

Gem**vision**



> Instruction Manual

Revo540

Multiple-Axis Milling System

Revo540

Multiple-Axis Milling System

> *Instruction Manual*

© 2005 Gemvision Corporation

Table of Contents:

Chapter 1: Features, Fixtures, and Accessories

About the Revo 540B Crate	1
Crate Disassembly	1
Reassembling Crate for Shipping	2
Technical Specifications	3
Revo 540B Tool Tray Assembly	4
Diagram of Parts	5
Revo 540B Milling Fixtures	7

Chapter 2: Assembling the Revo 540B Mill

Attaching the Spindle Control Unit	9
Installing the Mill Computer	10
Connecting the Tool Sensor	11
Attaching the Coolant-Dispenser	11
Assembling the Coolant Container	11
Assembling the Coolant Tank for Extended and Multiflip Fixtures	12

Chapter 3: Maintaining the Revo 540B Mill

Lubricating the Mill	14
Calibrating the Mill	15
Aligning the mill with Manual Offsets	16

Chapter 4: Preparing to Mill

Using the Revo Software	20
Loading a Revo file	20
Determining Milling Time	22
Controlling the G-Code	22

Stopping the Mill	22
Changing the Mill Speed	23
Checking the Mill Status	23
Machine Position	24
Seek Home	25
Jog Controls	25
Installing a Cutter	26
Seek Tool	27
Mill Control	28

Chapter 5: Using RevoCam

About the Software	29
Starting up RevoCam	29
About the Cutter Library	30
Edit the Cutter Library	31
Selecting a Strategy	32

Chapter 6: Preparing & Milling the Rotary One-Sided Strategy

About this Strategy	39
Preparing the Model in Matrix	39
Using the RevoCam Software	39
Understanding the Toolpath	42
Preparing the Mill	44
Using the Three-Jaw Chuck	44
Set & Show X Offset	45
Running more than one rotary ring on a single wax tube	48
4 Flats on Rotary	49
Understanding the 4 Flats on Rotary Toolpath	52

Setting up this Strategy on the Rotary Fixture	52	Assembling the Fixture	97
Chapter 7: Rotary Plus One & Flat on Rotary Plus One Strategies	53	Aligning the rotary fixture	99
Comparing the Two Strategies	53	3-Sided Auto-Flip Plus Rotary	101
Preparing the Model in RevoCam	54	Chapter 11: Advanced Options	106
Understanding the Toolpaths	58	An Overview	106
Assembling the Fixture	59	How to Use Advanced Options	109
Chapter 8: Preparing & Milling the Two-Sided Auto-Flip Strategy	61	Adjusting settings for conventional paths	109
Preparing the Model in Rhino	61	About the Path Type Menu	111
Using the RevoCam Software	62	Adding More Toolpaths	112
Understanding the Toolpath	69	Repeat a Conventional Milling Operation with Advanced Options	113
Preparing the Mill	70	To add an Advanced Milling Operation	113
Assembling Extended Fixture	71	Flat on Rotary	113
Assembling Multiflip Fixture	73	Follow Curve on Rotary	115
Set-up of Multiflip in Revo	73	Follow Curve on Flat	115
Running 3-Sided AF on Multiflip	75	Make & Save RVO File	115
Chapter 9: Preparing & Milling the Auto-Flip Flat Milling (3-Sided) Strategy	76	Custom Borders, Trimmed Toolpaths	116
About This Strategy	76	Freeform Borders w/ Flat Milling	117
Preparing the Model in Rhino	77	Trim / Custom Borders w/ Rotary	117
Using the RevoCam Software	78	Chapter 11: Tutorials Using Advanced Options	119
Understanding the Toolpath	85	I. Rotary Ring	119
Assembling the Fixture	86	II. Auto Flip Plus Rotary	123
Chapter 10: Preparing & Milling the Auto-Flip Plus Rotary Strategy	88	III. Auto Flip Flat Milling (2 Sides)	126
About this Strategy	88	IV. Auto Flip Flat Milling (3 Sides)	128
Preparing the Model in Rhino	88	V. Freeform Borders, 3-sided Auto Flip	132
Understanding the Toolpath	96	VI. Custom Borders & Trim Toolpath, Auto-Flip Plus Rotary	135

Chapter 1: Features, Fixtures, & Accessories

About the Revo540B Crate:

It is strongly recommended to store the REVO540 crate for 4 to 6 weeks in case the mill should need to be returned for troubleshooting. Instructions for reassembling the mill crate are found below. To store the crate, lay the boards flat on a level surface in a dry location. Crate boards will warp if stored in a moist, unlevel environment. Please disassemble and reassemble the crate and remove contents in accordance with the directions below to ensure the reusability of the crate.

Crate Disassembly:

1. Use a power screwdriver to remove all eight screws from crate lid (two on each side.)



2. Remove lid from crate.



3. Unload mill components from crate shelf.



4. Remove the shelf by grasping the edge of the center cut hole and pulling up/outward.



5. Use a power screwdriver to remove all screws from crate walls.



6. Use scissors to cut zip ties in front and rear of mill.

Front: (two zip ties through handle)



Rear: (four zip ties through handles)



The Revo540 mill is now ready to be removed from the crate.



Re-Assembling Crate for Shipment:

1. Push zip ties down through the holes cut in the bottom of the pallet base of the crate, then bend and push tie back through.

2. Repeat step one for all six pairs of zip tie holes.



3. Place mill so that base handles align with corresponding zip ties.

4. Thread ties through handles. (Two zip ties through front handle; four zip ties through rear handles.)

5. Fasten zip ties to mill handles and pull tightly to secure.

6. Cut off excess zip tie.

7. Align two side walls with crate base.



8. Drive three screws along the base of each wall. (Power screwdriver recommended)

9. Align the third wall and drive three screws along the base of the wall.

10. Drive two more screws about half way up each edge of back wall, securing the back wall to the side walls.



11. Align fourth side and drill screws according to instructions in steps 8 - 10.

12. Set crate shelf above mill on top of the crate wall boards. (Do not drive screws into the shelf!)



13. Place mill components on the shelf. These include the computer, monitor, attachment boxes, and the Rev0540 manual.

14. Place crate lid on top of crate, covering all items. Drive two screws into each side of the lid.



Technical Specifications and Industry Comparisons:

Travel X,Y, Z (Work Envelope): X = 172mm (6.7”), Y = 110mm (4.3), Z = 200mm (7.8”). This is the maximum travel range of each axis. ***NOTE:** Actual useable work envelope is reduced by the space occupied by fixturing.*

Table Size: 5.0” (127mm) x 16.75” (425.5mm). This is the actual size of the machine table surface. Revo’s Main Fixture Plate, which mounts on the table, measures 7.25” (184.2mm) x 16.75” (425.5mm).

Max Cutting Feed Rate: 1200mm/Min. This is the maximum feed rate possible regardless of cutting tool capabilities or part geometry. This will be limited by the slowest axis involved in multi-axis moves.

Lead Screw Type: Ball Screw – 15.24mm (0.600”) diameter. Industry offerings: Ball Screw, Acme Screw, 60 Deg. “Vee” Thread, Acme w/Zero Backlash Nut, 60 Deg. “Vee” Thread w/Zero Backlash Nut.

Drive Motors Type: Stepper Motor - Size 23 (56mm) – 3.0A 2.1V. Industry offerings: Stepper and Servo Motors.

Note: Linear Motors are becoming increasingly common on full scale machine tools and will probably find their way into Desktop machine designs in the future.

Way Type: Linear Rail Ball Ways (12mm diameter) Industry offerings: Dove Tail, Linear Slide (Plain type), Linear Slide (Ball Bearing Type), Linear Ball Way, Linear Slide (Sleeve Type).

Accuracy: +/- 0.05mm (0.002”) - How close the machine can move to an exact coordinate position.

Resolution: 0.004mm (0.00015”) - The finest increment of movement of which the machine is capable.

Repeatability: +/- 0.02mm (0.0007”) - How accurately the machine can return to a previous position.

Max Table Load: Currently Unknown.

Tool Holding Arrangement: Collet (NSK). Industry offerings: Collet, R-8, Morse Taper, and various proprietary taper shank tool holders.

Spindle Drive Type: Direct Drive. Industry offerings: Belt Drive, and Direct Drive.

Spindle Drive Motor Type: DC Variable Speed. Industry offerings: Step Pulley (belt drive), DC Variable Speed, AC Variable Speed. ***NOTE:** The AC drives are brush type motors and DC drives are brushless. Typically AC motors are significantly noisier and run hotter than DC motors.*

Spindle Speed Range: 0 to 25,000 RPM. Other Industry Spindle Speed Offerings: 200 to 2,500, 100 – 6,000, 350 to 3,500, 750 to 24,000, 0 to 25,000, 0 to 40,000 and 7,000 to 70,000. ***NOTE:** Step Pulley arrangements offer about 6 speeds which are not infinitely variable.*

Spindle HP: 105W (0.14 HP); Industry offerings: 0.33HP (253W) to 2.95HP (2270W).

Power Requirements: 3A @ 120V – or 360VA/216W – Minimum UPS 260W/450VA – 3 to 5 minutes back up.

Weight: 180Lbs. (82kg) machine only.

Shipping Weight: 380Lbs. (175kg) Industry range: 37Lbs. (17Kg) to 1210Lbs. (549Kg)

Class Type: Rapid Prototyping; Industry Class offerings: Hobby, Educational, RP, and Industrial.

Machine Only Dimensions: 41” W x 26” D x 36H

Minimum Recommended Bench Top Space Requirements: 29” Deep x 45” long.

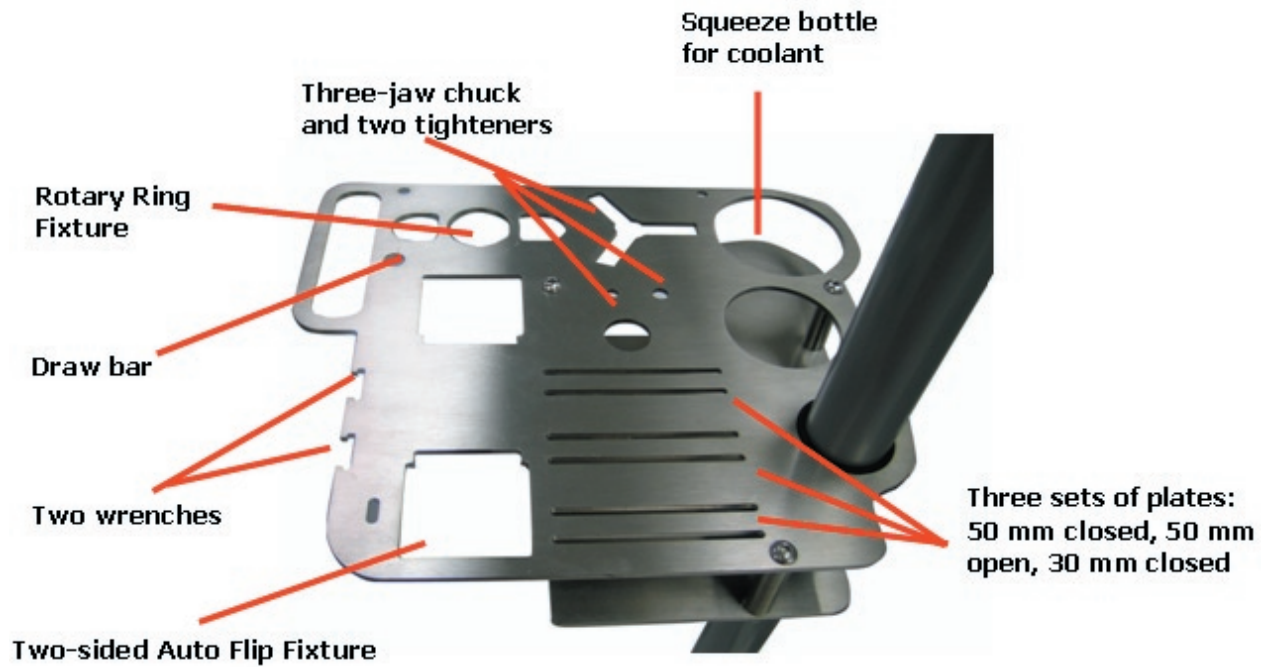
Coolant Type & Proper Usage: The only coolant recommended for use in the Revo 540 mill is Synspar GP by Spartan diluted with distilled water (NOT tap water) in a 20: 1 ratio. This coolant can be ordered by contacting Gemvision.

We strongly advise against the use of anything else in the coolant system of the mill. Tap water and other liquids can corrode the inner workings of the A axis, ruin the anodizing on the mill, stain the parts and pieces of the mill, and even clog or damage the coolant system.

REVO 540B Tool Tray Assembly:

The tool tray is attached to the mill prior to shipment. It is affixed to the left-hand side when facing the mill. The proper placement of the fixtures, tools, and accessories that fit into the tool tray is illustrated below.

Empty Tool Tray:



Full Tool Tray:

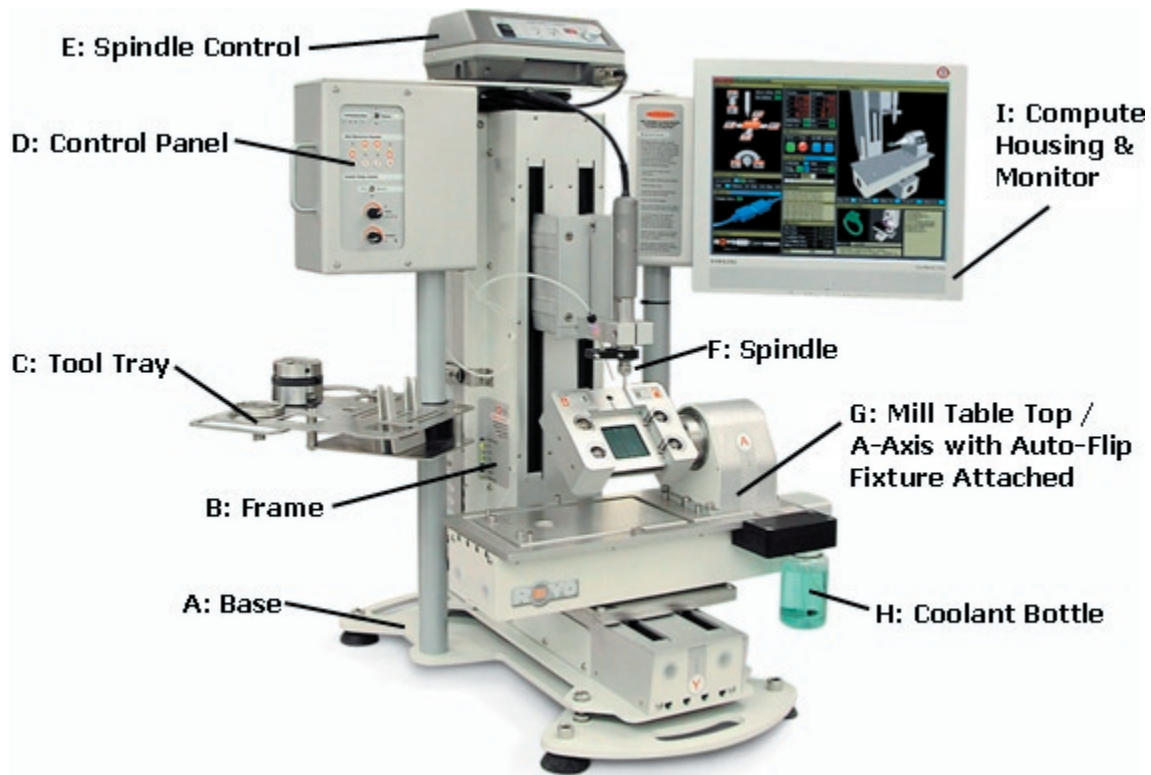


The tools that currently do not fit into the tool tray are:

- Two sets of Allen wrenches: Standard (black) and Metric (blue)
- Tightener and fixture kit for future use with thick wax tubes.
- Calibration arm and two dowel pins used for calibration



Diagram of Parts:



A – Base: Grip mill by handles on front and back of mill base to properly move it.

B – Frame: Internal electronics and mechanisms are housed in frame.

C – Tool Tray: Most fixtures and tools may be stored in tool tray for convenience.

D – Control Panel: (See more detailed illustration at right.) Use the control panel to monitor and manually reset the mill electronics:

1. Manually reset power for mill electronics.
2. Monitor communication to and from mill computer
3. Monitor movement of X, Y, Z, and A axes
4. Toggle coolant pump from auto to manual or off.
5. Adjust pump output rate (pulse and duration of pump activity)

D: Control Panel



E –Astro 250 Spindle Control: Manually controls the speed of the spindle:

- Power Button turns power on or off.

NOTE: *Do not turn off power during milling!*

- “Automatic” should be selected so that the spindle is turned on and off by the Revo toolpath.

- “Manual” will turn the spindle on continuous mode so that the spindle will not be turned on or off by the toolpath (not recommended).

Spindle Control:



F – Spindle: The spindle is turned on and off by the toolpath. Spindle rpms are set on the Astro 250 spindle control box on top of the mill.

The spindle consists of the following parts:

- Motor
- Shaft
- Collet (grips the cutter)



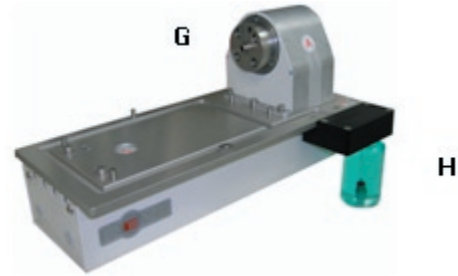
- Spindle cap (tighten / loosen to install or uninstall the cutter)

Parts of Spindle: (top to bottom -- shaft, collet, spindle cap)



G – Mill Table Top: Holds various fixtures in place during milling. (Shown with two-sided auto flip fixture attached to the A axis, above.)

H – Coolant Bottle: Re-circulates coolant via moat in fixture plate for efficient use of coolant.



I – Computer Housing & Monitor: Housing unit for computer is found in rear of monitor. Mouse and Virtually Indestructible Keyboard are also included and should be positioned at the base of the machine.

Monitor:

Housing & computer:



Mouse & Virtually Indestructible Keyboard:



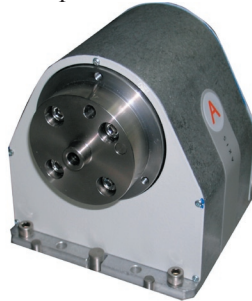
Revo540B Milling Fixtures:

Draw Bar: Use the draw bar to mount fixtures onto the face plate on the A axis.

Draw Bar:



Face plate on A axis:



Ring Fixture: Attach the **Ring Fixture** to the mill by lining up the pin in the fixture with the slot in the A axis face plate. Tighten the draw bar to secure the attachment to the A axis.

To attach each of the fixtures described below, line up the locator pin on the chosen fixture to the slot in the faceplate on the A axis:

Example: pin on Ring fixture:



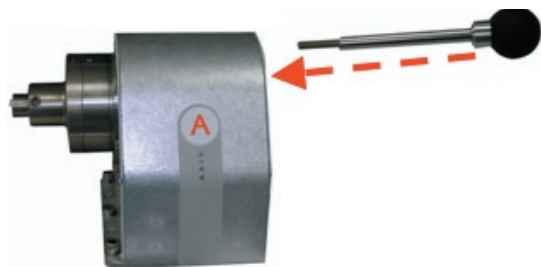
Ring fixture lined up with slot on A axis:



Mandrel 2-sides Plus Rotary: Secure this mandrel to the **Ring Fixture**, pictured above, to run the second half of the auto-flip plus rotary strategy. The three dowel pins line up with three holes in the center core in the ring model (further details are found in the related section of this manual).

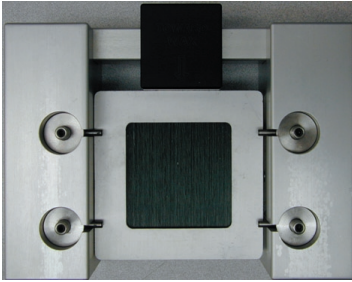
Insert the mandrel into the ring fixture, aligning the notch in the mandrel to the key on the ring fixture, and tighten the mandrel in place using the set screw located in the ring fixture.

Finally, insert the draw bar through the backside of the A axis and tighten, securing the fixture to the mill.

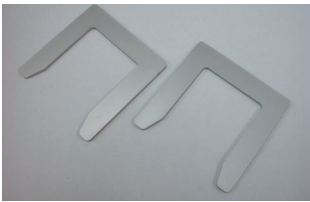


Two-Sided Fixture: 50-mm and 30-mm plates are included for use with the two-sided auto-flip strategy. Open 50-mm plates are also included for use with the 3-sided auto-flip strategy.

Two-Sided Fixture with gauge block for proper spacing & 50 mm closed plates installed:



50-mm open plates:



3 Jaw Chuck: Use the 3-jaw chuck with the wax tube to mill simple rotary rings. To assemble, insert the three “jaws” into their corresponding slots in the fixture (labeled 1, 2, 3). Align the locator pin with the slot on the A axis face plate and tighten with draw bar to attach the chuck to the A axis. Then, use the chuck wrench pins included to tighten them around the wax tube.

Chuck & 3 Jaws:



Chuck wrench pins / 3-jaw chuck assembled:



Calibration Arm: Use this arm as instructed in the Calibration interface in the Revo Control software to assist in the calibration process.



Cutters:

Identify the cutters by the color of their caps, being very careful not to switch caps!! It is VERY IMPORTANT to note that only the 10-degree and 0.032 mm flat cutters can cut rough wax. The other two cutters (0.016 mm flat and 6-degree tapered) are only designed for finishing work, as when running a second pass over a piece that was already cut by one of the other two cutters.

Cutters, left to right: 0.016 mm flat, yellow cap; 0.032 mm flat, black cap; 6-degree tapered, light blue cap; 10-degree tapered, dark blue cap.



Tools:

Wrenches: During cutter installation, use the two wrenches to tighten and loosen the spindle cap.



Electronic Digital Caliper: Use the Electronic Digital Caliper for measurement.



Allen Wrenches: Two full sets of Allen Wrenches, standard (black set) and metric (blue set), are included for use when assembling various fixtures.

Extra Coolant Needle: An extra coolant-dispensing needle is included should replacement be necessary.



Other Accessories:

- Wax: 2 packages of 10 mm thick wax, 12 pieces each; 1 package of 5 mm thick wax; 1 tube of 45mm diameter wax for thick single-sided rotary rings.

- Squeeze bottle for dispensing Synspar coolant

- Brake Caliper Grease and Jiffy Luber Hand Gun for lubricating mill (lubrication process described in full later in this manual).

- Sponge to place on fixture plate directly beneath cutter when changing tools to prevent cutter breakage.

Chapter 2: Assembling the Revo 540B Mill

The Revo540 mill is shipped with the following base components already assembled:

- Revo540 Milling Machine
- Spindle Unit
- Housing for Computer and Monitor
- Tool Tray

The following parts will need to be attached to the base components:

- Spindle control
- Computer & Monitor
- Tool Sensor
- Coolant dispensing needle

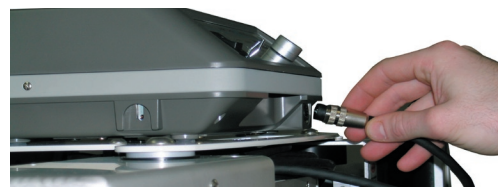
Attaching the Spindle Control Unit:

To begin, place the spindle control unit on the top shelf of the mill with the controls facing the front of the mill. In the front of the spindle control unit, connect the Motor Output Power Connector, which sends output from the spindle control unit to the spindle:

Spindle control unit (controls on unit should face the front of the mill)



Connect Motor Output Power Connector to front of spindle control unit:



Next, connect the two cords in the rear of the spindle control unit: the Sequence Control Cord (found on the left when facing the rear of the mill) and the Power Source Cord (on the right). The power source cord (shown being connected, below) must be plugged into an electrical outlet.

Rear of spindle control unit: (cord on left = sequence control cord; cord on right = power source cord)



Installing the mill computer & monitor:

Note: In order to attach the computer and monitor to the mill, it will be necessary to use a Phillips screwdriver, and it may be helpful to enlist the assistance of a second person.

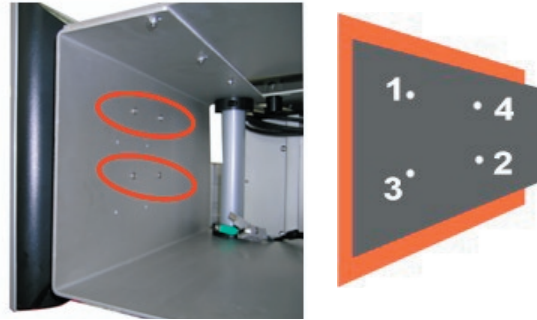
Begin by attaching the VGA cable and the power cord to the monitor. (These components will be difficult to connect after the monitor has been installed.)

(1) Power Cord; (2) VGA cable to computer



To affix the monitor to the computer housing unit on the mill, it is a good idea to enlist the help of a friend. One person should hold the monitor flush against the front of the housing unit, aligning the four mounting holes on the back of the monitor with the four corresponding holes in the computer mounting unit (screws circled in illustration below, for emphasis). It is a good idea to tighten the screws in a crisscross pattern, as pictured below:

(L) Monitor and housing unit; (R) Tighten screws in sequence indicated below:



Slide the computer into the housing unit and attach each connector in its appropriate slot at the rear of the machine. The computer should be facing out of the unit so that the CD tray and start / reboot buttons are easily accessible.

Computer / Housing Unit:



Monitor / housing unit / computer:

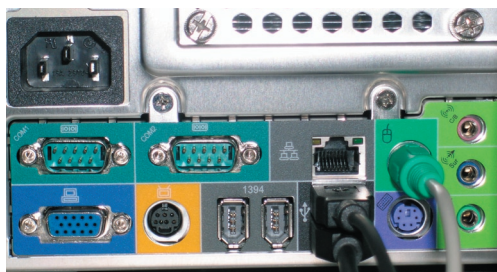


The cables at the rear of the computer, when plugged in, resemble the following image:

Rear view of machine (plugged in):



Corresponding ports only, not plugged in:



(Left side, top to bottom) Power cable; Communication cable connecting computer and mill; VGA cable connecting computer and monitor; (Bottom right: plugged in, above) USB Keyboard connection; Mouse

IMPORTANT: Plug the communication cable from the computer to the mill into the **FIRST** communication port (COM1), which is the port on the left-hand side just beneath the power cord!

Connecting the Tool Sensor:

The tool sensor connector is found on the left-hand side of the mill frame when facing it. Plug in the tool sensor and, when it is not being used, rest it in the left rear handle of the mill.

Left-hand side of mill, tool sensor highlighted:

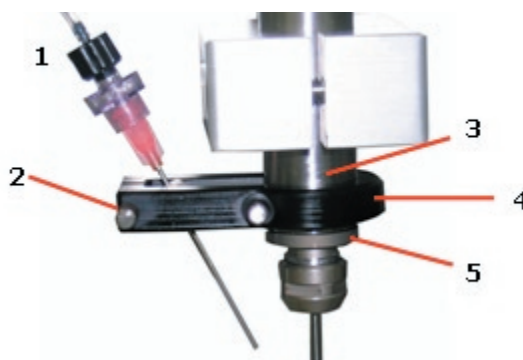


Plug in tool sensor / rest sensor on back left handle when not in use



Attaching the Coolant-Dispensing Needle:

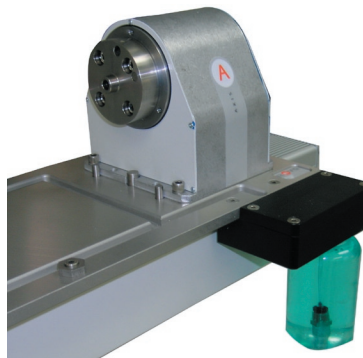
Slide the bracket that holds the coolant-dispensing needle up onto the spindle, positioning it on the spindle shaft just above the spindle collar. Be careful not to position the bracket over the spindle collar because this can prevent the spindle from rotating freely and can therefore damage the spindle. Tighten the thumb screw on the fixture to secure it to the shaft.



(1) Coolant-dispensing needle; (2) Thumb screw; (3) Spindle Shaft; (4) Needle bracket; (5) Spindle collar

Assembling the Coolant Container:

Fill the coolant container with coolant (be sure to follow the directions on the coolant for proper dilution). Drop the coolant tube into the container while guiding the container into position to screw it into the fixture on the mill. Make certain that it is filled before starting a new job. Then, keep it filled throughout the mill job: it is a good idea to check on coolant levels during a particularly long job and refill the coolant container as necessary.



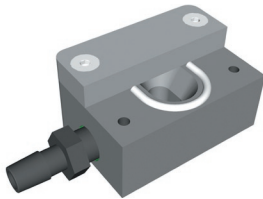
Assembling the Coolant Tank for use with Extended or Multiflip Fixtures:

Please note that this is a permanent change, replacing the smaller coolant jar from now on !!

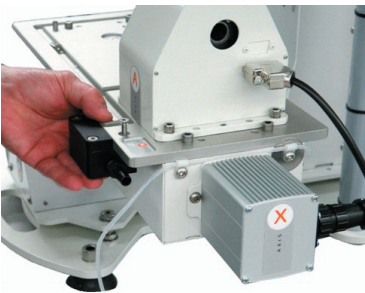
Start by removing the cutter for your own safety during this operation. Then, unscrew the existing coolant jar from the drain. Using a 1/8 in. Allen wrench, remove the existing drain, setting aside the screws that attached it to the mill since you'll be needing them momentarily.



Apply a 1/8 in. bead of silicone adhesive about 1/8 in. away from the edge of the new drain that came in your expansion kit.



Then, press the new drain into position at the location from which you removed the previous one, attaching it to the mill with the 1/8 in. wrench and the screws you just removed. Remove the excess adhesive from the drain cavity before it dries (a Q-Tip works well for this procedure).



Remove the filter from the existing tube and set it aside, as you'll be using it momentarily. Remove

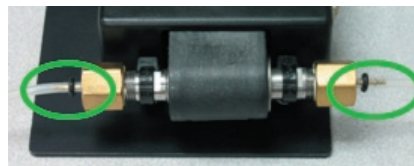
the existing "inlet" tube from the mill, threading it carefully through the eyelets that secure it to the mill while taking care to note each one as you go so you can re-thread the new tube through the identical eyelets. Also remove the "o-ring" from the end of the tube where it connects to the pump, setting it aside so you can find it again in a moment, as you'll need to re-attach it to the new length of tubing to the pump.

To replace the length of tubing you just removed, cut about 4 inches off the thinner tubing found in your Revo Expansion Kit (do *NOT* cut the thicker tubing included in this kit!!). Set aside the 4-in. piece for a moment, and thread the longer piece through the same eyelets on the mill from which you just removed the previous length of tubing. To more easily thread it through the eyelets, you may snip away a tiny bit of the end of the tubing, cutting it on an angle so it can fit more easily through the eyelets.

Secure the o-ring you removed earlier to the end of the tubing connected to the pump. The best way to do this is to first thread the o-ring over the smallest Allen Wrench in your set (0.050 in.). Then, insert the wrench into the end of the tube and roll the o-ring onto the tube.



Remove the wrench before affixing the tube to the pump.



After threading the new tubing through the eyelets, you should be left with about 19 inches of tubing between the mill table and the new coolant tank. Position the coolant tank to allow for enough slack to move the axes during milling (you may trim tubing if there is excess remaining at this time). Connect the end of the tube to the top of the smaller cap on the coolant tank.

Next, using the shorter length of tubing that you

trimmed away early on in this procedure (about 4 inches), attach the filter to one end and affix the other end over the inside portion of the small cap on the tank.

(L - R) Long length of new tubing; small cap on coolant tank; 4 - inch length of tubing, filter (removed from previous coolant assembly).

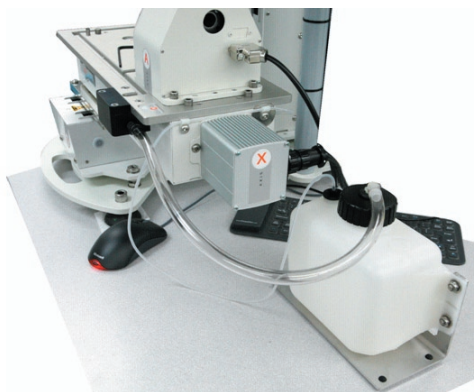


Drop the filter and its tube into the tank and attach the small cap to the top of the tank, thereby completing this portion of the connection.



To attach the drain to the tank, take the 19 inches of 3/8th in. diameter vinyl tubing and connect it from the end of the drain to the larger cap on the coolant tank.

Note: If the tank will be sitting on the counter or tabletop beside the mill, make certain that this tube is arranged so that the tank is "downhill" from the mill to ensure that coolant cannot get trapped in the tube and therefore overflow the drain.



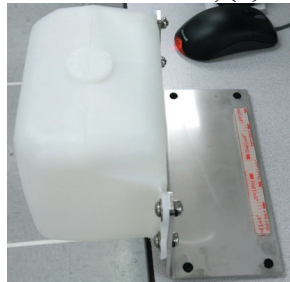
Finally, if you wish to secure the tank to the edge of the table or countertop beneath the mill, remove the tank from its bracket using a 7/16 in. nut driver

and a 5/32 in. Allen wrench, and turn the bracket upside-down so that the long, flat portion is flush with the tabletop.



Re-attach the tank to its mounting using the same hardware. To affix the entire assembly to the table, peel off one adhesive side of the velcro and attach it to the underside of the bracket, close to the edge away from the tank. Peel off the other adhesive side and press the entire assembly to the table or countertop. Now, to remove the bracket from the table, separate the two halves of the velcro, leaving one half on table or countertop and the other half attached to the mounting assembly for the coolant tank.

(L) View from bottom - velcro attached with adhesive to bracket; (R) Finished mounting:



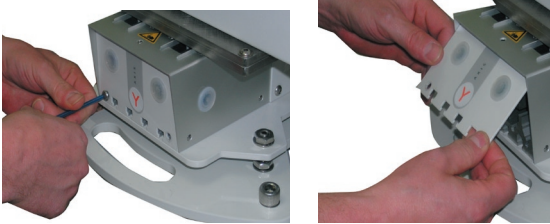
Chapter 3: Maintaining the Mill

Revo 540B Mill

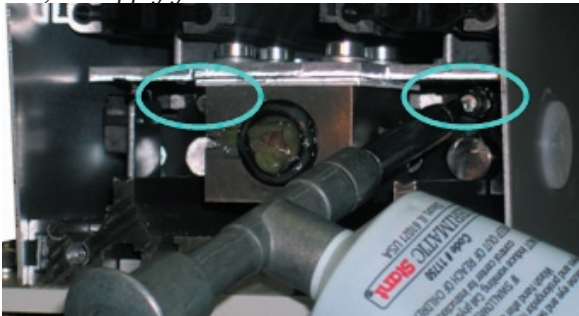
Lubricating the Mill:

About once every 6 months or so, lubricate the X, Y, and Z axes on the mill. To access the zerts (yes, that's what they're called!), remove the end panel for each axis by unscrewing three screws that affix it to the axis. Pop off the panel and insert the lube gun into the housing. Hold it against each zert (two for each axis: one on either side) and dispense.

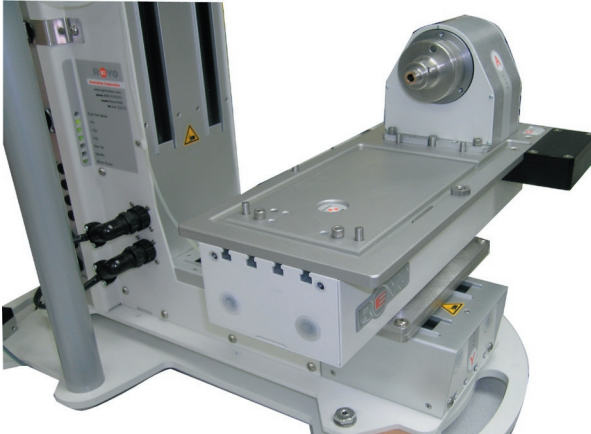
(L) Y axis: remove two screws on front panel and one on top of Y axis, above front panel; (R) Remove panel:



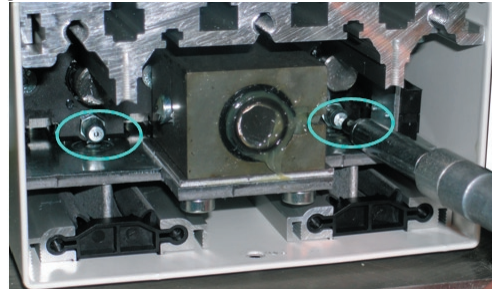
Locate two small, round metal zerts, one on each side, and apply grease:



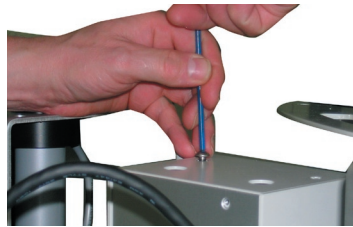
X axis: remove two screws from front of panel and one underneath panel.



Remove panel and hold lube gun to zerts to apply grease.



Z axis: unplug spindle control (two plugs in front and one in back) and remove spindle from cover. Remove bolt holding top shelf to the column that the coolant control panel is attached to and push top shelf out of the way.



Remove top cover by unscrewing two bolts from top and two from front of Z axis:



Find two zerts on either side of Z axis and apply grease. Don't fall!

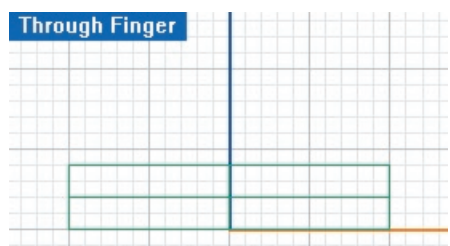
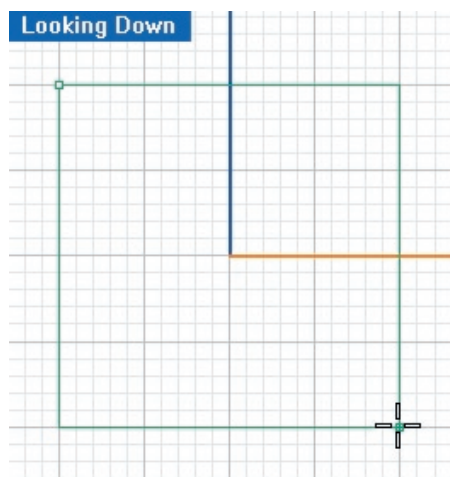


Calibrating the mill:

If there seems to be major problems with parts that have been produced by the mill, the mill's calibration might be off. When the mill is calibrated, it will properly calculate the center of the model being cut based upon the fixture offsets

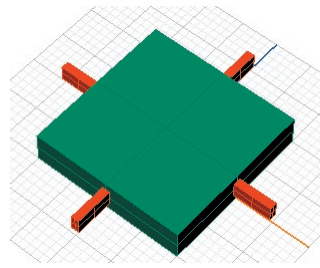
Checking the need for calibration: It is rather seldom that a user will need to calibrate the mill. To determine for certain whether or not the mill needs to be calibrated, create the following simple model in **Rhino** and run it on the mill using the two-sided auto flip strategy. Examining the resulting wax model will help the user determine whether or not the mill needs to be calibrated.

To build the model in Rhino, start with clear viewports. Turn on the **Grid Snaps** at the 1.0 mm distance. Select the **Box** command and create a rectangle, centered at 0,0,0 in the **Top** viewport, that is 15 mm X 15 mm square and 5 mm tall in the **Front** viewport.



Draw supports for this piece, as usual, and proceed through the steps in the **RevoCam Software** for use with the **Two-Sided Auto Flip** strategy (see the relevant portion of the manual for details). Then, save the .RVO file and transfer it to the mill computer. Load this RVO file into the **Revo**

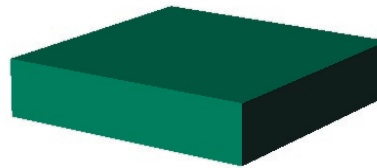
Control Software on the mill computer.



After the part has been produced on the two-sided auto-flip fixture, remove it from its supports and check the Side View (Right) of the model. If the top and bottom sides of the model do not line up, the mill needs to be calibrated.

Note: Although the illustration below is slightly exaggerated, this difference will be evident to the naked eye on a model produced by a mill that needs to be calibrated.

Normal model:



Model produced by a mill that needs to be calibrated:



Calibrating the mill: To calibrate the mill, click on the button in the upper left-hand corner of the Revo Control Software. This opens up the Calibration interface. Follow the directions posted in each step of the interface to calibrate the mill.



Aligning the Mill with Manual Offsets

Use these two alignment techniques to enter in Manual Offsets for both the Y and Z axes of your mill. This process will help you successfully “square up” the mill, telling the mill where the center of each axis is to the smallest increment possible, so that it can create parts that aren’t skewed in one direction. After completing this process once, it should not require adjustment again unless you notice that parts appear uneven on one side or the other.

Please note: when performing the steps in the calibration page does not seem to help problems of unevenness in your models. this is the next step to take. Also, it is fine to run this process pro-actively, before milling, to head off any possible issues with alignment of the axes.

To keep the mill in proper alignment after running these two operations, it is important NOT to run too many Seek Home operations, and especially not to run Seek Home one after another, with nothing in between them. Remember that the mill Seeks Home automatically upon startup and after running each job. Additional Seek Home operations should be limited to times when safety is at issue - as when stopping the mill in the middle of a job.

In addition, running Calibration Steps in the Mill Calibration page should be limited to times when they are absolutely necessary. Performing unnecessary Seek Homes or Calibration Steps can put the mill out of alignment, making the following two operations necessary once more to re-align the fixture offsets.

Auto-Flip Flat Milling (2-sided) Position Alignment:

The Problem: The toolpath looks fine and appears to hit all areas of the model necessary to achieve proper clean-up of details. However, the top and bottom halves of the piece appear to have a “shift” between them, where they do not line up properly with one another. This may be evident on any part that runs a two-sided operation, from 2-Sided AutoFlip to 3-Sided AutoFlip, as well as Auto Flip Plus Rotary or 3-Sided AutoFlip Plus Rotary.

These errors are not to be mistaken for calibration issues. Calibrations simply define the areas in which the part will be cut (or, where the fixture offset is set), and this is not the problem you are experiencing.

In the calibrations pages found in the Revo software, there is a manual calibrations tab that

Manual Calibrations Page:

Revo 540 Mill

Mill Control Setup

Mill Calibration

Manual Calibration

Rotary Plus One

The Manual Offsets are used while calibrating your mill, if during the calibration process the spindle/pin does not line up with the tool sensor these number should be adjusted.

The Manual Calibration Offsets will be added to the Automatic Calibration Values. These Offsets should only be change, if you are experiencing unacceptable levels of calibration error from the Automatic Calibration process.

Manual X Offset

0

Manual Y Offset

0

Show Manual Offset

Single Job Rotary :

3 Sided Jobs, Rotary Part :

Auto Flip Flat Jobs :

Auto Flip ,90 Edge :

Rotary Plus One, Rotary Part :

X Offset	Y Offset	Z Offset	A Offset
0	0	0	0
0	0	-0.035	0
0	0	-0.035	0
0	0	0	0
0	0	0	0

WARNING: Show manual Offset will move the spindle directly above the tool sensing position, check that there are no obstruction fixtures attached to the A axis before pressing this button.

RESET

Apply Changes

OK

exposes the manual calibration offsets. Open this to fine-tune the mill to cut accurately. Then, perform the following steps:

(1) Run the Test Block: In order to get the **2-Sided Auto-Flip Flat Milling** to cut on center without leaving a ledge, you'll need to cut the program **531 AFT Block 032.rvo** that you can download from the web at <http://www.gemvision.com/revo>.

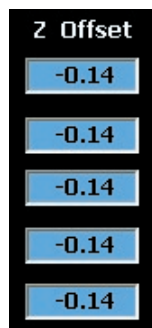
This program is designed to run on the Auto-Flip (2 sided) fixture using the **0.032 flat cutter with the black cap** and will require a **10mm thick piece of wax**. Remember to run a **Seek Tool** operation before running this toolpath if you have just installed the 0.032 flat cutter!

This tool-path will cut a 10mm X 10mm X 5mm block with an arrow cut into Side One pointing toward the machine. In other words, after the block has been cut, if you rotate the A-axis so that the open end of the fixture is pointing away from the machine, the arrow will be visible and pointing toward the machine.

(2) Correct the Z Offset: After completing the path, remove the fixture from the A axis and then remove the block from the wax. Begin by measuring the thickness of the part. If it is between 4.90 and 4.95 mm, you are set. If it is greater than 4.95 mm, determine the difference between the measured value and 4.93 mm and divide that value in half. Subtract the new number from the Z Offset column.

Example: Measured Thickness = 5.2 mm
 $5.2 - 4.93 = 0.27 \text{ mm}$
 $0.27 \text{ mm divided by } 2 = 0.14 \text{ mm, so ...}$
 Subtract 0.14 mm from value in Z Offset column.

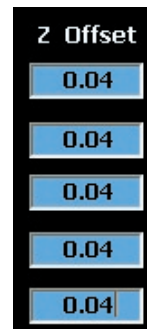
When correcting the Z offset column, correct it for *every row* or every job in the Manual Calibrations page.



If the Z thickness is less than 4.90 mm, *add* the difference between the measured value and 4.93 mm to the Z offset column.

Example: Measured Thickness = 4.85 mm
 $4.93 - 4.85 = 0.08 \text{ mm}$
 $0.08 \text{ divided by } 2 = 0.04$
 Add 0.04 mm to the value in the Z Offset column

REMEMBER: When correcting the Z offset column, correct it for *every row*, or, every strategy listed in the Manual Calibrations page!!

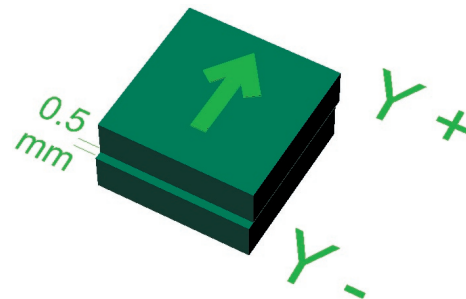


In either case described above, click **Apply Changes** in the manual calibrations page to save the values that you set.



If the thickness of the test block measures between 4.90 and 4.95 mm, no adjustment is necessary in the Z direction.

(3) Correct the Y Offset: Next, inspect the accuracy of the block by holding it so that the arrow is visible and pointing away from you. Check to see if there is a ledge in the front or back ends of the block, as shown below. Typically, this “step” is found along the Y axis. If you find a ledge, measure it.



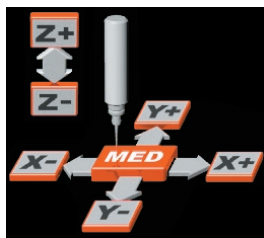
The value by which you will correct the Y Offset in the Auto Flip Flat Jobs field is equal to **HALF** the

value of the ledge measured on the block, above (in this case, 0.25 mm). Then, determine whether the ledge needs to be corrected in the Y+ or Y- direction.

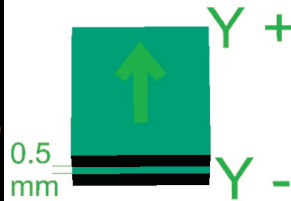
To do so, take a look at the Jog Controls on your mill. The arrow on the test block is pointing in the same direction as the Jog Controls, with Y+ in the direction that the arrow is pointing and Y- in the opposite direction. So, if your ledge is visible on the side away from the arrow, as illustrated in this example, the mill is going too far in the Y+ direction and needs to be adjusted back (Y-). In that case, you must *subtract* half the measured value to correct it.

Conversely, if the ledge is on the other side of the block, toward the tip of the arrow, your mill is going too far in the Y- direction, so you must *add* half the value of the ledge (move the spindle that distance in the Y+ direction) to correct it.

Jog Controls:



Correct Y Offset by negative 0.25 (-0.25)



	X Offset	Y Offset
Single Job Rotary :	0	0
3 Sided Jobs, Rotary Part :	0	0
Auto Flip Flat Jobs :	0	-0.25
Auto Flip ,90 Edge :	0	0

After changing this value, click on Apply Changes to correct the alignment.



(4) Repeat the Test: With both values (Z and Y) corrected as described above, mill the test block again to make sure alignment is correct. Continue milling test blocks and making these corrections until Z thickness is within 4.90 - 4.95 mm, and until the ledge is undetectable.

3-Sided Auto-Flip (90-Degree) Position Alignment:

The Problem: The toolpath looks fine and appears to hit all areas of the model necessary to achieve proper clean-up on all details. However, the front or back side of the ring appears to be “gouged out”. This may be evident on any part that runs a three-sided auto flip operation, including 3-sided auto flip plus rotary jobs.

These errors are not to be mistaken for calibration issues. Calibrations simply define the areas in which the part will be cut (or, where the fixture offset is set), and this is not the problem you are experiencing.

In the calibrations pages found in the Revo software, there is a manual calibrations tab that exposes the manual calibration offsets. Open this to fine-tune the mill to cut accurately. Then, perform the following steps:

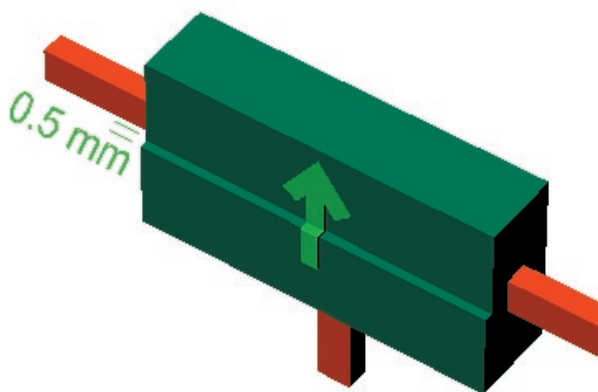
	X Offset	Y Offset	Z Offset	A Offset
Single Job Rotary :	0	0	0.075	0
3 Sided Jobs, Rotary Part :	0	0	0.075	0
Auto Flip Flat Jobs :	0	0	0.075	0
Auto Flip ,90 Edge :	0	-0.25	0.075	0

(1) Run the Test Block: In order to get the **3-sided Auto-Flip Flat Milling** operation to cut on the center line in the 90 degree (upright) position, you need to cut the program **531 3AFT Block 032.rvo** that you received on disk with this documentation. (You can also download this file from the web at <http://www.gemvision.com/revo>).

This program is designed to be run like a typical 3-Sided Auto Flip operation, using the 50-mm open plates. This file uses the **0.032 flat cutter with the black cap** and will require a **10mm thick piece of wax**. Remember to run a **Seek Tool** operation before running this toolpath if you have just installed the 0.032 flat cutter!

This tool-path will cut a 20mm X 10mm X 5mm block with an arrow cut into Side Two, pointing toward the open end of the fixture. In other words, after the block has been cut, if you rotate the fixture into the 90 degree position, the arrow will be visible and pointing up. After completion of the path you can remove the fixture from the A-axis and remove the block from the wax.

(2) Correct the Y Offset in the Auto Flip 90-Degree Column: After milling the test block, remove it from the wax and inspect the accuracy of the block by holding it so that the arrow is visible and pointing up. Check to see if there is a ledge in the front or back face of the part, as in the example shown in the first picture below. If you find a ledge, measure it:



When you have a ledge on the side with the arrow (-Y), **SUBTRACT** the SAME VALUE as the size of the ledge FROM the Y Offset value in the Auto Flip, 90-Degree Edge Row on-screen (if the ledge is on the other side, **ADD** the value of the ledge to this field):

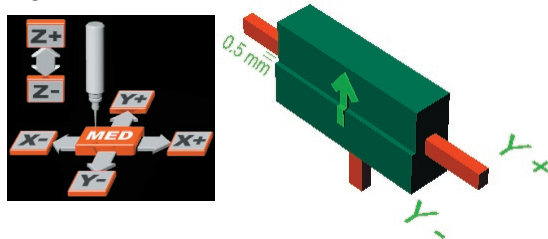
	X Offset	Y Offset
Single Job Rotary :	0	0
3 Sided Jobs, Rotary Part :	0	0
Auto Flip Flat Jobs :	0	0
Auto Flip ,90 Edge :	0	-0.5

(3) Repeat the Test: Click **Apply Changes** in the manual calibrations page to save the values that you set. Then, run the part again. If there is still a ledge, enter manual offset values again as described above. Repeat these steps until the test block has no visible ledge.

In this case, use the FULL VALUE of the ledge to change the number in the Auto Flip, 90-degree field under the Y Offset column. To determine whether you need to add or subtract this value from the number already in the column, look at which side of the block the ledge is on: if the ledge is on the side with the arrow, as shown above, **SUBTRACT** this value from the value in the box. If it is on the other side, **ADD** this value to the value in the box.

This is possible because, as in the above alignment block, the block is set up in line with the Jog Controls in the mill interface, with the arrow pointing in the positive Z direction. So, if the ledge is on the “negative Y” side, the mill is cutting too far in the Y+ direction and the spindle must be moved back in the negative Y (Y-) direction. Conversely, if the ledge is on the “positive Y” side, this means that the spindle is too far in the negative Y direction and must be moved forward in the Y+ direction.

Jog Controls: Arrow points up in +Z direction:



Chapter 4: Preparing to Mill

Using the Revo Control Software:

Before using the mill, make certain that the power switch found on the right-hand side of the mill (when facing the mill) is in the “ON” position. Also make certain that the communication cable that connects the mill computer to the mill (found above the power switch) is securely attached.

Right-hand side of mill: communication cable and power switch enlarged for emphasis



Turn on the mill computer by pushing the power button on the front of the computer. Next, turn on the monitor by pressing the power switch on the monitor.

The **Revo** mill software comes pre-installed on the mill computer. To open up the mill software, double-click on the **Revo** software icon found on the desktop of the Revo computer. Each part of the software interface is described in full below:

Loading a RVO file:

The **Revo** control software only opens the .rvo files created by the **RevoCam** software.

Note: It is important NOT to install RevoCam, Rhino, or any additional software on the mill computer!

Use of the **RevoCam** software varies based on the strategy being used and the type of model being produced. The proper use of each strategy is detailed further in this manual.

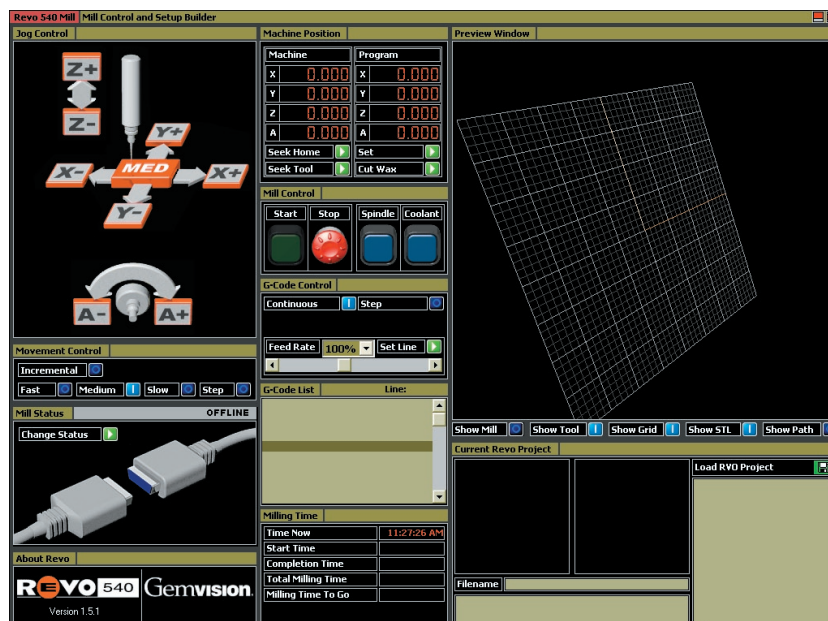
After using the **RevoCam** software to produce a .rvo file, there are three ways to save the .rvo file for use on the mill computer: (1) Save the .rvo file to a USB thumb drive (a small, portable hard drive) and insert that into the USB port on the mill computer; (2) Write the file to CD and place the CD in the CD-Rom drive found in the mill computer; or (3) If the mill computer and the computer on which Rhino is installed are both on the same network, save the .rvo file to a drive that is shared by both computers.

To open the .rvo file from either the CD-Rom drive, the USB drive, or the shared drive, click on the green disk icon beside the **Load RVO Project** label at the lower right-hand corner of the screen.

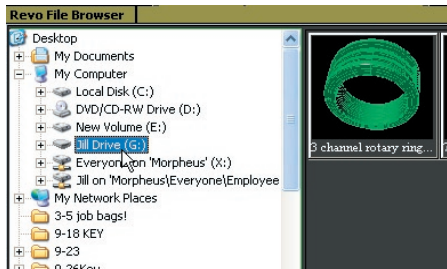
Load RVO Project



Within the **Revo File Browser**, navigate to the drive on which the .rvo file is stored. Clicking on any drive or folder containing .rvo files will cause



preview images of the .rvo files to appear in the gray space to the right of the menu.



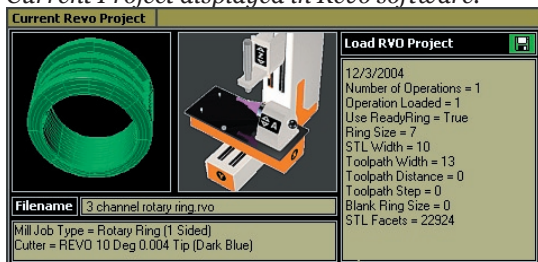
Click once on the preview image of the model that will be milled and the corresponding filename will appear in the box below the browser menu along with information about that file, including the number of operations ("jobs") it contains, the selected fixture plate, and the selected cutter.

Click once on the preview image in the browser window to display file information below the navigation menu:



Double-click on the preview image to load the selected file into the Revo software. A preview image of the file will be loaded under the **Current Revo Project** screen along with the **Filename** and information about the file. A preview image of the mill with the corresponding fixture attachment highlighted appears beside the thumbnail of the ring to show the location and fixture required for the selected toolpath strategy (in this case, "Rotary Ring 1-Sided").

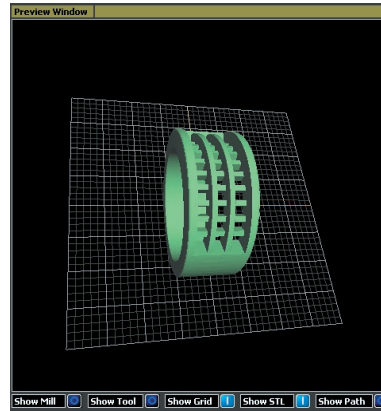
Double-click preview image to load selected file:
Current Project displayed in Revo software.



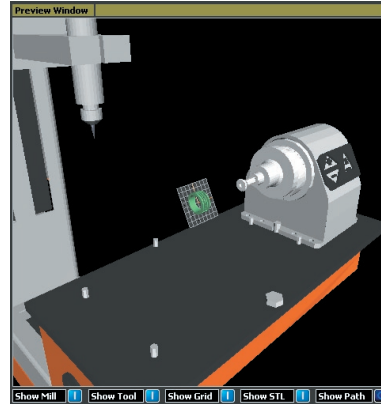
To preview the file on the grid found within the

Revo software, select one (or more) of the options found beneath the preview grid. It is possible to pan, rotate, and zoom in on the grid using the mouse in the same way it is used within **Rhino**.

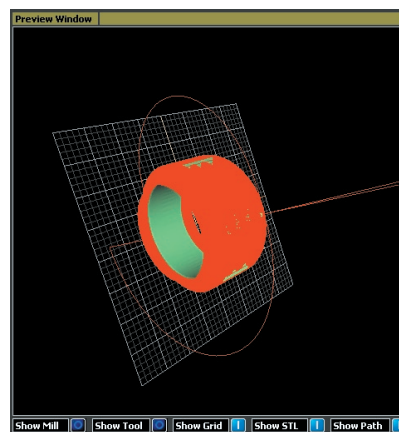
With **SHOW GRID** and **SHOW STL** selected:



With **SHOW MILL** and **SHOW TOOL** selected, too:



With **SHOW PATH**, **SHOW GRID**, and **SHOW STL** selected:



Determining Milling Time:

When a .rvo file has been loaded, content will appear in the **Milling Time** portion of the screen. The **Milling Time** portion of the **Revo** software displays the following items:

- (1) The present time (**Time Now**),
- (2) The time the job was started (**Start Time**),
- (3) The projected time the job will be completed, barring any complications (**Completion Time**),
- (4) The total time it will take to mill the job (**Total Milling Time**), and
- (5) The remaining time left on the mill job (**Milling Time To Go**).

Milling Time	
Time Now	1:31:22 PM
Start Time	1:23:13 PM
Completion Time	3:07:52 PM
Total Milling Time	1:44:39
Milling Time To Go	1:44:39
0%	

Controlling the G-Code:

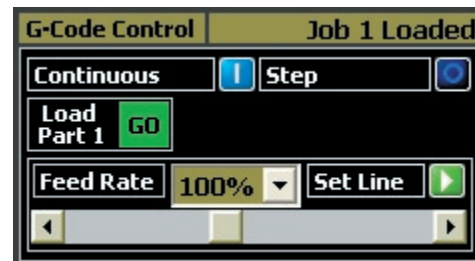
The **G-Code List**, which is found just above the **Milling Time** display, lists the code found in the **Revo** file. G-Code directs the mill to move and produce the model from the wax. There are usually thousands of lines of code found within this list.

G-Code List	Line 1
N1 G00 G17 G90 G54	
N2 G01 X00 F200	
N3 G00 A0	
N4 G00 X0	
N5 G00 Y0	
N6 M08	

The **G-Code Control** interface located above the **G-Code List** allows the user to see how the computer delivers the G-Code to the mill. As the job is loading, the olive green space to the right of the **G-Code Control** label will turn bright blue and display the word “**Loading**”, indicating that the software is working to load the .RVO file. Once the file is loaded, it will display **Job 1 Loaded**. In order to re-load the job, it is not necessary to open up the file again: simply click on the **Load Part 1 GO** button and the G-Code will be

re-loaded in full to re-start the entire job. If there is more than one part to the job, as with the Auto-Flip Plus Rotary strategy, **Load Part 2** will also be displayed. This process is fully described in the related section of this manual.

With **Continuous** selected, starting the mill job will cause the G-Code to begin directing the mill through all of the lines of code in the job without stopping. The list will automatically begin to scroll through very quickly. The highlighted line of code is the line that the mill is currently reading.



If it is necessary to run one line of G-Code at a time, click on the **Step** option, turning off the **Continuous** option. This option allows the mill to read one line of code at a time and move to position. It will pause at the end of each move and wait for the user to click on the **START** button again.



Stopping the Mill:

When using either option, it is possible to stop the mill for any reason by using the **Stop** button in the mill software or by pressing any key (or combination of keys) on the keyboard. If it suddenly becomes necessary to stop the mill, hit any key on the keyboard and it will instantly stop.



Under normal circumstances, it is not necessary for the user to change to the **Step** option or to do anything with the G-Code list. However, should the mill job need to be stopped for any reason (e.g. the power goes out and the user wishes to properly shut the mill down in the few minutes allowed by the UPS battery back-up), be certain to make note of the line of G-Code that was highlighted when the

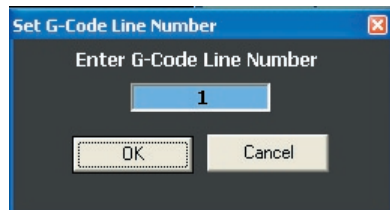
mill was stopped. The number of the G-Code line follows the “N” as the first thing in that line.

G-Code lines 1 – 6 (N1 – N6)

```
N1 G00 G17 G90 G54
N2 G01 X00 F200
N3 G00 A0
N4 G00 X0
N5 G00 Y0
N6 M08
```

After restarting the machine, click on **Load RVO File** and re-load the file that was previously stopped. Then, click on the **Set Line** option just above the **G-Code List** and enter in a value equivalent to the noted number minus 100. (It is not necessary to input the letter “N” during this process.)

This will cause the G-Code List to “jump to” the indicated number. A window will appear stating that the mill will now move to this position. Click **OK**. After the mill moves, turn on the **Coolant** button and click on **Start**. The mill will start cutting the model well before the point at which it was stopped, saving the time and aggravation of running the model from the beginning again.



Note: This will work ONLY IF the model was NOT REMOVED from the mill fixture (and the fixture was not removed from the machine) prior to re-starting the mill!! Also, it is VERY important to re-load the G-Code to allow the computer to read every line of code up to the selected line of code to start from. This ensures an accurate starting position and rotation of the piece.

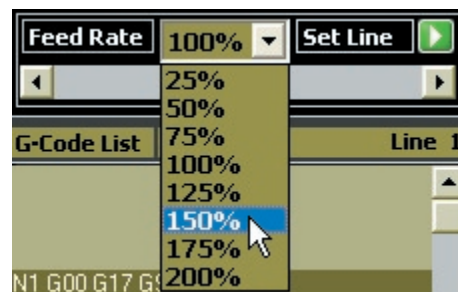
DO NOT attempt to “scroll down” through the G-code and highlight another line from which to start. ONLY the “Set Line” feature allows the user to start the G-Code from a point other than the first line of code!

Changing the Mill Speed:

The **Feed Rate** refers to the speed at which the G-Code is fed to the mill by the computer. It defaults to 100%. Raising this value speeds up the feed, which reduces the time it takes to mill the part. The feed rate may be adjusted in increments of 25%, with the lower limit at 25% and the upper limit at 200%.

The recommended setting is between 100-150%. Raising the feed rate above 150% is NOT recommended because it may reduce the quality of the final piece. To adjust the feed rate, use either the pull-down menu in which the **Feed Rate** value is displayed, or click and drag the sliding scroll bar beneath the **Feed Rate** menu until the desired value is displayed.

To adjust the feed rate: (1) Select the new rate from the drop-down menu: (this option can ONLY be used before starting the job!)



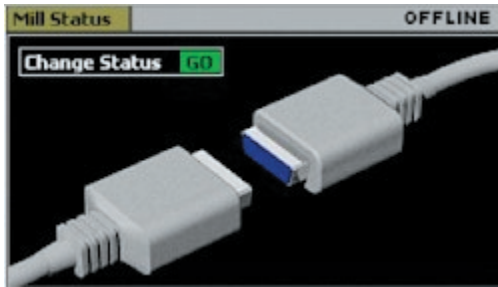
OR (2) click and drag the slider control: (this option may be used after the job has been started!)



Checking the Mill Status:

Before using the mill, it is important to note whether the computer is communicating properly with the mill. If the **Mill Status** indicates **Offline** (the plugs in the picture are gray in color and disconnected), then the mill computer is not communicating with the mill. To change this, click on the green **GO** button beside **Change Status** and the computer will attempt to reconnect to the mill. When the mill computer is properly connected, this illustration will be highlighted in bright blue and will show the plugs connected.

Offline:

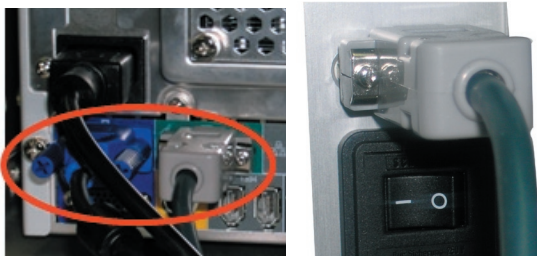


If connection is not possible, the following error message will appear:

“There was a communication error to the mill controller. Check that mill is connected and power is on.”

If this error message appears, check that the communication cable is securely connected to both the rear of the computer and the right-hand side of the mill frame. Also make sure that the power switch is in the “ON” position at the right-hand side of the mill frame. If it is already on, turn it off and then back on again. This will reset the mill control. Then, click on the “Change Status” arrow again to reconnect the mill to the mill computer and change the status to “Online”.

*Rear of computer, communication cable circled;
Mill communication cable and power button
(found on right-hand side of mill):*



Machine Position:

Each axis on the mill is a separate component. The G-Code controls the movement of each axis. Each axis moves independently of its counterparts. Each axis is labeled, as illustrated below:



Each time the Revo software is turned on, the mill “Seeks Home”, which means that it returns to the “o” position on each axis (X, Y, Z, and A). The “o” position for each axis is the furthest extent at the lower end of the axis: for the Z axis, this is the furthest “up”; for the Y axis, the furthest forward, for the X axis, the furthest to the right (when facing the mill).

When the mill is at “Home”, the machine coordinates for the X, Y, Z, and A axes should all equal o. These coordinates (o, o, o, o) differ from the grid coordinates familiar to many Rhino users. In Rhino, the center of the grid is equal to X= o, Y=o, Z= o; however, on the mill, a o value for all four axes means that each axis is at the furthest possible extent to the lower end of its travel.

For example, when the mill is homed, the X axis can move no farther to the right (when facing the mill); similarly, the Y axis can move no further forward, and the Z axis can move no further upward.

The **Machine Position** portion of the interface indicates the **Machine** coordinates, or, the location of each axis at the present time. When the mill is homed, X, Y, Z, and A will all equal o. The **Program** coordinates tell the position of the mill in relation to the “part zero”, or, the center of the piece that is loaded on the current fixture.

Each fixture has a fixture offset, which is the

location at which the part will be machined, associated with it. In addition, each strategy has a fixture (and, therefore, a fixture offset) assigned to it. That way, when the user selects a strategy, the computer automatically assigns a fixture and a fixture offset so that the mill can determine the center of the part.

The **Program Coordinates** are equal to 0, 0, 0, 0 at the center of the part. So, if the mill is homed, the **Program** coordinates will NOT be equal to 0, 0, 0, 0. However, as the mill moves closer to the center of the part, each number will move closer to 0. All of that is said to explain why there are two different sets of coordinates under **Machine Position**.

Machine Position			
Machine		Program	
X	0.000	X	- 6.500
Y	0.000	Y	42.428
Z	0.000	Z	72.667
A	0.000	A	-296.172

Seek Home:

Each mill job should be started with the mill in the “Home” position. To return the mill to home, click on the **Seek Home** command found under the **Machine Position** portion of the screen:



Note: If the mill has been moved out of the “Home” position, it will be necessary to click on Seek Home before it will be possible to start a job!

The mill automatically seeks home every time it is turned on and every time it is finished running a job. It is always a good idea to click on **Seek Home** as the final step before starting a milling job, just to make sure that the mill is properly oriented before it starts cutting the piece.

After running **Seek Home Go**, the mill will provide a “Homing Offset Error” dialog box. Essentially, this is reporting to the user how close to the previous “home” position the mill was able to get this time. Therefore, this should be a miniscule value, and the user can simply click on **OK** to move past it. If the “homing offset error” value seems higher than it should, click on **Seek Home GO** one more time and the machine should correct itself.

Jog Controls:

To manually control the positioning of each axis independently of the other three, use the **Jog Control** portion of the interface. **Jog Controls** allow the user to “jog”, or move, each axis to manually position the mill.

This feature is used most frequently during the **Rotary Ring** operation to set the X offset, or, the starting point of the toolpath, which allows the user to select where on the wax the toolpath will begin (for more details, see the related section of this manual).



The **Jog Control** interface displays which direction the spindle is moving in relationship to the mill; although, in reality, the axes are moving and the spindle remains stationary. To move the spindle along a given axis, click on the button that corresponds to the axis (X, Y, Z, or A) and the direction (+ or -) that the spindle needs to be moved along that axis.

For example, to move the spindle down along the Z axis, click on the **Z-** button. Then, if the spindle needs to be moved forward in the Y direction, click

on the **Y-** button. To move the spindle toward the left, click on the **X-** button. And to rotate the rotary fixture, click on **A-**, which rotates the axis toward the mill, or **A+**, which rotates it toward the user standing in front of the mill. (To remember which is which, just remember: you get an A+!).

Although this may sound confusing at first, the best way to learn what the jog controls do is to simply practice using them. As long as there are no fixtures, other objects, or body parts in the way of the spindle, and the spindle is not run down into the fixture plate, this should not be a problem!

To use **Jog Controls**, click on the control that indicates the direction in which the spindle will be moved and hold down the mouse button to move it in the indicated direction.

There are three speeds at which the mill can be jogged: **Slow**, **Medium**, and **Fast**. The speed with the blue indicator light on is the speed at which the mill will move (**Medium** set, below). This corresponds to the speed indicated by the orange box beneath the spindle in the interface screen.

Selected speed: Medium



Jog Controls Interface Display:

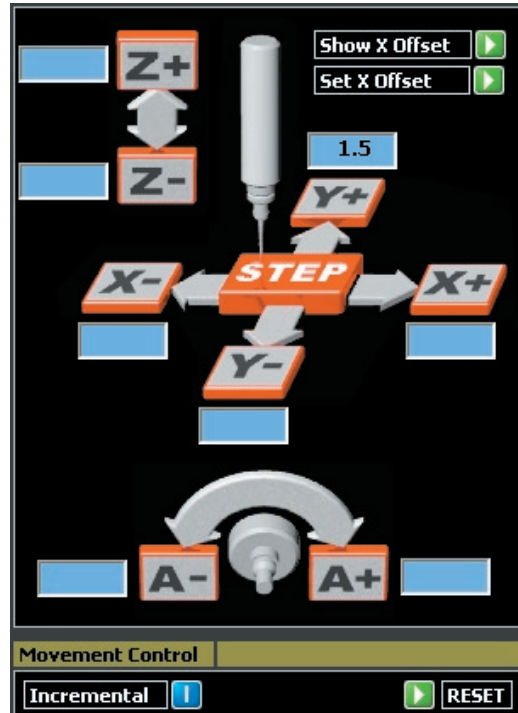


When the mill is set to **Step**, the axes will move in tiny increments each time they are clicked; holding down the mouse-button will not continue moving the axes as it does when on the other speeds. Each "step" movement equals 0.004 mm; therefore, it takes a LONG time to move any distance at this setting.

To indicate a specific distance, such as 10 mm, select the **Incremental** option and type the

distance into the appropriate direction box. Click on the green **GO** arrow that appears when the **Incremental** option is selected and the mill will move the indicated distance, in millimeters, in the corresponding direction.

Jog Controls: Incremental Option ON:



Installing a Cutter:

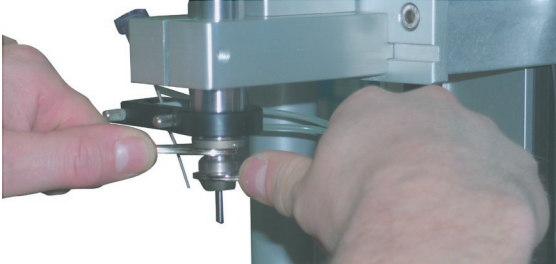
To install a cutter, position the two wrenches over the flats on the spindle and spindle cap and push them in opposite directions toward one another to loosen the cap. It is not necessary to remove the spindle cap completely! Carefully slide the cutter into the spindle cap by grasping it about a 1/4 inch away from the end, leaving the rest of the cutter extending out of the spindle cap.

Note: When handling a cutter, hold it on either side of its cover and DO NOT push on the tip of the cover, since the cutter can easily poke through the protective cover and injure the user.

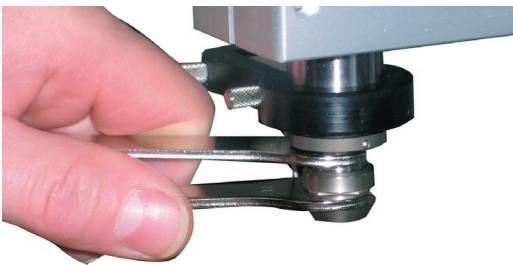
With the cutter installed, tighten the spindle cap by hand to hold the cutter in place, and then place the two wrenches over the flats on the spindle and spindle cap again. Tighten the spindle cap by moving the wrenches away from one another this time.

Note: Do not overtighten the spindle cap! Securing the cutter snugly is sufficient.

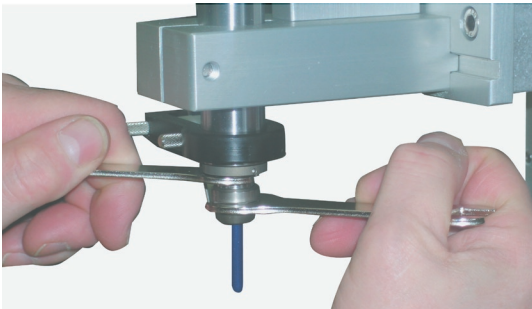
Loosen collet nut by pushing wrenches toward one another:



When collet nut is loose, remove dowel pin or previous cutter:



Slide in cutter till shaft is about 1/4 inch into collet nut and tighten with wrenches:



To remove the cover from the cutter, grasp it on either side: never push on the tip of the cover, since the cutter can easily poke through the protective cover and injure the user if pressure is placed on it.

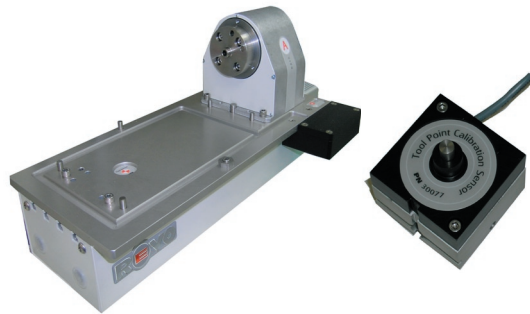
Seek Tool:

Each time a cutter is installed, it is necessary to run the **Seek Tool** operation so that the mill knows the length of the installed cutter. Each time any cutter is installed, a **Seek Tool** operation must be performed. Otherwise, the previous tool length offset will be applied and the part will be cut incorrectly.

*IMPORTANT: Before running **Seek Tool**, DO NOT FORGET to place the Tool Sensor into the Tool Sensor Center notch on the fixture plate!*

To prepare for the **Seek Tool** operation, place the **Tool Sensor** into the **Tool Sensor Center** notch found on the fixture plate. The shape of the locator button in the bottom of the **Tool Sensor** matches the shape and size of the **Tool Sensor Center** notch exactly.

Fixture Plate & Tool Sensor Center / Tool Sensor (cord MUST be connected in the appropriate slot on the left-hand side of the mill or this operation will not work properly!)

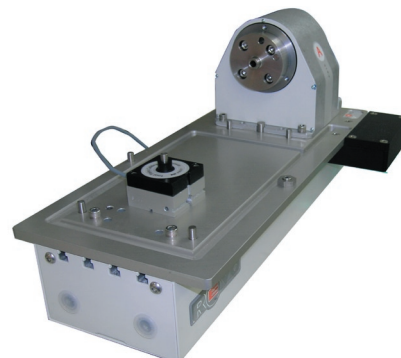


Placing the tool sensor into this notch ensures that the tool sensor will be correctly positioned for the **Seek Tool** operation.

Fit bottom of Tool Sensor into Tool Sensor Center notch:



Tool Sensor properly installed:



Note: If the user intends to use the same cutter again and that cutter was not removed from the spindle at the end of the last job that was run, the Seek Tool operation does NOT need to be performed - even if the mill was turned off between jobs! The mill will store the last recorded value and refer to it as the length of the tool.

After running **Seek Tool**, the mill will provide the values, in millimeters, for both the length of the current tool (the distance that the tool extends from the collet), and the amount that this value differs from that of the previous cutter that was installed.

Note: After running Seek Tool, remove the tool sensor from the table and return it to its resting place on the back left handle of the mill before proceeding! This will help to protect the tool sensor from damage.



The **Seek Tool** operation is performed so that the computer knows the length of the currently-installed cutter. It must be run each time the user changes cutters. However, if it is suspected that the cutter broke during a job, run **Seek Tool** again, right after the job, without changing the current cutter.

If a significant change in value (more than 0.1 mm) is reported by the mill since the last time **Seek Tool** was performed, then the cutter was likely broken during the job and will need to be replaced. If there was no breakage, the “Changed by” value should be right around 0 mm, since the same cutter is being measured during the **Seek Tool** operation.

Mill Control:

Once a file is loaded, the corresponding cutter is installed, **Seek Tool** is used to record the length of the new cutter, the mill is homed, and the appropriate fixture is installed, it is time to start milling the file!

Before clicking on **Start**, which starts the mill job, make certain that there are no foreign objects (or body parts!) in the way of the mill path. Use the **Mill Controls** to start the mill job once all of the above steps have been completed. Click on **Start** to start the file. This automatically controls the **Spindle** and the **Coolant**.

If the **Spindle** and the **Coolant** will be controlled separate from the **Start** command, use the corresponding controls. However, it is unlikely that this would be the case.

Notice that the manual controls on the **Control Panel** and the **Spindle Control Unit** allow for more precise adjustments of these factors; however, again, it is most likely that the user would simply click on **Start** and allow the program to automatically turn on and off the **Spindle** and the **Coolant** as directed by the G-Code.



In the event that the mill must be stopped, click on the **Stop** button. Refer to the section above about **G-Code** to learn how to start up the mill job from the point at which it was stopped.

It is also possible, in the event of an emergency, to stop the job by simply pressing any key (or many keys) on the keyboard. This stops the mill quickly without the need to position the mouse on the screen.

Chapter 5: Using the RevoCam Software

The RevoCam Software:

The **RevoCam** software guides the user through a series of steps that define a toolpath, or, a series of directions that the cutter follows in order to create the part. The toolpath is translated into the G-Code, which makes up the commands that the Revo computer sends to the mill. The toolpath that results from the **RevoCam** software is visible as a series of lines (or, a “path”) that the cutter (or, “tool”) must follow in order to cut the piece from the wax.

There are four strategies in **RevoCam**, or, four different ways in which the toolpath can be built. Each strategy is set up on the mill with a different fixture, and each one has a slightly different preparation process within the RevoCam software. For most of the strategies, the user must begin by preparing support tabs in Rhino before even entering the **RevoCam** software. The preparation necessary for each type of model varies; however, each one is explained in full in the next four chapters of this manual.

The **RevoCam** software guides the user through steps which include the following (the presence and sequence of these steps in each strategy vary):

- (1) input the model and its support tabs;
- (2) create an STL (stereolithography file, or, mesh) out of the entire model, which “wraps” the model in a mesh surface so that the software sees it as one complete piece even if it is a series of un-joined parts, rendering **Booleans** unnecessary;
- (3) define mill depths, or, the depth that the cutter will plunge into the piece to ensure that all parts of the model are machined;
- (4) draw custom borders, or, the area within which the cutter will move (the cutter will not extend beyond this border);
- (5) select the cutter to be used when milling the part;
- (6) define the finish quality (coarse through extra-

fine) with which the part will be cut;

(7) create the toolpath so that the user may inspect it for accuracy;

(8) save the toolpath as a .RVO file: the format that is read by the **Revo Control Software**.

Each strategy employs a different set of mill fixtures, which must be properly affixed to the mill prior to running the mill job. In addition, certain strategies require extra steps within the **Revo** control software to properly run them. Each of the strategies, including the preparation process in **Rhino** (if any), the specific steps to be performed upon it in the **RevoCam** software, the proper assembly of the milling fixtures, and the special processes (if any) required in the **Revo** software are detailed in the next four chapters (one for each strategy). Following that is a complete overview of the **Advanced Options** for each strategy and a series of tutorials whereby the user can practice using the advanced options.

Starting up the RevoCam Software:

With a model that has been properly prepared for milling located in the viewports, click on the **RevoCam** software icon located on your desktop following installation.

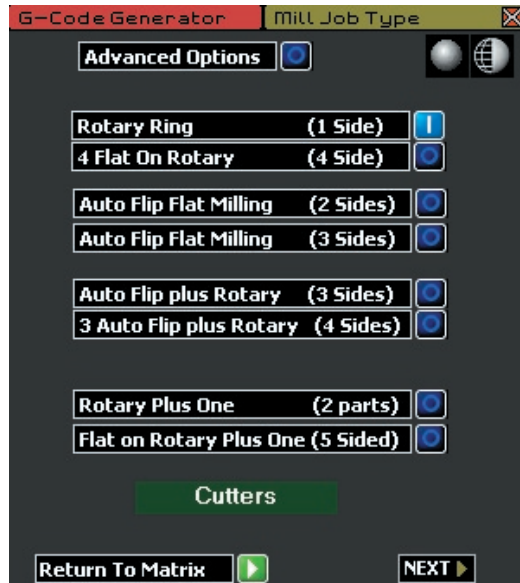


This will open up the **RevoCam** software. Click on the blue button beside a strategy to use it. Select a strategy based on the needs of the model in accordance with the criteria listed below (in the section entitled **Selecting a Milling Strategy**). To activate one of the four strategies, click on the dark blue button beside its name so that the button becomes highlighted in bright blue. Each strategy has a different series of screens that allow the user to control and define various parameters of the toolpath.

Only one strategy may be selected at a time (**Rotary Ring: 1 Side** is selected in the illustration below); however, the **Advanced Options**, which are detailed in full in another section of this manual, may be activated along with any of the strategies in order to change the options associated with that strategy. When a strategy is selected, the

related fixture is illustrated (and highlighted in bright pink) in the mill diagram above the menu.

Rotary Ring (1 Side) is selected below:

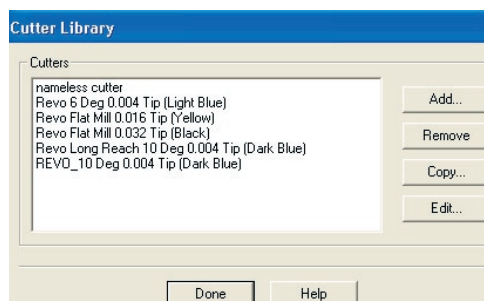


About the Cutter Library

It is usually NOT necessary to click on the green **Cutters** button found in the first screen of **RevoCam**. Doing so takes the user to a library of available cutters. It should be noted that this is NOT the place where the user selects a cutter for this particular mill job; that is done in a future screen in the software.

Here, instead, are the options to add custom cutters, edit the options on existing cutters, and remove cutters that were previously added. The cutters that are pre-programmed into this library are the ones that are included with the mill, and are the standard cutters that may be selected for the mill job.

WARNING: *It is NOT recommended to edit or remove any of the cutters that have been pre-programmed into this list!!*



Notice that each cutter is color-coded by its cap, corresponding to the four cutters included with the mill.

Note: *It is very important not to switch the caps on the cutters! This helps to ensure that the cutter selected in the software is the actual cutter that is installed in the mill. If the two do not match up, serious problems could result, such as the part detail being improperly cut or even cut away completely!*

Do not be alarmed that five cutters are actually listed in the Cutter Library: the “Revo Long Reach 10 Degree” cutter is actually just the Revo 10 Degree cutter (notice that “both” 10-degree cutters correspond to the cutter with the Dark Blue cover); however, it is necessary to select the “Long Reach” version in certain instances, which are described in full in another section of this manual.

The four cutters that come with the mill include two tapered cutters (10-degree and 6-degree) and two flat cutters (0.032 and 0.016). Tapered cutters, as the name implies, are not perfectly straight but taper slightly outward from the tip: the 10-degree cutter has a 5-degree taper on each side of the tip, and the 6-degree cutter has a 3-degree taper on each side of the tip.

ONLY two of the cutters (10 degree tapered and 0.032 mm flat) can cut from rough or uncut wax! The other two (6 degree tapered and 0.016 mm flat) should NOT be used to cut from rough. They are the cutters to select when creating a “second pass”, or, a second toolpath for the same piece, in order to improve definition on the part.

- 10 Degree 0.004 Tip: This tapered cutter has a spiral flute and is the best choice for most work on the mill. It has 16 mm of cut length and can successfully make rough cuts.

- 6 Degree 0.004 Tip: This tapered cutter is best used for secondary passes when the piece has already been cut with the 10 Degree cutter. Although it has 16 mm of cut length, it can only plunge 2.5 mm into wax before it will break, and therefore it cannot be used to cut from rough (uncut) wax.

Tapered cutters: (l-r) 6 degree cutter and 10 degree cutter



- **0.032 Flat:** A flat cutter that is 1 mm in diameter and 8 mm long, this cutter is best used when cutting out flat pieces such as a pendant or a coin, or any time when straight walls for raised lettering or other design elements are desired. It can make rough cuts.

- **0.016 Flat:** This single flute cutter is 0.5 mm in diameter and has only 3.5 mm of cut depth, meaning that it should not be used to cut through wax that is more than 3 mm deep or else it will very likely break. Because this cutter breaks when cutting from rough, it is best used for a second pass on a piece that has already been cut with another cutter. When used this way, it can add a clean finish to fine details and will provide straight-sided walls.

Flat cutters: (l-r) 0.016 mm flat & 0.032 mm flat

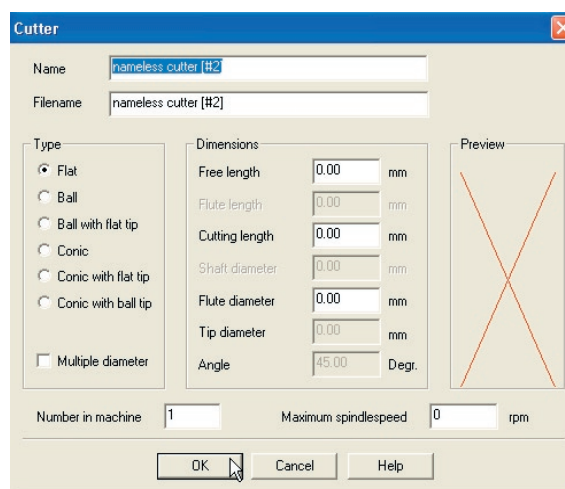


Note: During toolpath creation, the software will calculate for the geometry of the selected cutter so that the cutter does not unintentionally cut into the surface of the model. This means that, if the design elements (such as raised letters or artwork) are too close together, the toolpath will not attempt to direct the cutter to move between them but will instead cause it to pass over that part of the design. For best results, design elements should be a minimum of 0.5 mm away from one another to ensure that the cutter can reach between them.

Editing the Cutter Library:

The option to add, remove, copy, and edit cutters is an advanced function within **RevoCam** and is NOT generally recommended. However, the following instructions define how to use these controls:

Add: Click on **Add** to add a new cutter to this list. This will bring up the following dialog box, which allows the user to specify the name, type, and dimensions of the new cutter. A preview image will appear based upon the specifications that the user provides. If the user has a custom cutter, base these specifications upon that cutter. Whenever a cutter is added (or copied and edited, as described below), the new selection will be available to the user in the screen that facilitates cutter selection.



Remove: To remove a cutter presently in the library, select the cutter and click on **Remove**.

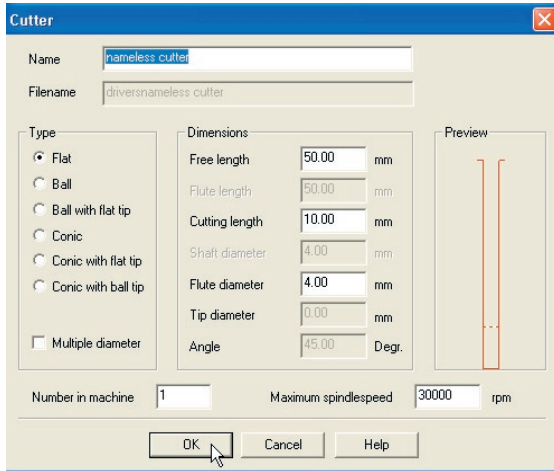
Note: It is strongly recommended NOT to remove any of the cutters that have already been preprogrammed into the library!!

Copy: To create a new cutter based upon a cutter that is already present in the library (either one that has been pre-programmed or one that was added by the user), select the cutter upon which the new one will be based and click on **Copy**. The new cutter dialog box will appear. Change the settings so that they reflect the dimensions of the custom cutter. When finished, rename the new cutter and click on OK to save it.

Edit: To change the settings for a cutter that is already present in the Cutter Library, select the cutter and click on **Edit**. This opens up the dialog

box that displays the current settings for that cutter. Edit the settings that need to be changed, rename the cutter, and click on **OK**.

*Note: It is strongly recommended NOT to change any of the settings for the cutters that have already been preprogrammed into the library! If it is absolutely necessary to edit one of these cutters, first **Copy** the cutter and then edit its settings.*



Selecting a Strategy:

There are seven basic strategies to use when milling models. Each strategy also has its own set of **Advanced Options** that are described in another section of this Help manual. The strategies are available within the **RevoCam** software.

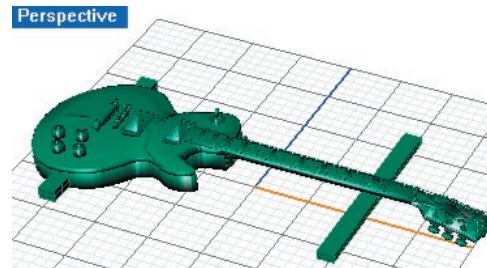
For each strategy, the preparation needed to run the strategy within the **RevoCam** software, the processes run within the **RevoCam** software, and the type of model with which the strategy works best all vary. An overview of each strategy, including when to use it, is found below. Following that are full, detailed directions on how to prepare a model for each strategy in **Rhino**, how to prepare the milling fixtures corresponding to the given strategy, and how to run the strategy on the **Revo** computer. Finally, the **Advanced Options** for each strategy are overviewed and a series of tutorials are provided for practice.

The seven basic strategies available for use in **RevoCam** are: **Rotary Ring (1 Side)**, **Auto Flip plus Rotary (3 Sides)**, **Auto Flip Flat Milling (2 Sides)**, and **Auto Flip Flat Milling (3 Sides)**, **Three-Sided Auto Flip Plus Rotary**, **4 Flats on Rotary**, and **Rotary Plus One / Flat on Rotary Plus One**.

Auto Flip Flat Milling: 2 Sides

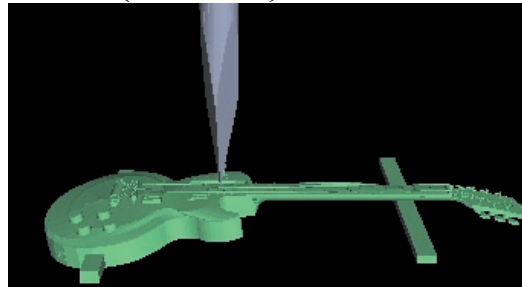
This two-sided strategy works best with a model that lays flat (or relatively flat) along the X/Y plane in **Rhino**.

Auto-Flip Strategy: The model should lie (relatively) flat on the X/Y Plane in Rhino:

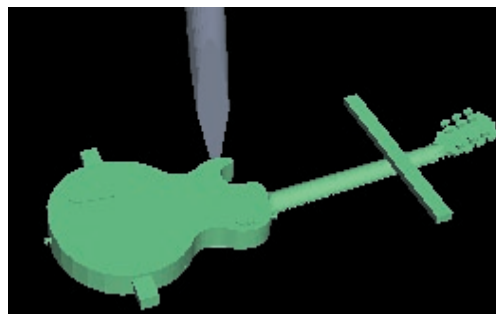


The mill cuts the model by holding the wax flat with the two-sided auto-flip fixture and moving the cutter straight down. The cut begins at the center of the part and works its way outward in concentric “circles” (the standard toolpath is actually made up of straight lines, so they are more accurately “concentric rectangles”). Then, after completing the top of the part, the model is rotated 180-degrees by the A axis and the back is cut. It is therefore fine to have details on the front and/or back of the model when using this strategy.

First side: (cutter shown)



Second Side: (after model is automatically flipped)



It is best to select the auto-flip strategy when the model does not have any important details on the sides, since these cannot be reached by the mill cutter. Notice that the cutter is limited as to the details it can mill on the sides of the piece. Vertical details, such as the “rope” design around this coin, are fine; however, any indentation that is horizontal will be passed over by the cutter.

Acceptable side detail: “rope” on coin below can be reached by mill from both sides.



Any indentation or horizontal cut in the sides of an autoflip piece will be missed during milling:



Note: Each of the following three strategies can be run on models that are centered at the normal orientation for a ring rail in Rhino, wherein the user can see “through” the finger hole of the model when looking into the Front viewport.

Rotary Ring: 1 Side

The rotary strategy is referred to as a “one sided” strategy because the wax is rotated on the A axis while the cutter is moved along the X and Z axes to cut only the details on the top side of the ring shank. Appropriate styles for use with the rotary ring strategy include simple bands and rings with lettering, artwork, or channel settings on the band.

As the A axis rotates the wax on the rotary fixture (1), the cutter moves up and down along the Z axis (2) and side to side along the X axis (3) to cut the part.



When choosing this strategy, a straight-sided shank, such as that in the ring pictured above, works best. If the side of the shank tapers toward the finger rail, or if there is an undercut in the side of the shank so that it curves in toward the finger rail, those details will be missed by the cutter.

In the first example, the curvature of the collar as it nears the finger hole will be missed by the cutter, creating straight-sided collars instead. Similarly, the sides of bezels or other designs that curve or taper inward toward the finger hole will not be cut properly; they will be straight-sided, instead.



The mill cuts next to, but not into, details on the sides of the shank during the Rotary Ring strategy. The engraving in the first design shown below will be missed by the rotary strategy. Conversely, a ring with straight-sided details on the sides of the shank will have no problem milling correctly using the rotary strategy.



For best results, select the rotary ring strategy when there are no details on the sides of the shank, since this strategy will not be able to cut these details properly. To properly mill models with any of the details listed above, try the next strategy: **Auto Flip Plus Rotary (3-Sided)**.

Auto-Flip Plus Rotary (3-Sided):

This strategy solves many of the limitations of both the auto-flip and rotary strategies. It starts with a two-sided auto-flip operation that reaches the details on the front, back, and inside of the shank, and then performs a rotary operation that reaches the details on the outside of the shank. It is the most comprehensive of the basic strategies, since it reaches all parts of the ring that can be cut without using the Advanced Options.

Auto-Flip Flat Milling: “Part 1” cuts both sides (front, back, and inside of the ring) using the two-sided flat operation; then “Part 2” cuts the ring on the rotary fixture to reach the remaining details:

Part 1: Same as two-sided auto-flip

Part 2: Same as single-sided rotary



Use the **Auto-Flip Plus Rotary (3-Sided)** strategy to mill a ring that is detailed all the way around the outside of the shank and has important details on the front and back of the shank, or one that tapers or curves down toward the finger hole, as in the case of a comfort fit shank.

Auto-Flip Flat Milling (3-Sided):

When the details on the front and back of the shank and the top of the head are significant, but the outside of the shank is relatively plain, it is best to use the Auto-Flip Flat Milling (3-Sided) strategy. This strategy holds the wax upright to cut out the head details first, and then returns it to the traditional two-sided auto-flip positions and completes the front and back of the model as usual.

The mill first cuts the head details out by holding the ring upright on the two-sided auto-flip fixture.

Then, the ring is laid down and the usual two-sided operation is completed:



This strategy is best used when the bottom and lower sides of the shank are not detailed. Plain shanks with elaborate heads are especially suited for use with this strategy.


Appropriate for the Auto-Flip Flat Milling (3-Sided) strategy:




Conversely, however, details on the bottom or sides of the ring shank will not be reached during this strategy. On the following rings, only the **Auto-Flip Plus Rotary (3-Sided)** will get to the details on the sides of the shanks. The **Auto-Flip Flat Milling (3-Sided)** will not “cut it” on these models (no pun intended). In each case, the **Auto-Flip Plus Rotary** strategy (the successful one, represented by the gold star) is compared to the **Auto-Flip 3-Sided Strategy** (the unsuccessful one, represented by the red slash) to demonstrate the ways that the auto-flip three-sided strategy may fail. In each case, the **Auto-Flip Plus Rotary (3-Sided)** strategy should be used, instead.

The holes in the side bezels will not be correctly cut


by the auto-flip 3-sided strategy. Use the rotary 3-sided strategy, instead.


 = Auto-flip flat milling (3-sided)

 = Auto-flip plus rotary (3-sided)




The split shank details will be missed by the auto-flip 3-sided strategy. Use the rotary 3-sided strategy to cut it, instead.


 = Auto-flip flat milling (3-sided)

 = Auto-flip plus rotary (3-sided)



The split shoulders in this ring will be missed by the 3-sided auto-flip strategy; however, the 3-sided rotary strategy will be able to cut them.

 = Auto-flip flat milling (3-sided)

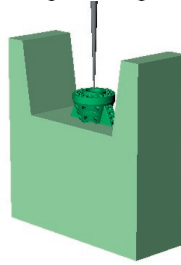
 = Auto-flip plus rotary (3-sided)



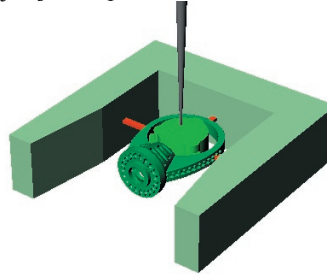
3-Sided Auto-Flip Plus Rotary:

This strategy combines the best parts of the last two strategies discussed above. First, the mill cuts the top-down angle of the 3-sided strategy. Then, when cutting the auto-flip operations, the inner hub used for the auto-flip plus rotary strategy is milled out. Finally, the model is moved to the A axis to cut a full rotary pass.

1: Top-down pass to complete head:



2: Front- and back- of ring are milled in **Two-sided Flat Milling** operations, together with hub for final operation:



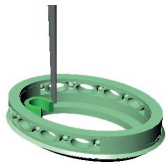
3: Ring is completed on **Rotary** strategy with hub and 3-pronged mandrel holding it to the A axis rotary fixture:



Rotary Plus One and Flat on Rotary Plus One Strategies:

The **Rotary Plus One** strategy begins with a one-sided flat operation and ends with a rotary pass. This strategy is appropriately used on detailed heads, pendants or earrings that are small and/or do *not* have a hole in the center large enough to include a center hub from the auto flip plus rotary operation. In such cases, as in the example below, this strategy is the next best choice.

Part 1: Flat (one-sided) operation cuts detail from top down:



Part 2: Rotary operation cuts detail all the way around

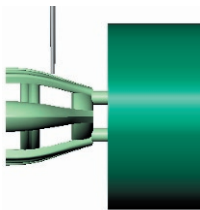


The **Flat on Rotary Plus One** strategy is similar to the **Rotary Plus One** operation in that it uses the same fixture and starts with a top-down pass. Next, instead of continuously rotating the piece on the A axis as it is being cut, this strategy holds the piece flat at a right angle, cuts the part, and then stops cutting and rotates it 90-degrees. After rotating, it cuts the part flat again, repeating this operation three times until all four sides of the part have been cut.

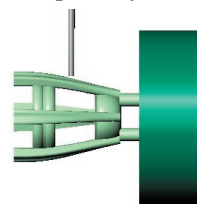
Side one: Top-down flat milling (one-sided ONLY)



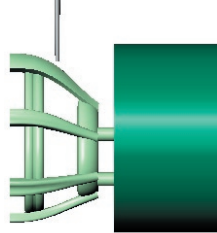
Side 2:



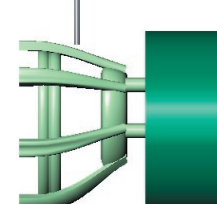
Side 4: (A axis rotates again, then mill cuts while part is flat)



Side 3: (A axis rotates part 90-degrees, then holds it flat while cutting this side)



Side 5: (A axis rotates again, then mill cuts while part is flat.)



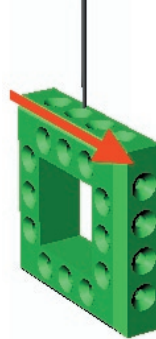
Like the **Rotary Plus One** strategy, the **Flat on Rotary Plus One** strategy is appropriate for use on heads; however, it is the preferred strategy to

use when accommodating heads with straight sides, or those with squared-off prongs that can best be reached by a cutter held at straight 90-degree angles.

In the illustration below, note the difference between a round-sided model and a flat-sided one: the flat-sided model is best cut with the **Flat On Rotary** operation and the round-sided model with the **Rotary Plus One** operation.

(L) Best Strategy for this part is Flat on Rotary Plus One: A axis holds model flat during cutting so that cutter movement is parallel to straight-sided parts. (R) Best Strategy for this part is Rotary Plus One: A axis is rotated while model is cut to properly finish domed sides or curved details.

Flat on Rotary Plus 1:



Rotary Plus 1:



In the same way, when milling parts with squared-off prongs, the most effective strategy is usually the **Flat On Rotary Plus One** strategy, because it can better clear the space around the prongs:

(L) Rotary Plus One strategy misses square edges of prongs; (R) Flat on Rotary Plus One cuts edges of prongs square:

*Rotary + 1:
Incorrect*



*Flat on Rotary + 1:
Correct.*



4 Flats on Rotary:

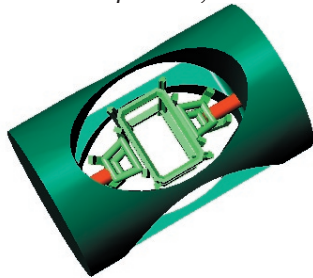
For models where you wish to obtain flat details on all four sides (you do not want the A axis to turn while cutting the model), use the more advanced 4 flat on rotary strategy.

Most useful for heads, pendants, and other small objects that will fit “inside” a wax tube, this strategy does not require the use of a finger size (or, ring rail), so it cuts the model away from the inside of the wax tube.

By carefully setting up a support structure and custom borders before you enter **RevoCam**, you define wax supports that hold the model to the wax tube during milling.

After cutting the top-down pass, the mill rotates the model 90-degrees, then stops turning and holds it still on the A axis while cutting the next quadrant. This continues until all four sides of the model have been cut.

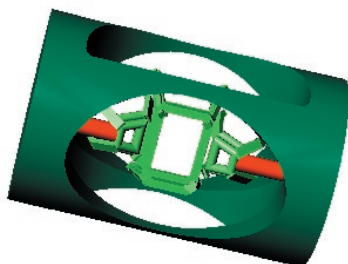
Mill cuts top-down;



Holds model still on A axis at 90-degree angles to cut both sides;



Also cuts bottom-up, finishing hard-to-mill models such as heads and pendants.



Sample models for use with each strategy:

Auto-flip 2-sided: details on top and bottom of model; minimal details on sides of model.



Rotary 1-Sided: details all around the sides of ring shank; minimal or no details on front or back of shank.



Auto-Flip Flat milling (3-sided): rings or other models with details on top (head), front, and/or back of shank (NO details at bottom or sides)





Auto-Flip Plus Rotary (3-Sided): rings or other models with details on front and back of ring and all around the shank.



Chapter 6: Preparing & Milling the Rotary One-Sided Strategy

About This Strategy:

Remember that this strategy is best used on a ring with details around the outside of the shank. For a model that is true to the design, the shank should be straight-sided, not tapered or curved toward the finger hole, and there should be no (or minimal) detail on the front and back sides of the ring.

The rotary strategy is referred to as a “one sided” strategy because the wax is rotated on the A axis while the cutter is moved along the X and Z axes to cut only the details on the outside of the ring shank. Appropriate styles for use with the rotary ring strategy include simple bands and rings with lettering, artwork, or channel settings around the band.

Appropriate for rotary strategy: Details around shank and straight sides.



Inappropriate for rotary (one-sided) strategy: Details on front and back of shank.

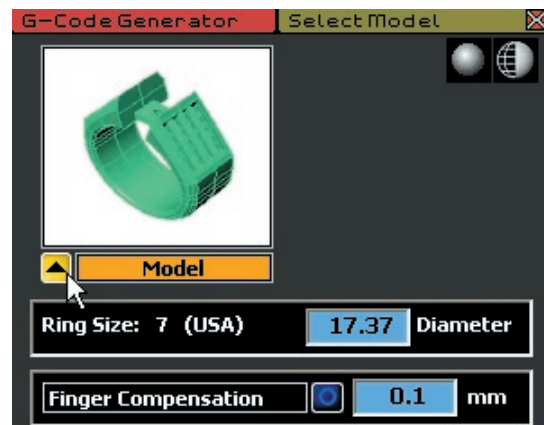
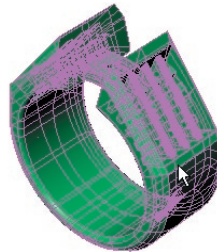


Using the RevoCam Software:

Open the **RevoCam** software by clicking on the icon found on your desktop after installing **RevoCam**. In the first screen of the software, select the **Rotary Ring** strategy.



Click on **Next** to proceed in the software. In the next screen of the software, select the model and input it into the **Model** preview window by clicking on the yellow “IN” arrow beneath the **Model** window:



If the model was created in Matrix, the ring rail will be read automatically. If not, the user will need to manually enter in a Diameter, in millimeters, for the finger size of the ring.

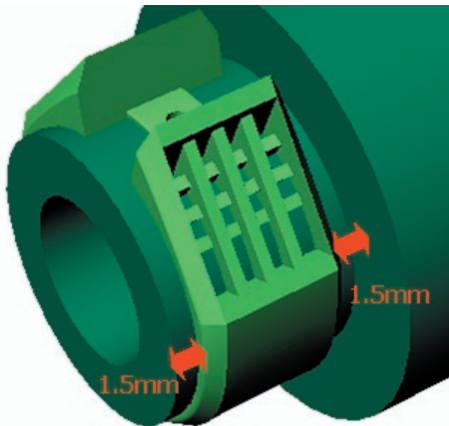
It is very important to include the correct ring size

because this value is used to “trim” the toolpath, ensuring that the cutter will not go any deeper than the depth of the ring rail.

The distance to which the mill will “Trim” the toolpath is 1.5 mm past the widest part of the ring, all the way around the ring. This ensures that there will be a small “overcut” on either side of the band so that the ring can be sawed off the wax tube after milling without any damage to the model. This value also creates a witness to assist in finishing the finger size of the ring, since the overcut describes the finger size of the ring.

Representation of finished model showing toolpath overcut: 1.5 mm is left on either side of the ring to leave space for the clean removal of the ring from the wax tube.

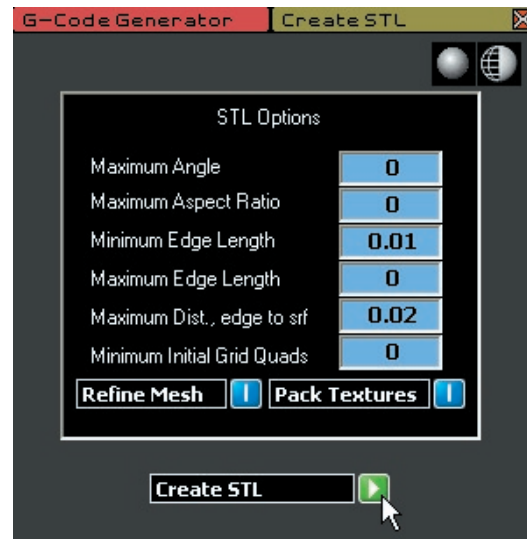
(Finished ring light green / Wax Tube dark green)



Create STL: After all of the settings are correct, click on **Next** to proceed to the **Create STL** screen in the software. In this screen, it is recommended that the user keep all of the default settings. Changing these defaults adjusts the density and composition of the mesh “skin” that is placed over the model in order to generate the toolpath, and can greatly increase the file size, which affects the performance of both **Rhino** and the **Revo Control** software.

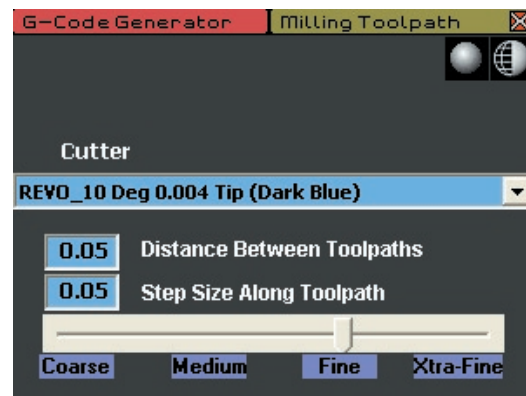
To proceed in this screen, simply click on the green **GO** arrow beside **Create STL**. A fine, steel gray mesh will be wrapped around the whole model, creating a tight seal over every part. Because the software uses this mesh to create the toolpath, all of its pieces are recognized as one by the mill, even if the model is not booleaned together. The toolpath is based upon the mesh, not upon the separate

pieces of the model, saving users time and sanity by eliminating frustrating Boolean union operations!



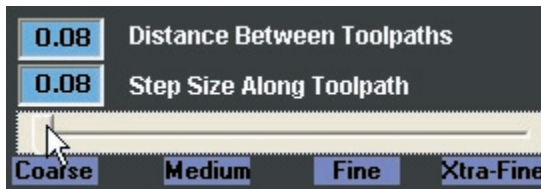
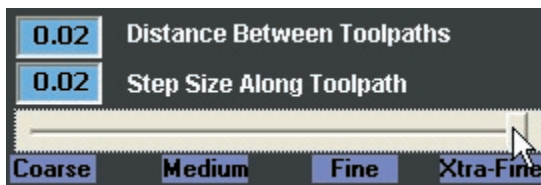
Click on **Next** to proceed in the software.

Selecting a Cutter: After creating the **STL** file, the software offers a **Cutter** selection screen. Recall that the best cutter for most jobs (especially for the first pass) will usually be the **Revo 10 Degree** cutter. This is the default setting in this screen. To change the cutter, click on the drop-down menu arrow and select another cutter from the list by double-clicking on it. The selected cutter will appear in the menu box.



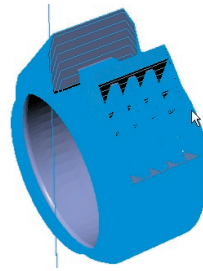


Defining the Quality of the Cut: There are two ways to use the controls that define the quality of the cut. The first is to click and drag the slider control, moving it along the scale from **Coarse** to **Xtra-Fine** until the desired setting is achieved. Raising the quality up to **Xtra-Fine** lowers these numbers. Lowering the quality raises these values.

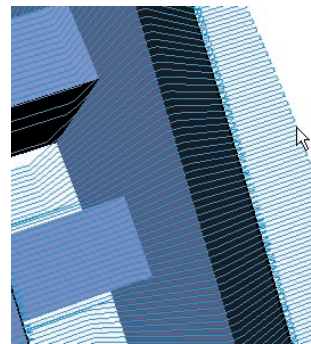


The second way to change these settings is to manually type a new value into the **Distance Between Toolpaths** and **Step Size Along Toolpath** text boxes. Lowering these values increases the quality of the mill job; however, it also increases the time that the mill job will take. For example, raising this value to **Xtra-Fine** will more than double the time it takes to run the job. It is not recommended to raise or lower these values beyond those available on the scale.

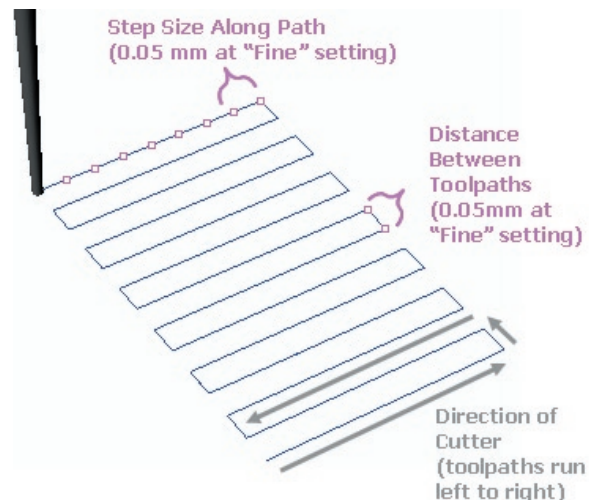
The blue text fields are provided so that the user can change one value but not the other. For example, leaving the slider setting at **Fine** and changing the **Distance Between Toolpaths** to 0.3 will increase the finish quality to basically the same as the “extra fine” setting without doubling the mill time: it only adds about 20 minutes to an average mill job. To understand what each value means, it is important to understand what a toolpath is. The following is an illustration of the toolpath for the ring used in this example.



Zooming in very close to the surface of the ring will reveal that the toolpath is actually several lines that define where the cutter will cut. It is, quite literally, the path (or line) that the tool (cutter) follows in order to produce the part.



Another representation of the toolpath, though not created to scale, is found below. The line that makes up the toolpath is essentially a lot of tiny “rows” that are connected at each end in a back-and-forth or “zigzag” pattern. After completing one “row”, the cutter turns around, creating the squared-off end, and completes another row, headed in the opposite direction. The space between each “row” is equal to the value listed as the **Distance Between Toolpaths**.



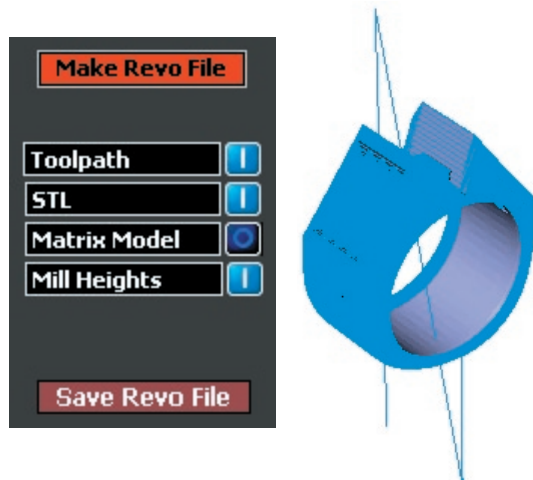
Step Size Along Path: This value refers to the distance between the reference points that the cutter uses to orient itself to the part. Because it cannot calculate every possible point along the line, since the mathematics would be staggering, the mill computer instead plots points along the toolpath and the cutter advances to these points.

The further away these points are, the less detail will be achieved in the finished part. The closer together these points are, the more detail will be achieved; however, the mill job will also take longer to run. The **Step Size Along Path** defines how close these points are along the toolpath. As illustrated above, for a “Fine” milling job they are 0.05 mm apart, which is pretty close. So, in a way, when the mill follows a toolpath, it is really just playing an elaborate game of “connect the dots”!

After selecting a cutter and setting the values for these two parameters, proceed to the **Next** screen in the software, **View and Save RVO File**.

View and Save RVO File: In the final screen of this software, begin by clicking on the red **Make RVO File** button. Wait a few minutes, and the toolpath will be created around the model. Now, use the remaining controls to view the toolpath, STL mesh, Rhino Model, and/or the Mill Depths as desired, checking the toolpath that was drawn by the software.

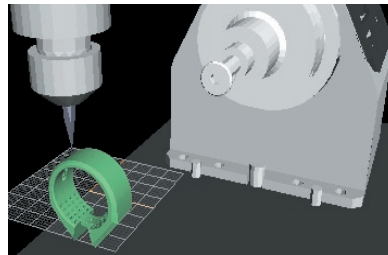
Using these controls, inspect the model to ensure that the toolpath is correct and then click on **Save Revo File**. In the illustration below, the default settings are shown; however, it is possible to turn on and off the **Toolpath**, **STL**, **Rhino Model**, and **Mill Heights** controls as needed to compare the toolpath to the model, ensuring its accuracy.



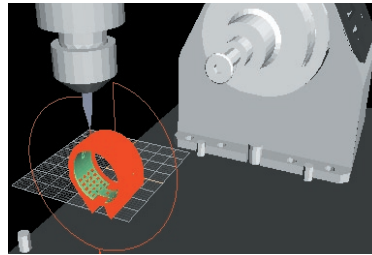
Understanding the Rotary Ring One-Sided Toolpath:

After the axes move to position the wax, this toolpath begins with the mill moving down on the Z axis toward the wax. Then, the A axis rotates the wax around so that the mill will start cutting at the bottom of the ring. This movement is depicted by the large “zigzag” line through the model. Once the toolpath is finished, the cutter returns to the **Home** position.

Starting and ending position of ring / cutter:

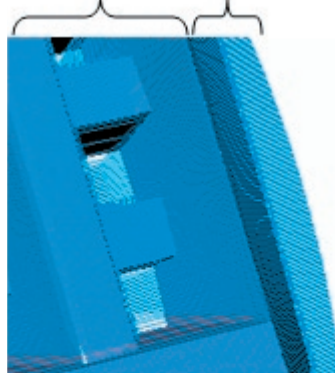


Same view, with toolpath shown:

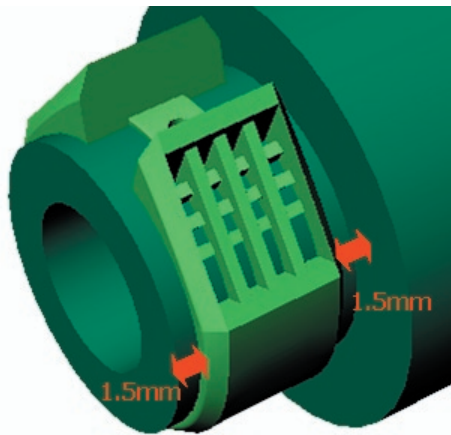


Also, remember that this toolpath has an “overcut” of 1.5 mm on either side of the ring that proceeds all the way around the ring to provide room for the user to cut the model away from the wax. This also creates a witness to assist in finishing the finger size of the ring, since the overcut describes the finger size of the ring.

Surface of ring Overcut: 1.5 mm



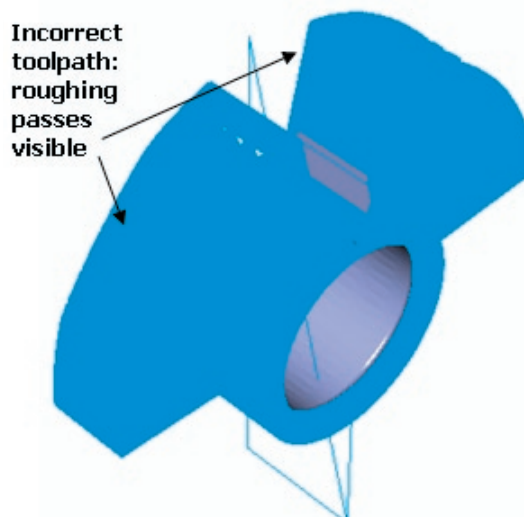
Representation of finished ring on wax tube:



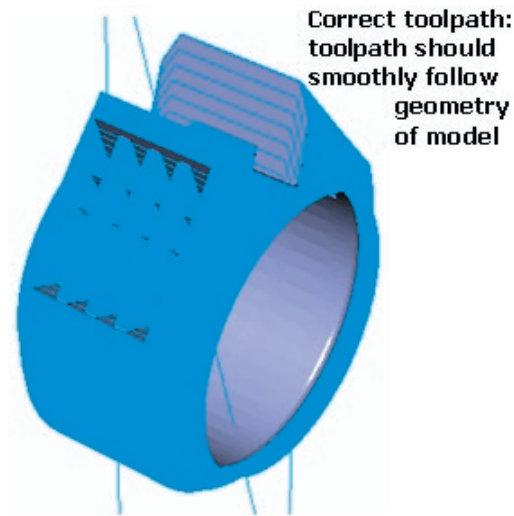
A bad toolpath on the rotary one-sided strategy is evident when there are roughing passes automatically added to the toolpath. These passes represent cuts in the wax that do not reach all the way to the surface of the ring. The roughing cuts free up space for the cutter so that, during the actual pass, the cutter would not be plunged so deeply into the wax that it would break.

If these roughing passes are evident, it means that the selected cutter is too short for the job. To repair this, go **BACK** to the **Cutter** selection screen and select a longer cutter. In the illustration below, compare a correct and an incorrect toolpath on the same ring:

Incorrect toolpath: (described cut length of selected cutter is too short, so the program includes a number of roughing passes)



Correct toolpath: (10-degree cutter is long enough that roughing passes are not needed)



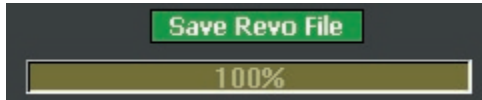
The illustrations above were created using the **Toolpath** and **STL** settings turned on in the final screen of the software. To view the original Nurbs model and compare the accuracy of the toolpath against that model, turn on the **Nurbs Model** setting, which turns off the **STL** setting. To see only the model, not the toolpath, turn OFF the **Toolpath** button.



In this strategy, the **Mill Heights** setting does not affect the view: instead, the mill height is determined by the finger rail – in effect, the toolpath is trimmed by the finger rail – so that the mill only cuts to the depth of the finger rail and no further. This also creates a witness to assist in finishing the finger size of the ring, since the overcut describes the finger size of the ring.

Once the toolpath has been deemed satisfactory, click on the **Save Revo File** button and indicate a name and location on the computer to which the file will be saved. When the progress bar reaches 100%, the file has been saved. To help you remember where the file is saved to, it is a good idea to first navigate to the desktop of the machine and save the file here. When naming the file, it is a good idea to include the selected cutter in the filename (e.g.:

“Ring_10dgr.rvo”). This will enable you to install the correct cutter before loading the ring in the **Revo** software. The file is now ready to be opened with the **Revo** control software found on the mill computer.



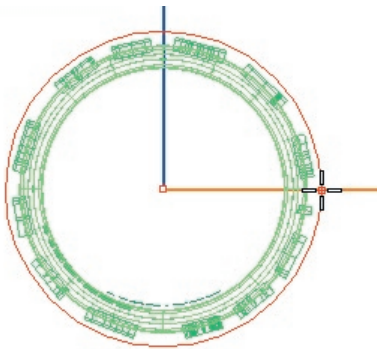
Preparing the Mill for a Rotary One-Sided Job:

The three-jaw chuck is suitable for holding wax tubes up to 40 mm in diameter. For best results, the ring should be slightly smaller than 40 mm in diameter.

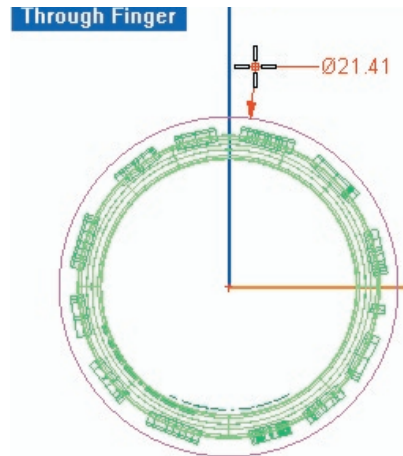
To determine the diameter of the model, open up the model in **Rhino** and select the **Circle** command. In the **Through Finger (Front)** viewport, press 0,0,0 to indicate the center of the circle and move the mouse until the circle just passes the outer edge of the model. Click once to place the circle and end the command.

Next, select the circle and click on the **Dimension Diameter** tool. The diameter will appear beside the circle. Click once to place it on-screen.

*Create a Circle just beyond the outer edge of the ring, using the **Circle** command in Rhino:*



Use the Dimension Diameter tool to measure the circle:

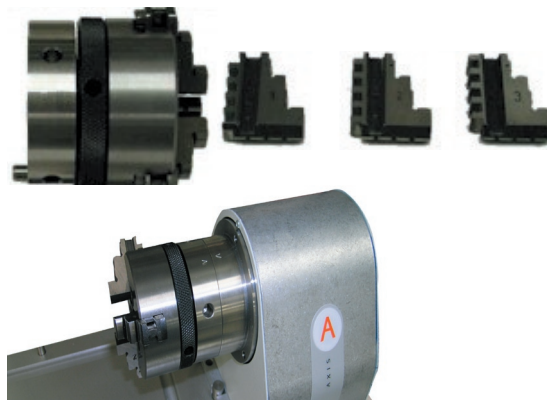


Next, compare the diameter of the circle (21.41 mm, above) with the diameter of the wax tube you have on hand. The tube should be a bit larger than the diameter of the ring; if it is up to 40 mm, and your ring is less than that, you are safe to use this fixture to mill this ring.

Assembling the Three-Jaw Chuck:

Use this fixture with the rotary ring strategy for a tube of wax with a diameter up to 40 mm (the ring should be less than 40 mm in diameter for best results). Prepare the three-jaw chuck fixture for assembly by placing each “jaw” into its corresponding slot in the chuck (jaws and slots are numbered). Tighten the collar by hand so that the jaws are held in place by the threads in the base of the chuck.

3-jaw chuck disassembled / assembled:

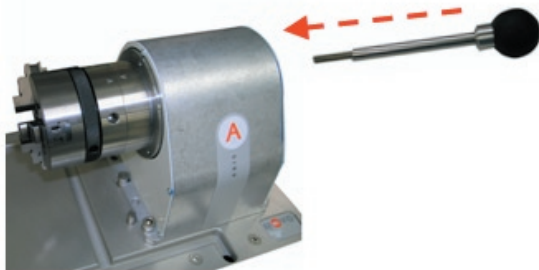


To insert the chuck into the A-axis fixture plate, line up the pin in the chuck with the slot in the plate (the “V” markers on the top of both the chuck and the plate should match up).



Next, insert the draw bar into the other side of the A axis fixture plate and tighten it, securing the three-jaw chuck to the fixture plate.

Insert draw bar into assembly and tighten:



Loosen the collar on the chuck, widening the space between the jaws to make enough room for the wax tube. Hold the tube between the three jaws and tighten the collar, clamping down on the tube with the jaws.

To tighten the jaws sufficiently around the wax, insert the two chuck wrench pins into the chuck: one on the collar and the other on the fixture itself. Pull the two chuck wrench pins toward one another, clamping the jaws around the wax until it is secure.

Pull chuck wrench pins together to clamp down on wax.



Starting the Rotary Ring in Revo Control Software: Set & Show X Offset

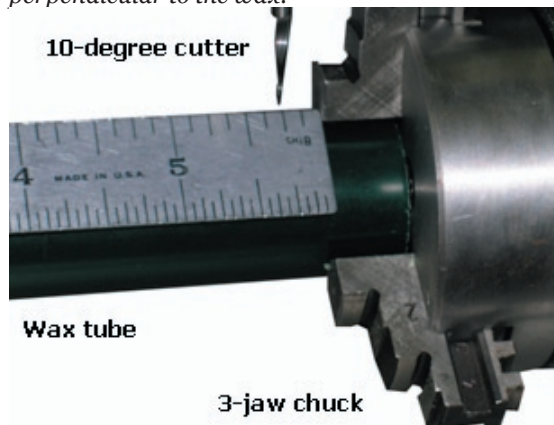
After starting up the mill and the **Revo** computer, click on the red “R” icon on the desktop to open up the **Revo** software. The mill automatically seeks the home position when the software is started. If the cutter is moved for any reason, make certain that the mill is in the “home” position (click on **Seek Home GO**) and click on the **Load RVO Project** icon (the green “disk”) to open the saved .RVO file from either a USB thumb drive, a CD-Rom, or a shared network folder. Navigate to the folder in which the file is saved and click twice on the file to open it in the **Revo** software. It may be necessary to wait just a moment for the file to load completely.

Install the cutter that was selected during the toolpath creation process in **RevoCam**. After the fixture has been properly assembled, with the wax securely in place, the next step is to tell the mill where on the wax to start and end the job. This process uses the **Set X Offset** and **Show X Offset** buttons that only appear in the **Jog Controls** interface during this strategy. Although this is a fairly simple process, it involves manually moving the mill with the **Jog Controls**, so it is imperative to take great care to ensure that the cutter is not inadvertently jogged into any fixtures, wax tubes, foreign objects, or body parts.

To begin, use the **Jog Controls** in the X and Y directions to jog the cutter until it is centered over the wax tube. Next, jog the cutter down on the Z axis until it is a few millimeters above the wax tube. Then, jog it in the “X+” direction until it is approximately 12 mm or 1/2 inch away from the three-jaw chuck. Finally, rotate the A axis until one of the jaws is at a 90-degree angle to the wax (or, until one of the jaws is perpendicular to the table).

Using the **Slow** setting on the **Jog Controls**, continue adjusting the spindle in small increments along each of the axes until the cutter is about a 1/4 inch away from the jaw and a few millimeters above the wax, as shown in the following diagram:

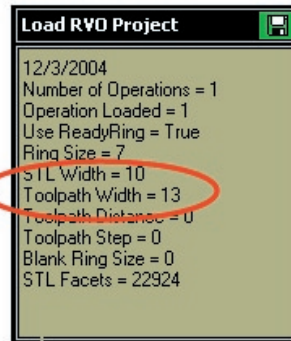
Use Jog Controls to position the cutter in the center of the wax on the Y axis, just above the wax on the Z axis, and about 1/4 inch away from the chuck jaw in the X axis. To ensure that the spindle cap will not hit the jaw, position the A axis so that the jaw is perpendicular to the wax.



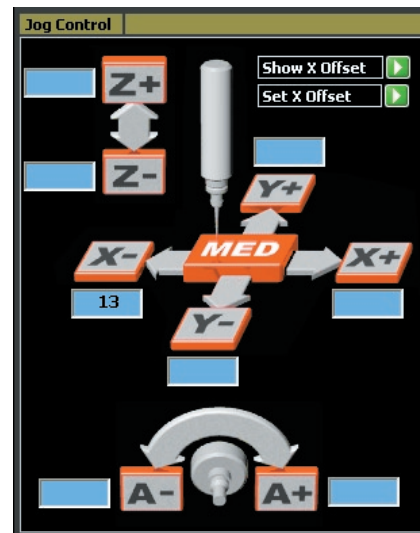
This position marks the right-hand extent of the toolpath, or, the point where the cutter changes direction and moves back toward the left-hand side of the part. Because toolpaths run in a left-to-right direction, the tool will not travel past this extent on the right-hand side of the wax tube, thereby ensuring that the tool will not hit the chuck jaw.

Placing the end of the toolpath this close to the chuck ensures that the user can run multiple rings on one tube of wax, saving as much wax as possible. This process also helps to ensure that the tool cannot run into the 3-jaw chuck during the mill job. Once the end position of the toolpath has been found, the next step is to set the starting point for the toolpath, or, the left extent at which the tool will start cutting the wax. To determine where this location is, check the file information on-screen, found in the lower right-hand corner of the interface, and note the **Toolpath Width**.

Note: If this value includes a decimal (such as 12.45 mm), it is important that you always round it UP to the next whole millimeter value (13 mm, in this case).



In this example, the **Toolpath Width** = 13 mm. To get the cutter to travel back to the left exactly 13 mm from its present position, click on the **Incremental** option beneath the **Jog Controls** in the **Revo** software. Blue text boxes will appear beside each of the directions in the **Jog Controls** interface.



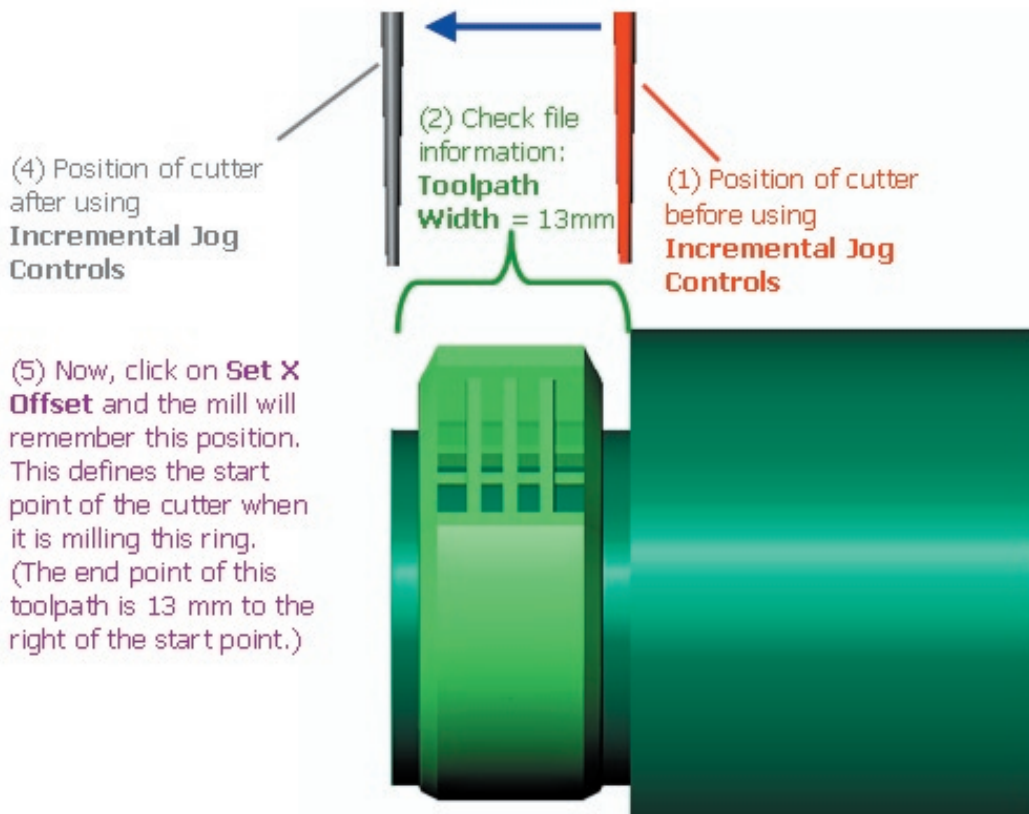
Since the direction that the mill needs to travel is 13 mm to the left of its current position, type "13" into the **X-** box. Then, click on the green **GO** button that appears when **Incremental** controls are turned on.



The mill will travel exactly 13 mm, the width of the toolpath, in the negative X direction. Turn **OFF** the incremental controls after the mill has stopped. Now, the mill is precisely positioned to start the toolpath. Click on the **Set X Offset GO** button found in the **Jog Controls** interface.



(3) Use **Incremental Jog Controls** to jog cutter 13 mm in X- direction.



After clicking on **Set X Offset**, the mill moves up in the Z direction and out in the Y direction. It also stores the **X Offset** value as a permanent fixture offset: therefore, this value does not change until it is re-set, following these same steps, by the user.

Even if the user clicks on **Seek Home**, or if a new job is loaded, the mill will remember this position until another position is set using **Set X Offset**. If you wish to return the spindle to this position to check that it is correct, click on **Show X Offset**.

After clicking on **Set X Offset**, the mill is ready to run (there is no need to click on **Seek Home** before starting the job). Click on the **Start** button in the interface and the operation will begin.



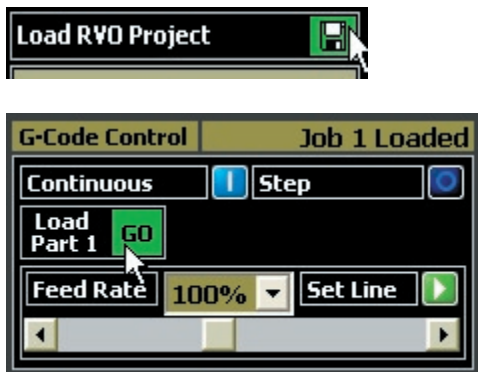
Running more than one rotary ring on a single wax tube:

To save both time and wax, it is possible to run another rotary operation on the same tube of wax without removing it from the fixture.

Note: Although it is fine to run more than one ring on a wax tube when using the three-jaw chuck fixture, it is important to note that the wax tube should be no more than 3 inches long. Not only will a longer tube block the Seek Tool operation, but, when the cutter has worked its way out past the 3 inch mark on the tube, the pressure of the cutter on the wax will push down on the wax, causing it to deflect, or flex, under the pressure of the cutter. This will create an inaccurate model. For best results, cut the wax tube so that it is no more than 3 inches in length, and then set it up for additional rings, as described below.

To begin, load the new job by clicking on **Load Revo Project** and opening the new job from the USB thumb drive, CD-Rom, or shared network drive. (Or, if the same job will be run again, click on **Load Part 1 GO**, found in the **G-Code Control** section of the interface, to re-load the same job so that it can be run from the beginning again.)

To load second rotary ring job, either (1) Load a brand-new project with the Load RVO Project button or (2) Load the same project using the Load Part 1 GO button under G-Code Control.

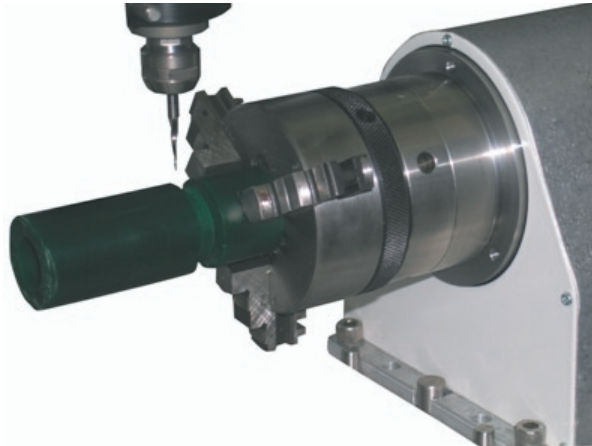


If a new cutter will be used, install it and place the **Tool Sensor** on the fixture plate. Run the **Seek Tool** operation to determine the length of the new cutter. If the same cutter will be used, this operation does not need to be run.

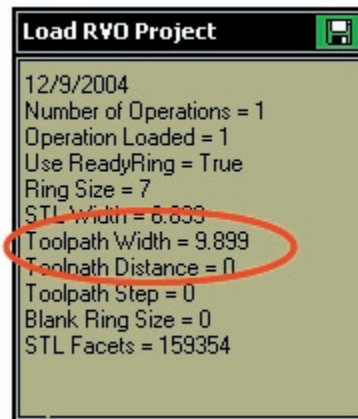
Click on **Show X Offset** and the mill will move to the start point of the toolpath that was run on the

previous ring. This position will now become the end, or, the right-hand extent, of the toolpath for the new ring.

Show X Offset returns the cutter to the X Offset value that was set for the previous rotary ring operation: or, in other words, the beginning of the toolpath for the previous ring. (This will become the end or right-hand edge of the toolpath for the next ring that will be run.)



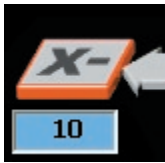
To set up for the next ring, examine the file information for the new file that was loaded and check the width of this toolpath.



In this case, the **Toolpath Width = 9.899** mm. Round this up to "10 mm" and select the **Incremental** setting for the **Jog Controls**.



Enter "10" into the **X-** text box to move the mill 10 mm in the left (X-) direction.



Click on **GO**.



The mill will move to position itself at the starting point for the toolpath of the next ring. After the mill stops moving, turn off the **Incremental** controls. Now, click on **Set X Offset**, which will set the start point for the new ring. The new toolpath will end where the toolpath of the previous ring began, leaving only a distance equal to the **Through Finger Overcut** for both rings between the two; or, a total of 3 mm. (The default **Through Finger Overcut** value for this strategy is 1.5 mm for each ring, or, 3 mm total between each ring.)

Setting up the next rotary ring on the same tube of wax: AFTER loading the new job, proceed with the following steps:

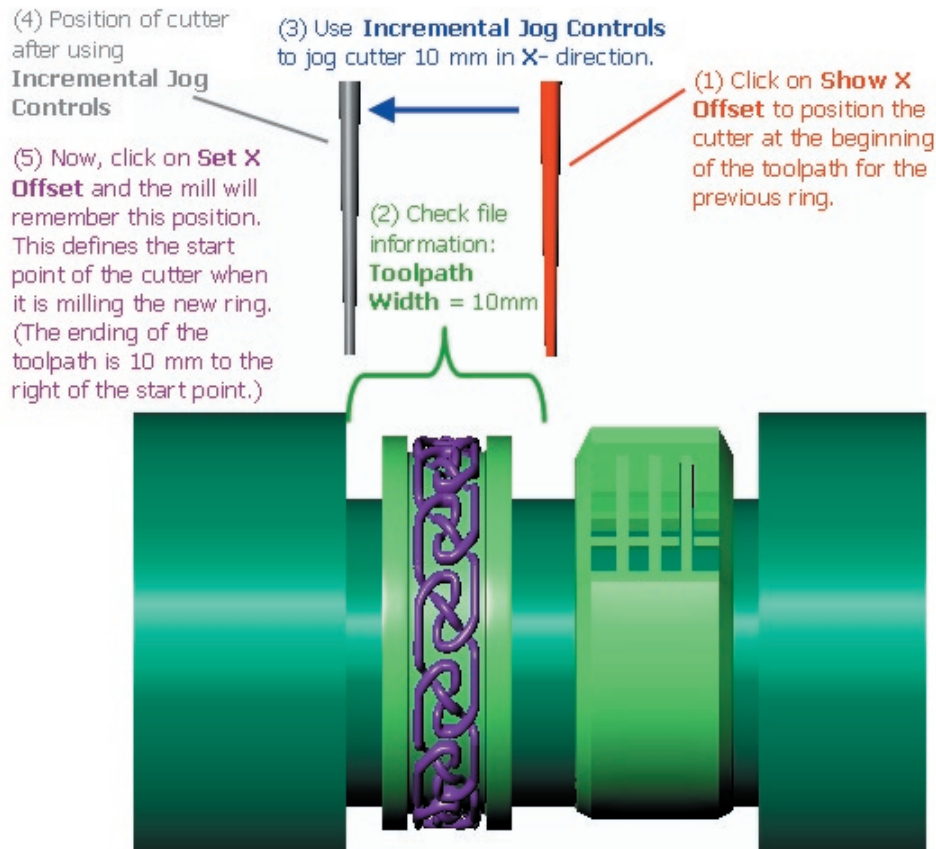
Using **Show** and **Set X Offset**, the user can fit as many rotary rings as possible onto a single tube of wax. However, it is important that the wax tube not be any longer than 3 inches because of the “flex” that occurs at the end of the wax. Past about 3 inches away from the fixture that is holding it, the wax will deflect under the pressure of the cutter.

After clicking on **Set X Offset**, the mill is ready to run the second job (there is no need to click on **Seek Home** before starting the job). Click on the **Start** button in the interface and the operation will begin.



4 Flats on Rotary:

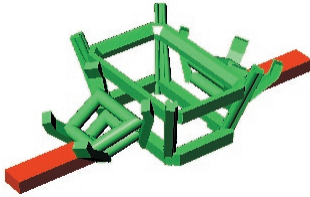
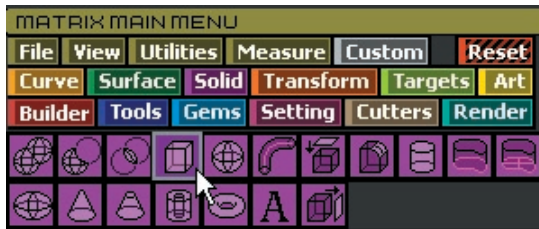
For models where you wish to obtain flat details on all four sides (you do not want the A axis to turn while cutting the model), use the more advanced 4 flats on rotary strategy.



Most useful for heads, pendants, and other small objects that will fit “inside” a wax tube, this strategy does not require the use of a finger size (or, ring rail), so it cuts the model away from the inside of the wax tube.

By carefully setting up a support structure and custom borders before you enter **RevoCam**, you define wax supports that hold the model to the wax tube during milling.

To prepare this model, picture how it will be anchored to the wax tube, and place supports at either end to hold it in that position. Use the **Solid** command **Box** to draw these supports about 1 mm W X 2 mm tall. Extend them far enough past the edges of the model that they will securely hold it to the wax.



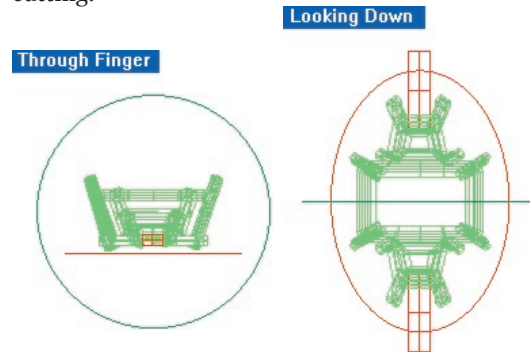
This model requires two borders: one when it is being cut from the top down / bottom up position; and the other when it is being cut from the left / right sides. The first one is drawn in the **Looking Down (Top)** viewport; the second one is drawn in the **Side View (Right)** viewport.

Use the **Ellipse** tool, found in the **Curve** flyout menu, to draw the borders, taking care that they are small enough to provide space for the wax supports that must hold one end of the wax tube to the other end.

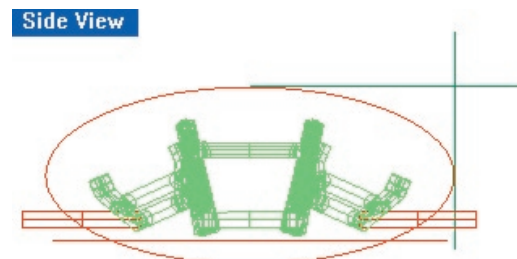
To determine this size, place your model at 0,0,0 in the viewports. Then, draw a **Circle** in the **Through Finger (Front)** viewport that represents the diameter of the wax tube from which you'll be cutting this model. When drawing the ellipse representing the custom border in the **Looking Down (Top)** viewport, DO NOT pass

this curve in space!!

And, of course, do not pass the ends of your supports, or they will fall away from the wax during cutting:



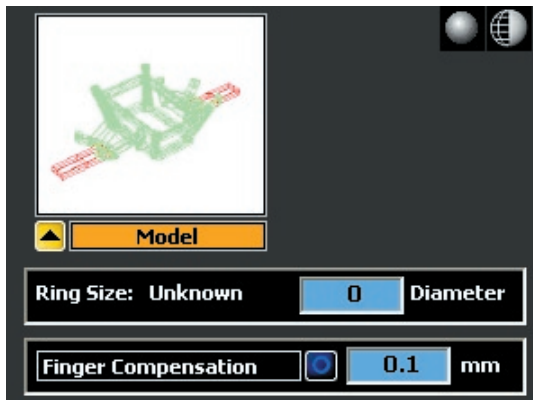
The second border must be drawn in the **Side View (Right)** viewport. To size this one correctly, draw a line from the top straight out to the side; and from the side of the ellipse straight up to the top line. The space in between these two lines represents the wax supports holding one end of the wax tube to the other where it surrounds the model, so they **MUST** be thick enough to hold during milling.



To create the toolpath for this model, enter **RevoCam** and select the **4 Flats on Rotary** toolpath option.



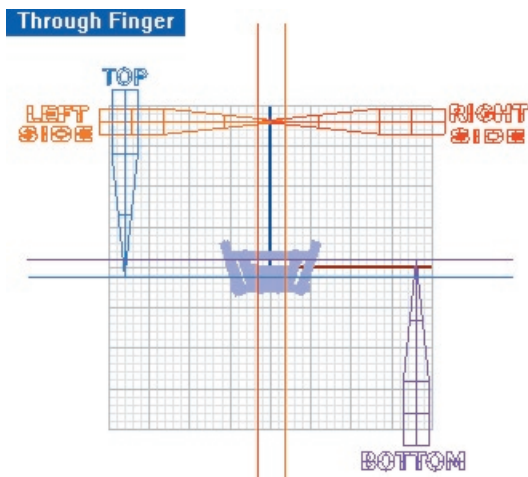
Insert the model into the first screen of the software, disregarding the finger size (the mill does not need to reference a finger size, since there is none used for this model).



Create the STL using the default settings provided by the software:

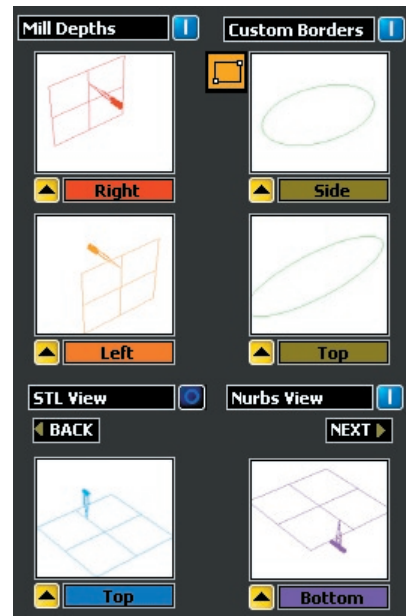


In the next screen of the software, set the Mill Depths by clicking and dragging the grouped Plane object about 1 mm past center on each of the four sides of the model. Insert the mill depths into their corresponding preview windows.

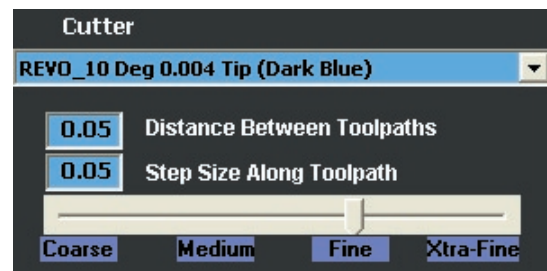


To select the Custom Borders you created above, turn on the Nurbs view of the model, revealing

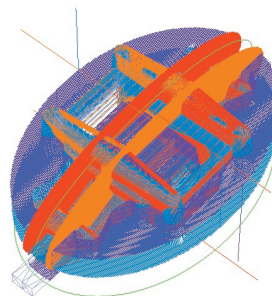
the curves that are turned off when the STL view is showing. When inserting custom borders, remember to select the border in the correct viewport: Select the Top border in the Looking Down (Top) viewport and the Side border from the Side View (Right) viewport!



In the next software screen, select the cutter and finish of choice: accepting the defaults should work fine for most models.



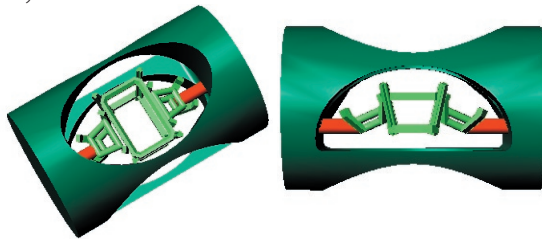
In the final software screen, use Make Toolpaths to see the results of the software. Then, save your Revo file and proceed to the mill to produce the wax.



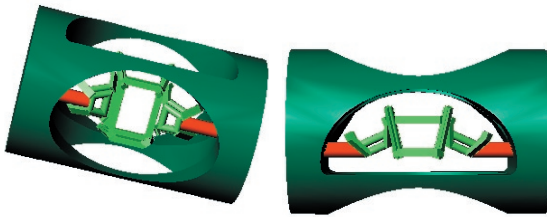
Understanding the 4 Flats on Rotary Toolpath:

After cutting the top-down pass, the mill rotates the model 90-degrees, then stops turning and holds it still on the A axis while cutting the next quadrant. This continues until all four sides of the model have been cut.

(L) Mill cuts top-down; (R) Holds model still on A axis at 90-degree angles to cut one side straight-on;



(L) Rotates Model again and holds it still to cut from the bottom-up; (R) Rotates third and final time to finish fourth of model:



Setting up this Strategy on the Rotary Fixture:

The 3-jaw chuck is set up on the A axis in the same way described earlier in this chapter. When selecting a wax tube for this operation, remember that the diameter must be larger than the supports by a large enough amount to leave sufficient supports to hold one end of the wax tube to the other during milling.

However, it must not be so thick that the shank will rub on the wax supports as it enters tube to cut the model. To prevent against this, take care when setting your mill depths on a very large model that the mill depth plane does not extend more than 15 mm from the edge of the circle representing the wax tube to the center of the part, since there is just 15 mm of cutting length on the 10-degree tapered cutter.

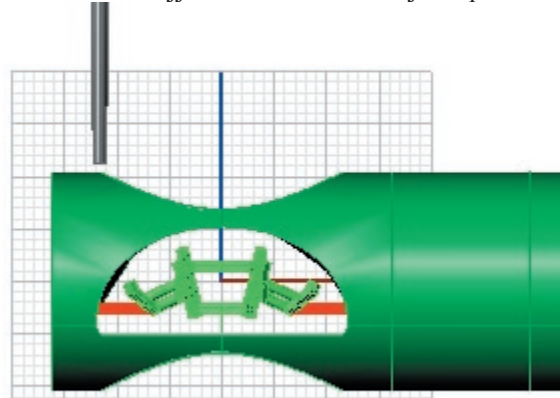
To determine the length of the wax tube, add (1) the length of the toolpath, (2) a 3 to 5-mm “stub” at the end of the wax to hold the “end” of the wax

to the supports of the model, (3) about 5 - 10 mm between the model and the 3-jaw chuck to maintain a safe distance between the cutter and the fixture, (4) about 3 - 5 mm for the jaw to hold onto the wax tube. So, for a 30-mm toolpath, you’ll need about a 50-mm long wax tube.

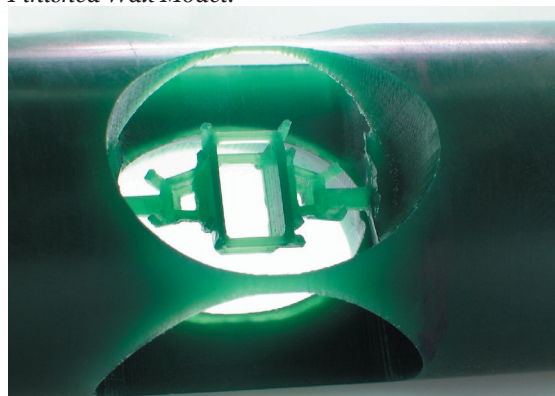
The main difference between set-up of this strategy and the Rotary 1-Sided strategy is the Set X Offset that you’ll run after Loading Part 1 of the job but before hitting Start. As when doing the Set X Offset in the previous strategy, use the Jog controls to position the mill.

However, this time it is ESSENTIAL that you leave about a 3 - 5 mm space between the cutter and the end of the wax tube, jogging the cutter over to the right about 3 - 5 mm up the wax tube! Remember: when using Set X Offset, you’re indicating the starting point of your toolpath (which runs from left to right). So, essentially, you’re leaving this space at the end of the wax tube for the wax to “hang onto” the supports of the model. Don’t make this area too long, but don’t make it too short, either. Between 3 and 5 mm is a good rule to follow!

Jog cutter to 3 - 5 mm in from end of wax tube and use Set X Offset to indicate start of Toolpath.



Finished Wax Model:



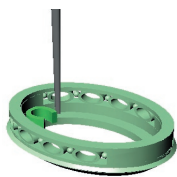
Chapter 7: Preparing & Milling the Rotary Plus One & Flat on Rotary Plus One Strategies:

Comparing the two strategies:

The **Rotary Plus One** strategy begins with a one-sided flat operation and ends with a rotary pass. This strategy is appropriately used on detailed heads, pendants or earrings that are small and/or do *not* have a hole in the center large enough to include a center hub from the auto flip plus rotary operation. In such cases, as in the example below, this strategy is the next best choice.

The one-sided flat operation is used to attain all of the details on the *top side* of the model *only*. Then, the rotary portion of the job cuts the side details of the model all the way around, as when cutting the details on a ring shank during a traditional rotary operation. This strategy is best used for heads and other models where the most important details are found on the top and sides of the model, and where the size or center detail of the model prevents any of the other strategies from being used.

Part 1: Flat (one-sided) operation cuts detail from top down:



Part 2: Rotary operation cuts detail all the way around



The **Flat on Rotary Plus One** strategy, though similar to the **Rotary Plus One** operation in that it uses the same fixture and starts with a top-down pass, runs the rotary part of the operation with a different *twist*, if you'll pardon the pun.

Instead of continuously rotating the piece on the A axis as it is being cut, this strategy holds the piece flat at a right angle, cuts the part, and then stops cutting and rotates it 90-degrees. After rotating, it cuts the part flat again, repeating this operation

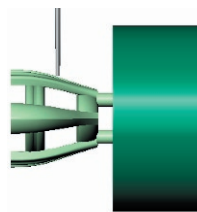
three times until all four sides of the part have been cut. Each of the four cuts is known as a “flat on rotary” pass. By employing this technique, this strategy can truly cut five sides of a piece (top-down plus four sides).

Side one: Top-down flat milling (one-sided ONLY)

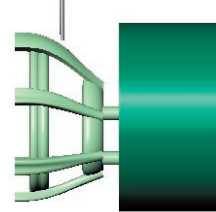


Sides 2-5: Mill does NOT cut the piece while it is being rotated, as in a traditional rotary pass: instead, it rotates the piece 90-degrees on the A axis (WITHOUT cutting it), then STOPS turning the A axis and holds the piece flat while milling only one quadrant of the piece. It repeats this three times until all four sides of the piece have been cut.

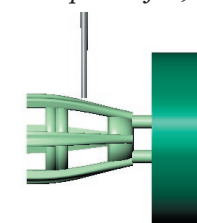
Side 2:



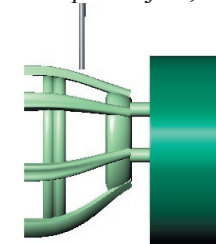
Side 3: (A axis rotates part 90-degrees, then holds it flat while cutting this side)



Side 4: (A axis rotates again, then mill cuts while part is flat)



Side 5: (A axis rotates again, then mill cuts while part is flat.)



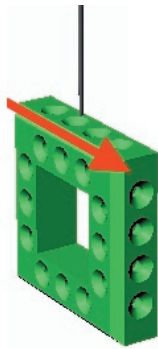
Like the **Rotary Plus One** strategy, the **Flat on Rotary Plus One** strategy is appropriate for use on heads; however, it is the preferred strategy to use when accommodating heads with straight sides, or those with squared-off prongs that can best be reached by a cutter held at straight 90-degree angles.

In the illustration below, note the difference

between a round-sided model and a flat-sided one: the flat-sided model is best cut with the **Flat On Rotary** operation and the round-sided model with the **Rotary Plus One** operation.

(L) Best Strategy for this part is Flat on Rotary Plus One: A axis holds model flat during cutting so that cutter movement is parallel to straight-sided parts. (R) Best Strategy for this part is Rotary Plus One: A axis is rotated while model is cut to properly finish domed sides or curved details.

Flat on Rotary + 1:



Rotary + 1:



In the same way, when milling parts with squared-off prongs, the most effective strategy is usually the **Flat On Rotary Plus One** strategy, because it can better clear the space around the prongs:

(L) Rotary Plus One strategy misses square edges of prongs; (R) Flat on Rotary Plus One cuts edges of prongs square:

Rotary + 1:
Incorrect



Flat on Rotary + 1:
Correct.



Both strategies are prepared the same way in **RevoCam** and the **Revo Control Software**, and both use the same **Revo Mill** fixture.

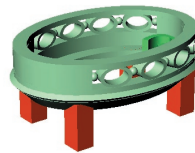
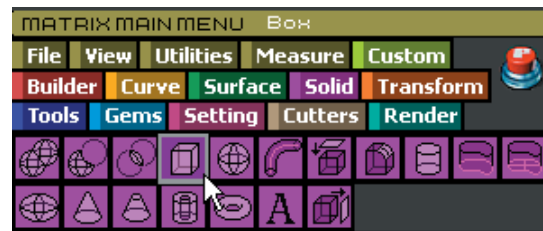
Preparing the Model in RevoCam:

Create the Supports: Before this model can be input into the software, supports must be attached to it that will hold it to the wax during milling. That way, it will not fall away from the surrounding wax while it is being cut. Create supports using the **Box**

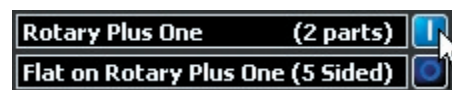
command in Rhino.

These supports will be used to attach the model to the wax in both the one-sided flat milling and the one-sided rotary positions. Because they will be cut during the rotary toolpath by default, it is important not to make them too long, so that the model remains rigid. They should be a *maximum* of 2 mm in length.

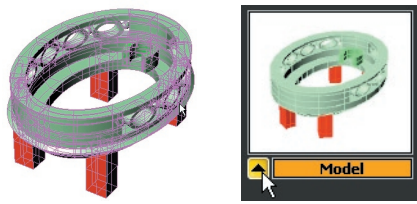
Make certain to place the supports at flat, detail-free areas on the model to ensure the least intrusion into the design and to provide the cleanest cutaway possible. To properly support the model, create boxes that fully sink into the surface of the design and are roughly 1 mm tall X 2 mm wide.



Select a Strategy: With Rhino open, click on the **RevoCam** software icon found on your desktop after installing RevoCam. Then, select the **Rotary Plus One (2 Parts)** strategy.

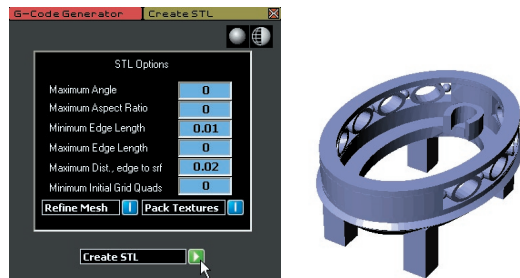


Input your Model: Click on **Next** to proceed in the software. In the next screen of the software, select the model, supports and all, and input it into the **Model** preview window by clicking on the yellow "IN" arrow beneath the **Model** window.



Create STL: After the model appears in the preview window, click on **Next** to proceed to the **STL Options** screen in the software. In this screen, it is recommended that the user keep *all* of the default settings. Changing these defaults adjusts the density and composition of the mesh “skin” that is placed over the model in order to generate the tool path, and can greatly increase the file size, which affects the performance of both **Rhino** and the **Revo Control** software.

To proceed in this screen, simply click on the green **GO** arrow beside **Create STL**. A fine, steel gray mesh will be wrapped around the whole model, creating a tight seal over every part. Because the software uses this mesh to create the toolpath, all of the pieces of the model are recognized as one by the mill, even if the model is not booleaned together. The toolpath is based upon the mesh, not upon the separate pieces of the model, saving users time and sanity by eliminating frustrating Boolean union operations!

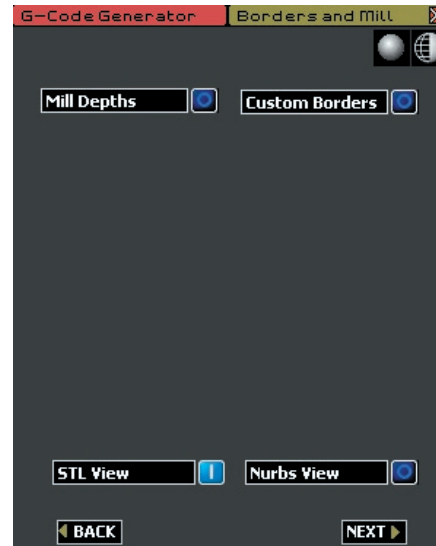


Click on **Next** to proceed in the software.

Borders and Mill Depths: In this screen, the user will input **Mill Depths** and **Custom Borders** using the tools found on-screen. **Mill Depths** define the maximum depth to which the cutter will plunge from the top side of the part during the flat portion of this operation, while **Custom Borders** define the area around the model to which the cutter path will be confined.

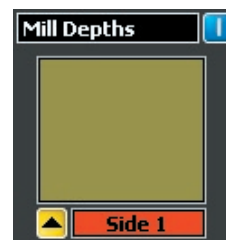
(The **Mill Depths** and **Custom Border** for the rotary part of this operation are automatically set by default to the extents of the model.) The

remaining two controls in this screen allow the user to toggle between the **STL** and **Nurbs** views of this model. You may need to toggle between these two options to select a free form border previously created in Rhino.



Setting Mill Depths: To begin, select the **ON** button beside **Mill Depths** to define the maximum depth to which the cutter will plunge when cutting the top of the part. This activates the tools that are used to define the mill depths: in this case, the **Side 1** mill depth (only *one side* is cut during the flat milling portion of this strategy).

Software screen: Preview Viewports: Mill Depths appear:



Side 1 mill depth is set in the Through Finger (Front) viewport;

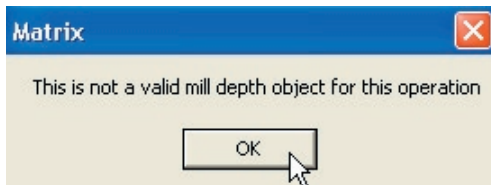


The mill depth is a “grouped” item that consists of a cutter, a plane, and a label. (Because the items are grouped, clicking on any of the items in the group will cause the entire “mill depth” to be selected.) The location of the plane portion of the Mill Depth defines the maximum depth to which the cutter will plunge when cutting the part.

When setting mill depths, it is *NOT POSSIBLE* to set the plane beyond the extents of the part. They must be intersecting the model in order for the tool path to be successfully generated. For best results, set the **Side 1** mill depth for this strategy *only as deep as needed* to complete the details from the top-down; in this case, just beneath the bottom of the bezel will do. Then, input the mill depth into the **Side 1** preview window in the software.



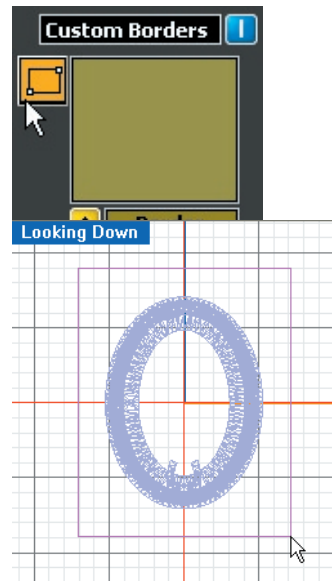
This screen of the software will *not* allow the user to mistakenly place anything *but* the correct mill depth into the **Mill Depth** box, so have no fear! If any other object is selected and the user attempts to input it into this box, the following error message will appear:



If this error message appears, click on **OK** and deselect everything in the viewports by clicking once on empty space. Then, select *just* the mill depth that corresponds to the label on the preview window and click on **IN** beneath that preview window again.

Setting Custom Borders: Once the mill depth has been input into its corresponding preview window, activate the **Custom Borders** control to see the tools used to define a custom border for this operation. This will cause the **Border** preview window and the **Rectangle** curve command to appear in the software screen.

Set the **Border** in the **Looking Down (Top)** viewport. This border represents the extents of the flat milling operation. To begin, select the **Rectangle** command and, working in the **Looking Down (Top)** viewport, draw a rectangle that is about 1 1/2 to 2 mm beyond the outer extents of the part.

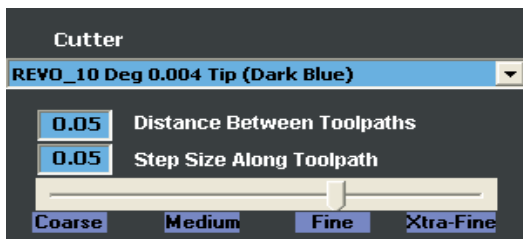


IMPORTANT: Drawing the **Border** in any viewport *EXCEPT* the **Looking Down (Top)** viewport during this part of the process will create an incorrect toolpath!

Once the correct border has been drawn, select the border and input it into the **Border** preview window by clicking on the yellow arrow beneath the window.



Selecting a Cutter: The software offers a selection box on-screen that allows the user to control the surface finish to which the model will be cut: from **Coarse** through **Xtra-Fine**. Recall that the best cutter for most jobs (especially for the first pass) will usually be the **Revo 10 Degree** cutter. This is the default setting on this screen. To change the cutter, click on the drop-down menu arrow and select another cutter from the list by double-clicking on it. The selected cutter will appear in the menu box.



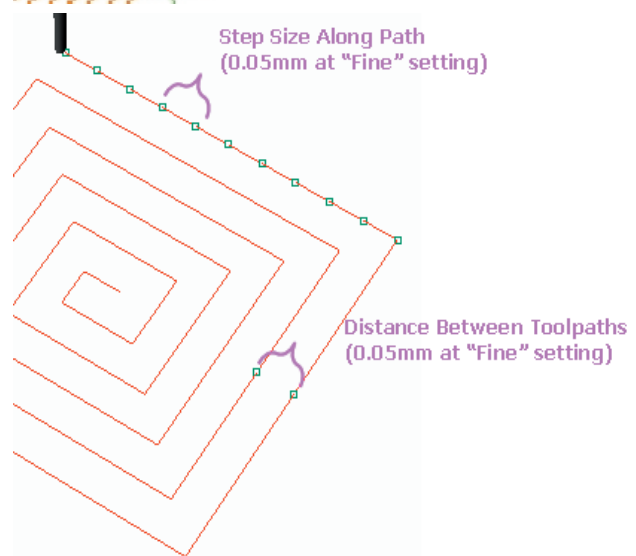
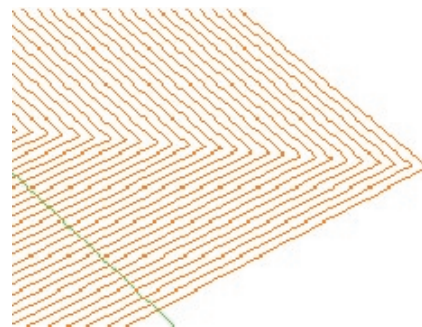
Defining the Surface Finish: There are two ways to use the controls that define the surface finish of a piece. The first is to click and drag the slider control, moving it along the scale from **Coarse** to **Xtra-Fine** until the desired setting is achieved. Dragging the slider to **Xtra-Fine** lowers the numbers in the **Distance Between Toolpaths** and **Step Size Along Toolpath** text boxes, and improves the quality of the finish. Moving the slider to **Coarse** raises these values, thereby decreasing the quality of the finish.



The second way to change these settings is to manually type a new value into the **Distance Between Toolpaths** and **Step Size Along Toolpath** text boxes. Lowering these values raises the quality of the finish; however, it also increases the time that the mill job will take. Raising this value to **Xtra-Fine** will roughly *double* the time it takes to run the job at the **Fine** setting. It is *not* recommended to raise or lower these values beyond those available on the scale. The manual control of these values is provided so that the user can change one but not the other (e.g., for a finer surface finish that only adds about 20 minutes to the average mill

job, set the **Distance Between Toolpaths** to 0.3 while leaving the **Step Size Along Toolpath** at 0.5).

To understand what each value means, it is important to understand how the toolpath is made. A flat toolpath starts from the center of a given side of the model and works outward in a concentric pattern toward the extents of the custom border. Zooming in close to one corner of the toolpath shows the corners of the concentric rectangles. A representation of the toolpath, though not created to scale, is found below. Remember that the start of the toolpath is at the exact geometric center of the given border. The space that is left between the concentric rectangular lines is equal to the value listed beside **Distance Between Toolpaths**.



The **Step Size Along Path** value refers to the distance between the reference points that the software uses to orient itself to the model in a straight line. The space between these points is equal to the value set for the **Step Size Along Path**. As illustrated above, for a "Fine" finish, the points are 0.05 mm apart. The further away

these points are, the coarser the finish will be. The closer together they are, the finer the finish will be; however, the longer the mill job will take. So, in a way, when the mill follows a path, it is really just playing an elaborate game of “connect the dots”!

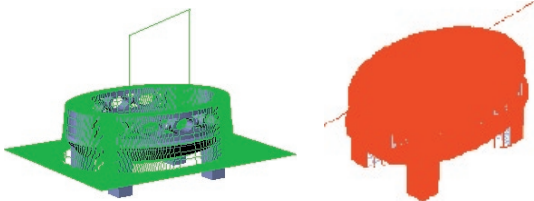
After selecting a cutter and setting the values for these two parameters, proceed to the **Next** screen in the software, **View and Save RVO File**.

View and Save RVO File: In the final screen of this software, begin by clicking on the red **Make RVO File** button. Wait a few minutes, and a toolpath will be generated around the model. Now using the remaining controls, you can turn on and off the STL mesh, Nurbs Model, and Mill Heights as desired to check that the toolpath was correctly generated by the software. If you are satisfied with the results of the toolpath, click on **Save Revo File**.



Understanding the Rotary Plus One Toolpath:

Recall that this toolpath starts in the 90-degree top-down position with a one-sided flat operation; then the fixture is moved to the rotary axis and the piece is completed with a rotary pass. The first operation (on the left, below), the flat milling pass, is the green toolpath; the second operation (on the right below), the rotary cut, is red.



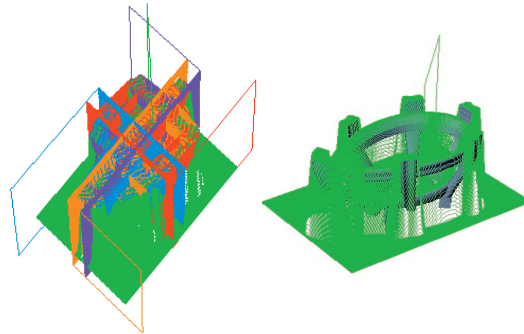
Understanding the Flat on Rotary Plus One Toolpath:

This is certainly the most colorful toolpath! This toolpath starts with the fixture in the 90 degree top-down position with a one-sided flat operation; then the fixture is moved to the rotary A axis

and the piece is completed with four more flat operations, each of them run by holding the rotary axis in a fixed positions at 90-degree angles until all four sides of the part have been cut.

The first operation, the flat-milling toolpath that is cut in the top-down position, is shown in green (on the right, below). The four flat on rotary operations are also shown below. Notice how each one reaches a different quadrant of the part until all four sides are complete.

Flat on Rotary Toolpath: First top-down pass:



Each flat on rotary toolpath (Looking Down view)



Adjust the controls for viewing the **Toolpath**, **STL**, **Nurbs Model**, and **Mill Heights** to see only what is necessary to check the accuracy of the toolpath. To view the original Nurbs model and compare the accuracy of the toolpath against the model, turn on the **Nurbs Model** by clicking on the **Nurbs Model** button, which turns off the **STL** button. To see only the model, not the toolpath, turn OFF the **Toolpath** button. To keep the **Mill Heights** from

blocking the shaded view of the model, turn them off. The default setting is shown below; rest assured that these controls do *not* affect the .RVO file that has just been generated: they only allow the user to better see both the part and the toolpath.

Once the toolpath has been deemed satisfactory, click on the **Save Revo File** button and select a location on the computer to which the file will be saved. When the progress bar reaches 100%, the file has been saved. It is now ready to be opened with the **Revo** software found on the mill computer. Remember: it is always a good idea to save this file to a separate folder on your computer's hard drive; then save a copy of it to the jump drive that you will use in the Revo computer.



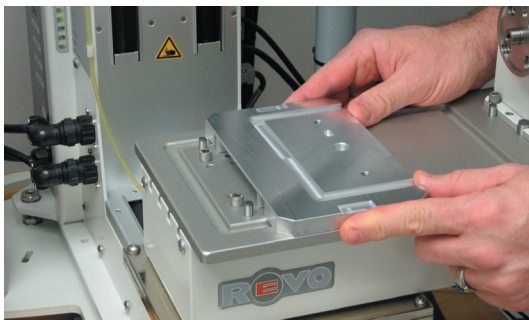
Assembling the Rotary Plus One

Fixture:

Note: To calibrate all new fixtures prior to use, go to the calibration interface of the Revo Control Software and follow the procedures outlined in the new steps.

Use the Rotary Plus One fixture for either strategy: Rotary Plus One or Flat on Rotary Plus One. In the **Revo Control Software**, **Load Part I** of this job and make certain that the proper cutter is installed.

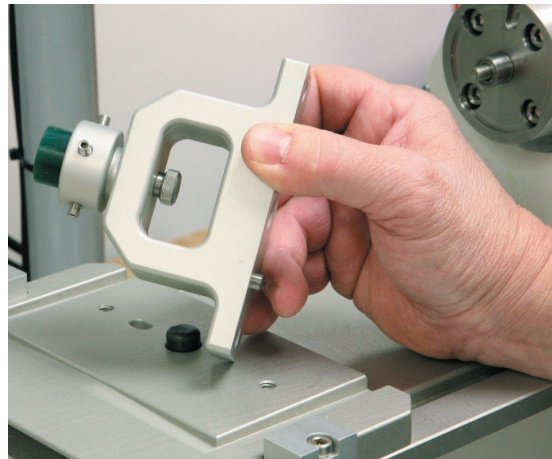
If you have just added a new tool, perform the **Seek Tool** function *before attaching the base plate* to ensure the cutter length is accurately recorded by the mill. Then, attach the base plate to the table, sliding it from right to left until it stops against the pins.



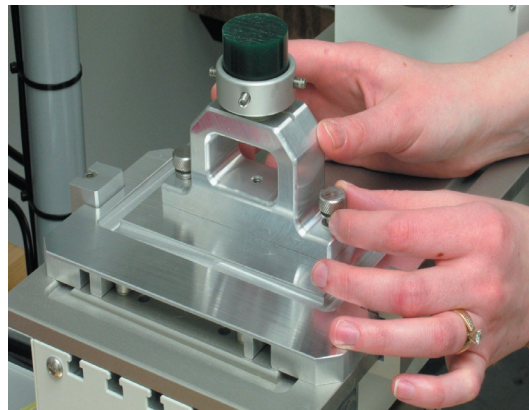
Affix the lock-down clamps that anchor the base plate to the table and tighten them with the 5/32nds Allen wrench.



Select the proper size cup for your wax blank and use an Allen wrench to tighten all four screws, securing the wax into the cup. Screw the cup into the upright fixture and tighten using thumb screw.



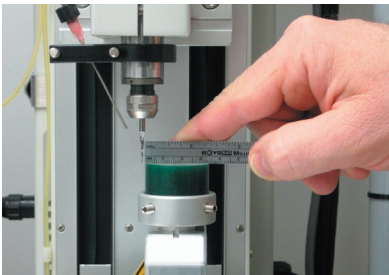
Then, align the pin in the upright fixture to the notch in the base plate and tighten the thumb screws to affix the upright fixture to the base plate.



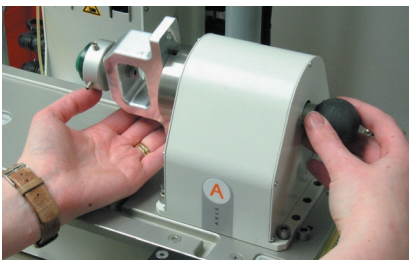
With the correct cutter installed, the hardware assembled, and the first part of the job loaded in the **Revo** software, the **Show Z Offset** and **Set Z Offset** buttons appear in the **Jog Control** portion of the interface. Jog the mill using the **Jog Control** interface until the tip of the cutter is located just beside the wax round. The tip should also be located *just below* the top of the wax. Click on **Set Z Offset** and the mill will remember this location and automatically return to the “o” position on the Z axis, ready to start this job. You’re now ready to click on **Start** in the Revo Control software.

Note: For the next job you run, the **Show Z Offset** button will return the cutter to the location set during the **Set Z Offset** command. Therefore, for each subsequent job, you can first use **Show Z Offset** to return the mill almost to the necessary position; you’ll only need to **Jog** it the rest of the way into position before using **Set Z Offset**.

Location of cutter for **Set Z Offset**:



After using **Set Z Offset**, the mill will record the position you set and return to the “o” position on the Z axis. You are now ready to **Start** the job. Once **Part I** has completed, *do NOT remove the wax cup from the upright fixture!!* Simply detach the thumb screws holding the upright fixture to the base plate and align the key in the upright fixture with the notch in the A axis. Insert the draw bar into the A axis and tighten to secure the fixture to the A axis.

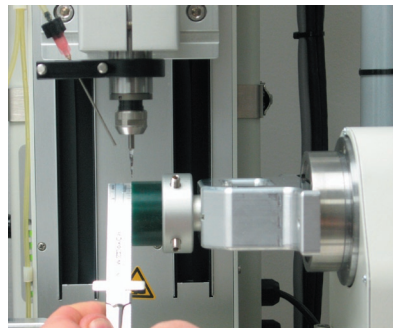


Now, you’re ready to use **Load Part II** to run the second part of the job. Click **Start** to finish this job.

Note: During the second part of the strategy, the **Show** and **Set X Offset** buttons appear in the jog control interface; however, there is no need to repeat the Set Offset command for the second part of the job. This is because the X offset is identical to the Z offset you’ve already set.

The reason that these controls are here is so that you can choose, if you wish, to run **Part II** of the job before **Part I**. To do so, first load the upright fixture with the wax blank onto the rotary A axis and tighten the draw bar. Then, **Load Part II** of this job and follow the directions above for setting the Z offset, jogging the cutter to the same location in the X direction this time, as indicated in the illustration below. With the cutter in this position, click **Set X Offset** and the mill will return to the o position on the Z axis. Then, **Start** Part II of this job. When running Part I after Part II, there is no need to run Set Z Offset in the Revo software.

Use **Set X Offset** ONLY when Running **Part II** of the job first:



Chapter 8: Preparing & Milling the Two-Sided Auto-Flip Strategy

About the Strategy:

Remember that this two-sided strategy works best with a model that lays flat (or relatively flat) along the X/Y plane in **Rhino** and has detail on its front and back and none on its sides. The mill cuts the model by holding the wax flat with the two-sided auto-flip fixture so that it is perpendicular to the cutter. The cutter begins at the center of the part and works its way outward in a concentric square or rectangular path.

Then, after completing the top of the part, the model is rotated 180-degrees by the A axis fixture and the back is cut. It is therefore fine to have details on the front and/or back of the model when using this strategy. It is best to select the auto-flip strategy when the model does not have any important details on the sides, since these cannot be easily reached by the mill cutter.

Preparing the Model in Rhino:

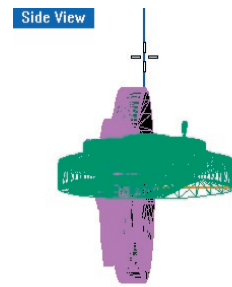
To begin, make sure that the model is mostly flat along the X, Y plane (the grid that is visible in the **Looking Down (Top)** viewport). If it is not, rotate it so that it is. To do so, select the entire model and click on **Rotate**. Working in the **Side View (Right)** viewport, indicate 0, 0, 0 on the grid as the center of rotation. Then, holding down Shift to engage **Ortho**, extend a straight line to one side and click. Still holding down Shift, swing the line up (or down) so that the model is flat along the X, Y plane (the top of the model is visible in the **Looking Down (Top)** viewport).

Incorrect: Through Finger (Front) view:

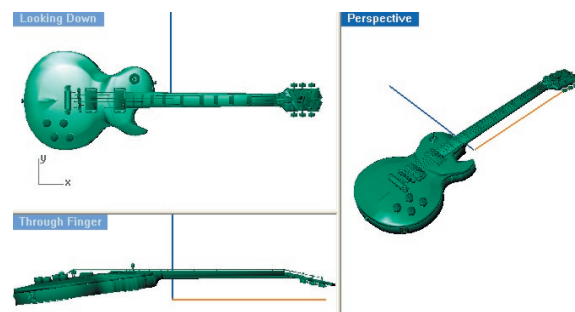
Through Finger



Rotate from 0,0,0 in Side View (Right) viewport:



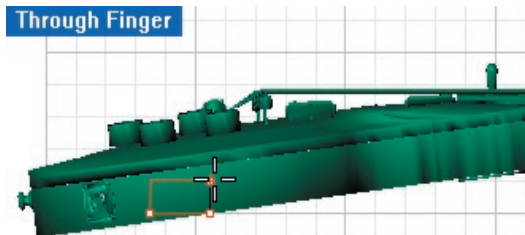
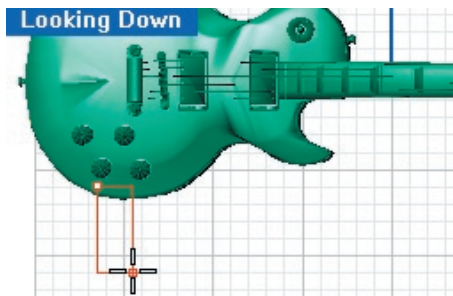
Correct: the flat, detailed part of the model is (roughly) parallel with the X, Y plane:



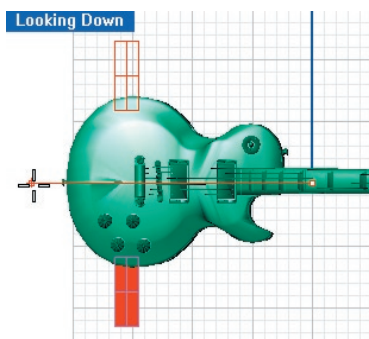
Before this model can be input into the software, support tabs must be attached to it that will hold it to the wax blank during milling. That way, it will not fall away from the surrounding wax while it is being cut. Draw support tabs using the **Box** command found in Rhino.

Make certain to place the support tabs at flat, detail-free places on the model to ensure the least intrusion into the design and the cleanest cutaway possible. To properly support the model, draw boxes that fully sink into the surface of the design and are roughly 1 mm tall X 2 mm wide.

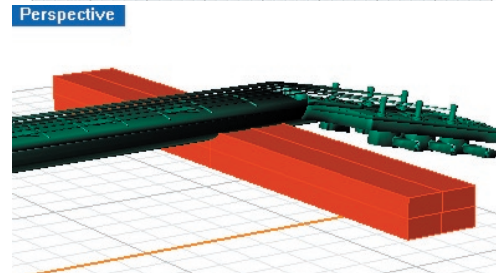
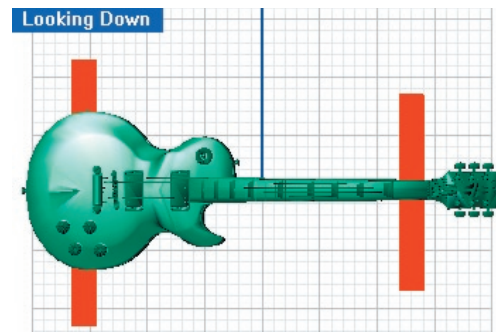
Draw them sufficiently long between the part and the surrounding wax so that there is enough clearance to cut the piece away from the wax after production. In this example, the supports were mirrored around 0,0,0, placing one on each side of the piece.



Use Mirror, with 0,0,0 as the “Center of Mirror Plane”:

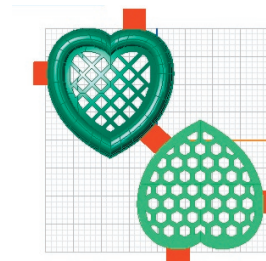


In the case of very thin details, such as the neck of this guitar, it might be necessary to create a full-length box to support the piece. In this example, too much strain would be placed on the thin neck of the guitar during milling, causing it to break away from typical supports. A box that only partly intersects the neck was drawn beneath it, extending from one side of the remaining wax to the other, in order to properly support the slender model.



Create supports that best accommodate the needs of the model. In some cases, two models may be milled on one piece of wax, as in the following example of the front and back of the same pendant. In this case, in addition to typical supports that anchor the pieces to the wax, supports were placed between the pieces, holding them together and ensuring their stability during milling:

Looking Down (Top) viewport:



With the careful placement of parts and supports, more than one piece can be milled at a time during the **Two-Sided Auto-Flip** strategy, as long as, together, the parts and supports all fit within the 50 mm square opening of the plates. Although it is possible to work without plates, it will be necessary to take a few precautions when doing so. These precautions are described in full below.

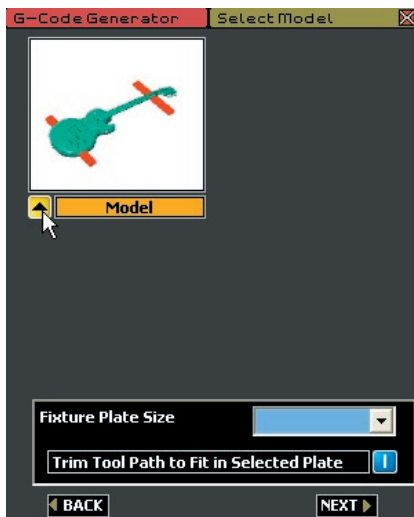
Using the RevoCam Software:

With Rhino open, click on the **RevoCam** software icon found on your desktop following installation.

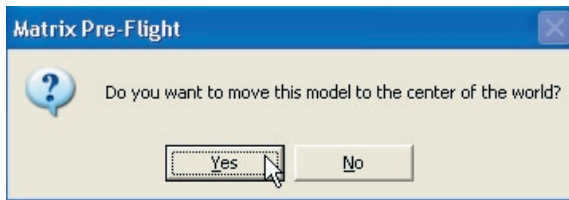
From the first screen, select the **Auto Flip Flat Milling (2 Sides)** strategy.



Click on **Next** to proceed. In the next screen select the model, supports and all, and input it into the **Model** preview window by clicking on the yellow “IN” arrow beneath the **Model** preview window.



The dialog box with the question “Do you want to move this model to the center of the world” will appear. The “center of the world” is 0, 0, 0 on the X, Y, Z grid in Rhino. Always click on **Yes** at this time. This operation ensures that the tool is cutting to the center of rotation of the A axis., thereby minimizing the required thickness of the wax blank used for milling the part.



Next, select the **Fixture Plate Size**, or, the size of the plates that will be used during this cut. The two sets of plates intended for use with the **Auto-Flip Flat Milling (2-Sided)** strategy are the 30 mm and 50 mm closed plates. Use the drop-down menu to select the plate that best fits the model (50 mm in most cases, except for the smallest of models)

and click once on the selected plate to place it in the menu box. Select “None” for use with the largest models (as long as they fit within a 75 mm X 75 mm area).

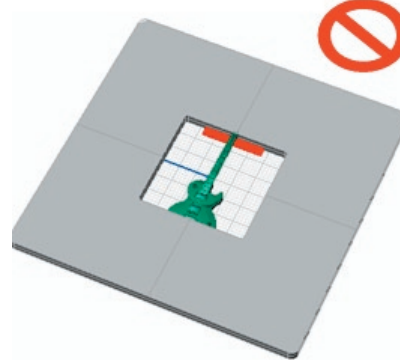


The selected plate will now be visible in the viewports, helping the user verify that the selection was correct. The plate should not overlap any part of the model: if it does, choose the next size up.

Correct:



Incorrect:



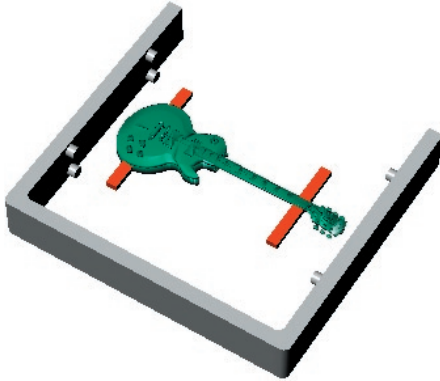
Finally, make sure that the option **Trim Tool Path to Fit in Selected Plate** is turned on. This selection helps ensure that the cutter will not crash into the metal plate, which would cause the cutter to break, by automatically trimming the toolpath to fit within the opening in the selected plate. With this option selected, the toolpath will be safely trimmed to 1.5 mm shy of the opening of the selected plates regardless of the size of the custom borders used. The default setting for this option is ON. (It is possible, though it is not recommended, to deselect this option.)

Trim Tool Path to Fit in Selected Plate

*Note: the **Trim Toolpath to fit in Selected Plate** option should ALWAYS be turned on in order to ensure that the tool and/or spindle will not crash into the fixture or the plate.*

Using no plates: If the model is too large to fit in even the 50 mm plates, select **None**. The two-sided auto flip fixture will appear. The model should not “pass” it on any side. If it does, or if the toolpath will cause the cutter and/or spindle to crash into the fixture or the pins, the part will need to be milled in two pieces, or more, to accommodate the large size.

No fixture plates:



There are a few more accommodations that need to be made prior to milling without plates. It is very important to check both the toolpath and the cutter length to ensure that the cutter and/or spindle will not crash into the fixture when milling any auto-flip strategy without plates:

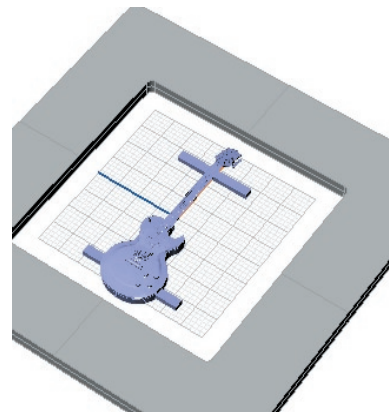
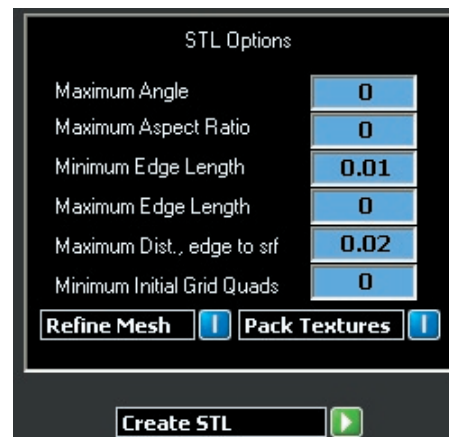
(1) Check the toolpath; then, double-check it! After creating the toolpath in the **RevoCam**, MAKE CERTAIN that it does not crash into the fixture or the pins!! This must be done visually, since the fixture plate is not accounted for during toolpath creation.

(2) Check the cutter length; then, double-check it! After installing the two-sided auto-flip fixture, use the **Jog Controls** section of the **Revo** interface to rotate the **A** axis until the fixture is flat. Then, jog the cutter down next to (not into!) the wax, roughly 1 mm past center, and visually check the depth of the cutter to make certain that the spindle cap will not hit the pins that hold the wax in place. If the cutter is too short, the spindle cap will likely hit the fixture. If it does appear that it will hit the fixture,

re-install the cutter, holding it just 0.25 inches from the end. Insert it into the collet and tighten. Then, check the depth again and perform the **Seek Tool** operation before cutting the part.

Create STL: After all of the settings are correct, click on **Next** to proceed to the **Create STL** screen in the software. In this screen, it is recommended that the user keep all of the default settings. Changing these defaults adjusts the density and composition of the mesh “skin” that is placed over the model in order to generate the tool path, and can greatly increase the file size, which affects the performance of both the **Rhino** and the **Revo** software.

To proceed in this screen, simply click on the green **GO** arrow beside **Create STL**. A fine, steel gray mesh will be wrapped around the whole model, creating a tight seal over every part. Because the software uses this mesh to create the toolpath, all of its pieces are recognized as one by the mill, even if the model is not booleaned together. The toolpath is based upon the mesh, not off of the separate pieces of the model, saving users’ time and sanity by eliminating frustrating Boolean union operations!

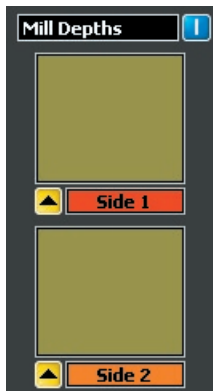


Click on **Next** to proceed in the software.

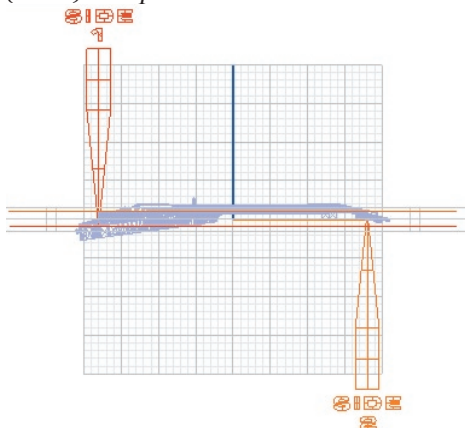
Borders and Mill Depths: In this screen, the user will input **Mill Depths** and **Custom Borders** using the tools found on-screen. **Mill Depths** define the maximum depth to which the tool will cut on both sides of the part, while **Custom Borders** define the area that the tool will cut around the model. The remaining two controls in this screen allow the user to toggle between the **STL** and **Nurbs** views of this model. If need be, select from among these two options to properly see all of the details on the model. This helps the user make certain that the **Mill Depths** and **Custom Borders** are properly set up.

Setting Mill Depths: To begin, select the **ON** button beside **Mill Depths** to define the maximum depth to which the cutter will plunge when cutting the part. This activates the two planes that are used to define the mill depths. In the software screen, the **Side 1** and **Side 2** preview windows will appear. In the viewports, the **Side 1** and **Side 2** mill depth planes will appear.

Software screen: Preview windows appear



Viewports: Mill depths appear in Through Finger (Front) viewport:



*Note: When observing the positioning of the mill depths, remember that Side 1 is the “top” of the model and Side 2 is the bottom. Between the operations that cut Side 1 and Side 2, the piece is rotated on the **Rotary A Axis** so that the bottom is now in the top position and vice-versa. So, even though the mill depth for Side 2 appears to come “up” from the bottom of the model, the cutter will actually come down from the top once the model is rotated.*

The mill depth is a “grouped” item that consists of a cutter, a plane, and a label. (Because the items are grouped, clicking on any of the items in the group will cause the entire “mill depth” to be selected.)

The location of the plane portion of the Mill Depth defines the maximum depth to which the cutter will plunge when cutting the part.

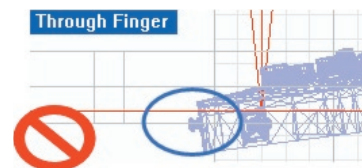
Make certain that this plane reaches through all of the details on that side of the part that the user wishes to cut (**Side 1** or **Side 2**). After setting the mill depth, select it and input it into the corresponding window (**Side 1**, below).

For example, in this model, the mill depth for **Side 1** must be dragged down so that the peg at the base of the guitar will be properly cut.

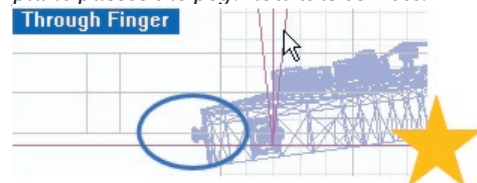
If the default mill depth for **Side 1** were used, the peg would not be reached by the cutter. To fix this, select the mill depth and drag it straight down until the plane is positioned past the peg:

Note: ALWAYS adjust the Mill Depths in the Through Finger (Front) viewport!

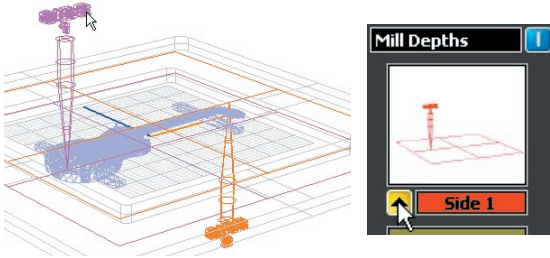
Default mill depth, Side 1: Incorrect (peg at end of guitar will not be cut)



Select the mill depth and drag it down until the plane passes the peg: now it is correct!

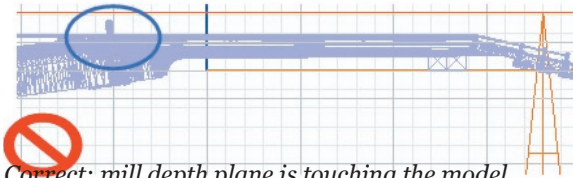


Once the cutter is correctly positioned, select it and input it into the corresponding **Preview Window** found in the software screen (click on the yellow **IN** button beneath the preview window to do so). Remember to insert the “Side 1” mill depth into the “Side 1” preview window and the “Side 2” mill depth into the “Side 2” preview window. The software will not allow the wrong plane to be inserted into the wrong preview window.

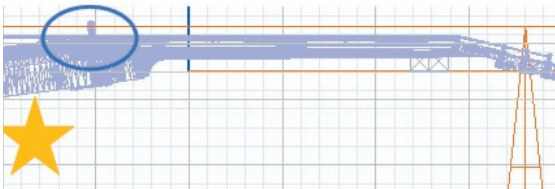


When setting mill depths, it is not possible to set the planes past the extents of the part. They must be touching the model in order to successfully generate the toolpath. For best results, set each mill depth to the furthest extents of the model without going past it. This will ensure that the maximum detail possible will be cut from each side of the model.

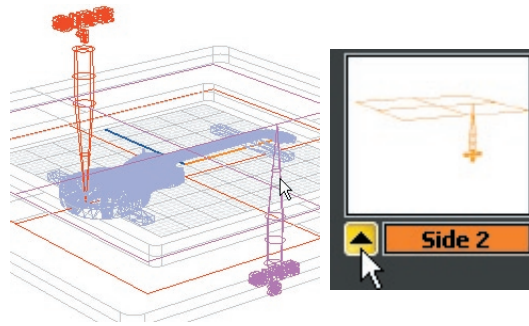
Positioning Side 2 Mill Depth: Incorrect: mill depth plane is placed past the furthest extent of the model.



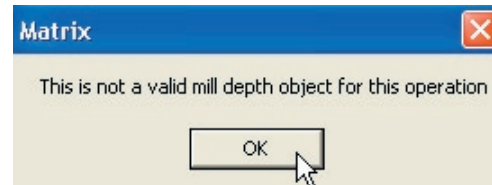
Correct: mill depth plane is touching the model (does not matter how far past center: the further past center, the more details are sure to be cut on this side of the part).



Select the **Mill Depth** for **Side 2** and place it into the appropriate preview window in the software, as well.



This screen of the software will not allow the user to mistakenly place anything but the correct mill depth into its corresponding **Mill Depth** preview window, so have no fear! If the other mill depth or any other object is selected and the user attempts to input it into this preview window, the following error message will appear:



If this error message appears, click on **OK** and deselect everything in the viewports by clicking once on empty space. Then, select just the mill depth that corresponds to the label on the preview window and click on the yellow **IN** arrow beneath that preview window again.

Setting Custom Borders: Once the mill depths have been input into their corresponding preview windows, activate the **Custom Borders** control to see the tools used to define a custom border for the part. This will cause the **Border** preview window and the **Rectangle** curve command to appear in the software screen.

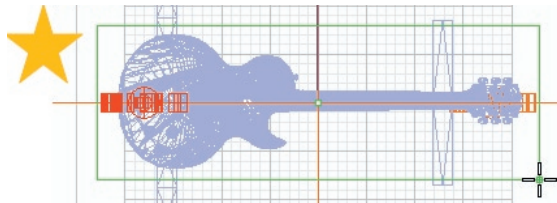


For this strategy, the custom border will be either a rectangle or a square that is drawn in the **Looking Down (Top)** viewport. To begin, select the rectangle tool, starting up this **Curve** command. Then, working in **ONLY** the **Looking Down (Top)** viewport, draw a border that surrounds the part

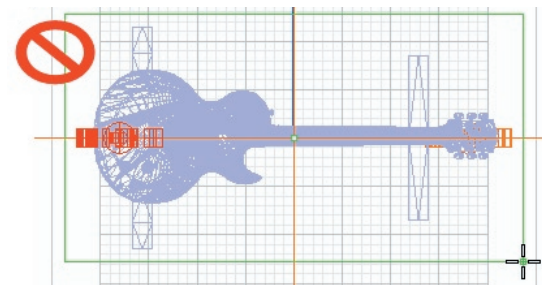
but falls within the extents of the support tabs. This rectangle is essentially defining the area that will be used to generate the toolpath region.

The cutter will remain inside this boarder or region, so be certain that the support tabs fall outside the extents of the border, or else the tabs will be cut away from the surrounding wax and the part will “fall through”! Also make sure that the border is at least 1.5 mm away from the extents of the part so that there is enough space to comfortably cut the part away from the wax.

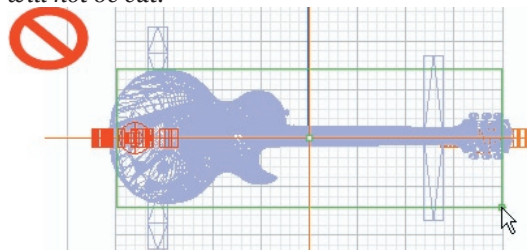
Use the Rectangle tool to draw a custom border surrounds the part. DO NOT extend the rectangle beyond the ends of the support tabs and DO NOT overlap, touch, or get too close to any part of the model itself!



Incorrect: rectangle is too big because it does not overlap the support tabs; therefore, the model will fall through!



Incorrect: the custom border overlaps the piece; therefore, those parts that fall outside the rectangle will not be cut.



IMPORTANT: Drawing the custom border in any viewport EXCEPT the Looking Down (Top) viewport will generate an incorrect toolpath!

Once the correct border has been drawn (as in the first example, above), select the border and input it into the **Border** preview window by clicking on the yellow input arrow beneath the window.

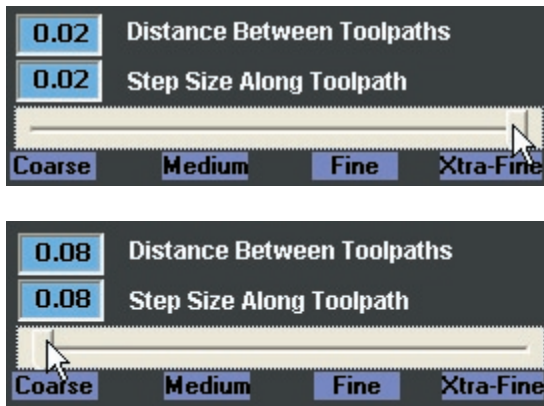


Congratulations! The hard part is done!! Click on **Next** to proceed in the software.

Selecting a Cutter: After the user sets the mill depths and custom border, the software offers a **Cutter** selection screen. When milling a flat surface, or when straight sides are needed for details, select the **Revo Flat Mill 0.032** cutter. When selecting this cutter, details MUST be a minimum of 1 mm apart! It is fine to use this cutter when cutting from rough, as well. For jobs that don't fall within these parameters, select the **Revo 10 Degree** cutter. This is the default setting in this screen. To change the cutter, click on the drop-down menu arrow and select another cutter from the list by double-clicking on it. The selected cutter will appear in the menu box.



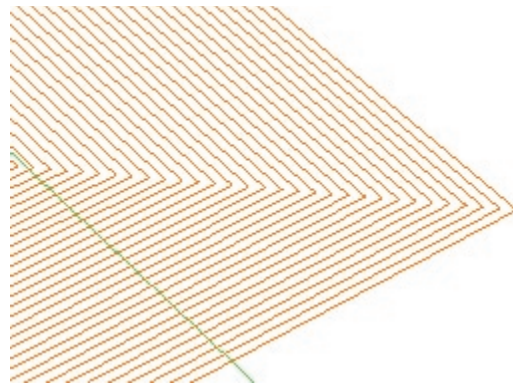
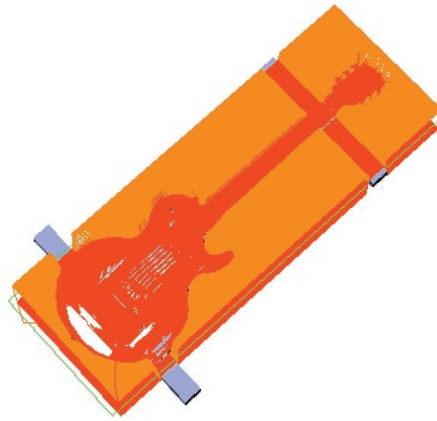
Defining the Quality of the Cut: There are two ways to use the controls that define the quality of the cut. The first is to click and drag the slider control, moving it along the scale from **Coarse** to **Xtra-Fine** until the desired setting is achieved.



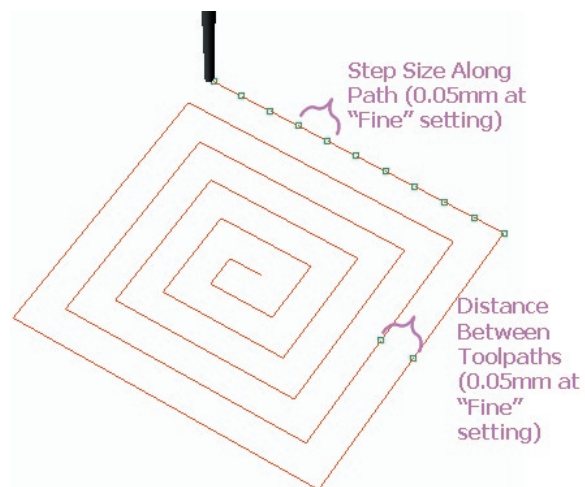
The second way to change these settings is to manually type a new value into the **Distance Between Toolpaths** and **Step Size Along Toolpath** text boxes. Lowering these values increases the quality of the mill job; however, it also increases the time that the mill job will take. For example, raising this value to **Xtra-Fine** will more than double the time it takes to run the job. It is not recommended to raise or lower these values beyond those available on the scale.

The blue text fields are provided so that the user can change one value but not the other. For example, leaving the slider setting at **Fine** and changing the **Distance Between Toolpaths** to 0.3 will increase the finish quality to basically the same as the “extra fine” setting without doubling the mill time: it only adds about 20 minutes to an average mill job.

To understand what each value means, it is important to understand what the toolpath is. The following is an illustration of the toolpath for the model used in this example. Notice that the toolpath for this piece starts from the center of the part and works outward in a concentric pattern toward the extents of the piece. Zooming in close to one corner of this toolpath shows the corners of the concentric rectangles.



Another representation of the toolpath, though not created to scale, is found below. Remember that the start of the toolpath is at the exact geometric center of the given border. The space that is left between the concentric rectangles is equal to the value listed beside **Distance Between Toolpaths**.

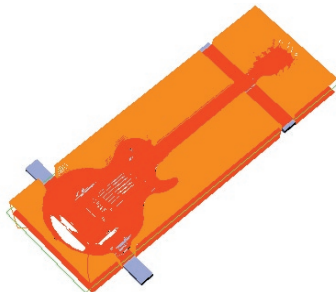
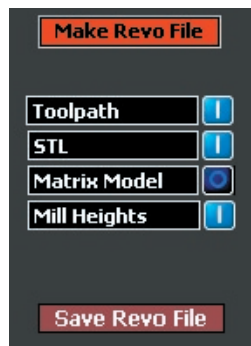


Step Size Along Path: This value refers to the distance between the reference points that the cutter uses to orient itself to the part. Because it

cannot calculate every possible point along the line, since the mathematics would be staggering, the mill computer instead plots points along the toolpath and the cutter advances to these points.

The further away these points are, the less detail will be achieved in the finished part. The closer together these points are, the more detail will be achieved; however, the mill job will also take longer to run. The **Step Size Along Path** defines how close these points are along the toolpath. As illustrated above, for a “Fine” milling job they are 0.05 mm apart, which is pretty close. So, in a way, when the mill follows a toolpath, it is really just playing an elaborate game of “connect the dots”! After selecting a cutter and setting the values for these two parameters, proceed to the **Next** screen in the software, **View and Save RVO File**.

View and Save RVO File: In the final screen of this software, begin by clicking on the red **Make RVO File** button. Wait a few minutes, and the toolpath will be created around the model. Now, use the remaining controls to view the toolpath, STL mesh, Nurbs Model, and/or the Mill Heights as desired to check the toolpath that was generated by the software. Using these controls, inspect the model to ensure that the toolpath is correct, and then click on **Save Revo File**.

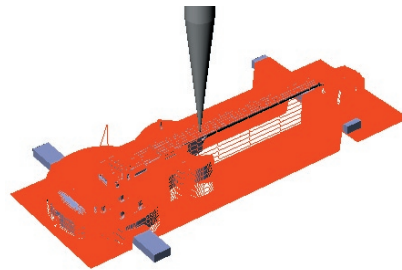


Understanding the Two-Sided Auto Flip Toolpath:

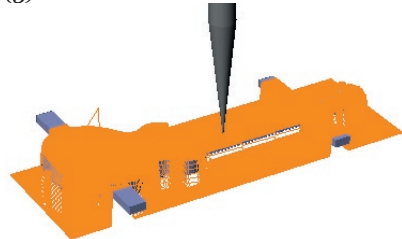
Recall that this toolpath starts at the geometric center of the border for the first side of the model and works outward in concentric rectangles actually toward the outer edges of the custom border that was set by the user. The red toolpath represents the **Side 1** cut and the orange toolpath represents the **Side 2** cut. Because the mill automatically rotates the fixture to start side two, there is no user intervention needed after the job has been started.

(1) Mill cuts Side 1; (2) Mill rotates AutoFlip fixture to reach Side 2; (3) Mill cuts Side 2: The cutter starts at the center of each side and works outward. Where there is empty space (no model or support tabs), the cutter plunges down to the depth that was indicated by the user when setting Mill Depths.

(1) Cut Side 1 (2) Mill rotates Autoflip Fixture



(3) Cut Side 2



Adjust the controls for viewing the **Toolpath**, **STL**, **Nurbs Model**, and **Mill Heights** to see only what is necessary to check the accuracy of the toolpath. To view the original Nurbs model and compare the accuracy of the toolpath against that model, turn on the **Nurbs Model** setting, which turns off the **STL** setting. To see only the model, not the toolpath, turn OFF the **Toolpath** button. To keep the **Mill Heights** from blocking the shaded view of the model, turn them off.

The default setting is shown below; however, these controls do not affect the RVO file that has been generated during this step: they only allow the user to better see the part and the toolpath.

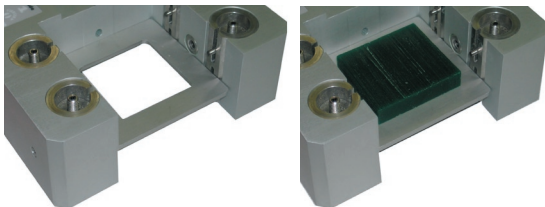


Once the toolpath has been deemed satisfactory, click on the **Save Revo File** button and select a location on the computer to which the file will be saved. When the progress bar reaches 100%, the file has been saved. It is now ready to be transferred to the **Revo** software found on the mill computer.

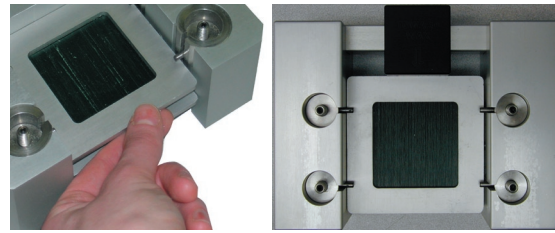


Preparing the Mill Fixture for a Two-Sided Auto Flip Strategy:

With the “**Side 1**” side of the two-sided auto-flip fixture facing up, place the gauge block over the spine of the fixture with the “Toward Wax” arrow facing the opening in the fixture. Place the first plate between the pins, setting it flush against the spacer. Then, center the wax block over the opening in the fixture plate (50 mm closed plates shown, below).



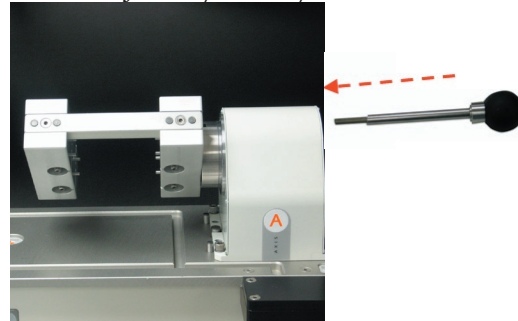
Place the second fixture plate over the wax but under the upper set of pins, pushing it back all the way so that it is flush with the spacer and lines up with the lower fixture plate:



Tighten the threaded screws with the Allen wrench so that the pins move towards one another and cause the plates to hold the wax securely. Remember that it is best to tighten in a “star” pattern, alternating screws diagonally across from one another until the plates are snug. A slight bowing may be noticeable in the plates when they are tight.

Finally, remove the spacer before attaching the two-sided auto-flip fixture onto the A axis fixture plate. Then, align the pin in the fixture with the slot on the face plate. Insert the draw bar through the A axis and tighten to secure the fixture to the axis. With the mill in the **Home** position, the fixture plate will be on a steep angle to the A axis. Fear not: this is correct!

Two-sided fixture / A axis / Draw Bar:

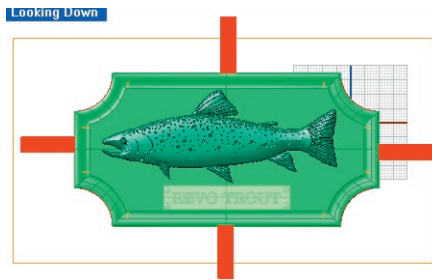


Using the Extended Auto-Flip Fixture & Tail Stock:

Preparing the Model in Rhino:

To run the two-sided auto flip strategy with the extended fixture using the traditional strategy (no advanced options), you should have a single large piece to mill. Attach supports as usual. Then, click on the **Mill Area** icon found in the **Software** flyout menu:

It is very important that the part AND ITS SUPPORTS fit FULLY INSIDE the curve that is placed by this software in the **Looking Down (Top)** viewport.



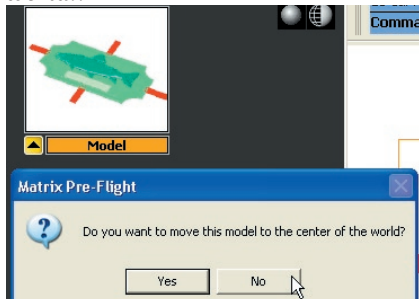
NOTE: This rectangle is NOT your custom milling border for this operation! It simply denotes the milling area for this part. Make certain there are NO parts of the model that fall outside of this border, or else the spindle could crash into the fixture!

Now, enter the **RevoCam** software and select the Auto Flip Flat Milling (2 Sides) strategy.



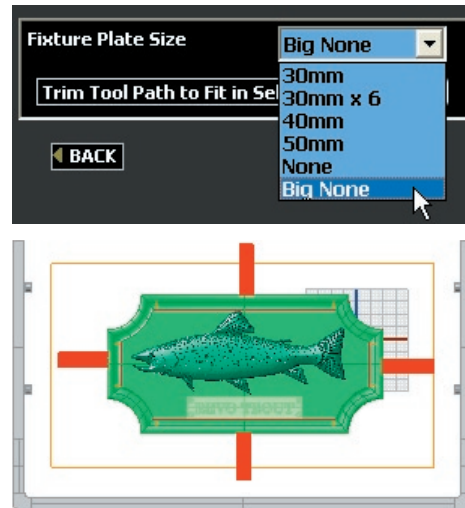
Complete the strategy much like you do the traditional 2-sided auto-flip strategy; however, in the first software screen, it is very important that you DO NOT elect to move the model to the “Center of the World”, as recommended by the pop-up window. This is because the Mill Area Border is correctly placed to accommodate the positioning of the larger fixture on the mill, and it is NOT centered. Moving it would cause the spindle to collide with the A axis!!

No, do NOT move this model to the center of the world!!



Also in this screen of the software, be sure to select the “Big None” fixture plate from the Fixture Plate

dropdown menu. This plate will appear onscreen, as shown in the illustration below:



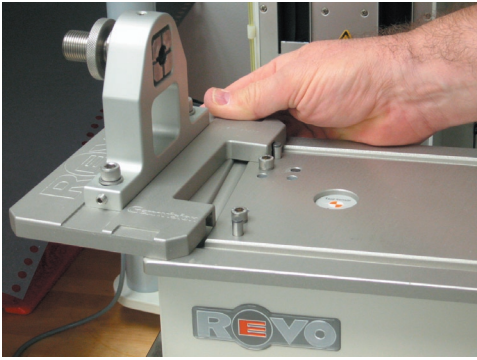
Now, complete the remainder of the **RevoCam** software as usual for the regular **Two-Sided Auto Flip** strategy. (Advanced Options are NOT needed unless you wish to put in multiple borders.) After outputting the .RVO file, proceed to the mill and assemble the proper fixture for use with the selected strategy.

Assembling the Tail Stock & Extended Auto-Flip Fixture:

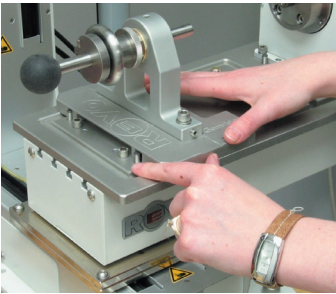
Note: To calibrate all new fixtures prior to use, go to the calibration interface of the Revo Control Software and follow the procedures outlined in the new steps.

To attach the **Tail Stock** to the mill’s table, slide the fixture into position from the left-hand side of the table over the 1/4” dowel pins located in the fixture plate and clamp into place for use with the **Extended Two-Sided** fixture. (Or, slide it to the right over the same dowel pins to position it for use with the traditional-sized Adjustable Two-Sided fixture.)

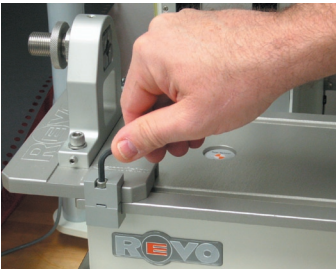
Position for use with Extended Two-Sided Fixture:



Position for traditional-sized Adjustable Two-Sided Fixture:

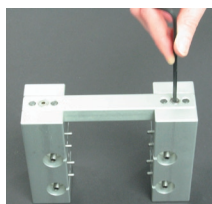
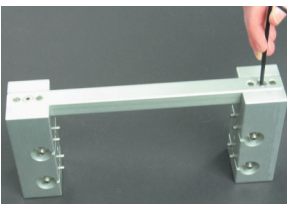


In either position selected above, affix clamps to secure fixture to plate and tighten with Allen wrench.



Select which spine length – **Extended** or **Traditional** – will be required for use with the adjustable **Two-Sided Auto Flip** fixture and screw it securely into the back of both arms of the fixture:

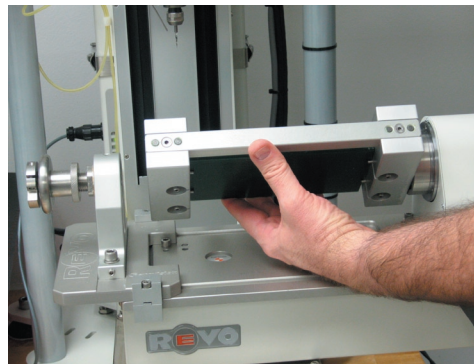
*Assemble appropriate spine for **Extended** fixture (on left) or traditional-sized fixture (on right):*



Insert plates (if needed for traditional length - no plates required when using extended fixture) and wax between pins in fixture and tighten with Allen wrench. When using the traditional-length spine, use the black spacer to properly position the wax at the edge of the fixture. Then, secure fixture to the A axis, using the draw bar to tighten.



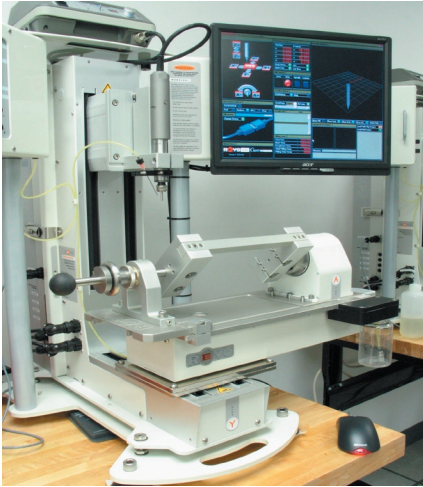
IMPORTANT: When using traditional-length spine, remove black spacer before running mill job.



Next, insert the rod support through the **Tail Stock** and into the pilot hole located in the back half of the Auto-flip fixture. Then, tighten the locking ring to secure fixture and support rod to the tail stock.

Insert rod support & Tighten locking ring:

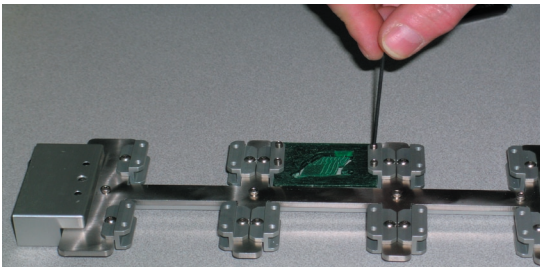


Extended size:

Set-up in Revo Control software for extended auto-flip fixture is identical to that for traditional two-sided auto flip strategy.

Assembling the Tail Stock & Multiflip Fixtures:

You will receive clamps for this fixture to accommodate 5, 10, and 15-mm wide waxes. Find the size you need for the models you are running and install them in the fixture, as shown below. Then, use an Allen wrench to close the clamps around as many wax blanks as you'll be running in this batch.



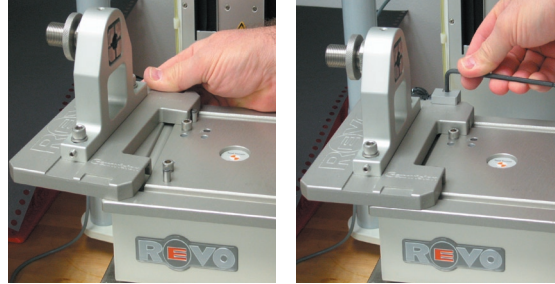
You'll need the tail stock attached to the mill's table to affix the Multiflip to the mill.

Note: *If the Tail Stock is being newly attached, as explained below, you'll need to run the Tail Stock calibration, found in the Calibrations tab in the Revo Control Software; then affix the Multiflip Fixture and calibrate it. If the tail stock is already attached, you'll only need to assemble the Multiflip Fixture and run its calibration step.*

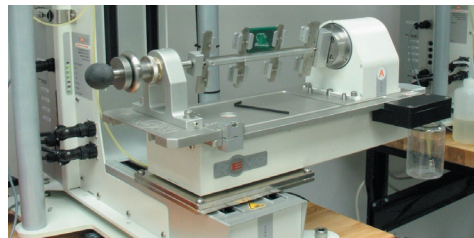
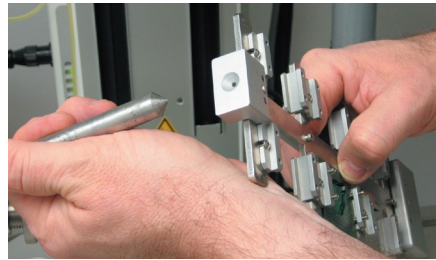
To attach the **Tail Stock** to the mill's table, slide

the fixture into position from the left-hand side of the table over the 1/4" dowel pins located in the fixture plate and clamp into place for use with the **Multiflip** fixture. Then, affix clamps to secure fixture to plate and tighten with Allen wrench.

Position for use with **Multiflip** fixture:



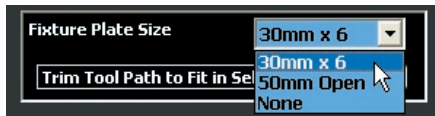
Slide the **Multiflip Fixture** into place and secure to both tail stock and rotary fixture. Be sure and use the tail stock rod with the pointed end that fits into the end of the Multiflip Fixture.



Set-up a Multiflip Project in Revo Software:

To run a Multiflip Project, you may run up to six models in any combination of the following strategies: two-sided auto flip, three-sided auto-flip, or the Auto-flip ONLY portion of two- or three-sided auto-flip plus rotary jobs. In addition, each model must fit into the 30-mm plates available for use with this fixture.

With these constraints in mind, set up a project you wish to run on this fixture the usual way in the RevoCam software. Be certain to select the "30mm X 6" drop-down option when selecting a fixture plate in the first screen of the selected strategy.



Complete the screens and save the .RVO file as usual for the selected strategy. Repeat for all 6 projects, using the SAME CUTTER so that the machine will not need to stop and wait for you to change cutters on a particular job. Copy the files to a memory stick, a networked drive, or another method you usually use in order to open them on the mill computer.

Now, enter the Revo Software on the mill computer. In the lower right-hand corner of the screen you'll see the "Load Multiflip Project" icon. Click this to enter the interface through which you will load the .RVO files you have just created above.



This will open the Revo MultiFlip Fixture interface. This interface displays a number of slots, each one representing the coordinating space on the multiflip fixture.



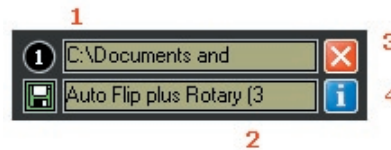
To load a Revo project into a "slot", click on the green "disk" icon located just above the preview window in that slot (or below, for the lower slots). Navigate to the location where you've saved this file, and Open it up to see the preview appear in the olive green box. Repeat for each project you wish to load.



Note: If you load these projects off of a memory stick, the memory stick MUST remain connected to the mill computer until every model has been cut! This is because the preview image you see in the box acts like a "shortcut" to the file on your memory stick: that file has not been permanently loaded into the software until the mill is ready to cut that model as part of the Multiflip job. So, keep this in mind as you are working, and do NOT remove the memory stick if you are acquiring models from it.

Or, as an alternative, simply copy the models from the memory stick into a folder on the mill's computer and Load them from here. With the RVO files saved elsewhere (on your memory stick or on another computer), delete them from the mill's computer when you are done running them, so as not to tax the available resources of the machine during future jobs.

With a .RVO file loaded into this interface, you'll see pertinent information just above the preview window that you can use to help you run this file. (For the operations on the lower side of the fixture, this information appears below the preview window).

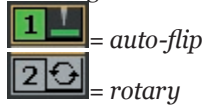


- (1) file location on the mill computer;
- (2) the strategy you selected in RevoCam with which to mill this file;
- (3) red "X" - click once to clear the preview window and cancel this job altogether;
- (4) blue "i" - click once to learn important information about this file, such as date and time of Revo file creation, fixture plate size, ring size, core size, toolpath width, etc.

Also, to the right of the preview window, you'll see several icons that you'll need to use to help you run each file. These icons reveal each "Load Part" of the given job. As when running a job the usual way in the Revo software, you'll have a different number of "Load Parts" for each strategy: 1 for two-sided auto flip, 2 for 3-sided auto-flip (one for cut-out curve, one for three sides), 2 for auto-flip plus rotary, (one for auto-flip, one for rotary) and 3 for 3-sided auto-

flip plus rotary (one for cut-out curve, one for three sides on flat, one for rotary).

The type of cut is apparent from the look of the icon found beside the preview window. For example, in an auto-flip plus rotary operation, you'll see the following two icons:



You'll notice that the rotary option is always grayed-out wherever it is found, since you cannot run a rotary operation on this fixture. So, following the flat milling operation (with center hub), you'll need to cut the models away from the wax supports (but NOT the center hub support!!) and run each rotary operation separately.

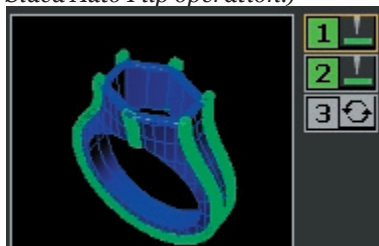
When you Load the model into the Revo software the traditional way, Load Part II ONLY so you can run JUST the rotary part of the model.

Running Three-Sided Auto Flip Jobs on the Multiflip Fixture:

It is important to note here that, when using the multiflip fixture, the mill is programmed to complete one model at a time: running both auto-flip operations in a two-sided strategy before proceeding to the next model, for example. With typical two-sided auto flip or two-sided auto-flip plus rotary jobs loaded into the software, it's fine to just hit **Start Multi-Mill Jobs**.

However, in the case of a three-sided auto flip model, keep in mind that there is an initial "Load Part" of the job (Part I) wherein the mill must follow the cut-out curve and wait for you to remove the cutaway piece of wax before it can proceed to Part 2.

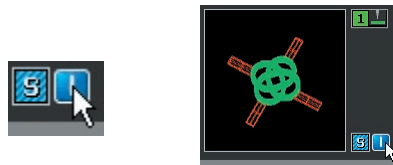
3-Sided Auto Flip Plus Rotary: Part 1 = Cut out curve; Part 2 = Two-sided auto flip; Part 3 = Rotary cut grayed out: it is NEVER run on this fixture! (This rotary part is non-existent on a 3-Sided Auto Flip operation.)



Unless you'd like to wait around all night for the mill to get to your three-sided job(s), stop, let you take out the wax, and start up again, you can limit which parts of which jobs it runs when using the multiflip fixture.

To do so, make certain that all jobs that do NOT include a cut-out curve are marked with the blue "Skip" indicator light (the aquamarine letter "S" to the right of each preview window).

Skip all but the three-sided auto flip jobs:



Then, click on "Start Multi-Mill Jobs" and wait a couple of minutes for the mill to cut the cut-out curve shapes for each of the 3-sided auto flip jobs.

When the mill stops, remove the cut-out shapes from each piece on the multiflip fixture by hand. Upon returning to the Multiflip interface, each completed job (the cut-out portions of the 3-sided auto flip jobs) will be indicated as complete with a checkmark beside them, and any of the jobs you had marked with a "Skip" will be returned to their original status.



Simply click "Start Multi Jobs" to complete the multiflip milling process. You're now ready to go home and have a good night's sleep, knowing your mill is working away, long into the night, without complaining or charging you time-and-a-half!

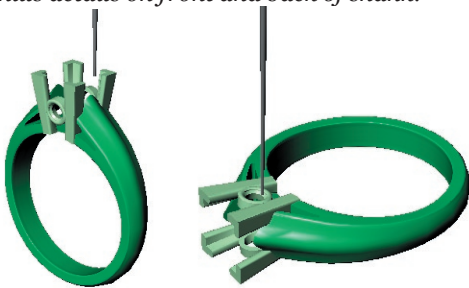


Chapter 9: Preparing & Milling the Auto-Flip Flat Milling (3-Sided) Strategy

About this Strategy:

The **Auto-Flip Flat Milling (3-Sided)** strategy starts by milling the top of the ring, completing head details and details at the top portion of the shank with greater precision than is possible with either the **Two-Sided Auto Flip** or **Rotary (1-Sided)** strategies alone. For rings with a combination of details in these areas and with a plain shank around the sides and bottom, this is the perfect strategy!

Mills details at head and top of shank:, and then mills details on front and back of shank:



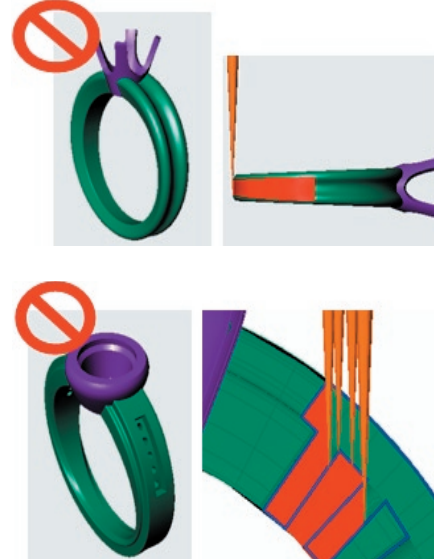
For best results, there should be little-to-no detail on the sides or bottom of the shank. A curved shank is acceptable; however, if the shank is grooved or split around the sides and/or bottom, the groove will be missed by the cutter.

Three-sided auto flip: shanks are straight and/or curve outward:



For best results, models like the next two rings should be run on the Auto-Flip Plus Rotary (3-

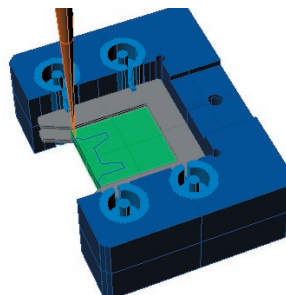
Sided) strategy: in the first one, the “groove” all the way around the shank will be missed by the Auto Flip Flat Milling (3-Sided) strategy; in the second, the sides of the channel will be missed by the top-down nature of the three-sided auto flip strategy. Both models require a rotary path to complete define these features. The errors are illustrated below:



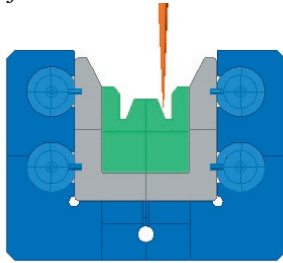
This strategy begins by running a quick two-sided auto flip job that mills a cut-away section out of the wax roughly following the shape of the head. After this is complete, the mill pauses in the process so that the user can remove the cut-away section from the wax. Next, the wax is rotated 90 degrees to mill the head and the top of the shank.

Because the cut-away section is removed, there is plenty of clearance for the cutter to mill the details at the head and top of the shank without breaking. After the head and top of shank is completed, the mill rotates the wax blank to the traditional Two-Sided Auto Flip positions at 0 and 180 degrees to finish milling the two sides (front and back) of the ring.

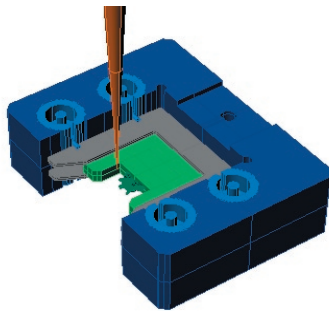
(1) The cutter follows a user-created cut-away curve to clear the way for milling details at head and top of shank.



(2) User removes cut-away piece. (3) Fixture is rotated 90-degrees to mill details at head and top of shank.



(4) A-axis rotates the two-sided auto flip fixture to the traditional two-sided position and cuts the front and back sides of the ring.

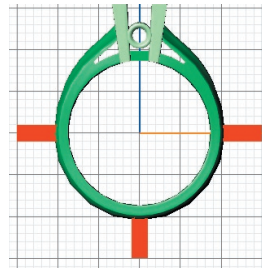


Preparing the Model in Rhino:

Drawing the Supports: Before this model can be input into the software, supports must be attached to it that will hold it to the wax during milling. That way, it will not fall away from the surrounding wax while it is being cut. Draw supports using the **Box** command found in Rhino.

Make certain to place the supports at flat, detail-free places on the model to ensure the least intrusion into the design and the cleanest cutaway possible. To properly support the model, draw boxes that fully sink into the sides of the shank and are roughly 1 mm wide X 2 mm tall. Create them with enough length so that there is sufficient space between the part and the surrounding wax to comfortably cut the piece away from the wax after production. In this example, one support was created and then rotated around 0,0,0, with “Copy = Yes” indicated in the **Command** line, to place three supports around the piece.

Note: When using the new Three-Sided Auto-Flip strategy, it is important NOT to place supports in the head area of the ring.

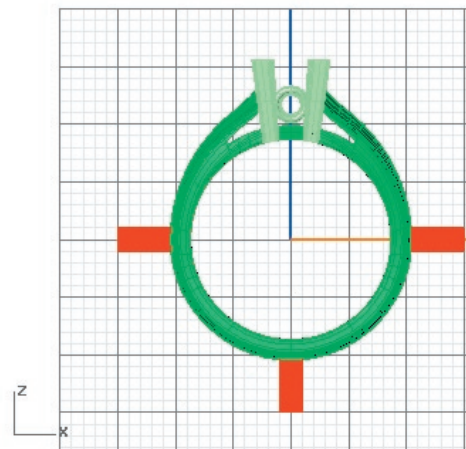


Drawing the Cut-Out Curve: Before entering the RevoCam software, draw a polyline curve in the **Through Finger (Front)** viewport that roughly follows the shape of the head and the top of the shank. This curve, known as the **Cut-Out Curve**, represents the wax cut-out that is milled during the first part of the strategy.

To begin, turn on the **1.0 mm Grid Snaps** and start up the **Polyline** curve tool. Start the curve 6 mm above the upper left-hand corner of the grid (move the mouse slowly, counting 6 “grid snaps” straight up above the upper left-hand corner of the grid). Do NOT start the curve to the left of the grid, or else the cutter could run into the fixture plate!

With 1.0 mm Grid Snaps ON:

Through Finger

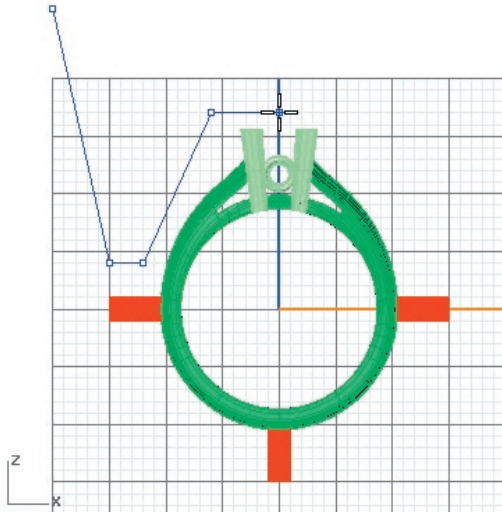


Extend the polyline curve down toward the supports and up the side of the ring, staying about 1.5 to 2 mm away from the support tabs and from the model. End the polyline when it touches the Y axis (vertical blue axis) in the **Through Finger (Front)** grid.

Draw a Polyline that starts 6 mm above upper left-

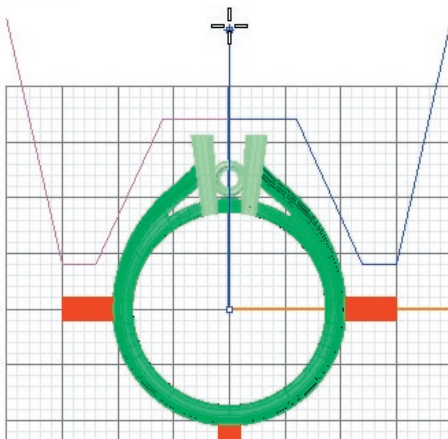
hand corner of grid and stays 1.5 – 2 mm away from the part and supports. End the line at the Y axis or the center of the grid.

Through Finger



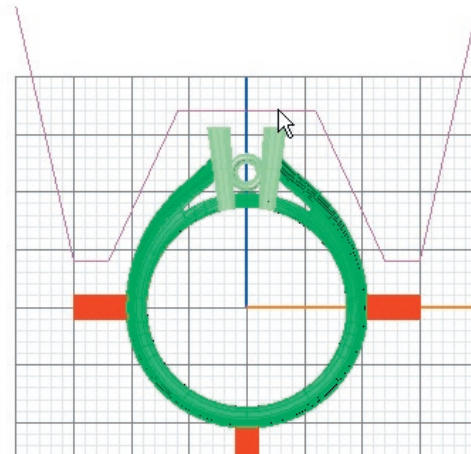
Still working in the **Through Finger (Front)** viewport, select the curve and click on **Mirror**. Press 0,0,0 on the keyboard to indicate the “Start of Mirror Plane” and, holding down Shift to engage **Ortho**, extend a line straight up to place the mirrored curve so that it touches its counterpart. Click once to end the **Mirror** command.

Mirror the line around 0,0,0, using Ortho so that both lines touch:



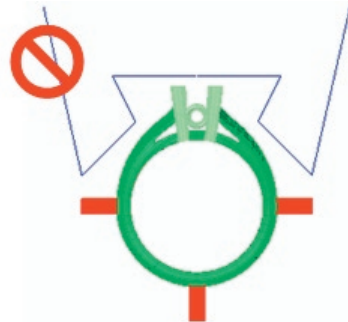
Select both curves and click on **Join**. This is the cutout curve that will be instrumental in preparing the part for this strategy within **RevoCam**.

Join both halves of cutaway curve:

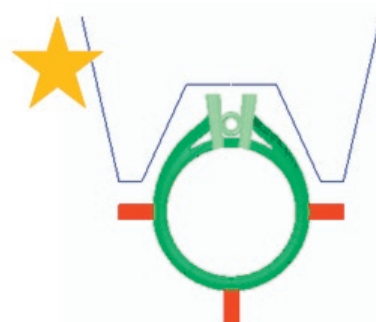


Warning: Do not inadvertently “lock” the cutout into the rest of the wax by creating an under-cut shape. Instead, be certain to taper the line upward, or draw it straight, so that the cutout will pull straight out of the flat wax after it is milled.

Incorrect cutout curve: (cutout piece will be “locked” into remaining wax)



Correct curve:

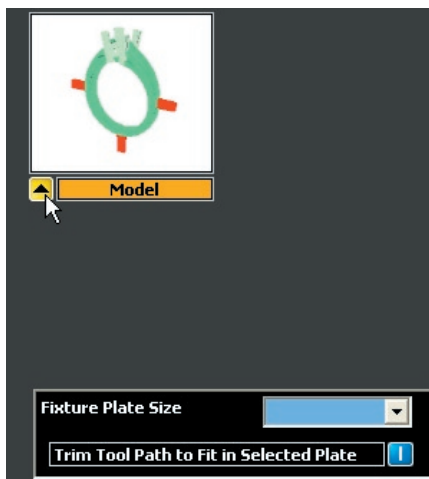
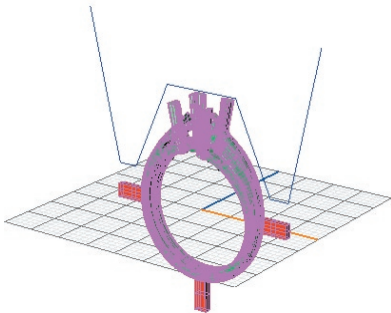


Using the RevoCam Software:

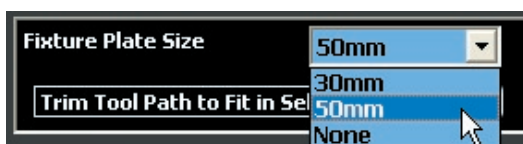
With Rhino open, click on the **RevoCam** software icon found on your desktop following installation. From the first screen, select the **Auto Flip Flat Milling (3 Sides)** strategy.



Click on **Next** to proceed in the software. In the next screen of the software, select the model, supports and all, and input it into the **Model** preview window by clicking on the yellow “IN” arrow beneath the **Model** preview window. Do not select the cutout curve during this step (it will be deselected when it is entered into the **Model** preview window). We will re-select it and input it into the software in a future step.

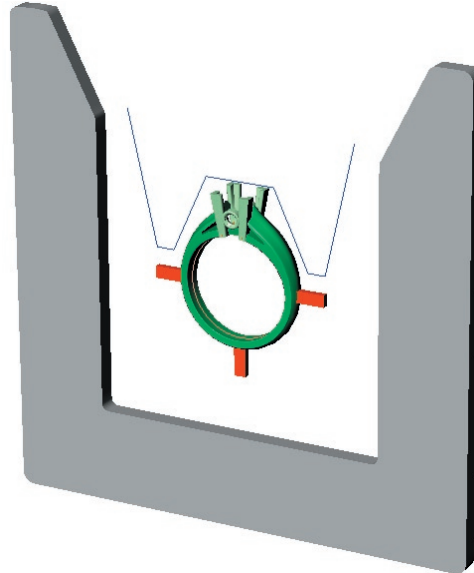


Next, select the **Fixture Plate Size**, or, the size of the plates that will be used during this mill job. The two options are 50 mm open and None. Use the drop-down menu to select the plate that best fits the model size and click once on the selected plate to place it in the menu box.



The selected plate will now be visible in the

viewports, helping the user verify that the selection was correct. The plate should not overlap any part of the model: if it does, choose “None”.



Finally, make sure that the option **Trim Tool Path to Fit in Selected Plate** is turned on. This selection helps ensure that the cutter will not crash into the metal plate, which would cause the cutter to break, by automatically trimming the toolpath to fit within the opening in the selected plate. With this option selected, the toolpath will be safely trimmed to 1.5 mm shy of the opening of the selected plates regardless of the size of the custom borders used. The default setting for this option is ON. (It is possible, though it is not recommended, to deselect this option.)



*Note: the **Trim Toolpath to fit in Selected Plate** option should ALWAYS be turned on in order to ensure that the tool and/or spindle will not crash into the fixture or the plate.*

Using no plates: If the model is too large to fit in even the 50 mm plates, select **None**. The two-sided auto flip fixture will appear. The model should not “pass” it on any side. If it does, or if the toolpath will cause the cutter and/or spindle to crash into the fixture or the pins, the part will need to be milled in two pieces, or more, to accommodate the large size. There are a few more accommodations that need to be made prior to milling when not using plates. It is very important to check both the toolpath and the cutter length to ensure that the cutter and/or

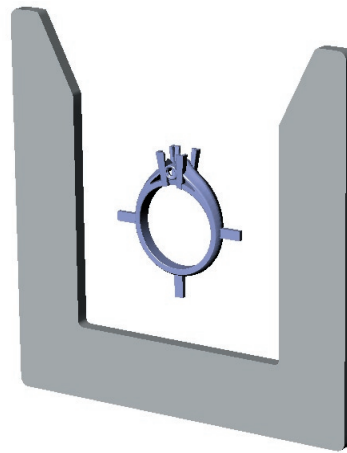
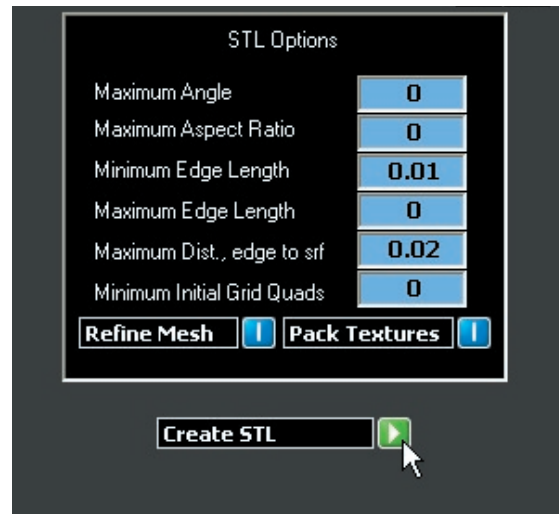
spindle will not crash into the fixture when milling any auto-flip strategy without plates:

(1) Check the toolpath; then, double-check it! After creating the toolpath in the **RevoCam**, MAKE CERTAIN that it does not crash into the fixture or the pins!! This must be done visually, since the fixture plate is not accounted for during toolpath creation.

(2) Check the cutter length; then, double-check it! After installing the two-sided auto-flip fixture, use the **Jog Controls** section of the **Revo** interface to rotate the **A** axis until the fixture is flat. Then, jog the cutter down next to (not into!) the wax, roughly 1 mm past center, and visually check the depth of the cutter to make certain that the spindle cap will not hit the pins that hold the wax in place. If the cutter is too short, the spindle cap will likely hit the fixture. If it does appear that it will hit the fixture, re-install the cutter, holding it just 0.25 inches from the end. Insert it into the collet and tighten. Then, check the depth again and perform the **Seek Tool** operation before cutting the part.

Create STL: After all of the settings are correct, click on **Next** to proceed to the **Create STL** screen in the software. In this screen, it is recommended that the user keep all of the default settings. Changing these defaults adjusts the density and composition of the mesh “skin” that is placed over the model in order to generate the tool path, and can greatly increase the file size, which affects the performance of both **Rhino** and the **Revo Control** software.

To proceed in this screen, simply click on the green **GO** arrow beside **Create STL**. A fine, steel gray mesh will be wrapped around the whole model, creating a tight seal over every part. Because the software uses this mesh to create the toolpath, all of the pieces of the model are recognized as one by the mill, even if the model is not booleaned together. The toolpath is based upon the mesh, not upon the separate pieces of the model, saving users time and sanity by eliminating frustrating Boolean union operations!



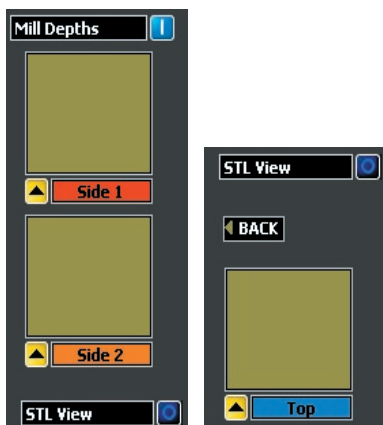
Click on **Next** to proceed in the software.

Borders and Mill Depths: In this screen, the user will input **Mill Depths** and **Custom Borders** using the tools found on-screen. This is also the screen where the **Cut Out** curve will be input into the software. **Mill Depths** define the maximum depth to which the tool will cut on both sides of the part, while **Custom Borders** define the area around the model that the tool will cut. The remaining two controls in this screen allow the user to toggle between the **STL** and **Nurbs** views of this model. If need be, select either of these two options to properly see all of the details on the model. This helps the user make certain that the **Mill Depths** and **Custom Borders** are properly set up.

Setting Mill Depths: To begin, select the **ON** button beside **Mill Depths** to define the maximum depth that the cutter will plunge when cutting the

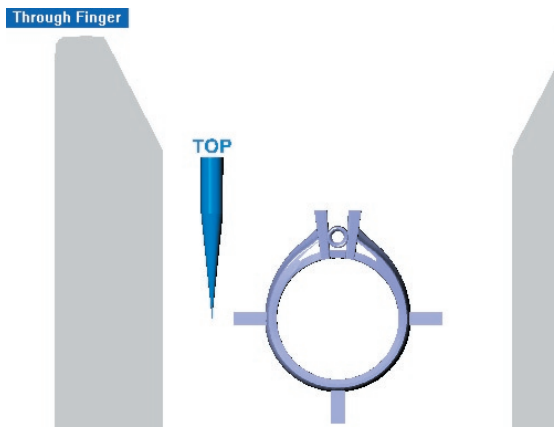
part. This activates the three planes that are used to define the mill depths. In addition to the two (**Side 1** and **Side 2**) mill depths provided in the two-sided auto flip process, there is now a third mill depth (**Top**) that is provided. The **Top** mill depth constitutes the third side of this three-sided operation and allows the user to indicate the depth that the cutter will plunge into the wax when cutting out the head and top portion of the shank. After turning on **Mill Depths**, the **Side 1**, **Side 2**, and **Top** preview windows will appear in the software screen. In the viewports, the **Side 1**, **Side 2**, and **Top** mill depths will appear.

Select “Mill Depths”; Preview windows appear:

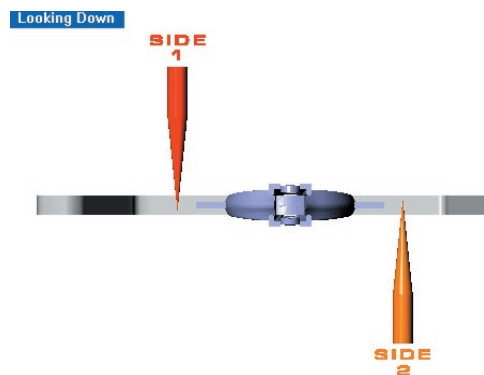


Viewports: Mill depths appear:

Top mill depth is set in **Through Finger (Front)** viewport:



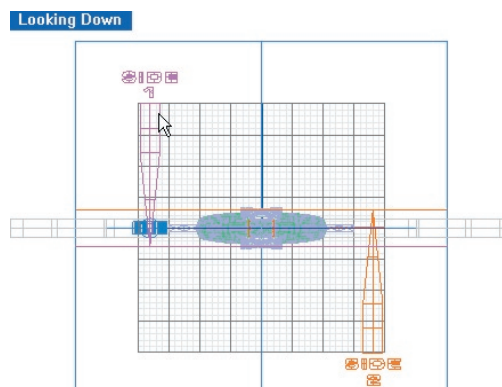
Side 1 & **Side 2** mill depths are set in **Looking Down (Top)** viewport:

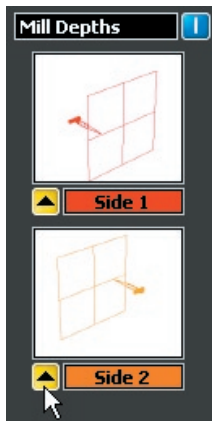


The mill depth is a “grouped” item that consists of a cutter, a plane, and a label. (Because the items are grouped, clicking on any of the items in the group will cause the entire “mill depth” to be selected.) The location of the plane portion of the Mill Depth defines the maximum depth to which the cutter will plunge when cutting the part.

When setting mill depths, it is not possible to set the planes past the extents of the part. They must be touching the model in order to successfully generate a toolpath. For best results, set each **Side** mill depth to the furthest extent of that side of the model without going past it. This will ensure that the maximum detail possible will be cut from each side of the model. In the case of the **Top** mill depth, this should be set only as deep as needed to complete the details on the head and top portion of the shank (recommended setting is 1 mm past the bottom of the head).

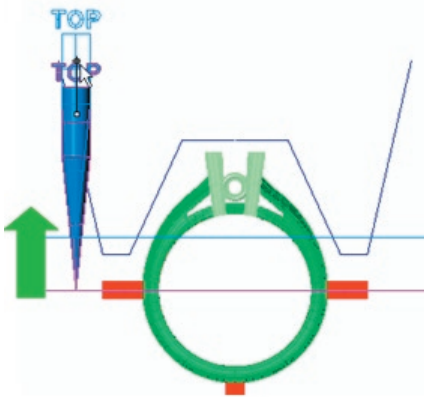
To set the Side mill depths, select and drag the mill depths in the **Looking Down (Top)** just past the center of the part, without passing the furthest extents of the part, and input them into their respective preview windows in the software screen.



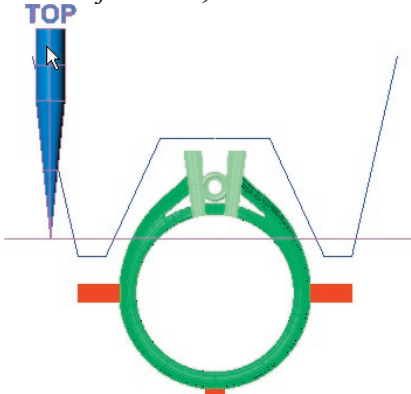


To set the **Top** mill depth, select and drag the mill depth up to just below the inside of the head and/or the details on the sides of the shank that will need to be cut from this angle. This saves the cutter from being plunged too far into the wax, which could cause the shank of the cutter to rub away features on the ring.

In this example, as in most, drag the Top mill depth up, since the plain shank does not warrant such a deep cut as is initially set by the default Top mill depth.



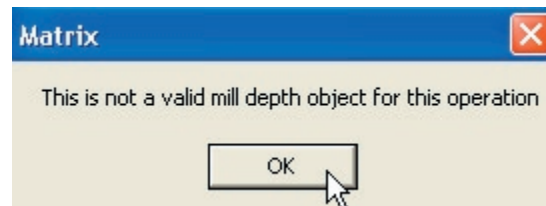
(Below: The Top mill depth is 1 mm below the bottom of the head)



Select the **Top** mill depth and input it into the corresponding preview window in the software.



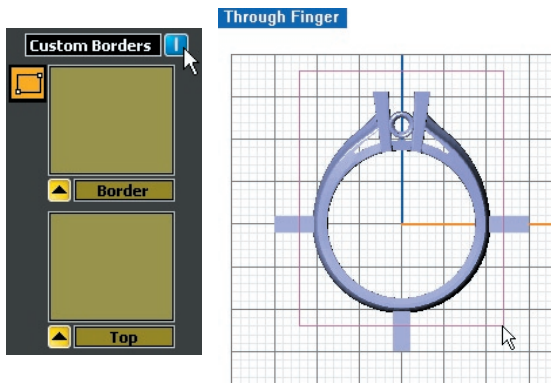
This screen of the software will not allow the user to mistakenly place anything but the correct mill depth into its corresponding **Mill Depth** preview window, so have no fear! If the other mill depth or any other object is selected and the user attempts to input it into this box, the following error message will appear:



If this error message appears, click on **OK** and deselect everything in the viewports by clicking once on empty space. Then, select just the mill depth that corresponds to the label on the preview window and click on the yellow **IN** arrow beneath that preview window again.

Setting Custom Borders: Once the mill depths have been input into their corresponding preview windows, activate the **Custom Borders** control to see the tools used to define custom borders for each portion of the operation: both the top 90-degree cut and the traditional auto-flip cut. This will cause the **Border** and **Top** preview windows and the **Rectangle** curve command to appear onscreen.

Set the first **Border** in the **Through Finger (Front)** viewport. This border represents the extents of the traditional auto-flip operation. To begin, select the **Rectangle** command and, working in the **Through Finger (Front)** viewport, draw a rectangle or square that is about 1.5 to 2 mm above and below the extents of the part and that does not extend past the support tabs of the model.



This rectangle (or square) represents the area that will be used to generate the toolpath region. The cutter will remain inside this border, so be certain that the border stays within the extents of the support tabs, or else the tabs will be cut away from the surrounding wax and the part will “fall through”! Also make sure that the border is at least 1.5 millimeters away from the extents of the part so that there is enough space to comfortably cut the part away from the wax.

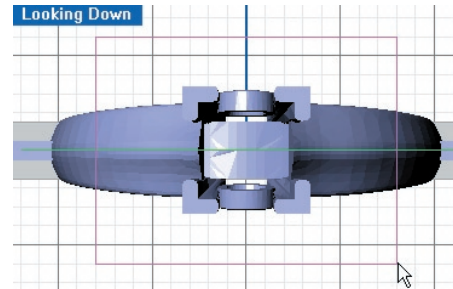
IMPORTANT: Drawing the Border in any viewport EXCEPT the Through Finger (Front) viewport will generate an incorrect toolpath!

Once the correct border has been drawn, select the border and input it into the **Border** preview window by clicking on the yellow input arrow beneath the window.

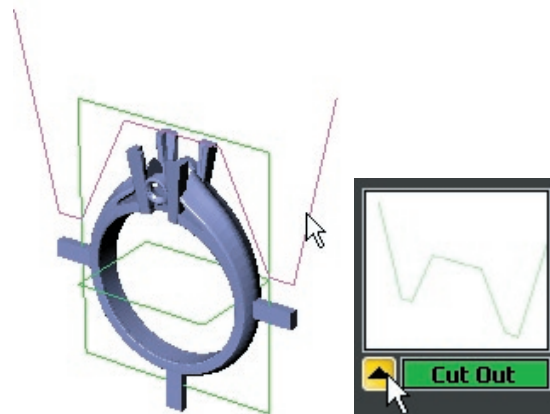


Now, to create the border for the top 90-degree operation that will cut out the head and upper portion of the shank of the ring, select the **Rectangle** command once more and draw a rectangle representing the **Top** border in the **Looking Down (Top)** viewport. Remember that this rectangle defines the area of what will be cut in the top-down operation. It should surround the

head and upper portion of the shank that needs to be cut from the top 90-degree angle, leaving about 1.5 to 2 mm on each side for clearance. If the shank is plain, it need not be included in the **Top** border: it is fine to only include the head, if this is the only detail that needs to be cut. Input this border into the **Top** preview window.

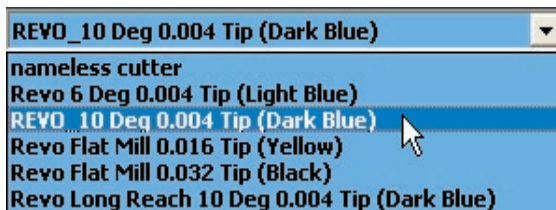
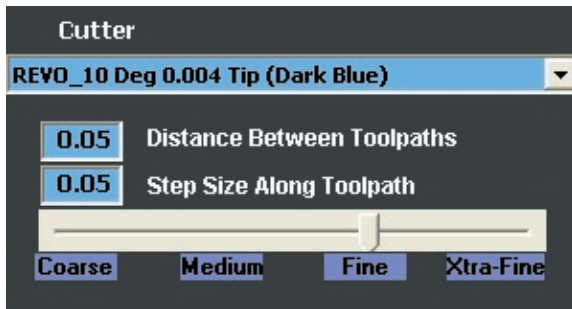


Finally, select the cut-out curve and input it into the **Cut-Out** preview window.

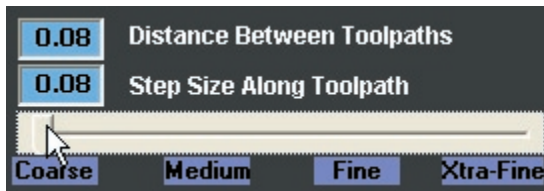
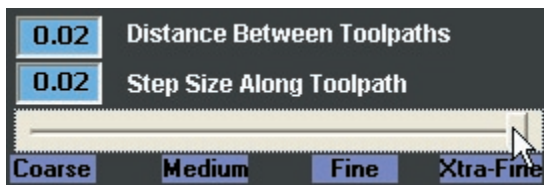


Congratulations! The hard part is done!! Click on **Next** to proceed in the software.

Selecting a Cutter: After the user sets the mill depths and custom borders, the software offers a **Cutter** selection screen. Recall that the best cutter for most jobs (especially for the first pass) will usually be the **Revo 10 Degree** cutter. This is the default setting in this screen. To change the cutter, click on the drop-down menu arrow and select another cutter from the list by double-clicking on it. The selected cutter will appear in the menu box.



Defining the Quality of the Cut: There are two ways to use the controls that define the quality of the cut. The first is to click and drag the slider control, moving it along the scale from **Coarse** to **Xtra-Fine** until the desired setting is achieved.

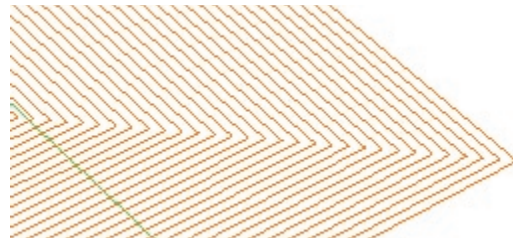


The second way to change these settings is to manually type a new value into the **Distance Between Toolpaths** and **Step Size Along Toolpath** text boxes. Lowering these values increases the quality of the mill job; however, it also increases the time that the mill job will take. For example, raising this value to **Xtra-Fine** will more than double the time it takes to run the job. It is not recommended to raise or lower these values beyond those available on the scale.

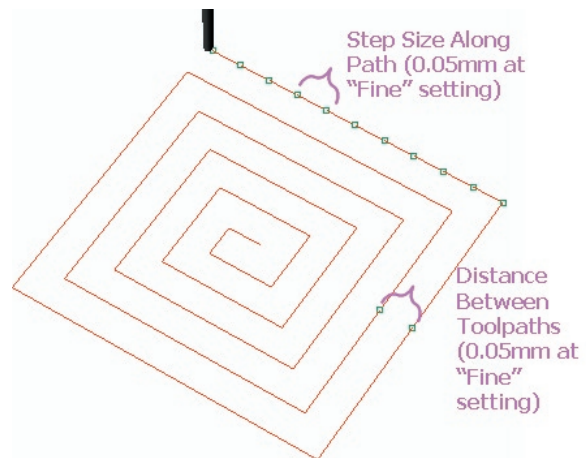
The blue text fields are provided so that the user can change one value or the other. For example, leaving the slider setting at **Fine** and changing

the **Distance Between Toolpaths** to 0.3 will increase the finish quality to basically the same as the “extra fine” setting without doubling the mill time: it only adds about 20 minutes to an average mill job.

To understand what each value means, it is important to understand how the toolpath is made. A flat toolpath starts from the center of a given side of the model and works outward in a concentric pattern toward the extents of the piece. Zooming in close to one corner of the toolpath shows the corners of the concentric rectangles.



Another representation of the toolpath, though not created to scale, is found below. Remember that the start of the toolpath is at the exact geometric center of the given border. The space that is left between the concentric rectangles is equal to the value listed beside **Distance Between Toolpaths**.



Step Size Along Path: This value refers to the distance between the reference points that the cutter uses to orient itself to the part. Because it cannot calculate every possible point along the line, since the mathematics would be staggering, the mill computer instead plots points along the toolpath and the cutter advances to these points.

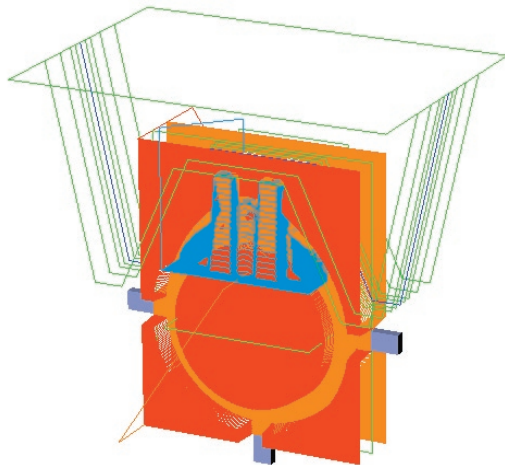
The further away these points are, the less detail will be achieved in the finished part. The closer

together these points are, the more detail will be achieved; however, the mill job will also take longer to run. The **Step Size Along Path** defines how close these points are along the toolpath. As illustrated above, for a “Fine” milling job they are 0.05 mm apart, which is pretty close.

So, in a way, when the mill follows a toolpath, it is really just playing an elaborate game of “connect the dots”!

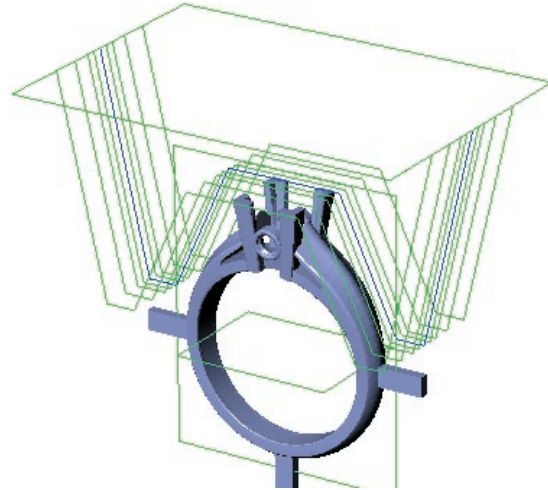
After selecting a cutter and setting the values for these two parameters, proceed to the **Next** screen in the software, **View and Save RVO File**.

View and Save RVO File: In the final screen of this software, begin by clicking on the red **Make RVO File** button. Wait a few minutes, and the toolpath will be created around the model. Now, use the remaining controls to view the toolpath, STL mesh, Nurbs Model, and/or the Mill Heights as desired to check the toolpath that was generated by the software. Using these controls, inspect the model to ensure that the toolpath is correct, and then click on **Save Revo File**.

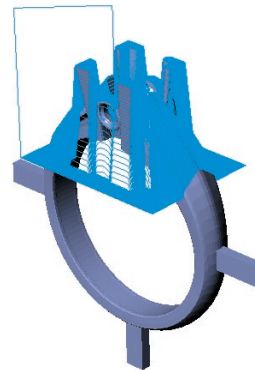


Understanding the Three-Sided Auto Flip Toolpath:

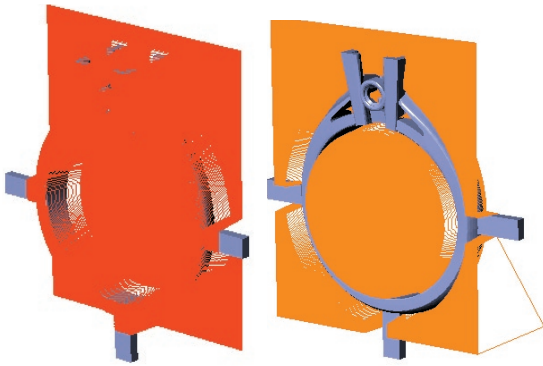
Recall that this toolpath starts with a trace cut that follows the cut-out curve to remove the cut-out shape at the top of the wax (the portion of the wax blank that faces the open ends of the plates). This will allow the cutter to better reach the head and top of the shank in the next step. This part of the toolpath is shown in green:



After the user removes the cutaway from the wax, the A-Axis rotates the wax 90-degrees and cuts the head and upper-shank details, as represented by the blue toolpath. When checking this toolpath, make certain that all of the details at the top of the shank that cannot be reached by the traditional two-sided auto-flip operation are included in the blue toolpath. If they are not, the custom border will need to be re-drawn and re-input in the **Borders and Mill Depths** screen of the software. Then, it will be necessary to re-generate the toolpath.



Next, the ring is rotated back into the traditional two-sided auto-flip positions and the front and back of the model are cut by the mill, as represented by the red and orange toolpaths:



Adjust the controls for viewing the **Toolpath**, **STL**, **Nurbs Model**, and **Mill Heights** to see only what is necessary to check the accuracy of the toolpath. To view the original Nurbs model and compare the accuracy of the toolpath against that model, turn on the **Nurbs Model** setting, which turns off the **STL** setting. To see only the model, not the toolpath, turn OFF the **Toolpath** button. To keep the **Mill Heights** from blocking the shaded view of the model, turn them off. The default setting is shown below; however, these controls do not affect the RVO file that has been generated during this step: they only allow the user to better see both the part and the toolpath.



Once the toolpath has been deemed satisfactory, click on the **Save Revo File** button and select a location on the computer to which the file will be saved. When the progress bar reaches 100%, the file has been saved. It is now ready to be transferred to the **Revo** software found on the mill computer.



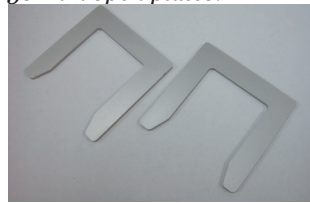
Assembling the fixture for a Three-Sided Auto Flip Strategy:

After turning on the mill and the mill computer, start up the **Revo Control Software** and click on the **Load Revo Project** icon and navigate to the file in which the mill job was saved. Select and open the mill job. Wait just a moment for the project to load.

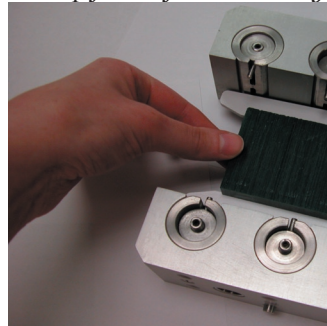
As with every mill job, begin with the selected cutter installed. If this is a new cutter, run the **Seek Tool** operation before starting the job.

VERY IMPORTANT: Remember to use the open fixture plates for this strategy!! To set up the fixture for the three-sided auto flip strategy, start by placing the first plate in the fixture, resting it on the lower set of pins, flush against the black spacer piece that should be positioned with the arrow towards the opening of the fixture. Then, center the wax in the plate by lining up the end of the wax with the inside edge of the “V” in the plate.

50-mm open plates:

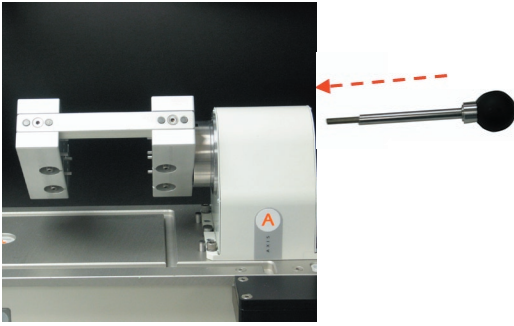


Line up front of wax with edge of “V” in plates.



Slide the second plate over the wax and beneath the upper pins. Tighten the threaded screws with the Allen wrench so that the pins move towards one another and cause the plates to hold the wax securely. Remember that it is best to tighten the screws diagonally across from one another, tightening them in sequence from 1-4 until the plates are snug. A slight bowing may be noticeable in the plates when they are tight.

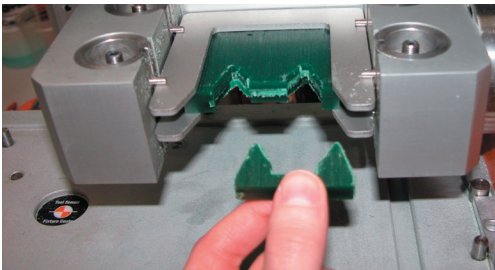
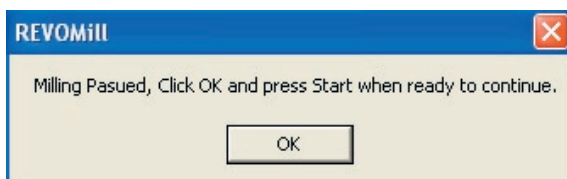
Affix the two-sided fixture to the mill, making sure that the pin lines up with the slot (marked with a “V”) on the mill. Tighten the draw bar to hold the fixture firmly in place. *Two-sided fixture / A axis / Draw Bar:*



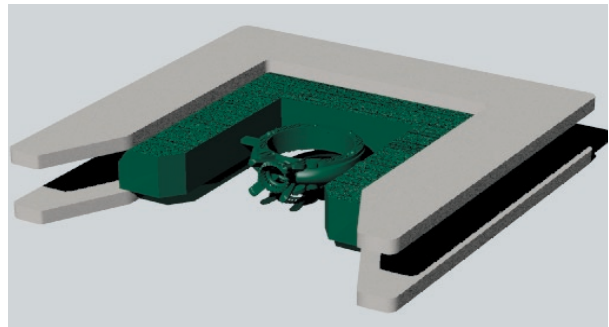
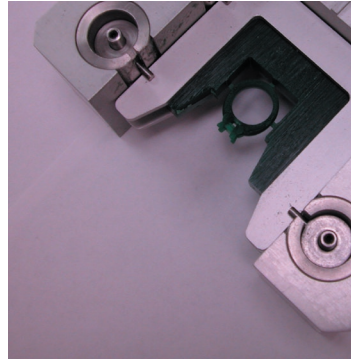
With the correct cutter installed, the mill homed, and the fixture properly secured to the mill, click on **Start**.



The first part of the toolpath, the cut-out, will be milled from both sides. The mill will pause once the shape of the cut-out has been milled through both sides of the wax. Gently slide the cut-out piece away from the remaining wax.



After removing the wax cut-out, click on **OK** in the Revo software, click the **Coolant ON** button, and then click the **Start** button to resume cutting. The remaining parts of the mill job are automated and do not require user intervention. The mill will now rotate the wax 90 degrees to complete the head (the "Top" cut). It will then return the wax to the familiar two-sided auto-flip positions and complete the milling process.



Chapter 10: Preparing & Milling the Auto-Flip Plus Rotary Strategy

About this Strategy:

The **Auto-Flip Plus Rotary** strategy starts with a traditional two-sided auto-flip operation, and then adds to that a one-sided rotary operation. It is best suited to a ring with details on both the front and back of the shank and with details that wind all the way around the shank.

If the ring has details that would be missed by either the rotary one-sided operation or by the three-sided auto flip operation, this is the strategy to use. In the ring shown below, for example, the three-sided auto-flip strategy would not be able to create the split shank that runs all the way down the sides of the ring; however, the rotary ring strategy would not be able to mill out the space between the gallery rails of the head.

First part of strategy, two-sided auto flip, mills details on front and back of ring. Then, the second part of the strategy, one-sided rotary, mills all the way around the outside of the shank:



The model is cut with a center hub and supports to hold the model to the rotary fixture during the second part of the job. The hub and supports are located in the center of the fingerhole and cut out during the first operation.

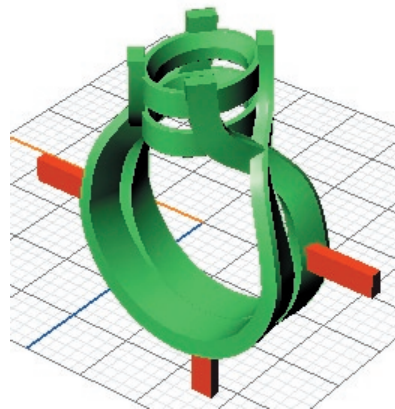
The user need not draw these supports (although the option to do so is available in the case of unique models): they are created as a step during the process of preparing the model within the **RevoCam** software.



Preparing the Model in Rhino:

Drawing the outer support tabs: Before this model can be input into the software, supports must be attached to it that will hold it to the wax during the first part of the job (the auto-flip operation). That way, the model will not fall away from the surrounding wax while it is being cut. Draw supports using the **Box** command found in Rhino.

To properly support the model, draw boxes that fully sink into the surface of the design and are roughly 1 mm wide X 2 mm tall. Create them with enough length so that there is sufficient space between the part and the surrounding wax to comfortably cut the piece away from the wax between the two-sided operation and the rotary operation. In this example, one support was created and then rotated around 0,0,0, with “Copy = Yes” indicated in the **Command** line, in order to place three supports around the piece.

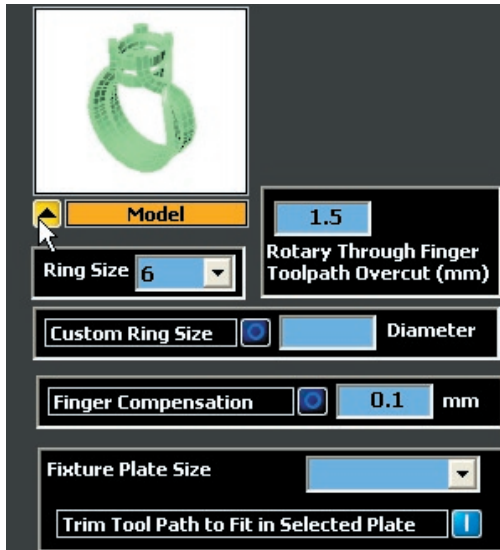
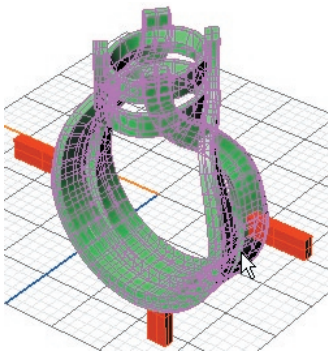


Using the RevoCam Software:

With Rhino open, click on the **RevoCam** software icon found on your desktop following installation. From the first screen, select the **Auto Flip Plus Rotary (3 Sides)** strategy.



Click on **Next** to proceed in the software. In the next screen of the software, select just the model and input it into the **Model** preview window by clicking on the yellow “IN” arrow beneath the **Model** window. Do not select the supports during this step: these will be input into the software in a future screen.



With a Matrix ring rail in the viewports, the software will be able to read the ring size, which will be automatically entered into the **Ring Size** drop-down menu. If a Matrix ring rail is not a part of the model, the user will need to enter the Millimeter value for the finger size in the Diameter text box.

It is very important to include the correct ring

rail size because this value is used to “trim” the toolpath, ensuring that the cutter will not go any deeper than the depth of the ring rail. This also provides a witness of the proper finger size to which the ring should be finished.

Custom: If a custom finger size is required, insert the value for the diameter of the rail, in millimeters, into this box.

Rotary Through Finger (Front) Toolpath

Overcut: The recommended setting for this value is the default: 1.5 mm. This ensures that there will be a small “overcut” on either side of the band so that the ring can be cut from the wax without any damage to the model. To widen the space that is left on either side of the ring, raise this value; to narrow this space (not recommended!

Finger Compensation: If ready ring wax is being used, this value will allow the cutter to cut into the butterboard about 0.1 mm instead of trimming the toolpath to the ring rail, which helps the user free the wax from the butterboard. This can be adjusted shallower or deeper if so desired.

Fixture Plate Size: Next, select the **Fixture Plate Size**, or, the size of the plates that will be used during the two-sided auto-flip portion of the operation. The closed fixture plates are used with this strategy. Use the drop-down menu to select the plate that best fits the model and click once on the selected plate to place it in the menu box.



The selected plate will now be visible in the viewports, helping the user verify that the selection was correct. The plate should not overlap any part of the model: if it does, choose the next size up.

Finally, make sure that the option **Trim Tool Path to Fit in Selected Plate** is turned on. This selection helps ensure that the cutter will not crash into the metal plate, which would cause the cutter to break, by automatically trimming the toolpath to fit within the opening in the selected plate. With this option selected, the toolpath will be safely trimmed to 1.5 mm shy of the opening of the selected plates regardless of the size of the custom borders used. The default setting for this option is ON. (It is possible, though it is not recommended, to deselect

this option.)

Trim Tool Path to Fit in Selected Plate

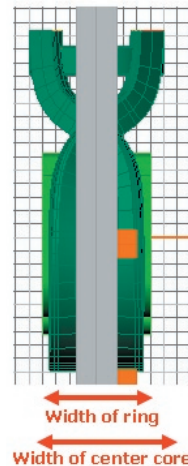
*Note: the **Trim Toolpath to fit in Selected Plate** option should ALWAYS be turned on in order to ensure that the tool and/or spindle will not crash into the fixture or the plate.*

Using no plates: If the model is too large to fit in even the 50 mm plates, select **None**. The two-sided auto flip fixture will appear. The model should not “pass” it on any side. If it does, or if the toolpath will cause the cutter and/or spindle to crash into the fixture or the pins, the part will need to be milled in two pieces, or more, to accommodate the large size. There are a few more accommodations that need to be made prior to milling when not using plates. It is very important to check both the toolpath and the cutter length to ensure that the cutter and/or spindle will not crash into the fixture when milling any auto-flip strategy without plates:

(1) Check the toolpath; then, double-check it! After creating the toolpath in the **RevoCam** software, MAKE CERTAIN that it does not crash into the fixture or the pins!! This must be done visually, since the fixture plate is not accounted for during toolpath creation.

(2) Check the cutter length; then, double-check it! After installing the two-sided auto-flip fixture, use the **Jog Controls** section of the **Revo** interface to rotate the **A** axis until the fixture is flat. Then, jog the cutter down next to (not into!) the wax, roughly 1 mm past center, and visually check the depth of the cutter to make certain that the spindle cap will not hit the pins that hold the wax in place. If the cutter is too short, the spindle cap will likely hit the fixture. If it does appear that it will hit the fixture, re-install the cutter, holding it just 0.25 inches from the end. Insert it into the collet and tighten. Then, check the depth again and perform the **Seek Tool** operation before cutting the part.

Creating the inner supports: In the **Next** screen of the software, the user specifies parameters for the inner supports that will be created by the software. The inner supports are attached to a center core. The thickness of this core should be just slightly more than that of the ring shank. In this case, the ring is 8 mm wide, so the size of the center core should be 8.1 mm. To set this value, type it into the text box beside **Center Core Size**.

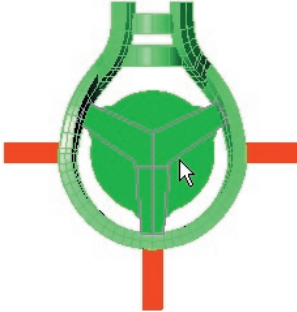


Note: After milling Part 1 of this job (the two-sided auto-flip portion of the job), it is important to measure the actual thickness of the Center Core so that the computer can compare it against the value input here. The mill will then compensate for the difference between the intended and actual thicknesses to re-center itself prior to running the second part of the job. This helps to ensure the accuracy of the model during this strategy.

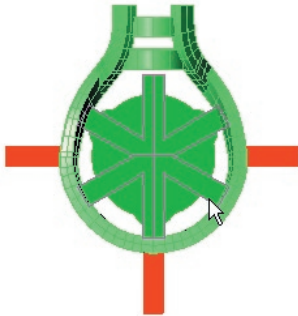
To select a number of inner supports to attach to the center core, consider the needs of the model. The available numbers of center supports are 3, 6, or none. In the illustrations below, it is apparent that there are two main parts to the inner support structure: the center core, which is rounded, and the inner supports, which connect the center core to the inner walls of the shank. The program will always create the Center Core, which works in conjunction with the three dowels located on the face of the mandrel in order to align the model to

the rotary fixture. Depending on whether the user selects 3 or 6 inner supports, the software will draw that number of extensions to hold the ring to the center structure. In most cases, only three extensions are necessary to support the model.

Three inner supports:



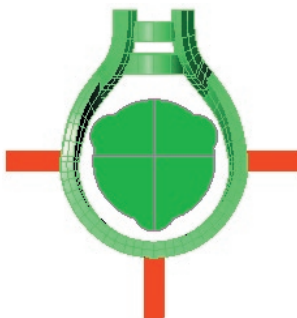
Six inner supports:



Selecting none means that the user has already drawn supports, in Rhino, that will affix to both the center core and the model.

Note: Select NONE if the ring has special features that will need extra support AND if these supports have ALREADY been drawn in Rhino. If they are needed but have not yet been created, go BACK into Rhino and create them. Then return to this screen in the software.

With no inner supports: (User must return to Rhino and draw them)



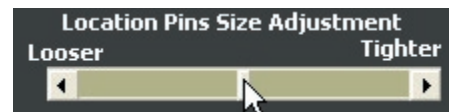
After selecting the number of inner supports, click on GO beside **Create Center**. After a moment, the center supports will appear inside the model. Select the entire inner support structure (core and Y-shaped supports, plus any additional supports created in Rhino) and input them into the **Inner** supports preview window in the software.

Note: All of the supports MUST be selected and entered into the software at the same time!

Select the outer supports, the ones that will hold the ring to the surrounding wax during the two-sided auto-flip part of the job, and input these into the **Outer** supports preview window.

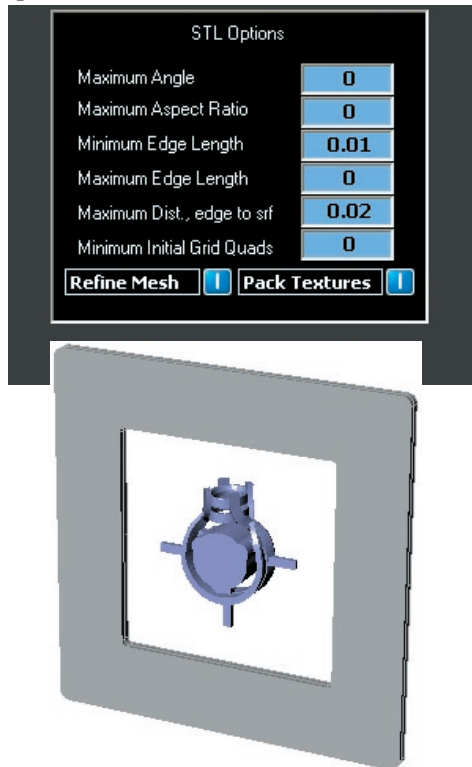


Finally, adjust the size of the holes for the location pins, if necessary. Though the default should be fine for most jobs, it is possible to adjust the size of the pinholes in the Center Core structure so that they fit more securely over the dowel pins on the face of mandrel. Moving the slider bar left or right Loosens or Tightens the size of the holes by tiny amounts for a looser or tighter fit.



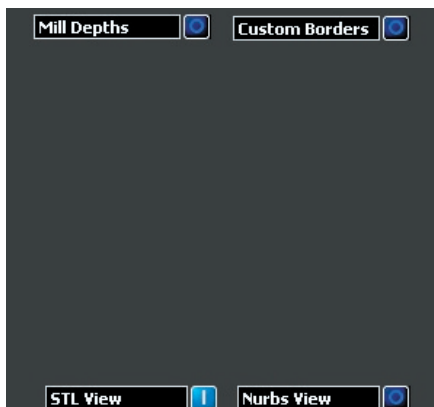
Create STL: After all of the settings are correct, click on **Next** to proceed to the **Create STL** screen in the software. In this screen, it is recommended that the user keep all of the default settings. Changing these defaults adjusts the density and composition of the mesh “skin” that is placed over the model in order to generate the tool path, and can greatly increase the file size, which affects the performance of both the **Rhino** and the **Revo** software. To proceed in this screen, simply click on the green **GO** arrow beside **Create STL**. A fine, steel gray mesh will be wrapped around the whole model, creating a tight seal over every part. Because the software uses this mesh to create the toolpath, all of its pieces are recognized as one by the mill, even if the model is not booleaned together. The toolpath is based upon the mesh, not off of the

separate pieces of the model, saving users' time and sanity by eliminating frustrating Boolean union operations!



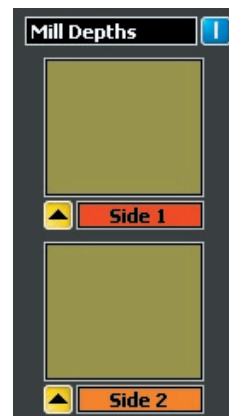
Click on **Next** to proceed in the software.

Borders and Mill Depths: In this screen, the user will input **Mill Depths** and **Custom Borders** using the tools found on-screen. **Mill Depths** define the maximum depth to which the tool will cut on both sides of the part, while **Custom Borders** define the area that the tool will cut around the model. The remaining two controls in this screen allow the user to toggle between the **STL** and **Nurbs** views of this model. If need be, select from among these two options to properly see all of the details on the model. This helps the user make certain that the **Mill Depths** and **Custom Borders** are properly set up.

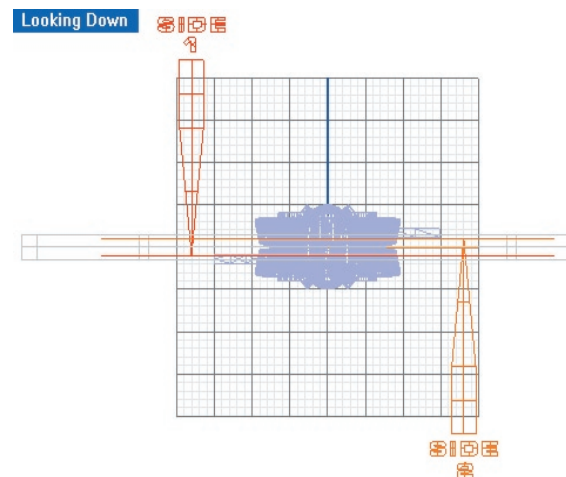


Setting Mill Depths: To begin, select the **ON** button beside **Mill Depths** to define the maximum depth to which the cutter will plunge when cutting the part. This activates the two planes that are used to define the mill depths. In the software screen, the **Side 1** and **Side 2** preview windows will appear. In the viewports, the **Side 1** and **Side 2** mill depth planes will appear.

Select "Mill Depths": Onscreen, two preview windows appear



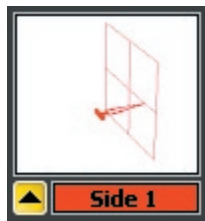
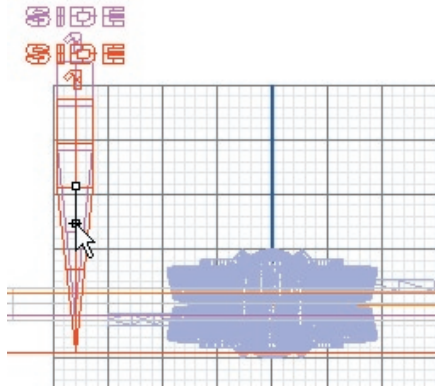
In the Viewports, two mill depths appear



*Note: When observing the positioning of the mill depths, remember that Side 1 is the "top" of the model and Side 2 is the bottom. Between the operations that cut Side 1 and Side 2, the piece is rotated on the **Rotary A Axis** so that the bottom is now in the top position and vice-versa. So, even though the mill depth for Side 2 appears to come "up" from the bottom of the model, the cutter will*

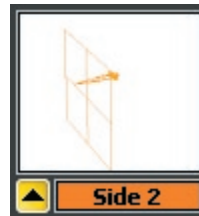
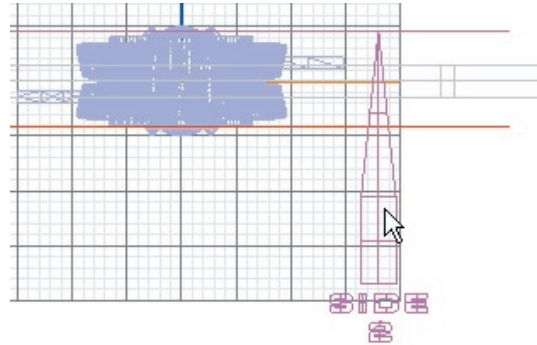
actually come down from the top once the model is rotated.

The mill depth is a “grouped” item that consists of a cutter, a plane, and a label. (Because the items are grouped, clicking on any of the items in the group will cause the entire “mill depth” to be selected.) The location of the plane portion of the Mill Depth defines the maximum depth to which the cutter will plunge when cutting the part. Make certain that this plane reaches through all of the details on that side of the part that the user wishes to cut (**Side 1** or **Side 2**). After setting the mill depth, select it and input it into the corresponding window (**Side 1**, below).

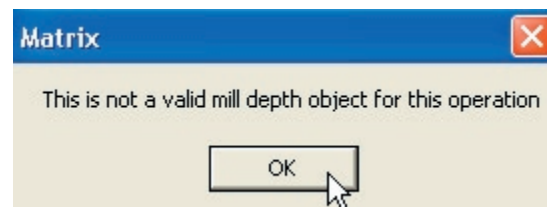


When setting mill depths, it is NOT recommended to set the planes past the extents of the part. They must be touching the model in order to successfully generate a toolpath. For best results, set each mill depth to the furthest extent of the model as possible without going past it. This will ensure that maximum detail will be cut from this side of the model.

Select the Mill Depth for Side 2 and place it into the appropriate preview window in the software, as well.



This screen of the software will not allow the user to mistakenly place anything except the correct mill depth into its corresponding **Mill Depth** preview window, so have no fear! If the other mill depth or any other object is selected and the user attempts to input it into this preview window, the following error message will appear:



If this error message appears, click on **OK** and deselect everything in the viewports by clicking once on empty space. Then, select just the mill depth that corresponds to the label on the preview window and click on the yellow **IN** arrow beneath that preview window again.

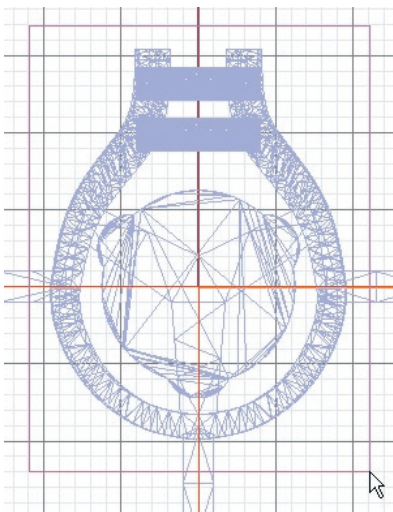
Setting Custom Borders: Once the mill depths have been input into their corresponding preview windows, activate the **Custom Borders** control to see the tools used to define a custom border for the part. This will cause the **Border** preview window and the **Rectangle** curve command to appear in onscreen.



For this strategy, the custom border will be either a rectangle or a square that is drawn in the **Looking Down (Top)** viewport. To begin, select the rectangle tool, starting up this **Curve** command. Then, working in ONLY the **Through Finger (Front)** viewport, draw a border that surrounds the part but falls within the extents of the support tabs. This rectangle is essentially defining the area that will be used to generate the toolpath region.

The cutter will remain inside this boarder or region, so be certain that the support tabs stay **WITHIN** the extents of the border, or else it too will be cut away from the surrounding wax and the part will “fall through”! Also make sure that the border is at least 1.5 mm away from the extents of the part so that there is enough space to comfortably cut the part away from the wax.

Use the Rectangle tool to draw a custom border surrounds the part. DO NOT extend the rectangle beyond the ends of the support tabs and DO NOT overlap, touch, or get too close to any part of the model itself!



IMPORTANT: Drawing the custom border in any viewport **EXCEPT** the Through Finger (Front)

viewport during this strategy will cause the software to generate an incorrect toolpath!

Once the correct border has been drawn, select the border and input it into the **Border** preview window by clicking on the yellow input arrow beneath the window.

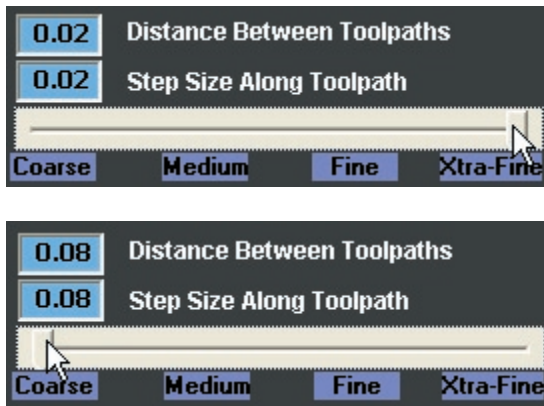


Congratulations! The hard part is done!! Click on **Next** to proceed in the software.

Selecting a Cutter: After the user sets the mill depths and custom borders, the software offers a **Cutter** selection screen. Recall that the best cutter for most jobs (especially for the first pass) will usually be the **Revo 10 Degree** cutter. This is the default setting in this screen. To change the cutter, click on the drop-down menu arrow and select another cutter from the list by double-clicking on it. The selected cutter will appear in the menu box.



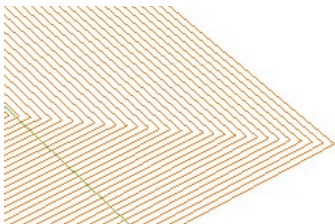
Defining the Quality of the Cut: There are two ways to use the controls that define the quality of the cut. The first is to click and drag the slider control, moving it along the scale from **Coarse** to **Xtra-Fine** until the desired setting is achieved. Raising the quality up to **Xtra-Fine** lowers these numbers. Lowering the quality raises these values.



The second way to change these settings is to manually type a new value into the **Distance Between Toolpaths** and **Step Size Along Toolpath** text boxes. Lowering these values increases the quality of the mill job; however, it also increases the time that the mill job will take. For example, raising this value to **Xtra-Fine** will more than double the time it takes to run the job. It is not recommended to raise or lower these values beyond those available on the scale.

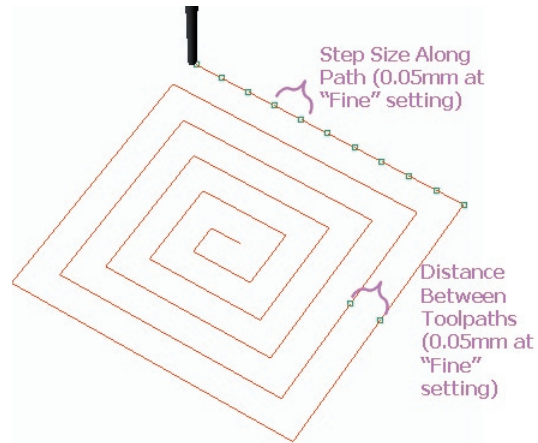
The blue text fields are provided so that the user can change one value or the other. For example, leaving the slider setting at **Fine** and changing the **Distance Between Toolpaths** to 0.3 will increase the finish quality to basically the same as the “extra fine” setting without doubling the mill time: it only adds about 20 minutes to an average mill job.

To understand what each value means, it is important to understand how the toolpath is made. A flat toolpath starts from the center of a given side of the model and works outward in a concentric pattern toward the extents of the piece. Zooming in close to one corner of the toolpath shows the corners of the concentric rectangles.



Another representation of the toolpath, though not created to scale, is found below. Remember that the start of the toolpath is at the exact geometric center of the given border. The space that is left between the concentric rectangles is equal to the value listed

beside **Distance Between Toolpaths**.

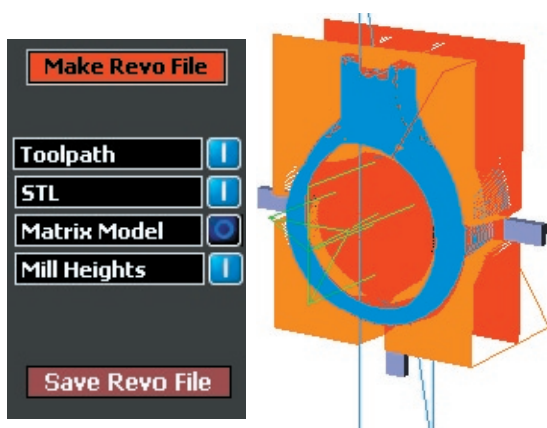


Step Size Along Path: This value refers to the distance between the reference points that the cutter uses to orient itself to the part. Because it cannot calculate every possible point along the line, since the mathematics would be staggering, the mill computer instead plots points along the toolpath and the cutter advances to these points.

The further away these points are, the less detail will be achieved in the finished part. The closer together these points are, the more detail will be achieved; however, the mill job will also take longer to run. The **Step Size Along Path** defines how close these points are along the toolpath. As illustrated above, for a “Fine” milling job they are 0.05 mm apart, which is pretty close. So, in a way, when the mill follows a toolpath, it is really just playing an elaborate game of “connect the dots”!

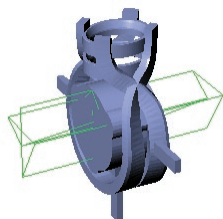
After selecting a cutter and setting the values for these two parameters, proceed to the **Next** screen in the software, **View and Save RVO File**.

View and Save RVO File: In the final screen of this software, begin by clicking on the red **Make RVO File** button. Wait a few minutes, and the toolpath will be created around the model. Now, use the remaining controls to view the toolpath, STL mesh, Nurbs Model, and/or the Mill Heights as desired to check the toolpath that was generated by the software. Using these controls, inspect the model to ensure that the toolpath is correct, and then click on **Save Revo File**.

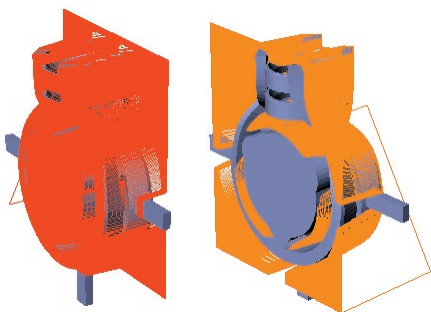


Understanding the Three-Sided Rotary Toolpath:

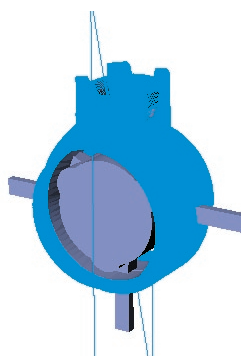
Interestingly enough, this toolpath starts by cutting the holes for the pins in the center support hub in the model. This part of the toolpath is shown in green:



The mill then proceeds to run the typical two-sided auto flip operation, cutting out both the front and back of the ring and the center core and supports that will be used during the next part of the job. These paths are represented in red and orange:



After this process is completed, the user removes the model from the outer supports and the surrounding wax and aligns the wax in the three-sided rotary fixture on the mill. The final process, a rotary one-sided mill job, is represented by the blue toolpath:



Adjust the controls for viewing the **Toolpath**, **STL**, **Nurbs Model**, and **Mill Heights** to see only what is necessary to check the accuracy of the toolpath. To view the original Nurbs model and compare the accuracy of the toolpath against that model, turn on the **Nurbs Model** setting, which turns off the **STL** setting. To see only the model, not the toolpath, turn OFF the **Toolpath** button. To keep the **Mill Heights** from blocking the shaded view of the model, turn them off. The default setting is shown below; however, these controls do not affect the RVO file that has been generated during this step: they only allow the user to better see both the part and the toolpath.



Once the toolpath has been deemed satisfactory, click on the **Save Revo File** button and select a location on the computer to which the file will be saved. When the progress bar reaches 100%, the file has been saved. It is now ready to be transferred to the **Revo** software found on the mill computer.



Before milling this strategy for the FIRST TIME only:

Before running the auto-flip plus rotary strategy for the first time, measure the thickness at the largest diameter on the mandrel that was sent for use with this fixture (the one with the three dowel pins).



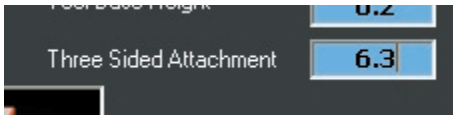
Note: This is a one-time measurement required to pre-set the location of the center of the wax model on the A axis. In most cases, this has already been pre-set by the assembly technicians.

Then, enter this value into the blue box labeled **Three Sided Attachment** in **Step One** of the Mill Calibration process in the Revo control software on the mill.

Click on maroon button to access Calibration interface:



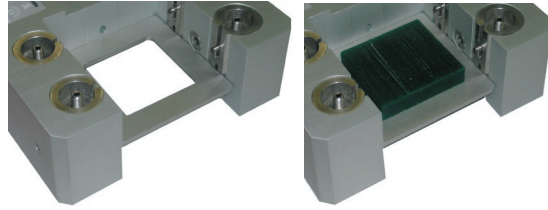
Enter width of mandrel hub into Three-Sided Attachment box:



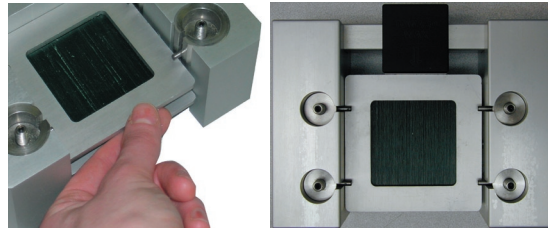
Assembling the fixture for the Auto-Flip Plus Rotary Strategy: Part 1

Recall that this strategy starts with the Two-Sided Auto Flip as Part I. This is a typical Two-Sided Auto Flip Fixture setup.

With “**Side 1**” side of the two-sided auto-flip fixture facing up, place the gauge block over the spine of the fixture with the “Toward Wax” arrow facing the opening in the fixture. Place the first plate between the pins, setting it flush against the spacer. Then, center the wax block over the opening in the fixture plate (50 mm closed plates shown, below).

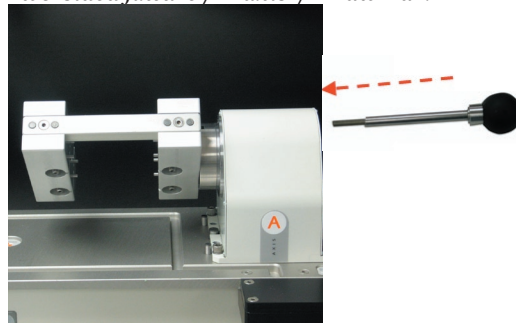


Place the second fixture plate over the wax but under the upper set of pins, pushing it back all the way so that it is flush with the spacer and lines up with the lower fixture plate:



Tighten the threaded screws with the Allen wrench so that the pins move towards one another and cause the plates to hold the wax securely. Remember that it is best to tighten in a “star” pattern, alternating screws diagonally across from one another until the plates are snug. A slight bowing may be noticeable in the plates when they are tight. Finally, remove the spacer before attaching the two-sided auto-flip fixture onto the A axis fixture plate. Then, align the pin in the fixture with the slot on the face plate. Insert the draw bar through the A axis and tighten to secure the fixture to the axis. With the mill in the **Home** position, the fixture plate will be on a steep angle to the A axis. Fear not: this is correct!

Two-sided fixture / A axis / Draw Bar:

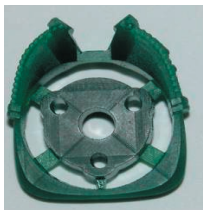


In the **Revo** control software, click on **Load Part 1** and **Start** to start the first part of the job. During this process, the mill will cut the Center Support structure attached to the center of the ring shank. This will be used during Part II of the strategy to attach the ring to the rotary fixture. The mill cuts

the locating holes first, on both sides of the Center Core, beginning with the larger center hole and working outward to the three smaller holes, which are used for timing and location on the mandrel. It might seem odd to watch the mill do this for the first time, but rest assured that it's working correctly.

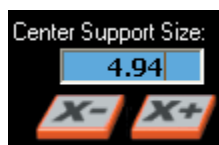
After Part I is finished, remove the ring from the outer block of wax ONLY. Because the inner supports are needed to complete the rotary part of the milling process (Part II), you must leave them intact.

Note: Model below shows an example of custom, user-created supports.



Assembling the fixture for Part 2:

Click on **Load Part 2**. Then, after the second part of the program has finished loading, measure the thickness of the Center Core and input this measurement into the "Center Support Size" field in the **Jog Control** area of the Revo mill interface. This will likely be slightly off of the value that was input in RevoCam for the thickness of the Center Support. Do not worry if this is so. Just input the value from the actual wax model after removing it from the two-sided auto-flip fixture.



To set up the mill for the rotary process (Part II), attach the rotary ring fixture to the A axis. To do so, line up the fixed pin on the fixture with the slot on the A-axis face plate.

Rotary ring fixture: A axis:



Finally, insert the draw bar into the end of the A axis and tighten, securing the fixture snugly to the mill.



Next, assemble the three-sided rotary attachment using the parts pictured below. Start by inserting the mandrel into the rotary ring fixture, making sure that the key on the fixture lines up with the notch on the mandrel stem, and tighten it into place with the 1/8 inch hex wrench.



Notice that, on the Center Support in the ring, there are three small holes that correspond with the three dowel pins on the mandrel. Slide the wax model onto the mandrel by lining up the holes in the Center Support with the dowel pins on the mandrel.

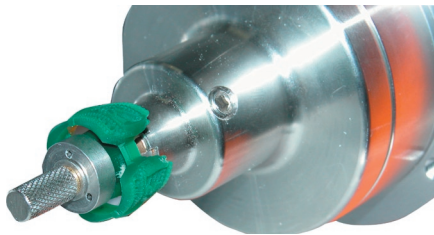
Also make sure that the head of the ring (or, the top of the model) is aligned with the key on the Ring Fixture. We've included a clue to help you do this: notice that there is an arrow pointing upwards towards the head of the ring that has been milled

onto the outer surface of the Center Support. Make sure that the arrow on the Center Support is facing away from the A axis, is visible, and is pointing towards the key on the Ring Fixture.

(Arrow simulated in illustration below, for emphasis)



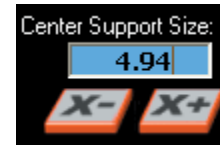
Now, place the mandrel cap over the wax so that the three holes in the cap are aligned with the three dowels on the mandrel. Insert the thumb screw through the center hole in the mandrel cap and use the thumb screw to tighten the cap to the mandrel (as shown below).



Aligning the mill to the rotary fixture:

When transferring an auto-flip plus rotary model from the autoflip fixture to the rotary fixture, it is necessary to align the mill to the center of the model.

After loading Part 2 of the job into the Revo Control software, the “Center Support Size” window and the “X-” and “X+” jog controls will appear in the Jog Control area of the interface. Remember that the value of the center support hub in the model should have already been input into the “Center Support Size” window in the interface:



Note: The following function is only available AFTER loading Part 2 of the Auto-flip plus Rotary job in the Revo Control software on the mill.

This step helps the mill find the true center of the model: click on **Start** to begin **Part 2** of the job in the **Revo Control Software**. However, **ONLY** let the mill make one cut (left to right) on the model, and then press **Stop**!



Now, turn off the **Spindle** and the **Coolant** using the buttons in the **Revo Control Software**. Use the **Jog Control** interface to move the spindle up (in the “plus” direction) on the Z axis, which helps ensure that the cutter is safely out of the way for this step. Now, using a loop and a flashlight to better see the ring’s surface, inspect both sides of the ring (left and right) where the first cut was made to see if the model has cutter marks on either side of the shank (for a flat-sided shank, there should not be cuts on either side of the shank).

Usually, cutter marks will be very faint, which is why the flashlight and loop are necessary. In the following illustration, the right-hand side of the model (the side closest to the fixture) was cut slightly, while the left-hand side of the model was not cut at all. This means that the cutter is slightly off-center, toward the right. Even this tiny distance (about 0.01 mm) can make a big difference to delicate parts such as prongs and lettering.

After running first cut, left to right, stopping mill, and jogging spindle in Z+ direction, inspect the ring closely with loop and flashlight:

Left side: (no cut marks at all)



Right side of ring: (slightly nicked by first cut)... therefore, cutter is off-center, too far to the right:

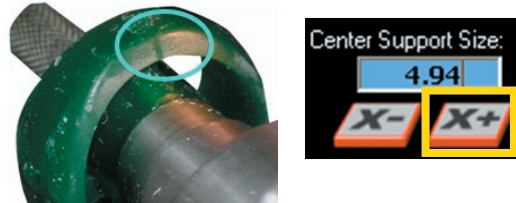


To move the spindle so it is not cutting into either side of the model, select the appropriate Jog Control setting for the distance you need to move the mill, in increments of Fast = 0.1 mm, Medium = 0.05 mm, Slow = 0.01 mm, Step = 0.004 mm.



So, as in this case when the mill is cutting into the right side, as shown above, set the Movement Controls to Slow and click the "X+" button once for every 0.01 mm you want the spindle to move in the X+ direction.

Right side is cut: use X+ to correct.



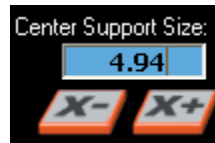
Or, if the mill is cutting into the left side of the piece, as shown below, click on the "X-" button once for every 0.01 mm you want the spindle to move in the X- direction.

Left side is cut: use X- to correct.



To determine how much the spindle should be moved, make an educated guess as to how far the mill is off. Then, click once on either button to move the mill at the set increments in the appropriate direction.

The rule is: if the right side is cut, click the X+ button. If the left side is cut, click the X- button.



Now, after making the proper adjustment, simply turn the Spindle and Coolant controls back on and click on the Start button to resume cutting the part.

This time, let the mill run back and forth through 4 cuts (left to right and back again four times) before stopping it. Again jog the spindle up in the Z direction to clear the cutter out of the way. Turn off the Spindle and the Coolant. Then, using a loop and a flashlight, check again to see that both sides are free of cutter marks. If they are, turn on the Spindle and Coolant controls in the software and then click on Start to finish the job.

If further adjustments are necessary, use the "X-" and "X+" controls to adjust the spindle again. Then, re-start the job and check once more. If it is necessary to adjust this value again, allow the mill to make 8 cuts the next time, then 12, and so on.

Continue checking and adjusting the X+ and X- direction jog controls as necessary until the sides of the ring are free from cutter marks, as shown below:

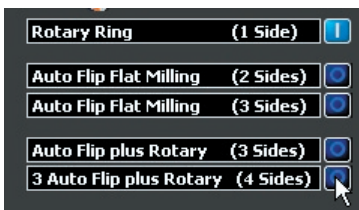


Once the cutter marks are no longer visible, the mill has successfully found the center of the part! Click on the **Spindle** and **Coolant** controls in the software to start them up, and then click on **Start** to complete Part 2.



3-Sided Auto Flip plus Rotary (4 Sides):

The fifth strategy, **3-Sided Auto Flip plus Rotary (4 Sides)**, combines the top-down flat milling power of the **3-Sided Auto Flip** strategy with the full rotary cut possible on the **Auto-Flip Plus Rotary (3 Sides)** strategy.

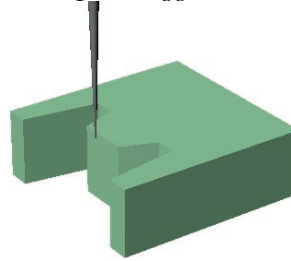


About this Strategy:

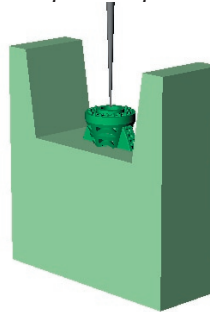
This strategy starts with the ring on the 50-mm open plate fixture. After creating the cut-away in the wax, it cuts the head from the top-down position first; rotates it 90-degrees to cut the

front side; then rotates it 180-degrees to cut the back side. The front-side and back-side cuts also complete the inner hub with supports and drill-holes that will hold the model to the **Auto-Flip plus Rotary** mandrel. After the first **Job** is completed by the mill, remove the model from the wax block and load it onto the 3-pronged mandrel to complete the final rotary pass.

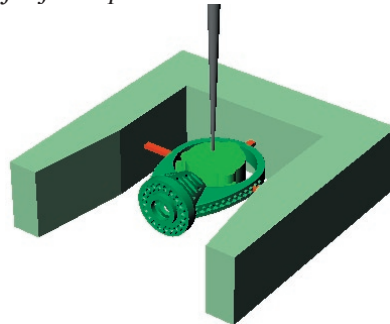
1: Cut-out curve is milled, as in the **Three-Sided Auto Flip** strategy:



2: Top-down pass to complete head:



3: Front- and back- of ring are milled in **Two-sided Flat Milling** operations, together with hub for final operation:

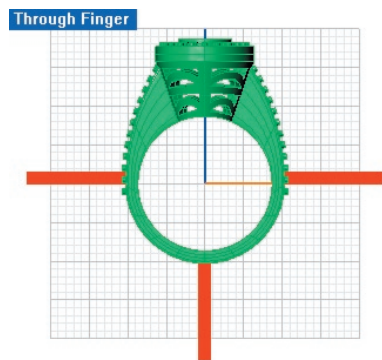


4: Ring is completed on **Rotary** strategy with hub and 3-pronged mandrel holding it to the A axis rotary fixture:



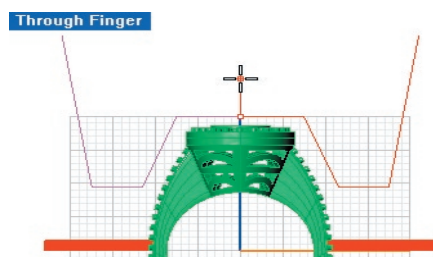
Using the RevoCam Software:

Start by placing standard supports into the model in preparation for the **3-Sided Auto Flip** strategy. Use the **Box** tool to place long, narrow boxes that are about 1 X 2 mm at the 3-, 6-, and 9-o'clock positions on the model in the **Through Finger (Front)** viewport:



It is also necessary to draw the cut-out curve for the standard **3-Sided Auto Flip** strategy. Turn on **Grid Snaps** at the **1.0 mm** setting and start the **Polyline** cut-out curve 6.0 mm above the grid. Head down toward the support, then up again toward the top of the ring, keeping about 1.5 to 2.0 mm away from the model at all times. Stop at the center of the grid and **Mirror** the curve to the other side of the ring. **Join** the two together to make a single cut-out curve:

*Draw curve with **Polyline** tool, **Mirror**, and **Join**:*

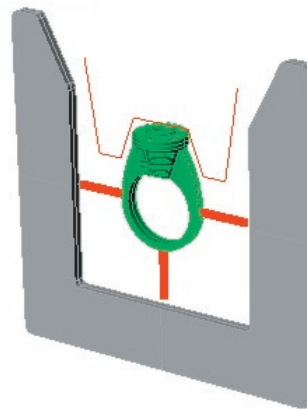
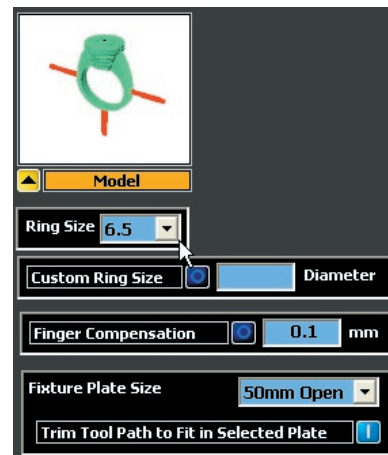


With the three support posts and one cut-out curve in place, you are ready to start up the **RevoCam**

software. Select the final strategy from the first screen of the software and click on **Next**.

3 Auto Flip plus Rotary (4 Sides)

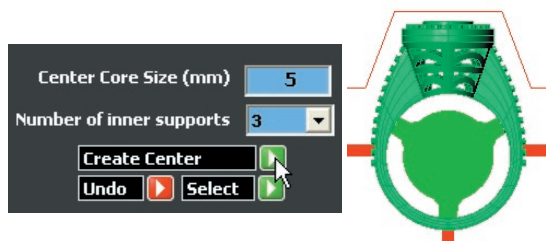
In the second screen of the software, select the entire model and input it into the **Model** preview window. If the ring rail has been entered into the **Models** window, the proper ring size will appear in the **Ring Size** screen. If not, select the ring size (**6.5**, below) from the **Ring Size** drop-down menu (or input a **Custom Ring Size** using the toggle button and **Diameter** box). Then, select the **50 mm Open Fixture Plates**, and select (as always), the **Trim Tool Path** option:



Note: It is acceptable to input the entire model, supports and all, into the **Model** preview window in this or the regular **Auto-Flip Plus Rotary** strategy. In the next screen of the software, the inner support hub is created. When the outer supports are at then entered into the outer supports preview window, they will be separated

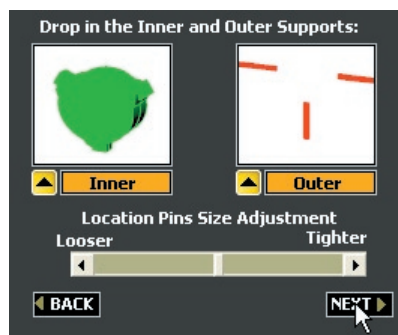
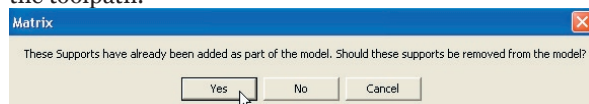
from the model so that the rotary pass will not recognize them during the cut.

In the **Next** screen of the software, create the **Center Core** by indicating the width, in millimeters. This value should be 1 mm wider than the narrowest width of the shank in the **Side View** viewport. Assign inner supports (3 is the usual) and click on the **Create Center** button.

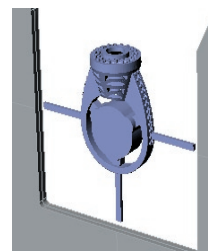


Select the inner supports (the new **User Green** hub) and the outer supports (the three boxes you placed at the 3, 6, and 9-o'clock positions back in step 1) and input them into the proper windows in the software.

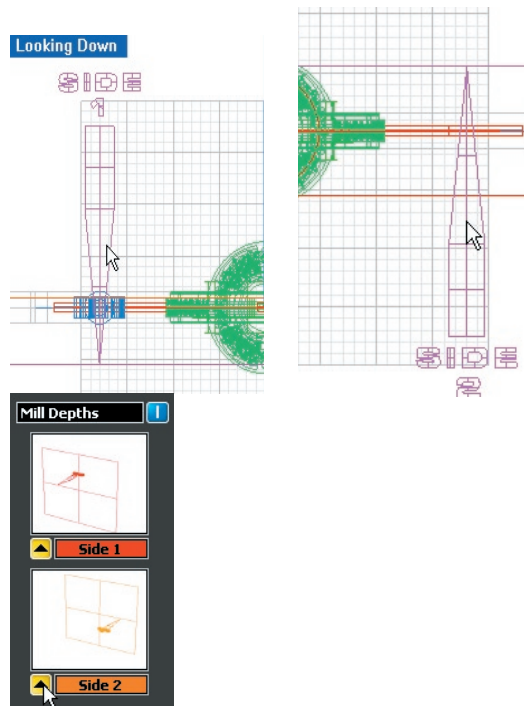
If the outer supports were already added in the **Model** window, the following error message will appear when you enter the outer supports into the outer support preview window. Click **Yes** and the outer supports will not be recognized as part of the model during the rotary pass: they will only be seen as support material during the autoflip portion of the toolpath.



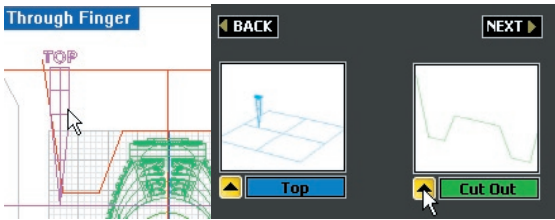
In the next screen of the software, accept the default values and click on **Create STL**. When the STL has been completed, click on **Next** to proceed in the software.



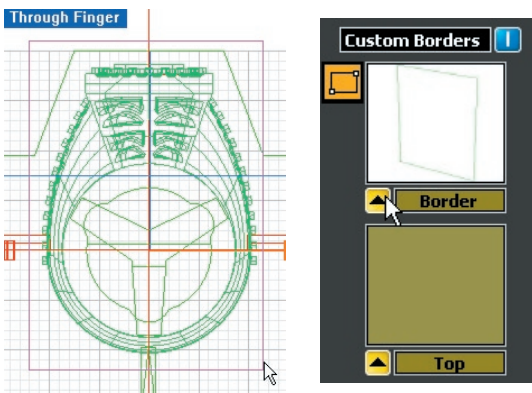
In the Looking Down (Top) viewport, set the Mill Depths for Sides 1 & 2 by selecting them and dragging them to the farthest extents of the model, past center, on their respective sides. Input each into their assigned preview windows: Side 1 into Side 1 preview window and Side 2 into the Side 2 preview window.



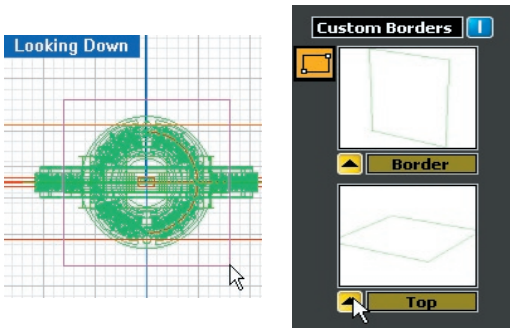
Set the **Top Mill Depth** to about 1 mm below the bottom of the head in the **Through Finger (Front)** viewport and input the **Mill Depth** into the **Top** mill depth preview window. Also, select the **Cut Out** curve and input it into the **Cut Out** preview window, which is found beside the **Top Mill Depth** window:



Select the **Custom Borders** window and draw the first rectangle in the **Through Finger (Front)** viewport about 1.5 to 2.0 mm beyond the extents of the model on all sides (*no* part of the border should extend past the ends of the outer support posts). Input this box into the **Border** preview window.

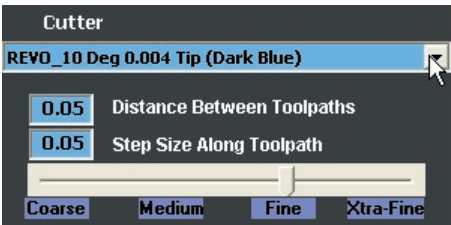
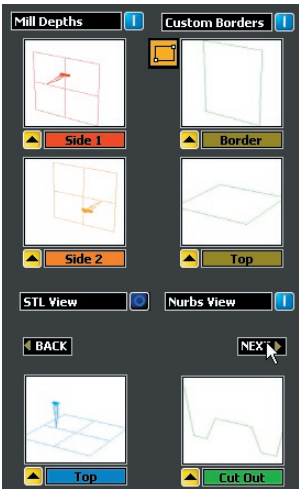


Select the **Rectangle** command again and draw the **Top** border in the **Looking Down (Top)** viewport. This border should only enclose the portion of the head you wish to mill during the first, “top-down” auto-flip pass. There is no need to enclose parts of the shank that will be reached during the rotary pass. Stay about 1.5 to 2.0 mm away from the details to be cut during this portion of the toolpath, as shown below. Input this box into the **Top** custom border preview window:



With all of the preview windows properly filled in, click on **Next** to proceed in the software. Select a cutter and surface finish quality, or accept the

defaults, and click on **Next**:

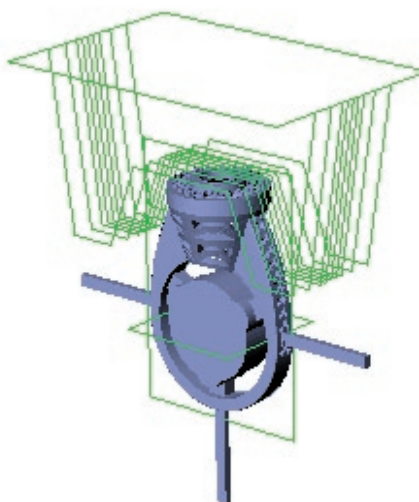


In the final screen of the software, click on **Make Revo File** to generate the toolpath:

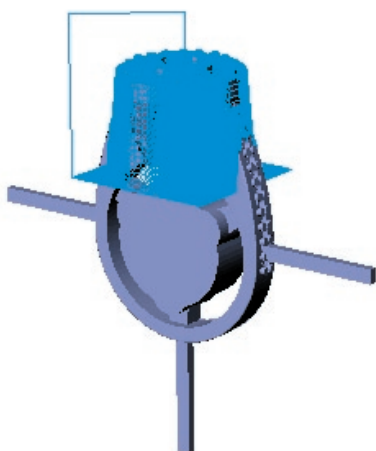


After toolpath creation, inspect the paths to make sure they have been created properly. The paths for this strategy include:

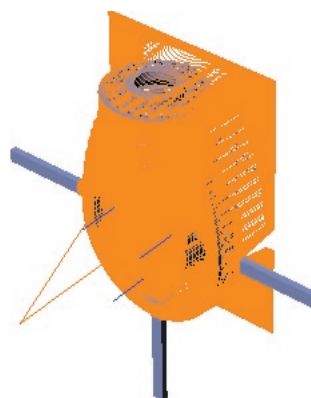
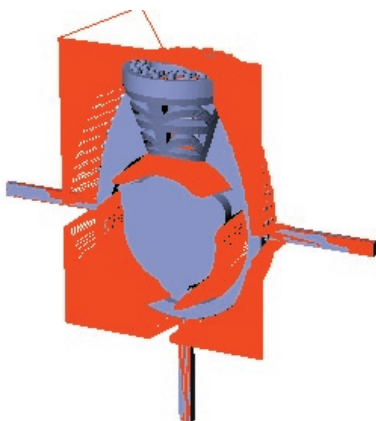
- (1) **Cut-Out:**



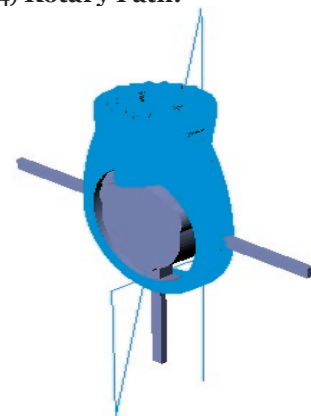
(2) Top-Down Path over Head:



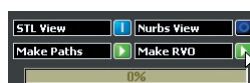
(3) Front side / back side of two-sided auto flip:



(4) Rotary Path:



If your toolpath checks out, as shown above, use the **Make RVO** button to save the **.RVO** file and you're ready to load it onto your mill.



On the **Revo Mill**, set up the first part of the job – the three-sided auto flip portion – the same way you set up the 3-sided auto-flip fixture with the 50-mm open plates. **Load** your 3-sided Auto Flip Plus Rotary Revo File and **Part 1** of the job will automatically load first. It runs the first three toolpaths (shown above): the cut-out, the top-down pass, and the front and back passes.

Then, run part two as though you were running the second part of the auto-flip plus rotary job: attach the ring fixture to the A axis using the draw bar to hold it firmly in place and fasten the auto-flip plus rotary mandrel to the A axis rotary ring fixture. Remove the model from the *outer block of wax ONLY* (leave the hub intact!) and place it onto the

auto-flip plus rotary mandrel. You're ready to load **Part 2** of this job. At that time, you'll need to align the center of the model using the X+ and X- jog controls that appear in the interface directly under the **Center Core Size** window.

Warning: *DO NOT use the standard X+ and X- Jog Controls for this procedure!! ONLY use the X+ and X- jog controls that appear in the interface DIRECTLY UNDER the **Center Core Size** window.*

With the model properly aligned as described in this manual, above, press **Start** to complete the part.

Chapter 11: Using Advanced Options

An Overview:

Advanced Options found within the **RevoCam** software are designed to give advanced users greater control over toolpath creation. Three new milling operations (**Flat on Rotary**, **Follow Curve on Flat**, or **Follow Curve on Rotary**) can be added to an existing toolpath, enabling the cutter to reach hard-to-mill areas and improving the finish and definition of finely-detailed models. **Advanced Options** for new and conventional operations include rotating the A axis to a custom angle, selecting a different cutter for different parts of the job, and indicating several different custom borders for the same model. When used in conjunction with conventional strategies, **Advanced Options** provide cleaner model detail and minimize additional milling time.

Advanced Options may be used with any of the typical six strategies. To engage them, turn **ON** the **Advanced Options** selector in the first screen of the **RevoCam** software. For any of the given strategies, there are no additional steps required when preparing the models in Rhino (although there are optional steps outline in the tutorials that may be taken). However, it is important to be aware that, even though all of the steps performed during a conventional operation are still required when **Advanced Options** are turned on, the sequence of the steps and their layout on-screen will differ.

After inputting the model into **RevoCam** and completing the early steps in the selected strategy, the user will be prompted to create the **STL** file as usual. Then, the **Advanced Setup Options** menu displays. This menu details the existing milling operations for the model. Here, specific settings for each part of the conventional milling operation can be adjusted: for example, the user can select a different cutter for each part of the job as well as include mill depths and borders just like when running a conventional strategy. Then, the user can insert additional milling operations to improve the quality and definition of the cut. Each new operation has its own set of parameters that the user can control, as well. Finally, the user can adjust the order in which each operation will be milled, if so desired.

When inserting additional operations, the user may choose to repeat operations already available in the conventional strategy, as when including a “second pass” over a portion of the model with a new cutter and a different border; or, the user may insert one of three advanced toolpaths. Each of the advanced toolpaths is available depending upon its compatibility with the conventional strategy already in use (e.g., a rotary toolpath cannot be added to a flat-milling job). The three advanced options are **Flat Milling on Rotary**, **Follow Curve on Rotary**, and **Follow Curve on Flat**.

Flat Milling on Rotary: The mill rotates the rotary ring fixture to the angle specified by the user (top, left, right, bottom, or a custom angle specified by the user) and holds it still during milling rather than continuing to rotate it. This provides the model with greater detail on any part specified by the user.

Flat Milling on Rotary

Top:



Custom:

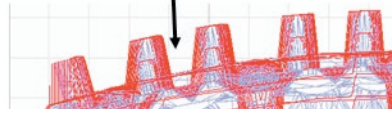


This operation solves a problem that typically occurs when running a rotary toolpath. When prongs, lettering, or other small details are milled on the head of a ring during a rotary operation, the toolpath (shown in red, below) is in fact angled, rather than straight, because the cutter is held at an angle (represented by the arrow in the illustration below, left) as the ring rotates. The resulting model reflects this, and often requires further clean-up prior to casting. The new **Flat Milling on Rotary** operation solves this problem by holding the ring flat during milling. The cutter is always perpendicular to the part, producing a cleaner toolpath. Compare the two toolpaths below.

Traditional Rotary (toolpath in red)

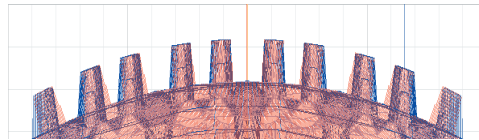


Flat Milling on Rotary (toolpath in red)



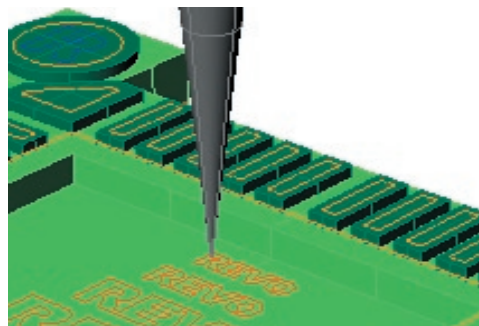
Using both operations together produces prongs, lettering, and other designs that are clean on both sides, rather than angled on one or both sides, thus improving model detail and reducing clean-up time. In the illustration below, both toolpaths are shown together: the original rotary toolpath in red and the **Flat Milling on Rotary** toolpath in blue.

With both Traditional and Flat Milling on Rotary strategies selected:



Follow Curve on Flat: After selecting this operation, the user may input curves that lie on the flat surface of a model and RevoCam will translate the curves directly into a toolpath. This may be used to produce an “engraved” effect or to clean up parts of the model that may have been cut with slightly less definition than desired. The interesting element in this strategy is that there need not be a surface around which the cutter is moving; simply a curve will do. As shown below, the word “REVO” will be engraved into the surface of the part. To produce this effect, the curves representing the shape of the letters were placed onto the surface of the part (they may also be placed just above or just below the surface of the part) and input into the software using this advanced option.

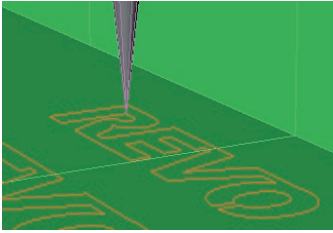
Follow Curve on Flat:



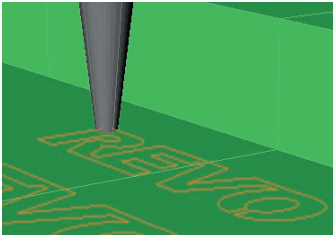
During this strategy, the user indicates the “Depth Below Curve”, or, the depth below the path curve

that the cutter will sink to when cutting the curves. The width of the resulting cuts depends upon the width of the selected cutter and the depth to which the cutter is plunged below the curves on the surface of the model. When a tapered cutter is sunk rather deeply into the surface, the width of the cut is determined by a thicker part of the taper, as illustrated in the two examples below:

When using a tapered cutter, a shallow depth below curve produces a narrower cut:

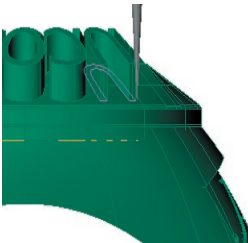


... and a deeper depth below curve produces a wider cut:

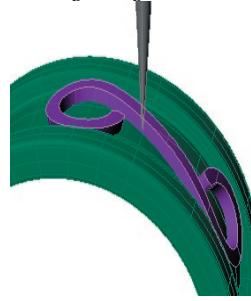


Follow Curve on Rotary: This strategy employs the same concept as **Follow Curve On Flat**, but it is run on the rotary fixture. If the curves are on the flat surface of a rotary ring, the user may direct the mill to hold the rotary fixture flat during cutting; or, if the curves wind around the shank of a rotary ring, the user may direct the mill to turn the ring during cutting.

Follow Curve on Rotary: Flat: ring is held still on A axis



Rotary: ring continues to turn on A axis.



In addition to the familiar processes of adding custom borders, adjusting mill depths, and specifying toolpath distance and step size, further options are available for use with advanced or conventional toolpaths. Among these are options to:

Specify different cutters for different toolpaths:

The mill will pause during the job to allow the user to change cutters when a new cutter is specified for the next toolpath.

Adjust the quality of milling for different toolpaths:

This option allows the user to indicate a coarse quality during parts of a job and a fine quality during other parts. For example, in the case of a finely-detailed head and a plain shank, or the detailed front of a pendant with a plain back, each operation may be milled with differing levels of coarseness to cut down on milling time.

Specify fixture angle: Top, Bottom, or Custom:

The mill can now rotate either the open two-sided fixture plate or the rotary fixture to a custom angle and run a flat milling operation at that angle. This way, details that were previously difficult to reach or had significant “stepping” because of the angle of the cutter can be reached with ease for improved detail.

Specify Fixture Angle: Custom

3-Sided auto flip: Use with 50 mm open fixture plate ONLY



Flat on Rotary: During 1- or 3-sided rotary strategies, ring is held flat on a custom angle during milling



Change Toolpath Type: The user can choose from among five new paths for the cutter, including a circular spiral pattern and a contour pattern that cuts only the inside and outside of the object. When used with a flat cutter, the **Contour** option smooths out raster lines to achieve a flat finish.

Complete a job with a Contour using a Flat Cutter:



How to Use Advanced Options:

To use advanced options, prepare the model for the appropriate strategy as usual in **Rhino** and enter the **RevoCam** software. Select the **Advanced Options** indicator. Then, select the strategy that is appropriate for the model being milled.



After activating Advanced Options, choose the standard strategy typical for your model. Complete the early steps of the software for the strategy selected just as before (input the model, create the STL, etc.).

After creating the **STL**, the **Advanced Setup Options** screen will appear. The different parts of this screen are labeled below:



(A) The milling operations to be run on the model are listed at the top of the screen. Upon first entering this screen, the list displays the typical operations associated with the selected strategy. To adjust the options for one of the operations in the list, click on it, which turns the black selection arrow to white. The options for the selected strategy appear below the list in the software screen.

(B) Indicator lights for each milling operation display when satisfactory **Pre-Flight** and **Toolpath** requirements have been met (**Passed** or **Failed** will display).

(C) Menu controls (**Insert**, **Delete**, **Up**, **Down**) enable the user to add, erase, and change the order of milling operations;

(D) View controls switch the on-screen view from the **STL** to the **Nurbs** model.

(E) Once all of the operations display **Passed** for their **Pre-Flight Parameters**, click on **Make Paths** to create the toolpath. Once the toolpath is complete, the **Make RVO** button will appear. Click this button to save the **RVO** file in the computer in preparation for milling.

Adjusting Settings for Conventional Paths:

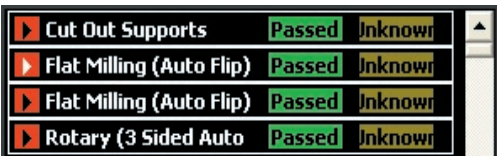
The **Advanced Setup Options** screen starts with a list of all operations that will be run on the model. For a simple rotary one-sided strategy, only the rotary one-sided operation is listed. For a more complex strategy, like the three-sided rotary strategy, all of the operations performed by the software are listed separately: the software first cuts out the supports, then runs two auto-flip

operations, then runs the rotary operation. Each of the operations run in each strategy are listed below:

Rotary Ring (1 Side) runs one milling operation called “Rotary (Single sided)”. This is the operation that is run during the conventional rotary 1-sided ring strategy.



Auto Flip plus Rotary (3 Sides) runs four separate operations: the first one cuts out the supports; the second and third consist of two “Flat Milling (Auto Flip)” operations (one on each side of the ring); and the fourth consists of the “Rotary (3 Sided Auto)” operation that completes the outside of the shank.



Auto Flip Flat Milling (2 Sides) runs two “Flat Milling 2 Sided” operations – one on each side of the model.



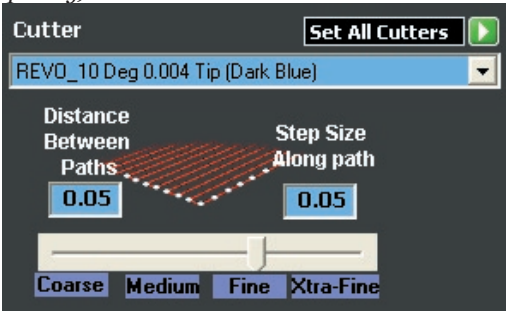
Auto Flip Flat Milling (3 Sides) begins with a two-sided auto-flip operation that cuts out the edge curve in order to better reach the head (or other) details; the second operation flips the ring 90 degrees so that the head (or other) details can be milled, and the third and fourth operations comprise the conventional auto flip “Flat Milling” operations.



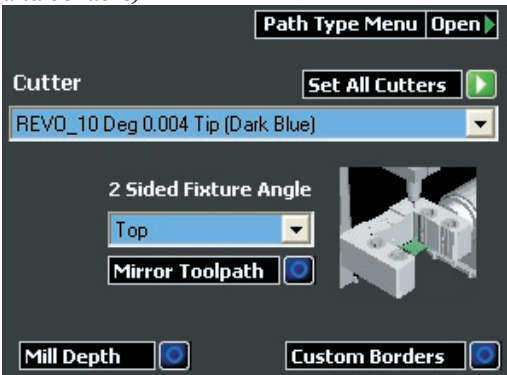
The best thing to do upon first arriving at the **Advanced Options** screen is to select each of the available items in the list, one-by-one, and check that the options for each one are appropriately configured. Select a list item by clicking on it. The arrow beside it will light up and the options associated with that operation will appear below

the list on the software screen.

Options for a rotary operation: (Set cutter and cut quality)

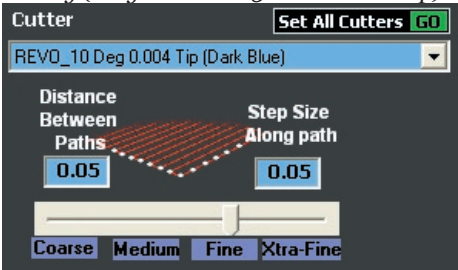


Options for a flat operation: (Set path type & cut quality, cutter, two-sided fixture angle, mill depth, and borders)



For all rotary (single-sided and three-sided) operations, the user is offered the option to select a new cutter and specify milling quality. When choosing cutters, remember to consider whether the current operation is cutting from rough or is running a second pass. Also note that, if a new cutter is selected for different parts of a strategy, the mill will pause between operations to allow the user to change the cutter.

Rotary (Single Sided & 3 Sided Auto Flip):



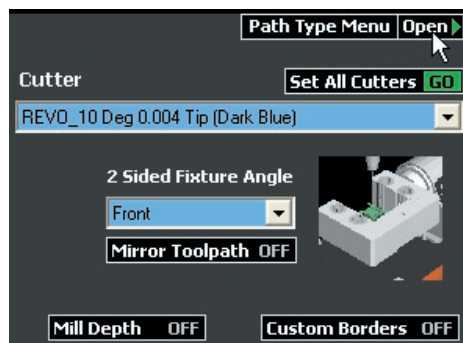
One element that appears as an option for every milling operation is the **Cutter** drop-down menu and the **Set All Cutters GO** command. As stated

earlier, a different cutter may be used for different milling operations. This is set by pulling down the **Cutter** drop-down menu and highlighting the desired cutter. The selected cutter will appear at the top of the list box and the other options will roll up again.

If, however, the user needs just one cutter for every operation, it is easy to set this up by selecting the desired cutter, as above, and clicking on **Set All Cutters GO**. Once the desired cutter has been set with this button, even future operations that are inserted into the menu will reflect this choice. For Flat Milling operations, the user can select a new cutter, change the path type (how the cutter will move during the operation), specify the angle at which the fixture is held, set the mill depth, and draw a custom border. Set the mill depths and custom borders just as described in the section of this manual pertaining to the conventional use of the strategy.

Note: When specifying settings for conventional operations, the angles should not be changed! This ensures that the piece is properly milled. When adding additional, advanced operations, it's fine to change the fixture angle; however, missing a "front" or a "back" operation on the two-sided flat fixture could be a grievous error, so it is best to leave these settings alone when setting up the conventional strategies necessary to properly cut the job.

Flat Milling: Two or Three-sided options:

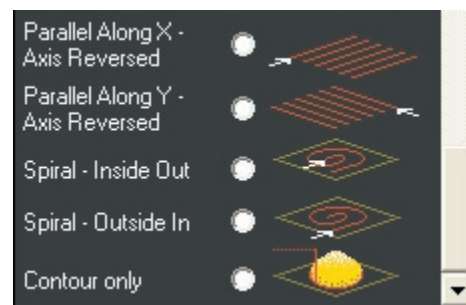
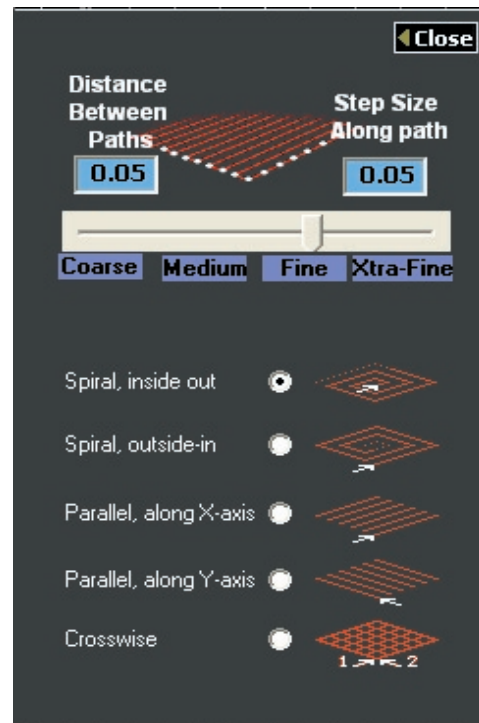


About the Path Type Menu:

Click on **Path Type Menu Open**, an option only available when using a flat milling strategy, to adjust the toolpath type if so desired. The **Path Type Menu** allows the user to adjust the finish (**Coarse** through **Xtra-Fine**), using the slider bar to set the default values for **Distance Between**

Paths and **Step Size Along Path**; or, the user may type in custom values.

Click on one of the tick boxes to assign a **Path Type** for the area being cut. The first five options displayed in this menu reflect the familiar milling patterns available to users with **DeskProto 3.1** (pictured on the left). The next five, shown on the right, below, are available to users with the new **DeskProto 4.0**. RevoCam automatically detects which version of the program is being run, and displays the menu options accordingly.

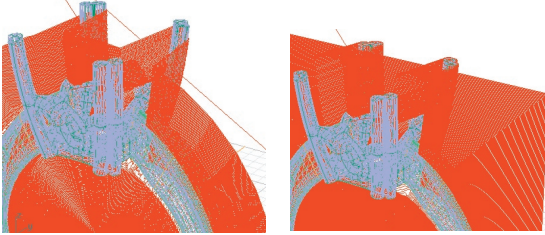


The new options are:

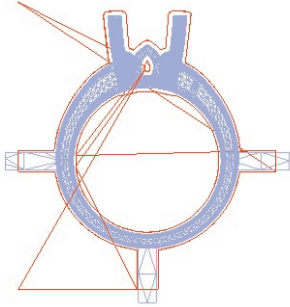
- (1) Parallel Along X-Axis Reversed: reverses the direction of Parallel along X-Axis;
- (2) Parallel Along Y-Axis Reversed: reverses the direction of Parallel along Y-Axis;
- (3) Spiral – Inside Out: runs the Spiral, Inside Out

path in a circular, rather than a block, pattern;
 (4) Spiral – Outside In: runs the Spiral, Outside In path in a circular, rather than a block, pattern.

Spiral vs. Conventional Block Pattern:



(5) Contour Only: runs one path that proceeds around the outer and inner edges of the model only. Switching to the **Flat** cutter and selecting this option for a standard 2-sided auto flip operation is a quick and easy way to smooth out the raster lines in a model at the very end of a mill job. Just be certain to add this toolpath as the FINAL toolpath on a job! Trying to run a **Contour** path type with a fine cutter cutting from the rough will likely cause the cutter to break.



Setting the **Mill Depths** and **Custom Borders** for flat milling operations in **Advanced Options** follows much the same procedure as setting them for a conventional operation. Turn on the **Mill Depths** indicator light. Click on the **Mill Depth** icon and the mill depth for only this side of the model (front or back) will appear. Set the depth to the desired setting and input it into the **Mill Depth** preview window. Turn on the **Custom Border** indicator light and draw a custom border using the **Rectangle** tool. Input the custom border. Then, select the next two-sided flat milling operation and repeat. It is best to use the same custom border when setting up the conventional portions of the mill job.

Note: When inputting a custom border and mill depth for the default Back and Front sides of the two-sided auto-flip operation, it is likely that the custom border will be the same for both sides of the model. The best way to handle this is to input the

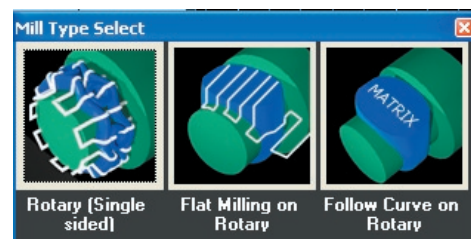
same custom border twice into the two separate operations: once into the “Back” operation and once into the “Front” operation. Although it is tempting to use the Mirror Toolpath button, this may actually confuse the issue, because Mirror Toolpath ON creates two toolpaths that are identical (back and front). The default list contains both “Front” and “Back” operations. If one of these was set to Mirror Toolpath ON, the other one would be redundant.

Adding more toolpaths:

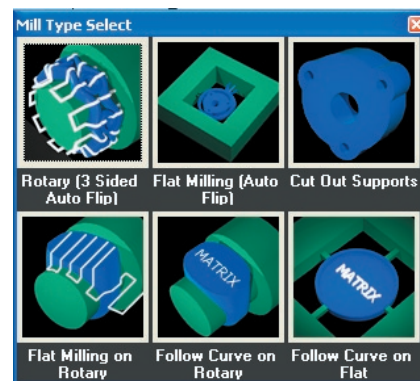
To add a new or conventional operation to the list, click on **Insert** and select a milling operation from the **Mill Type Select** menu that appears. The operations found in the **Mill Type Select** menu consist of all of the conventional and advanced operations available for use with the selected strategy and so, necessarily, differ by strategy:

Mill Type Select menu options available for each conventional strategy:

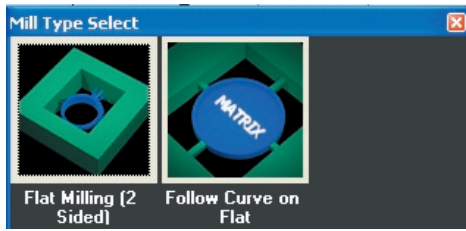
Rotary Ring (1 Side): Standard Operation – first; New Operations – last two



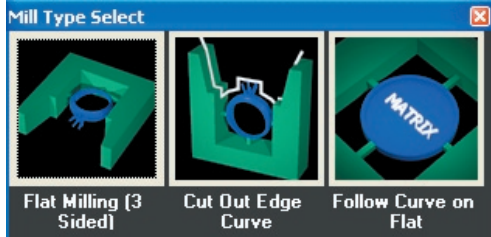
Auto Flip plus Rotary (3 Sides): Standard Operations – first three; New Operations – last three:



Auto Flip Flat Milling (2 Sides): Standard Operation – first; New Operation – second



Auto Flip Flat Milling (3 Sides): Standard Operations – first two; New Operations – last one



To repeat a conventional milling operation with advanced options:

Notice in the illustrations above that the first one, two, or three menu items are the same as the standard milling operations already listed in the menu for that strategy. Select one of these in order to run an additional conventional milling operation. Then, adjust the settings for that strategy in the options menu that appears below.

For example, run a “second pass” on a rotary or flat-milling operation by adding the identical strategy (“Flat Milling 2-Sided” or “Rotary Single-Sided”), this time selecting a finer quality setting and a smaller cutter with which to run the job.

For a flat milling operation, add definition to just one part of the model by changing the custom border to a smaller border: that way, just a small portion of the model will be cut during the second pass. Each time the cutter is changed, the mill will pause in the job and wait for the user to change the cutter. However, all of the toolpaths are still saved as one job, so that only this single job needs to be loaded into the **Revo Control** software.

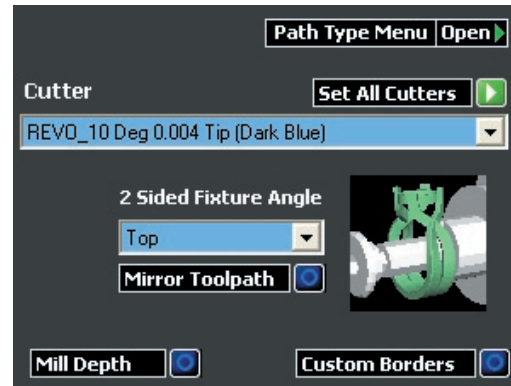
To add an advanced milling operation:

Each advanced milling operation that is compatible with the given part offers its own set of options that the user can adjust in the software screen.

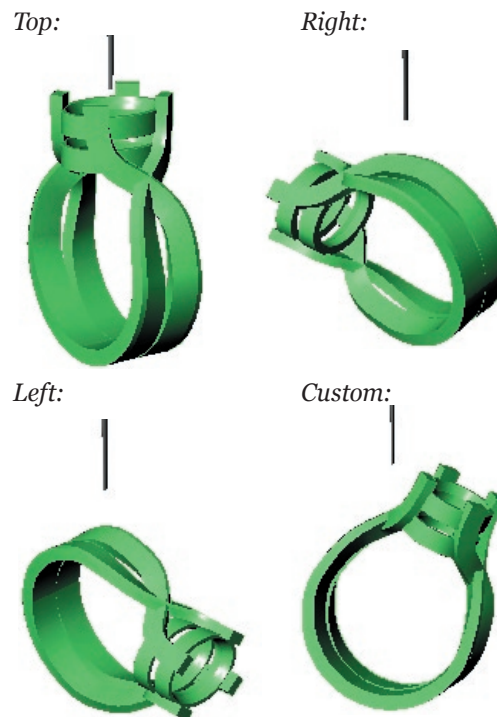
Flat on Rotary:

Adjust the path type, cutter, and fixture angle for

this strategy. Also, set a mill depth and custom border.



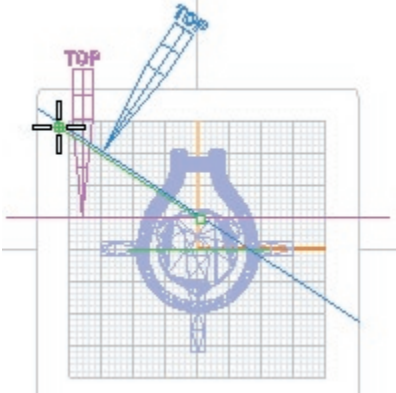
The fixture angle directs the the A axis how to hold the ring while the cutter is working. The part can be held still at the **Top**, **Right**, or **Left** angles while milling this strategy. Or, the user can set a **Custom** angle at which the mill will hold the part still.



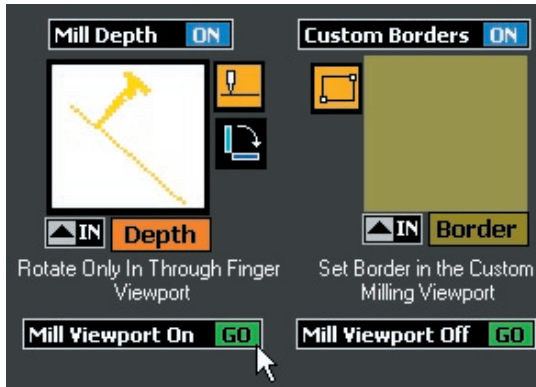
When electing to run the operation at a custom angle, the **Mill Depth** and **Custom Border** controls will work slightly differently. When the **Mill Depth** appears on screen, select it and click on **Rotate** (the option to **Rotate** the mill depth is not offered with any other fixture-angle selection: only “Custom”). Rotate the mill depth to the angle

at which the hard-to-reach details on the model can best be milled. Then, input the mill depth into the preview window and turn on the **Custom Borders** option. When the **Custom Borders** option is activated, the options **Mill Viewport On GO** and **Mill Viewport Off GO** will appear, allowing the user to open up a new viewport – the **Milling View** viewport – which appears on top of the other four viewports.

Set Mill Depth by rotating cutter:



Input mill depth and turn ON Custom Borders:

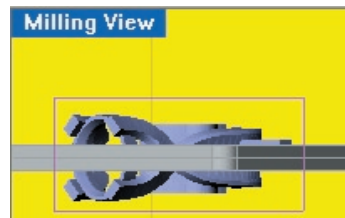


The **Milling View** viewport is set up with a nonstandard axis that changes based upon the angle to which the **Mill Depth** was rotated in the above operation. The axis is, in effect, projected onto the cutting plane so that the cutting plane is planar in this viewport. (This also shows a top-down view of the ring at the angle at which it will be held by the fixture during the flat milling operation.)

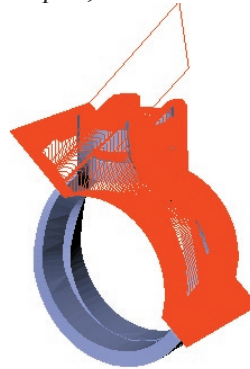
Turn on the **Mill Viewport** before selecting the rectangle curve tool to draw the custom border. Then, select the rectangle tool and be sure to draw the border in the Milling View viewport so that

it will be recognized properly by the software. After the custom border is drawn, input it into the **Border** preview window and click on **Mill Viewport Off GO** to restore the familiar four-viewport screen.

Turn ON Milling View viewport and draw Custom Border:

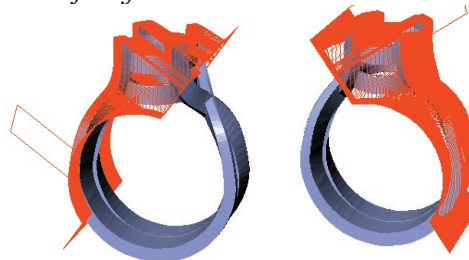


Resulting toolpath: (it will be planar in this viewport):



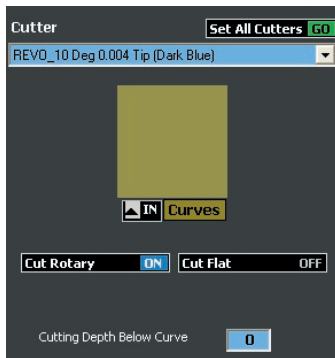
To automatically **Mirror** the specified toolpath to the other side of the ring, with the identical milling depths and custom borders set in the software (e.g., with **Right** fixture angle selected, toolpath will be mirrored to identical location on **Left** side of ring; or, with **Custom** select, toolpath will be mirrored to identical location on opposite side of ring), select the **Mirror** option beneath the **Two-sided** fixture angle:

With Mirror Toolpath selected, toolpath indicated for "custom" fixture position is applied to both sides of ring:

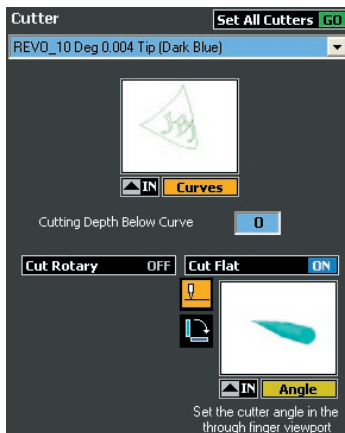


Follow Curve on Rotary:

This strategy provides the option to select a cutter, input the curves that will be followed during the toolpath, set a depth to which the cutter will be plunged beneath the curves during the cut, and select whether the curves will be cut “rotary” (where the fixture is rotated during the cut) or “flat” (where the fixture is held still on an angle during the cut). In order to input curves into the software, the user must switch to the **Nurbs** view of the model, select the curves, and click on the **IN** arrow to place them into the software.



If the ring will be held flat on the rotary fixture during this operation, the **Angle** preview window will appear.



To specify the angle, click on the **Angle Indicator** icon and the angle indicator will appear in the viewports. To indicate the angle at which the ring should be held flat, select the angle indicator and move it to the correct position on the ring using **Rotate**, if necessary.

Note: This is NOT a traditional Mill Depth, so the depth at which the angle indicator is placed is not important: just the angle.

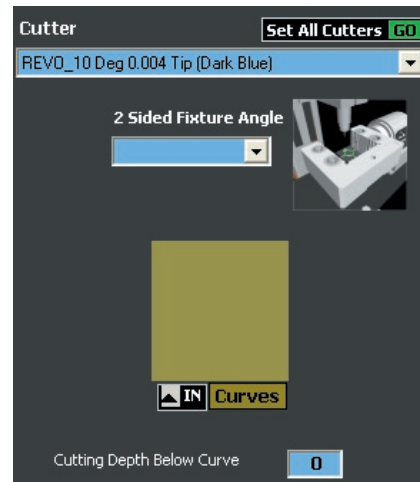


Input the angle indicator into the **Angle** preview window (shown above).

If **Cut Rotary** is **ON**, there is no option (and no need) to set the angle, since, when this option is selected, the fixture will continue turning the ring to cut out curves that proceed all the way around the outside of the shank.

Follow Curve on Flat:

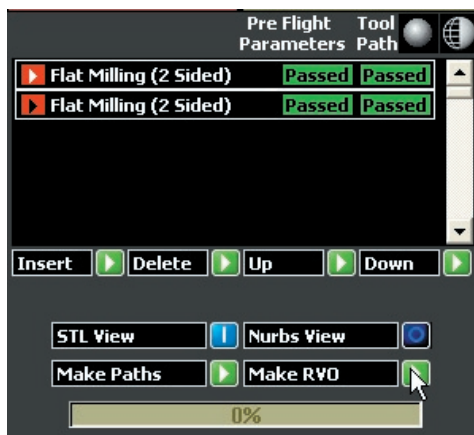
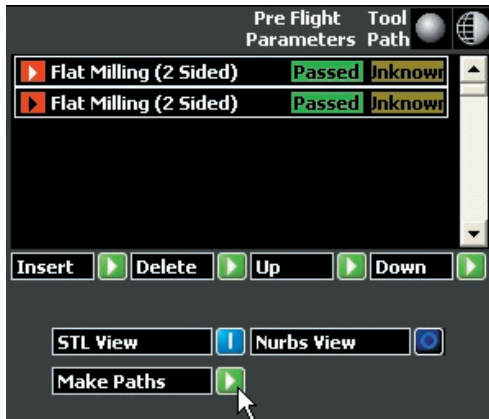
This advanced operation allows the user to input a cutter, select curves to input for the follow curve operation, and specify the depth below the curve that the cutter will be plunged during the cut. In addition, the user selects which angle (Front or Back of model) the fixture will be held during this operation. In order to correctly enter the curves, remember to select the **Nurbs** view of the model first and then select the curves and input them into the **Curves** preview window.



Making and Saving the RVO file:

Once all of the advanced options and additional toolpaths have been specified, click on the **Make Toolpaths** button. Each strategy in the list must display the yellow **PASSED** indicator beneath its Pre-Flight Parameters before this operation will be successful!! As each toolpath is generated, the **PASSED** indicator will appear beneath the **Toolpaths** option and, once all have passed, the **Make RVO File** button will appear. Click on this

button to save the revo file to the desired location on the computer. It is now ready to be opened in the **Revo Control Software**. As when using a conventional milling strategy, the file has been saved when the progress bar reaches “100%”.

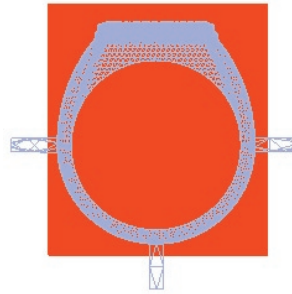


Custom Borders & Trimmed Toolpaths:

Use the following advanced options settings within any of the applicable strategies: with any standard or advanced flat milling operation, use **Freeform Custom Borders**. With any advanced rotary operation, use **Custom Borders** and **Trim Toolpath**.

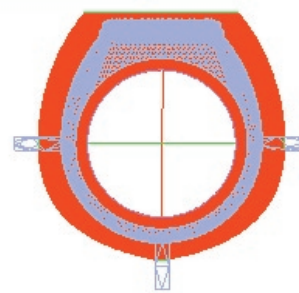
All three of these three new advanced options strategies reduce milling time significantly by minimizing the area where the mill will cut. Traditionally, the borders around the parts have been rectangular or square, meaning that the tool could be cutting a lot of empty space out around the part depending upon the model's shape. This was a disadvantage when using the auto-flip and rotary strategies.

Before: Flat Milling Path



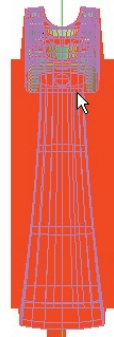
After: with **Freeform Borders**

Through Finger



Note: When freeform custom borders are used on a **Flat Milling** path, the **Path Type** will default to the **Parallel along the X-axis** path. This is the only setting available in the **Advanced Options** portion of the software with which the **Freeform Custom Borders** option will work. This means that, when **Freeform Custom Borders** are entered into the software, it will automatically **OVERWRITE** any of the user's settings in the **Path Type** section of the **Advanced Options** screen with this **Parallel along the X-axis** path.

Before: Rotary Path
(Side View (Right))
Toolpath



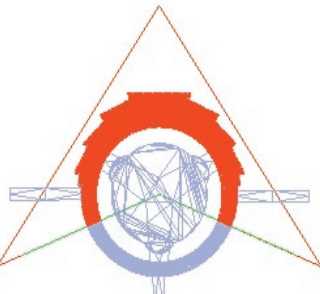
After: with
Trim



Before: Rotary Pass (Through Finger (Front view))



After: with Custom Borders



Using Freeform Borders on a Flat Milling Pass:

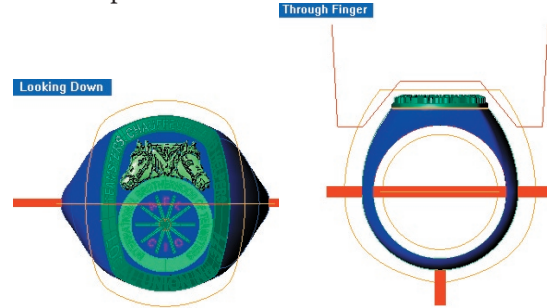
When using the standard strategies or the **Advanced Options** for any job that includes an auto-flip operation, it is now possible to enter freeform borders into the **Custom Borders** window to describe the area to be cut. This minimizes cut time by reducing the area the mill is required to cut and eliminating the cutting of empty space.

The freeform borders must be drawn *first* in **Rhino** before the user enters the **RevoCam Software: Advanced Options**. Use closed, planar curves and **Join** together any curves that may be separate. It's fine to enter more than one; you can even save additional time by ascribing an area *inside the finger hole* to leave untouched by the cutter. (This is only advisable when running the 2-sided or 3-sided auto flip strategies. The auto-flip plus rotary strategy and the new three-sided auto-flip plus rotary strategy both require the use of the central hub, so this is NOT recommended when running either of these two strategies.)

Note: It is possible to use Freeform Borders in the standard strategies, but they must consist of only one closed curve, and the border window will only receive one curve; unlike the advanced strategies, which will accept multiple freeform borders.

For the following three-sided auto-flip job, three separate freeform borders will be used: one in the

Top (Looking Down (Top)) view of the model to ascribe the area for the top-down pass; the next two in the **Through Finger (Front)** view of the model (shown below, right). Together, these two lines tell the mill where to cut: *only* between them both. The area inside the finger hole of the ring will not be touched by the cutter; it will therefore “fall away” from the piece after it is cut.

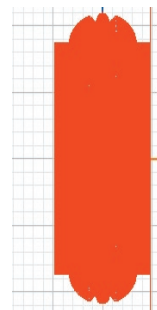


Note: Take special care when placing this additional curve inside the finger rail: if advanced milling passes will be run after this piece falls away, there is a slight chance the fallen piece may wedge between the auto-flip fixture and the mill's tabletop. Either remove the cut-out piece right after it falls out, place a support (shown above, right) that will hold the cut-out piece to the ring, or do not use the inner border when additional advanced toolpaths will be run subsequent to this one.

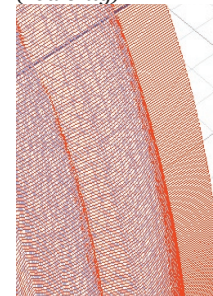
Using Trim Toolpath & Custom Borders on a Rotary Pass:

The **Trim Toolpath** option controls the “toolpath overcut” setting on the **Rotary** part of an **Auto-Flip Plus Rotary** strategy. Usually, this value defaults to 1.5 mm on either side of the ring, which means that an area of 1.5 mm of wax is left on either side of the ring between the ring. This space is represented below:

Silhouette of ring & Toolpath Overcut:



Edge of ring (silver & red) / Toolpath Overcut: (red only)



However, when milling a tapered ring, all this space is traditionally cut on either side of the *widest part* of the ring. Any parts of the ring that are narrower than this are left with *more* than a 1.5 mm overcut.

To eliminate the extra time that the mill spends cutting this “empty air”, it is possible to now fit the overcut to the contour of the ring as seen in the **Side View (Right)** viewport. This is accomplished using the **Advanced Options** settings “Trim Toolpath” during the rotary portion of an auto-flip plus rotary strategy. In addition, it is now possible to indicate a custom value for the overcut: instead of the default 1.5 mm, it is possible to indicate a new value, such as 0.5 mm, to save the most time possible:

Before **Trim** (ring selected)



After **Trim**, 0.5 mm Toolpath Overcut

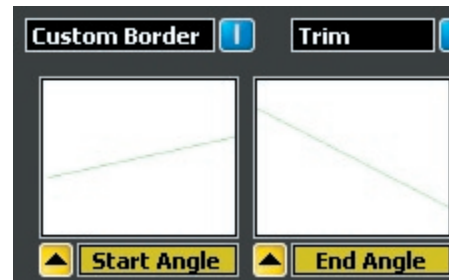
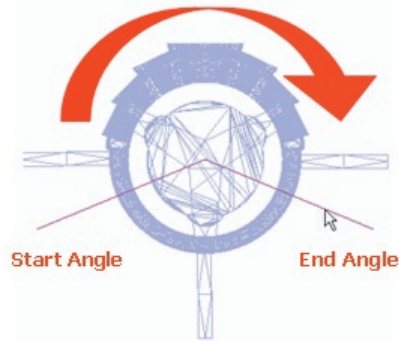


Warning: Although the **Trim Toolpath** option is available during the **Advanced Options 1-sided rotary** strategy, it is **NOT RECOMMENDED!!** This is because the toolpath overcut allows you to cut the ring away from the wax tube used during the 1-sided rotary strategy. Without leaving the standard toolpath overcut of 1.5 mm, it may become very challenging to cut the part away from the wax.

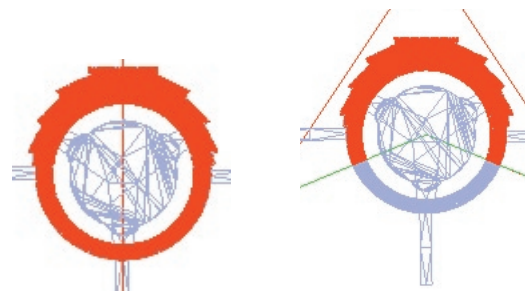
Yet you can save *even more* time when milling this ring if you use the **Custom Border** option now available on a rotary pass. This option controls the amount of the ring that is milled during the rotary operation: only the area between two angled lines ascribed in the **Through Finger (Front)** viewport will be milled. That way, if there are areas on one part of the shank that *don't* need to be run on the Rotary pass, these can be skipped in order to save time.

In this example, the **Flat Milling** passes will sufficiently complete the bottom of the shank, since there are no other details down there that need to be finished with a rotary pass. However, the sides and top of the shank *do* need the rotary pass in order to be milled correctly. Therefore, we'll use

Custom Border angles to ascribe a smaller area to be milled during the rotary pass. Set this angle starting in the *clockwise* direction, so that anything between the two angled lines will be milled at the settings indicated, with the first angled line as the *start angle* and the second angled line as the *end angle*.



Before Custom Border: After Custom Border:



Warning: As above, this **Custom Border** option **SHOULD NOT BE USED** during the **Advanced Options 1-Sided Rotary** strategy, even though it is available to the user. If the entire ring (360-degrees) is not milled during this strategy, which uses the wax tube, it will be very challenging to successfully cut it from the wax tube!!

Chapter 12: Tutorials Using Advanced Options

Based on the desired effect, the existing strategy-in-use, and the design of the model, many combinations of new operations and options can be used to enhance conventional milling strategies.

The following examples show how to use **Advanced Options** to achieve:

- a clean finish on intricate designs such as raised and sunken text;
- smoothness and definition in formerly hard-to-reach areas;
- a detailed finish on one side of a model and a coarse finish on the other;
- a flat finish on the inside and outside of a ring, erasing the raster lines in a fraction of the time;
- engraving on the inside bottom of a ring shank.

I. Rotary Ring (1 Side): Enhanced Details on a Signet Ring

1 In Rhino, prepare the model (below) for the Follow Curve on Flat and Follow Curve on Rotary operations.

Use the **Dupe Edge** tool, found in the **Curve** fly-out menu, to create curves that surround each of the edges on hard-to-mill details, such as sunken lettering.



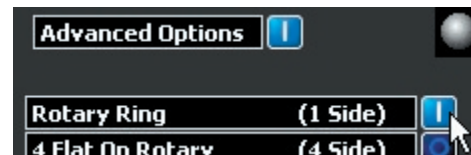
The curves may be placed on the top or bottom of the lettering: in the illustration below, the curves were placed at the top of the lettering, and so a **Cutting Depth Below Curve** will need to be specified.

Curves at top of letters: cutting depth below curve = .75 mm



2 Start the RevoCam software.

Click on icon on your desktop to start the **RevoCam** software and turn **ON** the **Rotary Ring (1 Side)** strategy. Select **Advanced Options ON** and click on **NEXT** to proceed in the software.

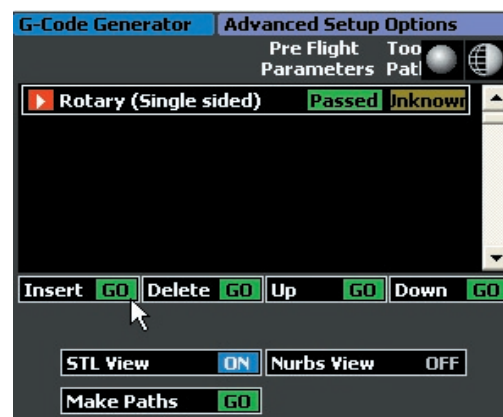


3 Input the model, as usual, and create the STL.

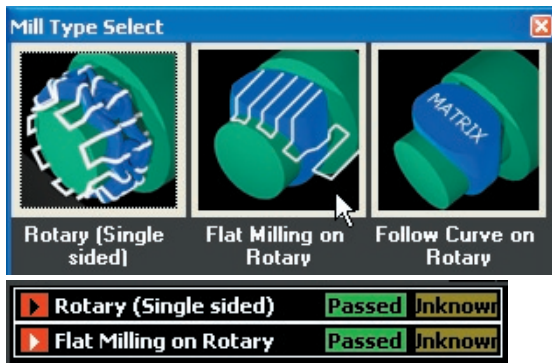
The steps in the **RevoCam** software are the same as those for each conventional strategy until after the **STL** file is created.

4 At the Advanced Options screen, click on Insert GO to add the first new milling operation to the existing list: Flat Milling on Rotary.

The default list contains the **Rotary (Single sided)** operation. To add a new operation, click on **Insert GO**.

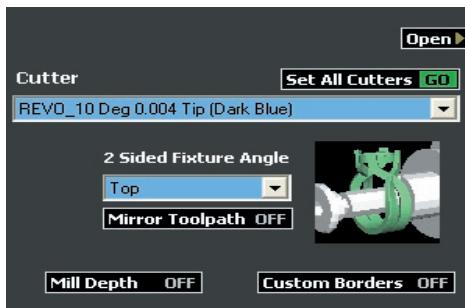


When the **Mill Type Select** menu appears (below), click on **Flat Milling on Rotary** and it will appear in the list



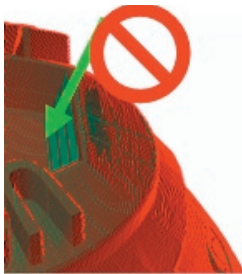
5 Set up the options for the Flat Milling on Rotary operation to mill the top of the signet ring.

Select the desired cutter and make sure that the **2 Sided Fixture Angle** is set to **Top**. (Note: The **Top** mill depth is blue in the viewports.)

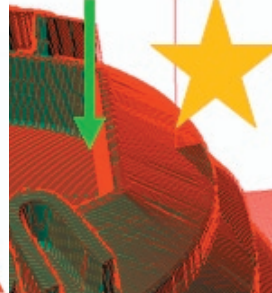


This operation will hold the ring still on the rotary fixture and re-mill the top of the ring. Notice the benefit this is over the former way (below, left). The angles at the top of the model are finished straight, clean, and true to the original model (below, right):

(L) Rotary 1-sided:



(R) Flat Milling on Rotary:



Keeping the **Mill Depth** and **Custom Borders** OFF leaves them at their default values:

- **Mill Depth:** 8 mm below the top surface of the ring.

- **Custom Borders:** 0.5 mm past the extents of the model.

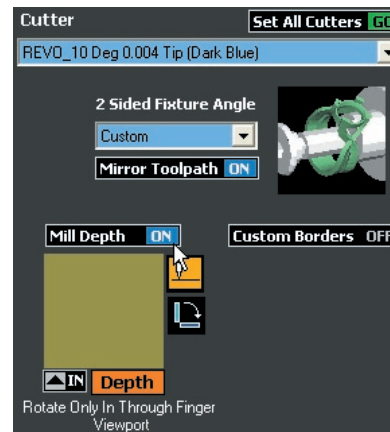
6 Click on **Insert GO** and select **Flat Milling on Rotary** once more.

Rotate the Mill Depth to mill both sides of the signet ring:

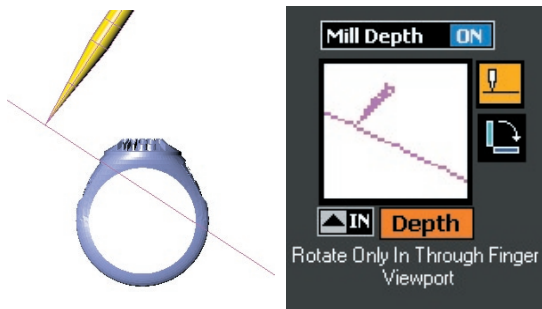


Select the desired cutter and change the **2-Sided Fixture Angle** to **Custom** by pulling down the drop-down list and clicking on **Custom**.

Turn **Mirror Toolpath** ON so that both sides of the signet ring will be milled with these options. Then, turn **Mill Depth** ON to see the **Depth** preview window.



Click on the **Mill Depth** icon to set the cutter and cutting plane in the viewports. Select the mill depth and click on **Rotate** to adjust it so that it is aligned with the cut-away on one side of the ring. Read the reminder below the preview window, and only rotate the mill depth in the **Through Finger** viewport!!



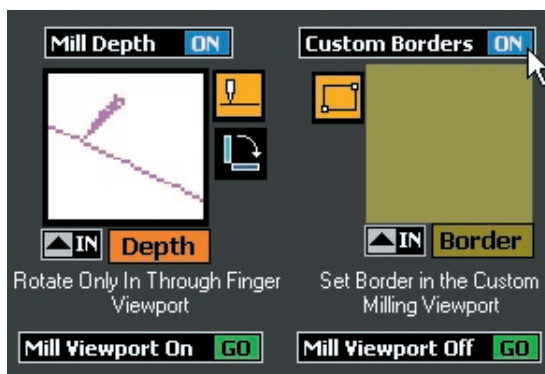
(Note: The Custom mill depth is yellow. The Right mill depth is red and the Left one is orange.)

Select the mill depth and input it into the **Depth** preview window. Because **Mirror Toolpath** was set to **ON**, the mill depth only needs to be input into the preview window one time and it will automatically be mirrored to the other side of the ring.

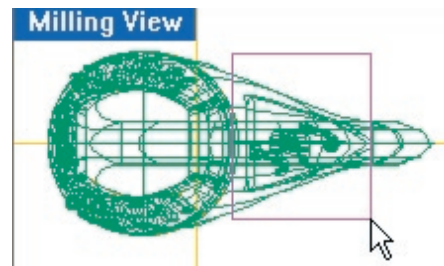
7 Provide a custom border to mill only the shield on the side of the ring at the custom angle:

Turning **Custom Borders** **ON** will now bring up the **Border** preview window and the **Mill Viewport** **ON/OFF** option.

Note: Before drawing the custom border, it is VERY IMPORTANT to click on Mill Viewport GO. This causes the custom viewport to appear. Do not click on the Rectangle tool until this viewport is visible!!

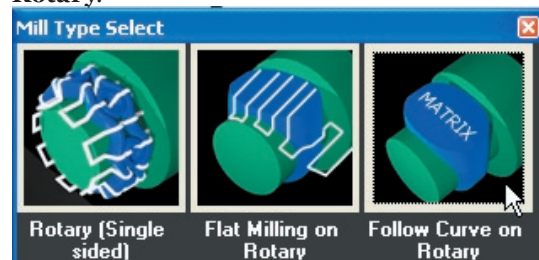


When the **Milling View** viewport is visible, click on the **Rectangle** icon to draw a custom border. Draw the rectangle so that it surrounds only the shield on the side of the ring, and input it into the **Border** preview window. Then, click on **Mill Viewport Off GO** to turn off the **Milling View** viewport.



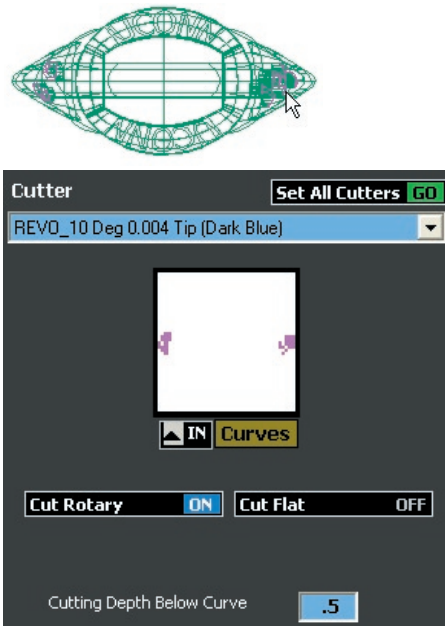
8 Click on Insert GO and select a new operation: Follow Curve on Rotary. Set up the options to better define the lettering on the sides of the ring.

Click on **Insert GO** and select **Follow Curve on Rotary**.



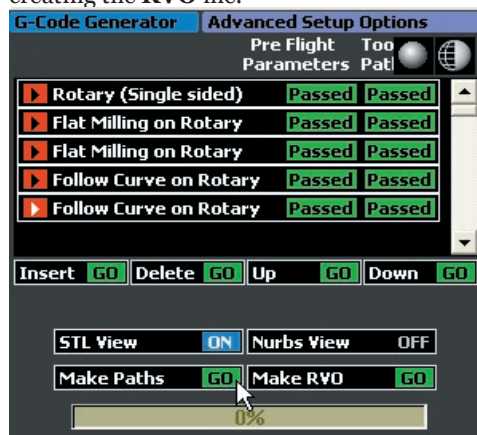
Highlight the curves on the sides of the model and input them into the **Curves** preview window. This time, the curves will be cut out in a rotary fashion to follow the curvature of the ring shank, so make sure that the **Cut Rotary** option is turned **ON**. Recall, in step 1, that these curves were created with the **Dupe Edge** tool at the top of the lettering, and so they must be milled at a depth of 0.5, which

is the depth to which the cutter was set into the ring shank to create the lettering.



10 Click on **Make Paths GO** to create the toolpaths.

Now that each **Pre-Flight Parameter** indicator light reads **Passed** for all of the operations that will be added to this mill job, click on **Make Paths GO** to create the toolpaths. As each toolpath is created, the **Toolpath** indicator lights will change from **Unknown** to **Passed**. In addition, the **Make RVO GO** button will appear in preparation for creating the RVO file.

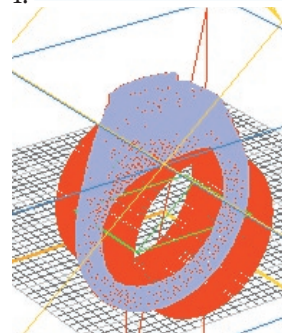


Notice that only the toolpath for the selected operation is visible. Click on the red and black arrows beside each item on the operation list to see its related toolpath.

In the illustrations below, each toolpath is shown

in red or highlighted in magenta for emphasis:

1. Rotary (Single sided)

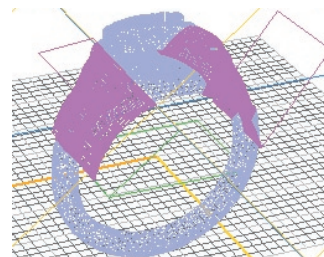


2. Flat Milling on Rotary

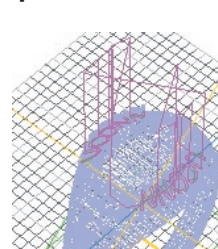


3. Flat Milling on Rotary

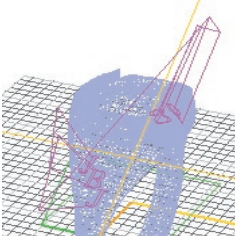
Recall that this toolpath was created with Mirror Toolpath ON and so it appears identical on both sides of the ring:



4. Follow Curve on Rotary



5. Follow Curve on Rotary

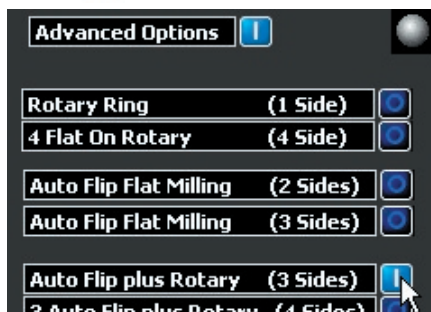


In this case, each of the five operations that make up this toolpath use the same cutter, and so each toolpath is really one mill job. As long as the same cutter was used in all of the operations, the mill will run each one in the order indicated without stopping and without any intervention from the user. If different cutters were selected, the mill will pause and wait for the user to change the cutter before proceeding on to the next step. If the toolpaths are all acceptable, click on **Make RVO GO** to create the Revo file.

II. Auto Flip Plus Rotary (3 Sides): Clean Milling for Hard-to-Reach Spaces

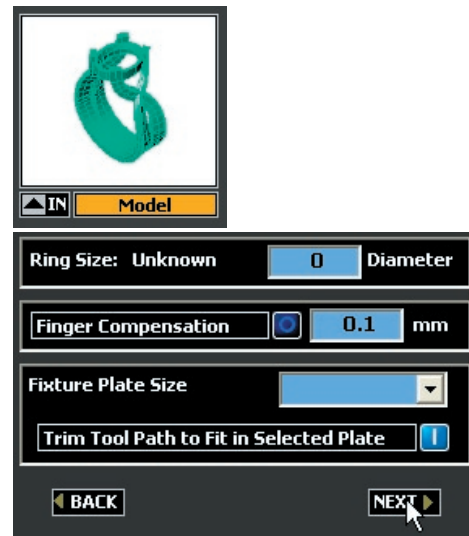
1 Prepare the Rotary with Auto-Flip Sides ring as usual, with outer supports, and click on RevoCam.

Make sure that **Advanced Options ON** has been selected and click on the **Auto Flip Plus Rotary (3 Sides)** option. Click **Next** to move on in the software.

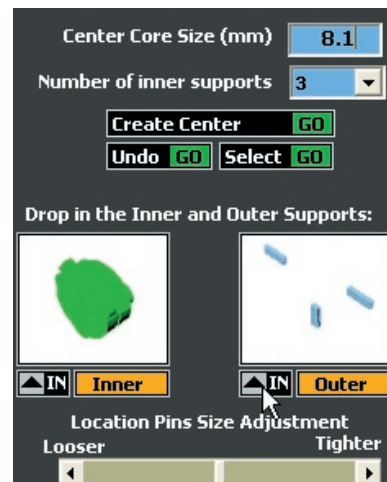


2 Proceed through the software as usual; then, create the STL

Input the model, minus the supports, into the **Model** preview window and set up the desired options for the conventional strategy.



Create the inner supports with a core size about 0.1 mm greater than the thickness of the ring and select the appropriate number of inner supports. Click on **Create Center GO** and then input the inner and outer supports into the corresponding preview windows. Adjust the location pins if desired and click on **Next** to create the **STL** file.



3 Set up the custom borders and mill depths for the default operations:

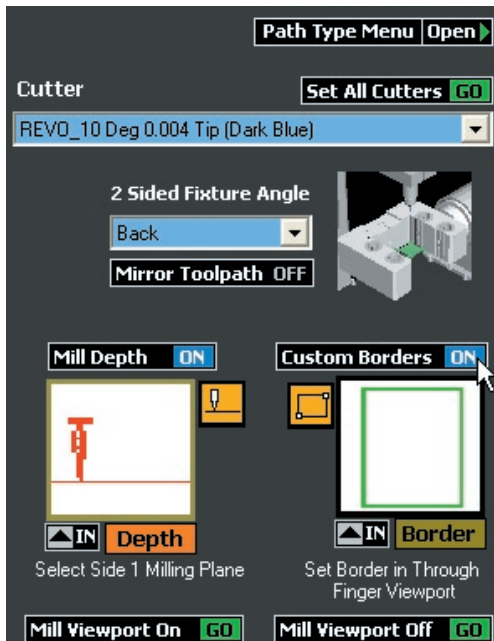
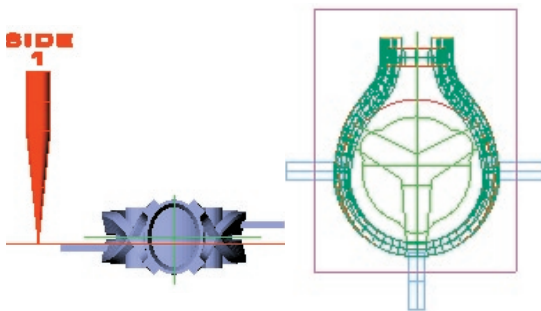
Select the first **Flat Milling (Auto Flip)** operation

by clicking on the red and black arrow beside it to change it to red and white.



The options will appear on the screen below. Select the desired cutter and turn ON **Mill Depths** and **Custom Borders**. Set the mill depth just past center and draw a custom border with the **Rectangle** tool. Input both into their respective preview windows.

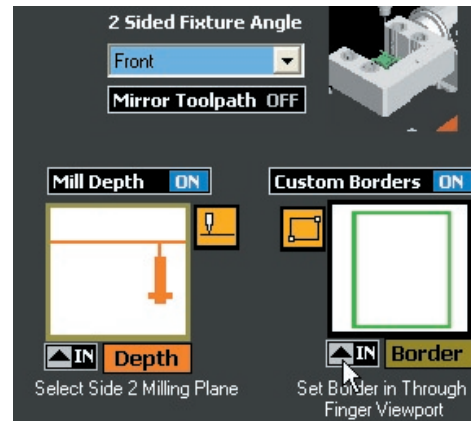
First Flat Milling (Auto Flip): Back



Important: Do NOT turn on Mirror Toolpath!! Even though the same custom border will be used for the other side of the ring, it is preferable to select the next default operation (Flat Milling Auto Flip) and not to mirror the border for the first. This is because, if Mirror Toolpath is turned on, the second operation will be redundant.

At this point, select the next **Flat Milling (Auto Flip)** operation, labeled “Front”, and input the SAME custom border. Adjust the mill depth accordingly.

Second Flat Milling (Auto Flip): Front



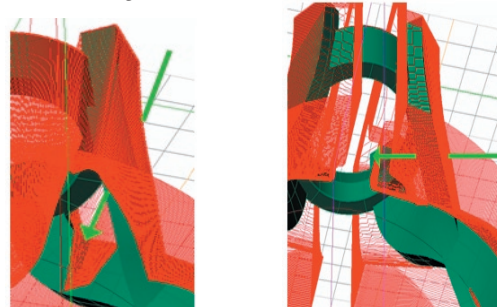
4 Click on Insert GO to select the first new operation for this job: **Flat Milling on Rotary**.



Because of the design of this ring, the conventional toolpath will not reach into the shank details fully. Therefore, rather than completing the shank details with straight, clean sides, the sides are milled at an angle and require further clean-up prior to casting.

However, because the **Flat Milling on Rotary** operation can hold the ring flat at a custom angle on the rotary fixture, the cutter will now be able to reach into the small spaces in this ring, achieving a cleaner finish (see illustrations below)

With Rotary 1-Sided: With Flat Milling on Rotary:

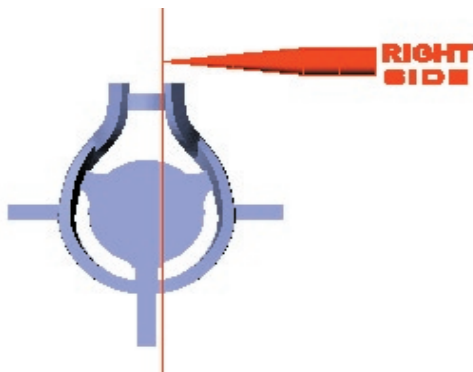
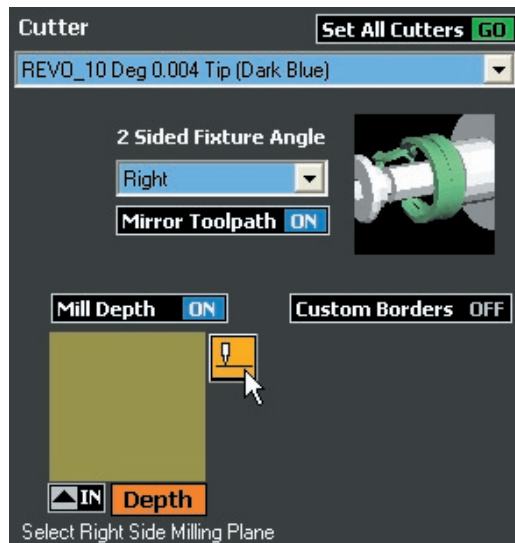


5 Set up the Mill Depth for the new milling operation.

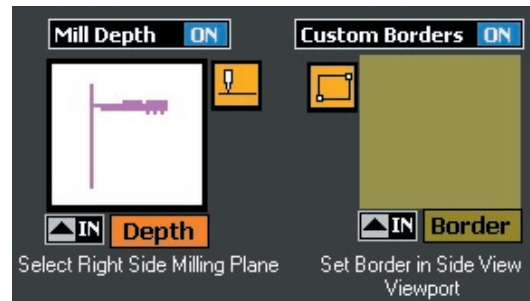
Select the desired cutter and set the **2 Sided Fixture Angle** to **Right**. Make sure that **Mirror Toolpath** is turned **ON** in order to set up the **Left** side without the need to create another new operation.

Next, Turn **ON** the **Mill Depth** and click on the **Mill Depth** icon beside the **Depth** preview window. The cutter and cutting plane will appear in the viewports. (The “Right” cutter is red and the “Left” cutter is orange.)

Select the mill depth and drag it just past the inner edge of the shank to ensure that the proper depth will be reached during milling.



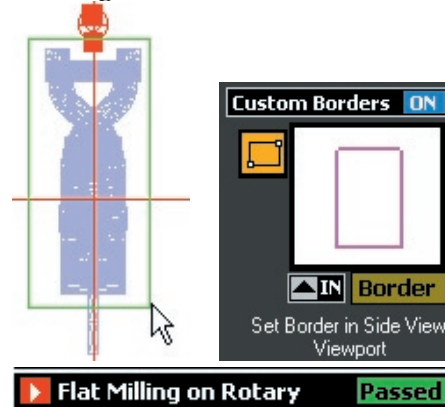
Then, highlight the mill depth and insert it into the **Depth** preview window.



6 Draw a Custom Border for the new operation.

Turn **Custom Borders** **ON** and click on the **Rectangle** icon. Be sure and follow the handy directions just below the **Border** preview window: draw the border in the **Side View (Right)** viewport only!!

Insert the border into its preview window and the **Pre-Flight Parameters** will now read **Passed**.



7 Make the toolpaths.

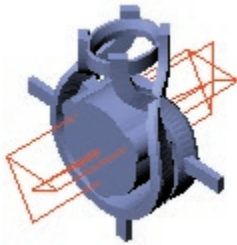
Click on **Make Paths GO** to create the toolpaths for each operation. Click on the arrow beside the new operation to see the results in the viewports.

As illustrated below the toolpath reveals that, following the conventional rotary operation, the ring will be held still on the rotary fixture at a 90 degree angle on the right, and then on the left, so that the shank details can be milled with straight, smooth sides.

Once all of the toolpaths are acceptable, click on **Make RVO GO** to create the Revo file.



1: Cut out supports



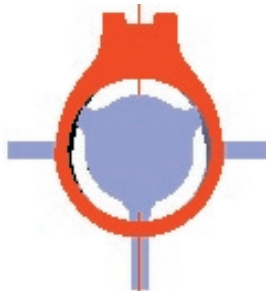
2: Flat Milling Auto-Flip (Back)



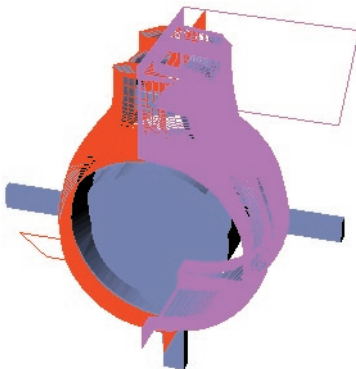
3: Flat Milling Auto-Flip (Front)



4: Rotary (3 Sided Auto)



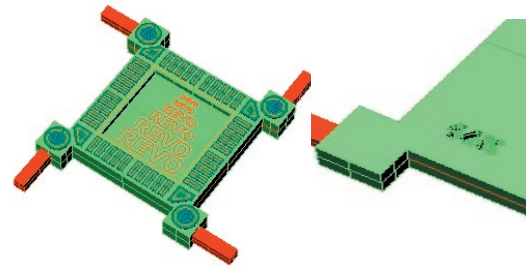
5: Flat Milling on Rotary (Mirror ON – two separate toolpaths, left and right, shown in two different colors, below, for emphasis)



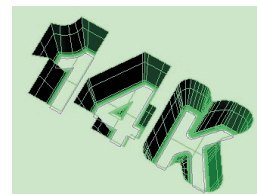
III. Auto Flip Flat Milling (2 Sides): Follow Curve on Flat / Adjusting Coarseness

1 Prepare the model for the conventional and new operations.

By now this should be old hat: put the supports on the model in preparation for the Two-Sided Auto Flip operation. The curves on the top of the model will be input into the **Follow Curve on Flat** operation to achieve the effect of curve engraving, and so nothing further needs to be done to them.

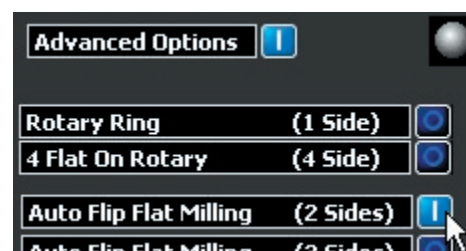


Notice that, in this model, there is a “14K” engraved on one section of the back of the model. Because the lettering is so tiny, it is unlikely that the conventional toolpath will be able to cleanly cut out the details. Use the **Dupe Edges** tool found in the **Curve** fly-out menu to create curves at the base of the lettering. These curves will later be input into the **Follow Curve on Flat** operation in order to enhance the detail for this engraving.

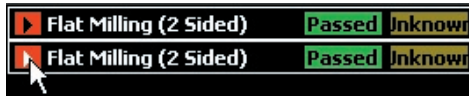


2 Start up the RevoCam software and complete the Auto Flip Flat Milling (2 Sided) strategy with Advanced Options turned ON.

Complete the familiar screens in the software and create the **STL** file.

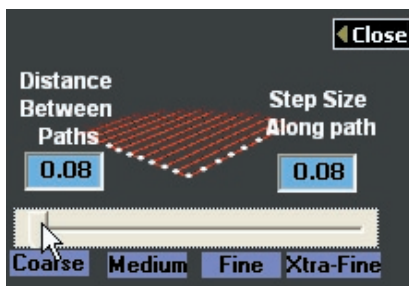
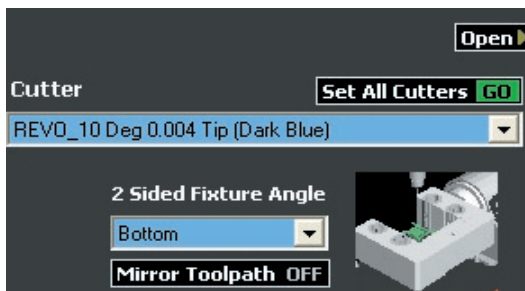


3 Click on the second **Flat Milling (2 Sided)** operation to adjust the options for this operation.



Because only one section of the underside of the object is detailed, it is not necessary to mill the entire back section with such high quality. Therefore, adjust the options that appear when the second milling operation is selected and switch to a coarser quality milling job.

If desired, select a new cutter. Then, click on **Path Type Menu Open** to adjust the toolpath options. Move the slider bar down to the **Coarse** setting. This will speed up the milling time. Then, click on **Close**.

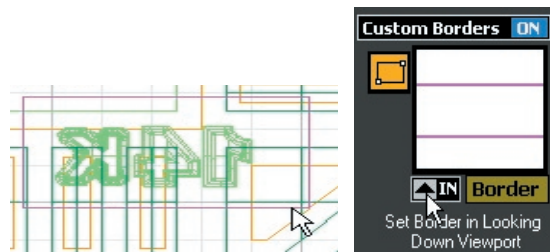


4 Click on **Insert GO** and select the **Flat Milling (2 Sided)** operation. Adjust the options.

In order to provide definition to the single, small area on the back that is engraved, insert a **Flat Milling (2 Sided)** operation and check that the **2 Sided Fixture Angle** is set to mill the **Bottom** of the model. **Mirror Toolpath** should be set to **OFF**, since this custom region need only be defined for the bottom of the model.

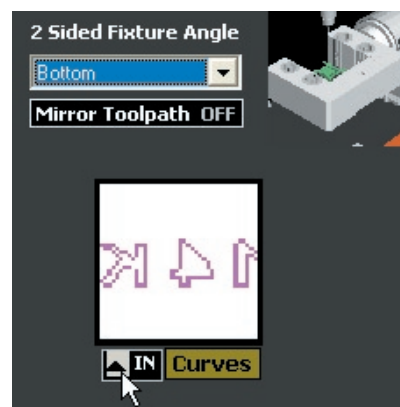


Accept the default **Mill Depth** and set a **Custom Border** in the **Looking Down (Top)** viewport so that only the "14K" is surrounded by the border. Input the border into the **Border** preview window.



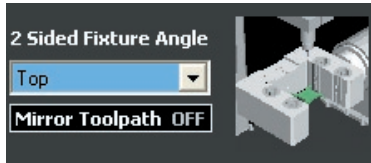
5 Insert a **Follow Curve on Flat** operation into the list.

To really define this engraving, insert a **Follow Curve on Flat** operation into the list. Select the **Bottom** fixture angle and turn **Mirror** OFF once more. Highlight the curves that were created in step 1 to define the bottom edges of the lettering. No **Cutting Depth Below Curve** needs to be indicated, because the curves were created at the bottom of the letters.

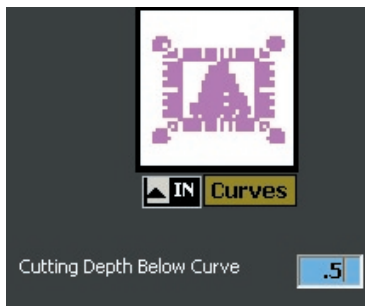
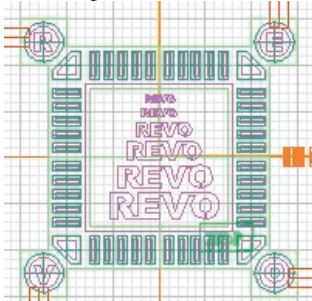


6 Insert one more Follow Curve on Flat operation to cut out the curves on the top of the model.

Make sure that the **Top** fixture angle is selected this time and **Mirror** is turned OFF.



Select the curves on the top of the model and input them into the **Curves** preview window. Because the curves are flat on the surface of the model, it is important to set a **Cutting Depth Below Curve** for this operation so that the curves are cut with some depth.

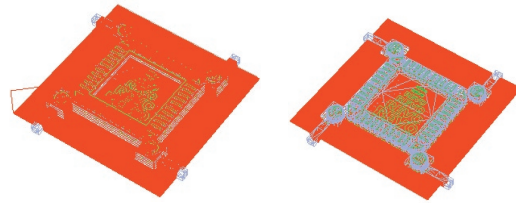


7 Click on Make Paths GO to see the results of the software.

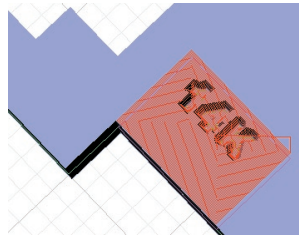
Once all of the toolpaths are acceptable, click on **Make RVO GO** to create the Revo file.



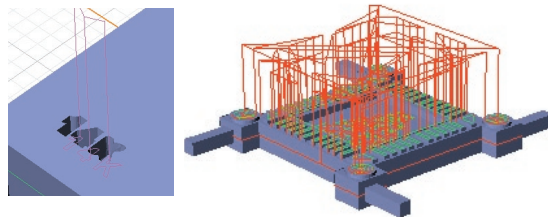
(L) 1: Flat Milling (Two Sided) Top; (R) 2: Flat Milling (Two Sided) Bottom



3: Flat Milling (Two Sided) Bottom, Custom Border



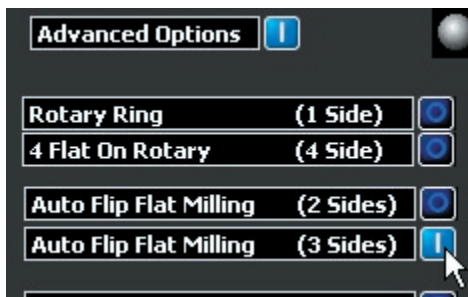
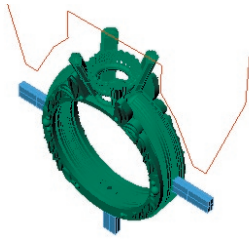
(L) 4: Follow Curve on Flat Bottom; (R) m; 5: Follow Curve on Flat Top



IV. Auto-Flip Flat Milling (3 Sides): Milling inside bottom of shank / Achieving flat finish

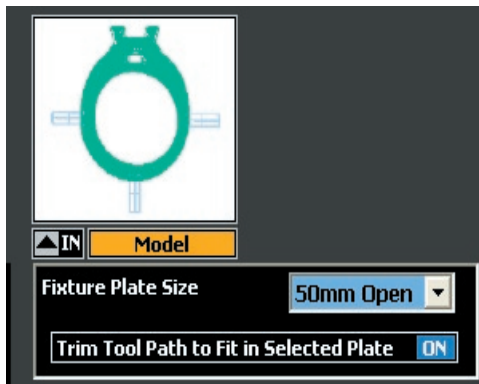
1 Prepare the model for Auto-Flip Flat Milling (3 Sided) operation.

Place outer supports to hold the shank and a cut-out curve at the top of the ring to define the shape of the wax cut-away that is milled in the first operation. For further instructions on creating this curve, consult the relevant section of this manual.



2 Start the RevoCam Software and turn **Advanced Options ON**. Select the **Auto Flip Flat Milling (3 Sides)** strategy.

Make sure to input the model and the supports during the second step of the operation. Then, create the **STL** file.



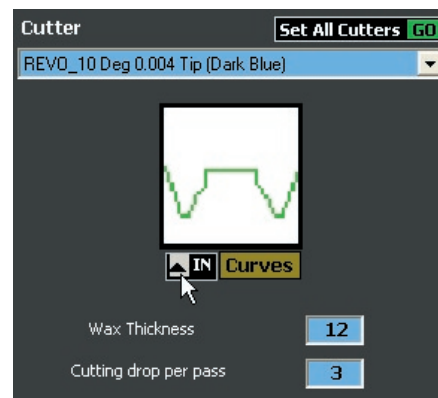
3 In the **Advanced Setup Options** screen, select the **Cut-out Edge Curve** operation to make the options visible and input the edge curve into the preview window.

The first operation in the **Advanced Setup Options** menu will indicate **Failed**. Simply click on the arrow beside it to select it, and the associated options will appear at the bottom of the screen. Select the desired cutter from the drop-down menu.

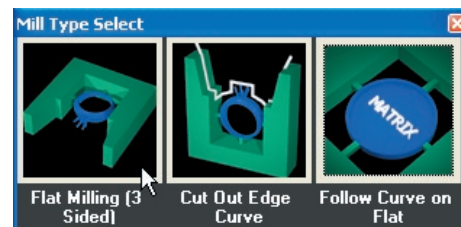
Then, click on the edge curve and input it into the software. If necessary, adjust the **Wax Thickness**

by inputting how many millimeters thick the wax is. (The default value "12 mm" is for a standard block of the green wax recommended for use with the Revo mill. The 3 mm cutting drop per path is also a recommended setting for this wax and therefore the default value in the software.)

Next, adjust the **Cutting drop per pass** by lowering it for firmer wax and raising it for softer wax. This number indicates the distance, in millimeters, that the cutter drops to perform the next pass. For firmer wax, the cutter should perform more passes (have a lower drop depth) to ensure a clean cut.

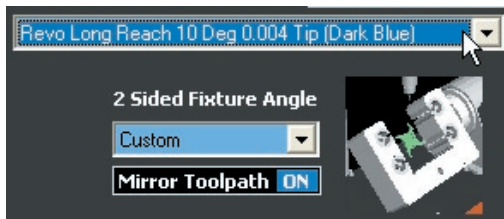


4 Click on **Insert GO** and select the **Flat Milling (3 Sided)** operation. Set up the **Mill Depth** to reach the inside bottom of the shank.



From the options screen for the **Flat Milling (3 Sided)** operation, select the **Revo Long Reach 10 Degree** cutter. It is very important to select this cutter in order to create a proper toolpath for this operation.

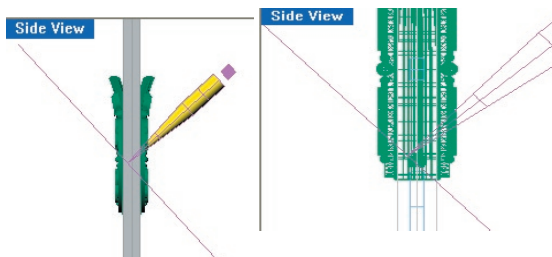
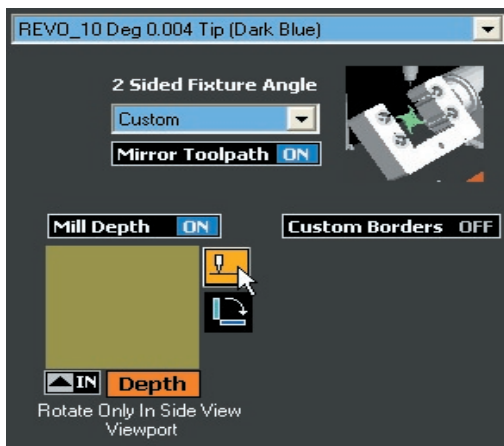
Set the **2 Sided Fixture Angle** to **Custom**. Turn **ON Mirror Toolpath** so that the operation will be performed on the other side of the ring, as well.



Next, turn **ON Mill Depth** and click on the **Mill Depth** icon so that the cutter and cutting plane appear in the viewports.

*Important: Take notice of the helpful reminder beneath the **Depth** preview window: "Rotate Only in Side View (Right) Viewport"! (Trying to rotate the mill depth in any other viewport will yield an invalid angle that the software will not accept.)*

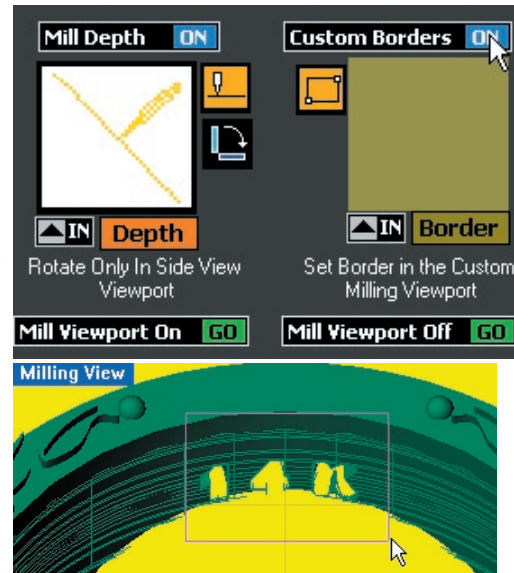
Then, highlight the mill depth and click on **Rotate**. Move the cursor over to the **Side View (Right)** viewport and rotate the mill depth to about a 45 degree angle. Still working in the **Side View (Right)** viewport, drag the mill depth down until the cutting plane intersects the shank completely.



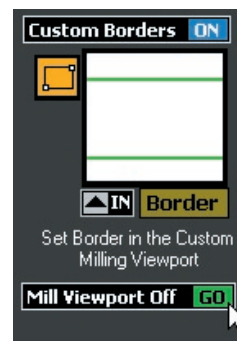
5 Click on **Custom Borders ON** to create a custom border in the Milling View viewport.

After inserting the mill depth into the **Depth** preview window, turn **Custom Borders ON** and click on **Mill Viewport On GO** to bring up the **Milling View** viewport.

Remember: Do not click on the Rectangle tool until after turning on the Milling View viewport!



Draw a custom border around the part of the shank to be milled and input it into the **Border** preview window. Then, click on **Mill Viewport Off GO** to return to the usual four-viewport view of Rhino.

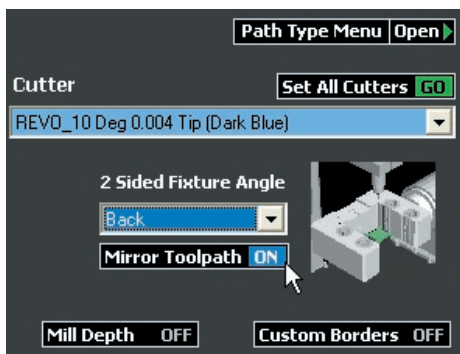
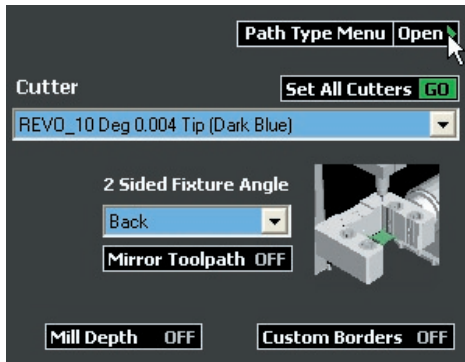
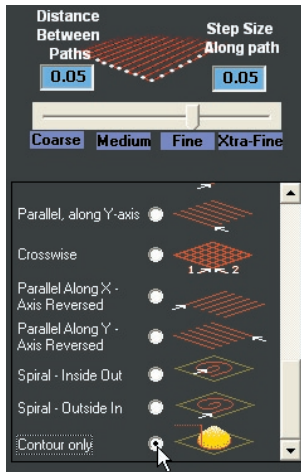


This toolpath will be mirrored to the other side of the ring to cut out the details on the inside bottom of the shank evenly from both sides.

6 Insert one more **Flat Milling (3 Sided)** operation and set up the options to complete a **Contour** cut with the flat cutter.

Start by opening the **Path Type Menu**. Select the desired level of coarseness and then scroll all the way down through the menu to **Contour only**.

Click in the tick box beside this option to select it, and then click on **Close**.



Turn on the **Mirror Toolpath** option. The contour cut can be run on either the front or the back **2 Sided Fixture Angle** and, as long as it is mirrored, the toolpath will be created on both sides of the ring. If they are acceptable, leave OFF the **Mill Depth** and **Custom Borders** options and the mill depth will default to 1 mm past the center of the model (on each side); the custom border will default to 1.5 mm in from the extents (supports) of the model and 1 mm above the top of the model.

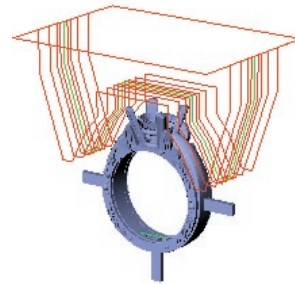
This is fine, because the **Contour** operation simply creates a toolpath around the inside and outside extents of the model.

Please Note: You cannot enter freeform or circular custom borders during a Contour pass: the border **MUST** be rectangular for the Contour toolpath to run as expected.

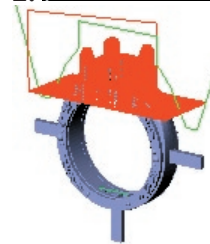
7 Click on Make Paths GO to create the toolpaths for this strategy. Once the toolpaths are acceptable, click on **Make RVO GO** to create the Revo file.



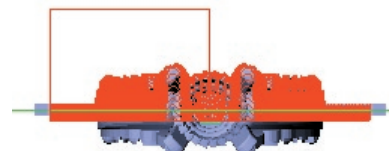
1: Cut Out Edge Curve



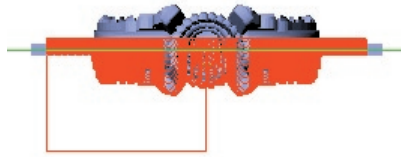
2: Flat Milling (3 Sided)



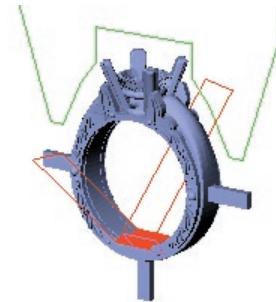
3: Flat Milling (3 Sided)



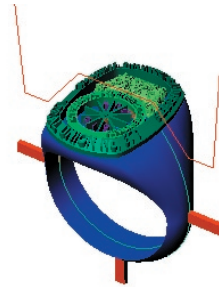
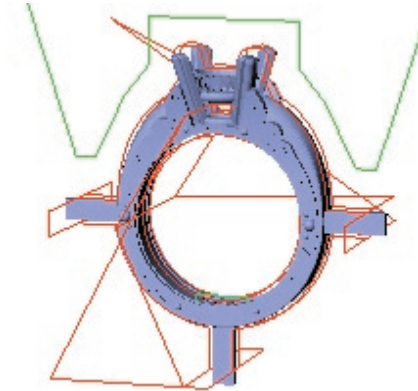
4: ▶ Flat Milling (3 Sided)



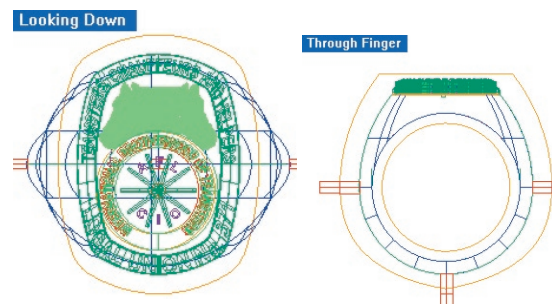
5: ▶ Flat Milling (3 Sided)



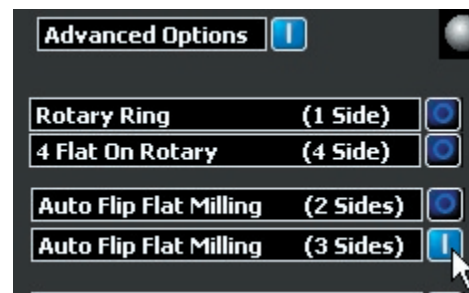
6 ▶ Flat Milling (3 Sided)



2 Then, using any **Curve** tools you wish, draw freeform custom borders in the proper viewport for the typical borders used in this operation: one that surrounds the head details in the **Looking Down (Top)** view (below, left); the other that surrounds the extents of the ring (and one that falls within the finger hole if the above standards have been met) in the **Through Finger (Front)** viewport (below, right). New freeform borders are illustrated in yellow, below:



3 Now, enter the **RevoCam Software** and select the **Auto Flip Flat Milling (3 Sides)** operation with the **Advanced Options** turned on. Proceed through the software screens as usual: input the model and supports, select the **50 mm Open** plates, and create the **STL** file.

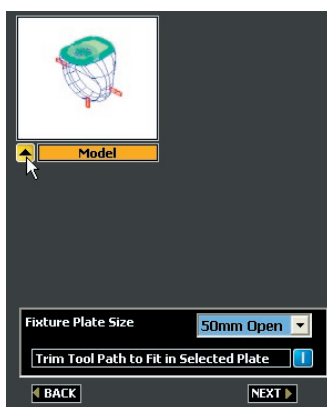


V. Freeform Borders on a Three-Sided Auto-Flip Advanced Milling Path:

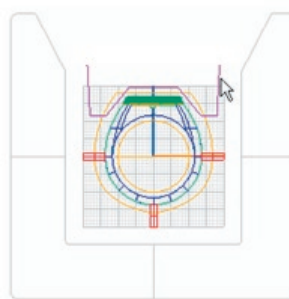
The following example will demonstrate how to set up the three-sided auto-flip job shown below. **Freeform Borders** are available on any auto-flip flat milling operation: two-sided, three-sided, or as part of an auto-flip plus rotary job.

To use this advanced option,

1 Prepare the model for milling, with the typical cut-out curve and supports.



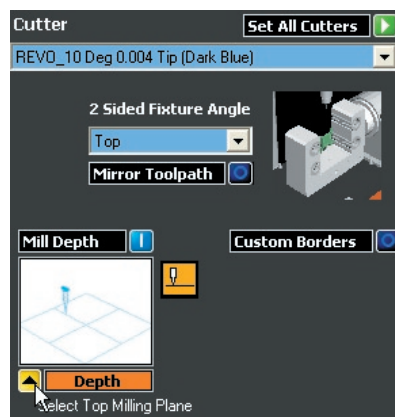
4 In the **Advanced Options** screen, start by setting up the first toolpath. Select the **Cut-out Curve** and enter it into the **Curves** preview window that appears when this toolpath is selected in the top part of the **Advanced Options** screen:



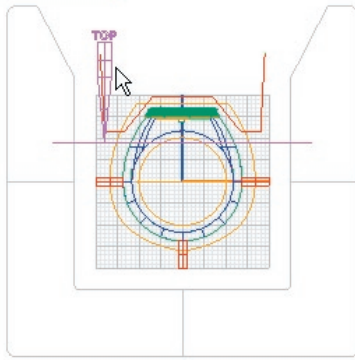
5 In the first **Flat-Milling** operation, the **Top-down** operation, select the **Cutter** of choice.

***Note:** When freeform borders are used on a **Flat Milling** pass, the **ONLY Path Type** option will be the **Parallel along the X-Axis** path. This is the only setting on which **Freeform Borders** will work. The software will **OVERWRITE** any of the user's settings in the **Path Type** section of the software with the **Parallel along the X-Axis** path.*

Toggle "ON" the **Mill Depth** indicator light and select the **Mill Depth** icon to place the **Top** mill depth in the viewports. Adjust the **Top** mill depth as usual (about 1 mm below the bottom of the head) and import it into the **Depth** preview window in the software.

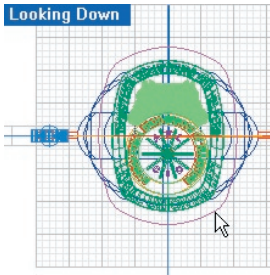


Through Finger



To import the freeform borders into the software, toggle “ON” the **Custom Borders** indicator light and select the appropriate freeform border (the one visible in the **Looking Down (Top)** viewport). Import it into the **Border** preview window and move on to the next operation.

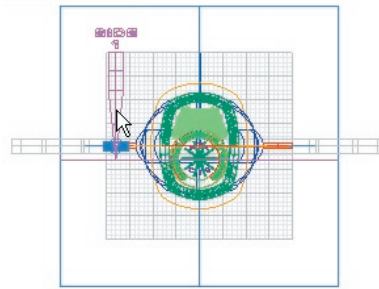
Looking Down



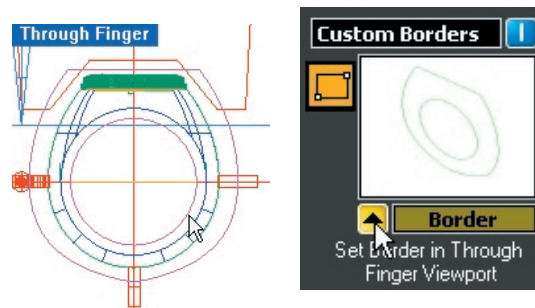
6 The next **Flat Milling** operation references the **Back** of the **Two-sided Auto Flip** portion of this job. Set the **Cutter** and **Mill Depth** as usual:



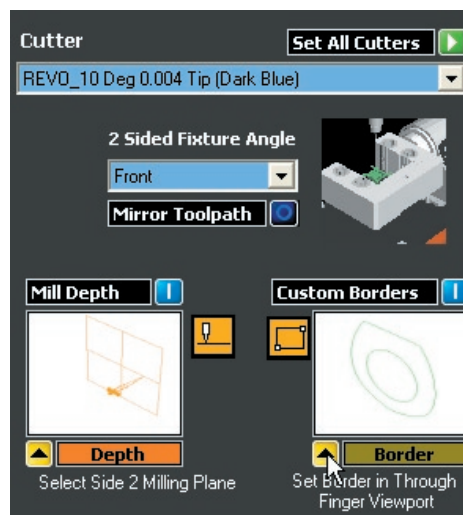
Looking Down

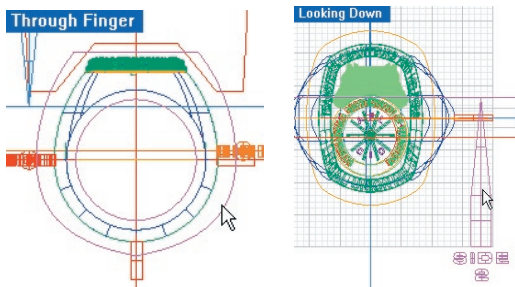


7 Then, toggle “ON” the **Custom Border** indicator light and select *all* freeform borders present in the **Through Finger (Front)** viewport: in this case there are two: the one for the outside of the ring and the one for the inside of the ring.

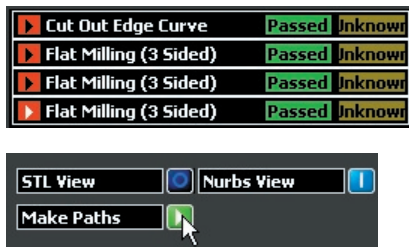


8 Repeat **Steps 6 & 7** for the final **Flat Milling** operation: the one that completes the **Front** of the ring:



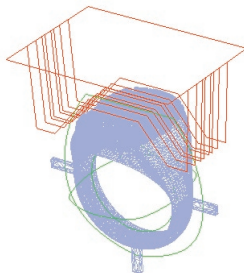


9 All of the toolpaths should now display **Passed**. Click on **Make Paths** to see the results:

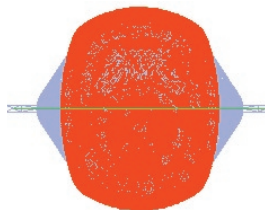


The toolpaths should somewhat match the following illustrations:

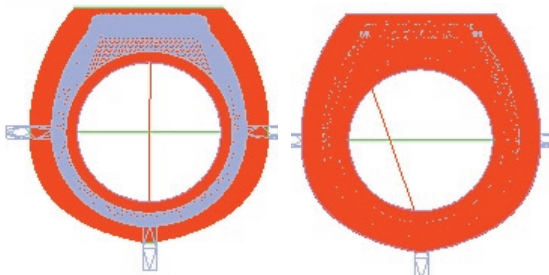
Cut-out Edge Curve:



*Flat Milling (Top-down)
Note freeform shape*



Flat Milling (Back – left and Front - right): Note freeform shape

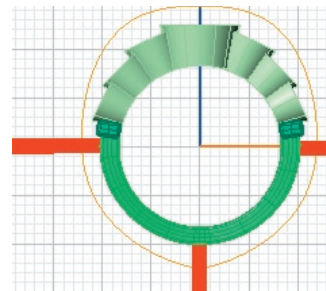
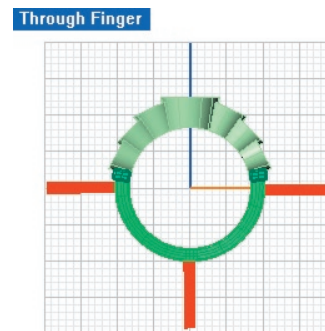


10 Click on “Make RVO” to create and save the RVO file to load it onto the mill.

VI. Using (Flat) Freeform Borders, Trim Toolpath, and (Rotary) Custom Borders with an Auto-Flip Plus Rotary 3-Sided Strategy:

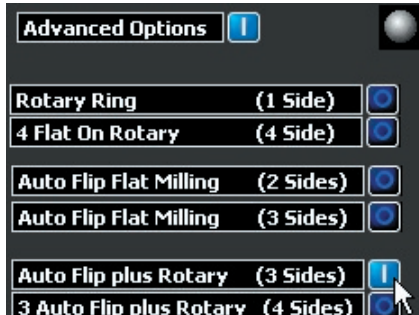
In the following **Auto-Flip plus Rotary (3 sides)** example, we’ll use all three of the new **Advanced Options** strategies: (1) **Freeform Borders** for Flat Milling, (2) **Trim Toolpath** for Rotary Milling, and (3) **Custom Borders** for Rotary milling:

1 To begin, create the supports at the 3, 6, and 9 o’clock positions as usual in **Rhino** using the **Box** tool. Then, using any of the **Curve Menu** commands, create **Freeform Borders** around the ring in the **Through Finger (Front)** viewport (since we’ll just be using the **Three-sided Rotary** strategy for this ring, there is no need to place a custom border in the **Looking Down (Top)** viewport.)

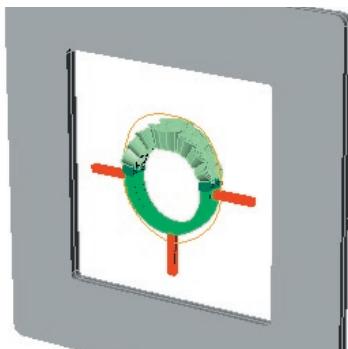
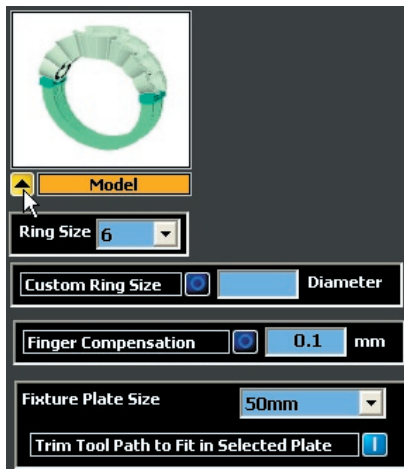


Note: Remember **NOT** to create a freeform border inside the finger hole during an **Auto-Flip plus Rotary** strategy. This extra time-saver is **ONLY** appropriate during the **Two- or Three-Sided Auto-Flip** strategies, since, during either of the two **Auto-Flip plus Rotary** strategies, this border would cancel out the creation of the hub, an essential part of these strategies!!

2 Now, start up the **RevoCam** software and select the **Auto Flip plus Rotary (3 Sides)** strategy with **Advanced Options** activated. Click on **Next** to proceed in the software:

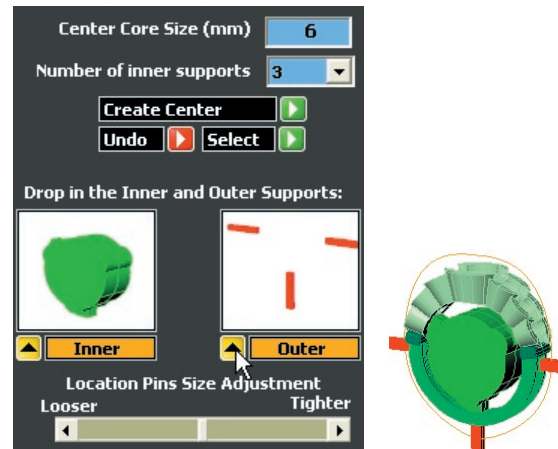


3 Select *just* the model and insert it into the **Model** preview window in the software interface. Select a **Fixture Plate Size** (50 mm indicated below) and make sure the ring size is correct. Click on **Next** to proceed.

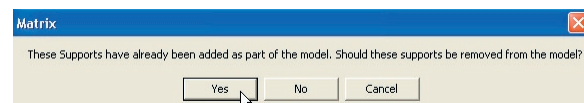


4 In the **Next** screen of the software, create a **Center Core** that is 1 mm larger than the narrowest part of the ring as measured in the **Side View (Right)** viewport. Input the center core

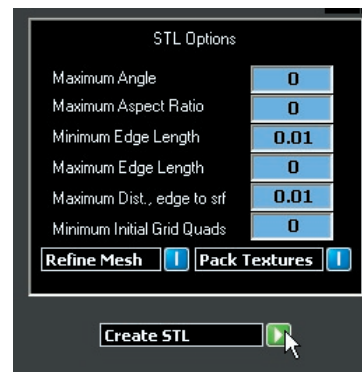
(**Inner** supports) and three **Outer** supports into the appropriate preview windows and click on **Next** to proceed in the software.

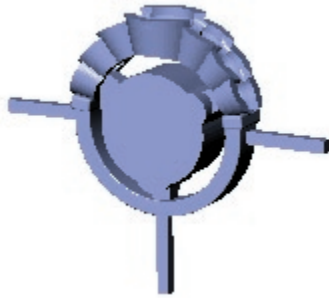


***Note:** If the following message pops up after inputting the **Outer** supports, it probably means that you accidentally input the supports into the **Model** preview window in the previous software screen. The software is just checking to make sure that you intended them for support material, not model material. Click on **YES** to remove them from the model and thereby treat them **ONLY** as supports. This will ensure that the rotary toolpath does not take them into consideration when cutting the ring.*

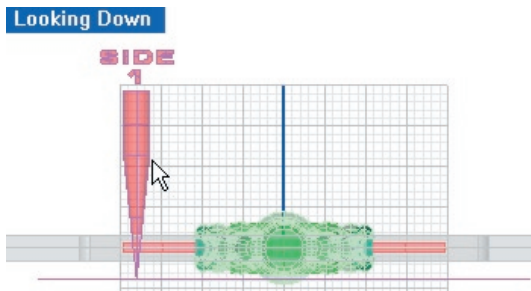
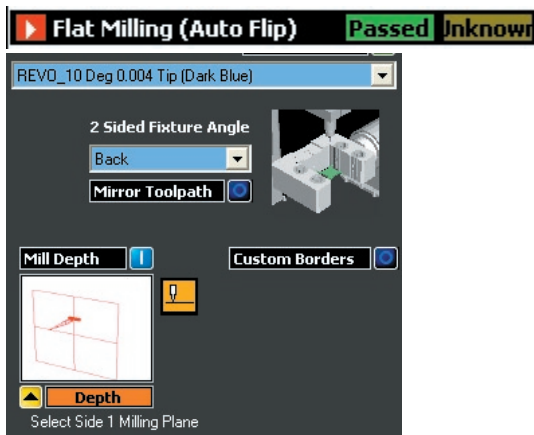


5 Click on **Create STL** in the next screen of the software.



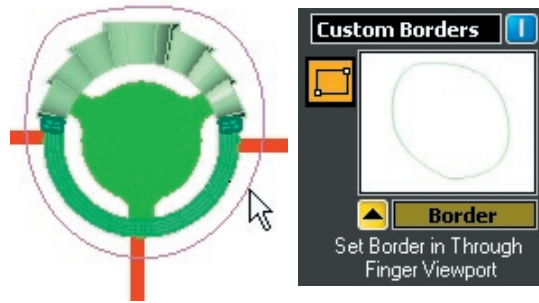


6 In the final **Advanced Options** builder screen, start by selecting the first **Flat Milling (Auto Flip)** operation. This creates the **Back** side of the **Two-Sided Auto Flip** portion of this strategy. Select the appropriate cutter and set up the **Mill Depth** for this portion of the job.

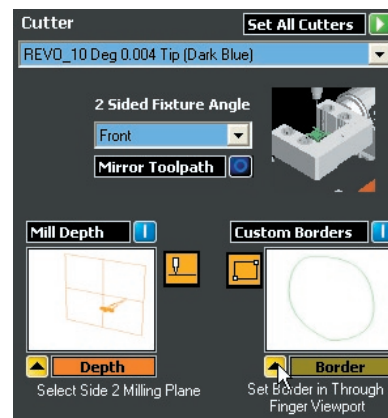


Note: you will need to turn on the **Nurbs View** in order to complete the next portion of this strategy.

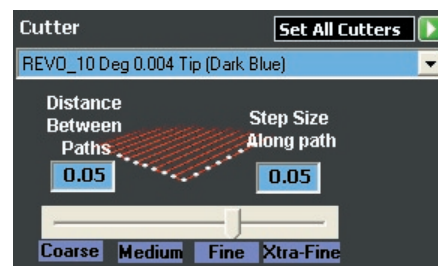
7 Next, click on the **Custom Borders** indicator light to activate this option and input the **Freeform Border** previously created in **Matrix** into the preview window.



8 Repeat steps 6 & 7 for the other **Flat Milling** operation: this mills the **Front** of the ring.

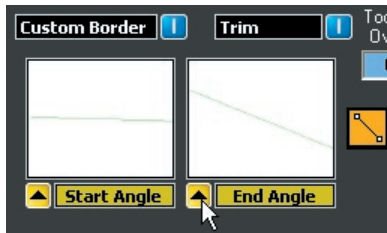
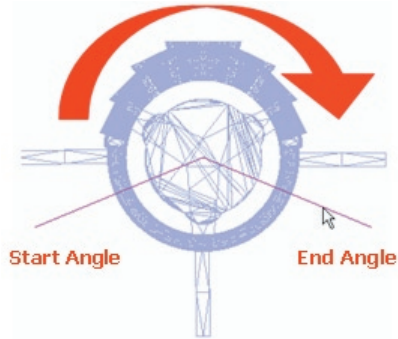


9 Now it's time to specify settings for the rotary pass. Select the appropriate cutter and finish type.



10 Select the **Custom Borders** option to bring up these interface controls. Click on the yellow **Line** tool that appears to create the "Start Angle". The line automatically starts at **F4** in the **Through Finger** viewport. Extend the line to the left side of the ring, adjusting the angle to the required starting position. Left mouse-click to set the start angle. Any parts of the ring that you enclose between the **Start Angle** and the **End**

Angle, working in the clockwise direction, will be milled using the settings indicated here. Input the **Start Angle** (left-hand line) into the **Start Angle Preview Window** and input the **End Angle** (right-hand line) into the **End Angle** preview window.



11 Now, select the **Trim** function if you wish to indicate a **Toolpath Overcut**. The default value is 0.5 mm; however, you may enter another value if need be.



12 If desired, here's another step you can try with the **Custom Borders** option. Imagine that you wanted to run a rotary pass along the bottom of this shank; however, you are not concerned about the surface finish in this location. You can save time by adjusting the surface finish from fine to medium or coarse on a section of your piece without many details. In that case, you would reverse the **Start** and **End** angles so that everything *between* them, working in a clockwise direction, would be milled with the new surface finish.

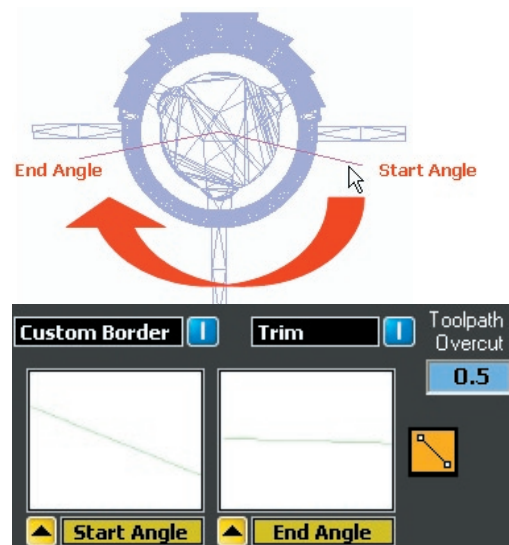
To try this for yourself, click on **Insert** and add another **Rotary (3-Sided Auto Flip)** pass to this piece:



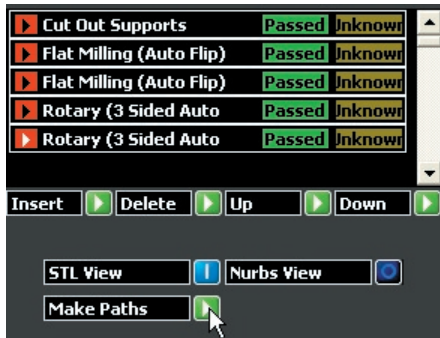
In this builder screen, select the desired cutter (it is fine to select a new cutter here, so long as you remember that the software will generate a separate cut file for the new tool selected and the mill will *stop* and wait for you to change cutters). Adjust the **Surface Finish** so that the bottom of the ring will be milled with a coarser finish than the top of the ring.



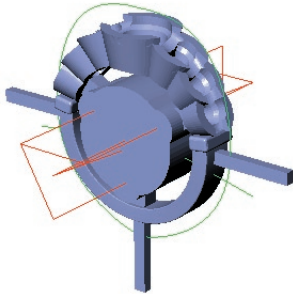
Now, select the **Custom Border** option. This time, *SWITCH* the **Start Angle** and the **End Angle** by *inputting each angle into the OPPOSITE* box from the way they were input in the previous step. Since these angles work to ascribe the cutting area in a clockwise direction, everything between them at the *bottom* of the shank will be milled this time with the coarser surface finish. To complete this strategy, indicate the desired **Toolpath Overcut** for this operation, if any.



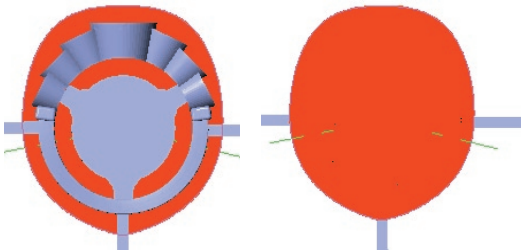
13 With all of the toolpaths indicating **Passed**, click on the **Make Paths** button. Once that has finished, inspect each toolpath to make sure it is correct.



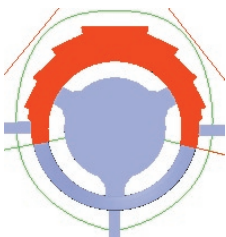
Cut Out Supports:



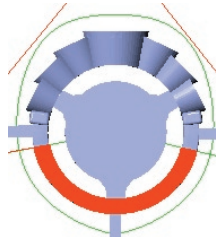
Flat Milling Autoflip: Front & Back



First Rotary Pass:
(finer surface finish setting)



Second Rotary Pass:
(coarser surface finish setting)



14 If they all look good, click on **Make RVO** file and select a place to save the RVO file in order to open it on your mill.

Keep updated on changes to Revo software on Gemvision's website: <http://www.gemvision.com/revo/>

Gemvision's Coolant Policy Statement

Please read this carefully.

Some mill users advocate using some sort of detergent or soap additive in conjunction with the coolant in order to minimize the clean-up process after a milling job. However, Gemvision cautions against this because many detergents (Tide, for example) contain abrasives or corrosives that may be detrimental to the Mill's systems.

Gemvision has not tested chemical reactions of anything other than Koolmist and the highly recommended Synspar GP coolant on the Mill machine and therefore has no way of knowing the possible chemical reactions that may occur, or even potential rusting of internal parts that may not be readily visible. User-created mixtures of anything other than the proper ratio of Koolmist or Synspar GP coolant and distilled water only may damage any associated machine parts, not limited to, but including the pump and coolant system.

Any damage to the mill resultant from user-created chemical mixtures voids respective portions of the Mill warranty. Therefore, Gemvision recommends only the use of Synspar GP which we have tested and found to best suit the requirements of the cutting application, coolant system, and the machine tool, which is why we indorse it and make it available in convenient quart packages. Be sure to only use distilled water and follow the mixture ratio as directed.

Gemvision's Mystic Cleaning Gun Warning Statement

Please read this carefully.

While these guns are relatively safe, if used properly, they are not completely without risk to the user. The 1,650 PSI stream produced by the gun is capable of puncturing human flesh. The result is an "injection type" injury which, if not properly cared for, could result in infection, tissue damage, vascular compression, and even amputation.

It is possible that an emergency room physician may not know the appropriate treatment procedures. You can download a copy of the proper treatment procedures from the website noted at the top of the page.

Vic Barocas, COO of ANC, and Dr. Raymond Wilson, ANC Corporate Technical Director, co-authored the treatment article. The article is also published on ANC's website.

With that being stated, the Mystic #01B Textile Cleaning Gun with Adjustable Nozzle is available from two companies:

American-Niagara Company (ANC)
6690 Jones Mill Court
Suites A & B
Norcross, GA 30092
888-490-1212
ervin@american-niagara.com
ANC's Website
ANC's Product Brochure

Westar Solutions
11653 Higgins Airport Way
Burlington, WA 98233
800-574-3477
erin@westarsolutions.com
Westar Solutions' Website