Building Your CAD Toolkit

An Introduction to Solidworks

Jack Naylor
Casual Academic - University of Sydney

31/08/21

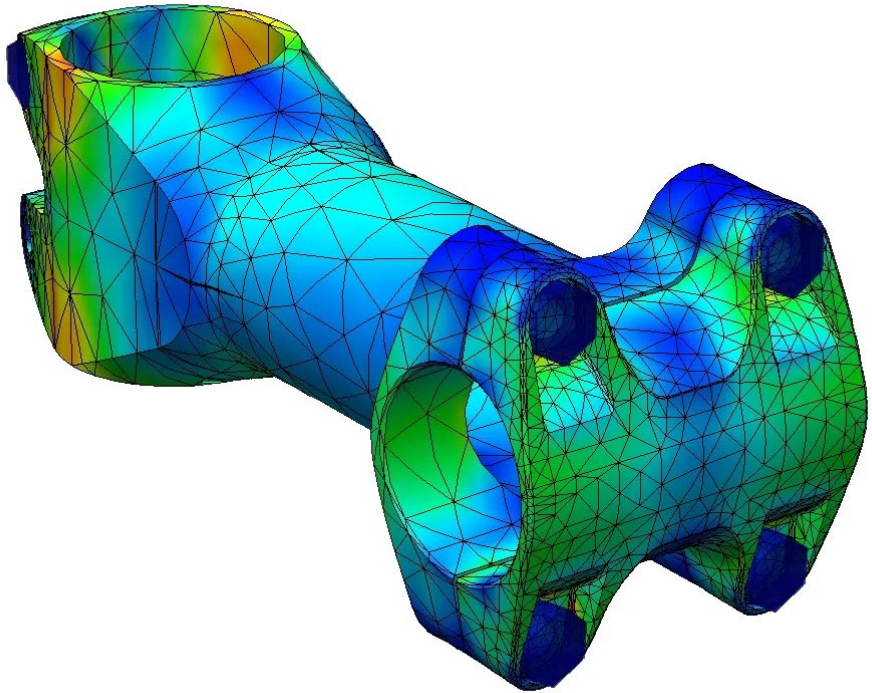
Today's Goals

1. Get you interested in Solidworks
2. Help you understand why you need to know CAD
3. Teachyou the basics
4. Get you feeling confident in using CAD going forward

What’s so good about Solidworks?

An Intro to Solidworks...

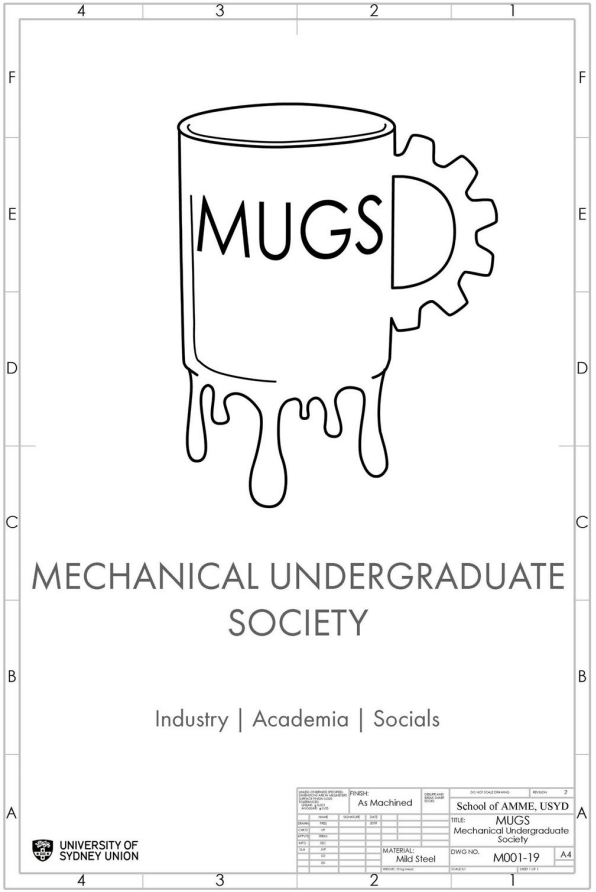
- Industry standard software for mechanical design
- In-built simulations and good interfacing with other packages
- Project management framework in-built



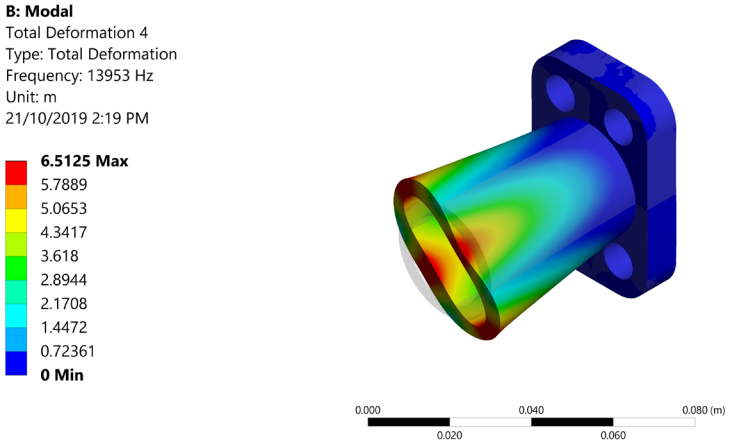
- Able to produce 2D engineering drawings from parts, making design to manufacture easier

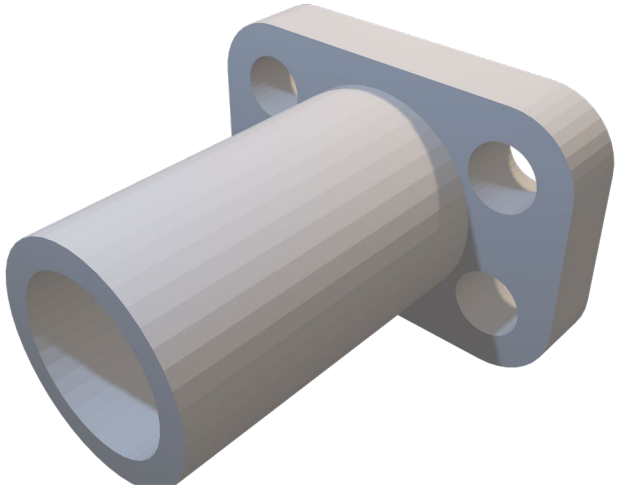
- Toolboxes for plastics, sheet metal (e.g. bending welding), CAM, electrical routing, pipes etc.

- Most versatile “general” CAD software out there



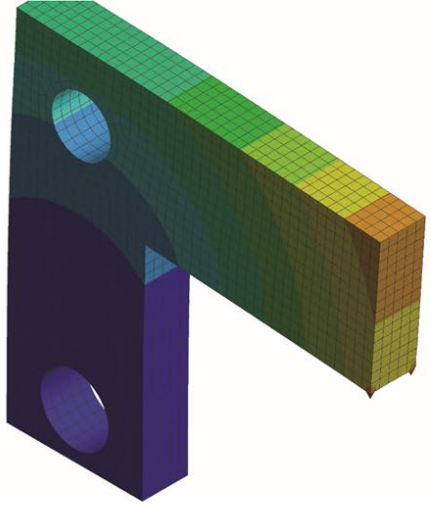
Also opens a whole new world of computational engineering!





D: Static Structural
Total Deformation
Type: Total Deformation
Unit: m
Time: 1
5/11/2019 12:45 PM

1.0381e-5 Max
9.2274e-6
8.0739e-6
6.9205e-6
5.7671e-6
4.6137e-6
3.4603e-6
2.3068e-6
1.1534e-6
0 Min

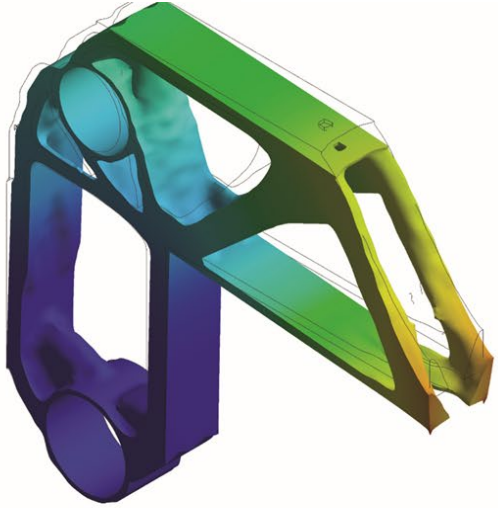


Solidworks models can be imported into ANSYS to do topology optimisation, design analysis and validation, CFD, magnetodynamics studies etc.

0.000 0.4
0.200

E: Model, Static Structural
Total Deformation
Type: Total Deformation
Unit: m
Time: 1
8/11/2019 10:31 AM

2.5121e-5 Max
2.2329e-5
1.9538e-5
1.6747e-5
1.3956e-5
1.1165e-5
8.3736e-6
5.5824e-6
2.7912e-6
0 Min



Key component of senior engineering units and of the real world!

Additive Vs. Subtractive Modelling

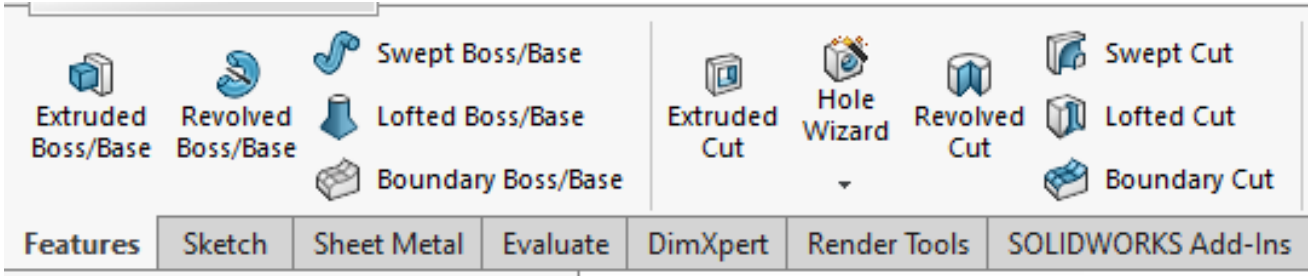
What is Additive Vs. Subtractive Modelling?

- Do we “add” or “subtract” material to create a model?
- Important when looking to build parts:
  - 3D printing is an additive process, want to use additive modelling
  - Milling is a subtractive process, want to use subtractive modelling
- Cannot use one or the other, need to use a combination

Solidworks is built for this!

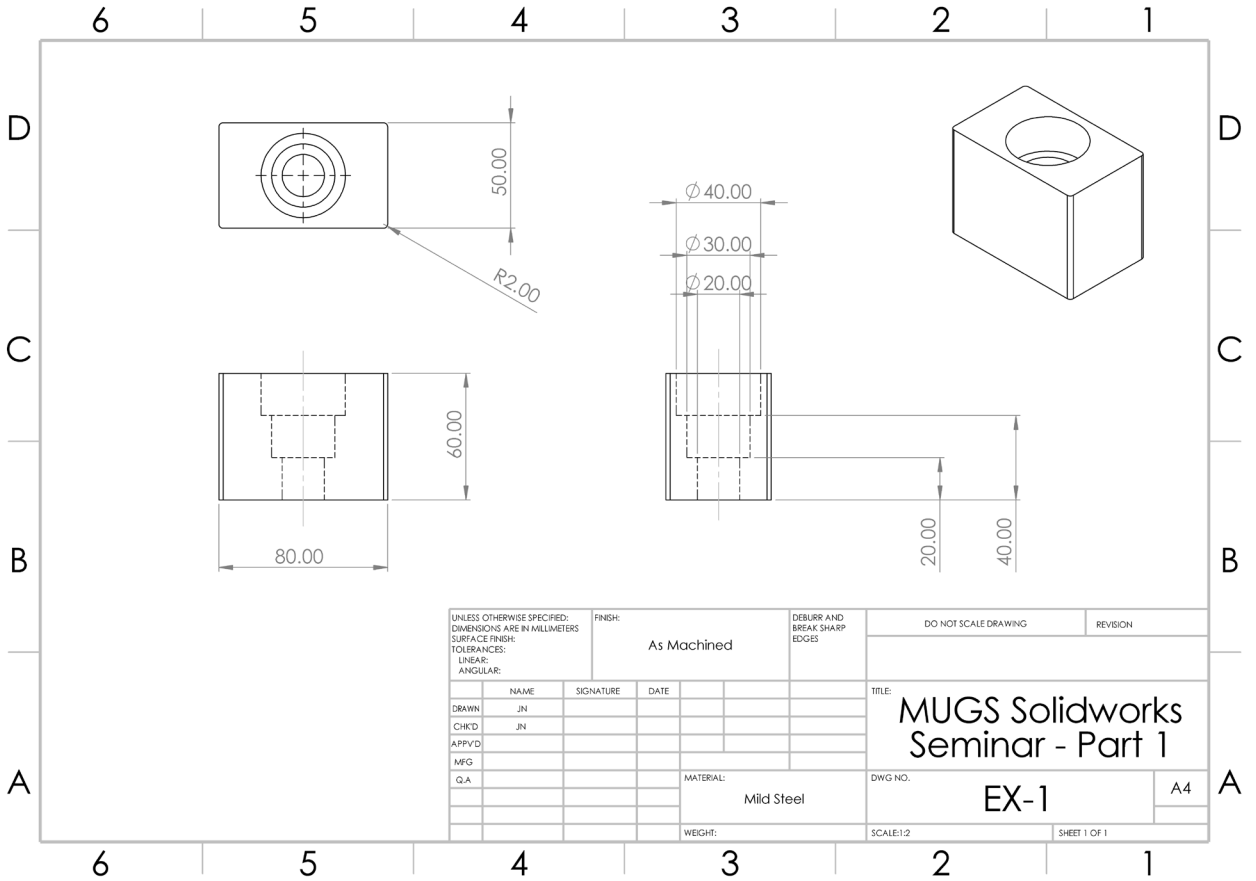
Additive Features

Subtractive Features



Consider this exercise:

- There are two ways of constructing the block
- I'll also use this to familiarise you with reading engineering drawings



Before attempting a CAD model…

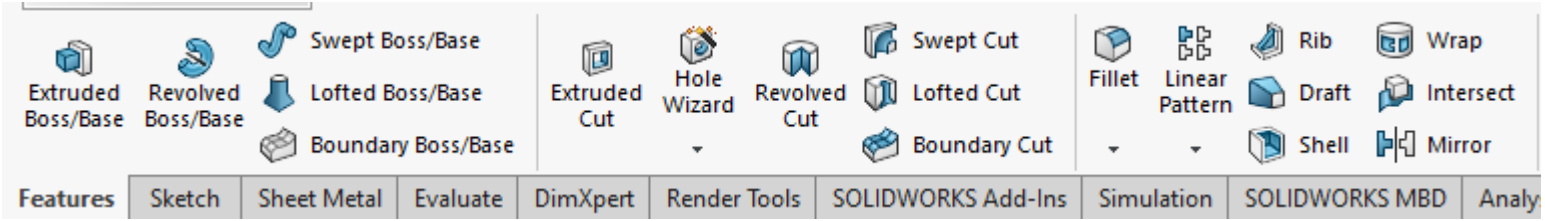
- Set out a game plan - start with a block and remove material OR start with a base sketch and start building things onto it
- Get an idea for proportions of the part, what is the largest and smallest features on the part and what's their relative size?
- Look for symmetry – CAD programs allow you to mirror parts, this can reduce the amount of modelling you need to do!

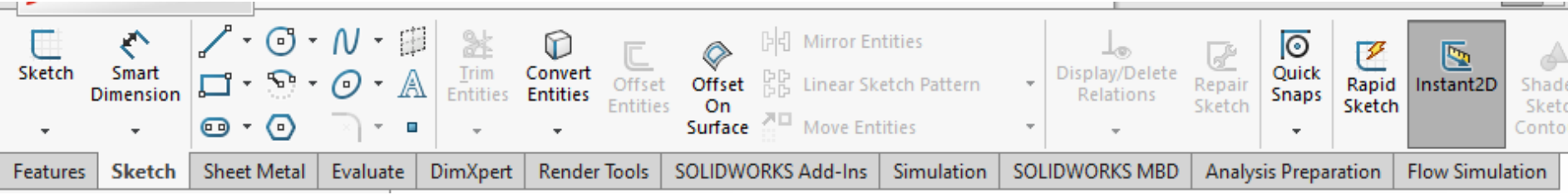
Parts

Brief Overview of Creating Parts

- To cut, or extrude we need a sketch on a plane or surface
- Able to mirror features/sketches, or rotate them around an axis
- For speed, you want to minimise the amount of features you need to sketch individually preplanning a part is key!
- Try to use the hole wizard wherever possible - if you use screws/boltsSolidworks can fill these in automatically!

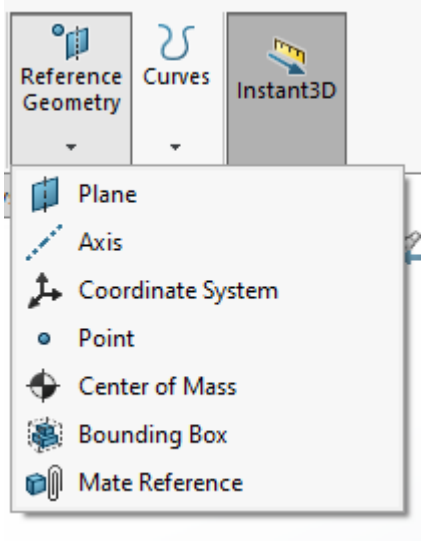
You want to create features , using sketches

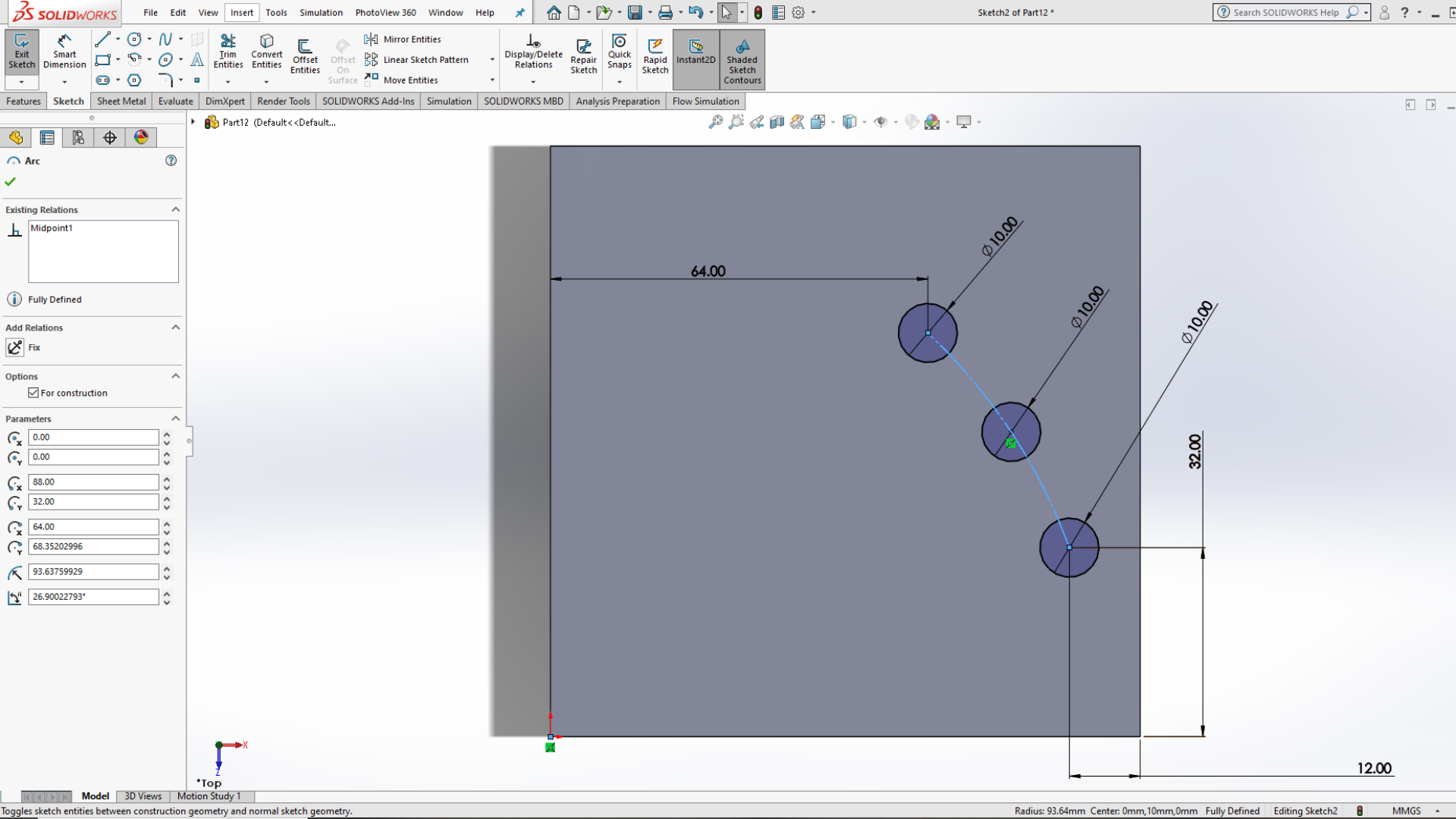


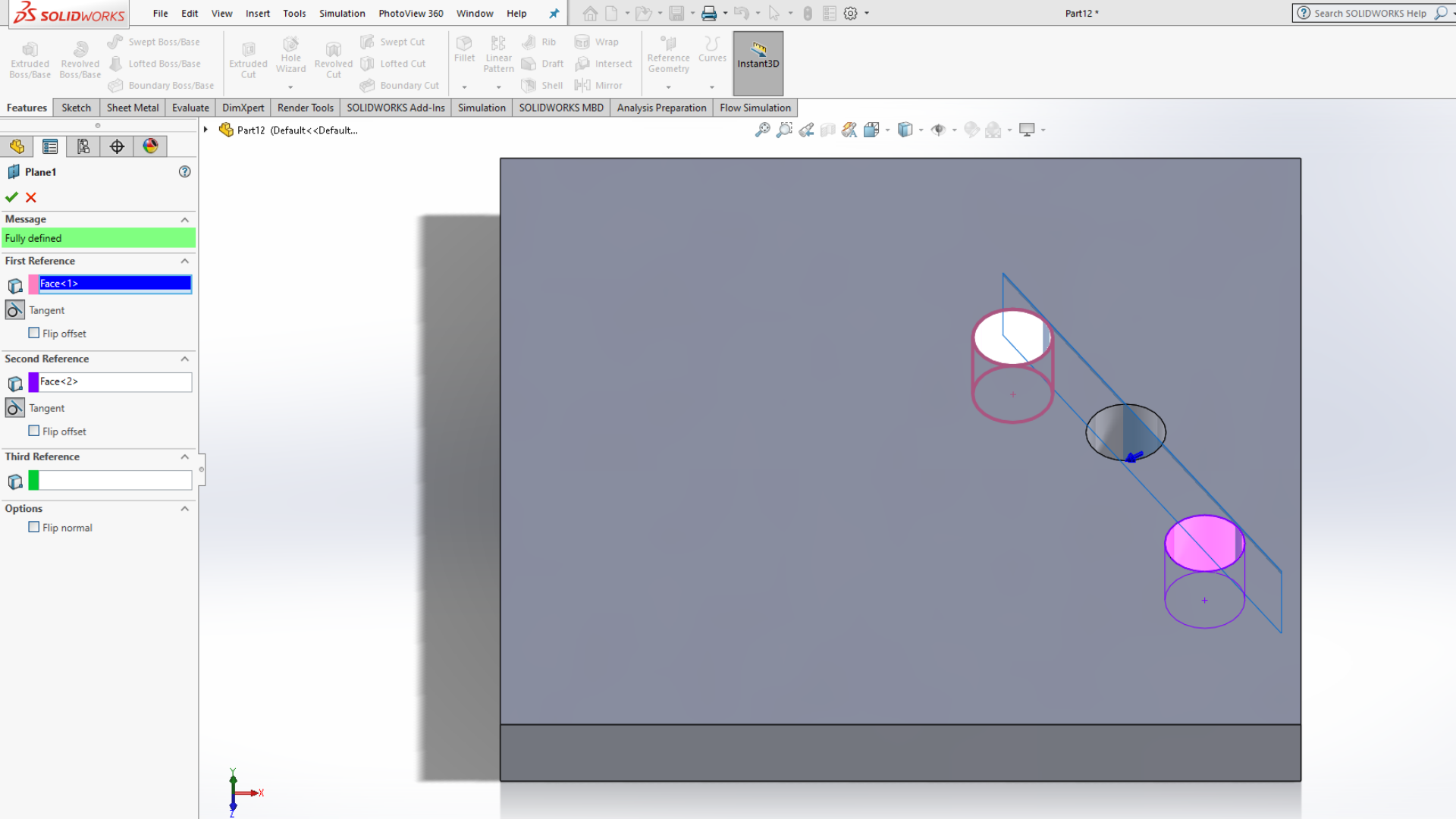


Reference Geometry

- Makes your job EASY!
- Allows you to start creating features off of surfaces to join later
- Another really good tool in sketches is “Construction Geometry”
    - These lines will not be used in the sketch, but rather to help you place things
    - For example, three holes along a curve in a square part

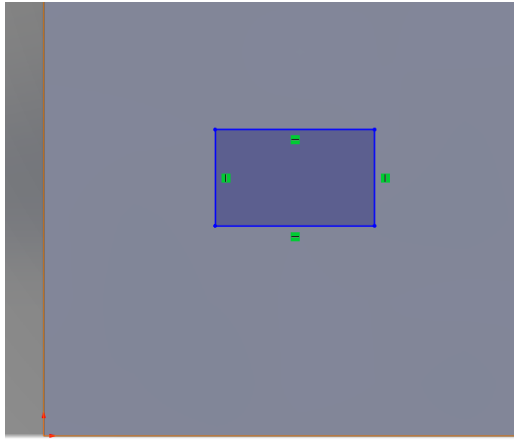


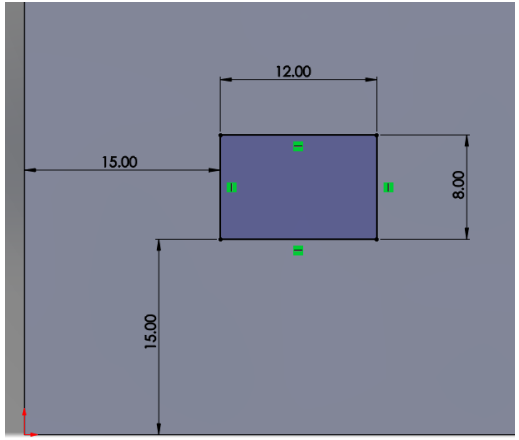

SOLIDWORKS
File Edit View Insert Tools Simulation PhotoView 360 Window Help
Sketch2 of Part12 *
Search SOLIDWORKS Help
Exit Sketch
Smart Dimension
Trim Entities
Convert Entities
Offset Entities
Offset On Surface
Mirror Entities
Linear Sketch Pattern
Move Entities
Display/Delete Relations
Repair Sketch
Quick Snaps
Rapid Sketch
Instant2D
Shaded Sketch Contours
Features Sketch Sheet Metal Evaluate DimXpert Render Tools SOLIDWORKS Add-Ins Simulation SOLIDWORKS MBD Analysis Preparation Flow Simulation
Part12 (Default<<Default...
Arc
Existing Relations
Midpoint1
Fully Defined
Add Relations
Fix
Options
For construction
Parameters
0.00
0.00
88.00
32.00
64.00
68.35202996
93.63759929
26.90022793°
64.00
Ø10.00
Ø10.00
Ø10.00
32.00
12.00
*Top
Model 3D Views Motion Study 1
Toggles sketch entities between construction geometry and normal sketch geometry.
Radius: 93.64mm Center: 0mm,10mm,0mm Fully Defined Editing Sketch2 MMGS

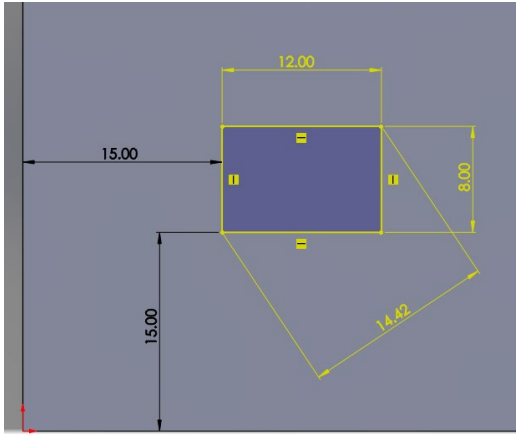

SOLIDWORKS
File Edit View Insert Tools Simulation PhotoView 360 Window Help
Part12 *
Search SOLIDWORKS Help
Extruded Boss/Base
Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base
Boundary Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Swept Cut
Lofted Cut
Boundary Cut
Fillet
Linear Pattern
Rib
Draft
Shell
Wrap
Intersect
Mirror
Reference Geometry
Curves
Instant3D
Features Sketch Sheet Metal Evaluate DimXpert Render Tools SOLIDWORKS Add-Ins Simulation SOLIDWORKS MBD Analysis Preparation Flow Simulation
Part12 (Default<<Default...
Plane1
Message
Fully defined
First Reference
Face<1>
Tangent
Flip offset
Second Reference
Face<2>
Tangent
Flip offset
Third Reference
Options
Flip normal

Sketches should be fully constrained

- Solidworks can only do what you tell it to do– it knows nothing about the part you are designing
- It does know whether it has enough information to build what you want it to
- 3 Types of Sketch Definitions: Under Defined, Fully Defined, Over Defined (red = unsolvable, yellow = over constrained)

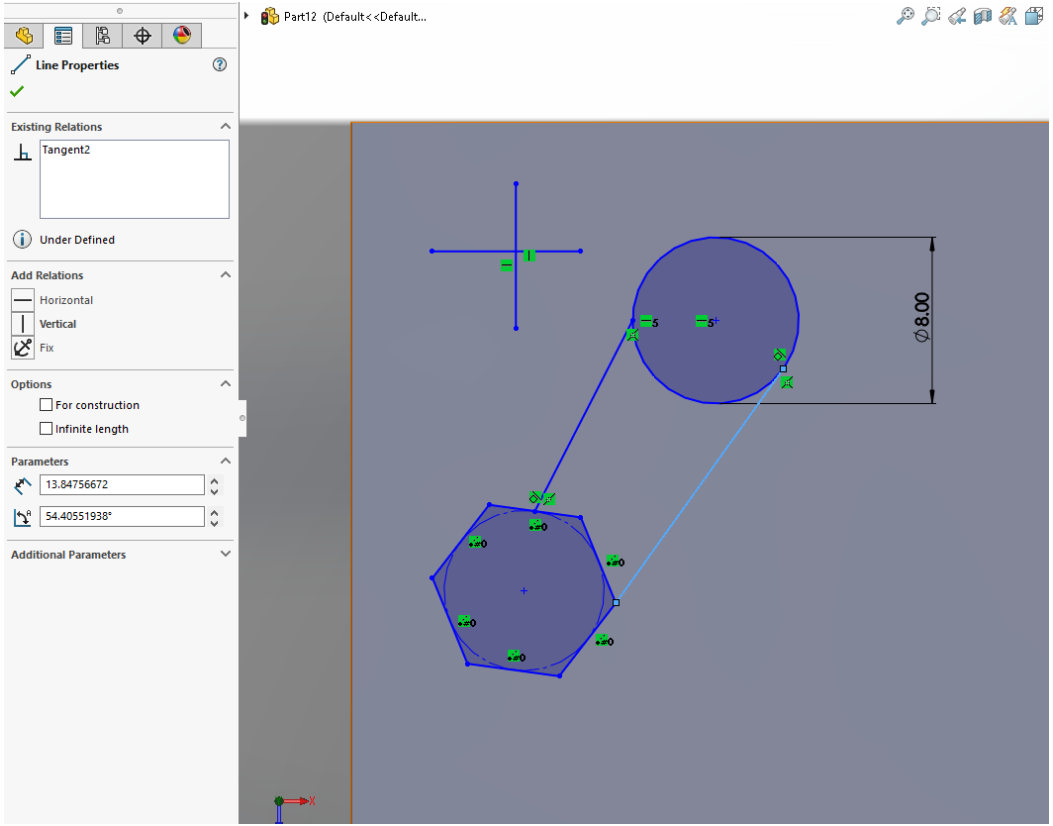






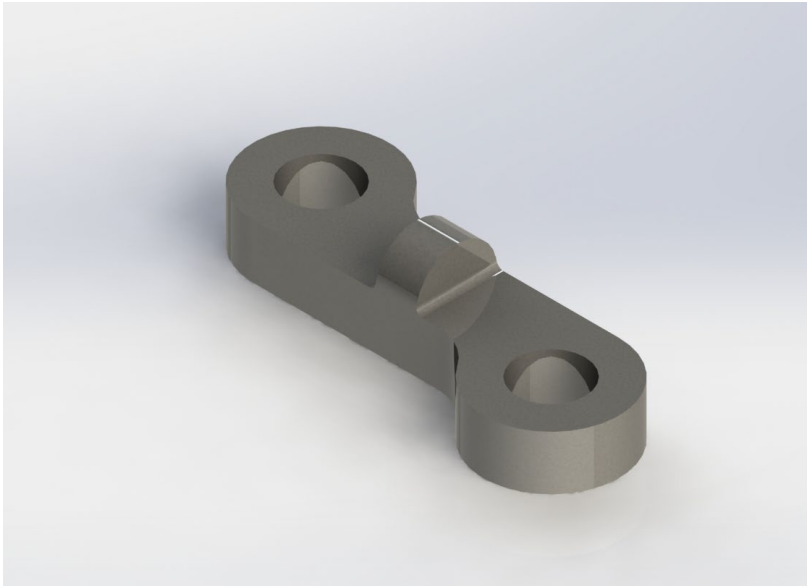
Geometrical Relations

- Very handy!
- Able to relate sketch entities to one another
  - Tangent relations
  - Horizonal/Vertical
  - Midpoint/Coincident
  - Etc.



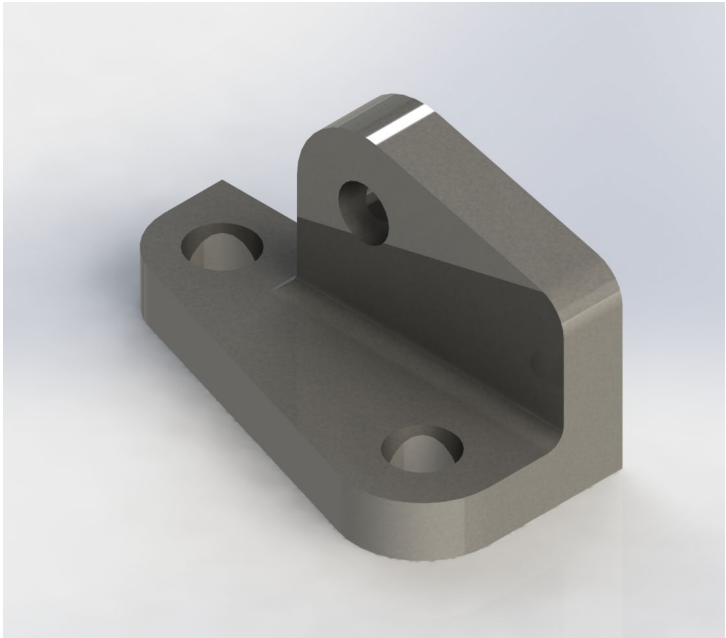
Let’s make a part!

- We’ll be taking a look at Exercise 2
- This will give some practice with reference geometry and geometrical relations



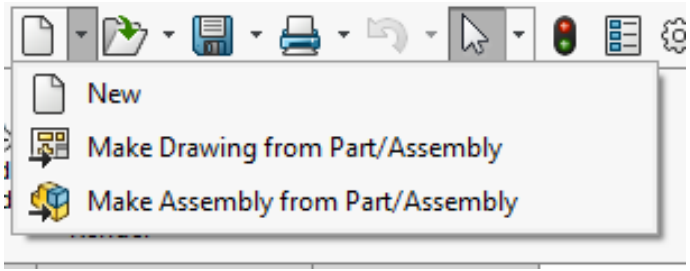
(Time permitting)
Let’s make another part!

- Slightly more complex geometry in Exercise 3
- More work with geometry relations



Drawings

- If you wanted to make a drawing from the part, you can click on New➢ Drawing from Part
- You don’t need to type the dimensions out again- it is all stored in the file.
- This is primarily how designs are interpreted in industry, and requires a whole undergrad unit on how to understand (all needs to be done to AS1100 as well)



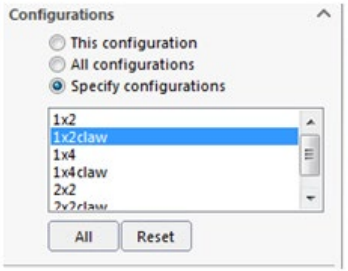
Assemblies

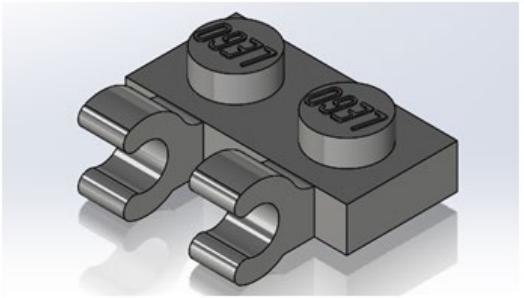
So we have our parts - now what?

- Parts fit together into a final product, called anassembly
- This allows us to “virtually construct” the part
  - See how it fits together
  - Check interferences, tolerances
  - Perform motion studies etc.

The good thing is – it’s just like Lego!

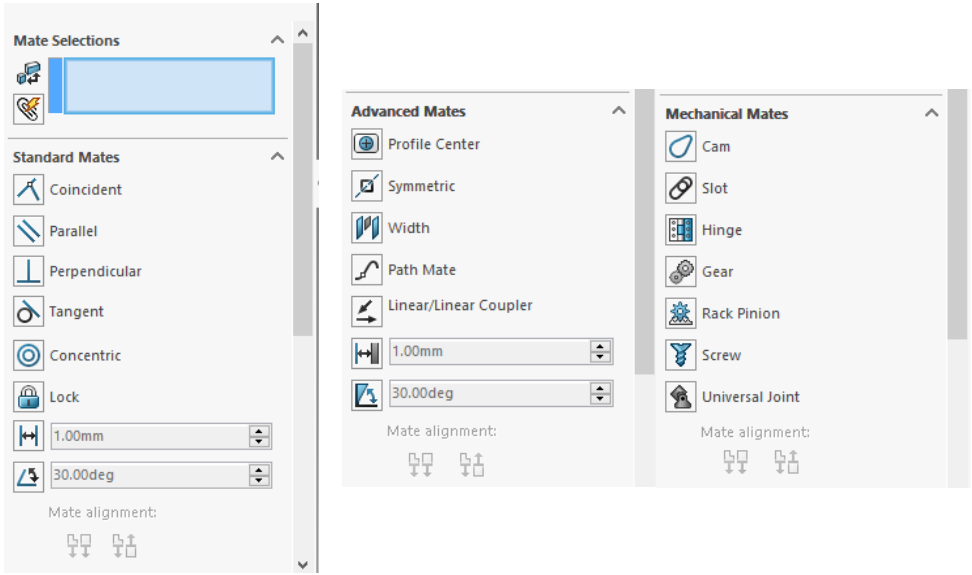
- Each “part” is a block
- We use “Mates” to push them together
- “Pins” for bearings/joints
- Can “play” with our design as we go along to make sure it all fits





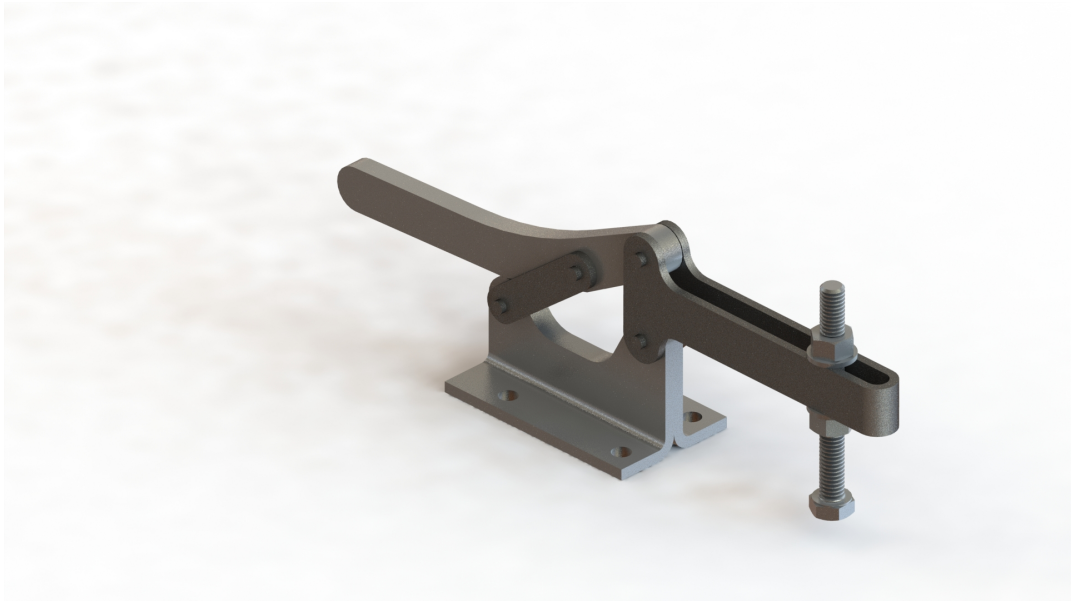
Mates

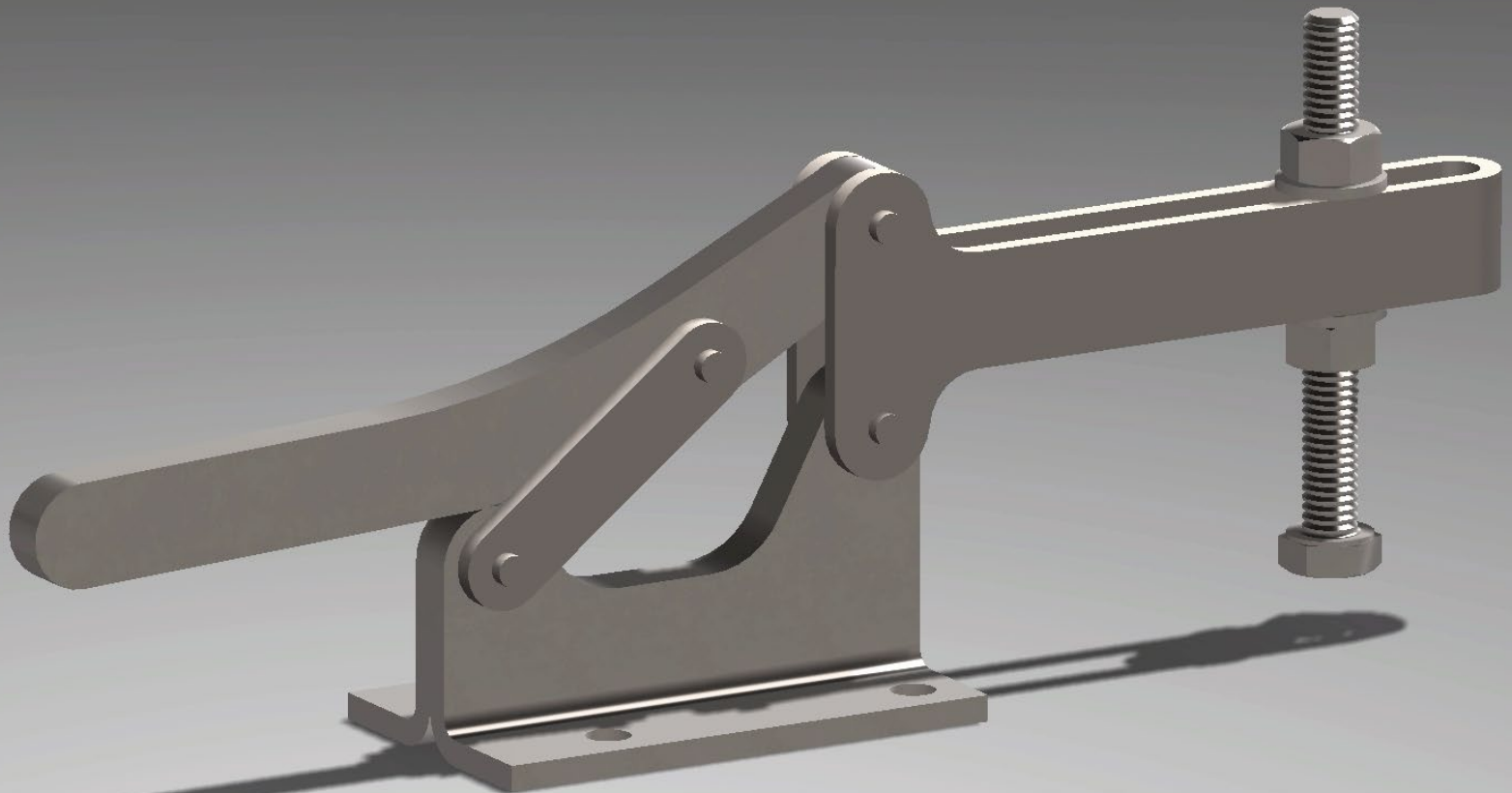
- These constrain the design, by placing hard, or range limits on placement of surfaces, edges or points.
- Can also be used to create mechanical constraints



What will we be making today?

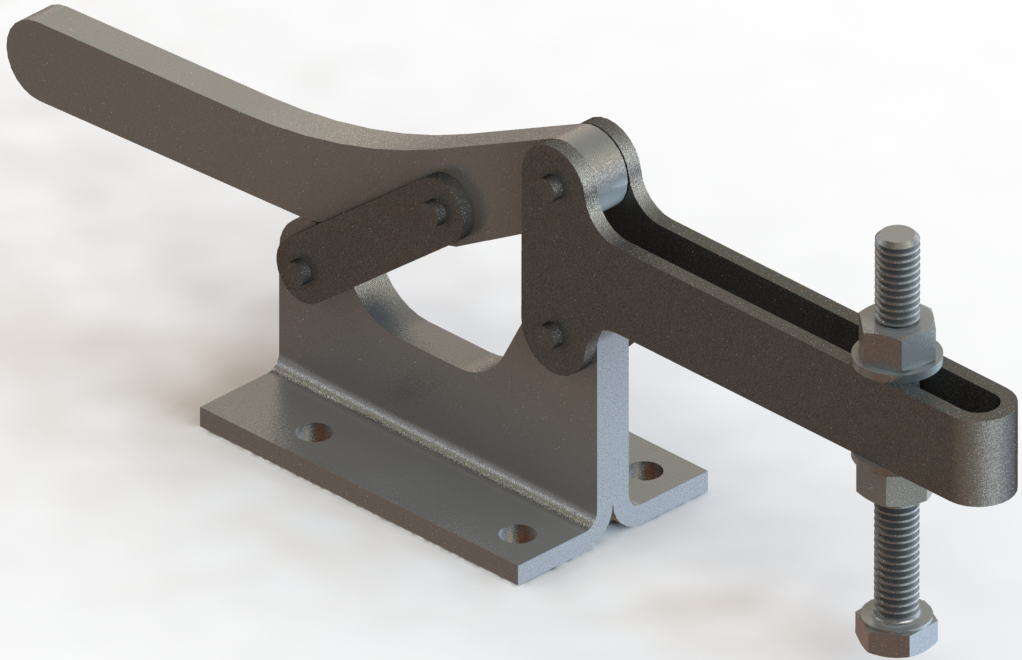
A clamp!





Open The Clamp Files

- Unzip the folder
- Open the “clamp_Unassembled.sldasm” file in the folder
- This has all the parts needed in it to build the clamp.
- We just need to assemble it all together!



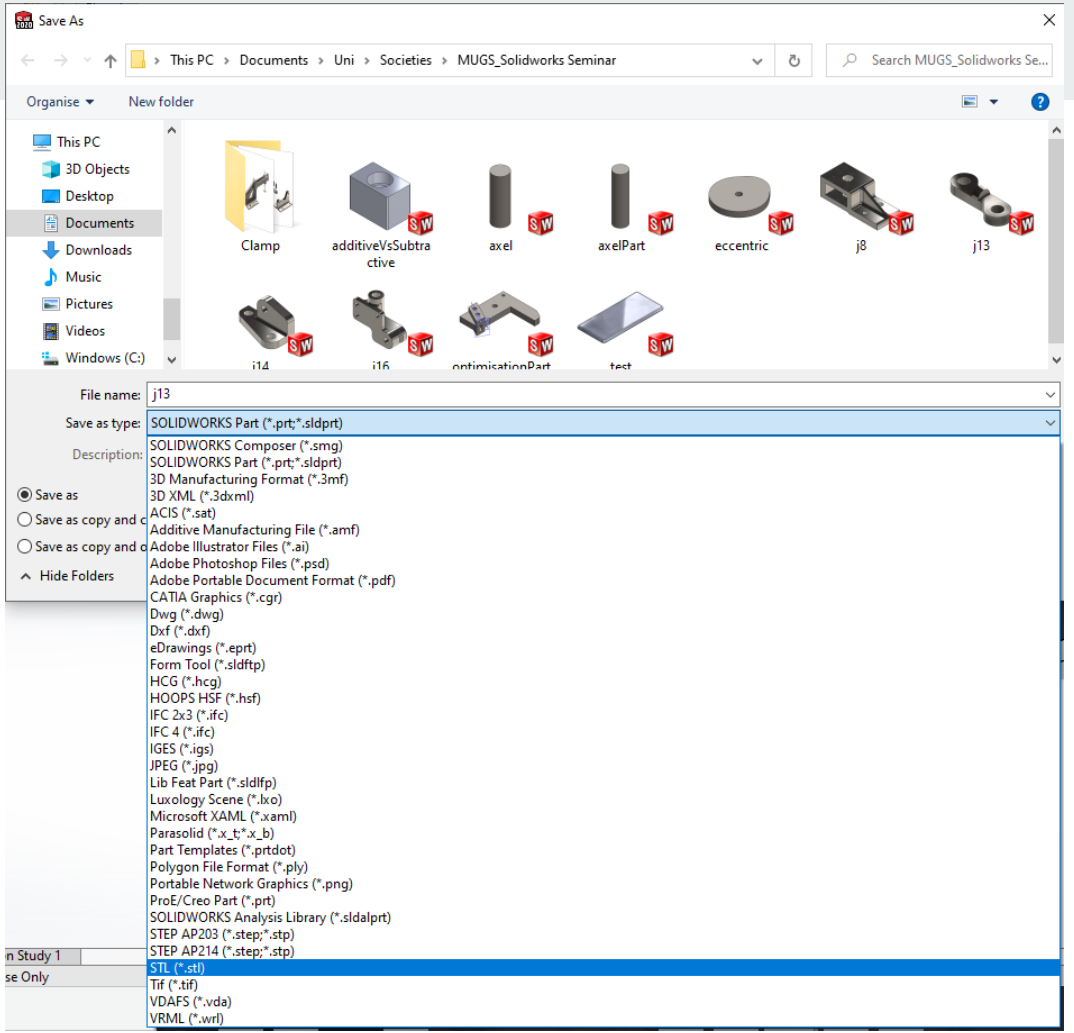
How do I manufacture a part?

We've designed a part…now what?

- Solidworks stores info in their own format
- Somesoftware can natively extract the information, others cannot.
- We'll be using 3D printing as an example.
- There are a couple of widely used formats, chief among them is the STL (stereolithography or standard triangle language depending on who you talk to.)

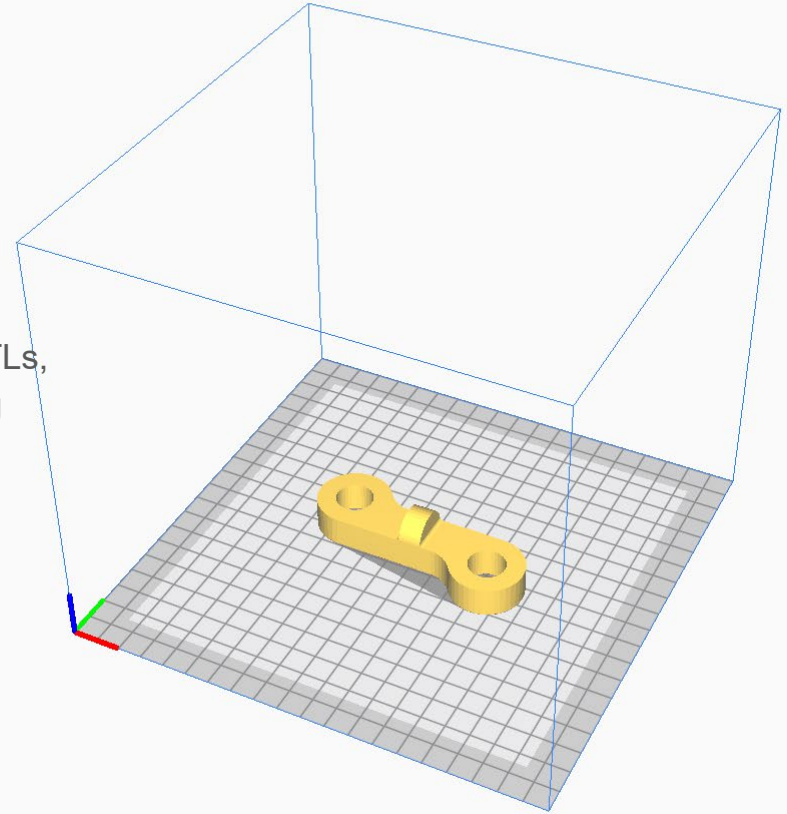
How to export for 3D Printing?

- Simple as saving as an STL!
- IGES is also common, and occasionally STEP.
- Less common but useful to know is PLY– used in computer graphics!



CURA

- Free software
- Drag and drop STLs, load printer config
- Slice and print!



Where to next?

Solidworks is good at what it does

- You were able to successfully transform 2D engineering drawings into 3D parts which could be sent for 3D printing or machining
    - To 3D print, just save your file as an STL!
- Knowing this much Solidworks will get you by, but will also open the doors to so many other things
    - Simulations (CFD, topology optimisation etc.)
    - Electrical routing and pipe work (complex to do in CAD!)
- Will have opened you eyes to a bit of design engineering work

A note on design engineering

- You don’t have to be a design engineer, but you should be able to communicate designs
    - Communication is key in engineering, and we communicate complex ideas visually through drawings
    - The best engineers can draw to a standard which everyone can understand, whether they’re other engineers, workman or the public.

- You never know when you’ll need it, and I guarantee you will at some stage in your career! So keep learning!

Some suggestions

- If you think you're going to use Solidworks at least once in a blue moon, I recommend the following:
  - Dual monitors
    - Allows you to have a drawing/part on one screen, and a part/assembly on the other
  - A good mouse (invaluable)
    - Can bind dimensioning tool to side buttons, easy too zoom and scroll – drastically improves workflow
  - A set of calipers and a micrometer
    - Can size things up, take measurements accurately from physical objects

Q&A

Jack Naylor
jack.naylor@sydney.edu.au