INTRODUCTION TO CAD

THROUGH ONSHAPE

onshape™

AGENDA

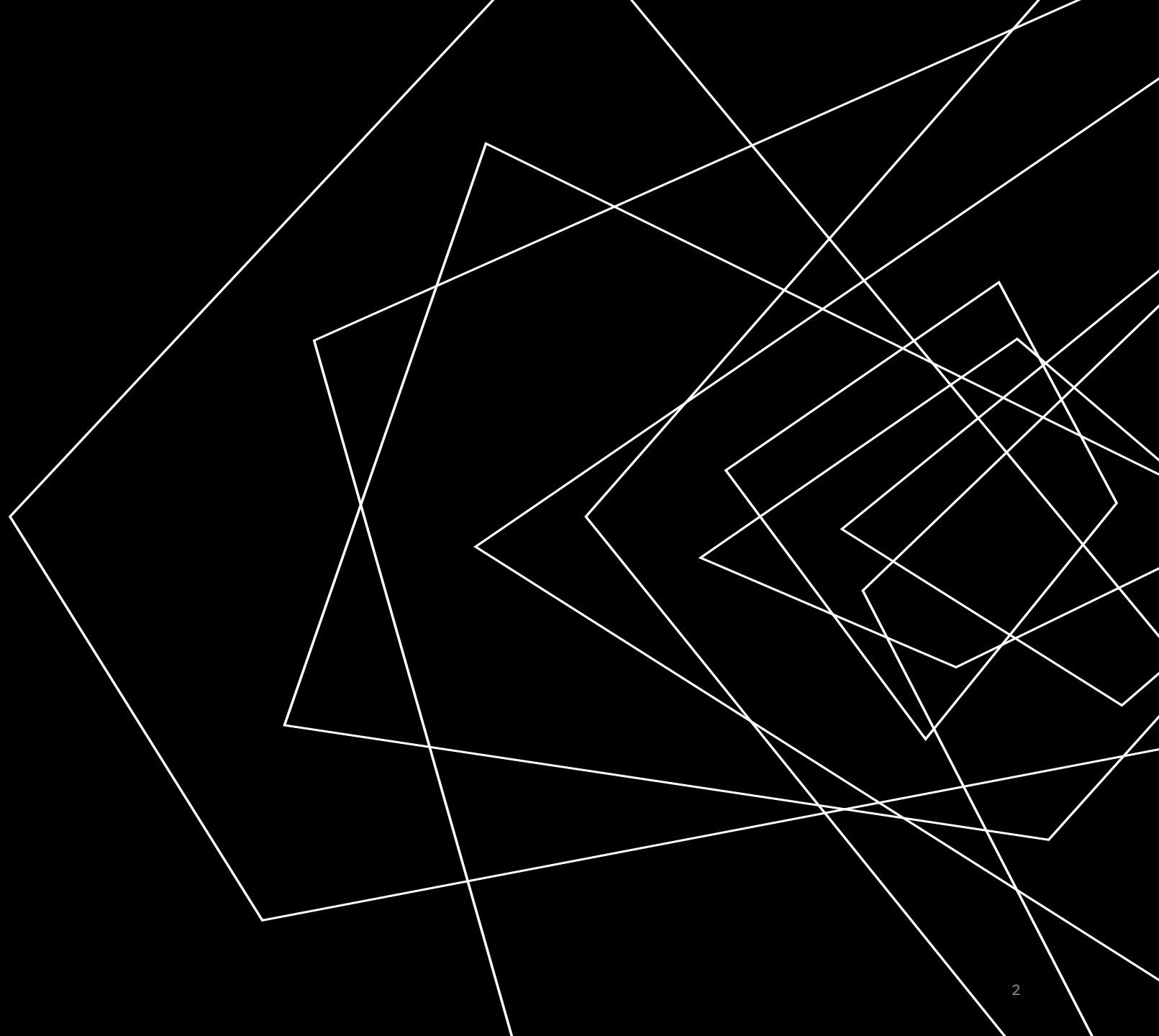
Creating an Account

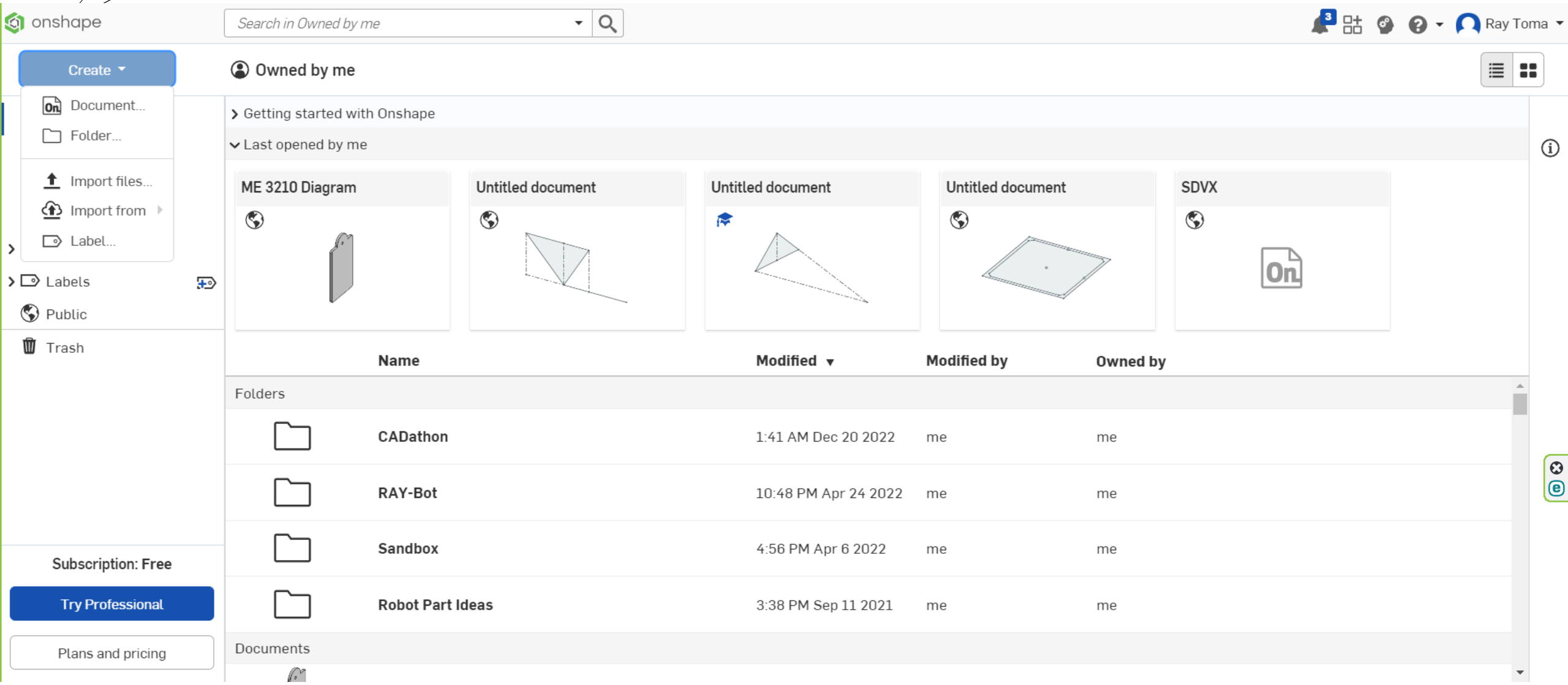
Navigation

Sketches

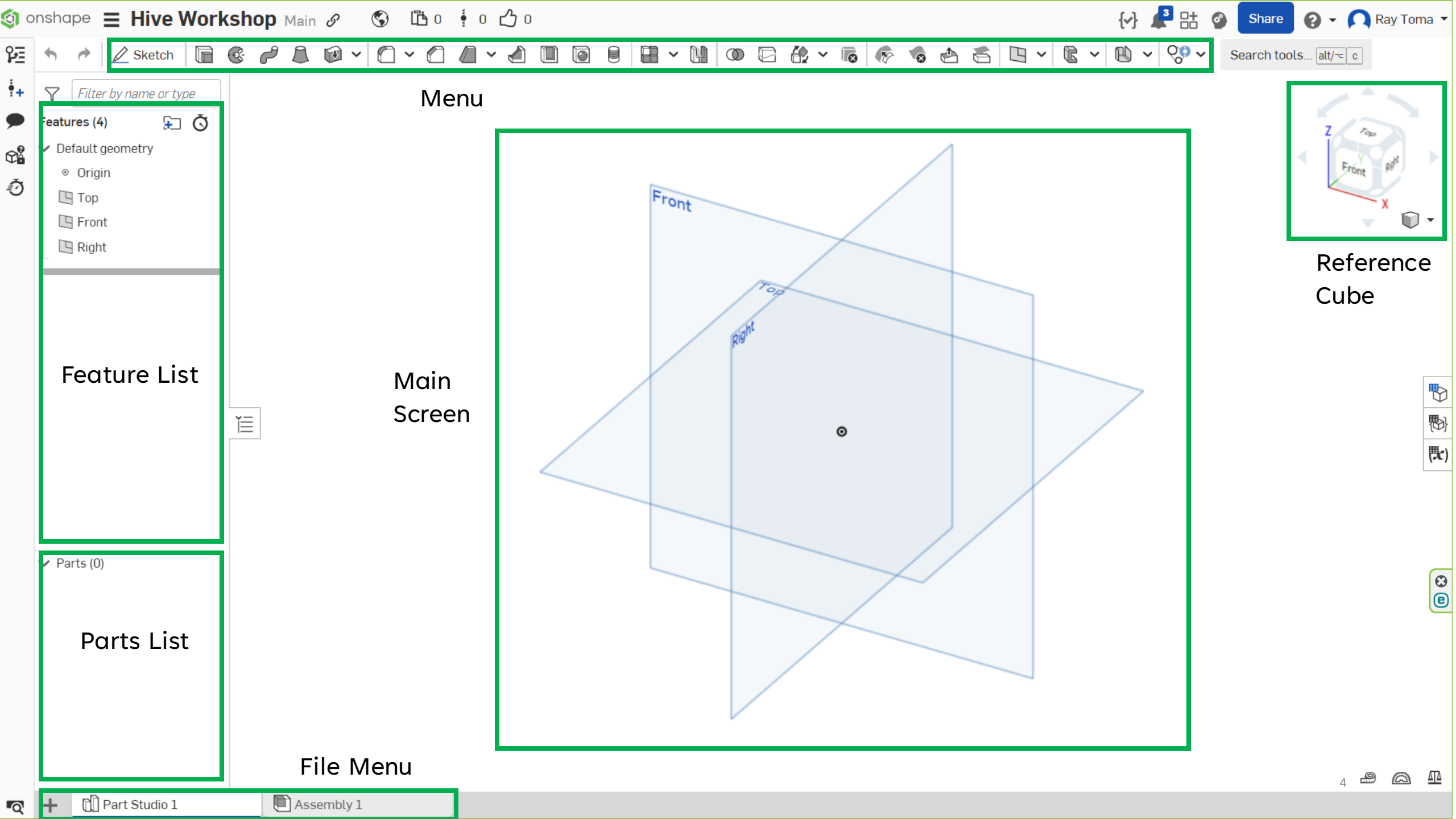
3D Parts

Assemblies





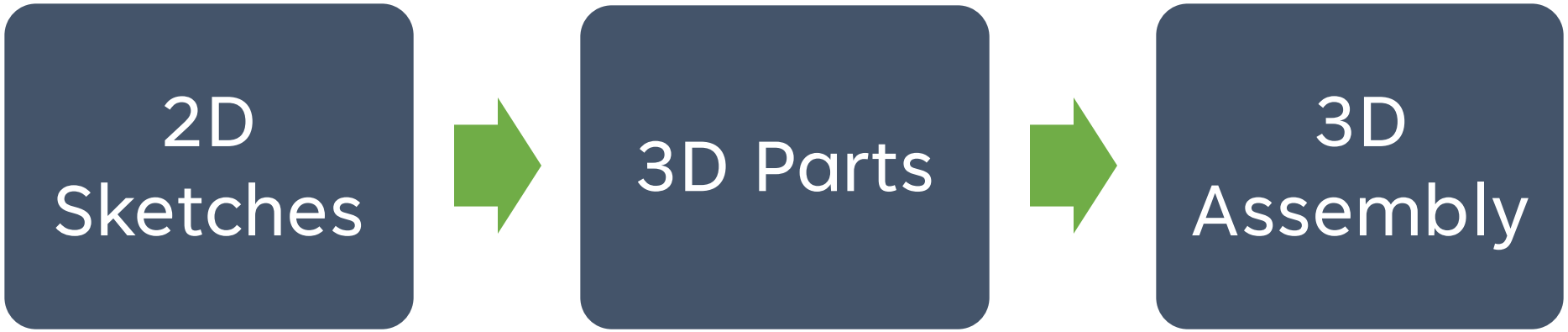
Create (top left) -> Document

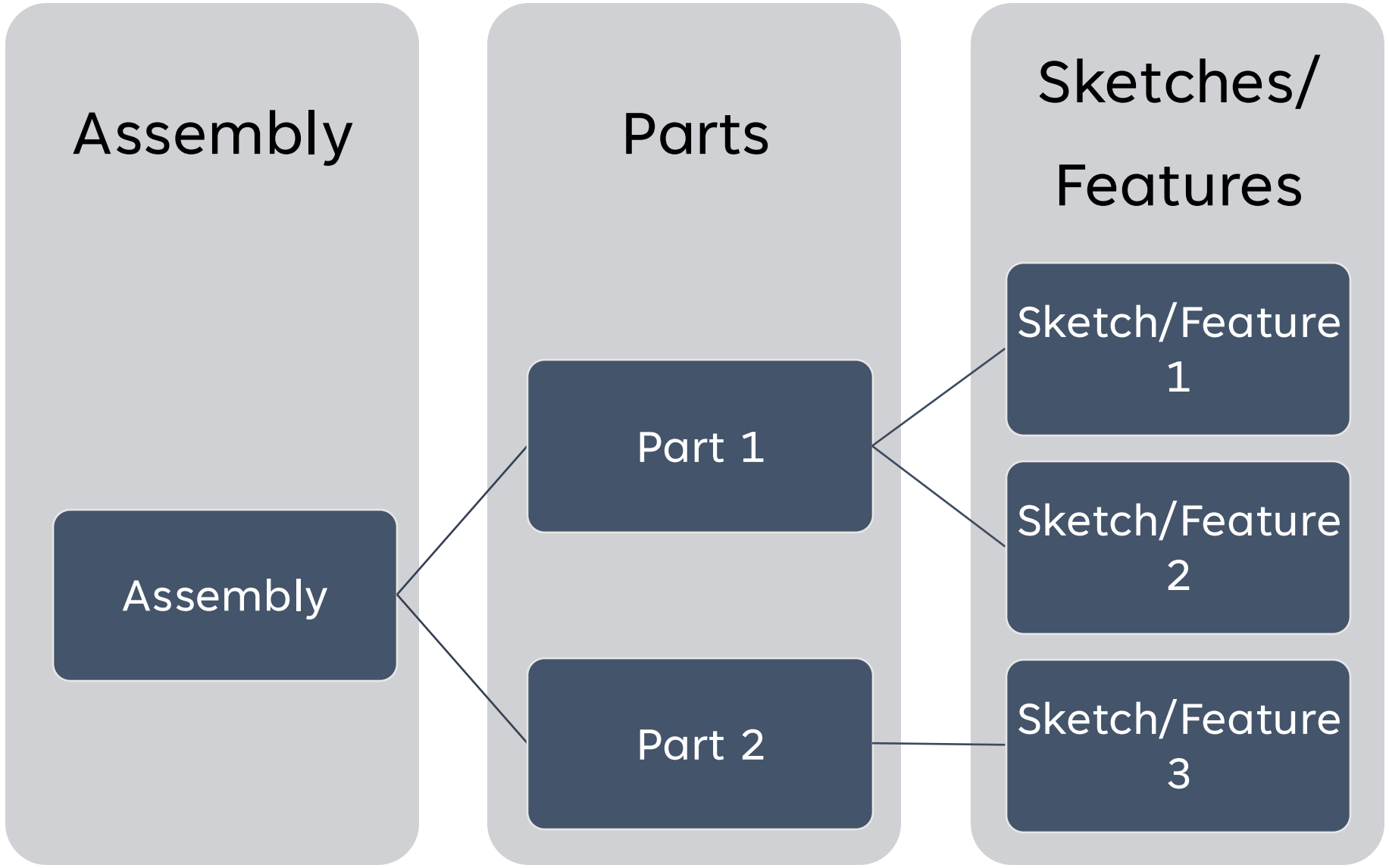

onshape
Hive Workshop
Main
Share
Ray Toma
Sketch
Search tools...
Menu
Filter by name or type
Features (4)
Default geometry
Origin
Top
Front
Right
Feature List
Parts (0)
Parts List
Front
Top
Right
Main Screen
Reference Cube
File Menu
Part Studio 1
Assembly 1

NAVIGATION CONTROLS

Control	Function
Left Click	Select (esc or click out to deselect)
Left Click + Drag	Multi Select
Right Click + Drag	Rotate
Scroll Wheel + Drag	Pan

GENERAL WORKFLOW



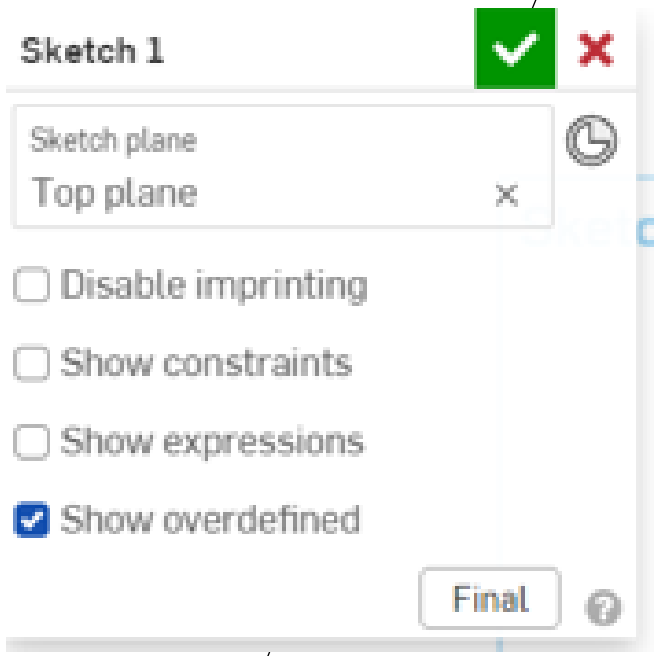

Assembly
Parts
Sketches/ Features
Assembly
Part 1
Part 2
Sketch/Feature 1
Sketch/Feature 2
Sketch/Feature 3

SKETCHES

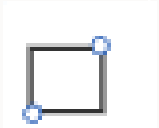
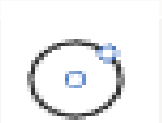
STARTING A SKETCH

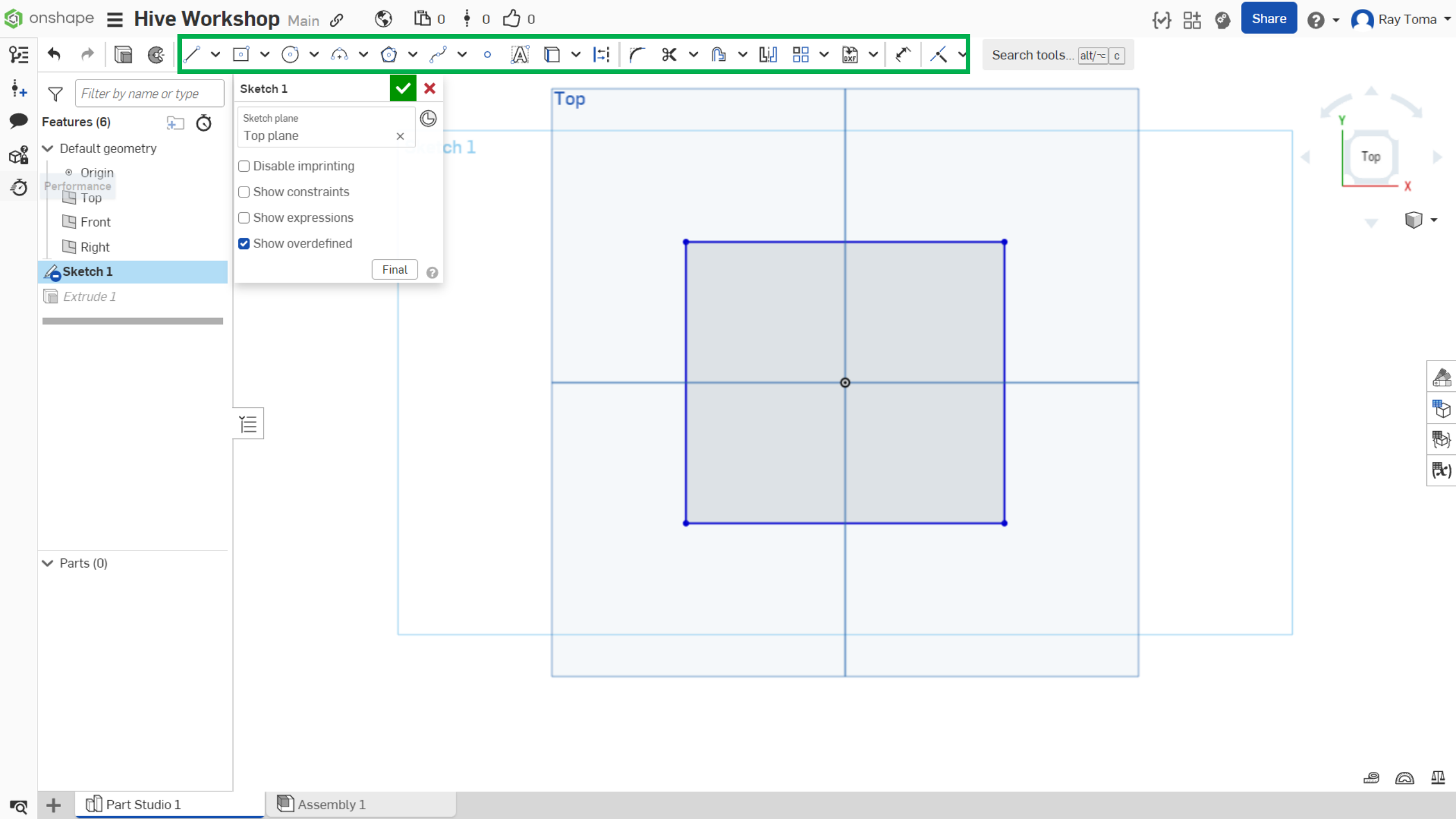
Sketches can be made on any plane or flat faces of parts

1. Click “Sketch” on top left of Part Studio
2. Select a plane or face to draw sketch
3. Add entities to sketch (lines, shapes, etc.)
4. Confirm Sketch (or Onshape won’t keep sketch)



BASIC TOOLS

Icon	Tool	Shortcut	Function
	Line	l	Creates a straight line
	Rectangle	g (corner) or r (center)	Creates a rectangle
	Circle	c	Creates a circle
	Dimension	d	“Dimensions set entities” (set lengths, distances, angles, etc.)


onshape
Hive Workshop
Main
0
0
0
Share
Ray Toma
Search tools... alt/⌥ c
Filter by name or type
Features (6)
Default geometry
Origin
Top
Front
Right
Sketch 1
Extrude 1
Parts (0)
Sketch 1
Sketch plane
Top plane
Disable imprinting
Show constraints
Show expressions
Show overdefined
Final
Top
Y
X
Top
Part Studio 1
Assembly 1

SKETCH RELATIONS (CONSTRAINTS)

- **Used to relate sketch objects to each other**
- **Needed to make a sketch defined (you want your sketch to be fully defined)**
- **Helps sketches stay consistent when changing dimensions**

Coincident. Positions selected faces, edges, and planes (in combination with each other or combined with a single vertex) so they share the same infinite plane. Positions two vertices so they touch.

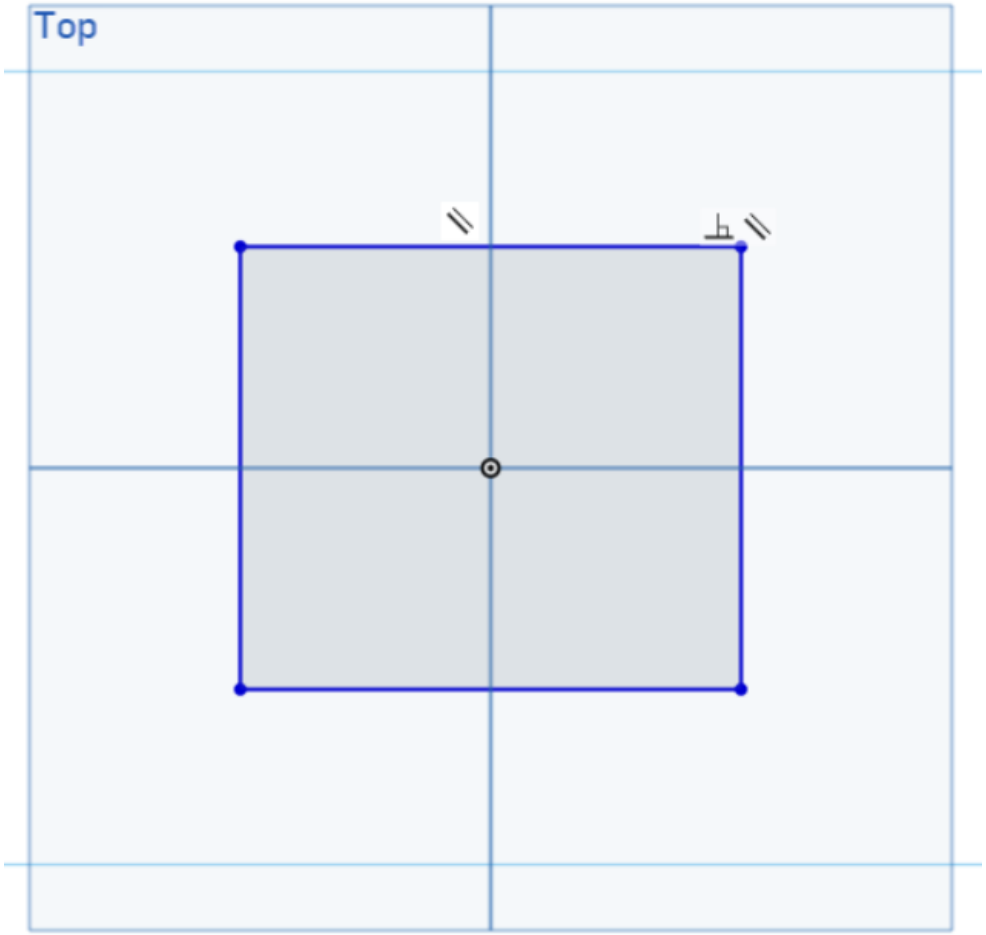
Parallel. Places the selected items so they remain a constant distance apart from each other.

Perpendicular. Places the selected items at a 90° angle to each other.

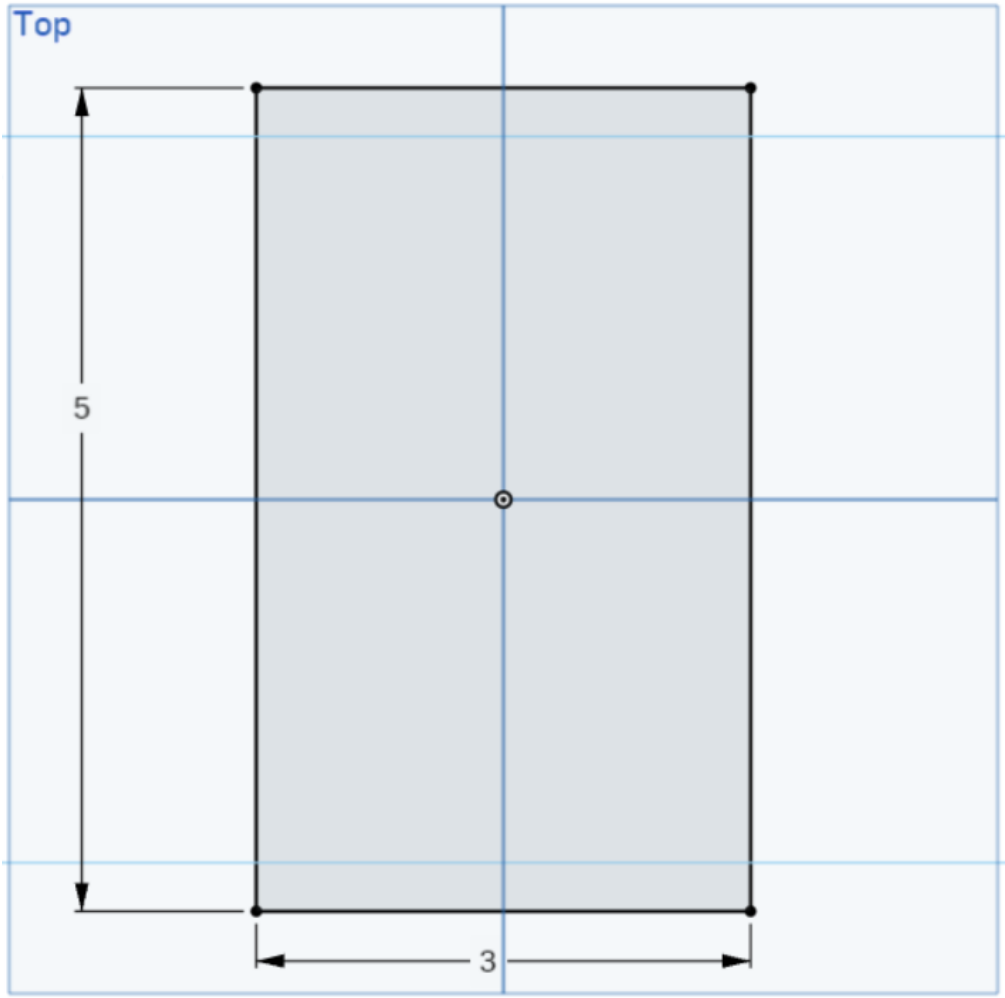
Tangent. Places the selected items tangent to each other (at least one selection must be a cylindrical, conical, or spherical face).

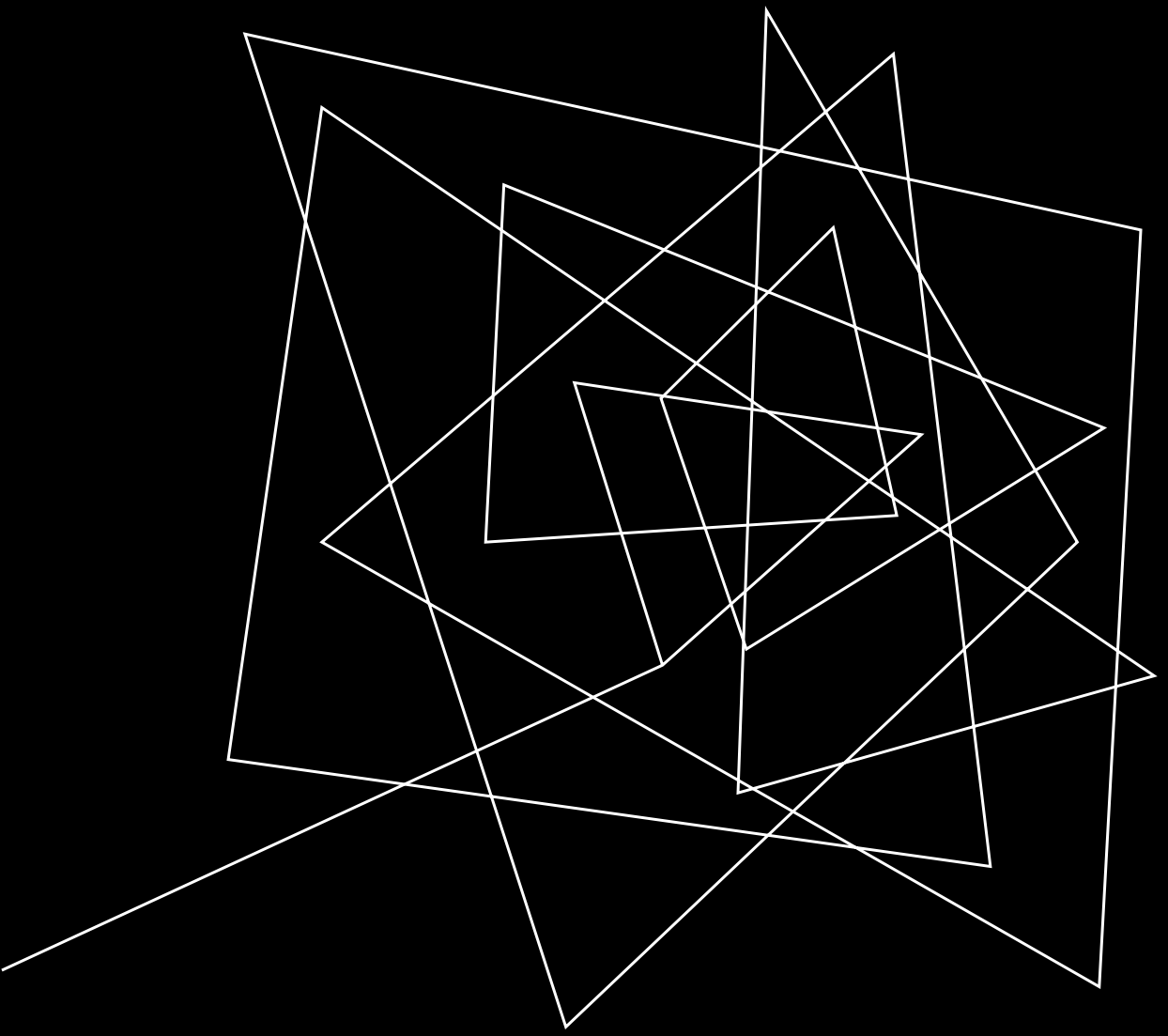
Concentric. Places the selections so that they share the same center line.

BAD



GOOD





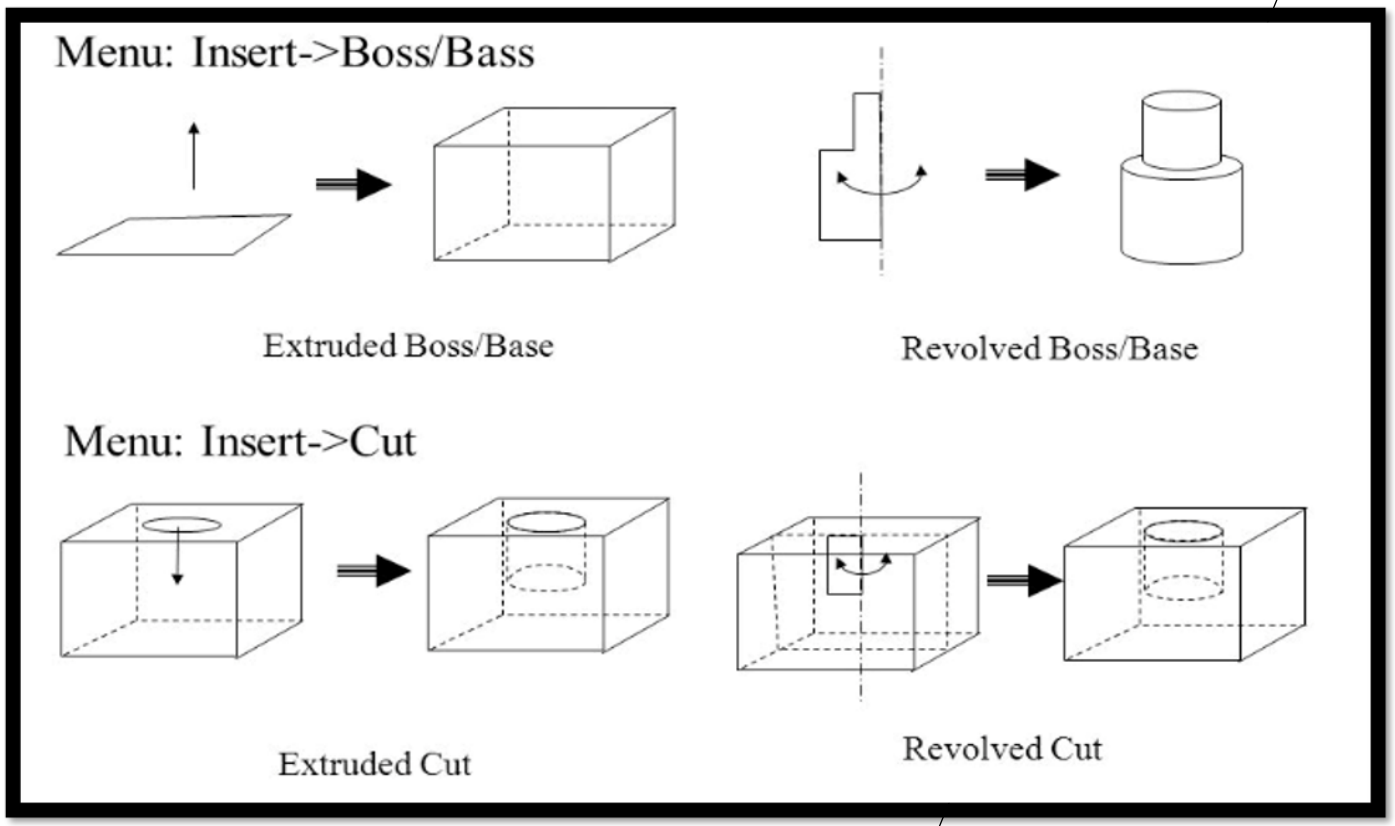
DEMONSTRATION

3D PARTS

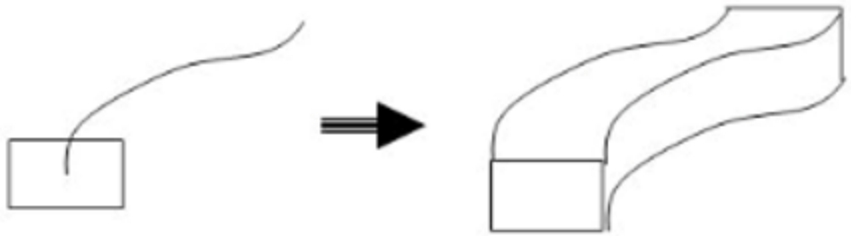
INTRODUCTION TO FEATURES

- Features convert 2D sketches to a 3D Part, or modifies existing parts

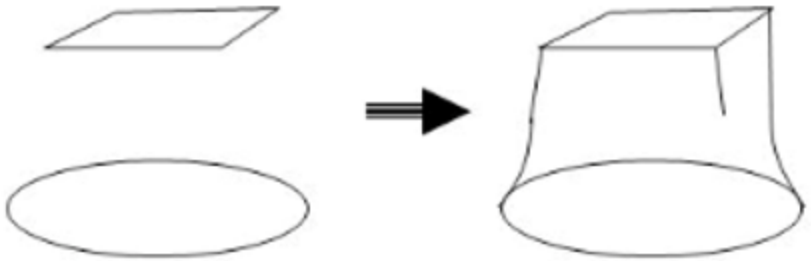
- Depending on the selection of features, the 2D sketch will be utilized in a different way



Menu: Insert->Boss/Bass



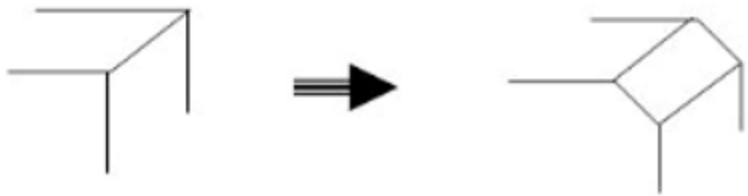
Sweep Boss/Base



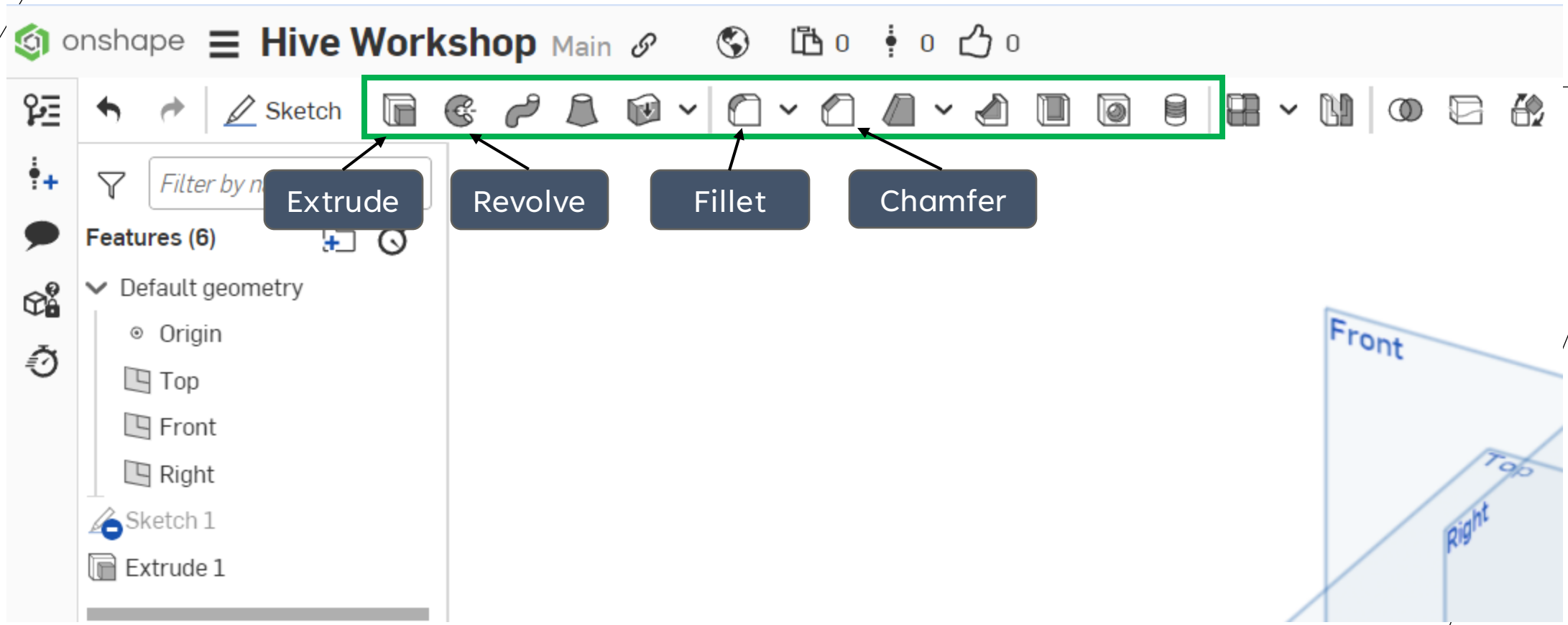
Lofted Boss/Base

Menu: Insert->Features

Fillet

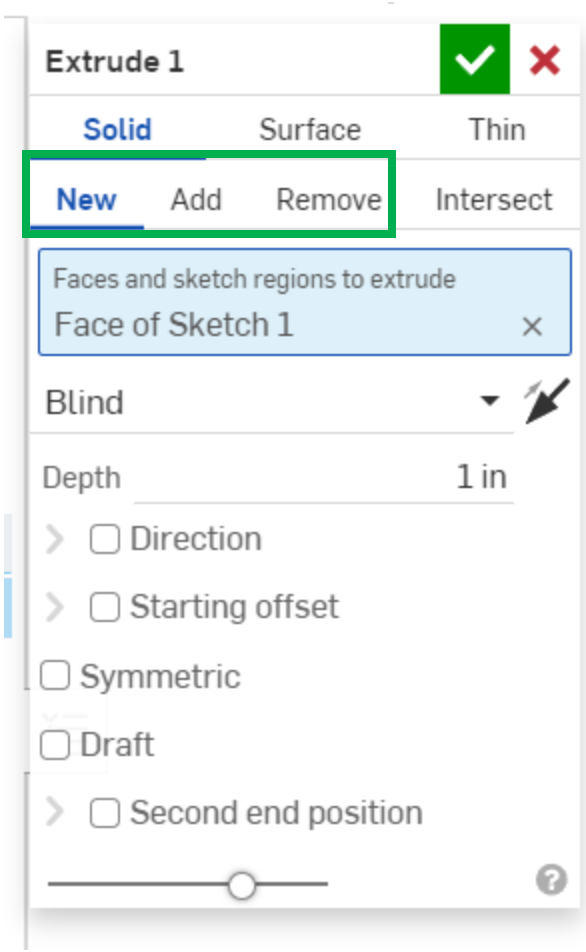


Chamfer


onshape
Hive Workshop
Main
0
0
0
Sketch
Extrude
Revolve
Fillet
Chamfer
Features (6)
Default geometry
Origin
Top
Front
Right
Sketch 1
Extrude 1
Front
Top
Right

FEATURE MENU

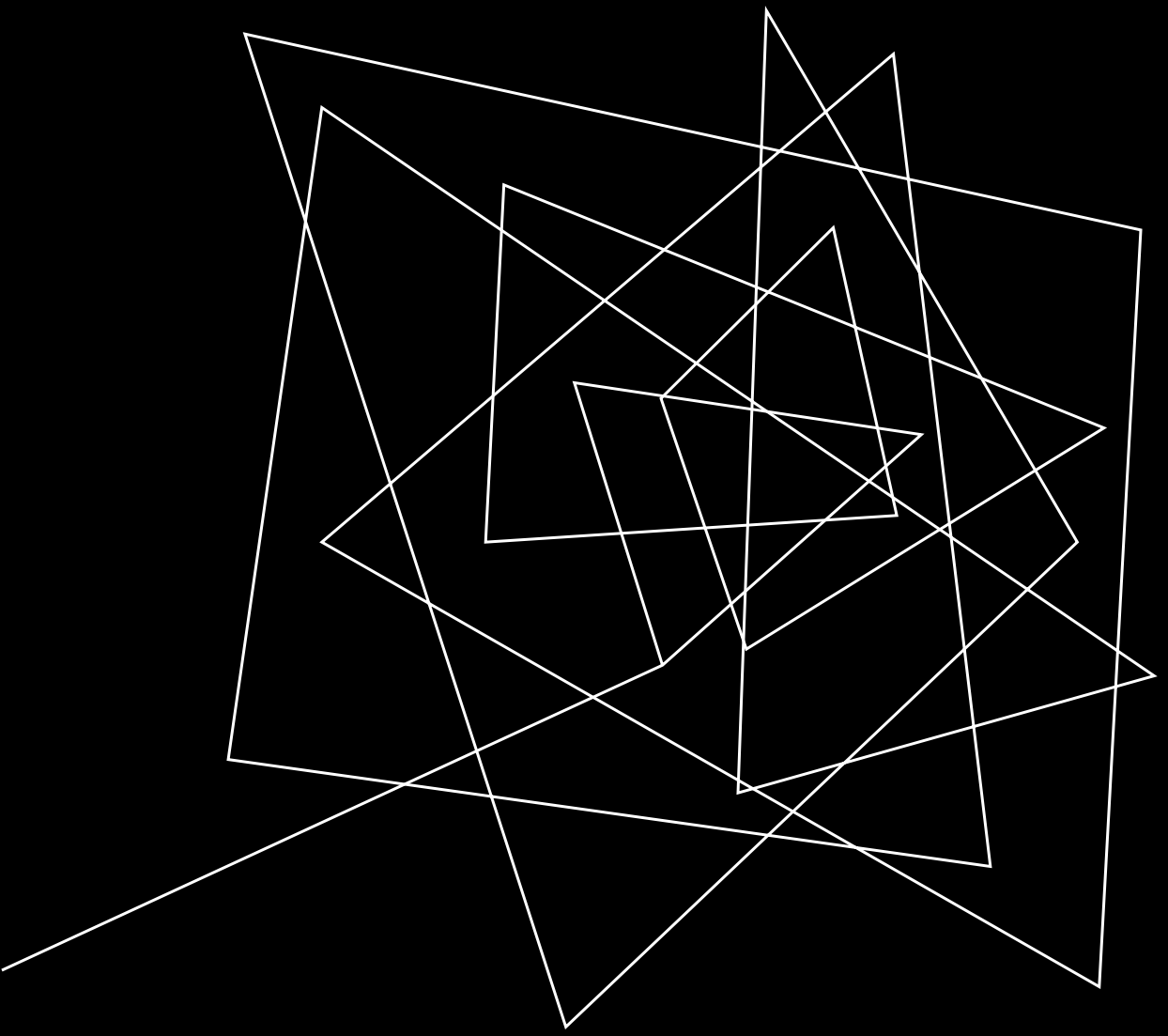
- New: Create a part
- Add: Add feature to an existing part
- Remove (Cut): Cut out feature from existing part



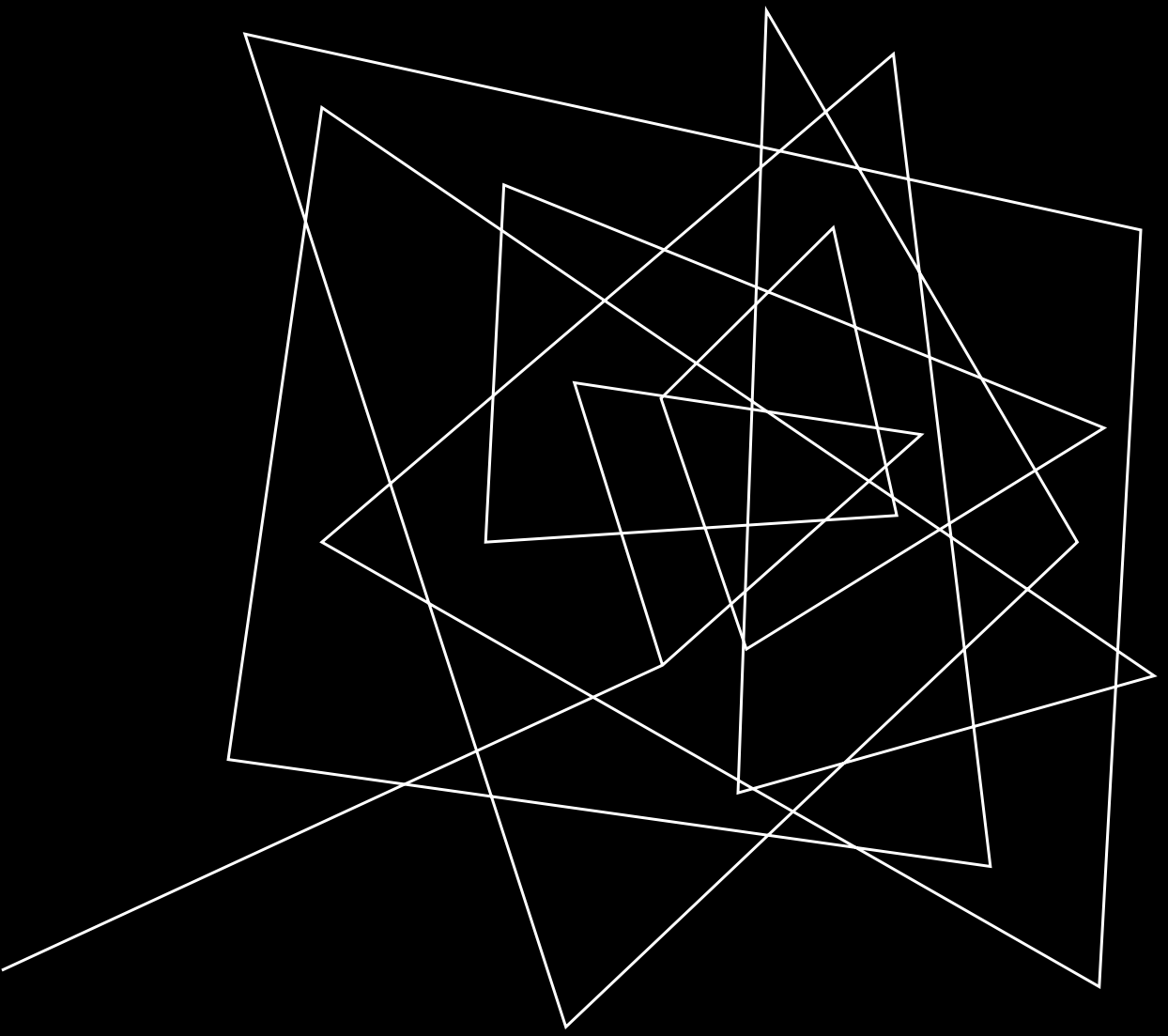
Sketch selection

Type of evaluation

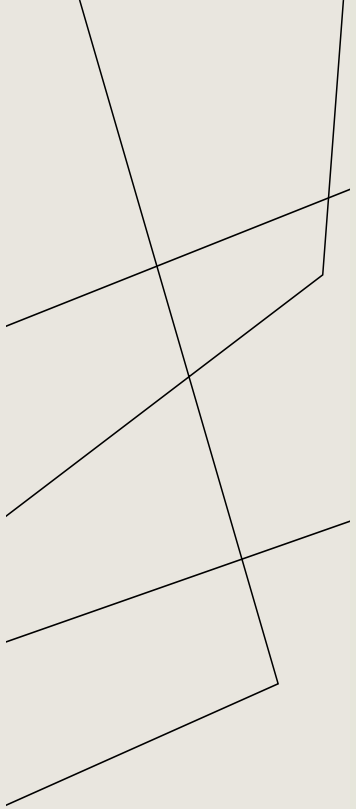
How much to extrude

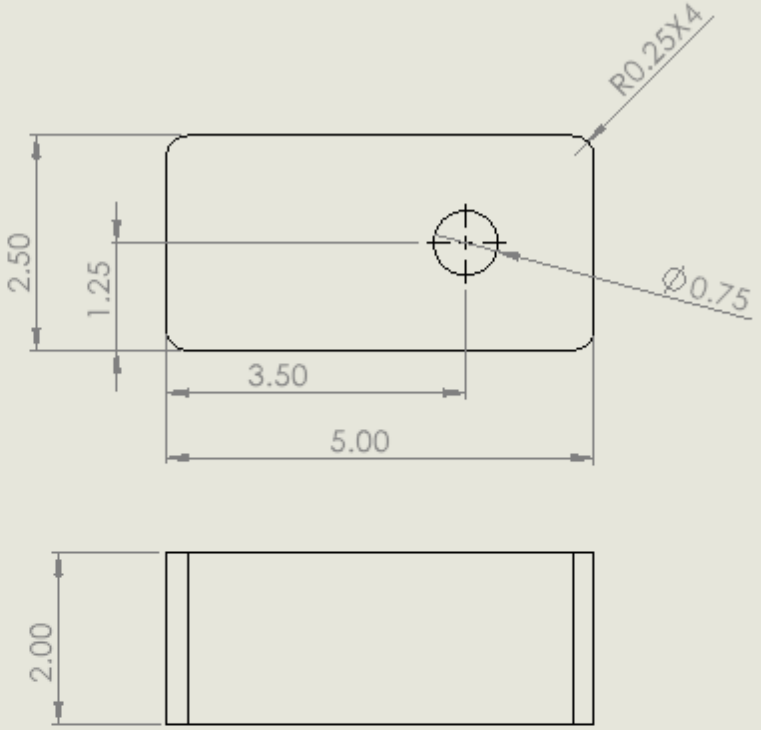


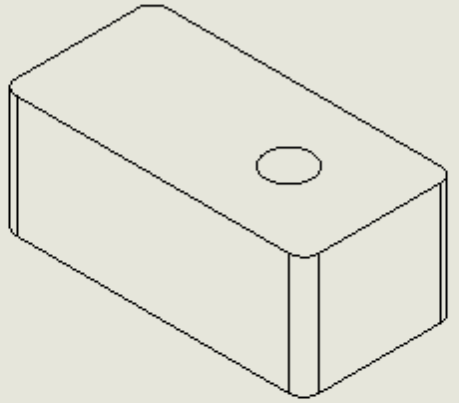
DEMONSTRATION



PRACTICE







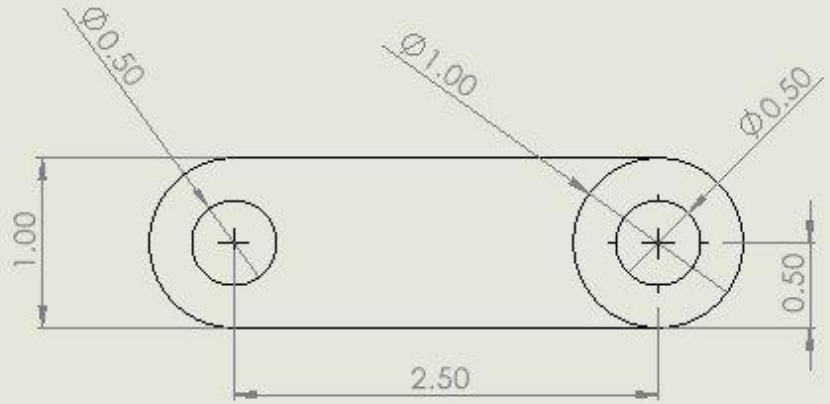
		UNLESS OTHERWISE SPECIFIED:		NAME	DATE
		DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL ± ANGULAR: MACH ± BEND ± TWO PLACE DECIMAL ± THREE PLACE DECIMAL ±	DRAWN		
			CHECKED		
			ENG APPR.		
			MFG APPR.		
		INTERPRET GEOMETRIC TOLERANCING PER:	Q.A.		
		MATERIAL	COMMENTS:		
NEXT ASSY	USED ON	FINISH			
APPLICATION		DO NOT SCALE DRAWING			

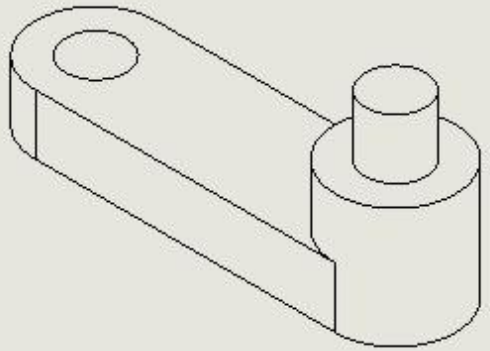
TITLE:

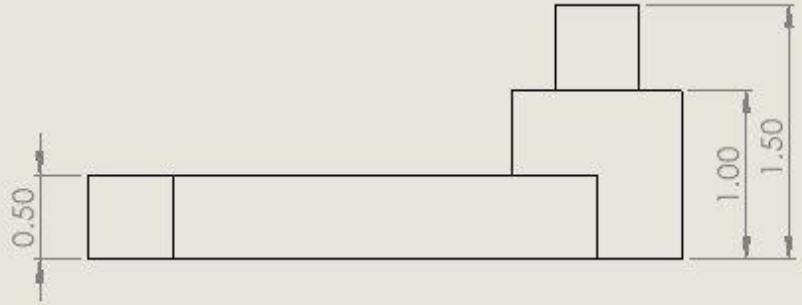
PracticeBase

SIZE	DWG. NO.	REV
A		

SCALE: 1:2	WEIGHT:	SHEET 1 OF 1







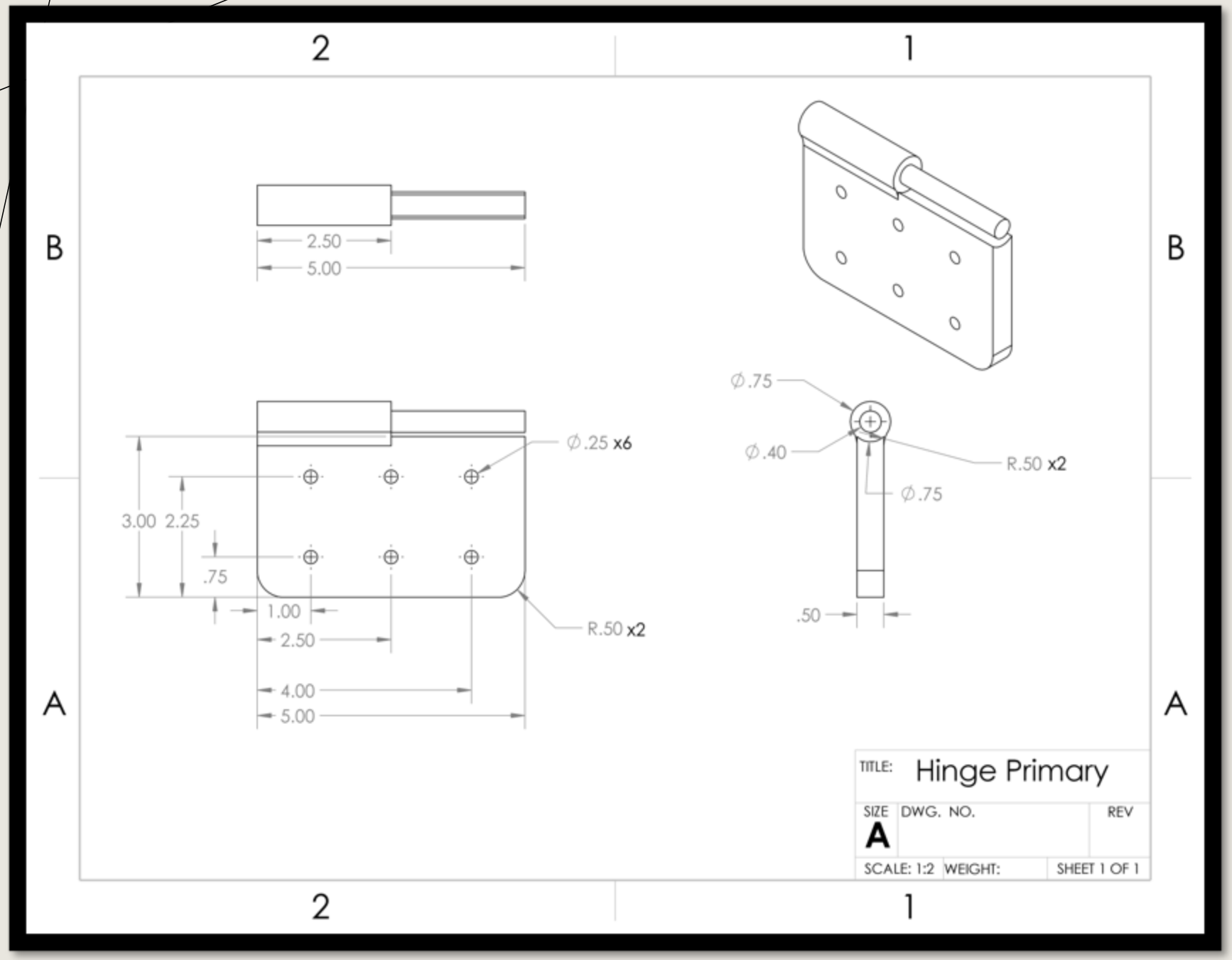
		UNLESS OTHERWISE SPECIFIED:		NAME	DATE
		DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL ± ANGULAR: MACH ± BEND ± TWO PLACE DECIMAL ± THREE PLACE DECIMAL ±	DRAWN		
			CHECKED		
			ENG APPR.		
			MFG APPR.		
			Q.A.		
		INTERPRET GEOMETRIC TOLERANCING PER:	COMMENTS:		
		MATERIAL			
NEXT ASSY	USED ON	FINISH			
APPLICATION		DO NOT SCALE DRAWING			

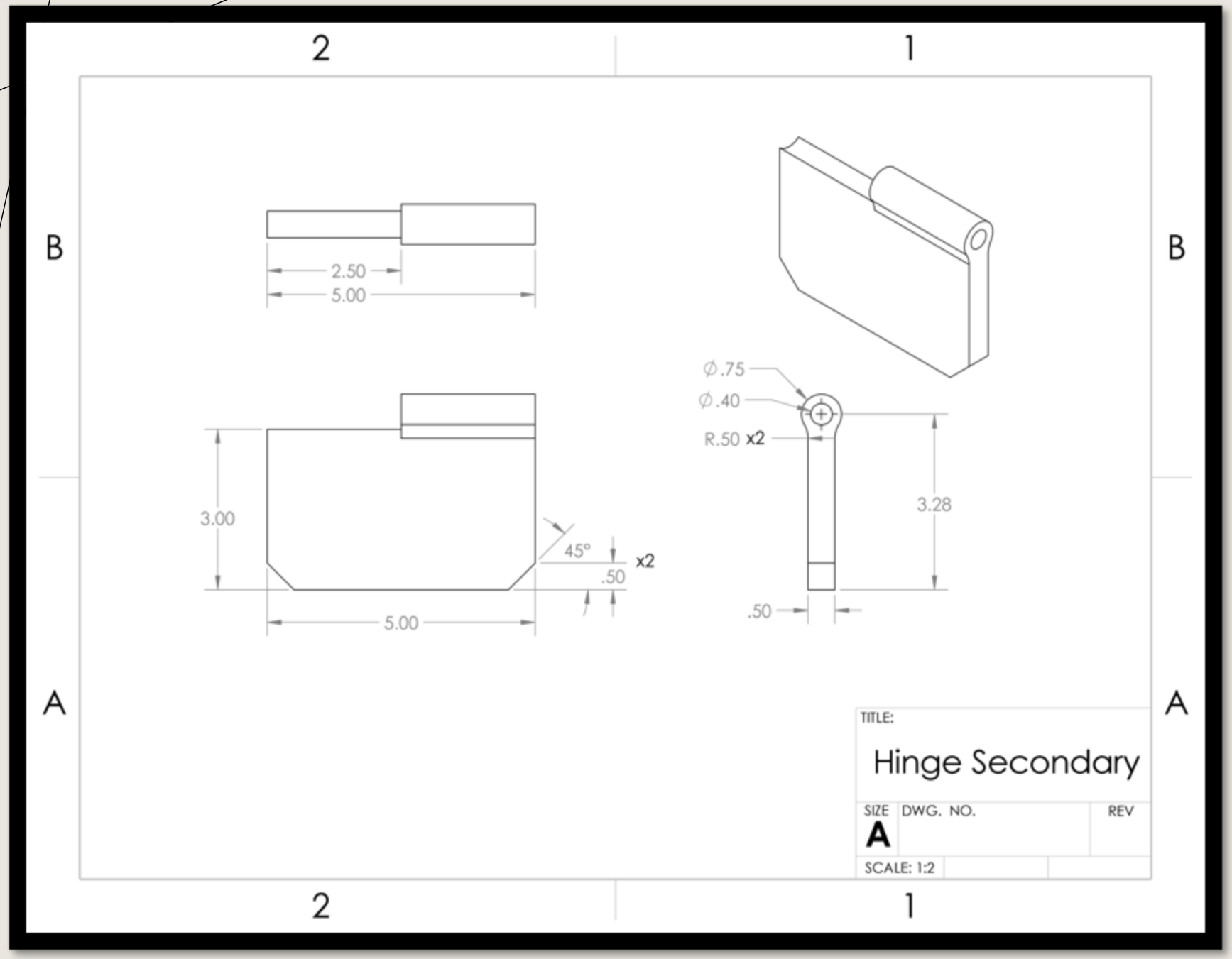
TITLE:

PracticeHandle

SIZE	DWG. NO.	REV
A		

SCALE: 1:1	WEIGHT:	SHEET 1 OF 1

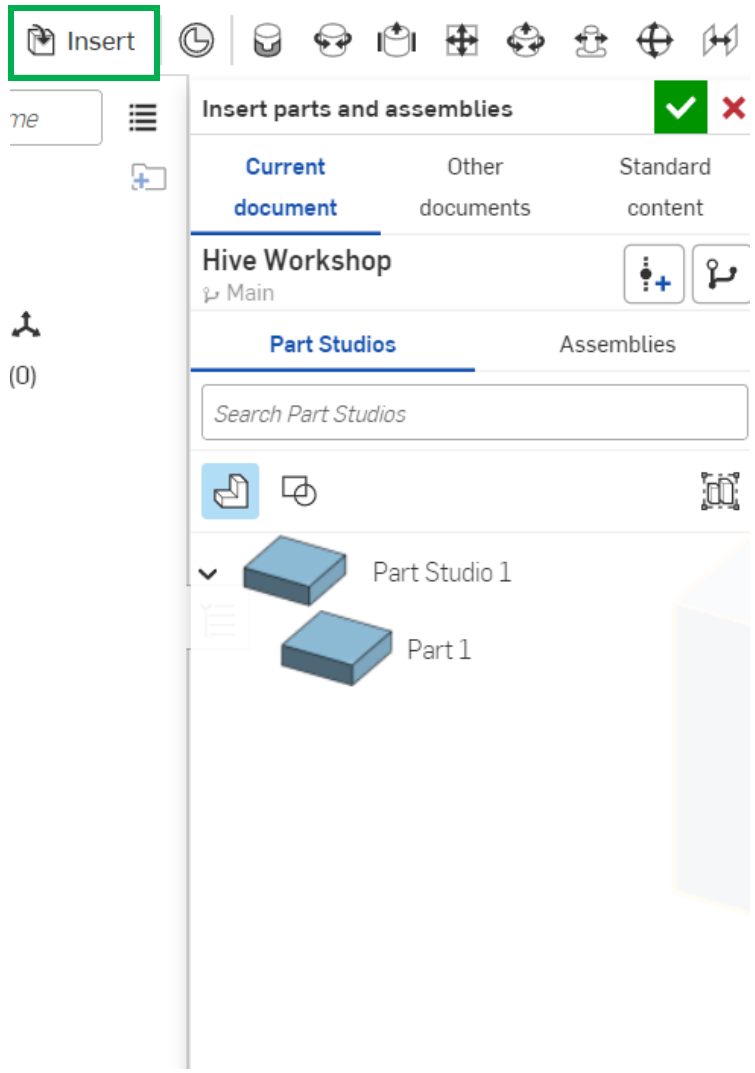

2
1
B
A
2.50
5.00
Ø.25 x6
3.00
2.25
.75
1.00
2.50
4.00
5.00
R.50 x2
Ø.75
Ø.40
R.50 x2
Ø.75
.50
TITLE: Hinge Primary
SIZE A
DWG. NO.
REV
SCALE: 1:2
WEIGHT:
SHEET 1 OF 1


2
1
B
A
2.50
5.00
3.00
45°
x2
.50
5.00
Ø.75
Ø.40
R.50 x2
3.28
.50
TITLE:
Hinge Secondary
SIZE
A
DWG. NO.
REV
SCALE: 1:2

ASSEMBLIES

CREATING AN ASSEMBLY

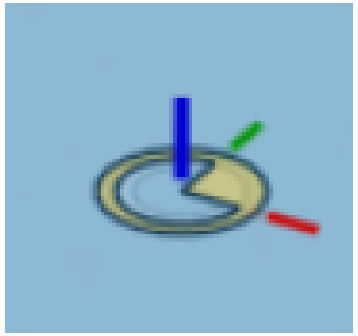
- From the bottom file menu, click “Assembly 1”
- To Insert a part, click “Insert”, and you should see a screen similar to the one on the right



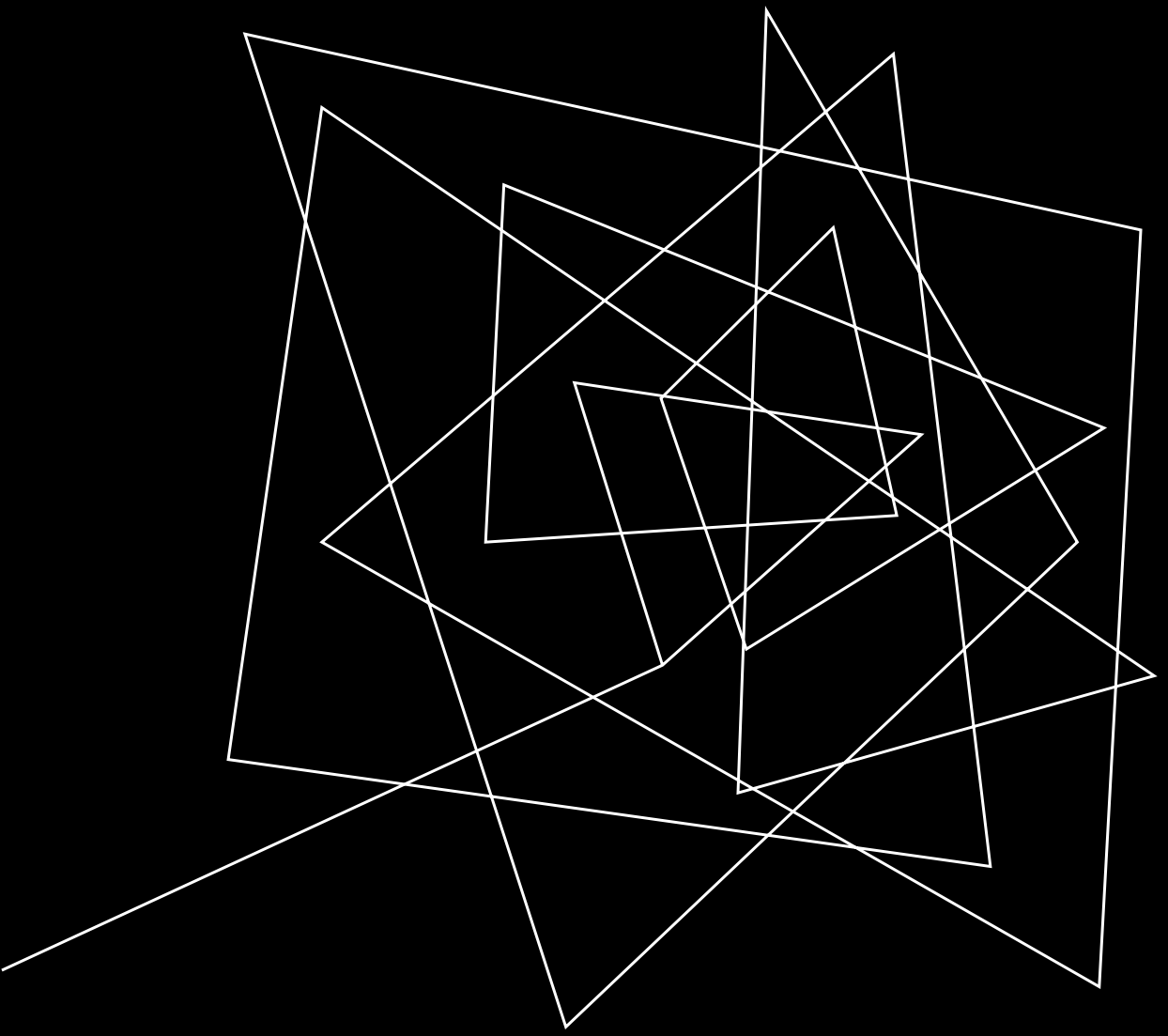
HOW TO DEFINE PARTS TOGETHER?

Using Mates

- Mates, similar to constraints, define relations between parts
- You can select Points, Edges, Faces to Mate
- Common Mates
  - Fastened: Overlaps Points, Edges, or Faces (be careful with orientation of selected entities)
  - Planar: Overlaps two flat faces together
  - Cylindrical: For two circular surfaces, aligns the center axis

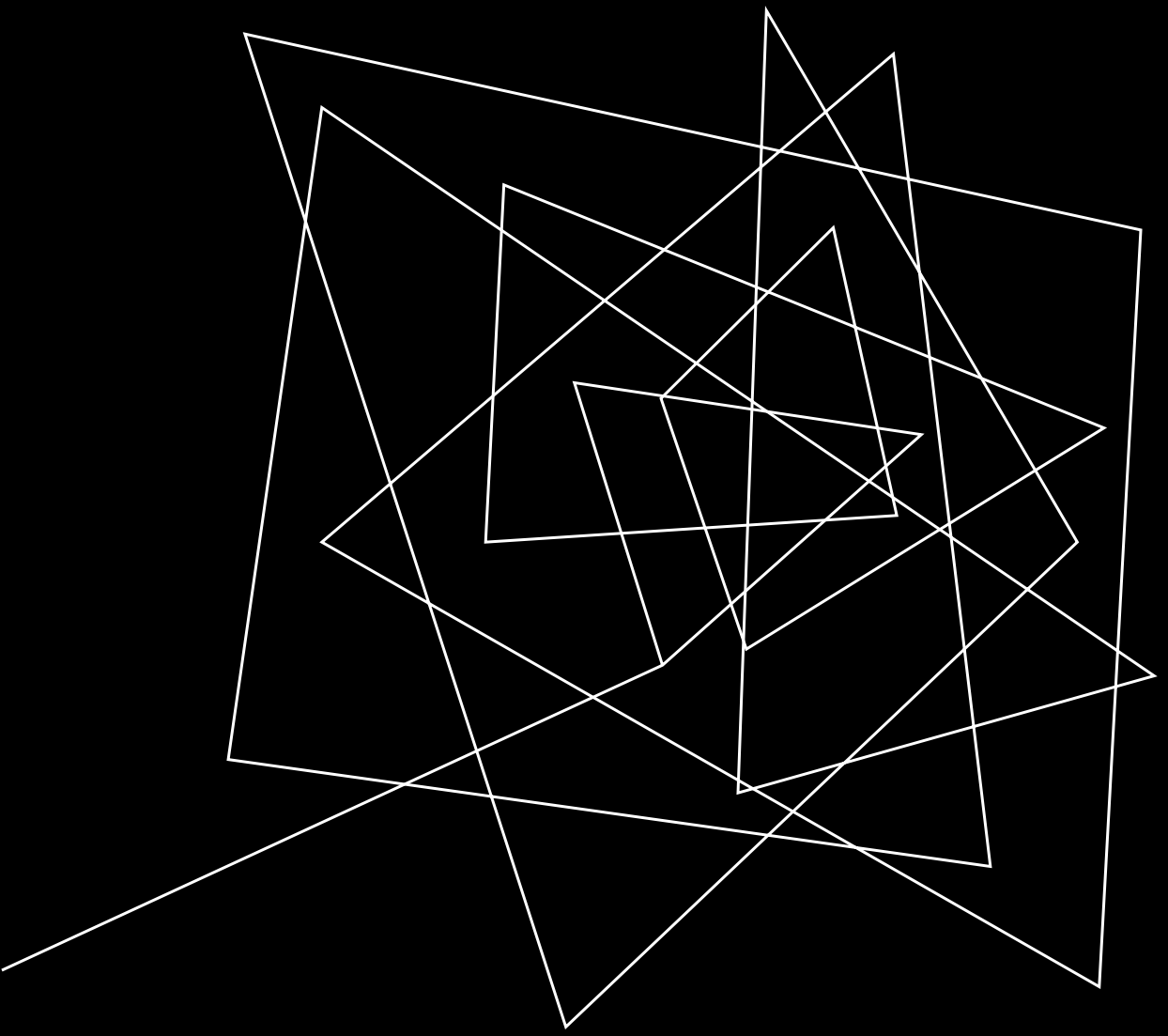


Mate Connector:
A point on part in which a mate is taken



PRACTICE:

- Mate the Base and Handle
- Use 1 Planar, 1 Cylindrical Mate



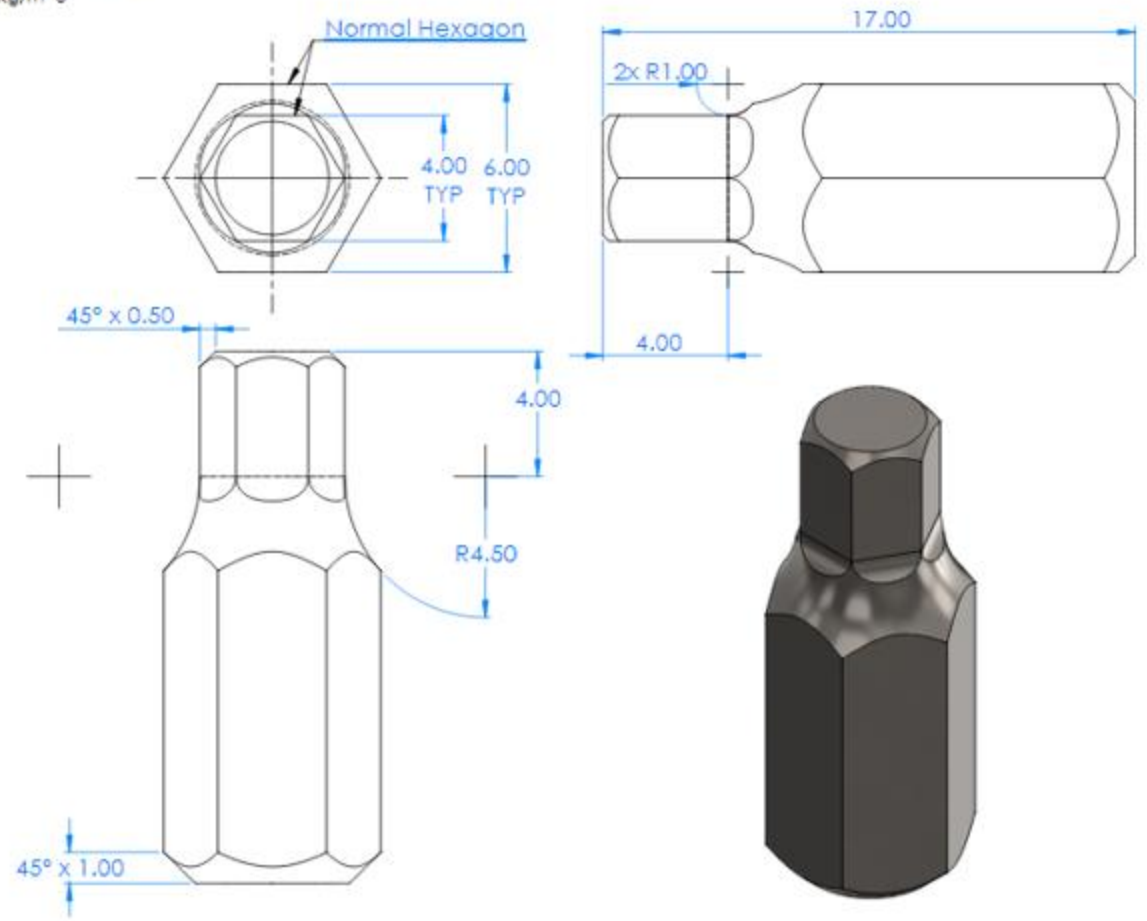
EXTRA PRACTICE:

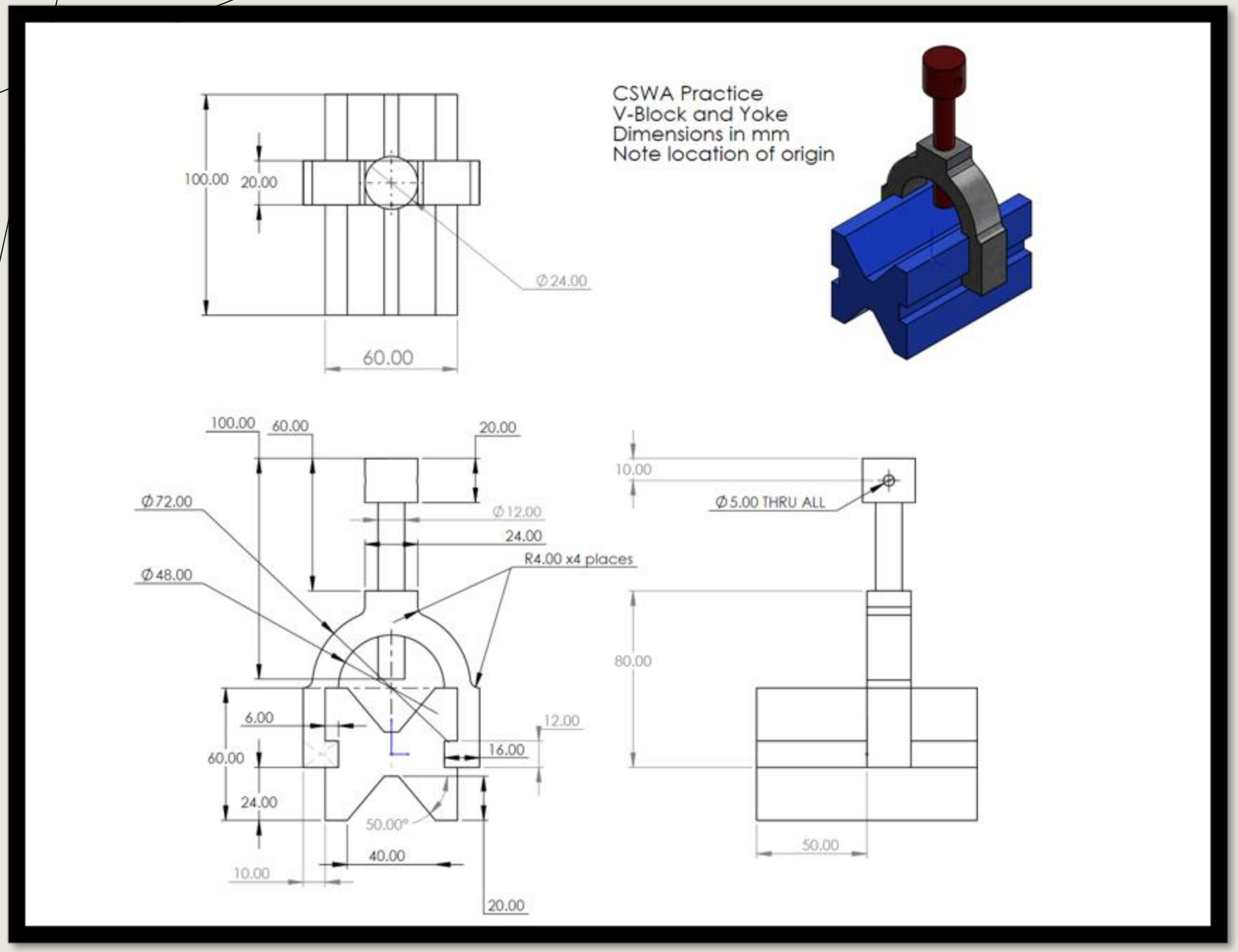
- Mate the Hinges together
- Bonus: Limit the angle of rotation

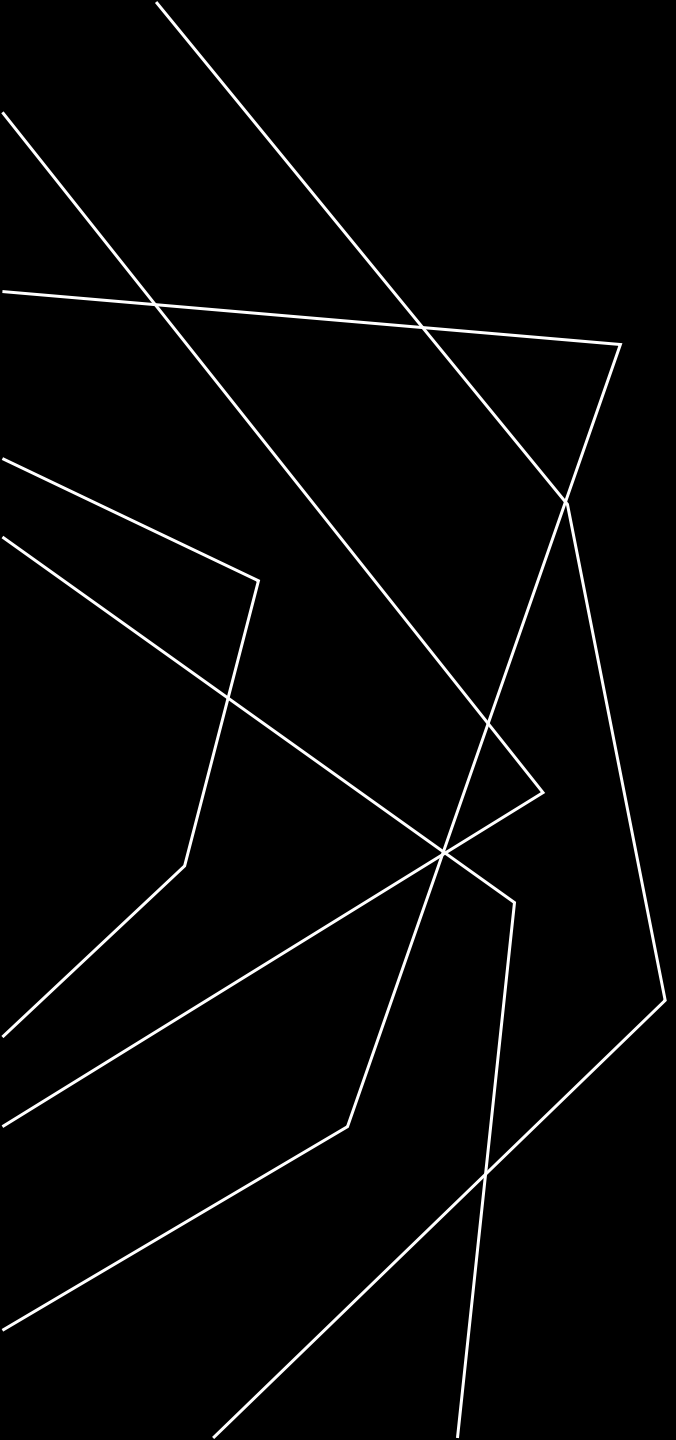
Screw Bit

Part A - Build this part in SOLIDWORKS. Use the values below as needed, then find the mass of the part (grams).

Unit System: MMGS
Decimal Places: 2
Part Origin: Arbitrary
Material Type: Steel
Material: Stainless Steel (ferritic)
Density: 7800 kg/m^3




CSWA Practice
V-Block and Yoke
Dimensions in mm
Note location of origin
100.00
20.00
Ø24.00
60.00
100.00
60.00
20.00
Ø72.00
Ø12.00
24.00
R4.00 x4 places
Ø48.00
6.00
12.00
60.00
16.00
24.00
50.00°
40.00
10.00
20.00
10.00
Ø5.00 THRU ALL
80.00
50.00



THANK YOU FOR COMING!

ADDITIONAL INFORMATION

Solidworks Course

- Primary Link to Canvas Course: https://gatech.instructure.com/enroll/DTCD39
- Secondary Link to Post (has some other info): https://www.reddit.com/r/gatech/comments/14bdixi/free_solidworks_cswa_cswp_course_and_unlimited/

If you ever want to 3D print a part:

- Right click on the part studio or specific part
- Click on export
- Specify format as STL
- As long as you have the STL file, the Hive will help you slice and print the part!