

A series of thin, black, overlapping lines forming various geometric shapes like triangles and polygons, creating a complex, abstract pattern in the upper left portion of the slide.

INTRODUCTION TO CAD

THROUGH ONSHAPE



AGENDA

Creating an Account

Navigation

Sketches

3D Parts

Assemblies

onshape

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Ray Toma

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Labels

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Getting started with Onshape

Last opened by me





ME 3210 Diagram

Untitled document

Untitled document

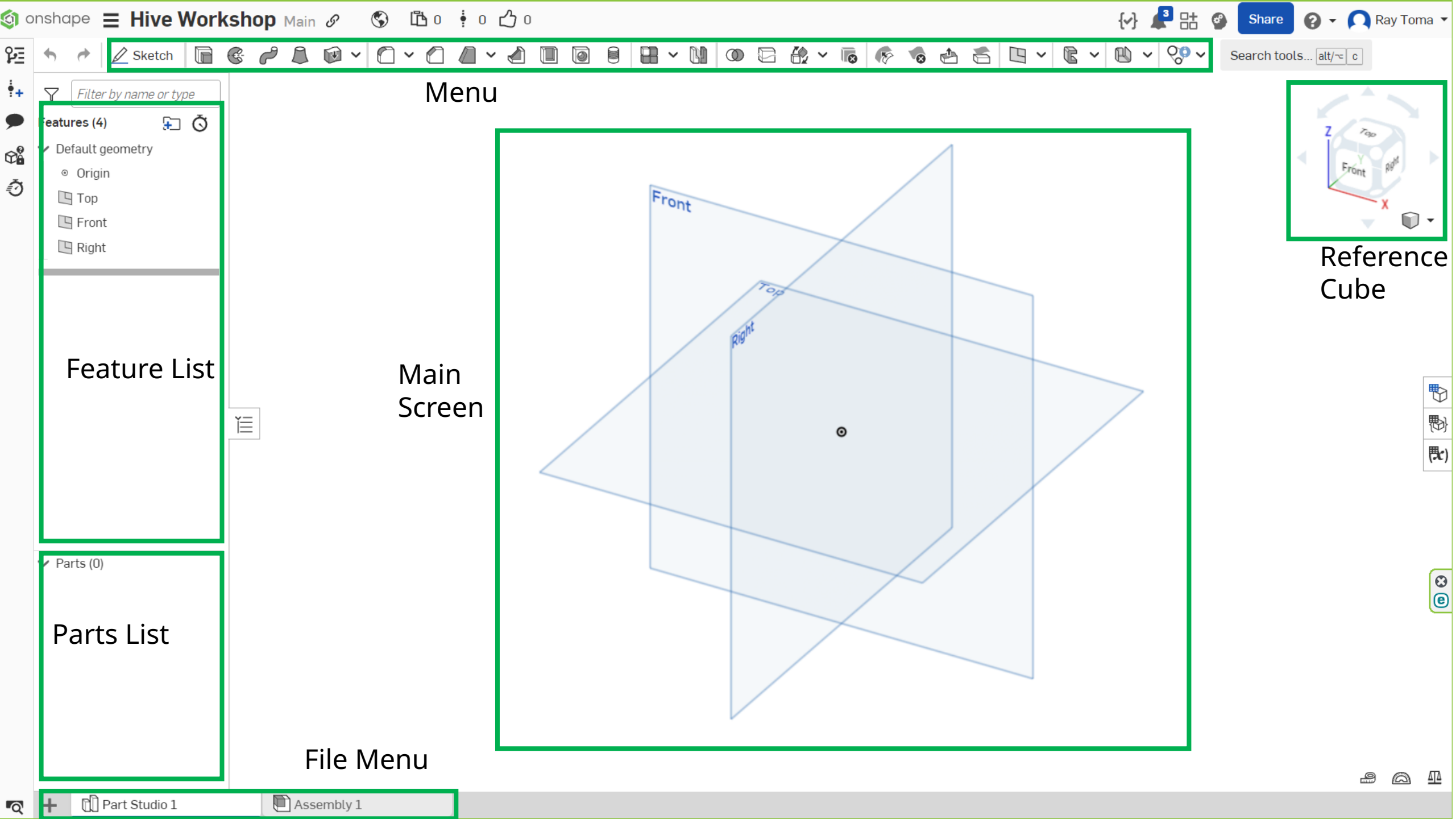
Untitled document

SDVX

Name	Modified	Modified by	Owned by
Folders			
 CADathon	1:41 AM Dec 20 2022	me	me
 RAY-Bot	10:48 PM Apr 24 2022	me	me
 Sandbox	4:56 PM Apr 6 2022	me	me
 Robot Part Ideas	3:38 PM Sep 11 2021	me	me
Documents			

Create (top left) -> Document

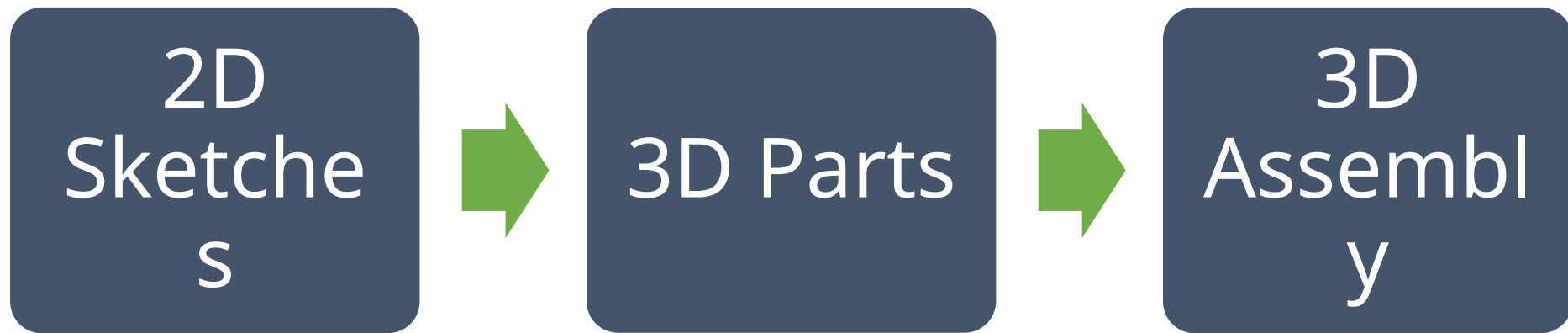
3

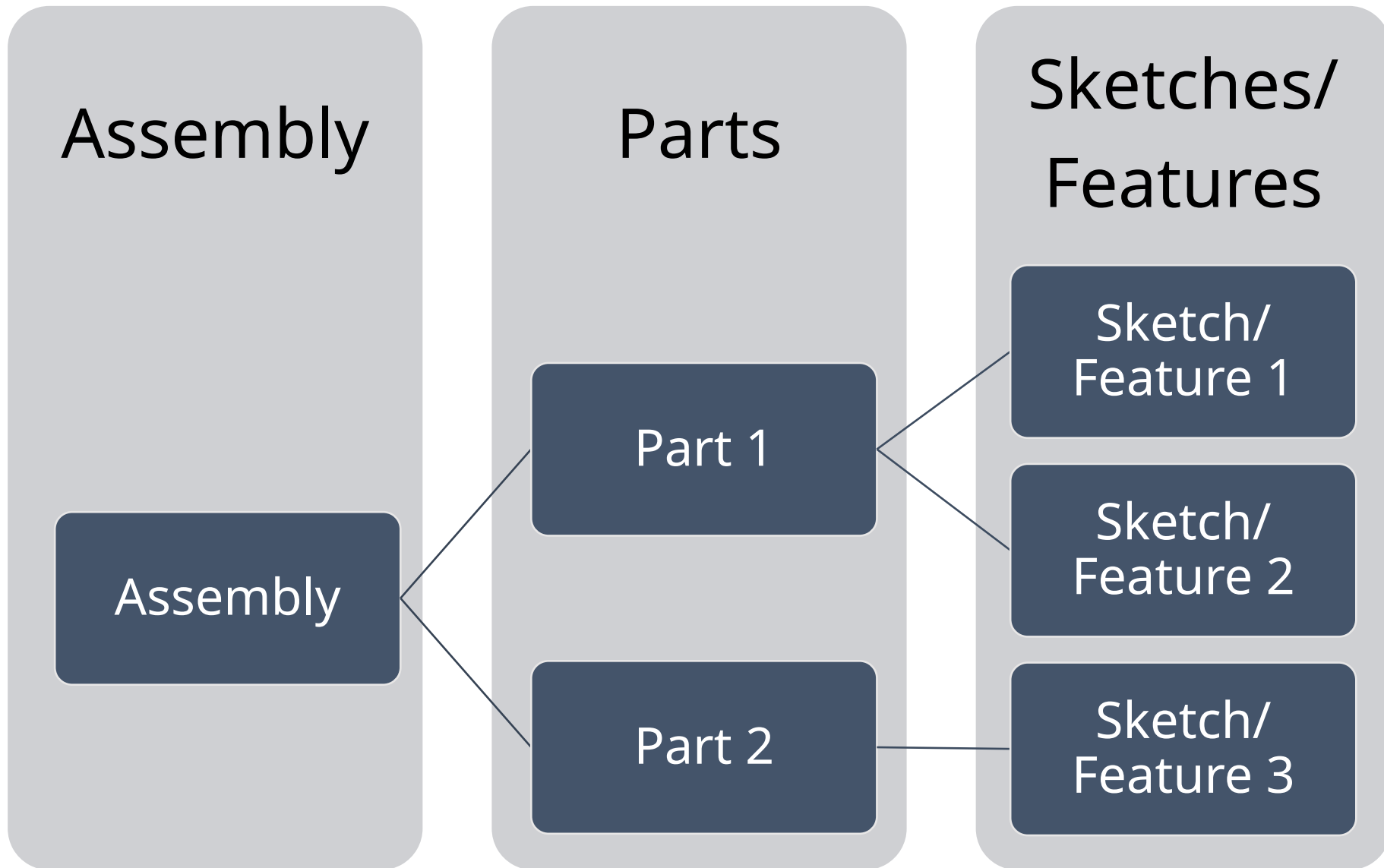


NAVIGATION CONTROLS

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





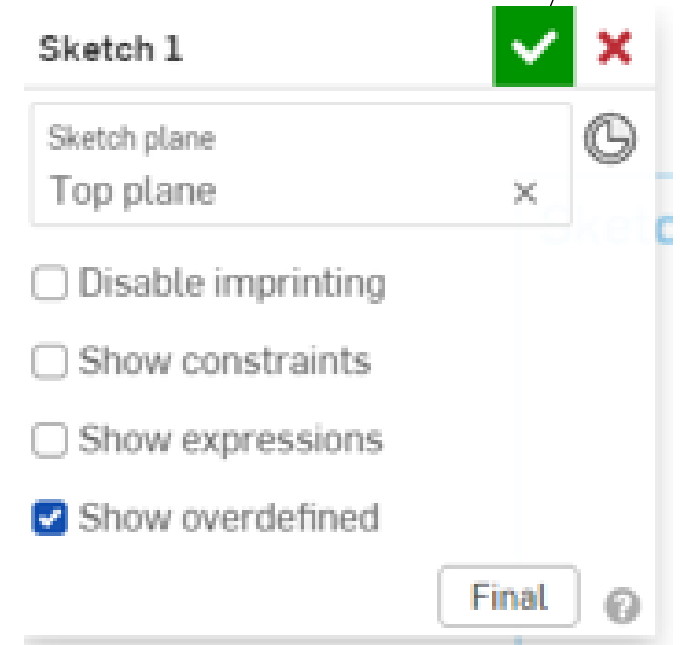
Two thin, dark gray lines intersect diagonally on a light gray background. One line slopes downward from the top-left towards the bottom-right, and the other slopes downward from the top-right towards the bottom-left. They cross each other in the upper-left quadrant of the image.

SKETCHES


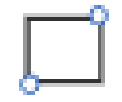


STARTING A SKETCH

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.)



Search tools... alt/~ c

Filter by name or type

Features (6)

- Default geometry
 - Origin
 - Performance
 - Top
 - Front
 - Right
- Sketch 1**
- Extrude 1

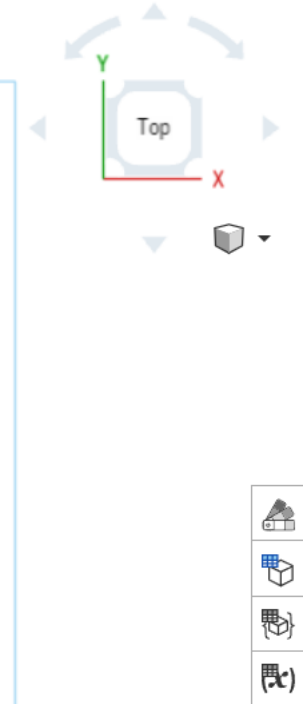
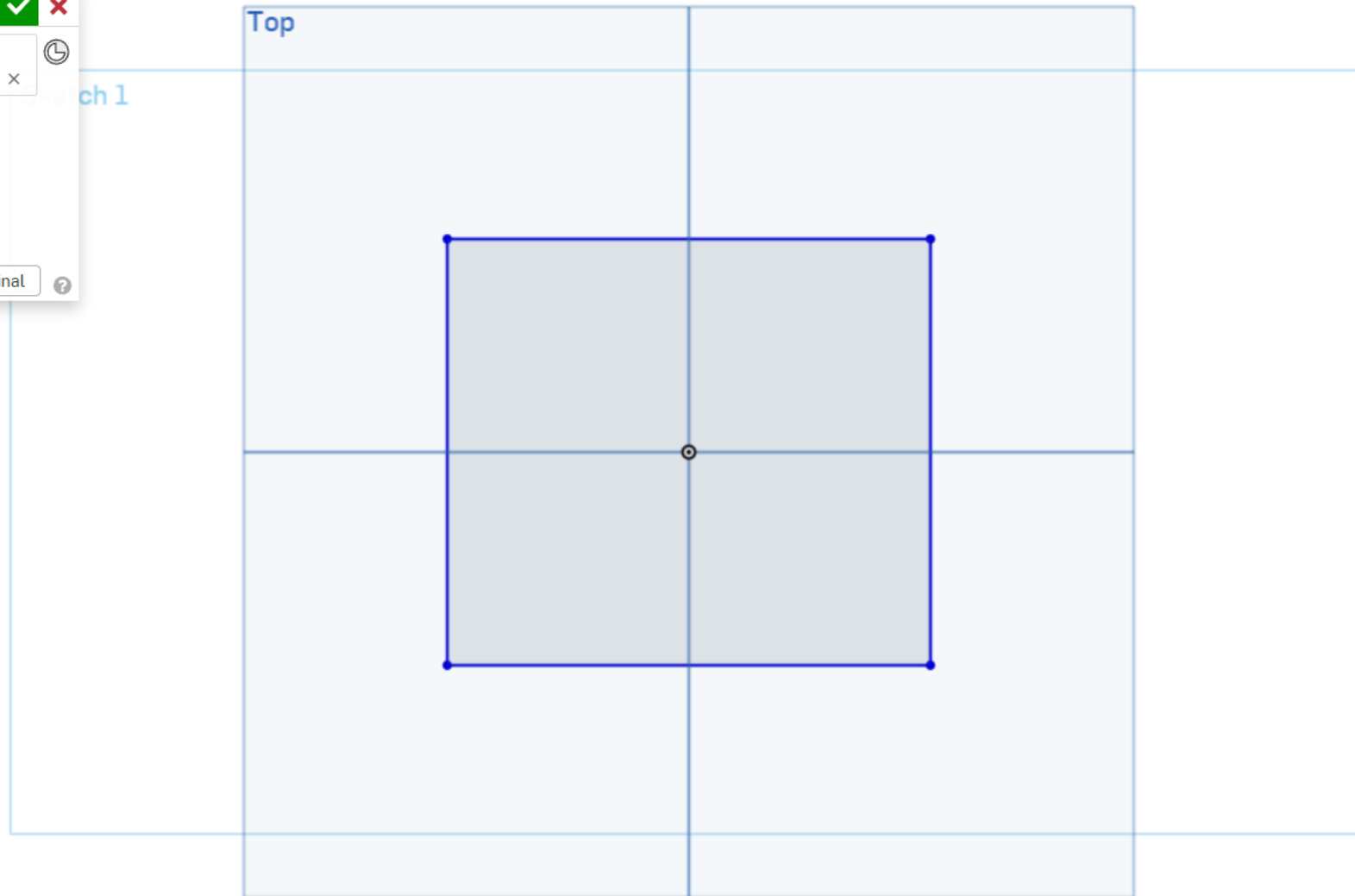
Parts (0)

Sketch 1

Sketch plane
Top plane


- ☐ Disable imprinting
- ☐ Show constraints
- ☐ Show expressions
- ☒ Show overdefined


Final ?





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.

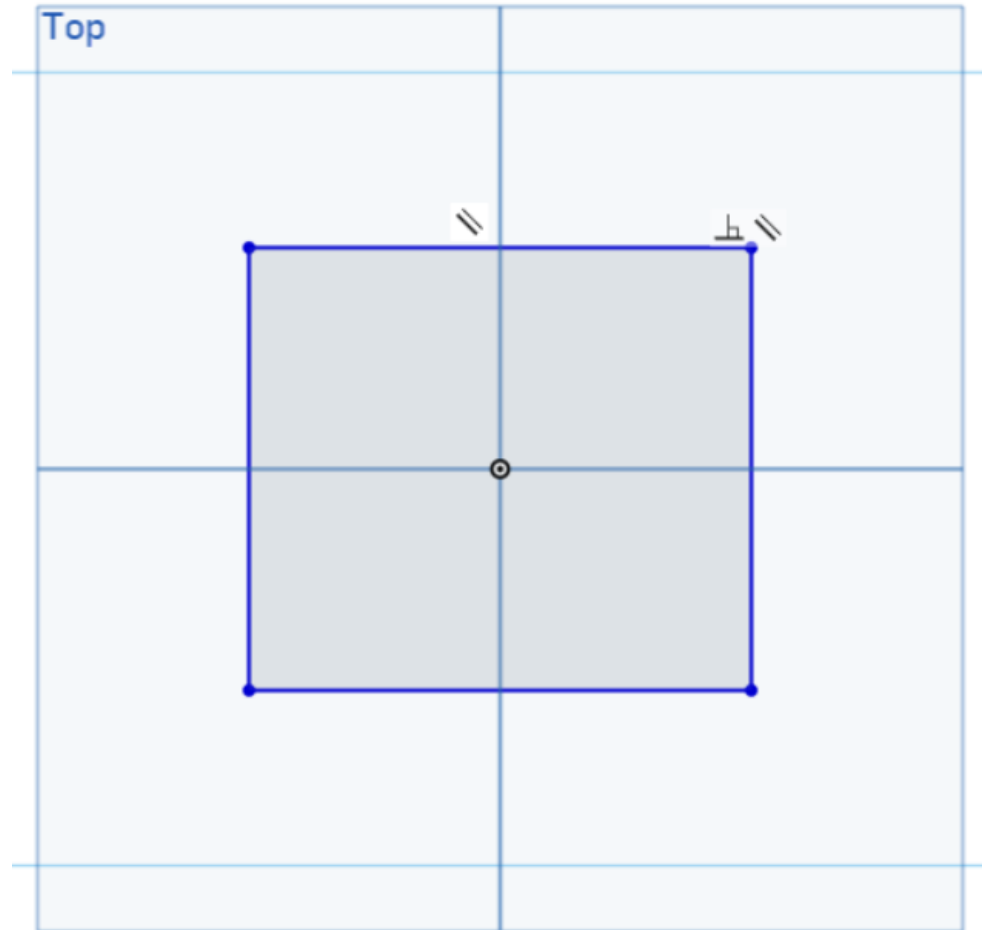
 Parallel. Places the selected items so they remain a constant distance apart from each other.

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

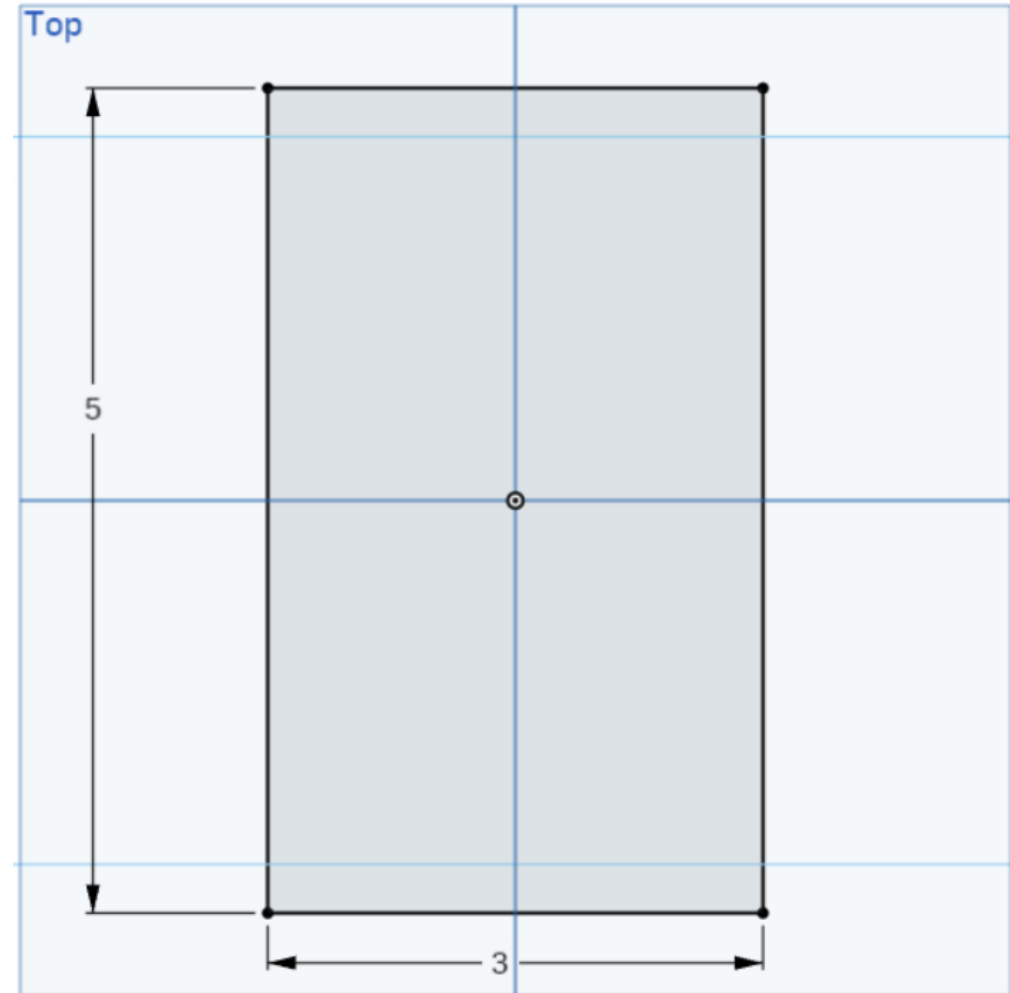
 Tangent. Places the selected items tangent to each other (at least one selection must be a cylindrical, conical, or spherical face).

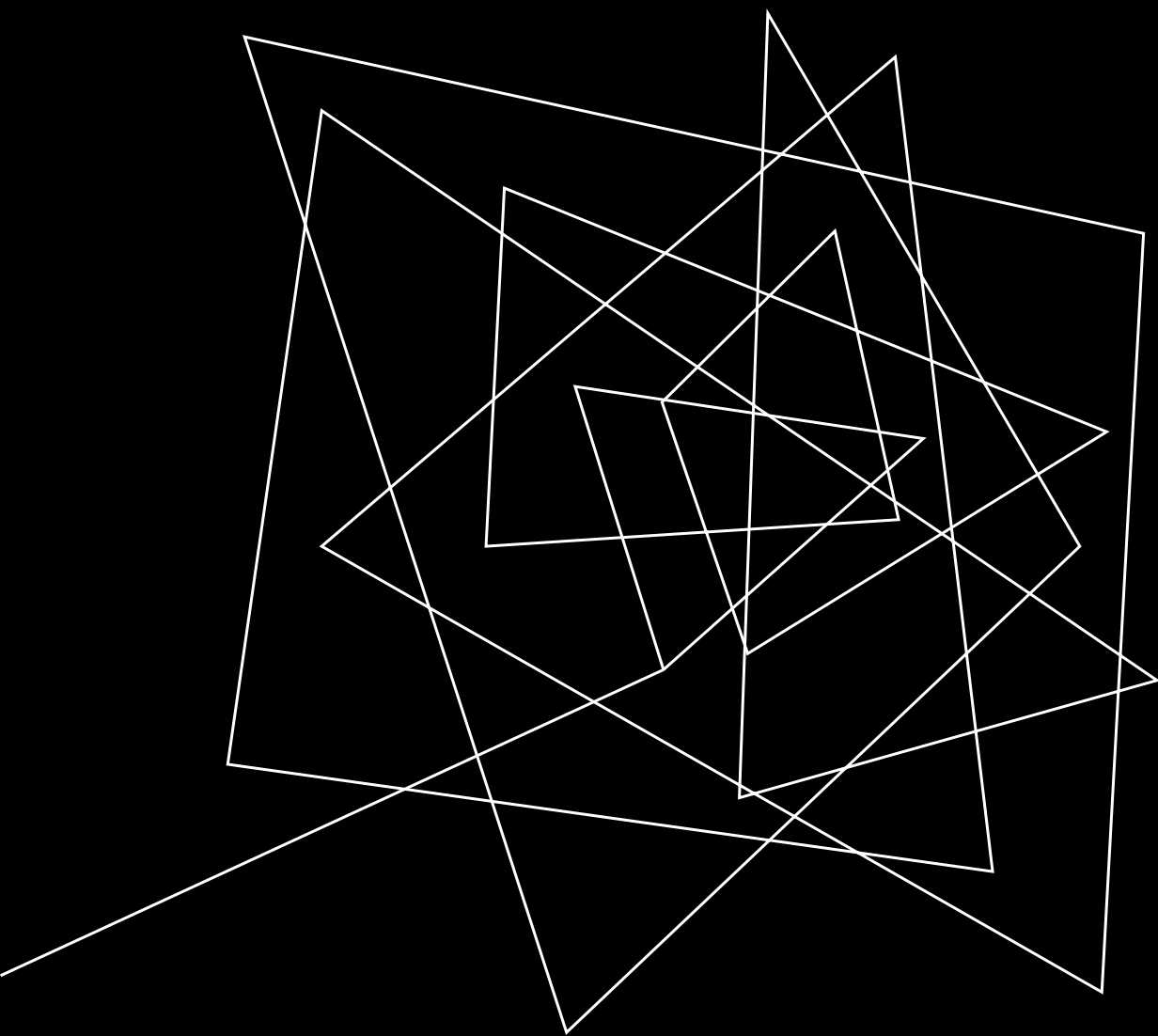
 Concentric. Places the selections so that they share the same center line.

BAD



GOOD





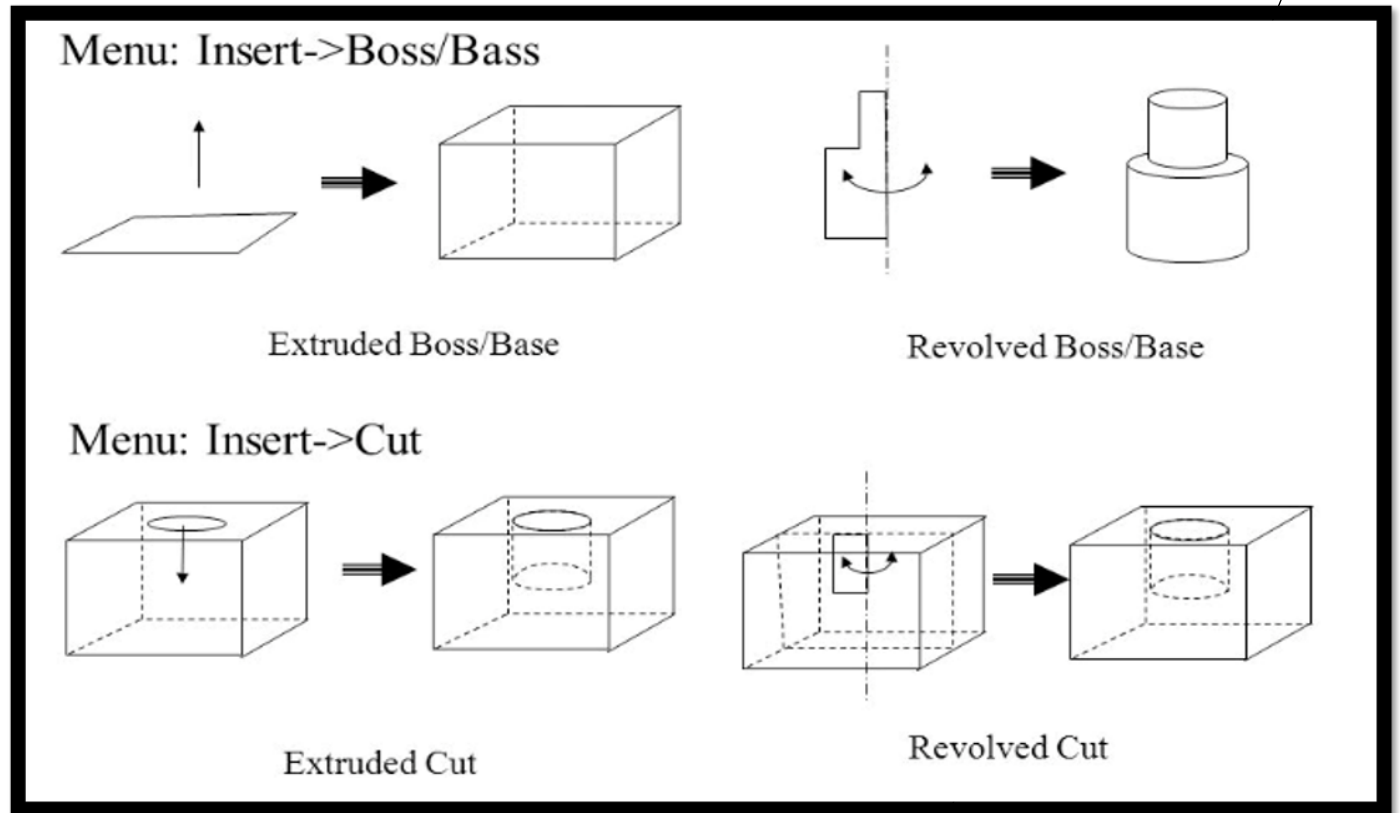
DEMONSTRATIO
N

An abstract graphic featuring two thin, dark grey lines that intersect on a light grey background. One line is oriented diagonally from the top-left towards the bottom-right, while the other is oriented from the top-right towards the bottom-left. To the right of the intersection point, the text "3D PARTS" is displayed in a bold, black, sans-serif typeface.

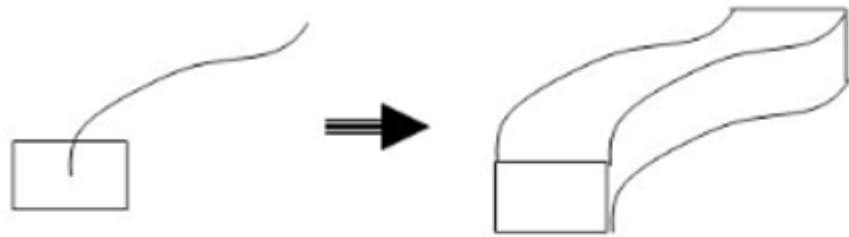
3D PARTS

INTRODUCTION TO FEATURES

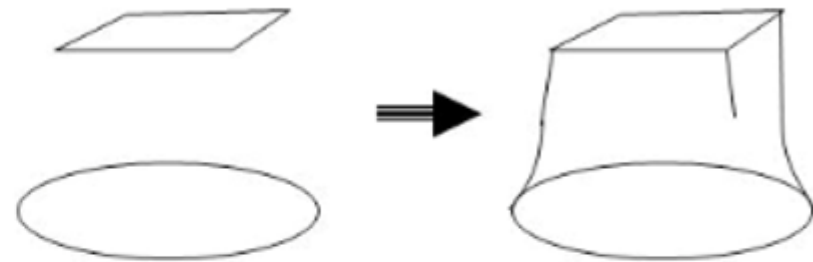
- Features convert 2D sketches to a 3D Part, or modifies existing parts
- Depending on the selection of features, the 2D sketch will be utilized in a different way



Menu: Insert->Boss/Bass



Sweep Boss/Base

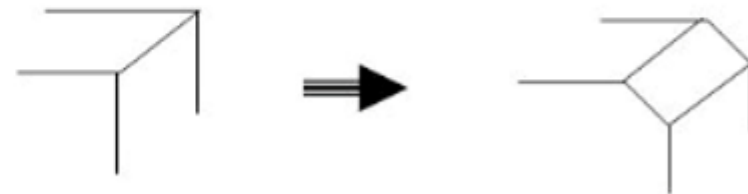


Lofted Boss/Base

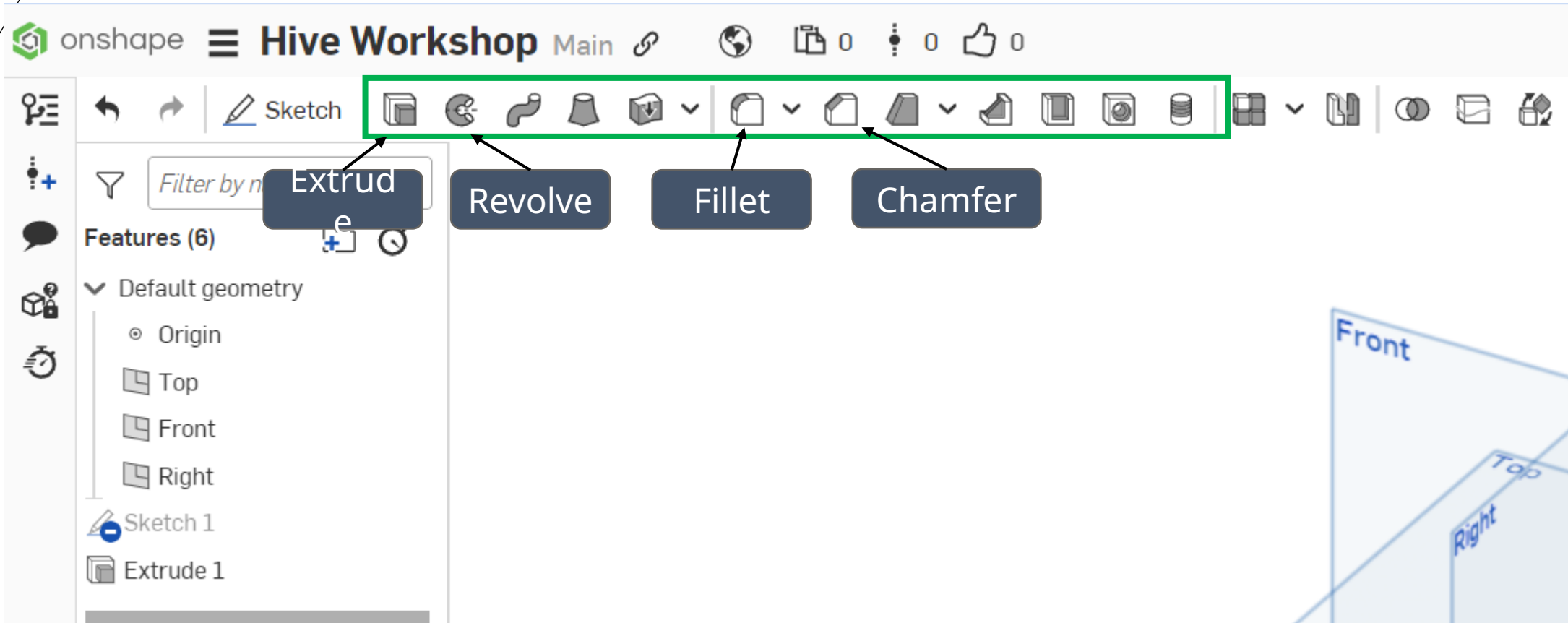
Menu: Insert->Features



Fillet

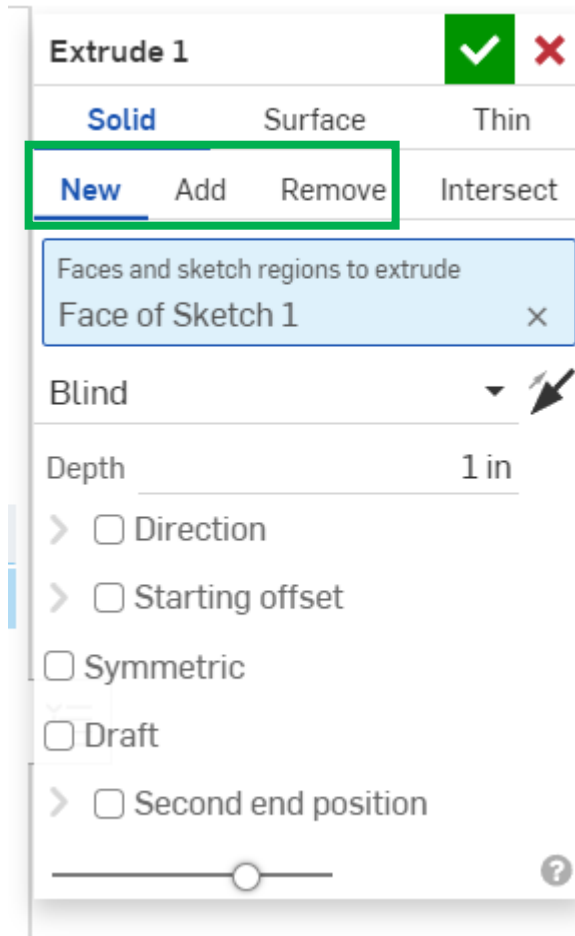


Chamfer



FEATURE MENU

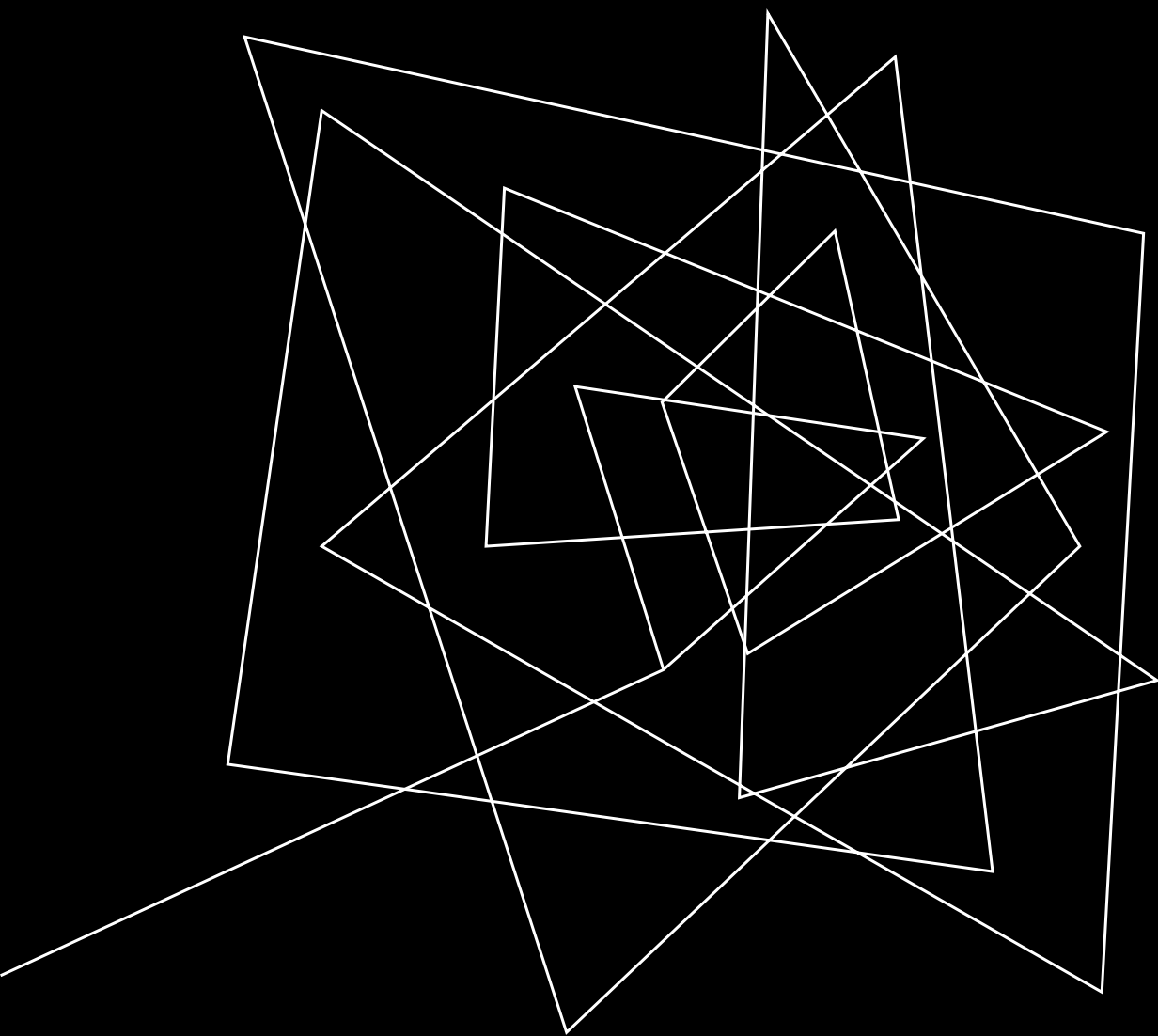
- New: Create a part
- Add: Add feature to an existing part
- Remove (Cut): Cut out feature from existing part



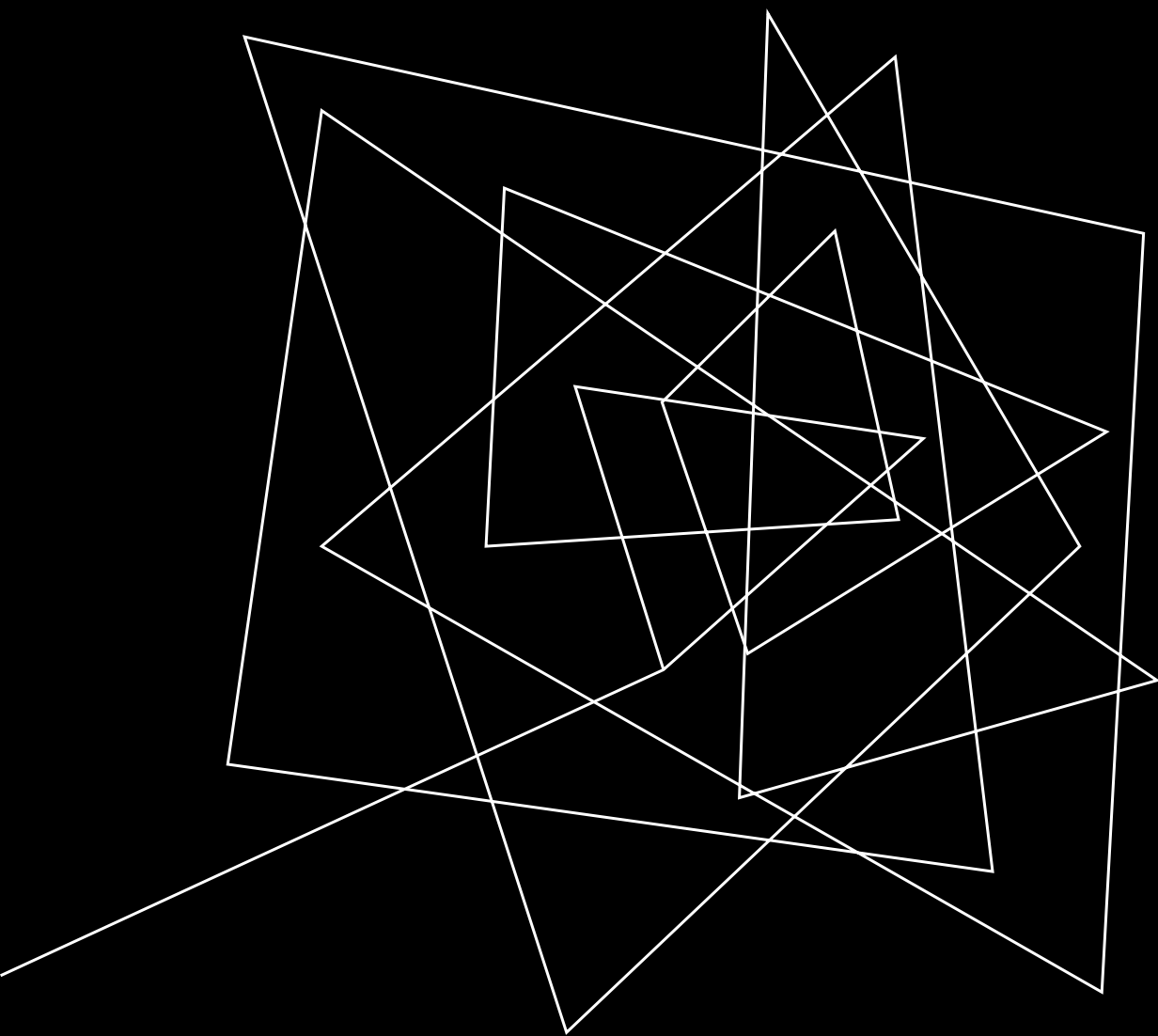
Sketch selection

Type of evaluation

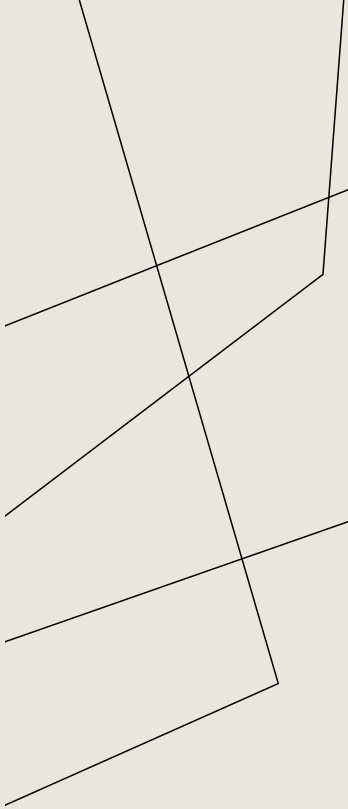
How much to extrude



DEMONSTRATIO
N

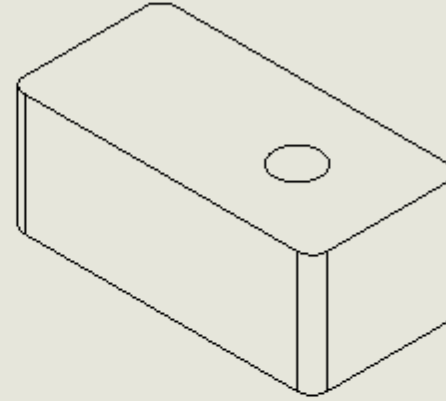
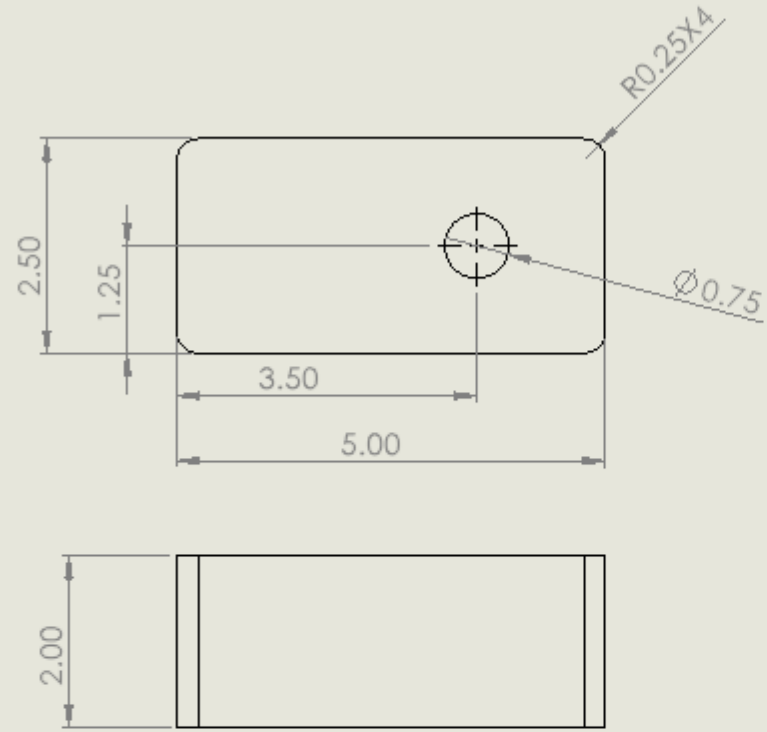


PRACTICE



B

A



B

A

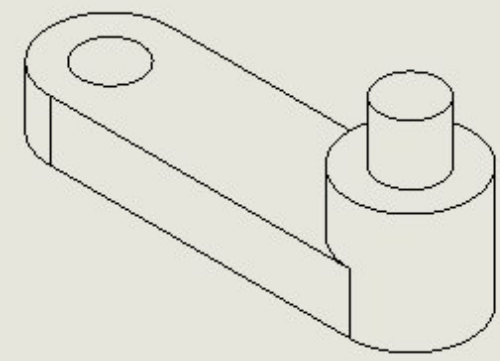
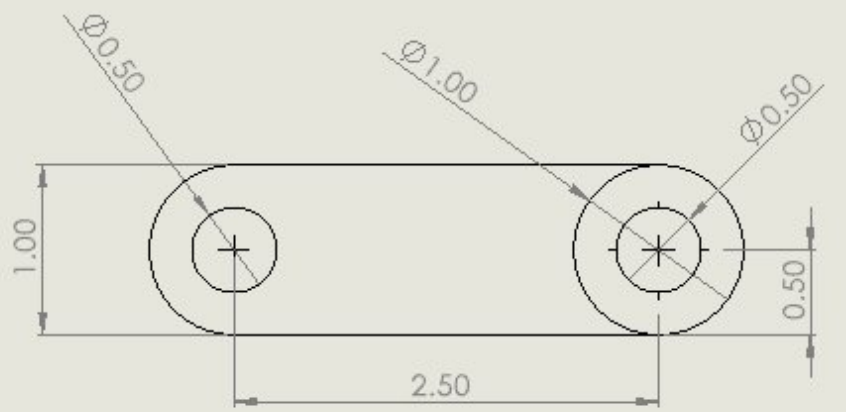
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REPRODUCTION IN PART OR AS A WHOLE
WITHOUT THE WRITTEN PERMISSION OF
<INSERT COMPANY NAME HERE> IS
PROHIBITED.

		UNLESS OTHERWISE SPECIFIED:		NAME	DATE	TITLE: PracticeBase					
		DIMENSIONS ARE IN INCHES							SIZE	DWG. NO.	REV
		TOLERANCES:		DRAWN					A		
		FRACTIONAL \pm		CHECKED							
		ANGULAR: MACH \pm BEND \pm		ENG APPR.							
		TWO PLACE DECIMAL \pm		MFG APPR.							
		THREE PLACE DECIMAL \pm		Q.A.		SCALE: 1:2 WEIGHT: SHEET 1 OF 1					
		INTERPRET GEOMETRIC TOLERANCING PER:		COMMENTS:							
		MATERIAL									
		FINISH									
NEXT ASSY	USED ON										
APPLICATION		DO NOT SCALE DRAWING									

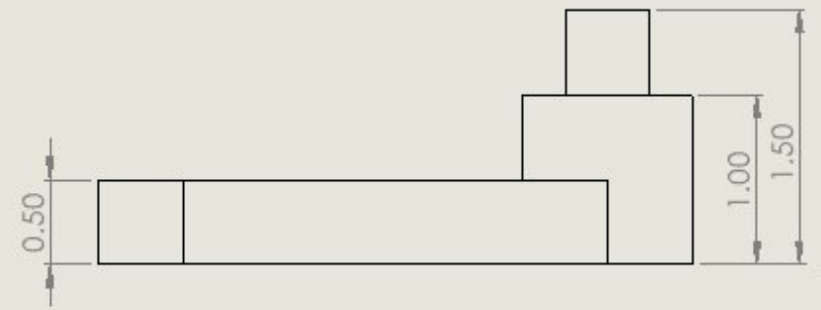
2

1

B



B



A

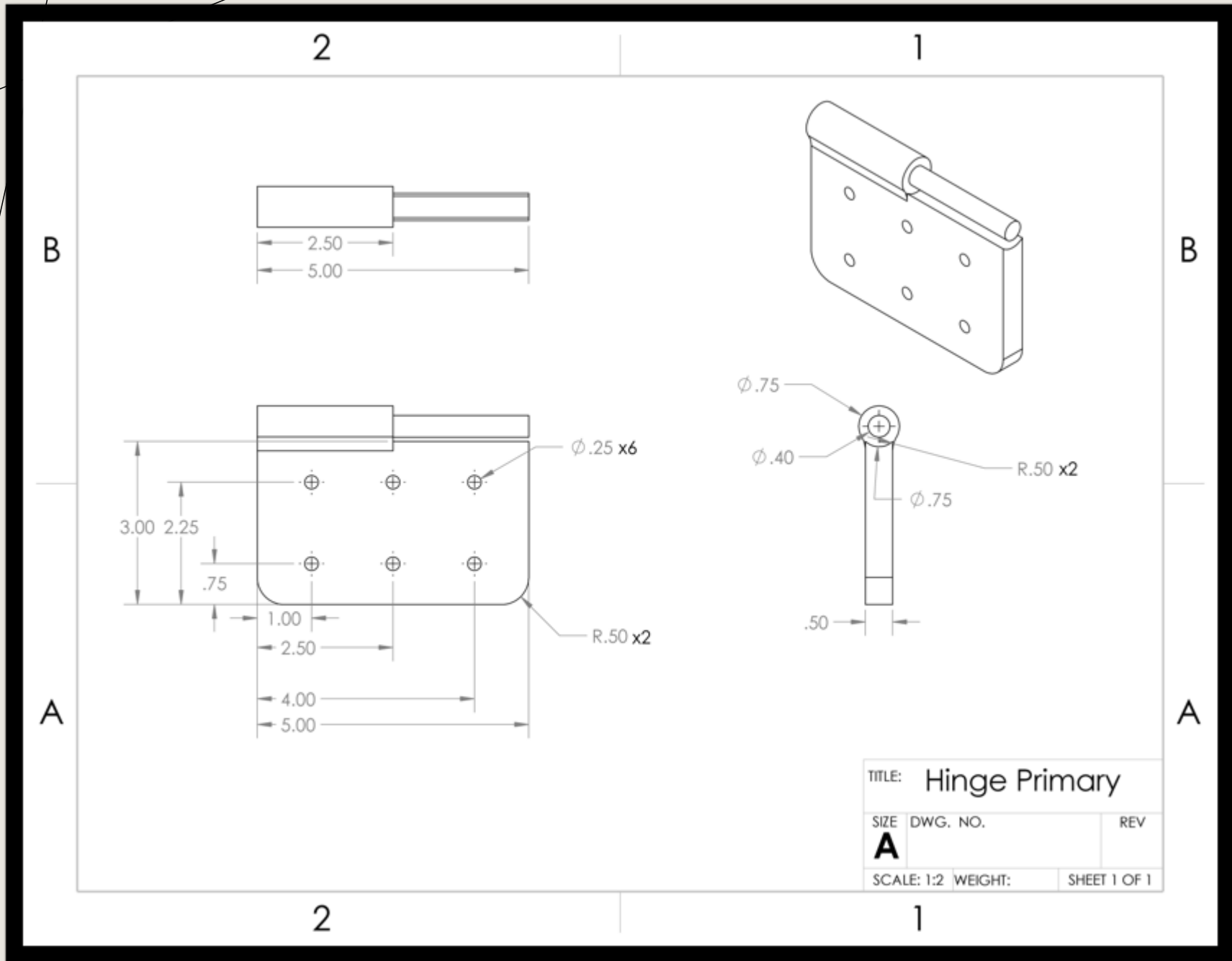
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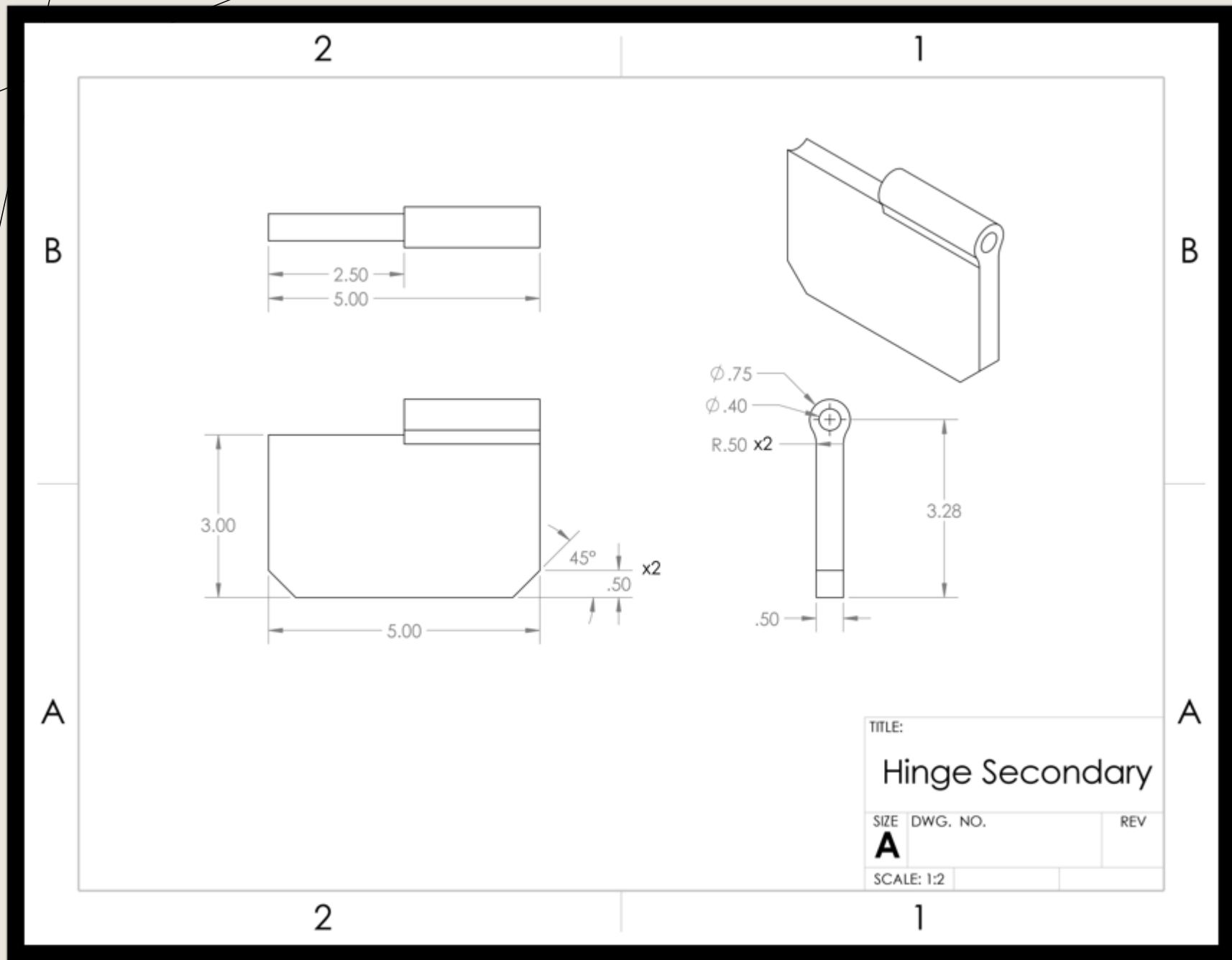
		UNLESS OTHERWISE SPECIFIED:		NAME	DATE		
		DIMENSIONS ARE IN INCHES		DRAWN			
		TOLERANCES:		CHECKED		TITLE:	
		FRACTIONAL \pm		ENG APPR.		PracticeHandle	
		ANGULAR: MACH \pm BEND \pm		MFG APPR.			
		TWO PLACE DECIMAL \pm		Q.A.			
		THREE PLACE DECIMAL \pm		COMMENTS:			
		INTERPRET GEOMETRIC				SIZE	DWG. NO.
		TOLERANCING PER:				A	REV
		MATERIAL					
		FINISH					
NEXT ASSY	USED ON						
APPLICATION		DO NOT SCALE DRAWING				SCALE: 1:1	WEIGHT:
						SHEET 1 OF 1	

A

2

1



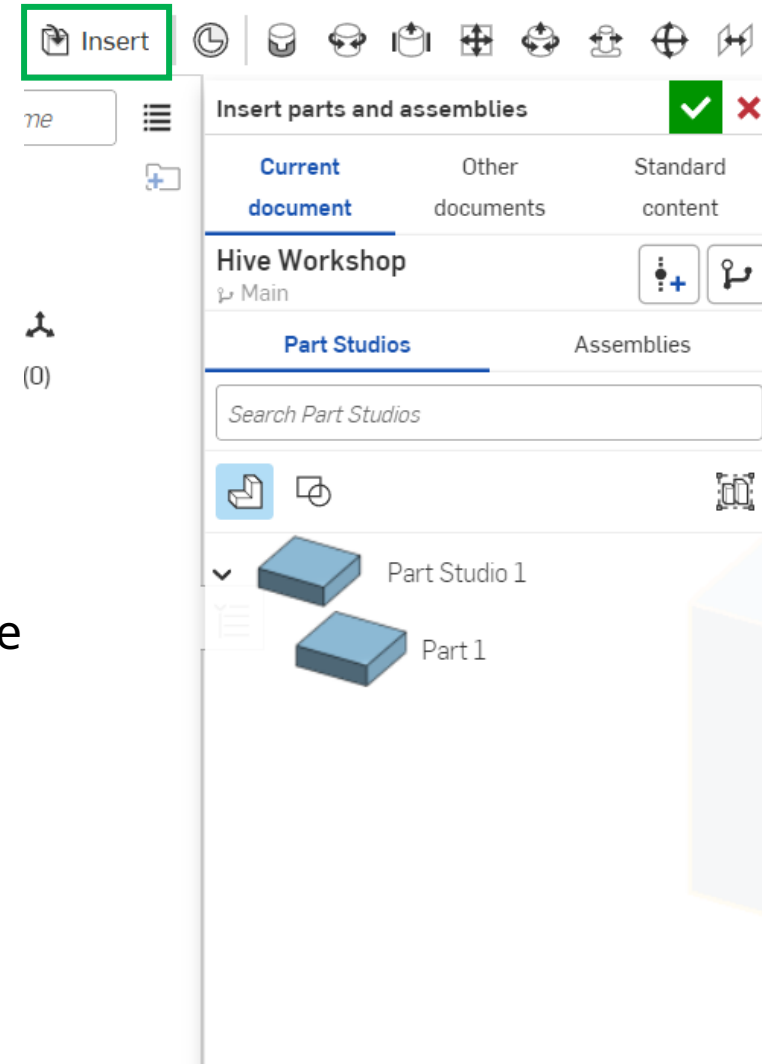


An abstract graphic featuring two thin, dark grey lines that intersect on a light grey background. One line runs diagonally from the top-left towards the bottom-right, while the other runs from the top-right towards the bottom-left. The word "ASSEMBLIES" is positioned to the right of the intersection point, rendered in a bold, black, sans-serif typeface.

ASSEMBLIES

CREATING AN ASSEMBLY

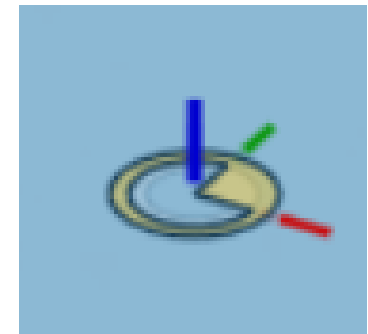
- From the bottom file menu, click “Assembly 1”
- To Insert a part, click “Insert”, and you should see screen similar to the one on the right



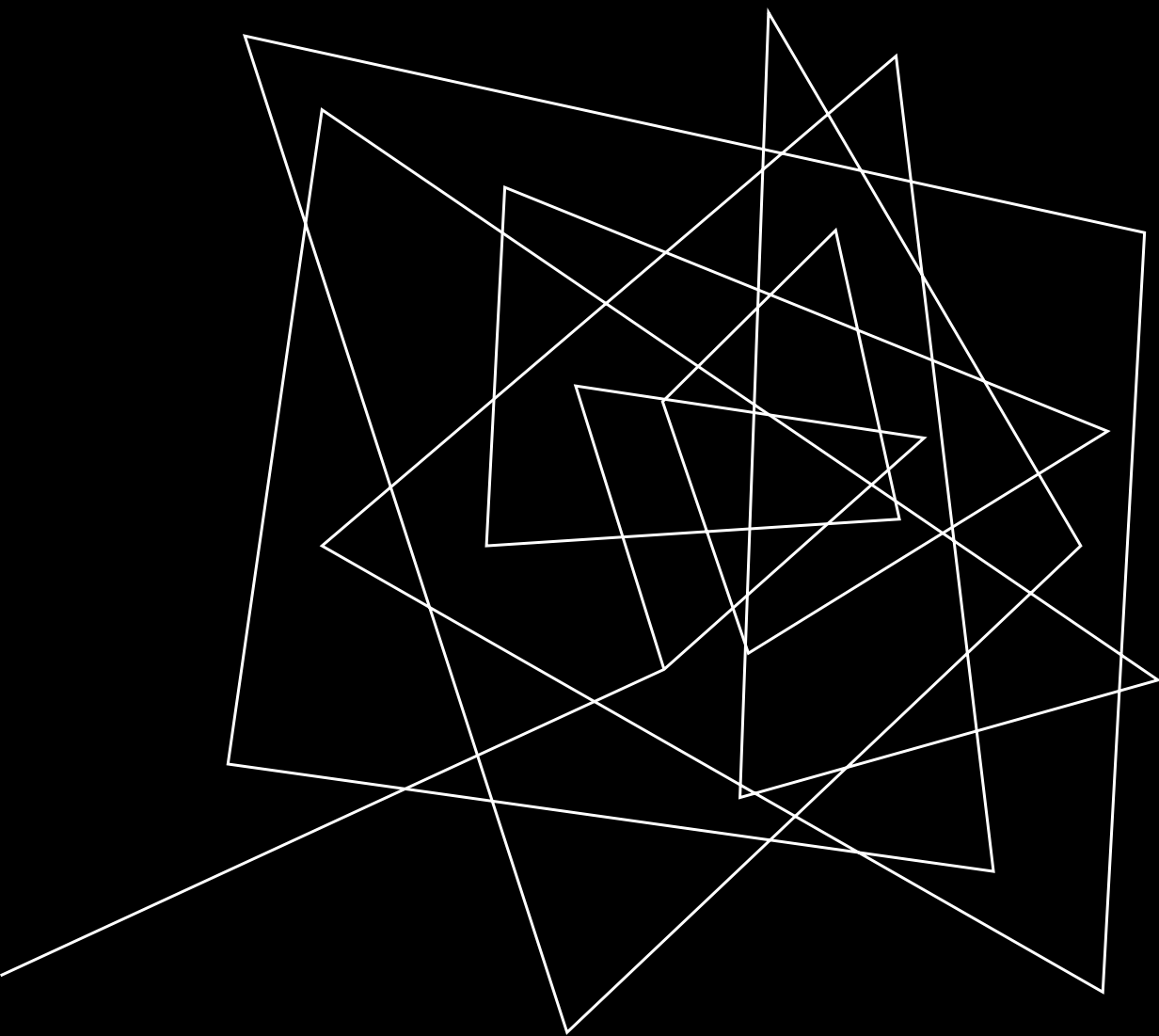
HOW TO DEFINE PARTS TOGETHER?

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

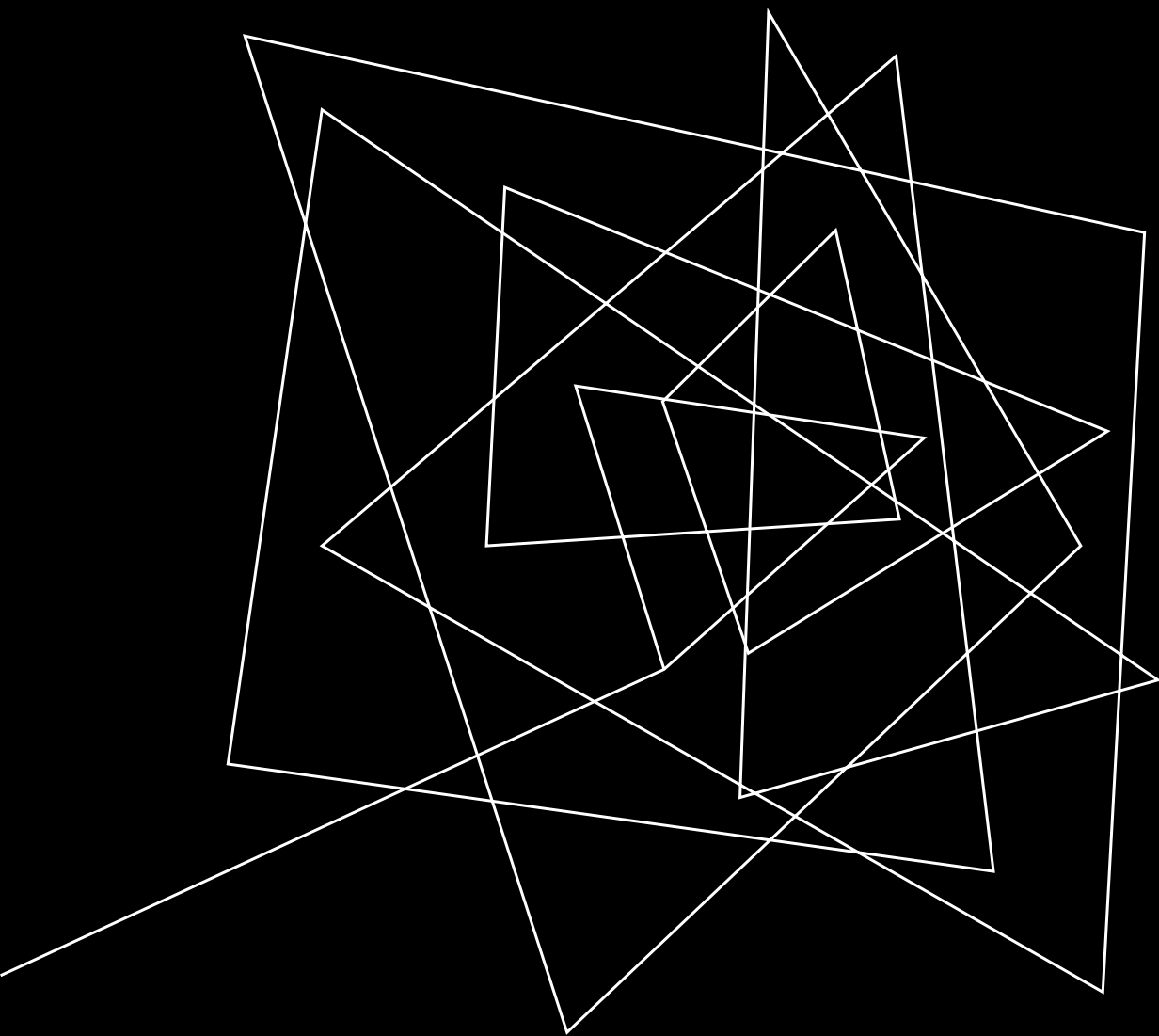


Mate Connector:
A point on part in
which a mate is
taken



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

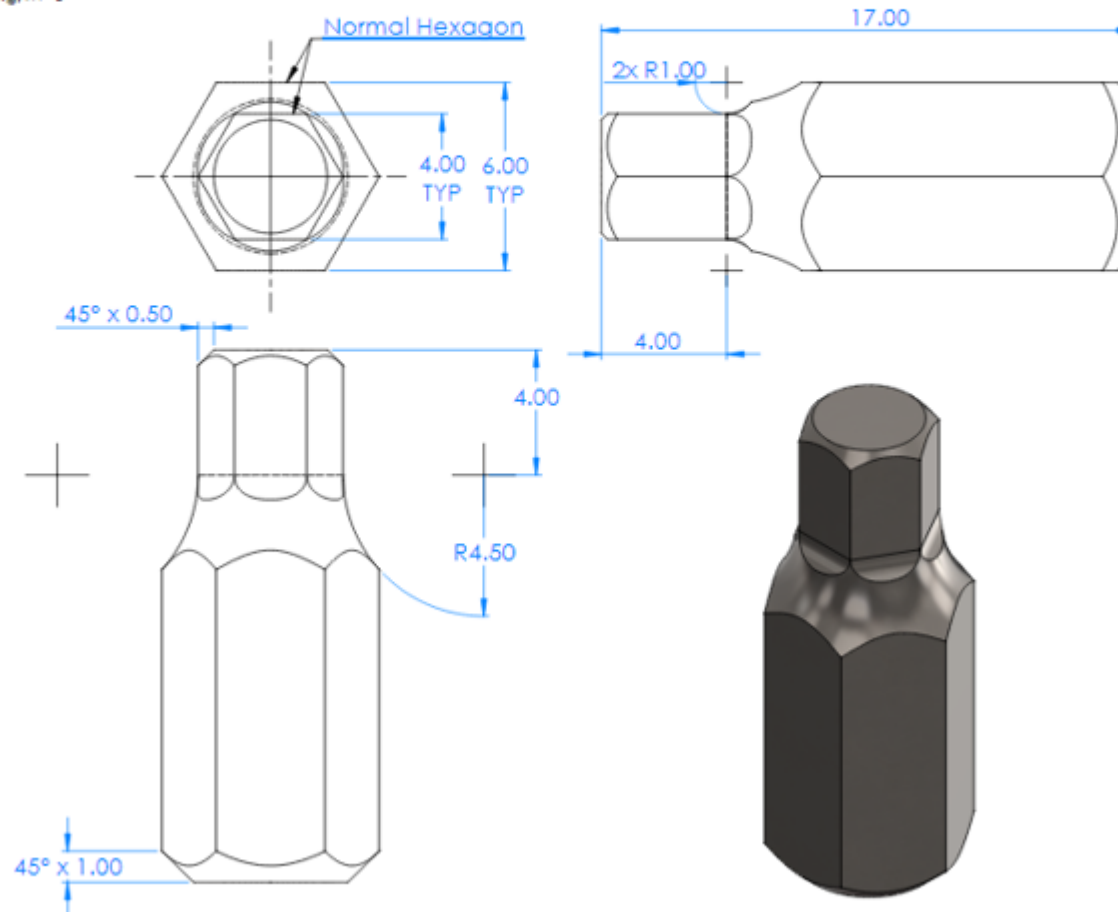
Decimal Places: 2

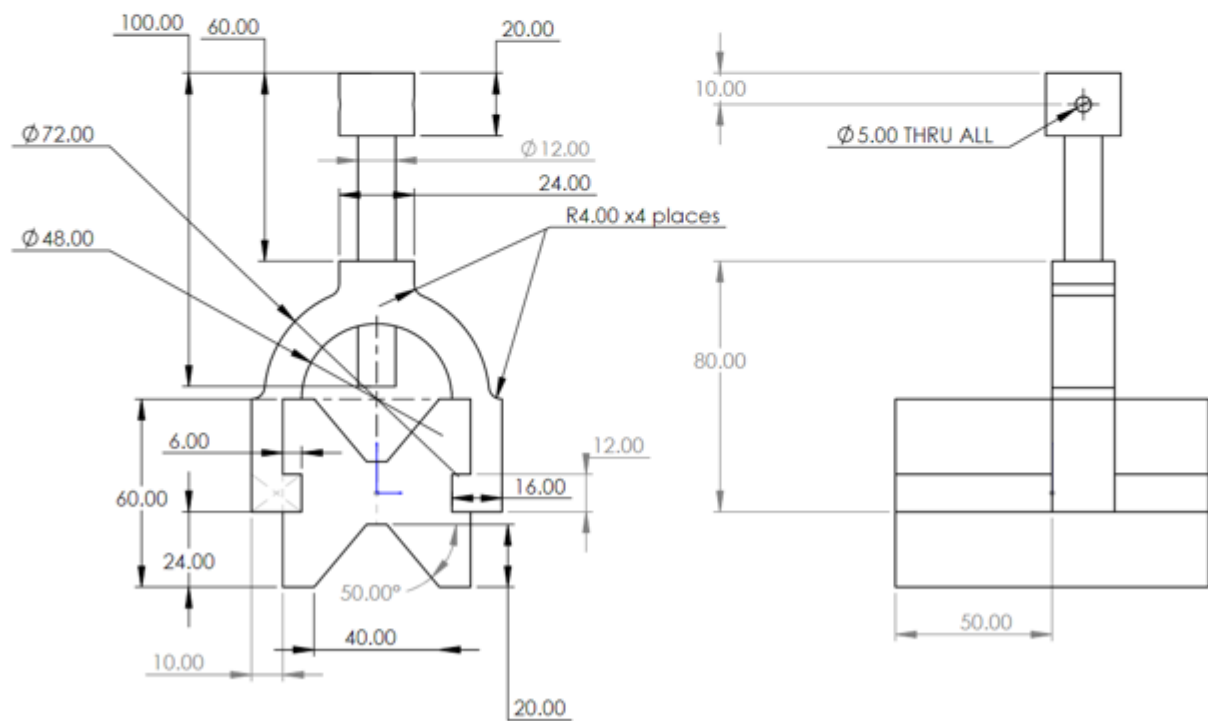
Part Origin: Arbitrary

Material Type: Steel

Material: Stainless Steel (ferritic)

Density: 7800 kg/m³





A series of white, thin, overlapping geometric lines on a black background, forming a complex, abstract shape on the left side of the slide.

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!