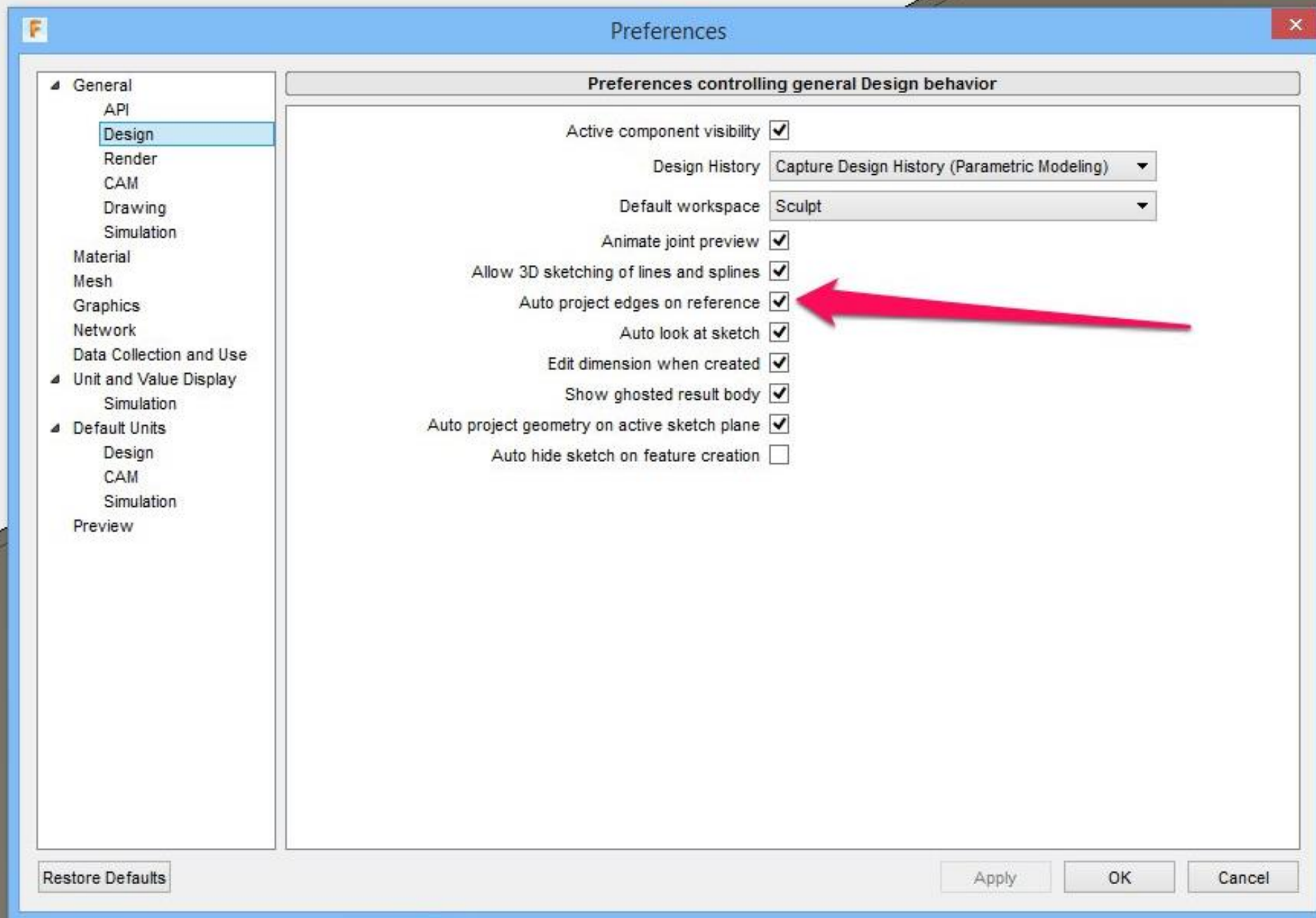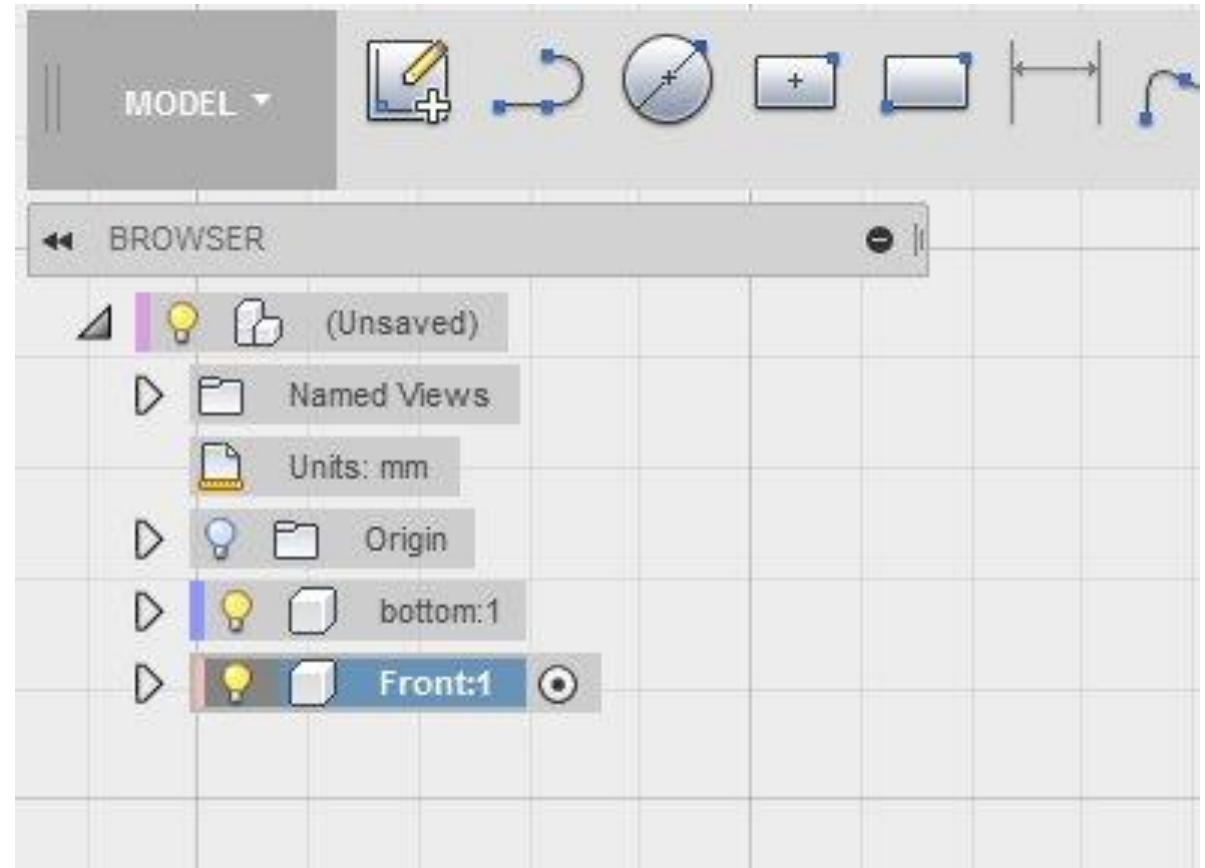# EASY BOX JOINTS

A NEW SKETCH, A NEW HOPE

- You may have noticed that we did not make any box joints in our sketch for the bottom of our box?

- We could have, that is certainly a legitimate way to work.

- There is a much simpler way, and I am lazy and I bet you are too

Check this now and avoid a lot of frustration!!!

- Go right now to preferences and make sure that "Auto project edges on reference" is checked
- Several people have had trouble with the next several steps and it looks like it always comes back to this
- For an expanded discussion see this video:
    - https://www.youtube.com/watch?v=PTn5BYcWFwY
- Back to the show….

Preferences

Preferences controlling general Design behavior

- General
  - API
  - **Design**
  - Render
  - CAM
  - Drawing
  - Simulation
- Material
- Mesh
- Graphics
- Network
- Data Collection and Use
- Unit and Value Display
  - Simulation
- Default Units
  - Design
  - CAM
  - Simulation
- Preview

Active component visibility ☑

Design History | Capture Design History (Parametric Modeling) ▾

Default workspace | Sculpt ▾

Animate joint preview ☑

Allow 3D sketching of lines and splines ☑

Auto project edges on reference ☑

Auto look at sketch ☑

Edit dimension when created ☑

Show ghosted result body ☑

Auto project geometry on active sketch plane ☑

Auto hide sketch on feature creation ☐

Restore Defaults | Apply | OK | Cancel

After you have finished your bottom, create another component, let's call it front

- Click the front face and then click the line icon or simply type L

- You will now be back in sketch mode looking at the front face of your bottom component

- Click the upper left corner and drag down to the lower left corner and release your mouse button

- You will now have a line that looks like this
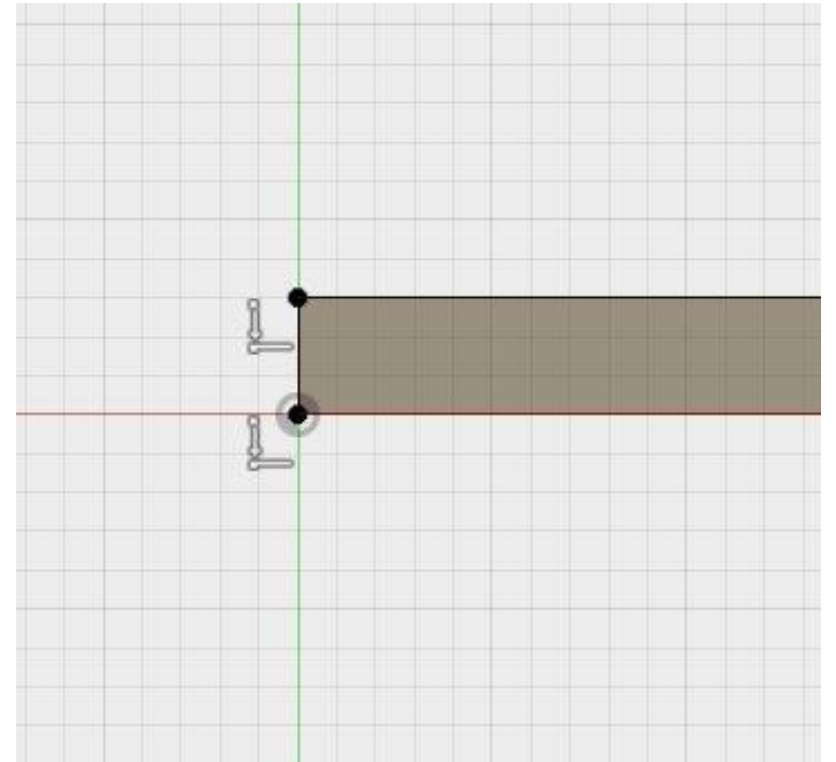
- Click "Rectangular pattern" from the pull down sketch menu

- As always, you can add this to the ribbon if you like

- You will get a dialog box asking you to select Objects

  - Select the line we just drew

- Drag the horizontal arrow across the part to the other end

  - You can also just type in your constant for this, I used "W"

  - See how much easier using parameters makes life?

- Now type in a quantity for how many lines we want

  - This is quite forgiving and will vary by the look you want

  - Always do an even number, you will see why in a moment

  - I did 10 for our example
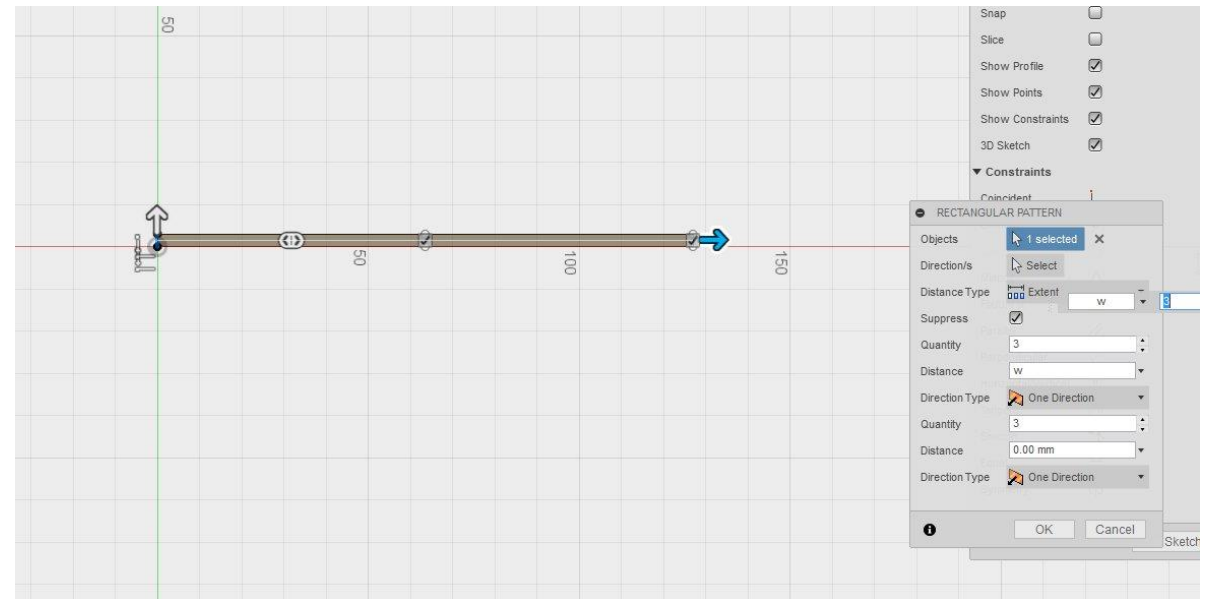
- Click "Stop sketch"

- Go back to an isometric view by either dragging the cube or clicking home

- Now your environment should look something like this
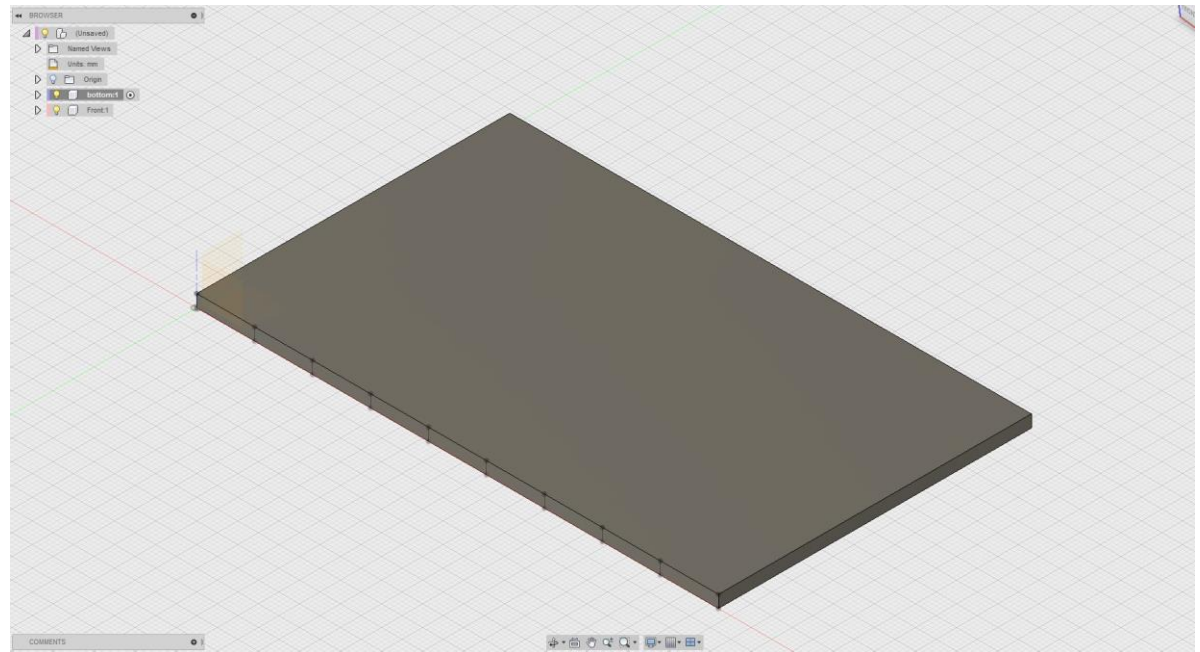
- Click extrude or push/pull
  - They will act the same in this case
  - Push/pull is an adaptive command that means many things at different times
- Click every other section and then start to push in
- Here things get MUCH easier if you are using parameters
  - Type - (material thickness constant), I used "I" so it is (-i)
  - If you want the joints to stand a bit proud, add a little so (-i-1) gives an extra mm for sanding
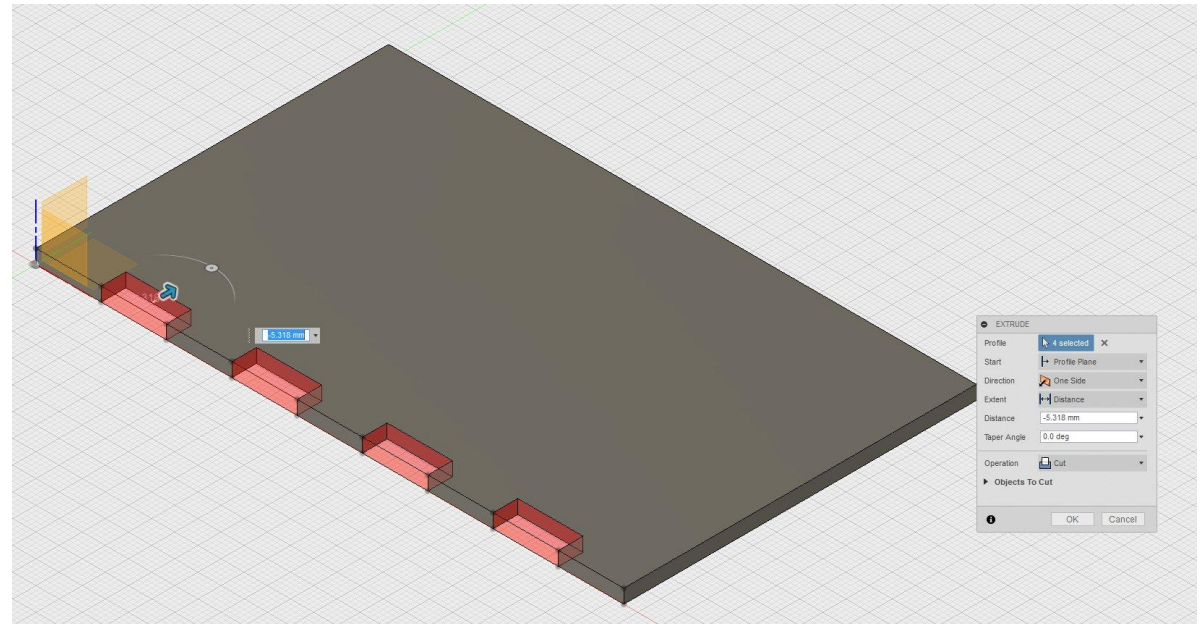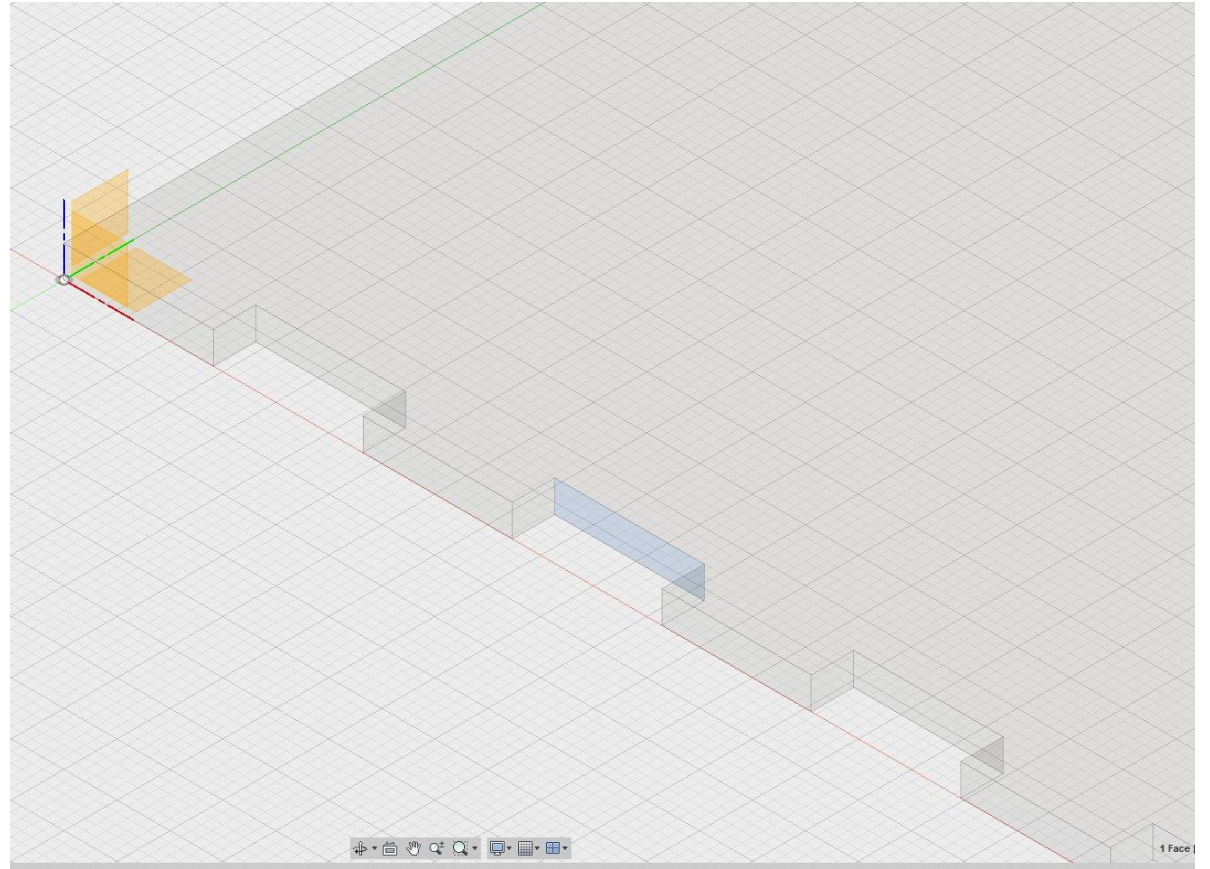
- Click OK and this is what you should have

- Now wasn't that easier than trying to put all that in the first sketch?
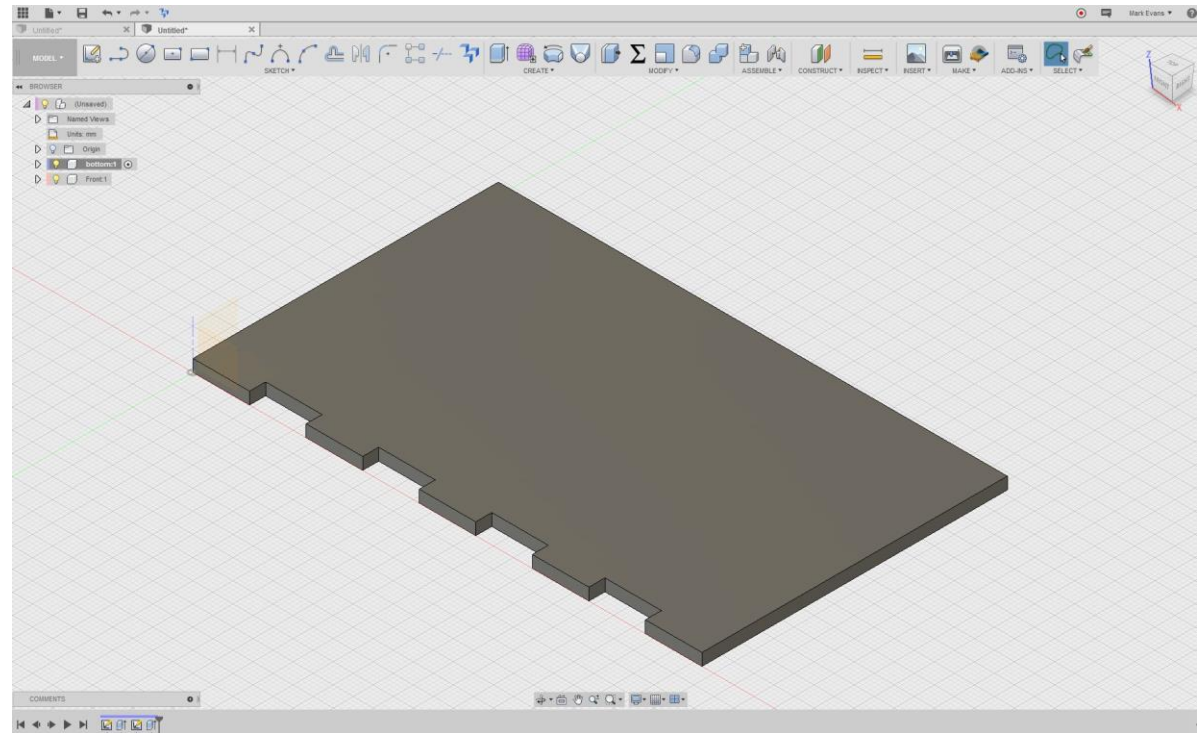
- Now do the same for one of the ends

- Create another component and name it end
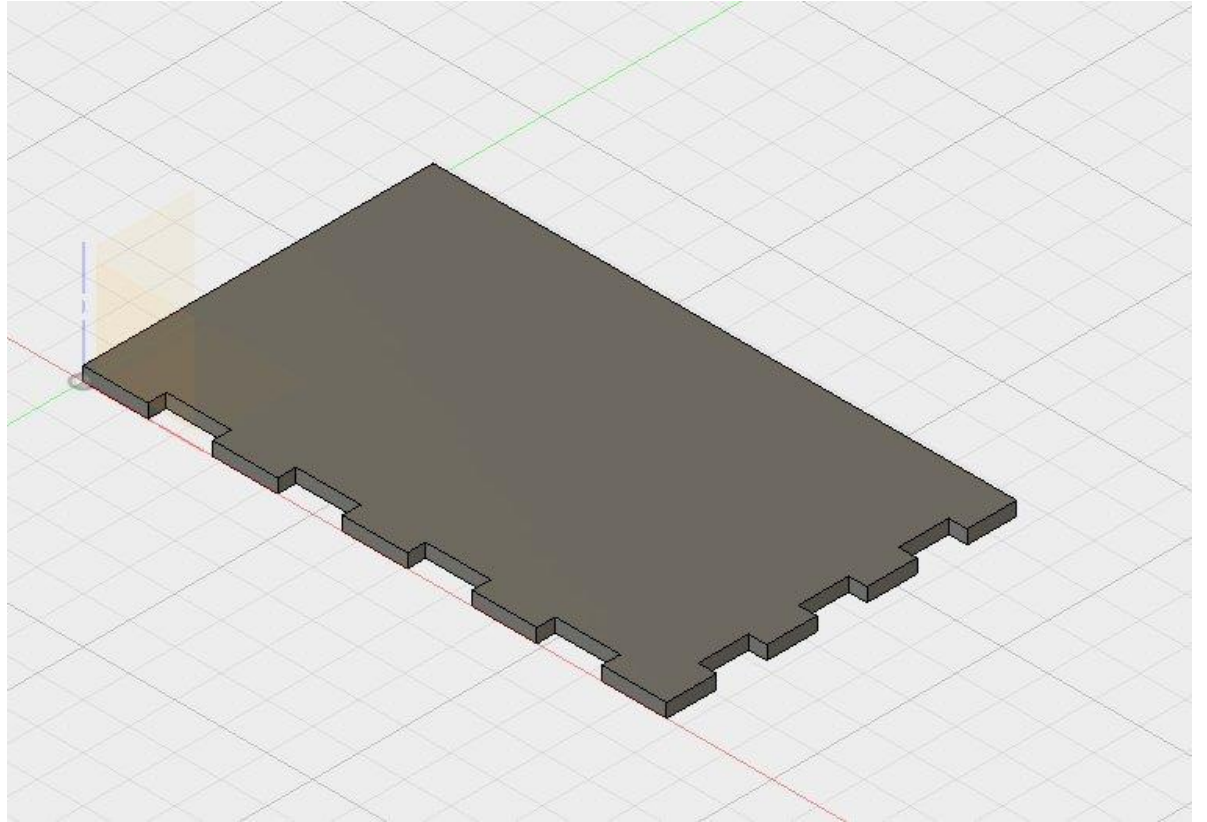
Activate front component and then select the bottom of any of the finger joints on the bottom

- Click two point rectangle and draw from lower left to the right at the height you want

- Again, much easier to type in parameters

- Click stop sketch
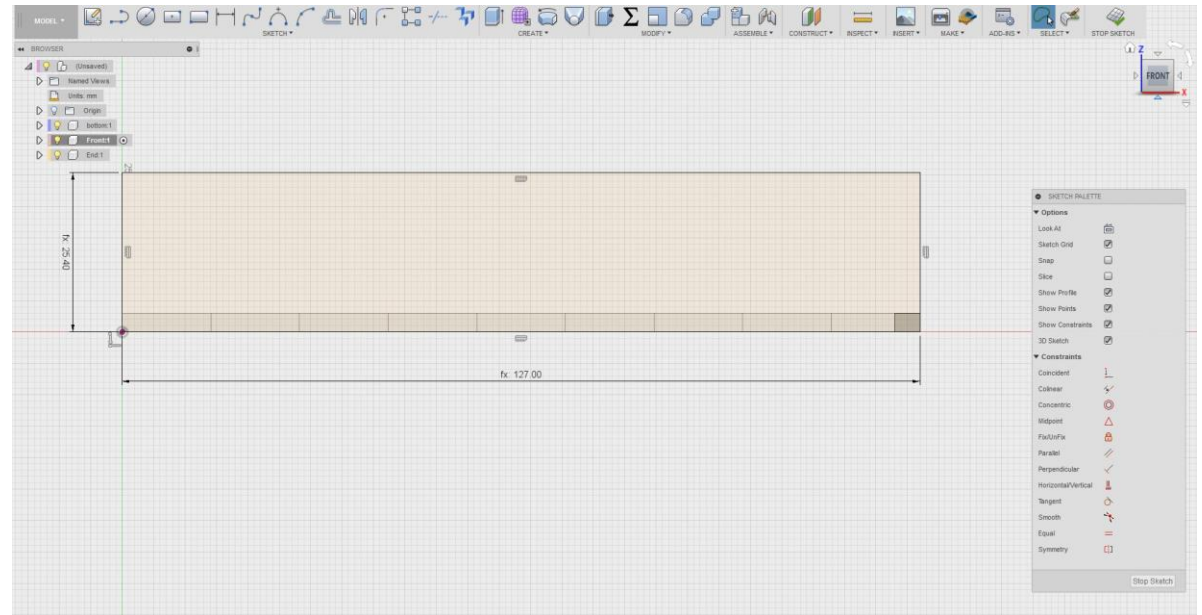
- Extrude the front as we have done before
- We now have two different components occupying some of the same space, we will fix that now
- Click "combine" under the modify ribbon
  - You will get another dialog box
  - Combine is very powerful, it is really four tools in one
  - For this operation we want to select the front as the target body, the bottom as the tool body, cut as the operation, and make sure "keep tools" is checked
  - Click OK and you have just cut box joints in the front
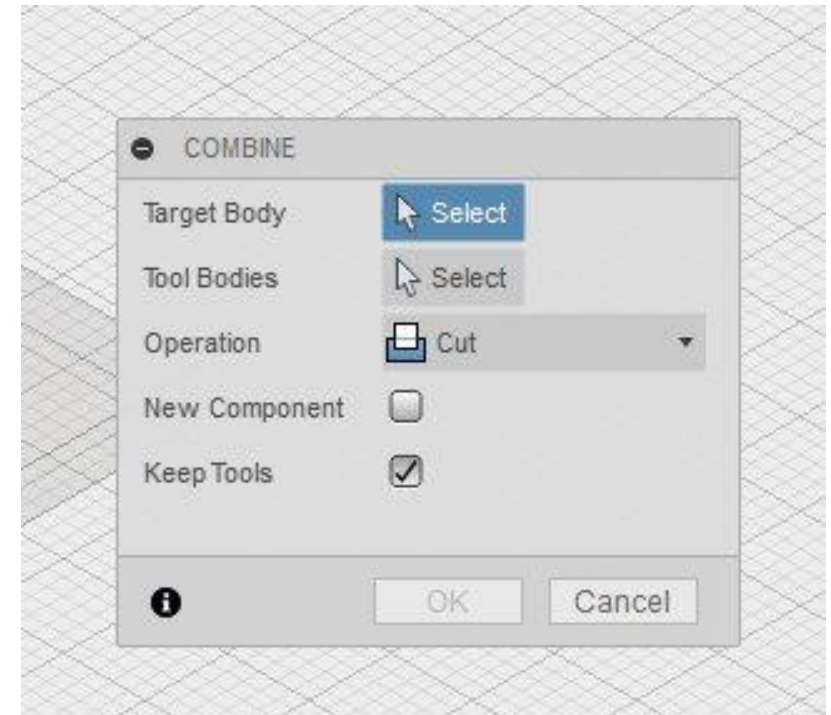
At this point you have all the tools you need to finish the box proceeding in similar manor as before but we will look at some different ways to proceed in the next tutorial
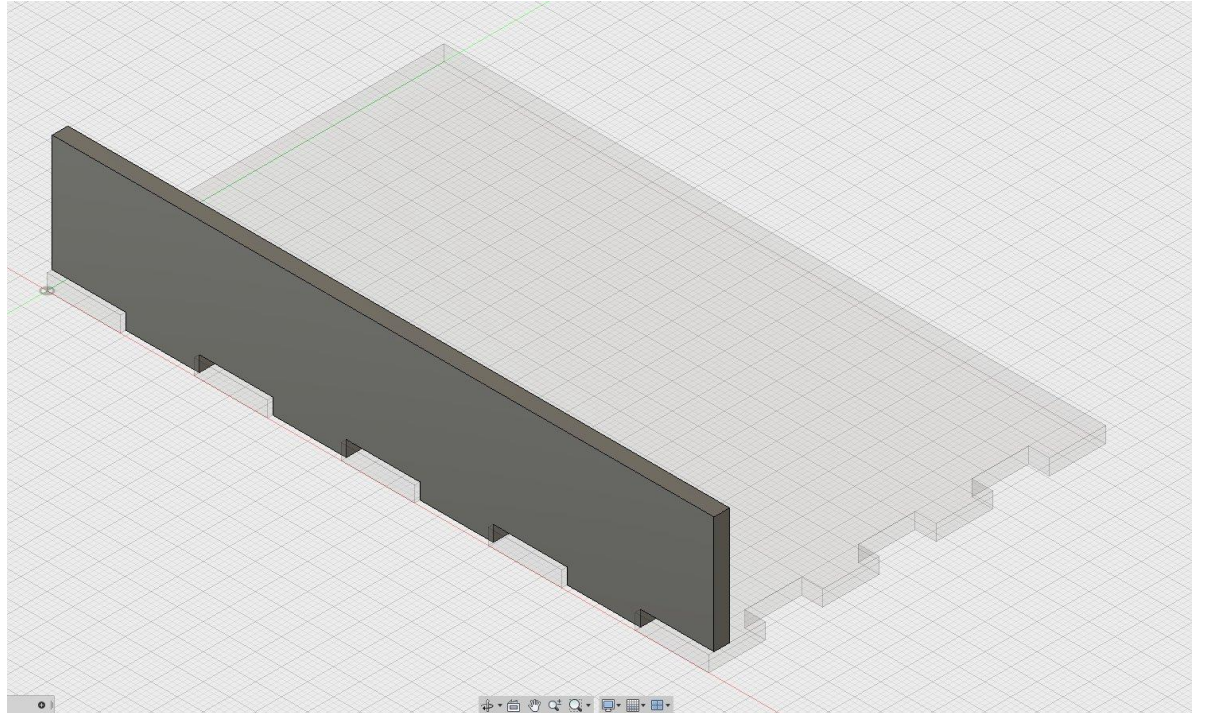
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