_NEWLAB

Building 128, Brooklyn Navy Yard
Brooklyn, New York 11205
718 488 7005

Product Realization & Prototyping at Newlab

Laser Cutter Programming & Operation

Updated 10/30/19

General Policies

- Review our general policies.
- If you have any questions, email prototyping@newlab.com or book a Digital Fabrication Consultation.

Access

- Use of the Laser Cutter requires the completion of both the DigiFab, Textiles & Electronics Lab, Casting Room, and Spray Booth Orientation as well as Laser Cutter Programming & Operation.
    - Book a spot in Laser Cutter Programming & Operation.
- Time must be scheduled in advance of use.
    - Schedule time on the Laser Cutter directly or by visiting the Laser Cutter page on New Lab OS.

Price Structure

- The Laser Cutter is reserved in 30 minute blocks, with a fee of $15 per block.
- Time reserved on the machine includes use of the computer workstation, material testing, set up. It is recommended you come with your file ready.
    - The computer closer to the door in the main room of the DigiFab Workshop has Laser Cut 5.3 on it and does not require booking for use.
    - If you require help or have questions about setting up a file or running the machine, please sign up for a Digital Fabrication Consultation.
- When you reserve a machine, you will be given the choice to pay the full amount for your booking or a 25% deposit. If you select the latter, you will be charged the remainder upon completion of your booking.
- It's always better to over-book, and then email us at prototyping@newlab.com when you're done so that we can adjust the cost for your booking.

    - Adjustments will be made rounding up to the nearest half-hour, so if you book 2 hours and only use 45 minutes, contact us and you'll only be charged for 1 hour.
    - If you run over your reserved time, you will be charged accordingly.

Machine Specs

- Model: Rabbit 1290-SE
    - http://www.rabbitlaserusa.com/Manuals/User%20Manual(DSP5.3).pdf
        - (useful information starts on page 12)
- Bed Size: 33" x 45"
- Maximum Cutting Thickness: 3/8" (depending on material)
- Maximum Material Thickness (for Engraving): 9"
- Kerf depends on material and settings used, but typically is around 0.01".

Materials

NOTE: Neither the list of banned or approved materials is conclusive. If you have a plastic, or any other kind of material, not covered by the approved or banned list, schedule a Digital Fabrication Consultation before trying to cut it. It is not allowed to cut an unidentified or unidentifiable piece of plastic.

Banned Materials

- Any plastic containing vinyl - creates chlorine gas when laser cut
    - PVC
    - Pleather containing PVC
    - Moleskine, or similar notebooks
    - EVA
- Polycarbonate
- ABS
- MDF
- Plywood (Cutting is banned - engraving is not)

Approved Materials

- Cuts: paper, cardboard, acrylic, polyester/mylar, leather, rubber (may require multiple passes), basswood and other wood sheets
- Engraves: all of the above, marble, glass, anodized aluminum, plywood

Using the Laser Cutter

Computer Access

- Login and password will be provided via email when you book time on the Laser Cutter.

Preparing Files in Illustrator

- The thickness of the path is irrelevant, but make sure that you are working in RGB, using black, centerline paths with no fill.
- All compound paths must be released.
- In some cases, paths may require ungrouping and being put on the same layer.
- Things like fonts must be transformed into paths. If given the option, use the "Expand" option as opposed to "Create Outlines".
- Go to File -> Export As, (NOT Save As) to make your DXF file.
    - The file must be formatted using AutoCAD version 2000.
    - The file must be exported in inches.
    - If you have "Export Selected Artwork Only" then you must have the artwork you want exported selected before selecting Export As.

Importing Your DXF to Laser Cut 5.3

- Click on the snowflake icon from the taskbar to open Laser Cut 5.3
- Go to File -> Import.
- Your file must be formatted in DXF using AutoCAD version 2000 - if it's not, please bring it into Illustrator and re-export it.
- The file must be in inches.
- If you make your model in Solidworks or another CAD software and there's any issues, import the file into Illustrator, then back out of Illustrator as a DXF2000 file.
    - Solidworks and other CAD programs may not join line segments by default, so you may need to join them yourself either in Illustrator or within Laser Cut 5.3.
- **Known Bug:** If your file name or the directory it's in has any non-English characters in it, the file may not load.

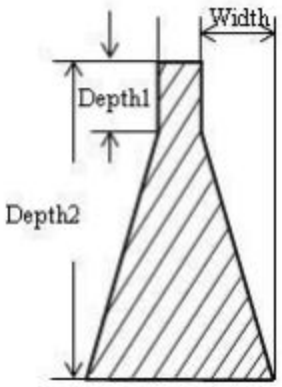
Using Laser Cut 5.3

- The hand tool "flings" the view, the selection tool - "Pick" - is next to it.
    - As an alternative to the hand tool, you can move around by zooming in and out using the mouse wheel, and the zoom will go towards or away from the mouse cursor.
- Paths which you will be cutting must fit completely within the grid. The grid correlates to the laser cutter's platform, however, the location of your paths on the grid do not (by default) actually correspond to their location on the laser cutter's platform.
    - Grid squares are in cm (everything else is in inches).
    - The software and hardware will have separate origins.
        - The two must correlate, but it's up to you to set it up so they do.
- Checking Vectors
    - Tools -> Data check
        - Check closed will check for paths which are not closed.

        - Check self-intersect will check for paths which intersect themselves (such as a figure-eight).
        - Check intersect will check for distinct paths which intersect each other.
        - Check overlaps will check for distinct paths which are directly overlapping each other in whole or in part (such as duplicate paths).
        - Any artwork with issues will become selected on clicking Check.
    - If your file has paths which should be joined but aren't, go to Tools -> Unite lines.
        - Tolerance (in mm) should be ~.01.
        - If specific artwork is selected, only that will be affected, otherwise Unite Lines will be global.
- Organizing Layers
    - Layers are operations - cut, engrave, etc - paths assigned to the same layer will be done together within the same time operation, and will use the same settings
        - Layers are designated by color.
    - Layers/operations will occur in order from the top down.
        - **Known Bug:** If the order of layers are changed, a few workflow procedures will cause them to revert to default. The order can be changed, but should be done as a last step before sending the job to the machine.
- Settings & Modes
    - Click on the layer's mode name to change it, then double-click on the power or speed to open that mode's settings.
    - In addition to the modes and settings below, scroll to the right in the layer window to find "Output" and "Times".
        - Output - whether or not that layer will be done as a part of the job.
            - Layers not set to output will still be considered for the purpose of the laser origin and the test bounding box on the laser cutter.
        - Times - the number of passes for a given layer. For some materials or material thicknesses, more than one pass will be required to cut through.
    - **Mode:** Cut
        - The laser will follow along the center of a path. Most often used for cutting out the as drawn shape.
        - Can also be used for scoring if the settings are adjusted to match the engrave settings for your stock.
        - Settings:
            - Speed - how fast the head is moving in mm/s.
            - Power - the intensity of the laser as a percentage.
                - **Never** set above 80%.
                - When experimenting to find appropriate settings for your material it is better to run the machine slower, rather than with more power, to achieve deeper cuts.
            - Corner power - the power at corner points, so that corners don't get double-exposed at the same intensity. Should always be 10-15 lower than Power.

            - Overlap - the distance in millimeters that the laser will overlap the starting point of a closed path in a cut.
            - Set to “Always Blow” every time.
            - **Do not change anything within the advanced settings menu.**
    - **Mode:** Engrave
        - Engraves a series of shallow lines from left-to-right, filling inside the perimeter of a **closed** path.
        - As the laser moves left-to-right, it will toggle on or off as it hits a vector on the same engrave layer.
            - I.E. - Engraving the letter “**O**”, which is actually two closed paths, won’t give you a solid circle as long as both the outside and inside circle are on the same engrave layer.
        - Settings:
            - Speed - how fast the head is moving in mm/s.
            - Power - the intensity of the laser as a percentage.
                - N**ever** set above 80%
                - When experimenting to find appropriate settings for your material it is better to run the machine slower, rather than with more power, to achieve deeper cuts.
            - Scan Gap - the resolution of the raster lines, the distance between engraved lines along the Y-axis.
                - A lower value will produce a higher resolution engraving, but the increase in heat in a localized area will cause the etch to be deeper, and may warp your material.
            - Bi-dir (Bi-directional) - when checked, the laser will engrave the raster lines as it goes back and forth, when unchecked the laser will only engrave lines as it goes in one direction. This brief cooldown can improve the quality of the engraving by depositing less heat in your material, but the operation will take longer.
            - Blow should always be **checked**.
            - ExpandScale should always be **unchecked**.
            - Reach out to Prototyping staff if you’d like to use the advanced settings under the “. . .” button **before** attempting to do so yourself.
    - **Mode:** GradeEngrave (Gradient Engrave)
        - Works like engrave, but engraves the inverse of the closed path and makes a beveled “gradient” from the surface to the engraved portion to the stock. This is useful for things like making stamps.

        - Settings (See diagram):



            - Min-Power is “Depth1” and Power is “Depth2” - you’ll need to test these settings to find what you like for your purposes. Refer to the settings sheet next to the laser cutter computer as a starting point for these settings, beginning your tests by using the power setting for Power and setting Min-Power as 5-10 lower.
            - Grade-width is “Width” - determines the angle of the gradient between Power and Min-Power.
            - Bi-dir (Bi-directional) - when checked, the laser will engrave the raster lines as it goes back and forth, when unchecked the laser will only engrave lines as it goes in one direction. This brief cooldown can improve the quality of the engraving by depositing less heat in your material, but the operation will take longer.
            - Blow should always be **checked**.
            - Repair should always be **unchecked**.
    - **Mode:** Hole
        - This mode does not work, and should not be used.
- Laser Origin
    - The origin is represented by a blue diamond somewhere within the grid.
        - Occasionally the blue diamond may be “beneath” a different representational icon and may not be visible.
    - Go to Laser -> Set Laser Origin
        - Can also be accessed by clicking this icon:

        - The Left Top/Left Center/etc. options will dynamically adjusts as more paths are added. The software creates an invisible bounding box around all artwork and aligns to the Left Top/Left Center/etc. of that box.
        - You can also set the origin based only on selected artwork.
        - Right Down is recommended so that you are aligning the head near where you are controlling the head’s position.

        - While the selection box is open you can use the crosshair cursor to manually select the origin. After clicking “OK” it will still show the crosshair cursor, so you may need to re-select the pointer cursor.
- Other Tools
    - Estimate Work Time:

        - Displays a time estimate for your job.
    - Simulate:

        - Simulates what the laser will do by displaying the toolpath.
    - Set Simulate Speed:

        - Adjusts the speed of the simulation. It’s recommended that the speed should almost always be set all the way to the right so that the preview is as fast as possible, as the slowest speed will make you watch the toolpath slower than real time.
- Setting Order of Operations
    - The order in which the layers/operations occur is defined by their order in the list at the top-right of the window, where the top one will occur first, followed by the second, and so on.
        - To change this order, select one of the layers and use the “Up” and “Down” buttons to move it accordingly.
    - To customize the order of the individual paths within a layer, go to Tools -> Adjust Output Order
        - When cutting, it’s recommended to cut things from the inside out. The software will attempt to set this as the default order when your file is imported.
        - **Known Bug:** If the order of layers are changed, a few workflow procedures will cause them to revert to default. The order can be changed, but should be done as a last step before sending the job to the machine.
- Sending to machine
    - As the grid within the software does not correspond to the laser cutter’s build area, it’s recommended that you always have “Immediate” checked before saving and downloading, like so:

    - Save the laser cutter file.

        - You will not be able to send the job to the machine until you have saved the laser cutter file.
    - DownLoad [sic] button
        - The laser cutter must be on for the download screen to come up, otherwise you'll get an error message.
        - Displays a queue of jobs, though it's recommended to only work with one file at a time.
        - Click "Delete All", to clear the queue, then "Download Current", this uploads the file as it is currently is (i.e. not how it was when last saved).
        - Your file name should appear in the queue.
            - Often a different file name will show up instead (laser.mol) because your file name is too long or has spaces or other characters that the software doesn't like. This will still be your file.

Operating the Laser Cutter

- Turn the key to turn on the laser cutter
    - The fume extractor (big black box behind the computer) should come on at the same time.
        - If it doesn't, make sure the fume extractor is working by pressing the power button.
        - **Do not operate the laser cutter if the fume extractor does not turn on.**
        - If you'd like, you can turn the fume extractor off while you're setting up, but it **must** be turned on before you hit "Start" on the laser cutter.
- Datum
    - Press Datum before doing anything else to home the head upon each start-up of the machine.
    - If you do not press Datum, you may find that the head moves very slowly and other positioning complications may arise. It is possible to damage the machine if the head has not been homed.
- Material
    - Check that the material you're using is flat. If it's not, use the magnets or the blue painter's tape from the top of the laser cutter to keep the material down.
        - Excessively warped material should not be cut.
    - If you use magnets, be careful that the head will not come in contact with the magnets during cutting.
- Moving the Head
    - The head will only move if nothing is selected on the screen, so hit escape to make sure that this is the case before using the arrow buttons to position the head where you need it.
- Focusing the Laser
    - Press the green "Down" button to lower the platform if the head will not make clearance over your material.
    - With your material in place, position the head in the approximate center of where you plan to cut.

    - Hold the focusing tool under the crook of the arm coming from the head.
    - Press the green “Up” button in small increments to move the platform up slowly.
    - The focusing tool should be snug between the arm and your material but move in and out smoothly.
- Origin
    - If you’ve checked “Immediate” in the software, wherever your head is positioned will correspond to the blue diamond laser origin you set in the software.
        - As the grid within the software does not correspond to the laser cutter’s build area, it’s recommended that you always have “Immediate” checked before saving and downloading.
    - While the file name is highlighted, press Enter/Carriage Return to open the logical origin menu.
        - If the file name is not highlighted because you were moving the head, press Enter/Carriage Return first.
        - The options will be “Set Logical Origin: Yes/No” and “Cancel Logical Origin: Yes/No”. Respectively, these mean set the logical origin, don’t set the logical origin, clear the local origin, or don’t clear the local origin.
        - Once the logical origin is set, no matter where you move the head it’ll return to the origin point you set. This applies only to that specific job.
- Test
    - Pressing “Test” will move the head from the origin you’ve set in a rectangular bounding box around all of the artwork present in your file. It will not fire the laser at this time.
    - The bounding box will include layers set to not output in this particular job, for the sake of positioning, but those layers still will not be cut or engraved when you press start.
    - If your bounding box would go outside of the printable area, the laser cutter will beep a few times and display “Soft Stop” on the screen. If this happens, you may need to:
        - Check that you’ve got space for your artwork relative to where the origin is set in the software.
        - Adjust the origin on the laser cutter.
        - Reset the logical origin.
        - Adjust the origin in the software.
        - Rearrange your files in the software to make sure that they all fit within a 33” x 45” space.
- Running Your File
    - Pre-Start Checklist:
        - Material is in position?
        - Laser is focused?
        - Origin is set?
        - Top is down?
        - Fume Extractor is on?
    - Once the checklist is complete, press “Start”
    - The laser cutter will emit a tone when it’s complete.

Fire Safety

- Some materials, such as cardboard, are more likely to cause more issues than others.
- Should a small flare-up occur and be immediately blown out by the laser cutter's blower, it may not be an issue, but if it happens more than twice you should stop the job and adjust your settings.
- When testing, if you don't know the settings you should be using for certain, start with lower power settings and work your way up with test cuts/engraves.
- If a fire should start and not be immediately blown out, press the red Emergency Stop button and open the top.
    - If the fire is small enough to be extinguished manually, do so.
    - If the fire is not small enough to be blown out manually, use the fire extinguisher on the wall.
    - If you've had to press the Emergency Stop, do not continue with operation of the laser cutter afterwards. Inform a member of staff immediately either by finding someone on-site or by emailing prototyping@newlab.com with "FIRE" in the subject line and including your phone number.
- If at any point you do not feel safe handling a fire yourself, even with a fire extinguisher, leave the space and immediately call 911. New Lab's address is:
  19 Morris Ave.
  Brooklyn Navy Yard, Building 128
  Brooklyn, NY 11205
    - After the fire department has been notified, inform a member of staff immediately either by finding someone on-site or by emailing prototyping@newlab.com with "FIRE" in the subject line and including your phone number.

Finishing Up

- Clean up after yourself
    - Return any magnets used to the top of the laser cutter.
    - Remove any tape from the sides of the platform.
    - Dispose of small pieces of scrap in the trashcan in the laser cutter room.
    - Dispose of any large pieces of scrap in the dumpsters outside.
- Log out of Windows

Known Bugs, Common Issues, & Basic Troubleshooting

- Layer Ordering
    - In some circumstances, the order of layers can change despite being set - there are a variety of different actions that can cause this behavior.
    - To avoid this, set the order of your layers last, just before saving and downloading.

- Soft Stop
    - When either testing or cutting, if the head is set to go outside of the printable area the laser cutter will beep a few times and display “Soft Stop” on the screen. If this happens, you may need to:
        - Check that you’ve got space for your artwork relative to where the origin is set in the software.
        - Adjust the origin on the laser cutter.
        - Reset the logical origin.
        - Adjust the origin in the software.
        - Rearrange your files in the software to make sure that they all fit within a 33” x 45” space.
- Origin Issues
    - Instead of starting where you’ve set the head, if it travels to a location that seems to approximately correspond to where your artwork is on the software’s grid, make sure “Immediate” is checked in the software, above the DownLoad button.
    - If the head travels to some other location, that means that a logical origin may have been set previously and the machine stored that position.
        - Move the head to where you’d like to set the origin, then press Enter/Carriage Return once to highlight the file name, and again to open the logical origin menu.
            - The options will be “Set Logical Origin: Yes/No” and “Cancel Logical Origin: Yes/No”. Respectively, these mean set the logical origin, don’t set the logical origin, clear the local origin, or don’t clear the local origin.
            - From here you can either set a new logical origin or clear the logical origin, allowing you to have the origin set to wherever the head is positioned.
- File Name Doesn’t Show Up on Download Current
    - Make sure your file is saved, as the software needs a file name to refer to.
        - Occasionally a different file name will show up instead (laser.mol) because your file name is too long or has spaces or other characters that the software doesn’t like. This will still be your file.
    - If the file will not show up under any circumstances, turn off the laser cutter, restart the computer, and then turn the laser cutter back on and try it again.
    - If it still doesn’t work after a full restart, something may have been corrupted and you will have to start a new file in Laser Cut and re-import your artwork.
- File Won’t Import / Parts are missing
    - There are a few reasons that your artwork may not import:
        - Your artwork did not follow the guidelines listed above under Preparing Files in Illustrator.
        - Your file name or the directory it’s in has any non-English characters in it.
        - Your .DXF file is corrupted, in which case you’ll need to re-export it from Illustrator.

- Broken Communication Error When Clicking DownLoad
    - The laser cutter must be turned on in order to download your job from the software to the machine.
    - If this issue persists even though the laser cutter is on, turn off the laser cutter, restart the computer, and then turn the laser cutter back on and try it again.

Still have questions?

Email prototyping@newlab.com or book a Digital Fabrication Consultation