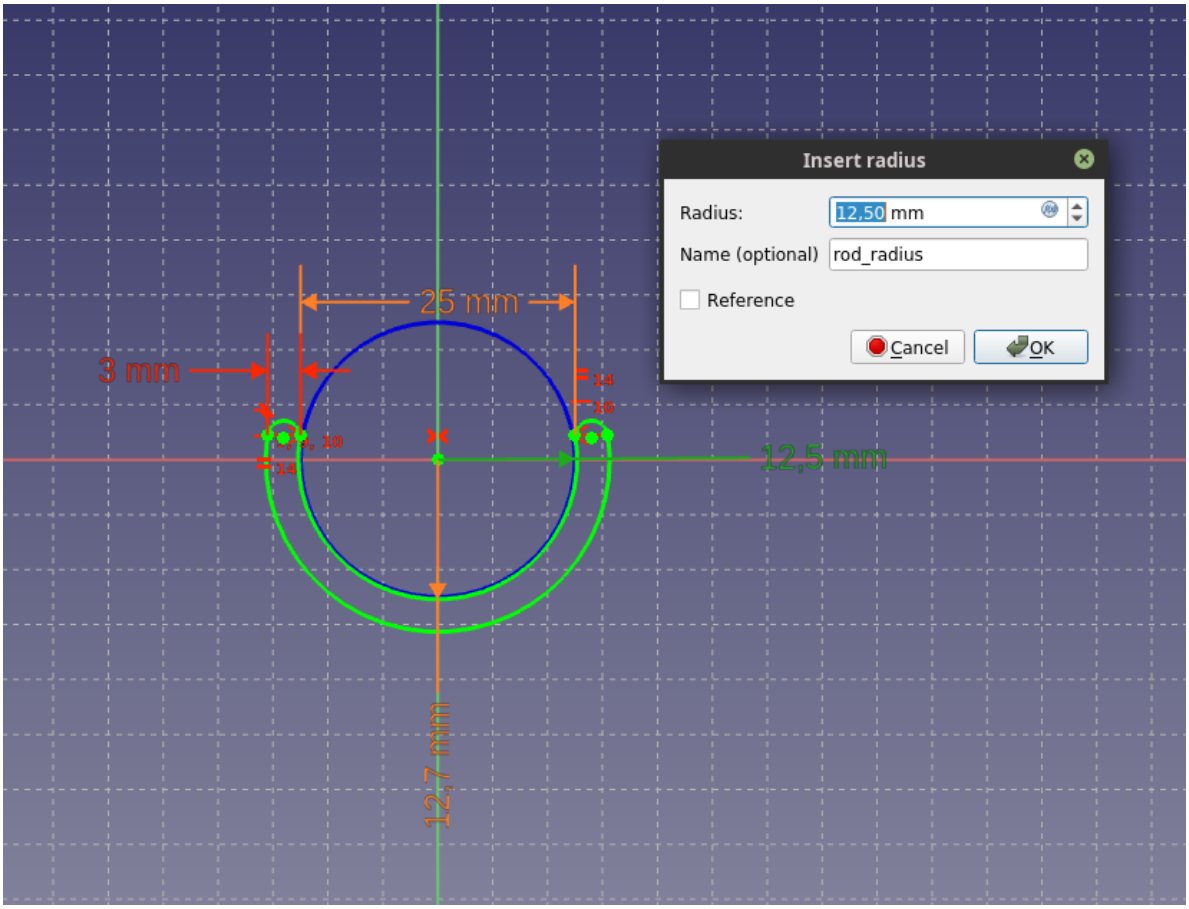
SIMPLE FULLY PARAMETRIC DESIGN WITH NAMED CONSTRAINTS IN FREECAD

(V2, by https://github.com/y1n-dev)

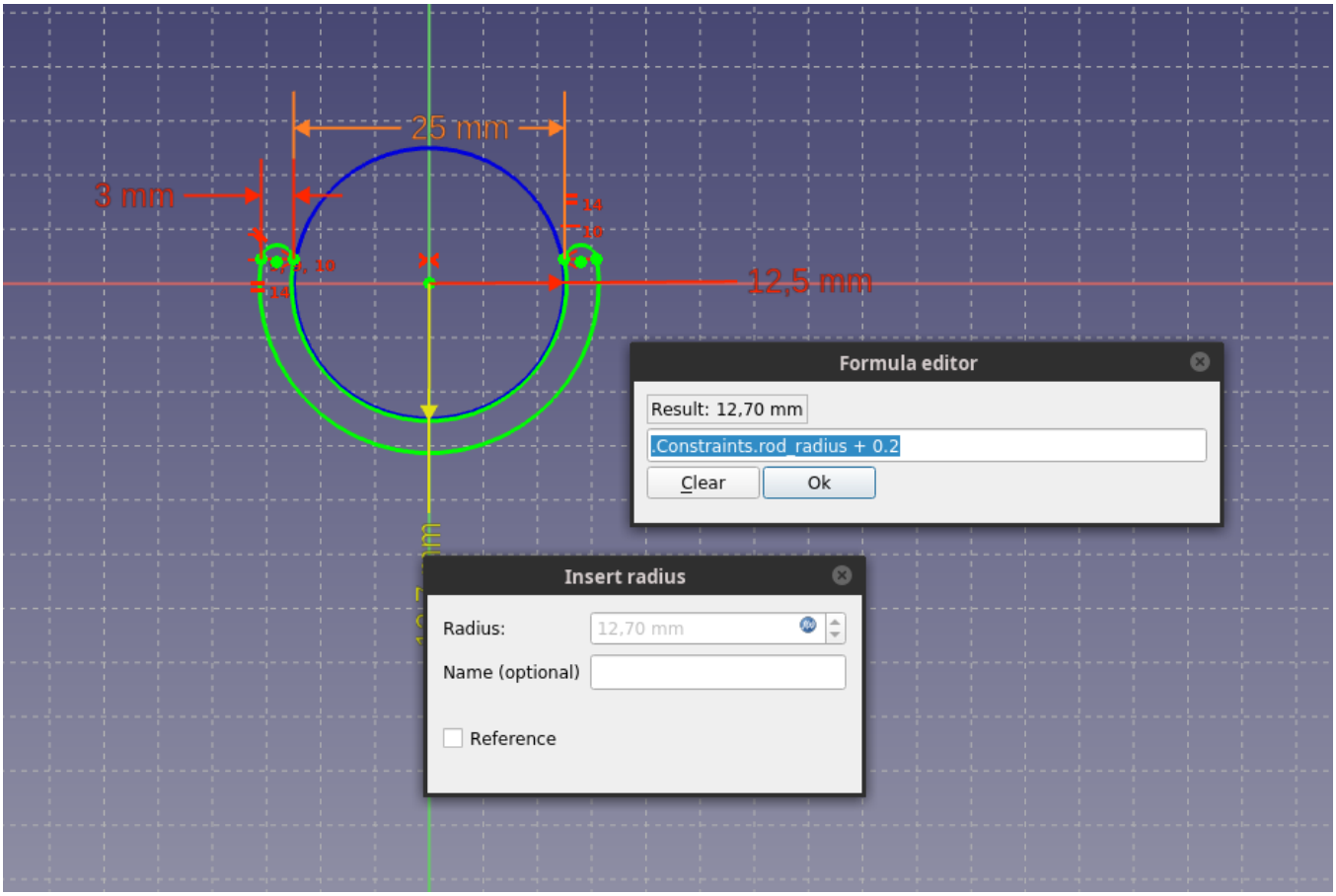
I was wondering why ppl strugggle with SPREADSHEETs, when there is a much simpler way by using NAMED CONSTRAINTS.
Especially for smaller designs this method is easy to implement or you can adopt your existing designs to it.

You can see a practically example and how to use it (and download it) here: https://youtu.be/_qdGifwaOi4 or take a look on the finale picture to get an overview.

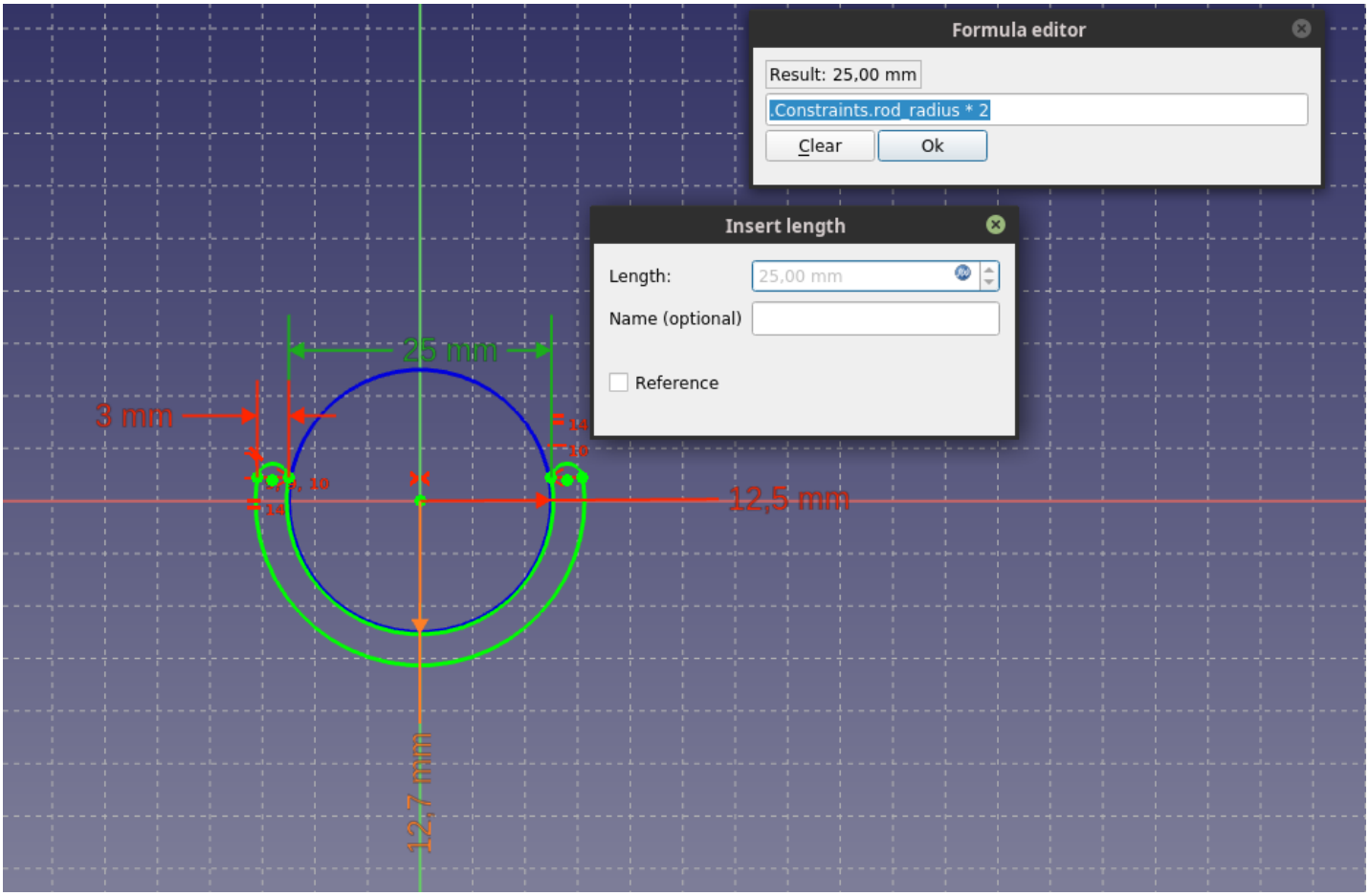
Things go easy when you have parametric design in mind and create at first a reference constraint (in this example the constructing geometry circle radius) and let other constraints point to it:



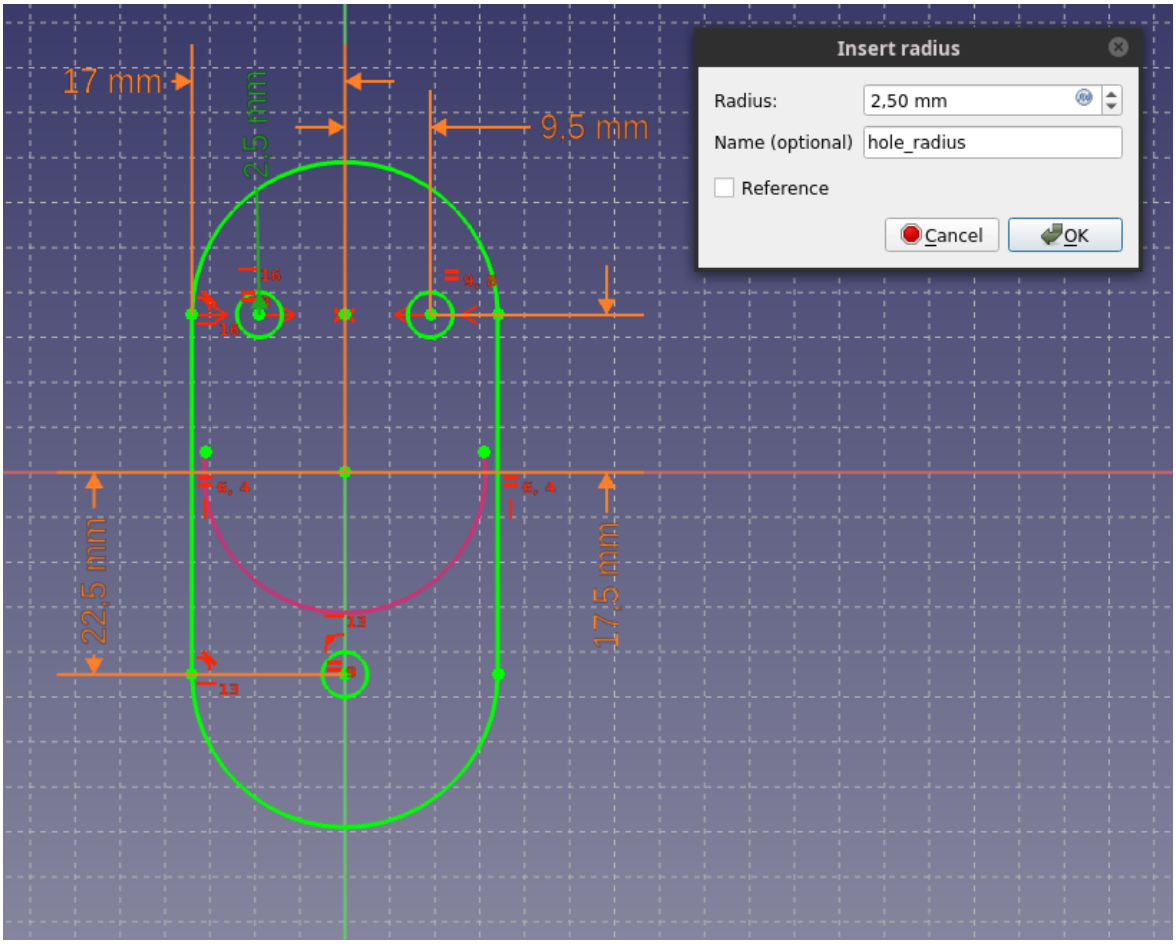
So the given radius is 12.5mm and the given name is 'rod_radius', in the next step step we calculate the radius of the arc with a formula (you can define it by clicking  the icon next to the value) '.Constraints.rod_radius' points to the value for the circle radius that we set and named in the previous step. '.Constraints.rod_radius + 0.2' gives a result of 12.7mm for the radius of the arc. If we have in mind to use a 25mm rod this give us a 0.2mm gap, but the opening of the mount shall be very tight and is therefor exactly like the rod-diameter (I had 3D-printing in mind while constructing). Please see below:



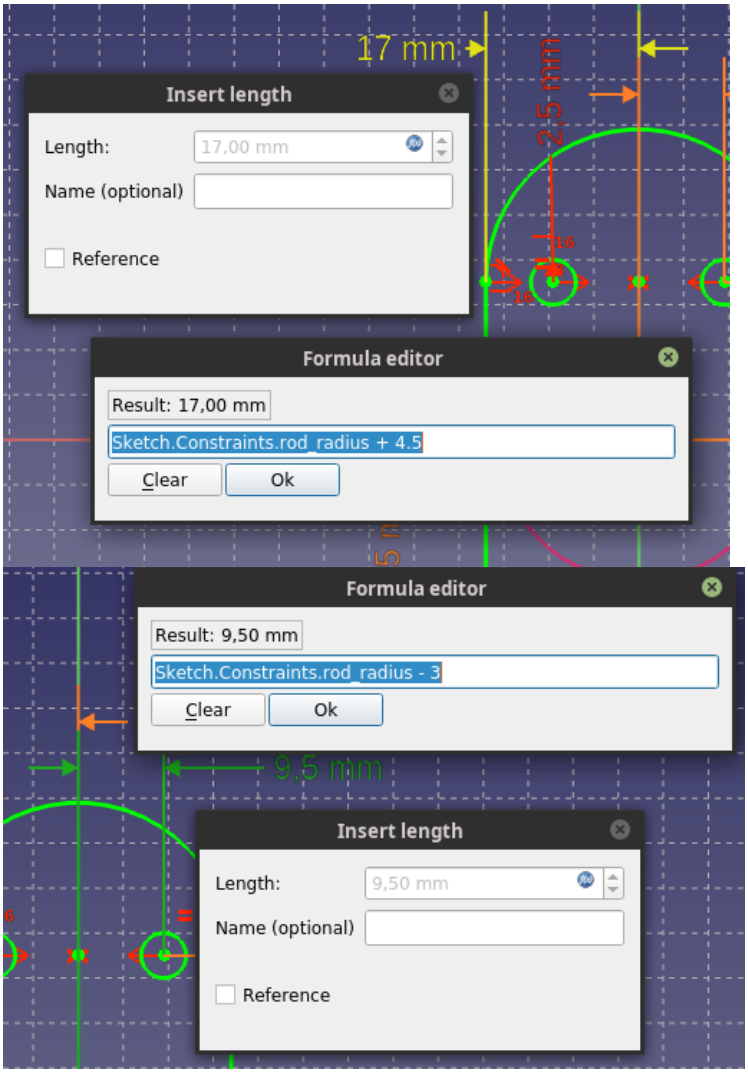
in the next step we set the opening for the rod mount by multiplying '.Constraints.rod_radius' with 2

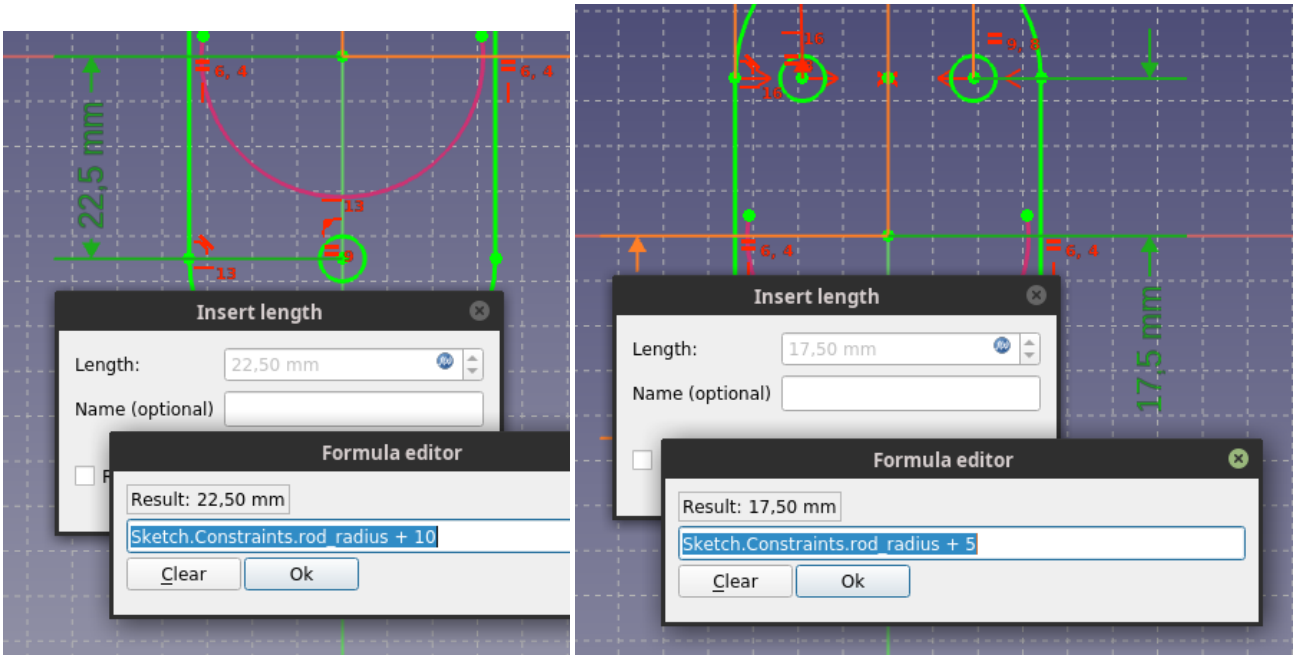


The radius for the holes get named too:

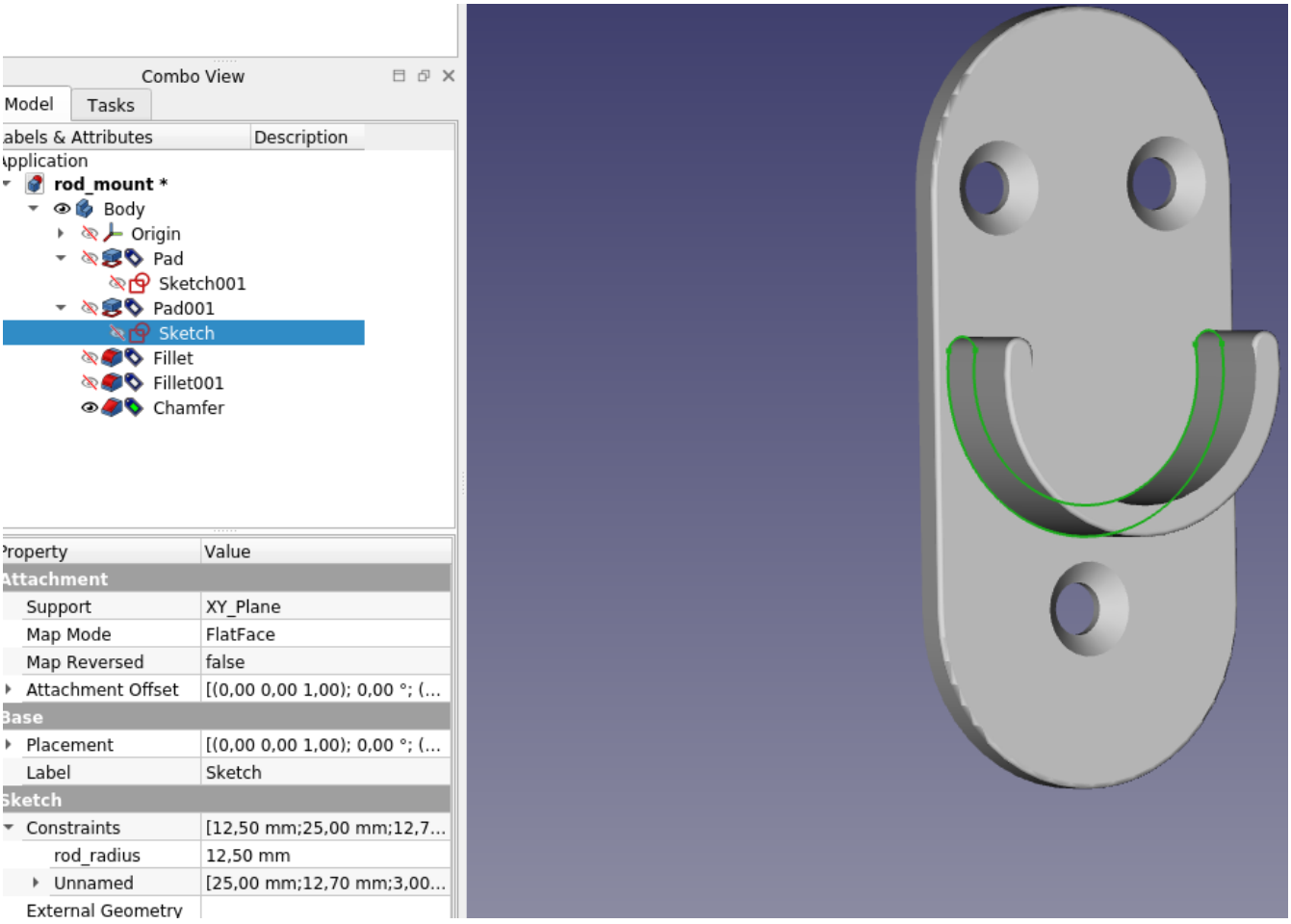


All values in orange (picture above) are all calculate referring to '.Constraints.rod_radius' (see below):





Now you can simple adapt the design by changing the values of your named constraints by selecting the sketch in 'Data-Sketch-Constraints-rod-radius' and click 'Recompute' (take a look at the Youtube-Video at the beginning).



Hope this helps one or another ☺
All the best to you

System-information:
FreeCad:
OS: Linux Mint 19
Word size of OS: 64-bit
Word size of FreeCAD: 64-bit
Version: 2020.10.22.22739 +2688 (Git) AppImage
Branch: LinkStage3
Python version: 3.8.6
Qt version: 5.12.9
Coin version: 4.0.0

OCC version: 7.4.0
Locale: English/United States (en_US)

Thanks to https://github.com/macdroid53 for proofreading the article.