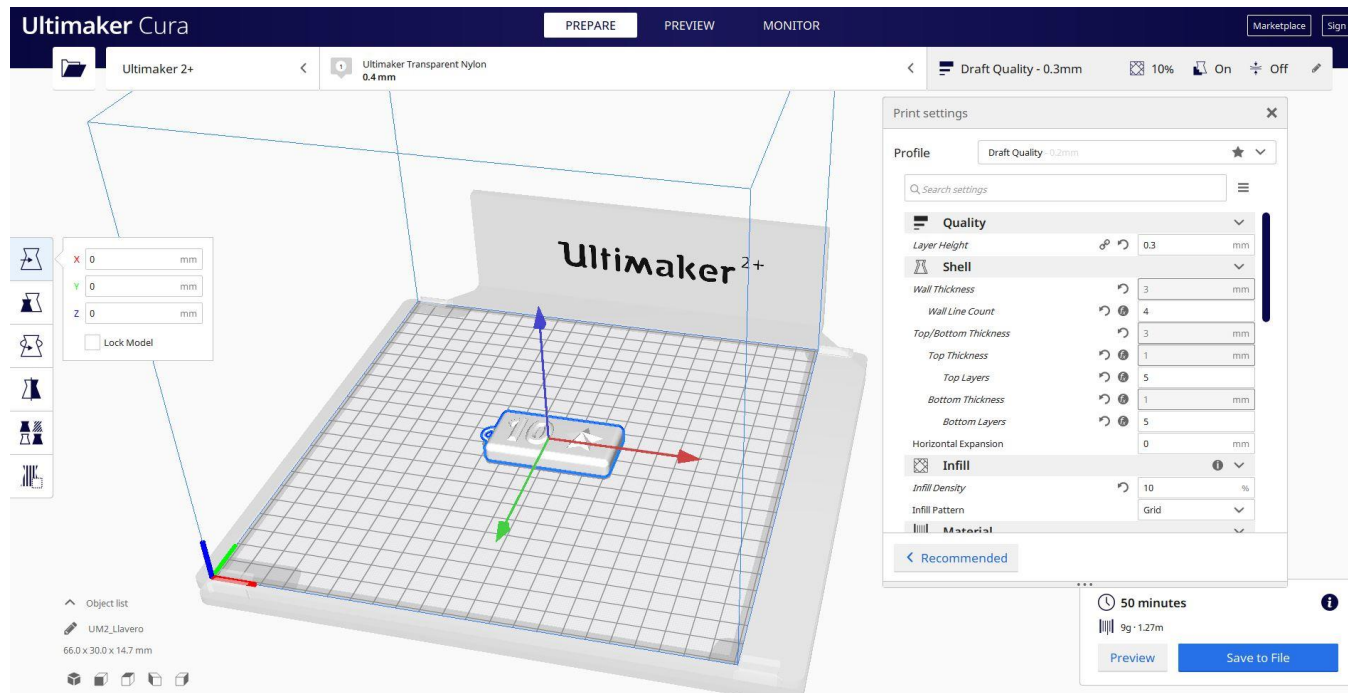




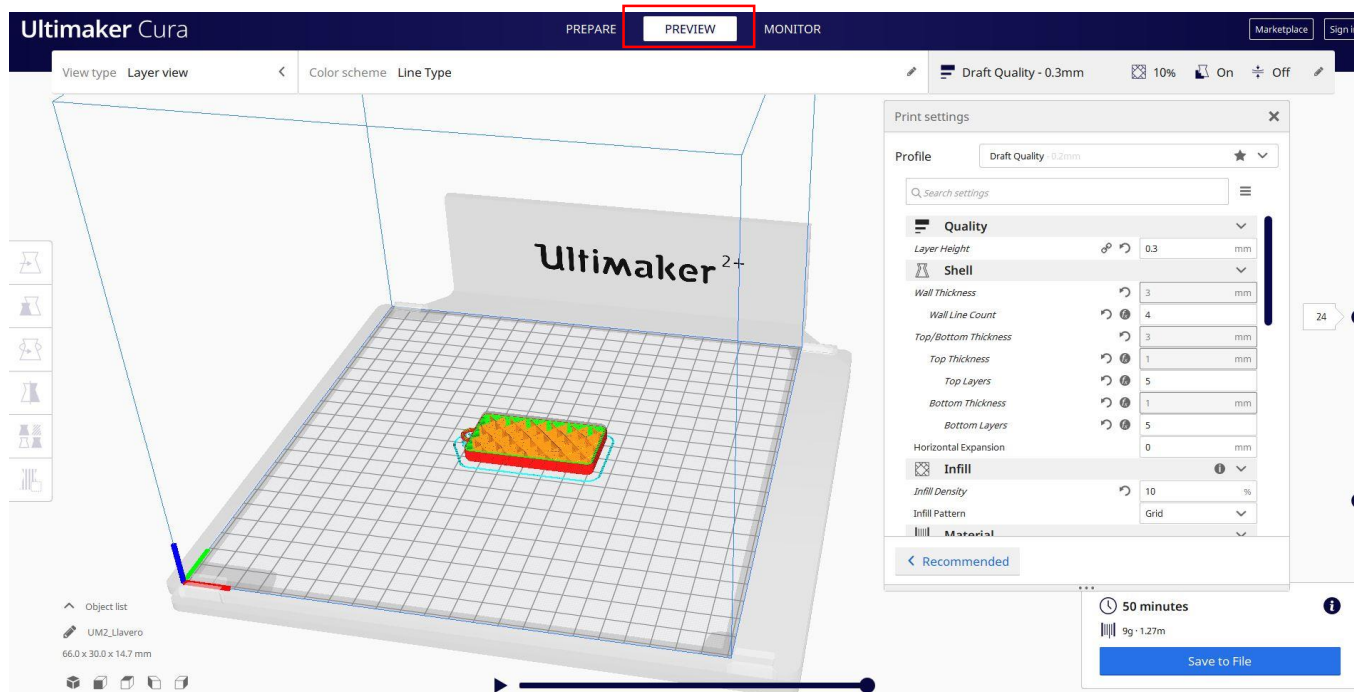
## Document 11 – LLAVERO

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 thickness, 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 shows the Ultimaker Cura interface. A 'Save to File' dialog box is open, displaying the file path '210423\_Cesar > File da stampare > Llavoro'. The 'Nome file' field contains 'UM2\_Llavoro' and the 'Salva come' field is set to 'G-code File (\*.gcode)'. The 'Salva' button is circled in red. In the background, the 'Print settings' panel is visible, showing the 'Draft Quality - 0.3mm' profile. The 'Quality' section includes settings for Layer Height (0.3 mm), Shell (Wall Thickness: 3 mm, Wall Line Count: 4, Top/Bottom Thickness: 3 mm), Top Thickness (1 mm), Top Layers (5), Bottom Thickness (1 mm), and Bottom Layers (5). The 'Infill' section shows Infill Density (10%) and Infill Pattern (Grid). A '50 minutes' timer is also visible in the bottom right corner.