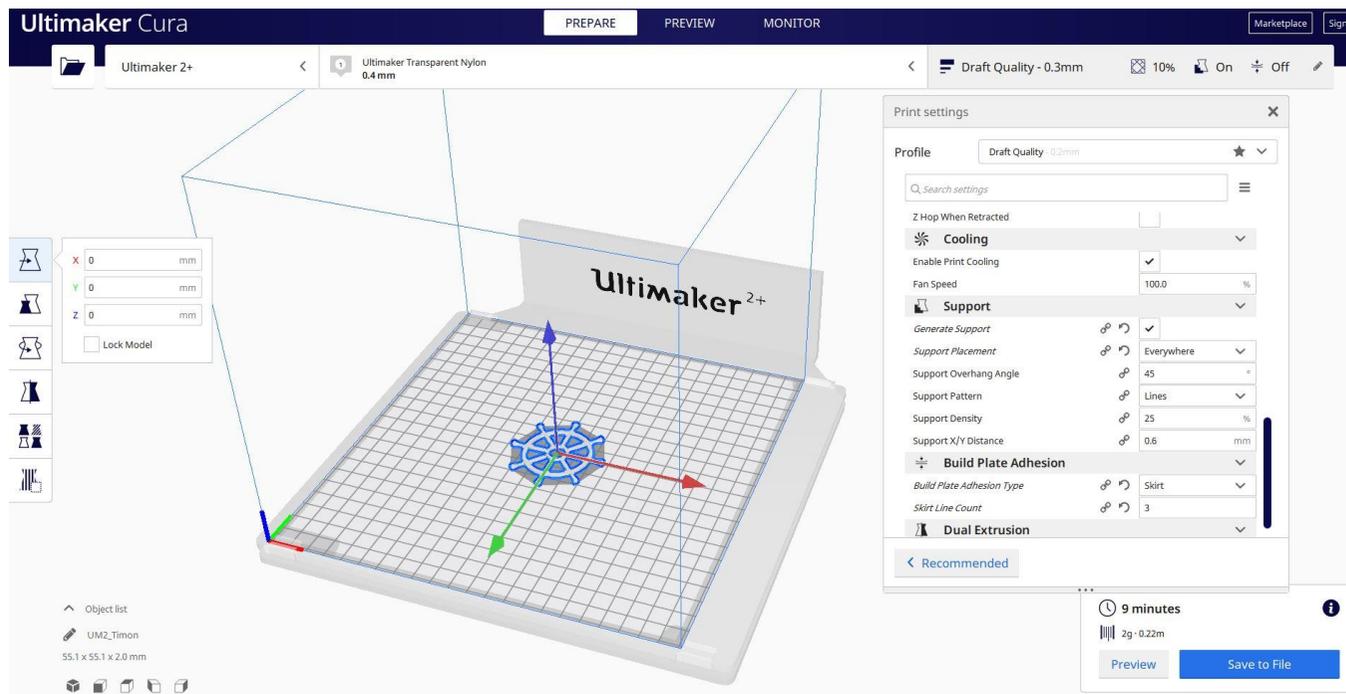




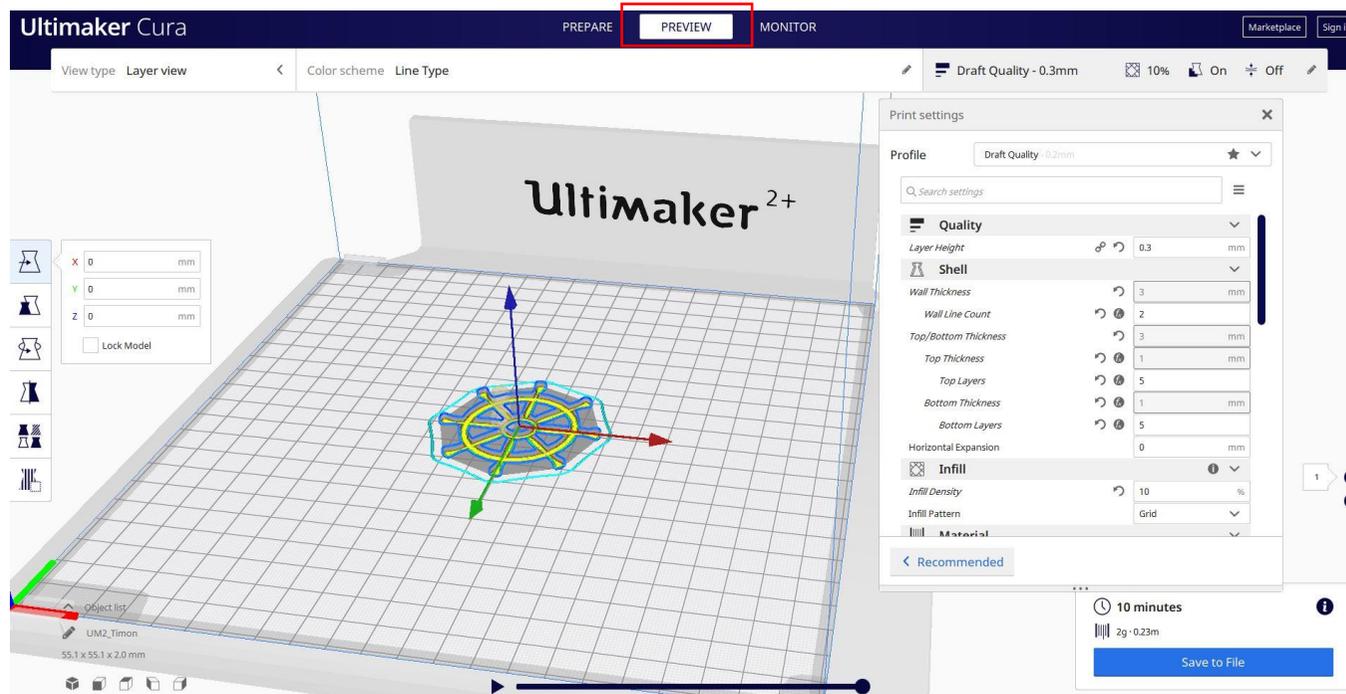
## Document 19 - TIMON

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 displays the Ultimaker Cura software interface. The main window shows a 3D model of a printed part on a grid. A 'Save to File' dialog box is open, showing the file name 'UM2\_Timon' and the file type 'G-code File (\*.gcode)'. The 'Salva' button is circled in red. The 'Print settings' panel is visible on the right, showing various parameters such as Layer Height, Shell, Top/Bottom Thickness, Top Layers, Bottom Thickness, Bottom Layers, Horizontal Expansion, Infill, and Material. The 'Draft Quality - 0.3mm' profile is selected. The bottom right corner shows a '10 minutes' print time and a 'Save to File' button.