Getting started with FreeCAD

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1 A few words about FreeCAD

FreeCAD is a parametric 3D modeler, free and open-source (under the LGPLv2+ license) that has been under continuous development since 2002. FreeCAD has an organisation in workbenches and features similar to CATIA, Creo, Autodesk Inventor, SolidWorks or Solid Edge. FreeCAD uses extensively other open-source libraries from the field of scientific computing such as Open CASCADE (a CAD kernel), Coin3D (an incarnation of Open Inventor) and the Qt GUI framework. In addition FreeCAD can be controlled from Python which allows for scripting or use of FreeCAD itself as a python module. The stable version you should use in the course is v16 or higher. The tutorials are written for v16, so small changes in the GUI may happen if you use a higher version.

FreeCAD is hosted on GitHub (https://github.com/FreeCAD). For a quick tour see the FreeCAD homepage (http://www.freecadweb.org/).

Learning a bit of FreeCAD is a good investment of your time. If you work with numerical simulations of realistic engineering components or systems, you will either have to use CAD or interact closely with people building CAD models for you. While there are many other CAD tools than FreeCAD, the general way of thinking is quite similar. More pragmatically, you will need to do a bit of CAD in the 3D printing module of the Fluid Mechanics course.

2 Installation

You should read this section if you want to install FreeCAD on your personal computer. If you cannot / do not want to install FreeCAD on your computer, it is also available as a module on the UiO Linux computers of the department.

General installation instructions for all OSs are given on the FreeCAD website, but read the whole present section before you install anything. In all the following, it will be assumed that Linux (Debian or Debian based distribution such as Ubuntu) is used. This is recommended as FreeCAD is most stable on Linux (several annoying bugs were observed on Windows, including failure to save work... Mac was found stable). If you want to install FreeCAD on Windows or Mac, follow the instructions given on their website:

`http://www.freecadweb.org/wiki/index.php?title=Install_on_Windows` http://www.freecadweb.org/wiki/index.php?title=Install_on_Mac

It is not recommended to install FreeCAD from the package manager of you Debian distribution, as the version packaged may be quite old (using sudo apt-get on Ubuntu 16.04LTS you will get v15, which do misses some important features present in v16). It is recommended to install the latest stable release version. This can be done following the instructions on FreeCAD webpage:

http://www.freecadweb.org/wiki/index.php?title=Install_on_Unix

Follow the instructions of the **Latest Stable Release from the stable releases" PPA or "daily" PPA** section. Install the version corresponding to the **stable PPA**, typing:

```
$ sudo add-apt-repository ppa:freecad-maintainers/freecad-stable
$ sudo apt-get update
$ sudo apt-get upgrade
$ sudo apt-get install freecad freecad-doc
```

3 Getting started

FreeCAD is supported by a growing community and many presentations, tutorials and forum discussions are available online. Here I will give a very basic introduction to get you started and I will refer to online tutorials and manuals if you need more details.

3.1 Open FreeCAD

To open freecad on UiO Linux computers, you will have to first load the freecad module:

```
$ module load freecad/0.16.6706
$ FreeCAD
```

The name of freecad module may change (if its version gets updated), but you can always list all available modules with:

```
$ module avail
```

To open freecad on your computer, just open a terminal and type:

```
$ freecad
```

This will take you to the FreeCAD start center. There you can either choose some of the options displayed on the main panel to get some FreeCAD introduction, or directly start working. You can open an existing project through the **File>Open...** menu or using the icon. You can create a new project through **File>New** or using the icon. For now we will consider that you opened a new project.

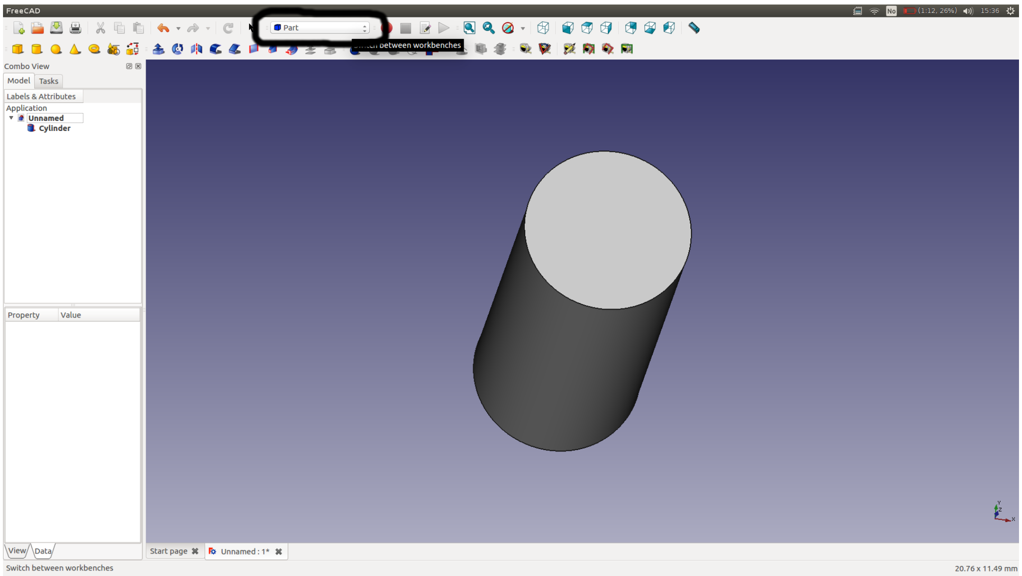

Figure 1: Selecting the right workbench is the most important step to get to build a good model.

3.2 Workbenches

The first key concept to understand about freecad is the notion of workbenches. A workbench is a collection of tools that make it easy to realize a task. You can select a workbench through the drop down menu in the icon bar, see Fig. 1. In the 3D printing module you will mostly use the **Part Design** and the **Part** workbenches. You should get more familiar with the workbenches available by reading at least the following documentations pages:

`http://www.freecadweb.org/wiki/?title=Workbenches`

`http://www.freecadweb.org/wiki/index.php?title=Part_Module`

`http://www.freecadweb.org/wiki/index.php?title=PartDesign_Workbench`

4 Example 1: using the Part workbench, primitives and FreeCAD most important tabs

Let us take a simple example. Here you will create a simple shape in FreeCAD.

- Click on the **Create a new empty document** button.
- Select the **Part** workbench
- Click the **Create a Cylinder** button. Note that the buttons space can become overpopulated and then some buttons are hidden. You can solve this by moving around button arrays by draging and dropping the vertical three dots visible before each button array.
- Click the **Set to axonometric view (0)** button

- Select the cylinder you just created. For this, click on the **Cylinder** line in the **Model** tab of the **Labels & Attributes** panel.

- You can Delete, Rename etc the cylinder you just create by right clicking on the **Cylinder** line. Rename the cylinder into **TubeVolume**.

- In the **Data** tab, you can edit properties of the selected object. For our newly created **TubeVolume**, this includes placement, label (that we already changed), and the properties that define the cylinder (Radius, Height, Angle). Set **Radius** to 4 mm by clicking on the field and typing the value. Set **height** to 20 mm.

- Now the cylinder is too big for your screen. You can train to move the camera relatively to the shape. Read and follow the instructions of the manual:

  `http://www.freecadweb.org/wiki/index.php?title=Mouse_Model`

  I advice you to use the defaults and to use a mouse (touchpad is not very adapted for working on CAD). You can also navigate the views by using the icons **Set to [...] view** and **Fit the whole content on screen** to get right zoom factor and axonometric or standard camera positions.

- Create a new cylinder, call it **TubeInside** and set a radius of 2 mm and a heigth of 20 mm.

- Select first the TubeVolume you created and then, holding the Ctrl key, the TubeInside so that both shapes are selected at one. Then, click the **Make a cut of tow shapes** button. This will create a new object, called **Cut**, in the Model tab. Rename it to **Tube**. You can expand the details of you Tube by clicking on the little triangle before its name. You can then display the hierarchical graph of you object. The graphical display of your object should look like in Fig. 2

- The underlying objects are still available and can still be accessed. They appear greyed in the Model tab as they are currently not displayed on screen. Click on **TubeInside**. Change the radius to 3 mm. The Tube object gets automatically updated. While TubeInside is selected, go to the **View** tab instead of the Data tab. There you can affect properties that have an effect on the rendering of your shapes. Set **Visibility** to **True** by clicking on the option. Toggle back the visibility to False by pressing space on your keyboard.

- Now we can change the placement of our object. Select the **Tube** object. Click on the **Placement** option in the Data tab, and open the detailed selection box by clicking the **...** symbol. If your tab is too narrow, you can rescale it by moving its right edge. The placement options are described in the manual:

  `http://www.freecadweb.org/wiki/index.php?title=Placement`

  Take a look at the manual and get familiar with the effect of each field. Then, apply a **Translation** of 3 mm in the Z direction and a 20 degrees rotation around the Y axis. Press OK to save you changes.

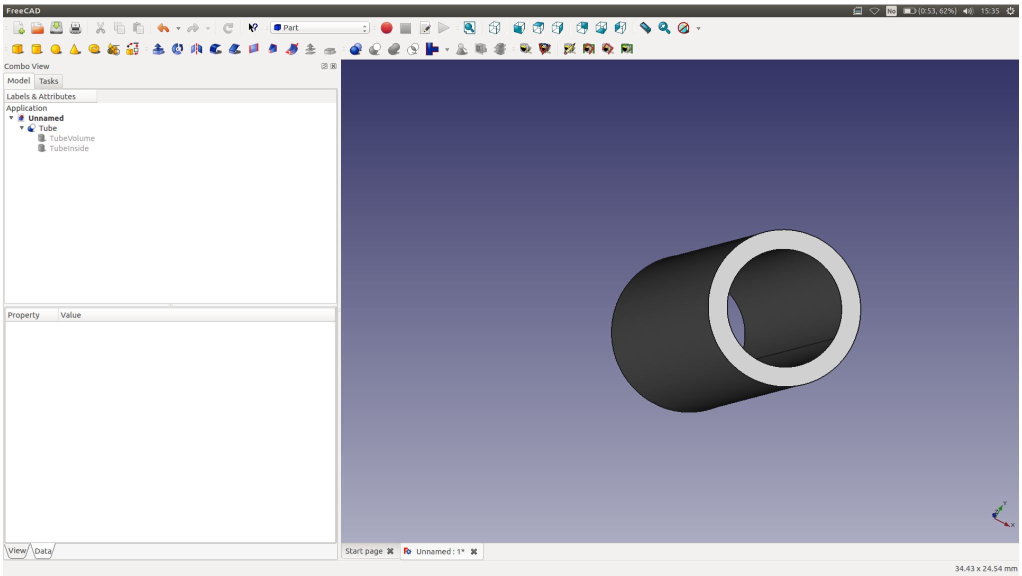

Figure 2: The goal of Tutorial 1: build a pipe from a boolean difference.

- You can save your work through the **File>Save** menu. This will save as a .fcstd, i.e. "FreeCad STanDard". You can export an object in many other formats by selecting the name of the object to export in the **Model** tab and then going through the **File>Export** menu. You can select between many file types. Most often in the 3D printing module you will use the **Alias Mesh (*.obj)** or the **STL Mesh (*.stl)** to export to the form labs software.

This should have given you an overview of some of the important panels and tabs of the standard FreeCAD window.

5 Example 1 bis: another way to design pipes

Often in CAD there are many ways to attain a given result. In the Example 1 we build a cylindrical pipe by taking the boolean difference between two solids. But FreeCAD also has a function that lets you perform the same task in one single step. First, start reading the FreeCAD documentation about the **Utility to apply a thickness**:

`http://www.freecadweb.org/wiki/index.php?title=Part_Thickness`

You should understand that this is well suited to build cylinders! Start a new document and:

- Move to the **Part** Workbench.
- Create a cylinder and select its top face by clicking on it in the main view window.
- Click on the **Utility to apply a thickness** button. The shape gets updated and you enter a dialog box looking like in Fig. 3 to set the fine tuned options of the tool. You can move around the modified cylinder to observe it. It is not yet a tube, so we should work a bit with the dialog box to get what we want.

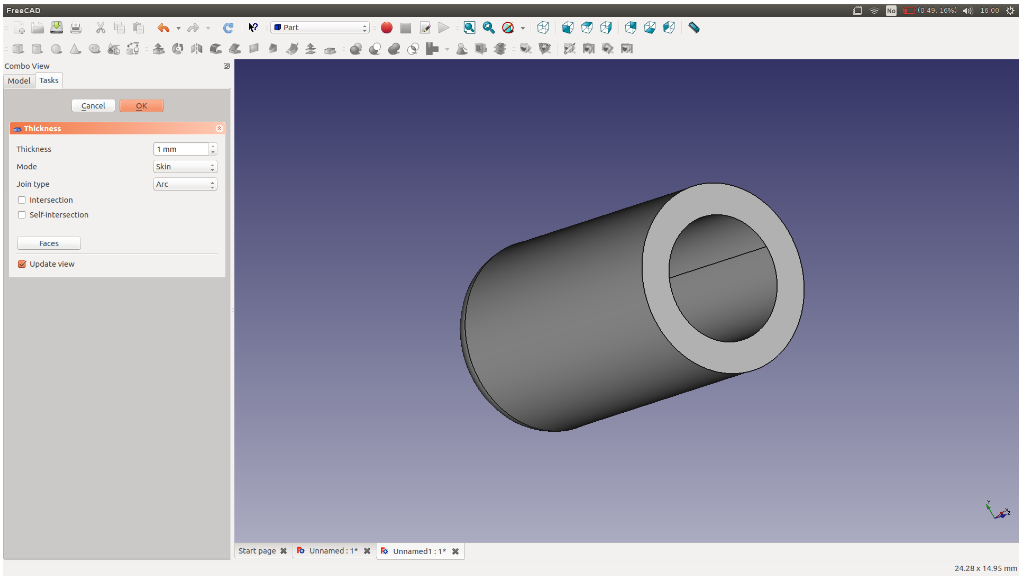

Figure 3: Dialog boxes are often used by FreeCAD to get user input.

- Click the **Faces** button in the dialog box. This will reset the view to the full cylinder. Select the top face and the bottom face of the cylinder by clicking one and then the other face while holding the crtl key. Then click **Done** in the dialog box. You now have a tube. You can adjust its **Thickness** by adjusting the corresponding option in the dialog box. Click **Ok** in the dialog box when you are satisfied with the result.

This small example should convince you to read quickly through FreeCAD documentation and to not hesitate to look for a bit of help on the FreeCAD forum before you start using a clumsy solution for designing your part! For example, if you should work with tubes in the future, look also at the documentation around the **Join walled objects** button!

6 Example 2: using the Part Design workbench and sketcher

Similarly to many other CAD tools, a powerful feature of FreeCAD consists in creating 2D geometries to extend them into 3D shapes. Here, I will take you through a simple example. But first, you shoud have a good look at the Sketch Workbench manual and go through the different buttons descriptions:

`http://www.freecadweb.org/wiki/index.php?title=Sketcher_Workbench`

- Create a **New** document
- Select the **Part Design** workbench
- Click the **Create a new sketch** button, then choose **XY-Plane**.
- We will draw a simple shape in 2D. Start by creating two **lines**, then one **arc** and one more **line** as indicated in Fig. 4. Note that if you create a line that is nearly horizontal, FreeCAD will automatically add it an horizontal constraint (visible as an additional

horizontal line icon next to your pointer). The same is true for vertical constraint. If you want to make some point of your current shape coincident with another point or line, click on it so that it gets highlighted. If FreeCAD does not get what you want and does not automatically enforce some constraints, this is not a problem - you can do it in the next step.

- Now that we have created a simple shape, we want to add constraints on it. First, we want to make it closed. For this, select the start and end points of your sketch (by clicking one then the other while pressing ctrl) and press the **Create a coincident constraint** button. The line at the top of your sketch should be horizontal (indicated by a small red icon next to it). If this is not the case, select the line and click on the **Create a horizontal constraint** button. Make sure the same way that the vertical line is vertical.
- Constraints are displayed in the main view and in the **Task** tab in the **Constraints** tile. By right clicking on a constraint you can delete or edit it. The **Solver messages** tab gives you indications about the current state of your sketch (under constrained, fully constrained, over constrained).
- Click on the arc of circle and then on the **Fix the radius** button. Set the circle radius to 15 mm. You can move the position of the 15 mm label by dragging it around. If you do not see the **Fix the radius** button, it probably means it is hidden in the buttons array - drag the constraints button array to a place with more space.
- Similarly, set the length of the horizontal and vertical lines (this can be done with the **Fix the horizontal distance** and the **Fix the vertical distance** buttons, or with the **Fix a length** button), the angle between the horizontal and the inclined line (use the **Fix the angle** button), make the top left corner of your sketch coincident with the origin, and set the angle of the arc of circle. Your sketch should then be fully constrained and turn green. Drag the labels around so that you sketch looks like the picture in Fig. 5.
- We can illustrate what happens if you try to make your sketch over constrained. Try to set the length of the inclined line to 50 mm. You will get an error message.
- Now that we have a 2D shape, let us make it 3D! Close the sketch editor, move to the **Labels & Attributes** tab. Rename your sketch to **Base**. Select your sketch, and click on the **Pad a selected sketch** button. This will open a dialog box. Set the **Length** of your pad to 150 mm and click Ok. Take a look at your new 3D shape by moving it around.
- Select your newly created **Pad** in the **Labels & Attributes** tab. Observe that you can set its properties through the **Data** tab. Set the length to 160 mm. You could also change, for example, its placement. Similarly you have access to the **View** tab. Set **Transparency** in the **View** tab to 50. Click in some empty part of the display view and move your solid around.
- Let us now say that we want to drill a hole in our shape, perpendicularly to the inclined face. Click on the inclined face to select it. Then, while it is selected, click on the **Create a new sketch** button. This will open the sketch editor with the plane in which you will be sketching set as the face that was selected. Move around as shown on Fig. 6 to get convinced about that. Once you are done looking at the position of your sketch in 3D, click on the **View sketch perpendicular to sketch plane** button to come back to a normal view.

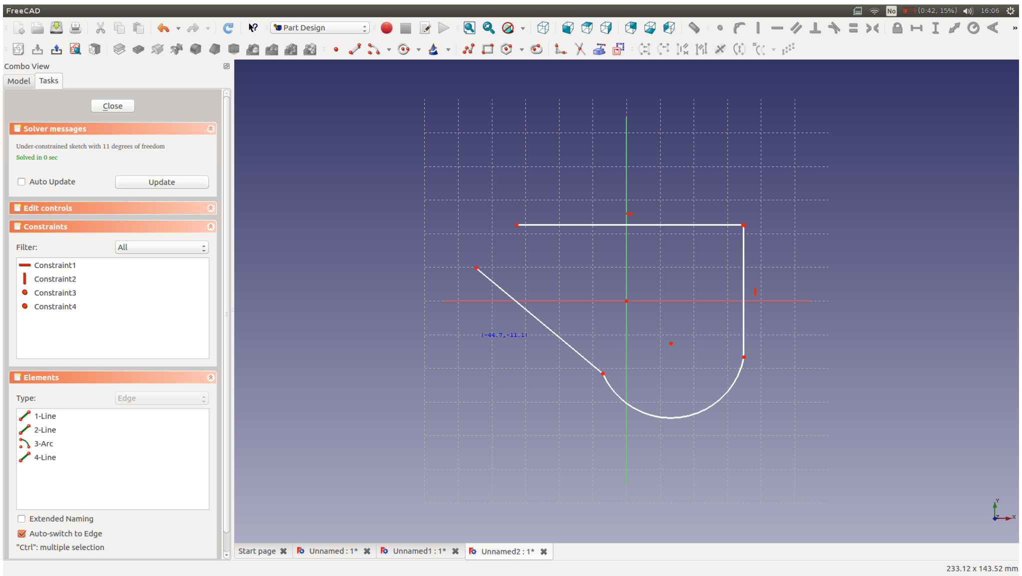

Figure 4: This is how a sketch will typically look like before you set constraints.

- Use the **Create a Rectangle in the sketch** button and set constraints to get a fully constrained sketch. You can use the **Create a lock constraint** button to set at once the X and Y position of the left bottom side of the rectangle. Click close when you are done.
- Back in the main view, select your new sketch and press the **Create a pocket** button. Set the type as **Through all** and press Ok. You can now check the results by navigating in 3D.
- Save the current state of your project (use the .fcstd), as we will continue building on it later on.

7 Example2 (continuation): A few more advanced sketch and 3D shape creation tools

By now you should already be familiar with the sketch tool and how to extend a 2D geometry into 3D. In addition, you should understand that the best way to get efficient is to read (quickly) through the documentation and discussion forum to get aware of what can be done using FreeCAD built in functionalities. However I will still help you a bit with a few sketcher and 3D shape creation tools.

- Open the project as you got it at the end of the **Example 2**. Move to the **Part design** workbench.
- Select the lateral face of your object, see Fig. 7. Create a new sketch there. As you can see, the part is still displayed under the sketch but you cannot access any of its edges. For this, we need to import some edges from the 3D object into the 2D sketch.
- Click the **Create an edge linked to an external geometry** button and then the inclined side of the 3D object. The edge will then turn purple, and you will gain access to it in the sketch.

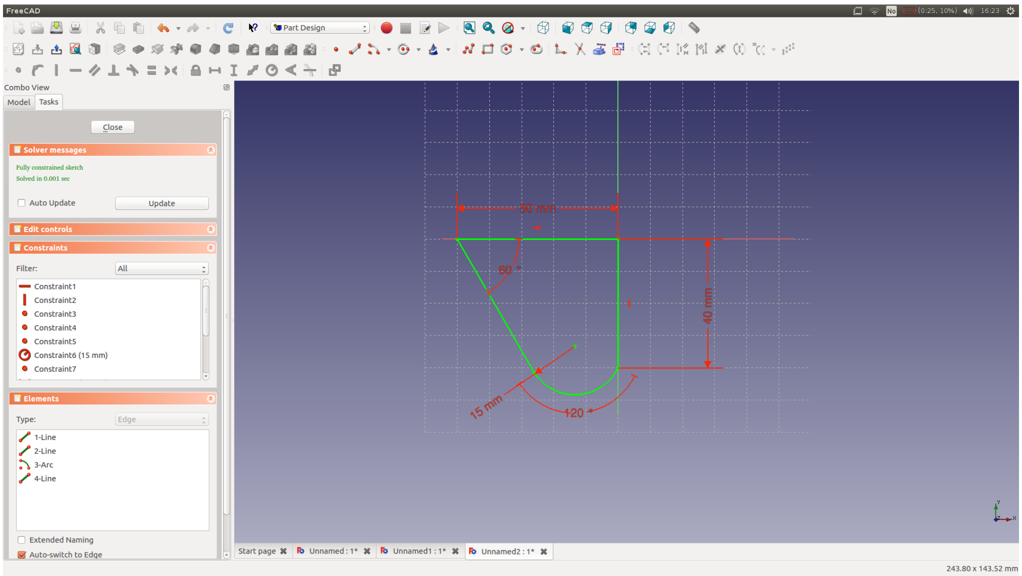

Figure 5: A fully constrained sketch turns green.

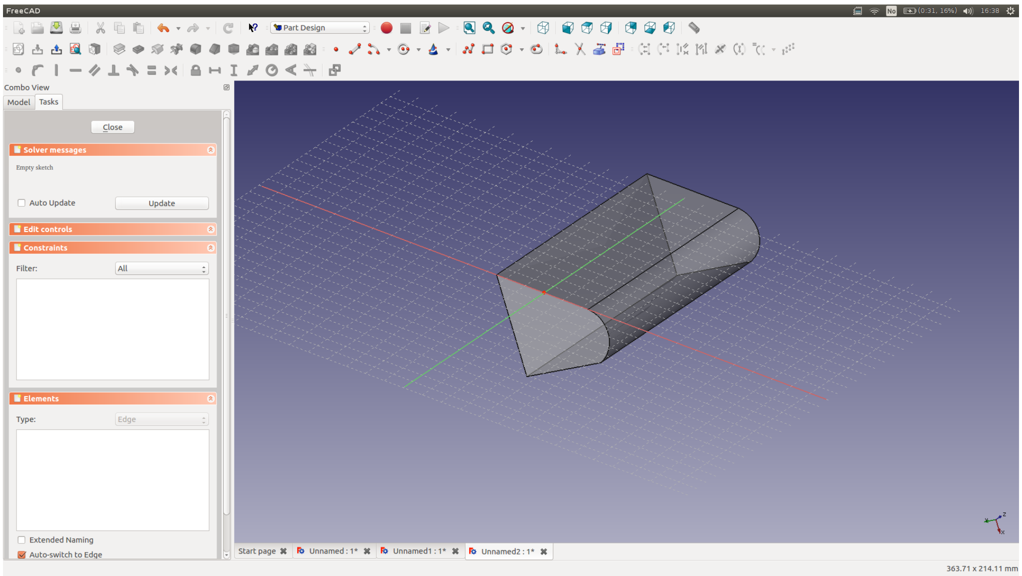

Figure 6: A sketch can be created on any planar face.

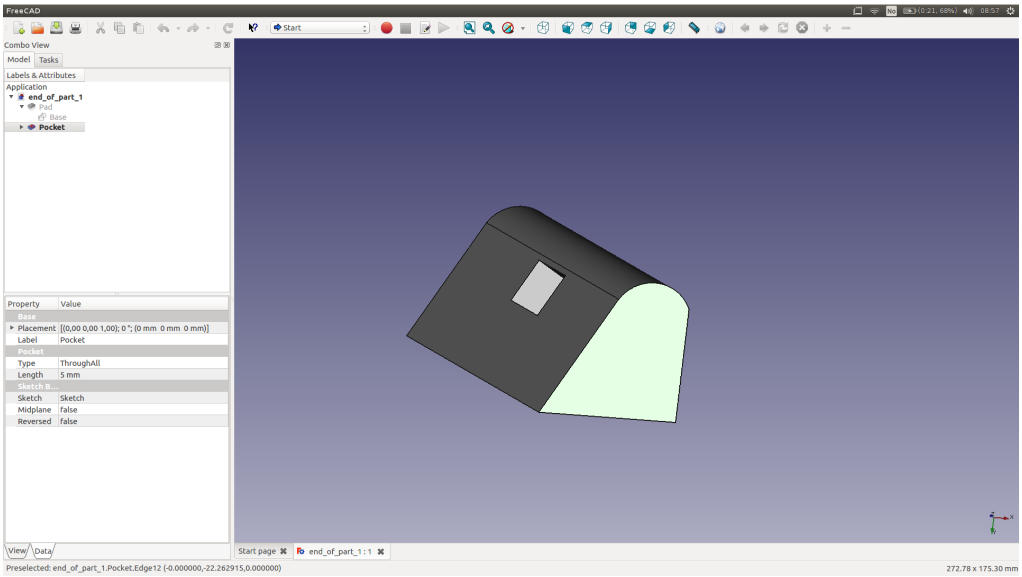

Figure 7: We will build our next sketch on a lateral face of our model.

- Build a sketch similar to the one shown in Fig. 8. Note that the imported external edge is available for connecting points on it and using as a constraint but is not a part of the sketch, so you will need to build a line on top of it to get a closed shape that can be extended to 3D. Close the sketcher. Rename the sketch as **SketchBase**.

- We will extend the geometry to 3D using a more sophisticated tool. Create a new sketch in the XZ plane, i.e. normal to your previous sketch. Build a sketch that looks like Fig 9. Note that the horizontal line is a construction line to set the position of the circle center. Close the sketch. Rename the sketch as **SketchPath**.

- Move to the **Part** Workbench. Click on the **Utility to sweep** button. In the **Sweep** tab, select the **SketchBase** and move it to the **Sweep** tab using the GUI arrow to the right. Then click on **Sweep Path**. Select the vertical line and arc of circle from the **SketchPath** in the graphical view window, holding the ctrl key pressed. Click Done. Click Ok.

- Inspect the shape you obtain in the 3D view. By default, FreeCAD generated an empty shell. Select the **Sweep** you just created in the **Labels & Attibutes** tab, and in the **Data** box set the **Solid** parameter to True. This will update you shape into a solid. It should look like Fig. 10. Here we generated a bulky looking shape, but this utility could be ideal to generate, for example, wings. Have a look in more details at the documentation for both the **Utility to loft** and the **Utility to sweep**.

There are afew more sketcher functions that you should look at by yourself, in particular the **Toggle the toolbar or selected geometry to/from construction mode**, the **Trim an edge with respect to the selected position** and the **Fillet** functions.

8 Example 4: using spreadsheet and formula

A neat functionnality available from FreeCAD v16 and upper is the possibility to use formula to define construction lengths, angles etc and to group the definition of those parameters in

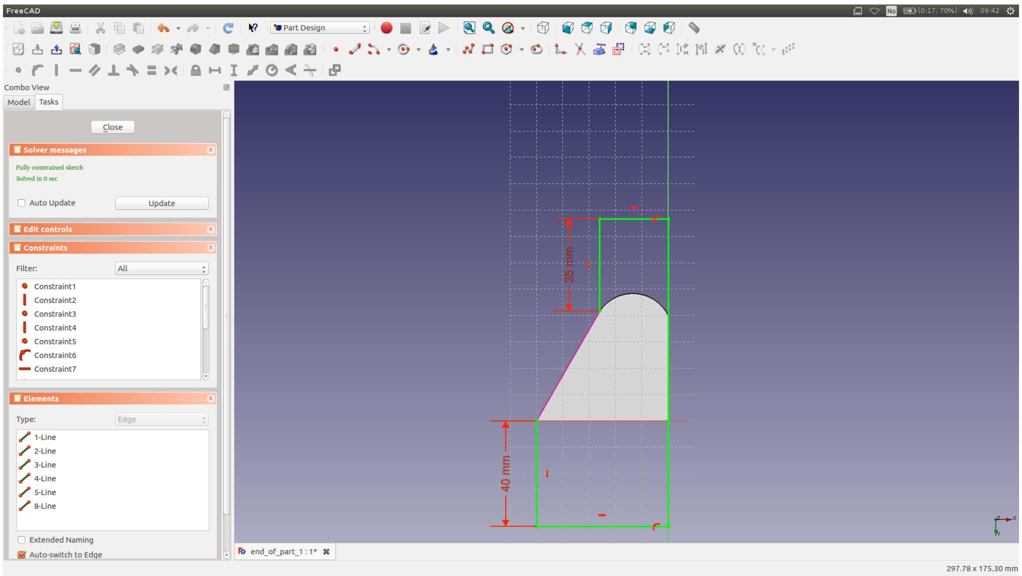

Figure 8

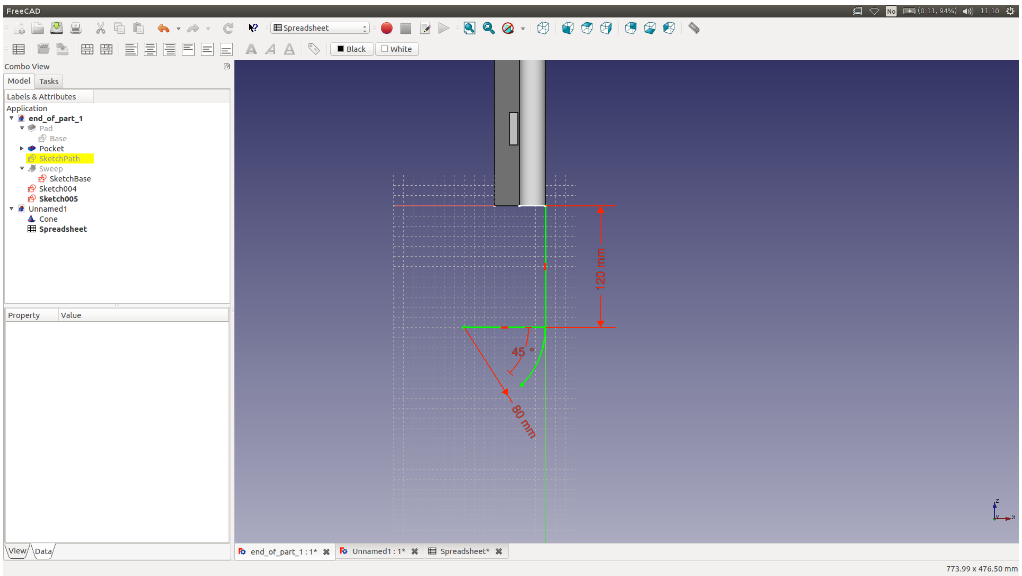

Figure 9

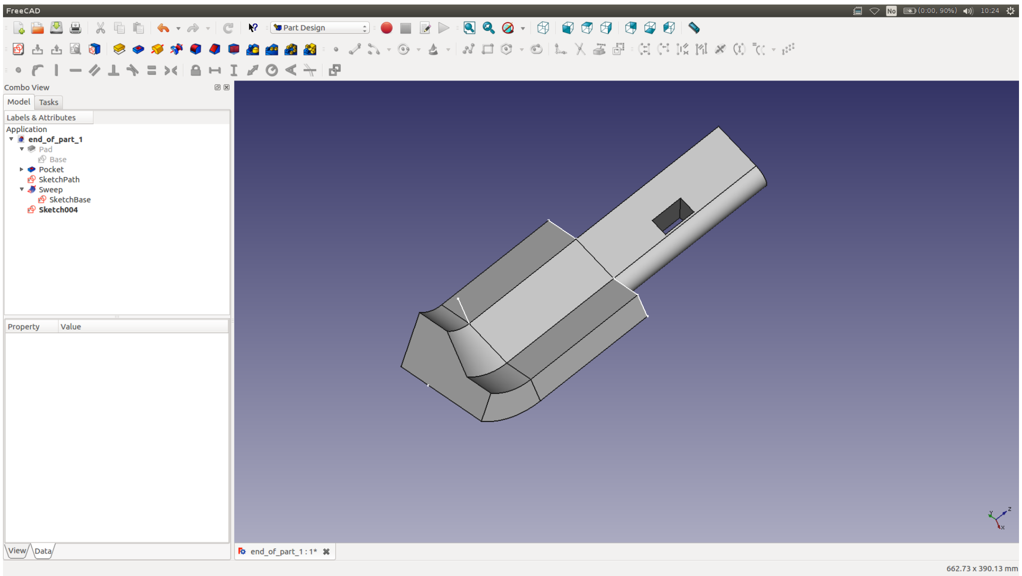

Figure 10

a spreadsheet. This way you can very easily build a parametric CAD model and generate different versions of your object by changing one or more attributes from the spreadsheet. We will illustrate this with a very simple example.

- Open an new project and go to the Part workbench.
- Create a **Cone solid**.
- Move to the **Spreadsheet** workbench and click on the **Create a new speadsheet** button. Open the speadsheet by double clicking on it in the **Labels & Attibutes** tab.
- The speadsheet works like any Excel-like tool. Get to a result similar to the one shown in Fig. 11. This is an engineering tool, **units are important!!** Right click on the cells which each of the values, enter the **Properties** tool. There, go to the **Alias** tab and enter the name of the parameter. The cell will get yellow to indicate that it is linked to an alias. Do this for all cells.
- Now that the aliases are defined, you can use them to set properties in your sketches and solids. In the **Data** tab, left click on the **Radius1** parameter of your Cone. The data filed will get available for setting a value, and you can notice that a small icon looking like a function symbol gets available on the right of the value box. Click on the function symbol and enter the name of the parameter you want to use. Note that you will have to use a Python-like convention, i.e. **Spreadsheet.param_radius_1** will refer to the **param_radius_1** defined in the **Spreadsheet** file. Click Ok. Do this for all three parameters Radius1, Radius2 and Height of the cylinder.
- Change some parameters in the spreadsheet. This will automatically update the parameters in your data field and the 3D view of your cone.

Here we used a speadsheet and formula in a very simple case, but this is a powerful feature when you get to a complex geometry with many parameters. The formula can use all usual mathematical functions, have a look at the documentation for more information.

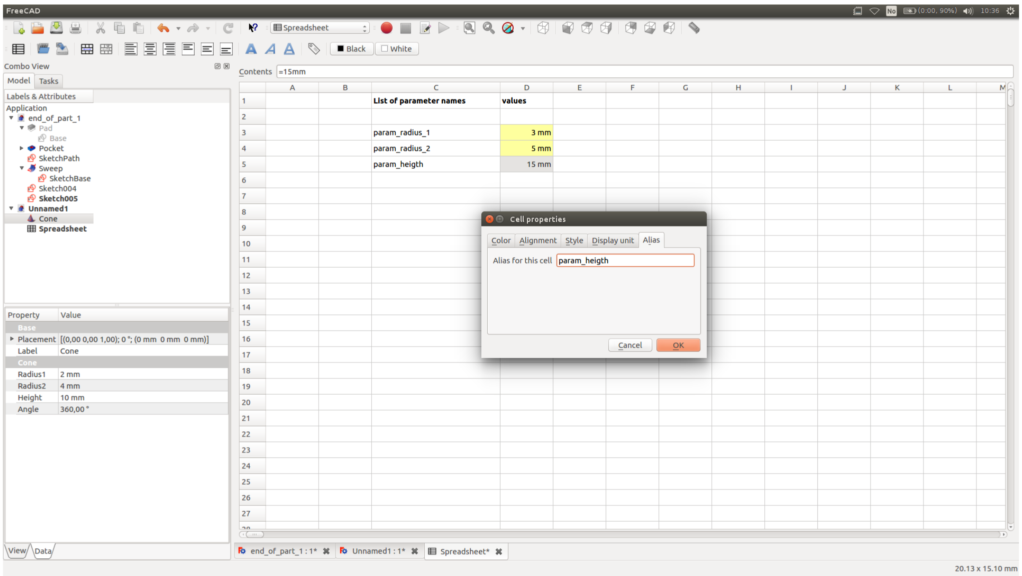

Figure 11

9 Going further

You know have used some of the most important features of FreeCAD including the **Part**, **Part Design** and **Sketcher** workbenches, as well as **spreadsheet** for easily controlling the parameters of your model. Often in CAD there are many ways of building a given geometry, and you should practise to get better. Google is your friend when working on CAD: search the internet for FreeCAD tutorials, videos, forum etc.

Since repetition is the mother of any learning, I strongly advise you to take a look at the FreeCAD manual:

`https://www.gitbook.com/book/yorikvanhavre/a-freecad-manual/details`

And the following on line tutorials:

Bram de Vries FreeCAD tutorials on Youtube:

`https://www.youtube.com/playlist?list=PLVrqzKSVAONHYXc9G9y9wHWus3ManpzUn`

FreeCAD lessons for beginners channel on Youtube:

`https://www.youtube.com/playlist?list=PL6fZ68Cq3L8k0JhxnIVjZQN26cn9idJrj`