SAVEnding – Designing Stage

EP1000 Module Project

Parameters

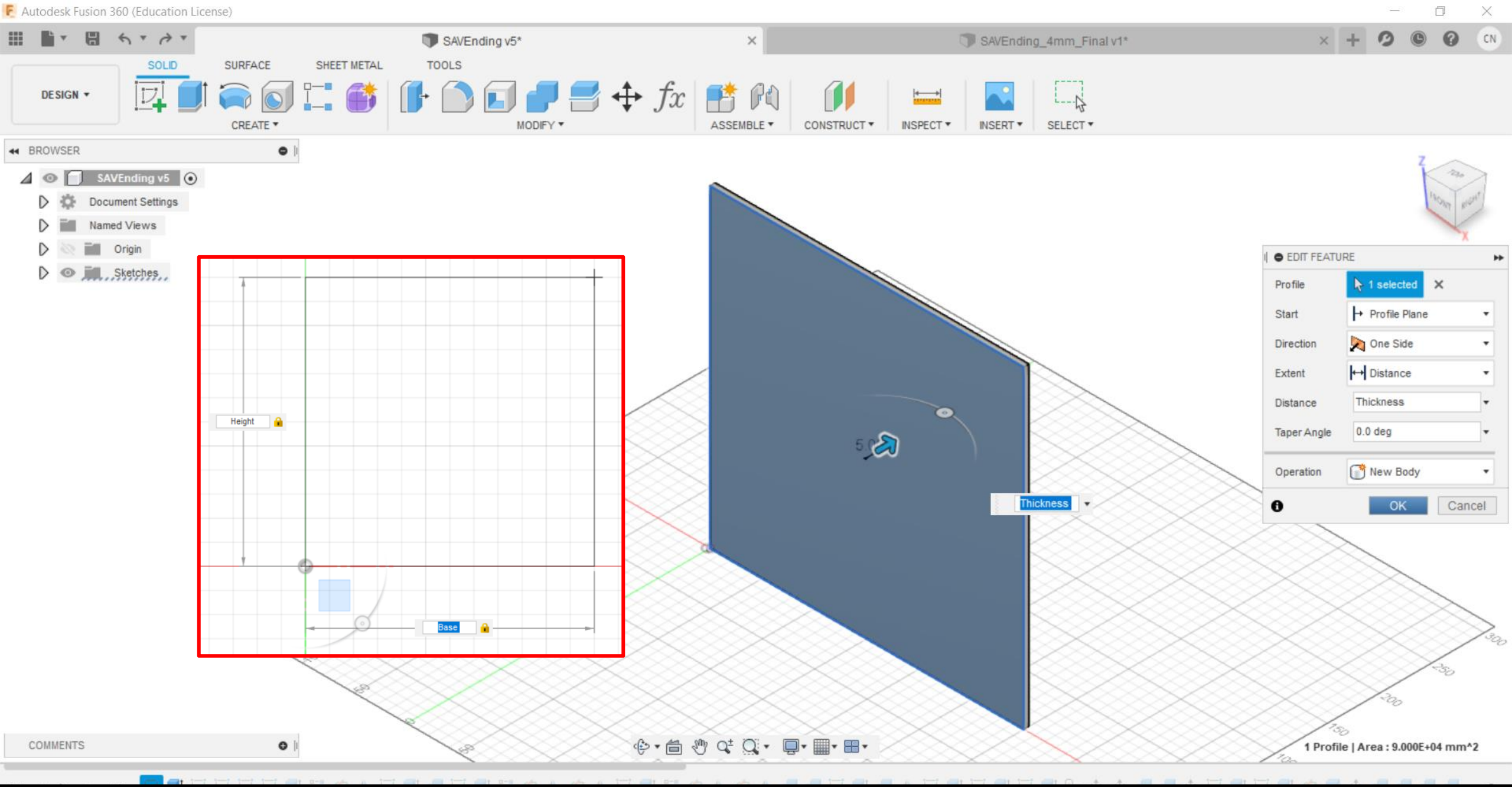
Parameter	Name	Unit	Expression	Value	Comments
User Parameters					
User Parameter	Thickness	mm	5 mm	5.00	
User Parameter	Base	cm	35 cm	35.00	
User Parameter	Height	cm	30 cm	30.00	
User Parameter	Width	cm	30 cm	30.00	
User Parameter	support_width	cm	20 cm	20.00	
User Parameter	divider_height	cm	6 cm	6.00	
User Parameter	divider_width	cm	21 cm	21.00	
User Parameter	opening_base	cm	16 cm	16.00	
User Parameter	opening_height	cm	6 cm	6.00	
User Parameter	box_base	cm	10 cm	10.00	
User Parameter	box_height	cm	6.5 cm	6.50	
User Parameter	box_width	cm	6 cm	6.00	
User Parameter	finger_base	mm	20 mm	20.00	
User Parameter	finger_height	mm	5 mm	5.00	
Model Parameters					
SAVEnding v1					

OK

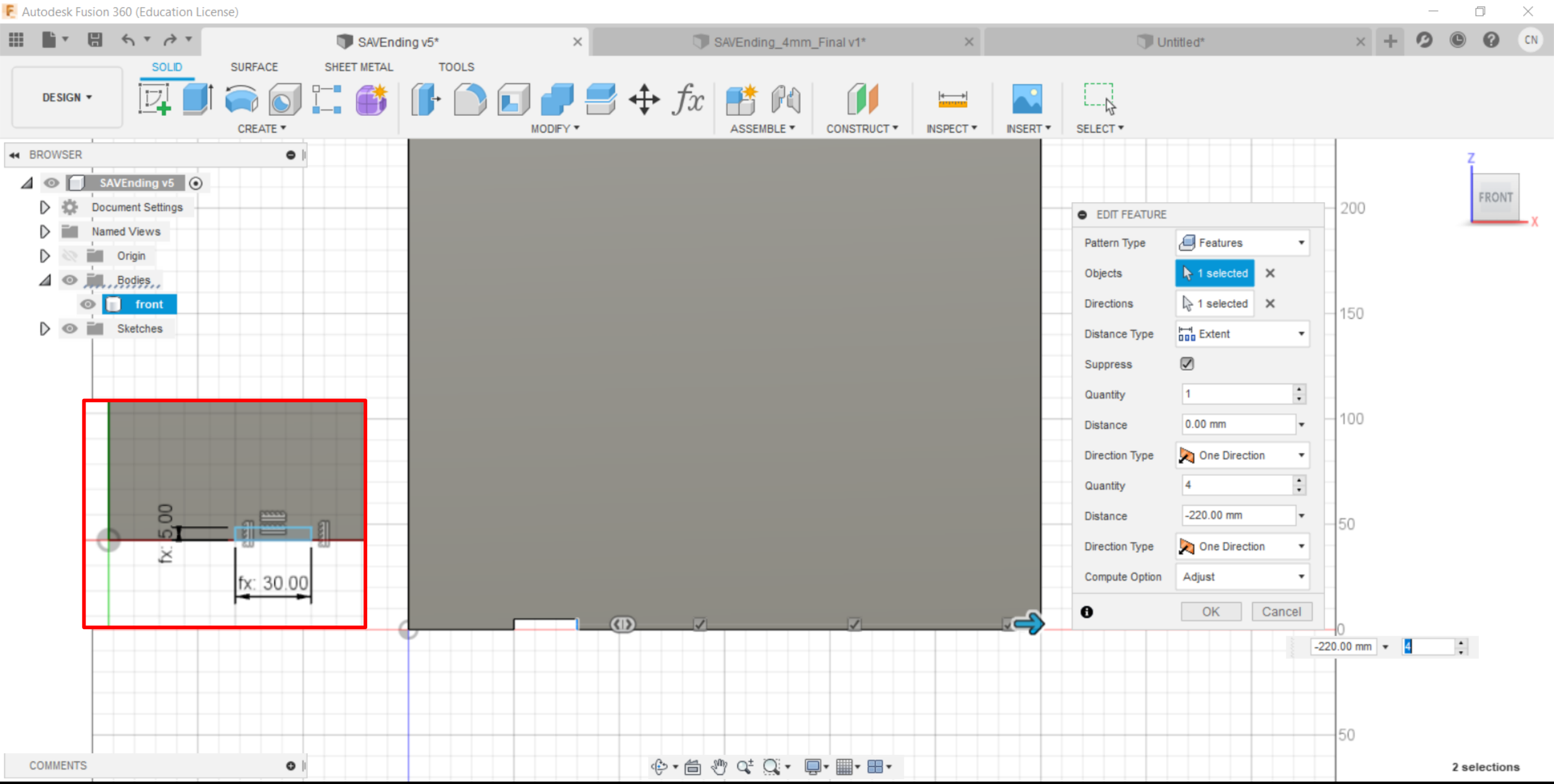
Parameters

Parameter	Name	Unit	Expression	Value	Comments
Favorites					
User Parameter					
User Parameter					
User Parameters					
User Parameter	Thickness	mm	5 mm	5.00	
User Parameter	Base	cm	30 cm	30.00	
User Parameter	Height	cm	30 cm	30.00	
User Parameter	Width	cm	28 cm	28.00	
User Parameter	support_width	cm	20 cm	20.00	
User Parameter	divider_height	cm	6 cm	6.00	
User Parameter	divider_width	cm	20 cm	20.00	
User Parameter	opening_base	cm	16 cm	16.00	
User Parameter	opening_height	cm	6 cm	6.00	
User Parameter	finger_base	mm	30 mm	30.00	
User Parameter	finger_height	mm	5 mm	5.00	
Model Parameters					

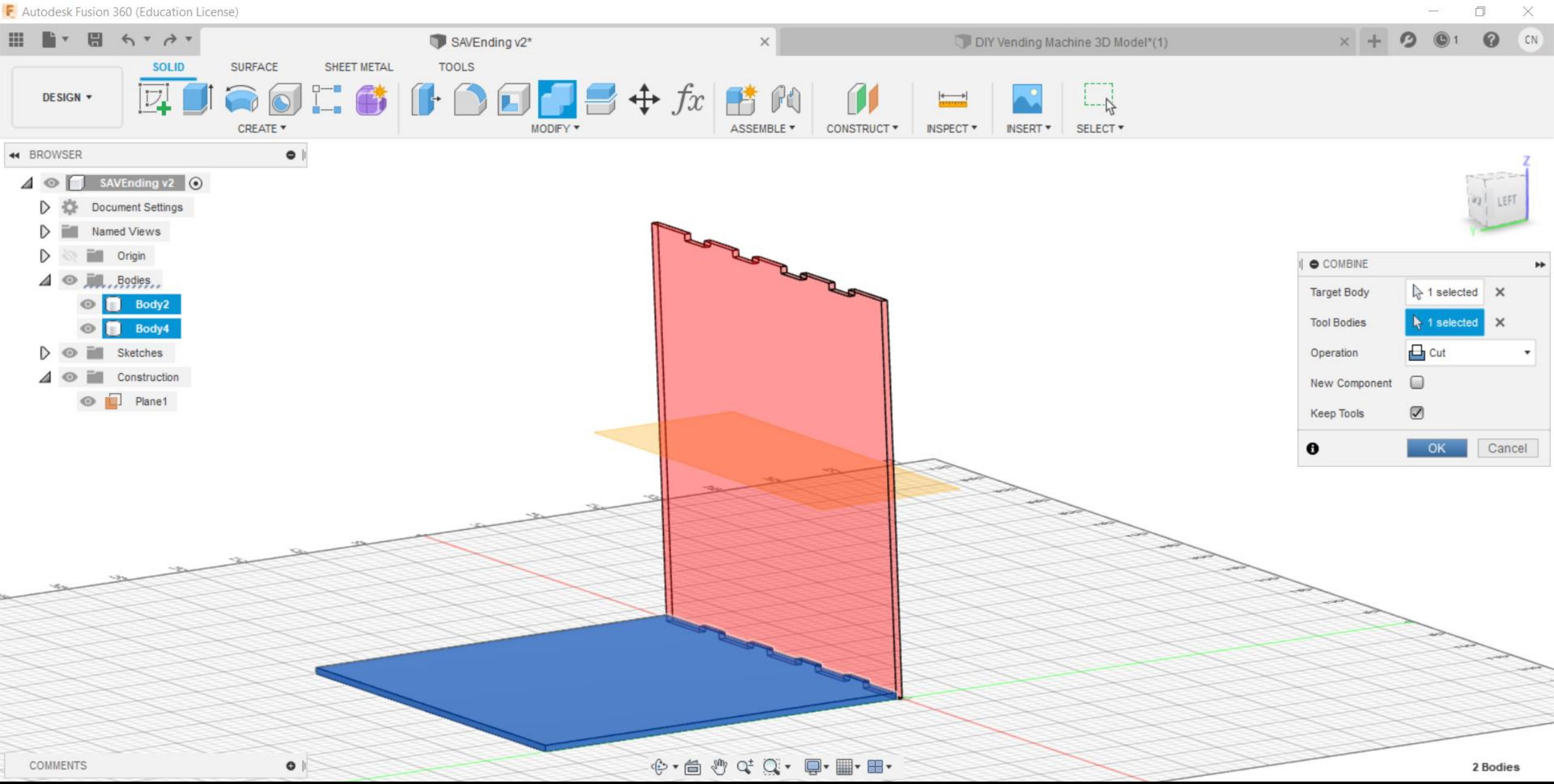
1. I set some parameters at the start, the final parameters are shown at the bottom.



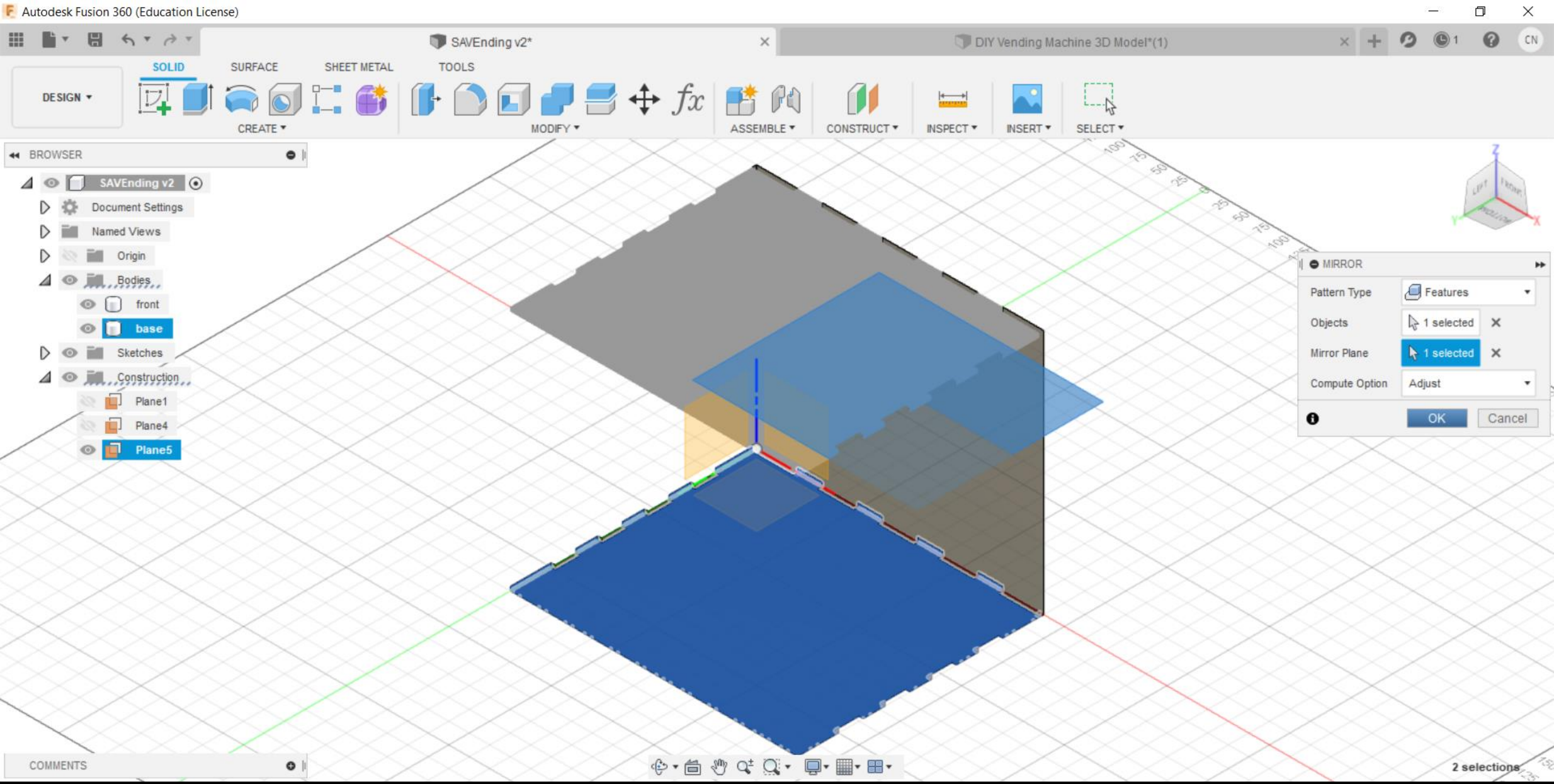
2. Create a sketch on the XZ plane, draw a 2-point rectangle of dimensions *Base x Height*, then extrude to *Thickness.*



3. It’s been long since I used Fusion 360, and I didn’t document that time, so I did not create the second piece to create the fingers then extrude. Here, I extruded a finger dent then used Rectangular pattern for other dents, as well as the other side.



4. I drew another rectangle, this time *Base x Width* on the XY plane. After extruding to *Thickness*, I used the front piece to cut it. Repeated the finger ‘dents’ for this piece again, on the left and right.



5. I constructed a midplane and mirrored the bottom piece. Then I make the finger dents again (silly of me not to recap), thankfully I have used snap to guidelines and mirror planes, hence I can still adjust my parameters easily.

Autodesk Fusion 360 (Education License)

SAVEnding v2*

DIY Vending Machine 3D Model*(1)

SOLID SURFACE SHEET METAL TOOLS

DESIGN

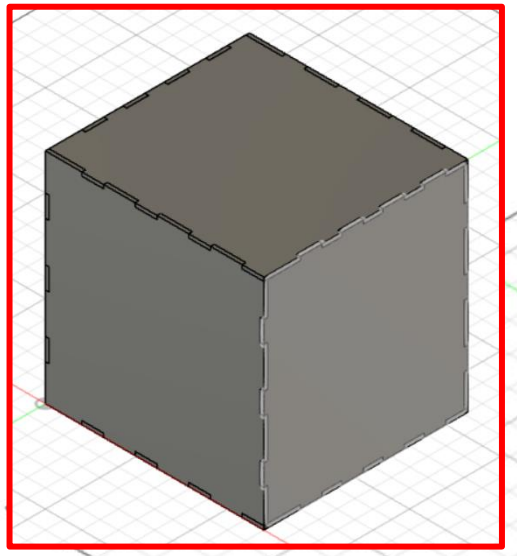
CREATE MODIFY ASSEMBLE CONSTRUCT INSPECT INSERT SELECT

BROWSER

- SAVEnding v2
  - Document Settings
  - Named Views
  - Origin
  - Bodies
    - front
    - base
    - base (1)
    - front (1)
    - Body8
  - Sketches
  - Construction
    - Plane1
    - Plane4
    - Plane5
    - Plane6
    - Plane7

COMMENTS

There we go, still a parametric box



COMBINE

Target Body: 1 selected

Tool Bodies: 4 selected

Operation: Cut

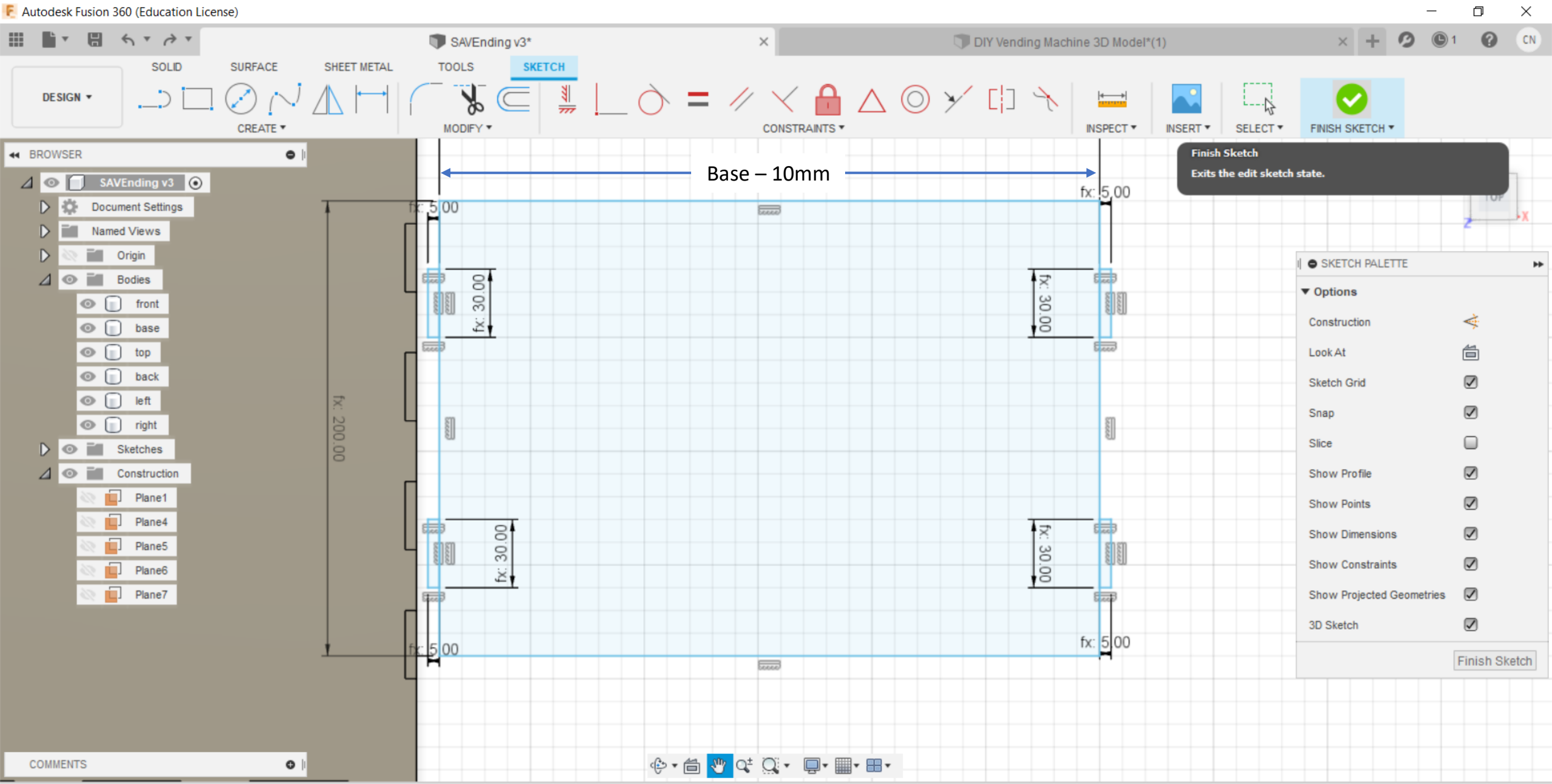
New Component

Keep Tools

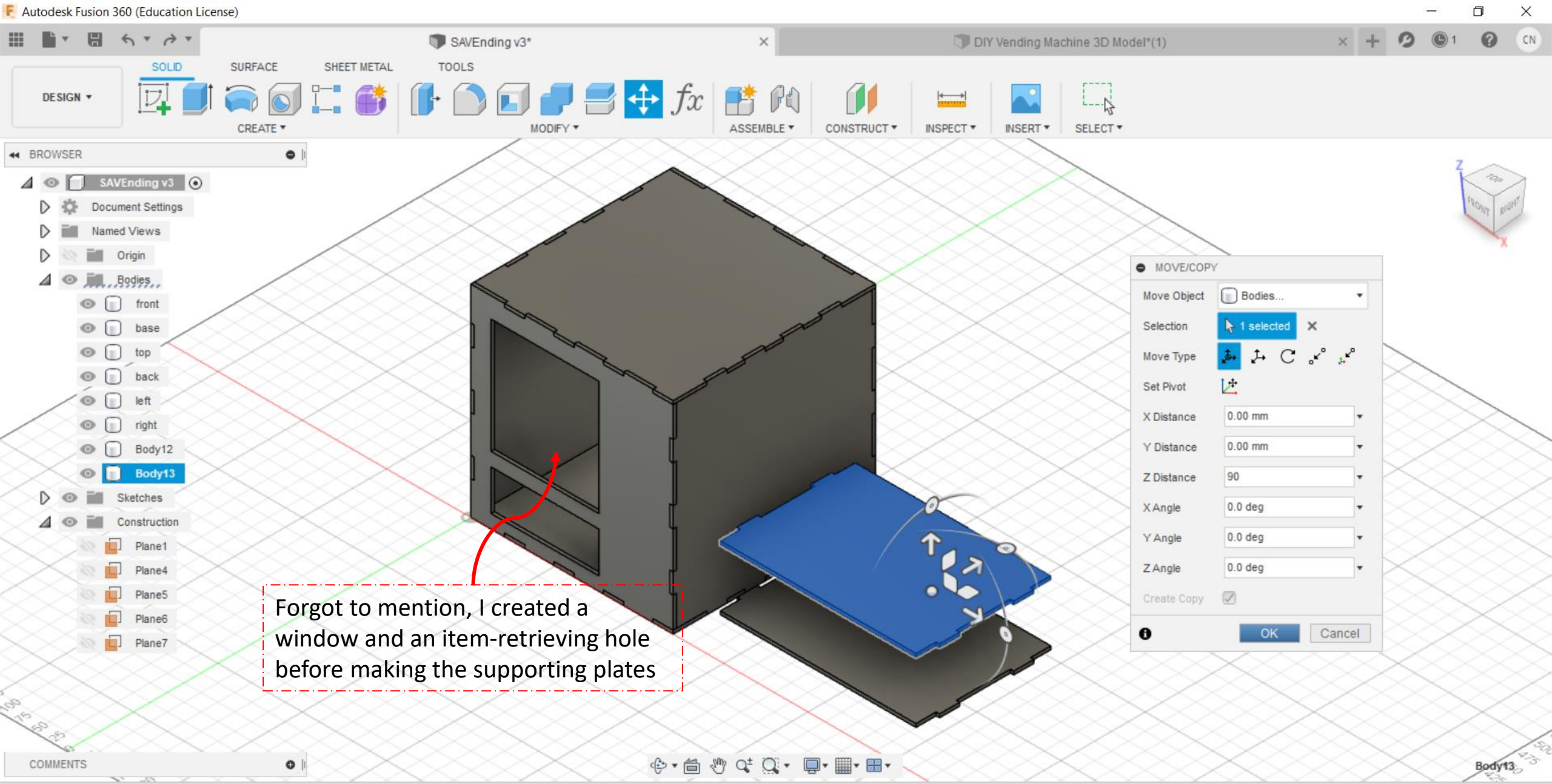
OK Cancel

Multiple selections

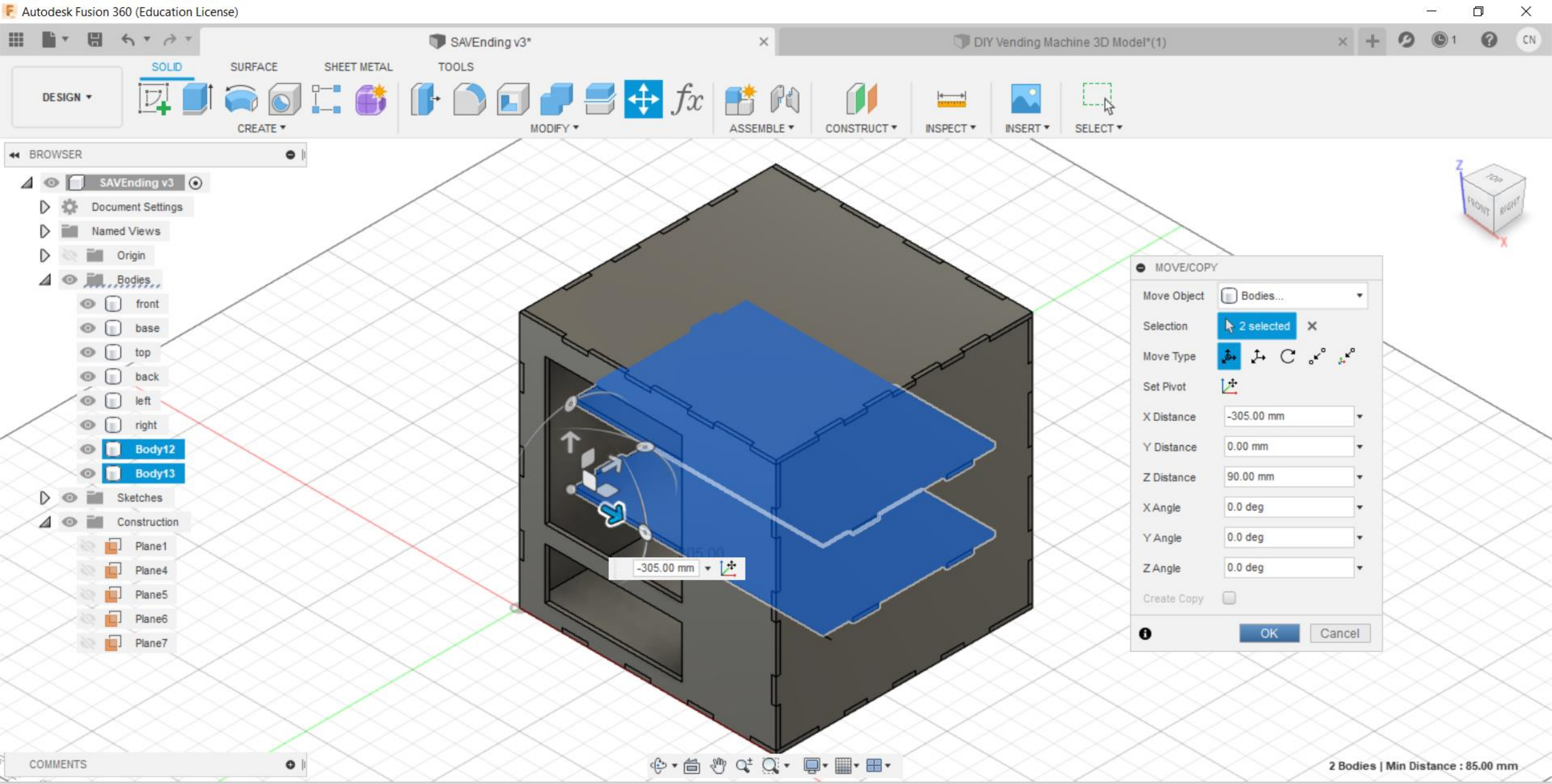
6. Continuing this mistake, I wasted much time repeating the same sketches, patterns, mirroring and cutting. Moving on, I used the 4 sides to cut a newly extruded left piece, then mirror again.



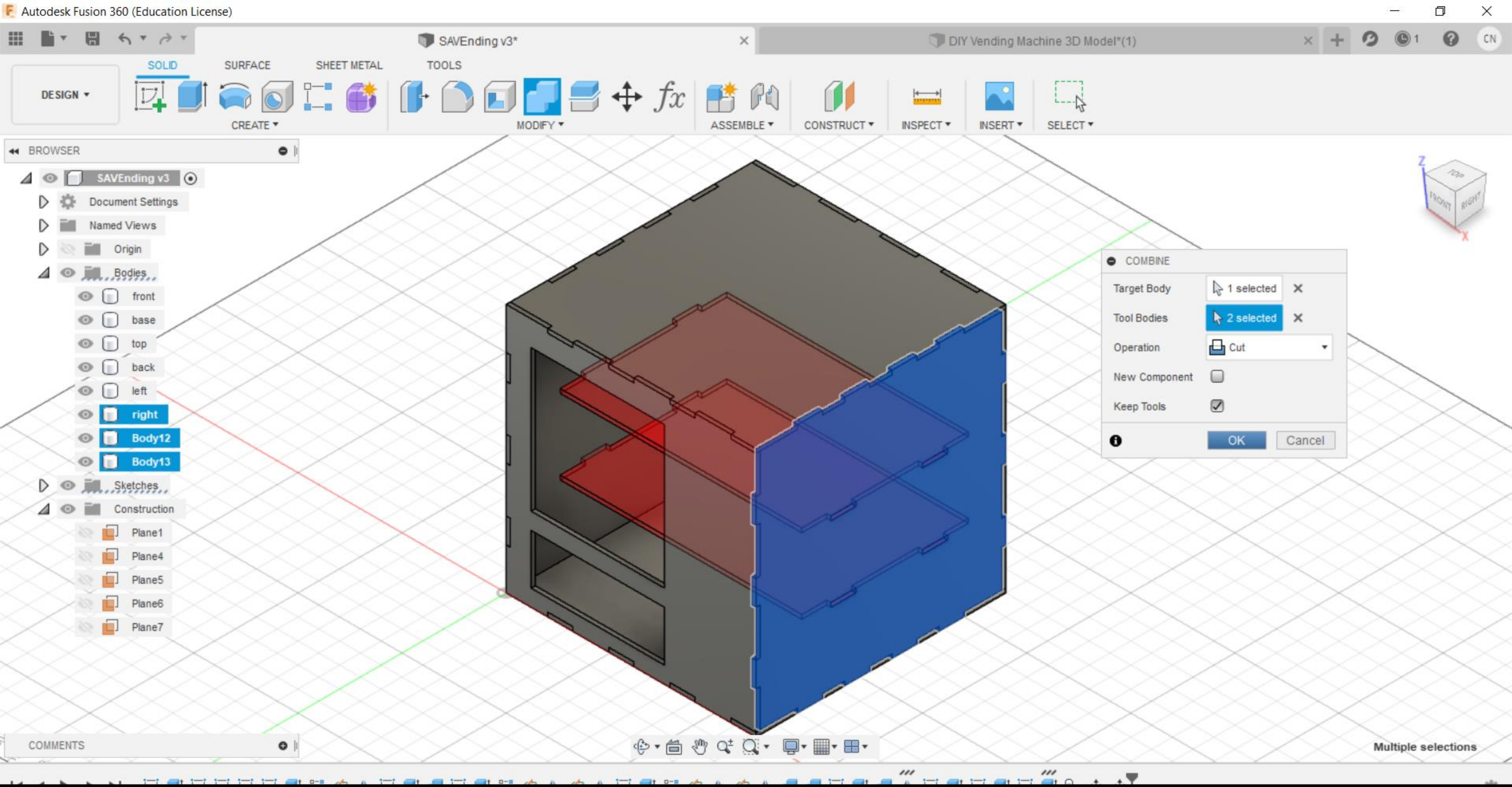
7. I drew these rectangles then extruded them. I sketched this just to the right of the actual position (so I can see the gaps).



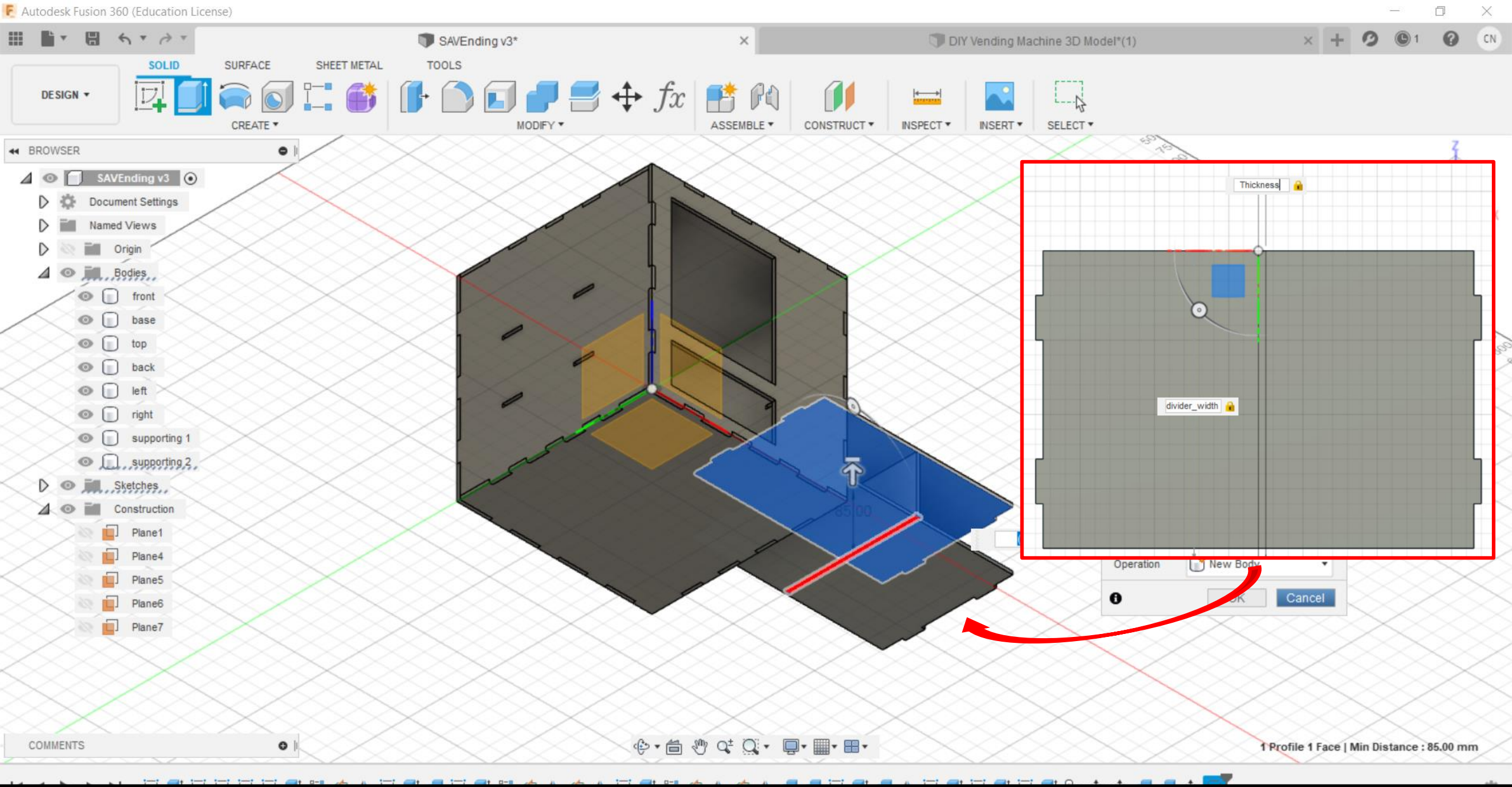
8. I extruded the sketch, then used Move/Copy function to Duplicate another one just 9cm above it.



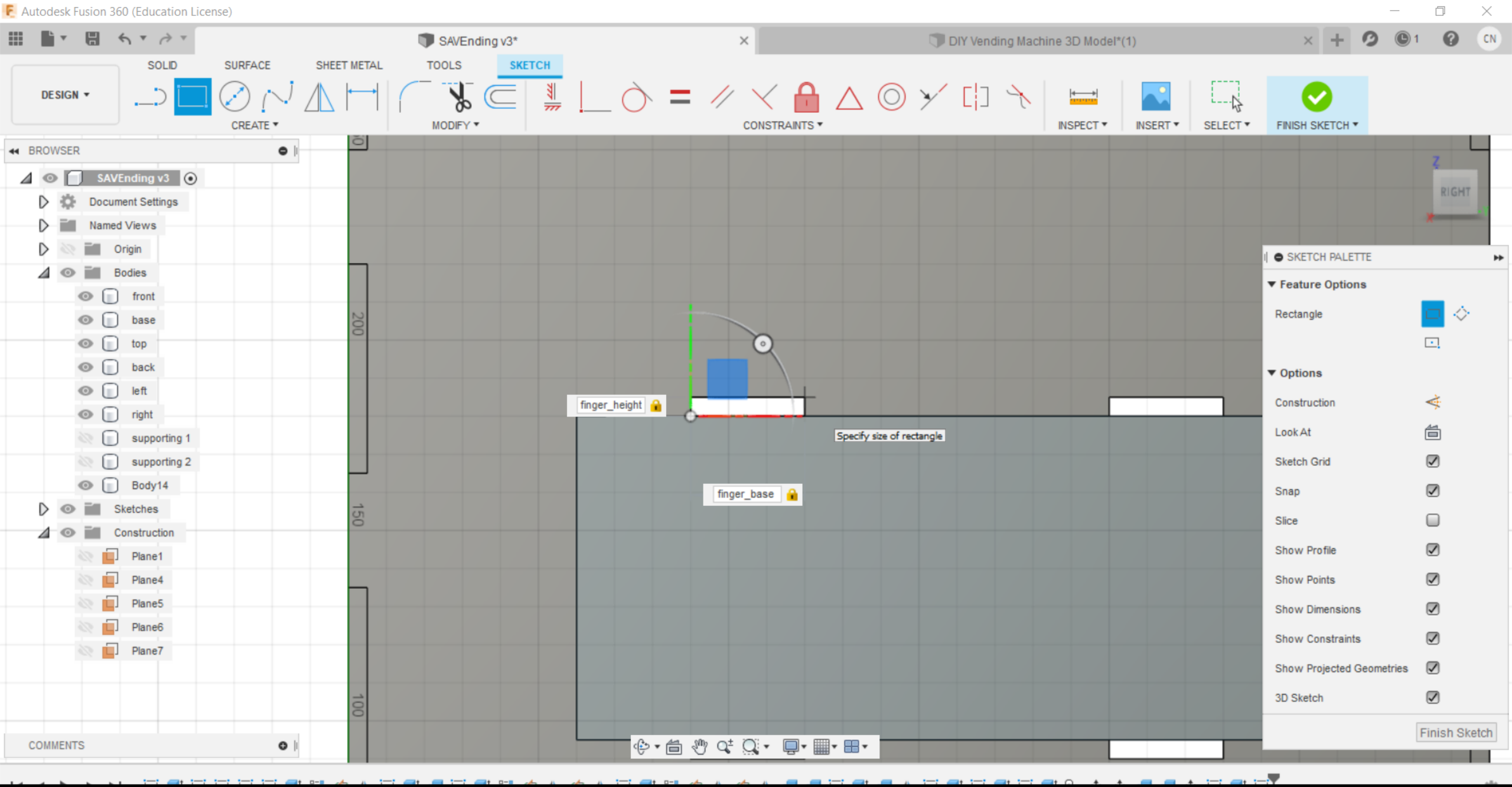
9. Select Move/Copy, but this time to move these 2 supporting plates into my box.



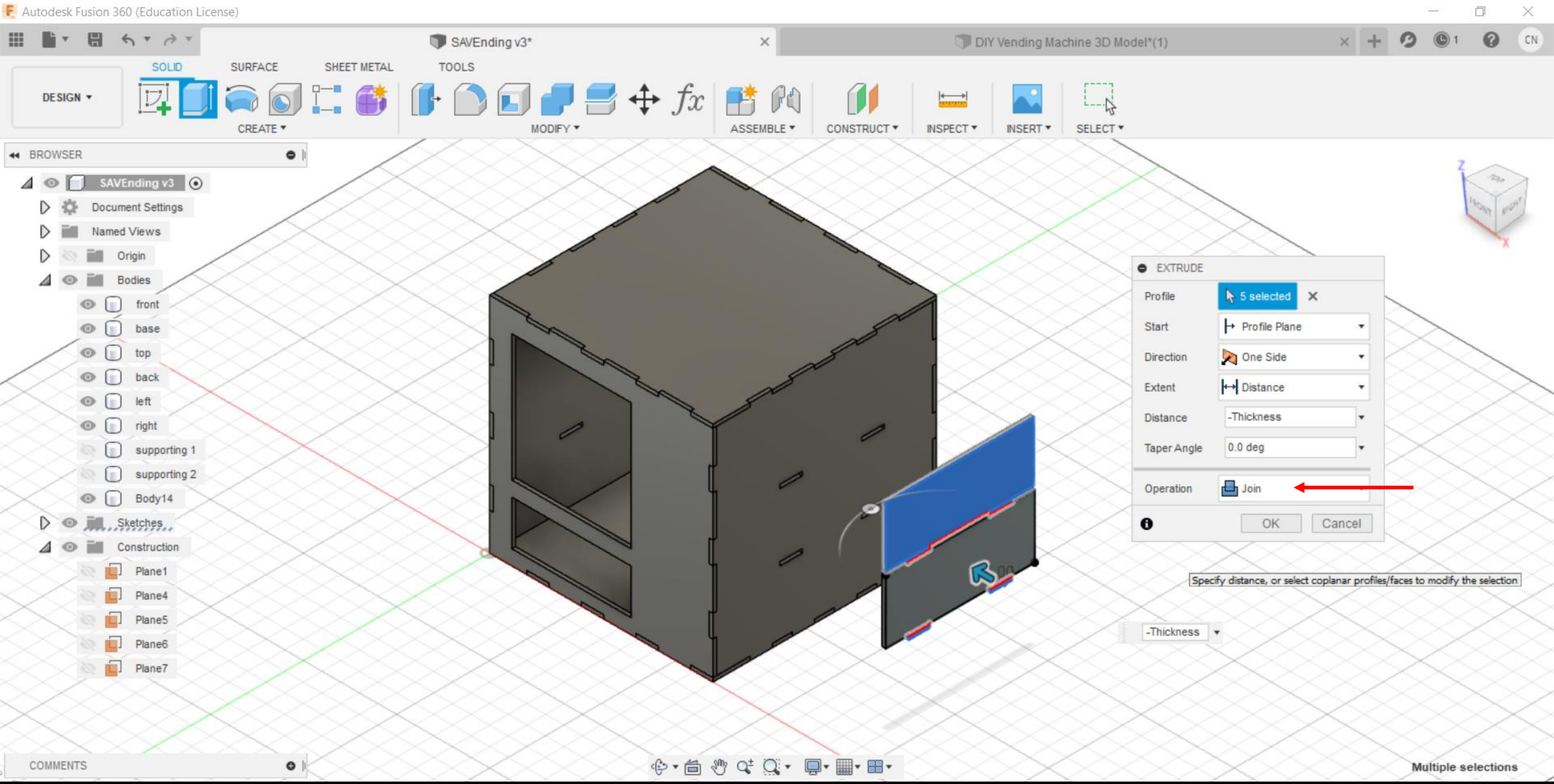
10. Cut holes on left and right walls for the supporting plates to fit nicely. It’s fun knowing that these plates will fit nicely.



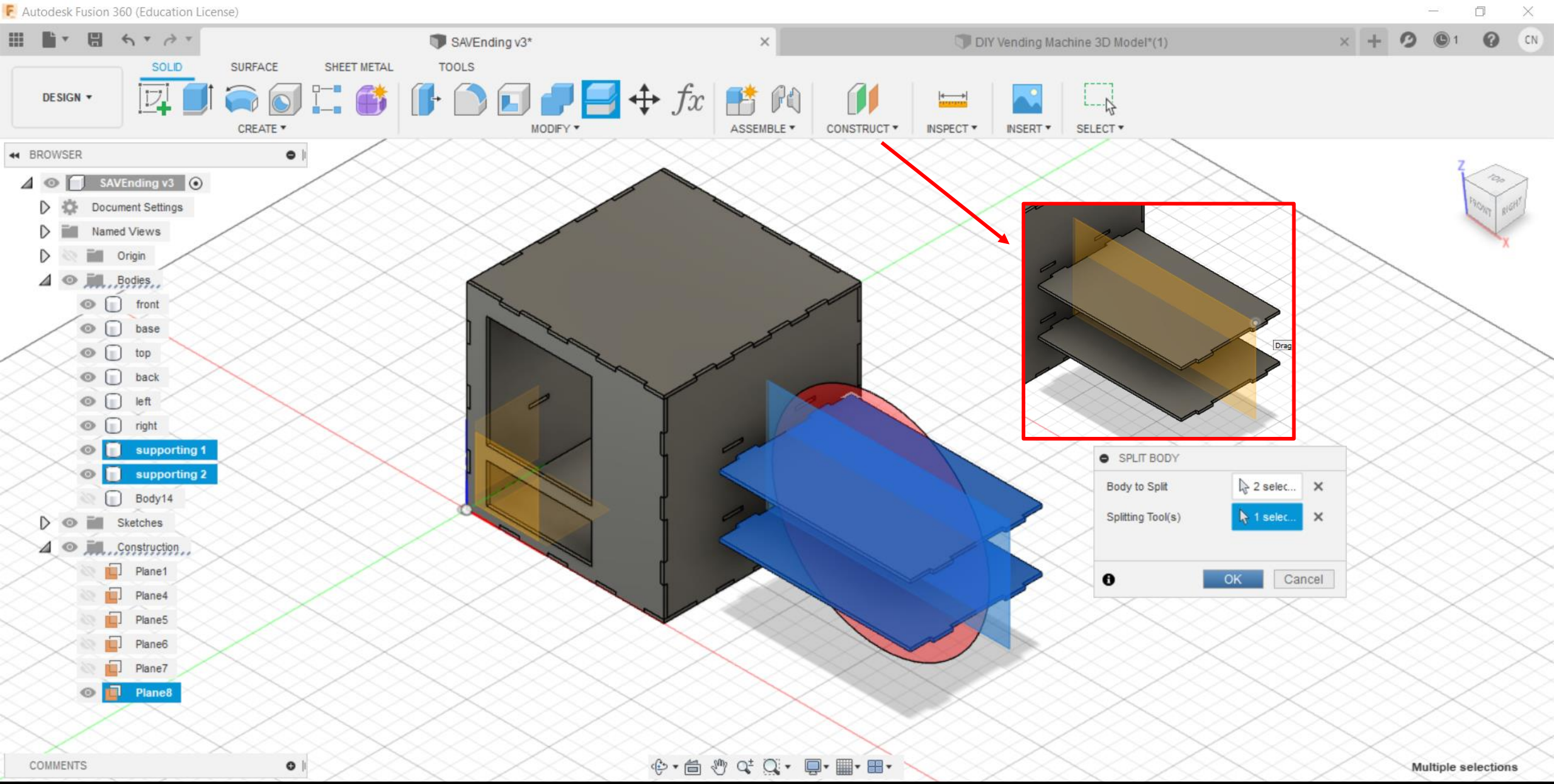
11. I drew a rectangle (*Thickness x Divider_width*) on the lower plate and extrude till it snaps to the bottom of the upper plate.



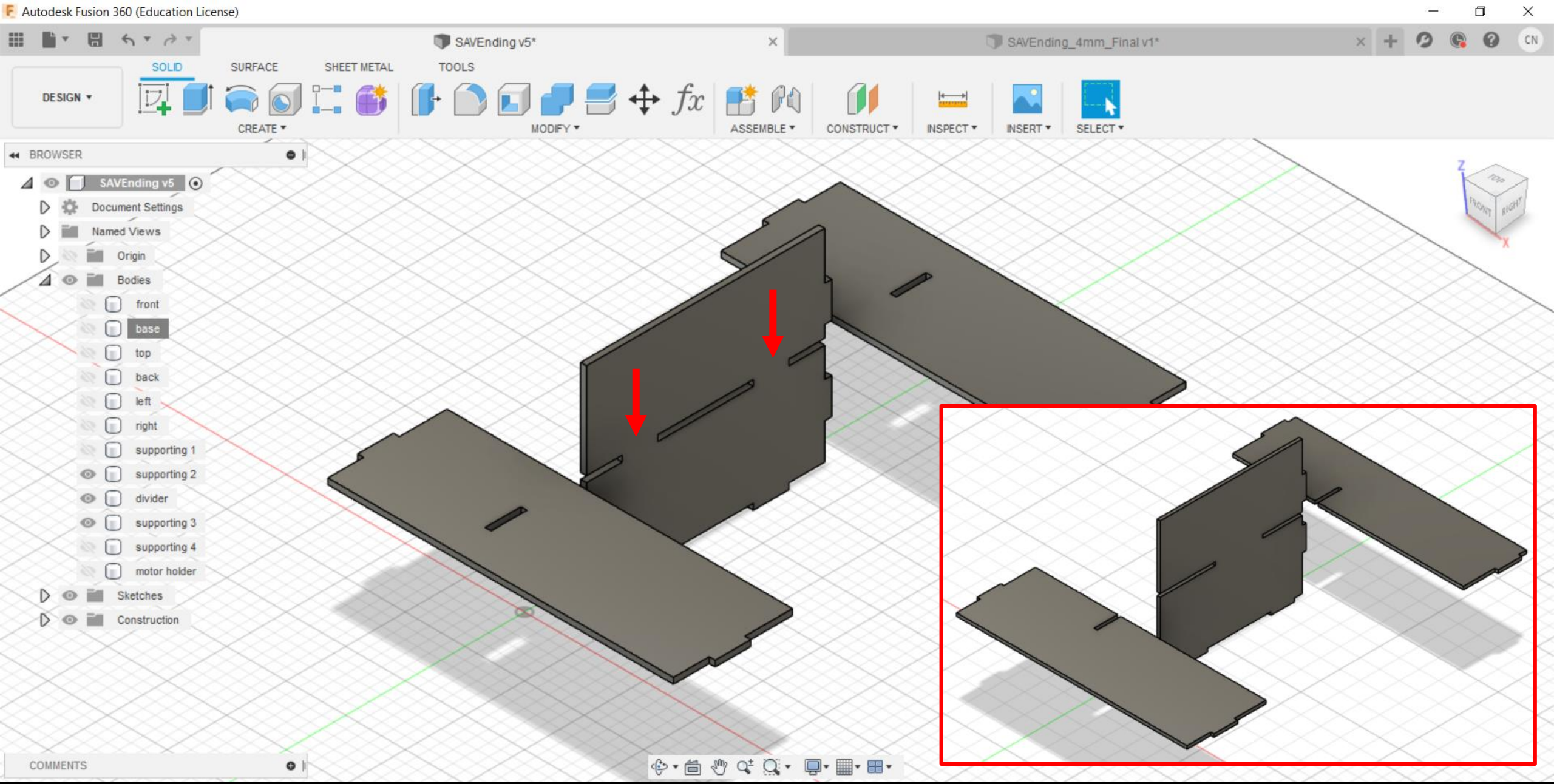
12. By facing left, I can create the fingers of the divider in the corresponding positions. Then extrude.



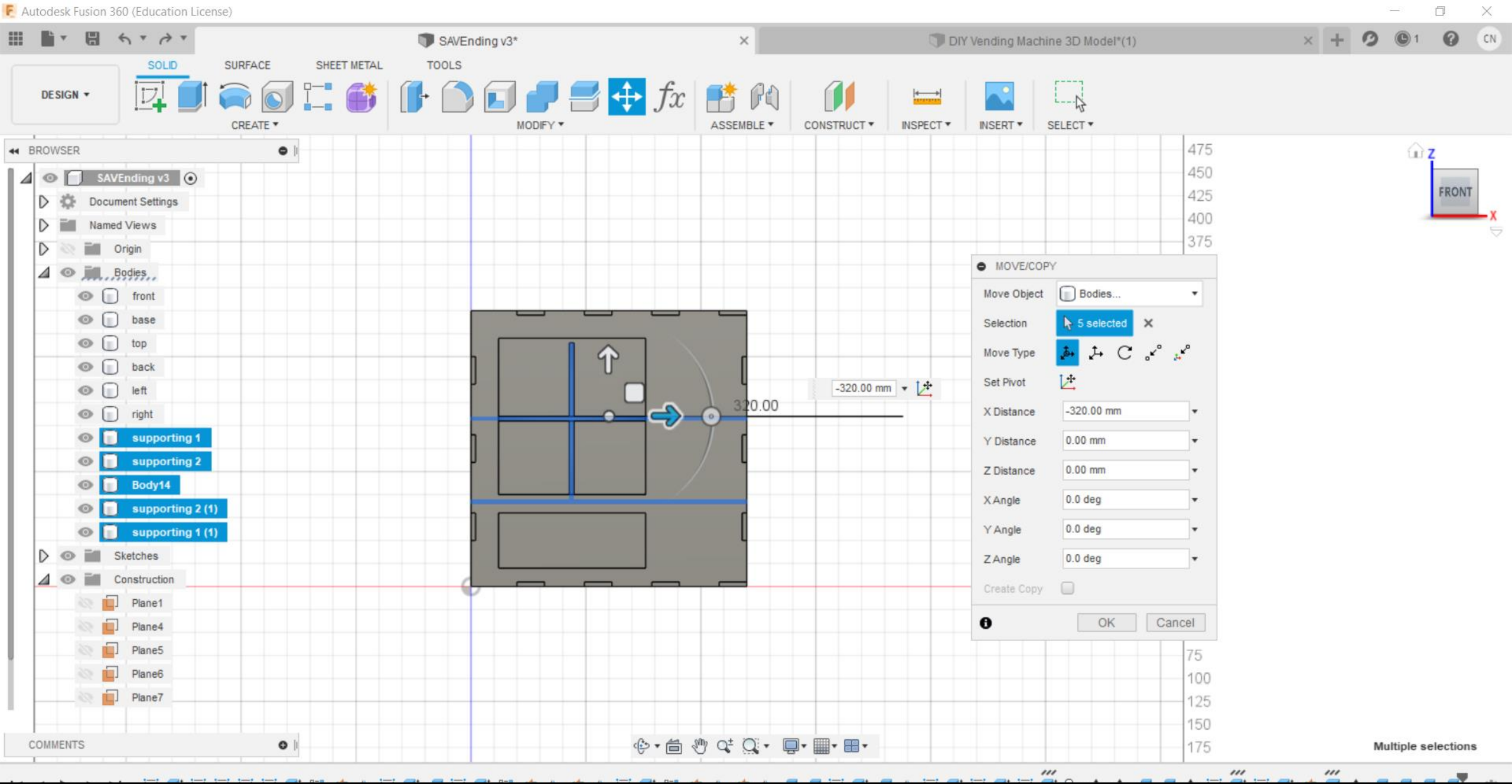
13. I created the top part of the divider, which I now realised I could have just make a whole sketch with holes spanning from one finger of the wall to another and towards the edges, then extrude desired rectangles.



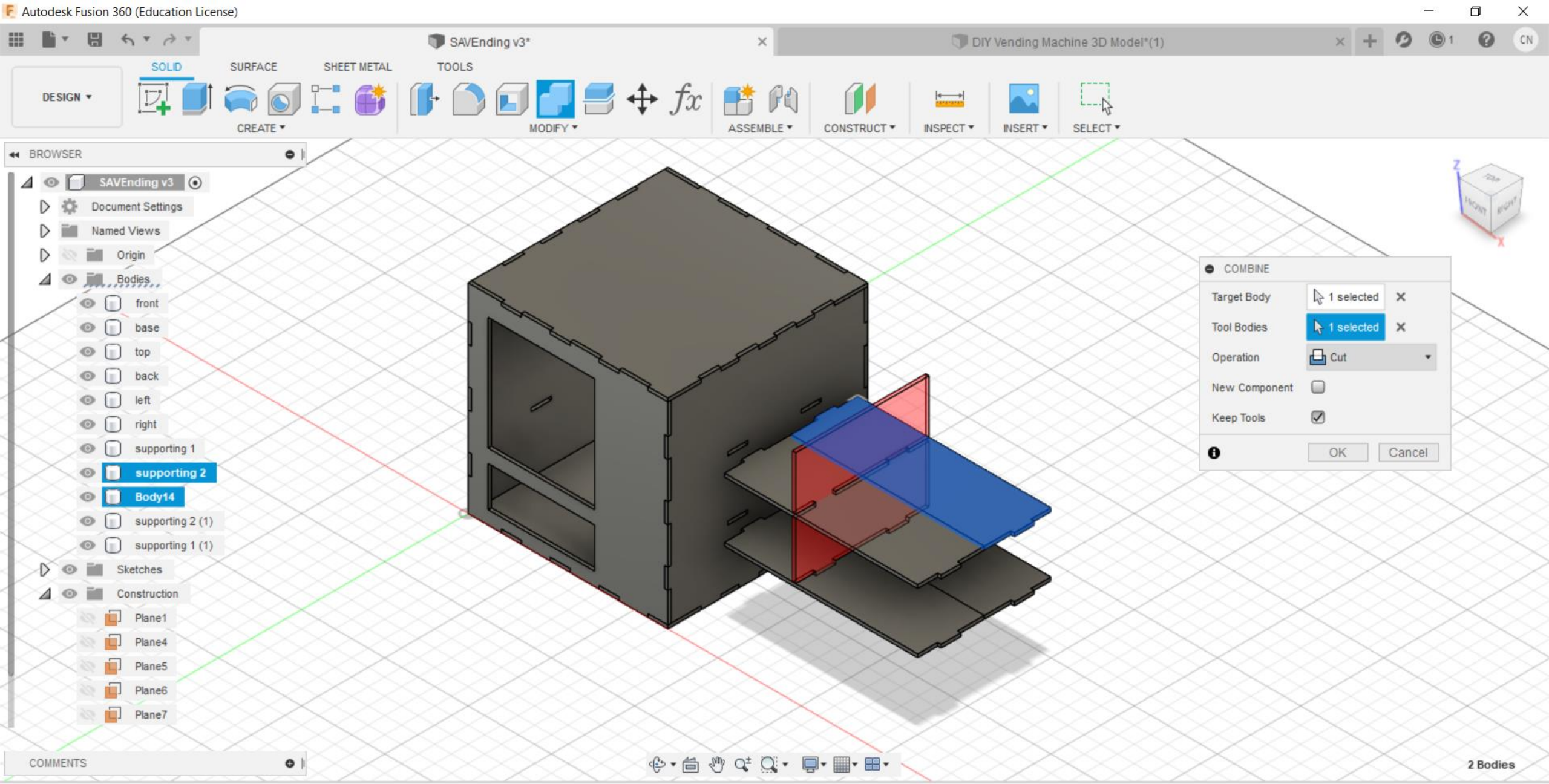
14. Constructed a midplane and used that to split the supporting plates.
The initial reason is to allow the divider to fit in, but something's wrong… It is only when I was about to laser cut then I realised.



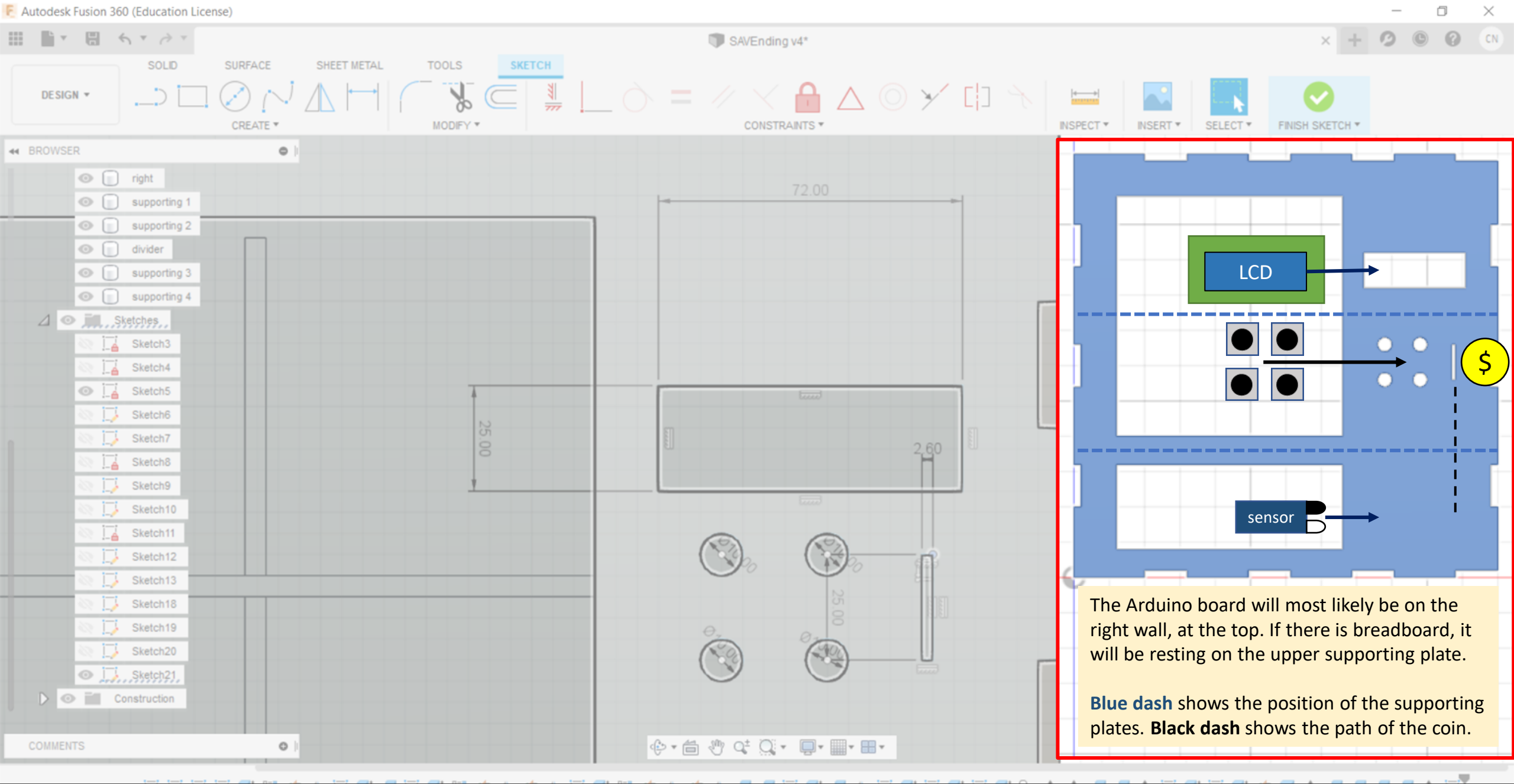
Answer: The divider will not fit the upper supporting plates at all as there is no entrance for the finger to slot.
Solution: I went back to edit the sketch and pulled the 2 centre rectangles together (merge). This way it creates press-fit plates.



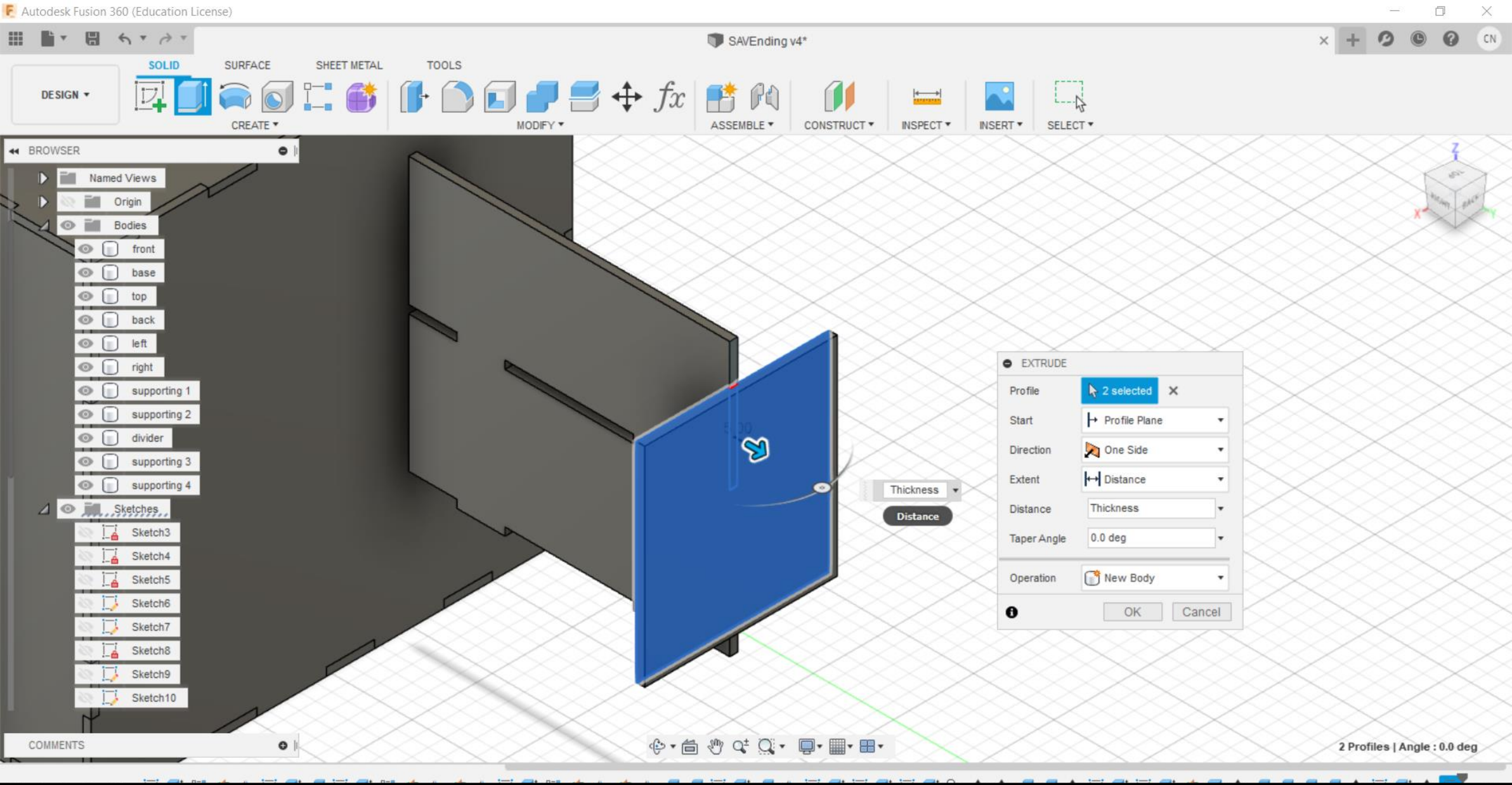
15. Problem isn’t solved here but it does not affect the following steps. I selected all internal plates and shift them into the box.



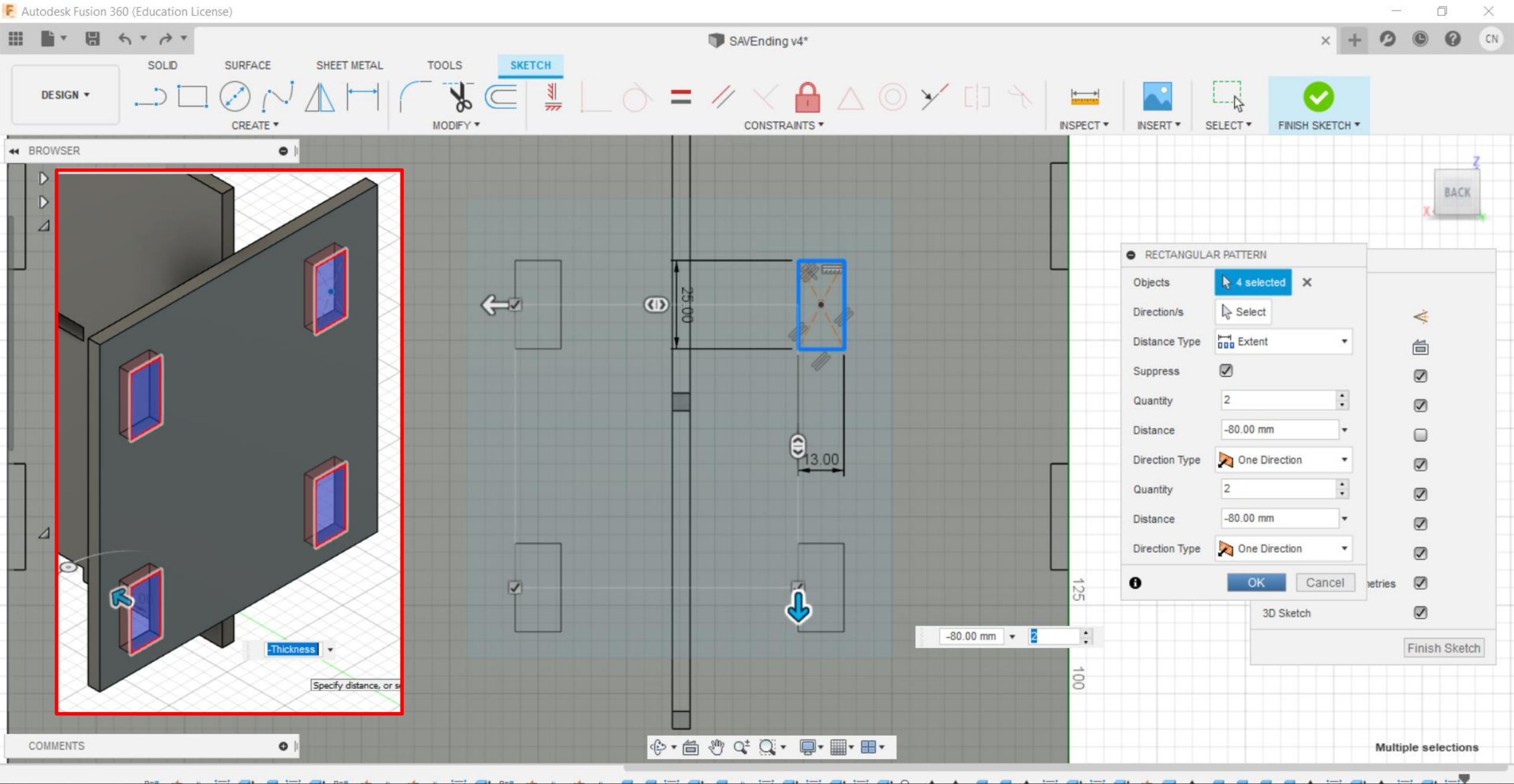
16. After fixing the position, I took them out and cut out the holes in the supporting plates using divider. Then return them.



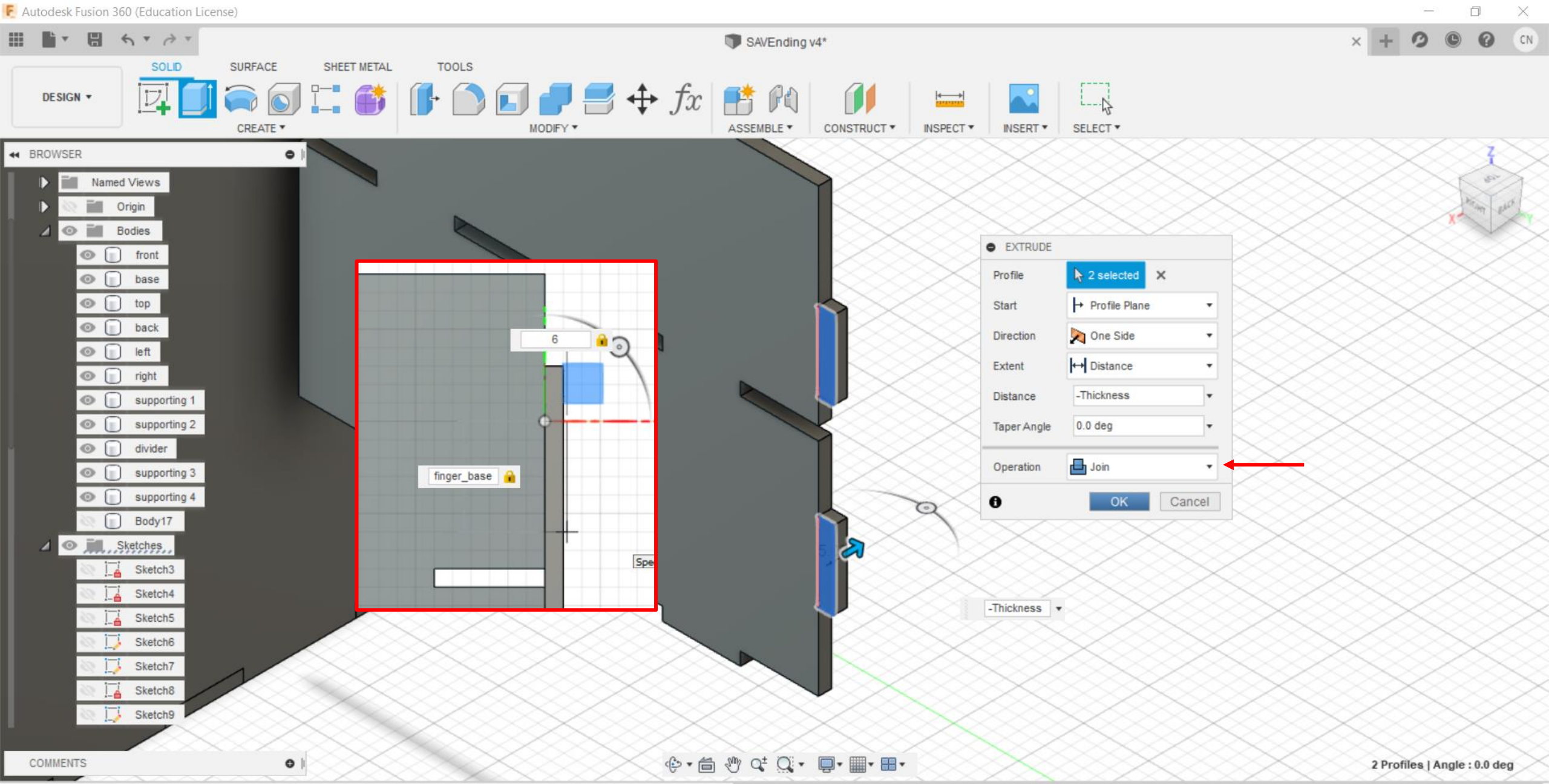
17. Now, time to consider where my components will be. The smaller image is the arrangement after taking into consideration.



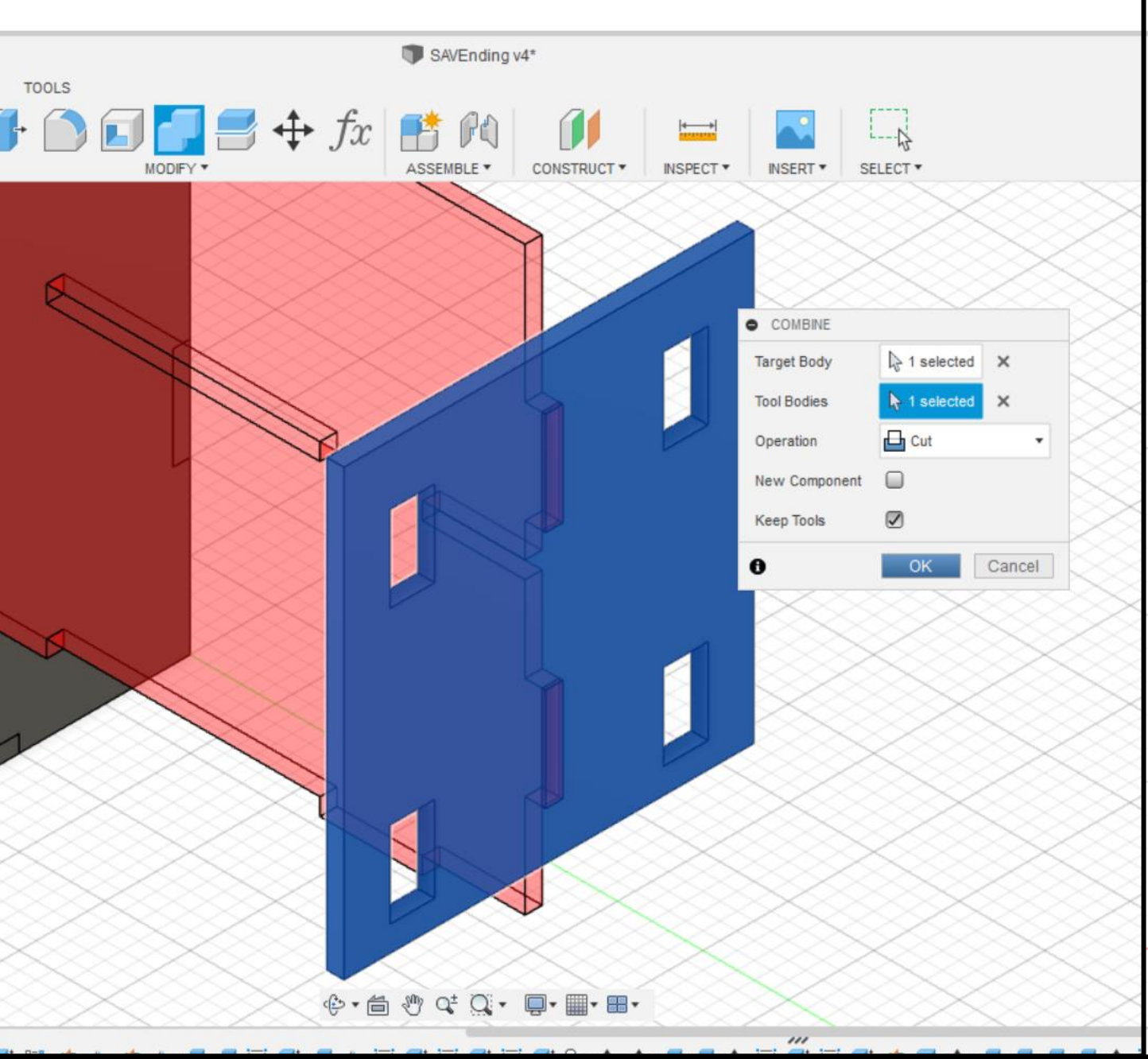
18. I need a holder for my servo motors. A small plate stuck behind the divider should do.

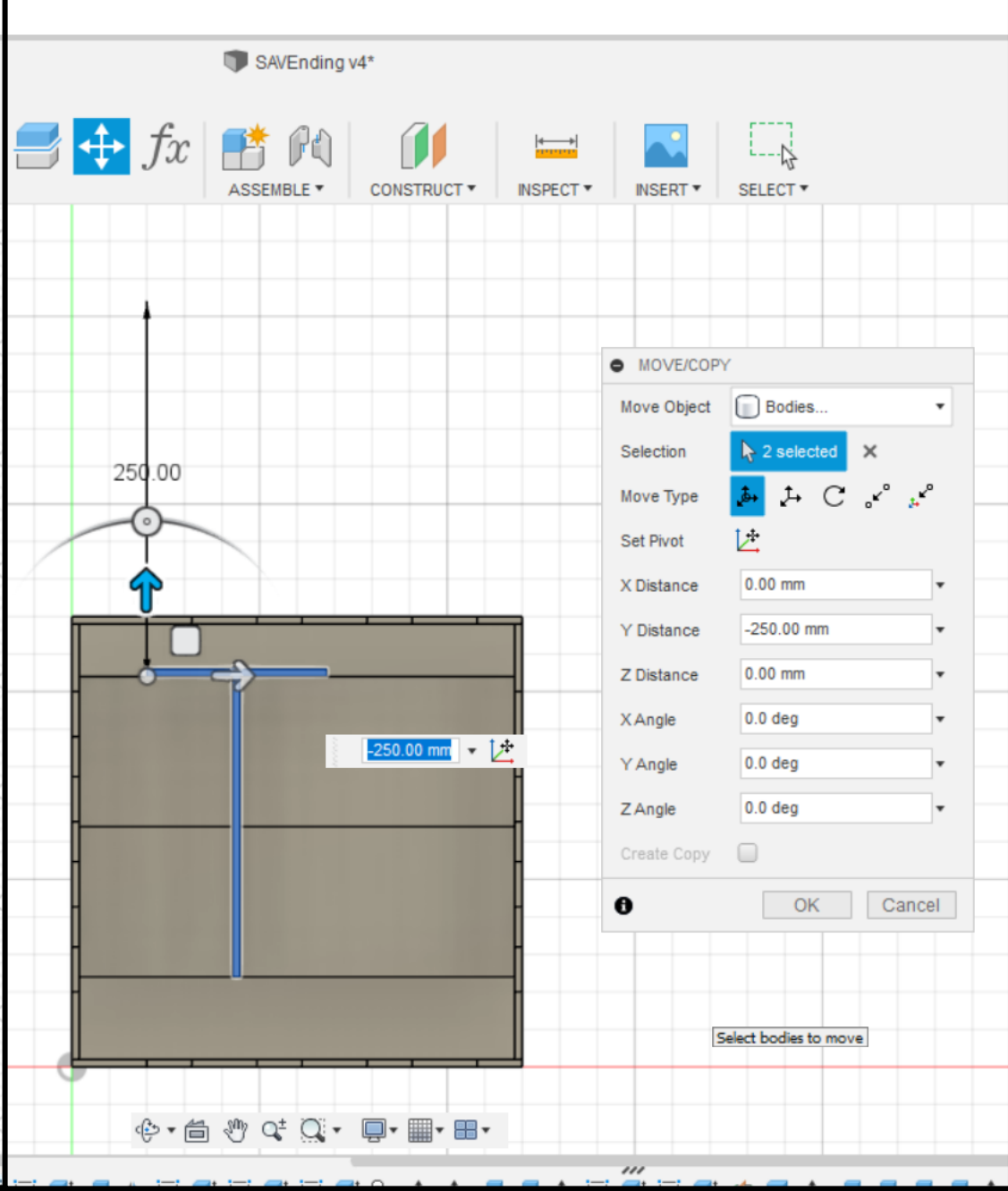


19. I drew according to the size of the motor, then used rectangular pattern to create 3 more holes, and extrude by *Thickness*.

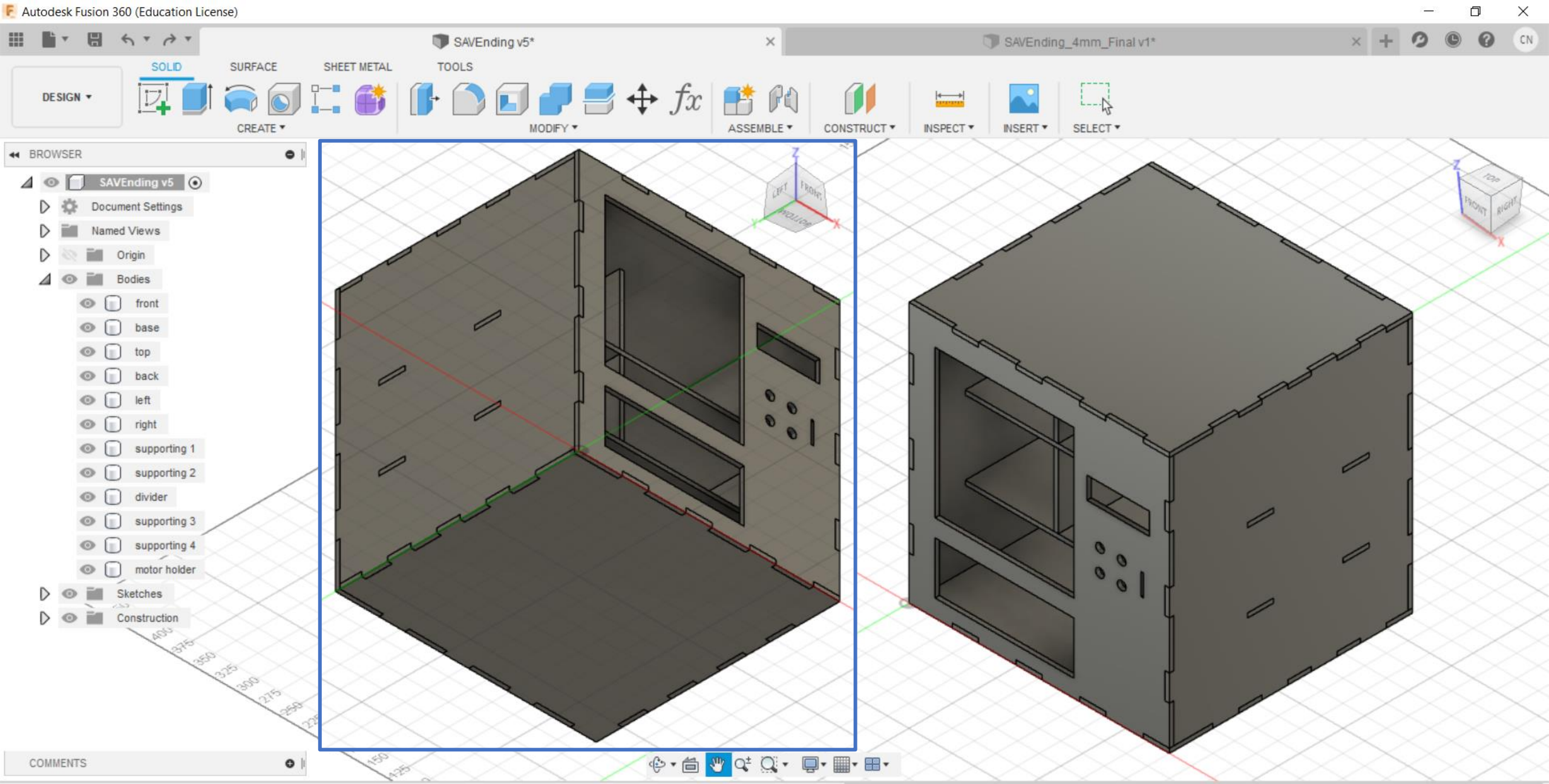


20. To hold the holder in place without glue, I created 2 fingers (instead of 1 for stability) from the back of the divider.





21. I cut the slots in the motor holder, then pushed these 2 plates into the box.



SAVEnding structure – Complete!!

What’s next?