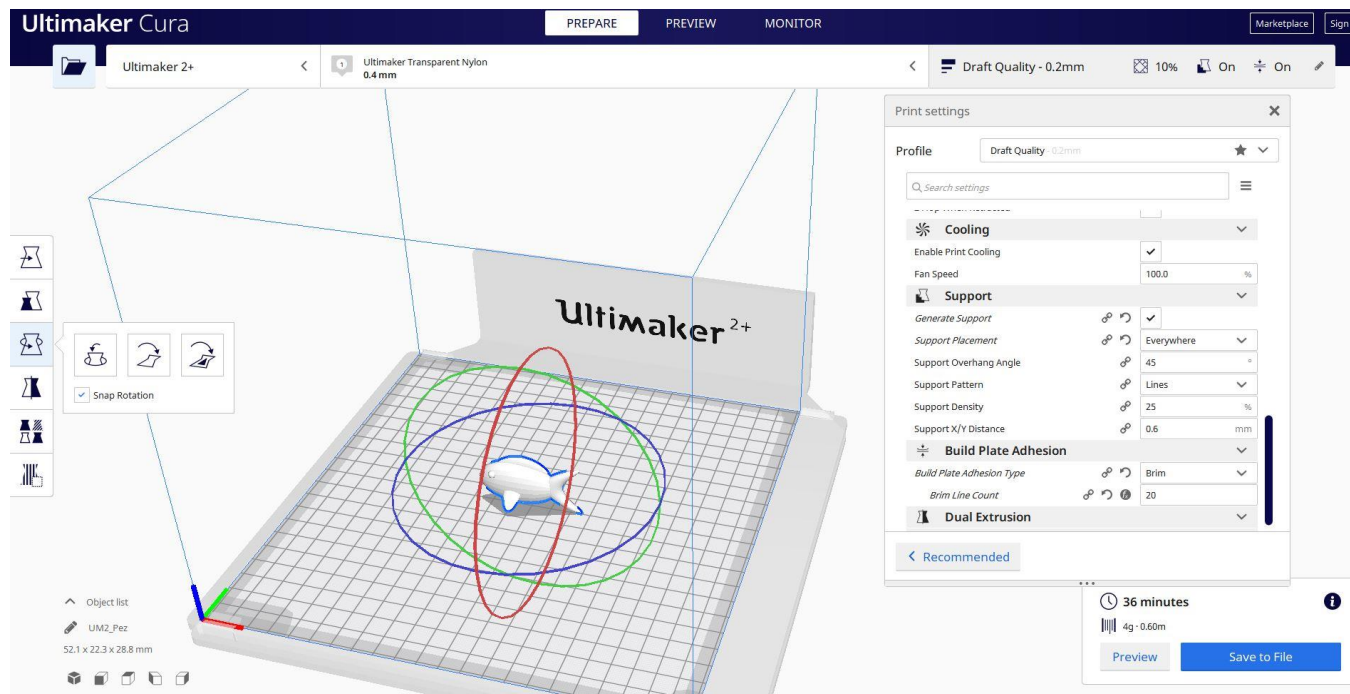




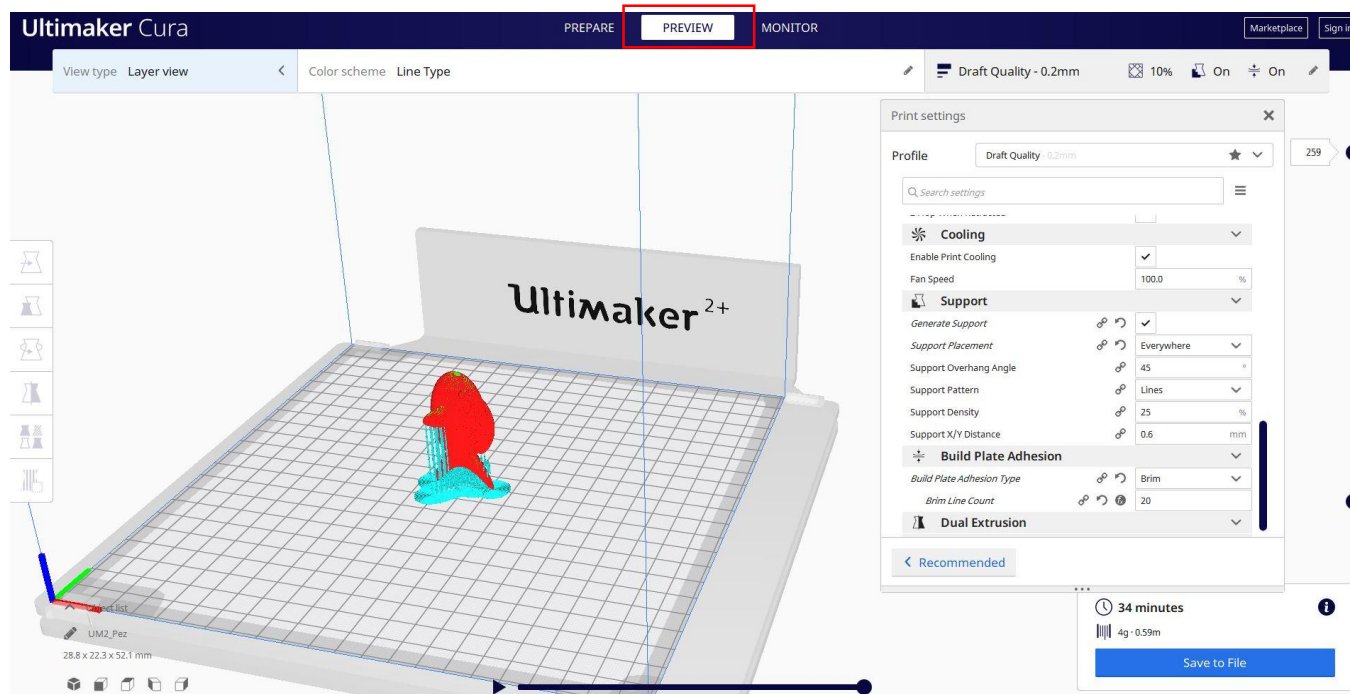
Document 16 - PEZ

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. At the top, the 'PREPARE' tab is active. A 'Save to File' dialog box is open, showing the file path '210423_Cesar > File da stampare > Pez'. The file name is 'UM2_Pez' and the file type is 'G-code file (*.gcode)'. The 'Salva' button is circled in red. To the right, the 'Print settings' panel is visible, showing the 'Draft Quality - 0.2mm' profile. The 'Cooling' section has 'Enable Print Cooling' checked and 'Fan Speed' at 100.0%. The 'Support' section has 'Generate Support' checked, 'Support Placement' set to 'Everywhere', 'Support Overhang Angle' at 45, 'Support Pattern' set to 'Lines', and 'Support Density' at 25%. The 'Build Plate Adhesion' section has 'Build Plate Adhesion Type' set to 'Brim' and 'Brim Line Count' at 20. The 'Dual Extrusion' section is also visible. At the bottom right, a summary box shows a print time of 34 minutes, a weight of 4g, and a size of 0.59m. The 'Save to File' button is highlighted in blue.