HARVARD GSD BEGINNER'S GUIDE TO ROLAND MDX 40-A

prepared by Alexander Matthias Jacobson

Gund Hall's CNC Milling Equipment

ONSRUD 96C18

3-Axis (X 48" , Y 96", Z 8")
Vacuum hold-down
Automatic tool change
Automatic Dust collection
Max Feed Rate: 1200 inches per minute
Max Rotations Per Minute (RPM): 24,000
Software: Mastercam X7

AXYZ 4008

3 Axis (X 96", Y 48", Z 6")
MDF Screw-down anchoring
Pneumatic non-automated tool change
Automatic Dust collection
Max Rotations Per Minute (RPM): 18,000
Max Feed Rate: 300 inches per minute
Software: Mastercam X7

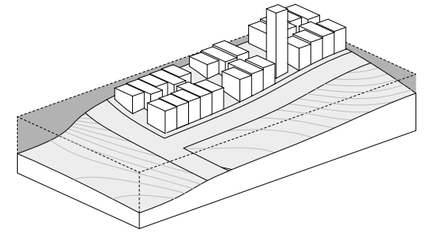
ROLAND MDX 40-A

3 Axis (X 12", Y 12", Z 4")
Double-stick tape anchoring
Hand tool change
No dust collection
Max Rotations Per Minute (RPM): 8,000
Max Feed Rate: 120 in/minute (3000mm/min)
Software: Mastercam X7 or Roland SRP

Typical Milling Procedures

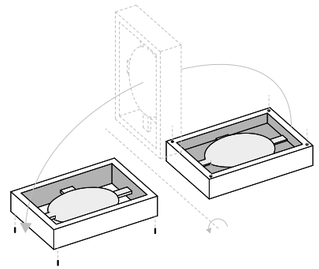
Topography Model

Goal: Represent landscape or surfaces, including roads, textures, topo lines and buildings.
Machine: ONSRUD, AXYZ, ROLAND
Typical Materials: blue foam, white foam, high-density foam, mdf plywood, solid wood, laminated pieces of solid wood



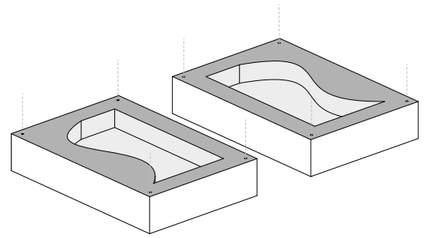
Flip Mill

Goal: Create a form milled on two or more sides.
Machine: ONSRUD, AXYZ, ROLAND
Typical Materials: blue foam, white foam, high-density foam, mdf plywood, solid wood, laminated pieces of solid wood



Mold Making

Goal: Create two halves of a mold which can be mated together and used for casting.
Machine: ONSRUD, AXYZ, ROLAND
Typical Materials: blue foam, white foam, high-density foam, mdf plywood, solid wood, laminated pieces of solid wood, wax



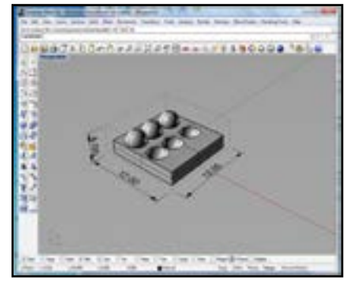
2D Contour Cut-out

Goal: Cut out 2D pieces of material.
Machine: ONSRUD, AXYZ
Typical Materials: plywood, acrylic sheet, mdf

Roland MDX 40-A Workflow Overview

1. Create Geometry

- Center on the origin
- Maximum size: 11” x 11” x 3.5”
- Max vertical wall height: 1.6”
- Export as closed .stl

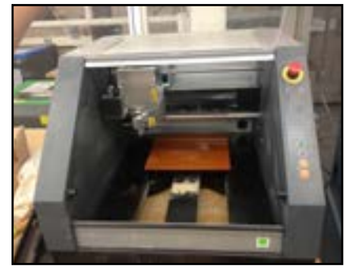


2. SRP Player Software

- Import geometry
- SRP Player mill settings
- Available bit sizes: .25” Ball /Flat
  .125” Ball/Flat
  Others can be purchased.
- Design tool motion to creat part

3. Preparing the Machine

- Starting the machine
- Anchoring material
- Installing endmills
- Setting the origin (X,Y and Z)



4. Run your job

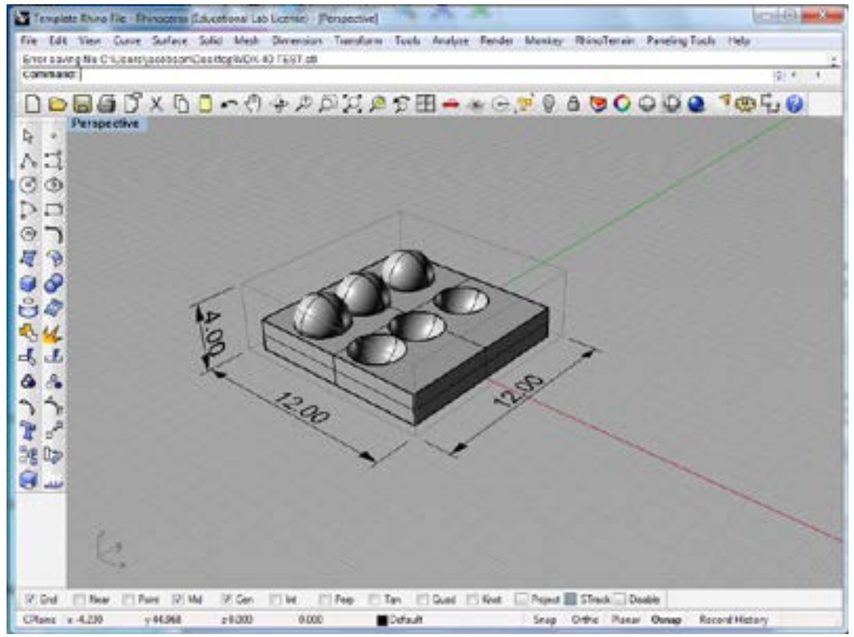
- VPanel controls
- Safety precautions + observations.
- Trouble-shooting
- Changing endmills

5. Extract Piece / Clean Up

- No compressed air, use a vacuum
- Shut down the machine

Detailed instructions follow...

Create Geometry



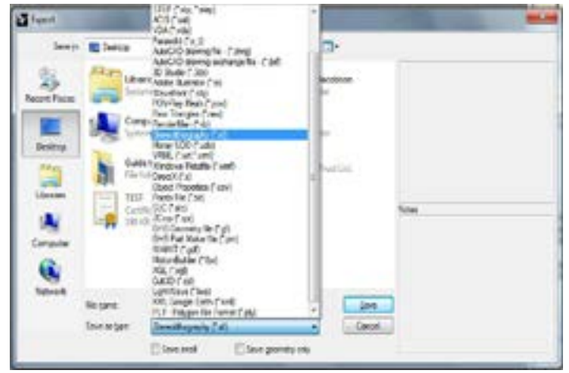
Steps:

1) **Create your geometry in Rhino**. The maximum size of anything that can be milled on the Roland MDX-40 is 4 tall (Z-axis)" x 12 wide (X-axis)" x 12 deep (Y-axis)". Note that the maximum pocket, hole or channel depth is limted by the dimensions of the cutting tools. On the Roland MDX 40-A, the maximum length of a 1/4" bit is 1.7", and the maximum length for a 1/8" bit is .9", so be aware that you will not be able to cut features deeper than this on the Roland MDX 40-A unless you cut from more than one side.

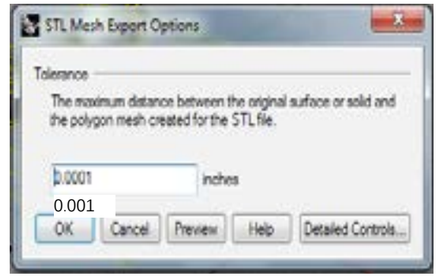
2) **Move to Origin.** The coordinate plane of Rhino's X,Y and Z axis correspond to the milling bed of the Roland MDX 40-A, where the point (0,0,0) is the center of the bed, the Z-axis points up, and X&Y are in the plane of the bed. Once you are satifsfied with your geometry, move it so that the bottom surface of your part it is centered on the origin, as pictured above.

3) **Export as .stl** SRP Player uses a file format called Stereolithography, which articulates solids and voids. It is best practice that the model is 'closed' when exporting as .stl. If you were exporting a cube, for example, it would have to have six sides that are all perfectly matched and touching without any gaps. A cube with 5-sides present and one side missing would b considered open. Select your geometry and type "Export" into the command line. You will be promted with the following three windows.

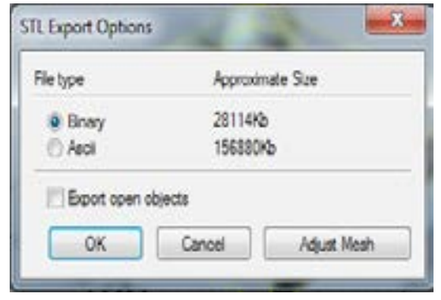
Exporting as Stereolithography (.stl)



Under "Save as type" select "Stereolithography (*.stl)."

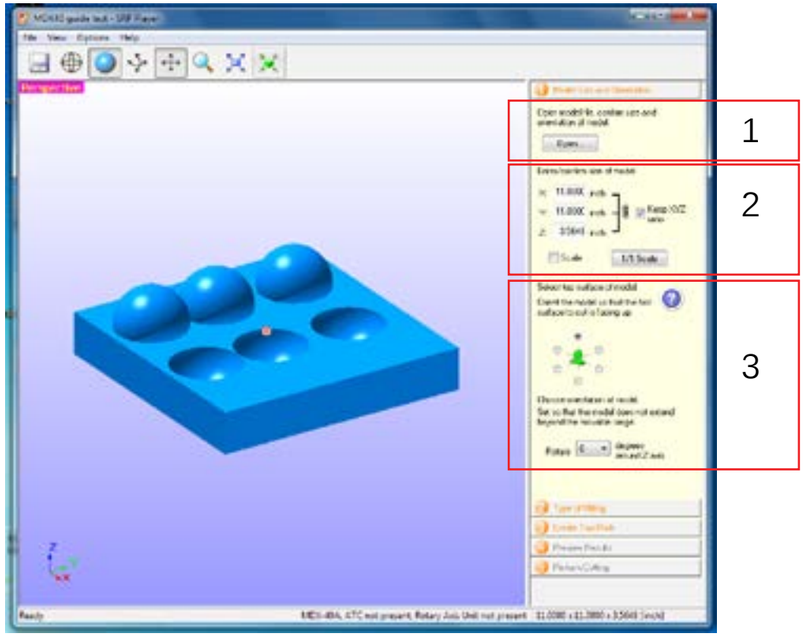


Set this to .001, so that your geometry is more precise than the machine.



Use binary. If you get errors resulting from your geometry in SRP Player, try unchecking "Export open objects".

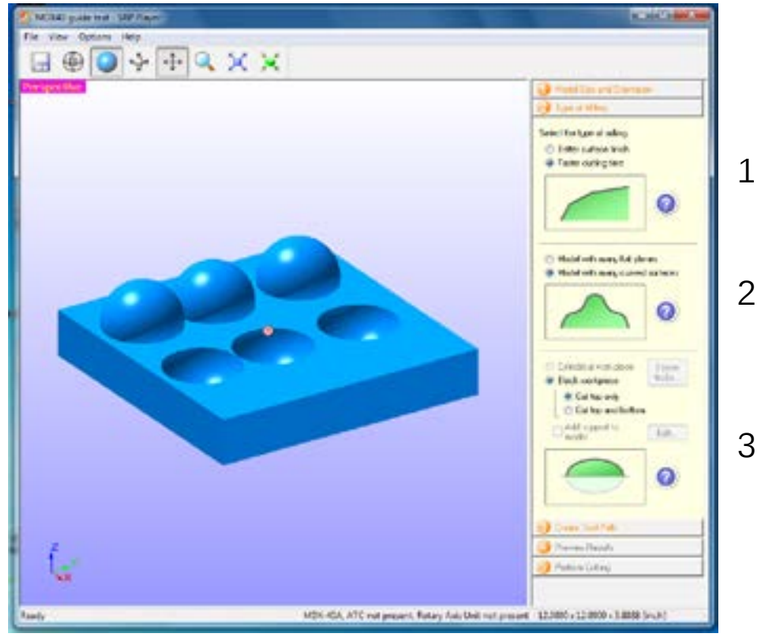
1 - SRP Player - Model Size + Orientation



Steps:

1) Open SRP Player software.
2) Open the .stl file you exported from Rhino.
3) Enter the scale at which you would like to mill this geometry. Note: if you would like to mill it exactly as you modeled it, either press 1/1 scale or enter in the dimensions of your geometry manually. If you need to mill all the way to the bounds of the material, you can reduce the amount of scrap material left around your part under Create Toolpaths > Edit > Roughing > Top Surface > Modeling Form
4) Typically this field can be left alone, but if you would like to reorient your material you can do so here. Note: this does not refer to orienting the direction of cutting, but rather the material in the bed.
5) Click ‘Type of Milling’ for next step.

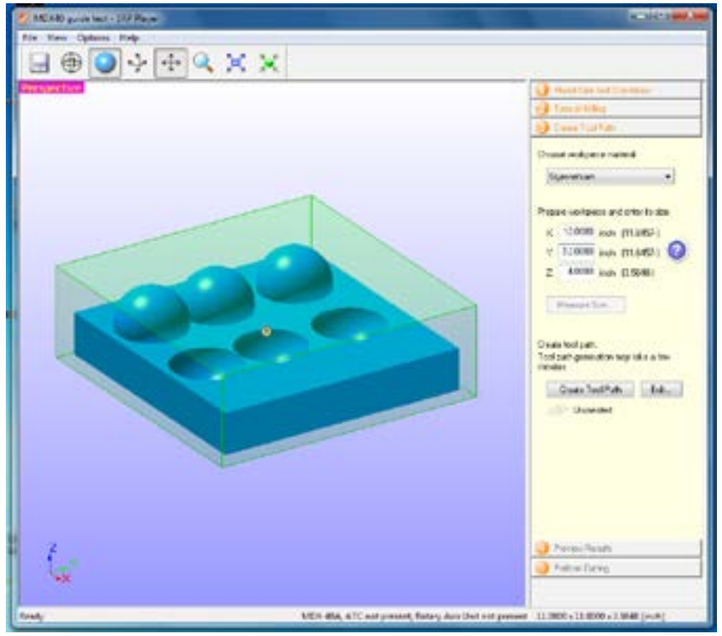
2 - SRP Player - Type of Milling



Steps:

1) Select for Quality vs Speed
2) Indicate the most prevalent characteristic of your geometry
3) For a single surface choose “Block workpiece” and “Cut top only.” “Cut top and Bottom” is used for flip-mills, or models milled on both the top and the bottom.
4) Click on ‘Create Tool Path’ to continue to next step.

3- SRP Player - Create Toolpath



1

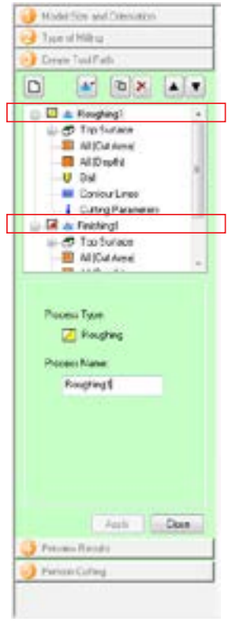
2

3

Steps:

1) Select your material from the drop-down list. If your material is not available, consult with a TA for selecting an appropriate substitute.
2) Enter in the size of your raw material. Your part should be parallel to the bed, and it will be centered in X and Y. Note: the maximum size is 12” (X Axis) x 12” (Y Axis) x 4” (Z Axis).
3) Clicking ‘Create Toolpath’ will instruct the SRP player to calculate the path that the tool should follow in order to cut out your desired geometry. For default settings, you can simply press “Create Toolpath”, however this often results in extremely long cut-times. If you would like to edit the settings on your toolpath, press “Edit” and proceed to the next page for guidance.

3.1 to 3.8 - SRP Player - Create Toolpath > Edit Toolpaths



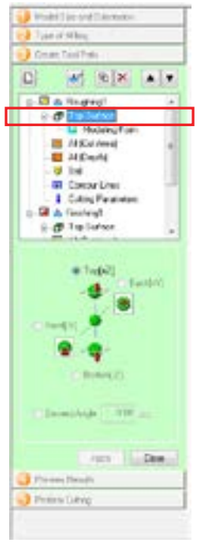
Here you can see and edit the details of the toolpaths SRP Player automatically generates. You must always run a roughing toolpath before you can run a finishing toolpath, but finishing tool paths need not cover your entire workpiece.

Roughing passes are intended to remove material as quickly as possible, but they do not leave as fine a finish. These passes cut as deep and as fast as is safe, according to the size of the tool and physical stiffness of the machine.

Finishing toolpaths are intended to control the quality of your finished piece. They remove a small portion of remaining material more slowly, leaving a finer cut. This is where you can finely control the finished appearance of your piece.

3.1 to 3.8 - SRP Player - Create Toolpath > Edit Toolpaths

3.1

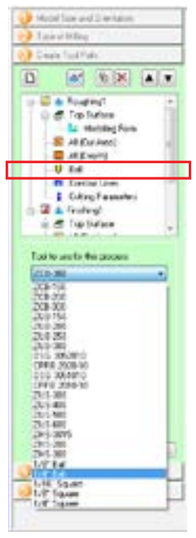


3.2

3.3

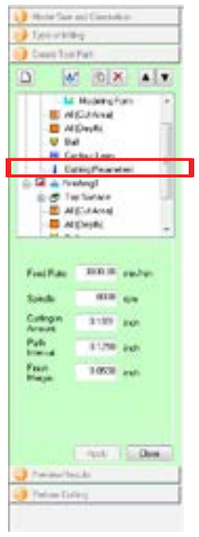
3.4

3.5



3.6

3.7



3.8

Each toolpath you create will have 8 sets of settings you can use to tailor the cutting. Repeat these steps for each toolpath you use.

3.1) **ROUGHING / FINISHING** assigns names to your toolpath, which may make organizing complex files easier. Note, you only need to run one roughing pass before beginning final passes.

3.2) **TOP OF SURFACE** is where you orient the tool relative to your part. Select “Top + Z” unless you want to cut into your piece at an oblique angle. Please consult with a TA if you would like to cut obliquely, this requires rotating the physical part on the bed.

3.3) **MODEL FORM** This is where you control the margins of scrap material SRP will leave around your part. If you intend to do a flip-mill (i.e. mill on both sides) this must remain at least .5”, and it may also be useful to create bounding walls for casting.

3.4) **ALL (CUT AREA)** These two fields are where you can instruct SRP to mill just a portion of your part. This can be useful for creating multiple finish passes with different surface qualities, or for milling only part of your stock material.

3.5) **ALL DEPTH** Here you can specify a maximum depth for your mill file. This is useful if you would like to specify different surface finishes for varying depths of your topography. Note that the depth is measured from the center of your material. If you want to cut one inch deep, in a two inch deep piece of material, you would set your starting depth to 1” and your ending depth to 0”, indicating you want it to go from 1” above the center to the center of your material. If you want your cut to go deeper than the center, you need to specify negative numbers, so cutting 1.25” into the material would mean that your start heigth is 1” and your ending height is -.25”.

3.6) **TOOL** Specify the tool size and the shape of the end. Currently, the Roland MDX 40-A is equipped with a .25” flat cutter, a .25” ball cutter, a .125” flat and .125” ball, all found in the black toolbox beside the machine. Note, you will need to manually change the bit and re-zero the machine for each tool change.

3.7) **CONTOUR LINE** Leave this setting on “up-cut” unless you’re using your own down-cutting endmills. Downcutting endmills can be useful for avoiding tear-out, or damage to the upper edge of the cut, however downcutting is not recommended for removing large amounts of material because it forces the chips into the material and they cannot be evacuated.

3.8) **CUTTING PARAMETERS** This is where you can control the speed at which the tool will cut through the material, as well as the amount of material it will remove with each pass. Refer to the chart in Appendix 1 of this document for Foam Cutting Parameters, and see Appendix 2 for Field Definitions.

4 - SRP Player - Preview Results

Maximum depth at which it is possible to cut into the material is defined by the distance the cutting bit sticks out of the collet.

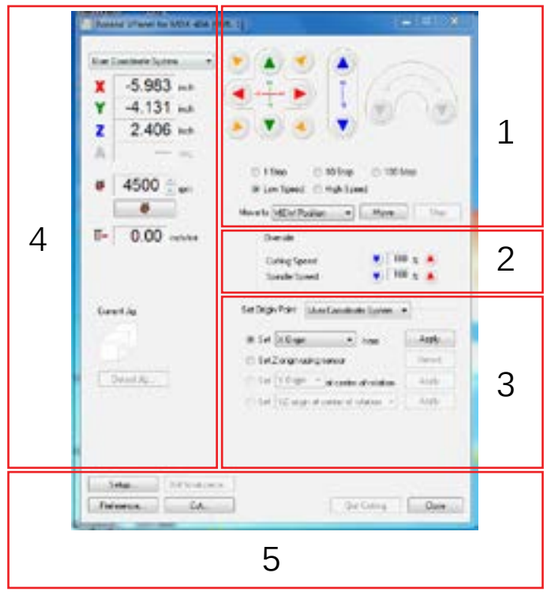
This is where you can see what the machine will cut, and estimate how long it will take. At this point it is your responsibility to ensure that you are not telling the machine to cut deeper than it is able.

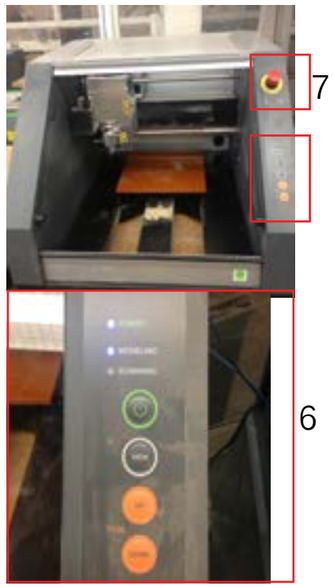
Steps:

1) Press “Preview Cutting” and wait for SRP Player to simulate your job.
2) As you watch the simulation, consider the deepest parts of your model. Note that the depth of vertical walls and deep pockets is limited by the length of the bit. The maximum depth at which a ¼” diameter tool can cut without the gantry colliding with your material is 1.7” and the maximum depth at which a 1/8” tool can cut without such a collision is .9” **It is your responsibility to ensure that the bit is at least as far out of the collet as your specified depth**. You need to check this value in order to prevent a collision between the collet or gantry and the material to be milled. Be aware that this is different from the amount of material the tool can cut in one pass-- that’s referred to as the “step-down” and it is determined by the flute length and chip load given a specific material.
3) If you need to reduce the depth of your model you can do so by either a) rescaling your geometry (under Model Size and Orientation), or b) specifying that the Roland MDX 40-A ignore the lower part of your model in under Tool Path Depth (see Step 3.5).
4) If you need to reduce the amount of time it is taking to cut your model, consider using a larger bit in step 3.6, a larger step-over for your Finish Passes in step 3.8, selecting a faster cutting time over quality in step 2, or (if you are using multiple finish-passes) eliminating redundant finishing passes in step 3.3.

Special Note: It is possible to use MasterCAM to generate code, which may increase the efficiency and/or allow more flexibility in determining tool lengths.

4.1 - 4.3 VPanel - Prepare for cutting



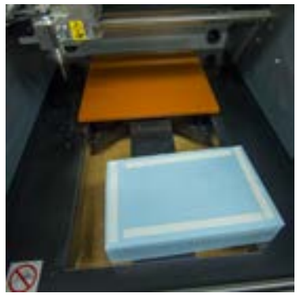
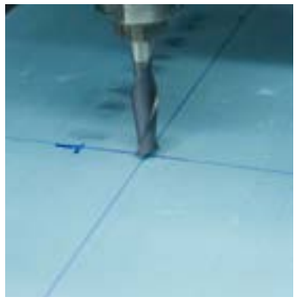


VPanel is the software that allows you to interact with the mill. You’ll need to use it as you fasten your material, install your endmill and “home” (define the origin) the axes.

Steps:

1) **Moving the gantry**. The red arrows will move in X (left-right), the green arrows will move in Y (back-forth). The movement in Y is inverted because the X axis is stationary, so move the Y axis as if you wanted to move the bit around your material. The blue arrows control movement in Z (up-down), these controls are duplicated in solid orange on the front of the machine itself. As you move you can control the speed using the step sizes. Use caution when moving at high speed, because if you collide with anything you might damage the bit, your material or the machine and you will need to re-zero the X, Y and Z axes.
2) **Overrides** allow you to adjust the spindle speed and feed rate after you’ve begun milling. The machine’s maximum spindle speed is 8000 rpm, and the maximum feedrate is 120 inches per minute, or 3000 mm/min. Overrides are also limited to 100% of the maximum specified in the job, so you can easily reduce speed, but you cannot increase speed at this stage.
3) **Origin Controls** are for communicating where the machine should place the origin, or the point (0,0,0). Use the top of your stock as zero for Z, and the center of your stock for zero in X-Y.
4) **Readout** will tell you where the tool tip is, according to the location your provide as the origin. It will also tell you how fast the bit is spinning (rotations per minute) and how fast it is moving (inches or mm per minute).
5) **Send to Cut** is where you access preferences, and tell the machine to start or stop cutting.
6) **MDX-40A controls** allow you to pause the job mid-way. Press “View” to do this, the spindle will stop and the gantry will retract. If you press and hold “View”, it will resume cutting. Green is the power button, and the orange buttons allow you to move the spindle up and down
7) **Emergency Stop** will stop the machine immediately and cause you to lose all work you’ve done so far. It is not possible to resume a job after an emergency stop.

4.1 - Fastening Material

Anchoring refers to the process of holding your material in place so the mill can begin cutting.

Steps:

1) **Mark the material** prior to mounting. make a mark on top of your material at the center that you can use to zero the X-Y Axes.
2) **Tape the material.** Double-stick tape your material to the bed of the machine (pictured in orange, above center). Be sure to cover all four edges of your material securely enough that it will not loosen during cutting.
3) **Mount material to the bed.** Be sure to mount your material onto the center of the bed, parallel to the edge in the same orientation in which you modeled it. The Y Axis goes from front to back, and the X Axis goes from right to left, refer to your Rhino Model if necessary.

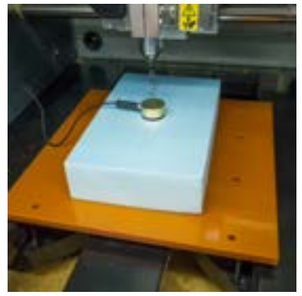
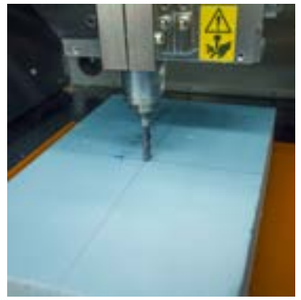
4.2 - Install/Change Cutting Tool



Steps:

1) **Choose Cutting Tool.** Install the tool you intend to cut with first. In most cases this will be either a 1/4" Flat or a 1/4" Round, because these are most often used for roughing.
2) **Place the bit in the appropriate collet.** There are two collet sizes, 1/8" and 1/4"
3) **Measure** how far you need the cutting tool to protrude from the collet, according to your desired geometry and maximum cutting depth.
4) **Finger tighten** the collet into the spindle. Ensure that the cutting flutes are not inside the collet, because tightening the collet on an area of the tool meant for cutting will damage the tool and the collet.
5) **Fully tigthen** the collet firmly into the gantry with two wrenches (pictured above, right). One prevents the spindle from rotating (top) and the bottom tightens the collet.
6) Repeat this for each tool change when prompted.

4.3 - Setting Origin in Z Axis + X-Y Axes

Setting the origin is how we communicate to the machine where (0,0,0) is located. Without this step, the machine will not know where to begin cutting. We will set the origin in Z seperately from in X & Y.

Steps for Setting the Origin in the Z-Axis:

1) **Connect Z-Sensor.** We store the sensor in the black toolbox that sits beside the machine. The sensor sends a signal to the machine when the cutting tool has touched its surface. Place the sensor in the bed and feed the cable through the black flap-port in the rear left of the machine. Pull the cable up gently and plug it into the port called 'Z0 Sensor' (pictured above left)
2) **Place the sensor** on top of your material (pictured above center).
3) **Position Mill**. Ensuring that the cutting tool is above the tallest part of your material before moving it, use the red and green X-Y controls in VPanel to position the bit directly above the center of the detector, or place the detector beneath the center of the bit.
4) **Detect.** Move the tool down until it is about .25" or 7 mm above the sensor, and then, in VPanel, select the blue bullet for "Set Z origin using sensor" and then press the button for "Detect". VPanel will ask you to confirm that the sensor is in place. Once you press "OK" , the machine will being to slowly move down in the Z direction until the tool tip touches the sensor. Your Z-zero point will be set at the bottom of the sensor, which should be at the top of your material. If you do not place the sensor beneath the tool, the MDX 40-A will lower the tool until it hits something and meets resistance, possibly damaging the tool or the material. After touching it will retract.
5) **Unplug** the sensor from the rear of the machine, remove it from the bed and place it back in the case.

Steps for Setting the Origin in the X-Y Axes:

1) **Measure** to the center of the material with a ruler, make a mark and place your material so that it s parallel to the edges of the bed (pictured above right).
2) **Navigate** using the red and green x-y arrows in VPanel to the center of your material, lower the bit so that it is within 1/4" of the top of your stock and you can easily see that it is centered, the tool center should be above the point you have drawn.
3) **Confirm**. Under 'Set' in Set origin Point click XY origin, and press "Set" when you have the tool centered on your material / point (picture above right).

5- SRP Player - Peform Cutting

We are now ready to begin the milling process. Ensure that the MDX lid is closed and that the workspace inside is clear of any tools or debris.

Steps:

1) **Press Start Cutting in SRP Player.**
2) **Confirm Tool** the first tool you use will almost always be a 1/4” Flat or 1/4” Round for roughing. If you do not see the tool you intend to use first here, press cancel and return to step 3.6 where we selected the bit. Confirm each of your toolpaths use the tool you intend and that the desired tool is actually installed in the machine.
3) **Set Surface** here you should select the top surface of the material, because we placed the z-sensor on the top of our material when we zeroed the Z-axis. It is also ok to set the z-sensor on the orange bed of the MDX, however you must then also check the bottom surface here to be consistent (otherwise the cutting tool will plunge into the table and cause damage to the tool, material and the machine).
4) **Proceed with mill.** Place your hand over the stop button and watch for problems.
5) Await instructions for changing tools. Refer to step 4.2 if necessary, but be certain that you are leaving enough of the bit out of the collet for your job.

Note: If the computer shuts off, you will lose the progress on your job.

Appendix 1: Foam Cutting Parameters

Type of toolpath	Roughing	Finish	Finish
Size of Bit	1/4”	1/4”	1/8”
Feed Rate	3000 mm/min (120 in/min)	3000 mm/min (120 in/min)	3000 mm/min (120 in/min)
Spindle Speed	8000 rpm	8000 rpm	8000 rpm
Cutting-in Amount	MAX .4”	.1”	.0625”
Path Interval	.18”	.125” to .05”	.0625” to .01”
Finish Margin	.05”	.05”	.05”

Appendix 2: Field Definitions

Cutting Speed

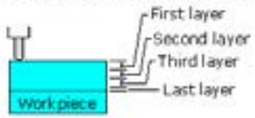
Enter the movement speeds (feed rates) for the tool during cutting.

Spindle

Enter the spindle rotating speed.

Cutting-in Amount

Enter the depth to which the tool cuts into the workpiece.
Cutting at the cutting-in depth is repeated as many times as necessary to achieve the target depth.
This setting cannot be made for finishing, except when [Contour lines + Zcut lines] has been selected as the path type.



Path Interval

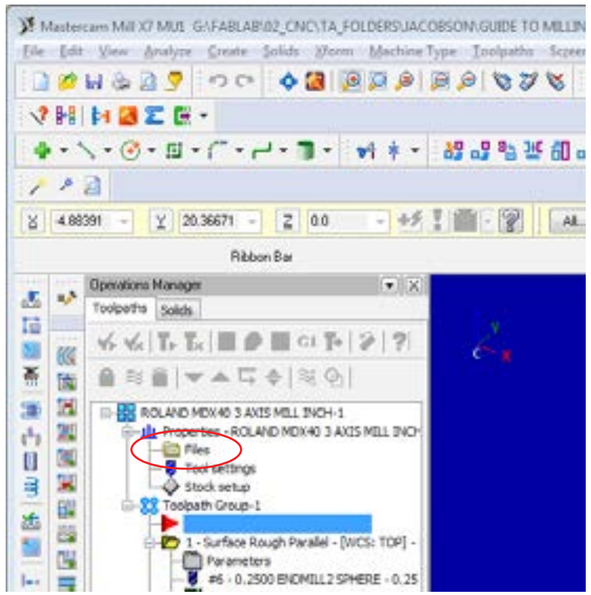
Enter the interval between adjacent paths.
You should generally make this narrower than the tool diameter.

Finish Margin

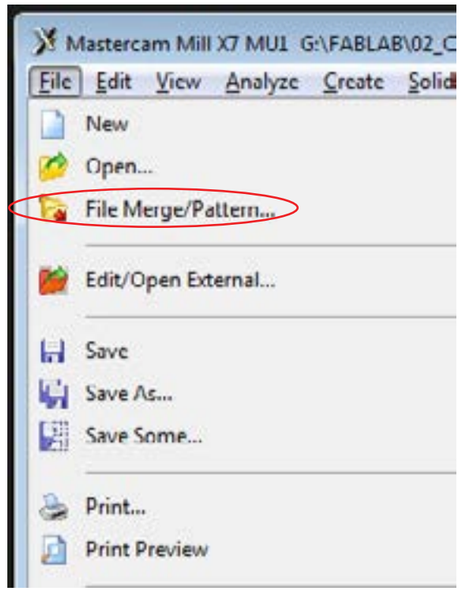
Enter the thickness of the remaining material surrounding the model.

Appendix 1: Posting from Mastercam: The Roland Template

2



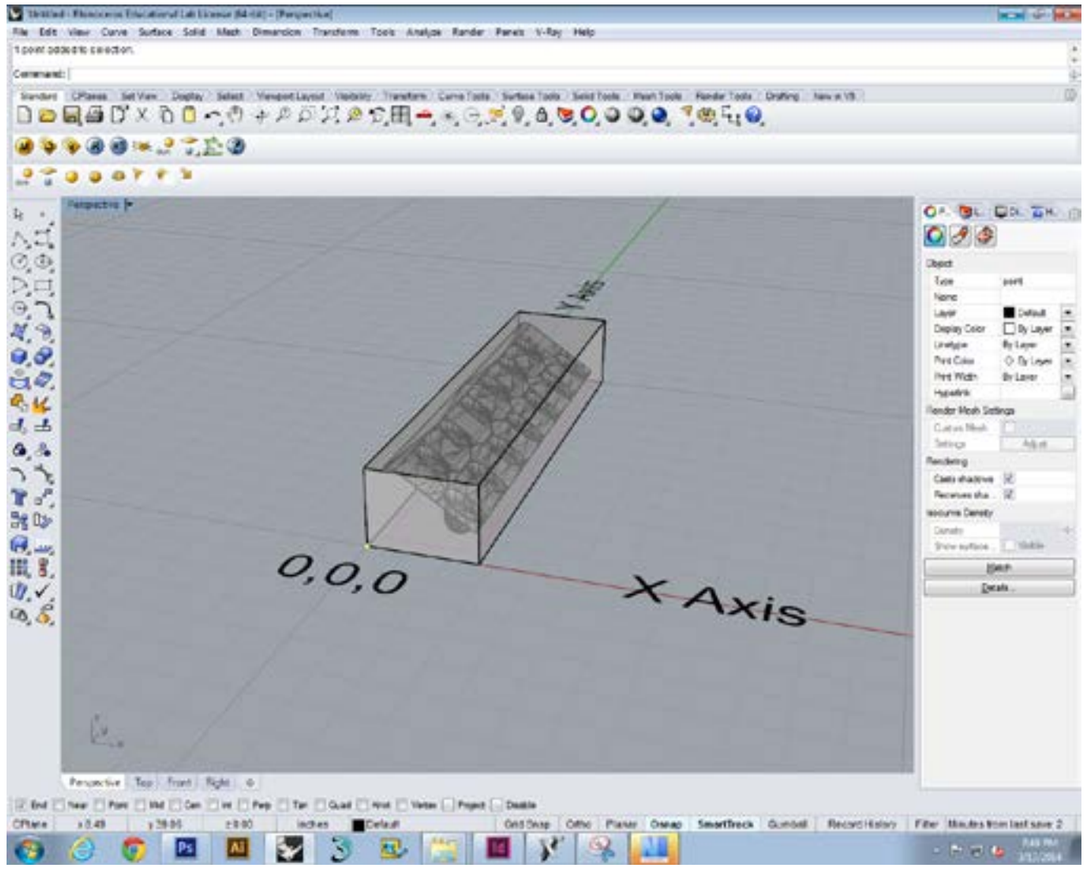
3



Mastercam provides more precise control over your milling job, however the process is not fully automated. The first step is to prepare toolpaths in mastercam for output to code that the Roland MDX 40-A can interpret.

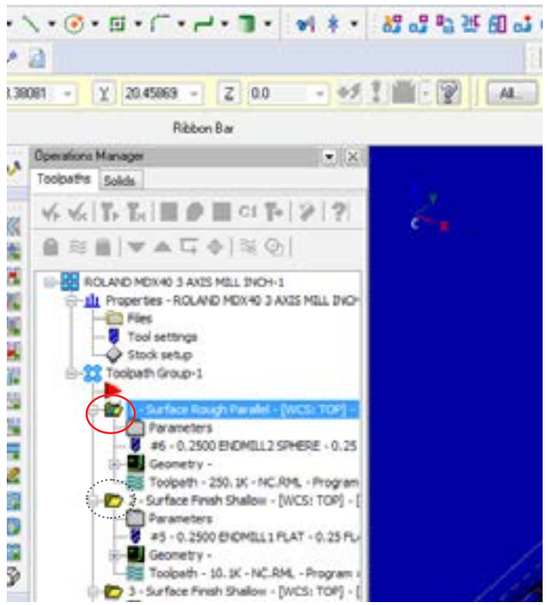
Steps:

1) Open the template Mastercam 7 file for the Roland MDX 40-A.
2) Ensure that the machine definition and posting protocol for the Roland MDX 40A have been loaded into the file. (Right-click 'Files' under the Properties tab of the Operation Manager and ensure C:\Users\Public\Documents\shared mcamx7\CNC_MACHINES\ROLAND MDX40 3 AXIS MILL INCH-1 is selected)
3) Import geometry into the model (File > "File Merge/Pattern" > Browse to your geometry). Your Geometry should be located in positive Z, positive X and positive Y. When in Rhino's TOP View, with Y positive facing UP and X positive facing RIGHT, locate the lower left hand coorner of the bounding box representing your material at the origin, with its lower face at 0 in the Z-direction. See image on next page.
4) Apply toolpaths to your geometry. Detailed tutorials for this process are available on the CNC tools main page, linked from http://www.gsd.harvard.edu/#/gsd-resources/fabrication-laboratory/
5) Post the NC Code, one fil

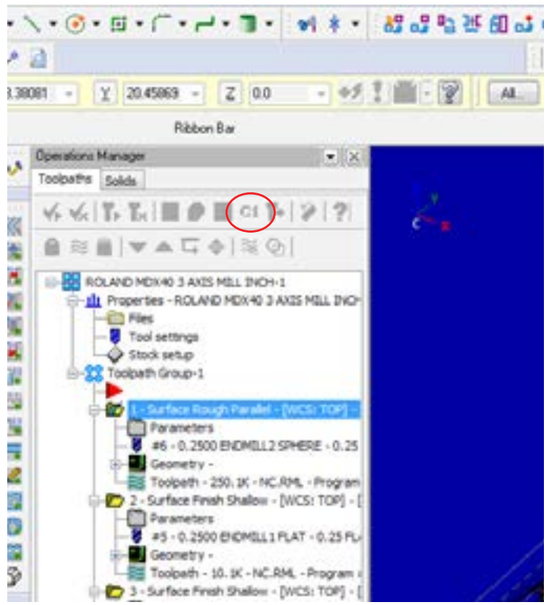


Exporting Toolpaths from Mastercam

1



2

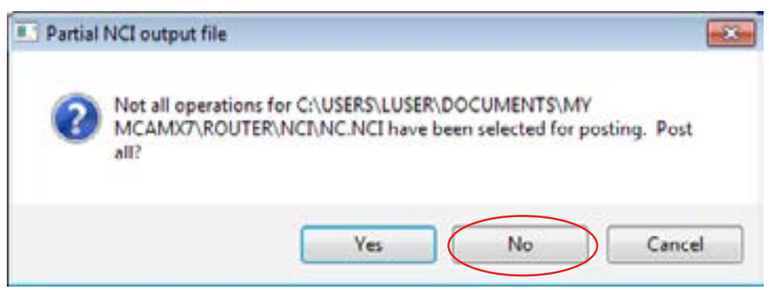


Now that we have prepared toolpaths, it is necessary to post them to G-code, which contains information about spindle speed, and movement. The the Roland MX40-A cannot automatically change tools, so it is necessary to post the toolpaths one at a time.

Steps:

1) **Select the first toolpath**. TA green check will appear after you have single-clicked on the name of the toolpath. Note the lack of a green check box on the other toolpaths (indicated with dotted lines).

2) Click on the icon for ‘G1’, which is for posting the toolpaths to NC-Code. You will receive a warning indicating that you have not selected all the toolpaths. This is normal, click No.



3) Repeat this step for all toolpaths in your file. Name the toolpaths with numbers indicating the sequence in which they will be milled, and the size of the bit. For example: “Path1_quarter inch”

Milling on the MDX 40-A

IMage of USB

Image of Power Button

With the NC files in hand, its now time to mill. Here we will walk through the process of starting the machine, and fastening your material to the bed.

Steps:

1) **Turn on the computer**. Sometimes the computer will have trouble booting if the Roland MDX 40-A is connected, simply unplug the USB at the base of the machine for startup, and plug it in again once you see the Windows Desktop.

2) **Turn on the Roland MDX 40-A.** First ensure that there is no material in the bed because the startup routine of the machine can cause a collision. Press the green Power Button on the Front of the machine.

3) **Install the tool for the first toolpath.** Each tool change will need to be done mannually. Install the tool that your first toolpath uses, ensuring that the length of the tool protruding from the collet is in line with the assumptions you made in Mastercam when programming the toolpaths.

Setting the Origin

Image of setting Z in machine

Image setting X-Y

Image of setting Z in software

Image setting X-Y in software

With your material fastened, its now necessary to instruct the Roland where 0,0,0 is relative to your part.

1) **Set the Z Origin.** , First, set the origin in Z, by moving the spindle away from your part, and lowering it to touch the bed. Use fast moves until you about 1/2" above the orange bed, then use slow moves until you are about 1/8" above the bed. Place a slip of paper on the bed and slide it back and forth as you incrementally lower the tool the rest of the way down to the bed, stop as soon as the tool touches the paper. With the tool in this position, select "Set Z Origin' in VPanel.

2) **Set the X-Y Origin.** Lift the spindle in the Z direction, and reposition it above your material, on the left hand corner closest to the door to the machine. With the tool in this position, select 'Set X-Y origin' in VPanel.

Running the Mill

Image for Sending NC code

Set the Machine in NC code mode

With the origin set, all that remains is to send the tool paths one at a time, and chang the tool when necessary.

Steps:

1) **Select 'Run' From inside the Vpanel Window,**
2) Load the first one of your toolpaths.
3) Press 'Cut'.
4) Keep watching the part through the glass, being prepared to stop the milling if anything unexpected happens.
5) When the toolpath completes, refer to 4.2 for changing the tool, and repeat the above steps for each toolpath.

Cleaning Up

Image Vacuuming Chips

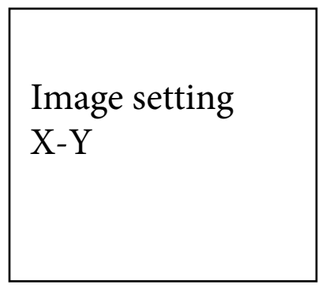


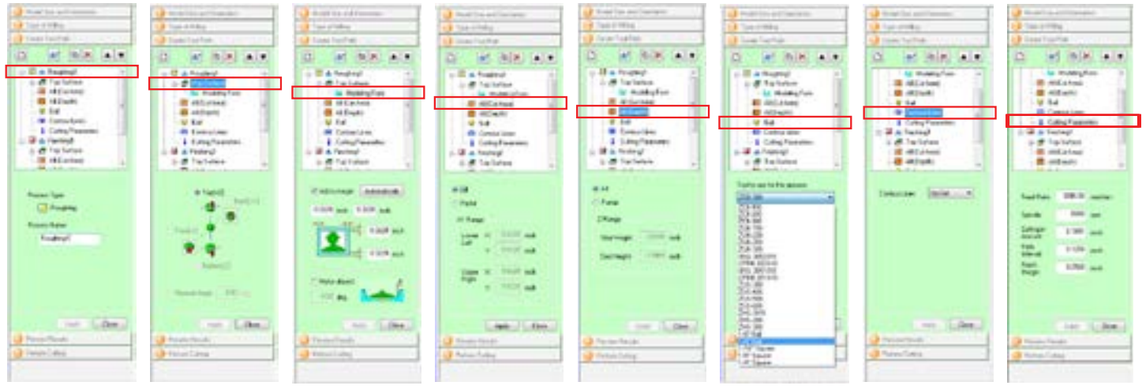
Image setting X-Y

Image of setting Z in software

Image setting X-Y in software

Please clean up the workspace after yourself by completing the following tasks.

1) Vacuum the bed. Do not use an airhose, as this may lodge debris in areas of the Roland MDX 40-A that will prevent it from operating properly.
2) Remove double-stick tape from the bed
3) Return all tools to their plastic sleeves and place them in the toolbox
4) Place the toolbox on top of the Roland MDX 40-A.
5) Shut down the computer.



1 2 3 4 5 6 7 8

Each toolpath you create will have seven sets of settings you can use to tailor the cutting. Repeat these steps for each toolpath you intend to use.

Steps:

1) Here you can assign names to your toolpath, which may make organizing complex files easier. Note, you only need to run one roughing pass before beginning final passes. If you would like to create multiple finish passes, duplicate one of the finishing passes and adjust the settings for each.
2) Here you orient the tool relative to your part. Select Top + Z unless you want to cut into your piece at an oblique angle, and please consult with a TA if you would like to cut obliquely.
3) This is where you control the margins of scrap material SRC will leave around your part. If you intend to do a flip-mill (i.e. mill on both sides) this must remain at least .5", but if you intend to mill just a single side it can be reduced to zero.
4) If you would only like to cut part of your material, here is where you define those bounds. This can be used for creating multiple finish passes with different surface qualities, or for milling only part of your stock material.
5) Here you can specify a maximum depth for your mill file. This is useful if you would like to specify different surface finishes for varying depths of your topography.
6) Specify the bit size. Currently, the Roland is equipped with a .25" flat cutter, a .25" ball cutter, a .125" flat and .125" ball. Note, you will need to manually change the bit and re-zero the machine for each tool change.
7) Leave this setting on up-cut.
8) This is where you can control how fast the machine cuts and how aggressively it removes material. Refer to the chart in the Appendix:Foam Cutting Parameters for filling in these settings.

COTNENTS:

TOPOGRAPHY MODELS:

- Size & Time considerations + planning the job
    - When to talk to a TA, expected scheduling
    - Maximum Dimension: 4’ x 8’ x 6” tall
    - Small: 6” x 12” x 2” (or whatever the Roland is)
    - Large: more than 6” x 12” x 2” up to a maximum of 4’ x 8’ x 6”
- Material choice + considerations
    - Wood
        - Plywood
        - Single piece of solid hardwood
        - Laminated solid wood
    - Foam
        - Blue Foam
        - White foam
        - High Density Foam
- Anchoring strategies

- Desired Geometry + Appropriate Milling Strategies
    - General overview of advantages to milling + limits of milling
    - Tooling considerations with regard to desired geometry
    - Buildings:
        - Pockets for inserting buildings
        - Solid Buildings integrated into topography
            - Overall strategy
            - Height limitations + relationship with the bit
            - Distance between buildings
            - Pocket Strategy
            - Rough Parallel Strategy
- Sloped topography
    - Smooth continuous surfaces
    - Stepped topography
    - Surface Articulation

- Recessed roads
    - Up-cutting + downcutting
- Contours and marking lines or points on the surface
- Holes for trees/inserting pins
    - Peck Drilling
    - Size + time considerations
    -

FLIP MILL

- Maximum size: 4’ x 8’ x 6” tall for your entire end piece
    - BUT, must be supported throughout
- Anchoring Strategy
- Registration Holes
- Multiple World Axes in Master Cam
- Tabs
- Thickness of resulting Geometry
- Tolerances
- Vibration + part stability
- Part removal

2D CUTOUTS IN SHEET MATERIAL
- (maximum 4 ‘ x 8’ x 1.5” tall)
- Tabs
- Size + Depth limitations
- Types of cutting bits
    - Up-cutting
    - Down-cutting
    - Material Specific
    - Engraving Bits
    -

MOLD FOR CASTING
- Size & time considerations + planning the job
    - When to talk to a TA, expected scheduling
    - Mill the positive, or the negative?
    - Maximum Dimension: 4’ x 8’ x 6” tall for each half of the mold)
    - Small: 6” x 12” x 2” (or whatever the Roland is)
    - Large: more than 6” x 12” x 2” up to a maximum of 4’ x 8’ x 6”
- Material choice + considerations
    - Wood
        - Plywood
        - Single piece of solid hardwood
        - Laminated solid wood
    - Foam
        - Blue Foam
        - White foam
        - High Density Foam
    - Wax, plastic or other mold making material
- Anchoring strategies

Desired Geometry + Appropriate Milling Strategies
- General overview of advantages to milling + limits of milling
- Tooling considerations with regard to desired geometry
- Female Molds - Surface milling
- Male Molds - Flip Milling (see next section)
- Registration Holes
- Grooves
    - Size + Depth limitations
    - Ramping
    - Speed Considerations
- Surface Geometry

    - Steep slopes
    - Deep holes
    - Surface Patterning
        - Flowline
        - Constant Scallop
        - Parallel
        - Radial
        -
-

Robotic Arm / 6-Axis Jobs

- When to consider the Robot
- Tolerances, Precision + expectations in planning the job
- Assembling readymade pieces
- Fiber-winding
- Circular Saw
-
- Size & time considerations + planning the job
- Material choice + considerations
- Anchoring strategies

Desired Geometry + Appropriate Milling Strategies

- Grasshopper + HAL -
-

GUIDE TO TOOL PATHS

BY TOOLPATH - Don't know what that toolpath does? Look here.

BY DESIRED GEOMETRY - Want to articulate a form, but don't know how? Look here.

SURFACES

GROOVES

POCKETS

HOLES

ABB 6 - Axis Small Robot

6 Axis

ABB 6- Axis Big Robot

6 Axis