FreeCAD Legacy v0.16 General Tutorial for Open Source Ecology

Unconstrained revolved sketch demo

a. Torus
b. Square
c. Constrain Circle to axis
d. Constrain Fully away from origin for revolve
e. Note how position about local origin results effects translations in global such as rotation
f. Add meta data for documentation
g. Add radius constraint as diameter /2 to show math

Constrained square frame

a. Demo symmetry constraints about origin
b. delete outer square
c. Show pad change
d. Redo outer square with equal sides
e. Discuss possible shapes and varied geometry with symmetry
f. Importance of symmetry in design and throughout freecad

Pad and Pockets settings

a. Demo with multiple pads and pockets
b.
c.
d.
e.
f.

Multi parts demo

a. Change sketch to break features
b. Restore sketch
c. Observe free sketch rotation
d. move free sketch for keyhole position
e. Adjust pad for depth
f. Reorient sketch
g. Remap sketch to same plane
h. Reorient sketch
i. Move sketch to same plane (defaults to origin- Note offset from plane only)
j. Note cube position x,y,z
k. Move sketch to cube origin @ x,y,z
l. Demo pad adjacent to cube
m. Move sketch off cube face
n. Reverse pad to demo pad opposite side adjacent to cube
o. Show all 3 folders and note created groups for folder organization
p. Review all and close

Applicable Logic to use Assembly 2 WB (PC v18.01 case study)

Parts/Modules with permanent fixed spatial relationships where potential scaling & changes are allowed by assembly constraints such that files update automatically through assemblies for faster editing and multiple part options.

a. Demonstrate how this logic can be used on simple frame assemblies
b.
c.
d.
e.
f.

Image tracing in FreeCAD

- The GC motor file is a good example for demoing a variety of concepts in freecad. https://wiki.opensourceecology.org/wiki/File:GC_Wheel_Motor.fcstd
- The 2D CAD images for parts are output from CAD and convey the basic info to recreate rough dimensions of parts, which is ideal for minimal part/file size and major dimensions needed for using the parts in larger machine design. Details like chamfers and fillets just add excess.
- The more literally the 2D CAD drawings are interpreted the quicker and more easily it can be drawn.
- Scaling and moving the image can be done in FreeCAD directly, but other apps may be ideal as well if more precision is needed.
- Tracing sketches over the image is a good start and check for scale, but in the end the sketches likely need to be constrained to the origin.
- The image can be moved or simply rotated for view the sketch started over the needed portion of the image and then constrained elsewhere after dimensions are verified.
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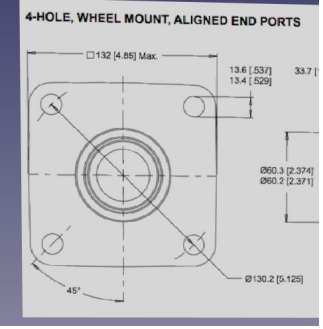
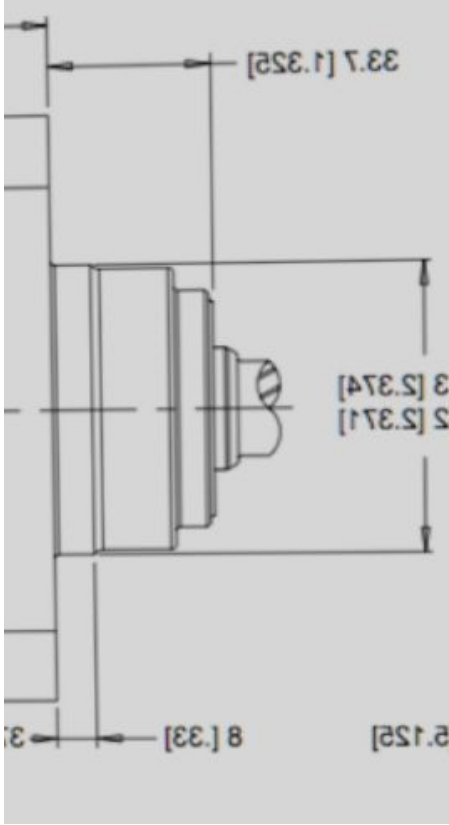
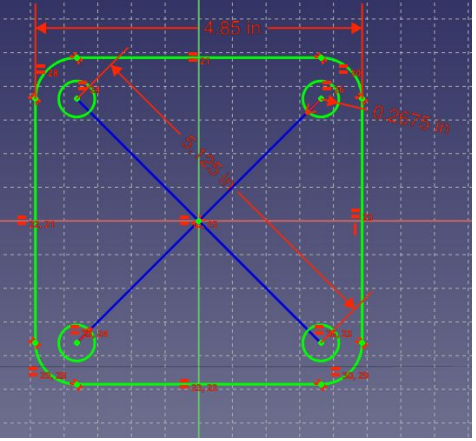


Typically I have started sketching these types of hydraulic parts starting by sketching the bolt mount pattern (left) and then adding more pads on each side of that. But, after looking closer at the drawing and thinking about the fastest process I think starting with the round front parts (right) would be faster due to fewer steps.

This is because the fastest way to draw it in 3D it is by tracing a profile of the 2D directly using the given dimensions and with some construction lines to eliminate the need to calculate the numerical values left out. Then that half profile can be revolved making a single feature instead of many pads. The shaft dimensions are not shown in this image, but it could be included in the revolved sketch as well. Other features can be added as pads later.





After the revolve the bolt pattern feature can easily be added as a pad to the back of the revolve. To continue using the reference image it will need to be rotated. The image could be moved to the axis origin for direct tracing, but this is time consuming and unneeded. The 2D drawing dimensions give the best clues as to how to quickly draw and constrain the sketches. The diagonal dimension is key.