3D Printing Checklist

1. Obtain 3D object files to print
2. Check object(s) for general printability (Check: correct file format for slicer, manifold and error checked)
3. Determine the required size/scale of object(s). Important for STL files (Effects: amount of print time and material used)
4. Insure the object(s) fit on the build plate. (Effects: additional print sessions, cutting object into multiple parts, placement and orientation)
5. Determine the required strength of the object(s). (Effects: Filament type, orientation, shells, and infill)
6. Determine the appropriate print speed needed for the print job [draft, iteration, final] (Effects: filament type, layer height, shells, infill and size)
7. Select the appropriate filament for printing. (Check: filament has been properly stored, is in good shape and appropriate for the printer)
8. Determine best orientation of the object(s). (Effects: supports, strength, object detail)
9. Import object(s) into slicer and do a preliminary slice to ensure the slicer has no initial issues with the object(s). (view: object slices within the slicer if capable).
10. Review if the object(s) require modification/redesign due to the above factors.
11. Check and set the appropriate 3D printer profile in the slicer.
12. Check and set the appropriate filament profile in the slicer. (If new: calibrate filament and make a new profile)
13. Determine the appropriate layer height for the object(s). (Effects: print time and vertical resolution)
14. Determine the appropriate shells/wall thickness for the object(s). (Effects: print time and strength)
15. Determine the appropriate infill for the object(s). Both pattern and density (Effects: print time and strength)
16. Determine if the object(s) require support. Add auto and check, then adjust support as needed. (alt: add manual support)
17. Determine if the object(s) require extra bed adhesion (Select: Brim, Raft, bed additive).
18. Determine any additional advanced settings that are required for this print.
19. Set any changes to slicer settings and record the information.
20. Slice and inspect the final slice (view: object slices within the slicer if capable).
21. Inspect printer for any physical issues. (Check: damage, free movement of axis, loose wires or fasteners, foreign object(s))
22. If required, inspect ventilation functionality and replace filters if needed.
23. Start the printer and ensure its correct power on and functionality. (Check: Fans running, if appealable lights and display on and functioning)
24. Tram the print bed if out of true manually or ensure auto tramming features are enabled.
25. Open filament container and inspect filament and spool (Check: filament is in good condition and tightly wound).
26. Heat and Load filament into printer (Ensure: filament extrude freely from hot end and can feed correctly from spool.)
27. Check and remove debris from print bed and if needed, clean the print surface.
28. Add any bed adhesion additive if required.
29. If required, warm up the printer/enclosure.
30. Note print environment temperature and humidity. (Effects: Environment qualities can affect print)
31. Send the file to printer and initiate the print.
32. Inspect the first printed layers to ensure these are laid down correctly. Stop the print process if there is a failure.
33. Inspect print as often as needed to ensure correct operation. Note any issues and note the sound/functionality of printer and quality of print.
34. Allow the print to finish and the printer to cool as needed for removal.
35. Remove print (use tools if needed). Note any issues with removal and remember safety.
36. When the printer is at room temperature, turn it off.
37. Move print to the processing area and inspect. Note any issues; any issues may result in changed slicer settings.
38. Record successful and unsuccessful print information for later use.
39. Finish print as needed.