



**SPARK
STUDIO
SALEM**
MAKER COMMUNITY

Intro to FreeCAD

Jon Arney

What is it for?

- Lots of “3d” software out there
- Each has a set of use-cases that it’s designed for
- OpenSCAD does solid geometry
- Blender does “surface” models as polygon meshes
- FreeCAD focuses on “engineering” use-cases (i.e. parts that can be machined)

Other Resources

- YouTube: MangoJellySolutions
 - Great tutorial series on FreeCAD
 - Suggest you watch each video and try out the demonstrations yourself for each one.
 - Reinforces the learning and lets you take it step by step

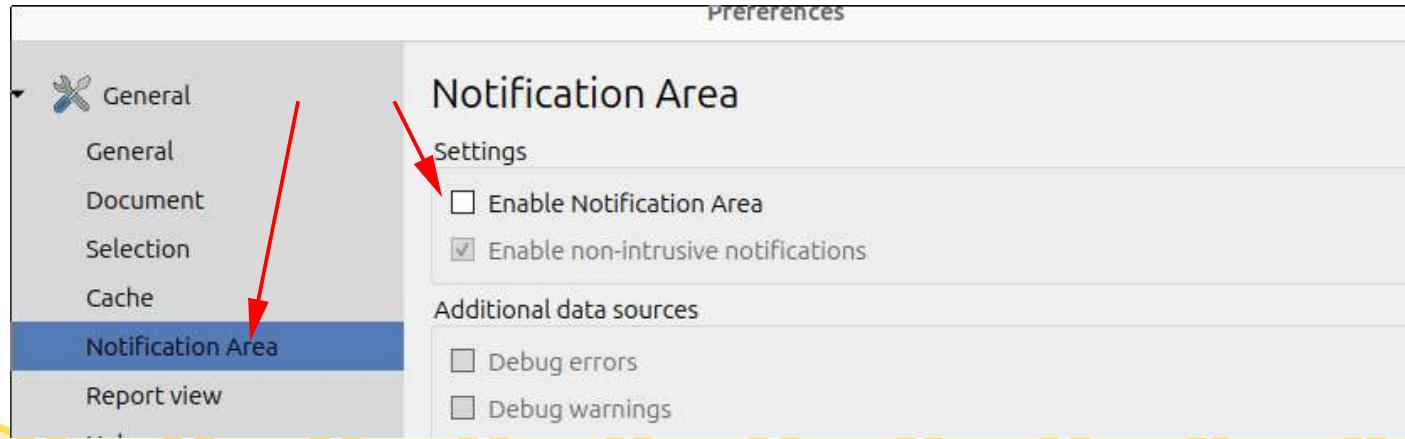
Versus other similar CAD lots of options out there

- FreeCAD
 - Open-source
 - Open XML-based format
 - Exports to STEP, STL, Gcode, SVG, (and more)
 - You own the files FOREVER
 - Bad (undeserved, IMHO) reputation
- Fusion360/AutoCAD
 - Requires cloud (very dependent on cloud for operation)
 - Much more fully-featured and polished
 - Much more industry-recognized, but can be expensive



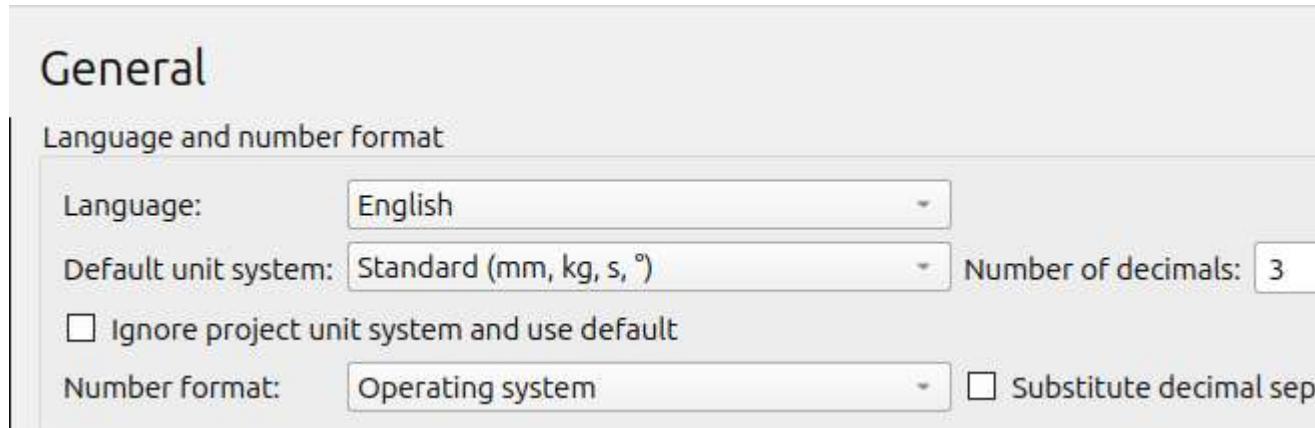
Getting Started

- Install it, see <https://www.freecad.org/> for installation instructions for your platform.
- Edit → Preferences
- Notification Area: Disable “Enable Notification Area” as shown

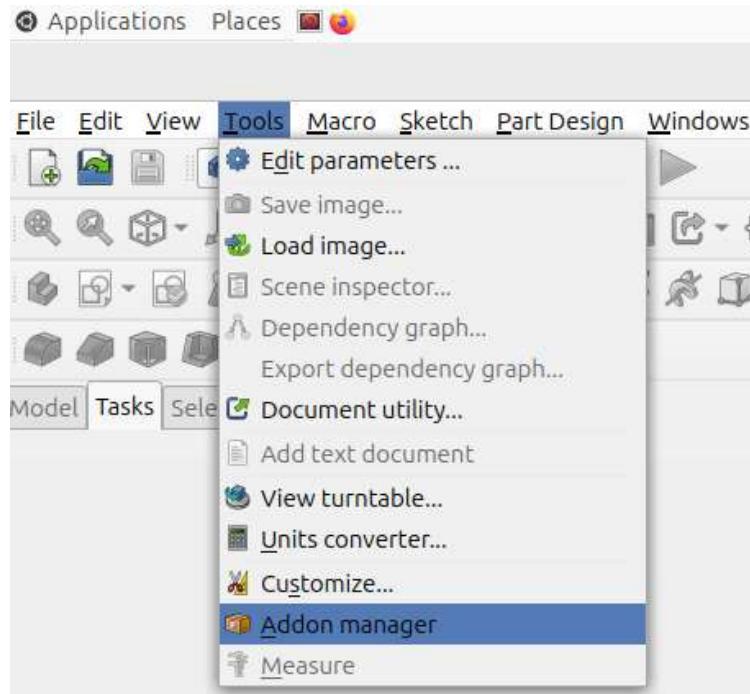


Units (Highly recommend mm for distances)

- Also recommend having a good set of calipers to reference real-world objects like bolts, shafts, bearings, etc.

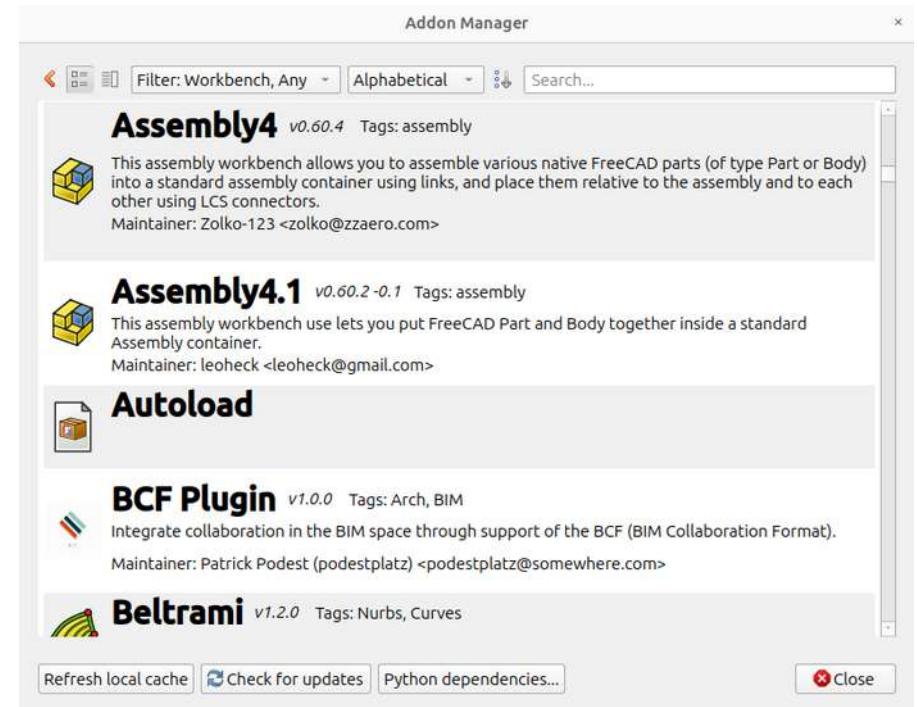


Add some useful add-ons



Install the add-ons

- Assembly4.1
- Fasteners (may be pre-installed)
- Freecad.gears
- LCInterlocking

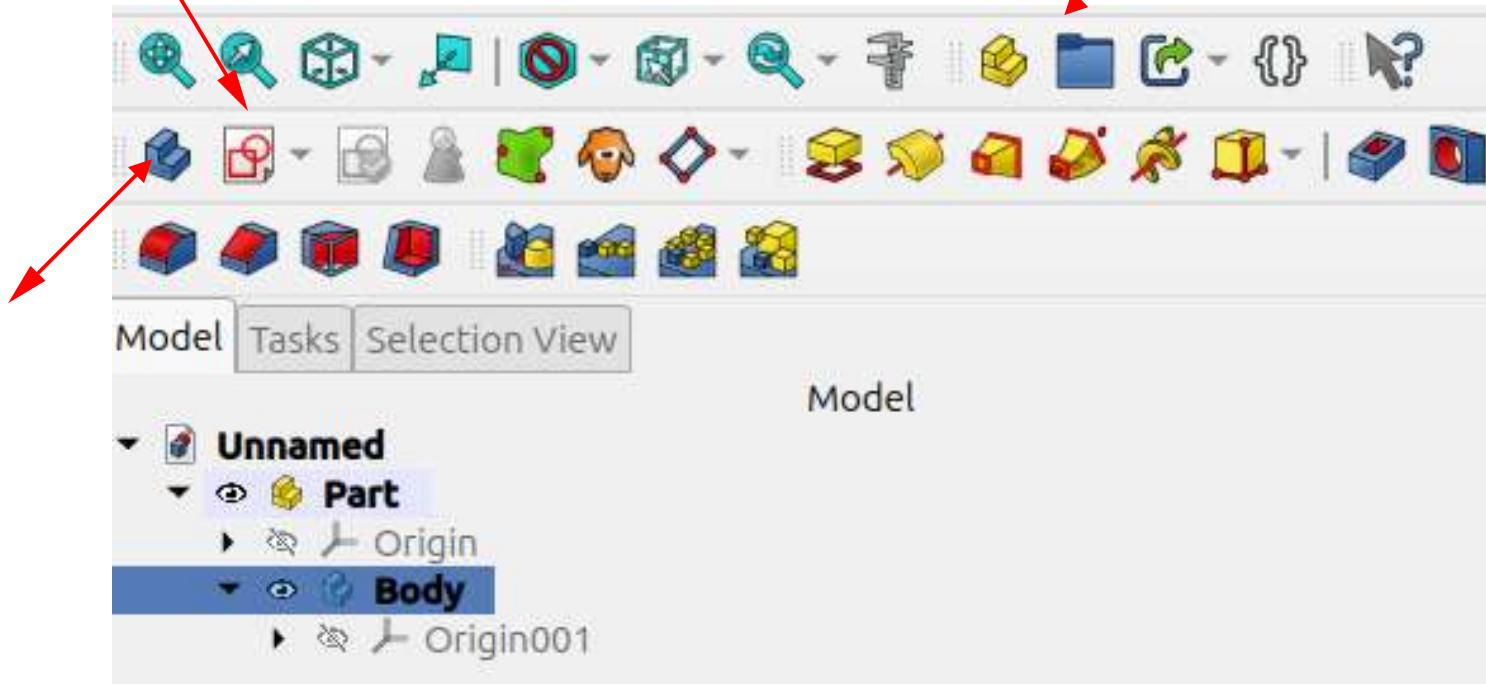


Normal workflow: Part Design Workbench

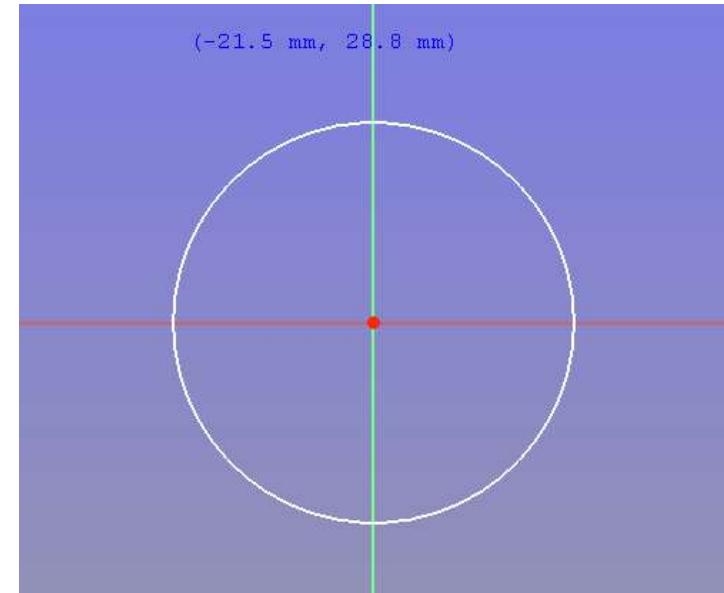
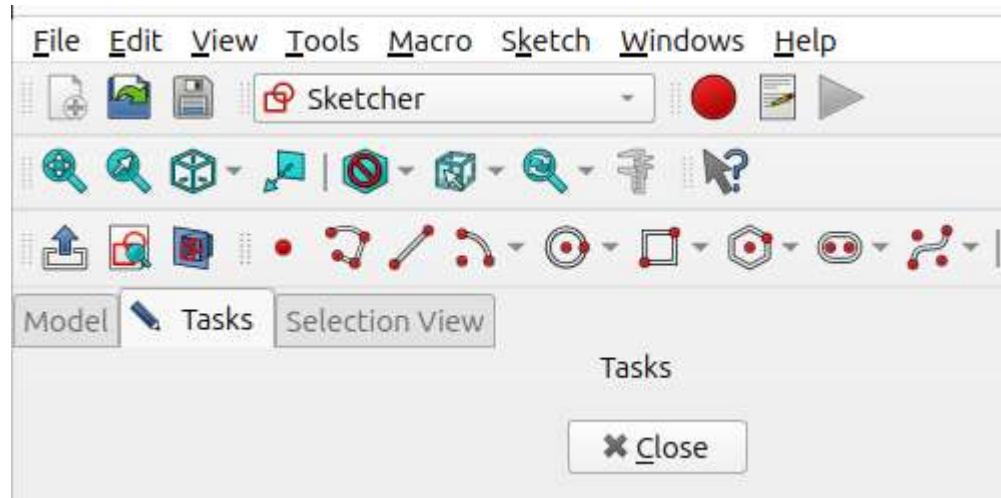
- Create a “Part”
- Create a “Body” inside the part
- Create a “Sketch”, a 2d-plane drawing
- Extrude the “Sketch” along an axis.
- Create another “Sketch” and cut shapes in it.
- Create a “Local Coordinate System” to note specific features so we can assemble it later.



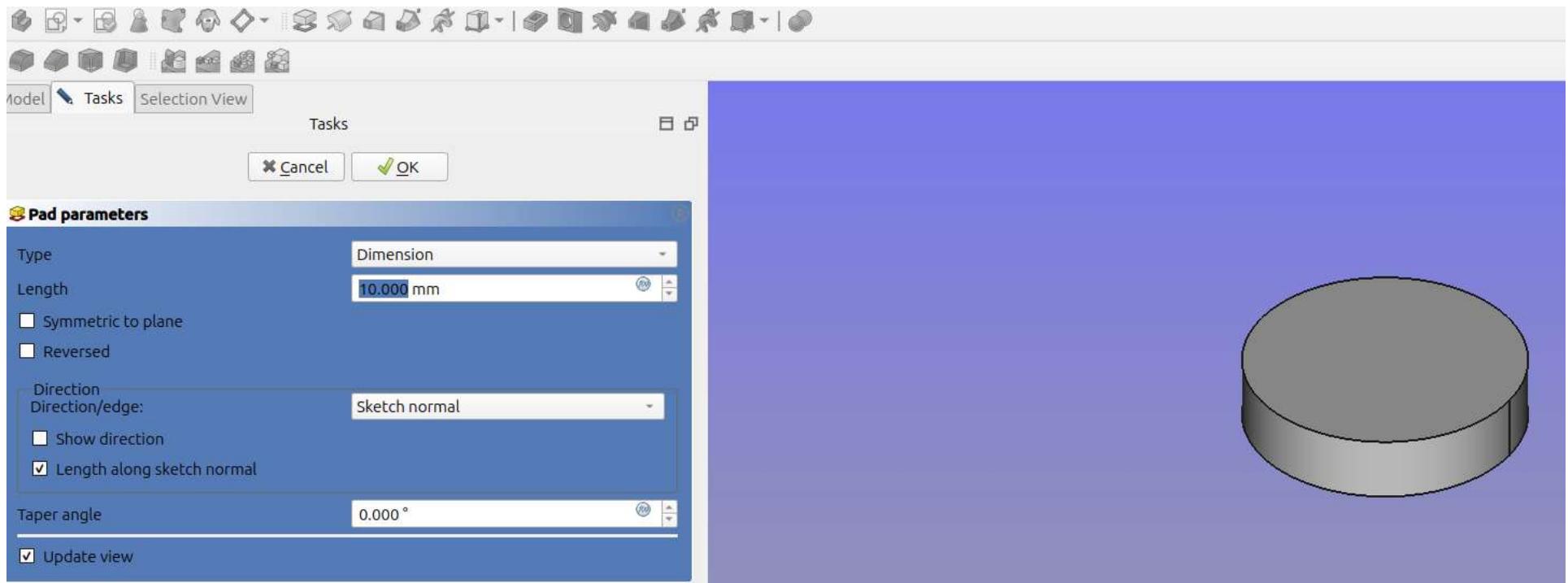
Add a “part” and to that, add a “body”



Create a circle and “Close” the sketch



“PAD” the sketch to extrude it



Modify the base shape (cut some holes)

- Another sketch perhaps at a different angle
- Sketch will be to cut a hole
- Use sketch to “pocket” the base shape
- Pattern the pocket to repeat the cut in different places
- Workflow follows similar manufacturing techniques
 - Start with a base of material
 - Cut away until the final shape is achieved
 - The workflow in the CAD software mirrors “traditional” manufacturing techniques
 - Resulting shape can be 3d printed OR machined using CNC mill or lathe

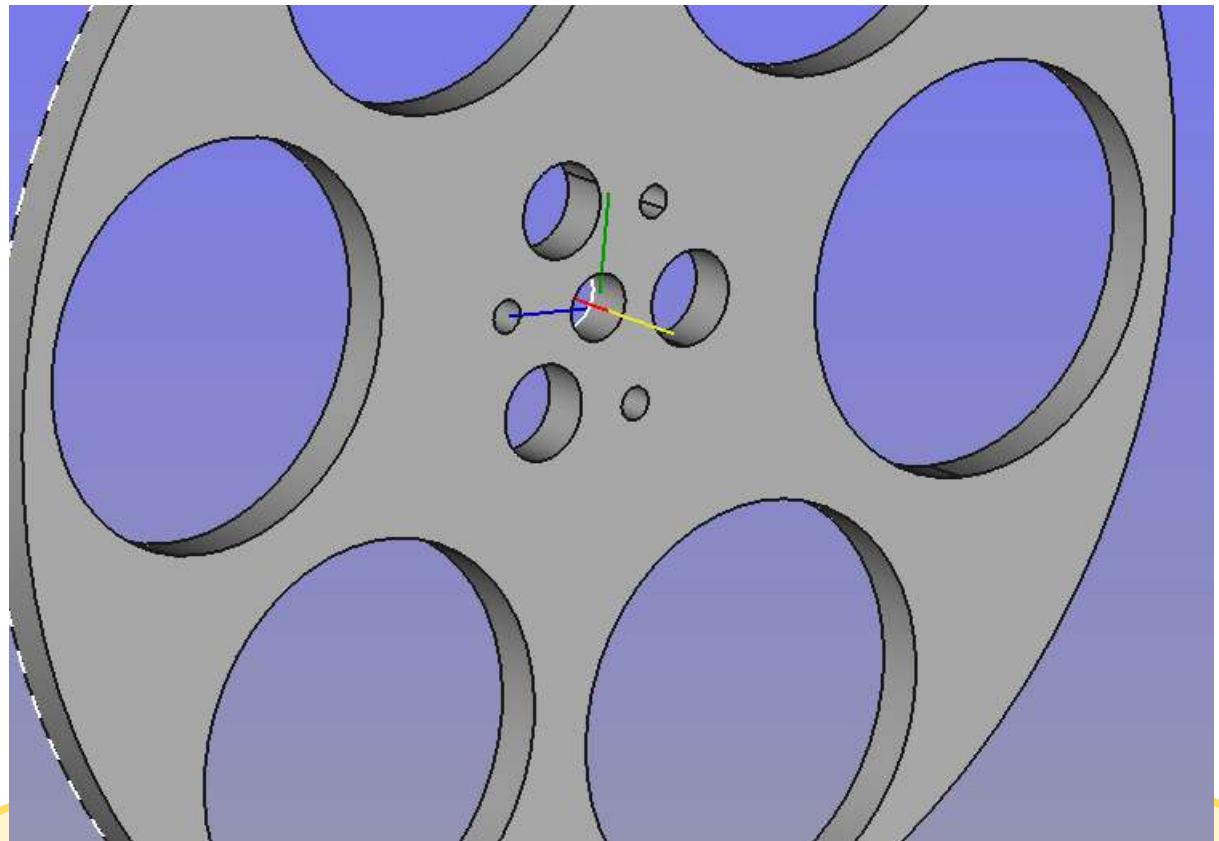
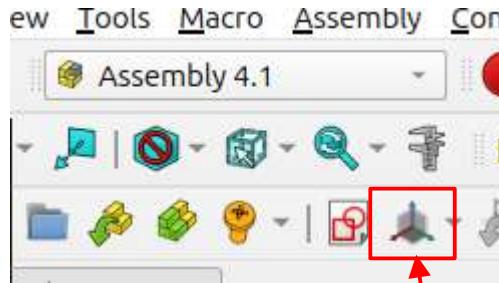


Parts to assemblies

- When you have a rough draft of the parts, you can assemble them into a larger machine.
- Assembly workbench does this.
- The approach is that each part gets one or more “Local Coordinate Systems (Datum)”.
- These are used to place parts on other parts
- In this way, an assembly can be put together from parts
- Workflow (for me) is to draft a “rough draft” of each part and assemble them.
- Then refine each part until it meets my needs.
- Limitation: One assembly per file
- Feature: Assemblies can use parts from other files, so you can create REALLY BIG assemblies if you need.

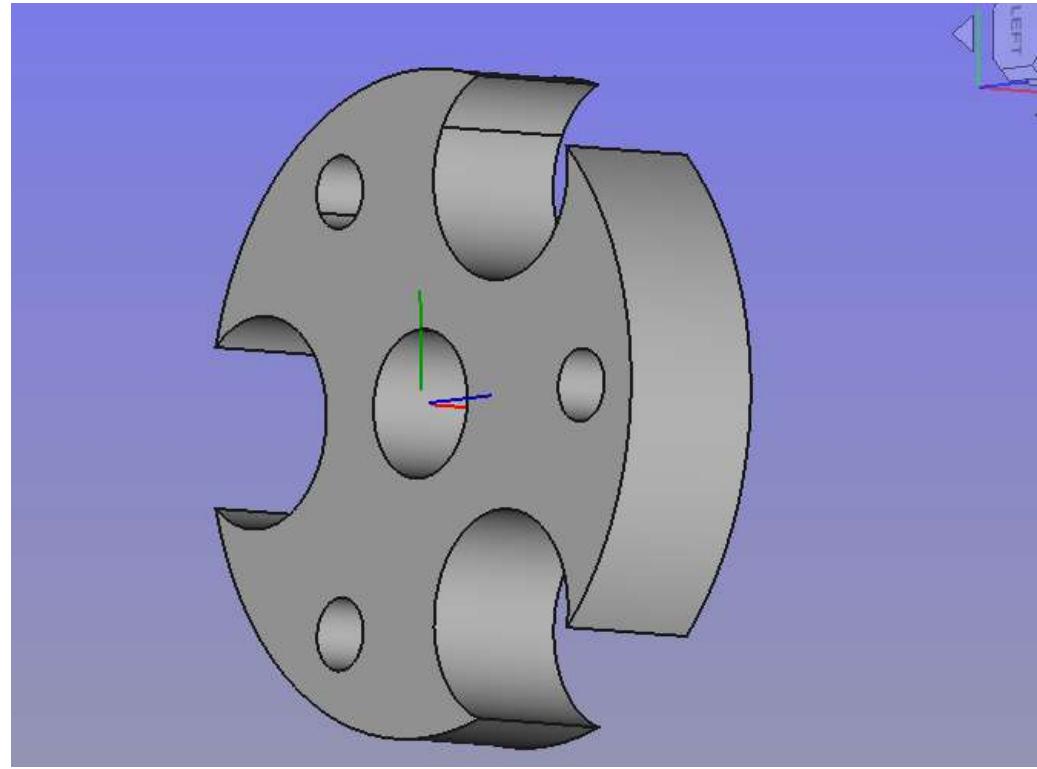
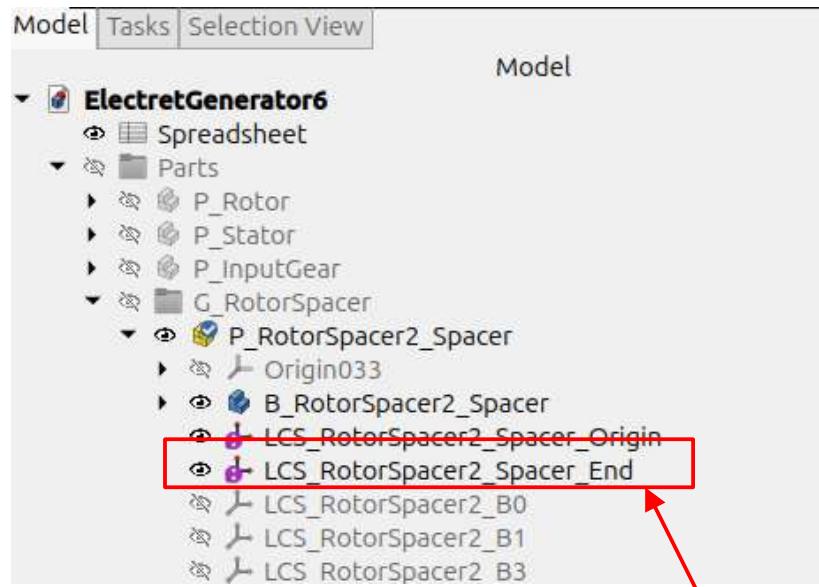
Place coordinate system on rotor

- Place a coordinate system on the rotor
- Give them meaningful names



Place coordinate system on spacer

- Another one on the spacer



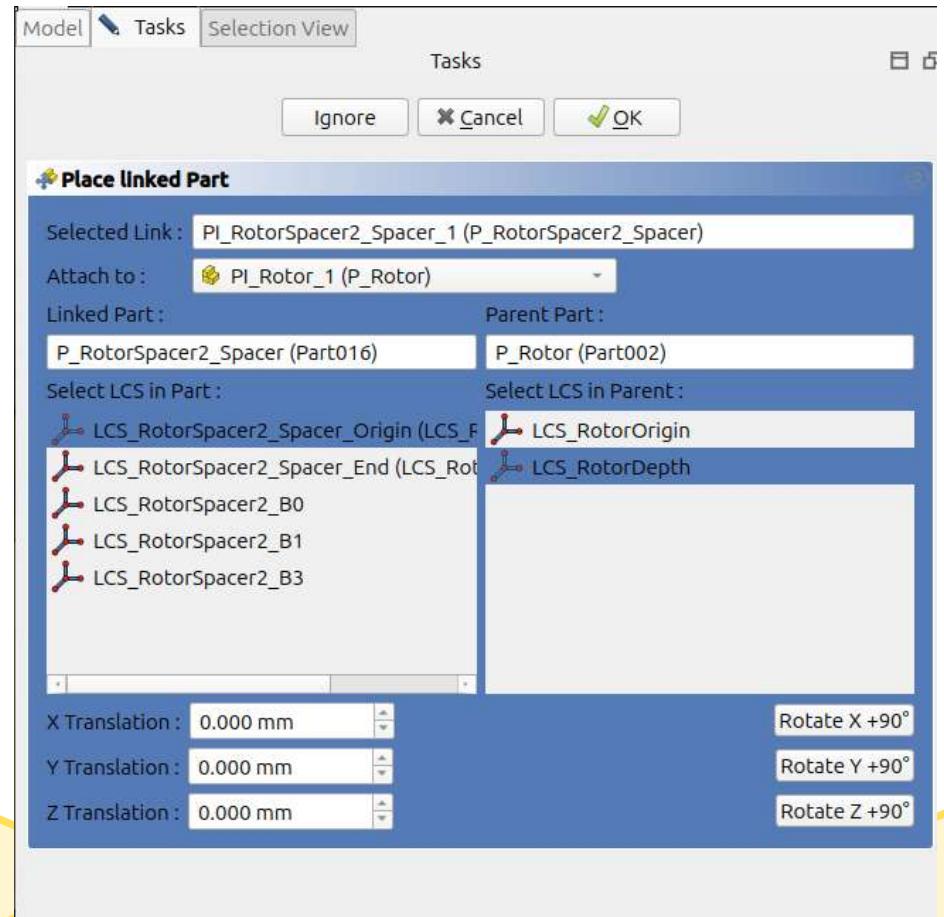
Insert Part, place at origin

- Select some part to serve as the “root” or “origin” and place it in the assembly
- All other parts will eventually connect back to it, this is usually the “base” or “case” of the machine.



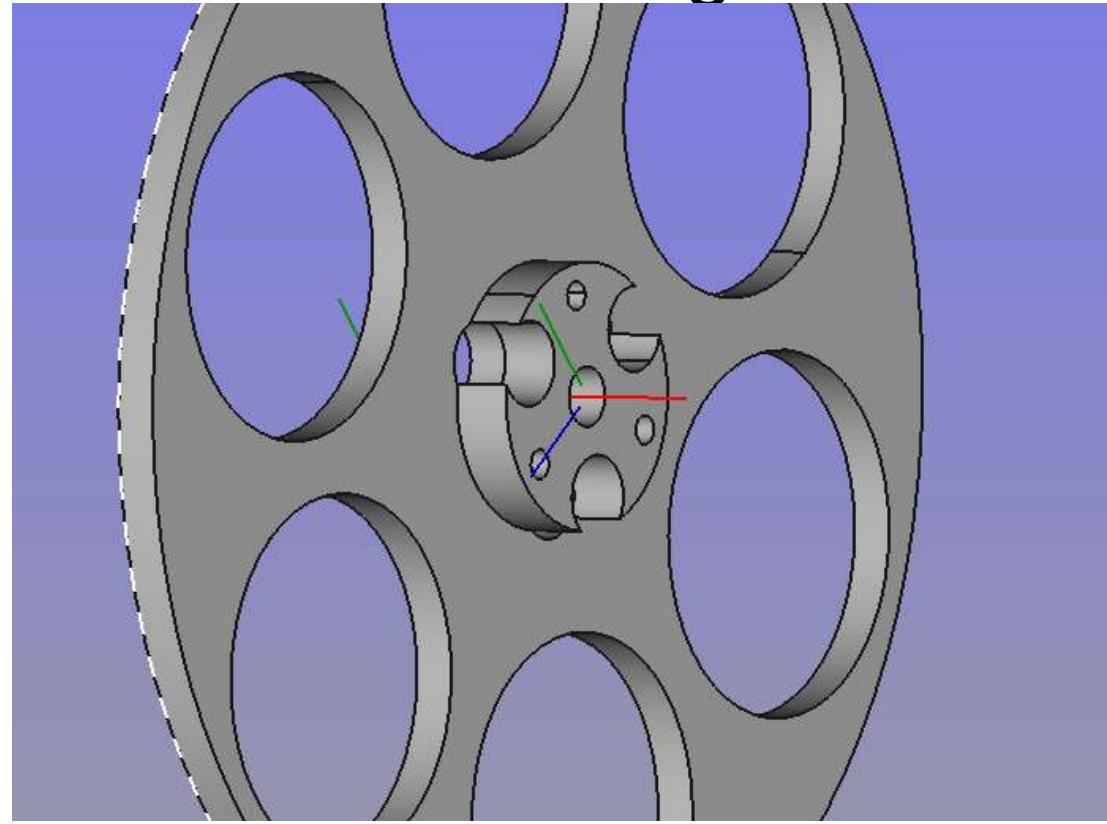
Attaching one part to another.

- Spacer part being inserted
- Attach spacer's origin to rotor's end
- If you change one or the other of the parts, the whole assembly will update



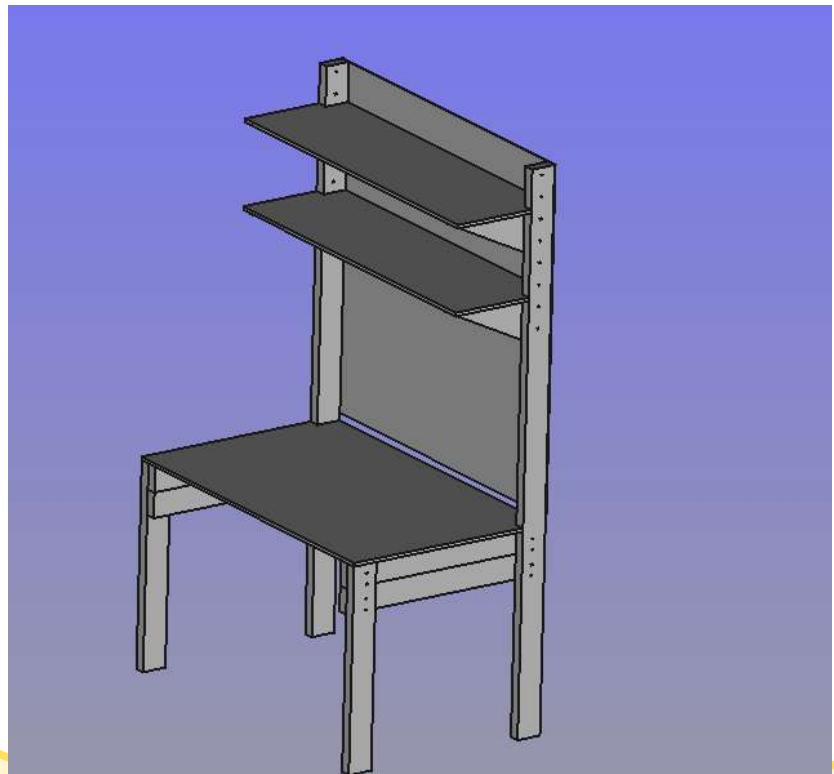
Insert next part and “attach” them together

- This is where names get really helpful
- Usually name the LCS for what is supposed to attach to it.

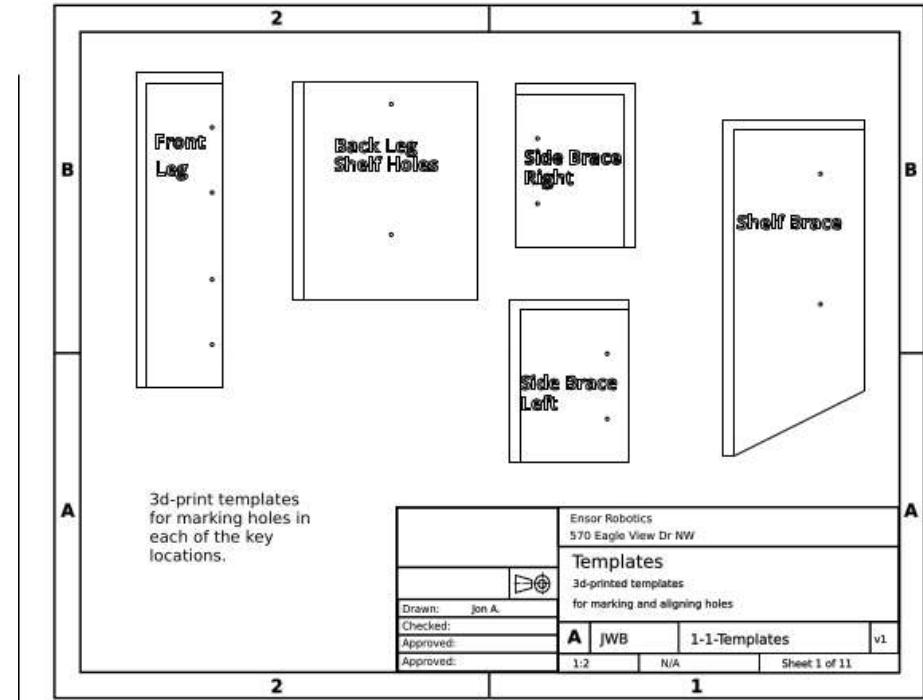
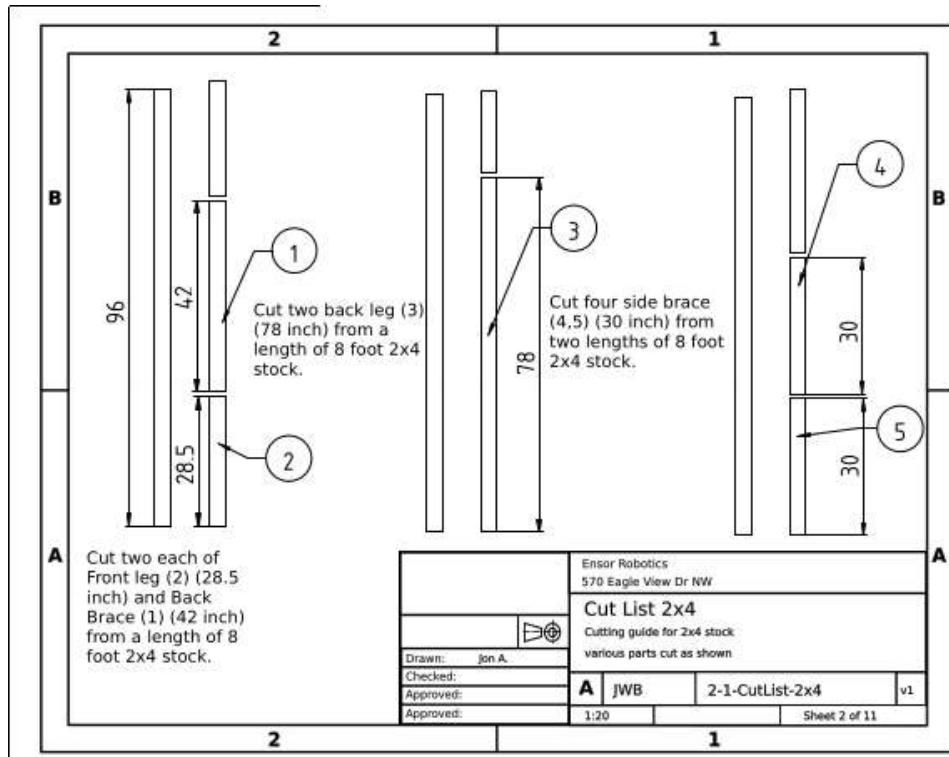


Assemblies: Not just for 3d printing

- The Idea
 - Work surface
 - Pegboard behind
 - Shelving above
 - Sturdy base
 - Repeatable



Export engineering drawings



Finished Project

- I have 3 of them now
- All built on the same plan
- Each takes about 4-5 hours to cut and assemble
- Cost is just the wood and the bolts



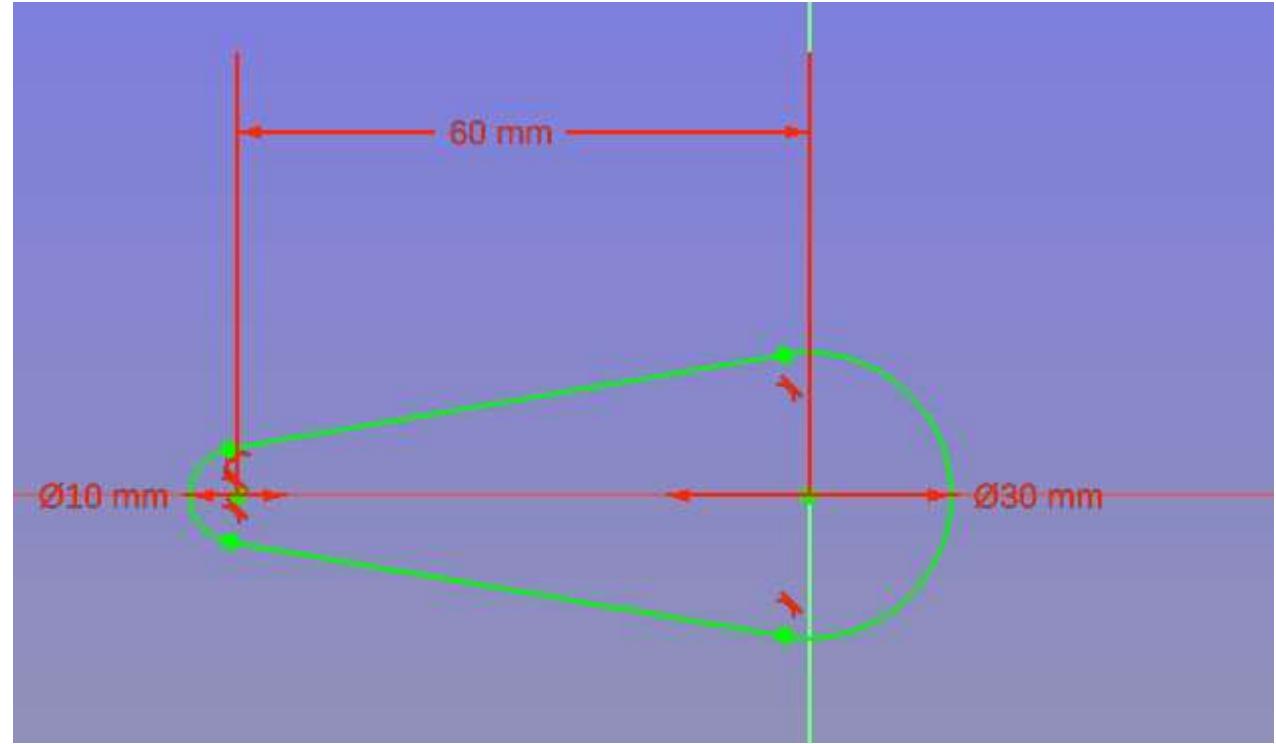
Advanced topics

- Fully constraining sketches
- Don't just "Draw" a sketch. Constrain it.
- If two lines are supposed to be tangent, that's a constraint.
- The constraint solver lets you specify things like parallel, equal length, specific angle, horizontally aligned, as well as specific lengths and diameters.
- Good practice to "fully constrain" designs so you're not taking any chances or "just winging it". Remember, FreeCAD is supposed to be about "engineering" not just drawing.



Constraints

- Lines tangent to arcs
- Arc centers specific distance
- Arc radii specified
- Only 3 independent numbers completely specify the shape



Advanced topics

- Parametric design
 - Instead of using physical dimensions, you can symbolically reference dimensions.
 - Spreadsheet to track “critical” dimensions
 - Reference those symbols in the design
 - When those dimensions change, the whole model updates.

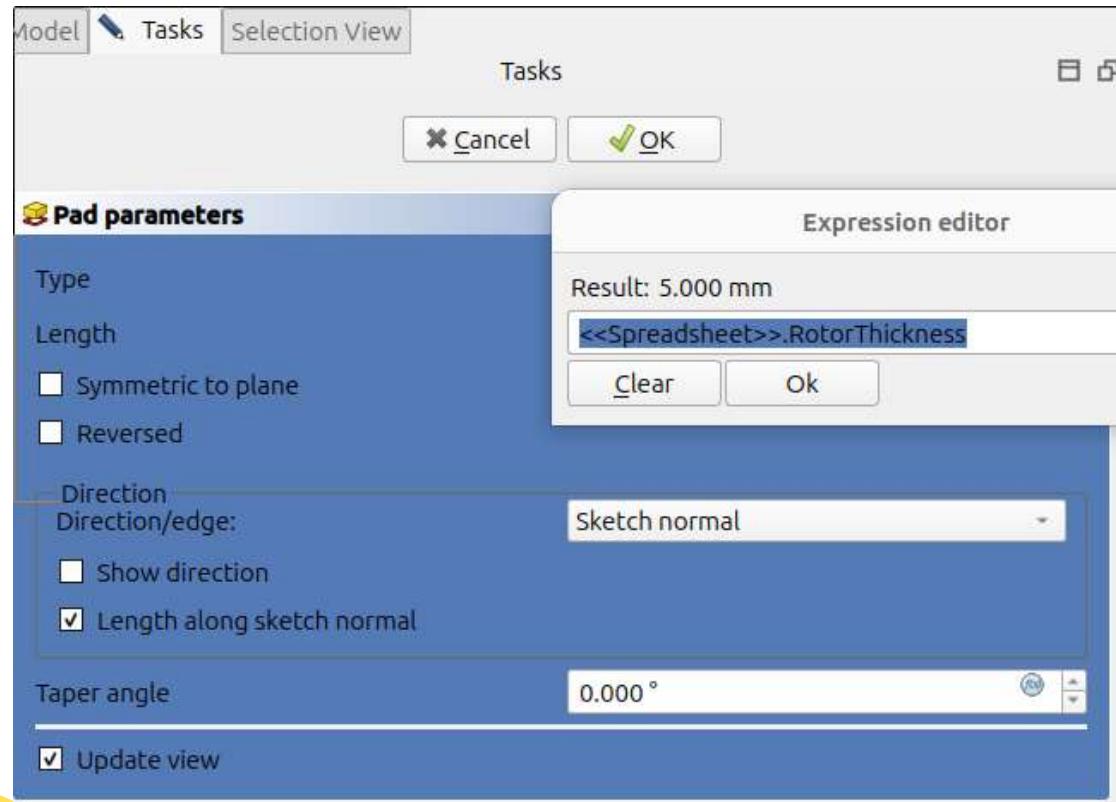
Spreadsheet to parameterize model

- Cells in spreadsheet can have an “Alias” in this case, “Rotor Thickness”.

	A	B	C	D
14				
15				
16				
17				
18	Rotor			
19	Thickness	5.000 mm		
20	Diameter	150.000 mm		
21	KeyholeDiameter	2.900 mm		3.
22	KeyholeOffset	12.000 mm		
23	Stator			

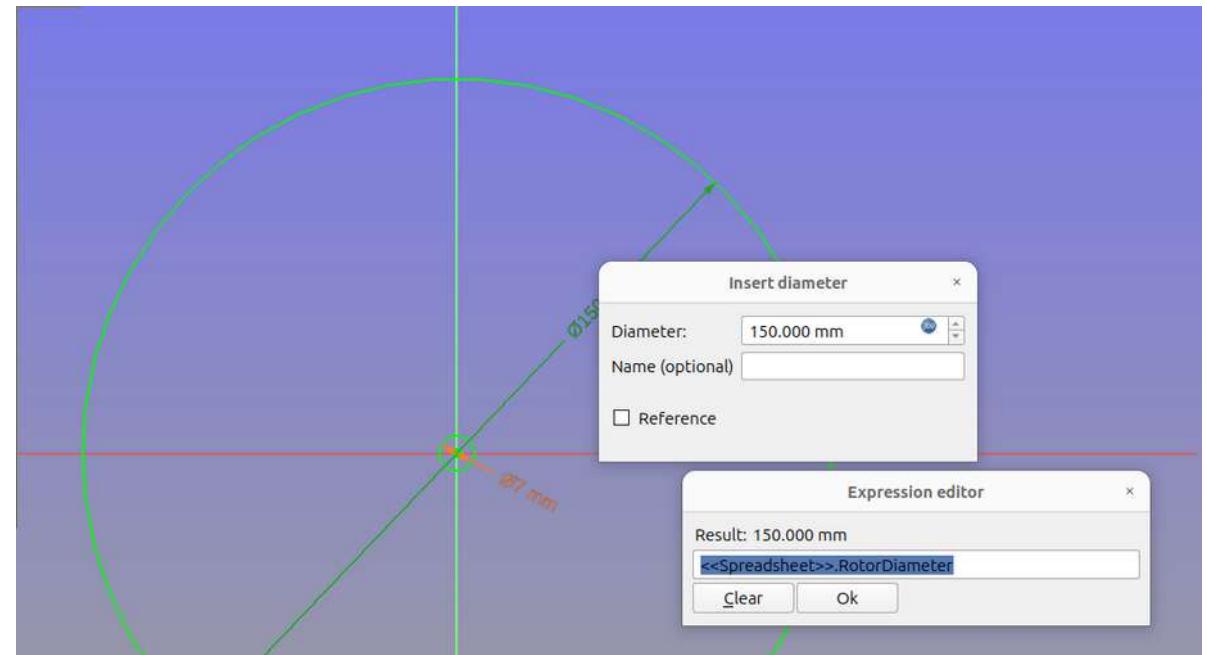
Using the symbol instead of dimensions

- Instead of padding by 5mm, we can say “pad by RotorThickness mm.
- This way, we can control the whole model centrally.
- Most places that accept numbers also accept aliases or expressions.
- You can also do math on them (+,-,*/,cos,sin,tan...)



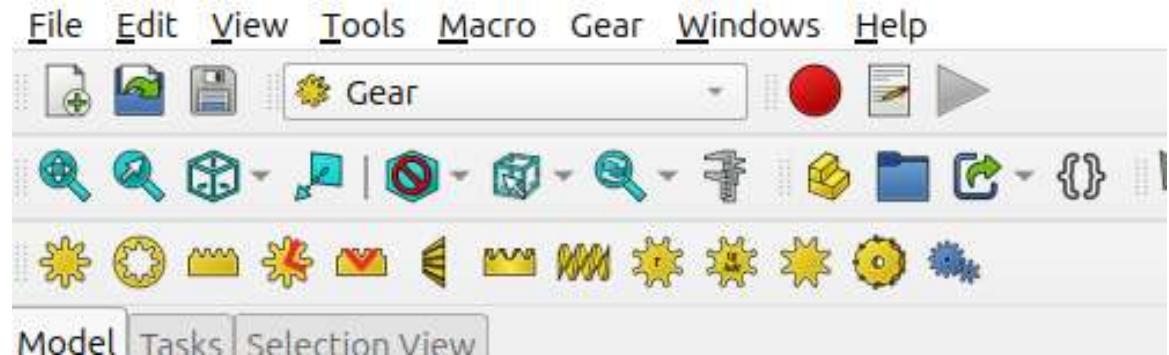
Sketches too...

- Also works for sketches to specify things like diameter, length, angle, etc.



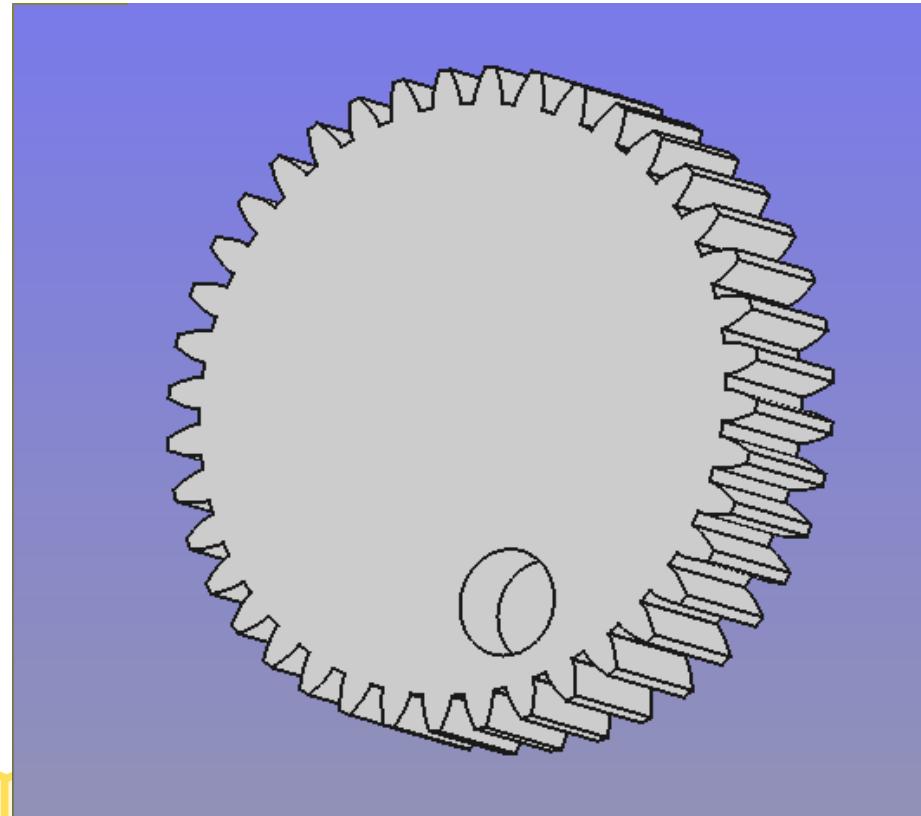
Gears are complicated

- Number of teeth is easy
- So is height
- “Module” is a technical “Gear” engineering term (close to teeth per mm but not really)



BUT, you can still cut features in them

- Create sketch and “pocket” a hole out of the gear, for example, to put a bolt or shaft through it.



Not enough time, BUT

- Fasteners add-on is SUPER useful for adding bolts, screws, etc.
- WARNING, you will need to become (somewhat) familiar with bolt ANSI, ISO, standards for bolts because they are modeled with precision to those standards.
- Super-annoying how many different types of bolts, screws, nuts, etc are out there, so the selections are overwhelming.
- Advice to pick a “favorite” fastener standard and stick to that.

Not enough time, BUT

- Laser Cut Interlocking can export shapes to SVG
- Useful for feeding a laser cutter with your parts
- Can also be used to make “boxes” with “tabs” that interlock when cut on a laser cutter
- Common technique and allows you to make your own box and box-like shaped using this technique and/or incorporate them into your design.
- I've barely scratched the surface, but nice to know it's there.



Thank you!

- I encourage all of you with interest to check out FreeCAD and give it a shot if you're interested in this style of CAD and modeling to "up your game" in designing new things.
- Your only limit is your imagination.
- Gambare! (Japanese for "Do your best")
- がんばれ