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 access 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: