

Onshape / CAD

- [Onshape - Lecture #1 Notes](#)

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<https://docs.google.com/document/d/1mGUbdagblQCSvaXb1p8mCmoFG--0c67WHuSI-37BeuY/edit>
Things to Go Through
DOCUMENT 1

<https://bit.ly/3zaudHe> write this on the board.

- Getting it setup using <https://onshape4frc.com/getting-started>
- Going through the tabs
 - Parts and Assembly Tabs
 - Each Subsystem (climber, swerve, intake) Gets it own **Document**
 - To make sure that the Robot doesn't get too cluttered, but you can organize the way you want.
 - **Assemblies** - different parts form an assembly (wheel, axles).
 - **Parts** - where you model all of your individual parts, show up in the bottom left
 - ****Parts Studio** - interrelated parts can be modeled in the same environment (studio), (Dumb things that don't move might be in one parts studio)
 - Documents can be broken up into different part studios.
 - **Feature Tree** - Every action you take is in the feature tree, shows the history of all of the changes you've made, above the parts,
 - Everything is moveable and customisable.
 -

Tab features, bottom left; **Tab manager** to the left of the plus

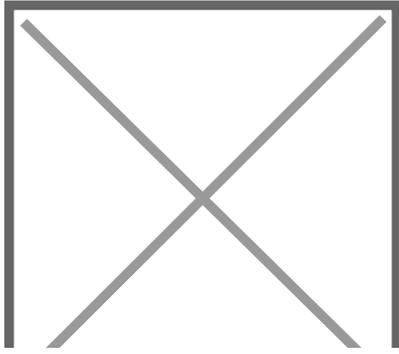
- **Version History**, under the onshape logo, branch and merge designs, high level feature tree
- We need to be able to move around, **Cube** to change the view, Press the drop down for more options
- Mouse Controls
 - Scroll in, **zoom in and out**, zooms were the pointer is
 - Clicking and holding the middle mouse button you can move **around one dimensionally**
 - **Double middle mouse click** to reset to the center
 - **Right mouse button** you can orbit the part
 - Apply to both parts and assemblies.
 - If you are really used to using solidworks or fusion go to your account, preferences, and pick the mouse settings

DOCUMENT 2

- Creating a **Sketch**
 - Outline of our model in 2D, we use it to make something 3D
- **Origin is** the central point and what we base the model off of.
- Press sketch, leave the dialog box up in the corner, **If you see a dialog box that's highlighted, if you click on something on the screen it will get put into the box.**
- If you want to do what the dialog box says leave it open

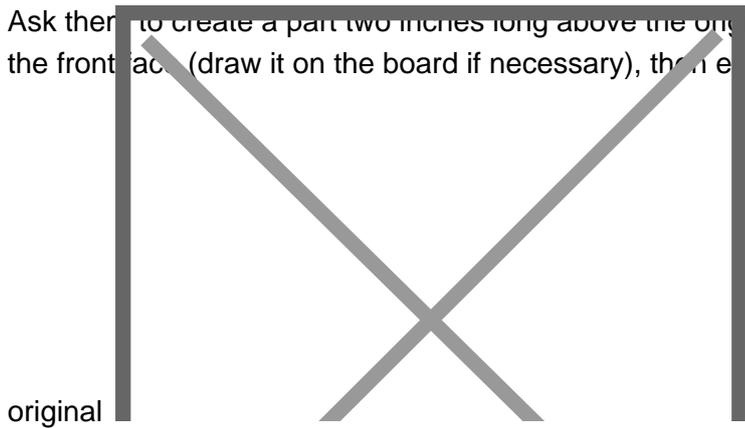
- You can toggle visibility on the planes in the feature tree.
- Toolbar has a bunch of shapes, you will most likely use a lot of these
- Pressing it **activates the tool**, if you press escape it will get you out of the tool, after you're done using a tool **PRESS ESCAPE**, so you don't get lost in a random tool.
- **DONT CLICK AND DRAG. its two clicks**
 - If you need a specific measurement just type it in.
- **If you click, and click again it selects both and doesn't get rid of the first**
- **To deselect**, click on the white space, or use the spacebar
- **Window Select**, click and drag
 - Left to right - box is blue, **only selects things completely inside the box**
 - Right to left - box is yellow, **selects everything it touches**
- If you're not **interacting with a shape the lines are blue**
- You can scale by clicking the corners but you can add **Constraints**
 - **Constraints** - are geometric rules that you can impose on a model and they have to follow then (hover over points and you can see parallel and perpendicular
- **Blue things are unconstrained**
 - **WHEN YOU ARE DONE WITH A PART EVERYTHING SHOULD BE CONSTRAINED**
- If you make a rectangle in accordance with the origin it will add a constraint with the origin.
- **Black means constrained**

- If you make a rectangle starting on the origin (yellow square) coincident constraint, a corner point is attached to the origin.
- **First thing you do in a sketch is constrain something to the origin**
 - So it doesn't change everything.
- **Dimension tool, Constraint** - just put it on the screen in a way that looks nice
 - You can type in measurements in different units.
 - You can dimension between two lines,
- **Extrude**
 - After the dialog box is open select a region, a closed in area (our rectangle), (highlighted blue), make sure its on solid, 4 inches width
- Add another rectangle on the front side using sketch tool
 - Dimension each side .25 inches from the sides
 - Extrude this rectangle, (anybody know what the two tabs do?)
 - **Add tab adds to the original part** (so a new part is not created in the parts list)
 - **New tab creates a new part** (a new part is created in the list)
 - **Remove tab deletes a part of the shape** (creates a hole)
 - Up to part, up to vertex instead of blind, or through all, mess around with them.
 - **Merge Scope** tells onshape what part to make changes to, **if you add multiple parts it will remove from multiple parts.**



- Add 1 inch

Ask them to create a part two inches long above the original rectangle with a gap of .5 inches, on the front face. (draw it on the board if necessary), then extrude it two inches perpendicular to the



Show off the **Feature tree**, right click and edit certain parts in history to make changes

- Make a circle on a side of the gray rectangle and dimension WHILE making the circle, if it's in a gray box it's waiting for some sort of input.
- Ask them to make this shape ?
- Finish the sketch and it should creates a hole