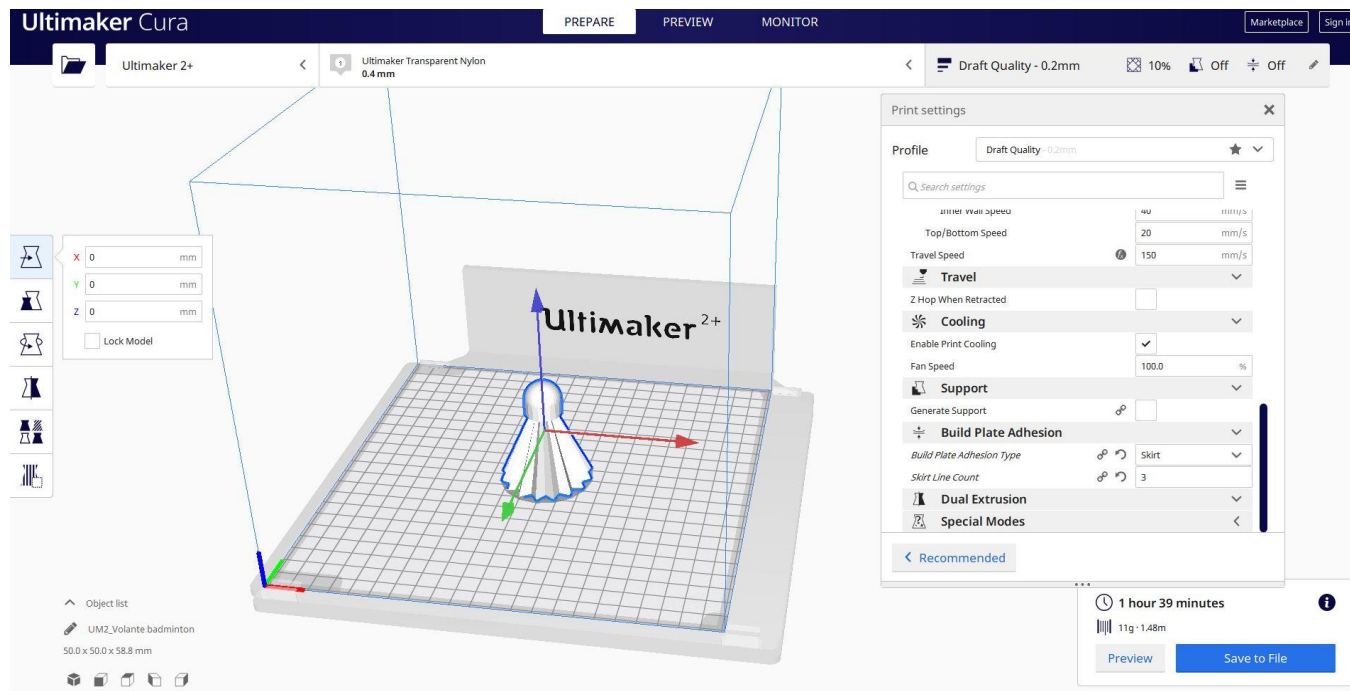




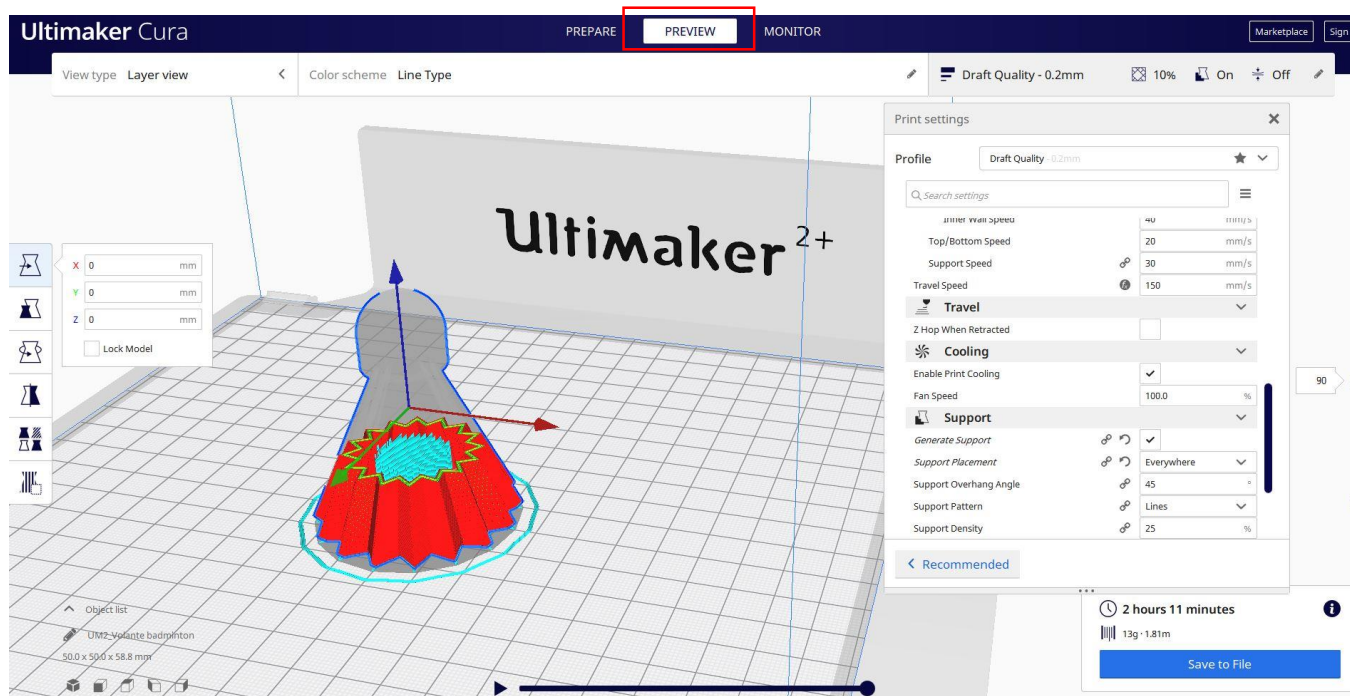
Document 20 - VOLANTE BADMINTON

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_Volante badminton' and the file type 'G-code File (*.gcode)'. The 'Salva' button is circled in red. To the right, the 'Print settings' panel is visible, showing various parameters for the 'Draft Quality - 0.2mm' profile, such as Top/Bottom Speed (20 mm/s), Support Speed (30 mm/s), and Travel Speed (150 mm/s). The 'Support' section is expanded, showing options like 'Generate Support' (checked), 'Support Placement' (Everywhere), and 'Support Density' (25%). The bottom right corner shows a 'Print' button with a timer of 2 hours 11 minutes and a 'Save to File' button.