Updated 10/1/2015
Local copies available from
\\Hardlab-Laser01\Public\VisioDrawings\LaserCutterTips&Tricks

Laser Cutters: Tips and Techniques

(Most of the information here can be applied to any laser cutter/engraver and CAD software.)

Using Microsoft Office Visio with the Microsoft Research's Universal Laser Systems PLS6.60 Laser Cutting System

By Mike Sinclair, Microsoft Research, sinclair@microsoft.com
1st published 8/7/2002. This is a continuous work-in-progress document.

Important Information and Disclaimers

- **Microsoft Corporation, through its Microsoft Research division, is providing this information as a courtesy to end users to assist you with using a laser cutter/engraver as well as generating designs in Microsoft's Visio. Microsoft is not a manufacturer of laser cutters or other materials described in this manual and assumes no responsibility for your use of any laser cutter or any materials you use with the laser cutter, regardless of whether you are applying this information. YOU USE THIS INFORMATION AT YOUR OWN RISK. MICROSOFT AND ITS AFFILIATES DISCLAIM ALL RESPONSIBILITY FOR HARM OR DAMAGE TO YOU, YOUR MATERIALS, SURROUNDINGS AND/OR YOUR USE OF YOUR LASER CUTTER.**

- **Familiarize yourself with the owner's manual for your laser cutter before using it. Refer often to the owner's manual when using your laser cutter.**

THIS DOCUMENT IS BASED ON A SPECIFIC LASER WE USE: In Microsoft Research, we have a Universal Laser Systems model PLS6.60, which is equipped with an 80-watt carbon-dioxide laser and energy wavelength output of 10.6 microns. This laser has a 2.0" fl lens. The PLS6.60 has a low-power, visible red laser, when enabled and the higher-power carbon-dioxide laser is disabled, for safely indicating the cutting or etching path. The laser cutter also has an aluminum honeycomb floor as its cutting platform. Whether your configuration is similar or not, you are advised to adjust the information in this document to match your specific configuration. ALL LASERS AND LASER SYSTEMS ARE DIFFERENT. USE CARE WHEN USING THIS INFORMATION WITH YOUR LASER CUTTER. MAKE SURE YOU ARE CLEAR TO USE YOUR MATERIAL IN THE LASER SYSTEM AND THAT THE EXHAUST AND FILTRATION SYSTEM IS APPROVED FOR THIS MATERIAL, ESPECIALLY IF HAZARDOUS GASSES ARE PRODUCED. MAKE SURE YOU HAVE READ AND ARE ABIDING BY THE MSDS SHEET FOR THAT MATERIAL.

- **Familiarize yourself with the locations of nearby fire extinguishers.**

- **The laser-cutting system should be used only in well-ventilated areas. Make sure the exhaust system and air assist are on.**

<u>**Background and Compatible Materials**</u>

This document describes how we use the laser cutter and Microsoft Office Visio® with our Universal Laser Systems laser cutter to engrave and cut *some* plastics, ceramics, woods, coated metals, and other materials. Make sure your management, venting system and filtering system are compatible with the materials.

a. **Plexiglas** (acrylic): One of the best, least-expensive plastics and the easiest to cut. Tends to be brittle in small or thin features. Comes in all shapes, sizes, and colors. Durability varies with type; some acrylics are less brittle than others.

b. **Delrin** (acetal): Structurally, perhaps the best plastic. It is extremely strong, durable, more flexible than acrylic, and naturally lubricated. It also is one of the most expensive plastics. Comes in black or off-white only. Delrin is a little harder to cut than acrylic Do not use glues or any other kind of adhesive on Delrin – few things stick to it. It can be welded with heat or fastened with screws. **Be very careful with Delrin. The vapors are very flammable, bordering on explosive, and can catch fire with an almost invisible blue flame. Avoid cutting very thin sections as these can (read *will*) catch fire.**

c. **Lexan** (polycarbonate): Much stronger, more durable, and more flexible than acrylic, but harder to cut. Edges tend to burn or yellow easily, making it difficult to get a clean cut. It's difficult to cut this material thicker than ~1/8". Keep the protective sheet on both sides while cutting, as this reduces yellowing of the non-cut portions.

d. **Styrene:** It burns easily and leaves a ridge on the edge. Flexible (not brittle), but not as strong as other plastics.

e. **Polyethylene Terephthalate PET** (Mylar, clear plastic bottles): Similar mechanical properties to polycarbonate, but edges cut a lot cleaner, though yellowing occurs near the edges – keep the protective film on. Cannot be glued with acrylic glue.

f. **ABS:** Similar to acrylic but more durable.

g. **HDPE:** Durable but hard to cut over ~1/4"

h. **PVC, vinyl – This material is <u>not allowed</u> in the laser cutter because of the highly toxic fumes.**

<u>**SPECIFICATIONS FOR THE LASER CUTTER**</u>**:**

1. **Work space:** 32" wide x 18" deep (for our device). Origin is the upper left corner.
2. **Spot size** (when focused): approximately 0.005" using the 2" FL lens
3. **Depth of field:** approximately +/- 0.1"
4. **Movement resolution of head:** 0.001"
5. **Kerf** (material removed during a cut): *approximately* +/- [0.003"-0.006"] ~(0.006"- 0.012" total, both sides)

<u>**Laser-Cutter Safety**</u>

When using your laser cutter:

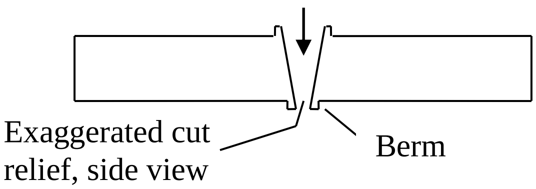
1) Ensure that you have a readily accessible fire extinguisher.
2) Only use your laser cutter in a well-ventilated area.

3) Remember that cutting certain materials, such as PVC, can create toxic fumes.
4) **Do Not** leave a laser job unattended. Monitor the laser cutter until the job is finished.
5) An approved fire extinguisher and/or fire blanket should be close at hand.
6) Adhere to the local regulations concerning your laser cutter.

Terms:

Kerf is the material removed during a cut: ~+/- [0.003" - 0.006"] ~(0.006"- 0.012" total, both sides).

The exit kerf will be slightly *smaller* (by 0.002"-0.004") at the bottom than at the beam entrance for thick plastic. This means the cut piece will be 0.002"-0.004" larger at the bottom that the top. This depends on the material property, thickness, and beam focus. For a slightly squarer cut, move the focus point about a third of the way down from the top surface. This might require a slight increase in power and could cause a slight rounding of the top edges of the cut.



Berm is the material build-up near cut edges, top and bottom.
There might be a noticeable lip or berm at the cut edges. Check with your fingernail. You might have to remove this with a file or scrape with a sharp metal edge, such as a razor blade, knife or a pair of scissors, if this is an issue when stacking multiple layers or gluing. Some acrylics (cast or 'Chemcast', 'Optix' with a brown paper film) will produce *less of a berm* than other types (extruded - with blue plastic film).

PREPARING THE LASER CUTTER:

1. **A new job:** If starting a new laser cutting job, there are rulers along the X=0, Y=0 borders that are black with white markings that become dirty quickly. A short time after you have used your laser cutter, these markings will become unreadable from the affluent produced and may not be able to be cleaned. Cover the markings with thick transparent tape (packing tape). When the markings become unreadable, simply replace the tape. A separately etched plastic strip can also be used. Etch deeply.
2. **Placing material in the cutter:** Open the lid and place material in the cutter, firmly against the upper-left 90-degree reference stop or any other place on the honeycomb floor, but note the position where the cutting will take place. Be careful not to gouge the expanded metal bed near the origin when loading large sheets of material.
3. **Setting the cutting position:** You can draw the piece to be cut at the appropriate place on Visio's 32" x 18" drawing area, keeping the origin at the upper left of the drawing. Do not draw anything that touches or crosses the 32" x 18" border. Refer to the manual.
4. **Focusing: Note:** Our new PLS 6.60 laser cutter has "automatic focusing" where all you specify is the Z-thickness of the material being cut**.**
    a. **For manual focusing:** Eyeball the height of the piece <u>with regard to</u> the end of the air-assist nozzle to make sure it won't crash into the piece when moved from its home resting place (the upper right) to the cutting area. If it appears that the head won't clear when rapidly moved to the focus position, remove the material, lower the table to a safe height, using the down arrow

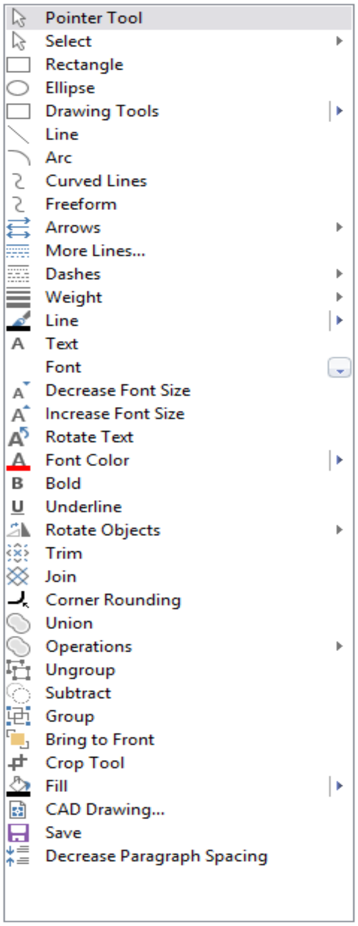
when in the focus mode. As soon as you push the **Z** button to enter the focus mode, the head will move rapidly to the focus position, usually near the origin in the upper left. With the lid open, press Z on the cutter, and use the manual focusing tool to focus in the area you will be cutting, moving the head with the X, and Y buttons as described in the manual. After focusing, remove the tool and press Z to re-home the head. Use SELECT and ESCAPE on the laser cutter to get to the file menu, the normal mode of use.

5. **Printing:** From the local PC, "print" the job on the laser cutter. You can set the laser mode, power, speed, and **PPI** from the Preferences button with the Print selection. More later.
6. **Viewing laser path before cutting:** To see the laser-cutting path before cutting, open the lid and press START. A non-cutting red laser spot will be emitted instead of the invisible carbon-dioxide laser and trace the cutting/engraving path. You can increase the cutting speed **temporarily** to expedite this check. Make sure you reset the speed before cutting/etching.
7. **Cutting, stopping and pausing:** When you're ready, close the lid and press START. If you want to stop the process prematurely or temporarily, raise the lid or press **PAUSE**. The former option provides the *fastest response, but you can't resume*. Press the PAUSE button to home the head after completing the present operation. Raising the lid will stop the carbon-dioxide laser immediately; pressing PAUSE without raising the lid might take awhile. You can press the **PAUSE** button again to **resume** the job from PAUSE, or you can start over by pressing **START**. Raising the lid, then pressing PAUSE stops the cutting path after the laser has been shut off, and you cannot resume the cut.
8. **Don't leave lid open:** The laser stays "warmed up" and quieter with the lid closed.

1. **CUTTING MODES WITH THE LASER CUTTER:** There are two modes of operation, **vector** and **raster**. In raster (engrave) mode, the head sweeps back and forth, "filling in" the shape, a (fat line, a filled polygon, or a bitmap image). In vector mode, which creates thin lines, the head traces the vector path. Use **raster** for engraving and **vector** for cutting. The vector mode will cut only along thin lines of near-zero width, whereas the raster mode will try to fill everything designated with that color, including filled areas, alphanumeric characters, and thick lines. Be careful about **not** leaving a vector outline to your engraved polygons. You may end up with a cut-out polygon! *This is easy to forget.*
2. With material that is not opaque, you'll know you're cutting completely through the material when you see the aluminum honeycomb cells fill up with smoke. You also will see short, bright flashes of light when the laser hits the honeycomb edges after cutting through a clear or translucent piece. If you don't observe either of these, you're probably not cutting completely through.
3. When cutting completely through material placed atop the honeycomb, a faint but permanent outline of the honeycomb might be visible in the vicinity of the cut line on the bottom of the piece being cut. If you don't want this, elevate those pieces with scraps or leave the protective paper or film on or place a sheet of paper between the honeycomb and plastic. Make sure the elevated objects aren't in the cutting path. Don't crash the head into the elevated piece. If you don't cut completely through the piece but have to break plastic (especially acrylic) to release the part, **the broken edges should be filed down as they can be as sharp as broken glass and cause injury if not removed**.
4. The energy used to cut or engrave is a function of laser power and inversely to the speed set in the Print ->Properties dialog box. There is also a direct relation to PPI though it's probably best to use 1000 PPI when cutting. Use just enough energy to do the job. These notes and the ULS manual provide a starting point for some materials. Laser pulses are metered synchronously to the head's position, so multiple passes will yield the same spatial pulse pattern.
5. The laser-cutting order**,** from first cut to last**,** is:

a. All rasterized engraving operations first, as drawn from back to front in the same color, then in color order, as listed in the laser Properties setting. For example, black areas will be engraved before red areas. Overlapped engraved regions will receive only one pass. Ghosting of other layers, both vector and raster, can occur within an engraved area. It's best to engrave with only one type and color per area—no overlapping raster or vector objects within an engraved area—to be safe**.**
b. Vector, cutting operations follow, from back to front**,** then in color order as listed in the Laser Setting. For example, all black lines will cut before any red lines. Multiple overlapping lines will receive multiple passes.
   Part cutouts should be done after all internal part operations are completed. Otherwise, a slight offset from part movement may result**.** Sometimes, the gases produced accumulate in the honeycomb cells, and small explosions might occur, popping the part up, and sometimes over, if it is completely released. Cut the part out last.

6. **Importing your data into Microsoft Office Visio®:**
   You can use virtually any 2-D drawing package that outputs files in the following formats: .igs, ,dxf, .dwg, .dgn, .ai, .emz, .cmx, .cgm, .cdr, .eps, .emf, .png, .ps, or .wmf. Visio will import most bitmapped files, or you can use programs such as CorelTRACE, InkScape or Vextractor to produce a vector outline as a .wmf or .dxf file from a bitmap. If you use a different CAD package, learn what formats work.
7. **Drawing in VISIO:** Here's a start at a usable Quick Access Tool Bar you can use in VISIO that has all the tools you need to create drawings for the laser cutter. Under **File/Options/Quick Access Tool Bar**, download and import the "LaserQuickAccessToolBar.exportdUI" file from **\\hardlab-laser01\public\VisioDrawings\LaserCutterTips&Tricks** **or**, choose **All Commands** and manually **Add** the following commands:



8. **Printer settings:** Note: Our crack IT department has created a startup file that should setup all of these as defaults automatically.
   a. The Visio Print Setup ***mus***t be set up for a 32” x 18” printer and drawing size which is the size of the cutting bed of the PLS 6.60 laser cutter. **If you can’t get past this stage, don’t print! It may ruin your material. The VISIO auto-startup file on MSR Hardlab-Laser01 and Hardlab-Laser02 computers should do the following:**
      i. Choose the P6.60 printer.
      ii. In Print Setup, choose *User Defined Landscape* and *Adjust* to *100* percent.
      iii. In Page Size, choose *Custom Size, 32 x 18, Landscape*
      iv. In Drawing Scale, choose 1:1.

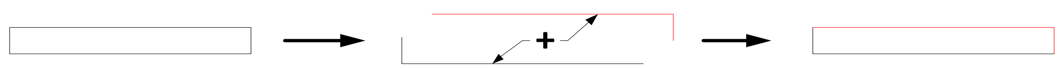
      Make sure the above is correct and do not proceed until this is correct!

   Note: if you are having problems with the above, check Home->Options->LaserCutterSettings. You should see “User-Defined LANDSCAPE” filled in as Page Size. This is a common problem where it is left blank.

   b. Use different colors for lines and engraving fills to designate different power, speed, and PPI settings. Power, speed, and PPI for each of eight preselected colors can be set in File->Print->Properties->LaserSettings. The colors, in cutting order, are black, red, green, yellow, blue, magenta, cyan, and orange. These can be set for raster (engrave) and/or vector (cut), or skip (off) in the“Laser Settings described above. **The colors must be pure** colors such as RED = RGB 255,0,0 and blue = RGB 0,0,255, YELLOW = RGB 255,255,0…...

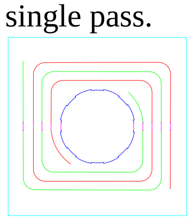
c. A single color setting applies to both vector and raster operations drawn in that color. The raster and cut depths are not the same for power, speed, and PPI. It’s best to keep engrave and cut colors separate (IE avoid using the Rast/Vect setting for a single color).
d. For the Quality/Throughput setting, use 5, the highest setting, in Print Properties. The ‘6’setting may cause weird scaling and offset errors.
e. If engraving a gray-scale image (using the color black), the engraving power will be inversely proportional to the gray level, and black will result in the highest power/deepest engraved depth.
f. If cutting closely spaced features, high powers and low speeds might cause distortions or re-melting of these features, especially along the thin walls separating the pieces. Use multiple passes at lower power and/or cooling off periods between cuts on adjacent sides of thin features. NoLaserActionPauses can be implemented by slow speed and low power on scrap areas.

You should be aware that Delrin is extremely susceptible to catching on fire and not self-extinguishing under these conditions. When cutting close to a recent cut, such as a thin wall, or cutting close to an edge, in our experience, the thin piece might catch on fire with a dim blue flame and might not go out immediately. In our experience, you can avoid this situation either by not cutting near thin pieces or close to an edge or by delaying the adjacent cuts on thin-wall features. You can force a vector color change to cause a pause when cutting lines near each other.



Based on our experience, **we strongly recommend you avoid cutting where thin pieces of Delrin will result that could catch fire easily**. In the above diagram, the black lines are cut first, then the red lines. If there are other black and/or lines in the diagram, this will insert an effective delay between when these black and red lines are cut, enabling the thin part to cool down between cuts. Alternately, you can execute a PAUSE – a slow, low-power vector cut in a scrap area to generate an effective pause between colors cut. We also recommend this for other materials as well.

Another tip to reduce thin-wall heat distortion is to leave small “tags” that will continue to support the thin structures through cutting. When done with the thin structure cutting, use a single pass to cut through the tags. The example below demonstrates this. The multiple thin wall structure cuts are separated in time with multiple colors. They are also supported through cutting with small (~.03”) “tags” that keep them aligned where they would normally be heat distorted if released. The final operation (before the final outline cutout) is to cut the tags (magenta) free with a single pass.



    g. In Print->Properties->LaserSettings, you can choose Skip, Vect/Rast, Vect, or Rast for each color.
        i. Skip will result in no action to objects drawn or filled in that color.
        ii. Vect/Rast will raster/engrave fat lines and filled areas and vector/cut thin lines. I don’t recommend using this. You should explicitly direct the operation as a cut or an engraving.
        iii. The Vect operation will only cut thin lines. Fat lines or filled areas will be ignored for a Vect operation.
        iv. Rast will raster/engrave fat lines and filled areas. It also will try to raster/engrave any thin lines in that color.
        I usually use only Rast for engraving only and Vect for cutting only and use separate colors for engraving and cutting to keep these operations separate.

9. Engraving will produce a rough surface. Don’t expect to engrave a filled polygon and end up with a smooth finish at a precise depth within the polygon. Engraving also takes quite a bit of time.
    a. For raster/engraving acrylic with no masking required to avoid fogging, try 10 percent power at 50 percent speed.
    b. Masking - When engraving, the heat and effluent gases produced might cause discoloration and/or permanent distortion to un-engraved surfaces nearby. Use the protective film or masking tape over these areas to protect un-engraved areas. Burned masking tape itself can cause discoloration, so experiment. To help smooth (melt) an already-engraved area, print the same image but at 50 percent power and 50 percent speed and ~0.5” out of focus (further away). For vector marking (engraving thin lines) acrylic with no masking required, try less than 50 percent power at 35 percent speed and experiment. The laser maker recommends covering unprotected plastic with thick dish soap before cutting or engraving then rinsing off afterwards.
    c. For some reason, the laser cutter might not engrave small areas. Specifying less than 200 dpi might help. Starting on a larger engraved area might help. Experiment.

10. **Cutting/Engraving recommendations: Cutting -** use the 2” FL lens, at 1000 dpi—
    Most plastic material can be purchased locally at TAP Plastics**,** 12021 Northup Way, Bellevue. For larger and lower cost quantities, try Calsak Plastics in Seattle.
    Light vector engraving of Delrin, acrylic or polycarbonate - use power 20 percent, 60 percent speed.
    CUTTING, *suggested starting values* (Note ‘P’ = Power, ‘S’ = Speed) YMMV so experiment!
    a. 0.125” Delrin: P=100, S=2.
    b. 0.25” Delrin: P=100, S=1.3.
    c. 0.517” Delrin: P=100, S=0.5. Try to focus ~0.2” below the top surface. It might take two passes. **Watch for dim blue flames that don’t automatically extinguish!** Don’t repeat a cut until the previous cut has cooled.
        i. For cuts that leave thin walls/pieces (<1/8”), trim the vectors into multiple pieces and colors so one side of cut is executed after the other to enable cooling in between. Cutting too close in time and distance can cause severe deformation.
        ii. Make sure all cuts are done so previous passes have time to cool. You can implement an effective delay by cutting a different color line with 0.5 power and 0.5 speed.

d. 0.046” acrylic: P=100, S=7.
e. 0.114” acrylic: P=100, S=2.5
f. 0.232” acrylic: P=100, S=1.4.
g. 0.440” acrylic: P=100, S=1.0.
h. 0.022” polycarbonate: P=100, S=30.
i. 0.116” polycarbonate: P=100, S=2. Leave protective film on until after cutting to reduce yellowing of the surface near the cut.
j. 0.133” styrene: P=100, S=2. Expect slightly yellow edges.
k. 0.11” rubber: P=100, S=1.5. Wash thoroughly afterward to remove black, sooty residue.
l. Xerox transparency film: P=20, S=20. Use two passes at P=10, S=20 for intricate features. Anchor the film with small weights well outside cutting boundaries to prevent warping and moving from air currents.
m. Aluminum tape: Cut from the non-aluminum side at P=100, S=3.
n. Aluminized Mylar film: P=40, S=30.
o. Cutting 3” camera-case foam: P=100, S=1.
p. Engraving acrylic (suggestions only):
    i. 0.150”: P=50, S=8.
    ii. 0.078”: P=50, S=20.
    iii. 0.053”: P=50, S=30,
    iv. 0.043”: P=50, S=40.
    v. 0.036”: P=50, S=50.
    vi. 0.030”: P-50, S=60.
q. ~~Cutting vinyl letters without cutting through the paper backing (Scotchcal 220 film): P=50%, S=100% (suggestion only – you need to experiment here to be able to just cut through the vinyl and not the backing)~~
r. Engraving paint on metal:
    i. Raster: P=40, S+100, 150 PPI.
    ii. Vector: P=40, S=20, 1,000 PPI.

11. Miscellaneous cutting recommendations
    a. 0.115” birch plywood: P=100, S=4.
12. Discoloration and/or surface problems near cuts: It might be helpful to leave the protective paper/plastic film on the plastic while cutting to preserve the quality of the surface. Just remember that you’ll have to peel the cover film/paper off manually, so try to leave a continuous surface to make the film removal easier. Placing wet tissue paper or thick liquid detergent over the material might reduce this problem. The manufacturer suggests that a thin coat of dishwashing soap will reduce this problem. Wash afterward.
13. **Caution again** – plastic that is not cut completely through but forced to separate (IE hammering to break apart) can leave ***V E R Y*** sharp edges, as sharp as a freshly sharpened knife. **FILE THESE DOWN** before using or it can cause serious injury.
14. **Removing small parts** (wanted and/or unwanted) – For removing many small parts (IE screw holes), the hole centers will many times be slightly raised (on one side of the plastic) but still attached to the sheet. Place the raised side down on a flat surface and hit with your hand with a medium force. All the hole centers should pop out. Sometimes just hitting the cut sheet against the table will dislodge them. If some of the small parts are to be saved but indistinguishable from the scrap, put a superficial etched mark on them like an “X”.

VISIO (and other CAD programs) NOTES:

Microsoft provides the following information for your benefit. We recommend that you experiment or test a given method to ensure that it produces the proper result. YOU USE THIS INFORMATION AT YOUR OWN RISK. MICROSOFT DISCLAIMS ALL RESPONSIBILITY FOR HARM OR DAMAGE TO YOU, YOUR MATERIALS, AND/OR YOUR LASER CUTTER.

1. **Using AutoCAD's .dxf files** (works most of the time):
   a. File Open (specifying Files of Type .dxf) your file.
   b. Edit (or right-click) ->CAD Drawing Object->Convert->OK to convert to the Visio format. Use defaults.
   c. Select and Ungroup all objects.
   d. Delete the drawing frame (the sometimes invisible rectangle around the drawing).
   e. Select the rest of the objects, which might have invisible lines, and choose an appropriate line color. We have noticed on our laser cutter, black is the only color choice.
   f. Check for correct scale and correct if necessary.
   g. Note, the color of the lines cannot be changed. Not sure why.
   h. Make sure the scale is correct.
   i. Curved lines are usually made up of many tiny lines (polylines).
   j. When subsequently trimming/ filling/Boolean operations, you may note some unusual behavior such as inverted polygons. Try to correct these by removing the offending lines.
2. **Vector Cuts**: Zero-width lines cause the laser to trace the line (cut) when in the vector cut mode. Use Format->Line->Weight->Custom = 0.
3. **Polygons**: Inside/outside does not have significance when cutting. The kerf is of finite width and there is no tool compensation as there is in CAM for a milling machine. You must add this compensation manually if it is important. The laser will trace any cut-enabled, zero-width line. If you're engraving a polygon, you must fill it with an engrave-enabled color, and inside/outside will dictate what gets engraved. Turn off the perimeter line (specify No Line), or the polygon will be cut out after engraving.
4. **Origin**: 0,0 is at the upper left of the 32" x 18" workspace. Stay at least 1/8" away from the cutter's aluminum borders. Place drawn objects no closer than 1/8" to the X and Y axes.
5. **Environment**: (Note – our great IT department has generated an automatic startup file to effect all laser cutter defaults)
   a. First, open a File->New Drawing. Before drawing anything, check/change the Line Weight (Format/Line/Weight/Custom) to 0. This enables the printer driver to turn all drawn lines automatically into thin vectors rather than trying to engrave a fat line. You can use a thicker line, but the cutter will try to engrave it. After you complete the drawing, select-all and make sure the formatted size for all cut-only lines is 0.
   b. Select No Fill in the paint-bucket icon for polygons or circles to be cut unless you want to engrave the filled area. If engraving without cutting the polygon, select No Line in the paintbrush icon, or the area will be cut after it's engraved.
   c. Use the View->Size and Position window, in which you can enter numbers for size and offset. It executes immediate math: For example, it will replace 4+15/16 with 4.9375.
6. **Cutting Order**: The order of laser operations is from back to front within the same color, then by color order (see PRINT PROPERTIES). Use these attributes (Shape-> Order and/or color) to

specify the cutting/engraving order. If mixing engraving and cutting, the laser will engrave first. Make the part cut out the last operation (with Shape->Order->Bring-to-Front) to avoid material movement, because a part will move slightly when it is completely cut out

7. **Raster/engraving speed**: When engraving multiple areas filled with a specific color, the head will sweep back and forth to include all horizontally overlapping areas. If these areas of like color are spaced far apart horizontally, it might take a long time to engrave. If possible, position these like-color engraved areas close together horizontally or arrange so there is little horizontal overlap.
8. **CorelTRACE (or, Vextractor, InkScape (free))** is great for converting a bitmapped image to a vector object. (flat-bed scan the object to a bitmap image and process in Photoshop). Sometimes the program will trace with multiple overlapping lines. In Advanced Trace, try Medium Complexity, Medium Node Reduction and Smooth Node Type with a large minimum object size, such as 300. I've had great luck in flat-bed scanning an object, tracing it, then cutting out a nearly identical part. If you need to compensate for the lost material in the kerf, I've used Photoshop's Select->Modify->Expand or Contract by a discrete number of pixels to add or subtract kerf.
9. **No Cut:** If you don't want specific objects on your drawing cut or engraved, make it a non-cutting color or a dashed line, or simply position it completely off the 32" x 18" page onto the background. **Make sure no objects or lines cross or straddle the 32" x 18" border.**
10. **Hole-Cutting Example:** Cutting a 0.162"-diameter hole in Delrin produces approximately a 0.165"-diameter hole and a 0.154"-diameter slug, indicating a hole .003" too big in diameter and a slug .008" too small in diameter. This varies considerably. If it's important, EXPERIMENT.
11. **Screw Holes**: To cut holes for screws (tap, clearance, and body sizes), use this data in your CAD:

Tap holes: subtract ~0.004" from the desired tap drill diameter for the kerf compensation. Tapping is a handy, strong, and quick way to fasten two pieces of laser-cut material together and to enable later disassembly. Using our fast tapping tool or a tap in a portable drill, apply light pressure on the tap in the tap hole, and the rotating tap will pull itself into the hole. When at the required depth, quickly reverse the direction to back the tap out. If you wait too long, the heat generated by the tap melts some of the plastic, which then cools and solidifies. This might seize the tap in the plastic and break it when you try to remove the tap. For a Nyloc-like friction fit, don't tap all the way through, to enable the inserted screw to thread itself through the small, untapped portion and provide a tight friction fit that will withstand vibration.

Body holes: subtract 0.004" from the desired body drill diameter for the kerf compensation.

Clearance-fit holes: subtract 0.008" from the desired diameter for the kerf compensation.

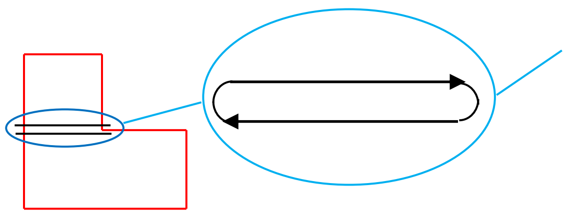
SCREW SIZE	TAP HOLES			BODY HOLES			0.001" CLEARANCE HOLES		
	VISIO RADIUS	**VISIO DIAM.**	DESIRED DIAM.	VISIO RADIUS	**VISIO DIAM.**	DESIRED DIAM.	VISIO RADIUS	**VISIO DIAM.**	DESIRED DIAM.
2--56	0.033	**0.066**	0.070	0.042	**0.084**	0.086	0.039	**0.080**	0.084
4--40	0.0425	**0.085**	0.089	0.056	**0.111**	0.113	0.051	**0.106**	0.108
6--32	0.0515	**0.103**	0.107	0.069	**0.138**	0.140	0.064	**0.130**	0.134
8--32	0.066	**0.132**	0.136	0.082	**0.164**	0.166	0.077	**0.156**	0.159
10--32	0.0775	**0.155**	0.159	0.099	**0.197**	0.199	0.090	**0.183**	0.186
1/4--20	0.0985	**0.197**	0.201	0.124	**0.248**	0.250	0.120	**0.240**	0.246
5/16--18	0.1265	**0.253**	0.257	0.155	**0.311**	0.313	0.152	**0.303**	0.309
3/8--16	0.1545	**0.309**	0.313	0.187	**0.373**	0.375	0.181	**0.362**	0.368
7/16--14	0.182	**0.364**	0.368	0.218	**0.436**	0.438	0.000	**0.000**	

Typical screw head and hexagonal nut, actual sizes

Screw Size	Pan Head Diameter	Flat Head Diameter	Nut Size Flat-Flat	Nut Thickness
2-56	0.165	0.160	0.185	.062
4-40	0.209	0.209	0.245	.095
6-32	0.260	0.253	0.308	.105
8-32	0.314	0.302	0.338	.123
10-32	0.371	0.346	0.370	.128
1/4-20	0.483	0.468	0.438	.220

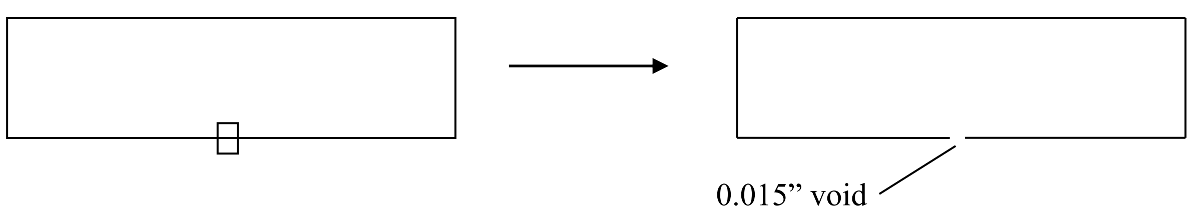
12. **Quick Bevel Cut:** To produce a beveled finish atop a normal cut:
    a. Cut the part out normally. You might have to design small "tags" to hold the part in place for the next operation if the part will shift much after the initial cutout. Leave everything untouched in the cutter after the cuts.
    b. Lower the floor by 1.0".
    c. Make a copy of the vectors you want to bevel, change to a unique color, and exactly overlay these atop your original object(s).
    d. Move these new vectors in X and Y on your drawing, up .01" and to the left by 0.015" (add -0.015 to X and +0.01 in Y in the SIZE-POSITION window). This accounts for this machine's AZ/EL (L/R) offset per 1.0" out-of-focus for our printer.
    e. Remove the protective film and vector-cut only that unique color at 2x to 4x speed, same power or a half to a quarter of total power. Keeping the film on will weld the film to the plastic.
    f. Return the floor back to the normal focus height.
    g. You may experience some removable or permanent fogging to the plastic because you removed the film. You can execute a superficial cut just through the protective film (~50% pwr, 100% speed) about 1/8" just inside the outside perimeter of the finished piece. Do this initially and remove the film from all but the inside area then execute steps a-f above..
    h. Experiment
    i. ULS Corporation – Please add final mirror adjustment screws to compensate for beam skew past the focus point.
13. **Precision bending acrylic, Delrin or polycarbonate:**
    This method can produce a near-perfect bend in thin acrylic, acetal or polycarbonate by using the laser to heat only portions of the plastic. The downside might be a weaker section of plastic at the bend. Experiment. The following works well for 1/16" polycarbonate:
    a. Cut out the part using regular cutting procedures. Keep the plastic's protective layers on when cutting but remove for bending.
    b. Lower the workpiece 2" below focus position by using the FOCUS buttons.
    c. Draw two lines 1/16" apart, slightly longer (~1/8" longer on each end) than the place you want to bend. Draw the first line top down (or right to left) and the second, 1/16" away, bottom up (or left to right). Select these two lines and copy/paste 19 times. Select these 19 copies, place atop the original 2 lines, and group all 20 copies—40 lines total—together. This will cause the laser cutter to apply non-cutting heat in the area of the line without having the head retrace. Cut at 30 percent power and 45 percent speed for 1/16" polycarbonate.

d. Immediately remove plastic and bend to desired position.
e. For multiple bends, it’s best to do one bend at a time (change the color of the other bend places to a non-cutting color). Make sure the head won’t hit the piece.
f. It’s great for making project boxes. Because any bending will consume a portion of the material, you will have to account for the material used in the bending. Experiment first.



g. **Laser Origami:** A good friends and colleagues at Hasso Plattner Institute in Germany, created a technique where you can use gravity to automatically bend plastic at right angles. Check out http://www.hpi.uni-potsdam.de/baudisch/projects/laserorigami.html .

14. **Making Lenses:** Crude optical lenses can be made by rasterizing shapes in clear acrylic. Describe the lens’ Z-shape as a black-and-white gray-scale image—the darker the image, the deeper the engrave—and engrave in 3-D, out of focus. Use Photoshop to describe a radially symmetric shape. A radial linear gradient from white to black, with black in the center, will produce a conical concave shape. Change the gradient profile, using Adjust/Image/Levels, to approximate a circular arc to achieve a spherical lens output. Try cutting 1” (long) out of focus at 150 PPI, with power = 100 percent and speed=30 percent**.** Change to 3D in Print Properties. Shift the image by X = -0.015” and Y=+0.01” to compensate for the lateral angular shift when 1.0” out of focus. This will create a negative lens with a ~0.05” Z depth in clear acrylic. Return the Print Properties to Normal.
15. **Cutting thin materials**: As the thin part is cut out, it might fly away, making it difficult to find and creating the potential for interference with subsequent cuts (or clogging your exhaust fan). As a final operation, install a small “tag” on one or more of the lower parts of the outer-most cutout; a ~0.015” void in the cutout vector should work and is easy to later tear apart. Overlay a 0.015” square on the vector line, select it and the line, and choose Shape/Operations/Trim. Then remove the square and the 0.015” line to produce the uncut tag.

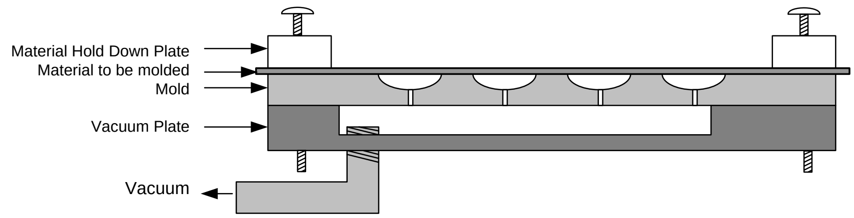


16. **Reducing fogging when etching**: To reduce fogging (sometimes permanent marring of the plastic surface that happens when the gasses from etching touch un-etched plastic and causes fogging of the surface) you can leave the protective film/paper on but you may end up having to peel 100’s or 1000’s of isolated paper islands (no fun!) or remove the all protective film/paper and etch with a reduced power. Another method is to remove all of the top film/paper (so you

won't have to "weed" later) and brush on a thin film of thick liquid detergent. Etch your plastic then wash away the remaining detergent. Works great.

17. **Annealing to remove residual stress:** Often, with many closely spaced cuts or deep etching, the plastic piece will come out warped. This is from thermal stresses trapped in the plastic. One way to relieve these stresses is to anneal at a high temperature. I've had luck putting the plastic in a constant temperature oven for a few minutes. Let ~1/4" Acrylic anneal at 100 degC for around 5 minutes, resting on a flat surface in the oven.

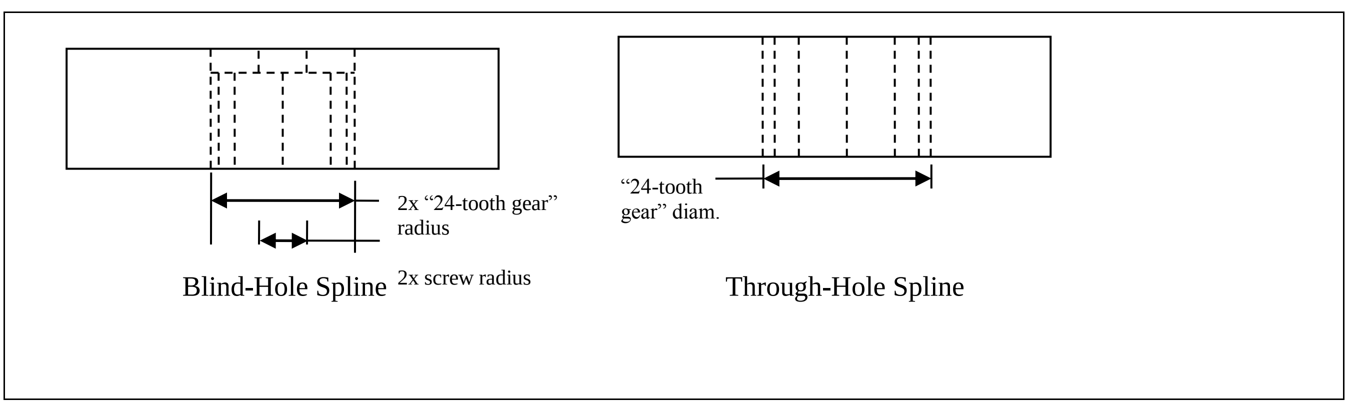
18. **Vacuum forming**: Molds and a vacuum-forming jig can be made quickly for 3-D heat-molding thin materials such as thin polycarbonate and acetate.



Cut the material hold down and vacuum plates out of acrylic. The mold can be out of acrylic but will depend on what's being molded. You may have to remove the berms on the acrylic cuts for a good seal. I've had luck using gray-scale images in the laser cutter and engraving out of focus by about 1". This will make a smooth surface, but you can't get much fine detail. Cut small holes (0.008"- 0.010") in strategic places in your mold to pass the vacuum. Make sure to cut plenty of aligned bolt holes in all plates and molding material to clamp the fixture together and seal for the vacuum. Cut a tap hole in the vacuum plate and tap to avoid having to add a nut, but don't over-tighten. Once everything's clamped and the vacuum is pump connected, use a heat gun to heat the top (the mold material) uniformly and distort to the mold shape. Allow it to cool before unclamping.

19. **Blind tap holes:** For blind tap holes (threaded holes that don't go all the way through the piece) in plastic (Delrin in this example), use concentric vector circle cuts, starting with the outside diameter as in the table above, decreasing the *radius* by .010" at each step, down to .010" minimum. In other words, for a #2-56 tap hole in Delrin, 0.033", 0.0200", and 0.010" radius cuts at 50 percent power at 2.5 speed will yield a ~1/16" deep hole per pass. Allow to cool, then tap, using a **bottoming** tap. Alternative**:** For a blind 4-40 tap hole, engrave at P=50, S=14 for 0.186" deep in acrylic.
20. **Blind Splines** for hobby radio-controlled (R/C) servo shafts, for example:

These splines appear to be 24-tooth with varying diameters. Use a 24-tooth gear as a close geometry.



Blind-Hole splines in 1/8" DELRIN. Use 24-tooth gear as spline geometry:

-

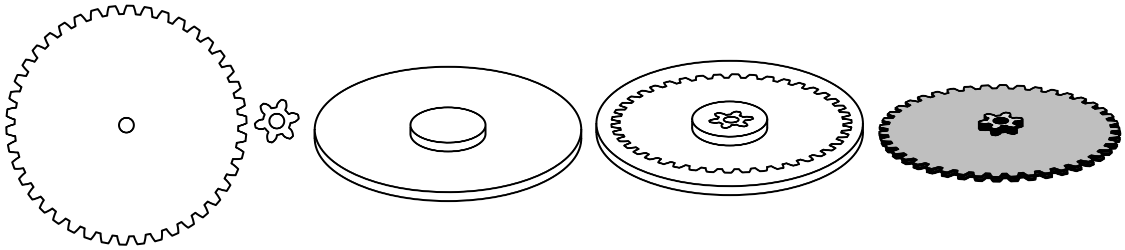
Use These Numbers in Visio

Type	Spline Radius	Screw radius	Power	Speed	Remarks
Large servo	.133"		100	45	raster (filled gear), 2 passes. .bmp image=.276"
		.048"	50	2	Cut vector (.047" radius screw hole), 1 pass
Medium servo	.123"		100	45	raster (filled gear), 2 passes. .bmp image=.254"
		.047"	50	2	Cut vector (.047" radius screw hole), 1 pass

Through-Hole splines, 24 tooth:

Type	Gear Diam.	Power	Speed	Remarks
Large Servo	.245"	50	2.5	Cut vector gear, 2 passes
Medium Servo	.225"	50	2.5	Cut vector gear, 2 passes

21. **Gears**: You can make good spur gears with the laser cutter. Delrin is what we use for this because of its toughness and self-lubrication. Here is an inexpensive, clever gear-generator program: http://woodgears.ca/gear/. Note that the gears produced do not have any kerf compensation. The new version does have a variable called "Slop". Entering a negative value will "grow" the outline, effectively causing a correct *kerf compensation*. The program is on our laser cutter computer.
22. **Stacked and Welded Parts**: You can make stacked parts (spur gears in this example) welded to each other. You can also do this for other non-gear items as well. Start by making your gear pair using apps like woodengears.com (a great program!!). Make sure to add "slop" to add back the kerf, especially for smaller gears. Cut two acrylic disks about 20% larger than their respective gears. Make sure there are no berms on the edges that would not let them stack in surface-to-surface contact.
    a. Place them in the laser cutter so they will stack with the larger disk against the X=Y=0 edge.
    b. Concentrically stack them as shown.
    c. Cut the axis hole with focus, power and speed that will go through both thicknesses.
    d. Cut the small gear with appropriate focus, power and speed to go through the top layer **AND** only through **part of the bottom layer**. This causes the layers to be welded to each other at the cut location…..AS LONG AS THEY ARE CLEAN AND IN CONTACT.
    e. Cut the larger gear with the power and speed to cut through a single layer.
    f. Carefully pry apart the smaller disk's outside part to remove it.
    g. Depending on the calibration of your laser, non-circular hole results, etc, you may have to experiment to add offsets to insure the gears and axis hole are concentric, especially for smaller gears.

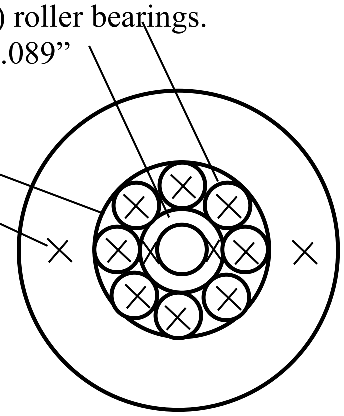


I’m working on ways to make out of Delrin and other more gear-friendly plastics.

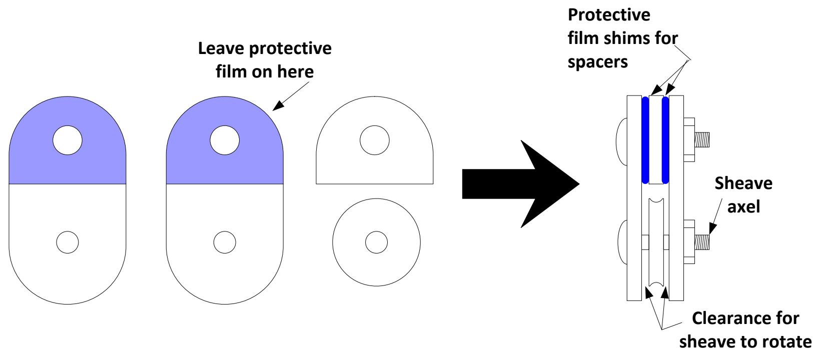
23. **Smallest Feature Size**: Visio (or the ULS PLS6.60 printer driver) doesn’t appear to pass a circle of less than 0.008”diameter. No object will be cut below this dimension. Also, if you need to cut a very small hole (< ~.050”), use concentric circles to ablate the hole as you may not be able to remove the remaining slug. As an example, draw concentric circles of diameter = .045”, .035”, .025”, .015” for an approximate .050” hole.

24. Delrin roller bearing example (as shown in the library .vsd drawing):

a. Bearing diameter: Cut 8x 0.1104”-diameter (0.0552” radius) roller bearings.
b. Cut inner sleeve: 0.108” I.D. (0.054” radius), 0.178” O.D. (0.089” radius). The I.D. will accept a 4-40 screw for mounting.
c. Cut inner race of outer part 0.37” I.D. (0.185” radius).
d. Mark (‘X’) the top of **every part**. Rollers assemble upside down with regard to races.
e. Cut two inner-sleeve geometries out of Xerox transparency material or acetate film for spacers. Spacers can also be realized by leaving the protective film on in the correct places on the bearing holders. This is done by laser scoring and removing just the film in areas where the thin spacer is not needed. This works very well. See **Easy Shims** below.



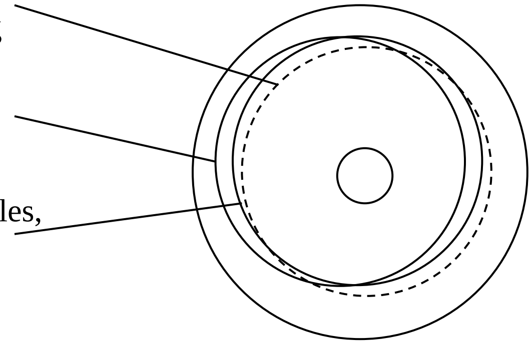
f. **Easy Shims (thin spacers):** To make thin spacers for clearance, leave part of the protective foil/paper on the plastic (~ 0.002” per film layer) . For example, to make a pulley where the sheave rotates even though the cheeks are fastened tightly with screws, leave areas of the cheek parts with the protective film/paper intact. Use low power/high speed to just cut through the film on sections that don’t need to be cut through the plastic:



.

25. –Small bearing race (or O-Ring groove): This is tricky but works well (for this particular laser cutter at least). Start with this - The race is “cut” as vector circular paths with the lens 1.0” out of focus. Position the table 1.0” lower than the focus point. At this grossly out-of-focus position, the laser beam cuts a smooth, wide, rounded bottom trough. The beam has an astigmatism, as well as a slight angular Z-axis misalignment when out of focus (to the lower right in our case) which must be accounted for by moving the drawn race feature to the upper left. It’s important to make all measurements with regard to the desired center. The following is based *only on our laser cutter* that has a tilt to its beam in X & Y. YMMV.

a. Draw a desired race-diameter circle (use a non-cutting color) line for non-cutting) where you want the bearing race to be.
b. Draw two additional race-diameter circles (solid lines), one 0.015” to the left and one 0.015” to the right of the original race circle. Group these two, and make them a separate color from each other. You now have two circles, 0.03” apart in X. This partially corrects for the beam’s astigmatism.
c. Displace this two-circle group by 0.015” in the –X direction and 0.010” in the -Y direction (displace toward the upper left). This compensates for the misalignment of the beam along the Z-axis when mis-focused (long) by 1.0”. Do not scale or rotate the result! Use these same displacements for all race sizes.



d. With the system out of focus (long) by 1.0” (use the **Z** button in the Focus mode), vector-cut the two displaced circles at 100 percent power, 10 percent speed, and three passes for Delrin, two for acrylic (The number of passes is variable, but this seems to work for 3/32” -> 5/32” ball bearings obtained at the hardware store). For acrylic, make two passes per side. This will be just deep enough for 3/32” ball bearings and a 0.022” polycarbonate bearing keeper.
e. Raise the table back to focus height, taking care not to crash the head, and cut the center hole, the outer diameter, and anything else that needs to be cut or engraved. Make sure the part cut out is last, or a small displacement error might result.
f. A bearing keeper can be cut out of thin (0.022”) polycarbonate. Cut holes slightly larger than the balls.

26. **Snug Fit:** For a tab-in-slot snug fit in ~0.2” acrylic, cut the slot ~0.012” smaller than the thickness of the acrylic. You may have to experiment. Remember the entrance size of a hole will be larger than the exit (bottom) when cutting through. See the section on **KERF.**
27. **Press fit for ¼-20 cap-screw head:** Use a 0.179”-radius (0.358”-diameter) hole in acrylic. Groove the screw head parallel to the screw shaft with the Dremel cutoff wheel or grinder. This should help relieve the pressure and provide additional holding force to the plastic. Gently tap the screw head into place, entering from the laser (larger-diameter) side. This is good for making a fixed, screw-in plastic knob.
28. **Smooth, Wide Groove:** A relatively smooth groove can be produced with equally spaced lines cut out of focus. Cut as parallel vectors, 0.028” apart, 2” out of focus (long). Avoid cutting adjacent lines sequentially to allow for cooling. Color adjacent vectors with sequential colors—allow to cool between cuts for short grooves. You can use zero power cuts after regular cuts and use multiple copies. P=7, S=10, 1000 PPI yields about 0.014” of cut depth per copy in acrylic.
29. **Removing solder mask over copper from a printed circuit board:** Engrave the area to be removed at P=50, S=100, 500 PPI. Scrub off remaining film with acetone. Acetone won’t touch the un-lasered solder mask. Be careful not to engrave over any non-copper-filled areas, because engraving directly on FR4 material will burn and turn it to conductive carbon.

30. **Flexible, multilayer printed circuit board:** A workable PCB can be made using aluminized Mylar film. You should know which side of the Mylar is conductive (sometimes both sides are but insulated from each other. Investigate with an ohmmeter) Also, the aluminum coating, a few 10s of nm thick, is easy to scratch and break electrical connection so handle with care. Start with a flat plate of metal (aluminum works fine). Thoroughly clean it and coat with a low-tack spray (repositionable) adhesive. Bake it in an oven at ~100 deg C for around 15 minutes to make sure the adhesive sticks to the plate which also reduces the tackiness of the adhesive to around that of a PostIt Note. This is to keep isolated pieces from flying away when being cut. Apply a piece of aluminized Mylar to the sticky plate and thoroughly press it to the plate so all parts stick. Be careful not to scratch the aluminum coating. Laser cut (suggest P=40, S=30 for a start) your first layer outline. Weed (remove) parts of the film that aren't wanted. Press clear tape (packing tape) over the remaining circuit and press/rub all areas firmly so that the tape sticks to all parts of the circuit. Carefully remove the tape from the plate which should bring with it all circuit parts, fixed in their original arrangement. Apply the tape to your desired substrate. Electrical connection can be made with conductive ink or epoxy. For multi-layer circuits, apply a piece of double-stick tape to a clean metal plate. Laser cut apertures where you want vias (vertical electrical connections) between adjoining circuits. Remove the tape from the plate and apply it to the conductive side of the circuit produced above. Apply a small quantity of conducive epoxy to the middle of each via and apply the next circuit layer, conductive side down, to the aperture film side. Procedures can vary – experiment!
31. **Paint-filled etched lines or markings:** Painting a piece of metal with dark paint then etching with a vector cut through the paint makes a beautiful "etching". Etching through the colored layer in anodized aluminum can also produce a finely etched drawing. Painting just the laser cuts: (from the artists in the lab) -
    a. With the protective paper/film still on, etch through just the protective paper or plastic film on acrylic or polycarbonate.
    b. Spray the plastic with etched paper/film with a contrasting color paint.
    c. Remove protective paper/film after the paint dries.
32. **Heat stress causing sheet stock to warp:** Many times when cutting a lot of parts out of large pieces of sheet stock, the accumulated heat built up in the sheet over a short time will cause unequal top/bottom mechanical stress, resulting in the sheet warping out of plane and may result in these cuts being cut grossly out of focus and/or alignment. This can be mitigated by placing weights (or otherwise clamping) on the non-cut part to stay in-plane. I use a heavy thin steel bar that I lay over the sheet stock. First verify the lens carriage won't hit the bar. Do a practice cut without the bar to the verify path of the lens carriage before you cut. I use a thin enough bar that the carriage head won't hit but pass over it.
33. **Generating** Vector Fonts:

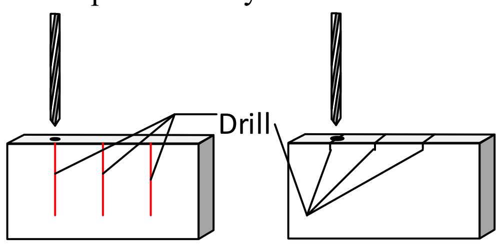
Generating vector text for laser cutter

a. (not the fastest as just typing but beats tracing rasterized fonts)
b. Open InkScape
c. Go to Extensions/Render/Hershey Text
d. Render text
    i. Choose Live Preview
    ii. Font Face – choose a font
    iii. Action – "Typset that text"

iv. Text – enter a line of text

e. Select Apply
f. Select and copy the line of text rendered to the InkScape screen
g. Copy into your VISIO document.
h. Make sure to reduce the line width to 0 if you want it as a "cut-able" vector

34. **Marking drill guides:** If you will need to drill parallel to the plane of the sheet plastic (edge drill), you can make an accurate marking by vector cutting a superficial marking line on your piece. If I need to drill into the edge of my piece for threading a screw for example, I mark a line on the plastic using 20% power + 20% speed. This way I can see the line and line up the drill press and guess at the midpoint of the thickness of the plastic. Another method is to make a short cut (~.01") across the edge of the plastic I later want to drill. If I use cut-through values, I will wind up with a very thin "notch" where I need to drill which is easy to find and start with drill bit.



Additional information:

[Plastic Instructional Videos](http://www.tapplastics.com/info/video.php) by TAP Plastics: http://www.tapplastics.com/info/video.php

TAP Plastics has a store in Bellevue, WA, at 12021 Northup Way, (near 20th Street), 425-861-0940. There is also a TAP Plastics in Seattle near Westlake: 710 9th Ave. N, 206-389-5900.