



# 190q-1c-cad-freeCAD-lesson-hl- 2024-7-10.odp

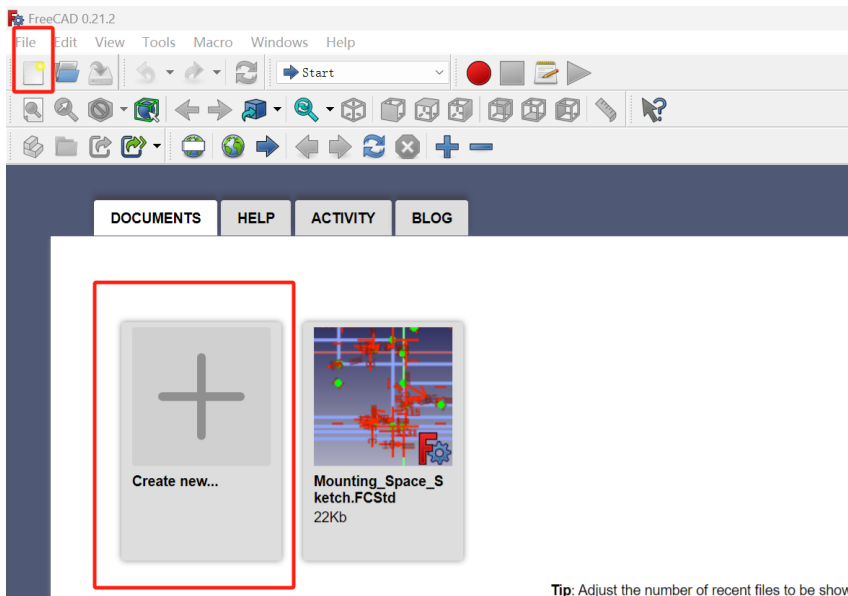
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Version 2, 2024-7-3  
CTI One Corporation is located in the  
Silicon Valley in Santa Clara, California.  
We design, develop and manufacture AI  
enabled smart rollators/walkers and  
exoskeleton devices for mobility  
solutions.

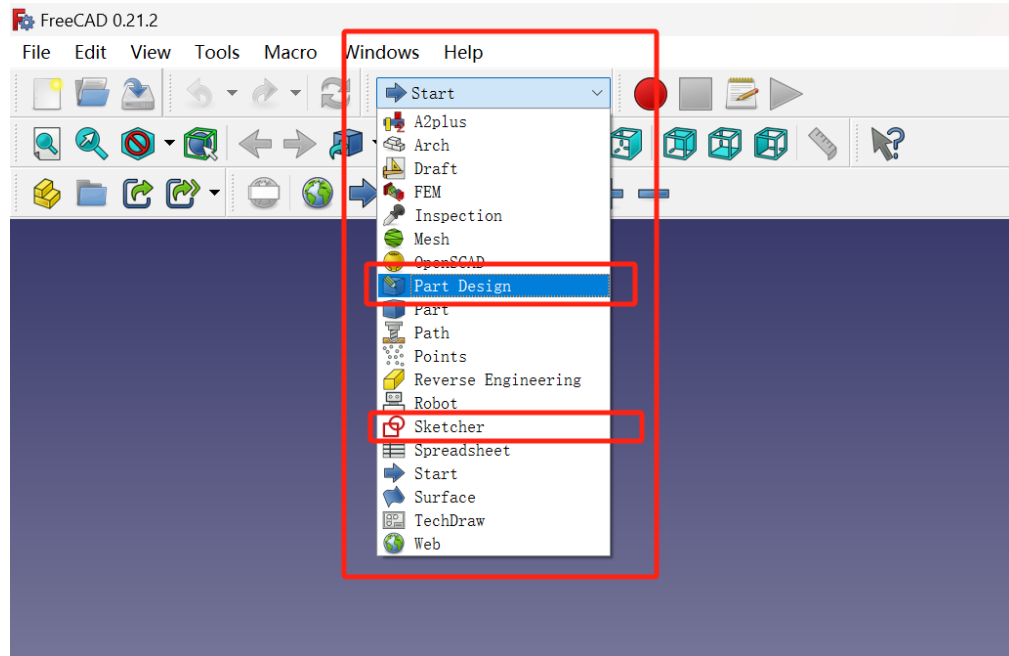
CTI One Corporation  
This document is created by:  
Harry Li, Ph.D.  
Chris Sun  
Ritai Su  
Jackson Meng

July 10, 2024

# FreeCAD Configuration (7/10,2024)



Step 1: Open FreeCAD, click on one of these two icons to create a new file. Each of them serves the same function



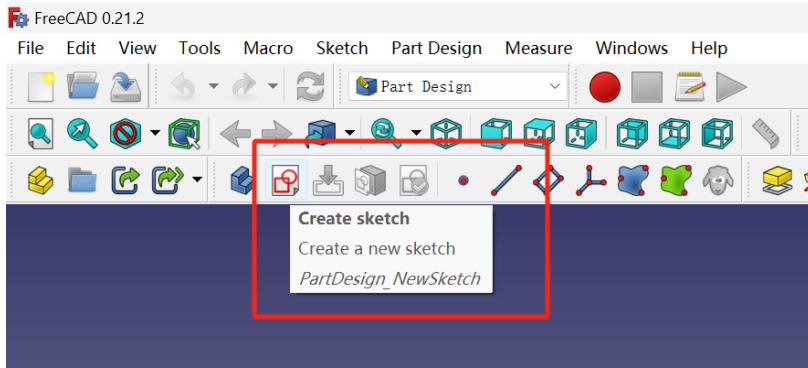
Step 2: Go to the drop-down column at the top. Select one of these under different conditions.

Sketcher: Only for sketches. The sketches you done here cannot be extruded into a 3D part

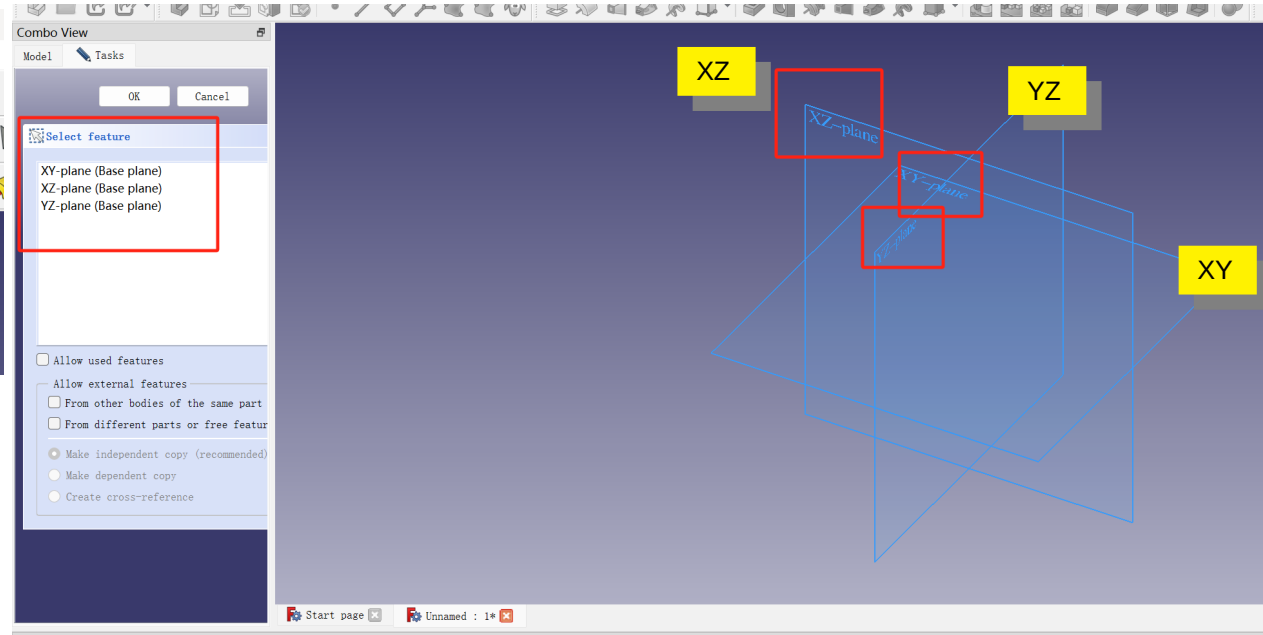
Part Design: For all the functions. Your sketches created over here can be extruded into 3D parts with further operations. (Recommended \*\*)



# Space Sketch Design for 2D Poly (7/10,2024)



Step 3: After selecting the drop down menu, select “Create Sketch” over here to start the 2D sketching process



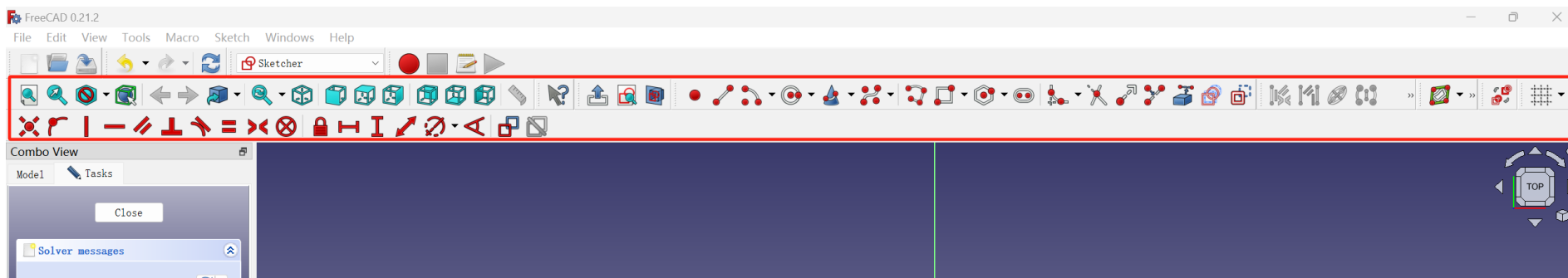
Step 4: After clicking sketch, you will instantly go into this page. You need to select one of the default planes as the datum plane of your sketch (Where your sketches will be placed on)

XZ plane: Also called front view

YZ plane: Also called side view

XY plane: Also called top view  
(Recommended \*\* We always want our part to be placed on a surface from the top)

# Space Sketch Design for 2D Poly (7/10,2024)

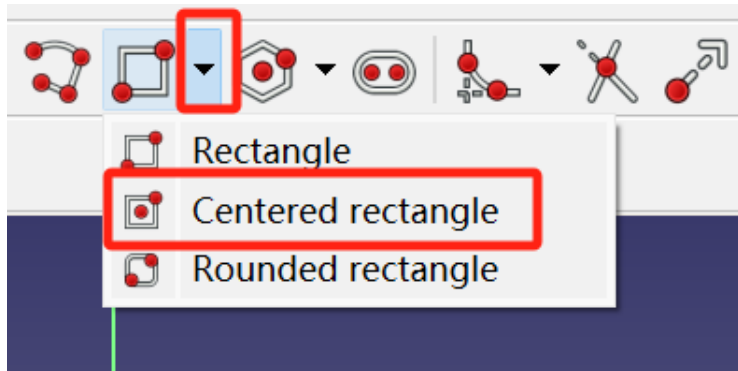


Step 5: After selecting the datum planes. You can see these icons on the top of your screen. These are called “sketch tools”. Basically they are all the tools you will use during the 2D sketching. Play around with it and you can get more familiar with each of these.

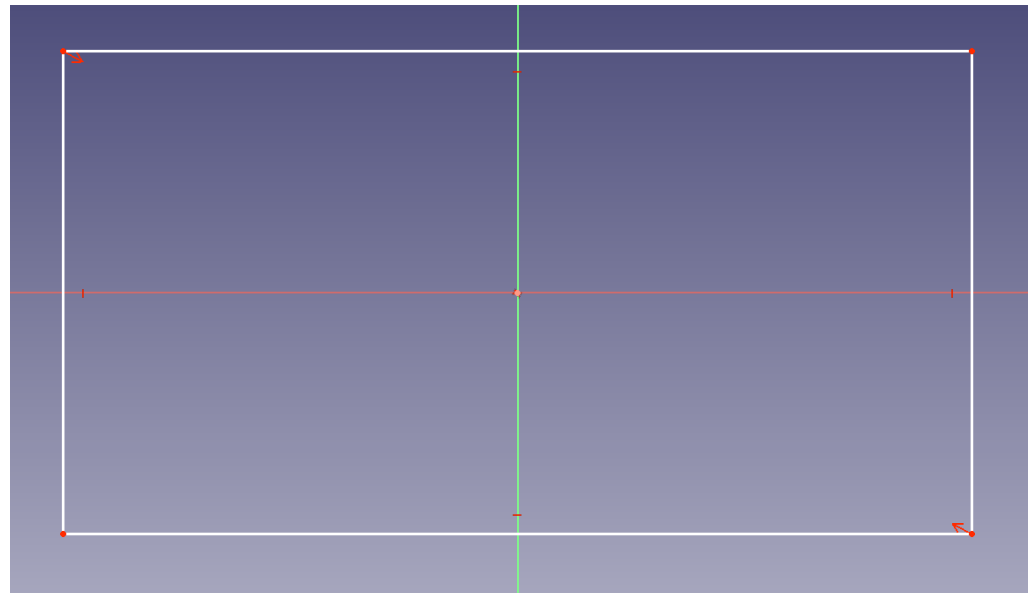
Top Part: The top part has the tools for you to generate simple shapes

Bottom Part: The bottom part helps you to formalize (define) the simple shapes you generated. For example, define the dimensions, angles, coincidence with reference to the coordinate system.

# Space Sketch Design for 2D Poly (7/10,2024)

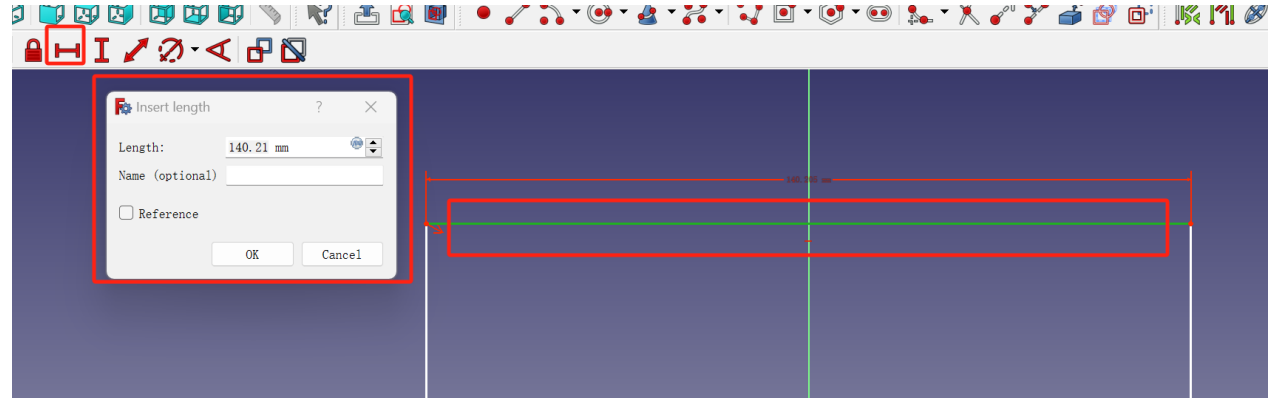


Step 6: For our case. We want to do a space sketch for the instrumentation panels. So we will focus on rectangles and circles. To draw a rectangle, move your mouse to the drop-bar beside the rectangle icon. Centered rectangle is highly recommended every time since you always want to start your sketch from the center of the coordinate system.



Step 7: Draw the rectangle FROM THE CENTER OF THE COORDINATE SYSTEM

# Space Sketch Design for 2D Poly (7/10,2024)

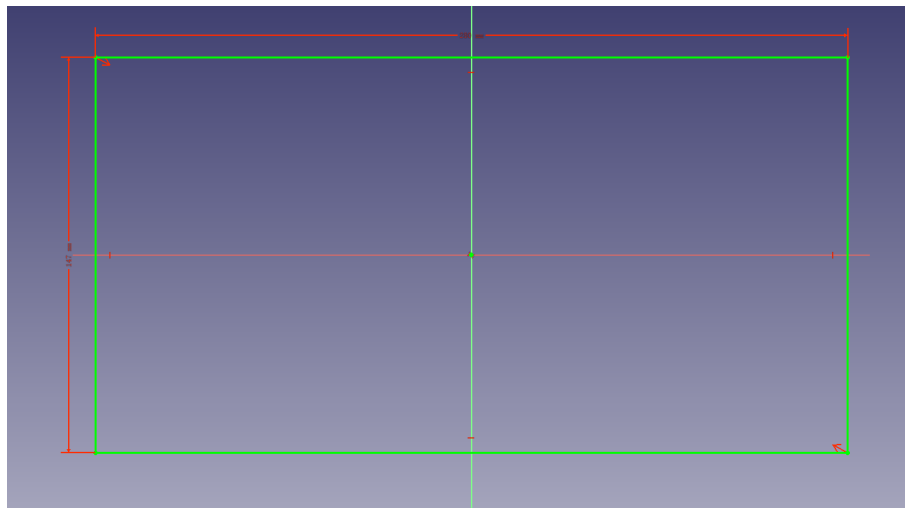


Step 8: Use these tools to define the dimensions of your rectangle.

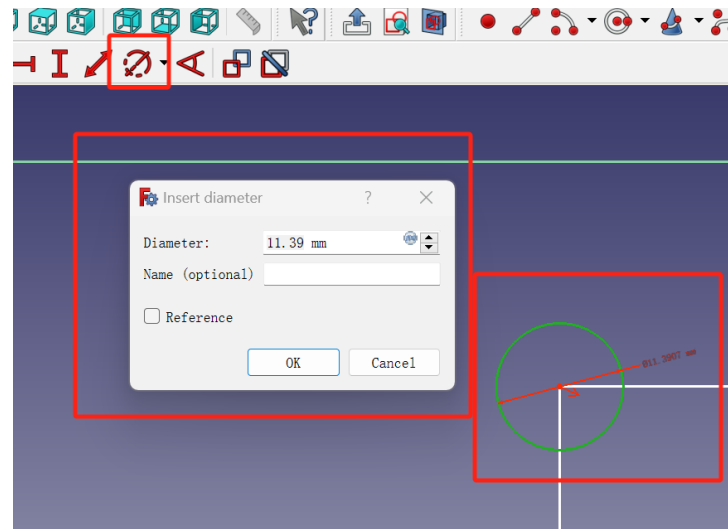
- 1<sup>st</sup> icon: Horizontal distance
- 2<sup>nd</sup> icon: Vertical distance
- 3<sup>rd</sup> icon: Any type of distance between the selected two points
- 4<sup>th</sup> icon: Diameter of the circle

To do this. Select the line you want to define the dimensions. And then click the corresponding icon. In this case, we want to define the horizontal line of the rectangle, then choose the horizontal distance icon from the tool bar. And then enter the ideal value. In this case, it would be 280mm.

# Space Sketch Design for 2D Poly (7/10,2024)

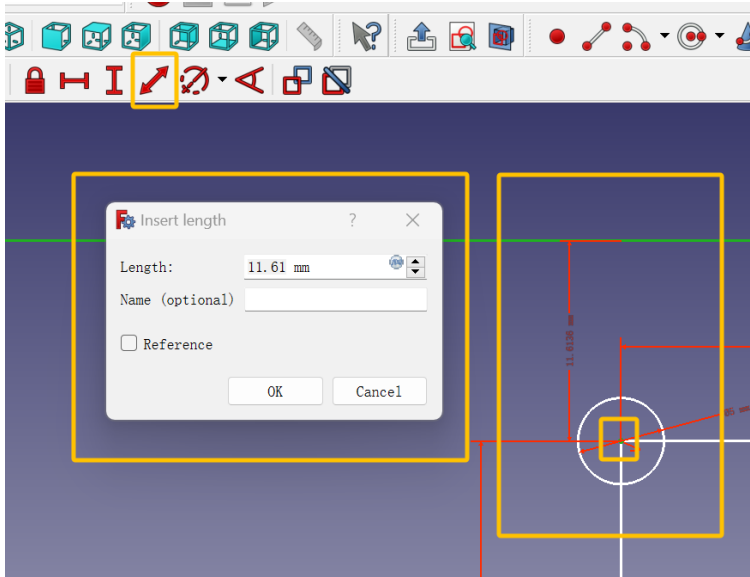


Step 9: After your rectangle is fully defined. The fully-defined shape will turn into green color. REMEMBER: Always define your shape before go into the next step. You don't want any random shape to appear later in the design process.

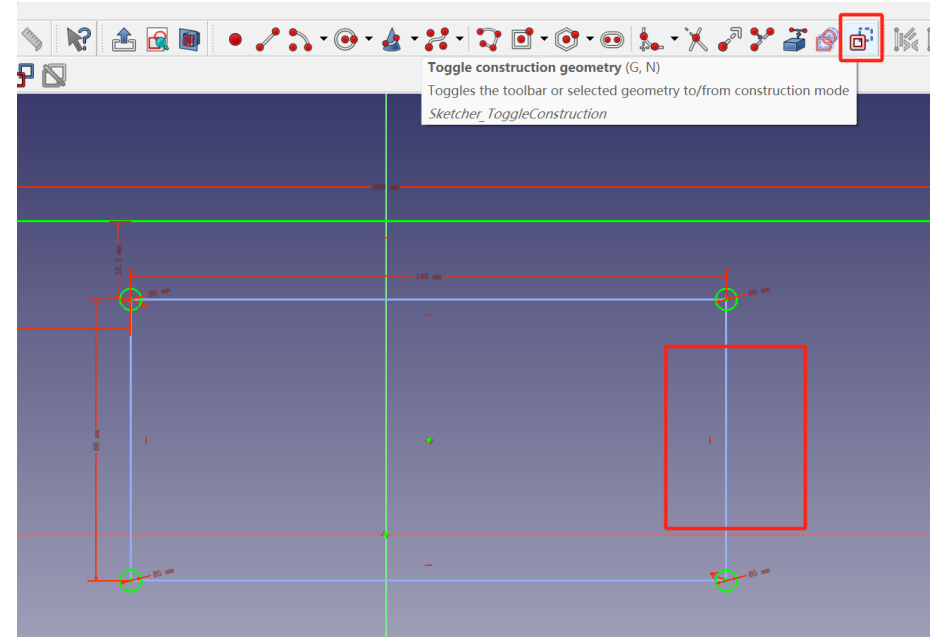


Step 10: Add holes and other components space sketch onto the sketch you have already done. For circle cases, use the “diameter” icon to define the shape of the circle.

# Space Sketch Design for 2D Poly (7/10,2024)



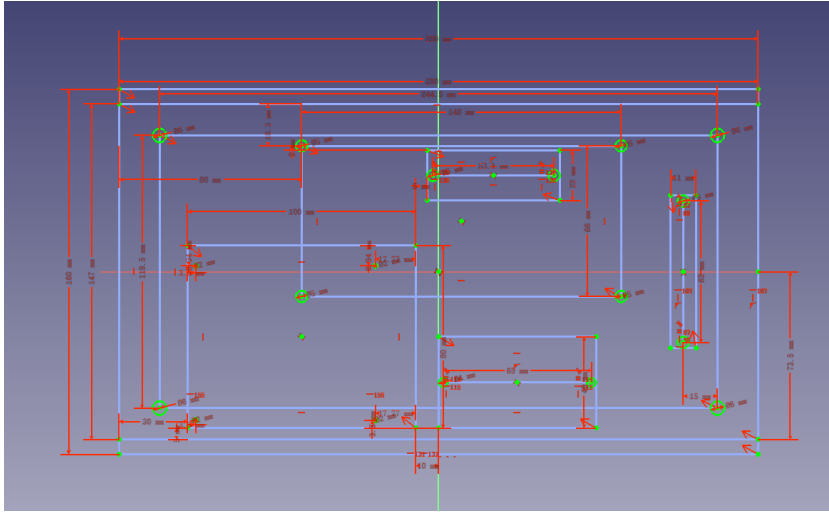
Step 11: In most of the cases, only define the vertical and horizontal distance is not enough, since you have not defined where the general shape is located. Therefore, you might also need to add more dimensions onto your shape. For example in this case, to define the distance between the point and the side, click on the icon in the box and enter the value.



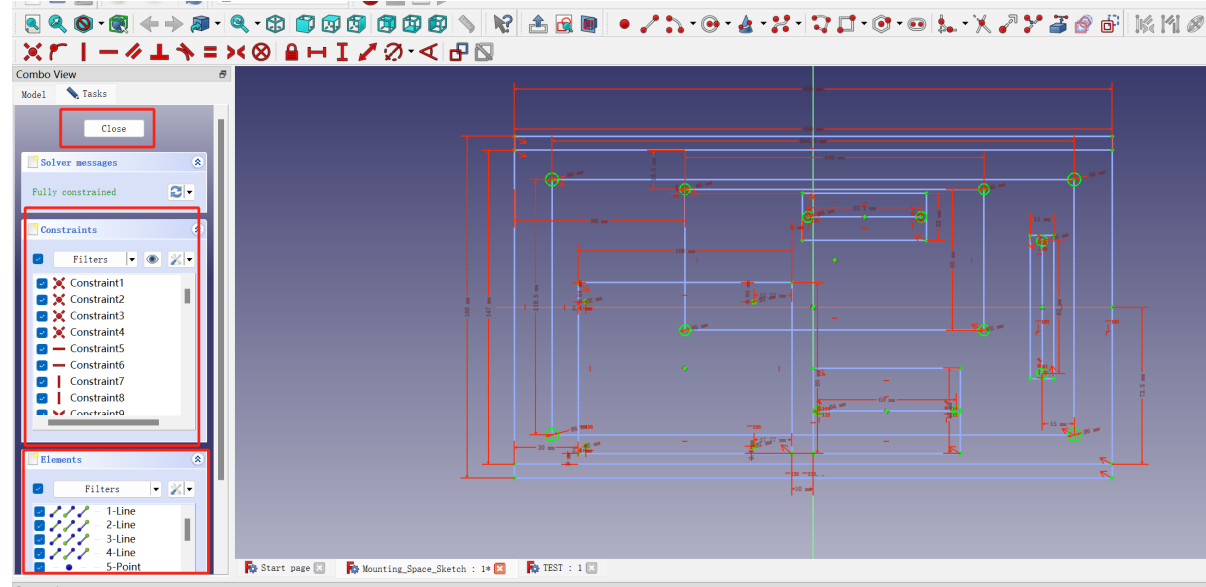
Step 12: It's also important to identify which part of the drawing is necessary to be shown on the space sketch. In this case, the main point of our space sketch is to locate the holes and align the electrical components. So the main point is the holes. Therefore, we could make other lines representing the electrical components "construction lines". Click the icon on the top to make the existing line constructional. The construction lines will be shown in BLUE.



# Space Sketch Design for 2D Poly (7/10,2024)

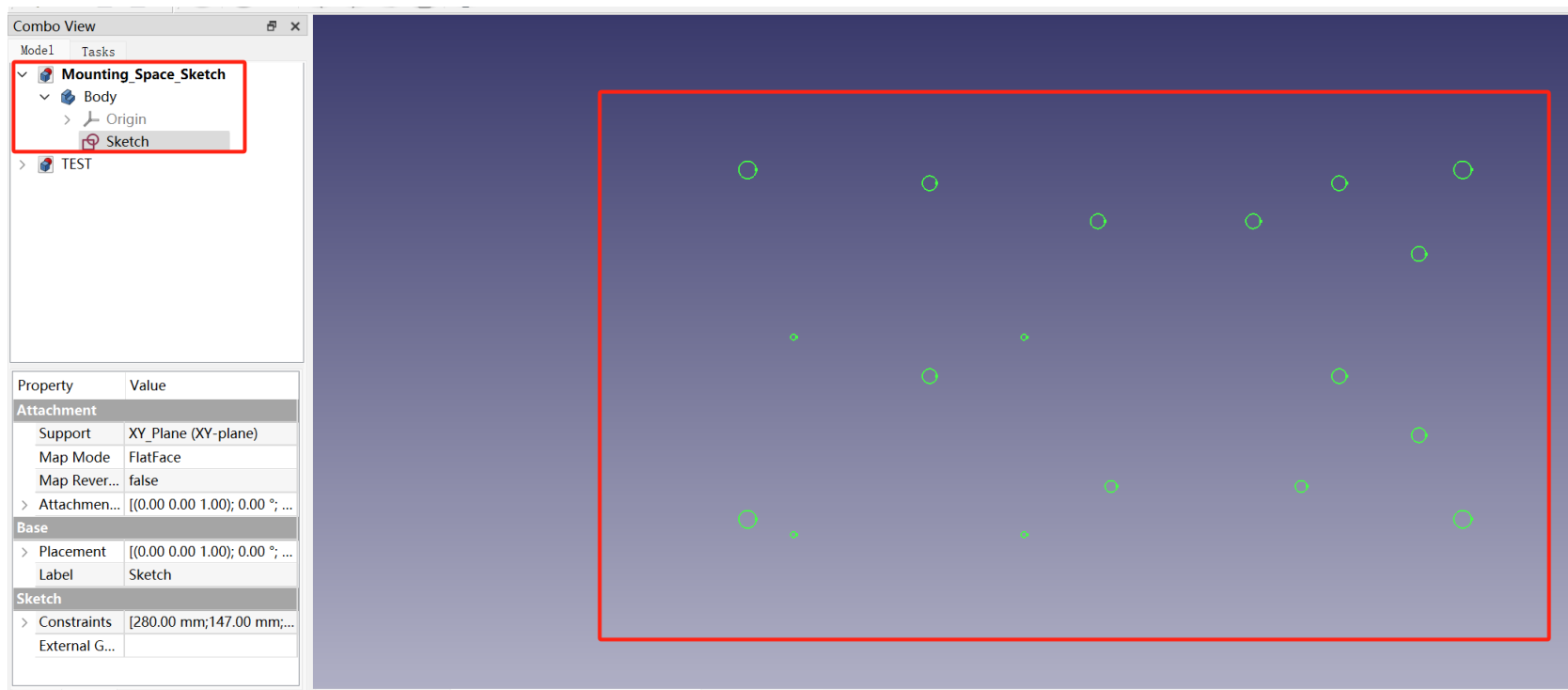


Step 13: Continue this process for every other electrical components to fit in the instrumentation panel. This is what I got in the end. The general space sketch with all the components included.



Step 14: If you want to change any of the dimensions. Check the left menu. All the constraints being added before could be found over here. Just select the constraints you want to modify and delete them. After deleting, follow the same procedures to add a new updated one.

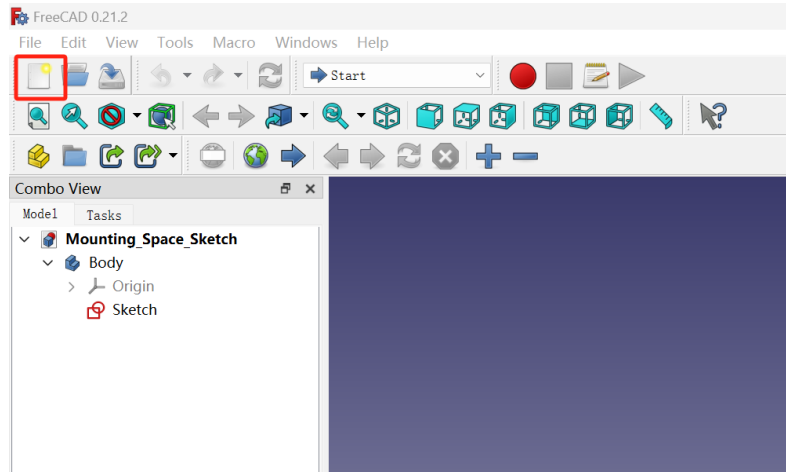
# Space Sketch Design for 2D Poly (7/10,2024)



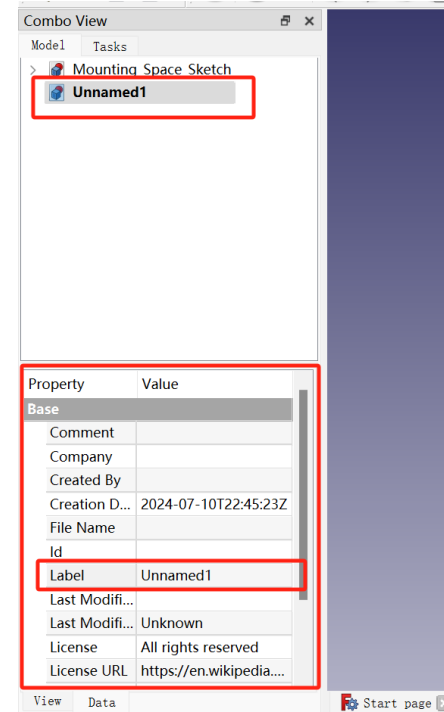
Step 15: After finishing the drawing. You can see the left menu become a property tree. This simply shows all the bodies, components, as well as sketches you have got in the drop-down menu. And you can see the corresponding sketch (parts) you selected in the property tree to appear on the right. Realizing all the construction lines you added will not shown on the big screen after quitting the sketch section.



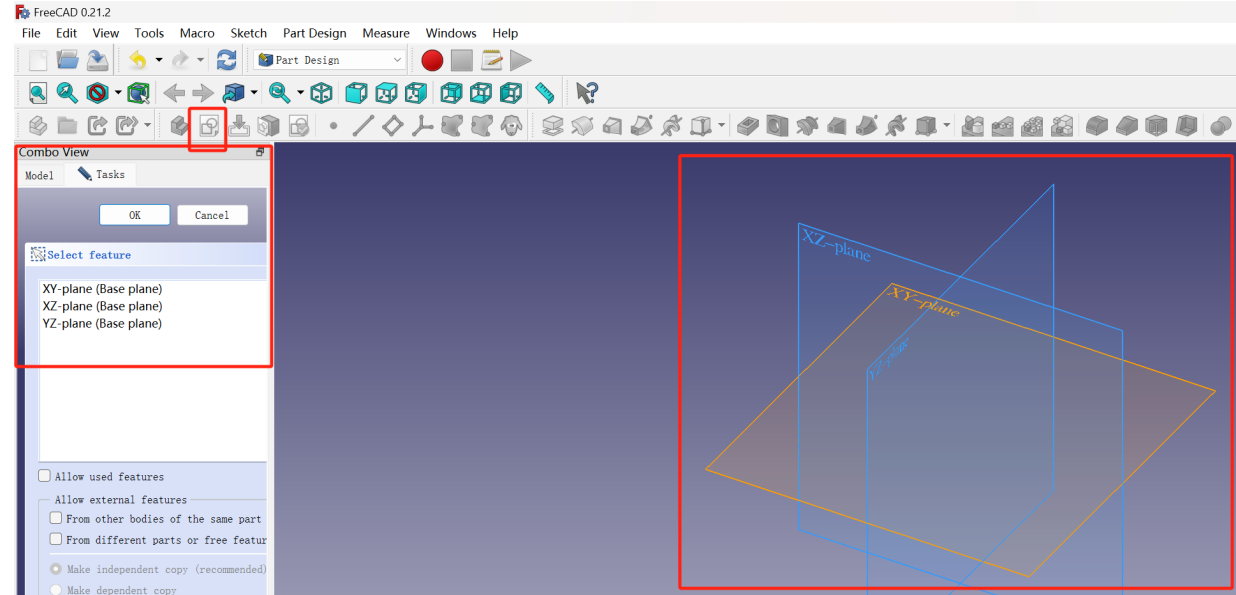
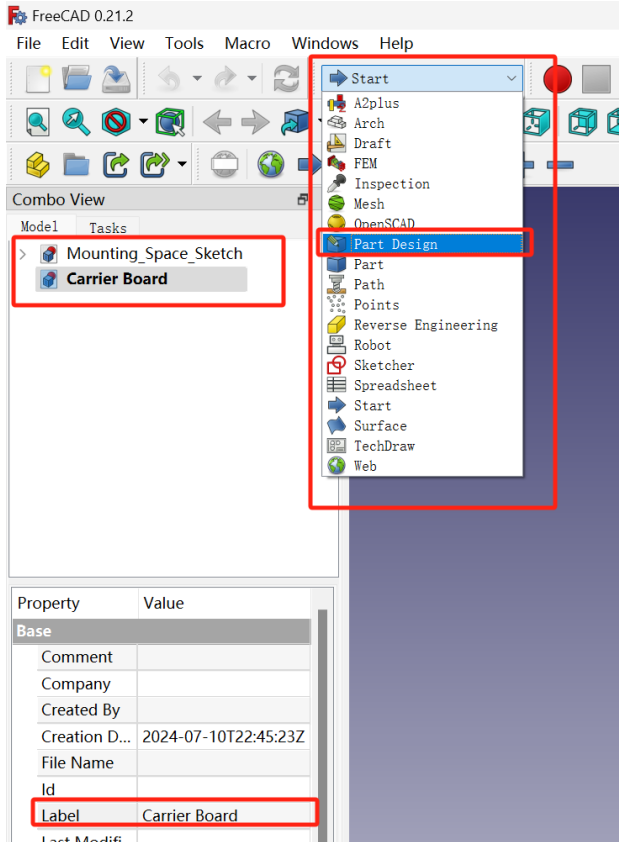
# Design 2D Carrier Board Based on the Space Sketch (7/10,2024)



Step 16: After finishing the space sketch, start a new CAD to design the actual carrier board you want to make. The new CAD will also appear on the property tree. All of the information will be shown below. REMEMBER to change the label of your part into the actual design name. (e.g. Carrier Board)



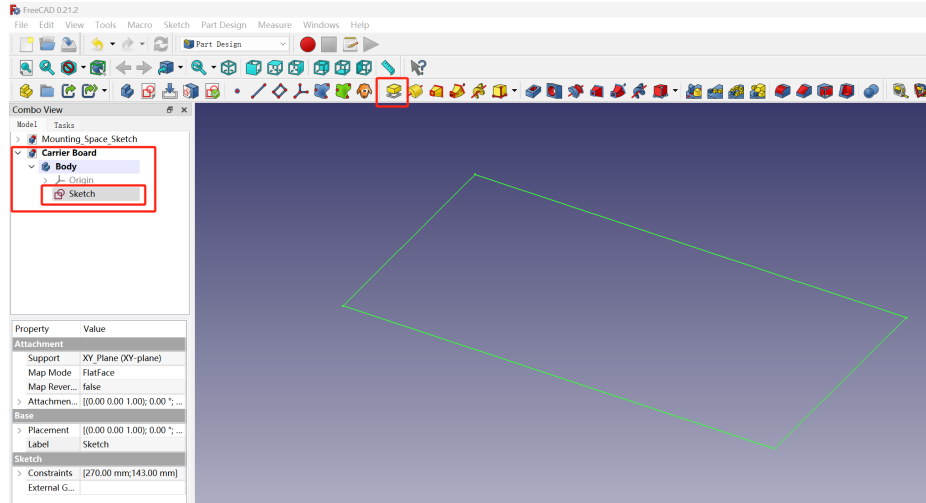
# Design 2D Carrier Board Based on the Space Sketch (7/10,2024)



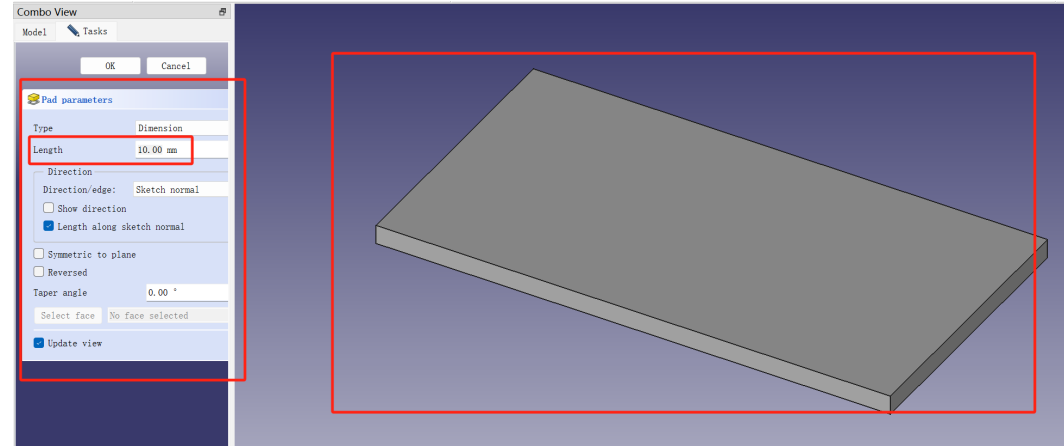
Step 17: Follow the same procedure as the space sketch. Draw the 2D fully-defined rectangle as the carrier board



# Extrusion and Further Added Features from 2D to 3D (7/10,2024)



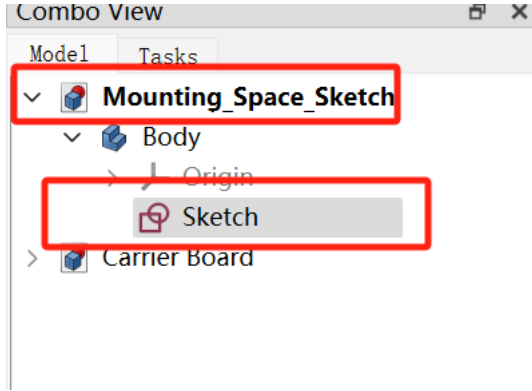
Step 18: To extrude the part, select the targeted sketch in the property tree. Then, select the “Pad” feature on the top tool bar.



Step 19: After clicking on “Pad”, focus on the left menu. “Length” means how thick you want to extrude your part. In our case, it would be 4mm, exactly same with our current panel.

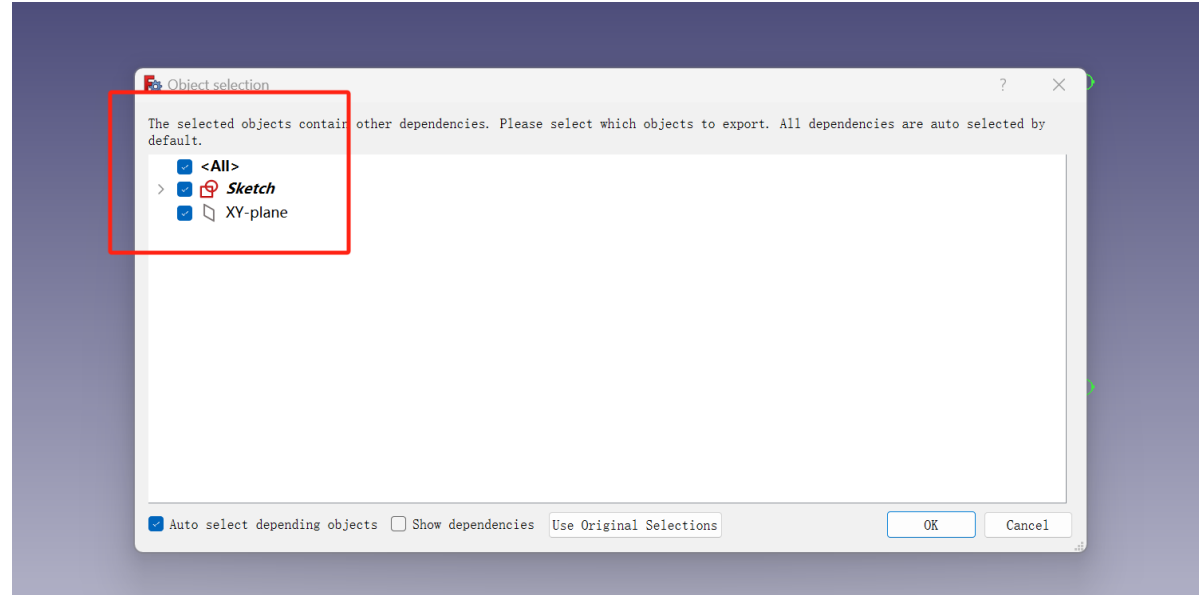


# Extrusion and Further Added Features from 2D to 3D (7/10,2024)



Step 20: Now, we want to put the space sketch on top of the carrier board, in order to exactly locate the hole positions. (The reason why we don't want to draw it separately is to minimize potential errors and maximize the time efficiency.)

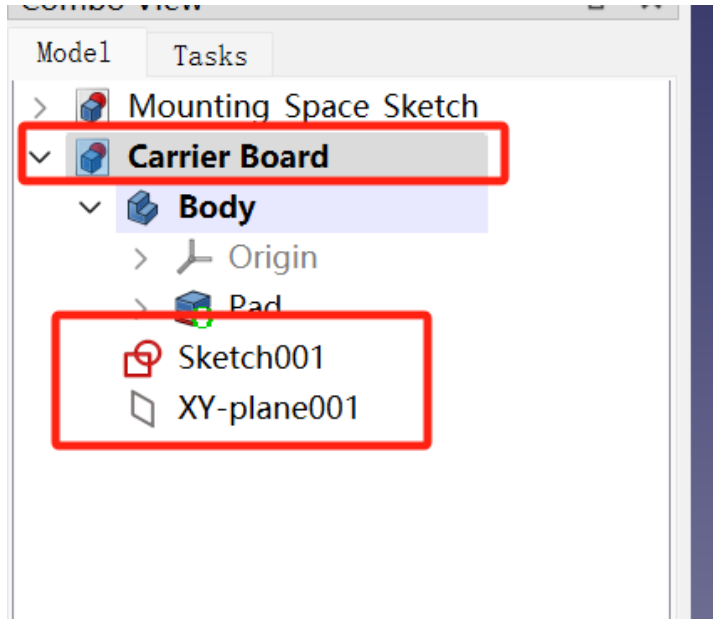
Select the space you created in the property tree (Remember to select another part instead of the carrier board)



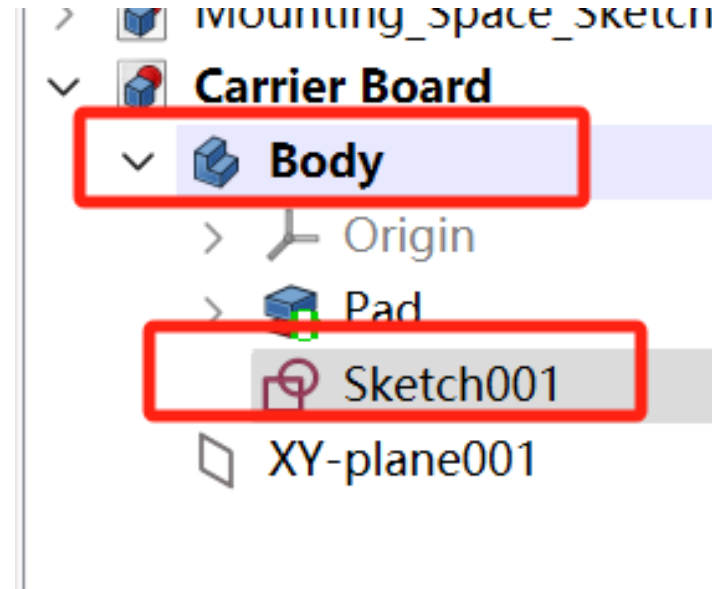
Step 21: Click CTRL C on your keyboard to copy the space sketch, select all parts appeared in this pop-up window, click OK.



## Extrusion and Further Added Features from 2D to 3D (7/10,2024)



Step 22: Select the carrier board part in the property tree. Click CTRL V to paste it under this section

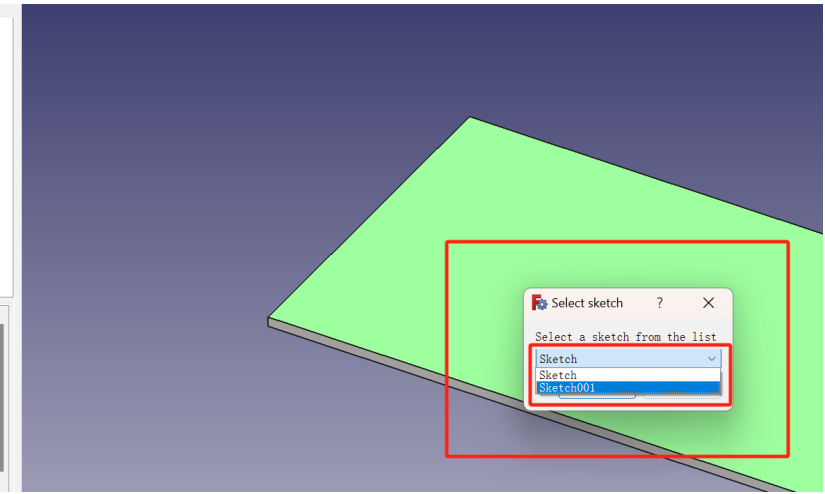
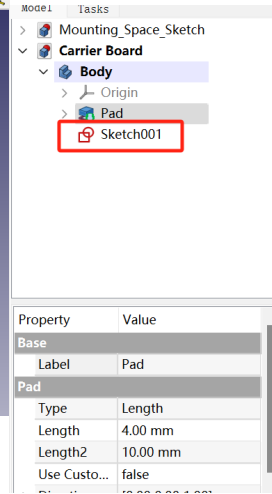
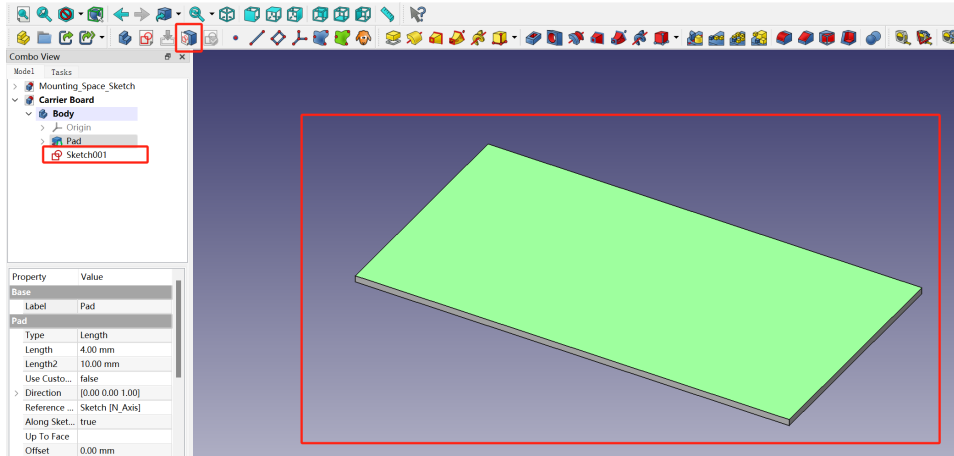


Step 23: Drag your space sketch under the column of BODY.  
(After pasting your sketch, your sketch will not be shown under Body section, which will cause problems for extrude cut later in the process. So PLEASE REMEMBER to drag it under the Body column)

You can delete the Plane you copy paste here after doing this operation



# Extrusion and Further Added Features from 2D to 3D (7/10,2024)

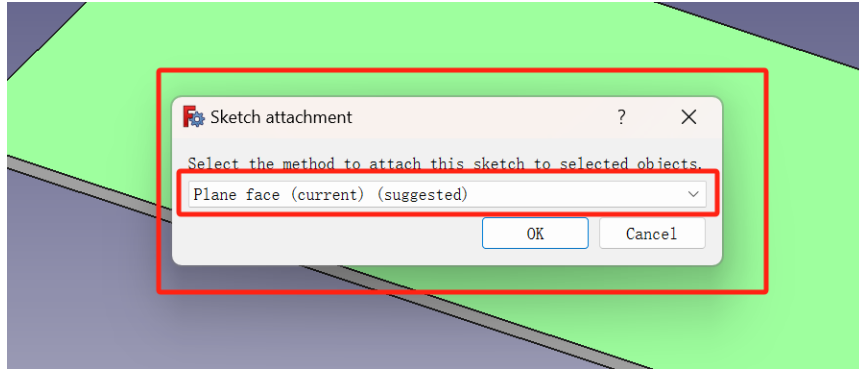


Step 24: Select the targeted surface you want to put your space sketch on. (In this case, it would be the top surface of your carrier board.) Then, select "Map Sketch to Face" icon on the top tool bar

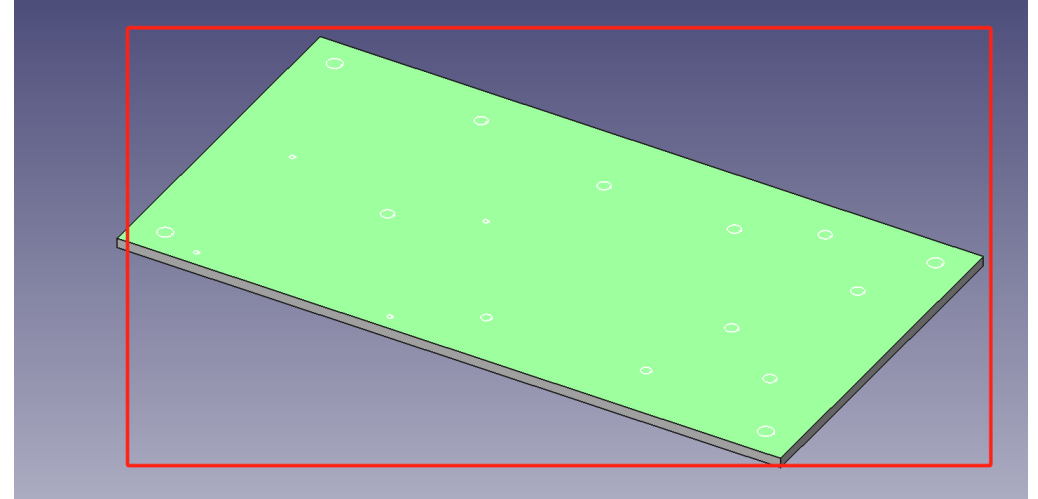
Step 25: Select the right sketch at the pop-up window. PLEASE REMEMBER to match the name of the sketch. In this case, I would select "SKETCH 001" instead of "SKETCH"



# Extrusion and Further Added Features from 2D to 3D (7/10,2024)



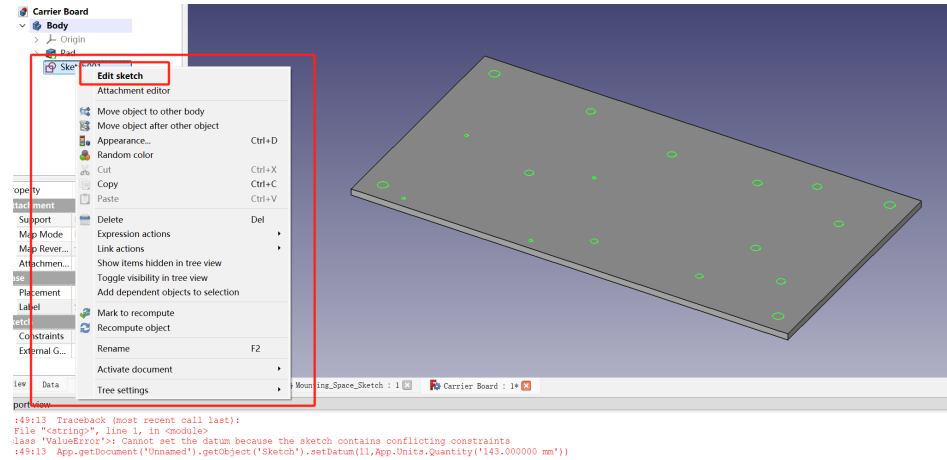
Step 26: Don't change anything in this pop-up window. Plane Face (Current) just means the surface you just highlighted before clicking the icon. Just click OK and move forward. (Double check whether the surface is correct)



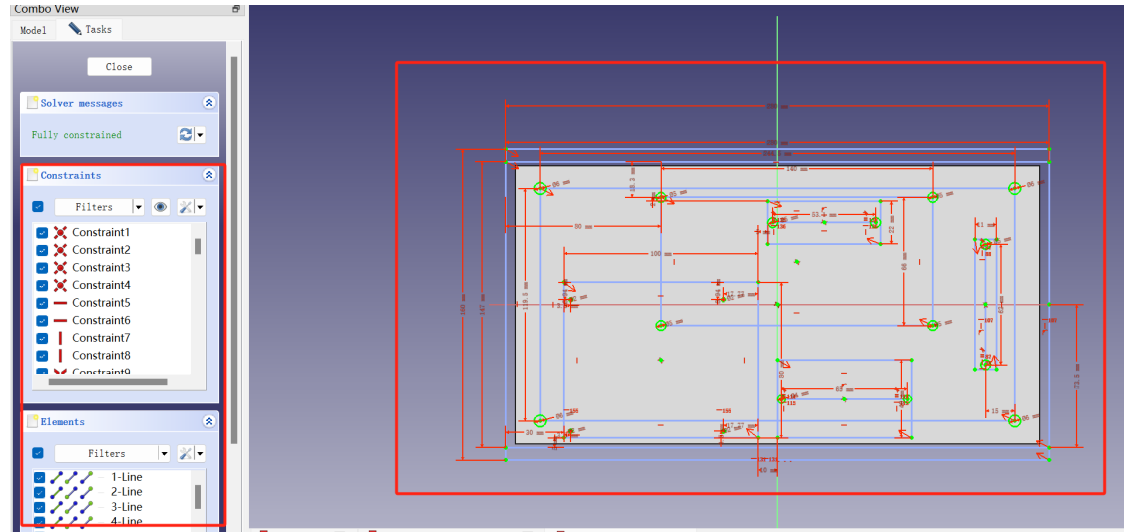
Step 27: After hitting OK, you can see your space sketch appeared to be on the surface you selected.



# Extrusion and Further Added Features from 2D to 3D (7/10,2024)



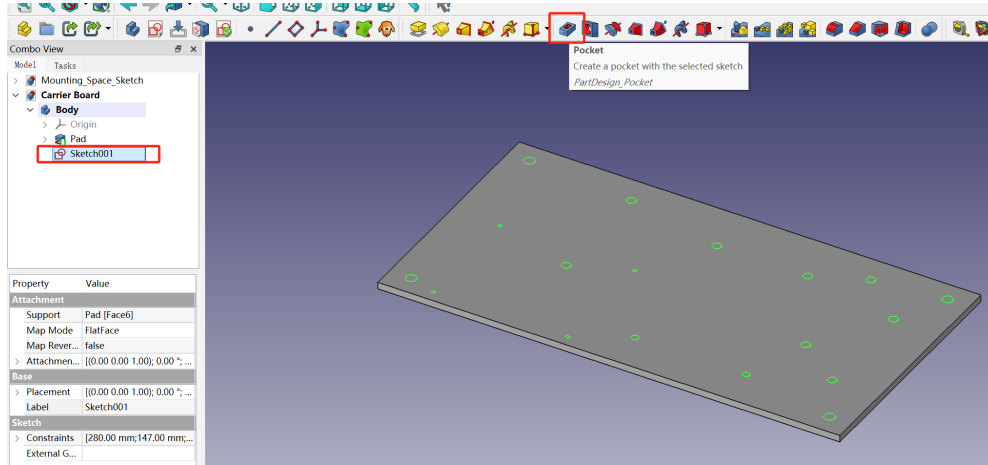
Step 28: If you still want to change anything at this stage, you can right-click the sketch you just put onto the surface in the property tree. Then click Edit Sketch



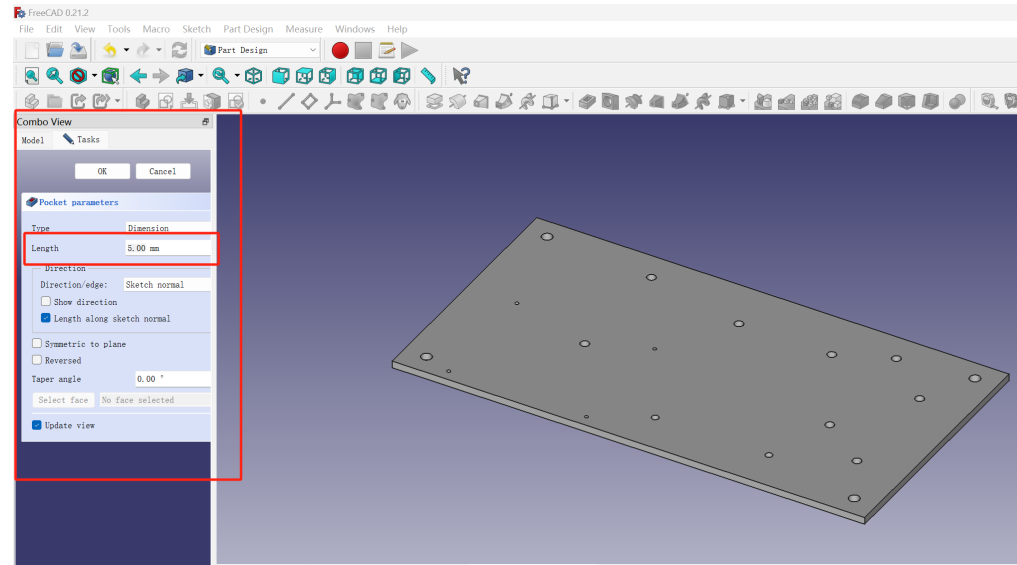
Step 29: Then you can do the similar operations that I shown in STEP 14 to make any modifications you want.



# Extrusion and Further Added Features from 2D to 3D (7/10,2024)



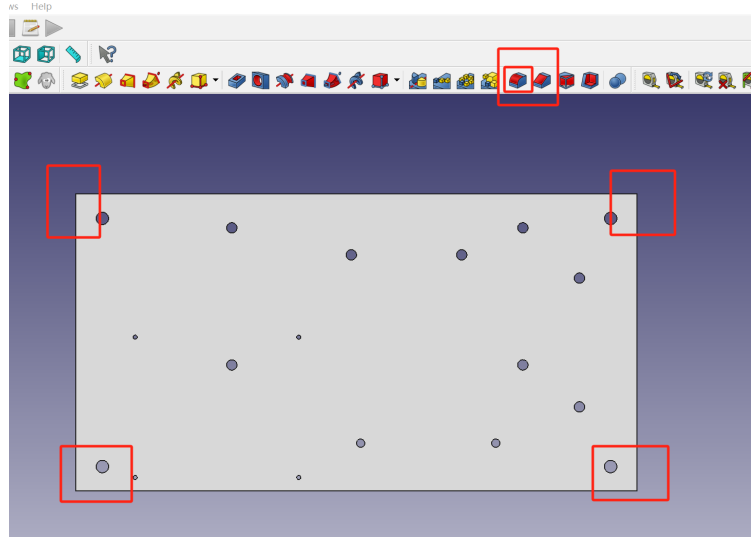
Step 30: To make holes based on your sketch. Select the sketch in the property tree, then click “Pocket” icon on the top of the tool bar.



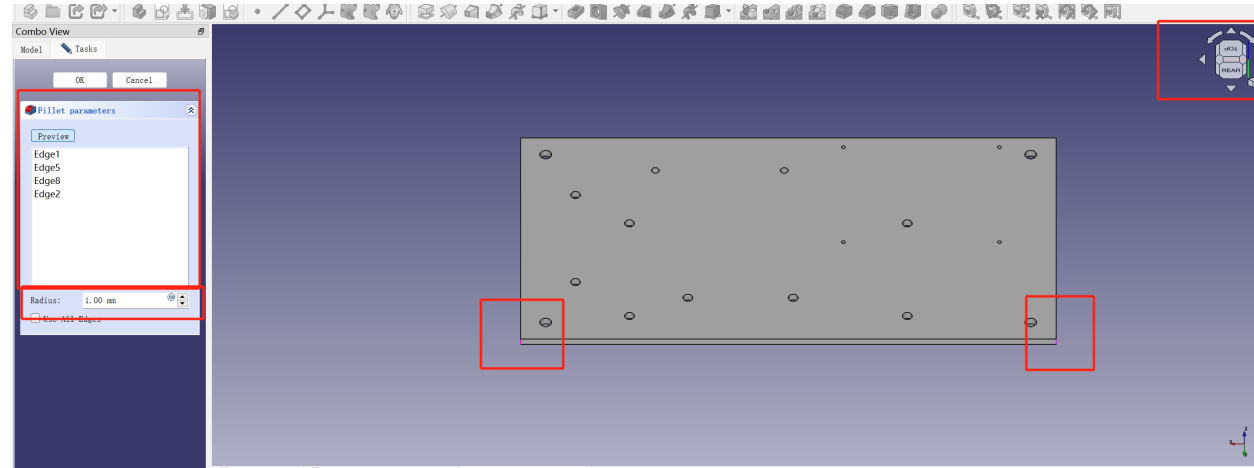
Step 31: Just like extrusion, “Pocket” follows the same type of operations. Length means how deep you want to cut the shape



# Extrusion and Further Added Features from 2D to 3D (7/10,2024)



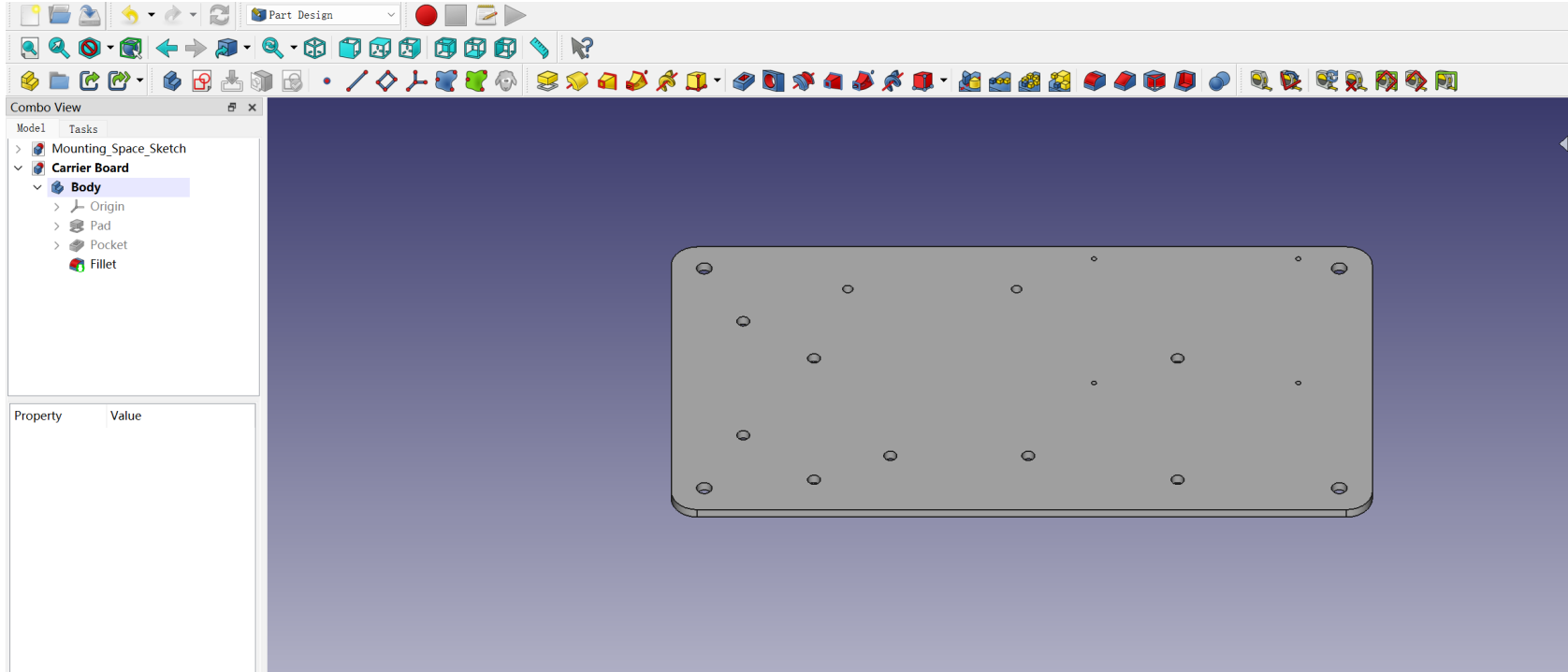
Step 32: After everything is done, don't forget to round the corners. We will use "Fillet" under this situation. Click "fillet" icon on the top of the tool bar.



Step 33: Rotate and move around your part with the top right arrows. Select the edges you want to round. The selected edges will be shown in the left menu. Choose your fillet radius. Generally, we start from 10mm, and change according to the result.



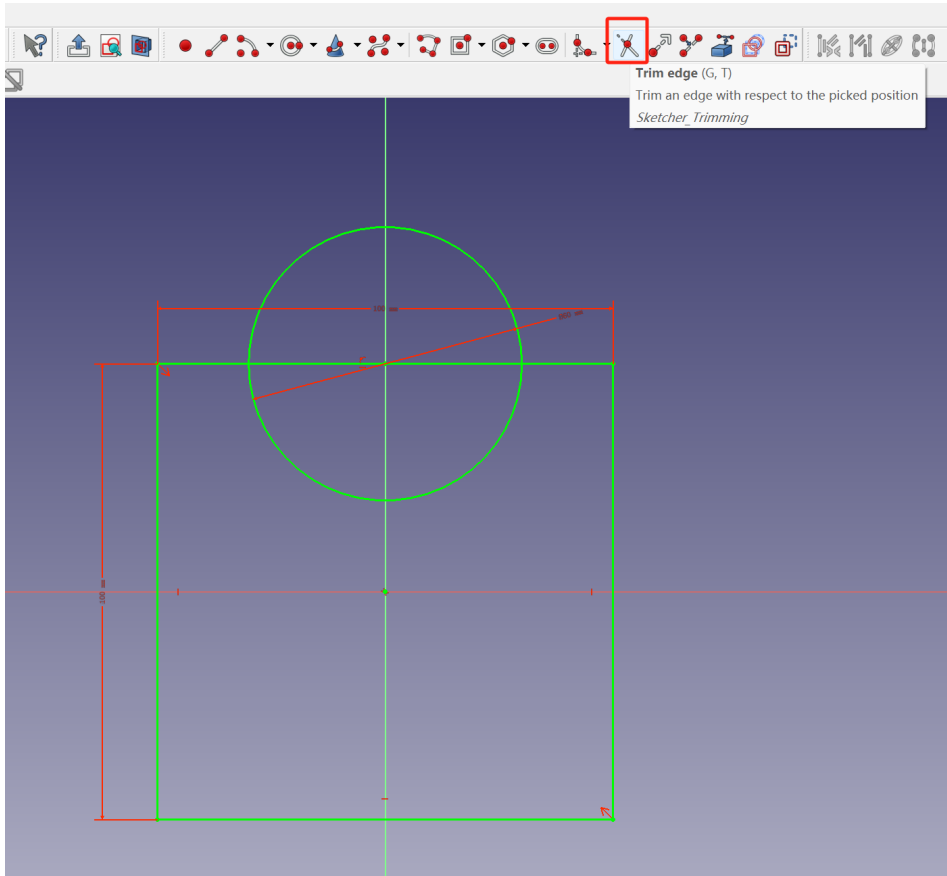
# Extrusion and Further Added Features from 2D to 3D (7/10,2024)



Step 34: After clicking OK. The carrier board is DONE



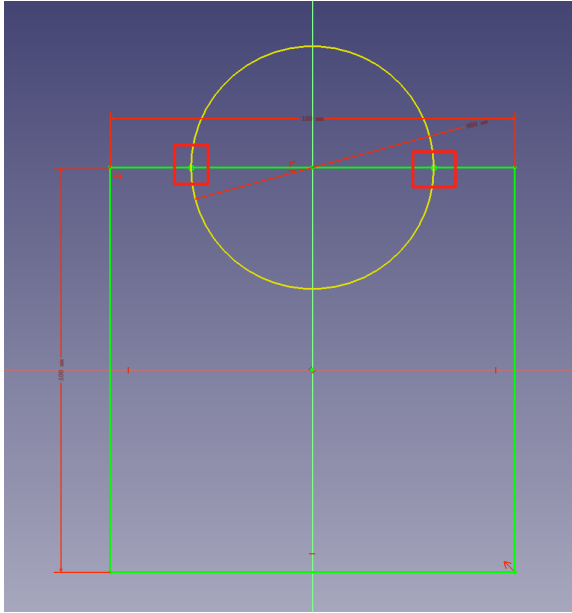
# Trim Tool to fulfill Boolean Operation in 2D Sketches (7/10,2024)



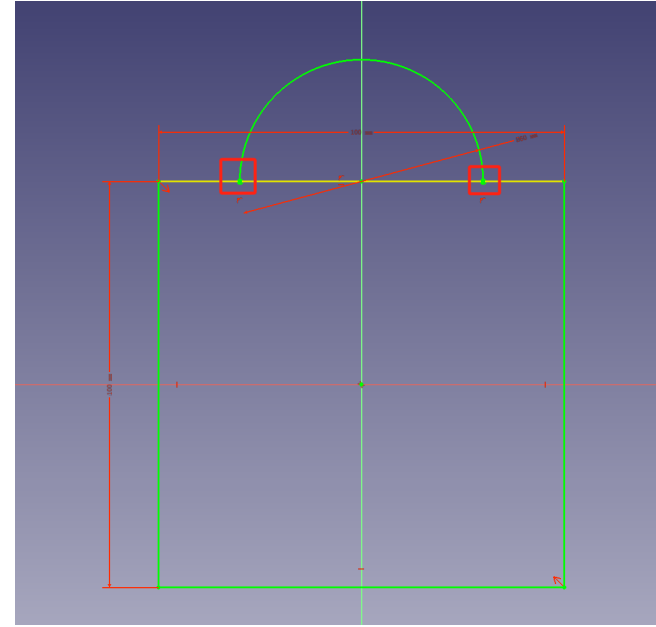
Sometimes you might have similar sketch like this. You know the radius and the dimensions of the circle and rectangle. But all you want is the outer rim instead of the inner lines. In order to get rid of the inner lines, we want to pursue Boolean Operations. However, FreeCAD doesn't have boolean operations for 2D sketches, but only for 3D parts. Therefore, what we are going to do here is called: Trim.

Step 1: Select the trimming tool on the top tool bar

# Trim Tool to fulfill Boolean Operation in 2D Sketches (7/10,2024)

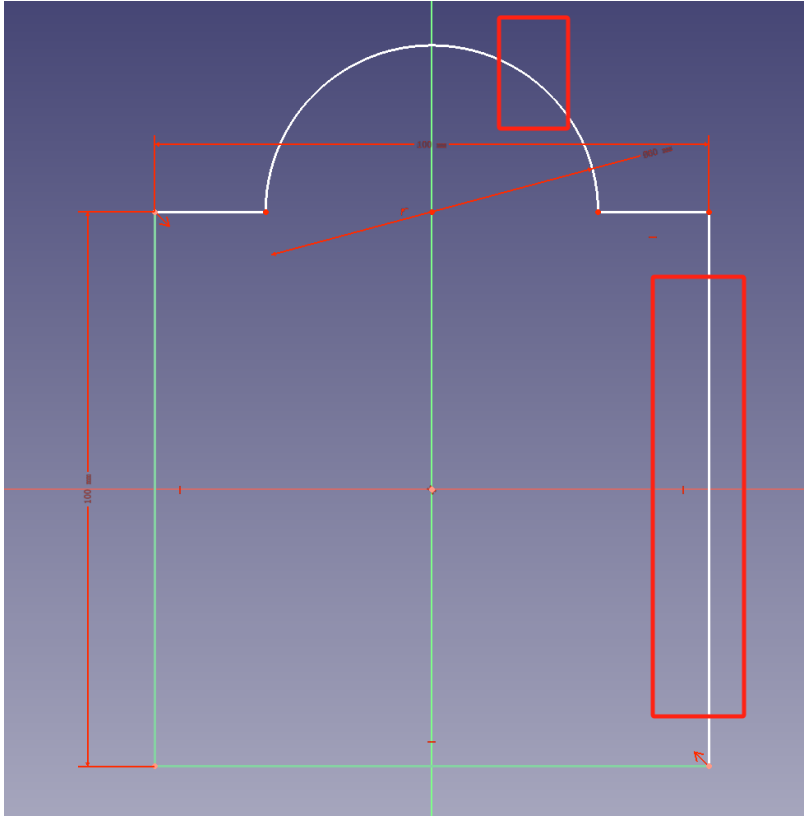


Step 2: Move your cursor to the part you want to delete (Inner lines). FreeCAD will identify the nearest two joint points on the graph. (Shown in the red box). After you see these two joints are been highlighted, left click your mouse

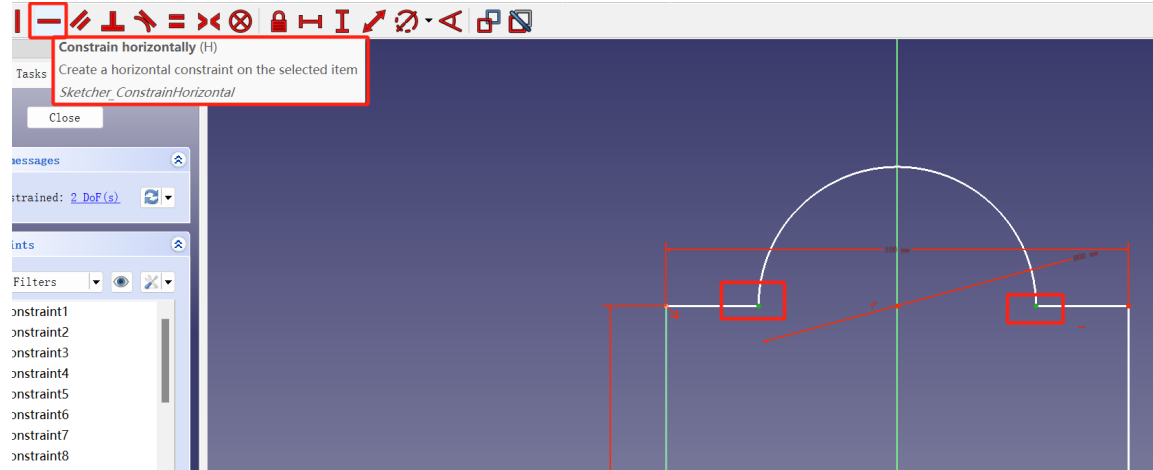


Step 3: Do the same operation again for the other lines you want to delete. For example in this case, it would be the inner horizontal line underneath the sphere.

# Trim Tool to fulfill Boolean Operation in 2D Sketches (7/10,2024)



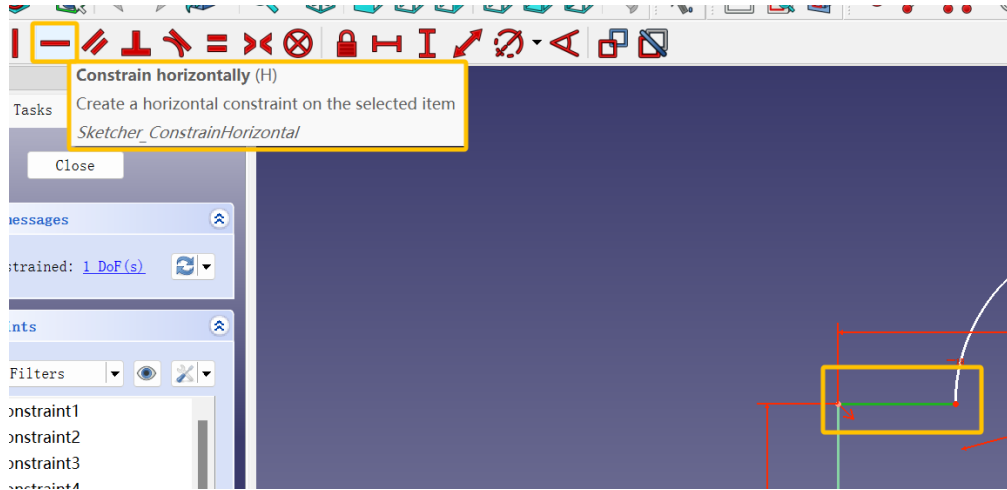
Step 4: After trimming down the shape. You can see they are not fully-defined. This is because they are not full rectangles anymore. You need to add more dimensions to make it fully-defined before moving on to the next part.



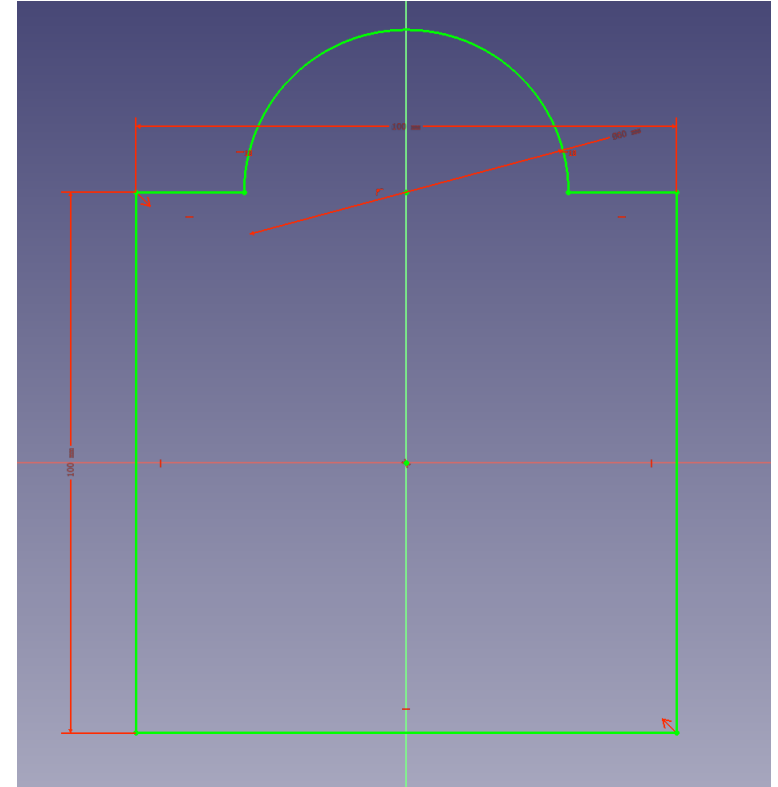
Step 5: In this case, these two points are not on the same horizontal line anymore. Therefore, I need to add a “horizontal constraint” to these two points. Simply select the two points you want to modify, then click on the “horizontal constraint” icon.



# Trim Tool to fulfill Boolean Operation in 2D Sketches (7/10,2024)



Step 6: In this case, this line is not defined to be horizontal anymore. Therefore, we need to follow the similar procedure to this line to add the horizontal constraint. Simply select the targeted line, then click on the "horizontal constraint" icon on the top tool bar.



Step 7: After adding extra dimensions, if the shape is fully defined. You can see the color turns from white to GREEN.



# FreeCAD Installation For 3D Design (12/15,2023)

<https://ubuntuhandbook.org/index.php/2019/04/install-freecad-0-18-ubuntu-18-04-16-04/>

Use FreeCAD for 3D CAD design as a complement tool to LibreCAD which is only for 2D CAD design

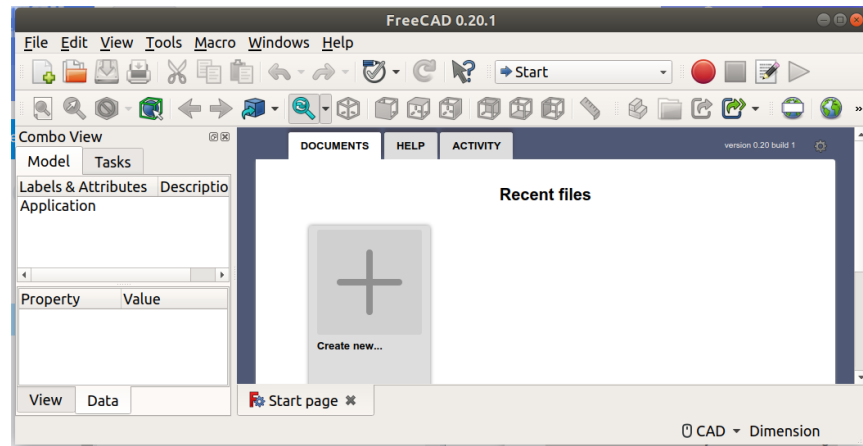
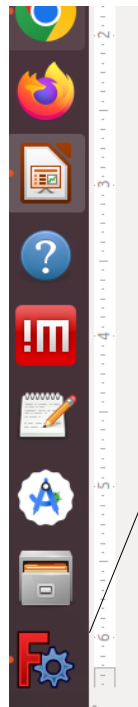
```
harry@harrys-gpu-laptop: /media/harry/easystore3/backup-2020-2-15/CTI/3proejcts/3-8-smart-tech/3-8-4-6-products/AIV200/190p-pcb-sch-assembly/190p-1-pcb-sch/190p-1e-EasyEDA$ sudo apt install freecad
File Edit View Search Terminal Help
/backup-2020-2-15/CTI/3proejcts/3-8-smart-tech/3-8-4-6-products/AIV200/190p-pcb-sch-assembly/190p-1-pcb-sch/190p-1e-EasyEDA$ sudo apt install freecad
Reading package lists... Done
Building dependency tree
Reading state information... Done
The following additional packages will be installed:
  calculix-cx fonts-lyx freecad-common freecad-python3 libcoin80c
  libfreecad-python3-0.20 libmedc1v5 libocct-data-exchange-7.5
  libocct-foundation-7.5 libocct-modeling-algorithms-7.5
Suggested packages:
  calculix fonts-lyx-doc freecad-doc libcoin80c-doc libfreecad-python3-doc libmedc1v5-doc libocct-data-exchange-7.5-doc libocct-foundation-7.5-doc libocct-modeling-algorithms-7.5-doc
```

Once the installation is done, then run it from the ICON

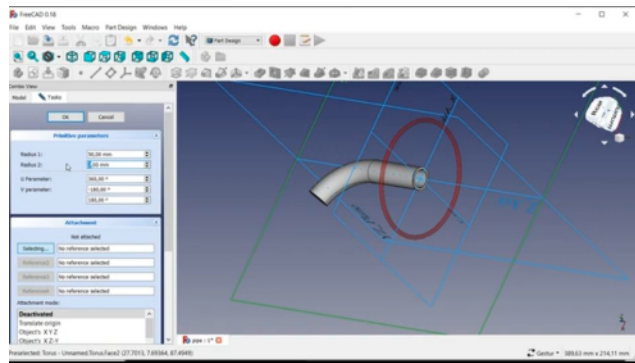
Dodo > add on manager

HL 2024-7-8, udey classes

[https://www.udemy.com/course/freecad-course/?utm\\_source=adwords&utm\\_medium=udemyads&utm\\_campaign=Search\\_DSA\\_GammaCatchall\\_NonP\\_la.EN\\_cc.US&campaigntype=Search&portfolio=USA&language=EN&product=Course&test=&audience=DSA&topic=&priority=Gamma&utm\\_content=deal4584&utm\\_term=.\\_ag\\_162530792106\\_.ad\\_700852016379\\_.kw\\_.de\\_c\\_.dm\\_.pl\\_.ti\\_dsa-1456167871416\\_.li\\_9031968\\_.pd\\_.&matchtype=&gad\\_source=1&gclid=Cj0KCQjW-ai0BhDPARIsAB6hmP6p8iZYkkesQIFi8OaAQnLEEneejbxOI3J1K82qf3QwDuQwysWrRpQaAgxAEALw\\_wcB&couponCode=NVD20PMUS](https://www.udemy.com/course/freecad-course/?utm_source=adwords&utm_medium=udemyads&utm_campaign=Search_DSA_GammaCatchall_NonP_la.EN_cc.US&campaigntype=Search&portfolio=USA&language=EN&product=Course&test=&audience=DSA&topic=&priority=Gamma&utm_content=deal4584&utm_term=._ag_162530792106_.ad_700852016379_.kw_.de_c_.dm_.pl_.ti_dsa-1456167871416_.li_9031968_.pd_.&matchtype=&gad_source=1&gclid=Cj0KCQjW-ai0BhDPARIsAB6hmP6p8iZYkkesQIFi8OaAQnLEEneejbxOI3J1K82qf3QwDuQwysWrRpQaAgxAEALw_wcB&couponCode=NVD20PMUS)



**FreeCAD How To Make Pipe Basic Tutorial For Beginner**  
<https://www.youtube.com/watch?v=HsPhw8Q545g>

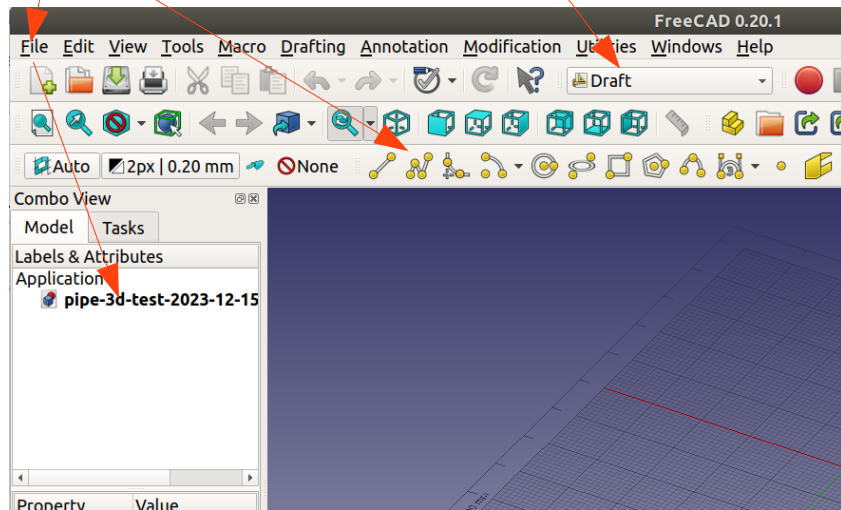


# Steps for 3D Pipe Design (12/15,2023)

## FreeCAD How To Make Pipe Basic Tutorial For Beginner

<https://www.youtube.com/watch?v=HsPhw8Q545g>

File > new project > given a name of the new project and save it in a proper directory as you under “File” to navigate to the right folder location; > select “Draft” at the top window to enable polyline for drawing



Note: Install **GitPython** and **git** are installed on your system, additional macros will be loaded from [FreeCAD-macros](#)

<https://gitpython.readthedocs.io/en/stable/intro.html>

```
#install GitPython
$pip install GitPython
```

<https://www.digitalocean.com/community/tutorials/how-to-install-git-on-ubuntu-18-04>

```
#install git
$sudo apt install git
```

[https://wiki.freecad.org/How\\_to\\_install\\_additional\\_workbenches](https://wiki.freecad.org/How_to_install_additional_workbenches)



# Addon Manager to Add Dodo (12/15,2023)

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Starts from Tools from the tag > addon  
manager > dodo

