General Setup in Rhino:

- Scale your object and make sure that it is no bigger than:
  10" (254mm) x 6"(154mm) x 6.7" (170mm) (X,Y,Z)
  The bigger your object the longer it will take to print

- All of your surfaces must be closed. Fix your work as you are modeling to make sure that you are maintaining a water tight model.

- Using boolean operations is the best way to insure that you are not left with open surfaces. If you are unsure of which boolean operation to use; the command "**Boolean2Objects**" allows you to click through your options before selecting

- Non-manifold objects, even if closed, will sometimes give you problems. Explode the object and rejoin all surfaces. Most of the time this will fix the issue.

- Check the thickness using the command "**ThicknessAnalysis**". If your model is too thin it will not print correctly. We suggest not modeling smaller than 1/8" or 3mm. Thinner will print, but we can not insure that it will print correctly.

- Ensure your objects are all on the same base plane. Floating objects will create more need for supports. The "**Align**" command in a side view can help fix this.

- Use the command "**Mesh**" to mesh your model. More polygons will create a smooth surface, but take longer to print. Try keeping the slide bar in the center.

- Check that your mesh is good. Use the command "**MeshRepair**" and click on "check mesh". If your mesh is bad click "next" and try to fix it through the repair options provided.

- The Dremel DigiLab 3D Slicer units are in mm. Check that your units are set to mm. Export to an .STL file and open it in the Dremel DigiLab 3D slicer program.

Helpful Commands:

ExtrudeSrf- Extrudes a surface in one direction

OffsetSrf- Extrudes a surface based on surface direction

Cap- Fills in any planar holes

EdgeSrf- Creates a surface from 2-4 open curves

PlanarSrf- Creates a surface from closed planar curves

Loft- Creates a surface through multiple curves

Revolve- Creates a surface by revolving a curve around an axis

ShowEdges- selecting naked edges shows where your object is open

Material:
Hatchbox, Matter Hackers or eSUN PLA 1.75mm Filament

You may supply alternate colors. Do not purchase any material that is not PLA.
We **cannot** print ABS, Carbon fiber, nylon, Pet-G, glow-in the dark, wood, metal carbon fiber or any filament labeled abrasive.
**Please make sure that it is the correct filament before buying**

General Setup in Dremel DigiLabs

- Download and install the Dremel DigiLabs Slicer. Add a Dremel DigiLabs 3d45 printer and open your file in the Dremel DigiLabs program.

- On the right hand side of the program are icons that will allow you to Move, Scale, Rotate and Mirror your object when it is selected.

- On the left hand side of the program is where your print settings are set. Use the drop down menu under profile to select the proper settings for your project.

  Low Quality/High Speed- Will print the fastest. This is very good option for massing models or quick iterations of your model

  Medium Quality- This is very good option for any model.

  High/Ultra Quality- This is overkill for most models and takes a considerable amount of time. This is a good option if you are creating pieces with built-in hinges, or pieces that are meant to fit together.

- Under Print setup make sure that **General Support** and **Build Plate Adhesion** are checked off.

- Click on **Custom** up next to where it says **Print Setup.** Make sure Generate support is checked and Support Placement is set to Everywhere.

- Change to an appropriate Build Plate Adhesion based on your model:

A **Skirt** is the default on DigiLabs for bed adhesion. This doesn't supply any support really, but acts as a good warm up for the printer. It is good if your model is flat at the bottom w/o supports
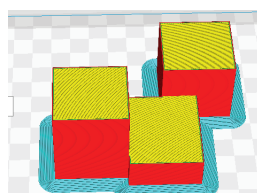
A **Brim** creates several layers that are flat next to your model. This thin layer helps keep your supports from failing and reinforces small pieces
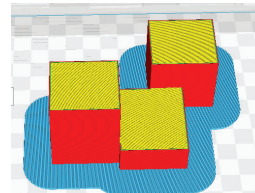
A **Raft** creates an entire surface underneath your print. This can add a considerable amount of time to your print. We suggest trying a brim first.

- Click Prepare and wait for it to slice your object. You are not done yet; check that your object has sliced properly before saving your file.

- On the left side down at the bottom is an icon that looks like an eye. Click on this and choose the option that says **Layers**. Set your color scheme to **Line Type.**

- Use the slide bar on the right to scan through the layers of your model. Check where supports are being placed and that your bed adhesion is correct.

- If your bed adhesion is intersecting your model there is a problem, something in your model is not on the build plate. Move your object down by 0.1mm at a time and re-slice until it is all flat, or fix your model in Rhino.

- If everything looks correct you may save your file and print from an available machine. Follow all guidelines posted: use the glue provided, watch your file until the first layer is down

You are only permitted one machine at a time plan accordingly