TEXAS A&M UNIVERSITY
ROBOTICS TEAM & LEADERSHIP EXPERIENCE

SolidWorks (CAD) Foundation

TURTLE Hatchling

Schedule

Week 1: Introduction
Week 2: SolidWorks (CAD) Foundation
Week 3: SolidWorks 3D
Week 4: Tools, Project, and Process
Week 5: Design Review and C++
Week 6: SolidWorks Assembly
Week 7: Programming and Git/GitHub
Week 8: Electronics and Soldering
Week 9: Prototype Week
Week 10: Build Week

Attend at least one meeting a week. If there is a conflict, let an officer know.

Information will be the same across meetings, however, feel free to attend more than one. Slides are posted on the website.

Note: Weeks with project milestones are in orange

Disclaimer

We are cramming multiple 200, 300, and 400 level department courses and their prerequisite into 8 weeks of 2-hour lectures

Treat the slides as reference documents and don’t hesitate to ask any questions.

SolidWorks

Two options

Texas A&M students have free access to SolidWorks Student Premium through the Software Center. **Recommended**

https://software.tamu.edu/

DO NOT INSTALL IN ONEDRIVE

DO NOT INSTALL IN ONEDRIVE

The software can also be accessed through the Virtual Open Access Lab (VOAL) which can be used without downloading SolidWorks.

For Mac users [Omissa Horizon Clients]

and

People who are currently installing SolidWorks

https://voal.tamu.edu/

Note: SolidWorks is the recommended software for TURTLE Advanced Projects. Please try to utilize SolidWorks, however, other CAD platforms may be accommodated.

VOAL Instructions

- Virtual Windows Machine that works anywhere
- Connect through TAMU VOAL site.
  - Login: Same as Howdy
  - {Engineering Desktop} click “...” then “Launch from Client”
- Do **NOT** save files in “(C:)” (You will lose the files)
  - Should be restricted by default
- Save Locations
  - “NetID (H:)” (Access only available in VOAL)
  - “Network or External Drive” (Need to grant VOAL access to the specific drive/folder)

VOAL

Texas A&M University https://connect.voal.tamu.edu

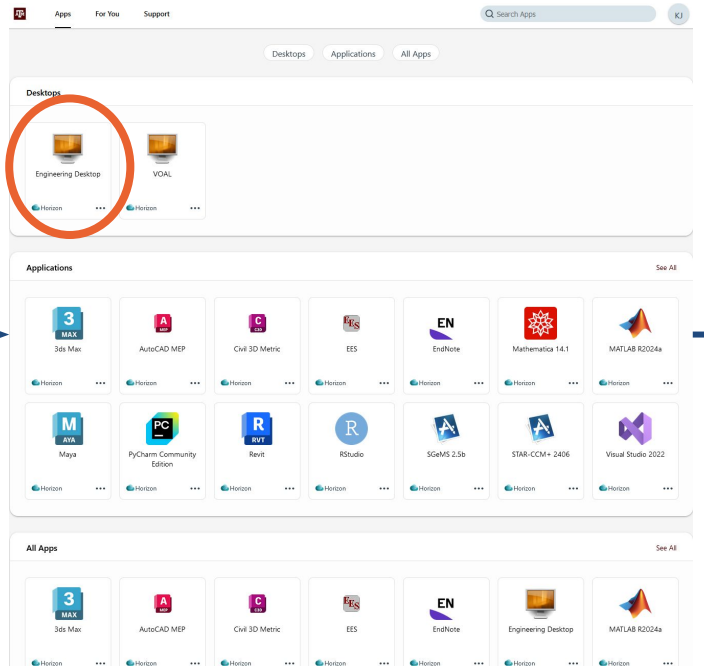
TAMU VOAL

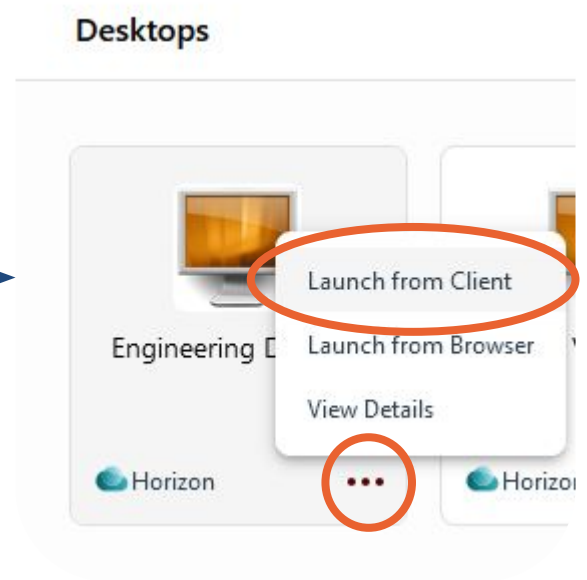
TAMU Virtual Open Access Labs. We recommend use of the Horizon Client for best performance.

Download VMware Horizon Client. Launch Native Client.

Missing: ~~tau~~ | Show results with: tau

Click the first time and download **Omnissa Horizon Clients**





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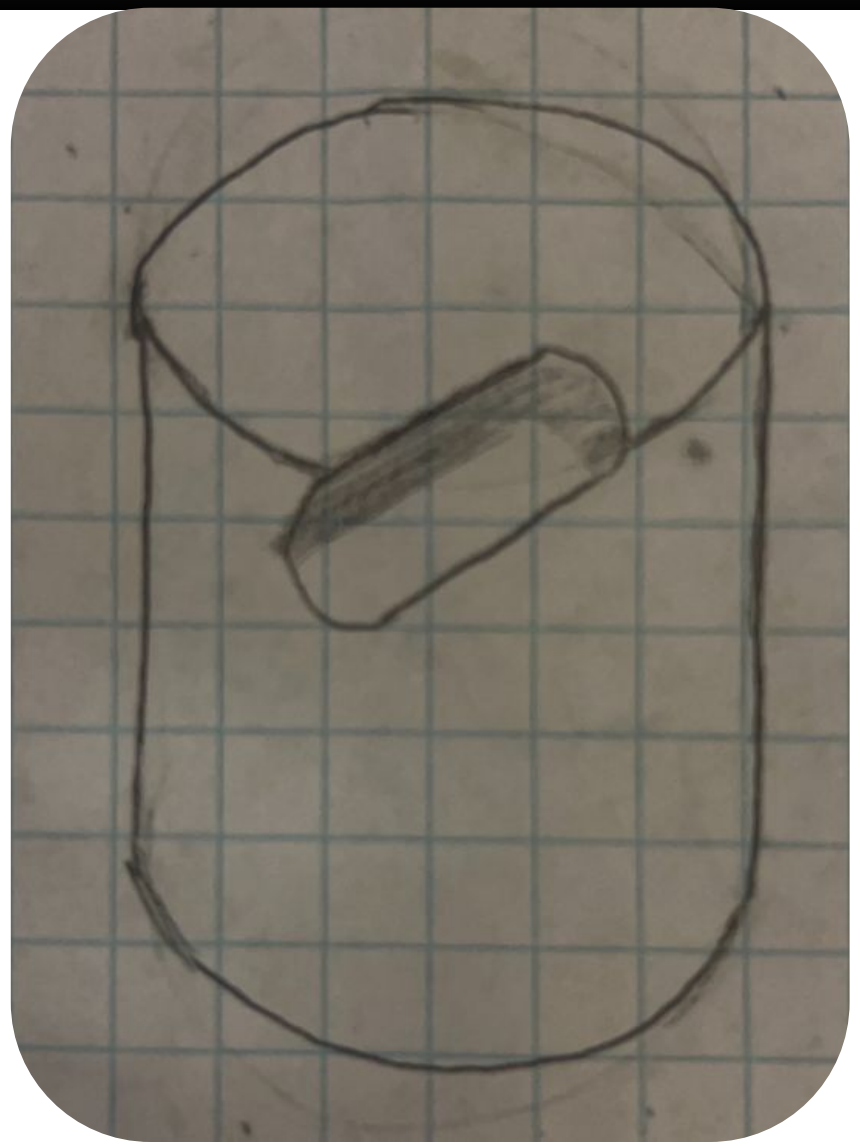
What is CAD?

CAD

C - Computer
A - Aided
D - Design

CAD programs are software **tools** that do the complicated tasks of defining geometry and storing the information in a commicatable format. **The user is still required to do the design.**

How can we define geometry?



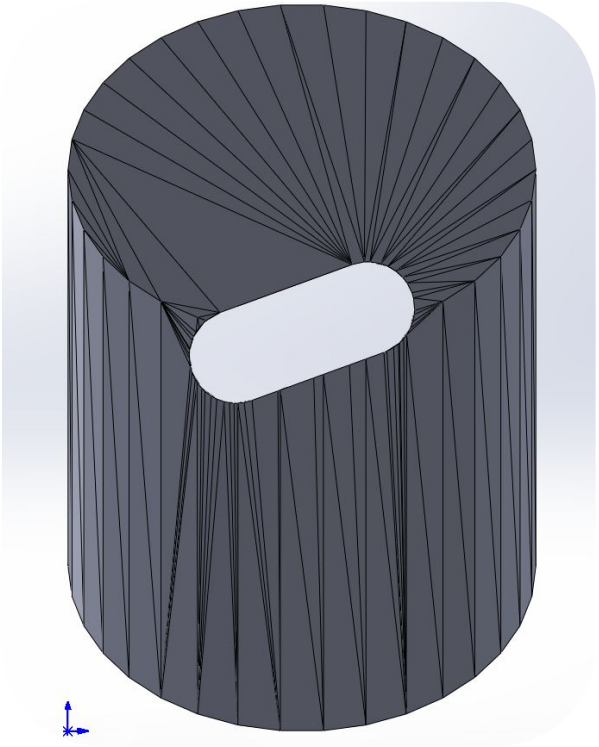
An easy method is providing a visual representation.

Without visual cues, it is a challenging task. Especially as the geometry becomes more complicated

Two common methods

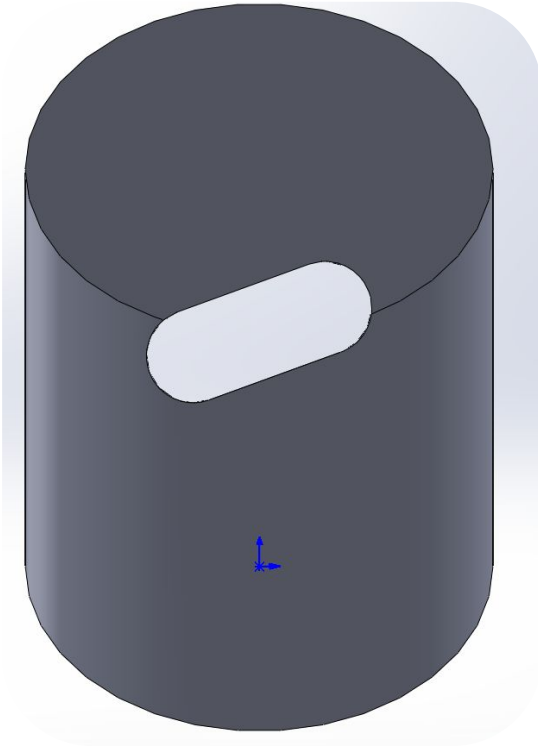
Direct Modeling

- Points define a mesh surface through polygons



Parametric Modeling

- Equation defined surfaces



CAD Softwares

- SolidWorks
- CATIA
- Blender
- Autodesk Fusion
- Autodesk Inventor
- Creo
- Onshape
- TinkerCAD

The tools and capabilities may change between softwares, but a common concept will improve your designs in all.

What is this common concept?

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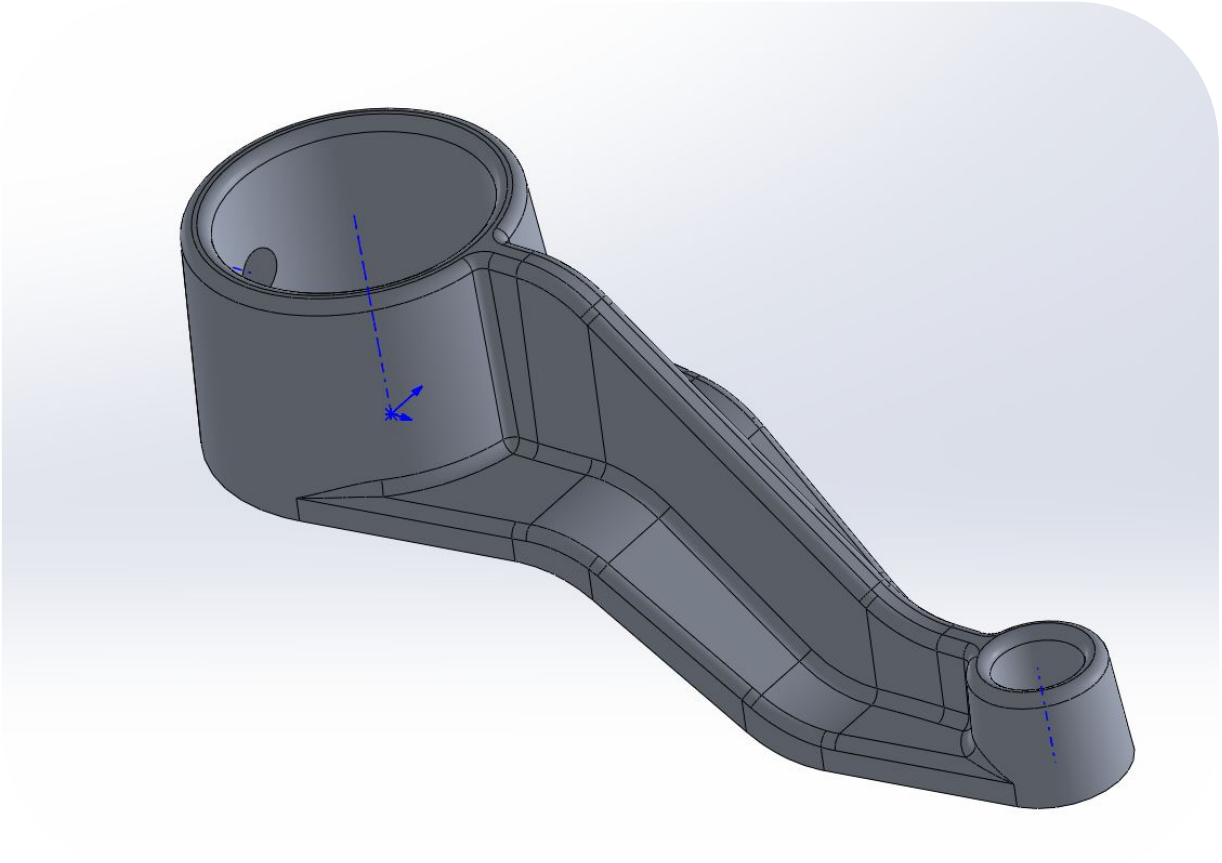
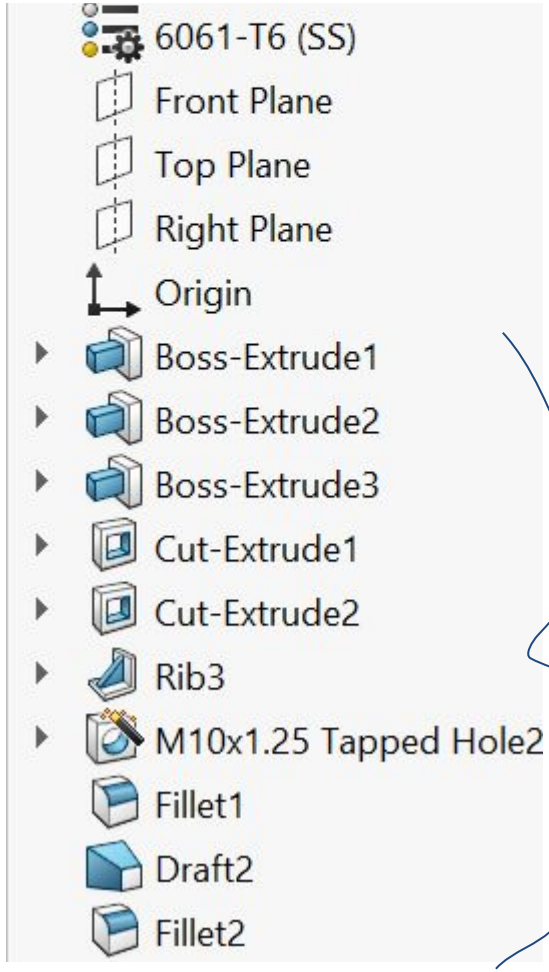
Design Intent

Designing robust parts that allow easy **modification** in the future.

Importance of Design Intent

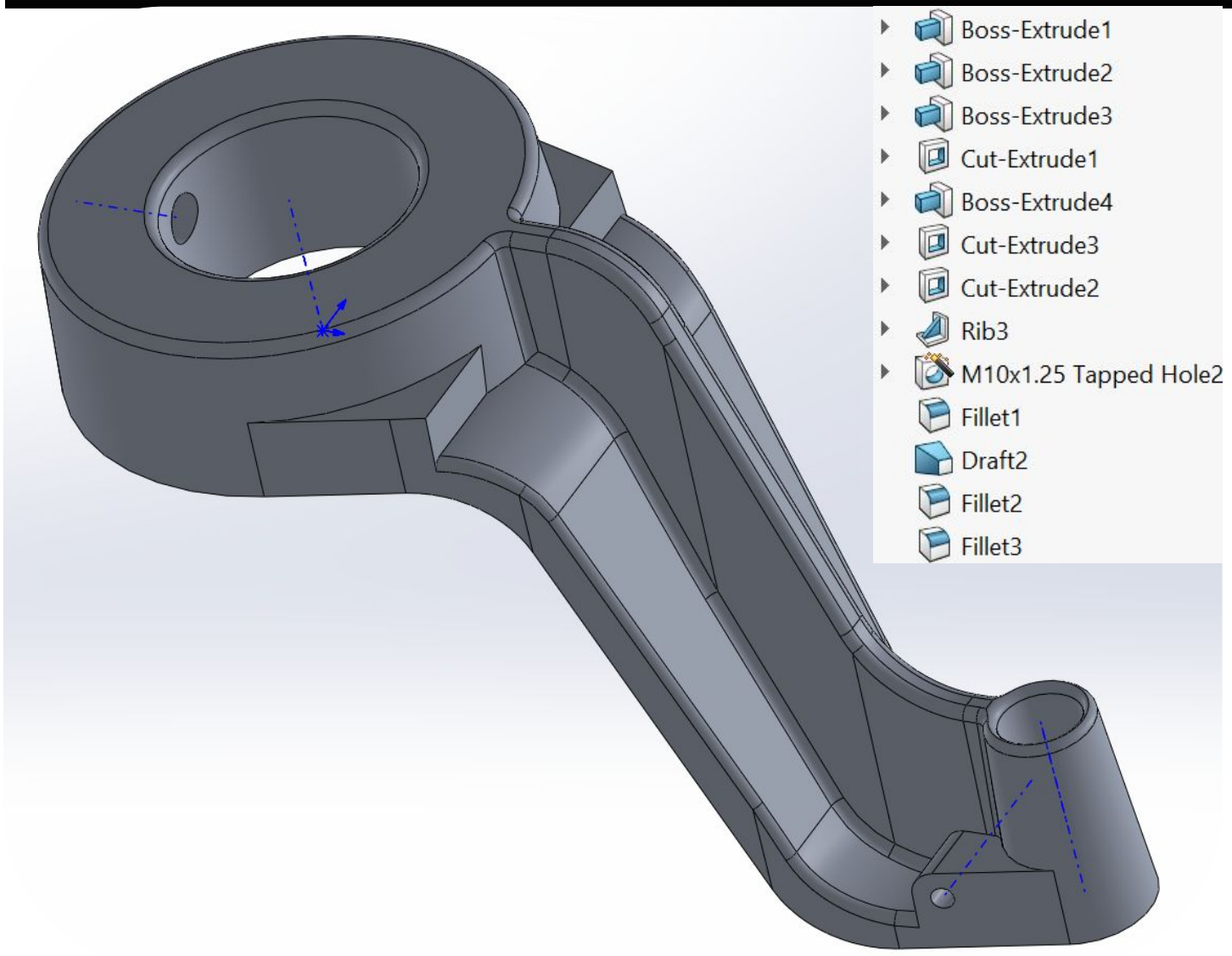
This is the model from the 2009 Model Mania





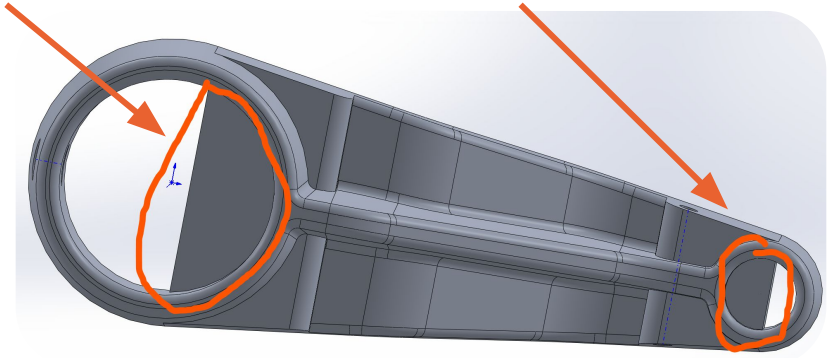
This is an ideal feature tree

Why?



You can make some rather silly changes quickly without destroying the model.

It also helps prevent unwanted geometry like this



Implementing Design Intent

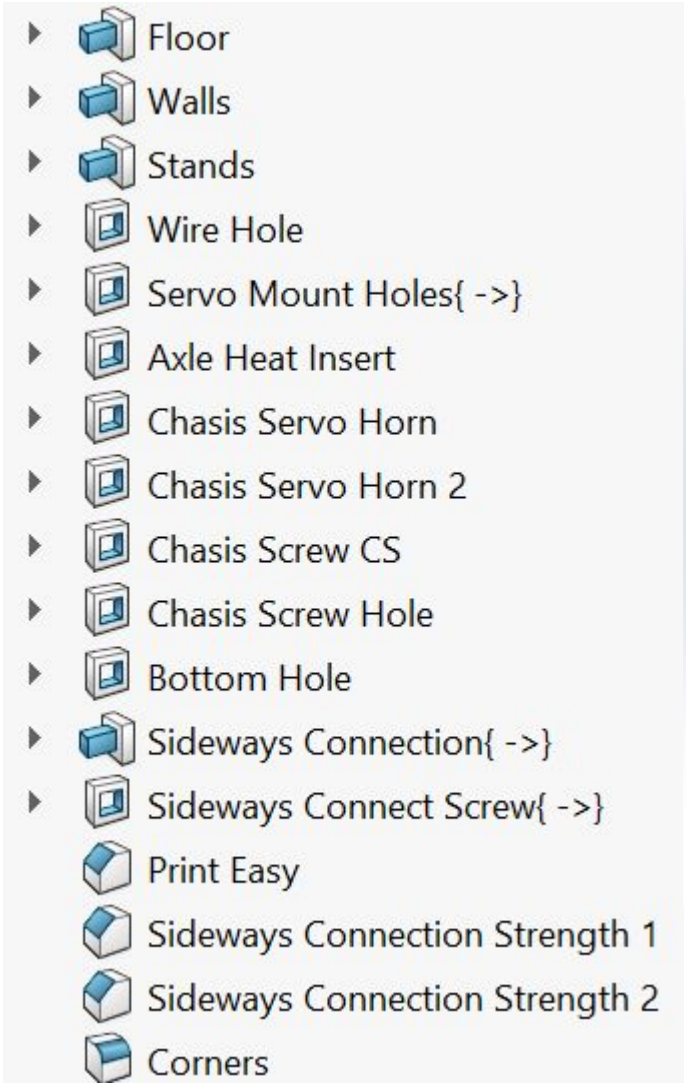
1. Feature Tree Management
    a. Group similar feature tools together
    b. Leave aesthetic finishing tools to the end

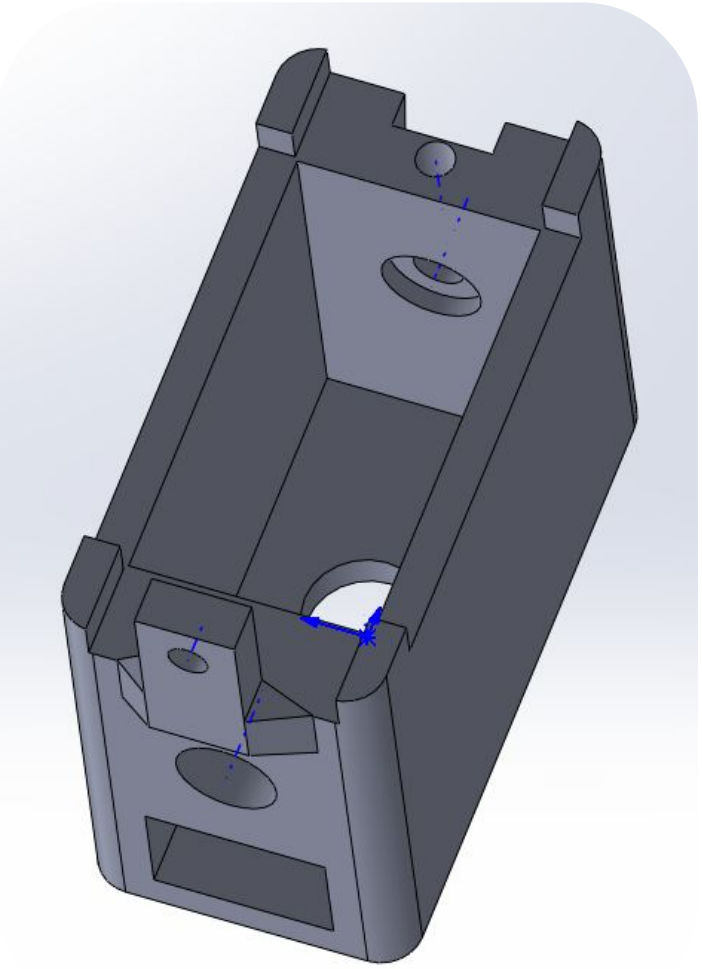
2. Dimensioning Structure
    b. Dimension the important tolerancing points

3. Design for Manufacturing
    c. Technically a different concept, but just as important

Resilient Modeling Strategy

1. Add Material
2. Subtract Material
3. Special Operations
   a. Shell, Patterns, Mirror
4. Finishing Operations
   b. Chamfer, fillets
5. Repeat Loops
   c. if necessary





Renaming features was a personal preference

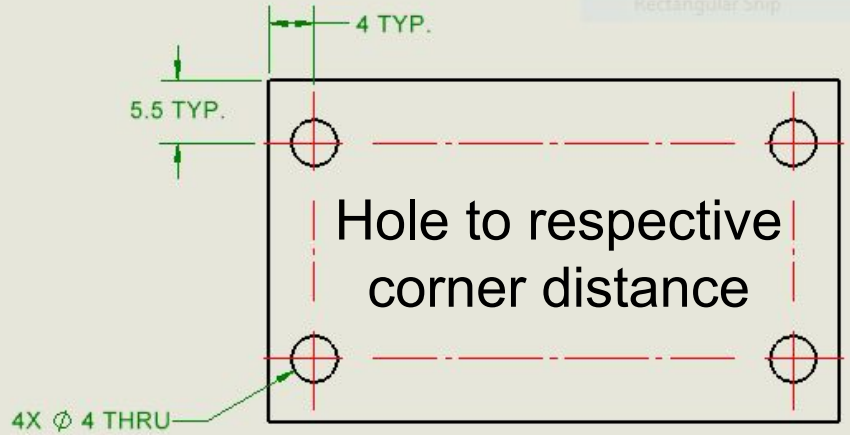
Dimensioning and Tolerance (GD&T)

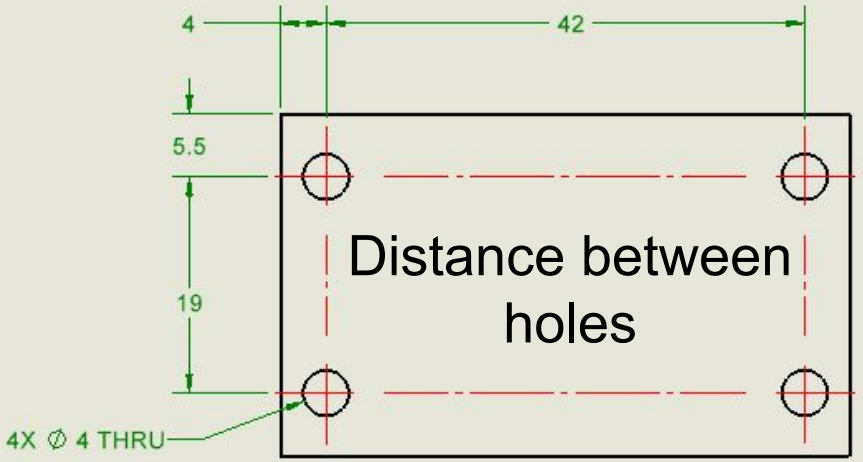
- Tolerance is the acceptable deviation from the designed measurement before the part is defective.
- A smaller tolerance range is achievable at a higher cost through more precise machines
- Tolerances are in relation to a reference and can stack together
- Sketch dimensions you define should be the important dimensions
    - Use the same dimension points as you would in a drawing

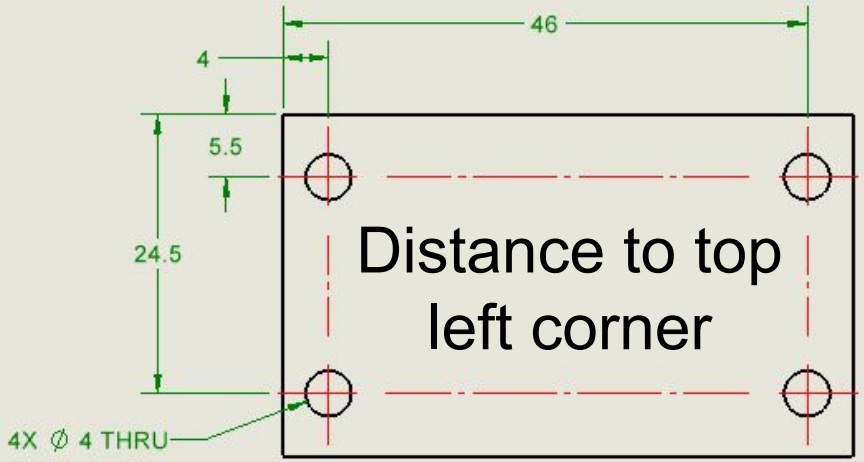
Dimensioning has Meaning

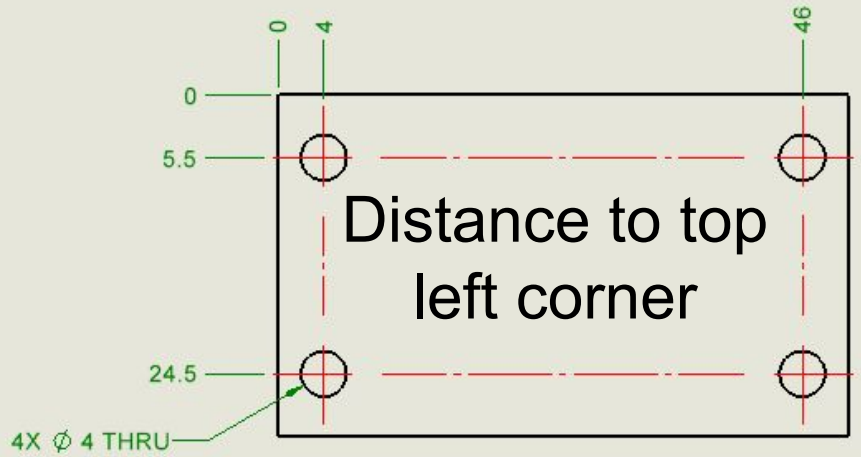
Note: The edges would have their own tolerance

Top right is typical for a screw pattern.



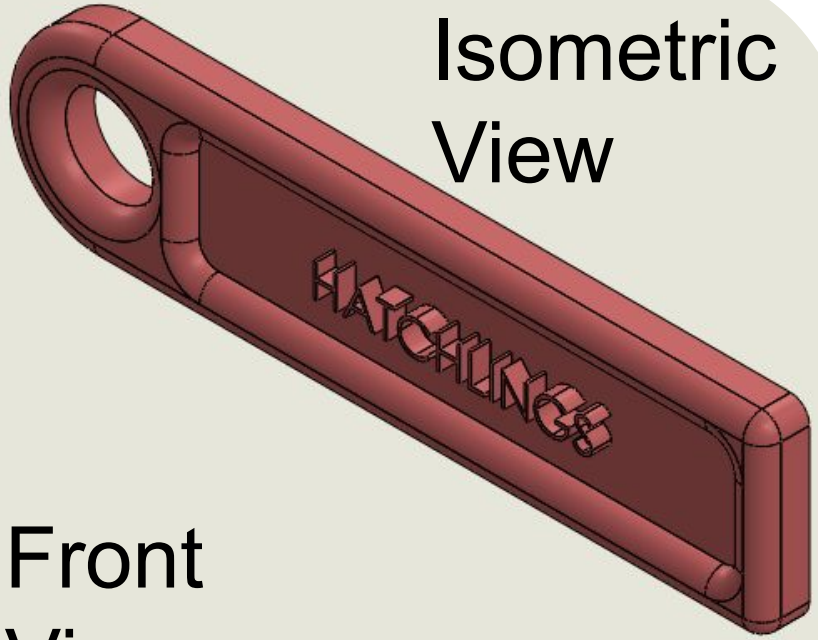




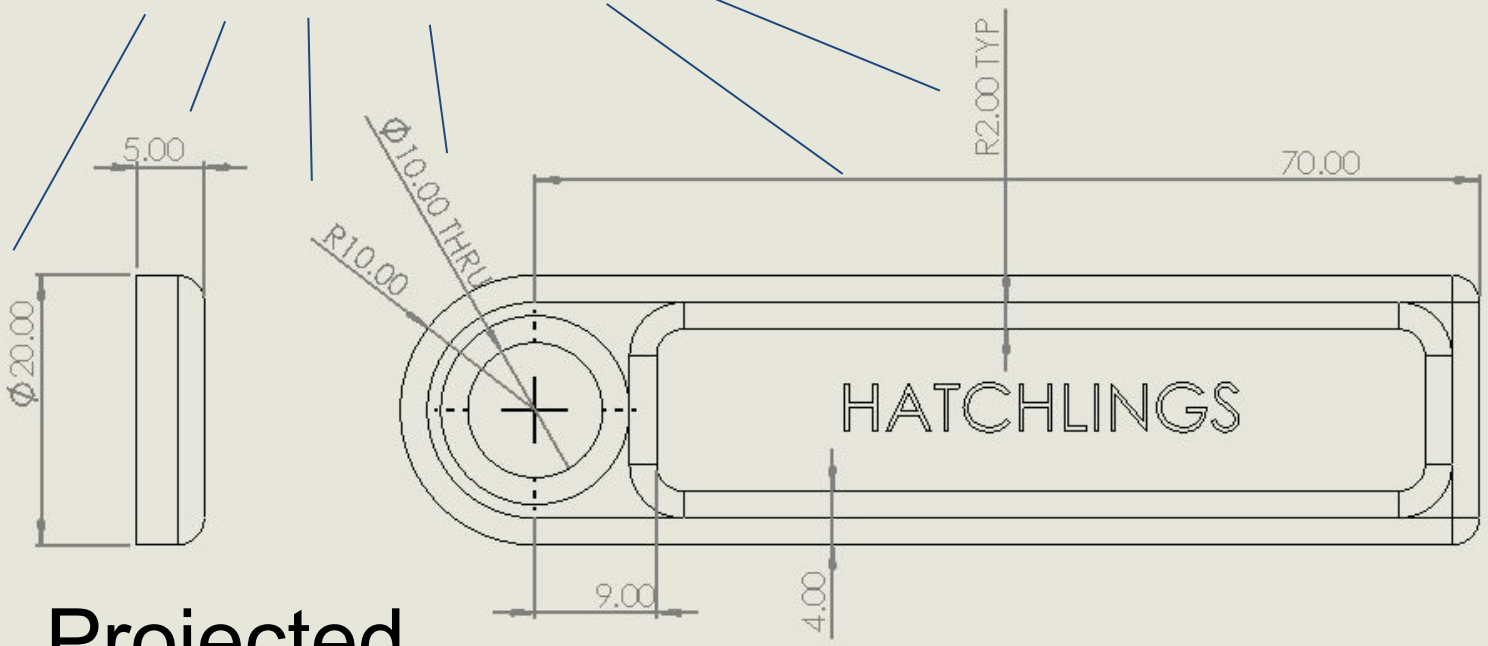


Reading a Drawing

This looks complicated. Just copy the Dimensions.

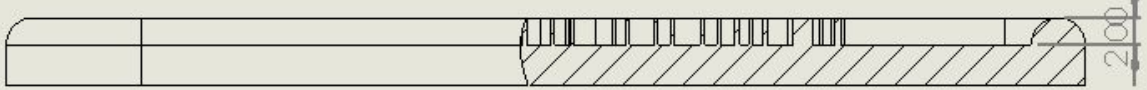


Isometric View



Front View

Projected Left



Projected Bottom

CSWA Callouts

TYP - Unless specified, the measurement is constant across the feature. (Every fillet on last slide is 2mm radius)

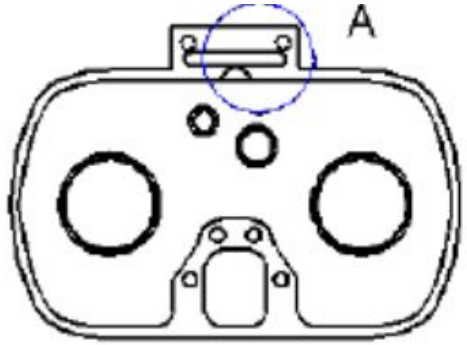
<#> X - States the same dimension is used # of times.

THRU - Hole goes all the way through the part

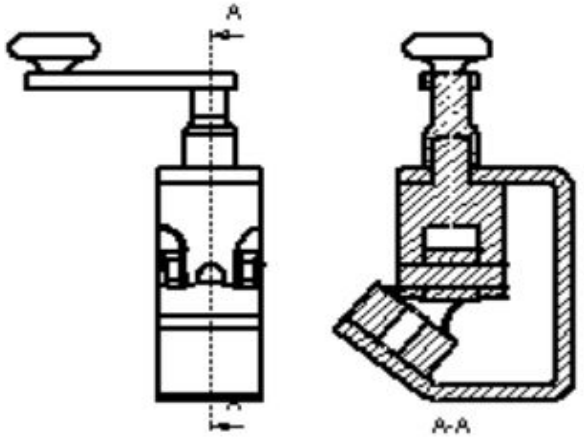
R <#> - Radius of #

∅ <#> - Diameter of #

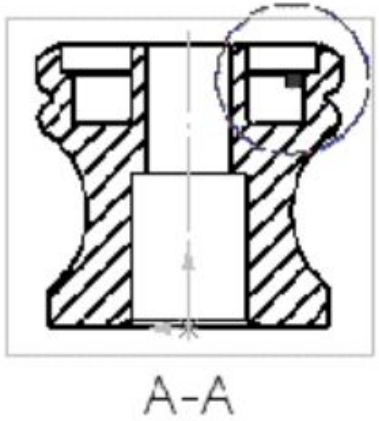
Drawing Views (CSWA Knowledge)

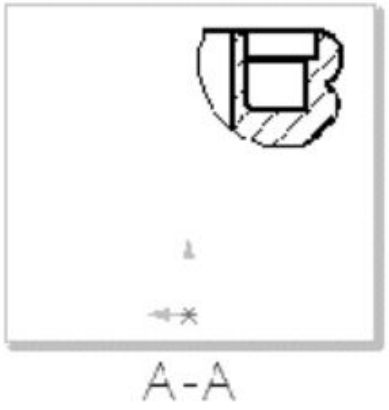


Detail

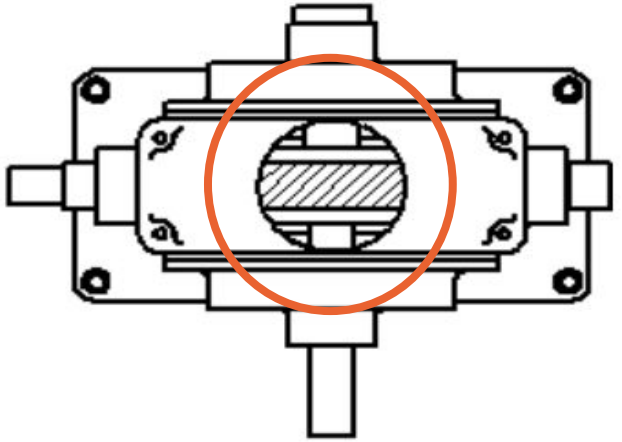


Section

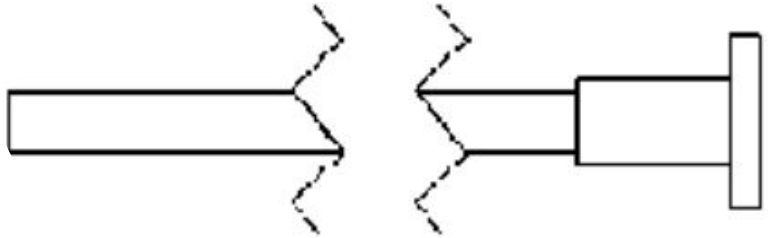




Crop



Broken-Out Section



Break

Taken directly from [SolidWorks]

Design for Manufacturing

This is unimportant until you’re designing custom parts

We will talk about DFM in Week 4 - Project and Process

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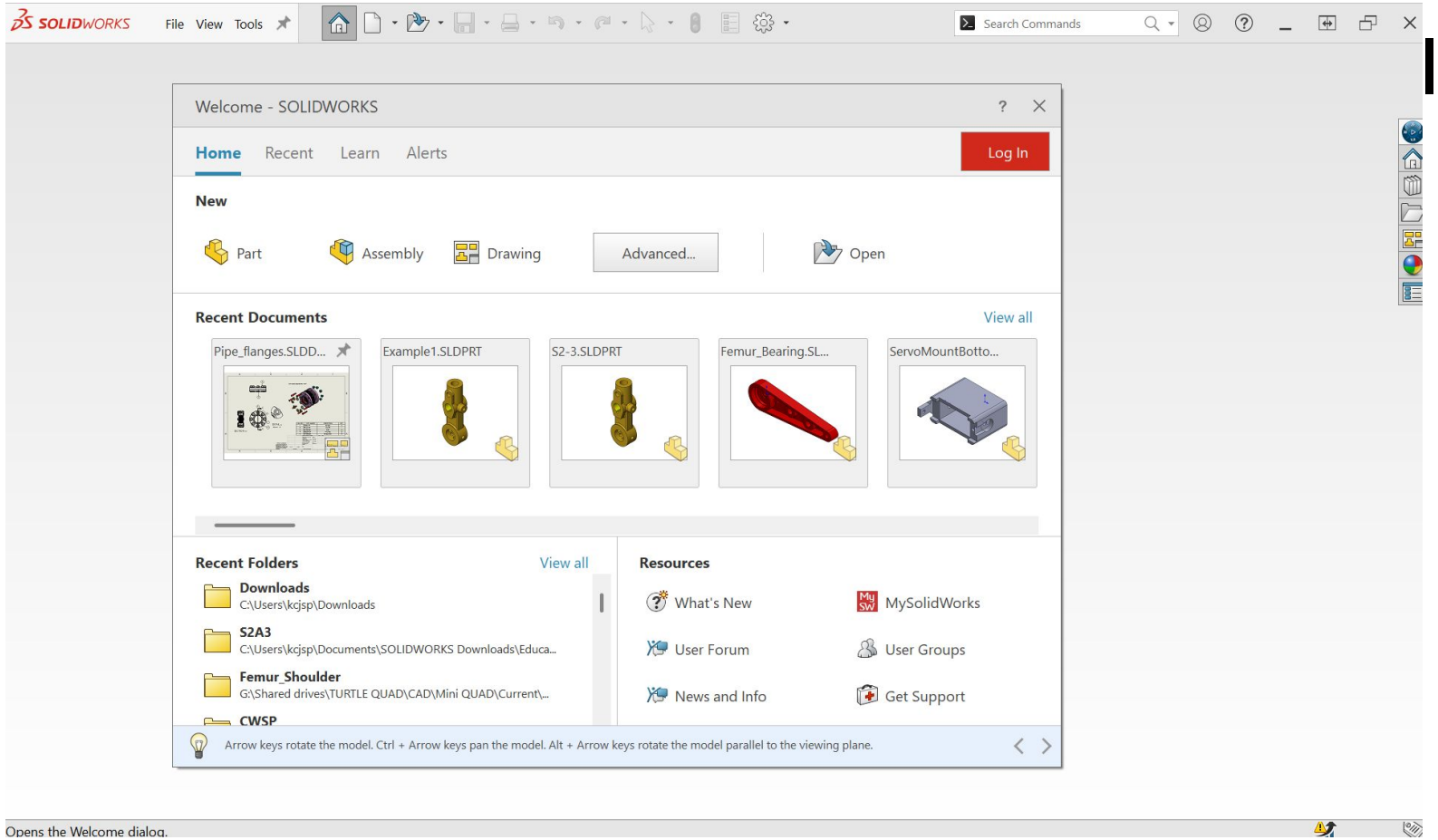
SolidWorks

What is SolidWorks?

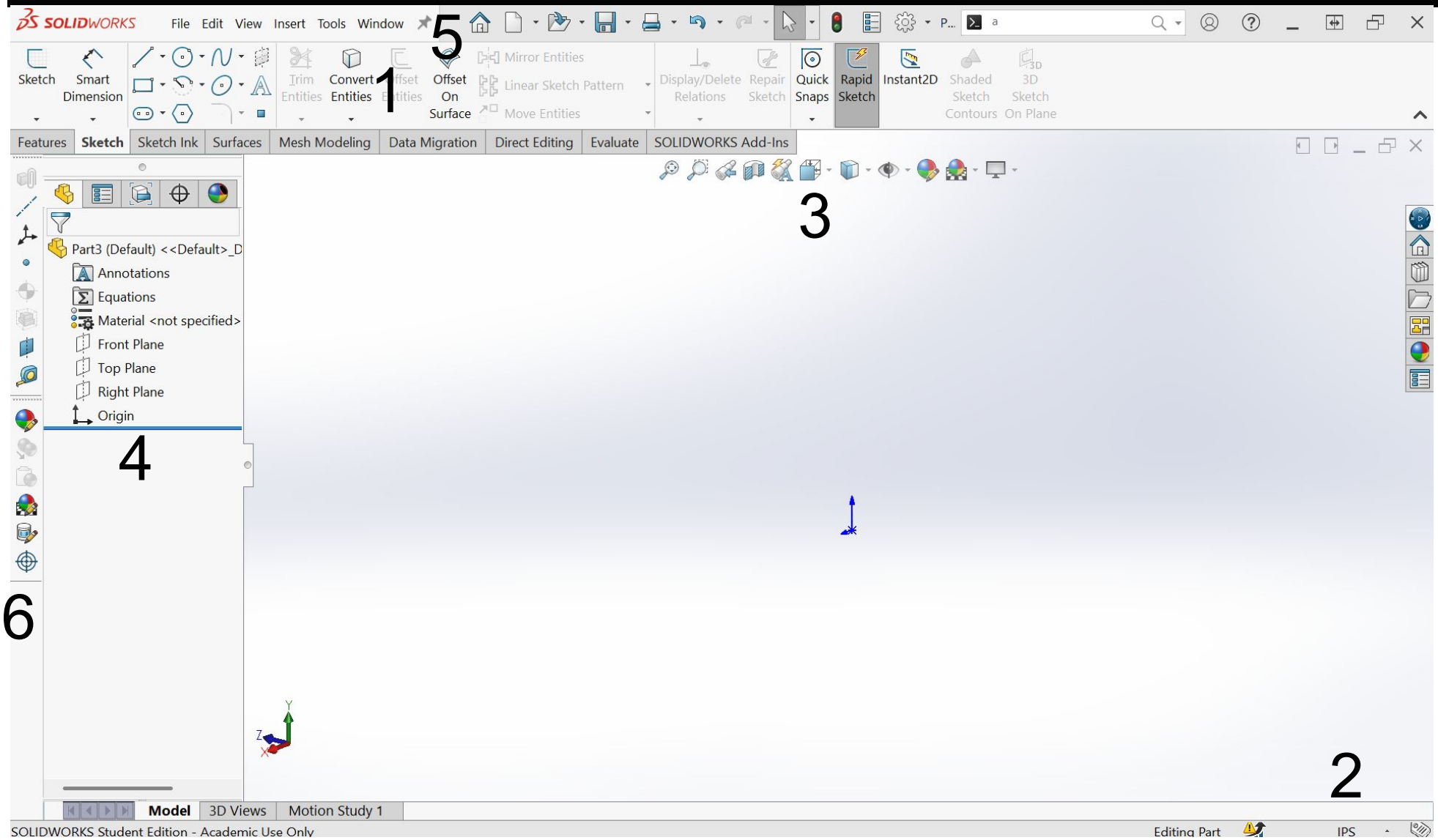
SolidWorks is a feature tree based parametric CAD Software.



Included Capabilities:

- Part Creation
- Assembly Creation
- GD&T Drawings
- Engineering Analysis
- So much more

SolidWorks Layout

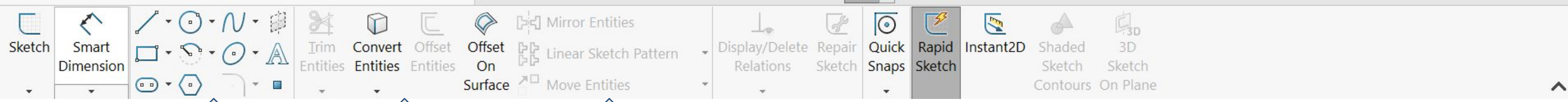


Next slide has the key

Key

1. Command Manager
   a. Holds the toolbars based on different grouping tabs.
2. Change the Units
3. View (Heads Up)
   a. Quick useful view changes
4. Feature Tree
   a. Tabs for other useful stuff. (Configurations, Appearances, Etc)
5. Standard Toolbar
   a. File, Settings, Rebuild and a lot more important stuff
6. Custom Toolbar

Sketch Tab



Drawing Tools

Specialty Drawing Tools

Patterns

- Use “Drawing Tools” to make a general shape.
- “Trim Entities” is like an eraser
- Convert Entities projects selected geometry onto the plane
- Offset Entities does as it says
- Try not to use sketch patterns or fillets/chamfers

The Basic CAD Journey

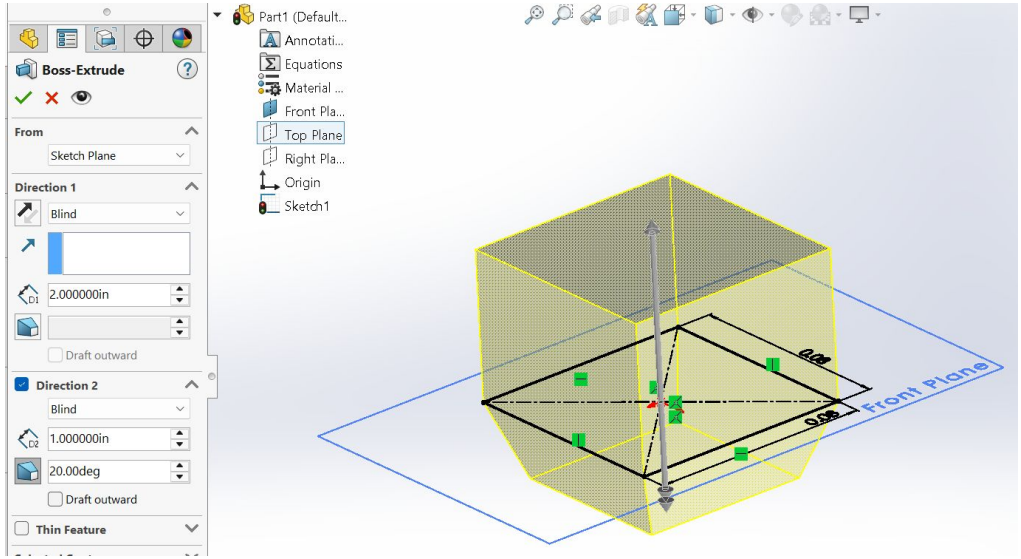
1. Create/Select a Plane (A face is a plane)

2. Create a Sketch
   a. Draw general shape w.r.t the origin
   b. Add relations
   c. Dimension the rest from largest to smallest

3. Turn Sketch into a 3D Feature

4. Save

5. Repeat



Navigation in SolidWorks

Scroll Wheel : Zooms to cursor

Hold Middle Mouse Button : Rotates about the object

“Ctrl + Hold MMB” : Pans camera on the viewing plane

“Alt + Hold MMB” : Rolls on the viewing plane

Planes

Planes are flat surfaces with zero thickness and zero curvature. Includes flat faces on objects

SolidWorks starts with the 3 Datum Planes:

- Front
- Right
- Top

Planes are required to make a sketch

Want a Unique Reference Plane?

It is often advantageous to use a custom reference plane when designing parts. You can do this through the “Reference Geometry” tool.

A plane can be uniquely defined by:

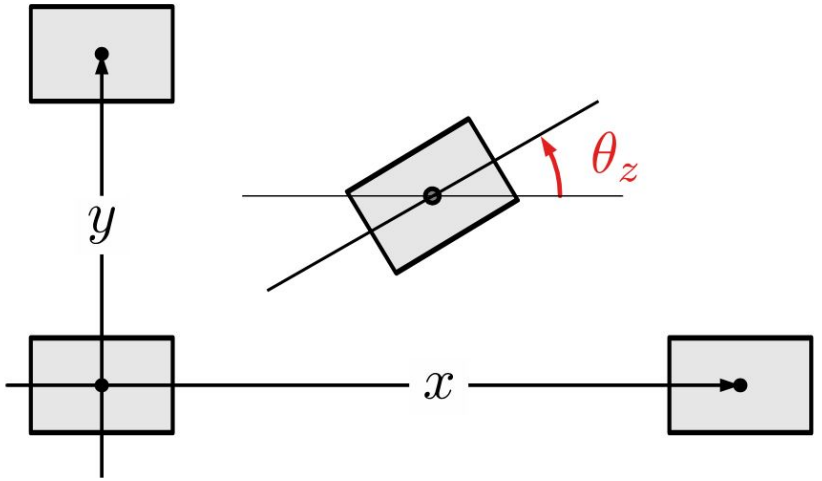
- Three non-collinear points
- A point and a line not on that point
- Two distinct intersecting lines
- Two separate parallel lines
- A flat face

Sketches

Sketches are 2D “drawings”.

A plane has 3 Degrees of Freedom:

- X (Translational)
- Y (Translational)
- θ(z) (Rotational)



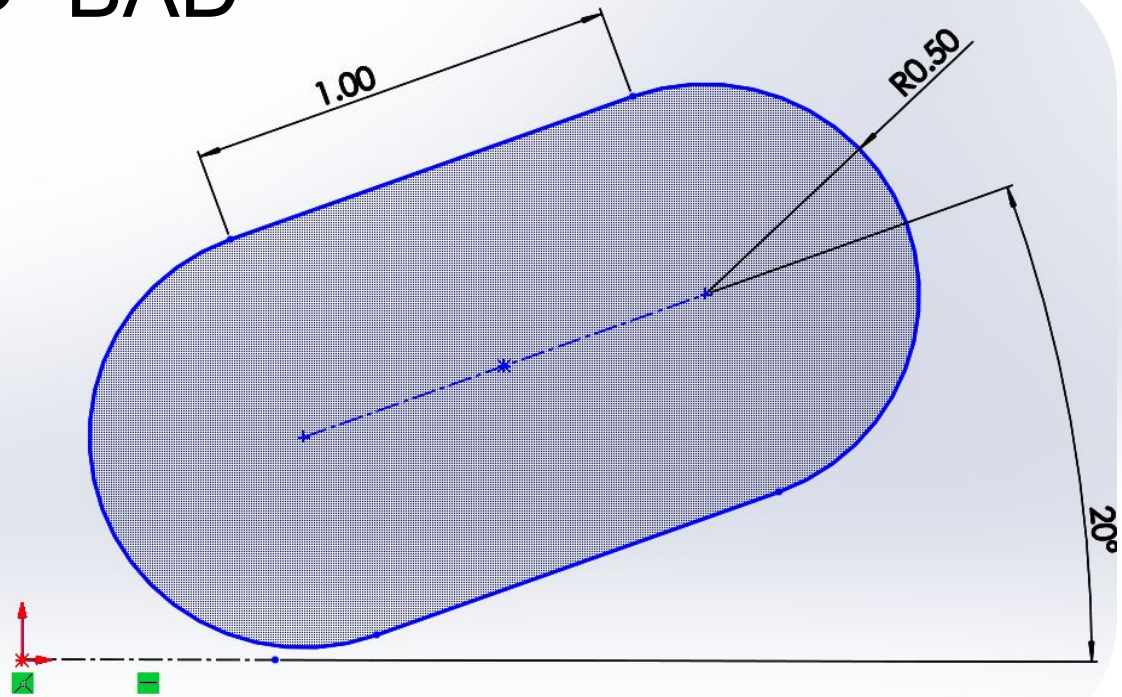
engineeringstatics.org

An ideal sketch will have one enclosed region and fully defined (cannot move) in space.

Is my sketch fully defined?

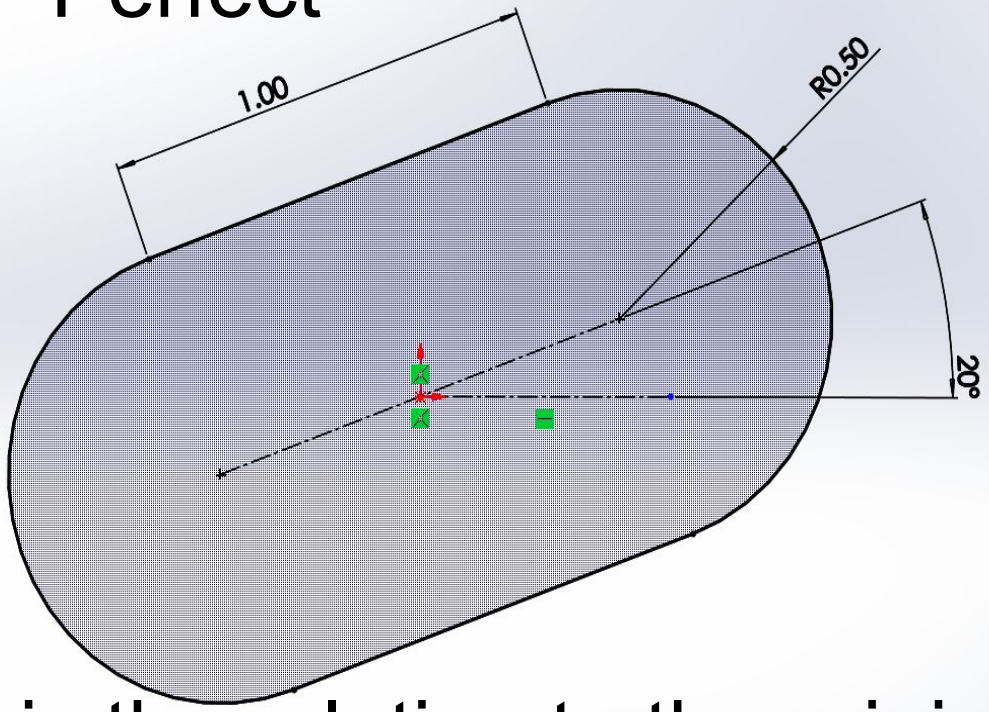
Under defined:

- Blue Lines
- Multiple Potential Solutions
- BAD



Fully defined:

- Black Lines
- One Solution
- Perfect

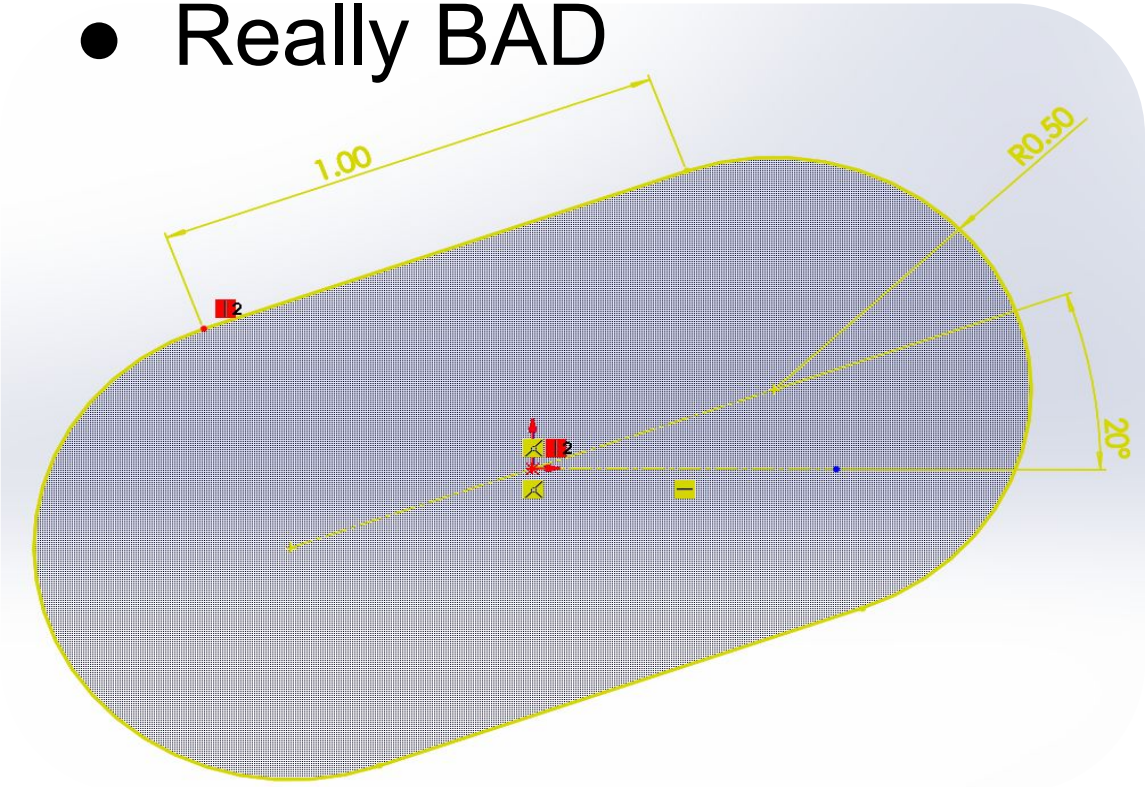


The difference between these is the relation to the origin.

Other Defined States

Over defined:

- Yellow/Red Lines
- Zero Potential Solutions
- Really BAD



Construction (Centerlines) lines:

- Reference Geometry indicated as a dashed line
- Has no direct impact on potential solutions
- Very useful for defining curves

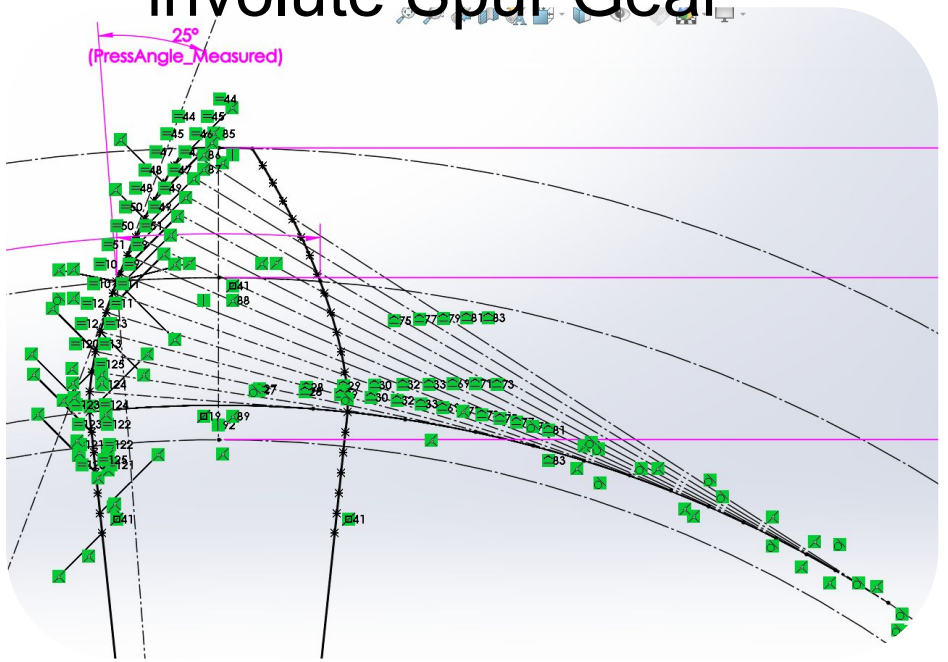
Sketch Relations

Geometric relationships that show up as little icons.

Benefits of relations over dimensions:

- Easier to read
- Reduces workload
- More robust to modification
- Shows purpose (Design Intent)
- Easier to define geometry

Example: A tooth of an involute Spur Gear



Common Sketch Relations

Most are self explanatory, these can be tricky to understand

Please **NEVER** use the “Fix” relation. 

Relation	Entities to select	Resulting relations	Icon
Collinear	Two or more lines.	The items lie on the same infinite line.	
Coincident	A point and a line, arc, or ellipse.	The point lies on the line, arc, or ellipse.	
Equal	Two or more lines or two or more arcs.	The line lengths or radii remain equal.	
Concentric	Two or more arcs, or a point and an arc.	The arcs share the same center point.	
Coradial	Two or more arcs.	The items share the same center point and radius.	
Merge	Two sketch points or endpoints.	The two points are merged into a single point.	
Pierce	A sketch point and an axis, edge, line, or spline.	The sketch point is coincident to where the axis, edge, or curve pierces the sketch plane. The pierce relation is used in sweeps with guide curves.	

Note: Standard assembly mates are nearly identical

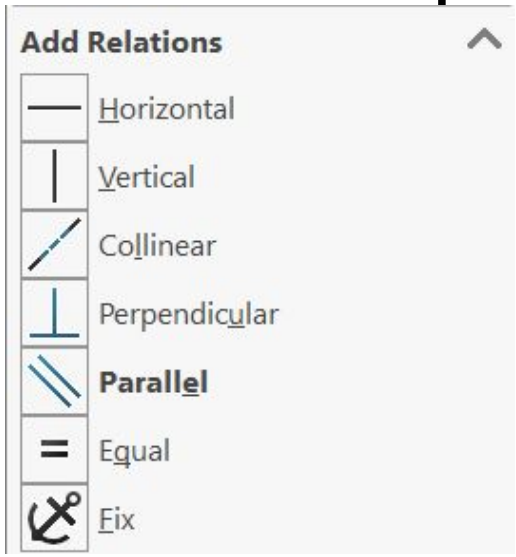
All relations from/on SolidWorks Help

Creating Relationships Methods

- Drag and drop
  1. Click and hold a point
  2. Drag until relationship shows up
  3. Release click

Note: Only works for simple relationships

- Preselect
  1. Control click entities
  2. Select intended relationship



- "Add Relation" tool
  1. Click the down arrow on "Display/Delete Relations" tool within "Sketch" tab
  2. Click "Add relation"
  3. Click entities
  4. Select intended relationship

Helpful Shortcuts

****You can customize shortcuts for your workflow****

Hold Right Click : Customizable Quick Toolbar
“q” : Shows all planes
“f” : Orients view to fit the model
“Shift + c” : Fully Minimizes Feature Tree
Feature Tree Search “*” : Fully Expands Feature Tree
“Spacebar” : Allows Viewing Orientation Selection
“a” : Switches type within a drawing tool.

Example Custom Shortcuts

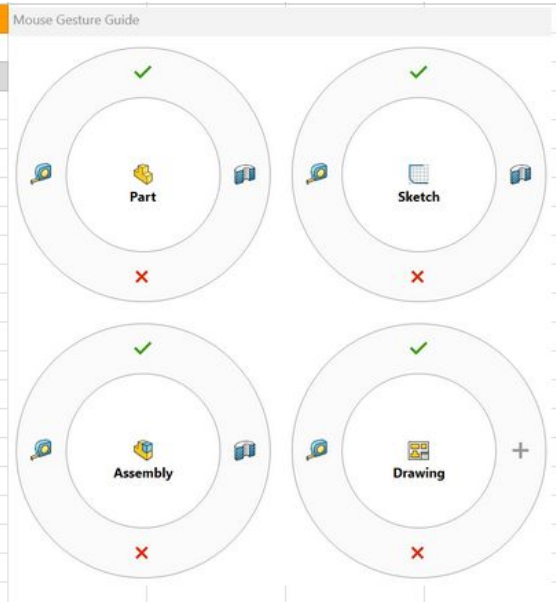
May increase CAD speed

Use a layout that you will remember and find comfortable

	Tab	q	w	e	r	t	y	u	i	o	p
Regular	Hide Component	Show Refrences	Line	Circle	Create Reference Plane	Arc	Sketch		Search Files		
Ctrl		Make Coincident	Make Collinear	Make Equal	Create Referece Axis	Flat Tree Toggle	Redo				Print
Shift	Show Component	Make Horizontal	Make Vertical	Extrude	Revolve						
Alt				Extruded Cut	Revolved Cut	Design Table				Open	
Ctrl+Shift	Show All Components										
Ctrl+Alt											

	a	s	d	f	g	h	j	k	l
Regular	Command Option Toggle	Rectangle	Smart Dimension	Fillet	Chamfer	Hole Wizard			
Ctrl	Select All	Save	Make Concentric	Merge Points	Make Tangent				
Shift	Mate	Sweep	Loft						
Alt	Show Temporary Axes	Sweeped Cut	Lofted Cut	Zoom to Fit					
Ctrl+Shift		Save All							
Ctrl+Alt		Save As							

	z	x	c	v	b	n	m		Enter	Repeat Command
Regular	Trim	Make Construction Geometry	Construction Line	Midpoint Line		Next Edge	Measure		Spacebar	Orientation
Ctrl	Undo	Cut	Copy	Paste	Rebuild					
Shift			Collapse Tree		Rebuild All					
Alt					Force Rebuild	New				
Ctrl+Shift			Copy Appearance	Paste Appearance						
Ctrl+Alt										

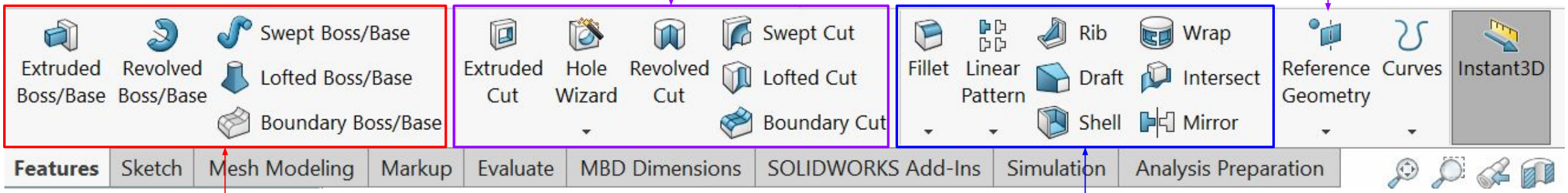


Features Tab

More on this next week.

Cutting Tools for removing volume

Extremely Powerful



Boss/Base tools for adding volume

Notice: Cutting and Boss/Base tools are inverses

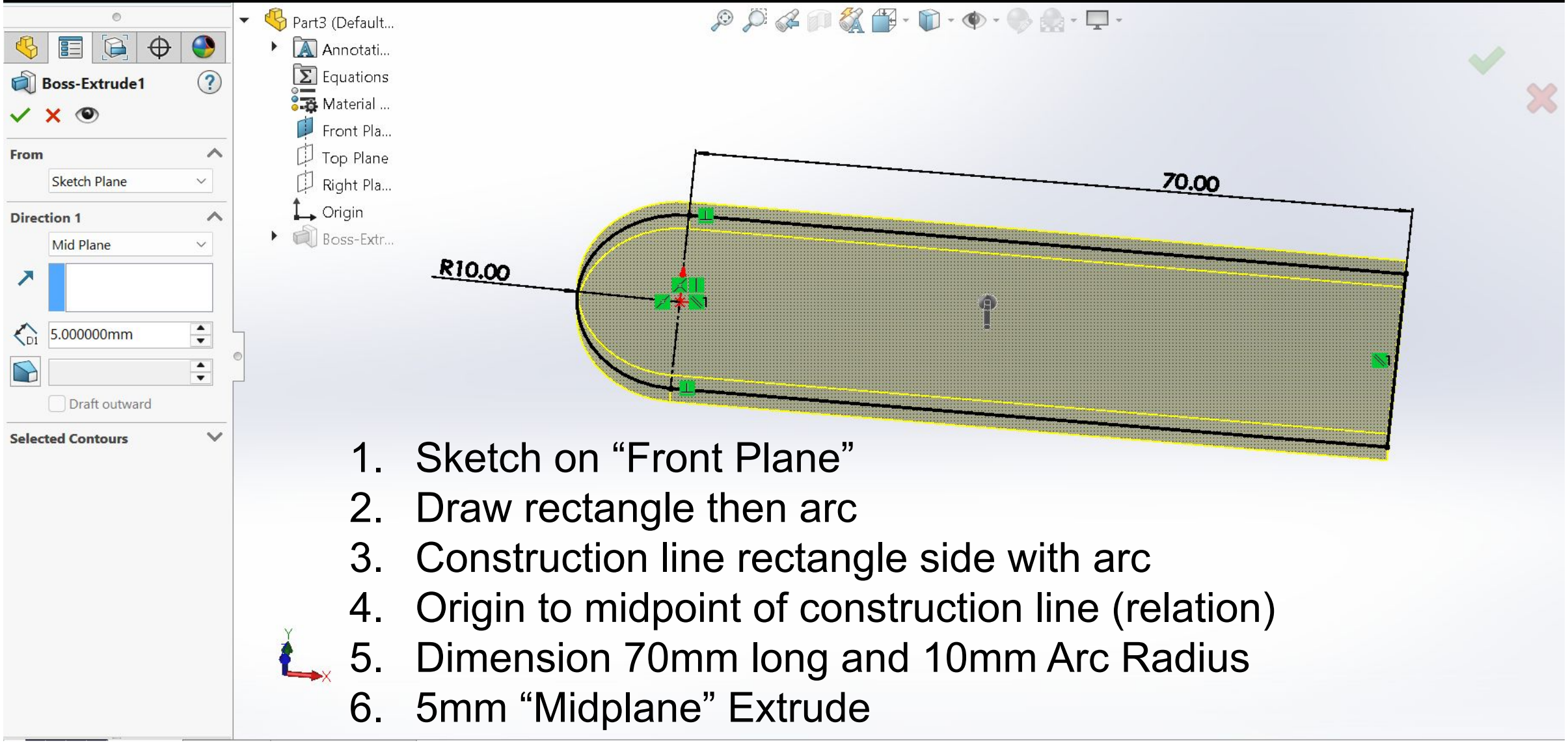
Speciality Tools

Making a name tag

Note:

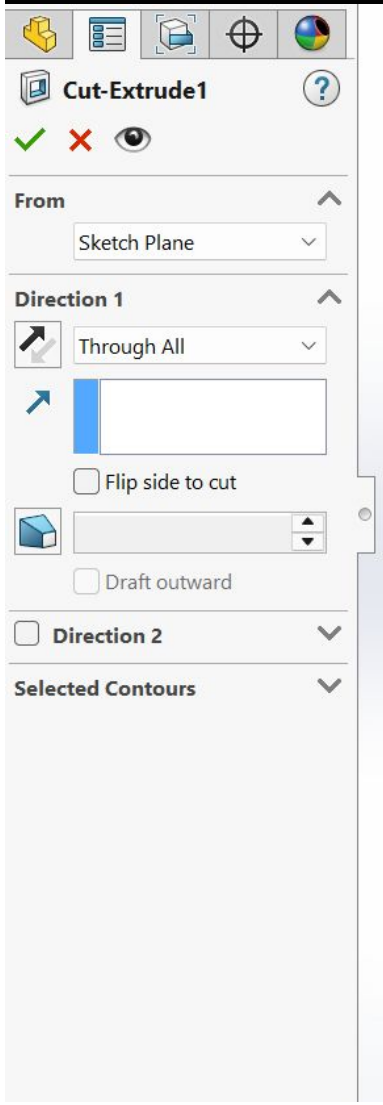
Text is one of the few things that can be under defined.

Feature #1 (Boss/Base)



1. Sketch on “Front Plane”
2. Draw rectangle then arc
3. Construction line rectangle side with arc
4. Origin to midpoint of construction line (relation)
5. Dimension 70mm long and 10mm Arc Radius
6. 5mm “Midplane” Extrude

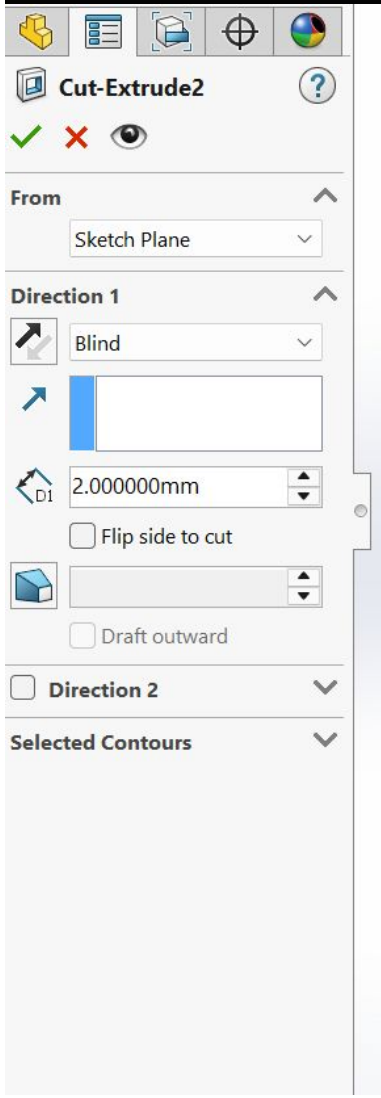
Feature #2 (Extruded Cut)

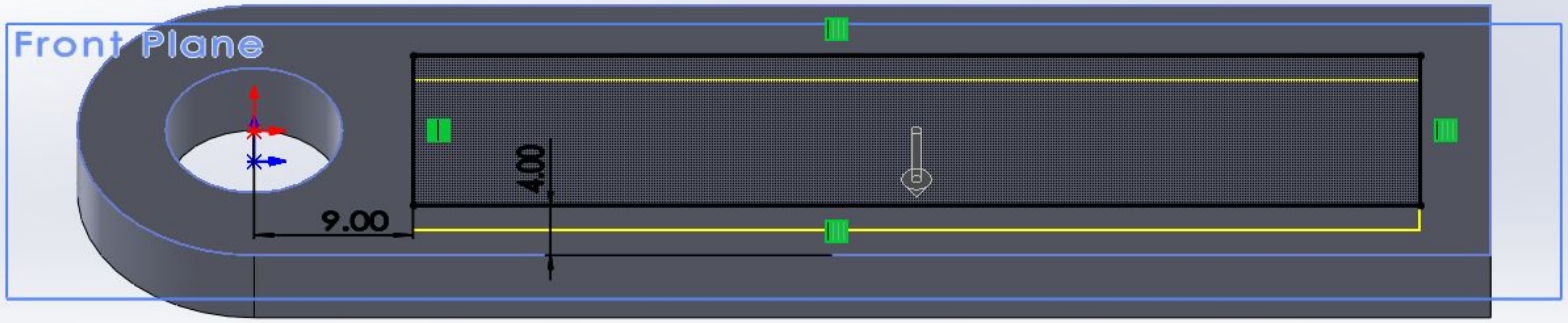


1. Sketch on “Top Face”
2. Draw circle
3. Coincident circle to origin (relation)
4. Dimension to 10mm diameter
5. Extruded cut “Through All”

Note: Extrude vs. Extrude Cut

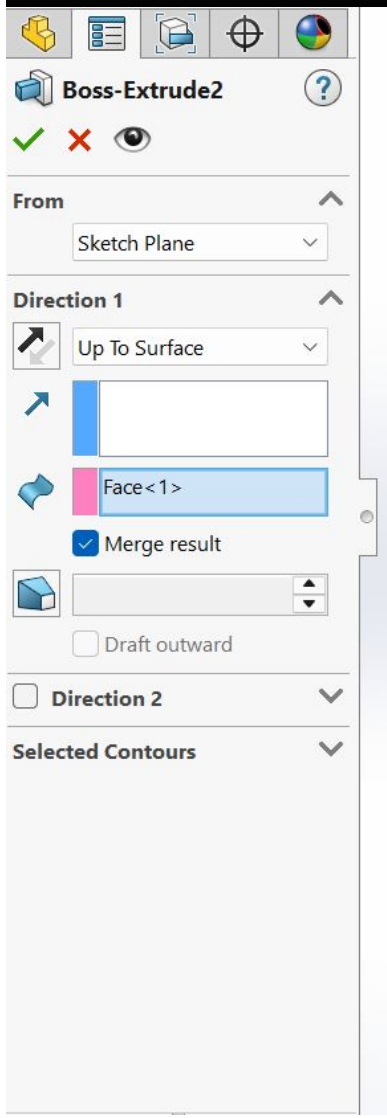
Feature #3 (Extruded Cut)





1. Sketch on “Top Face”
2. Offset face 4mm and Draw vertical line to make a rectangle
3. Trim everything besides the rectangle
4. Dimension vertical line 9mm away from the origin
5. Extruded Cut “2mm Blind”

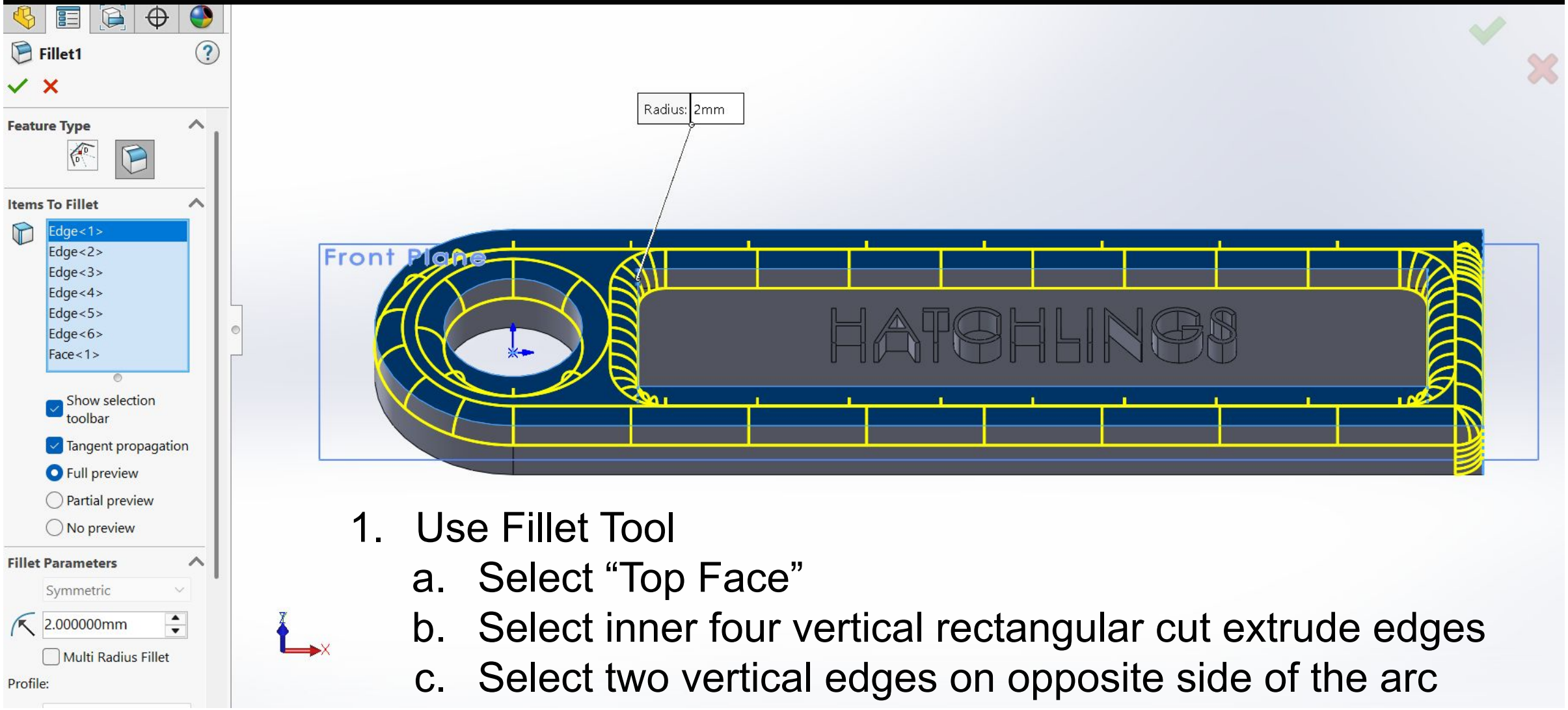
Feature #4 (Boss/Base)



Note: Text is finicky and rarely used.
The bolded default font works well.

1. Sketch on embedded face
2. Draw horizontal construction line
3. Dimension 7.54mm from the top of cut rectangle
4. Use text tool to write name (First Last_initial) using the line as the curve. **Bold the text.**
5. Extruded “Up to Surface” face is “Top Face”

Feature #5 (Fillet)



1. Use Fillet Tool
   a. Select “Top Face”
   b. Select inner four vertical rectangular cut extrude edges
   c. Select two vertical edges on opposite side of the arc
   d. Apply 2mm fillet length

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SolidWorks 3D

Next Week

“Louis, I think this is the beginning of a beautiful friendship”

Rick (Casablanca 1942)

Hatchling