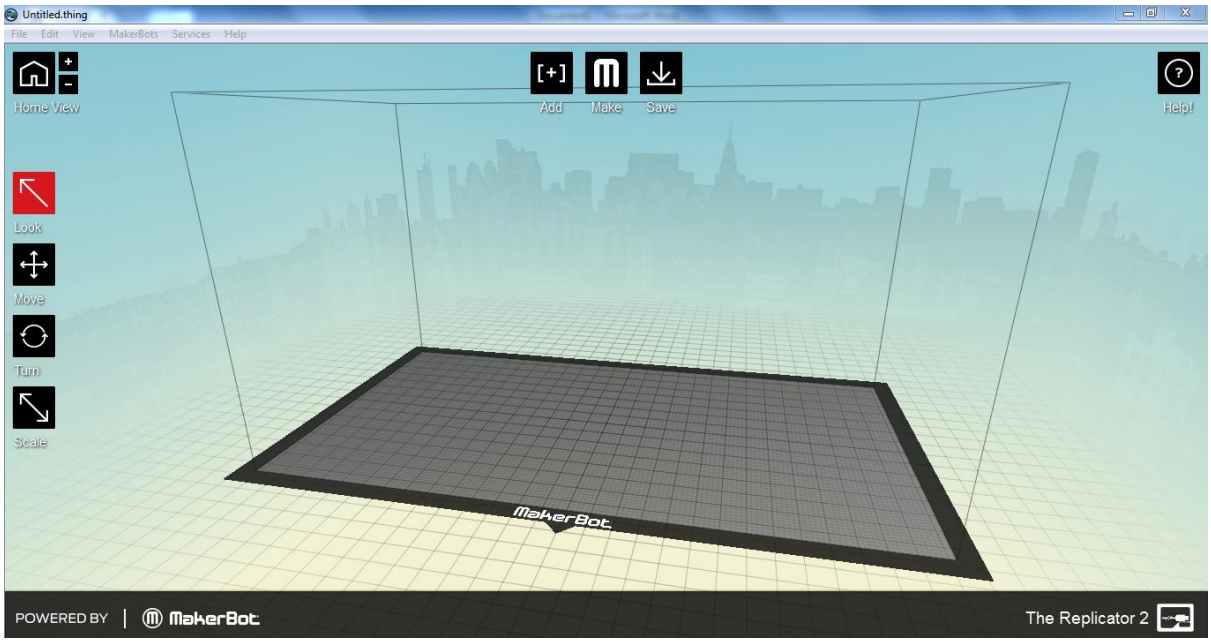
3D printing from Makerware

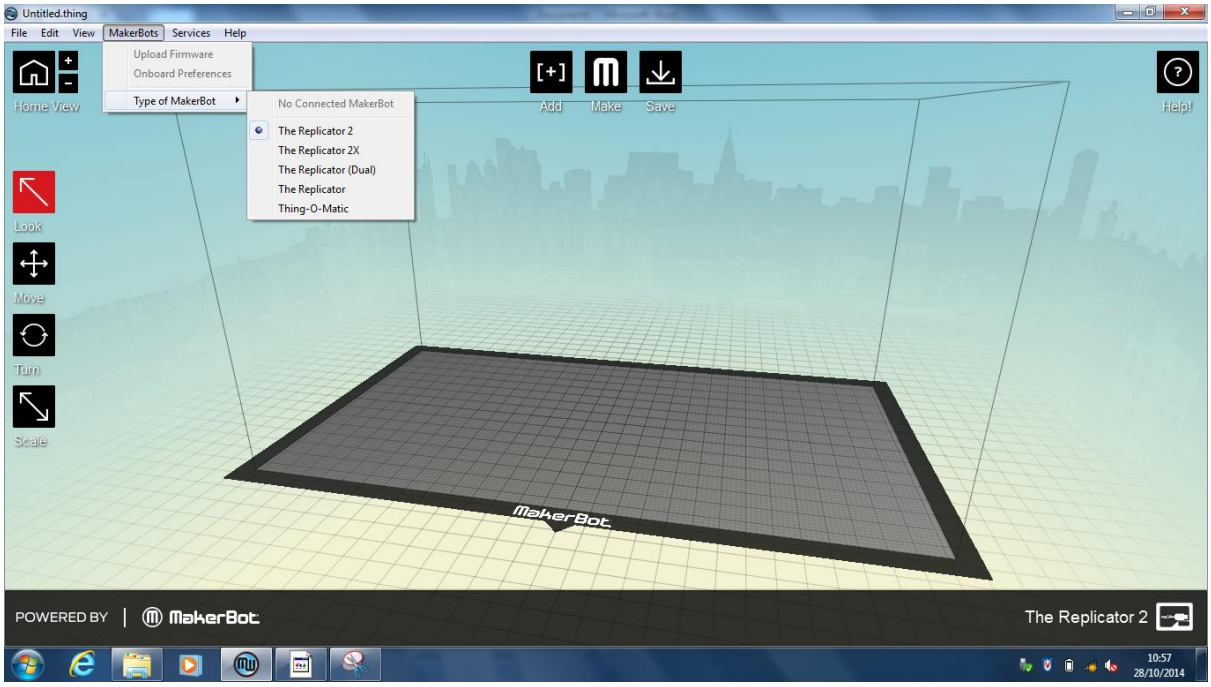
These are guidelines I use for printing PLA on a makerbot R2 and using an SD card to print from.

Save from solidworks as an STL file with the options needs…. – look at website link

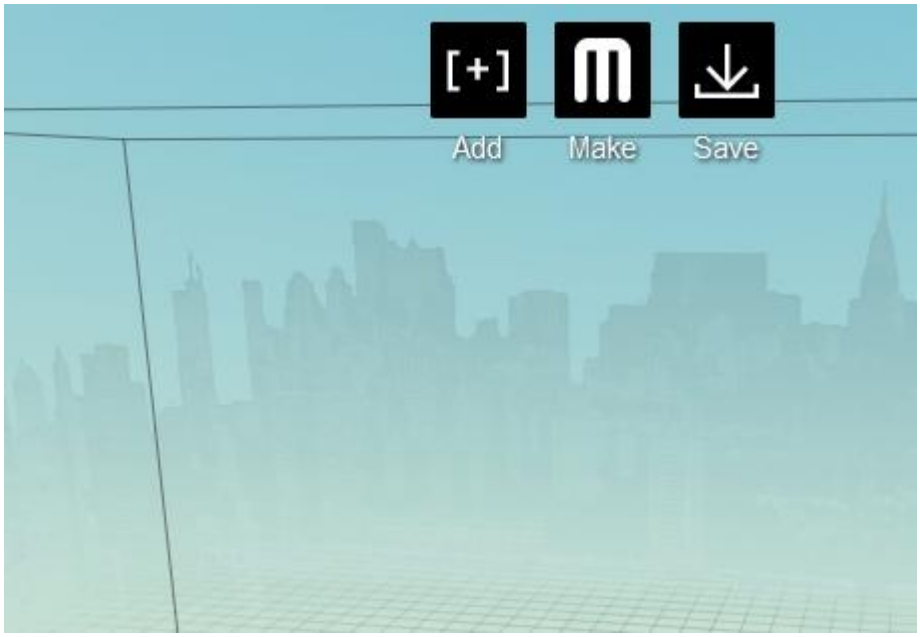
Open Makerware



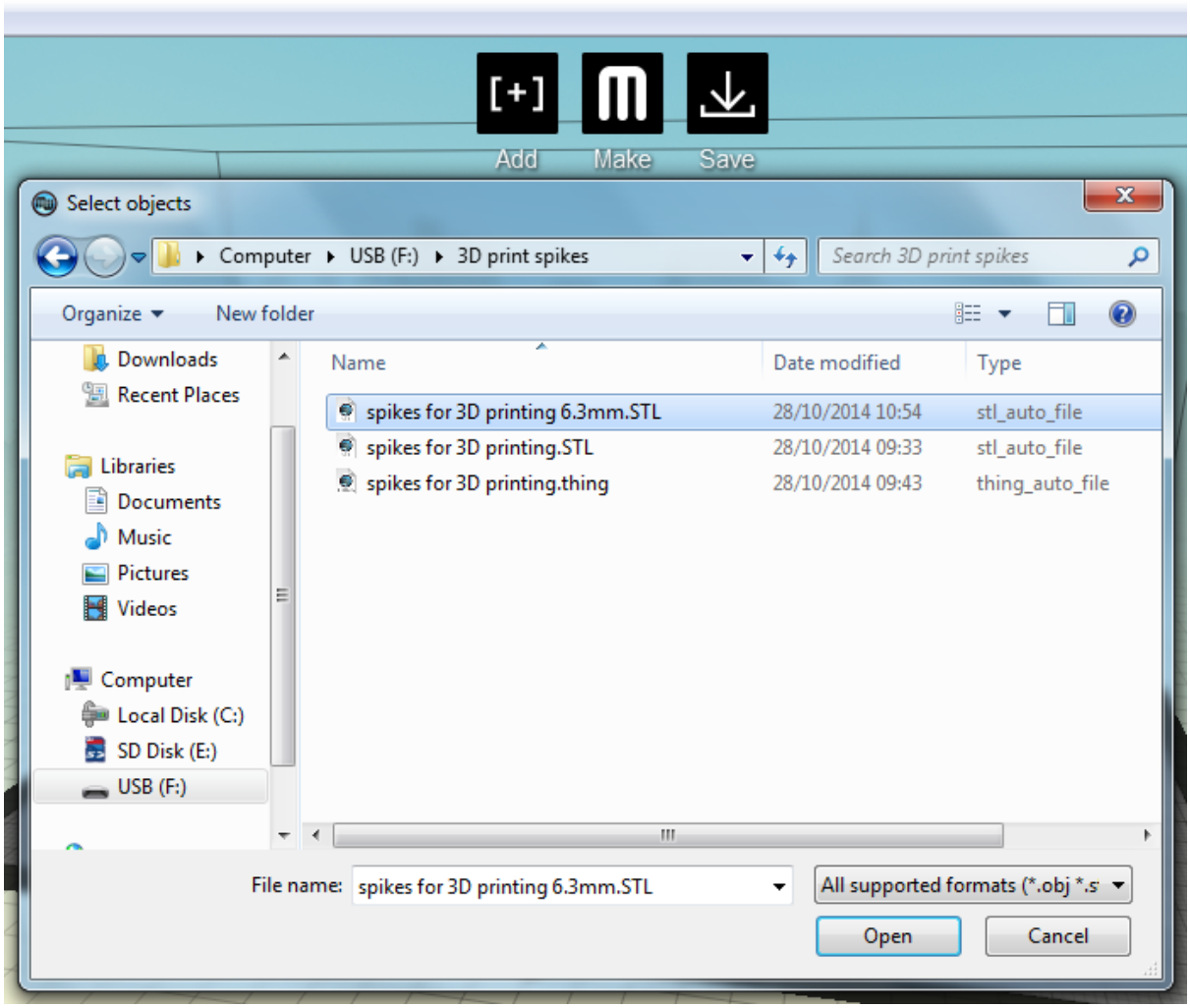
Select what model of makerbot you are going to print onto



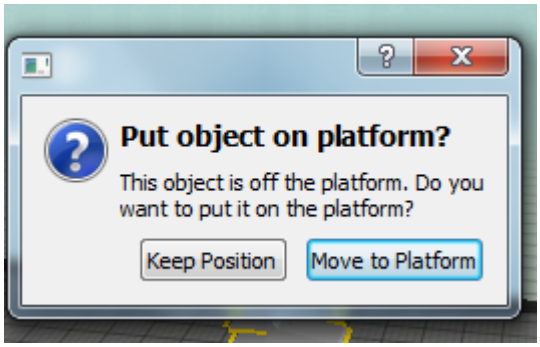
Add (import) your part (stl model) into makerware by clicking on the add icon in makerware



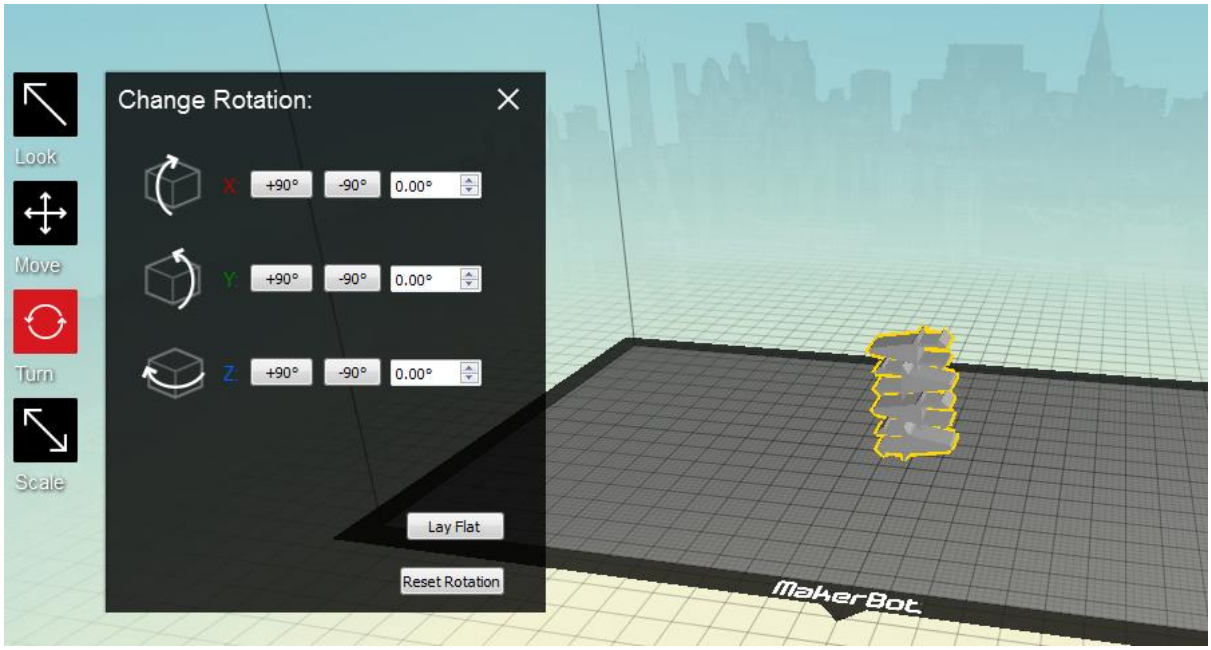
Navigate to the file



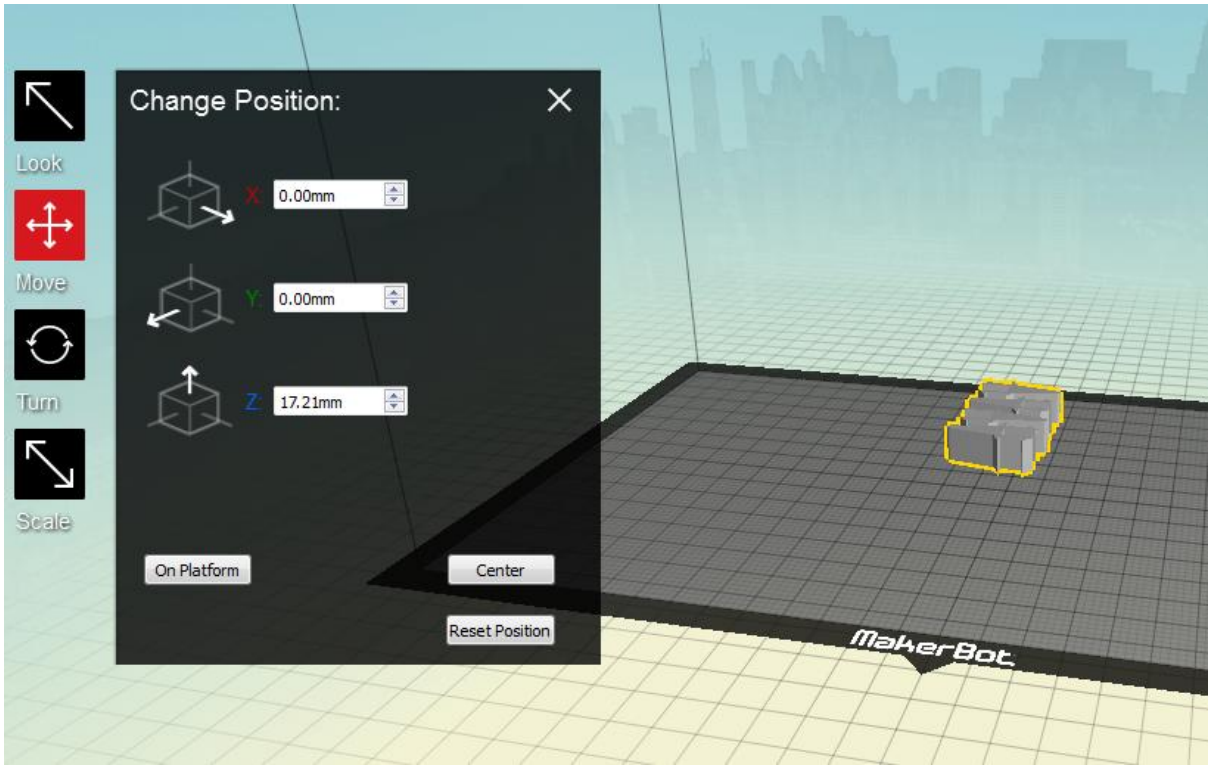
Move the part to the platform



When the object is selected double click on the turn icon so you can rotate the part so it has miminal support material and is positioned for the best printing outcome

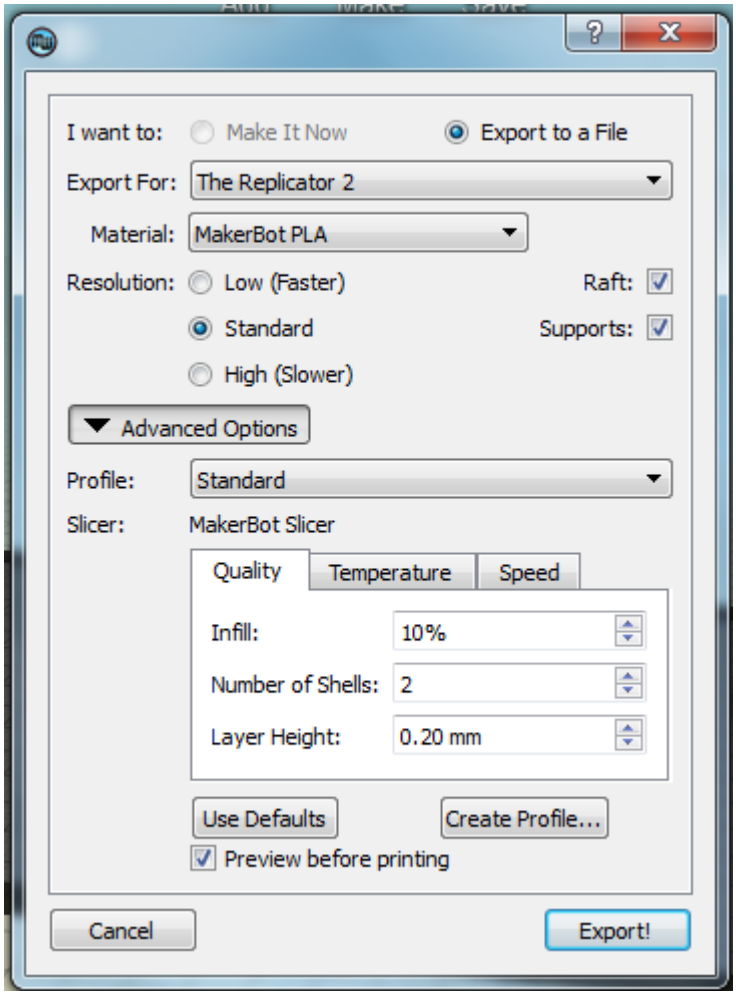


After you've rotated it. Move the part to sit on the platform so the printer doesn't think the object is in mid air

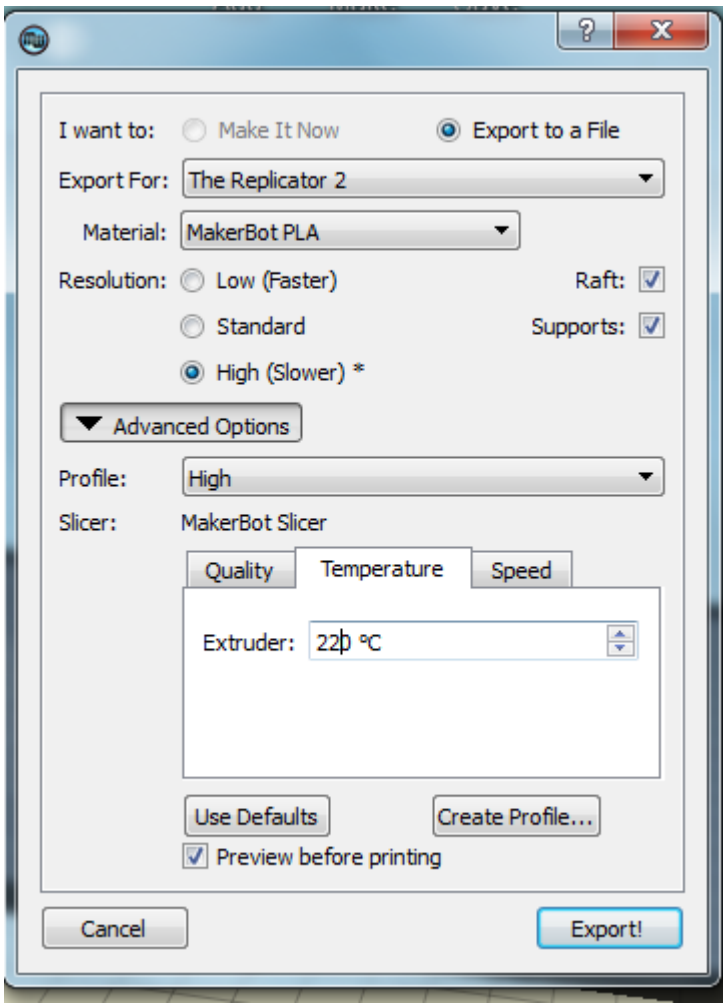


Next we're going to export the file for printing. Select 'Make' icon

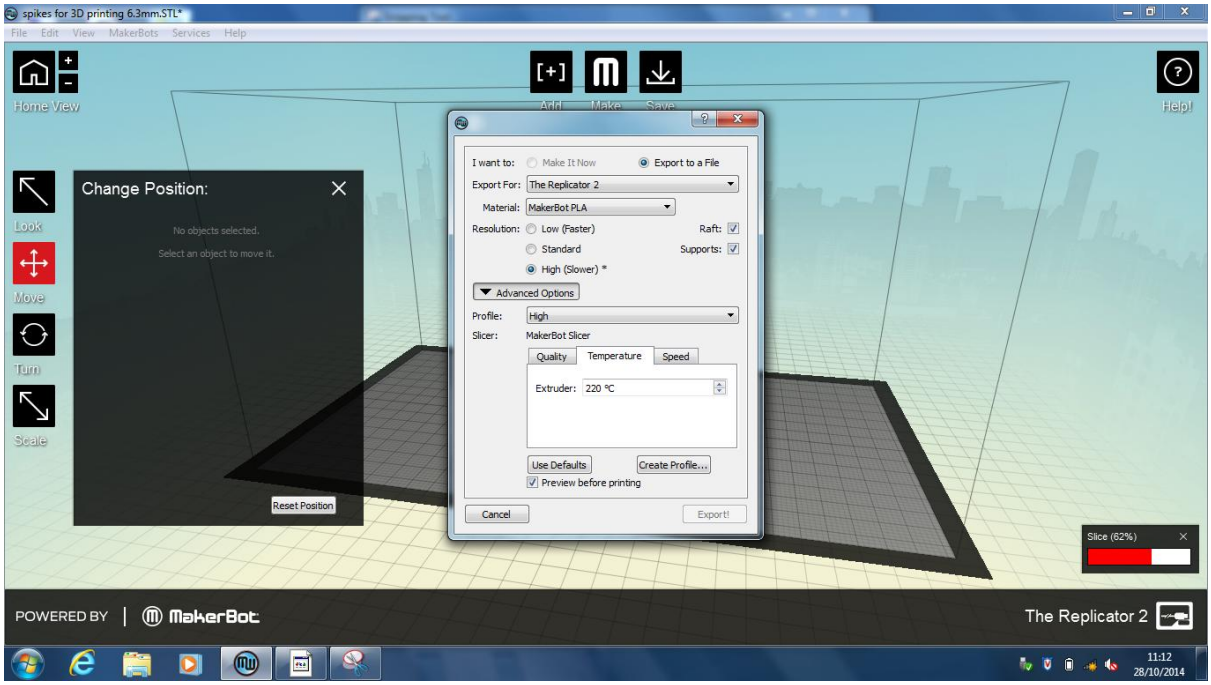
Depending on the quality you select different options but make sure you select print preview so you can see how long it will take and get an overview of how it will print


I want to:
Make It Now
Export to a File
Export For:
The Replicator 2
Material:
MakerBot PLA
Resolution:
Low (Faster)
Standard
High (Slower)
Raft:
Supports:
Advanced Options
Profile:
Standard
Slicer:
MakerBot Slicer
Quality
Temperature
Speed
Infill:
10%
Number of Shells:
2
Layer Height:
0.20 mm
Use Defaults
Create Profile...
Preview before printing
Cancel
Export!

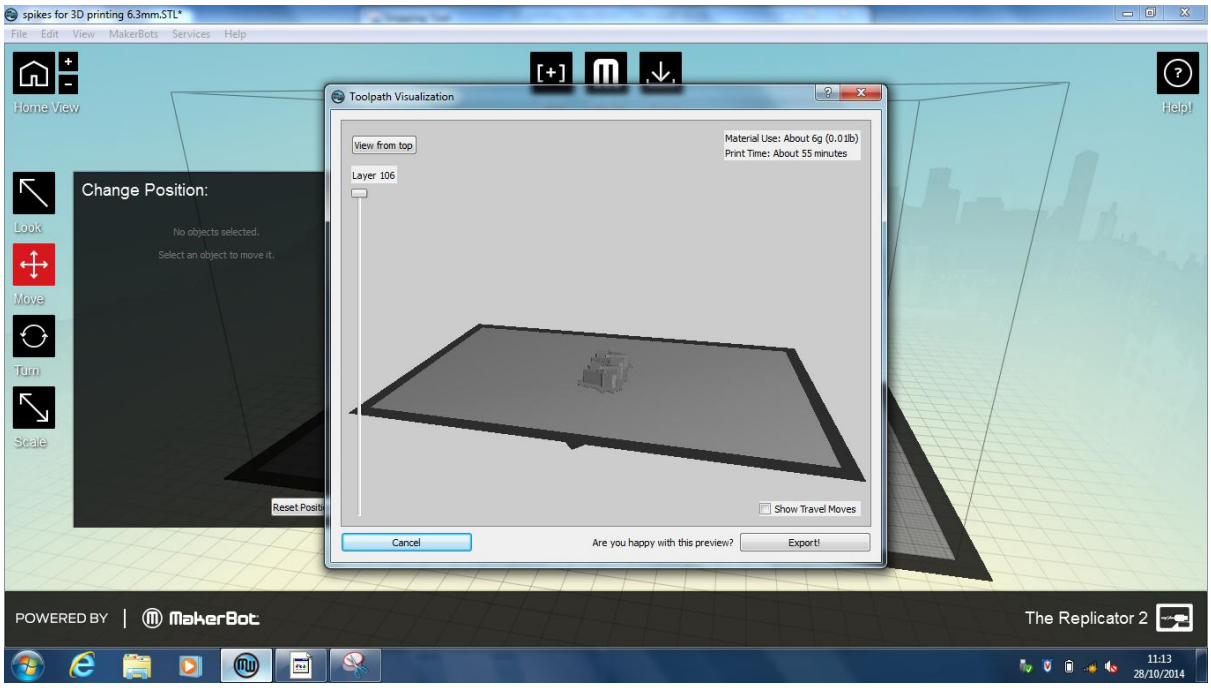
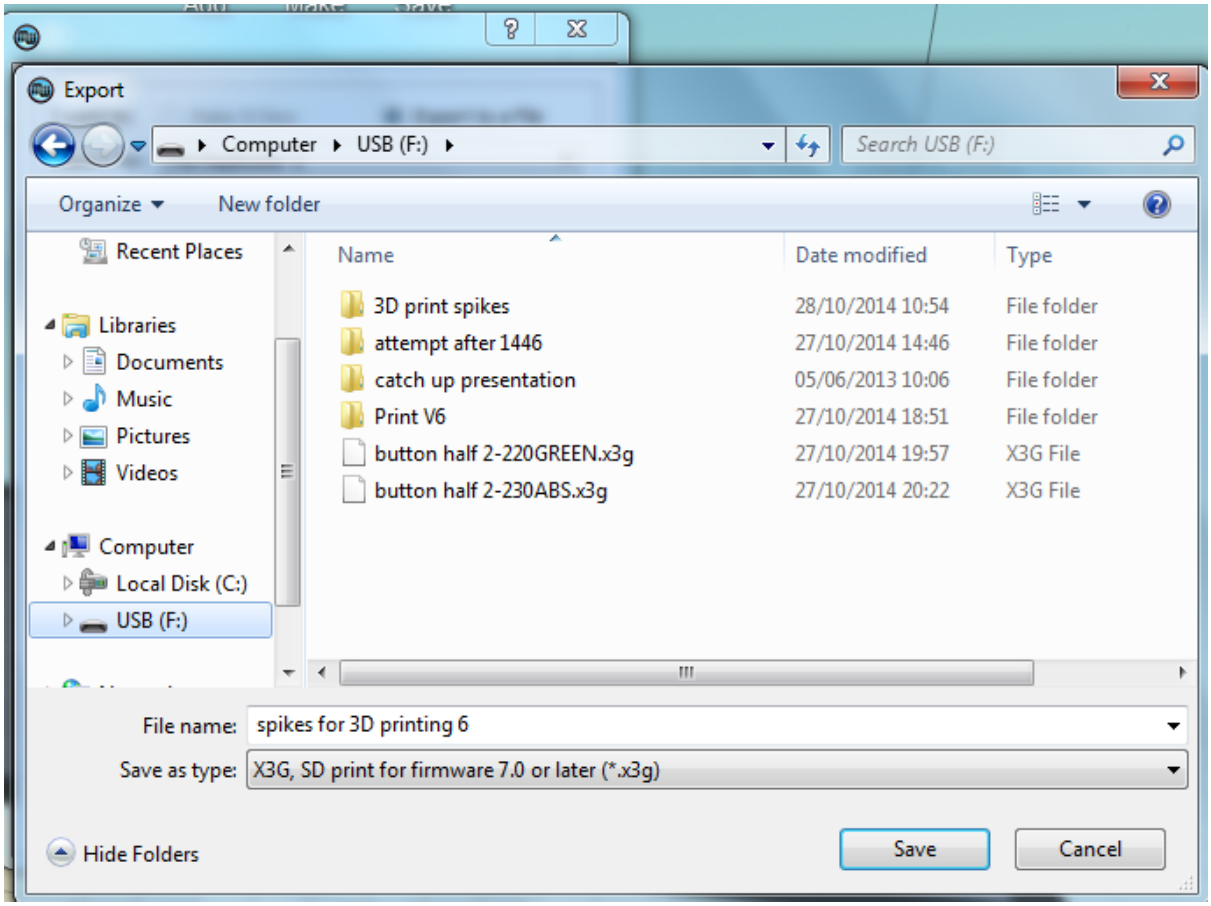
Check the temperature. For PLA the temperature can vary even from different filaments of PLA. We use 220 to print green PLA and this works nicely. This might be too hot for most PLA filaments but it works for out free PLA we have in the lab.



Select export. When you do you'll see a slicing exporting loading bar processing the part for printing



Print preview



Then select export and save the file to the SD card to put into the printer