chapter 1 • Introduction

In the world of modern mechanical design, 3D tools have become an indispensable resource. They allow the interactive modeling of instrument cases and compartments, control elements, as well as the complex arrangements of all bits and pieces that form a complete apparatus. During the design process, the packing density of components can be maximized, which saves volume. Alongside this process, it can be verified that no collisions between the different parts occur. This saves time and disappointment, when the assembly is completed later on, meeting with the real world.

For the longest time, 3D design tools have been available only to commercial businesses, which were able to spend 5-digit amounts for acquisition and maintenance. For students, academic packages exist at more affordable price tags, as well as floating campus licenses. In contrast, private users and hobbyists did not enjoy easy access to these tools.

Especially during the last decade, suppliers of circuit boards opened business to private customers, and the availability of 3D printers at affordable prices dramatically increased. Bundled together with interesting software, this propelled a wide distribution of 3D printing technology. With these advances, the community developed two remarkable software projects, which have meanwhile matured into a highly useful state: KiCad for circuit boards and FreeCAD for mechanical design work. KiCad allows, besides the design of the schematic and the circuit board, also the export of step models — an exciting interface to the mechanical world, which can be used to introduce the 3D circuit board models directly into mechanical designs.

This book is intended to deliver a useful take-off angle for technologists who were mainly developing electronics before and want to extend their designs towards a mechanical framework. It can by no means be a complete description of the FreeCAD universe, which has countless facets and extensions. Also, with a complex tool like FreeCAD, there usually exist many ways to achieve a result. With that in mind, the special pathways to the examples in this book are given for the reader's reference and experimentation, and to ease the start of their learning process with the software.

Usually, integrating a circuit board into a compartment, or a little chassis, only requires a minor subset of FreeCAD's possibilities. Therefore, the focus will be cast onto some small, but typical projects: The design of a front panel, or the assembly of a simple apparatus with front- and back panels, chassis, and cover. Once these design processes are mastered, the extension to more complex applications is straightforward. Following the described pathways should allow the successful assembly of the projects later on, with hopefully not too many surprises.

All examples, also the circuit board design, are electronically available in the supplemented materials online. Fully appropriate in the spirit of FreeCAD they can be modified, freely shared, and used.

In this book, as one example, the assembly of a small lab safety transformer is described. Here is an important statement concerning the use of electricity:

According to VDE code, voltages exceeding 50 VAC or 120 VDC constitute life-threatening hazards. Only trained and certified electricians are allowed to do the electrical work should a project be put forward towards practical implementation.

Chapter 2 • Starting Out With FreeCAD

FreeCAD is available nowadays in the package management systems and repositories of some Linux distributions. In this book, FreeCAD 0.20 will be used on a Tumbleweed platform from openSUSE. It is, however, also possible to install FreeCAD on the Microsoft Windows® operating system.

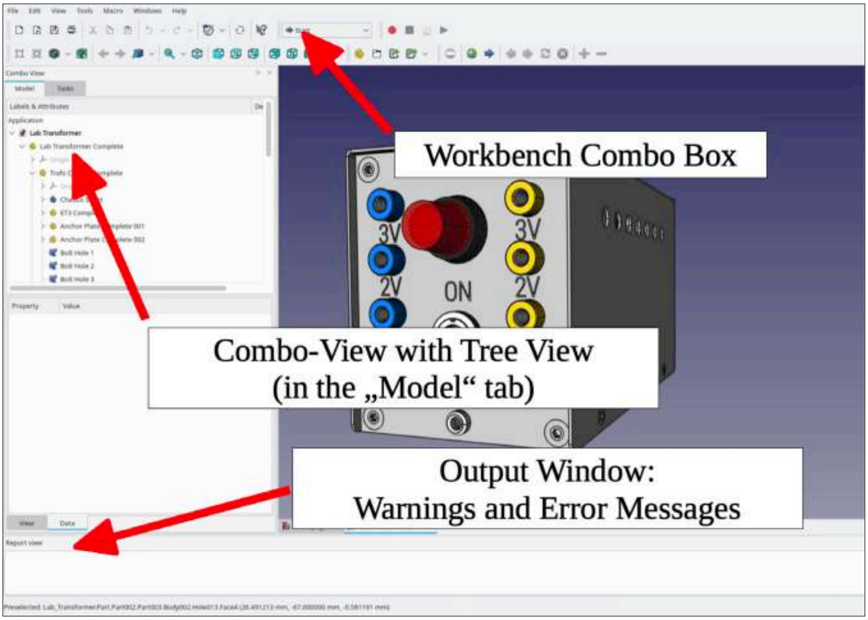


Figure 2-1

The functionality of FreeCAD is grouped into different workbenches. The selection of workbenches is contained in the Combo Box (Figure 2-1). The most frequently used workbenches in this book are Part Design to create 3D Objects, and Part, mainly for the attachment of 3D objects during an assembly. Sometimes, Draft will be utilized to generate geometry and character contours for engraving, and TechDraw for the extraction of drawings from the models, as well as for the DXF export of data to the milling machine. For sheet metal parts, the add-on called Sheet Metal by Shai Seeger will be used, and the placement of fasteners is exercised with the extension called BoltsFC.

2.1 Extending the Functionality of FreeCAD

We will start off with the installation of the extensions that will be used in this book. The way of installing the extensions depends on the form in which they are available (e.g., macro or workbench). Most conveniently, add-ons are installed with the integrated AddOn-Manager (Figure 2-2).

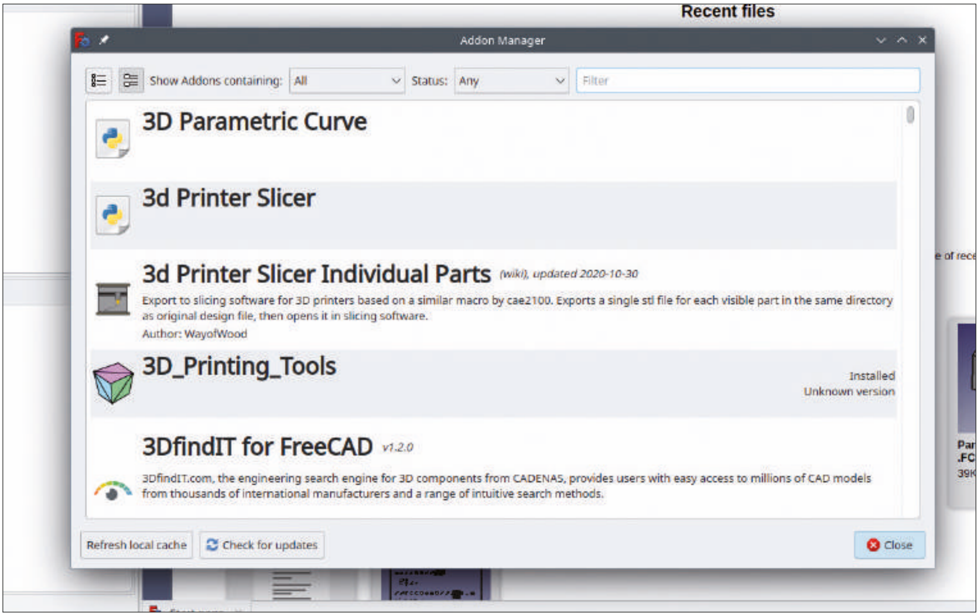


Figure 2-2

2.1.1 AddOns

In order to operate the AddOn Manager, a connection to the internet is required. The AddOn Manager is started by selecting Tools | AddOn Manager from the main menu. A warning appears, which states that the offered extensions are not checked or approved by the FreeCAD developer team. After acknowledging the warning with OK, a world of interesting additional functionality opens which is made available by the user community at no charge. This may be a good moment to reflect on reciprocating action to these impressive gifts later.

2.1.2 Workbenches

For working with sheet metal, we select the Sheet Metal workbench from Shai Seeger from the list. When the list entry is clicked, a partly animated description of the new functionality is presented. Take some time and scroll down the window to enjoy the collected information and source citations for further reading. Clicking the Install button at the top right of the AddOn Manager installs the workbench, very similar to the app stores on a smartphone. A convenient luxury! The workbench is then listed in the Combo Box of the main window (Figure 2-1).

2.1.3 Macros

Whereas the installation of the Sheet Metal workbench went quasi fully automated, the next extension requires a small intervention, before it can be used. The additional step is also described in the online documentation for BoltsFC, which is displayed in the AddOn Manager window.

If not already done, please click the Back button in the AddOn Manager (top left, the arrow pointing to the left side), in order to return to the main selection page. Then select BoltsFC and click the Install button again.

To render BoltsFC usable, the downloaded macro has to be manually installed. In order to retrieve the file, activate the display of hidden files in your file manager. Then, navigate to '\home\username\.local\share\FreeCAD\Mod\BOLTSFC' and look for the file 'start_bolts.FCMacro'. Copy this file into the location:
\home\username\.local\share\FreeCAD\Macro\

Unfortunately, with Windows© systems, the installation of BoltsFC is not as easy. As an alternative, consider installing the Fasteners workbench and use that instead. [FRE 2023].

2.1.4. Starting the new Extensions

Whereas workbenches, like the previously installed Sheet Metal workbench, are launched by selecting the corresponding entry from the Combo Box of the main window (Figure 2-1), the BoltsFC macro is executed by selecting Macro from the list in the main menu entry Macro | Recent Macros.

2.2 Workbenches and Context Menus

The amount of functionality present in FreeCAD necessitates the splitting into various workbenches. In each selected workbench, only certain menu entries and tool buttons will be visible — FreeCAD would be difficult to handle and probably rendered unusable if this would not be the case. Because of this separation of functionality, it is important to remember the current workbench selection. Sometimes, a function or menu button cannot be found. If this is the case, check the workbench selection as that may reveal the cause. Some of the workbenches are not compatible with each other. Luckily, it is only rarely necessary to swap between them.

The workbench concept can be frustrating when you start out using FreeCAD. However, this is just a matter of exercise and will quickly get easier after you have used the program for some time.

2.3 Automatic Workbench Switchover

Only some workbenches are automatically selected. For example, if a blue 'body' icon is double-clicked in the tree view of the model. FreeCAD will then select the Part Design workbench.

The same is true for the Sketcher workbench, when you click the Sketcher tool button (in Part Design), or you double-click a sketch in the tree view. These switchovers are invoked by the selection of corresponding tasks and follow the user entry in a straightforward way, so there is little probability for confusion with missing tool buttons or the changed context menu.

2.4 The Tree View

On the left-hand side of Combo View (Figure 2-3), when you select the tab Model, the project hierarchy is depicted in the shape of a tree view. Std-Part-Containers are shown in yellow, and the bodies of 3D geometry are shown in blue.

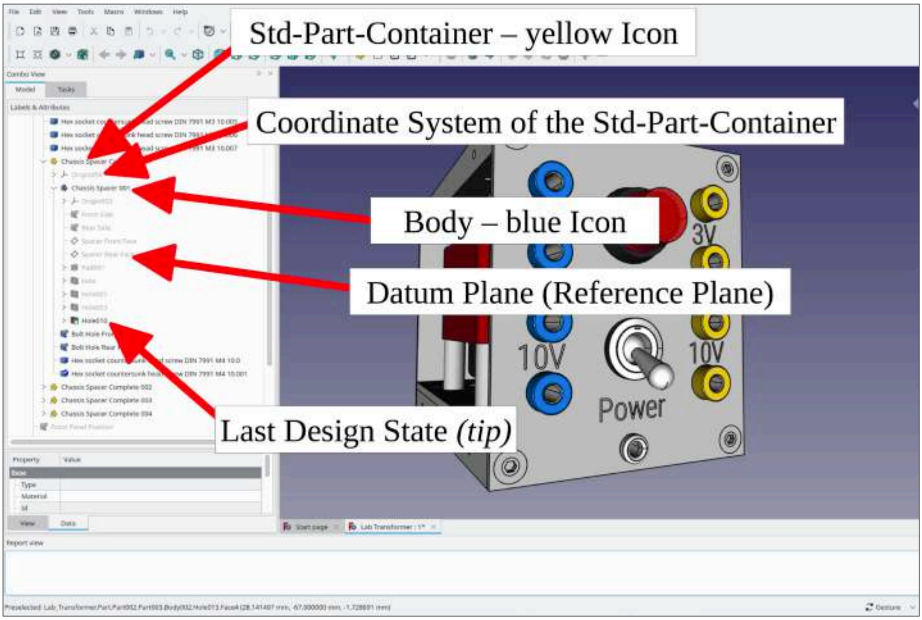


Figure 2-3

In the tree view, each object can be expanded to inspect the components it contains. If a body object is expanded, all invisible entries of its structure list appear grayed out (Figure 2-3, Tree View). It is possible to display or hide every object in the tree view by marking the listed entry and pressing the SPACE key. The reaction of FreeCAD will be different, depending on the category of the selected object: Sketches, datum planes or other reference objects will be shown or hidden as intuitively expected. This is, however, different with generated 3D geometry within a body. The last state of the body design, called 'tip' (imagine the visible tip of an iceberg), will turn gray if an earlier state of the body design is made visible. Marking such a state, and subsequent pressing of the SPACE key, reverts the view up to the selected, now displayed design step. All later steps, including the previous tip, are then hidden (grayed out) and suppressed. In this way, changes can be invoked on the selected, earlier design step. After completion of the modification, it is good practice to activate the tip for display again so that all details following the change are invoked again. Possibly, some other, later, design features will then show the need for changes as well. If you omit displaying the tip again, and possibly collapse the view of the modified body, this can lead to confusing situations when, for example, some details of the assembled object appear to have vanished.

2.5 The Output Window

The output window is located at the bottom of the main window. There, warnings and error messages are listed. Warnings can be triggered, for example, by scope violations when references across the boundaries of objects are asserted. Even though the design and its associative properties could function properly despite these warnings, it is good practice to investigate the root cause for their appearance and proceed to provide a suitable remedy. Whereas warnings are displayed in orange, errors are listed in red. Some error messages result from temporary issues — possibly, a figure for a dimension was typed in as a number without units, which is the normal procedure during practical work. FreeCAD, in this case, will supply the missing unit(s) automatically.

The output window can be emptied of accumulated messages by right-clicking into the window and selecting Clear from the context menu. After that, it is advisable to initiate a recalculation, in order to find out whether an error message or warning persists. Objects which received changes are not always recalculated automatically. To make sure this happens, right-click on the questionable object in the tree view and manually select 'Mark for recalculation' from the context menu. Otherwise, you could conclude that some changes resolved an error that reoccurs later on (when the whole object is recomputed for some other reason). This can cause headaches when working on several things concurrently.

Chapter 3 • Project Organization

When starting a new design project with FreeCAD, it pays to reflect on its organization right from the start. This prevents the project from turning into a mess with too many items residing at the same hierarchy level in the tree view. Also, a lot of extra work can result from inserting a component into a project (like a bunch of spacers) and adding the associated screws later, one by one. The goal of a properly organized project is not only to view a nicely pruned project tree but also to prevent many repetitive tasks when finalizing the design with all its cosmetic additions.

Furthermore, the structuring consideration “what belongs to whom?" is useful to move related objects together within the context of a larger assembly. When, for example, far down the project a potentiometer is shifted, and miraculously the mounting holes in the front panel follow, as well as the engraving and the control knob, then the highly desirable state of associativity has reached perfection. This is very useful for achieving, for example, enclosure volume optimization together with an agreeable front panel design. On top of it, the probability of errors is minimized because none of the relevant, now coupled, details can be forgotten.

If a project structure is changed later in the process, the tools to do so naturally exist. But changes of this nature, like moving objects between different Std-Part-Containers, could result in unexpected side effects like the invalidation of references at object scope boundaries. These surprises, and strategies for their prevention, will be examined in some depth later, when the more complex assemblies are described.

3.1 Std-Part-Container versus Body

The design of an individual part as well as that of an assembly, should always start with the generation of a new Std-Part-Container. This choice is not automatically invoked by FreeCAD. It is also possible but less useful to start with a new body object right away (for example, when the project consists of only of this one piece of material). The initial choice to generate the Std-Part-Container may look like an unnecessary complication at first. But it buys a lot of convenience later: All bodies which are inserted into the new Std-Part-Container are placed inside their own, encapsulated coordinate system. Right from the start, they can therefore be moved together by just affecting the references of the enclosing Std-Part-Container, even when the contained bodies themselves have no other relations with each other. The placement of the contained objects can be manipulated by changing their position and angles in the Property list. To do this, expand the Placement property and click on the Value field that you want to modify. A slight touch of virtual reality is delivered with the possibility to change the selected property by turning the mouse wheel — the live update of the 3D view is excitingly close to actually touching the object itself!

In FreeCAD, every Body (not: everybody 😊) can consist of just one continuous piece of material. This is not a limitation for the creation of a complex part though, because the Std-Part-Container can contain many bodies. If pockets and other subtractive features are designed into one body, it is important that the actual object is not separated into multiple pieces by the removal of the material, as that would trigger an error message and be aborted as a process.

When a part is finished, consider inserting it into another assembly by just copying and pasting its enclosing Std-Part-Container. All the locally defined placement and attachment parameters are preserved by this move. Nevertheless, they can be modified by accessing the corresponding items in the Property list, for example, to turn a potentiometer control knob, or flip a toggle switch.

3.1.1. Placement versus Attachment

As already mentioned, newly generated bodies are initially only placed in the enclosed Std-Part-Container. Two ways to manipulate the placement parameters work. You can either expand the line Placement and enter values into the corresponding edit fields (or use the mouse wheel) as described above. Alternatively, you can click into the edit field which lines up with the label Placement. This click triggers the appearance of a small ... button. If you click that, a task window opens, in which the values for the placement parameters can be accessed as well. The task window needs to be closed with the OK button when done.

Alternatively, you can decide to attach a body to another body within the container. In order to define an attachment, it is necessary to switch to the 'Part' workbench. Next, the body to be attached has to be marked in the tree view. On selection of 'Part | Attachment' from the main menu, a task window opens in which the attachment relation can be defined. The attachment overrules the prior placement, which therefore appears gray in the property list of the body once the attachment relation is in place.

When defining an attachment relation, a placement needs to be selected (e.g., 'On plane' or 'Concentric'). This method resembles the creation of an assembly, the only difference being that the related objects are bodies and that the scope is limited to the enclosing Std-Part-Container. More precisely, the scope of an individual body is already encapsulated with respect to another body in the same Std-Part-Container. With the aid of a reference object, the 'Shape Binder', these boundaries can be made transparent for some references, as described below.

Once the attachment has been defined (or even an attachment definition was unsuccessfully canceled), a special section, with the heading 'Attachment', appears in the property list of that body. In that section, it is possible to access the 'Attachment Mode' task window by clicking into the 'Value' field of the 'Map Mode' line and issuing a subsequent click the ... button appearing in that field. This opens a task window in which the attachment mode is changed by the selection from the list, and by the entry of suitable offsets and rotation angles below.

Clicking on the 'Support' field in the same way opens the 'Link' dialog, in which a reference object can be selected to which the attachment is intended to relate. Because it is possible to relate to more than one reference object, it is important to remember that the previous reference is not removed automatically if you pick a new one. If you want to abandon the old reference, mark the highlighted entry in the list and click the 'Clear' button. If you clicked in error somewhere and wish to revert to the initial reference, you can click the 'Reset' button. To set a new reference object, after having cleared the old ones, just mark it in the list and click 'OK' to close the 'Link' dialog.

Embedding multiple bodies in one Std-Part-Container also make great sense to provision fasteners to a part. Fasteners are often needed to fix it to a real assembly and include items like screws, washers, and nuts. The supplied attachment modes including the concentric orientation, allow the quick and associative arrangement of items like washers directly at the edge of mounting holes or material boundaries.

3.1.2 Assembly or Std-Part-Container?

In version 0.20.1, FreeCAD lacks an official assembly workbench. However, a Std-Part-Container can also contain more Std-Part-Containers. This offers the possibility to construct an assembly merely from a set of nested Std-Part-Containers, with their relationships defined as described in the preceding paragraph. In principle, these are all the tools needed to furnish assembled designs. Throughout this book, this is the preferred method. As a result of each design step, a Std-Part-Container is again generated, which offers a high degree of portability and compatibility. Other methods to generate assemblies will be discussed in Chapter 6.

3.2 The Topological Naming Problem

Chugging on happily in 3D land, you are likely to bump into a severe but manageable limitation of FreeCAD (which you may sometimes even find with the commercial systems): The core system enumerates all facets of the generated 3D geometry. This in itself would go mainly unnoticed if there wasn't any chance of an unexpectedly triggered re-enumeration. Triggering events include the deletion of a chamfer or fillet.

Even that would again be harmless if there was no detail referenced to a certain facet, e.g., a hole in the end face of a cylinder. After the re-enumeration, the former reference number of that end face could be assigned to any other facet, taking the hole with it to an unexpected site. If worse comes to worst, the hole (if not the whole part) is rendered impossible to generate, and error messages pile up in the output window. The former design breaks and can only be recovered by redefining all the affected references one by one.

Such events can be quite nerve wrecking and time consuming. Luckily, the appearance of these phenomena can be fenced to some degree by safeguards and strategies that will be described below.

3.3 Sketches and Datum Objects

One way out of this topological misery can be taken by the definition of datum objects. These objects, as well as references pointing to them, are not affected by re-enumeration. Furthermore, references to sketches or parts thereof — like individual contours or curves — also survive the number storm. Unlike jumping from facet to facet in 3D land, it is a bit harder to climb up the geometry mountains when nailing in these safety anchors.

The additional effort can pay dividends if the defined planes refer to important design parameters like the length of a spacer. In this case, it could be sufficient to change just that parameter in order to generate many variants later. Besides this straightforward concept, in a complex project, it would also suffice to reposition many related parts by changing just one number. It is obvious that the associativity empowered by these assignments plays an important role when, for example, the volume of a module needs to be minimized, or some other part needs to be inserted in between during the design update.

3.4 Other Strategies to Avoid the Topological Naming Problem

It will not always be feasible to strictly avoid references to generated facets. One pragmatic strategy could be to define all principal shapes of a part until no noticeable need for additions exists anymore. An exception from this could include the adaption of certain dimensions. Only then, cosmetic features like chamfers or fillets should be added — they usually lead to many, newly generated facets. But here, too, a major pitfall lurks: for example, in case an altered dimension leads to the loss of a major detail — like a surface on the principal shape — a re-numeration could be triggered, which irreversibly breaks the design. 'Irreversible' in the sense that even clicking the helpful 'Undo' button will not restore the previous shape anymore. In that case, all earlier design steps of the affected body need to be reviewed and their references updated manually — one by one.

3.5 Avoidance of Repetitive Work by Project Structure

The previously described situation demonstrates an obvious idea: It is meant to divide a project into many easy-to-overlook, separate parts. Each of these could be defined and finalized in its own Std-Part-Container before the sphere of larger assemblies is entered. On the side, these separate parts, saved into individual files, can be grouped, and subsequently stored to form a nice and valuable library of design resources that saves a lot of time later on. This does not limit the option to modify an individual instance of such a part when it is copied into an assembly.

3.6 Always Start with the Std-Part-Container

As a golden rule in the making: The Std-Part-Container should always be your first step, which is very easy to do: start a new file by selecting 'File | new' from the main menu and save it with a descriptive name. As the initial step, switch to the workbench 'Part Design' and click the yellow 'Part' tool button (Figure 3-1). In the tree view, the yellow icon of the Std-Part-Container appears. The title of this container can be changed by a right-clicking it, and the selection of 'Rename' from the Context Menu. Alternatively, it can be edited by pressing the F2 key once the container has been marked in the tree view. Using a descriptive name is rewarding even though it may seem superficial at times, because you will use Copy and Paste later on to insert the Std-Part-Container into assemblies. Watch out where the mouse points to when you press F2. If the mouse points into the 'Description' column of the tree entry (which is not visibly separated below), the cursor will appear in that column, which can be confusing. To delete unwanted input from the 'Description' column, just point the mouse to it again and press F2 to access the entry.

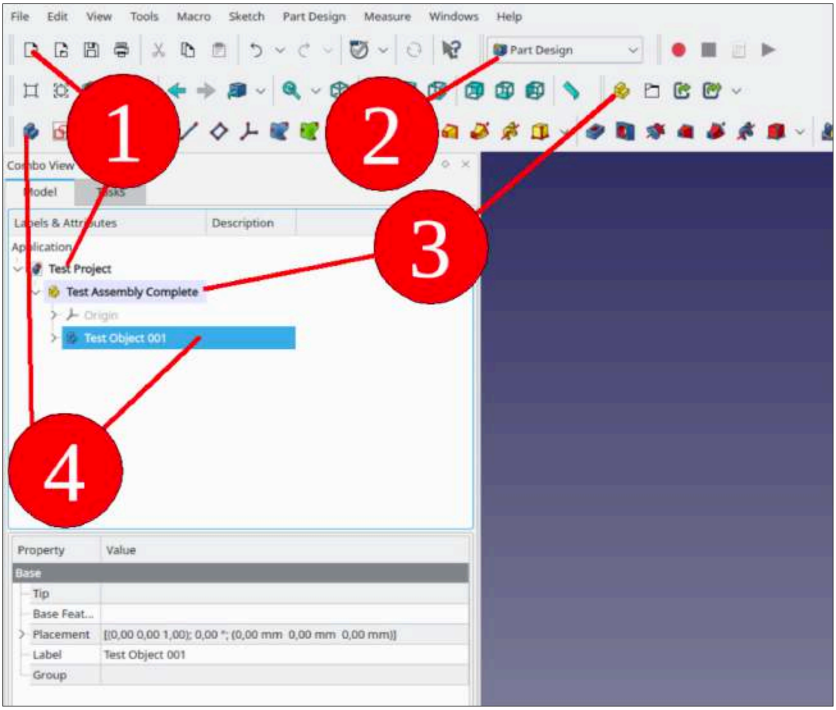


Figure 3-1:

It's also possible to start off with a single, new body object right away. This, however, would block many opportunities for structuring the project with special appearance attributes, various placement options, and others. For example, a rocker switch could have a casing made of black plastic, a movable actuator, which could be red and transparent, and contacts, which could appear metallic and shiny.

Also, operations like the projection of sketches to curved surfaces required for, say, lettering on switch actuators, will generate new body objects. These should be grouped together with the body in an enclosed Std-Part-Container to allow easy insertion of the part into assemblies later on.

3.7 Shape Binder and SubShapeBinder

When a project is assembled from different Std-Part-Containers, there is sometimes the necessity to modify a part in relation to others. An example of such a 'creative carving' process is the insertion of openings into a front panel, which relate to the contours of the corresponding control elements placed on it. An associative relation between the mounting holes and the control elements implies communication between body objects residing in different Std-Part-Containers. Because the scope for such details is very limited, special reference objects exist which allow making the connection across Std-Part-Container and body limits: The Shape Binder (blue icon) and the SubShapeBinder (green icon).

3.7.1 The Shape Binder (blue)

The Shape Binder is always created in an activated body object. The body can be activated by double-clicking its tree view entry. In the activated state, the title of the body will appear in bold letters in the tree view. If it was forgotten to activate one, FreeCAD presents a dialog, from which a body can be selected. As an alternative, listed in the selection dialog, the creation of a new body object could also be chosen, into which the new Shape Binder is subsequently placed.

With a valid selection, be it in tree view or in the selection dialog, a task window opens for the definition of the Shape Binder. This task window contains two collectors, one named Object and one named Add Geometry. Each collector is activated by clicking its button, which turns dark gray. Then the entry field next to the button needs addressing by another click into it — the rim of it turns blue. This multi-click feature is a thing to get used to.

If the collector called 'Object' is chosen, it is possible to switch the Combo View tab to 'Model' — with the task staying alive. In the tree view, sketches or datum planes can be selected even when they are currently hidden from the 3D view. Also, objects, which may be difficult to pick from the 3D view, can be selected in the tree view with greater ease. Alternatively, it is possible to point and click details from the 3D view directly. After one item has been selected, it will be listed in the 'Object' field of the task window. Then, in the second step, the 'Add Geometry' button allows the selection of the desired detail.

Often, clicking the collector 'Add Geometry' is sufficient for updating 'Object' and 'Geometry' list entries simultaneously, for example, when borders or facets of an object in the 3D view are selected. It is also possible to add multiple details of an object to the 'Geometry' list. In order to do so, the 'Add Geometry' button must be clicked each time again to enable the next selection. Importantly, Sketches do not need to be selected as an entity only — also individual contours from a sketch can be picked, e.g., the mounting holes of a complex footprint.

Finally, the task window can be closed with the 'OK' button. In this book, a constant effort is made to name each item in the tree view with a self-explanatory name. This is also good practice for shape binders — and makes it easy to understand the concepts embedded in a design, also after months, if not years, of lying dormant in the project folder.

If shape binders get momentarily piled up in a hasty manner, their references can be displayed or hidden in the 3D view like any other object for identification (mark them in the tree view and press the SPACE key). The other way also works — if a Shape Binder receives a click in the 3D view, FreeCAD will expand the tree view and expose the corresponding, highlighted item.

Last but not least, the Shape Binder does not follow changes of the referenced object automatically, unless its property 'Trace Support' in the property list is set to 'true'.

3.7.2 The SubShapeBinder (green)

In contrast to the Shape Binder, the SubShapeBinder (green icon) can also be generated outside of bodies or Std-Part-Containers.

The selection of the referenced objects also differs: At first, the detail is selected by marking it in the 3D view or tree view. Multiple details can be selected as well by holding down the CTRL key and marking all of them. Only then, the green tool button 'Create a sub object(s) shape binder' from the workbench menu should be clicked. The green SubShapeBinder object icon then appears either in the branch of an activated body object, Std-Part-Container or — if none was activated — in the root node of the tree view.

The SubShapeBinder could also reference objects from external files — but in this book, you will not utilize this option. By default, the SubShapeBinder is always actively tracking its reference. You can change its behavior using the property 'Bind Mode'. The default value 'Synchronized' can be changed to 'Frozen' to disable the tracking, or to 'Detached', in order to copy the referenced shape once, and reference the independent copy only.

The SubShapeBinder can show slightly unexpected behavior when it is referencing a plane (or a datum plane), e.g., the XY plane of an object. When the Std-Part-Container, in which the SubShapeBinder resides, is attached later to a plane of different orientation (e.g., the YZ-Plane of the assembly into which it is embedded), then the SubShapeBinder will retain its original orientation literally (e.g., reference the XY plane of the assembly). In contrast, the blue Shape Binder with activated 'TraceSupport' will follow these transformations, albeit with different side effects, as discussed later.

3.7.3 How to Reference Cut-and-Pasted Std-Part-Containers

The supplemented materials include a small sample project allowing the effects of the different types of shape binders to be studied. The file location is: 'Sample Projects | Shape Binder Test | Test Front Panel.FCStd'. It contains a front panel and two pilot lamps attached in different ways. The blue pilot lamp uses the Shape Binder, and the green one the SubShapeBinder. The two pilot lamps each have a fastener on the back side, and the Shape Binders reference the position of these fasteners with respect to the pilot lamp body.

The 'Trace Support' property of the blue Shape Binder is convenient. In the Std-Part-Container of the blue pilot lamp, the fastener is added such that it follows the position of the pilot light casing body. This seems a bit fancy in this example case with only one single fastener but makes more sense if there are more parts in the Std-Part-Container, like multiple stacked nuts and washers, and multiple mounting holes.

If the Std-Part-Container of the blue pilot lamp is inserted into the project by cut-and-paste, the Shape Binder reference breaks up and an error message is produced (Figure 3-2):

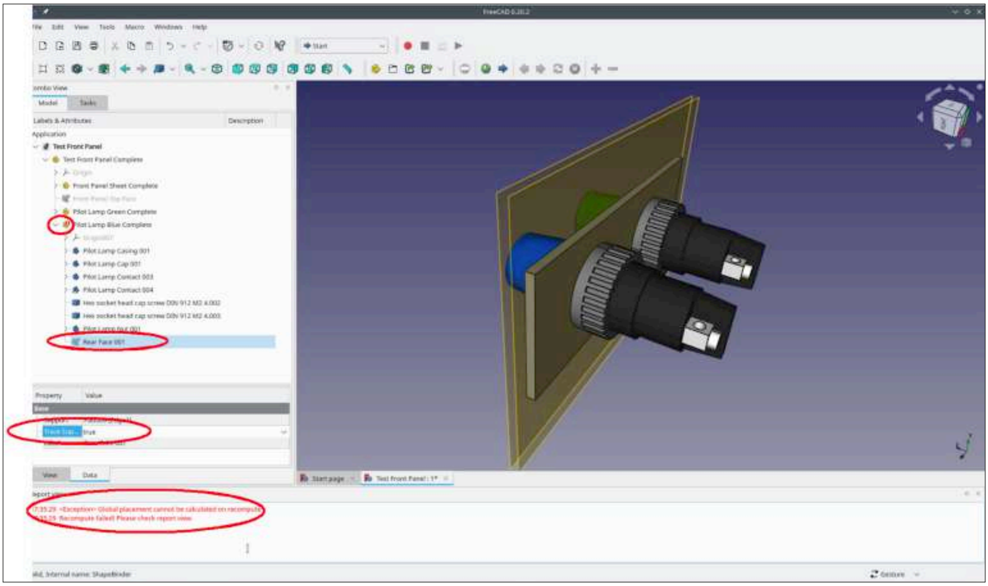


Figure 3-2:

```
<Exception> Global placement cannot be calculated on recompute
Recompute failed! Please check report view.
```

In contrast, if a SubShapeBinder reference is used as with the green pilot light, no error message results.

A simple and often sufficient method to resolve these issues consists of omitting the attachment of supplementary parts and fasteners, and simply leaving them inside the just placed enclosing container. Then, however, all of these parts have to be taken to their final positions in the assembly manually, one by one.

Alternatively, the property 'Trace Support' of the conflicting Shape Binder could be set to 'false' after the sub-assembly has been finished. This suppresses the error message but freezes the position of the fastener (Figure 3-3). If things are to be moved around for optimization later on, the frozen positions might increase the error probability.

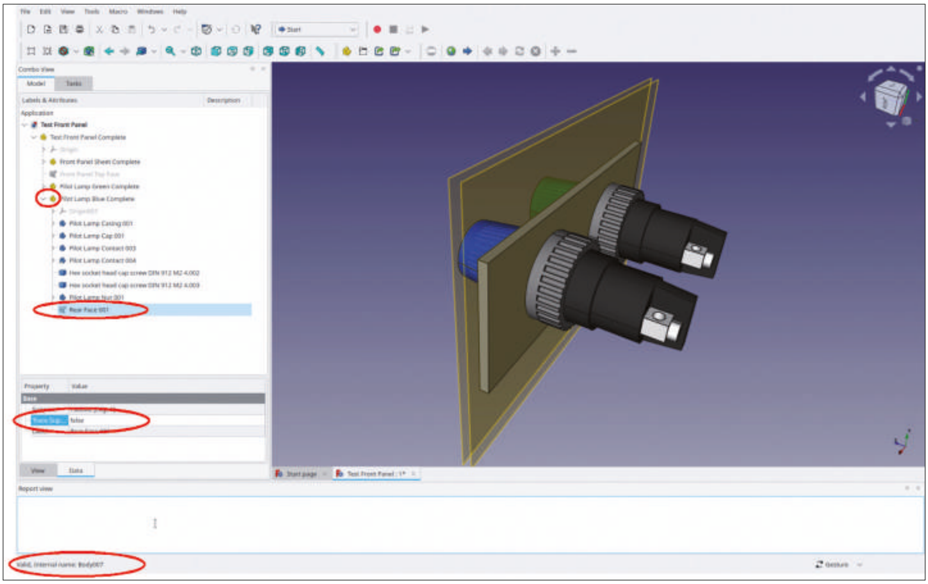


Figure 3-3:

As an intermediate result, the use of the SubShapeBinders avoids errors when recomputing the global coordinates. However, this comes at a cost: Their reference sometimes does not follow re-orientations of the parent Std-Part-Container when being rotated to different assembly planes. Therefore, some experimentation remains to be done, in order to find the smoothest solution.

It can also be useful to redefine the references of the inserted components in the final assembly. This adds some more work, and the components are to some extent less portable when copied out of the assembly for recycling at a later time.

3.7.4 Redefining the References after Paste

As an example of the redefinition of the references, in the supplementary materials, there is another version of the sample project, in the file:
'Sample Projects | Shape Binder Test | Test Front Panel Shape Binders on Backside.DCStd'.

There, an attempt is seen to reference the back surface of the panel for the positioning of the fasteners with both types of shape binders, as a demonstration of certain error cases. To change the reference for the blue Shape Binder, double-click the blue shape binder entry in the tree view to display the task window. There, delete the old entries for 'Object' and 'Geometry'. Then, activate the 'Add Object' collector, switch to the model tab, and select the reference plane in the 'Panel' object by clicking on it. Close the task window with the 'OK' button. Because the Shape Binder is now referencing a plane, change the attachment mode of the 'Pilot Lamp Nut' to 'XY on Plane' by editing the property 'Support'. Again, the computation of the global coordinates fails.

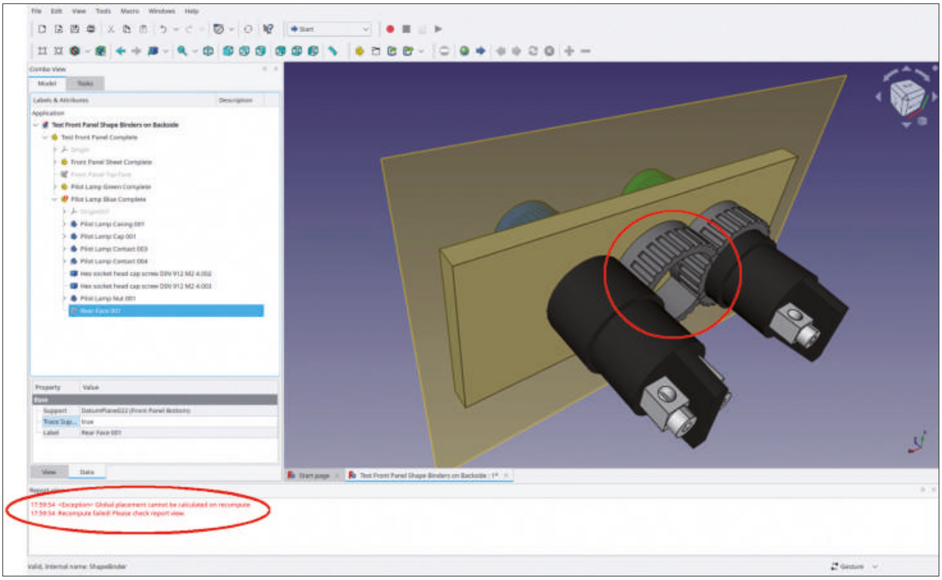


Figure 3-4

This can also be seen in the 3D view — the fastener is not located concentrically with the pilot light casing anymore (Figure 3-4). Resetting the 'Trace Support' property to 'false' restores the orientation but disables the position tracking of the reference plane — this not a useful result either (Figure 3-5).

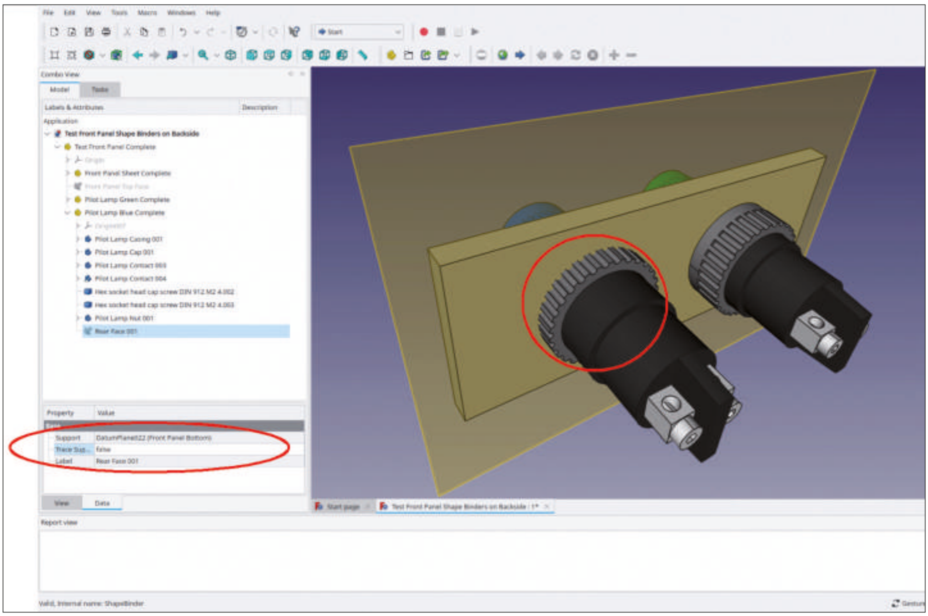


Figure 3-5

The green SubShapeBinder cannot be edited with a task window. It is easiest to delete it and regenerate it from scratch. After deleting the SubShapeBinder in the tree view, click the defining reference plane in the body 'Front Panel Sheet' and click the tool button 'SubShapeBinder' in the workbench menu.

In the green pilot lamp, the attachment of the fastener can be changed by editing the property 'Support'. In the selection dialog, delete the former entry and select the new SubShapeBinder. This brings up yet another error message:

```
PositionBySupport: TopoDS::Face
```

The green SubShapeBinder cannot create a reference of the fastener to an infinite object (the reference plane). In order to resolve the issue, you could choose to define the SubShapeBinder as a reference to a non-infinite object (e.g. the sketch of the panel).

As a working remedy, you could create a datum plane locally in the Std-Part-Container of the pilot lamp and define the SubShapeBinder reference between these two planes (plane to plane — works). The fastener, you can then set up to follow this new, local datum plane (by editing the 'Support' properties).

Even though the concept with the plane-to-plane reference looks a bit awkward — an 'intermediate' datum plane is necessary — it can be an advantage, for example, when referencing many fasteners to this 'intermediate' plane and associating that later on with a single assignment to, say, the back of the panel.

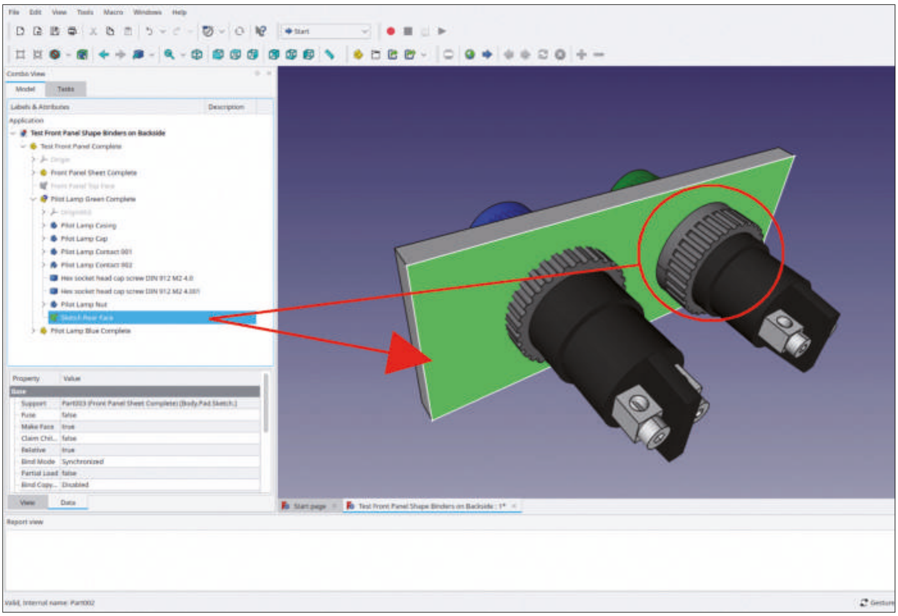


Figure 3-6

In the example file, you will find that the sketch of the 'Pad' for the front panel was chosen as a reference mainly because it coincides with the back of the panel. (Figure 3-6). The fastener is now following the back side of the panel.

As a result, when references are updated, the green SubShapeBinder seems a bit more complicated (no task window to edit!) but responds more robustly to the recalculation of the global coordinates. This is favorable and easier, as long as no re-orientations of the enclosing Std-Part-Container are invoked by its attachment in an assembly. Later, you get to study an example where such a re-orientation has to be dealt with.

Chapter 4 • Creating Parts — First Steps

After the description of some concepts, it is now the time to move toward the practical application.

4.1 How Much Fidelity is Needed?

A digital caliper is useful to determine the component's dimensions but flipping datasheet pages to sections with mechanical drawings is even better and time saving.

As a pragmatic approach, it is meaningful to realize that the 3D models created from the components are often used to define mounting holes and cutouts in a given panel. That task is easier to complete when the parts are drawn such that they describe the openings rather than the part itself — by adding some excess to the corresponding contours. With this often subtle modification, an appropriate fit between component and mounting hole results, by simply referencing the component contours for pockets or holes.

Although it is tempting to focus on the beautiful outside of a front panel, with 3D design it is as important to have a meaningful representation of the parts that extend to the inside. These parts can appear simplified, as long as they allow the analysis of collisions, the accessibility for the wiring harness, and other important spatial properties.

On the outside of a panel, it is often sufficient to have a coarse outline of the control elements only. But the true pleasure of using FreeCAD comes when beautiful and realistic representations of the work are created and viewed. This can deliver strong motivation, if not appetite, to finalize often lengthy and elaborate projects. Therefore, the effort to master details and appearance seems well invested, besides all the necessary technicalities.

4.2 Simple Parts

A simple part could be a spacer. It consists of only one body object, more precisely an extruded, round plastic tube. Such objects can be generated with little time and be associated with realistic appearance attributes. With these parts, it is easy to decide whether their design is finalized. After the assignment of all the properties (color, material), they can be stored in a nice library for picking later during assembly.

4.2.1 The Simple Spacer, Step by Step

1. **Preparations**

   Start FreeCAD and select from the main menu 'File | New'. Save the file as 'Spacer'. Select the 'Part Design' workbench from the Combo Box. In the Combo View on the left-hand side, a task window opens, offering basic things to do. For the moment, this can be left aside. Select the 'Model' tab in the Combo View to display the still empty tree view.

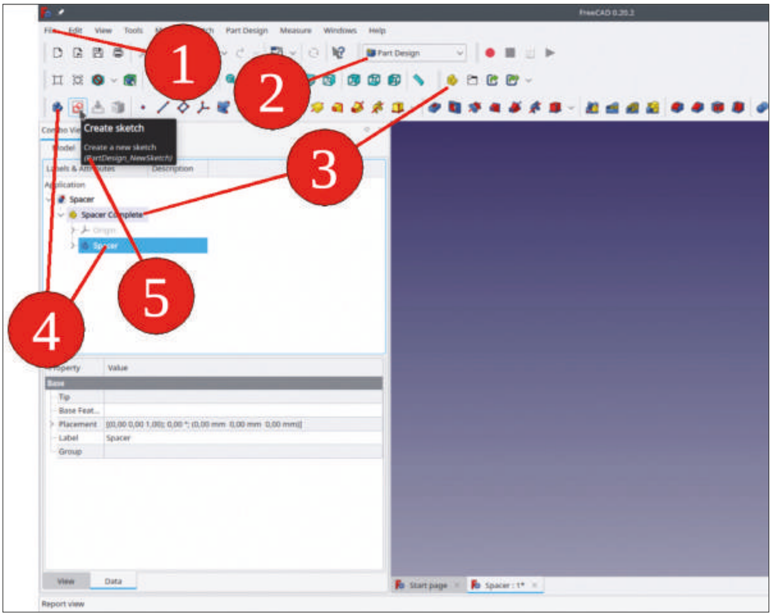


Figure 4-1

Click on the yellow 'Part' tool button in the workbench menu (Figure 4-1), to generate a new Std-Part-Container. In the tree view, you can see that the Std-Part-Container is created together with its own coordinate system named 'Origin' (grayed). Rename the Std-Part-Container to 'Spacer Complete' by right-clicking the entry in the tree view and selecting 'Rename' from the context menu. Alternatively, it is possible to mark the Std-Part-Container in the tree view and press F2. But caution — then, it depends on the mouse pointer position, which field is updated. This can be confusing: If the mouse pointer is pointing underneath the 'Labels and Attributes' column, renaming works as expected. If the mouse pointer rests underneath the 'Description' column, the description field opens for editing. The name 'Spacer Complete' sounds a bit overkill — as the spacer consists of just one body object. But it is good practice to think of multiple bodies right away. Along with the spacer, you will later add the screw, washer and nut, to the same container.

As a last preparation, you activate the new Std-Part-Container by double-clicking on its tree view line (the title of the item is then displayed in bold letters) and selecting the blue 'Body' tool button from the workbench menu. Rename the new body object in the tree view to 'Spacer', using one of the methods described above. In Figure 4-1, all these steps have been completed for your reference.

2. Now, in order to create the first part, define the cross-section of the spacer as a sketch. Click on the 'Create Sketch' tool button in the workbench menu (Figure 4-1, number 5). A task window opens, in which it is possible to select the sketch plane. You can either select the plane from the list or click on a plane representation in the 3D view. As the sketching plane, select the XY-Plane001 from the list and close the task window with the 'OK' button (on the top of the task window). The sketcher workbench opens, which changes the selection of tool buttons and context menu items. Also, in the Combo View, two new list boxes appeared ('Constraints' and 'Elements'), which will be populated in the course of the sketch creation.

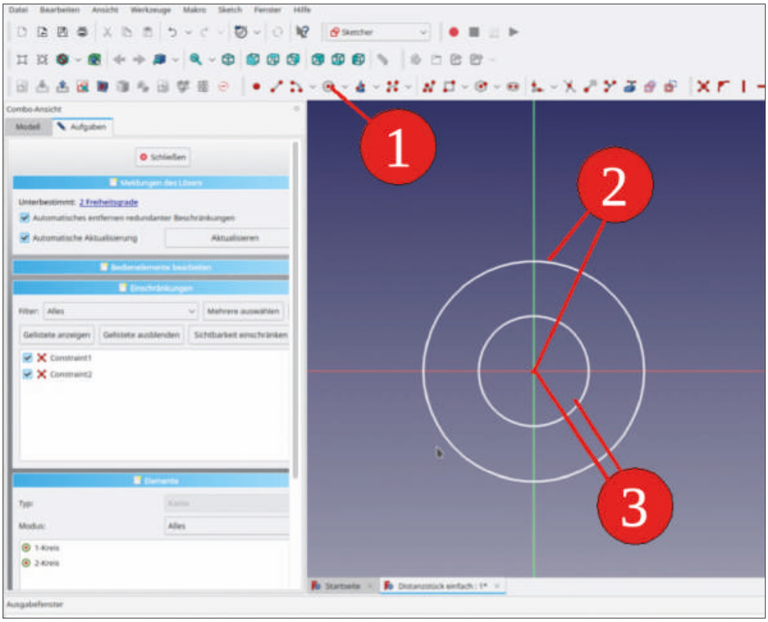


Figure 4-2

The work with the sketcher is not always straightforward. Sketches need to be properly defined by constraining, but not over-constraining, all degrees of freedom. The designer, with his ideas, may sometimes feel entrenched with this methodology. But without these strict definitions, how could the computer know which requirement has priority? Some time and nerve-saving strategies that result in fully constrained sketches will be shown in detail later. With the simple example here, you do not need to worry too much (yet).

The spacer has a simple structure — its cross-section is described by just two circles, concentric to the origin. To start the sketch, select 'Create circle' from the sketcher menu (Figure 4-2, step 1). The mouse pointer will change its shape into a little crosshairs with an attached symbol. It is easy to overlook the small crosshairs, but as tiny as they are, they are the center of action. To start a circle, approach the origin with the little crosshairs, until the red dot representing the origin changes its color to yellow. Only then, the pointer is locked to the origin (any other point object which you want to select behaves like that). This is important! If you miss the origin (or target point) only by small amounts, additional degrees of freedom would result (here for the position of the circle center). With the origin clicked, you can pull the circle to some diameter, which will be specified more precisely in a later step. Using the same method, draw a second circle, concentric to the first one. When the second circle is finished, the 'circle' symbol still sticks to the mouse pointer, happily expecting more circles to come. To end the command, just issue a right-click into the blue. This works with all the other drawing commands in the same way.

3. In the Combo View to the left, the 'Elements' list has been populated with the two circles (Figure 4-2). Also, in the list 'Constraints', two symbols for 'Coincidence' are listed, because the two center points of the circles are constrained to coincide with the origin.

The two circles are still displayed in white. In the list 'Solver Messages' on top of the Combo View, the first line reads 'Underconstrained: 2 DoF(s)' (degrees of freedom). In order to display the under-constrained offenders, click on the hyperlink '2 DoF(s)'. In contrast to the white ones, elements that are fully constrained will then show in light green, when the constraints have been properly defined. The art is now, to select the degrees of freedom such, that just the correct number of constraints results. This can be quite a headache if the shapes are more complex. Luckily, the present example is simple: As the origins of the circles are already fixed, the only remaining freedom is the selection of their diameters (or radius).

4. To define the constraint, in the 'Elements' list, right-click the outer circle. Its color changes to green, in order to indicate that work with it is in progress. In the top part of the context menu, the available constraints are listed with red symbols (Figure 4-3).

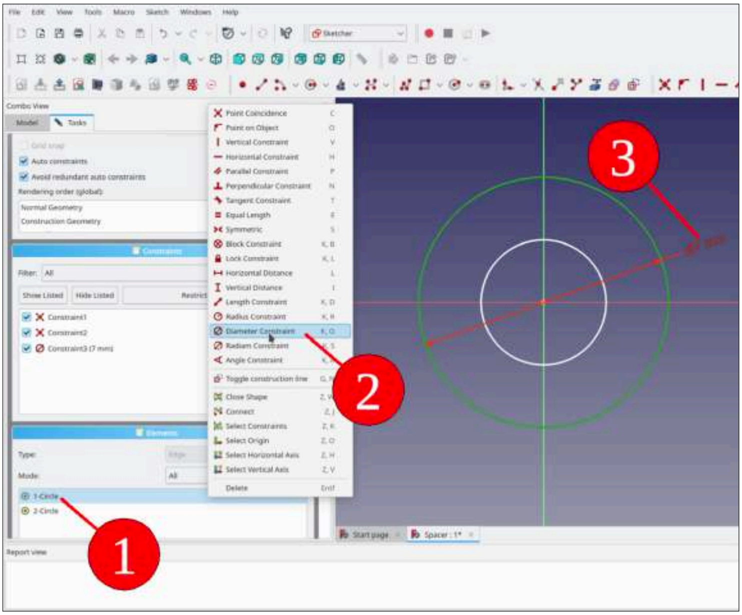


Figure 4-3

Select 'Diameter Constraint' from the context menu and enter 7 mm in the small parameter dialog which then opens. Now, the green color of the circle is persistent, indicating it is fully constrained. The green flavors are important: If you click now on the hyperlink '1 DoF(s), the remaining underconstrained object is taking the green color, whereas the fully constrained one is displayed in light green. This color play is very useful, especially when a sketch stubbornly refuses to be fully constrained.

In the same manner, change the diameter of the inner circle to 3.5 mm. The success appears twofold: The whole sketch changes the color to bright green, and in the 'Solver Messages' window, in green letters 'Fully constrained' is proclaimed (Figure 4-4).

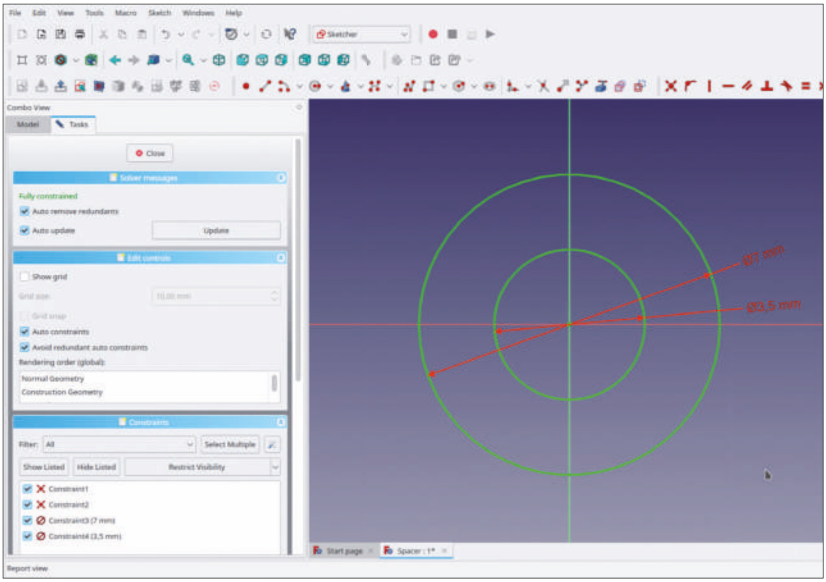


Figure 4-4

In case an error occurred during the entry of the diameter, or a change of it is required, a single left-click on the corresponding item in the 'Constraints' list opens a dialog window, in which the parameter can be edited. When the sketch looks finalized, close the sketcher by clicking the 'Close' button on top of the task window. Sometimes, it is difficult to find because the other lists have expanded. Soon, you will scroll automatically to the top of the sketcher task to find the 'Close' button.

5. After closing the sketcher, the sketch is displayed in the 3D window to the right. In order to create the spacer, mark the sketch in the tree view and select the 'Pad' tool button from the workbench menu (Figure 4-5).

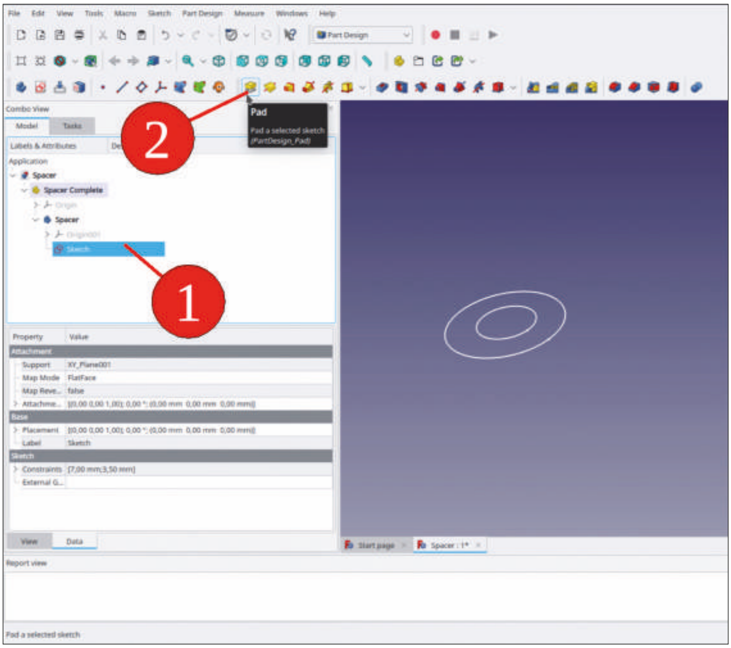


Figure 4-5

The 3D geometry appears, and a task window opens. There, select for the type 'Dimension' (which is preset), and for the length 10 mm (also selected by default). The 3D-geometry updates when you change parameters, e.g., by clicking into the 'Length' edit field and rolling the mouse wheel (Figure 4-6).

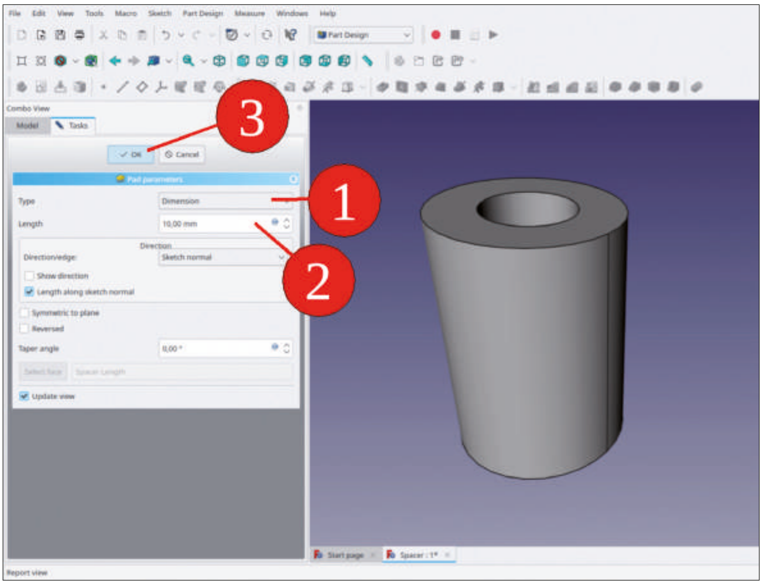


Figure 4-6

Close the task window with the 'OK' button. This flips the tab of the Combo View back to the 'Model' tab.

6. In the 3D view, the spacer still appears in its standard color (Figure 4-7). With a right-click of the 'Spacer' body, you can select the item 'Appearance...' from the context menu (Figure 4-7).

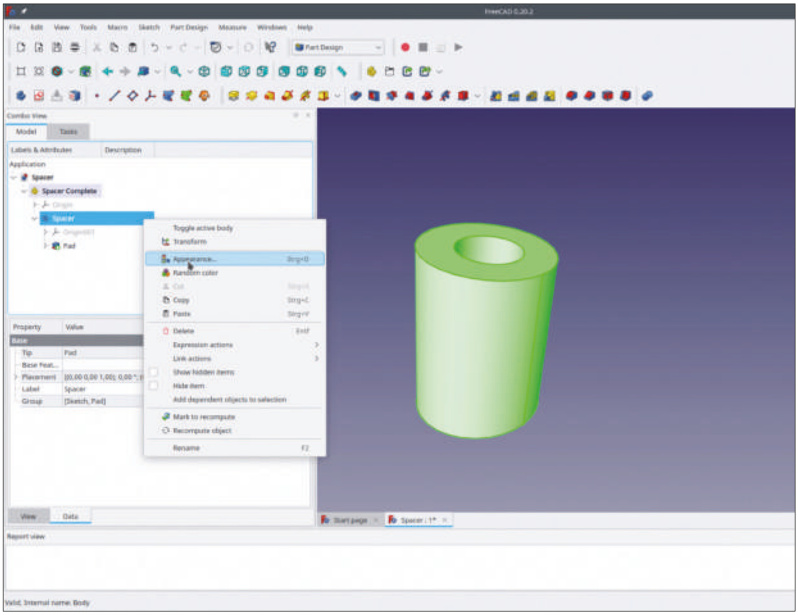


Figure 4-7

A new task window opens (Figure 4-8), which allows selecting the viewing mode (to suppress the lines in the objects, you can select 'Shaded'), but also color and reflectivity presets for different materials: From the combo box 'Material' select 'Shiny Plastic', which defaults to a black shape color (sometimes dark gray can be better distinguishable). This task window is closed with the 'Close' Button at the bottom.

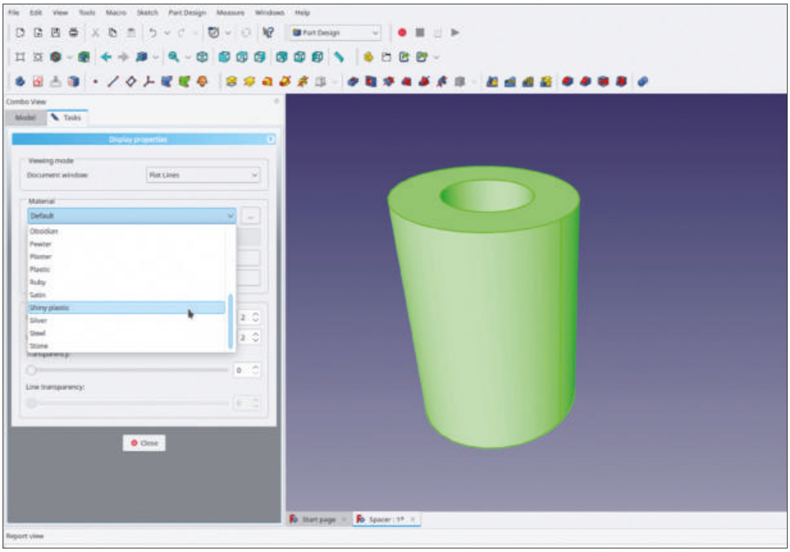


Figure 4-8

After the closure of the task, the new spacer gloriously appears in the 3D view, with an astonishing resemblance to something you have used probably thousands of times before (Figure 4-9). <CTRL-S> saves it for later use.

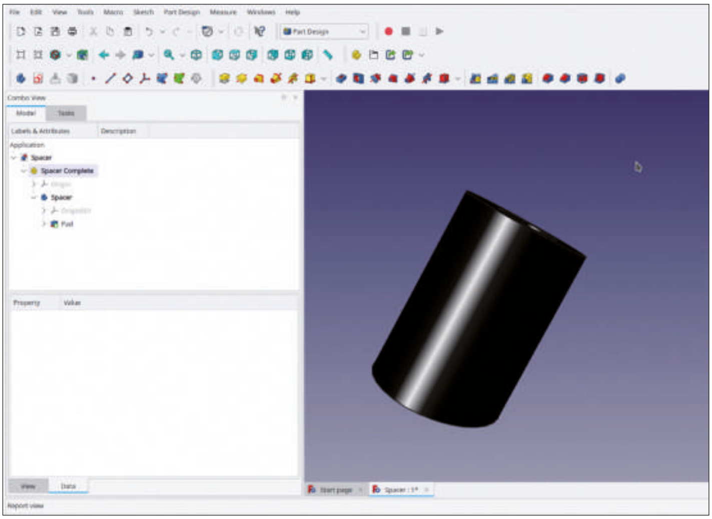


Figure 4-9

4.3 Color versus Appearance

When newly created, 3D geometry is displayed with the gray default color. If a project is assembled, special colors can render the individual components more distinguishable. On top of that, as just seen with the spacer, setting the 'Appearance' attributes can render the display very realistic and beautiful.

In the tree view, if you right-click a tip of a body, you can find the item 'Set Colors' in the context menu. With that function, you can paint individual facets of a body to your liking. Unfortunately, the coloration is bound to that tip, which means that it will disappear once another design step is undertaken with that body. They only reappear if you revert the state of the body by making that special design step visible again, by marking and pressing the SPACE key.

Therefore, it is more meaningful to use the 'Appearance' attributes with the different bodies in a project, as was done with the spacer in the previous example. These attributes are persistent with respect to design changes. Also, the variety of preset material profiles, which determine color, reflectivity, and transparency at the same time, allow the creation of an astonishingly realistic representation of the work.

4.4 Compound Parts

A part composed of multiple bodies could be, e.g., a rocker switch which has a casing, an actuator and contacts. The actuator could carry lettering (like 1 and 0) and should be movable in order to represent a switch position. Rocker switches with curved and flat actuators are provided with the supplemented material online. The advantage of having the Std-Part-Container as an encapsulation for all these different body parts is evident with these objects. Because some details can be more involved (like lettering attached to a curved surface), you will continue to use our simple spacer and extend the example to some degree.

4.4.1 Fasteners for the Simple Spacer

Our spacer could be nicely extended to a compound part by adding fasteners to it. This comes with an increased utility: When the spacer is pasted into an assembly later, there is no need to add the mounting screw, washer, or nut one by one. Plus there is associativity between fasteners and spacer, because all these internals refer to the same coordinate system of the enclosing Std-Part-Container. The use of a macro-based functionality in FreeCAD can be demonstrated by utilizing the extension BoltsFC.

In the first pass, you will intentionally introduce a small, but instructive error, when defining the relations between the fasteners and the spacer.

1. If it was closed in the meantime, reopen the file 'Spacer.FCStd'. In order to activate the spacer body, right-click the body 'Spacer' and select 'Toggle active body' from the context menu (alternatively, double-click the body line in the tree view). The title of the body will be displayed in bold print when it is activated. Also, the enclosing Std-Part-Container is activated by this step.

2. From the main menu, select 'Macro | Recent Macros | start_bolts'. On the right-hand side of the main window, the BOLTS Parts selector opens.

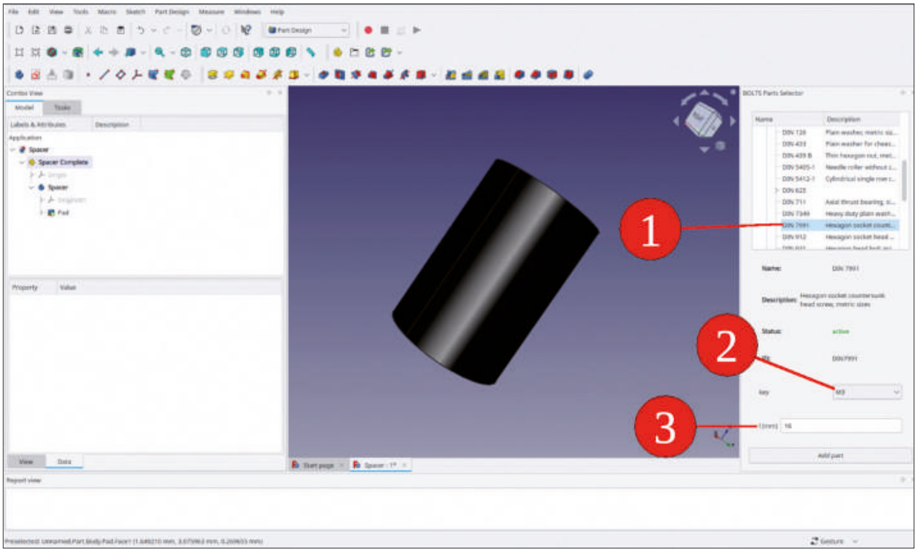


Figure 4-10

3. From the list, select 'Standard | DIN | DIN 7991Hexagon socket countersunk head', leave the key at the default 'M3' and set the length l to 16 mm (Figure 4-10). Then click the 'Add Part' button at the bottom of 'BOLTS Parts selector'.

4. The screw appears in the 3D view. In the tree view, however, it is not yet listed as a part of the Std-Part-Container 'Spacer Complete'. Also, the surface of the screw head is displayed discontinuously (Figure 4-11).

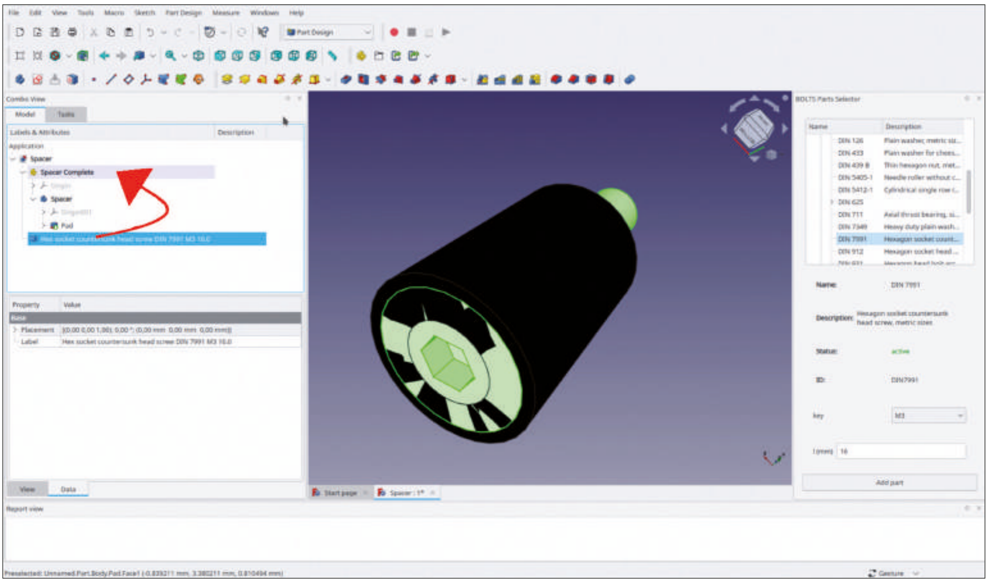


Figure 4-11

This effect is caused by the coincidence of the screw head surface with the spacer surface – the association of the plane with an object is not unique, so FreeCAD displays them all at once.

In order to associate the screw with the spacer, in the tree view, it can be moved by drag-and-drop into the Std-Part-Container 'Spacer Complete'.

5. After the move, the screw appears as an object embedded in 'Spacer Complete' (Figure 4-12). In the Property list, only 'Placement' parameters are listed.

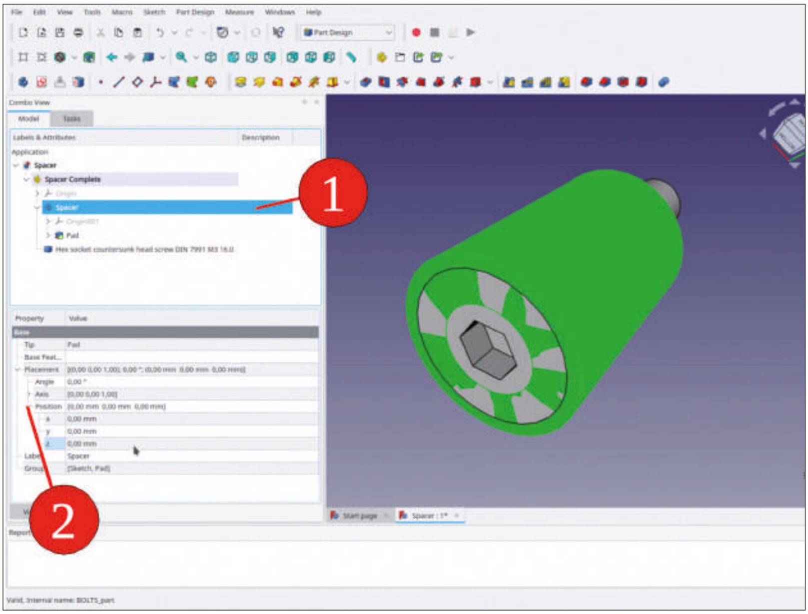


Figure 4-12

The body of the screw is thus only placed, and not attached to other bodies. You can expand the property list by using the arrow keys on the left-hand side (Figure 4-12). The arrows allow you to navigate to the z coordinate (listed in the item 'Position'). Here, usual the usual thickness for a chassis plate could be entered. Once the edit field next to a coordinate identifier is clicked, it is also possible to manipulate the displayed values with the mouse wheel. In this way, screws could be 'tightened' also later on.

6. The position of the screw is still unsatisfactory. When a 2-mm aluminum sheet is usually taken for the chassis, it is useful to move the screw to the correct position right away. In this simple case, you do not have to define attachments. Simply click the edit field for z and roll the mouse wheel, or enter the value of -2.01 mm manually, with the small offset of 0.01 mm added for the clear display of the screw (Figure 4-13). The resulting offset of the screw with respect to the spacer is displayed in the 3D view with an immediate update.

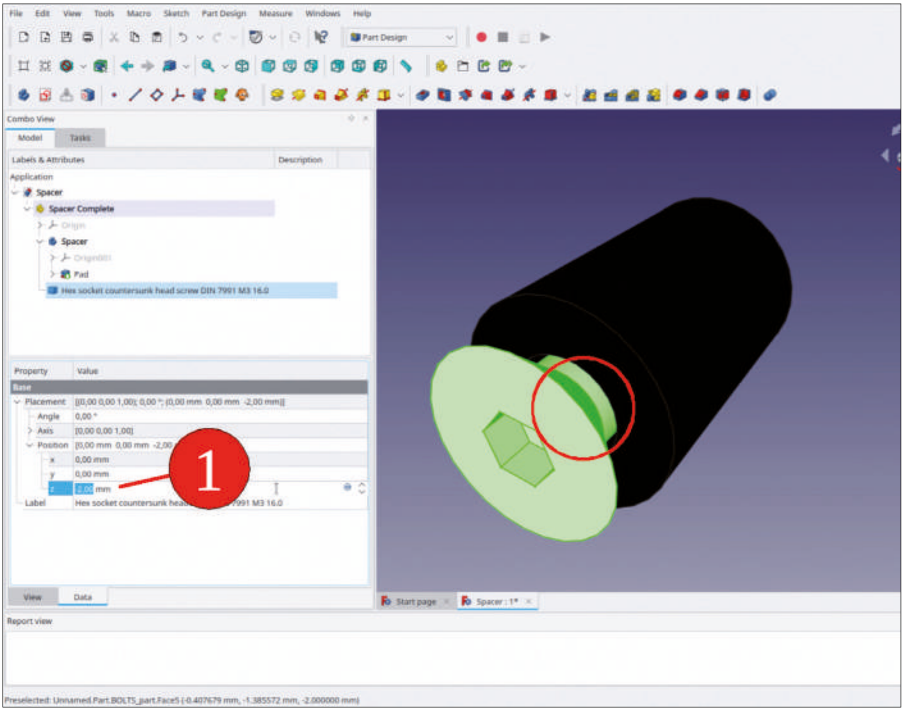


Figure 4-13

7. From the 'BOLTS Parts Selector', choose 'DIN | DIN12 | DIN125 A Plain washer, metric sizes', with key M3 (preset), and click the 'Add Part' button (Figure 4-14, steps 1 and 2). The washer appears in the 3D view. In the tree view, it needs to be dragged-and-dropped into the Std-Part-Container 'Spacer Complete', in order to gather the belongings.

8. Again, the placement parameters could be used to bring the spacer to an appropriate position. However, it is better to define an attachment for the automation of that step. This is useful, if later e.g., variants of the part are needed which differ in length. Then, after the change of just the length parameter, the washer (and also nut, later) will follow the designer diligently. In order to define the attachment relation, switch to the 'Part' workbench (Figure 4-14, step 3).

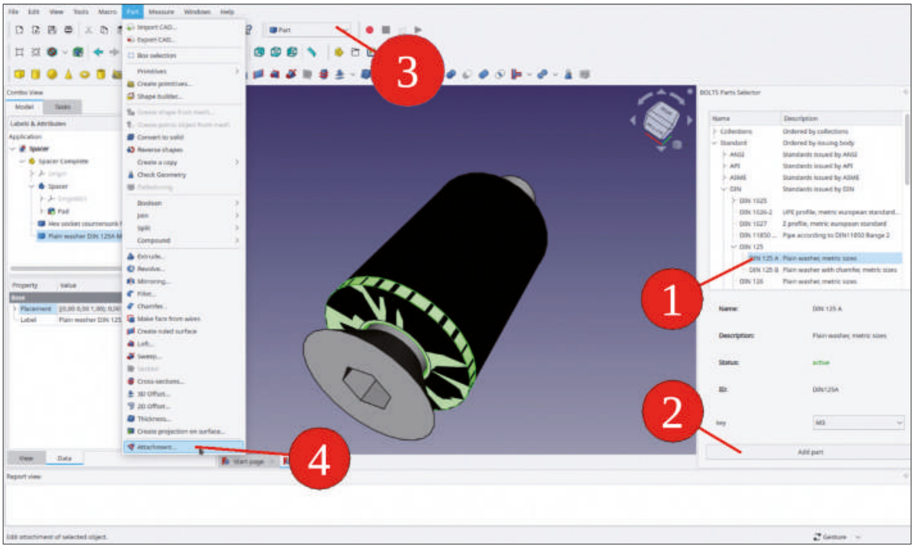


Figure 4-14

9. In the tree view, because of the preceding steps, 'Plain washer DIN125A M3' should be still marked, otherwise click the item. From the main menu, select 'Part | Attachment' (Figure 4-14, step 4). A task window opens, in which you can proceed with defining the new relation.

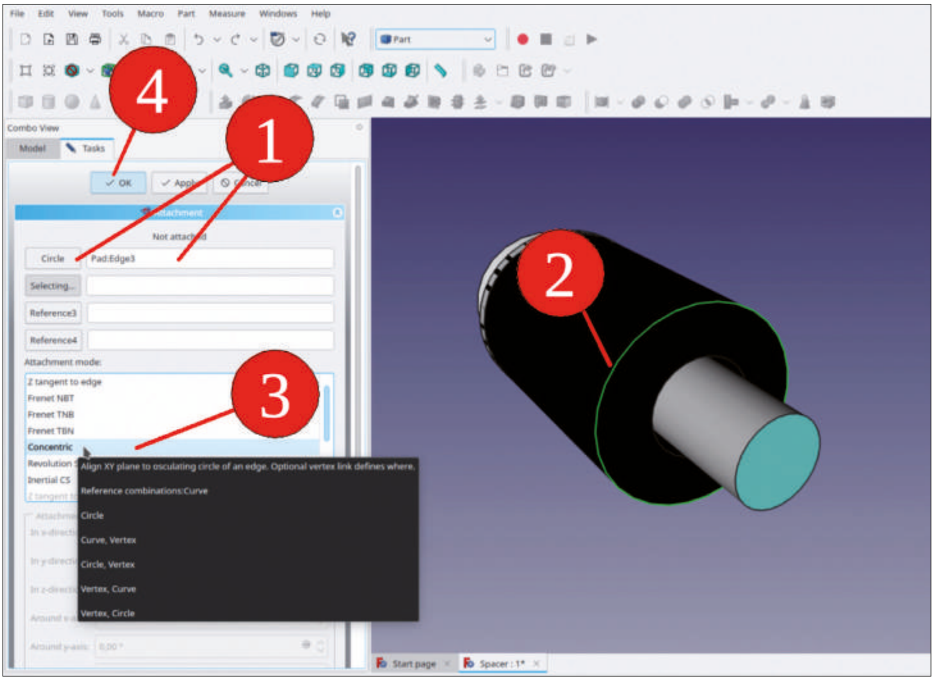


Figure 4-15

Upon opening the task window, the collector for 'Reference1' is already activated (the button label reads 'Selecting…'). Click into the edit field next to it, which then receives the focus (i.e., gets a blue rim). In the 3D view, click the rim of the spacer, which is opposite to the head of the screw, where the washer should go to (Figure 4-5). In the task window, select the attachment mode 'Concentric' (Figure 4-15, step 3) and close the task window with the 'OK' button. This brings the washer already close to the end of the spacer (Figure 4-16). Sometimes, the selection of the reference in the 'Attachment' task window is not functioning right away (maybe something else was clicked in between the steps). Then, the collector needs to be activated again, by clicking its button until its color is dark gray, and the label reads 'Selecting…' again. After that, click the edit field next to it, to grant it the focus (indicated by the blue rim). Only then, the collector is ready to receive a feature.

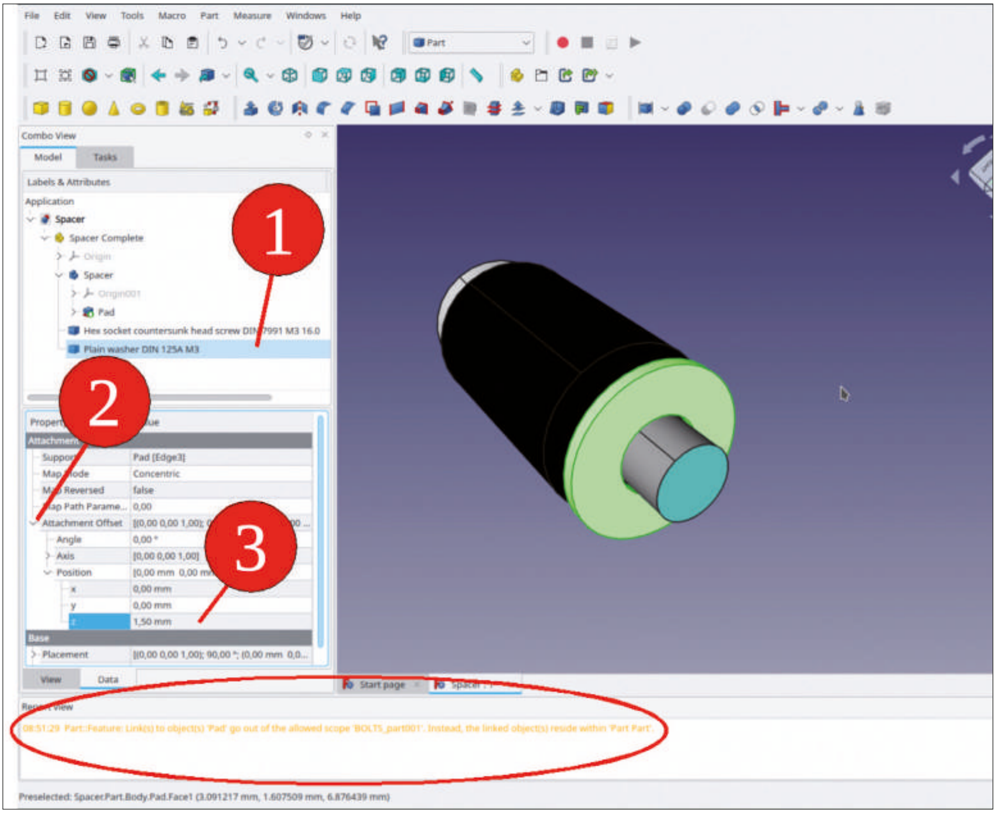


Figure 4-16

10. Often, a circuit board is mounted with the spacer. It is therefore useful to define a default attachment Z offset, which accounts for the board thickness. This usually amounts to 1.5 mm (or 1.55 mm, depending on the unit base). The parameter can be accessed by expanding the 'Attachment Offset' property in the Combo View (Figure 4-16, number 2).

When you decided to enter the offset in the 'Attachment' task window already, the 3D view is updated when you either roll the mouse wheel with one of the parameters selected or click either the 'Apply' or 'OK' button, the latter closing the task window.

If a small blue check mark appears on the tree view item of the washer, the component has not been recalculated yet. Then, trigger the recalculation with the F5 key. A warning message is issued: (Figure 4-16; output window at the bottom):

```
'Part::Feature: Link(s) to object(s) 'Pad' go out of the allowed scope 'BOLTS_
part001'. Instead, the linked object(s) reside within 'Part Part'.'
```

This is (just) a warning but flags a formal shortcoming of the defined relation. You can inspect what's going on by selecting 'Tools | Dependency graph' from the main menu (Figure 4-17).

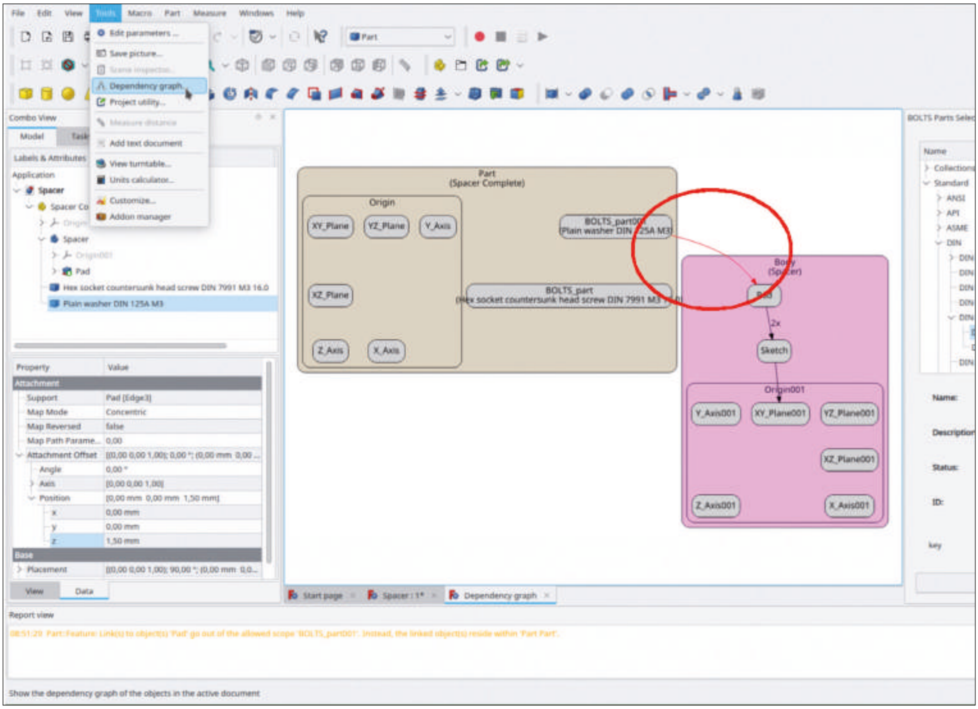


Figure 4-17

The dependency graph shows that the body 'Spacer' resides in its own, enclosed name space (scope). The arrow for the offending relation between the washer and spacer is displayed in red.

It is better to correct the problem right away. A valid reference is needed across the scope boundaries. For this purpose, you already met with two objects in Chapter 3, the two flavors of shape binders.

As another useful addition, a datum plane controlling the spacer length will help later on with the attachment of other parts:

11. Close the dependency graph and return to the 'Model' tab.

12. To be able to control the spacer length by it, the datum plane needs to be created in the 'Spacer' body. The 'Spacer' body must be activated to do this, otherwise the datum plane is created in the currently activated entity, which could hide somewhere else. Activate 'Spacer' by double-clicking its item line in the tree view. Expand the coordinate system in it and select the XY plane by clicking the item line (Figure 4-18, number 1), as the alignment of the datum plane should be parallel to it.

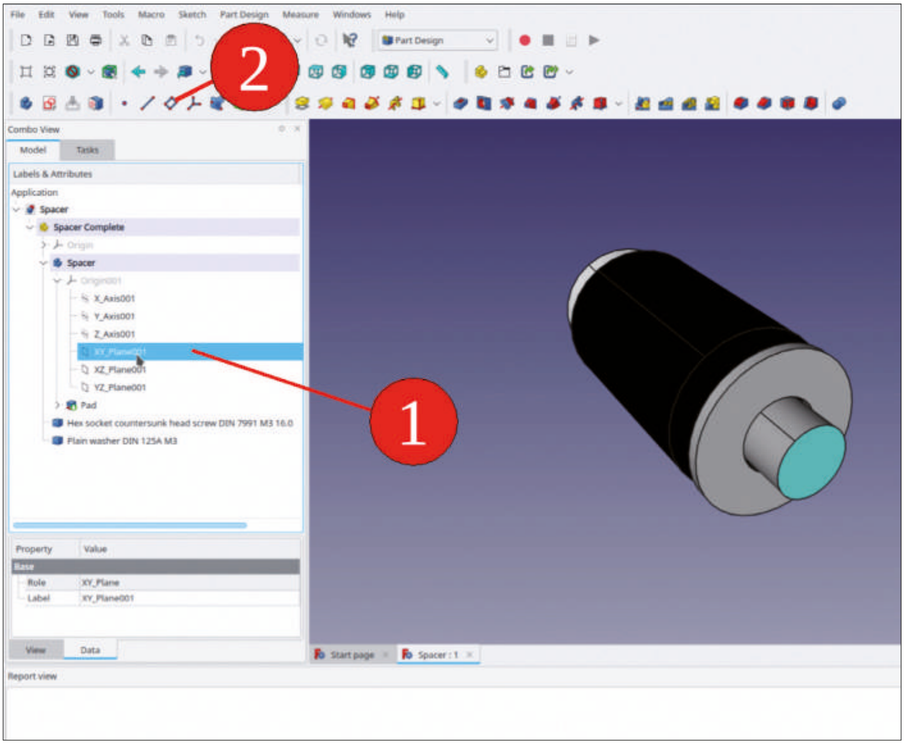


Figure 4-18

Click on the 'Create a Datum Plane' tool button (Figure 4-18, number 2). A task window opens. Because the XY plane was marked before the task was started, the plane is already attached to it and the attachment mode is preset to 'Plane Face' (Figure 4-19). Set the attachment offset in the z-direction to 10 mm, i.e., the intended length of the spacer. Then close the task window with the 'OK' button (top of window). The trick is now to define the spacer length with respect to the new datum plane and refer the washer to the plane position. Then, the washer will diligently follow length changes of the spacer. This looks a bit indirect but provides a rock solid reference to the spacer end face.

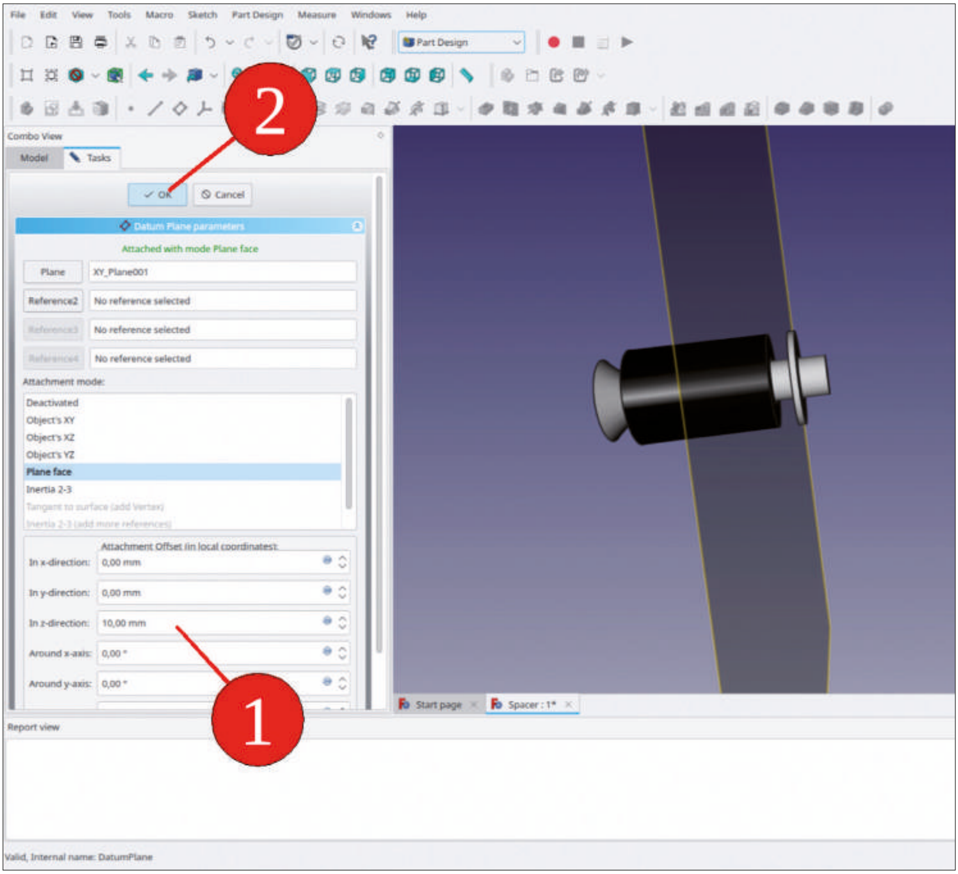


Figure 4-19

13. In the tree view, the new datum plane appears in the body 'Spacer' as 'Datum-Plane'. If a number of them piles up, that can become confusing, so rename it to 'Spacer Length'. This helps later to catch up with the ideas again (Figure 4-20).

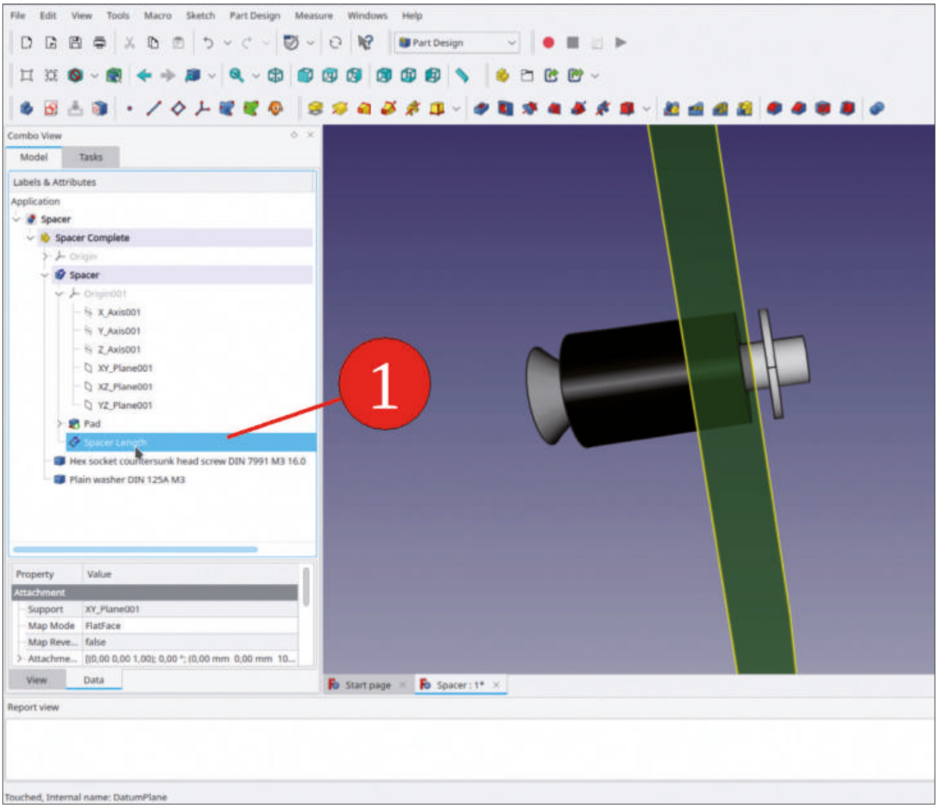


Figure 4-20

14. Now, in order to set the associative connection, the length of the spacer needs to be controlled by this new datum plane. Click twice on the tip (the last design state) of the spacer in the body 'Spacer' to edit it. It is listed as 'Pad'. The task window of the pad opens. Select for the 'Type' 'Up to face' from the combo box. By this selection, the collector 'Select face' already has a focus (dark gray). Then, click the yellow datum plane in the 3D view to select it. The term 'Datum-Plane' appears in edit field of the collector (Figure 4-21). Close the task window with the OK button (on top). Hide the datum plane by marking it in the tree view and pressing the SPACE key.

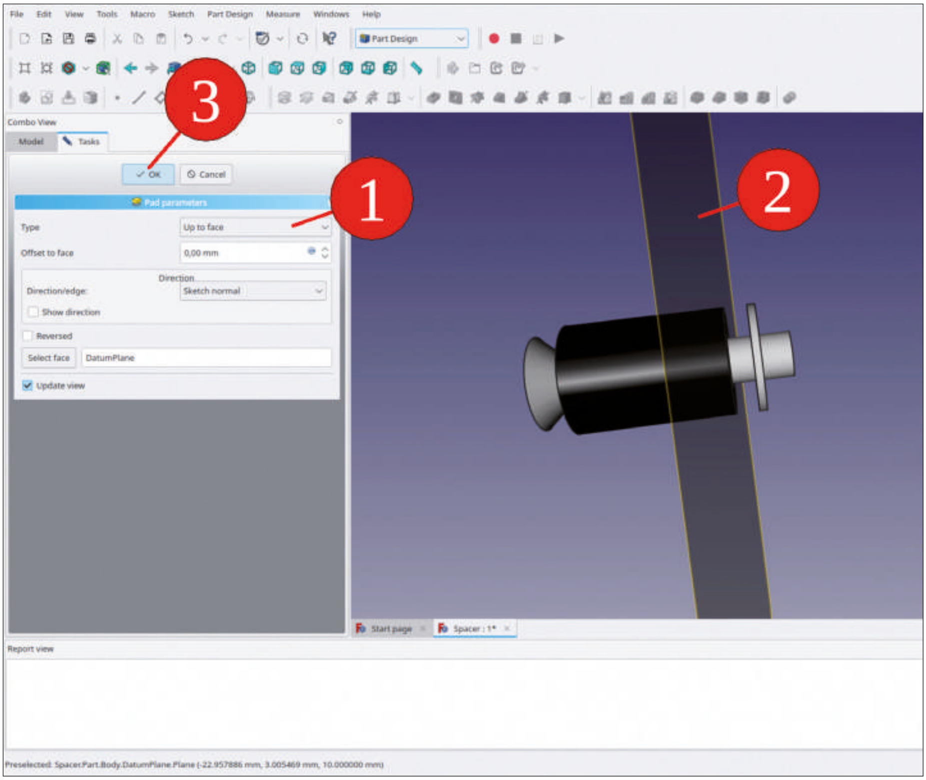


Figure 4-21

15. Next, the SubShapeBinder can be created. From the last step, the body 'Spacer' should be still activated (its title been shown in bold in the tree view). In the 3D view, mark the boundary of the spacer as shown in Figure 4-22 and click the green 'Create SubShapeBinder' tool button in the workbench menu. (If you would want to fix the SubShapeBinder even more rigorously, you could sketch a circle onto the new datum plane, which coincides with the spacer, and use that as input for the SubShapeBinder).

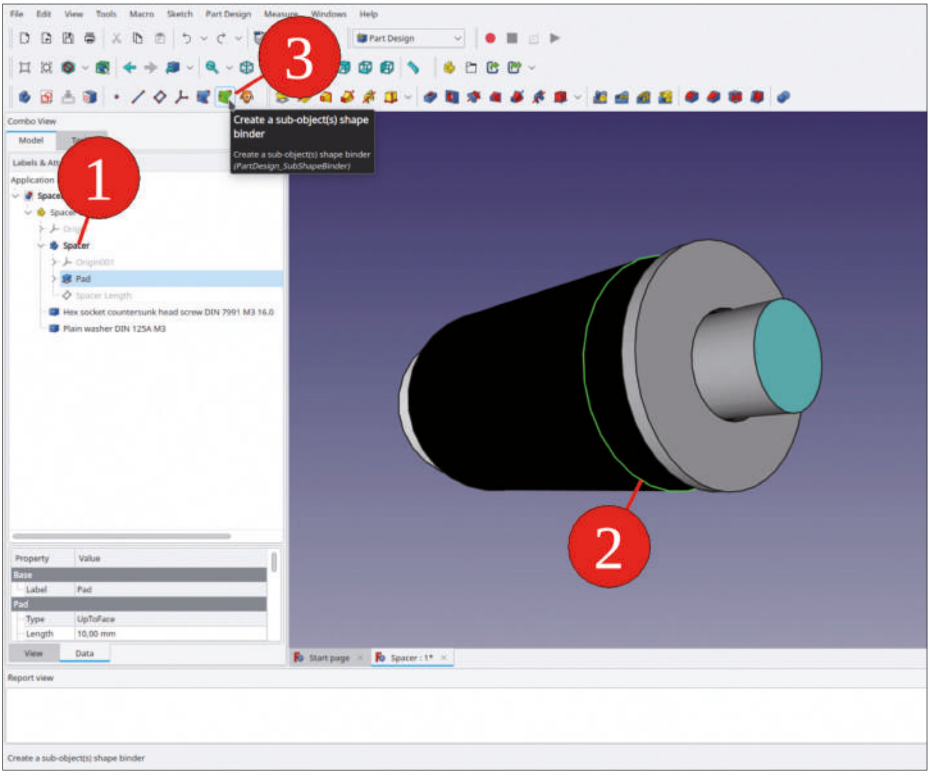


Figure 4-22

16. In the tree view, rename the new SubShapeBinder as 'Spacer End Face'.

Instead of the green SubShapeBinder, the blue Shape Binder could have been used. In the context of a single Std-Part-Container, this works just as well. However, complications can arise when the Std-Part-Container is copied into assemblies later on. Then, the recalculation of the global coordinates will fail, as described in Chapter 3.

17. In order to allow other body objects in 'Spacer Complete' to refer to the new SubShapeBinder, drag and drop it into the Std-Part-Container 'Spacer Complete'. It is then listed at the same hierarchical level as the washer and screw (Figure 4-23).

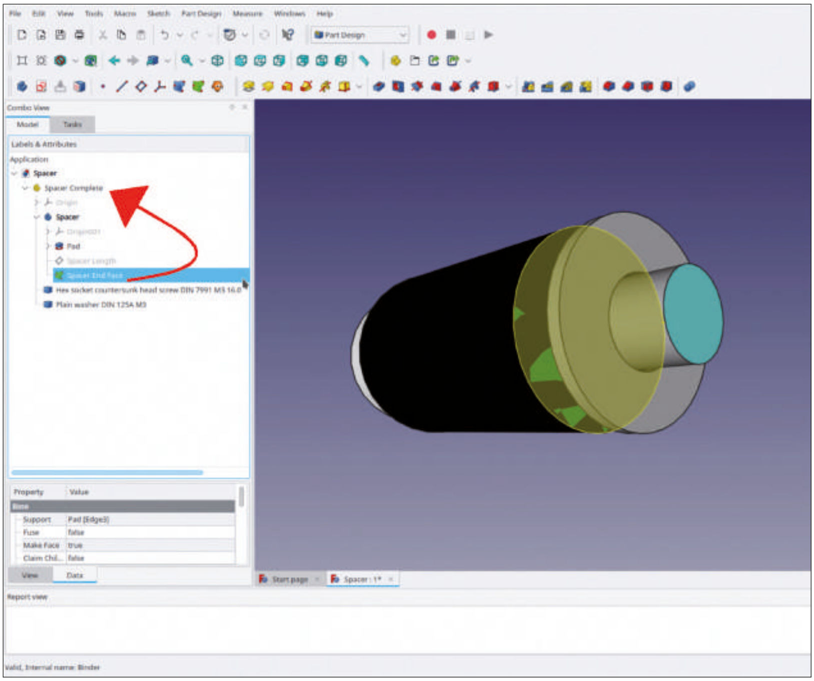


Figure 4-23

18. Now, the attachment of the washer needs to be repaired. Click on the washer in the tree view, and on to the property field 'Support' (Figure 4-24).

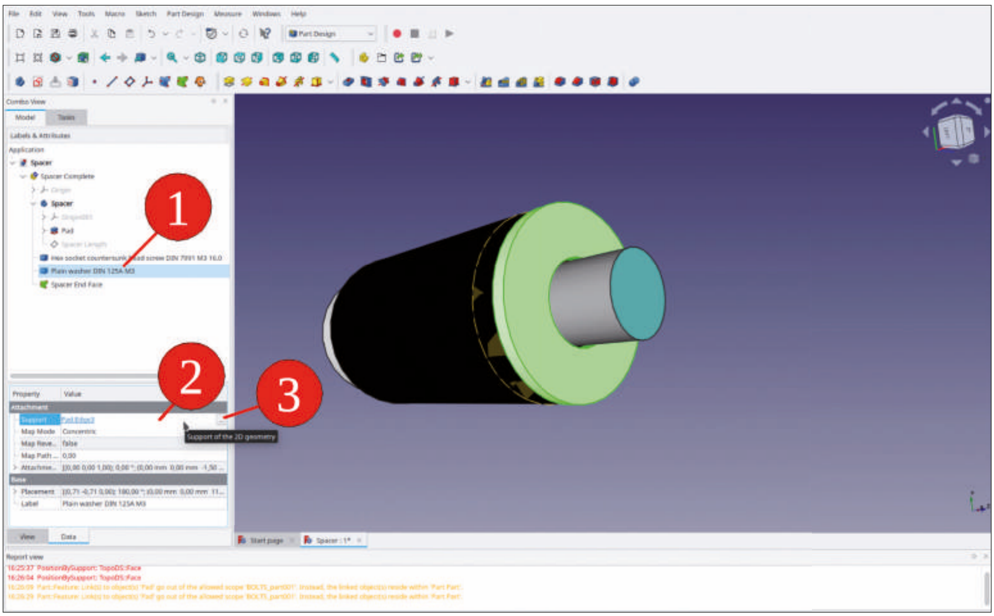


Figure 4-24

Clicking the [...] button on the right-hand side of the edit field opens a selection dialog in which the object 'Pad' appears still marked. Click on the 'Clear' button to remove this selection (Figure 4-25) and mark the SubShapeBinder 'Spacer End Face' instead (Figure 4-26). Then, close the list with the 'OK' button.

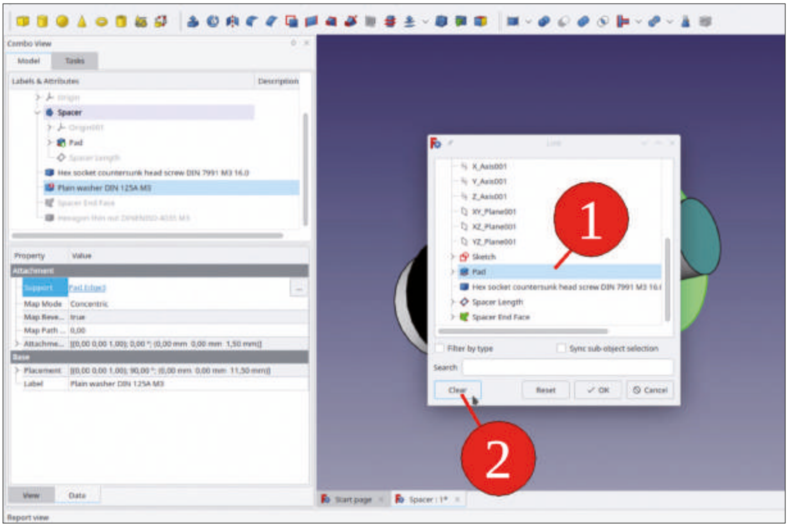


Figure 4-25

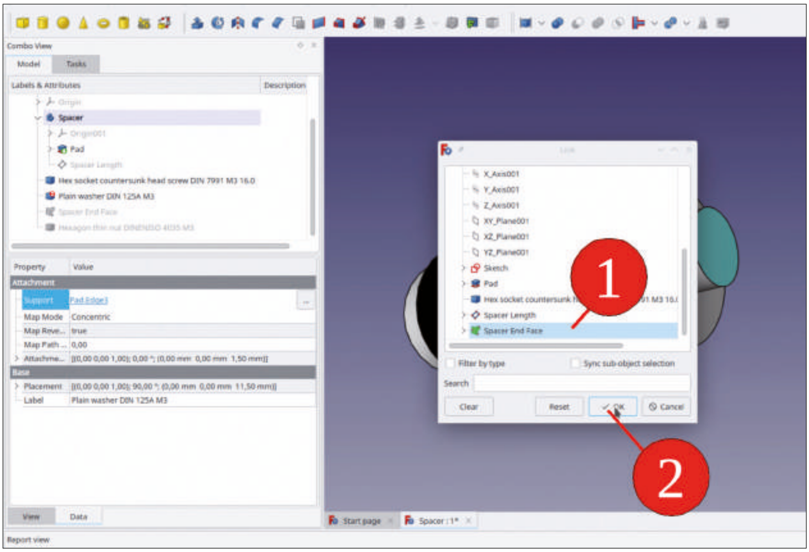


Figure 4-26

19. In the Property list, click the entry field for 'Map Mode', and then on to the [...] button appearing on the right-hand side. This opens a task window. There, the attachment mode 'Concentric' is still selected, which makes no sense with respect to a plane. From the list, select 'XY on plane' instead (Figure 4-27).

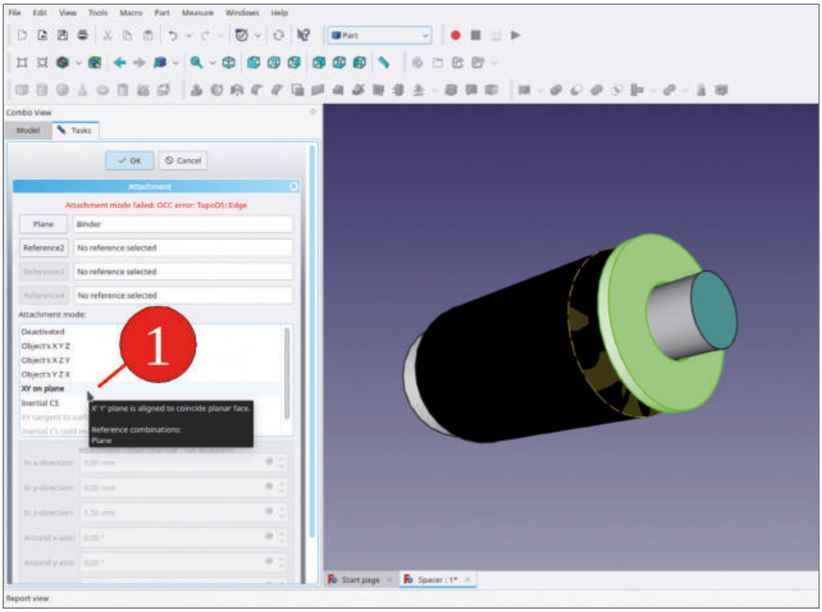


Figure 4-27

To allow the circuit board to be inserted between spacer and washer later, select a z offset of 1.5 mm (Figure 4-28).

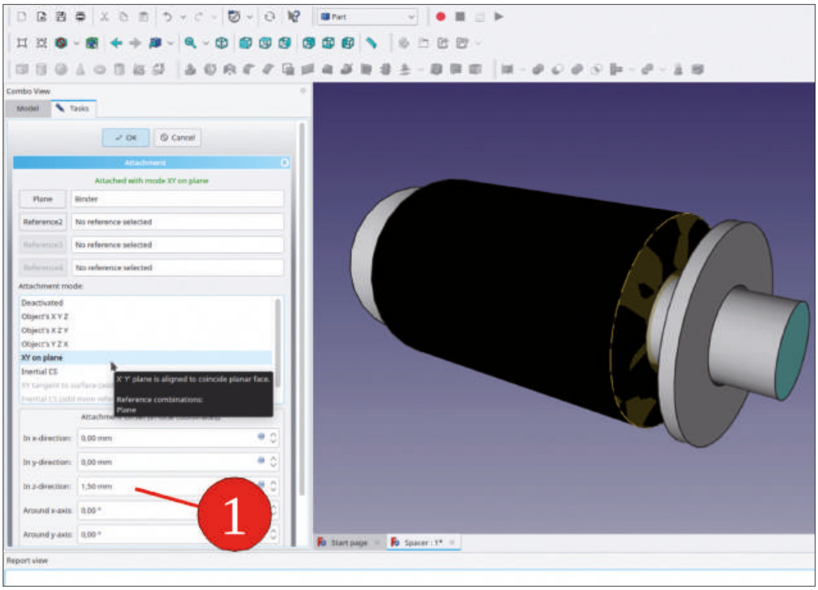


Figure 4-28

After closing the task window with the 'OK' button (top), the washer appears at the correct position.

20. In order to check whether the scope violation has been rectified, clear the output by right-clicking the selection of 'Clear' from the context menu. Then, right-click the Std-Part-Container and select 'Recompute object' from the context menu. No new warnings should appear in the output window.

21. Now, add the hex nut: From the BOLTS Parts selector, pick 'DINENISO | DINEN-ISO 4035 thin hexagon nut, metric sizes' from the list. The M3 key is already preset. Click the 'Add part' button to insert the nut into the project. Drag and drop the nut into the Std-Part-Container 'Spacer Complete'.

22. The nut is usually attached to the corresponding washer. To define the attachment, mark the nut in the tree view and switch to the 'Part'-workbench. From the main menu, select 'Part | Attachment'. The task window opens with a previously activated collector for Reference1. Click the top edge of the washer (e.g., the inner edge, Figure 4-29, step 1). For the attachment mode, select 'Concentric'. Then, close the task window with the 'OK' button. The nut now appears at the correct location.

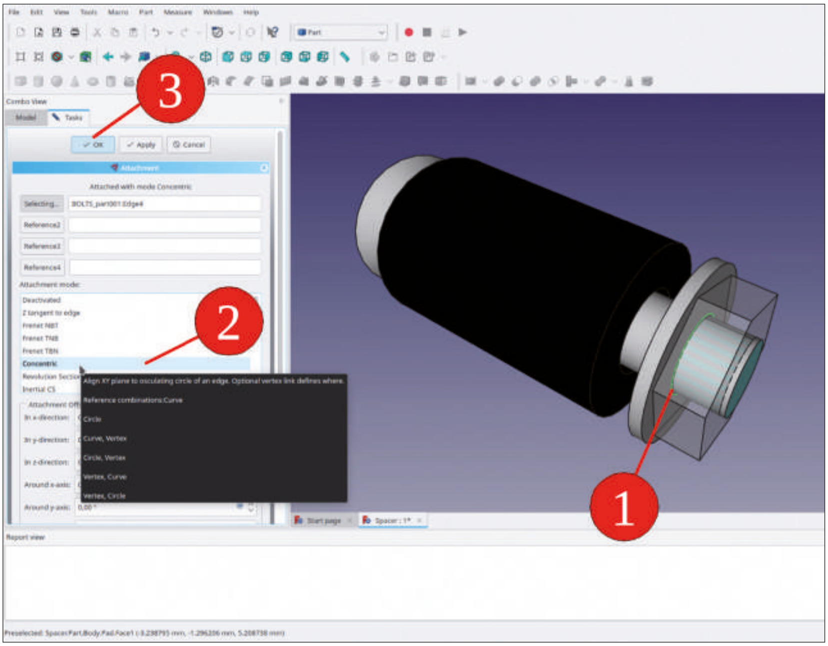


Figure 4-29

23. The associativity can be evaluated by changing the position of the datum plane 'Spacer Length'. In the tree view, double-click the datum plane, and change the attachment z offset to, for example, 5 mm. This time, the 3D view will not follow in live fashion. After closing the task window with the 'OK' button, there is still a recalculation needed in order to update the 3D view (Figure 4-30).

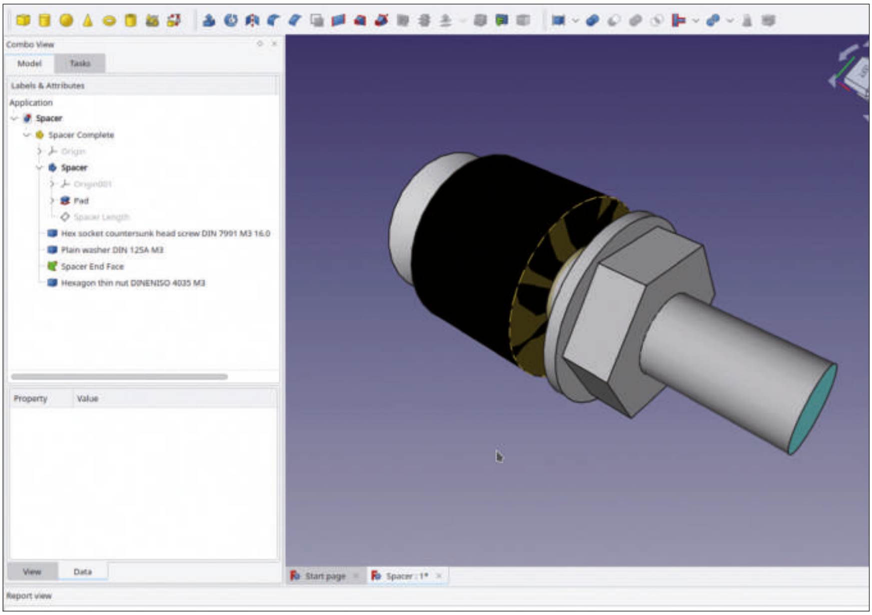


Figure 4-30

24. In the final step, set the length of the spacer to 10 mm again and hide the Sub-ShapeBinder 'Spacer End Face' by selecting it in the 3D view and pressing the SPACE key. Now, the spacer is ready for use (Figure 4-31).

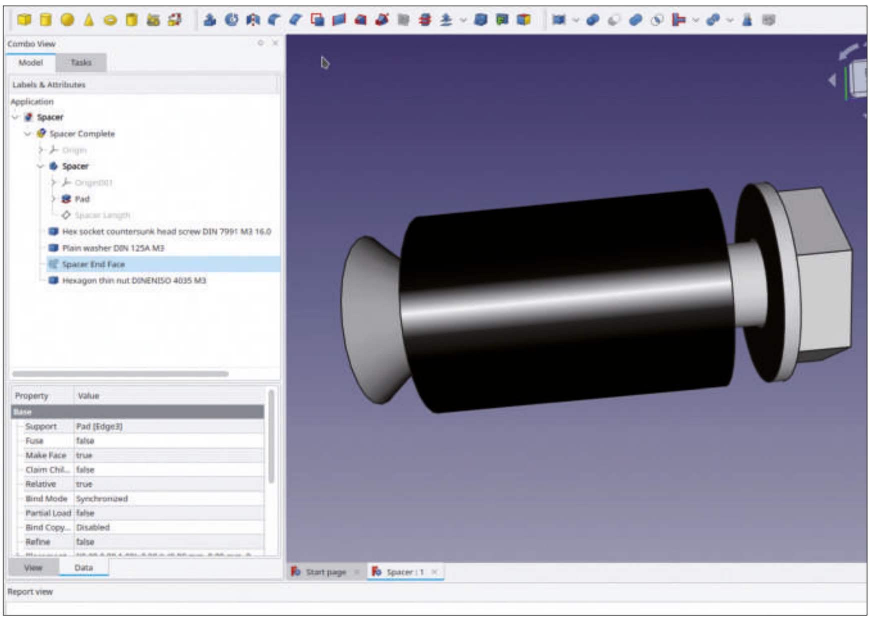


Figure 4-31

The procedure may look complicated, especially when following the step-by-step description. It is, however, not particularly difficult. The associativity will pay off later, for example, when components are fitted into an enclosure, or a protective cover needs to be added to a display — in that case, the length of spacers might need adapting in some cases.

On top of that, the encapsulation of parts and fasteners later allows the insertion in a single step, by means of cut-and-paste.

4.5 Parts Packaged with Fasteners Save Time

With the spacer, it became evident that grouping the parts together with their accessories in a Std-Part-Container can be a big time saver. This is especially true if a part is introduced in a design multiple times, like a 4-mm banana jack with its with insulating bushing and nuts. If a standard thickness for the panels is known in advance, the fasteners can be preset to the correct positions right away. But it is not difficult to edit the positions of these parts later on, in order to 'secure' the hardware.

4.6 Displaying and Hiding Elements

Sometimes, elements like a datum plane can be disturbing. It is easy to hide objects by selecting them in the tree view and by pressing the SPACE key. This is useful not only during a visual inspection of an assembly, but also when engravings and lettering have to be added to surfaces, as described below.

4.7 More Examples of Compound Parts

For the example projects described in this book, further components are needed. In Appendices A through G, the creation of the following components is described in step-by-step instructions:

- The Rotary Switch
- The Potentiometer
- The (4 mm) Banana Jack
- The Pilot Lamp
- The Toggle Switch
- The IEC Power Inlet
- The 9 V Battery (PP3; 6LR61; 'Block')

All other components in the example projects are available with their supplementary materials online and can be inspected for their specific way of making. Working with the Sketcher is covered in a multitude of instructions both online and in books. Some information about these sources is given in Chapter 11, *Community Resources*.

Chapter 5 • Working with Sheet Metal

The 'Sheet Metal' workbench is not included with the standard functionality of FreeCAD. It can, however, be smoothly installed with the Addon Manager — if that wasn't done already while digesting Chapter 2.

5.1 Example: 9 V 6LR61 Battery Holder

For the 6LR61 (a.k.a. 'Block' or 'PP3') battery holder, a representation is needed first. With some excess tolerance in the dimensions of that model, you can make sure that the real battery will fit into the finished holder with not too much pressure exerted onto the casing. The step-by-step instructions to create the battery model are given in Appendix G.

The battery model is copied as a Std-Part-Container into the Std-Part-Container of the holder. The battery holder is thus already a mini-Assembly in its own right, with the nested Std-Part-Container of the battery (not forgetting … the fasteners!).

The usefulness of this project organization method is evident: All parts for the battery 'complex' come together in one Container. They are not scattered all over the tree view, but nicely sorted into their branch. Also, when the holder has been mounted, one click, and the SPACE key displays the contained battery and the necessary collection of other little additions, too. This saves a lot of time and meddling, especially when the component is inserted more than once. Also, this grouping of components again has the outer form of an Std-Part-Container, which can reappear as a node in a more complex assembly again. And again...

1. The first steps follow the standard procedure: Create a new file, rename it to '9V Battery Holder', generate the parent Std-Part-Container, and designate it '9V Battery Holder Complete'.

2. In the next step, insert the battery which has been created following the steps in Appendix G. If you decide to skip the battery model creation, you find the file in the online materials. The insertion of components by cut-and-paste is used quite extensively in this book.

3. Locate and open the file '9V Block Battery'. Now, both files are visible in the tree view. Click once the yellow Std-Part-Container '9V Block Battery Complete' and copy it with CTRL-C. A dialog window appears, in which you could also limit the selection of details that will be copied. Do not change the default selection and confirm the copy operation with the 'OK' button.

4. In order to activate the destination of the copied battery model, double-click the previously created new (and yet empty) Std-Part-Container 'Battery Holder Complete'. Paste the battery model by CTRL-V. If you forgot to activate the correct destination, the battery model may have been pasted to the root of the tree view. Then, drag-and-drop the model into 'Battery Holder Complete'. In the tree view, close the file '9V Block Battery' (right-click, 'Close' from the context

menu). These preparations result in a tree view shown in Figure 5-1. The typical accident happens when the model is pasted into the source document node again. Then, the pair should be deleted there, just to keep things in good order, or you could simply reverse the erroneous copy operation with CTRL-Z. Close the document node '9V Block Battery'.

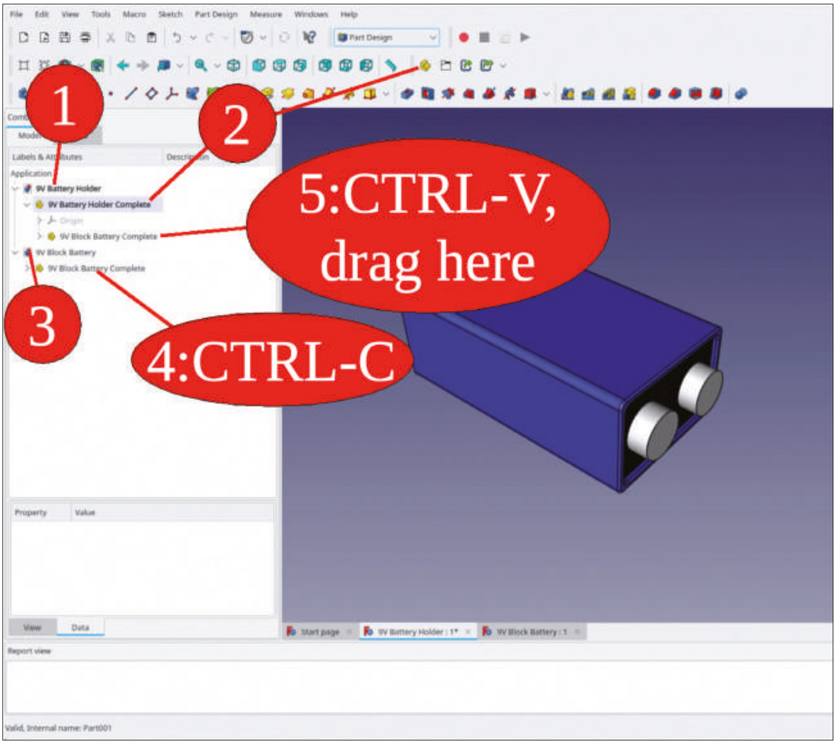


Figure 5-1

5. In order to start with the battery holder sheet metal, a reference to the top of the battery is needed. To identify the top face, you can briefly show the coordinate system of the parent Std-Part-Container. The reference will be a shape binder. To generate that in the correct location, activate the body 'Block Battery' (the title appears in bold letters if activated). Then, in the 3D view, mark the top face and click the 'Create a Shape Binder' tool button (Figure 5-2). Now, you have created a reference to a facet – which could break by re-enumeration. But you may allow for that because the battery can be considered as a established part, for which no further changes are expected. If you decide to add more elaborate contact representations later, the design could break! In such a case, more robust references with a datum plane and a sketch should be considered. For the sake of simplicity, that's skipped here.

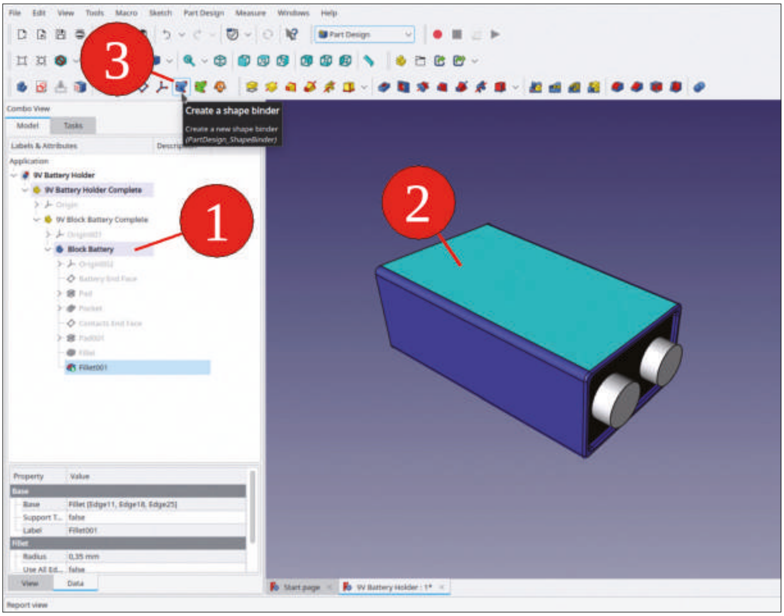


Figure 5-2

A task window opens, in which the object and geometry are already preset (Figure 5-3). Close the task window with the OK button. If details are well accessible in the 3D view, marking them before clicking a tool button can save time, because the selection then often appears correctly preset in the tool task windows.

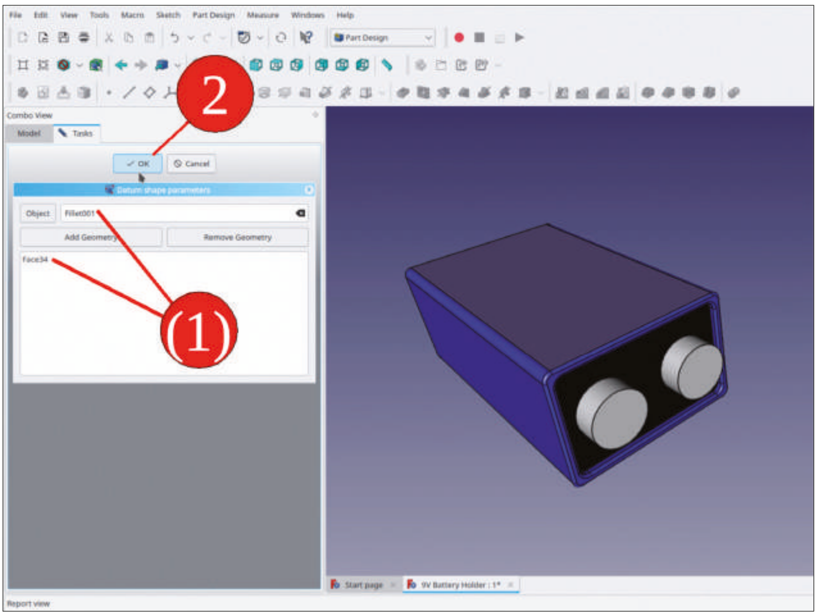


Figure 5-3

6. Rename the shape binder to 'Battery Top Face'. In order to make it accessible for other objects in the parent Std-Part-Container, drag and drop it into '9V Battery Holder Complete'. The Std-Part-Container '9V Battery Complete' can be collapsed in the tree view (Figure 5-4). Leave the trace support of this shape binder set to 'false', because you do not expect changes to the battery anymore.

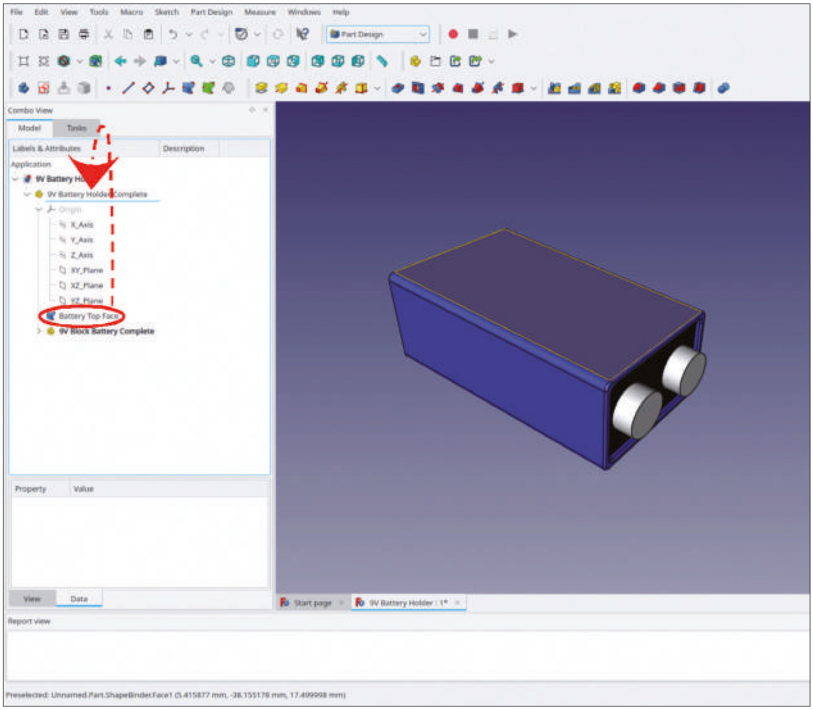


Figure 5-4

7. Activate the Std-Part-Container '9V Battery Holder Complete'. For our battery holder, click the blue 'Create Body' tool button and rename the new body to 'Holder Sheet Metal' (Figure 5-5).

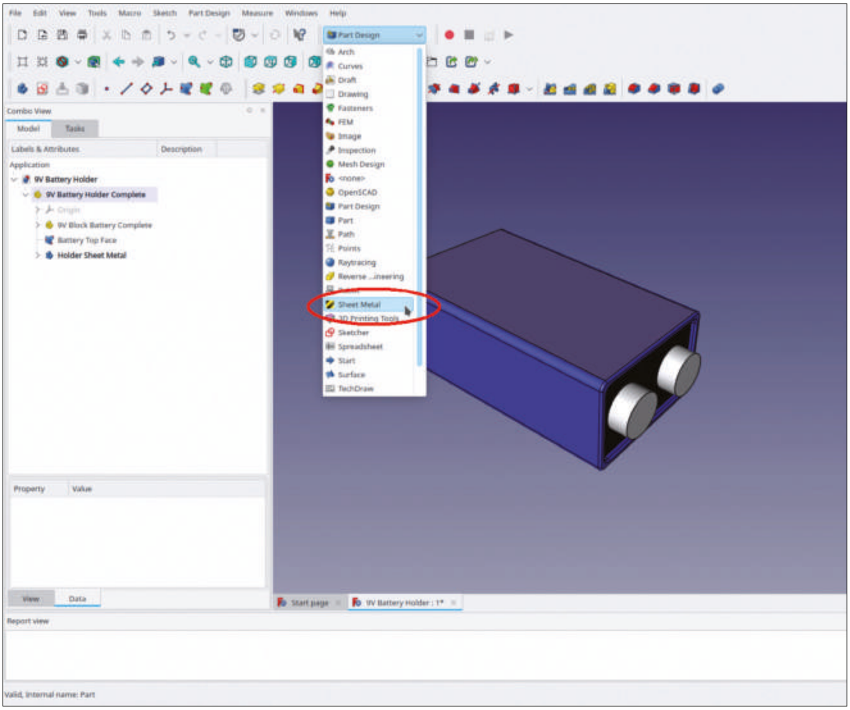


Figure 5-5

8. Switch to the 'Sheet Metal' workbench (Figure 5-5).

9. In the tree view, mark the shape binder 'Battery Top Face' and click the 'Make Base Wall' tool button (Figure 5-6). In the tree view, expand the body 'Holder Sheet Metal' and mark the new object 'base bend'. Insert a value of 1.5 mm into both property fields 'radius' and 'thickness' (Figure 5-6).

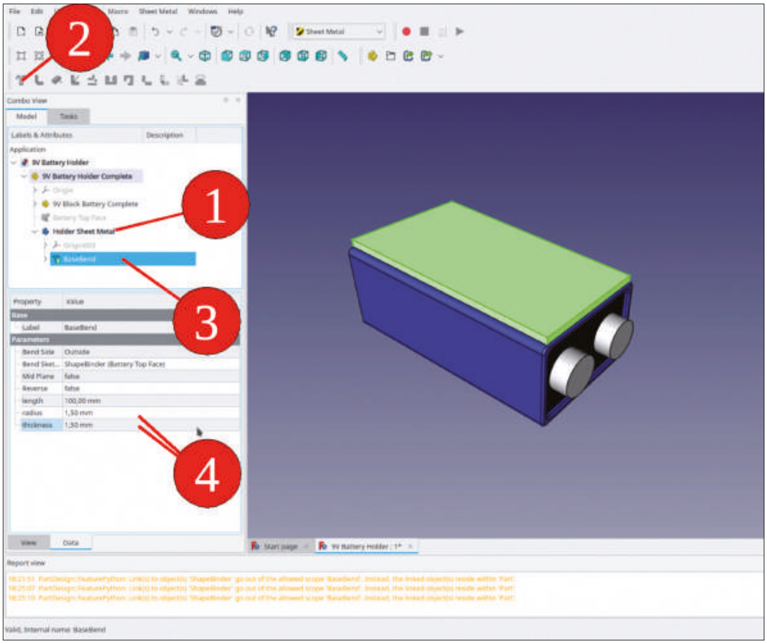


Figure 5-6

10. Select one top edge of the new sheet metal piece and click the 'Make Wall' tool button (Figure 5-7).

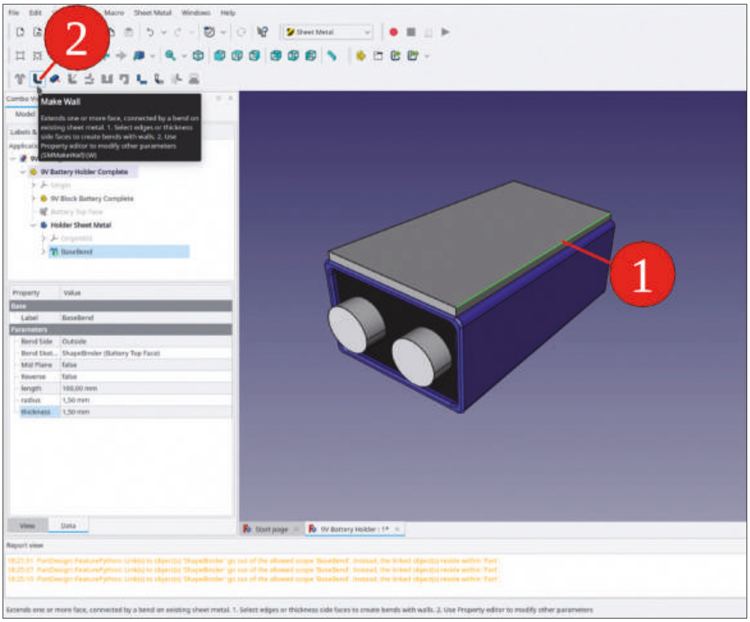


Figure 5-7

11. In the tree view, click the new wall 'Bend' and set its length property to 14 mm (Figure 5-8).

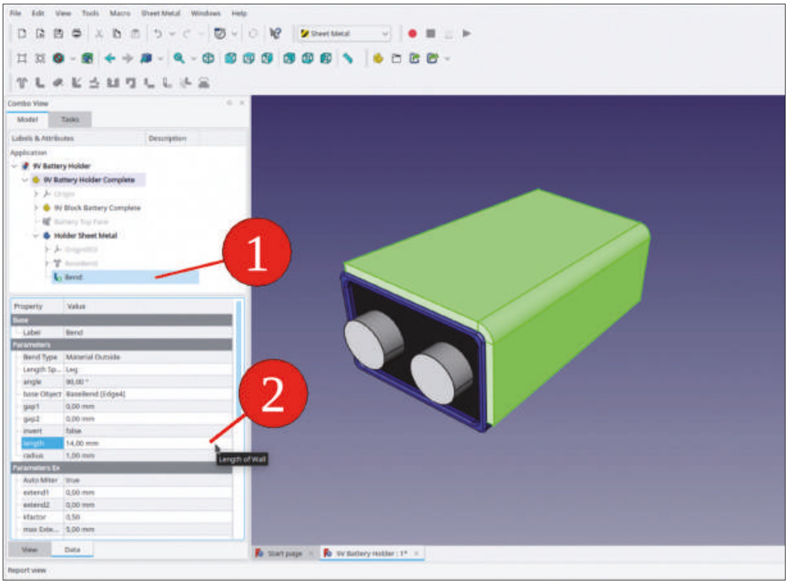


Figure 5-8

12. In the 3D view, mark the outer, lower edge of the sheet metal piece and again click the 'Make Wall' tool button (Figure 5-9).

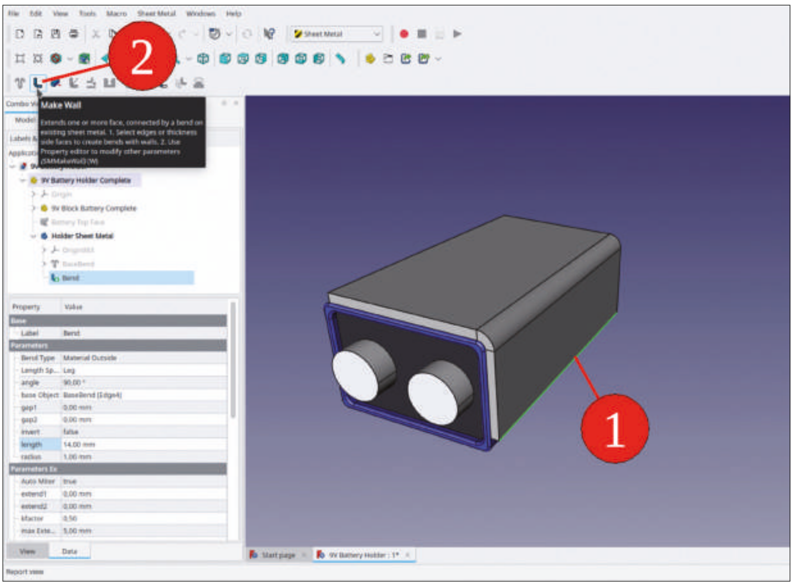


Figure 5-9

13. In the tree view, mark the new wall ('Bend001') and set the 'Invert' property to 'true'. For the length, insert 8 mm (Figure 5-10).

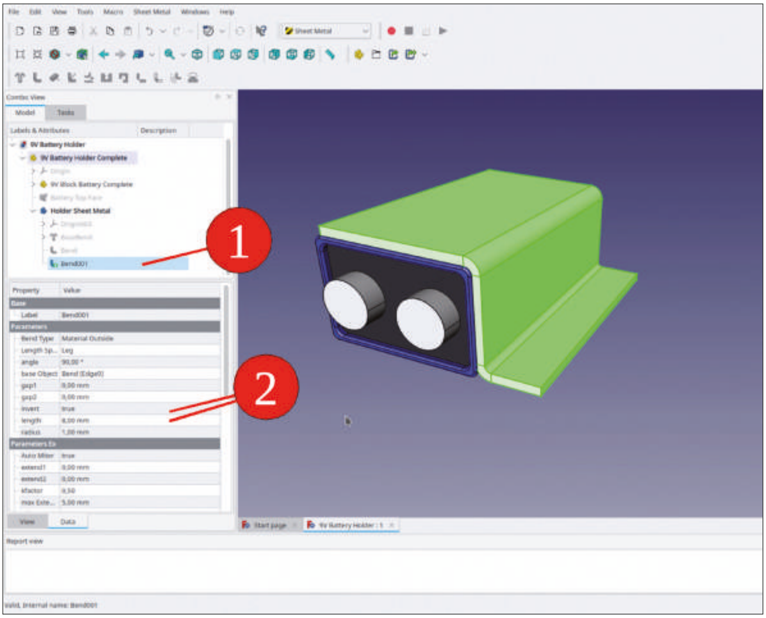


Figure 5-10

14. In the same way, attach two walls to the opposite side of the battery holder (Figure 5-11).

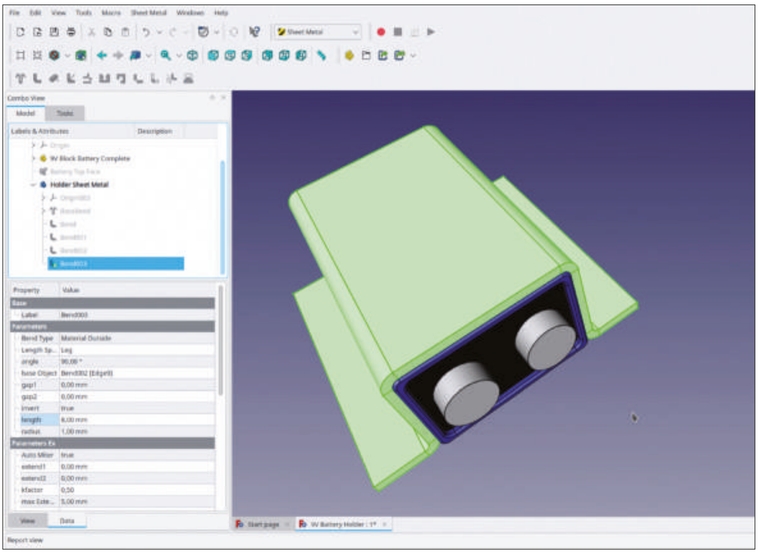


Figure 5-11

15. On the back side of the holder, a little stop would define the position of the battery nicely. Mark the back edge on top and generate a small bend there (Figure 5-12).

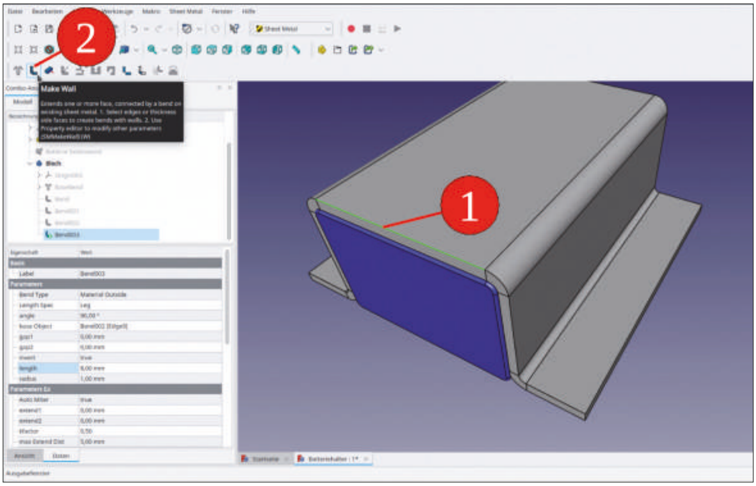


Figure 5-12

16. In the tree view, click the new wall to display its properties. In the property list, enter the following property values:
Length = 3 mm; gap1 = gap2 = 2 mm (these are the lateral distances to the edges); reliefd = reliefw = 0.00 mm (these are cuts to reduce strain when bending the part, which you do not use here, Figure 5-13). To find the relief properties, scroll down the property list.

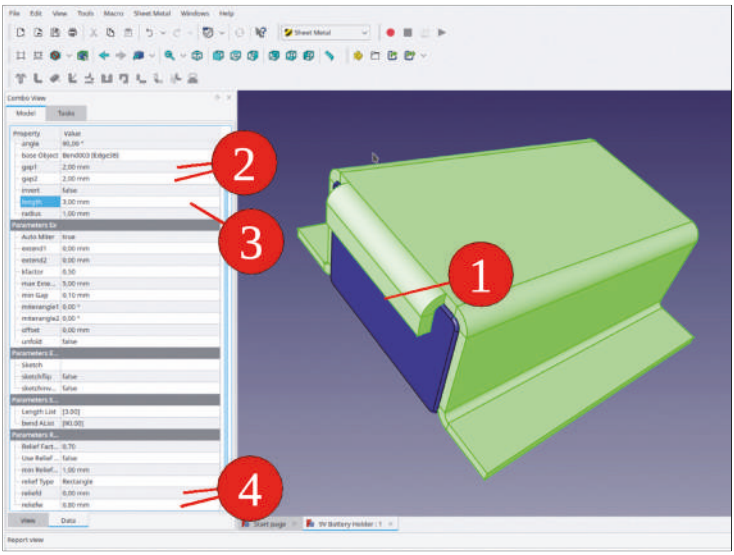


Figure 5-13

The sheet metal creation is finished at this point. Switch back to the 'Part Design' workbench to add some more features, which make the work with the part more agreeable, like some rounded edges. Also, the mounting holes need to be added. These will be addressed next:

17. The mounting hole will be sketched first. Click on the sketcher tool button and select the XY plane (on which the battery rests). Close the sketch and reopen it by double-clicking its icon in the tree view (this will show the previously generated geometry). To eliminate distortions by perspective, and to have a clearer view, select from the main menu 'Sketch | View Section' and 'View | Orthographic View'.

18. Click the 'Create Circle' tool button and draw a circle approximately where the mounting hole should go. No special precision is required with this drawing yet (Figure 5-14).

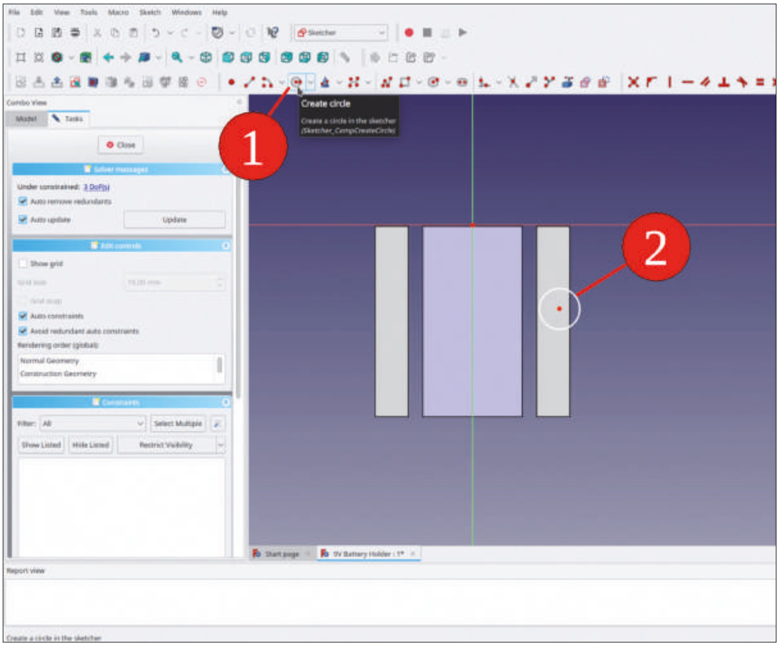


Figure 5-14

19. Click the 'External geometry' tool button and mark two edges of the sheet metal flange where the hole should be located on. The edges appear highlighted in violet, indicating that they are available as construction lines for the sketch (Figure 5-15).

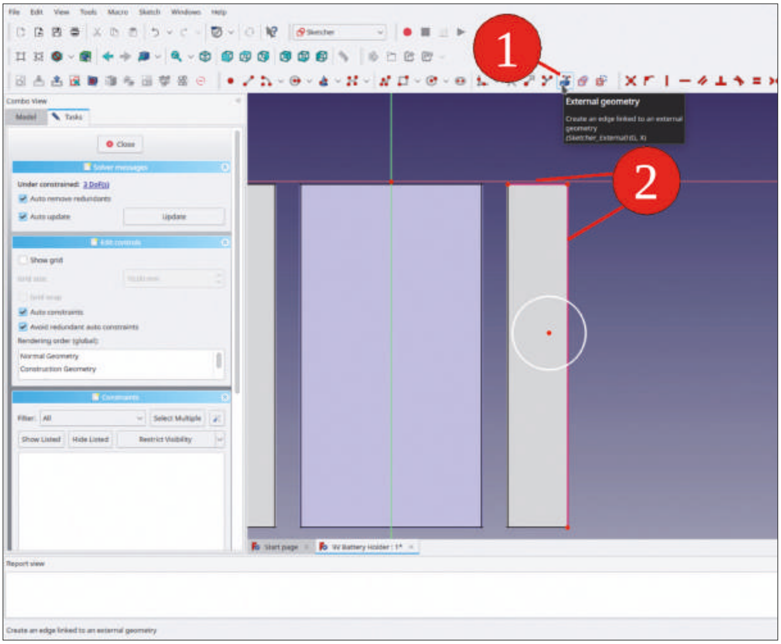


Figure 5-15

20. To place the circle in the center of the flange, mark two diagonally placed corner points of the new construction lines, and then the center of the circle. Clicking the 'Constrain symmetrical' tool button brings the circle to the center of the flange (Figure 5-16).

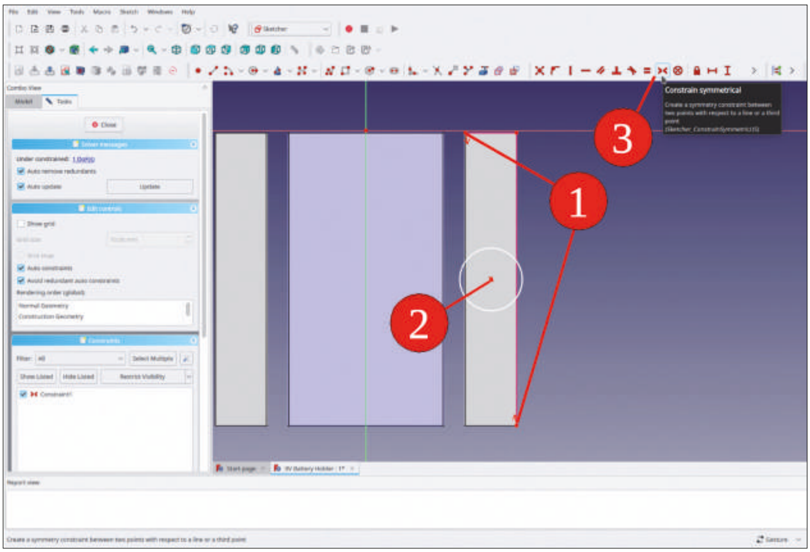


Figure 5-16

21. Right-click the circle in the 'Elements' list and select 'Diameter Constraint' from the context menu. Set the diameter to 3.2 mm (for an M3 screw, Figure 5-17). Close the sketch.

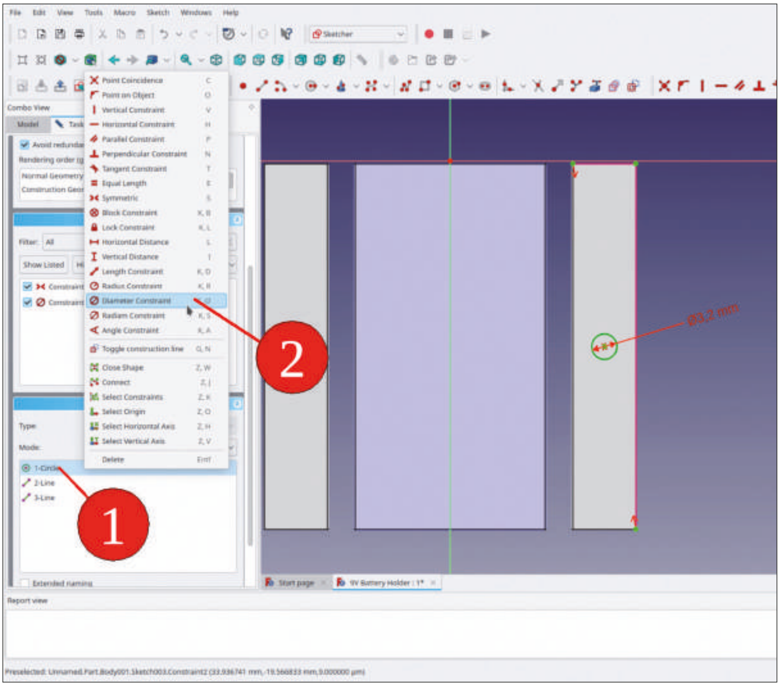


Figure 5-17

22. In the tree view, mark the new sketch and click the 'Pocket' tool button (Figure 5-18).

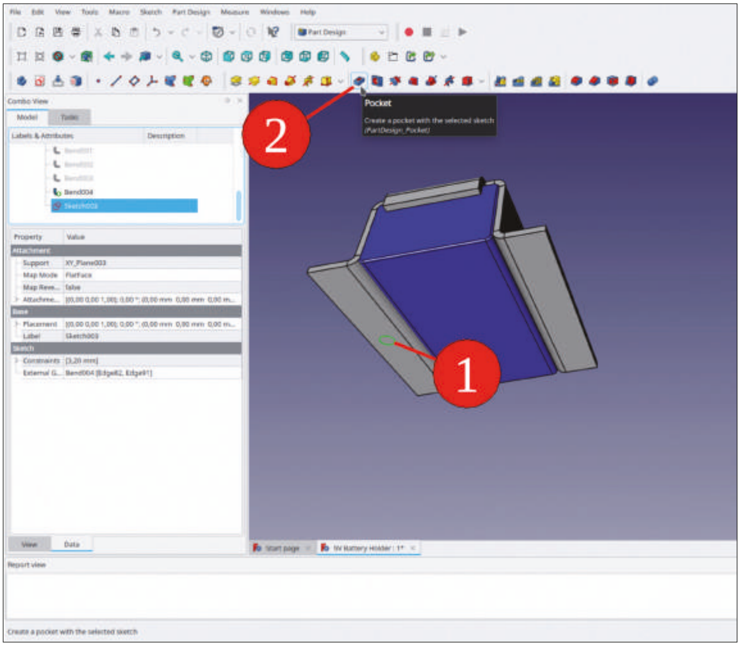


Figure 5-18

23. A task window for the pocket opens. Select 'Through all' for the type and check the 'Reversed' check mark (Figure 5-19).

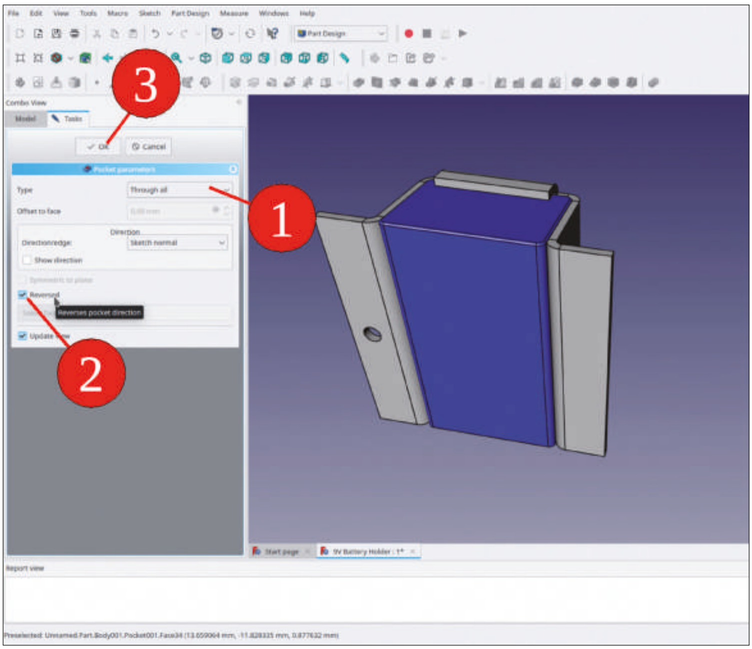


Figure 5-19

24. In the tree view, mark the tip and select 'Part Design | Apply a pattern | Mirrored' from the main menu (Figure 5-20).

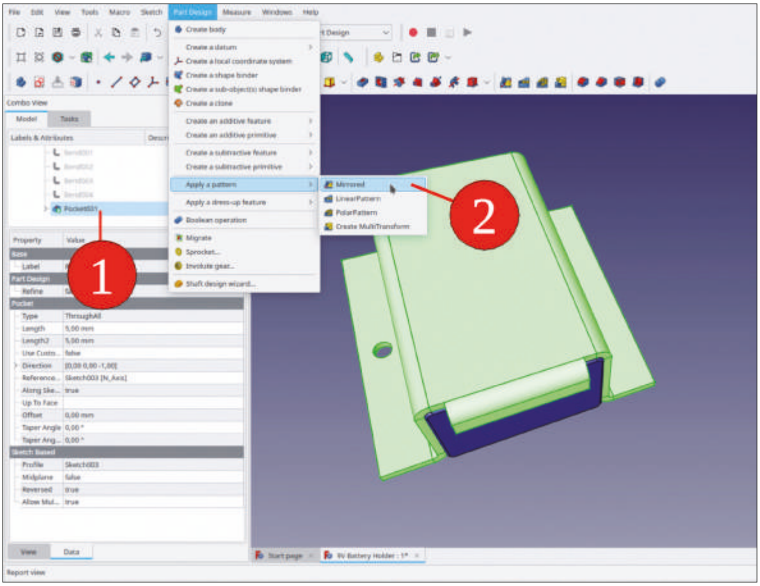


Figure 5-20

25. A task window opens, and possible mirror planes are shown in the 3D view. Select the YZ plane (Figure 5-21, step 1) and close the task window with the 'OK' button.

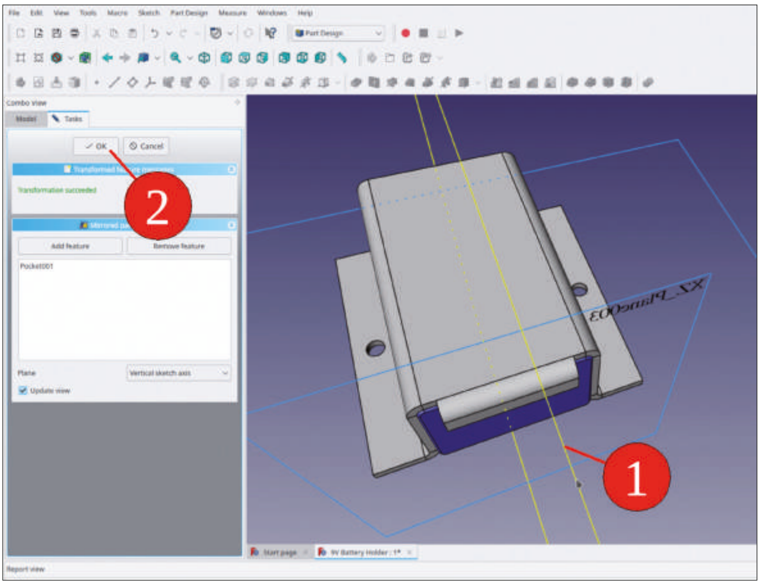


Figure 5-21

26. To avoid sharp corners on the part, mark the four outer edges of the sheet metal piece and select 'Fillet' (Figure 5-22).

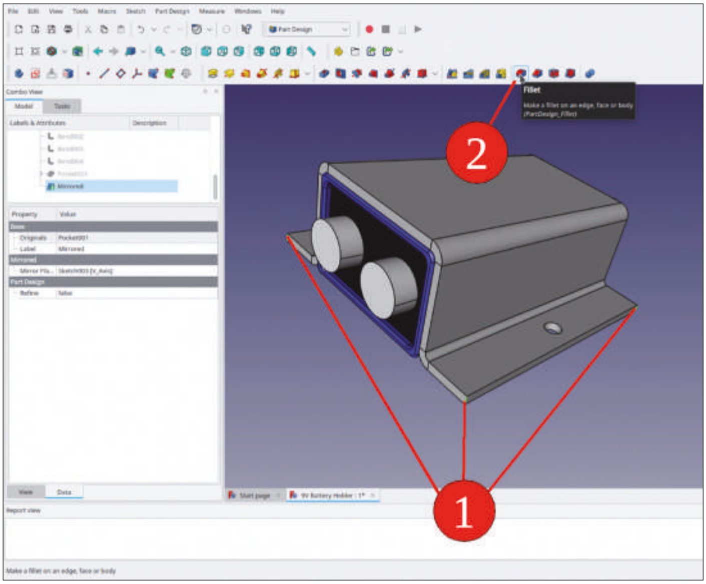


Figure 5-22

27. In the task window, enter for the radius: 5 mm (Figure 5-23), and close the task window.

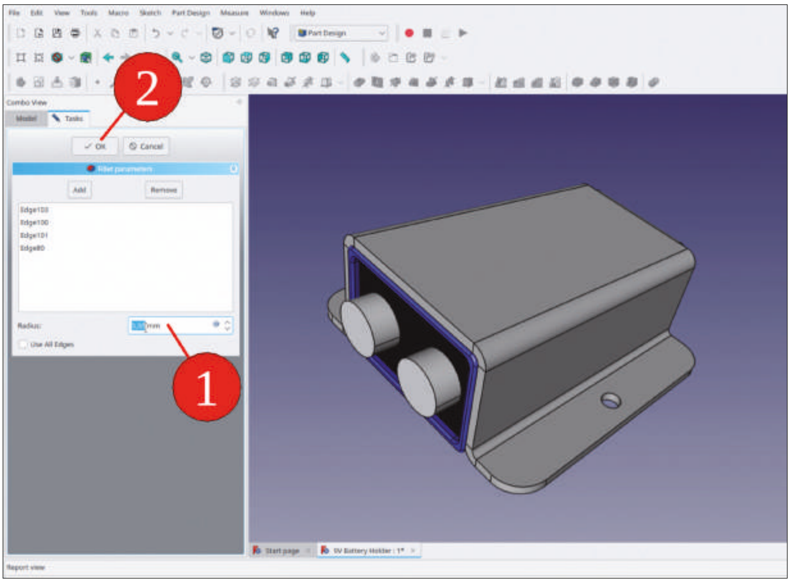


Figure 5-23

The sheet metal part is finished now. With the fasteners added, the holder would be all self-contained and ready to be inserted into an assembly. You can add the fasteners using the same methods described in the previous chapter. At this point, however, it is more important to cover some other steps required to actually manufacture the part.

5.2 Creating the Unfold

When a sheet metal part has been designed, during the first step of its making, it is cut out of a larger, plain sheet of material. In order to know the contour of the flattened part, you need to unfold the 3D design. Fortunately, there is functionality provided with the 'Sheet metal' workbench — this saves a lot of time and effort. Besides showing the flattened contour, the positions of the bends are also given, which are needed to bring the part into shape once it has been cut.

28. First, in the tree view, hide the battery so it's not disturbing the view of the next steps (mark it and press the SPACE key).

29. Switch to the 'Sheet Metal' workbench. In the 3D view, mark the top face of the battery holder. Then, click the 'Unfold' tool button (Figure 5-24).

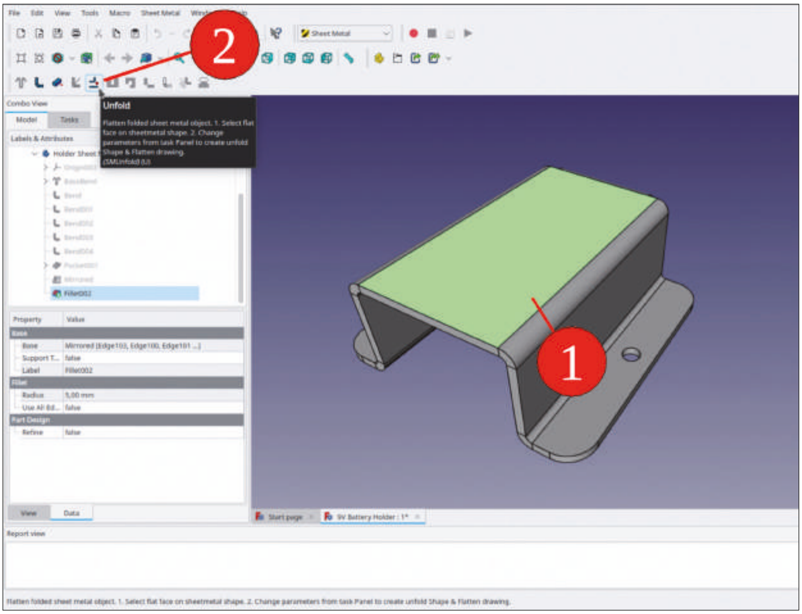


Figure 5-24

30. A task window opens. Check the box for 'Separate projection layers', select DIN, and leave the K factor at its default value (0.4). Then, close the task window with the OK button (Figure 5-25).

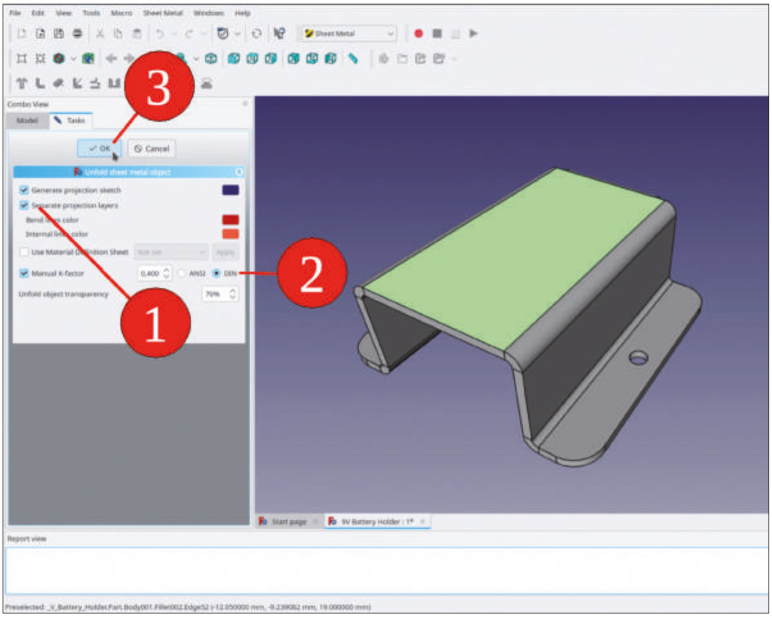


Figure 5-25

31. In the stem of the tree view, some new items appear (Figure 5-26). One is the object 'Unfold', which represents (in the 3D view transparent) a model of the flattened part. Furthermore, there are nicely sorted out sketches 'Unfold_Sketch_Outline (the outer contour), Unfold_Sketch_Internal (with all the internal openings of the sheet), and Unfold_Sketch_bends (the positions where the part needs to be folded). You may find this division useful because your milling machine has to add radius corrections that differ for outlines and internals.

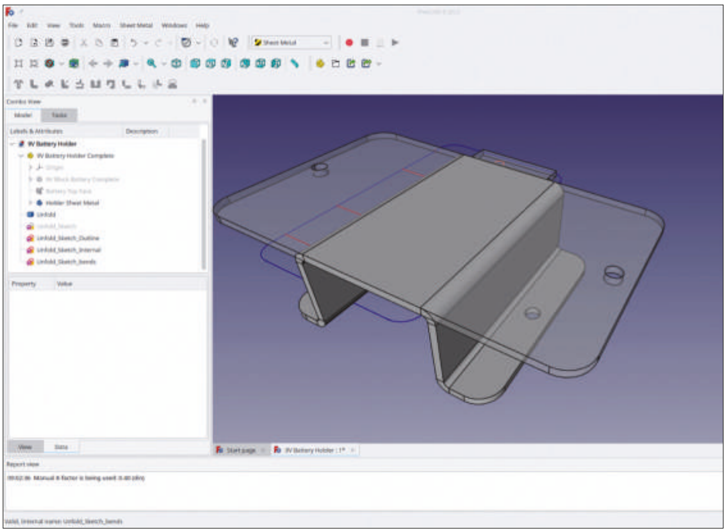


Figure 5-26

32. Now, consider if it makes sense to have the manufacturing data inside the Std-Part-Container of '9V Battery Holder Complete'. If you opt for this, drag and drop the required items into the container. This makes the file slightly larger but it is evident which data has been used to fabricate the prototype. On the other hand, you could always extract the data later, following the method described next.

5.3 Exporting the Unfold

For the actual making of parts, it is necessary to export the data into a useful, portable format. These days, many machine controllers can read DXF files. This is also true for, for example, NC EAS(Y)5 from the EAS company [EAS 2022]. Exporting and re-importing are demonstrated here with the battery holder example. For this, a drawing of the flattened part is needed: Figure 5-27.

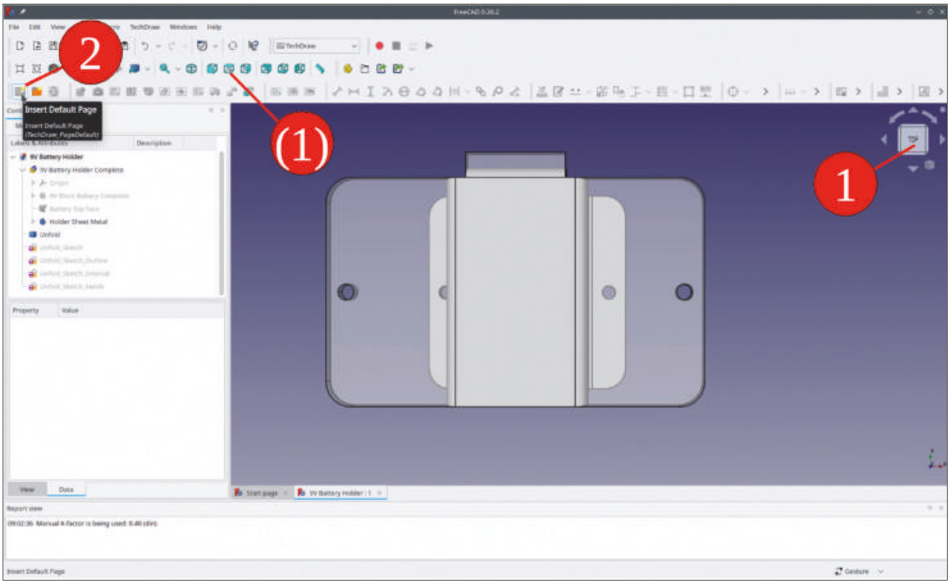


Figure 5-27

33. First, bring the battery holder into the exact 'top' orientation. To achieve this, click either on the control cube (Figure 5-27, step 1) or the 'Top' tool button (same figure, alternative step 1). The precise orientation is important: A slight misalignment of the object would be projected onto the drawing, changing the dimension values. This is difficult to spot by eye and can be disappointing later.

34. Switch to the 'TechDraw' workbench. Click the 'Insert Default Page' tool button (Figure 5-27, step 2). In the tree view, a new object 'Page' appears, with a 'Template' object for the frame of the drawing. Rename the 'Page' object to 'Unfold Drawing'.

35. In the tree view, mark 'Unfold_Sketch_Outline' and 'Unfold_Sketch_Internal'. Then, click the 'Insert View' tool button (Figure 5-28, steps 1 and 2). Mark the 'View' object, and adjust X, Y and 'Rotation' parameters (step 3), until the view object appears in the center of the drawing. For the translations, you could also drag-and-drop the view. Check the 'Scale' to be set to 1.00 (important!). Then, scroll to the 'Projection' section of the property list, and verify the 'Perspective' is set to 'false' (this avoids distortions), and that the 'Direction' property reads '[0,00 0,00 1,00]' (the unit vector of the z direction = top view, Figure 5-28, step 4).

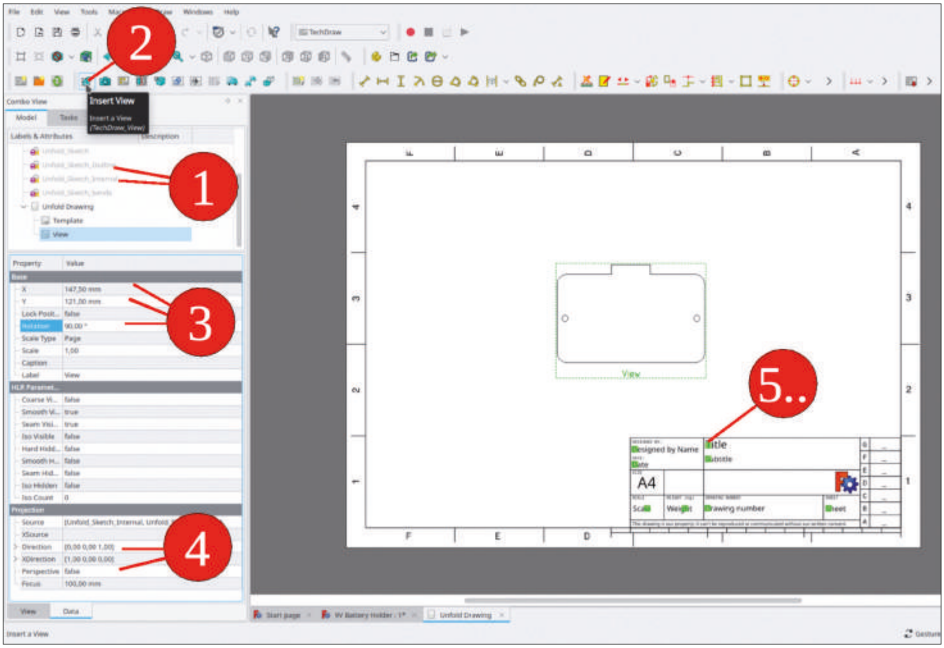


Figure 5-28

36. On the drawing, the view as well as the fields of the frame are displayed with small, green markings. Click those in order to edit the content of the fields (Figure 5-28, step 5). Some work on the documentation makes it easier to understand later what the intentions were. For the DXF export, the frame around the viewport is disturbing. You can right-click it and select 'Toggle Frames' from the context menu to hide the frame as well as the green edit markings on the other fields. (Figure 5-29).

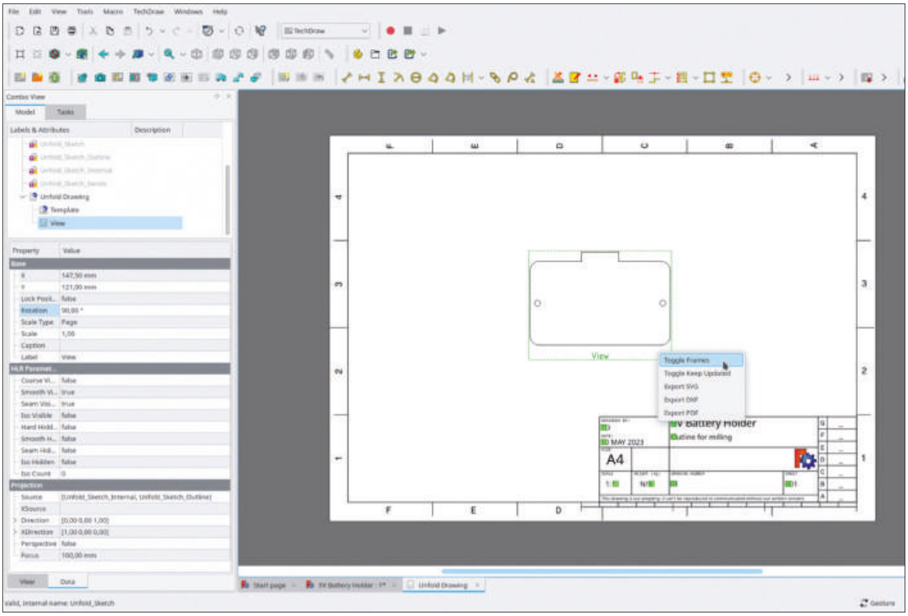


Figure 5-29

37. In order to export the view contents, right-click the view again (with the frame hidden — it's not needed for exporting). Then, select 'Export DXF' (Figure 5-30).

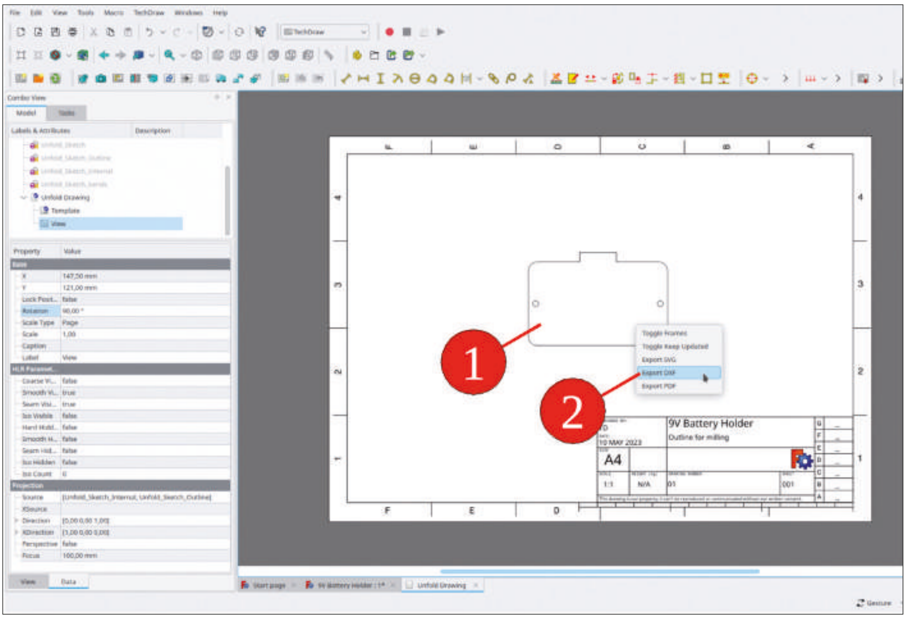


Figure 5-30

38. The import to NC EAS(Y) works straightforward (Figure 5-31), which is running on a Windows© platform. The precision of the data is very good, which has been verified by manufacturing several parts in the way described.

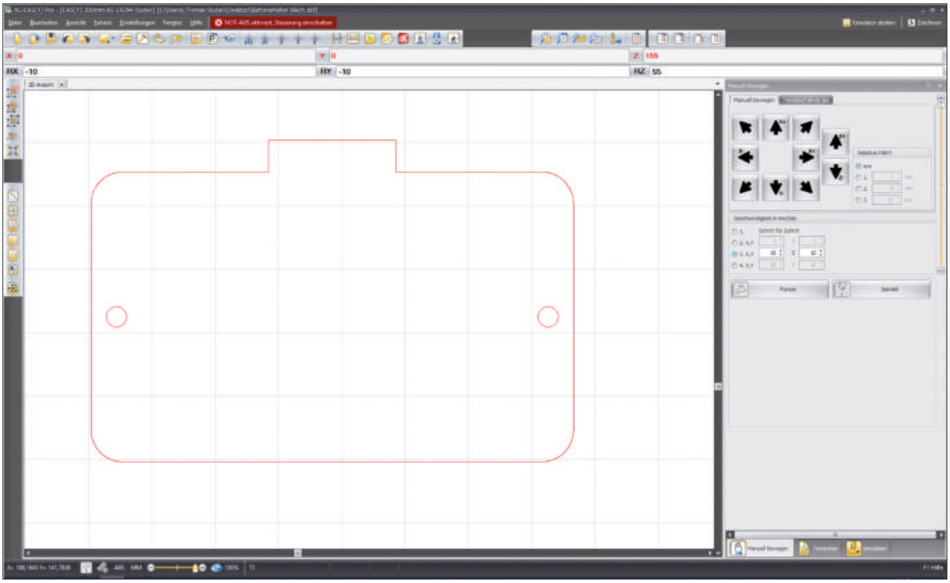


Figure 5-31

5.4 Creating a Drawing of the Battery Holder

Parts are not always manufactured with the aid of CNC machines. Sometimes, the simplicity of a part does not justify the programming effort, or a machine is simply not available. In these cases, a drawing needs to be furnished, which contains all the necessary information needed by a mechanical workshop. Technical drafting is a profession in its own right, requiring training. The following sections provide only a very brief introduction into the matter. For those who want to read more about the topic, reference manuals and books exist, like [Hoi 2000, Arz 2001]. This in mind, you can proceed and use the battery holder as a simple example.

1. Switch to the 'TechDraw' workbench and create a new drawing page, similar to the one created in 5-3-2. Rename this page to 'Mechanical Drawing' and fill in the fields of the drawing frame.

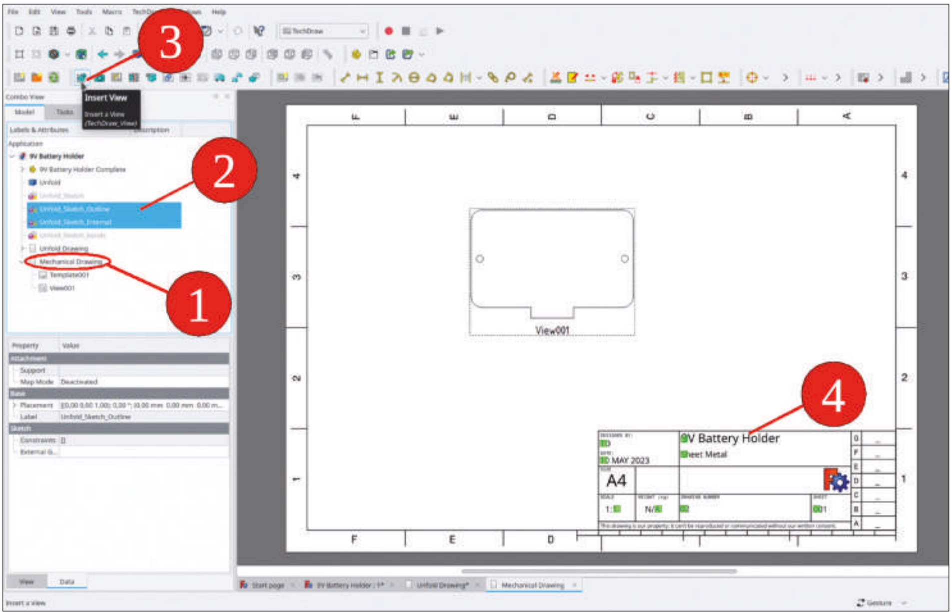


Figure 5-32

2. In the tree view, mark 'Unfold_Sketch_Outline' and 'Unfold_Sketch_Internal' and click the 'Insert View' tool button (Figure 5-32).

3. Move the view upwards, to accommodate for the measures that will be inserted next. To do this, you can either drag the view with the mouse, or edit the X and Y parameters in the 'Base' property list. When you want to drag the view, pick a part of it that is not occupied by the part itself. Hide the frame with a right click, and the selection of 'Toggle frames' from the context menu.

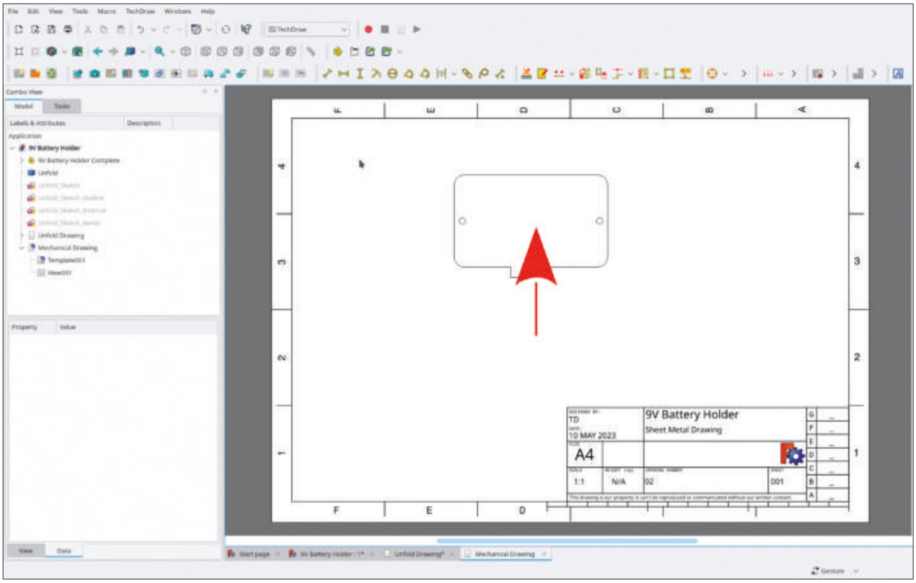


Figure 5-33

4. The part is symmetric. A symmetry line could deliver this information to the machinist. To add this line, click the part and select 'TechDraw | Add Lines | Add center line to faces' from the main menu (Figure 5-34).

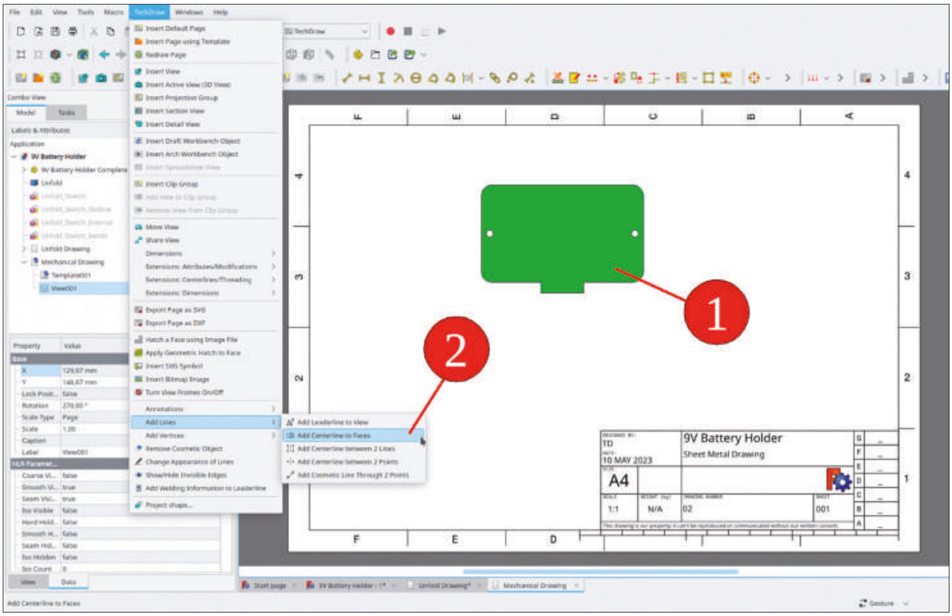


Figure 5-34

5. In the task window, select a weight (width) of 0.25 mm, and set the 'Extend By' parameter to 10 mm (Figure 5-35).

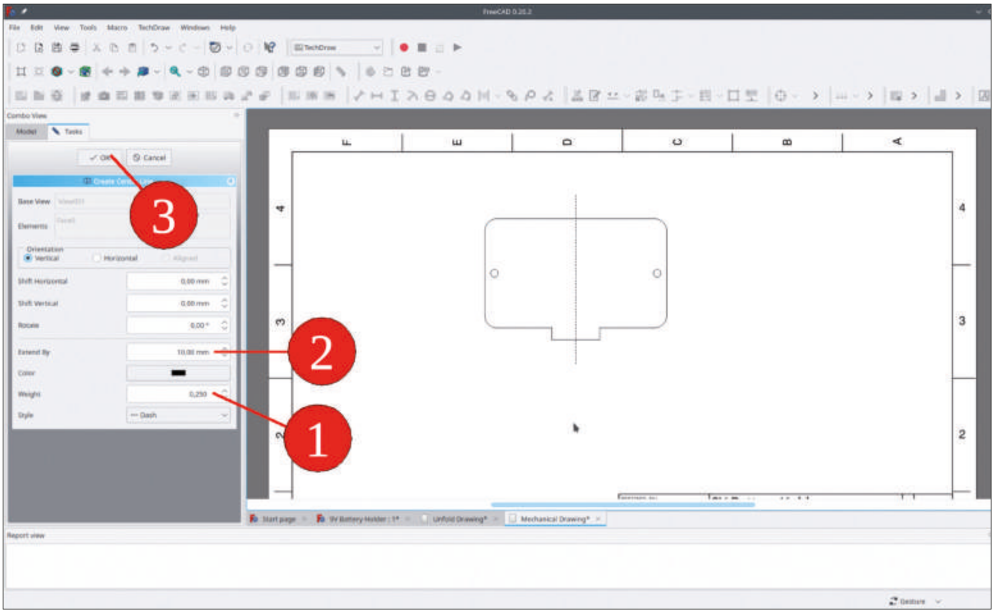


Figure 5-35

Because of the symmetry, it would be accepted practice to add measures between opposite details. This is a nice shortcut and keeps the drawing simple. However, this is done in practice, but is at hand not helpful when you have to actually make the part. In the real world, it is better to have dimensions that help you to mark the sheet with a scribe:

6. First, the distance of a mounting hole to the edge of the part could be specified. To pick the circle position, center lines for the hole are needed. Mark the hole contour on the drawing and click the 'Add Circle Center lines' tool button (Figure 5-36).

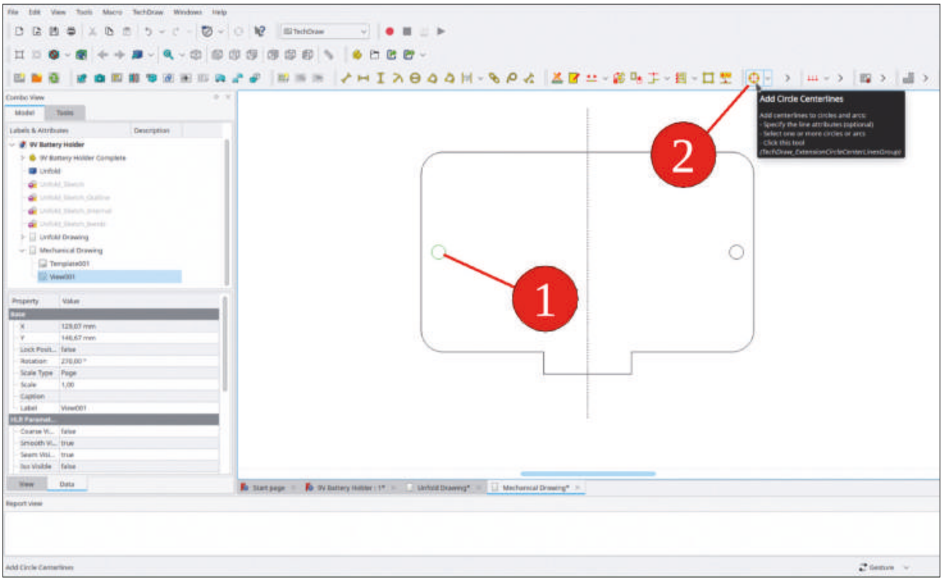


Figure 5-36

7. Mark the vertical edge close to the hole, and the vertical center line of the hole. Then, click the 'Insert horizontal dimension' tool button (Figure 5-37). Drag the dimension to a free space underneath the part contour. In order to move a dimension, drag on the image itself, not on arrows or leader lines.

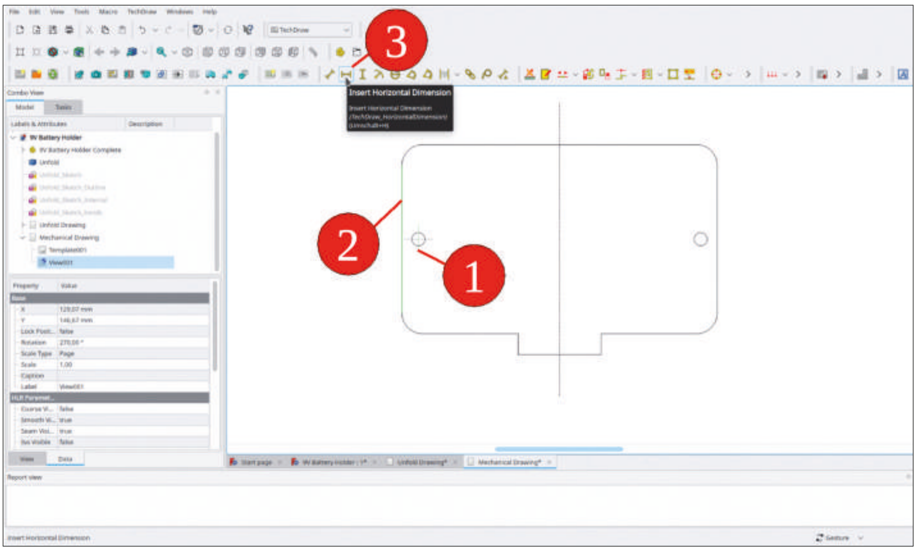


Figure 5-37

8. In the same fashion, mark the vertical edge on the side, the corresponding vertical edge of the lower protrusion and add the dimension (Figure 5-38).

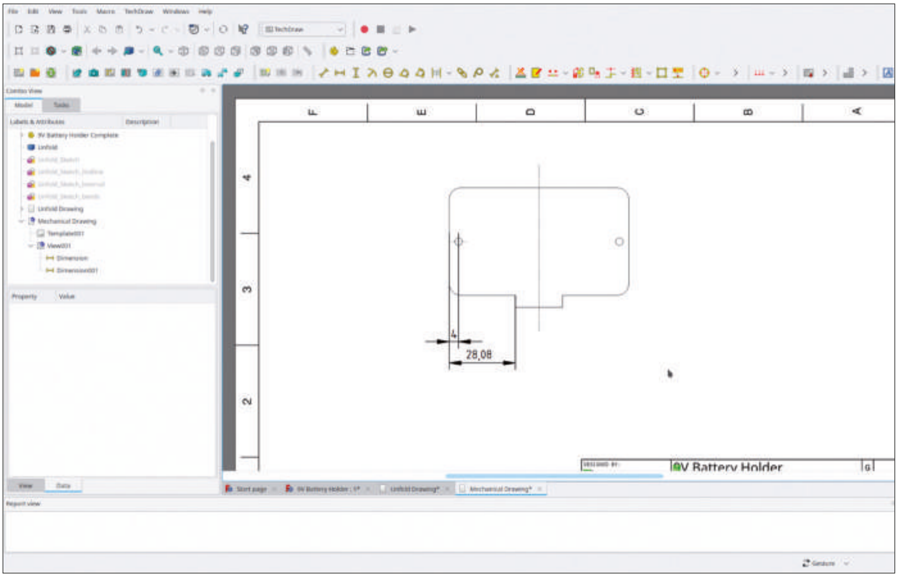


Figure 5-38

9. To complete the horizontal dimensions, add the total extension of the part, edge to edge.

10. Then, add the center lines to the other hole, as described in step 6, and add all the vertical dimensions displayed in Figure 5-39.

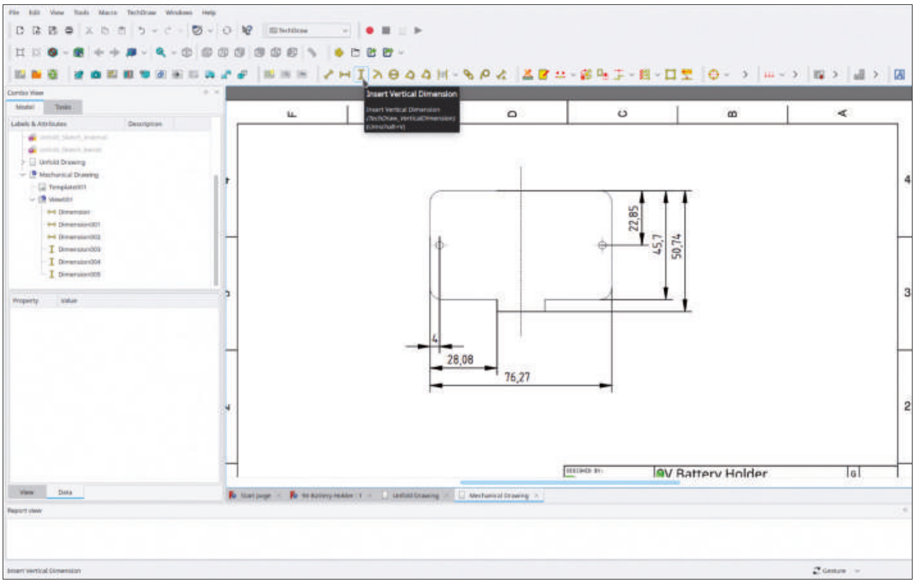


Figure 5-39

11. In the drawing, mark one of the two circles and click the 'Insert Diameter Dimension' tool button. The circle is present twice. Not always, this is as evident as here. Machinists usually like a hint to the total number of similar features. Therefore, in the tree view, double-click the diameter dimension. A task window opens. In the field for 'Format Specifier', add '(2×)' behind the cryptic format string. That addition will show on the drawing as typed (Figure 5-40). Close the task with the OK button.

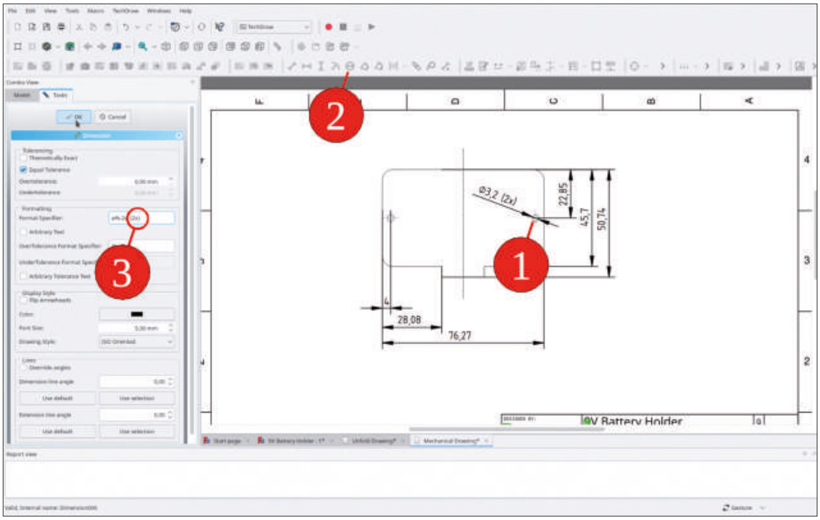


Figure 5-40

12. Then, mark one of the rounded corners and click the 'Insert Radius Dimension' tool button. In the tree view, double-click the radius dimension and add '(4×)' to the 'Format Specifier' field (Figure 5-41).

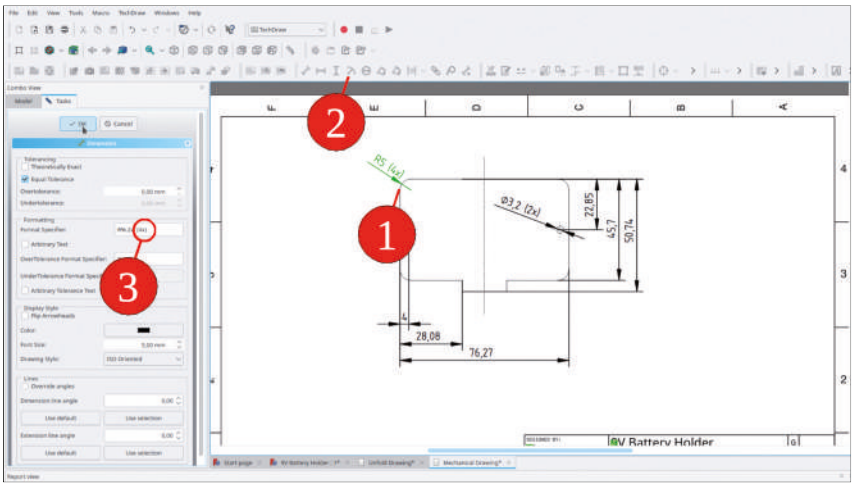


Figure 5-41

13. Now, the drawing is nearly done. Some dimensions are given with a precision or 1/100 mm – this is very precise, maybe too precise for manual production with the punching machine. When given to a machine shop, such drawings can raise – for a good reason – discussions about the necessary effort. Specifying the dimensions with 1/10 mm seems already fairly good for this part. In order to change all the questionable figures at once, in the tree view, mark all the linear dimensions showing two digits behind the comma. In the property list, locate the field 'Format Spec', and change the format string to '%.1w'. The figure in front of the 'w' designates the number of digits shown.

14. Now, hopefully there will be no discussions about the precision coming up. A comment specifying the material is still missing. From the main menu, select 'TechDraw | Annotations | Insert Annotation' (Figure 5-42).

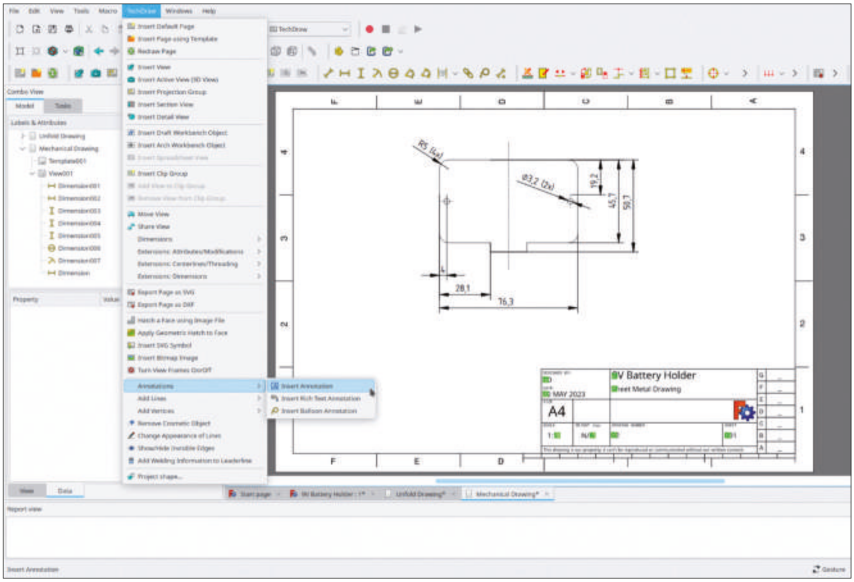


Figure 5-42

15. The annotation can be dragged to a free space on the drawing. It is eventually difficult to pick – in this case, right-click the drawing and select 'Toggle frames' from the context menu. Then, it is easier to pick the annotation frame for dragging.

16. The text of the annotation can be modified by clicking the property list entry for 'Text', and subsequent clicking the … button to the right. A dialog opens, into which the text can be entered. Type some additions for the material and the tolerances, as shown in Figure 5-43.

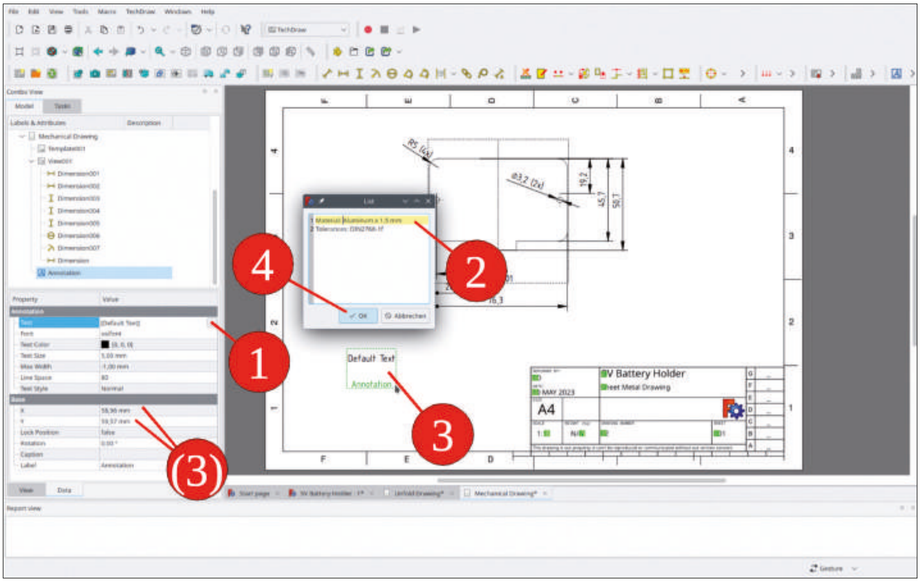


Figure 5-43

With these steps, the drawing is completed. You could already dare to see a machine shop with it (Figure 5-44). As already mentioned, technical drafting is a profession in its own right, and the creation of fully constrained drawings is a kind of art. However, a few small steps are taken. On top of it, the drawing will help you in the shed wielding the saw.

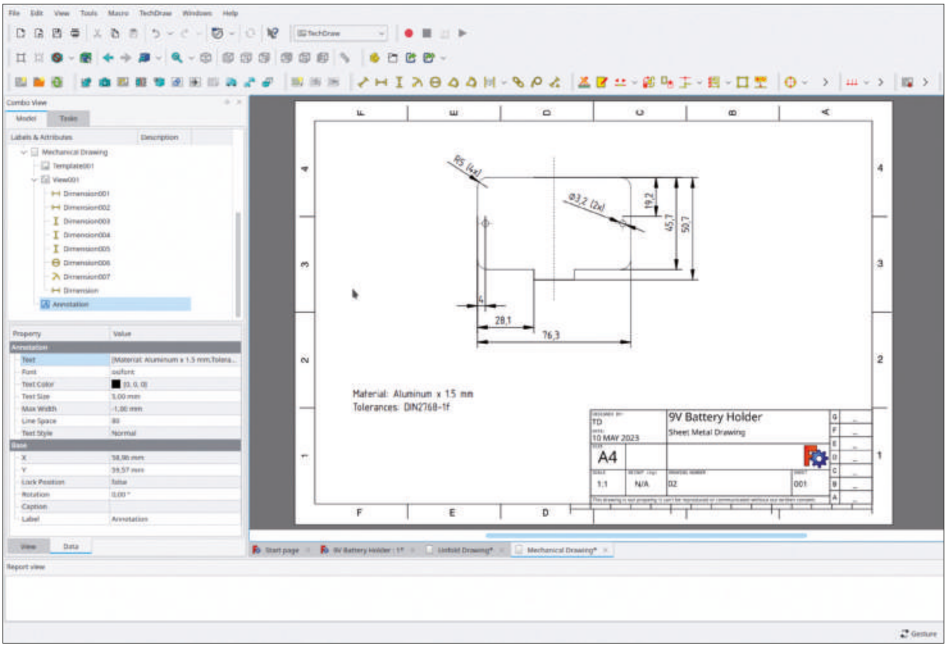


Figure 5-44

5.5 Create a Plan for the Bending Machine

For the bending, a plan is needed. It contains information about the position of the bends, the direction of the bending and eventually a sequence in which the bending should be done.

1. For the bending plan, generate a new page by clicking on the 'Insert Default Page' tool button (leftmost in the workbench menu). Rename the new page to 'Bending Plan'.

2. In the tree view, mark the sketches 'Unfold_Sketch_Outline', 'Unfold_Sketch_Internal' and 'Unfold_Sketch_Bends'. Then, click the 'Add view' tool button. (Figure 5-45). When the components in the view are not aligned properly, delete the view again, and sync the placement parameters of the different unfold sketches.

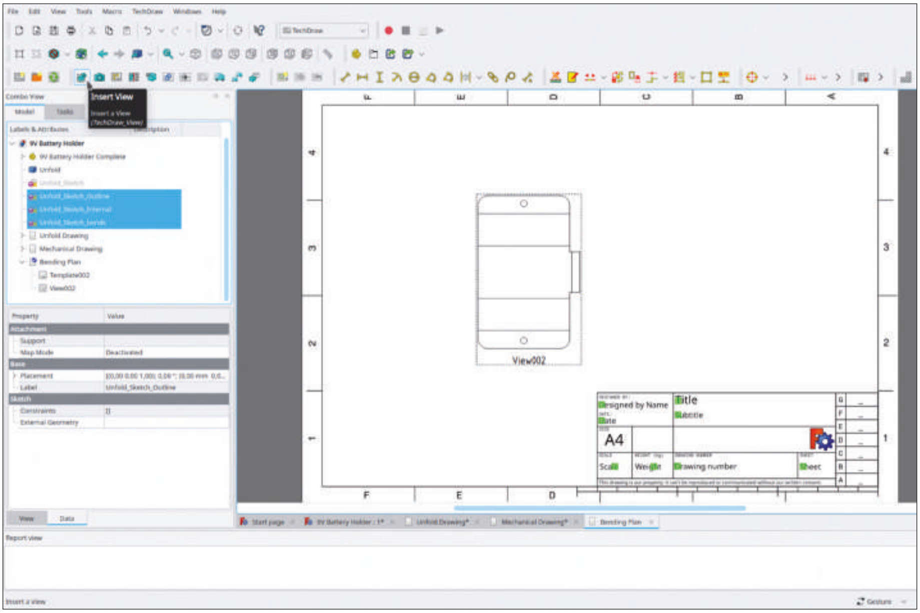


Figure 5-45

3. Pick the view in the frame, or a free area inside, and drag it to a free spot on the drawing surface. Adjust the 'Rotation' property if needed. Fill in the drawing frame fields with name, date, etc.

4. As described in 5-4-4, add a vertical symmetry line. If the setup is failing, you can remove the line again by marking it in the drawing and then selecting 'TechDraw | Remove Cosmetic Object' from the main menu.

5. Add the dimensions given in Figure 5-46. These dimensions connect bending positions with outer edges, which is useful for marking with a scribe. If you have very good equipment, try the 1/100 mm for the marks on the sheet metal, or

the bending machine guides. If you use a ruler and a manual scribe, you could reduce the precision of the dimensions to 1/10 mm, as described in 5-4-12.

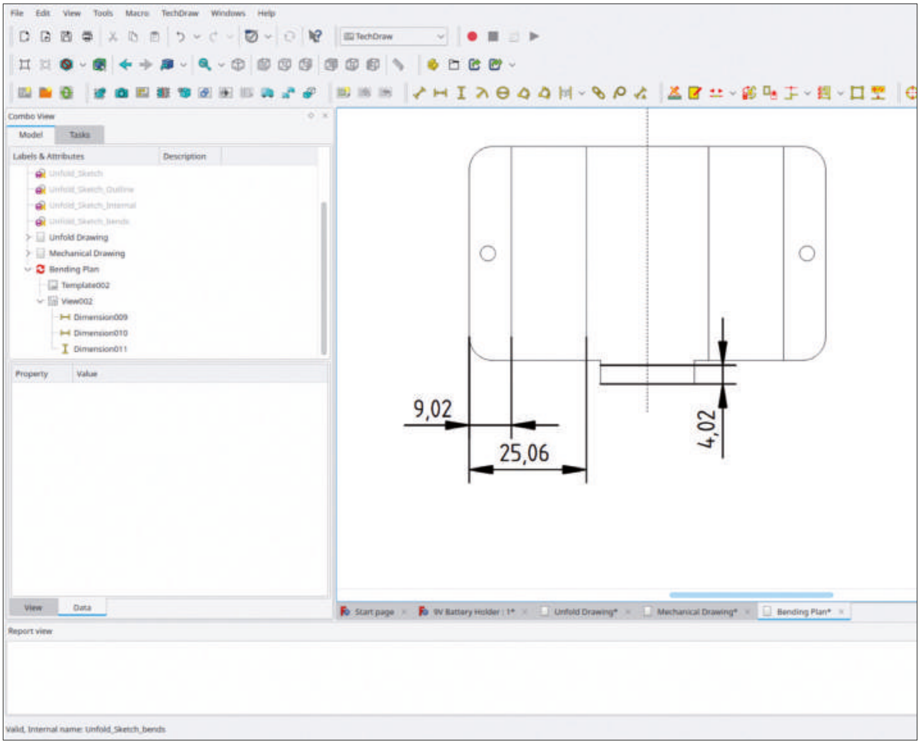


Figure 5-46

6. With the bending plan, the succession of the different bends is important, because some of them, when done, might make others inaccessible for the bending tool. Anticipating this in advance may save some frustrations later. It is already evident that the small stop which points to the bottom is not easy to bend after the large flaps on the sides have been folded. To give the small bend on the bottom the highest priority, mark it and add an annotation by selecting 'TechDraw | Annotations | Add Balloon Annotation' (Figure 5-47). A task window opens. Insert for the text the order ('1') and the bending direction('d').

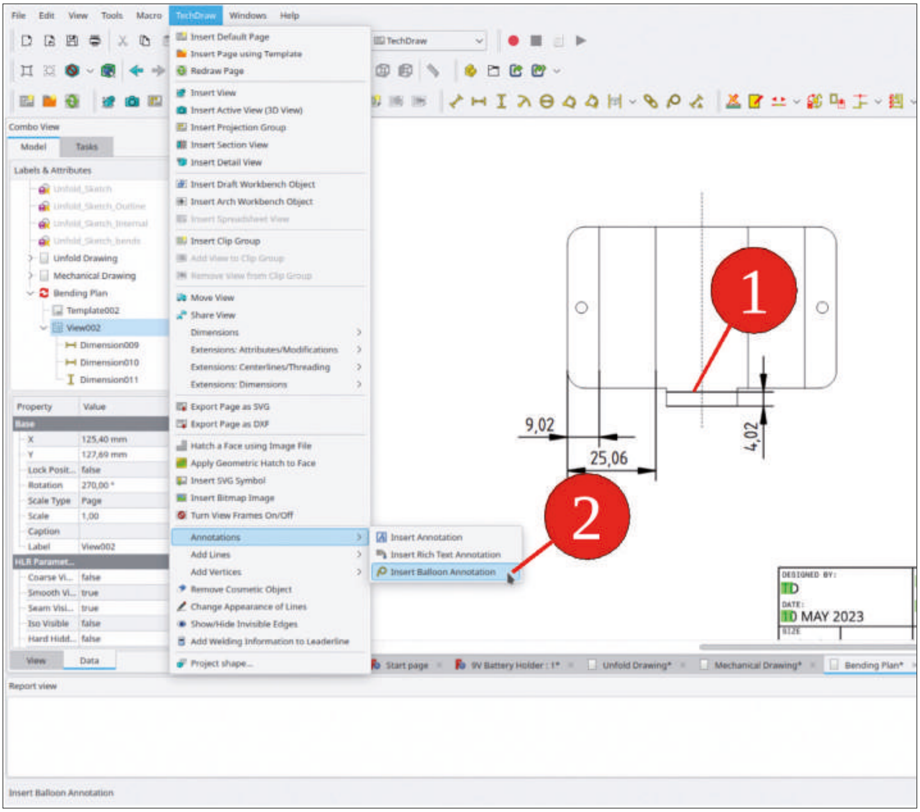


Figure 5-47

7. In the same way, add the annotations given in Figure 5-48. Please note: The order of the bends depends on your bending machine and its capabilities.

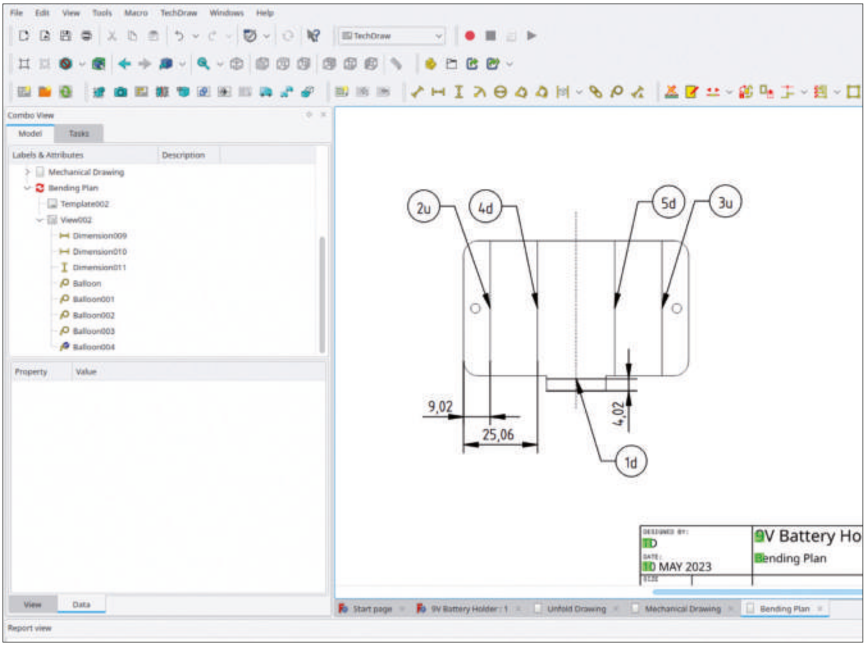


Figure 5-48

5.6 Example Photos of the Battery Holder

After all this work in the virtual world, the question remains whether the result is meaningful. The sheet metal part was made in the classical way: With your scribe, punching machine, file, and calipers. The bending was done 'blind', without intermediate checking with a battery. The only difference being that for the battery dimensions, the center of the tolerance field was used. In consequence, some batteries might not fit well.

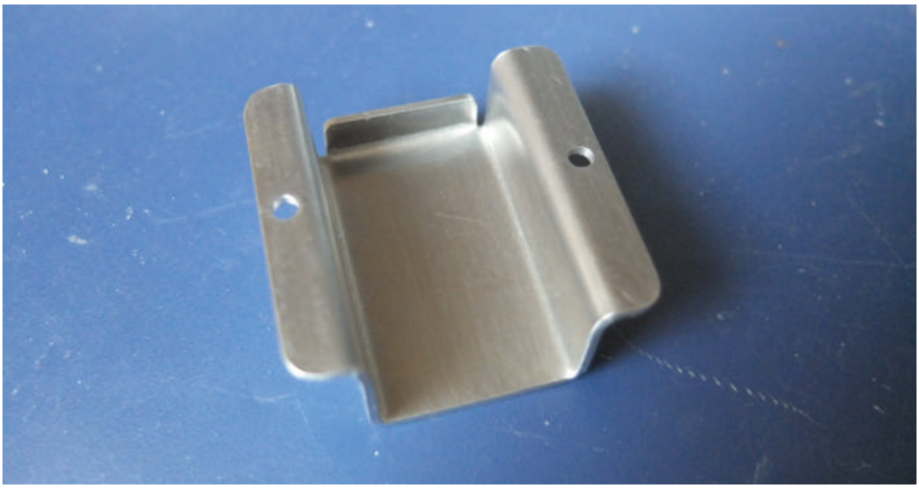


Figure 5-49

Figure 5-50

Figures 5-49 and 5-50 show the result: The battery fits tightly but is not overly pressurized by the part. The 'Sheet Metal' workbench from Shai Seeger works like a charm! It is also good to look a bit more into the documentation which comes with it. With this simple example, you have just scratched the surface of the possibilities available.

Chapter 6 • Assemblies

6.1 “Assembly” Workbenches or Std-Part Container?

For FreeCAD, various assembly workbenches exist as AddOns. The ones used the most are probably Assembly2, A2plus and Assembly4. The gigantic advantage of these AddOns is evident: the ease of assembling components with defined degrees of freedom, plus a whole catalog of meaningful relations. Whereas Assembly2 offers conditions that are intuitive and easy to utilize, Assembly4 exclusively uses local coordinate systems for the link between different objects. An introduction to Assembly2 is given in [Kis 2018], in Chapter 3, while A2Plus is explained in [Kis 2019] and Chapter 9.

The ease of mounting assemblies with a few clicks also comes with a small caveat: The assembled parts have changed their format away from the Std-Part-Container, since the defined relations are now integrated into the component files. This makes the design recycling or porting of components between assemblies more difficult. Also, the ability to open and modify such assemblies relies on the availability of the workbenches themselves.

Another drawback is surfacing, i.e., when one of the parts is formed with its 'bodies' and modified in relation to other parts during the assembly. You will get to work on such an example in the next chapter: The design of a front panel, onto which components are placed. In this case, it is desirable to link some of the parts such that, for example, mounting holes or footprints show associative connections with the defining objects. Then, moving the component will cause all the related details to follow in a synchronized way. Therefore, it is necessary to shape the body objects of, for example, the dependent panel.

Furthermore, it is desirable to retain the ability to nest the design entities in arbitrary numbers and hierarchy levels. This allows the creation of nicely structured, complex designs, which are broken up into manageable smaller entities. Following this path, a completed assembly could resurface as a component of the next-level assembly.

This portability is already built into the basic concept of FreeCAD, with the inclusion of the Std-Part-Container. Even though the popular 'Assembly' workbenches with their comfortable utility can be of great help, also when simulating mechanisms, you will not proceed to use them in the following chapters and stick with the Std-Part-Container only. That may seem purist, but once this method is internalized, you can even better appreciate the “Assembly”-workbenches too and utilize them with applications they excel in — like making a complex mechanism move.

Chapter 7 • An Assembly Example: The Elektor ESR Meter as a Front Panel Project

Sometimes, the compartment for a small piece of equipment exists before the idea for the insides arrives. This was true with the author as well. From the flea market, there was a leftover wooden box with a hinged lid, which seemed just right to use for the Elektor ESR meter. To turn boxes like this into an instrument, the only structural part needed is a front panel, maybe with some spacer feet in the corners, for the assembly. A meter in a wooden box? Looks a bit retro. But then, why not? The circuit has been around for a long time, and the addition of a recycled mechanical panel meter kind of reinforces the style.

7.1 Project Organization

Some initial thought about the organization of a new project is always meaningful. There is one special part which will change and be adapted when other parts are merged into the project tree: The front panel itself. For that reason, it should be packed into the project tree right on top and in its own Std-Part-Container. The associative cutouts and mounting holes that accumulate during the assembly can be conveniently collected there. If it is the first branch in the tree view, it can be easily located for access.

Into the project, other components are added, like the binding posts. In their own Std-Part-Containers, they are completely self-contained but eventually get associated with engraved labels later on which should relocate together with these entities when they are moved, for example, during the design optimization.

Other components are additions to already inserted objects like the control knob for the potentiometer, which controls the scale zero position. Furthermore, these control elements will very likely have associated labels and guide lines on the front panel. The same is true for the rotary switch (ON/OFF and battery test position). The circuit board will rest on spacers and have associated fasteners.

It is meaningful to group these related parts in enclosing Std-Part-Containers. This has the nice side effect of generating a tidy and concise project tree. Furthermore, a lot of the required associativity follows from this embedding practice, by reference to the coordinate systems of the enclosing Std-Part-Containers — even when the grouped parts are only placed (not attached) in there.

7.2 Preparation – Starting the New Project Tree

Start a new file and save it as 'ESR Meter'. Then, start a new Std-Part-Container and rename that to 'ESR Meter Complete'. For the front panel, start another Std-Part-Container and rename that to 'Front Panel Complete'. Drag and drop the new Std-Part-Container into 'ESR Meter Complete'. The front panel is the only part that will be created and modified over the course of the assembly. Switch to the 'Part Design' workbench and click the 'Create body' tool button. Rename the new body object to 'Front Panel'. Figure 7-1 shows the structure.

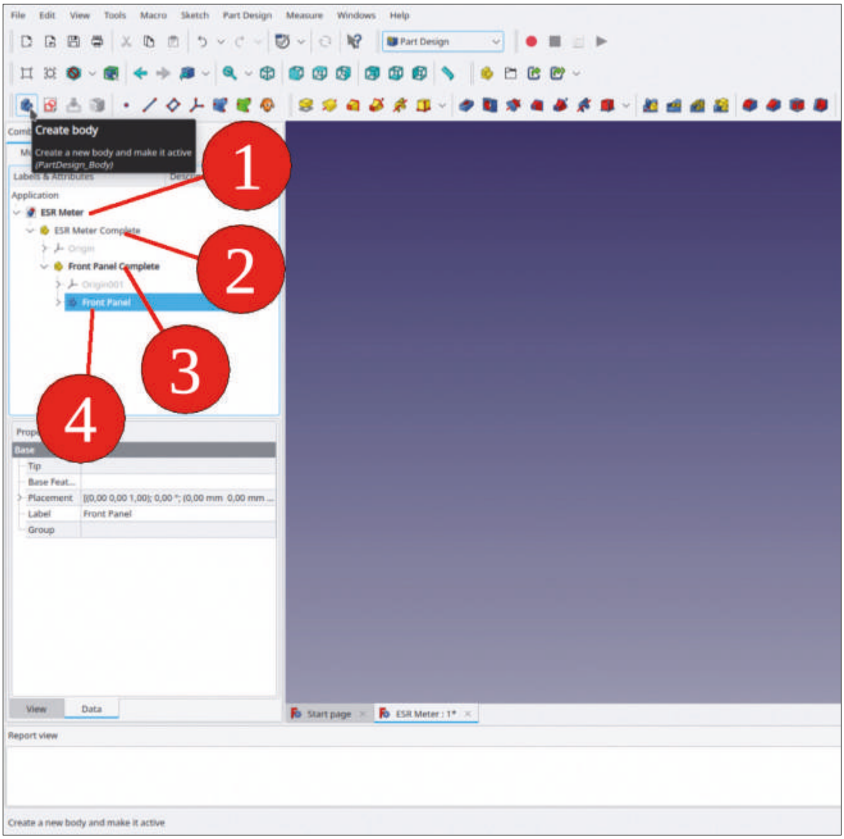


Figure 7-1

7.3 Modeling the Front panel

1. The wooden box constrains the dimensions of the front panel: Both the width and the height have to equal 160 mm in order to fit inside. To prepare the design, double-click the 'Front Panel' body to activate it (its title is then shown in bold letters). This selects the 'Front Panel' body as the destination for the operations to be applied next. In a complex project tree, it is important to track which object is activated.

2. Start the sketcher and select the XY plane from the initial selection dialog. Click the 'OK' button to proceed. Click on the 'Create a centered rectangle' tool button and draw a rectangle, which is centered around the origin (Figure 7-2). End the drawing command with a right click. Mark a horizontal line of the rectangle and click the 'Constrain horizontal distance' tool button. Enter a value of 160 mm. In a similar way, constrain the height of the rectangle to 160 mm. Close the sketch.

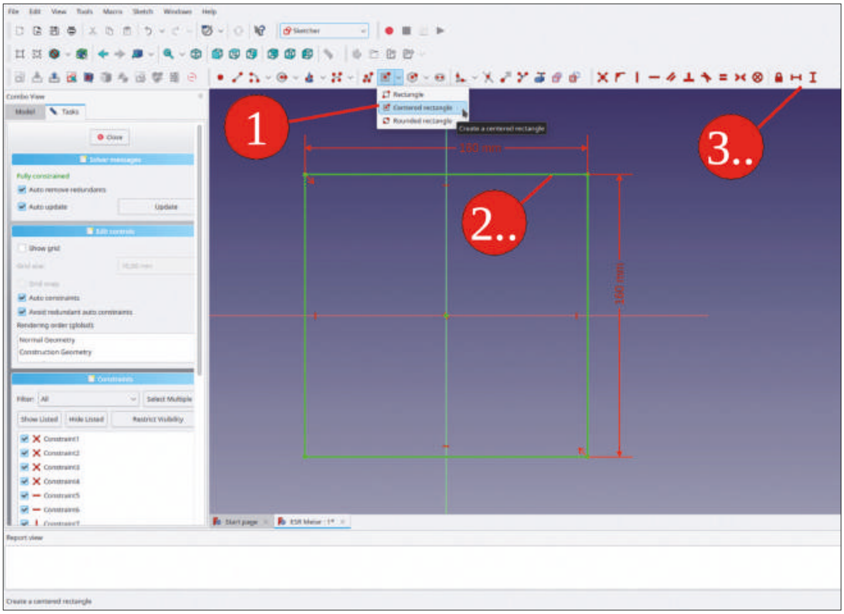


Figure 7-2

3. Parts that are attached to the front side of the front panel need a reference, which is immune against re-enumeration (the front panel will change a lot during the assembly process). This is easiest done with a datum plane. In the tree view, display the coordinate system of the 'Front Panel' body (mark it and press the SPACE key).

4. In the 3D view, click on blue to deselect the whole coordinate system. Then mark the XY plane. Click on the 'Create a datum plane' tool button. In the task window, thanks to our prior marking, the XY plane is already preset. Set the attachment Z offset to 2 mm (the sheet metal thickness, Figure 7-3). Hide the coordinate system again with the SPACE key. Rename the new datum plane to 'Front Panel Top Face'.

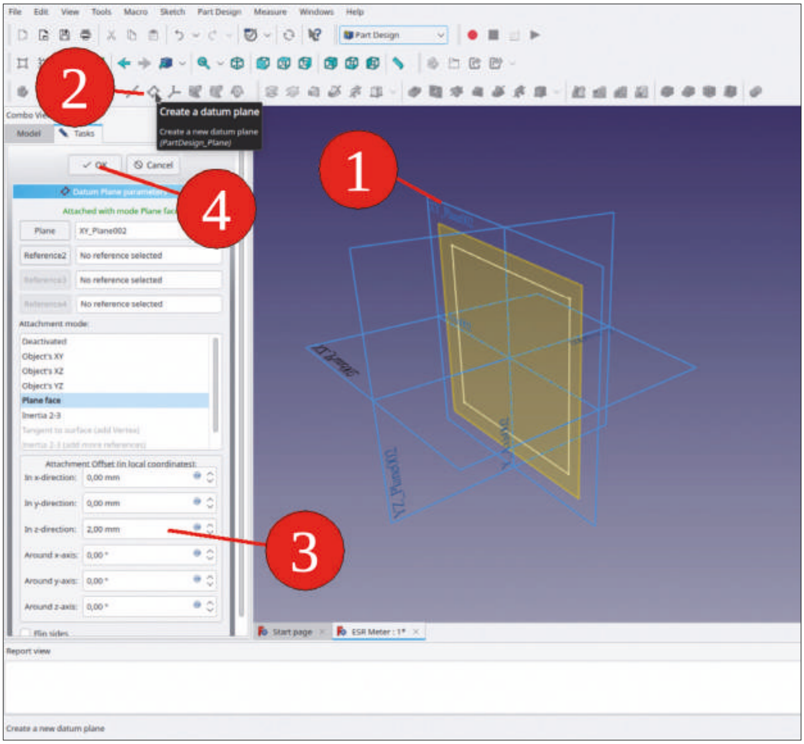


Figure 7-3

5. In the tree view, mark the new sketch and click the 'Pad' tool button. In the task window, set the type to 'Up to face'. Then, in the 3D view, click the representation of the datum plane (Figure 7-4). Close the task window with the 'OK' button.

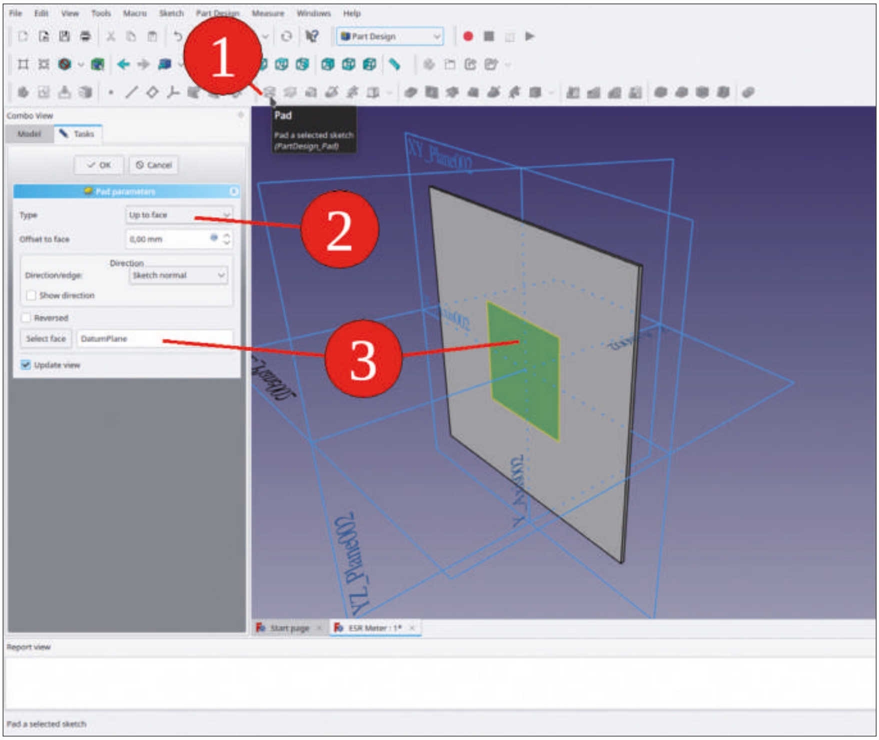


Figure 7-4

6. In the tree view, mark the datum plane and hide it with the SPACE key. Also, hide the coordinate system. Collapse the 'Front Panel Complete' branch.

7.4 Insertion of the Rotary Switch

1. In the directory 'Sample Projects | ESR Meter | Components', locate the file 'Rotary Switch' and open it. In the file just opened, mark the Std-Part-Container 'Rotary Switch Complete' and copy it by pressing CTRL-C. A selection dialog opens (Figure 7-5). Do not change the standard selection, and click the OK button to proceed.

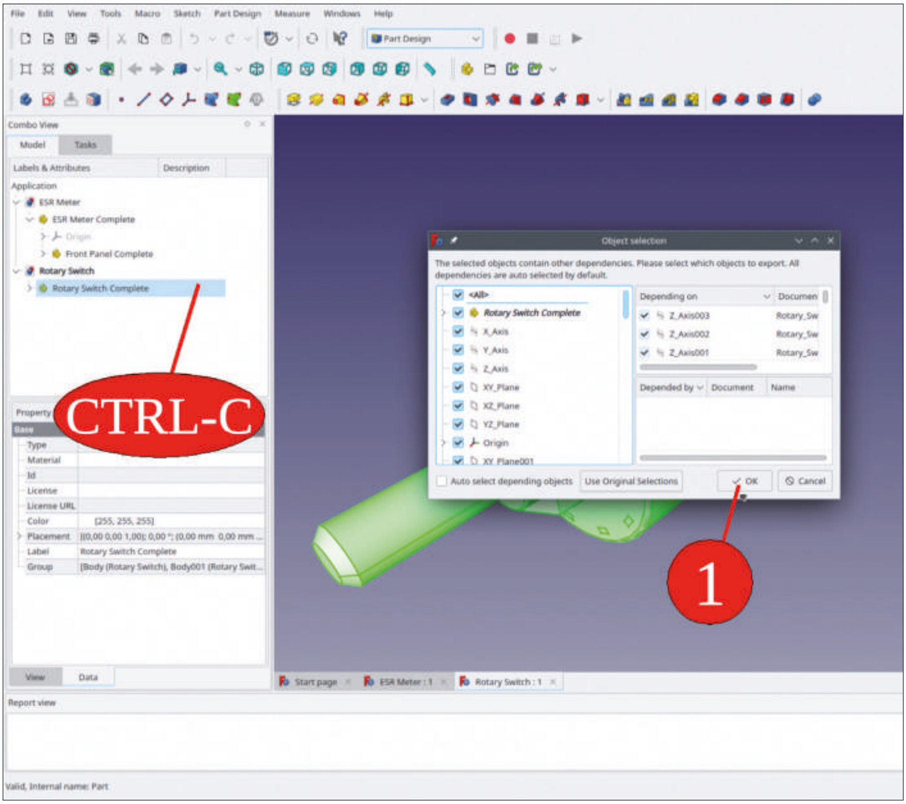


Figure 7-5

2. In the tree view, double-click the root of the file 'ESR Meter', in order to activate it as the destination for the following paste operation. Then, paste in the rotary switch with CTRL-V.

3. Drag and drop the Std-Part-Container 'Rotary Switch Complete' into the Std-Part-Container 'ESR Meter Complete'. Close the file 'Rotary Switch' by right-clicking its root node and selecting 'Close document' from the context menu (Figure 7-6). Sometimes, if the activation of the destination file has been omitted, the pasted object appears in the source document. If this happens, delete the misplaced object (or press CTRL-Z) and start over.

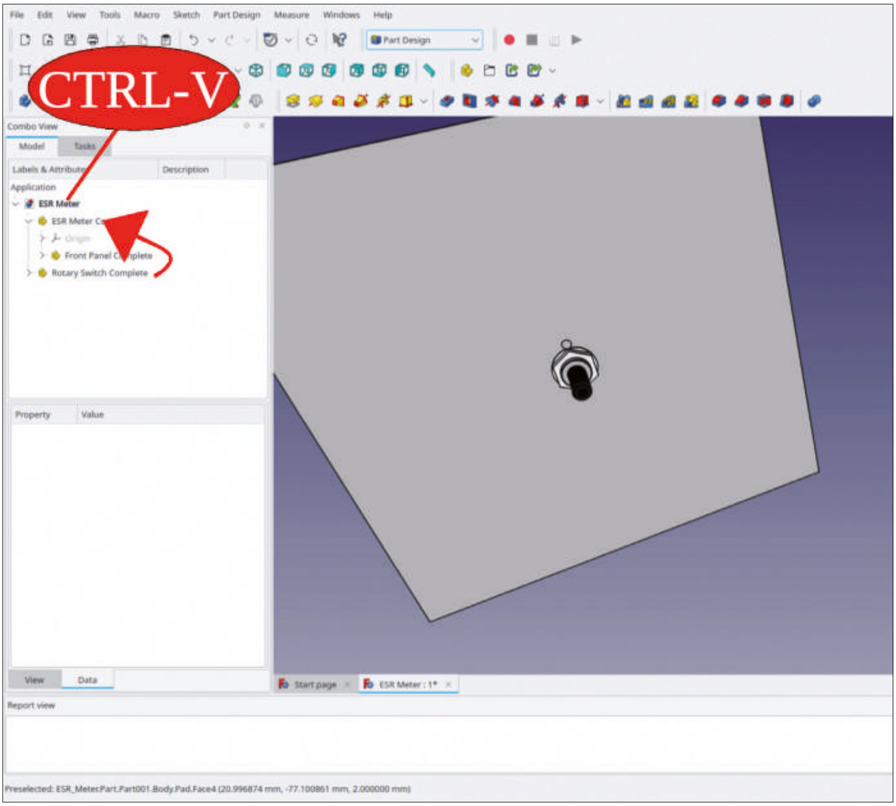


Figure 7-6

4. The position of the rotary switch can be adapted by editing the 'Placement' parameters. To display a task window with these parameters, in the tree view, mark the Std-Part-Container 'Rotary Switch Complete'. In the property list, click into the 'Placement' edit field, then click the … button appearing on the right-hand side of that field.

5. In the task window, click the X Translation field and roll either the mouse wheel (this is a fun part, due to the live update!) or enter the value manually. Set the X translation to −41 mm. In the same way, set the Y translation to −60 mm. If you enter the value manually, do not press the ENTER key if further changes are intended, as the ENTER key will close the task window. Just click the next edit field for more changes. If you are done, close the task window with the 'OK' button (Figure 7-7).

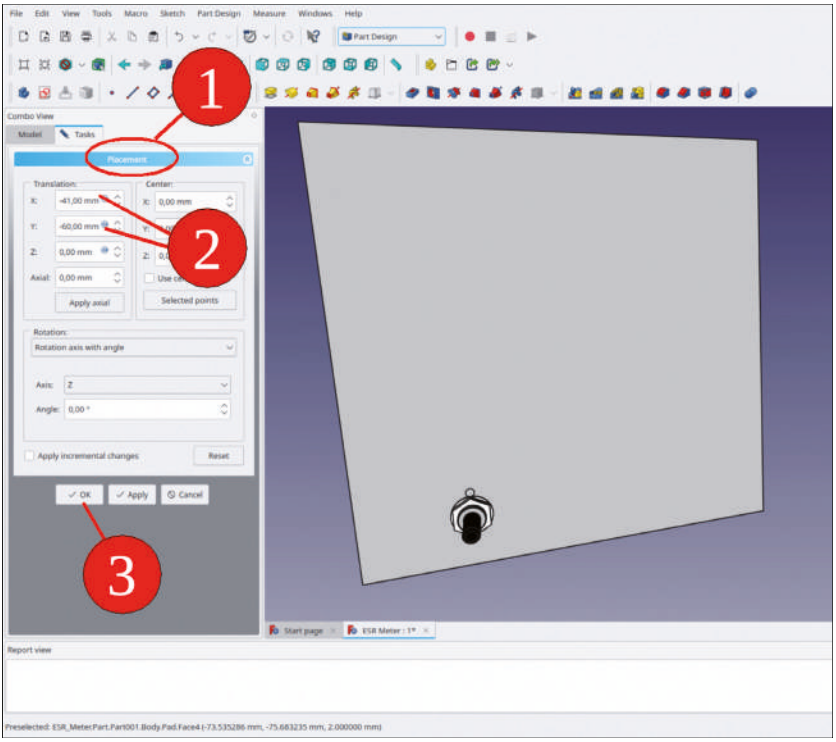


Figure 7-7

7.5 Generate an Associative Footprint with the SubShapeBinder

With a nice CAD program, there is no need to draw the mounting holes for a component manually. When a SubShapeBinder is used, all of this can be done with a few clicks:

1. In the tree view, hide all components except the 'Rotary Switch' body by marking the items and subsequently pressing the SPACE key.

2. In the tree view, double-click the Std-Part-Container 'Front Panel Complete' to set the destination for the new SubShapeBinder. This works even though the container is hidden. The title of the container appears in bold letters once it is activated. In the container, also double-click the 'Front Panel' body to activate it. Here, you want the SubShapeBinder to go.

3. In the 3D view, while holding down the CTRL key, select the two contours that define the front panel footprint. Mark the contours at the base of the geometry, because for the following, the height of the curves has to match. Then, click the 'Create a sub object(s) shape binder' tool button (Figure 7-8).

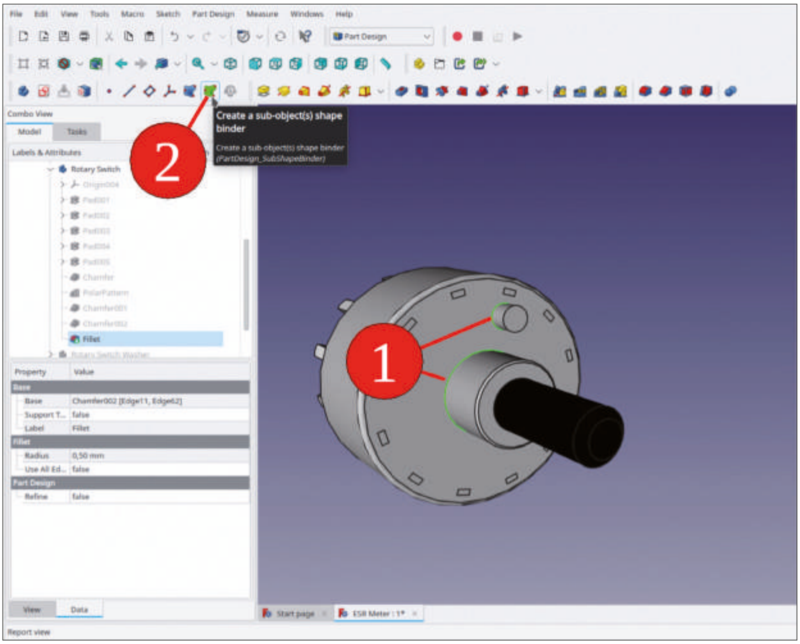


Figure 7-8

4. Scroll up the tree view and locate the new, green SubShapeBinder in the body 'Front Panel'. Rename it to 'Footprint Rotary Switch' (Figure 7-9).

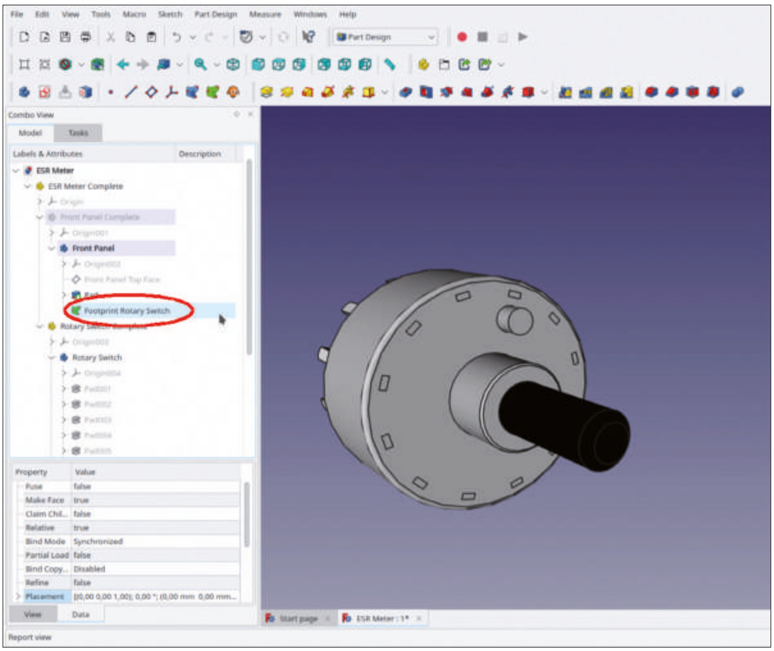


Figure 7-9

5. In the tree view, show again the Std-Part-Container 'Front Panel Complete' (mark it and press the SPACE key).

6. In the tree view, mark the new SubShapeBinder and click the 'Pocket' tool button (Figure 7-10).

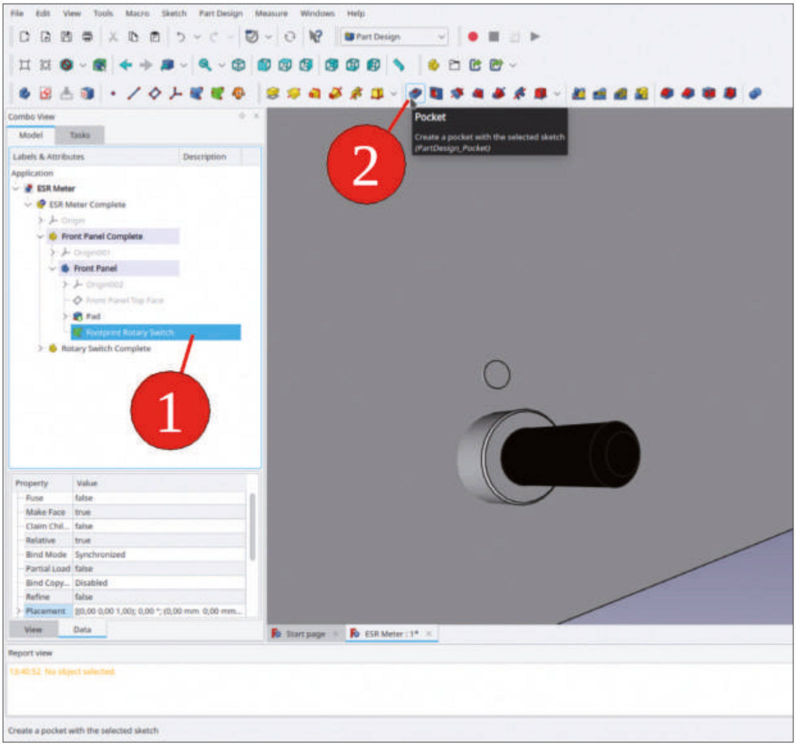


Figure 7-10

7. In the task window, select for the type 'Through all' and check the 'Reversed' checkbox (Figure 7-11).

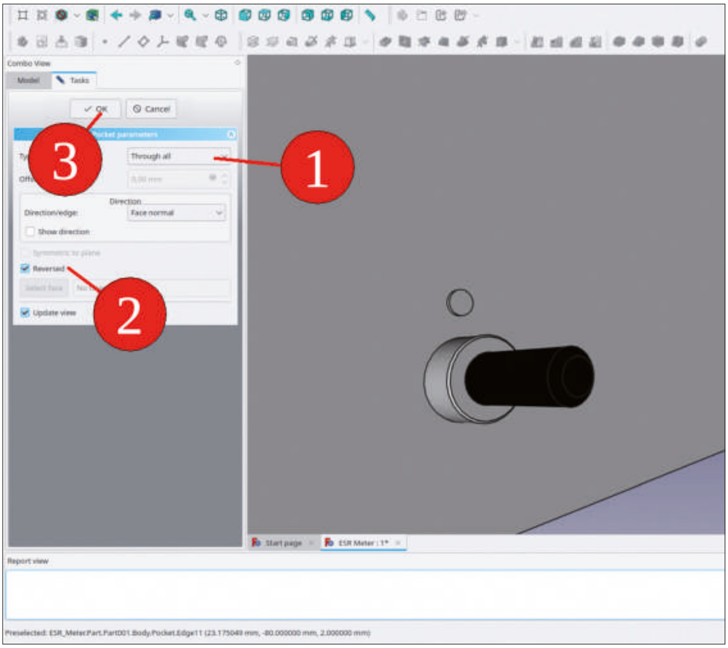


Figure 7-11

8. In the tree view, display the washer and the nut of the rotary switch again (with the SPACE key).

If you hide the Std-Part-Container 'Rotary Switch Complete' (mark in the tree view, and press the SPACE key), you can see the mounting holes in the front panel (Figure 7-12).

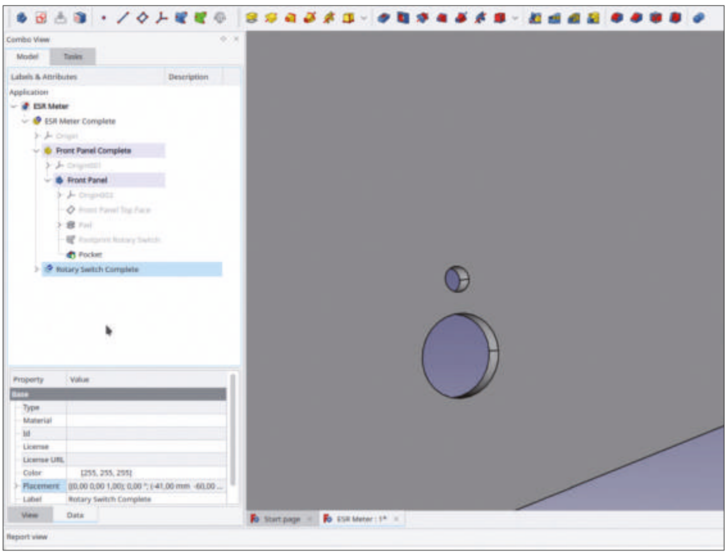


Figure 7-12

The SubShapeBinder keeps track of the referenced item by default (in contrast, the ShapeBinder has a 'Tracking' property that needs to be set to True to do that). Therefore, the mounting holes will follow changes of the rotary switch placement — very comfortable!

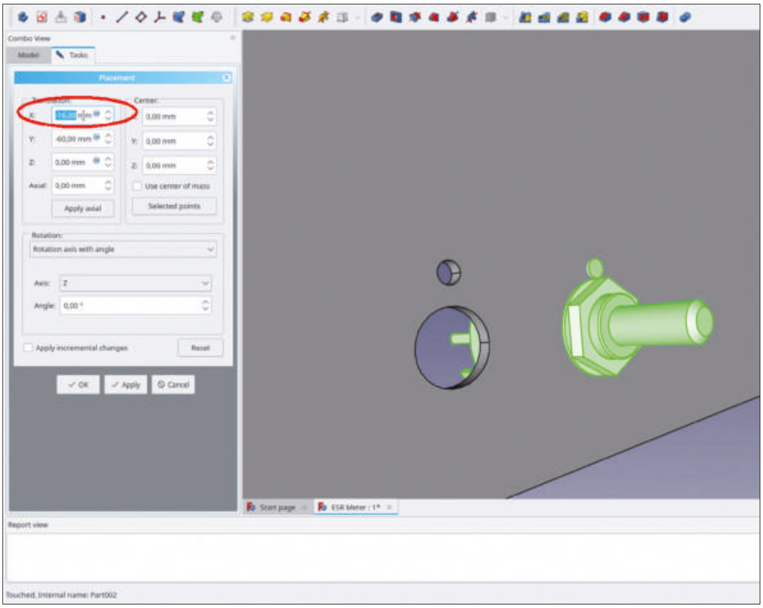


Figure 7-13

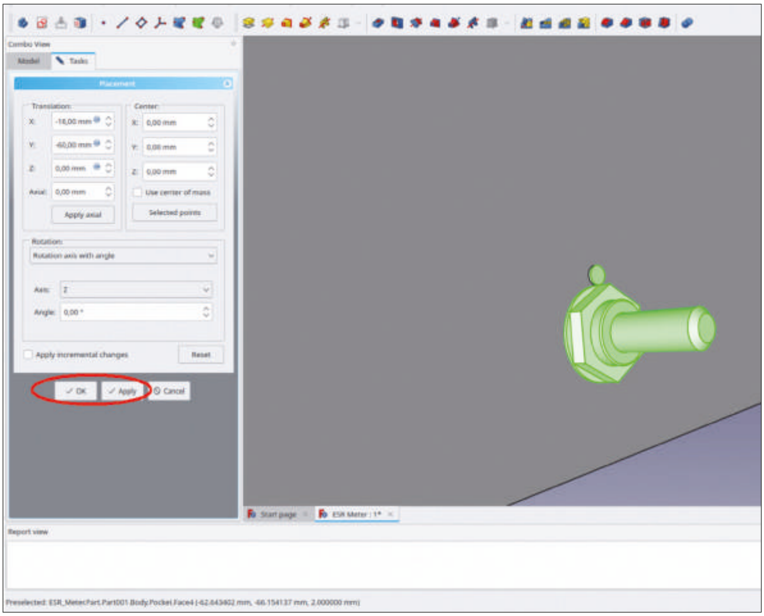


Figure 7-14

Please note that the rotary switch will follow modifications of its placement parameters instantaneously (Figure 7-13), but not the footprint in the front panel. In order to update the front panel, either click the 'Apply' or the 'OK' button (Figure 7-14).

7.6 The Knob and its Place in the Tree View

Now, a control knob can be added for the switch. This is easiest done by adding the knob to the 'Rotary Switch Complete' Std-Part-Container. Embedded there, the knob will then follow true and steady if the placement of the switch is changed. Furthermore, a nice tree structure results, because all the bits and pieces belonging to the switch are now contained in one branch, which can be collapsed to gain more overview. As a candidate for the control knob, a model made for a recycled one is contained in the directory with the components. Yes, with FreeCAD, such lovely old pieces can also obtain a second life!

1. In the directory 'Sample Projects | ESR Meter | Components', locate the file 'Knob Recycling', and open it. Copy the Std-Part-Container 'Knob Recycling Complete' with CTRL-C. In the displayed selection dialog, do not change the default selection and click 'OK' to proceed.

2. In the tree view, double-click the root node 'ESR Meter' (to set the paste destination) and insert the knob with CTRL-V. Then, drag-and-drop the new container into the Std-Part-Container 'Rotary Switch Complete'. Close the document 'Knob Recycling' (Figure 7-15). The placement of the knob is already correct in X and Y coordinates.

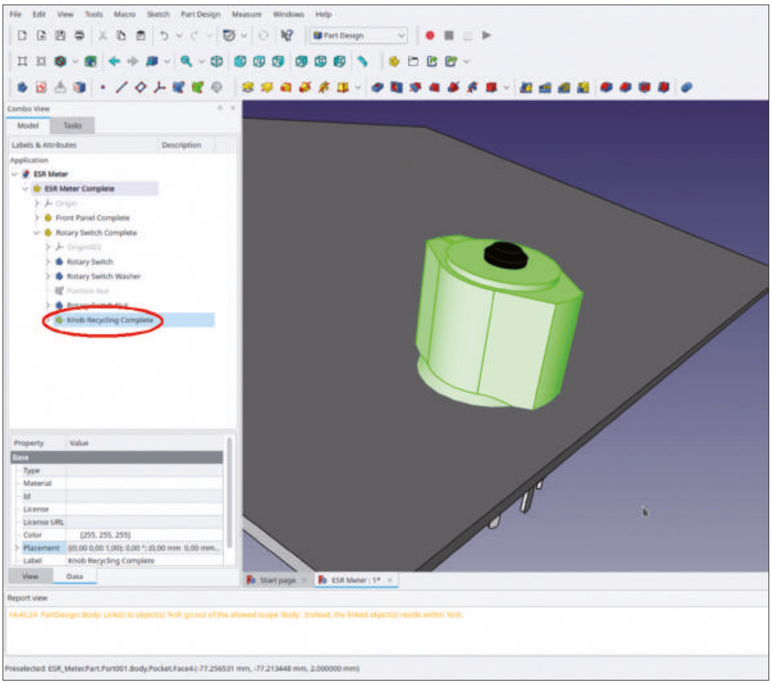


Figure 7-15

3. The knob still scratches on the front panel. In order to give it some Z offset, in the tree view, mark the Std-Part-Container 'Knob Recycling Complete', and edit its placement in the property list. Either expand the list entries until you can access the Z position ('Placement | Position | Z') or open a task window by clicking into the 'Placement' edit field, and then on the ... button to the right. Set the Z translation to 4 mm (Figure 7-16).

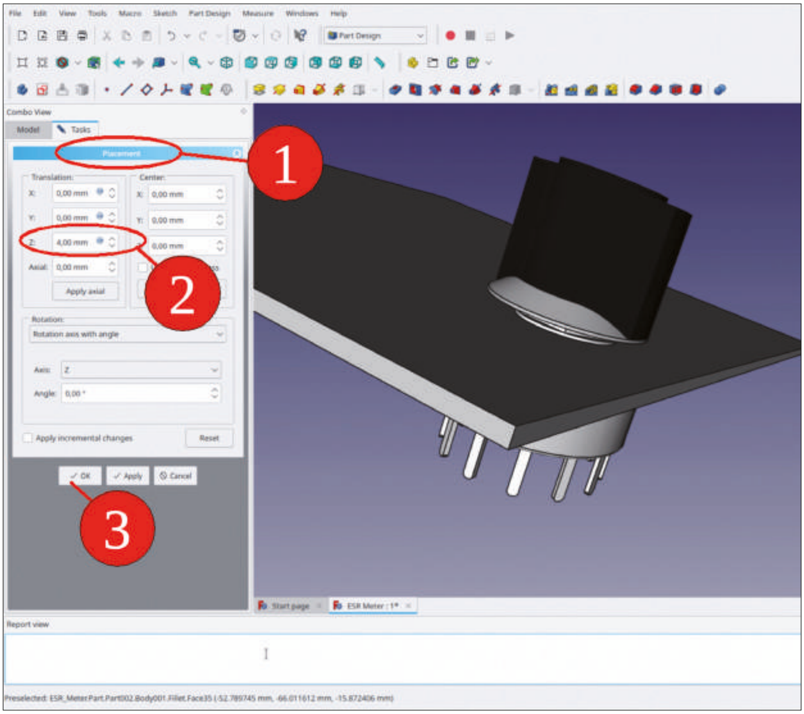


Figure 7-16

7.7 Testing the Associativity

A small test demonstrates the associativity: Change the placement of the rotary switch, i.e., set the X translation to zero (Figure 7-17). As expected, the objects move and the footprint waits for a recalculation.

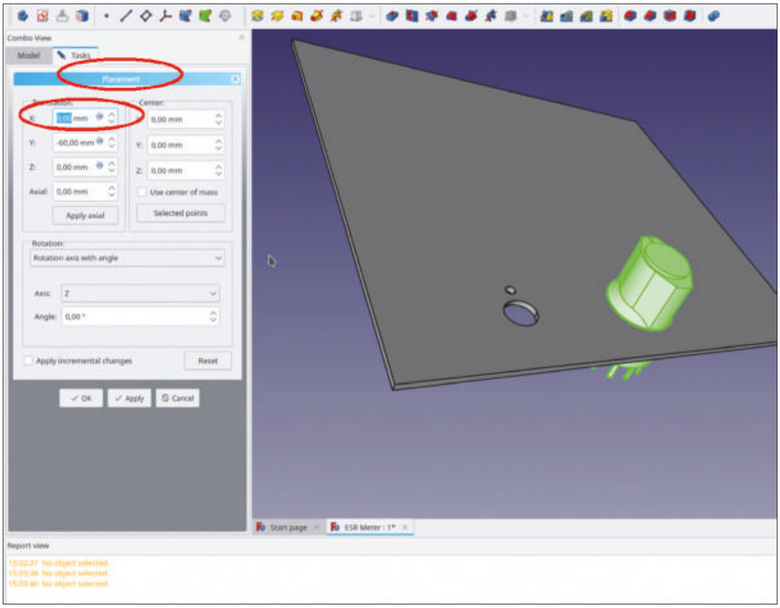


Figure 7-17

To update the front panel, in the task window, either the 'Apply' or the 'OK' button must be pressed (Figure 7-18). If you press the 'Apply' button too early, the objects get deselected. This also happens if you click somewhere in the 3D view. In this case, you need to close and reopen the task window again and start over.

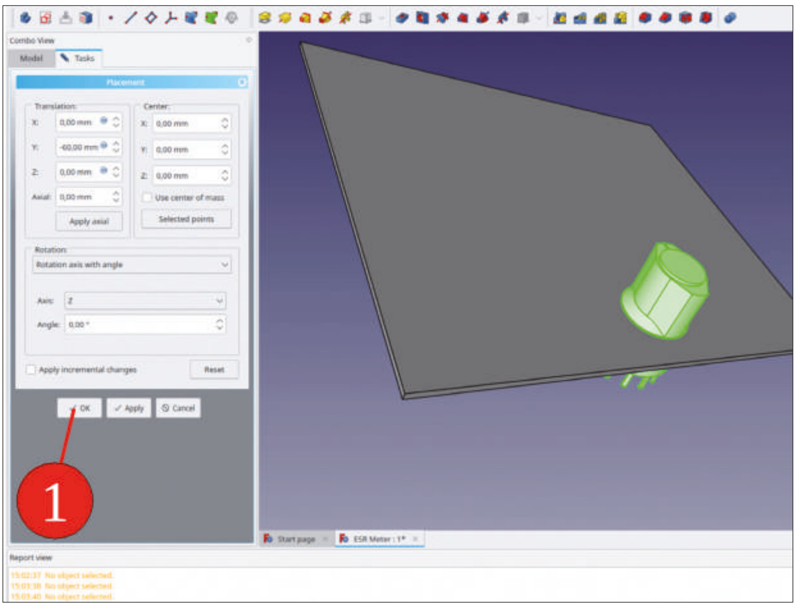


Figure 7-18

As an exercise, you can add the potentiometer with its control knob to the assembly. The steps are similar to the ones for the rotary switch. The coordinates for the potentiometer are: X = 11 mm, Y = –60 mm. The result is shown in Figure 7-19.

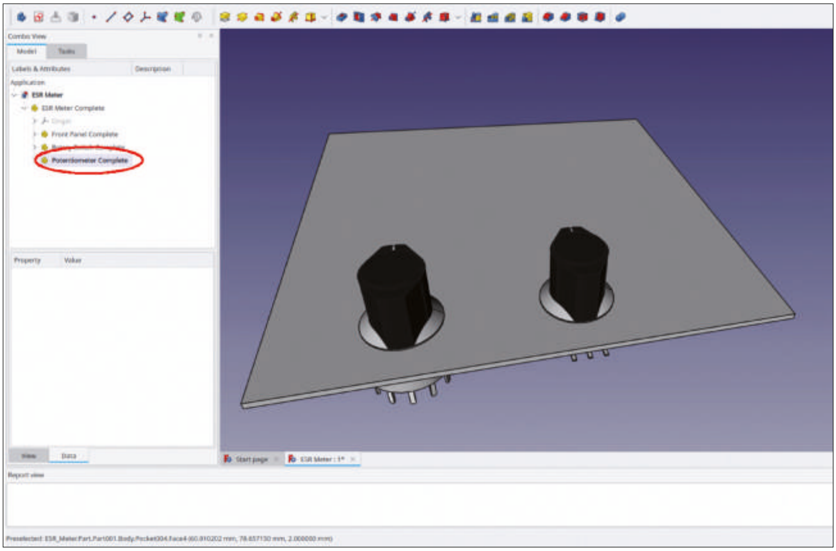


Figure 7-19

On the rear of the front panel, the terminals of the potentiometer point in an unfavorable direction (Figure 7-20).

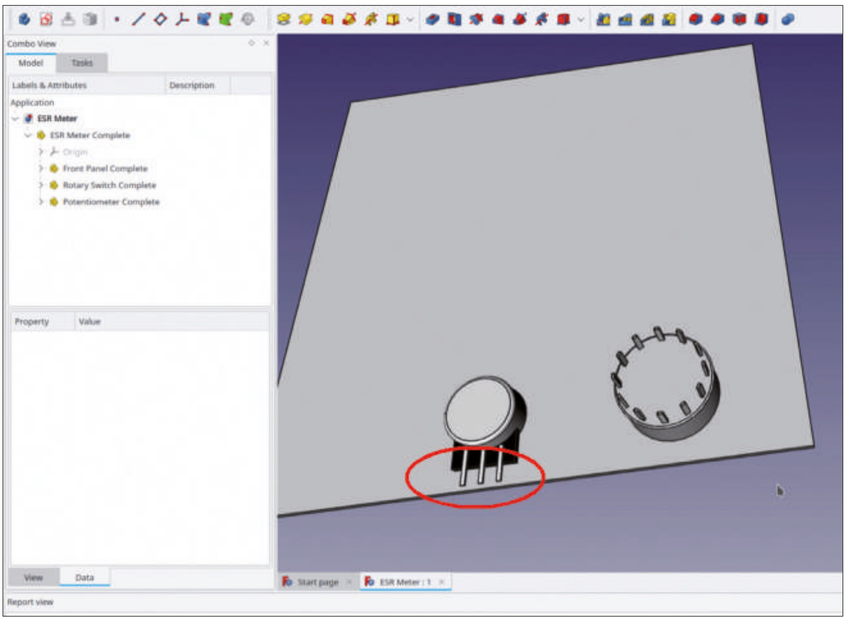


Figure 7-20

The orientation of the potentiometer is easily changed. In the tree view, mark the Std-Part-Container 'Potentiometer Complete'. In the property list, click into the edit field of the 'Placement' field and then on the ... button. Change the rotation angle to –90° (Figure 7-21). In the 3D view, turn the front panel around. You can change the orientation of the control knobs in a similar way. Locate the Std-Part-Containers of the knobs by expanding the corresponding branches of the tree view.

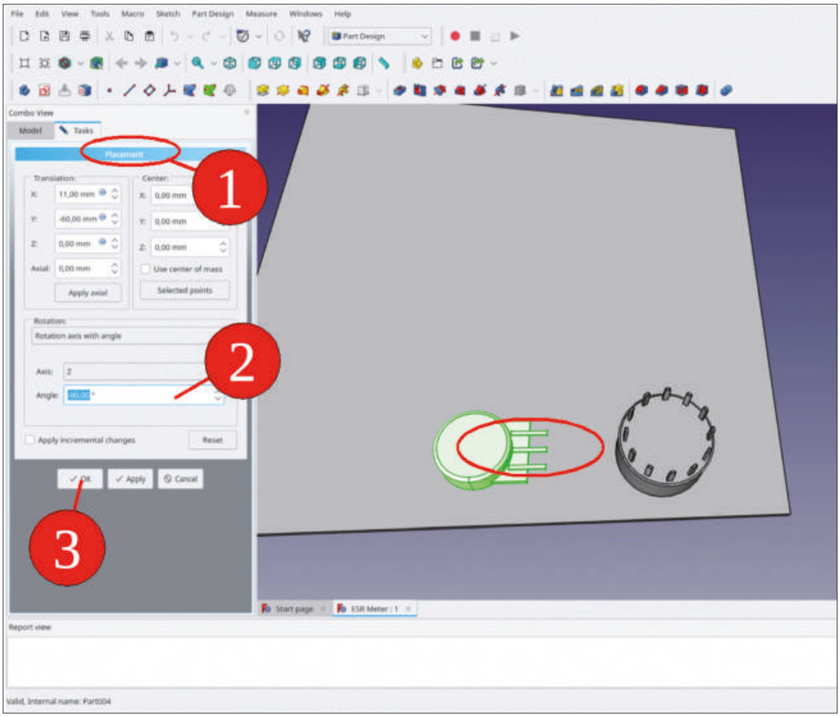


Figure 7-21

7.8 The Binding Posts

1. Locate and open the file 'Sample Projects | Components | Binding Post.FCStd'. Mark the Std-Part-Container 'Binding Post Complete' and copy it, pressing CTRL-C. In the pop-up selection dialog, do not change the standard selection and click the 'OK' button to proceed.

2. Double-click the 'ESR Meter' root node of the tree. Press CTRL-V to insert the Std-Part-Container 'Binding Post Complete'. Drag-and-drop the new Std-Part-Container into 'ESR Meter Complete'.

The binding post is placed too low – the front side insulating washer coincides with the rear of the front panel (Figure 7-22). It should, however, be attached to the front side of the panel.

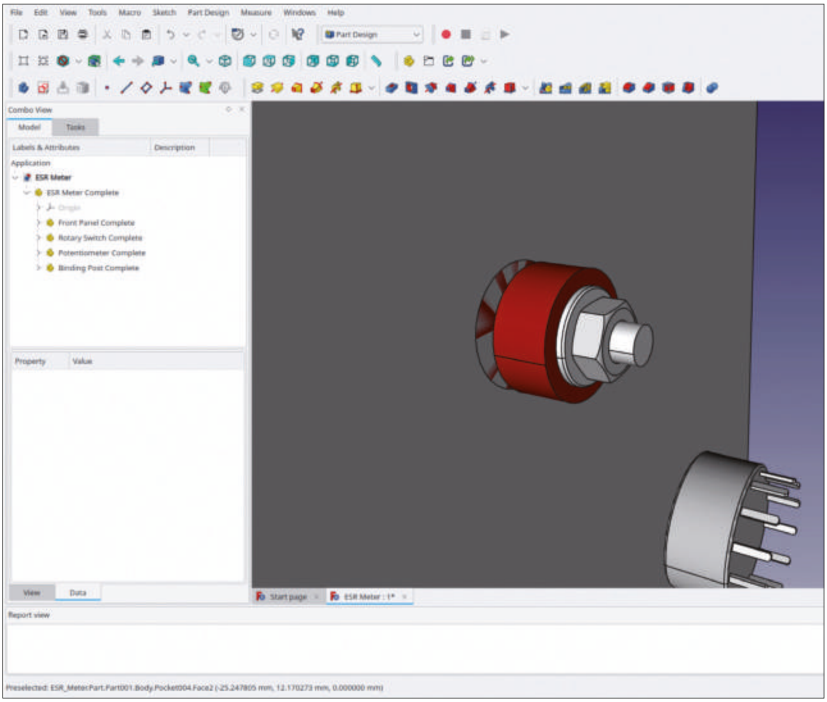


Figure 7-22

To correct the issue, several possibilities exist:

3. The front panel thickness is assumed to never change (there is only one piece of aluminum sheet in the basement). In this case, editing the 'Placement' parameters can transport the binding post to the correct position. For most of the cases, this quick and simple approach is fully sufficient.

4. The front panel is the top Std-Part-Container, and it will not be copied further and pasted into other assemblies. In this case, you can define a blue shape binder which refers to the top face of the front panel. To make that accessible for the attachment of other objects, it can be dragged and dropped to, for example, the parent Std-Part-Container.

An issue may arise though: Should the front panel be copied and pasted later on, then the calculation of the global coordinates will fail with the shape binder. Still, you could use the shape binder while designing the panel and set its 'Tracking Support' to False when the design stage is finished.

5. The most complex case arises if the front panel is a sub-assembly of a larger project and the insertion with copy-and-paste is intended, plus the associativity with the panel thickness has to be provided. As a solution, a green SubShapeBinder could be used. One inherent limitation has yet to be overcome though.

If the SubShapeBinder references an infinite shape like a datum plane (e.g., 'Font Panel Top Face' in the 'Front Panel' body), it will not allow you to attach other objects. As a solution, you can create a sketch of the outline on the 'Front Panel Top Face' and refer the SubShapeBinder to that (finite) object. Then, the attachment of components to the top face of the panel works, and the completed sub-assembly can be copied and pasted into other work (we will do that in another project later).

This discussion seems a tad academic. But let's spend a little extra time now to shed light on these methods so you can proceed faster at a later time. Let's add the sketch to the front panel:

6. Double-click the body called 'Front Panel' (expand the 'Front Panel Complete' Std-Part-Container to locate it). Mark the 'Front Panel Top Face' datum plane and start the sketcher (Figure 7-23).

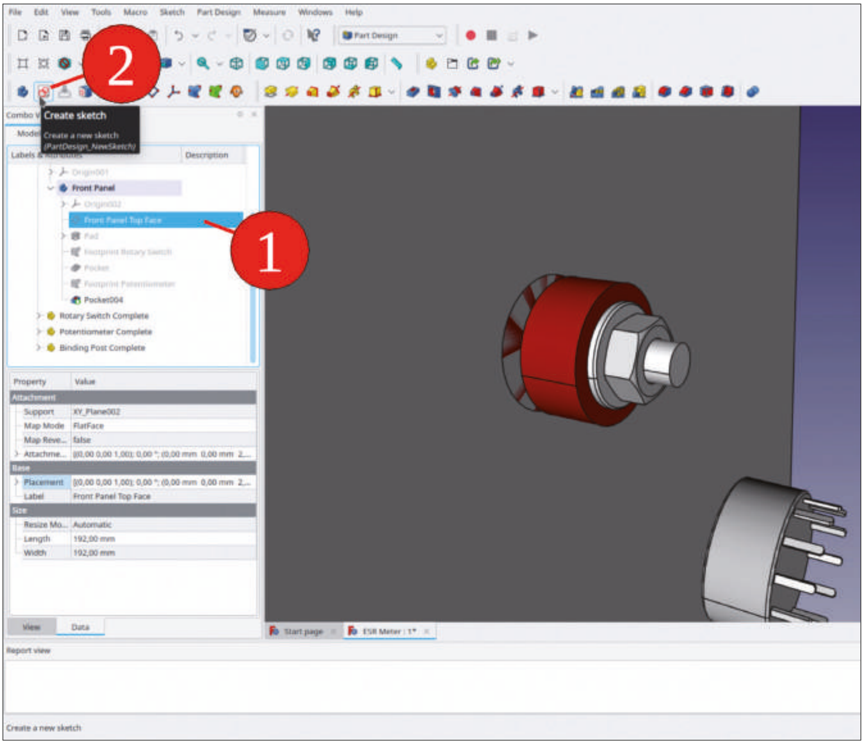


Figure 7-23

7. Click the 'External geometry' tool button and mark all 4 lines of the outer front panel contour (Figure 7-24).

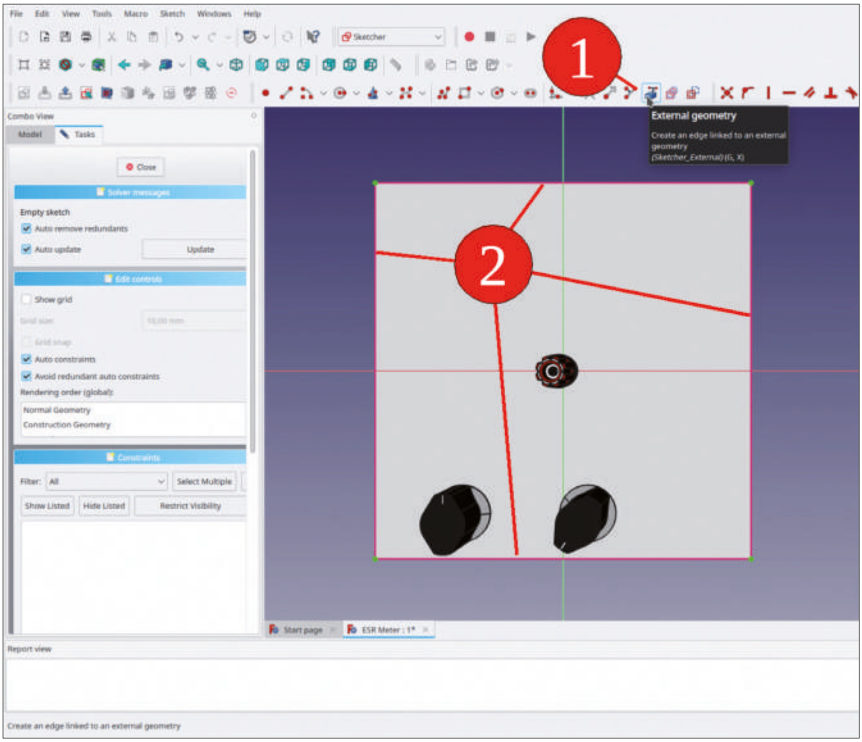


Figure 7-24

8. Click the 'Create a rectangle' tool button and draw a rectangle on top of the violet construction lines (Figure 7-25). Close the sketch with the 'Close' button (top). Rename the sketch to 'Front Panel Top Face Contour'.

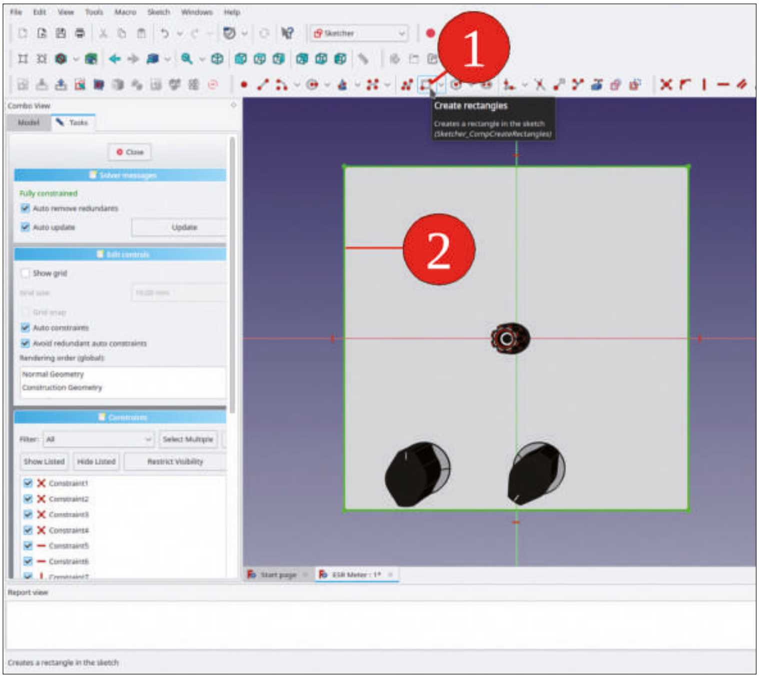


Figure 7-25

9. Now, you create the SubShapeBinder. It will appear in the currently activated object. The SubShapeBinder is needed at a location where it is accessible for other objects. The most meaningful destination is the Std-Part-Container called 'ESR Meter Complete'. To deactivate the body 'Front Panel', right-click it in the tree view, and select 'Toggle active body' from the context menu. The title of the deactivated 'Front Panel' body now appears in normal type.

10. In the deactivated body, mark the sketch 'Front Panel Top Face Contour'. Then, click the green 'Create a sub object(s) shape binder' tool button (Figure 7-26). Hide the sketch 'Front Panel Top Face Contour'.

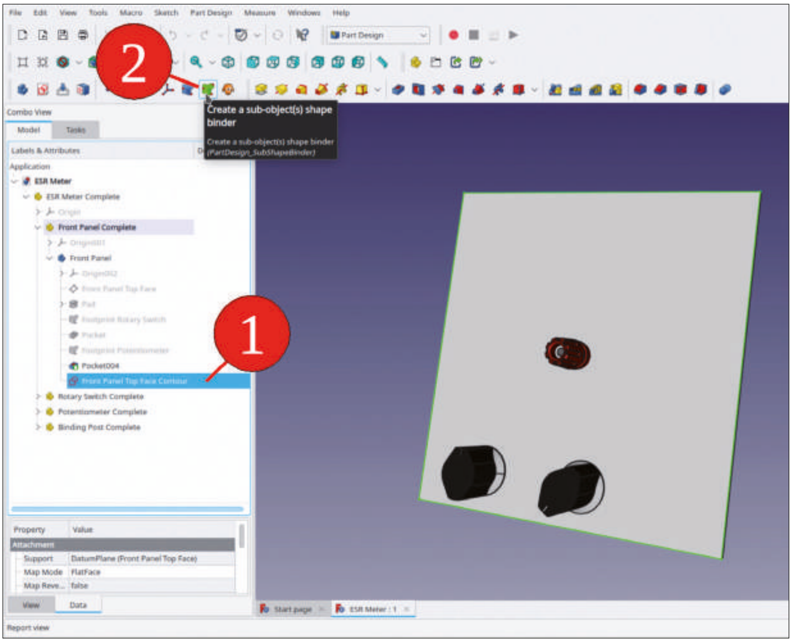


Figure 7-26

11. Rename the new SubShapeBinder to 'Front Panel Top'. Drag-and-drop it in the Std-Part-Container 'ESR Meter Complete' (Figure 7-27). Then, hide the new SubShapeBinder.

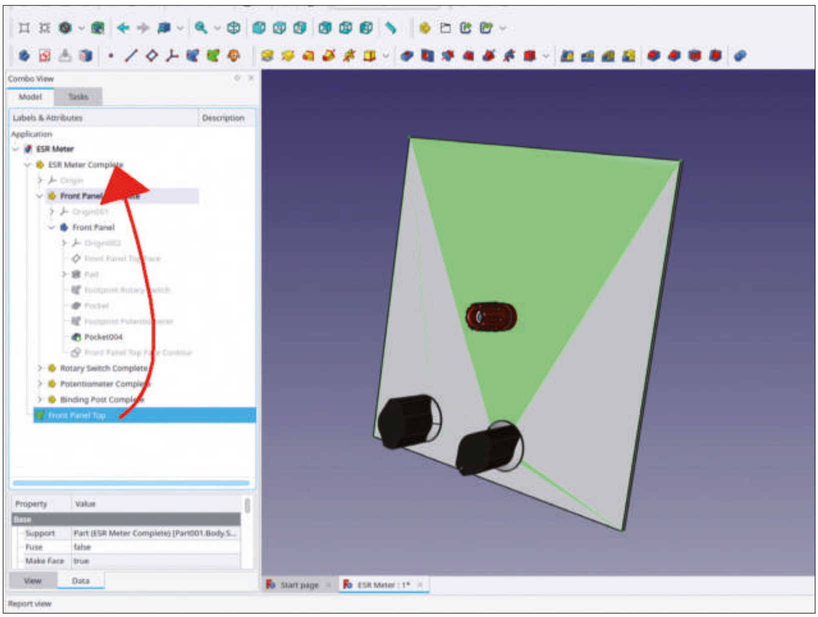


Figure 7-27

Now, you have created a robust reference to the front panel top face. In order to have the maximum use of it, all related elements could be attached to this reference. The resulting associativity would make all attached parts follow any thickness variations of the front panel. As an example, you can correct the positioning of the binding post:

12. In the tree view, mark the Std-Part-Container 'Binding Post Complete'. Then, switch to the 'Part' workbench. From the main menu, select 'Part | Attachment'. In the task window, click into the edit field for Reference 1 (which displays the label 'Selecting...').

13. In the Combo View, switch to the 'Model' tab and click the SubShapeBinder 'Front Panel Top'. Then return to the 'Tasks' tab. From the attachment mode list, select 'XY on plane'. This updates the position of the binding post, which moves to the front face of the front panel (Figure 7-28). Close the task window with the 'OK' button.

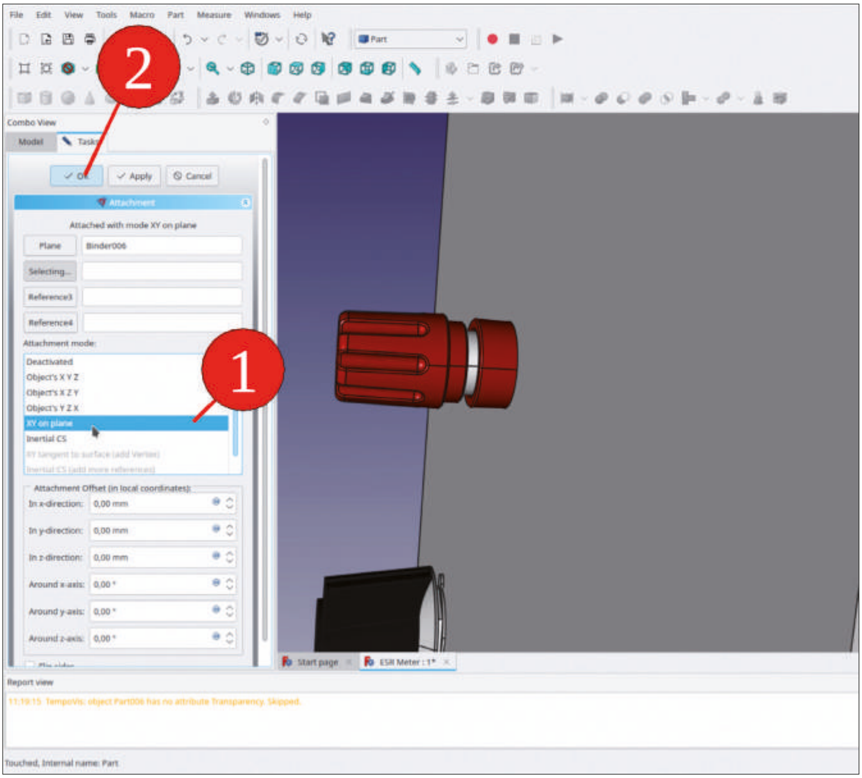


Figure 7-28

14. In the tree view, mark the Std-Part-Container 'Binding Post Complete'. In the property list, click the edit field 'Attachment' (we have to use the attachment now, which overrides the placement). Then, open the task window by clicking

the [...] button to the right. (The task window has the title 'Placement' – slightly confusing, but you can safely ignore this). Enter the following values for the translations: X = 57 mm, Y = 36 mm (Figure 7-29).

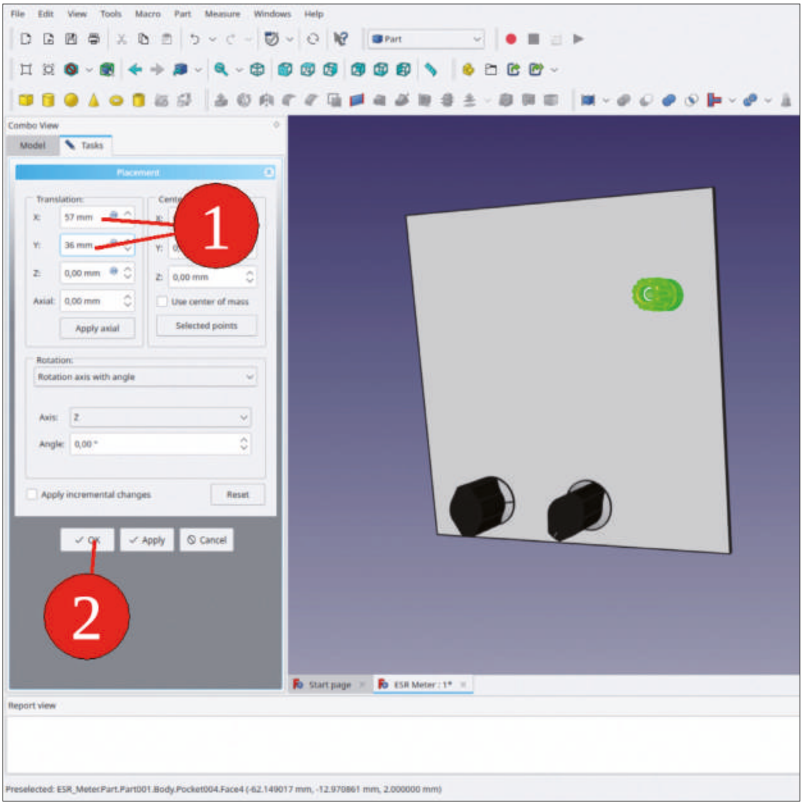


Figure 7-29

Now you can test the associativity so achieved: The front panel thickness is controlled by the datum plane 'Front Panel Top Face', in the 'Front Panel' body object. Mark the datum plane. Open the task window for the 'Attachment' parameters in the property list by clicking into the edit field, and then on the [...] button. If you insert a value like 5 mm for the Z translation, and click the 'Apply' button, the binding post remains correctly placed on the front side of the front panel (Figure 7-30).

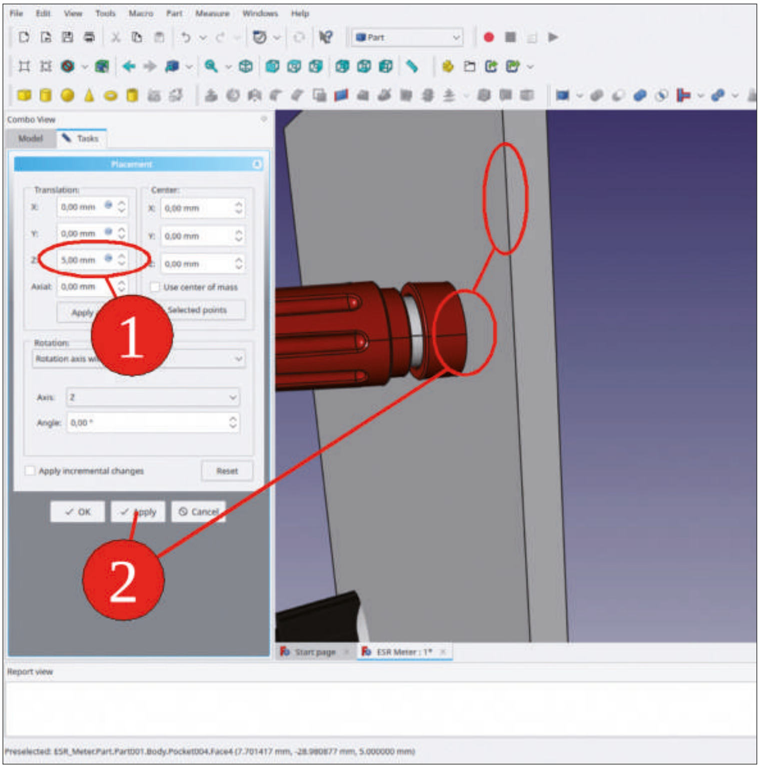


Figure 7-30

As a small drawback, the rear insulating washer now collides with the rear of the front panel. To fix this issue, another attachment relation would need to be declared for the components that refer to the rear face of the front panel. This would increase the length of this step-by-step instruction sequence further, so you may try to perfect the model on your own in the described way (e.g., make the control knobs and fasteners follow the front panel thickness, etc.).

The second binding post can be generated with a link. This saves time. As an additional advantage, changes on the parent object are also transmitted to the linked copies of it (like the placement adaptions on fasteners, appearance, and color selections). In contrast to this, a clone would have resulted in an independent copy of the parent object.

15. In the tree view, rename the binding post container to 'Binding Post Complete 1'.

16. In the tree view, mark 'Binding Post Complete 1' and click the 'Make link' tool button (Figure 7-31). Rename the new Std-Part-Container to 'Binding Post 2'. Drag-and-drop it into the Std-Part-Container 'ESR Meter Complete' (Figure 7-32).

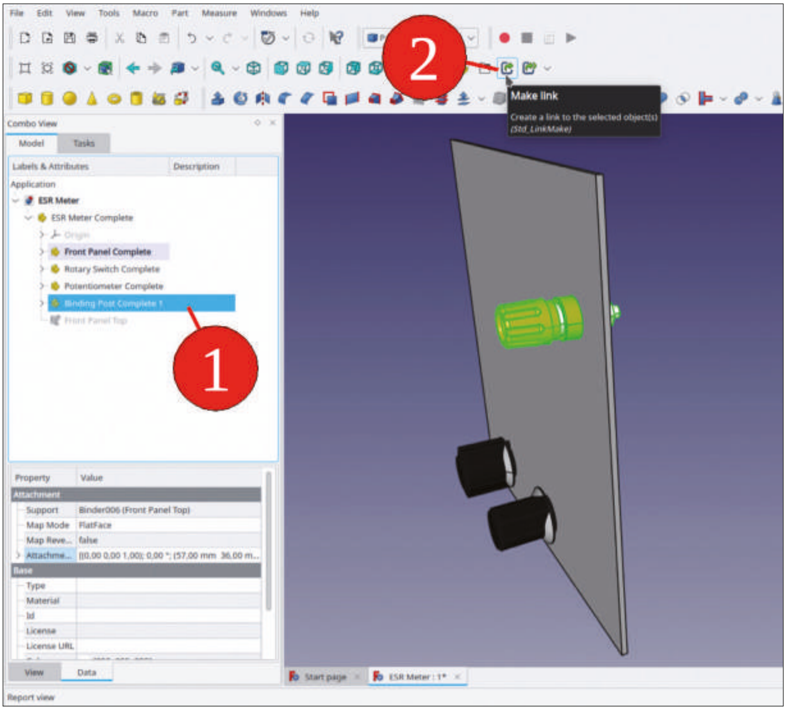


Figure 7-31

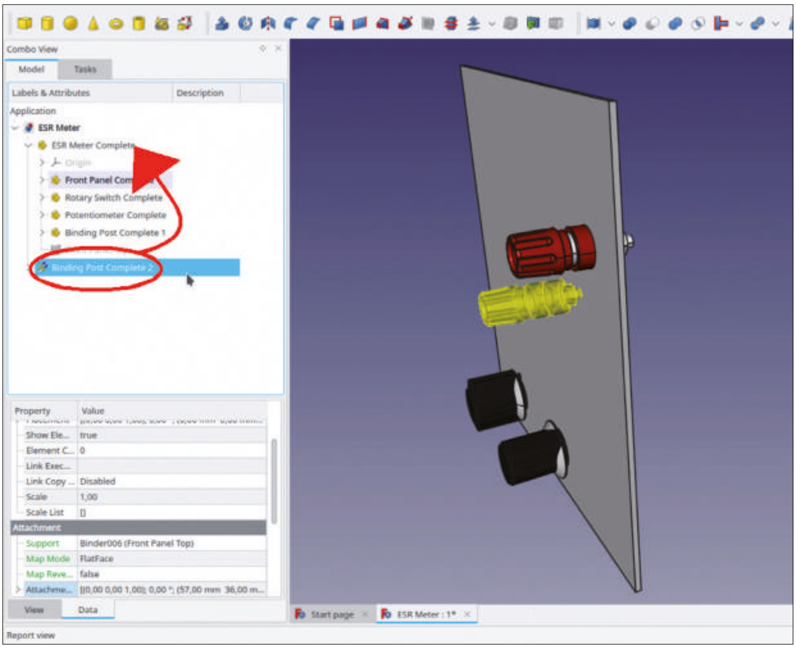


Figure 7-32

17. In the tree view, mark 'Binding Post Complete 2'. In the property list, set the property 'Link Transform' to 'true' (Figure 7-33). With this choice, the 'Link Placement' parameters are defined relative to 'Binding Post Complete 1'.

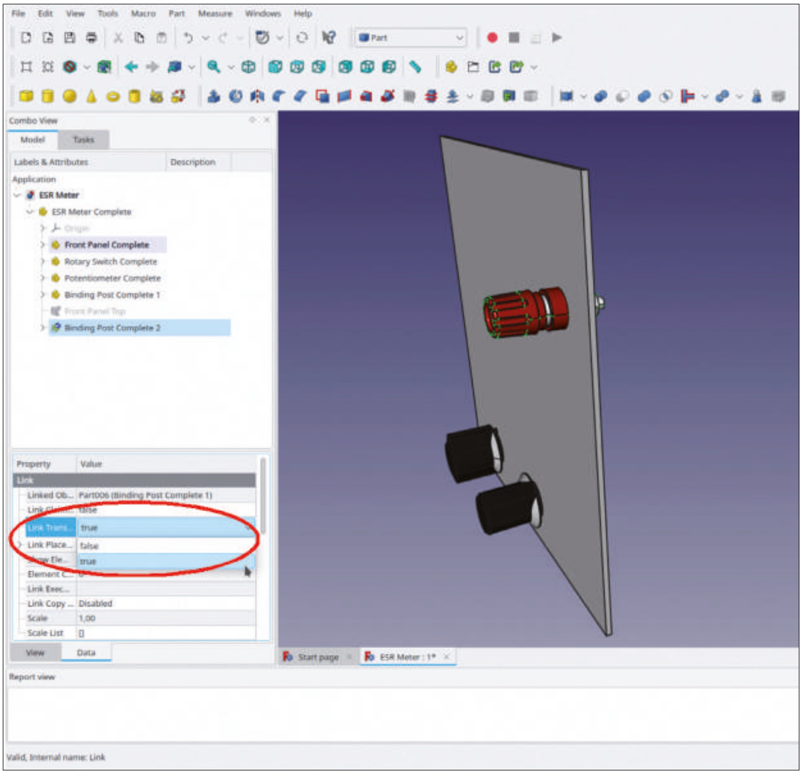


Figure 7-33

18. Edit the 'Link Placement' parameters and enter a value of -38 mm for the Y translation (Figure 7-34).

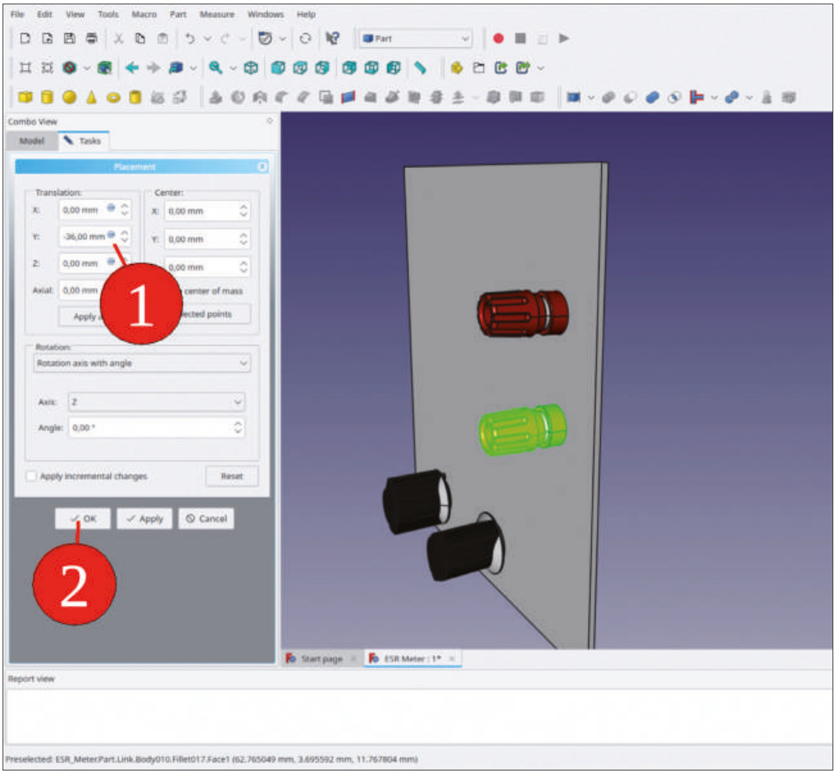


Figure 7-34

19. In the tree view, expand 'Binding Post Complete 1' and hide 'Insulating Washer 2', 'Washer' and 'Nut' (because the second binding post is a linked object, it follows).

20. If the body 'Front Panel' is still activated (its title is shown in bold), deactivate it (otherwise, with the next steps, a cyclic error would be thrown).

21. On the binding posts, mark all edges which define the footprint. Hold down the STRG key for this multiple selection (Figure 7-35).

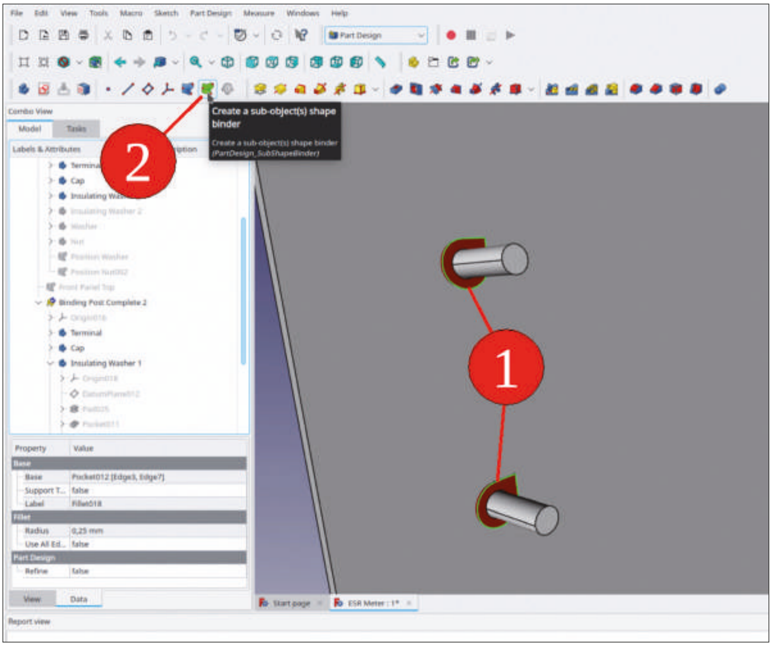


Figure 7-35

22. Click the green 'Create a sub object(s) shape binder' tool button. The shape binder will appear in the root branch of the tree. Rename it to 'Footprint Binding Posts'. Drag-and-drop it into the body 'Front Panel' (Figure 7-36).

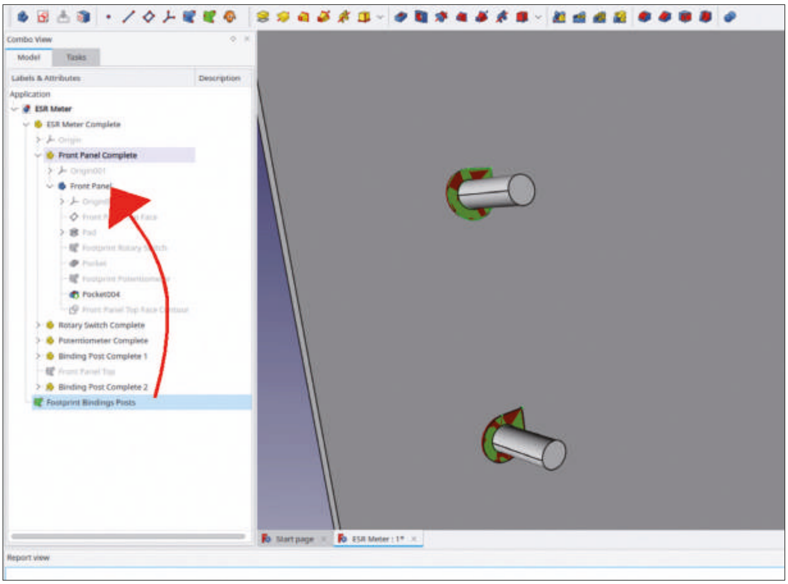


Figure 7-36

23. Double-click the body 'Front Panel' to activate it (title shown in bold letters). In the tree view, mark the SubShapeBinder 'Footprint Binding Posts' and click the 'Pocket' tool button (Figure 7-37). In the task window, select the type 'Through all' and check the 'Reversed' checkbox (Figure 7-38). Close the task window with the 'OK' button.

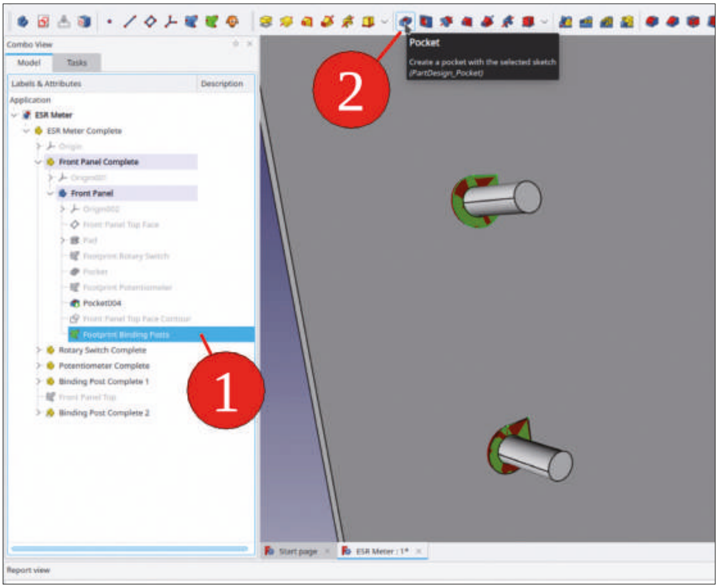


Figure 7-37

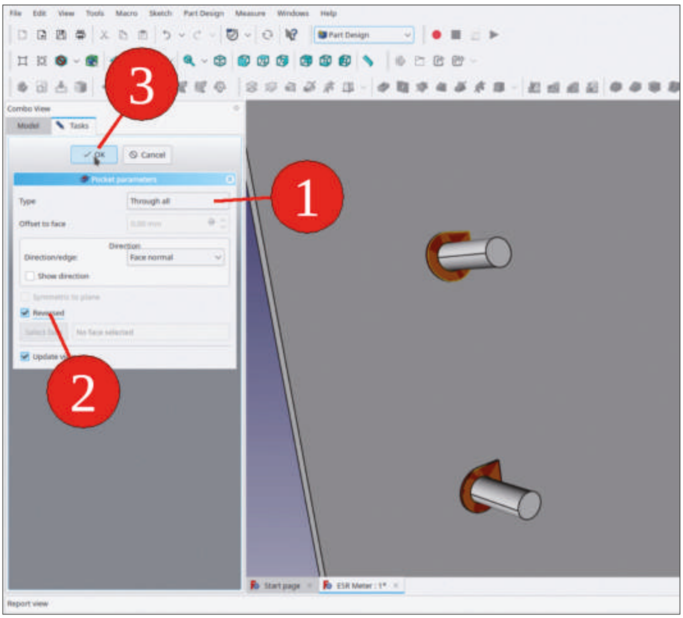


Figure 7-38

24. The punched-out footprint can be shown by hiding 'Binding Post Complete 1' and 'Binding Post Complete 1' (Figure 7-39).

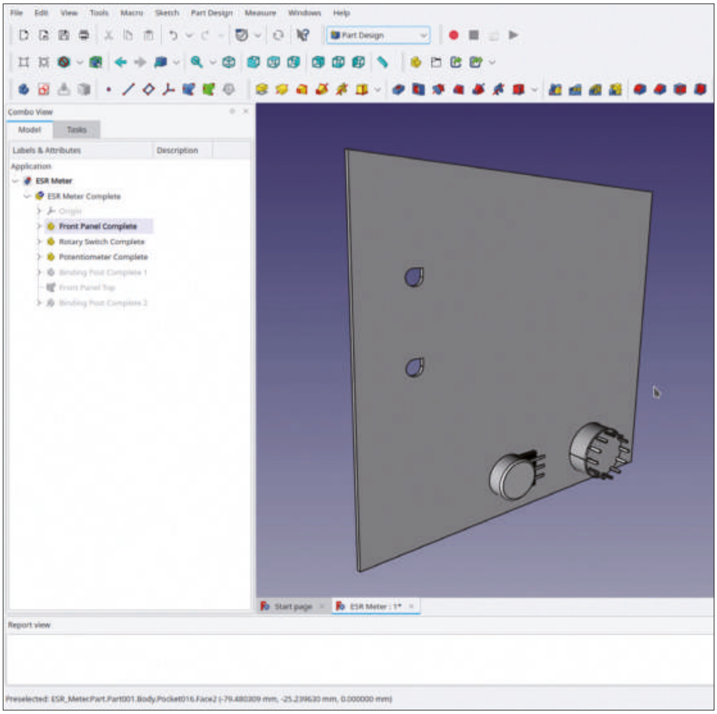


Figure 7-39

25. In the tree view, display 'Insulating Washer 2', 'Washer' and 'Nut' again with the SPACE key. Also, display the Std-Part-Containers 'Binding Post Complete 1' and 'Binding Post Complete 1' again.

7.9 Placement of the Panel Meter

The placement of the panel meter is left here as an exercise for you. The file is contained in 'Sample Projects | ESR Meter | Components'. You can attach it to the 'Front Panel Top' Sub-ShapeBinder, like the binding posts, and select 'XY on plane' as the attachment mode. For the attachment offsets, use: X = –17 mm, Y = 7 mm. The result is shown in Figure 7-40.

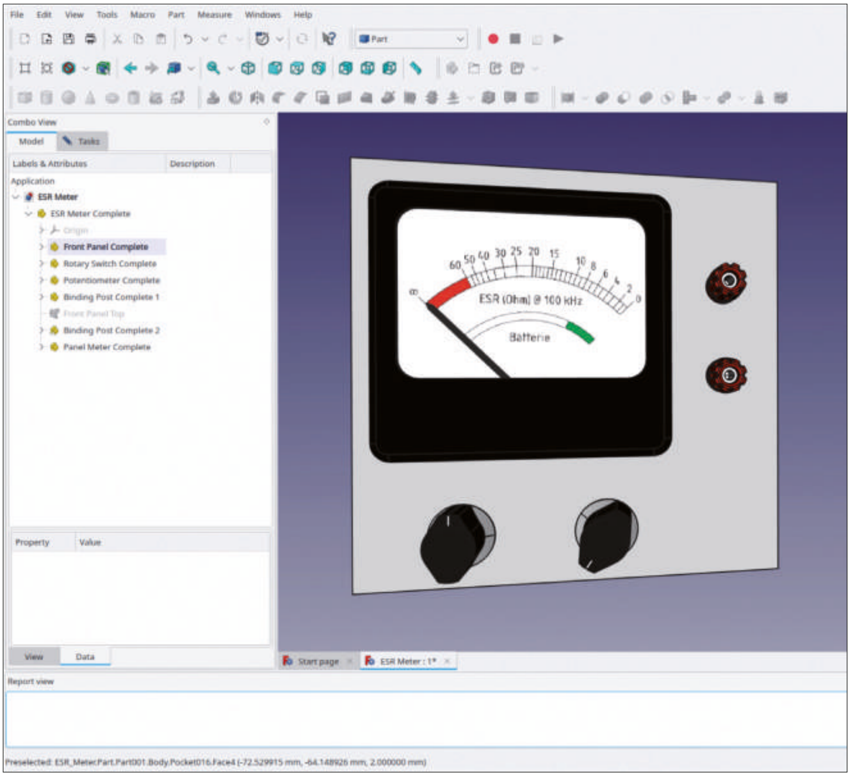


Figure 7-40

For the footprint of the panel meter, the mounting bolts as well as the cylindrical part of the casing must be selected for the definition of the SubShapeBinder. If you select the top faces of these objects, they may reside at different heights. This can lead to an error message later (like 'Curves are not coplanar'), when the 'Pocket' tool is used to model the footprint later. For this reason, it is better to define the SubShapeBinder with the contours at the base of the features, where they are attached to a plane facet to the casing. To proceed, in the tree view, hide the Std-Part-Container 'Front Panel Complete' as well as all the nuts and washers of the panel meter. That exposes all edges necessary for the footprint. Mark them in the 3D view (Figure 7-41). Verify that the 'Front Panel' body is still activated (as the destination for the SubShapeBinder) before you click the 'Create a sub object(s) shape binder' tool button. Locate the new SubShapeBinder at the destination (with larger project trees, this invokes scrolling back and forth). Rename it to 'Footprint Panel Meter'.

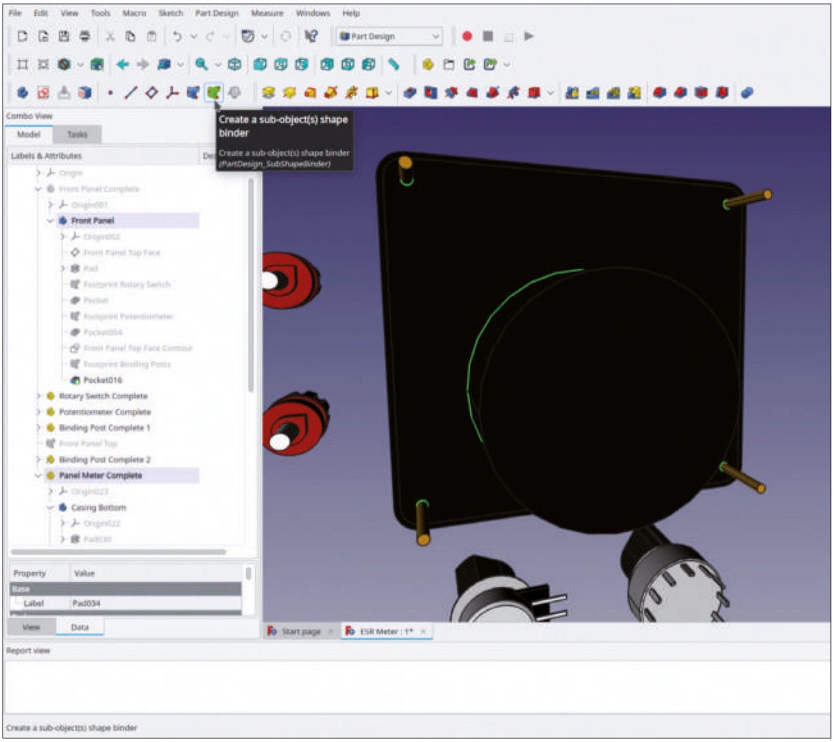


Figure 7-41

Mark the new SubShapeBinder and click the 'Pocket' tool button. In the task window, select the type 'Through all'. Because the pocket extends from the front side towards the back, the checkbox 'Reversed' can be left unchecked in this case (Figure 7-42). The punched-out footprint is shown by hiding the panel meter in Figure 7-43.

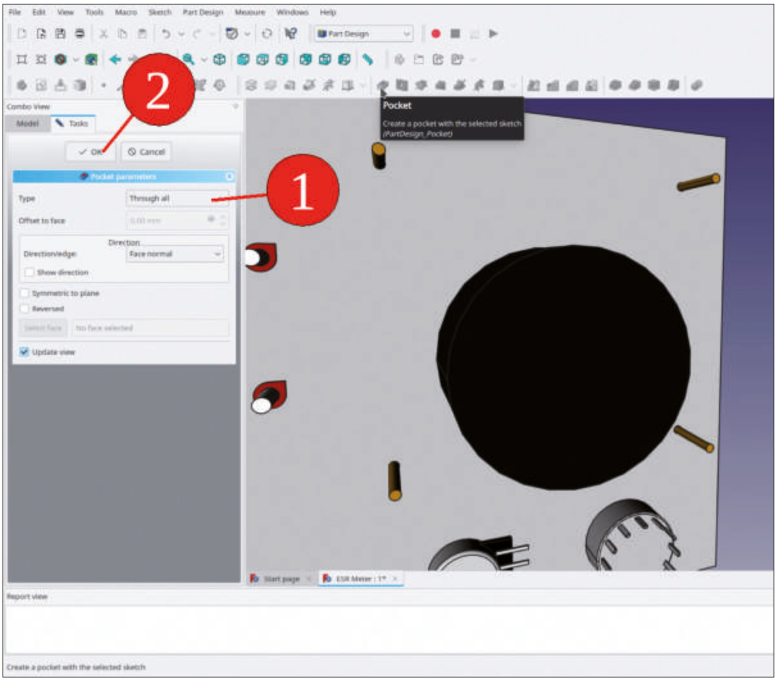


Figure 7-42

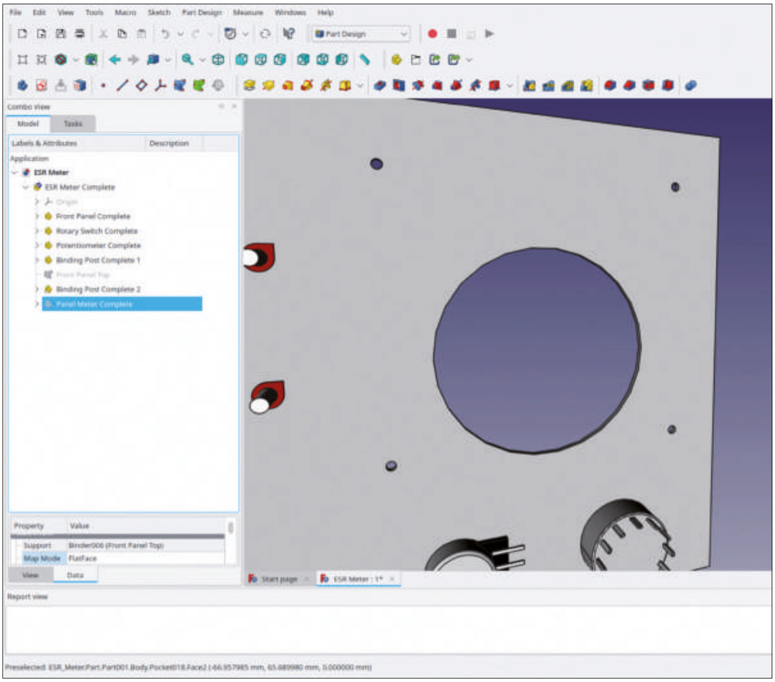


Figure 7-43

7.10 Placing Components to the Rear Surface: The Anchor Plates

When the design grows on the computer screen, it is also good to conceive how the wiring should proceed later. Some anchor plates can hold the wiring harness nicely in place with some cable ties strapped to it.

In most cases, the anchor plates reside on the rear face of the front panel, like with our project. The following table summarizes the positions and rotation angles:

Anchor	X [mm]	Y [mm]	Rotation around Z [°]
1	−13	−23	−90
2	46	−23	−90
3	−29.5	−2	90
4	−29.5	−14	180

1. In the directory 'Sample Projects | ESR Meter | Components', locate and open the file 'Anchor Plate'. Copy the Std-Part-Container 'Anchor Plate Complete' and press CTRL-C. In the pop-up selection dialog, do not change the default selection and click 'OK' to proceed.

2. In the tree view, double-click the root node 'ESR Meter'. Insert the Std-Part-Container with the anchor plate by pressing CTRL-V. Drag and drop the new Std-Part-Container into the Std-Part-Container 'ESR Meter Complete'.

3. In the tree view, mark the 'Anchor Plate Complete' Std-Part-Container, and switch to the 'Part' workbench. From the main menu, select 'Part | Attachment'.

4. In the task window, click into the edit field next to 'Reference 1' (the button is shown in dark gray, with the label 'Selecting...'). Click on the 'Model' tab. In the tree view, expand the coordinate system node of the Std-Part-Container 'ESR Meter Complete'. Click on the XY plane and return to the 'Tasks' tab. For the attachment mode, select 'XY on plane'. For the attachment offsets, enter the values from the table above. Finally, check the 'flip sides' checkbox, as the anchor plate is attached from the back (Figure 7-44). Close the task window with the 'OK' button.

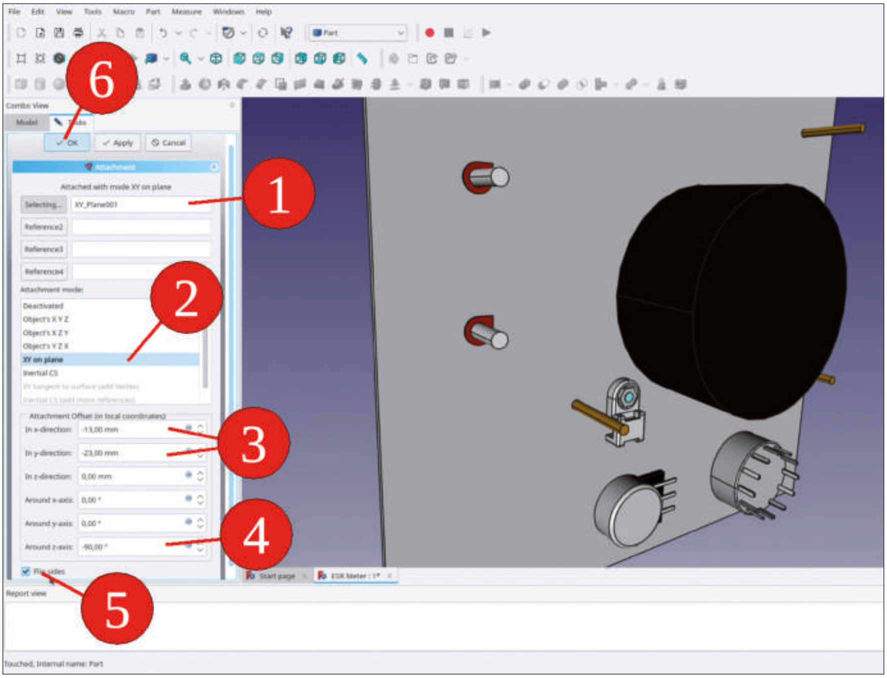


Figure 7-44

5. In the tree view, rename the anchor plate to 'Anchor Plate Complete 1'.

6. In the tree view, mark 'Anchor Plate Complete 1' and click the 'Make link' tool button (Figure 7-45).

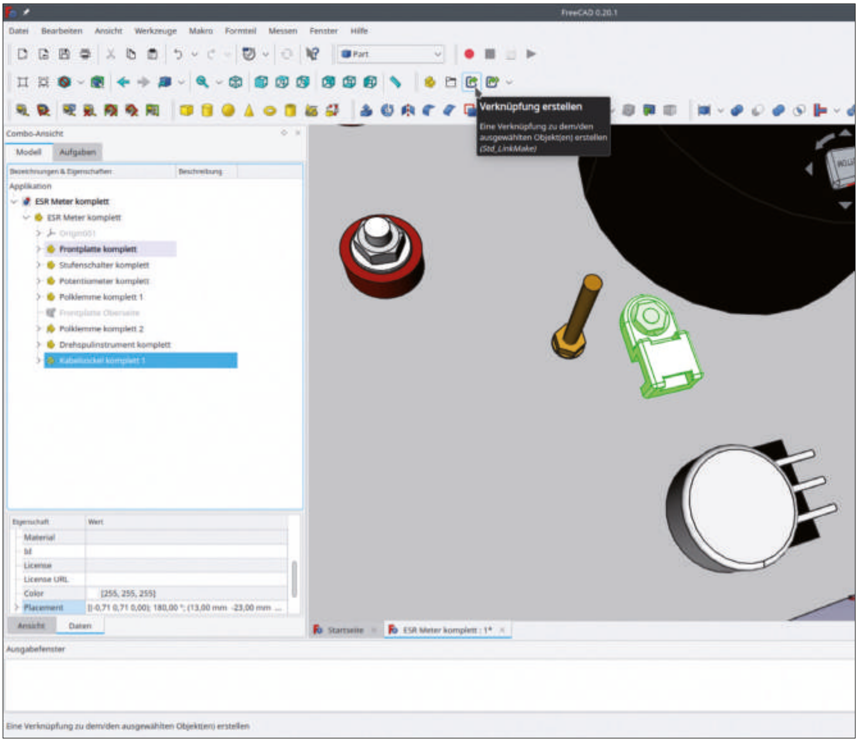


Figure 7-45

7. Drag-and-drop the new anchor plate into 'ESR Meter Complete'. Rename it to 'Anchor Plate Complete 2'. Repeat the positioning, as it was done for the first anchor plate in steps 3 and 4, using the attachment offset values from the table above.

8. In a similar way, create the anchor plates 3 and 4. The result is shown in Figure 7-46.

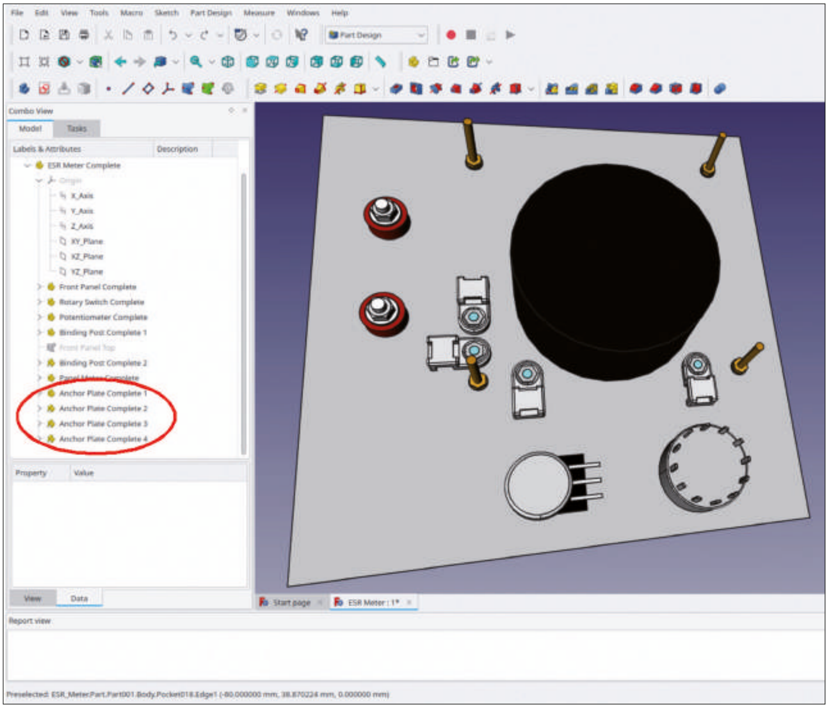


Figure 7-46

9. For the mounting holes in the front panel, a SubShapeBinder has to be defined. If necessary, double-click the 'Front Panel' body (In 'Front Panel Complete'), to activate it. This is setting it as the destination for the SubShapeBinder, created in the next steps.

10. In 'Anchor Plate Complete 1', hide the screw and the nut. The linked objects follow automatically. Very convenient!

11. Mark the edges of the holes in the anchor plates. Then, click the 'Create a sub object(s) shape binder' tool button (Figure 7-47).

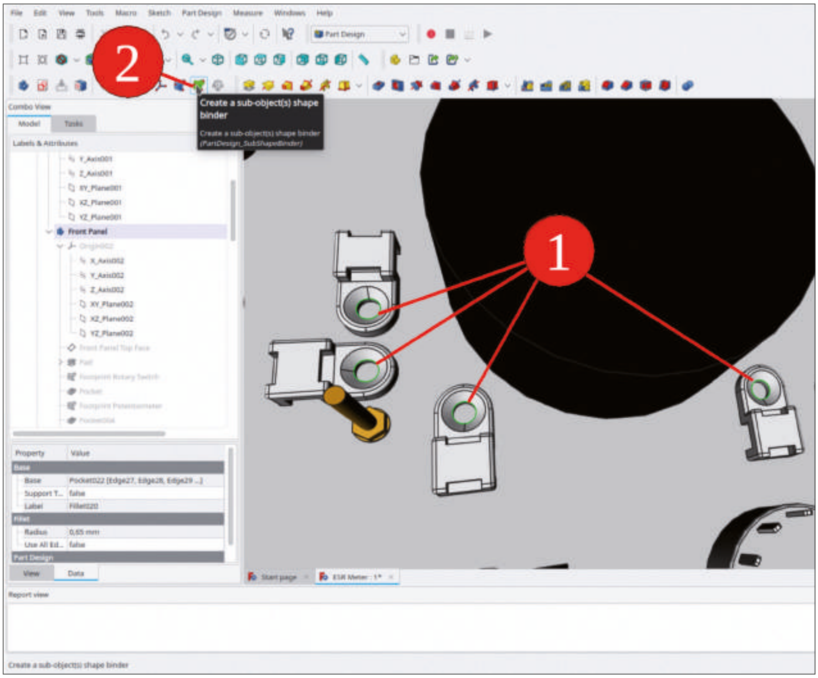


Figure 7-47

12. In the body 'Front Panel', rename the new SubShapeBinder to 'Footprint Anchor Plates'. Mark the new SubShapeBinder. In the property list, set the parameter 'Make face' to 'false'.

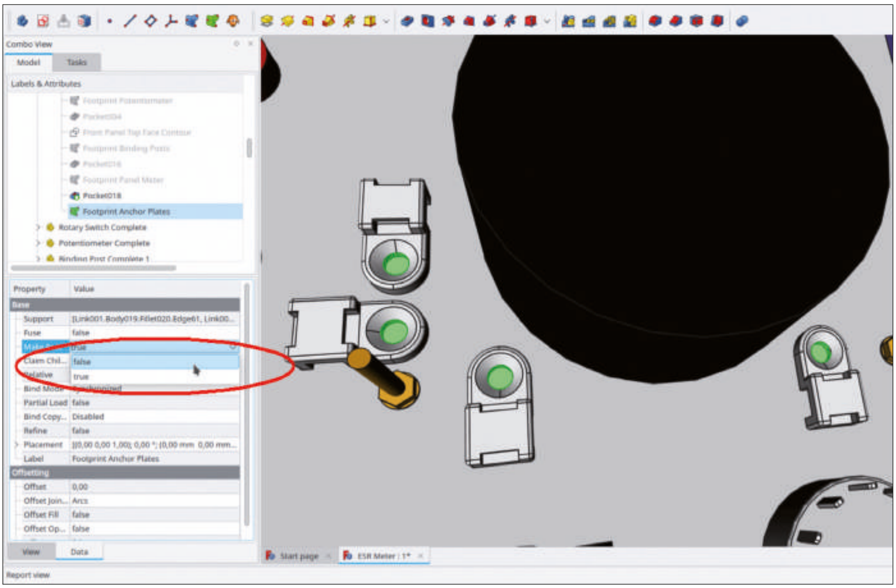


Figure 7-48

13. In the tree view, mark the new SubShapeBinder, and click the 'Hole' tool button (Figure 7-49).

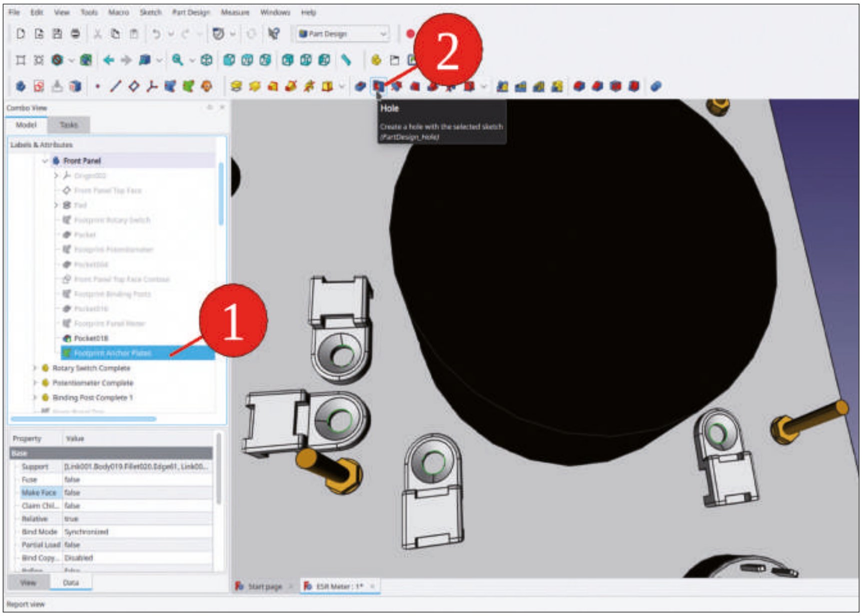


Figure 7-49

14. In the task window, enter a diameter of 3.2 mm and select 'Through all' for the depth. Check the 'Reversed' checkbox (Figure 7-50). Close the task window with the 'OK' button.

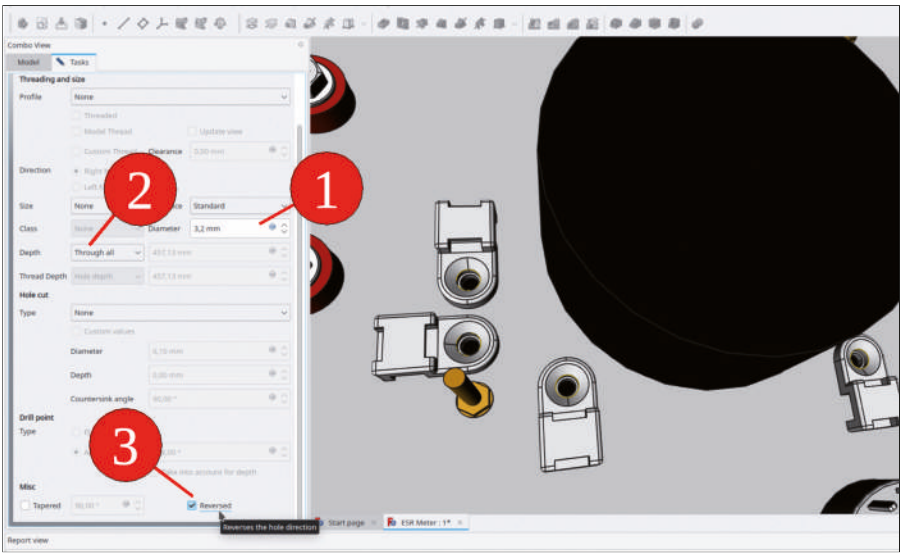


Figure 7-50

15. In the tree view, expand 'Anchor Plate Complete 1'. Display the screw and the nut again with the SPACE key.

7.11 Inserting the Battery Holder

Like the anchor plates, the battery holder is attached to the rear of the front panel. As an example, we've selected a sheet metal design in order to highlight some of the nice 'Sheet Metal' workbench capabilities.

This decision could be debated: Batteries usually leak an aggressive liquid when they fail. This leakage could attack and corrode the aluminum pieces. To increase the longevity of the instrument, a plastic sheet could be wrapped around the battery for protection. As an alternative, the battery holder could be redesigned and made from plastic using a 3D printer. To proceed with the example case, let's stick to the aluminum version.

16. Run the standard procedure: In the directory 'Sample Projects | ESR Meter | Components', locate and open the file '9V Battery Holder'. Copy and paste the '9V Battery Holder Complete' Std-Part-Container into the 'ESR Meter' document tree. Then, drag and drop it into the ESR Meter complete' Std-Part-Container (Figure 7-51). Close the '9V Battery Holder' document tree.

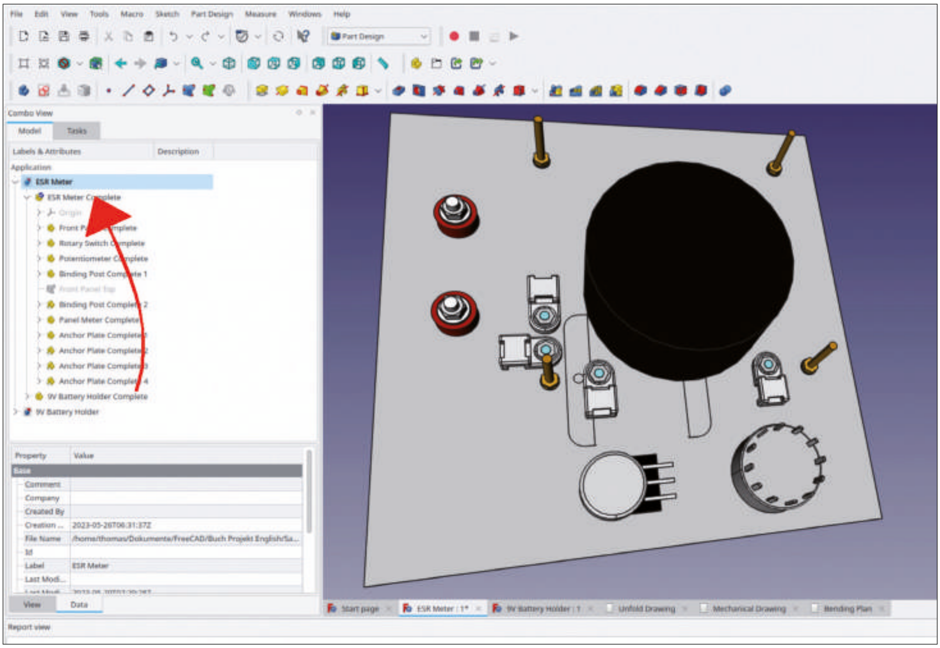


Figure 7-51

17. Switch to the 'Part' workbench. In the tree view, mark the '9V Battery Holder Complete' Std-Part-Container. From the main menu, select 'Part | Attachment'. Click into the edit field for Reference 1, of which the label reads 'Selecting...' (Figure 7-52). Switch to the 'Model' tab.

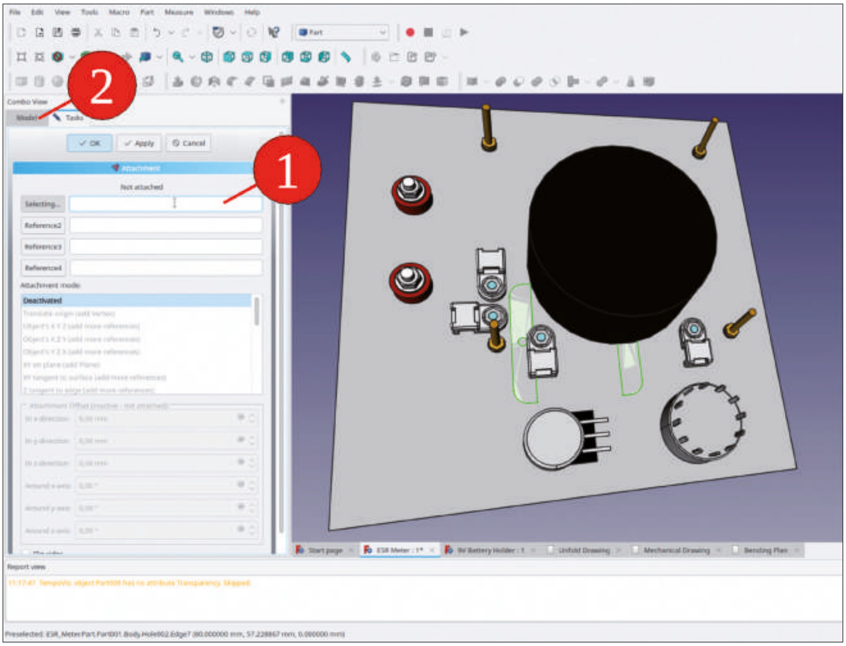


Figure 7-52

18. In the tree view, select the XY plane of 'ESR Meter Complete' (Figure 7-53).

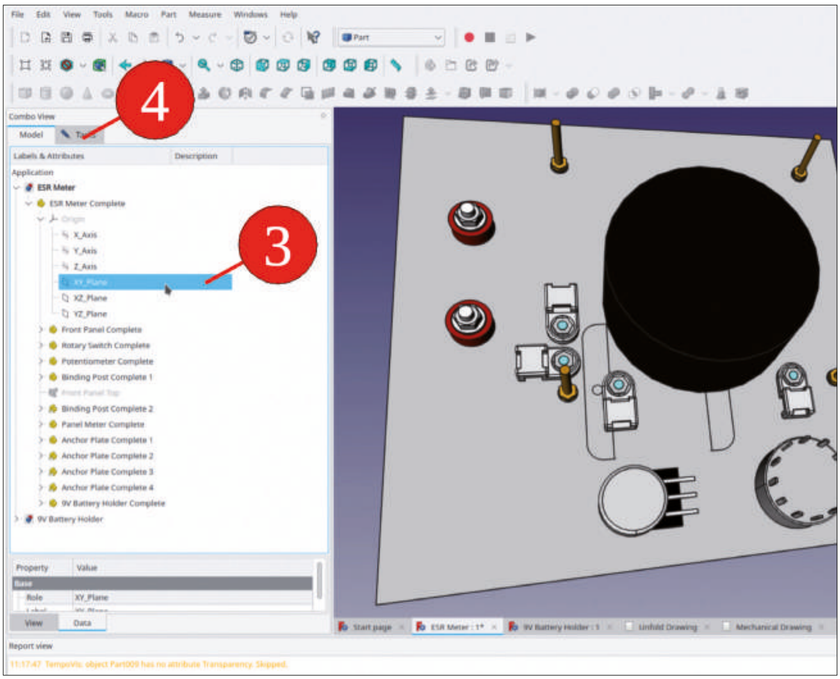


Figure 7-53

19. Return to the 'Tasks' tab. For the attachment mode, select 'XY on plane'. For the attachment offsets, enter X = −24 mm, Y = −52 mm and a rotation angle of −90° around the Z axis. Check the 'Flip sides' checkbox. (Figure 7-54). Close the task window with the 'OK' button.

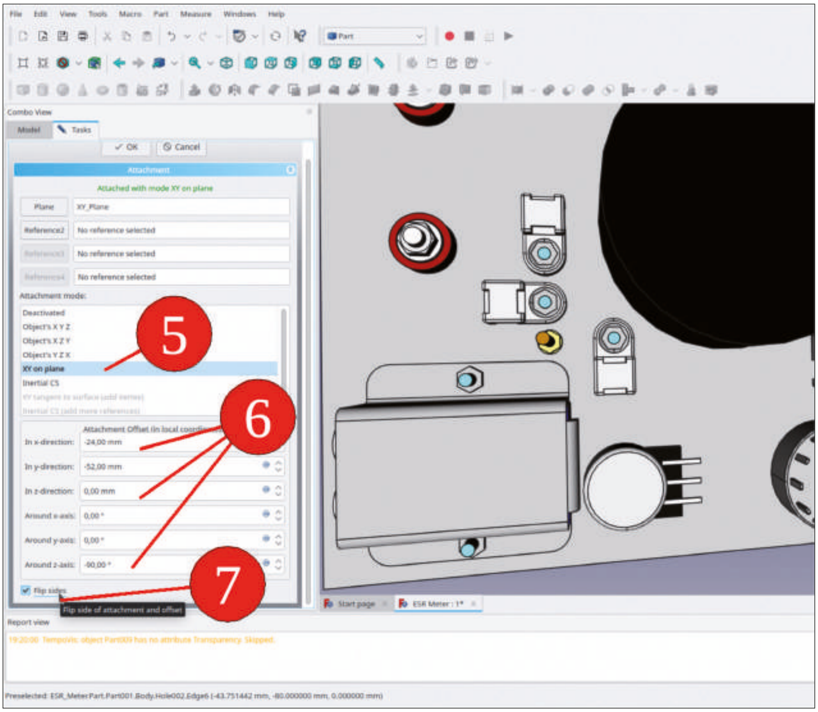


Figure 7-54

20. In order to place the mounting holes for the battery holder, in the tree view, double-click the 'Font Panel' body. This again sets the destination for the Sub-ShapeBinder which you are about to create.

21. In the tree view, expand the '9V Battery Holder Complete' Std-Part-Container. Hide all screws and nuts with the SPACE key to expose the view of the hole edges. Scroll up to the 'Front Panel Complete' Std-Part-Container and double-click the 'Front Panel' body to activate it, and to set it thereby as the destination for the SubShapeBinder.

22. Hold down the CTRL key and, in the 3D view, mark both mounting hole edges on the battery holder (Figure 7-55). Then, click the green 'Create a sub object(s) shape binder' tool button.

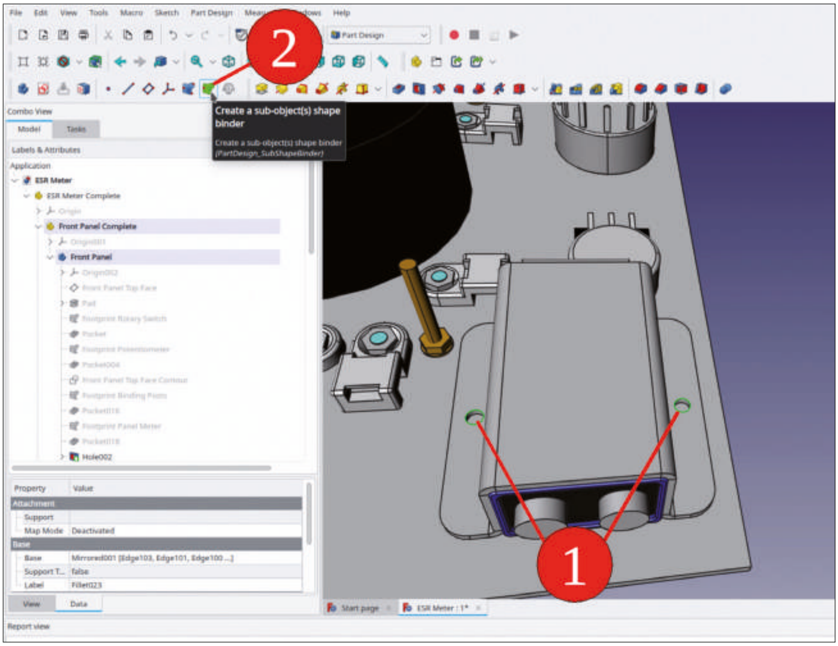


Figure 7-55

23. Rename the new shape binder (scroll back to the 'Front Panel' body object, the tree view keeps jumping) to 'Footprint Battery Holder'. In the property list, set the 'Make face' property to 'false' (Figure 7-56).

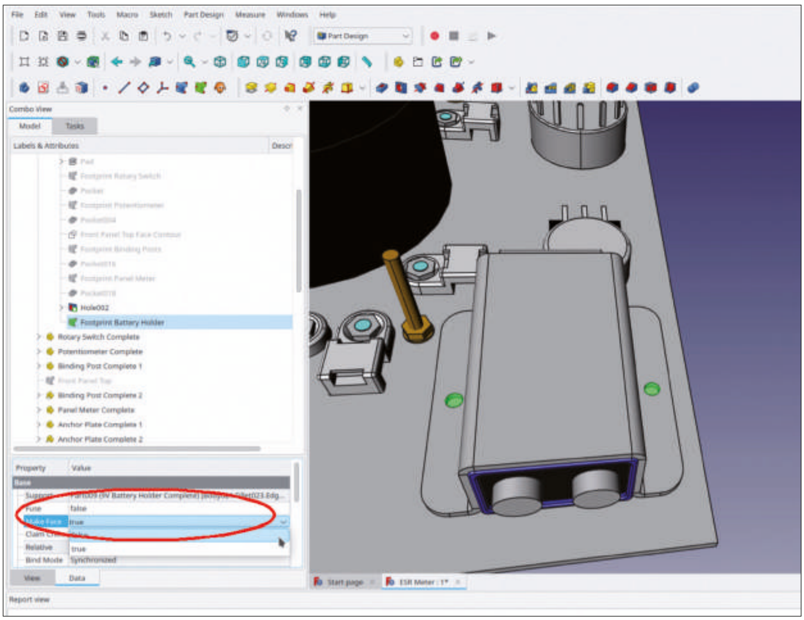


Figure 7-56

24. In the tree view, mark the new SubShapeBinder, and click the 'Hole' tool button. In the task window, enter for the diameter 3.2 mm, and select for the depth 'Through all'. Also, check the 'Reversed' checkbox (Figure 7-57). Close the task window with the 'OK' button.

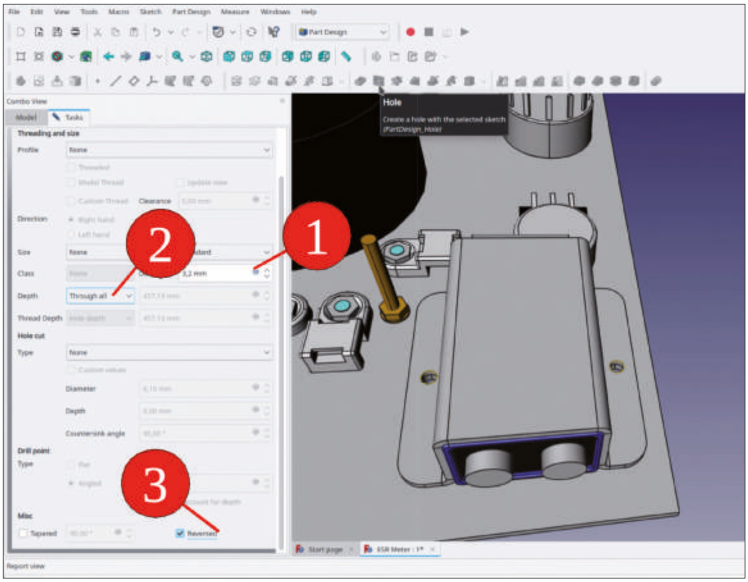


Figure 7-57

25. In the tree view, mark the fasteners for the 9V battery holder, and display them with the SPACE key (Figure 7-58).

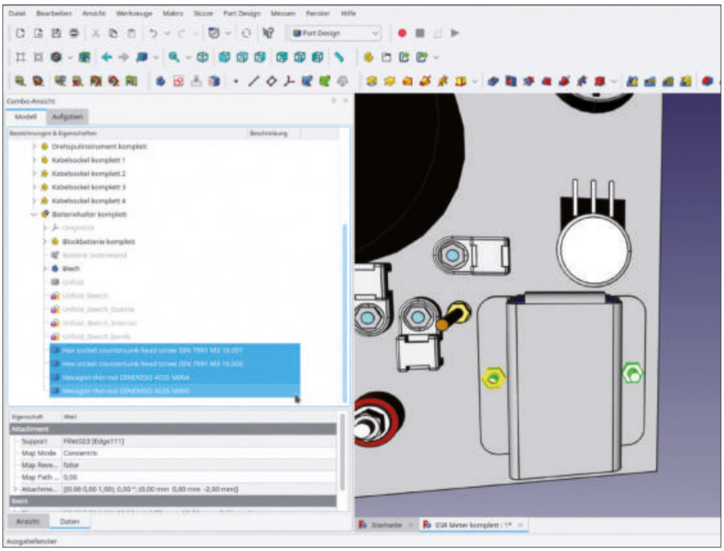


Figure 7-58

7.12 Inserting the Circuit Board

A fascinating opportunity with FreeCAD comes into play when circuit boards designed with KiCad [DAL 2022] are combined with mechanical elements. KiCad has an option to export a 3D model of the circuit board as a STEP file. Importing to FreeCAD is achieved with only a few mouse clicks:

1. In the directory 'Sample Projects | ESR Meter | Components', locate and open the file 'ESR_Meter_001.step'. FreeCAD creates a new document and inserts the STEP file contents in an Std-Part-Container. Rename the container to 'ESR Meter PCB Complete' (Figure 7-59).

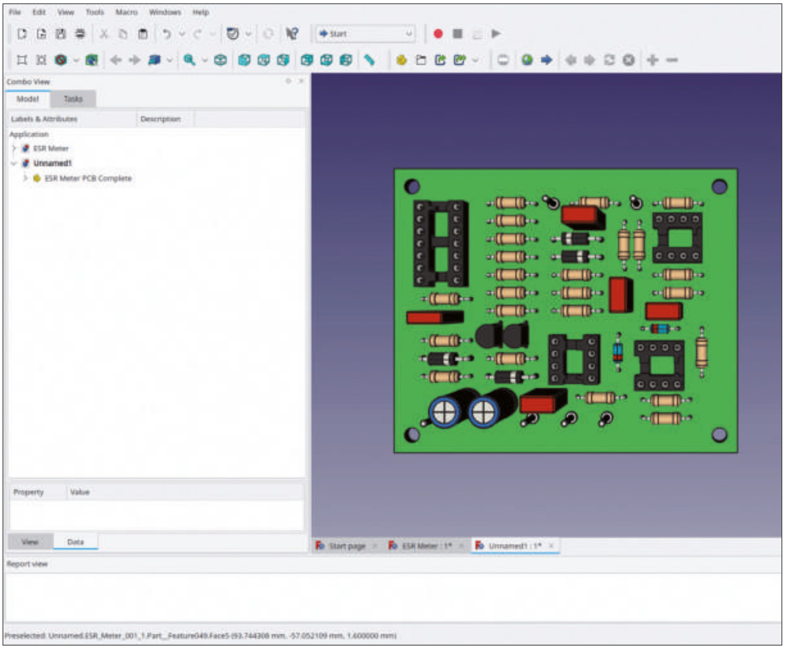


Figure 7-59

2. Copy the renamed Std-Part-Container with CRTL-C.

3. Double-click the document node 'ESR Meter' to set the destination for the paste command.

If you forget this step, the components get copied into the source again. That can involve many objects due for manual removal afterwards. Eventually, the helpful CTRL-Z will undo the result as well. It is a good practice to close the source document node after the Std-Part-Container got copied.

4. Drag and drop the new Std-Part-Container into 'ESR Meter Complete'. That's all there is to it – simple and efficient (Figure 7-60).

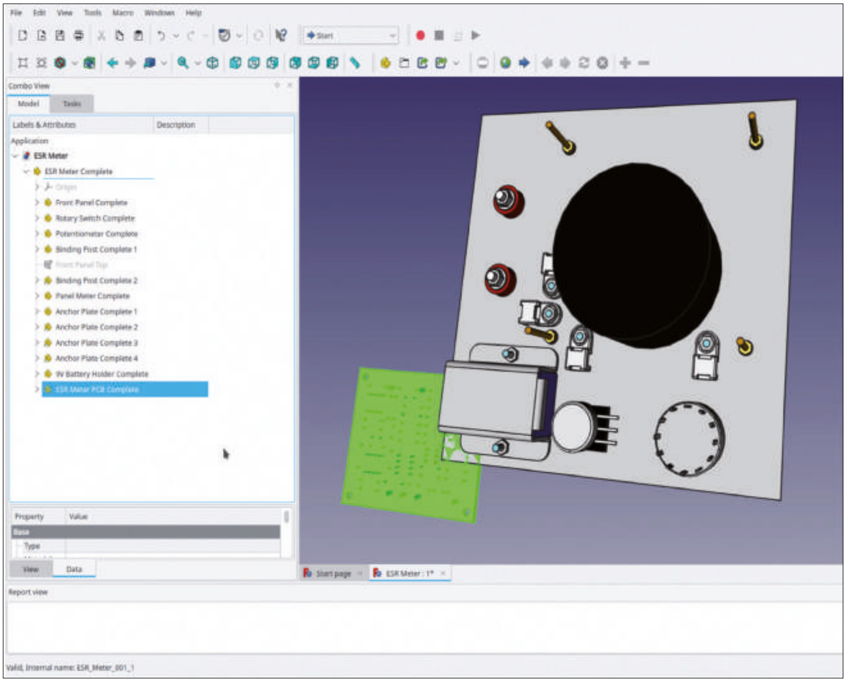


Figure 7-60

5. Now, you attach the circuit board to the assembly: Mark the Std-Part-Container 'ESR Meter PCB Complete' and switch to the 'Part' workbench. From the main menu, select 'Part | Attachment'. In the task window, the collector for 'Reference 1' is already activated. From the tree view in the 'Model' tab, select the XY plane to the 'ESR Meter Complete' Std-Part-Container. Return to the 'Tasks' tab. Check the 'Flip sides' checkbox and enter the following attachment offsets: X = -122 mm, Y = –78 mm, Z = 18 mm, Rotation about Z = 90° (Figure 7-61). Close the task window with the 'OK' button.

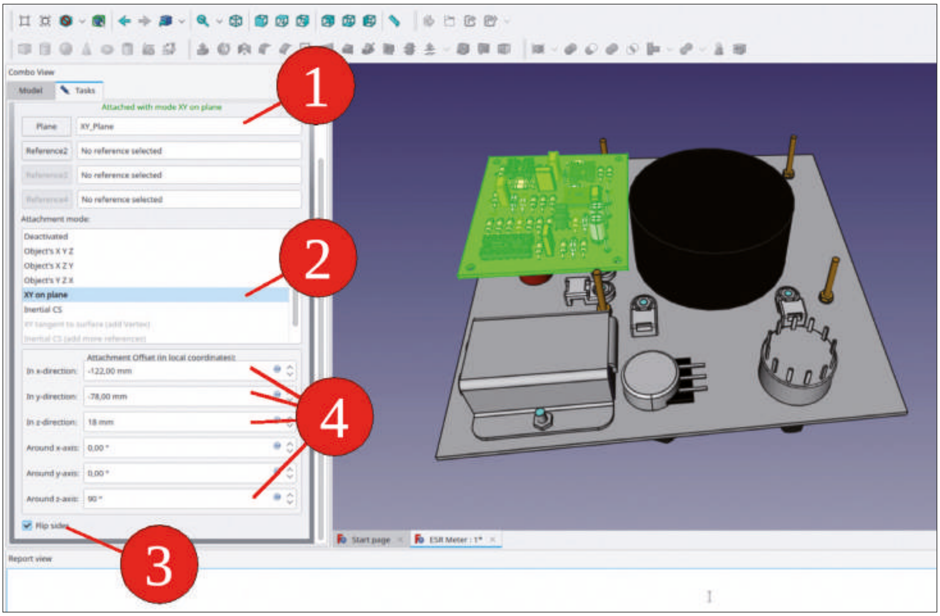


Figure 7-61

6. In order to set the destination for the upcoming SubShapeBinder, double-click the 'Front Panel' body. This will activate it. Mark the 4 edges of the PCB mounting holes. Then, click the 'Create a sub object(s) shape binder' tool button.

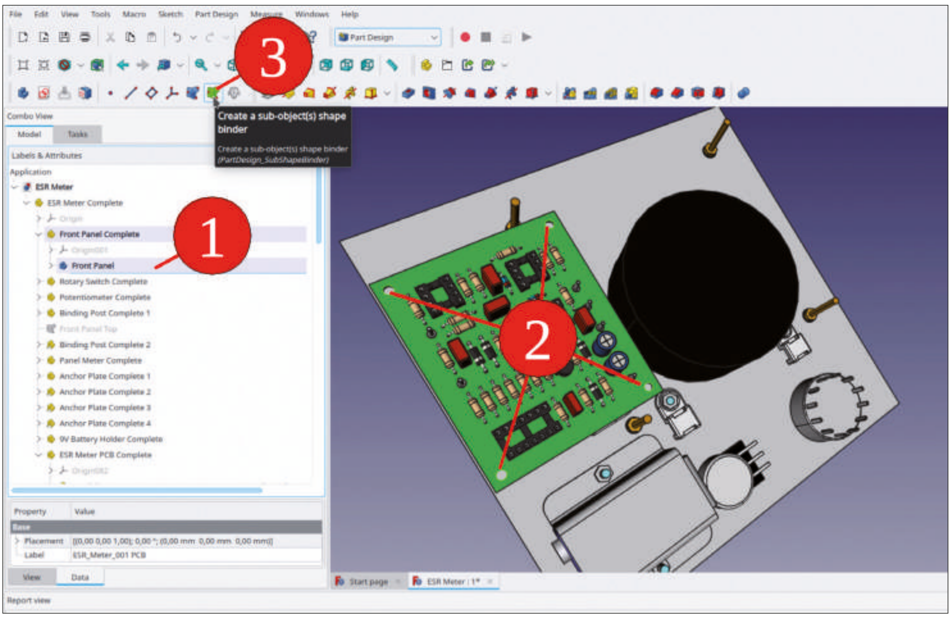


Figure 7-62

7. The tree view will jump away from the destination object and expose the node of the last item which was marked in the 3D view. Scroll up to locate the new SubShapeBinder in the 'Front Panel' body. Rename it to 'Footprint PCB' and set its 'Make face' property to 'false' (Figure 7-63). Then, click the 'Hole' tool button.

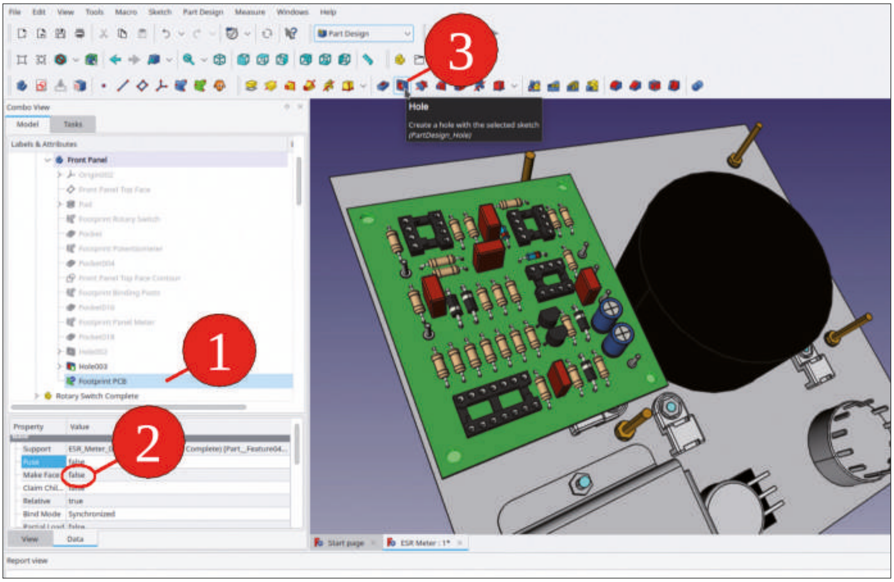


Figure 7-63

8. In the task window, set the diameter to 3.2 mm, select for the depth 'Through all', and check the 'Reversed' checkbox (Figure 7-64). Close the task window with the 'OK' button.

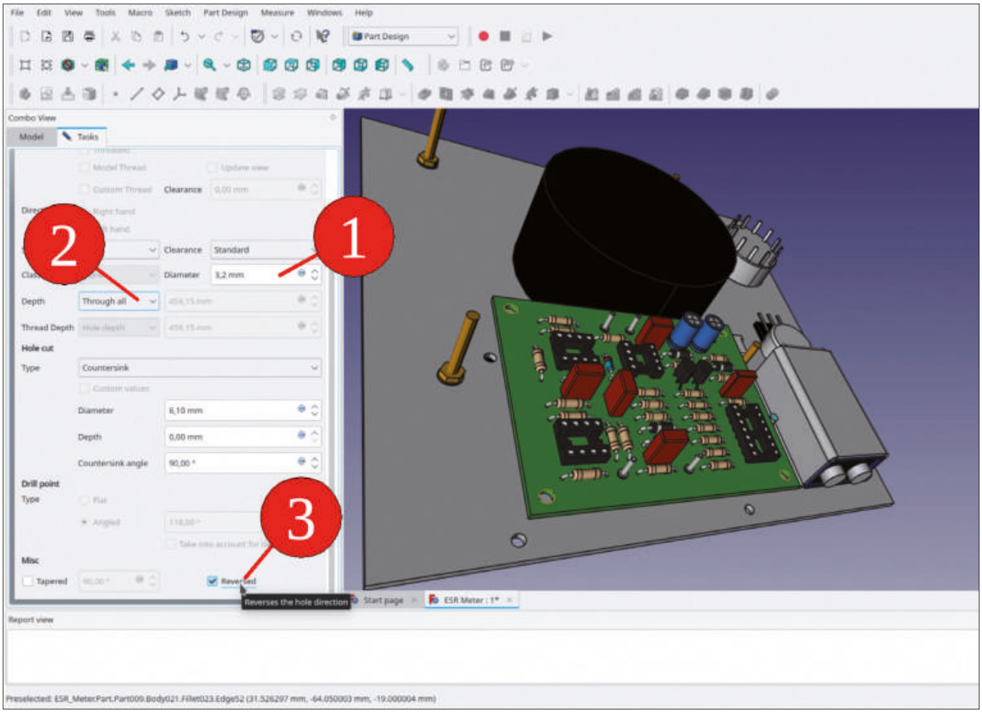


Figure 7-64

The circuit board is still missing its fasteners. You have several options where to place them within the project tree. One option is to attach the fasteners to the holes in the front panel. Alternatively, the fasteners could be added to the Std-Part-Container of the circuit board. Because the second possibility results in a less complex project tree, let's pursue that:

9. In the directory 'Sample Projects | ESR Meter | Components', locate and open the file 'Spacer'. It is an 18 mm version of the file that you have created in Chapter 4. Copy the Std-Part-Container 'Spacer Complete' with CTRL-C, and do not change the default selection in the pop-up selection dialog. Click on the OK button to proceed.

10. Close the document node 'Spacer'. This activates the document node 'ESR Meter' again. (if more than one document remains in the tree, you have to check which one is activated, with the title shown in bold letters. Insert the spacer with CTRL-V.

11. Rename the Std-Part-Container 'Spacer Complete' to 'Spacer Complete 1'. Drag-and-drop it into the Std-Part-Container 'ESR Meter PCB board Complete' (Figure 7-65).

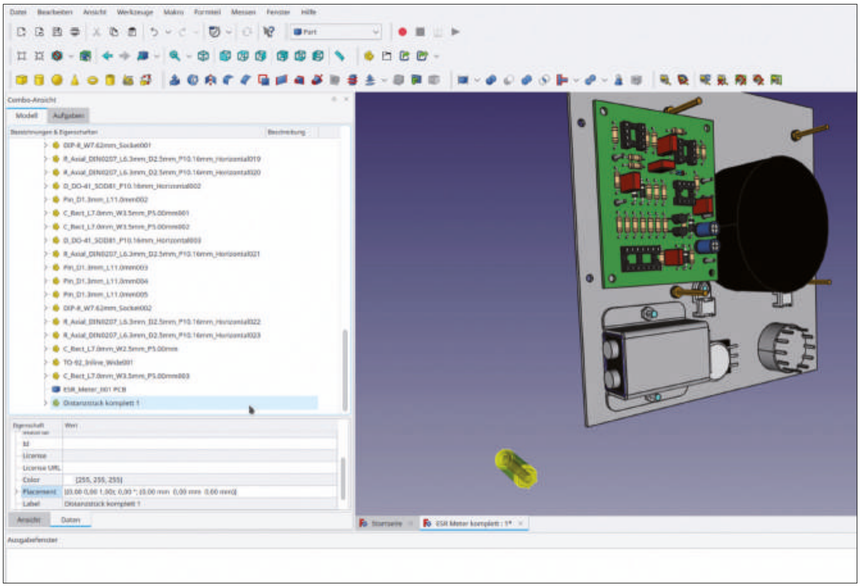


Figure 7-65

12. In the tree view, mark 'Spacer Complete 1' and switch to the 'Part' workbench. From the main menu, select 'Part | Attachment'. From the rear of the circuit board, select the edge of a mounting hole (hiding the front panel with the SPACE key will make that easier). For the attachment mode, select 'Concentric' (Figure 7-66). Set the attachment offset Z to –18 mm (Figure 7-68) and close the task window with the 'OK' button.

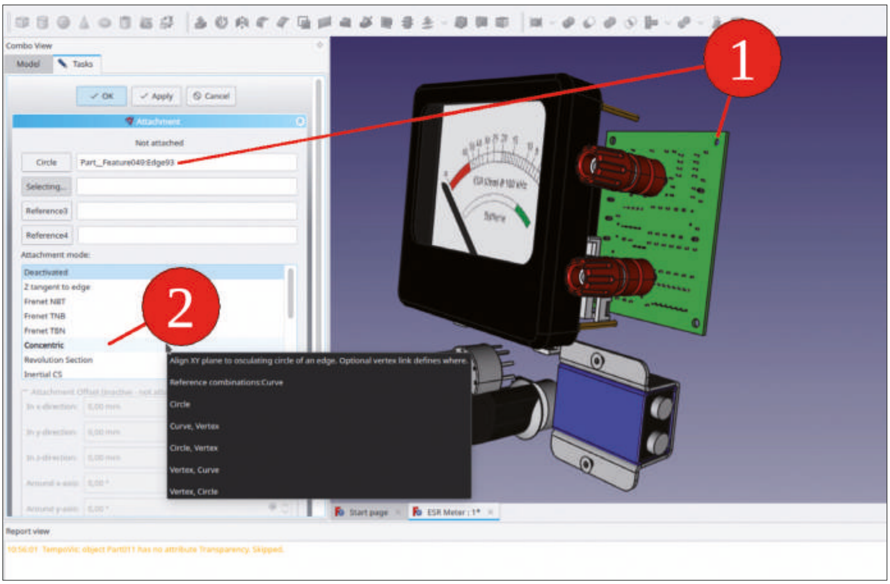


Figure 7-66

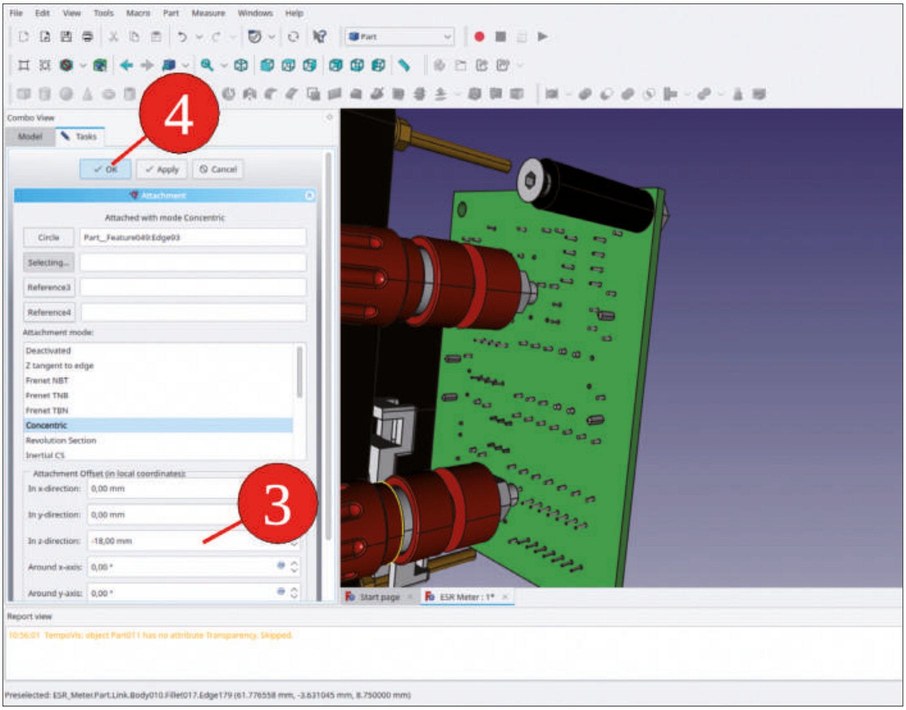


Figure 7-67

13. The other spacers are created as link objects. In the tree view, mark 'Spacer Complete 1' and click the 'Make link' tool button. While you are there, repeat that two more times, in order to create all the spacers at once. In the tree view, rename them to 'Spacer Complete 2' to 'Spacer Complete 4'. Make sure to always mark again 'Spacer Complete 1' as the parent of the link again, before you click the 'Make link' button.

14. Repeat the attachment procedure as described for 'Spacer Complete 1' for all the spacers, distributing them to the different mounting holes of the PCB. The result is shown in Figure 7-68.

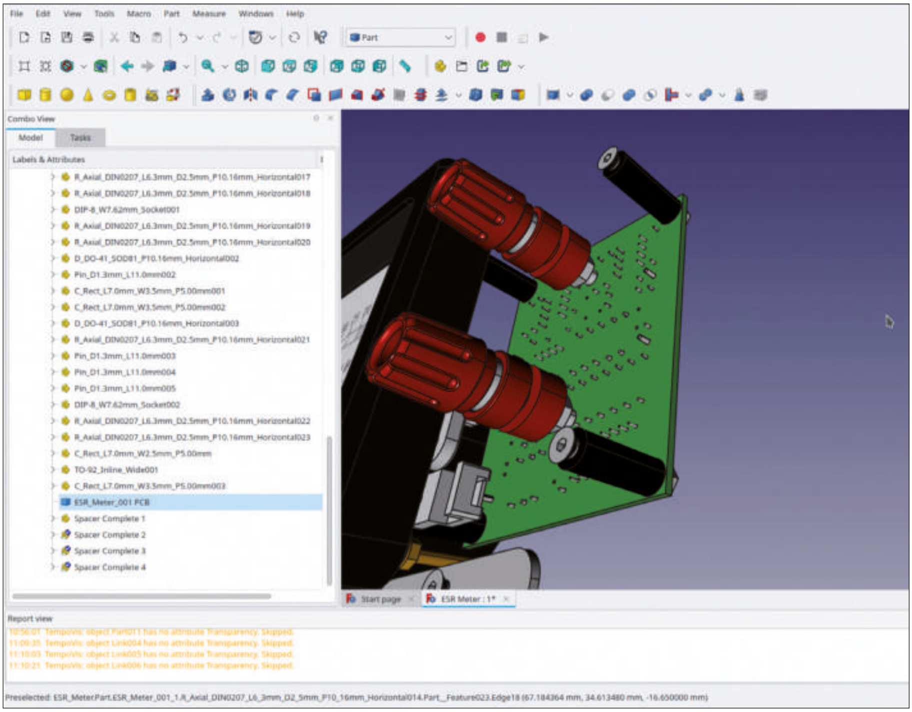


Figure 7-68

For the attachment of the front panel to the wooden box (or the enclosure of your choice), four studs are still missing. Again, the first stud will be copied from a file, and the residual ones are then generated as link objects:

15. In the directory 'Sample Projects | ESR Meter | Components', locate and open the file 'Spacer Casing'. Copy the Std-Part-Container 'Spacer Casing Complete' with CTRL-C. In the popup selection dialog, do not change the default selection and click the 'OK' button to proceed.

16. In the tree view, close the document node 'Spacer Casing'. Insert the copied object by CTRL-V. Rename the new Std-Part-Container to 'Spacer Casing Complete 1'. Drag and drop it into the 'ESR Meter Complete' Std-Part-Container.

17. In the tree view, mark 'Spacer Casing Complete 1'. In the property list, edit the placement parameters and enter the translations: X = −75 mm, Y = −75 mm (Figure 7-69).

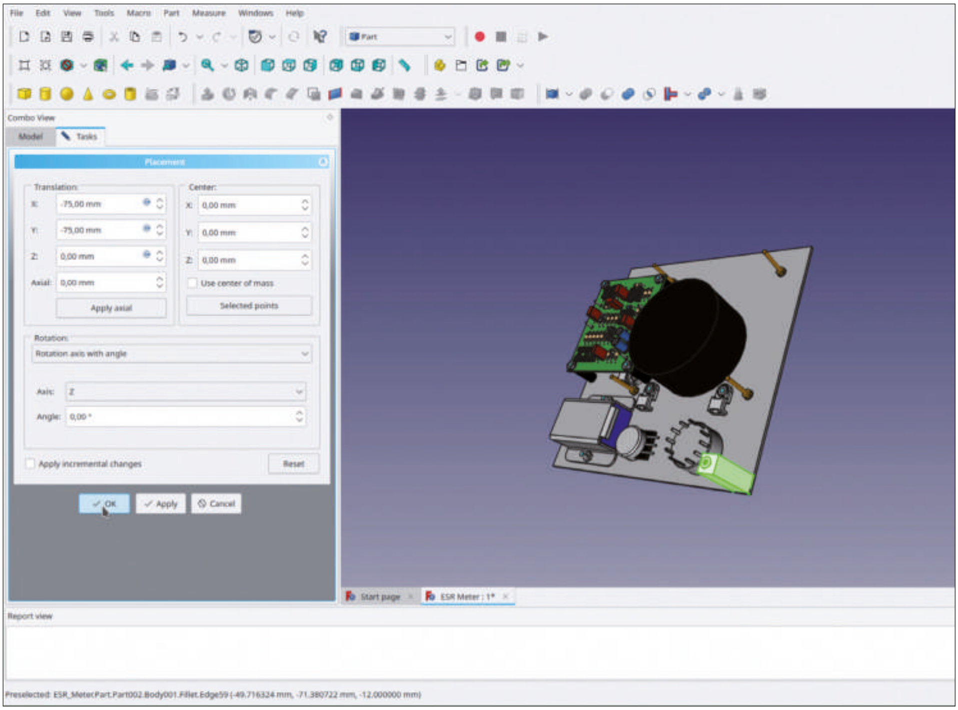


Figure 7-69

18. In the tree view, mark 'Spacer Casing Complete 1' and click the 'Make link' tool button. Repeat this step 2 more times, in order to generate all the spacers at once.

19. Rename the new spacer containers to numbered versions of the parent spacer. Drag-and-drop the new spacers into the 'ESR Meter Complete' std-Part-Container. Edit the placement parameters of the added objects such that the spacers are distributed into all the front panel corners (X, Y = +/− 75 mm). The result is shown in Figure 7-70.

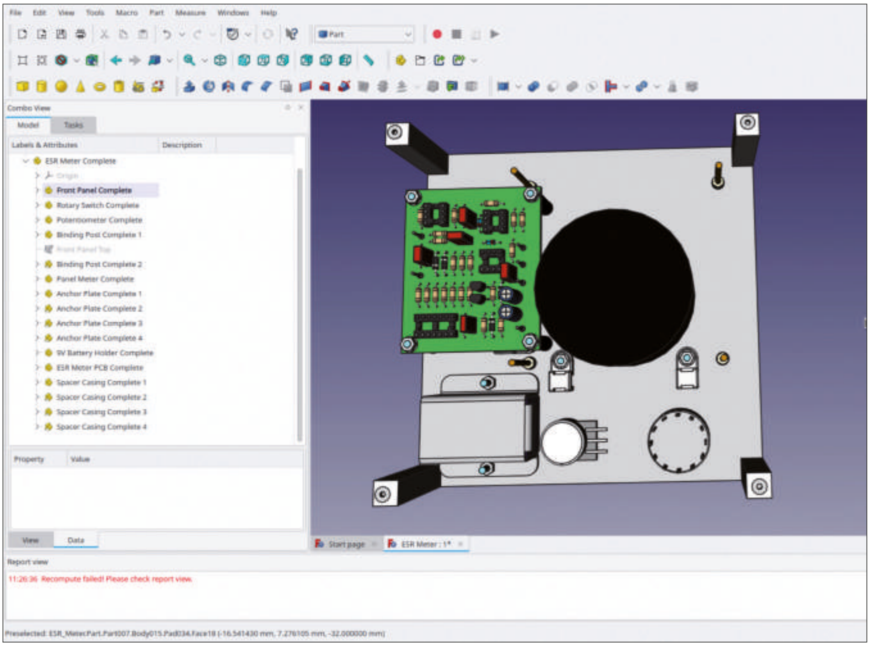


Figure 7-70

20. In order to expose the edge of the mounting holes, in 'Spacer Casing Complete 1', hide the front-side screw with the SPACE key (all the linked ones follow automatically).

21. To set the destination for the SubShapeBinder generated next, activate the body 'Front Panel' with a double click. Then, hide it also with the SPACE key.

22. Switch back to the 'Part Design' workbench.

23. In the 3D view, mark the edges of the mounting holes in the casing spacers. Then, click the green 'Create a sub object(s) shape binder' tool button (Figure 7-71).

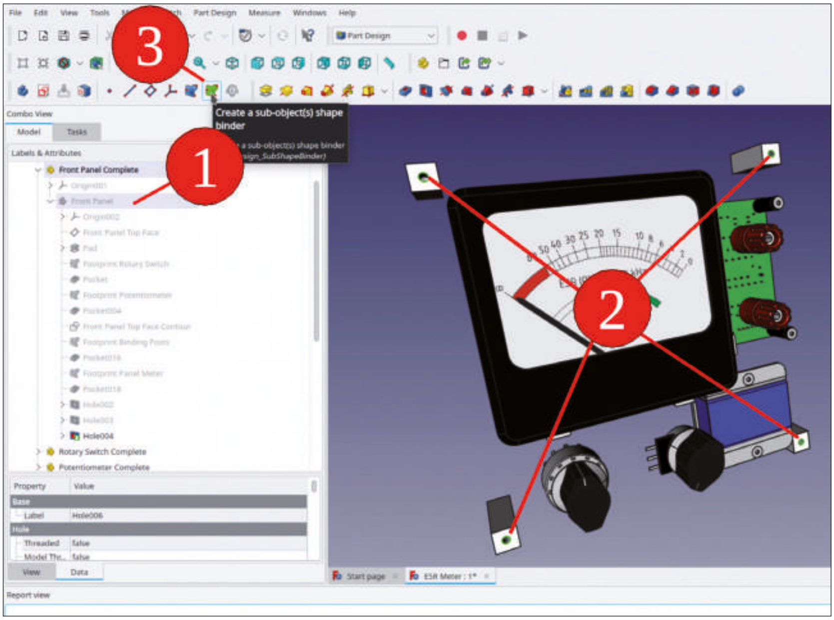


Figure 7-71

24. Rename the new SubShapeBinder in 'Front Panel' to 'Footprint Spacers Casing'. In the property list, set the property 'Make face' to False (Figure 7-72). Then, click the 'Hole' tool button.

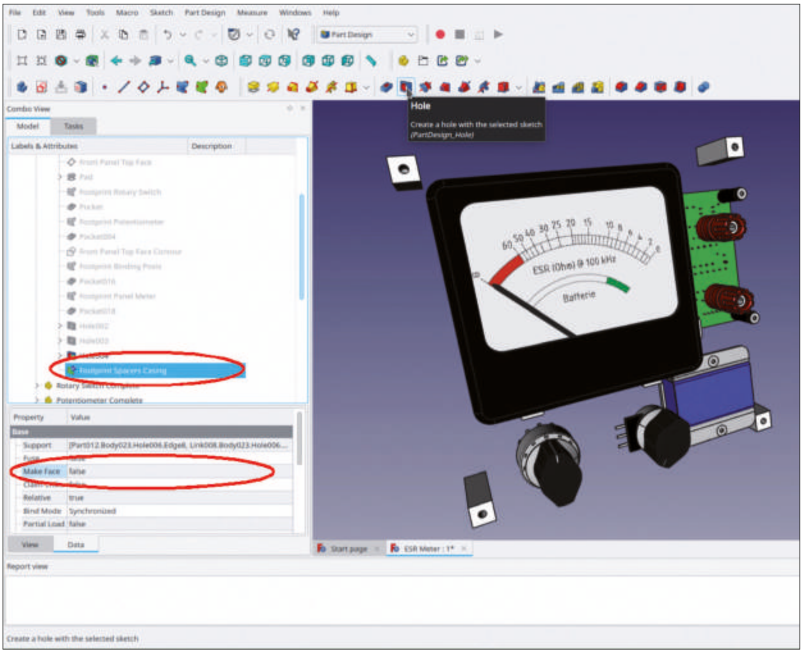


Figure 7-72

25. In the task window, select enter a value of 3.2 mm for the diameter, select 'Through all' for the depth, and check the 'Reversed' checkbox (Figure 7-73). Close the task window with the 'OK' button (top).

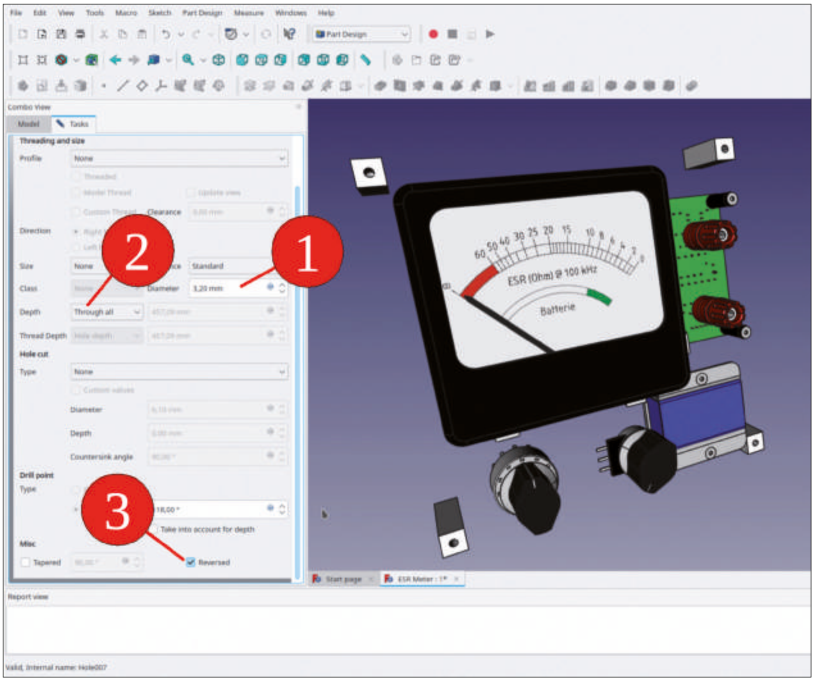


Figure 7-73

26. In the tree view, display the body 'Front Panel' again, with the SPACE key, as well as the hidden screw of the 'Spacer Casing 1' (Figure 7-74).

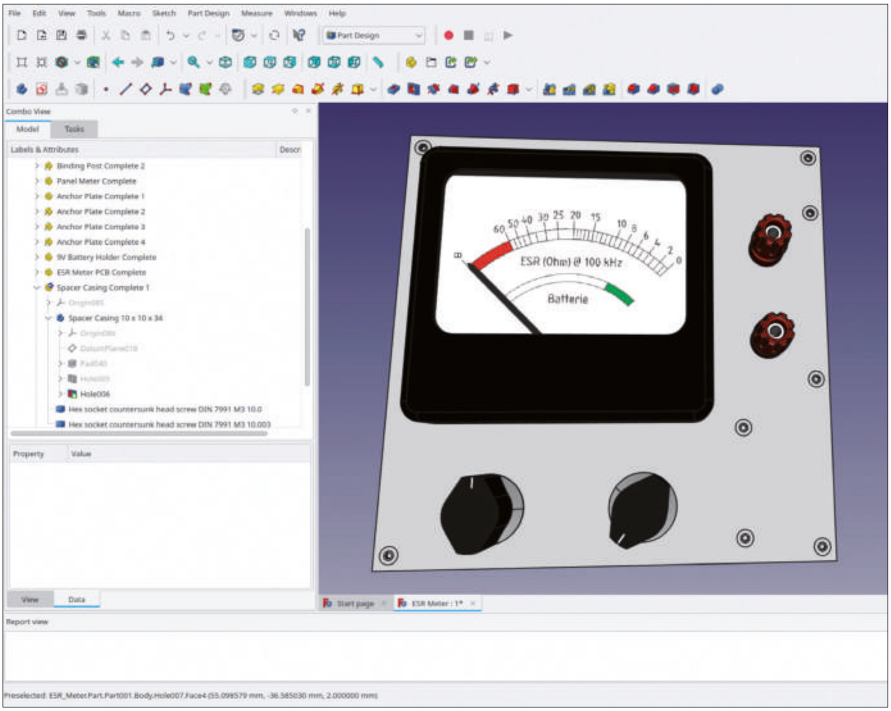


Figure 7-74

This concludes the mechanical design of the ESR meter. There are a few things best left to you — the holes have no countersunk edges yet. It can be tricky to add those because the holes are generated from the rear, in 'Reverse' mode. You could try to apply a chamfer with the front edge of these holes. But eventually, it is even better to omit those details because that could make the data transfer to the milling machine slightly easier.

However, one small error is still concealed in the design, a bit intentional, to demonstrate the advantages of an associative design. If you inspect the assembly from the rear, you can spot a collision between an anchor plate and a spacer (Figure 7-75).

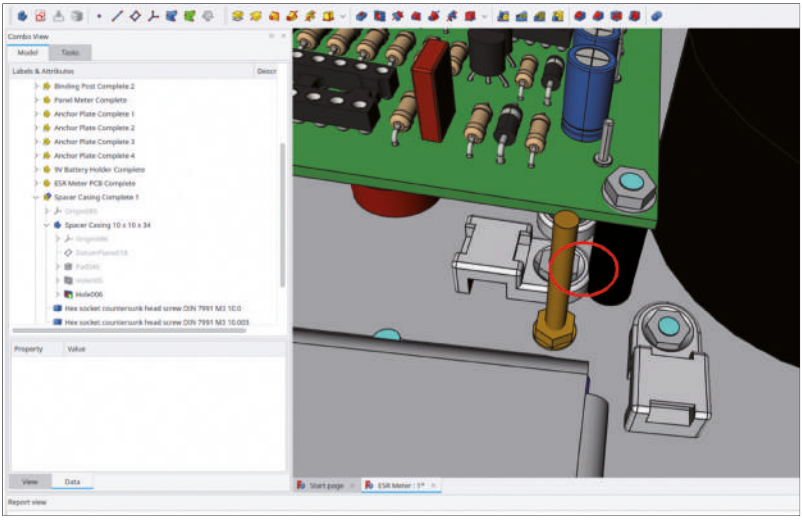


Figure 7-75

The issue can be resolved by shifting the circuit board in the X direction: Click on the 'ESR Meter PCB Complete' Std-Part-Container and expand the attachment parameters. By setting the attachment offset X to −121 mm, the collision is avoided (Figure 7-76). It is evident, that by the associative relations between the circuit board, the spacers, the fasteners and the holes in the front panel, the operation could be achieved by changing only one parameter.

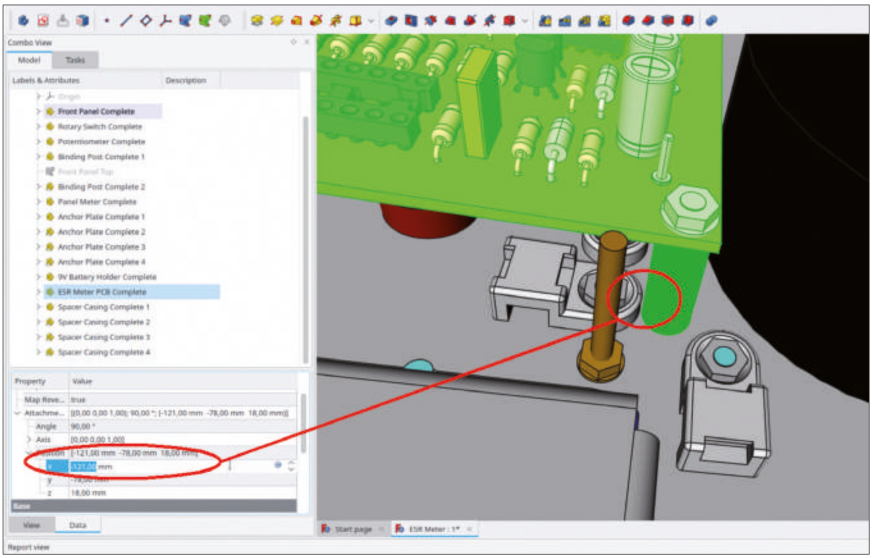


Figure 7-76

Although this may look like a trivial example, there are a multitude of possibilities arising for optimization and serialization of designs once the associativity is properly defined.

Chapter 8 • Finalizing the Front Panel with Graphic Elements

For the professional look of an instrument as well as the provision of intuitive user guidance, well-designated control elements can make a big difference. While customized stickers from a label printer may be sufficient for a quick demo or prototype, engraved lettering will add to the longevity of a released instrument. If you are lucky and own some thin milling cutters (with 0.5 mm diameter, for instance), it is not difficult to produce such engraving. Filled with paint, it will give good contrast with even the ability for color highlight important sections. For rotary switches and potentiometers, guide lines provide additional order and structure with the panel design.

8.1 Drawing Guide Lines

8.1.1 Sketch versus Path

When defining elements for engraving, it is essential to make the results compatible with the interface the milling machine provides. With its 'Path' workbench, FreeCAD has built-in support to generate trajectories for CNC machines. This can be helpful when milling complex pockets or 3D surfaces.

Many smaller CNC machines are found in electronics workshops these days. Meanwhile, not just the software for the smaller varieties, but the large ones too in the machine shop, are able to read DXF files and translate them into production data.

Consequently, let's follow a simplified way here and draw sketches for the guide lines, which can be displayed on TechDraw sheets and exported to DXF format from there.

8.1.2 Guide Lines for the Rotary Switch

With rotary switches, it is useful if the designation of a switch position is clearly associated with the switch position. With only one switch in the design, you might get away by nicely placing the text around the knob. But with only a few switches, and consequently many possible switch positions, more structure is needed on the front panel.

When planning the sketches, it is worthwhile to attach them to objects that are resilient against re-enumeration. Intuitively attaching them to the front facet of the panel might not be the strategy of choice. Luckily, you already have created a datum plane ('Front Panel Top Face'), which is rock solid compared to generated facets. Onto this, you will draw the following sketches and outlines.

1. Turn the 3D view to the 'Top' orientation and zoom into the corner with the rotary switch.

2. In the tree view, if necessary, expand the 'Front Panel Complete' Std-Part-Container. Inside, activate the body 'Front Panel' with a double click.

3. In the tree view, mark the datum plane 'Front Panel Top Face' and start the sketcher.

4. From the main menu, select 'View | Orthographic view' and 'Sketch | View section' (Figure 8-1).

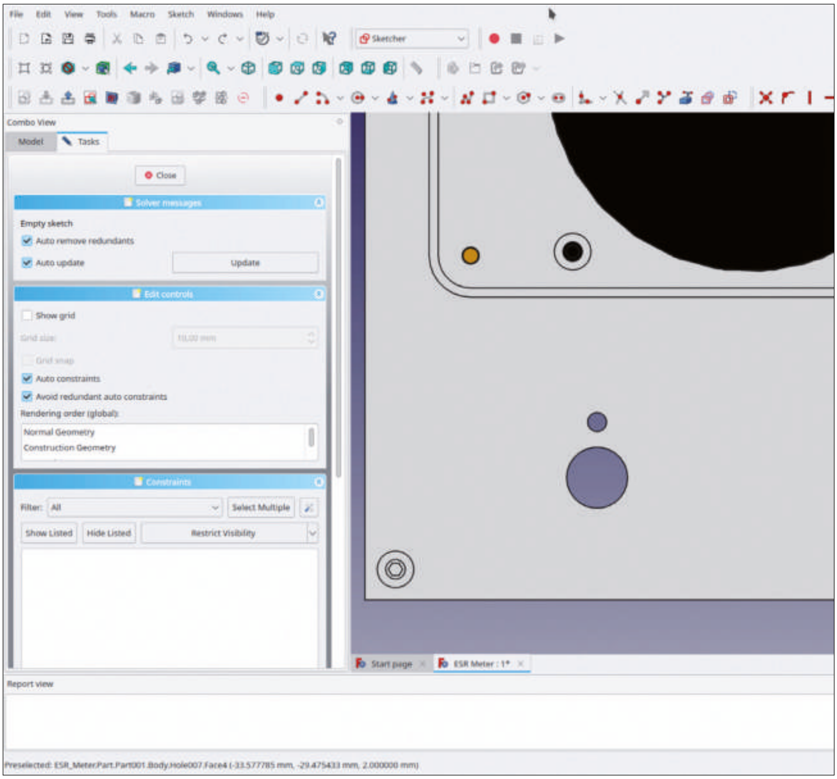


Figure 8-1

Now, let's draw some construction elements. This may look complicated but helps to keep the sketch fully constrained. Also, when one parameter of the drawing is changed, less distortion (if any) of the other elements will result. Changes would be very difficult if you had drawn the sketch right away, specifying a lot of dimensions for each point in it.

5. Click on the 'External geometry' tool button and mark the large hole of the rotary switch footprint (Figure 8-2), which is then displayed in violet as a construction element.

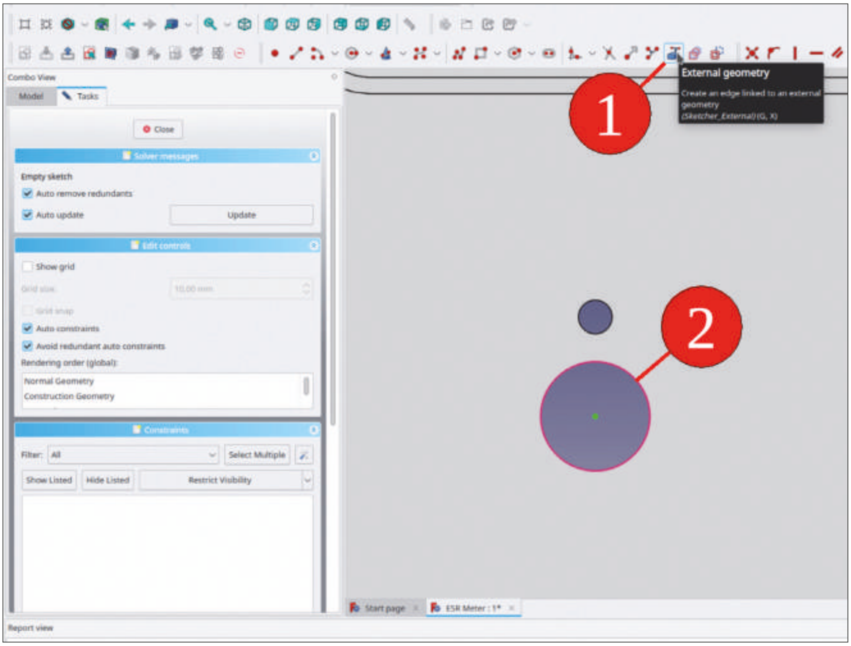


Figure 8-2

6. Draw two circles centered on the construction element. End the drawing command with a right-click. In the 'Elements' list, right-click one of the circles and select 'Diameter constraint' from the context menu (Figure 8-3). Enter a value of 38 mm.

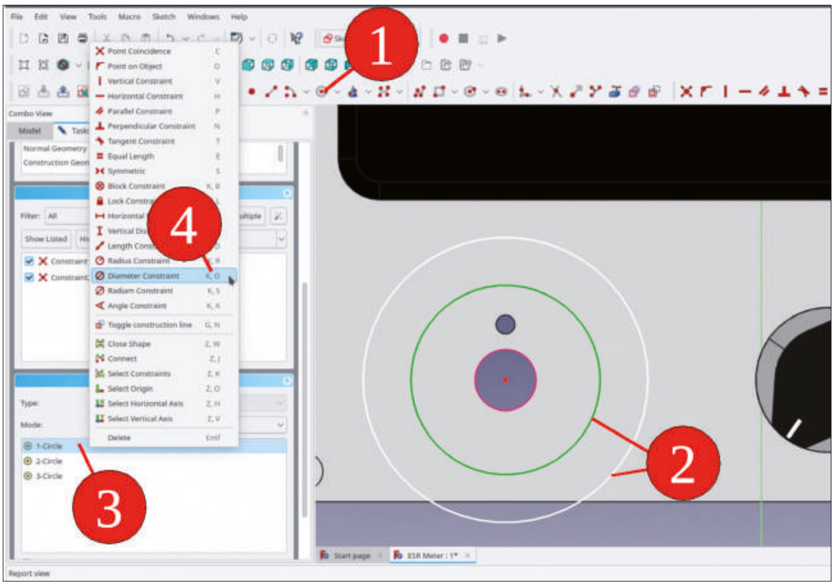


Figure 8-3

7. In a similar way, constrain the diameter of the second circle to 22 mm. The two circles are marking the start and end points of the radial guide lines.

8. In the 'Elements' list, mark the two circles. Right-click the selection. From the context menu, select 'Toggle construction line' (Figure 8-4). The two circles appear in blue color now but will be invisible with the sketch.

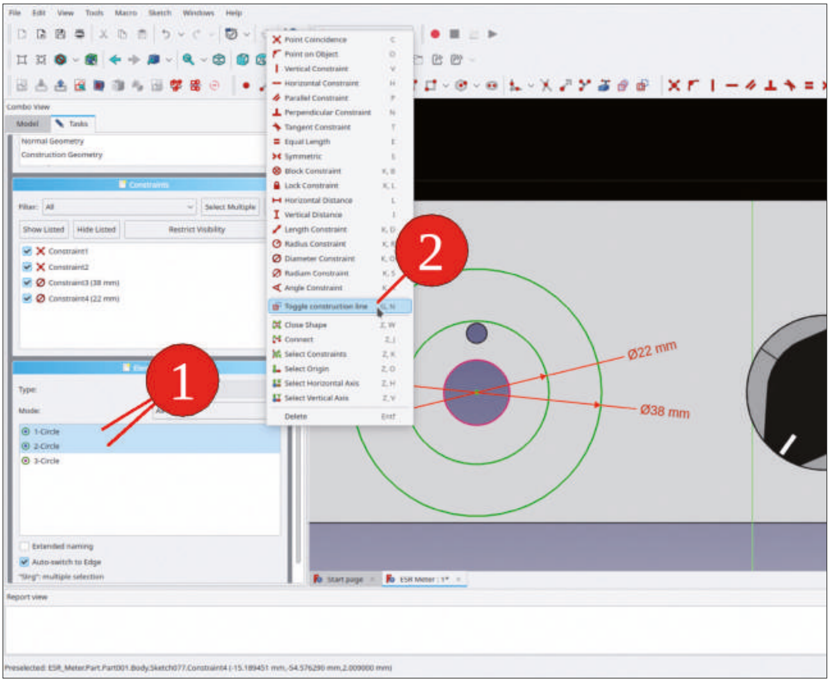


Figure 8-4

9. From the center of the two circles, draw three lines. One must be vertical (to indicate that implicitly selected constraint, a small, red, and vertical bar appears next to the line, Figure 8-5, step 1), when you try to draw it vertically. Take care to attach the lines to the center point. It takes some aiming with the crosshairs, until the center point switches color to yellow (Figure 8-5). If the lines, or even one of them, is not properly attached, there will be some time needed for debugging the sketch later.

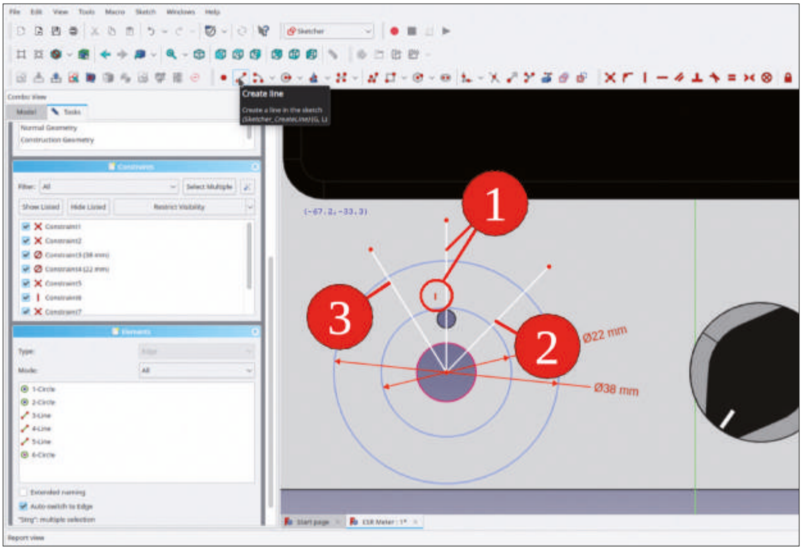


Figure 8-5

10. On the sketch, mark the vertical and one of the other lines. In the 'Elements' list, right-click the selection, and select 'Angle constraint' from the context menu. Set the angle to 30° (our switch has 12 positions). Repeat the procedure for the other line (Figure 8-6). Sometimes, the sketcher complains when you try to apply a constraint. In that case, it is likely that some other element was still selected from prior operations. To resolve this, click somewhere into the empty space, which deselects all elements, and start over.

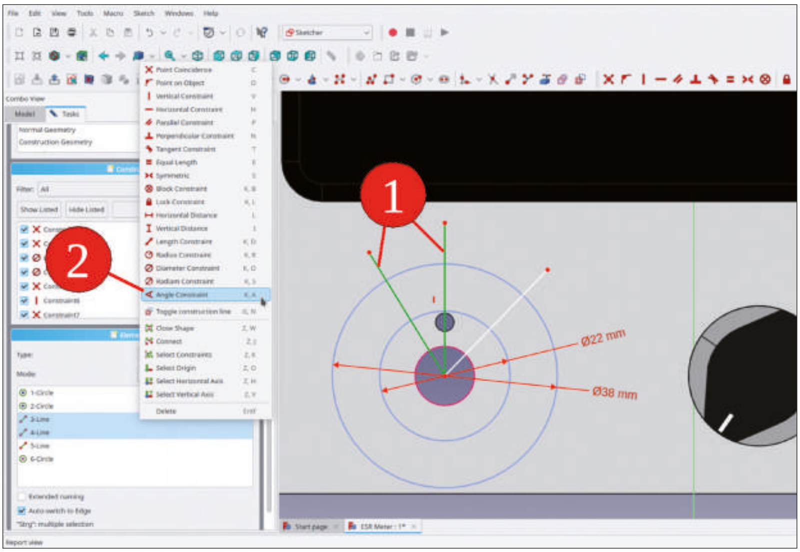


Figure 8-6

11. In the 'Elements' list, mark the three new lines. Right-click the selection and select 'Toggle construction line' from the context menu (Figure 8-7). The three lines also switch to blue color.

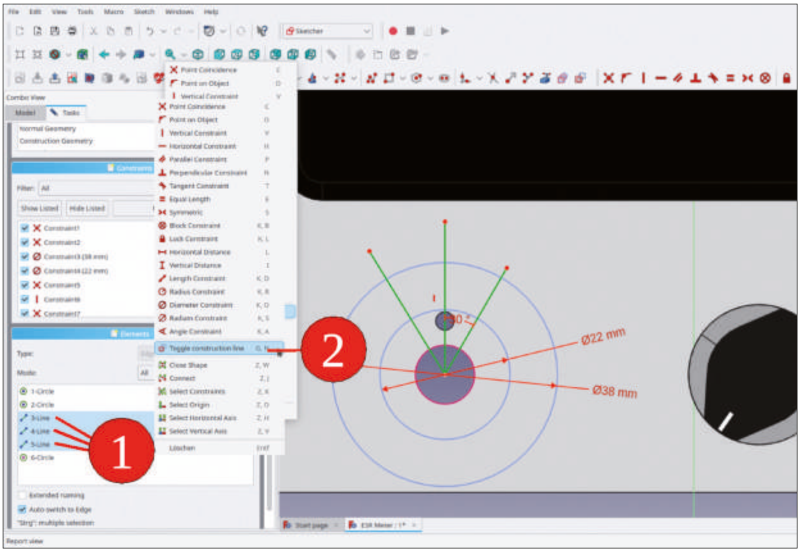


Figure 8-7

12. Click the 'Trim edge' tool button and trim all parts of the lines which are located outside of the bigger circle (Figure 8-8).

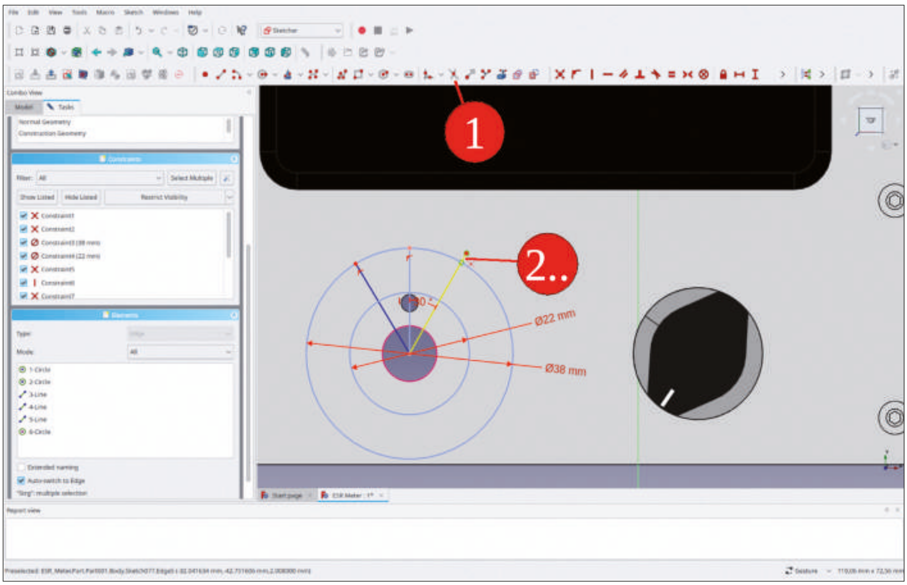


Figure 8-8

13. Draw three lines, which start at the outer endpoints of the construction lines. Place the endpoint on the inner circle, with arbitrary orientation of the lines. This is easier than trying to define all the constraints at once. The result is shown in Figure 8-9. End the drawing command with a right click.

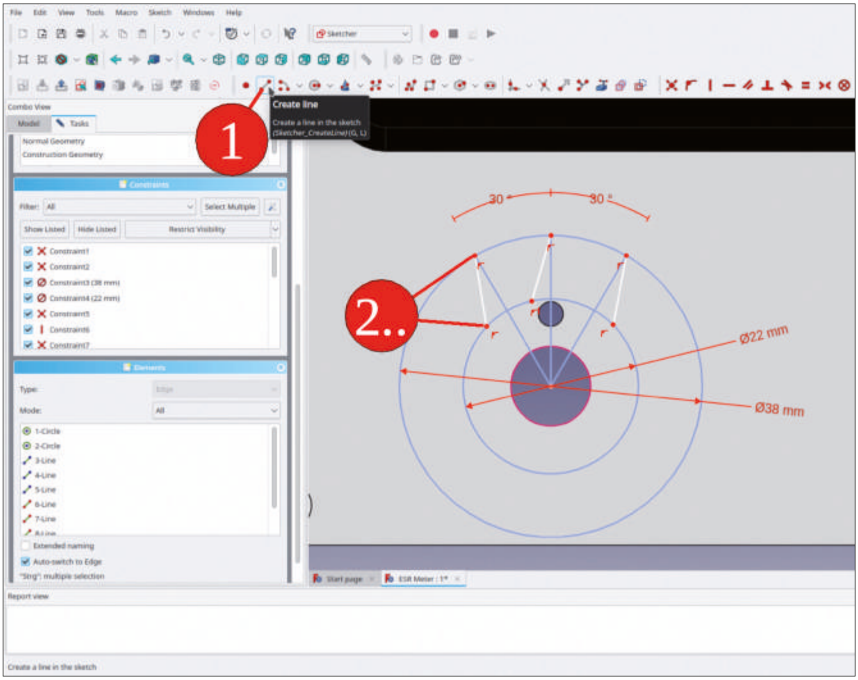


Figure 8-9

14. For each of the three cases, mark one of the new lines and the corresponding construction line. Then, click the 'Constrain parallel' tool button (Figure 8-10). The result is shown in Figure 8-11.

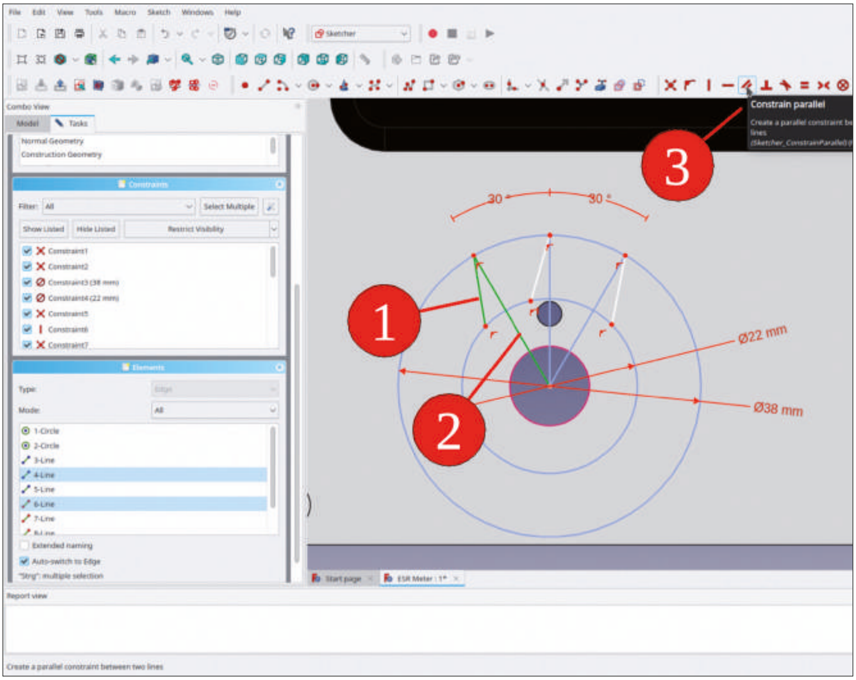


Figure 8-10

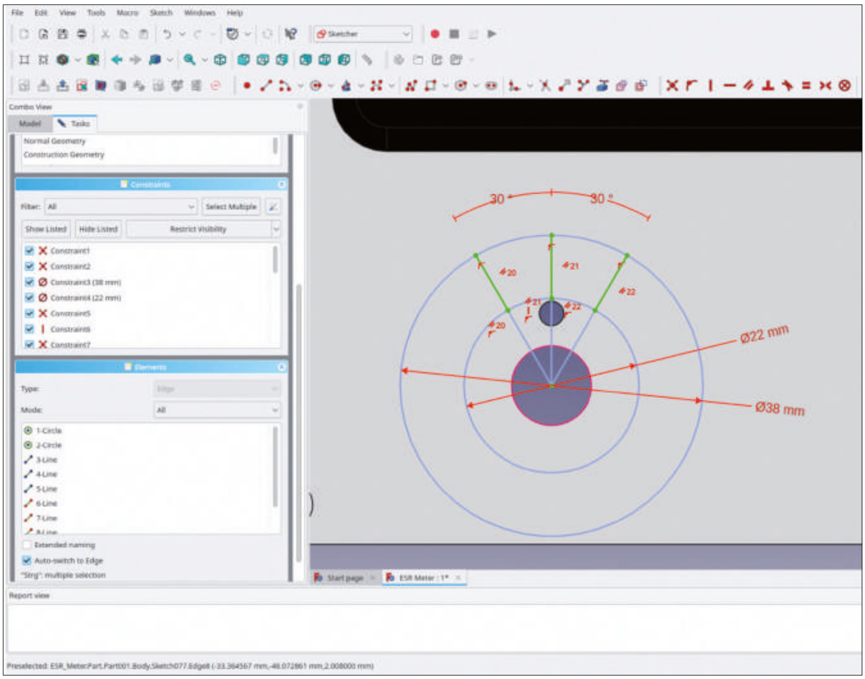


Figure 8-11

The three lines are displayed in bright green, meaning the sketch is fully constrained. When you find yourself defining (too) many dimensions, it is always a good idea to think about some helpful construction geometry.

15. Draw two horizontal lines which start at the outer endpoints of the outer guide lines: For the first click, aim at the endpoint, then pull the line out and watch for the little horizontal red bar, indicating the horizontal constraint. The exact length of the lines does not matter. End the drawing command with a right-click.

16. Mark one of the horizontal lines and click the 'Constrain horizontal distance' tool button (Figure 7-12). Enter a value of 7 mm.

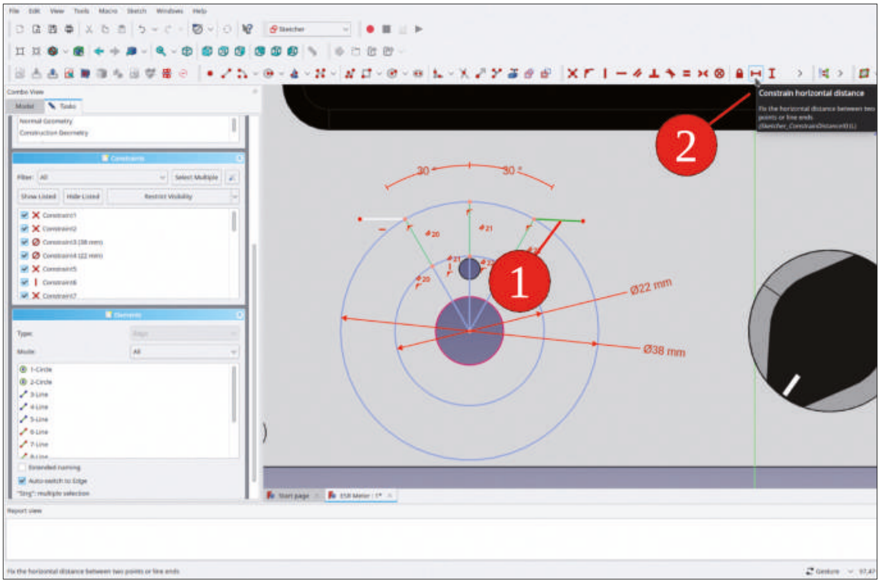


Figure 8-12

17. Mark both horizontal lines and click the 'Constrain equal' tool button (Figure 7-13). This sets the length of the two lines to the same value. 'Constrain equal' works with different objects – e.g., circles will be set to equal diameter.

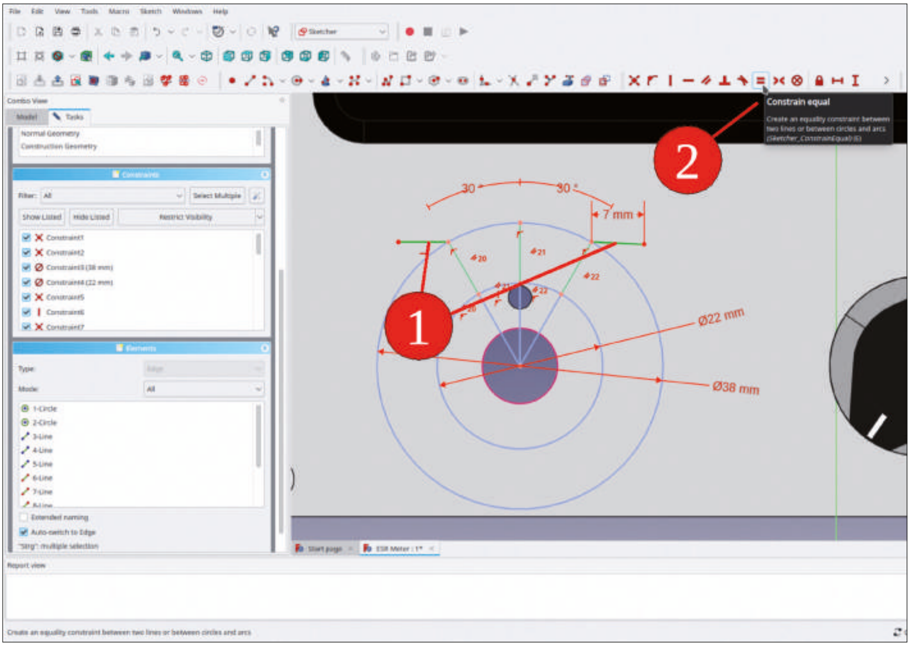


Figure 8-13

18. Now, the sketch appears bright green, and the solver says: 'Fully Constrained' – done! (Figure 8-14). Close the sketch.

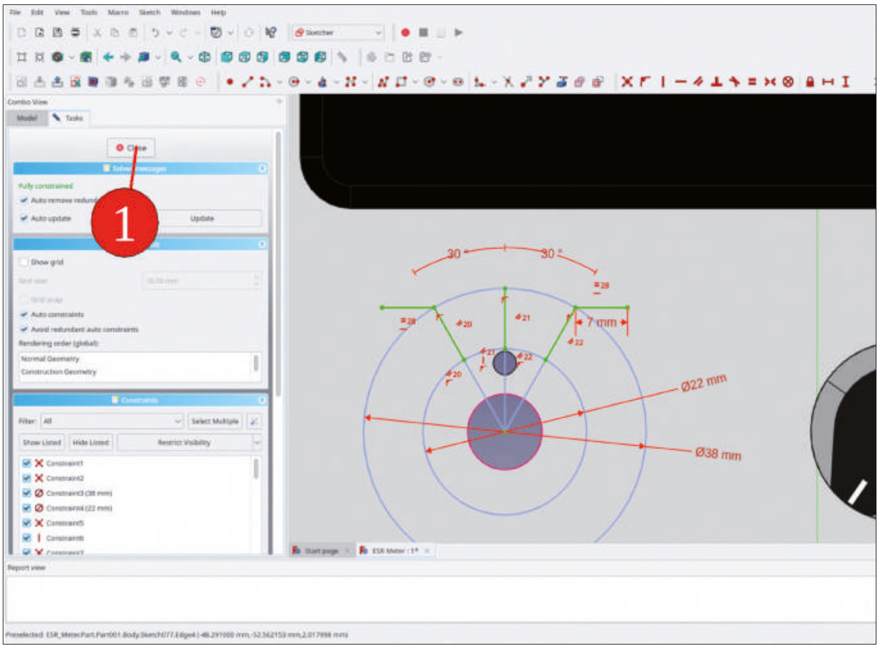


Figure 8-14

Now, you can display the rotary switch again (mark in the tree view and press the SPACE key). It is a question of style whether you like the guide lines ending underneath the control knob. If you don't, thanks to the construction geometry, this is easy to change:

19. In the body 'Front Panel', reopen the new sketch with a double click. Either double-click the dimension of 22 mm in the sketch directly, or, in the 'Constraints' list, click the line 'Constraint4 (22 mm)'. In the popup dialog, enter a value of 26 mm (Figure 8-15). Because the sketch is now shown together with the control knob, the diameter can be nicely optimized.

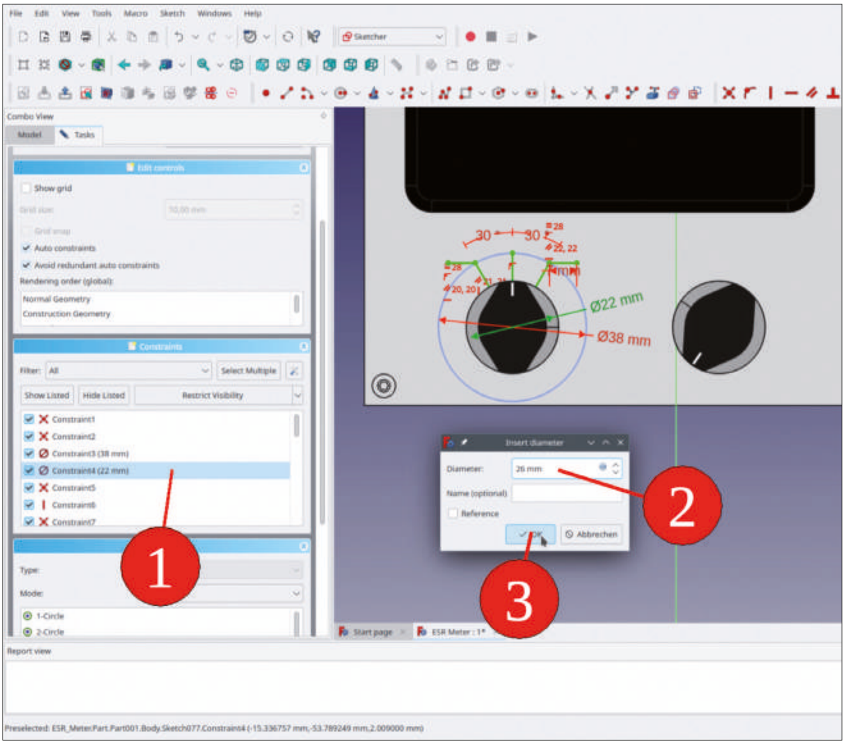


Figure 8-15

20. In the tree view, rename the sketch to 'Guide Lines Rotary Switch' (Figure 8-16).

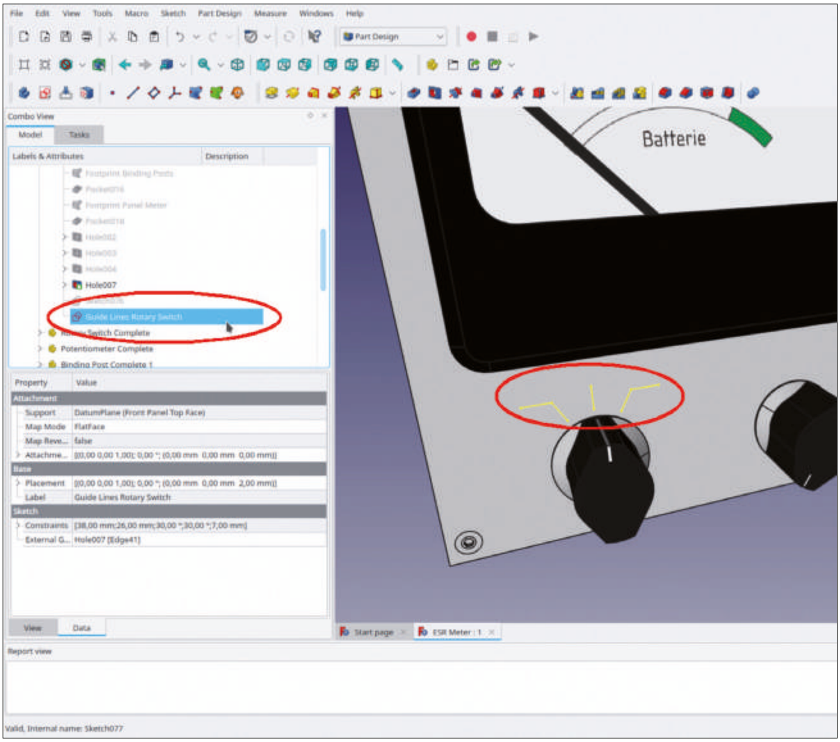


Figure 8-16

As long as the sketch is not hidden, it will also appear in TechDraw views. From there, it is then available for DXF export. Furthermore, the sketch is also associated with the rotary switch because you have attached it to the mounting hole via an external geometry reference. It will thus follow position changes of the rotary switch.

8.2 A Guide Line for the Potentiometer

A potentiometer should be turned and set to any position between the end points. An arc with two end marks can provide a hint to the user regarding the input expected by the control.

1. In the tree view, hide the Std-Part-Container 'Potentiometer Complete' with the SPACE key.

2. The body 'Front Panel' is still marked by the previous steps. Again, mark the datum plane 'Front Panel Top Face' and start the sketcher. From the main menu, select 'View | Orthographic view'.

3. Click on the 'External geometry' tool button and mark the edge of the mounting hole for the potentiometer (Figure 8-17).

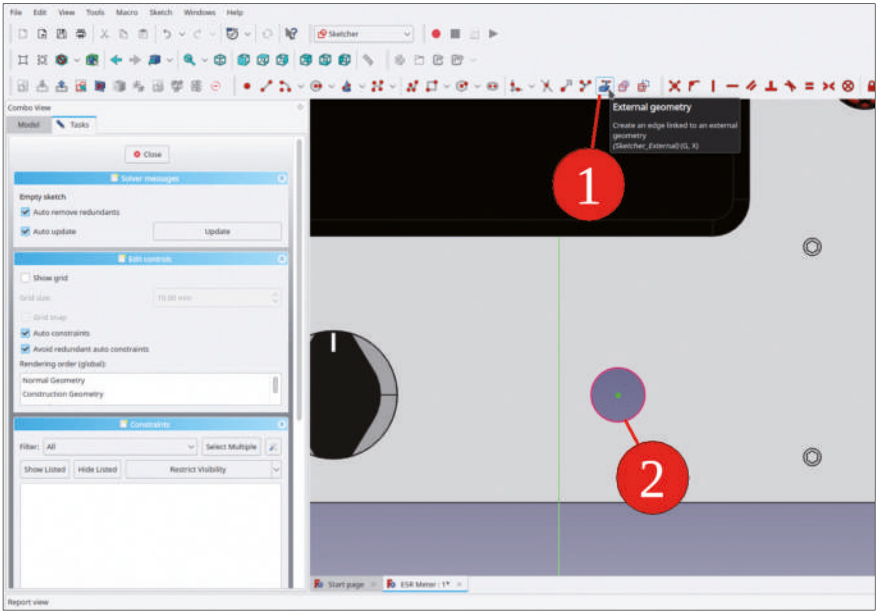


Figure 8-17

4. As was done for the rotary switch, draw two circles, which are centered on the violet construction circle, and constrain their diameters to 27 mm and 33 mm (Figure 8-18).

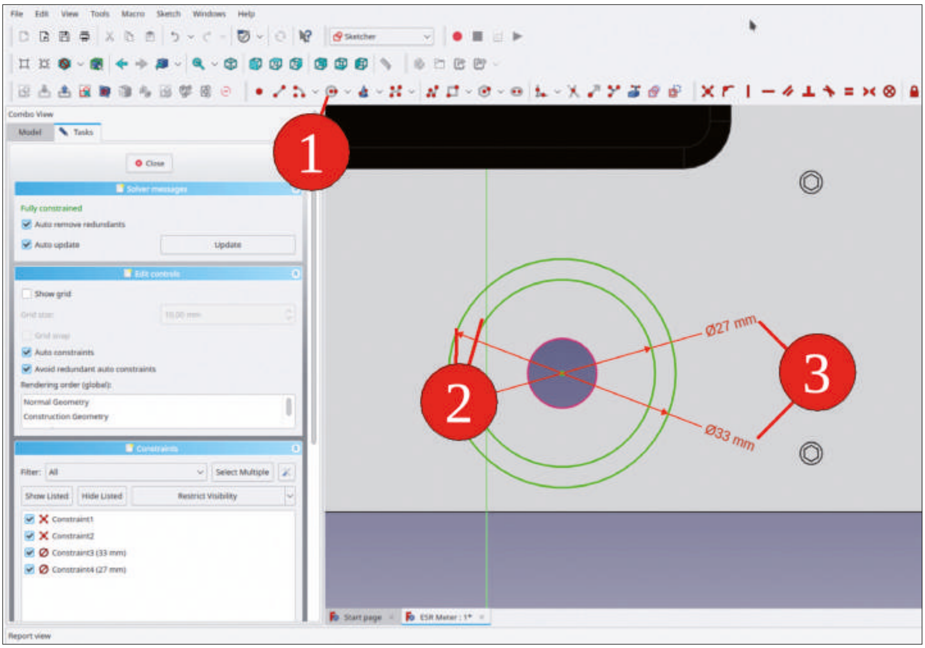


Figure 8-18

5. Draw three lines, each one with a starting point in the center of the circles. One of them must be oriented vertically (Figure 8-19). Mark the vertical line, and one of the outer lines, and click the 'Constrain angle' tool button (Figure 8-19).

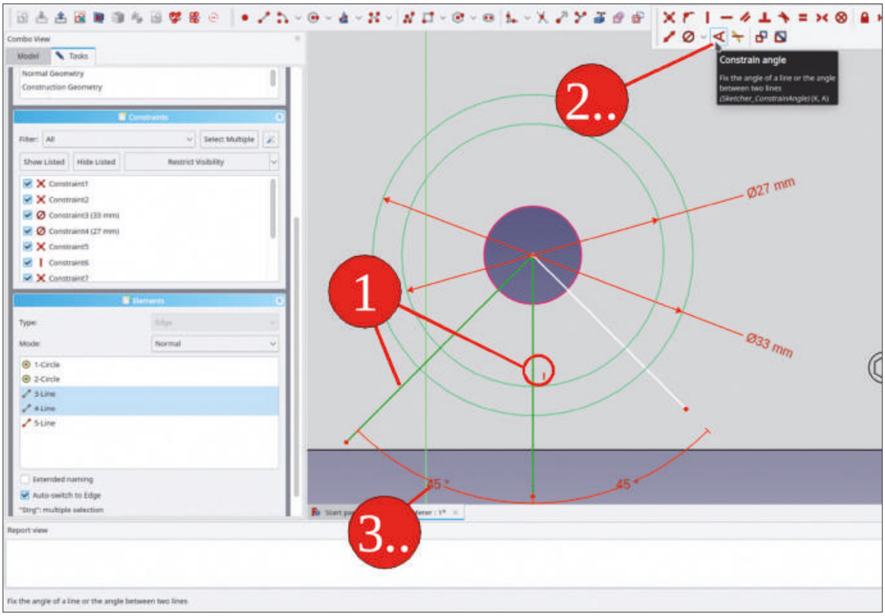


Figure 8-19

6. Trim the parts of the lines which are located outside the larger circle (Figure 8-20).

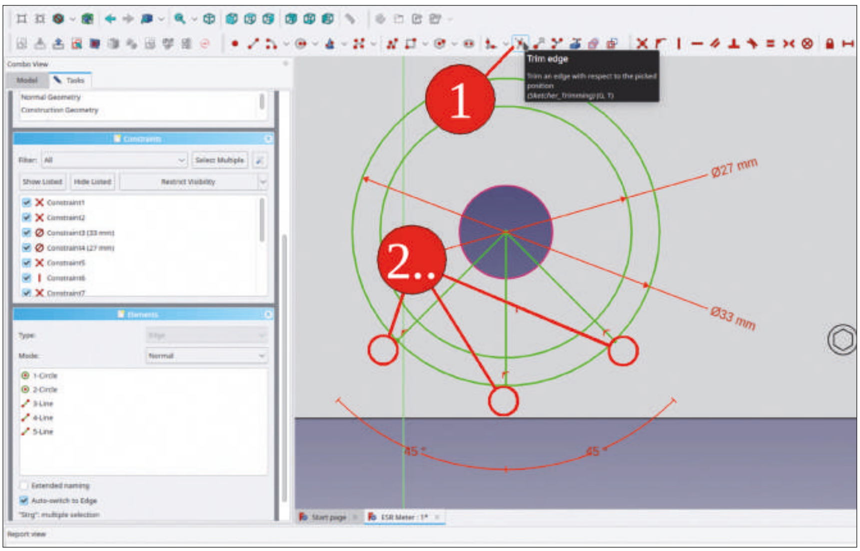


Figure 8-20

7. In the 'Elements' list, mark the two circles and all lines. Right-click the selection and select 'Toggle construction line' from the context menu (Figure 8-21).

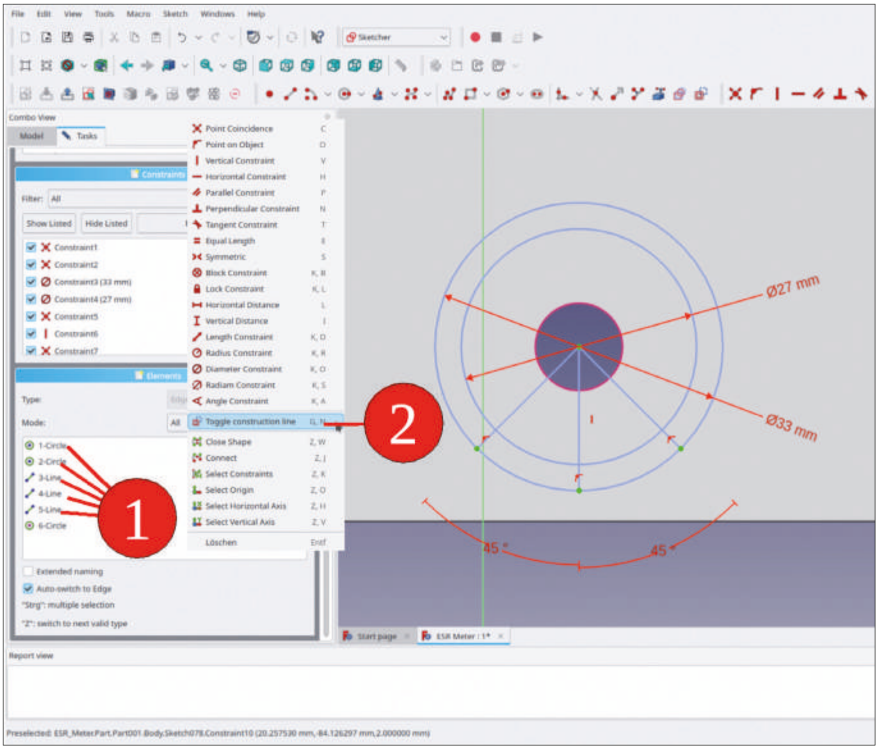


Figure 8-21

8. Click on the 'Create arc' tool button and draw an arc that is centered about the circles. Place an endpoint onto each of the inclined lines. The diameter of the arc can be arbitrary (Figure 8-22).

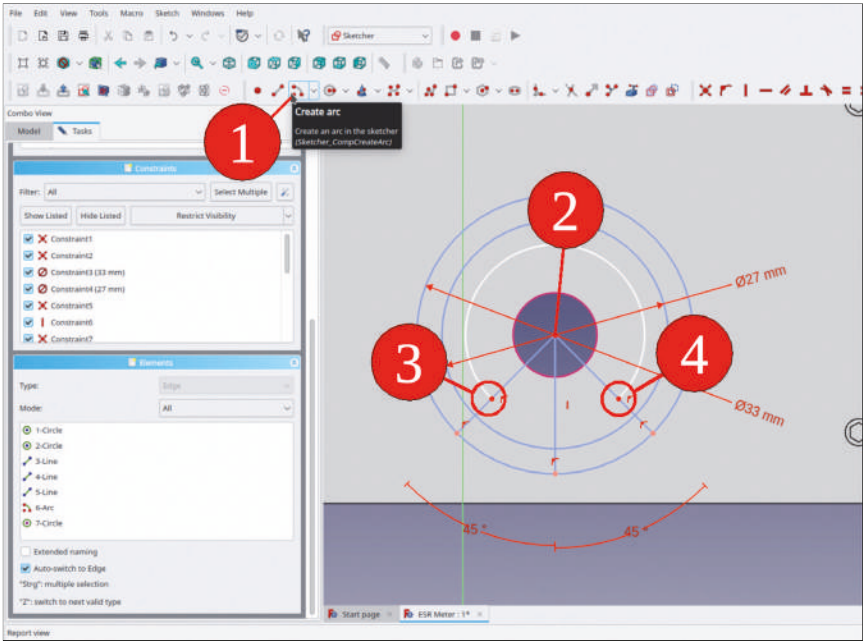


Figure 8-22

9. On the sketch, mark the arc and the inner construction circle. Then, click the 'Constrain equal' tool button (Figure 8-23).

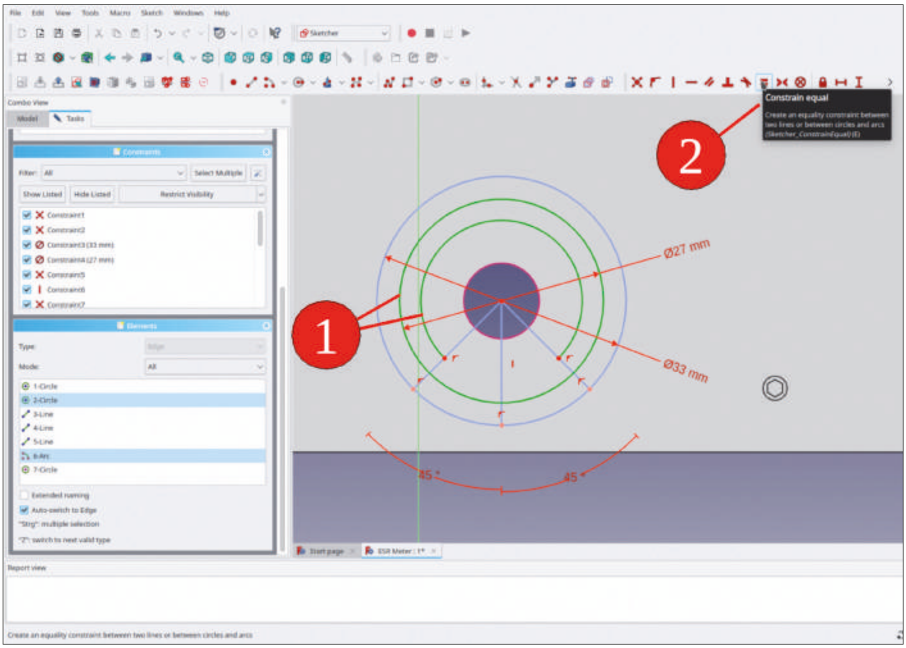


Figure 8-23

10. For the end marks, draw two lines between the endpoints of the arc and the corresponding endpoints of the construction lines (Figure 8-24).

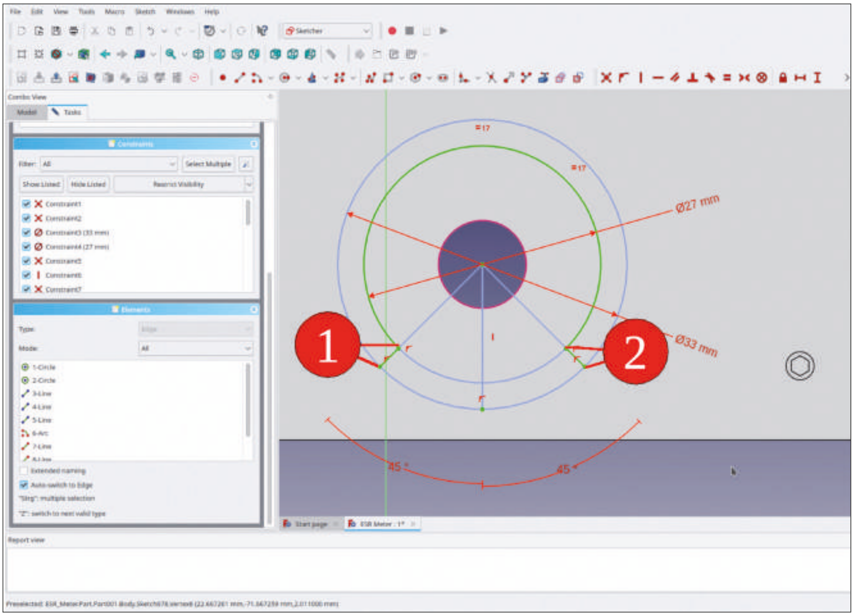


Figure 8-24

11. Close the sketch, and, in the tree view, rename it to Guide Line Potentiometer'. Display again the 'Potentiometer Complete' Std-Part-Container with the SPACE key (Figure 8-25).

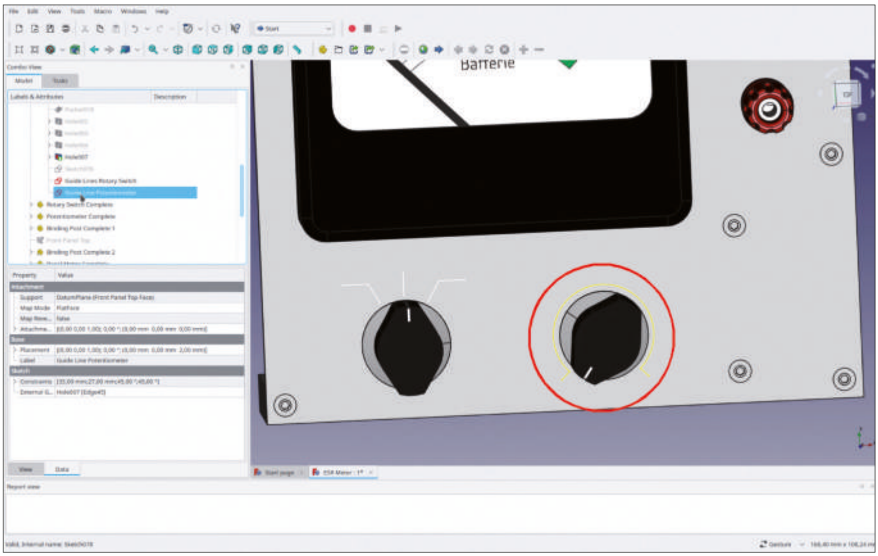


Figure 8-25

With these guide lines, the front panel already has a good structure. Descriptive text is, however, still missing.

8.3 The Engravings

In the 'Draft' workbench, FreeCAD offers functions, which allow the creation of engraved text. When you switch to the 'Draft' workbench, a grid representing the draft surface is automatically displayed. First, the orientation of the drafting surface needs to be set properly. Click on the 'Current working plane' tool button, which opens a task window. You can either set the working plane with the buttons provided (e.g., top), you can click an object in the 3D view, or you can switch to the 'Model' tab and click a tree view node. Use the last method: When the task window opens, switch to the 'Model' task and click the 'Front Panel Top Face' datum plane in the 'Front Panel' body. This brings the drafting plane up front (Figure 8-26, step 3). The grid representing the drafting plane stays visible, even when the 'Draft' workbench is left. To hide it, use the button shown in Figure 26, step 4.

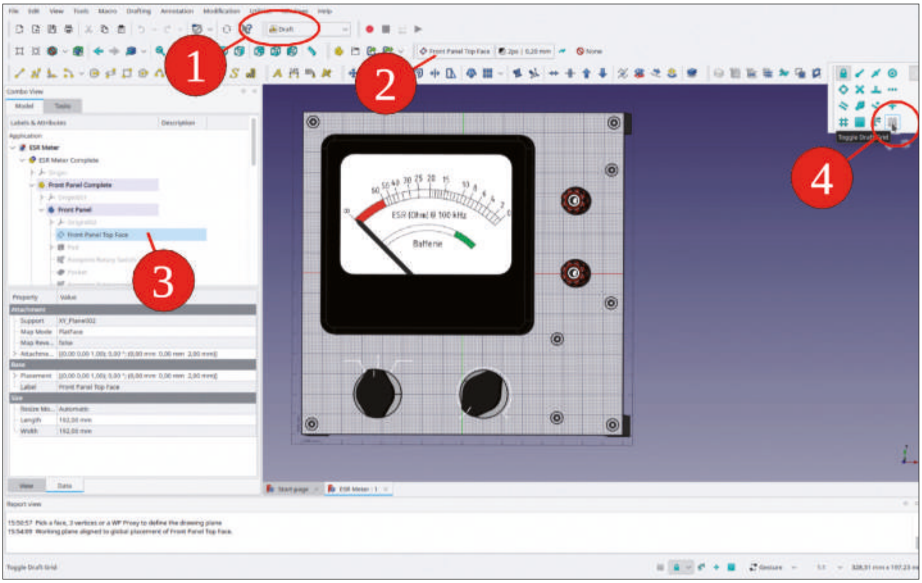


Figure 8-26

The outline of lettering can be generated with the 'Shape from text' tool. Let's now add labels to all the elements on the ESR meter front panel:

1. Click on the 'Shape from text' tool button (Figure 8-27).

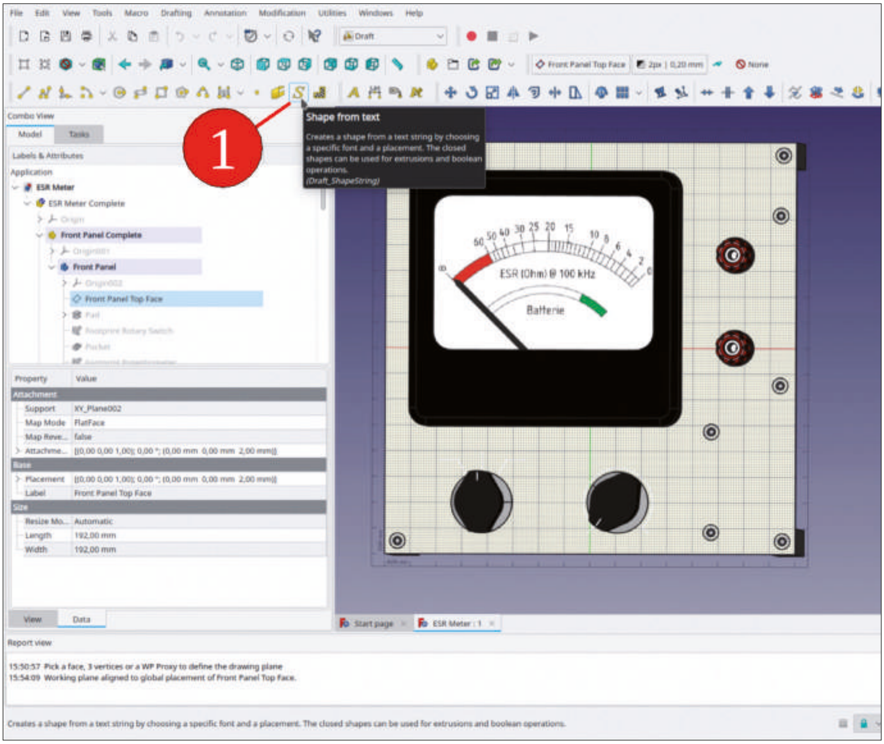


Figure 8-27

2. In the task window, enter 'Batt.' (designates 'battery testing') for the string and set the height to 6.5 mm. Click approximately at the location of the front panel, where you want to bring the text outline. Then, for the Z offset, enter 2 mm (the front panel thickness), as Z is reset to 0 when you clicked (Figure 8-28).

3. Also, the font file has to be specified. On Linux systems, you can find the freely distributed file 'RobotoCondensedLight.ttf', of which a copy is provided in the 'Sample Projects | ESR Meter | Components' directory.

4. Close the task window with the 'OK' button. Locate the 'Shape String' object in the tree view, grouped in the 'Front Panel Complete' Std-Part-Container, and mark it (Figure 8-28).

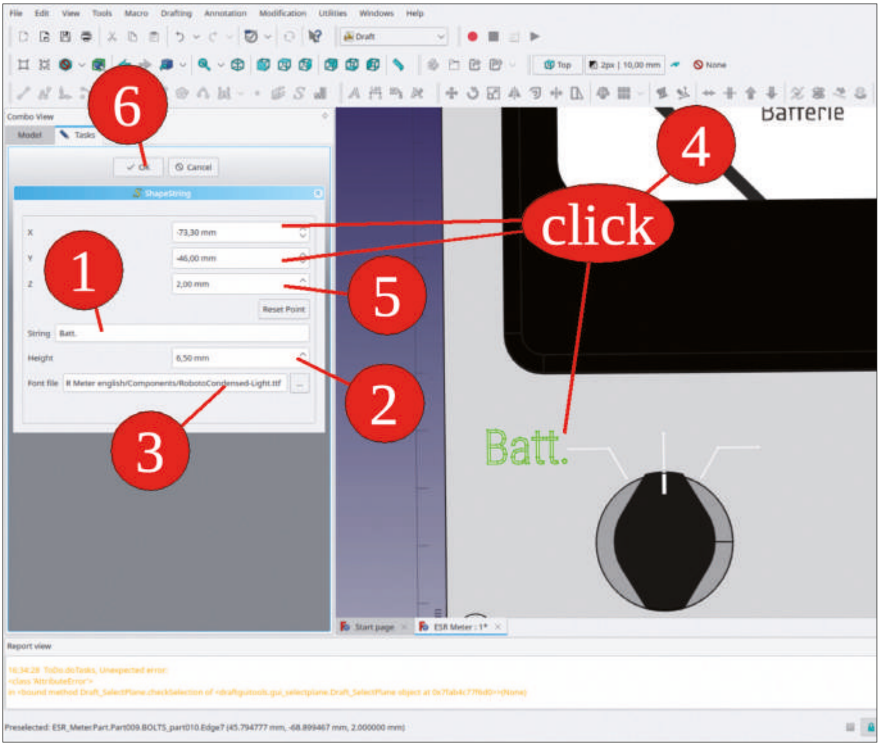


Figure 8-28

5. Edit the placement parameters (there, the placement can be modified with live update) and move X and Y coordinates with the mouse wheel, until the label appears at a good position (Figure 8-29). Close the task window with the OK button.

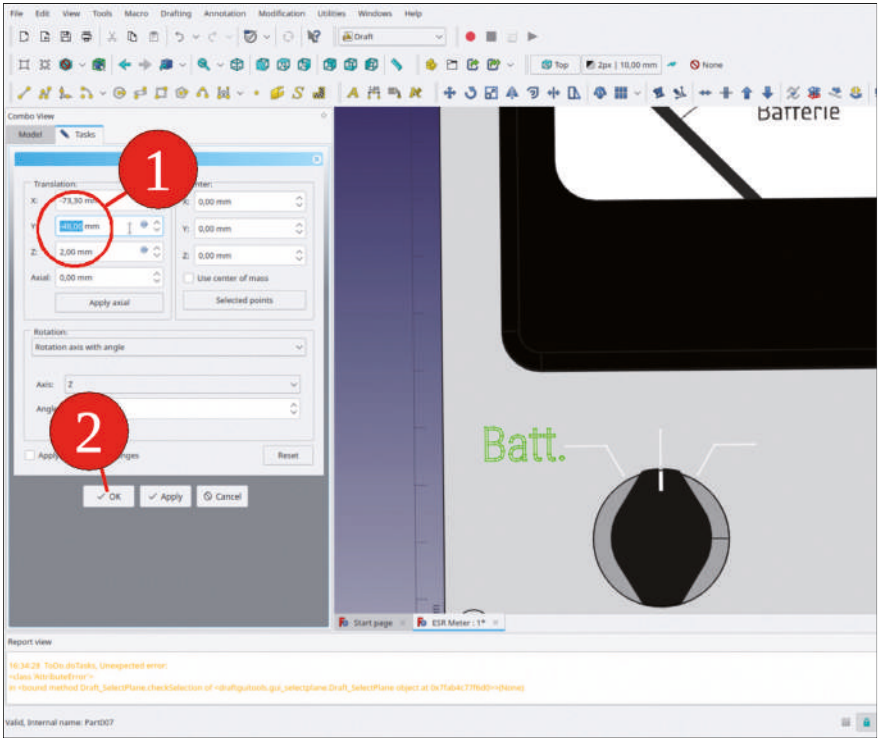


Figure 8-29

6. In similar fashion, create the labels 'OFF' and 'ON', and then move them to the locations shown in Figure 8.30.

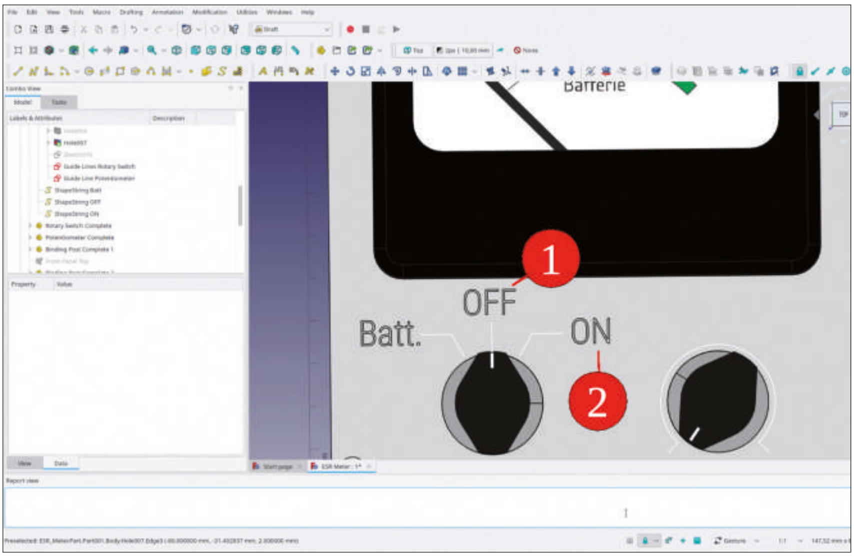


Figure 8-30

7. Above the potentiometer, create the label 'Set Zero' (Figure 8-31).

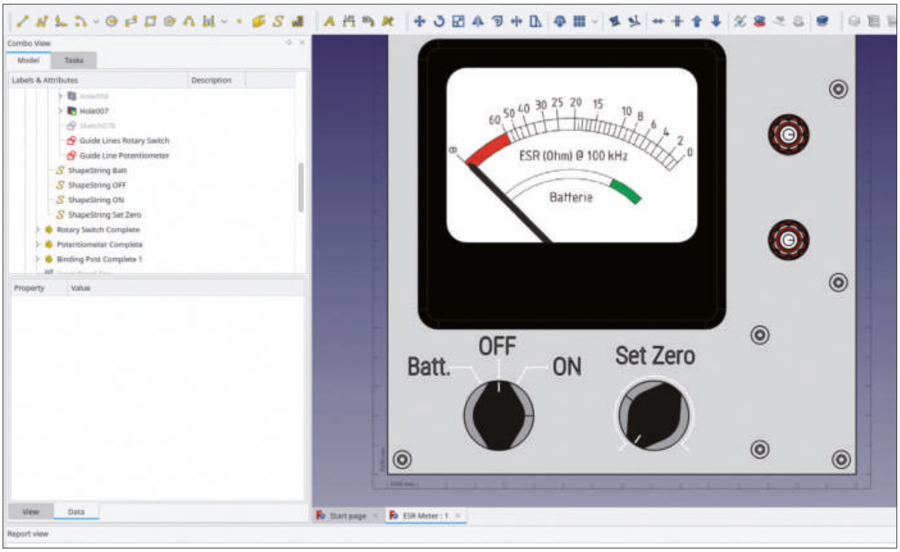


Figure 8-31

8. Between the binding posts, position the letters 'C' and 'x' as shown in Figure 8-32. The x should be a subscript, i.e. a bit smaller in type.

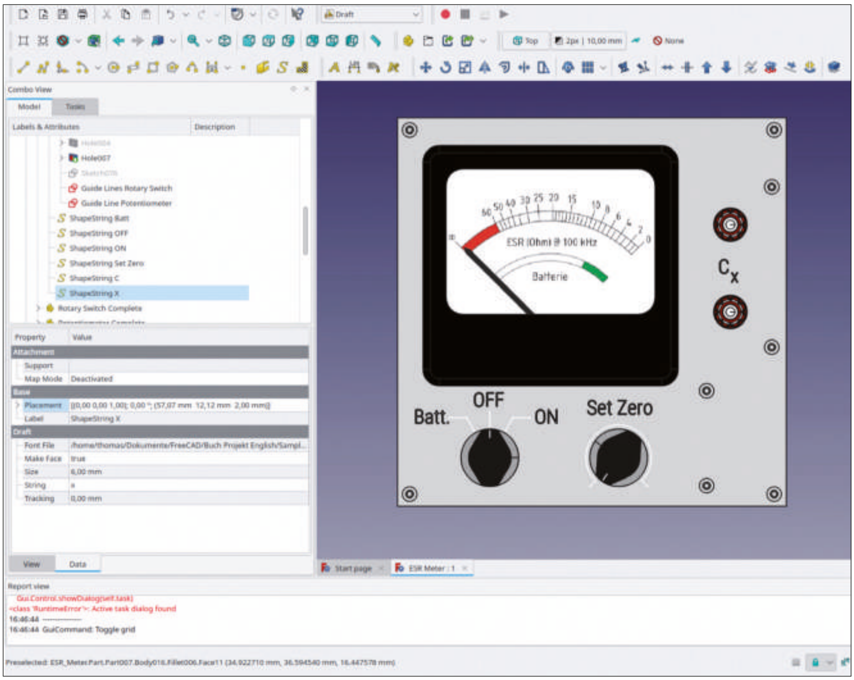


Figure 8-32

The Elektor ESR Meter front panel design is now completed.

8.4 The 'TechDraw' Workbench and the DXF Export

Already with the battery holder, you have used the 'TechDraw' workbench. The procedure is quite similar here, except that no unfolding is necessary for the straight front panel. For the engraving, let's exploit the fact that sketches are also displayed in the views created in 'TechDraw' documents.

1. In the 3D view, select the orientation 'Top', either by clicking on the control cube or by clicking the 'Top' tool button (Figure 8-3). The precise orientation is important. Otherwise, a slightly distorted projection will be displayed in the views, in which the dimensions deviate from the plain object. Figure 8-33

2. Switch to the 'TechDraw' workbench. Click on the 'Insert Page using Template' tool button (Figure 8-34).

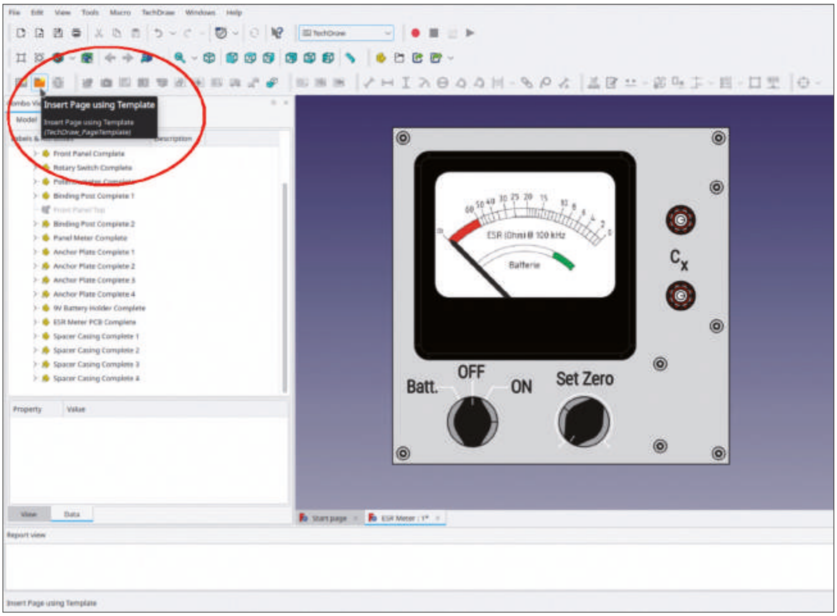


Figure 8-34

3. In the popup selection dialog, as a template, select 'A3 Landscape blank'.

4. In the tree view, mark 'Front Panel Complete' by a click (this selects what gets into the view in the next step).

5. Click on the 'Insert View' tool button. The actual view of the front panel is inserted in the TechDraw page (Figure 8-35). Right-click the frame and select 'Toggle frames' from the context menu, in order to hide the frame.

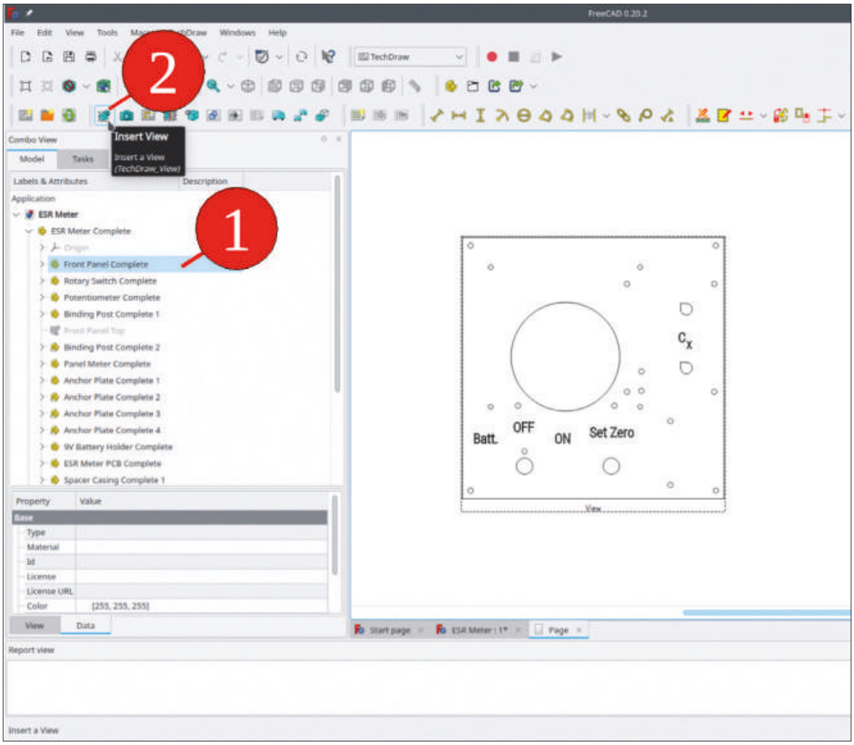


Figure 8-35

6. The export of the view is now the easiest part: Right-click the view and select 'Export DXF' from the context menu. (Figure 8-36).

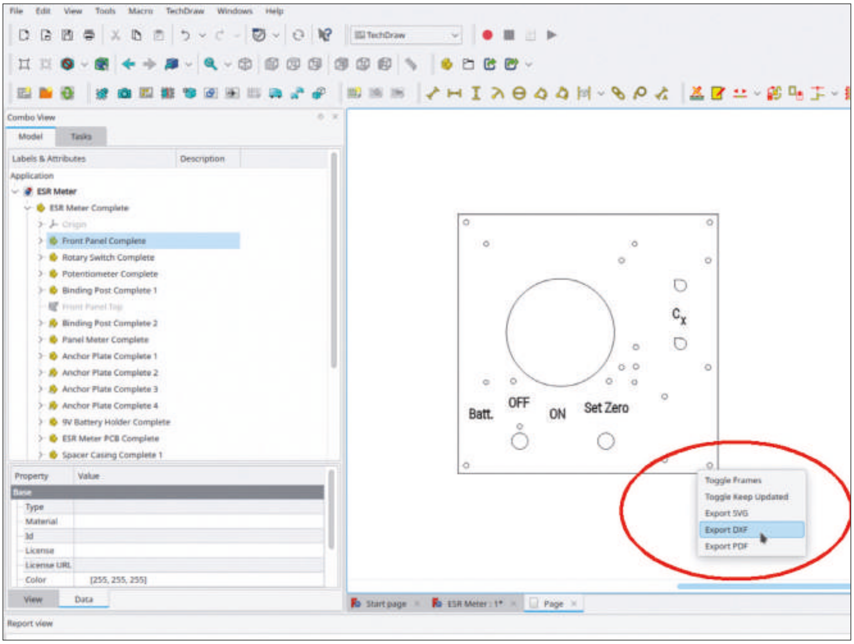


Figure 8-36

With this astonishingly small number of clicks, our first FreeCAD project is prepared for fabricating on a real machine. The next four photos show that this was not just theory. The effort invested during the design phase pays off with an easy assembly and with no collisions or other surprises (Figures 8-37, 8-38, 8-39).

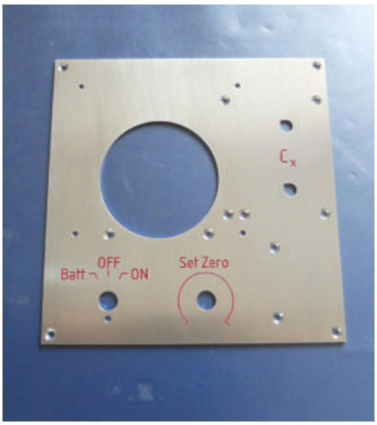


Figure 8-37

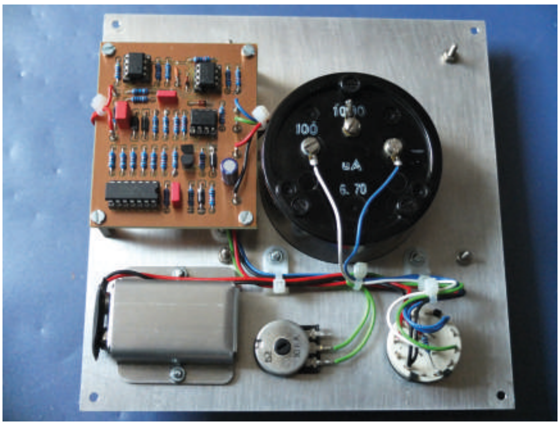


Figure 8-38

Figure 8-39

Figure 8-40

Chapter 9 • A More Complex Design: The Lab Transformer

With the tools established in the preceding chapters, you now have all options on the table to design a complete apparatus. Often, a piece of equipment consists of a front panel, a rear panel, and a chassis in between. The chassis carries components like a circuit board or — with similar project tree structure – more sub-assemblies packed into nested Std-Part-Containers. Finally, a sheet-metal cover will protect the inner workings from prying fingers and dust.

As an example, a laboratory transformer is selected. Admittedly, this is a bit of an old-school device, but still, such a device can give useful service in research and development even in today's era of switch-mode power supplies.

For the lab transformer, several components will be assembled. A few of these are described in Appendix A through F. If you prefer to start with the existing component files, just fetch them from here: Sample Projects | Lab Transformer | Components.

In order to keep things nicely encapsulated in smaller units, let's start to create the front and rear panels in separate FreeCAD files. The sub-assemblies can then be inserted into the final design by cut and paste. The chassis will be added during the course of the assembly as it ties together the panels and will be affected through associativity when the panels are repositioned.

9.1 The Front Panel

On the front panel, the user expects the secondary coil terminals of the transformer to which the test setup is then attached. Regarding simplified wiring, all connections to the mains (powerline; AC power) can reside close to the rear panel, including the mains switch, pilot light, and power inlet.

This constellation, however, is not necessarily the most practical one. When arranging experiments, it is easy to lose sight of the pilot light or the mains switch. Also, in some situations, you may want to remove the power quickly and not have to reach across a whole pile of things to locate it by 'blind' finger poking.

With FreeCAD, the different constellations of an assembly are easily compared, and the spatial requirements are evaluated. Therefore, you can opt for improved usability and position the pilot light and the mains switch onto the front panel. Whether or not this leaves space for the wiring can then be explored with the 3D model.

9.1.1 Setting up File and Std-Part-Container

1. Create a new folder for the lab transformer. Then, with FreeCAD, create a new file and save it as 'Transformer Front Panel'. In the document, start a new Std-Part-Container, and rename it to 'Trafo Front Panel Complete'. ('Trafo' is a common abbreviation for 'transformer' in German). The container is used later to collect all the assembled front panel elements. Also, the container is later inserted into the final design, with a cut and paste.

2. Inside the container, start a new Std-Part-Container and rename it to 'Front Panel Sheet Complete'. Within this container, you will later collect all engraving. Switch to the 'Part Design' workbench. In 'Front Panel Sheet Complete', start a new body, and rename it to 'Front Panel Sheet'.

3. Save the result, which is illustrated in Figure 9-1.

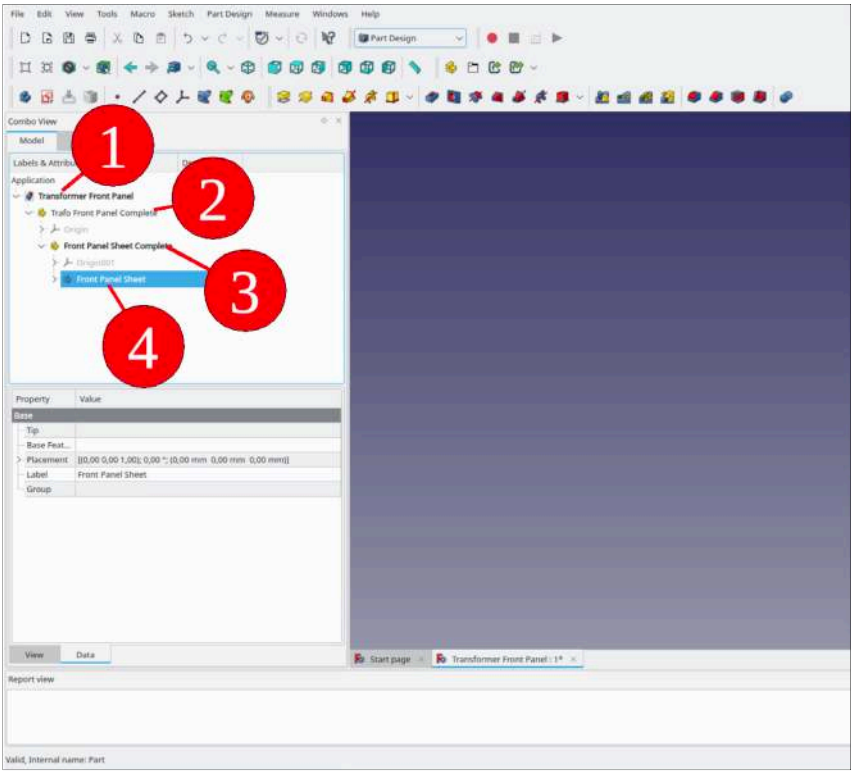


Figure 9-1

9.1.2 Creating the Panel Sheet

1. Double-click the body 'Front Panel Sheet' to activate it.

2. Click on the 'Sketcher' tool button and select the XY plane in the initial selection task window. Click 'OK' to proceed (Figure 9-2).

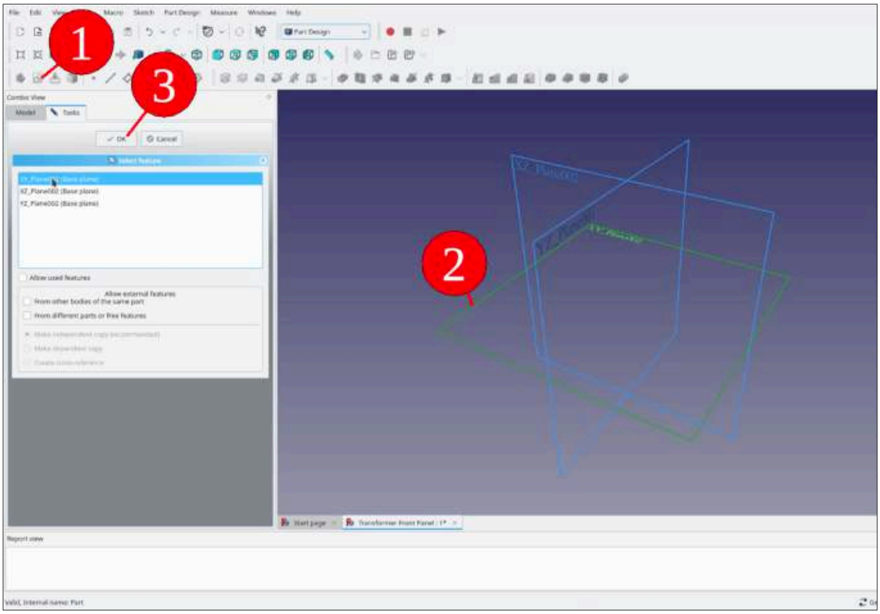


Figure 9-2

3. Draw a rectangle whose bottom side coincides with the X-axis (Figure 9-3). End the drawing command with a right-click.

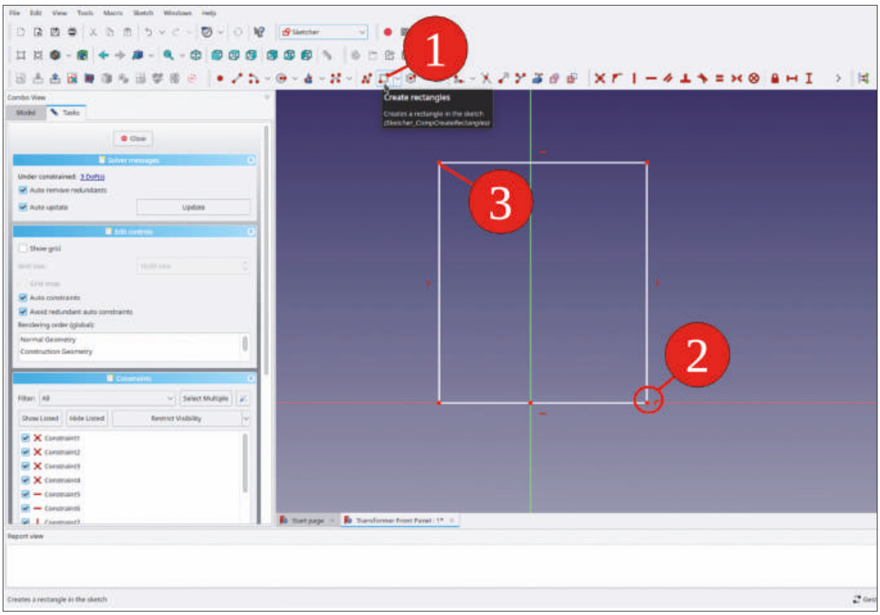


Figure 9-3

4. Mark the lower two corners of the rectangle, and the Y-axis. Then, click the 'Constrain symmetrical' tool button (Figure 9-4).

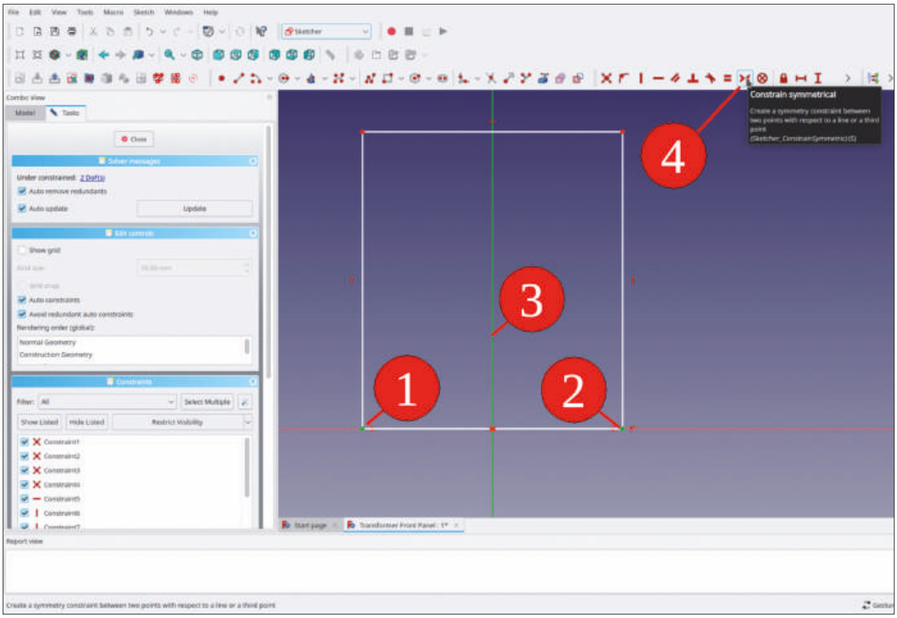


Figure 9-4

5. Mark a horizontal line of the rectangle and click the 'Constrain horizontal distance' tool button. Enter a value of 75 mm for the width (Figure 9-5). In a similar way, constrain the vertical dimension of the rectangle to 90 mm. The sketch is then shown in bright green, as fully constrained. Close the sketch (top).

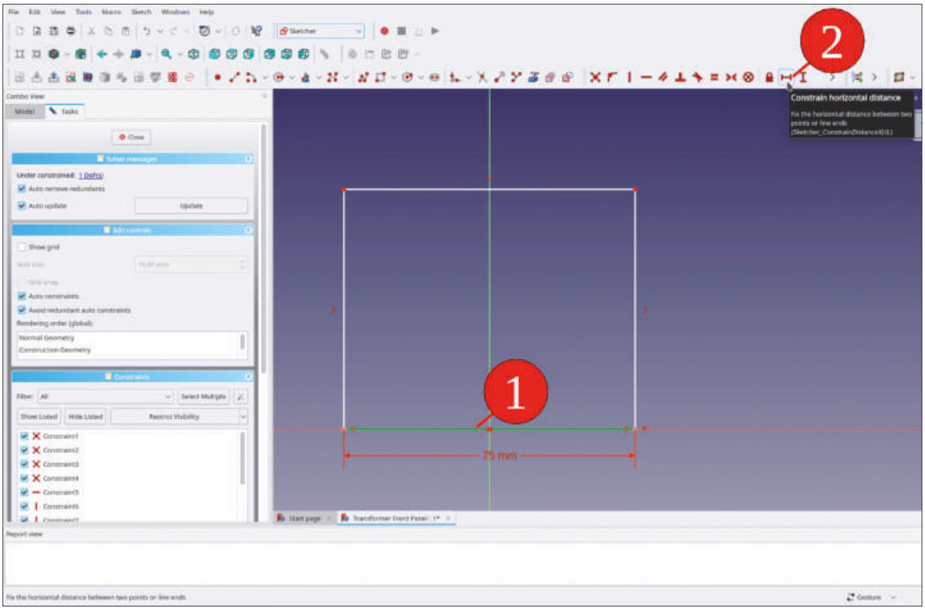


Figure 9-5

6. For the definition of the front panel top face, create a datum plane: In the tree view, expand the coordinate system of the body 'Front Panel Sheet' and mark the XY plane. Then, click the 'Create a datum plane' tool button (Figure 9-6).

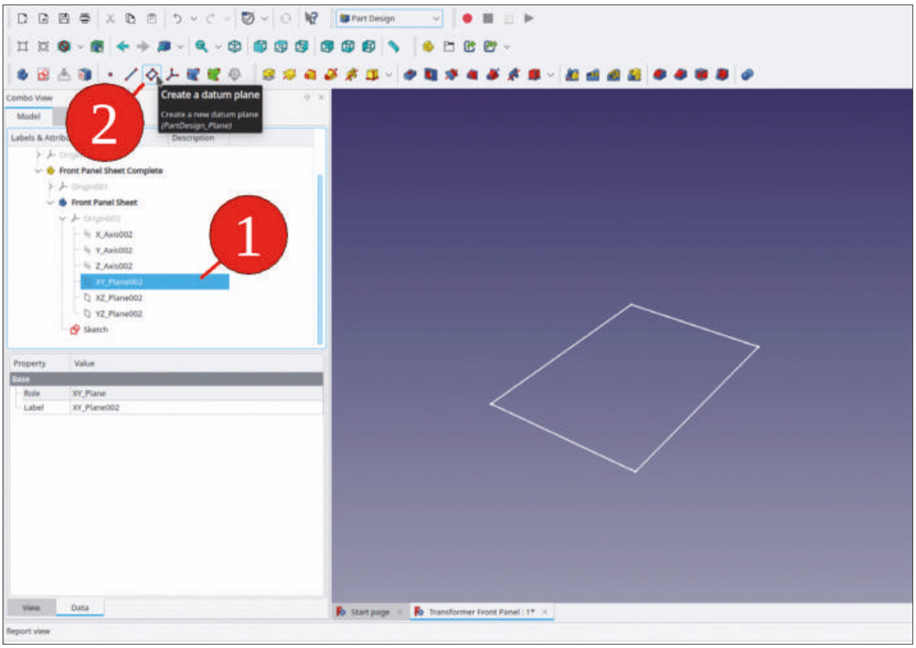


Figure 9-6

7. For the attachment Z offset, enter a value of 2 mm (Figure 9-7) and then close the task window with the OK' button. This offset represents the panel thickness.

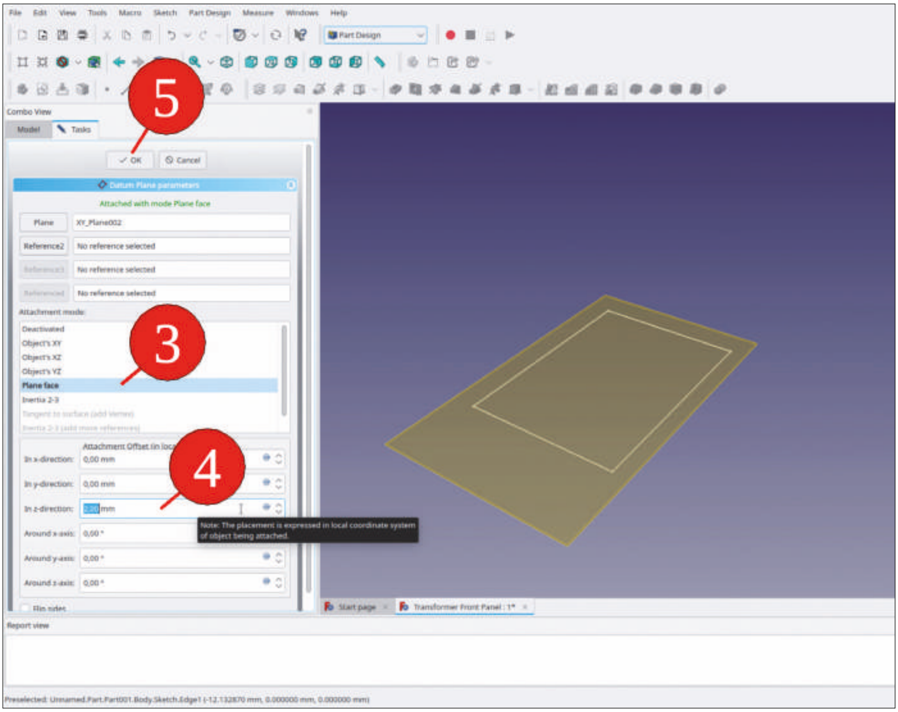


Figure 9-7

8. In the tree view, mark the sketch of the panel outline, and click the 'Pad' tool button (Figure 9-8).

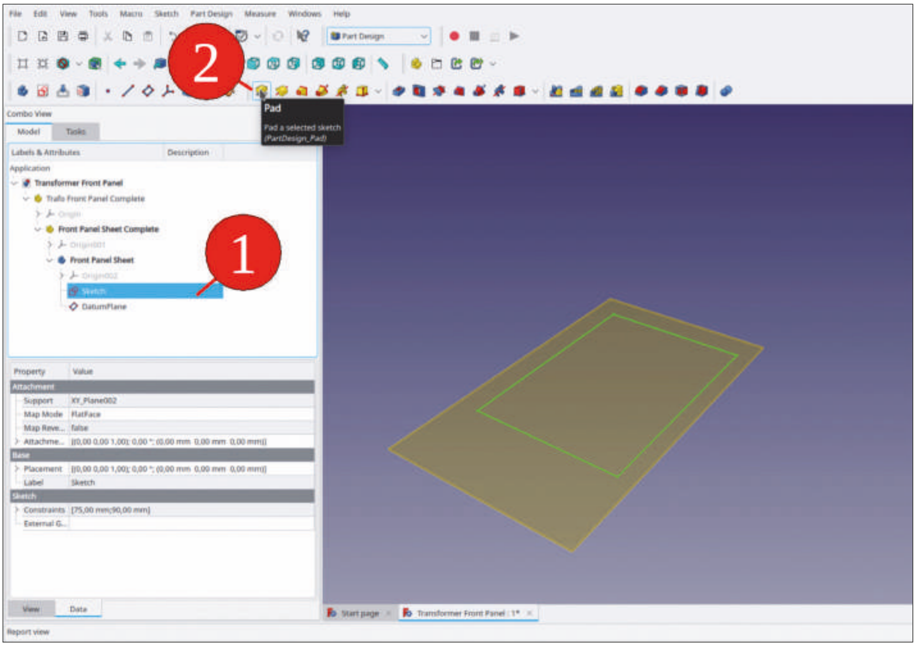


Figure 9-8

9. In the task window, select for the type 'Up to face'. Then, click the datum plane representation in the 3D view (Figure 9-9). Close the task window with the 'OK' button.

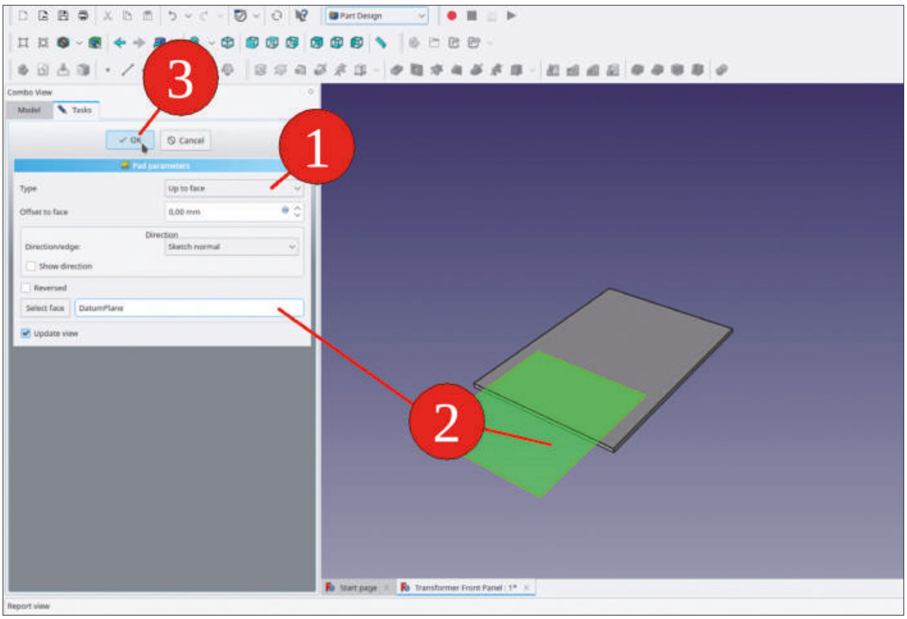


Figure 9-9

For the attachment of components to the front panel top face, a resilient reference is needed. As in the preceding chapter, you can provide it with a sketch on the datum plane, to which you then reference a SubShapeBinder.

10. In the tree view, rename the datum plane to 'Front Panel Sheet Top'. In the tree view, mark the datum plane and start the sketcher. To display the existing 3D geometry, close and reopen the sketch right away.

11. Click on the 'External geometry' tool button and mark two adjacent lines of the rectangle (Figure 9-10), which are then shown in violet as construction lines.

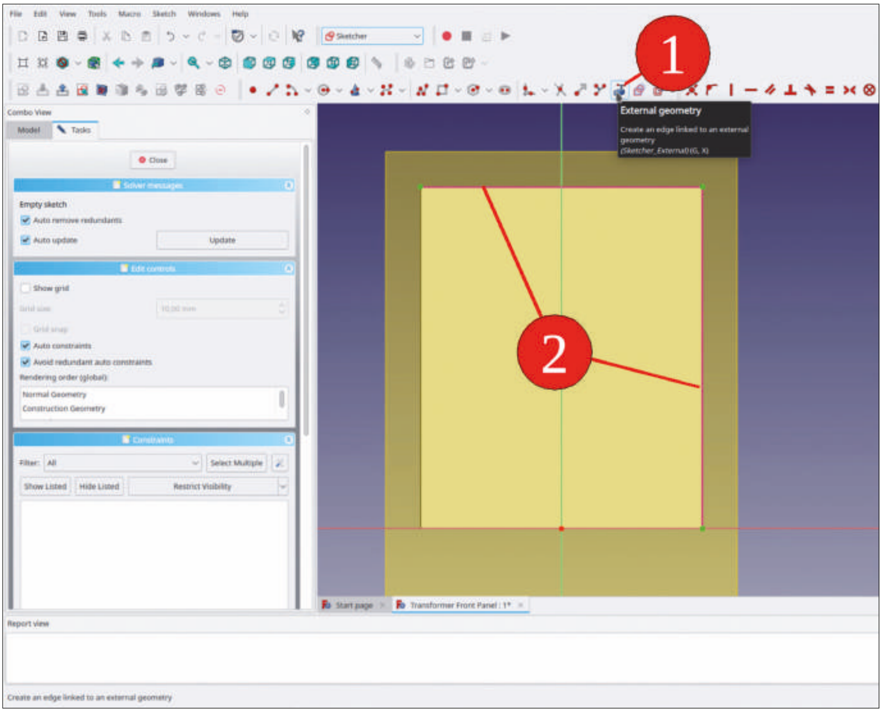


Figure 9-10

12. Click on the 'Create a rectangle' tool button and draw a rectangle locked onto the two endpoints of the construction geometry (Figure 9-11). Close the sketch.

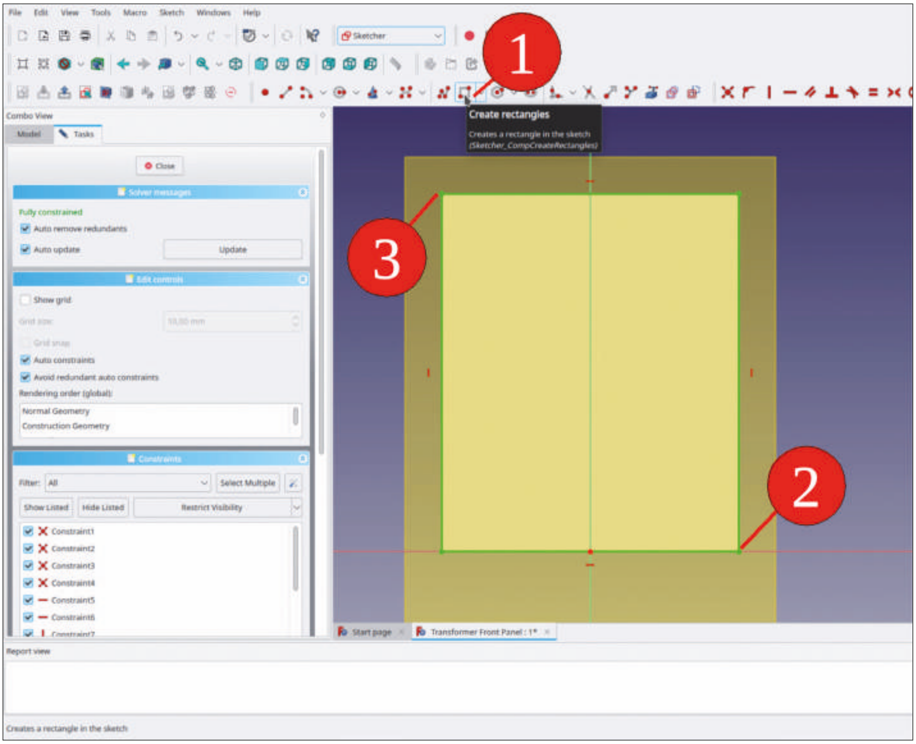


Figure 9-11

13. Rename the new sketch to 'Front Panel Top Contour'. Deactivate the body 'Front Panel Sheet' by a right-click and select 'Toggle active body' from the context menu.

14. In the tree view, mark the new sketch and click the green 'Create a sub object(s) shape binder' tool button. Rename the new SubShapeBinder to 'Front Panel Top Face'. Drag and drop it into the Std-Part-Container 'Trafo Front Panel Complete' (Figure 9-12).

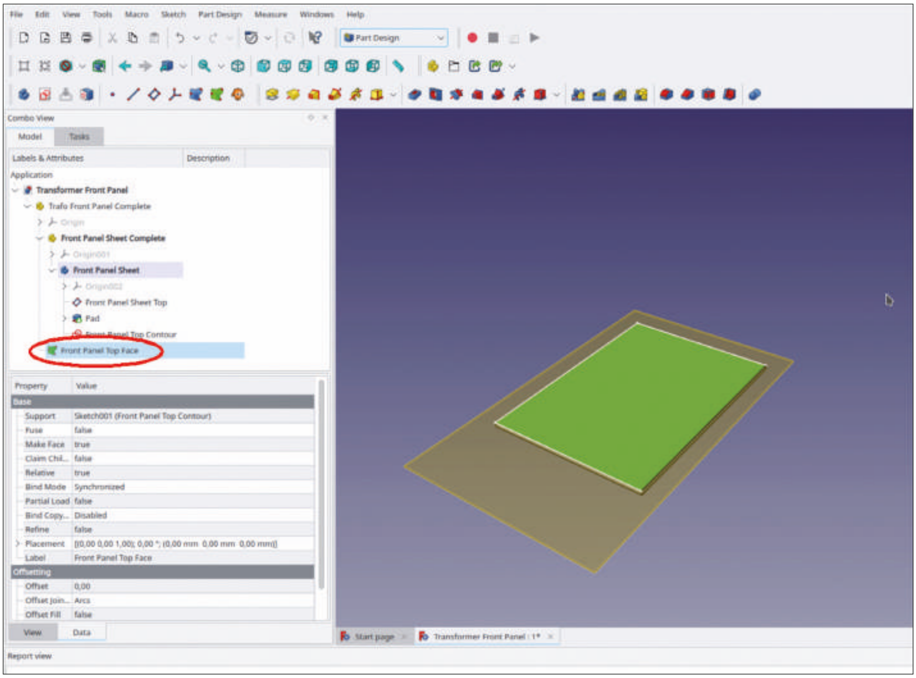


Figure 9-12

15. In the tree view, hide the datum plane 'Front Panel Sheet Top, the sketch 'Front Panel Top Contour' and the new SubShapeBinder 'Front Panel Top Face'.

16. Up to this point, the rear panel equals the front panel (except some designations in the tree). Therefore, save a copy of the file as 'Transformer Rear Panel', in order to save time later on.

The procedure looks a bit unwieldly, but the references defined here help to copy and paste the front panel (sub) assembly later on.

9.1.3 Placement of the Components

1. In the directory 'Sample Projects | Lab Transformer | Components', locate and open the file 'Toggle Switch.FCStd'. Copy the Std-Part-Container 'Toggle Switch Complete' with CTRL-C. In the popup dialog, do not change the standard selection and click the 'OK' button to continue.

2. In the tree view, close the document node 'Toggle Switch'. If the title is not shown in bold letters (which may be the case with more documents open), double-click the node 'Transformer Front Panel'. Then, paste the switch with CTRL-V. Drag and drop the Std-Part-Container with the switch into 'Trafo Front Panel Complete' (Figure 9-13).

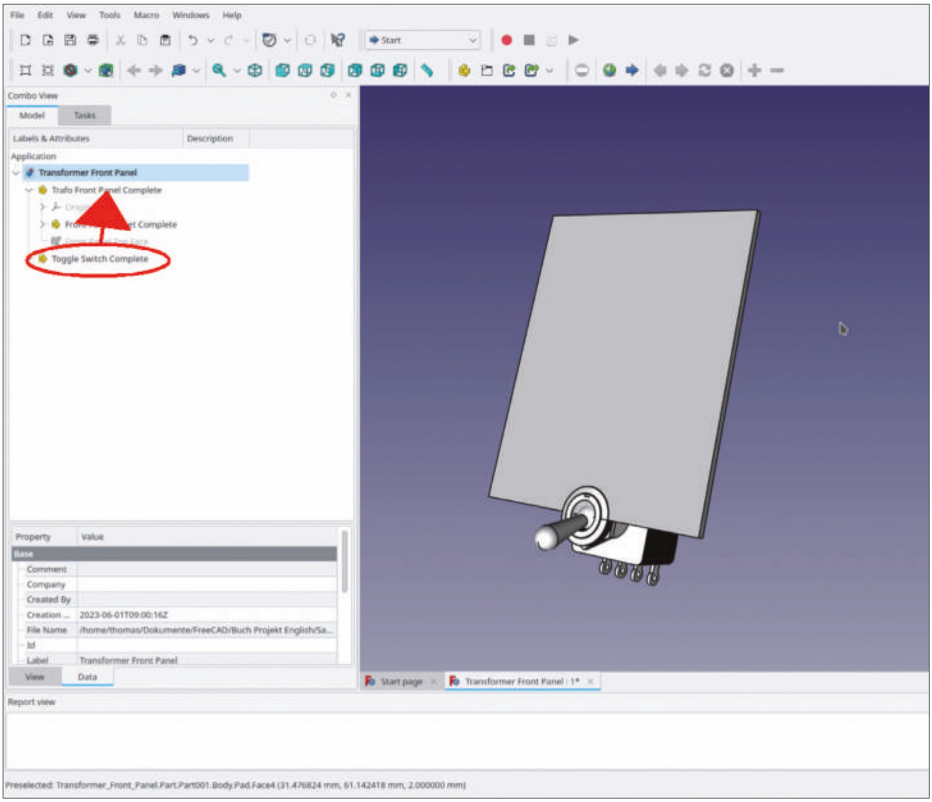


Figure 9-13

3. In the tree view, mark 'Toggle Switch Complete'. Switch to the 'Part' workspace and select 'Part | Attachment' from the main menu.

4. In the task window, click into the entry field next to the button for Reference 1, which is displayed in dark gray, and of which the label reads 'Selecting…' (Figure 9-14).

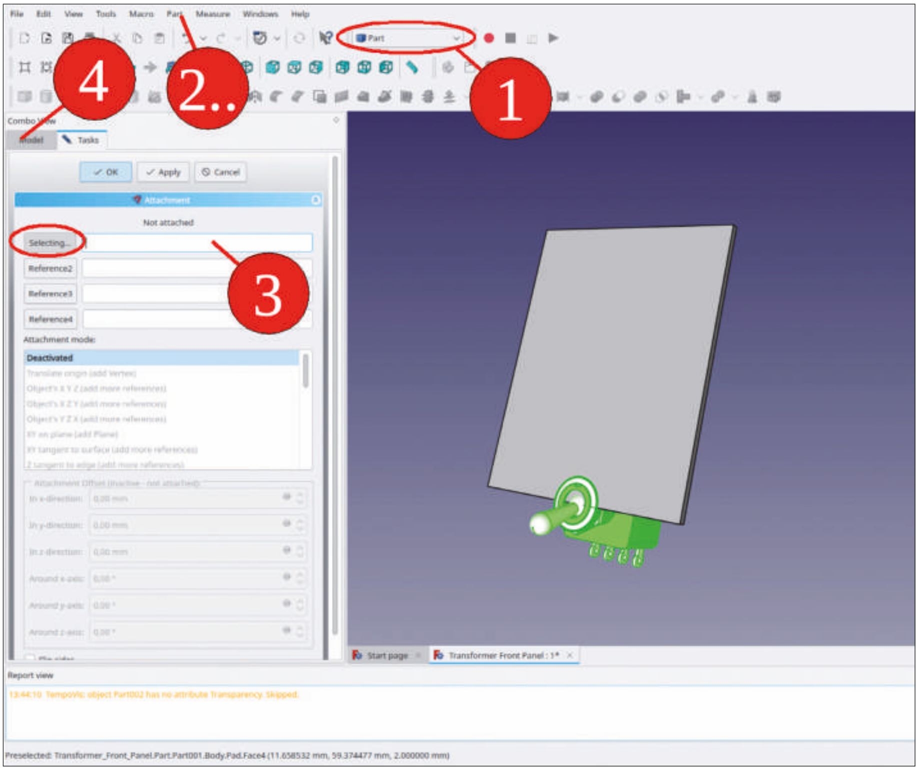


Figure 9-14

5. Switch to the 'Model' tab (Figure 9-14, step 4) and, in the tree view, click the SubShapeBinder. Switch back to the 'Tasks' tab. For the attachment mode, select 'XY on plane', and set the attachment offset Y to 32 mm (Figure 9-15). Close the task window.

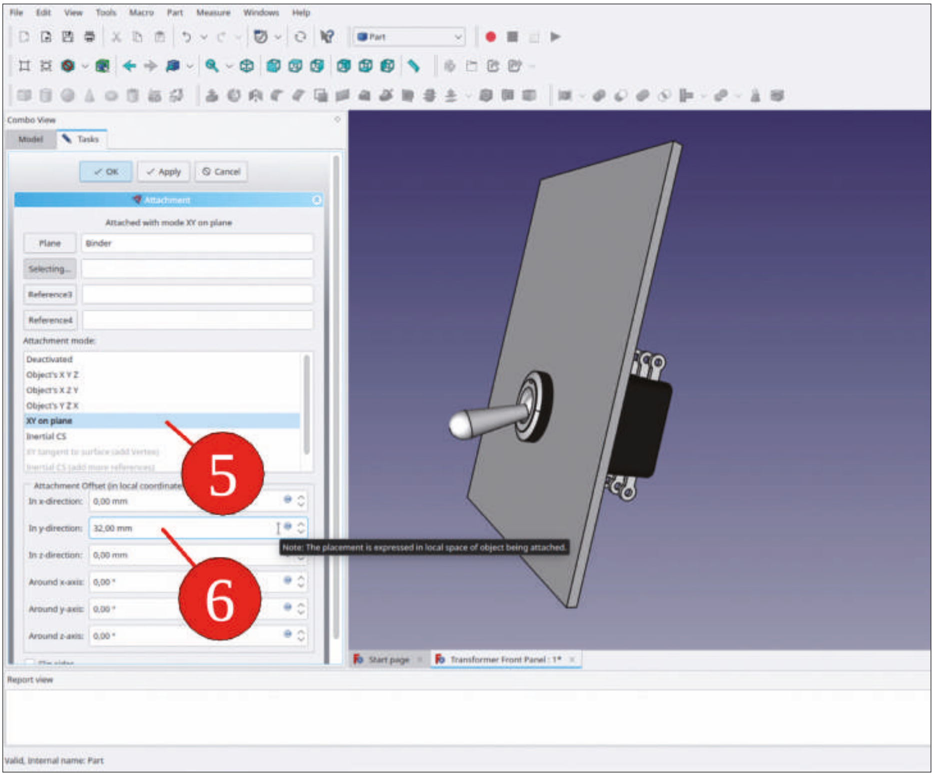


Figure 9-15

When you enter values for dimensions, occasionally, error messages appear in the output window. This happens when you just type the figure without units (which is usually the case). These error messages can be safely ignored and deleted, because FreeCAD will automatically add the dimension again.

Following the same procedure as described in the steps 1 to 5 above, place the following components onto the front panel and position them with respect to the front face:

Object	X [mm]	Y [mm]
Pilot Lamp Complete	0	65
Banana Jacket Metal Complete	0	6
Banana Jack Complete	24	22
Banana Jack Complete	−24	22

6. Rename one of the Std-Part-Containers called 'Banana Jack Complete XXX' to 'Banana Jack Complete 005', and the other one to 'Banana Jack Complete 001' (Figure 9-16). In the tree view, mark 'Banana Jack Complete 001' and click the 'Make link' tool button.

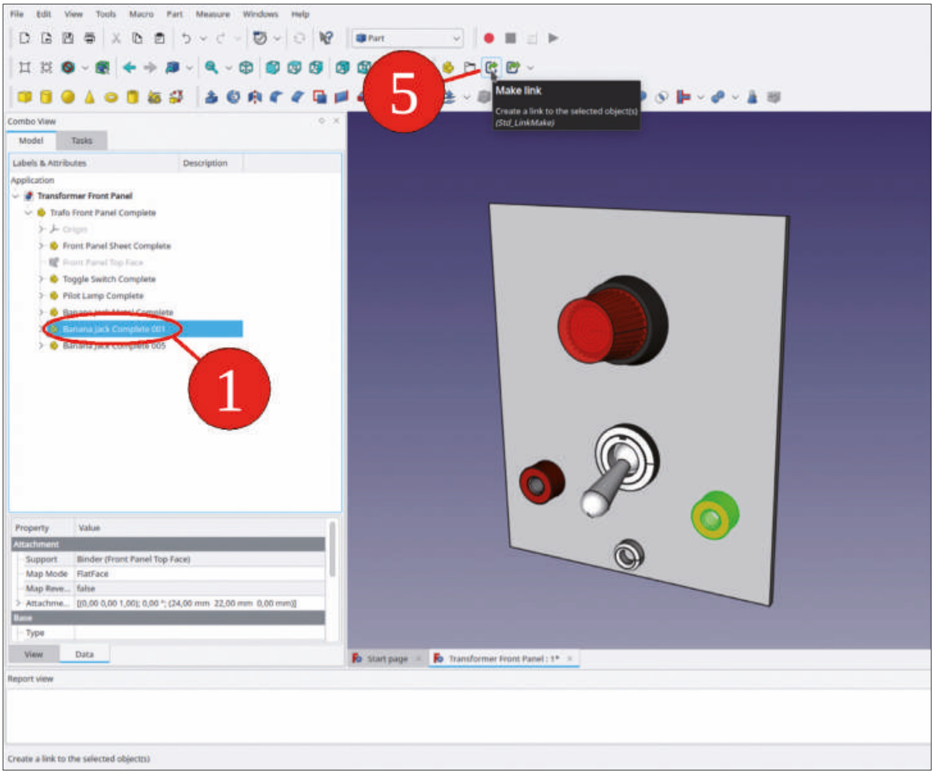


Figure 9-16

7. Create three more links of 'Banana Jack Complete 001' (Figure 9-17). Drag-and-drop the three new objects into the Std-Part-Container 'Trafo Front Panel Complete'.

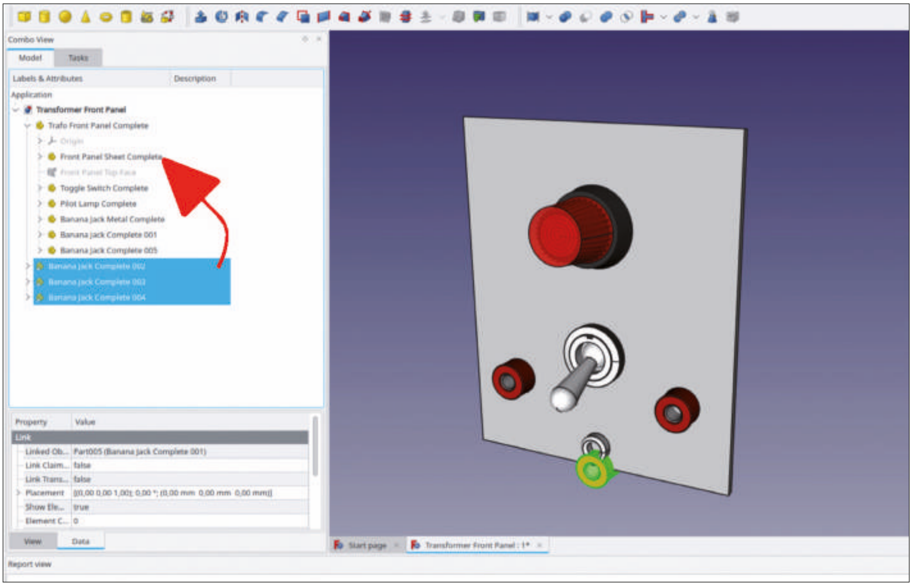


Figure 9-17

8. In the tree view, mark all the new banana jack link objects. In the property list, set their 'Link transform' property to True (Figure 9-18). Now all the positions can be specified as 'Link placement' parameters relative to the linked object 'Banana Jack Complete 001'.

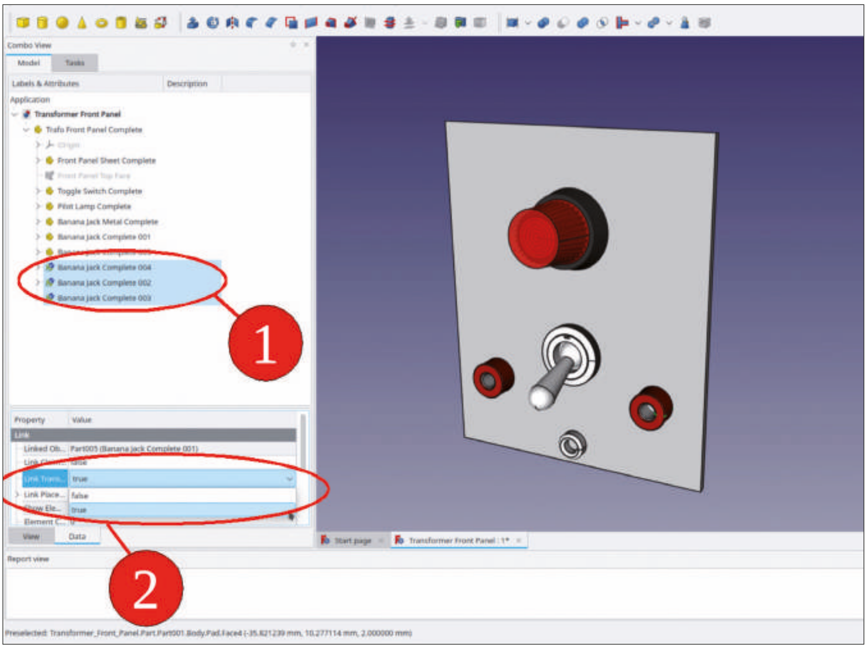


Figure 9-18

9. Adjust the link placement Y parameters according to the following table (Figure 9-19):

Object	Y [mm]
Banana Jack Complete 002, 006	17
Banana Jack Complete 003, 007	34
Banana Jack Complete 004, 008	51

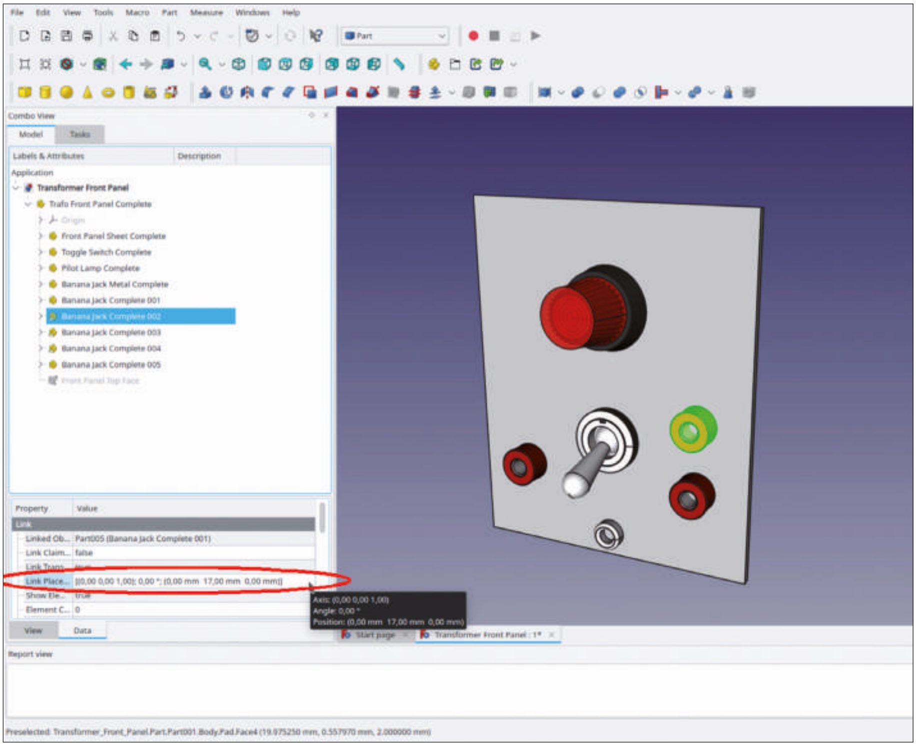


Figure 9-19

11. In the tree view, expand 'Banana Jack Complete 001'. Mark 'Banana Jack Cap' and 'Banana Jack Washer'. Right-click the selection and select 'Appearance...' from the context menu (Figure 9-20).

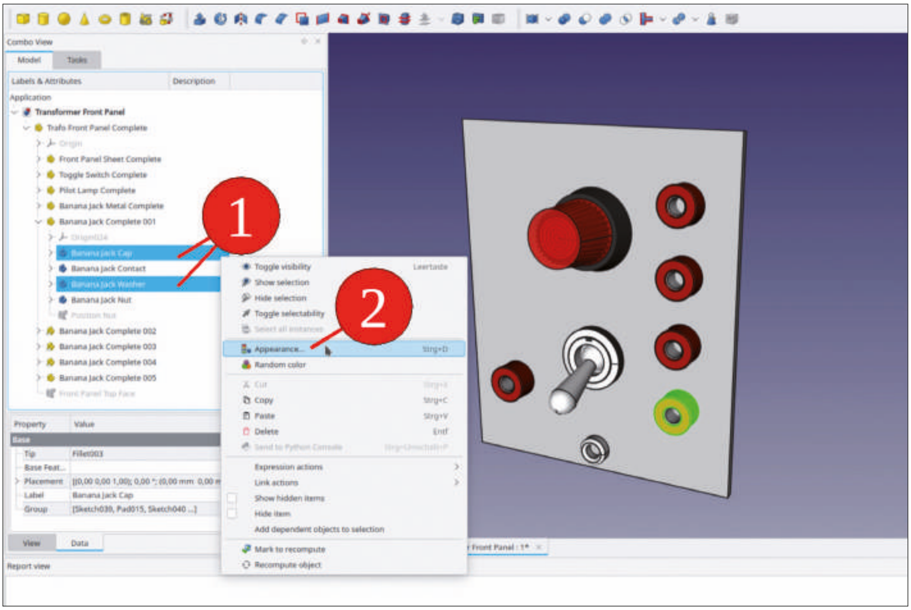


Figure 9-20

12. In the task window, select 'Shiny Plastic' for the material, and set the color to yellow (Figure 9-21). Close the task window.

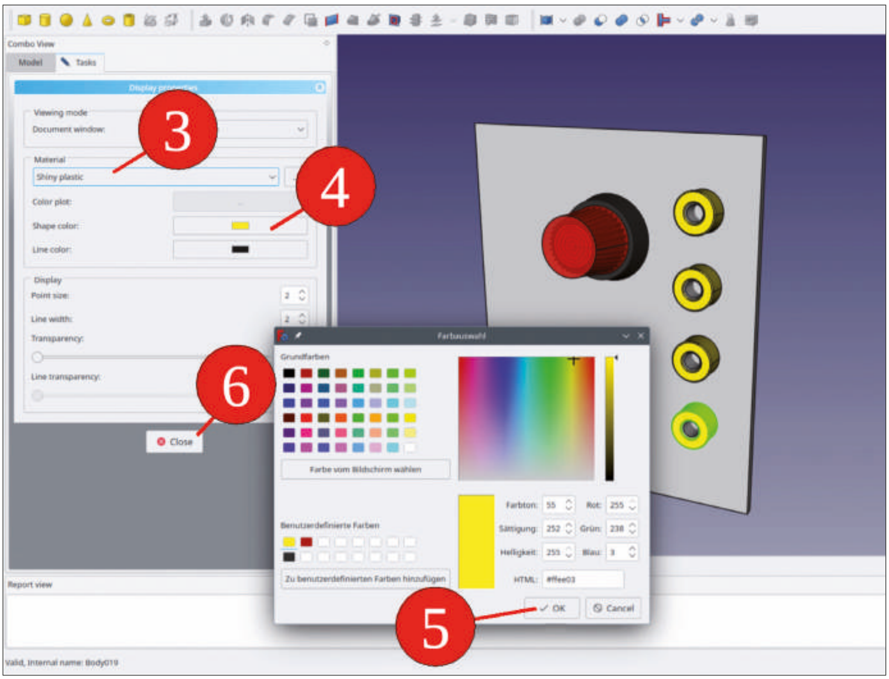


Figure 9-21

13. Repeat the creation of 3 link objects but click 'Banana Jack Complete 005' as the parent object (Figure 9-22). Drag-and-drop the linked jacks into 'Trafo Front Panel Complete'.

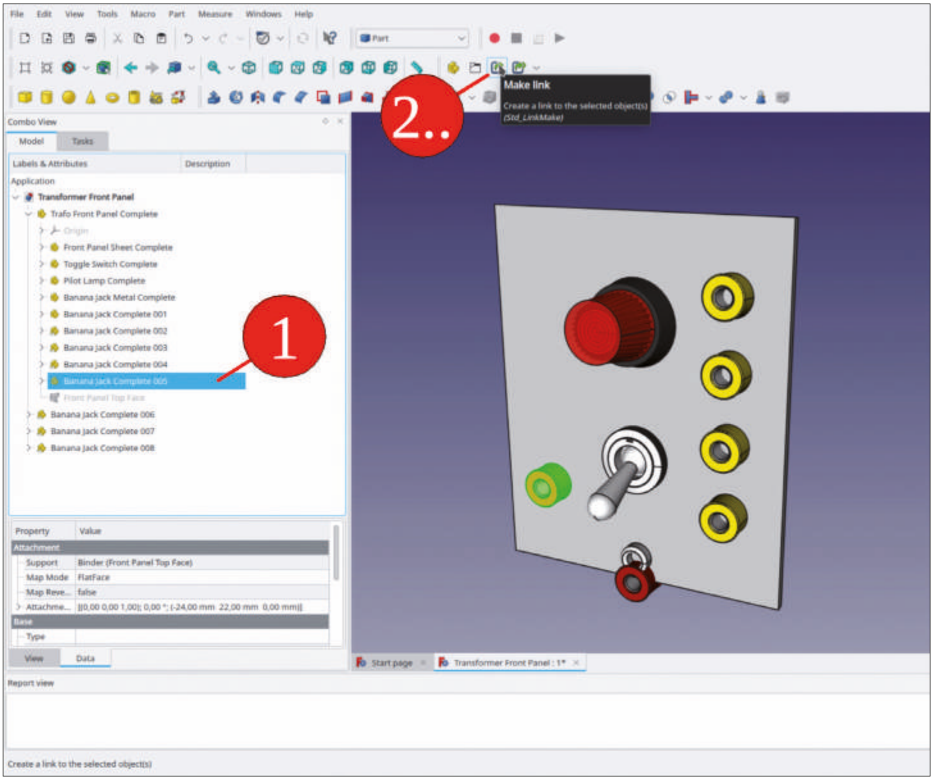


Figure 9-22

14. Again, mark the three new banana jack link objects, and set their 'Link transform' property to 'true'.

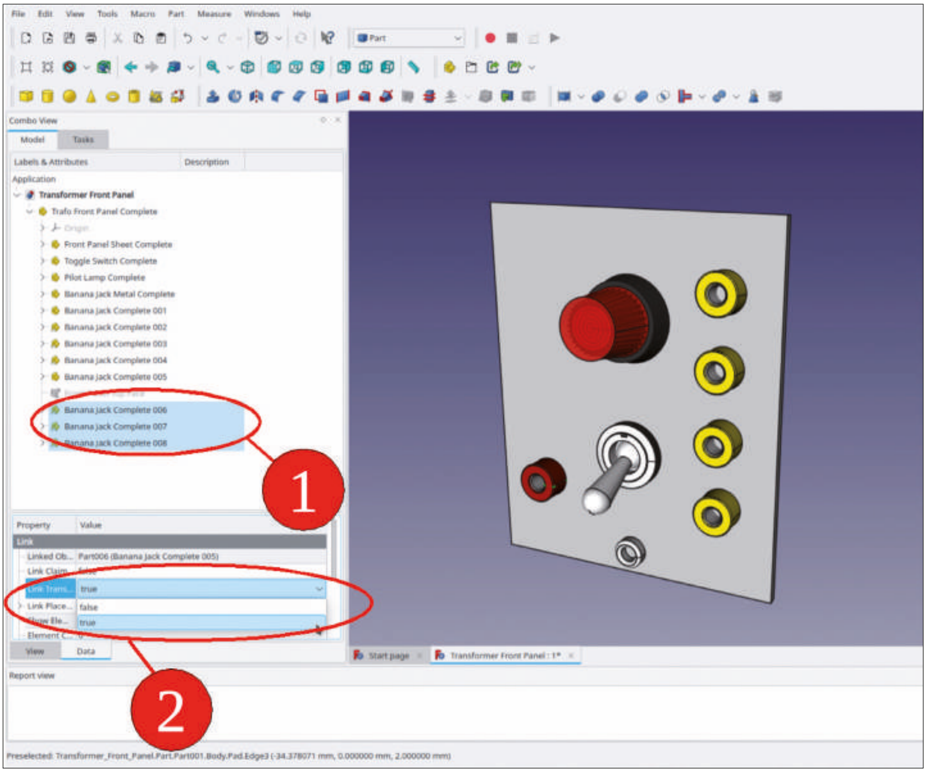


Figure 9-23

15. For the three new banana jack link objects, in the property list, set the link placement Y values according to the table above (in step 4).

16. Expand 'Banana Jack Complete 005' and mark 'Banana Jack Cap' and 'Banana Cap Washer'. As in step 5, set the appearance of these objects to the material 'Shiny Plastic', but select a bright blue for the color (Figure 9-24).

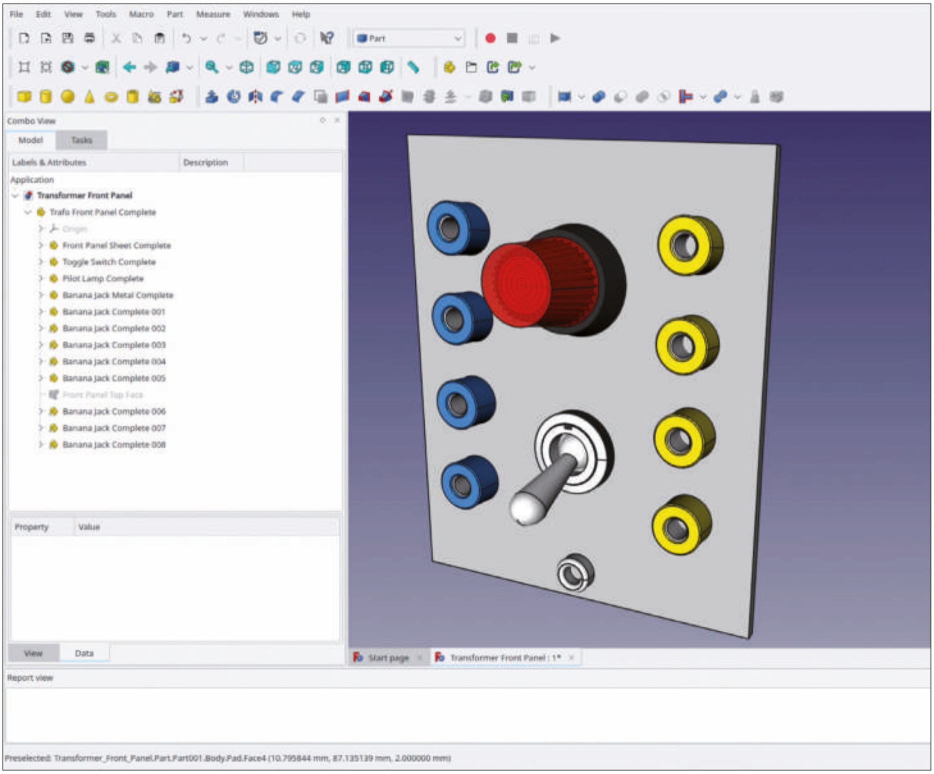


Figure 9-24

Now, all front panel elements are attached and positioned. Inside the jacks, and also in the casing of the switch, the sheet metal is still present. This will be cut away in the next section.

9.1.4 Punching the Holes

The footprints for the openings of the front panel elements will be referenced with SubShapeBinders again. In order to make the SubShapeBinders appear in the body called 'Front Panel Sheet', make sure that this body is always activated (the title shown in bold letters), before you click the 'Create a sub object(s) shape binder' tool button. Otherwise, the SubShapeBinder will be added to the currently activated object and you have to search it. The tree view will expand when you select reference objects, and it is the selection of the referenced objects rather than the created SubShapeBinder which is then centered in the tree view. As a consequence, the following procedure includes quite a bit of scrolling to and fro.

1. To set the target for the SubShapeBinder, activate the body 'Front Panel Sheet' located within the Std-Part-Container 'Front Panel Sheet Complete'.

2. To expose the edges of the front panel elements, hide the body 'Front Panel Sheet' with the SPACE key.

3. In the 3D view, mark the toggle switch contour, which determines the shape of the hole in the front panel (Figure 9-25), and click the green 'Create a sub object(s) shape binder' tool button.

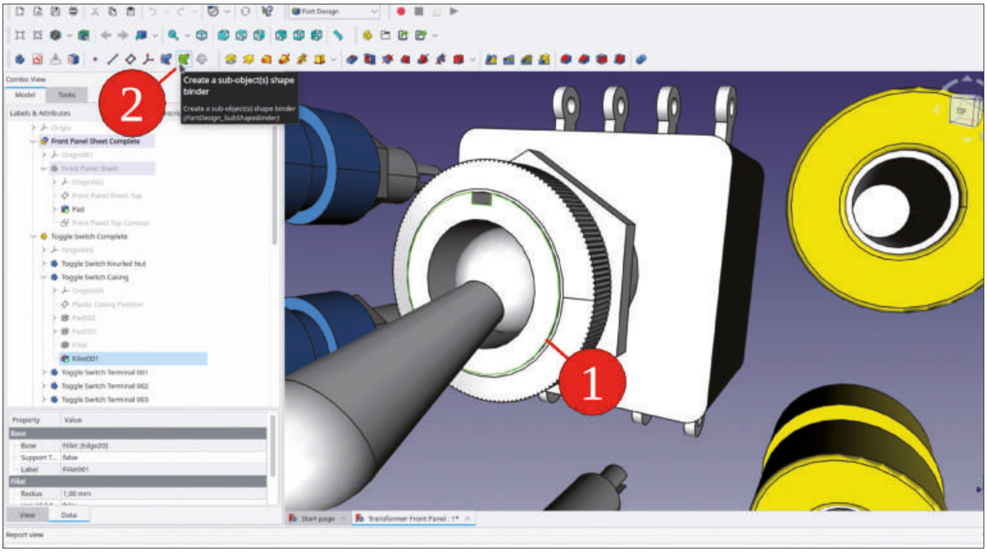


Figure 9-25

4. Locate the new SubShapeBinder in the body 'Front Panel Sheet' and rename it to 'Footprint Toggle Switch' (Figure 9-26).

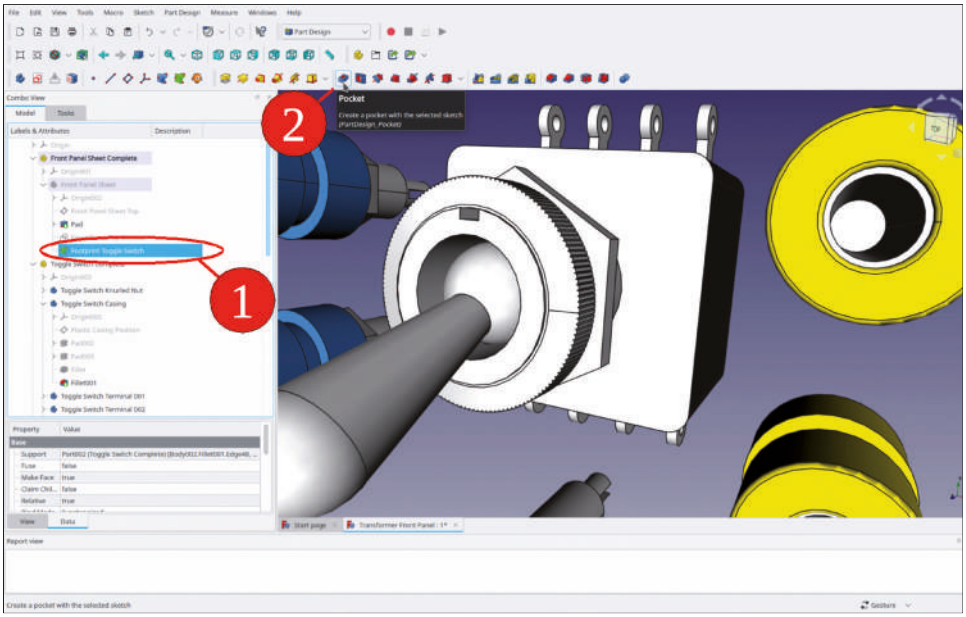


Figure 9-26

5. With the SubShapeBinder marked in the tree view, click the 'Pocket' tool button. In the task window, for the type, select 'Through all' (Figure 9-27). Close the task window with the 'OK' button.

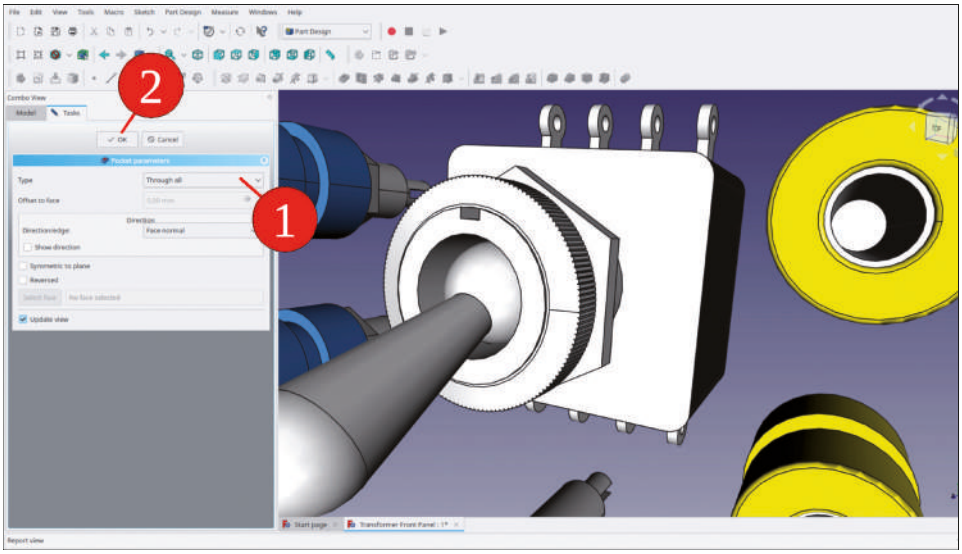


Figure 9-27

6. In order to check for success, temporarily hide the toggle switch and show the front panel (with the SPACE key, Figure 9-28).

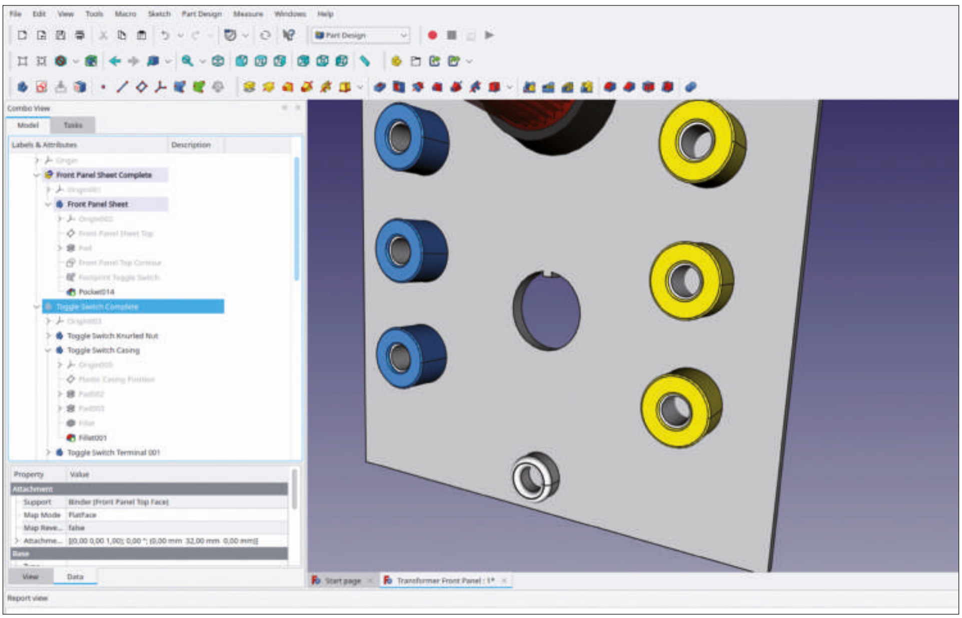


Figure 9-28

7. In the 3D view, rotate the assembly such that the rear of the front panel elements is visible. In the Std-Part-Container 'Pilot Lamp Complete', hide the 'Pilot Lamp Nut' body to expose the lamp casing. Make sure the 'Front Panel Sheet' is still activated (title shown in bold print).

8. Holding down the CNTRL key, mark the contour defining the front panel cutout for the pilot lamp. The chain of elements must be a closed loop (Figure 9-29). When you mark something that does not belong to the loop, just click it again to deselect it. Then, click the green 'Create a sub object(s) shape binder' tool button.

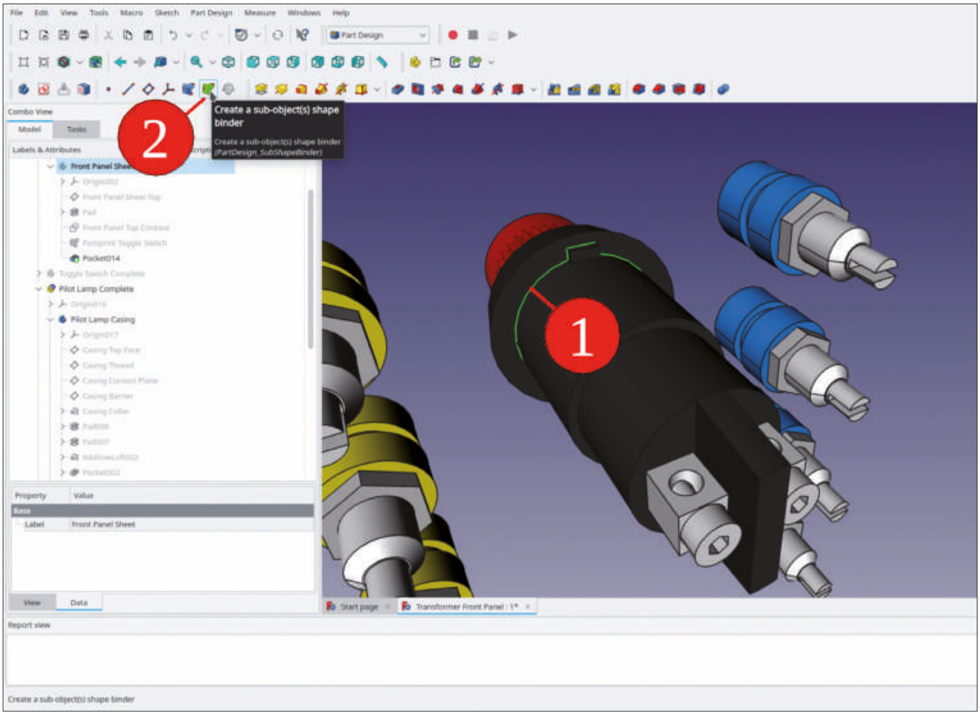


Figure 9-29

9. Scroll back to 'Front Panel Sheet' and rename the new SubShapeBinder to 'Footprint Pilot Lamp' (Figure 9-30). Then, with the new SubShapeBinder marked in the tree view, click the 'Pocket' tool button.

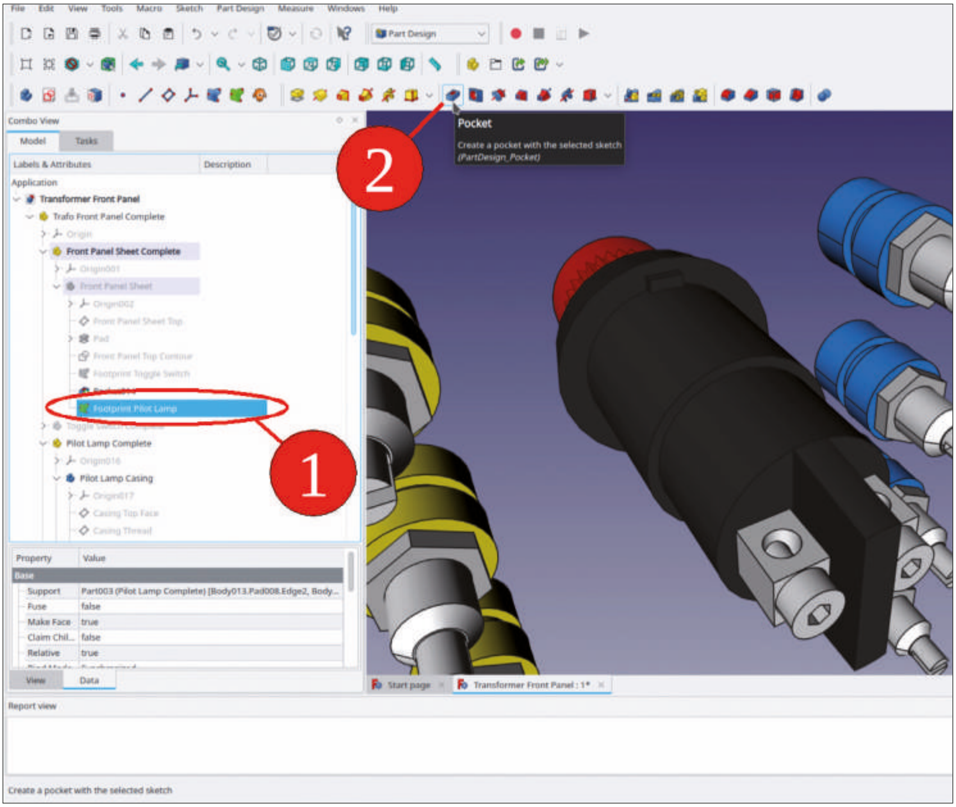


Figure 9-30

10. In the task window, for the Type, select 'Through all' and close the task with the 'OK' button.

11. In a similar way, mark all contours of the banana sockets defining the mounting holes and then create a SubShapeBinder. Rename that to 'Footprint Banana Jacks' (Figure 9-31) and create the holes with the 'Pocket' command (Figure 9-32).

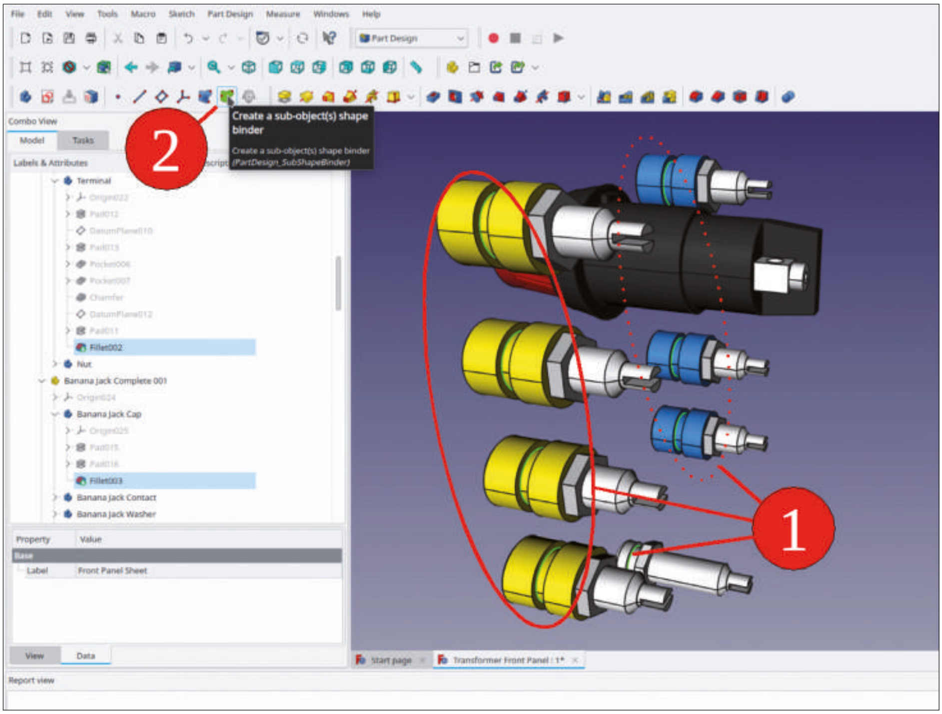


Figure 9-31

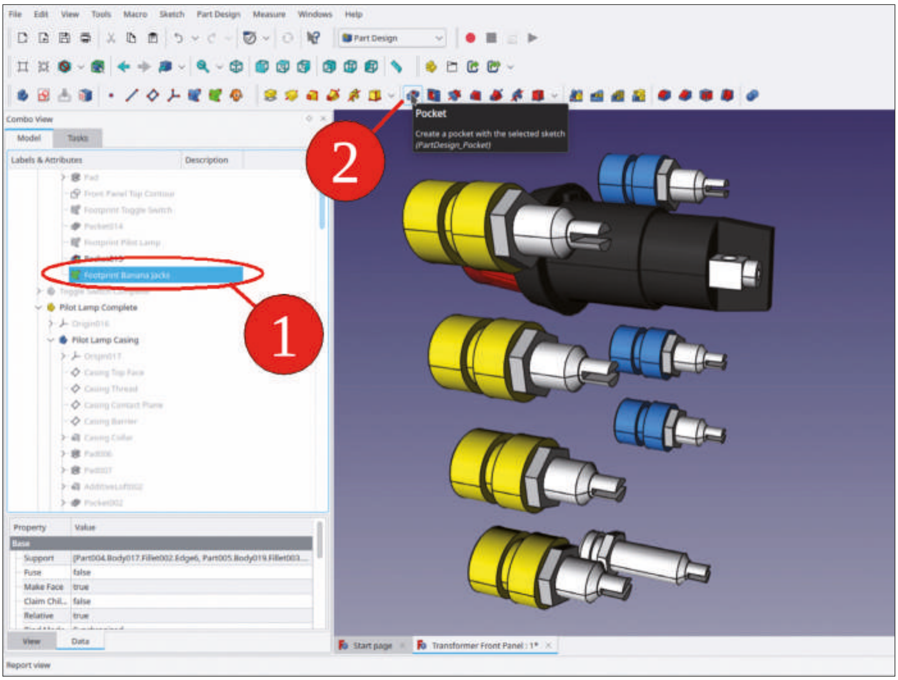


Figure 9-32

12. Hide all front panel elements and reveal the 'Front Panel Sheet' body to check all the created front panel cutouts (Figure 9-33).

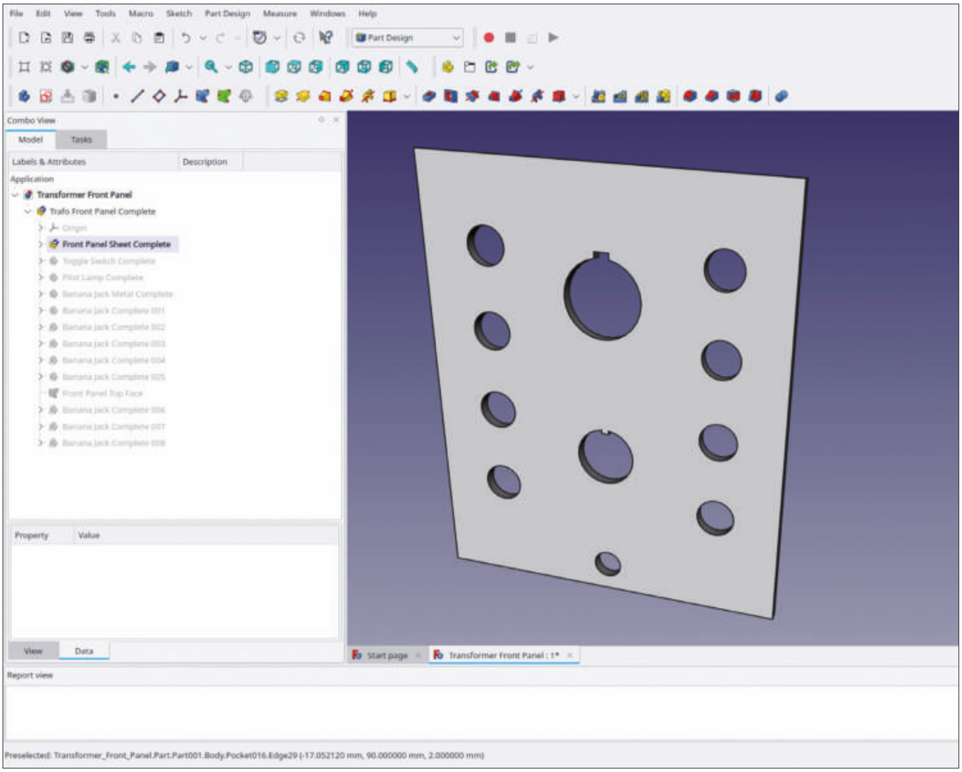


Figure 9-33

The mounting holes for the attachment of the front panel itself are not modeled yet because they will be defined later, in association with the spacers of the chassis.

9.1.5 Adding the Engraving

On the front panel, no guide lines are necessary, just engraved text. We will use the tool 'Shape of text' in the 'Draft' workspace, and the font 'RobotoCondensed light.ttf', which is freely available with Linux systems. A copy of it is contained in the 'Sample Projects | Lab Transformer | Components' directory, so you can find and select it easier.

1. In the 3D view, set the viewing direction to 'Top' (control cube or 'Top' tool button).

2. Switch to the 'Draft' workbench. Click on the 'Current Working plane' tool button. In the task window, set the 'Offset' parameter to 2.00 (your current panel thickness) and click the 'Top (XY)' button (Figure 9-34). The order of these steps matters!

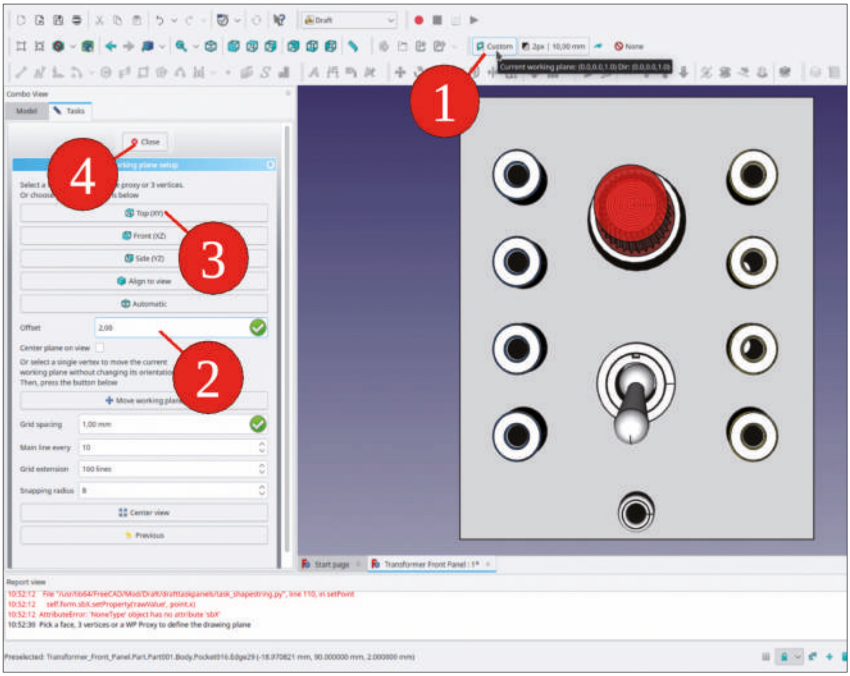


Figure 9-34

3. Click on the 'Shape from text' tool button. (Figure 9-35).

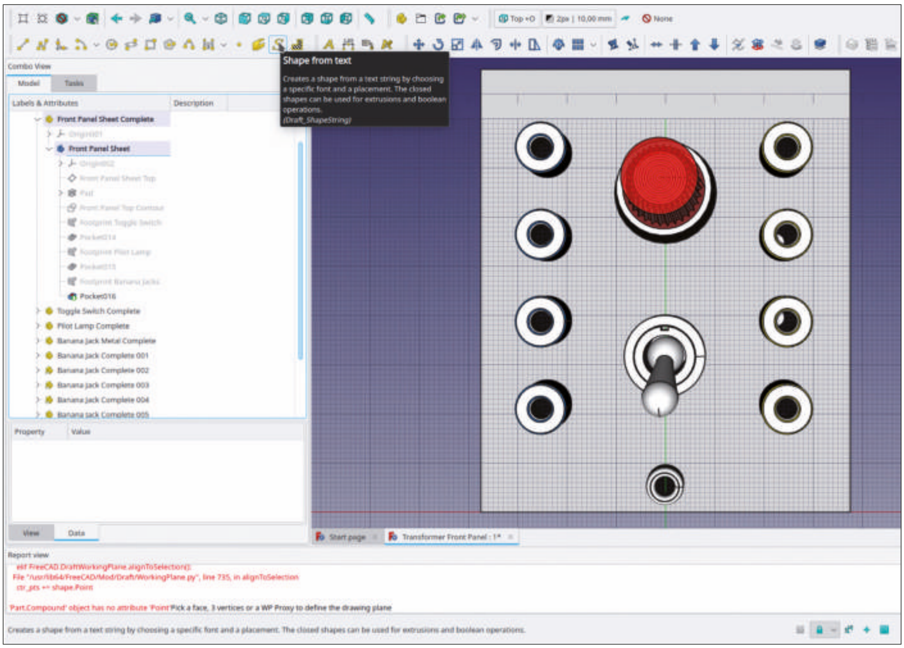


Figure 9-35

4. In the task window, enter 'Power' as the string, 6.5 mm for the height, and select 'RobotoCondensed-Light.ttf' for the font (from the 'Components' directory of the project, Figure 9-36). Click roughly on the panel where the text should be located. Then, close the task window with the 'OK' button. Only then, the text will be shown in the 3D view. In the tree view, rename the new 'ShapeString' object to 'ShapeString Power'.

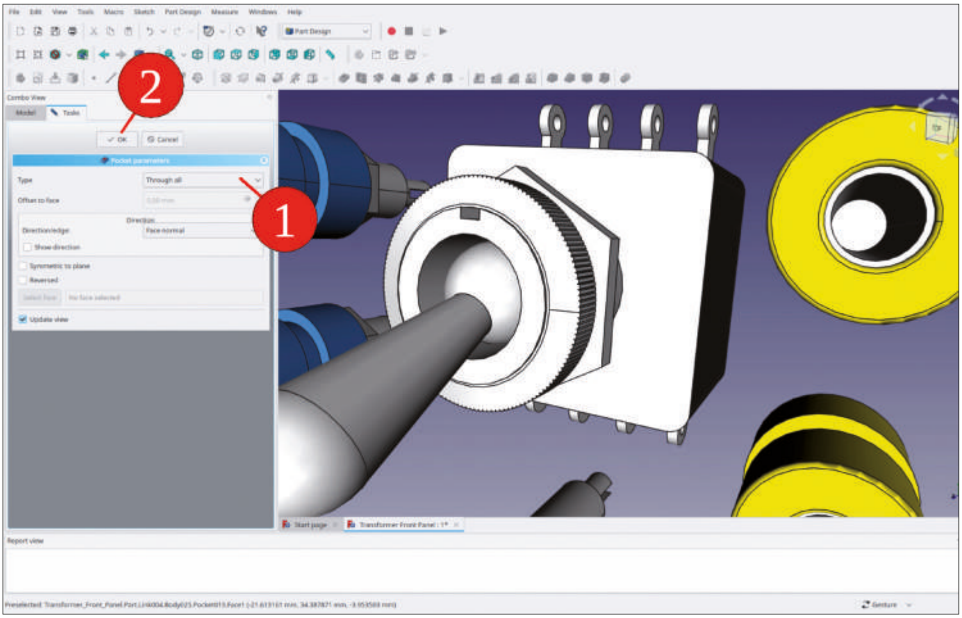


Figure 9-36

The position of the ShapeString objects can be tweaked later by editing their placement parameters. Doing so allows live updating of the view, which makes the process easier.

5. In the same fashion, label the top position of the toggle switch with 'ON', and place the designation 'GND' next to the lowest banana jack.

6. As shown in Figure 9-37, in each row, place a '10V' label between the lower two insulated banana jacks, between the center ones a label reading '2V'. Between the upper ones, place a label with '3V'.

7. The result of this labeling work is shown in Figure 3-37. Switch off the drafting grid with the 'Toggle draft grid' tool button (Figure 9-38). Save your work.

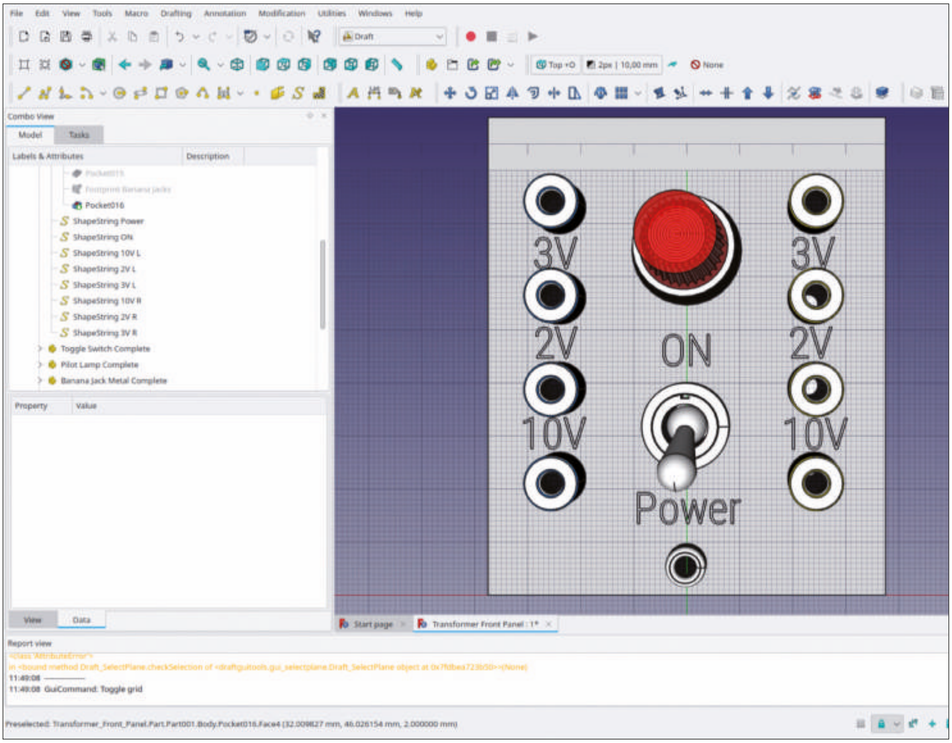


Figure 9-37

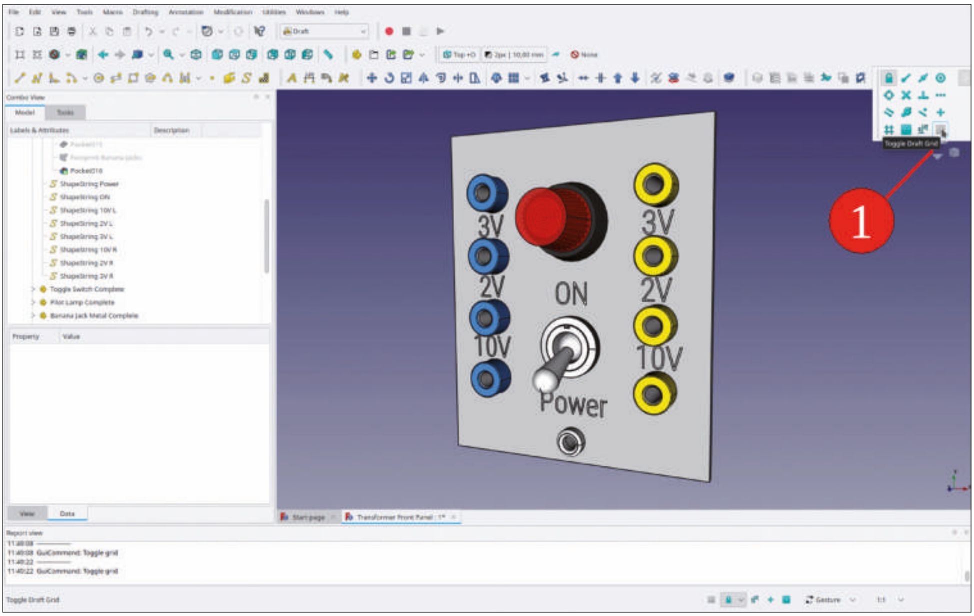


Figure 9-38

9.2 The Rear Panel

The AC power inlet and two fuse holders are to be placed onto the rear panel. Later, you are ready to place a screw terminal on the chassis, for the protective earth conductor.

9.2.1 Prepare File and Std-Part-Containers

1. Open the file 'Transformer Rear Panel.FCStd', which you saved as a blank rear panel in 9.1.2, step 16.

2. Change the following designations:
   - 'Trafo Front Panel Complete' to 'Trafo Rear Panel Complete',
   - 'Front Panel Sheet Complete' to 'Rear Panel Sheet Complete',
   - 'Front Panel Sheet' to 'Rear Panel Sheet',
   - 'Front Panel Sheet Top' to 'Rear Panel Sheet Top',
   - 'Front Panel Top Contour' to 'Rear Panel Top Contour',
   - 'Front Panel Top Face' to 'Rear Panel Top Face'

The result is shown in Figure 9-39.

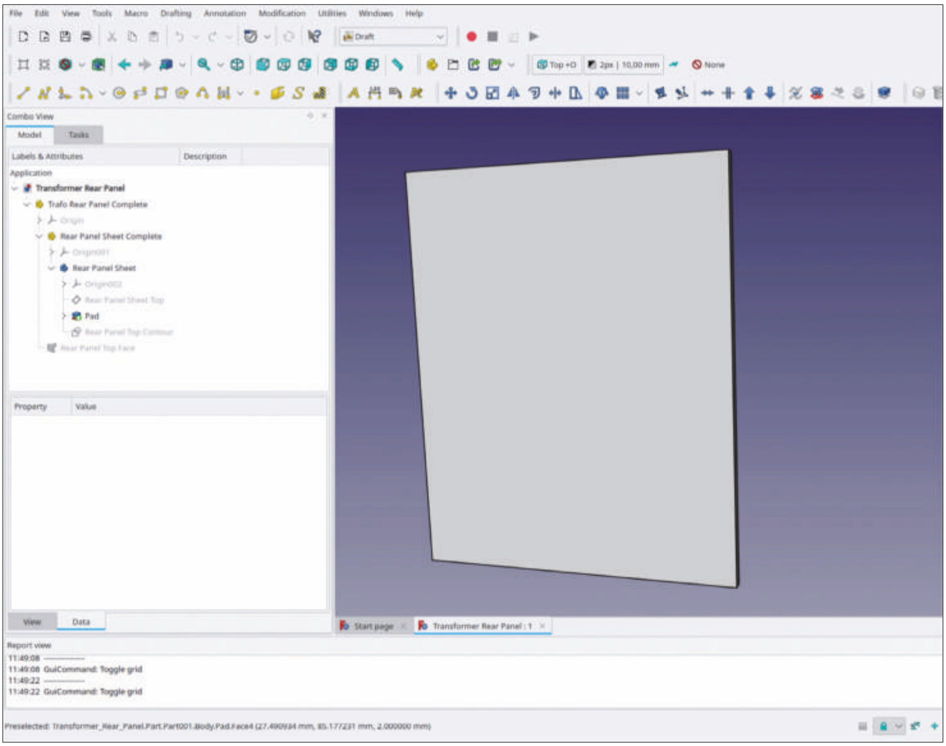


Figure 9-39

9.2.2 Placing the Components

1. Locate and open the file 'IEC Power Inlet' in the directory 'Sample Projects | Lab Transformer | Components'. Copy the Std-Part-Container 'IEC Inlet Complete' with CTRL-C. In the selection dialog, do not change the standard selection and click the 'OK' button to proceed. In the tree view, close the document node 'IEC Power Inlet'. Press CTRL-V to insert the Std-Part-Container into the document node 'Transformer Rear Panel'. Drag-and-drop the Container into 'Trafo Rear Panel Complete' (Figure 9-40).

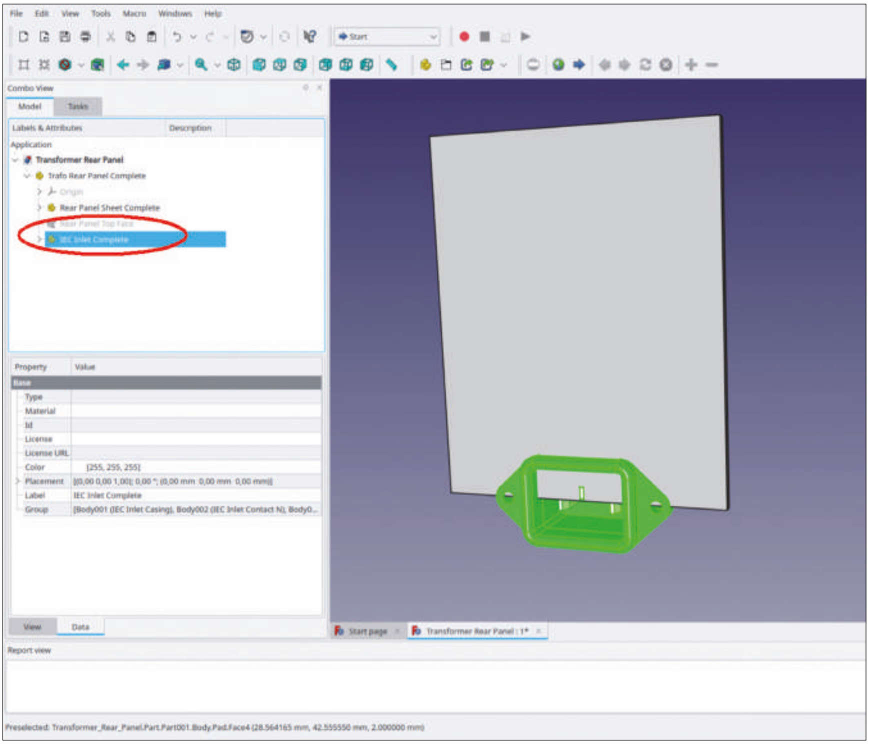


Figure 9-40

2. In the tree view, mark the Std-Part-Container 'IEC Inlet Complete' and switch to the 'Part' workbench. From the main menu, select 'Part | Attachment...'. When the task window opens, the collector for Reference 1 is already activated. Switch to the 'Model' tab, and, in the tree view, click the (hidden, grayed out) shape binder 'Rear Panel Top Face'. Return to the task window and select 'XY on plane' as the attachment mode. For the attachment offset Y, enter a value of 25 mm (Figure 9-41). Close the task window with the 'OK' button.

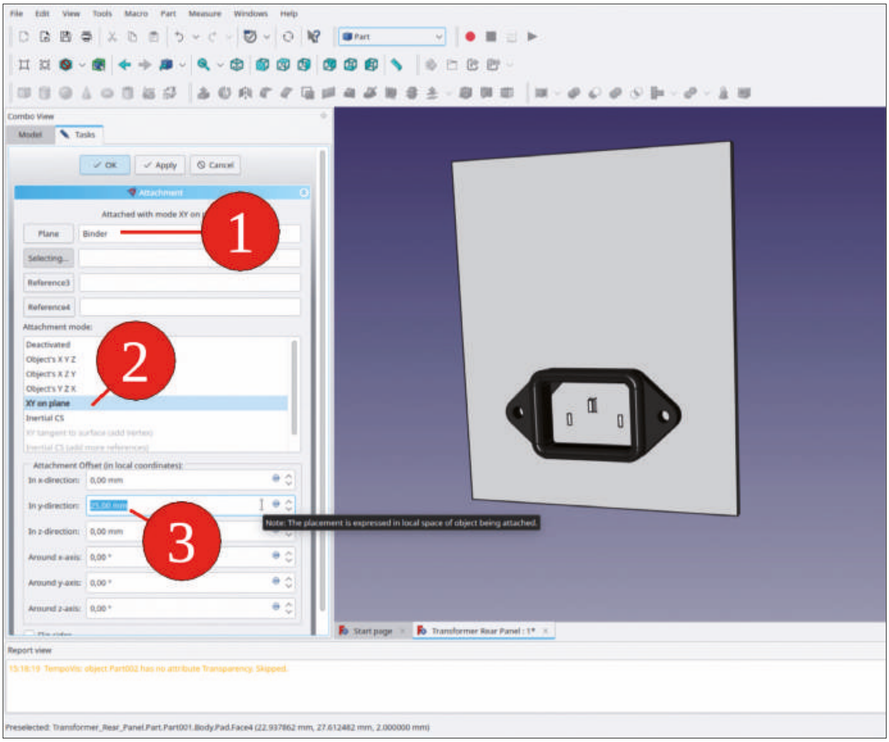


Figure 9-41

3. Following the same procedure as for the AC power inlet, open the file 'Fuse Holder.FCStd', and insert the Std-Part-Container 'Fuse Holder Complete' into the 'Trafo Rear Panel Complete' Std-Part-Container.

4. Rename the new container to 'Fuse Holder Complete 001'. Repeat the attachment procedure and attach the fuse holder to the SubShapeBinder 'Rear Panel Top Face'. Enter for the attachment offsets X = 15 mm and Y = 58 mm (Figure 9-42). Close the task window with the 'OK' button.

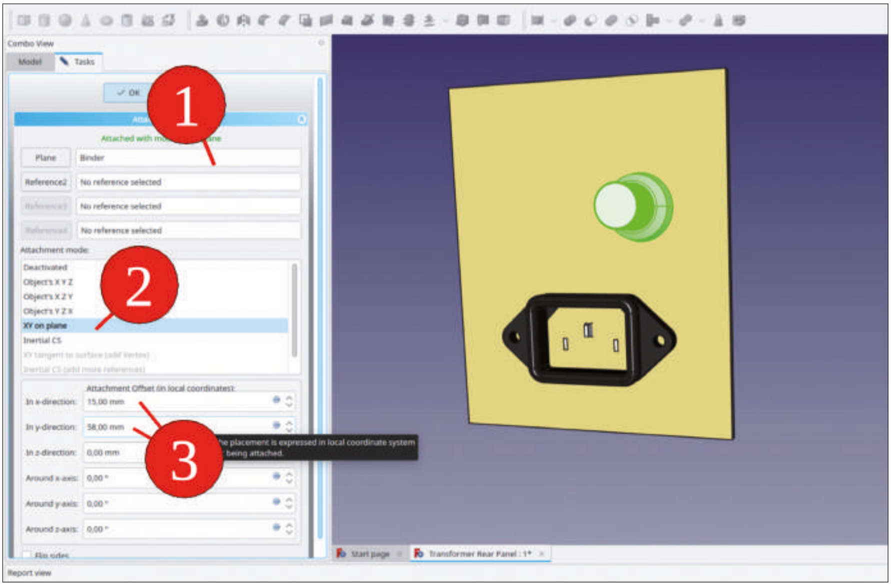


Figure 9-42

5. In the tree view, mark 'Fuse Holder Complete 001'. Then, click the 'Make link' tool button (Figure 9-43). Drag-and-drop the new 'Fuse Holder Complete 002' into the Std-Part-Container 'Trafo Rear Panel Complete'.

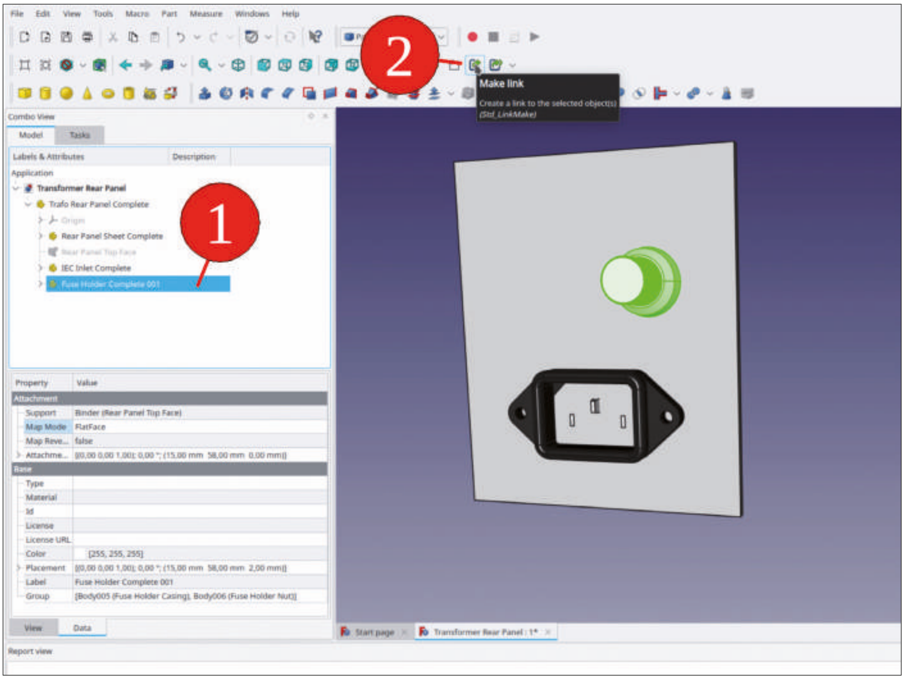


Figure 9-43

6. In the tree view, mark 'Fuse Holder Complete 002'. In the property list, set the property 'Link transform' to 'true', in order to allow placement relative to the parent object.

7. Edit the 'Link placement' parameters and set the translation to -30 mm (Figure 9-44).

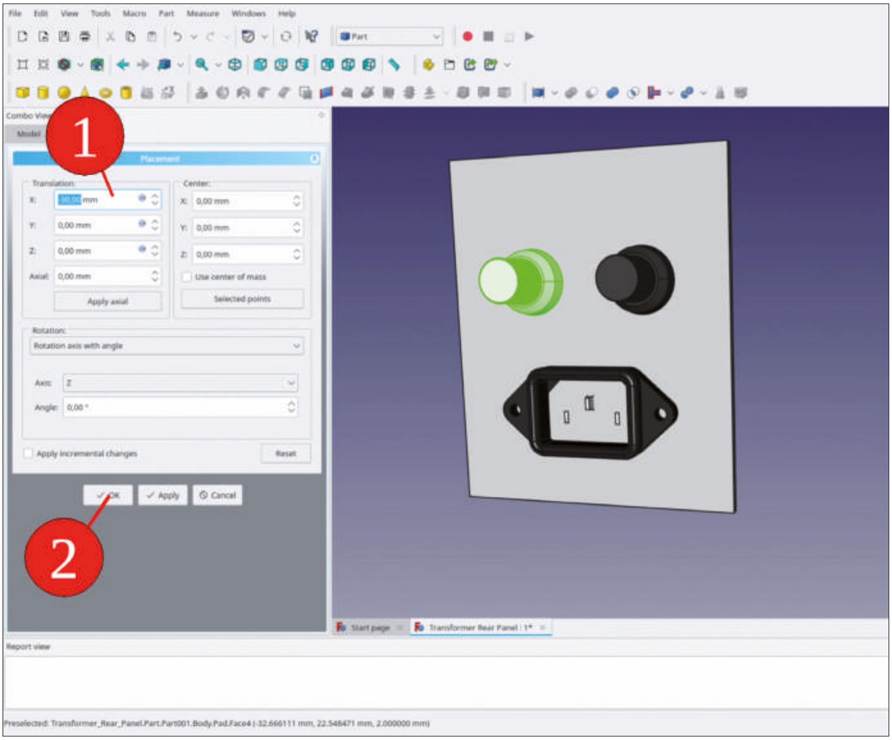


Figure 9-44

9.2.3 Punching the Holes

1. In the Std-Part-Container 'Rear Panel Complete', activate the body 'Rear Panel Sheet' with a double click. Besides activating the body, FreeCAD also switches automatically into the 'Part Design' workbench, if it was not previously selected.

2. In the tree view, hide the body 'Rear Panel Complete' to expose the edges of the other assembly elements. Rotate the view, until the rear details of the elements are visible.

3. Hold down the CTRL key and select from the 3D view all edges of the IEC power inlet, which determine the cutout and screw holes in the panel (Figure 9-45). Then, click the green 'Create a sub object(s) shape binder' tool button.

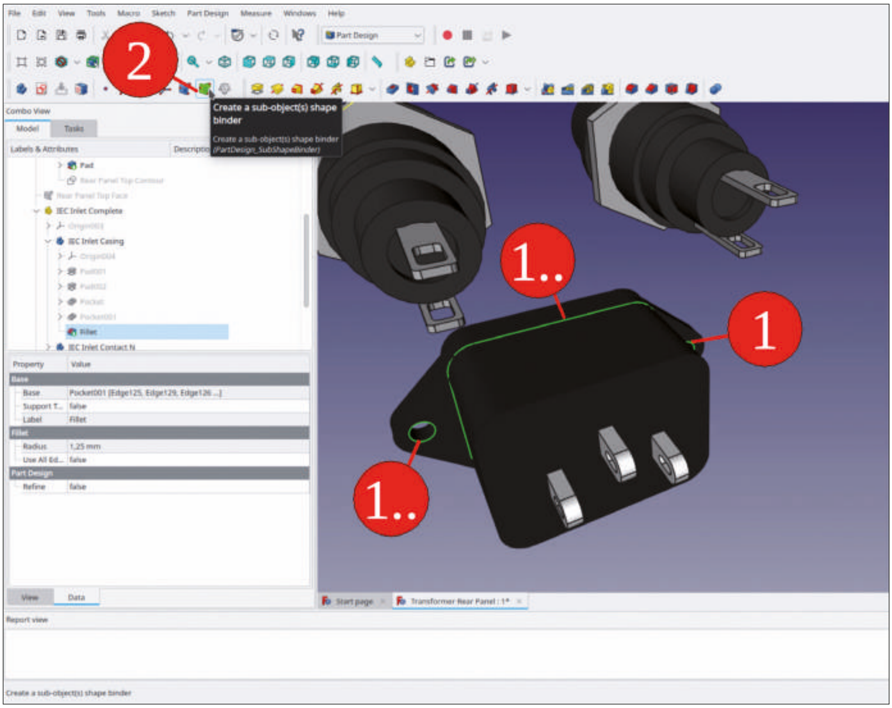


Figure 9-45

4. Rename the new SubShapeBinder in 'Rear Panel Sheet' to 'Footprint IEC Inlet'.

5. With the new SubShapeBinder marked in the 3D view, click the 'Pocket' tool button. In the task window, select 'Through all' for the type and click the 'OK' button.

6. In the tree view, expand 'Fuse Holder Complete 001' and hide the 'Fuse Holder Nut'. Because the second fuse holder is a linked object, it will follow on its own.

7. In the 3D view, hold down the CTRL key and select all edges on the two fuse holders which define the cutout in the panel (Figure 9-46). Then, click the green 'Create a sub object(s) shape binder' tool button.

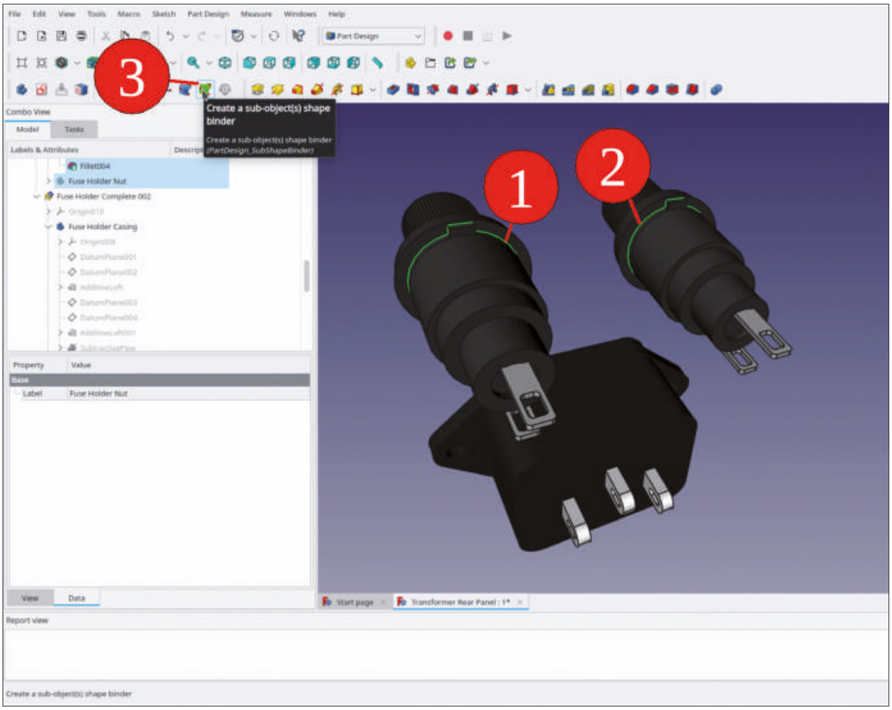


Figure 9-46

8. In the body 'Rear Panel Sheet', rename the new SubShapeBinder to 'Footprint Fuse Holders'.

9. With 'Footprint Fuse Holders' marked in the tree view, click the 'Pocket' tool button. In the task window, select for the type 'Through all' and click the 'OK' button.

10. To check whether all the holes are properly inserted, display the 'Rear Panel Sheet' body again with the SPACE key (Figure 9-47). The rotation locks of the fuse holders and the bolt holes for the IEC inlet are visible. For the next steps, display all previously hidden items again.

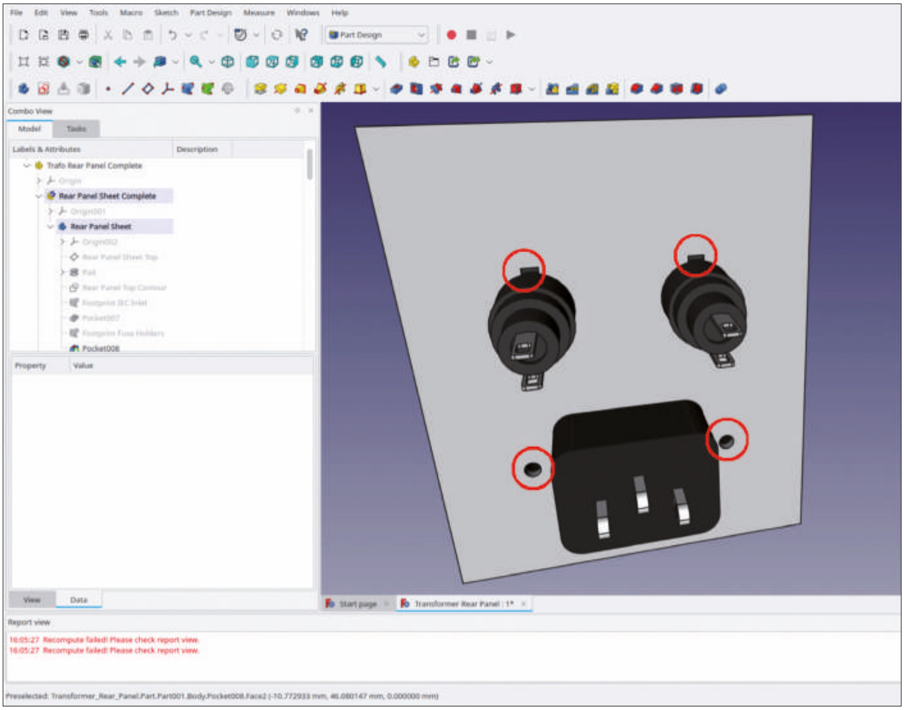


Figure 9-47

9.2.4 Adding the Engraving

The method is the same that you have used for the front panel. You can run through all the preparation steps as described in section 9.1.5, steps 1 and 2. On the rear panel, information about the mains voltage, the frequency range and the power intake are useful. You may also want to think about a unique designation of the apparatus, the manufacturer, and the IP class. For the example, however, let's keep things easy:

1. In the Draft workbench, create a shape string object with the text: '230V/50Hz/30VA' (this is the specification for the example transformer). The text can be placed below the IEC power inlet.

2. The second text could describe the strength and the timing characteristics of the fuses. This depends on the transformer, of course. For the example transformer used here, enter '2 × 0.5 AT' (= 0.5 amp, slow blow). This text could be placed centered below the fuse holders.

The result is shown in Figure 9-48. If you want, you could add fasteners to the IEC inlet. Then, the file can be saved and closed for later use.

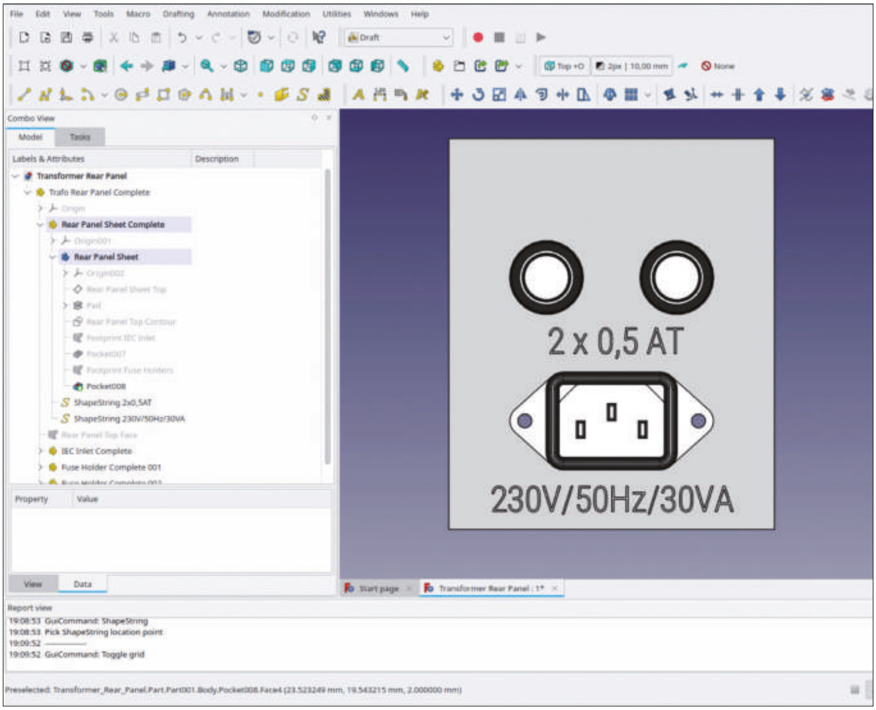


Figure 9-48

9.3 The Chassis

The chassis could play an important role when the volume of the apparatus is optimized later on, for example, by a length variation. Eventually, it would be useful to be able to move the front panel and rear panel of the assembly independently of each other. Therefore, meaningful references are needed right from the beginning when the chassis is designed.

9.3.1 Preparation of File and Std-Part-Container

1. Start a new file and save it as 'Transformer Chassis'.

2. Create a new Std-Part-Container and rename it to 'Trafo Chassis Complete'.

3. Start a new body object and rename it to 'Chassis Sheet'. In order to do this, switch to the 'Part Design' workbench (where the blue 'Create body' tool button is available).

4. To control the position of the front panel, create a datum plane: Show the coordinate system of 'Chassis Sheet' with the SPACE bar. In the 3D view, mark the XZ plane. Then, click the 'Create a datum plane' tool button. In the task window, the attachment mode 'plane face' is already preset. Enter an attachment offset Z value of 65 mm (Figure 9-46). Note: A Z offset designates here a shift parallel to the normal vector of the plane, not the Z coordinate direction. Close the task window with the 'OK' button.

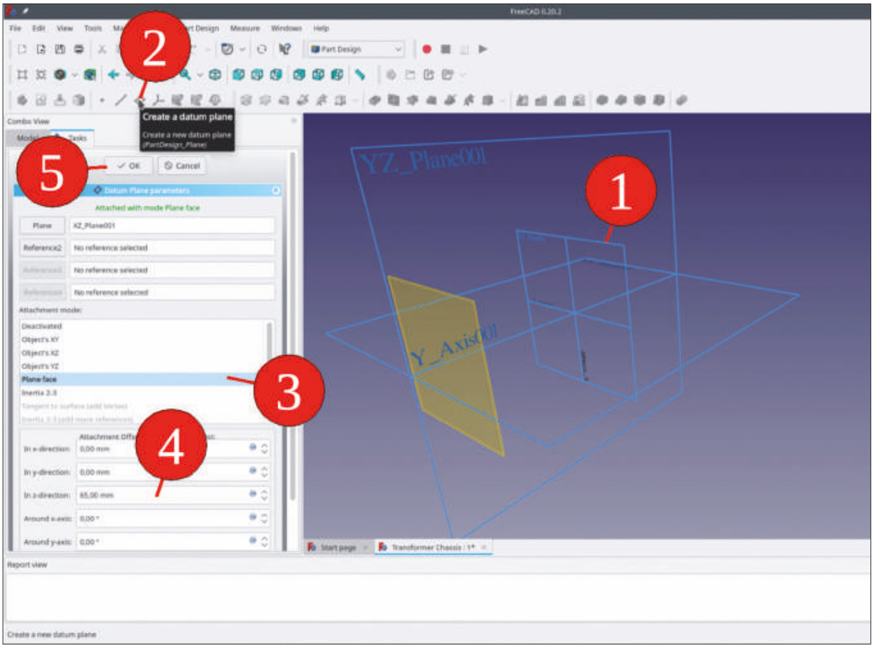


Figure 9-49

5. Rename the new datum plane to 'Position Front Panel'.

6. In the same way, create a datum plane and set the attachment Z offset to -65 mm. Rename that datum plane to 'Position Rear Panel'.

7. Now you will sketch the cross-section of the chassis sheet onto the plane 'Position Front Panel': In the tree view, mark this datum plane and start the sketcher.

8. From the main menu, select 'View | Orthographic view' and 'Sketch | View section'.

9. Sketch a rectangle that is attached to the X-axis (Figure 9-50).

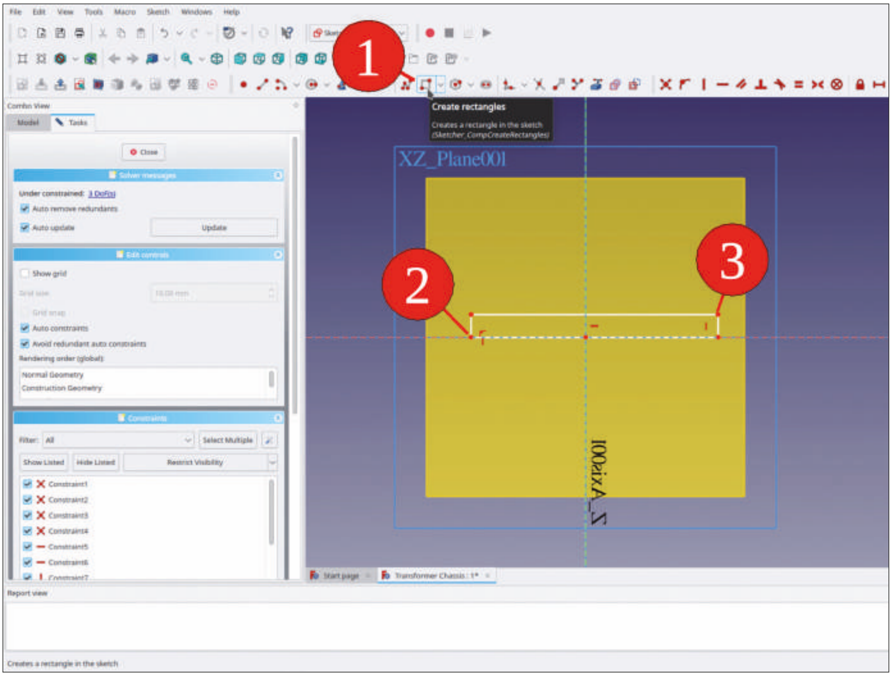


Figure 9-50

10. Mark the lower two corners of the rectangle, and then the Z (= vertical) axis. Click on the 'Constrain symmetrical' tool button (Figure 9-51).

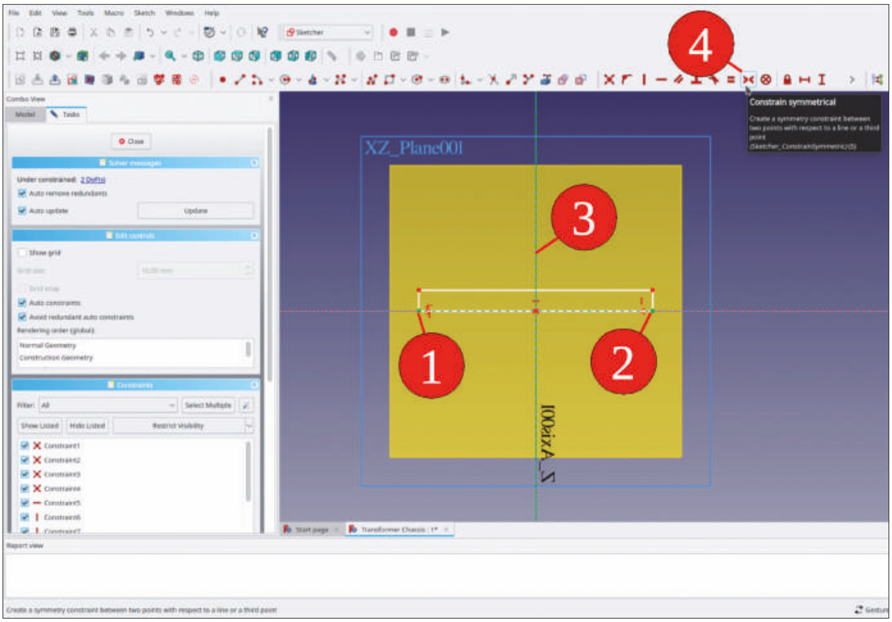


Figure 9-51

11. Set the height of the rectangle: Mark one of the vertical sides of the rectangle and click the 'Constrain vertical distance' tool button. Enter a value of 3 mm for the thickness of the sheet (Figure 9-52).

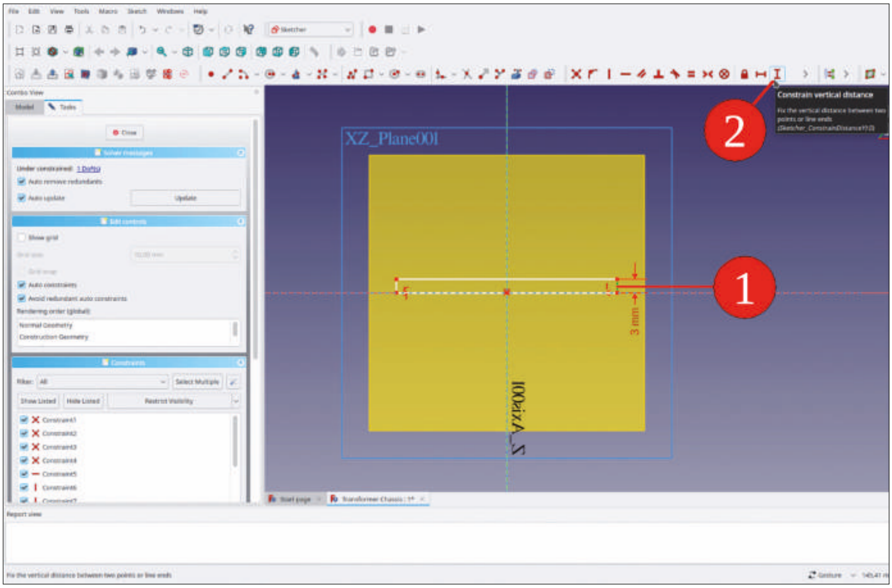


Figure 9-52

12. In a similar way, constrain the width of the rectangle to 75 mm (Figure 9-53). Close the sketch.

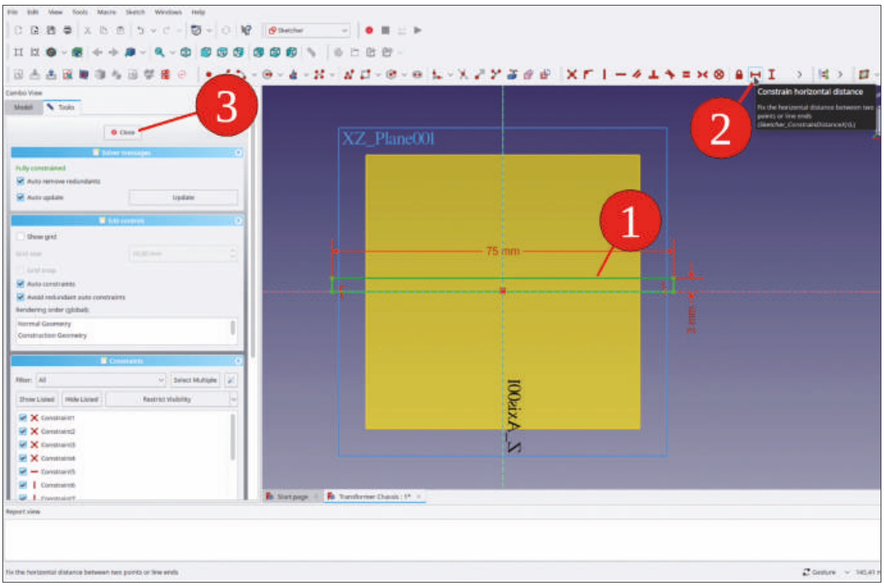


Figure 9-53

13. In the tree view, mark the new sketch and click the 'Pad' tool button.

14. In the task window, select for the type 'Up to face'. Select the datum plane 'Position Rear Panel' as the face, either by clicking on it in the 3D view, or by switching to the 'Model' tab and clicking on its tree branch. (Figure 9-54). Close the task with the 'OK' button.

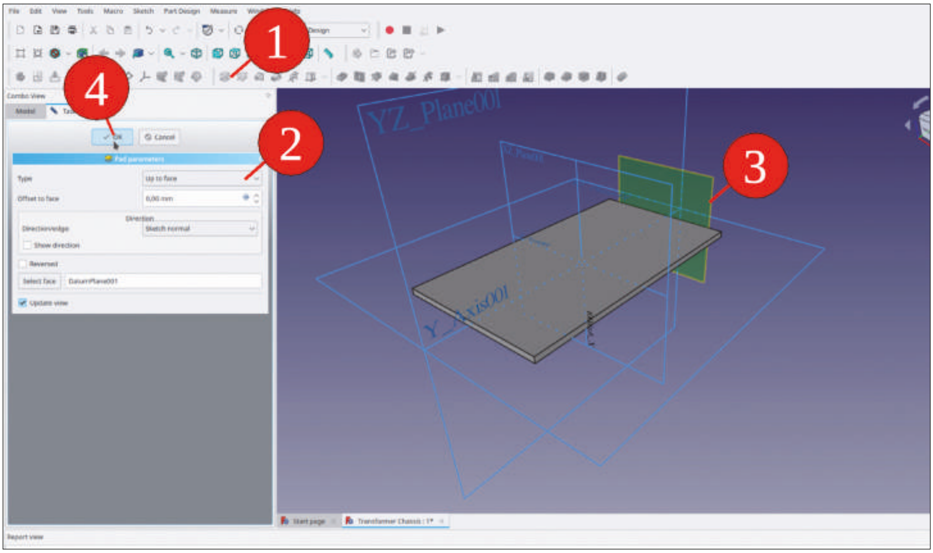


Figure 9-54

The chassis is just a sheet metal piece now. To form the connection with the rest of the assembly, square aluminum spacers can be added. We need to draw only the first one of them, the others can be added as link objects. In order to associate the length of the chassis sheet with the length of the spacers, you need two SubShapeBinders which reference the two datum planes.

15. In the tree view, hide the body 'Chassis Sheet'.

16. Create a new body and rename it to 'Chassis Spacer 001'.

17. Drag-and-drop the new body into 'Trafo Chassis Complete'.

18. The body 'Chassis Spacer 001' should still be activated (title shown in bold letters). If not, double-click it in the tree view.

19. Click on the blue 'Create a shape binder' tool button. In the task window, click the button 'Object' until this turns dark gray. Then, click into the edit field next to the 'Object' button, which receives the focus (gets a blue rim). Switch to the 'Model' tab and select the datum plane 'Position Front Panel'. Return to the 'Tasks' tab and close the task window with the 'OK' button (Figure 9-55).

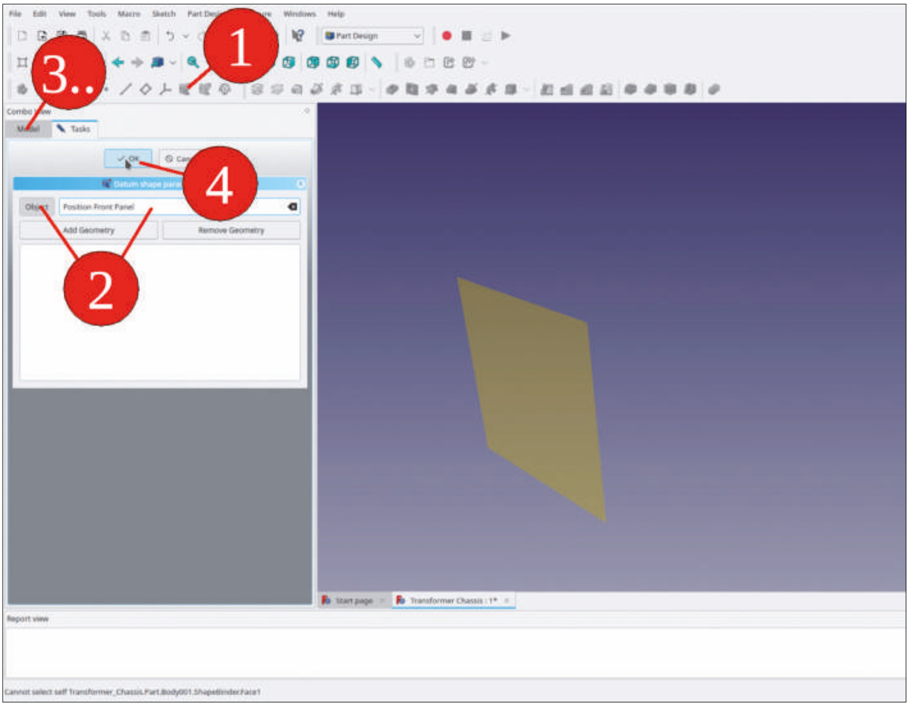


Figure 9-55

20. Rename the new shape binder to 'Front Side'. In the same way, create a shape binder for the rear datum plane and rename it to 'Rear Side' (Figure 9-56).

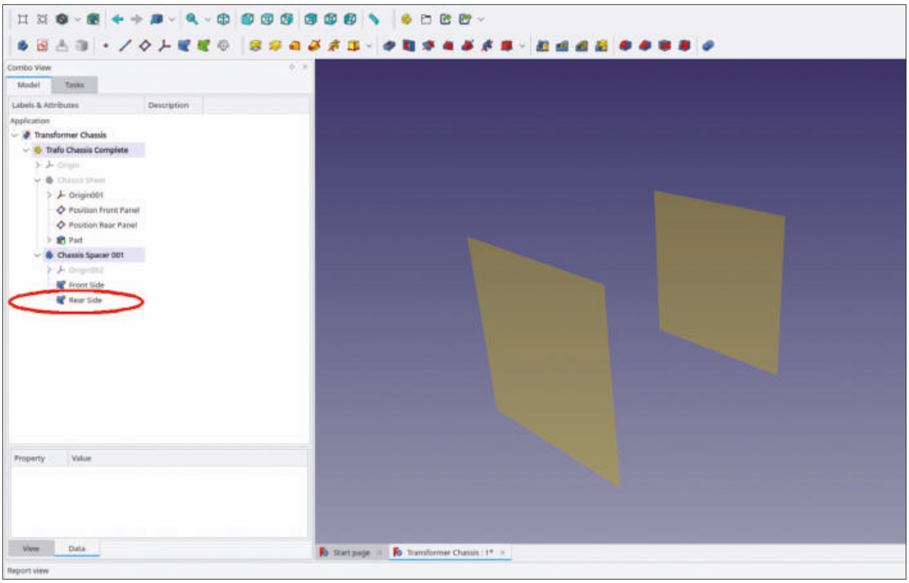


Figure 9-56

21. Mark both shape binders. In the property list, set the property 'Trace Support' to 'true' (Figure 9-57).

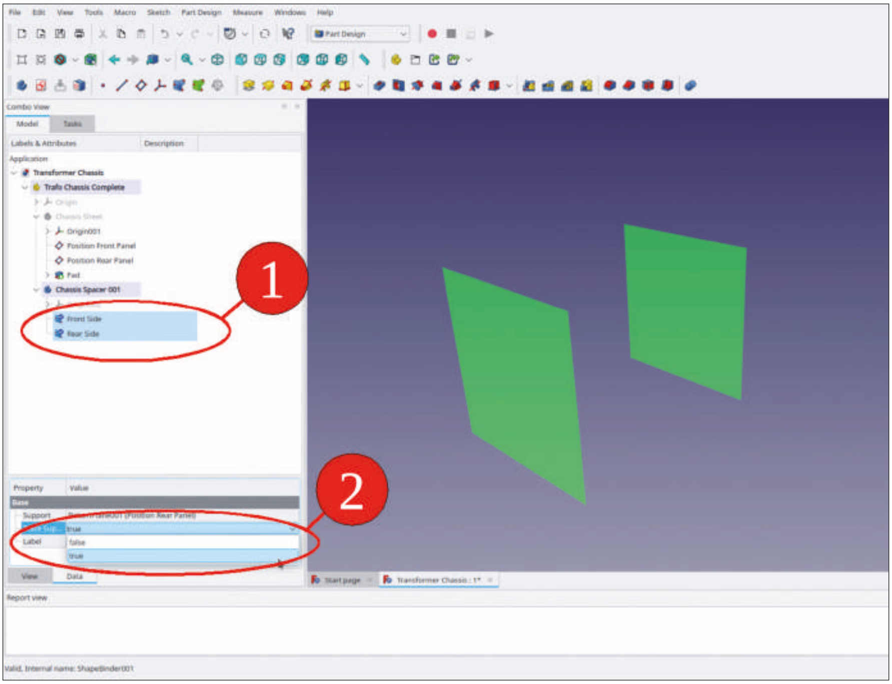


Figure 9-57

It is possible to sketch onto these shape binders. However, it turns out, that the placement of the spacer cannot be adjusted any more, as the shape binders track the origin of the referenced datum plane. As a remedy, two local datum planes can be defined, which reference the corresponding shape binders. The placement of the local datum planes can then be used to define an offset to the shape binders. This will get more obvious when you proceed:

22. Click on the 'Create a datum plane' tool button. In the task window, the collector for Reference 1 is already activated when the task window opens. Switch to the 'Model' tab and select the datum plane 'Front Side'.

23. Return to the task window. The attachment mode 'Plane face' is already preset. Close the task window with the 'OK' button.

24. Rename the new datum plane to 'Spacer Front Face'.

25. In the same way, create another datum plane for the shape binder 'Rear Side' and rename it to 'Spacer Rear Face' (Figure 9-58). In the tree view, hide the two shape binders 'Front Side' and 'Rear Side'.

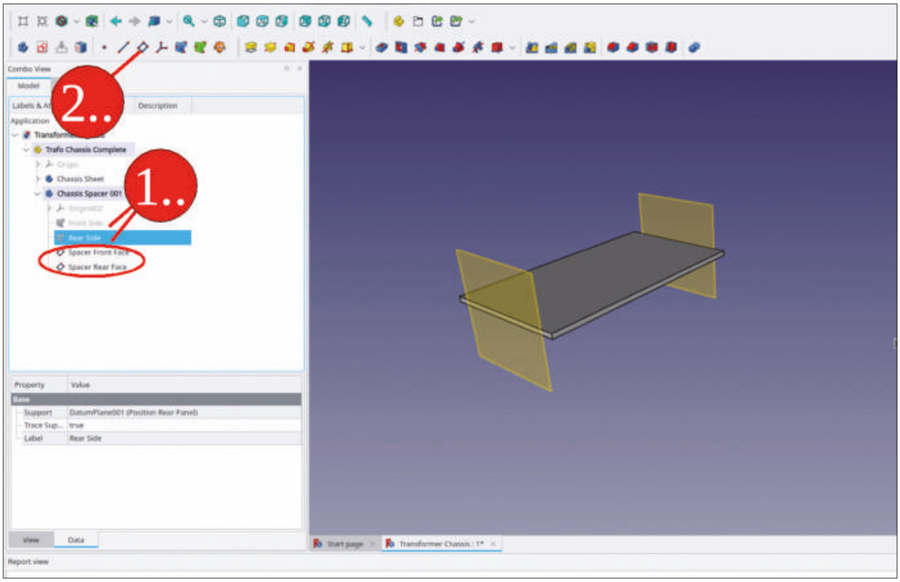


Figure 9-58

26. In the 3D view, mark the datum plane 'Spacer Front Face' and start the sketcher. From the main menu, select 'Sketch | View section' and 'View | Orthographic view'.

27. Draw a rectangle that is centered around the origin. Mark one horizontal and one vertical side of the rectangle and click the 'Constrain equal' tool button (Figure 9-59).

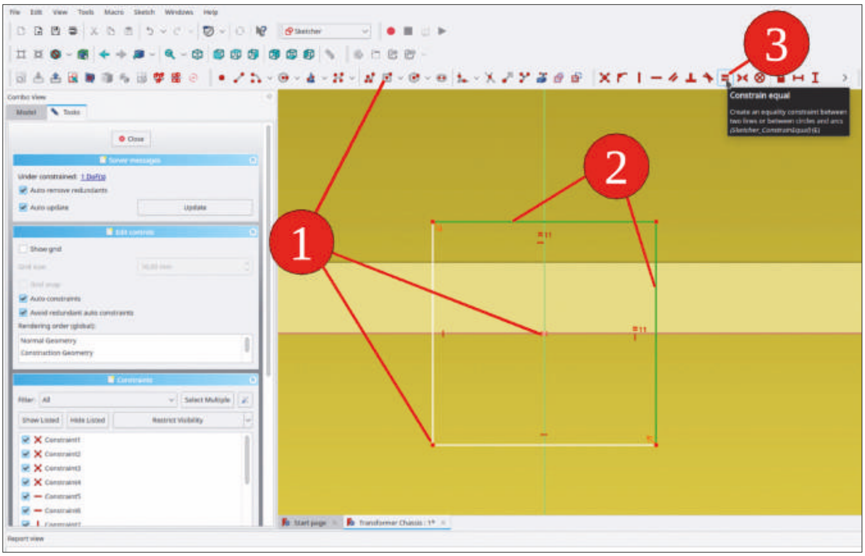


Figure 9-59

28. Click on one horizontal side of the rectangle. Click on the 'Constrain horizontal distance' tool button. Set the length to 10 mm (Figure 9-60). Close the sketcher.

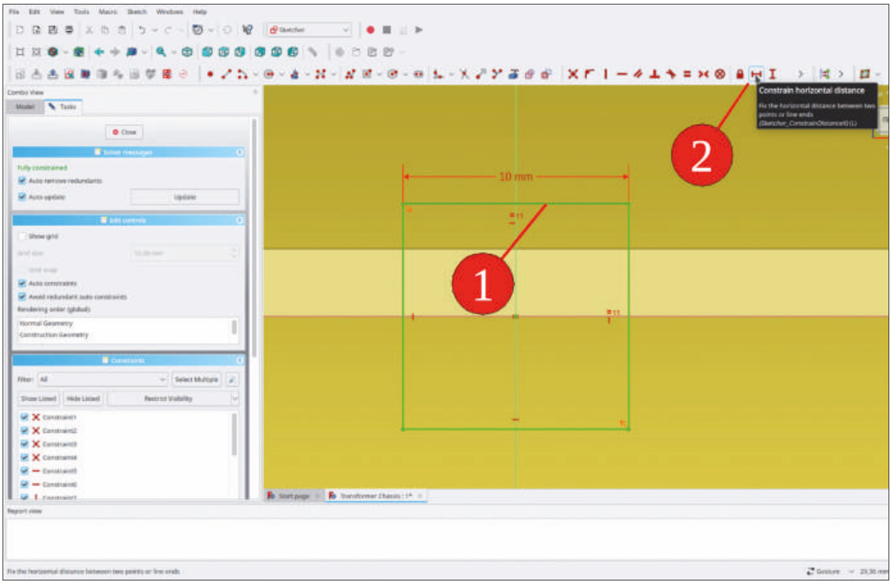


Figure 9-60

29. In the tree view, mark the new sketch and click the 'Pad' tool button.

30. In the task window, select for the type 'Up to face'. In the 3D view, click the rear datum plane representation (Figure 9-61).

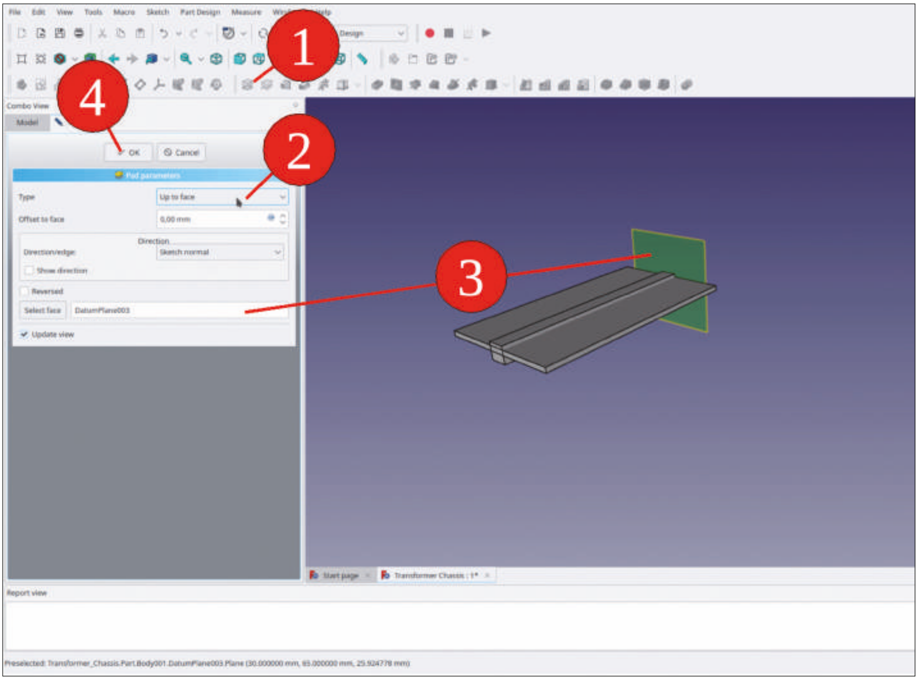


Figure 9-61

31. Close the task window with the OK button. Hide the datum planes with the SPACE key.

The spacer is now associated with the reference planes 'Position Front Panel' and 'Position Rear Panel' in the body 'Chassis Sheet'. This you can test later on, by changing the position of these reference planes. Please note that the changes will only take visible effect when a recalculation is executed (by pressing the F5 key, with the objects marked for recalculation). The holes for the fasteners are still missing. For these, you can create sketches on the datum planes – with the advantage that the holes will thus not be affected by a re-enumeration of the facets.

32. In the 3D view, mark the datum plane 'Spacer Front Face' and start the sketcher. Draw a circle that is centered around the origin and constrain its diameter to 3 mm (the diameter is not important, see next step). Close the sketch.

33. In the tree view, mark the new sketch and click the 'Hole' tool button (Figure 9-62).

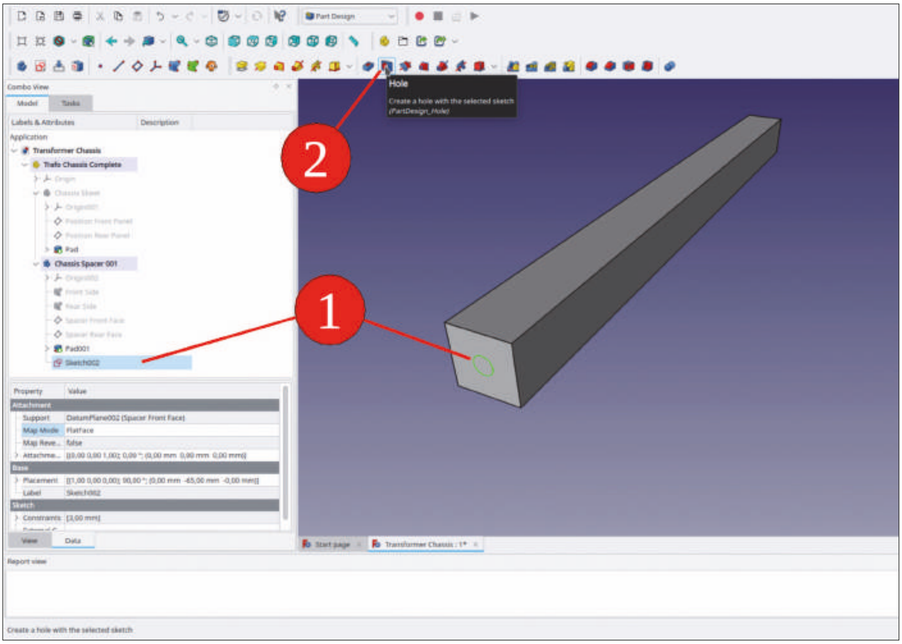


Figure 9-62

34. In the task window, select the profile 'ISO metric regular' and set the size to M4. Check the 'Threaded' checkbox and set the depth of the hole to 15 mm (Figure 9-63). Close the task window with the 'OK' button.

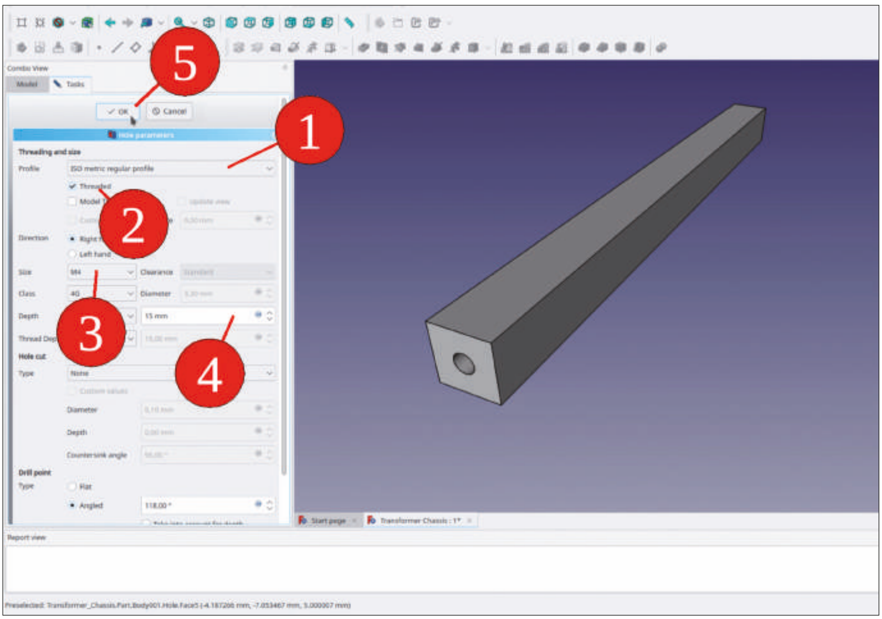


Figure 9-63

35. In the same way, create a hole in the rear face of the spacer. Because it is drilled from the back side, the checkbox 'Reversed' needs to be checked in the task window of the 'Hole' command.

Now, the placement of the spacer can be adjusted. To view the spacer together with the chassis sheet, display the body 'Chassis Sheet' again with the SPACE key (Figure 9-64). The placement of the spacer cannot be done with its 'Placement' parameters – because of the shape binder reference, adjusted offsets are reset when a recalculation is taking place. The placement is achieved via the attachment parameters of the datum plane 'Spacer Front Face' instead, where the sketch of the spacer profile is located.

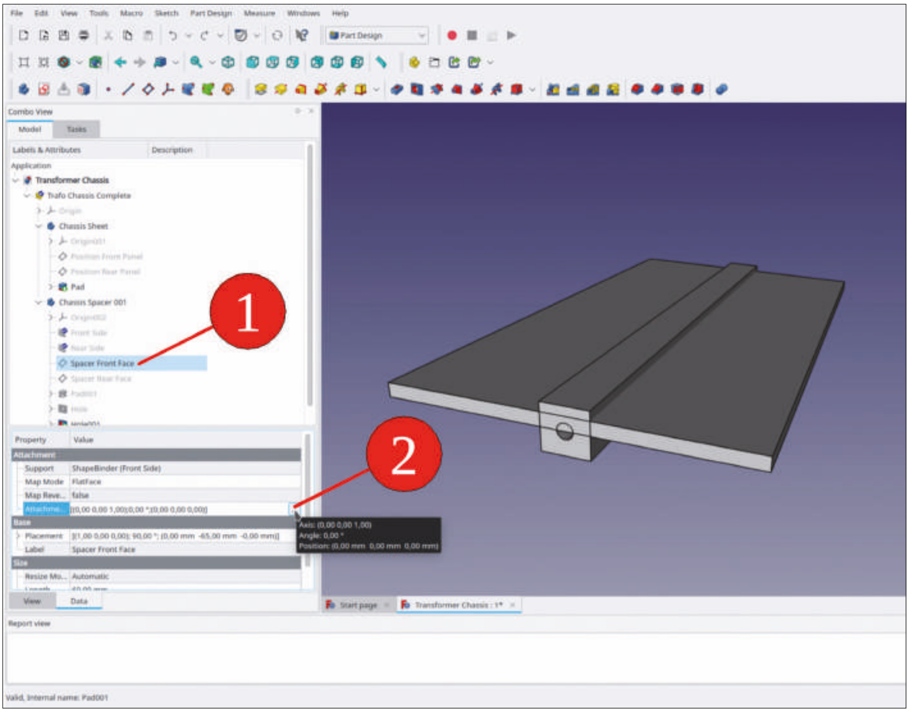


Figure 9-64

36. In the tree view, mark the datum plane 'Spacer Front Face'. In the property list, edit the 'Attachment' parameters. Open the task window with the ... button in the edit field. Set the translation to X = 32.5 mm and Y = -5 mm (Figure 9-65). The view will be updated only if the 'Apply' or the 'OK' buttons have been clicked. Close the task window with the 'OK' button.

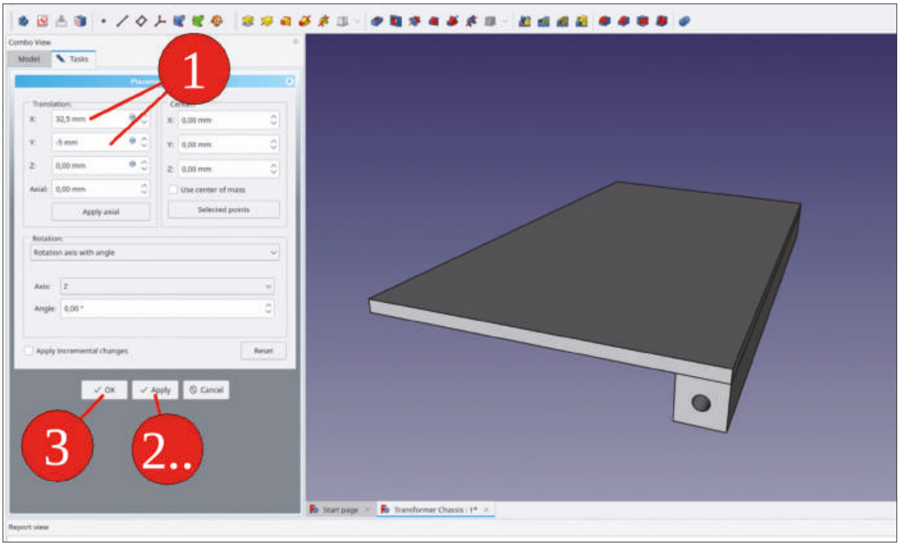


Figure 9-65

37. The offset has to be inserted also for the attachment of the 'Spacer Rear Face' datum plane, in order to not lose the hole in the rear face.

38. Using the 'Create link' tool button, create three links of 'Spacer 001'. Drag-and-drop them into 'Trafo Chassis Complete' (Figure 9-66).

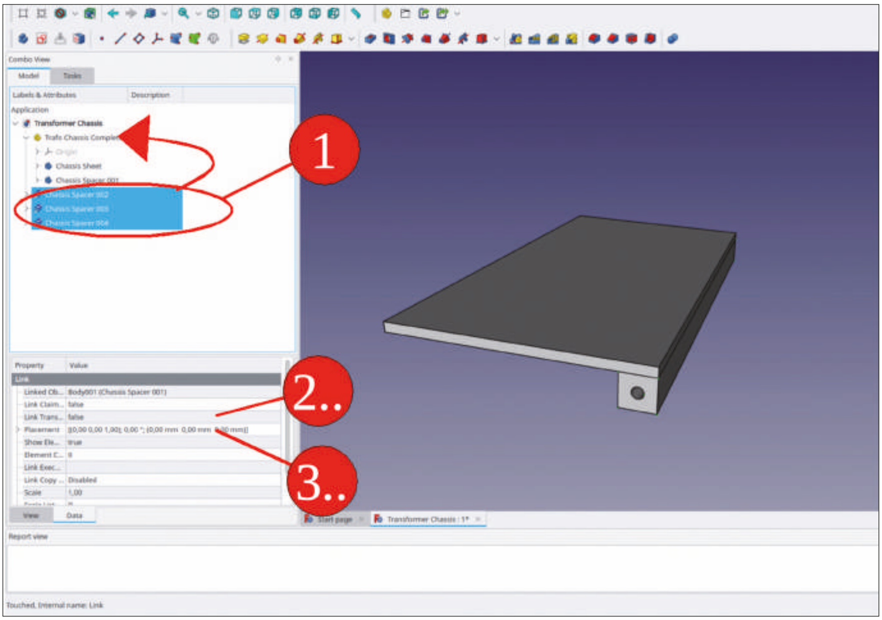


Figure 9-66

39. Mark the three new spacer bodies. In the property list, set the property 'Link transform' to 'true'.

40. Set the 'Link Placement' parameters of the three new spacers according to the table below (the result is shown in Figure 9-67):

Object	X [mm]	Y [mm]
Chassis Spacer 002	–65	0
Chassis Spacer 003	0	80
Chassis Spacer 004	–65	80

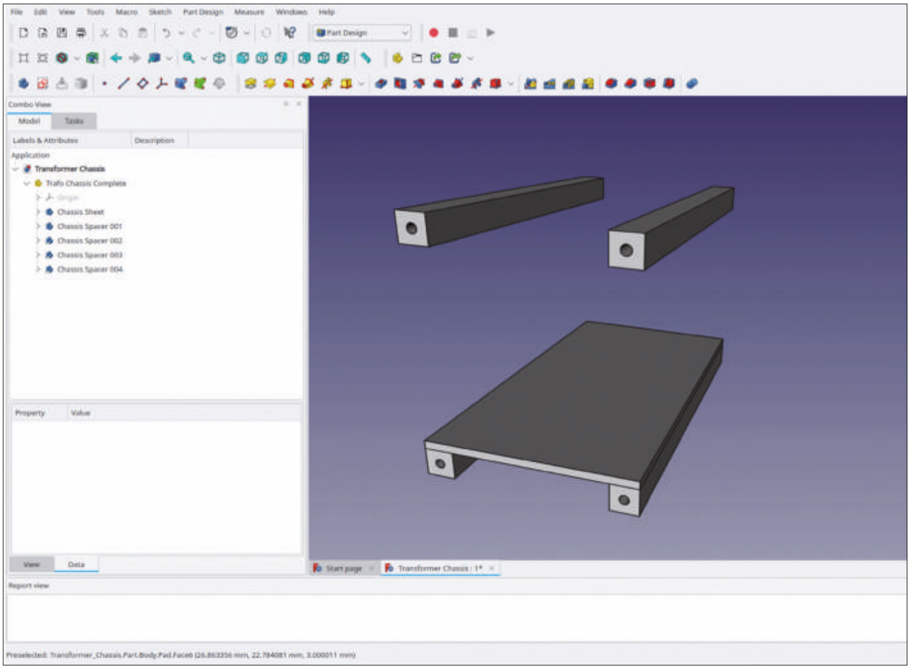


Figure 9-67

9.3.2 Bolts for the Chassis Sheet

The chassis sheet still floats because the bolts, which attach it to the spacers, are still missing. We can sketch the holes for the bolts on the XY plane of the chassis sheet and communicate their position to 'Chassis Spacer 001' by shape binders. The other linked spacers will follow this operation on their own:

1. To have an unobstructed view, in the tree view, hide 'Chassis Spacer 003' and 'Chassis Spacer 004' with the SPACE key.

2. In the tree view, activate the body 'Chassis Sheet' by a double click.

3. Expand the coordinate system of 'Chassis Sheet' and mark the XY plane. Then, start the sketcher. From the main menu, select 'Sketch | View section' and 'View | Orthographic view'.

4. Click on the 'External geometry' tool button. Click on the top and bottom horizontal edges of the chassis (Figure 9-68).

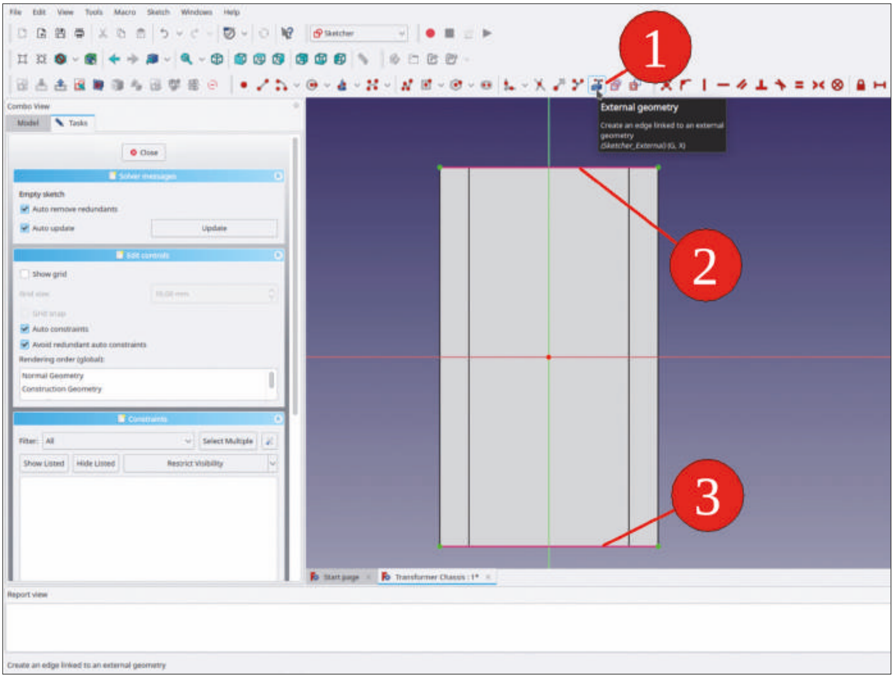


Figure 9-68

5. Draw a rectangle. Mark two corners that are on equal height. Mark the Y-axis and then, click the 'Constrain symmetrical' tool button (Figure 9-69).

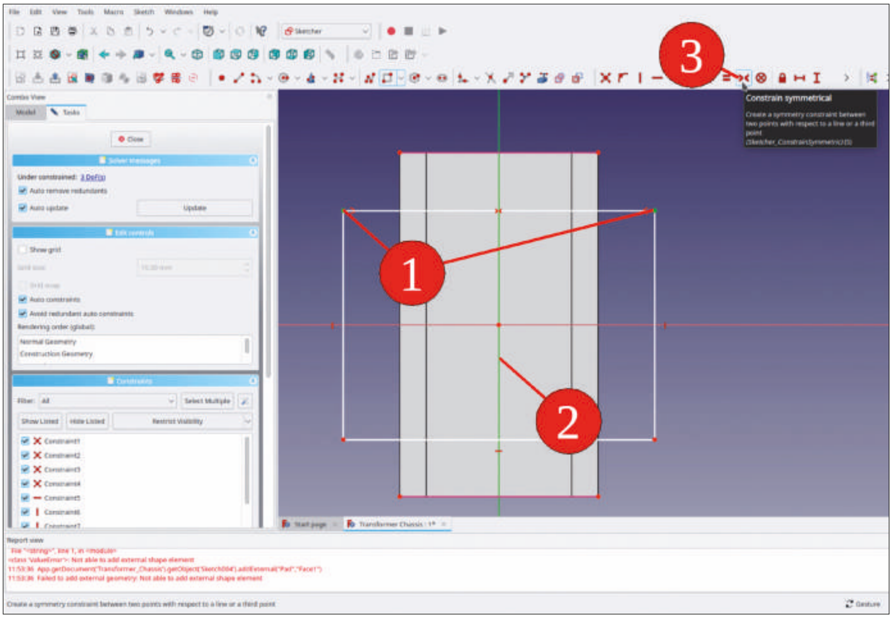


Figure 9-69

6. Mark two upper, corresponding corners of the rectangle and of the construction geometry and constrain the horizontal distance to -5 mm (this will bring the hole to the center of the spacers).

7. Again, mark the corresponding upper corners and constrain the vertical distance to 25 mm. Repeat the vertical constraint for two corresponding lower corners (Figure 9-70)

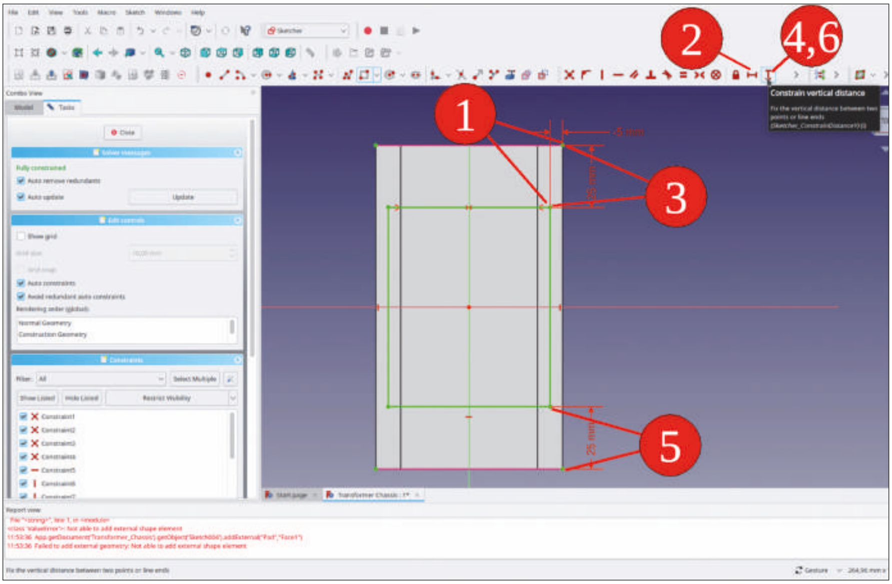


Figure 9-70

8. In the 'Elements' list, mark the 4 lines of the rectangle. Right-click the selection and select 'Toggle construction line' from the context menu (Figure 9-71). This turns the rectangle into a useful construction geometry which helps to keep the holes in place, even when front panel and rear panel are moved by unequal amounts.

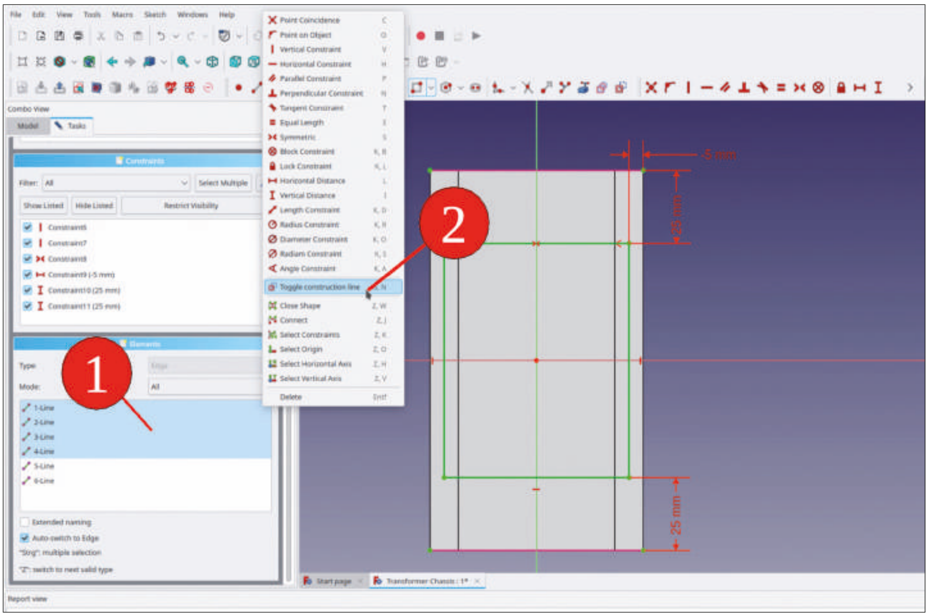


Figure 9-71

9. Draw 4 circles onto the corners of the rectangle. Mark the 4 circles and click the 'Constrain equal' tool button (Figure 9-72).

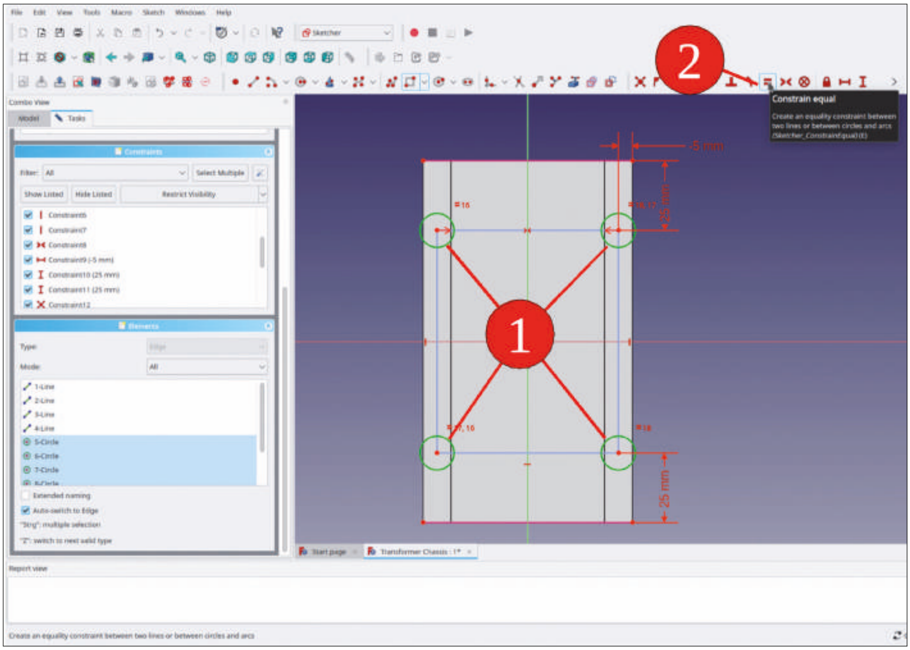


Figure 9-72

10. In the 'Elements' list, right-click one of the circles and constrain the diameter to 3 mm. Close the sketch (top).

11. In the tree view, mark the new sketch and click the 'Hole' tool button. In the task window, enter 3.2 mm for the diameter and select 'Through all' for the depth. Check the 'Reversed' checkbox (Figure 9-73). Close the task with the 'OK' button (top).

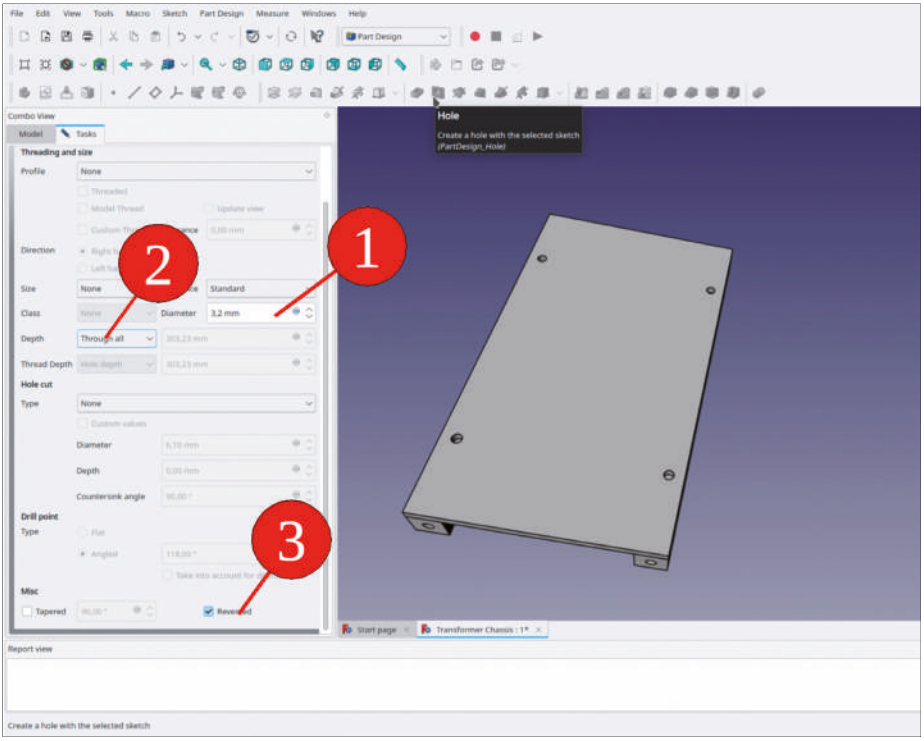


Figure 9-73

The holes are in the sheet. Now they need to be continued in the spacers, with the help of shape binders.

12. As the destination for the new shape binders, activate the body 'Chassis Spacer 001' by a double click (title then shown in bold letters).

13. In the body 'Chassis Sheet', hide the tip (Hole002), and display the sketch of the holes (Sketch004). From the sketch, mark the two holes that need to be transferred to 'Chassis Spacer 001' (Figure 9-74). Then, click the green 'Create a sub object(s) shape binder' tool button.

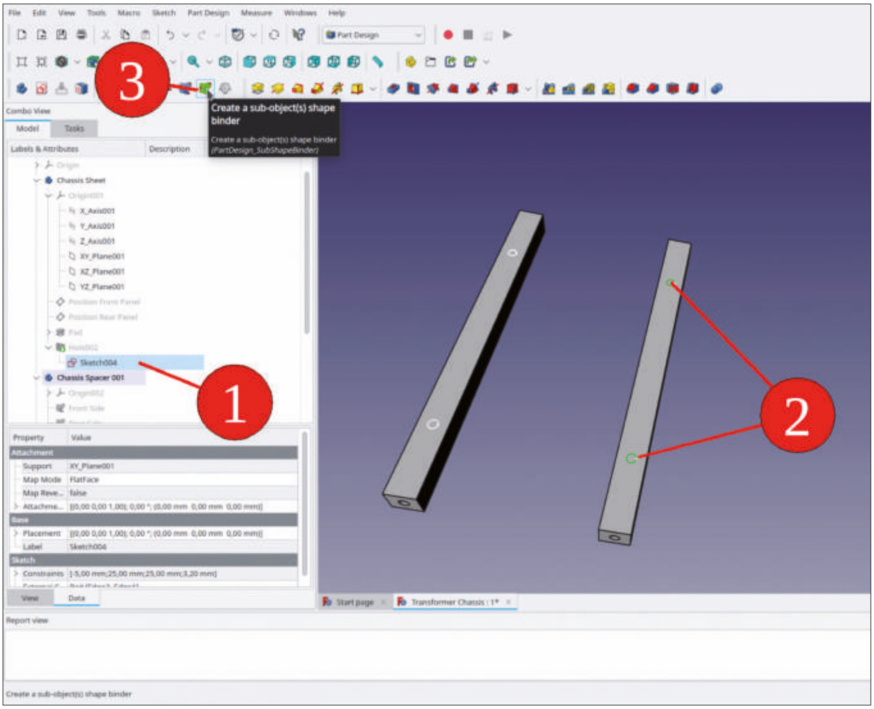


Figure 9-74

14. Rename the new SubShapeBinder to 'Mount Holes Chassis'. In the property list, set the property 'Make face' to 'false'.

15. In the tree view, mark the new SubShapeBinder and click the 'Hole' tool button. For the profile, select 'ISO metric regular profile', check the 'Threaded' checkbox. For the depth, select 'Through all' and for the key M3. Close the task window with the 'OK' button.

Because the other spacers were linked, the new threaded holes are transferred to those as well (Figure 9-75). They could be used to hold the secondary wiring harness with anchor plates. We will proceed with the assembly of other components to the chassis, when the front- and back panels are assembled. Then, it is easier to spot spatial constraints inside the apparatus.

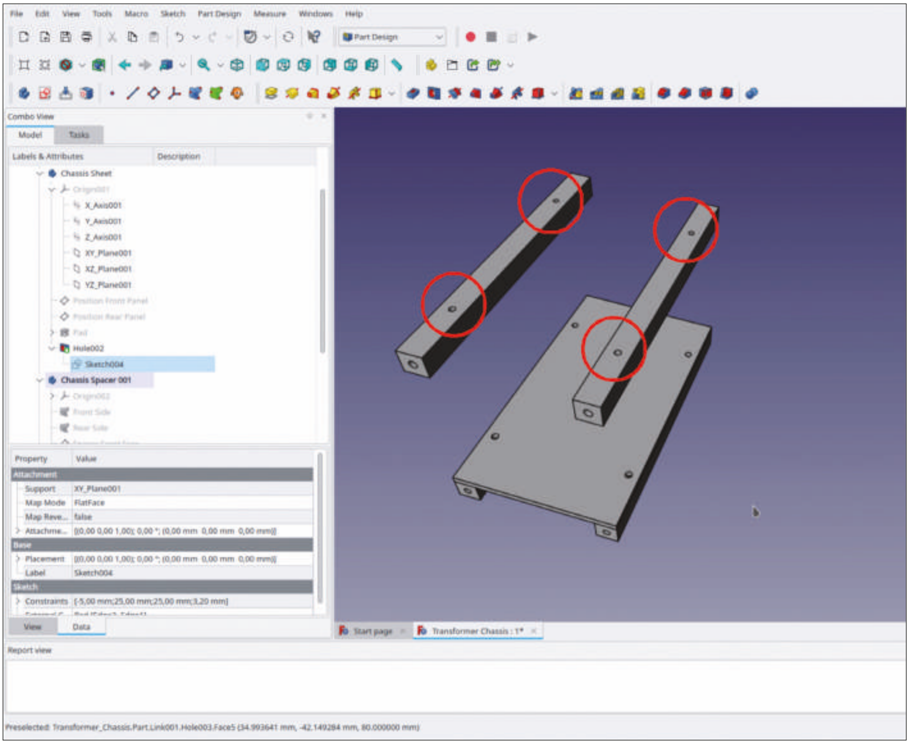


Figure 9-75

9.4 Assembling the Chassis, Front and Rear Panel

For the final assembly, you create a new file. Merging the already prepared sub assemblies resembles the insertion of elements into, e.g., the front panel, with cut-and-paste. Some referencing may aid in optimizing the lengths.

1. Create a new file and save it as 'Lab Transformer'.

2. In that file, start a new Std-Part-Container and rename that to 'Lab Transformer Complete'.

3. From the file 'Transformer Chassis', copy the Std-Part-Container 'Trafo Chassis Complete' and paste it into the new document. Drag-and-drop the container into 'Lab Transformer Complete'.

4. Similarly, copy 'Trafo Front Panel Complete' from the file 'Transformer Front Panel', paste it into the new document, and drag-and-drop the container into 'Lab Transformer Complete' (Figure 9-76).

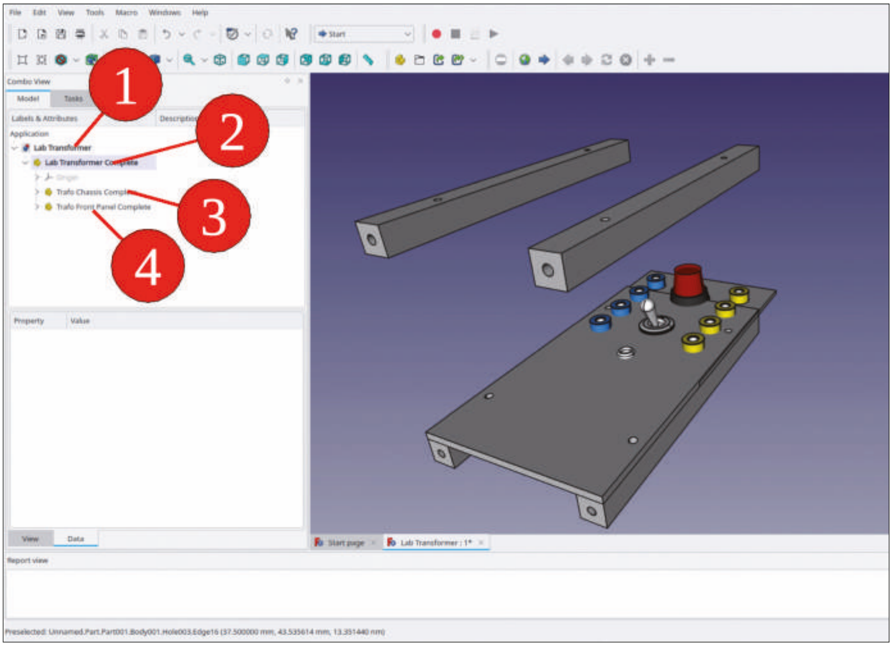


Figure 9-76

The orientation of the front panel is not aligned with the 'Front' direction of the final assembly. When creating the front panel, you selected the XY plane for the top face. This can be fixed by the definition of a shape binder and a proper attachment:

5. In the tree view, activate the body 'Chassis Sheet' by a double click. Click on the blue 'Create a shape binder' tool button. In the task window, click in the 'Object' button, which turns dark gray. Then, click the edit field next to the button (Figure 9-77).

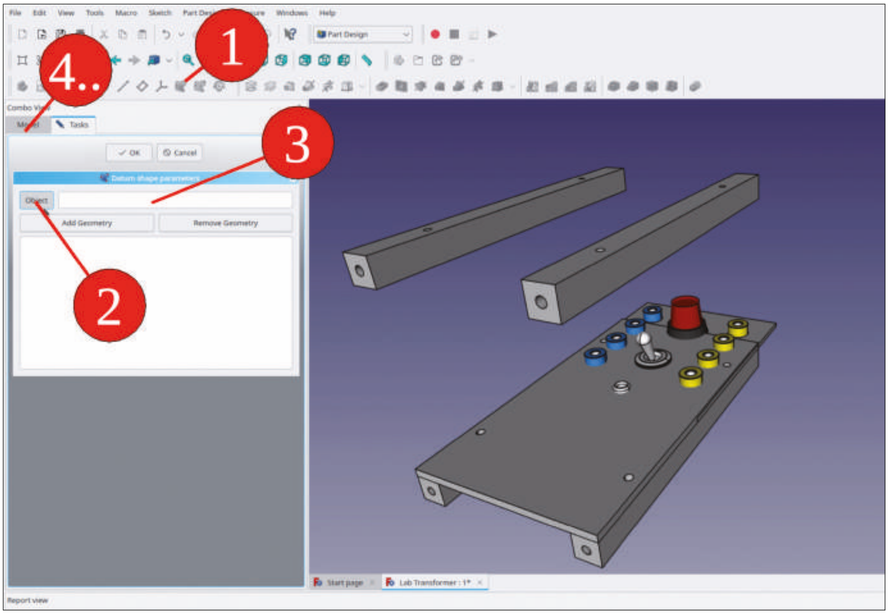


Figure 9-77

6. Switch to the 'Model' tab. Click on the datum plane 'Position Front Panel' (Figure 9-78) and return to the task window. Close the task window with the 'OK' button.

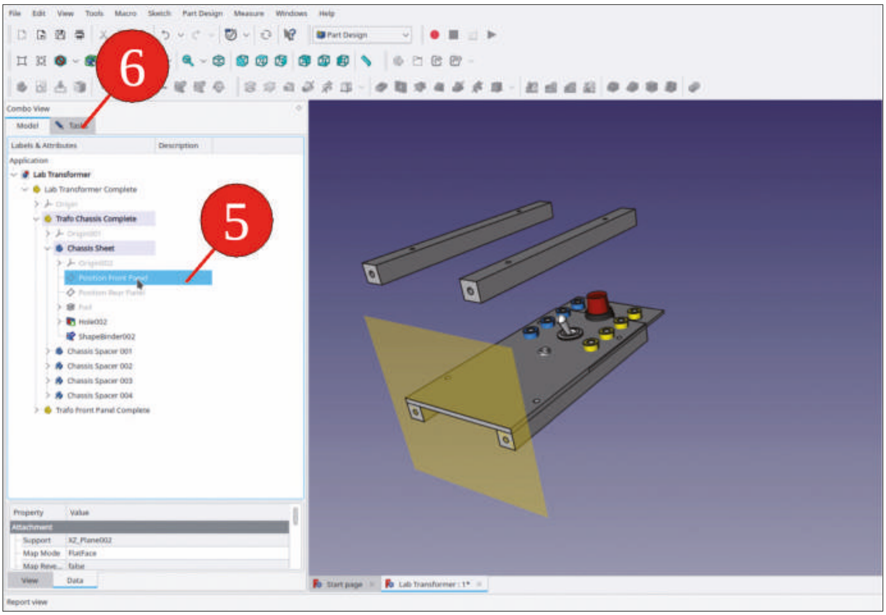


Figure 9-78

7. Rename the new shape binder to 'Front Panel Position'. Drag-and-drop it into the Std-Part-Container 'Lab Transformer Complete' (Figure 9-79). Set its 'Trace Support' property to 'true'.

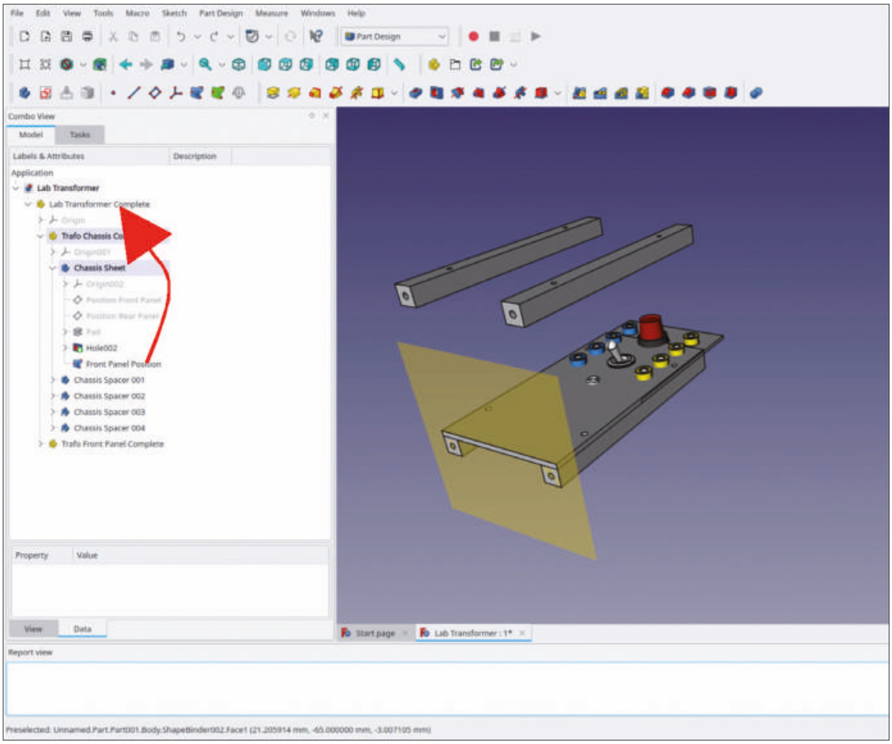


Figure 9-79

8. Now you fix the attachment: In the tree view, mark the Std-Part-Container 'Trafo Front Panel Complete' and switch to the 'Part' workbench.

9. From the main menu, select 'Part | Attachment...'. In the task window, the collector for reference 1 is already waiting for input. Switch to the 'Model' tab. In the tree view, click the shape binder 'Position Front Panel'.

10. Return to the 'Tasks' tab and select 'XY on plane' as the attachment mode. (Figure 9-80). For the attachment offset Y, enter 10 mm and close the task window with the OK button (Figure 9-81). Hide the shape binder 'Position Front Panel'.

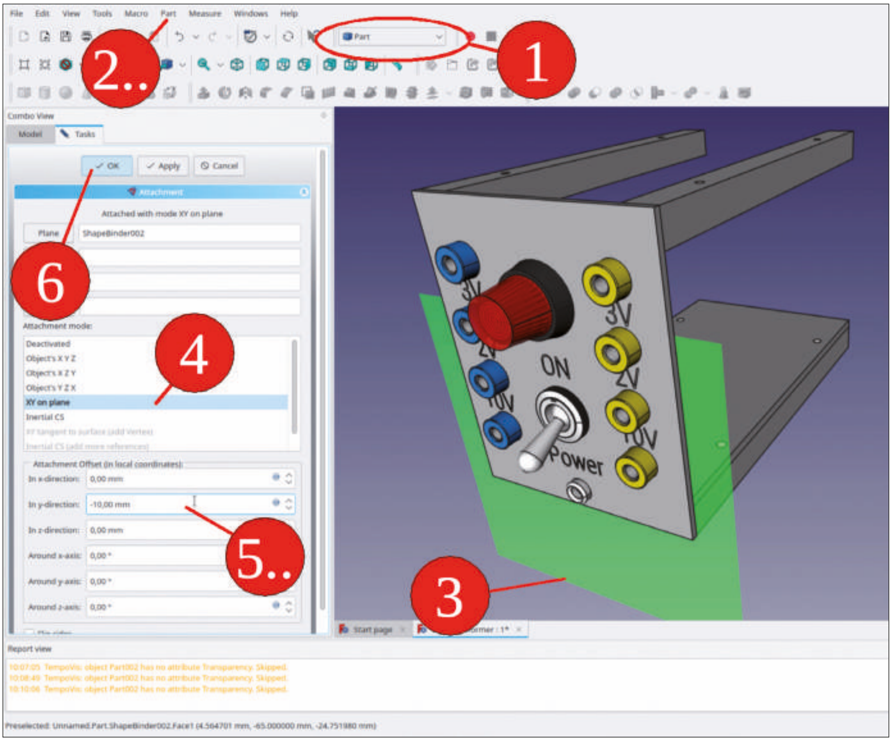


Figure 9-80

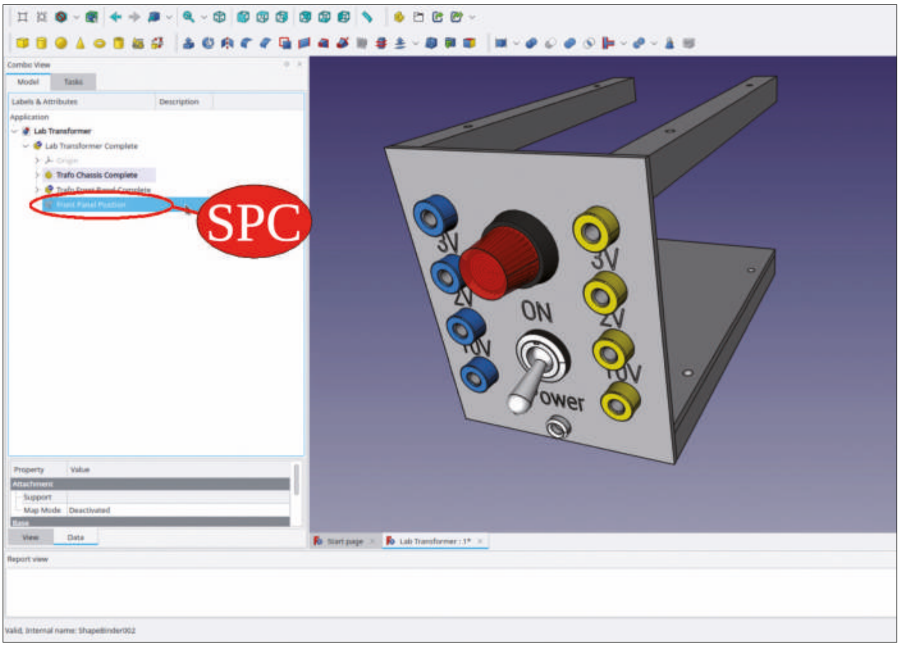


Figure 9-81

The designations for the directions are not always obvious. In this case, you have selected 'XY on plane' as the attachment mode. Even though the global coordinate system has the Z axis in the vertical direction, it is designated as the 'Y' direction for the attachment offset ('Z' is the direction normal to the referenced plane). Some trial-and-error helps.

The back panel is inserted into the assembly in just the same way:

11. Create a shape binder 'Rear Panel Position' referencing the datum plane 'Position Rear Panel' in 'Chassis Sheet'. Set its 'Trace Support' property to 'true'.

12. From the file 'Transformer Rear Panel', copy and paste the Std-Part-Container 'Trafo Rear Panel Complete' into the document 'Lab Transformer'. Drag-and-drop the Std-Part-Container into the Std-Part-Container.

13. Mark 'Trafo Rear Panel Complete' and switch to the 'Part' workbench. From the main menu, select 'Part | Attachment…'. Attach the rear panel to the shape binder 'Position Rear Panel', select 'XY on plane' as the attachment mode and enter a value of -10 mm for the attachment offset Y. Then, check the 'Flip sides' checkbox (Figure 9-82).

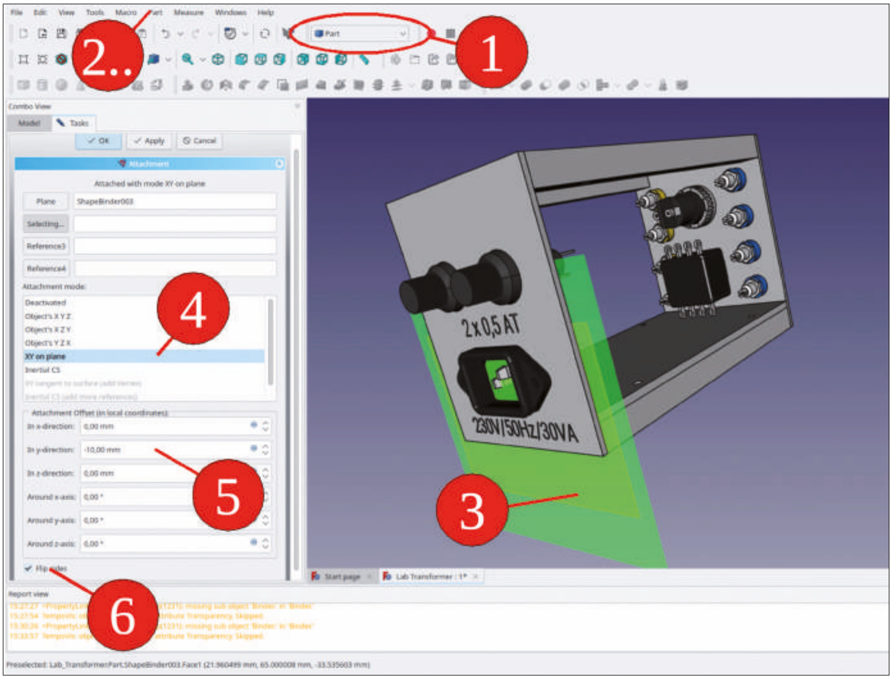


Figure 9-82

14. In the tree view, hide the shape binder 'Rear Panel Position' (Figure 9-83).

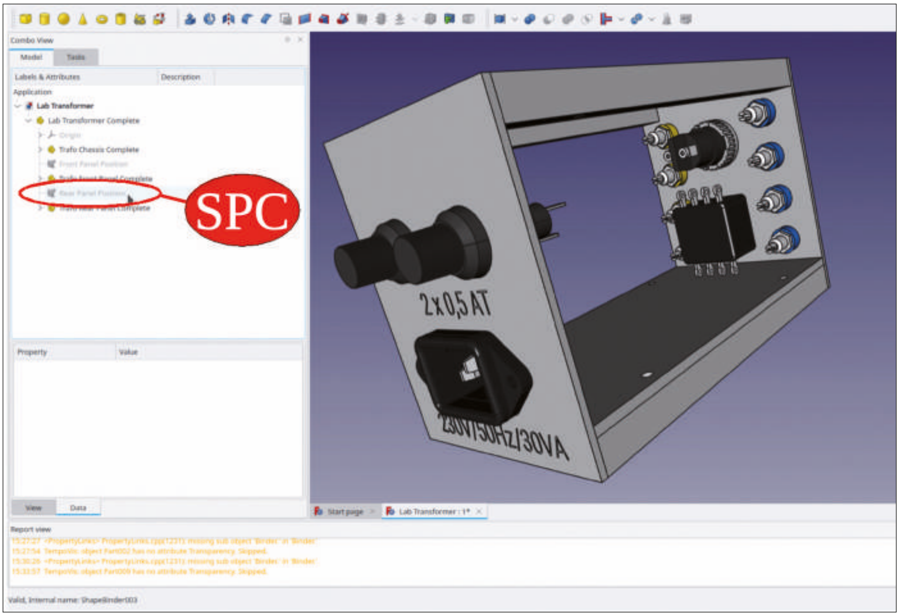


Figure 9-83

In the front and rear panels, the mounting holes for the spacers are still missing. Because it is not yet warranted that the spacers stay in the corners of the panels during, e.g., an optimization, the holes should be modeled via an associative relation.

15. In the tree view, locate the body 'Front Panel Sheet'. Activate this body by a double click.

16. Hide the body 'Front Panel Sheet', which you just activated, to expose the holes in the spacers below (Figure 9-84, SPACE key). Mark the edges of the 4 holes.

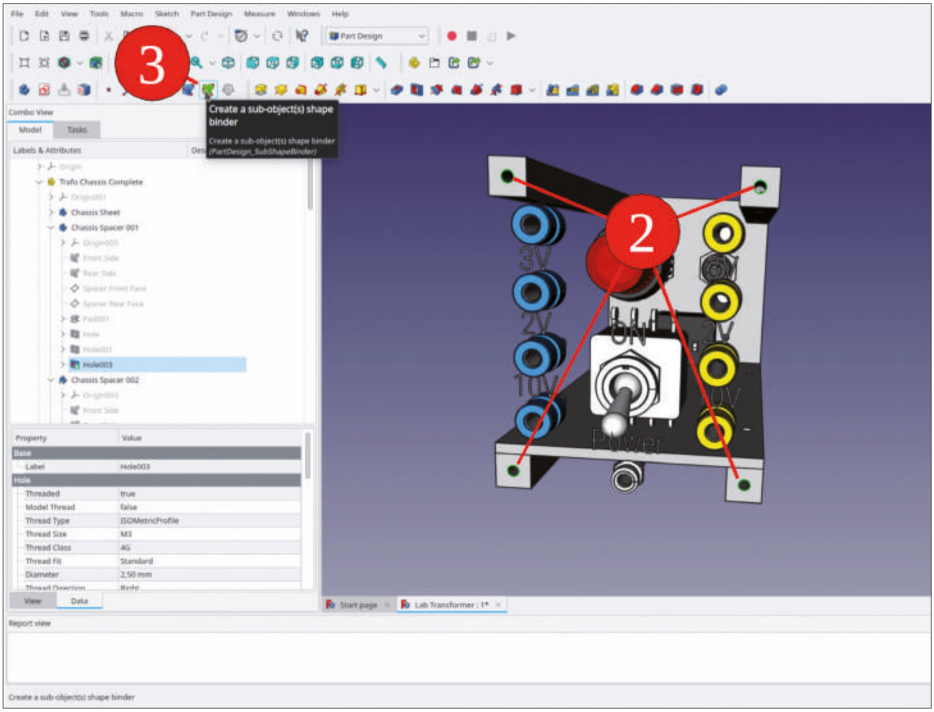


Figure 9-84

17. Click on the green 'Create a sub object(s) shape binder' tool button.

18. Scroll up to locate the body 'Front Panel Sheet' again, in which the new SubShapeBinder has been placed. Rename the SubShapeBinder to 'Footprint Spacers'.

19. With the SubShapeBinder marked in the tree view, click the 'Hole' tool button. In the task window, set the diameter to 4.2 mm, select 'Trough all' as the depth, and check the 'Reversed' checkbox. Then, display the 'Front Panel Sheet' again with the SPACE key.

20. In the similar way, insert the mounting holes in the back panel.

The chassis with all the elements on front and back panel is now completed (Figure 9-85). As an example for the innards, a transformer will be added next. For the placement of inside components, it is important to have a good representation of the casing around, in order to avoid collisions by optimization of the positions.

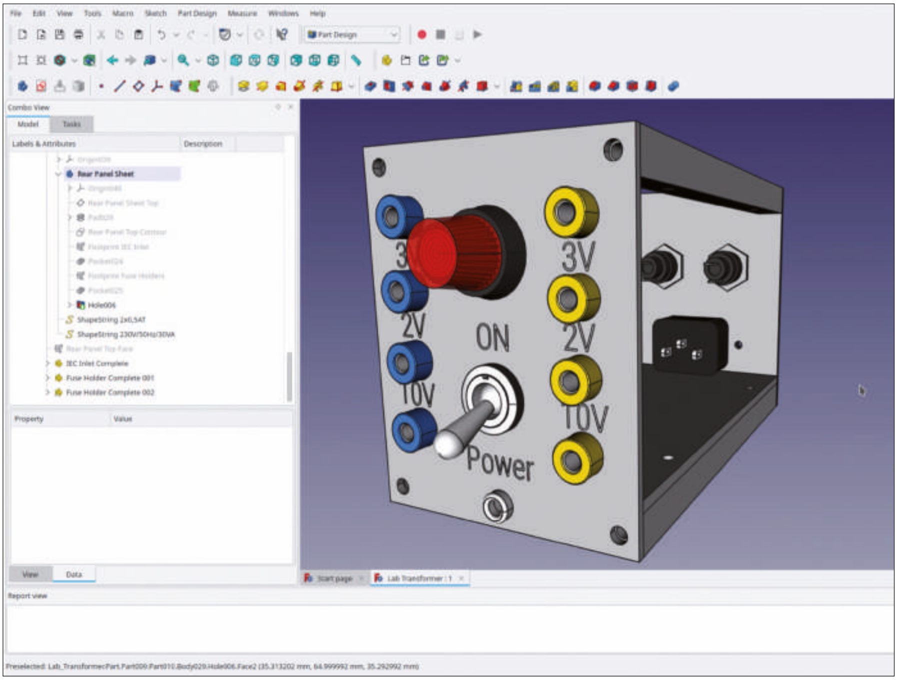


Figure 9-85

9.5 Placing the Transformer

In the components directory, there is a model file with an example Transformer. The file describes the vintage type 'Engel ET3'. Even though the piece was quite old, it had a tap on the primary coil allowing for a 230 VAC primary voltage, as well as the necessary screens for the protection of the secondary winding. If you want to actually create the example project, you need to purchase an up-to-date transformer from your supplier. Very likely, you would have to match the number of banana jacks, and the engraving on the front panel with your device. A few sentences with respect to safety need to be added here: If you plan to use this lab transformer on your bench, it has to deliver protected low voltage at the secondary side (SELV). To meet this specification, special insulation and screens are necessary, which are usually proclaimed in the data sheet as well as on the type label. Another important item is the protection of the transformer against overload.

The fuses for the primary coil, provided on the back panel, are a safeguard against gross failures (e.g., a short in the secondary circuit, or melting insulation, shorting out parts of the primary coil ...). The transformer is not yet protected against slow and painful overheating. This has to be done by a thermal fuse which could be strapped to the outside of the coil although it is better located inside the coil — some types have these fuses already integrated. Insulated versions of thermal fuses exist, and otherwise a suitably rated material must be added for insulation. If you are not already an experienced developer in the field, you can easily see that these matters are not an easy task to address, involving critical material and component choices. As yet one more issue to ponder, the transformer has no secondary fuses, because it is unclear how the user combines the various coils with short lab jumper cables. Thus, the transformer is no toy, but needs to be used with care and oversight. If it is operated neglectfully and carelessly, it can represent a fire hazard. Construction and use of the equipment must thus be performed by certified electricians only.

21. In the 'Sample Projects | Lab Transformer | Components' directory, locate and open the file 'Engel Transformer ET3.FCStd'. From this file, copy the Std-Part-Container 'ET3 Complete' and paste it into the document 'Lab Transformer' (double-click the destination document, so its title is shown in bold, before you paste, Figure 9-86).

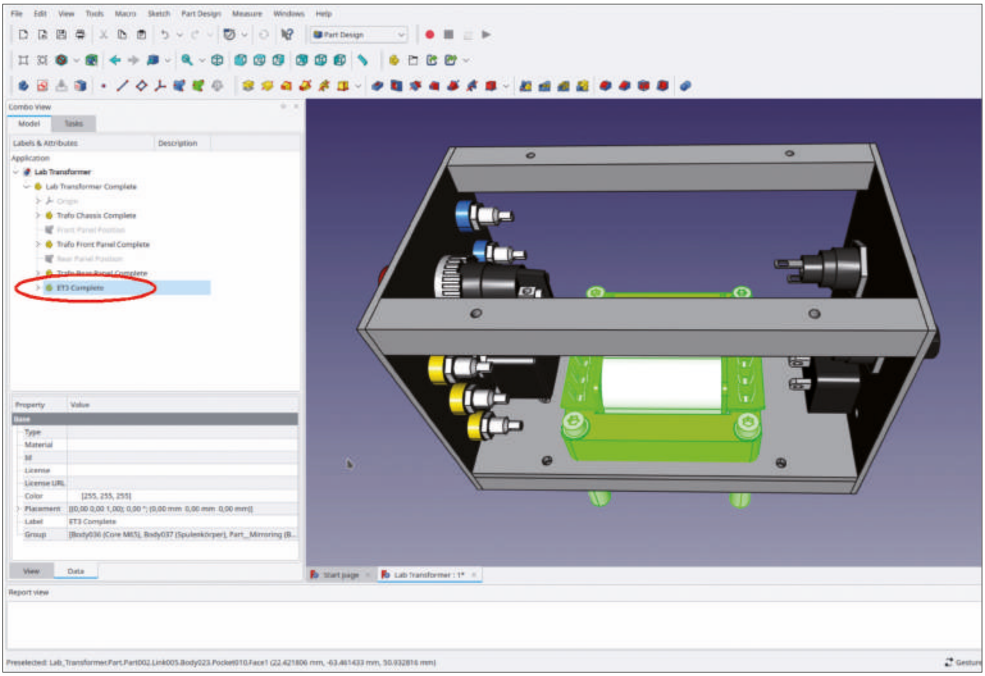


Figure 9-86

The transformer looks displaced (Figure 9-86). In order to lift it onto the top of the chassis, the placement parameters could simply be adjusted to match. The placement would refer to the whole assembly, and not be associated with the chassis sheet, though. A short consideration provides a shortcut for this special case: In principle, the transformer is a part of the 'Trafo Chassis Complete', because it will bolt to that part. If you drag-and-drop the transformer into that Std-Part-Container, the placement already refers to the chassis, and the intended association is thus provided, except for chassis sheet thickness variations. We take this shortcut here, in the final assembly:

22. Drag-and-drop the Std-Part-Container 'ET3 Complete' into the 'Trafo Chassis Complete' Std-Part-Container. Edit the placement of the transformer container and set the Z translation to 40.5 mm (Figure 9-87).

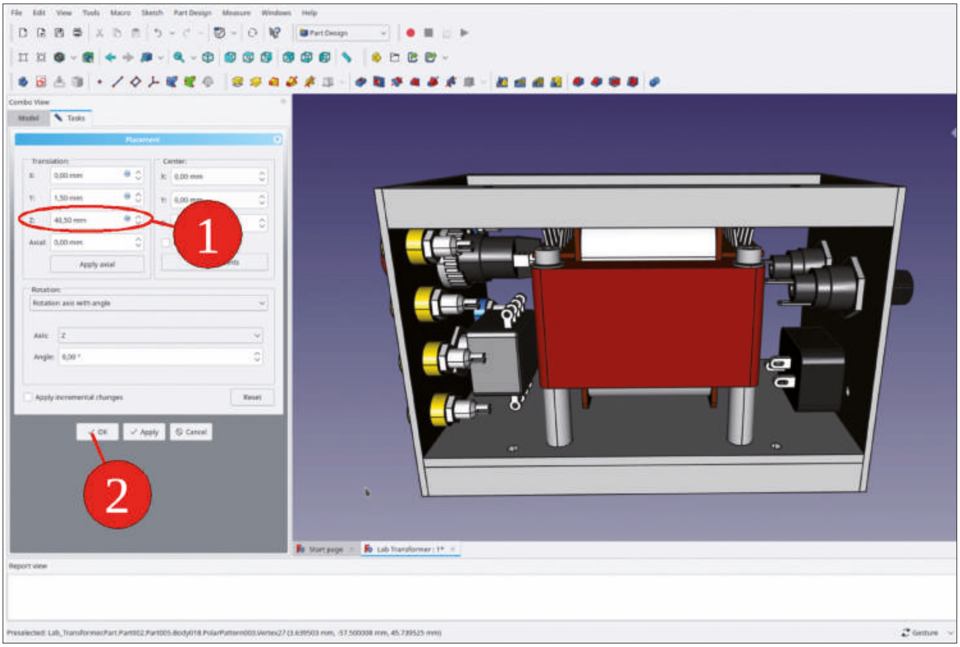


Figure 9-87

23. The visual inspection shows that the pilot lamp is located very close to the transformer terminal strip (Figure 9-88). Even though it should be possible to assemble the unit, the transformer could be moved a bit, in order to gain a little more space. Switch the view to 'Top' and, from the main menu, select 'View | Orthographic view'.

24. Edit again the placement of the transformer. Set the Y translation to 1.5 mm (Figure 8-89).

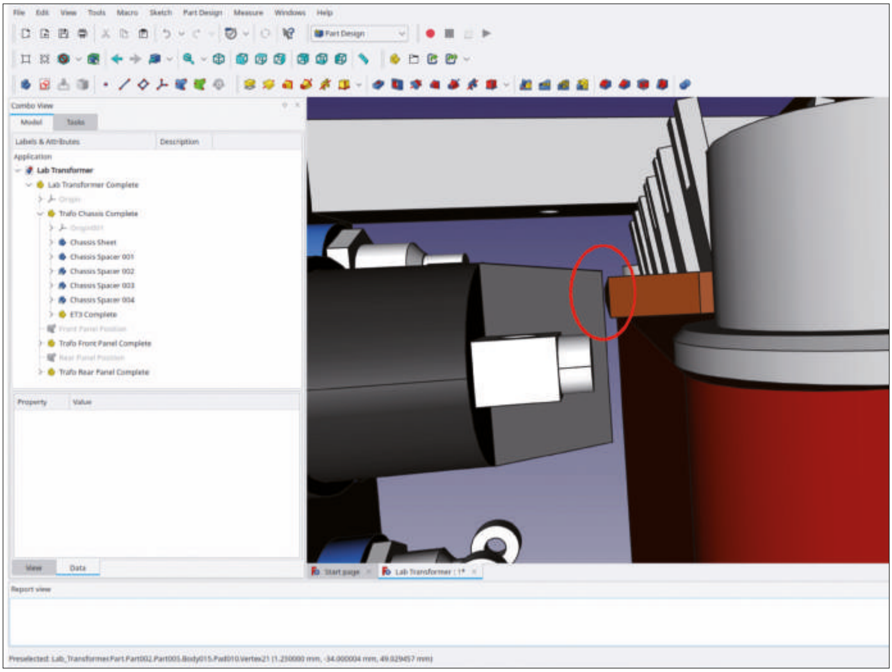


Figure 9-88

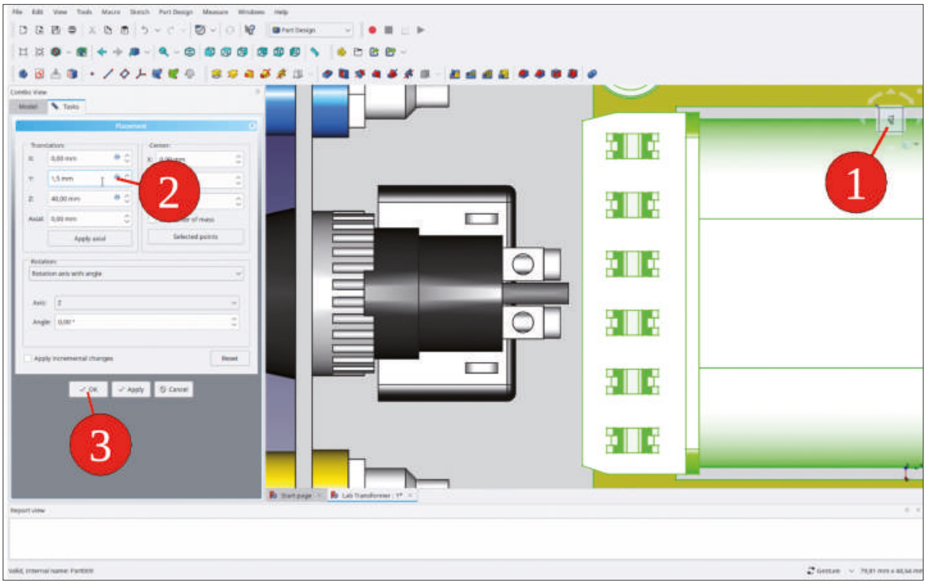


Figure 9-89

9.6 Placing the Anchor Plates

With transformers, it is good practice to separate the wiring for primary and secondary circuits. With our example project, it is possible to route the wires for the primary in the center of the casing, underneath the transformer. Two anchor plates help to fix the little wire harness later on. With respect to the project structure, the anchor plates belong to 'Trafo Chassis Complete', because their fasteners fix it to that part.

1. In the tree view, hide 'ETD3 Complete' with the SPACE key, in order to expose the chassis sheet again. From the 'Sample Projects | Lab Transformer | Components' directory, open the file 'Anchor Plate.FCStd'. Copy the Std-Part-Container 'Anchor Plate Complete' and paste it into the document 'Lab Transformer'. Drag and drop it into the 'Trafo Chassis Complete' Std-Part-Container. Rename it to 'Anchor Plate Complete 001'.

2. In the tree view, click the new anchor plate. In the property list, open the 'Placement' task window by clicking on the … button in the edit field of the placement property. Set the translations X = –9 mm, Y = –33 mm and Z = 3 mm (Figure 9-90). Close the task window with the OK button.

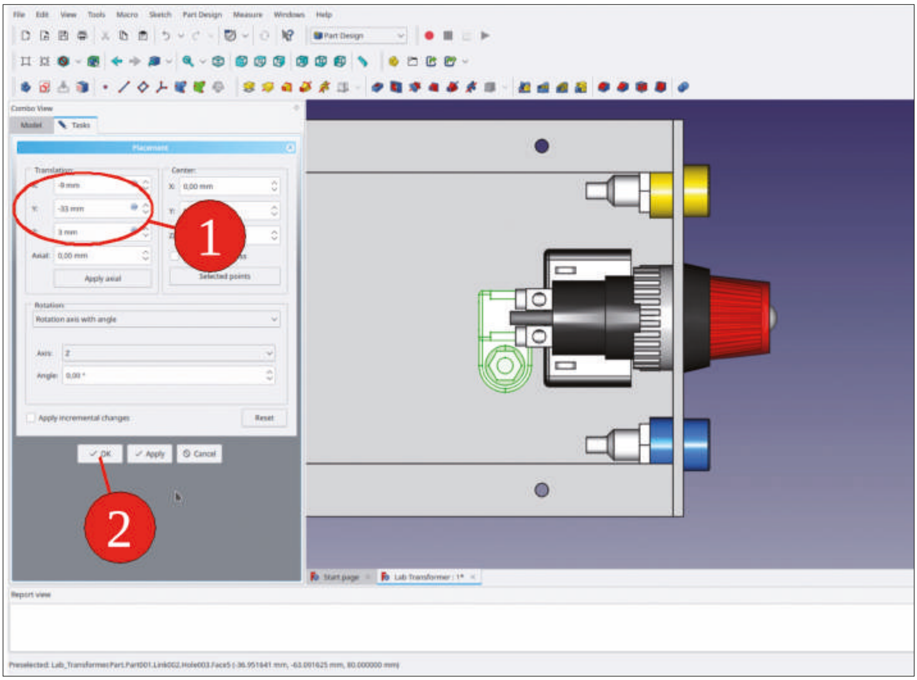


Figure 9-90

3. In the tree view, click the new anchor plate and click the 'Make link' tool button (Figure 9-91).

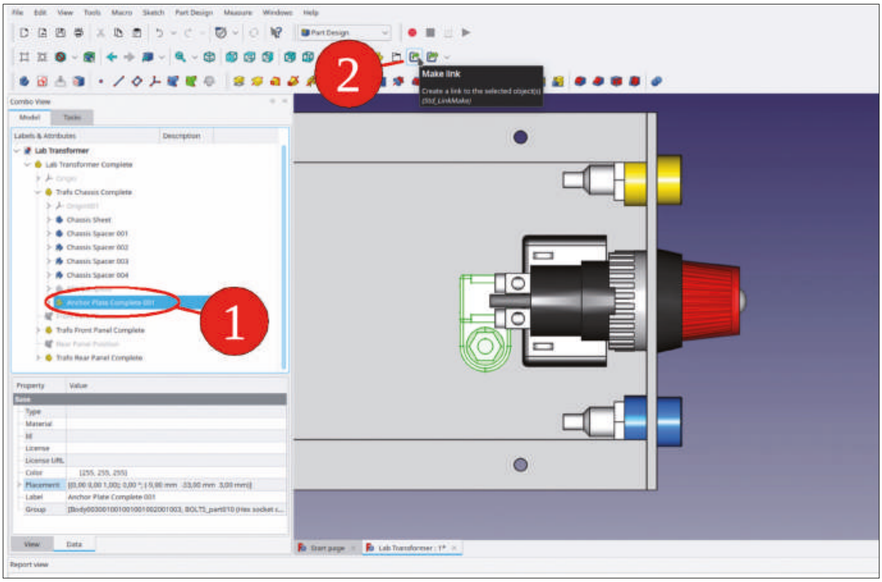


Figure 9-91

4. The new anchor plate appears as 'Anchor Plate 002'. Drag-and-drop it into the Std-Part-Container 'Trafo Chassis Complete'.

5. In the tree view, mark 'Anchor Plate 002'. In the property list, set 'Link transform' to 'true'. With this, the further 'Link placement' parameters can be specified relative to the parent part. Open the 'Link placement' task window by clicking on the ... button in the 'Link placement' edit field. Set the Y translation to 72 mm (Figure 9-92).

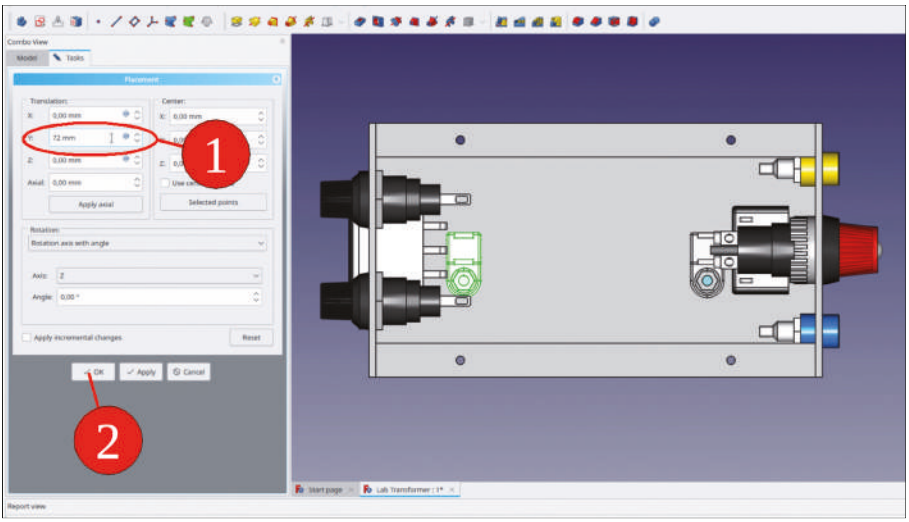


Figure 9-92

6. In the tree view, display the transformer 'ETD3 Complete' again with the SPACE key. Visually inspect the anchor plates in their locations (Figure 9-93).

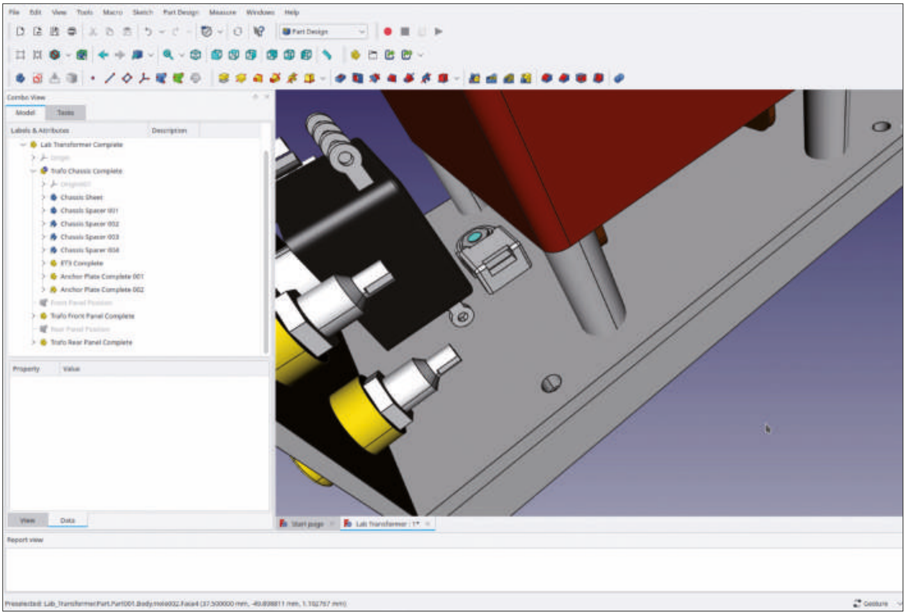


Figure 9-93

In the chassis sheet, the bolt holes for transformer and anchor plates are still missing.

7. In the tree view, activate the body 'Chassis Sheet' by a double click, which also sets it as the destination for the SubShapeBinders created in the following steps.

8. Hide the just activated body to expose the component edges necessary to define the holes.

9. In the 3D view, from the bottom of the transformer, mark the inner contours of the transformer spacers. Then, click the green 'Create a sub object(s) shape binder' tool button (Figure 9-94).

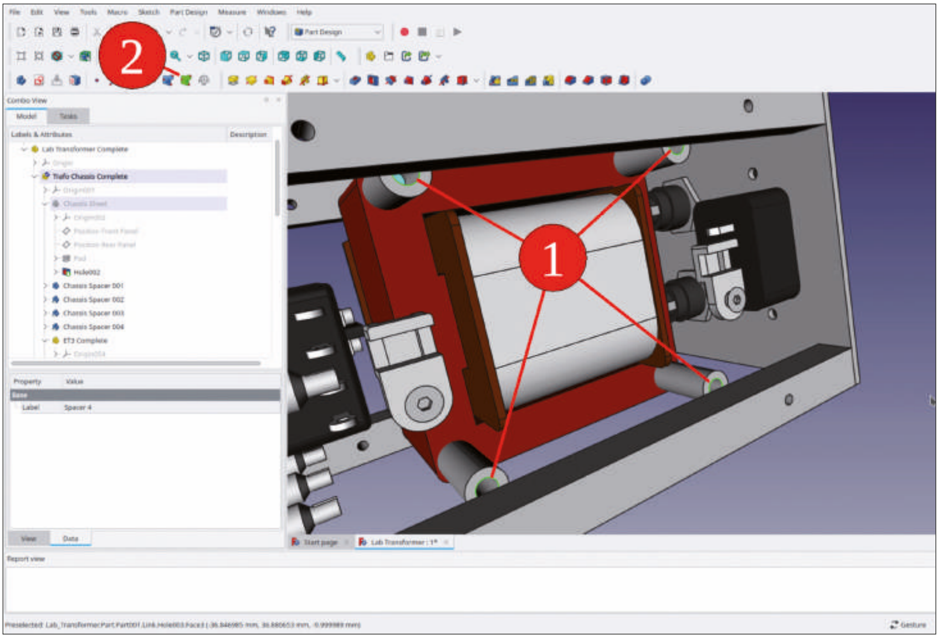


Figure 9-94

10. In the tree view, scroll back to 'Chassis Sheet'. Rename the new SubShapeBinder to 'Footprint Transformer'.

11. With the new SubShapeBinder marked in the tree view, click the 'Hole' tool button. In the task window, set the diameter to 4.2 mm and select 'Through all' as the depth. Temporarily show the 'Chassis Sheet' body again, to check the results of the 'Hole' command (Figure 9-95). Close the task window with the 'OK' button.

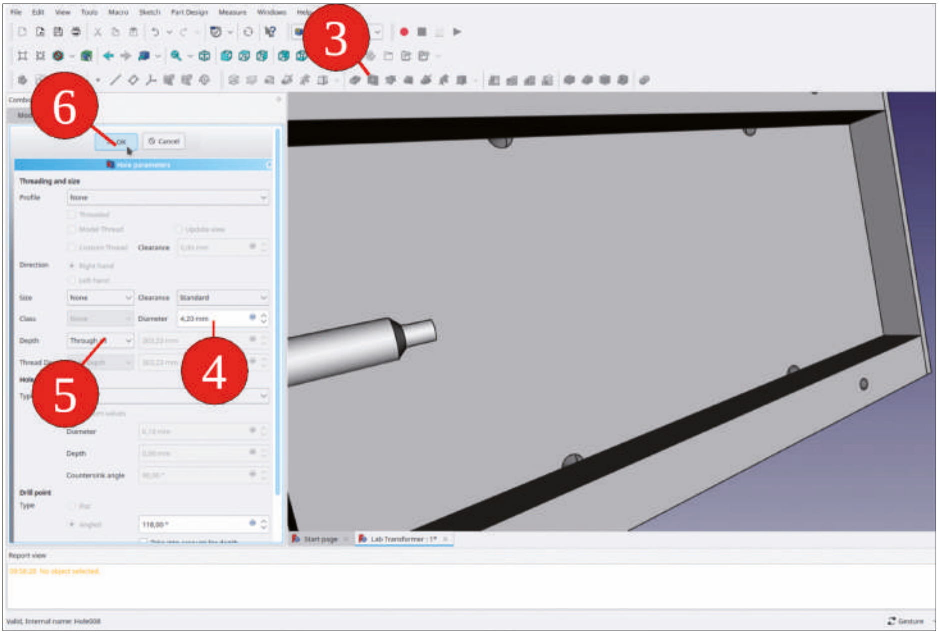


Figure 9-95

12. In the same way, create the bolt holes for the anchor plates. Set the diameter for those to 3.2 mm.

13. With the anchor plates, the heads of the countersunk screws are located inside the material. Expand 'Anchor Plate Complete 001' and edit the placement for the screw. Set the Z translation to −3.01 mm. With the addition of 0.01 mm to the chassis thickness of 3 mm, the head is displayed clearly, in front of the sheet (Figure 9-96). The screw for the second anchor plate follows, as that is linked to the first one.

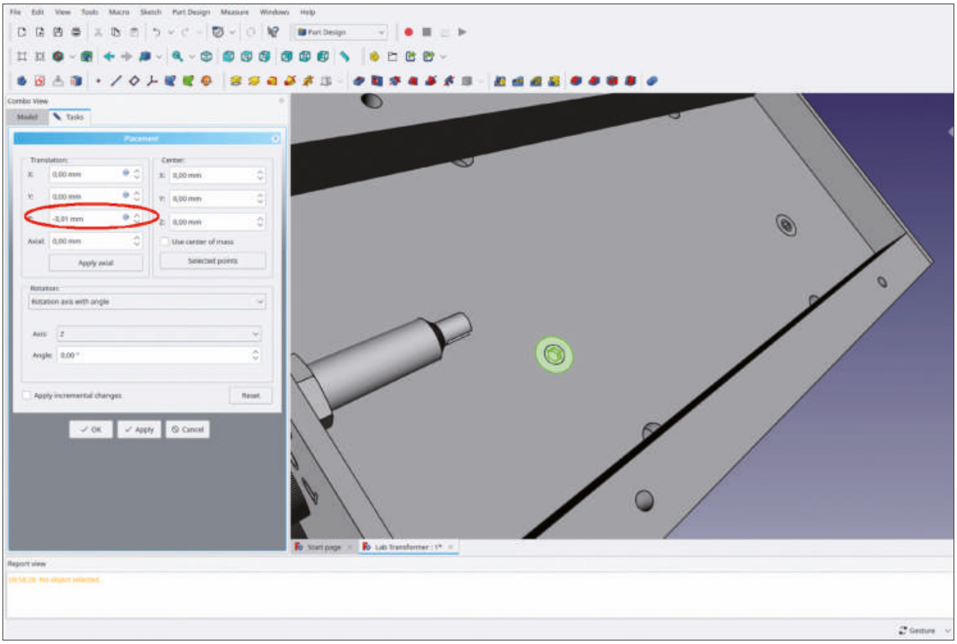


Figure 9-96

For the transformer, some vent holes in the chassis would allow the cooling air flow to enter from the bottom. In this way, the air could also access the coil from the bottom. Entering from holes in the lower part of the cover side walls, the air would probably not get there as much. To create the holes, the 'Chassis Sheet' needs to be touched. For this change, the late point in time is plausible: Only now, with the maturing 3D model, these questions can be addressed with visual feedback from the design.

14. In the tree view, activate the body 'Chassis Sheet' by a double click, if necessary.

15. Start the sketcher and select the XY plane as the sketching plane.

16. From the main menu, select 'Sketch | View section' and 'View | Orthographic view'.

17. Draw two circles, approximately where the vent holes are expected. Mark the two centers of the circles and the Y-axis. Then, click the 'Constrain symmetrical' tool button (Figure 9-97).

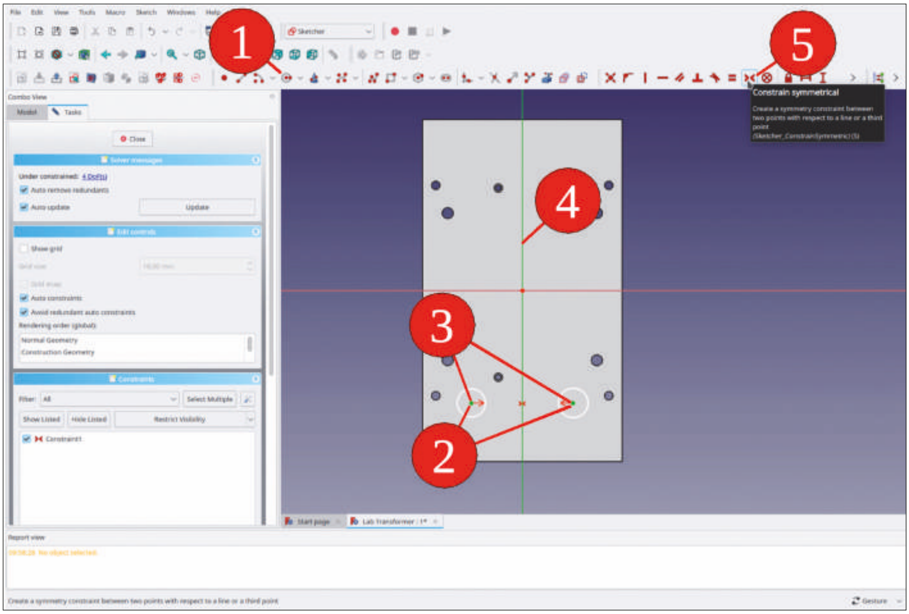


Figure 9-97

18. Mark the two circles and click the 'Constrain equal' tool button (Figure 9-98, steps 1, 2).

19. In the 'Elements' list, right-click one of the circles. From the context menu, select 'Constrain diameter' and enter a value of 7 mm (Figure 9-98, steps 3, 4).

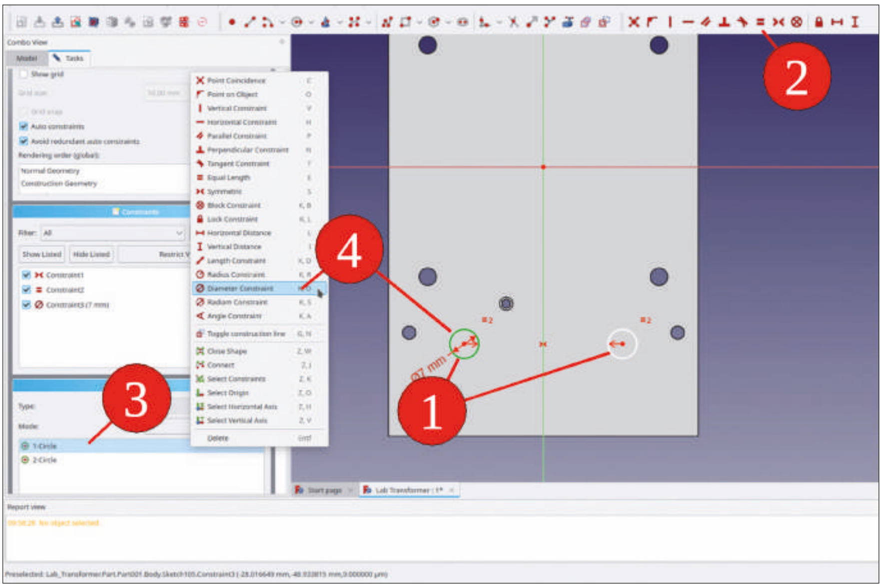


Figure 9-98

20. Mark the two circle centers and constrain the horizontal distance to 34 mm. Then, mark one of the circle centers and the origin, and constrain the vertical distance to 40 mm (Figure 9-99). Close the sketch.

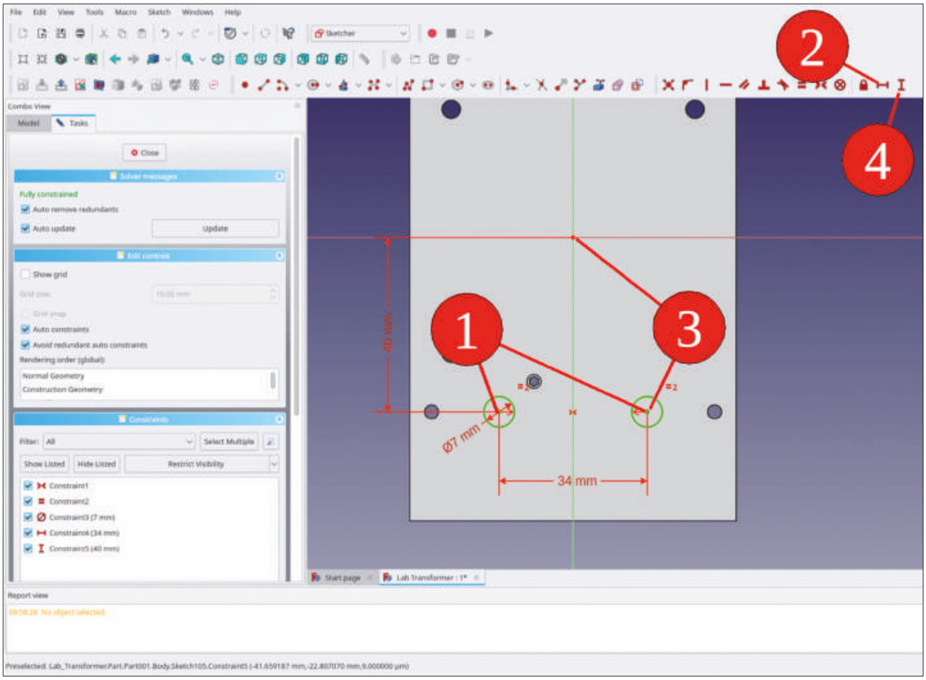


Figure 9-99

21. In the tree view, mark the new sketch and click the 'Pocket' tool button. In the task window, select the type 'Through all' and check the 'Reversed' checkbox (Figure 9-100). Close the task window with the 'OK' button.

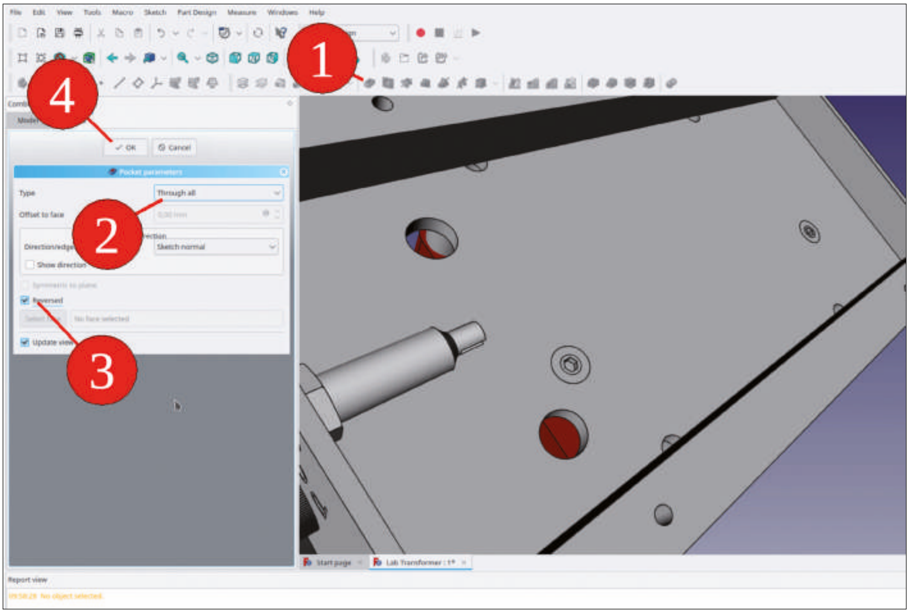


Figure 9-100

22. In 'Chassis Sheet', mark the last design state (tip), and select 'Part Design | Apply a pattern | LinearPattern' from the main menu. In the task window, select 'Vertical Sketch axis' as the direction, and set the number of occurrences to 7 (Figure 9-101). Close the task with the 'OK' button.

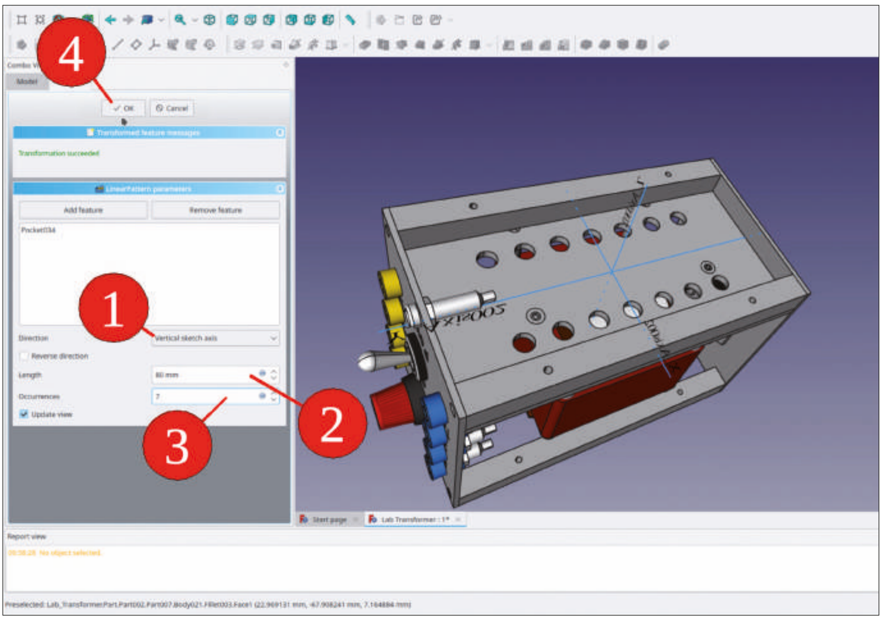


Figure 9-101

9.7 Associative Generation of the Sheet Metal Cover

For the generation of the sheet metal cover, robust relations to the other parts of the assembly are necessary. It is also worthwhile to think about which parts could be the source of these references. The length of the apparatus is defined by the two datum planes 'Position Front Panel' and 'Position Rear Panel' in the body 'Chassis Sheet'. The width and height of the apparatus are controlled by the width and height of the front (and rear) panel. With the cover extended a little longer than the apparatus itself, the references could be derived from the front and back panel.

Defining the references turns out to be not trivial. Of course, shape binders could be generated, referencing the edges of the panels directly. But then, they get dependent on facet numbers, and the design could break by re-enumeration, after the panels have been touched again.

Therefore, it is meaningful to create a reference to a sketch located on a datum plane, which would be a resilient anchor in 3D space.

9.7.1 Where to Pick the References

For the top of the new cover, it would be good to define references to the outer top edges of front panel and back panel. Fortunately, inside the bodies 'Front Panel Sheet' and 'Back Panel Sheet', there already exist sketches of the outer panel outlines: The sketches 'Front Panel Top Contour' and 'Rear Panel Top Contour'. We will make use of the fact that a shape binder can also be defined using only parts of sketches, which saves us work.

1. In the tree view, display the sketches 'Front Panel Top Contour' and 'Rear Panel Top Contour' with the SPACE key (Figure 9-102).

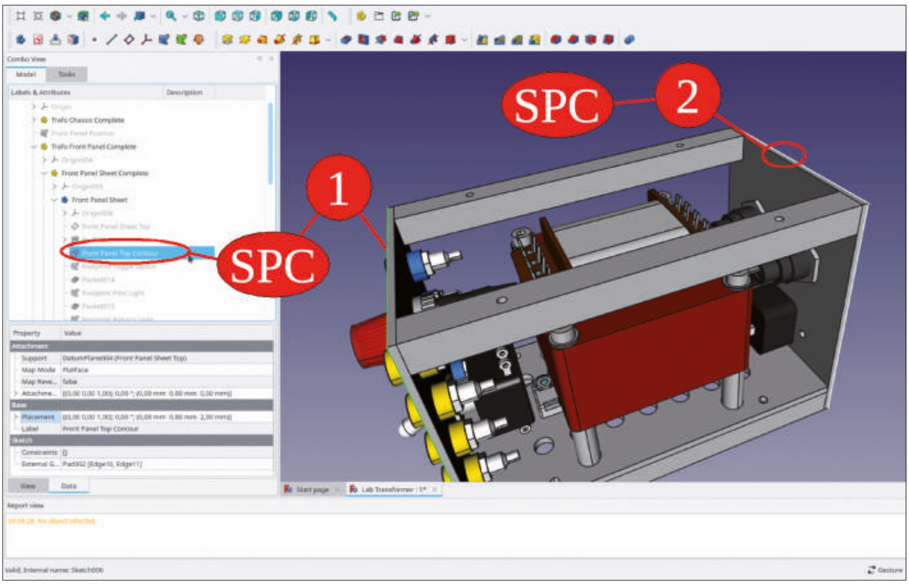


Figure 102

From these sketches, you will pick the necessary edges for the shape binders, with which the cover top will be defined. As a result, the cover will follow length changes of the chassis as well as width and height changes of the front cover. All of this may look a bit indirect, but a lot of associativity is returned.

9.7.2 Creating the Cover

1. For the cover, start a new Std-Part-Container and rename it to 'Trafo Cover Complete'. In that container, start a new body and rename it to 'Trafo Cover Sheet' (Figure 9-103).

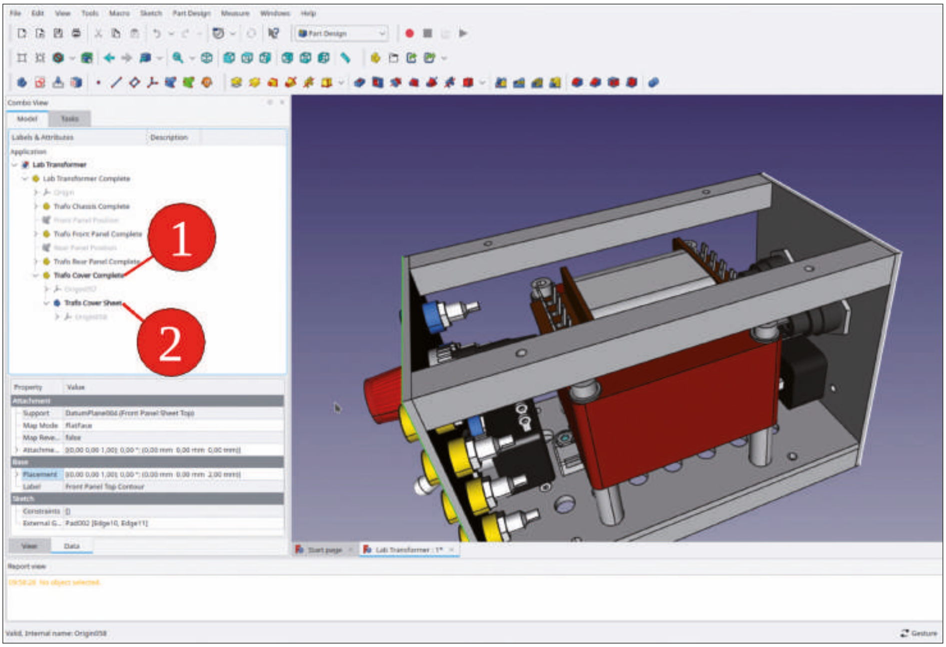


Figure 9-103

2. In the 3D view, hide the tip (last design state) of 'Front Panel Sheet'. From the sketch of the front panel top outline, mark the upper line. Then, click the blue 'Create a shape binder' tool button (Figure 104).

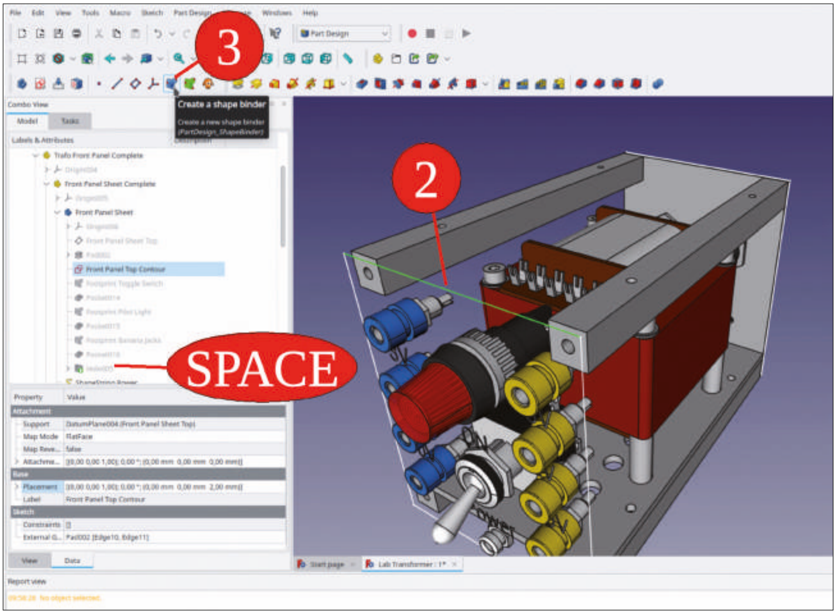


Figure 9-104

3. In the task window, the selection of the object (the sketch 'Front Panel Top Contour') and the upper line (edge 1) is preset (Figure 9-105) because of our prior markup. Close the task window with the OK button.

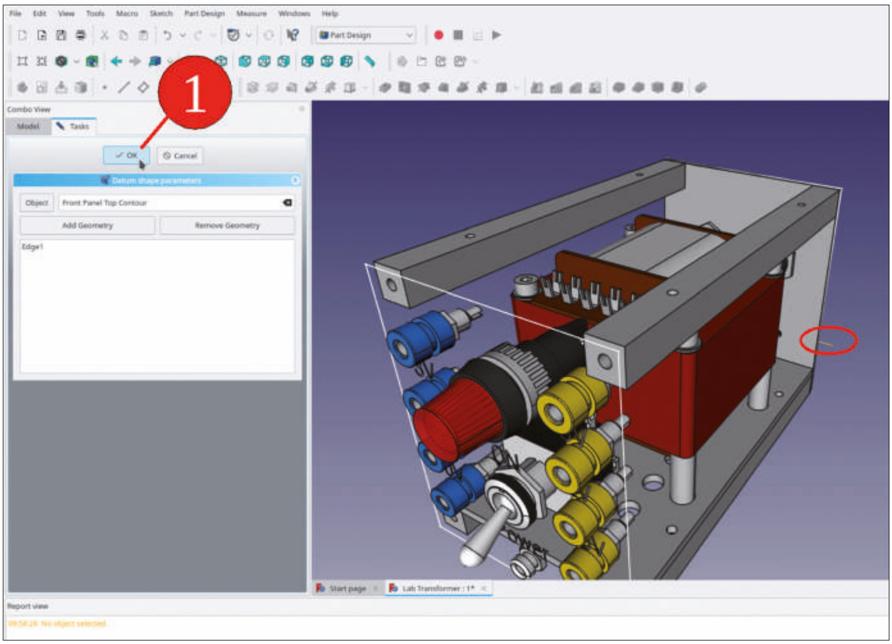


Figure 9-105

4. In 'Trafo Cover Sheet' (scroll down), rename the new shape binder to 'Front panel Outline Top'. The shape binder will jump to its correct location after setting its 'Trace Support' property to 'true', followed by a recalculation (F5 key, Figure 9-106).

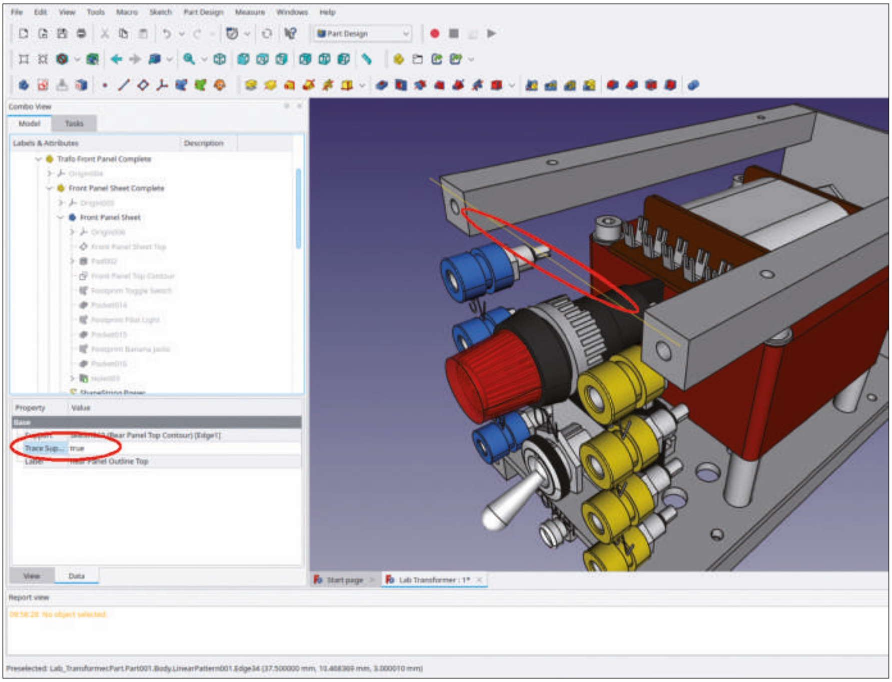


Figure 9-106

5. In the same way, create a shape binder for the back panel top outline.

6. In the bodies 'Front Panel Sheet' and 'Back Panel Sheet', hide the sketches of the top contours again. In the two bodies, display again the tips. Then, in the tree view, hide all Std-Part-Containers except 'Trafo Cover Complete' with the SPACE key (Figure 9-107). All this hideaway is making the selection of the shape binders easier.

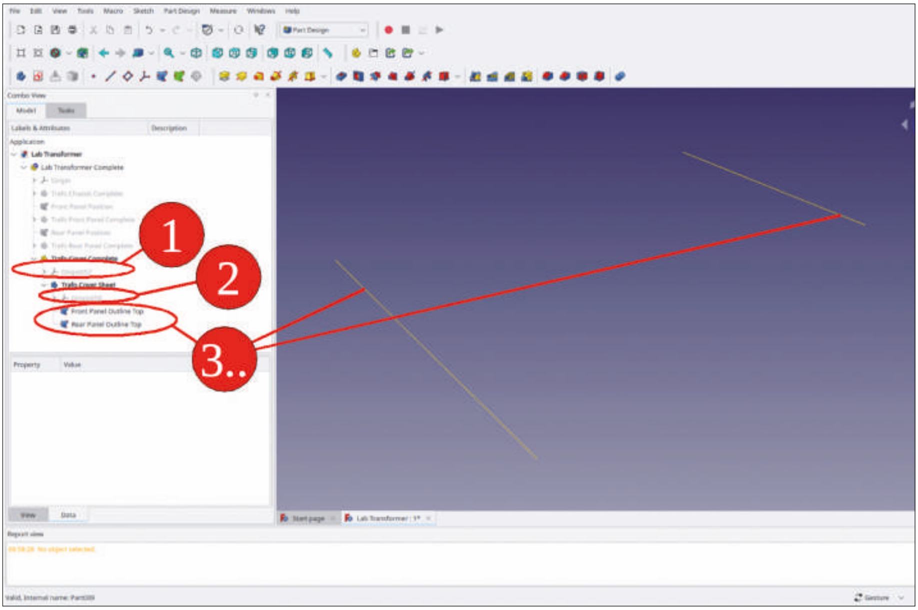


Figure 9-107

7. In the tree view, mark the two new shape binders and click the 'Create a datum plane' tool button (Figure 9-108). Do not change the preset values and close the task window with the 'OK' button. Rename the new datum plane to 'Trafo Cover Top Plane'.

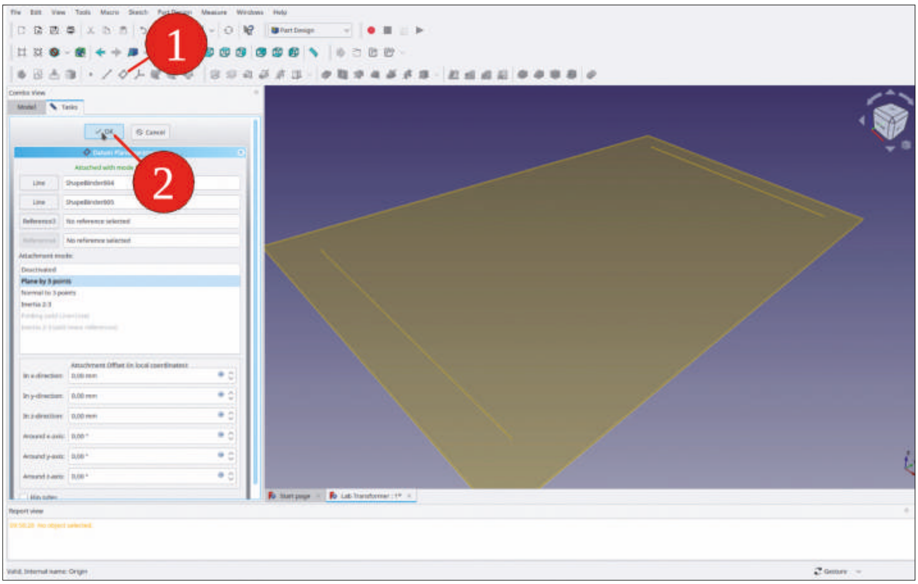


Figure 9-108

8. Mark the new datum plane and start the sketcher.

9. Close and reopen the sketch in order to show external geometry (Figure 9-109). The shape binders appear as yellow lines. Temporarily display the 'Trafo Front Panel Complete' to see which of the shape binders corresponds to the front (Figure 9-109).

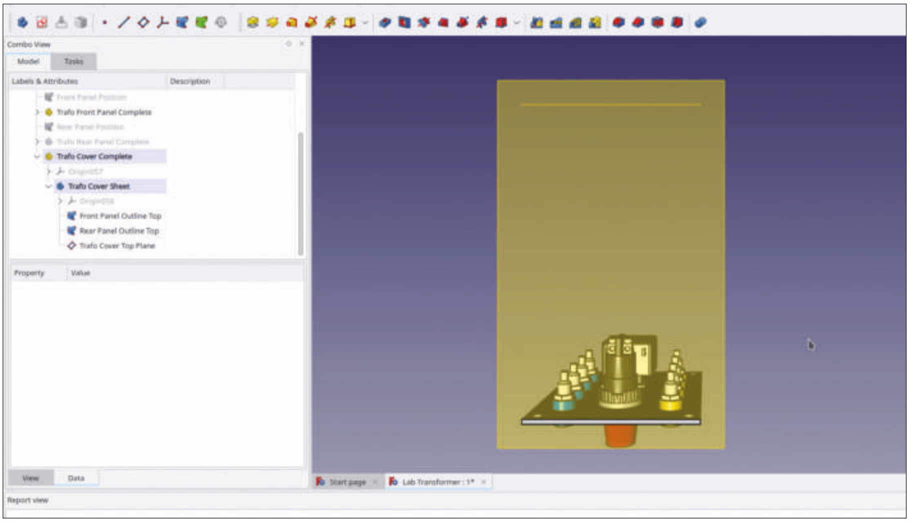


Figure 9-109

10. Click on the 'External geometry' tool button and mark the two shape binders (Figure 9-110).

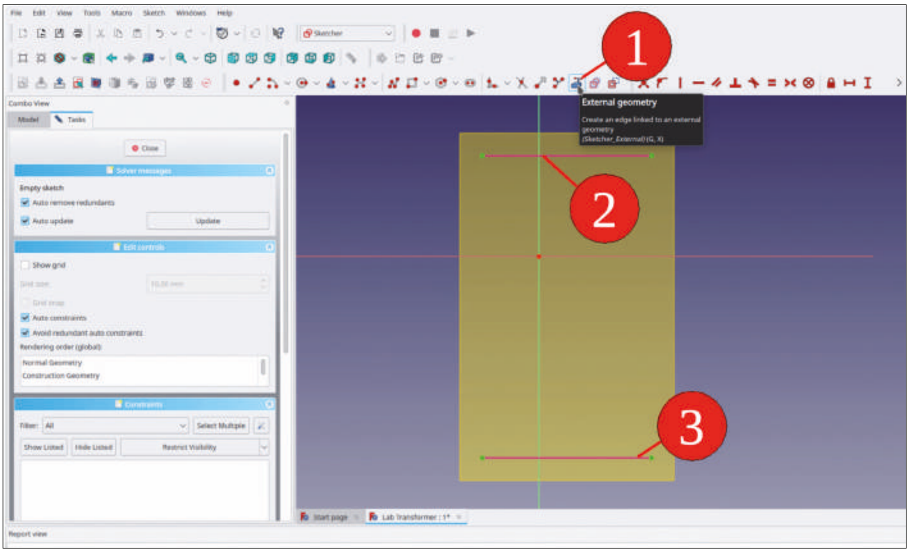


Figure 9-110

11. Draw a rectangle. The cover should overlap to the front and back side somewhat. Mark a corner of the top reference line and the corresponding corner of the rectangle. Constrain the vertical distance to 3 mm (Figure 9-111).

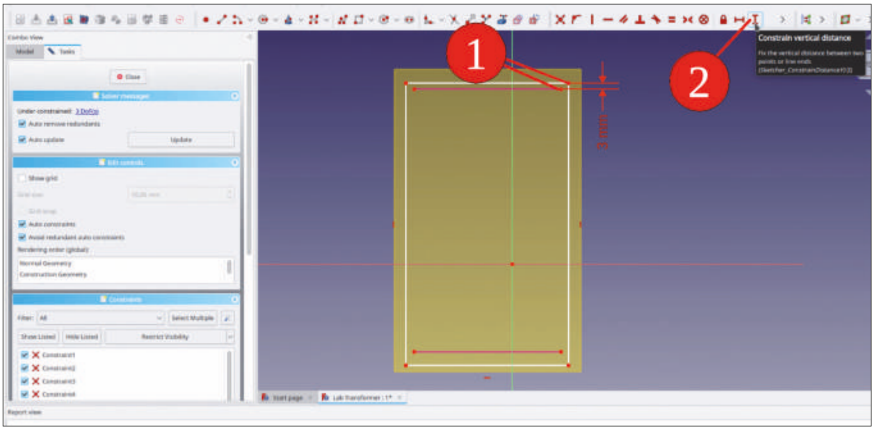


Figure 9-111

The upper shape binder corresponded to the rear panel. It is important that only the vertical position of the rectangle top is controlled by this shape binder. We will assign the control to the width of the cover to the front panel, a few steps ahead.

12. Mark two lower, corresponding end points of rectangle and construction geometry. Also constrain the vertical distance of these points to 3 mm.

13. Mark the two lower points again and constrain their horizontal distance to zero. (Figure 9-112).

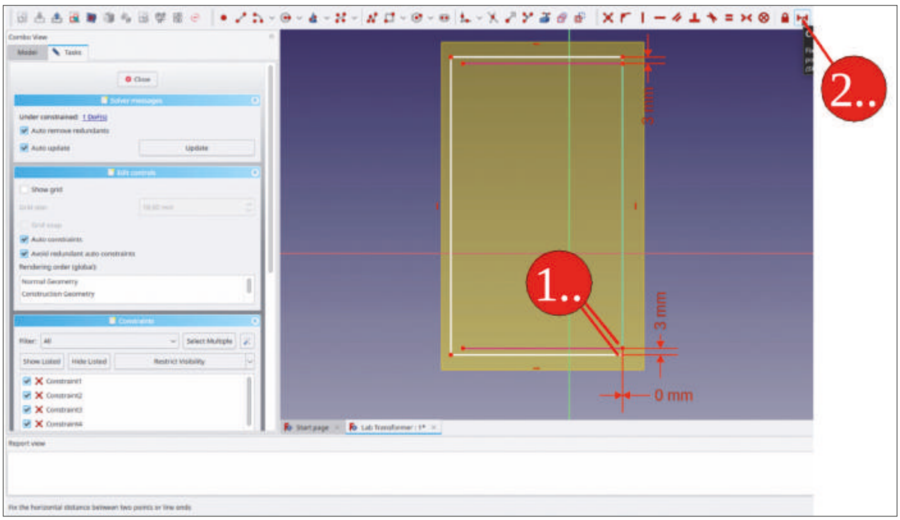


Figure 9-112

14. On the opposite side, mark the lower point of the rectangle and the corresponding end point of the lower construction line. Constrain the horizontal distance to 0 mm (Figure 9-113). Close the sketch.

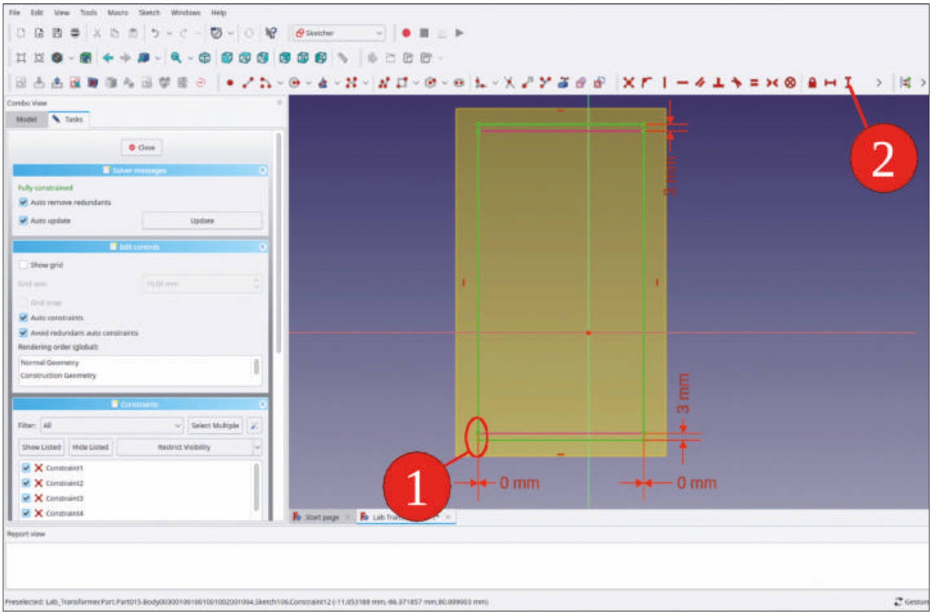


Figure 9-113

Now, you are ready to create the cover. Thanks to a wonderful 'Sheet Metal' workbench, this can proceed with only little effort:

15. Switch to the 'Sheet Metal' workbench.

16. In the tree view, mark the new sketch and click the 'Make Base Wall' tool button (Figure 9-114).

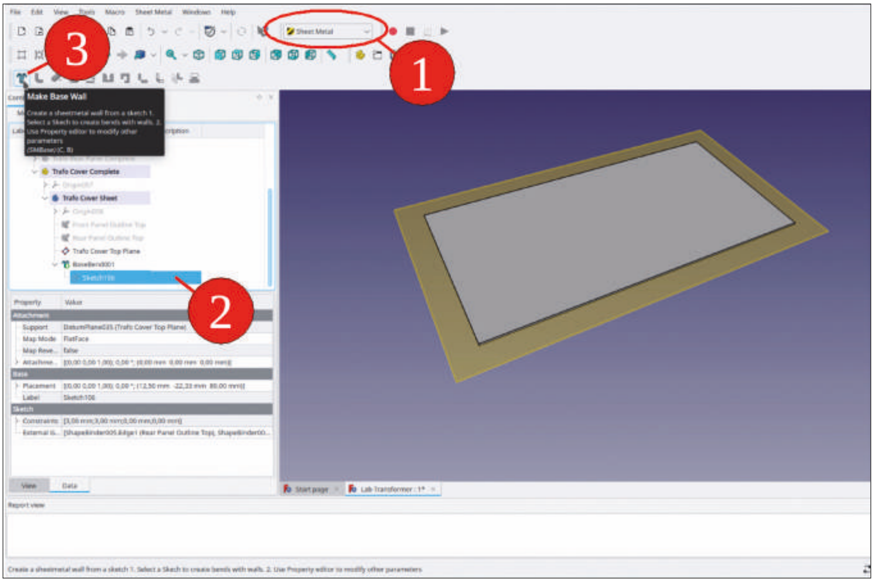


Figure 9-114

17. In the tree view, hide the shape binders 'Front Panel Outline Top' and 'Rear Panel Outline Top'. Also hide the datum plane 'Trafo Cover Top Plane'. Then, display again all the Std-Part-Containers (Figure 9-115).

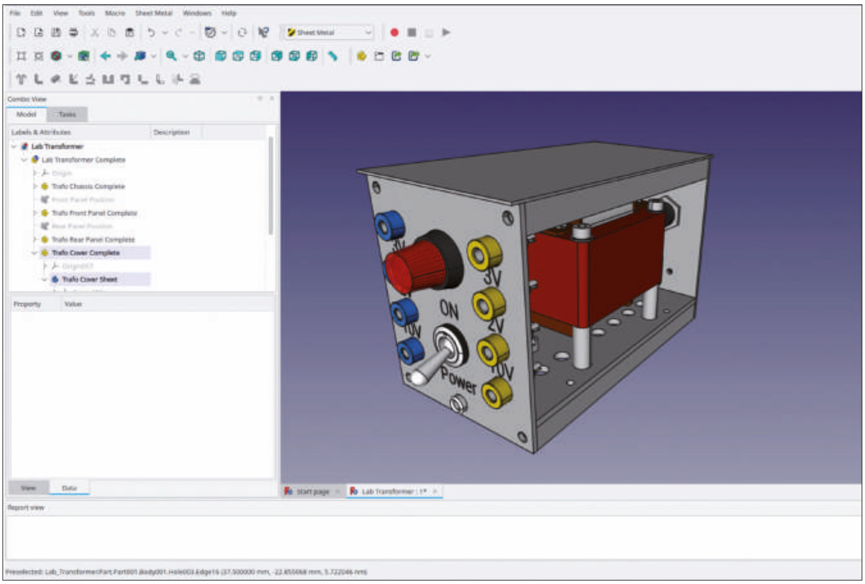


Figure 9-115

18. Mark a long top edge of the cover and click the 'Make Wall' tool button (Figure 9-16).

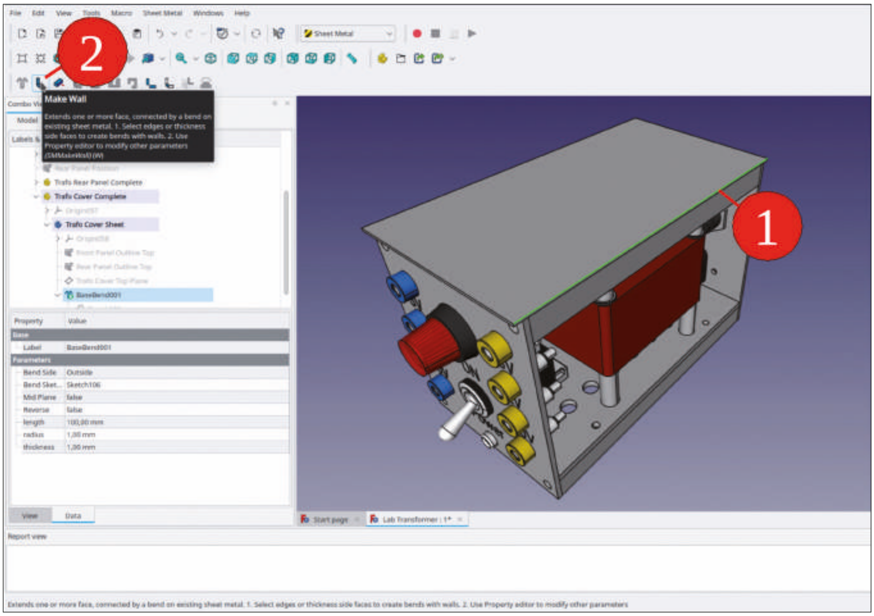


Figure 9-116

19. In the tree view, mark the new bend object. In the property list, select 'Thickness outside' as the bend type. Set the length to 89 mm. The bend will only update, if a recalculation has been triggered (e.g., by the F5 key, Figure 9-117).

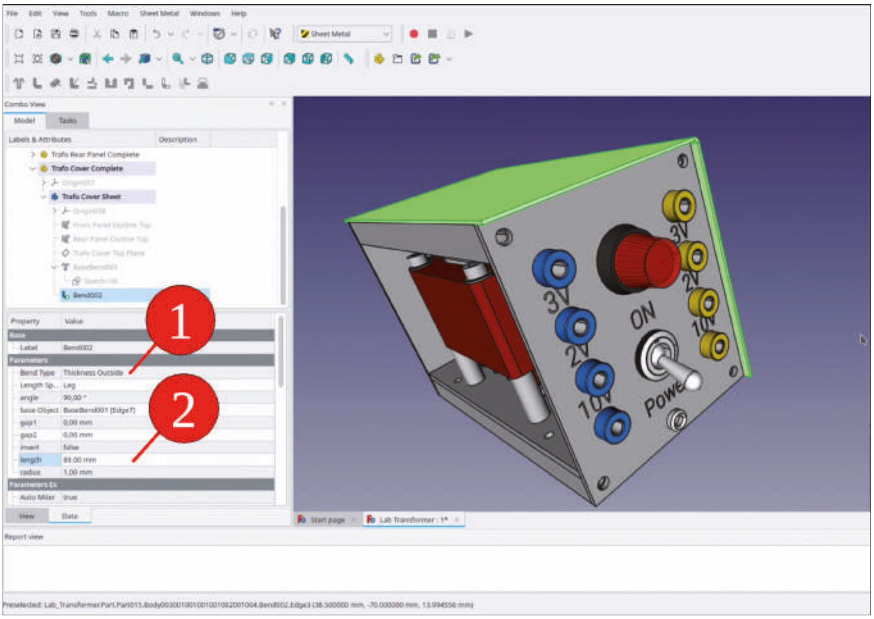


Figure 9-117

20. In the same way, create the opposite wall of the cover (Figure 9-118).

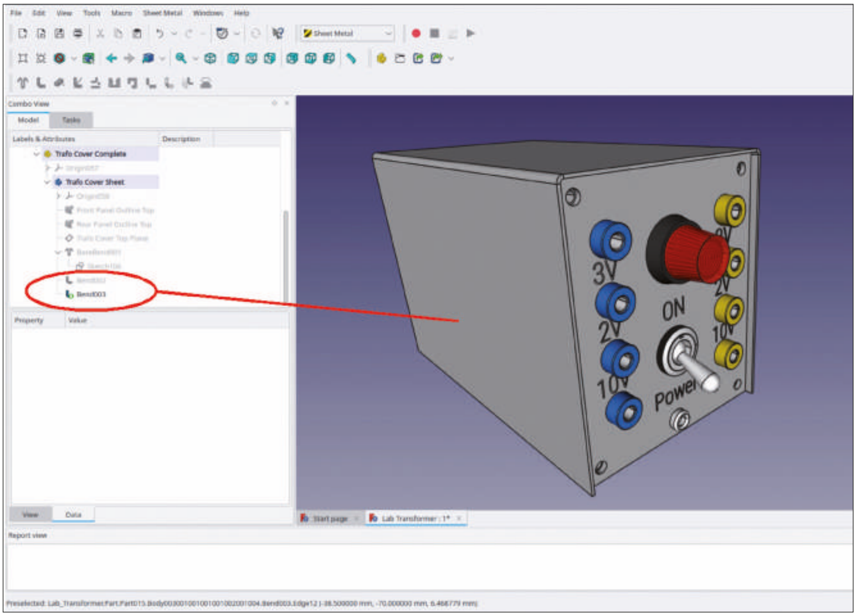


Figure 9-118

21. Switch back into the 'Part Design' workbench.

Before the fillets for the lower corners of the cover are added, all other features of the cover should have been created — in order to lower the danger of facet re-enumerations. Near the top of the cover, rows of vent holes could allow air circulation from below.

22. Consider sketching the holes onto the YZ plane of the cover and creating symmetric pockets from there through the whole part, which saves the effort to define additional planes: Start the sketcher and select the YZ plane as the sketching plane. If the present 3D geometry is not visible, close and reopen the sketch.

23. From the main menu, select 'Sketch | View section' and 'View | Orthographic view' (Figure 9-119).

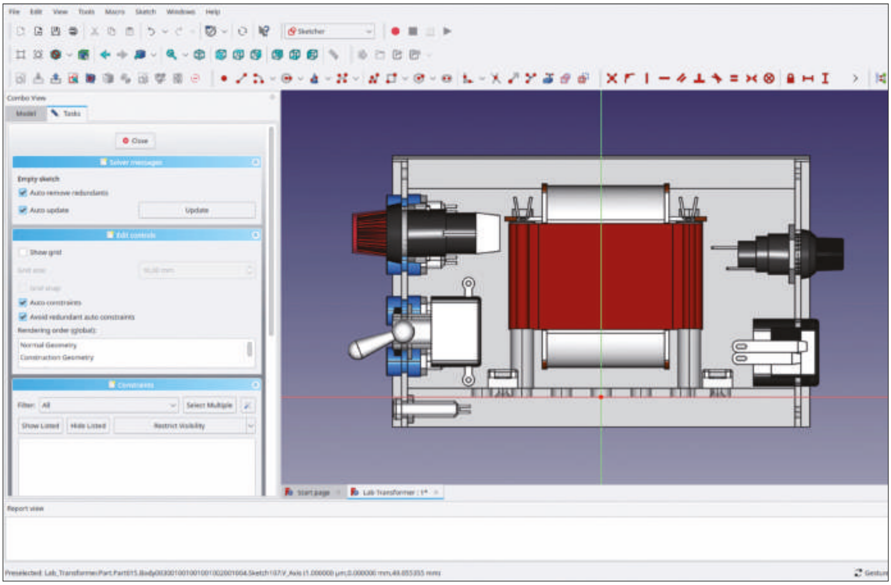


Figure 9-119

24. Draw a circle of 7 mm diameter. Constrain the horizontal distance to the center line to 40 mm. Constrain the vertical distance from the origin to 60 mm (Figure 9-120). Close the sketch.

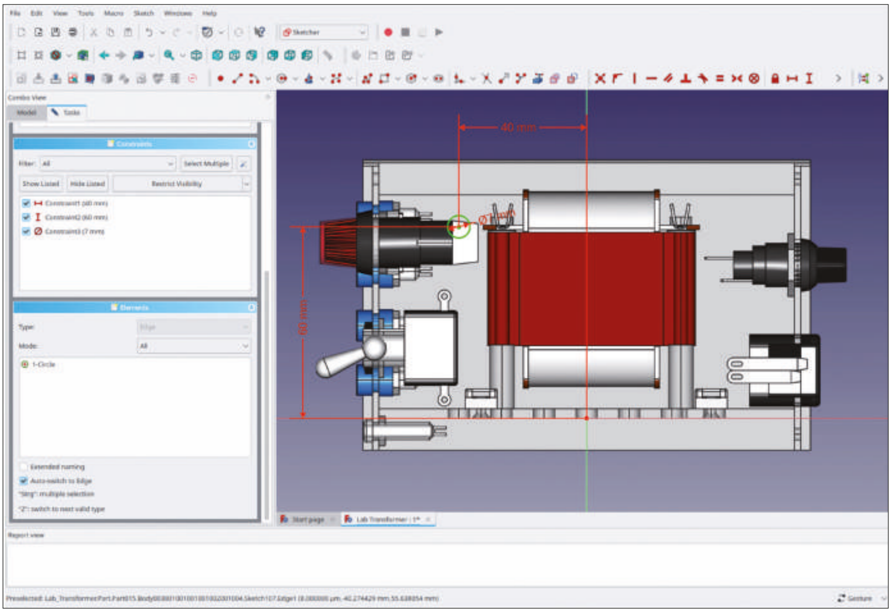


Figure 9-120

25. With the new sketch marked in the tree view, click the 'Pocket' tool button. In the task window, select for the type 'Through all' and check the 'Symmetric to plane' checkbox (Figure 9-121). This creates a hole in both sidewalls at the same time. Close the task window with the 'OK' button.

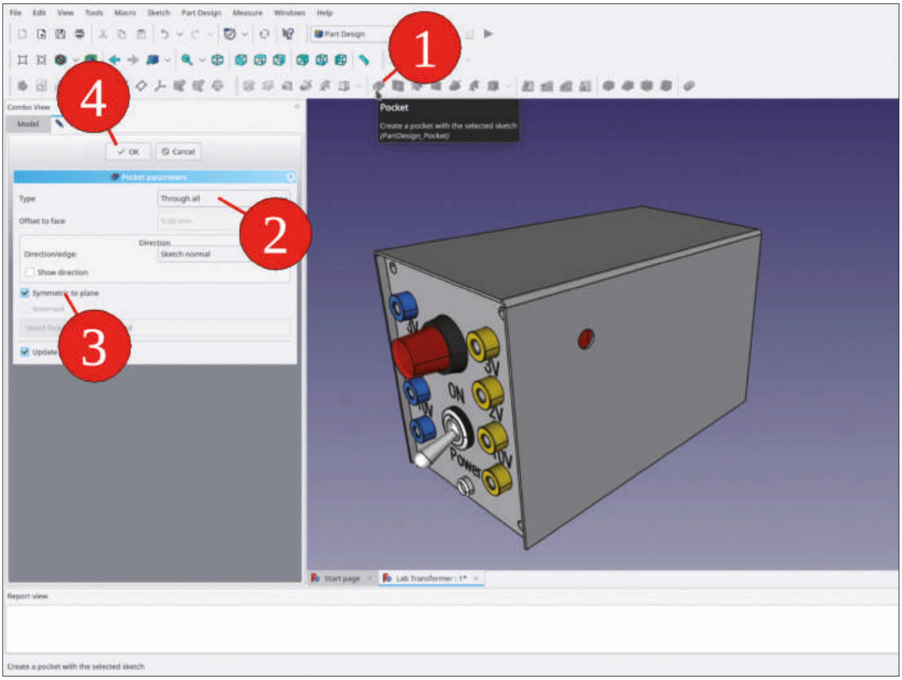


Figure 9-121

26. In the tree view, mark the last design state of the cover. Then, select from the main menu 'Part Design | Apply a pattern | LinearPattern' (Figure 9-122).

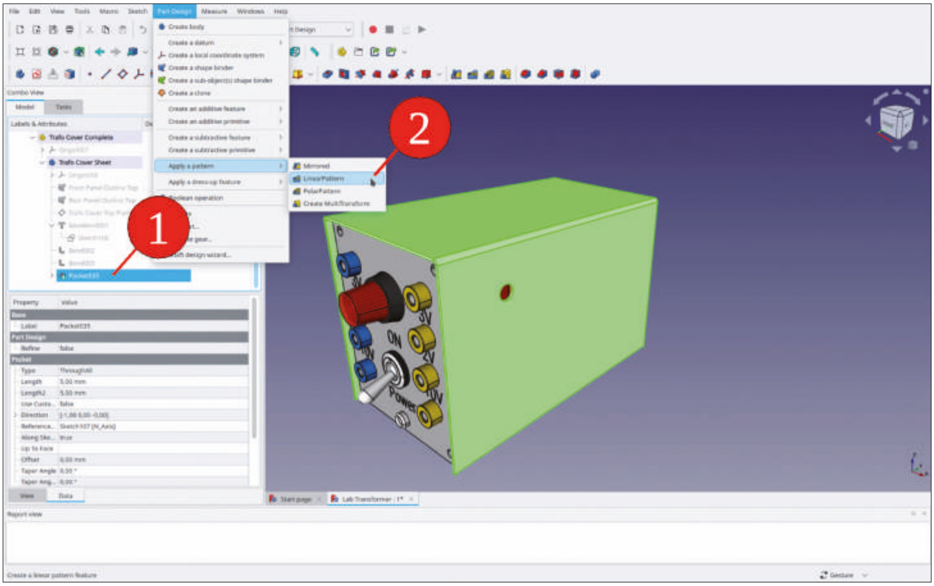


Figure 9-122

27. In the task window, set the length to 80 mm, and the number of occurrences to 7 (Figure 9-123). If you apply a linear pattern and it is not being shown as expected, try to select another axis from the 'Direction' control.

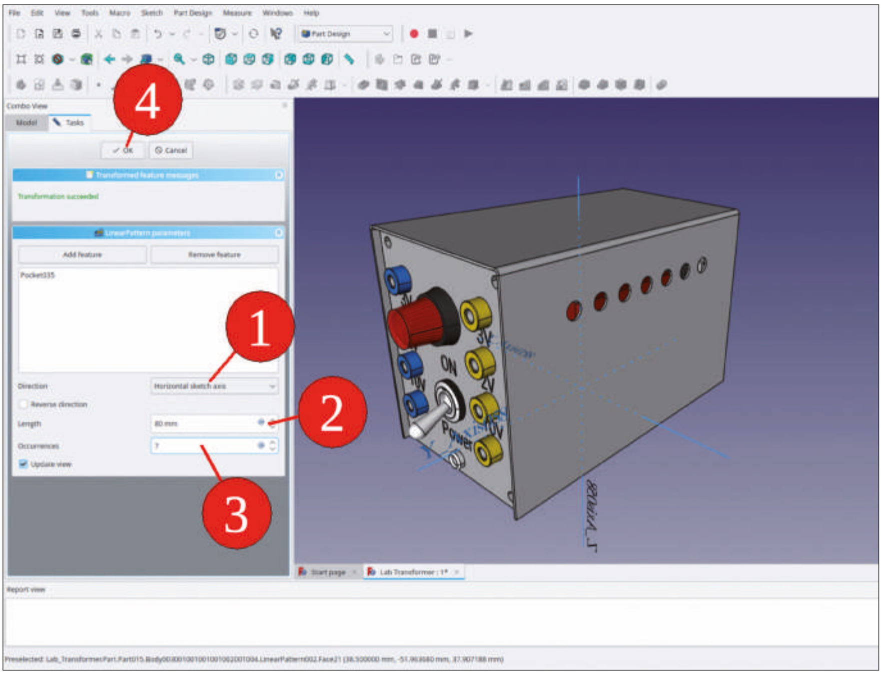


Figure 9-123

The bolt holes for the attachment of the cover are still missing. Which element should control the position of these holes requires a worthwhile thought. The bolts are ending in the lower chassis spacers, in which threaded holes for them will be cut. When thinking about the manufacturing of the spacers, it seems important that the threaded holes are positioned in the center of the square profiles, and not wander out of the profile cross-section by some other dimension changes. Therefore, you start to define the threaded holes in the chassis spacers first, and then create a reference to them in the cover design.

28. In the tree view, hide the Std-Part-Containers 'Trafo Cover Complete', 'Trafo Front Panel Complete' and 'Trafo Rear Panel Complete' with the SPACE key.

29. In the Std-Part-Container 'Trafo Chassis Complete', activate the body 'Chassis Spacer 001' with a double click.

30. In the Std-Part-Container 'Trafo Chassis Complete'. Hide the other spacers.

31. Start the sketcher and select the YZ plane as the sketching plane. From the main menu, select 'View | Orthographic view'.

32. Click on the 'External geometry' tool button and mark both end faces of the spacer (Figure 9-124).

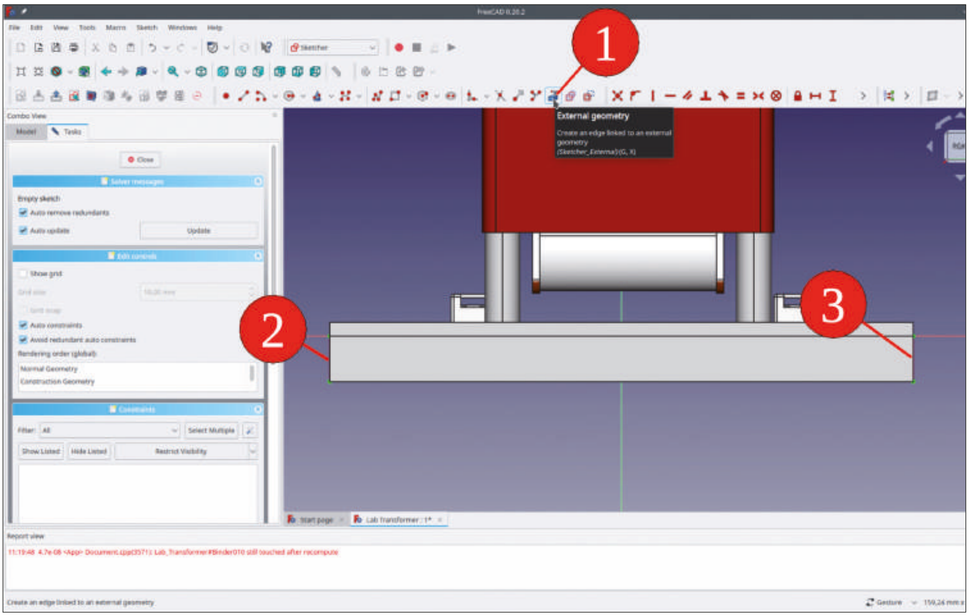


Figure 9-124

33. From the main menu, select 'Sketch | View section'. Draw two circles. In the 'Elements' list, mark the two circles and click the 'Constrain equal' tool button. Mark the two centers and constrain their vertical distance to 0 (Figure 125).

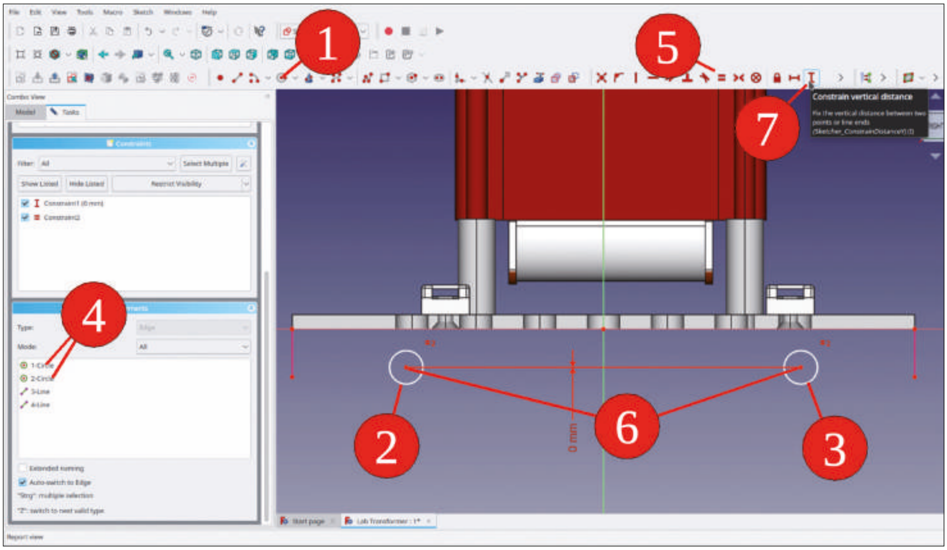


Figure 9-125

34. Mark one of the circle centers and one endpoint of the closest construction line. Constrain the horizontal distance to 20 mm. Repeat that for the opposite circle (Figure 9-126).

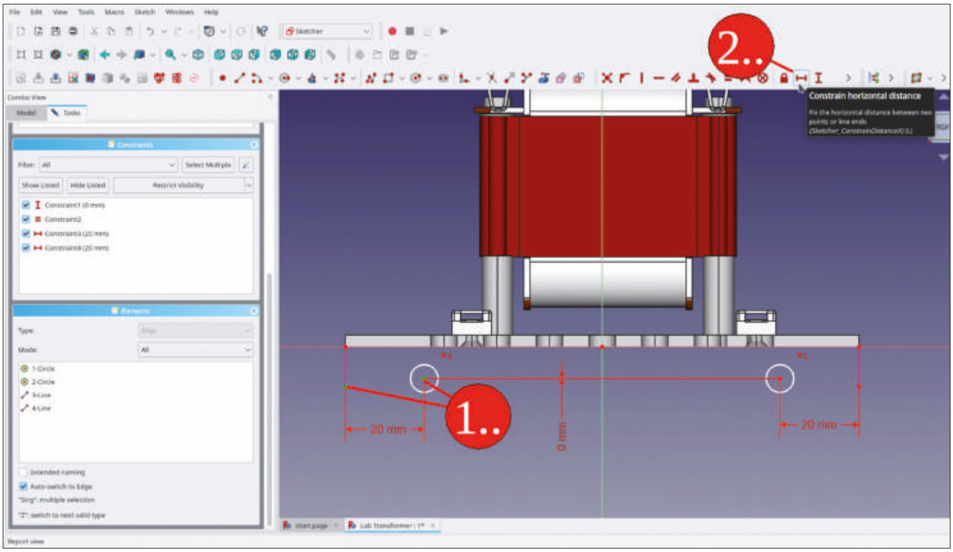


Figure 9-126

35. Draw a point (Figure 9-127). Mark the point and the closest construction line. Then, click the 'Constrain symmetrical' tool button (Figure 9-128).

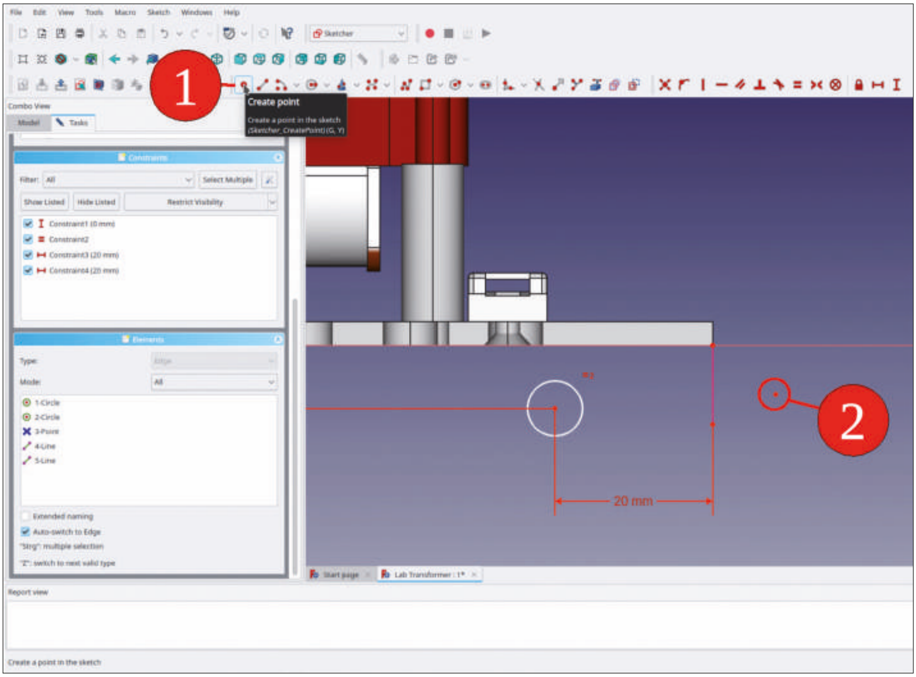


Figure 9-127

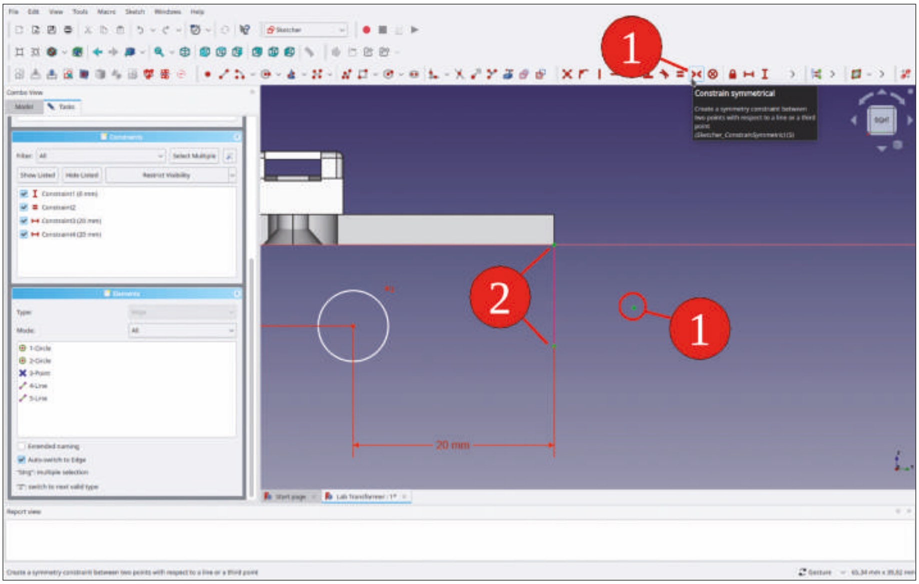


Figure 9-128

36. Mark the new point and one of the circle centers. Constrain the vertical distance to zero (Figure 9-129).

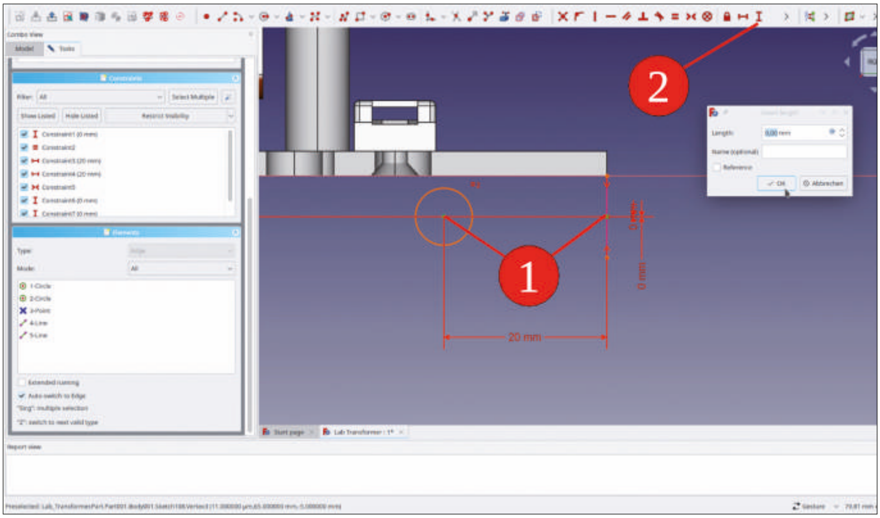


Figure 9-129

37. In the 'Elements' list, right-click one of the circles. Select 'Diameter Constrain' from the context menu and enter a value of 2.5 mm (Figure 9-130). Close the sketch.

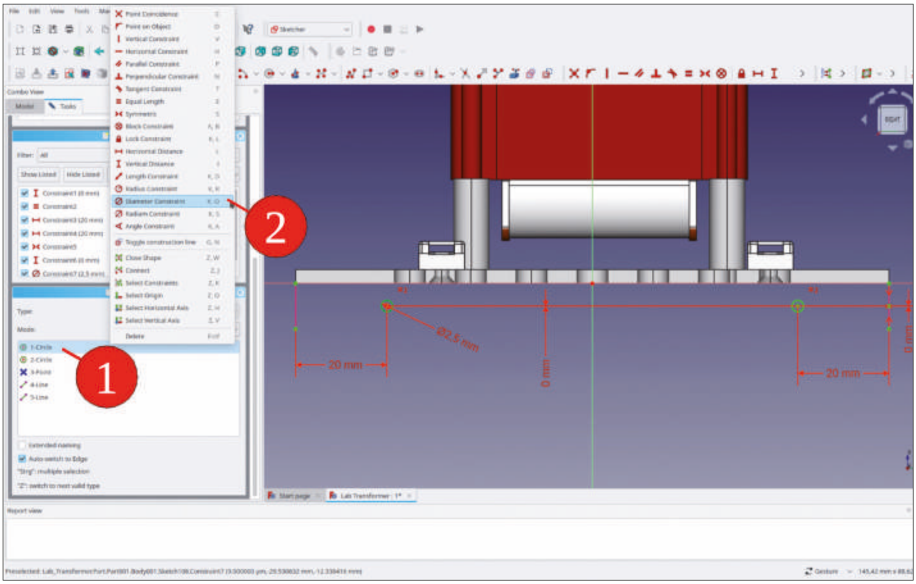


Figure 9-130

38. In the tree view, mark the new sketch and click the 'Hole' tool button. In the task window, select 'ISO metric regular' for the profile and check the 'Threaded' checkbox. For the size, select M3. Set the depth to 'Through all' and check the 'Reversed' checkbox (Figure 9-131). Close the task window with the 'OK' button.

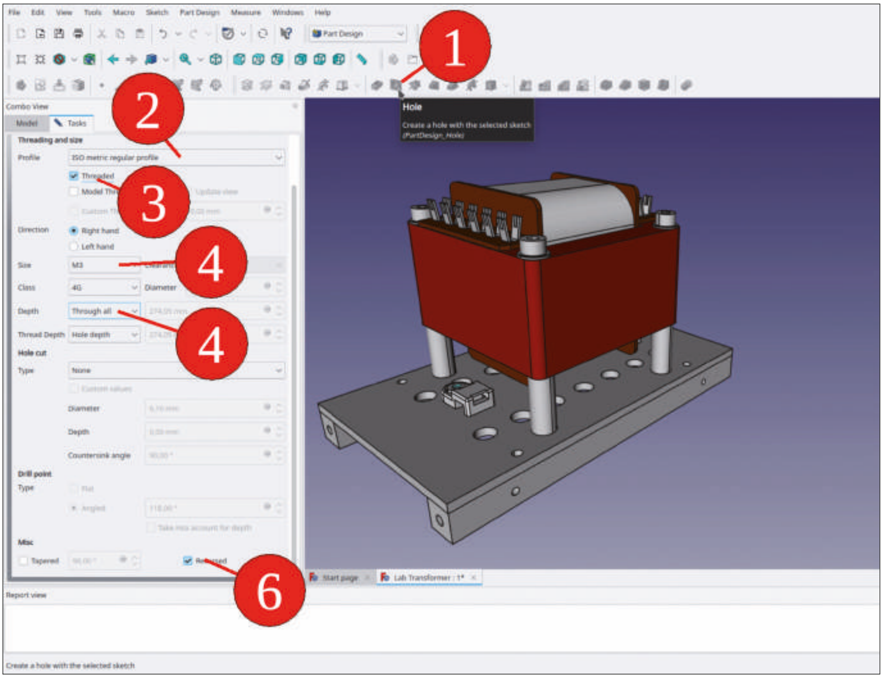


Figure 9-131

39. In the tree view, display all the linked chassis spacers again with the SPACE key. The holes have been propagated to all of them.

40. When you hover over 'Chassis Spacer 001' in the tree view, the 3D view updates with a highlighted, transparent representation of the spacer. With this view, you can quickly check whether any of the holes collide with each other (Figure 9-132).

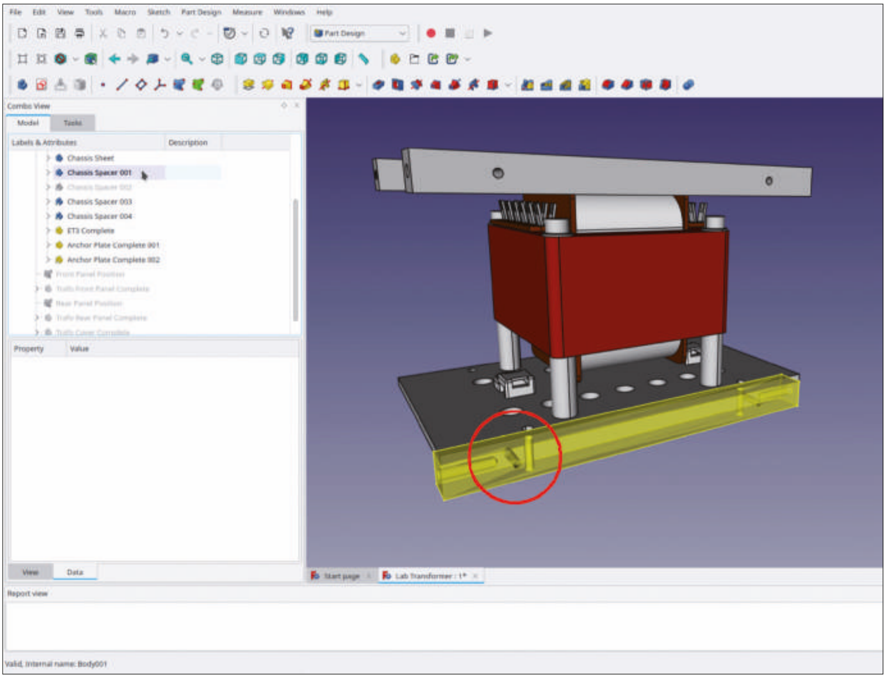


Figure 9-132

After these preparations, you can create the mounting holes in the cover. For the associative reference, shape binders can be used:

41. In the tree view, display the Std-Part-Container 'Trafo Cover Complete' with the SPACE key. Expand it and activate the body 'Trafo Cover Sheet' by a double click, in order to set the destination for the new shape binders.

42. In the 3D view, navigate to the last design state of 'Chassis Spacer 001' (Hole…), and expand that to expose the sketch. Mark the sketch with a click (Figure 9-133).

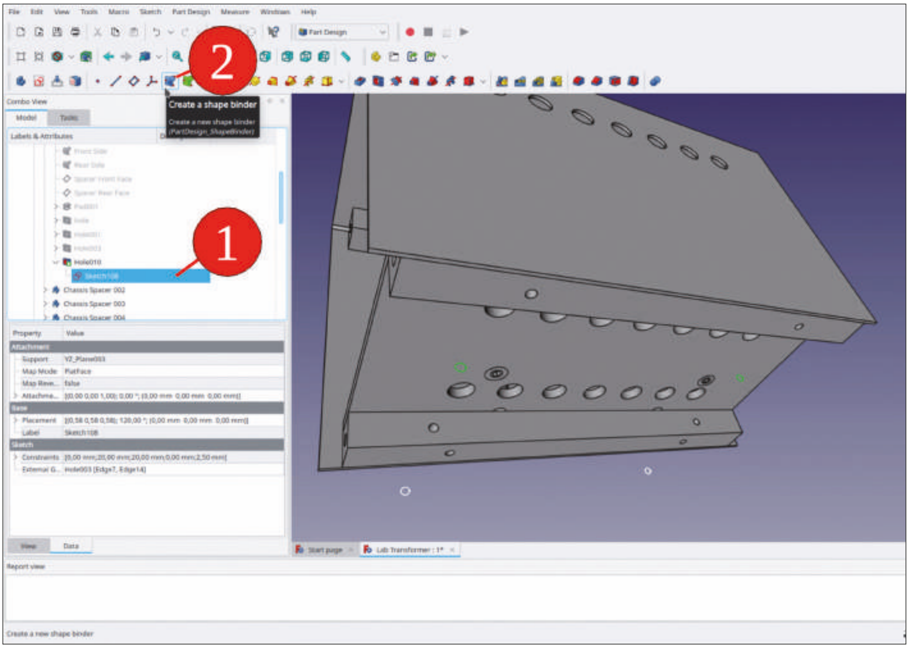


Figure 9-133

43. Click on the blue 'Create a shape binder' tool button. Thanks to our prior selection, the sketch is already preset. Close the task with the OK button.

44. Scroll down and locate the new shape binder in 'Trafo Cover Sheet'. Rename it to 'Footprint Chassis Spacers'.

45. With the new shape binder marked, click the 'Hole' tool button.

46. In the task window, set the diameter to 3.2 mm and select 'Through all' for the depth. Check the 'Reversed' checkbox (Figure 9-134). Close the task window with the OK button.

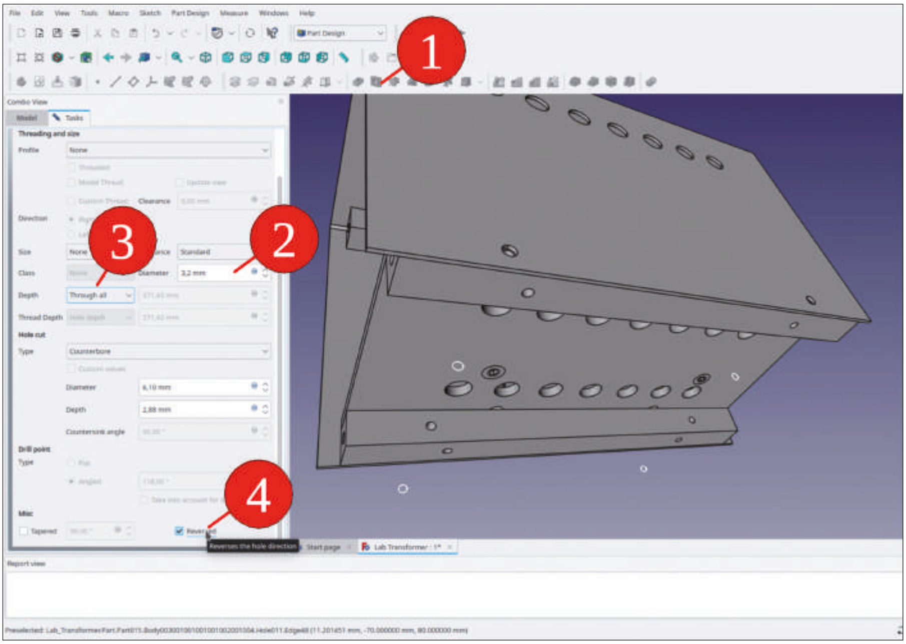


Figure 9-134

47. In the tree view, expand the last design state of 'Trafo Cover Sheet' to expose the shape binder. Mark the shape binder with a click, and then click again on the 'Hole' tool button.

48. In the task window, repeat the entries described in step 6 above, but do not check the 'Reversed' checkbox (Figure 9-135). Close the task window with the OK button.

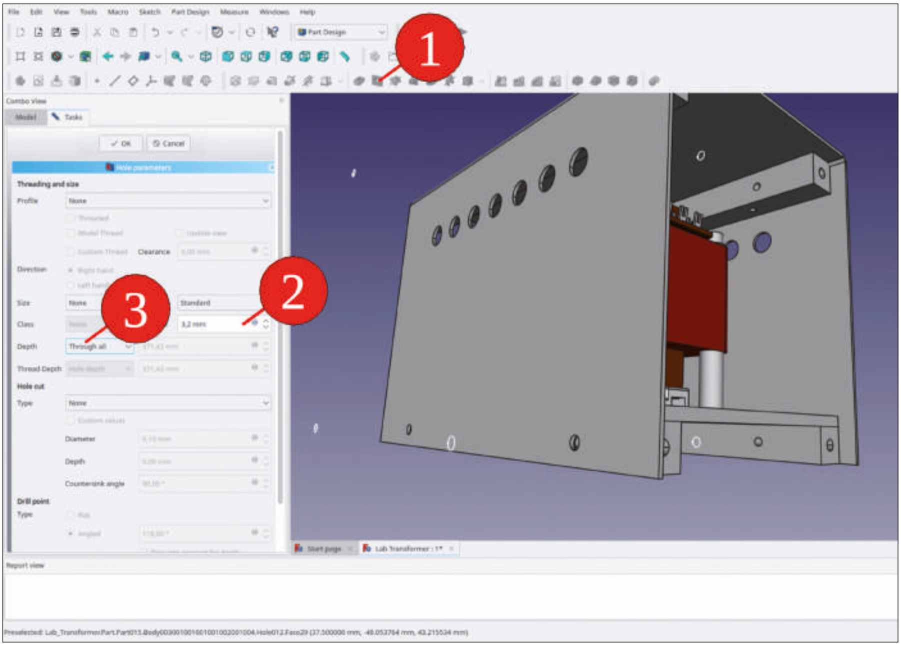


Figure 9-135

49. In 'Chassis Spacer 001', hide the sketch in the tip again with the SPACE key. Display the two panels again.

The cover is nearly completed. Now, cosmetic features can be applied. A fillet on the lower corners of the cover saves ugly scratches on the workbench later on. These fillets are attached to facets! We create them in the hope, that this is the last addition, and re-enumeration is not likely to occur with the cover…

50. In the 3D view, mark the four lower edges of the cover sheet (Figure 9-136). Then, click the 'Fillet' tool button. In the task window, set the radius to 3 mm (Figure 9-137).

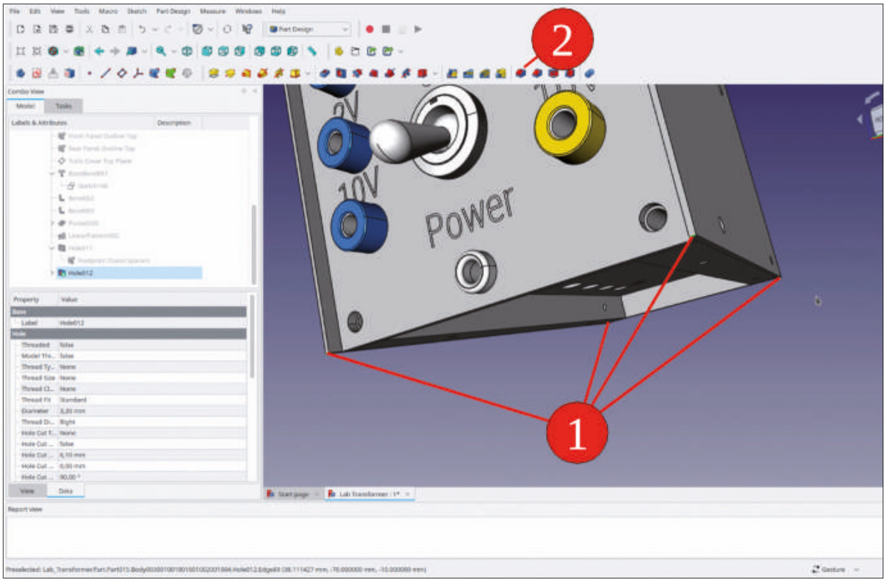


Figure 9-136

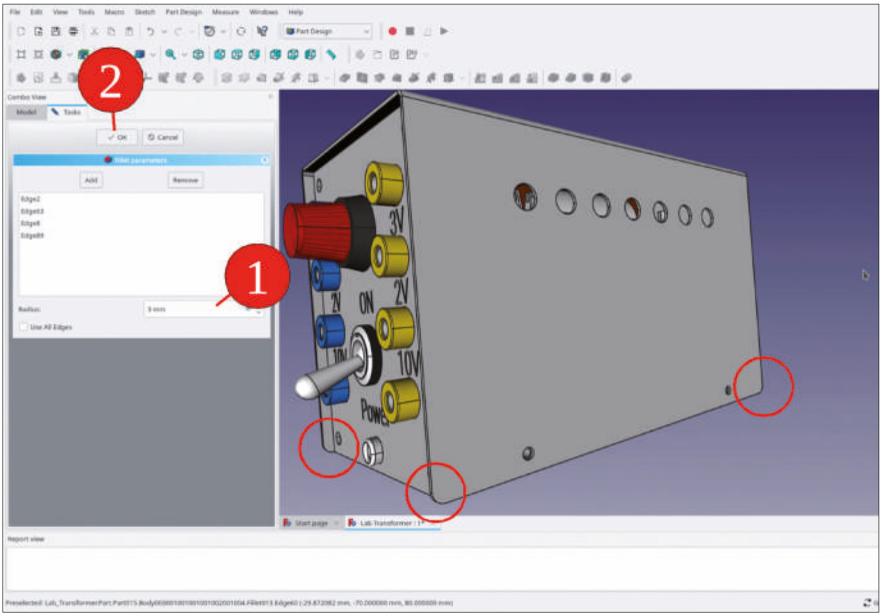


Figure 9-137

The parts of the apparatus are now readily designed, at least to the point where you could start to actually manufacture the parts. If you want to test whether the associativity works, you can, e.g., move the front panel 20 mm towards the front, changing the attachment parameters of the datum plane 'Position Front Panel' in the body 'Chassis Sheet' (Figure 9-138, 9-139).

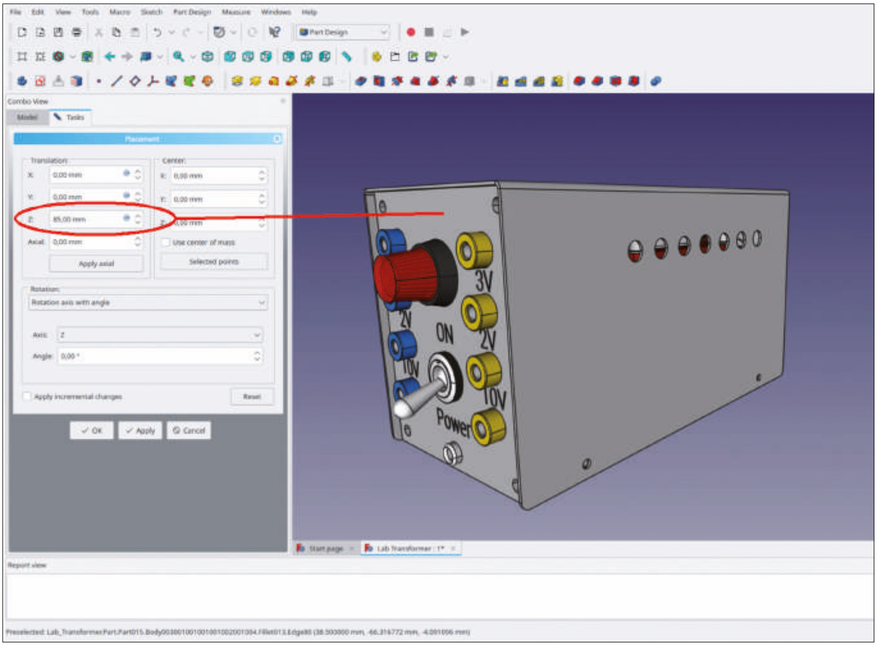


Figure 9-138

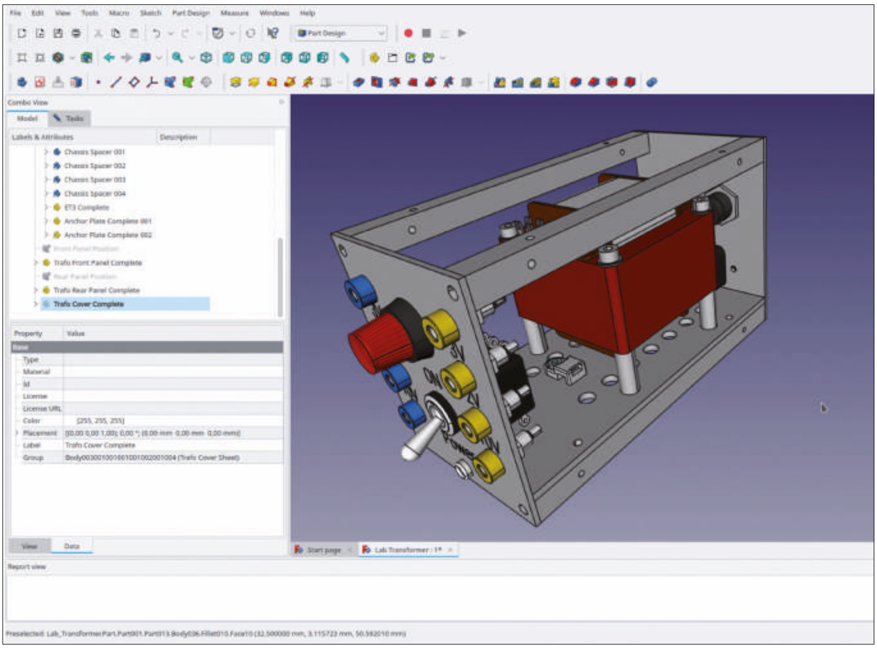


Figure. 9-139

You could also try to make the front panel a bit wider. In order to do this, you need to edit the sketch of the first state of the front panel ('Pad…'). In the following example, the width was set to 85 mm (Figure 9-140). The back panel does not follow, because it brought its own dimensions (Figure 9-141). Maybe you can change that with a suitable reference later, on your own.

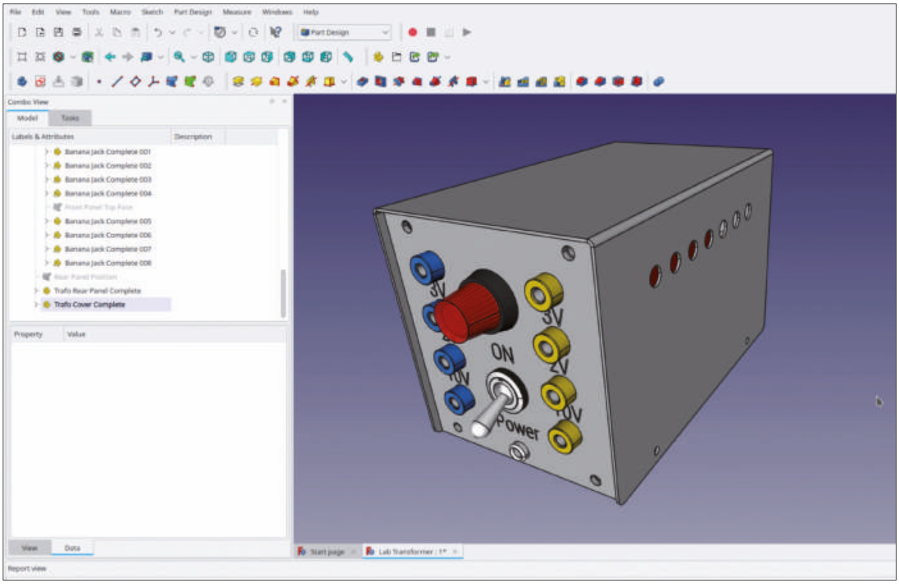


Figure 9-140

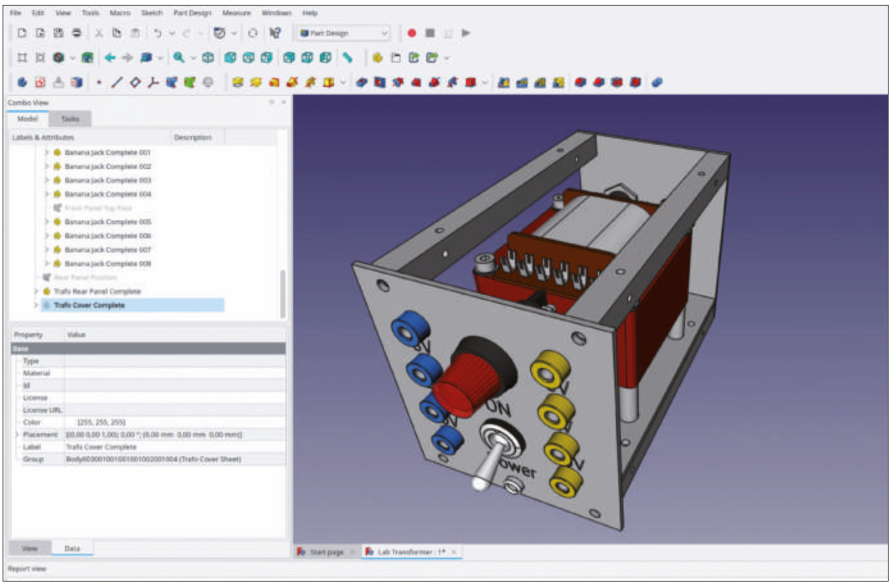


Figure 9-141

After a parameter has been changed, the assembly needs to be updated with a recalculation (F5 key). This already takes some time. Now, you can add the hardware to bolt the transformer together.

9.8 Where do the Fasteners Belong?

In the past chapters, as well as in the examples described in the Appendices, the fasteners are packed into the Std-Part-Containers of the objects, which are mounted with that hardware. Especially with parts that are used multiple times, this is a time-saving practice.

In the current project, you've also generated parts dynamically, like the cover or the chassis spacers. With those parts, the positions of the bolt holes were created using shape binders referencing other parts, in which special requirements for the positioning of the holes exist. The fasteners could simply be attached to the edges of the resulting bolt holes, e.g., in the cover. However, this would reference their position to facet edges, and thus expose these attachments to the risk of displacement by re-enumeration. If the screws are regarded as the very last, cosmetic addition of an assembly, that would probably be still tolerable.

However, it is preferable to attach the fasteners to more robust references. These exist in the form of the sketches, with which the hole positions were defined in the first place. For the cover, the hole positions were defined in the chassis spacers, in order to constrain those to the spacer profile centers. From the defining sketch, two more derived shape binders will help to position the screws.

9.8.1 Fasteners for the Cover

1. In 'Trafo Chassis Complete', expand the body 'Chassis Spacer 001' and display the sketch in the tip (Hole010) with the SPACE key (Figure 9-142).

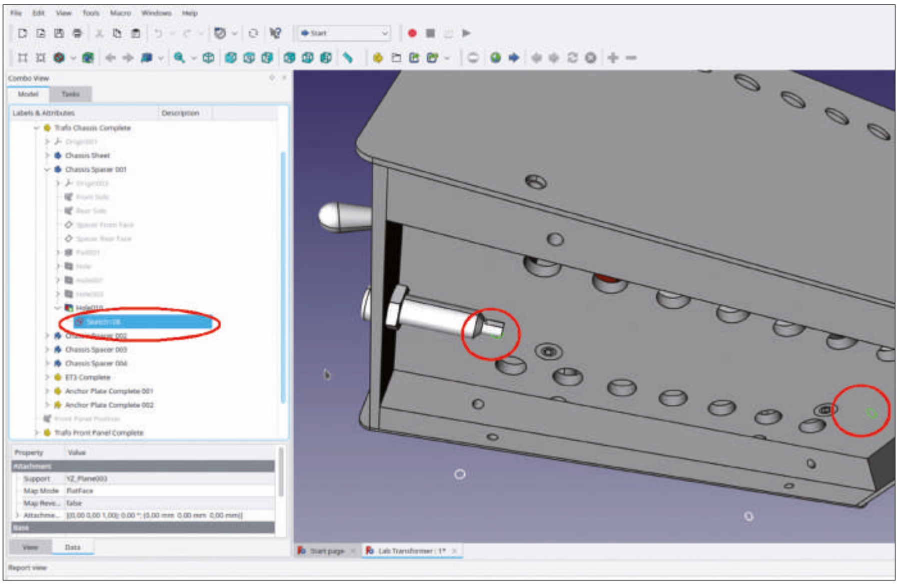


Figure 9-142

2. In the Std-Part-Container 'Trafo Cover Complete', activate the body 'Trafo Cover Sheet' to set it as the destination for the new shape binder.

3. Click on the blue 'Create a shape binder' tool button. In the task window, click the 'Add Geometry' button, which then turns dark gray, to indicate the collector is activated. In the 3D view, mark the circle of the sketch, which is located close to the front panel (Figure 9-143). Close the task window with the 'OK' button.

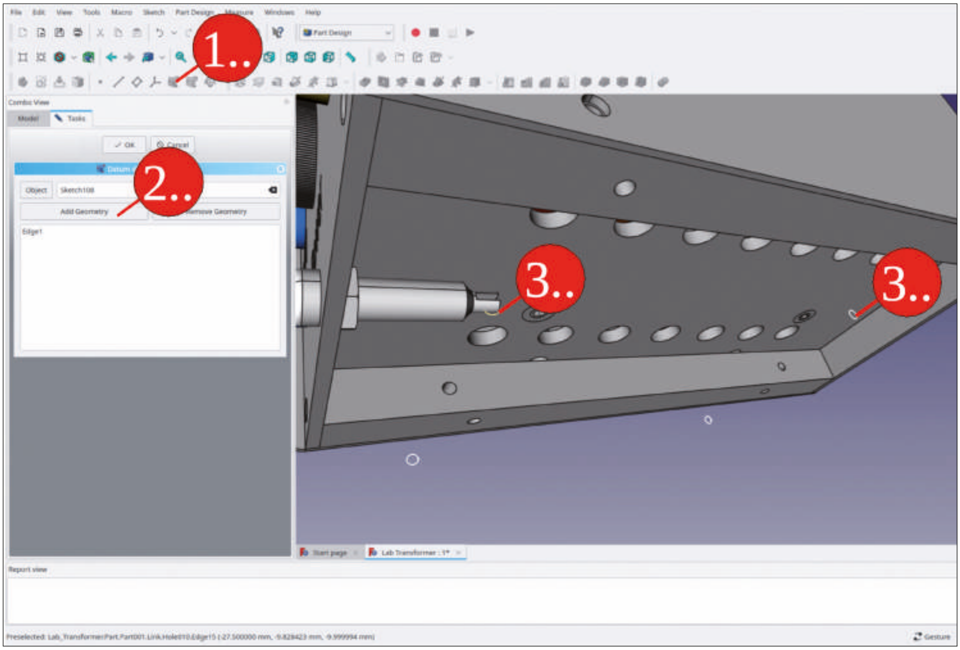


Figure 9-143

4. In the same way, create a shape binder for the rear circle of the sketch.

5. Scroll down to the 'Trafo Cover Sheet' body and rename the new shape binders to 'Mount Hole Positions Front' and 'Mount Hole Positions Rear' (Figure 9-144). Set the property 'Trace support' for both shape binders to 'true'.

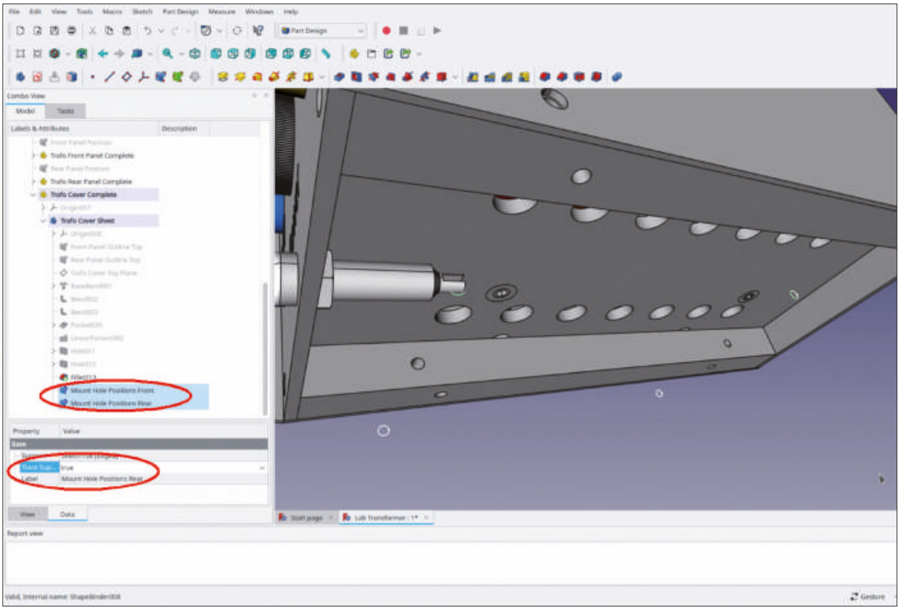


Figure 9-144

6. Drag-and-drop the two shape binders into the Std-Part-Container 'Trafo Cover Complete'. Hide the sketch in 'Chassis Spacer 001' (Figure 9-145).

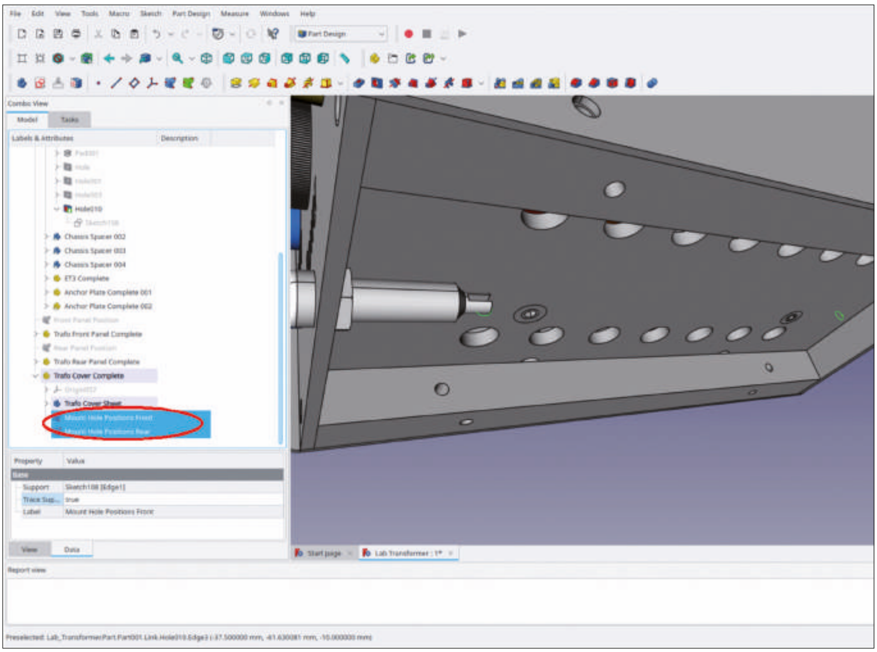


Figure 9-145

7. Now, you add the screws: From the main menu, select 'Macro | Recent Macros | start_bolts'. From the BOLTS Parts selector, select 'Standard | DIN | DIN7991'. Leave the key at M3 and set the length to 10 mm. Click the 'Add part' button 4 times to add all the screws at once.

8. In the tree view, mark all the new screw objects. Drag-and-drop them into the Std-Part-Container 'Trafo Cover Complete'.

9. Now, you create the attachments. Switch to the 'Part' workbench. Mark the first screw object. From the main menu, select 'Part | Attachment…'.

10. In the task window, click into the edit field next to the button for reference 1, which is dark gray and reads 'Selecting…'. Then, switch to the 'Model' tab, and click the shape binder 'Mount Hole Positions Front'. Return to the 'Tasks' tab.

11. In the task window, select 'Concentric' as the attachment mode. Set the attachment offset Z to -38.51 mm and check the 'Flip sides' checkbox. (The additional offset of 0.01 mm to the border of the casing helps to display the screw more clearly, Figure 9-146). Close the task window with the 'OK' button.

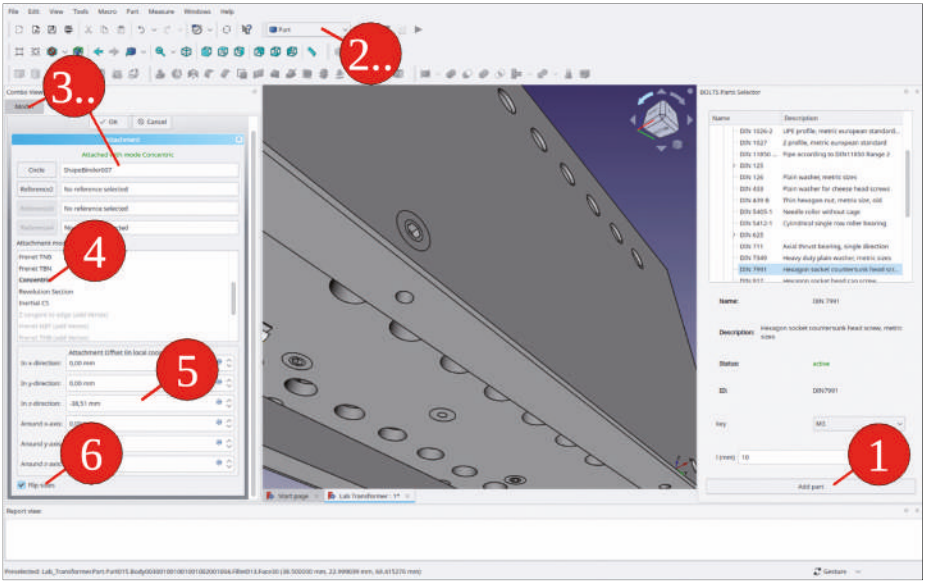


Figure 9-146

12. Attach the other screws in a similar way, also using the shape binder 'Mount Hole Positions Rear'. For the screws on the opposite side of the casing, do not check the 'Flip sides' checkbox.

13. In the body 'Trafo Cover Sheet', hide the shape binders 'Mount Hole Positions Front' and 'Mount Hole Positions Rear'.

9.8.2 Fasteners for the Chassis

1. In the tree view, hide all Std-Part-Containers except 'Trafo Chassis Complete'. In 'Trafo Chassis Complete', hide all objects except 'Trafo Chassis Sheet', and in that body, also hide the tip. Locate and display only the sketch used to create the four mount holes (Figure 9-147).

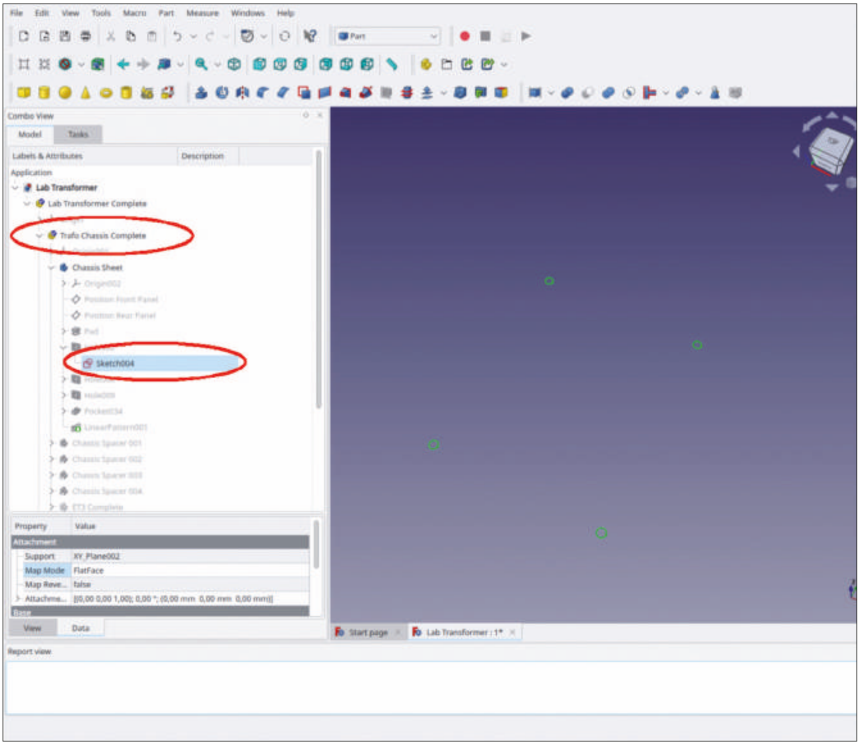


Figure 9-147

2. In the tree view, activate the body 'Trafo Chassis Sheet' with a double click.

3. One by one, mark each circle of the sketch and then click the blue 'Create a shape binder' tool button. Do not change the preset values and close the task window with the 'OK' button.

4. Rename the four new shape binders to 'Bolt Hole 1' to 'Bolt Hole 4'. Set the trace support for all of them to 'true'.

5. Drag-and-drop the four new shape binders into the Std-Part-Container 'Trafo Chassis Complete' (Figure 9-148). Display the objects 'Chassis Spacer 001', 'Chassis Spacer 002' and the tip in 'Chassis Sheet' body again, with the SPACE key.

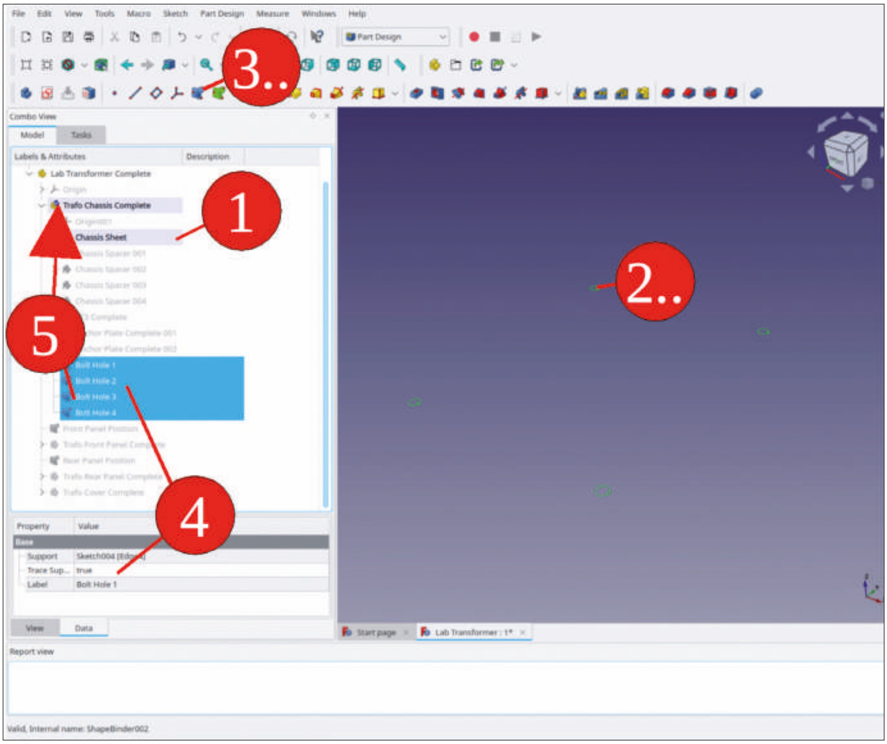


Figure 9-148

6. With the 'BOLTS Parts Selector', add four screws, type DIN 7991, length 10 mm, to the design. Drag-and-drop them into 'Trafo Chassis Complete'.

7. Switch to the 'Part' workbench. Again, mark each screw and select 'Part | Attachment…' from the main menu. As described in the previous section, attach the first screw to one of the new shape binders, selecting the attachment mode 'Concentric', an attachment offset of -3.01 mm (Figure 9-149) and checking the 'Flip side' checkbox.

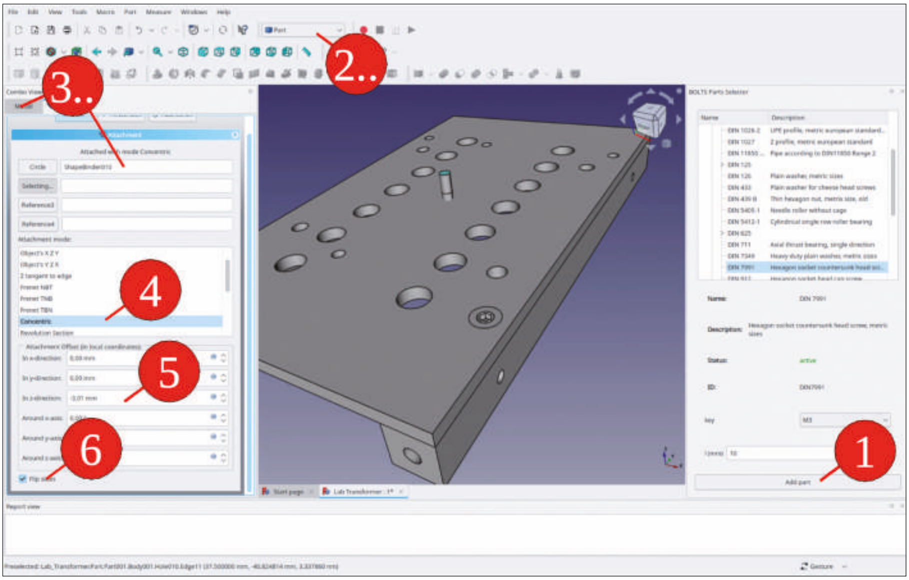


Figure 9-149

8. In the same way, attach the remaining screws to the other shape binders (Figure 9-150).

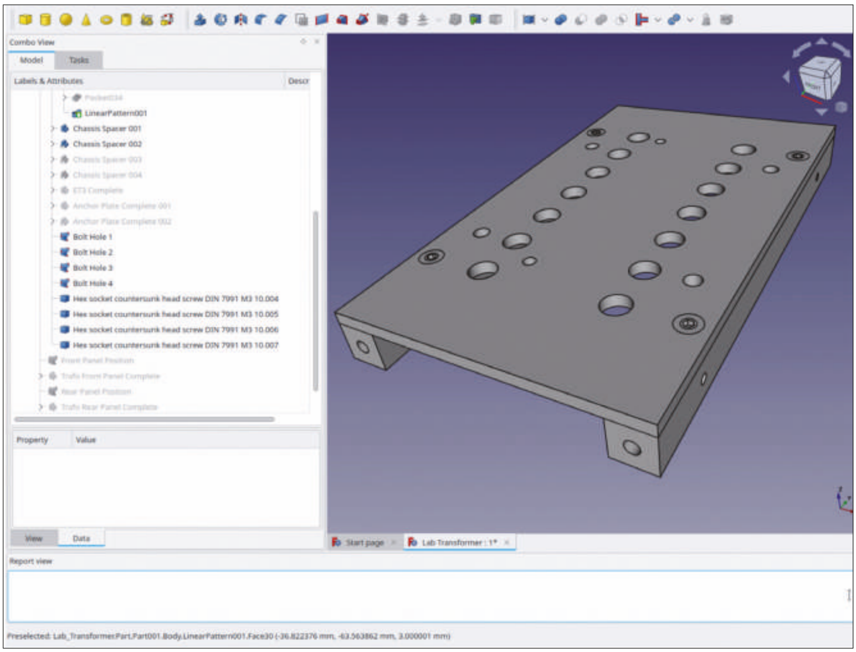


Figure 9-150

9. Display all previously hidden objects again with the SPACE key.

9.8.3 Fasteners for Front and Rear Panel

The defining objects for the mount holes in the front and rear panel are the chassis spacers. For this reason, it would now be practical to add the screws to the first spacer (all others are link copies of the first one and would therefore follow automatically). However, expanding the tree of 'Trafo Chassis Complete' now reveals a weakness of our selected project structure: The spacers are only defined as body objects, and it is therefore not possible to add the fasteners to just one of them. One solution is to add the eight screws manually to the enclosing Std-Part-Container 'Trafo Chassis Complete'. Because this is an instruction book, you may find it fitting to correct the structural weakness of the project in a few more steps. The following would need to be undertaken: The link copies of the chassis spacer are deleted. A new Std-Part-Container is created in order to house the spacer and the fasteners. From this, you will create new link copies. Then the references for the holes in the front and back panel need to be restored:

1. In the tree view, hide 'Trafo Front Panel Complete' and 'Trafo Rear Panel Complete' with the SPACE key.

2. Create a new Std-Part-Container and rename it to 'Chassis Spacer Complete 001'.

3. Drag-and-drop it into 'Trafo Chassis Complete'.

4. Drag-and-drop the body 'Chassis Spacer 001' into the new Std-Part-Container.

5. Delete the link copies 'Chassis Spacer 002' through 'Chassis Spacer 004'.

6. Double-click the body 'Chassis Spacer 001' to activate it (Figure 9-151) and set it as the destination for new shape binders.

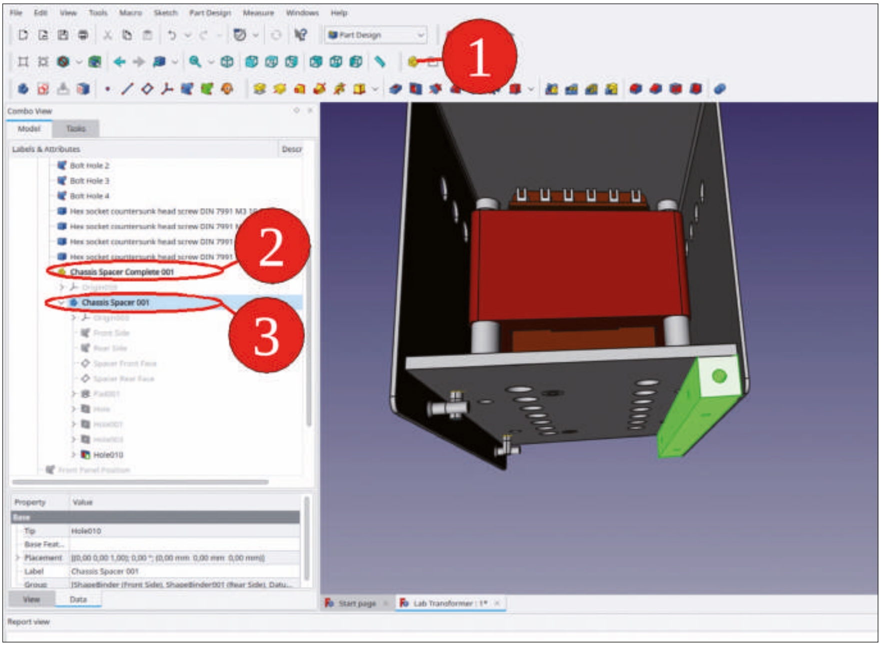


Figure 9-151

7. In the tree view, display the two sketches for the holes in the spacer end faces (Figure 9-152). Hide the tip of the body 'Chassis Spacer 001' (designated as 'Hole010').

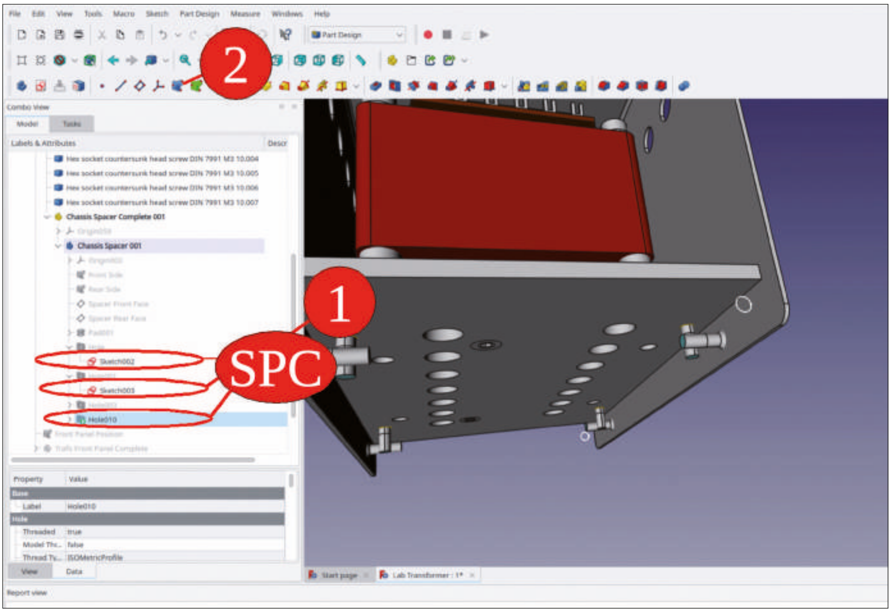


Figure 9-152

8. Click on the blue 'Create a shape binder' tool button. In the task window, click the 'Add Geometry' button, which turns dark gray.

9. Click on the circle of the front-side sketch. It is important that the exact edge is listed in the task window, not only the sketch object (Figure 9-153). Close the task with the 'OK' button.

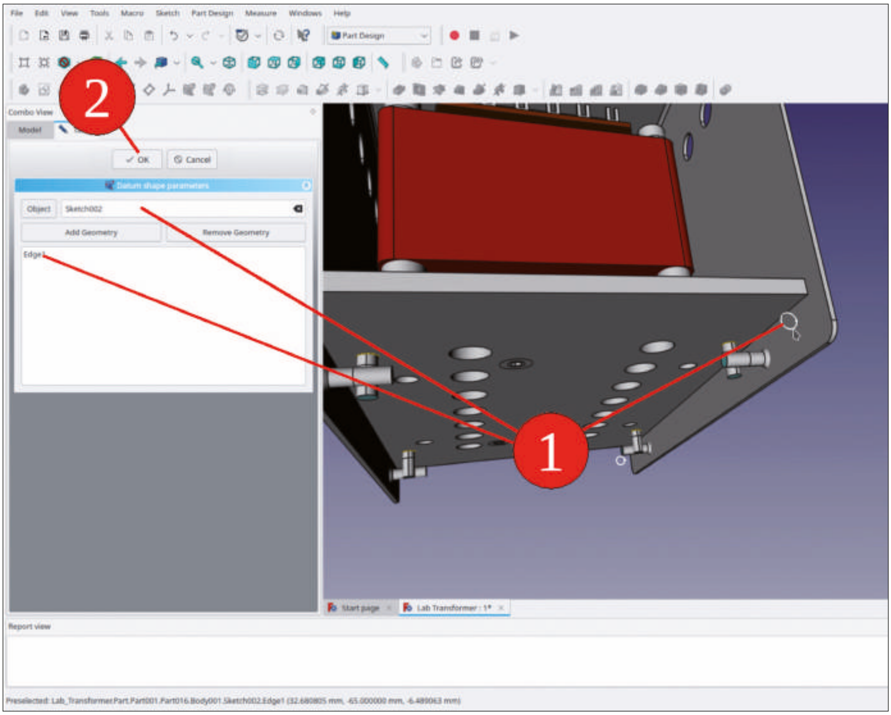


Figure 9-153

10. Rename the new shape binder to 'Bolt Hole Front Face'.

11. In the analog way, create a shape binder for the rear sketch, and rename it to 'Bolt Hole Rear Face'. Take care when clicking the blue 'Create a shape binder' tool button. If the other shape binder is still marked in the tree view (because you just renamed it), the task window of that shape binder is reopened, instead of the creation of a new one.

12. Set the property 'Trace Support' for both new shape binders to 'true'. Drag-and-drop both shape binders into the Std-Part-Container 'Chassis Spacer Complete 001' (Figure 9-154).

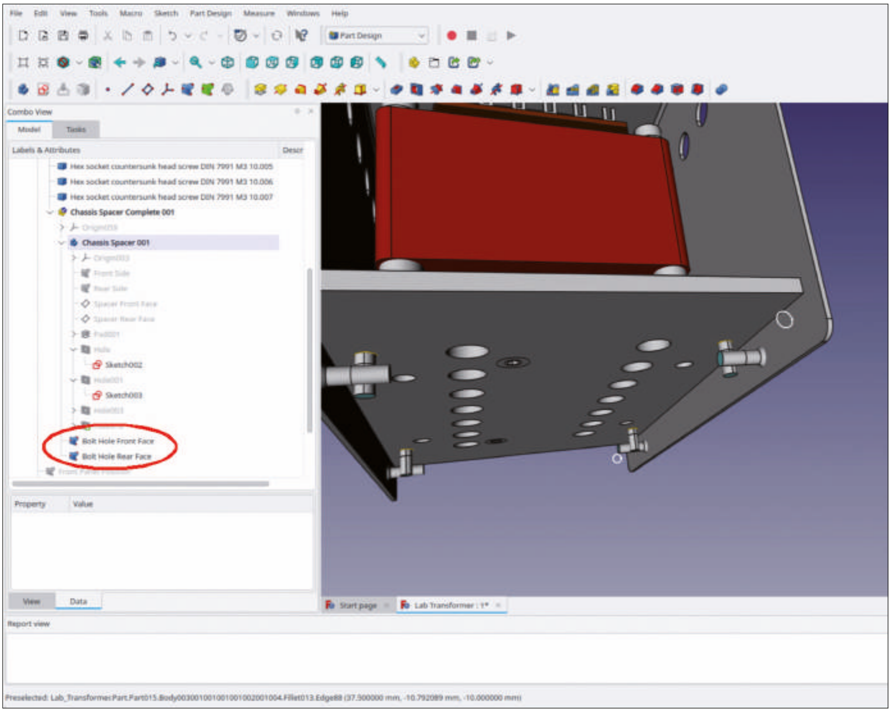


Figure 9-154

The shape binders will be needed to attach the screws. We leave the related sketches still set as visible, they will be needed to define the holes in the panels later.

13. Select the macro 'start_bolts', and add two screws DIN 7991, size M4, length 10 mm to the assembly.

14. Drag-and-drop the new screws into the Std-Part-Container 'Chassis Spacer Complete 001' (Figure 9-155).

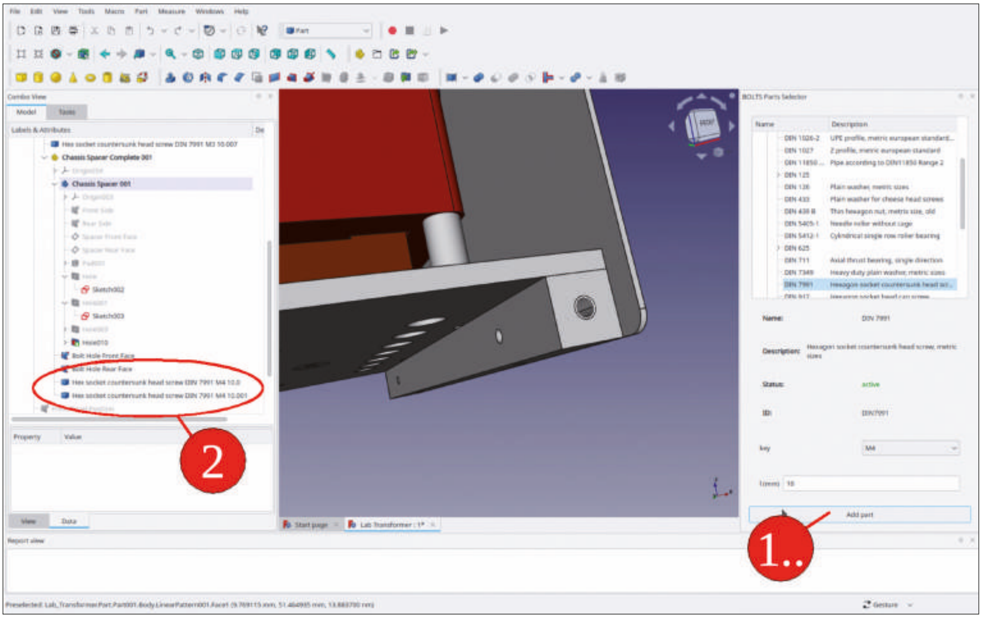


Figure 9-155

15. In the tree view, display the tip of the body 'Chassis Spacer 001' with the SPACE key. Mark the first screw and switch to the 'Part' workbench. From the main menu, select 'Part | Attachment…'.

16. For the selection of reference 1, switch to the 'Model' tab and select the shape binder 'Bolt Hole Front Face'. Switch back to the 'Tasks' tab. For the attachment mode, select 'Concentric', enter a value of -2.01 mm for the attachment offset Z and check the 'Flip sides' checkbox (Figure 9-156). Close the task window with the 'OK' button.

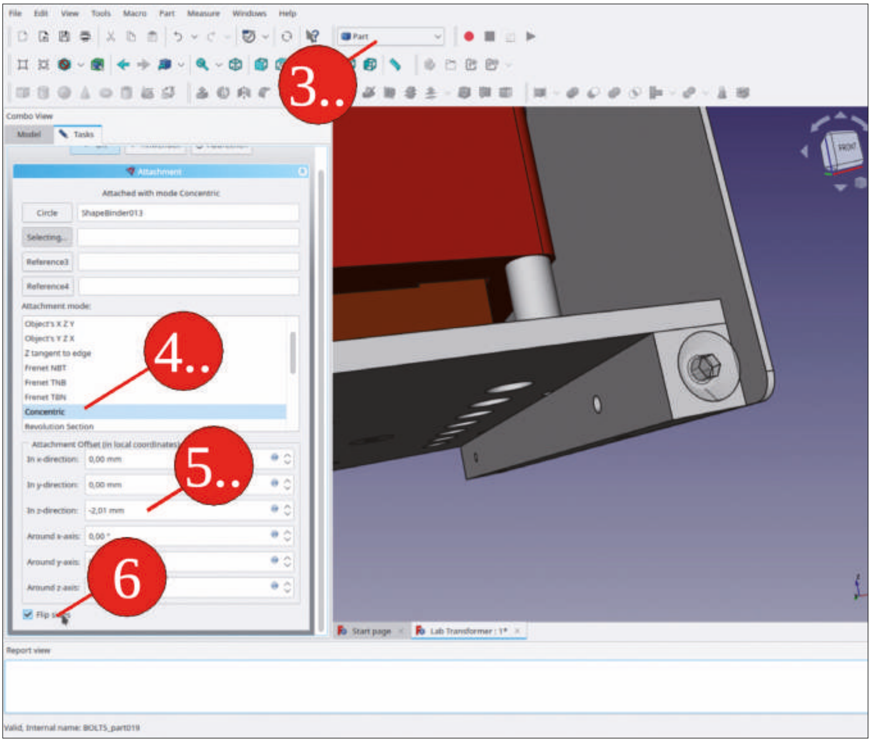


Figure 9-156

17. In the similar way, attach the other screw to the rear face shape binder. There, you do not need to check the 'Flip sides' checkbox.

18. Using the 'Make link' tool button, create 3 links of the Std-Part-Container 'Chassis Spacer 001'.

19. Drag-and-drop the link objects into the Std-Part-Container 'Trafo Chassis Complete' (Figure 9-157).

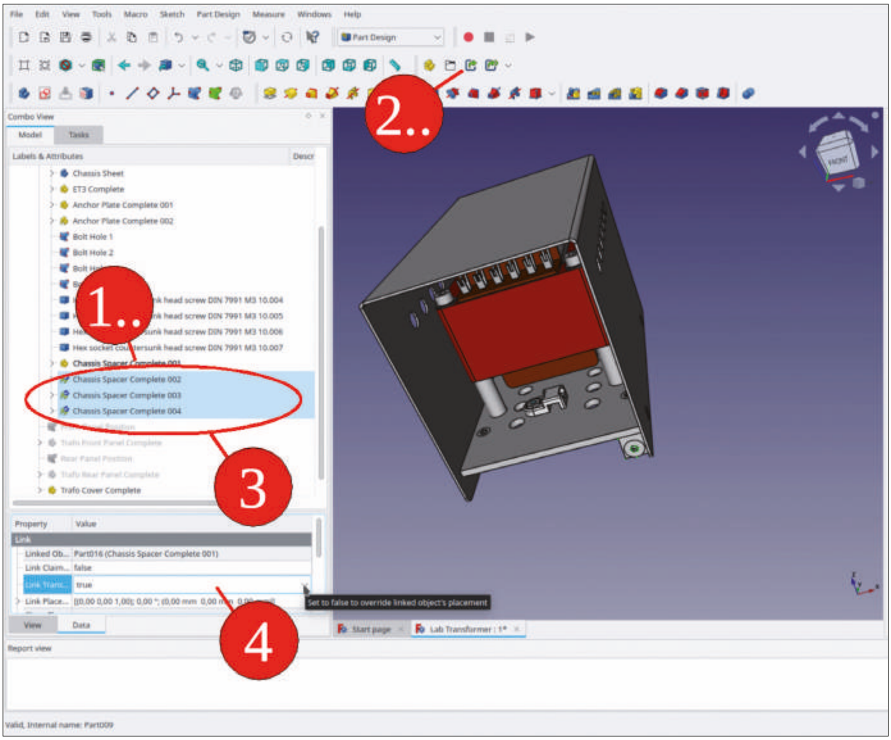


Figure 9-157

20. Edit the 'Link placement' properties of the 3 link objects, according to the table in section 9.3.1. The translations distribute the 4 spacers again in the corners of the unit.

As an advantage, you only need to change the placement of the screws on 'Chassis Spacer Complete 001'. Unfortunately, by invoking the changes to the project structures, you need to repair the references of the holes in the front and back panel.

21. In 'Chassis Spacer Complete 001', hide the two screws.

22. In 'Front Panel Complete', activate the body 'Front Panel Sheet'. Delete the last design state ('Hole005'), which represents the creation of the mounting holes. This reveals the SubShapeBinder which was contained in the tip (Figure 9-158).

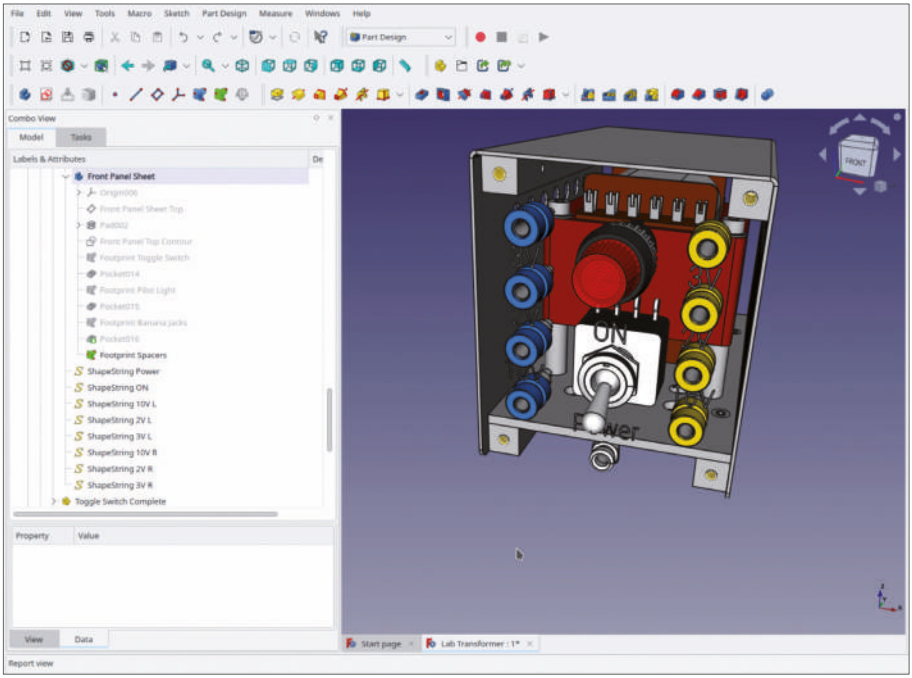


Figure 9-158

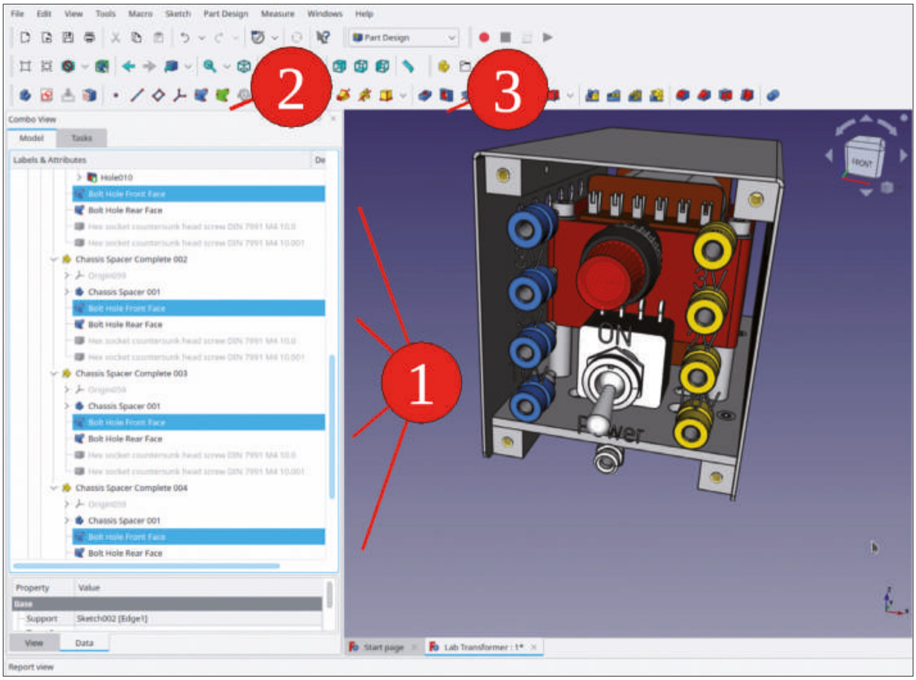


Figure 9-159

23. Delete the SubShapeBinder 'Footprint Spacers' also (after the deletion of the referenced objects, it is not easy to edit it).

24. In the tree view, in all versions of the 'Chassis Spacer Complete', mark the shape binder 'Bolt Hole Front Face' (Figure 9-159).

25. Click on the green 'Create a sub object(s) shape binder' tool button.

26. Rename the new SubShapeBinder in 'Front Panel Sheet' to 'Footprint Spacers'. With the new SubShapeBinder marked in the tree view, click the 'Hole' tool button.

27. In the task window, set the diameter to 4.2 mm, select 'Through all' for the depth and check the 'Reversed' checkbox (Figure 9-160).

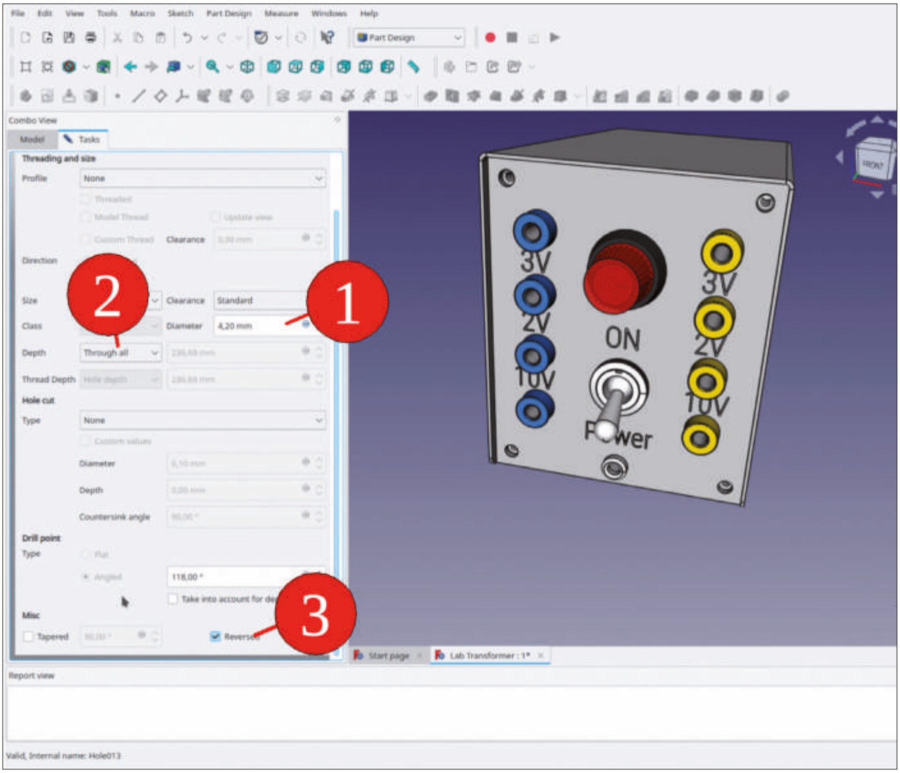


Figure 9-160

28. Close the task window with the 'OK' button.

29. In a similar way, create the bolt holes for the rear panel. Display the screws in 'Chassis Spacer Complete 001' again with the SPACE key, which updates all the linked screws also.

The design is now finished (Figure 9-161). As you have seen, it is perfectly possible to correct features in a more complex project structure. But it will not always be as easy as in this example, where you only had to undo the last design steps in the panels.

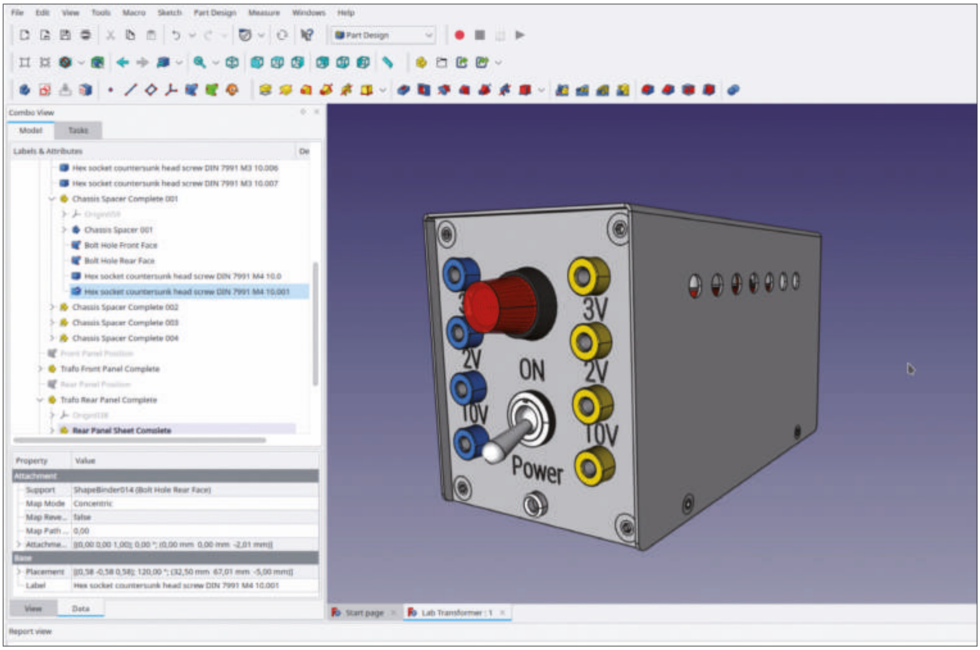


Figure 9-161

You might have noticed that the holes for the bolts were not countersunk. With respect to the documentation, this may look incomplete. Also, an external service provider would not have created the features. In the home workshop, however, such short cuts could be permissible, as planning and manufacturing are done by one person.

9.9 Creating the Unfold of the Cover

The methods to create the unfolded version of the cover, and also drawings of it, are described in section 5.4.1. With this part, outline and bends are easy to distinguish. Therefore, the sample project file contains only one sheet with a drawing of the unfolded cover.

9.10 Example Photos – Meeting the Reality

Some photos of the lab transformer are shown below. The effort invested in the 3D design paid off – all parts could be assembled without difficulties.

9.11 What to do Next

The lab transformer was a demonstration project, maybe a bit old school. Yet, all the steps undertaken to create sub-assemblies and the final assembly can be seen as quite generic. Following the method, it does not matter what the chassis contains. What about a nice 2 × 100 W high-end amplifier, as described in [ELE 2022]?

9.12 Component Libraries – Design Recycling

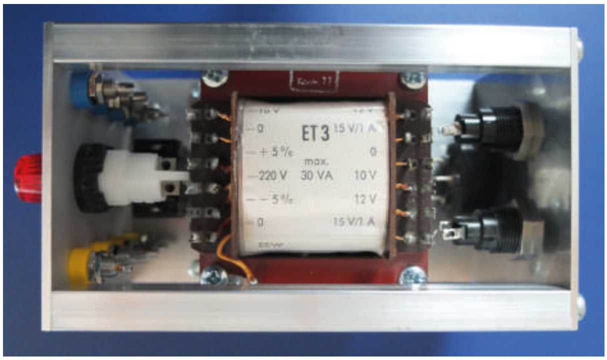
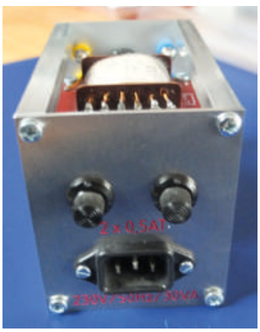
A collection of beautiful components makes the start into the 3D world easier. In the directory 'Extras' of the online material, some more parts are contained: Two versions of snap-in mains rocker switches, and a safety banana jack, which you can use and share. Be aware of the dimensions, though. Some components may look fairly similar, but have slightly different sizes, depending on the manufacturer and model.

9.13 Importing STEP Model Files

The insertion of STEP models into assemblies is fairly simple, as you have seen with the circuit board of the ESR meter earlier. Many manufacturers offer access to model files, with a sometimes-required prior registration. Parts like e.g., plastic enclosures often have inclined walls on in- and outside, which ease the release from the mold during the production process. Against such details, you may not want to fight with a caliper, but rather snug your artwork into the casing, guided by some shape binders to the model file.

When you open a STEP file with FreeCAD, the Std-Part-Container for it is automatically generated – lavishness!

Quite often, the STEP files turn out to be large. Then, it could be beneficial to switch to the 'Part' workbench and to create a body object of it, selecting 'Part | Create a Copy | Create simple copy' from the main menu. This efficient method, however, lets the color selections of the different objects in the STEP file disappear. A circuit board may then be shown in the plain standard gray.


Chapter 10 • FreeCAD and KiCad

10.1 Importing Data from KiCad to FreeCAD

All relevant information for the STEP data import has been presented in Section 9.7.1. KiCad is delivering very neat, colorful representations of the circuit boards in STEP files. This possibility is so fascinating in its own right, that it can also provide motivation for getting started with FreeCAD. However, the connection between KiCad and FreeCAD is not all plain sailing!

10.2 Creating 3D Models for KiCad

This section was inspired by [And 2022]. A detailed description of the KiCad library management is contained in [Dal 2022]. For circuit elements, which do not yet have a 3D model in a KiCad library, you can use FreeCAD to actually build one! This is very useful when you develop, say, a small breakout board for a sensor, maybe with a special holder attached. Or you have found an antique treasure in a dusty drawer, which you wish to bring back to life.

The latter case motivated the author: A number of small 7-segment LED displays were found in a box of surplus electronics (Figure 10-1). Evidently, they have the look and feel of a Texas Instruments TI-30 Pocket Calculator ('old hands' might remember the device), but they could still find useful applications (Figure 10-2).

Figure 10-1

Figure 10-2

A step-by-step instruction for the creation of the 3D model would be too exhaustive here (many more instructions follow in the appendices). The FreeCAD file for the display is contained in the directory 'LED Display' of the online material. In case of your interest, you could open the file and inspect the methods of its making.

The association of a 3D model to a circuit element footprint does not require black magic. The silkscreen patterns already represent such associations, with 2D drawings residing in the plane of the board. On many boards, the component density is quite high. Therefore, it is important to invest good effort in precise positioning of the pins and of the casing when creating the 3D model. A good fit will make the assembly of the device easier. Also, undesirable mechanical stresses, for example, by pressing the device into poorly defined eyelets, can be avoided. Many data sheets have mechanical drawings of the casings, which can be used as a reference during the design of the 3D model.

When the 3D model with suitable pins has been created (Figure 10-3), the way back into KiCad is not difficult. You need to find the path to the local directory for user-owned libraries. With openSUSE Linux, it is:

home/your-username/.local/share/kicad/6.0/3dmodels

Export (or copy) a step file of the model to this location.

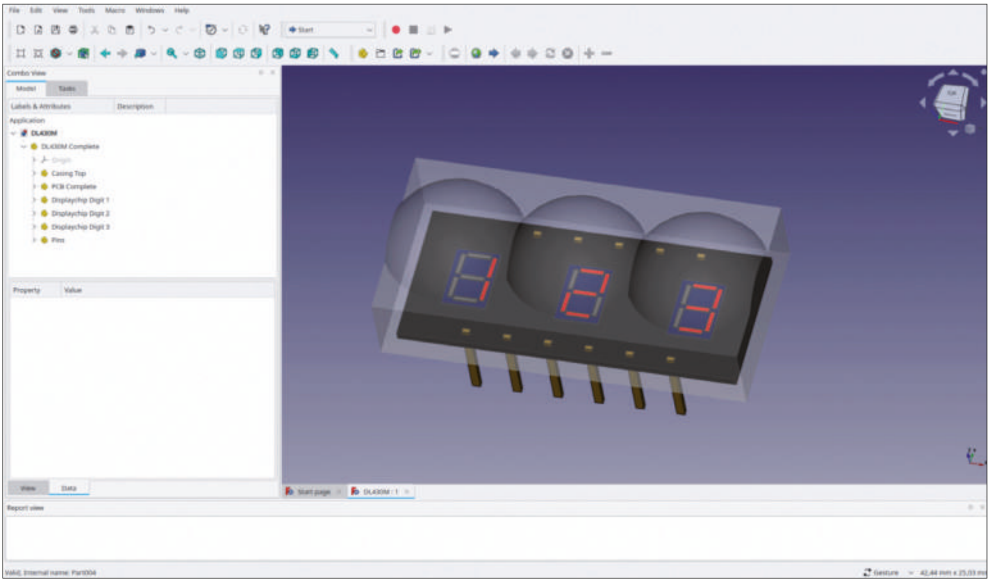


Figure 10-3

In KiCad, [Dal 2022] describes in detail, how to start a new library with the footprint editor. The footprint editor is then used to define the array of eyelets or pads, and the silkscreen symbol (Figure 10-4). After clicking on “Footprint properties” (Figure 10-5), you can select the tab '3D Models' and add the newly created model file (Figure 10-6, steps 1 and 2).

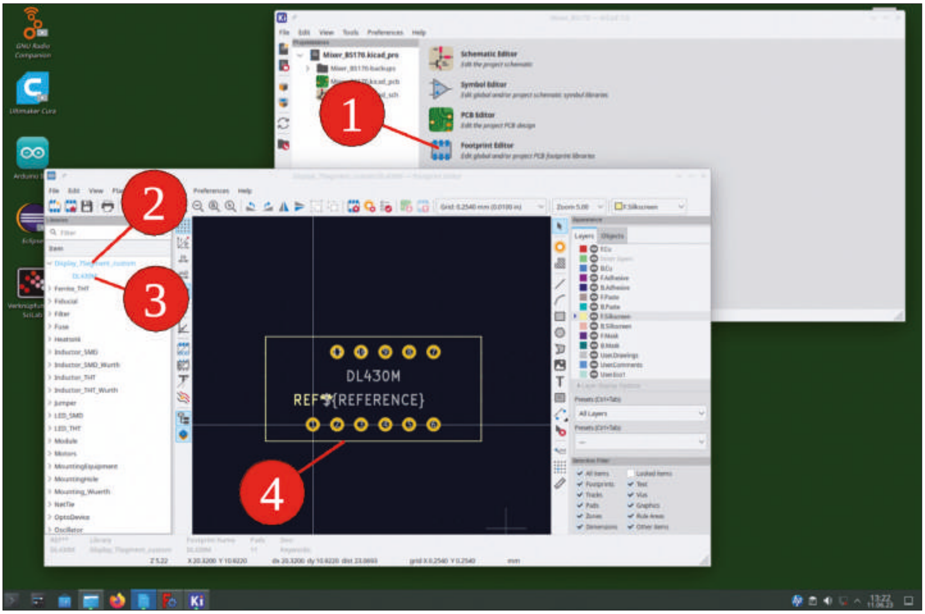


Figure 10-4

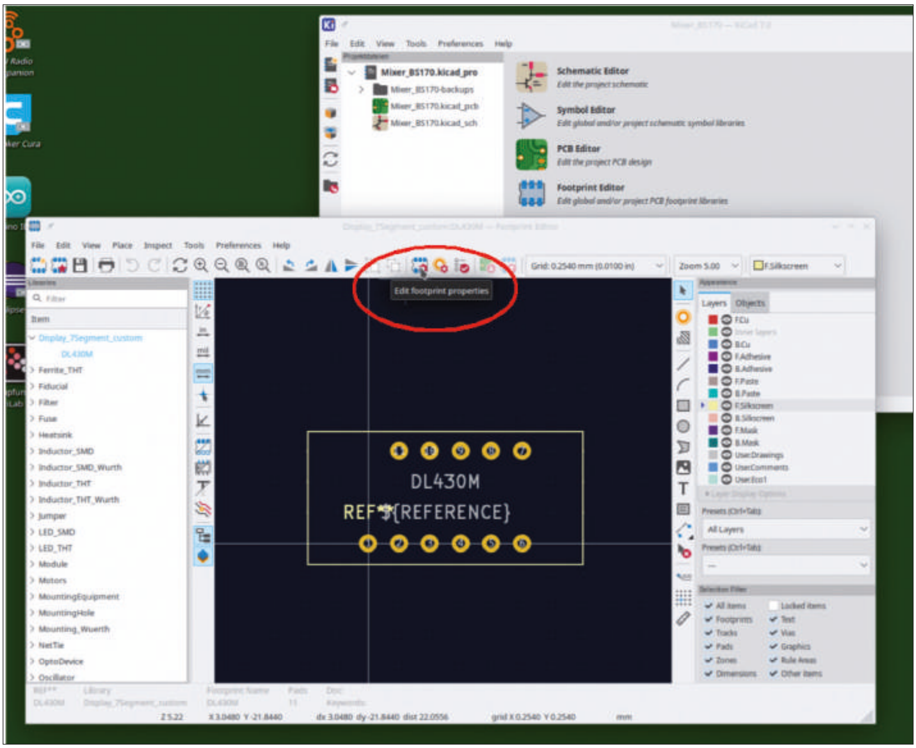


Figure 10-5

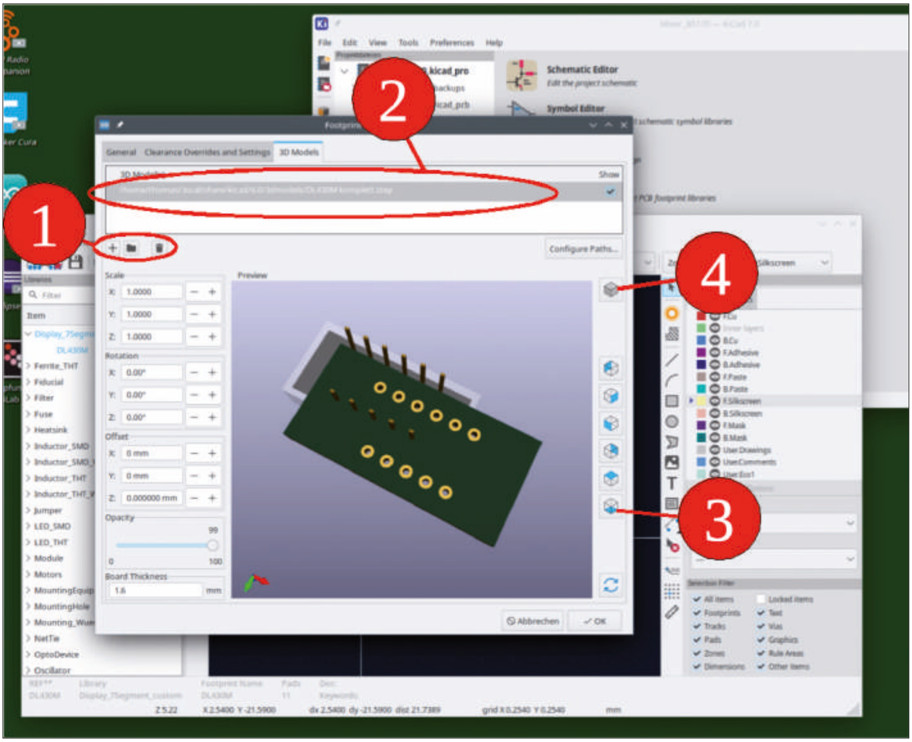


Figure 10-6

It is necessary to align the 3D model with the pads or eyelets with perfection. For a better representation, switch the view to “bottom” (Figure 10-6, step 3). Also, switch the perspective to 'orthographic' (Figure 10-6, step 4). Align the 3D model with the parameters rotation and offset (Figure 10-7, step 5). In the final state, the casing and the silkscreen print are perfectly aligned (Figure 10-8).

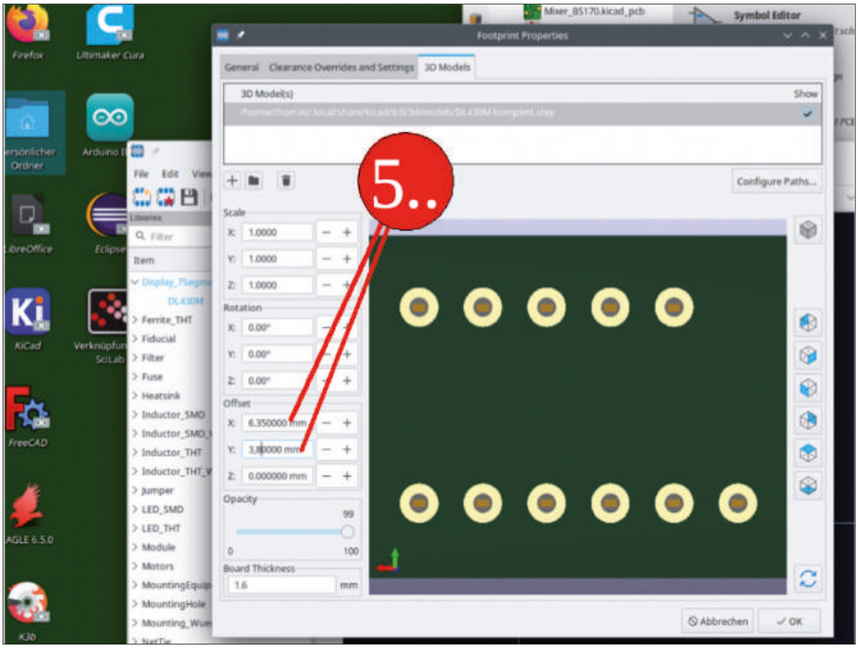


Figure 10-7

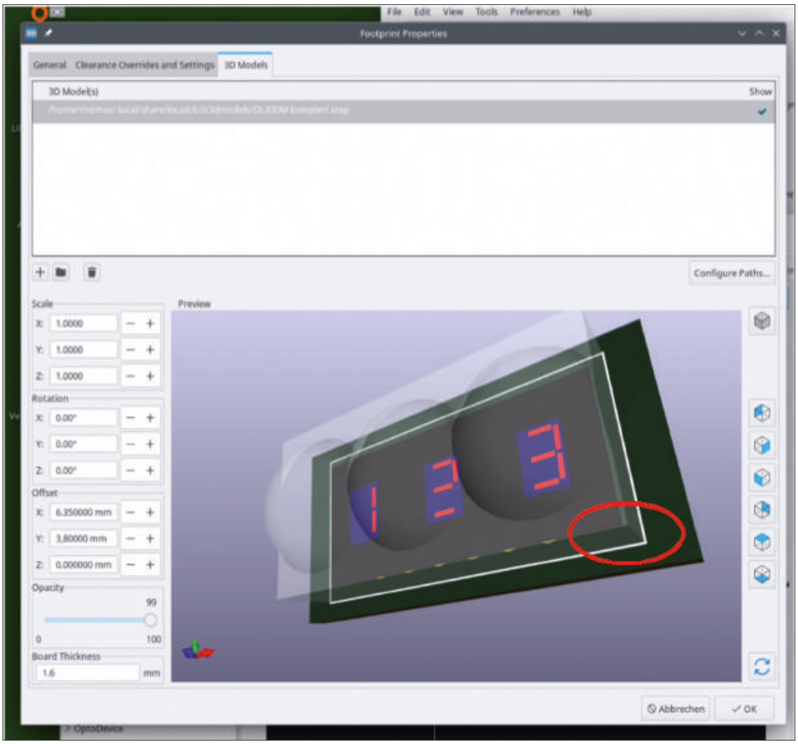


Figure 10-8

In the next step, create a symbol library with the symbol editor. With that, couple a suitable circuit diagram symbol to the just created footprint (Figure 10-9). That's all you have to do to use the new (actually, 'ancient') element in KiCad (Figure 10-10). thanks to KiCad, the steps are quite intuitive to undertake.

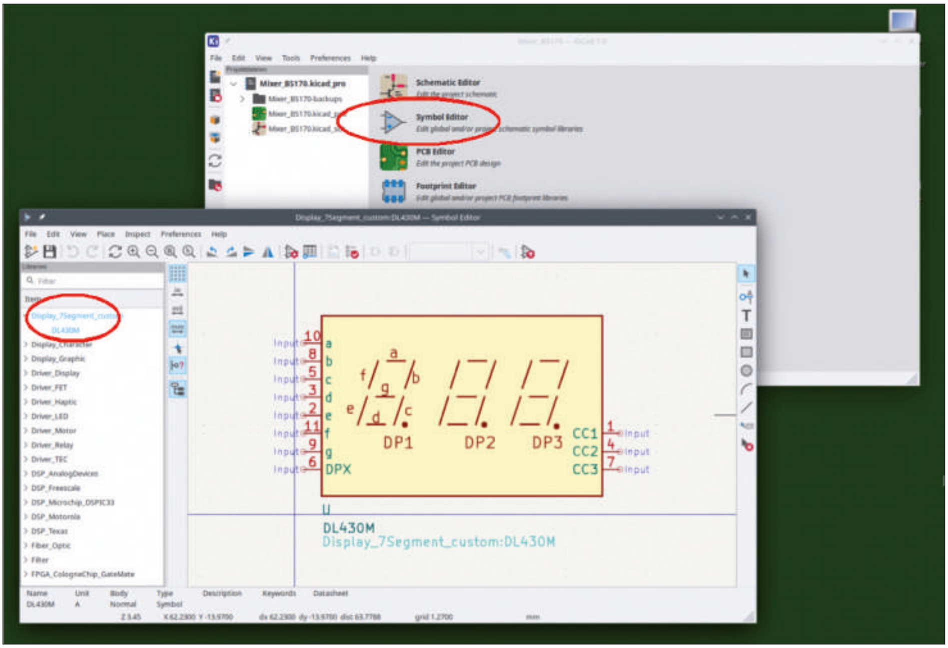


Figure 10-9

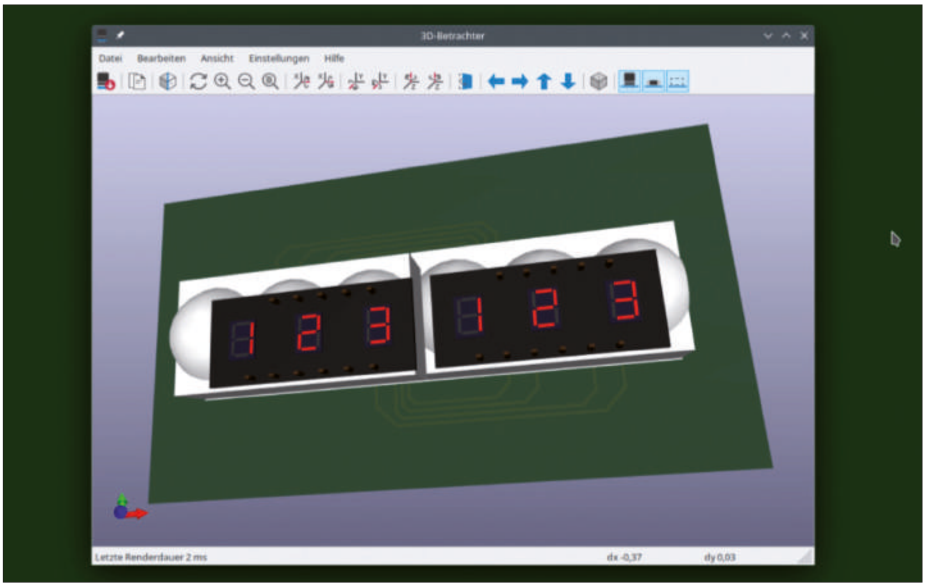


Figure 10-10

Chapter 11 • Community Resources

FreeCAD is freely available and has become quite popular over te past few years. Consequently, there is a whole universe of blogs, video tutorials on YouTube, and other websites which cover the topic. The following, rudimentary, list is intended to contribute just a few starting points for your own research.

1. The main service point is definitely the FreeCAD project homepage:
   https://www.freecadweb.org/

2. Terms and concepts are explained at:
   https://wiki.freecadweb.org/

3. This website also has the manual for free downloading:
   https://wiki.freecadweb.org/Manual:Introduction/en

4. Many topics and pitfalls are explained nicely and clearly by Ulrich Rapp here:
   https://www.ulrich-rapp.de/stoff/freecad/index.htm

5. Useful hints for the use of the sketcher are available here (with examples for 3D printing):
   https://ubit-rc.de/downloads/PartDesignTut1.pdf
   https://ubit-rc.de/downloads/PartDesignTut2.pdf

6. Another introductory website, though partly under construction, can be found here:
   https://www.freecad.info/

7. The following website offers many articles related to FreeCAD, as well as the free e-book FreeCAD for Makers:
   https://www.workshopshed.com/

8. The free download of the e-book is available from:
   https://www.workshopshed.com/2022/10/freecad-for-makers/

Sources

[And 2022] https://www.workshopshed.com/2019/04/creating-a-3D-model-for-KiCad/

[ARZ 2001] Paul Arzberger et al, Tabellenbuch Mechatronik, Gehlen Verlag 2001, ISBN 9783441921202 (in German).

[Dal 2022] Peter Dalmaris: KiCad 6 Fundamentals and Projects, Elektor International Media 2022, ISBN 9783895764967, p. 242-257; p. 381.

[EAS 2022] https://easgmbh.de/media/downloads/nc-easy-5/

[ELE 2022] Elektor Labs: Fortissimo-100 High End Amplifier, Elektor Magazine November & December 2022.

[FRE 2023] https://www.freecadweb.org/Fasteners_Workbench

[Hoi 2000] Hans Hoischen: Technisches Zeichnen, Cornelsen Giradet Verlag 2000, ISBN 9783464480083 (in German).

[Kis 2018] FreeCAD Basics Tutorial, www.tutorialbooks.info, 2018, ISBN 9781792706318.

[Kis 2019] FreeCAD 0.19 Learn by Doing, freecadtuts@gmail.com, 2019, ISBN 9798540073226.

Appendix A • The Rotary Switch

The rotary switch will be designed as a simple part with only one body. Then, the washer and nut will be added. Create a new file and save it as 'Rotary Switch'. Select the 'Part Design' workbench and create an Std-Part-Container. Rename it to 'Rotary Switch Complete'.

Click the blue 'Create body' tool button. In the tree view, rename the new body object to 'Rotary Switch' (Figure A-1).

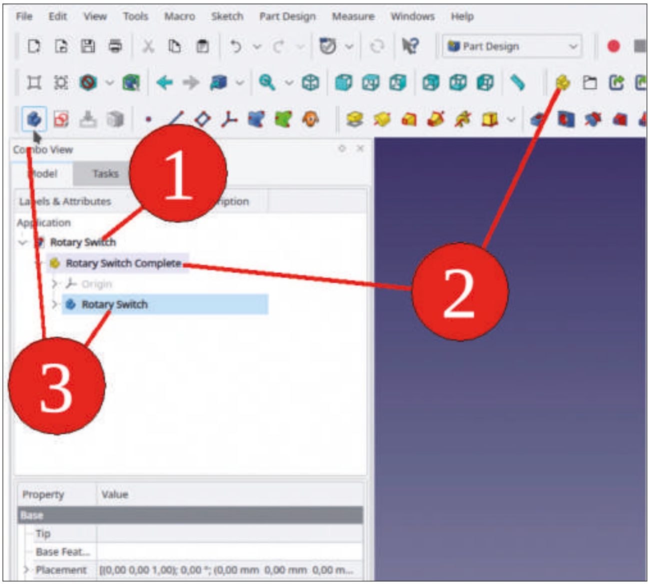


Figure A-1

Now create the casing of the switch. In the tree view, expand the body object, and show the coordinate system by marking it and pressing the SPACE key. Click once 'into the blue' to deselect the coordinate system, then mark the XY plane. Click on the 'Sketcher' tool button (Figure A-2).

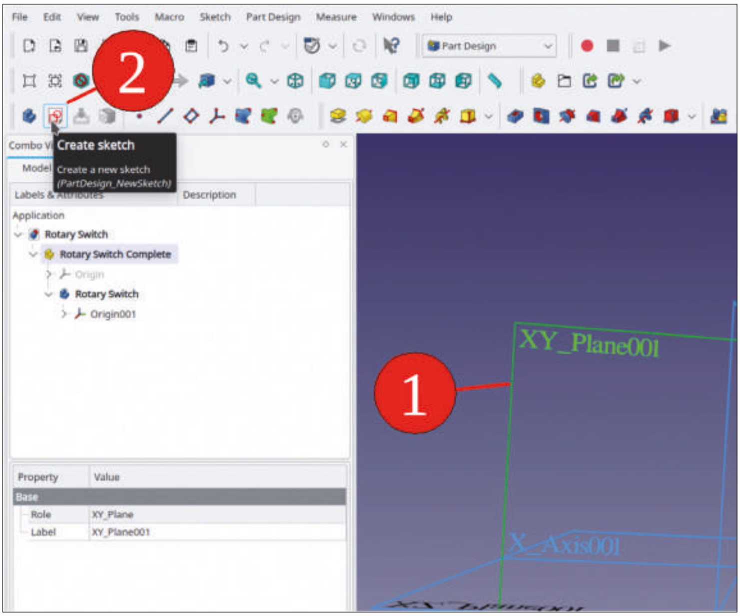


Figure A-2

Click the 'Create circle' tool button and draw a circle that is centered around the origin. Point with the little crosshairs, which is sometimes difficult to see, until the origin color changes to yellow. Only then, the tool is locked onto the origin as the target. Draw the circle, by pulling its diameter with the mouse, to some arbitrary size. One left click ends the operation. Then, terminate the drawing command with a right click. If you forget to do this, the next click starts a new circle.

In the task window on the left, locate the entry '1-circle' in the panel 'Elements'. Right-click the entry and select 'Diameter Constraint' from the context menu. Set the diameter of the switch casing to 27.5 mm. The circle turns bright green, and the solver says, 'fully constrained'. End the sketcher with the 'Close' button (top, Figure A-3).

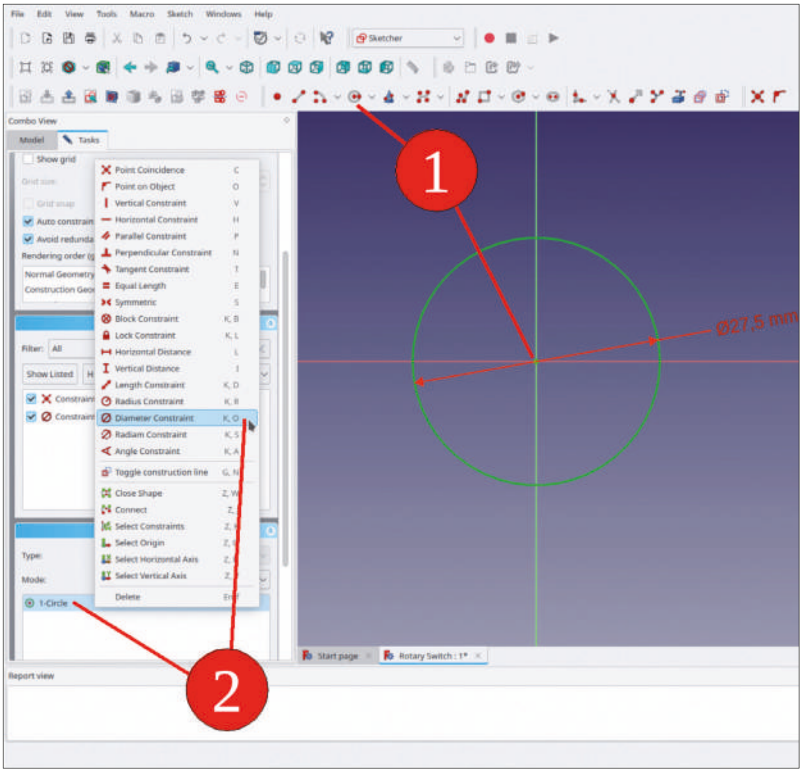


Figure A-3

In the tree view, mark the new sketch and click the yellow (= additive) 'Pad' tool button. In the task window, set the length to 12.5 mm, and check the 'Reversed' checkbox: In this way, you can set the XY plane to be the mounting plane (front panel back side) of the switch. The casing protrudes therefore to the back (Figure A-4). Close the task window with the OK button.

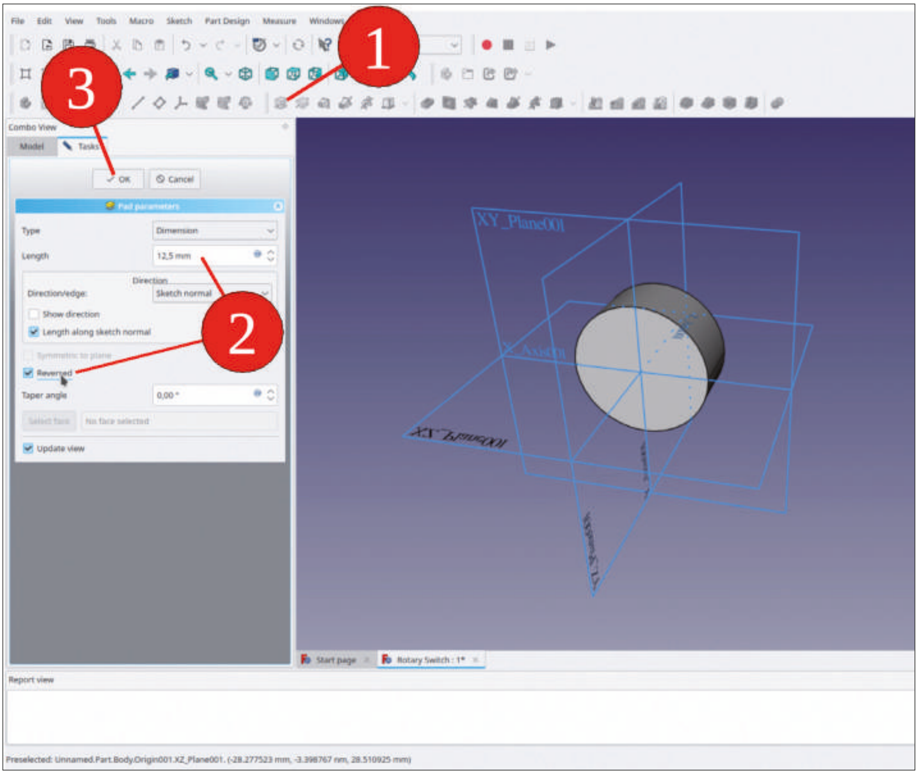


Figure A-4

Now create the threaded stub, with which the switch is mounted later on. Just like in the preceding steps, sketch a circle onto the XY plane, which is centered on the origin. We omit the thread itself for simplicity.

In the tree view, mark the new sketch and click again the yellow 'Pad' tool button. In the task window, set the length to 7 mm, and do not check the 'Reversed' checkbox, because the stub is protruding to the front (Figure A-5).

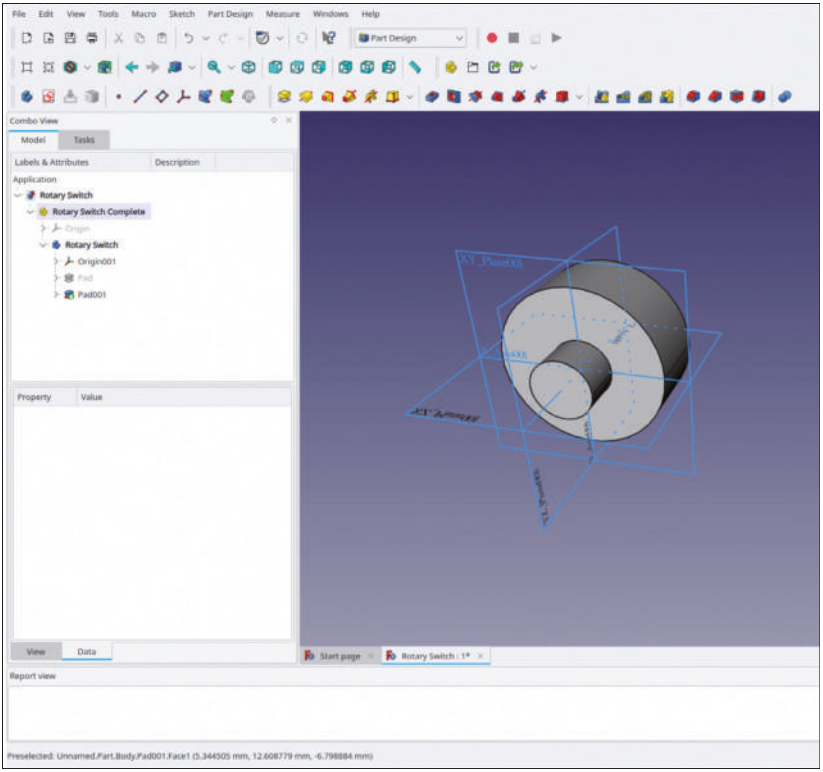


Figure A-5

The switch has a rotation lock, which is also adding to the footprint. It is a little pin, offset from the center. For this, you need a circle of diameter 3.1 mm, which is centered on the Y axis, above the origin: Start a new sketch on the XY plane. When it opens, the 3D geometry may be obscuring the sketching plane. From the main menu, select 'Sketch | View section'. If the perspective is distorting the view, from the main menu, select 'View | Orthographic view'. Then click the 'Create circle' tool button. Target the Y axis with the mouse and place the circle center. Complete the circle with the second click somewhere close. End the drawing command with a right click. Then, right-click the circle in the 'Elements' panel and select 'Diameter constraint' from the context menu, to set the value of 3.1 mm (Figure A-6). In order to set the offset value, mark the circle center and the origin. Both points are displayed in green color when selected. Then, click the 'Constrain vertical distance' tool button. In the pop-up dialog window, enter 9.1 mm and close it with the 'OK' button. Now the circle changes color to bright green, the sketch is fully constrained (Figure A-7). Close the sketch with the 'Close' button (top).

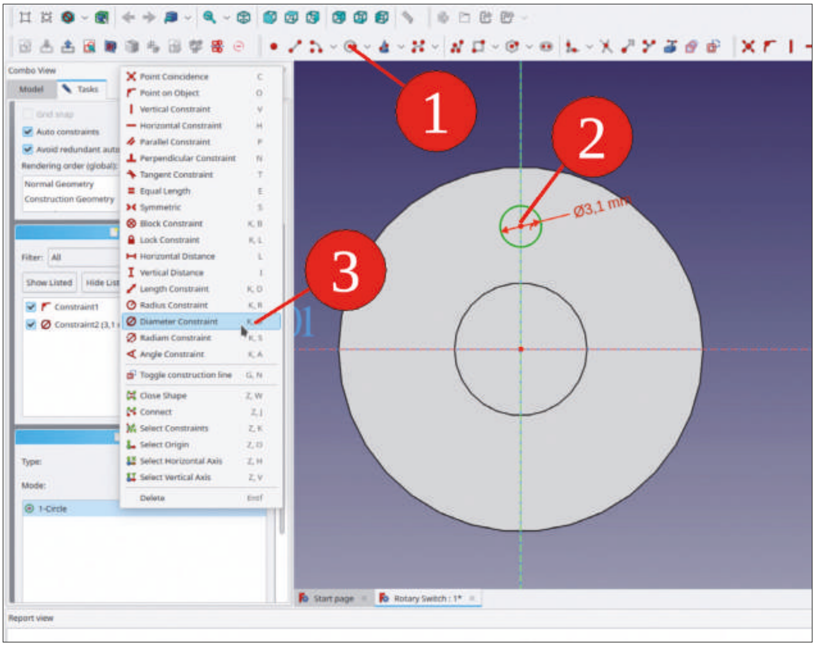


Figure A-6

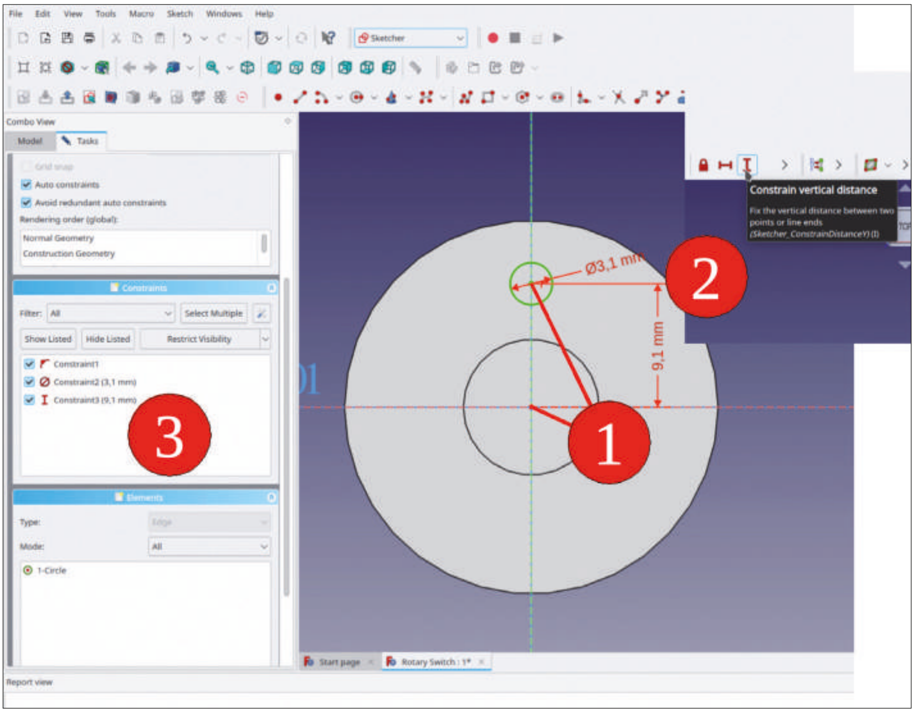


Figure A-7

In the tree view, mark the new sketch and click the 'Pad' tool button. In the task window, enter 2.5 mm for the pad length (Figure A-8).

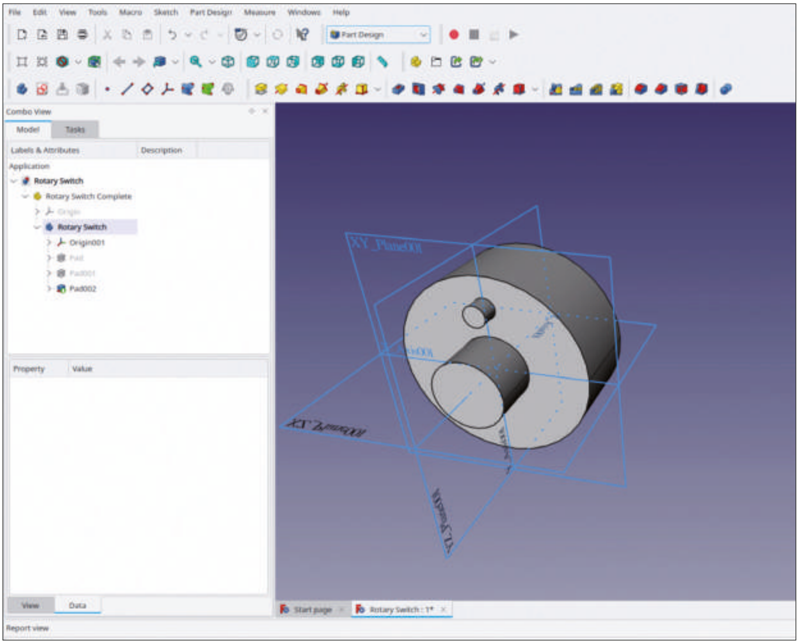


Figure A-8

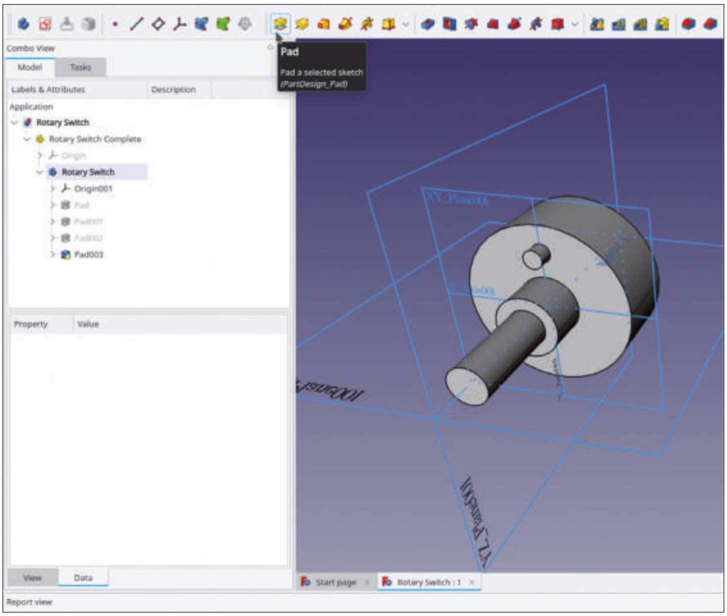


Figure A-9

Also place the sketch for the switch axis on the XY plane. In principle, the axis would thus collide with the switch casing. However, since there will be one single body for the switch model, that does not matter. By selecting the XY plane instead of the casings top facet as the starting point for the axis, it is less prone to vanish in case the facets get re-enumerated. To create the switch axis, sketch a circle onto the XY plane, which is centered on the origin, with a diameter of 6 mm. Then, using the 'Pad' tool button, create the axis by extruding the sketch to 22 mm (Figure A-9).

To make them re-enumeration immune, the contacts on the back side can also be created by sketches on the XY plane. From the main menu, select again 'Sketch | View section' and 'View | Orthographic view', to enable a better view of the sketch plane. To draw the profile, mark the XY plane again and click the 'Sketcher' tool button. From the sketcher menu, select 'Centered rectangle' and, after targeting the Y axis as the center, draw the rectangle. End the drawing command with a right click (Figure A-10).

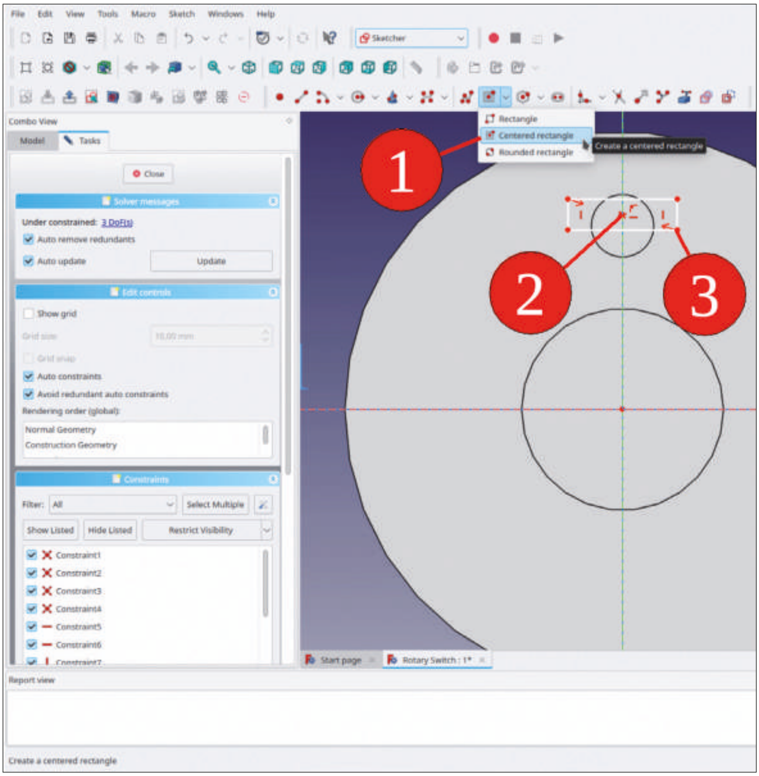


Figure A-10

Mark a horizontal line of the rectangle and click the 'Constrain horizontal distance' tool button. In the popup dialog, set the width to 1.5 mm. In a similar fashion, set the value for the height of the rectangle to 0.7 mm, using the 'Constrain vertical distance' tool button (Figure A-11).

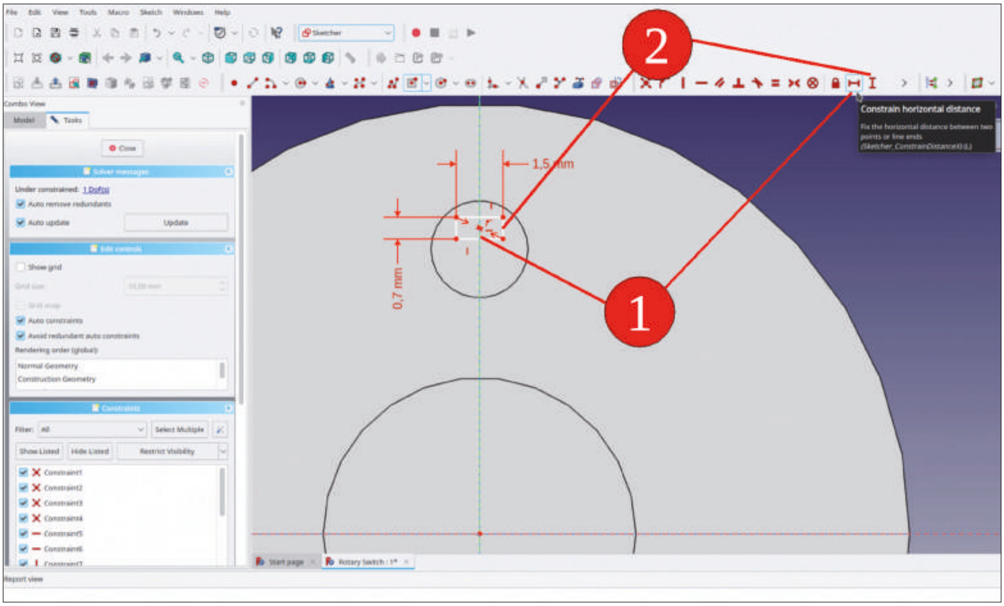


Figure A-11

Mark the center of the rectangle and the origin and set the distance to 12 mm (all the measures were picked from the switch with a caliper before). The sketch appears bright green as fully constrained (Figure A-12). Close the sketch with the 'Close' button (top).

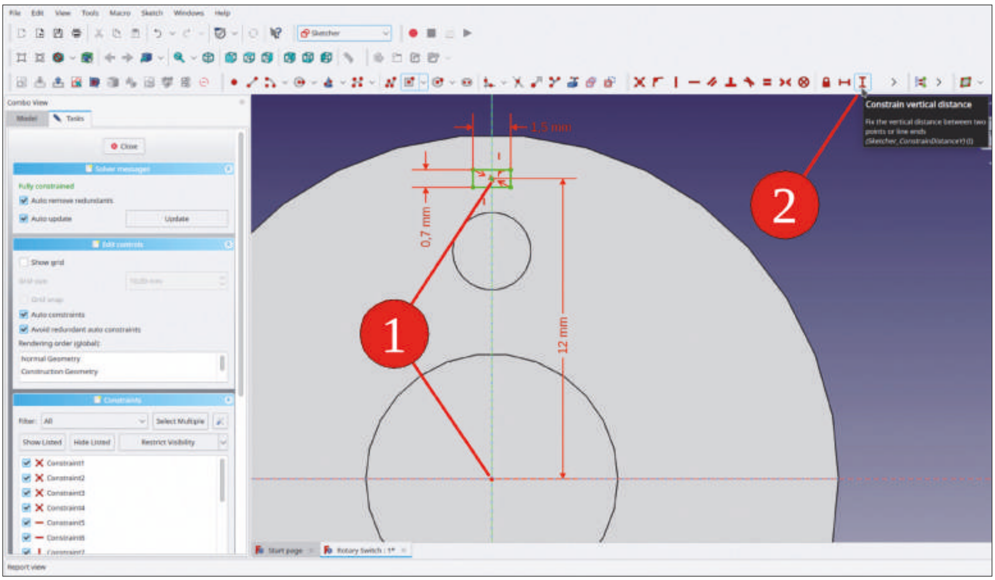


Figure A-12

The contacts protrude by 7 mm from the back of the casing, which itself has a thickness of 12.5 mm. Therefore, mark the sketch and, using the 'Pad' tool button, enter 19.5 mm for the pad length in the task window. Also, check the 'Reversed' checkbox, as the contacts protrude to the rear. Close the task window with the 'OK' button.

The contacts get chamfered edges, which ease the insertion into a circuit board. Mark the two short edges of the contacts in the back (Figure A-13) and select the 'Chamfer' tool button. In the task window, for the size, enter a value of 0.4 mm. The type 'equal distance' is already preset. If you enter a value that is too large to be generated, an error message is thrown. After the value has been reset to a permissible amount, the chamfers will be displayed again. Close the task window with the 'OK' button.

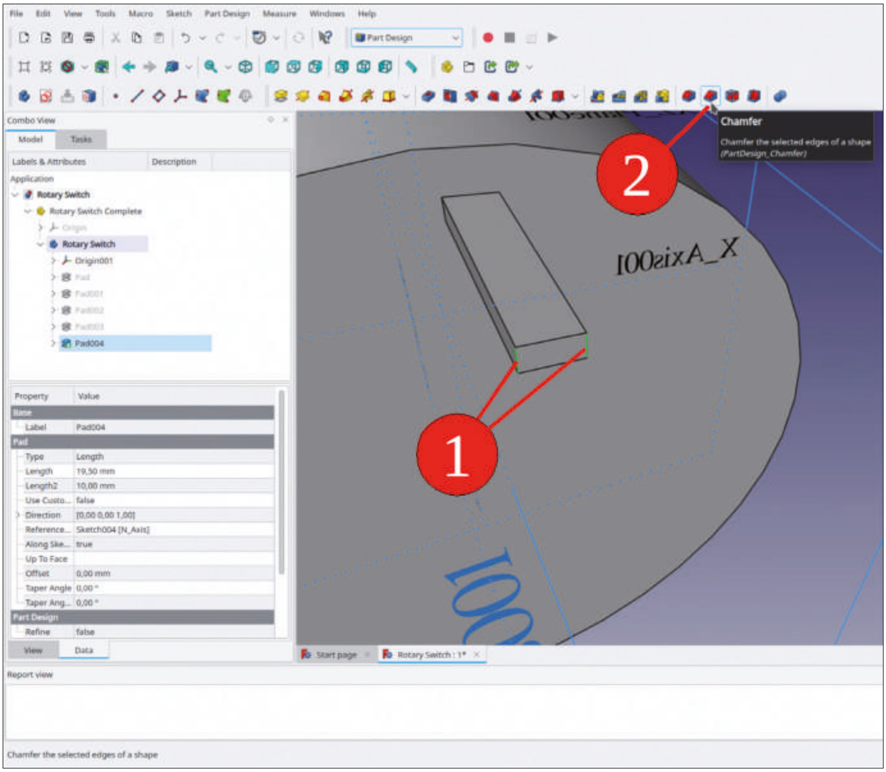


Figure A-13

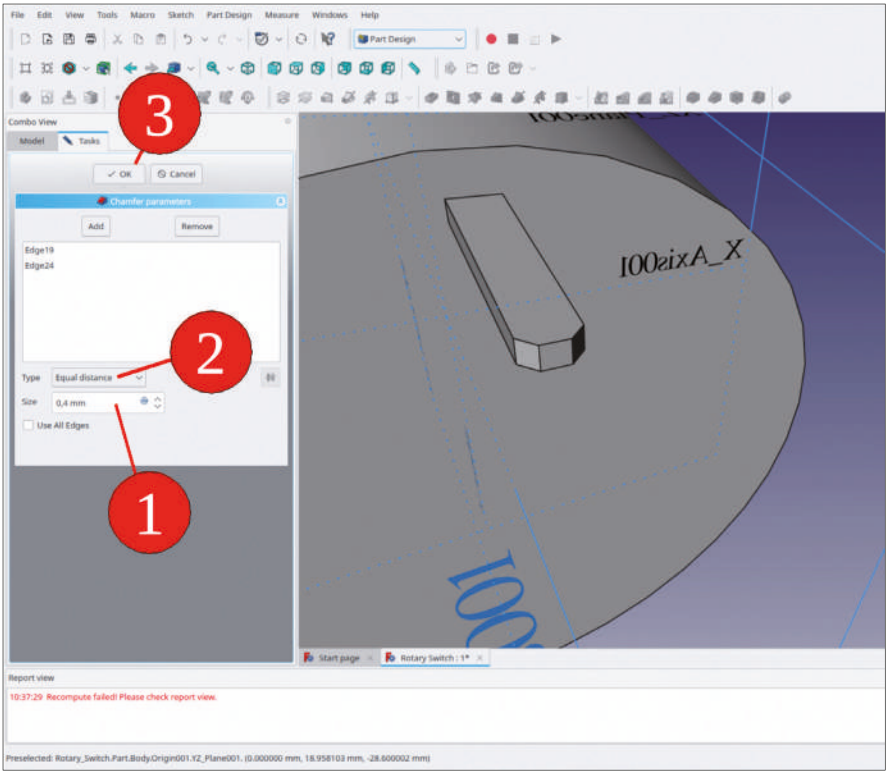


Figure A-14

In order to generate all the 12 contacts, in the tree view, mark the last pad and the chamfer (holding down the CTRL key). Even though the whole switch body is now highlighted in the 3D view, the following command will only have an effect on the marked items in the tree view.

From the main menu, select 'Part Design | Apply a Pattern | PolarPattern' (Figure A-15). In the task window, set the number of occurrences to 12. For the axis, 'Normal sketch axis' is preset (otherwise, select this axis). Close the task window with the 'OK' button (top). Hide the coordinate system of the switch by marking and pressing the SPACE key.

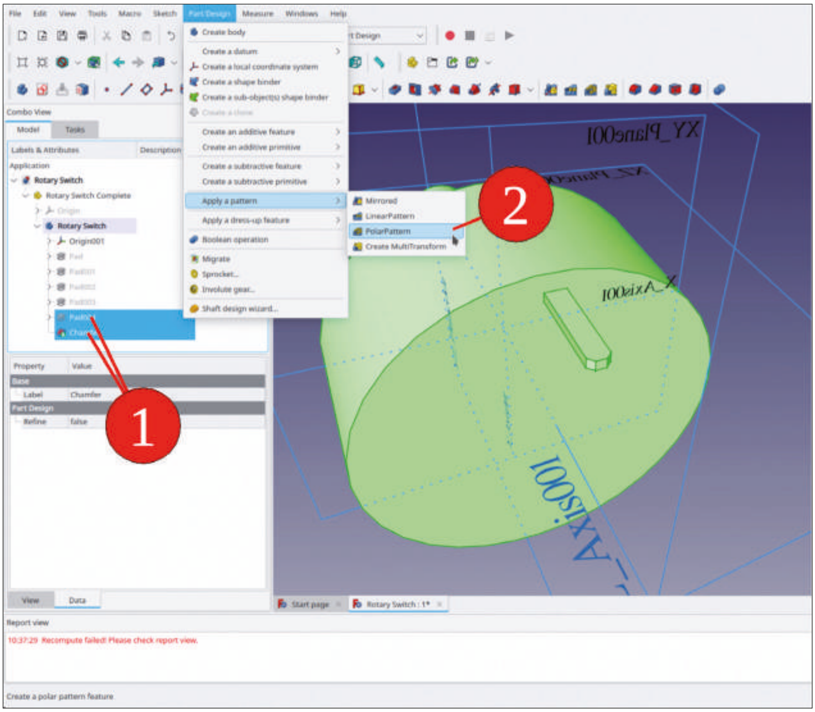


Figure A-15

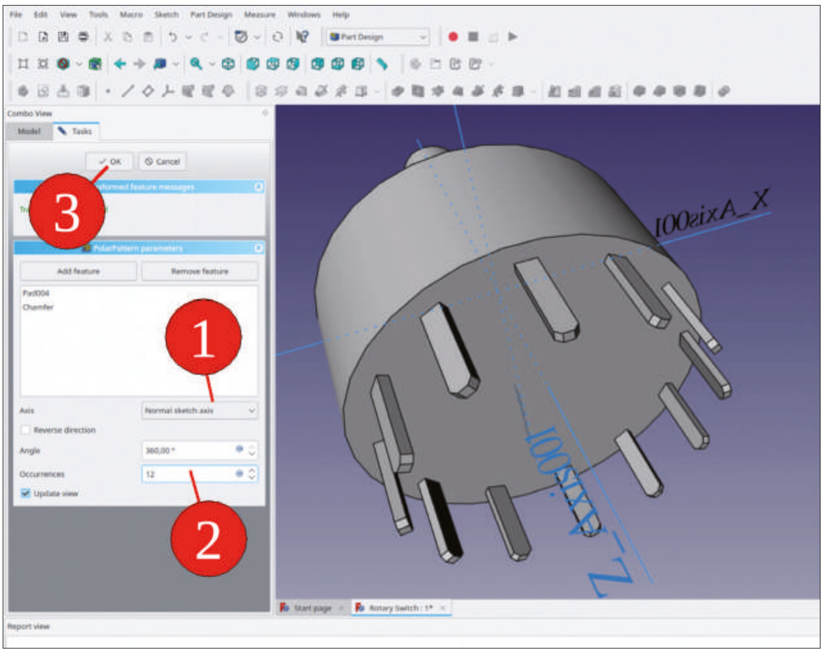


Figure A-16

Some cosmetics result in a nicer view of the switch. The end of the mounting stub could get a chamfer. Mark the sharp edge of the stub and click the 'Chamfer' tool button (Figure A-17). The type 'equal distances' is already the default, just enter a value of 0.25 mm for the size. Close the task window with the 'OK' button. In the same way, add an equidistant chamfer of 1 mm size to the front edge of the axis (Figure A-18).

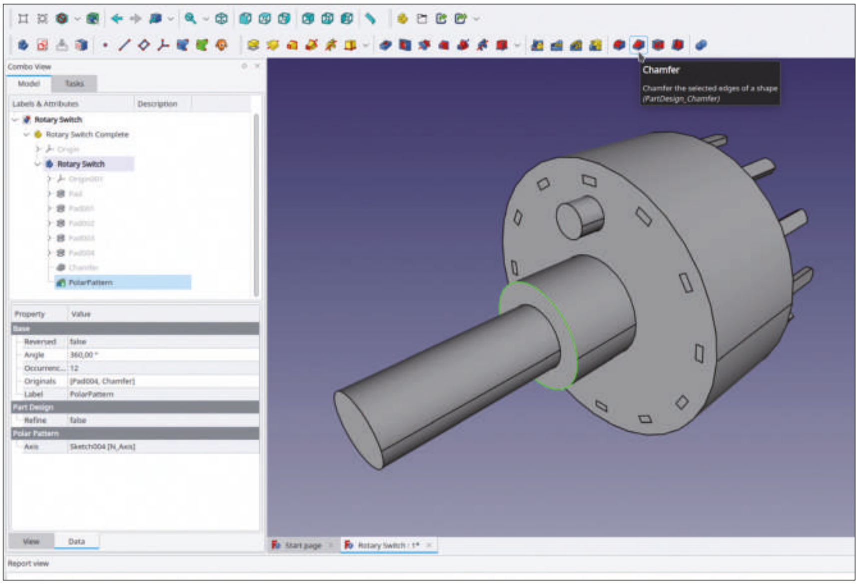


Figure A-17

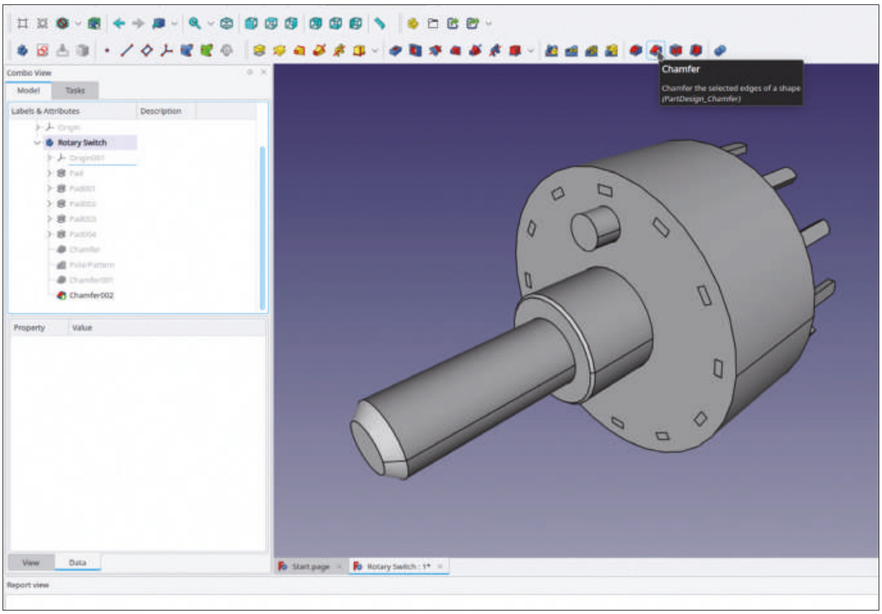


Figure A-18

In the 3D view, mark the two edges of the switch casing. In reality, the casing as rounded edges. Click on the 'Fillet' tool button (Figure A-19). In the task window, set the radius to 0.5 mm.

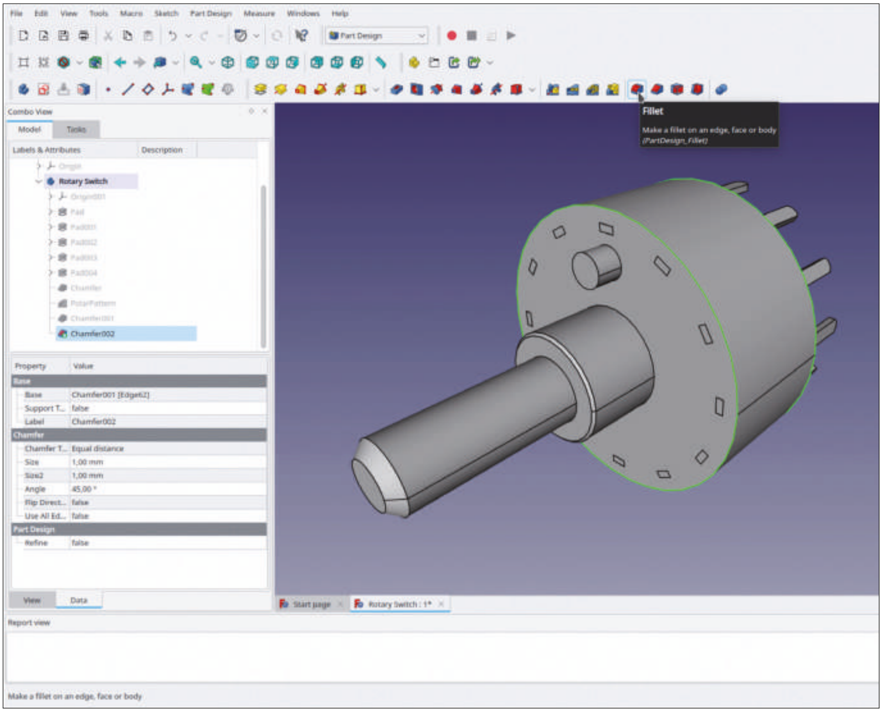


Figure A-19

Because the switch has been built as a single body part, the coloration can only be achieved by assigning colors to individual facets. These colors are bound to the tip and would vanish when some other change is applied to the body. But – this rotary switch is simple enough to appear completed. Therefore, let's limit it to simply painting the facets.

In the tree view, right-click the tip (the last design state) of the switch body and select 'Set colors' from the context menu. A task window opens (Figure A-20). In the 3D view, mark all facets of the switch axis (holding the CTRL key). Then, click the button with the color field, and pick 'black' from the color selection pop-up dialog. In the same way, you could also assign the color white to the contacts. Instead of clicking the individual facets, you could click the button 'Box selection' and mark a rectangular selection of objects, which are then subject to further coloration. Close the task window with the 'OK' button (top).

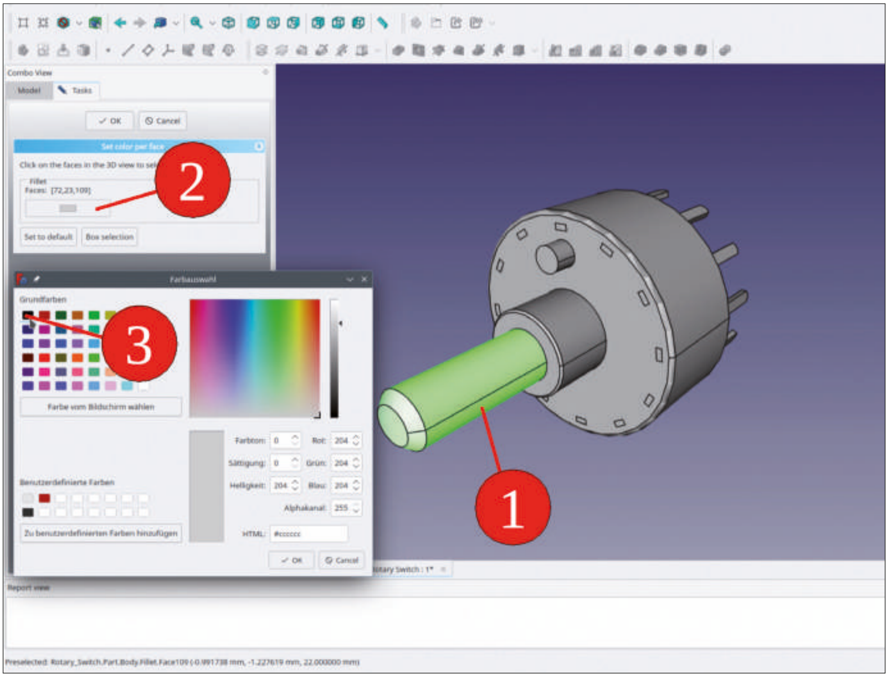


Figure A-20

The switch is now done. As an addition, fasteners could be useful later, when inserting the rotary switch into an assembly. In the tree view, collapse the 'Rotary Switch' body branch. For the fixture, a washer and a nut are needed. Let's start with the washer:

Click the blue 'Create body' tool button to start the washer. Rename the new body to 'Rotary Switch Washer'. If it is not residing in the Std-Part-Container 'Rotary Switch Complete', drag-and-drop it into there. Check whether it is activated (its title is shown in bold letters). If not, double-click it, so all of our following operations are applied there.

In the tree view, expand the 'Rotary Switch Washer' body and show the coordinate system by marking it and pressing the SPACE key. Click once 'into the blue' to deselect the coordinate system, and then mark its XY plane. Click on the 'Sketcher' tool button to start the sketch onto the selected plane right away.

Usually, when a sketch is started anew like that, the existing 3D geometry is not shown. If it shows (because you opened the sketch, e.g., for the second time), the 3D geometry can obscure the sketching plane. In these cases, it is useful to select 'Sketch | View section' from the main menu.

For the washer, sketch two circles, which are both centered to the origin. Right-click each of the circles in the 'Elements' panel and select 'Diameter constraint'. Set the diameter of the inner circle to 10.1 mm, and the diameter of the outer circle to 15 mm (Figure A-21). Close the sketch with the 'Close' button (top).

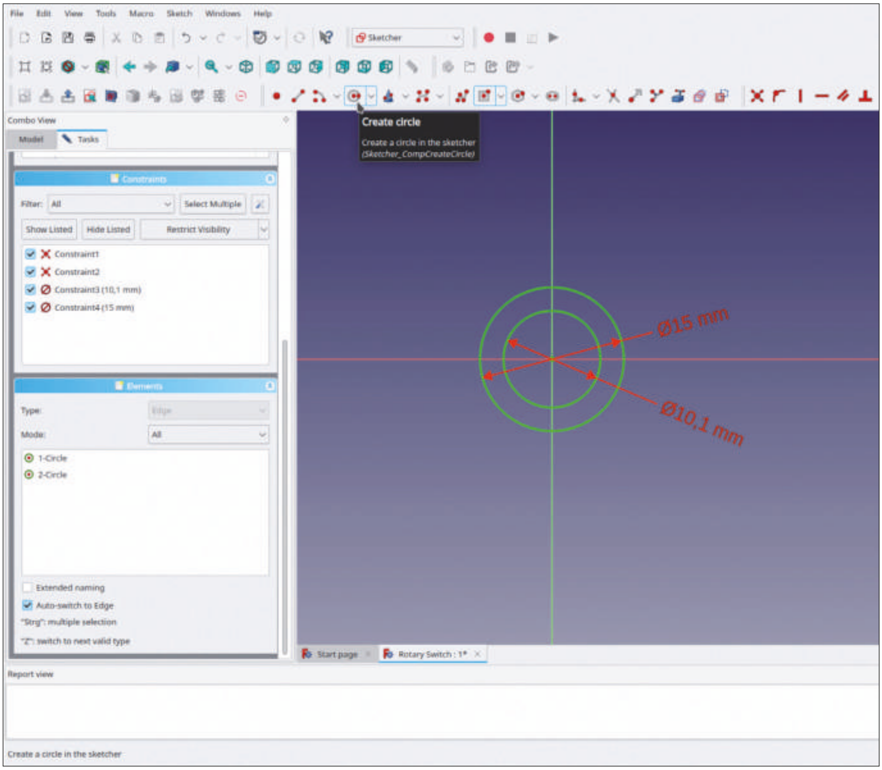


Figure A-21

In the tree view, mark the new sketch, and click the 'Pad' tool button. In the task window, set the length of the pad to 0.5 mm. Hide the coordinate system of the washer with the SPACE key.

In the tree view, right-click the 'Rotary Switch Washer' body and select 'Appearance...' from the context menu. In the task window, select 'Chrome' from the 'Material' list, and click the 'Close' button (Figure A-22).

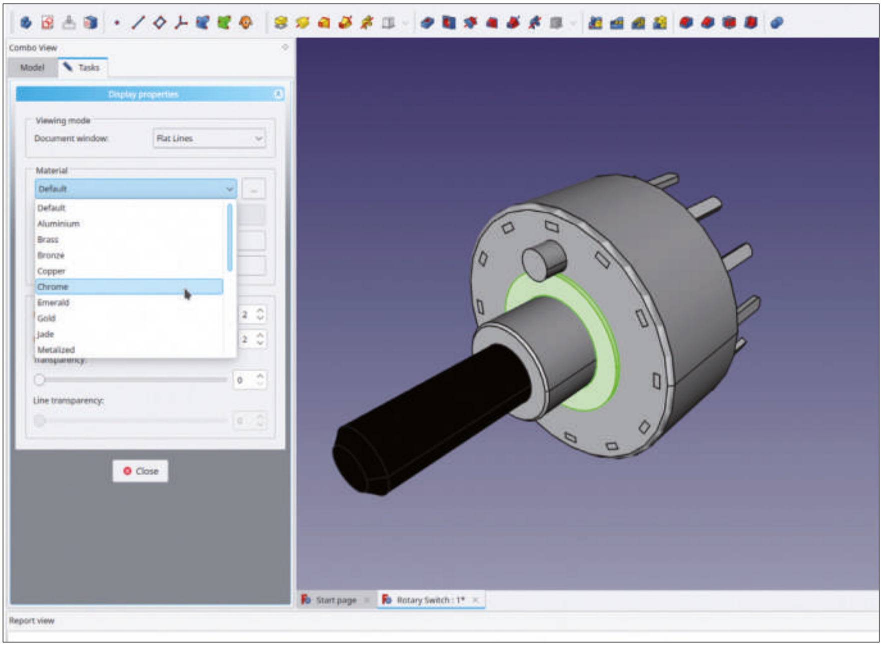


Figure A-22

In the tree view, click the 'Rotary Switch Washer' body. In the property list below, click the field 'Placement', and then the ... button, which appears to the right (Figure A-23).

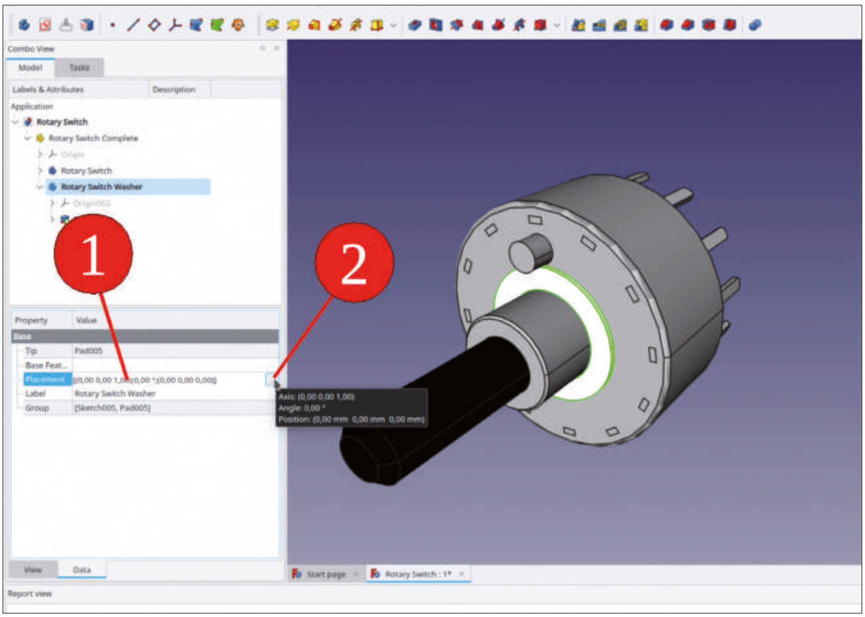


Figure A-23

In the task window, enter an offset of 2 mm for the washer, a typical front panel thickness (or the thickness value of your favorite aluminum sheet in the shed). Close the task window with the 'OK' button (Figure A-24).

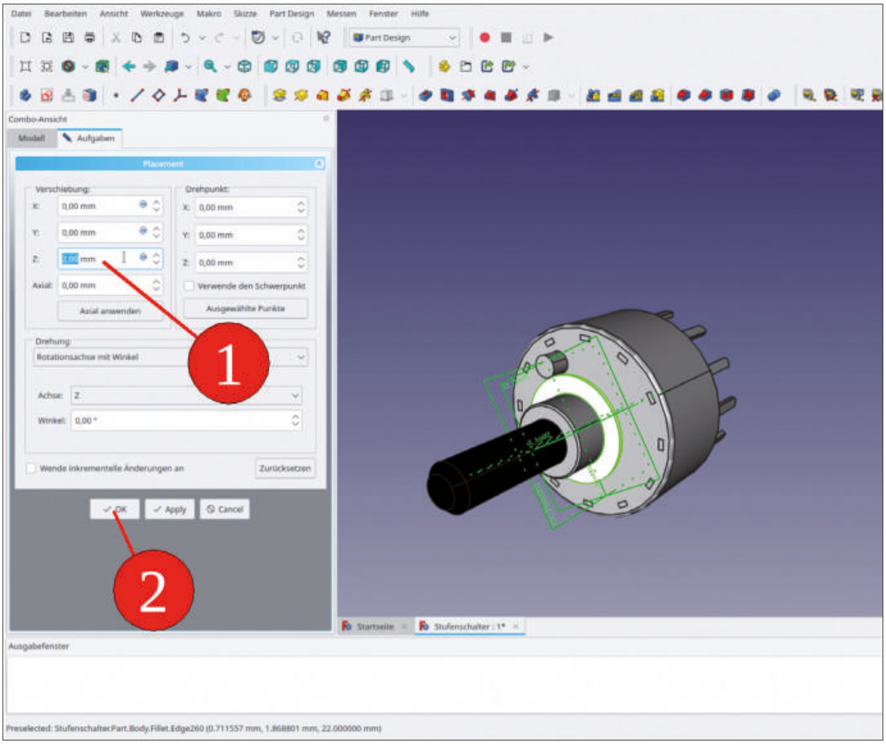


Figure A-24

For the associative positioning of the nut, you need a 'SubShapeBinder'. This reference must be visible to the nut, so you need to relocate it by a drag-and-drop into the parent Std-Part-Container once it has been created.

If the 'Rotary Switch Washer' object is still activated (title appears in bold letters), then right-click it and select 'Toggle active body' from the context menu, in order to deactivate it.

In the 3D view, mark the top inner edge of the washer (to which the nut should be bound to). Then click the green 'Create a sub object(s) shape binder' tool button. (Figure A-25).

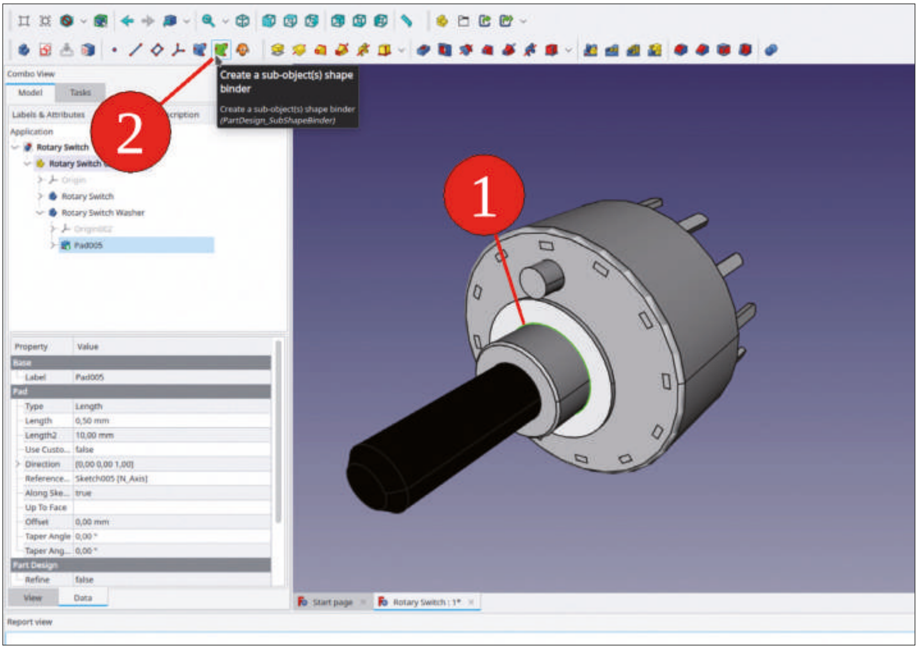


Figure A-25

Drag the green SubShapeBinder into the Std-Part-Container 'Rotary Switch Complete' (Figure A-26) and rename it to 'Position Nut'.

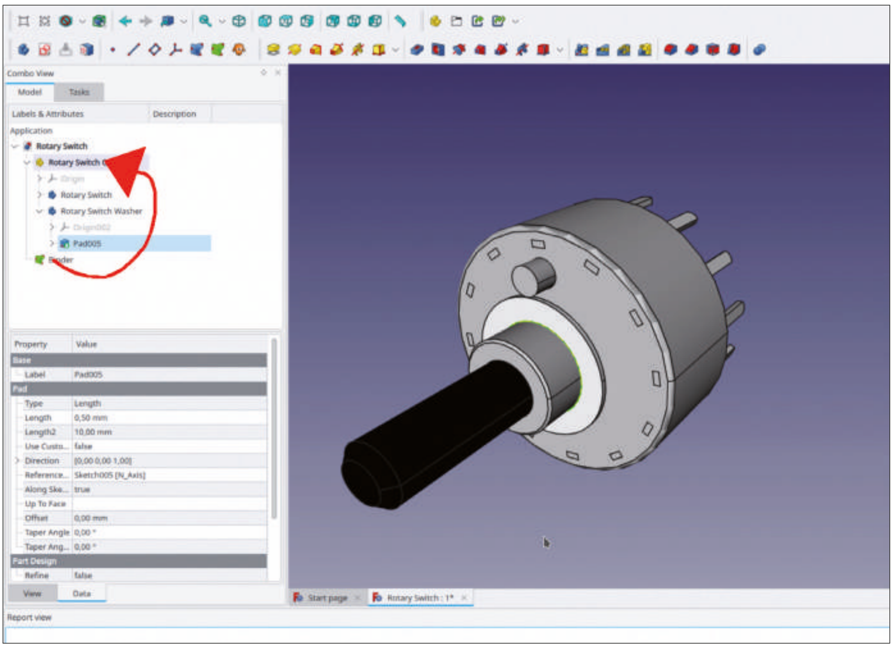


Figure A-26

If the SubShapeBinder is still marked in the 3D view, click the Std-Part-Container 'Rotary Switch Complete', to remove the marking. If you miss removing the marking, the binder will be used as a base feature in the nut. The 'Undo' button brings you back if this happens. For the nut, create a new body by clicking the blue 'Create body' tool button. Rename the new body to 'Rotary Switch Nut'. In the tree view, expand the new body object, and display its coordinate system by marking and pressing the SPACE key.

Select the XY plane (remember — deselect the coordinate system first, then select the XY plane), then, start the sketcher.

Click on the 'Create regular polygon' tool button to create the hexagonal outline of the nut. Stop the drawing command with a right click (Figure A-27). The first click of the command defines the center point (lock it to the origin), and the second click determines the size and orientation of the hexagon (if you lock it to an axis, there are less degrees of freedom to constrain later). End the drawing command with a right click.

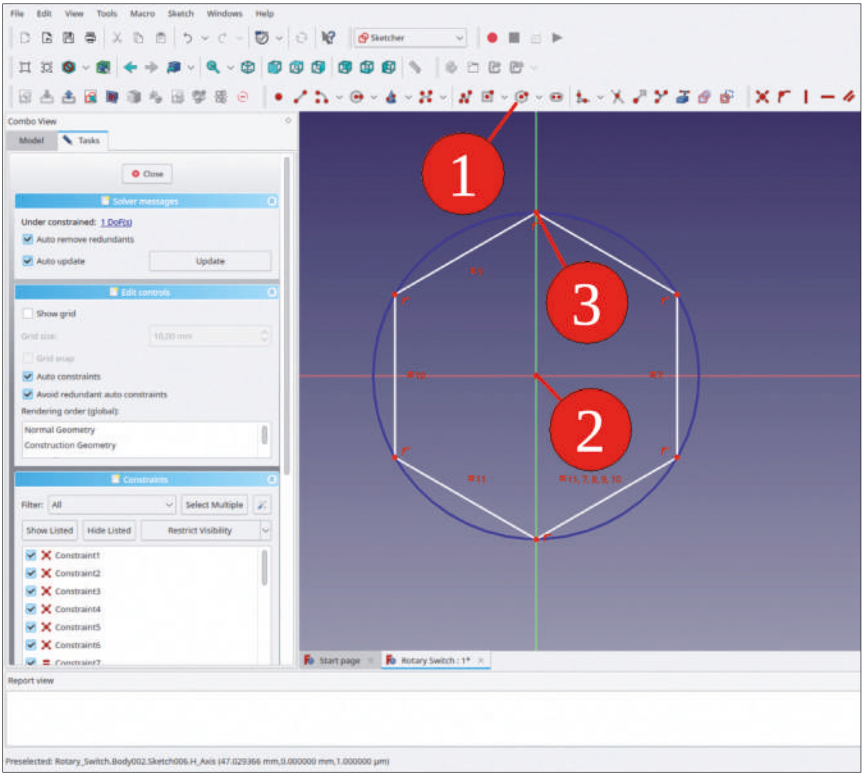


Figure A-27

On the sketch, mark two opposite points which define the key width of the nut. Then click the 'Constrain horizontal distance' tool button and enter a value of 13 mm for the key width (Figure A-28).

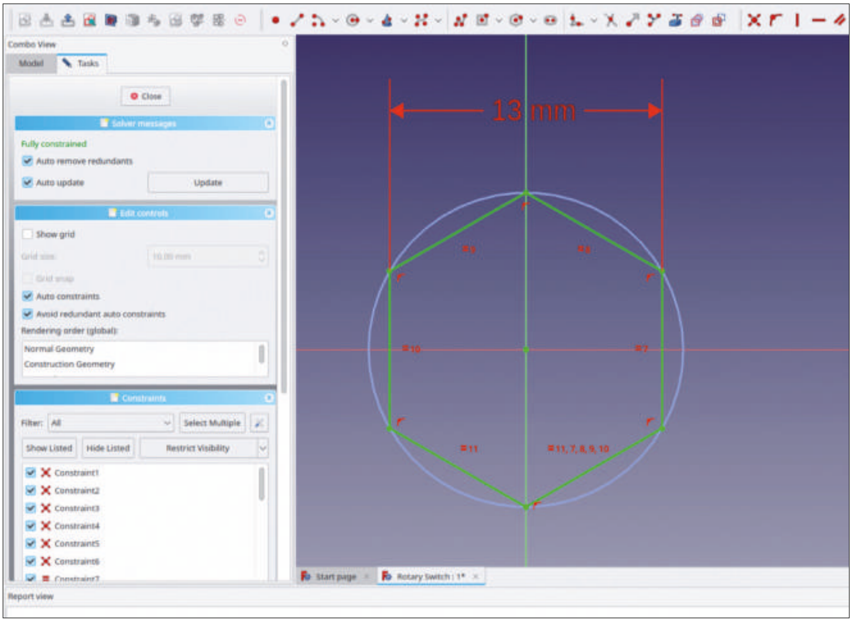


Figure A-28

Draw a circle centered on the origin and constrain its diameter to 10 mm (Figure A-29). Close the sketch (button top of the task window).

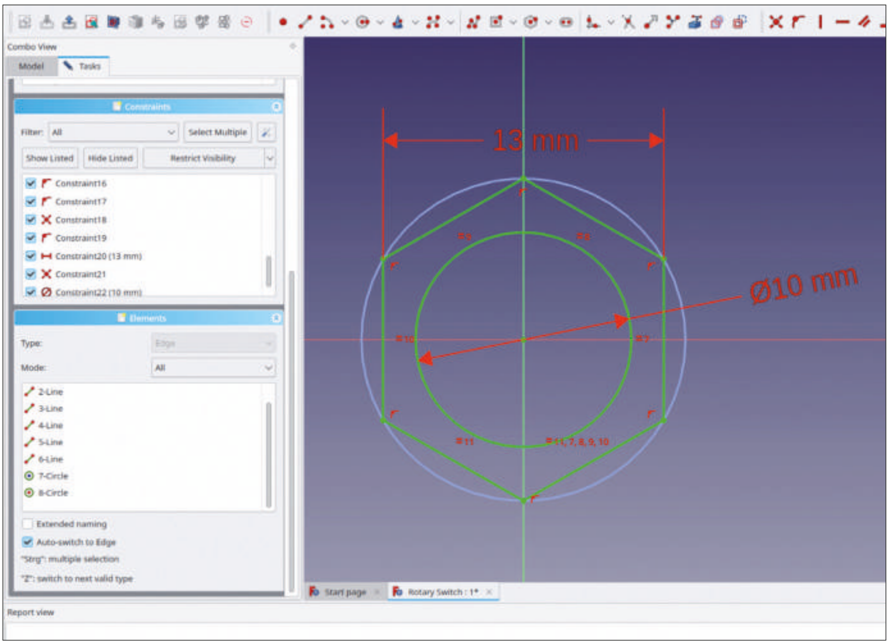


Figure A-29

In the tree view, mark the new sketch, click the 'Pad' tool button and extrude the sketch to a nut with a thickness of 2.4 mm. Right-click the body of the nut and select 'Appearance' from the context menu. In the task window, set the 'Material' entry to 'Chrome'. Then, close the task window.

The new nut is only placed into the Std-Part-Container. It is possible to bring it to the correct position by editing the 'Placement' parameter fields. But then, for each change of the front panel thickness, washer and nut would have to be relocated one-by-one. It is better to define a relation between nut and washer, in order to benefit from some degree of associativity.

Hide the coordinate system of the nut.

In the tree view, mark the body 'Rotary Switch Nut', as you want to modify its relations.

Switch to the 'Part' workbench. From the main menu, select 'Part | Attachment'. When the task window opens, the collector for reference 1 is already activated. In the Combo View, switch to the 'Model' tab and click the SubShapeBinder 'Position Nut'. Then, return to the 'Tasks tab'. In the collector, there is now the 'Binder' object.

As the attachment mode, select 'XY on plane' from the list (Figure A-30).

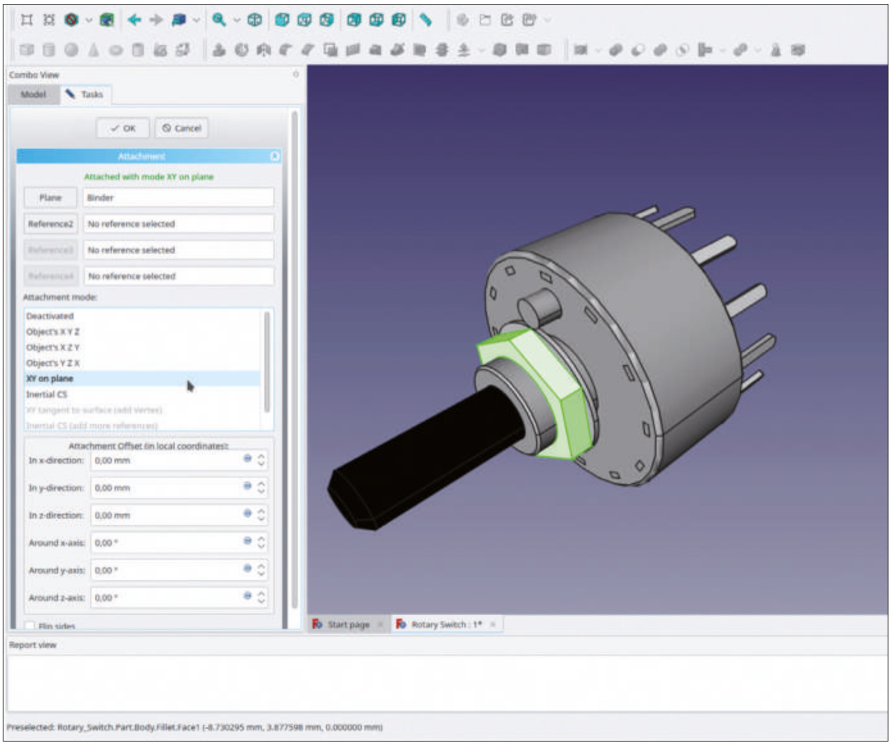


Figure A-30

Close the task window with the 'OK' button. The nut is now appearing at the correct position.

You could test the associativity by changing the placement of the washer. While the washer responds immediately to variations of its placement parameters, the nut will keep track only after a recalculation has been triggered with the F5 key.

Appendix B • The Potentiometer

Close all open documents. Start a new file and save it as 'Potentiometer'.

Switch to the 'Part Design' workbench and start a new Std-Part-Container by clicking the yellow 'Create part' tool button. Rename it to 'Potentiometer Complete'.

Click the blue 'Create body' tool button to start a new body object. In the tree view, rename that to 'Potentiometer Casing'.

Click on the 'Sketcher' tool button to start a new sketch. Because you have not marked any plane before doing so, a task window for the selection will be shown. Mark the XY plane on the list and close the task window with the OK button.

For the casing, click the 'Create circle' tool button and draw a circle that is centered on the origin. Right-click the circle in the 'Elements' list (scroll down the Combo View to see it) and select 'Diameter Constraint'. Set the diameter to 20.2 mm. Close the sketch with the 'Close' button (top).

In the tree view, mark the new sketch and click the 'Pad' tool button.

In the task window, set the length to 8.8 mm and check the 'Reversed' checkbox (the potentiometer casing, like one of the rotary switches in Appendix A, protrudes to the back (Figure B-1). Close the task window with the OK button.

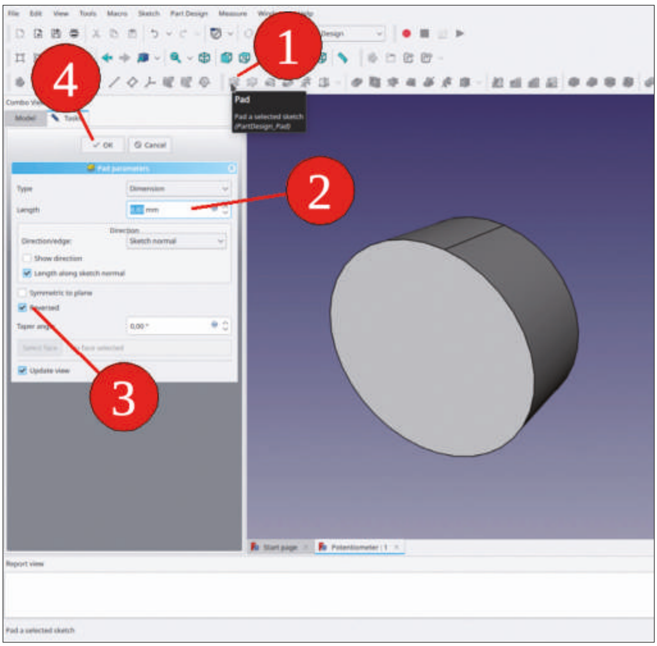


Figure B-1

For the threaded stub of the potentiometer casing, sketch another circle onto the XY plane, with a diameter of 10 mm. In the tree view, mark the new sketch and click the 'Pad' tool button. Set the length to 7.5 mm. This time, leave the 'Reversed' checkbox unchecked (the mounting stub points to the front, Figure B-2). The thread, you omit for simplicity.

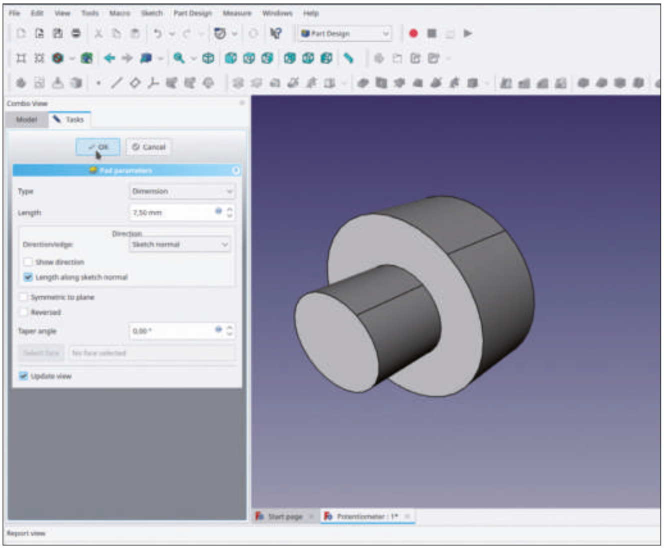


Figure B-2

To create the axis of the potentiometer, sketch a circle centered around the origin, this time with a diameter of 6 mm. The axis should protrude 10 mm from the potentiometer casing, therefore set the pad length to 17.5 mm (Figure B-3).

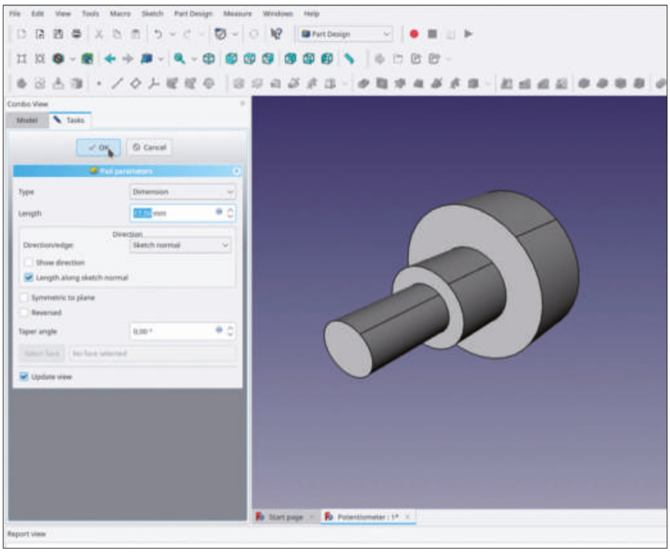


Figure B-3

Mark the top edge of the axis and click the 'Chamfer' tool button (Figure B-4). In the task window, leave the type at 'equal distance' and set the size to 0.5 mm. Close the task window with the OK button.

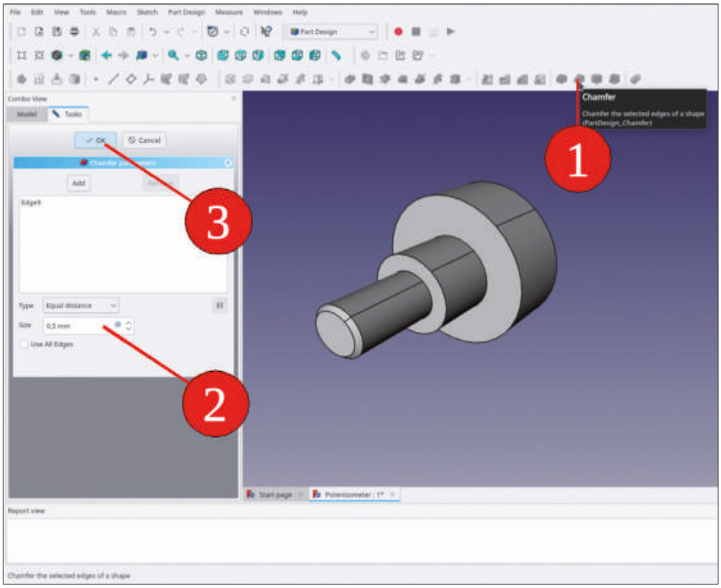


Figure B-4

The potentiometer usually has a plate that carries the contacts (and also the resistive element inside). For the sketch of this plate, you create a datum plane. To specify the orientation of the datum plane, in the tree view, show the coordinate system of the body 'Potentiometer Casing' (mark, and press the SPACE key). Then select the XY plane (if the whole coordinate system is still shown green, click into the blue to deselect it, and then, mark only the XY plane).

With the XY plane marked, click the 'Create a datum plane' tool button. The task window opens. Because of the prior selection of the XY plane, the datum plane is already attached to the XY plane, with attachment mode 'Plane face'. Set the attachment offset for Z to −3.2 mm (Figure B-5) and close the task window with the OK button.

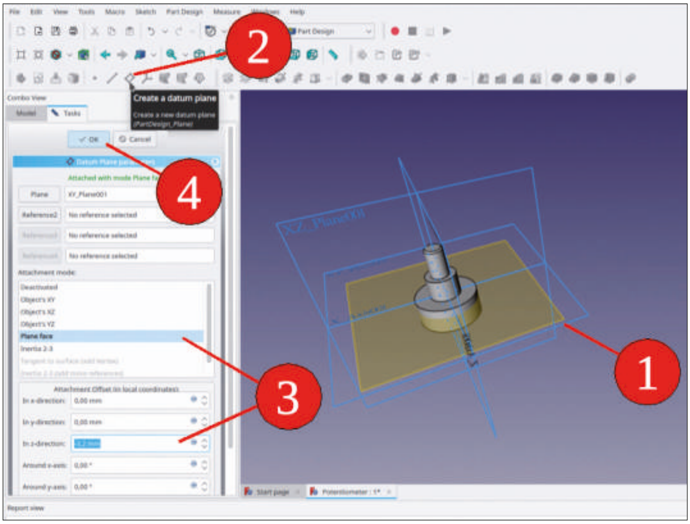


Figure B-5

Rename the new datum plane to 'Contact Plane'. With the datum plane marked, click the 'Sketcher' tool button to start the drawing for the contact plate. When the sketcher is started for the first time, the already existing 3D geometry is not shown. Close the sketch and open it again by double-clicking it in the tree view.

Click the 'Create a centered rectangle' tool button and draw a rectangle that is centered on the Y axis (first click), and the top coinciding with the X axis (second click). End the drawing command with a right click. Mark a horizontal side of the rectangle, click the 'Constrain horizontal distance' tool button and set the width of the rectangle to 17.7 mm. In the same fashion, set the height to 14.4 mm (Figure B-6). Close the sketch with the 'Close' button (top).

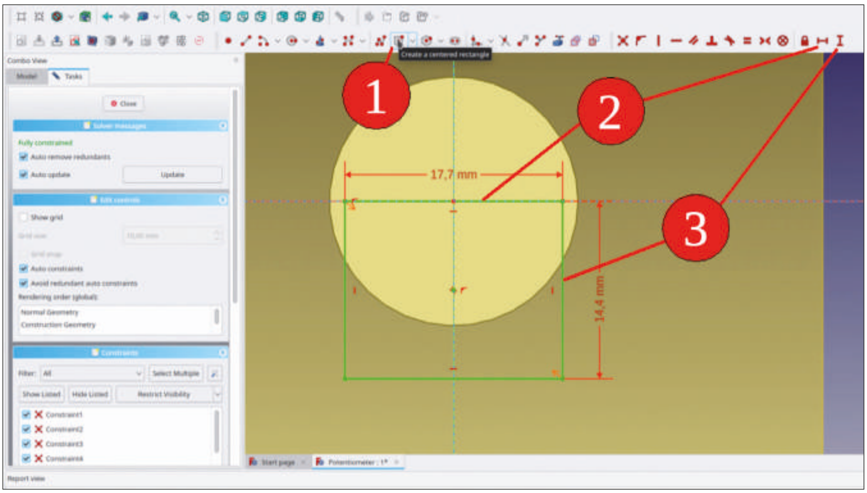


Figure B-6

Pad the new sketch with a length of 2 mm (Figure B-7). In the tree view, hide the datum plane and the coordinate system (mark, and press SPACE key).

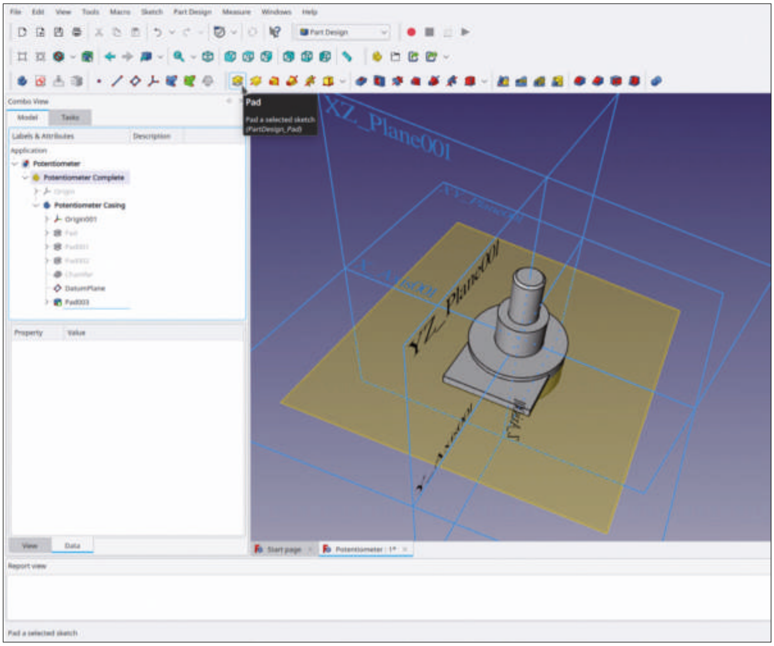


Figure B-7

Now, the contacts will be created. In order to sketch the datum plane 'Contact Plane', mark the hidden datum plane in the tree view (an alternate way to clicking on it in the 3D view, if shown there). Then, click the 'Sketcher' tool button. To display the already-present 3D geometry, close the sketch right away, and reopen it by double-clicking on it in the tree view.

From the main menu, select 'Sketch | View section' and 'View | Orthographic view', for a plain and unobstructed view of the sketching plane.

To reference the new sketch to already present details, click the 'External geometry' tool button (Figure B-8) and mark the lower edge of the contact carrier plate. The edge is shown as a violet construction line, to which drawing elements can now be locked.

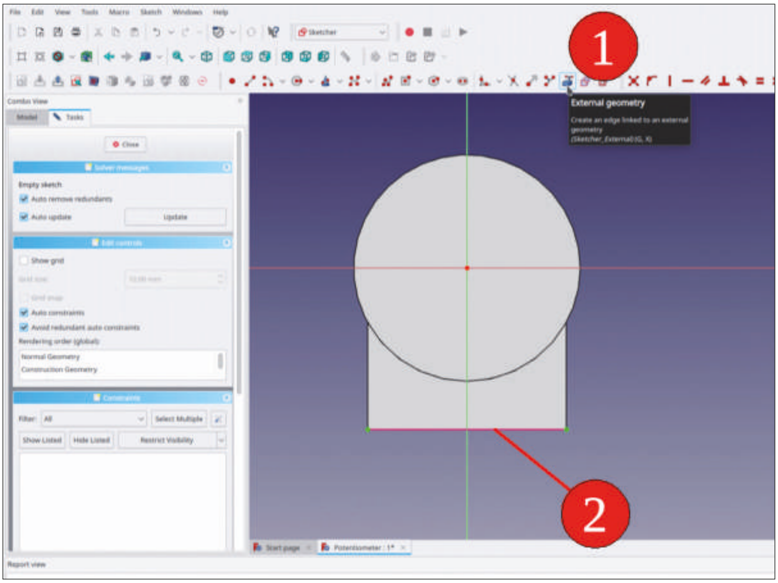


Figure B-8

Click the 'Create a centered rectangle' tool button and draw the rectangle with the midpoint locked to the Y axis. Set the height of the rectangle to 10 mm, and the width to 1.2 mm. Then, mark the center point of the rectangle, the violet construction line, and click the 'Constrain point on object' tool button (Figure B-9). Close the sketch with the 'Close' button (top).

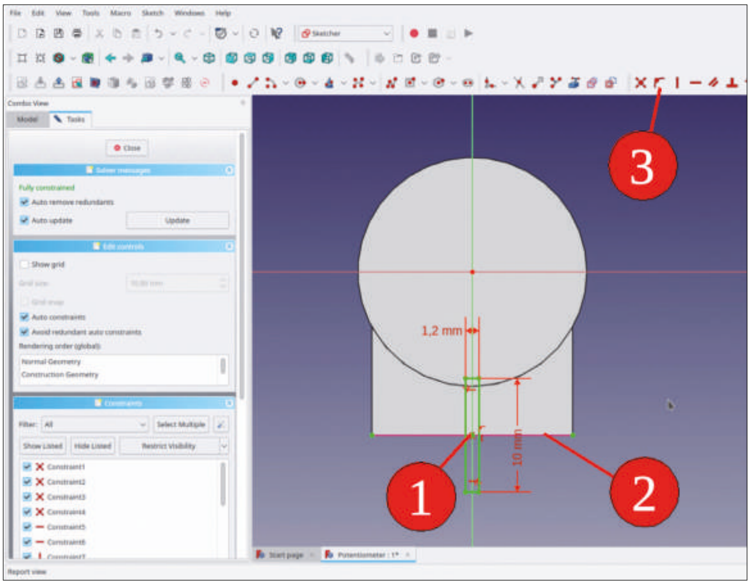


Figure B-9

In the tree view, mark the new sketch. Clicking the 'Pad' tool button, and extruding it towards the back (with 'Reversed' checkbox checked), to a thickness of 0.5 mm.

From the main menu, select: Part Design | Apply a pattern | LinearPattern.

In the task window, the directions should read 'Horizontal sketch axis'. Set the length to 5 mm and leave the number of occurrences at 2 (Figure B-10). Close the task window with the 'OK' button.

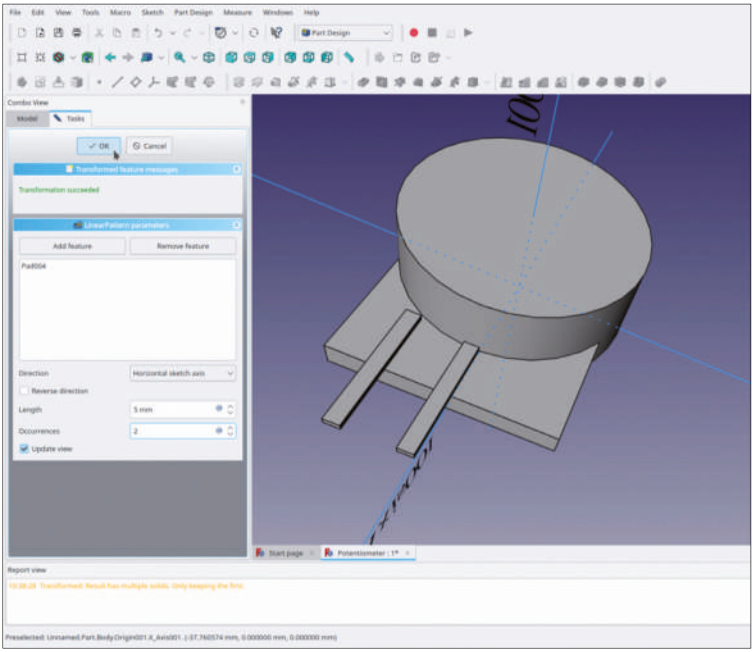


Figure B-10

In the tree view, again mark the design state with the center contact (the last one before 'LinearPattern'). Again, select 'Part Design | Apply a pattern | LinearPattern' from the main menu. In the task window, all the settings are identical to those of the preceding step, except that the 'Reverse direction' checkbox must now be checked (Figure B-11) to add the opposite contact. Close the task window with the OK button.

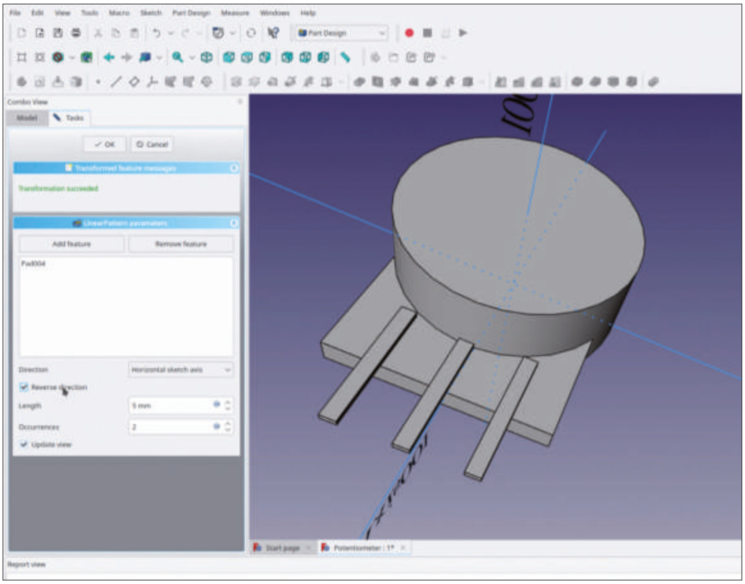


Figure B-11

Like with the rotary switch, as cosmetic features, add a fillet of 1 mm radius to the bottom edge of the potentiometer casing, and a small chamfer of 0.25 mm size to the top edge of the mounting stub.

The potentiometer looks finalized now. Except for the length of the axis, no changes are expected for the part. Therefore, it seems permissible to color the facets of the single body part. To get a nicer reflectivity for the whole part, you could right-click the body 'Potentiometer Casing' and select 'Appearance' from the context menu. In the task window, set the 'Material' to 'Shiny Plastic', but set the color to a bright gray (the default color for plastic is black, which renders shapes difficult to distinguish).

After that, right-click the tip of the body (the last design state, which is displayed non-grayed). Select 'Set colors' from the context menu. Select all facets of the axis, either by clicking them one-by-one (with the CTRL key down), or by a box selection. Set the axis color to black. Then, select all facets of the contact carrier plate, and also set their color to black. With a box selection, select all contacts and set their color to a very bright gray. (Figure B-12).

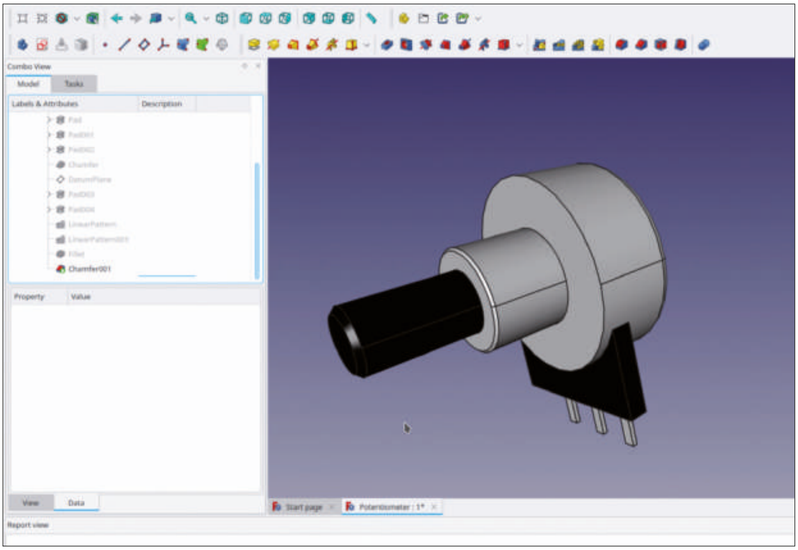


Figure B-12

Now, you add the washer as a separate body object. Click on the blue 'Create body' tool button. If the body is not listed inside the 'Potentiometer Complete' Std-Part-Container, drag-and-drop it there. Rename the new body to 'Potentiometer Washer'. If it is not activated (the title is not shown in bold letters), activate it with a double click.

Click on the 'Sketcher' tool button and select the XY plane from the list as the sketching plane. Like with the rotary switch, draw the contour of the washer with 2 circles, which are both centered around the origin. The inner circle has a diameter of 10.1 mm, set the diameter of the outer one to 16 mm (Figure B-13). Close the sketch.

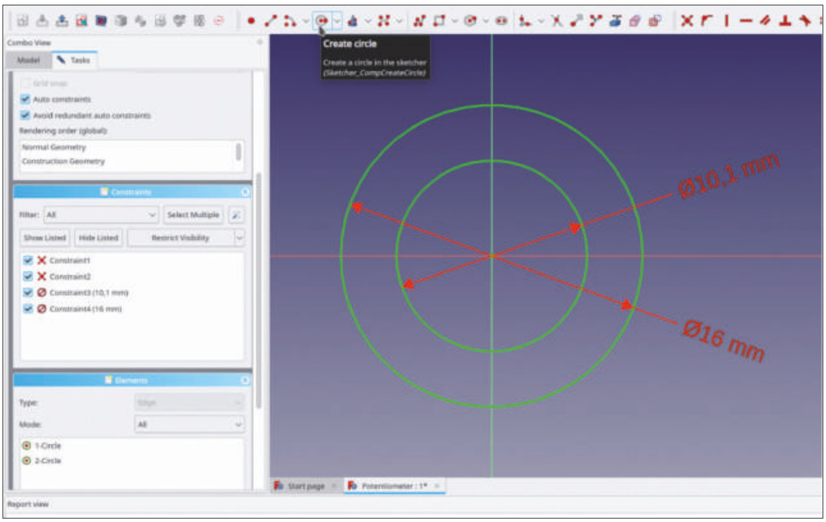


Figure B-13

In the tree view, mark the new sketch and click the 'Pad' tool button. For the thickness of the pad, enter 0.5 mm.

In the tree view, mark the washer body and edit the placement parameters (click into the 'Placement' property line, click the [...] button to open the task window). Set the Z offset to a typical front panel thickness like 2 mm (Figure B-14).

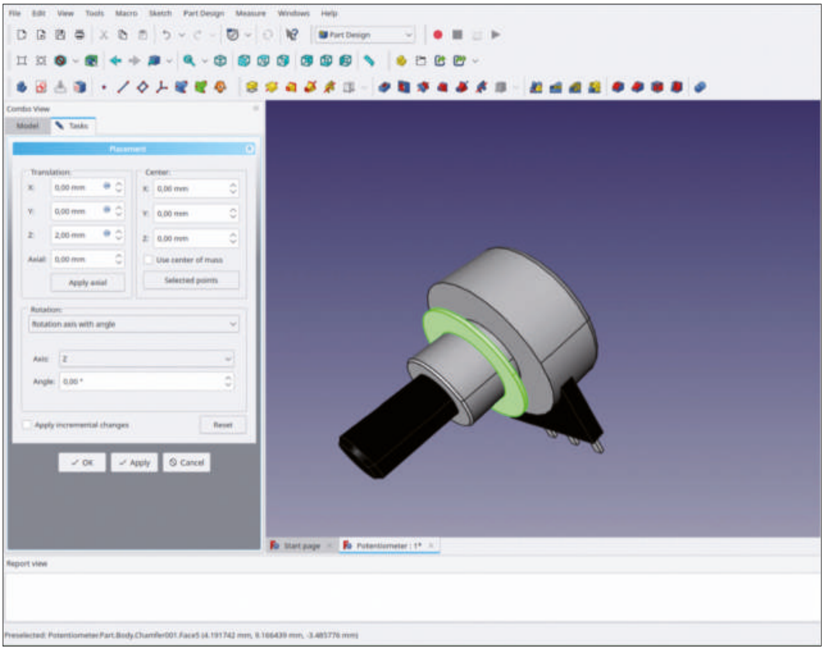


Figure B-14

Add the nut by clicking the blue 'Create body' tool button. Rename the new body to 'Potentiometer Nut'.

By the preceding step, the body 'Potentiometer Nut' is still activated. If not, double-click it to activate it (title is shown in bold letters). Click on the 'Sketcher' tool button and select the XY plane as the sketching plane.

Like with the rotary switch, draw a hexagon centered on the origin, with one corner located on the Y axis. Set the distance (the key width of the nut) between the sides to 14 mm, by selecting two opposite points and clicking the 'Constrain horizontal distance' tool button.

For the inside of the nut, draw a circle, which is centered on the origin, and set its diameter to 10.1 mm (Figure B-15). Close the sketch with the 'Close' button (top).

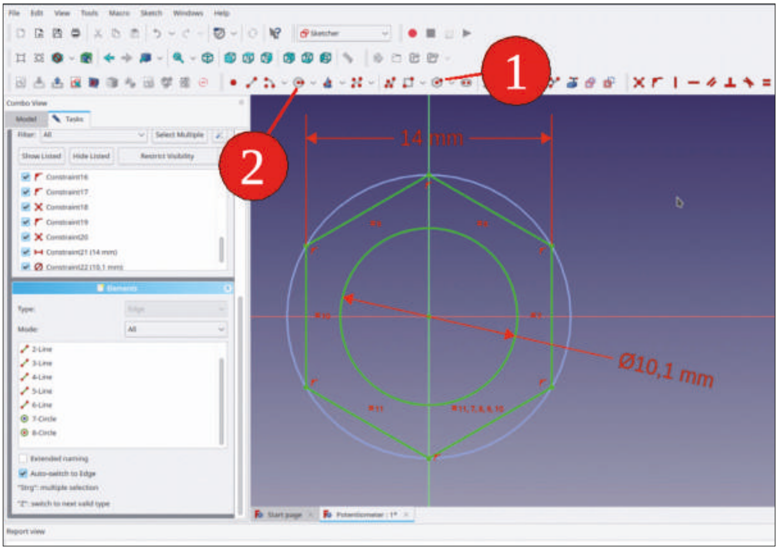


Figure B-15

In the tree view, mark the new sketch, and create a pad with 2 mm thickness.

The nut position should be depending on the washer's location. For the attachment relation between the two bodies, a SubShapeBinder is needed.

In the tree view, hide the 'Potentiometer Nut' body. Also, deactivate it by right-clicking it, and selecting 'Toggle active body' from the context menu. The title of the body object is then shown in normal letters (not bold). In the 3D view, mark the inner edge of the washer, then click the 'Create a sub object(s) shape binder' tool button (Figure B-16).

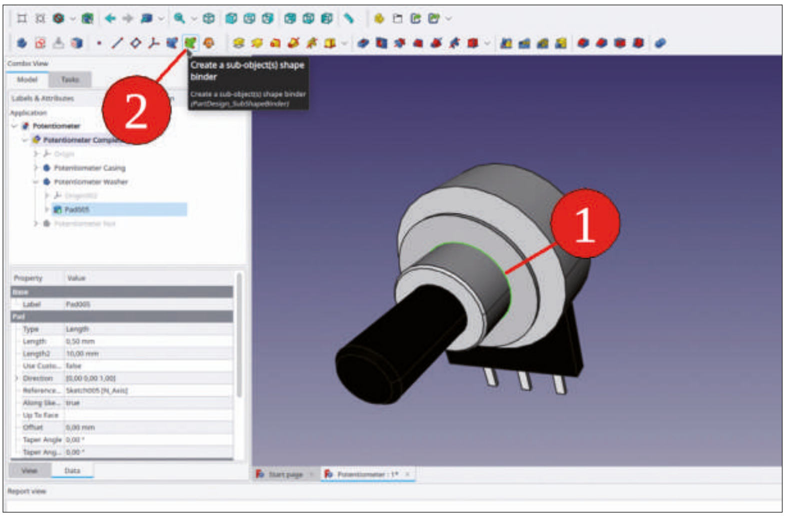


Figure B-16

Drag-and-drop the green SubShapeBinder object into the Std-Part-Container 'Potentiometer Complete' and rename it to 'Position Nut' (Figure B-17).

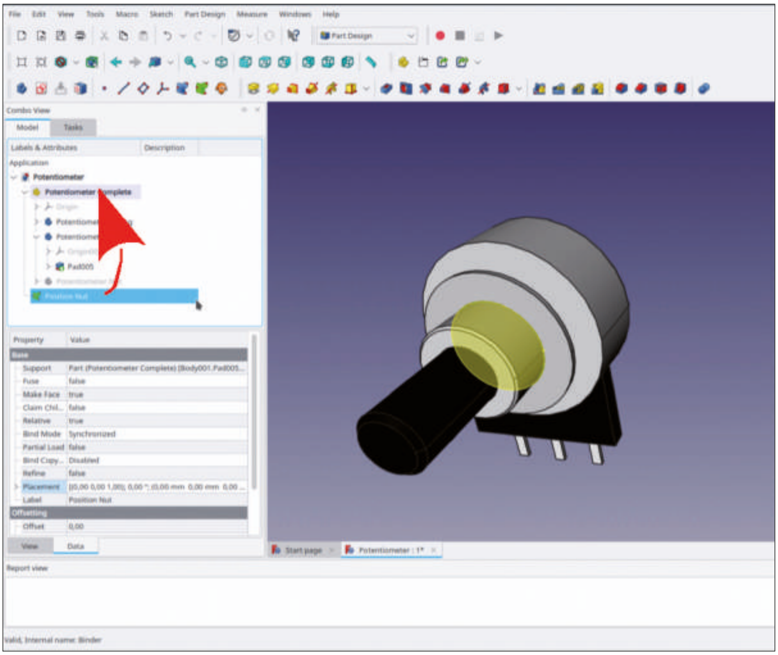


Figure B-17

In the tree view, mark the body 'Potentiometer Nut', and switch to the 'Part' workbench. From the main menu, select 'Part | Attachment...'. The collector for reference 1 should be active by default. Otherwise, click the button 'Reference1' until it reads 'Selecting...', in the edit field next to the button. In the Combo View, switch to the 'Model' tab and select the new SubShapeBinder. Then, return to the 'Tasks' tab.

For the attachment mode, select 'XY on plane' from the list (Figure B-18).

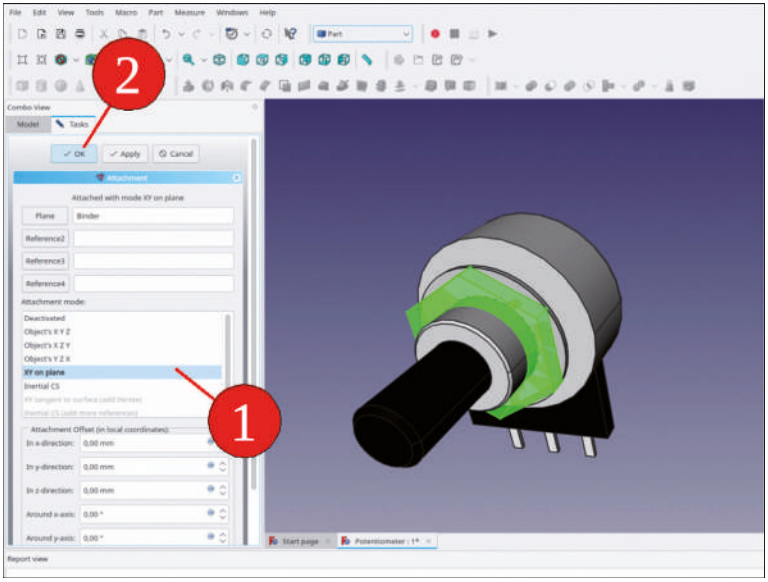


Figure B-18

In the tree view, mark the body objects of nut and washer. Right-click the selection and select 'Appearance' from the context menu. In the task window, set the material to 'Chrome'.

In the tree view, show the nut, and hide the SubShapeBinder 'Position Nut' (Figure B-19).

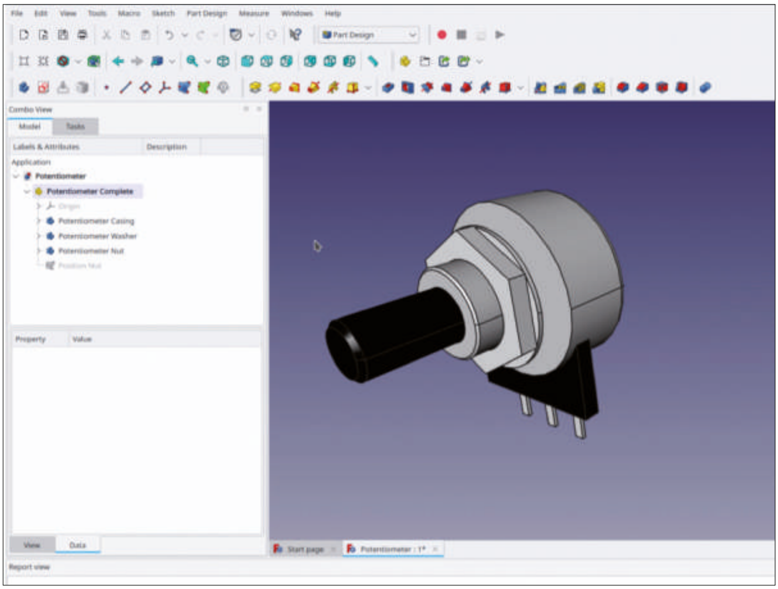


Figure B-19

Save your work.

Appendix C • The Banana Jack

It is not evident from the connector proper how the reference to a banana came about. The name may be due to the contact spring in early versions of the plugs, where the body of the contact pin had a milled groove, in which – indeed - a banana-shaped spring inserted, with the back of the 'banana' pointing outward, protruding from the slot. The mating socket is also called 'wander socket' by veterans in electronics.

Let's generate the Banana Jack as a compound part with several items embedded in its Std-Part-Container.

First, follow the usual procedure: Create a new file and save it as 'Banana Jack'.

Switch to the Part Design' workbench and start a new Std-Part-Container. Rename that to 'Banana Jack Complete'.

Create a new body object and rename that to 'Banana Jack Cap'. Show the coordinate system of the new body object by marking it in the 3D view and pressing the SPACE key (Figure C-1).

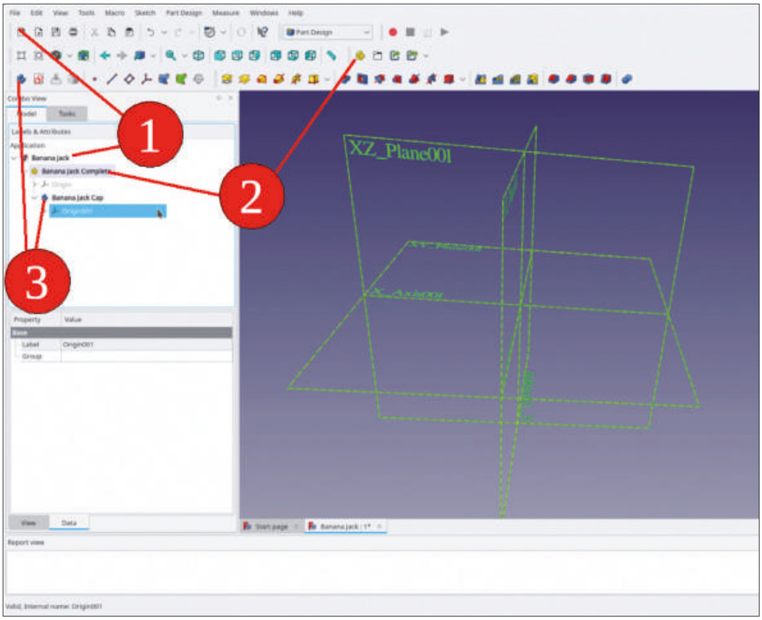


Figure C-1

Click the 'Sketcher' tool button and select the XY plane from the list, as the sketching plane. Alternatively, in the 3D view, mark the XY plane and then start the sketcher.

Click on the 'Create circle' tool button and draw two circles centered around the origin. End the drawing command with a right-click.

Scroll down to the 'Elements' panel. When doing that, beware of a pitfall: if you scroll with the mouse wheel, and the mouse happens to hover over the 'Mode' Combo Box, quite possibly, instead of scrolling the view downwards, the mode selection is changed (e.g., to 'External'). Then, there are no entries in the list. Make sure that 'Mode' is set to 'All' in order to list all the elements of the sketch. Right-click the inner circle and select 'Diameter Constraint' from the context menu. Enter 4.6 mm for the diameter value (Figure C-2).

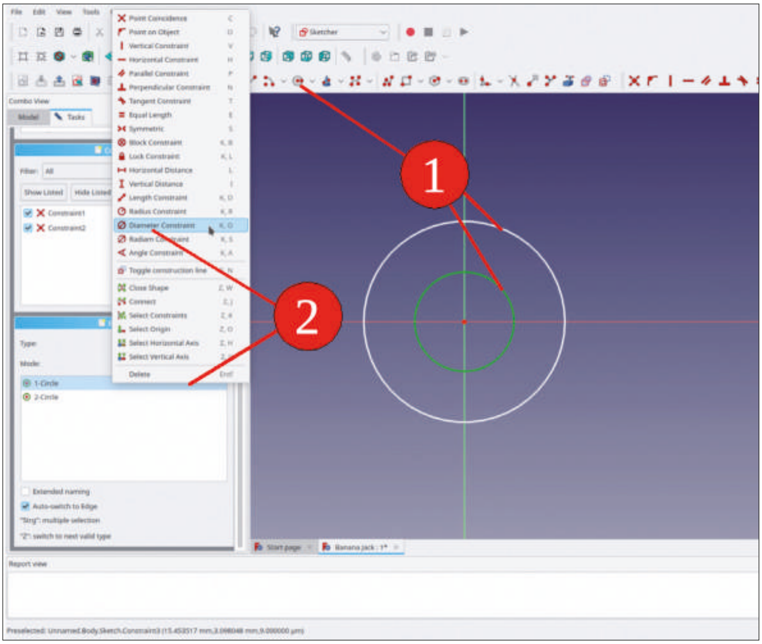


Figure C-2

In the same way, constrain the diameter of the outer circle to 9.9 mm (Figure C-3). Close the sketch with the 'Close' button (top).

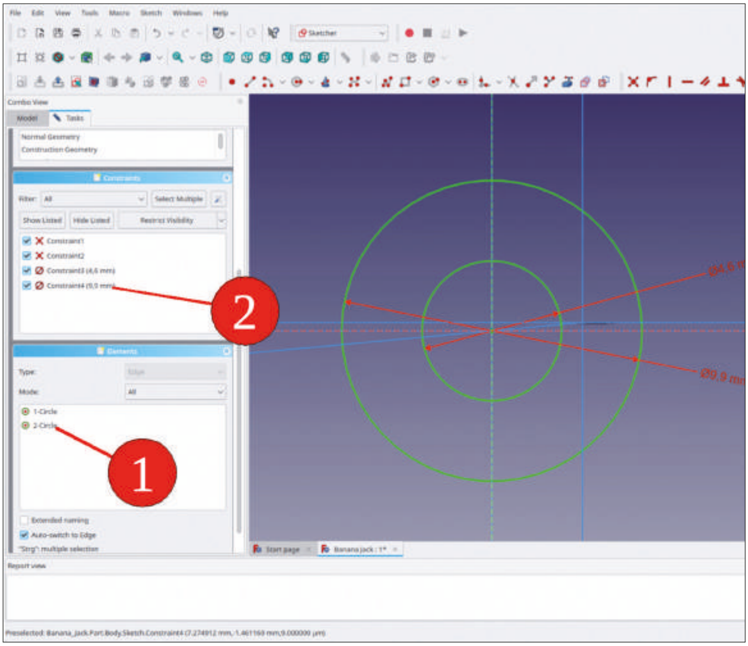


Figure C-3

In the tree view, mark the new sketch and click the 'Pad' tool button. In the task window, enter 5 mm for the length. Close the task with the OK button (Figure C-4).

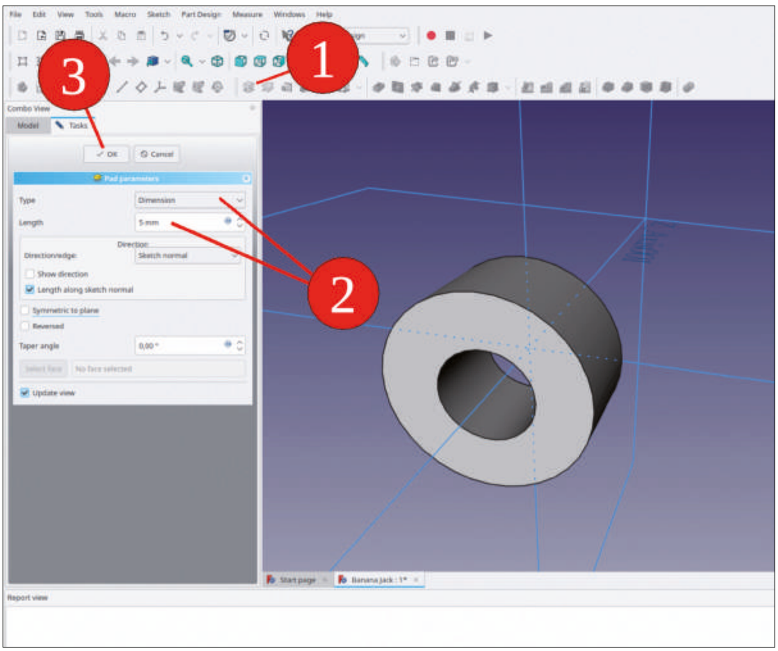


Figure C-4

In the 3D view, select the XY plane again and click the 'Sketcher' tool button. If the cap already created is not shown, close and reopen the sketch. Now you can create the part of the cap which protrudes through the front panel to the rear. To have a less distorted view, select 'Sketch | Show section' and 'View | Orthographic view' from the main menu.

To create a reference to the hole in the cap, click the 'External geometry' tool button, and then mark the inner edge of the cap. The edge is displayed in violet now (Figure C-5) and is added as a construction line to the 'Elements' list. Again, draw two circles that are centered around the origin.

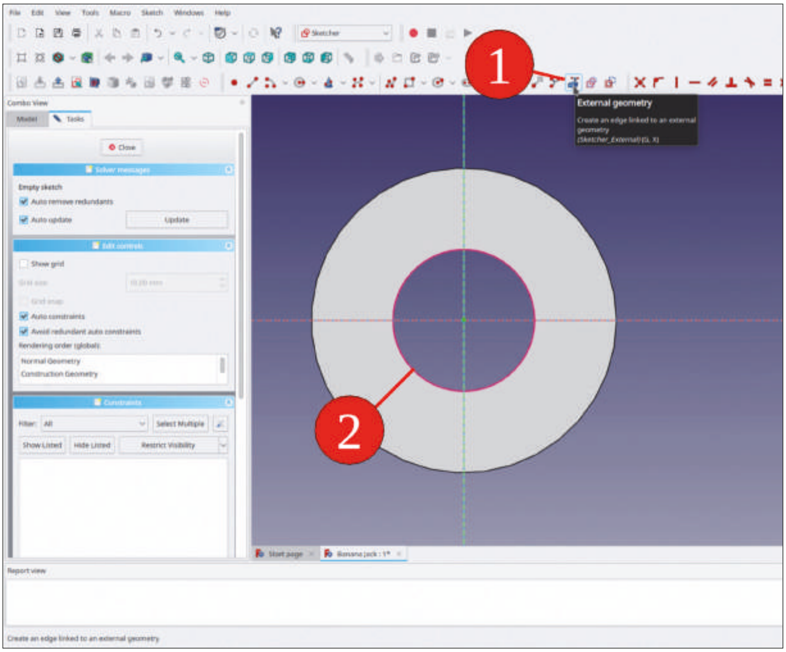


Figure C-5

On the sketch, mark one of the two new circles and the violet reference. To match the diameter of the circles, click the 'Constrain equal' tool button (Figure C-6).

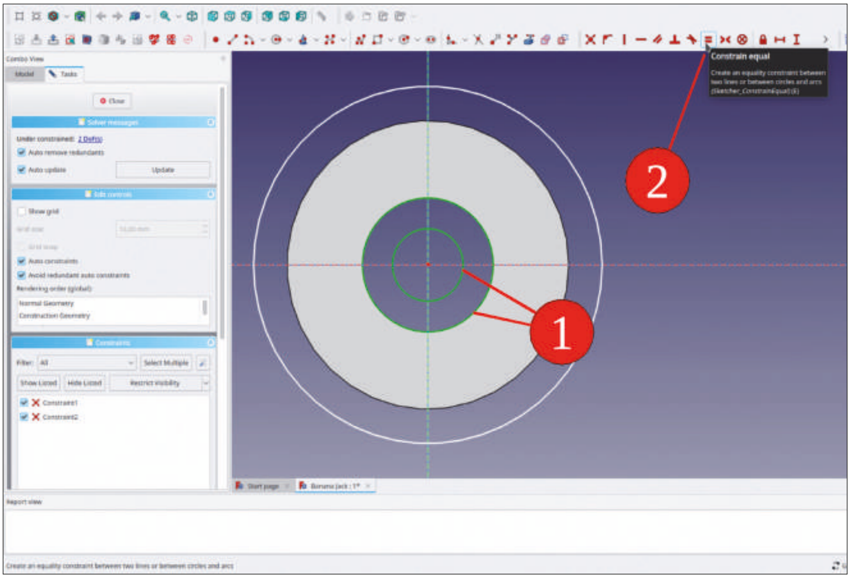


Figure C-6

In the 'Elements' list, find the outer circle (by clicking entries, the corresponding sketch element is highlighted in green). Right-click the outer circle and select 'Diameter constraint' from the context menu. Enter a value of 7.85 mm for the stub (Figure C-7).

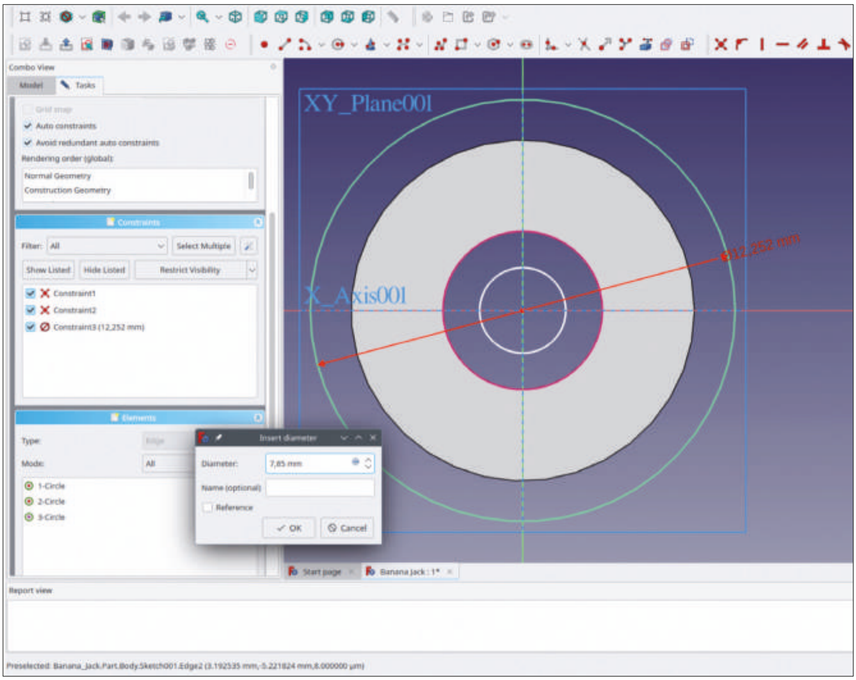


Figure C-7

In the tree view, mark the new sketch and click the 'Pad' tool button. Set the length of the extrusion to 4 mm and check the 'Reversed' checkbox, as the shaft protrudes to the back (Figure C-8). Close the task window with the 'OK' button.

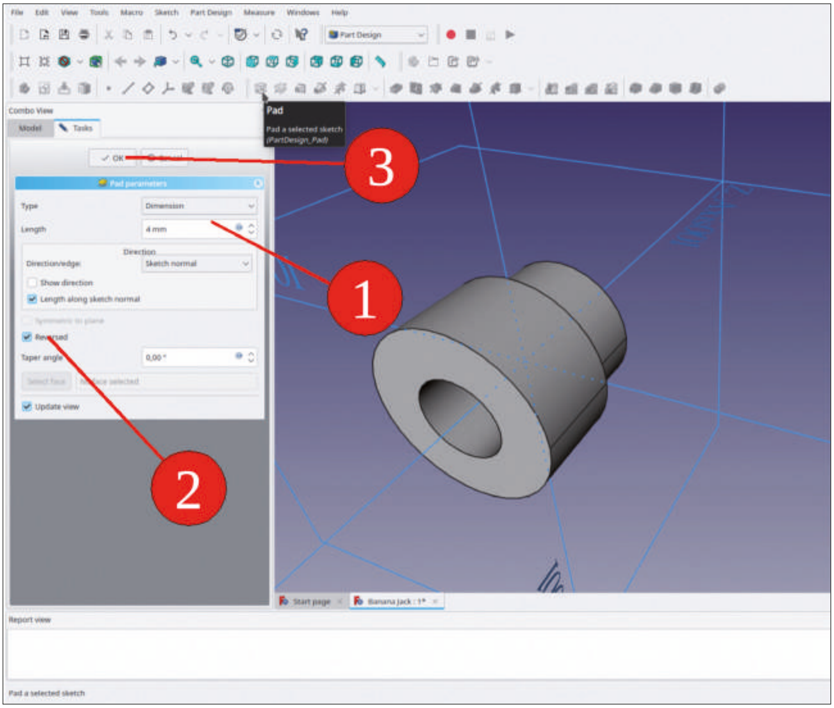


Figure C-8

Because no further changes are expected for the banana jack cap, the front edges could be rounded as cosmetic features.

In the 3D view, mark the two front edges of the cap (CTRL key down) and click the 'Fillet' tool button (Figure C-9). In the task window, set the radius parameter to 0.25 mm. Now, the cap starts to look realistic (Figure C-10).

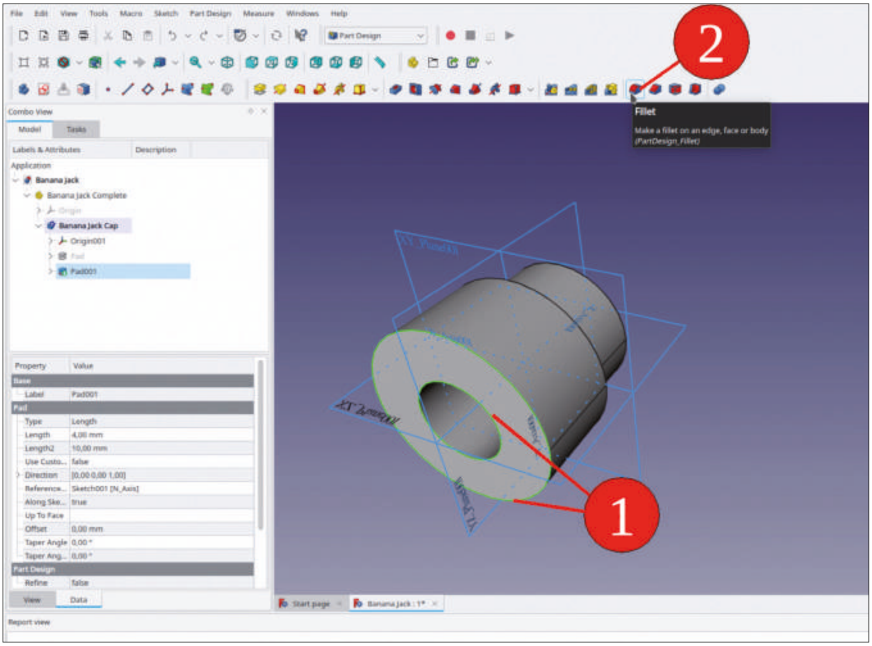


Figure C-9

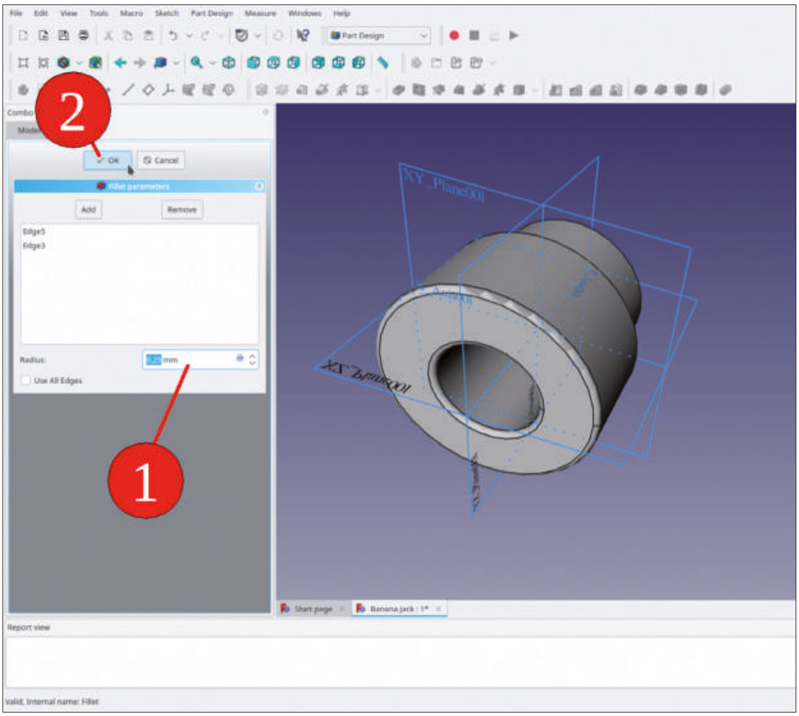


Figure C-10

We made the selection before the tool 'Fillet' was called, which already added both features to the list in the task window. If you would like to add more details, it would be necessary to click the 'Add' button prior to each new selection from the 3D view.

The realistic impression can be further enhanced by appearance attributes: Right-click the 'Banana Jack Cap' body and select 'Appearance...' from the context menu. For the material, select 'Shiny Plastic', and for the color, a dim red. Close the task window with the 'Close' button in the bottom. Hide the body's coordinate system (Figure C-11).

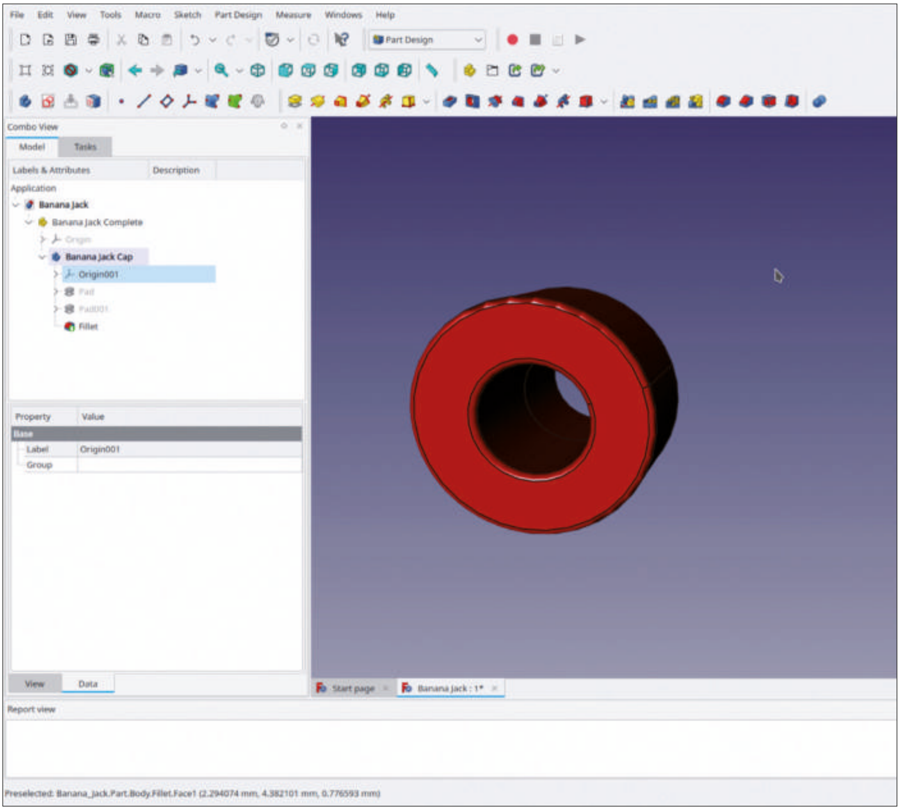


Figure C-11

This appearance is not cast in stone. Later, the color of each banana jack can be individually changed in an assembly. This is especially important for these jacks, which come in a multitude of colors, and usually carry a special meaning.

Now, the contact of the banana jack is created. Start a new body object by clicking the blue 'Create body' tool button. Rename it to 'Banana Jack Contact'. If it has been created outside of the 'Banana Jack Complete' Std-Part-Container, just drag and drop it in there.

In the 3D view, display the coordinate system of the new body (mark it in the tree view and press the SPACE key). The contact has a zone where the diameter reduces towards a soldering stud. The reference to the soldering stud can be created with a datum plane. In order to define the datum plane, in the 3D view, mark the XY plane of the coordinate system. Then, click the 'Create a datum plane' tool button.

In the task window, set the distance between the planes with the 'Attachment Offset' Z parameter. Set it to −14.5 mm (Figure C-12).

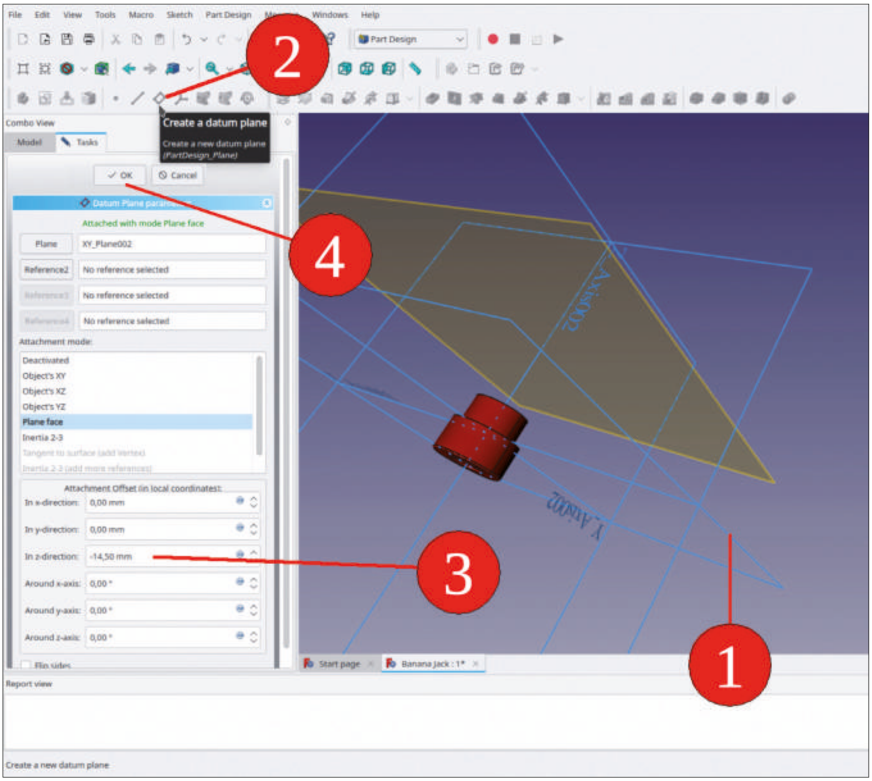


Figure C-12

Draw the outline of the contact: In the 3D view, mark the XY plane and start the sketcher. Draw a circle which is centered about the origin and set its diameter to 5.9 mm. Close the sketch.

In the tree view, mark the new sketch, and click the 'Pad' tool button. In the task window, select the type 'Two dimensions'. For the length, enter a value of 4.5 (protruding to the front), for the 2nd length, enter 14.5 mm (Figure C-13). Close the task window with the 'OK' button.

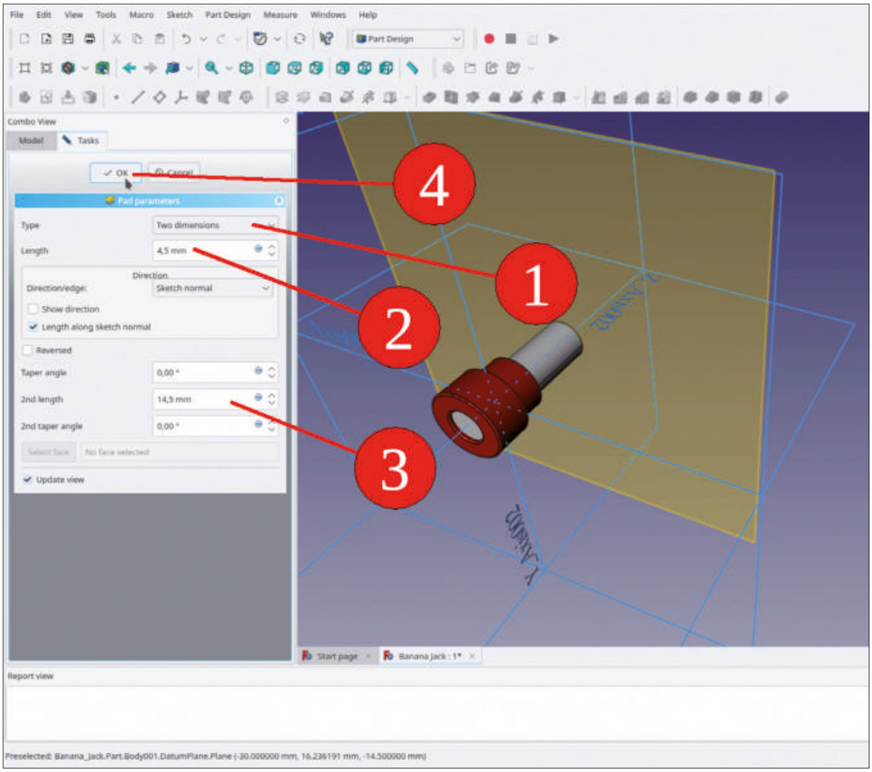


Figure C-13

Rename the datum plane to 'Position Soldering Pad'. With that plane marked in the tree view, start the sketcher. For the soldering pad, draw a circle centered around the origin, and set its diameter to 3.2 mm. Close the sketch with the 'Close' button (top).

In the tree view, mark the new sketch and click the 'Pad' tool button. In the task window, set the length to 4 mm and check the 'Reversed' checkbox (Figure C-14).

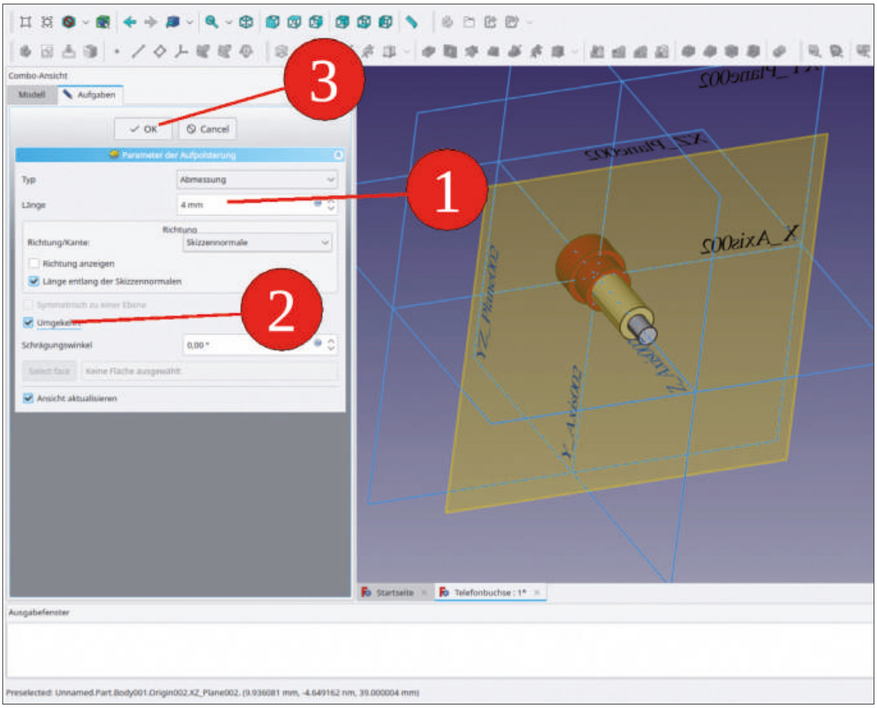


Figure C-14

In order to create the hole for the plug, in the 3D view, mark the XY plane again and start the sketcher. Draw a circle centered around the origin again and constrain its diameter to 4 mm (the diameter which accepts the plug).

In the tree view, mark the new sketch and click the 'Pocket' tool button. In the task window, select for the type 'Two dimensions'. Set the length to 5 mm (pointing to the front) and for the 2nd length 13 mm (pointing backwards). Check the 'Reversed' checkbox. Close the task window with the OK button (Figure C-15).

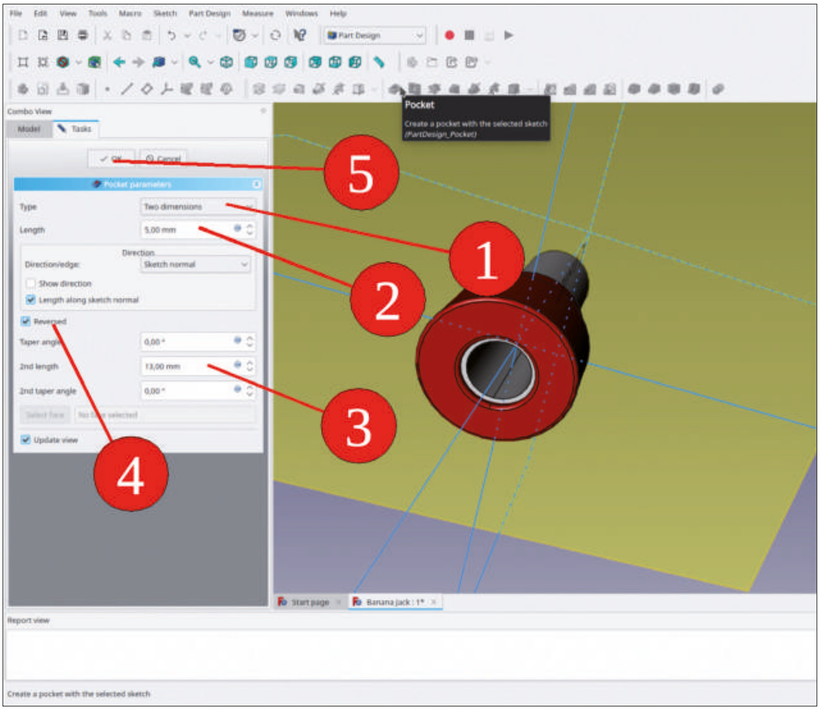


Figure C-15

The solder stud has a slot where a wire can be placed. To sketch that, in the 3D view, mark the YZ plane and start the sketcher. In order to display the already-present 3D geometry, close the sketch and reopen it by double-clicking it in the tree view. In order to have a less distorted view, select 'Sketch | Show section' and 'View | Orthographic view' from the main menu.

Sketch a rectangle that is centered on the vertical axis, and constrain dimensions and distance as shown in Figure C-16. Close the sketch.

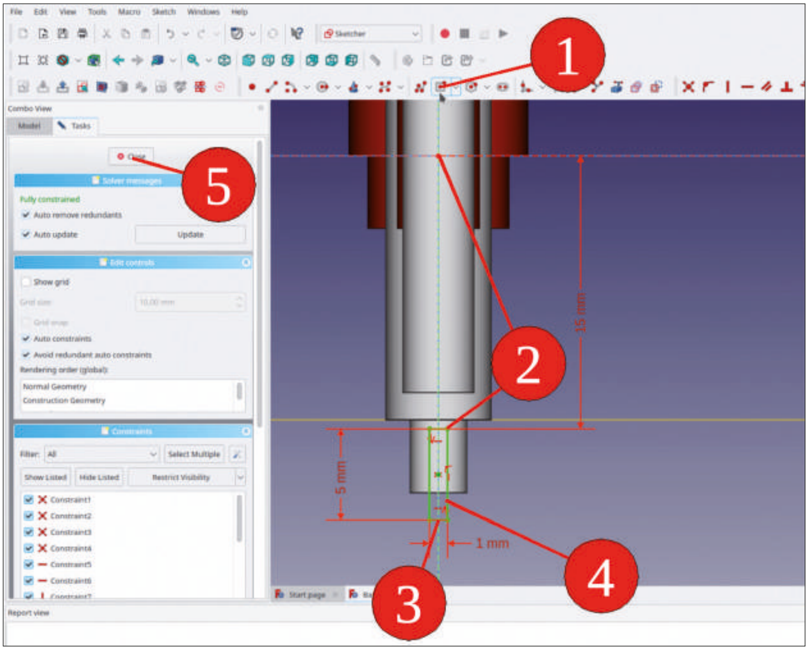


Figure C-16

In the 3D view, mark the new sketch and click the 'Pocket' tool button. For the type, select 'Through all', and check the 'Symmetric to plane' checkbox (Figure C-17).

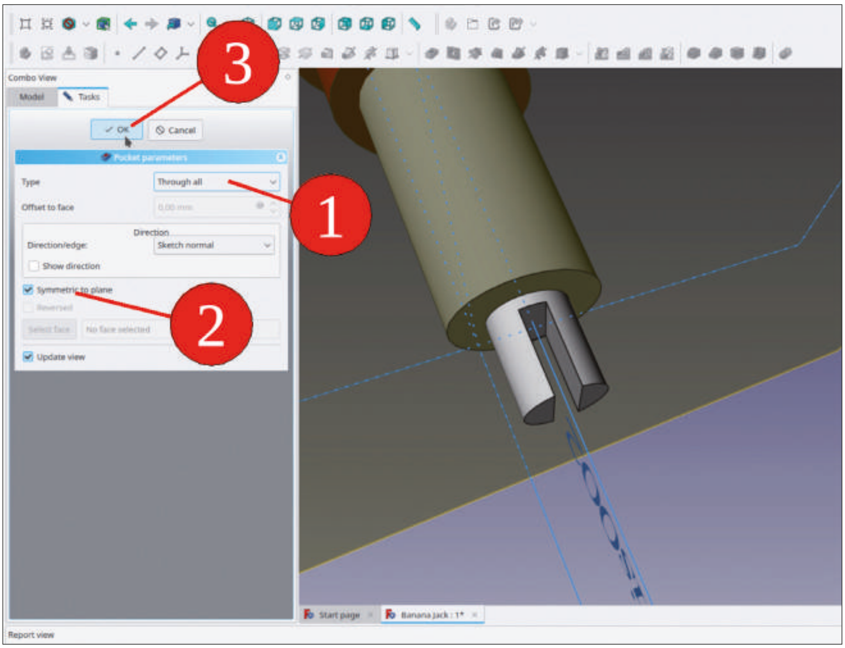


Figure C-17

Create the tapered section between the soldering stud and contact: Mark the larger rear edge of the contact and click the 'Chamfer' tool button. For the size, enter 1.2 mm (Figure C-18).

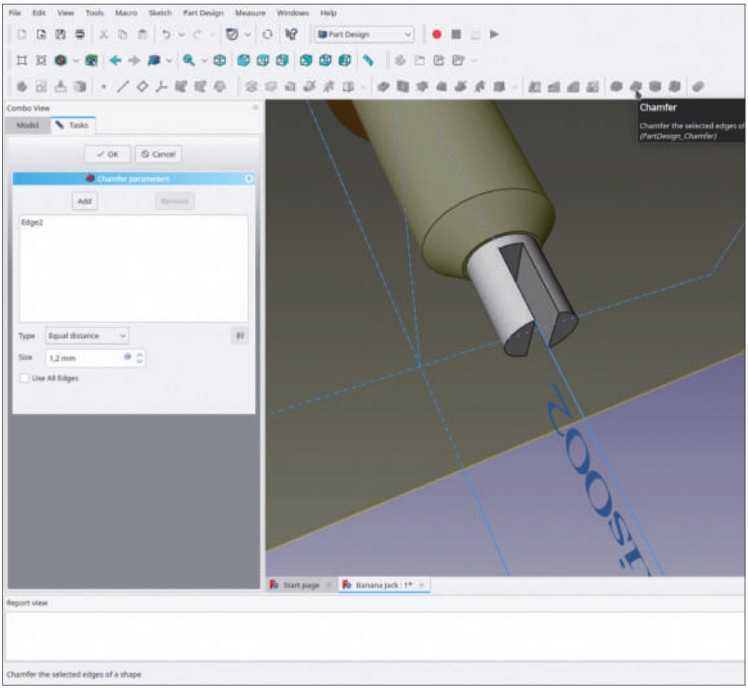


Figure C-18

In the tree view, hide the datum plane and the coordinate system. Right-click the 'Banana Jack Contact' body and select 'Appearance…' from the context menu. Set the material to 'Chrome' and close the task window (Figure C-19).

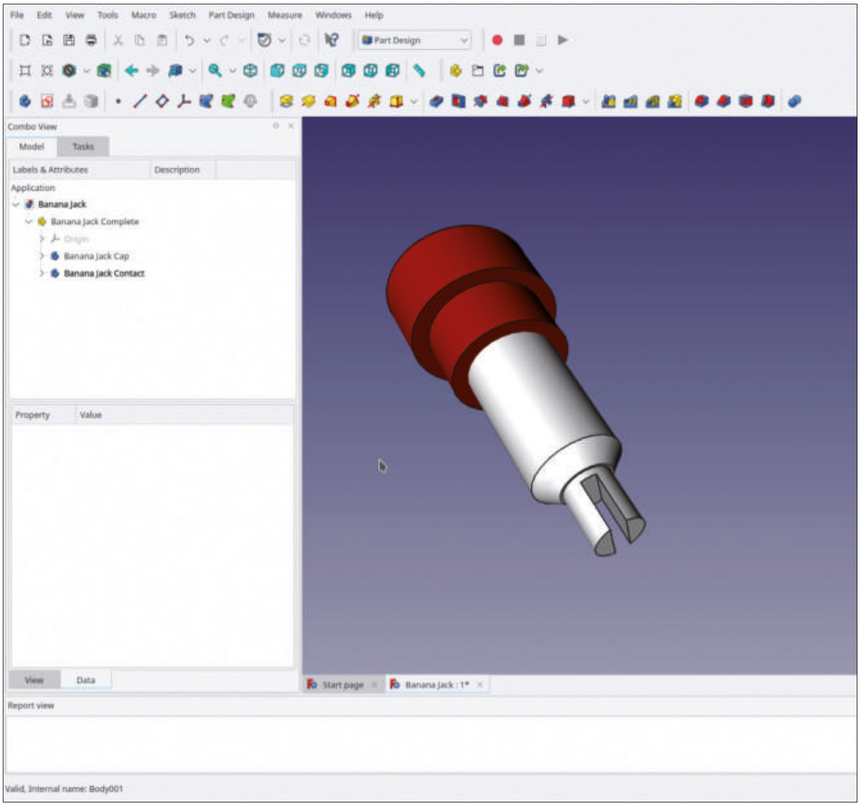


Figure C-19

These banana jacks have an insulating washer for the rear side, which can accommodate the thinner part of the cap. Start the creation of the washer by clicking on the 'Create body' tool button. Rename the new body to 'Banana Jack Washer'.

Display the coordinate system of the new body (mark in tree view and press SPACE key).

In the 3D view, mark the XY plane (only that appears green then!) and start the sketcher.

For the outline of the washer, draw two circles that are centered on the origin. Constrain the diameter of the two circles to 6.4 mm and 10.4 mm. Close the sketcher.

In the tree view, mark the new sketch and click the 'Pad' tool button. Set the length of the pad to 4.4 mm and check the 'Reversed' checkbox, to extrude it towards the back (Figure C-20).

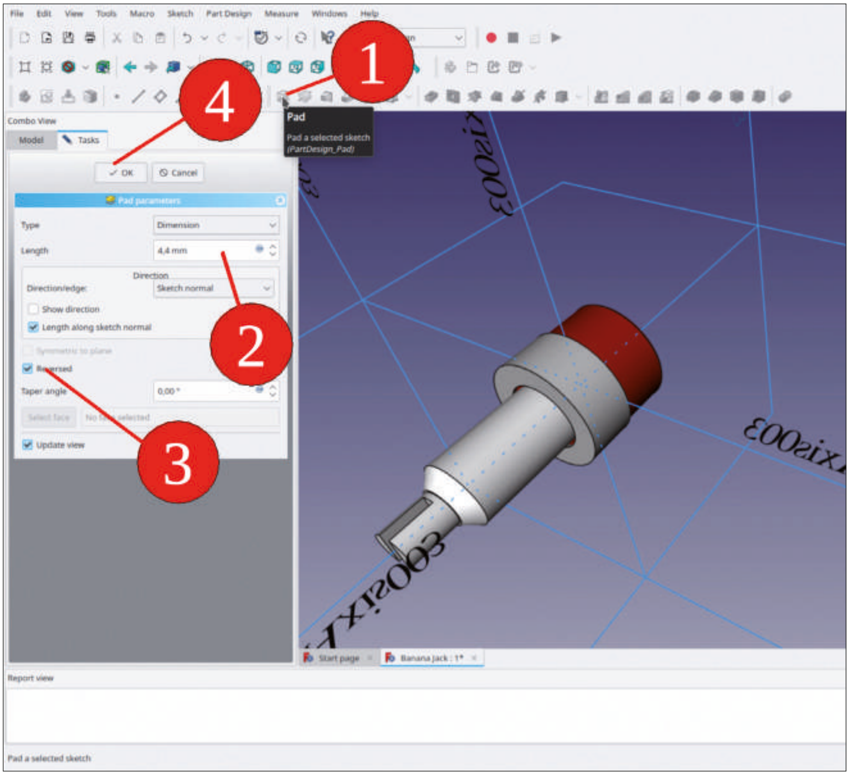


Figure C-20

For the washer, the inner pocket which houses the thinner part of the cap is still missing. In order to create that pocket, in the 3D view, mark the XY plane again and start the sketcher.

For the pocket, draw a circle centered around the origin, and constrain the diameter to 8 mm.

In the tree view, mark the new sketch, and click the 'Pocket' tool button. In the task window, set the length to 3.4 mm (Figure C-21).

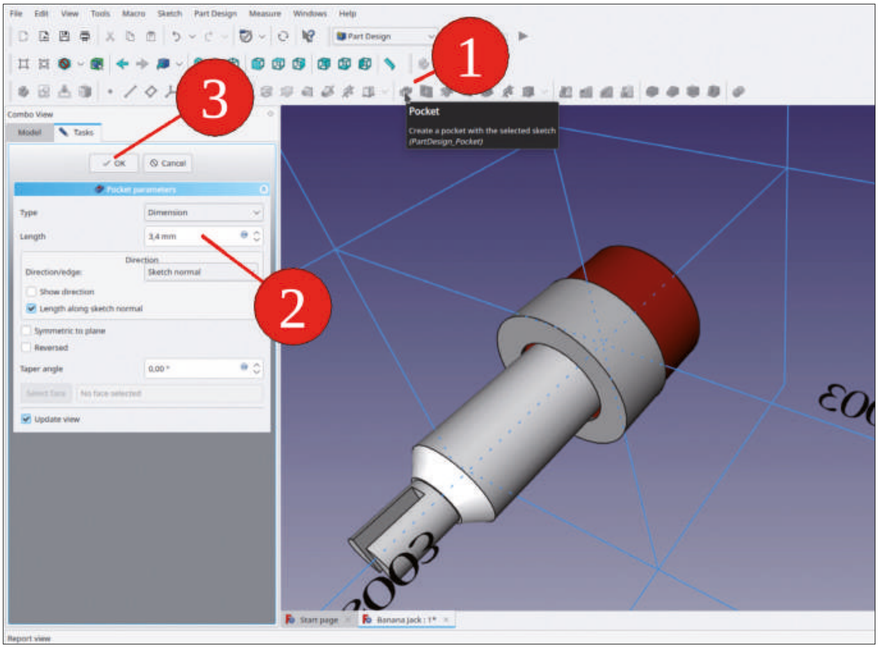


Figure C-21

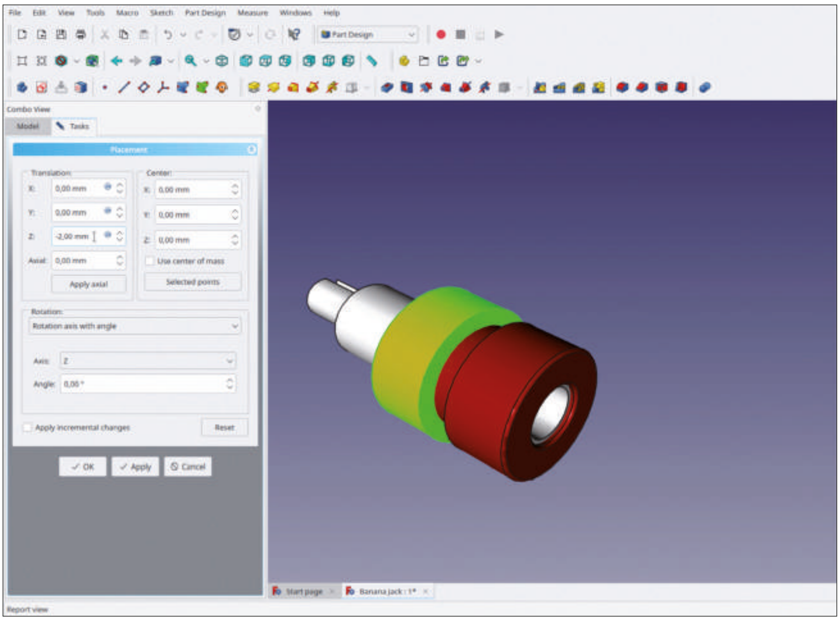


Figure C-22

Right-click the body of the washer and select 'Appearance…' from the context menu. For the material, select 'Shiny Plastic'. Set the color to the one of the cap.

In the tree view, mark the washer body. In the property list, click the 'Placement' edit field, and then on the ... button. In the task window, move the washer by a typical front panel thickness to the rear, e.g., by 2 mm (Figure C-22). This setting can easily be matched to different panels later on. Close the task window with the OK button.

Next, the nut is added. Create a new body by clicking on the blue 'Create body' tool button. Rename it to 'Banana Jack Nut'.

Hide all the body objects, except for the nut, and display the coordinate system of the nut body.

In the 3D view, mark the XY plane and start the sketcher.

Draw the outline of the nut: Click the 'Create regular polygon' tool button (the hexagon is selected by default). With the first click, select the origin. Lock the second click the Y axis at some distance, this will make it easier to define the dimension (Figure C-23).

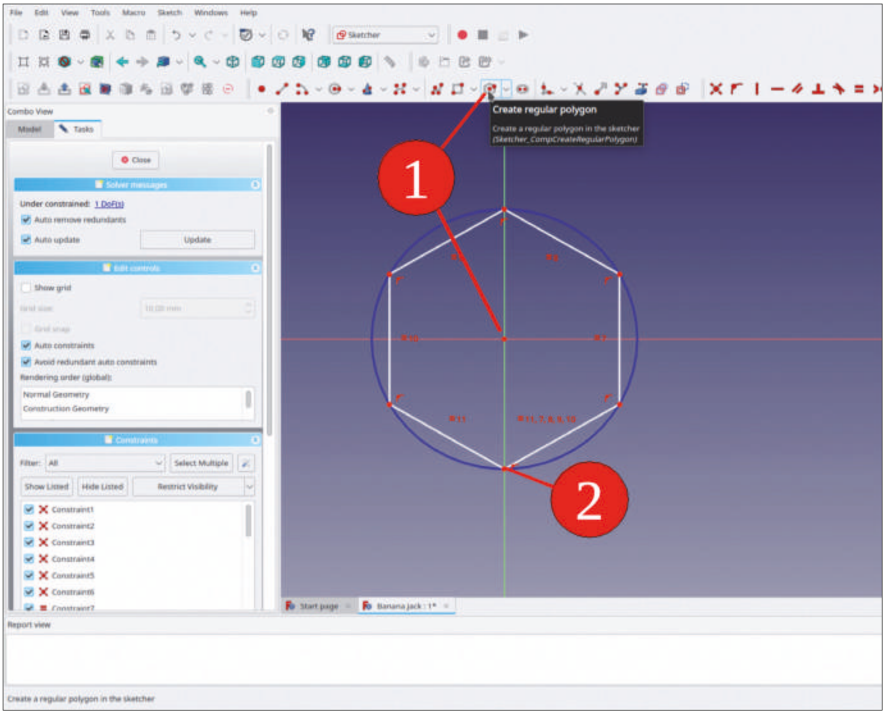


Figure C-23

Mark two points opposite to each other, which are not located on the Y axis. Constrain the horizontal distance to 8 mm.

Draw a circle centered around the origin and constrain the diameter to 5.9 mm (Figure C-24). Close the sketch.

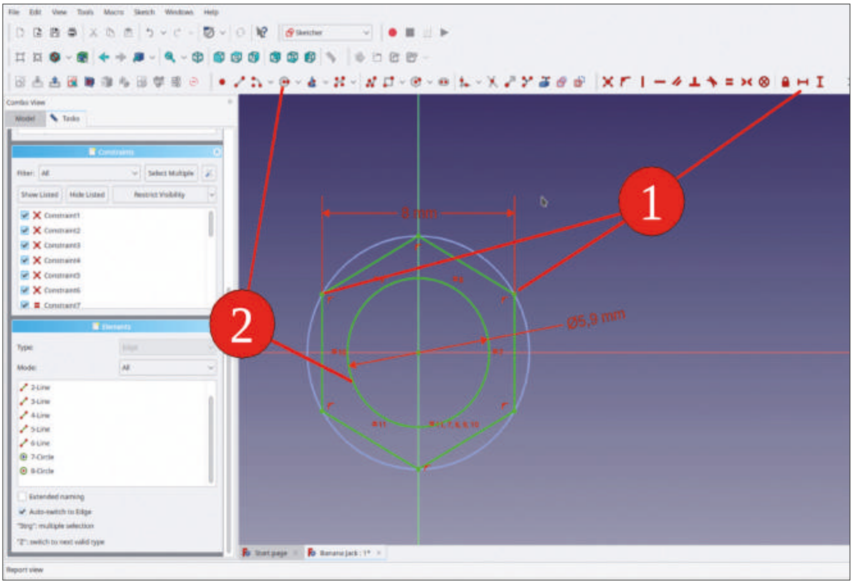


Figure C-24

In the tree view, mark the new sketch and click the 'Pad' tool button. In the task window, set the length to 2 mm and check the 'Reversed' checkbox (Figure C-25).

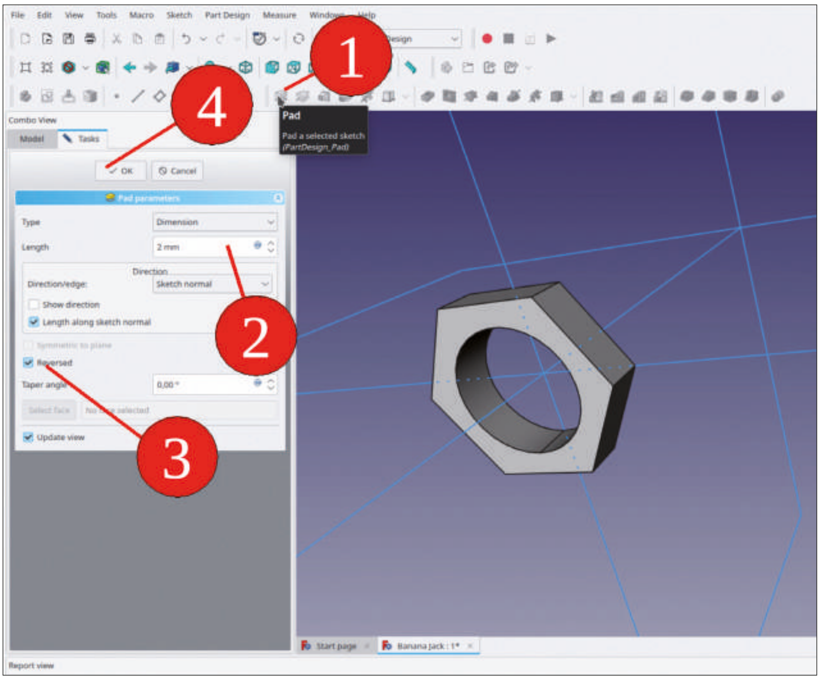


Figure C-25

The nut should always be located directly behind the washer. This can be automated, when an attachment relation to the washer is defined. To communicate this across the scope of the individual bodies, a SubShapeBinder is needed.

In the tree view, deactivate all activated objects except the Std-Part-Container 'Banana Jack Complete (the destination of the new SubShapeBinder). You can deactivate any activated body object by right-clicking it and selecting 'Toggle active body' from the context menu. Eventually, you need to select this item twice.

Hide the coordinate system of the nut body, then hide the nut body itself. Display the 'Banana Jack Washer' object (Click on it in the tree view and press the SPACE key).

In the 3D view, click the inner rear edge of the washer, and then on the green 'Create a sub object(s) shape binder' tool button (Figure C-26).

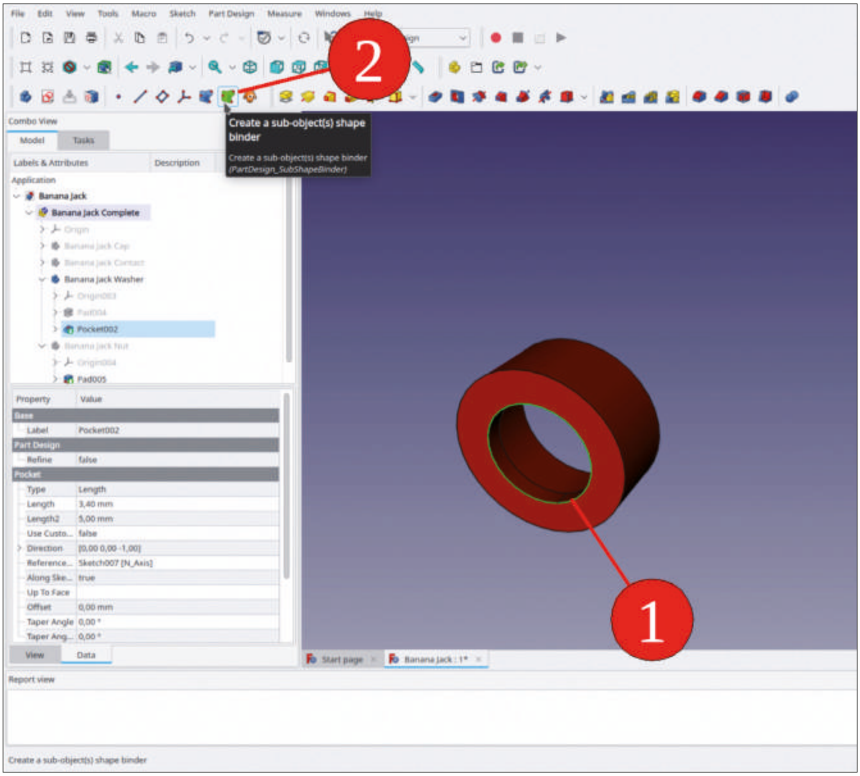


Figure C-26

In the tree view, rename the SubShapeBinder to 'Position Nut'. Drag-and-drop it into the Std-Part-Container 'Banana Jack Complete' (Figure C-27). In this way, things stay together when inserting the Banana Jack Container later. In the tree view, display the body of the nut (click and SPACE key).

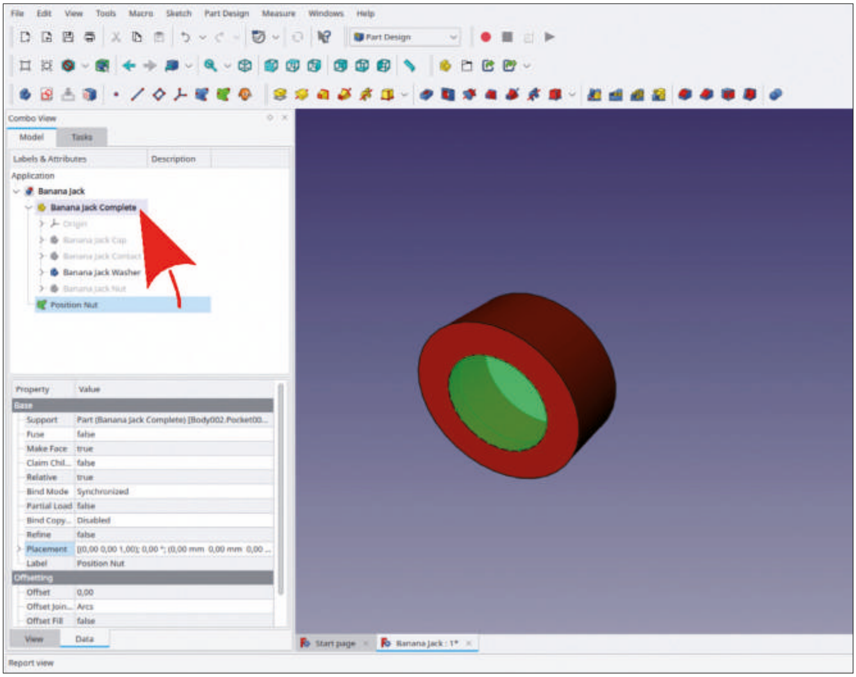


Figure C-27

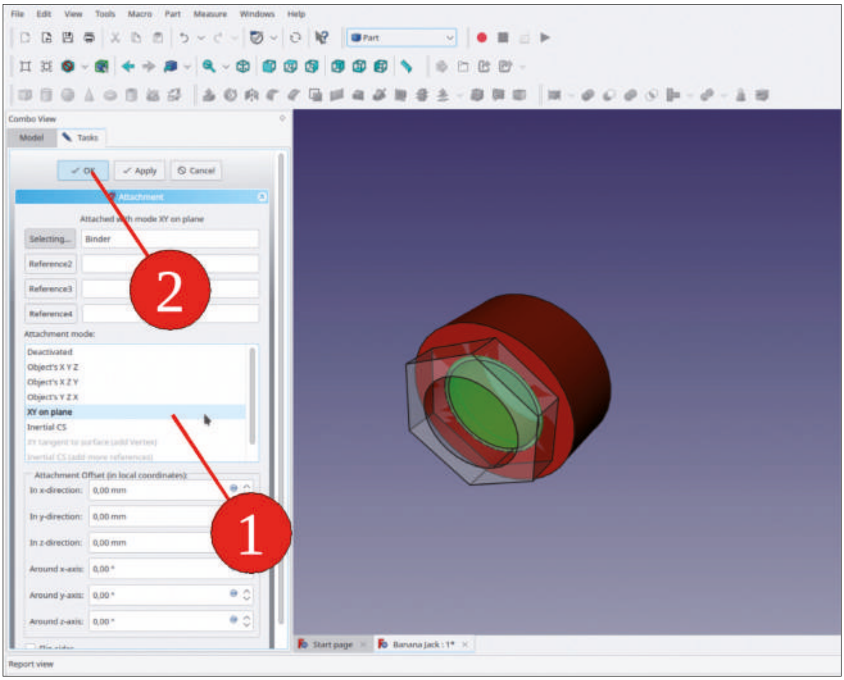


Figure C-28

With the nut body marked, switch to the 'Part' workbench and select 'Part | Attachment' from the main menu.

In the task window, the collector button for 'Reference1' should read 'Selecting…'. Otherwise, click that button, until this is the case, and then into the corresponding edit field.

Switch to the 'Model' tab and, in the tree view, click the SubShapeBinder.

Return to the 'Tasks' tab. As the attachment mode, select 'XY on plane' from the list (Figure C-28).

In the tree view, hide the SubShapeBinder and display all body objects.

Test the associativity: In the tree view, click the 'Banana Jack Washer' body, in the property list, click the 'Placement' field, and there, click the … button to display the task window.

In the task window, click into the Z translation field and change the washer position with the mouse wheel. The washer moves instantaneously. To update the nut position, click either the 'Apply' or the 'OK' button. The latter also closes the task window.

Revert the washer to a Z translation of 2 mm (Figure C-29). Save your work!

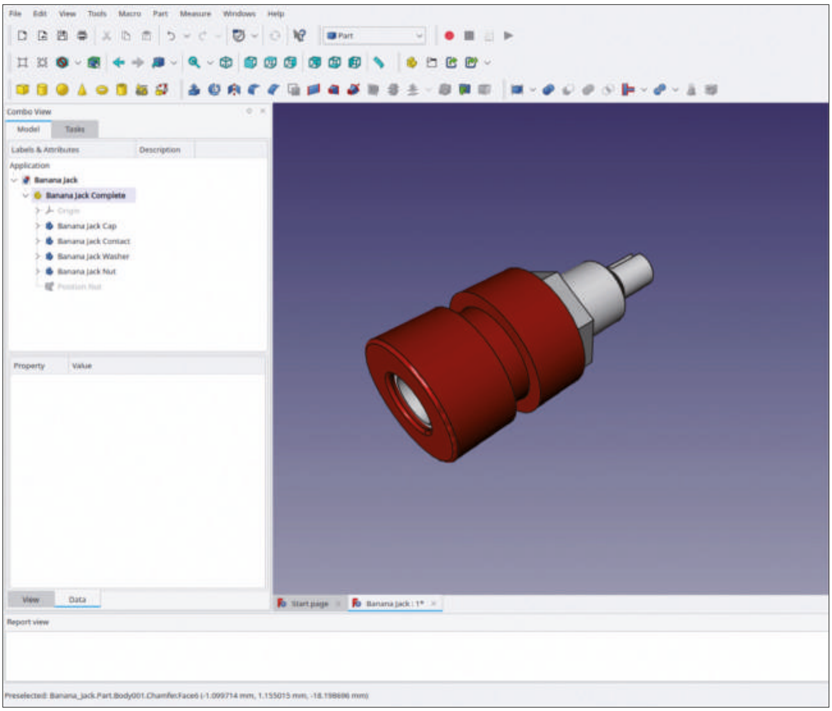


Figure C-29

Appendix D • The Pilot Lamp

Again, run the standard procedure: Start a new file and save it as 'Pilot Lamp'. Switch to the 'Part Design' workbench and create a new Std-Part-Container. Rename it to 'Pilot Light Complete'. Then, start a new body object and rename that to 'Pilot Lamp Casing'.

We plan the casing to be oriented and positioned like the potentiometer and the rotary switch — with the XY plane coinciding with the front panel surface. The pilot lamp casing has several features, for which you can generate the reference planes right away. These features are:

- The top of the opaque casing protruding from the front panel (Z = 5 mm).
- The length of the large fine threaded part, where the fastener is later on attached (Z= –14.9 mm).
- The plane where the contact terminals reside on (Z = –25 mm).
- The end face of the insulating barrier between the two contacts (Z = –33 mm).

In the tree view, display the coordinate system of the new body object with the SPACE key. In the 3D view, select the XY plane (and only that — click the blue to deselect the whole coordinate system, then the XY plane).

Then, click the 'Create a datum plane' tool button. With the XY plane preset, the task window for the datum plane opens already with the datum plane attached to the XY plane. Set the Z parameter to the value of 5 mm for the first feature listed above. Close the task window with the OK button. In the tree view, rename the new datum plane to 'Casing Top Face'.

In similar fashion, create and rename the other datum planes for the features listed above. It may look a bit tedious to rename all the planes with self-explaining names. Later, this can be a good time saver. The achieved state of the project is depicted in Figure D-1.

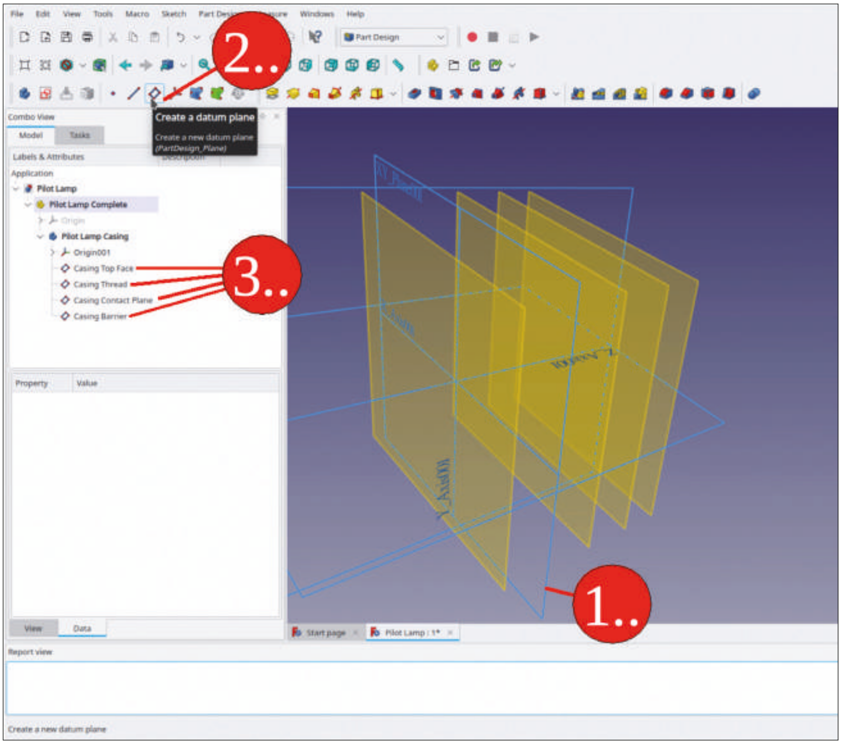


Figure D-1

With these preparations done, the casing can be modeled. Start with the conical part that protrudes from the front panel. For this, a 'Loft' object is needed, which is defined by two sketches defining the cross-sections, at the start and at the end of the profile. With a caliper, the diameter of the conical part can be found to be 20.7 mm at the front panel and 18.5 mm where the casing has the top face.

To begin the loft, sketch the first circle: Mark the XY plane and click the 'Sketcher' tool button. Draw a circle that is centered around the origin and constrain its diameter to 20.7 mm. Close the sketcher.

In the same way, sketch a circle centered around the origin onto the 'Casing Top Face' datum plane. Constrain the diameter of this circle to 18.5 mm.

In the tree view, mark the two sketches, and click the 'Loft' tool button (Figure D-2).

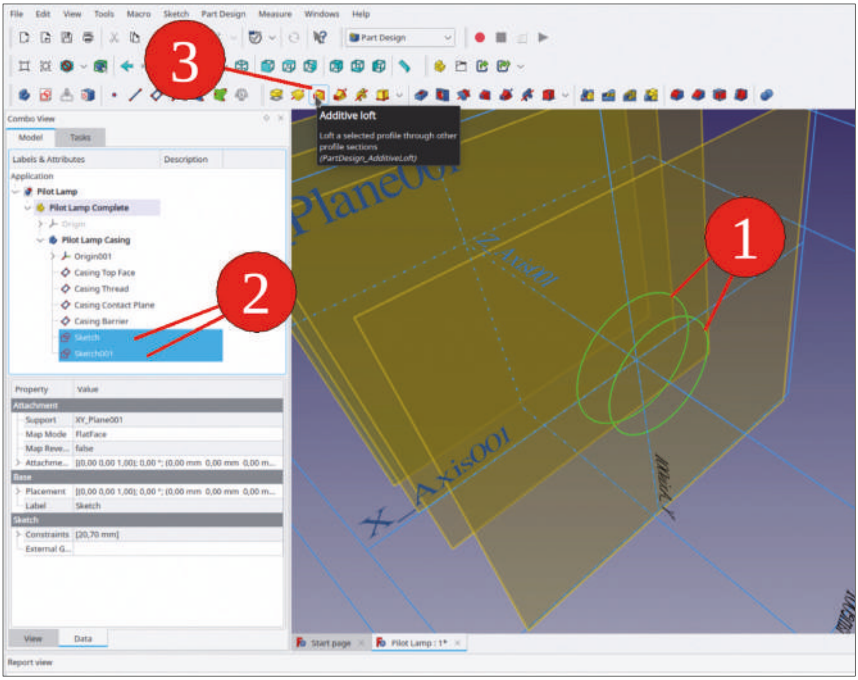


Figure D-2

In the task window, check the 'Closed' checkbox. Close the task window with the OK button (Figure D-3). In the tree view, rename the 'AdditiveLoft' feature to 'Casing Collar'.

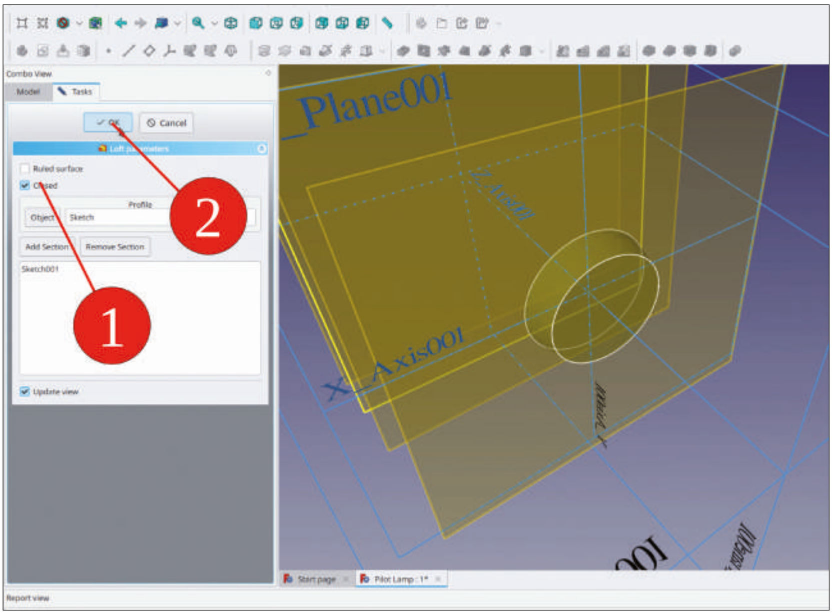


Figure D-3

For the threaded part of the casing, draw another circle on the XY plane, which is centered around the origin. The thread has an outer diameter of 15.8 mm (with a little extra tolerance added, to have a lose fit later). If the sketching plane is obscured by already present 3D geometry, select 'Sketch | View section' from the main menu. Close the sketch after constraining the circle diameter.

In the tree view, mark the new sketch and click the 'Pad' tool button (Figure D-4).

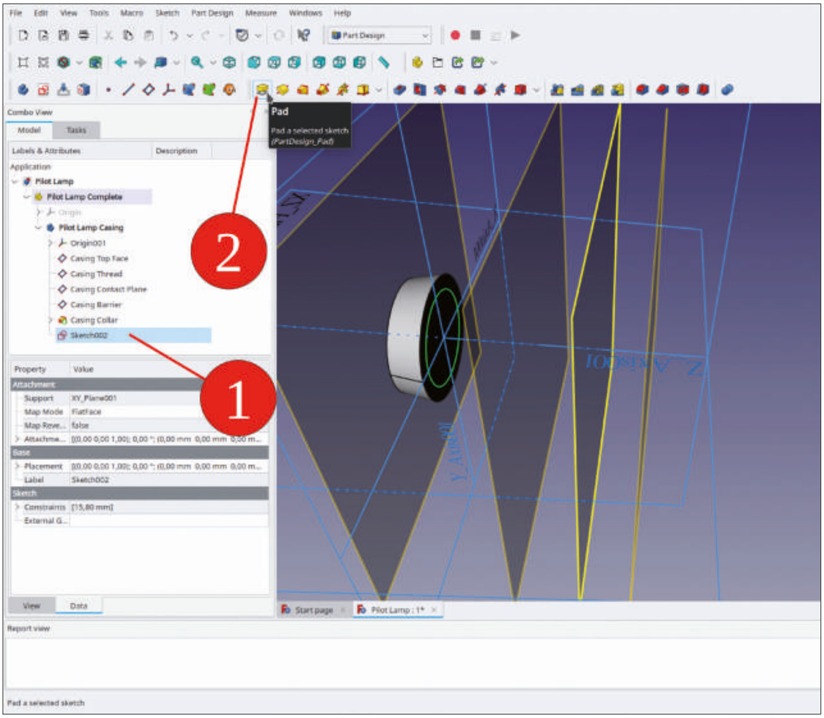


Figure D-4

In the task window, select the type 'Up to face', which activates the face collector below (the button 'Select face' appears dark gray). If the collector is not activated, click again on the button 'Select Face', until it is dark gray. Then click the 'Casing Thread' datum plane (Figure D-5). You can do that either in the 3D view, or by switching to the 'Model' tab and clicking on the datum plane tree view item itself (and then returning to the 'Tasks' tab).

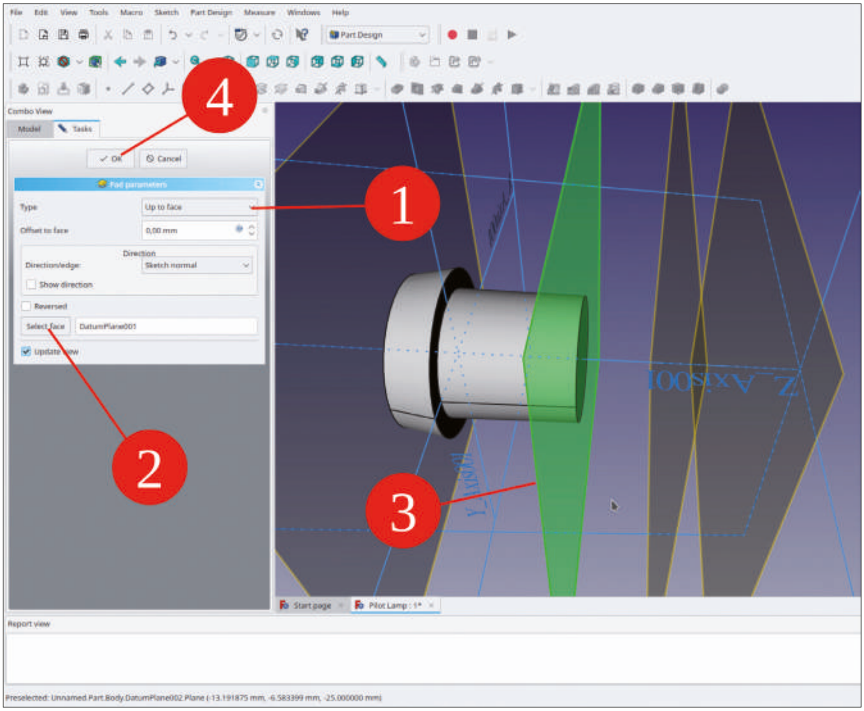


Figure D-5

The thread of the casing is followed by a cylindrical part of the casing, which has a diameter of 14 mm. As described in the preceding steps, draw a circle with diameter 14 mm on the 'Casing Thread' datum plane, and extrude it with the 'Pad' command to the 'Casing Contact Plane' datum plane.

The isolating barrier between the contacts is formed as a section of a conical protrusion. As for the front collar of the casing, for the definition of the conical 'Loft' object, two cross-sections are needed. To draw the first circle, in the tree view, mark the 'Casing Contact Plane' and start the sketcher. Draw a circle that is centered around the origin and constrain its diameter to 14 mm (which equals the diameter of the cylindrical part).

In a similar fashion, draw the circle for the rear face of the casing, and set the diameter to 12.2 mm.

In the tree view, mark the two new sketches and click the 'Loft' tool button. In the task window, check the 'Closed' checkbox and close the task window with the OK button.

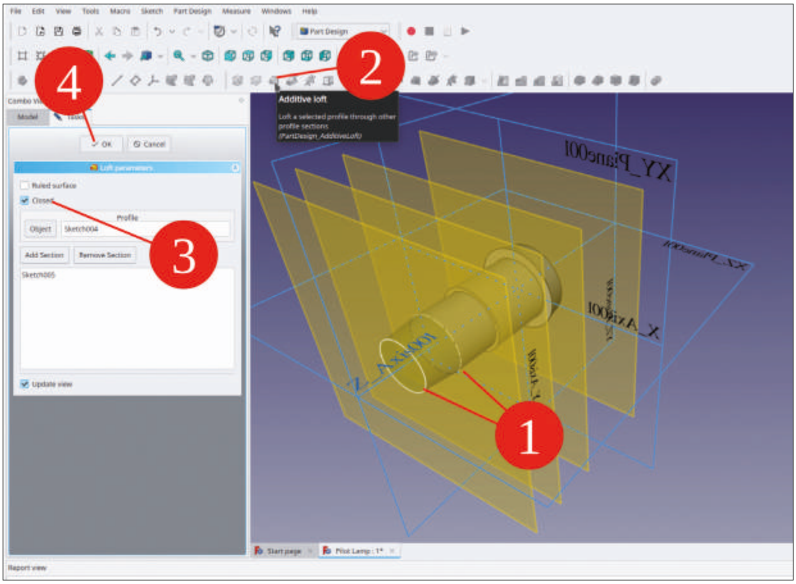


Figure D-6

The space for the contacts is defined on the 'Casing Contact Plane' datum plane. Mark it in the tree view and start the sketcher. When the sketch is started like that, the present 3D geometry is not displayed. To display it, close and reopen the sketch. Select the 'Centered rectangle' drawing tool and draw a rectangle that is centered on the X axis (Figure D-7).

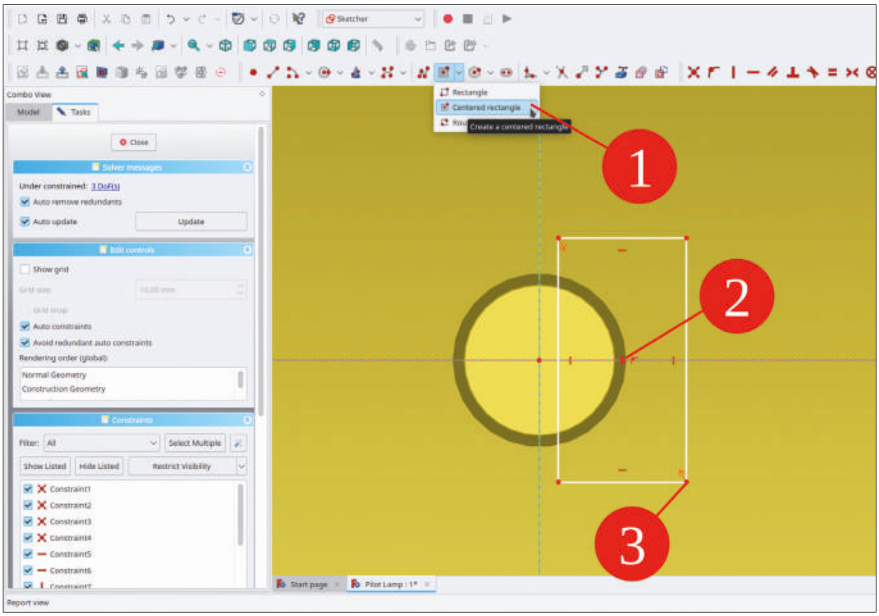


Figure D-7

Now, the dimensions are needed for the rectangle. Mark the Y axis and one corner of the rectangle, which is next to it, and constrain the horizontal distance to 1.25 mm (Figure D-8, steps 1,2).

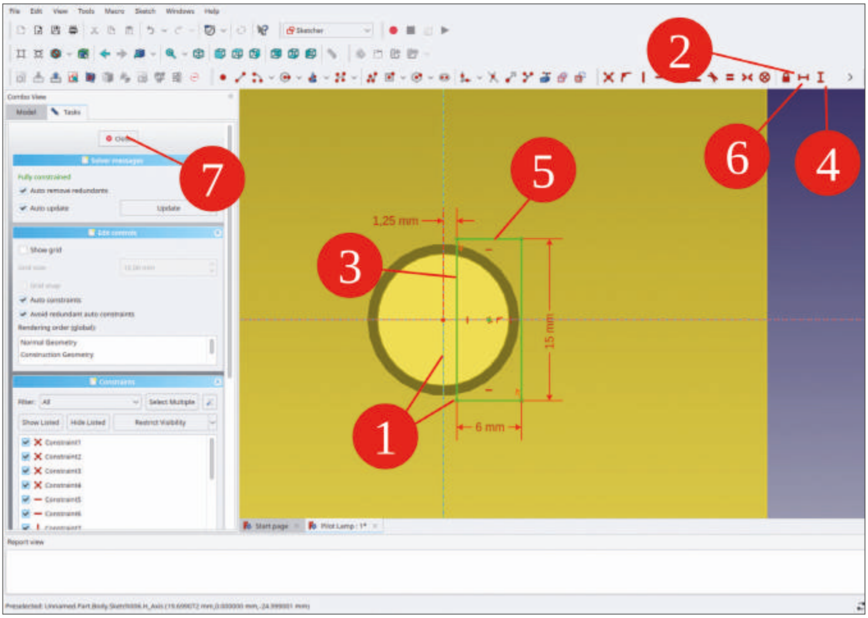


Figure D-8

Then, click a vertical line of the rectangle and constrain the height to a value large enough to cut the whole casing (e.g., 15 mm, Figure D-8, steps 3,4).

Next, select a horizontal line of the rectangle and constrain the horizontal distance to 6 mm, which is wide enough to cut the whole casing (Figure D-8, steps 5, 6).

In the tree view, mark the new sketch and click the 'Pocket' tool button (Figure D-9). In the task window, select the type 'Up to face' and click the 'Casing Barrier' datum plane, either in the 3D view or in the tree view (Figure D-10). Close the task window with the OK button.

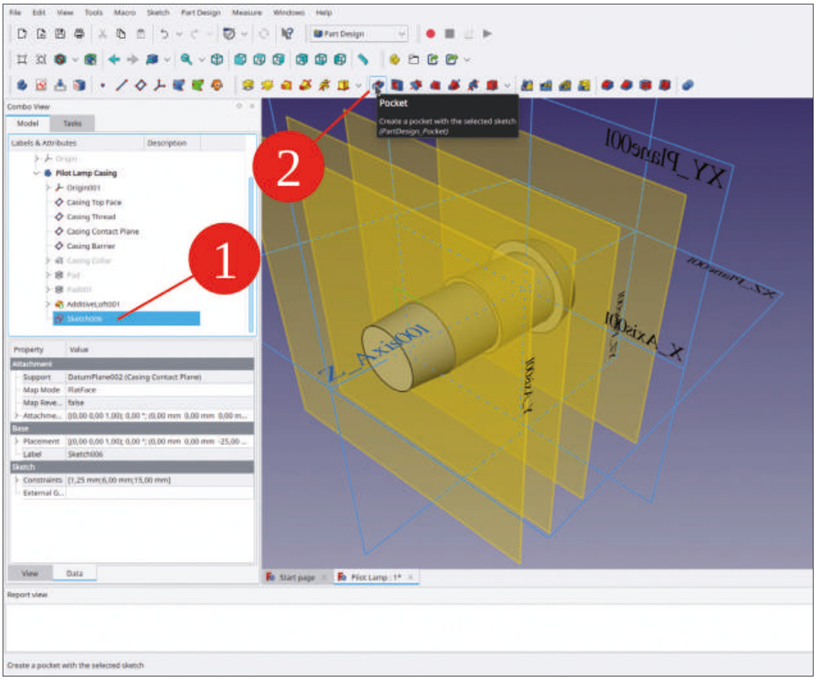


Figure D-9

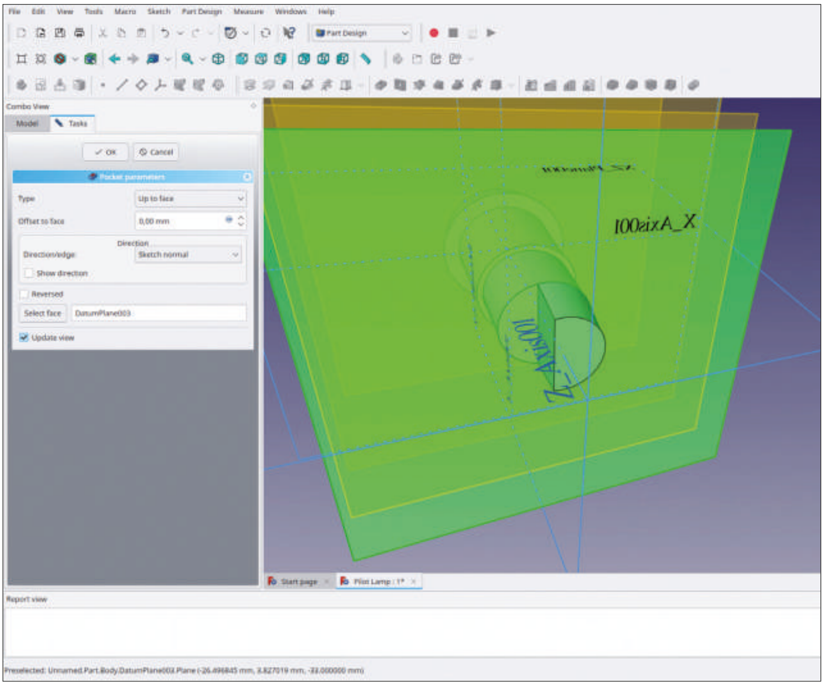


Figure D-10

In the tree view, mark the new design state 'Pocket'. From the main menu, select 'Part Design | Apply a pattern | Mirrored'.

By this selection, in the task window, the object to be mirrored is already listed. For the mirror plane, select the 'Base YZ' plane, and close the task window with the OK button (Figure D-11).

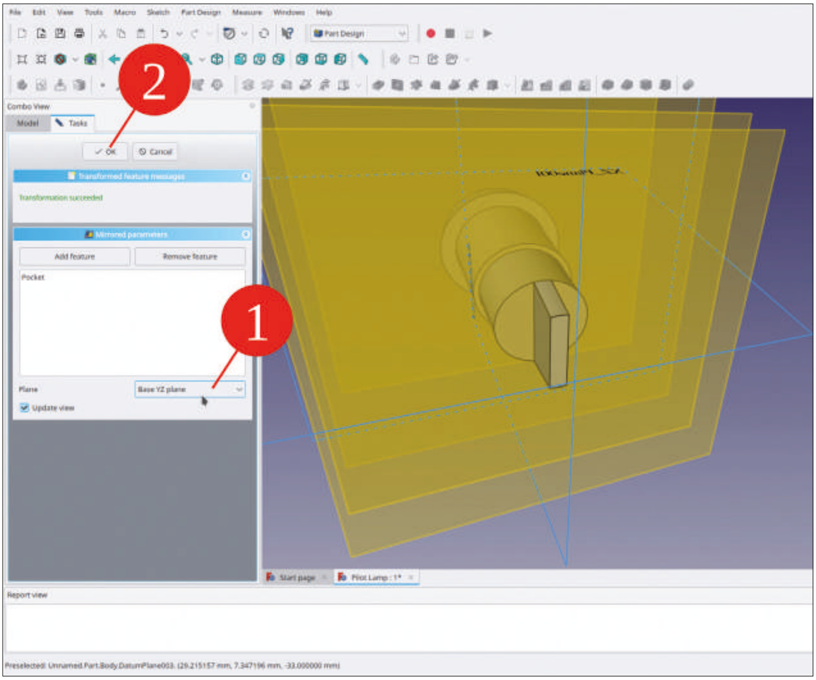


Figure D-11

A small, but important detail is yet missing: The rotation lock at the rear side of the front collar, which must be considered for the footprint of the pilot light. The rotation lock is modeled by a sketch on the XY plane, as a rectangle that is centered on the Y axis. The sketch is depicted in Figure D-12.

Mark a horizontal line of the rectangle and constrain the width to 3 mm (Figure D-12, step 2).

Mark the bottom corner of the rectangle and the origin. Constrain the vertical distance to 6 mm (Figure D-12, step 3).

Mark the top corner of the rectangle and constrain the vertical distance to 9 mm. The sketch is then displayed as fully constrained (Figure D-12). Close the sketcher and hide all datum planes.

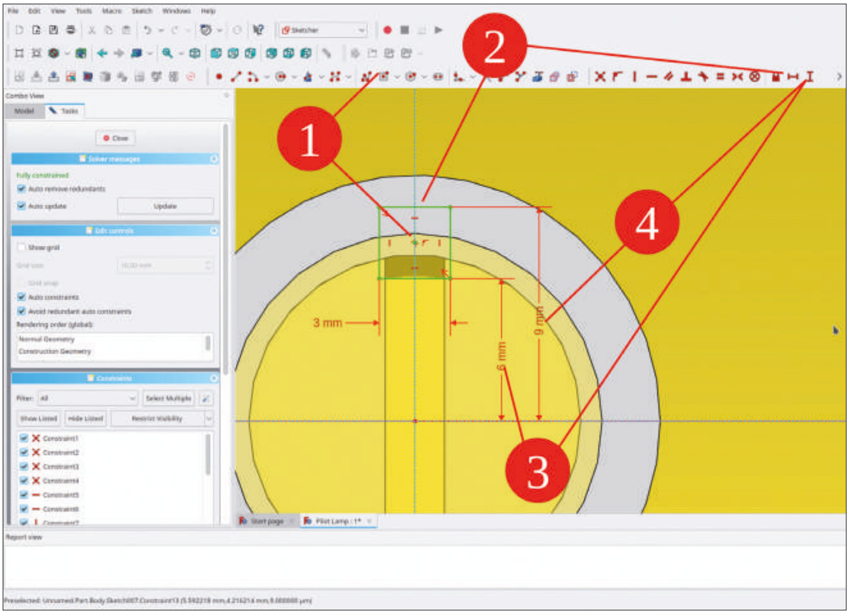


Figure D-12

In the tree view, mark the new sketch and click the 'Pad' tool button. In the task window, set the length to 1 mm and check the 'Reversed' checkbox (Figure D-13). Close the task window with the OK button.

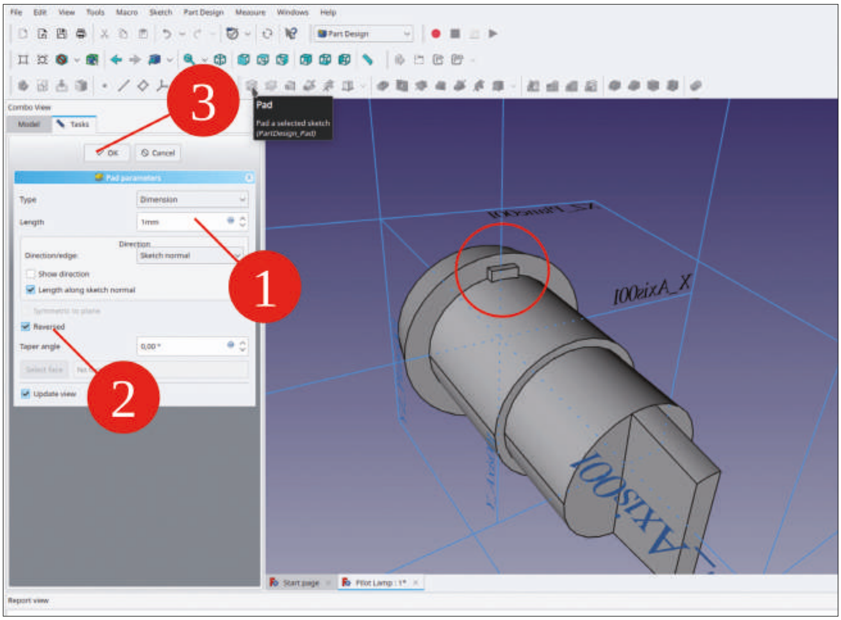


Figure D-13

In the tree view, hide all datum planes and the coordinate system. Right-click the body of the casing and select 'Appearance…' from the context menu. For the material, select 'Shiny Plastic'. This auto-selects black as the color. Sometimes, a dark gray color can give more guidance to the eye.

For the cap of the pilot light, create a new body object. If necessary, drag-and-drop it into the Std-Part-Container 'Pilot Light Complete'. Rename the new body to 'Pilot Light Cap' (Figure D-14).

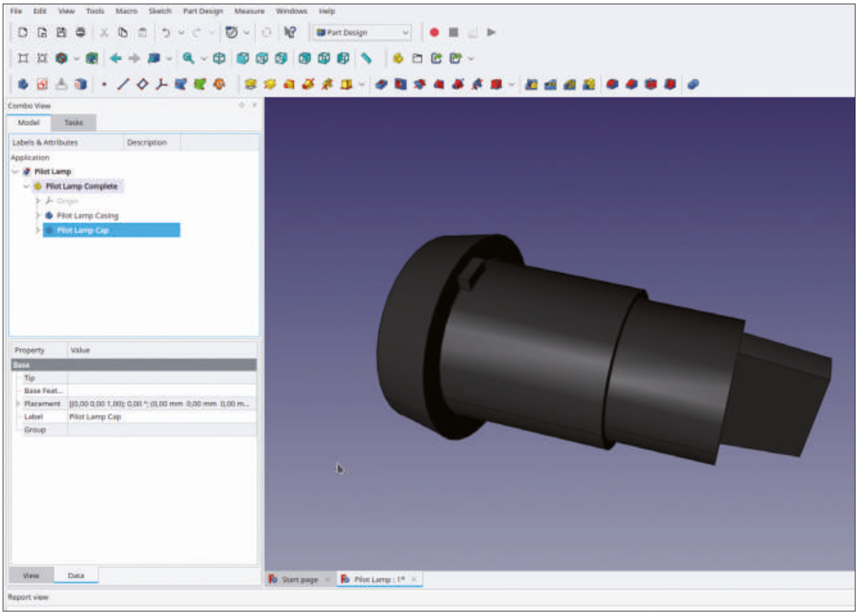


Figure D-14

First, adjust the position of the placement for the new cap body: In the tree view, mark the body of the cap. In the property list, expand the 'Placement' parameters, and enter a value of Z = 5.01 mm for the cap. The additional offset of 0.01 mm with respect to the casing top face helps to render the cap more clearly. If the bottom face of the cap and the top face of the casing would coincide (at Z=5 mm), FreeCAD would try to display them both at the same time. This results in a discontinuous display of the boundary between the two objects.

While working on the cap, hide the 'Pilot Light Casing' body and display the coordinate system of the cap body (mark in 3D view and SPACE key).

First, define two datum planes that are parallel to the XY plane. One is located at the position of the top face of the cap, and the other is on the top face of the hollow inside of the cap. To generate the first datum plane, in the 3D view, mark the XY plane and click the 'Create a datum plane' tool button. In the task window, set the attachment Z offset to 11.4 mm. Close the task window with the OK button. In the tree view, rename the plane to 'Cap Top Face'.

The cap is hollow. It will be displayed with a transparent body later. For a realistic representation, the inside of the cap has to be modeled also. For the inner surface, again create a datum plane parallel to the XY plane. Set the attachment Z offset to 9.8 mm and rename the plane to 'Cap Inner Top Face'.

Now, the cone for the cap outline needs to be generated. This is again a 'Loft' object, for which two cross-sections have to be defined. For the outline, sketch one circle onto the XY plane, and constrain its diameter to 16 mm. On the 'Cap Top Face' datum plane, sketch a circle with a diameter of 13.5 mm. Both circles are centered around the origin.

In the tree view, mark both new sketches and click the 'AdditiveLoft' tool button (Figure D-15). Check the 'Closed' checkbox and close the task window with the 'OK' button.

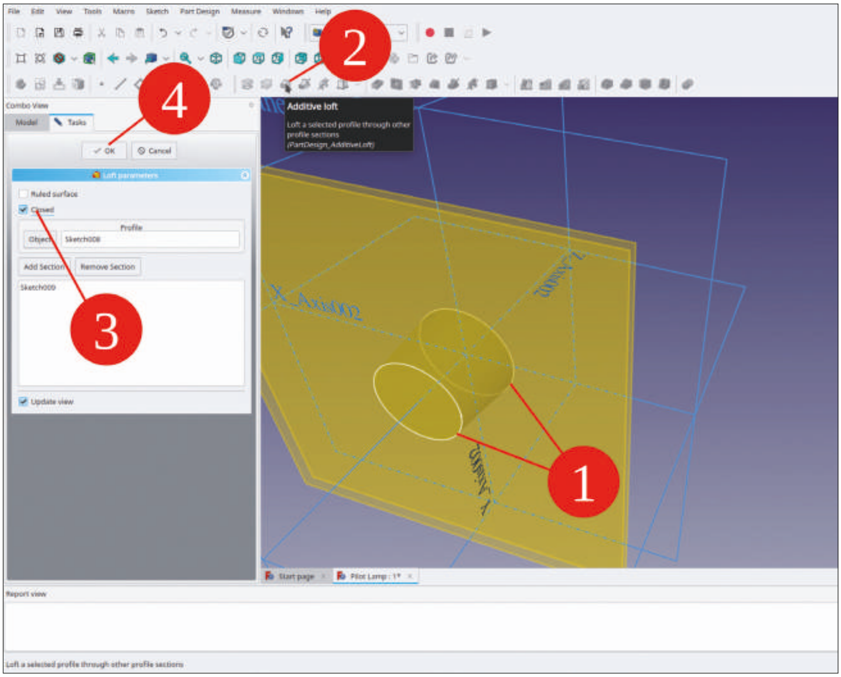


Figure D-15

Now, the hollow space inside the cap is created. To define it, sketch another two circles: One located on the XY plane, with a diameter of 13.5 mm. The other one is located on the 'Cap Inner Top Surface' datum plane, with a diameter of 10.5 mm.

In the tree view, mark the two new sketches and click the 'Subtractive loft' tool button (subtractive tool buttons show up in red, additive ones, in yellow). Close the task window with the OK button (Figure D-16).

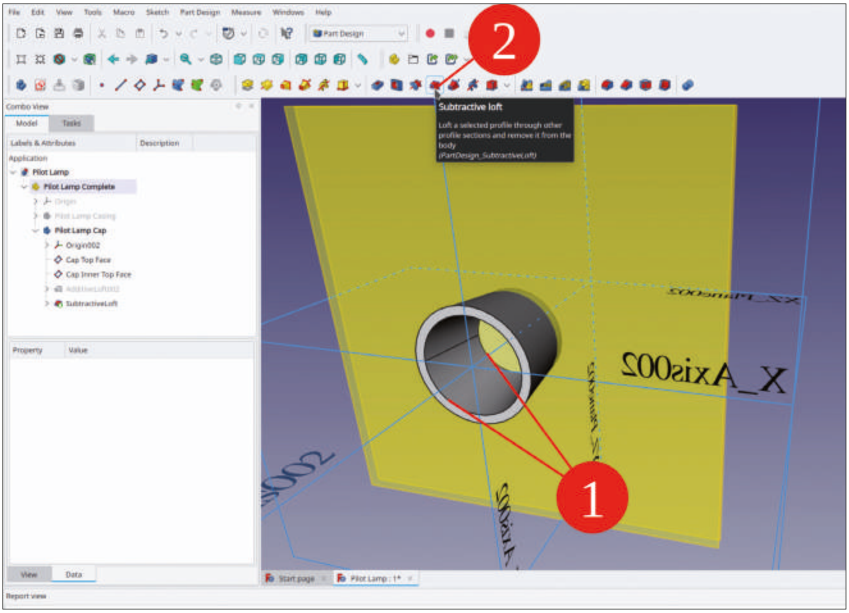


Figure D-16

In the tree view, hide the datum planes and the coordinate system of the cap. Show the body 'Pilot Light Casing'.

In the tree view, right-click the body of the cap and select 'Appearance...' from the context menu. For the material, select 'Shiny Plastic'. Set the color to red, and the transparency to 40% (Figure D-17).

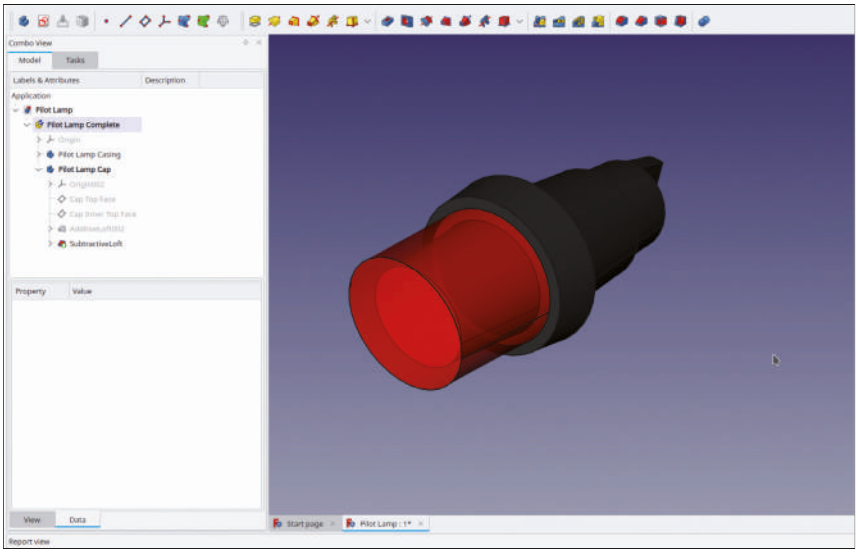


Figure D-17

The pilot light looks quite realistic now. It could already be used in assemblies, and to define the footprint in a front panel. In a book like this, however, you add two more cosmetic details which also give some guidance for the use of other tools. First, there is a groove pattern inside the pilot light cap, which is oriented along the conical surface. To model such a groove, you need a guide line. Luckily, the inner surface already has one on display – where the defined profile has its boundary line. For the following, you keep track of this useful line. In the tree view, hide the 'Pilot Light Casing' body again, and rotate the cap until you can spot the line (Figure D-18).

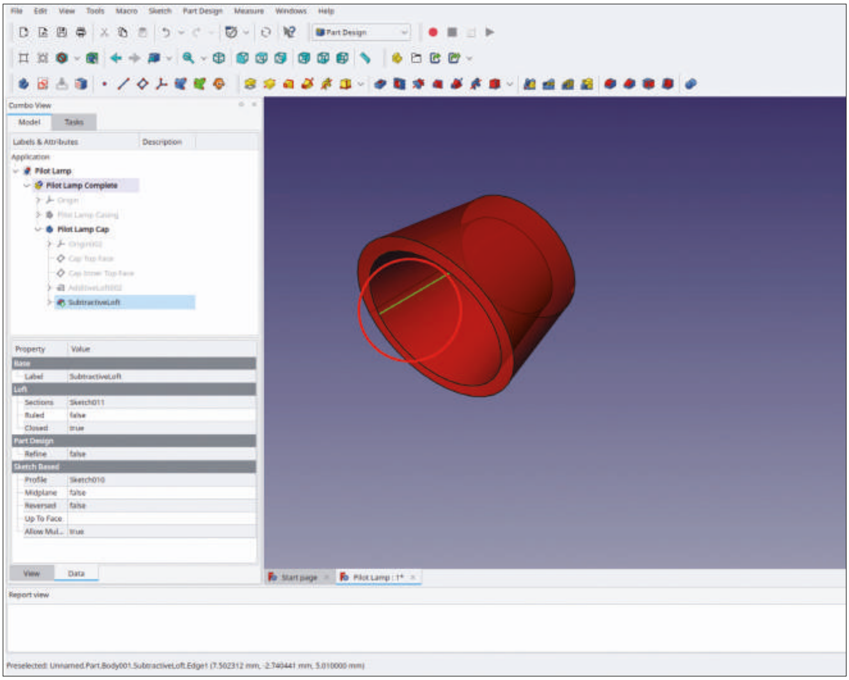


Figure D-18

Now, the profile of the grove is sketched on the XY plane. To do this, start the sketcher and select the XY plane from the selection dialog. Close the sketch right away, and reopen it, in order to display the cap with the sketch. If the cap appears white or gray now, this is due to the illumination of the scene. First, turn the cap into the 'Bottom' orientation (use a tool button or the control cube). Then slightly move the cap, until you can see the guide line again.

Click the 'External geometry' tool button and mark the guide line. In the sketching plane, the guide line is represented by two projected points – the little green dot with the cross marks as the starting point, the dot as the end point of the line (Figure D-19, step 1 and 2).

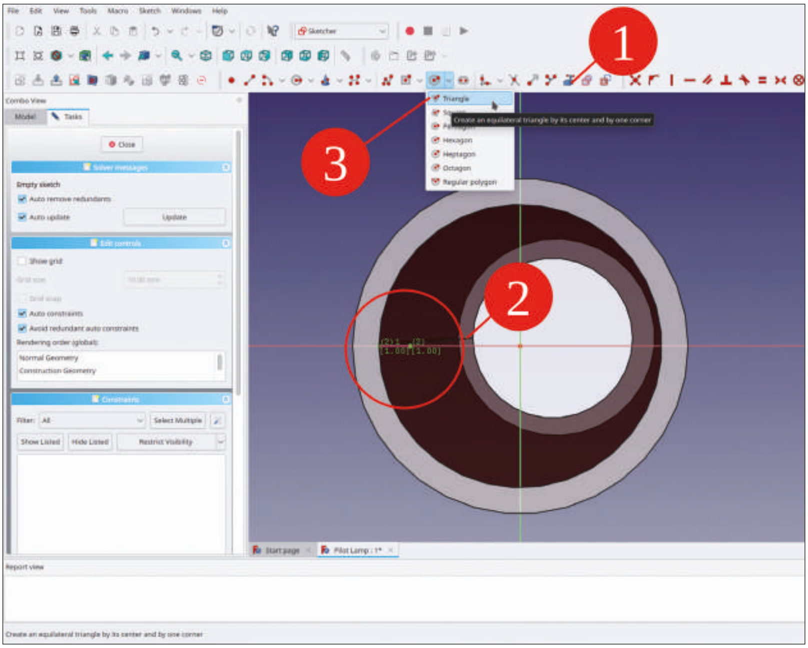


Figure D-19

Click the 'Create an equilateral triangle by its center and by one corner' tool button. (Figure D-19, step 3).

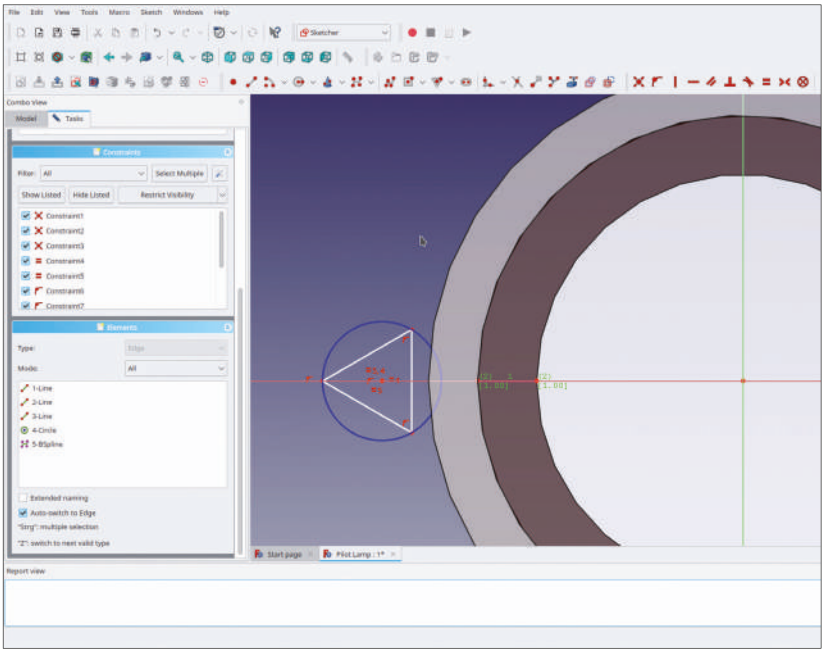


Figure D-20

With the first click, place the center point of the triangle on the X axis. With the second click, to the left of the center point, place one corner of the triangle onto the X axis. It is eventually hard to lock the corner onto the X axis – aim with the crosshairs, until the axis turns yellow (Figure D-20). If you find it difficult to see, set the zoom to high magnification, select 'View | Orthographic view' from the main menu.

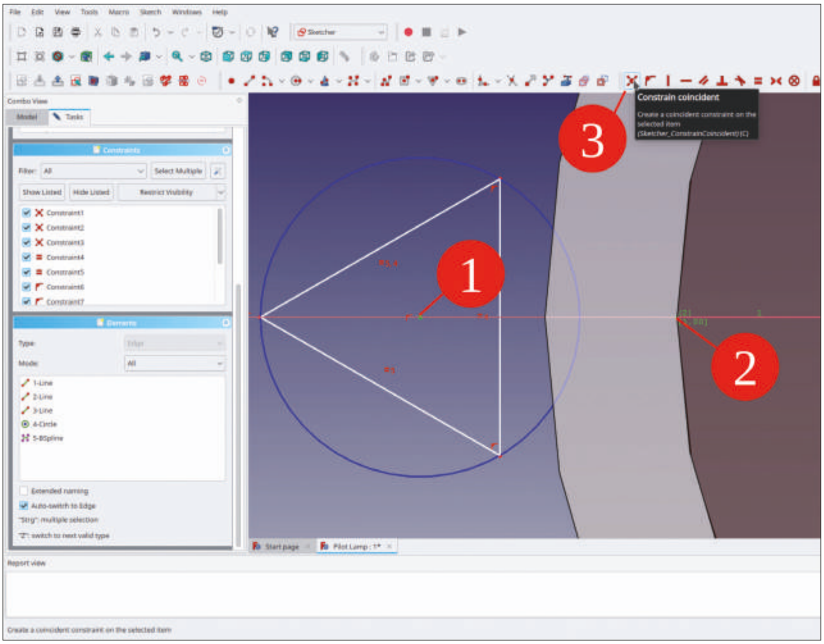


Figure D-21

Mark the center point of the triangle, and then the projected starting point of the guide line. If the center point is difficult to select, mark it with a small rectangular selection with the CTRL key down. Then, click the 'Constrain coincident' tool button (Figure D-21).

Mark one line of the triangle. Click on the 'Constrain distance' tool button (hidden, click down arrow in menu bar, in the constraints tool section). Enter 1.2 mm for the distance (Figure D-22). Close the sketch.

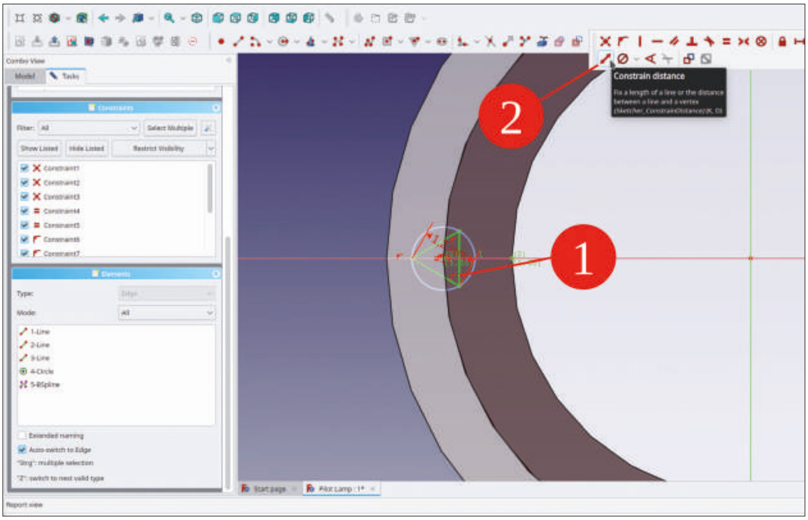


Figure D-22

In order to create the first groove, in the 3D view, mark the guide line and the sketch of the triangle (CTRL key down, Figure D-23). Then, click the 'Subtractive pipe' tool button. Close the task window with the OK button (Figure D-24).

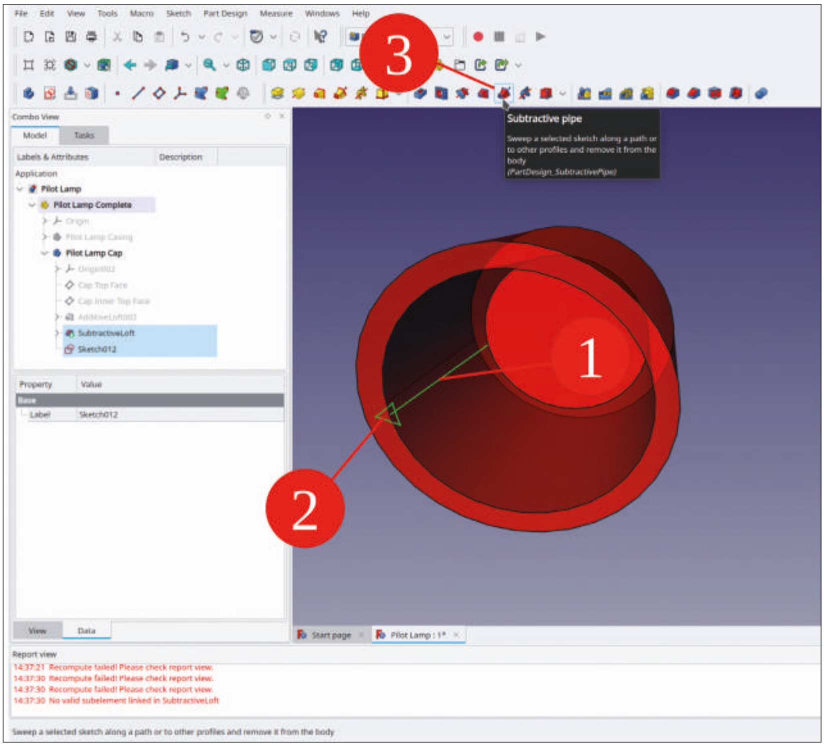


Figure D-23

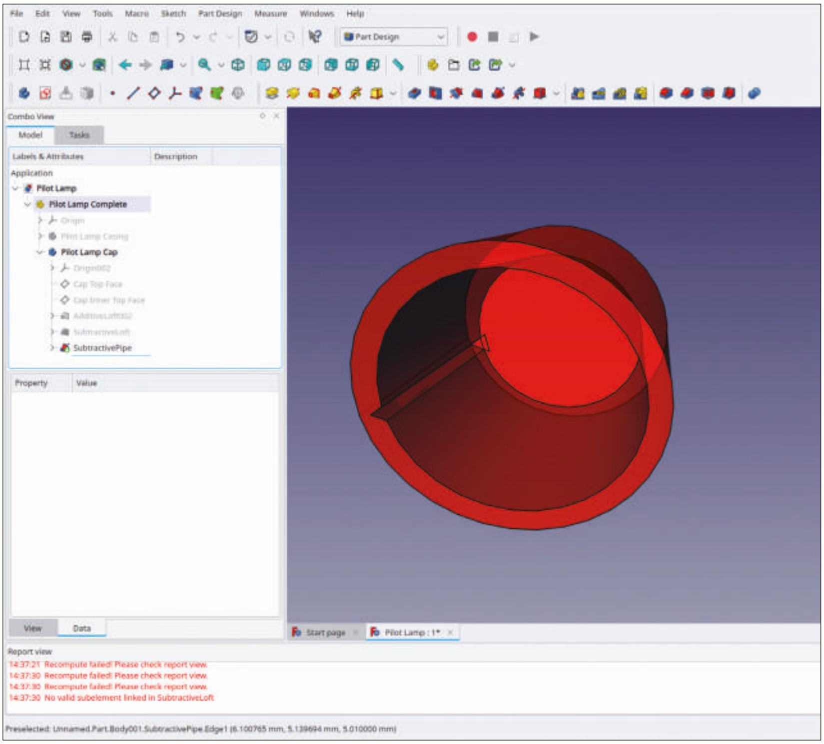


Figure D-24

For the generation of all the grooves, a polar pattern is needed. From the main menu, select 'Part Design | Apply a pattern | PolarPattern'.

A selection dialog opens. Scroll down the list of possible elements for the pattern. The last entry is 'Subtractive Pipe'. Mark that entry and click the OK button (Figure D-25).

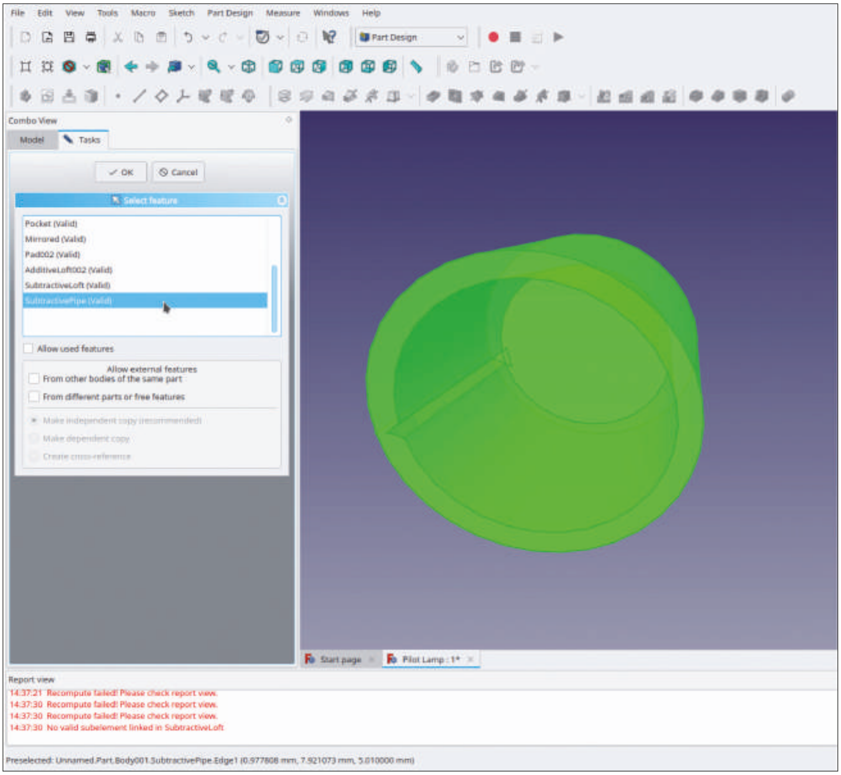


Figure D-25

A task window opens. Enter a number of 30 for 'Occurrences' and close the task window with the OK button (Figure D-26).

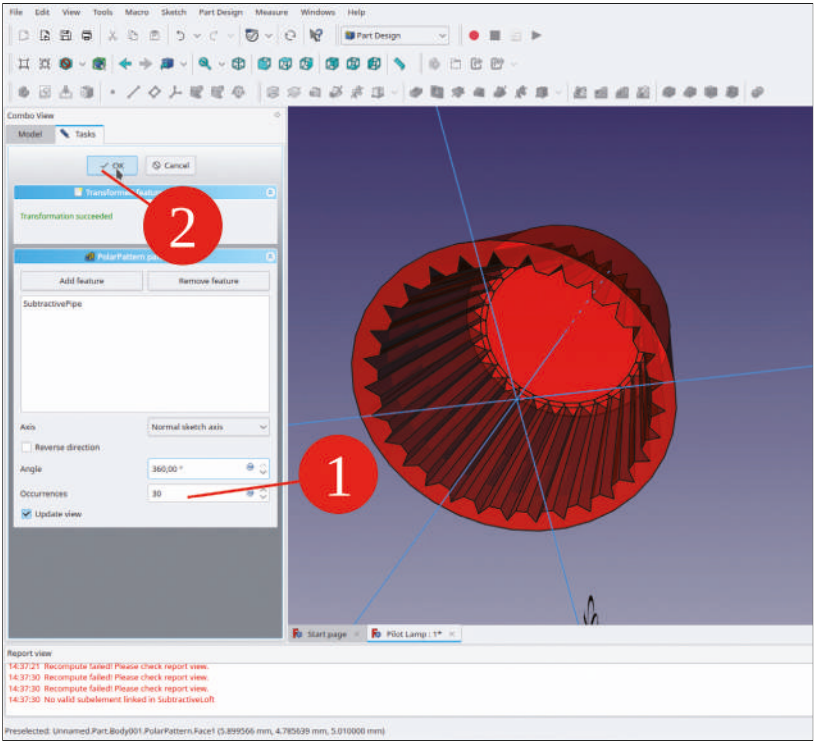


Figure D-26

In the top surface of the cap, there are concentric, circular grooves. The profile of these groves needs to be sketched on the XZ plane. Click on the 'Sketcher' tool button and select the XZ plane in the initial selection dialog. Close and reopen the sketch to display the cap with the sketch. Select 'View | Orthographic view' and 'Sketch | View section' for a better view.

Switch to the 'Model' tab and show the 'Cap Inner Top Surface' datum plane (mark in the tree view and SPACE key). Switch back to the 'Tasks' tab.

Click on the 'External geometry' tool button and mark the 'Cap Inner Top Surface' datum plane. You could also switch to the 'Model' tab and pick the plane in the tree view). The datum plane is shown as a violet construction line (Figure D-27, steps 1 and 2).

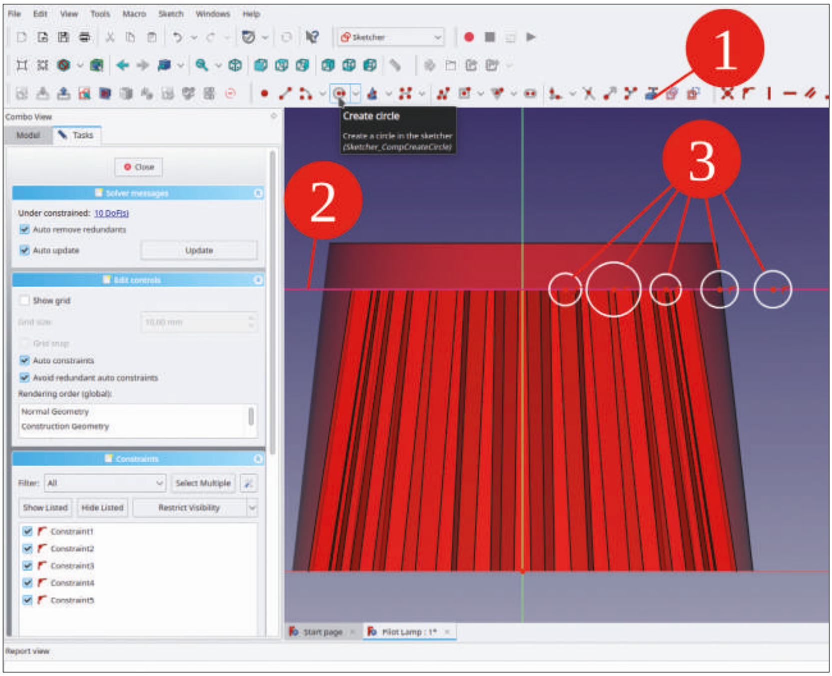


Figure D-27

Sketch 5 circles centered on the datum plane representation (Figure D-27, step 3).

On the sketch (alternatively in the 'Elements' panel), mark all the circles and click the 'Constrain equal' tool button (Figure D-28).

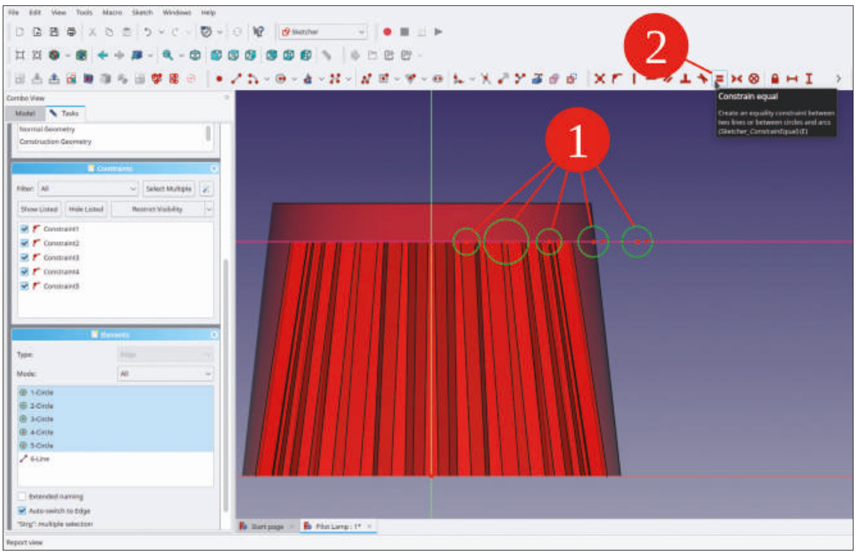


Figure D-28

In the 'Elements' panel, right-click the first circle and select 'Diameter constraint' from the context menu. Set the diameter to 0.5 mm — notice that all other circles follow because of the previous constraint.

Mark the origin and the center of the inner circle. Click the 'Constrain horizontal distance' tool button and set the distance to 0,5 mm. In the same way, set the other distances to 1.5 mm, 2.5 mm, 3.5 mm and 4.5 mm (Figure D-29) Close the sketch.

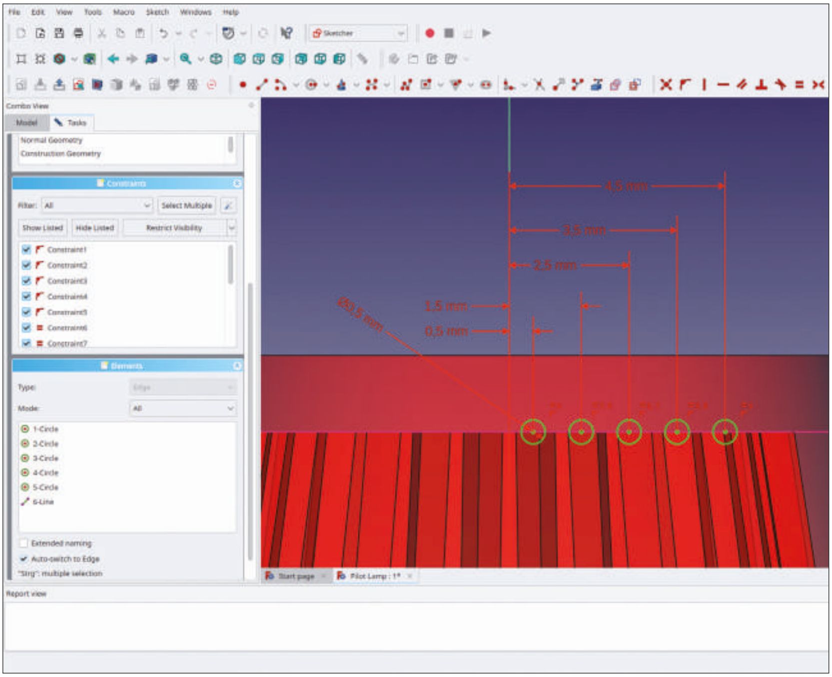


Figure D29

Hide the 'Cap Inner Top Surface' datum plane. In the tree view, mark the new sketch and click the 'Groove' tool button. In the task window, angle and axis are correctly preset. Close the task window with the OK button (Figure D-30).

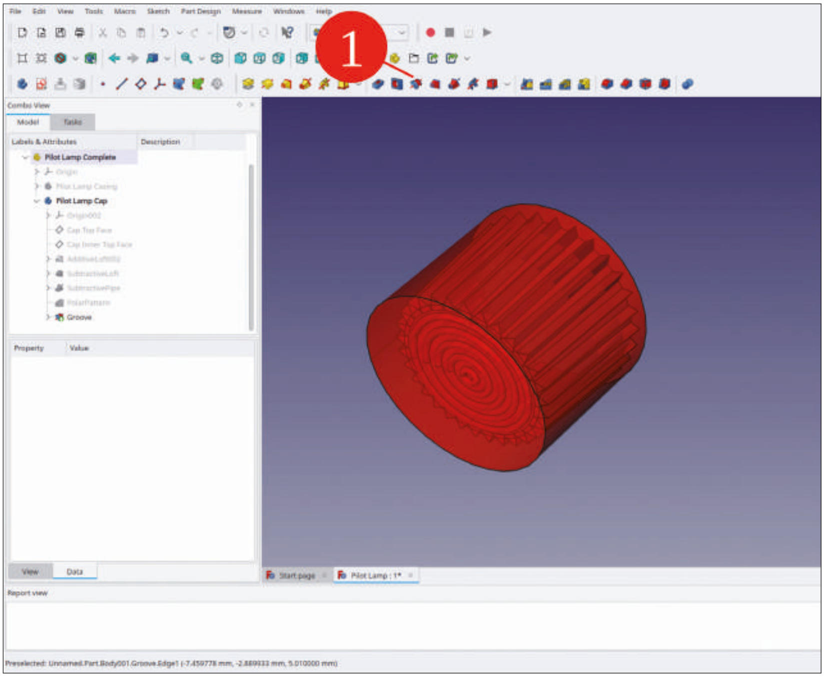


Figure D-30

At this point, a realistic appearance of the pilot light has been achieved. It could, however, be useful to add the contacts as well as the fastener. With these additional items, the accessibility of the fastener and the space for the wiring harness could be examined later, in the assembly.

Click the blue 'Create body' tool button. Rename the new body object to 'Pilot Light Contact 1'. If the body is not located within the Std-Part-Container 'Pilot Light Complete', simply drag and drop it there.

Start the sketcher. Because no plane was marked before this call, the plane needs to be selected now. Check the checkbox 'Allow external features | From other bodies of the same part' and pick 'Casing Contact Plane' from the list (Figure D-31). Close the sketch and reopen it, in order to display already present geometry.

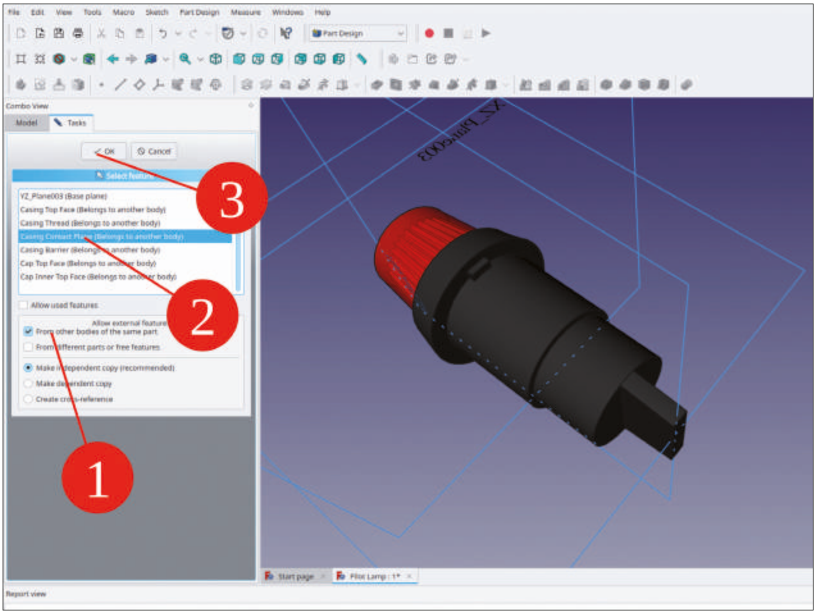


Figure D-31

From the main menu, select 'View | Orthographic view' and 'Sketch | View section'.

Click on the 'Create a centered rectangle' tool button and draw a rectangle that is centered on the X axis (Figure D-32).

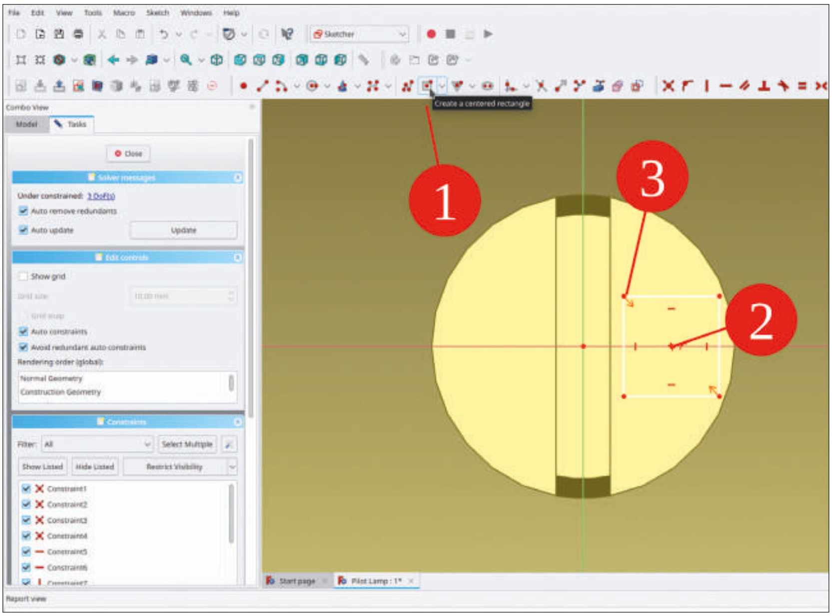


Figure D-32

Mark a horizontal line of the rectangle and click the 'Constrain horizontal distance' tool button. Set the distance to 4 mm. Mark one horizontal and one vertical line of the rectangle and click the 'Constrain equal' tool button. Then, mark the origin and the center point of the rectangle and constrain the horizontal distance to 3.5 mm (Figure D-33). Close the sketch.

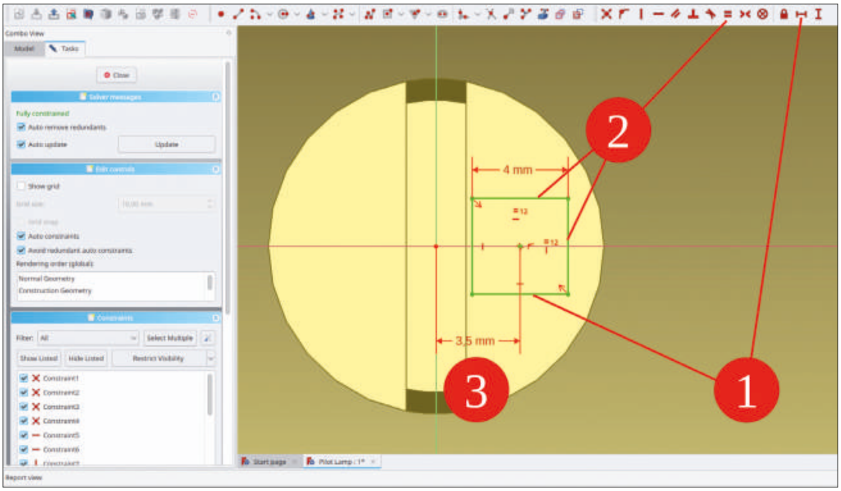


Figure D-33

In the tree view, mark the new sketch. Click on the 'Pad' tool button. In the task window, enter 5 mm for the length, and check the 'Reversed' checkbox (Figure D-34). Close the task window with the OK button.

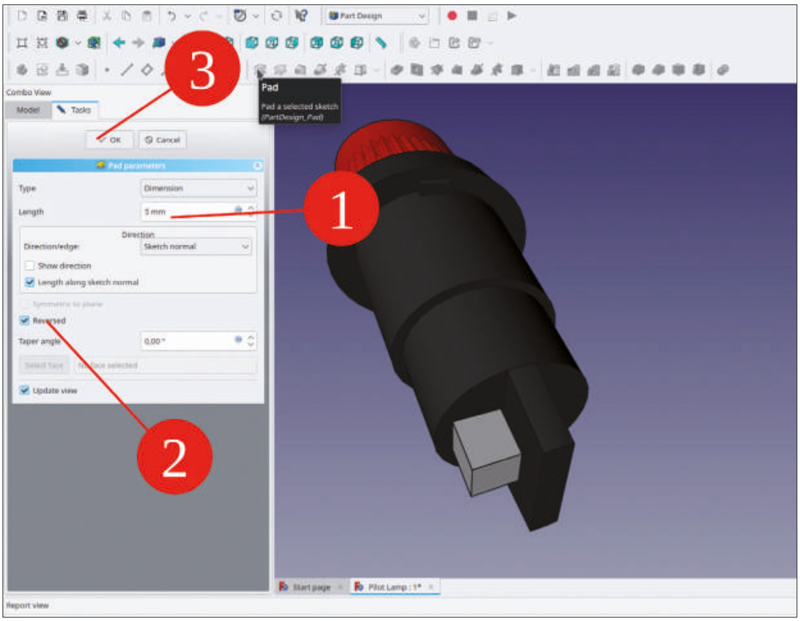


Figure D-34

In the tree view, mark the local copy of the datum plane in the body of the contact. Start the sketcher. Rotate the view by either clicking on the 'Top' tool button, or by clicking on the control cube.

From the main menu, select 'View | Orthographic view' and 'Sketch | View section'.

Click on the 'External geometry' tool button and select two adjacent sides of the contact rectangle. Then, sketch a circle of arbitrary size and position (Figure D-35).

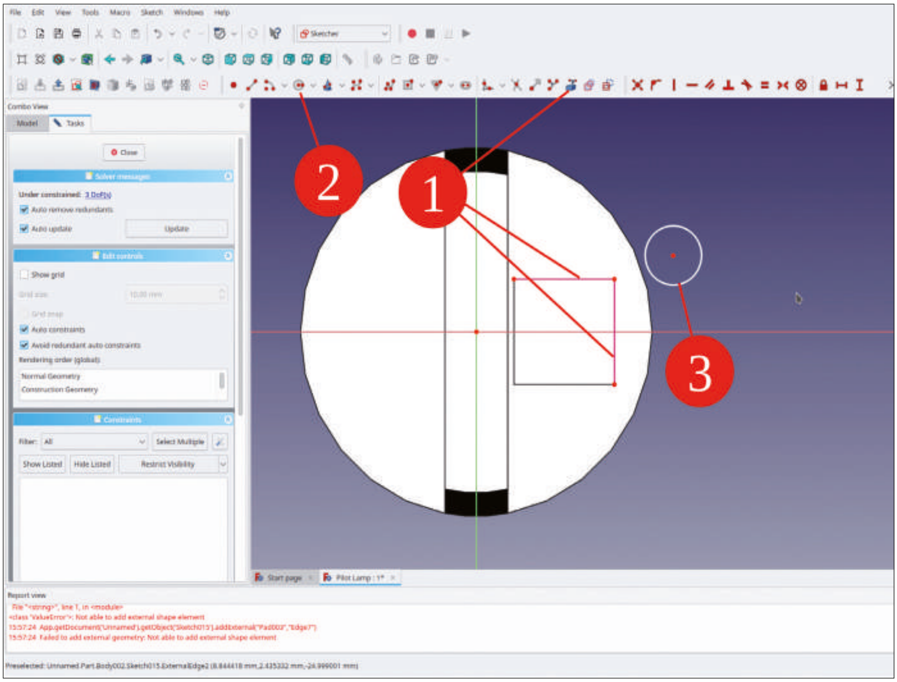


Figure D-35

Mark two diagonal corners of the violet construction lines, and the center of the circle. Then, click the 'Constrain symmetrical' tool button (Figure D-36). Close the sketch.

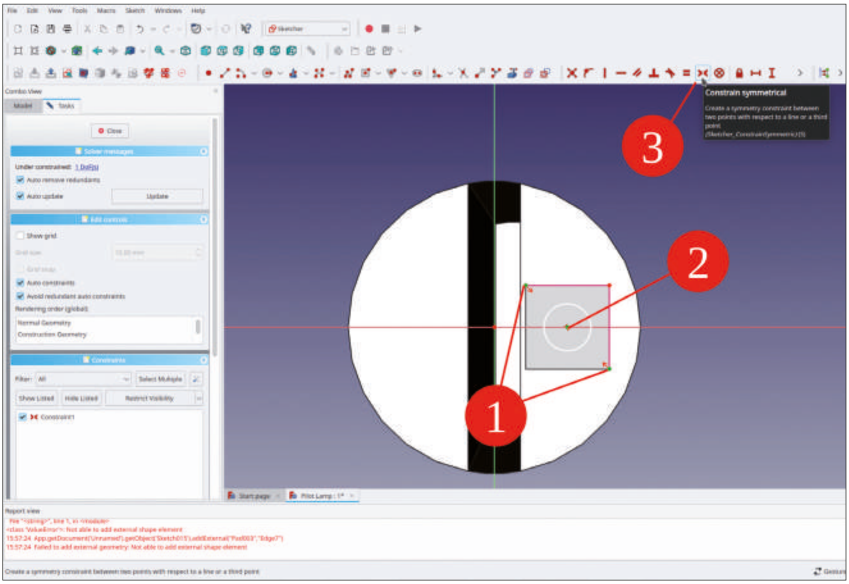


Figure D-36

In the tree view, mark the new sketch. Then, click the 'Hole' tool button.

In the task window, select for the profile 'ISO metric regular profile', check the 'Threaded' checkbox and set the size to M2.5 (i.e., metric 2.5 mm). For the depth of the hole, select 'Through all' (Figure D-37). Close the task window with the OK button.

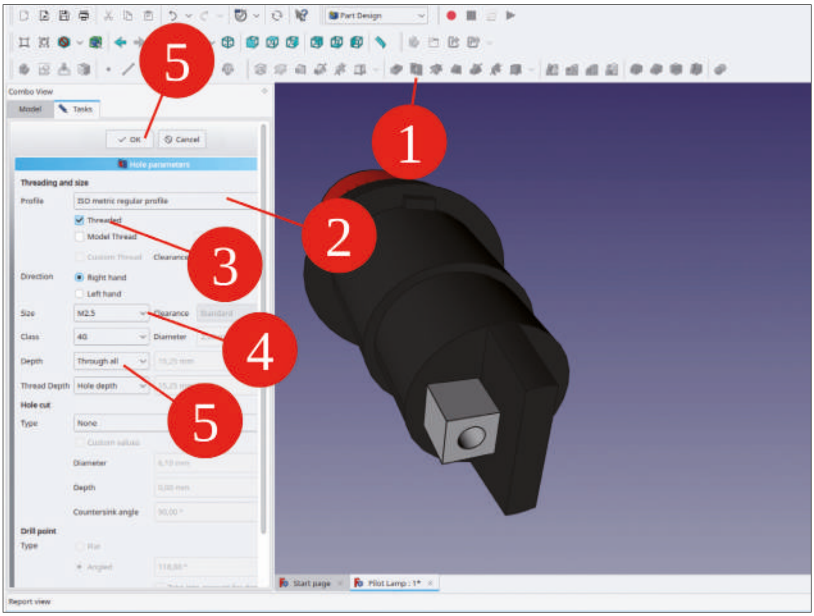


Figure D-37

To accommodate the wiring, another hole is needed, crossing the threaded hole. The procedure is similar to the one just executed for the threaded hole. Start the sketcher and select the XZ plane. Close and reopen the sketch, to display the already present geometry. From the main menu, select 'View | Orthographic view' and 'Sketch | View section'.

Click on the 'External geometry' tool button and mark two adjacent sides of the contact. Then, draw a circle somewhere, and constrain its diameter to 2.5 mm (Figure D-38).

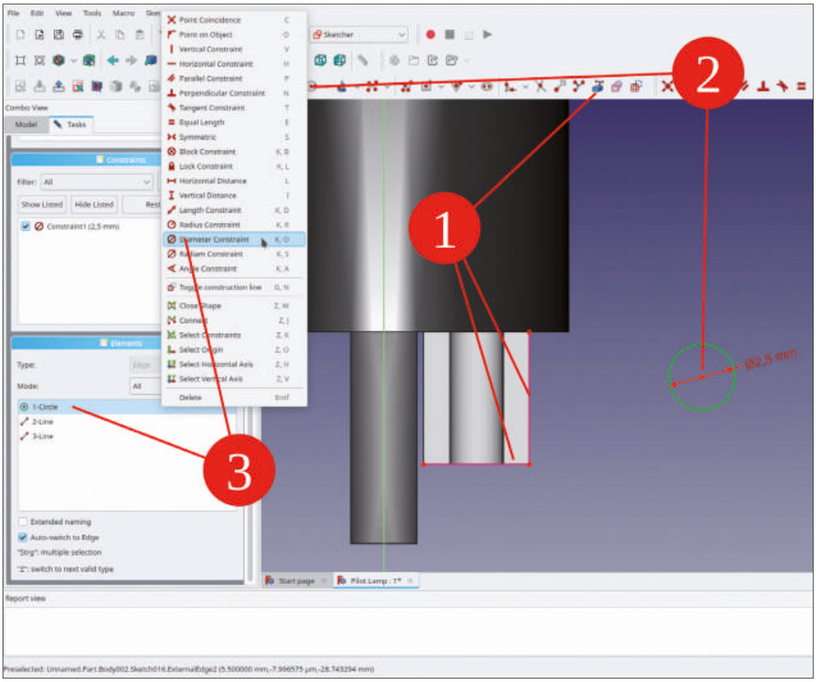


Figure D-38

Mark two corners of the construction lines, which are diagonal with respect to the contact contour. Mark the center of the circle and click the 'Constrain symmetrical' tool button (Figure D-39).

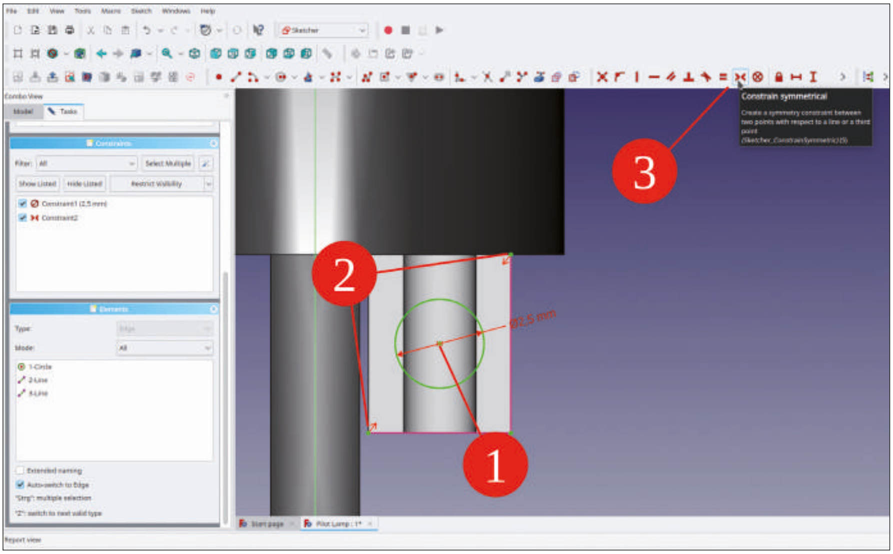


Figure D-39

Close the sketch.

In the tree view, mark the new sketch and click the 'Pocket' tool button.

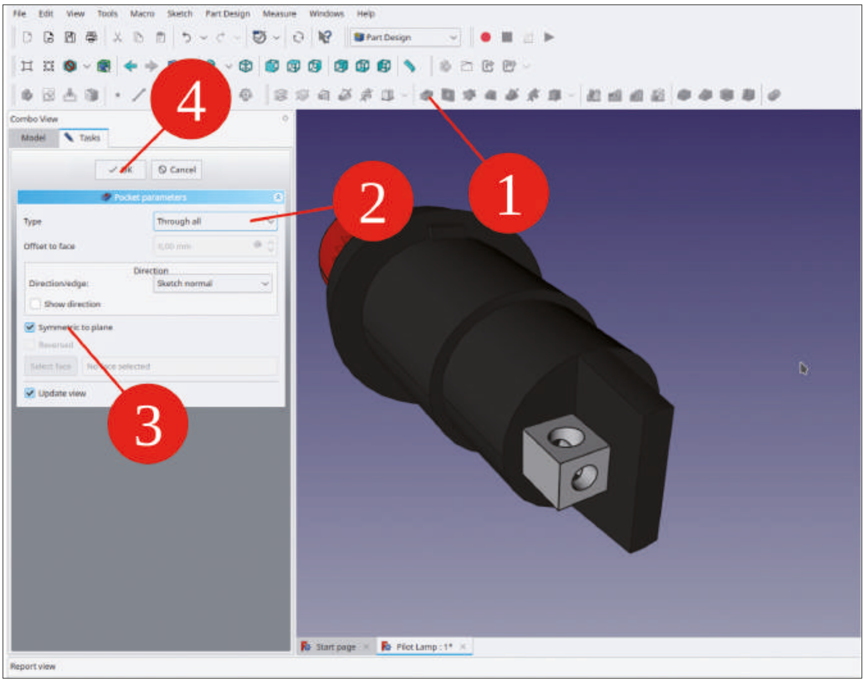


Figure D-40

In the task window, select the type 'Through all' and check the 'Symmetric to plane' check-box (Figure D-40). Close the task window with the OK button.

In the tree view, right-click the body 'Pilot Light Contact 1' and select 'Appearance…' from the context menu. In the task window, select 'Chrome' as the material and close it.

We can generate the second contact with a reference: In the tree view, mark the body 'Pilot Light Contact 1'. Then, click the 'Make link' tool button (Figure D-41). In the tree view, a new body object appears. Drag-and-drop it into the Std-Part-Container 'Pilot Light Complete'. Rename it to 'Pilot Light Contact 2', for consistence.

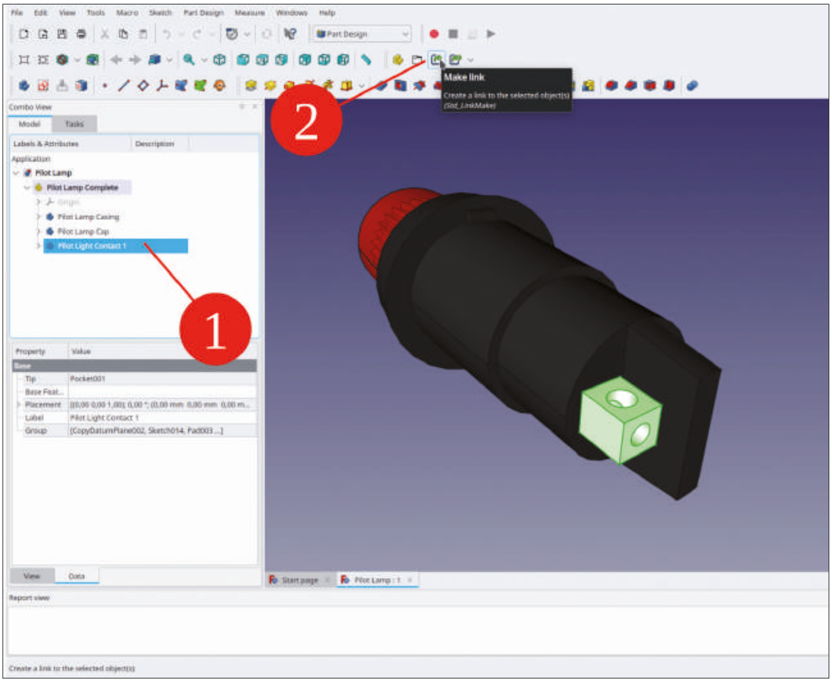


Figure D-41

Mark the new contact body and edit the placement coordinates. These coordinates are relative to that of the linked parent object. Open the placement task window by clicking the property line 'Placement' and then on the … button to the right. Set the X offset to −7 mm and close the task window with the OK button.

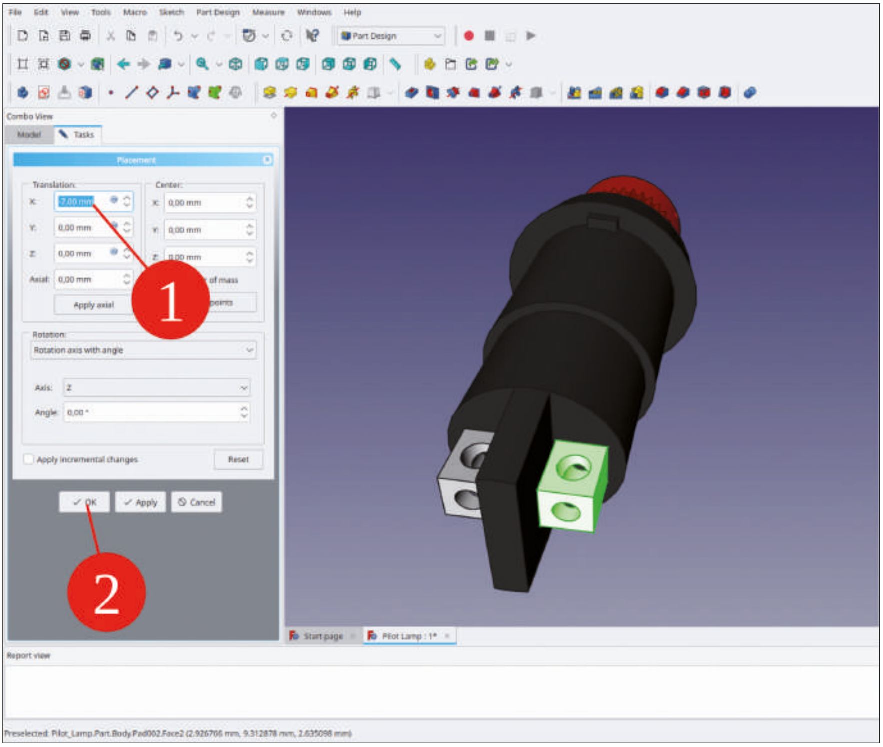


Figure D-42

For the terminal screws, from the main menu, select 'Macro | Recent macros | start_bolts'. From the 'BOLTS Parts selector', select 'Standard | DIN | DIN912'. Set the key to M2 and the length to 4 mm. Click twice on the 'Add part' button.

Drag-and-drop the two screw bodies into the Std-Part-Container 'Pilot Light Complete'. Rename the screws to 'Terminal Screw 1' and 'Terminal Screw 2'.

The pilot lamp casing has no variants and will therefore not change (except e.g., in color). Therefore, you can leave the screws just placed and not worry about an attachment relation. Click on the first screw and edit the placement: Set X = 3.5 mm, Y = 0 mm, Z = −32 mm (Figure D-43). Then, set the placement of the second screw to X = −3.5 mm, Y = 0 mm, Z = −32 mm.

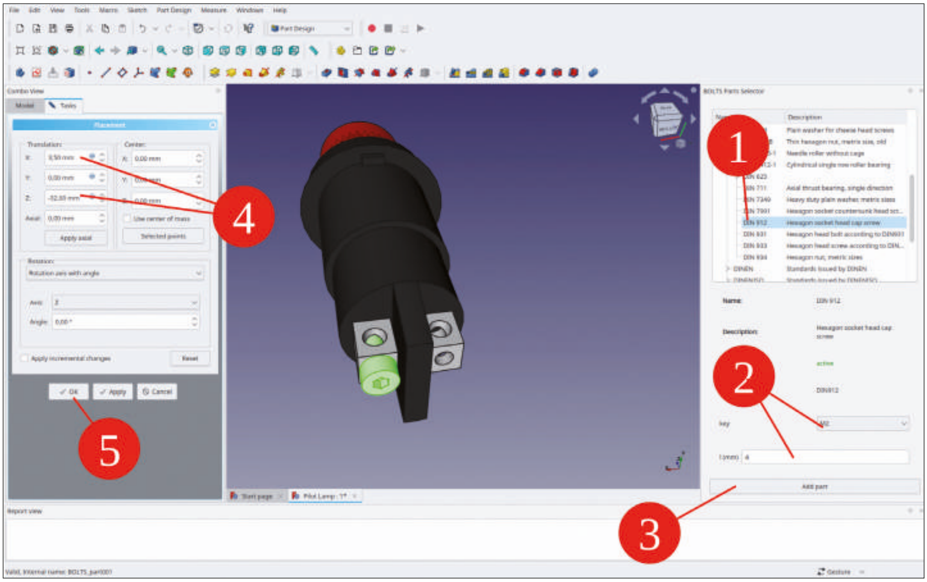


Figure D-43

In the tree view, change the appearance of the screws and select 'Chrome' as the material.

As a last thing, the fastener nut is missing. Create a new body object (blue 'Create body' tool button) and rename it to 'Pilot Light Nut'. If necessary, drag-and-drop it into the Std-Part-Container 'Pilot Light Complete'.

In the tree view, hide all other body objects.

Start the sketcher and select the XY plane as the sketching plane.

Draw two circles that are centered around the origin. End the drawing command with a right click. Constrain the diameters to 16 mm and 19.9 mm. Close the sketch.

For the end face of the nut, a datum plane is needed, because some details originate there. In the 3D view, click 'into the blue' to deselect the sketch that was just created. Then, click the 'Create a datum plane' tool button. In the task window, the collector for Reference 1 is activated when the window opens. Switch to the 'Model' tab. In the tree view, expand the coordinate system of the new body and mark the XY plane. Return to the 'Tasks' tab and enter an attachment Z offset of –7.5 mm. Close the task window.

In the tree view, rename the new datum plane as 'Nut Top Face'.

In the tree view, mark the sketch and click the 'Pad' tool button. In the task window, set the type to 'Up to face' and, in the 3D view, click the datum plane (Figure D-44).

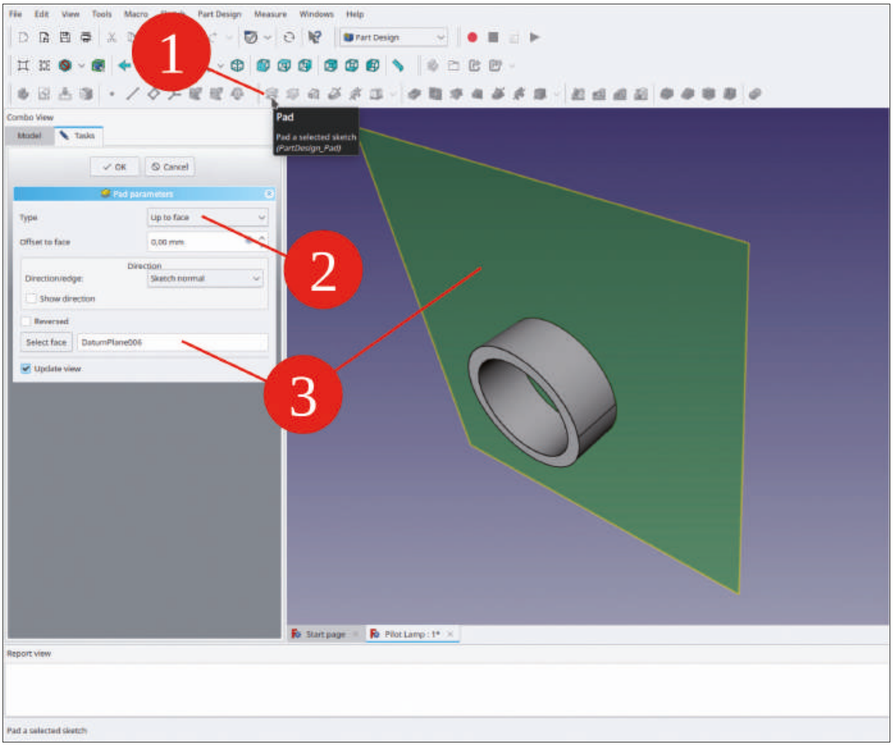


Figure D-44

The profile of the fastener nut has a coarse, grooved pattern. The pattern starts at the end of the face. To sketch the first groove, in the 3D view, mark the 'Nut Top Face' datum plane and start the sketcher. Close and reopen the sketch to display the already-present 3D geometry. From the main menu, select 'View | Orthographic view' and 'Sketch | View section'.

Sketch a circle that is centered on the X axis and constrain its diameter to 1.2 mm.

Click on the 'External geometry' tool button and click the outer contour of the nut, which is then displayed in violet, as a construction geometry. End the 'External geometry' command with a right click.

Mark the center of the new circle, and the violet construction geometry. Click on the 'Constrain point onto object' tool button (Figure D-45). Close the sketch (top).

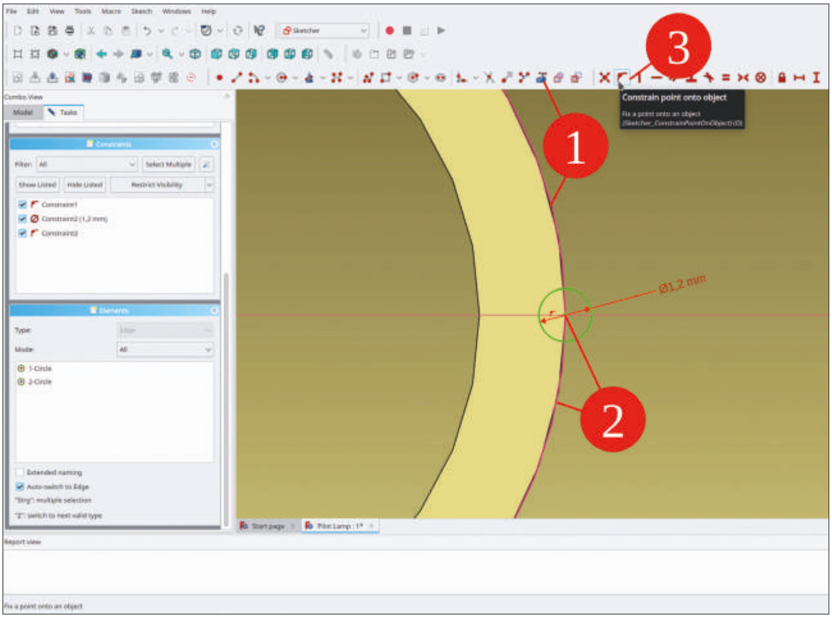


Figure D-45

Now create the grove: In the tree view, mark the new sketch and click the 'Pocket' tool button. In the task window, select the type 'Dimension', the length 5 mm and check the 'Reversed' checkbox (Figure D-46). Eventually, you need to move the 3D view a bit to see the groove. Close the task window with the OK button.

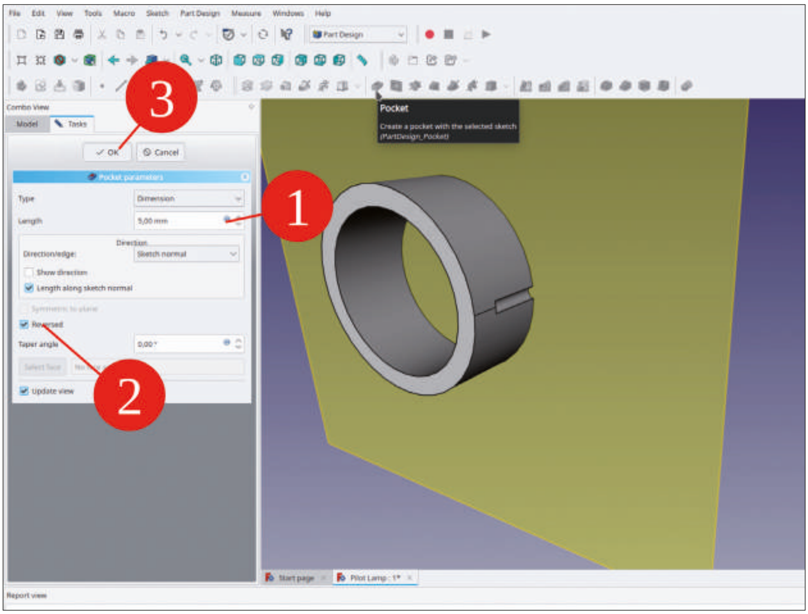


Figure D-46

From the main menu, select 'Part Design | Apply a pattern | PolarPattern'. A selection window opens. Scroll down and mark the last entry, the pocket object just created. Click the OK button to proceed.

In the task window, the axis ('Normal sketch axis') and the total angle are preset. Enter a number of 30 for the occurrences and close the task window with the OK button (Figure D-47).

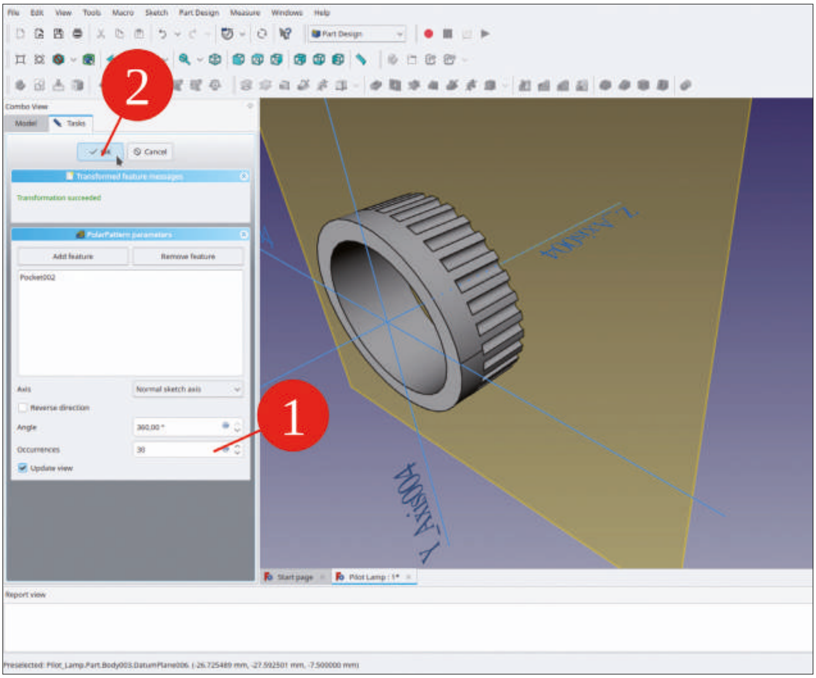


Figure D-47

The nut has further featured on the rim, for which a mating key tool exists. As a preparation, in the tree view, hide the 'Nut Top Face' datum plane. Start the sketcher and select the YZ plane as the sketching plane.

Close and reopen the sketch to display the already-present 3D geometry. From the main menu, select 'View | Orthographic view' and 'Sketch | View section'.

Click on the 'External geometry' tool button and click the projected top face of the nut. Draw a rectangle that is centered on the Z axis, with one edge locked to the violet top face reference (Figure D-48). Set the Width of the rectangle to 5.4 mm and the height to 1.2 mm. Close the sketcher.

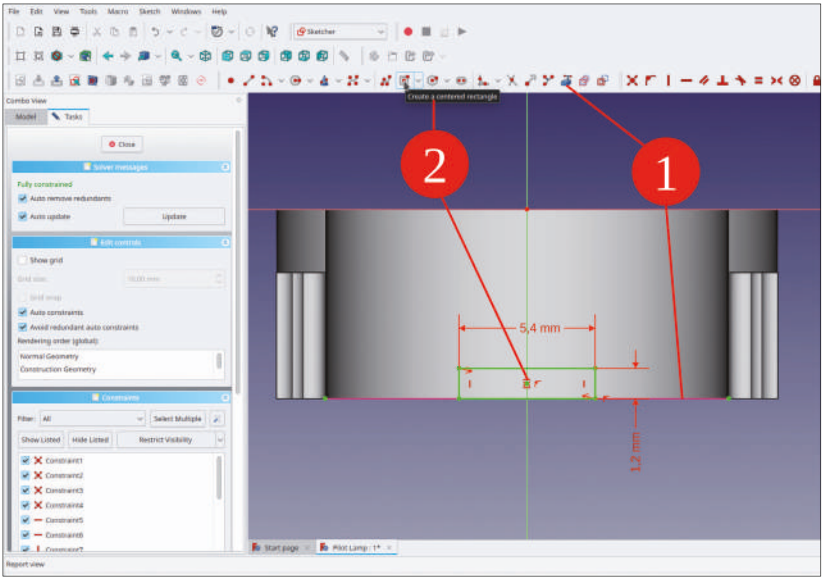


Figure D-48

In the tree view, mark the new sketch and click the 'Pocket' tool button. In the task window, select 'Through all' for the type (Figure D-49). Close the task window with the OK button.

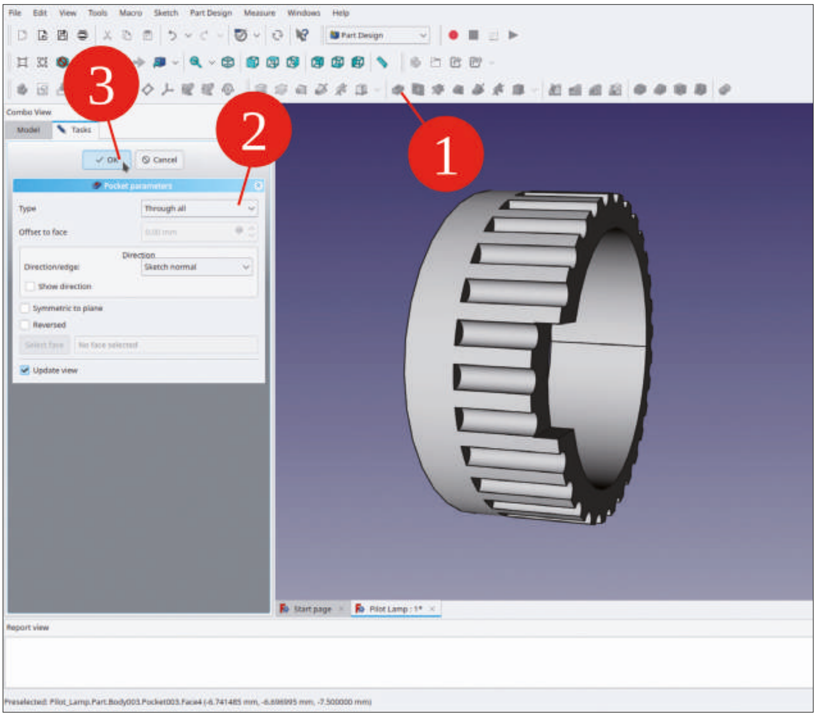


Figure D-49

From the main menu, select 'Part Design | Apply a pattern | PolarPattern'. A selection window opens. Scroll down and mark the last entry, the pocket object just created. Click the OK button to proceed. In the task window, select the axis 'Base Z axis' and set the number of occurrences to 4 (Figure D-50). Close the task window with the OK button.

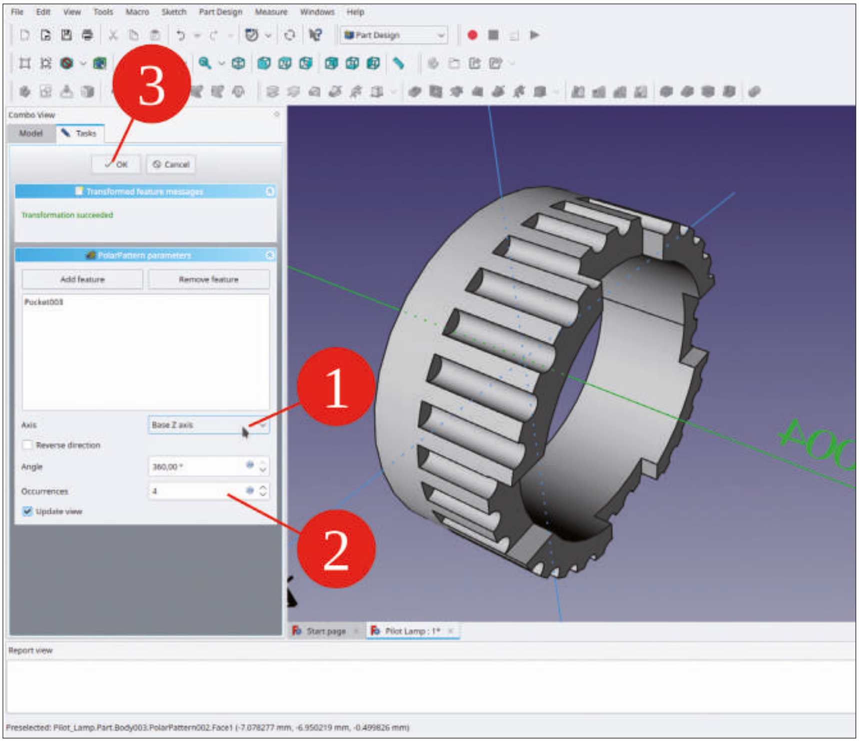


Figure D-50

In the tree view, right-click the nut body and select 'Appearance…' from the context menu. Select 'Shiny Plastic' as the material, and a bright gray for the color.

In the tree view, mark the body of the nut. In the property list, open the placement task window by clicking the 'Placement' field and on the … button. Set the Z translation parameter to a typical front panel thickness, e.g., –2 mm. Close the task window with the OK button and save your work.

The finalized pilot light is depicted in Figure D-51. The creation of this object was quite complex, especially with the step-by-step mode. But now, the little gadget is ready to be inserted by just coy-and-paste into as many designs as you like!

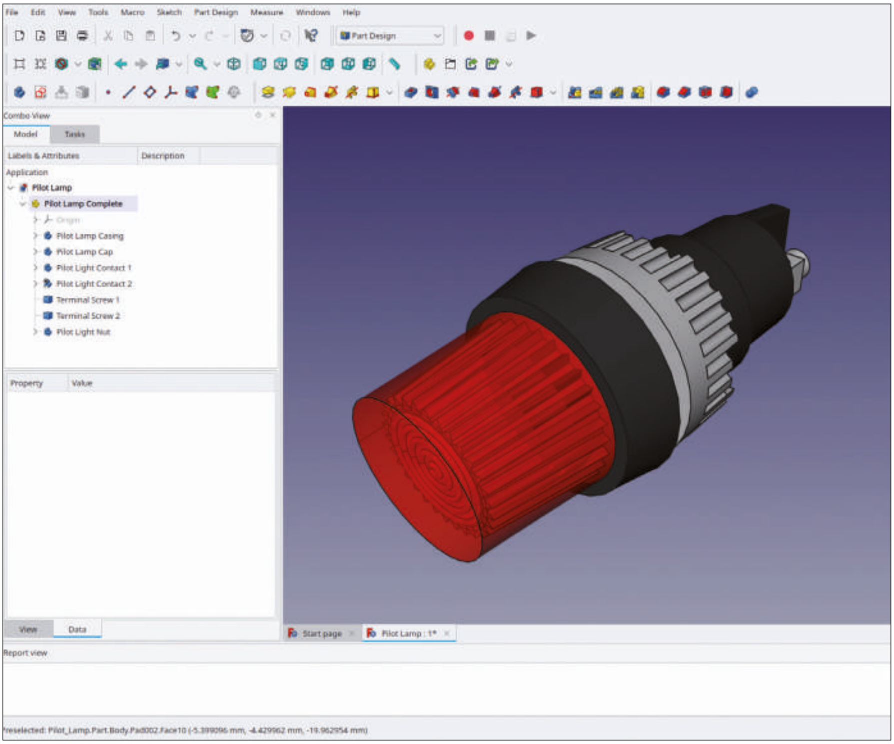

File
Edit
View
Tools
Macro
Sketch
Part Design
Measure
Windows
Help
Part Design
Combo View
Model
Tasks
Labels & Attributes
Description
Application
Pilot Lamp
Pilot Lamp Complete
Origin
Pilot Lamp Casing
Pilot Lamp Cap
Pilot Light Contact 1
Pilot Light Contact 2
Terminal Screw 1
Terminal Screw 2
Pilot Light Nut
Property
Value
View
Data
Start page
Pilot Lamp : 1*
Report view

Figure D-51

Appendix E • The Toggle Switch

As always, follow the standard procedure: Start a new file, save it as 'Toggle Switch'. In that file, create a new Std-Part-Container and rename it to 'Toggle Switch Complete'. Start with the front-side fastener of the toggle switch. Switch to the 'Part Design' workbench and create a new body object and rename that to 'Toggle Switch Knurled Nut'.

The knurled nut is attached later to the front panel. Therefore, you create it as the first thing here, and refer to it with all other following parts. The knurled nut has a thickness of 2.3 mm, and a slightly conical top face.

A few words towards its practical use: The nut is knurled to improve the grip for your fingertips, not the pliers' grip! After the knurled nut has been properly set, the switch is tightened with a wrench from the back side of the panel. If you tighten the knurled nut with pliers instead, it's sure to be mashed, and probably some scratches will be left on your front panel, too.

The sketch of the end cross-section of the loft object is located on a datum plane:

In the tree view, show the coordinate system of the nut body (mark it and press the SPACE key). In the 3D view, mark the XY plane and click the 'Create a datum plane' tool button. In the task window, set the attachment Z offset to 2.85 mm (Figure E-1). Close the task window and rename the new datum plane to 'Knurled Nut Top Face'.

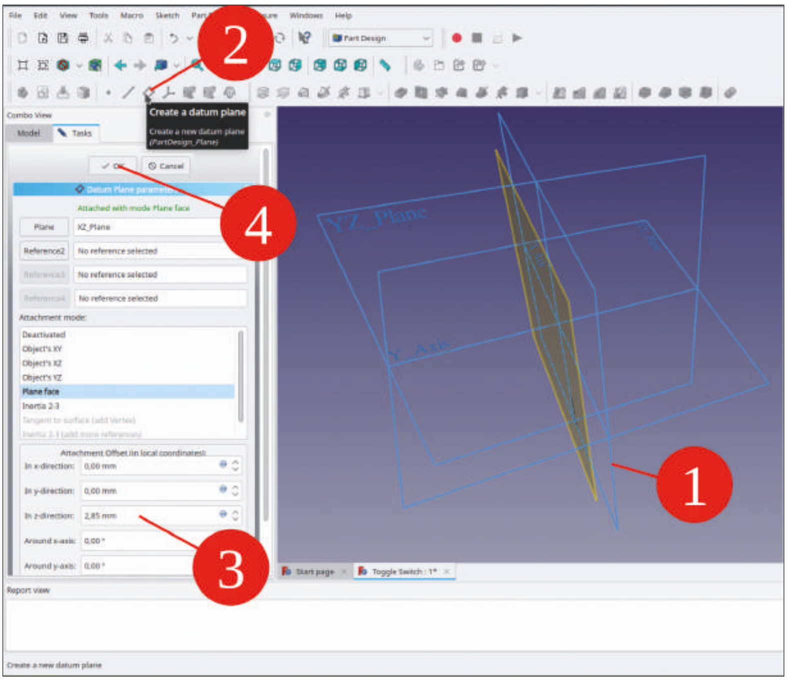


Figure E-1

In the 3D view, mark the XY plane and start the sketcher. Draw a circle that is centered around the origin. End the drawing command with a right click.

In the 'Elements' list, right-click the circle and select 'Diameter Constraint' from the context menu. Enter 16.2 mm for the diameter (Figure E-2). Close the task window.

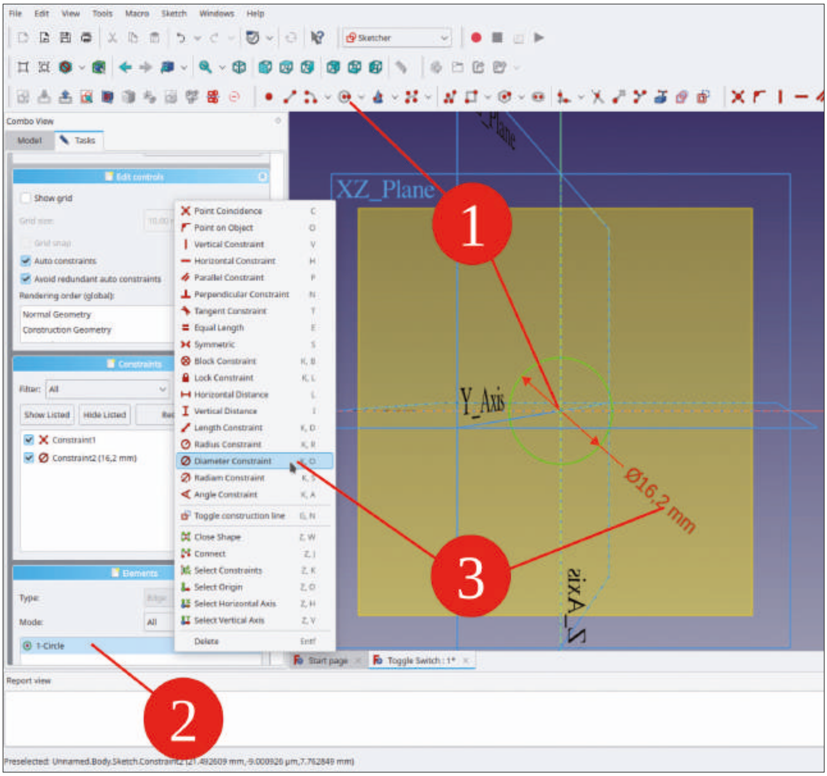


Figure E-2

In the tree view, mark the new sketch and click the 'Pad' tool button. In the task window, set the length to 2.3 mm (this models the cylindrical part of the nut, Figure E-3 step 1).

In the tree view, mark the 'Knurled Nut Top Face' datum plane and start the sketcher. Draw a circle which is centered around the origin and constrain its diameter to 12.4 mm. Close the sketch.

In the tree view, hide the 'Knurled Nut Top Face' datum plane.

Hold down the CTRL key. In the 3D view, mark the top face of the cylinder and the new sketch (Figure E-3, steps 2 and 3). Click on the 'Additive loft' tool button. Close the task window with the OK button (Figure E-4). You have certainly noticed — the loft refers to a facet. This could cause havoc but the simple nut will be finished soon, so any re-enumeration damage that might occur should be limited.

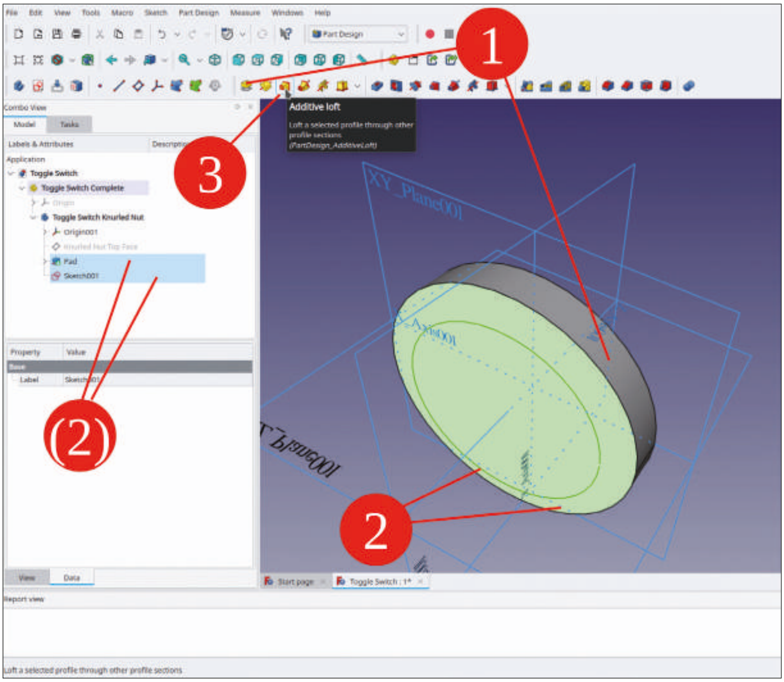


Figure E-3

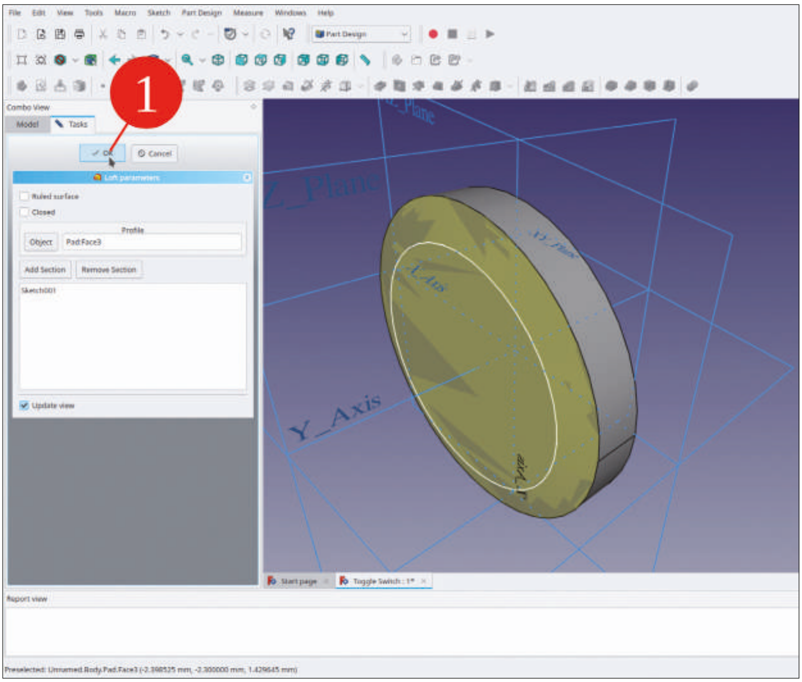


Figure E-4

For the threaded hole of the nut, sketch a circle on the XY plane, which is centered around the origin. Constrain its diameter to 12 mm (Figure E-5). If the view of the sketching plane is obscured by the nut body, from the main menu, select 'Sketch | View section'.

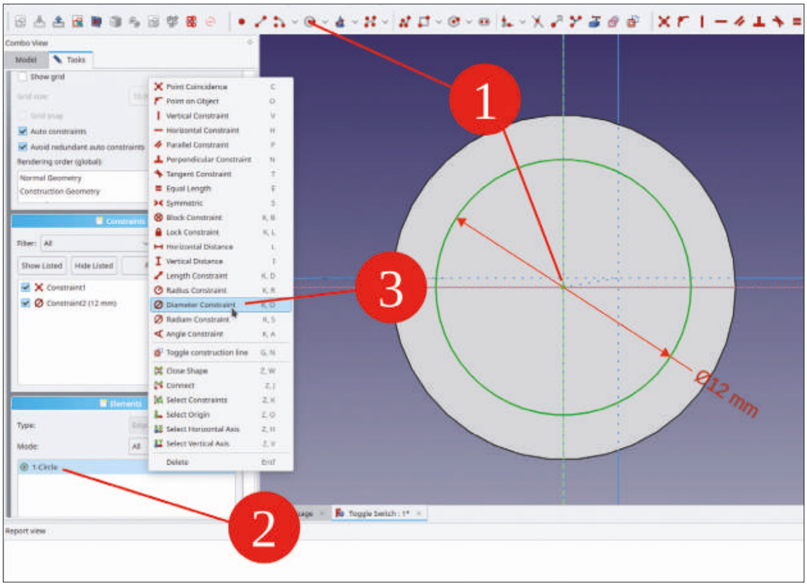


Figure E-5

In the tree view, mark the new sketch and click the 'Pocket' tool button (Figure E-6). In the task window, select the type 'Through all' and check the 'Reversed' checkbox (Figure E-7). Close the task window with the OK button.

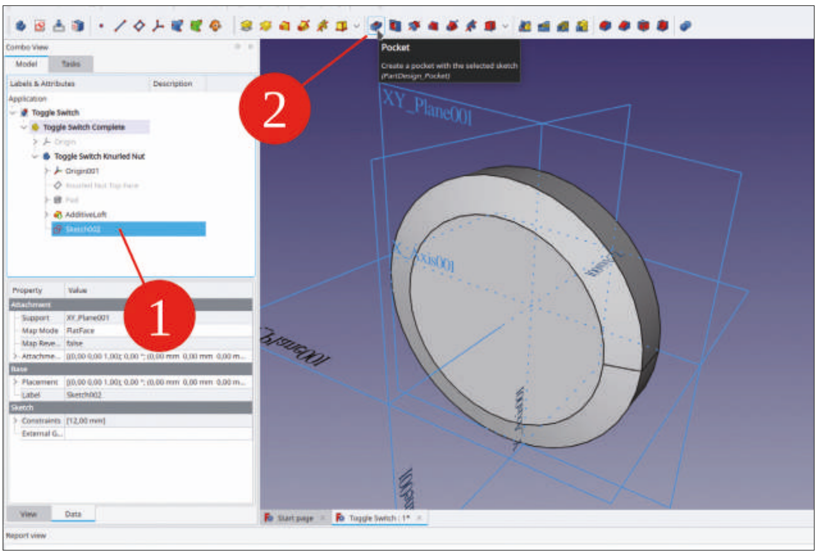


Figure E-6

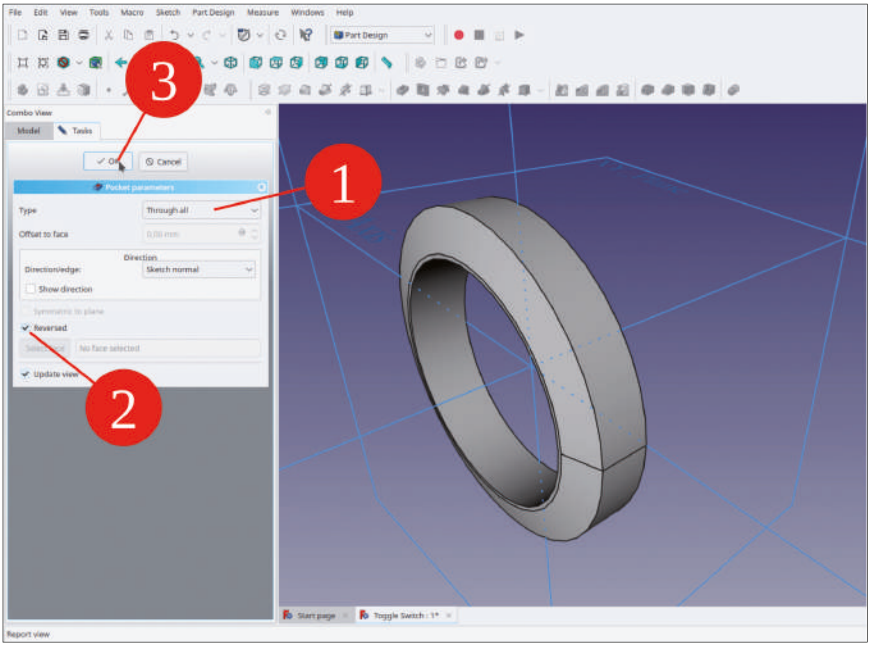


Figure E-7

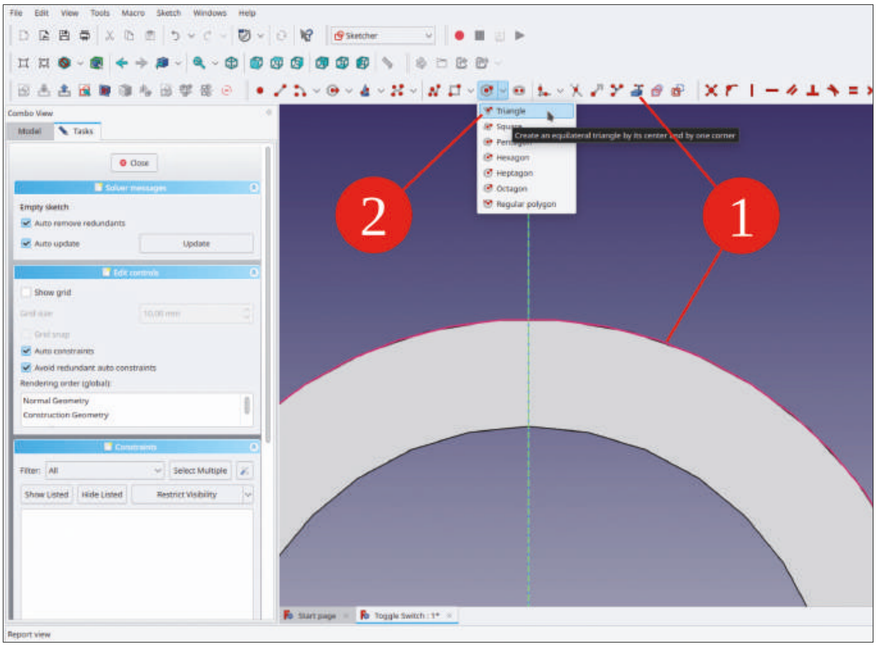


Figure E-8

For the knurled surface, start by drawing one profile on the XY plane. Mark the XY plane and start the sketcher. From the main menu, select 'View | Orthographic view' and 'Sketch | View section'.

Click on the 'External geometry' tool button and mark the outer contour of the nut. The outline will be displayed in violet, indicating that a construction object has been created, to which you can refer. Then, click the 'Create an equilateral triangle by its center and a corner' tool button (Figure E-8).

With the first click, lock the center of the triangle to the Y axis. With the second click, place one corner of the triangle also on the Y axis, somewhere below (Figure E-9). It takes some aiming with the cross hair cursor, until the target object highlights (especially with the corner point). Being precise with these items reduces the degrees of freedom right away.

Mark the horizontal line of the triangle and constrain its length to 0.2 mm (Figure E-9, steps 3, 4). Now, the triangle is really small. Zoom in to display the center point. You can drag the center closer to the violet circle if that helps with the next step. The center point will stay nicely locked to the Y axis.

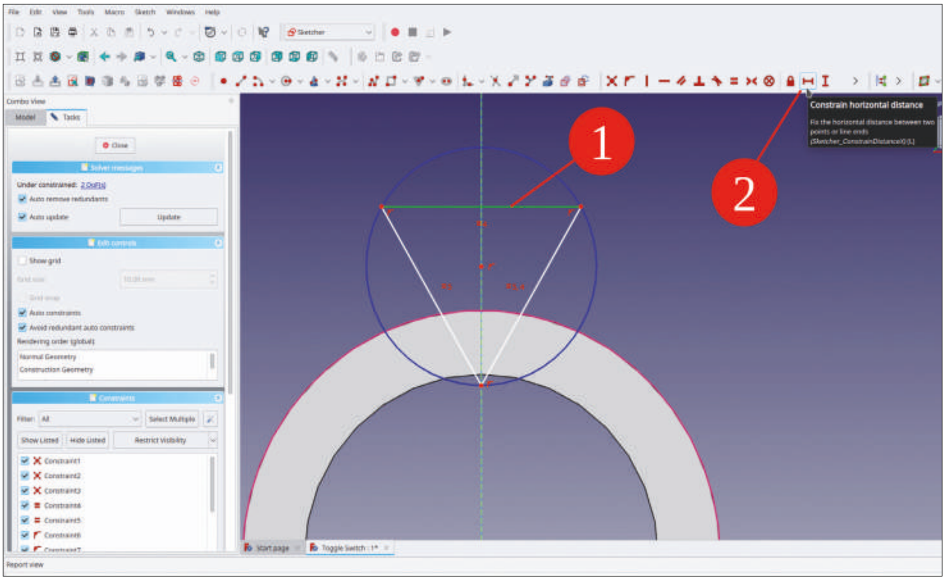


Figure E-9

Mark the center point of the triangle and the violet nut outline. Then, click the 'Constrain point onto object' tool button (Figure E-10). The sketch is displayed in bright green — fully constrained. Close the sketch with the 'Close' button (top).

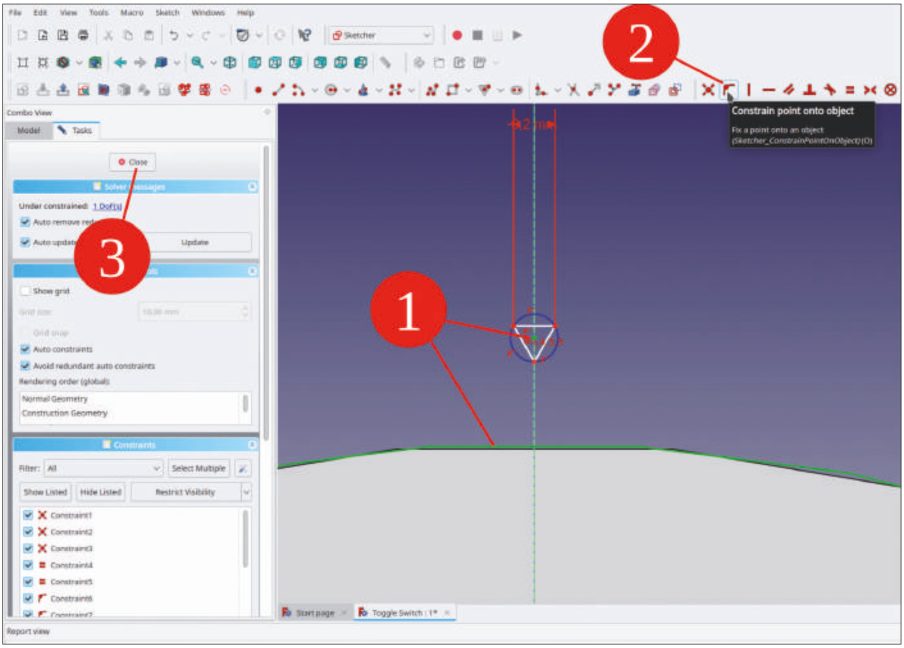


Figure E-10

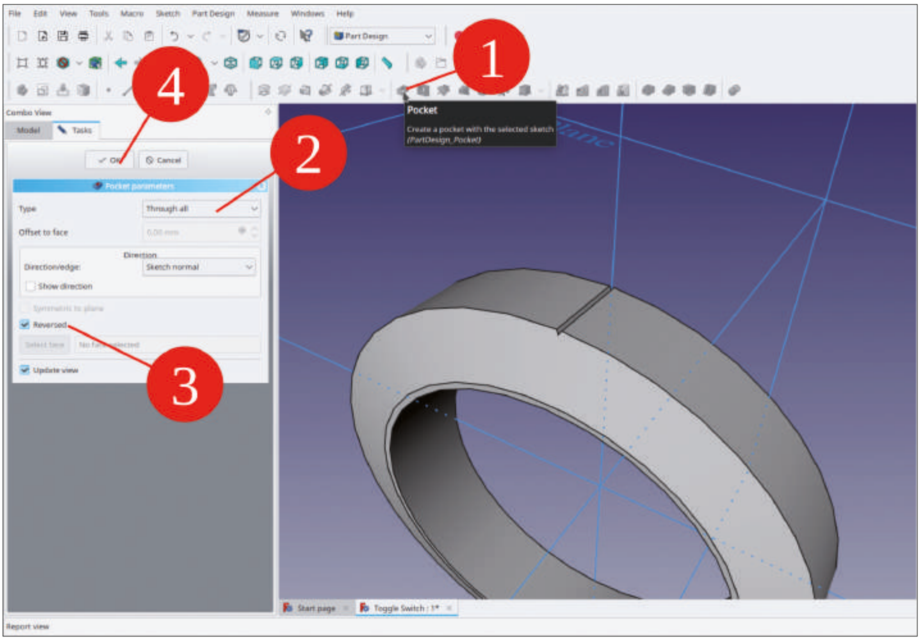


Figure E-11

In the tree view, mark the new sketch and click the 'Pocket' tool button. In the task window, select for the type 'Through all' and check the 'Reversed' checkbox (Figure E-11). Close the task window with the OK button.

In the tree view, mark the last design state 'Pocket'. From the main menu, select 'Part Design | Apply a pattern | Polar pattern' (Figure E-12).

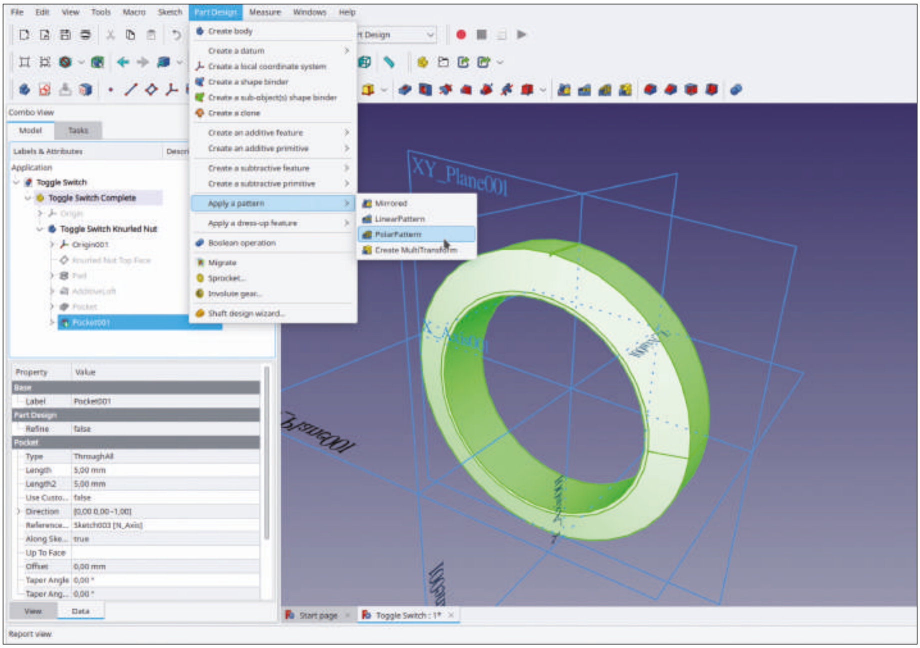


Figure E-12

In the task window, set the number of occurrences to 140 (the generation takes a little time, Figure E-13). Then close the task window with the OK button. In the tree view, hide the coordinate system of the nut (mark it and press the SPACE key).

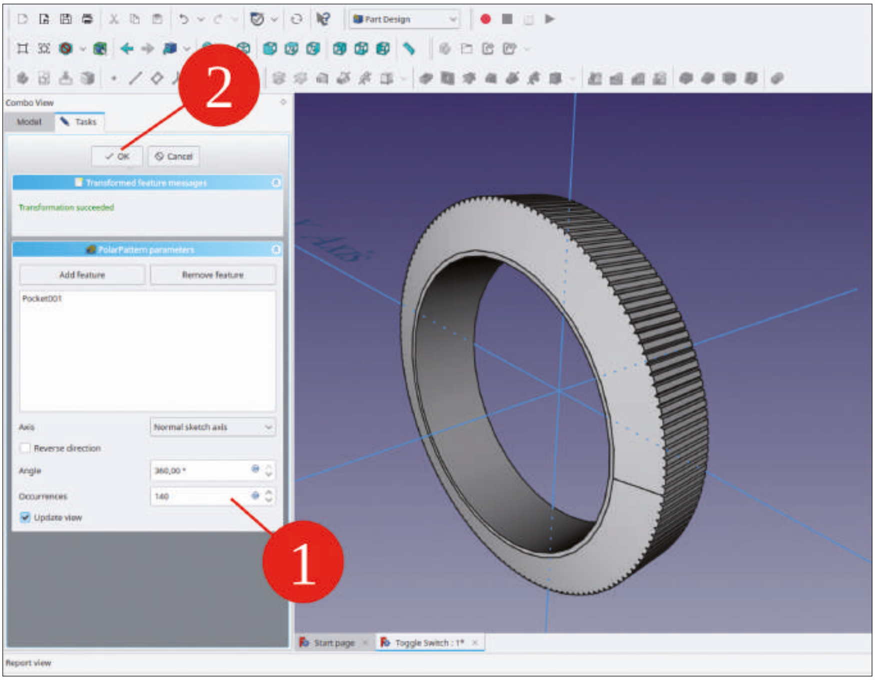


Figure E-13

Now the nut is nearly done. To make it shine a bit more, in the tree view, right-click the 'Toggle Switch Knurled Nut' body and select 'Appearance…' from the context menu. For the material, select 'Chrome' and close the task window.

The switch casing consists of a cylindrical shaft. As a rotation lock, the shaft has a groove. The plastic casing for the contacts is attached to the rear of the shaft. When modeling the cylindrical part, it is a good question how to consider the rotation lock. It is, of course, possible to simply sketch the outline with all features in a sketch at once. However, a sketch with many degrees of freedom could be frustrating to constrain properly. Our simple example implies that it may be easier to sketch the cylinder and the grove in two separate, less troublesome sketches. Sometimes, that is not satisfactory, too, as it may make the structure of the part more complex or obscure a design idea. With sketches, this is a recurring question. We will therefore use the groove as an example case and define a construction element. This helps to reduce the degrees of freedom, and it keeps all related features together in one sketch.

For the casing, create a new body object and rename it to 'Toggle Switch Casing'. If necessary, drag-and-drop it into the 'Toggle Switch Complete' Std-Part-Container.

Start the sketcher and from the selection dialog, select the XY plane as the sketching plane. (An alternative way to start the sketcher on a certain plane is to mark its item line in the tree view before the 'Sketcher' tool button is clicked).

Draw a circle that is centered around the origin, and constrain its diameter to 12 mm.

Draw a second circle, also centered on the origin, and constrain the diameter to 8.5 mm (Figure E-14).

Expand the 'Create Rectangle' tool button and click the 'Create a centered rectangle' tool button (Figure E-14). Draw a rectangle that is centered on the Y axis. Constrain the dimensions of the rectangle: Set the width to 1.5 mm, and the height to 3 mm (Figure E-15, steps 2 and 3).

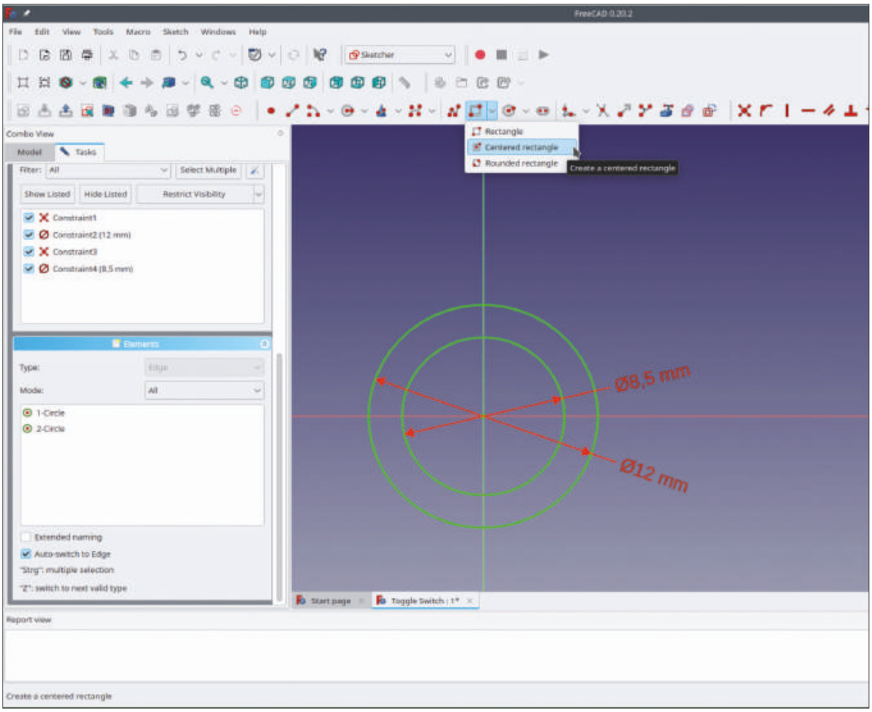


Figure E-14

Mark the lower corner of the rectangle, then the origin. Constrain the vertical distance to 5.2 mm (all the measures were taken with a caliper from an existing switch, Figure E-15, step 4).

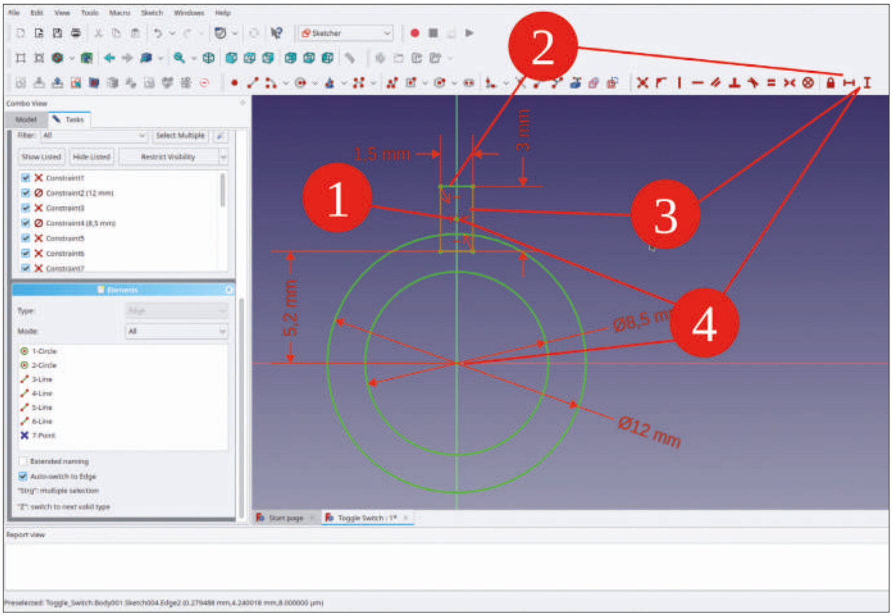


Figure E-15

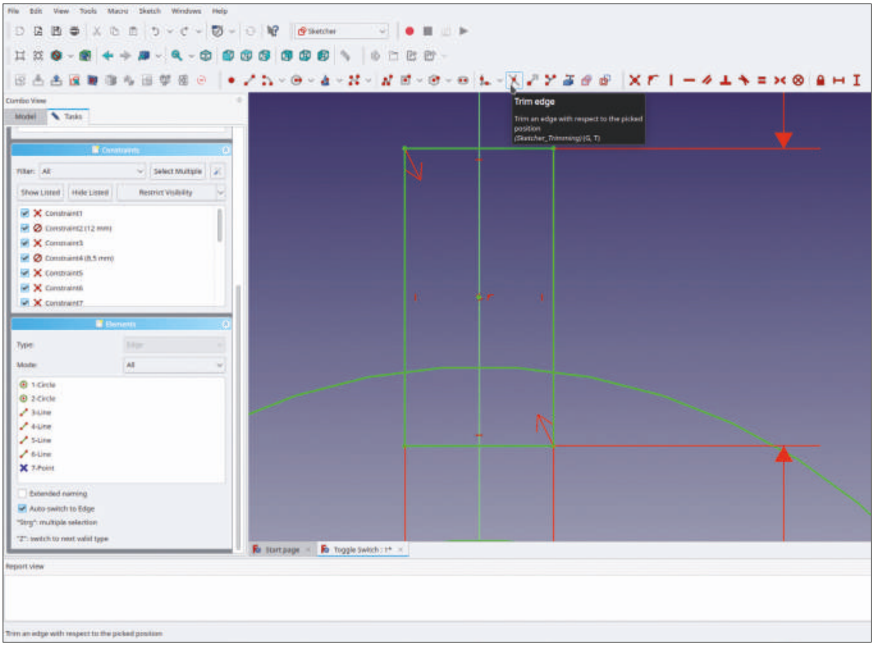


Figure E-16

At this point, it is tempting to use the 'Trim edge' tool, and to cut the circle with the rectangle (Figure E-16). Note that the sketch is fully constrained now, before this operation. If you proceed, constraining the resulting sketch will get hairy: When dimensions change later on, complicated sketches can end up distorted. It is therefore better to redefine the rectangle as a construction element:

Fortunately, this is simple: Mark the four-line objects in the 'Elements' list, and right-click the selection. From the context menu, select 'Toggle Construction Line'. The rectangle is now displayed in light blue, and the sketch is still fully constrained.

Now, you can safely click the 'Trim edge' tool button and cut away the part of the circle which is located inside the rectangle (Figure E-17). As depicted in Figure E-17, the sketch is still fully constrained. Click on the 'Create line' tool button and draw the lines which define the groove, as shown in Figure E-18. The sketch stays fully constrained, and you would even have the chance to change, e.g., the width of the slot without distorting the rest of the sketch. Close the sketch (top).

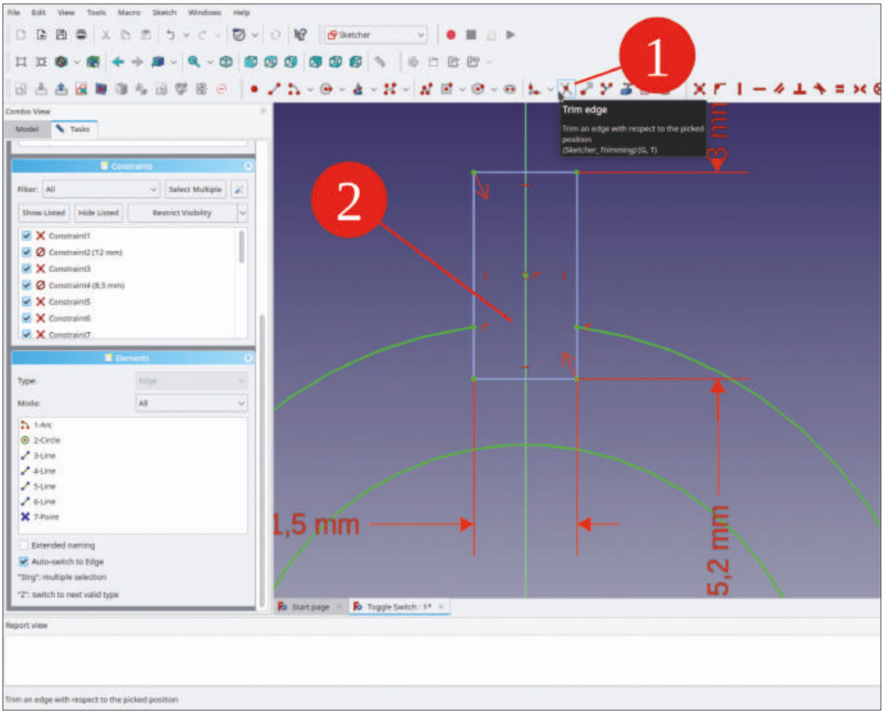


Figure E-17

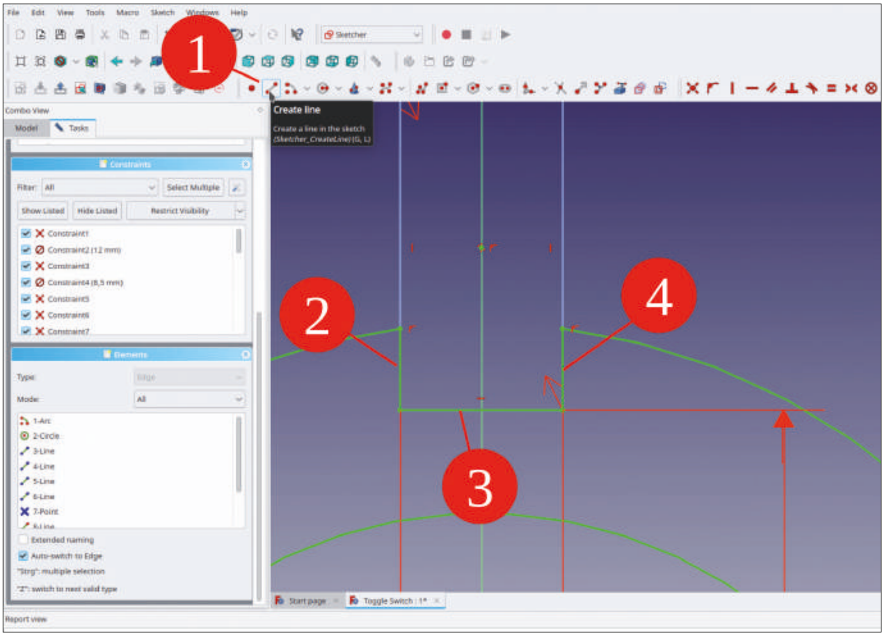


Figure E-18

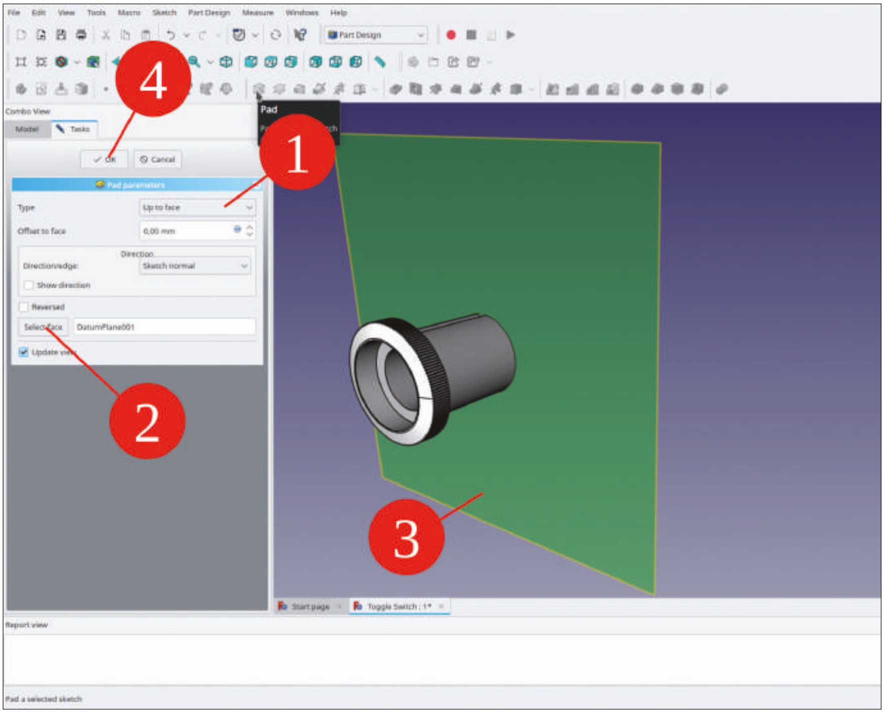


Figure E-19

Before padding the new sketch to form the shaft, a datum plane for the definition of the plastic part of the casing is a useful addition. In the tree view, expand the coordinate system of the 'Toggle Switch Casing' body (named Origin002) and mark the XY plane. Click on the 'Create a datum plane' tool button. Enter a value of –13 mm for the Z attachment offset and close the task window. Rename the new datum plane to 'Plastic Casing Position'.

In the tree view, mark the new sketch and click the 'Pad' tool button. In the task window, select 'Up to face' as the type, and click the datum plane, either in the 3D view, or in the tree view which you can access by switching to the 'Model' tab (Figure E-19). Close the task window with the OK button.

In the tree view, click the 'Toggle Switch Casing' body. In the property list, edit the placement parameters: Either click the 'Placement' property line and open a task window by clicking the ... button to the right, or expand the property list entries 'Placement' and 'Position' to find the line for Z. Enter a value of 3 mm manually (by this amount, the switch casing shall later protrude from the front panel).

Now you add the housing for the contacts. In the 3D view, mark the datum plane 'Plastic Casing Position' and start the sketcher. From the main menu, select 'View | Orthographic view' and 'Sketch | View section'. Draw a rectangle centered around the origin. Constrain the width to 26 mm, the height to 23.2 mm (Figure E-20). Close the sketch.

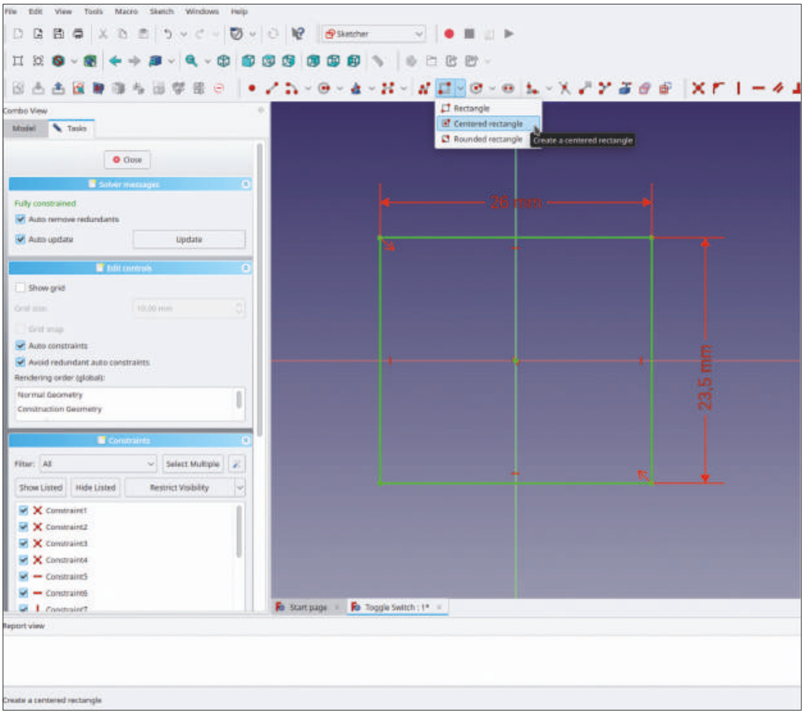


Figure E-20

In the tree view, mark the new sketch and click the 'Pad' tool button. In the task window, enter a value of 16.6 mm for the length and check the 'Reversed' checkbox. Close the sketch. In the tree view, hide the datum plane 'Plastic Casing Position'.

In the 3D view, mark the four outer boundaries of the newly extruded casing body.

Click on the 'Fillet' tool button (Figure E-21). For the radius, enter a value of 2 mm and close the task window with the OK button.

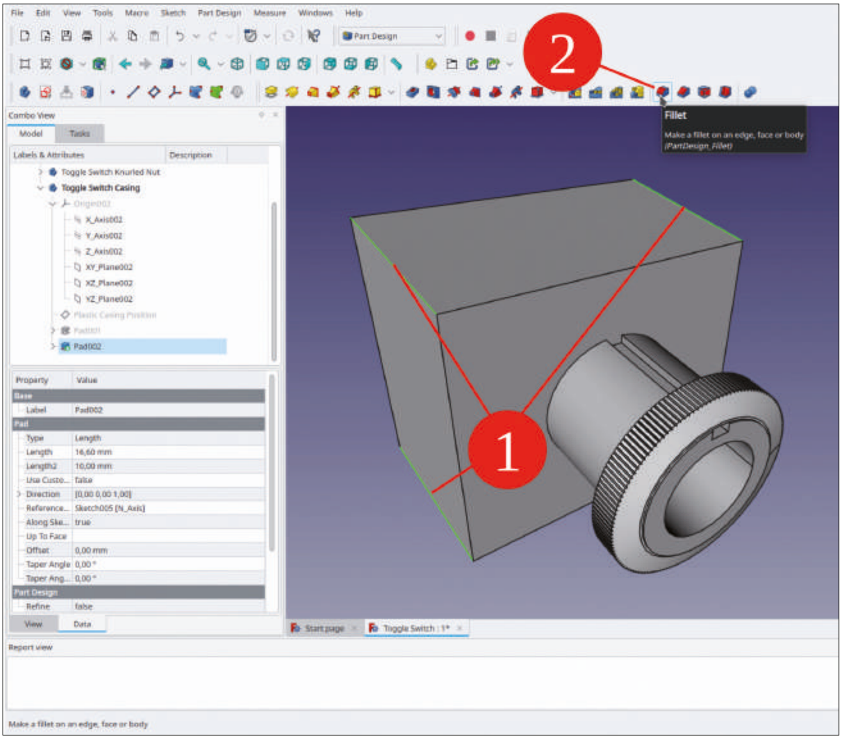


Figure E-21

In the 3D view, mark one segment of the lower casing boundary. Again, click the 'Fillet' tool button. Leave the radius at the default value of 1 mm. The rounded edge is automatically completed (Figure E-22). In this case, this is very useful. If the fillet cannot be completed due to e.g., a too-large radius value, the responses to the command can be confusing. In these cases, start with a very small value for the radius, and spot the cause for the problem by slowly increasing the radius. Close the task window with the OK button.

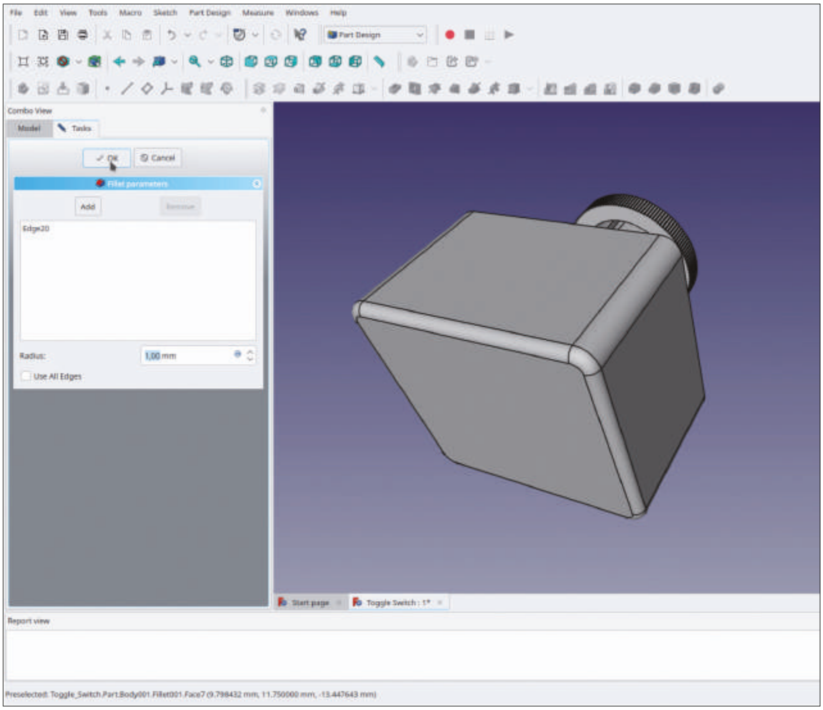


Figure E-22

The switch casing is now completed. In order to have a better representation, some coloring is useful. First, right-click the body of the casing in the 3D view and select 'Appearance…' from the context menu. For the material, select 'Chrome' and close the task window.

The plastic part of the switch is usually made of Bakelite. Because the switch casing is completed, let's limit to coloring facets of the body. In the tree view, right-click the last design state in the 'Toggle Switch Casing' body (the tip) and select 'Set colors' from the context menu.

In the 3D view, mark all facets which belong to the bottom of the casing, and which are not directed to the front. Either click the facets one-by-one, holding the CTRL key, or rotate the switch into a comfortable position, click the 'Box selection' button and pull a rectangle over the rear end of the switch casing, which is faster (Figure E-23). Set the color to black.

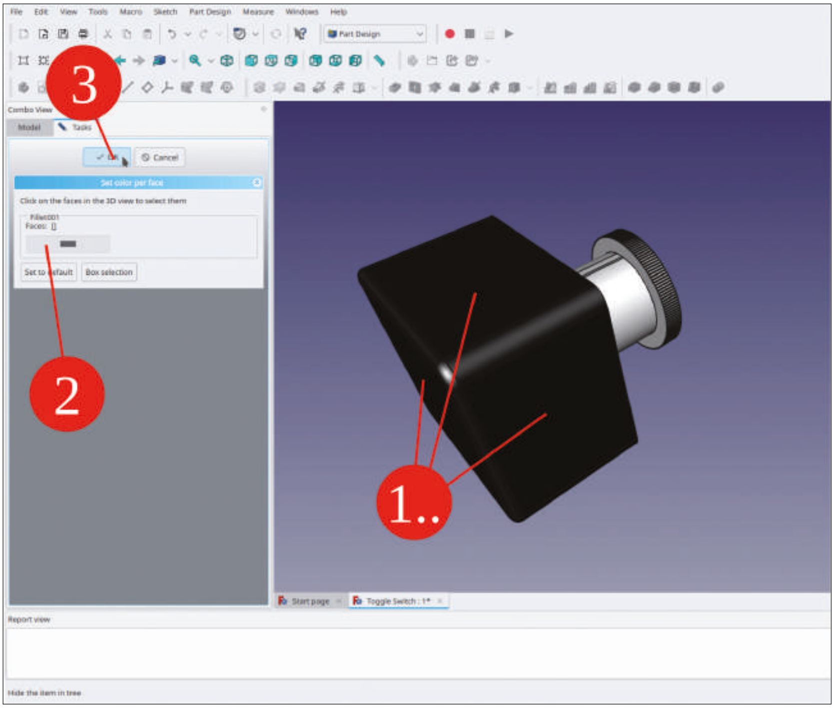


Figure E-23

Now the terminals need to be modeled. For the first contact, click the blue 'Create body' tool button. Rename the first contact to 'Toggle Switch Terminal 001'. If necessary, drag-and-drop the new body into the Std-Part-Container 'Toggle Switch Complete'.

Expand the new body and display its coordinate system (mark and SPACE key).

In the tree view, hide the nut and the casing. In the 3D view, mark the YZ plane and start the sketcher.

Draw a centered rectangle: With the first click, lock the center to the Y axis, with the second click, lock one corner to the Z axis (Figure E-24, steps 1, 2). End the drawing command with a right click. Mark a horizontal line of the new rectangle and constrain its width to 3 mm. Mark a vertical line and constrain the height to 2.4 mm.

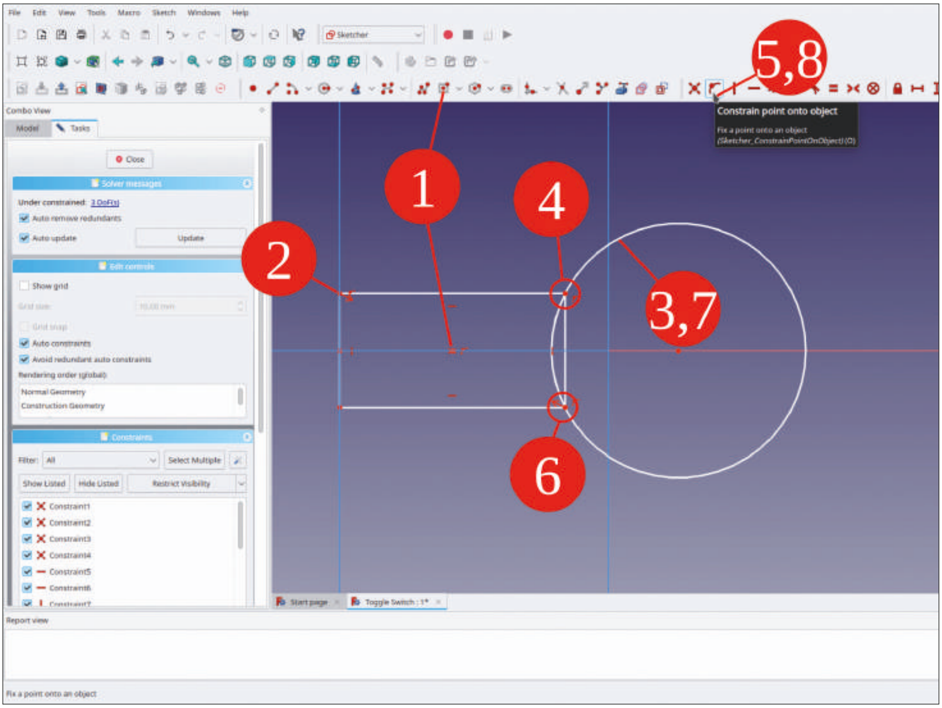


Figure E-24

For the soldering pad, draw a circle of arbitrary position and size. End the drawing command with a right click. Mark the circle and one outer corner of the rectangle. Then, click on the 'point on object' constraint button (Figure E-24, steps 3, 4, 5). Mark the other outer corner of the rectangle, and then the circle, and apply again the 'point on object' constraint (Figure E-24, steps 6,7,8).

In the 'Elements' list, right-click the circle and select 'Diameter constraint' from the context menu. Set the diameter to 4 mm.

Trimming away the surplus line and an arc from the circle would lead to many degrees of freedom that would need to be specified. It is better to keep the line of the rectangle, and to redefine it as a construction line: In the 'Elements' list, right-click the outer, vertical line of the rectangle. From the context menu, select 'Toggle construction line' (Figure E-25).

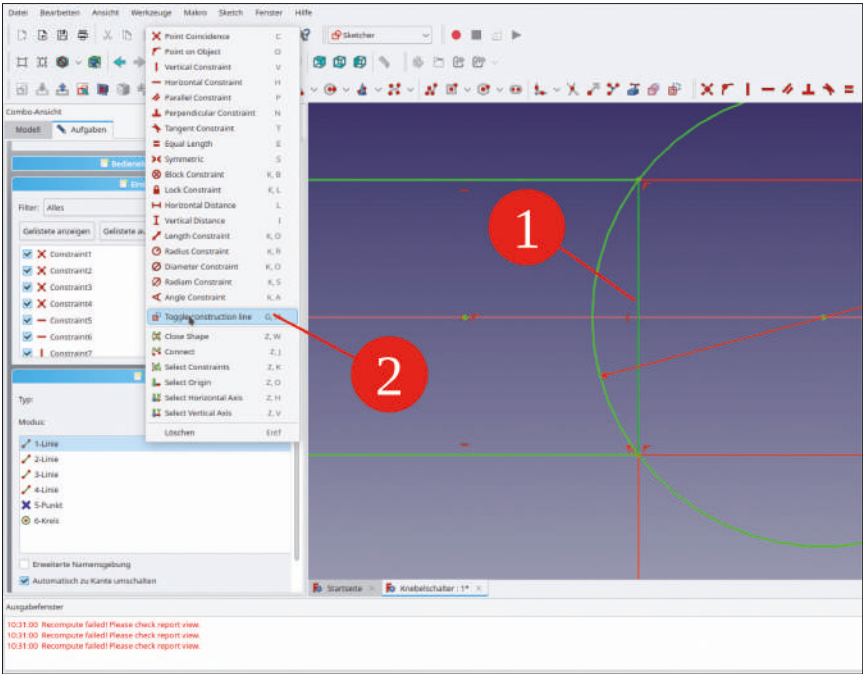


Figure E-25

Now, the arc which is located inside the terminal area can be cut away without disturbing the fully constrained state of the sketch. Click on the 'Trim edge' tool button, and then on the circle, within the rectangle (Figure E-26). The sketch remains fully constrained!

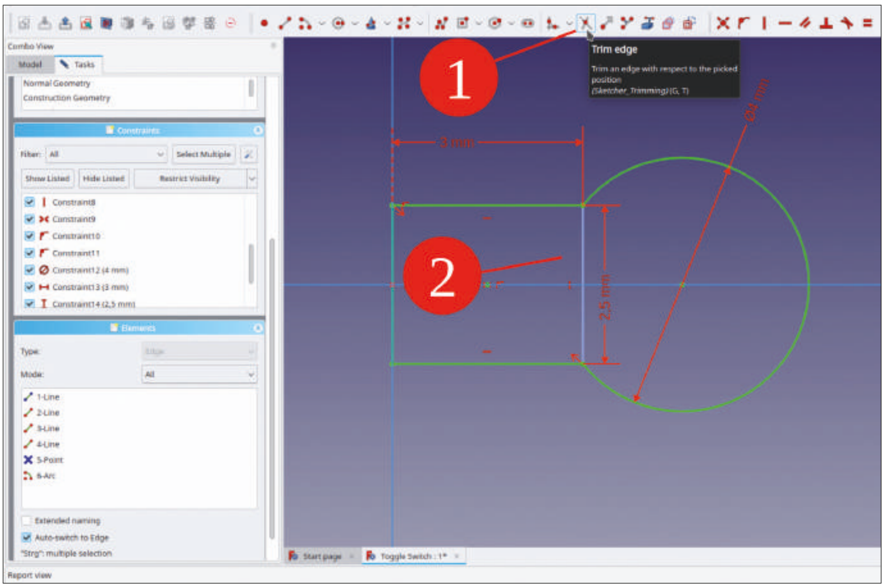


Figure E-26

The hole for the wire is still missing. Draw a circle with its origin locked to the center of the outlining arc. Constrain the diameter of the circle to 1.8 mm (Figure E-27). Close the sketch (top).

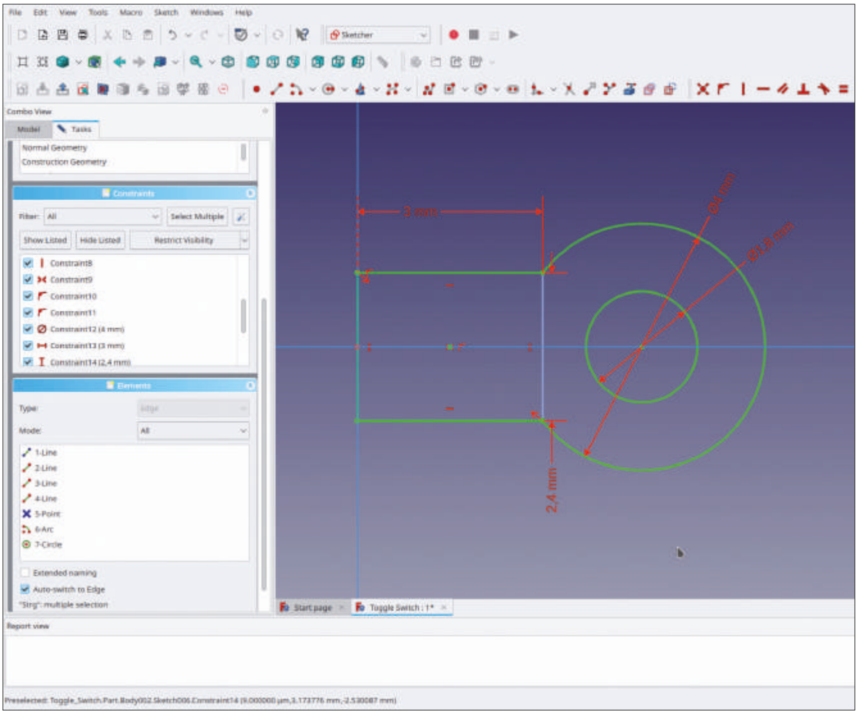


Figure E-27

In the tree view, mark the new sketch and click the 'Pad' tool button. In the task window, enter a value of 1.2 mm for the length, and check the 'Symmetric to plane' checkbox. Close the task window. In the tree view, right-click the terminal body and select 'Appearance…' from the context menu. Select 'Chrome' as the material.

The contact needs to be positioned. In order to see where it moves to, display the nut and the casing (in tree view, mark, and press SPACE key). Then mark the contact body again. In the property list, click into the 'Placement' property field and open the task window with the ... button. In the task window, set the placement translations to X = –9.00 mm, Y = 11.60 mm, Z = –22.50 mm (Figure E-28).

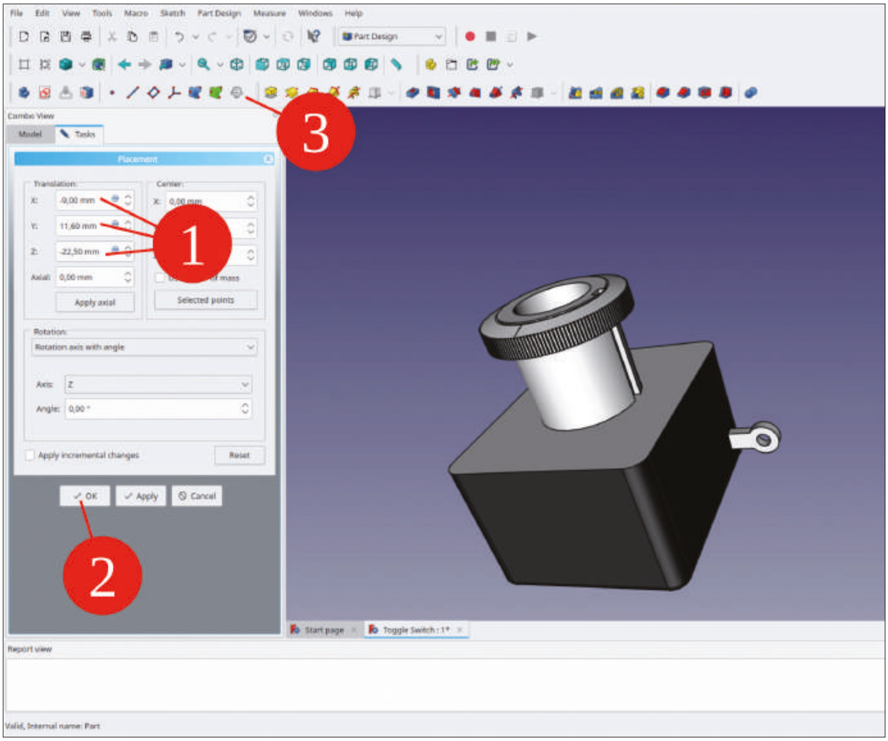


Figure E-28

Close the task window with the OK button. In the tree view, mark the terminal body and click the 'Create a clone' tool button (Figure E-28, number 3). In the tree view appears a new body object. Rename it to 'Toggle Switch Terminal 002' and drag and drop it into the 'Toggle Switch Complete' Std-Part-Container.

In the tree view, mark the cloned terminal. In the property list, expand the placement parameters and set the X translation to 6 mm. In the same way, generate two more clones, rename them, and set the X translations to 12 and 18 mm (Figure E-29).

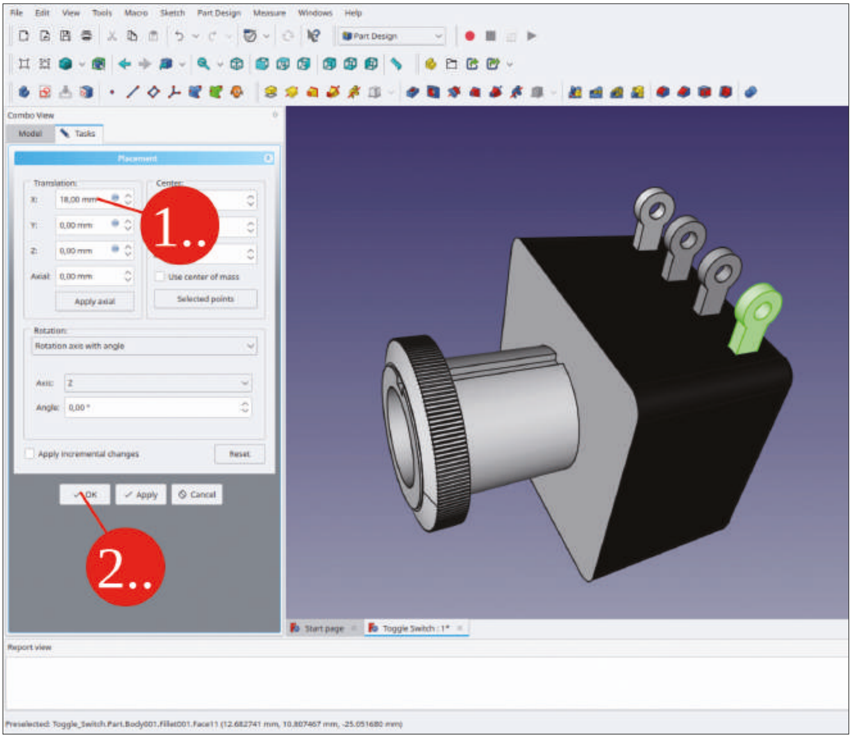


Figure E-29

Set the appearance of all the new terminals also to 'Chrome'.

Now clone each of the terminals again, rename them in sequence (Toggle Switch Terminal 005 ... 008) and drag-and-drop them all into the Std-Part-Container.

In the tree view, mark all the newly generated terminals. In the property list, click into the 'Placement' edit field, and then on the ... button to display the task window. In the rotation section, the Z axis is already preset. Enter an angle of 180° to move all the new terminals to the opposite side at once (Figure E-30).

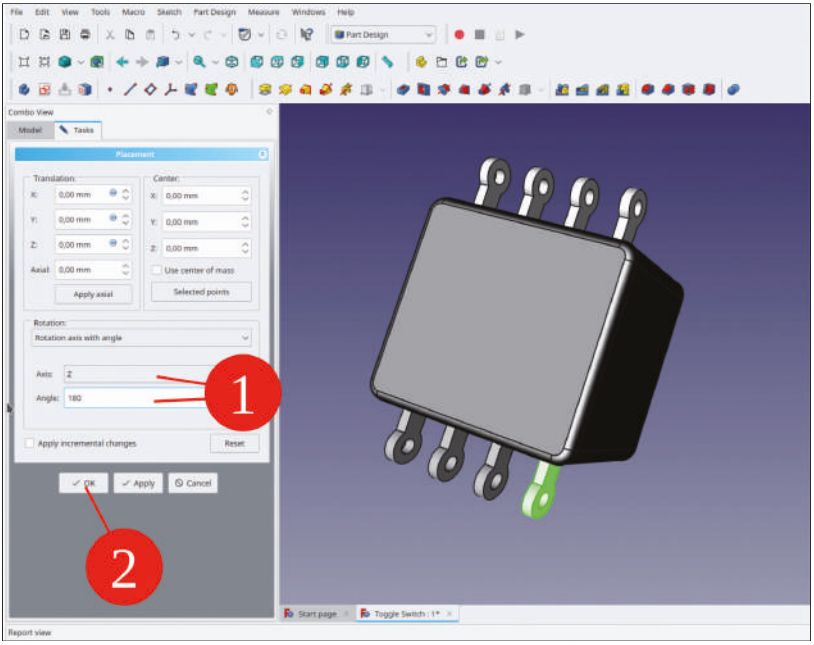


Figure E-30

In the tree view, mark all new contacts and set the appearance to 'Chrome' (Figure E-31). Close the task window.

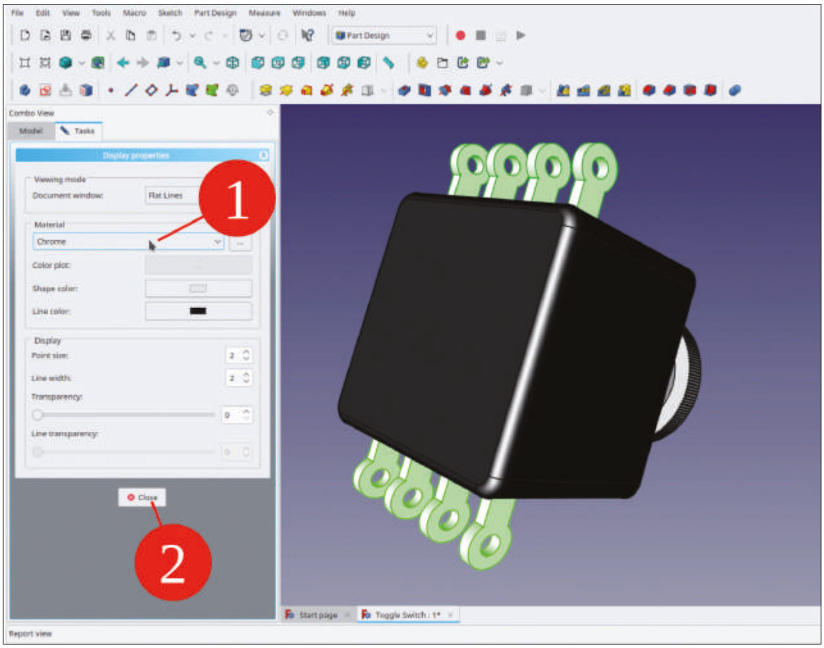


Figure E-31

As a last thing, the lever will be modeled. Create a new body object and rename it to 'Toggle Switch Lever'. Hide all other bodies while the lever is designed. Display the coordinate system of the lever body.

Mark the YZ plane and start the sketcher. Draw a circle that is centered around the origin, and constrain its diameter to 7.6 mm. Then, draw a second circle above the first one, which is centered on the Z axis. Constrain its diameter to 6.5 mm. In the 'Elements' list, mark both circles, right-click the selection and select 'Toggle construction line' from the context menu (Figure E-32).

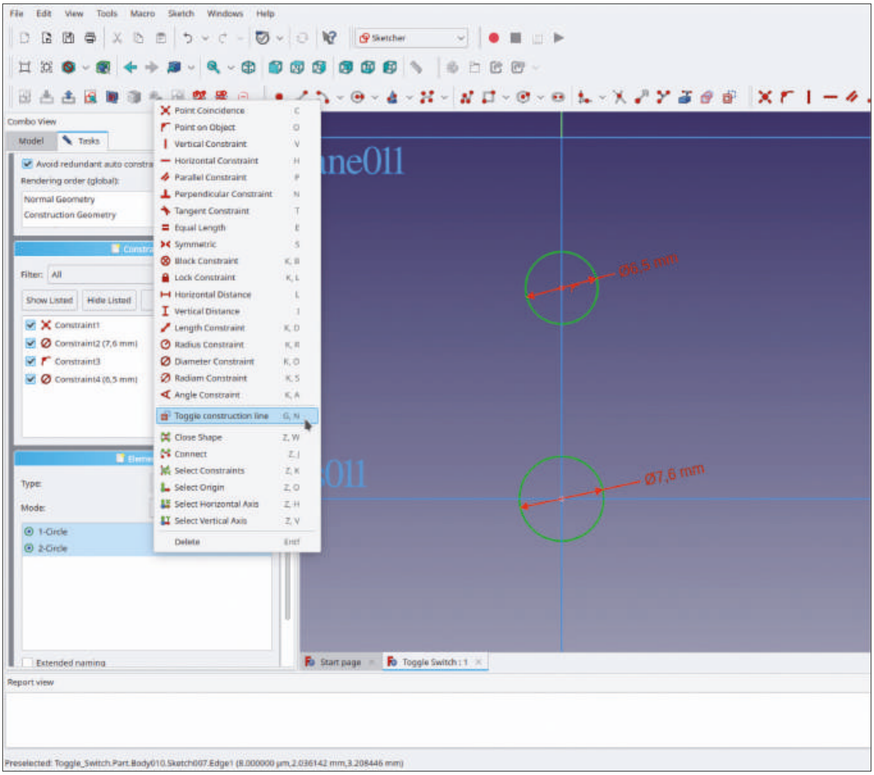


Figure E-32

Mark the center points of the two circles and constrain the vertical distance to 16.3 mm (Figure E-33).

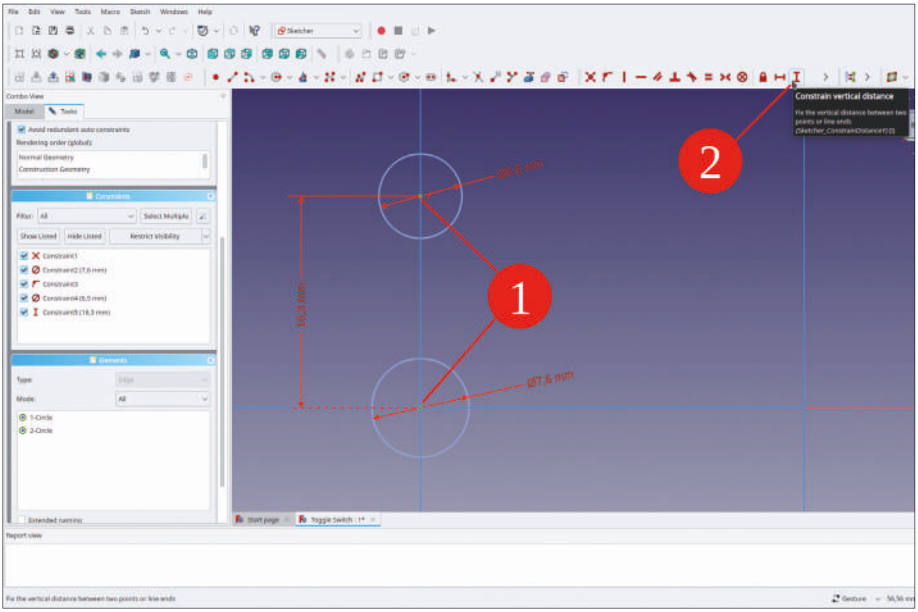


Figure E-33

For the lever, draw a line of approximate length and position. End the drawing command with a right click. Mark the upper endpoint of the line and the upper circle and click the 'Constrain tangent' tool button (Figure E-34).

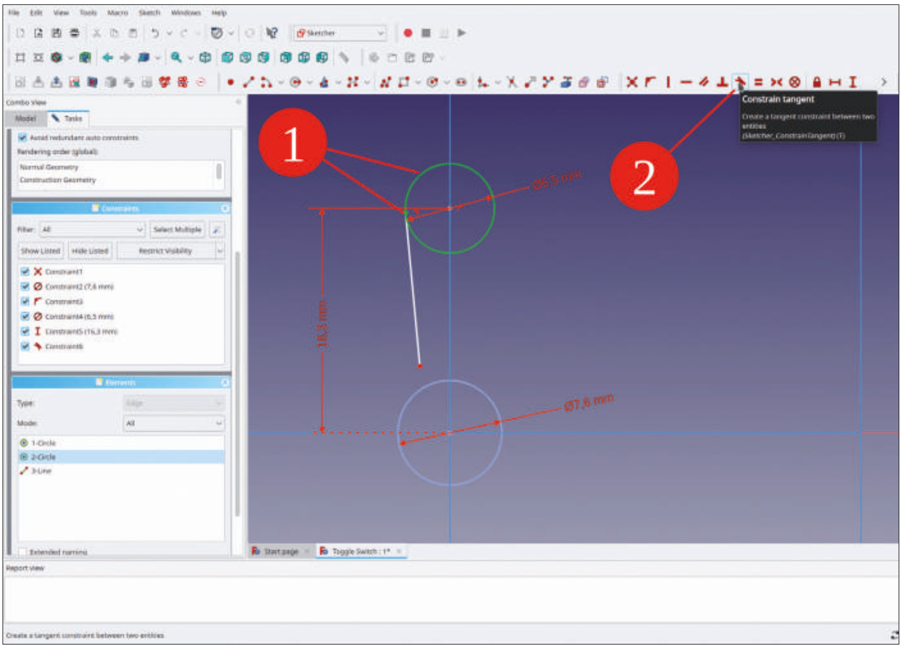


Figure E-34

Mark the lower endpoint of the line and the lower circle. Then, click the 'constrain point onto object' tool button (Figure E-35). Sometimes, the solver complains about too many selected objects. Then, some drawing element from a prior step was still marked. To resolve this, click the blue background to deselect all, and start over.

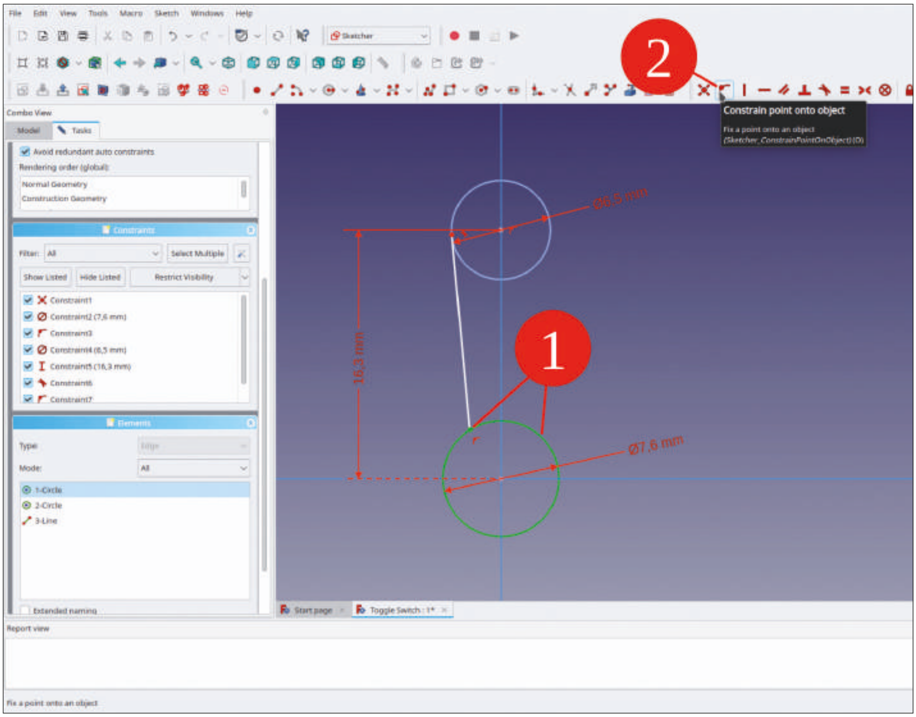


Figure E-35

Mark the line and the Z axis. Then, click the 'Constrain angle' tool button (Figure E-36). Set the angle to 7°. The line is already fully constrained.

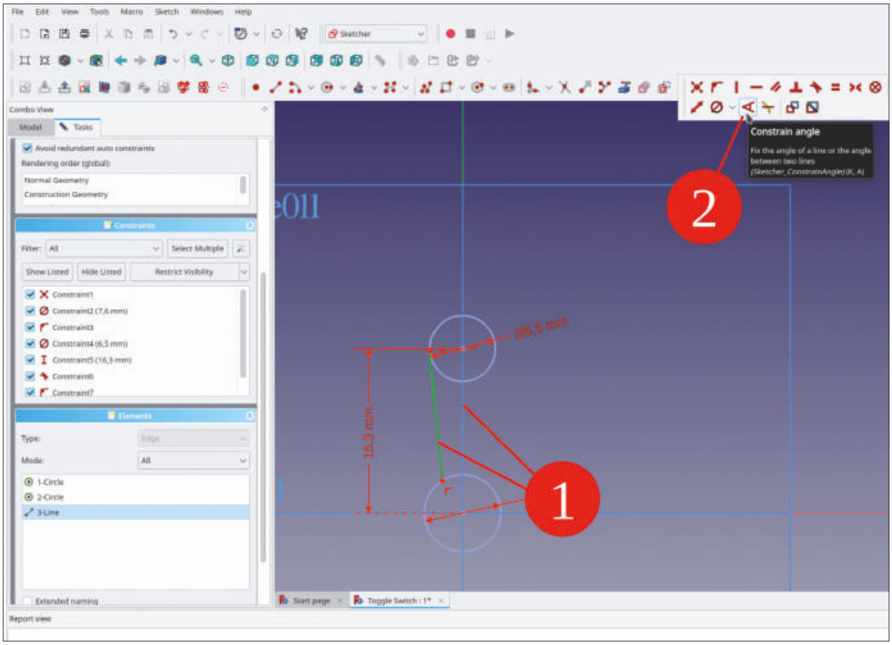


Figure E-36

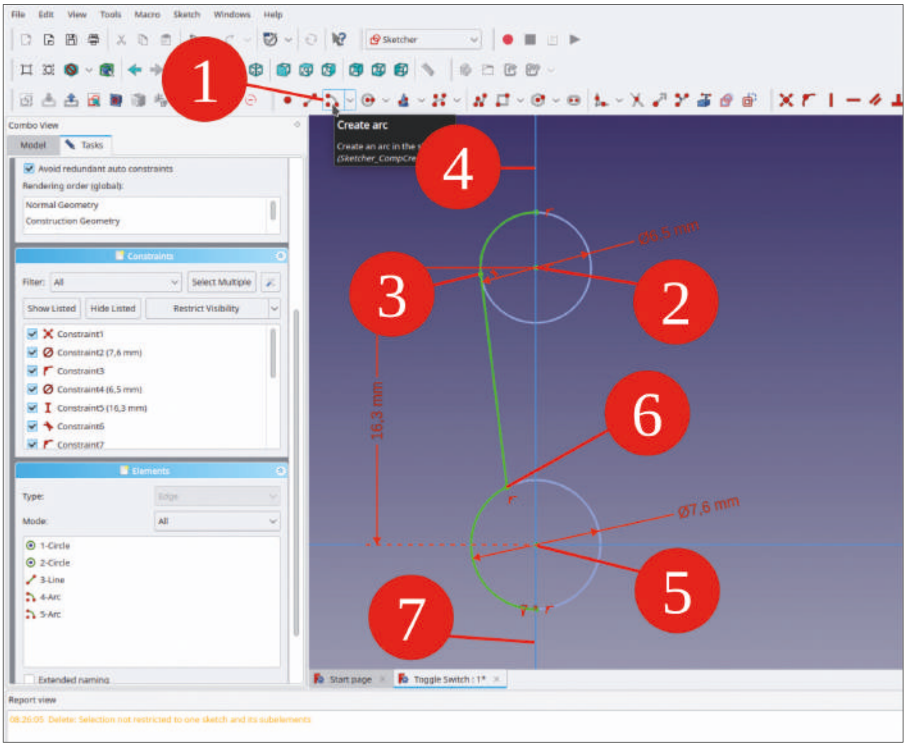


Figure E-37

Draw two arcs, as shown in Figure E-37. To achieve this, click first the origin of the corresponding circle, then lock one end of the arc to the endpoint of the line which is attached to that circle. Define the other endpoint of the arc by clicking on the Z axis with the crosshairs. It is difficult to see whether you are locked to the axis when the view is zoomed in (because of the blue axis representation). If you zoom out a bit, you can see the axis changing color to yellow when the cross hairs cursor is on it. If you successfully hit the two endpoints, the arc is shown bright green — fully constrained again.

The profile that you draw here will be used for a revolution. It needs to be closed, so draw a line on the Z axis, connecting the two endpoints (Figure E-38). Close the sketch.

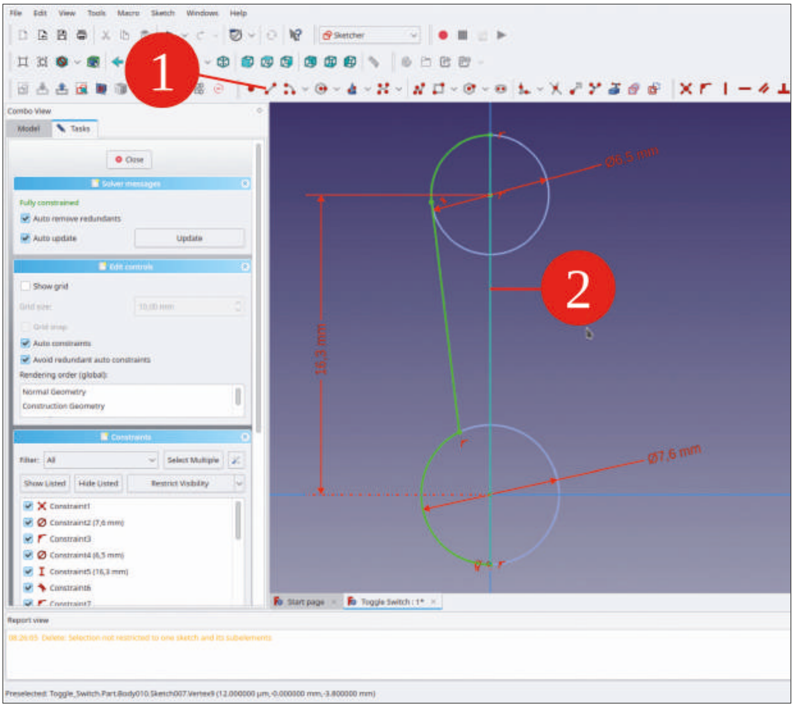


Figure E-38

In the tree view, mark the new sketch and click the 'Revolution' tool button. The defaults for axis and angle are already sufficient to generate the part (Figure E-39). Close the task window with the OK button.

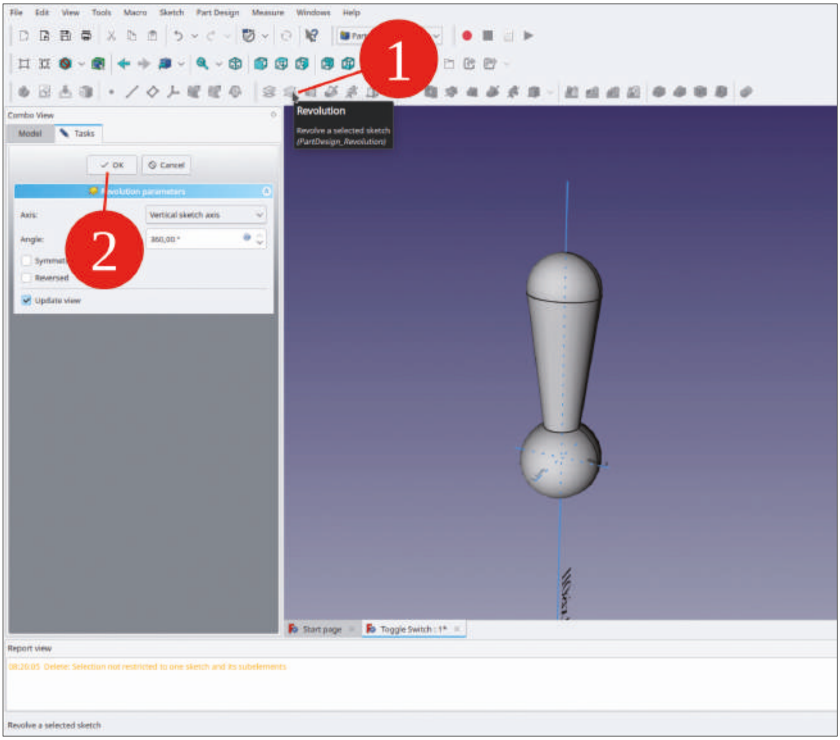


Figure E-39

In the tree view, hide the coordinate system of the lever body with the SPACE key. Set the appearance of the 'Toggle Switch Lever' body to the material 'Chrome'.

In the tree view, show all other bodies of the switch (mark, and SPACE key).

In the tree view, mark the 'Toggle Switch Lever' body. In the property list, click into the 'Placement' edit field and then on the ... button to show the placement task window. Set the Z translation to -0,5. For the rotation, select the X axis and enter a value of -22° for the angle (Figure E-40). You can actually 'toggle' the switch with the mouse wheel, while the cursor is inside the 'Angle' field. Virtual reality as close as it gets. Close the task window with the OK button.

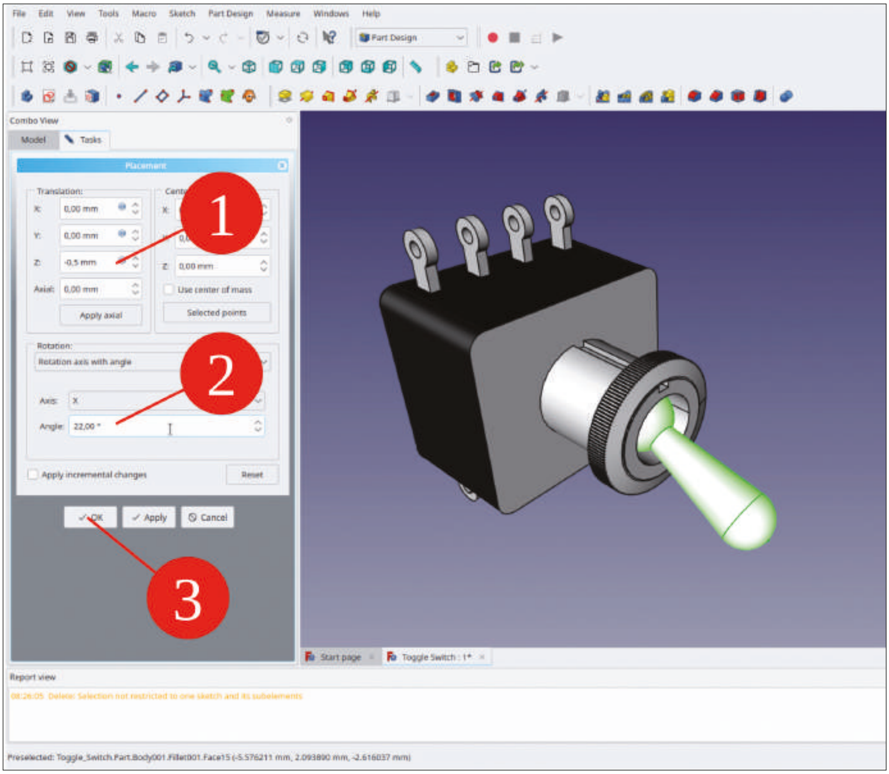


Figure E-40

The lower nut is still missing. This is the one that you want to tighten with a tool, not the knurled one (which saves your equipment many scratches and deformations). Create a new body object. If necessary, drag-and-drop it into the Std-Part-Container, and rename it to 'Toggle Switch Nut'.

Start the sketcher and, in the selection dialog, set the XY plane as the sketching plane.

Click on the 'Create regular polygon' tool button and draw a hexagon which is centered around the origin. With the first click, define the center, with the second, place a corner of the hexagon onto the Y axis (Figure E-41).

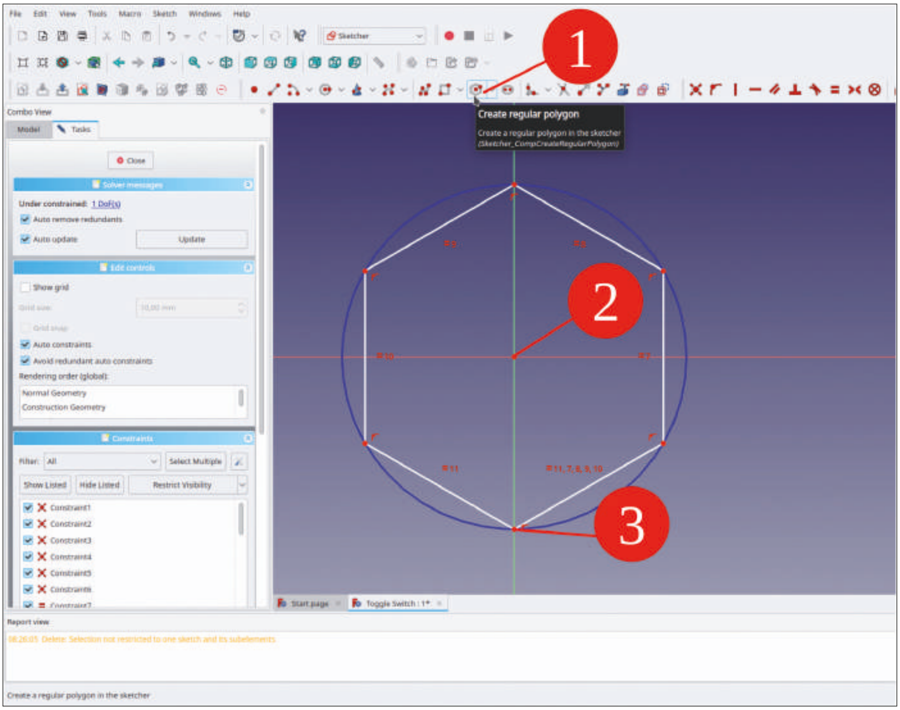


Figure E-41

Mark two corners of the hexagon which are on the same height (Y coordinate). Constrain the horizontal distance to a key of 16 mm (Figure E-42, steps 1 and 2).

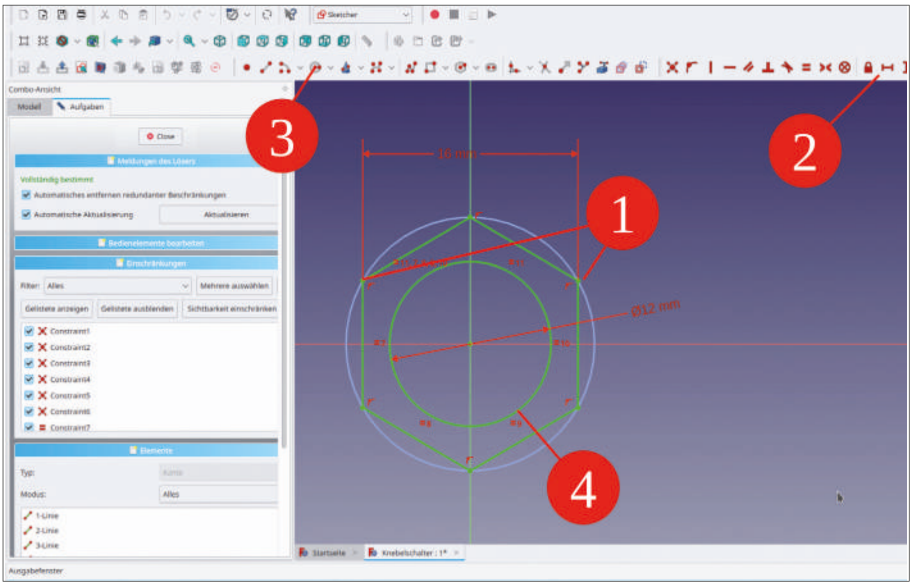


Figure E-42

Draw a circle which is centered around the origin and constrain its diameter to 12 mm (Figure E-42, steps 3 and 4). Close the sketch.

In the tree view, mark the new sketch and click the 'Pad' tool button. For the length, enter a value of 1.5 mm. Because the nut extends from the rear of the front panel towards the back, check the 'Reversed' checkbox (Figure E-43).

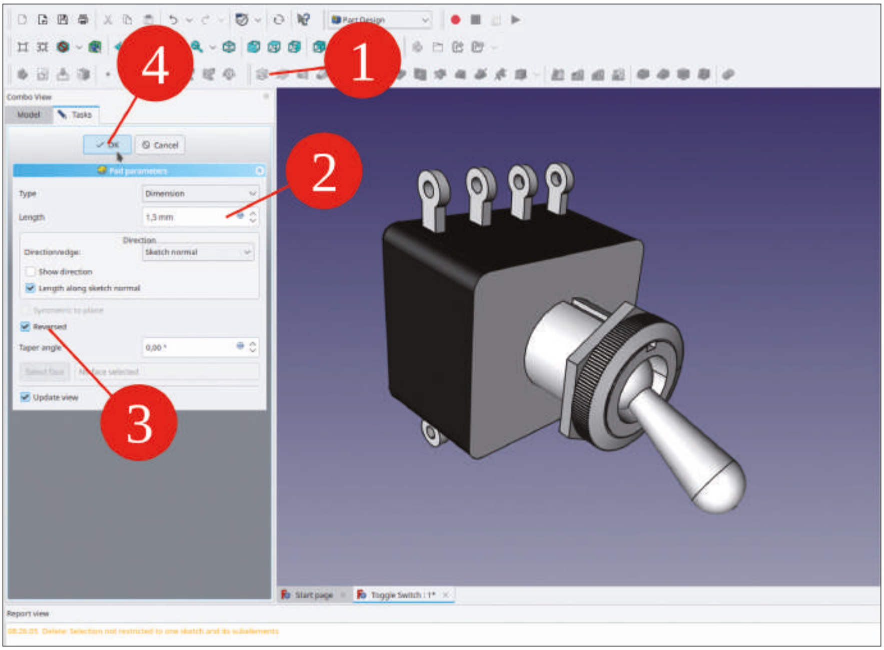


Figure E-43

In the tree view, mark the body of the new nut and set the appearance to 'Chrome'.

The nut position could be set to account for a standard front panel thickness. In the tree view, click the nut object. In the property list, click into the 'placement' edit field and then on the ... button to show the task window. Set the Z translation to −2 mm (Figure E-44). Close the task window and save your work.

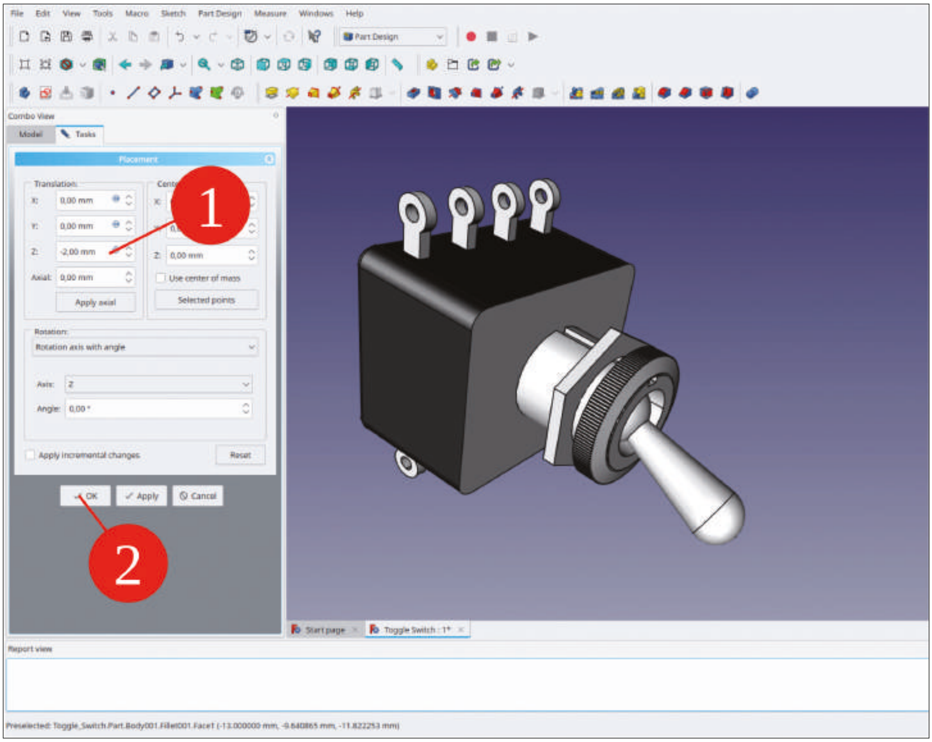


Figure E-44

Appendix F • The IEC Power Inlet

First, run the standard procedure: Create a new file and save it as 'IEC Inlet'. Then, create a Std-Part-Container, and rename this to 'IEC Inlet Complete'. Within this container, create a new body object and rename that to 'IEC Inlet Casing'. Display its coordinate system (mark it and press the SPACE key).

Start the sketcher. In the selection dialog, select the XY plane as the sketching plane.

Draw a rectangle that is centered around the origin. Constrain the width of the rectangle to 30.2 mm, and the height to 22.5 mm (Figure F-1). For a better fit, some extra tolerance has already been added to the measured dimensions.

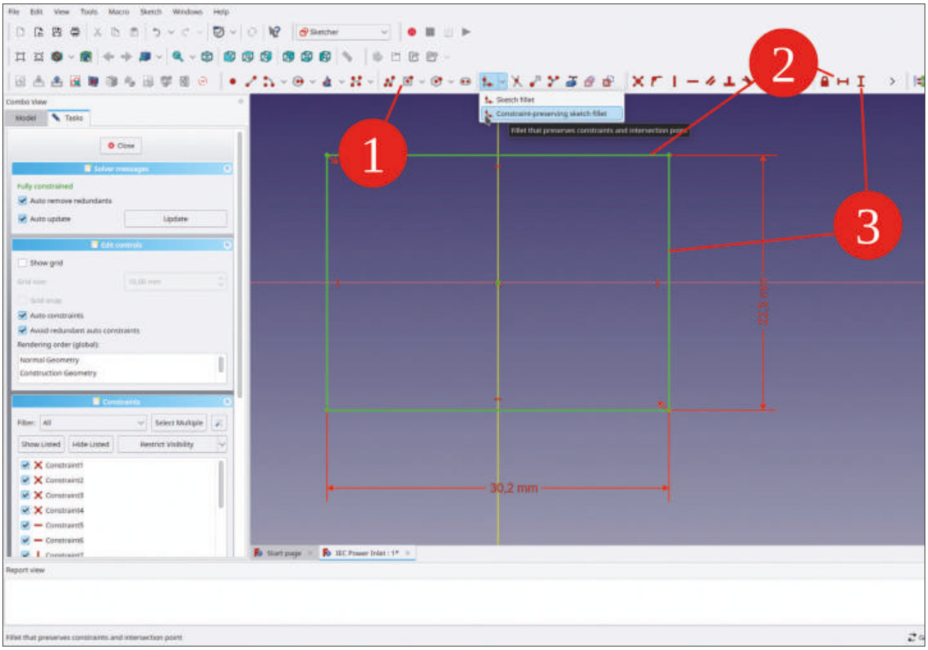


Figure F-1

Click on the 'Constraint preserving sketch fillet' tool button (Figure F-2). Then click two adjacent lines of the rectangle to create the round edge. Repeat this for each corner.

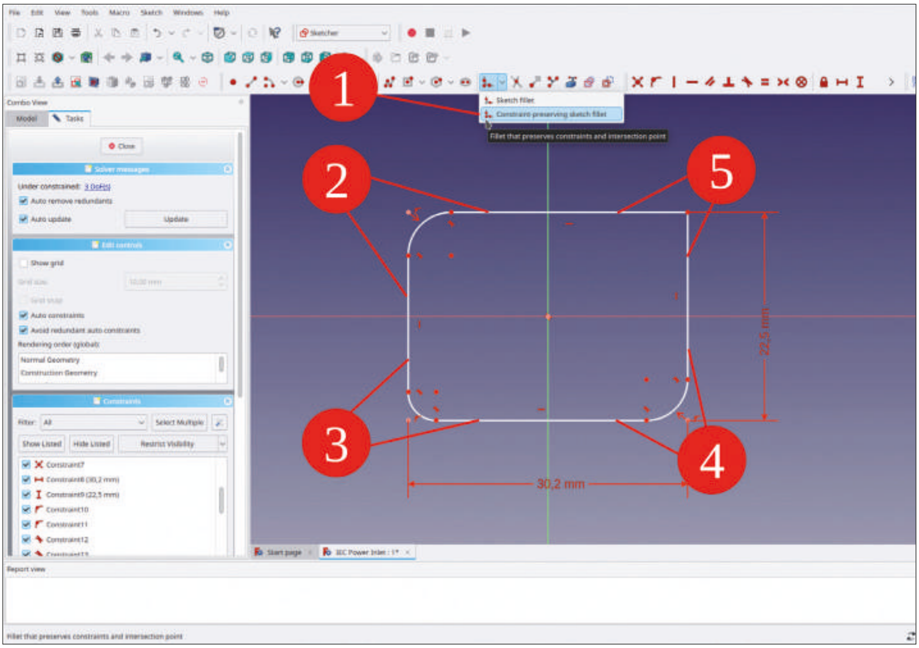


Figure F-2

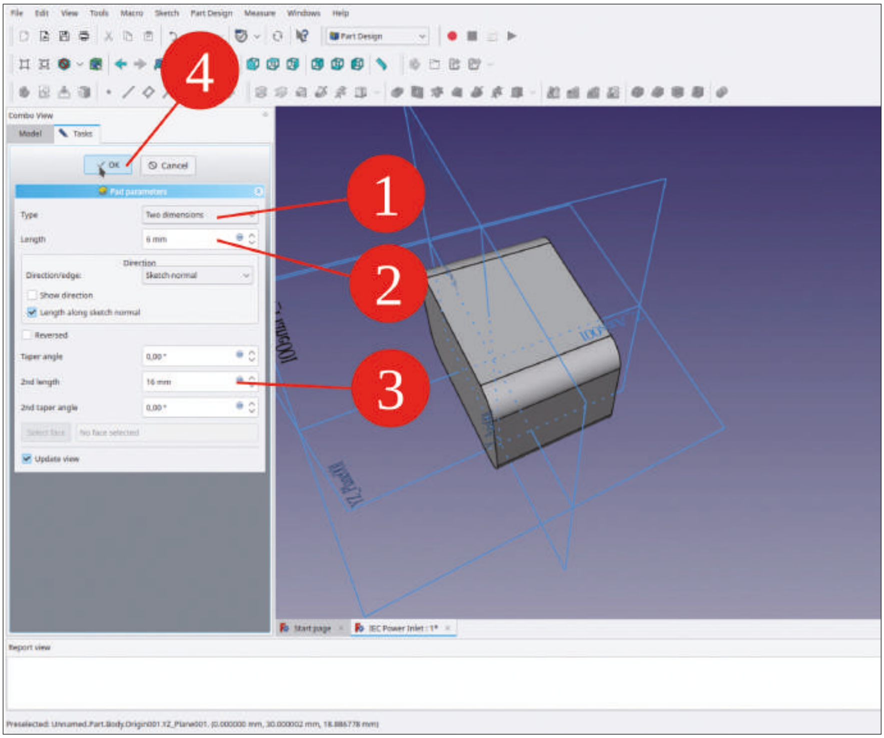


Figure F-3

In the 'Elements' list, mark all 'Arc' objects, right-click the selection and select 'Constrain radius' from the context menu. Enter a value of 4 mm. Close the sketch. If you would have selected 'Sketch fillet' instead of 'Constraint preserving sketch fillet', the change of the radius would have distorted the sketch.

The new sketch describes the contour of the casing itself. It will protrude to the front and to the back. Mark the sketch and click the 'Pad' tool button. In the task window, select for the type 'Two dimensions'. Set the length to 6 mm (protruding to the front), and the '2nd length' to 16 mm (protruding to the rear, Figure F-3). Close the task window with the OK button.

Now, generate the flange for the inlet. In the 3D view, mark the XY plane and start the sketcher. Close and reopen the sketch to display the already-present 3D geometry. From the main menu, select 'View | Orthographic view' and 'Sketch | View section'.

Click on the 'External geometry' tool button and then on each of the four arcs. These will be highlighted in violet, as construction elements (Figure F-4).

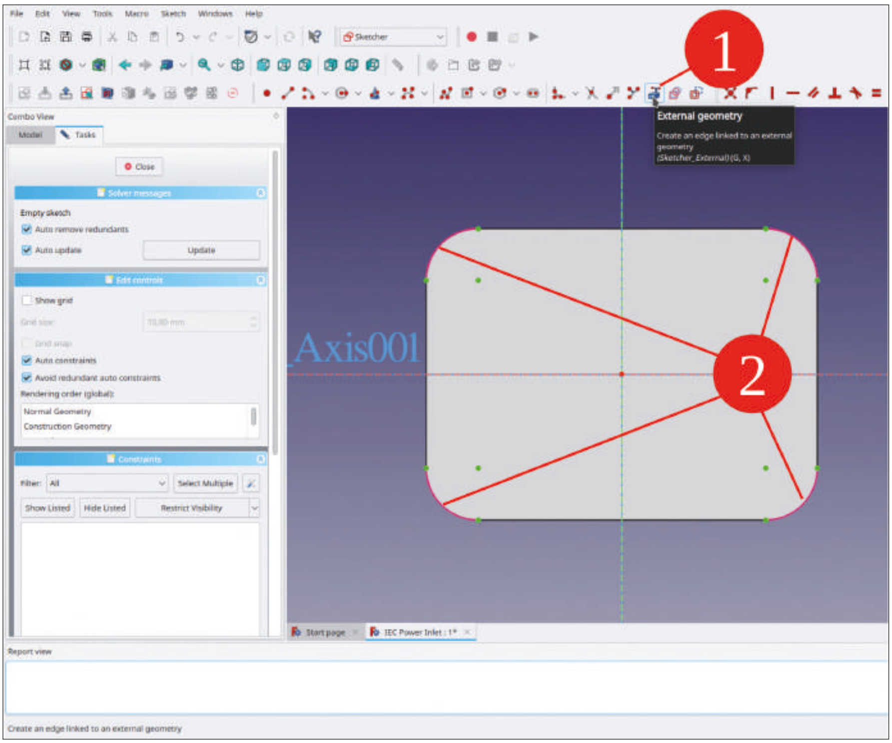


Figure F-4

For the mounting holes, draw two circles that are centered on the X axis. Mark the two circles and click the 'Constrain equal' tool button. In the 'Elements' view, right-click the two circles and select 'Diameter constraint' from the context menu. Enter a value of 3.2 mm to accommodate M3 (3 mm metric) screws later.

Mark both circle centers, then the Y axis. Click on the 'Constrain symmetrical' tool button.

Mark both circle centers again and constrain the horizontal distance to 40 mm (Figure F-5).

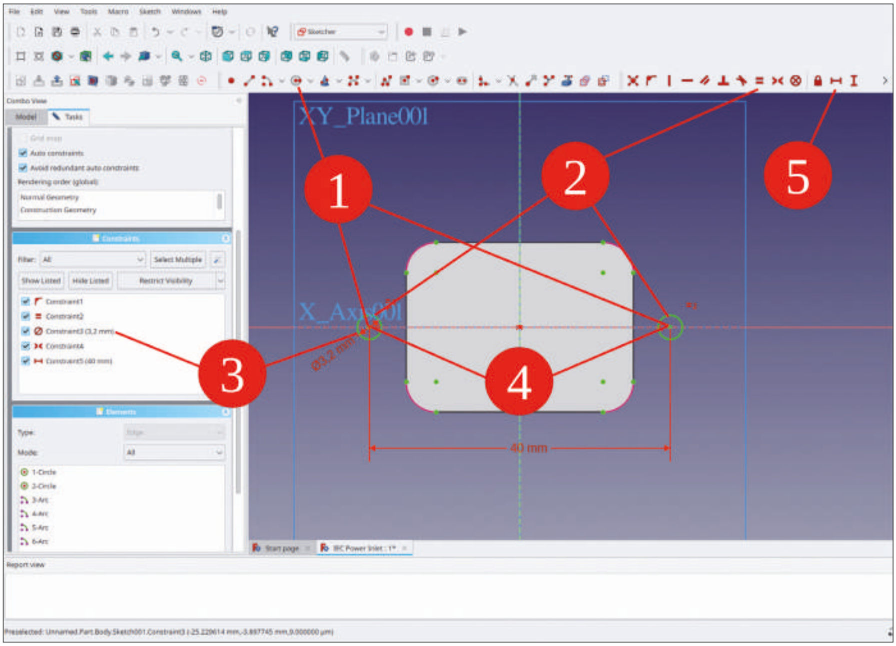


Figure F-5

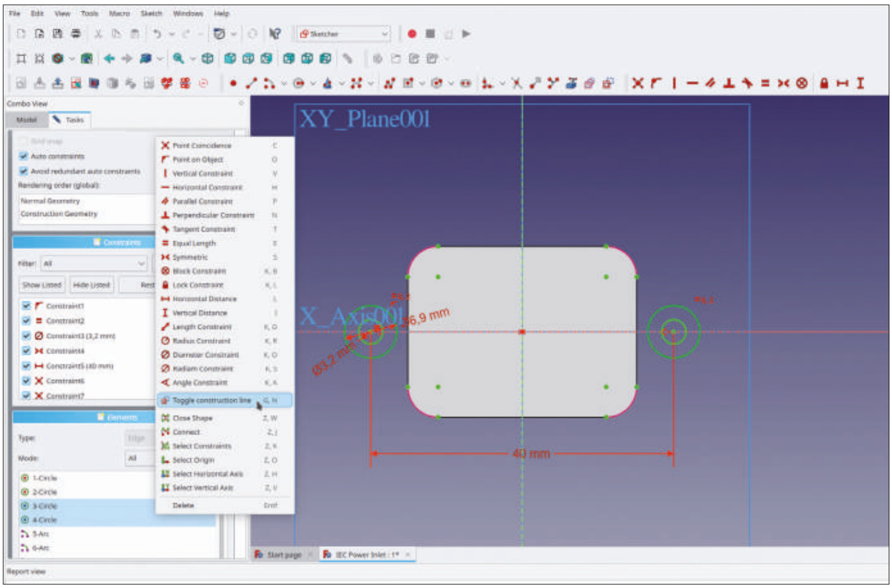


Figure F-6

Around each circle, draw a concentric, larger one. Mark both of the new circles and click the 'Constrain equal' tool button. Then constrain the diameter of one of the new circles to 6.9 mm.

In the 'Elements' list, mark the two larger circles. Right-click the selection and select 'Toggle construction line' from the context menu (Figure F-6).

Draw a line of arbitrary position and length close to one of the circles. End the drawing command with a right click. Mark one endpoint of the line and the closest (violet) arc. Click the 'Constrain tangent' tool button (Figure F-7).

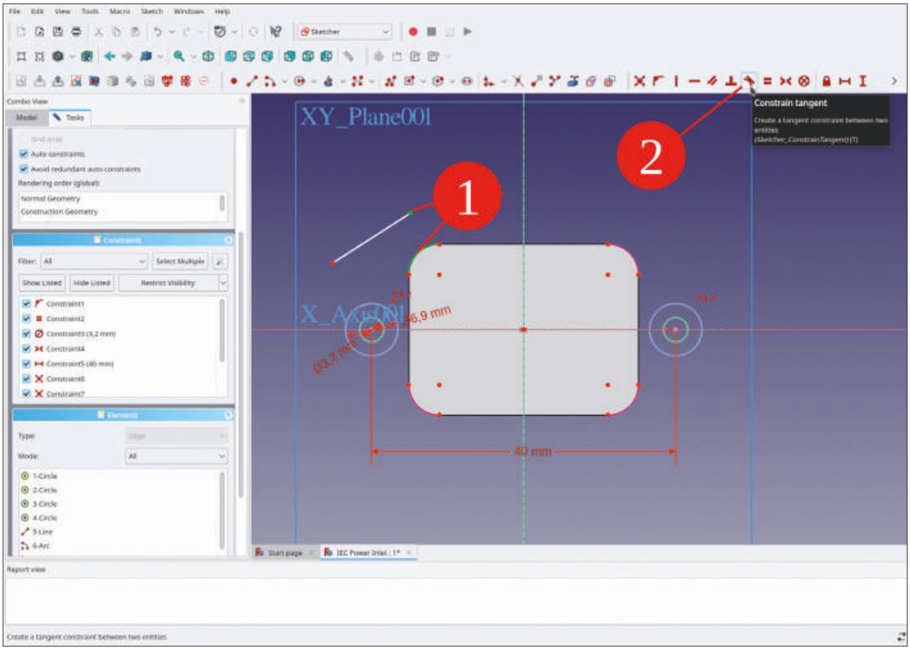


Figure F-7

In the same way, mark the other endpoint of the line and the larger circle close to it. Click the 'Constrain tangent' tool button again (Figure F-8).

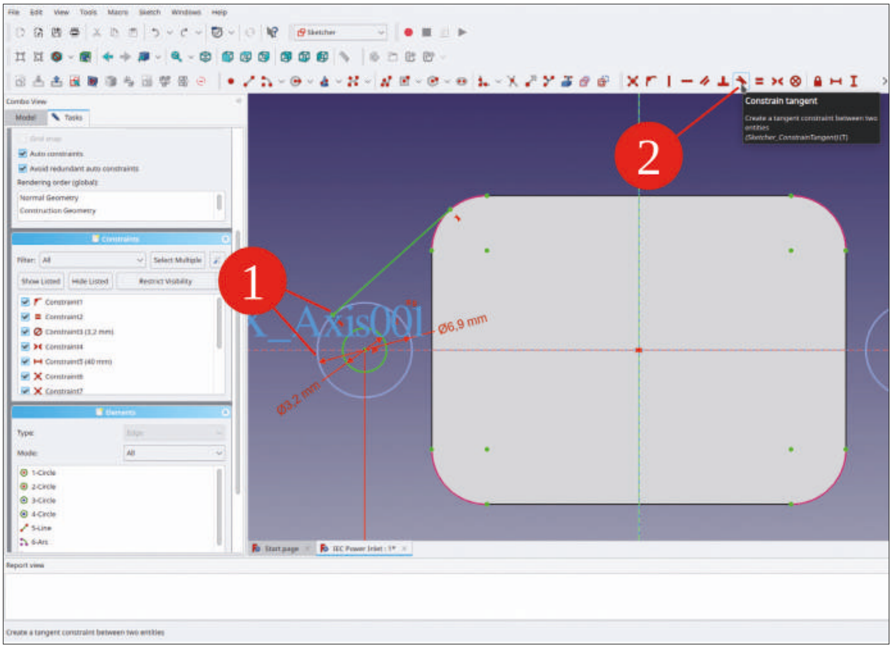


Figure F-8

Repeat the procedure to generate lines for all four arcs (Figure F-9).

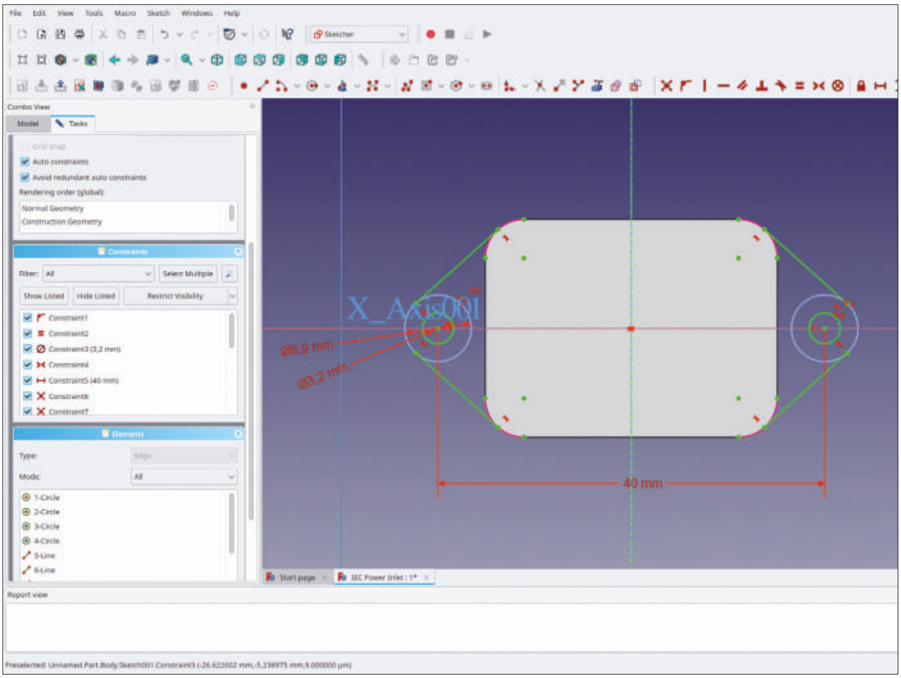


Figure F-9

Click on the 'Create arc' tool button and draw all arcs which are shown in Figure F-10 in bright green.

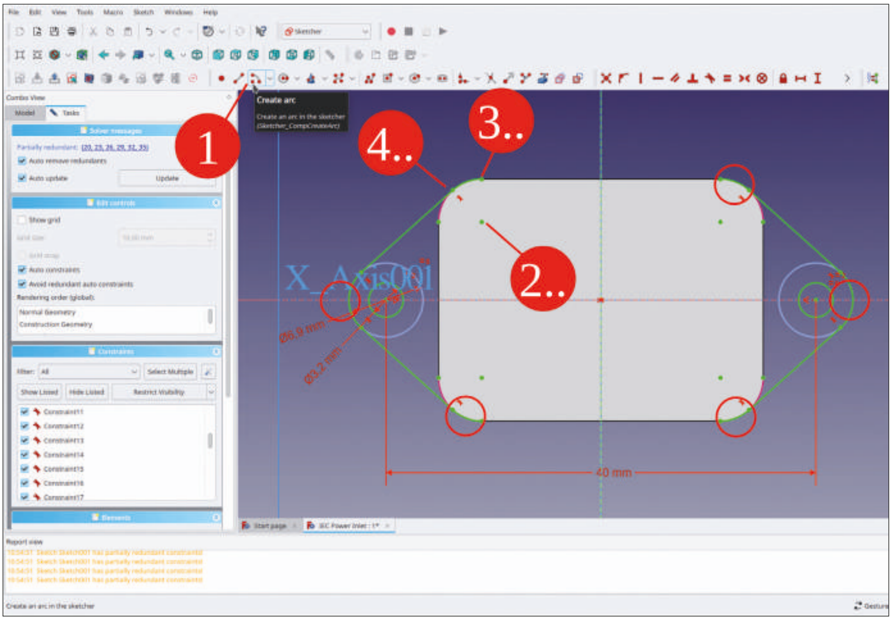


Figure F-10

Close the flange contour by adding two horizontal lines (Figure F-11). Close the sketch.

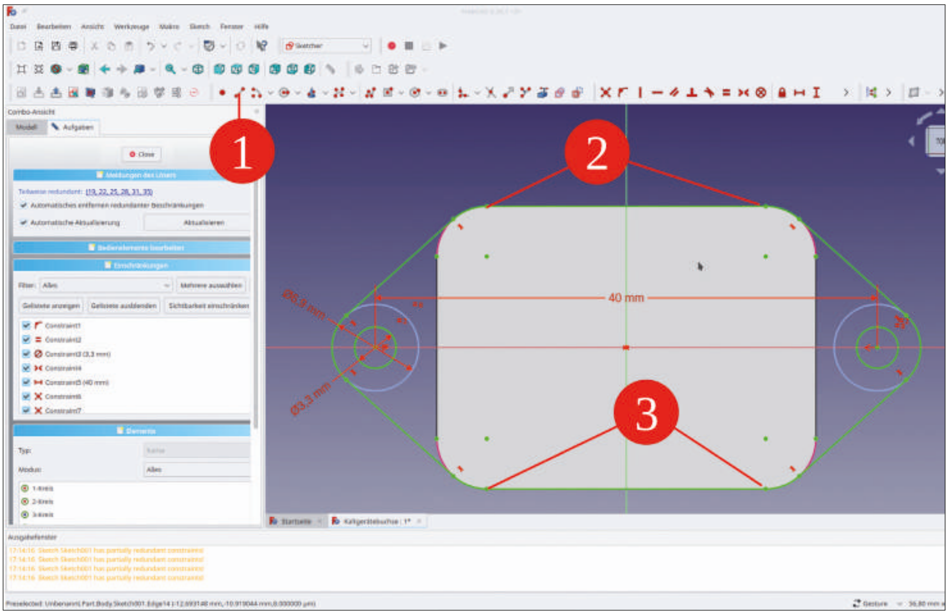


Figure F-11

In the tree view, mark the new sketch. Click on the 'Pad' tool button. In the task window, enter a value of 3 mm for the length (Figure F-12).

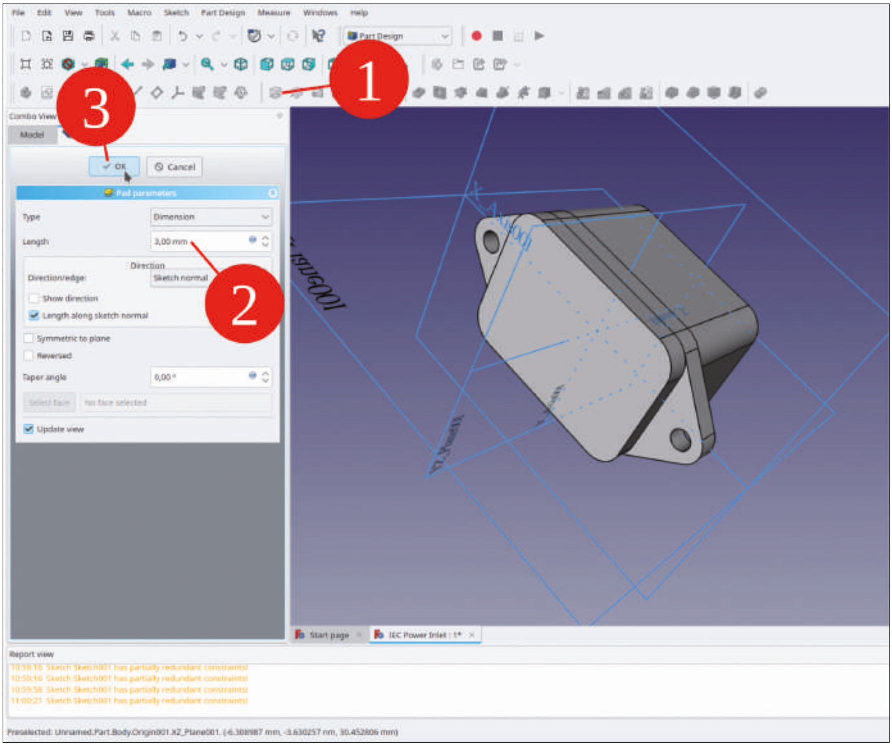


Figure F-12

Define the opening in the socket: Mark the XY plane and start the sketcher. Close and re-open the sketch, to display the already-present 3D geometry. From the main menu, select 'View | Orthographic view' and 'Sketch | View section'.

Draw a rectangle that is centered around the origin. Constrain the width to 24.25 mm, the height to 16.8 mm.

As with the outer contour of the casing, click the 'Constraint preserving sketch fillet' tool button and apply the fillet to all four corners of the new rectangle. In the 'Elements' view, mark all arc objects, right-click the selection and select 'Constrain radius' from the context menu. Set the radius to 1 mm (Figure F-13).

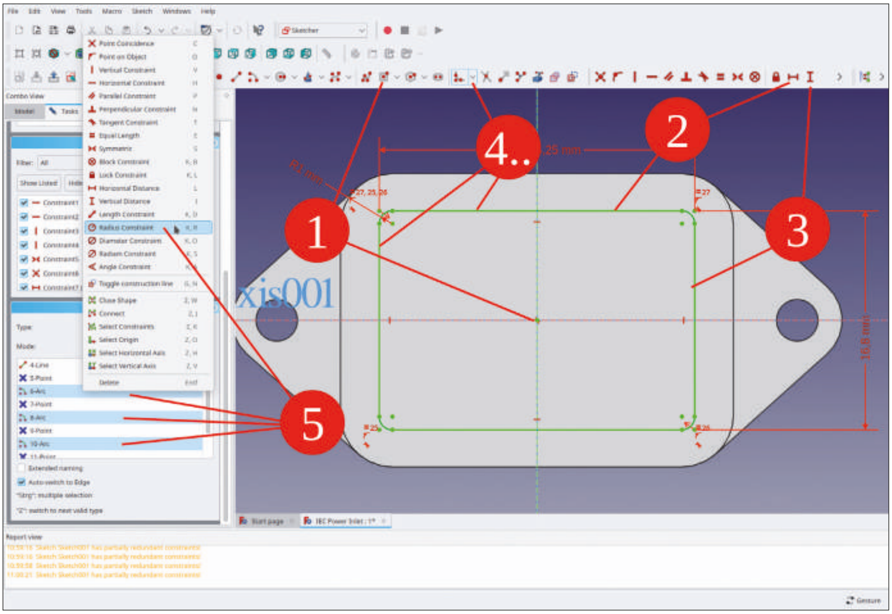


Figure F13

Close the sketch (top).

In the tree view, mark the new sketch and click the 'Pocket' tool button. In the task window, for the type, select 'Through all' and check the 'Reversed' checkbox (Figure F-14). Close the task window with the OK button.

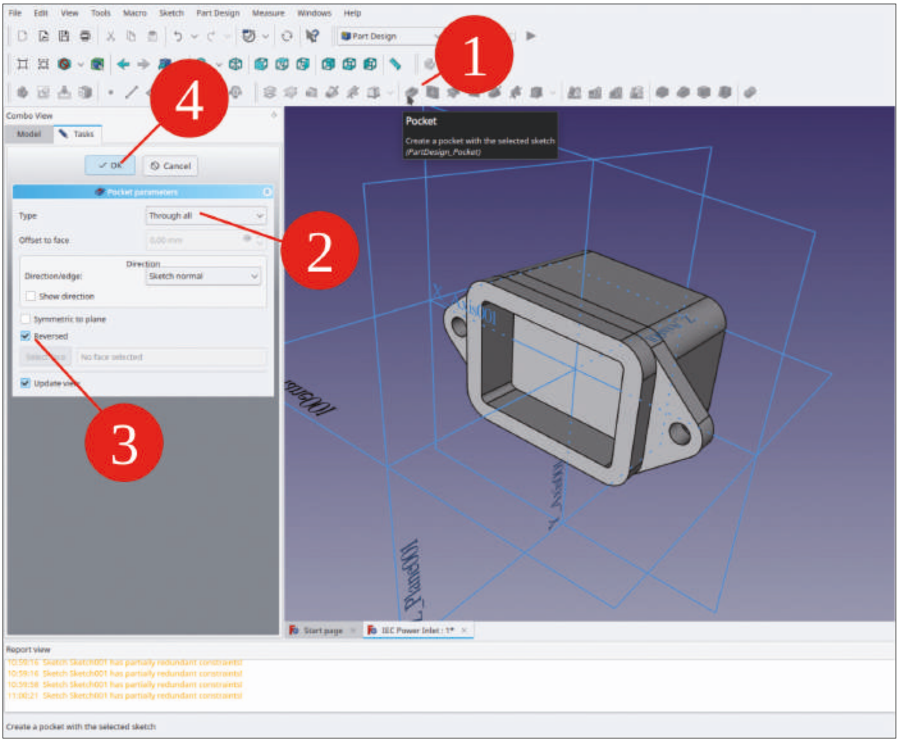


Figure F-14

The rear end of the opening has a slightly different profile. Again, in the 3D view, mark the XY plane and start the sketcher. Close and reopen the sketch, to show the already-present 3D geometry. From the main menu, select 'View | Orthographic view' and 'Sketch | View section'.

Click the 'External geometry' tool button. Mark all parts of the inner contour, except for the two upper arcs (Figure F-15). For the coded corners of the inlet, draw two inclined lines (Figure F-16). Mark one line and the Y axis. Click the 'Constrain angle' tool button and enter a value of 45°. Repeat the procedure for the other line (Figure F-17).

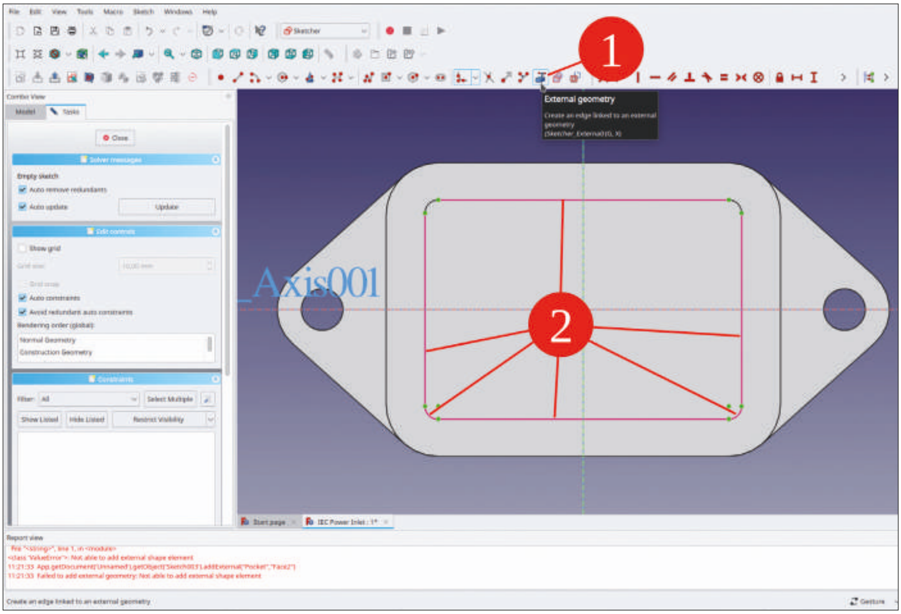


Figure F-15

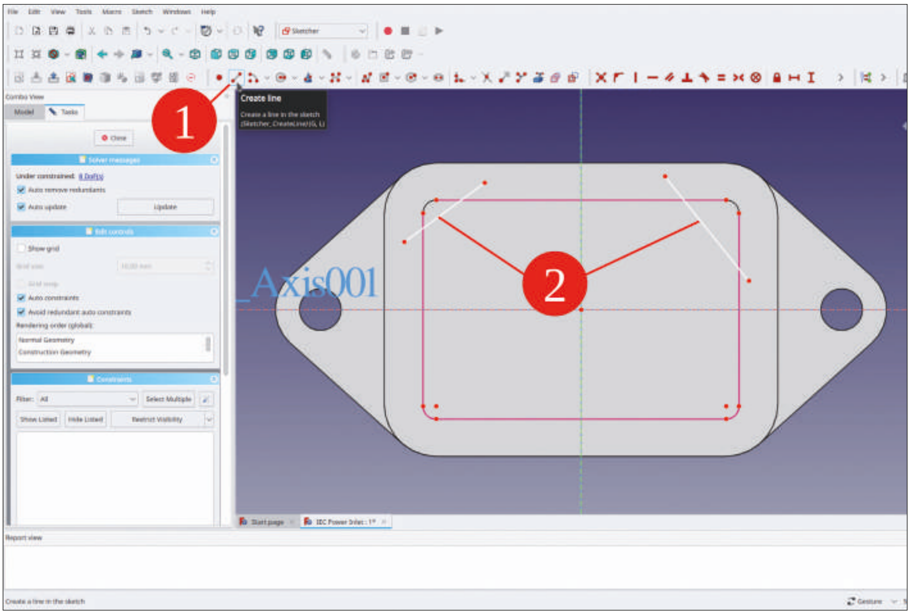


Figure F-16

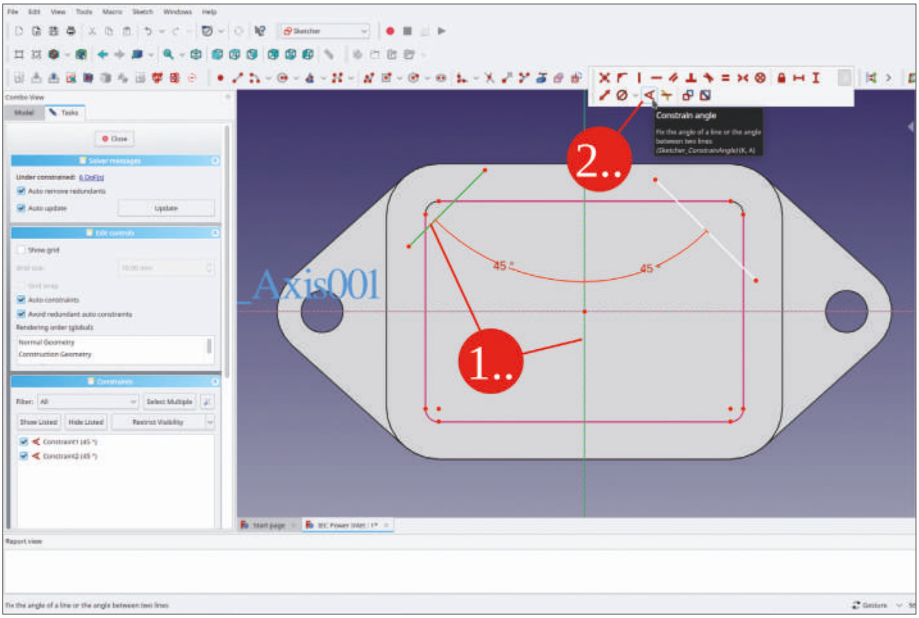


Figure F-17

Trim all parts of the two lines which are located outside of the inner contour (Figure F-18).

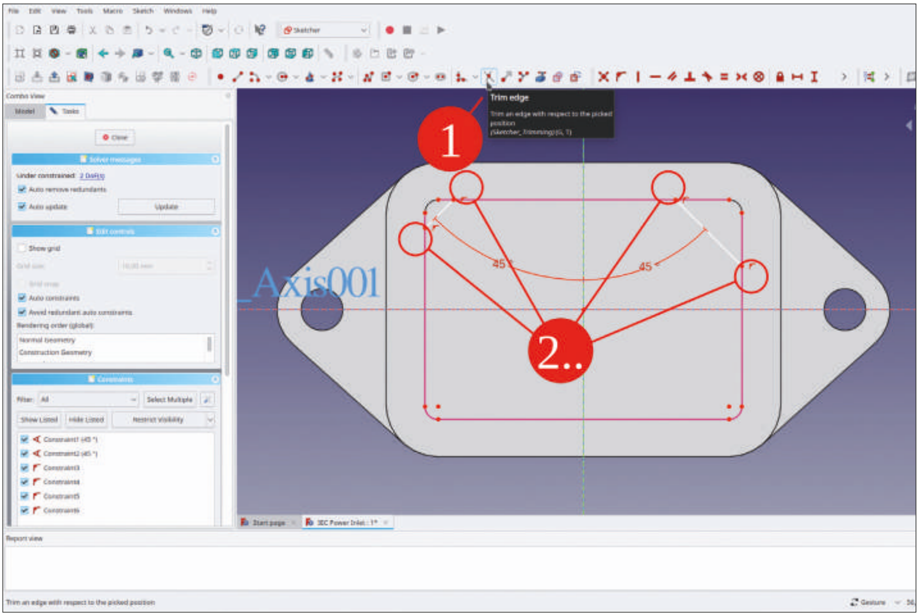


Figure F-18

On each side, mark the bottom end point of the line and the endpoint of the lowermost line of the contour, and constrain the vertical distance to 12.7 mm (Figure F-19).

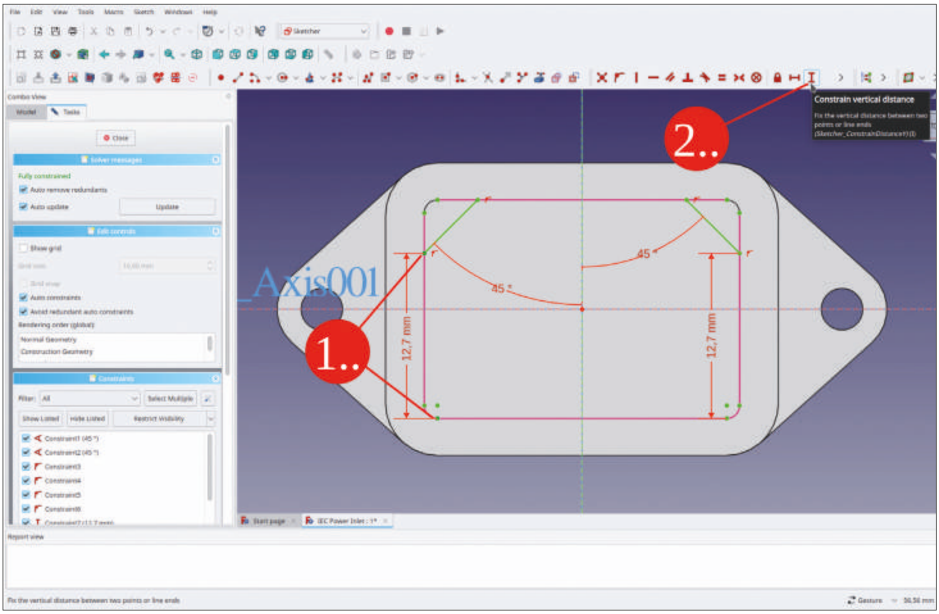


Figure F-19

Add the missing lines to the bottom profile, using the line command, as shown in Figure F-20. Take care to target the endpoints properly – only then, the sketch stays fully constrained.

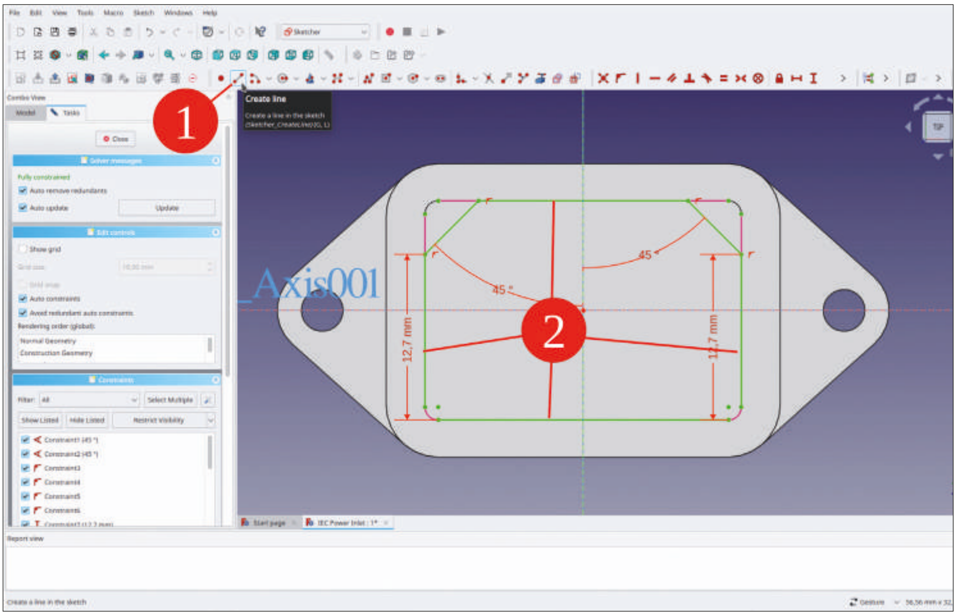


Figure F-20

Click on the 'Create arc' tool button and complete the profile by adding the two lower arcs, as shown in Figure F-21. Again, it is important to aim with great care with the crosshairs, to keep the sketch fully constrained, or at least leave it with only partially redundant constraints.

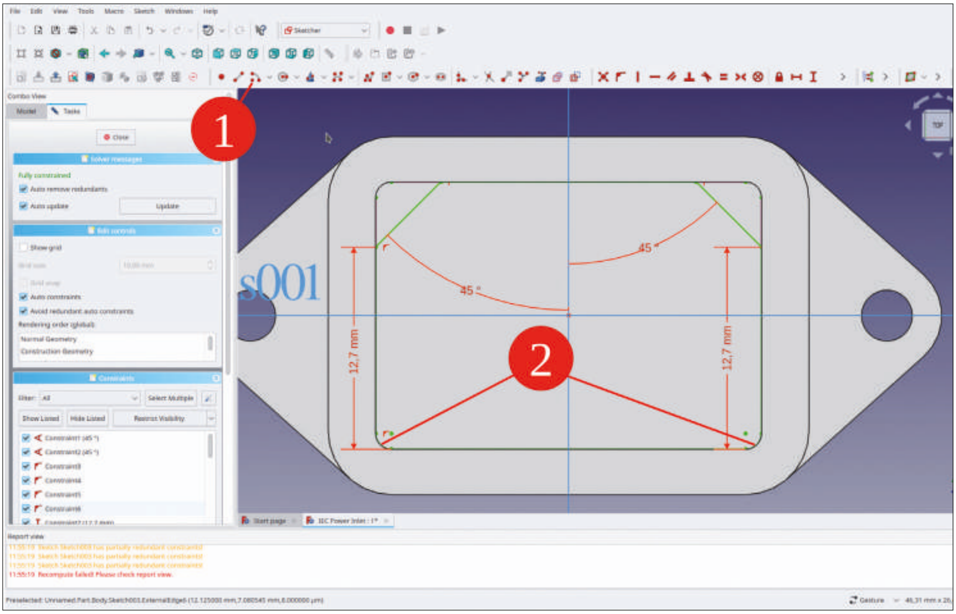


Figure F-21

Close the sketch. In the tree view, make sure that the new sketch is marked and click the 'Pocket' tool button. Set the length to 10 mm and close the task window with the OK button (Figure F-22).

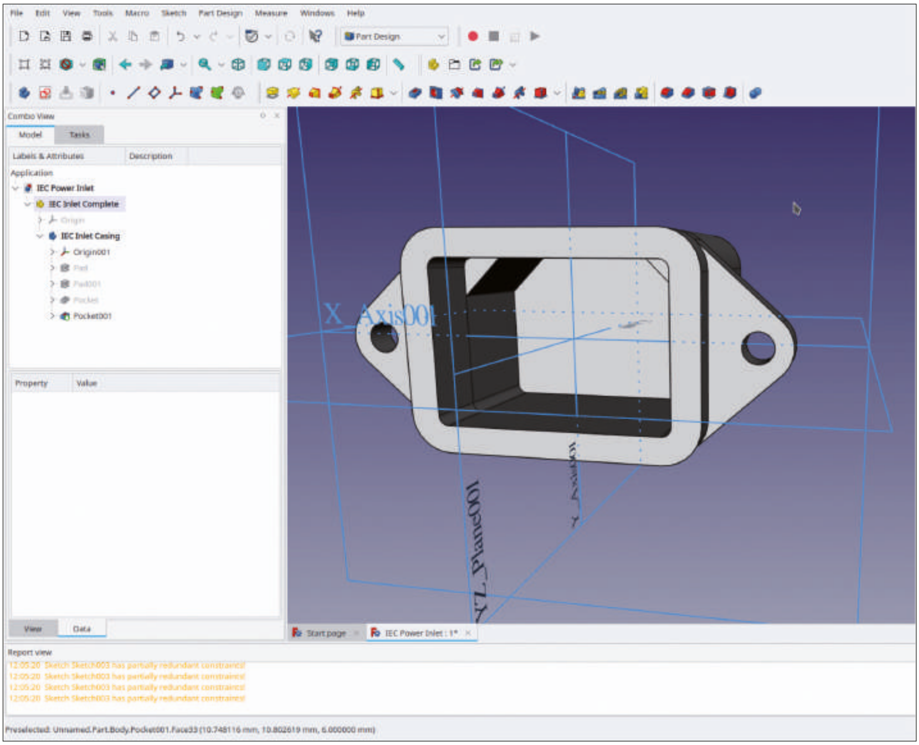


Figure F-22

In the 3D view, mark all edges which appear green in Figure F-23. Click on the 'Fillet' tool button. In the task window, enter a value of 1.25 mm for the fillet radius. Close the task window with the OK button.

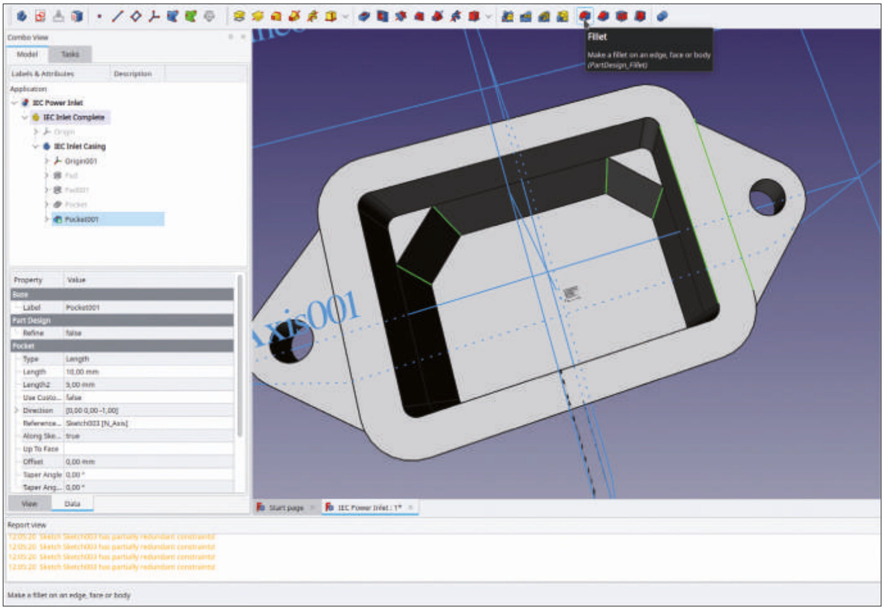


Figure F-23

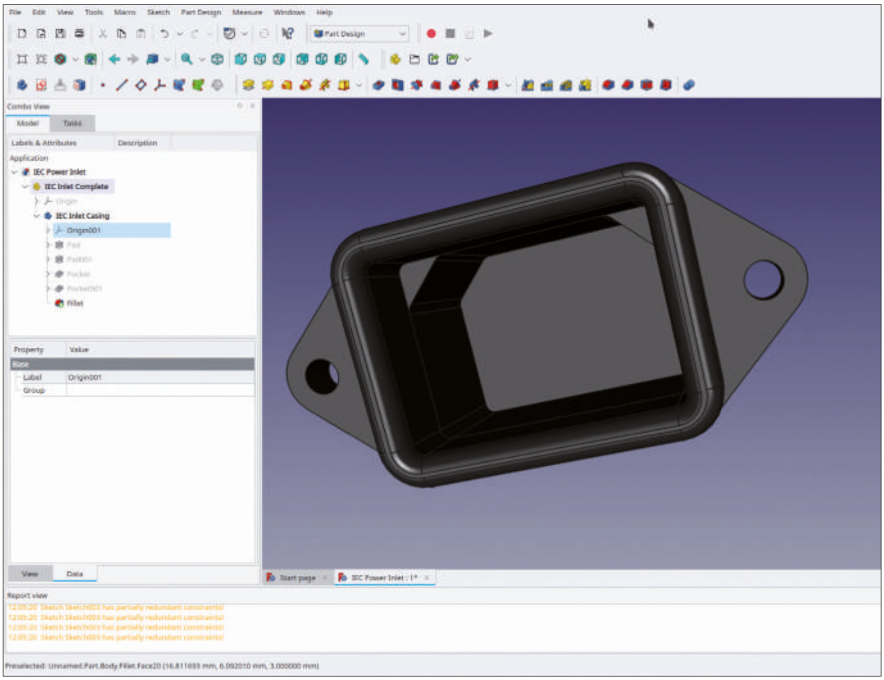


Figure F-24

In the tree view, right-click the body of the casing, and select 'Appearance…' from the context menu. For the material, select 'Shiny Plastic'. This presets the color to black. Sometimes it is better to select dark gray, which gives better contrast (F-24). Close the task window.

Now, the contacts need to be modeled. In the tree view, hide the 'IEC Inlet Casing' body. Create a new body and drag-and-drop it into the Std-Part-Container, if necessary. Rename the new body to 'IEC Inlet Contact N'.

Start the sketcher and select the XY plane as the sketching plane.

Draw a rectangle that is centered around the origin. Constrain its width to 2 mm, its height to 4 mm (Figure F-25). Close the sketch.

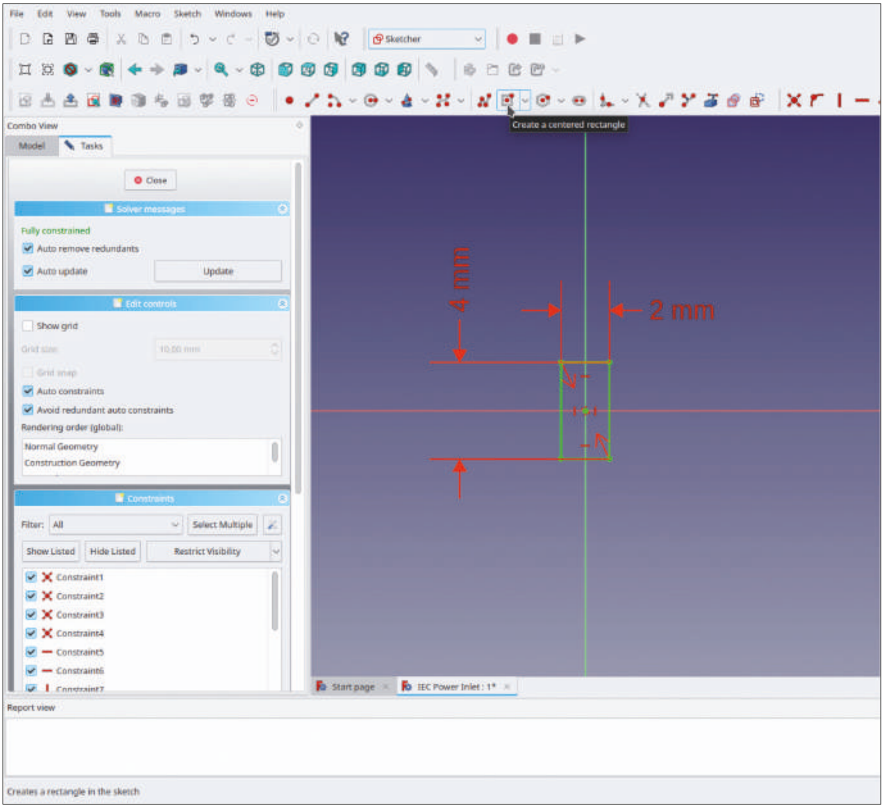


Figure F-25

In the tree view, mark the new sketch and click the 'Pad' tool button. Set the length of the pad to 23 mm and check the 'Reversed' checkbox (Figure F-26). Close the task window.

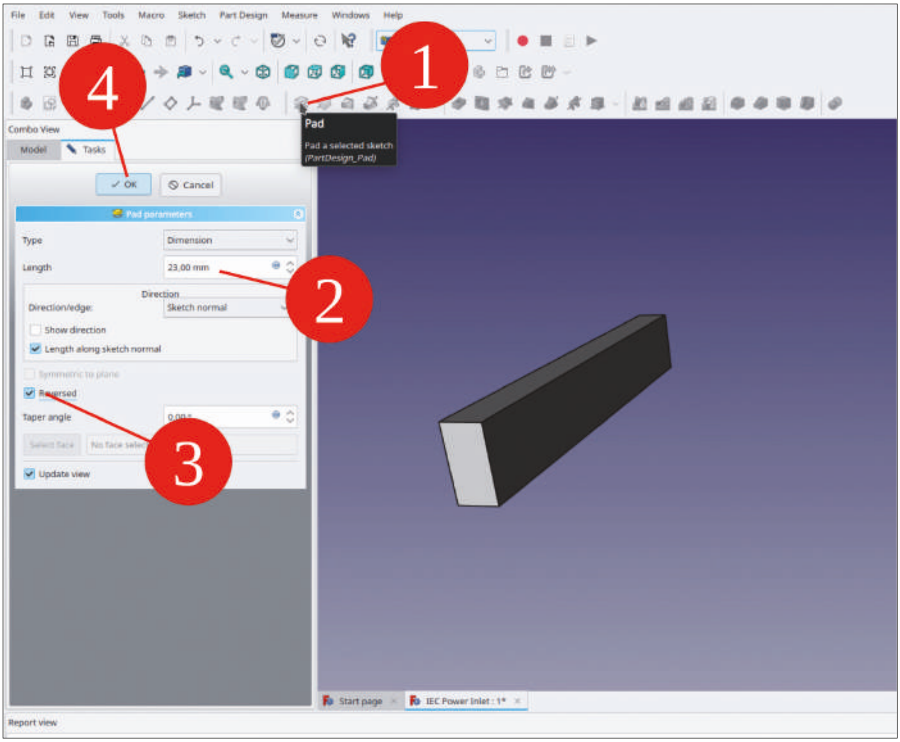


Figure F-26

For the tapers at the contact front, in the 3D view, mark the front face of the contact. Then, click the 'Chamfer' tool button. In the task window, select the type 'Two distances', set the size to 0.5 mm and the size 2 to 3 mm (Figure F-27). Close the task window with the OK button.

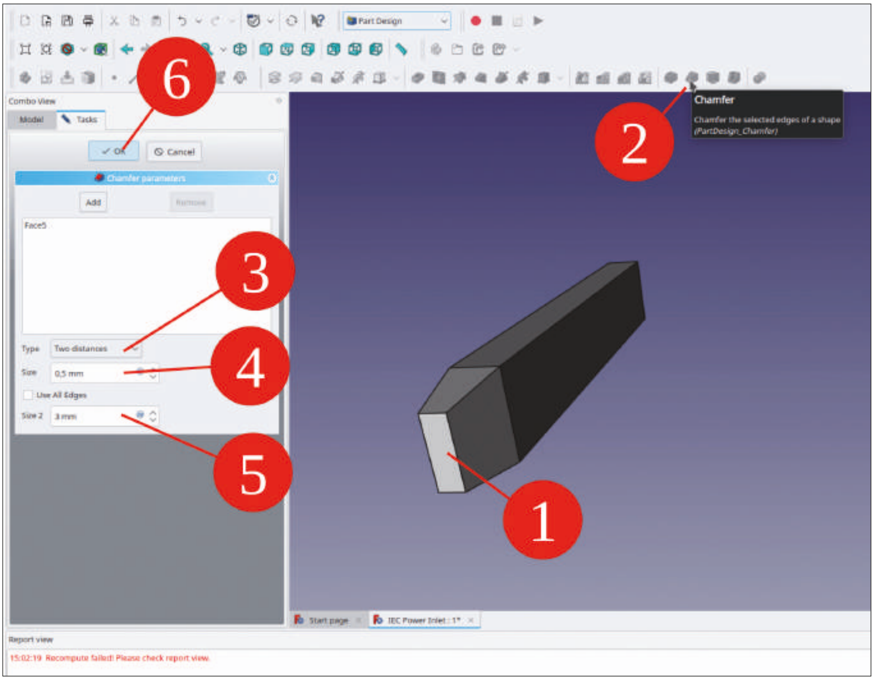


Figure F-27

In the 3D view, turn the contact around and mark the two shorter rear edges, as shown in Figure F-28.

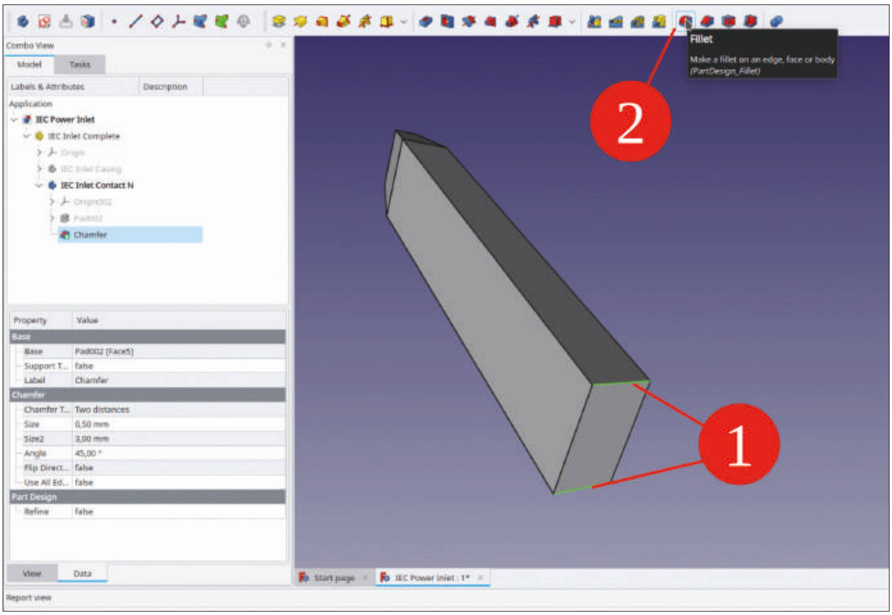


Figure F-28

Click on the 'Fillet' tool button. In the task window, set the radius to 1.99 mm (with 2 mm, the tool fails). Close the task window with the OK button (Figure F-29).

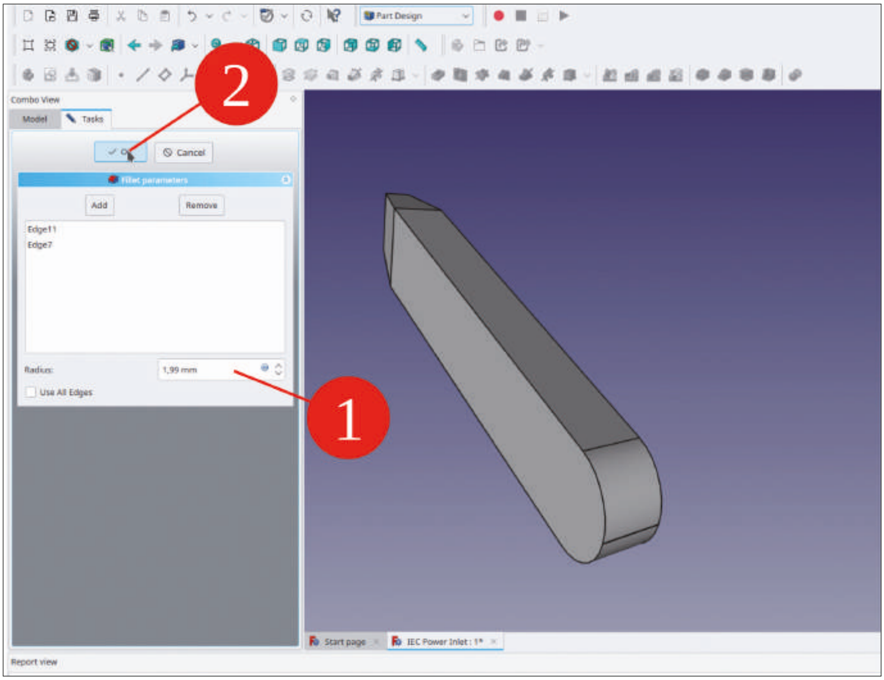


Figure F-29

For the eyelet in the contact, start the sketcher again and select the YZ plane as the sketching plane.

To show the geometry of the contact, close and reopen the sketch. From the main menu, select 'View | Orthographic view' and 'Sketch | View section'.

Click on the 'Create slot' tool button. Click on two locations of the Z axis to draw the slot, centered on the Z axis. In the 'Elements' list, right click one of the arcs and constrain its diameter to 1.5 mm (Figure F-30, step 2).

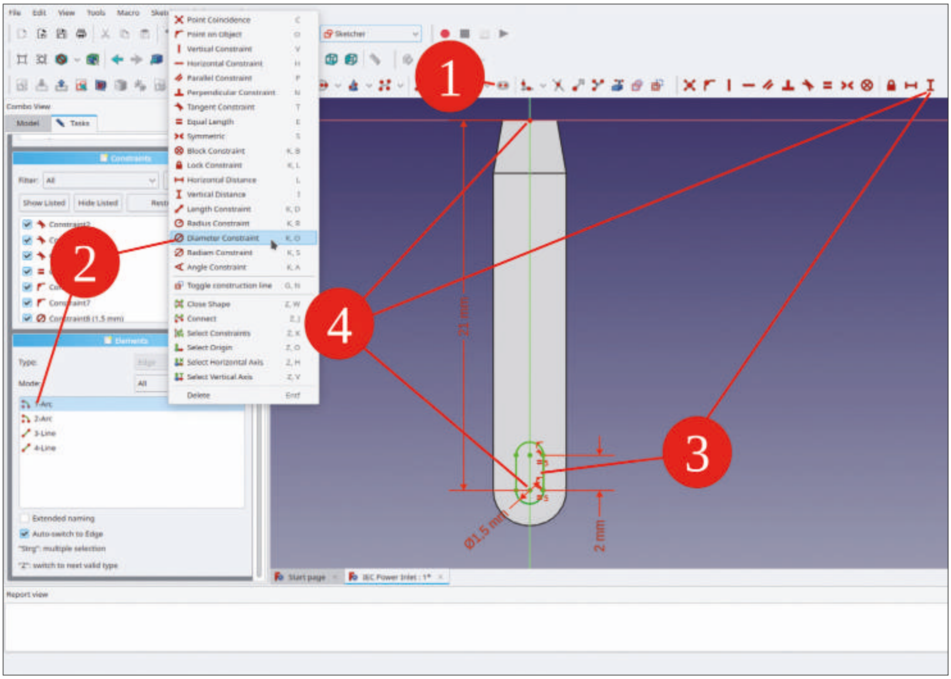


Figure F-30

Mark a straight line of the slot and constrain the vertical distance to 2 mm (Figure F-30, step 3).

Mark the lower arc center point and the origin and constrain the vertical distance to 21 mm (Figure F-30, step 4). Close the sketch (top).

In the tree view, mark the new sketch and click the 'Pocket' tool button. Select for the type 'Through all' and check the 'Symmetric to plane' checkbox (Figure F-31). Close the task window with the OK button.

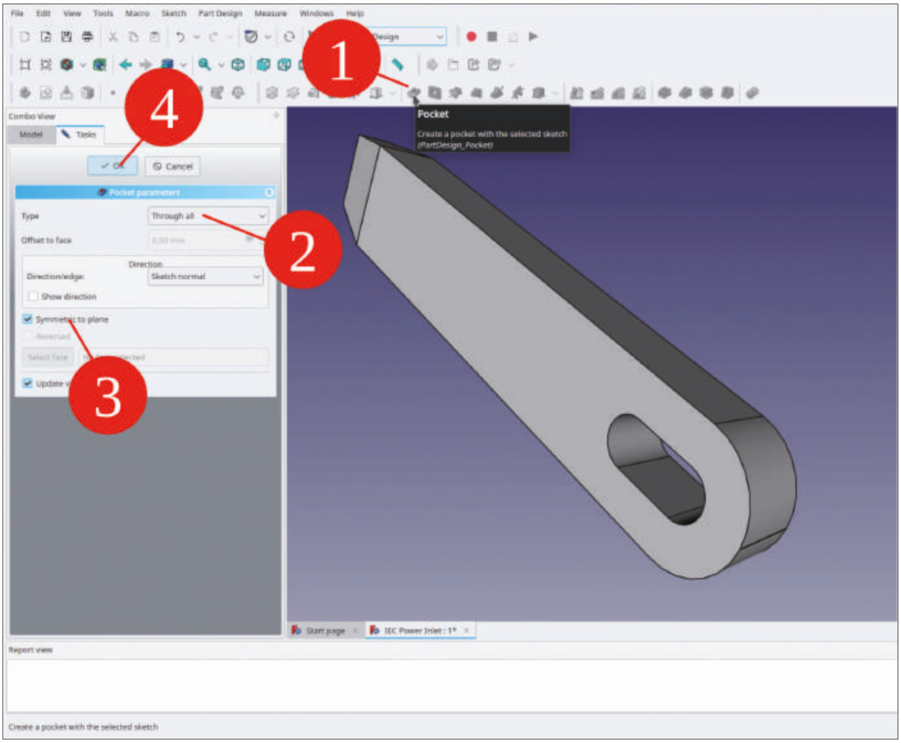


Figure F-31

In the tree view, mark and copy the 'IEC Inlet Contact N' body (CTRL-C). A selection dialog appears. Do not change the items, just click 'OK'. Double-click the Std-Part-Container to activate it and paste a copy of the contact with CTRL-V. Drag-and-drop the pasted body into the Std-Part-Container and rename it to 'IEC Inlet Contact PE'. Note: PE stands for *Protective Earth.*

In the tree view, display the body 'IEC Inlet Casing' again (mark it and press the SPACE key). Mark the body 'IEC Inlet Contact N'. In the property list, click into the 'Placement' edit field and open the task window with the ... button.

Set the Y translation to −2 mm, the X translation to −7.125 mm and close the task window with the OK button.

In the same way, edit the placement of the PE contact. Set the Y translation to 2 mm. The PE contact protrudes more than the other contacts, therefore also set the Z translation to 2 mm (Figure F-32).

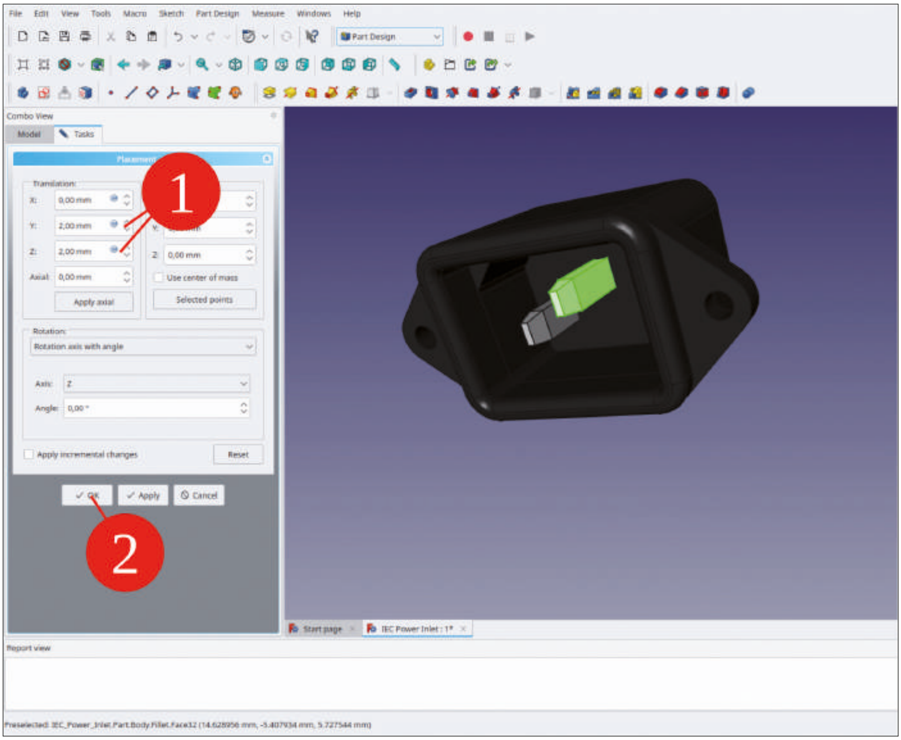


Figure F-32

The PE contact is longer than the other contacts. To account for this, in the tree view, expand the 'IEC Inlet Contact PE' body, and double-click the first design step (Pad). Change the length to 25 mm. And close the task window with the OK button. Now the eyelet is misplaced.

Now correct the position of the eyelet: In the tree view, expand the last design step (the tip, 'Pocket') and double-click the contained sketch. From the main menu, select 'View | Orthographic view' and 'Sketch | View section'. Note that the sketch is centered on the Y axis, while the contact is shown with its placement translations. Change the distance of the lower slot arc from 21 to 23 mm (Figure F-33) and close the sketch.

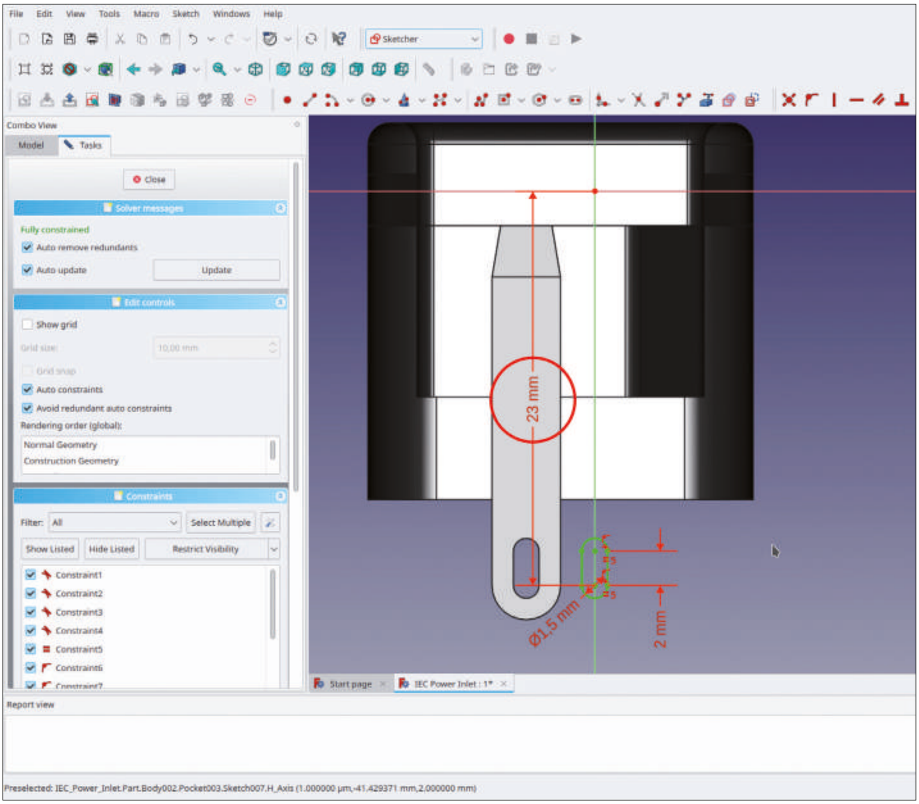


Figure F-33

In the tree view, mark the 'IEC Inlet Contact N' body and click the 'Clone' tool button.

Drag and drop the clone into the Std-Part-Container and rename it to 'IEC Inlet Contact L'.

In the tree view, mark the clone and edit its placement (the placement is relative to the parent contact). Set the placement X translation to 14.25 mm (Figure F-34).

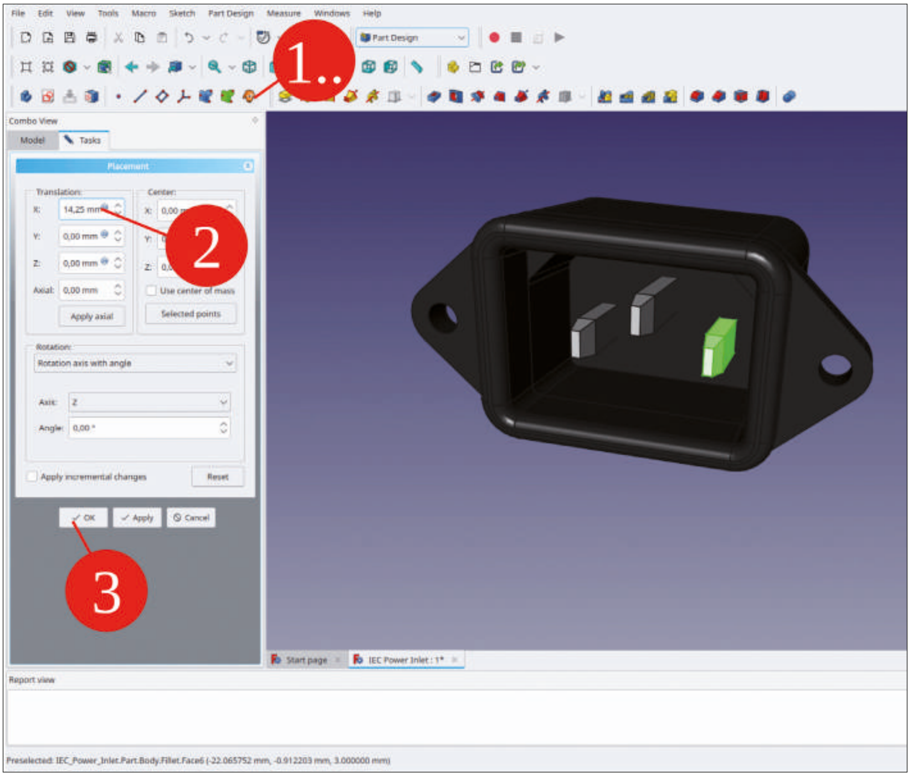


Figure F-34

In the tree view, mark all contacts and set their appearance to 'Chrome'. Save your work.

Appendix G • The 9 V Block (6F22) Battery

Start a new file and save it as '9V Block Battery'. Switch to the 'Part Design' workbench. In order to create the parent Std-Part-Container, click the yellow 'Create Part' tool button, and rename that to '9V Block Battery Complete'. Then, create the body for the battery casing by clicking the blue 'Create body' tool button, and rename that to 'Battery' (Figure G-1).

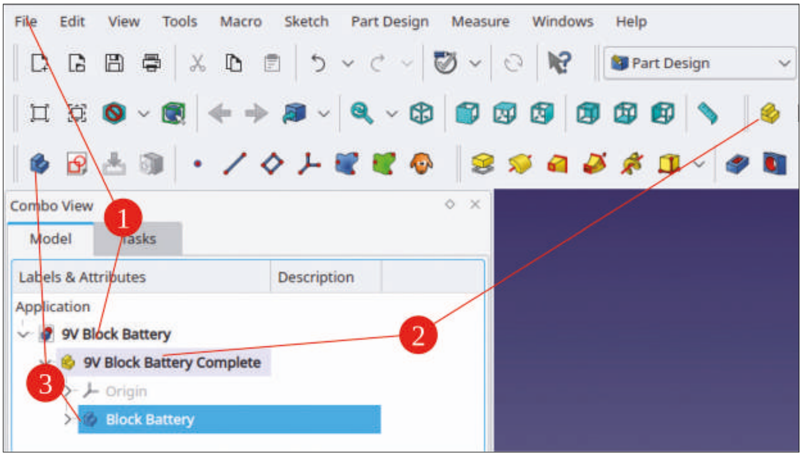


Figure G-1

With the larger distributors of electronic parts, you can find data sheets with mechanical drawings for the 9V block battery, commonly designated as type 6F21 or PP3. The mechanical dimensions are given with tolerance. You could use the upper tolerance limits for height, width, and thickness. It does not matter much which of the sources you take — the numbers are almost identical.

In order to draw the battery body, click the 'Sketcher' tool button from the workbench menu. When no plane is preset (e.g., by marking it with a click in the 3D view), a task window opens in which the sketch plane has to be selected. Select the XZ plane for the sketch and close the task with the OK button.

If not already preset, in the panel 'Solver Messages', check the boxes for 'Auto remove redundant' and 'Auto update', which is somewhat helpful (Figure G-2).

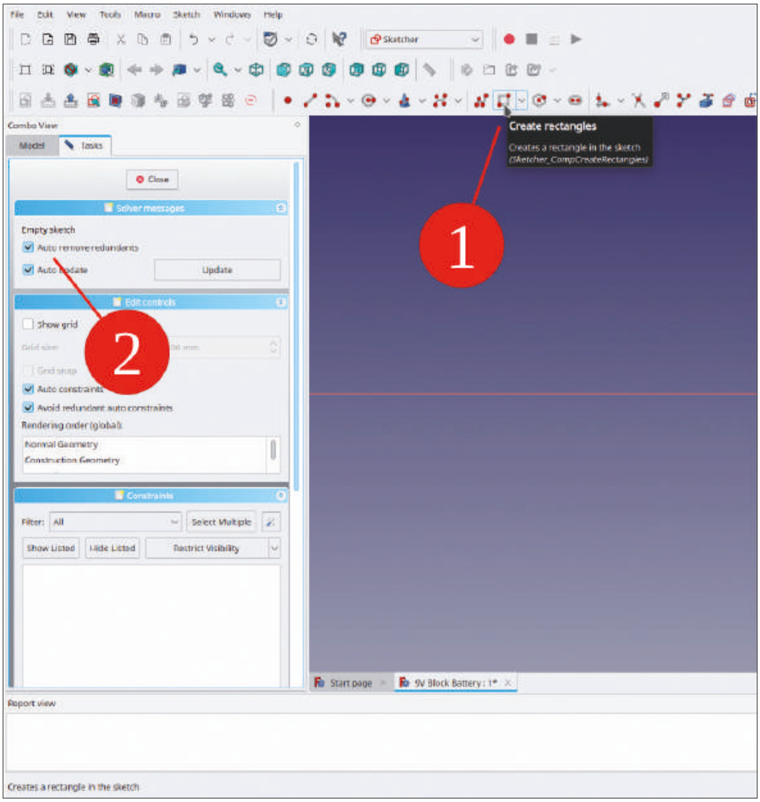


Figure G-2

From the sketcher menu, select the 'Create Rectangles | Create a rectangle' tool button and draw a rectangle with one side coinciding with the X-axis. End the drawing command by a right click (Figure G-2, G-3).

From a horizontal side of the rectangle, mark the two end points. Then, mark the vertical axis. From the sketcher menu, select the 'Constrain symmetrical' tool button. (Figure G-3).

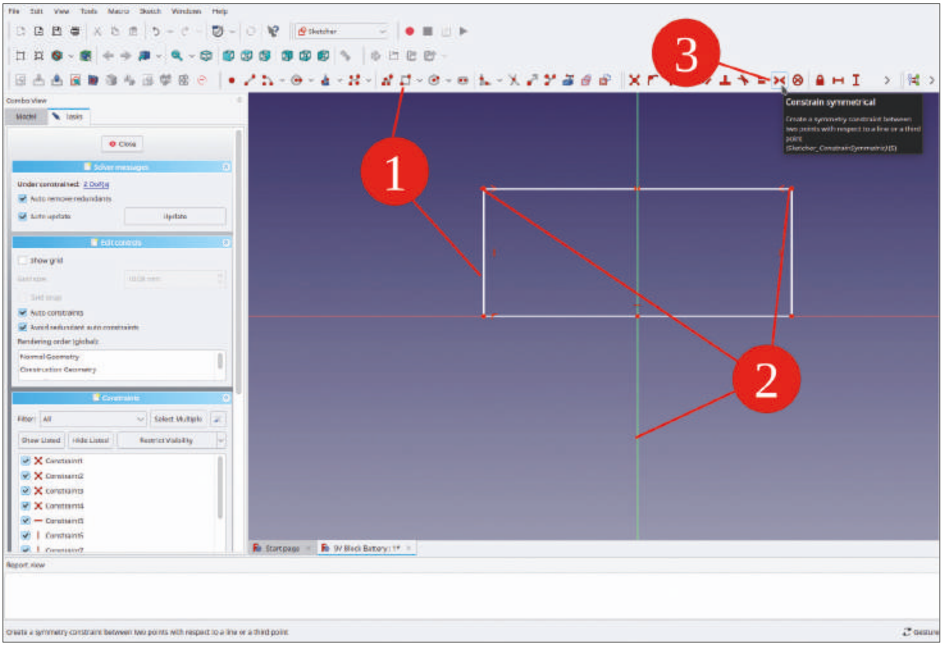


Figure G-3

Mark one horizontal side of the rectangle and click the 'Constrain horizontal distance' tool button. In the appearing dialog, enter 26.5 mm (the maximum width from the data sheet, Figures 4 and 5).

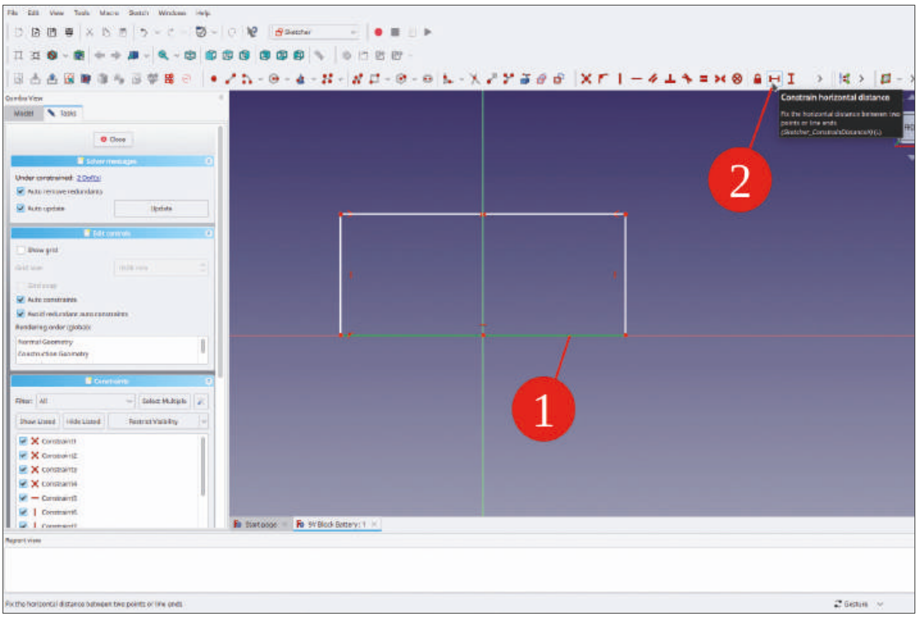


Figure G-4

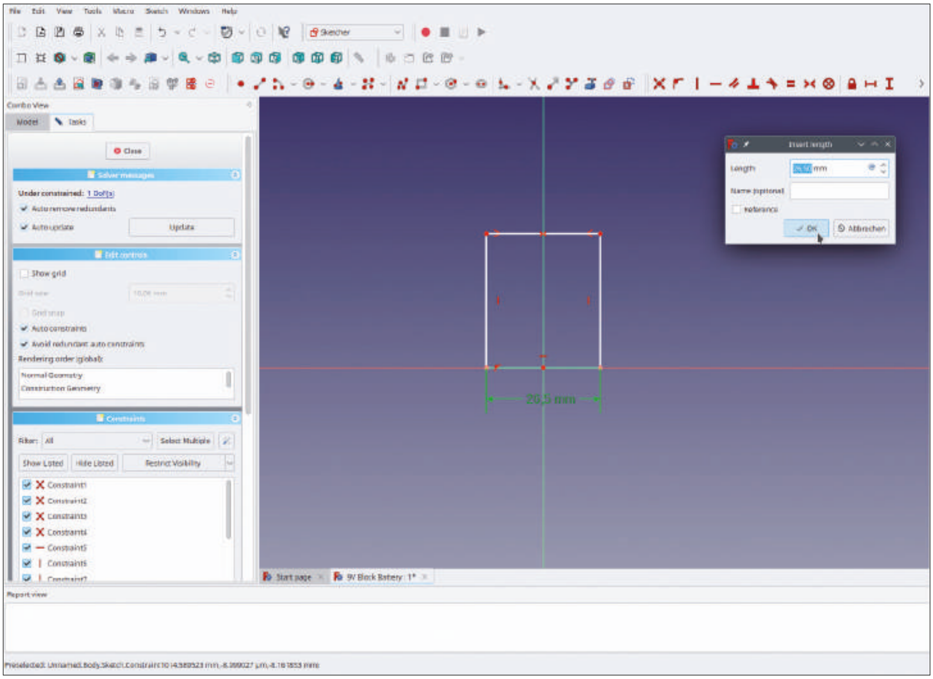


Figure G-5

Similarly, mark a vertical side of the rectangle and select the 'Constrain vertical distance' tool button. Enter 17.5 mm for the height. Now the rectangle is displayed in bright green as fully constrained. Close the sketcher task window with the 'Close' button on top.

With these dimensions, the battery will eventually have a loose fit in the holder. It would also be possible to set the dimensions to the center of the tolerance fields (then, some batteries could be difficult to insert). As another solution, some thin, elastic tape on the inside surfaces of the battery holder might help to accommodate all the cases.

In the tree view, mark the coordinate system of the body 'Block Battery', and show it with the SPACE key. In the 3D view, click 'into the blue' to deselect the coordinate system, and then once onto the XZ plane (to select only that, will turn green). Then, click the 'Create a datum plane' tool button. Thereby, the new datum plane shown is already attached to the XZ plane. In the attachment offsets, enter the battery height from the data sheet, which amounts to 46.6 mm (Figure G-6). Finally, close the task window with the OK button on top.

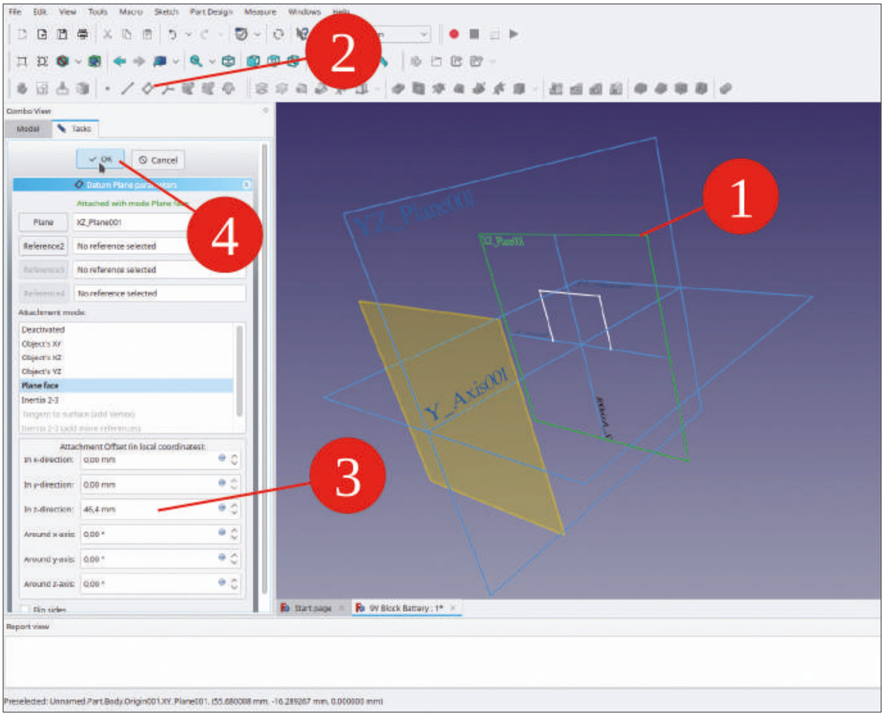


Figure G-6

It is somewhat counter-intuitive that the Z offset has to be used — as the Z axis of the coordinate system is pointing upwards. The 'Z' designates here a coordinate local to the datum plane and is therefore aligned parallel to the normal vector of the plane (orthogonal to the plane).

In the tree view, rename the new datum plane to 'Battery End Face'.

In the tree view, mark the sketch with the rectangle, then click the tool button 'Pad' (Figure G-7). A task window opens. Select the pad type as 'Up to face' and click the datum plane in the 3D view (Figure G-8). Close the task window with the OK button (top).

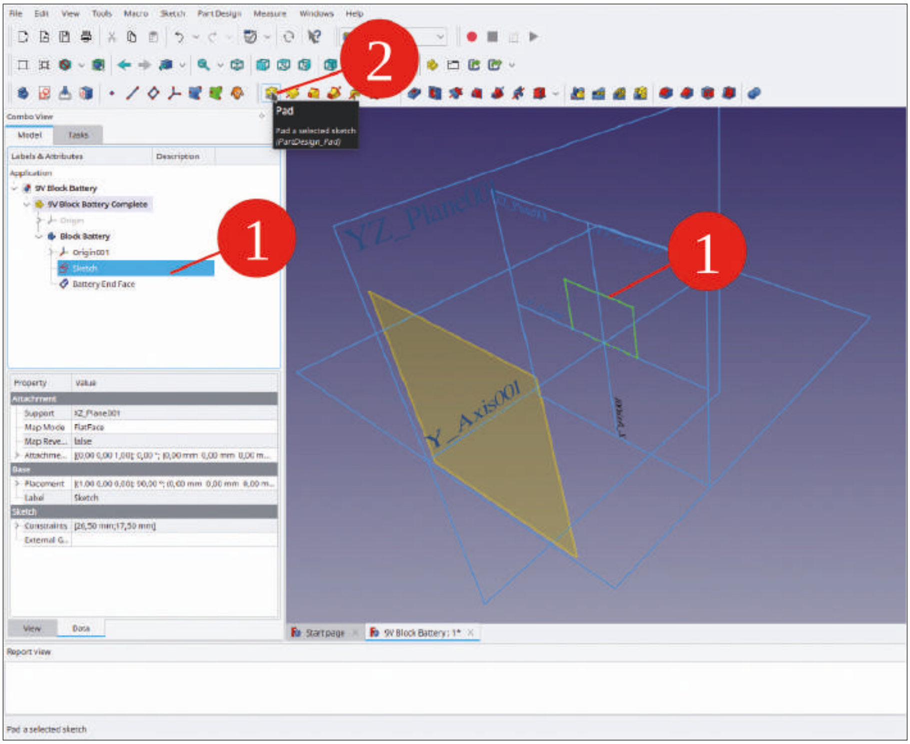


Figure G-7

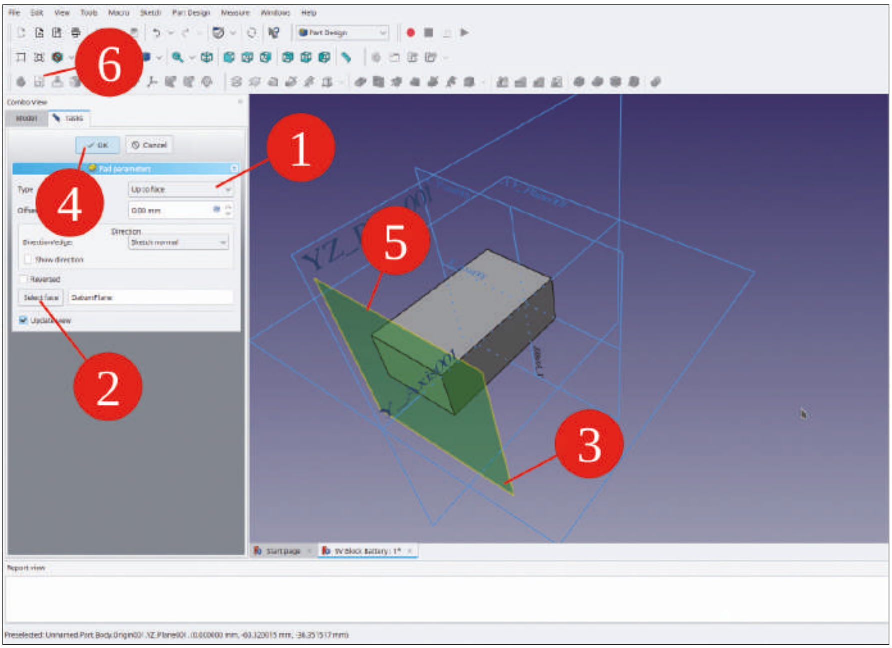


Figure G-8

At the top of the battery, create a recessed plane for the contacts. The outline needs to be sketched onto the top of the pad. Click the datum plane 'Battery End Face' and then the 'Sketcher' tool button (Figure G-8 steps 5 and 6). When the sketch opens for the first time, the other geometry is not shown. Close the sketch task window and reopen it by double-clicking on the new sketch in the tree view, to display everything.

To enable references to the batter casing, click the 'External geometry' tool button (Figure G-9). Then, click the 4 sides of the casing end face. The sides appear in violet, indicating that they can be referred to by the sketch.

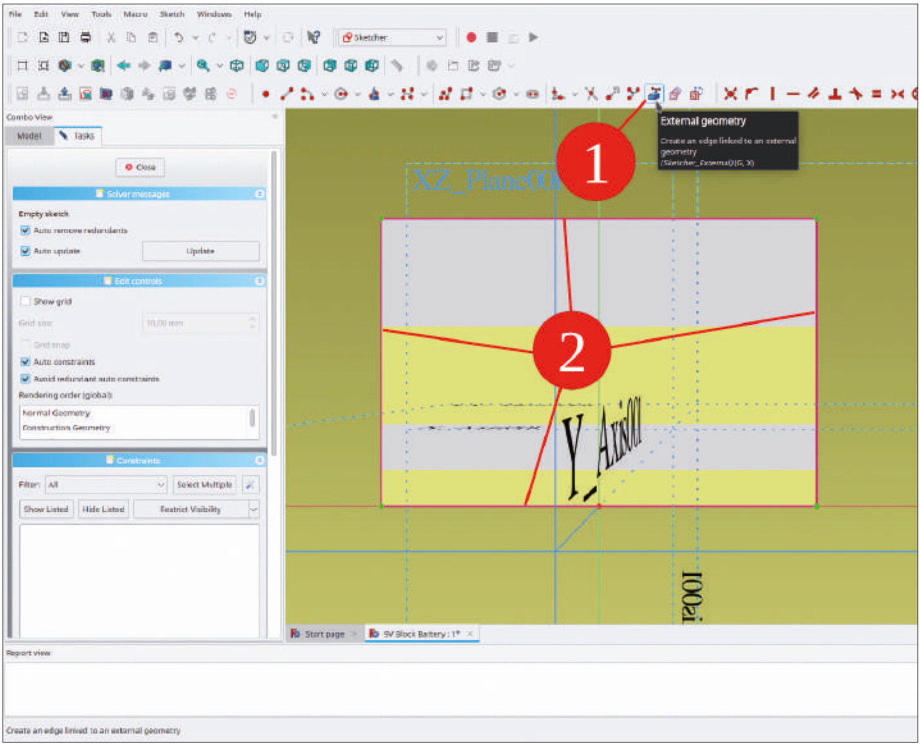


Figure G-9

In order to draw the recessed area outline, click the 'Create a rectangle' tool button and draw a rectangle into the area bounded by the violet lines (Figure G-10, step 1). The exact position of the corners is not yet important. End the drawing command with a right click.

Mark the two lower corners of the rectangle, then the vertical center line and select the tool button 'Constrain symmetrical' (Figure G-10, steps 2 and 3).

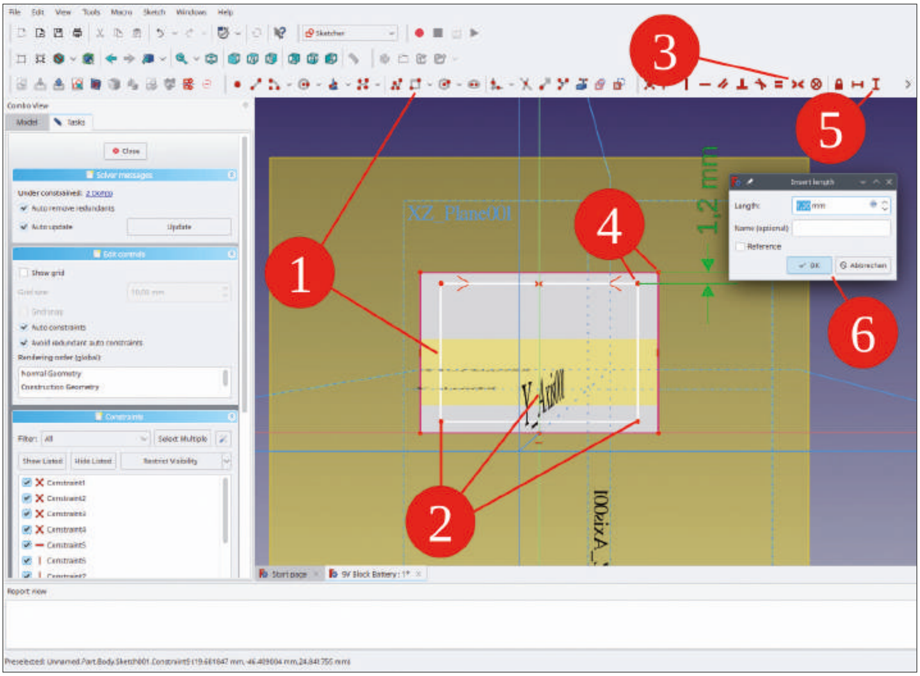


Figure G-10

Mark two adjacent corners of the nested rectangles (Figure G-10, step 4) and click the 'Constrain vertical distance' tool button. For the rim thickness, enter 1.2 mm in the pop-up dialog, and close that with the OK button (same figure, steps 5 and 6).

In the same manner, mark the two corner points again and set the horizontal distance to 1.2 mm as well. Because of the symmetrical constraint, the condition is set on both sides simultaneously (Figure G-11).

As the last degree of freedom, fix the distance at the bottom: Mark two adjacent corner points and set the vertical distance to again 1.2 mm. The sketch is now fully constrained (Figure G-11). Close the task window with the 'Close' button on top.

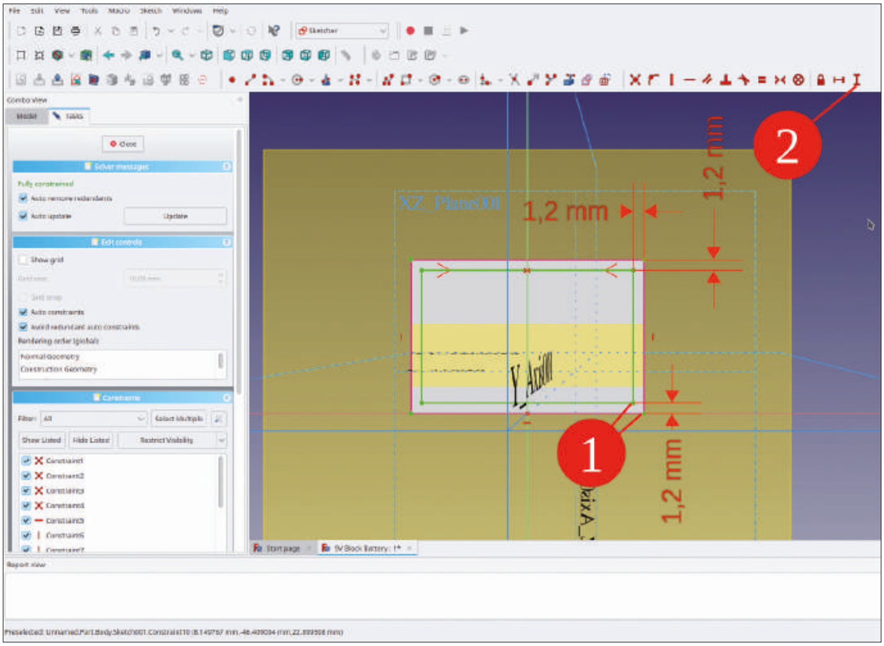


Figure G-11

In the tree view, mark the new sketch and click the 'Pocket' tool button (Figure G-12). A task window opens. Leave the pocket type at 'Dimension' and change the length to 1 mm. Close the task window with the OK button (top).

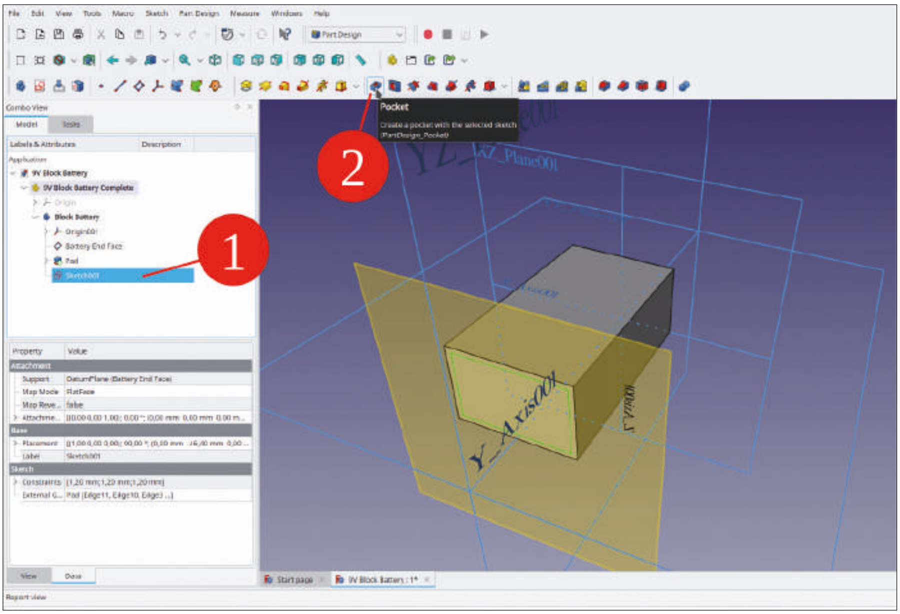


Figure G-12

In the 3D view, hide the datum plane 'Battery End Face' by marking it with a click and pressing the SPACE key.

Now you need another reference plane for the battery contacts. In the 3D view, again mark the XY plane of the coordinate system and click the tool button 'Create datum plane'. For the distance from the bottom of the battery, you take the total height from the data sheet, which amounts to 49 mm. Enter that value into the attachment offset field for the z direction. Close the task window and rename the new datum plane to 'Contacts End Face'.

The contacts are drawn and extruded to pads from this plane. In the tree view, mark the new datum plane (to preset it as the sketching plane) and click the sketcher tool button. The sketch opens, but the view is still empty. Close the sketch right away and reopen it by a double click in its tree view item line. Then, the battery geometry that is already present will be displayed.

In order to draw the (simplified) contacts, click the 'Create circle' tool button and draw two circles, into approximate position, and with arbitrary radius (Figure G-13). End the drawing command with a right click.

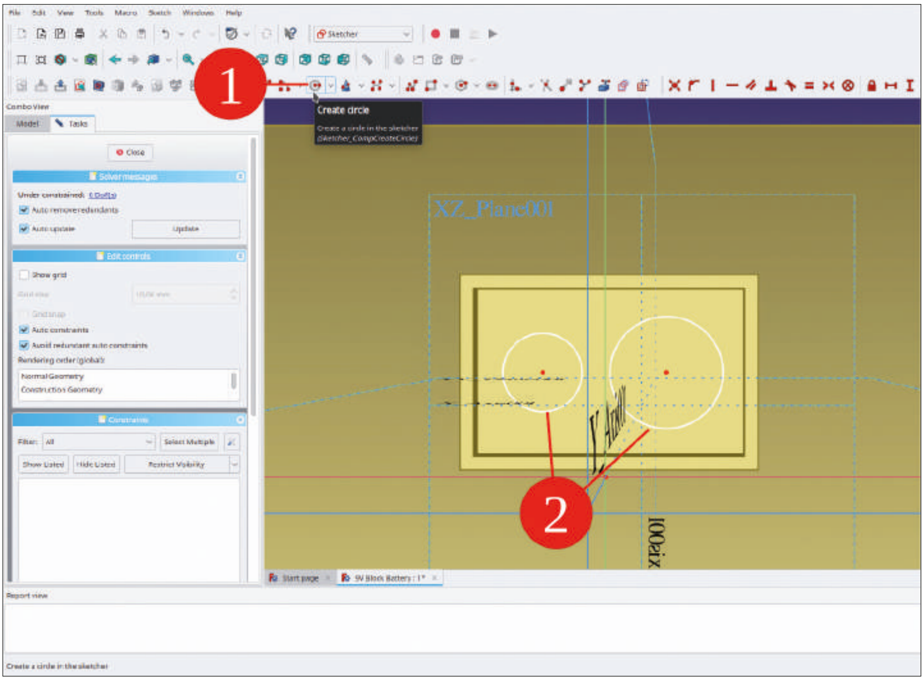


Figure G-13

To set both circles to the same diameter, mark both of them and click the 'Constrain equal' tool button (Figure G-14, step 1).

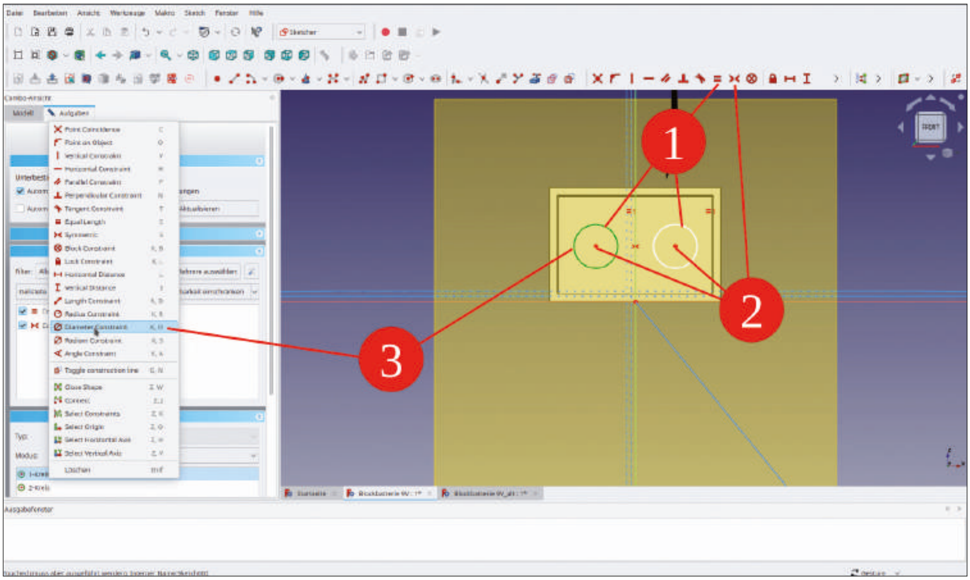


Figure G-14

Then, mark both center points of the circles and the vertical center line. Click the 'Constrain symmetrical' tool button to equalize the distances of the contacts from the battery center. (Figure G-14, step 2).

In the Combo View, scroll down to display the 'Elements' list. Right-click one of the circles and select 'Diameter constraint' from the context menu (Figure G-14, step 3). Set the diameter to 8 mm, this approximate measure is sufficient for further work.

Mark the two center points of the circles and click the 'Constrain horizontal distance' tool button. Set the horizontal distance to a value taken from the data sheet, 12.95 mm.

The distance of the contacts to the long side of the casing is still missing. Mark one of the center points and the origin of the coordinate system. Click the 'Constrain vertical distance' tool button and set the distance to 8.75 mm, which is half of the battery thickness taken from the data sheet. The sketch is now shown in bright green, as fully constrained (Figure G-15). Close the task with the 'Close' button (top).

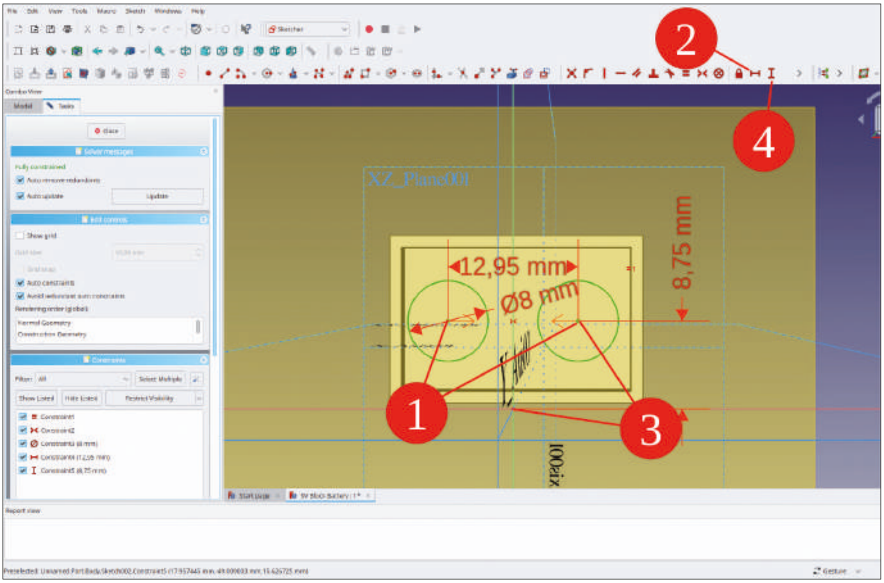


Figure G-15

In the tree view, rename the new sketch to 'Contacts Profile'.

In the tree view, mark the new sketch and click the 'Pad' tool button. An error message is thrown:

```
'Recompute failed! Please check report view. '
```

The generation of the contacts would eventually result in several material pieces – this is currently not supported. If the contacts are defined such that they are connected with the battery body, this error can be avoided:

In the task window, select the Type 'To first' (= next object face), and check the 'Reversed' checkbox below (Pads want to grow into positive direction, usually). (Figure G-16, step 3). With this resolution, the contacts successfully appear in the 3D view. Close the task window with the OK button.

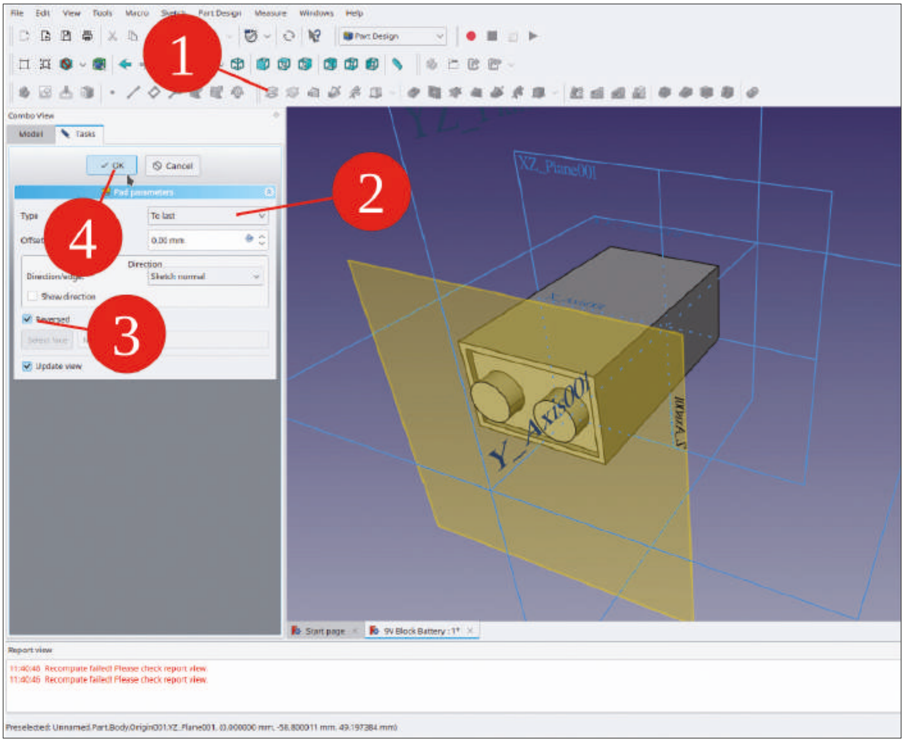


Figure G-16

In the tree view, hide the datum plane 'Contacts End Face' and the coordinate system 'Origin001'.

Holding down the CTRL key, mark the 4 long edges of the battery casing, as well as the edges at the bottom. Then, click the 'Fillet' tool button. In the task window, enter for the radius 1.2 mm (Figure G-17).

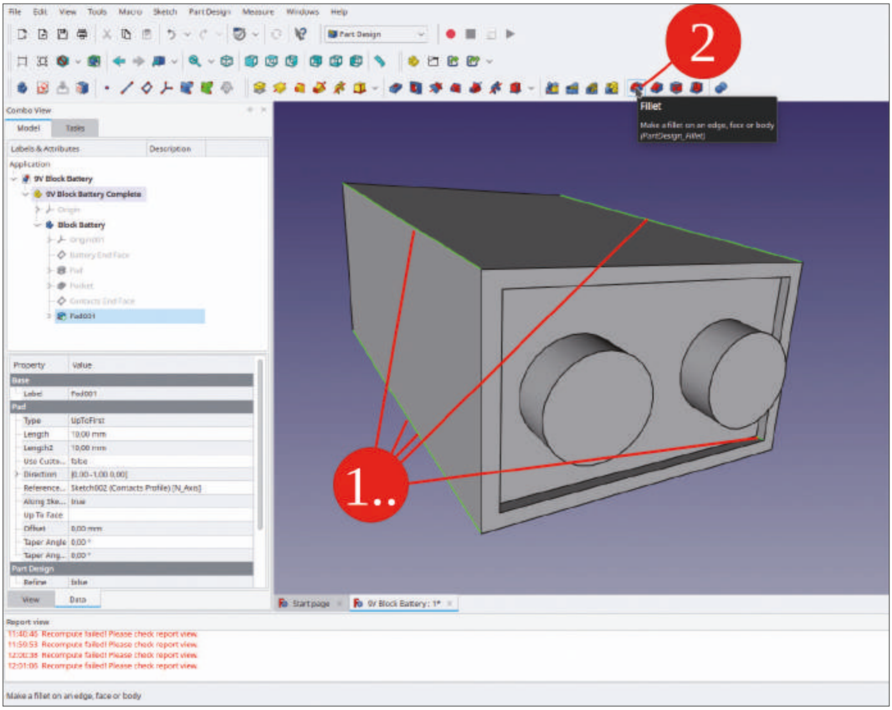


Figure G-17

Then, mark one edge of the inner and outer contour of the battery top face, as well as one of the bases, and click again on the 'Fillet' tool button (Figure G-18). Set the radius to 0.35 mm. If the radius is too large to be generated, error messages will be issued.

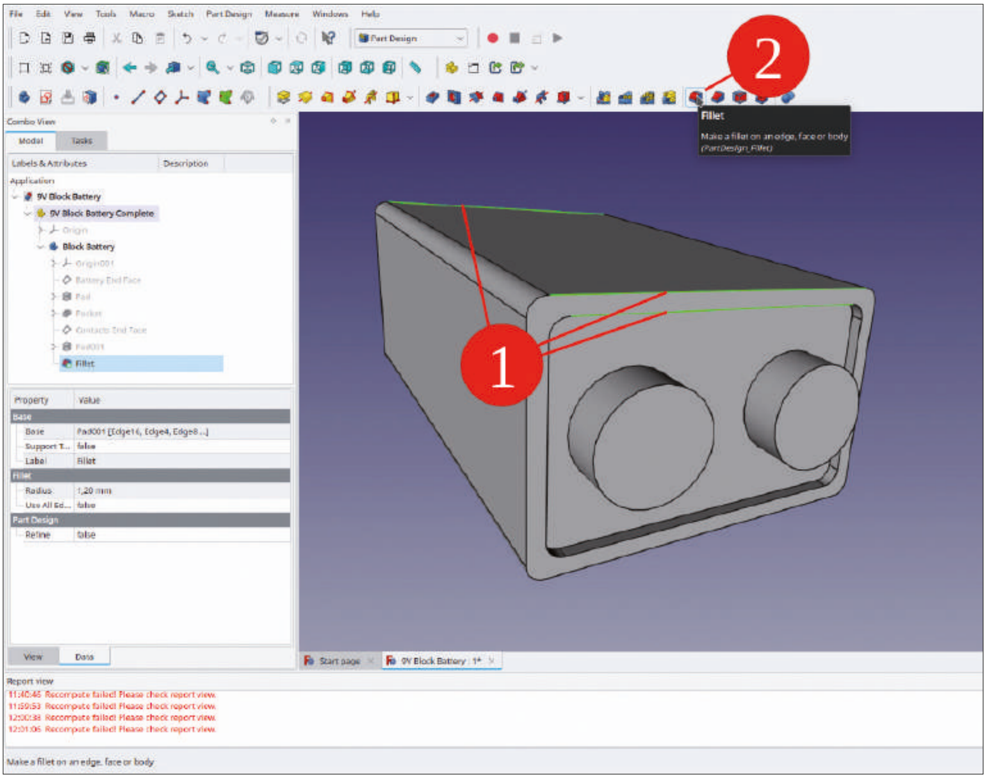


Figure G-18

The battery body can now get a more appealing color. In the tree view, right click it and select 'Appearance' from the context menu. Set the 'Material' property to aluminum, and the color to dark blue.

Now, the whole battery appears painted. Because it will not change anymore, you can color some facets. In the tree view, right-click the tip, and select 'Set colors…' from the context menu. A task window then opens.

Click the base plate below the contacts. In the task window, a color selection window is open. Click the button, which shows the actual blue color and set the facet color to black. In the same way, set the contact facets color to light gray.

Close the task window. Now the battery is ready to be used in other designs (Figure G-19).

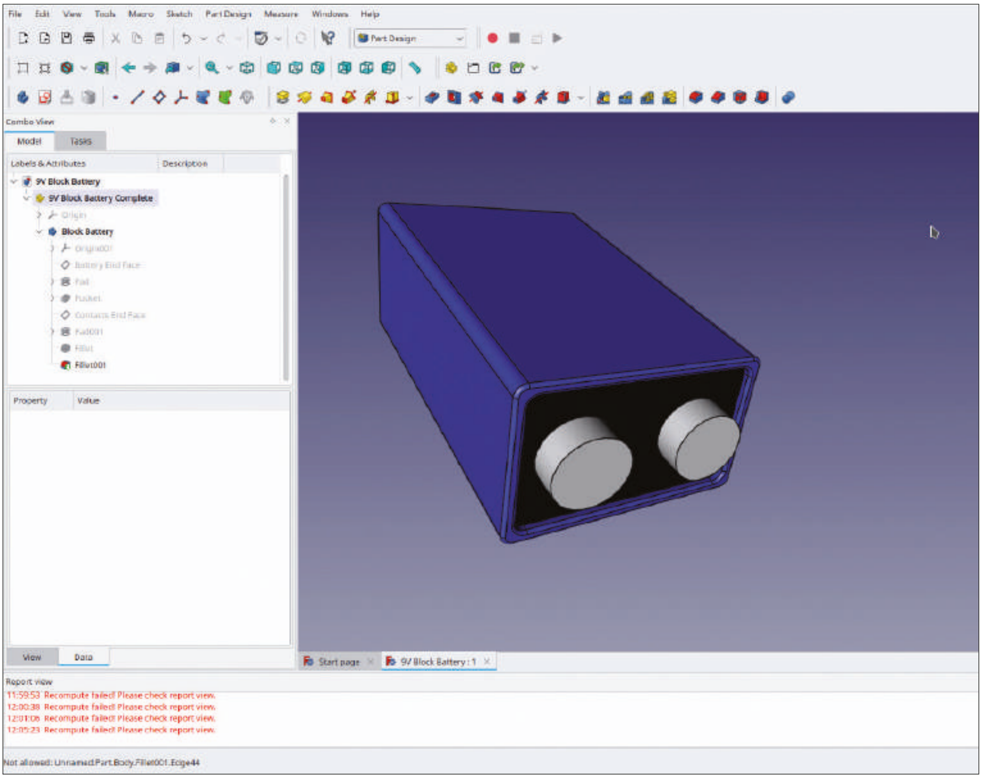

File
Edit
View
Tools
Macro
Sketch
Part Design
Measure
Windows
Help
Part Design
Combo View
Model
Tasks
Labels & Attributes
Description
9V Block Battery
9V Block Battery Complete
Origin
Block Battery
Battery End Face
Contacts End Face
Fillet001
Property
Value
View
Data
Start page
9V Block Battery : 1
Report view
11:59:53 Recompute failed! Please check report view.
12:00:38 Recompute failed! Please check report view.
12:01:06 Recompute failed! Please check report view.
12:05:23 Recompute failed! Please check report view.
Not allowed: Unnamed.Part.Body.Fillet001.Edge44

Figure G-19

Index

FreeCAD for Electronics Applications

Practical Introduction to 3D Modeling from Enclosure to Front Panel

Embedding a vintage component, creating a professional looking home for a circuit board, or even designing a complex apparatus complete with a chassis – these and many other challenges turn into a stimulating pleasure with FreeCAD. Once you have internalized the basic processes, there are virtually no limits to your imagination.

Starting to use a new software is never straightforward – especially with a tool as versatile as FreeCAD. Manageable, but at the same time easily usable individual components provide the starting point in this book. Putting these components together later results in assemblies.

In the FreeCAD universe, a workable trajectory is demonstrated. The described procedure is illustrative so the examples are easily applied to custom tasks. The devices were made by the author and illustrated with photos.

Creating a 3D design is requiring some effort but the initial investment pays off soon. Besides the impressive spatial representation of the projects, the extracted drawings yield a solid base for documentation and production. Extended FreeCAD capabilities like the unfolding of sheet metal parts enormously add to efficiency and pushes models forward into practical assembly.

Soon you will definitely not want to do without FreeCAD!

Thomas Duden studied Physics in Clausthal-Zellerfeld, where he also received his PhD. For research purposes, the experiments required the design of many instruments. These developments also included electronics (mainly stable high voltage and current sources), programming of PCs and microcontrollers plus the calculation of electron-optical components with their aberrations. Meanwhile, mechanical designs covered ultrahigh vacuum systems, components for electron microscopy and diffraction instruments, often in combination with spin-polarized electron beams. Longer work phases abroad took place at the IBM Almaden Research Center (San Jose), at Arizona State University, and at the National Center for Electron Microscopy of the Lawrence Berkeley National Laboratory. Thomas Duden obtained his ham radio license at the age of 16 and currently operates a consulting service for the development of scientific instruments.

www.elektor.com

ISBN 978-3-89576-574-2