FreeCAD Part Design 101

From sketch to printable 3D part

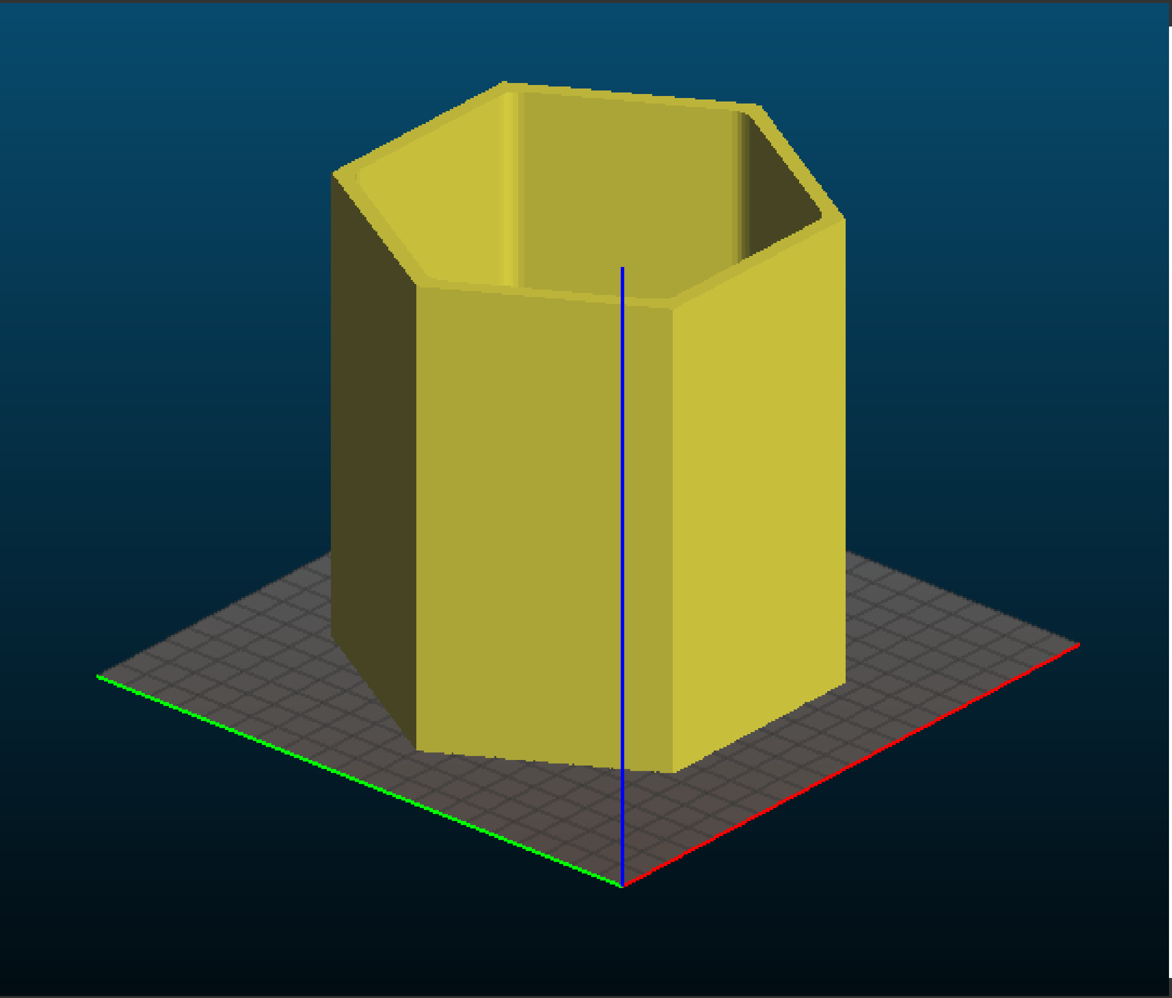
If you haven't yet, Install it quick!

Kai

Who Am I

- Karel “Kai” Trachet
- InfoSec by day
- I love repairing stuff
- Learned to use FreeCAD recently
- No previous CAD experience

What are we going to do



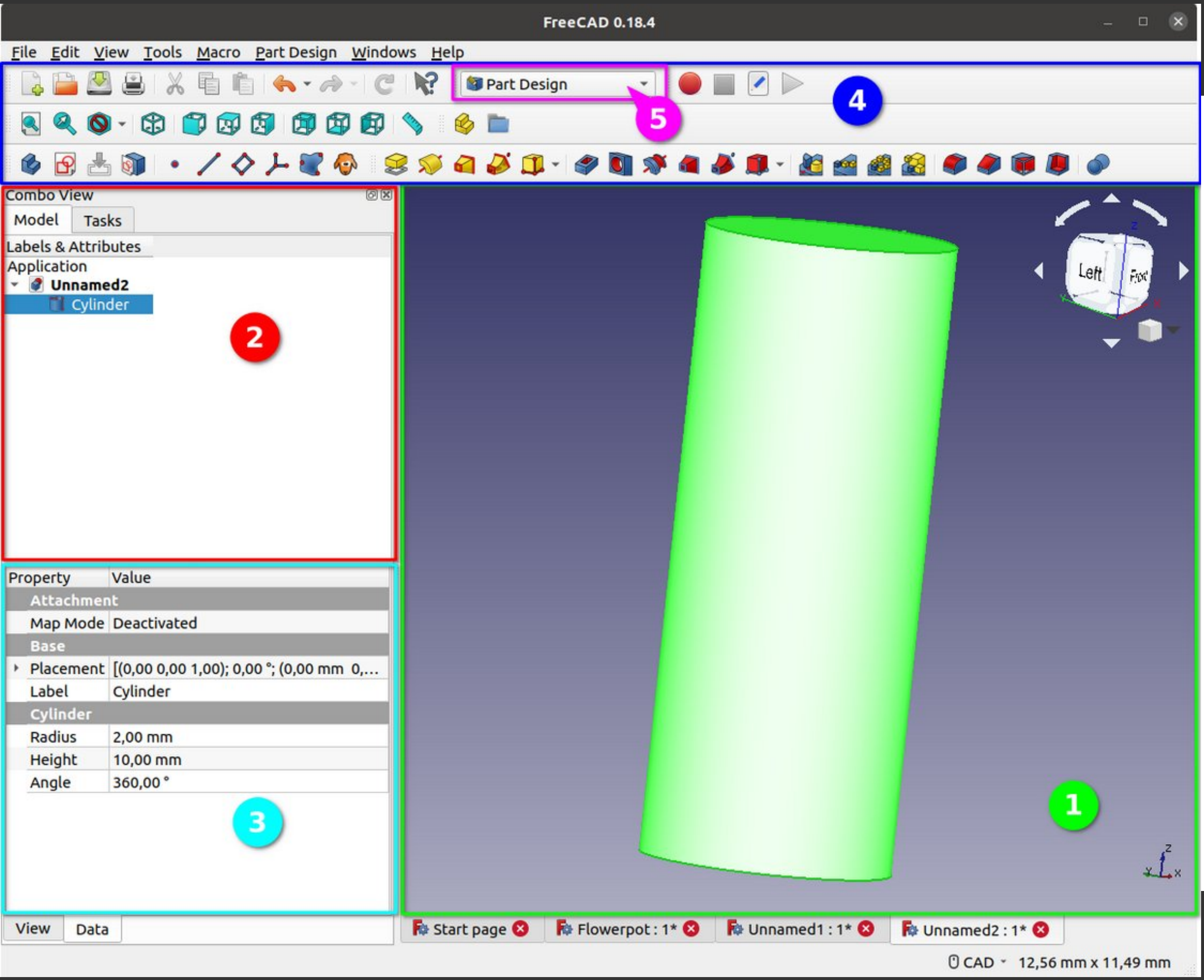
- 1 hour(!)
- FreeCAD is Huuuuuge!!!
- Focus on part-design
- Some tools
- Simple design
- Hexagonial Flowerpot!

FreeCAD

From www.freecadweb.org:

FreeCAD is a 3D parametric modeling application. It is primarily made for mechanical design, but also serves all other uses where you need to model 3D objects with precision and control over modeling history.

The Interface



- 1)The 3D view
- 2)The Tree View
- 3)Property Editor
- 4)Toolbar Area
- 5)Workbench

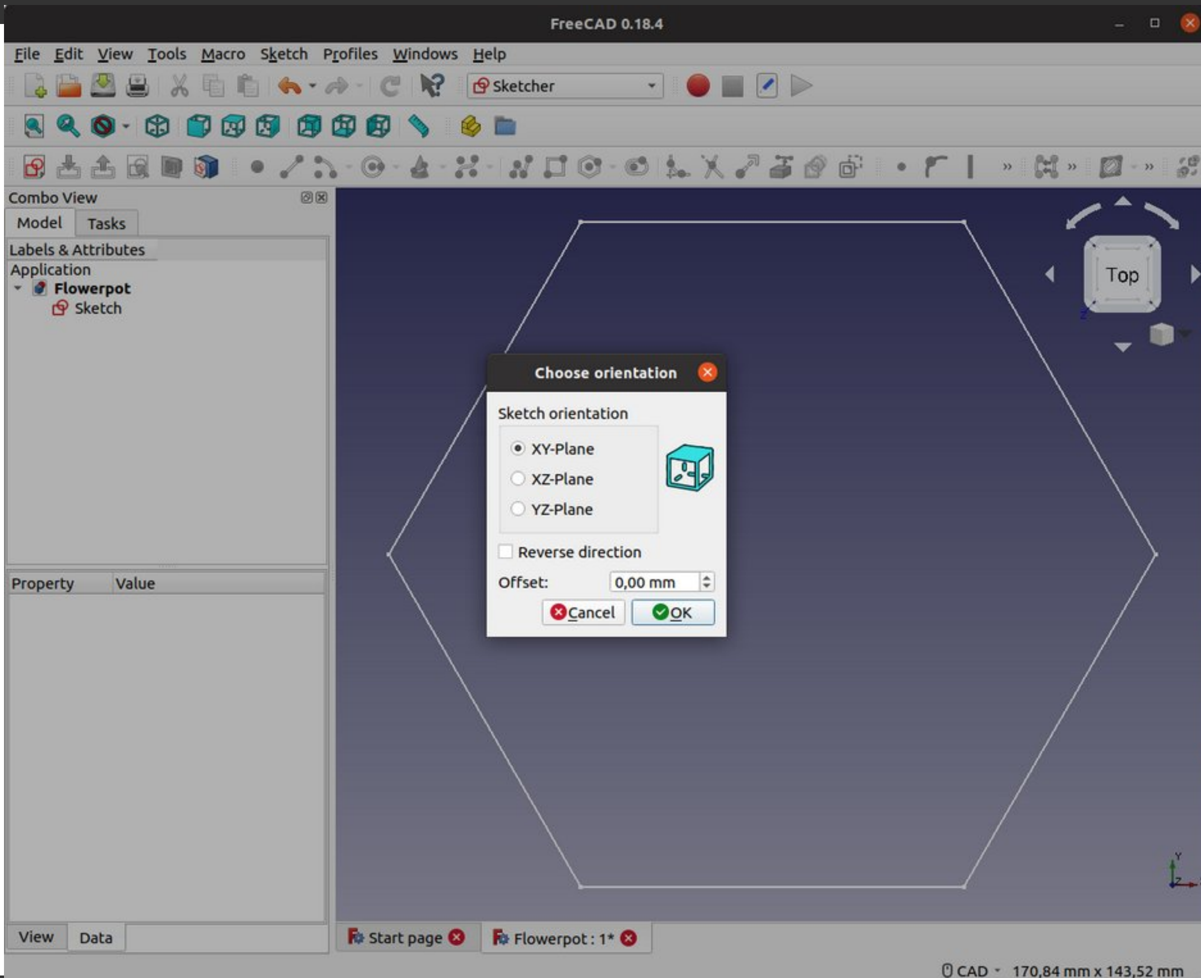
- Workbench is a set of tools grouped together for a certain task
- FreeCAD has a bunch
- More available as add-ons

- We'll be using:
  - Part Design
  - Sketcher

Mouse controls in 3D View

- Left Mouse => Select
- (CTRL + Right Mouse) or (Middle Mouse) => Pan
- SHIFT + Right Mouse => Rotate
- Scroll wheel => Zoom

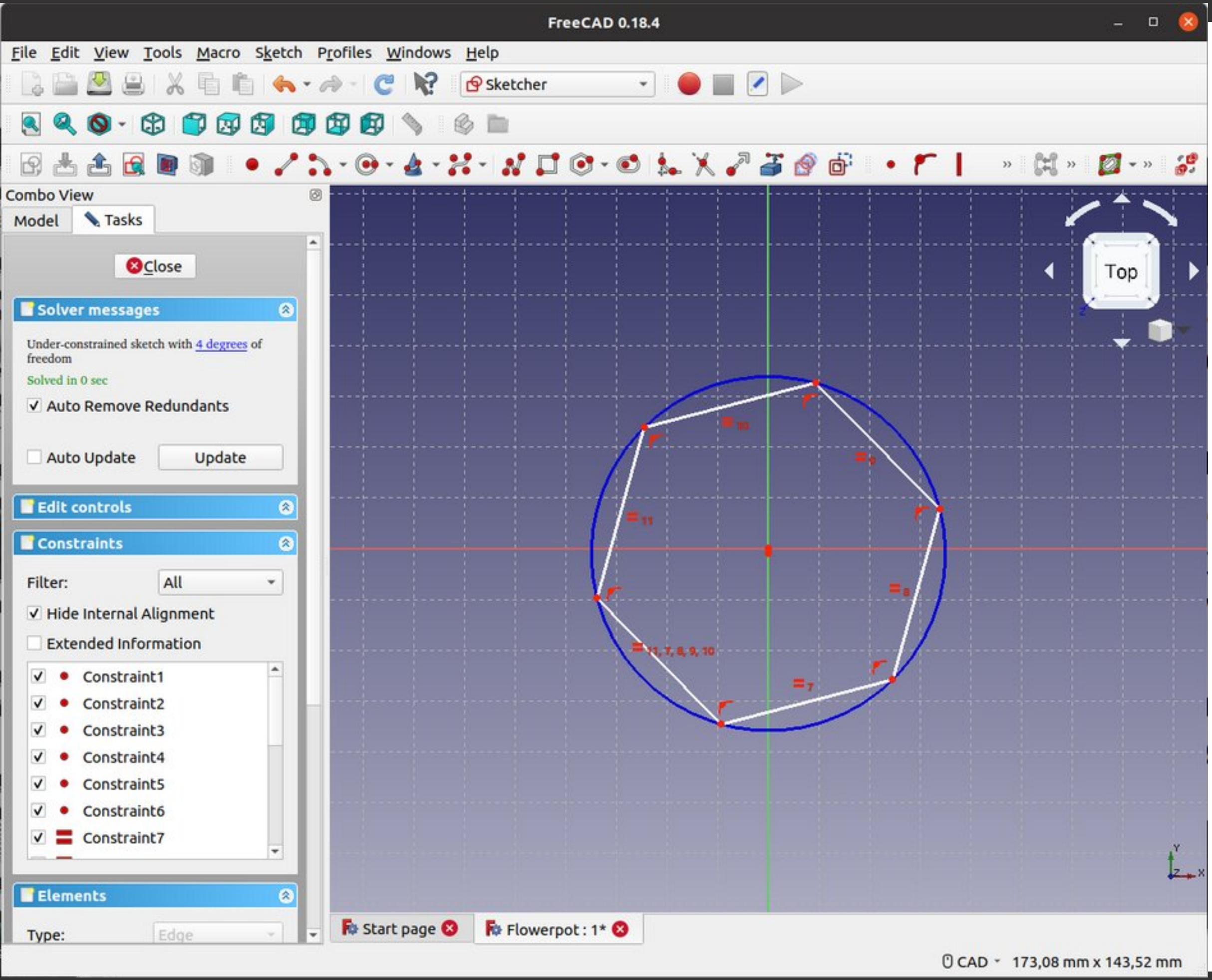
Sketch Workbench

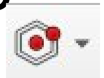


- Before going 3D we will first create a 2D sketch
- This is done in the sketch workbench
  - Click “New Sketch”
  - Choose “xy-Plane” as orientation

Out of Scope Note: The Sketch workbench is not made for Technical Drawings => Use Draft workbench instead

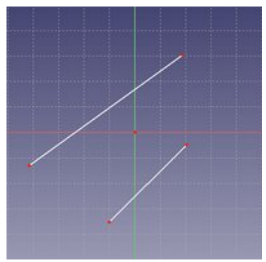
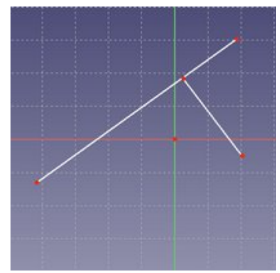
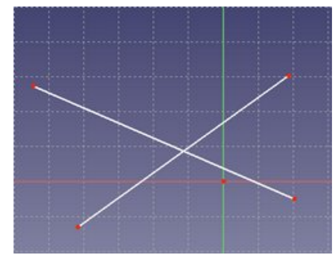
Creating a shape



- Click the “create a regular polygon” drop down button 
- Click “hexagon”
- Click and drag the shape in the 3D view around the 0,0 coordinate to draw the shape
  - Don’t worry about the size or position yet

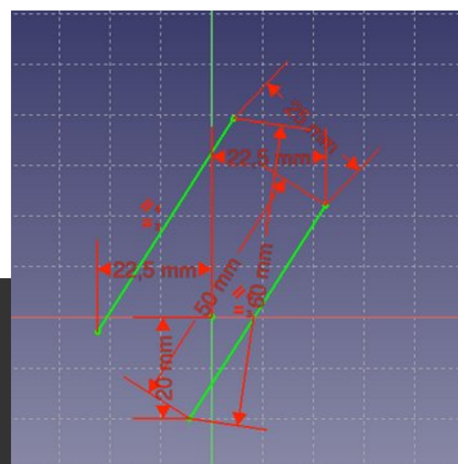
- A constraint is a logical rule that defines the relationship between 2 elements (points, vertices, point of origin).
- A fully constraint sketch has all it’s elements constrained so there is no ambiguity in the way it’s drawn.

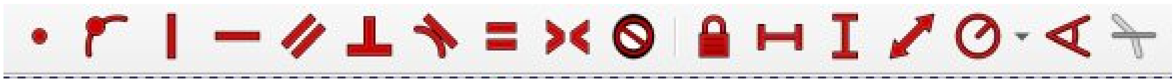
2 unconstrained lines:

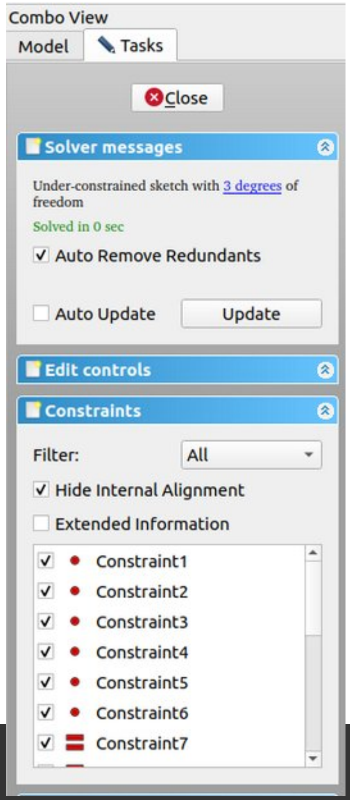




...

2 constrained lines:

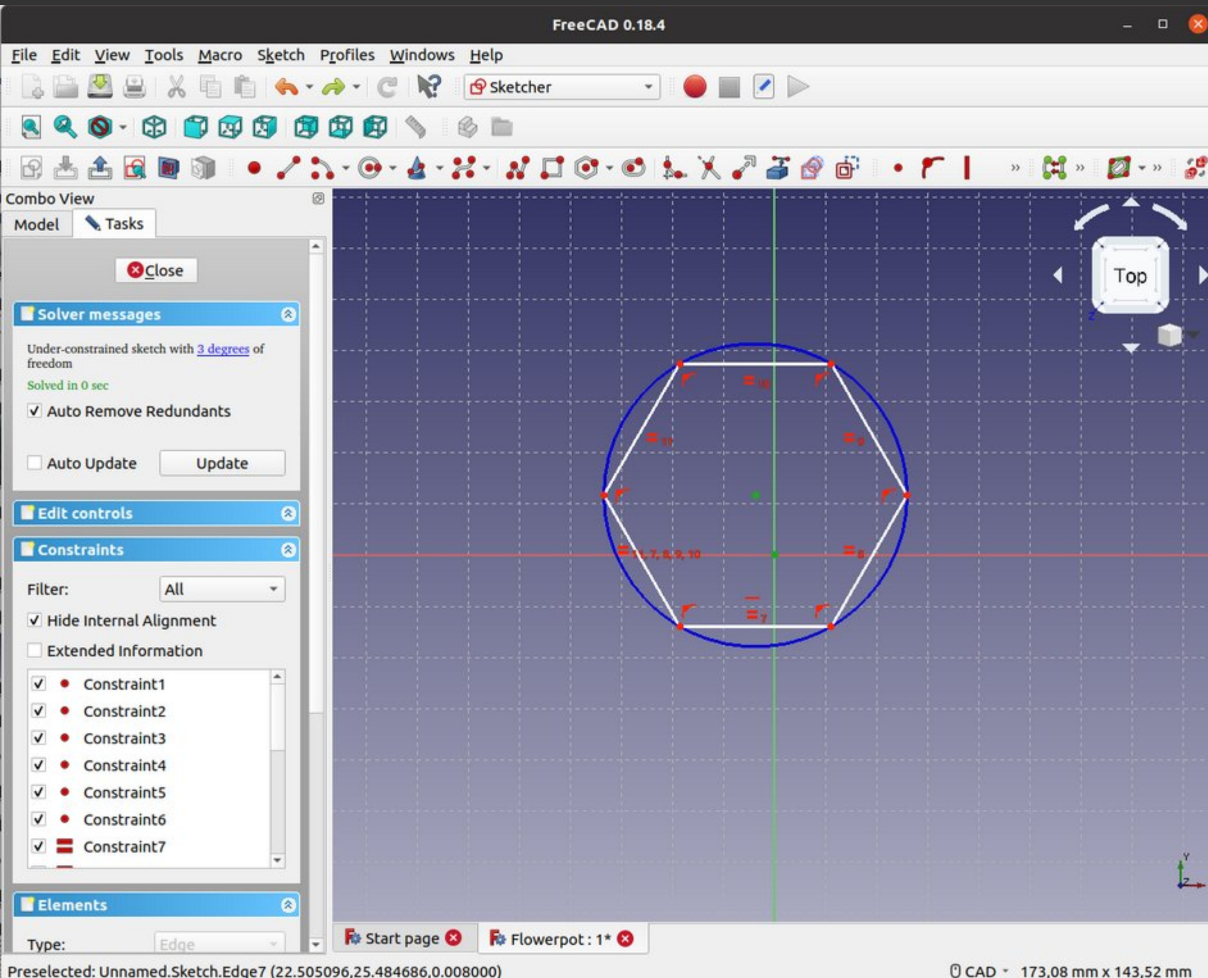


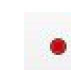


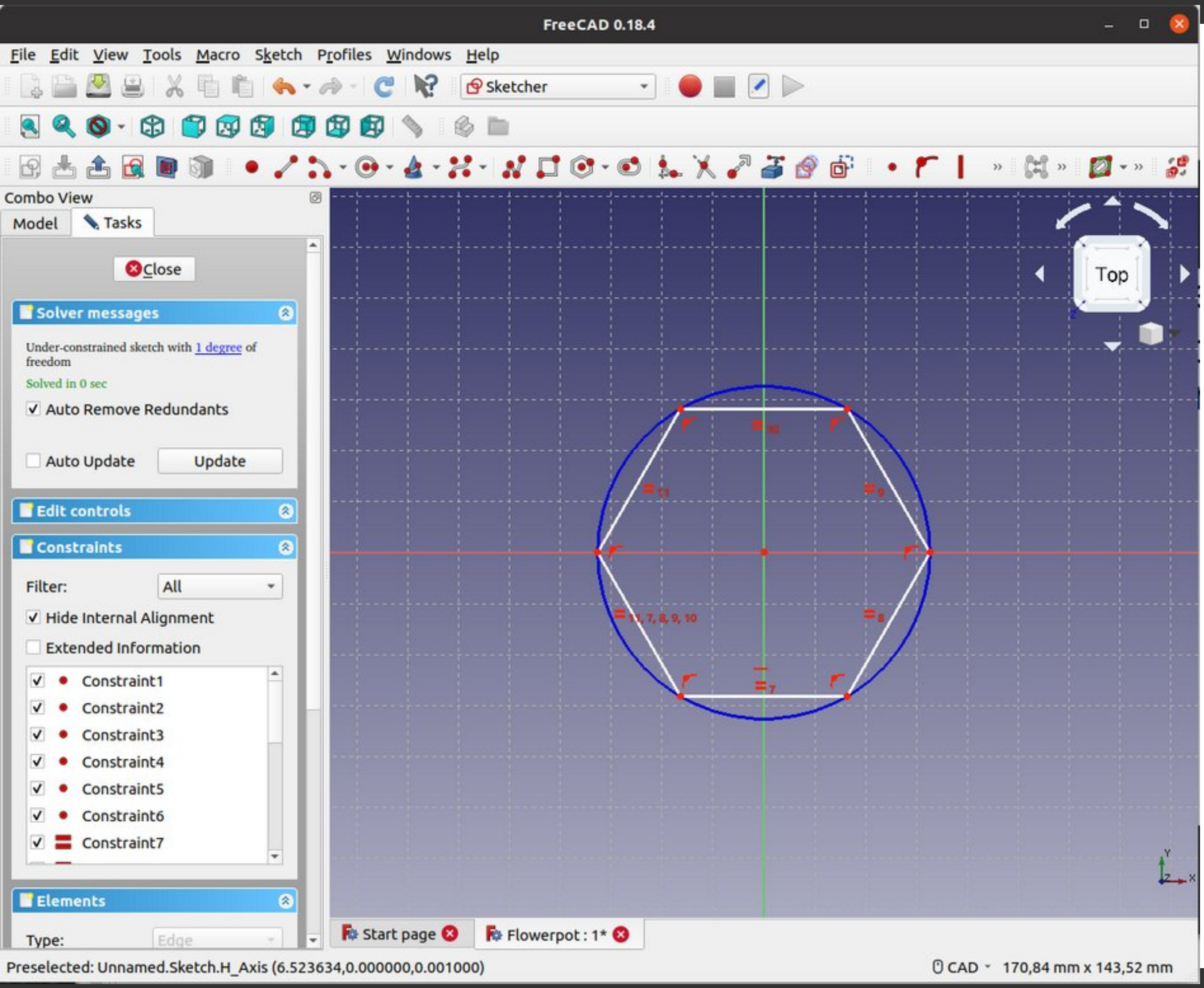


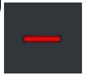
- The shape can still be moved, stretched and rotated
- We need to constrain the sketch
  - First we'll add geometric constraints
  - Then dimensional constraints
- Some are already defined for the hexagon

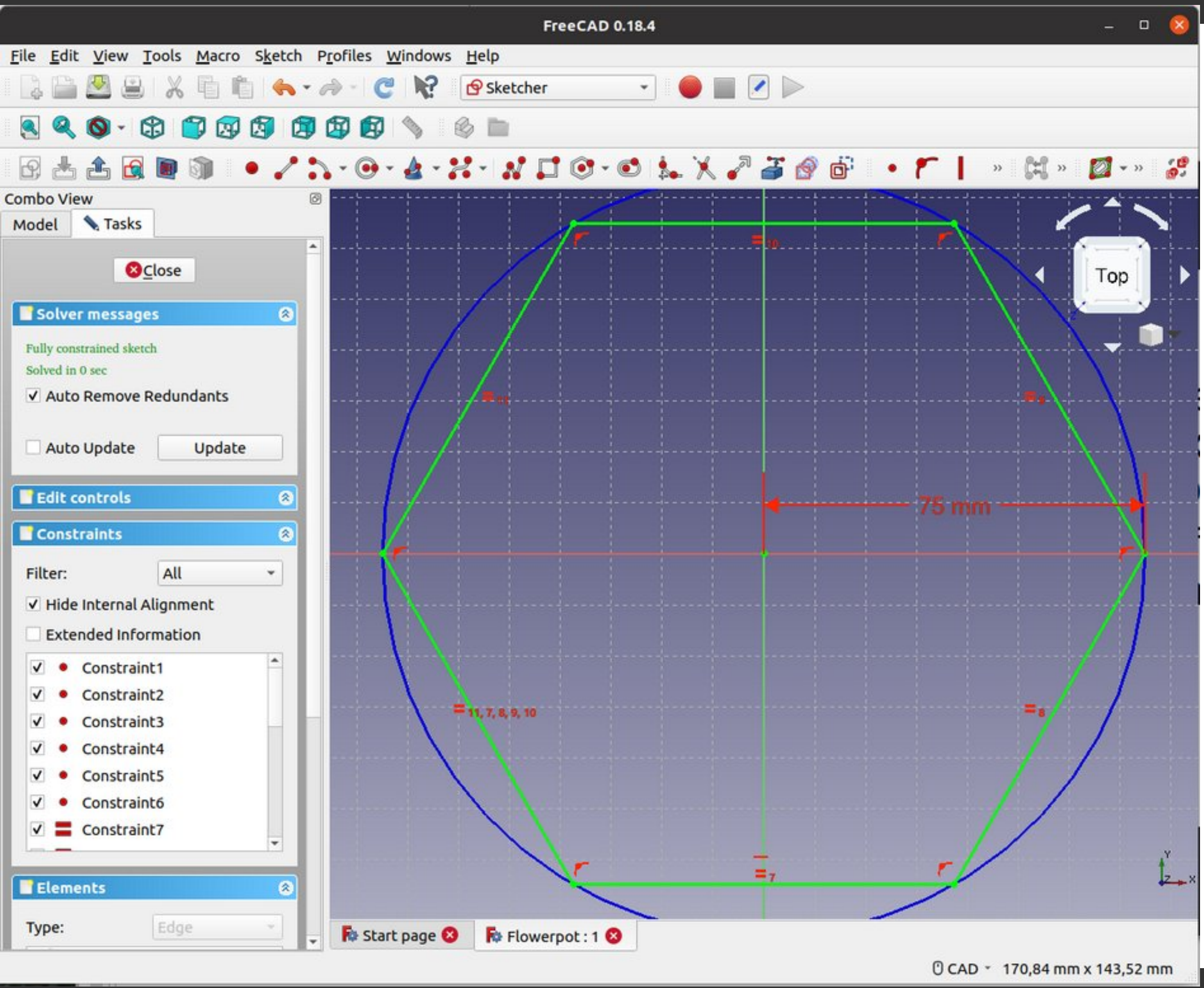
Position



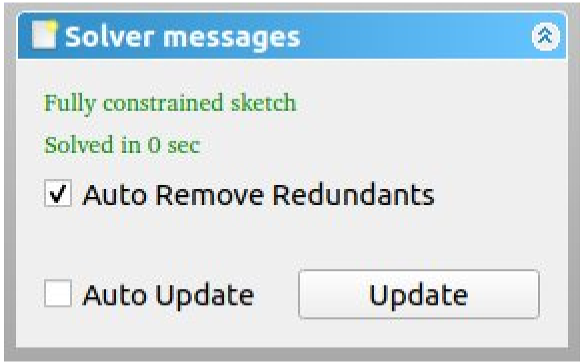
- Center the sketch to the 0,0 point
- First select the central point of the shape and the 0,0 coordinate by clicking on them
  - Click on an empty part of the 3D view to deselect everything
- Constrain them by clicking on “create coincident constraint” 



- Select one of the sides
- Constrain with “create a horizontal constraint” 



- We are going to define the radius
- Select one of the 2 vertices on the x axis and the center point
- Constrain with “fix the horizontal distance”
- Insert the radius you want for the flowerpot
  - ie. 75mm



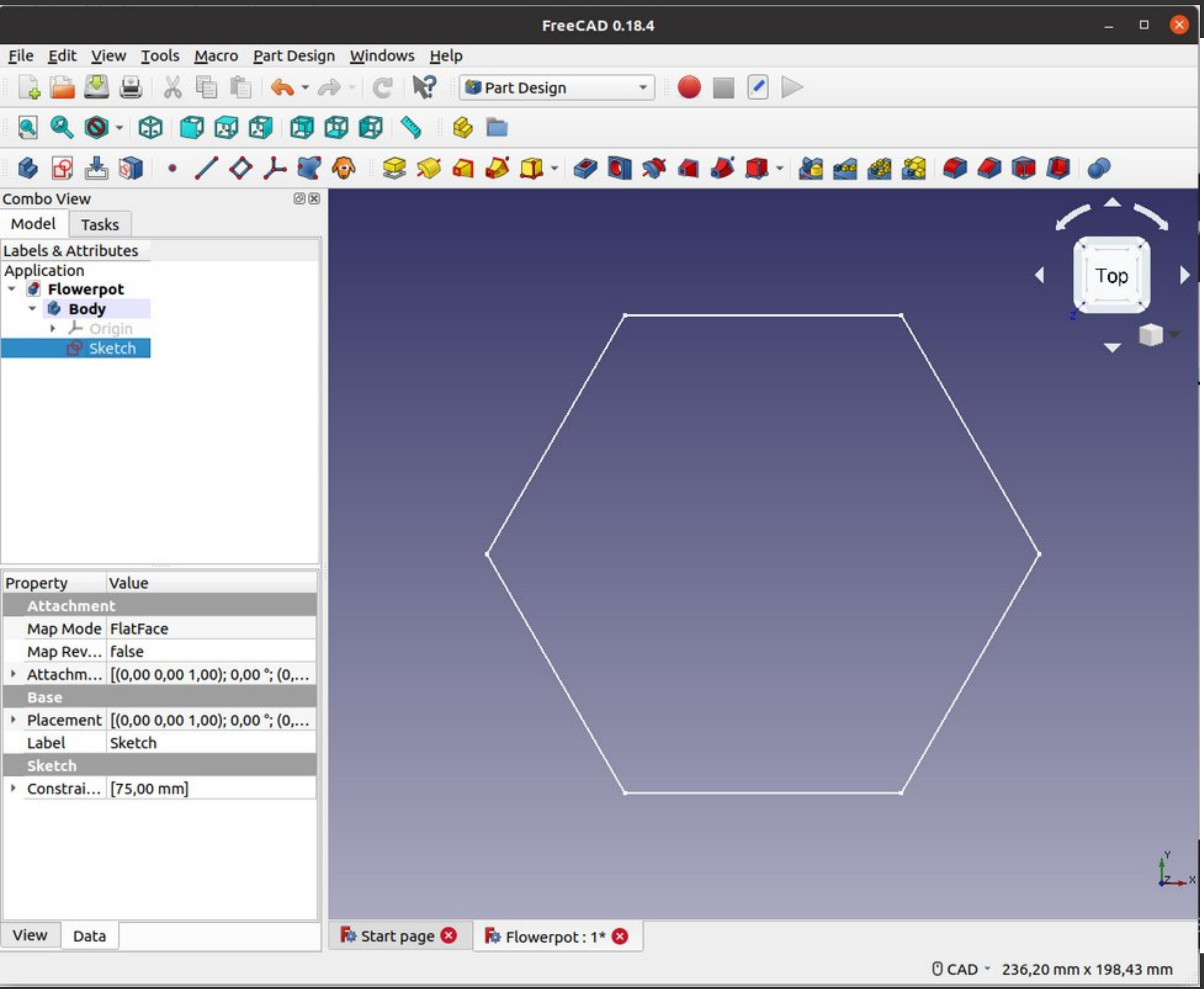
- The sketch is now fully constrained
- Close the sketch task 

- We can now make our 3D shape

Part Design Workbench

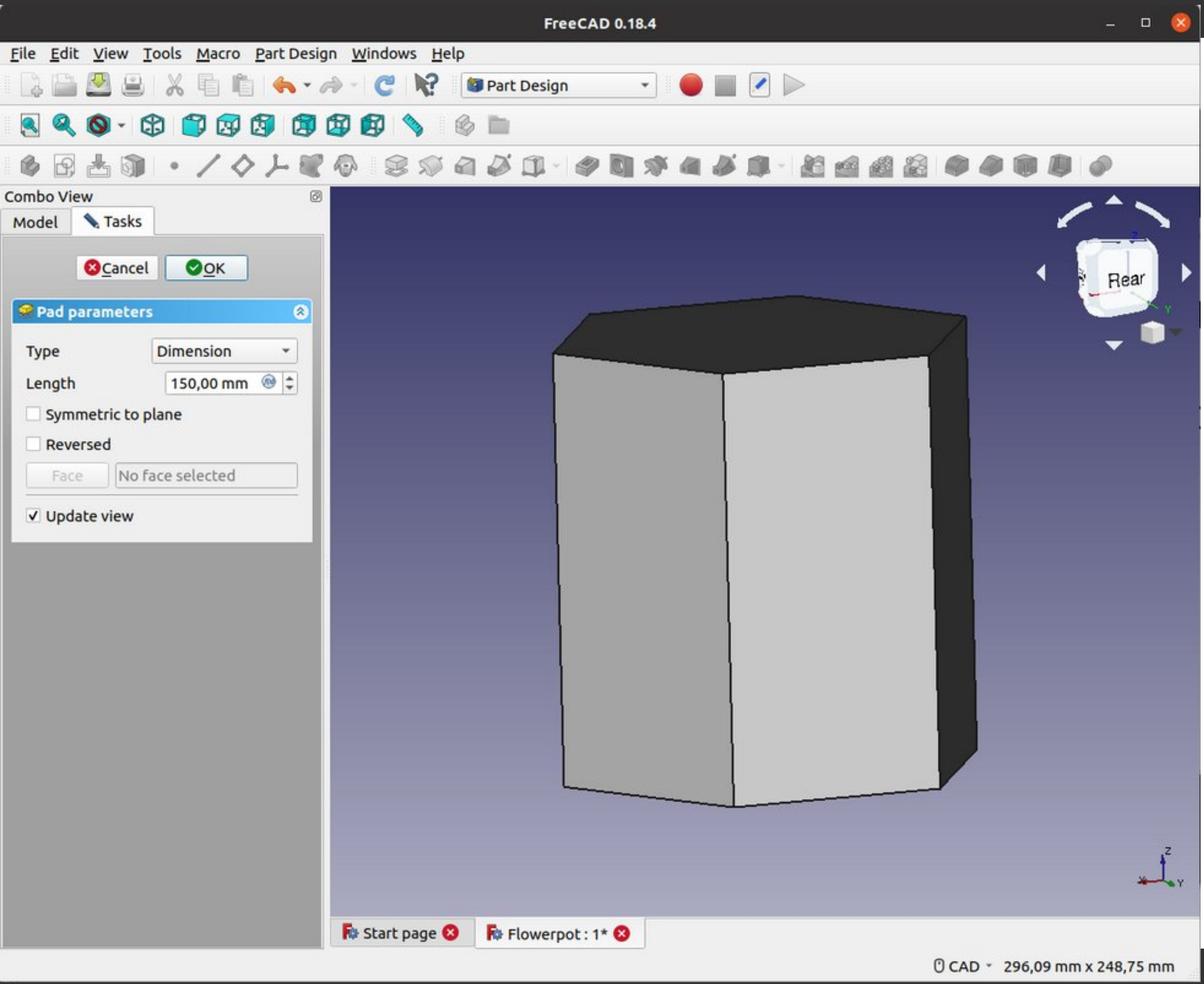
- Change to “Part Design Workbench”

- Datum Tools
- Additive tools
- Subtractive tools
- Transformation tool
- Dress-up tools



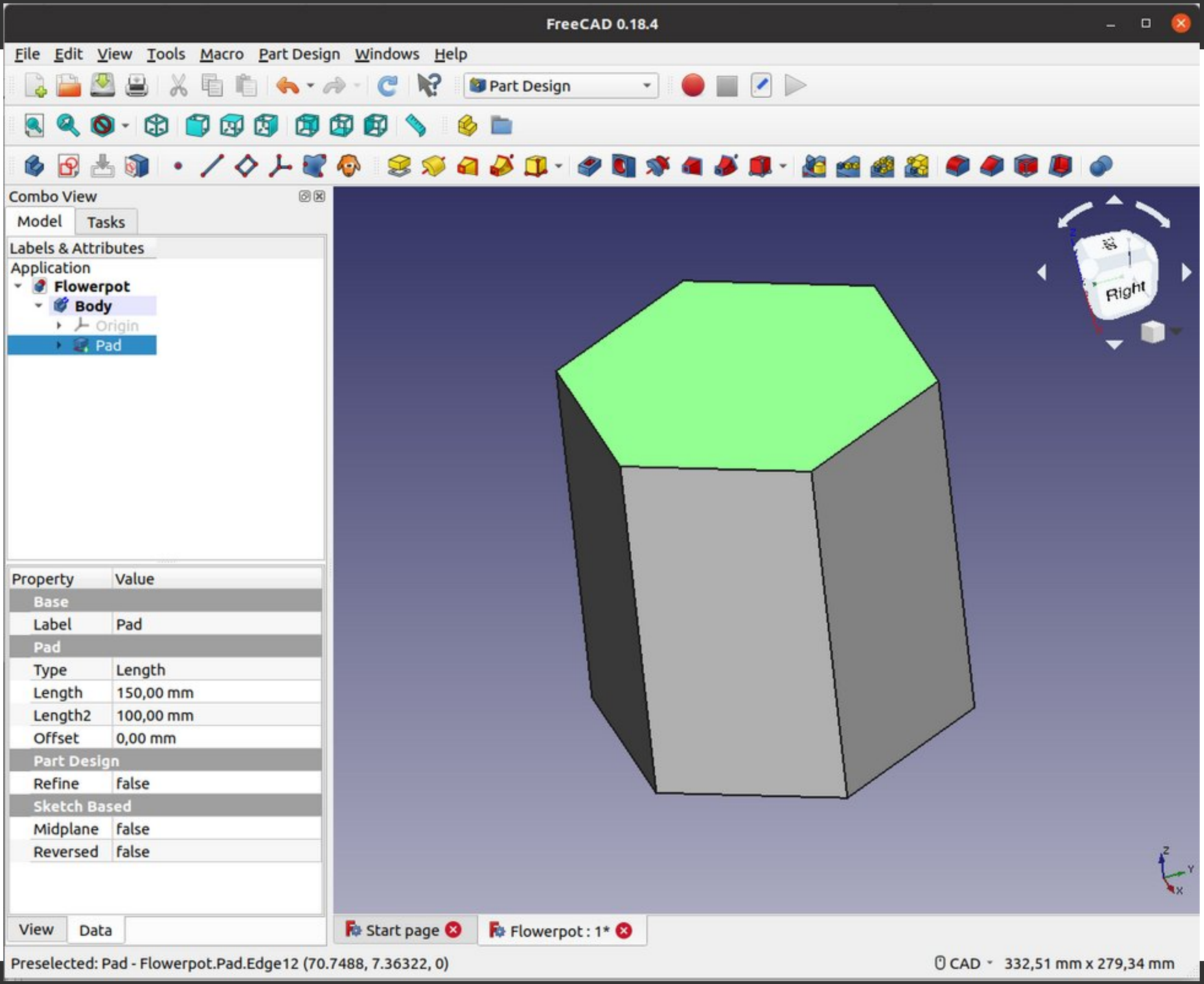
- With the sketch selected click “Create a new body and make it active” 
- Choose the xy-Plane
- The sketch should be under the body in the Tree-view

Padding



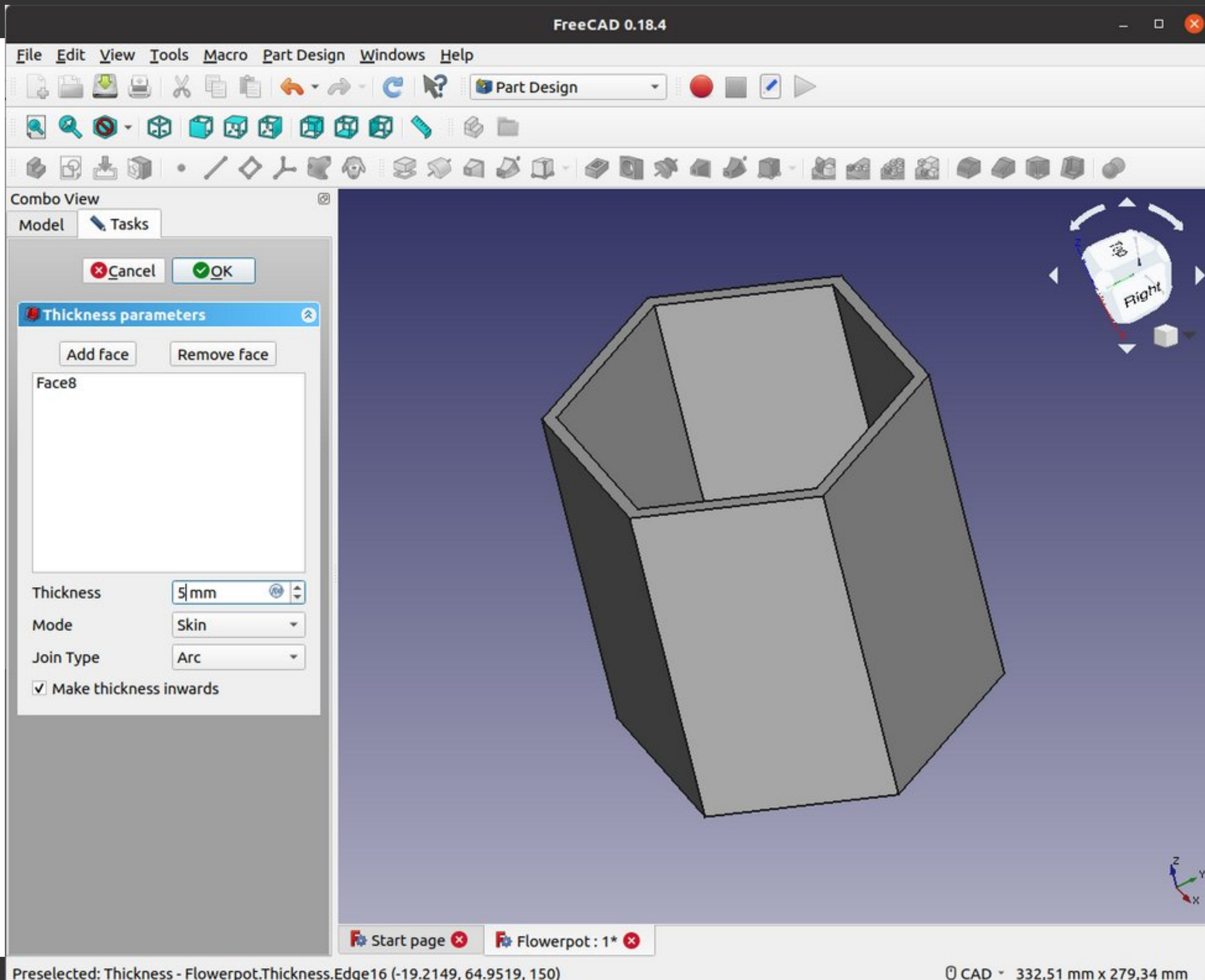
- Let’s go 3D
- Select the sketch in the Tree-View
- Click “Pad a selected sketch” 
- Fill in the height (length)
  - ie. 150mm
- Click “OK” 

Make the part hollow

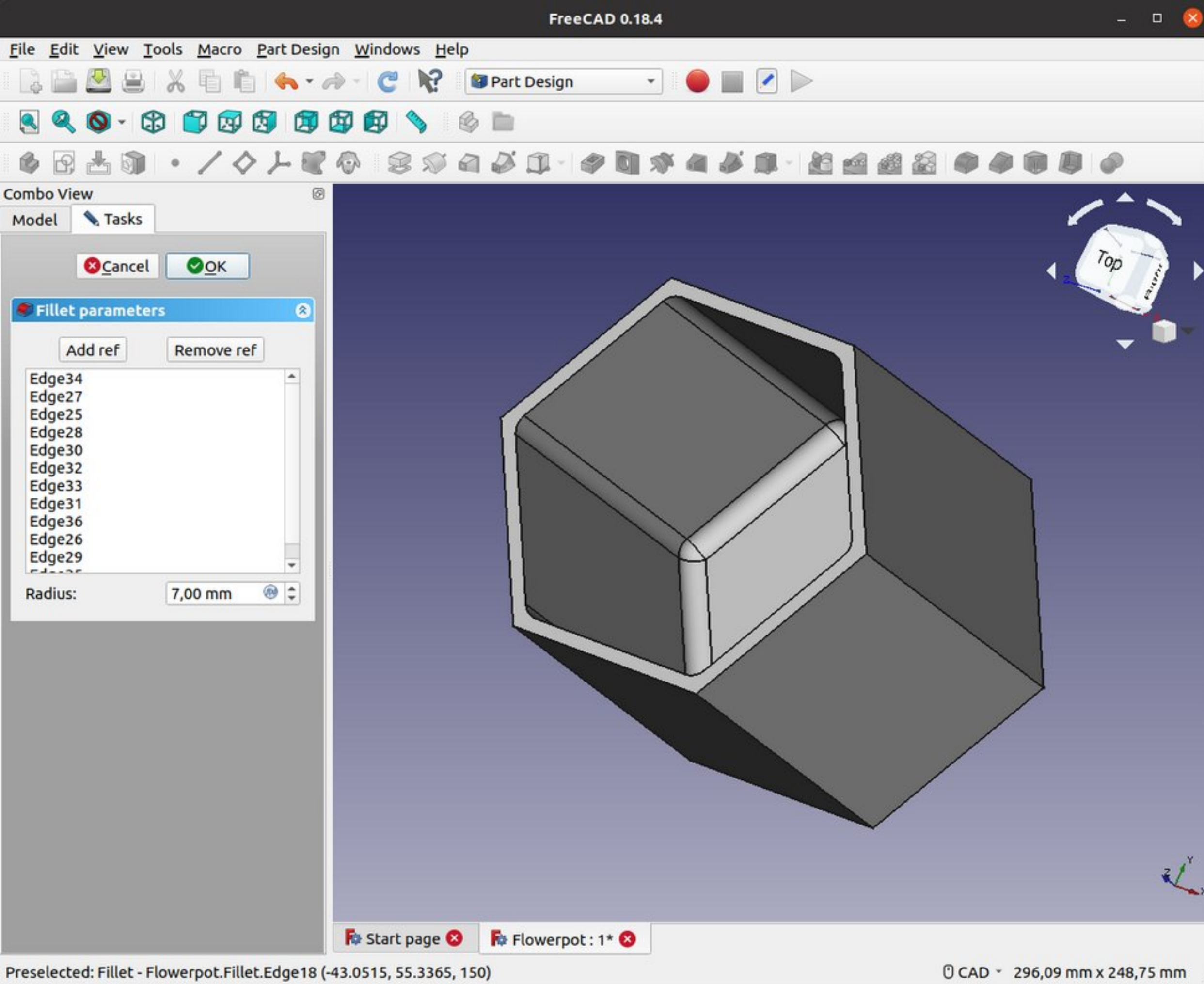


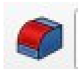
- 2 ways of doing this
  - Sketch + Pocket
  - Make a thick solid
- Let Freecad do the work!
- Click “Make a thick solid”

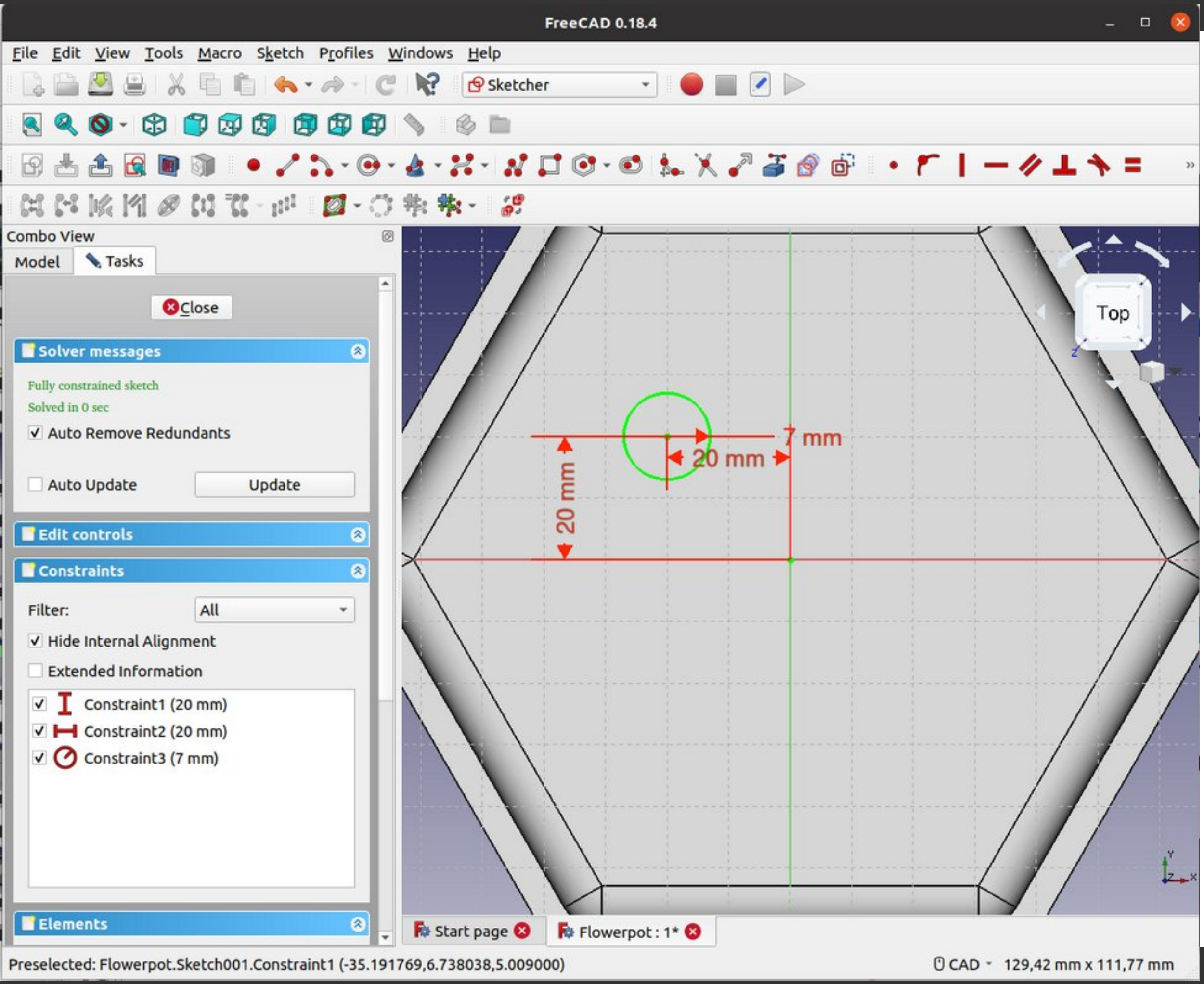
Make the part hollow

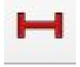


- Let's take the easy route
- Select the top face of the Hexagonal Prism
- Click "Make a thick solid" 
  - Thickness: ie. 5mm
  - Check "make thickness inwards"
  - Click "OK" 

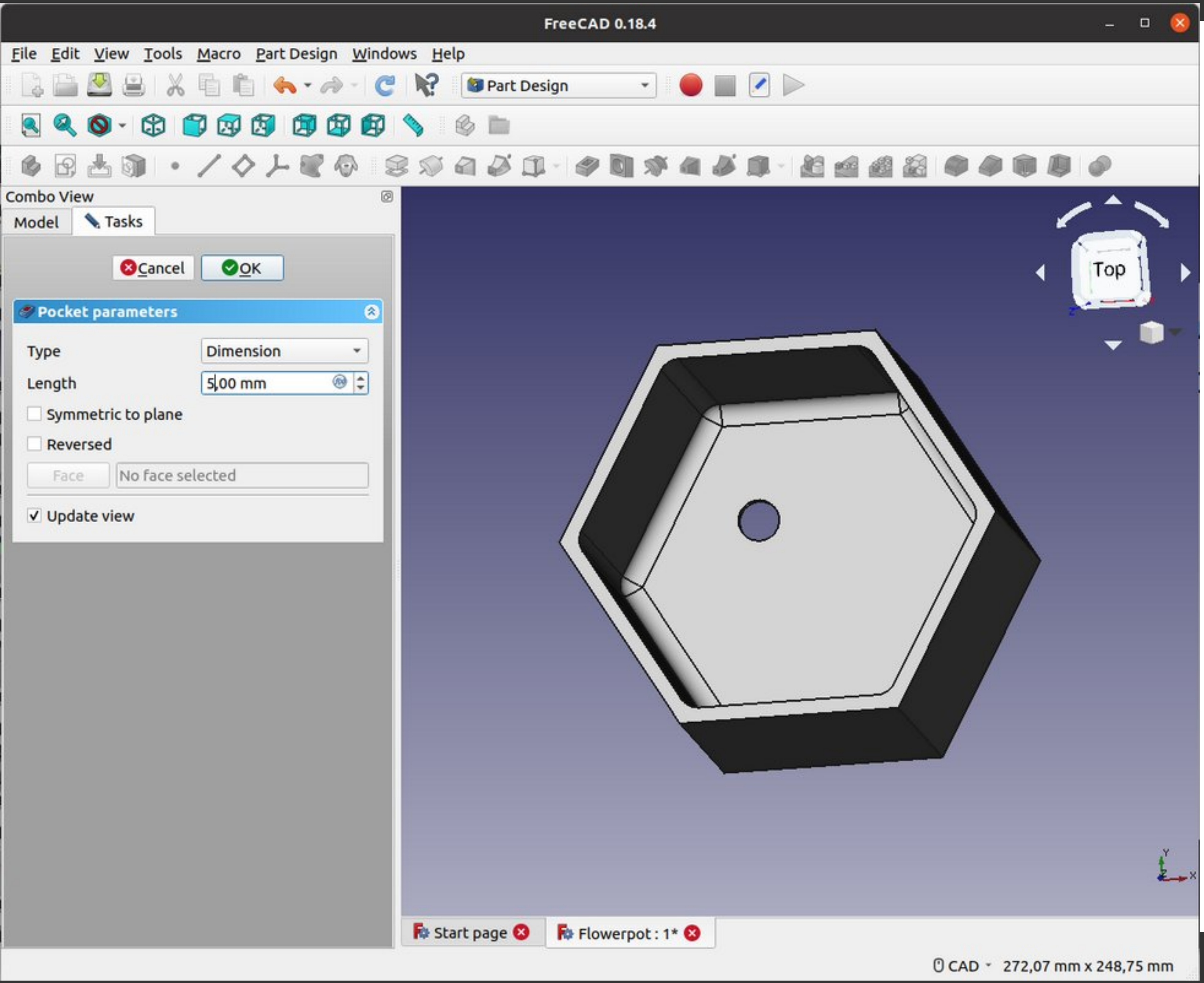


- Let’s smooth out the inner corners
- Select all the vertices on the inside of the pot
  - This can be a bit tricky
- Click “Make a fillet” 
  - Set a radius (ie. 7mm)
  - Click “OK” 

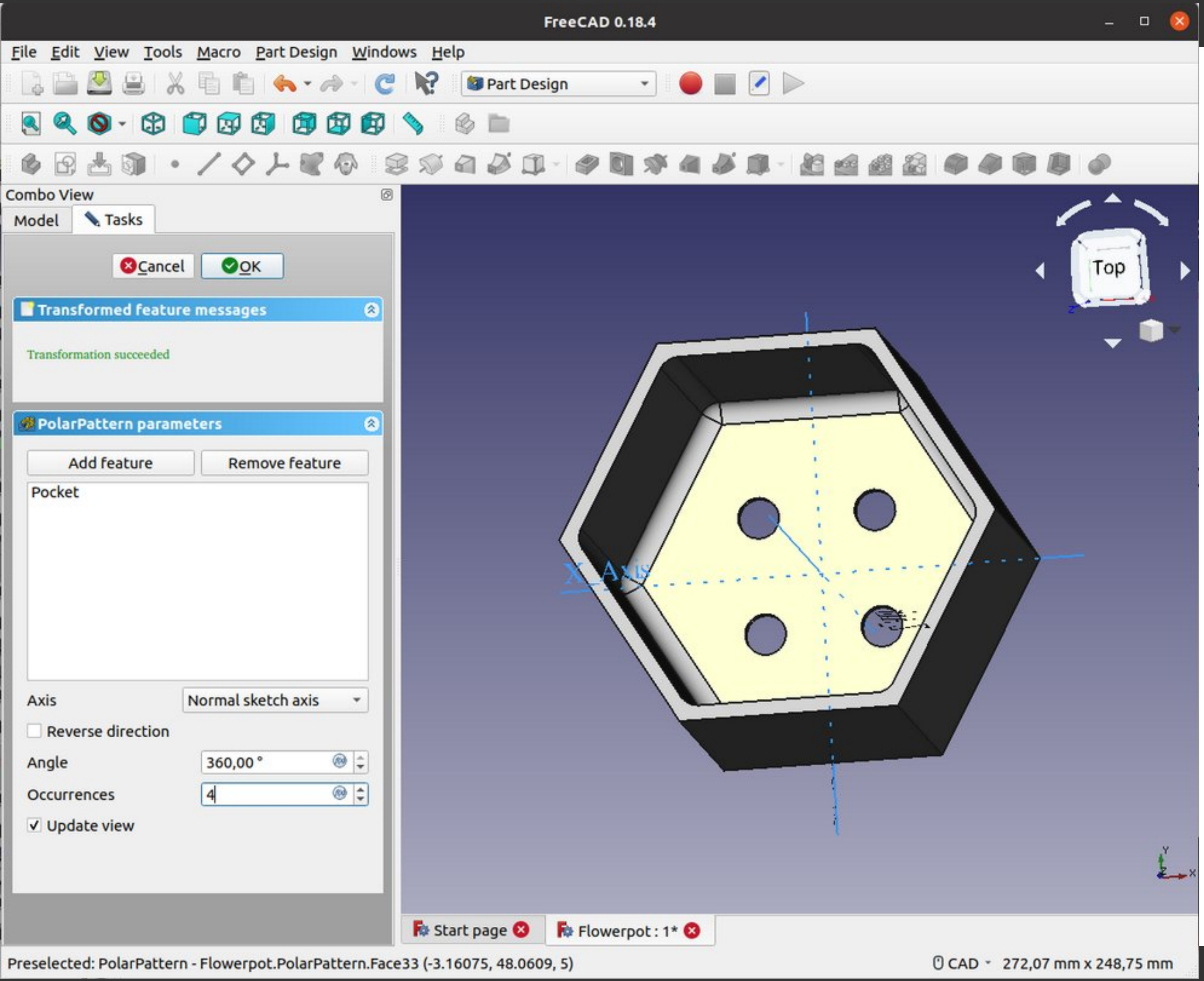


- We'll add drainage holes
- Select the bottom face
- Click "Create a new sketch" 
- Make a circle:
  - Add a vertical  and horizontal  distance constraint on the axes
  - Add a radius constraint (ie. 7mm) 

Pocket 2



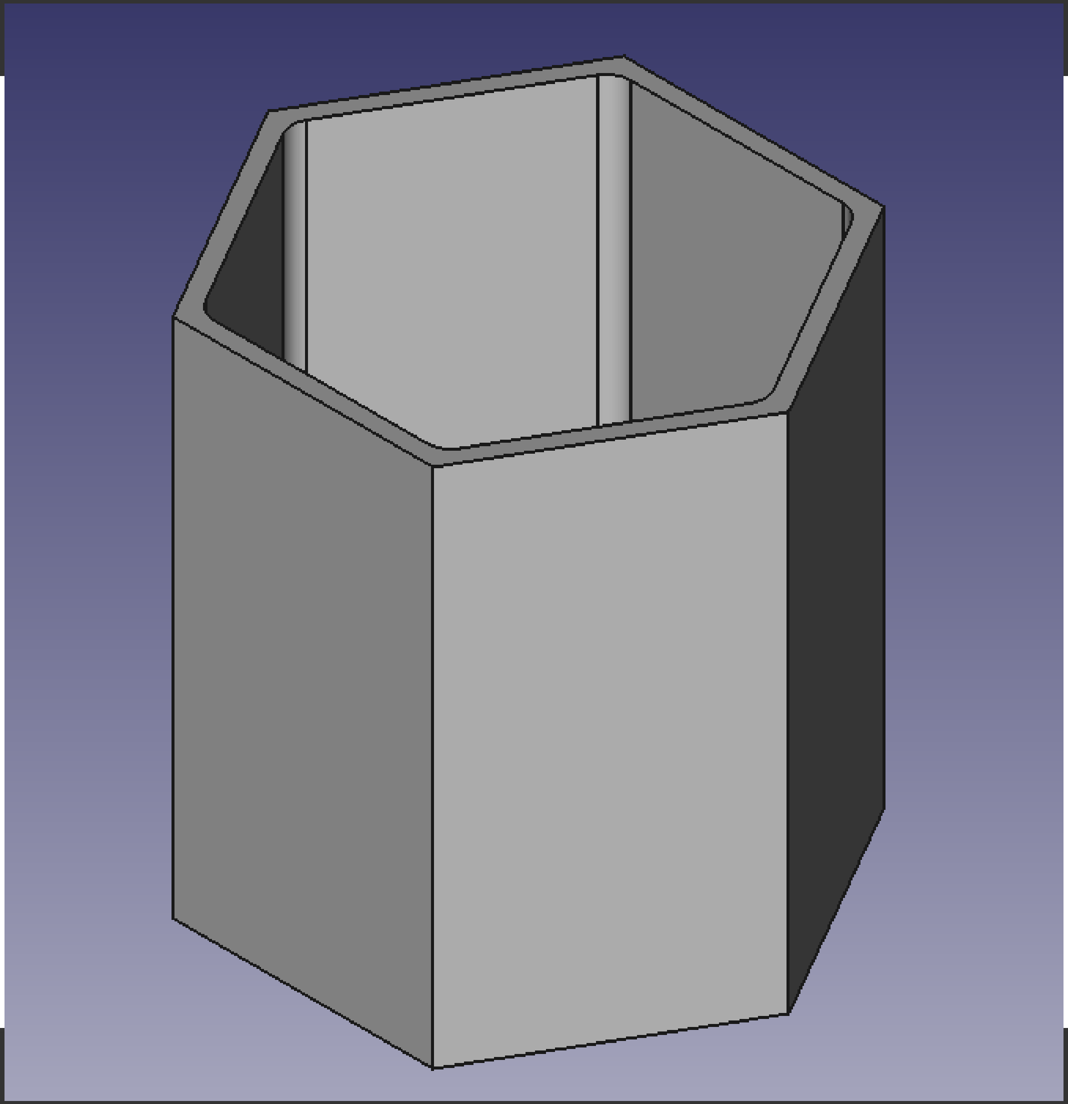
- Click “Create a Pocket”
- A hole should be created



- But we want 4!
- Click “Create a Polar Pattern Feature” 
  - Set occurrences to 4
- Click ”OK” 

Optional additions

- Try to add a design on one of the sides (sketch + pocket).
- ...or try to improve the design in some other way



- And we’re done!
- To print we need an STL(mesh) file
- Export with Ctrl + E or “File” >> “Export”
- Save as stl
- Import in your slicer software and start printing

Learn More

- www.freecadweb.org/Getting_started
- http://www.help-freecad-jpg87.fr/index.php
- Great video tutorials available on not so free, well known platforms