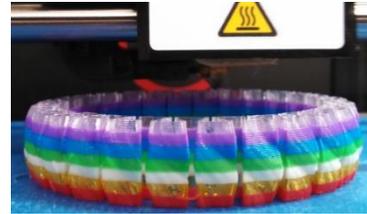# 3D Printing Checklist   revised: 5/22/15

☐ **Choose your Color:**
The library has red, white, blue, green, purple, translucent yellow, and translucent colorless colors. Consult a librarian to check if additional colors have been added. In addition, you can specially request manual horizontal banding in the colors of your choice.

☐ **Need Supports? (Have overhangs steeper than 45°?):**
If your model has overhangs, supports may be needed to stop printed plastic that is still hot from drooping. Beware that it may be difficult to snap off the supports without snapping off delicate parts of your object, and it will add some plastic to your total cost. Some solutions:

*Rotate the model:*
The easiest solution if it is possible for your object

*Using a sloping overhang:*
The 3D printer can handle slopes of 45° or steeper.
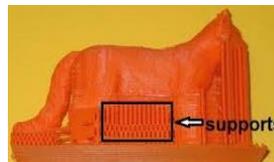
*Auto-generated Supports:*
Removable supports can be easily added using Makerbot Desktop by the librarian.

*Custom Supports:*
Making your own supports using a 3D modeler.

Also, the 3D printer sometimes prints in a way called "bridging," when it has to build a horizontal surface between 2 points. This is more successful when the distance between the 2 points is smaller, if too large the underside of the "bridge" may be stringy without supports.

**1.25mm**   **25mm**   **50mm**

☐ **Readable file format**:
The library accepts 3D modeling files in .stl, .obj., or .thing format given in an email or memory stick.

☐ **Scale:**
Check that your object is your desired size. The library's printer can only print things that fit within a 6" wide x 6" high x 11" long space. Keep your object small to save money and time.

☐ **Tolerances (gaps):**
The 3D printer has ~.4mm variance, meaning gaps should be at least ~0.4mm wide. Otherwise, for example, a peg won't fit in a hole, or 2 gears may fuse together.

≥ 0.4mm
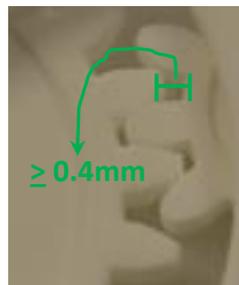
☐ **Check the Print Preview:**
Examine your model and check for anything that doesn't look right, such as a hole in a wall that was too thin for the program to recognize. Minus the effects of gravity, what you see in a print preview is what you'll get. Also check the weight (aka. cost) and build time estimates.
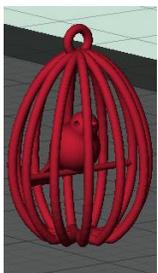
1. Import the .stl or .obj of your 3D model into Makerbot Desktop, which you can download at makerbot.com/desktop.
2. Go to SETTINGS. Selecting a lower Resolution will let it print much faster, but it will be less smooth. Also, check the box for rafts or supports if needed. Click SAVE SETTINGS.
3. Go to EXPORT PRINT FILE. Makerbot Desktop will slice your 3D model into the horizontal layers that the 3D printer actually prints.
4. Click on PRINT PREVIEW instead of "Export Now" to see the print preview.

☐ **Minimum thickness (~0.4mm):**
If any part of your model is thinner than ~0.4mm, it will probably not print that part. Can be checked by using Makerbot Desktop's preview option.
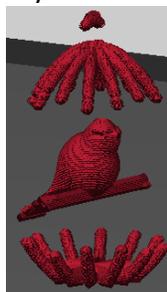
# The following are usually not issues:

- **Need Rafts?:**
  Most objects do not need it. If your object has lots of corners, or not much surface touching the ground, it may need a raft. Rafts are used to prevent "cookie sheeting," warping that results from uneven cooling that can cause your object to warp and crack or fly off of the printer before it's done. Beware it may be difficult to remove and will add some plastic to your total cost.

- **Polygon threshold:**
  Usually not an issue unless your model was 3D scanned or sculpted. All .stl 3D models consist of a mesh of triangles. Those with a lot of triangles have gigantic file sizes that some programs and computers struggle to handle. Consult the library's 3D printing specialist if this is an issue.

- **Model Integrity:**
  Usually not an issue. The 3D printing program cannot understand models that do not have closed (watertight aka. manifold) meshes with correct normals. Consult the library's 3D printing specialist if this is an issue.

**Katonah Village Library**
26 Bedford Rd
Katonah, NY 10536
katonahlibrary.org
914-232-3508
katref@wlsmail.org