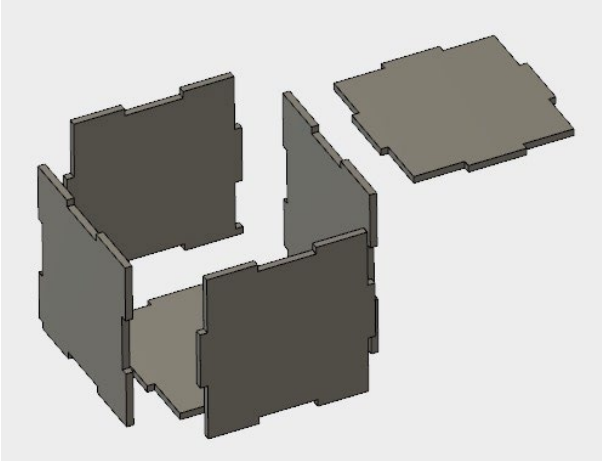
Laser Cutting - Box

Fusion 360 commands:
1. Learn to set up parameters (Modify > Change Parameters)
2. Use dimensions to constrain drawing using parameters (Sketch > Dimension)
3. Defining & sketching on surfaces
4. Project existing faces & bodies as guides for new sketch (Sketch > Project)
5. Extruding selected faces to create slots (Create > Extrude)
6. Using one body as tool to shape a second body (Modify > Combine)
7. Using construction planes as aid in drawing (Construct > Midplane)
8. Using mirror command to duplicate objects (Create > Mirror)

Goal:

Our goal is to create a 3D cube (size: 60mm x 60mm x 60mm), made of either plywood (thickness 2.5mm) or acrylic (thickness 3.0mm), with fingers & slots between the faces, to ensure proper alignment. The tab width (finger) is 20mm, or 1/3 of the length of each side.

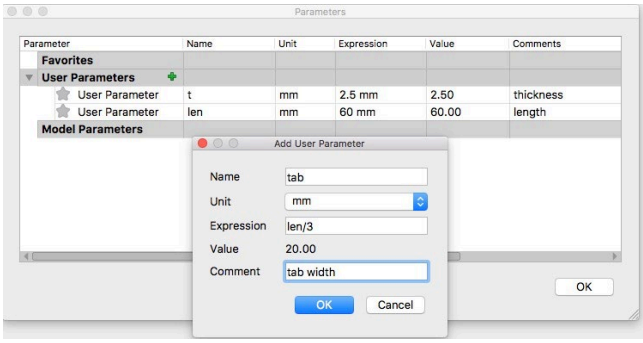
An illustration of the 3D cube:



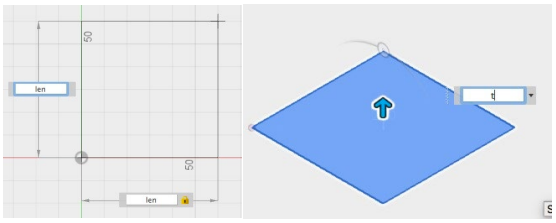
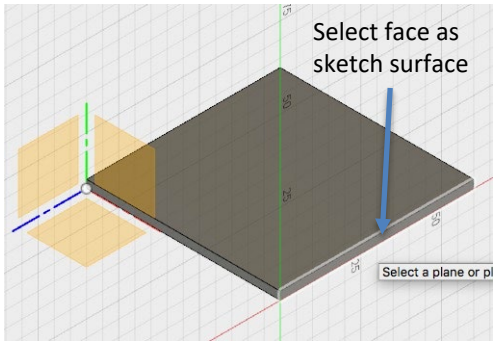
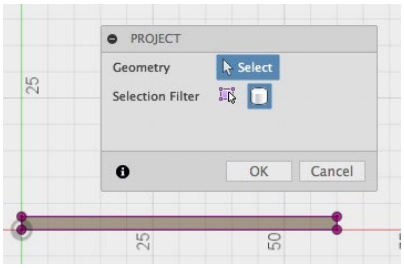
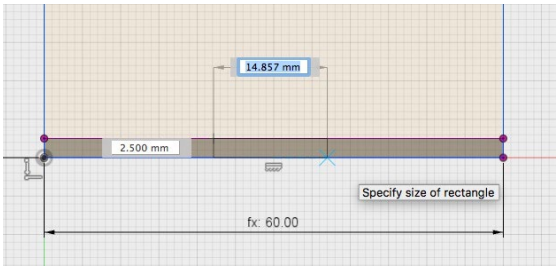
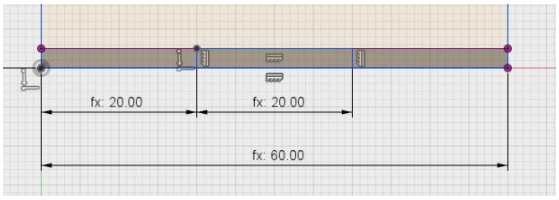
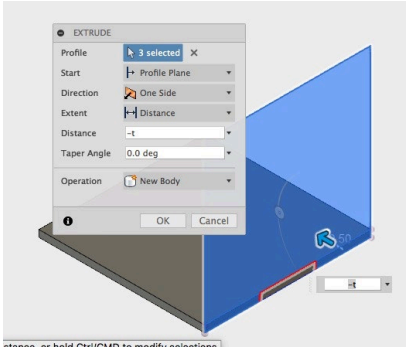
Note:
Length of each side: 60 mm
Length of each tab: 20 mm
Thickness of material: 2.5 mm

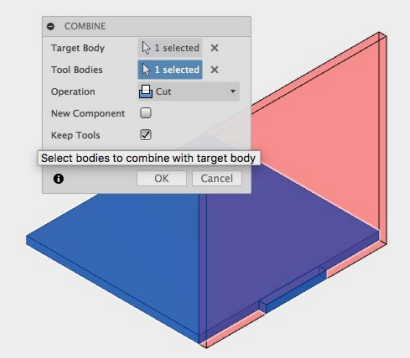
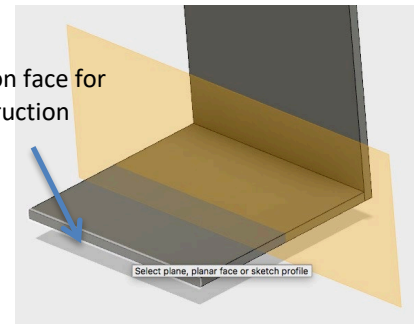
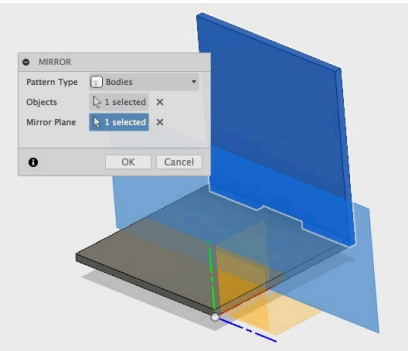
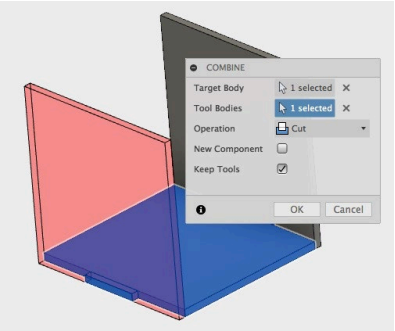
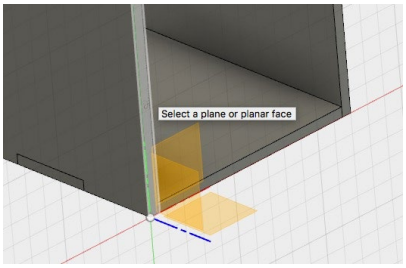
Parameters:
len - 60 (length)
t - 2.5 (thickness)
tab - len/3 (tab width)

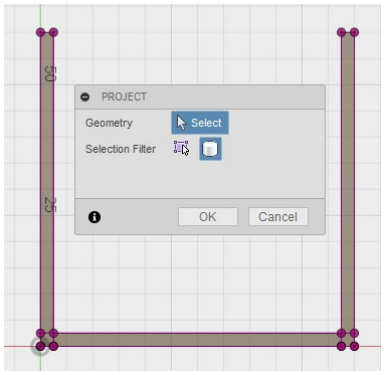
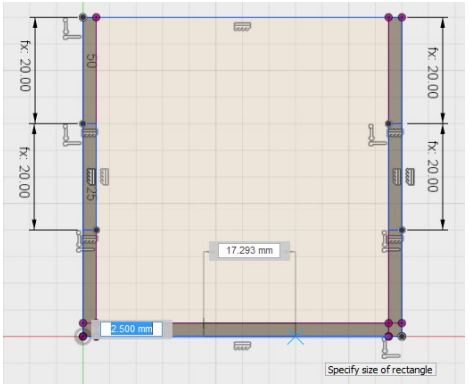
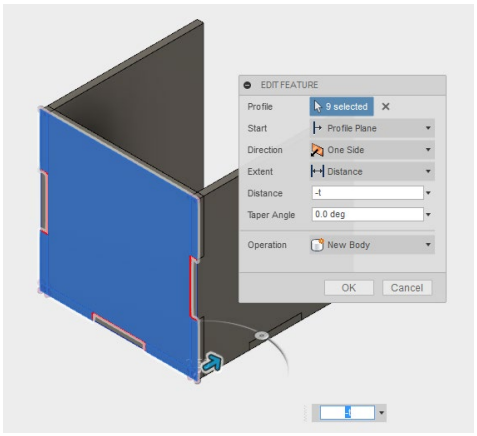
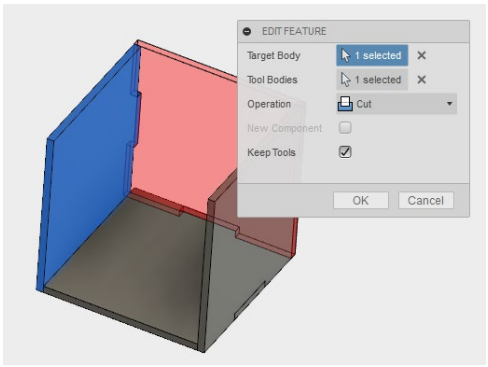
Process:

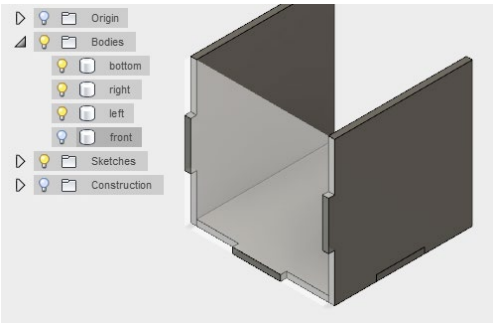


1. Start a new Fusion 360 design (File > New Design)
2. Create parameters for your 3D cube:
   a. t – 2.5mm (thickness)
   b. len – 60mm (length)
   c. tab – len/3 (tab width)
3. Click OK when done.

	4. Create a new sketch on the X-Z plane. 5. Draw a rectangle from the origin, with dimensions len x len. 6. Stop Sketch and extrude the rectangle to height t.
	7. Create a new sketch. Select the right face of the bottom plate as the sketch surface.
	8. Project the existing body (bottom) onto the sketch surface using Sketch > Project/Include > Project. 9. Select Body for Selection Filter and bottom body for Geometry and click OK.
	10. Draw a rectangle starting at the bottom left (origin), with dimensions len x len. 11. Draw a 2nd rectangle, using the bottom profile as a guide. Note the "X", indicating that the rectangle lies on the profile line. Width of this rectangle is tab (20mm).
	12. Use the dimension command (Modify > Dimension) and set the rectangle to be tab (20mm) away from the left edge. 13. Click Stop Sketch to end the sketch.
	14. We want to create a new body based on the sketch profile that we just drew. 15. Select Create > Extrude and click on the profiles for the right face. Enter –t for the thickness and New Body for the Operation. Click OK. 16. Congratulations! You have just created the right face.

	17. We now want to use the new face to cut out the slot for the bottom face. Click Modify > Combine. 18. Select the bottom plate as target body and side plate as tool body. Select Cut as the Operation and check Keep Tools. Click OK. 19. Hide and show each body one at a time to see the parts that you have created.
	20. We now want to draw the left face of the cube. Rather than redraw everything, we will mirror the right face about a center plane. 21. Create a construction plane, midway between the left and right face of the bottom plate (Construct > Midplane). 22. Click on the right face, rotate the view, then click on the left face.
	23. With the construction plane created, click Create > Mirror. Select Bodies as the Pattern Type, right face as the object and the construction plane as the mirror plane. Click OK.
	24. Using the left plate as tool body, cut a slot on the bottom plate using Modify > Combine. Remember to check “Keep Tools”. 25. Selectively hide and show each body to see the parts that you have created.
	26. We not want to draw the front plate. Create a new sketch, selecting the front face of one of the bodies as the sketch surface.

	27. Project the profiles of the 3 existing bodies onto the sketch surface by clicking Sketch > Project/Include > Project, then selecting the 3 bodies. Click OK. 28. Draw a rectangle (len x len), from the top left to bottom right corner of the cube profile.
	29. Draw 3 rectangles for the tabs. Make sure that the rectangles are on the profile lines ("X" appears when you are on the line). 30. Add dimension constraints, setting spacing & tabs to 20mm (tab). Click Stop Sketch.
	31. You are now ready to extrude the front face of the cube. Click Create > Extrude, then select the profiles to extrude. Distance is –t and Operation is New Body. Click OK.
	32. We will now use the front plate to cut out the tabs for the first 3 sides. Click Modify > Combine. Select the right plate as target, front plate as tool and Cut as Operation. Tick the Keep Tools checkbox and click OK. 33. Repeat for the other 2 sides.

	34. Selectively show/hide each face to make sure that the tabs on each face is correct. 35. We have now created 4 sides of our 3D cube. 36. Complete the cube by creating the remaining 2 sides, using the commands that you have learnt.