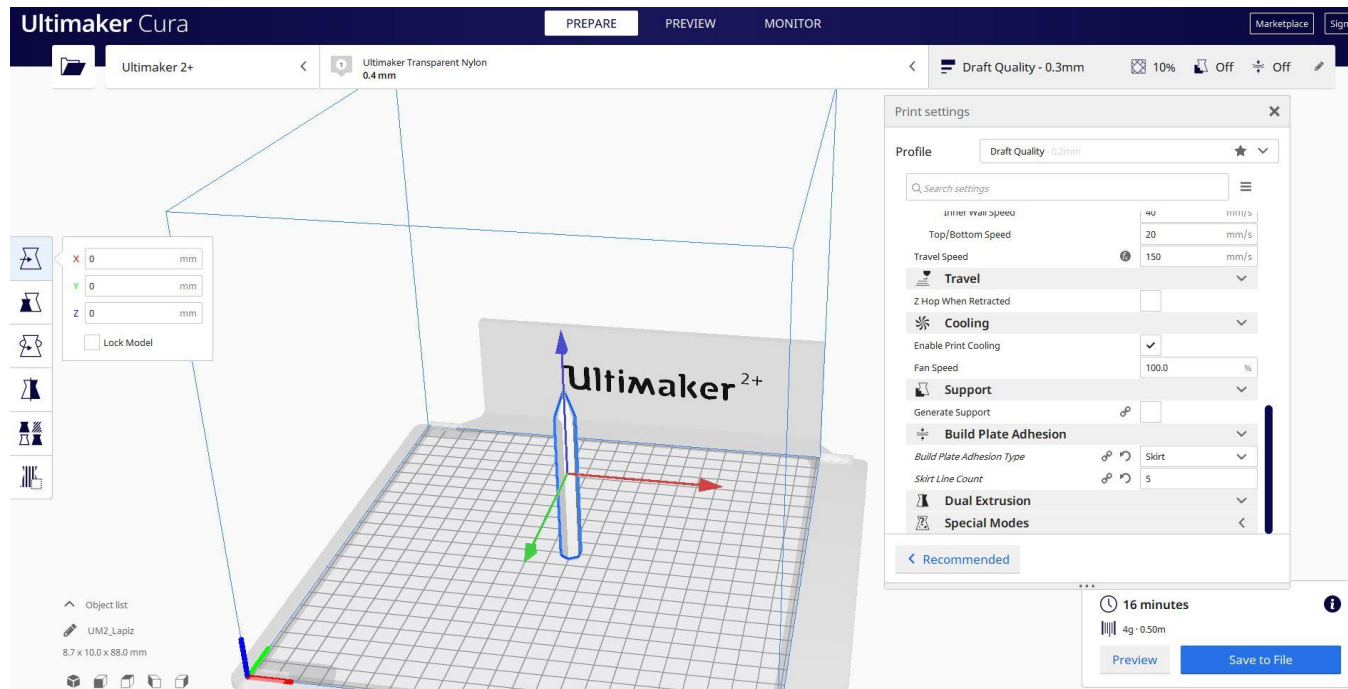




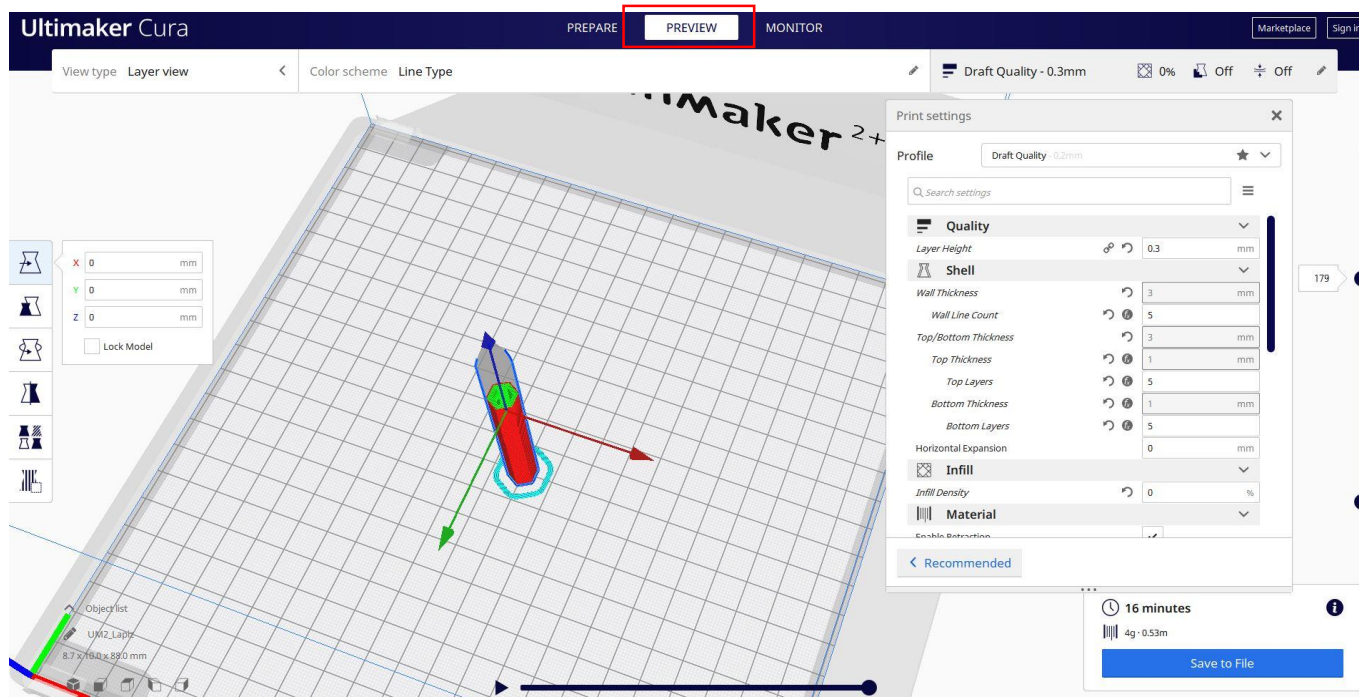
Document 9 – LAPIZ

1. Import the file on the Slicing Software ("Cura") and orient the piece in the best way to be printed.





2. I enter all the correct printing parameters (layer height, wall tickness, infill, support, speed, temperature, ...) and check for any problems from the "Preview"





3. At this point I can save the ".Gcode" file to send to the machine.

The screenshot displays the Ultimaker Cura software interface. The main window shows a 3D model of a printed part on a grid. The 'Print settings' panel is open on the right, showing the 'Draft Quality - 0.3mm' profile. The 'Save to File' dialog box is open in the center, showing the file path '210423_Cesar > File da stampare > Lápiz'. The file name is 'UM2_Lapiz' and the file type is 'G-code File (*.gcode)'. The 'Salva' button is highlighted with a red circle. The 'Print settings' panel shows various parameters for the print job, including Layer Height (0.3 mm), Wall Thickness (3 mm), and Infill Density (0 %). The 'Save to File' button is also visible in the bottom right corner of the software interface.

Parameter	Value	Unit
Layer Height	0.3	mm
Wall Thickness	3	mm
Wall Line Count	5	
Top/Bottom Thickness	3	mm
Top Thickness	1	mm
Top Layers	5	
Bottom Thickness	1	mm
Bottom Layers	5	
Horizontal Expansion	0	mm
Infill Density	0	%