

## How to Make an STL File from a Revit File

This tutorial will go through the steps to creating an STL file using Revit and Fusion 360.

This method can be used with both Revit 2019 and Revit 2020.

### Converting the Revit File to a DXF

Start by making a copy of the floor plan you are choosing to 3D print. In the copy of the floor plan remove all components that are not floors and walls. To ensure having the best quality print it is necessary the body being printed only includes floor and walls.

Once your model is only composed of floors and walls go to File -> export -> CAD Formats -> DFX (as show in Figure 1). Once you have selected DFX you will be prompted to the window show in figure 2.

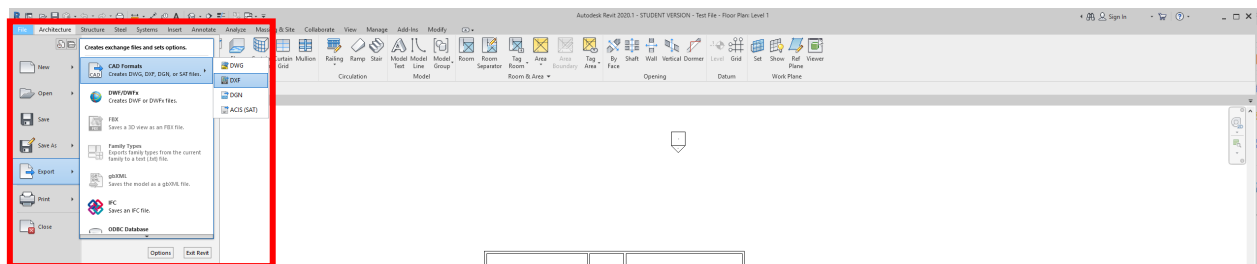


Figure 1: Finding the Export Feature on Revit

In the DFX Export window verify the correct floor plan is being exported and select next.

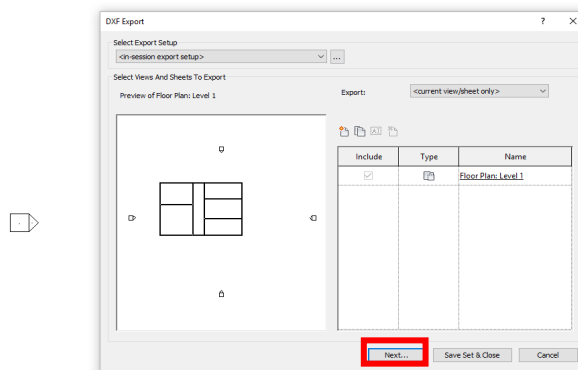


Figure 2: DFX Export Window

You will be directed to save the file as shown in figure 3. Once ok is selected the floor plan will be exported into a DXF format.

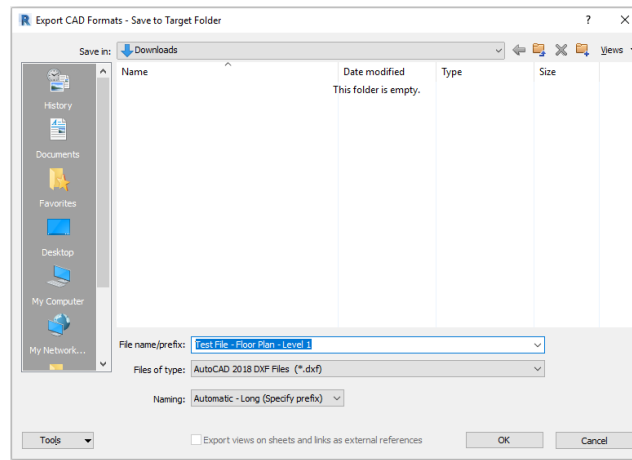


Figure 3: Saving the DXF File

## Exporting a DXF File into Fusion

First open a blank Fusion File. Then proceed to Open -> Open from my Computer -> Select the correct file -> Open. Back on the Fusion Design Space, the floor layout should load. Please note that it is only two dimensional. All these steps are illustrated in figures 4-6.

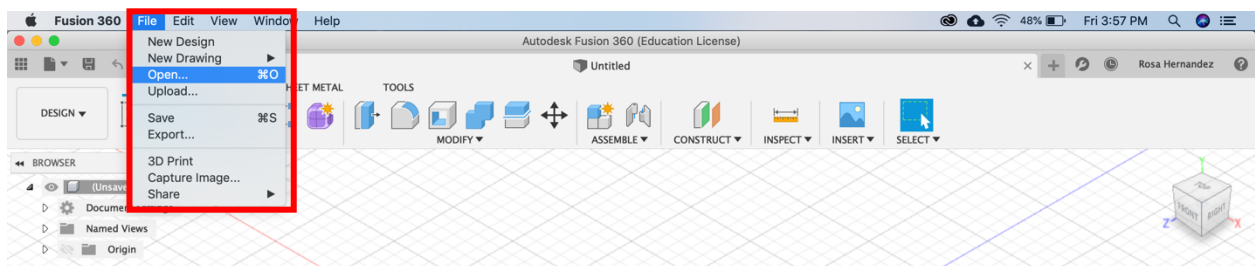


Figure 4: Starting a New Fusion File

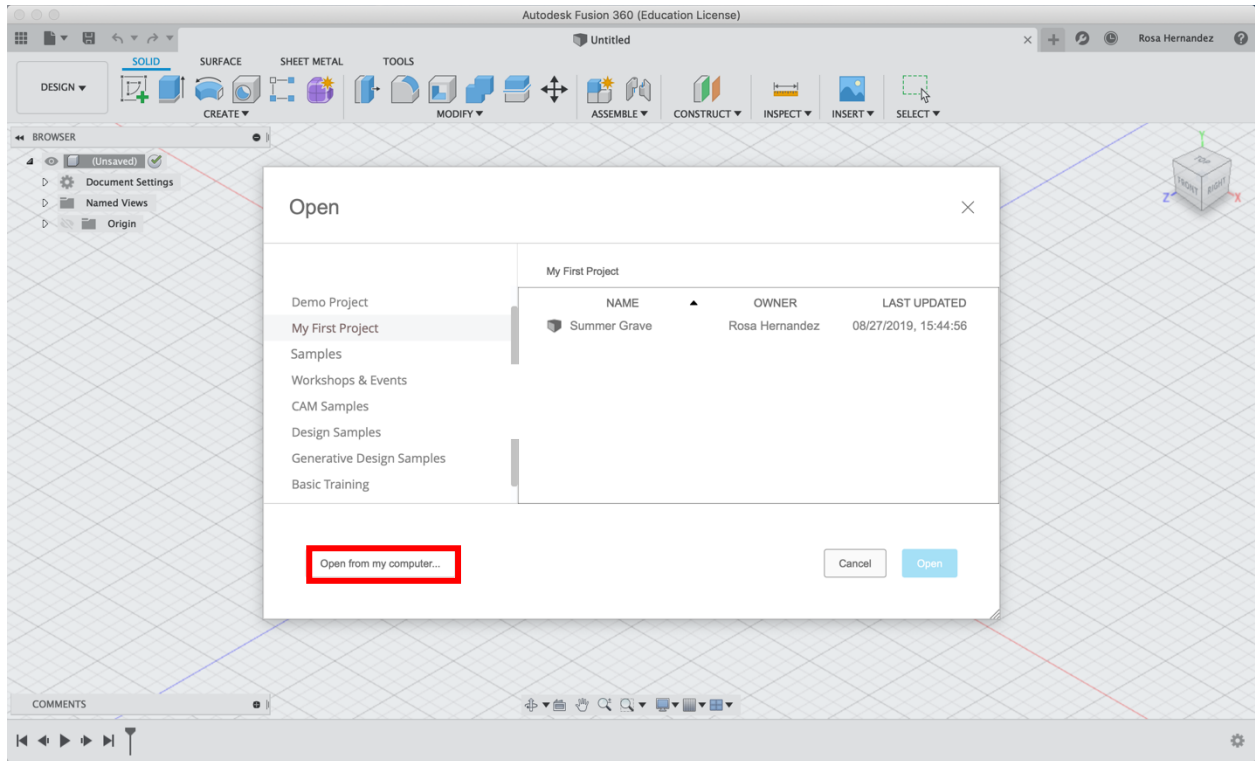


Figure 5: Choosing Where to Open the File From

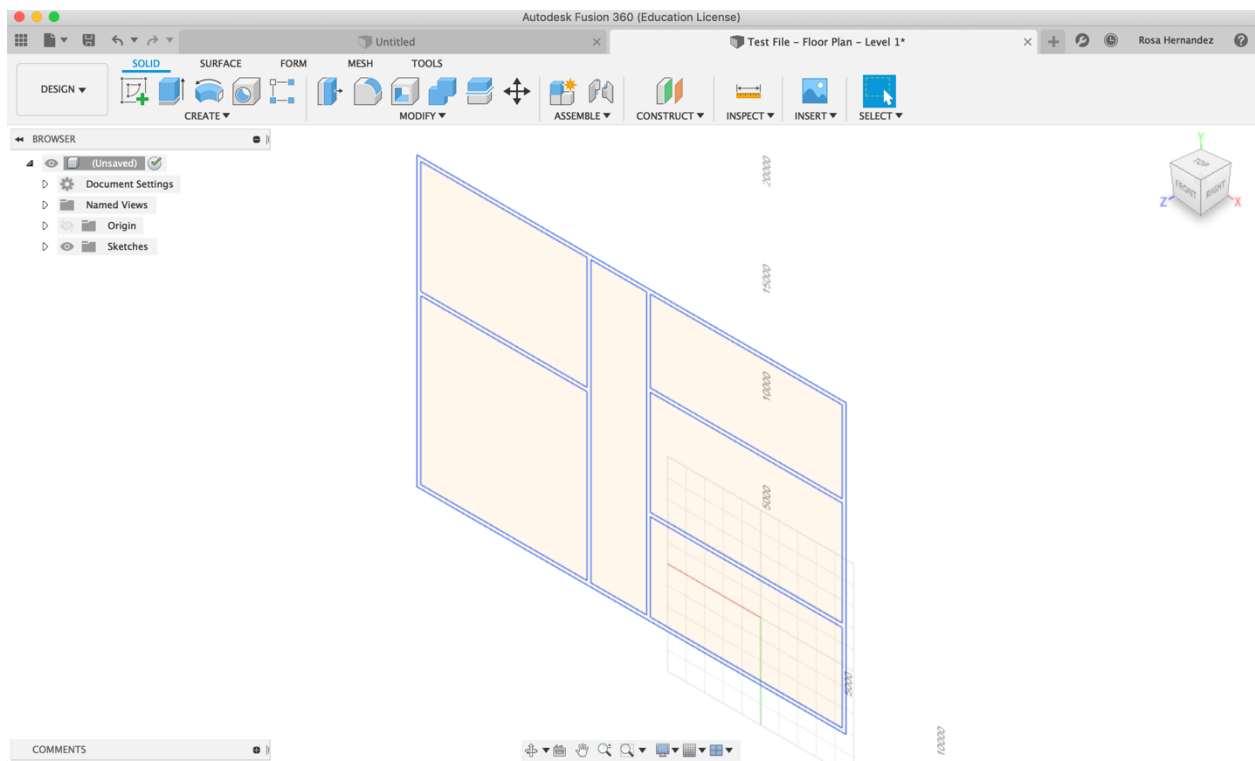


Figure 6: Floor Layout in Fusion

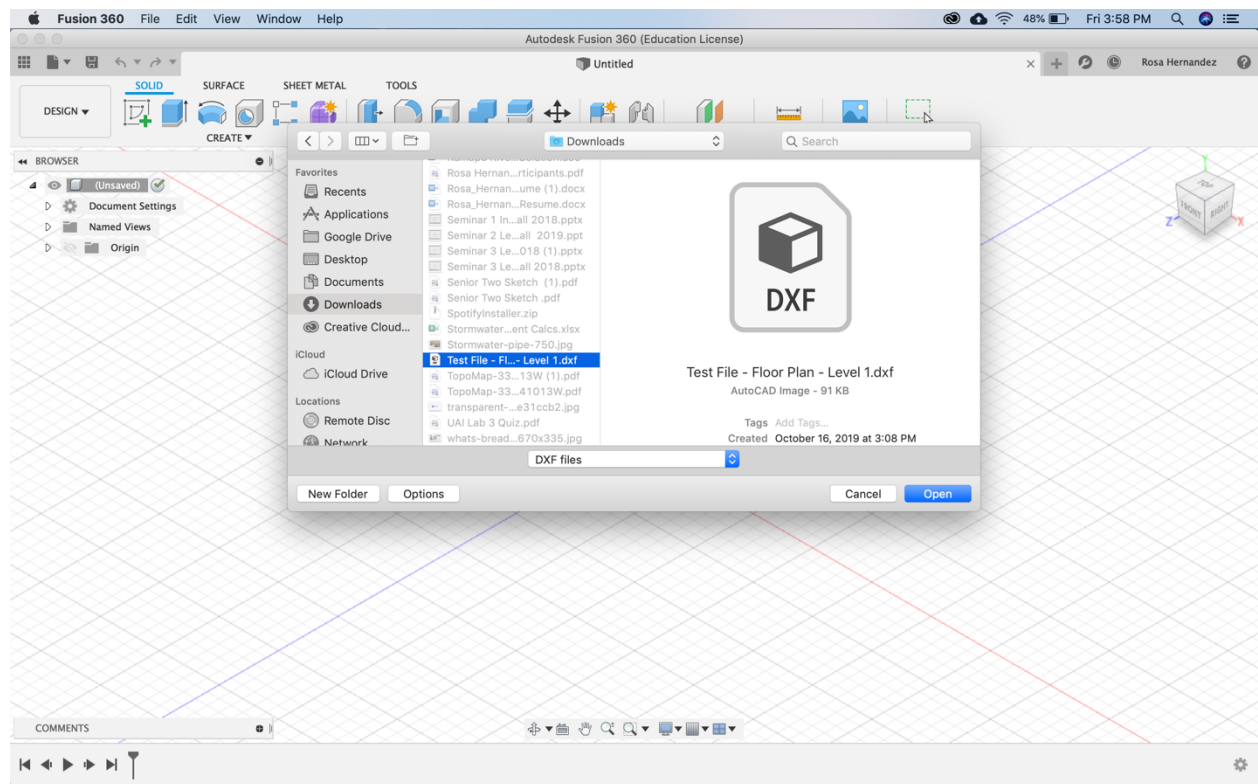


Figure 7: Selecting the DXF File



## Scaling the Model

In order to ensure that the walls and floors are extruded correctly, the model needs to be first scaled down. The model will be scaled so the length and width of the building does not exceed 7 inches (177.8 mm). To scale the model the sketch for the floorplan has to be edited. In the browser tab, under sketches, find the sketch for the walls, right click and select edit sketch.

In order to use the scale feature on Fusion you will need to select a point that is not part of the components being scaled. As shown in figure 8 click on the dropdown menu on the Create tab and select point, this will allow you to create a sketch point.

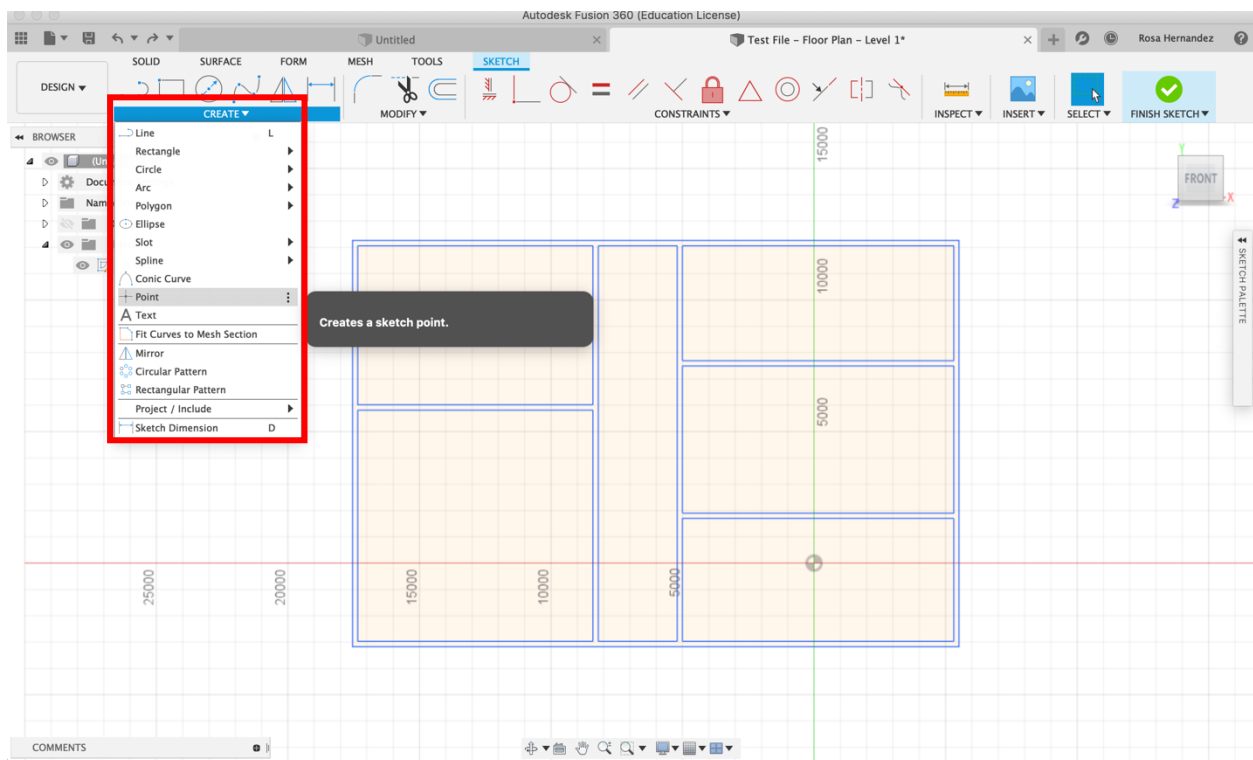


Figure 8: Finding the Point Feature

Once you have selected the point feature on Fusion click anywhere on the sketch to set the point. Please ensure the point you place is outside of the floorplan as shown in figure 9.

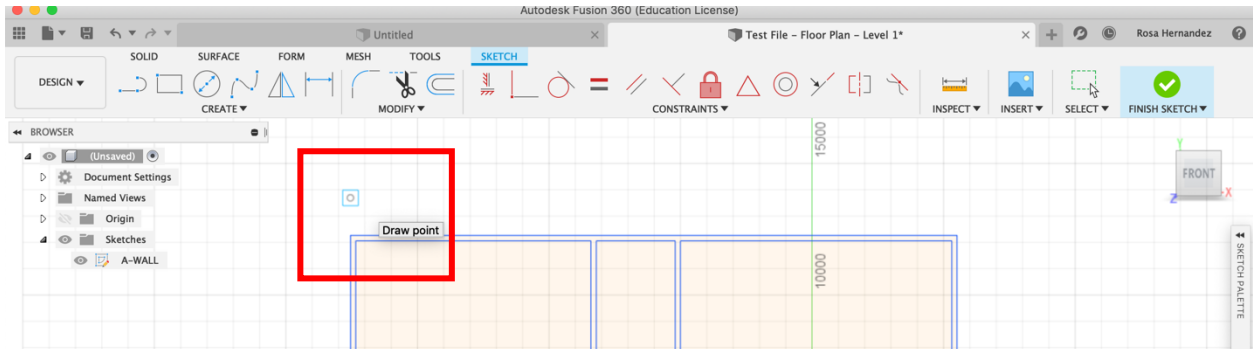


Figure 9: Creating a Point

Once the point has been placed the next step would be to use the sketch scale feature. Click on the dropdown menu in the modify tab and select sketch scale (shown in figure 10).

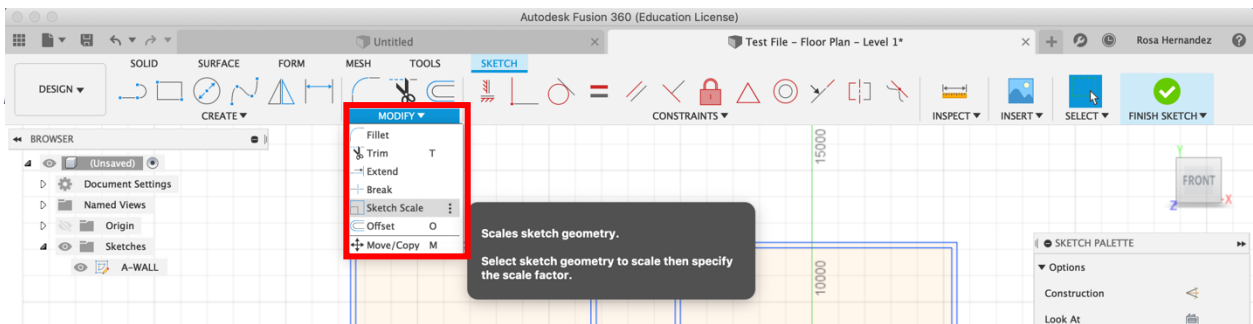


Figure 11: Finding the Sketch Scale Tool

To use the sketch scale tool, first select the entities that are to be selected. Make sure that everything that is part of the floor plan is selected, that includes everything in the sketch but the point that was created. In the sketch scale pop up box (shown in figure 12) select point and select the point that was drawn.

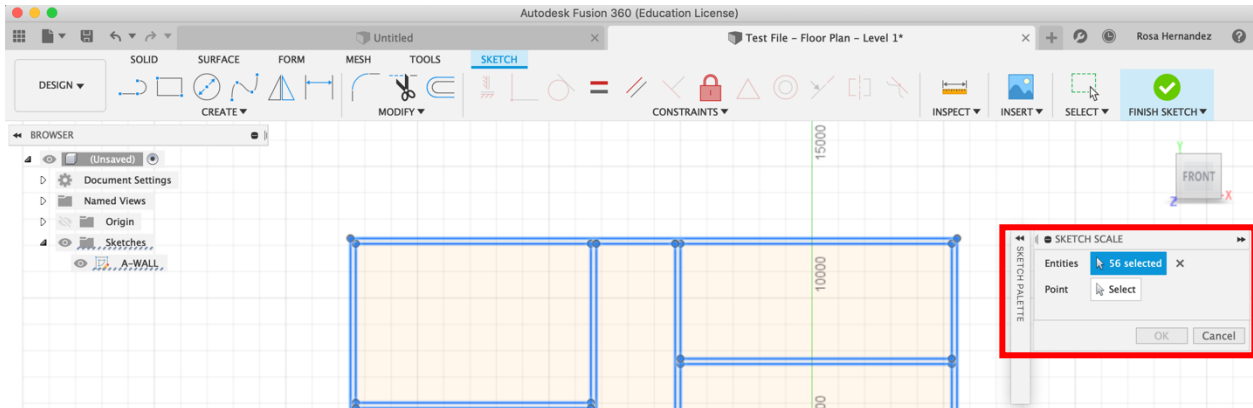


Figure 12: Sketch Scale Pop-Up

Once the entities and point have been selected, the scale factor can be designated in either of the pop-up boxes shown in figure 13. The floorplan should be scaled down so neither the width or length exceeds 7 inches (1778.8 mm), this is to ensure that the floor plan fits on the Ultimaker build plate. It may take multiple attempts of scaling to achieve the desired dimensions.

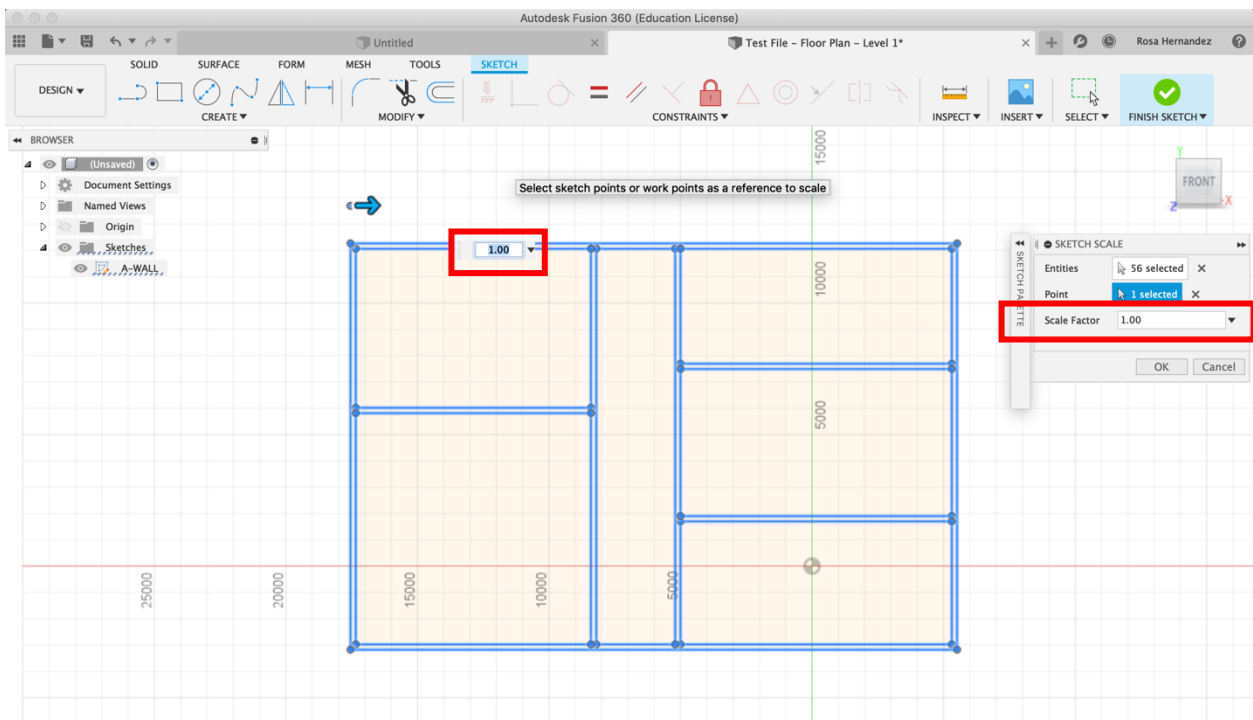


Figure 13: Designating the Scale Factor

To check if the length and width do not exceed 7 inches, the measure feature on Fusion can be used. Click on the dropdown menu under the Inspect tab and select Measure as shown in figure 14. To use the measure tool simply select two points, the distance between those two points will be displayed.

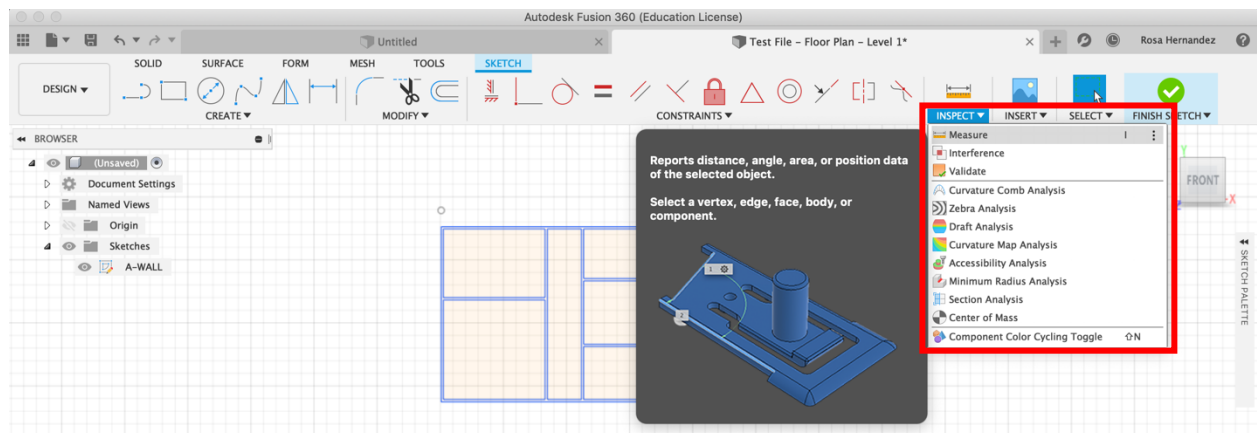


Figure 15: Finding the Measure Tool on Fusion

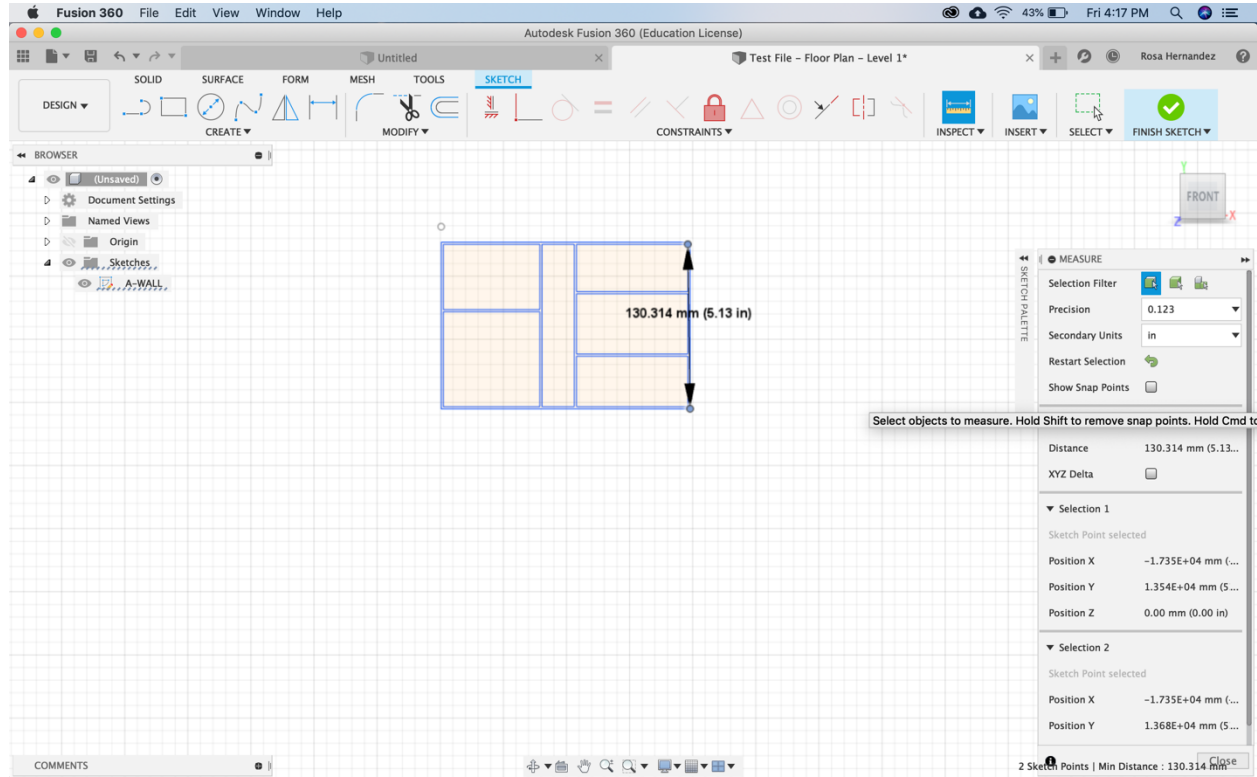


Figure 14: Using the Measure Tool

## Extruding the Walls

Click on the drop-down menu under the Create Tab and select Extrude (as shown in figure 16).

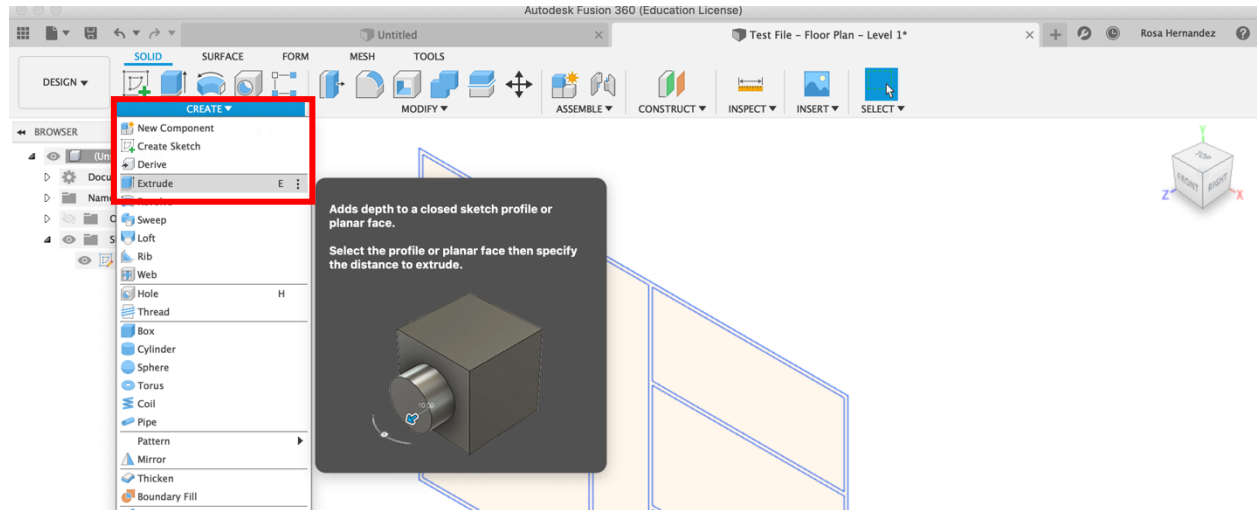


Figure 16: Finding the Extrude Tool on Fusion

Using the setting shown in the Extrude pop-up in figure 16, select the floor plan and extrude it 12.5 mm.

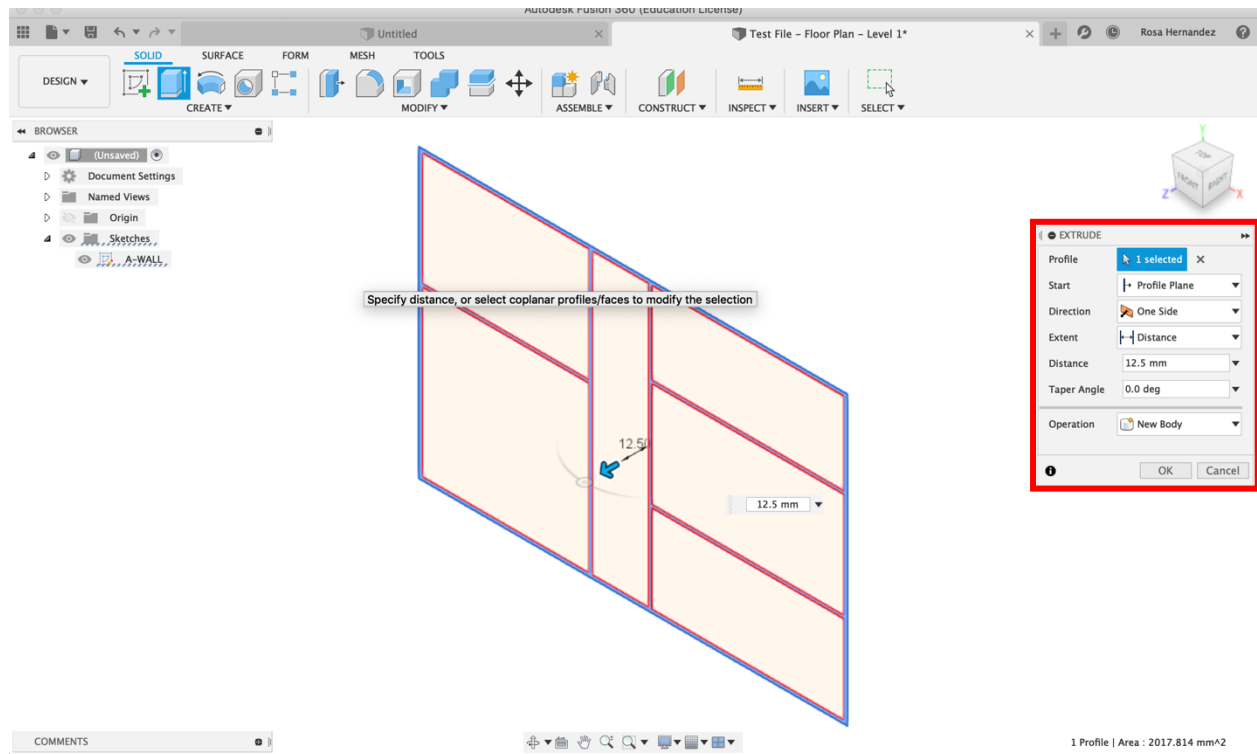


Figure 17: Using the Extrude Tool

Figure 18 shows the how the model should look like once the floors have been extruded.

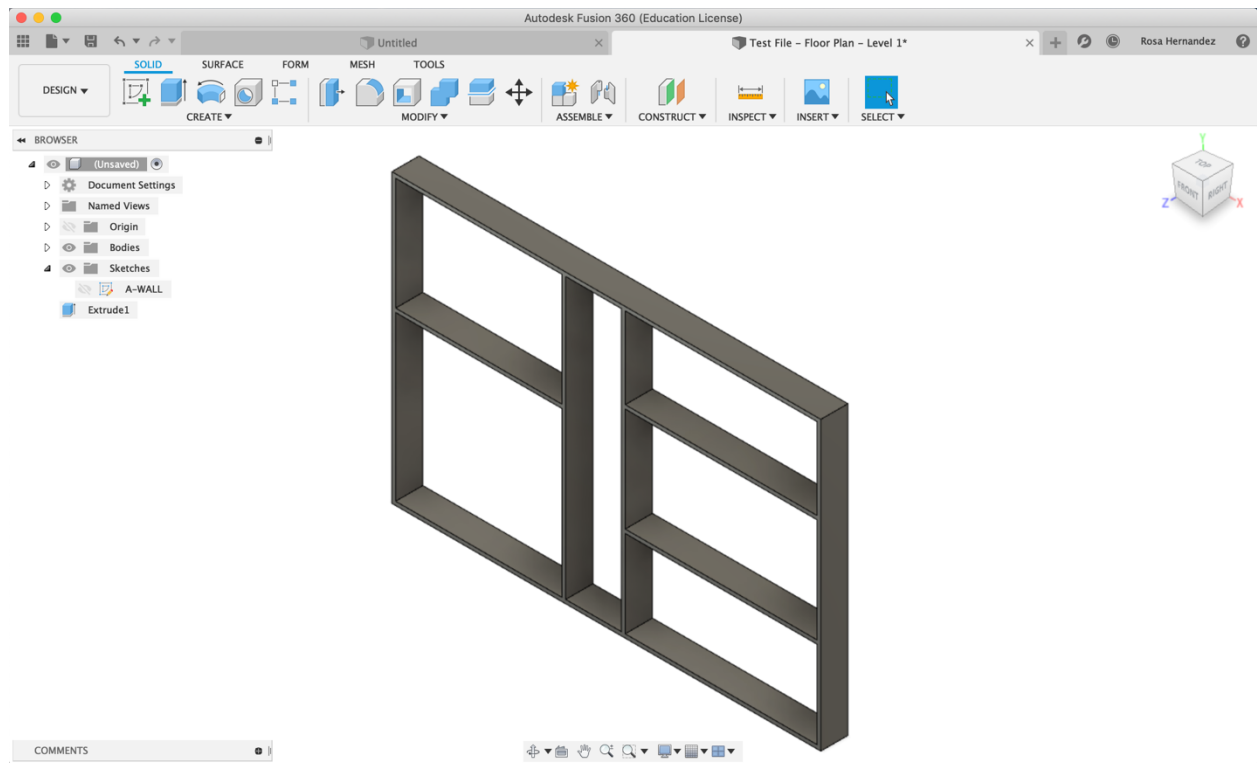


Figure 18: Extruded Floors

## Creating the Floor

In order to create a floor a new sketch needs to be made. Under the Create Tab select Create Sketch.

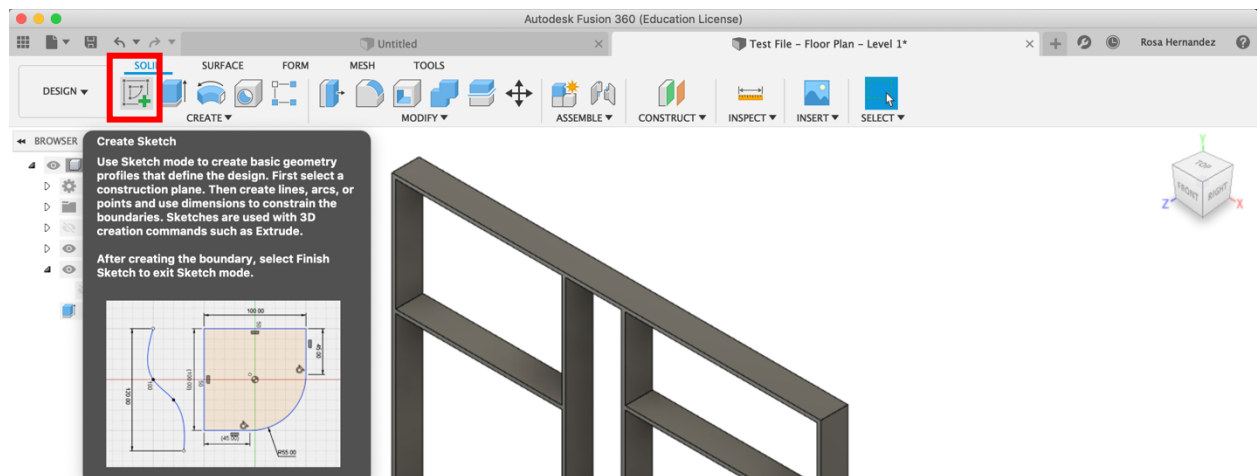


Figure 19: Finding the Create Sketch Tool

When selecting a plane to create the sketch select the outer face of the floor plan as shown in figure 20.

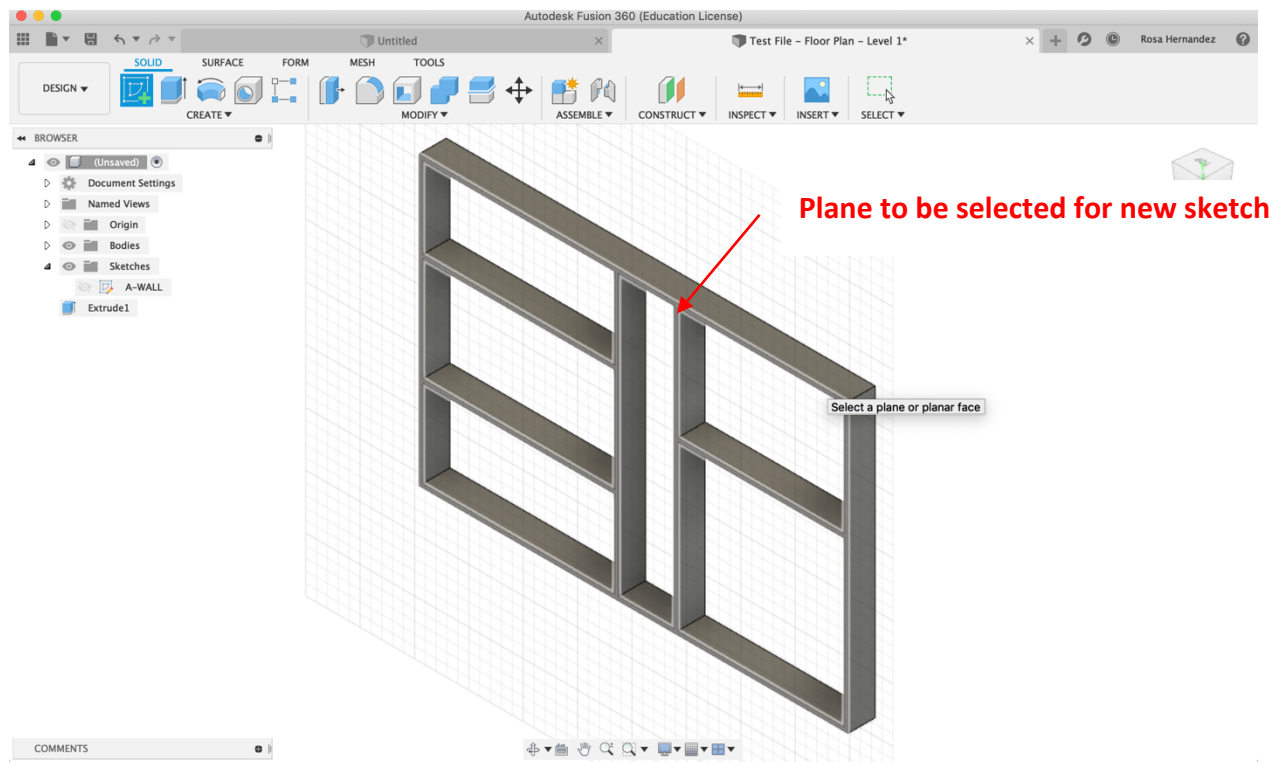


Figure 20: Selecting the Plane for the New Sketch

Once in sketch mode draw a rectangle around the perimeter of the floor plan. To draw a rectangle, go the Create Tab and select Two-Point Rectangle. Draw the rectangle by selecting two points on the floor plan as seen in figure 21.

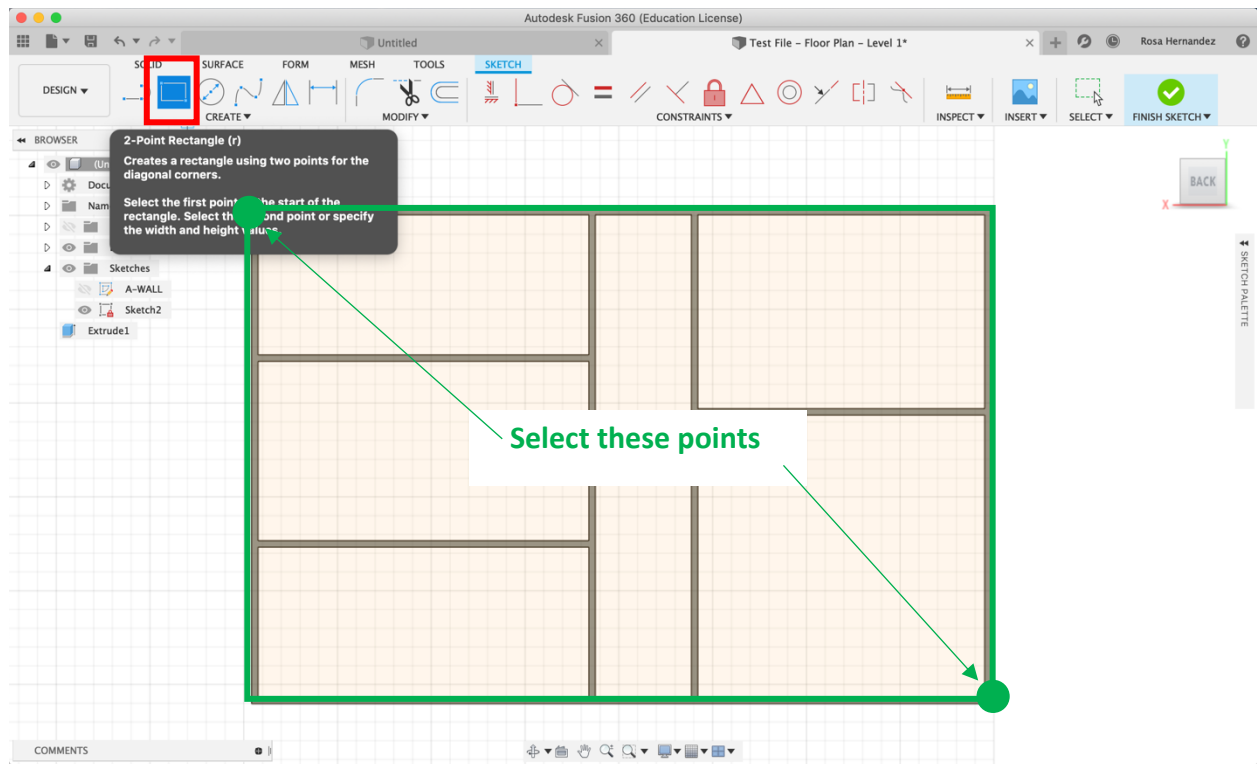


Figure 21: Drawing a Rectangle Around the Perimeter

Exit sketch mode once the rectangle has been created.

## Extruding the Floor

Using the same procedure described earlier extrude the floor by 2.5 mm. When selecting the profile that needs to be extruded, it may be needed to select multiple profiles.

The finished model should like the one in figure 22.



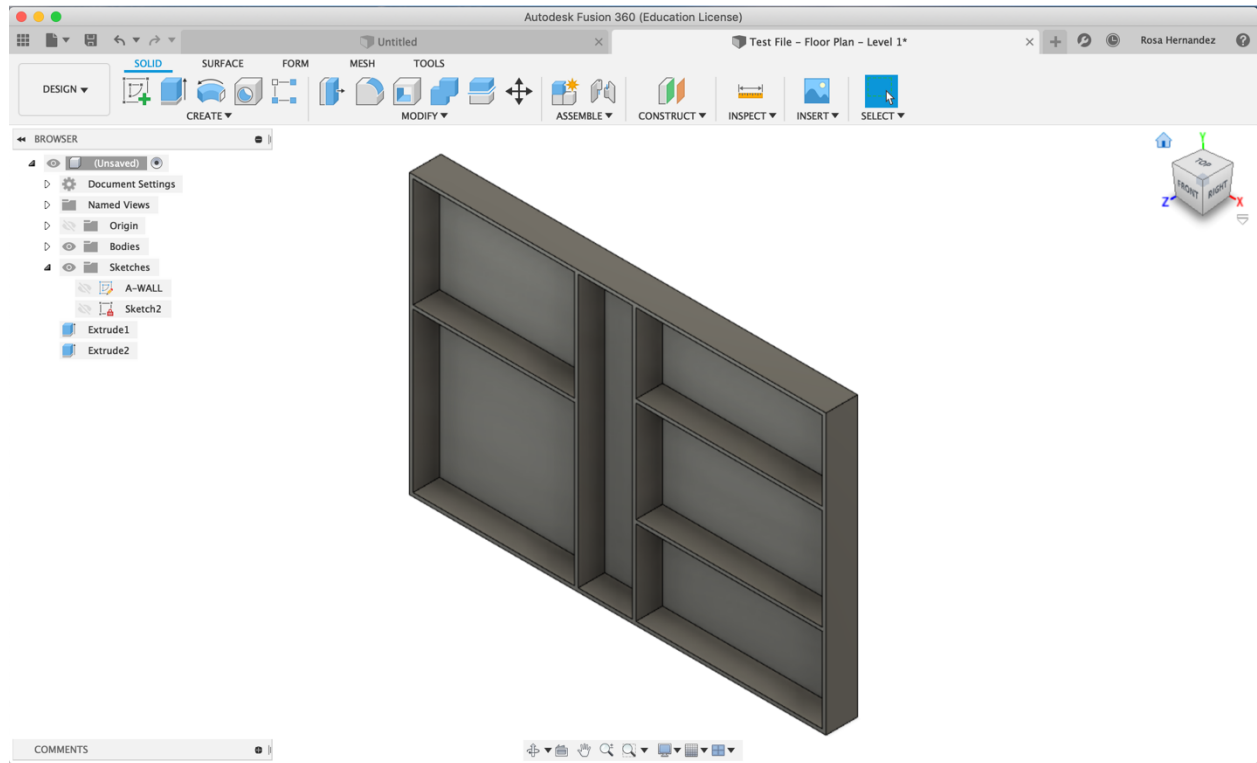


Figure 22: Finished Model

## Exporting as STL File

To export as a STL file, go to the Browser Tab. Expand the bodies section and right click on the body that corresponds to the floor plan. From the menu that appears select Save As STL (shown in figure 23).

In the Save As STL pop-up do not change any of the options just hit ok and save the file on the computer.

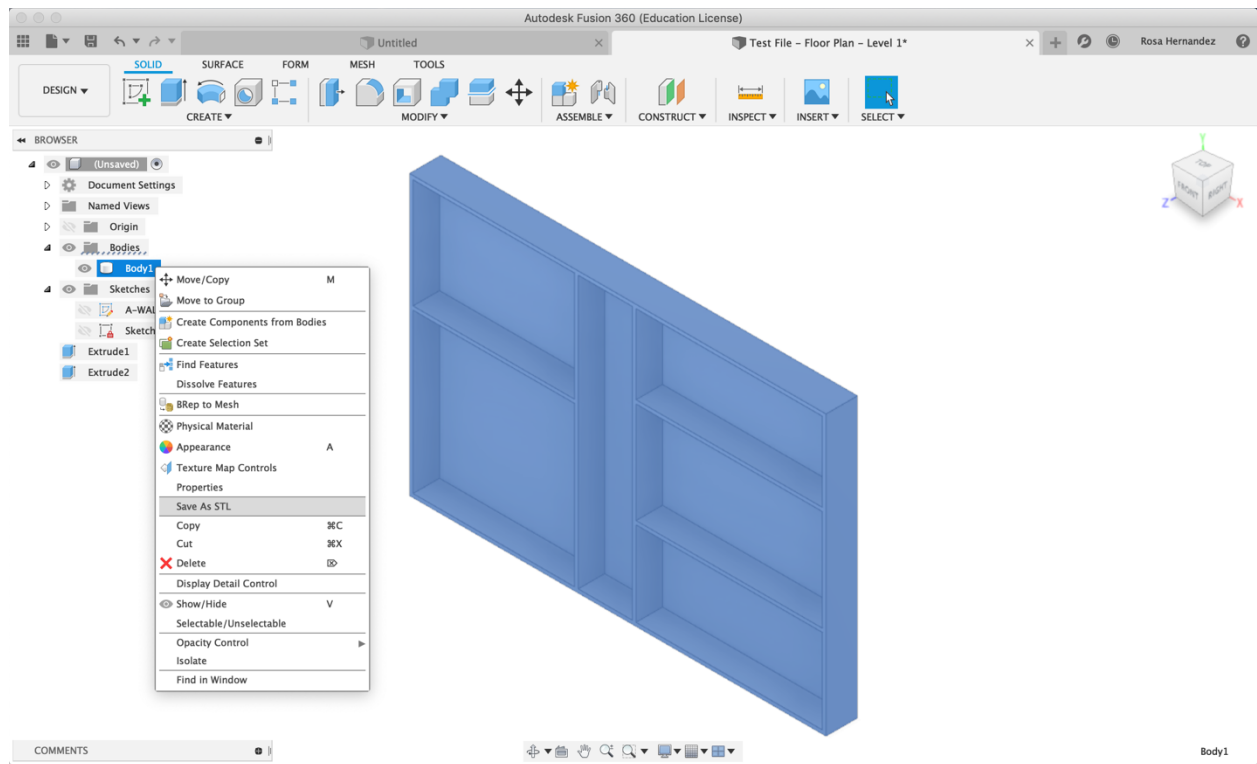


Figure 24: Finding the Save As STL Option

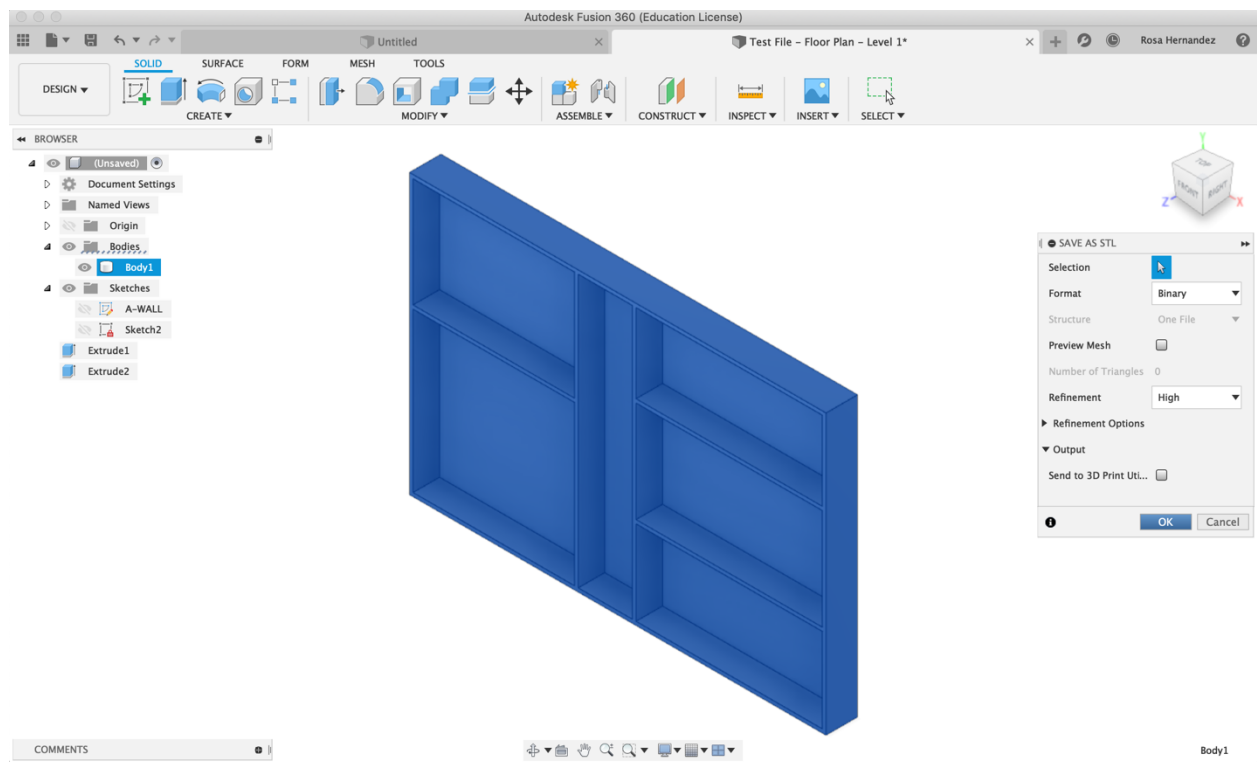


Figure 23: The Save as STL Pop-Up