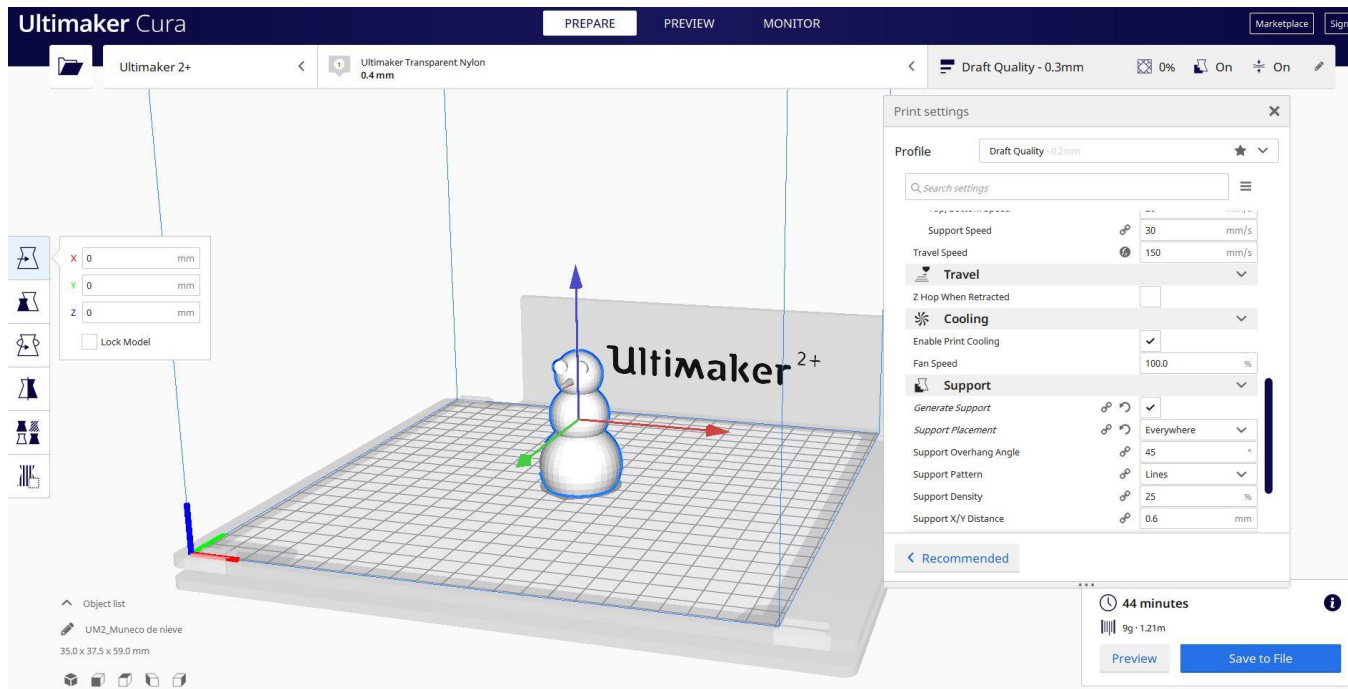




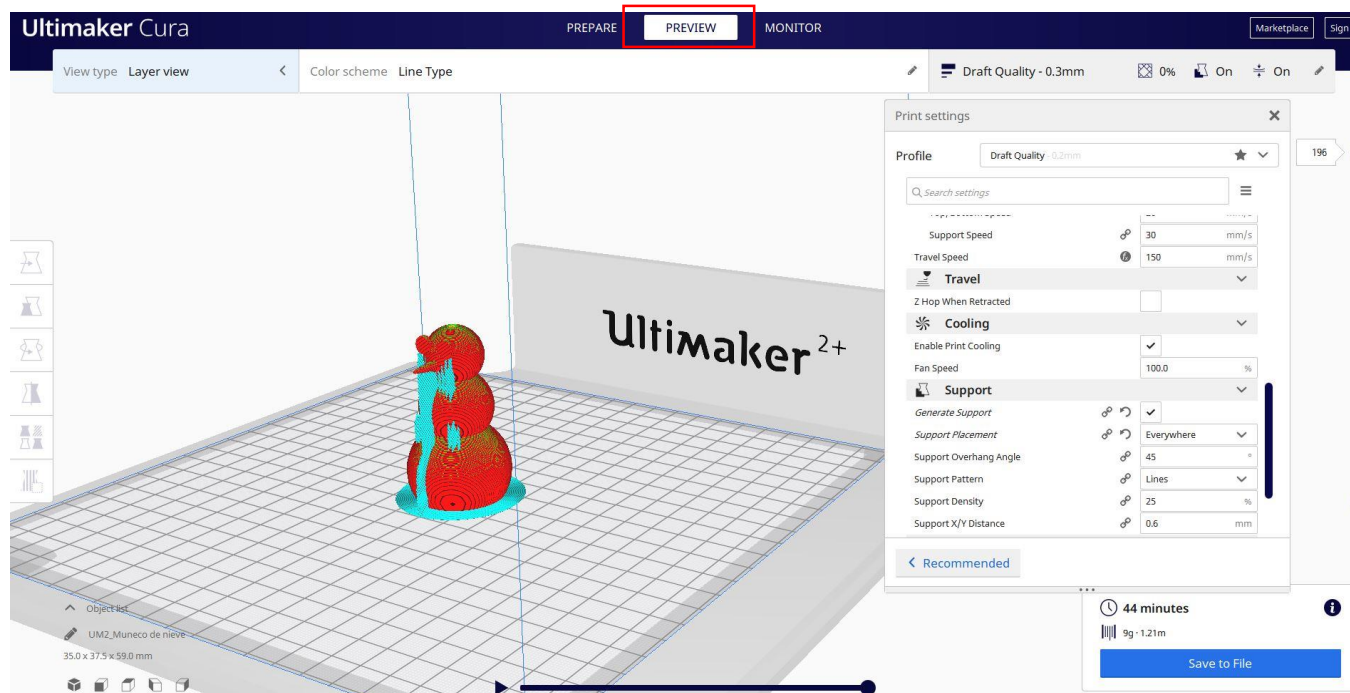
## Document 13 – MUNECO DE NIEVE

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. A 'Save to File' dialog box is open, showing the file path 'File da stampare > Muñeco de nieve'. The file name is 'UM2\_Muneco de nieve' and the file type is 'G-code File (\*.gcode)'. The 'Salva' button is circled in red. The background shows the print settings for 'Draft Quality - 0.3mm' with a 44-minute print time. The print settings panel includes sections for Profile, Travel, Cooling, and Support, with various parameters like Support Speed, Travel Speed, Fan Speed, and Support Density.

Section	Parameter	Value	Unit
Profile	Support Speed	30	mm/s
	Travel Speed	150	mm/s
Cooling	Enable Print Cooling	<input checked="" type="checkbox"/>	
	Fan Speed	100.0	%
Support	Generate Support	<input checked="" type="checkbox"/>	
	Support Placement	Everywhere	
	Support Overhang Angle	45	°
	Support Pattern	Lines	
	Support Density	25	%
Support X/Y Distance	0.6	mm	