

Introduction to Fab Academy

By Fablab Kamakura

Instructor: Rico Kanthatham

Session 3: Introduction to 3D Design

Session Objectives:

- Review previous session **Assignment & Note Taking** progress
- Introduction to **Parametric 3D CAD design**
- Introduction to **Fusion 360 & FreeCAD 1.92**
- **Final Project Ideation**...progress review

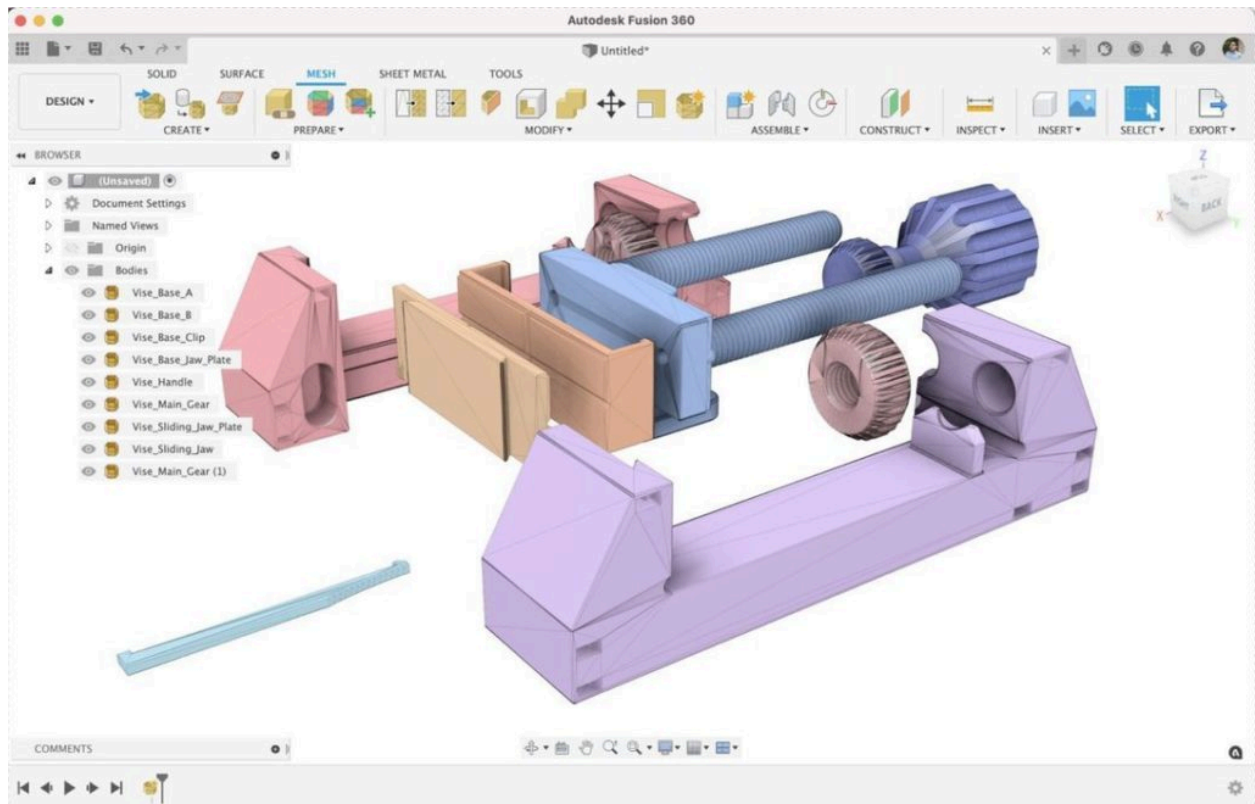
Preparations for Next Session: Microcontroller & Electronics

- Install **Arduino IDE**
- Prepare **Arduino Kit**

Session Learning: 3D Design

3D design for Fab Academy

Digital fabrication often uses precise digital models.

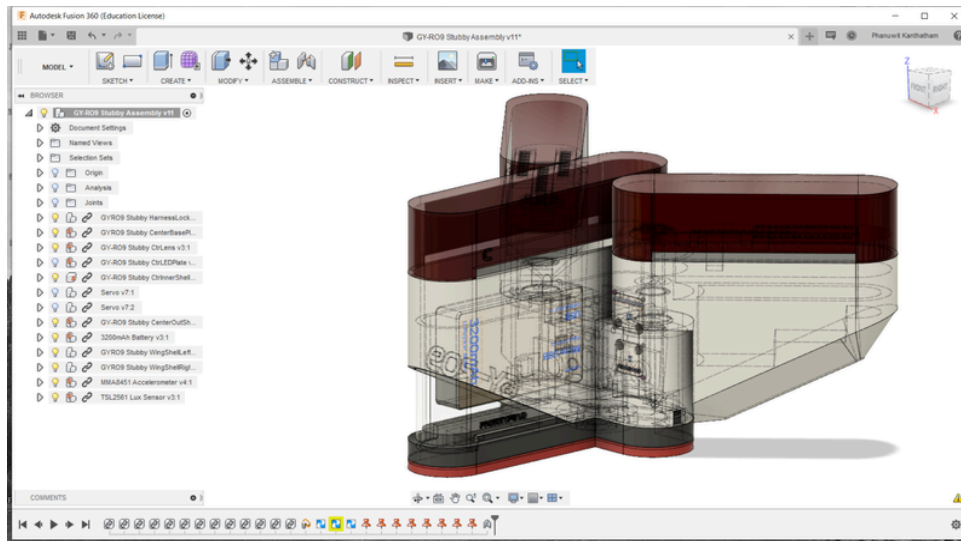


3D modeling is the process of developing a mathematical coordinate-based representation of any [surface](#) of an object (inanimate or living) in [three dimensions](#) via [specialized software](#) by manipulating edges, vertices, and polygons in a simulated 3D space. ^{[1][2][3]}

3D modeling for Fab Academy

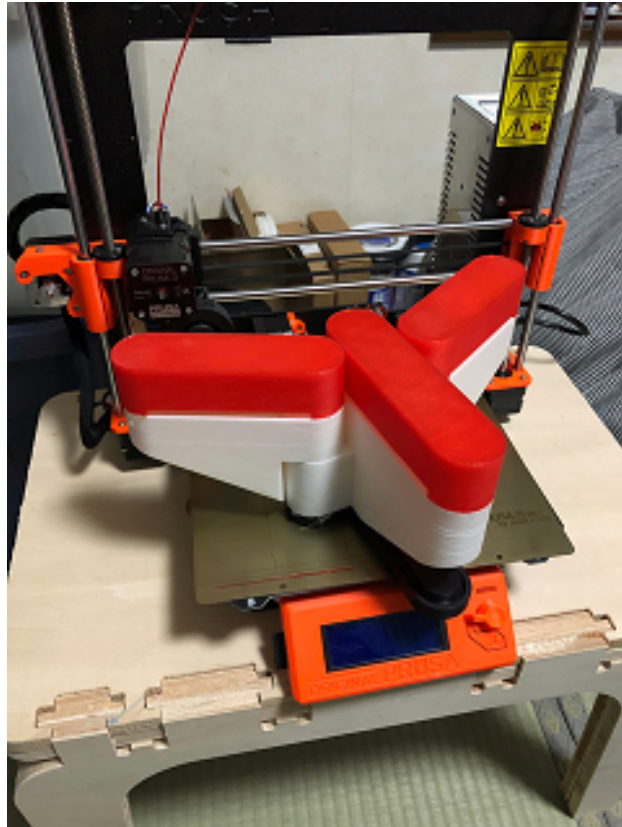
Virtual Prototyping

- Testing shapes, sizes, assembly relationships...What form? How big or small? How will they connect or join together?
- Answer many questions in virtual space before making the object physical



3D Models for Digital Fabrication

- Used to generate tool paths for CNC machine (3D printers, CNC Milling machines, etc.) >> export .STL file



Computer Aided Design (CAD) Software

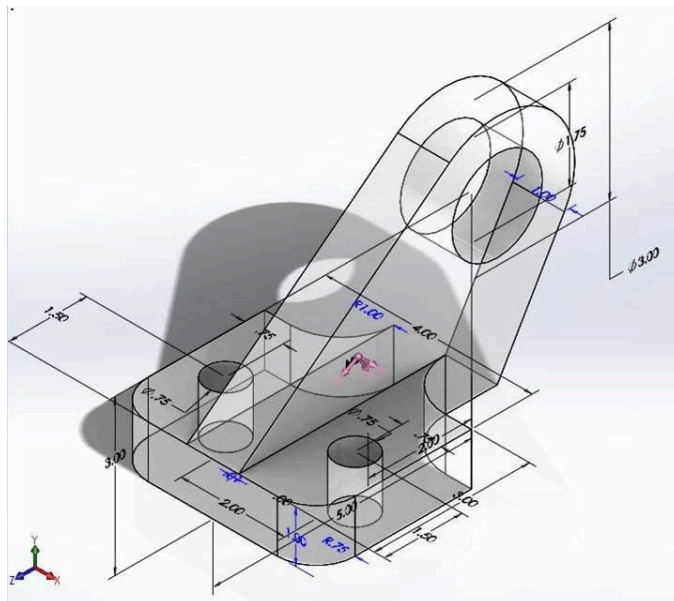
3D models are made in Computer Aided Design (CAD) software. For Fab Academy, a CAD software capable of “Parametric Modeling” is recommended.

Parametric Modeling

“Parametric modeling (or parametric design) is the creation of a digital model based on a series of pre-programmed rules or algorithms known as ‘parameters’ . That is, the model, or elements of it are generated automatically by internal logic arguments rather than by being manually manipulated” Designing Buildings, Wiki

Parametric modeling

- Usually requires a clear initial understanding of the object, its specific dimensions and functions
- Establishes size and/or position relationships between the shapes making up the 3D model...usually in a spreadsheet
- Allows for more efficient, “automatic” updates to a model...without having to require the rebuilding of the model from scratch
- Records an action history of the building process of a model...allowing adjustments to a previously executed step that will reverberate through subsequent actions.



Parametric CAD Modeling Workflow

Very similar in any Parametric CAD software...only the names and location of tools differ in the software' s interface.

1. Create a **SPREADSHEET** of parametric dimensions
2. **2D SKETCH** on a drawing plane
3. **DIMENSION & CONSTRAIN** the drawn Sketch lines
4. **EXTRUDE** the 2D Sketch into a 3D object (often called a Body)
5. **MANIPULATE, COMBINE, TRANSFORM** bodies...into the final object
6. The final object can then be **RENDERED & ANIMATED...**to aid visualization

Fusion 360 vs FreeCAD



They both dance. One is **elegant and graceful** but **expensive**...the other is **awkward** but **open-source and free**. Both are **parametric modeling** programs capable of **high precision sketching** and **3D extrusions**, **rendering**, **limited animation** and **limited simulation studies**.

Fusion 360 has a **better, more intuitive user-interface** and **smoother workflow**...but costs money

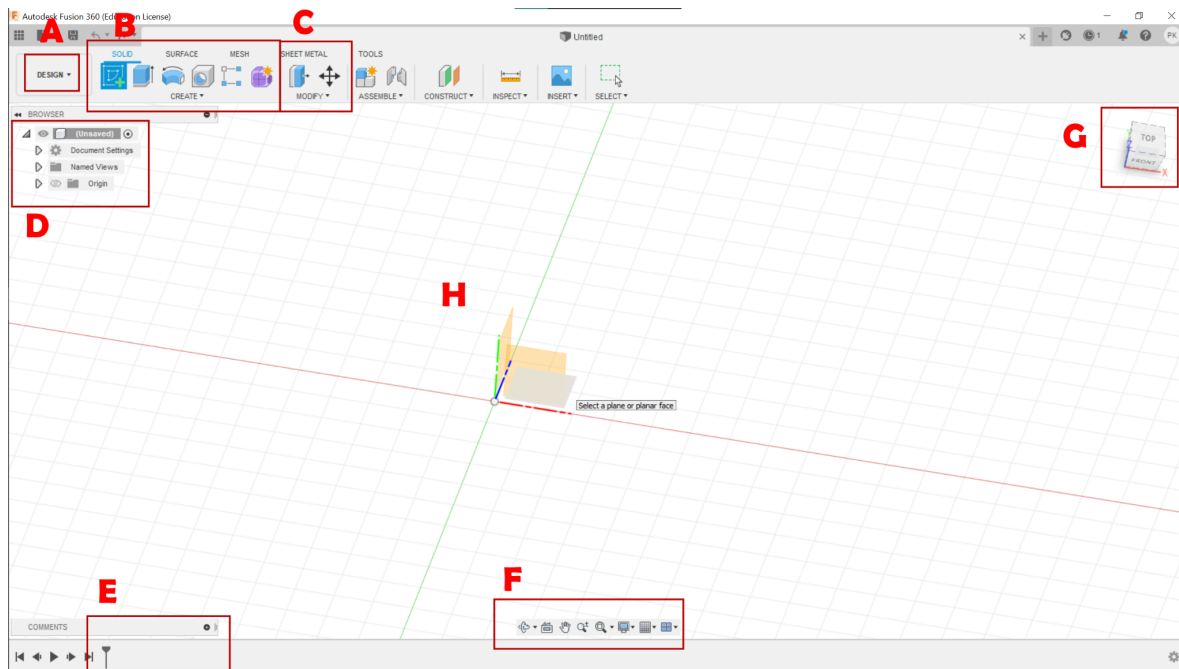
- Free version is available but limited in functionality...Full version **expensive** for individual users
- **NOT open source**...so new features and updates depend on the software developer
- **Limited customizability**
- Some 'Best Practices' need to be learned to keep complex models from breaking
- Beautiful built in **rendering engine**
- **Generative design** and **simulations** also possible

FreeCAD is...free and full-featured...but has an **awkward, cobbled-together interface**.

- **Open source**...new features and functionalities are added in frequent updates and Add-ons.
- **VERY Full Featured**...both built-in and Add-on functionalities
- **Infinitely customizable**...the interface, add-on features...can be adjusted to the user's exact preference
- **Great documentation** in many languages (<https://www.freecadweb.org/?lang=id>)
- **Programmable**...OpenSCAD and Python integration
- ...**NO built-in rendering!!**

Fusion360 by Autodesk

Interface Overview



A = Workspace selection (Design, Render, Animation, Simulation, Manufacture, Drawing)

B = Modeling tools

C = Model transformations

D = Model elements

E = Model history

F = View options

G = View selection

H = Drawing area

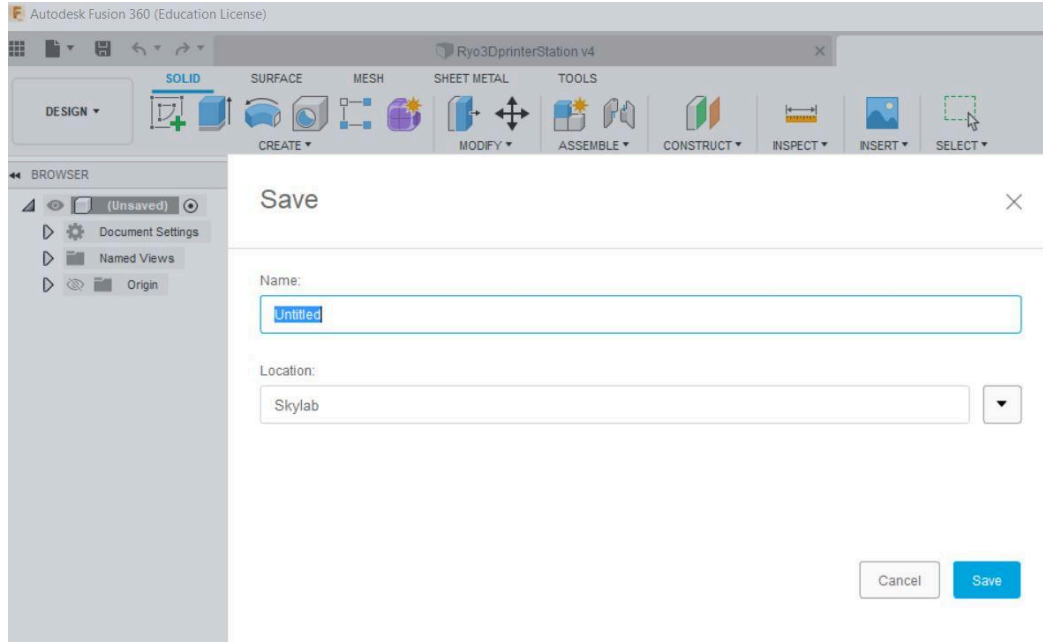
Basic Navigation

- **PAN** (Left/Right, UP/Down) >> MMB hold and drag
- **ZOOM** >> MMB scroll wheel Up/Down
- **ORBIT** >> SHIFT + MMB hold and drag

Fusion360 Procedure:

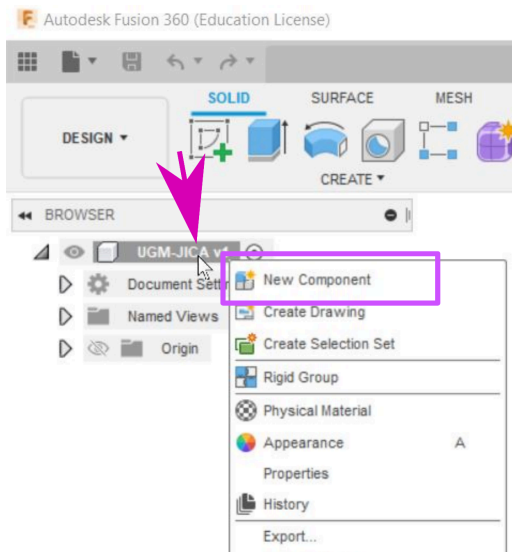
1. Save [File Name]...Rule #0

- File >> Save...
- Triggers Auto-Save



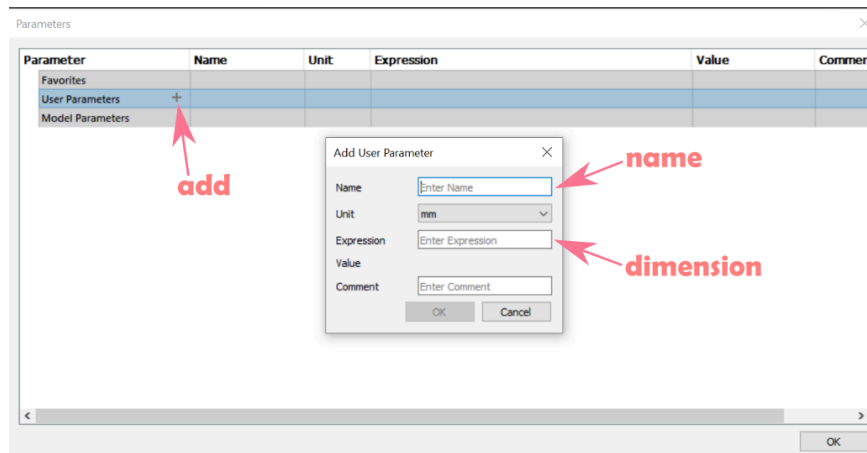
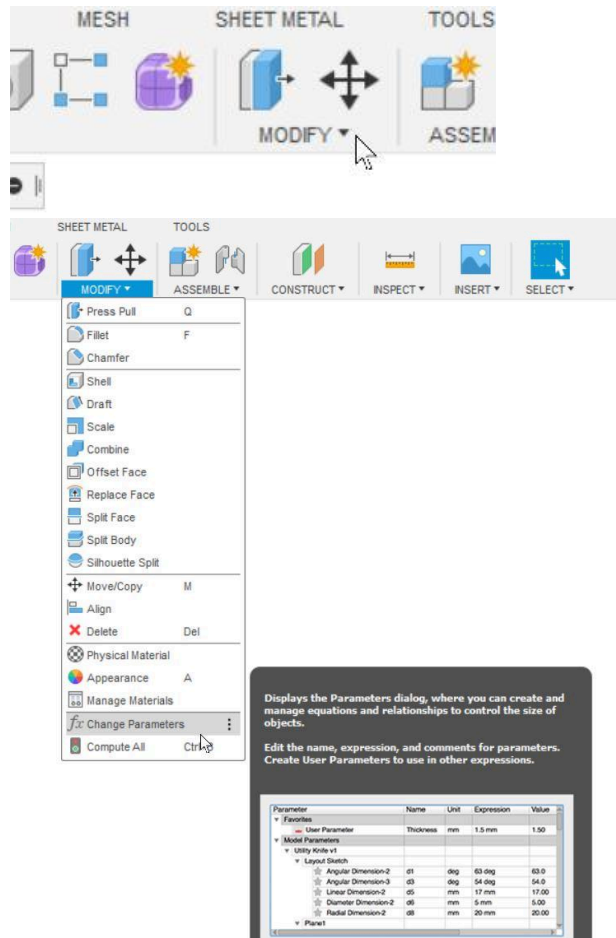
2. Create a Component (to build inside)...Rule #1

- Create component
- Activate component
- Do all modeling in that component

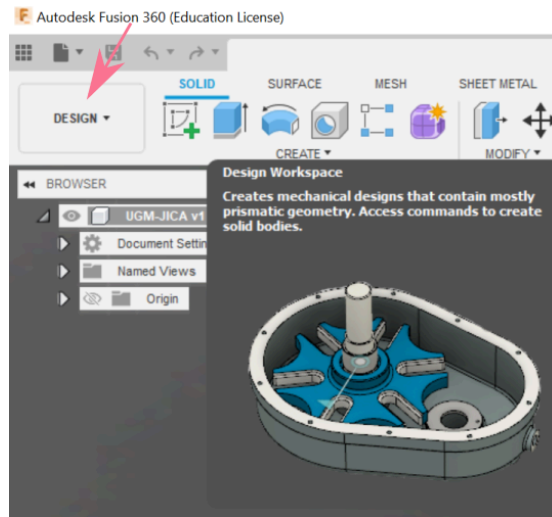


3. Define Parameters (make a spreadsheet of dimensions)

- Use simple, easy to remember, sensible name for parameters
- Use a very short name for the spreadsheet (ex: a)

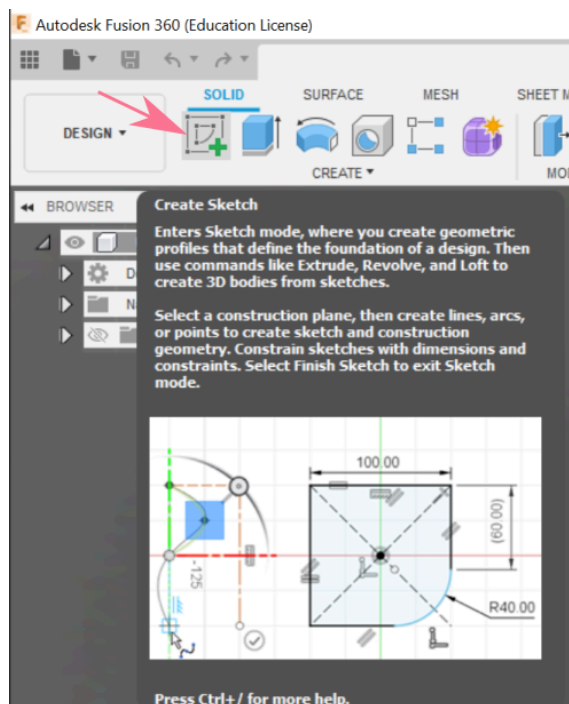


4. Work in the Design Workspace



5. Sketch

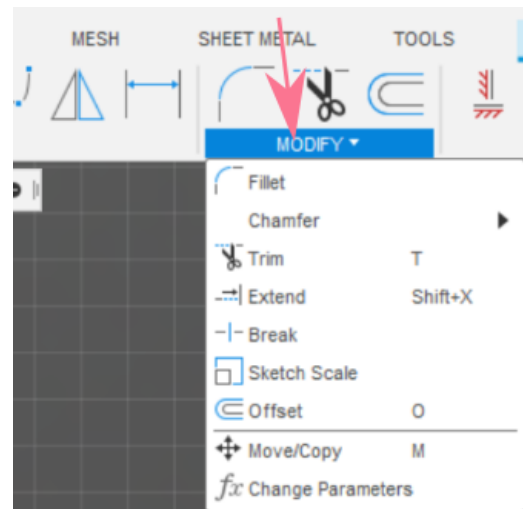
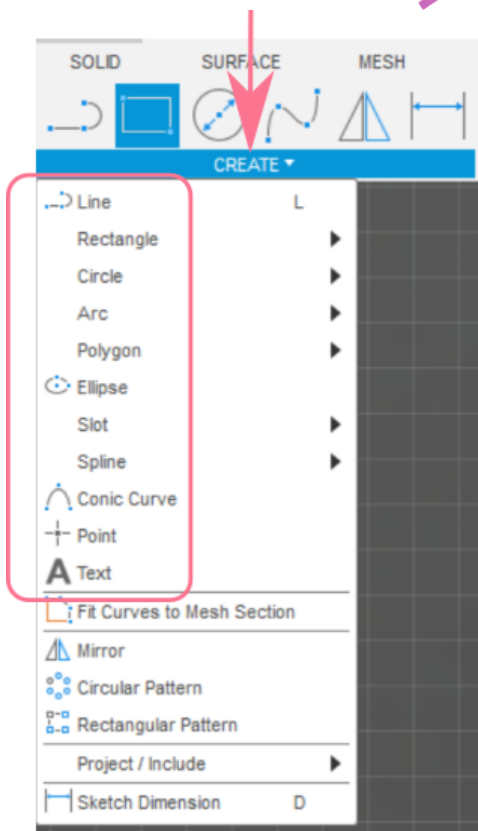
- Select **Plane** (or Faces)
- **Draw** (rectangles, circles, lines, construction lines)
 - i. Use **parameter** dimensions for shape size, spacing and location (TAB to switch)
 - ii. Use **constraints**
 - iii. Use grid and surface snap
 - iv. Click **“Finish Sketch”** ...when done

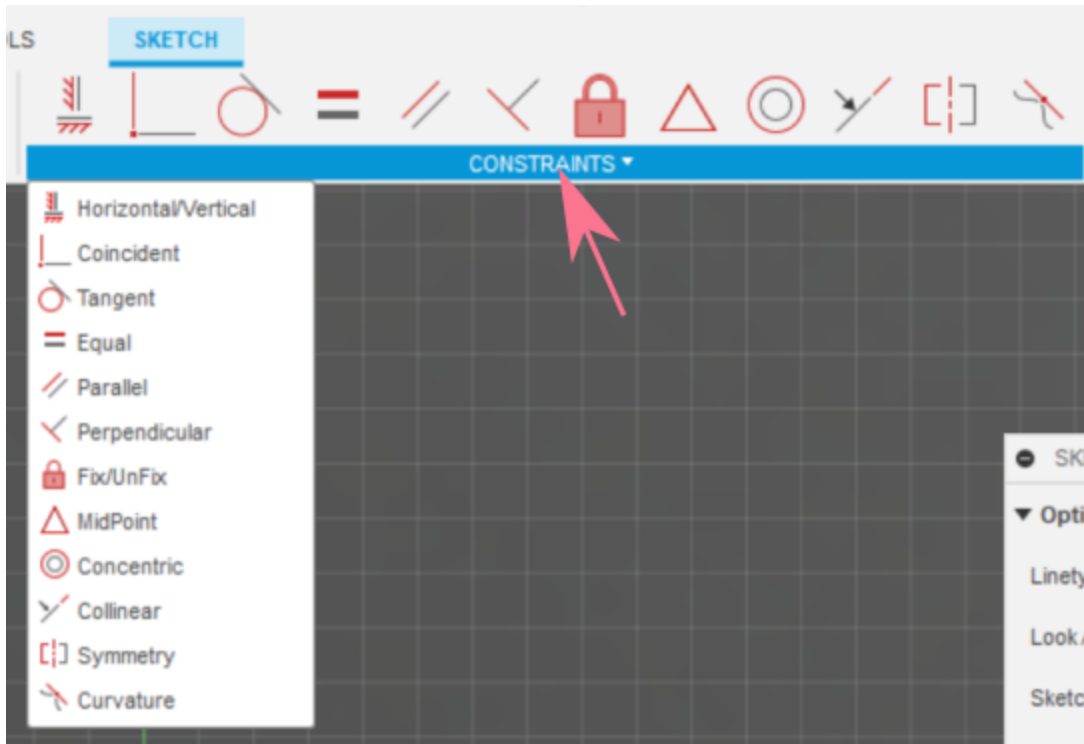


Offset
Trim
Fillet



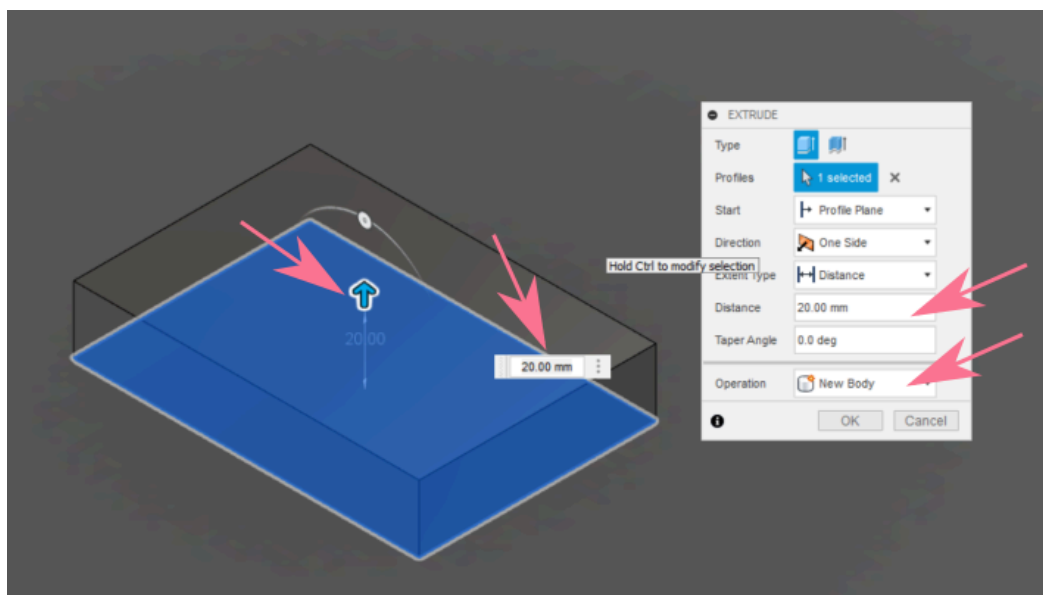
Dimension
Mirror
Spline
Circle
Rectangle
Line



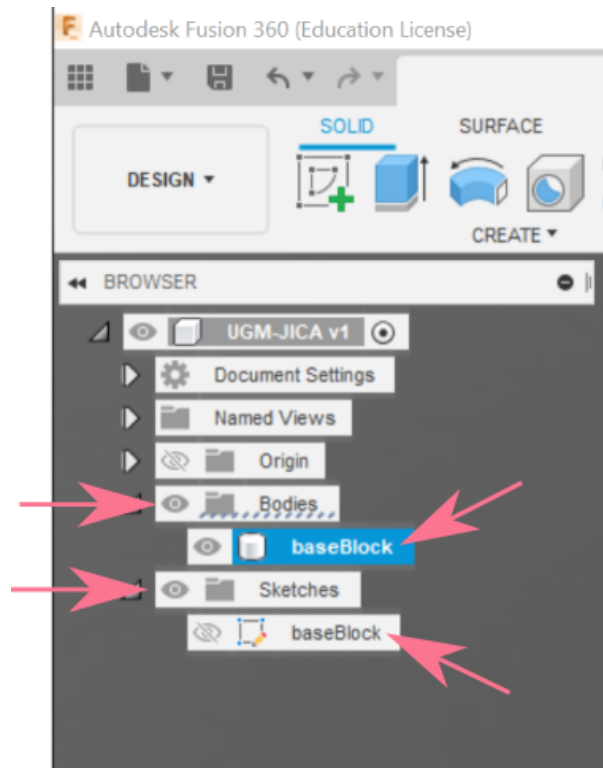


6. Extrude

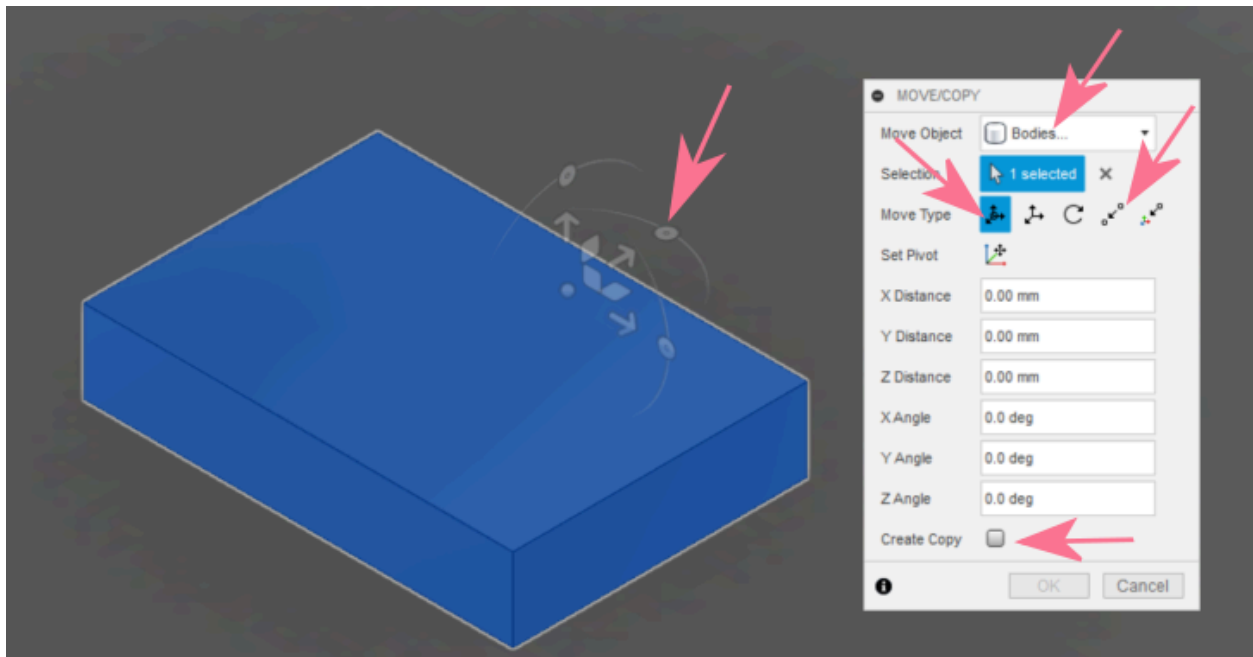
- Select shape to extrude >> E
- Define extrude Direction and Distance
- Define if...New Body, Join, Cut, Intersect, or New Component



7. Change Body and Sketch names to something sensible and easy to remember...**Rule #2**

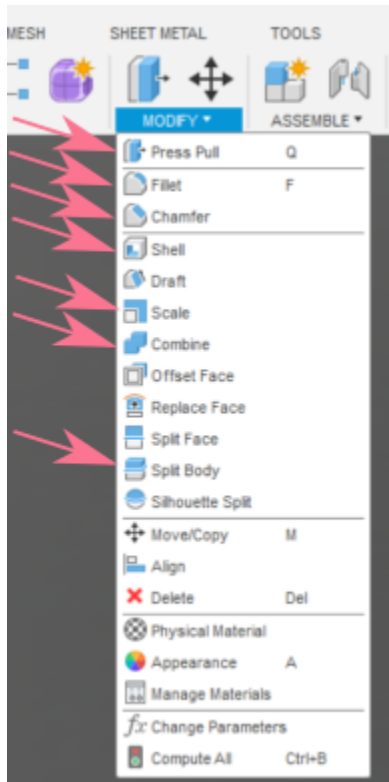


8. Move/Copy



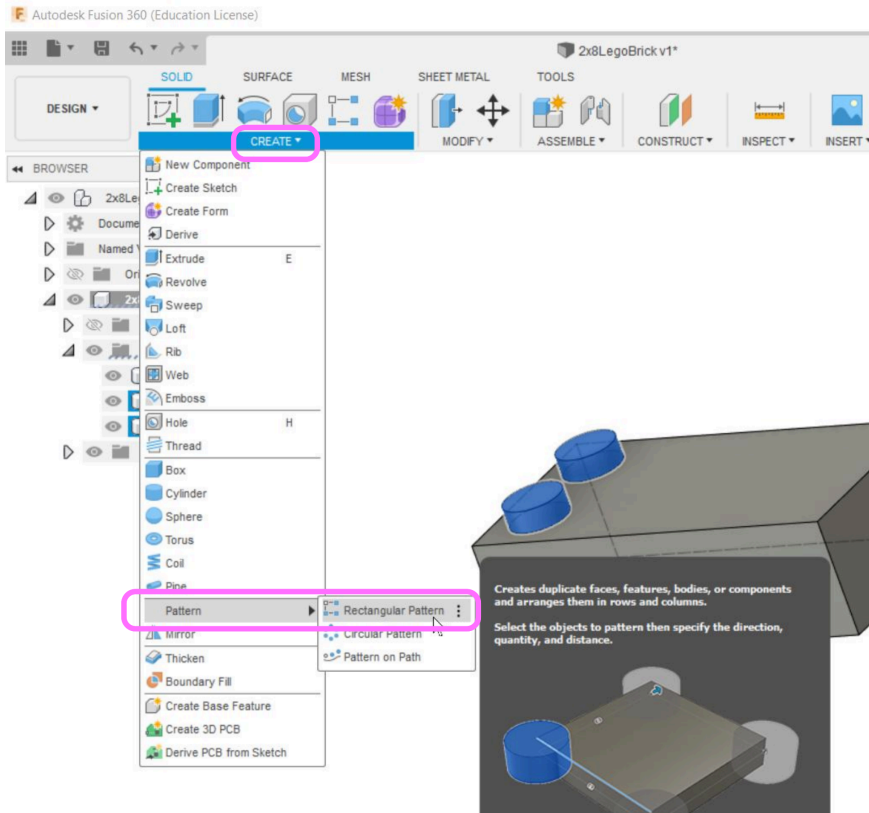
9. Modify Menu

- Press/Pull
- Fillet & Chamfer
- Shell
- Scale
- Combine
- Split



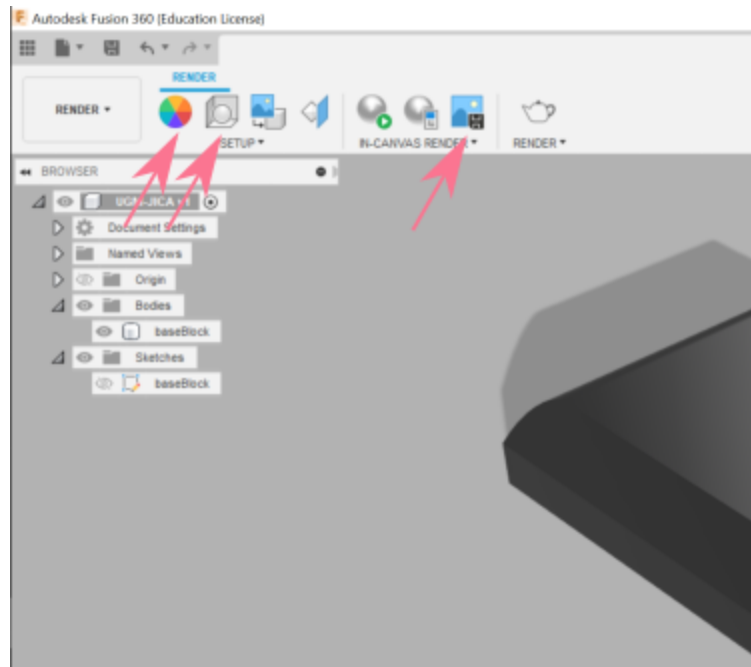
10. Pattern (Array)

- Make repeating copies of an object

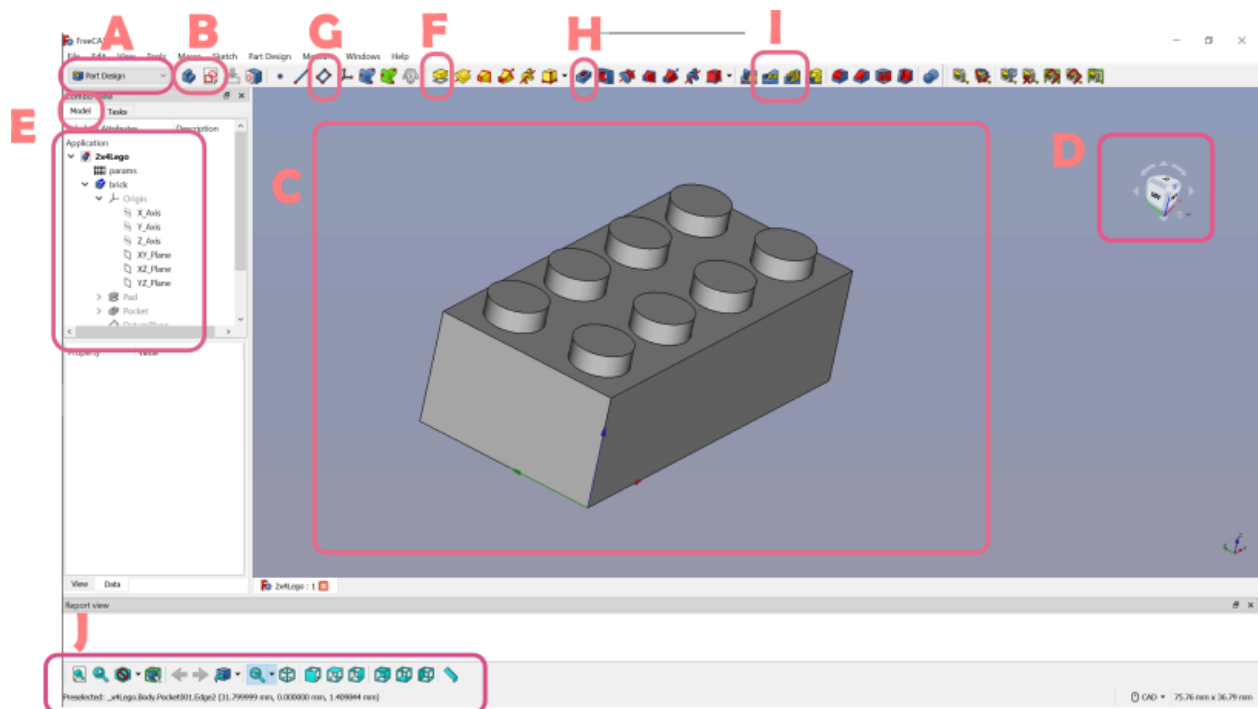


11. Render Workspace

- **Appearance** – Assign materials to objects
- **Scene Settings** – Specify settings for the rendering environment
- **Capture Image** (Lower Quality, Fast) – to save display image as PNG, JPEG or TIFF
- **Render** (Higher Quality, Slow) – to save render as PNG, JPEG, or TIFF (Local Render, Standard (50 times))



FreeCAD



Interface Overview

- A = workbench selection (recommend to start with 'Part Design')
- B = model/sketch
- C = drawing area
- D = view selection cube
- E = model history
- F = pad (extrude)
- G = new datum plane
- H = shell
- I = array
- J = view selection

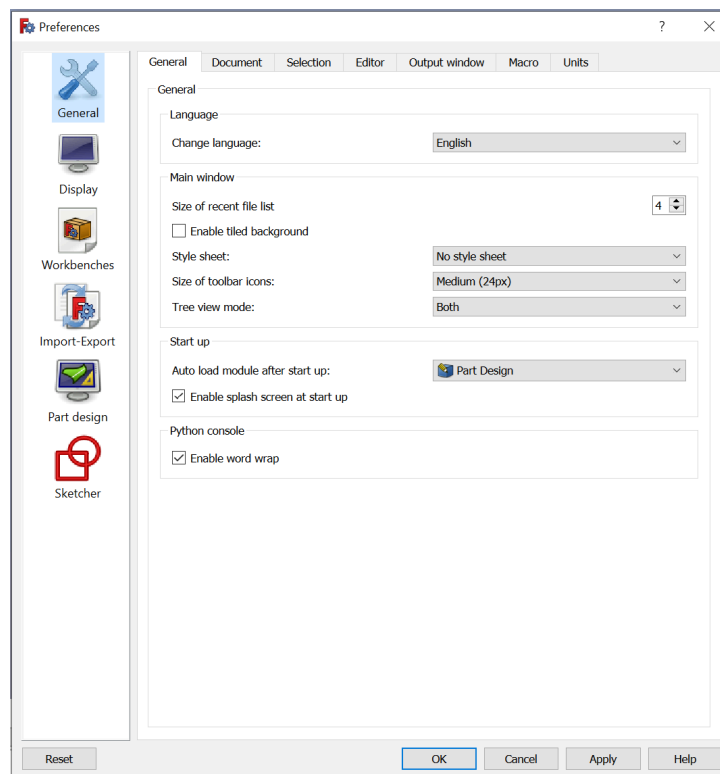
Basic Navigation

- Orbit = SHIFT + RMB
- Zoom = MMB scroll
- PAN (any direction) = CTRL + RMB

FreeCAD Procedure:

1. Edit >> Preferences

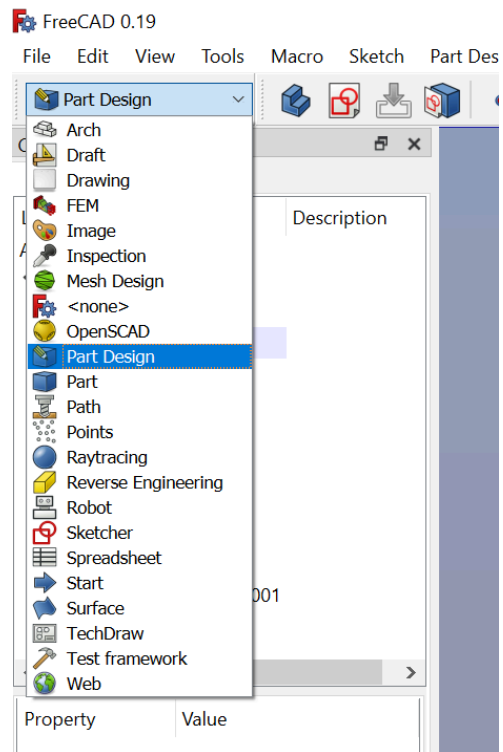
- General >> Default startup workspace >> Parts Design
- General >> Selection >> toggle Auto expand tree... & Preselect object in 3D...
- Display >> 3D View >> toggle Show Axis Cross
- Display >> Colors >>



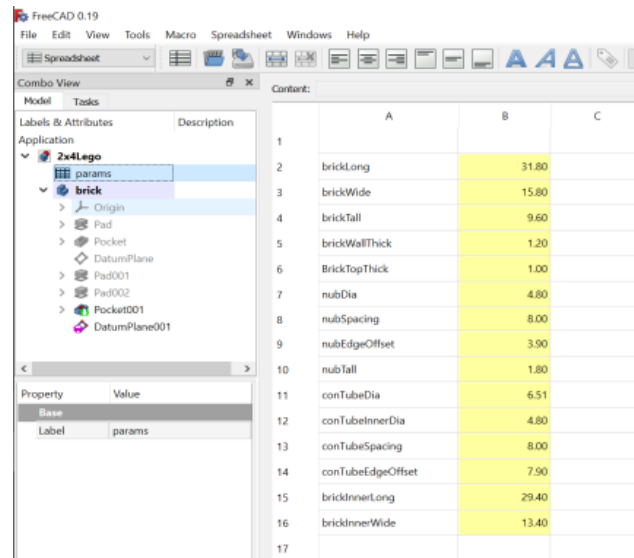
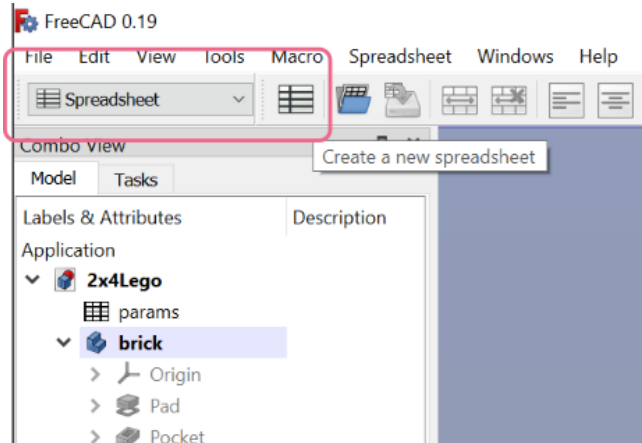
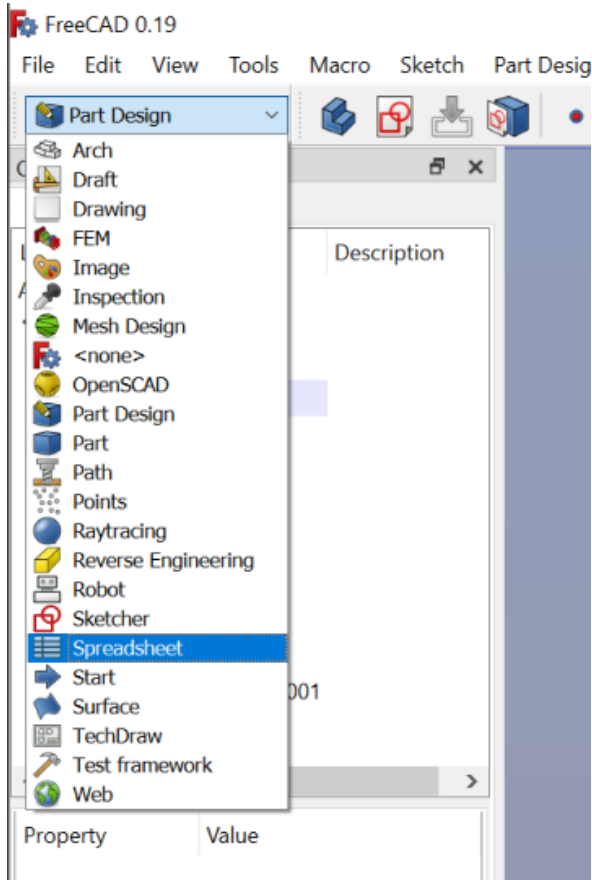
2. Organize toolbars

- Drag and drop toolbars to where you find most comfortable

3. Start in Part Design workspace



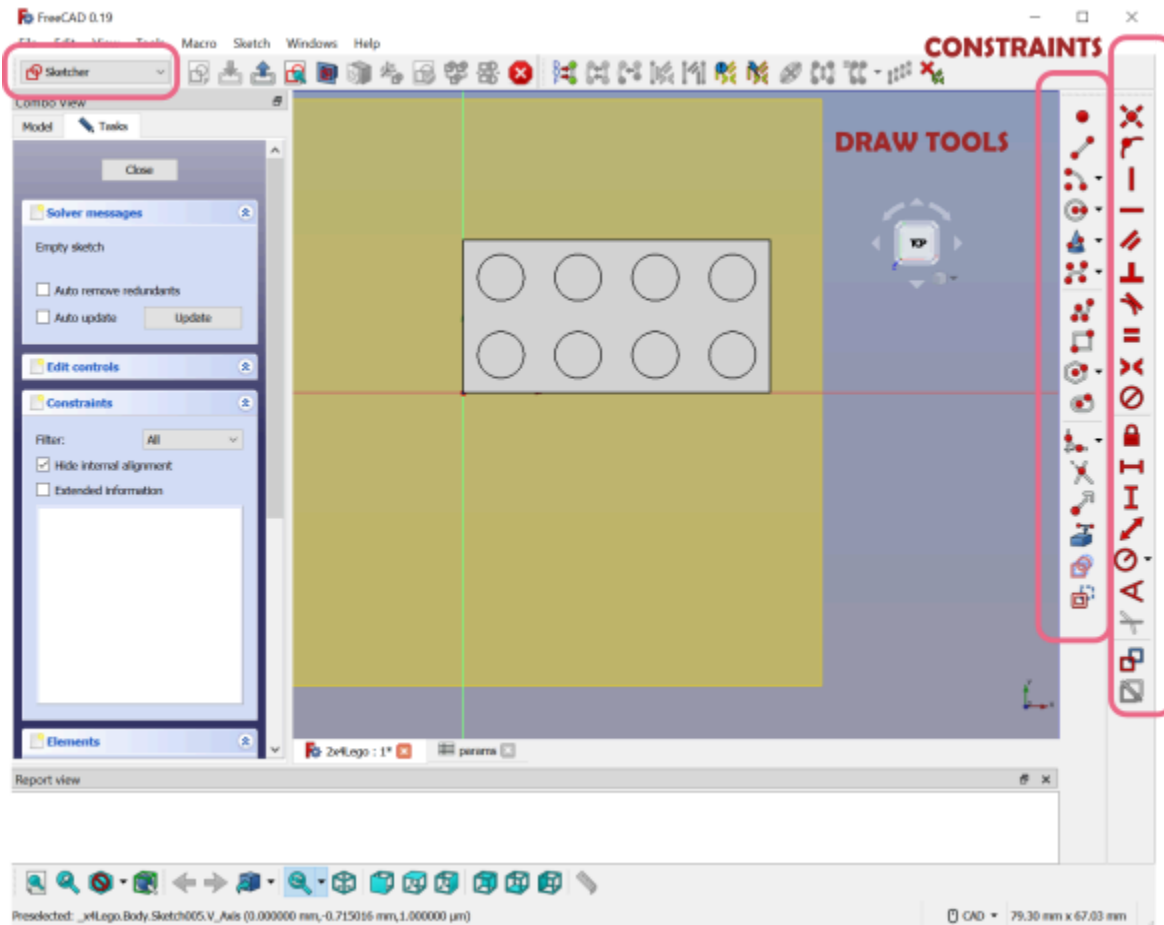
4. Spreadsheet workspace to create parameters



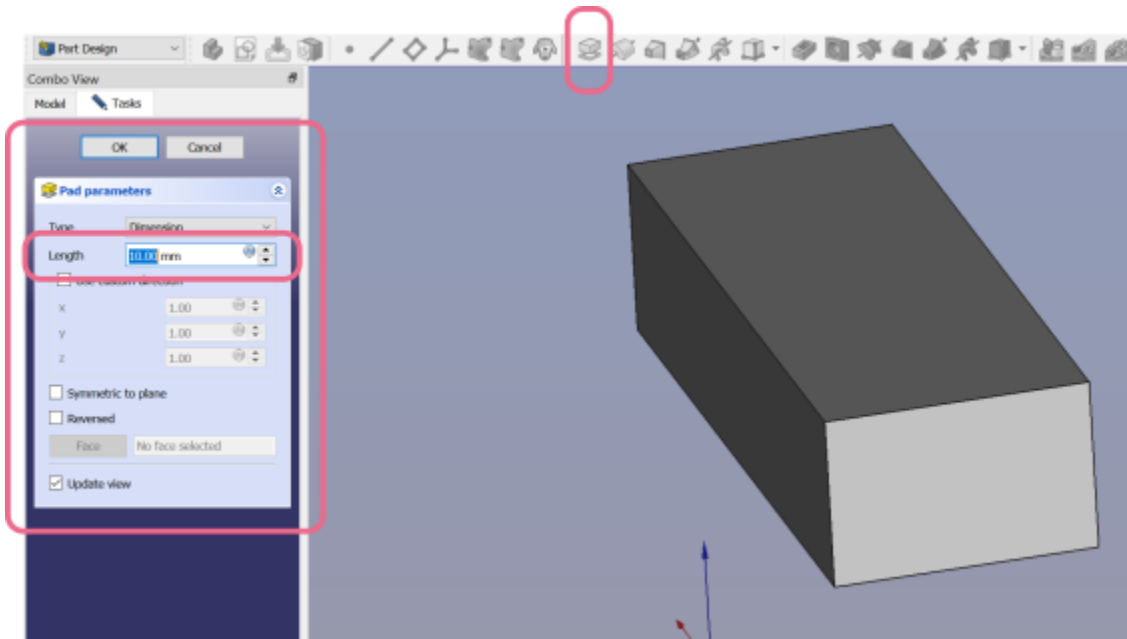
- Enter text description of the parameter in one column
- Next to it...enter the parameter values
- Click on each value to give it an (easy to remember, short, simple) "alias" >> cell turns yellow after Alias assigned
- Right click value cells >> units = mm
- Change spreadsheet name to something short and simple

5. Draw

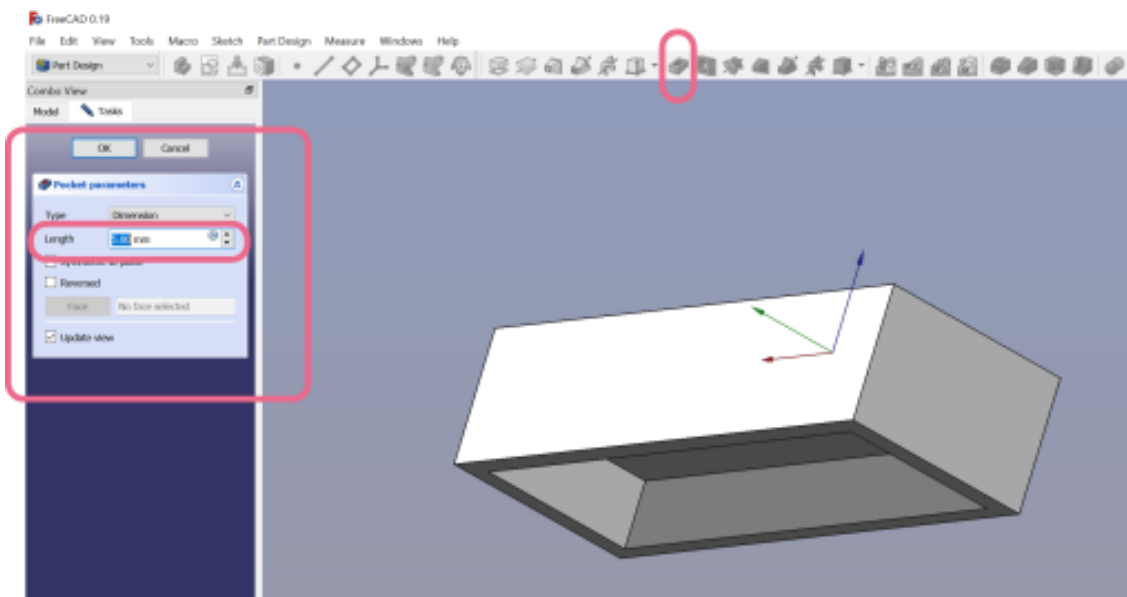
- LMB to add
- ESC to exit
- RED vertice manipulatable after draw
- Add constraints
- Add parameters (= to toggle Formula Editor)
- Remove all DOF



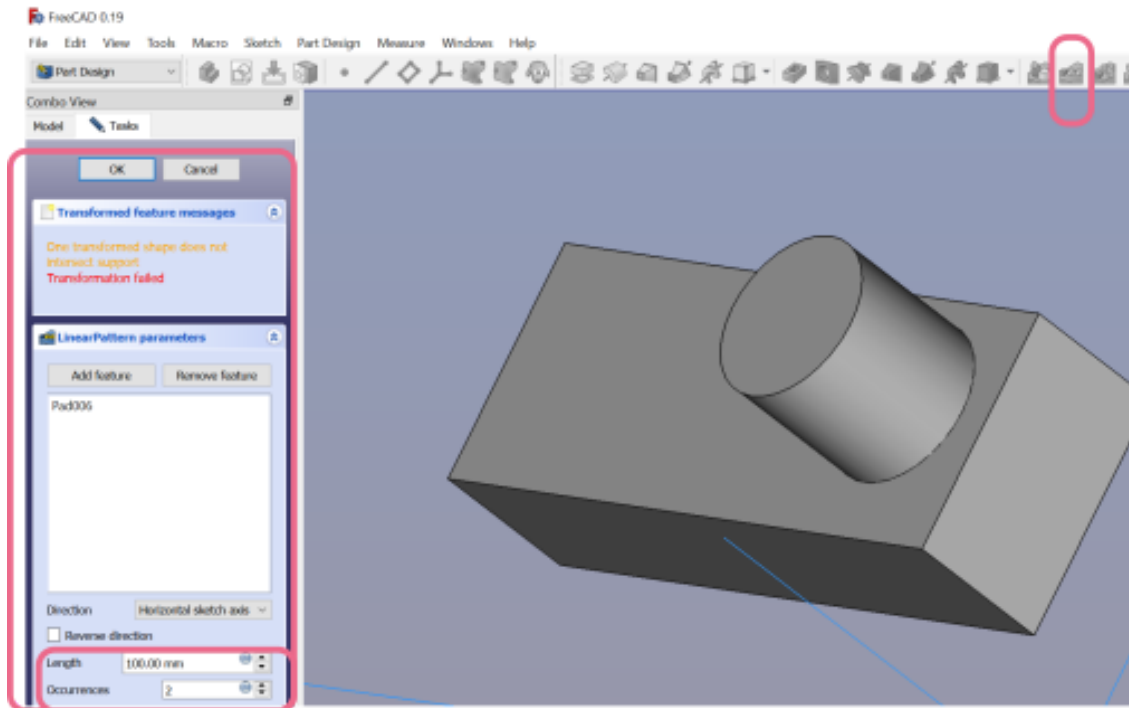
6. PAD = extrude Sketch to be a Positive 3D body



7. Shell = extrude Sketch to be a Negative 3D body (...used to cut holes)



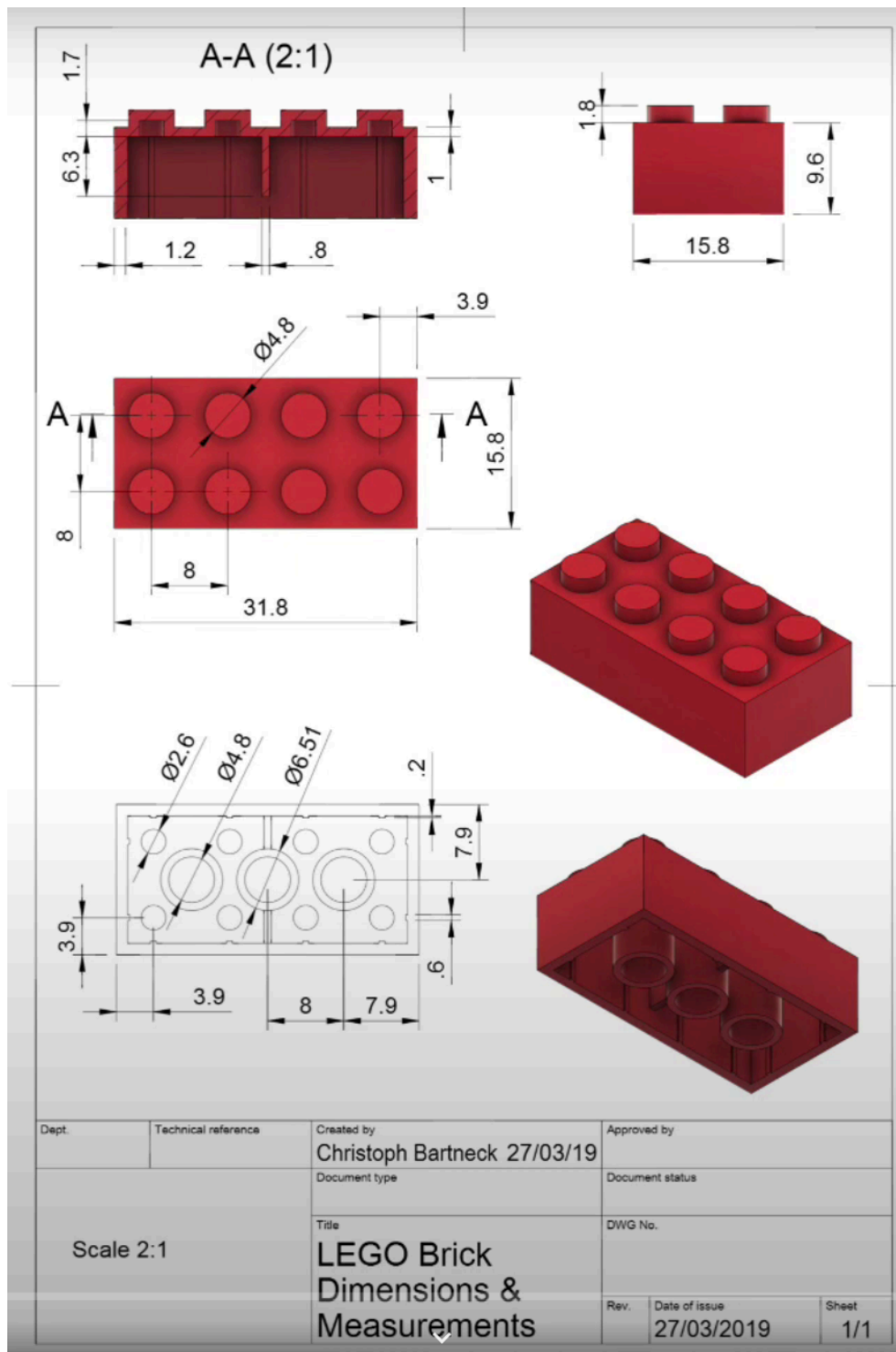
8. Array



9. Windows tile...for multiple windows

10. SPACE...toggles visibility of tree elements

Session Learning: Modeling A 2x6 Lego Brick



Session Assignment:

- 1. Practice using Fusion360 or FreeCAD...make a 3D model of some aspect of your Final Project idea**
 - a. Use Parametric Modeling techniques
- 2. Generate a Rendered image of your model**
 - a. Output as JPEG (Fusion 360 only)
- 3. Continue refining your 2D graphics for your 1-page Final Project Presentation slide**
 - a. Deadline: Dec 1
- 4. Document your 3D Design learning and modeling work process**
 - a. Text
 - b. Images
 - c. 1 link

Resources:

Fusion360

Product Design Online (Kevin Kenedy)

https://www.youtube.com/channel/UCooViVfi0DaWk_eqxIXXi0Q

Lars Chirstensen

<https://www.youtube.com/channel/UCo29kn3d9ziFUZGZ50VKyWA>

FreeCAD

Sketcher for Beginners

<https://www.youtube.com/watch?v=gbNg3mzm84s>

Parametric Modeling

<https://www.youtube.com/watch?v=fXoRAYv1wHQ&t=114s>

Constraints & Arrays

<https://www.youtube.com/watch?v=nfbrfItvOCA>

Topological Naming Problem

<https://www.youtube.com/watch?v=6p2vqEEWq4>