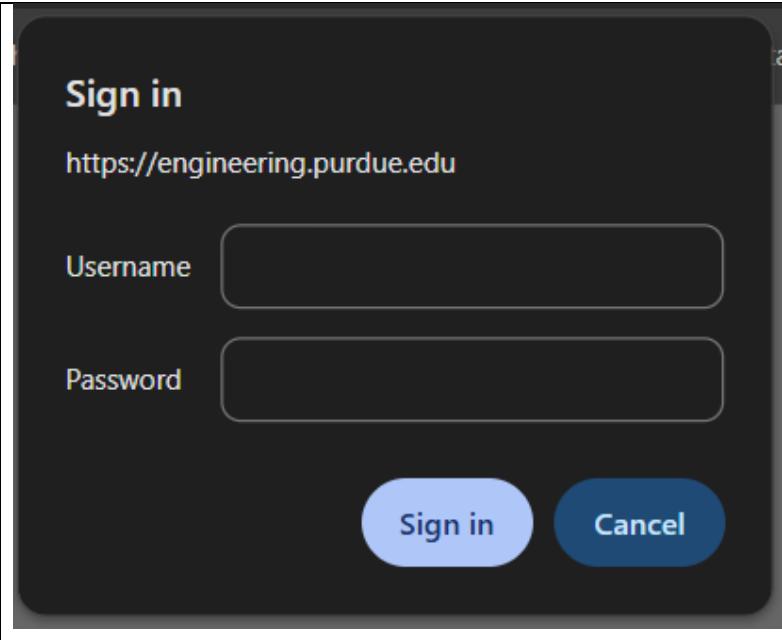


CAD (SolidWorks) Onboarding Project

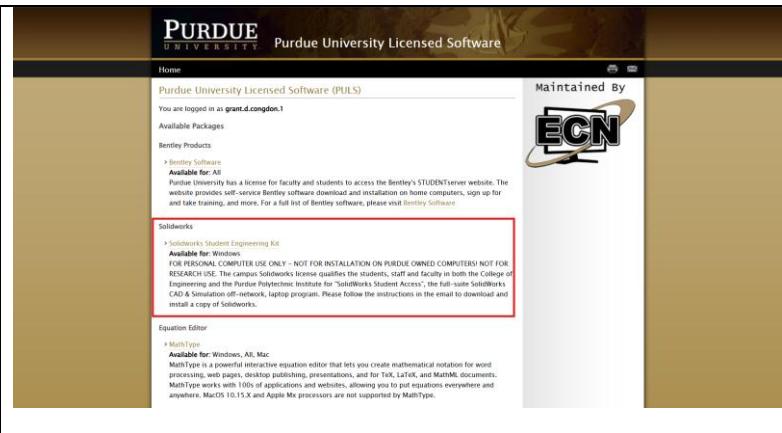
Installation

1. Navigate to engineering.purdue.edu/PULS

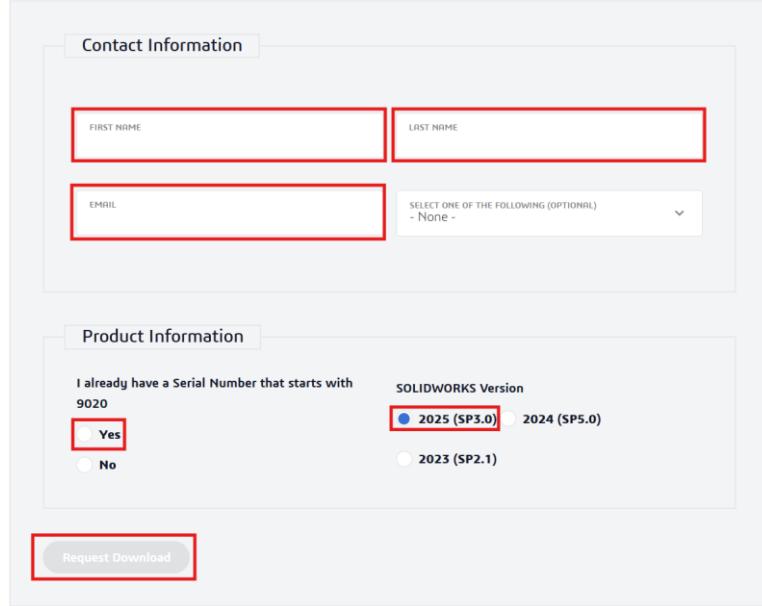
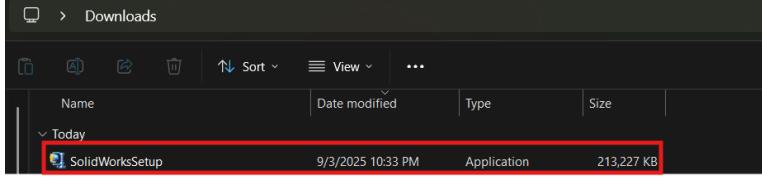


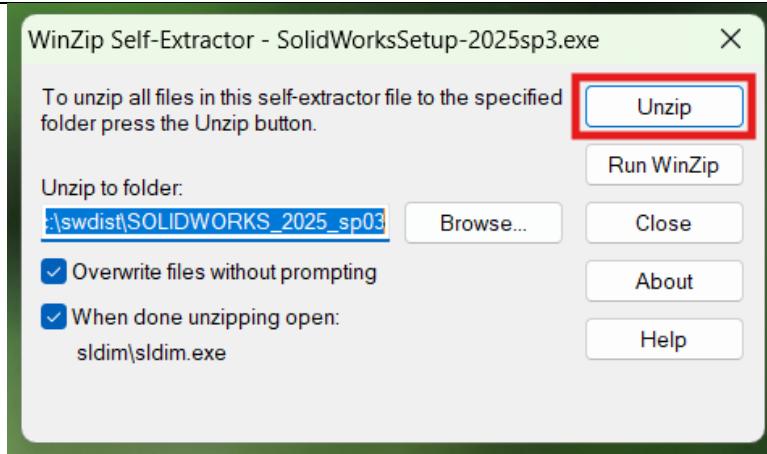
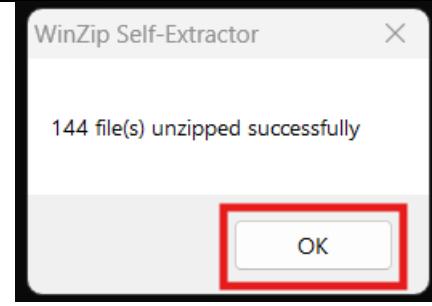
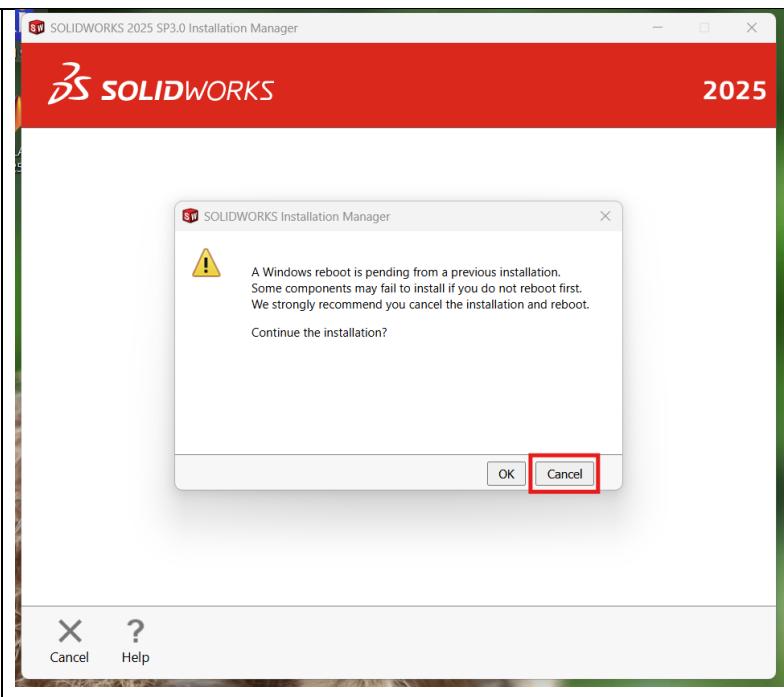
2. Enter your Purdue username and password

3. Navigate to the most recent version of SolidWorks

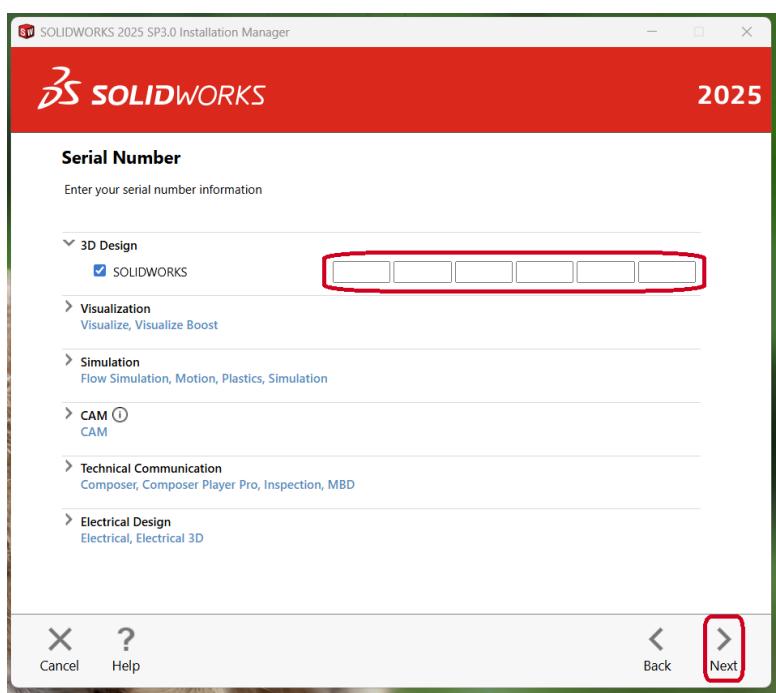
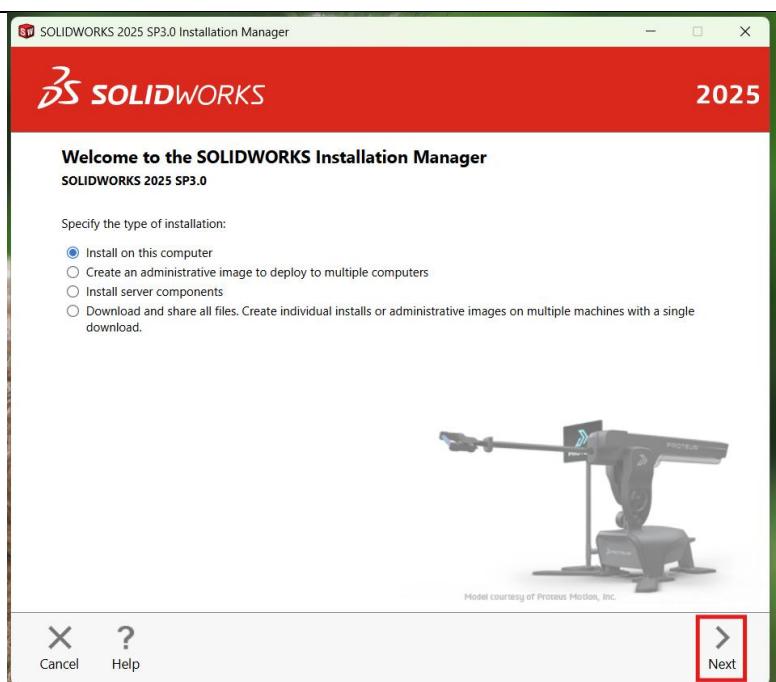
A screenshot of the "Purdue University Licensed Software (PULS)" website. The header reads "PURDUE UNIVERSITY Licensed Software". The main content area shows a list of software packages. A red box highlights the "Solidworks" section, which includes a link to the "Solidworks Student Engineering Kit". Below this, there is a note about the license being for "STUDENT USE ONLY - NOT FOR INSTALLATION ON PURDUE OWNED COMPUTERS! NOT FOR RESEARCH USE." To the right of the main content, there is a sidebar with the text "Maintained By ECN" and the ECN logo.

<p>4. Enter your Purdue email, keep the number of planned installations at 1 and click “Download”</p> <p>NOTE: This will download a SolidWorks installer – DO NOT USE THIS. Follow the steps listed below</p>	
<p>5. Navigate to your Purdue email, and click on the quarantine notification; this may be in your spam/junk folder</p>	<div style="background-color: #e0f2f1; padding: 10px;"> <input type="checkbox"/> quarantine@messaging.microsoft.com Microsoft 365 security: ... 1:49 AM External Email: Use caution with a... </div>
<p>6. Release the email from software@ecn.purdue.edu</p>	

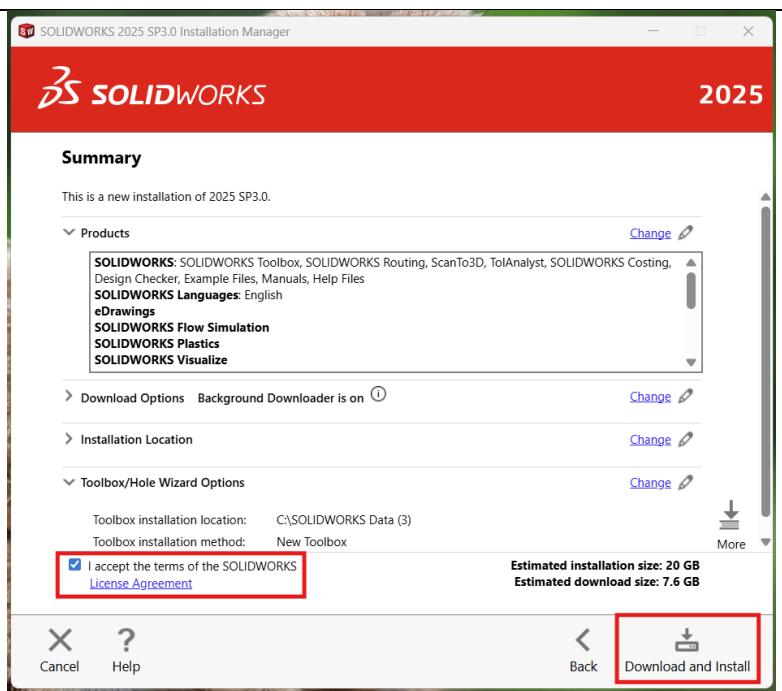
7. Click on the long link	<p>Purdue qualifies for SolidWorks Student Access, the full-suite SolidWorks CAD & Simulation off-network, laptop program.</p> <p>THIS SOFTWARE IS NOT TO BE USED FOR ANY RESEARCH PURPOSES!</p> <p>Download Instructions</p> <ol style="list-style-type: none"> 1. Uninstall any/all previous version of SOLIDWORKS 2. Go to https://nam04.safelinks.protection.outlook.com/?url=https%3A%2F%2Fwww.solidworks.com%2FSEK&data=05%7C02%7Cgongdon%40purdue.edu%7C73651f2bde374e156d4408ddea58210%7C4130bd397c53419cb1e58758d6d3f21%7C1%7C0%7C638925042613891257%7CUnknown%7C1WFpbGZsb3d8eyJfbXB0eU1hcGkiOnRydWUsIYiOiwLjAuMDAwMClsIAiOijXaW4zMislkFOljoiTWFpbClslldUiIjgQ%3D%3D%7C0%7C%7C&sdata=P1P75%2BcDk5yPl6cN3h%2BBqXCLDMl6mn8IEldYIlgVn2Q%3D&reserved=0 3. Enter all relevant information into the form 4. Choose 2025 (SP3.0) for the most recent version of SOLIDWORKS 5. Download and run the SOLIDWORKS installer. Your Unique Download Serial Number is
8. Enter your information a. Ensure you use your Purdue email b. Indicate you have a serial number c. Select the 2025 version of SolidWorks d. Click “Request Download”	
9. Click “Accept and Continue”	<h3>SOLIDWORKS Community Access Initiative</h3> <p>Dassault Systems SolidWorks Community Access Eligibility and Terms of Use Agreement (Please read this agreement and ACCEPT or DECLINE its terms and conditions at the bottom of the page.)</p> <p>"Community Access", "Student Access" or "Student Access product" refers to Student Standard (formerly known as the Student Design Kit/SDK), Student Premium (formerly known as the Student Engineering Kit/SEK), or 60-Day Trial version of SOLIDWORKS software.</p> <p>Eligibility for SOLIDWORKS Community Access Licenses:</p> <p>SOLIDWORKS Community Access licenses available at this website are provided only to: Students and Educators at academic institutions that have been authorized by Dassault Systems SolidWorks Corporation ("DS SolidWorks") to offer SOLIDWORKS Community Access; and Students and Educators who have received Student Access for trial purposes. SOLIDWORKS Community Access also includes military and sponsored organizations.</p> <p>I decline Accept and Continue (The 'Accept and Continue' button is highlighted with a red box)</p>
10. Double click the downloaded file and unzip it	

	
11. You should see a popup like this once the file is unzipped, click "Ok"	
12. Allow the installer to make changes to your computer	
13. If you receive this popup, cancel installation and reboot your computer then return to step 10 , otherwise proceed to the next step	

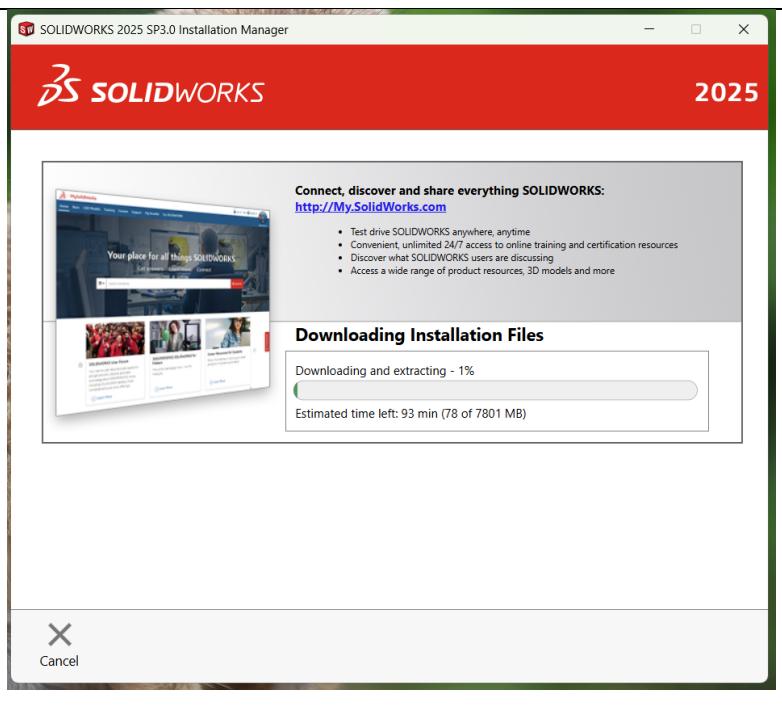
14. Follow the default installation instructions
- “Install on this computer”
 - Enter the serial numbers found in the email from [step 7](#)
 - Click “Next”

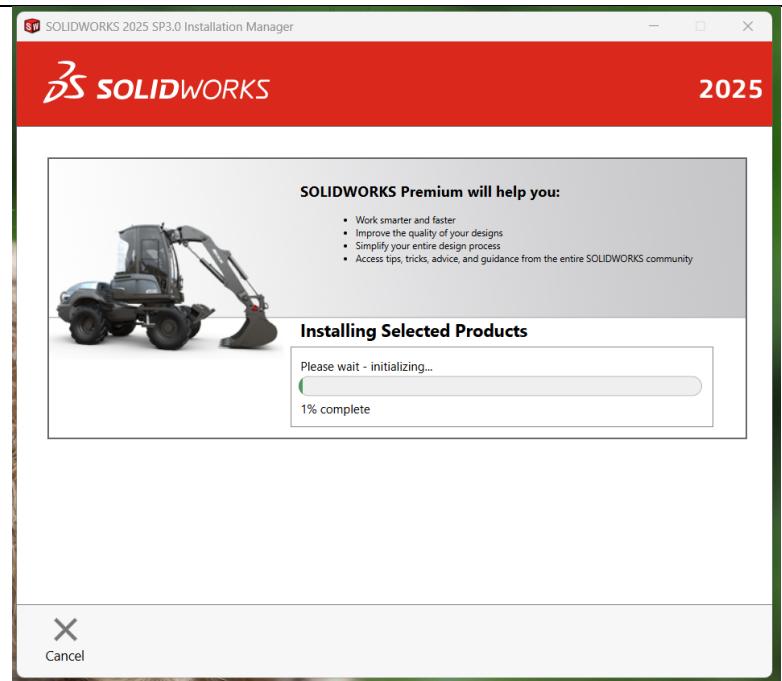


15. Agree to the terms of service and click “Download and Install”

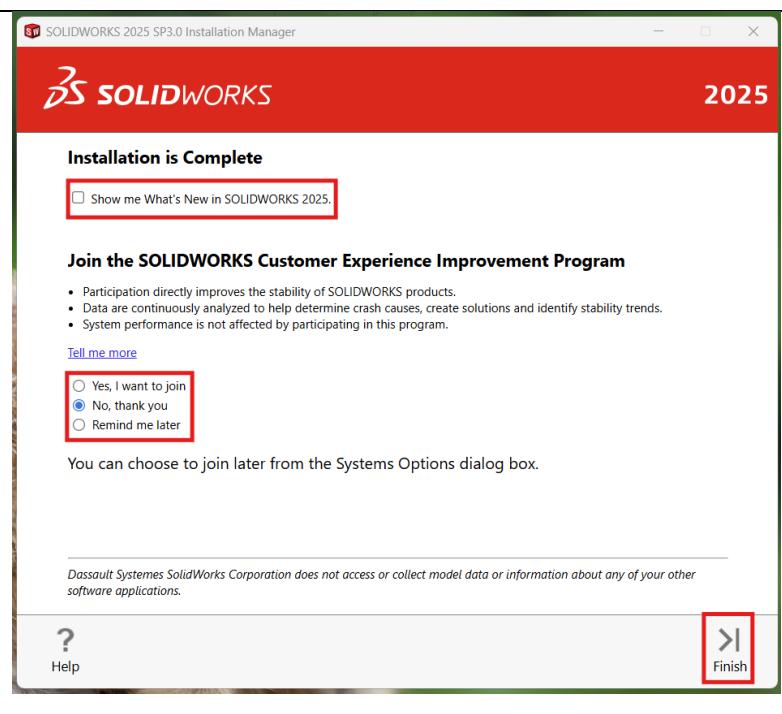


16. Wait for download and installation to finish

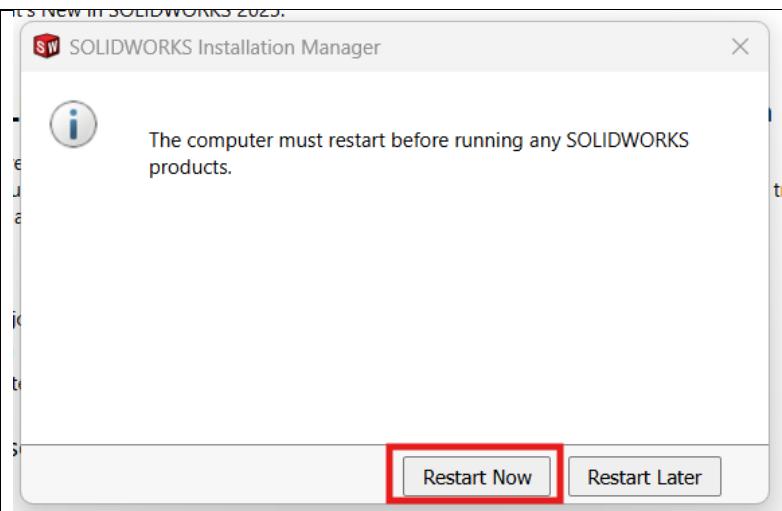




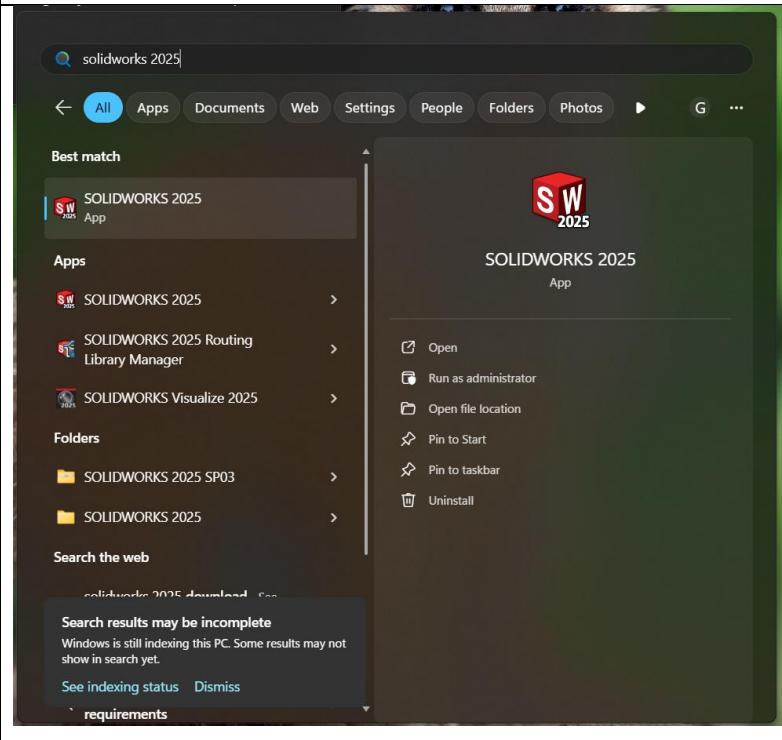
17. Fill out the post-installation form
- Uncheck the box to show what's new in SOLIDWORKS 2025
 - Decide whether you want to be a part of the customer
 - Click "Finish"



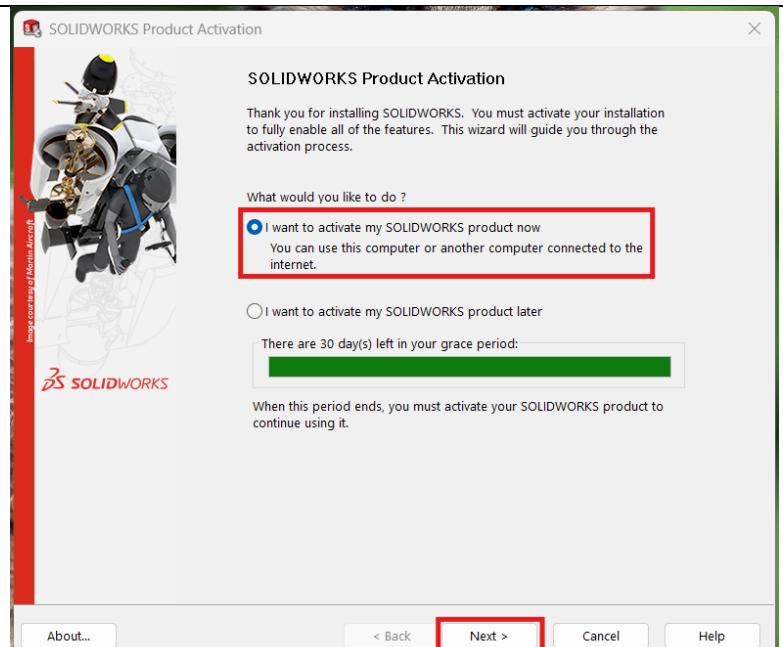
18. Restart your computer



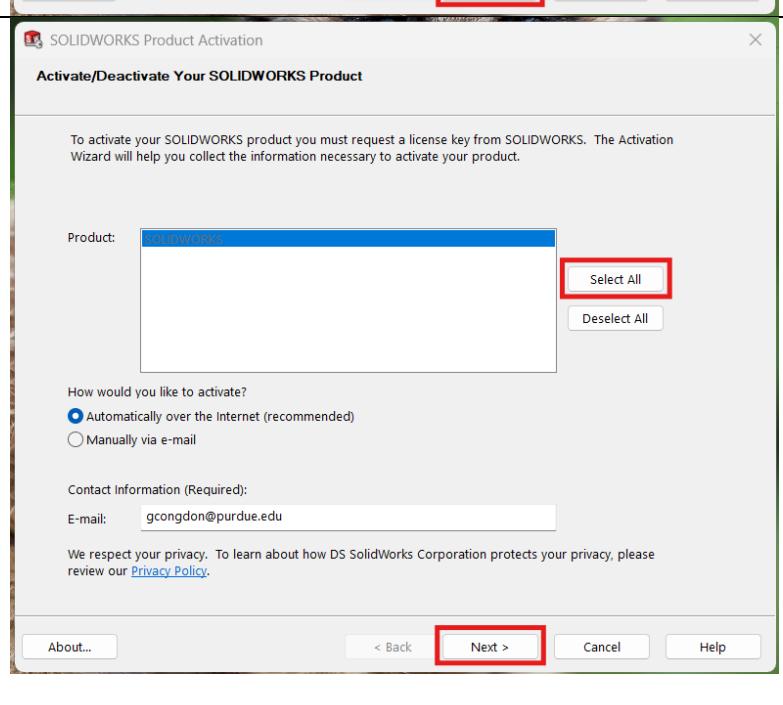
19. Open SolidWorks 2025



20. Select the option to activate your SolidWorks product now then click "Next"

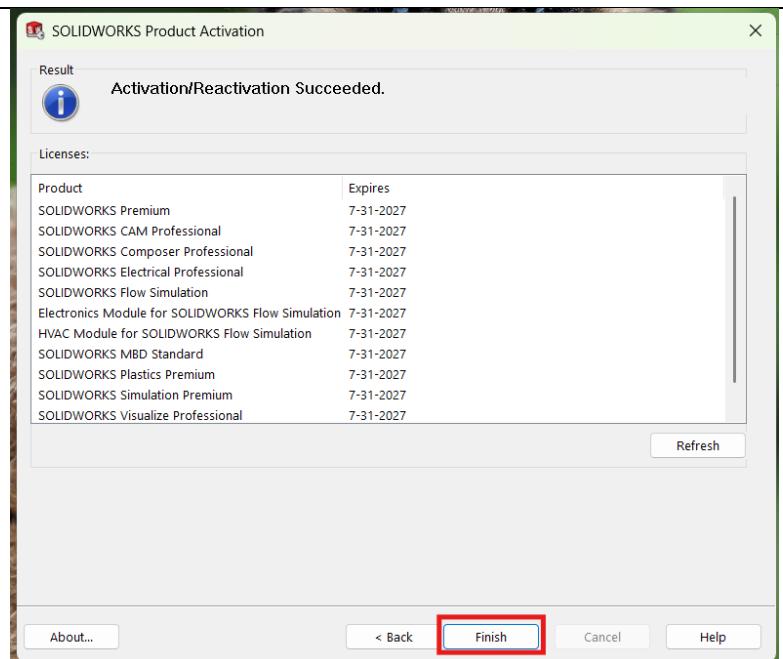


21. Click "Select All" then "Next"

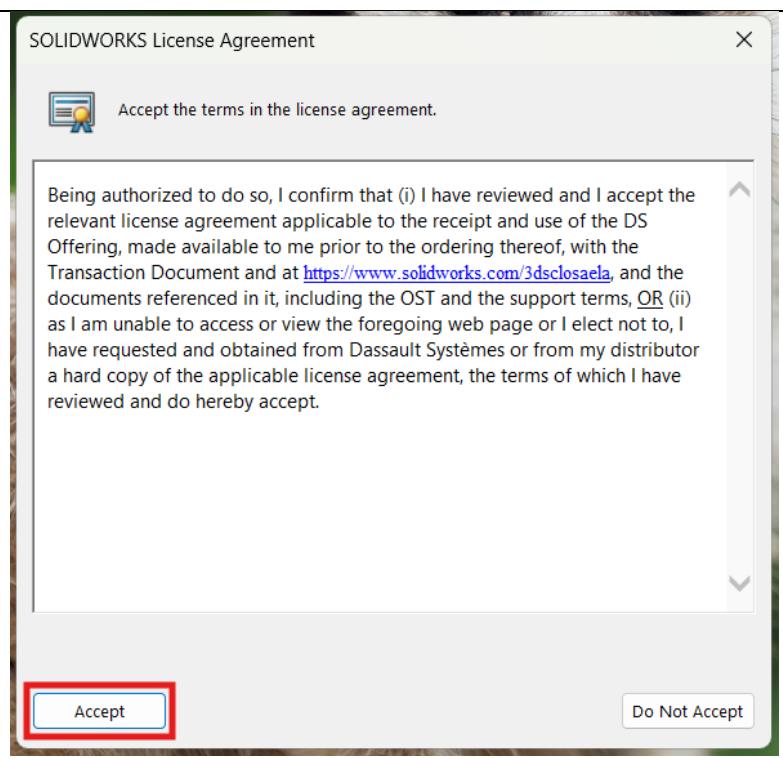


22. Click “Finish”

Note: If you don't see this screen, ask an exec member for help

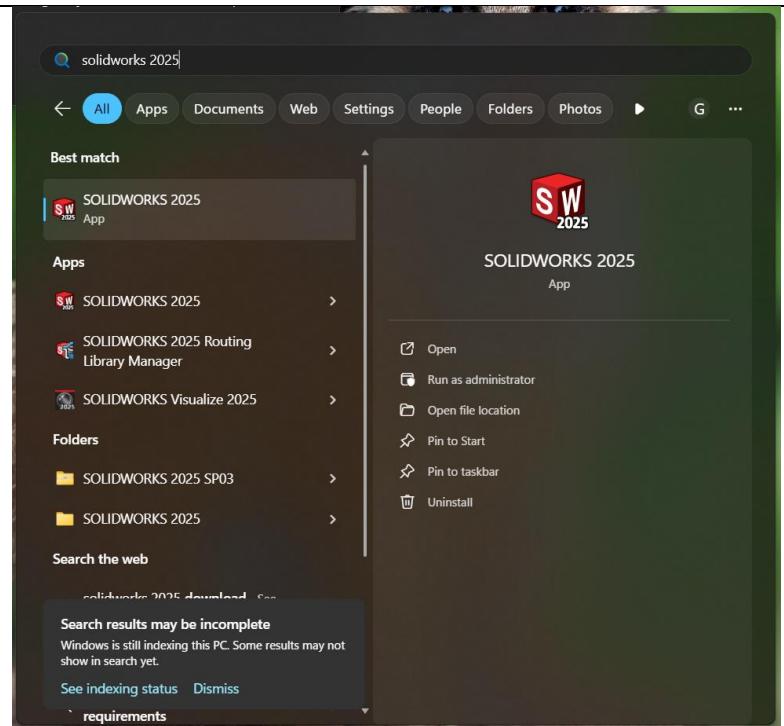


23. Accept the terms in the license agreement

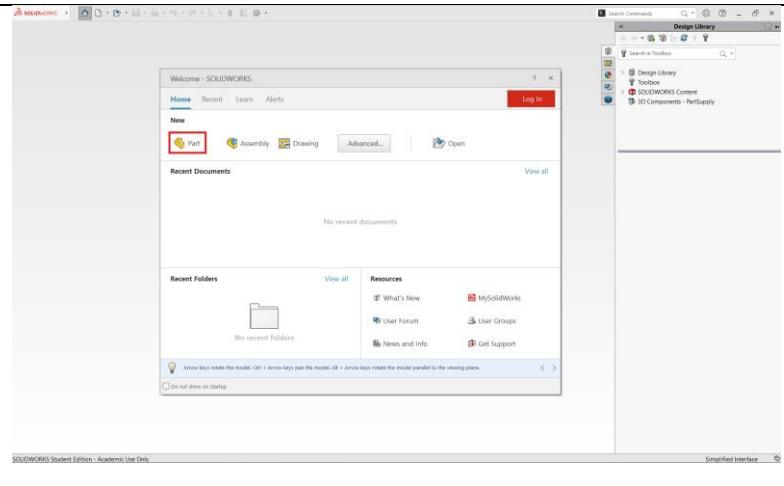


Catapult Base

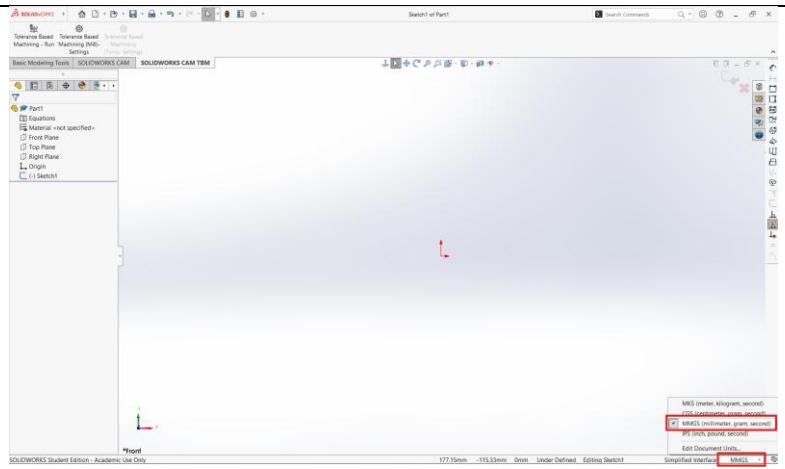
1. Open SolidWorks 2025



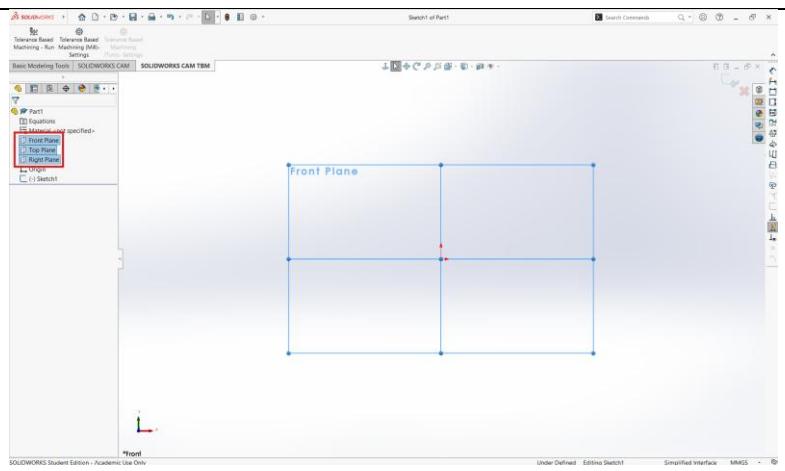
2. Create a new part



3. Ensure your units are MMGS (millimeters, grams and seconds)

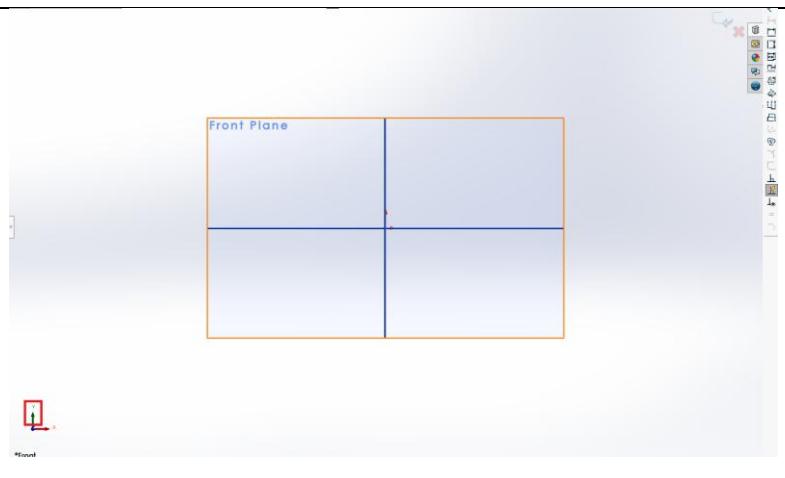


4. Select the 3 planes on the left pane, right click then click the eye to “show” them

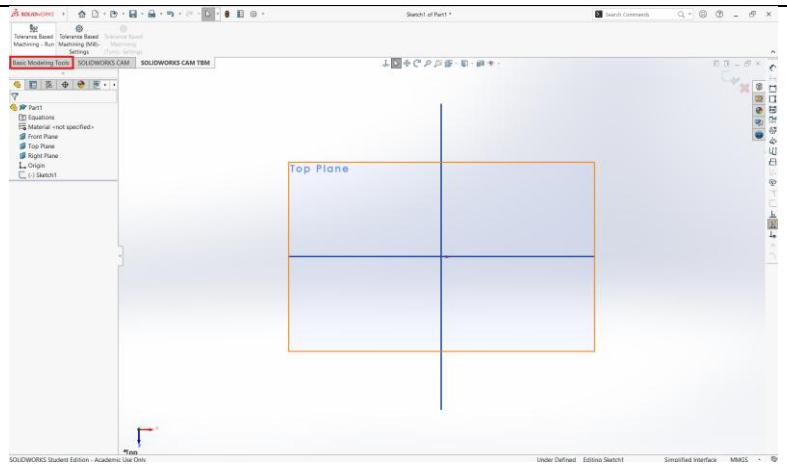


5. Click the y-axis in the lower left corner to view the top plane.

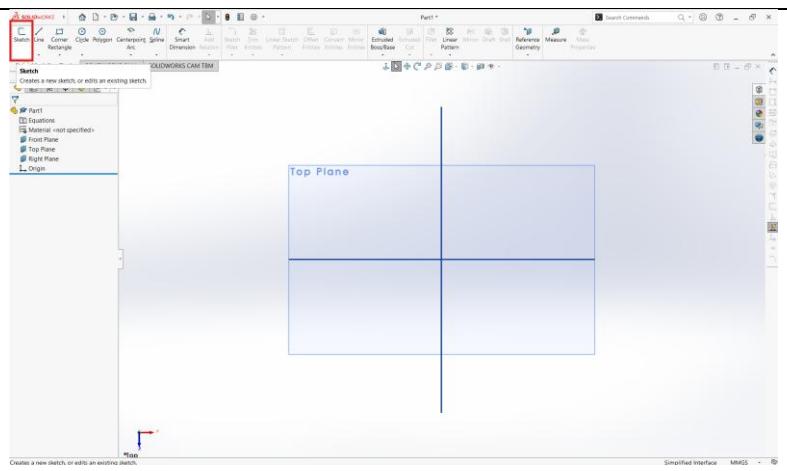
Note: You can also use the middle button of your mouse to drag your perspective around



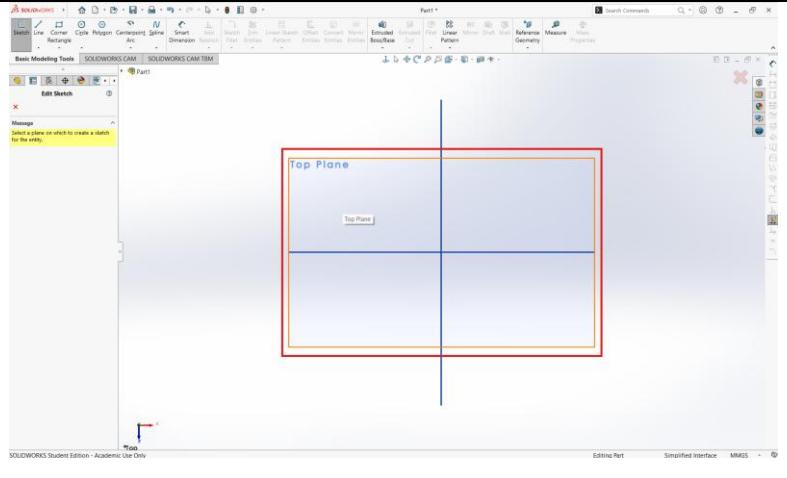
6. Select “Basic Modeling Tools”



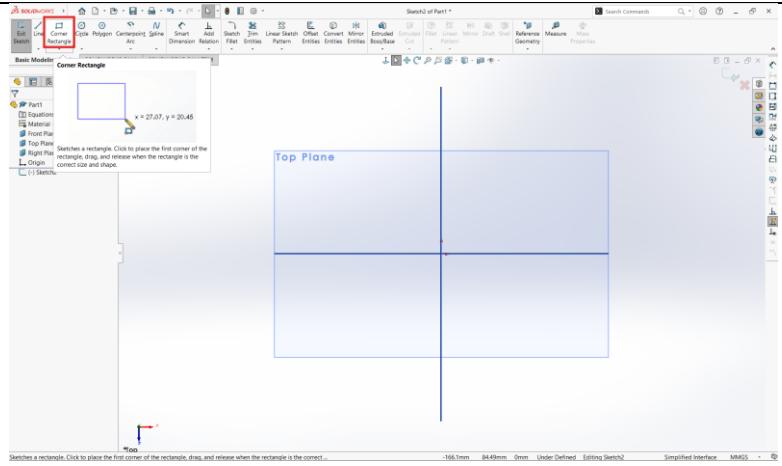
7. Click “Sketch”



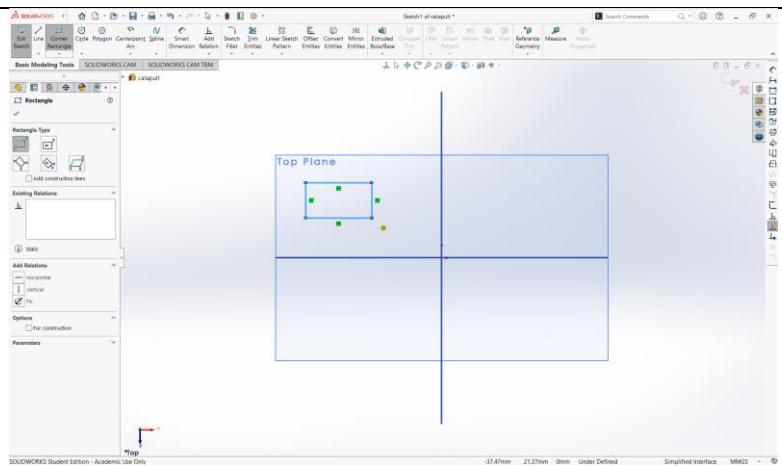
8. Click on the top plane, this tells SolidWorks on what 2D plane to put your sketch



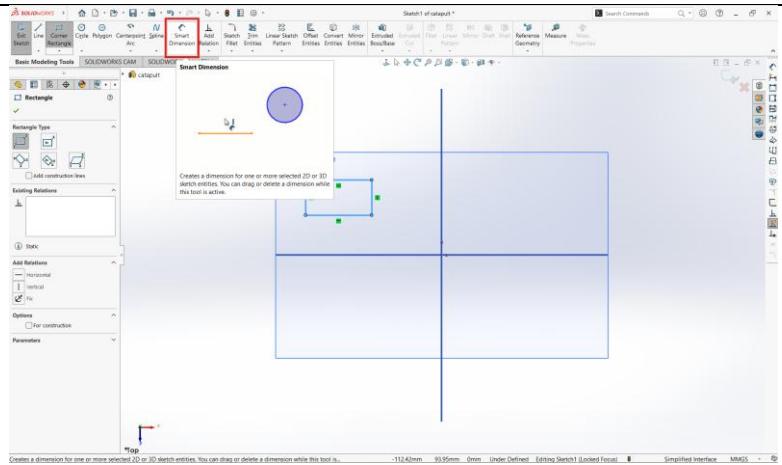
9. Click on the “Corner Rectangle” tool



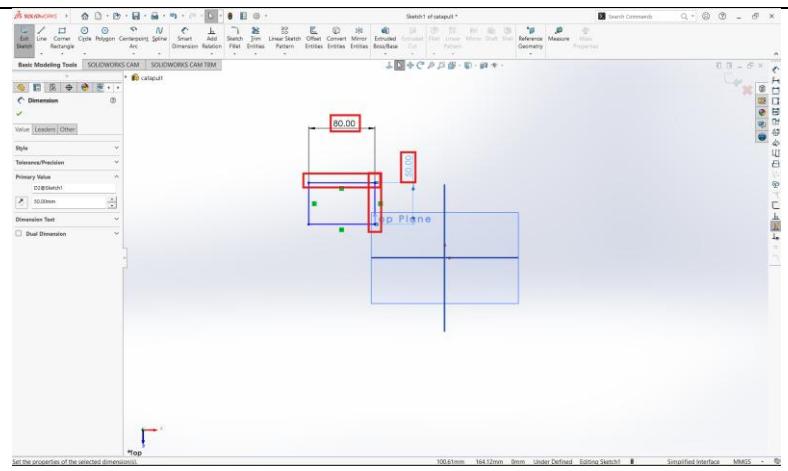
10. Click any two points to create a rectangle



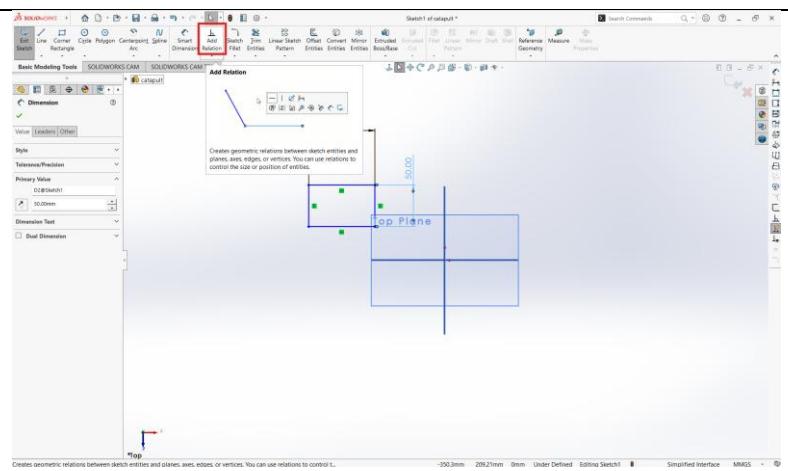
11. Click on the “Smart Dimension” tool



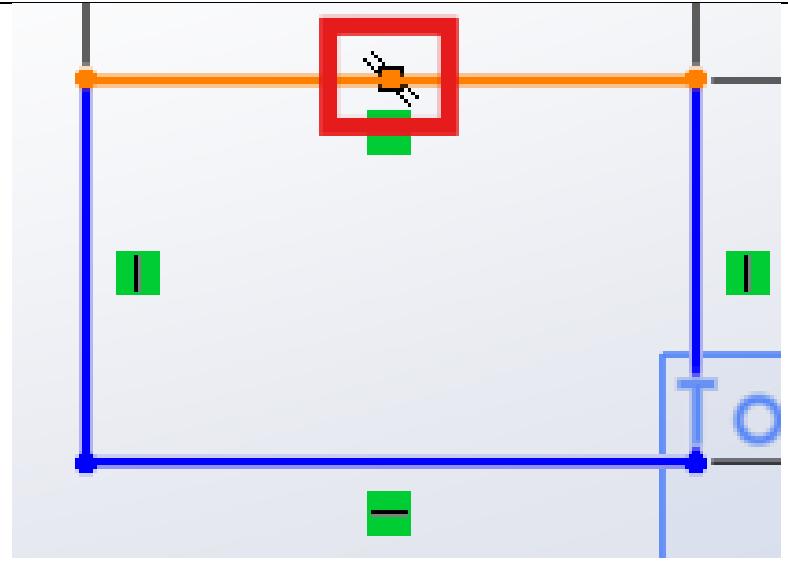
12. Click on the edges of the rectangle you want to dimension; make the rectangle 80mmx50mm



13. Click on the “Add Relation” tool

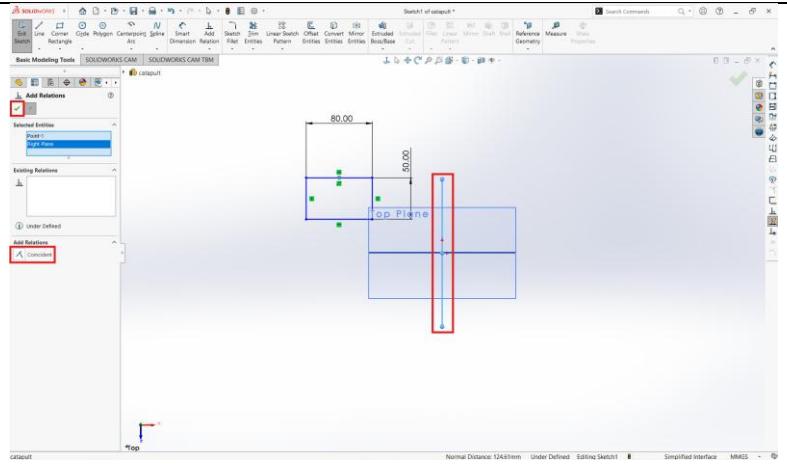


14. Move your mouse around the center of the top edge of the rectangle until the midpoint appears – click it

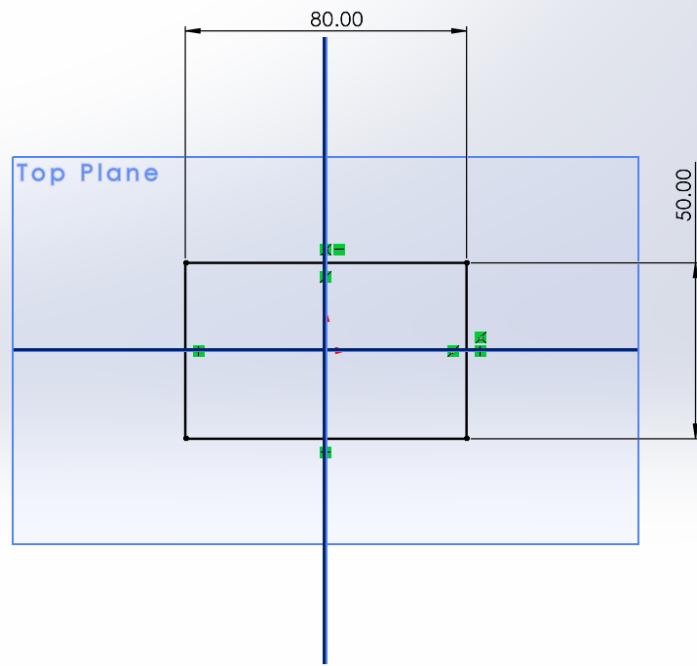


15. Click the right plane
- Click “Coincident” in the left pane
 - Click the green check mark in the left plane

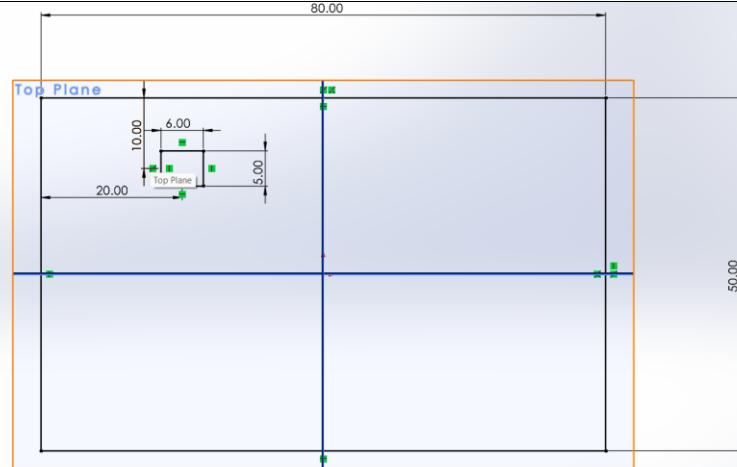
This will center your rectangle horizontally



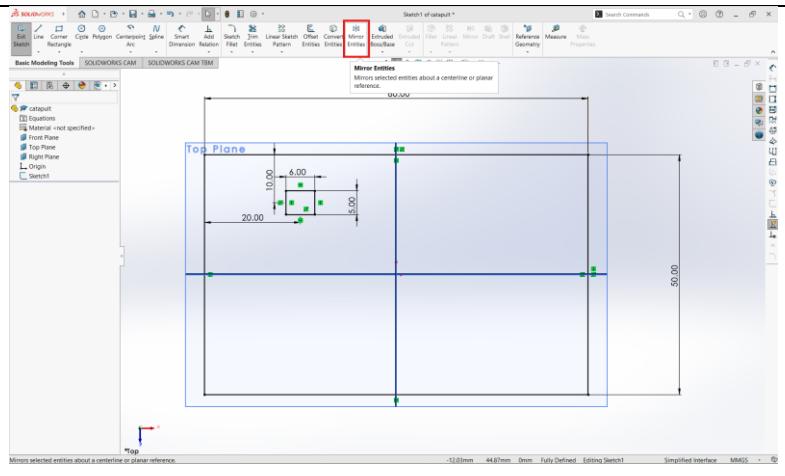
16. Hit escape then follow the same procedure (steps 14-15) to center your rectangle vertically. It should look like the picture to the left when you are finished



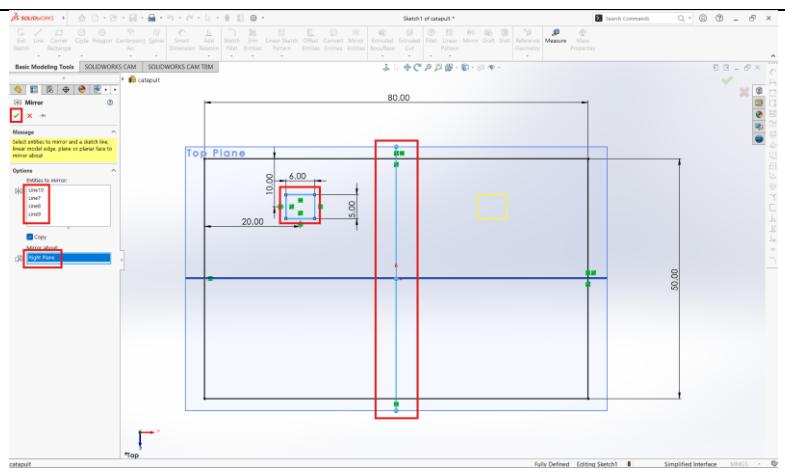
17. Add another rectangle to the top left corner with dimensions as depicted in the image to the right



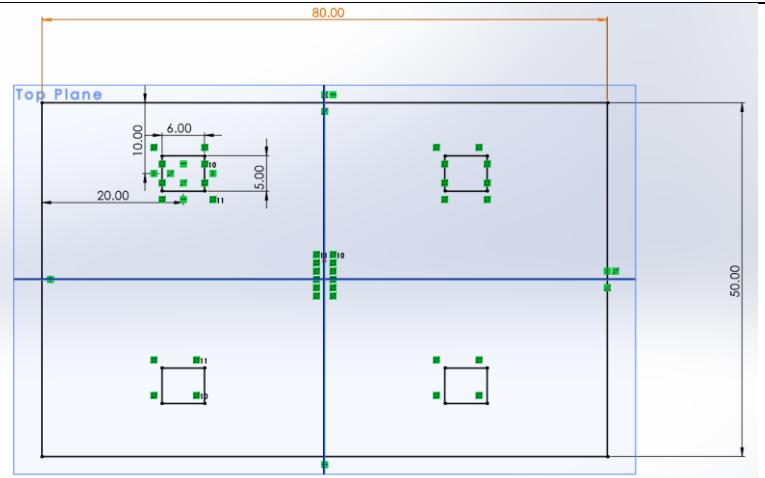
18. Click on the “Mirror Entities” tool



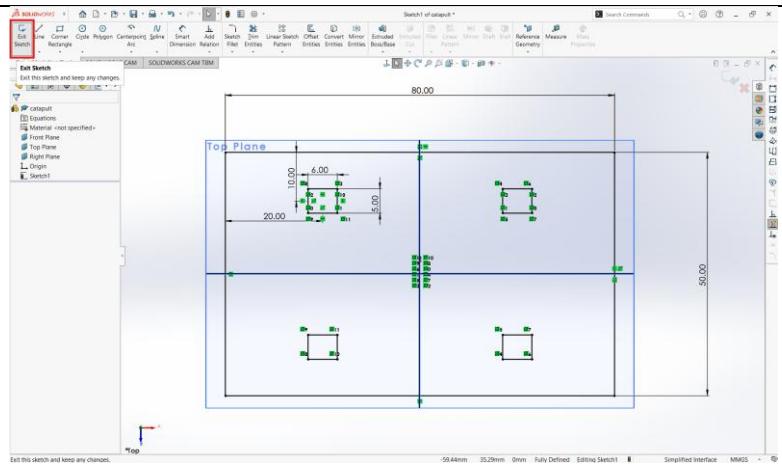
19. For “Entities to mirror,” select the edges of the small rectangle
a. For “Mirror about” click on the right plane
b. Click the green checkmark



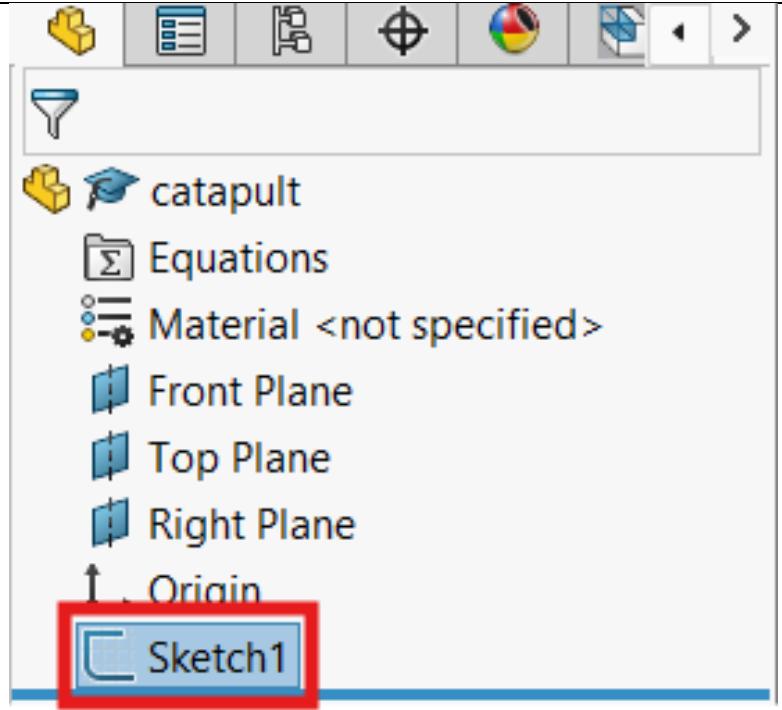
20. Repeat the same process (steps 18-19) to mirror the rectangles about the front plane



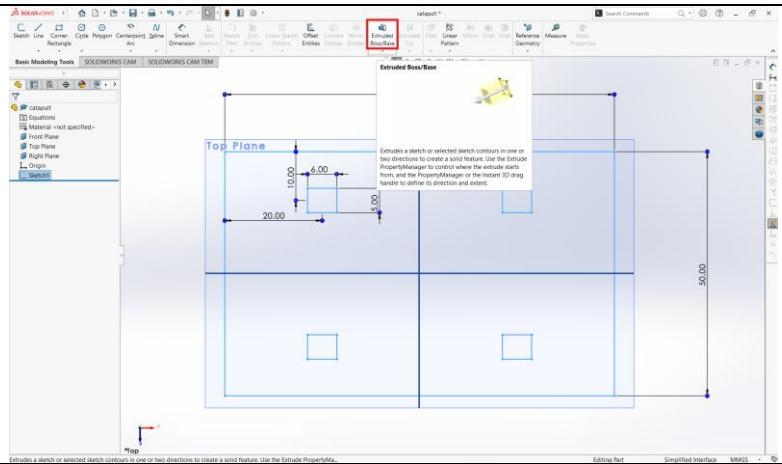
21. Click “Exit Sketch”



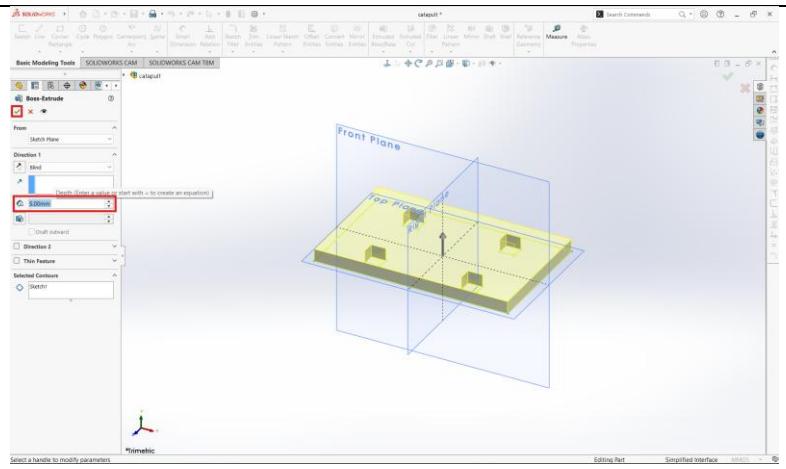
22. Click on the sketch you just made in the design tree



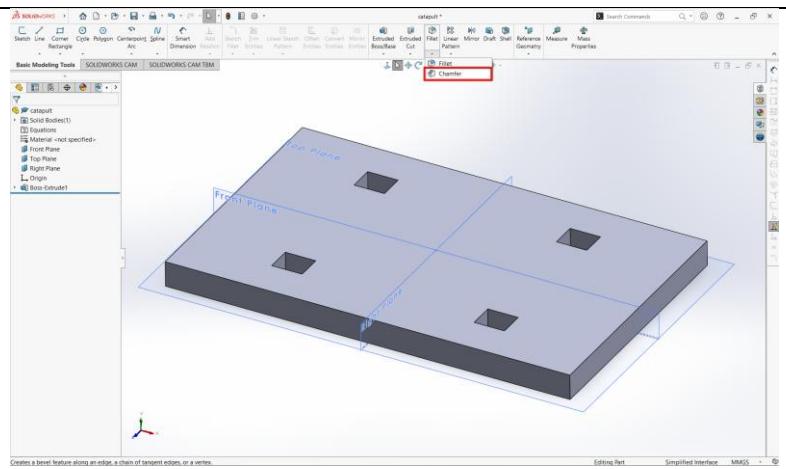
23. Click “Extrude Boss/Base”



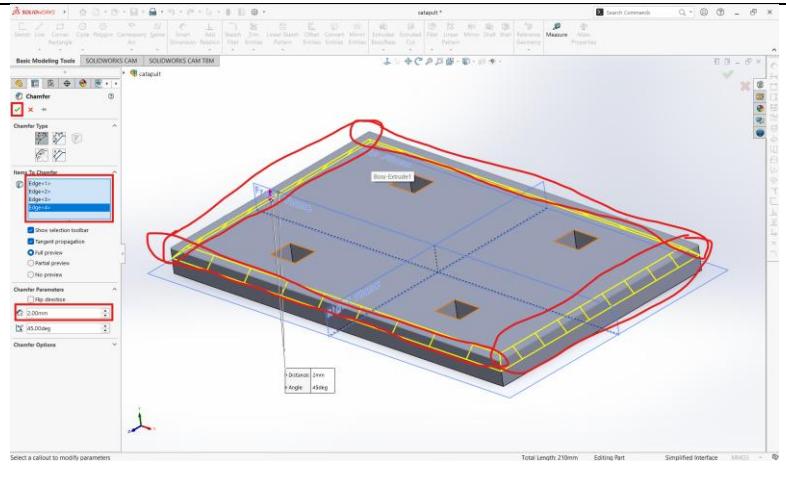
24. Enter a depth of 5mm and click the green checkmark



25. Select the “Chamfer” tool

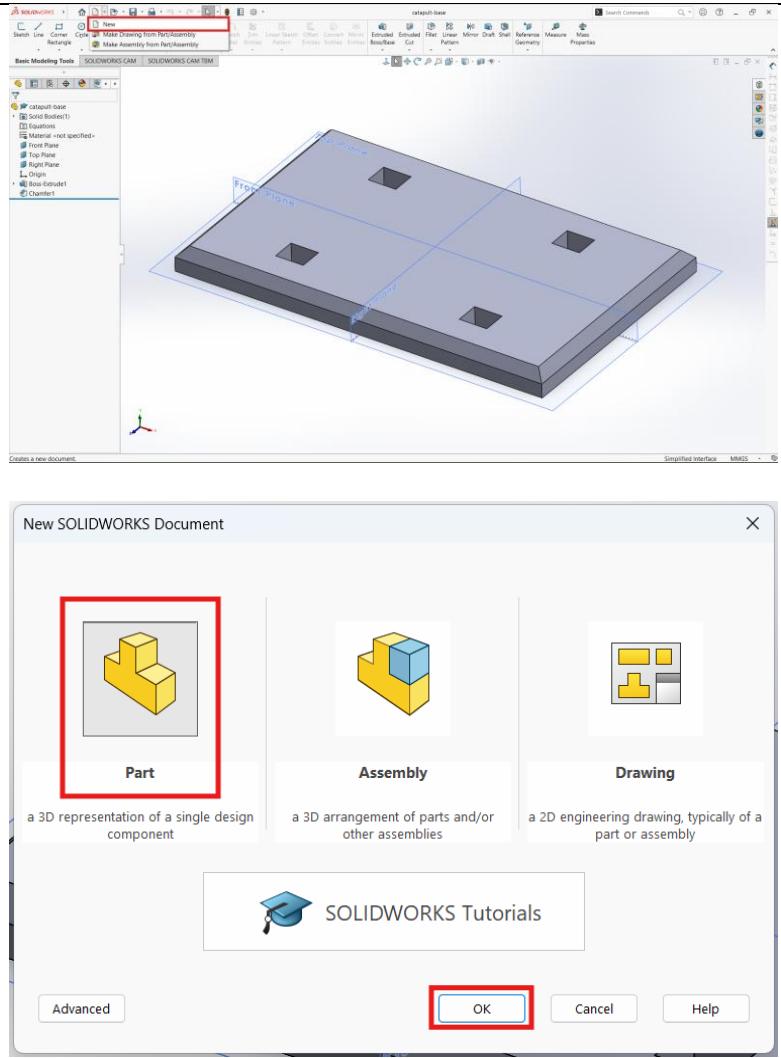


26. Select all 4 upper edges of the rectangle you extruded
a. Set the chamfer distance to 2mm
b. Click the green checkmark

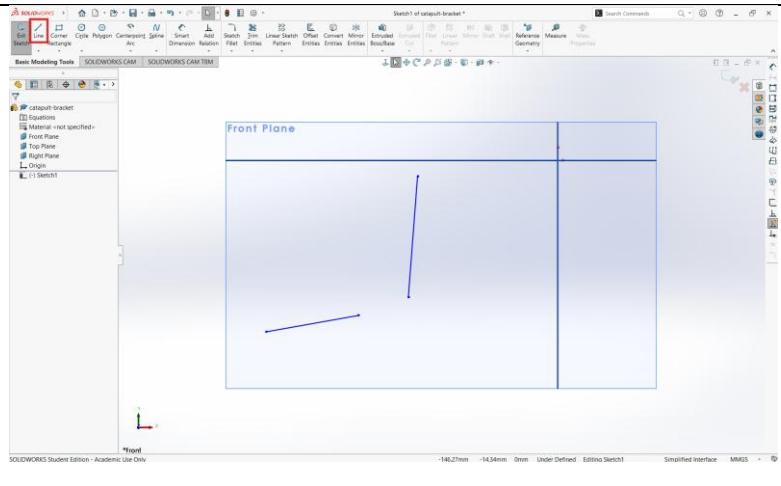


Catapult Bracket

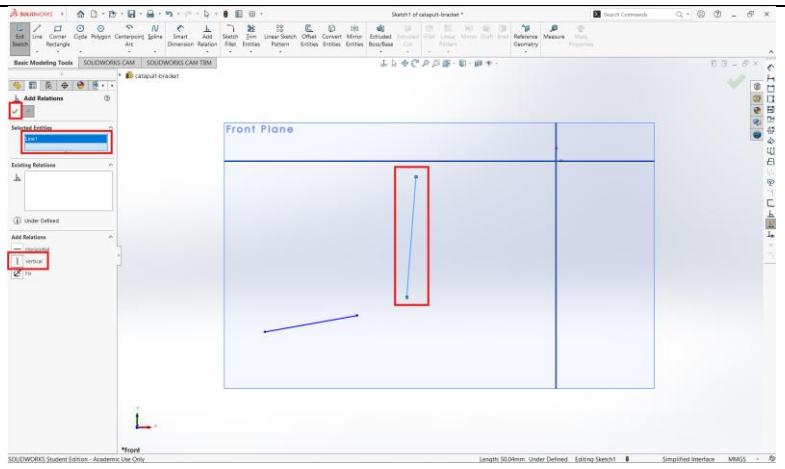
1. Create a new part



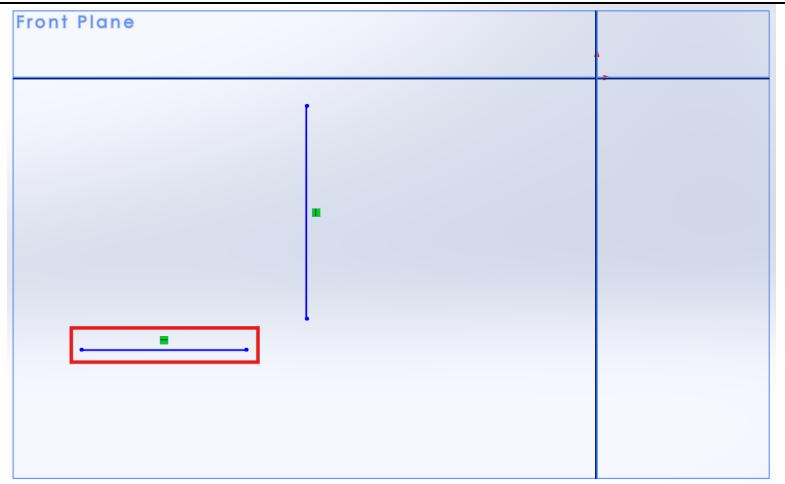
2. Use the line tool to create 2 lines



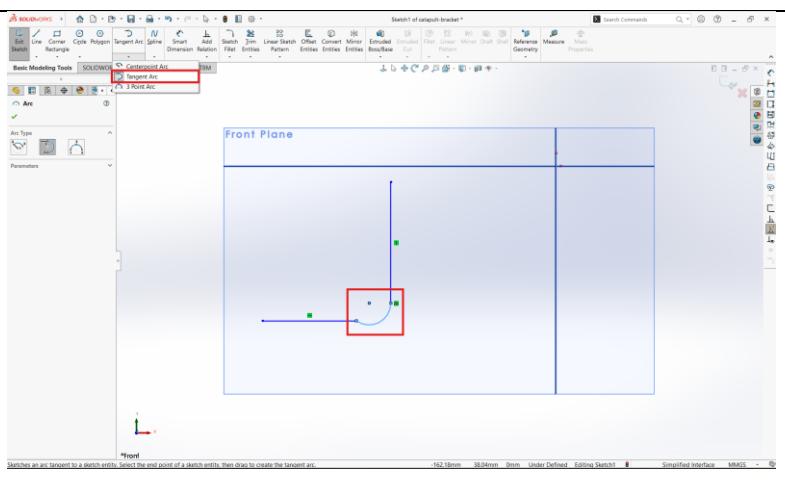
3. Use the “Add Relation” tool to make one of the lines vertical



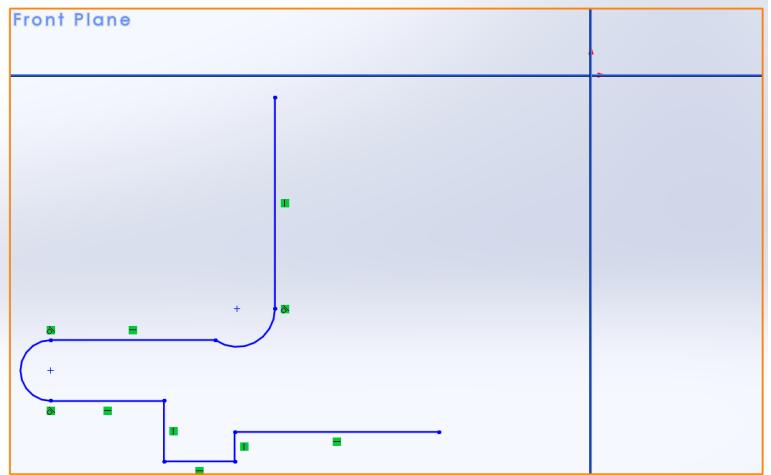
4. Add a horizontal constraint on the second line



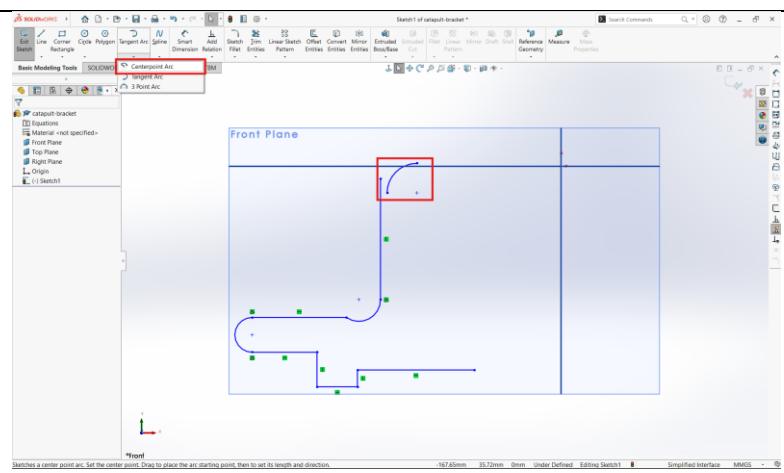
5. Use the “Tangent curve” tool to connect the 2 lines



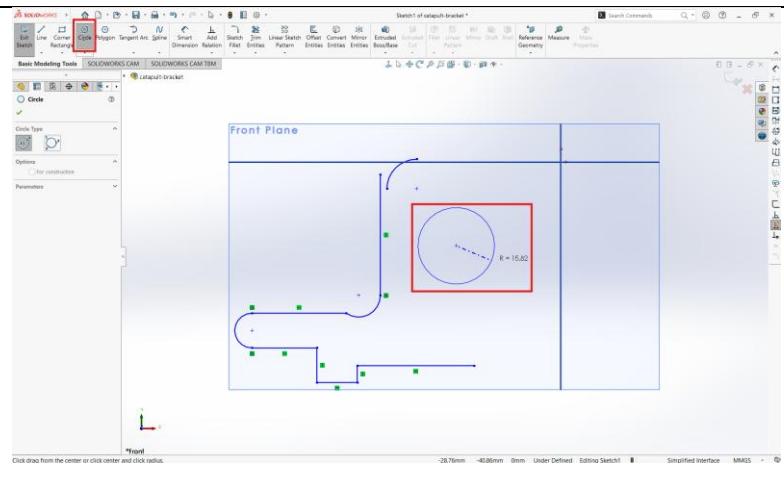
6. Add more lines and curves and constrain them



7. Use the “Center point arc” tool to draw a quarter circle



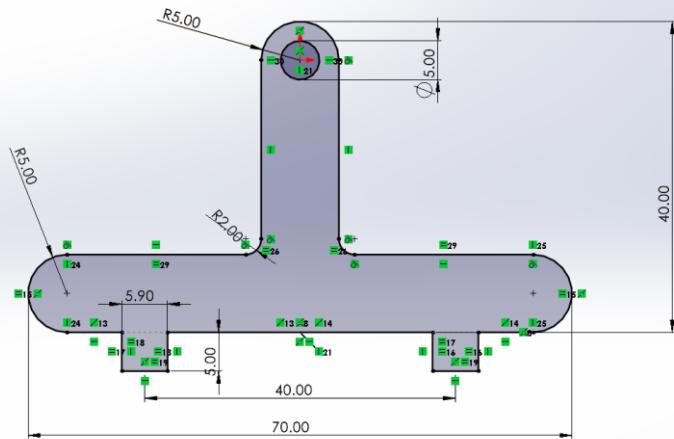
8. Use the “Circle” tool to draw a circle



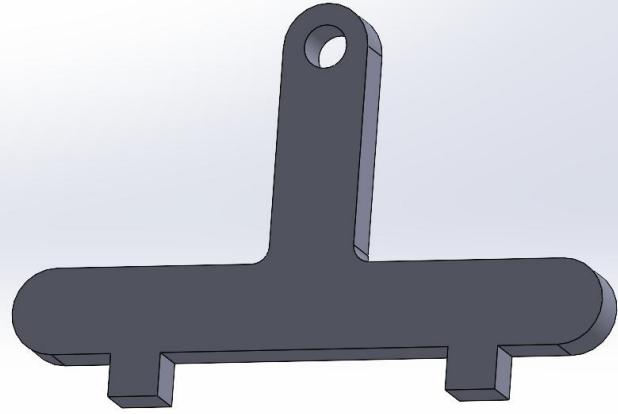
9. Use what has been shown so far to create the sketch depicted on the right

Note: The sketch is symmetrical, and you need to use a variety of sketch tools

All your lines need to be black to indicate the sketch is fully defined

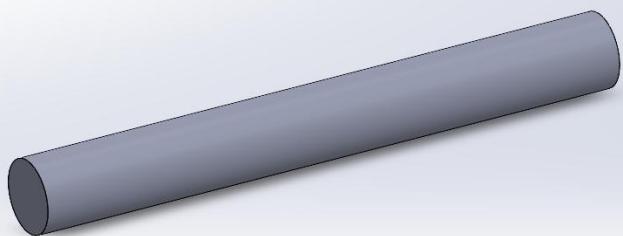


10. Extrude the sketch to a depth of 4.9mm

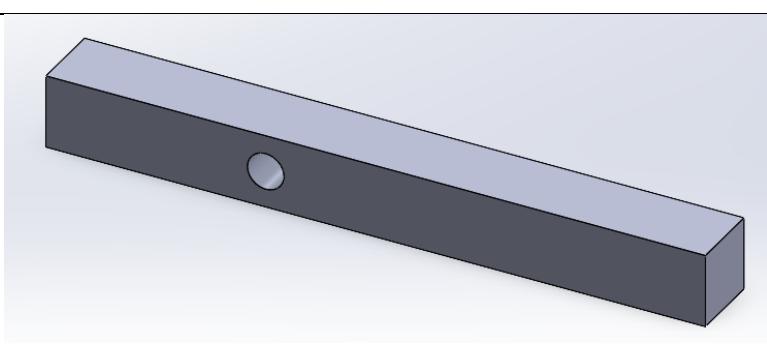
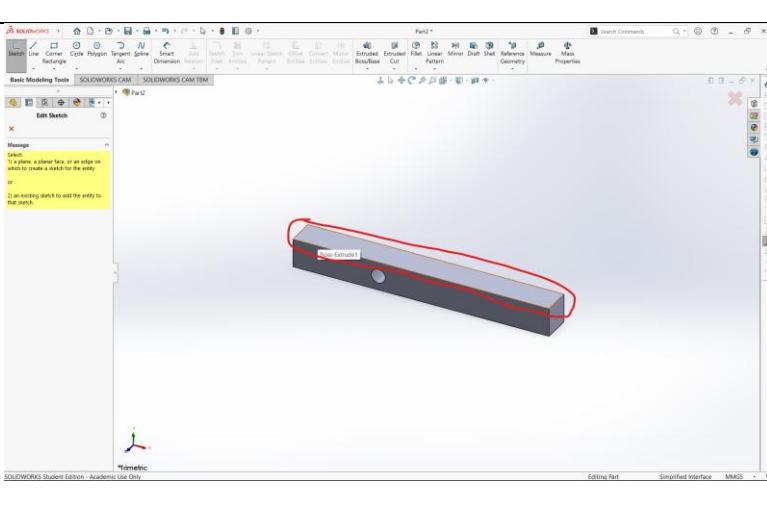


Axle Pin

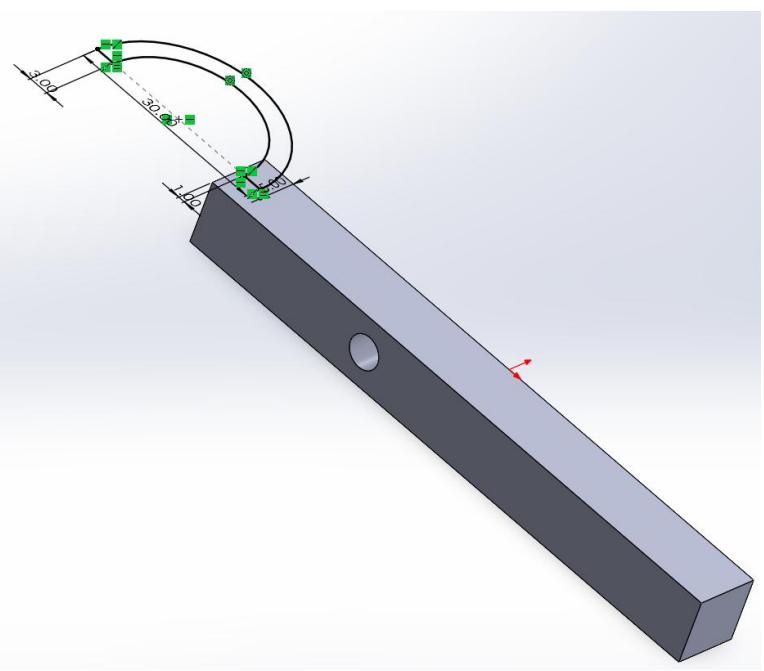
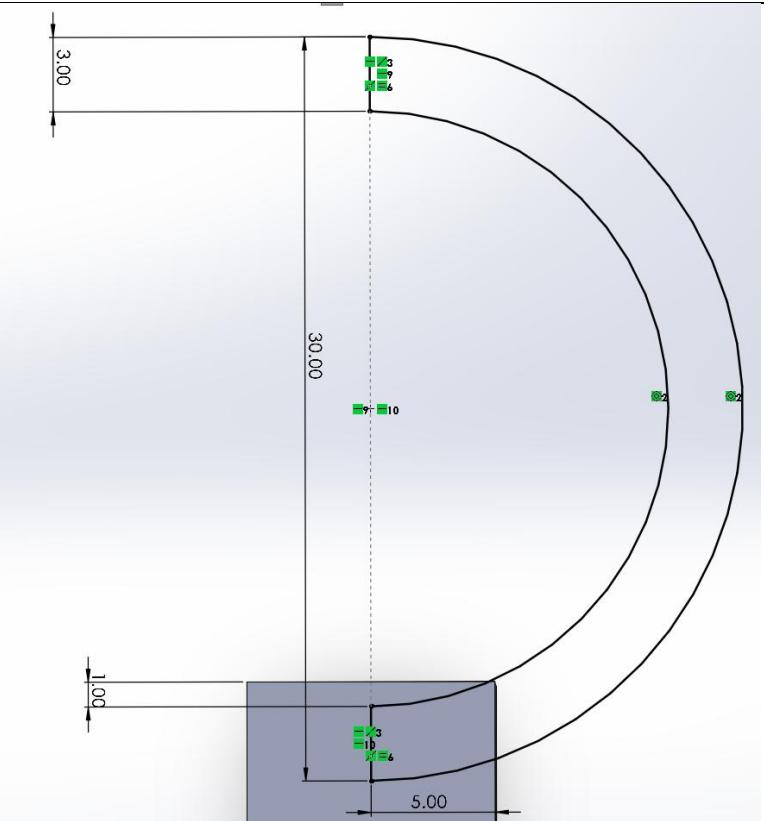
1. Create the part to the right which is a cylinder with a diameter of 4.8mm and length of 40mm



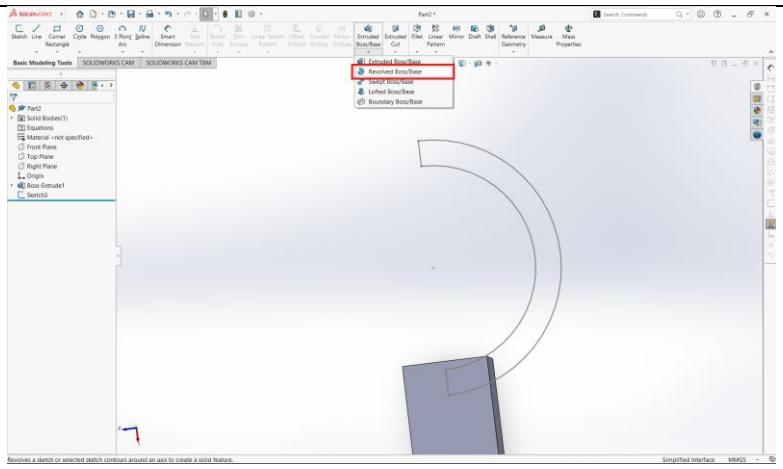
Throwing Arm

1. Create a new part and the sketch depicted on the right	
2. Extrude the sketch 10mm	
3. Create a sketch on the top of the arm	

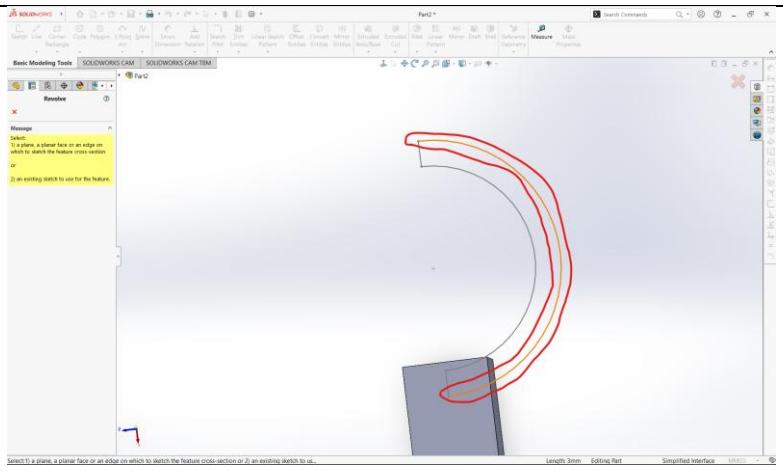
4. Create the following sketch



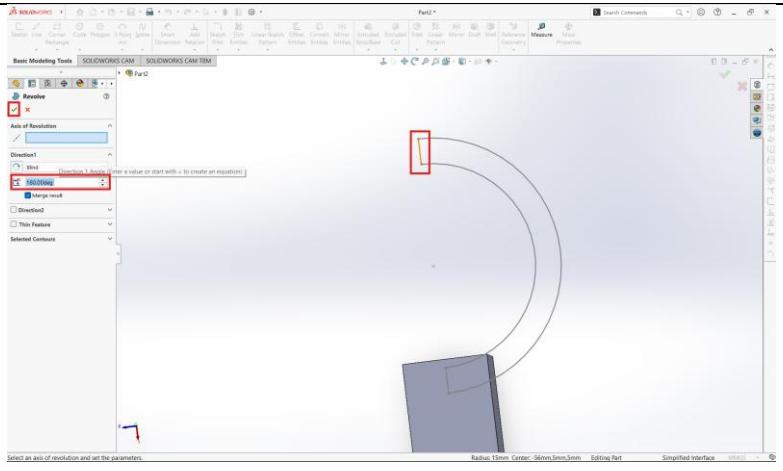
5. Select the “Revolve Boss/Base” tool



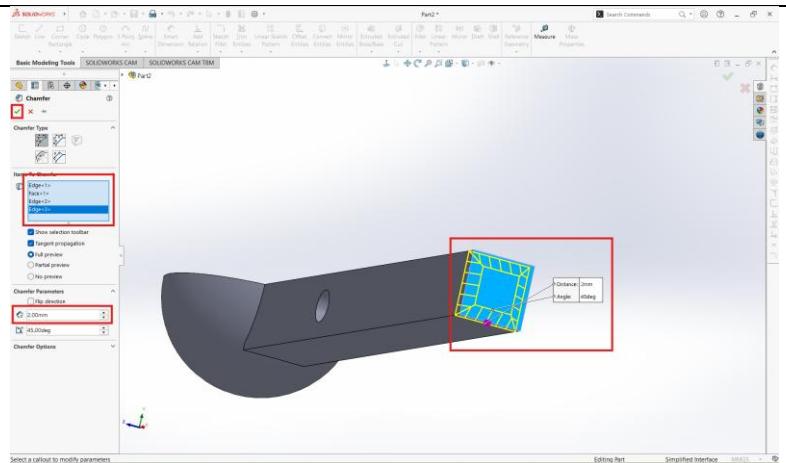
6. Click the outer edge of the sketch to select what you are revolving



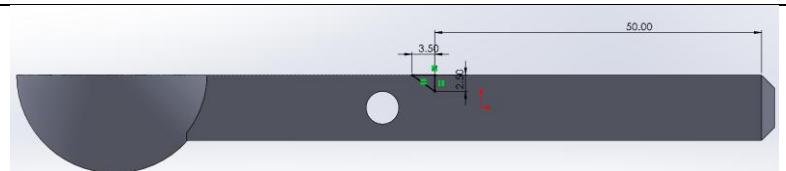
7. Click one of the sketch's straight segments to determine an axis of rotation
 a. Change the angle to 180 degrees
 b. Click the green checkmark



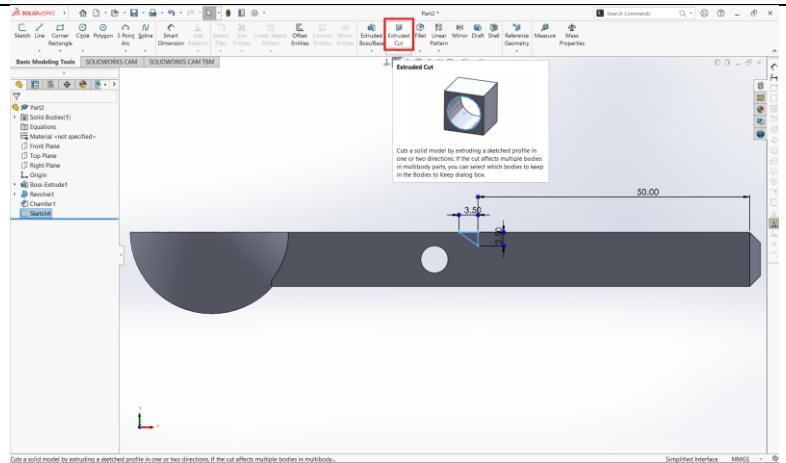
8. Chamfer the bottom 4 edges



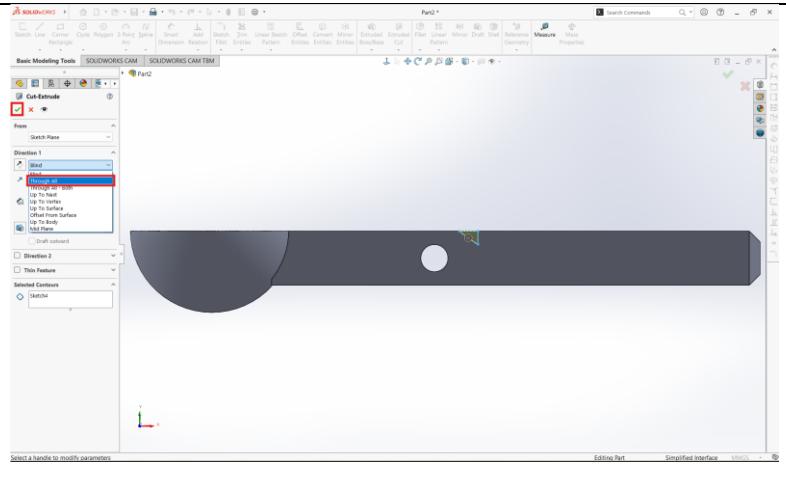
9. Create the sketch depicted on the right



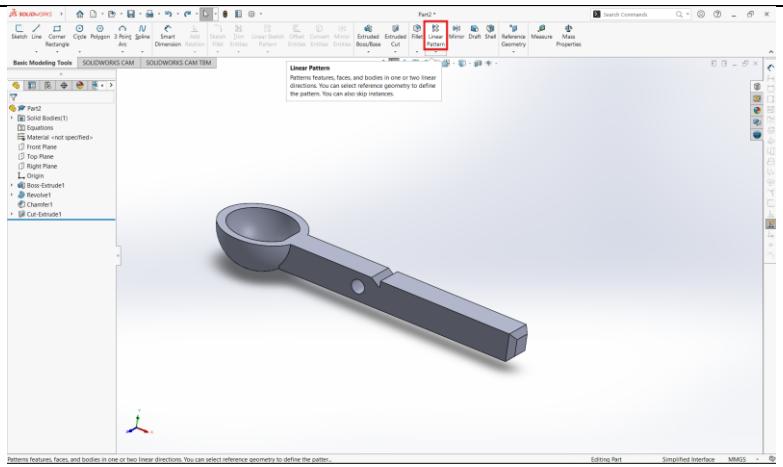
10. With the previous sketch selected, click on the “Extrude Cut” tool



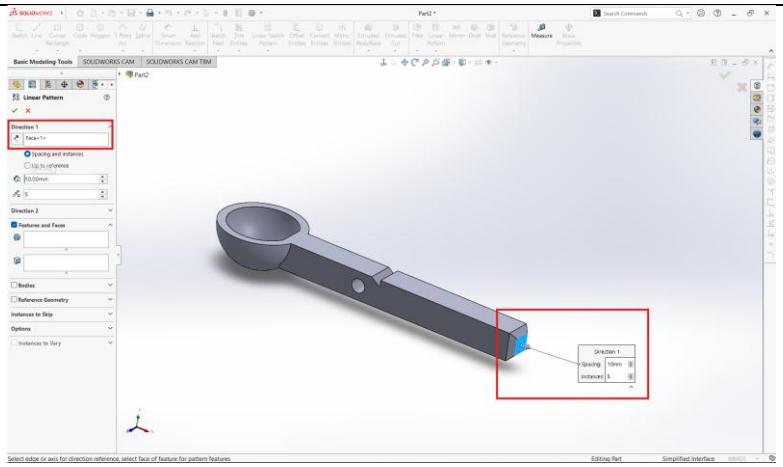
11. Select “Through All” from the dropdown then click the green checkmark



12. Click on the “Linear Pattern” tool

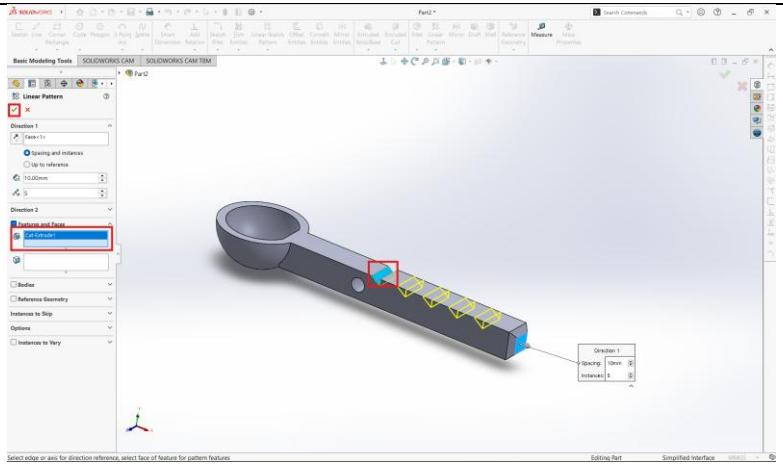


13. Click on the end of the throwing arm to specify the direction



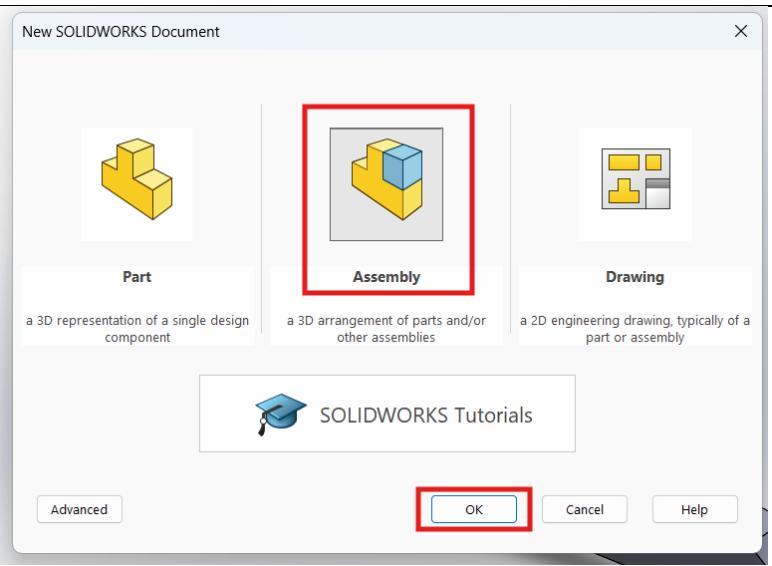
14. Click on the “Features and Faces” box

- Click on one of the faces of the previous cut extrude
- Click the green checkmark

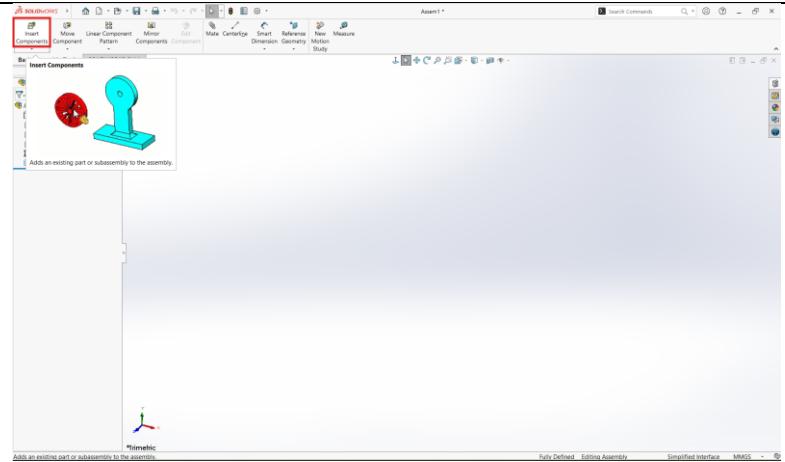


Assembly

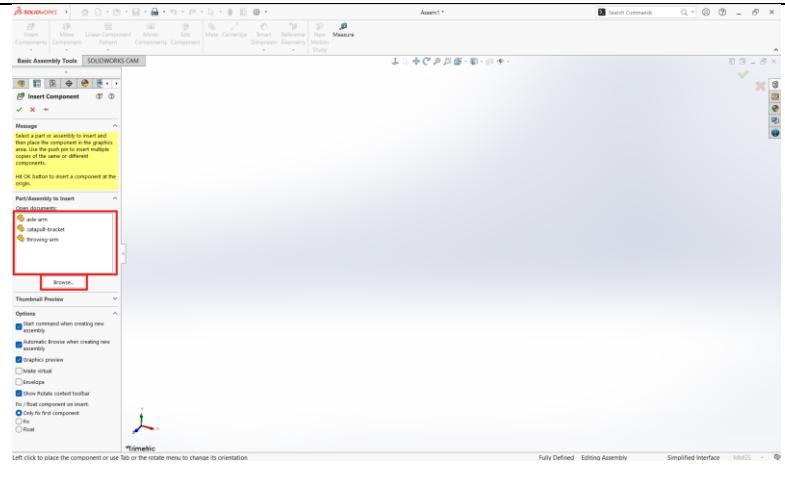
1. Create a new assembly



2. Click “Insert Component”

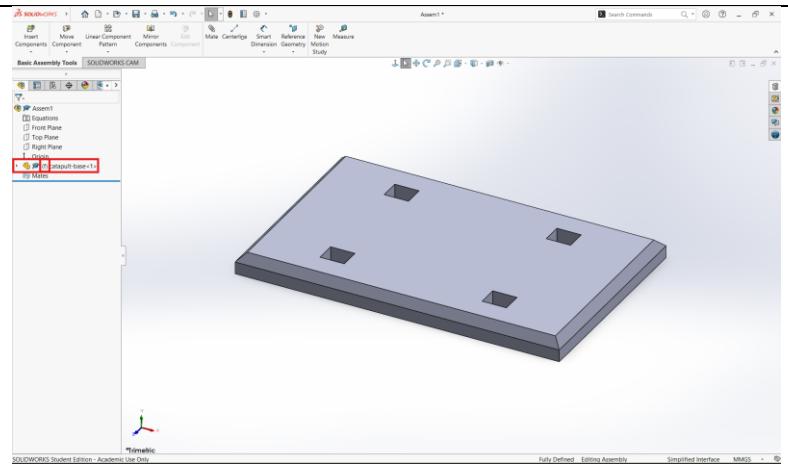


3. Select the catapult base from the list or click “Browse” to find it in your files

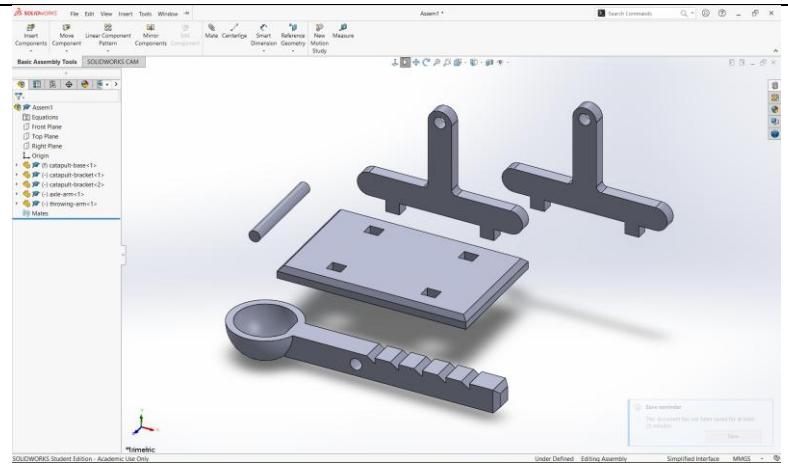


4. Click to place the part

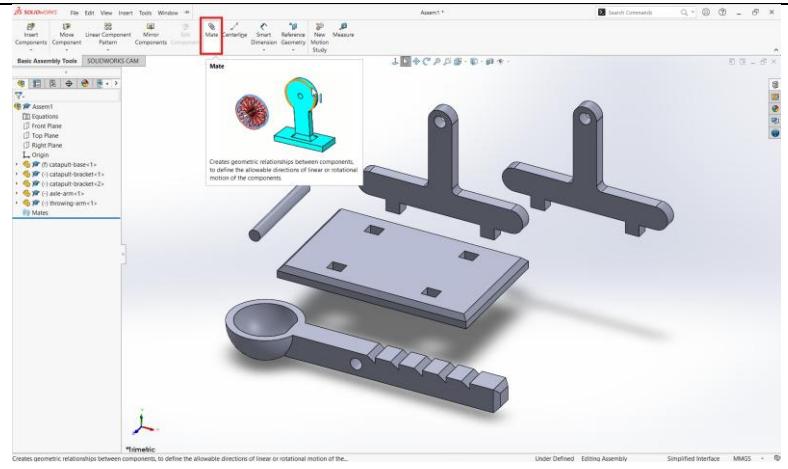
Note: Ensure the part has an (f) next to it, indicating it is a fixed part. This should be the only part that is fixed



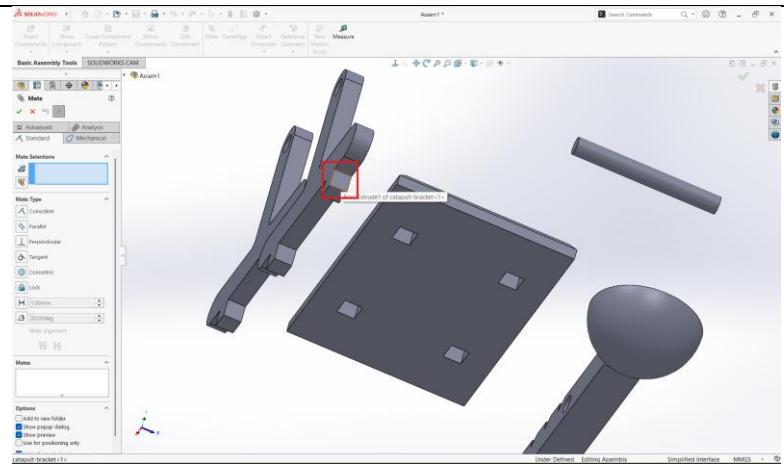
5. Insert the other parts



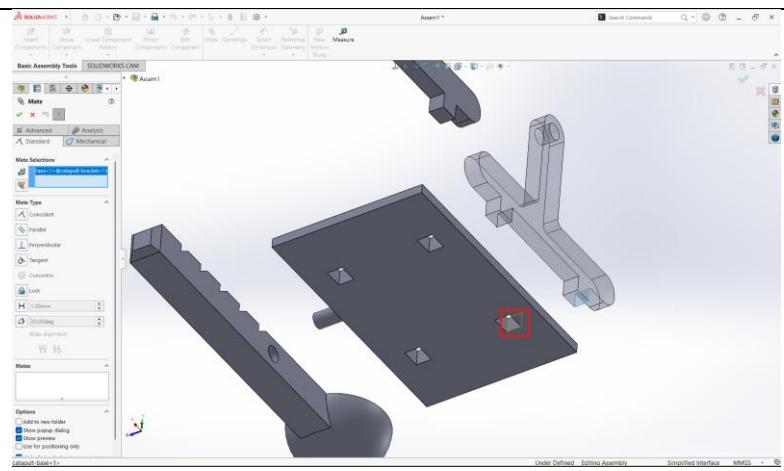
6. Select the “Mate” tool



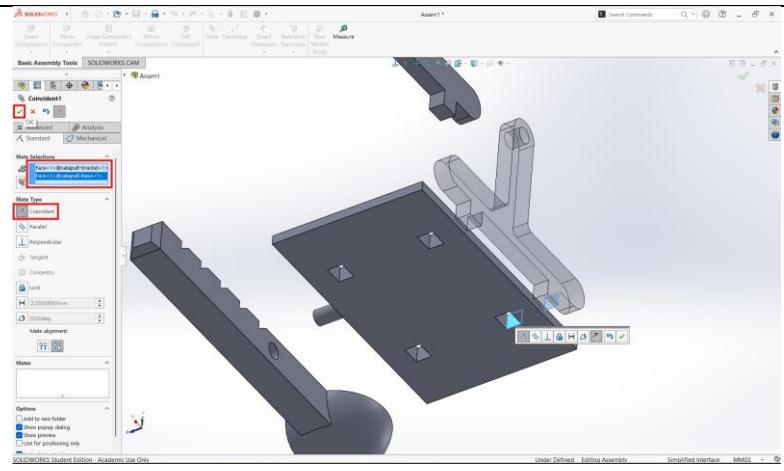
7. Select the face of one of the catapult brackets as shown in the image to the right



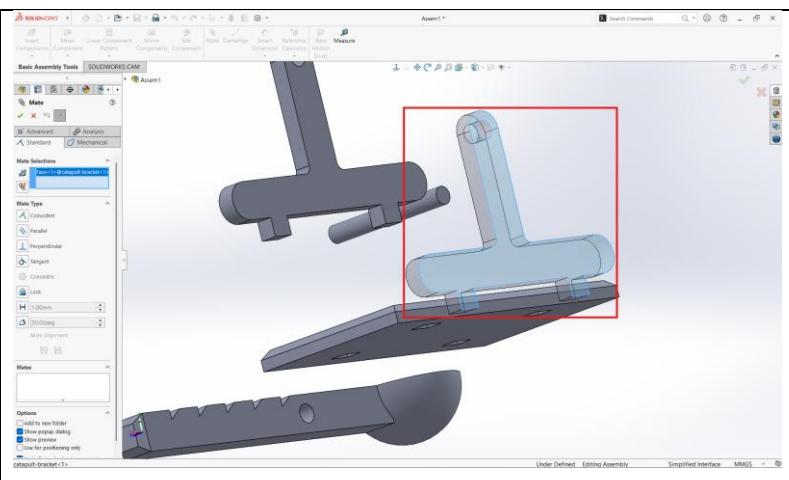
8. Now select this face from the catapult base as shown in the picture on the right



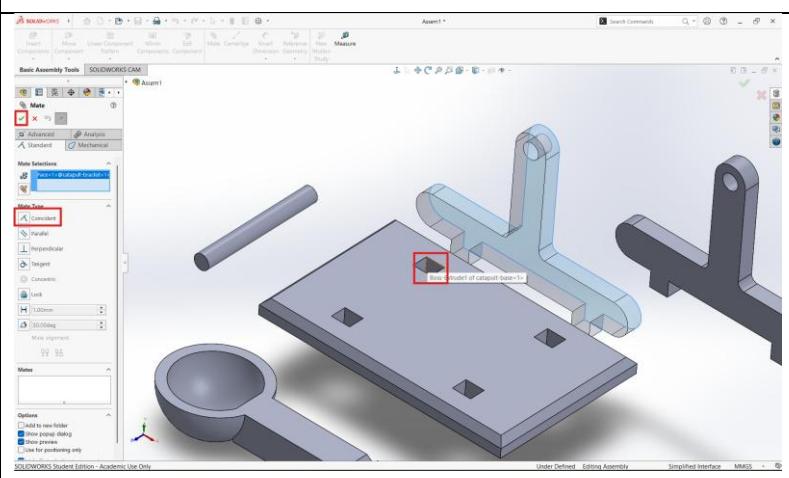
9. Select a coincident mate and click the green checkmark



10. On the same bracket, click the shown face

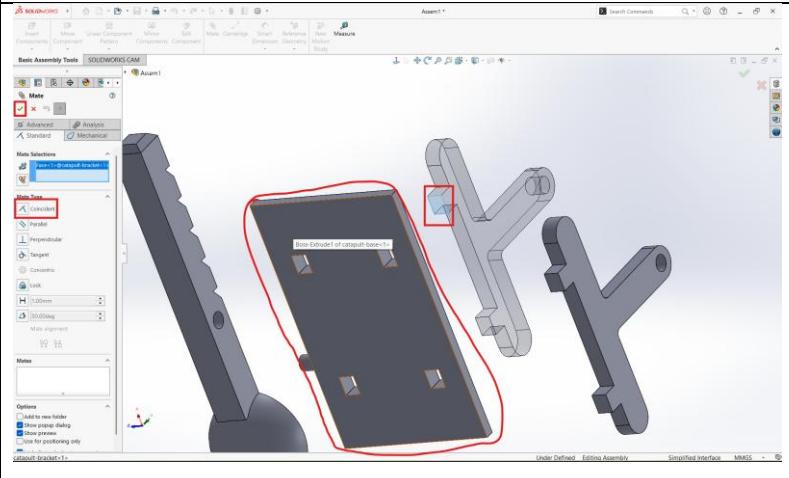


11. Select this face on the base, followed by clicking the coincident mate, and the green checkmark

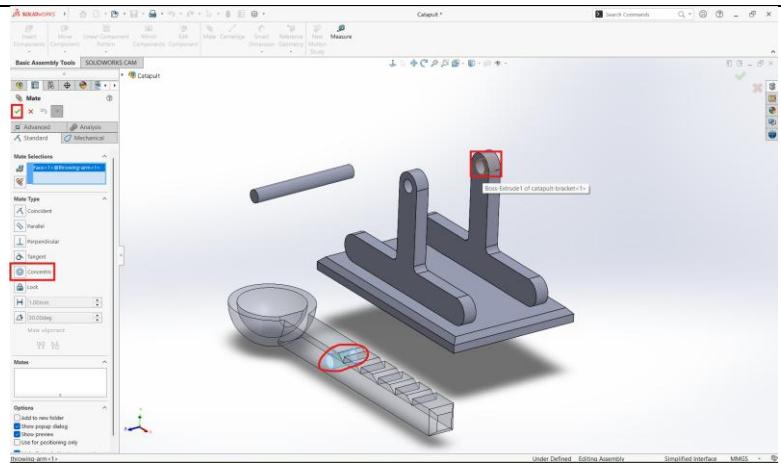


12. Click the following faces and create another coincident mate between them. This should fully constrain one of the brackets

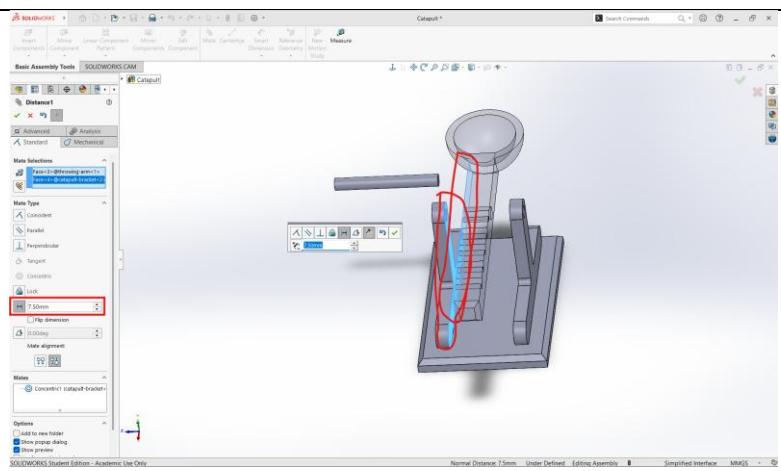
a. Repeat steps 6-12 for the other bracket



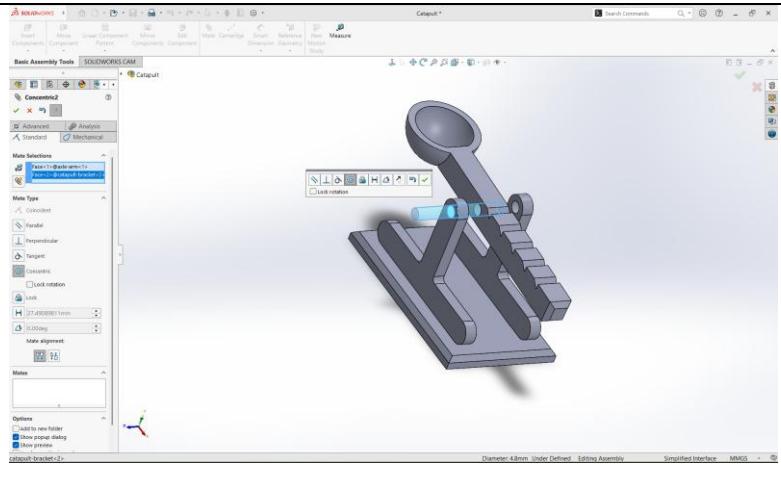
- 13. Create a concentric mate between the hole in the throwing arm and the hole in the bracket**



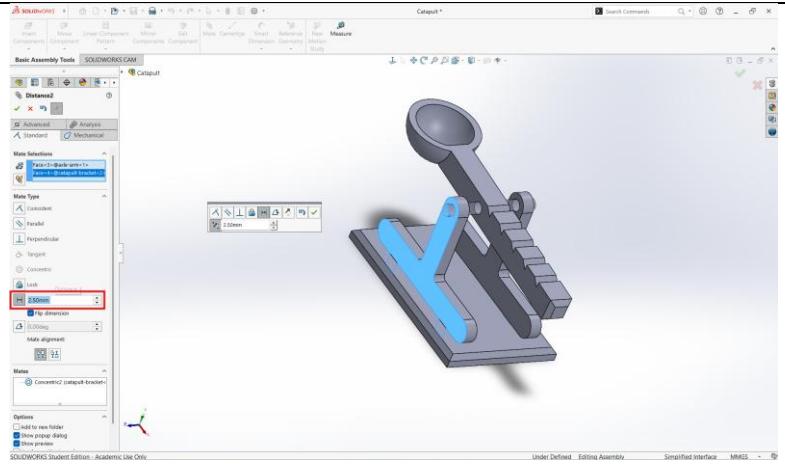
- 14. Create a distance mate of 7.5mm between the outer face of the thrower arm and the inner face of the bracket**



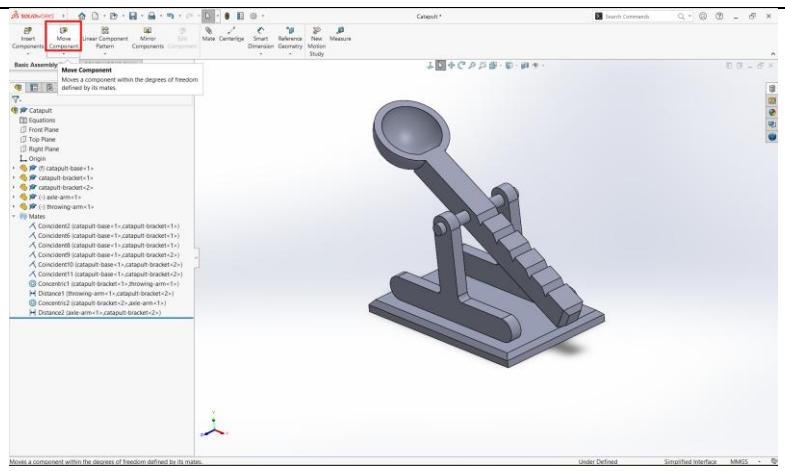
- 15. Use another concentric mate to insert the axle pin**



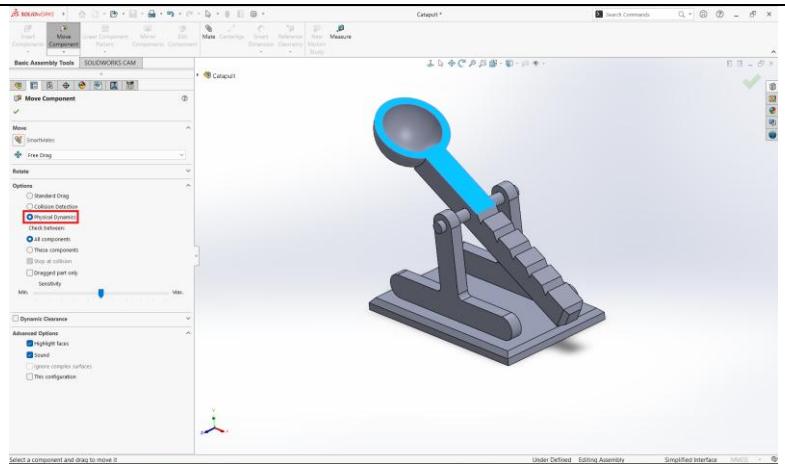
- 16. Create a distance mate of 2.5mm between the outer face of the axle pin and the bracket**



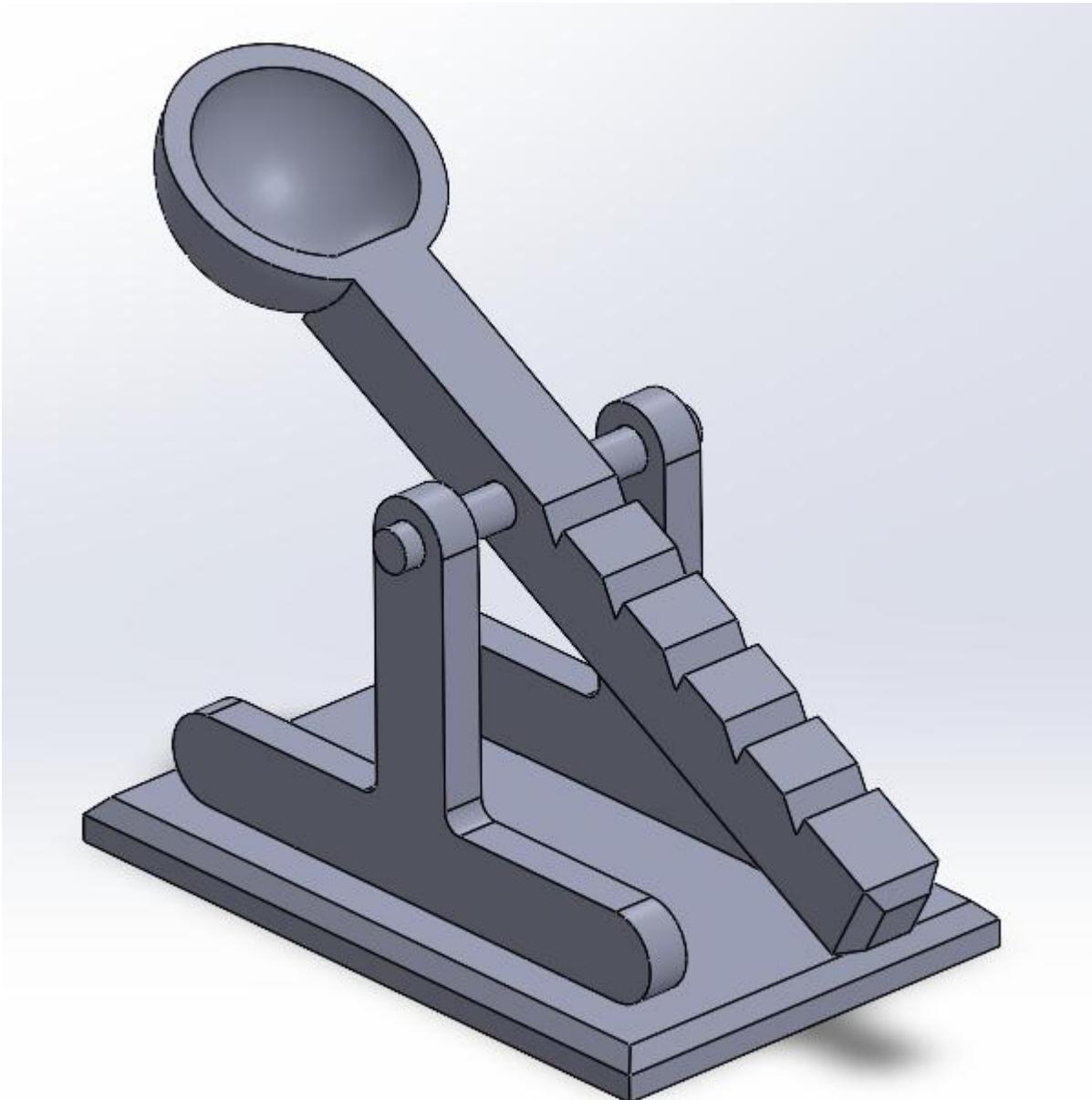
- 17. Click the “Move Component” tool**



- 18. While in this tool with the “Physical Dynamics” option, you can move the catapult arm and see how it interacts with the base**



Final Assembly



Bonus!

- Try to add extra design features which can make the catapult shoot objects farther
- 3D-Print the catapult
 - o Add small fillet to each part's sharp corners of 1-2 mm radius
 - o Ensure all parts are saved as STL files