Tab and Slot in Fusion 360

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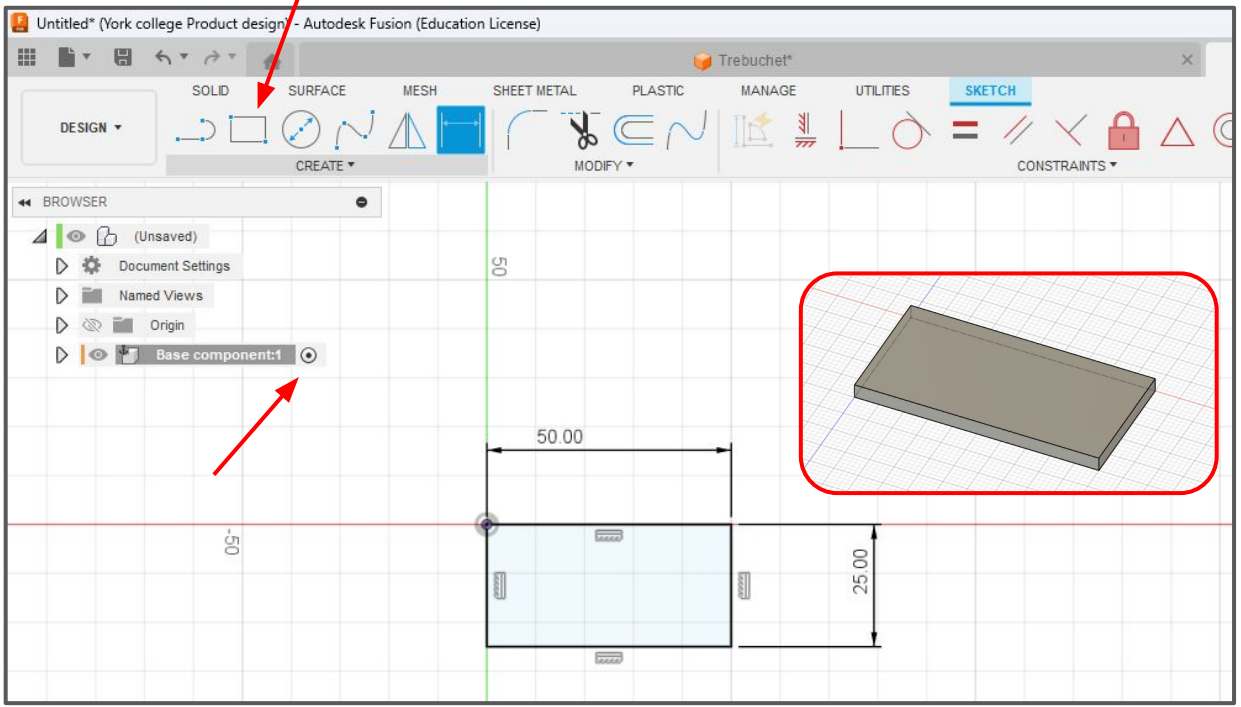
Step 1

Create a new component

Then create a new sketch rectangle 50x25mm

You can then title that base and extrude it by 3mm (the thickness of your MDF)

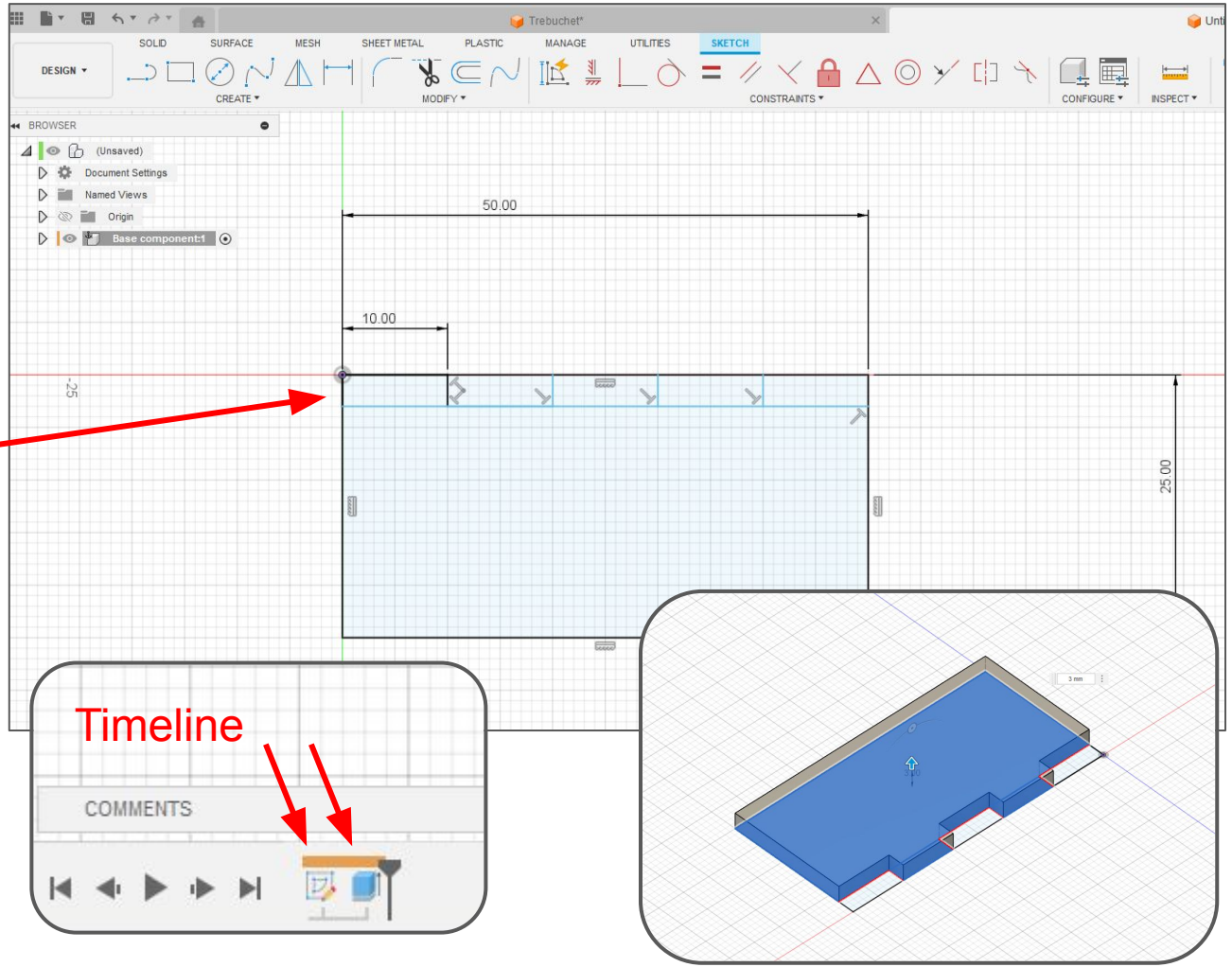
You can select extrude tool or press “E”



Step 2

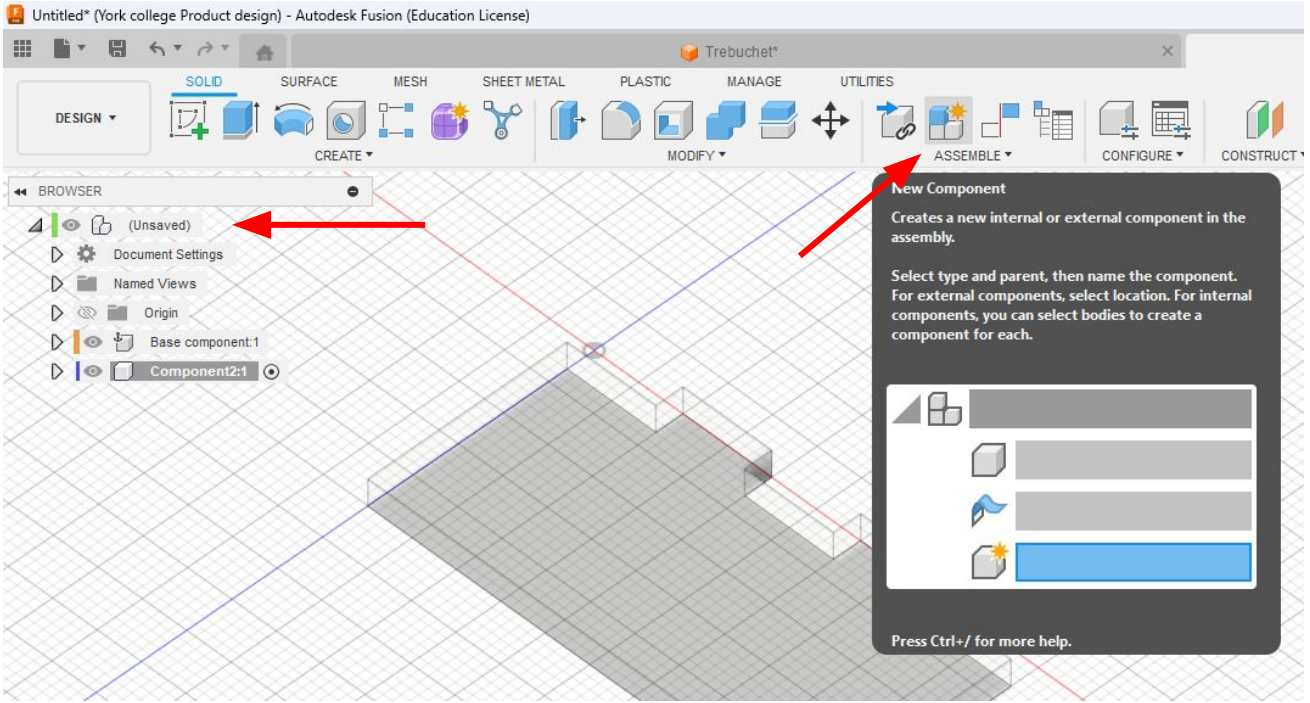
Edit your original sketch (double clicking the sketch function in the **parametric timeline** at the bottom) to include a 3mm finger join at your chosen length.

You can now edit the extrude command in the same way by “ctrl-left clicking” the teeth you've just made



Step 3

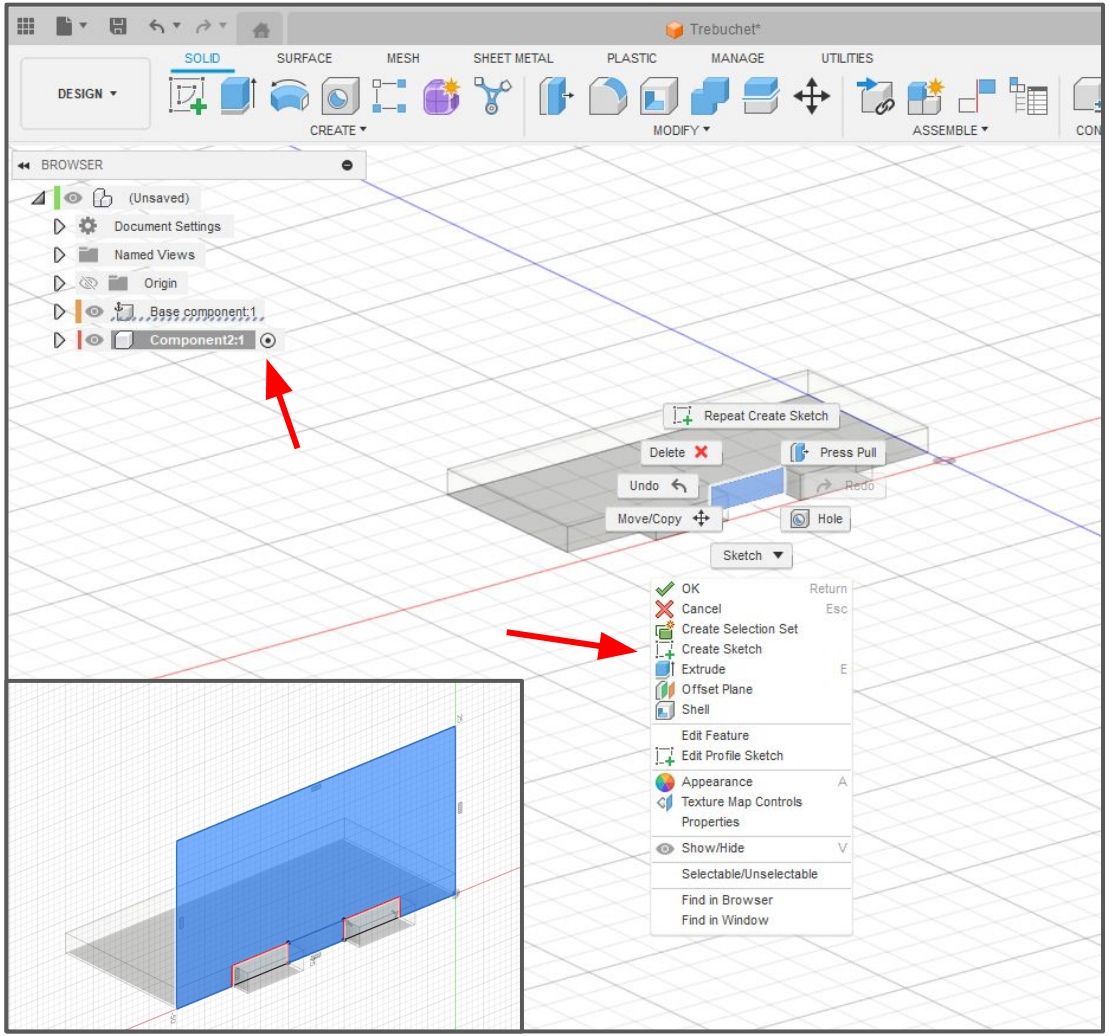
Create another new component - Remember to select the project at the top of your menu (labeled unsaved on this example)



Step 4

With the new component selected you can now right click a face on the existing component “base” and select create sketch from the drop down menu.

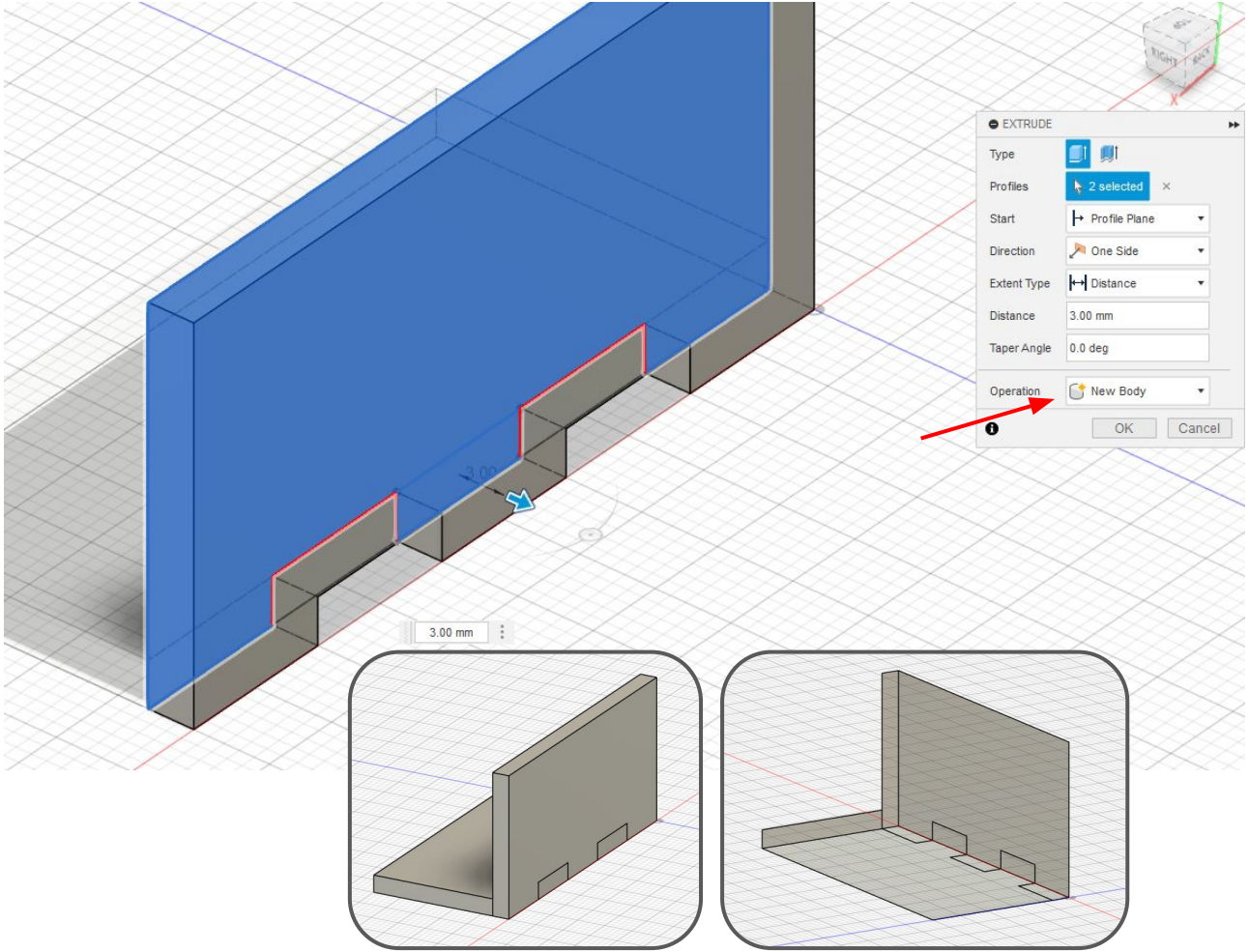
You will now draw your second sketch to be the inverse of the teeth on the original base piece.



Step 5

Extrude this as a **new body** and you will now have 2 components that should interlock.

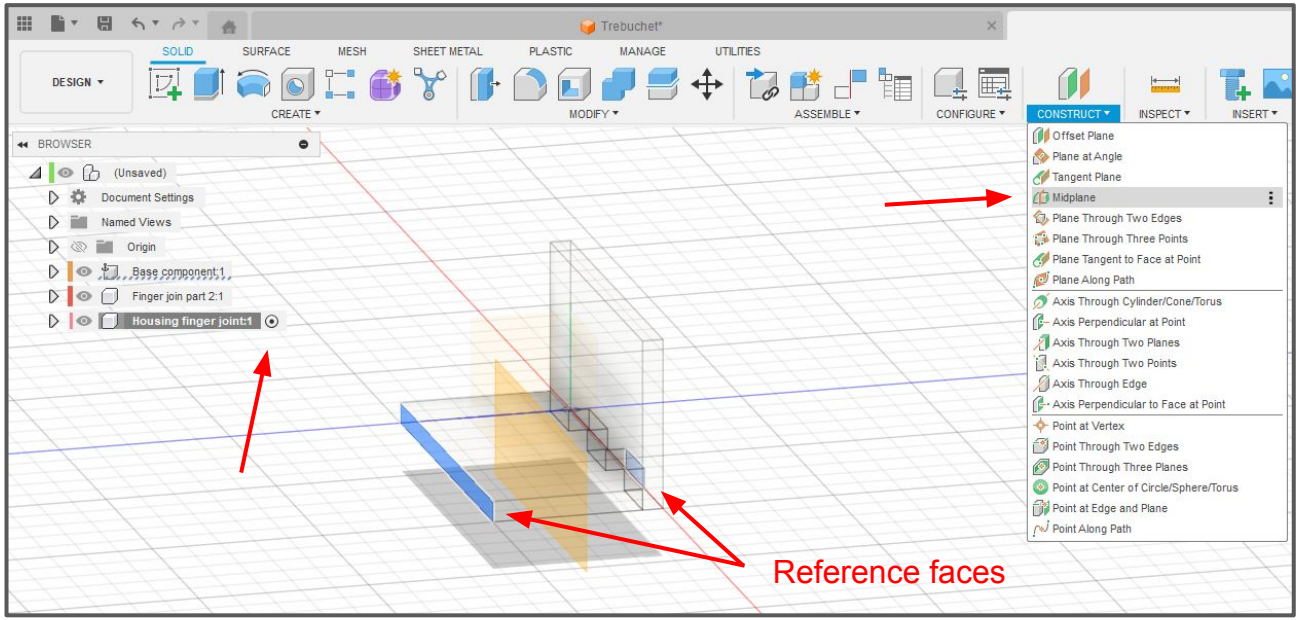
This is called a finger or comb joint and is a great way of connecting 2 pieces of material together



Step 6

After creating another new component, (remember to select the project first) you should then create a midplane using the existing base as a reference component.

This is found in the construct option

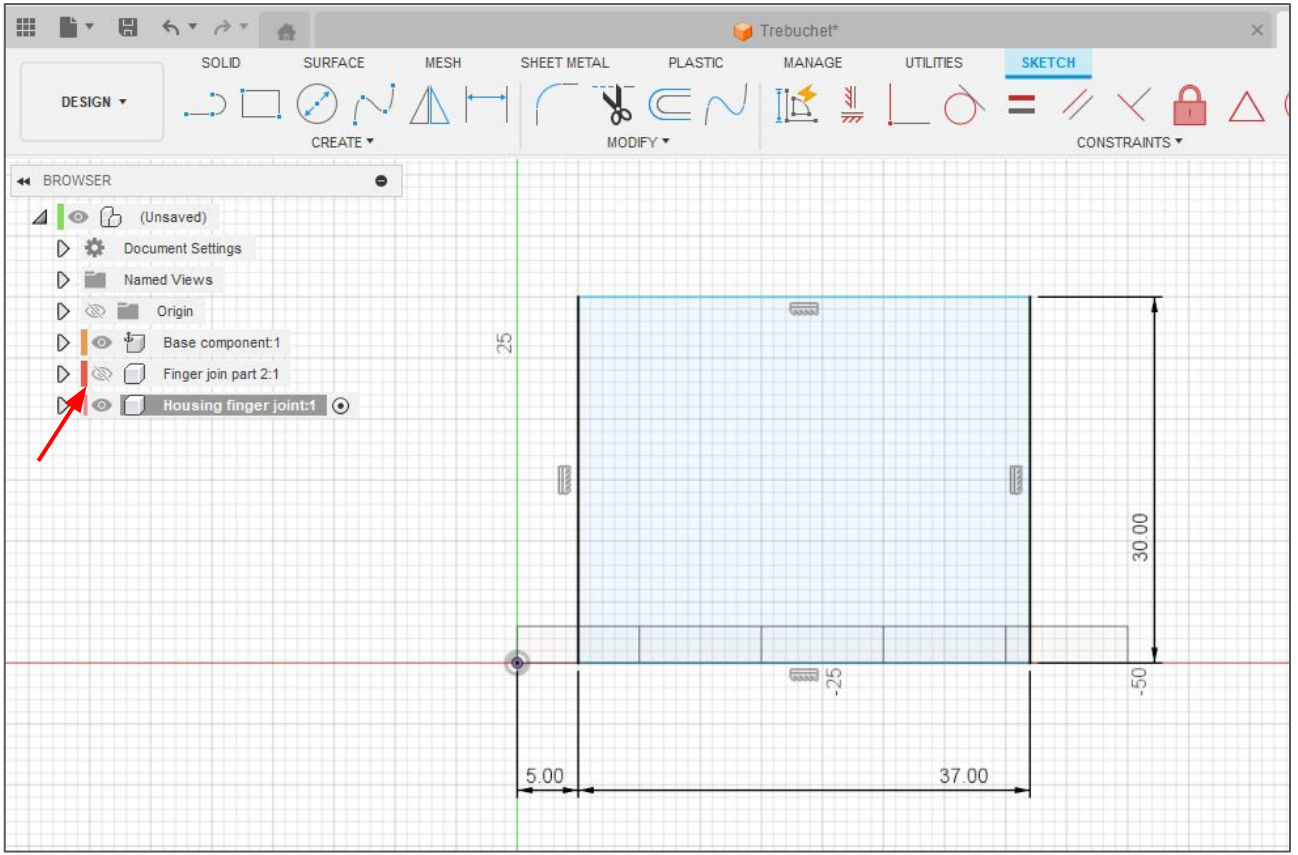


Step 7

Selecting the midplane we created you should now right click it and create a sketch with the new component selected.

You can hide the old components if they are in the way by selecting the “view” / eye icon.

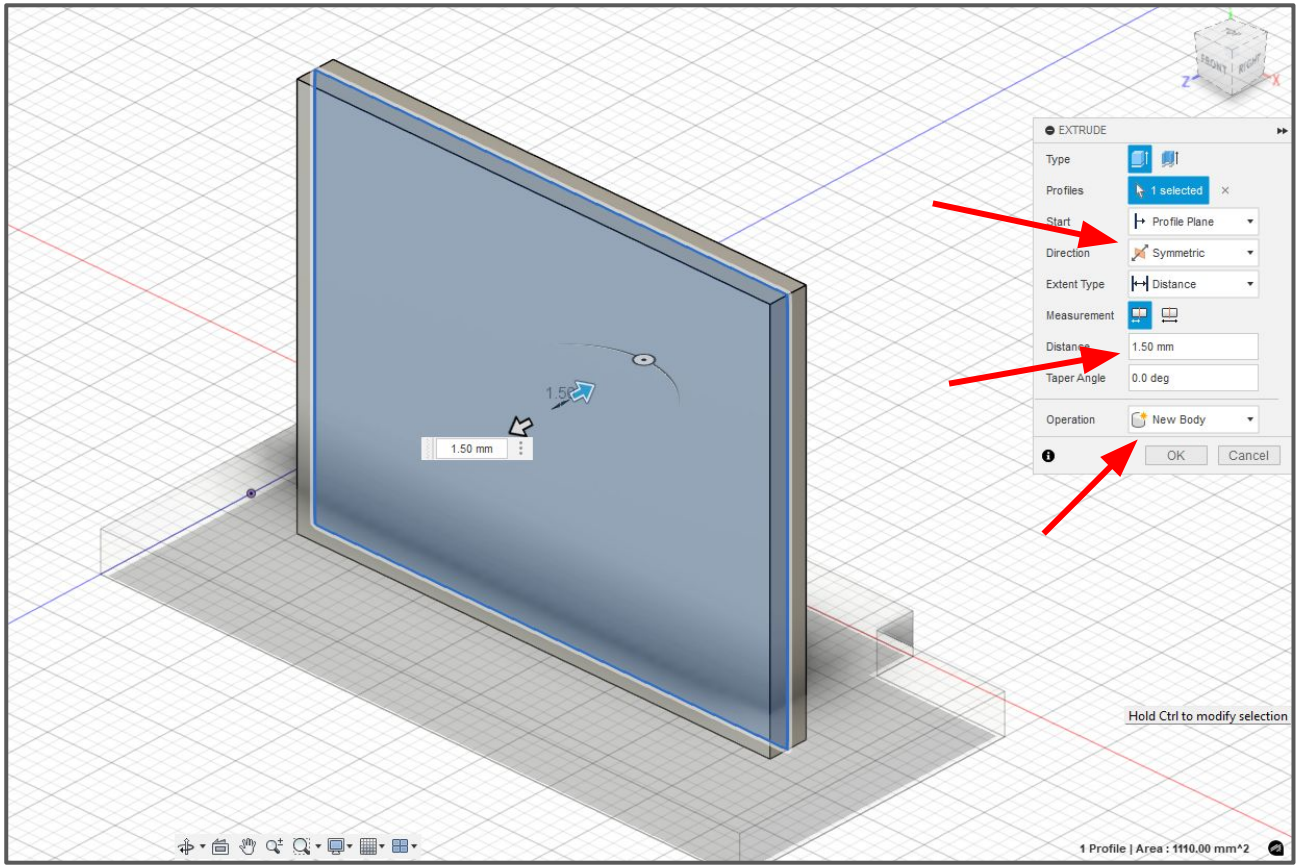
Create a rectangle that is narrower and does NOT span the length this time.



Step 8

Now extrude this component (symmetrically) by 1.5mm totalling 3mm.

It is important to ensure you create a NEW BODY and NOT join the component to the original



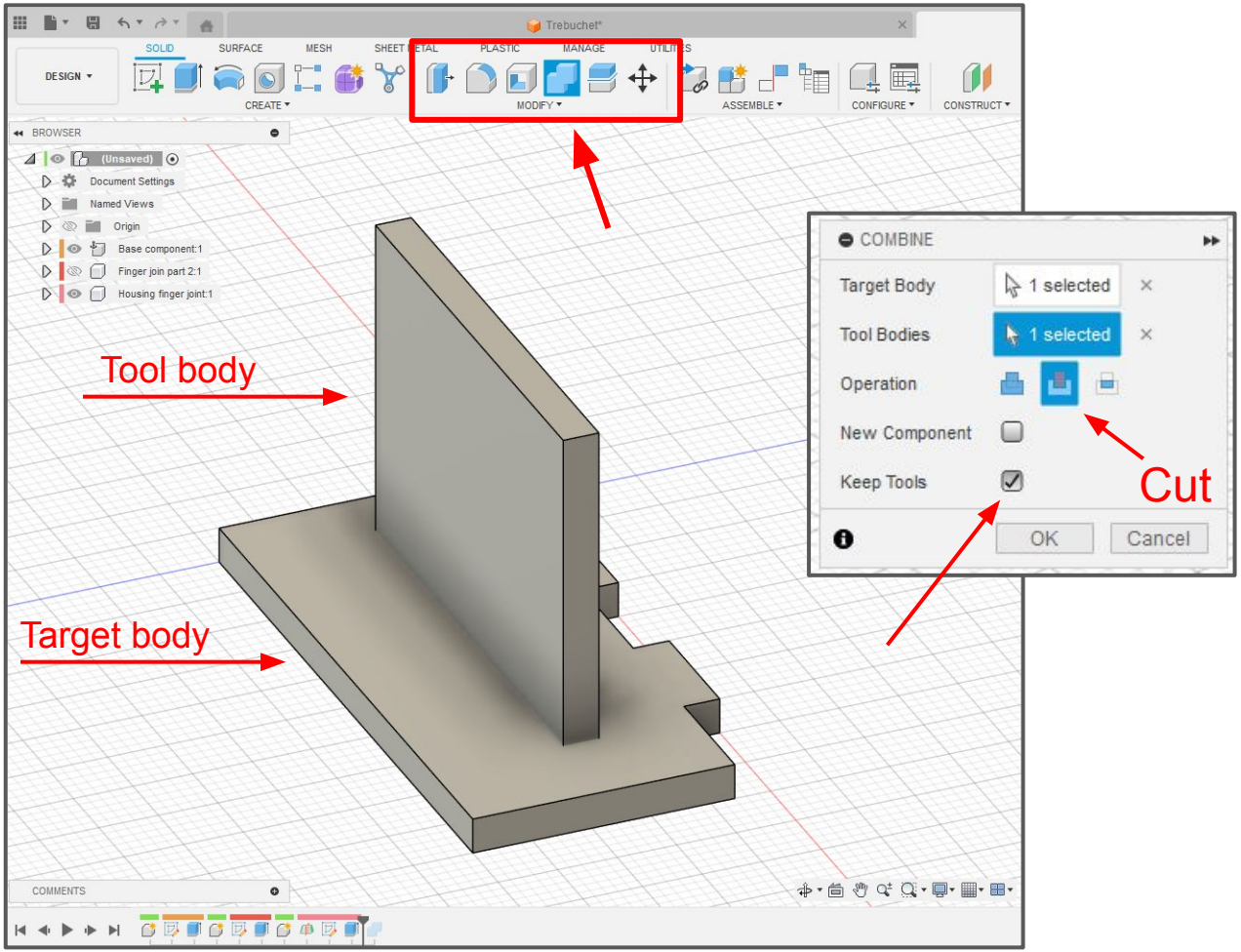
Step 9

Select the project at the top of your menu and then select the “combine” tool under the modify tab.

In the pop up window ensure “target body” is selected and click the base.

Then select tool body and select the rectangle we just created.

Ensure cut is selected and keep tools.

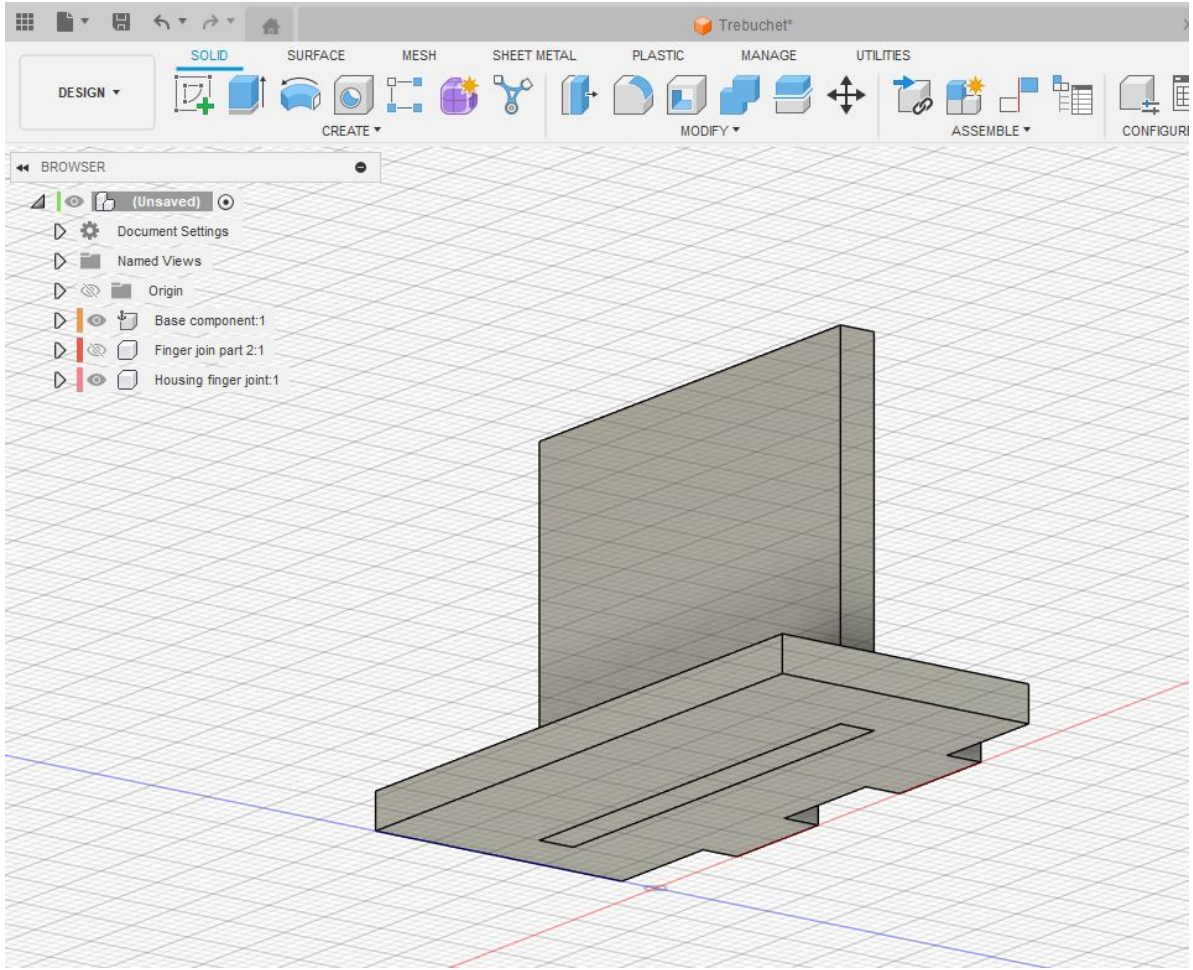


Step 10

This creates slots that you can use to piece together a variety of components.

Your task is to now create a box with these methods.

Ensure all your pieces are 3mm thick and are labeled as individual components.
(If you're clever you may be able to copy paste components to reduce work time)



Step 11

With a little patience and using a larger variety of the sketch tools you can create snap fitting components such as this that slot through cut holes in your work.

Try it when you have finished your box