

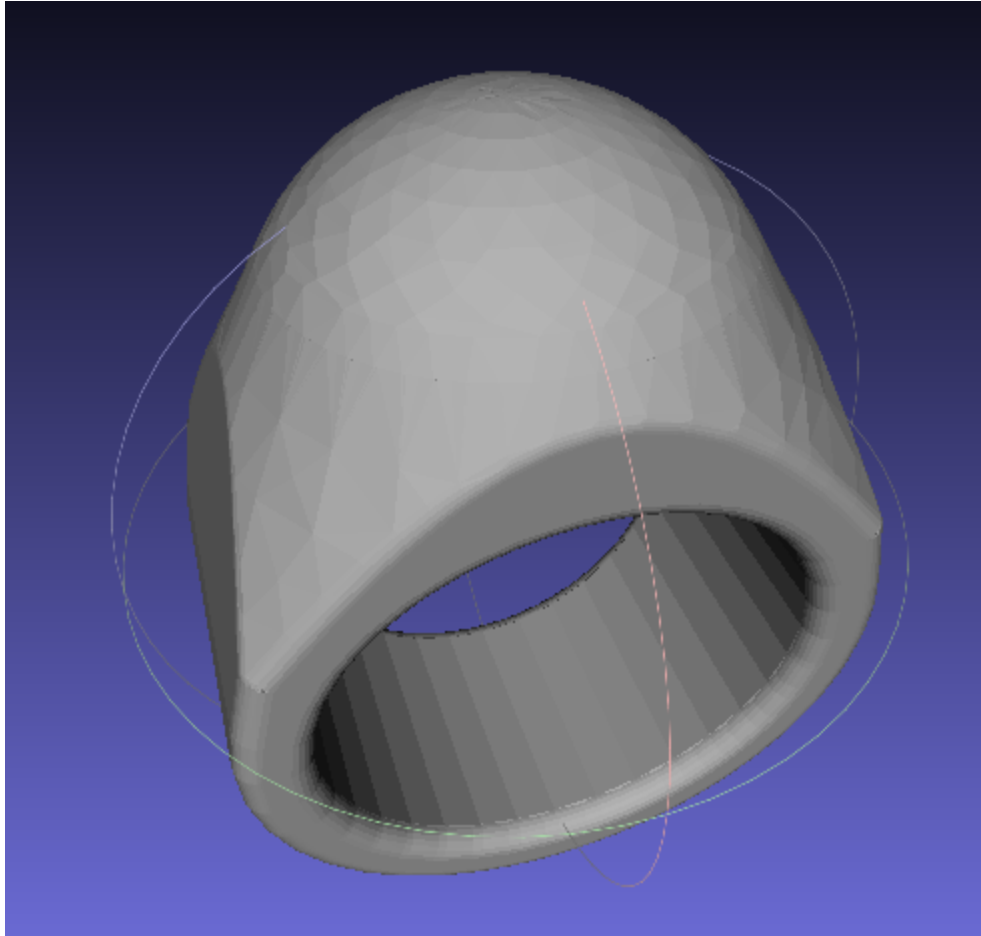
freecad ring demo

by michael philetus weller (philetus@gmail.com)

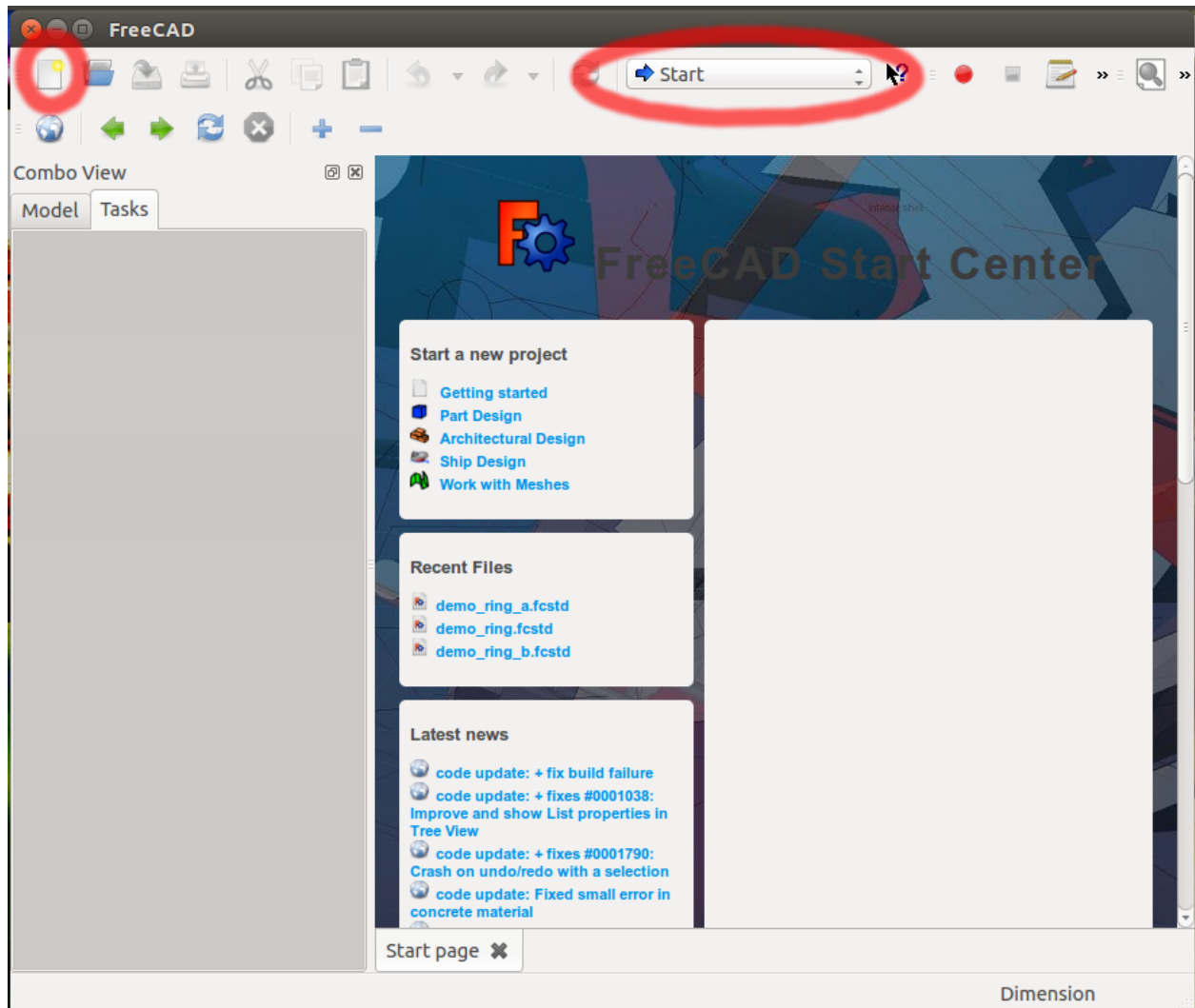
december 2014

an introduction to using freecads part design mode to model objects to be printed on an fdm
2d sketches expressing design constraints are used to generate 3d geometries

finished stl mesh exported from freecad:

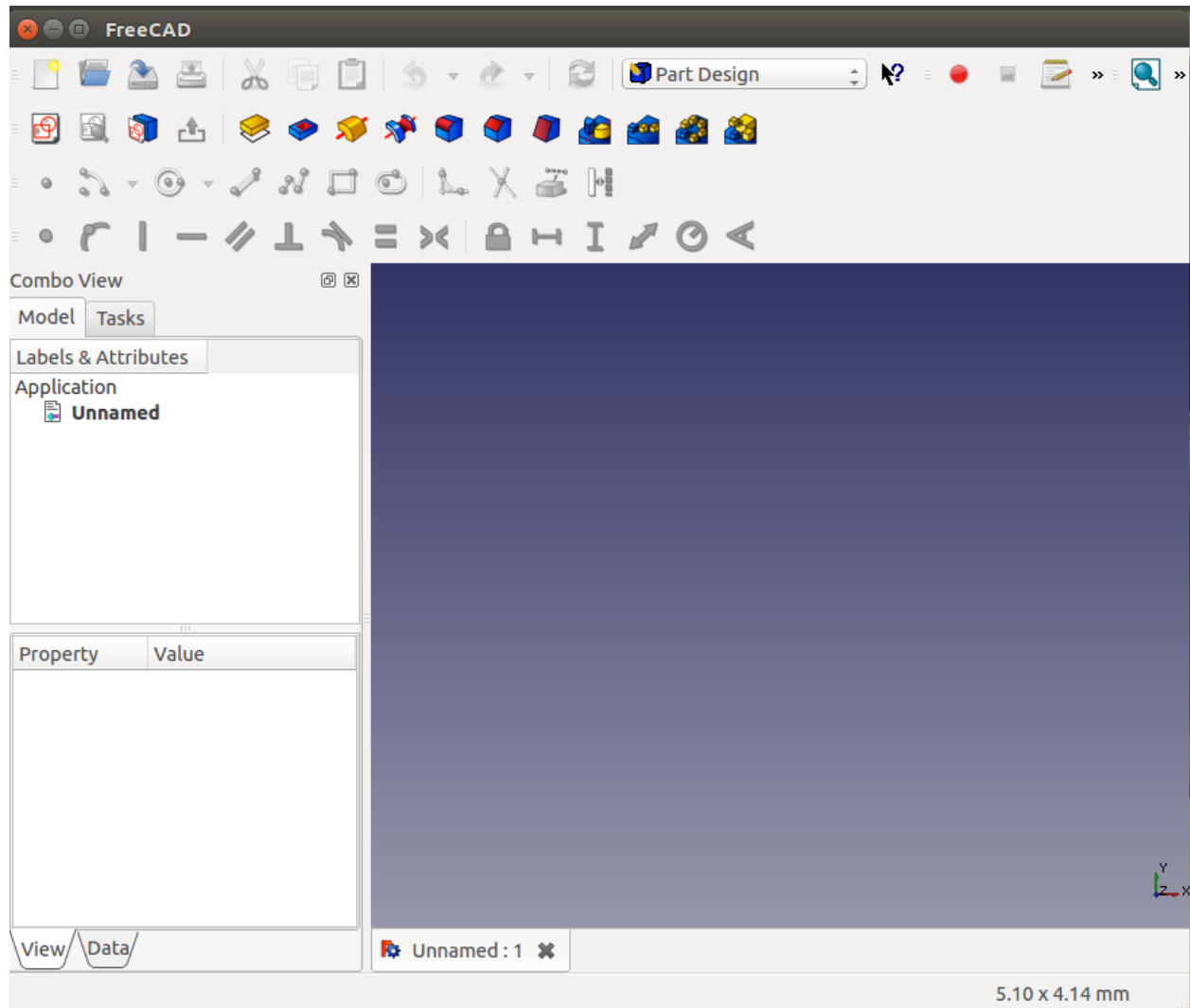


step 0:



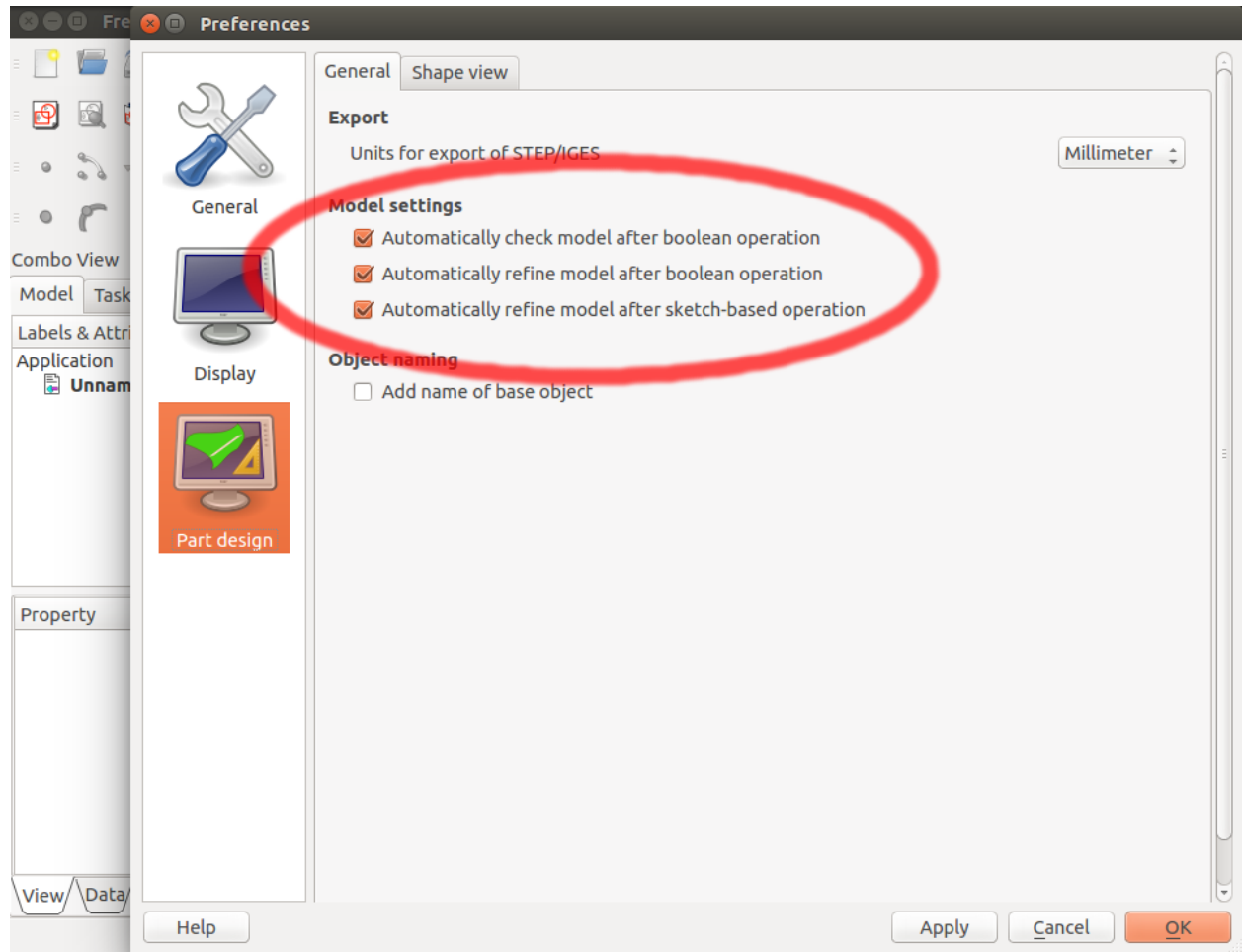
- get freecad (at least version 0.14, used in this demo) here: <http://www.freecadweb.org/>
- start the program

step 1:



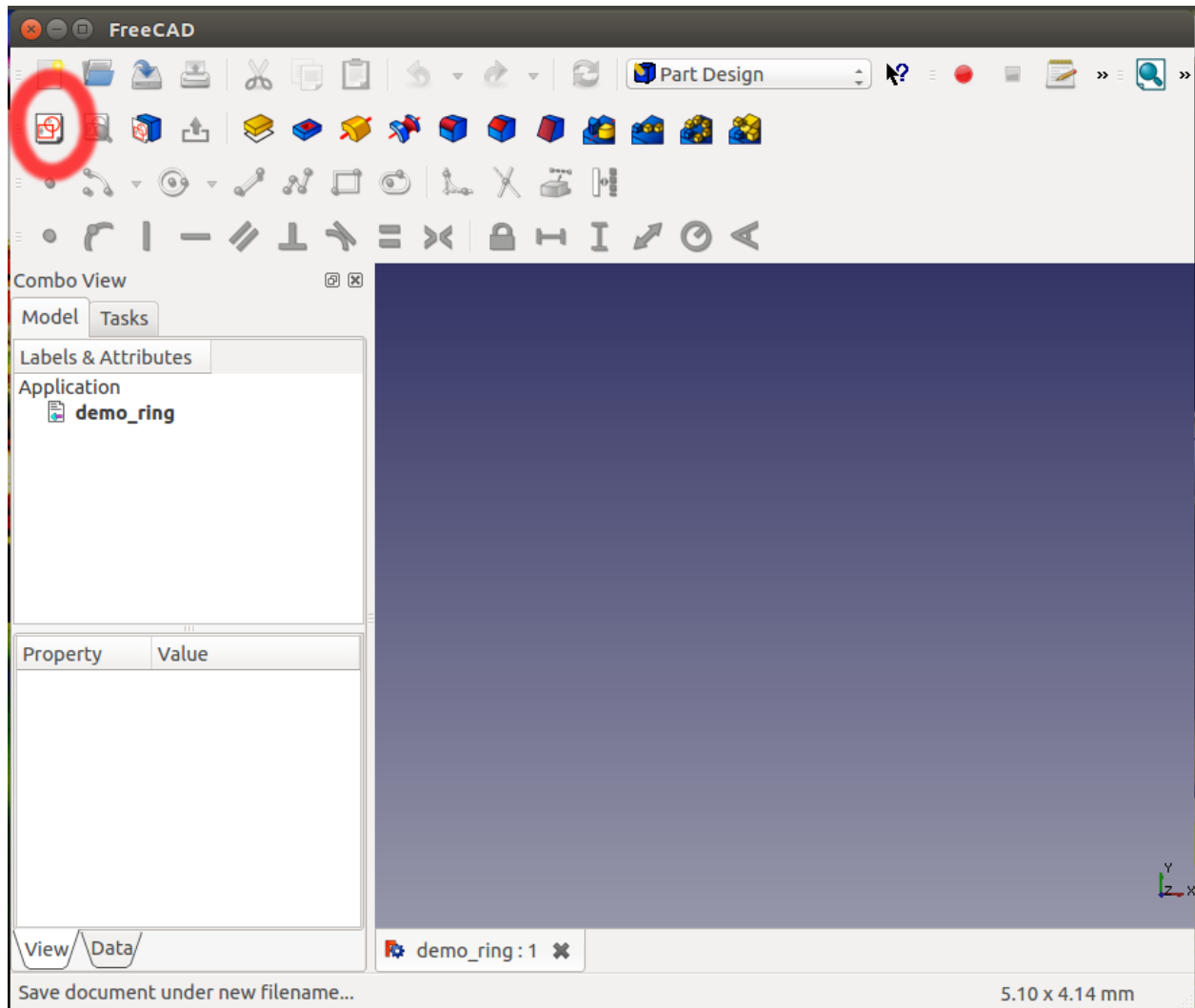
- use the mode pulldown (highlighted in step 0) to select part design mode
- use the new document button (highlighted in step 0) to start a new document

step 2:



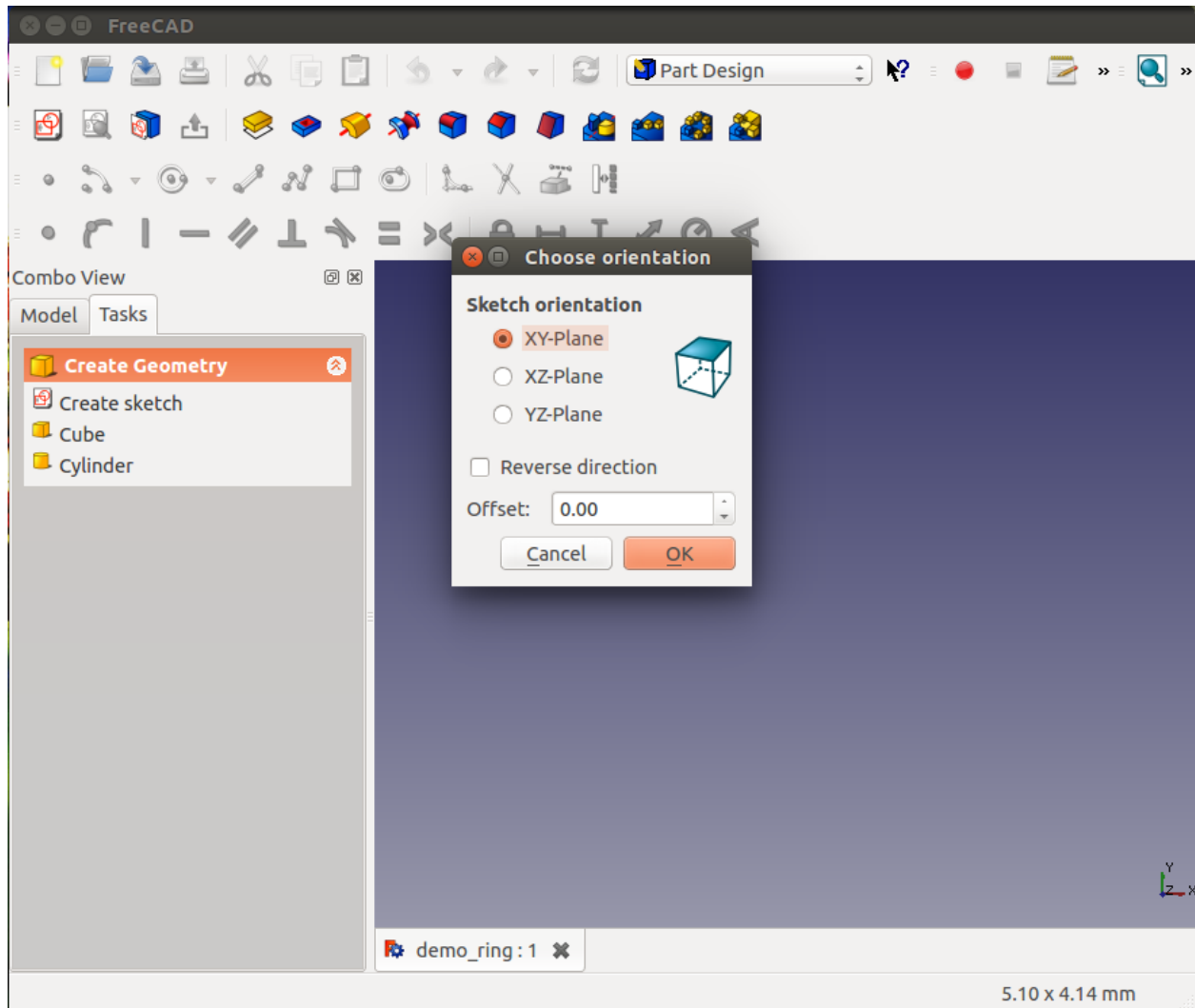
- open edit->preferences dialog and check the 3 boxes shown under part design

step 3:



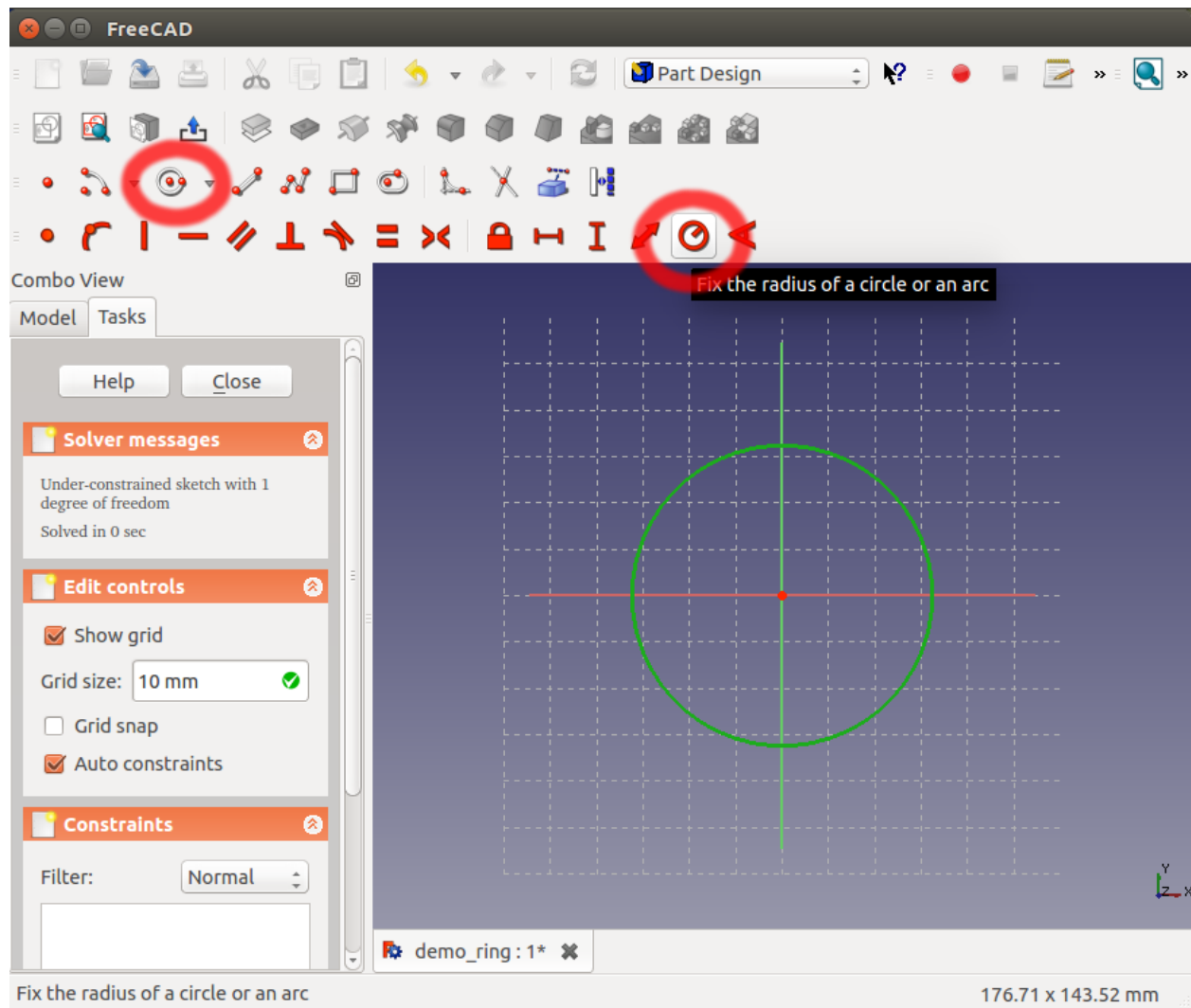
- save your document (i saved mine as demo_ring) (freecads undo is rather unreliable and it frequently corrupts save files, consider either using version control to track versions of your model, or save each time with an increasing version number to avoid losing your work)
- start by creating a new sketch with the new sketch button

step 4:



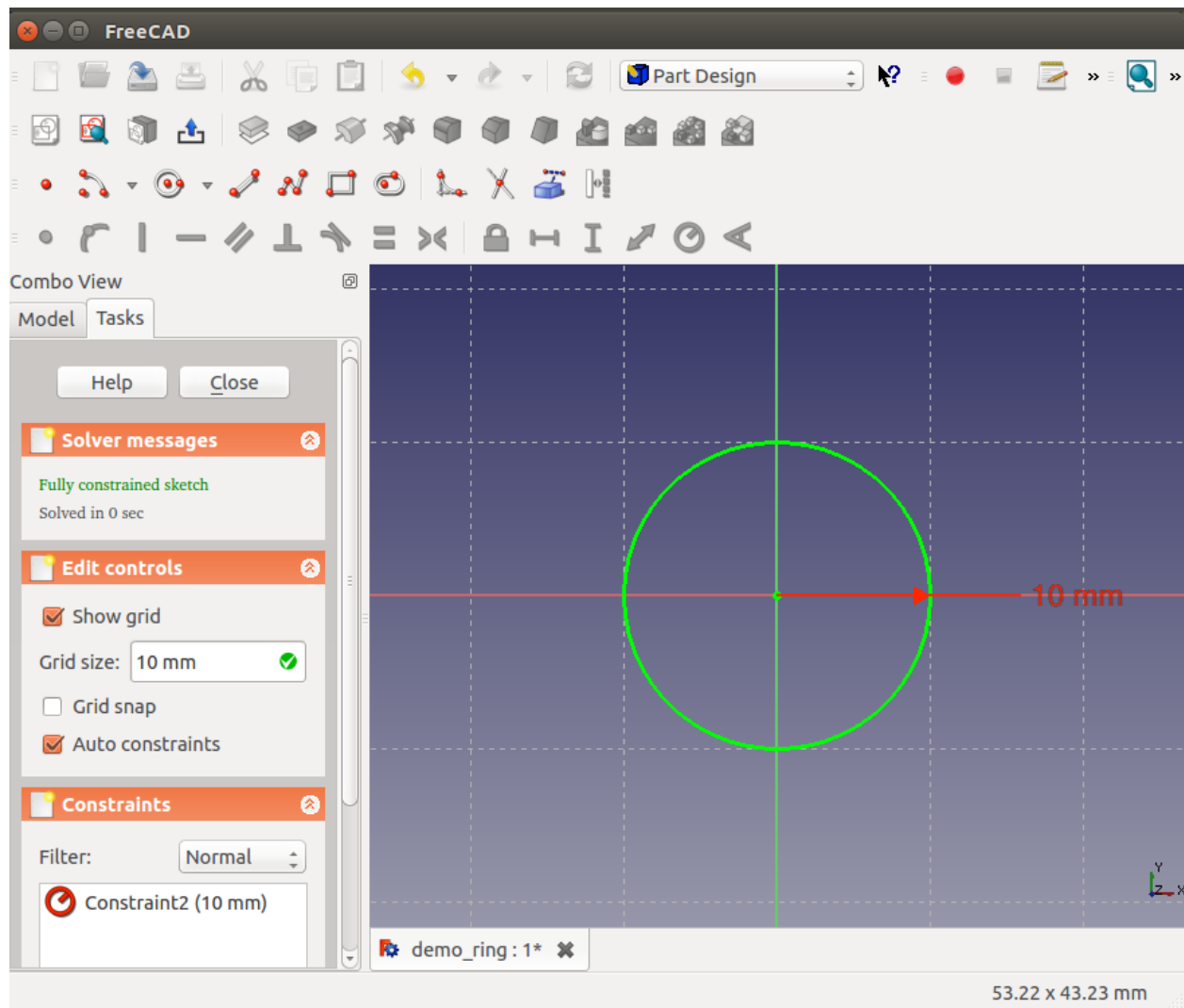
- one major current deficiency of freecad is the inability to create drawing planes and pad (extrude) / cut (pocket) from them. supposedly they are already in the development version, hopefully they will be in the next stable release
- for now we will start our sketch on the xy plane and then only draw on the faces of our in-progress ring

step 5:



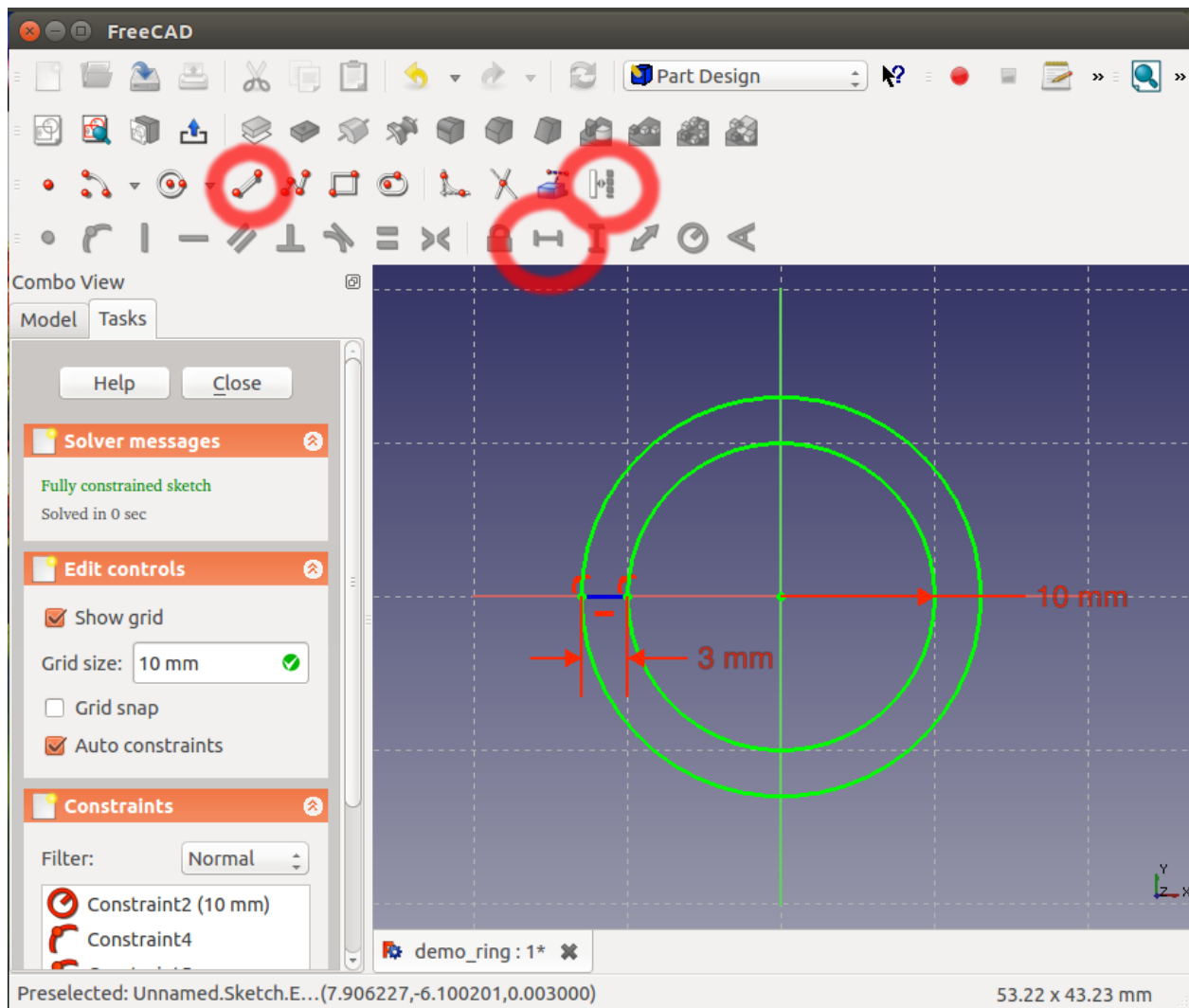
- we are ready to start sketching! the sketch tools are in the 3rd row, and the constraint tools are in the 4th row. in general we will create some sketch geometry and then add constraints to it to control its size and position
- click on the draw a circle button (highlighted)
- move your mouse over the origin point until it changes color and then click. this automatically adds a position constraint holding the center of the circle at the origin
- click a second time to create a circle of an arbitrary size (we will add a size constraint in a moment)

step 6:



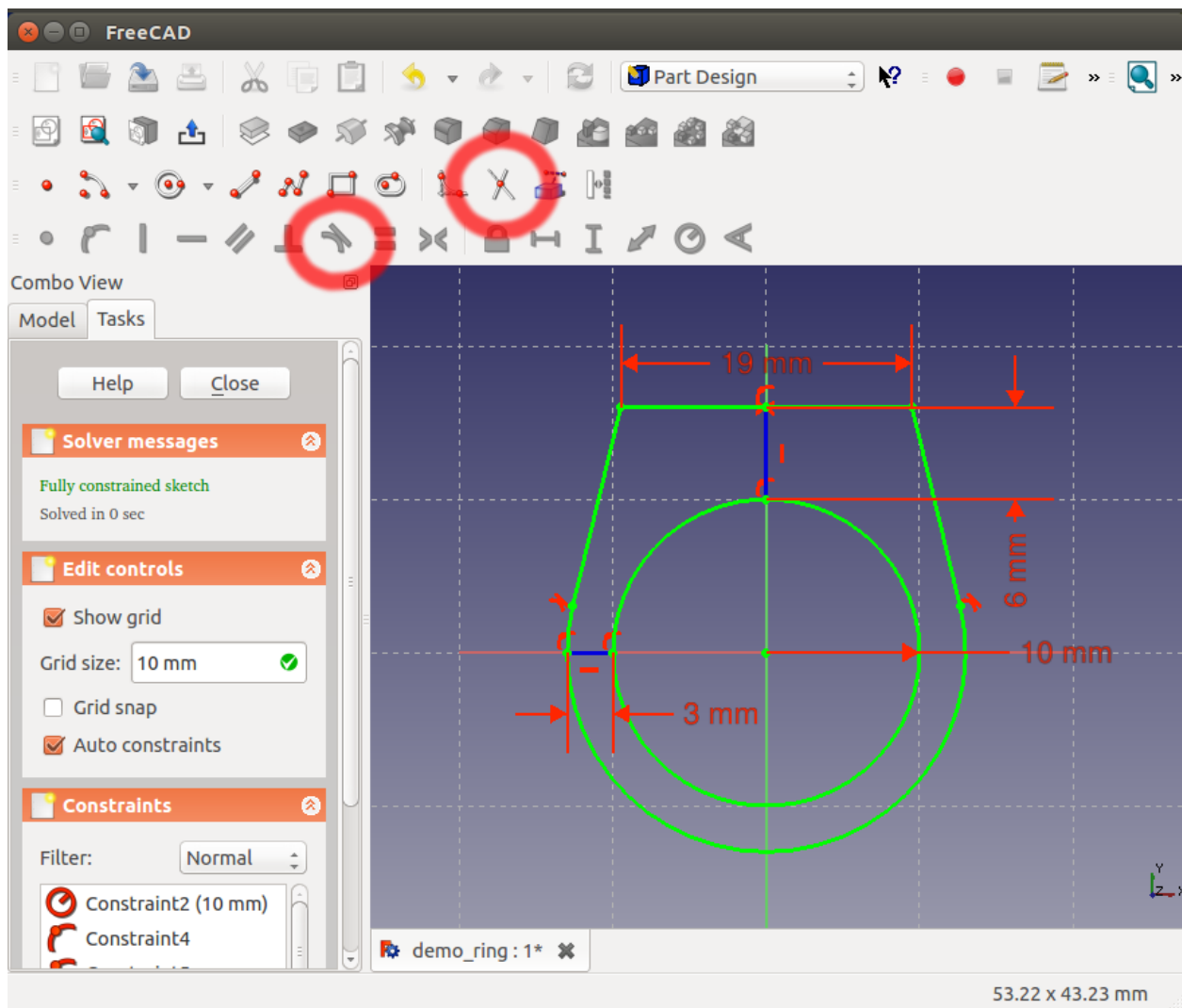
- click on the circle to highlight it, then click on the radius constraint button to define the size of the circle, and enter 10 (mm) in the box
- this circle will define the hole your finger will fit thru! go measure your finger, double click on the radius constraint and enter the size you really want it to be
- the sketch probably got really small, if you have a mouse with a scroll wheel you can use the wheel to zoom in and out. you can check out the navigation modes in edit->preferences->display->3d view->3d navigation (i like blender mode, but if you dont have a mouse try touchpad mode)

step 7:



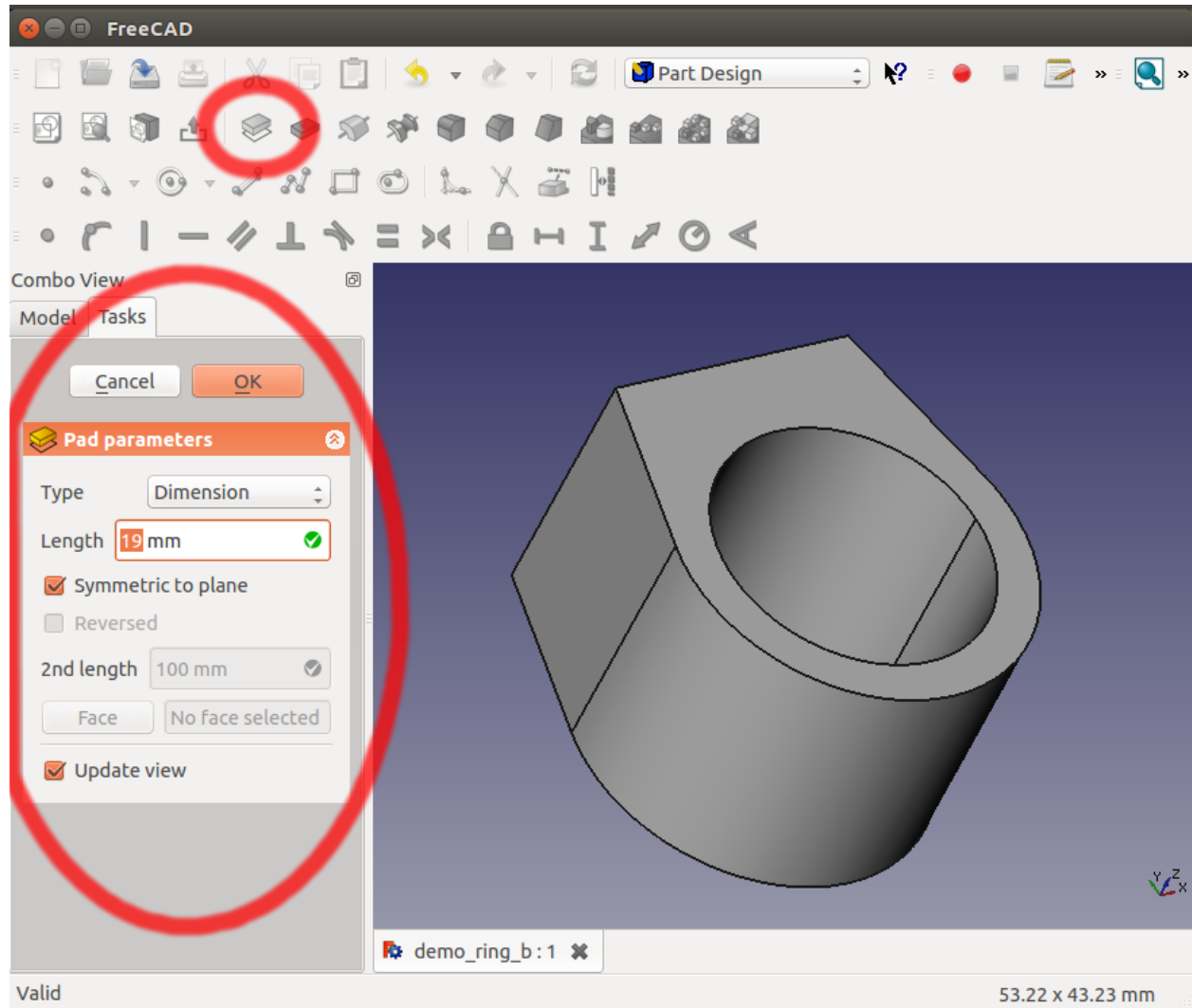
- add a second circle to define the outside of the band (make sure its outside the first circle, we will define the size further in a moment)
- create a line (with the highlighted line button) that goes from the inside circle to the outside circle
- use the highlighted construction line tool to specify that the line shouldn't be used as a boundary for 3d geometry, but is just there to help constrain the position of the other geometry (it should turn blue)
- use the point on edge constraint (if it wasn't created implicitly when you made the line) to fix the ends of the line to the inside and outside circles
- use the point on edge and horizontal constraints to fix the line to the x axis
- click on the line to highlight it and then apply a horizontal distance constraint and set the width of the band to 3mm (if we make it too much thinner there could be printing issues on the fdm)

step 8:



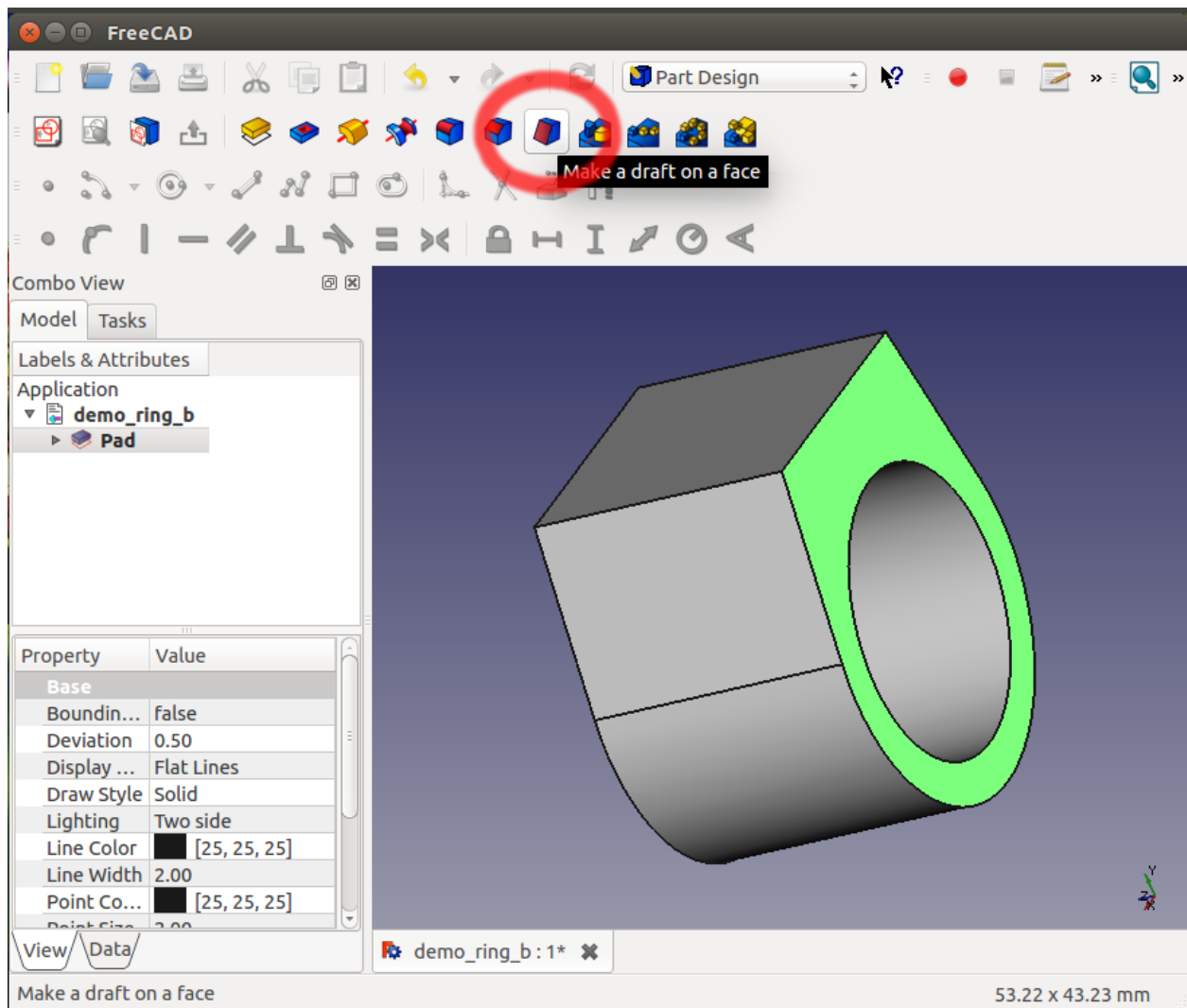
- now we are going to add a pedestal to the top of the ring, use the polyline tool to draw three connected edges extending from the outer circle
- constrain the ends of the polyline to the outer circle and then use the highlighted trim tool to trim away the top of the circle inside the pedestal
- make sure the ends of the polyline are cleanly fixed to the ends of the arc (nee circle) and then add tangent constraints to both sides
- add a symmetric constraint across the y axis for the two upper points of the polyline
- add a horizontal constraint making the top of the pedestal 19mm across
- create a construction line from the inner circle to the center of the pedestal and constrain it to be 6mm tall
- the sketch should turn green and say 'fully constrained' in the left panel

step 9:



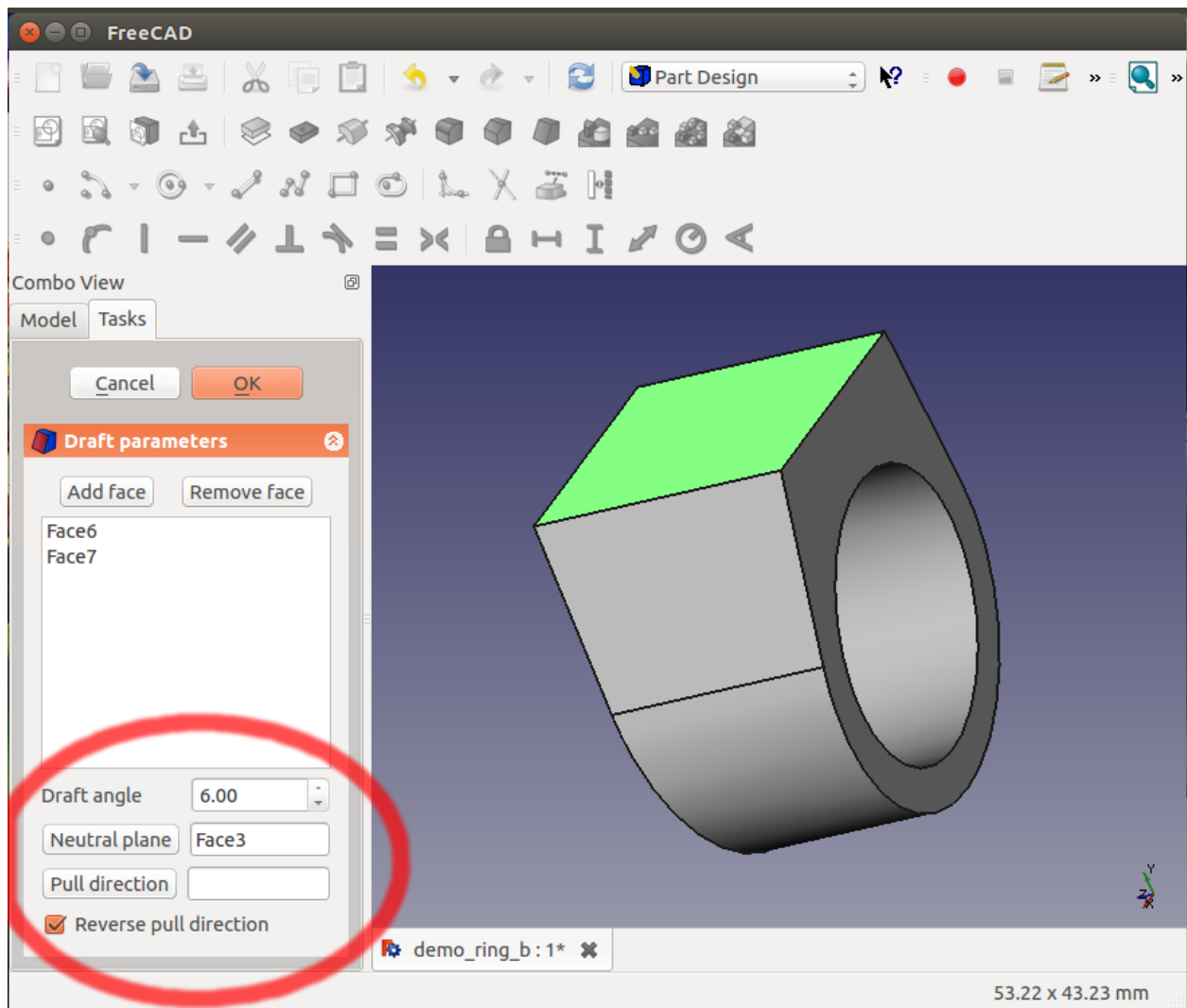
- close the sketch
- make sure the sketch is still highlighted in the history panel and click on the pad button to extrude the 2d sketch into a solid body
- make it 19mm tall and click on the symmetric to plane button so that we can draw things at the origin in the middle of the pedestal (this wouldn't be important if we could create arbitrary sketch planes)

step 10:



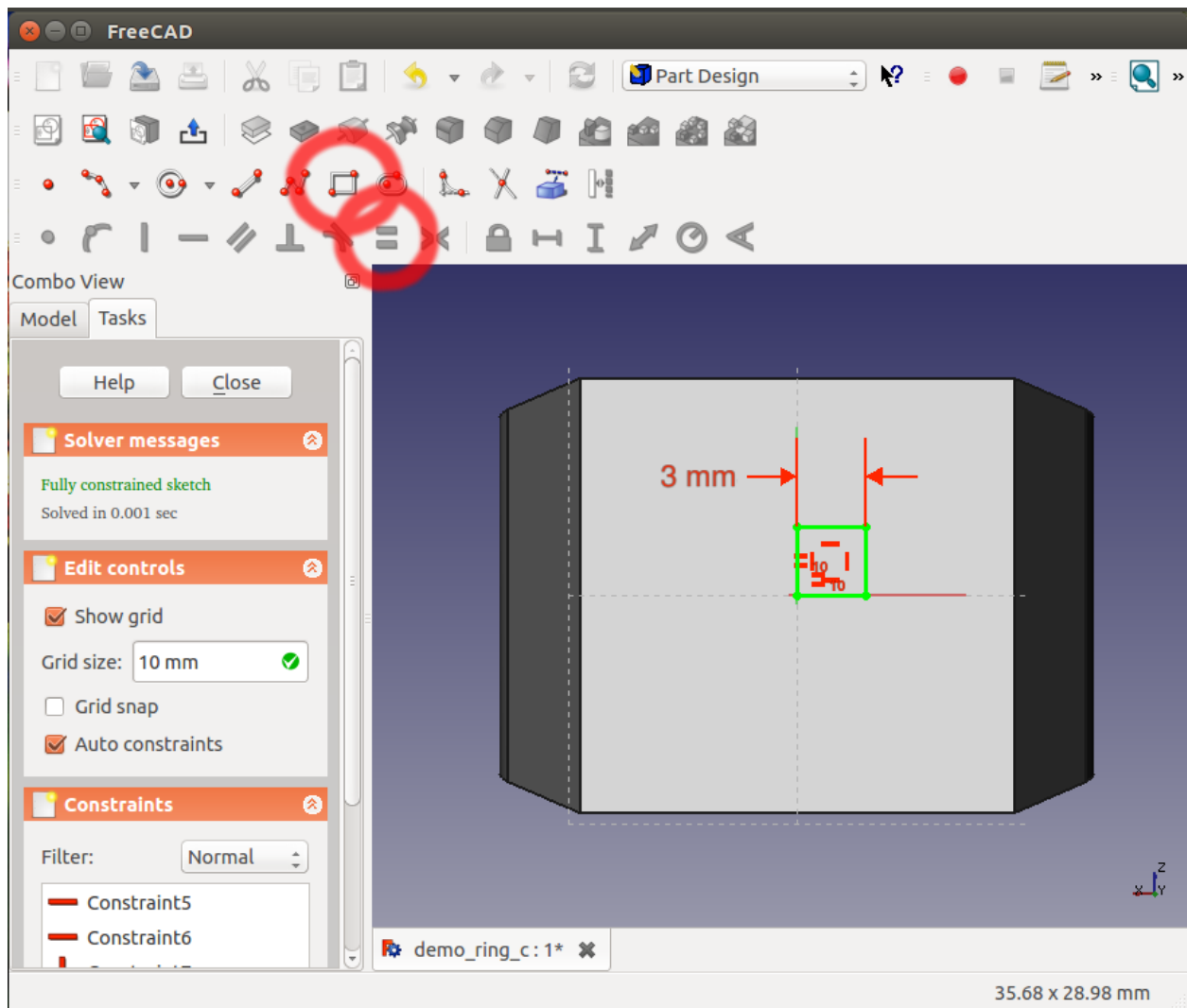
- we have a pretty wide, chunky ring. we can add a draft to the sides to make the bottom narrower and more comfortable to wear. select the two opposite faces (hold down control to select more than one geometry)
- hit the draft button

step 11:



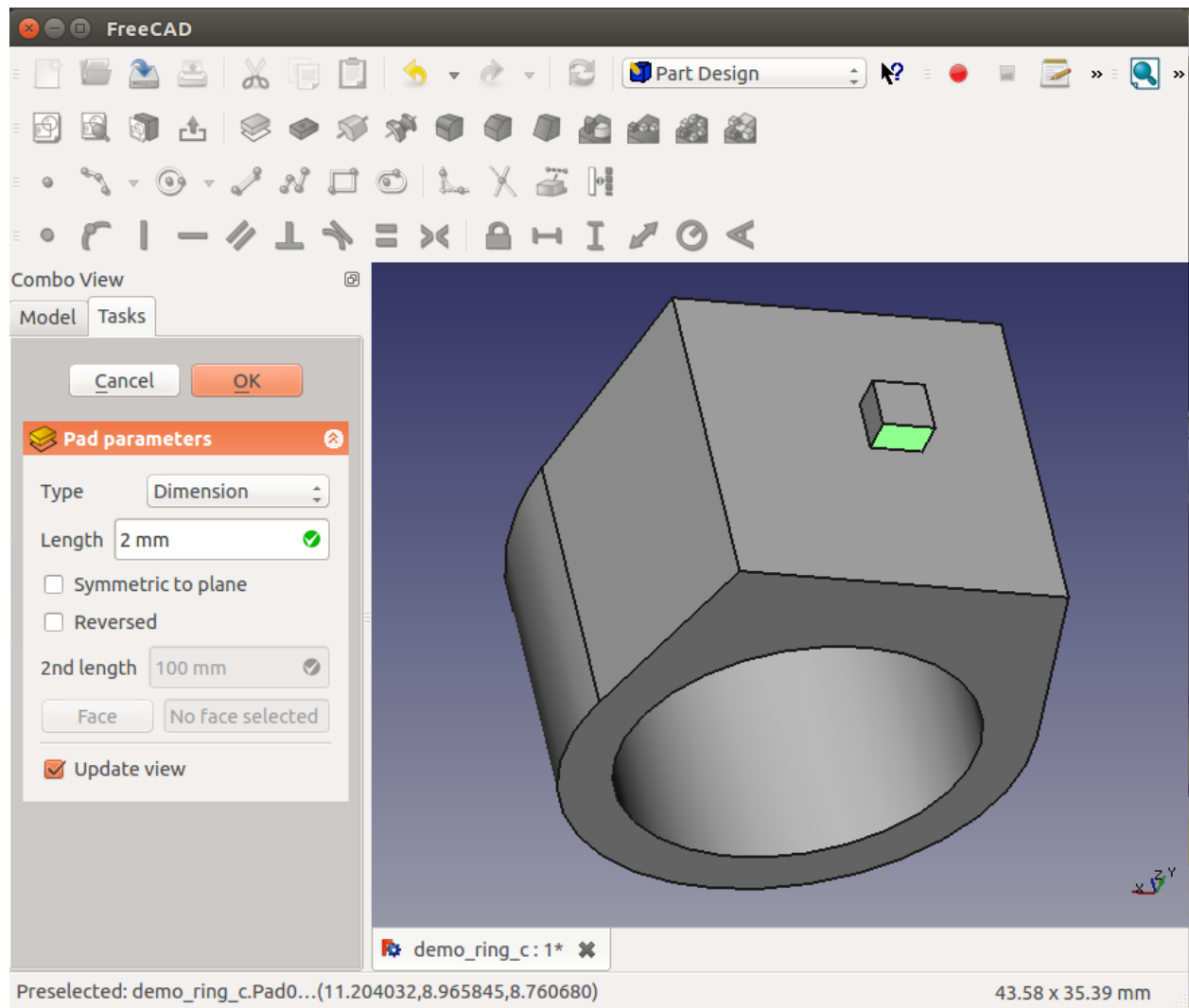
- we want the pedestal to stay the same size, so hit the neutral plane button and click on the top face of the pedestal
- check the reverse pull direction box to make the ring get skinnier moving away from the pedestal
- change the draft angle to 6 degrees
- hit ok

step 12:



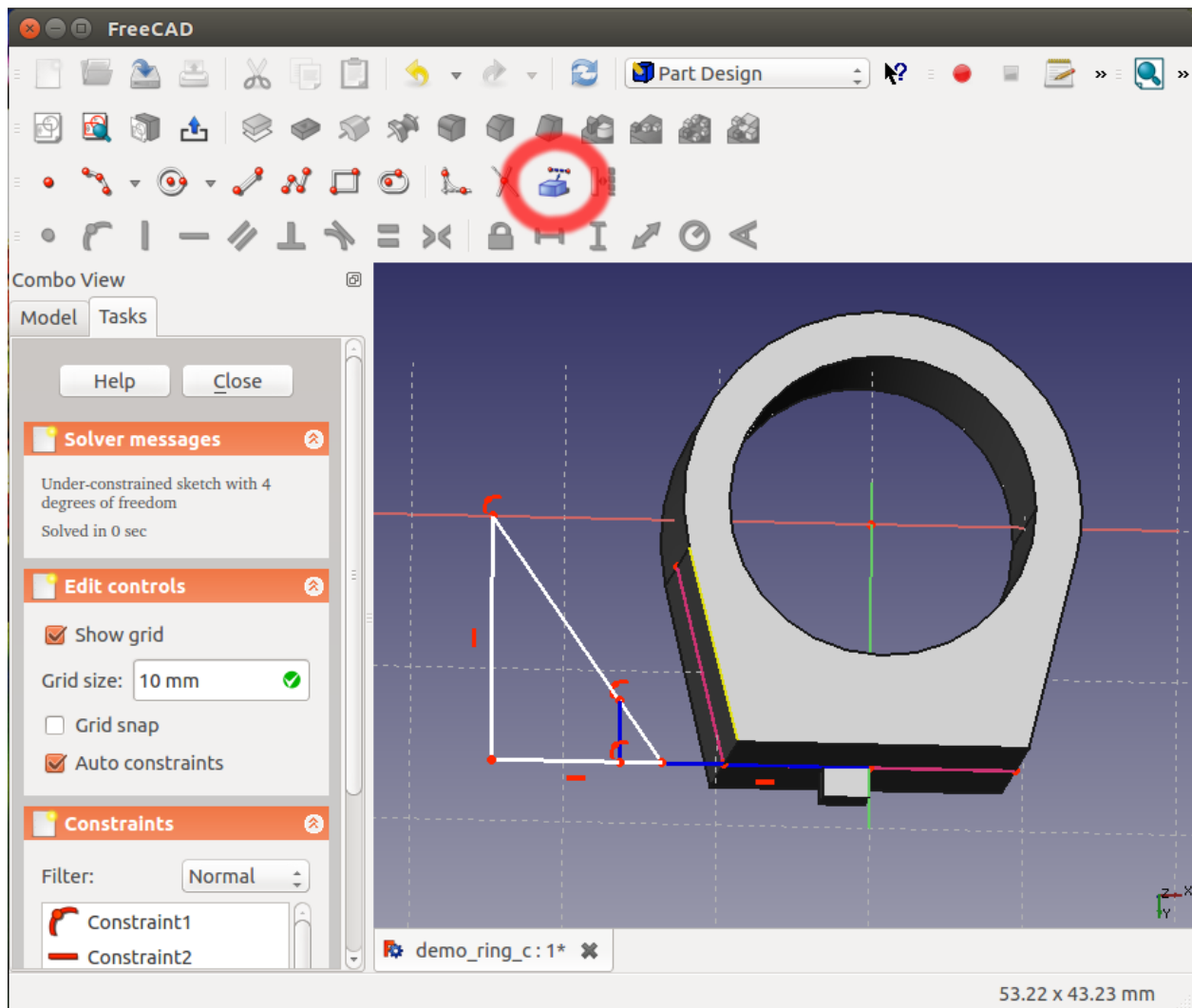
- this step is kind of goofy, because this version of freecad doesnt have drawing planes we have to create a temporary object with faces at the origin so we have faces to draw on
- click on the top face of the pedestal, and then the sketch button
- draw a 3mm square so that one corner touches the origin

step 13:



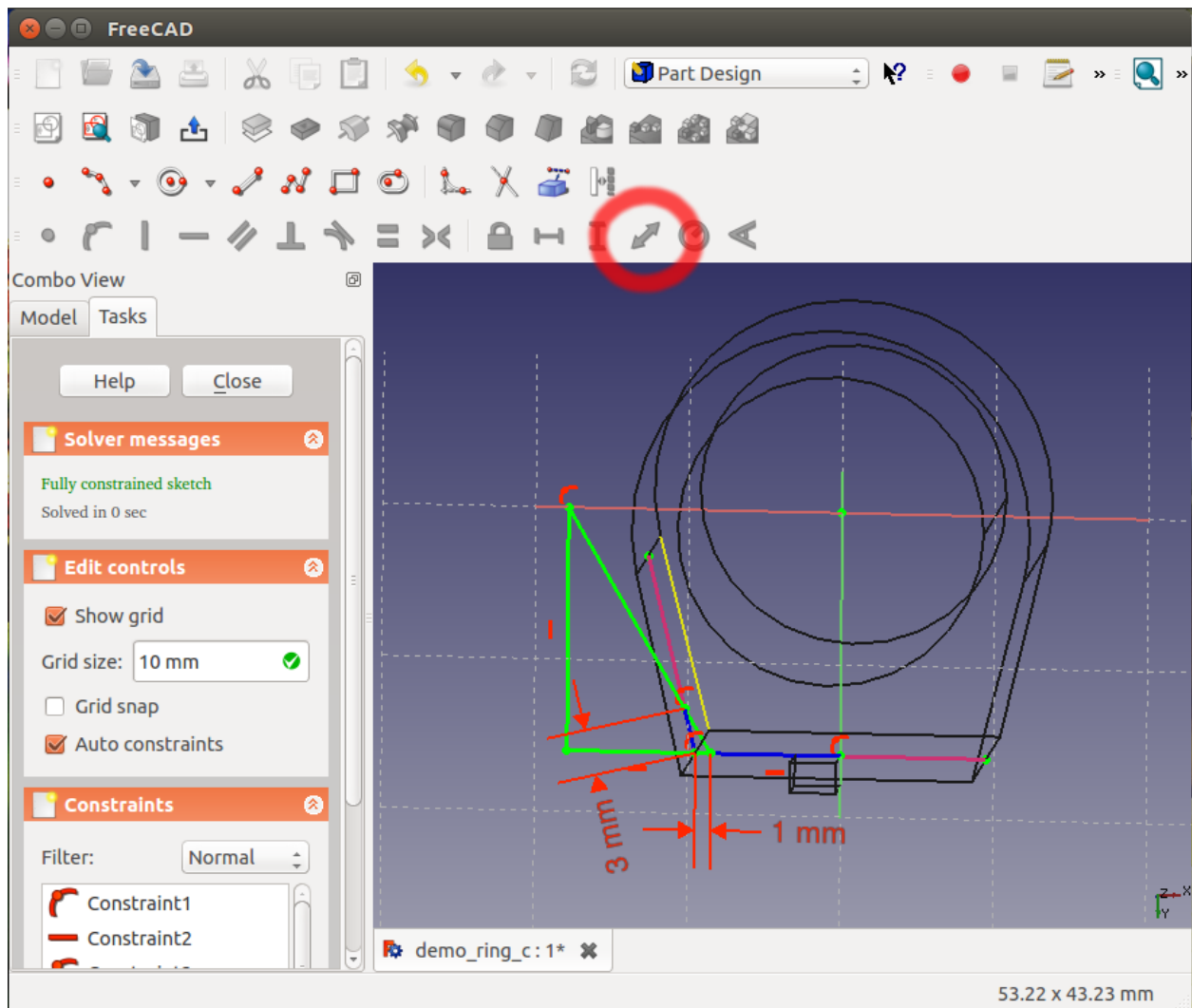
- make a 2mm tall pad from the sketch
- start a new sketch on the face shown in green!

step 14:



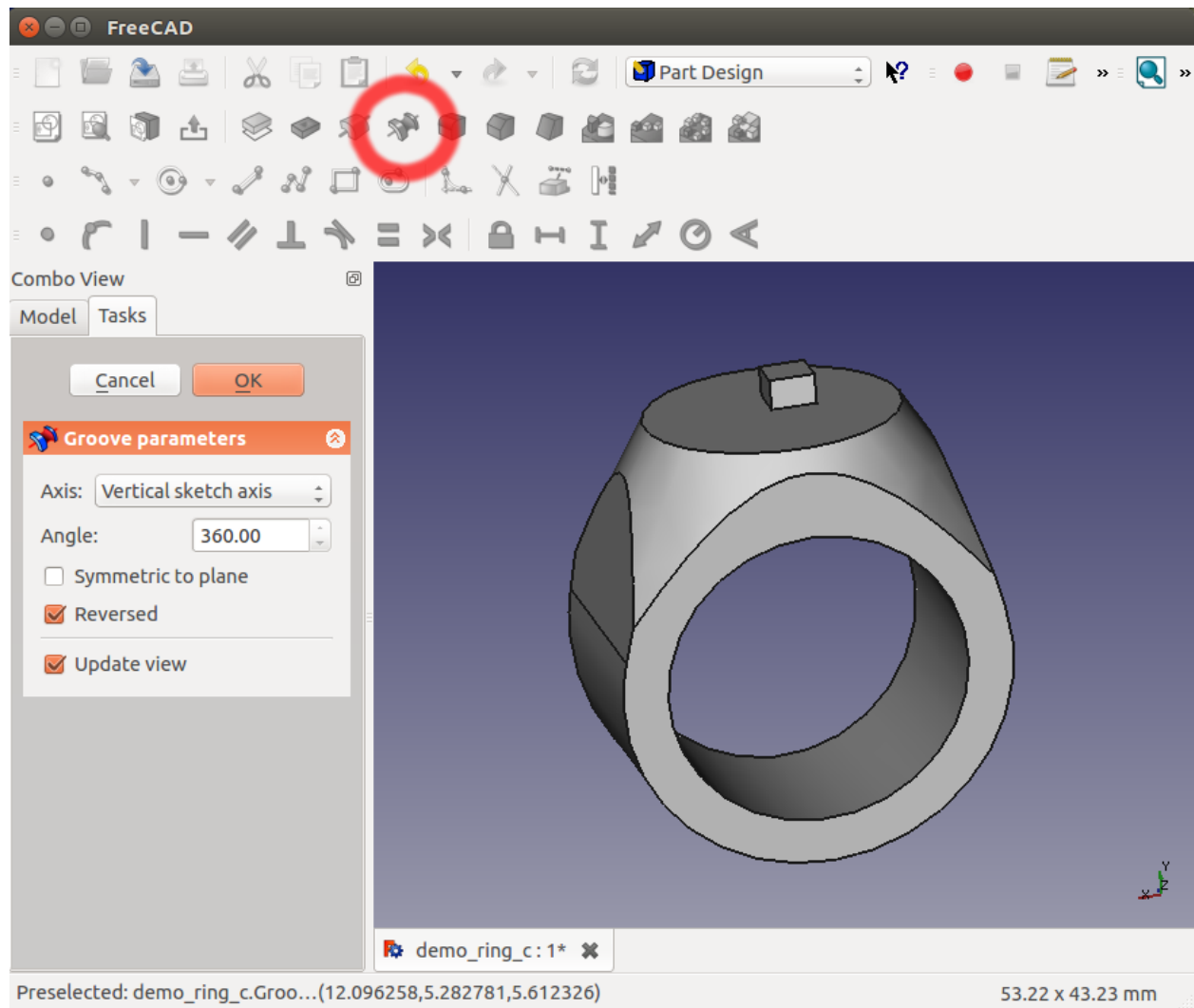
- up until now we have just made extruded bodies and cuts, now we will sketch a geometry that we will make a rotated cut with (what freecad calls a groove) to make the top of the pedestal round
- use the existing geometry tool to bring the top and side edge of the pedestal into this sketch
- draw a triangle with construction lines as shown

step 15:



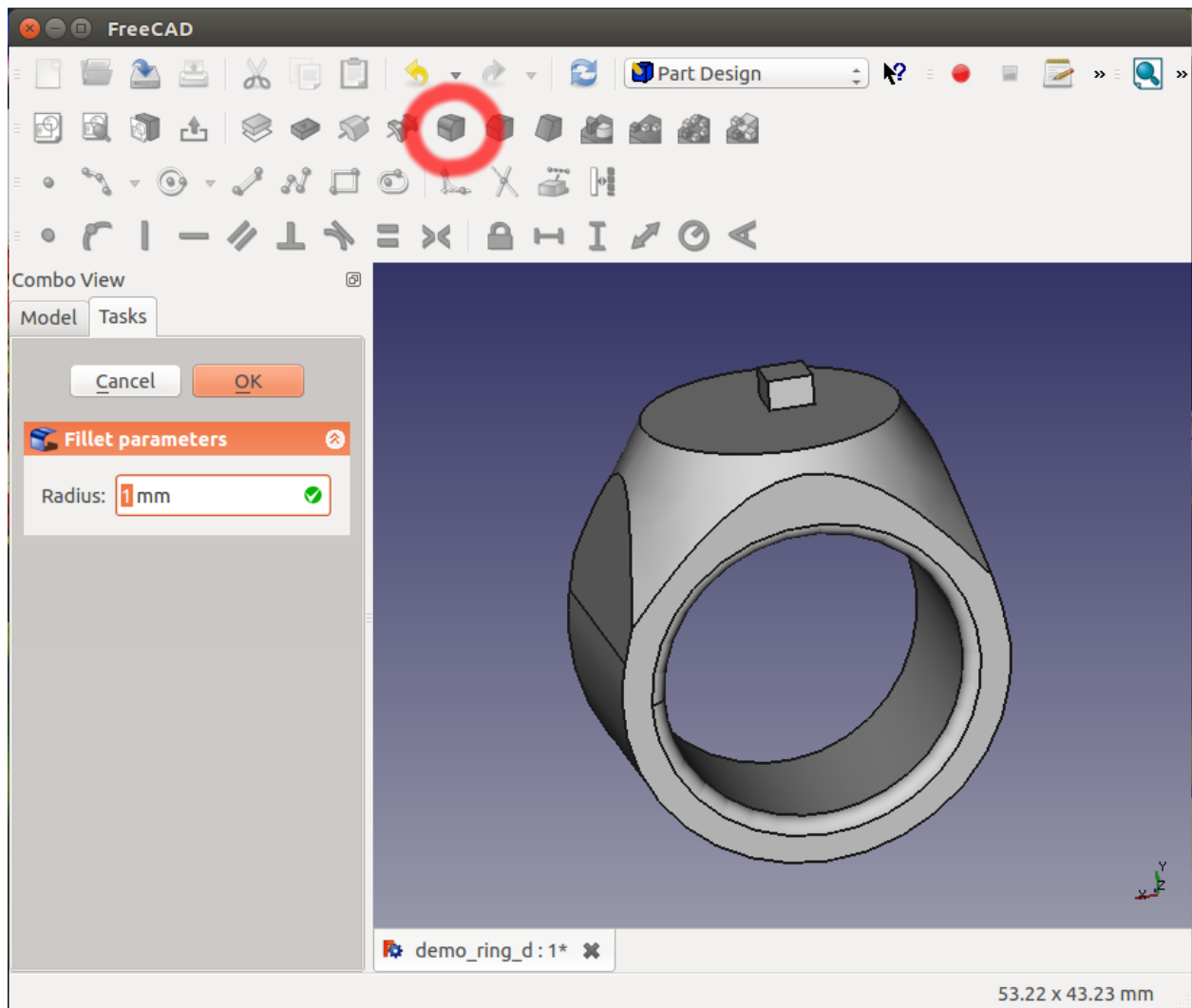
- it is hard to see our sketch through the solid body of the ring, so click on view->draw style and select wireframe mode
- constrain our cutting triangle to the edge of the pedestal as shown (use the highlighted edge length constraint to set the length of the construction line along the side of the pedestal)

step 16:



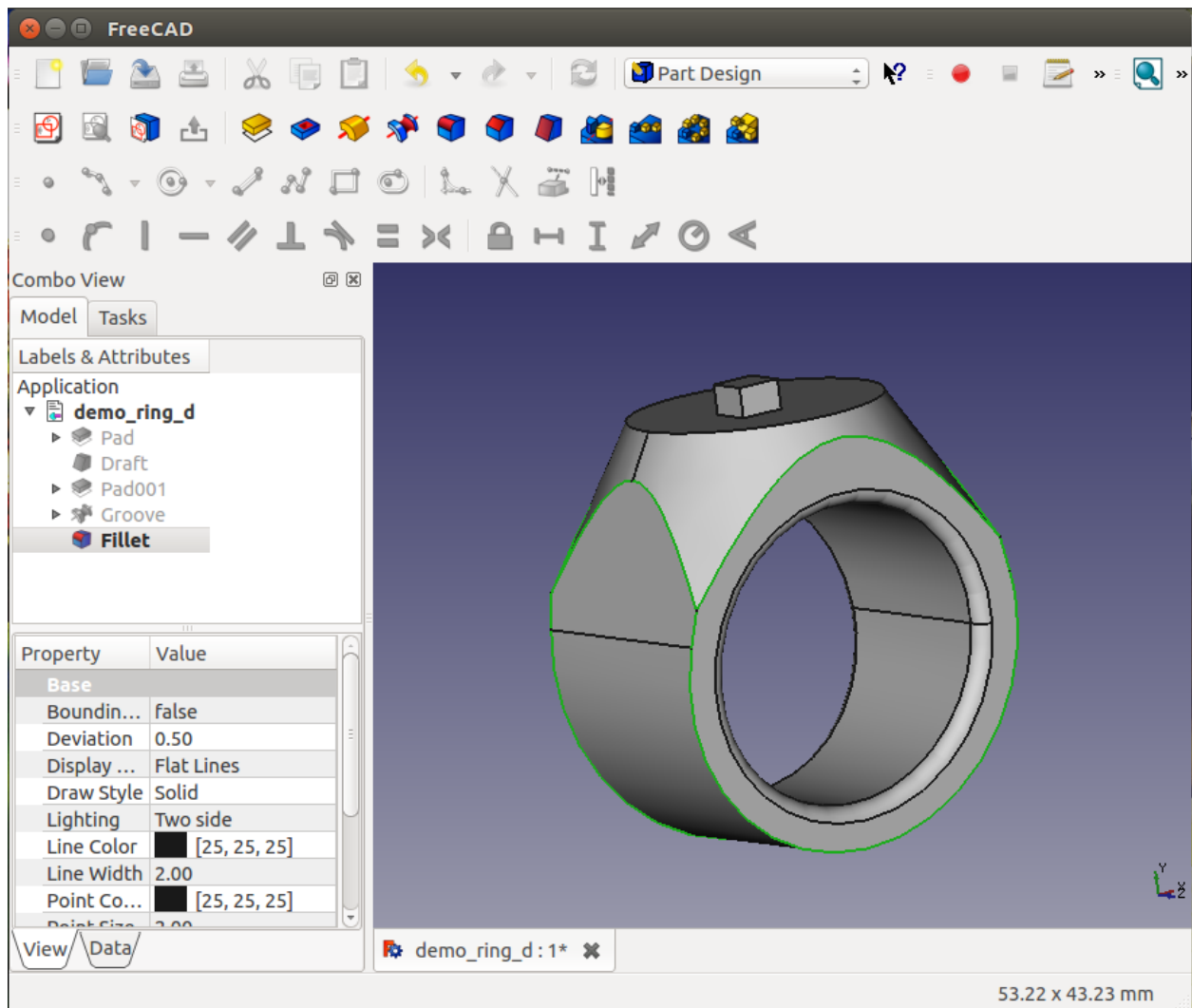
- close the sketch and set the draw style back to 'as is'
- cross your fingers, select the sketch and then click on the groove button
- if it looks ok, as above, click ok

step 17:



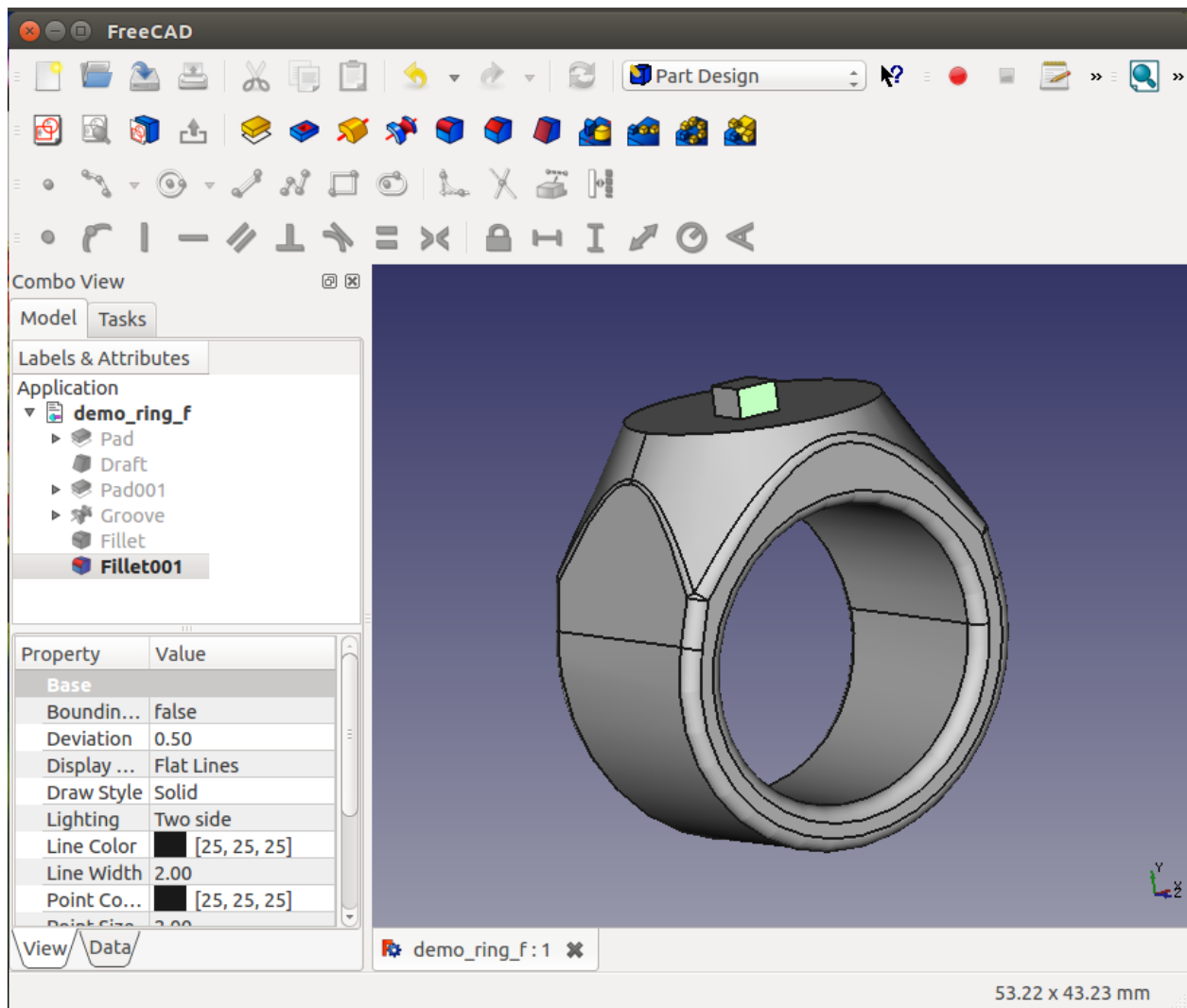
- select the two inner edges of the ring (remember to hold control) and then select the fillet tool, and set the fillet radius to 1mm

step 18:



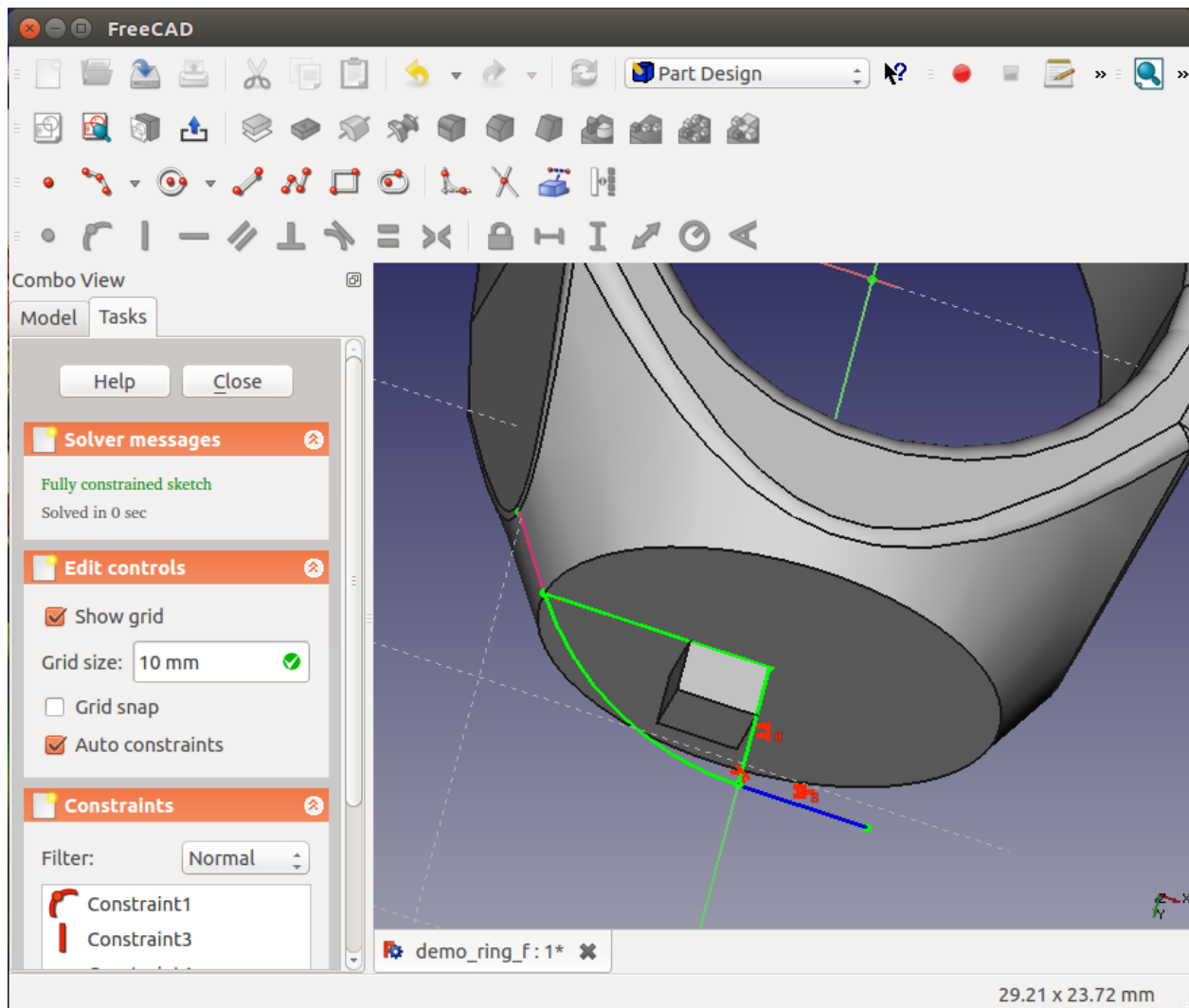
- select the outer edges to fillet

step 19:



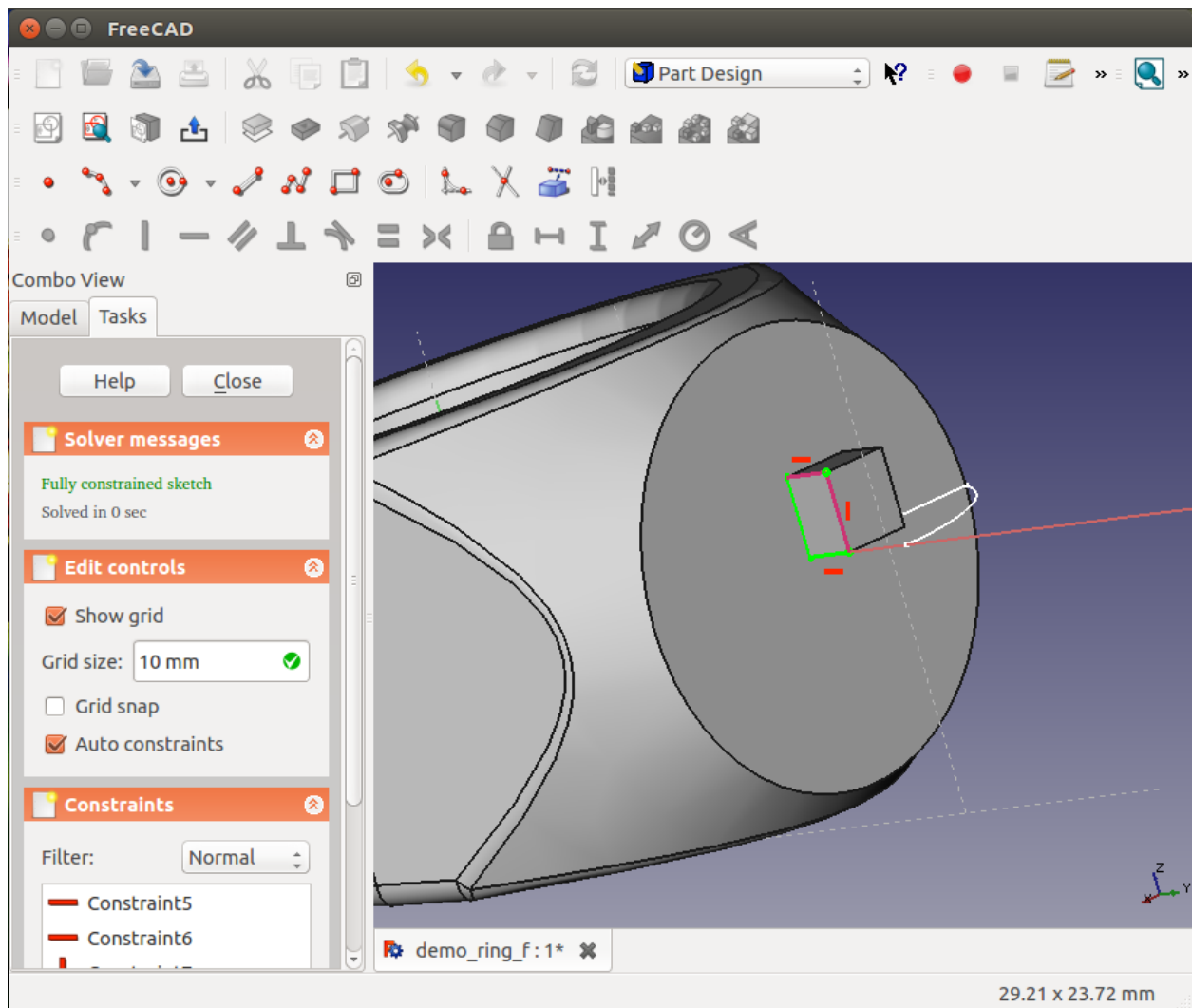
- select the face of our temporary geometry again and create a sketch on it

step 20:



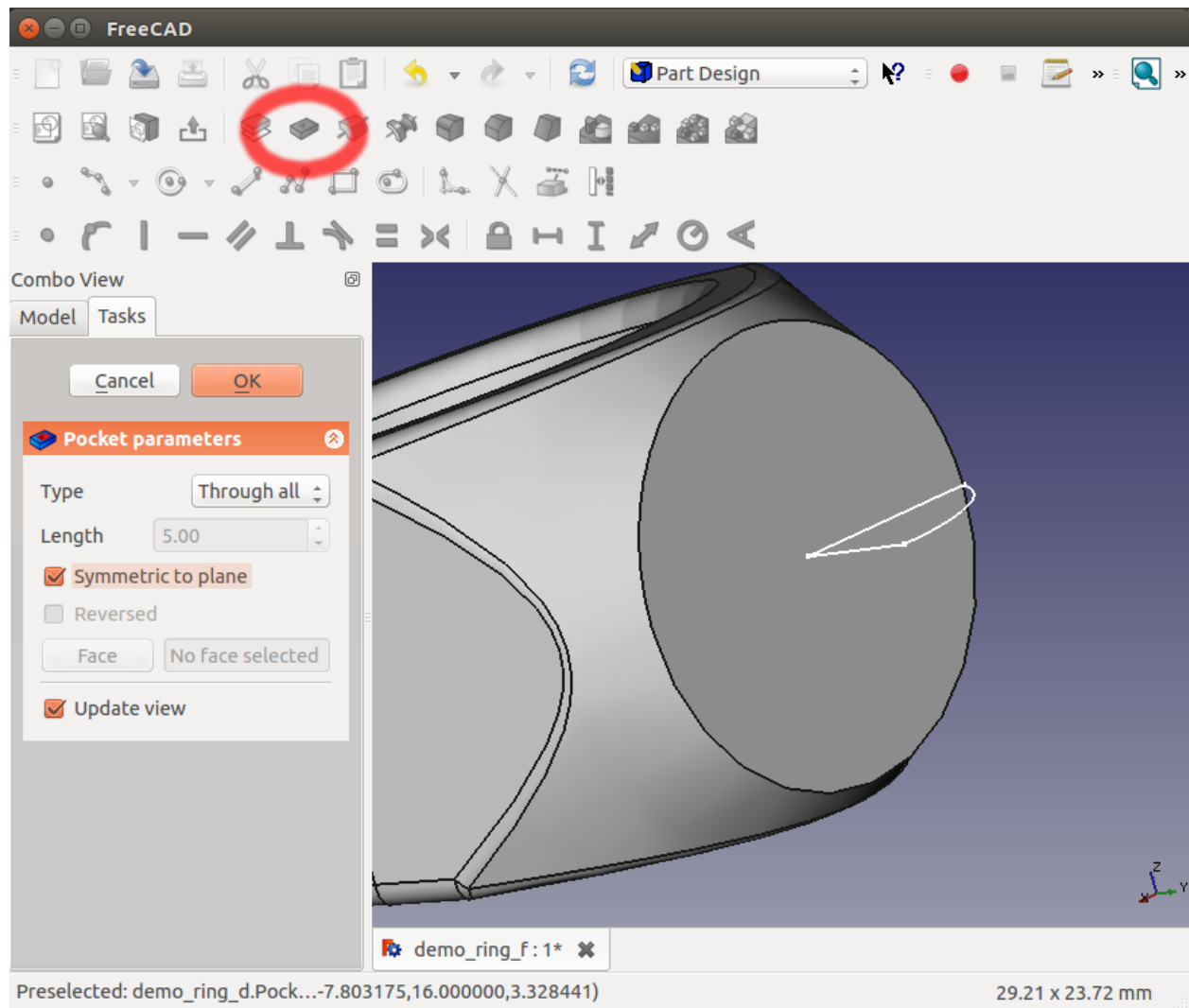
- use the existing geometry tool to bring the edge of the round pedestal into the sketch
- draw two lines centered at the origin
- draw an arc tangent to the edge of the pedestal and tangent to a construction line perpendicular to the vertical edge (make the construction line equal to the other edge just to make the sketch constrained)

step 21:



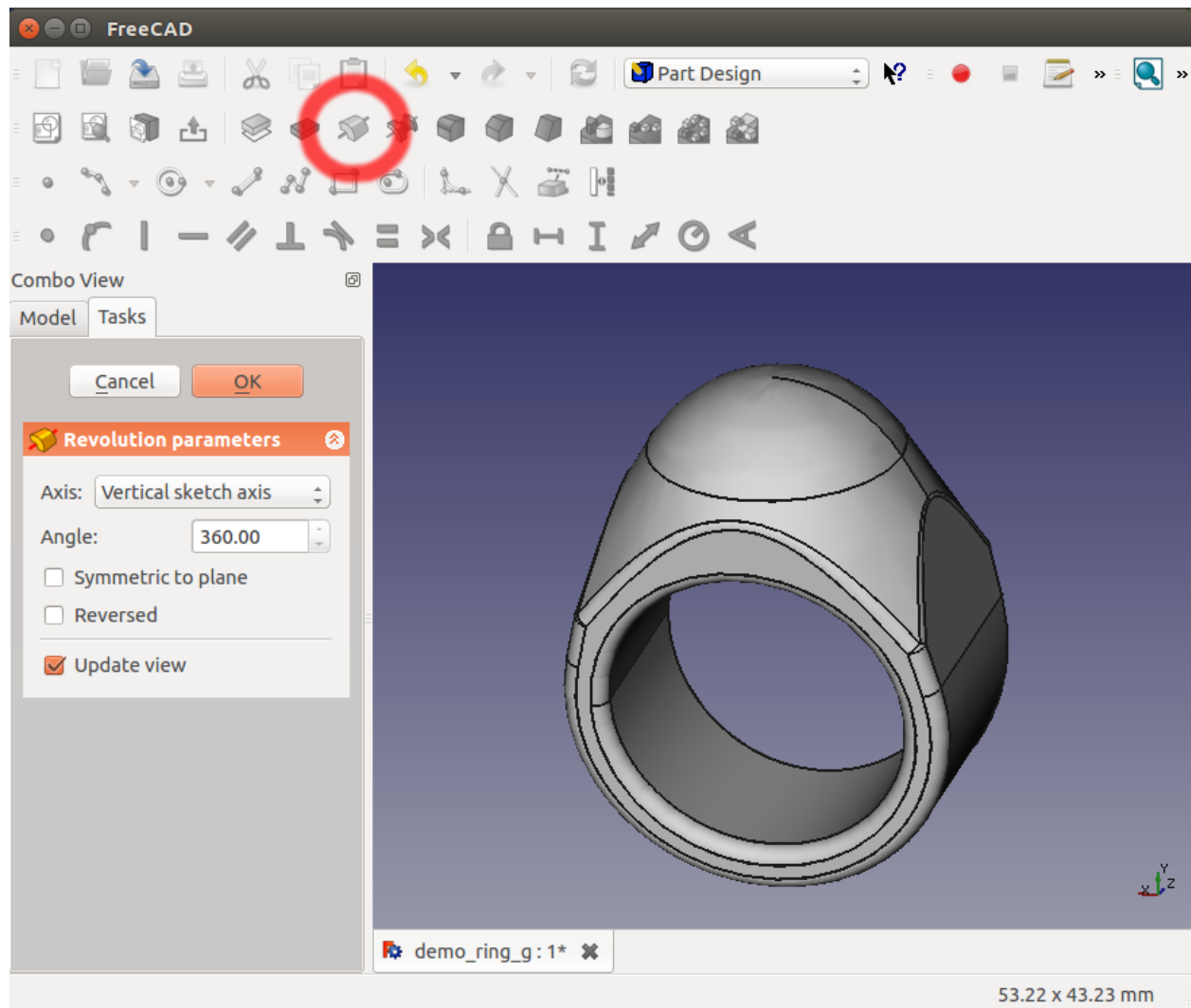
- although theoretically we should be able to revolve the sketch and have it cover our temporary pad, when i tried that it made freecad unhappy
- try drawing a rectangle on the side of the pad and making a pocket with it to get rid of the temporary pad

step 22:



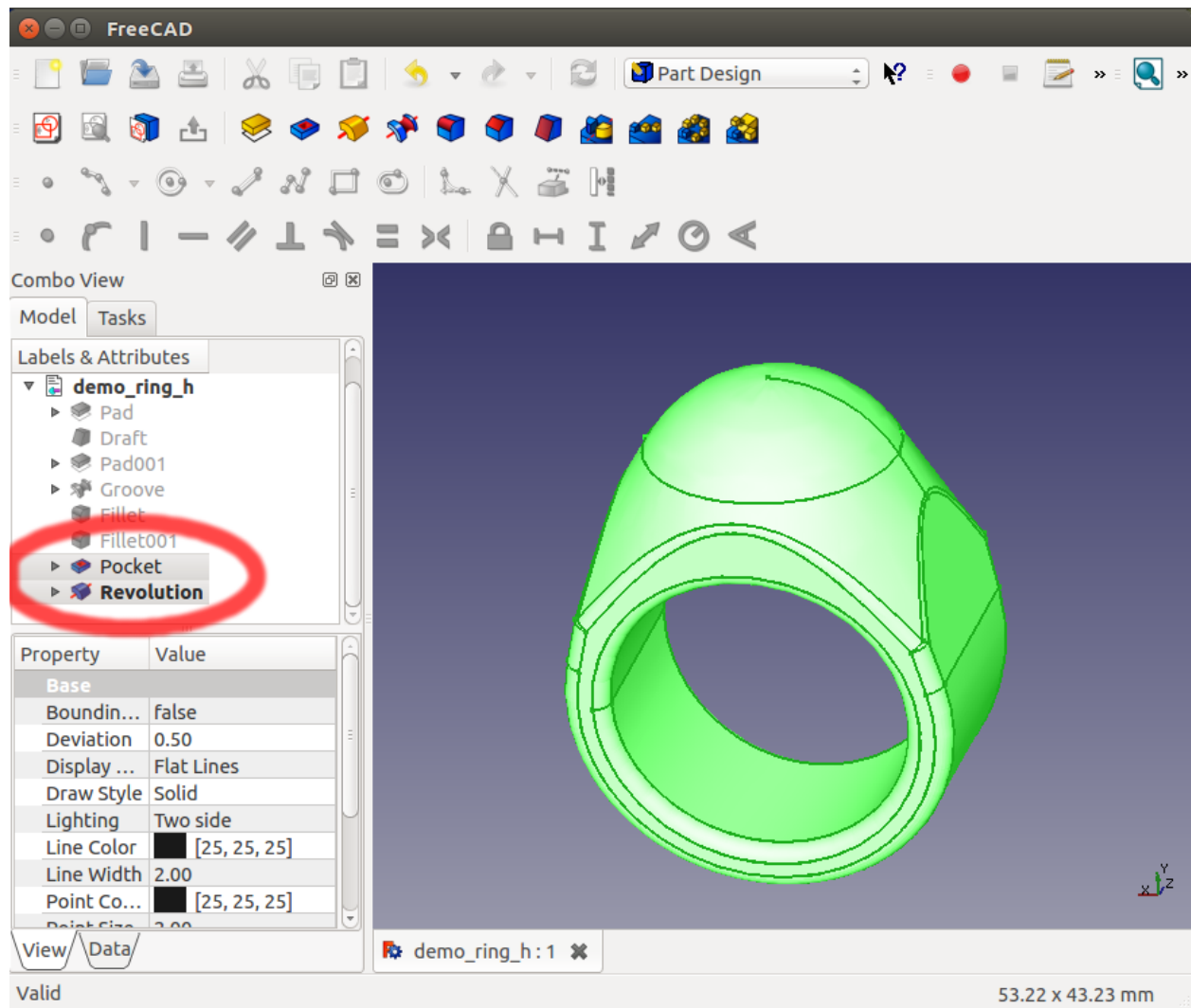
- set the pocket to go 'through all'

step 23:



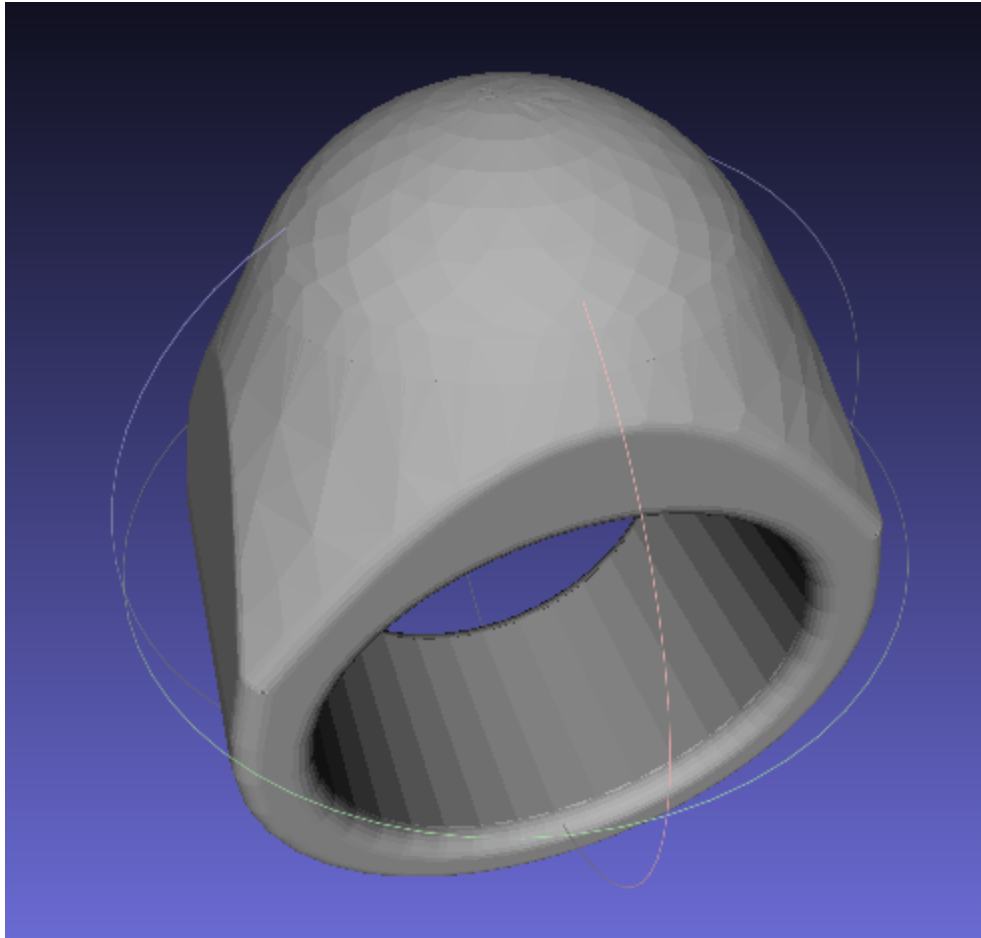
- select the dome sketch and the revolution tool to generate the dome for the top of the ring (freecads revolution tool is rather fussy, if you get bizarre geometry you may have to backtrack and try again)

step 24:



- yay i got freecad to generate the ring i wanted!
- to fabricate the ring you need an stl mesh -- select the last node of the history graph (or if only part of your model lights up green try selecting more nodes, i had to select two) -- then select file->export and pick 'mesh type' from the file type pulldown and give your file a name ending in '.stl'
- open the stl file in meshlab or your slicing program and see if freecad really made you something nice or if you have to start over now

final stl mesh shown in meshlab:



- if you can see any holes there is something wrong, go back to freecad and try again
- if there are any black faces they are probably facing the wrong way, you could try to fix it in meshlab or netfabb, or start over in freecad