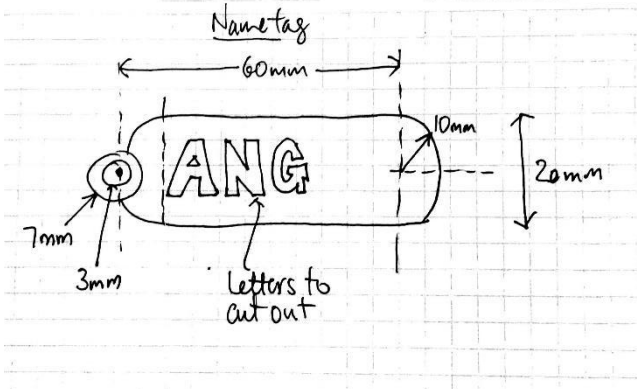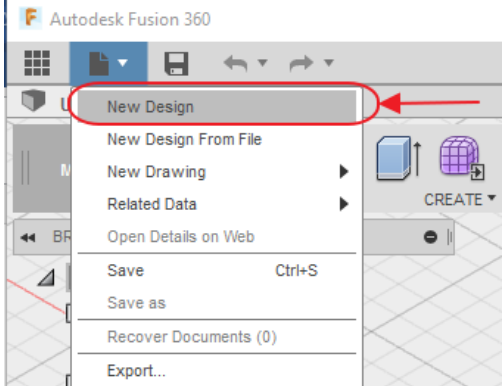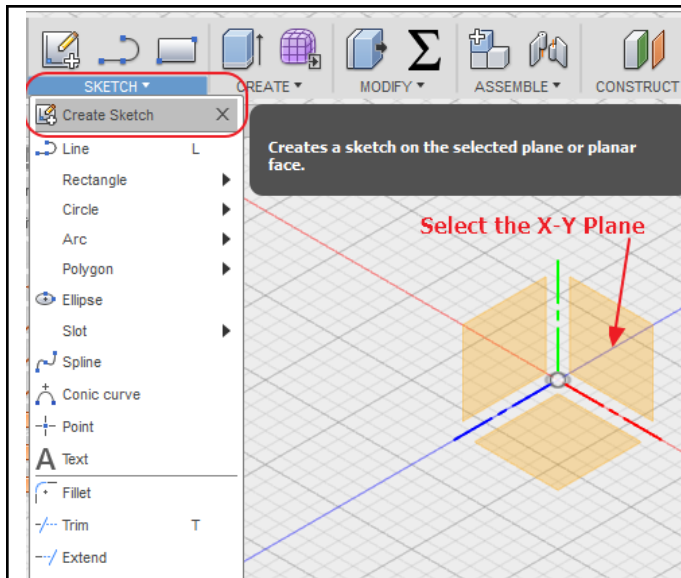# Laser Cutting - Keychain

- Light Amplification by Stimulated Emission of Radiation
- Can be used for cutting and engraving
- Focussed light produces very fine cuts
- A 2-D Vector drawing is used as the cutting template and sent to the Laser cutter
- Cutting is based on
  - Power of light
  - Number of Pulses Per Minute
  - How slow/fast the laser travels
- Carbon Dioxide Lasers (CO_2) can cut paper, cardboard, wood, acrylic
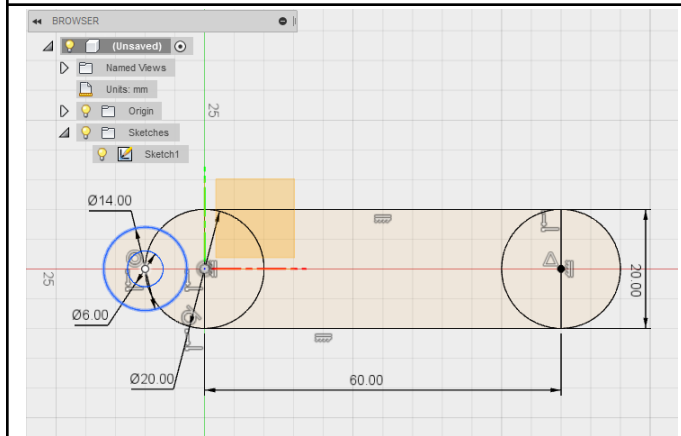
Let's make a personalised keychain

Process:

1. Draw the design on paper
2. Create the design using Fusion 360
3. Convert the design to a 2D drawing and export it (DXF file)
4. Read the DXF file into the Laser Cutter System
5. Cut

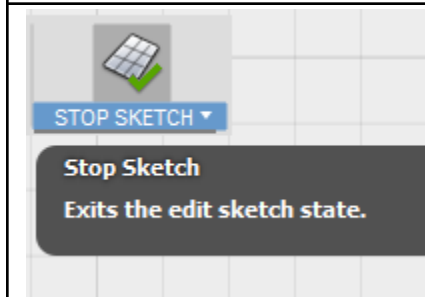|	
	Make a drawing of your keychain Approximate size: 20mm x 70 mm x 3mm (thickness) The small ring attachment should have a diameter of about 6mm and 14mm
	Start Autodesk Fusion 360 Sign in to your account Create a new design

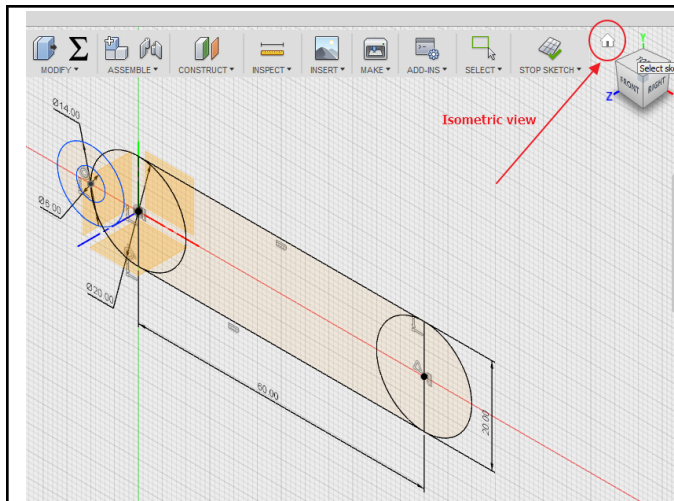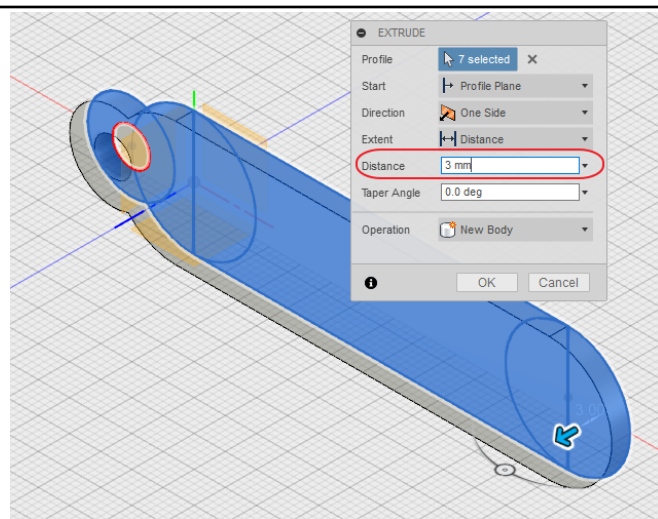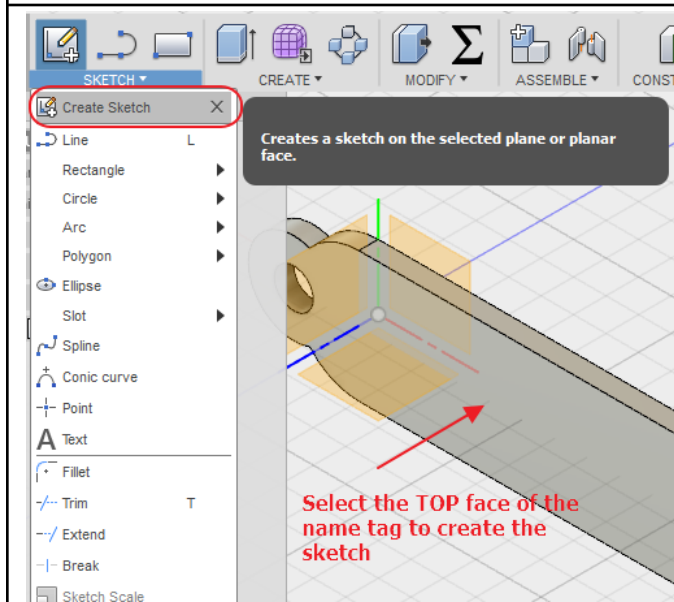|	
	Create a new sketch, select the X-Y plain for the drawing
	Using the drawing tools: ● Sketch the name tag, use circle, rectangles and lines to draw it. ● Dimension the name tag to the correct sizes ● Stop Sketch when finished
	Stop Sketch when finished

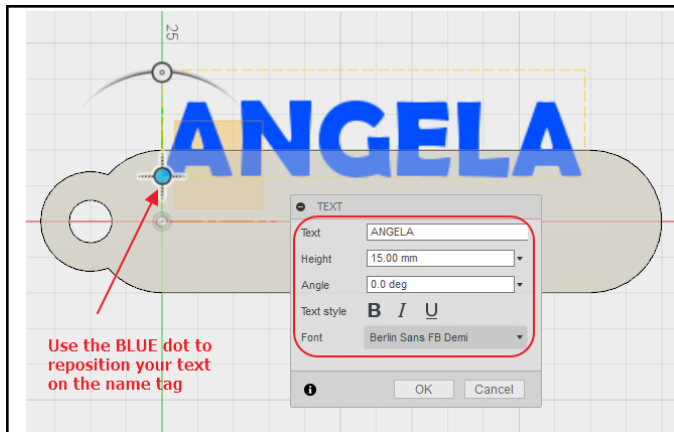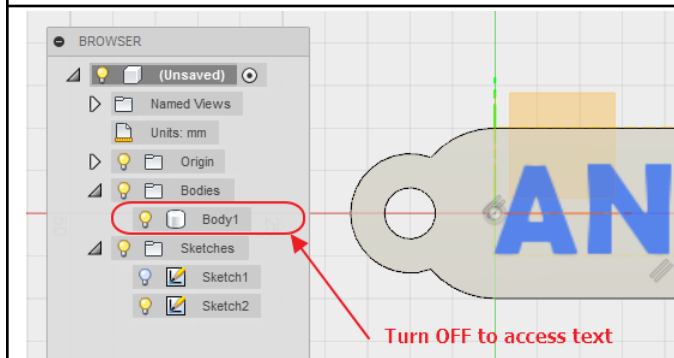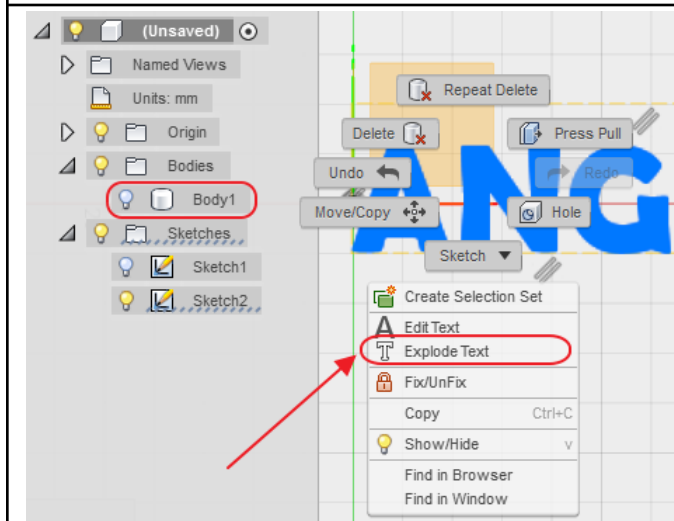|	
	Select the ISOmetric view
	Select EXTRUDE (E) Select ONLY the parts which you would like to extrude to make the name tag (in blue) Key in the extrusion distance (i.e. the thickness of the material) – 3mm Click OK
	Create a new sketch Select the top face of the name tag. You will draw your name on this face

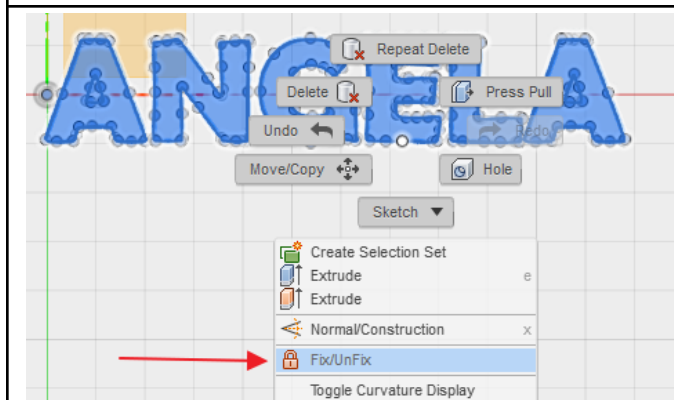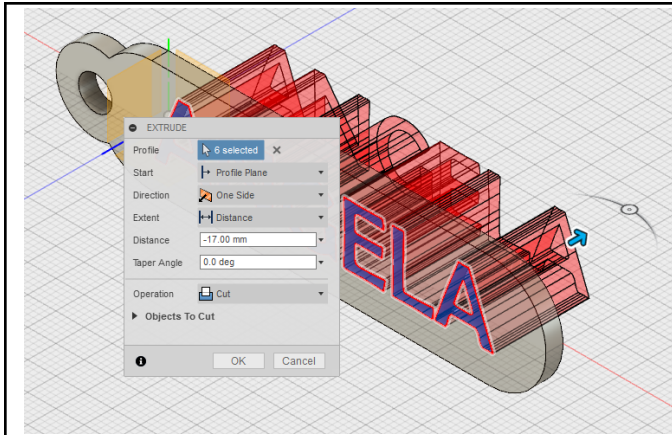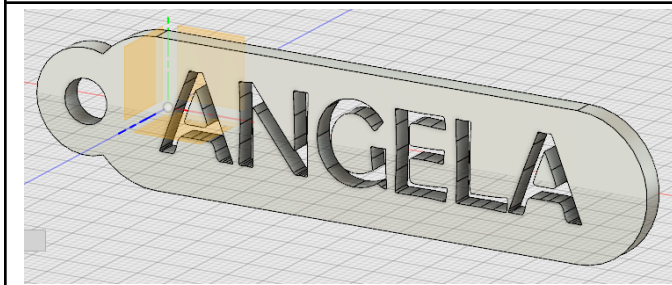|	
Use the BLUE dot to reposition your text on the name tag	Select TEXT Position a point on the name tag to place your text Key in your Name Change the size to fit (e.g. 15mm) Change the font to suit your style Use the BLUE dot to position your text exactly on the name tag Click OK
Turn OFF to access text	Turn OFF the body so that you can manipulate the text. Click the Lightbulb on the side, such that it becomes unlit
	Place your cursor over the BLUE part of text Right-Click Select EXPLODE TEXT This converts your text into line drawings You should see individual letters appear as outlines
	Use your mouse to draw a selection window around all the text. (L-click & hold down, move cursor to select all the text, release left mouse button) R-Click and select the Fix/UnFix This action allows you to move the letters individually should you need to edit them. You should see the letter outline turn BLUE

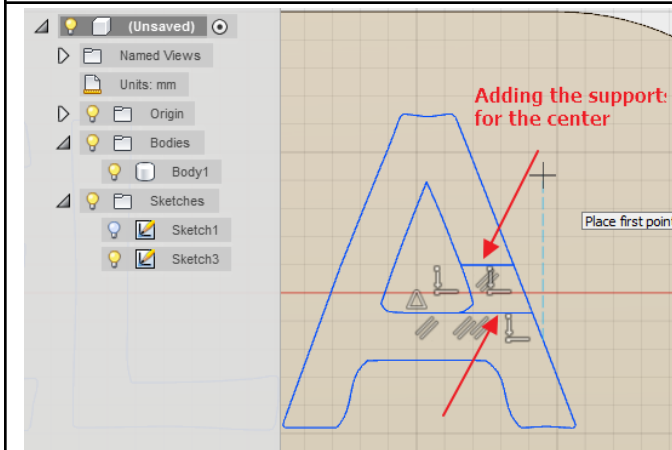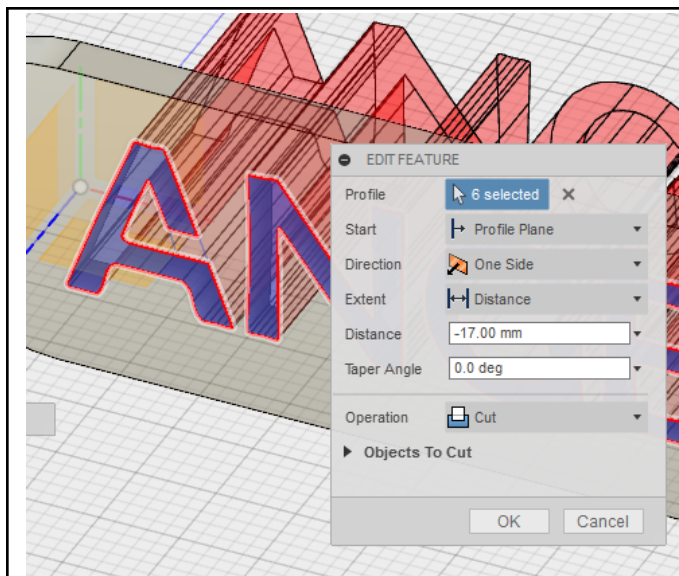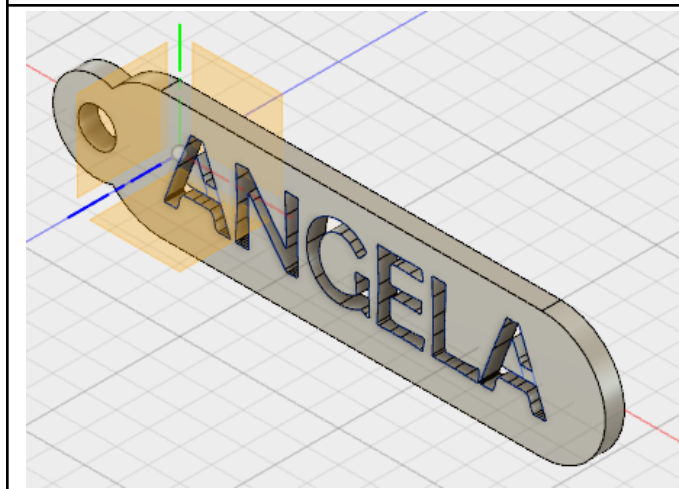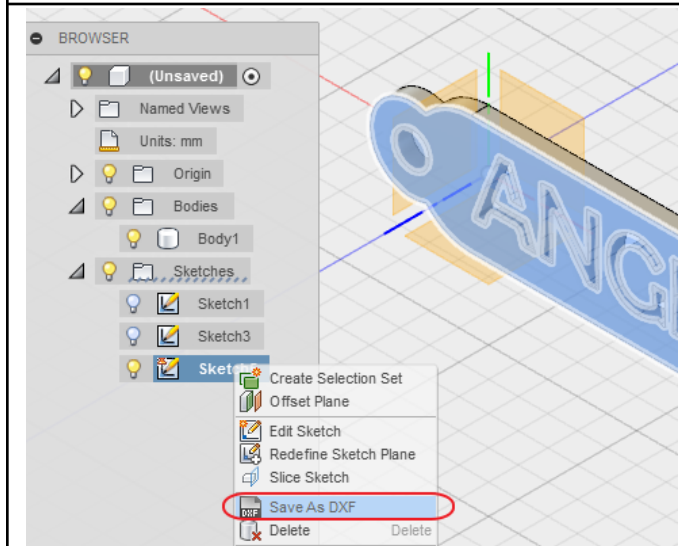|	
	Stop Sketch Turn on the BODY visibility Select ISOmetric view Select EXTRUDE (E) Select all the letters that you have typed in for your name and CUT them from the name tag by entering an appropriate cutting depth or by pulling the BLUE arrow
	You should see the appearance of your name tag wit the cut out letters. Can you spot any errors? (Look at the letter "A") How can you fix them?
	Fixing the "A" by 1. Reversing the tasks until the point of the sketch 2. Correcting the sketch to include the support (so that the center of A does not drop out) 3. Forwarding the task 4. Editing the extrude feature

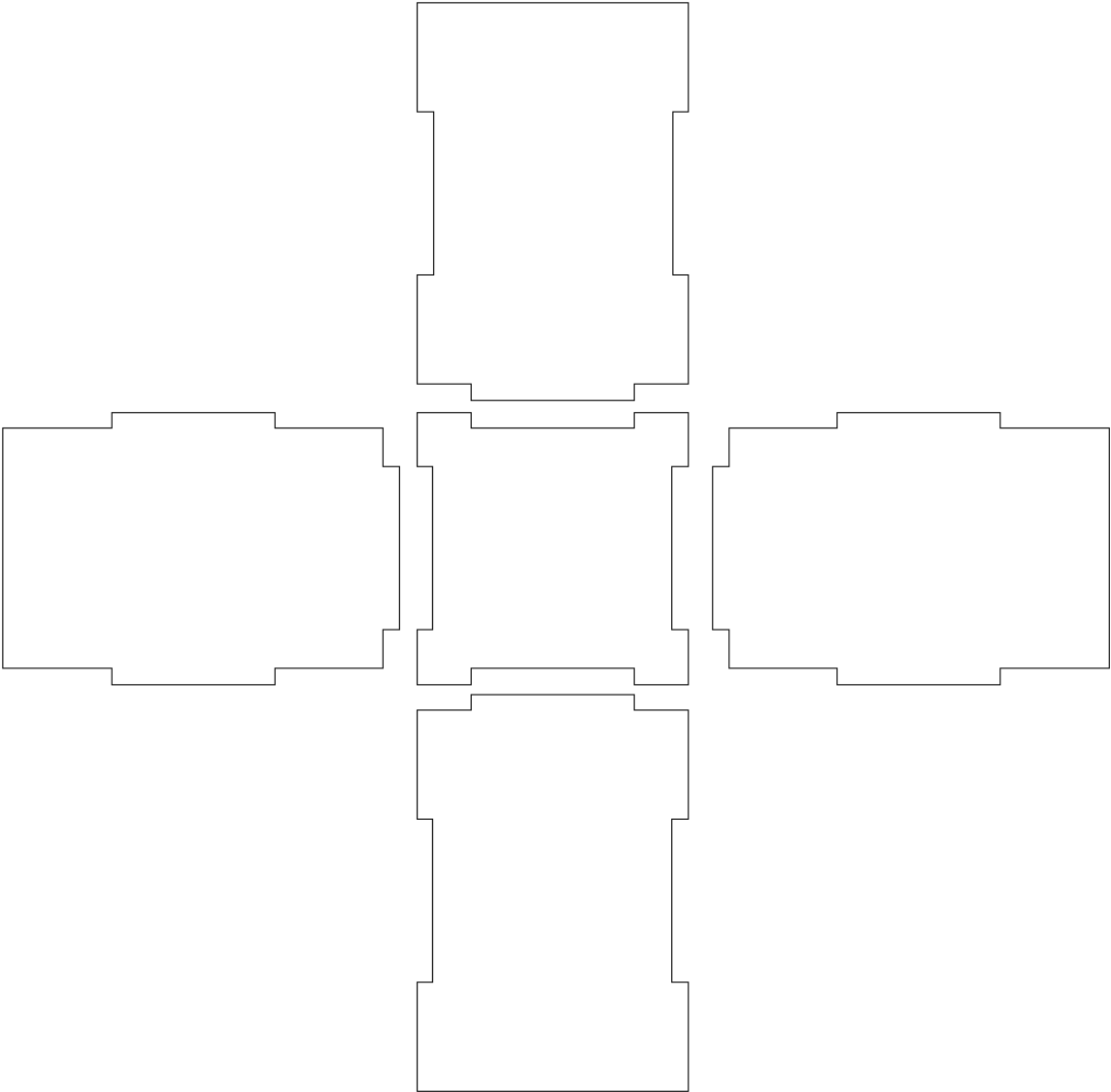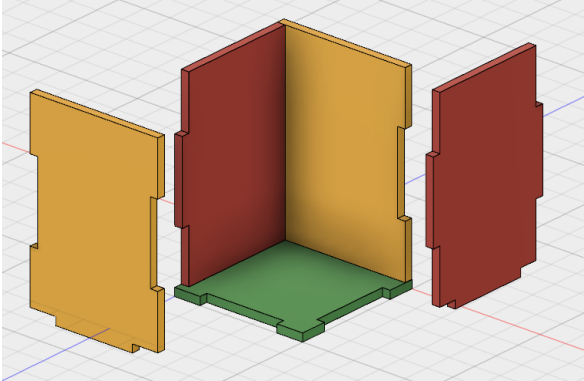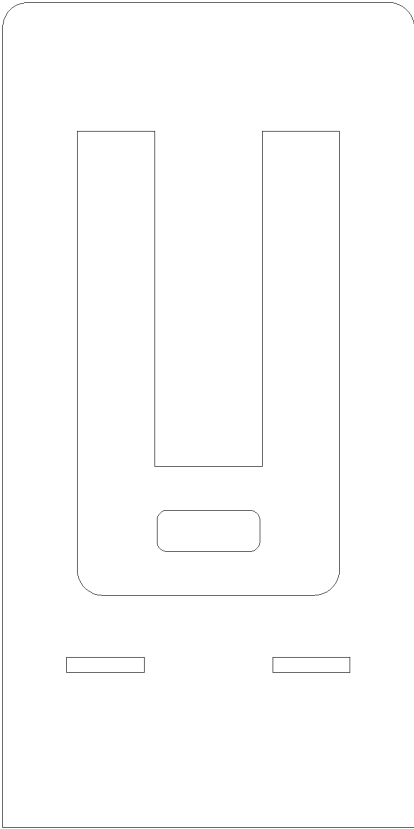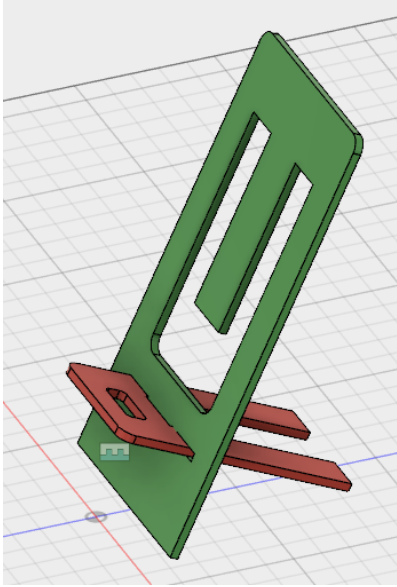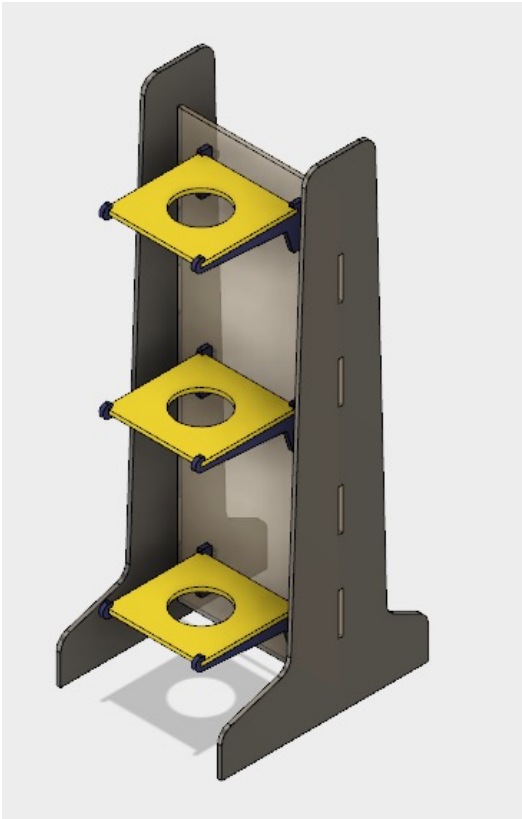|	
	The New extrude
	Completed name tag in ISO view We now need to EXPORT the top face to a DXF file for Laser cutting
	Follow the following steps to create a sketch for export Create sketch Select the TOP plane for the sketch face Stop sketch You will see a now Sketch being created and it you be highlighted in blue You can then R-click on this sketch and export it as a DXF file. Use your name e.g. Angela.dxf for the filename. Your lecturer will instruct you on how to send the file for laser cutting.
	Save your fusion 360 file for further editing or creation of new name tags.

Other Examples to try

Box

Simple Phone Stand

Vertical Drip Irrigation System

Raft Hydroponics System