

Introduction to CAD

August 17/18, 2019

What is CAD?

- Computer Aided Design
 - Creates a 3D model
 - Allows cheap iteration of designs
 - A close-enough prediction of final product
- Does not replace real visualizations
 - Hand-drawn sketches are cheapest

CAD Terminology

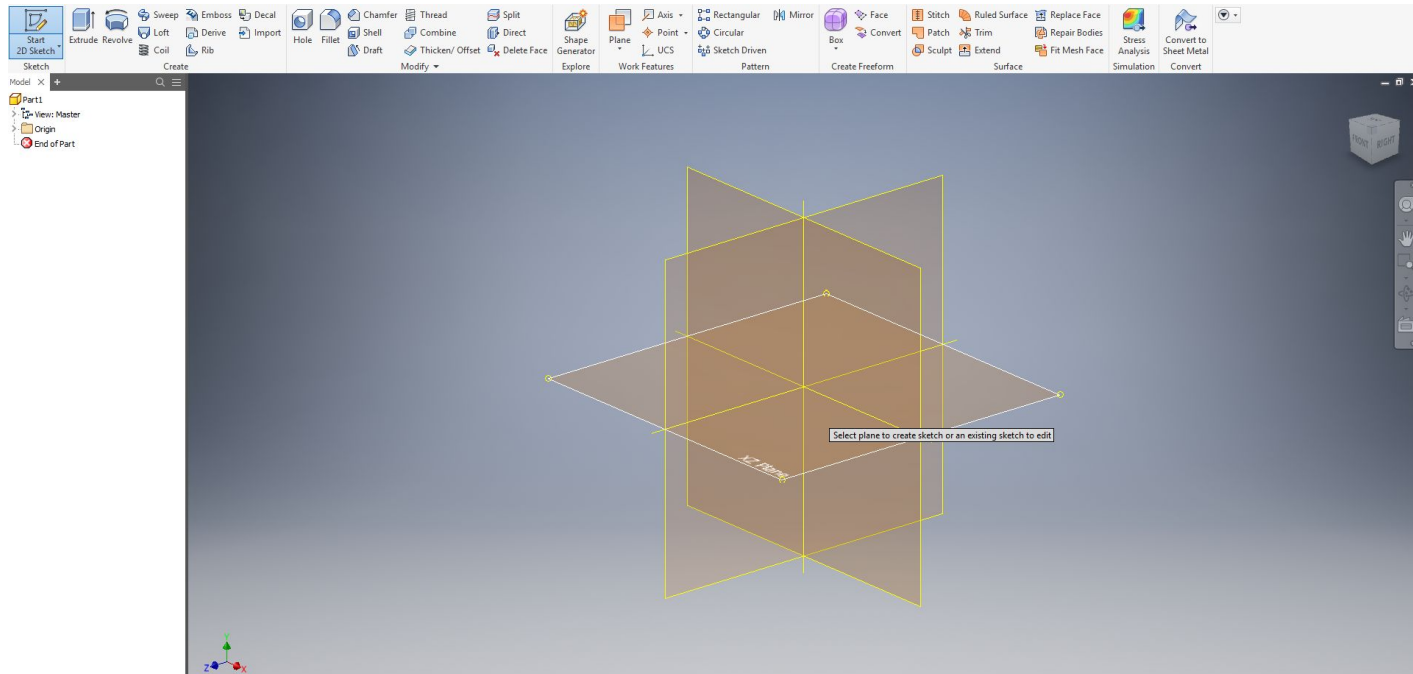
- Sketch
 - Lowest level of organization - 2D drawing that will be extended later
- Feature
 - A 3D aspect of a part. Usually consumes a sketch but can be built off of another feature
- Part
 - Collection of features and sketches that represent a real “thing”
- Assembly
 - Collection of parts stuck together by faces or edges

CAD Space

- CAD is a 3D space
 - Has coordinates like geometry
- All design makes use of geometric shapes
 - Drawing circles before axles
 - Drawing rectangles before metal plates
- Complex designs start with simple geometries
 - Break down complex shapes into a series of simple ones

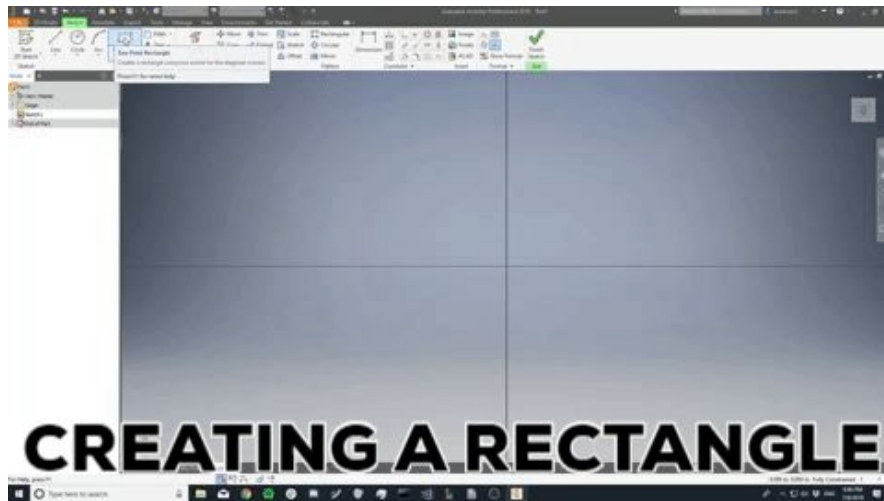
Sketch

- Represents a 2D plane



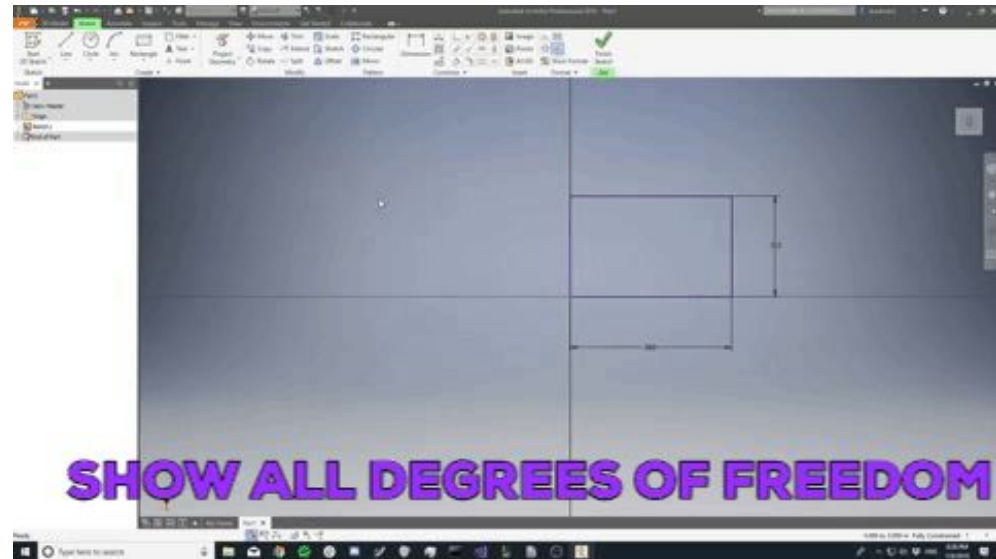
Drawing on a sketch

- Computers aren't that smart
 - If you don't tell the CAD something, it won't know it
 - “Constraining” is the act of defining some measurement or relationship in the CAD
 - “This line is 3 inches long” | “These lines are parallel” | “This angle is 10 degrees”
 - A quick rectangle (*Why are the lines green?*)



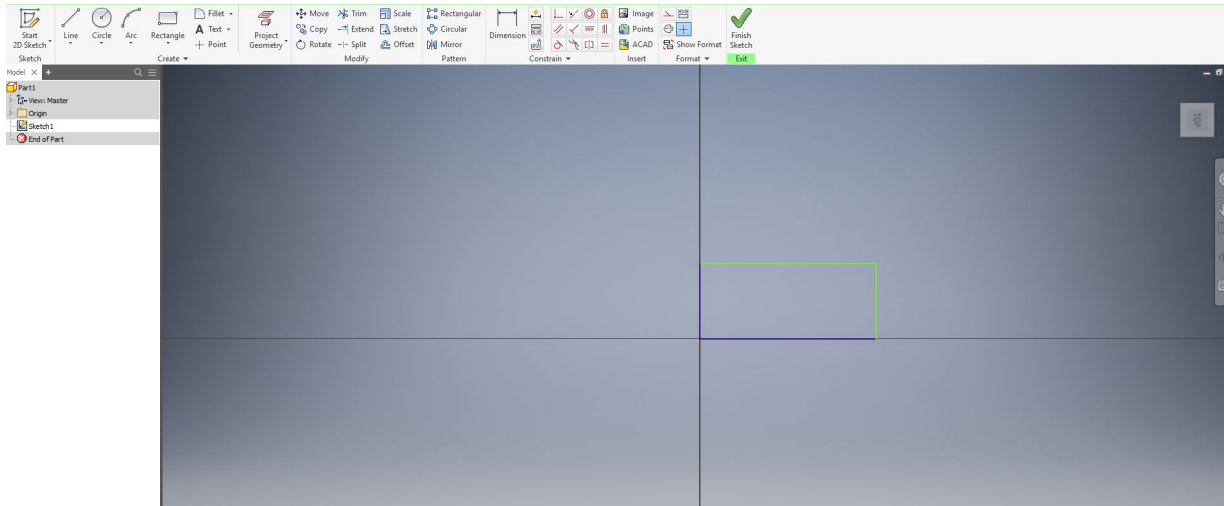
Oh No! My CAD Won't Constrain!

- Something isn't constrained and I don't know what
 - “Show Degrees of Freedom” is your best friend
 - Right click anywhere on the sketch
 - Red arrows indicate where lines can move



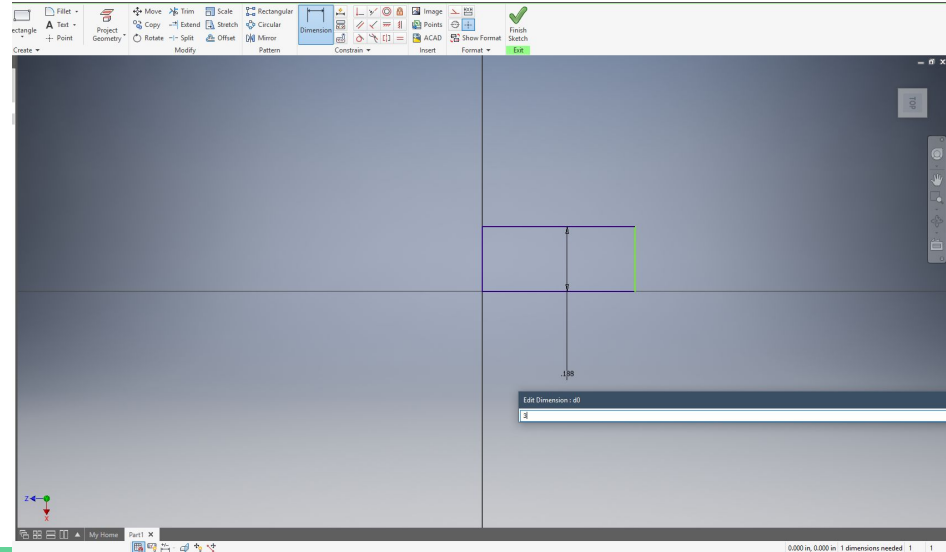
Sketch (cont.)

- “Constraining” a sketch tells the CAD *where* everything is
 - In the example, the green lines differ from the blue
 - **Blue** means their locations are *known*
 - **Green** means their locations are *unknown*
 - Right now it is guessing until you clarify



Sketch Constraints (cont.)

- How do we fix it?
 - Define a green position relative to a blue position (or an axis)
 - In the example, we will say the green horizontal line is 3 inches from the blue
 - Sketch > Constrain > Dimension



Sketch Constraints (cont.)

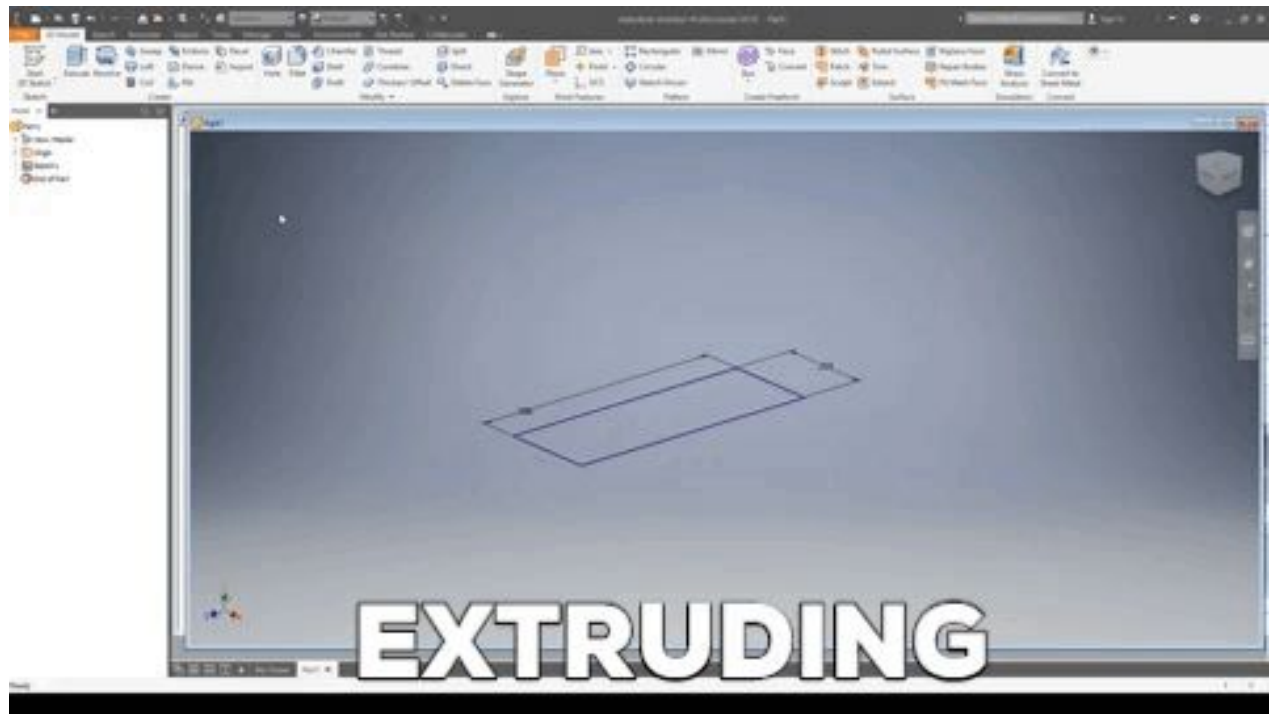
- There are constraining methods available in inventor
 - Those will be covered in future lessons
 - The important part is to be aware of what constraints you use
 - Using the correct constraints will make complex geometries **much** easier
 - Always ask yourself “If I moved one line, what will break?”

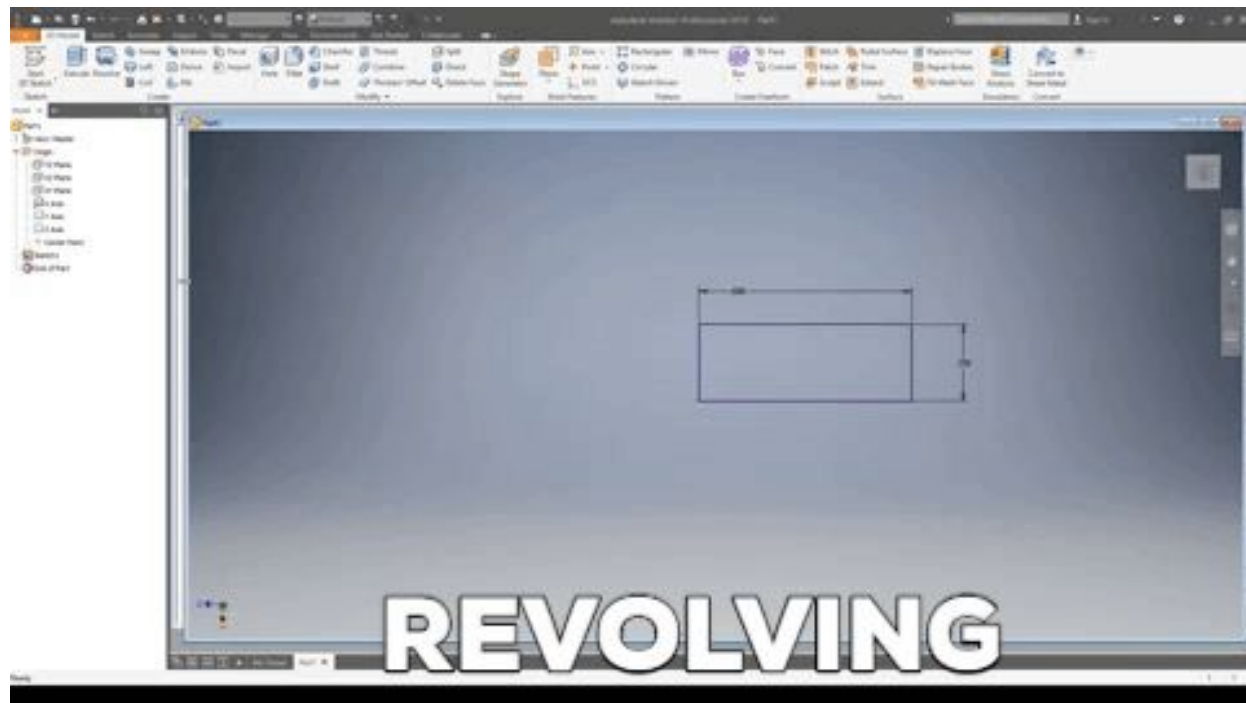


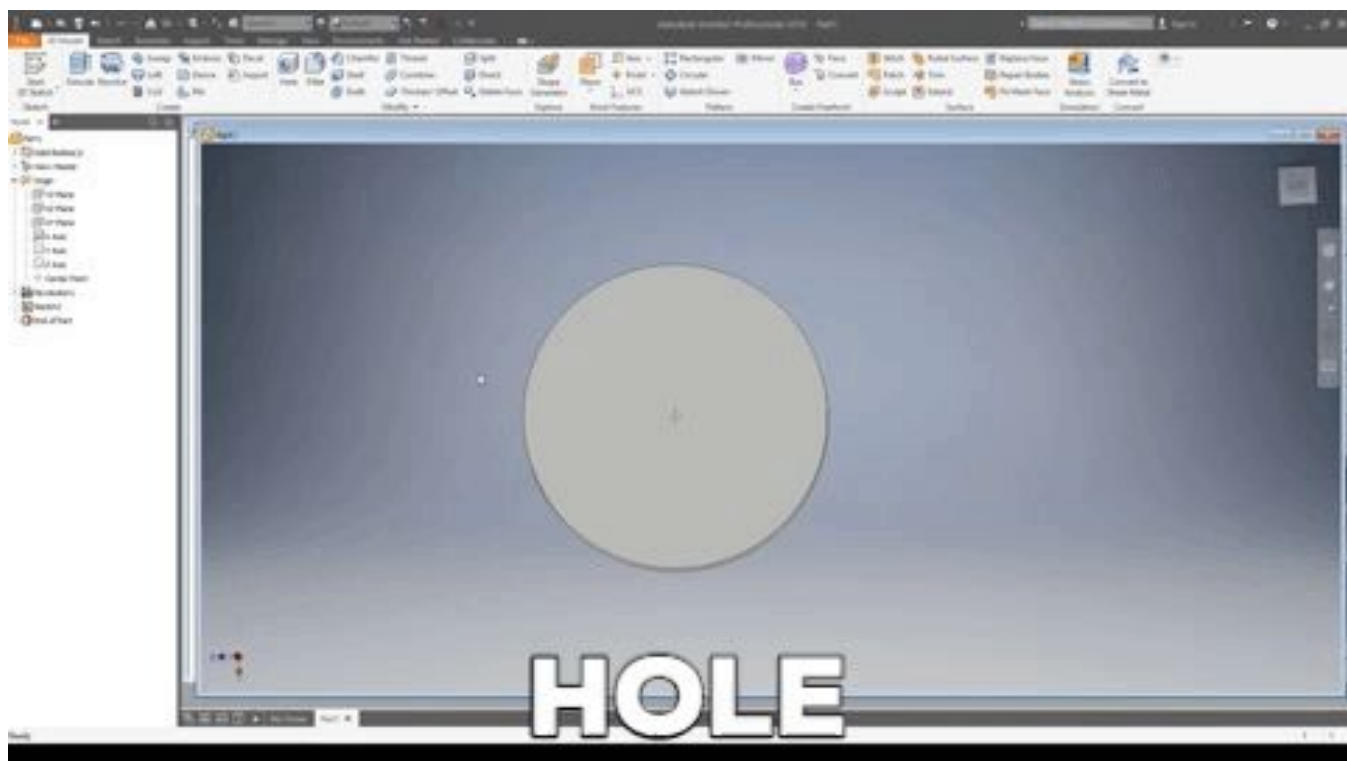
Features

- The 3D representation of manipulating a sketch
 - Extrude
 - Imagine dragging a piece of paper in one direction
 - Revolve
 - Same thing except drag the paper in a path orbiting an axis
 - Others
 - Hole
 - Useful for screw holes (Think hole puncher)
 - Sweep
 - Uncommon, used for something like a paperclip



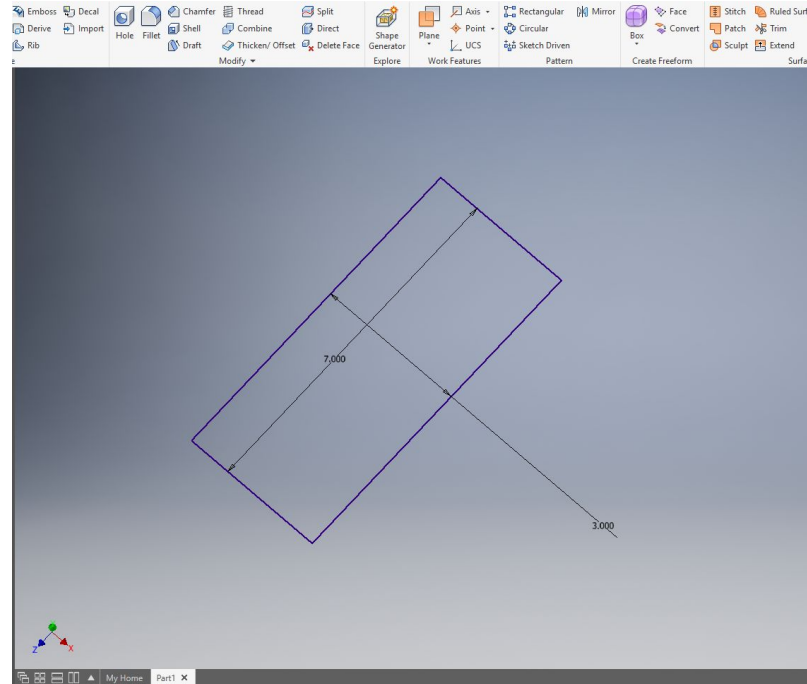






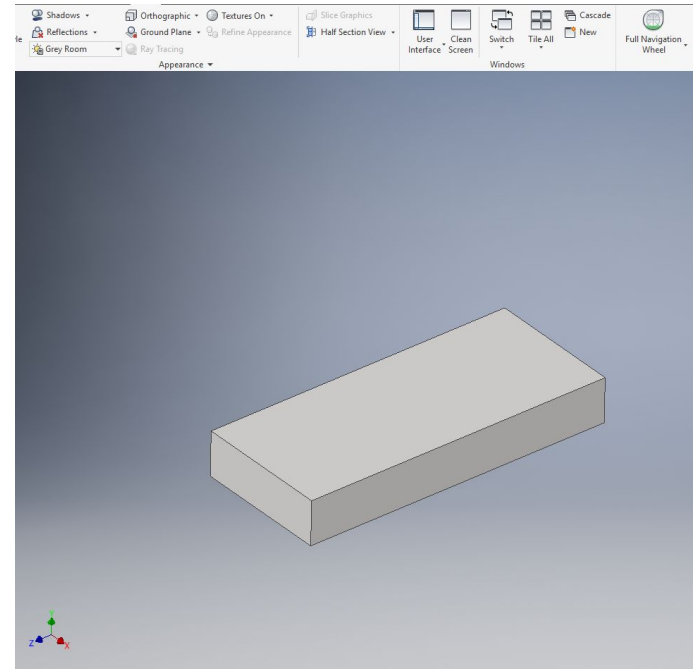
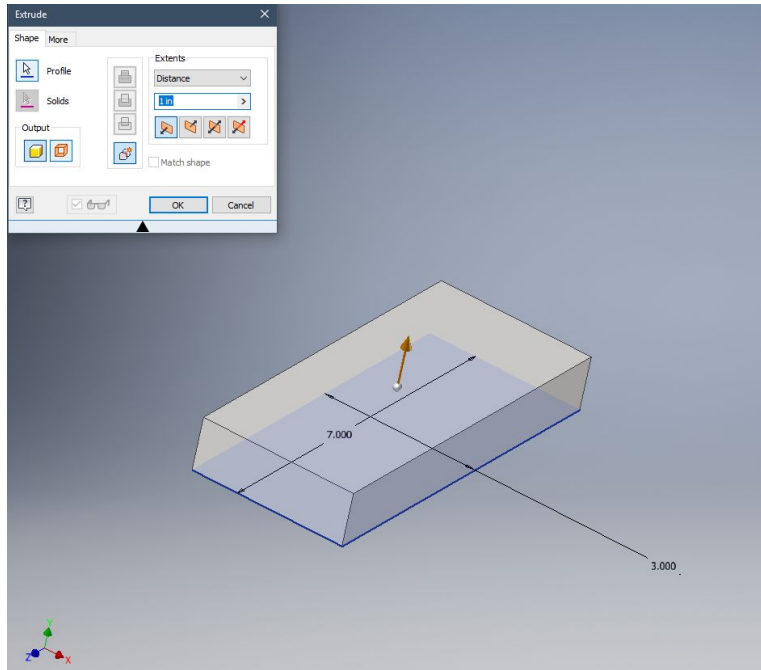
Back to the sketch

- Creating a rectangular prism with the previous sketch we get...



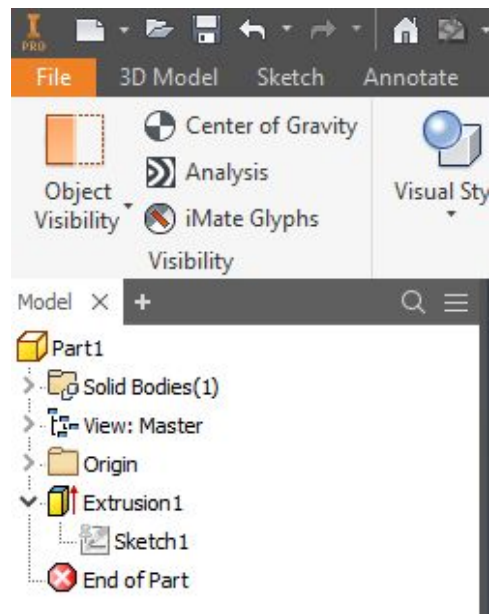
Back to the sketch (cont.)

- The sketch is used as a building block for this feature
 - This is also a part even though it is only made up of one feature



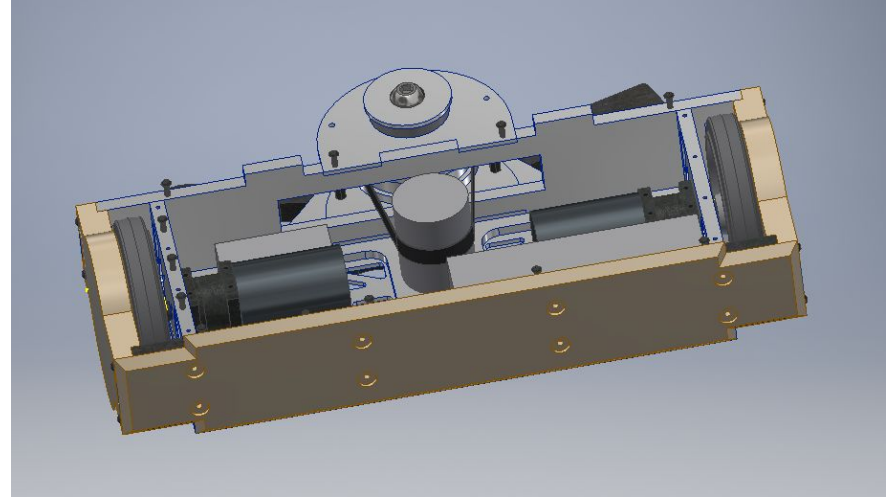
Parts

- Simply a collection of features performed in the same space
 - These files will have an .ipt ending
 - They have a “filesystem” to help understand all their parts
 - As seen below
 - Part > Feature (Extrusion) > Sketch



Assemblies

- This is where it all comes together (literally)
 - Individual “part” files are added to the assembly file
 - Several options exist to constrain them
 - Basically imagine where the real parts will touch and use that
 - Each shaded line is a division between parts



Recap

