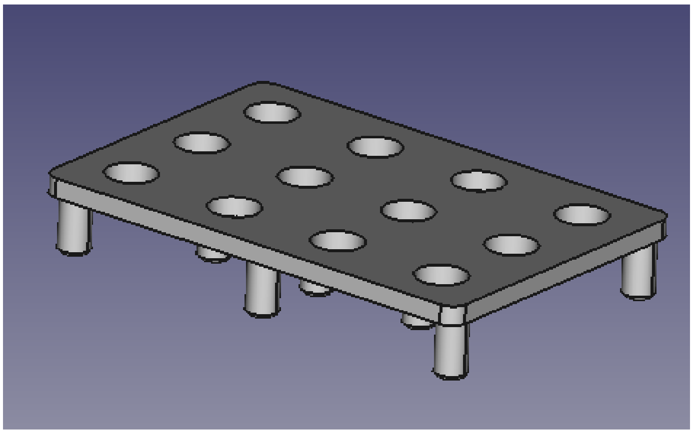
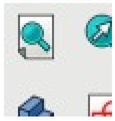
FreeCAD – ER 16 Collet Holder

By Thor

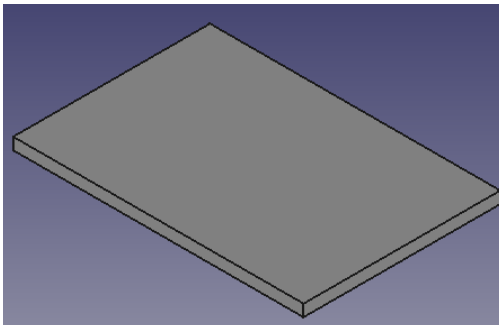


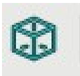
To store all my ER 16 collets in their cardboard boxes makes it cumbersome to find the right collet, and letting them float around in a cupboard is not a good idea, what is needed is a holder with holes a bit smaller than the max. diameter of the collets. So why not print a holder, a friend of mine has acccess to a 3D printer at work..

Set up FreeCAD (v 0.18) workbox to *Part Design* and create a new project (*File*, *New*).

In the Combo **View** *Tasks* tab click *Create* Body, *Create Sketch* and select *XY*-Plane. Set *Grid Size* to 10mm and tick *Grid Snap* and draw a *rectangle* (*Sketch*, *Sketcher Geometries*, Create *Rectangle*) 148 x 100 mm in size, click *Close* when finished. If the rectangle doesn't show up on screen click the fit  button.

In the Tasks tab of the Combo View select the *Pad* option and extrude to a *Depth* of 6mm and click *OK*.

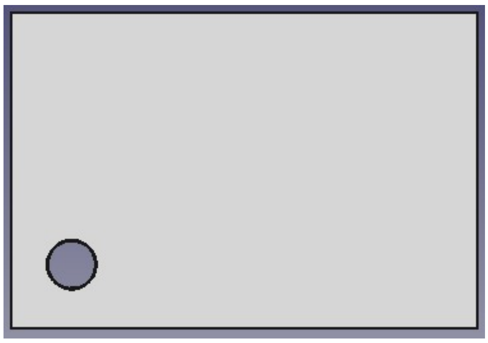


If you click the isometric view button  you should get this:

Switch back to Top View  and click inside the rectangle.

This brings up the different options in the *Tasks* tab, click *Create Sketch* and set *Grid size* to 5mm and tick *Grid Snap*.

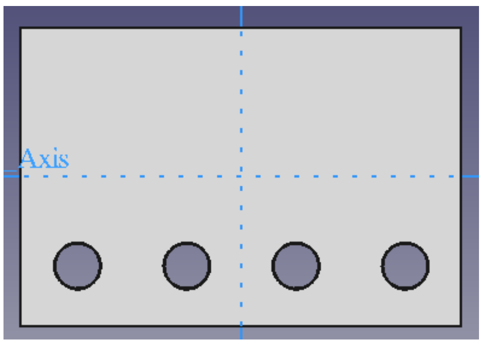
Draw a circle (Sketch, *Sketcher Geometries*, *Create Circle*) with centre at -55,-30 and with a *radius* just under 8mm, click *close* when finished.



Now select *Hole* in the *Tasks* tab, and set *Profile* to *None*, *Depth* to *Through all* and *Diameter* to 15.8mm. You can also set Drill Point Type to Flat and tick *Tapered* to get a tapered hole (86 deg), click *OK* when finished.

Your model should look something like this:

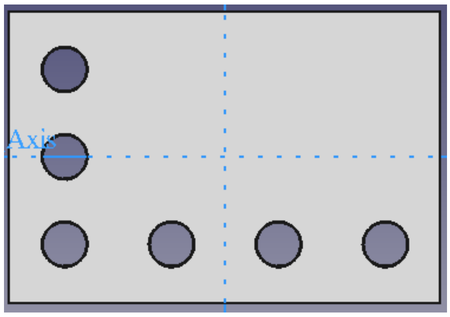
If you click the *Model* tab int the *Combo View* you can right-click on Hole and rename Hole to *FirstHole*, then click the *Tasks* tab.



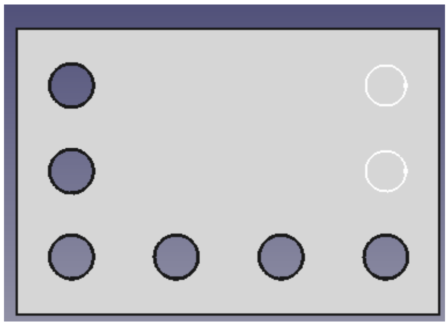
Now select *Part Design*, *LinearPattern* and select FirstHole and set *Direction* to *Horizontal Sketch axis* and *Length* to 110mm and *Occurrences* to 4, click OK when finished.

Your model should look like this:

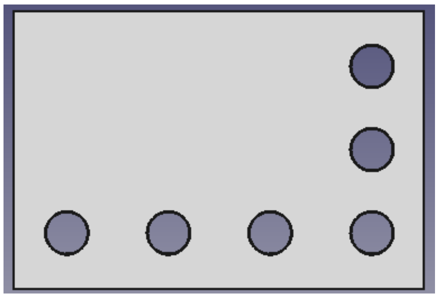
To make it easier to create the other two rows of holes you can use the *LinearPattern* option to create 3 holes in the other direction, just remember to set Direction to *Vertical Sketch axis*, *Length* to 60mm and *Occurrences* to 3. The two new holes will be deleted later, your model should look like this:



Now click inside the rectangle (not in the holes) and create a new *Sketch* and create two new *circles* in the right hand part of the rectangle, like this:

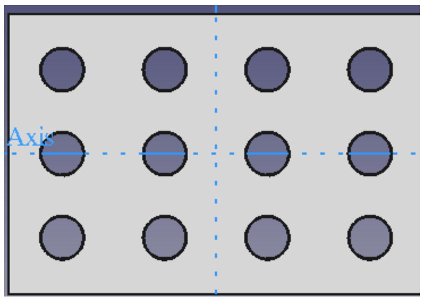


Now select *Hole* in the tasks tab (or click on one of the circles you just made) and create two new *holes* with *Profile* set to *None*, *Diameter* to 15.8 (as you did with the first hole). The hole can be renamed to SecondHole. You can also select the two temporary holes to the left and press the *Delete* button. You may have to click the circle you just made and repeat the *Hole* procedure. Your model should look like this:



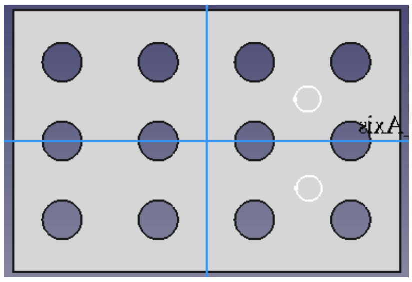
You can now use the LinearPattern feature to make the last two rows of holes, click on one of the SecondHole and just remember to tick *Reverse Direction*, *Length* to 110mm and *Occurrences* to 4.

Your model should now look like this:



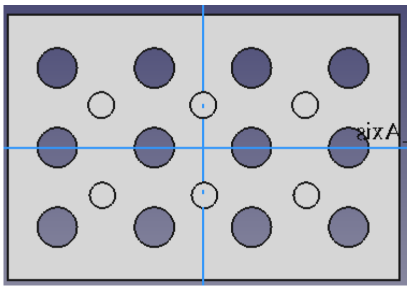
Next job is to make some feet on the underside so click  *to select Bottom View,* and click inside the rectangle (not inside a hole).

In the *Tasks* tab click *Create Sketch* and create a new sketch and draw two *circles* with *radius* 5mm between the rows, and click *Close*. Your model should look something like this:



In the *Tasks* tab select *Pad* to extrude to a *length* of 18mm and click *OK*.

Then click on one foot to select it and then *Part Design*, *LinearPattern* and set *Occurrences* to 3 and adjust length so the middle foot ends up in the middle, like this:



To make a foot in each corner you just repeat the procedure but set *Occurrences* to 2 and adjust Length so the feet end up in the corners.

Your model should look something like this when seen from the bottom: