



SPARK  
STUDIO  
SALEM  
MAKER COMMUNITY

# Intro to FreeCAD

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# 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

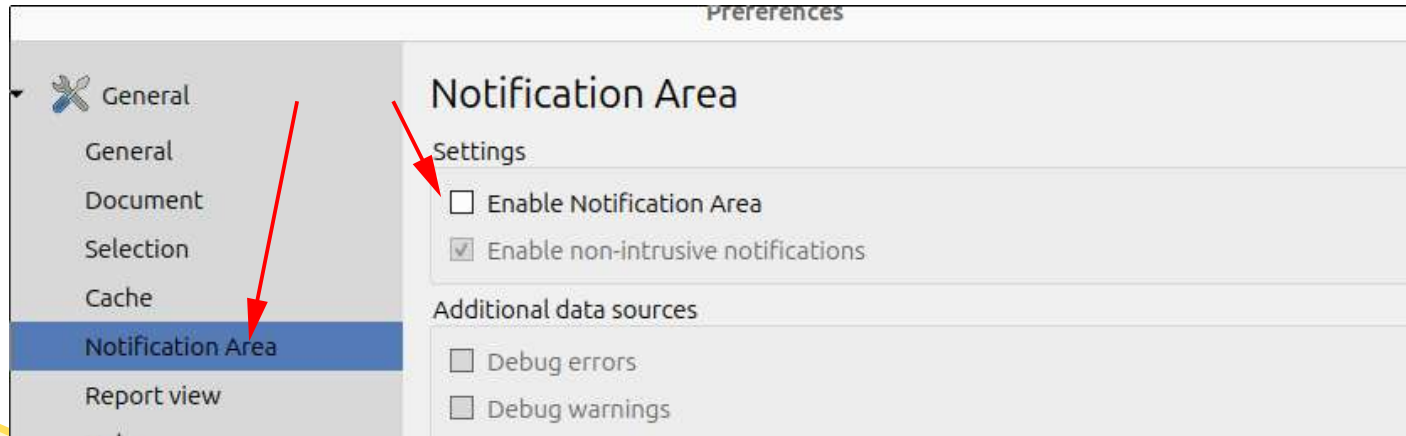
## lots of options out there

- 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



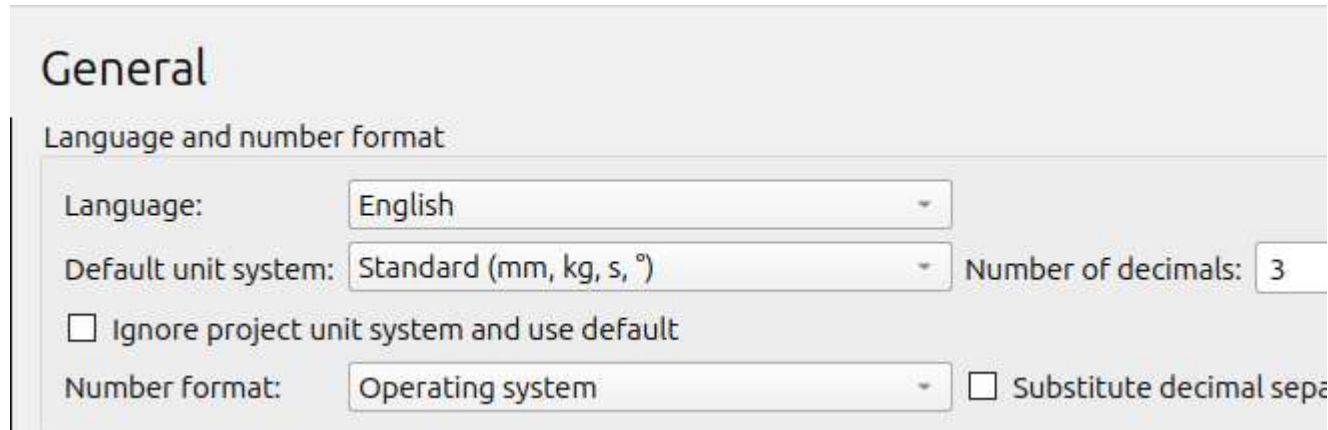
# Getting Started

- 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.



The image shows a 'General' settings window. Under the 'Language and number format' section, the 'Language' is set to 'English'. The 'Default unit system' is set to 'Standard (mm, kg, s, °)', and the 'Number of decimals' is set to '3'. There is an unchecked checkbox for 'Ignore project unit system and use default'. The 'Number format' is set to 'Operating system', and there is an unchecked checkbox for 'Substitute decimal separator'.

General

Language and number format

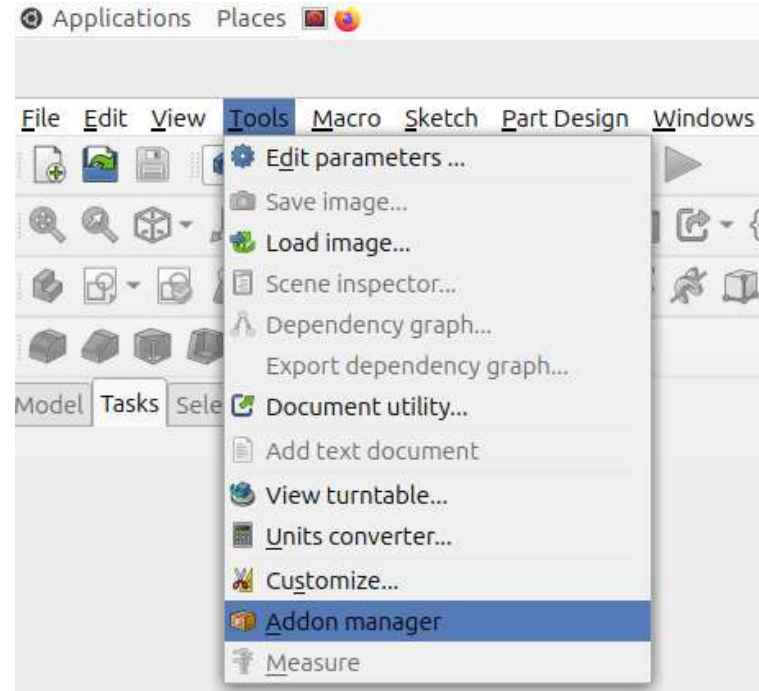
Language: English

Default unit system: Standard (mm, kg, s, °) Number of decimals: 3

☐ Ignore project unit system and use default

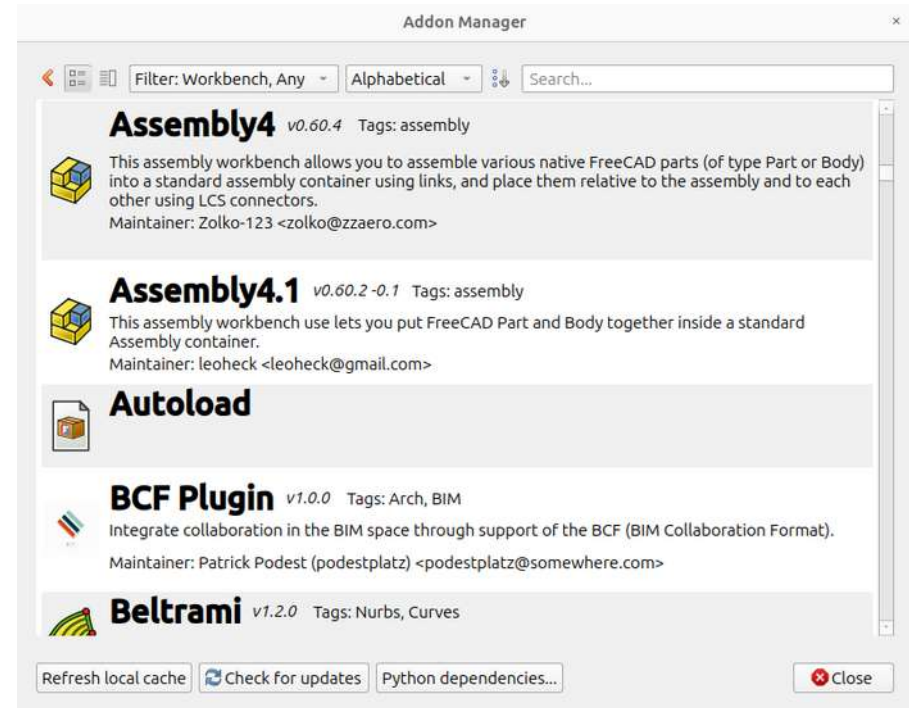
Number format: Operating system ☐ Substitute decimal separator

# Add some useful add-ons



# Install the add-ons

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



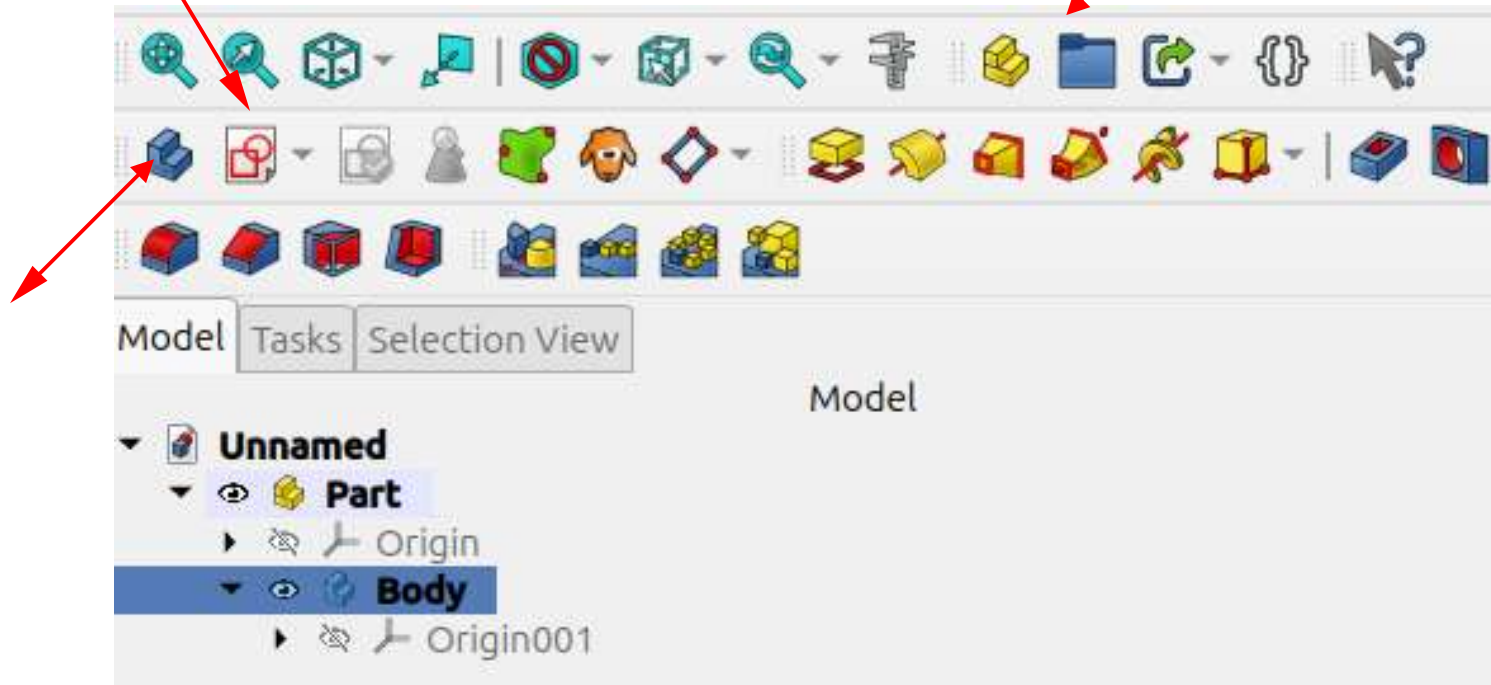


# Normal workflow: Part Design Workbench

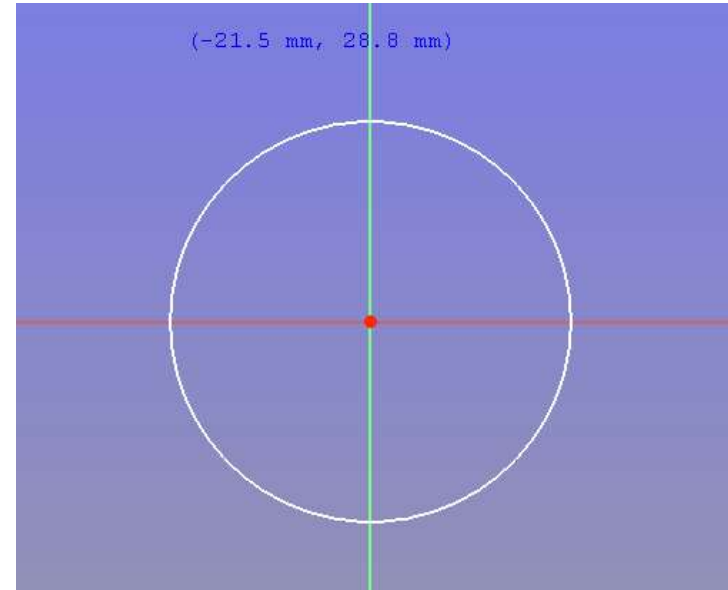
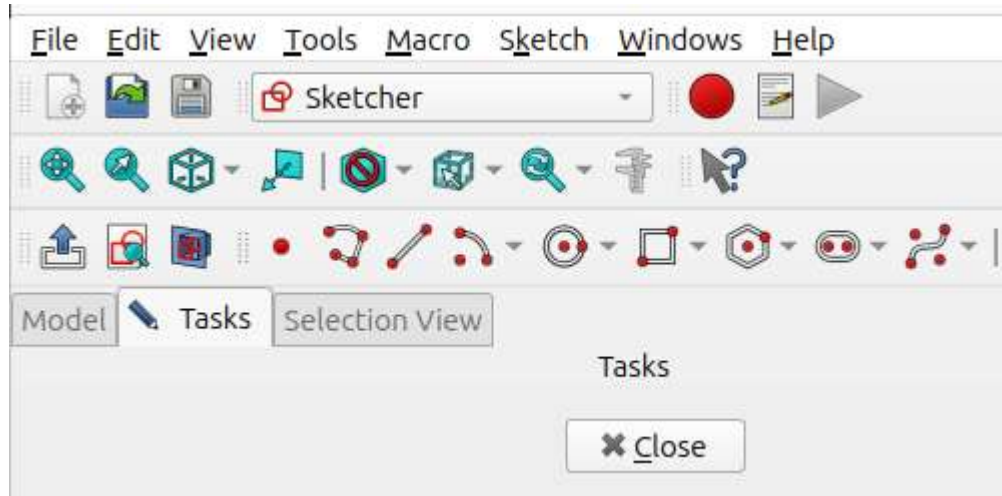
- Create a “Part”
- Create a “Body” inside the part
- Create a “Sketch”, a 2d-plane drawing
- Extrude the “Sketch” along an axis.
- Create another “Sketch” and cut shapes in it.
- Create a “Local Coordinate System” to note specific features so we can assemble it later.



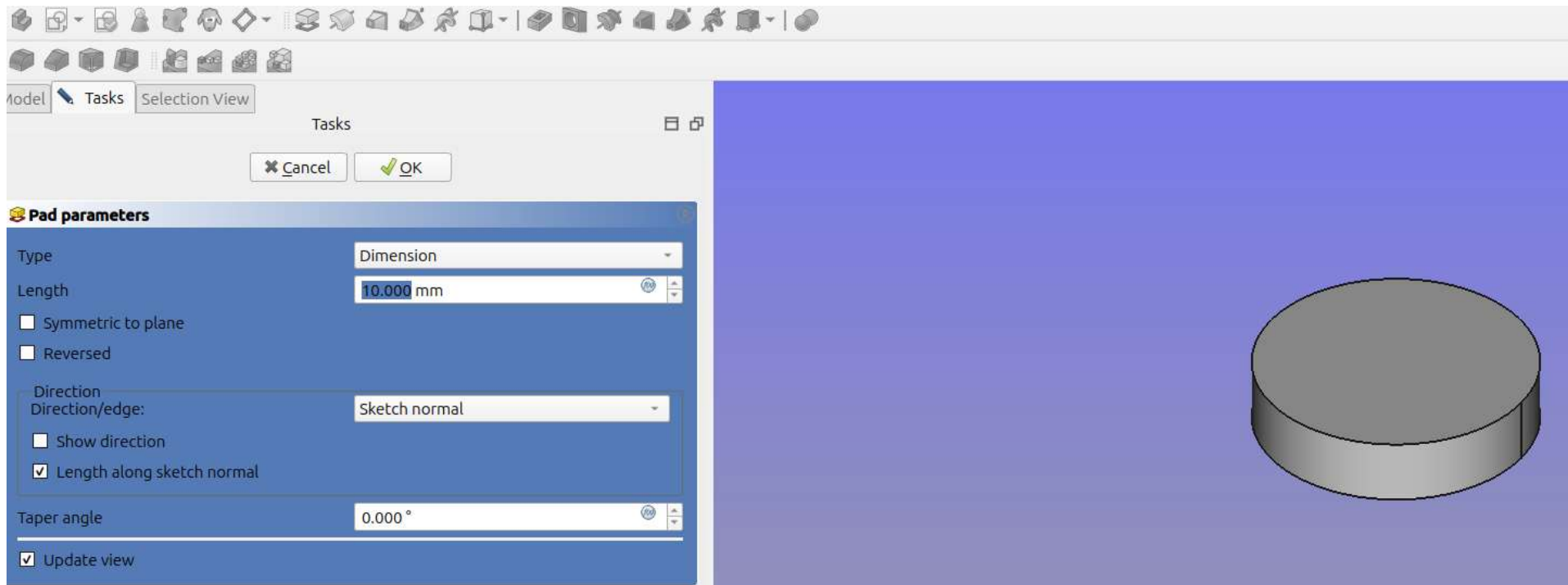
Add a “part” and to that, add a “body”



# 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



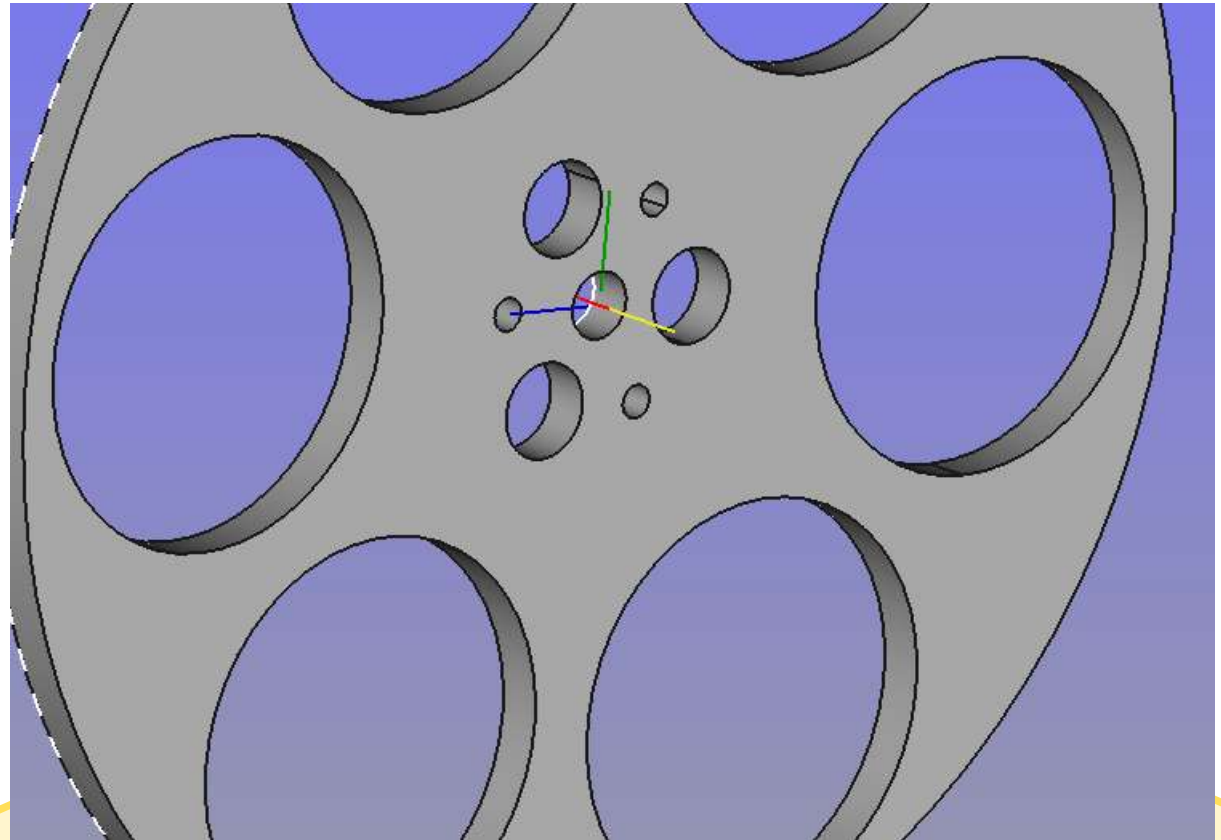
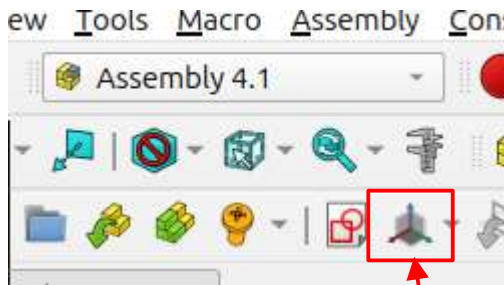
# Parts to assemblies

- 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

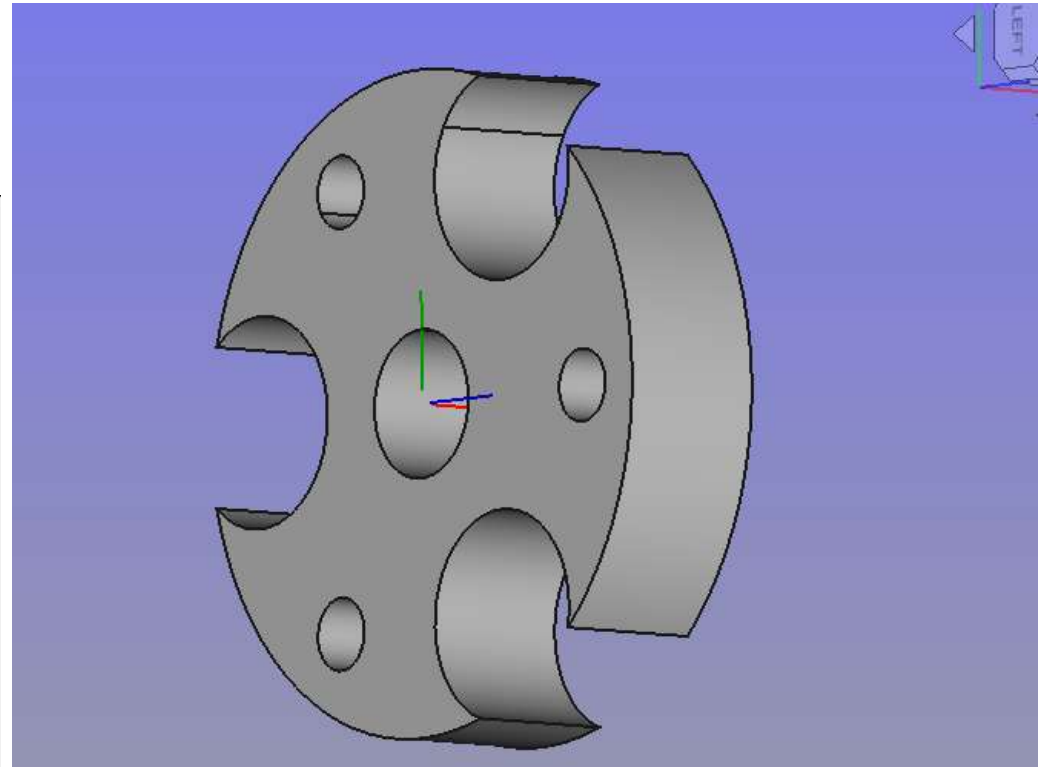
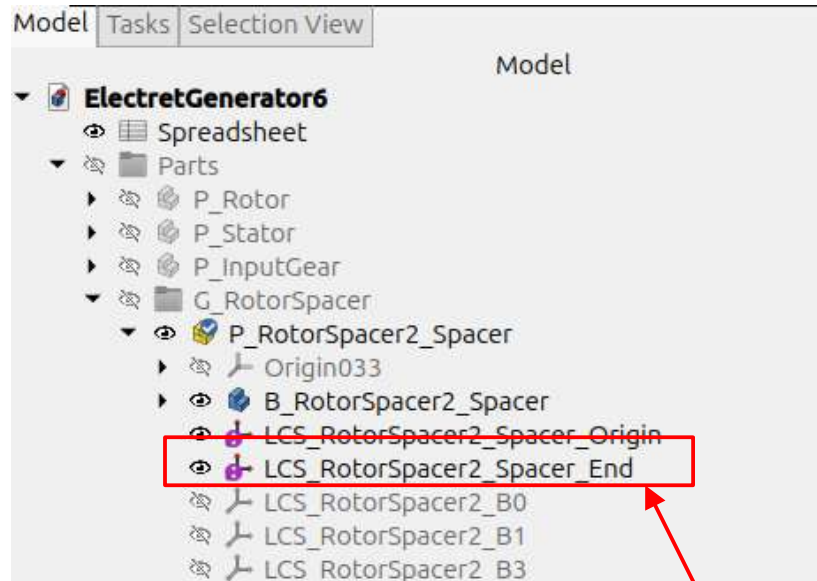
- Place a coordinate system on the rotor
- Give them meaningful names





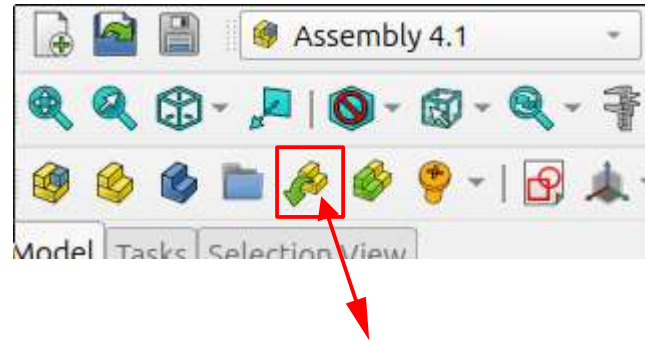
# Place coordinate system on spacer

- Another one on the spacer



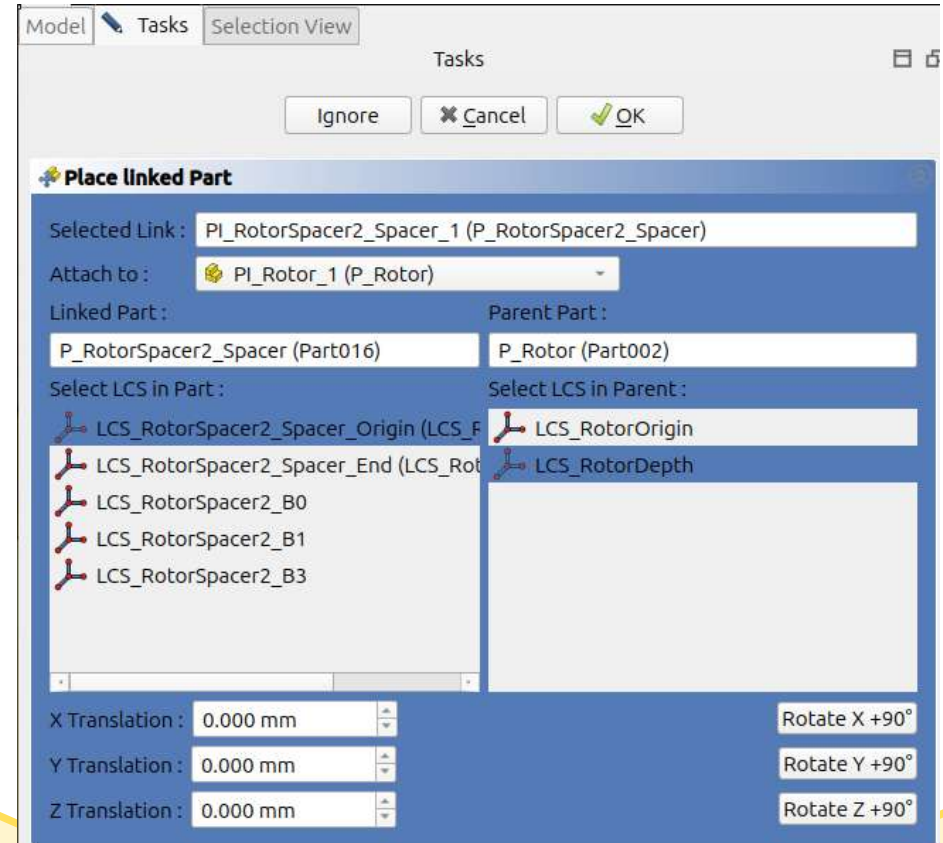
# Insert Part, place at origin

- 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.



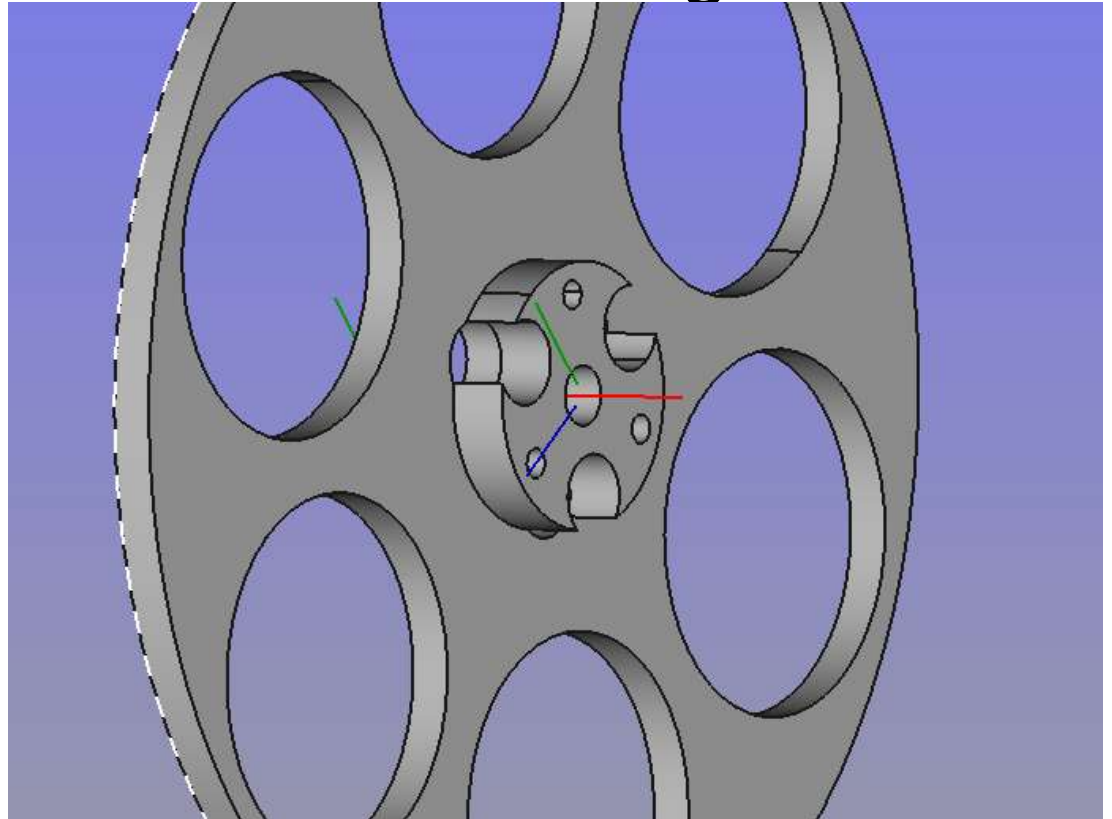
# Attaching one part to another.

- 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



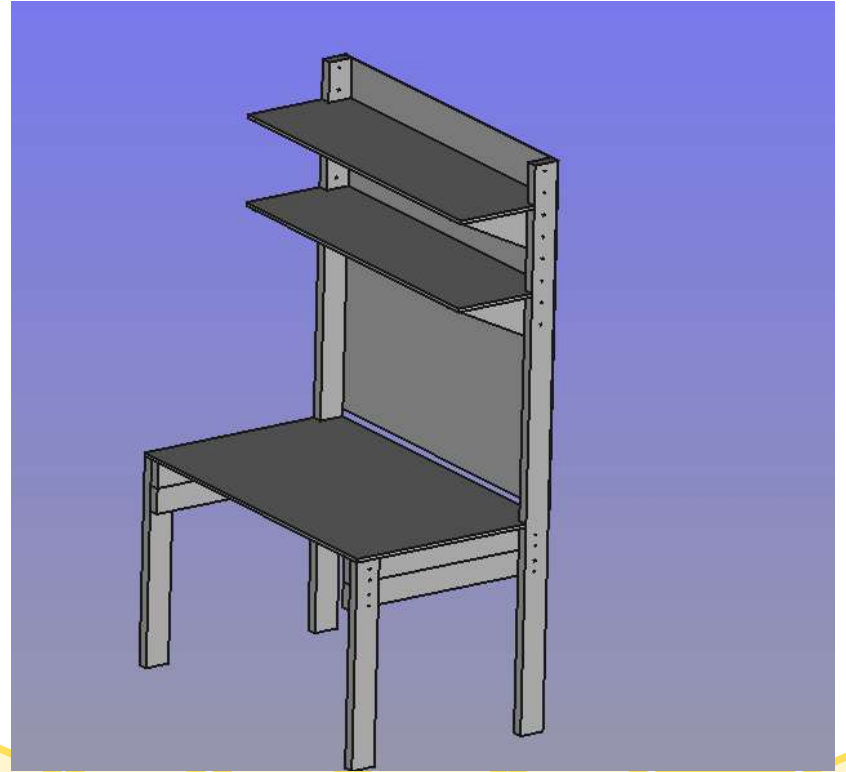
# Insert next part and “attach” them together

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

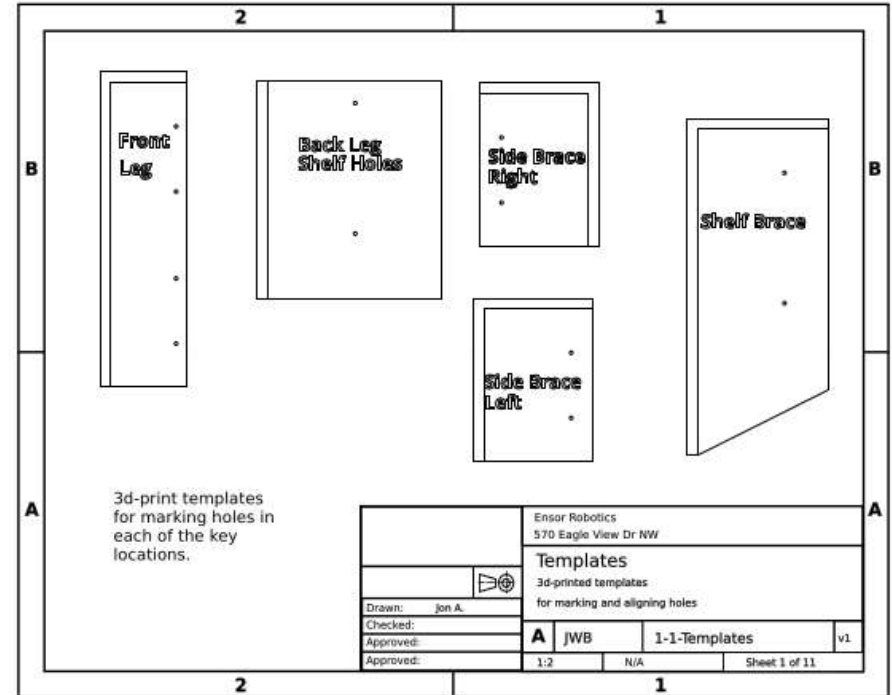
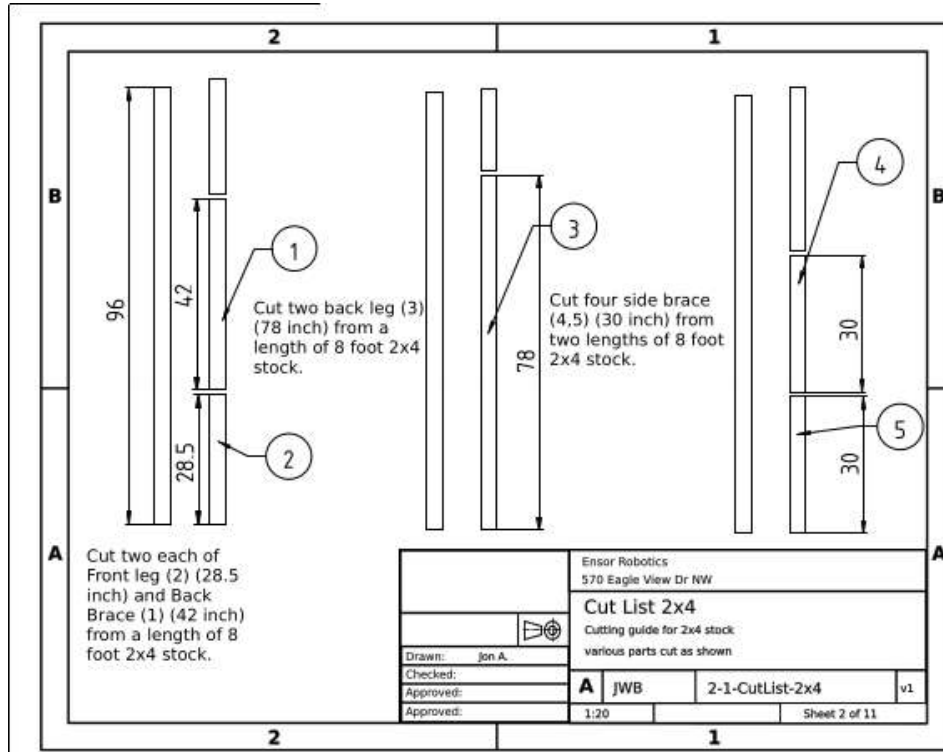


# Assemblies: Not just for 3d printing

- 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

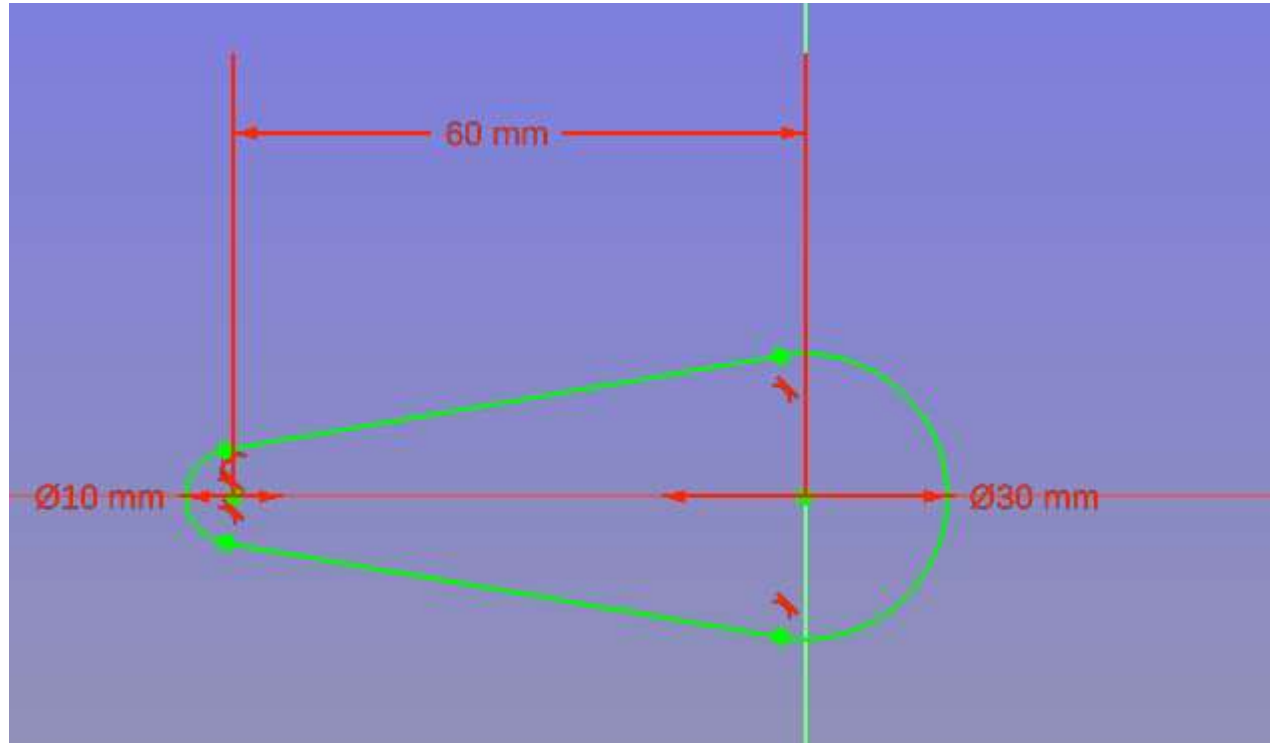
- Fully constraining sketches
- Don't just "Draw" a sketch. Constrain it.
- If two lines are supposed to be tangent, that's a constraint.
- The constraint solver lets you specify things like parallel, equal length, specific angle, horizontally aligned, as well as specific lengths and diameters.
- Good 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



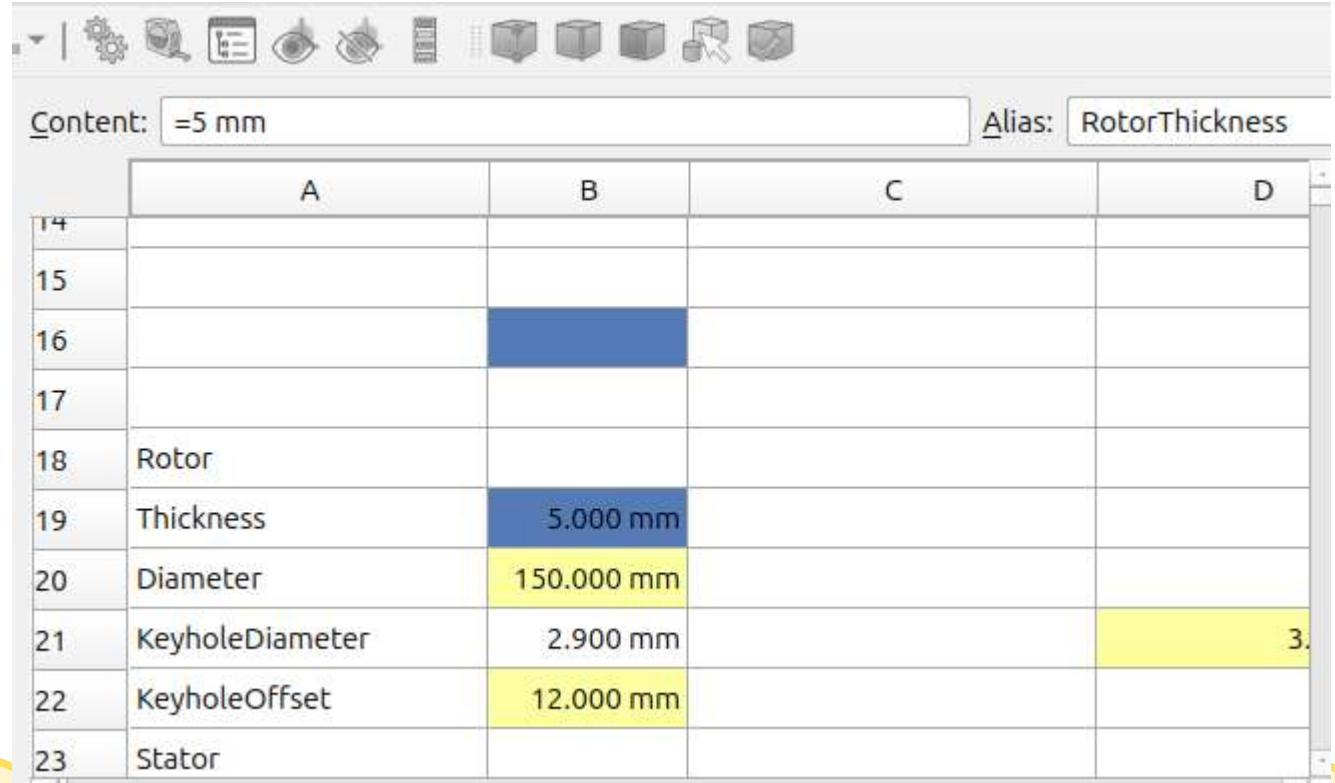
# Advanced topics

- 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”.

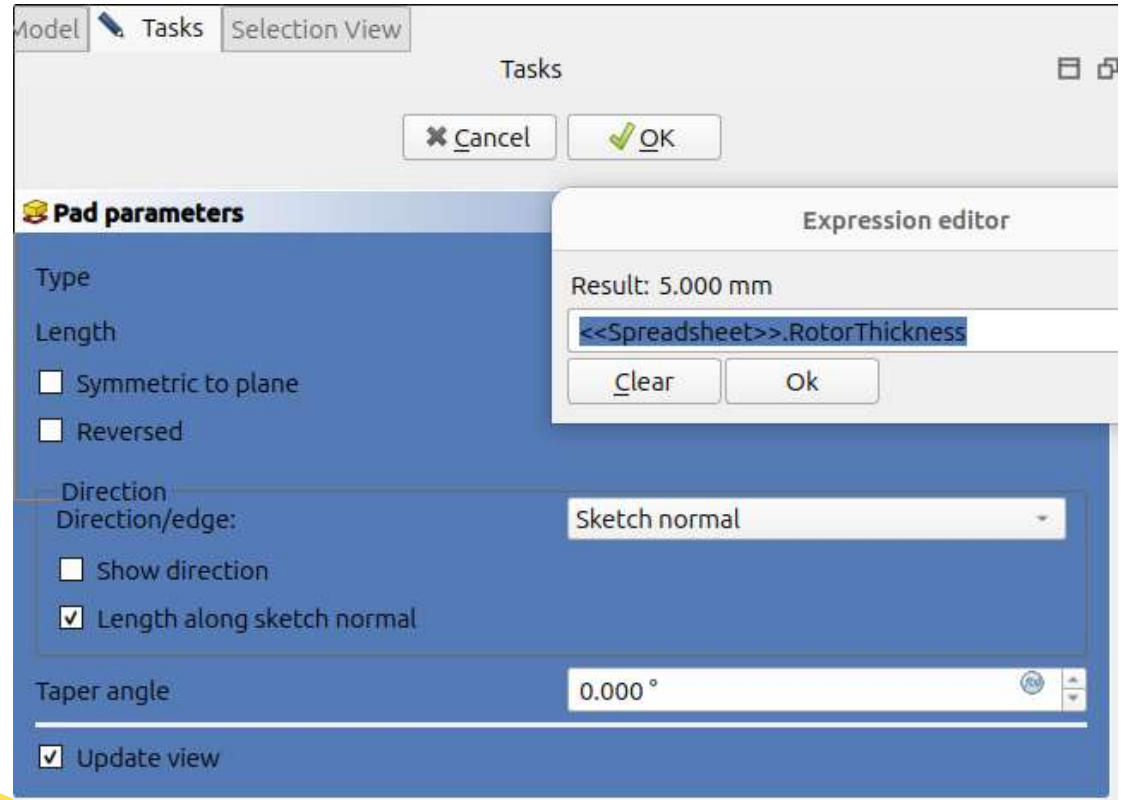


Content: =5 mm Alias: RotorThickness

	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			

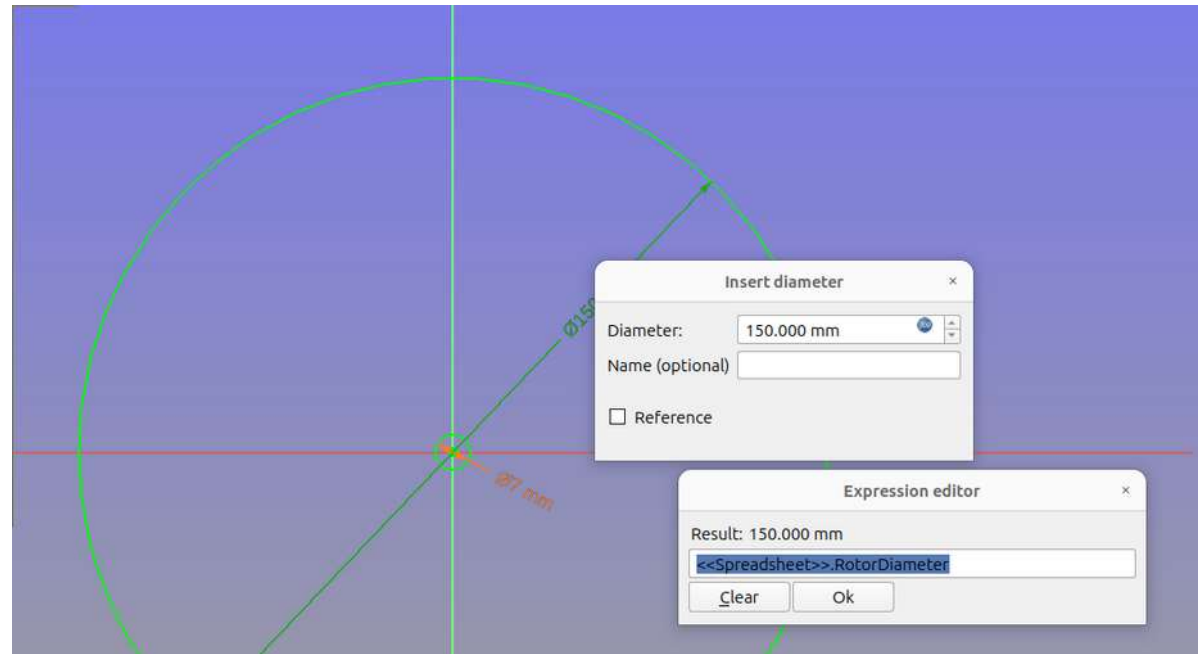
# Using the symbol instead of dimensions

- 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...)



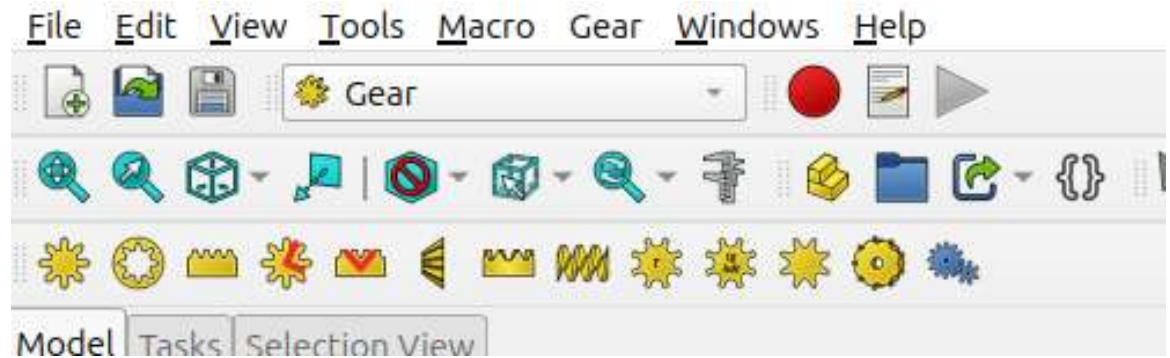
# Sketches too...

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



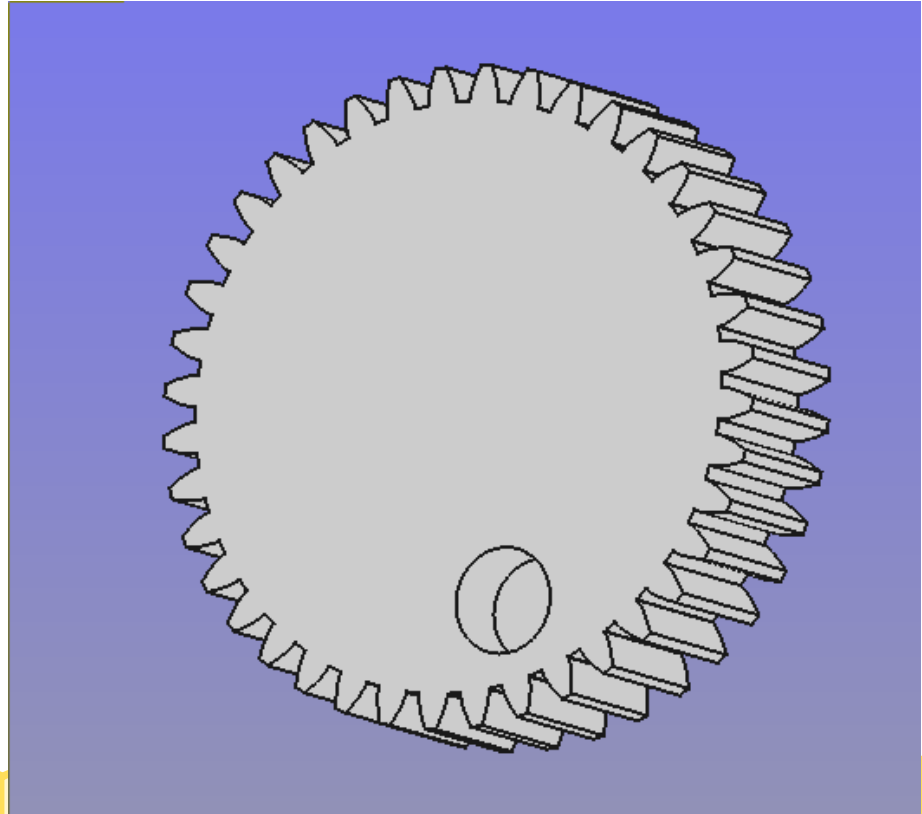
# Gears are complicated

- 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

