REVO

DUAL SPINDLE

540C

INSTRUCTION MANUAL

Revision April 2011

Gemvision

Revo 540C Instructional Manual

Table of Contents

Chapter 1

Assembly, Parts, & Features

Follow steps to uncrate and assemble Revo 540C, and learn to use and maintain its software and hardware components.

An Introduction to your Revo 540C Mill

Welcome to the Revo C instruction manual! This chapter will introduce you to each part and accessory of the Revo 540C mill that is shipped to you. You will then learn the steps for uncrating/ re-crating; assembling the mill; powering-up; and finally installing and maintaining important hardware components and accessories that accompany the mill, including the coolant system, cutters and spindle components, and fixtures.

The Revo Software & Wax Calibration

This chapter will then explore the Revo (Revo Project Management) software, which is found on the Revo computer and controls the mill. Learn how to run the Wax Calibration Block: the first wax you should cut on your mill BEFORE you use the mill to cut models you created in Matrix. This quick and easy process allows the mill to detect and correct for any movement of its axes that may have occurred during shipping.

After the mill cuts this block, you'll take several measurements using the digital calipers included with the mill and input these into the mill's computer. The mill will use this information to automatically correct for any shifting that may have occurred during shipping. After completing this process, you're ready to begin creating models in the Matrix Mill Builder to run on the Revo 540C (explained in Chapters 2-3).

Chapters 2 - 3: The Matrix Mill Builder

The next two chapters in this manual explain how to set up basic (Chapter 2) and "Detailed Options" or more advanced (Chapter 3) jobs on the Revo C mill. They cover all the steps for producing the ".RVOC" file in the Matrix software, which gets transferred to the mill, and set-up on the Revo, which can vary depending on the fixture you're using.

Step-by-step tutorials are also included to walk you through each type of job. You'll learn how to correctly prepare the software and hardware for various types of models.

This chapter will take you through uncrating, assembly, set-up, installation, maintenance, and the very first wax you must cut... from a crated-up mill (above) to the fully-functioning Revo 540C (below).

Tutorial 1: Uncrating & Set-Up of Revo 540C

Follow these instructions to uncrate your mill upon first receiving it. It is HIGHLY recommended that you save the mill crate for future use.

Use a power screwdriver to remove the screws from the top of the mill's crate. Then, remove the contents from the top of the crate and remove the shelf over the mill. Disassemble the front and back of the crate FIRST; then the sides of the crate. Cut the zip ties holding the mill to the base of the crate, and the mill will be free. The mill should ONLY be lifted by the handles on its base, and ONLY by two or more people, as it weights 230 lbs. Place it on a level, sturdy table or countertop so it will not wobble during milling. DO NOT try to "shim" (level) the mill by adjusting ANYTHING on its frame. If necessary, add card stock or another material beneath the unlevel foot of the table or the mill to level it.

Materials Needed

Included with Mill:
- **Crate (L X W X H = 36" X 34" X 48")**

Not Included with Mill:
- **Power screwdriver**
- **At least 2 people to lift the mill**
- **Level, sturdy table / countertop**

1 - Remove screws from cover & set aside. Remove Cover.

1 Remove Screws from Crate Cover Begin by using the power screwdriver to remove the screws from the top and sides of the crate's cover. Lift off the cover and set aside.

2 Remove Contents & Shelf Remove the contents from the shelf on top of the mill. DO NOT discard any of its contents until you've verified that all mill parts & accessories (listed in the next section of this manual) are accounted for. Lift out the shelf and set aside.

2 - Remove contents (above, left) and set aside. Remove shelf (above, right) before disassembling remainder of crate.

3 Remove Screws from Front and Back of Crate With all contents removed from the shelf on top of the mill, and with the shelf itself removed, use a power screwdriver to remove the screws from the front and back of the crate. Remove front and back walls of crate BEFORE removing sides.

3 - Remove front and back of crate BEFORE removing sides. Set aside screws for when you need to re-use the crate.

Uncrating & Set-Up cont'd.

4 Remove Screws from side walls of crate Remove screws from the side walls of the crate last of all, and remove side walls of crate. Set all screws aside for future use (explained in Step 6).

5 Remove Zip-Ties from Base of Mill Zip ties attach the mill to the base of the crate. Cut these with scissors to remove the mill from its crate.

6 Crate Storage It is recommended that you store the crate for future use, if you should need to ship your mill. To store the crate, lay the boards flat on a level surface in a dry location. Boards will warp if stored in a moist, unlevel environment. Also store the screws that came with the crate for re-use (longer screws for sides, front and back; shorter screws for top of crate)

7 Crate Re-assembly Should you need to re-use this crate when you are shipping the mill (recommended), follow these directions in reverse. Take care to attach the mill to the crate base with zip ties first, attach the sides of the crate before the front and back (longer screws for crate sides), re-install the shelf, and screw down the top (use shorter screws for crate top).

Topics Covered in this Chapter:

A Location for your Mill

Locate your mill in a temperature-controlled environment that does not fluctuate (i.e. do not locate it under a heating or air-conditioning vent). An unheated space is NOT appropriate for the mill. A solid surface of at least 30" X 36" that is anchored to the wall, such as a countertop, works best.

Important Warnings

Do not install Matrix or any other software on your mill P.C. Any software that draws on the computer's RAM or takes up hard drive space can interrupt communications between the mill and the control computer, which can cause the mill to "stutter" during the job and lose steps, throw communications errors, or pause when it shouldn't.

Similarly, do NOT connect the Revo Computer to the internet, or to a network on which other computers have access to the internet. This can also cause the communication errors described above, and if the mill should acquire a virus, this could void the warranty that accompanies your mill at the time of purchase. Although it would sound sensible to put antivirus software on your mill, we also advise against this. Software which attempts to connect to the internet to regularly update or run virus scans at regular intervals can interrupt communication between the mill and its control computer, which can negatively affect the quality of the parts being milled.

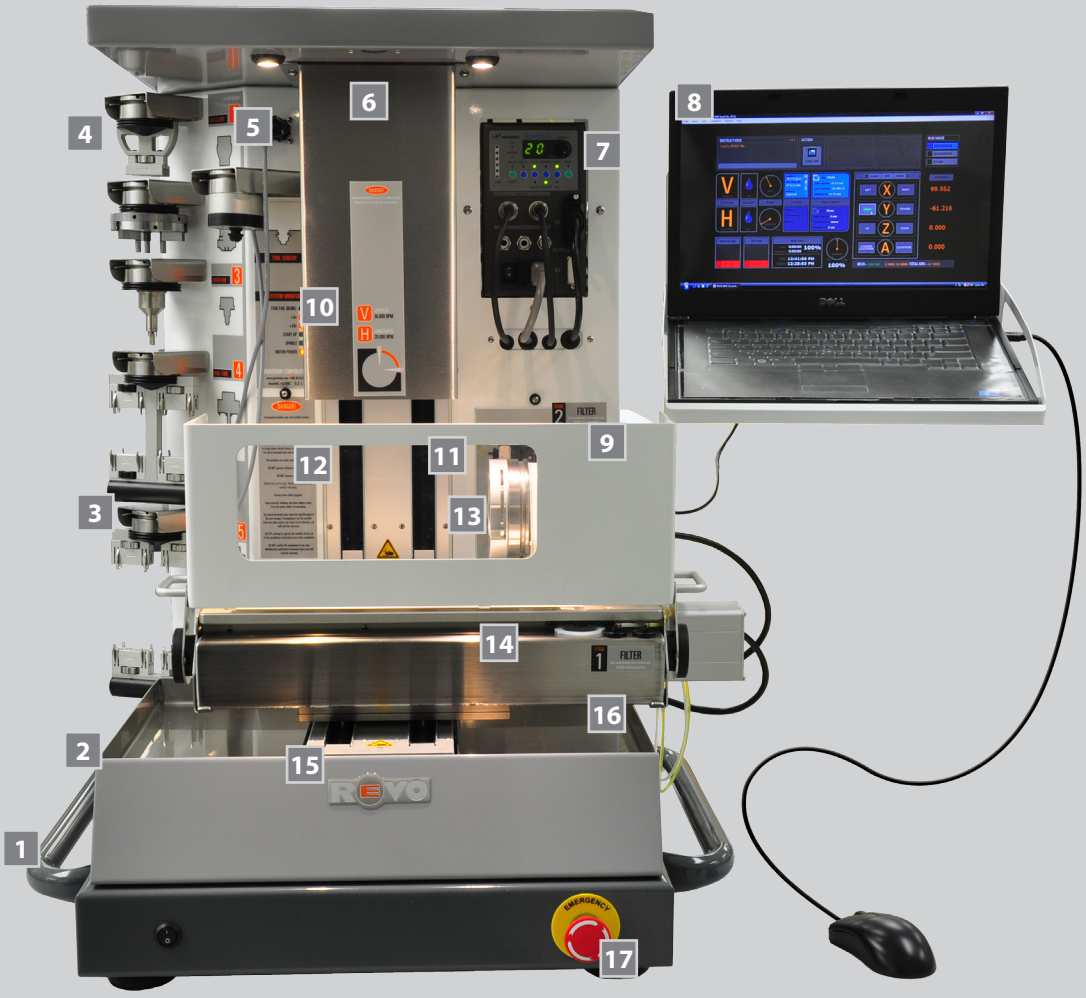
If you are upgrading from the regular version of Matrix to the Mill Version, you'll need to contact Gemvision Technical Support to request an update for your Matrix dongle. This update only takes moments. If you purchased the Mill and Matrix together, this will not be necessary.

It IS safe to leave your mill and mill computer on when it is not in use. Turn it off if you will be away from it for an extended period (longer than a weekend) or during a severe lightning storm, which can damage the mill despite running it with a UPS battery backup.

Assembly, Parts & Features...

Revo 540 Parts & Accessories

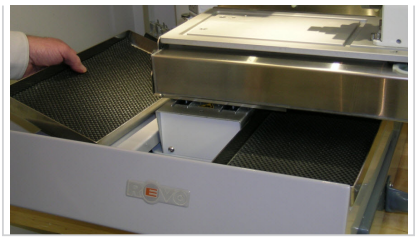
The following diagram highlights the important parts & accessories included with the Revo 540C mill. Each part is described with an overview of its use.



1 - Handles for carrying
2 - Drip trays
3 - Removable, tilting splash guard
4 - Fixtures with storage mounts
5 - Tool sensor with storage mount
6 - Worklight
7 - NSK dual spindle control unit
8 - Laptop mount with laptop
9 - Dual filters with removable cover
10 - Spindle housing
11 - Dual spindles with cutters installed
12 - Z axis (up and down movement)
13 - A axis (rotational movement & fixture mount)
14 - X axis (left and right movement)
15 - Y axis (forward and backward movement)
16 - Coolant reservoir with filters & tubes for coolant recycling
17 - Emergency stop (E-Stop) button

Revo 540C Parts and Fixtures An Overview cont'd.

Important parts & accessories included with the mill are described here.



2 - (Left) Slide out main tray & remove tool trays for cleaning. 3- (Above and Right) Slide on splash guard before a job.

1 - Handles for Carrying The mill is only to be carried, by at least two adults, using the handles on both sides of its base. Remove all possible fixtures to avoid injury when doing so.

2 - Drip trays Slide main tray and/or two smaller tool trays out away from mill, as shown in illustration (above, left) for cleaning.

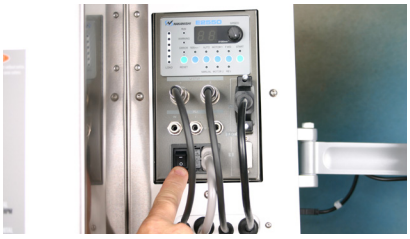
3 - Tiltable Splash Guard To install, slide onto mill table. Remove, or tilt down, to clean.

4 - Fixtures with mounts All five fixtures are designed to mount to the mill frame for easy storage. When not in use, slide each fixture onto its storage mount. Each mount is labeled.

5 - Tool Sensor with Storage Mount Similarly, slide the tool sensor onto its storage mount when not in use.

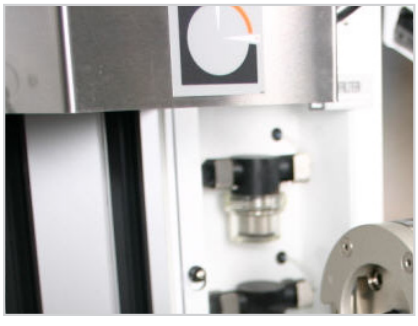
6 - Work Light LED light illuminates the workspace of the mill.

7 - NSK Dual Spindle Control Vertical and Horizontal spindles are controlled by this unit and powered on here.

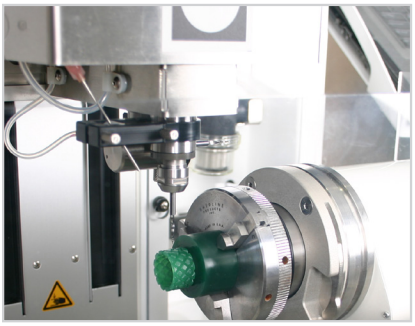


8 - Laptop Arm, Laptop Complete installation instructions for the laptop arm and laptop computer accompanying the mill follow this section of the manual. IMPORTANT: For best results, ALWAYS protect laptop keyboard with the plastic keyboard protector included with the mill.

9 - Dual Filters with Removable Cover The dual filters - one for each spindle - are housed behind a removable Lexan cover. To clean the filters, slide the Lexan up and over the screws to remove it; slide each filter jar up and over its mount and unscrew to remove for cleaning.

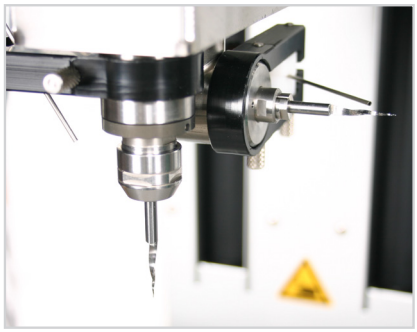


10 - Spindle Cover The spindles retreat up into the Spindle Cover when the mill is returned to the Home position, as pictured above. This position represents the furthest extent of each axis AWAY from the work area. This protects both the cutters and the mill operator. While cutting a part, the Z axis lowers the spindles down out of the cover and into cutting position (pictured above). When the mill is running, it is VERY important to keep all body parts and other foreign objects out of the path of the cutters!

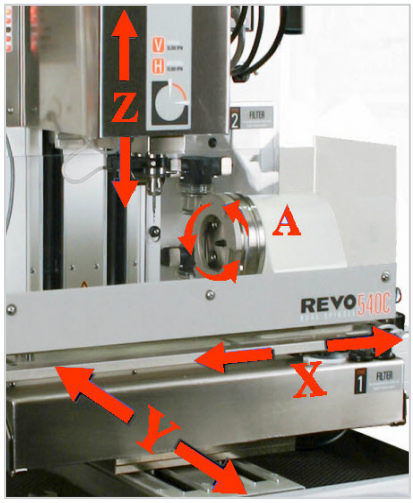


Revo 540C Parts and Fixtures An Overview cont'd.

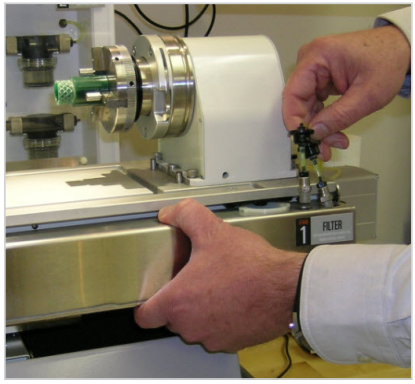
Important parts & accessories included with the mill are described here.



11 - Dual Spindles The vertical and horizontal spindles consist of a shaft, collet, spindle cap (in the case of the Vertical spindle only) and cutter - when it is installed. At times, you may be required to install a callibration pin (installed in the spindle during shipping) instead of a cutter. For this reason, be sure to keep the callibration pins that were shipped with the mill. NEVER tighten the collet without first having a cutter or calibration pin inserted. Coolant needles with mounts are installed on each spindle using clamps and thumb screws. They should be aimed so that the coolant hits the part of the model currently being cut in order to prevent wax debris (swarf) from melting back onto the part and causing undesired results. Full instructions to install and use the coolant system are covered later in this chapter.



12 through 15 - X, Y, Z and A Axes The axes move the spindles (Z) and the model (X, Y, and A) to position the wax so that it is in the proper relationship to each cutter so the model can be cut accurately from the wax. The toolpath (created in Matrix) directs the mill to move each axis to properly align the cutters and the model. The Z axis (#12 in original diagram) moves the cutters up and down. This is the only axis that moves the cutters: the remainder of the axes move the fixture holding the wax to position it correctly for the cut. The A axis (#13 in illustration) holds the fixture and rotates it so the wax is at the proper orientation to the cutters. The X axis (#14) is the axis on which the A axis is mounted, and it moves left and right to position the A axis and the model correctly for the cut. Finally, the Y axis (#15) is the axis on which the X axis is mounted, and it moves toward or away from the mill in order to get the X and A axes in proper alignment for the cutters. All four axes work together by following the toolpath instructions to move the wax toward & away from (Y), left and right (X), and properly rotated (A); and to move the cutters up and down (Z) so that the model will be produced accurately.



16 - Coolant Reservoir, Filters & Tubes Fill the coolant reservoir with coolant each and every time you run a mill job, to ensure the wax is properly cooled and doesn't heat up due to the friction of the cutters. This causes the wax debris (swarf) to melt and adhere to the part, which can cause unexpected results. A full description of installation, use, and maintenance of the coolant system is explained in a future section of this chapter.

17 - Emergency Stop (E-Stop) Push in the E-Stop (Emergency Stop) to stop the mill if there is an emergency, such as the cutter colliding with another part of the mill, or a power outage, lightning storm, or earthquake preventing proper operating of the mill. Pushing the E-Stop will stop all systems (including stopping the cutters from spinning) and therefore may result in a broken cutter. Follow directions later in this chapter to learn how to tell if the cutter was broken, and to learn how to re-start a mill job after it has been stopped. To Release the E-Stop, turn it towards the right: the direction indicated by the arrows on the button. This will release the Emergency stop (systems will STILL REMAIN STOPPED) so you can return the mill to safety (to the Home position), a process described later.

Revo 540C Parts and Fixtures An Overview cont'd.

Important parts & accessories included with the mill are described here.

Digital Calipers Used for measuring the wax calibration model and useful for checking the accuracy of your finished waxes, a set of digital calipers are included.

5/64" & 3/16" Allen Wrenches
You'll use the 3/16" Allen Wrench right off the bat to attach the laptop mount to the mill (full instructions follow). The 5/64" Allen Wrench opens the Coolant Pump Access Panel should you need to trouble-shoot the coolant pump (but ONLY do so at the recommendation of Gemvision Technical Support personnel).

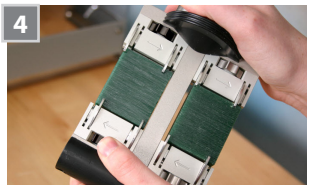
Spindle Tools These include three 1/8" X 1 1/2 " Pins; two 12 / 14 mm wrenches used to install Vertical Cutters; one 5.5 / 7 mm wrench and one 5 / 8 mm wrench used to install Horizontal Cutters. Proper use of these tools for cutter installation, removal, and maintenance is explained below.

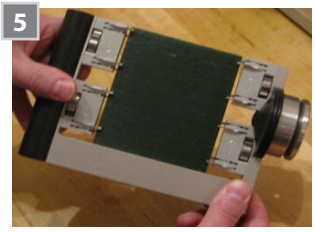
Wash Bottle & Spartan Coolant
Synspar coolant must be diluted with a 20:1 distilled water mixture before it may be used. Fill the wash bottle with this mixture if you wish to wash the swarf (wax debris) off the model while it is being milled. (It's no fun to wait for the swarf to fall off when you want to check out your model during milling.)

Coolant Nozzles (2) Installation of coolant needles is explained later in this chapter. Two are included: one each for the Vertical and Horizontal spindles. Brackets with thumb screws attach them to spindles.

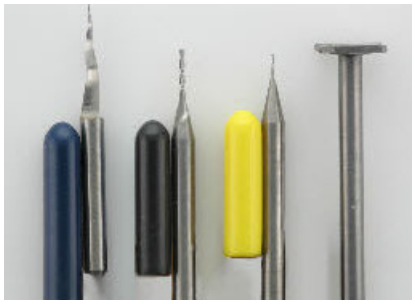
Fixtures Five fixtures, which are best stored by mounting them to the mill frame beside the label indicating which is which, are included with the mill. Proper use, installation, and wax installation for each fixture is explained in this chapter.
- Fixture 1: Base Clamp (with 10mm adapter shown on right)
- Fixture 2: 3-Jaw Chuck (includes two dowel pins for assembly)
- Fixture 3: Center Hub Fixture (includes cap, screw and two washers)
- Fixture 4: Dual 3-sided Fixture
- Fixture 5: XL Flat Fixture

10mm Base Clamp Adapter





Wax Blanks Each mill includes a starter kit of wax containing:
- 12 pieces of Base Clamp Wax
- 1 round tube wax (1 5/16" Outside Diam. X 7/16" Inside Diam. X 6" L)
- 12 pieces of Dual 3-sided wax (10 X 46X42 mm)
- 6 pieces of XL Flat Wax (105 X 95 X 10 mm)



Cutters: Each mill also contains a starter kit of cutters, including:
- five 10 degree tapered cutters - *dark blue cap*
- two 0.032"tip straight cutters - *black cap*
- one 0.016" tip straight cutter - *yellow cap*
- one 12 mm hollowing cutter

Note: To order additional wax or cutters, visit http://www.gemvisionstore.com.

Tool Sensor This is shipped already installed on the mill, and should be removed from its packaging and mounted to the mill frame in its position beside the fixtures when it is not in use. When and how to use the tool sensor, as well as the different positions in which it should be installed, are explained in this chapter.

Tutorial 2: Assembling & Powering Up the Mill

With the mill uncrated and set on a sturdy table or countertop, install the laptop arm, connect the laptop, & plug it into a UPS device.

To set up your mill for the first time, begin by attaching the laptop arm to the side of the mill using the Allen wrench included with your mill. Then, set the laptop on the laptop mount and connect it in three locations (laptop to mill USB communication cable, laptop power, USB mouse). BEFORE running your mill for the FIRST TIME and EVERY TIME, connect it to a UPS (Uninterruptable Power Supply) with a minimum of 500 VA (Volt Amps) and a surge protector. Also, for best results, do NOT have your mill on the same circuit with or nearby other large electrical equipment that produces interference. (For your reference, the mill draws 3 amps of power.)

Materials Needed

Included with Mill:

- **Laptop Mount**
- **Laptop**
- **USB Mouse**
- **3/16" Allen Wrench**

Not Included with Mill:

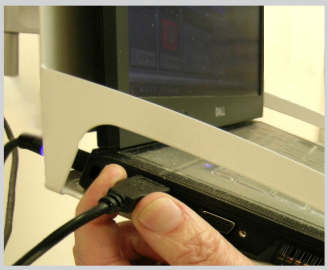
- **Minimum 550 Volt Amps UPS (Uninterruptable Power Supply) with Battery Backup and Surge Protection (4 outlets)**

1 Install Laptop Mount

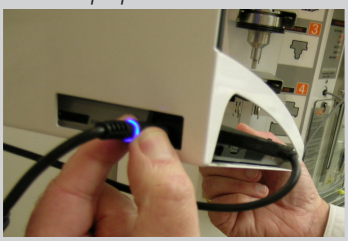
To begin installing laptop, attach laptop mount to the mill frame (WITHOUT the laptop on it) using the 3/16 " Allen Wrench included with the mill.

2 Set Laptop on Mount and Connect Laptop

Set the laptop on the laptop arm and connect the USB cable from the mill to the back of the laptop (at right, above). Also connect the laptop power cord to the back of the laptop (at right, below).



Above: Mill communication USB.
Below: Laptop Power cable.



3 Attach USB Mouse

Attach the mouse, which is included with the mill, to the USB connection found on the right-hand side of the mill. Important: For best results, ALWAYS protect laptop keyboard with the plastic keyboard protector included with the mill.

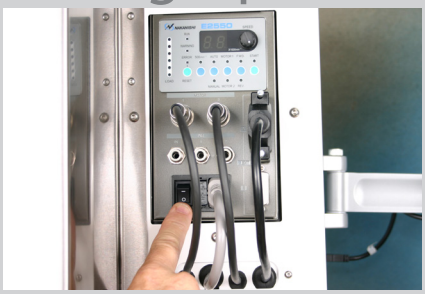
4 BEFORE Powering Up, Connect Mill to UPS

FIRST, connect your mill to an uninterruptable power supply with battery back-up and surge protection, and connect this to a power outlet. The minimum requirements for this device are 550 VA (Volt Amps) and at least 4 sockets with battery back-up AND surge protection (of course, plug the mill and its components into THESE outlets only). Important: It is NOT recommended to run your mill without a UPS device. This device helps protect the electrical components of the mill from energy surges, which can damage them, and it ensures the mill can keep running for several minutes after the power goes out, which gives you time to shut it down safely and properly so you can resume your mill job again later (a process explained later in this chapter).

Warning: Even with a UPS device, for best results DO NOT run your mill on the same circuit with other large, power-draining electrical machinery, and try not to locate it within range of machines that emit high levels of electronic interference, which can also interrupt communications between the mill and the mill computer, which may cause errors in finished parts.

Assembling & Powering Up the Mill cont'd.

5 Power On the Mill With the mill plugged into a UPS device that is connected to a power outlet, you may power on the mill. Power on the mill in three locations: first, toggle on the switch on the front, left-hand side of the mill frame.

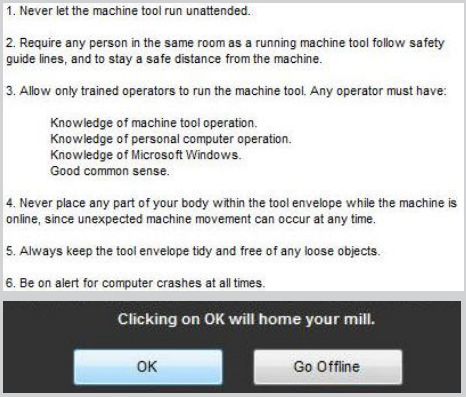


6 Power Up the Spindle Control Unit Next, power on the NSK dual spindle control unit by toggling on the switch on the front of the unit. Leave these controls in their factory settings (Indicator lights are "Auto, Motor 1, Forward" and "30" (in thousands of Revo) for Vertical and "20" for Horizontal spindles.

7 Power on the laptop Press and hold the power button on the laptop until it starts up. Note: you may leave these components ON overnight or over the weekend. Turn OFF the mill in all 3 locations when you will be away from the mill for a longer period or if there is a lightning storm.

8 Double-click the Revo Software Icon On the desktop of the Revo computer you'll see the red icon for the Revo 540C software. Double-click it to start the Revo (Revo Project Management) software. Upon starting up the software, a safety warning will appear. Important: Read this safety warning carefully before accepting the message, because accepting it will cause the mill to move to its "Home" position.

NEVER impede the travel of the axes when they are in motion. Doing so can damage the mill. "Homing" the mill moves each of its axes to the furthest extent it can go away from the working envelope of the mill (where the wax is cut). For safety, the mill is shipped with the spindles lowered down against the table, protected by a film case and pressed against a stack of cardboard. The first time you turn on the mill, accept this message, and "Home" the mill, these pieces will shift and fall away. Do not be alarmed. You may discard them, unless you wish to save them for safe mill shipping in the future.



Read safety message carefully the first time and every time; click "OK" to bring mill online and establish connection to mill computer.

9 Stopping the Mill in Case of Emergency If there is an emergency while the mill is running, press the E-Stop button on the front of the mill (above) or press any key on the laptop keyboard to stop the mill. To release the E-Stop, turn it to the right (above, right): the direction indicated by the arrows on the button. You CANNOT bring the mill back online without releasing the E-Stop button. However, systems will NOT be re-started upon releasing the button. After the button is released, you'll need to use the Revo software to prompt the mill to go back online ("Status" > "Go Online"). This will return the mill to its "Home" position, moving the cutters to safety. The Revo software is fully explained later on in this chapter, including directions to return the mill to the "Home" position and to "Jog" or move the axes to safety. Remember: it is VERY important to read this instruction manual thoroughly before operating the mill.

Important Note About using the E-Stop: Push the E-Stop in to stop the mill in case of an emergency. This will stop all systems (including stopping the cutters from spinning) and therefore may result in a broken cutter. Follow directions later on in this chapter to learn how to tell if the cutter was broken, as well as how to re-start a mill job after it has been stopped.

Tutorial 3: Use & Maintain the Coolant System

Learn to install, use, and maintain the various components of the coolant system.

To assemble the coolant system, install each coolant needle on its spindle. With each use, fill the coolant reservoir (#1 in illustration at left) with coolant to ensure the wax is properly cooled and doesn't heat up too much with the friction of the cutters. This would cause the wax debris (swarf) to build up on the part, which can cause unexpected results. The pump (#2) pulls coolant out of the reservoir up the tubes, through two sets of filters, and expels it through the coolant needles (#3) onto the model. Carefully aim the coolant needles to ensure the coolant hits the part of the model currently being cut. Coolant is caught by the "moat" on the table so it can run back down into the reservoir and be recycled through a system of filters (one on the intake for the used coolant, two on the ends of the tubes leaving the coolant reservoir, and two main filters in the dual pump system).

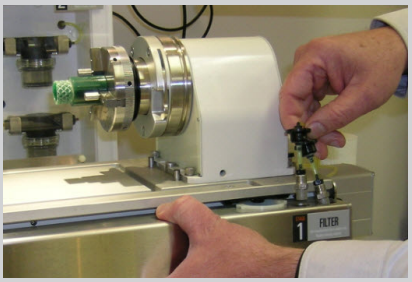
Materials Needed

Included with Mill:
- **2 Coolant Needles & Brackets**
- **Splash Guard**
- **Synspar Coolant**
- **Wash Bottle for Diluted Coolant**

Not Included with Mill:
- **Distilled water**

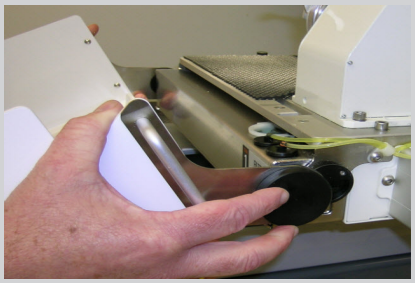
1 To assemble, install Coolant Needles on both Spindles

To assemble the coolant system and before milling your first model, install each coolant needle, with its bracket and thumb screw, onto each spindle (Vertical shown at left, Horizontal shown at right). To assemble, install needle in bracket; slide bracket over spindle onto shaft; and tighten thumb screws. MAKE SURE THEY ARE TIGHT, or else the bracket could slip down during milling and become damaged, which can also break the cutter and damage your model. IMPORTANT: Aim the coolant needles at the tip of each cutter so coolant flows onto the part of the model currently being cut. Check position throughout milling.



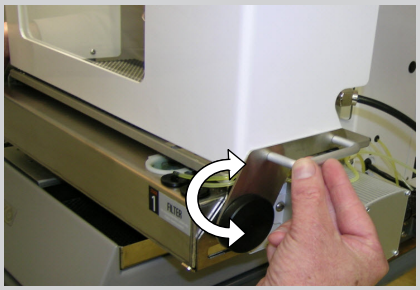
2 Before & during every job, check that tank is full

Pour diluted coolant mixture into reservoir and check that levels are high before EVERY job. Periodically check that they remain high and refill as necessary throughout the entire job, especially during long jobs. About once a week, empty and fully clean coolant tank.



3 Slide on Splash Guard

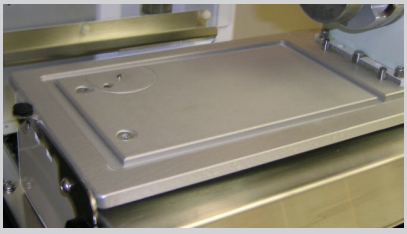
Before running a part on the mill, the final recommended step after checking the coolant levels is to install the Splash Guard. This protects other parts of the mill, and the mill's surrounding environment, from coolant drips. As shown in the lower right-hand corner of the image above, sliding on the Splash Guard by it's hinge fixture until it clicks in place. Your splash guard manufactured of a sturdy material and tilts out for easy cleaning.



Coolant System cont'd.

4 Caring for small & large filters

Coolant is pumped out of the reservoir; travels up the tubes through two smaller filters (shown above) which catch wax debris; passes through two larger filters in the dual pump system (shown above, right); and is expelled from the coolant needles onto the part as it is being cut. To ensure smaller filters do not plug up with wax, clean them after every job by blowing air through them in the reverse direction from the coolant flow. The larger filters - one for each spindle - are housed behind a removable Lexan cover (shown above and at right). To clean the filters, slide the Lexan up and over the screws to remove it (above, right); slide each filter jar up and over its mount (center, right) and unscrew to remove for cleaning (below, right). To clean, remove and rinse filters. This should only be accomplished on an as needed basis.



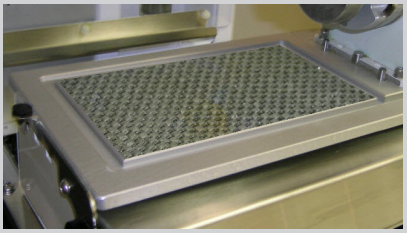
5 Moat in table for recycling coolant

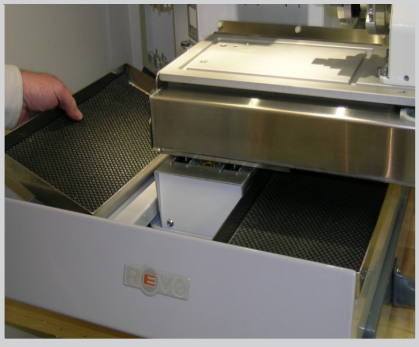
Coolant drips onto model and is collected in the coolant moat on the mill's table. It drips through another filter into the coolant reservoir, where it is recycled by the pump and filtration system for re-use. How environmentally friendly of the Revo! Clean the filter between the moat and the reservoir frequently, as this is where wax can get backed up and cause the coolant moat to overflow, reducing the amount of coolant being recycled and creating one very drippy mill table.

Note: As shown at right, some users place a piece of felt or foam pad on the mill table to further filter coolant as it enters moat.

6 Maintenance of Coolant System: Every Use

Check coolant levels before each and every job. For especially long jobs, check coolant levels throughout the job to ensure you don't run out of coolant or have overflow in the coolant moat that can drip onto other parts of the mill or the mill's surroundings. Also, after each and every time you use the mill, clean up any and all swarf (wax pieces) and coolant spills that land on your mill. Several Lexan "guards" are included to protect the axes: remove these, clean them off, and replace them, along with the Splash Guard, after EVERY use. It is ESPECIALLY IMPORTANT that coolant and swarf do not leak into any of the axis housings and affect the axis controls. Coolant overflow from the moat can cause this, so clean the filter between the moat and the reservoir, and the filters on the tubes exiting the reservoir, after every use. For general cleanliness, it is also a good idea after every use to remove the drip trays (shown below), wash them off, and wipe up the mill base underneath them, to keep built-up coolant and swarf from collecting there (this creates a gross, slimy build-up that can harm the protective coating on your mill).





7 Maintenance of Coolant System: Once Monthly

At least once a week, fully remove the coolant reservoir, dump out all coolant, and clean reservoir thoroughly. As described above, you may disassemble the larger filters and rinse them out as well, but do so only as needed.

Tutorial 4: Use & Maintain Cutters & Spindles

Learn about each cutter and when to use it; and learn the parts of the spindle and how to install and measure cutters.

L - R: 6-dgr tapered, 0.032" straight, 10-dgr tapered, 0.016" straight, 12 mm hollow.

Toolpaths are designed to use ONLY the cutter(s) specified when preparing the project in the Matrix Mill Builder. This section therefore begins by defining each cutter and its proper use. Learn how to install a cutter in the vertical and horizontal spindles, how to measure a cutter during part set-up with the Tool Point Sensor, and the proper care and maintenance of the spindles and cutters. All Gemvision cutters have a helical shape to draw wax debris away from the part. Cutters are straight, tapered, and circular for different uses.

Materials Needed

Included with Mill:

- **5 - 10-dgr tapered cutters**
- **2 - 0.032" straight cutters**
- **1 - 0.016" straight cutter**
- **1 - 12 mm hollowing cutter**
- **Spindle tools (four wrenches for cutter installation)**

Not Included with Mill:

- **6-dgr tapered cutter**

1 About each cutter (L-R in illustration at right)

10-Degree Cutter (dark blue cap): The 10-degree cutter has the longest cut length, at 16-mm, of any of the cutters. It is therefore the cutter you'll choose the most, since it can cut through more "rough" or uncut wax (up to 15 mm deep without breaking) than the others. It also offers great part definition thanks to its tiny tip: just 0.004" wide. Just note: this cutter has a 5-degree taper on either side (for a total of 10 degrees of taper), and therefore produces ever-so-slightly tapered sides even on straight-sided models. Use other cutters described below and software techniques in "Detailed Options" in Matrix if you need to add a "finishing pass" to achieve straight sides (see Ch 3).

6-Degree Cutter (light blue cap): This cutter is for secondary passes ONLY. Although it has a "cut length" of 16 mm, it can only cut to 2-mm deep and it therefore is not rigid enough to cut uncut wax. Because it only has a 3-degree taper on each side, it can be used to reach into tight spots between details where the 10 deg. cutter cannot reach. But beware, as it is a very fragile cutter.

0.032" Cutter (black cap): This straight cutter can cut from uncut wax but its cut length is only 8 mm, so it can't cut as deep as the 10-degree cutter. It is used most often for flat bezels and inlay work because it can create straight sides where a tapered cutter cannot. Yet its tip, at just under 1 mm diameter, is not as small as the 10 degree cutter's tip, so it cannot cut in as tight spots.

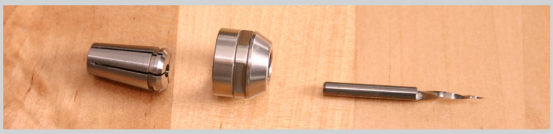
0.016" Cutter (yellow cap): This straight cutter is for secondary passes ONLY, because it has just a 3-mm cut length and cannot move sideways

through uncut wax. It is intended to be used after one of the tapered cutters to produce straight sides on a model. Also, it requires at least 0.5 mm of space between details to fit in and complete clean-up work.

12 mm hollowing cutter: This cutter should not plunge deeply into rough (uncut) wax; therefore, during set-up in the Matrix Mill Builder, a cut-out curve is automatically inserted before the path for this cutter, removing rough wax so it can reach its intended position. It is used to create the interior cut on a hollowed or comfort-fit model, or to contour (finish with straight sides) the inside of the ring.

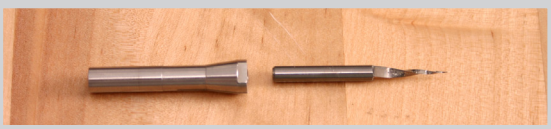
2 Parts of Vertical Spindle

The Vertical spindle consists of (left to right in illustration at right, above) a collet, which holds the cutter, and a spindle cap, which is used to secure and protect the collet. The cutter (or a callibration pin, which is installed when the mill is shipped) fits into the collet. Note: Store all callibration pins in a safe place, as these may be needed for calibration in case Gemvision Technical Support instructs you to do so.

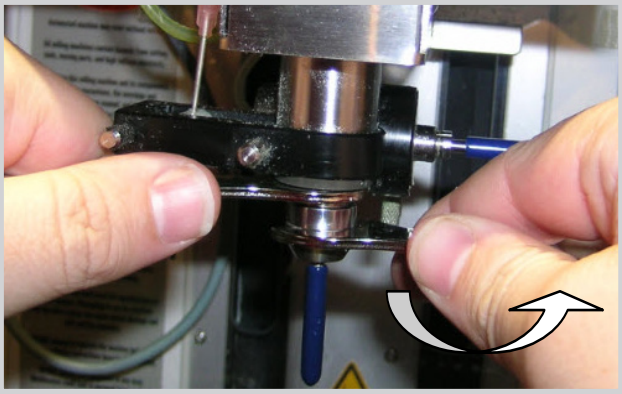


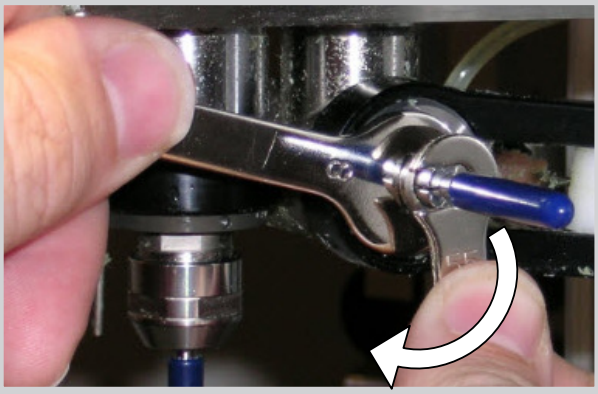
3 Parts of the Horizontal Spindle

The horizontal spindle consists of the collet only. The cutter or callibration pin fits into this directly, without the need for a spindle cap.



Cutters & Spindles cont'd.

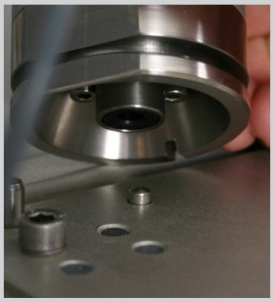




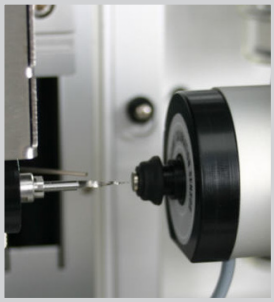
4 To Install and Remove Vertical Cutter During set-up of a mill job, the software on the Revo computer will prompt you to install the cutter you selected when preparing the Revo file in the Matrix Mill C Builder. When prompted to do so, place cutter in spindle cap and hand-tighten. Then, grip spindle cap with wrenches as pictured above.
To tighten: Push wrenches away from each other.
To loosen and remove cutter: Pull wrenches toward each other. When you feel it loosen, untwist cap by hand just until you can remove cutter. Always have the rubber caps on your cutters when adding or removing them from the mill.

5 To Install and Remove Horizontal Cutter Insert cutter and grip collet with the two smaller wrenches.
To tighten: Turn the smaller, outside wrench clockwise.
To loosen: Turn the smaller wrench counter clockwise.

Note: We recommend placing a sponge or another soft object under each cutter during installation and removal to help prevent it from breaking should it fall.



6 Tool Sensor Position, Vertical After the Revo software prompts you to install each cutter, it will need to measure the cutters with the tool sensor so that the mill can calculate the correct position at which to cut the part. To measure the vertical cutter when prompted to, align the notch on the bottom of the tool sensor to the pin on the mill's table to center the tool sensor under the cutter (at left, above). "Confirm" in the software that you've done so; then, instruct the mill to "Move", and it will automatically measure the cutter (at right, above).



7 Tool Sensor Position, Horizontal Install the tool sensor in the A axis the way you do the other fixtures: align the flat part of the chuck with the flat part of the puck (receiving piece on the A axis) and lower the clamps on the A axis to secure it in place. As in step 6, "Confirm" it is installed and then instruct the mill to "Move". The mill will position itself to measure the cutter against the tool sensor.

Tutorial 5: Installing Fixtures & Wax

Each fixture is easily installed onto the A axis without the need for tools. Learn to install wax into each fixture, as well.

When you start a job on the Revo C mill, the software will request a wax blank and fixture corresponding to the strategy you chose in the Matrix Mill Builder when creating the toolpath. Each fixture is numbered, and each is just as easy as the next to install on the A axis: simply line up the flat side of the fixture chuck with the flat side of the accepting piece ("puck") on the A axis. The fixture will click into place. Then, lower the A axis clamps to secure the fixture to the axis. The proper procedure for installing wax in each fixture is explained below:

Materials Needed

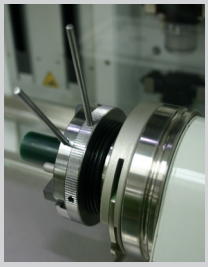
Five Fixtures:

- **#1 - Base Clamp**
- **#2 - Three-Jaw Chuck**
- **#3 - Hub Fixture**
- **#4 - Dual Fixture**
- **#5 - XL Fixture**

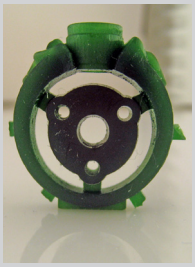
Wax Blanks included with mill:

- **12 pieces of Base Clamp wax**
- **1 round tube wax (1 5/16" OD, 7/16" ID, 6"L)**
- **12 pieces for Dual (10 mm)**
- **6 pieces for XL (10 mm)**

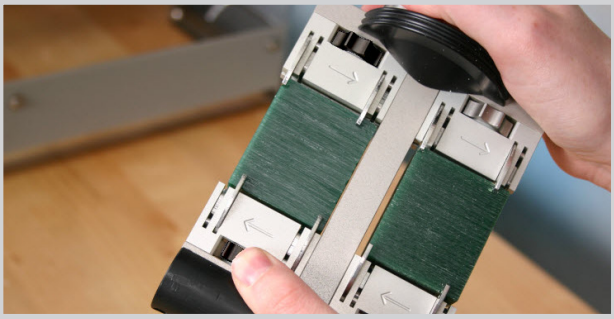
1 Base Clamp Slide grooved wax into fixture and tighten screw in direction of arrows. DO NOT overtighten. For the 10mm wide wax, first insert the adapter, then the 10mm wax blank.



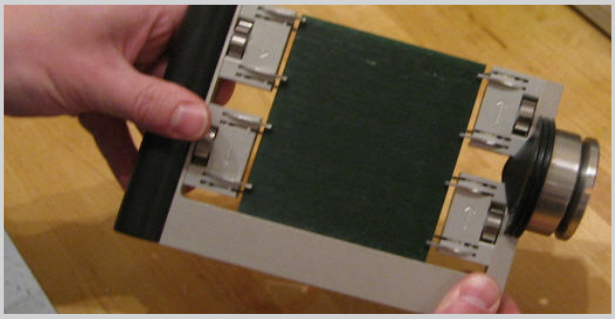
2 Three-Jaw Chuck Turn fixture to tighten jaws around tube wax. Upon installing onto A axis (as described and illustrated above, left), insert dowel pins, (shown above, right) and push in opposite directions to make certain wax is tightly held in fixture and will not move during cutting.



3 Hub When using this fixture, which adds a secondary rotary operation to any flat job, the software will create a wax piece in the center of the ring which secures the model to the hub. Slide the hub over the three pins with the arrow in the wax pointing up and facing you when the fixture is installed on the A axis. Slide the cap over the pins and insert and tighten the screw with its two washers.



4 Dual 3-Sided Fixture. Insert wax blocks between the clamps and turn in direction of arrows to tighten clamps around wax. Clamps are self-centering, so wax will be centered in fixture. In Revo software, black clamp screws indicate "Side 1" of Dual Fixture (closer to mill when fixture is installed and mill is "Home") and silver clamp screws indicate "Side 2" (closer to user).



5 XL Fixture. Insert wax into XL fixture clamps and turn in direction of arrows to tighten clamps around wax. Clamps are self-centering, so wax will be centered in fixture.

Getting Started ...

Revo Software Interface

The Revo software controls the mill and directs set-up and preparation for every mill job. Learn about each part of this software.

About the Revo Software Interface

Note: This section of the manual will review each part of the Revo interface and how to use it to cut your first part: a Wax Calibration model you MUST cut before running any other job on the mill.

The Revo Procedure Manager software (Revo) is already installed on your Revo computer when you receive your mill. A total redesign from the previous Revo control software on earlier models of the mill ("Revo 540A" and "B"), Revo is the software interface that controls the movement of the mill. Its most significant update is that it walks the user step-by-step through the set-up and assembly of the mill necessary to create a part, with clear "Instructions" to perform, "Actions" to direct mill movement and confirm steps are completed, and graphics to display visual representations of each step.

Dropdown Menus

Along the top of the screen are the dropdown menus *File, Status, Seek, Calibrate* and *Help*. Under "File", you may "Open" a Revo file you created in the Matrix Mill C Builder (you may also do so by clicking the Action "Load" when no job is loaded). This initiates the step-by-step "Instructions" section of the interface explained fully below. To close the Revo software, click "File" then"Exit".

The "Status" drop-down menu offers the options *Go Online* or *Go Offline*. If the mill is "offline" - or, not being controlled by the Revo software - click "Go Online" to reconnect the mill and its control computer. This is automatically performed when the Revo software is first opened. Use this in case the mill and its control computer lose communication for any reason.

Click "Go Offline" to take the mill offline. It will no longer listen to the instructions being sent to it by the control software. The best way to stop the mill in case of an emergency, however, is with the large red "Emergency Stop" button on the front of the mill frame, as explained elsewhere in this chapter.

The "Seek" drop down menu offers the options *Seek Home, Homing Report, Seek V Tool* and *Seek H Tool*. "Seek Home" is automatically performed by the mill when the Revo software is first started up, and at the end of each job. This action causes the mill to return to the "0" positions for each of its four axes: X (moves furthest right, when you are facing the mill, that it can go), Y (furthest toward you, when facing the mill), Z (furthest up, so cutters are beneath spindle housing), and A (rotates until it stops). During a Seek Home operation, the spindle will be housed in the spindle cover so the cutters are not exposed. You may wish to perform this action if you have "jogged" (moved) the axes using the Jog Controls (explained in full below), or if the mill has been stopped inadvertently. Further instances when it is important to home the mill are explained later on in this manual. However, the mill "homes" itself, when necessary, during the set-up and preparation of a mill job (as you follow the Instructions and Actions), so it is usually not necessary for you to click this button.

A Guide to the Revo Software Interface

Dropdown Menus are located at the very top of the interface. Click one to see and access its available options.

Instructions walk you through what to do for each step in the set-up and assembly of the mill in order to run a .RVOC file you created in Matrix.

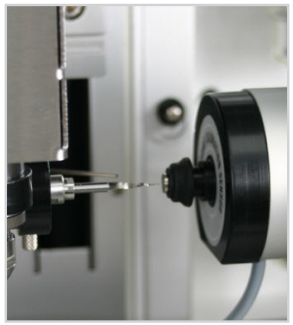
Actions follow each "Instruction". After reading and following the Instruction, click the Action button (e.g., "Move" the mill or "Confirm" that a step has been completed) to proceed to the next step in the Instructions for this job.

Graphics display a visual image of the instructions and actions the mill is directing you to perform, such as loading a cutter, installing the tool sensor, or clearing the mill of all fixtures (below).

"Homing Report" compiles a list of the mill's history of "Seek Home". This information can provides insight into where the mill started and ended at the beginning and ending of each mill job. It is likely you will not have to create a Homing Report unless a Gemvision support technician requests this information.

"Seek V Tool" measures the location of the tip of the cutter loaded in the vertical spindle. "Seek H Tool" measures the location of the tip of the cutter loaded in the horizontal spindle. Clicking either "Seek Tool" option initiates the instructions to position the tool sensor in the appropriate location for the "Seek Tool". When performing a "Seek V Tool" align the notch on the bottom of the tool sensor to the pin on the mill's table to center the tool sensor under the cutter.

When performing a "Seek H Tool" align the flat part of the tool sensor with the receiving piece on the A axis and lower the clamps on the A axis to secure the tool sensor in place (see below). You may wish to Seek Tool if you believe you broke the tip of a cutter. This process is explained later in the manual.



Revo Procedure Manager An Overview

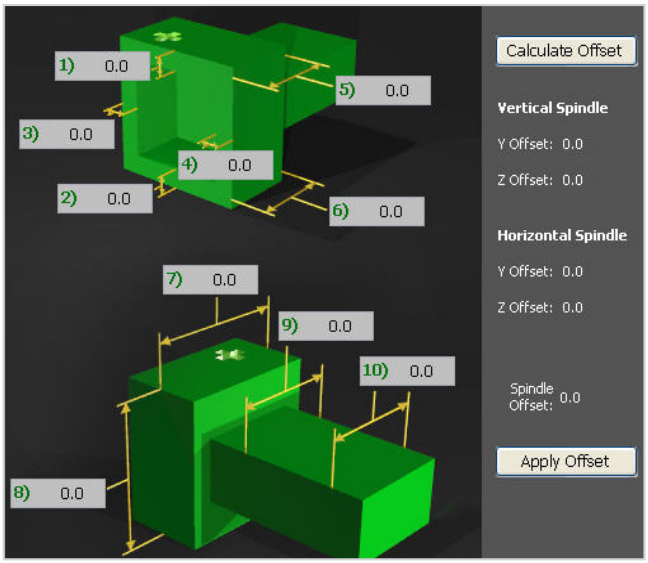
The remaining dropdown menus are explained here.

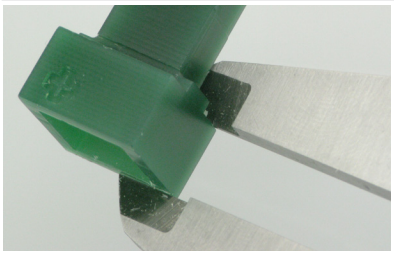
"Calibration" Dropdown Menu The most commonly used tool within the Calibration menu is the Wax Calibration Block. The other options within this menu are used by the mill assembly technicians; valit is unlikely you will have to use these tools.

The "Wax Calibration Block" is the FIRST job you should run upon receiving and assembling your mill, so we'll walk you through this procedure step-by-step later on in this chapter. This process will help your mill detect and correct any shifting that may have occurred in its axes during shipping. Your mill is fully calibrated (using the rest of the tools found in this menu) at Gemvision prior to shipping. However, should any movement occur during shipping, the Wax Calibration Block will help the mill correct these changes.

It's easy to use: you'll set up the fixtures, wax, and cutters indicated to quickly cut a simple block of wax; remove the block from the support materials, and take a few measurements from it using the digital calipers included with the mill.

Enter these measurements into the software, and the mill will be able to correct for any inconsistencies it finds between the actual measurements and the theoretical measurements of the block. Usually, these differences are miniscule and, after the mill has corrected for them, no further adjustments are necessary. You should only have to run the Wax Calibration Block one time,

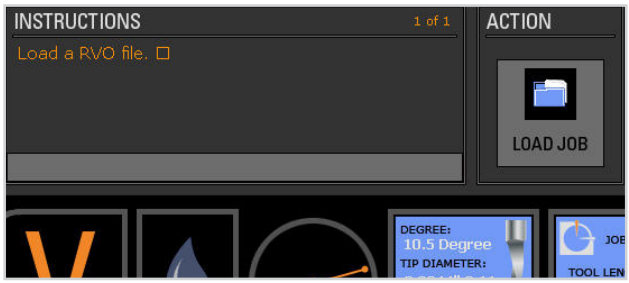




Cut the wax block. As instructed by the software, enter all 10 measurements, noting where the symbol on the block appears in order to correctly orient each measurement.

unless you relocate the mill, ship the mill, replace a mill or have a mill collision, it is recommended that you run this file again to ensure the axes did *not* shift too far during that event.

However, if in the course of running your mill you see inconsistent results, contact Gemvision Technical Support, which may advise you to run this Wax Calibration Block again (or, rarely, other procedures in this menu) to correct for any inconsistencies. Of the remaining calibrations, you should NOT run ANY of these procedures unless you are advised to do so by Gemvision Technical Support. At that time, they will supply you with the necessary instructions to do so.

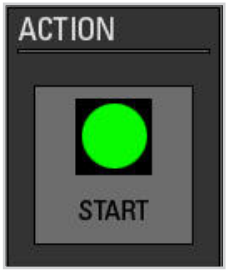


Instructions The "Instructions" section of the interface tells you what to do next. With no job loaded, it enthusiastically prompts you to load a job, since Revo likes to see you making money. Sometimes, it will tell you what the mill is about to do next, so you can make sure all body parts, loose clothing, small children, and animals are out of the way of the cutter. Other times, it will instruct you to remove a cutter, install a cutter, install the tool sensor, place wax into a fixture, or attach a fixture to the A axis.

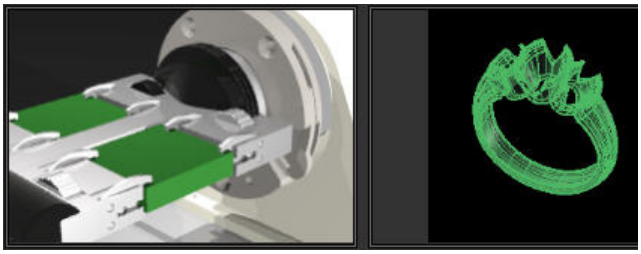
Action The "Action" button indicates to the mill that you've read, understood, and completed the Instructions portion of the interface. If the "Instructions" are urging you vehemently to load a job, the "Action" is "Load Job". This button will take you to the directory of your mill computer so you can select a job to mill. If the "Instructions" are telling you the mill is about to move, the "Action" is "Move". The mill won't move until after you ensure it is safe to do so and click "Move". How courteous of the Revo! At other times, the "Action" will simply be to "Confirm" that you've read and followed the instructions, so you can get to the next step. As you "confirm" each step, you will see other parts of the interface verify this (explained below). Do not lie to the mill, as it cannot detect whether or not you've properly followed the instructions. At other times, "Next" will appear in the "Instructions" interface, allowing you to skip the "Action" (if, for example, the mill thinks you already have the correct cutter installed). However, it is ALWAYS necessary to read and follow all instructions carefully, to ensure safe and proper use of the mill.

Revo Procedure Manager An Overview cont'd.

Each section of the Revo interface is explained here.



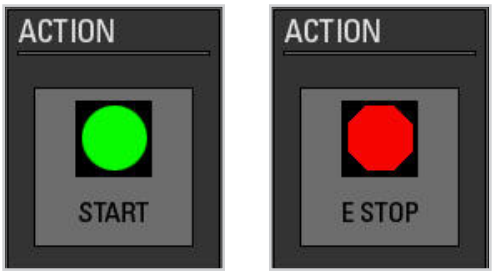
"Start" and "Stop" When you have completed all of the required steps before starting a job, the "Start" action button appears. Click it to start cutting your job. The mill will move into position, and each spindle and its associated coolant pump will turn on automatically when it is needed. After clicking "Start", the "E-Stop" button will appear. Click it, press any key on the keyboard, or use the "Stop" button on the front of the mill if you need to stop the mill for any reason. More information on stopping the mill in the middle of a job is provided later.



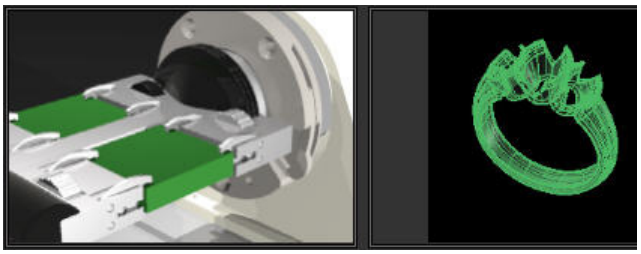
Graphics Upon loading a file, the graphics section displays a representation of the fixture and wax needed to create this model (L) and a preview image of the file (R). This is to help you verify that you've chosen the correct file to mill. Then, as you begin to follow the instructions, the first box will display which object you need to perform the steps indicated - such as a cutter, a fixture, or a piece of wax - and the second one will show the completed step (installed fixture, e.g.). When the mill starts cutting the job, the initial graphics return.

Revo Procedure Manager An Overview cont'd.

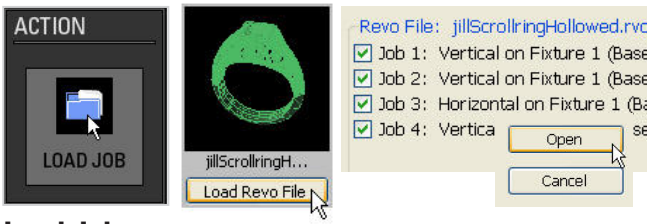
Each section of the Revo interface is explained here.



"Start" and "Stop" When you have completed all of the required steps before starting a job, the "Start" action button appears. Click it to start cutting your job. The mill will move into position, and each spindle and its associated coolant pump will turn on automatically when it is needed. After clicking "Start", the "E-Stop" button will appear. Click it, press any key on the keyboard, or use the "Stop" button on the front of the mill if you need to stop the mill for any reason. More information on stopping the mill in the middle of a job is provided later.



Graphics Upon loading a file, the graphics section displays a representation of the fixture and wax needed to create this model (L) and a preview image of the file (R). This is to help you verify that you've chosen the correct file to mill. Then, as you begin to follow the instructions, the first box will display which object you need to perform the steps indicated - such as a cutter, a fixture, or a piece of wax - and the second one will show the completed step (installed fixture, e.g.). When the mill starts cutting the job, the initial graphics return.

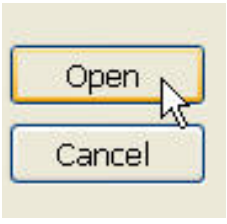


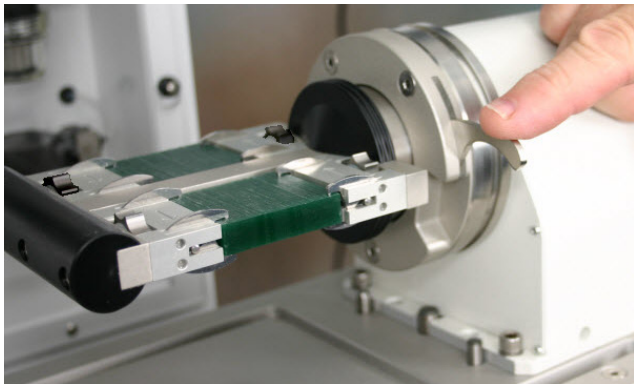
Load Job Click this button and choose a thumbnail image of the part to load, which you'll learn how to create using Matrix in Chapter 2. Upon selecting the thumbnail, a description of each "Job" required to run this part (different cuts performed by the mill) will appear onscreen. A full description of each type of job is explained in Chapter 3. You CAN complete just one part of a job, come back to the mill WITHOUT removing the wax or fixture, and Load just the latter part(s) of the job to cut. However, it is not recommended that you do so. In most cases, make sure ALL parts of the job are selected and click "Open" to load the file.

(Above) Click on a file made for the Dual Fixture and the options "Load Revo File 1" or "2" will appear. Select one.

Two Positions on the Dual Fixture To load a model into either of the two locations on the Dual fixture, click a job to load and the options "Load Revo File 1" and "Load Revo File 2" will appear (for Dual Fixture only). "File 1" runs in "Position 1" on the fixture, with the black clamp screws (closer to the mill when the fixture is installed and the mill is at "home"). "File 2" loads in "Position 2", with the silver clamp screws (closer to the user). Load the same job in both positions, or load a different job in each position, so your mill can complete two jobs with only one set-up operation.

(Above) After selecting a job for position "1" or "2", both positions will be populated in the dialog box onscreen. Click Open to return to the Revo software and continue setting up the mill to run these two files.

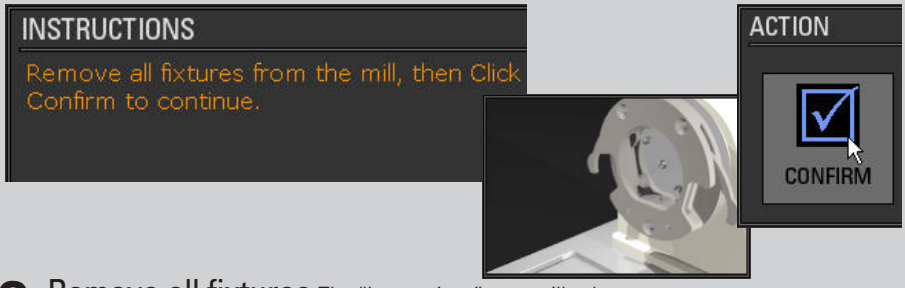




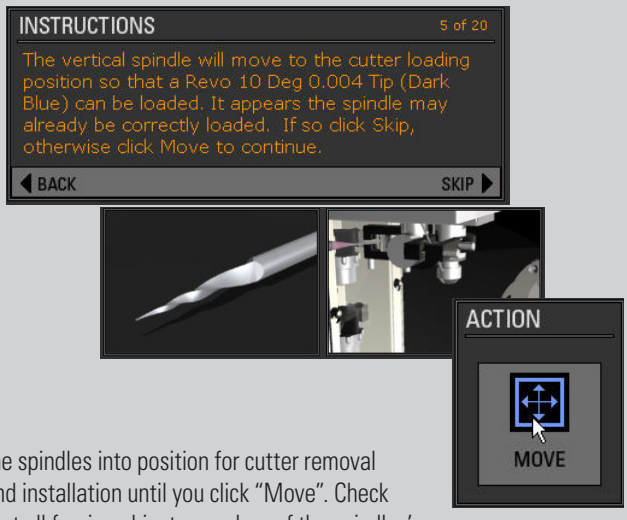
(R) When the Dual Three-Sided Fixture is first installed on the mill (i.e. , the mill is in its Home position),"Position 1" is closest to the mill, with the black clamp screws, and "Position 2" is closest to the user, with the silver clamp screws.

Tutorial 6: Instructions & Actions for a Sample Job

1 "Load Job" The "Load Job" button appears whenever a job has finished and the Revo C is waiting to work on the next task. Click it to navigate the Revo computer for an "RVOC" file you'd like to mill, click the file, and click "Open". Or, go to File > Open to load a new job (this is NOT possible while "Job is Running" appears in Instructions window). Remember: in the case of a file set up for the Dual Fixture, two positions (1 and 2) will appear onscreen. Choose one or both. You may choose the same or different models for each position. .

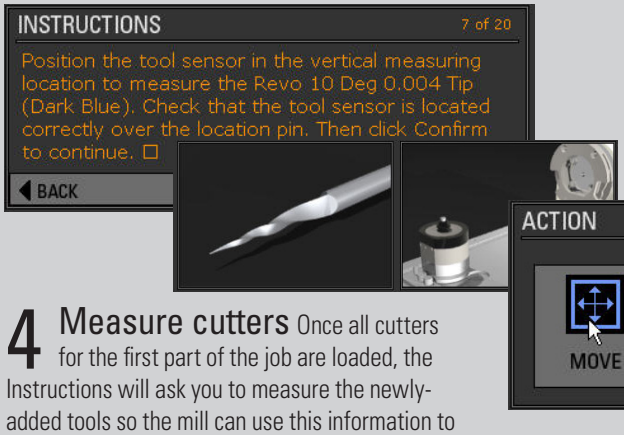


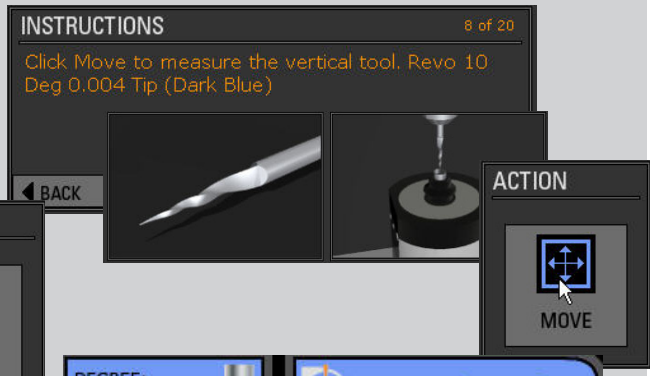
2 Remove all fixtures The "Instructions" area will ask you to remove all fixtures currently attached to the A axis. Do so, and click Confirm to indicate you've done so. Note: When a new job is loaded, the "Wax Blank" and "Fixture" areas of the interface will read "Not Set" until you "Confirm" those steps of the Instructions during set-up. Then, the mill is "trusting" that you did this - it cannot detect that they are correct. Same goes for the Cutter section of the interface: it "remembers" the cutter it told you to load in the previous job, so it may offer you the chance to click "Next" instead of "Move", even if you've removed the cutter yourself in the meantime. It's simply "remembering" the last cutter you confirmed installing. This means that, if you changed or removed cutters at some point (other than at set-up) or installed an incorrect cutter, the mill cannot detect this. It is VERY IMPORTANT, for the success of your parts AND for the safety of your mill, not to "lie" to the mill by clicking "Confirm" when you really haven't completed the task. If you complete the wrong task or "lie" to the mill, it could crash and damage sensitive hardware, damage a cutter, or damage your model.



3 Load correct cutters The Instructions will now walk you through removing the existing cutters in both spindles (Vertical and Horizontal) and installing the correct cutters for the start of the job. For your own safety, the mill will not move the spindles into position for cutter removal and installation until you click "Move". Check that all foreign objects are clear of the spindles' path before you do so. Each time you remove and add a cutter, click "Confirm". The cutter definition portion of the Revo interface will display the cutter to assist you with determining which one to load. Also, the cap color is listed in the Instructions, so be sure to replace the correct cap when the cutters are not in use.

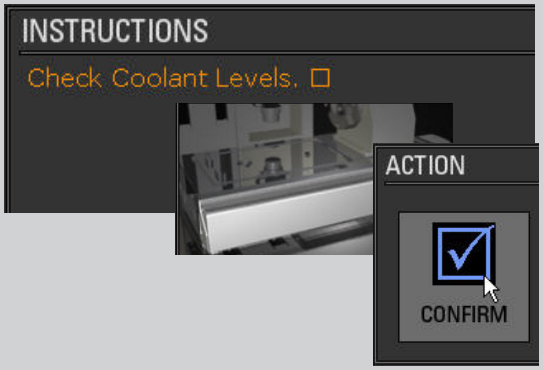
Instructions & Actions for a Sample Job cont'd.





4 Measure cutters Once all cutters for the first part of the job are loaded, the Instructions will ask you to measure the newly-added tools so the mill can use this information to correctly cut your part. To do so when you are prompted by the software, position the tool sensor as indicated for one spindle - then the next - and click "Move" at each location to measure that tool. Tool length will now be displayed.

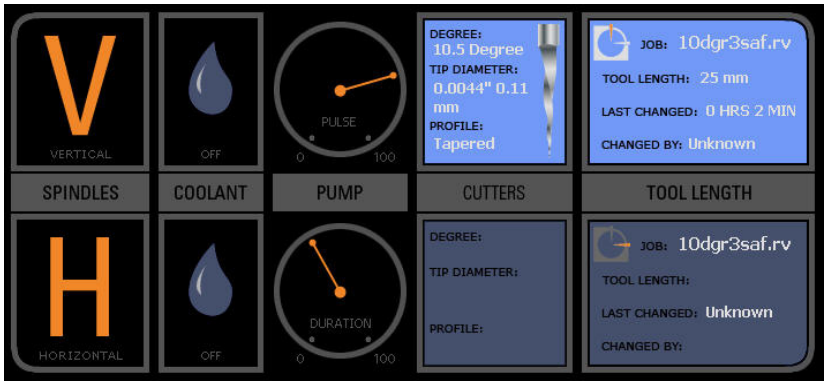
5 Load Wax & Fixtures The last steps instruct you to load wax of the required size into the fixture; and then to load the fixture with its wax onto the mill. Confirm each step. This will cause the "Wax" and "Fixture" displays to indicate that the wax / fixture it told you to install for this job was installed. (Remember: Don't "lie" to the mill.)



6 Check Coolant and Click "Start" With the mill all ready to go, click "Start". Depending on the strategy, the mill may pause after a few minutes (time indicated in "Time Remaining" section of screen) and offer further instructions (clear cut-away wax, change cutters, etc). Follow all Instructions and click "Confirm". The job will be completed after all required user interventions are performed. See Chapter 2 for a complete description of each user intervention necessary.

Revo Procedure Manager An Overview cont'd.

Each section of the Revo interface is explained here.



Spindle & Coolant Display Each spindle (V - Vertical and H - Horizontal) is represented beside a read-out of its respective coolant pump. As each spindle turns on during a mill job, it's frame is lit up in white. When it is off, it is dark gray. Similarly, when its coolant pump turns on, it is lit up in bright blue, and grayed-out when off.

Pulse & Duration Controls Control the "pulse" (frequency at which coolant is expelled) and "duration" (length of time coolant is expelled with each pulse) using these two digital controls (0 - 100%). Click and drag the indicator to the right (to increase) or left (to decrease) each setting. Or, just left-click at the position to which you want to move the indicator. These controls replace the knobs that were used to perform the same functions on the coolant control unit of previous models (A and B).

Cutter & Tool Length Display The cutter currently loaded in each spindle - IF you correctly followed the Instructions for the current job - is indicated in this section of the interface beside the spindle (V or H) in which that cutter is loaded. The Tool Length currently recorded for this cutter (recorded by the mill during the section of the Instructions that uses the Tool Point Sensor) is also displayed here for each cutter. IMPORTANT: if you think you may have broken a cutter and wish to check, DON'T remove the cutter! Instead, measure it again by clicking the "Seek" dropdown menu > "Seek (V or H) Tool". The software will prompt you to load the tool sensor in the correct location before it measure the tool. The tool should measure the SAME as this value. If it varies MORE THAN 0.04 mm, this means you've broken the tip off your cutter, and will need to replace that cutter in order to create accurate models. This interface also shows when the last "Seek Tool" (tool-measuring step during set-up) was performed. If it hasn't been performed for a while and you're getting unexpected results, it might be time to check, using "Seek > Seek (V or H) Tool" and make sure you haven't broken a tool.

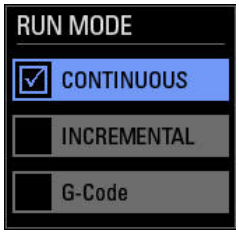
Wax Blank & Fixture Display Essential parts of any mill job, the "Wax Blank" and "Fixture" sections will keep track of when the proper wax and fixture were installed for this mill job (actually, when you TOLD the mill they were installed, since the machine cannot detect these). Before you "Confirm" each step, these displays read "Not Set" in bold red graphics, warning you that you're not ready to mill until you've loaded each one by following the Instructions. They display "confirmed" after you have followed the instructions and clicked "Confirm". Important: You must follow the instructions correctly for these to be correct: Revo cannot "detect" that the correct wax and fixture are installed. It simply takes your word for it. (And please, for both your sakes, don't lie to the mill)

Run Time The Run Time section of the interface displays the "Total" amount of time this mill job will run; the "Remaining" time, once it starts running, until it will be finished; the percentage of the job it has completed; the current "Time" (according to the Windows clock on your computer); and the time at which the job is estimated to "Finish". That way, you'll know when to return to the mill and load on the next money-making job. All times are an approximation. Incidentally, the "Total Time" your Revo has been running is added up and displayed in the lower right-hand corner of the Revo interface. Consider this your mill's "odometer", so you'll know when maintenance is due (explained elsewhere in this manual). Revo Tech Support may also ask you for this number should you call in for assistance.

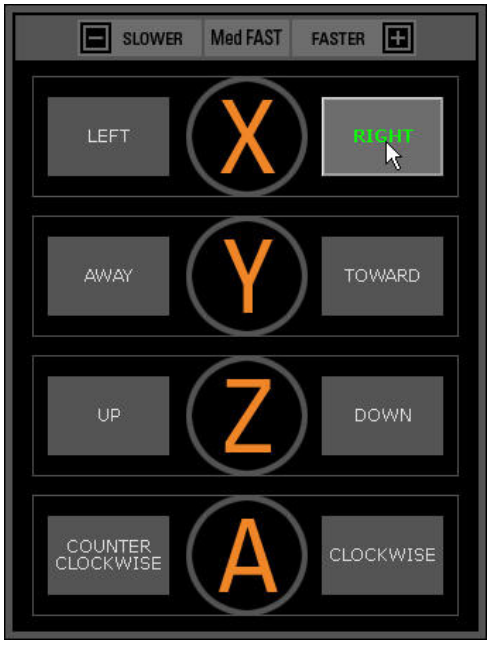
Feed Rate Just to the right of the Run Time interface is the Feed Rate control (50 - 150%). At any time before or during a job, you may click and drag the indicator to speed up or slow down the spindle rate (both spindles) for your mill job. (Or, just click the spot on the dial to which you want to move the Feed Rate.) We STRONGLY recommend leaving this at 100%. However, you may increase the speed as high as 150%, to speed up the job (and risk reducing surface finish quality); or, you may reduce the speed as low as 50%, causing the job to take longer and possibly improving surface finish, depending on the model.

Revo Procedure Manager An Overview cont'd.

Each section of the Revo interface is explained here.



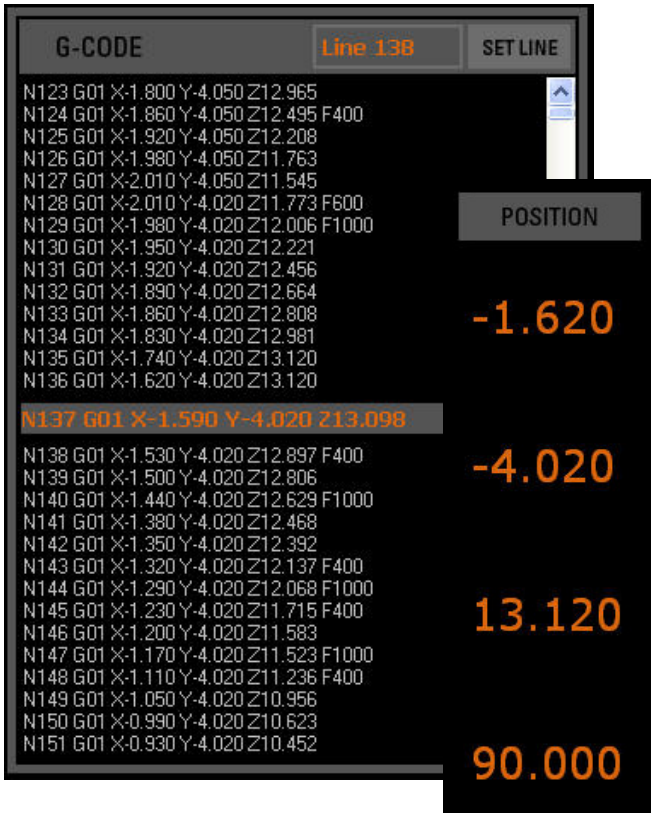
Jog Controls In "Run Mode > Continuous", you'll be able to see the Jog Controls (shown at right). While the mill is working, the letter representing the axis that is currently moving will light up in green. When the mill is NOT running, use these controls to move, or "Jog", the mill yourself during certain strategies, a process we'll explain later in this manual. It's easy to do: simply select a speed by clicking the "Slower" or "Faster" controls above the axes. The current speed displays between these two controls. PLEASE choose the slowest possible rate of movement you can stand: don't rush those cutters around, especially when they get close to something (or somebody). After selecting a speed, click one of the two controls beside the letter representing the axis to move it in the indicated direction (Left, Right, Away, and Towards are relative to YOU, the operator; not the mill).



Incremental Movement To move an axis "incrementally", a process in which you assign the distance it should move in mm, choose this "Run Mode" and a field will appear beside each direction the mill can move. Type in a value for it to move and click "Go". Use "Reset" to clear all the fields and move the mill again in a new direction/distance.

G-Code Toggle on the G-Code option to see the programming language that the mill computer is using to instruct the mill to cut your part. The G-Code is read from the toolpath you created in Matrix, meaning that the regular, closely-spaced "points" along your toolpath are converted into coordinates for the mill to travel to - while the spindles are spinning quickly - so that the cutters follow the right path to cut your part out of the wax blank. What you're seeing in the G-Code interface are all of the many thousands of coordinates to which the mill needs to travel in order to cut your part. The large gray highlighted line - which scrolls through very quickly, depending on your Feed Rate - is the line of code the mill is working on right now. See how fast that thing goes? If you like to watch this stuff work, switch to the G-Code view. There are times you'll need to work in this view, which we'll discuss next.

Position Display The current "position" of each axis is displayed here, where "0" for each axis is the "Home" position it returns to during "Seek Home" (described earlier). These numbers will change while you are jogging the mill, and while the mill is running a job. Usually, they're not needed by the average user, but be aware that they'll be changing - and quickly - as the mill moves each of its 4 axes to complete its work.

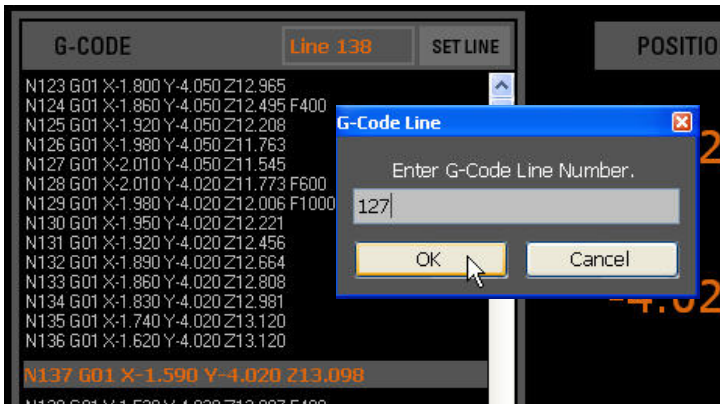


G-Code shown at right; Position Display at far right.

Revo Procedure Manager An Overview cont'd.

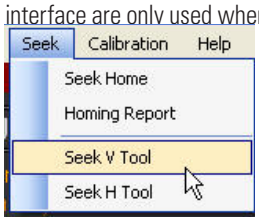
The remaining parts of the Revo interface are explained here.

Set Line See those numbers, starting with N, that appear at the beginning of each line of G-Code? They indicate the number of the G-Code line the mill is currently working on. After you stop the mill in an emergency situation (such as after the power goes out and before the battery backup gives way), write down the number of the line of G-Code on which the mill stopped (it will be highlighted). Then, DO NOT remove the wax from the fixture from the mill. When it is safe to do so again, click "Load Job" in the Action window and load the SAME job part(s) again. With the job set up and before clicking "Start", click "Set Line". Type in the G-Code line 10 lines PRIOR TO the number you wrote down. If you wrote down "137", indicate "127". There is no need to use "N"; just the number. This way, you won't need to run the entire job again.



If the mill job was stopped at line "137", write this number down. DO NOT remove fixture or wax. "Load Job" again when it is safe to do so ("skip" all cutter loading and measuring steps) and, before clicking "Start", choose "Set Line". Type in 10 lines earlier ("127"), click OK, and click Start.

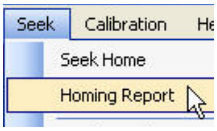
Troubleshooting the mill Certain areas of the Revo interface are only used when troubleshooting the mill.



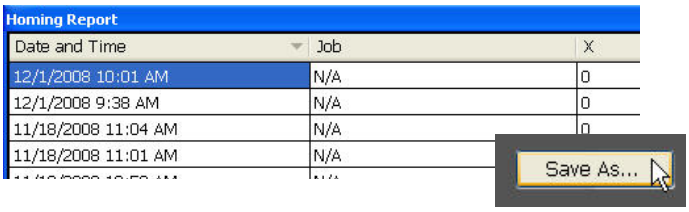
Do you see part inconsistencies? "Seek > Seek V (or H) Tool". Check that the cutter didn't break.

Broken Cutter? The leading source of part inconsistencies or unexpected results when using the Revo mill is a broken cutter. Measuring the cutter is part of every job you run on the mill; however, you don't need to run a job to measure a cutter. To check and see if your cutter broke between the last time it was measured and right now, select the "Seek" drop-down and choose "Seek V (or H) Tool", depending on which tool may have broken. Instructions will guide you to place the tool sensor for either the vertical or horizontal spindle to measure the length of the cutter. Remember: its length should have changed by LESS THAN 0.04 mm. If it changed by more, chances are it broke. Unless of course, you inadvertently loaded a broken cutter into the collet at the beginning of the job. Compare the existing cutter to a new/fresh cutter and examine the tips with a loupe. Remember, if you remove or replace a tool from the spindle, *always* measure the length of the cutter before milling.

Homing Offset Error / Homing Report Each time the mill "seeks home" or returns to its default (0,0,0,0) position, it records the difference between how many steps it took to get to its destination and how many steps it took to get home. Like balancing a checkbook, this number is usually off by a few steps (equivalent of a few pennies when balancing a checkbook - only it's Bill Gates' checkbook); however, when this number is off by more than 24 steps



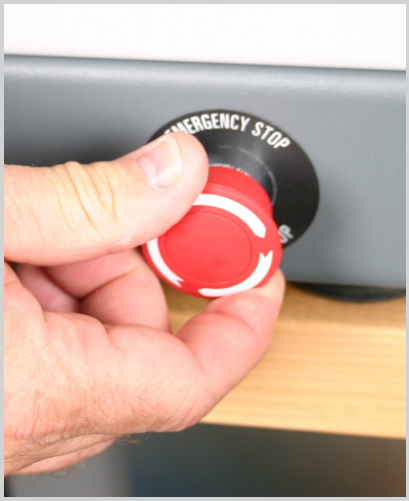
in any direction, a status box will alert you. The outcomes of every "Seek Home" operation the mill performs are therefore recorded "behind the scenes". This is not important information for the user to know UNLESS an error of more than 24 steps in any direction occurs. If an error occurs, a "Homing Offset Error" will appear onscreen with a notice to call Gemvision Technical Support. Tech support will ask you to locate your "Homing Report", which has recorded this value every time your mill went home, so they can determine if there is an error that can be traced to a certain element of the mill over time. To find this report, locate the "Seek" dropdown menu and the "Homing Report" that appears. Click "Save As" when this appears, and save it to a safe place on your computer where you can retrieve it later and email it to Gemvision Technical Support.



Other Log Files Additional log files may also be requested by Gemvision Technical Support when troubleshooting an issue with your mill. To locate these logs, access the dropdown menu "Help" > "Save Logs" and navigate to a location on your computer where you can find this file again easily to email it to Gemvision Technical Support as an attachment. This file is saved as a ".zip" file.

Tutorial 7: Stop a Job & Use "Set Line" to Resume

Follow these steps to stop a job; re-load it, and use "Set Line" to restart it a few lines of G-Code before it stopped.



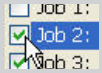
After stopping a job for any reason, write down the line of G-Code on which the mill stopped. It is VERY important not to remove the wax from the fixture or the fixture from the mill prior to resuming the job. It is also important to know which "Part" of the job the mill was running when it was stopped (explained earlier). When it is safe to do so, "Load" this Job part and click through Mill Set-up. PRIOR to clicking "Start", switch to the "G-Code" view of the Revo interface and click "Set Line". Indicate a G-Code line 10 lines before the mill stopped and click "Set". Click "Start" to resume running the part.

Icons used

"E-Stop" to stop a job

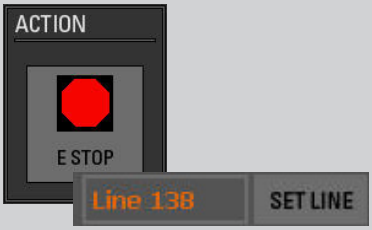
"Load Job"

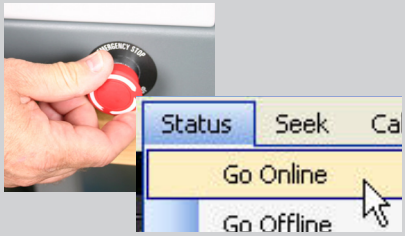
G-Code mode

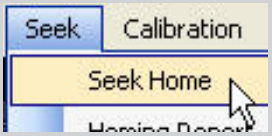


Job Controls

"Set Line"



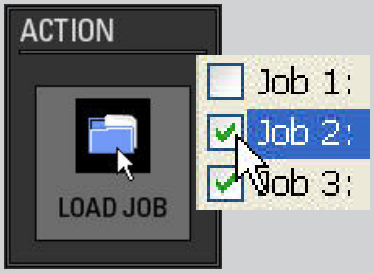


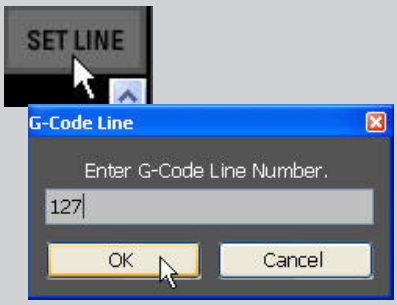


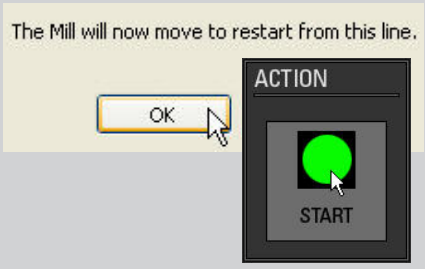
1 Stop the Mill & write down the G-Code line If a problem occurs during milling (e.g. the power goes out), hit the E-Stop or any key to stop the mill. To see the current line on which the mill stopped, click "G-Code" Run Mode. The current line is displayed left of "Set Line".

2 Reset the E-Stop and bring the mill online If you hit the physical E-Stop button on the mill, turn it to the right to re-set it. (Mill will not go back online if E-Stop is still depressed.) Click "Status > Go Online" to restore mill's communication with laptop.

3 "Home" the mill Bringing the mill back online should "Home" it automatically. If it does not, complete this step yourself by clicking "Seek > Seek Home". Remember: DO NOT remove wax fixture from mill! If you do, these steps will NOT work.







4 Re-load the Job Load the SAME file that was running when the mill stopped. Take care to ONLY load the job parts that were not completed yet: i.e., if the mill stopped partway through Job 2, only load Jobs 2 and 3: not Job 1. Complete all the required steps until the "Start" button shows.

5 BEFORE starting job, click "Set Line" BEFORE clicking Start, go back to "G-Code" run mode and click "Set Line". Type in the G-Code line from Step 1 MINUS 10, to start a few lines back from the line at which the mill stopped, ensuring better results.

6 Mill will move to line. "Start" the job When warning appears that mill will move into the new position, click "OK" and wait a few moments. Click "Start" and the job will start from the new line of G-Code.

Tutorial 8: Check for a Broken Cutter

An important part of set-up and the first line of defense when troubleshooting inconsistencies in finished parts, learn to detect a broken cutter.

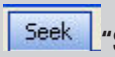
Whether measuring a cutter during part set-up on the mill or using the "Seek" Dropdown menu in the software, this is an important step for both running and maintaining the mill. When a cutter breaks during a mill job, the part may reveal inconsistencies from the original design. If you notice part inconsistencies and/or you think the cutter has broken, DO NOT remove the suspect cutter! If you do, it will be much harder to tell whether or not it has broken. Next, run "Seek" > "Seek (V or H) Cutter" and measure the cutter. If the "Changed By" information in the next screen reveals a value of more than 0.04 mm (yep, that small), you have broken a cutter. Remove and discard the broken cutter (or, read about possible uses for broken cutters in Chapter 3). The cutter may still be broken, however, if it was broken before it was installed. To determine this, use a new cutter you know isn't broken to run the part. If the part improves, the first cutter was broken.

Materials Needed

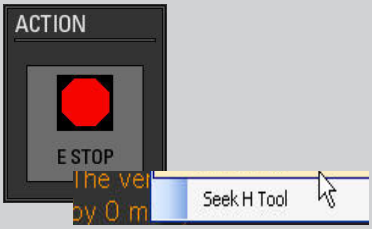
Hardware:

- **Suspect Cutter**
- **Tool Sensor**
- **New Cutter (in case it's broken)**

Revo Software:

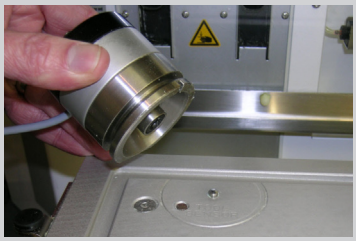


"Seek" Menu (Seek V or H)



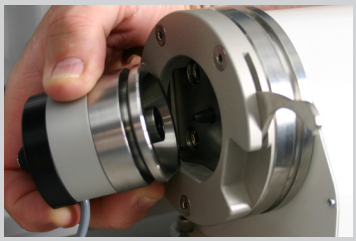
1 Seek V (or H) cutter

Use this dropdown menu when you think you may have broken a cutter. IMPORTANT: DO NOT remove the suspect cutter before clicking one of these options!
Note: This process will also be completed automatically during set-up. It is the same.



2 Position Tool Sensor as Indicated

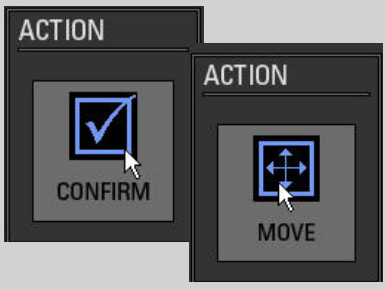
For either selected position, the software will prompt you to position the tool sensor on the mill. *(V - above Left)* To measure the vertical cutter, locate the sensor in the notch on the table, aligning the bottom of it with the pin, to place it in the Vertical measuring location.

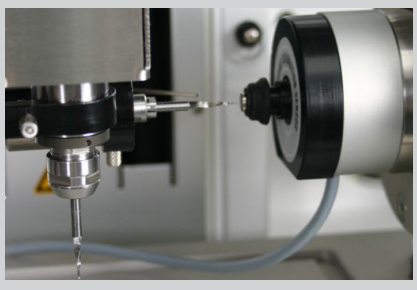


(H - above Right) To measure the Horizontal cutter, install the tool sensor on the A axis like any other fixture, aligning the flat part of the chuck with the flat part of the puck (receiving piece on the A axis) snugly, so that it snaps into place, and lowering the clamps to secure it.

3 "Confirm" Tool Sensor is loaded & click "Move"

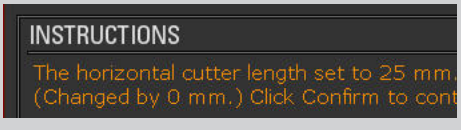
Since the mill will not move until you tell it to, "Confirm" that you have loaded the tool sensor, check that it is safe to do so, and click "Move" so that the mill can measure the cutter against the tool sensor.





4 Check "Changed by" value

After the tool is measured, the software will report the tool length and a "changed by" value. Check that this value is not more than 0.04 mm. If it is, the tool has broken. If it is not, your tool STILL may be broken, if it was already broken the LAST time it was measured (obviously, there would be no change). In that case, install a brand-new cutter that you are CERTAIN has not broken and run the part again. If the new part is more successful, you'll know the first cutter was broken. If no improvement occurs, contact Gemvision Technical Support.



Getting Started ...

Calibrating the Mill

After assembling the mill and before cutting your first part, it is important to cut a calibration part. This corrects for any shifts that may have occurred during shipping.

When to run Wax Calibration Block

It is ESSENTIAL to run the "Wax Calibration" block after your mill is all set up (i.e. you've followed the rest of the steps in this chapter and you understand the assembly, use, and maintenance of each part of your mill) and BEFORE you run ANY parts you've created in Matrix. It is easy to run the Wax Calibration, and it is the first job you will cut on your mill.

It is important to run the Wax Calibration Block BEFORE any others jobs because Wax Calibration is designed to correct for any shifting in your mill's axes that may have occurred during shipment from Gemvision to you. Shifting can produce part inconsistencies, so you'll want to correct these before you run any models. You should also run this file after moving or shipping your mill, such as when you've relocated it from one room to another, or when you've shipped it from one location to the next. It may also be necessary to run the Wax Calibration block if you damage a fixture and we send you a replacement fixture. A GV Technical Support representative will help detemine if his is needed. You may also be instructed to run Wax Calibration by Gemvision Technical Support if certain inconsistencies appear in your models. A support technician can help you verify whether these inconsistencies are due to a broken cutter, the need to calibrate, or another error in hardware or software set-up or mill functioning.

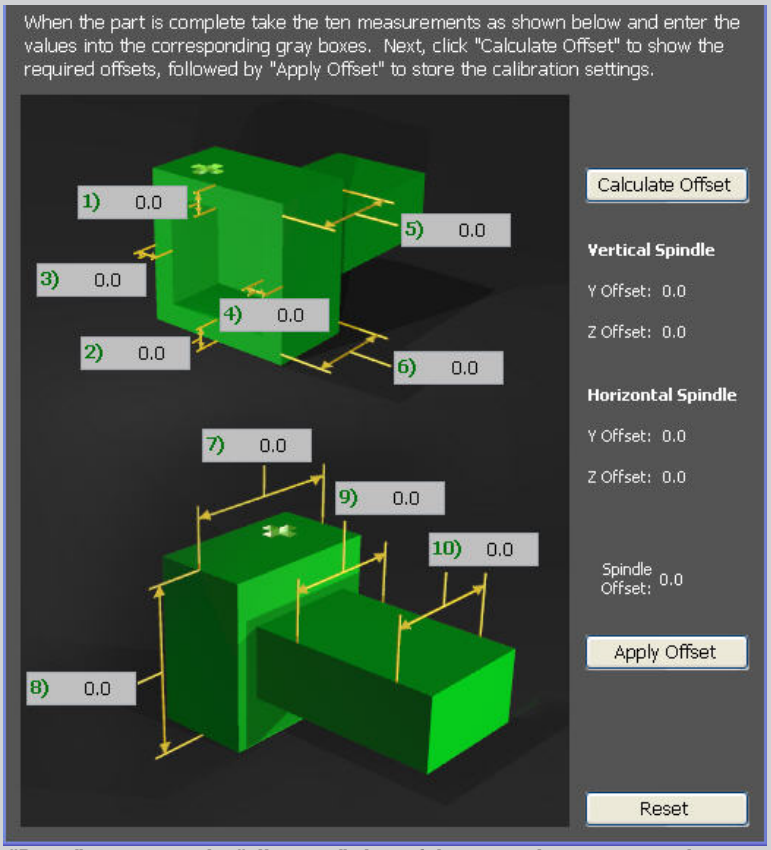
Using the Wax Calibration Block Interface

After mill completes the wax calibration block, the "Wax Calibration Block" interface appears onscreen, prompting you to measure the part and enter the measurements into each gray box.

1- Take Measurements Be VERY careful to measure and record the correct dimensions on the block, keeping track of the location of the symbol milled into the block as a reference point. The dimensions to measure are: (1 - 4 in illustration below) the thickness of all four walls of the box; (5 - 6) the height of the box in two locations - side with symbol and opposite side; (7 - 8) the width of these two sides of the box; (9 - 10) the width of the support box in two locations: one at each side of its center.

2- "Calculate Offset" With all measurements correctly entered, click "Calculate Offset" to see the results of your changes beside each axis, indicating how each will be corrected for the Vertical and Horizontal spindles.

3- "Apply Offset" Click "Apply Offset" to keep these values. Finally, close the Wax Calibration Block dialog.



"Reset" to return to the "all zeroes" view of the screen in case you made an error when entering values. Values are only saved upon "Apply Offset".

Tutorial 9: Mill your first model: Wax Calibration

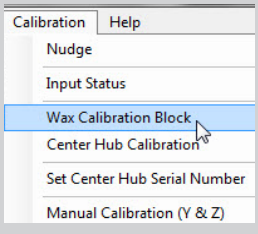
The VERY FIRST model you should mill lives in the "Calibration" dropdown menu under "Wax Calibration".

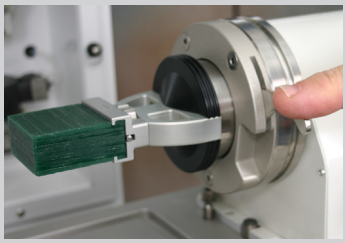
The VERY FIRST model you should run on your mill, after fully reading and understanding the set-up tutorials in this chapter and before ANY other models are milled, is the Wax Calibration model found in the "Calibration" dropdown menu onscreen. Set up the mill as prompted to by the Instructions and Actions section of the interface, using the 0.032" black flat cutters in BOTH the Vertical and Horizontal spindles; the 45X39X20 mm Base Clamp wax; and the Base Clamp (Fixture 1). Once the part is produced, it will fall away from the supporting wax. Retrieve it and CAREFULLY take the measurements indicated onscreen, taking care that each is entered in the correct box. Click "Calculate Offset" and "Apply Offset" before closing the dialog box.

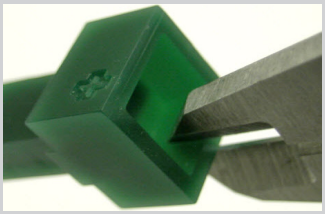
Materials Needed

Hardware: (Included with Mill)
- **Base Clamp (Fixture 1)**
- **Base Clamp Wax (45X39X20mm)**
- **Two (2) 0.032" straight cutters (V & H)**
- **Digital Calipers**

Revo Software:
- **Calibration > Wax Calibration Block**







1 Click "Wax Calibration Block"

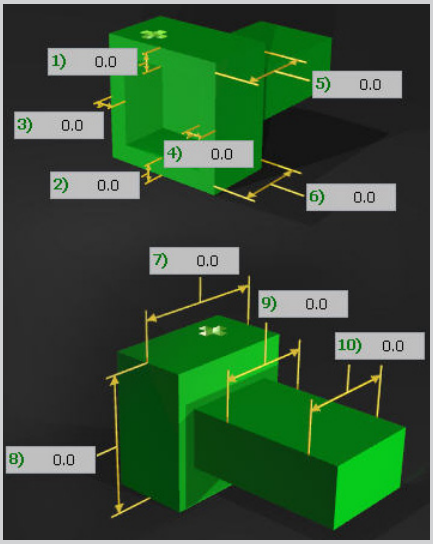
From the "Calibration" drop-down menu at the top of the screen, select the "Wax Calibration Block". This will cause the model to automatically load, and the "Revo Cal" graphic will appear onscreen.

2 Install the cutter, wax, & fixture

The "Instructions" onscreen will walk you through removing the current cutter; installing the 0.032 " straight cutter in the Vertical spindle, and measuring the cutter with the tool sensor in the Vertical measuring location. Also install the correct wax blank in the Base Clamp Fixture and install the fixture on the A axis. "Confirm" you've done so in the software, and check the coolant levels to complete set-up. Install the splash guard and "Start" the job.

3 Remove Wax Block, Enter Measurements

Instructions will appear onscreen. Remove the block from the wax, clean and measure it in the ten locations indicated. Use your digital calipers: Measure the thickness of the box on all four sides; the height of the box on the top (marked with symbol) and bottom; the width of the box on the top and the side; and the width of the box's support at two locations, each to one side of center.

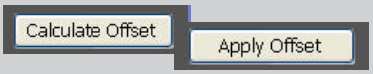


Note: Best Use of Calipers

To best use the calipers for measuring this Calibration Block, keep these tips in mind:
- use the same set of calipers every time you take a measurement from your mill;
- caliper jaws should be clean; no debris;
- when shut, the calipers should read "0". If needed, click "Zero" to zero them out when shut;
- the face of the part being measured should be parallel with the jaws of the calipers;
- apply the same amount of pressure at each location you measure; if possible, use the same part of the jaws (tips vs. flat edges, e.g.) to measure each location.

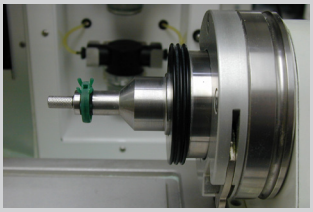
4 "Calculate Offset" & "Apply Offset"

Click "Calculate Offset" and you will see the Offset values on the far right-hand side of this screen change to reflect your measurements. If you determine you've made an error during measurement, click "Reset" to return all fields to 0. ONLY use "Reset" BEFORE Apply Offset. After re-entering values, click "Calculate Offset" again. When you are CERTAIN you are ready to do so, click "Apply Offset" to complete calibration. Close the window when done.



Tutorial 10 : The Center Hub Calibration

If Technical Support instructs you to run the Center Hub Calibration or if the models you are milling on the Hub fixture are producing undesirable results as seen in the picture below, follow these instructions to align the A & X Axis on the Hub fixture.



If you notice that prongs are consistently clipped from front to back (as seen when mounted on the hub fixture) you may be required to run the Center Hub Calibration to correct this offset. This calibration is used to align the A & X axis when using the dual + hub strategy.

Materials Needed

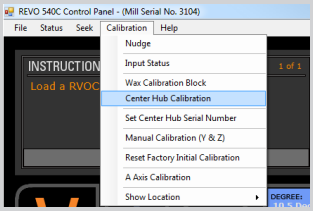
Revo Revo Software:
- **Calibration menu > "Center Hub Calibration"**

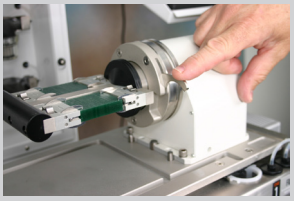
Revo:
- **Dual Three-Sided Fixture**
- **Center Hub Fixture**
- **One (1) 46X42X10mm wax block**
- **10 deg. cutter (dark blue cap)**

Optional but recommended:
- **Loupe or magnifying glass**

1 Choose Center Hub Calibration

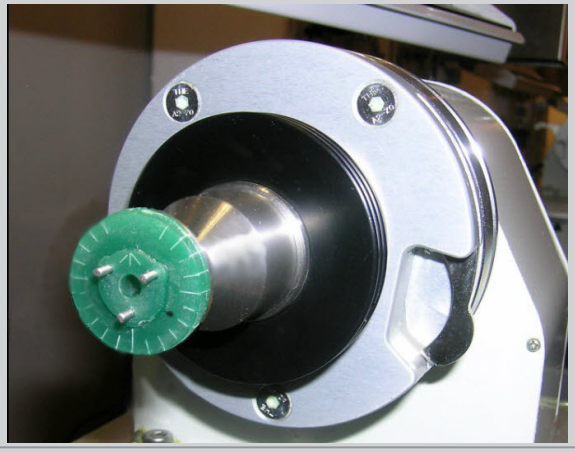
From the Calibration drop-down menu at the top of the Revo interface select the "Center Hub Calibration". This will cause the model to load automatically.





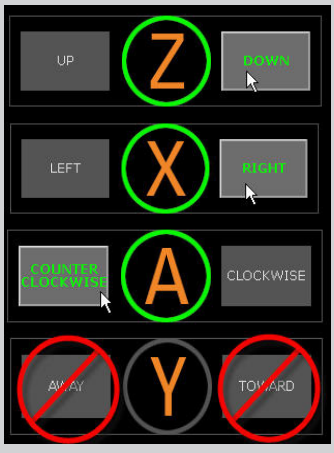
2 Install the cutter, wax & fixture

The Instructions will prompt you to remove the horizontal cutter, install a 10 degree cutter in the vertical spindle, measure the cutter using the tool sensor, and load a 46 X 42 X 10mm wax blank into Fixture 4, the Dual 3 Sided Fixture. Click "Start" to run the first portion of the Center Hub Calibration.



3 Load the Wax on the Hub fixture, Fixture3

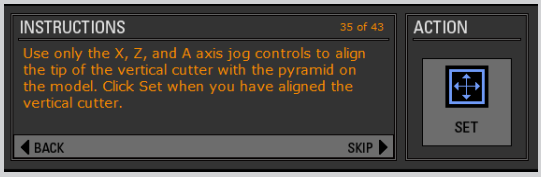
Rinse the Fixture. The Center Hub Calibration Coin will automatically be freed from the Dual fixture. Attach the Coin onto Fixture 3 (Center Hub); making certain the model is properly aligned. The Arrow will face up and should be seen when the coin is attached through the 3 pins of the Center Hub Fixture. Attach the thumbscrew and Click Start.



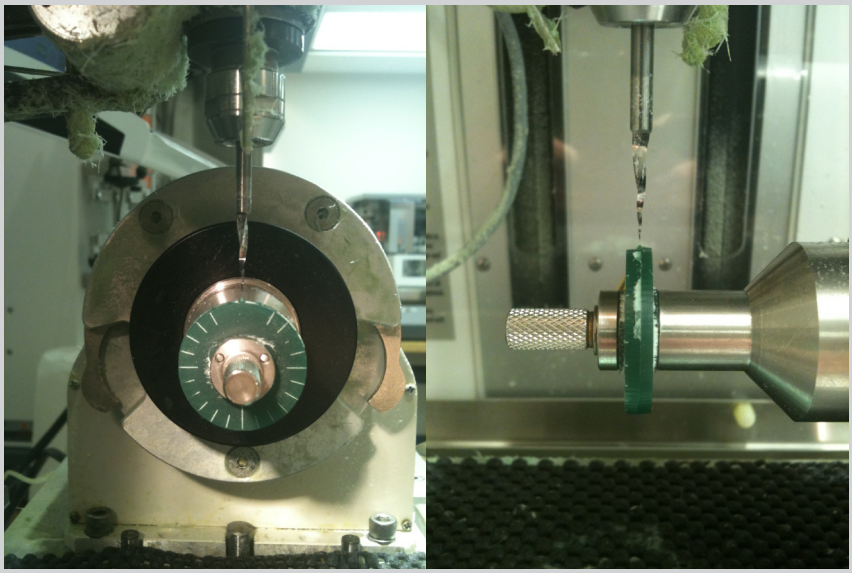
4 Center Hub Calibration Alignments

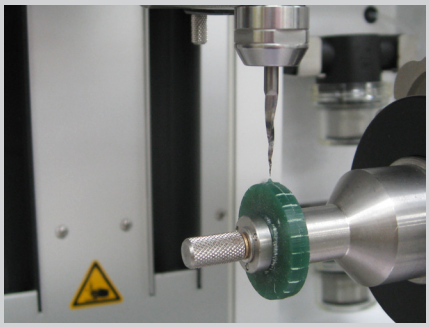
The mill will move to the Home position and then move to the approximate location where the tip of the cutter will align with the small pyramid on the top of the coin. First use the Z axis jog control to bring the cutter within close proximity to the pyramid. Next use the X axis jog control to align the tip of the cutter with the pyramid when viewing the model on edge. Lastly, use the A axis to align the tip of the cutter with the pyramid when viewing the coin straight on. It is important to make sure you use the proper vantage point when making the adjustments. If you are viewing the model on angle, optical illusions will make it difficult to properly calculate this adjustment. When you are confident the alignment is correct, click "Set". This will cause the mill to run the final portion of the toolpath.

When prompted to do so, Jog the mill using any Jog Controls EXCEPT the Y controls to align tip of cutter with "pyramid" on model.

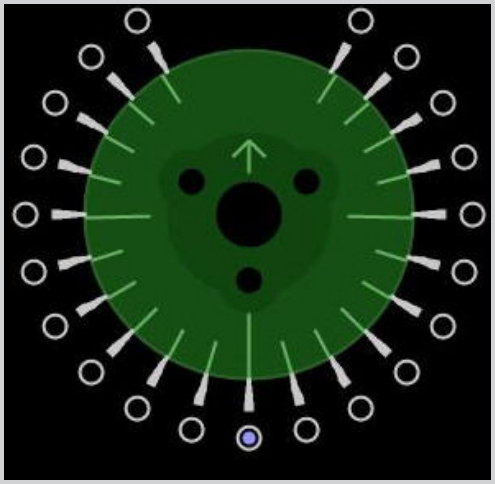


Calibrating the Hub Fixture cont'd.





Make sure to view the cutter and pyramid alignment straight on during the alignment process. Viewing the cutter at an angle can cause an optical illusions making it seem as if the cutter is aligned.



(L) This disk is correct: the ticmarks line up best at the 6:00 position. Click here in diagram (default position for diagram).

9 Final Alignment Menu The final Center Hub Calibration menu will appear. Remove the model from the fixture. Use a loupe or microscope to closely examine the Coin. Find the location on the coin where the tic marks on the face of the coin align perfectly with the tic marks on the side of the coin. Place a small mark on the coin where the proper alignment occurs. Next view the onscreen menu and select the corresponding location. Click Set A Cal. This will complete the Center Hub Calibration process.

(L) This disk represents a misaligned hub: the ticmarks do NOT line up at the 6:00 position.

Chapter 2

Building Toolpaths in Matrix

This part of the Matrix software allows the user to create a "toolpath", or, the file that directs the mill how to cut the model out of wax.

About the "Toolpath Builder" in Matrix

Mill C Builder is found in the Builder Menu.

Found in the Builder menu of the Matrix software (Mill C Version), the Mill C Builder walks you through a series of steps that end in the creation of a toolpath for the model you have in the viewports.

A toolpath is a curve representing the path the cutter will take to cut the model from the selected wax block or tube. The toolpath is saved as a ".RVOC" (Revo C) file after using the mill builder. This file is written in "G-Code", a numerical language which tells the mill how to move using a series of coordinates (X, Y, Z, and A). Put simply, by doing what the G-Code tells it to, the mill follows the toolpath you create in the Mill Builder to properly cut your model. After creating the .RVOC file in Matrix, you should save it to a USB drive or similar device and transfer it to your Revo computer. The Revo computer then walks you through a series of steps to set up the mill so it can follow the toolpath and cut your model out of wax, as described in Chapter 1. Certain steps in the Revo software are unique to each fixture, so we'll cover those in this chapter, as well.

Preparing to Use the Mill Builder

Always put the model into a Job Bag prior to entering the Mill Builder, so you have a saved version of it with History, Gems, and other important elements. Also make sure that, if you're milling a ring, you have a ring rail in the viewports prior to entering the builder. If you do not, and you are milling a ring, the builder will prompt you to enter the diameter of the finger rail, in mm, so you should know this value if you don't have a ring rail present. Obviously, it's easier to place the Ring Rail first, in Matrix. The Mill C Builder can even read a custom size placed with the Ring Rail tool (mm or in. diam. or circumf.), so make use of this handy tool prior to entering the Mill C Builder. Note: If you wish to mill a model created in a version of Matrix prior to 5.3, you will need to open the model in version

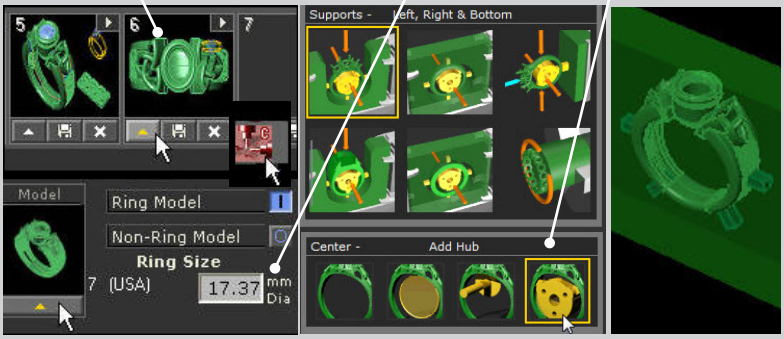
An Overview of using the Mill Builder in Matrix

1: Prepare Model
In Matrix, Job Bag the final model. Delete gems and any additional items that will not be milled. If you don't have a ring rail and you are creating a ring, add one. If you will be using your own supports, create them. Job Bag again, and enter the Mill Builder.

2: Input Model
In the Mill Builder, select the size of your model. Select the actual size you'll be using. A preview appears.

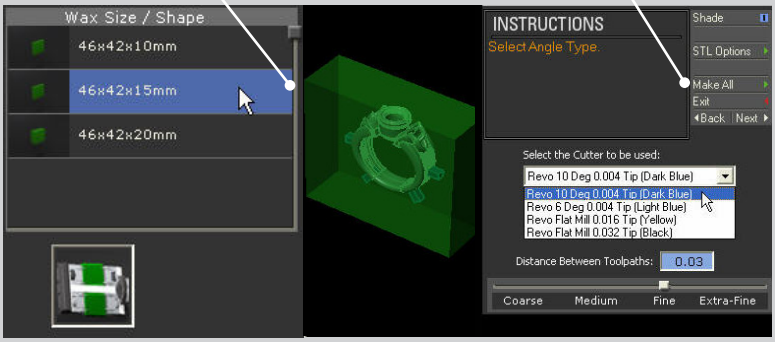
3: Supports & Center
Choose supports (how a model is held to the wax) and the builder will add them to the model. Next, choose a center strategy: the disc cutter (for hollow or comfort-fit), the "Hub" (adds a rotary pass to a flat cut), "Ignore" (mills out center) or "Center cut" (for faster results).

4: Fixture & Wax Size
The builder offers a fixture and wax size based on the supports you chose and the model and click the "In" arrow beneath the Model preview window to input it into the builder. Choose whether or not it is a ring. Check that the ring size is correct or input a value for its mm diameter. Additional options will appear.



5: Additional Angles
For two strategies - 3-sided and base clamp - you'll be able to assign 2, 3 or 5 angles; or 4 or 8 angles, respectively. Choose based on model details.

6: Make Revo
Lastly, choose a cutter and part definition (Coarse to Extra-Fine) and click "Make All" to create and save the ".RVOC" file that you'll transfer to the Revo mill for cutting.



6.0, delete the current ring rail and add a new ring rail. By doing so, the model can be recognized as a *ring* by the builder. Finally, if you have objects in the viewports that will NOT be cut, it is important to delete them from the model. The only thing onscreen should be the parts of the model that will be cut by the mill and a ring rail (if the model is a ring). It is also a good idea to Job Bag this version of the model - with no gems or additional items present - so you can input it back into the mill builder quickly and easily if you need to use it to create another toolpath in the future. The program will ignore gems so they can remain.

Job Bag before modifying for Mill Builder ...

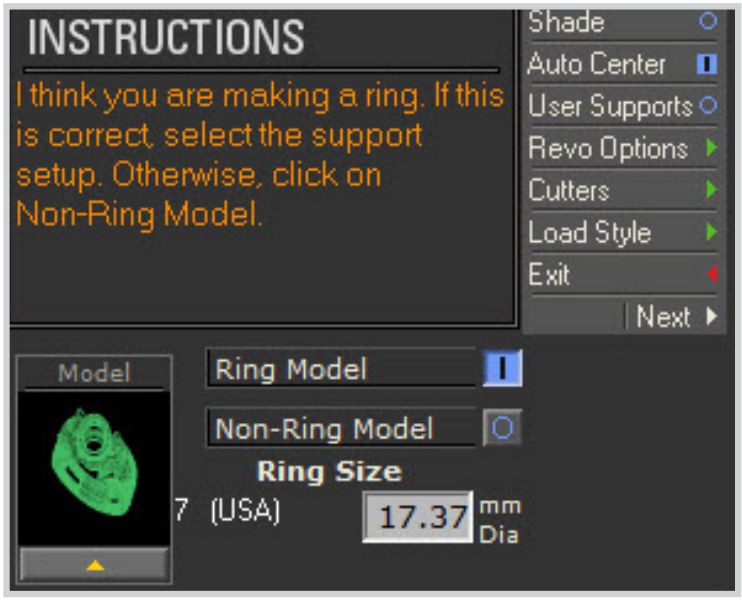
... AND Job Bag after modifying for Mill Builder

The Mill Builder: At-A-Glance

The following steps walk you through preparation of a ring model in the Mill Builder & completion of a toolpath for use on the Revo C. Each step is described fully in this chapter, along with the process for a non-ring model at the end of this chapter.

1 - Input Model. Ring or Not?

Upon first entering the Mill Builder, input your model and choose whether it is a ring or not. The builder can determine when you have input a ring rail - which should only be present for a ring model - and calculate the ring size. However, you should ALWAYS double-check that this selection is correct! If you have a ring to mill but no ring rail, you'll need to manually select "Ring Model" and input the mm diameter of your ring. So unless you know this value, or are prepared to measure it using Rhino tools from inside the Mill Builder, (using the "Dimension" menu in gray bar along the top of the screen) it is best to add a Ring Rail in Matrix first.

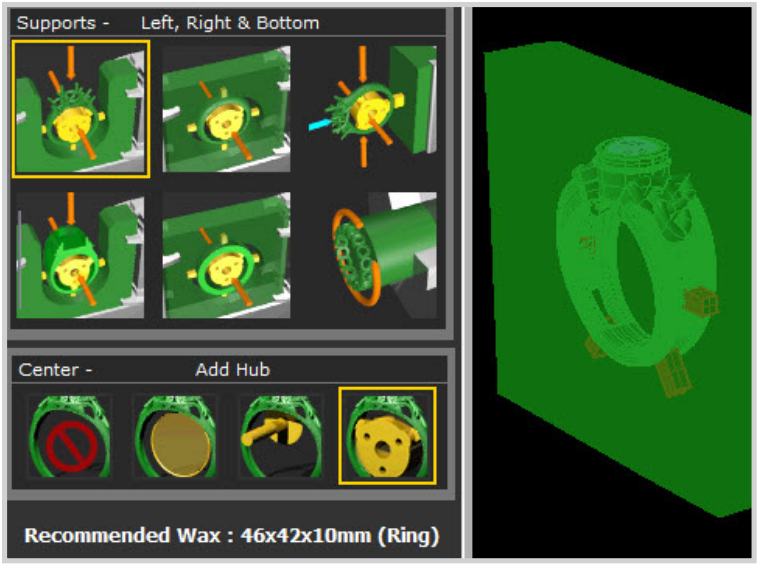


(Above) Input model into Mill Builder and select Ring or Non-Ring. Important: For ring models, if Ring Size does not display, input mm diameter of ring.

2 - Choosing Supports, Ring Only

The next step is to select the best support strategy for your ring model based on where its details are located. Supports hold on to the wax model during milling, connecting it to the block or tube of wax from which it is being cut so it does not fall out of place or flex under the pressure of the cutter. Choosing the location of the supports determines which fixture(s) you'll be using to mill this model, as explained later in this chapter. Choose support locations based on the details in your model so as to not leave out any key details when milling but insure the support intersect the model. When creating a non-ring model, or if you wish to use your own custom-created supports, you will need to build supports ahead of time in Matrix, turn on the "User Supports" option in the Mill Builder, and input them into the "Supports" in-box that appears. The process for building these supports is fully described in this chapter.

(Below) Choosing a Support strategy will place supports on the model. This is ONLY for rings. For non-ring models, input supports created in Matrix.



3 - Choosing a Ring Center

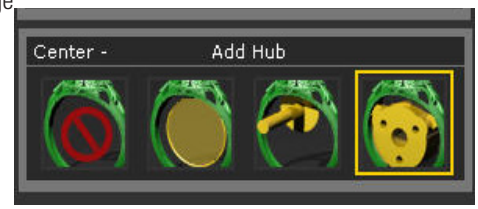
When you are milling a ring, you will have the opportunity to choose a "Center" option - or, how the mill will treat the center of your ring. These are mutually exclusive options, and are explained in full - how and when to choose them - later on in this chapter. From left to right, they are overviewed here:

(1) Mill Center ("Ignore Center" on 3-Jaw Chuck) treats the center of the ring like any other empty space in your model. This can take a while to cut, but since it does not require user intervention, select it when you wish to start up a job and let the mill run without your intervention.

(2) Cut Out Center runs a single cut-out curve 2 mm inside the center of the finger hole. Once it completes this section of the job, the user must "pop out" the wax piece it leaves behind and prompt the mill to keep cutting the remainder of the job. Because it has less to cut now (the center is already cleared away), this is a faster solution.

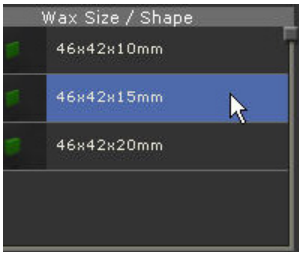
(3) Cut Out and Hollow runs "Cut-Out Center" first to clear the way for the disk cutter; then, the mill prompts you to change cutters so it can complete the interior cut on a hollow or comfort-fit model with the disc cutter. It will also prompt you to change the cutter back to complete the remainder of the job.

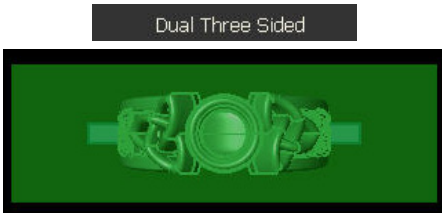
(4) Add Hub cuts a "hub" in the center of the model. The hub support attaches the model to the Hub fixture so that a rotary pass (around the outside of the model) may be added to any flat cut, completing details that may have been missed on the flat.



4 - Choose Fixture, Wax

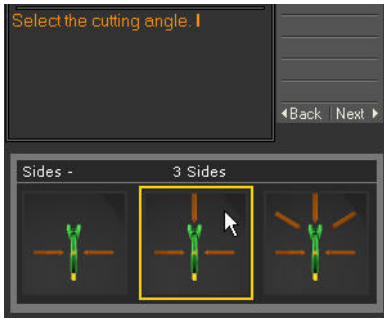
The support method, center strategy, and model size will be used by the Mill Builder to determine which fixture(s) will be used to mill your model. This information will be given to you by the Mill Builder software, and also by the Revo software when you are setting up the mill to build your model. The proper use of each fixture, and how the mill cuts a model from each of them (Ring and Non-Ring), is fully described in this chapter. Different wax sizes are available with each fixture. The Mill Builder can calculate the wax size your model will require, and will offer you the smallest size allowable for your model; or, you may choose a larger size wax if that's all you have. Note: During this step, it is VERY important to choose the ACTUAL wax size you will use to cut the model! The Mill Builder will not let you choose a wax size which is too small. Also note: if you choose a flat strategy (with smaller wax sizes) and the only wax large enough to create your model is a large tube wax, the Mill Builder will not let you proceed.



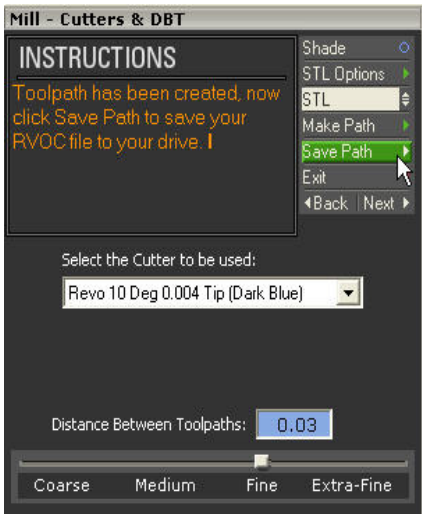


5 - Choose Custom Angles

Certain strategies allow you to choose custom angles: 2, 3, or 5 sides on the Dual Fixture (shown at right); or 4 or 8 sides on the Base Clamp fixture. What each of these options do and when to choose them is also described in this chapter.



(Below) Choose a cutter and Distance Between Toolpaths, or keep defaults. "Make Path". With the path saved, you may click "Save Path". Or, if you wish, click "Next" to view and edit each part of toolpath (described in "Detailed Options" chapter next in this manual). If you change any parts of the toolpath in this way, you must use "Make Path" and "Save Path" again.



6 - Make the RVOC File

There are several steps left on the last screen of the builder: choosing a cutter; choosing the part definition (or, the level of detail at which the mill will cut your part - Coarse through Extra-Fine), and finally making the .RVOC file that will be transferred to the mill for cutting. Begin by clicking "Make Path". This performs two tasks: (1) It creates an "STL" or mesh version of your model, which it must use to create the toolpath; and (2) it creates the toolpath - the aforementioned "map" that tells the mill how to move the cutters and axes to create your part. The toolpath looks like a long curve running back and forth over the model. It's a good idea to go to the Detailed Options screen (explained in the next chapter) and inspect this curve, to make sure it cuts all parts of your model. Finally, click "Save Path" to create and save the ".RVOC" file, which is written in G-Code, the computer language which directs the mill to follow the toolpath. This is the file that should be transferred, via USB device, to the mill. Each of these steps will be described in this chapter.

Topics Covered in this Chapter:

Clockwise, from lower left: XL Fixture; Hub Fixture; Base Clamp Fixture, Three-Jaw Chuck; Dual Three-Sided Fixture

Understanding the Ring Strategies ...

About Each Fixture

The many strategies of which the mill is capable are all performed using one of the five fixtures that attach to the A Axis. Learn about them here.

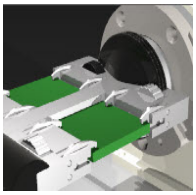
About the Fixtures

Choosing a Support Strategy...

Every model that is milled needs to be supported or "held" at some location, so it does not fall away from the wax or flex under the pressure of the cutter. These connecting pieces are called "supports". They are added to your model in the Mill Builder, or you may add them yourself in Matrix.

However, you don't want the supports to interfere with highly detailed areas of your model. Analyze where the details are located and where they are NOT located (also where supports can be placed), and the best angles at which to approach these details will determine which fixture will be best to run a job. The mill builder will help you accomplish this. This section presents an overview of how each fixture orients the wax for the approach of the cutters and supports the model by attaching it to the surrounding wax.

Based on the description that follows, choose the support strategy and center option which will cut the most details on each model. The support strategy and center option you choose will determine which fixture(s) will be used to mill your part, as well as which wax sizes (block or tube) are available for the job. The best support strategy for each type of ring model - depending on where its details are - is covered next.



... determines the fixture (Dual Three-Sided, here) ...

Other Factors in Toolpath Creation

Also factoring in as the software builds the toolpath, are (1) the thickness of the wax necessary to complete the job; (2) the type and length of the cutter(s) in each spindle; and (3) the orientation of each cutter (Vertical and Horizontal) in relation to the details on the model.

Also keep in mind how each axis moves in order to position the cutters and the model at the best angle for the cut: the A axis rotates the fixture to the correct position within all 360-degrees of rotation; X moves to the left and right, taking the A axis with it as it goes; Y moves forward and backward, taking both X and A along with it; and Z moves up and down to position the cutters at the height assigned by the toolpath.

... and available wax sizes / type (block or tube wax).

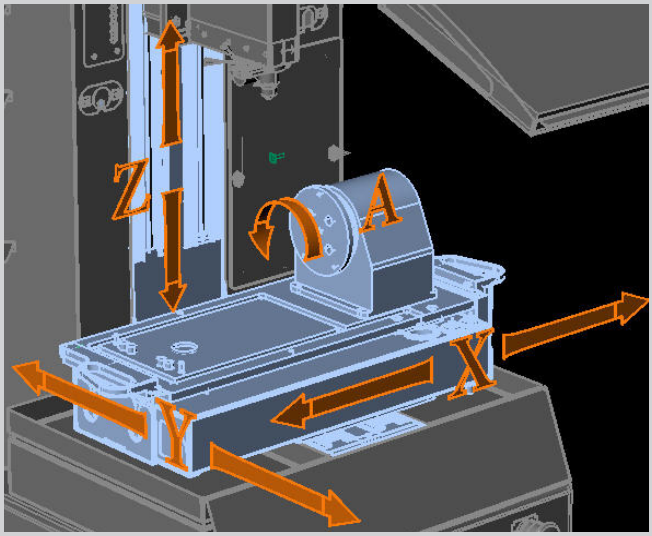
Each fixture described herein is attached to the A axis, so technically it is the fixture that is moved by the A, X, and Y axes to position the wax it holds in correct relationship to the cutters; and the cutters are moved up and down along the Z axis to attain the correct height as they cut each element of the model. Armed with that knowledge, we can study each of the fixtures - and its related ring strategies - in turn. Non-Ring strategies differ slightly, and so will be covered at the end of this chapter.

Two Methods: Flat Milling vs. Rotary

Recall the range of motion of which the mill is capable: The A axis, to which the fixture that holds the wax is mounted, can rotate 360-degrees; the X axis travels left and right and the Y axis travels forward and backward to get the fixture & wax properly positioned under the cutters; finally, the Z axis travels up and down to follow the toolpath & cut the wax.

The two main methods for cutting are: (1) Flat methods, whereby the model is held flat on the A axis while the cutters and X, Y axes move to mill the wax.

The other methods are (2) Rotary cuts, wherein the A axis turns the model while it is being cut by the cutters. The X and Y axis' movement allows the cutter to travel back and forth over the model.

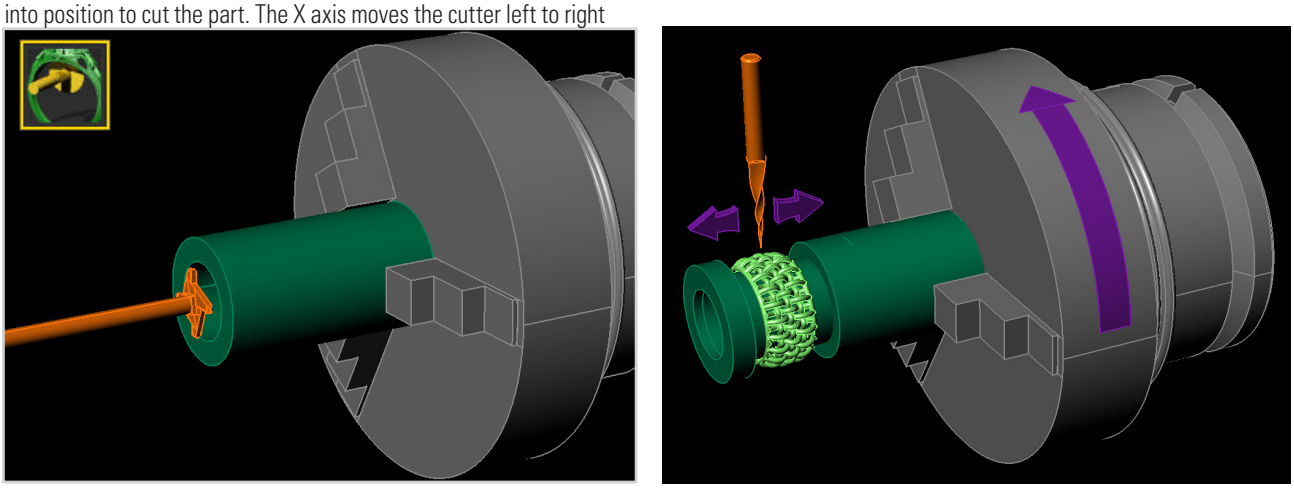


About Each Fixture An Overview

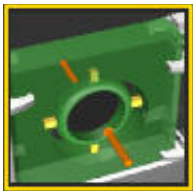
Each fixture and its associated ring strategies are summarized in this section.

The 3-Jaw Chuck The easiest strategy to understand, the Rotary strategy cuts the model out of a tube of wax held by the 3-Jaw Chuck. Best-suited for bands or rings with detail all the way around the outside, the only details missed on these models (when using the basic Rotary strategy) are those on the front and back of the band (as seen in the Through Finger view). To get these details, you may add the Hub fixture to one of the flat strategies, a process explained later. As shown below, at right, during milling the A axis rotates the tube of wax held by the fixture as the Z axis moves the vertical spindle into position to cut the part. The X axis moves the cutter left to right across the model to complete it. During set-up, the user must "Jog" (move) the mill to tell it where the edge of the wax tube is, a process explained later.

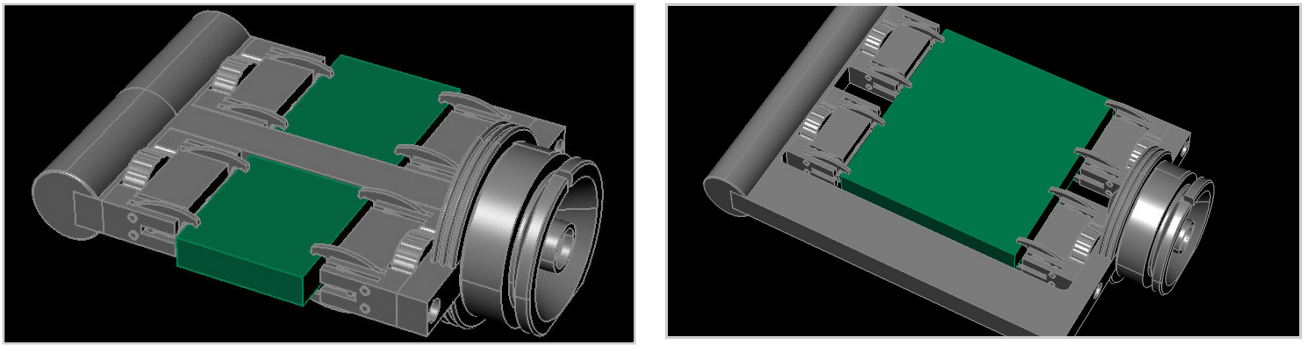
Border Distance There are NO supports necessary for this strategy. To support the model, the mill automatically leaves a thin amount of wax between the completed ring and the wax tube known as the "Border Distance". (Seen below, between wax tube and model.) This is created automatically, so there are no supports added when choosing this strategy. This was called the Toolpath Overcut on the A&B models.



(L) A hollowing cut (which can simply be used to cut the ring to finger size when the model is not hollow) is completed by selecting a hollow wax tube and installing a hollowing cutter on the Horizontal spindle. When selected, the hollowing cut is completed first.



The Dual & XL Fixtures Two different two-sided fixtures are included with the mill. The "Dual" fixture holds two smaller pieces of wax (46X42 mm) and is capable of creating two ring-sized models in one job (set up for this is explained in Chapter 1). The "XL" fixture is designed for a single, large piece of wax, which is suitable for larger jobs or as many smaller models as you can fit, with supports, into the milling area. Both fixtures hold the wax at two 180-degree angles, parallel to the table. The vertical cutter mills the details on one side of the model; the A axis rotates the fixture 180 degrees, and the vertical cutter completes the other side of the model. These fixtures are used mainly for flat or two-sided pieces such as pendants, pins, earrings, belt-buckles, cufflinks, etc; or for simple rings with no details around the outside of the band, an area missed during this strategy. The Dual fixture can also hold the model at a 90-degree angle to add a top-down cut to a typical two-sided cut - the "three sided" strategy.

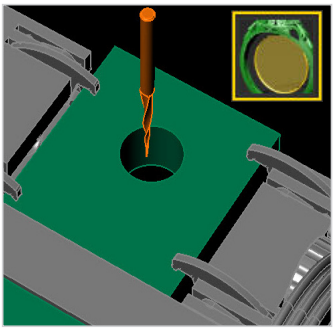


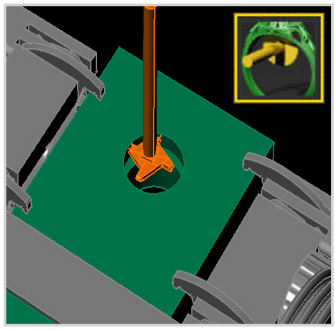
The two 2-sided fixtures are: Dual (Above) and XL (Right). When you input a large two-sided part or a number of smaller two-sided parts into the builder, XL is selected for you.

About Each Fixture An Overview, cont'd.

Each fixture and its associated ring strategies are summarized in this section.

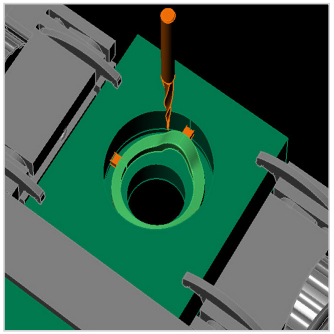
"Cut Out Center" When either the "Cut Out Center" or "Cut Out and Hollow" center option is selected along with the two-or three-sided strategy, the first step when cutting the model is for the 10-dgr cutter to ream out a center hole from both sides (front and back). The software will pause for the user to remove this piece. If the disc cutter will be used, the mill software will prompt you to change to this cutter before completing the hollowing cut or comfort-fit cut. It will then prompt you to change cutters back, before completing the two- or three-sided job as usual.

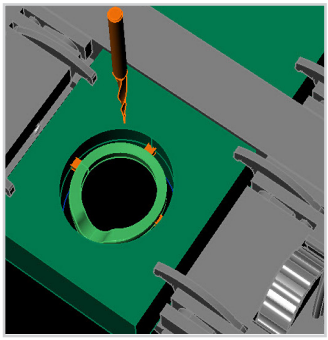




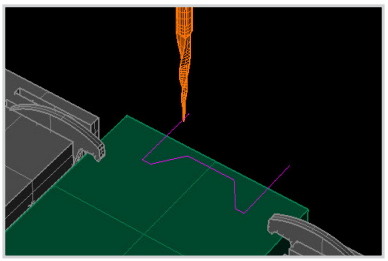
(1) When the user chooses the Cut Out Center (L, above) or Cut Out and Hollow (L and R, above), this is completed before the 2- or 3-sided cuts, while the wax is most rigid.

"Three Sided" Strategy The Dual fixture may also be used for the "three-sided" strategy, which adds an additional rotation to the beginning of the two-sided strategy, holding the wax at a 90-degree angle to the table so the vertical cutter can cut the top-down details of a ring (crown, shoulders), and then complete the two-sided cut. The Mill C Builder adds three supports (left, right, bottom) to hold the model to the wax (no supports are placed at the top). This strategy can also perform an additional two angled cuts (45 degrees & 135 degrees), at the end of the toolpath, to cut only the details missed during the other three cuts. Set-up in the Mill Builder for this strategy (3 or 5 angles) is explained later.

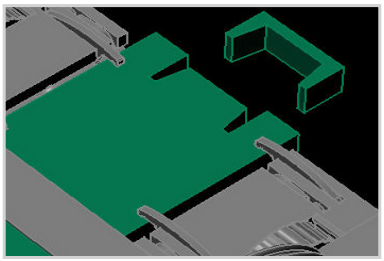




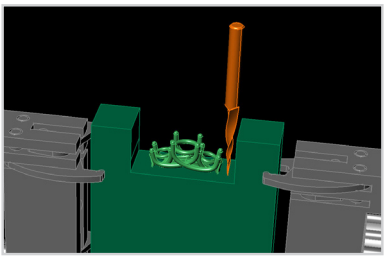
(2) During a 2-sided cut on the dual fixture, two sides of the model are cut: just the front and the back (L and R, above).



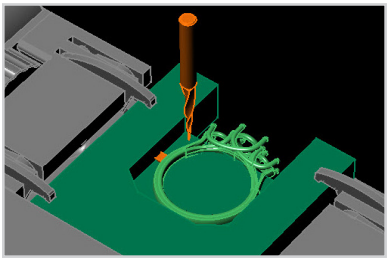
(1) The Three-Sided strategy begins with a cut-out curve, which clears away space for the top-down cut to be made without roughing out all that wax.

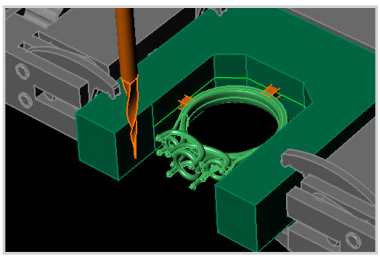


(2) The cut-out piece falls away from the wax. The user must clear it from the mill table before the mill can proceed.

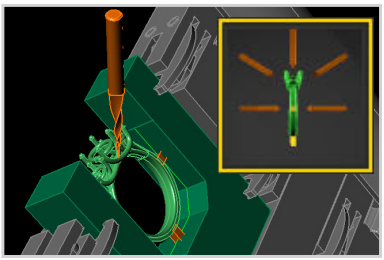


(3) The Top-Down cut mills out the top of the model first. It defaults to a depth of 1 mm inside the finger rail, unless the depth of the cutter is adjusted in Detailed Options.





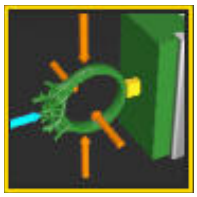
(4, Left & Right, Above) The A axis rotates the part to the typical 180-degree and 0-degree positions for a two-sided cut to complete the front and back sides of the model.



(5) If the user chose"5 sides", the A axis will rotate the fixture to 45 & 135 degrees to complete the angled cuts.

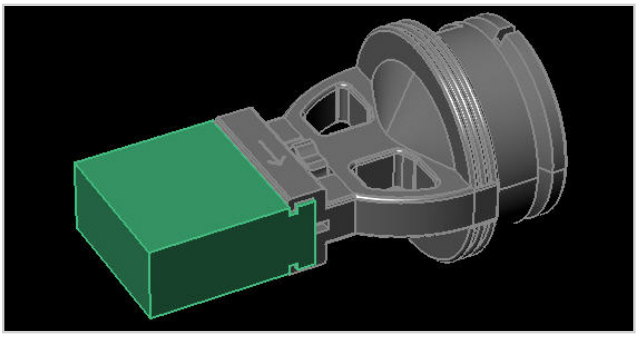
About Each Fixture An Overview cont'd.

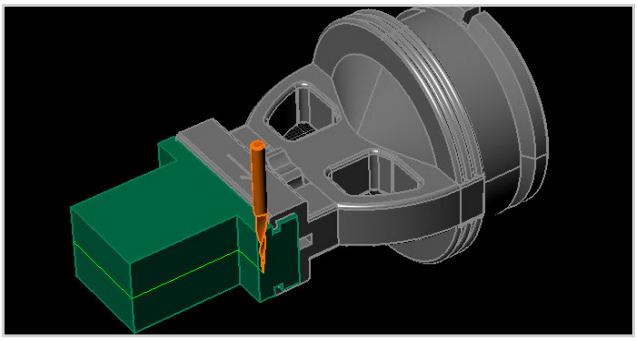
Each fixture and its associated ring strategies are summarized in this section.



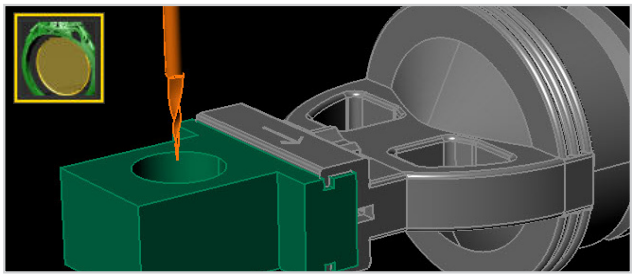
The Base Clamp The newest and most versatile fixture, the base clamp, has a number of strategies associated with it, each of which is easy to understand based on the basic premise of the fixture. This fixture can complete a number of widely-varied model types. Details everywhere on the model will be cut except for those which continue all the way around the bottom, outside of the band. This is thanks to its bottom-support of the wax, its ability to maximize the reach and angles available to both cutters, and its ability to never leave the object unsupported up until the very last cut. The best kinds of models to use with this strategy are rings with details on the top (crown), shoulders, sides, front, and back. It is also great for pendants or other models that have side to side and top to bottom detail (see "Non Ring Strategies" in this chapter). The only part of the ring it doesn't mill is the bottom, outside of the band, so it is not suitable for eternity-style rings or bands with detail all the way around the bottom.

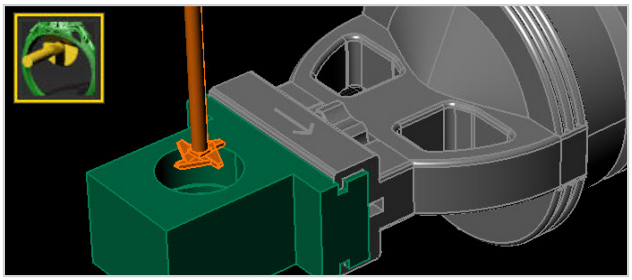
Cut-Out Curve The base clamp strategy begins with the wax parallel to the table. The vertical spindle follows a "Cut Out Curve", created automatically by the Mill Builder to remove wax that would otherwise have to be "roughed away" or cut away by a number of extra passes and extra mill time. The "Cut Out Curve" is cut from the top of the wax first; then the bottom - just past the center of the wax each time - to avoid burying the cutter during either pass. The cut-out wax will fall out or be removed by the user. Clear it from the workspace and indicate in the Revo software this is done to continue the job.



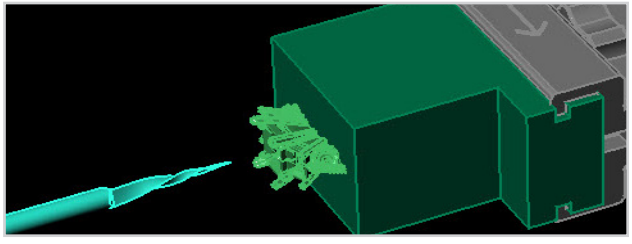


"Cut Out Center" &/or "Hollow" When either the "Cut Out Center" or "Cut Out and Hollow" center option is selected, the next step AFTER the cut-out curve is for the 10-dgr cutter to ream out a center hole from both sides (front and back). The software will pause for the user to remove this piece. If the disc cutter will be used, the mill software will prompt you to change to this cutter before completing the hollowing cut or comfort-fit cut. It will then prompt you to change cutters back before it completes the typical steps in a base clamp cut, which are explained below. Cut-out curves and hollowing are completed early-on in the process, when the most possible wax is left, so the piece is best supported and will not flex during these two cuts.





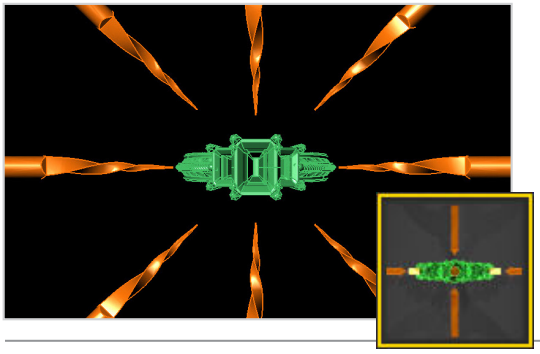
Top-Down Cut On a typical Base Clamp strategy, the next step (after any cut-out and hollowing cuts) is to approach the wax from the "top down" view of the model with the horizontal cutter, milling out the crown of a ring, for example, as well as other details from the "top" or "Looking Down" view that the horizontal spindle is able to reach (based on cutter length and your specific model).



About Each Fixture An Overview, cont'd.

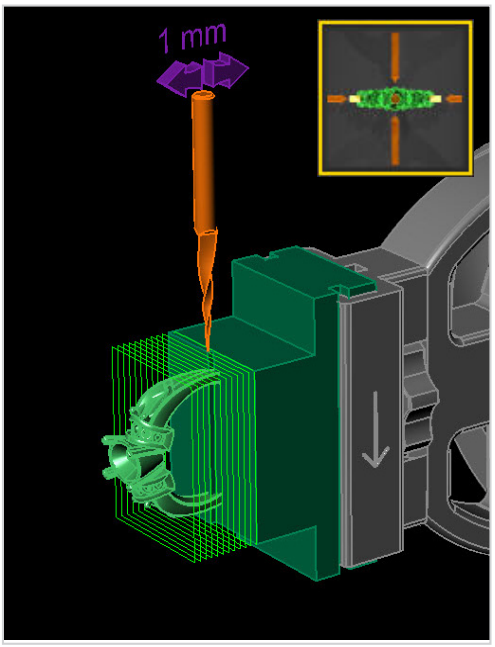
Each fixture and its associated ring strategies are summarized in this section.

4 or 8 Sides It's the job of the vertical spindle next to cut the front, back, and sides of the model as the A axis rotates the fixture at 90-degree angles (for the 4-sided Base Clamp strategies) or 45-degree angles (for the 8-sided strategy). This strategy cuts a 1-mm wide piece of the model; rotates the model 90-degrees (for 4-sided strategy) or 45-degrees (for 8-sided strategy) and cuts another 1 mm-wide piece, continuing to rotate and cut each 1 mm-wide piece of model from the top to the bottom, where the support is. As you can probably guess, this strategy takes the longest to run. During the 8-sided rotation, the mill will only cut on the 45-degree angles what it MISSED cutting at the 90-degree angles, which does help make it more efficient, both in time & part finish.



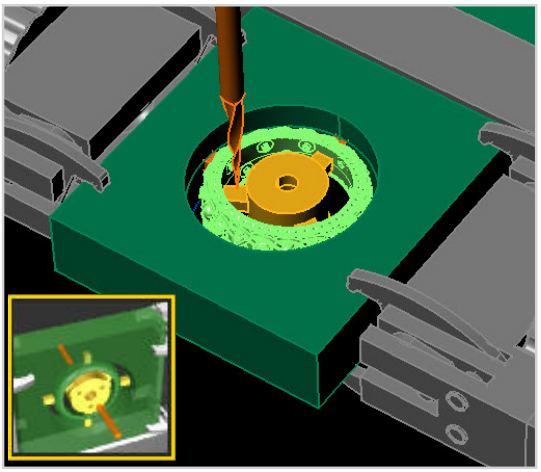
(R) 4-sided: rotates model 90-degrees and cuts a 1-mm wide piece of model at each rotation. Starts at top and works down, 1 mm at a time.

(L) 8-sided: rotates model 45-degrees and cuts a 1-mm wide piece of model at each rotation.

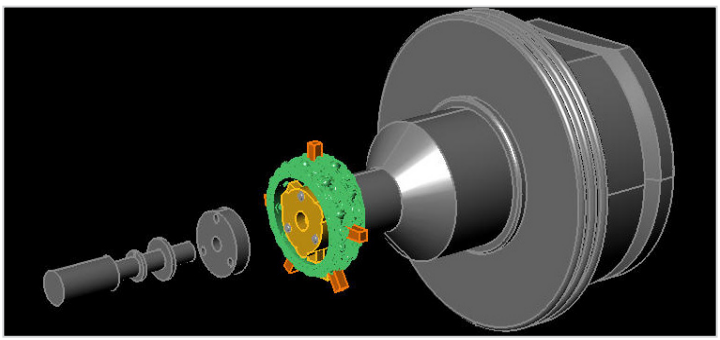


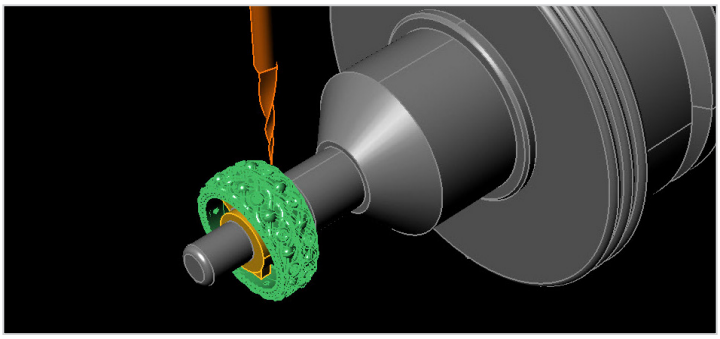
The Hub Fixture While using the mill builder, selecting the "hub" center option together with either the 2- or 3-sided strategy on the Dual fixture or the Base Clamp will add a small wax hub in the center of your model, making it possible to install the model on the Hub Fixture and add a rotary cut to any model that has been cut on the flat.

Order of Operations Prior to the flat cuts, the hub cutouts (a center screw hole, three pin holes, and an arrow) are milled out. Then, the hub is cut as part of the model during the "Front" and "Back" parts of the mill job.



The Hub is cut during the "Front" and "Back" cuts of the 2- & 3-sided strategies on the Dual fixture (2-sided above).



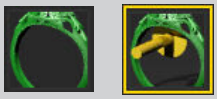


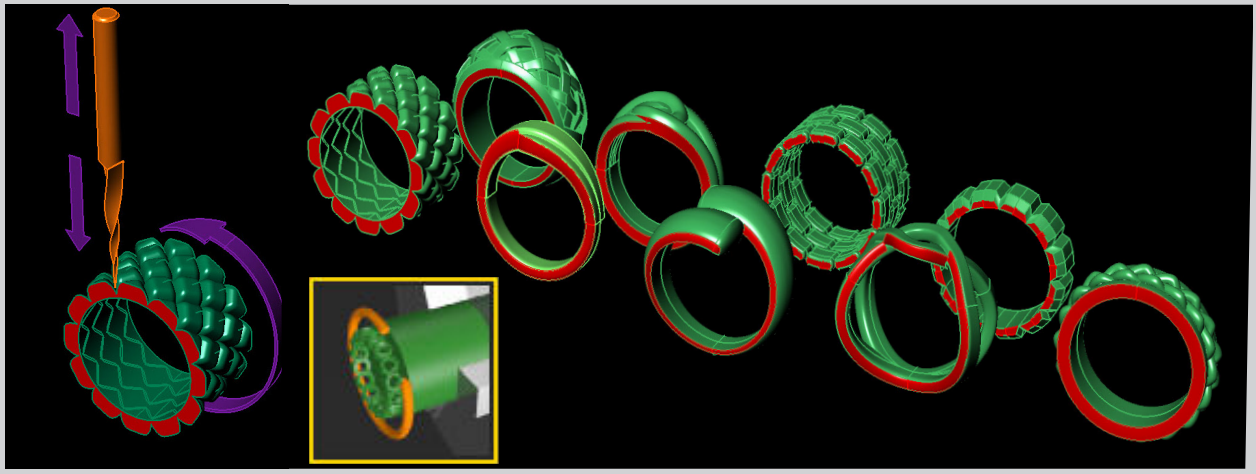
Transfer Model to Hub Fixture To transfer the model from the original fixture to the hub fixture, the user removes the model (with the hub intact) from the original wax and installs it on the Hub Fixture, aligning the three pin holes in the hub with the three pins on the fixture. The arrow milled into the wax should be pointing up and facing you. Apply the cap with three pin-holes; then, slide the two washers onto the screw and tighten the model to the fixture using the screw (above, at left). A simple "centering strategy" to find the midpoint of the model in the X direction at its new location is then completed by the user at the prompting of the Revo software interface. This process is described later in this chapter.

At a Glance: Types of Models Suitable for each Fixture

Rotary Strategy. Models with details all the way around the outside of the band are best-suited to the simple rotary strategy on the 3-Jaw Chuck. If the model has details on the front or back, as seen in Through Finger, or an undercut, inward taper, or groove / other detail at this location, choose one of the flat strategies (two or three sides on Dual fixture) and add a Hub center to complete a rotary cut following the flat cuts.

Rotary Plus Interior Cut. If the rotary model is hollow or comfort-fit, choose the "Cut Out and Hollow" center. An interior cut, performed with the hollowing cutter, will be added to the job. In this case, hollow tube wax MUST be used. The other center option for the Rotary Cut - "Ignore Center" - does not address the center of the ring, so the user must ream out the finger hole after the model is milled.





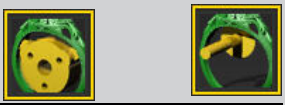
(Above) As the A Axis turns, the Z axis moves the spindle up and down, and the X axis moves the model left to right, to cut the ring. Details in red (side of ring) are missed. The front and back of the ring may ONLY be straight during simple rotary strategy: No tapering, undercut, grooves, gem seats, etc. If these are present, use either the 2-or 3-sided strategy plus the hub.

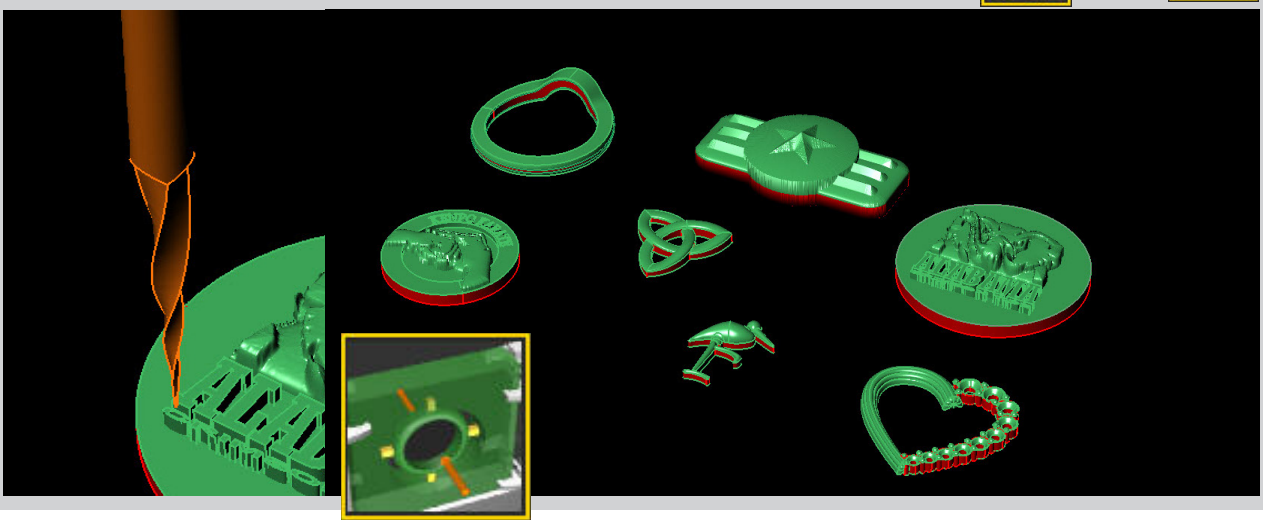
Two-Sided on Dual Fixture. Although this strategy is rarely used for rings, it is quick, and if you are milling a simple ring with no details on the outside of the band that cannot be reached by the cutter, it might be your best choice.

Details that would not be cut using this strategy alone include those around the outside of the band such as grooves, gem seats, lettering, or other details.

Two- or Three-Sided Plus Hub. If you DO have the aforementioned components all the way around the outer face of the design, choose the hub center with either the 2- or 3-sided strategy to add a rotary cut after the flat cuts & reach these details.

Two-Sided Plus Interior. If the model will be hollowed or comfort-fit, select this option to chase the inside of the finger rail with the disc cutter to complete these details.





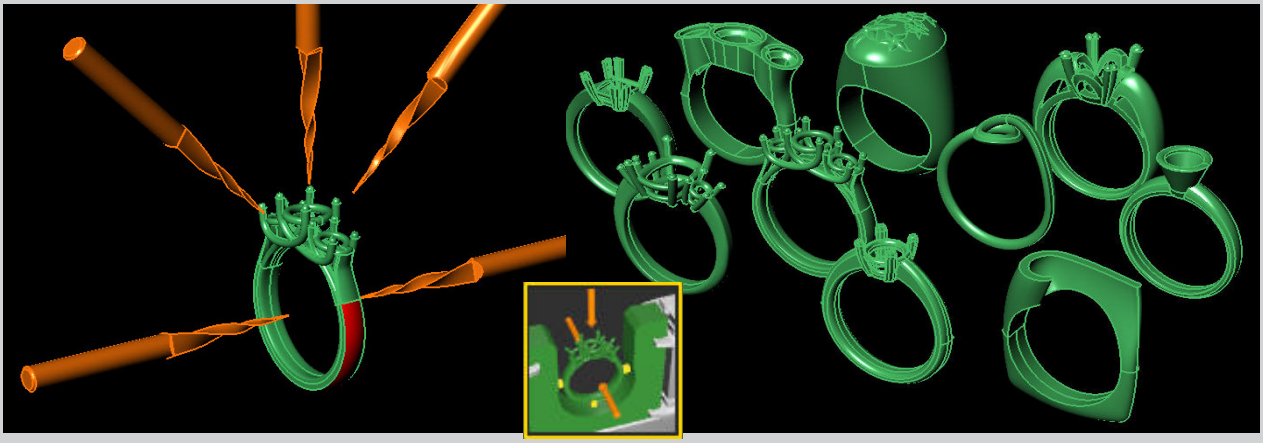
At a Glance: Types of Models Suitable for each Fixture

Three-Sided on Dual Fixture. Select this strategy when the model has head details that can best be reached by a top-down cut.

For this strategy, the top angle will default to a depth of 1 mm inside the finger hole, unless you change this in Detailed Options (explained in a future chapter). Therefore, details on the shoulders or sides of the ring will NOT be milled if they continue too far down the sides. In this case, use the Base Clamp, which completes the top-down cut as well as four (or eight) side-angle cuts, to reach details such as shoulders and side details on the band.

Three or Five Angles? The five-angled cut will take slightly longer; however, it will only mill details which could NOT be achieved by the three-sided cut. So, depending on the model, it may take a only slightly longer - or it could take a lot longer - than the three-sided cut.

Select the five-sided option when there are details such as cuts or gem seats angled up on the front and back of the model; heads or bezels that can benefit from additional angled cuts (as seen in the Side View of the model), or any open work at the top of the ring which additional angles can help clean out.



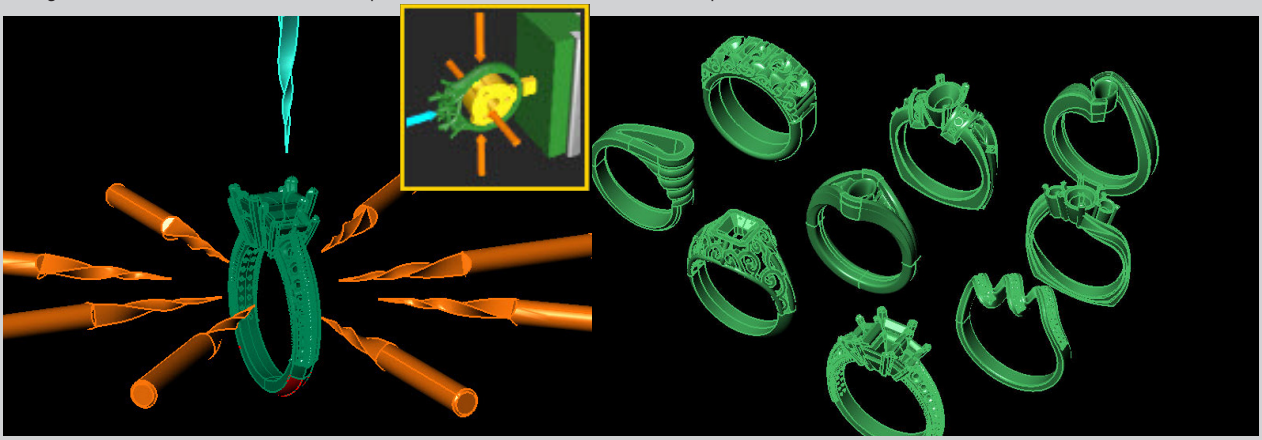
3-Sided Models have details on the top, front and back of the ring, with no details around the outside, bottom of the band (red surface above, left), as these would be missed during this strategy. (Yellow cutters represent 2 additional angles in 5-sided cut.)

Base Clamp. The only details that will be missed by this strategy are those on the outside, bottom of the ring, where the support lives. So if you don't have an eternity band, text, or other important details at that location, this is an ideal selection for reaching pretty much all outer details on a ring.

Base Clamp Plus Hub. If you need to get to those details on the outside, bottom of the ring, but you still wish to capitalize on the angles and reach of the Base Clamp, you may choose the Hub center option with this strategy and transfer the model to the Hub fixture when it's done running on the base clamp.

4- or 8-Sides? Although it will add time to the job, the 8-sided option will try to reach every part of the model that would be missed by the 4-sided cut. Just how much time is added will be determined by how many details can best be reached by these additional angles: it depends on the model.

Generally, if there are angles in the Side View and Looking Down view that angle up and away from the flat sides of the model, on the 45-degree angle, it is a good idea to pick the 8-sided strategy. If the sides of the ring are largely flat, 4-sided will be able to complete it for you.



Base Clamp Models should only be LACKING detail on the very bottom, outside of the ring, where the sprue is usually placed for casting. All other angles: top, left, right, front, and back, can be reached, along with 4 additional angles in yellow, above left.

At a Glance: Types of Models Suitable for each Fixture

Adding the Hub: Add the hub to one of the two flat strategies based on the needs of the model.

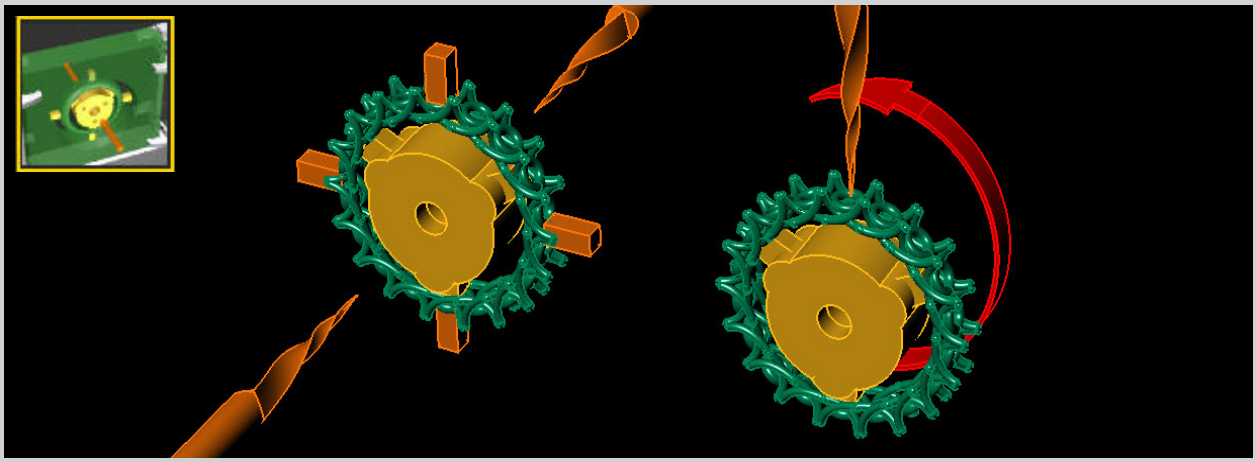
To save time, it is best to use the quickest strategy required for the job. The following are in order from quickest through slowest. As you may have already guessed, one model can be completed using several different strategies.

However, the suggested Hub strategy to use for each type of model is listed below, along with a detailed description of how each one works and when to choose it.

Two Sides on Dual Fixture Plus Hub Select the Center Option "Hub" along with a two-sided model to cut a model with details on the front and back (Two-Sided portion of the job) and around outside of the band (Rotary portion of the job).

If the model has a crown, it is best to choose the three-sided strategy, to add a flat, top-down cut to the model.

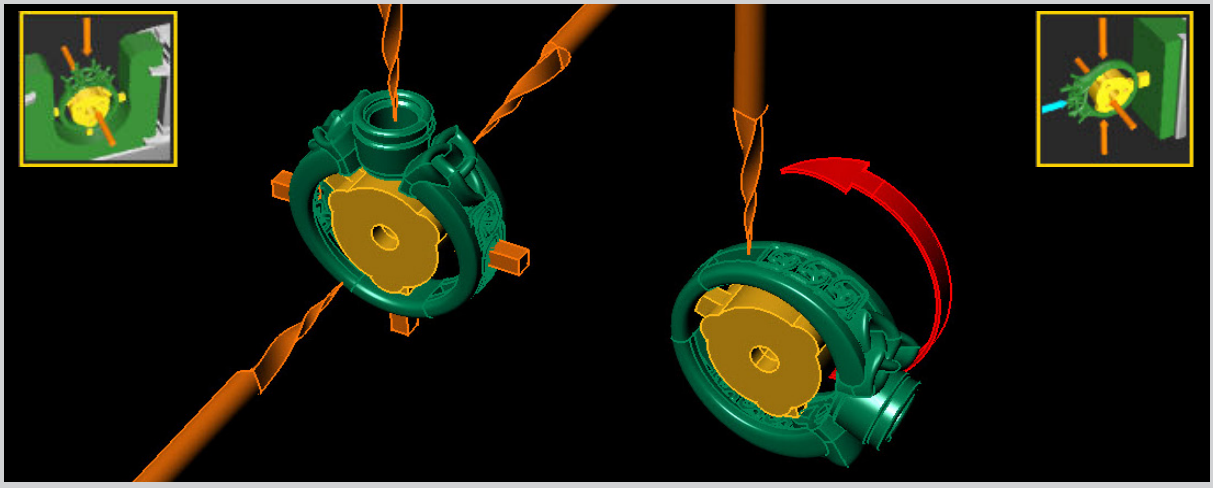
If there are details on the 45-degree angle, such as on a knife-edge shank profile, choose the Base Clamp (8-sided) option.



(L) During the "Two-Sided" part of the job, the mill can create the "Trellis" details on the front and back of the model. (R) During the "Rotary" part of the job, thanks to the hub fixture, the mill can cut the bezels and prongs around the outside of the band.

Three-Sided on Dual Fixture plus Hub Select the Center Option "Hub" with the three-sided cut on the Dual fixture to cut a model with details on the front and back of the model, a head or crown assembly on the ring (Three-Sided part of cut), and details around the outside of the band (Rotary cut on hub fixture). Select 5-sided when there are details on the 45-degree angles in the Side View viewport.

Base Clamp Plus Hub Select the Center Option "Hub" along with a base clamp model to cut the details around the bottom, outside of the band which are missed by the Base Clamp cut. This is pretty rare, since the other flat strategies plus the hub can usually get all the details you'll need, and even when there are details around the bottom, outside of the band, the sprue is usually placed here. Because this strategy takes so long, try and use one of the other flat strategies with the hub if it is possible to do so.



(L) During the "Three-Sided" part of the job, a top-down crown or head / shoulder details can be milled, along with the "Front" and "Back" of the model (including the hub). During the "Rotary" part, on the hub, details all the way around the band are cut.

Using the Mill Builder ...

Screen 1: Supports & Center for Rings

The support strategy you choose for your design will determine which fixture will be used to cut your model. All of the strategies and their associated fixture(s) are described below.

Choosing a Support Strategy

Every model needs to be supported - or, held to the wax from which it is created so that it does not fall off during milling. The first step in Mill Builder places supports automatically based on the ring rail and which support strategy you choose. Choose which set of supports to add to your model based on where the details are: that is, where the model can be connected to the surrounding wax so that the mill will not miss any key parts of your model. The support strategy you choose will determine which fixture will be used to cut the model. Each support strategy is associated with one of the five fixtures that accompany the mill. These five fixtures, and the types of models (or, where the details are on each model) which they can cut best are as follows:

1 & 2) The Dual 3-Sided Fixture & XL Fixture are suitable for models with details on two (front and back) or three (front, back and top) sides. The supports for a two-sided model are placed on the top, bottom, left and right sides of the model. The supports for a three-sided model are placed on the left, right, and bottom of a model. Use the XL fixture for large flat parts or a number of smaller flat parts.

3) The 3-Jaw Chuck Rotary Fixture is suitable for models with details all around the outside of the ring but not on the front or back of the ring. Supports aren't required for a rotary model: instead, the mill knows to cut a "Toolpath Overcut; or, a small amount of wax that connects the model to the wax tube from which it is cut.

4) The Base Clamp Fixture is suitable for models with details at the top, sides, front and back of a model but not at the very bottom, outside of the band. This is because a single support is placed at the bottom, center of the model to connect it to the wax block from which it is being cut.

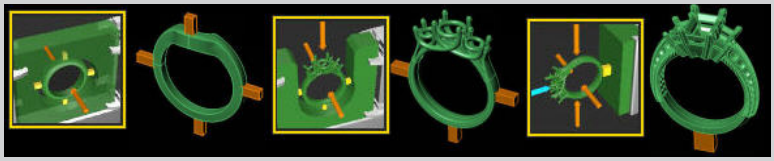
5) The Hub Fixture is used when the "Hub" center is chosen as the center strategy. This fixture makes it possible to add a rotary pass to any of the flat strategies (not the three-jaw chuck), so that virtually no detail on a model is missed.

Builder-Created Supports: Where are they placed?

Top, Bottom, L & R: 2-Sided, Dual Fixture
This strategy uses the Dual fixture for a standard (ring-sized) model. The fixture rotates only to 180 and 0 degrees to cut the front and back of the part. Supports are placed by the Mill Builder at the top, bottom, left, and right of the model.

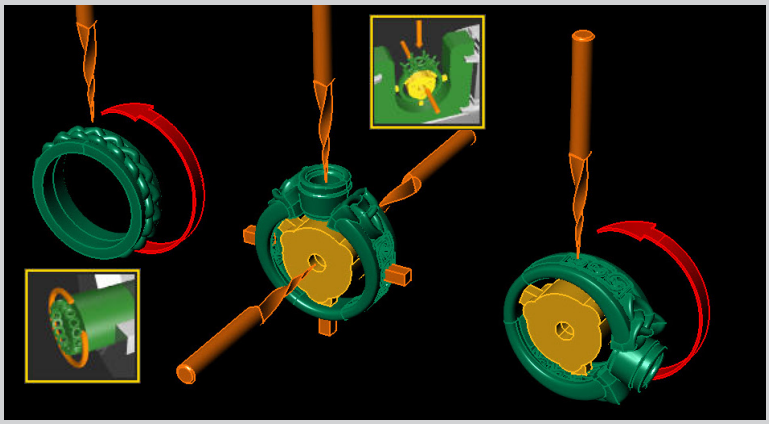
L, R, & Bottom: 3-Sided, Dual Fixture
This strategy also uses the Dual fixture, but it completes a top-down (90 degree) cut before cutting the front and back of the model. Supports are placed on the left, right, and bottom of the model so the mill can cut it from the top, as well.

Bottom Only: Base Clamp
Cut the top, front, back, left, and right, reaching 5 sides (at least) with this fixture. Additional angles are possible, as explained later on in this chapter. Just one support will be placed at the bottom of the model, so only details here will be missed.



No Supports: Rotary
This strategy cuts the ring from a wax tube while rotating the A axis. Details on the front and back of the ring will be missed: only details on the outside of the band will be cut. There are NO supports added, because the mill leaves a "Border Distance", or, a small portion of wax that connects the model to the wax tube.

Adding a "Hub" Center Combine the usefulness of the rotary strategy with other strategies so you don't miss any part of a model by adding a Hub center. After cutting the model flat, you can transfer it from the flat fixture to the hub fixture, which allows the mill to complete a rotary cut on the outside of the band AFTER the flat cuts have been made, helping to ensure all details are cut.



(Left) Rotary; (Center & Right) Three-sided plus Rotary cut on Hub Fixture.

For example, the 2-or 3-sided strategies on the Dual fixture misses the outside of the band. Add a "hub" to one of these and you will be adding a rotary pass to a two- or three-sided strategy: allowing the mill to access nearly every part of the ring.

Click the Hub center option to add the Hub fixture to either of these two strategies. Just remember: The Hub is exclusive of all other center strategies (contour, hollow, e.g.); they cannot be used together.

Building your own Supports

The Mill Builder draws supports automatically based on which support strategy you choose. To draw them yourself, use the "Box" tool found in the Solid menu to prepare them ahead of time in Matrix at the locations where the builder would usually put them; or, at locations which work best for the model you are building and the strategy you plan to use for it. IMPORTANT: Supports you create must be AT LEAST 2 X 2 mm thick and, for best results, 3 mm long so they will be strong enough to support a majority of the models you will encounter and long enough to touch the wax block when a custom border is added around the outside of your model by the builder. Supports must INTERSECT the model!

Turn on "User Supports" to input your own.

Then, in the Mill Builder, turn ON the "User Supports" option and an "In Box" will appear. Select your pre-drawn supports and input them into this preview window. You may also add your own inner supports to affix the hub to your model; simply choose "None" while constructing the hub (explained later). When you choose the hub, only the hub will be drawn, not the supports connecting it to the ring. Select and input your own inner supports for the hub into this preview window.

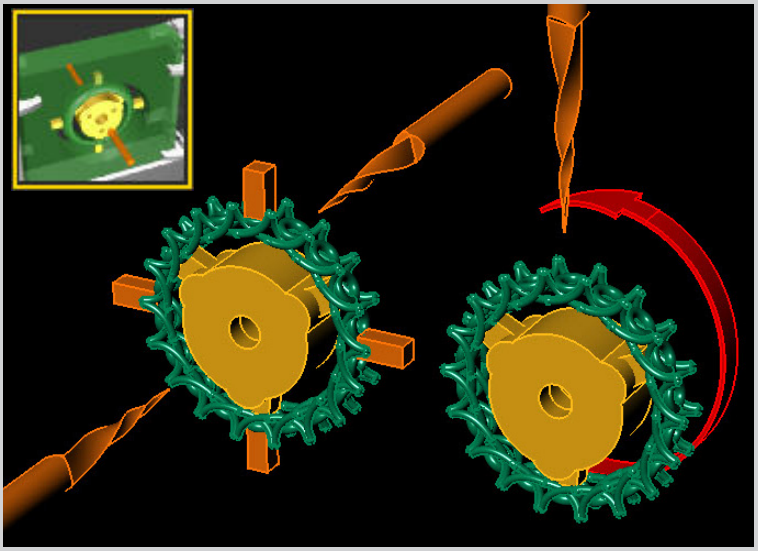
Moving Builder Supports

If you don't wish to draw your own supports but the default placement of the builder-drawn supports don't work for your model, you may move the supports created by the builder: however, you MUST leave them on the SAME SIDE of the model (Left, Right, Top, or Bottom, e.g.) so as not to confuse the builder *and* be sure they intersect the model. There is no need to input them: simply move them after they are placed by the builder and click "Next".

About the Hub Fixture

The "Hub" Center

Select the "Hub" center option when you wish to add a rotary cut to any of the flat strategies. After cutting any of the flat strategies (Base clamp, 2- or 3-sided on the Dual), adding the Hub center allows you to add a rotary cut after all the flat cuts are completed. The mill builder adds a "hub" center to the model, which - after the model (with center hub intact) is removed from the flat wax - attaches the wax to the hub fixture so the outside of the ring can be completed with a rotary cut, in which the ring is turned on the A axis while it is cut.



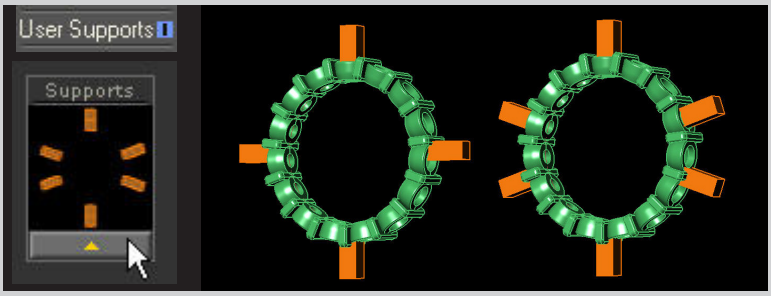
Moving Builder Supports or Building your Own

Build your own supports in Matrix

Once you get used to the placement of the supports by the Revo software, you may wish to create your own when the model calls for it. In Matrix, use the Box tool to create a support 2 X 2 mm thick and at least 3 mm long. Place the supports as close as possible to where they are commonly placed in the Mill Builder. Then, in Mill Builder, turn on "User Supports" and place your supports into the "Supports" In Box when prompted to do so. Make sure they are placed correctly for the strategy you've chosen.

Moving Supports

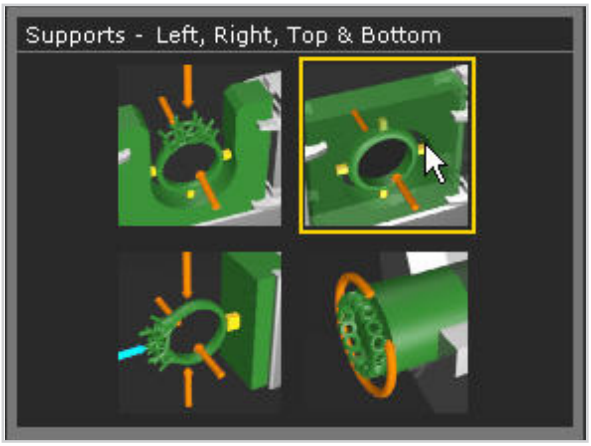
Alternately, you can use the builder to create supports (with "User Supports" turned off) and simply move them. Just remember to keep the builder-created supports on the SAME SIDE of the model where they first appeared so the builder will still recognize them. Auto generated supports offset 1.5mm from the ring rail. Be sure to verify that they intersect the model. This is true for ALL supports.

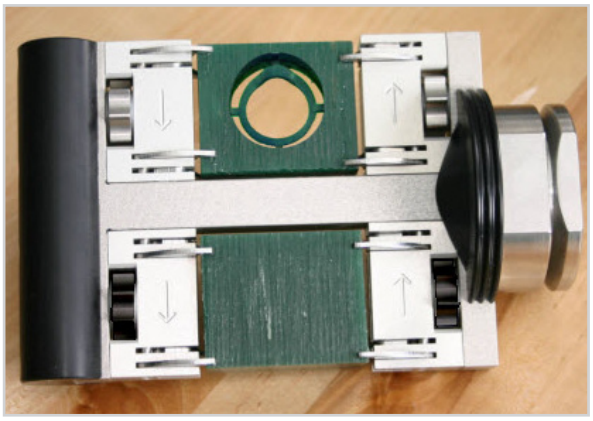


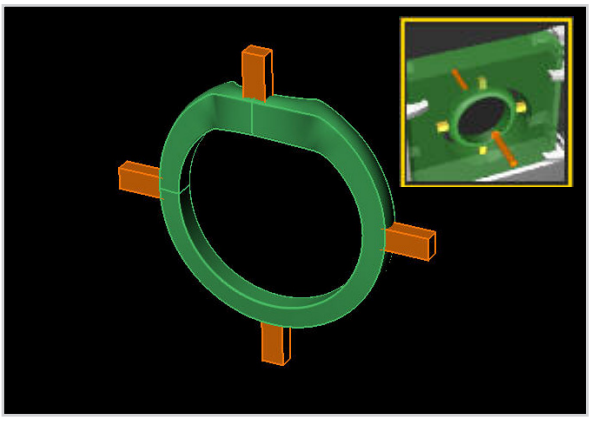
(L) You ONLY need to enter custom supports if you created them in Matrix. Turn on "User supports" to get this box. Or, (Center & R) add supports with the builder and move / copy / rotate them. Keep them on the same side, and just click Next.

At a Glance: Placement of Ring Supports for Each Fixture

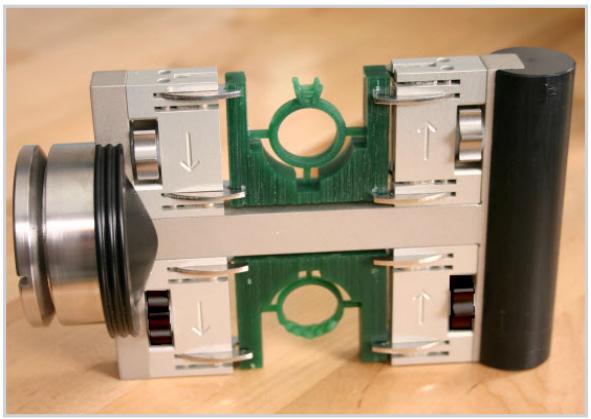
The Mill C Builder found in the Builder menu in Matrix will add supports automatically based on which support strategy you choose. Or, you may wish to build them yourself in Matrix, before entering the Mill Builder, using the "Box" tool found in the Solid menu. Draw a box that is AT LEAST 2 mm X 2 mm thick and 3 mm long (for the Base clamp, it should be AT LEAST 4 mm thick), and position it at the locations indicated in the following illustrations; or as near to these locations as the design of the model will allow. You'll recall that each fixture described in the previous section of this manual requires a different support structure. This quick review will help you match up the supports with the fixtures and review them all in one spot.

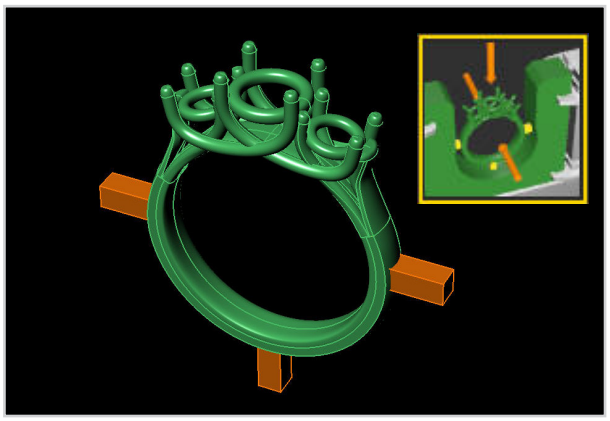




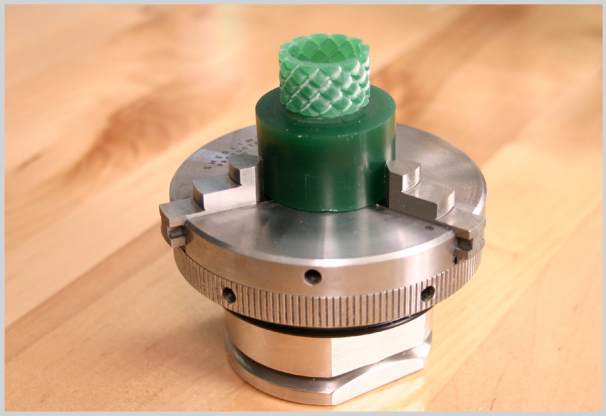


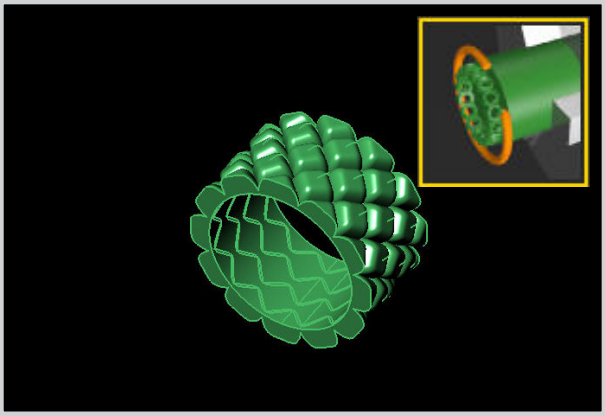
"Left, Right, Top & Bottom": Two Sides on Dual or XL Fixture (Applies to both two-sided on XL fixture and two-sided on Dual fixture) A two-sided part should have supports at the top, bottom, and two sides of the model. This model is usually flat work, with details on the front and back but no details on the side, since the mill cannot cut from the sides during this strategy. A ring with no details around the outside of the band, as pictured above, is also well-suited to this fixture.



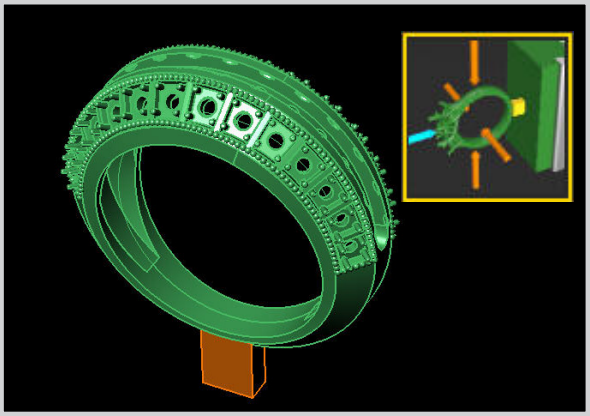


"Left, Right, and Bottom": Three-Sided on Dual Fixture This strategy uses the dual 3-sided fixture and holds it first straight up-and-down (90-degrees to the A axis) to mill the top of the model; then holds it at 0- and 180-degrees to mill the front and back of the piece. Supports belong at the bottom, left side, and right side of the piece when viewed in the THROUGH FINGER viewport.

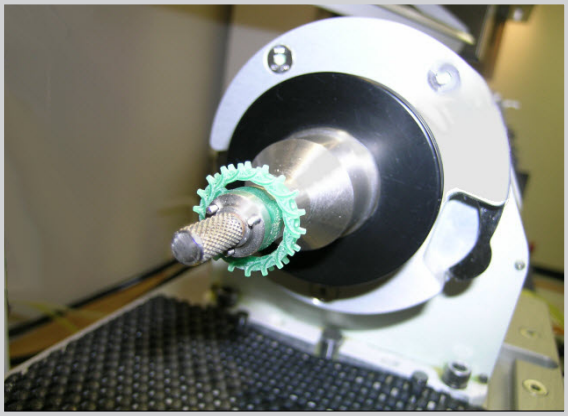


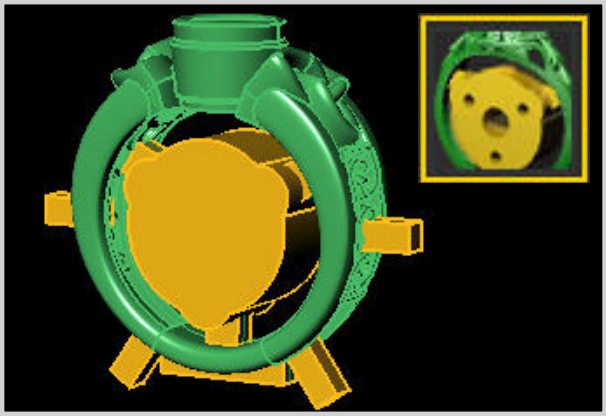


"No Supports": Rotary A Rotary model does not always need user-created supports. Remember that, as the A axis rotates, the part is cut out of the wax tube by the cutter moving up and down in the Z (and left and right in the X). The toolpath automatically leaves a small piece of wax between the part and the wax tube known as the "toolpath overcut", which secures the model to the wax.



"Bottom" Support: Base Clamp The base clamp holds the wax at the base of the model. To attach the model to the wax block, a single support is attached to the bottom of the model (as seen in Through Finger) which should be no less than 4X4 mm thick and 3 mm long, so that it is not too short (spindle could run into fixture) and not too long (must be sturdy enough to support model until all is cut).





Hub Since pieces milled with the hub strategy begin on the Dual-fixture, first choose the strategy (2- or 3-sided) you will start with. Then choose the hub center and the supports will be re-created, with the bottom support becoming two: one on each side of center. In a future screen, the builder will place the hub and accompanying calibration piece at the base of the model.

Using the Mill Builder ...

Choosing a Center Option

After choosing supports in Screen 1, determine how you will cut the center of a ring model based on the following description of each option.

About The Center Options

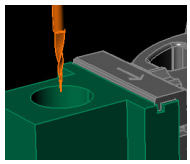
The four center options - "Mill Center", "Cut Out Center", "Cut Out & Hollow", and "Add Hub" - provide four different methods for completing the center of a ring, as well as providing for the use of the Hub fixture described earlier in this chapter. Please note a few important details about the center strategies before reading on:

1 - Center Options are Mutually Exclusive

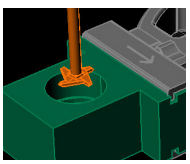
First, notice that each center strategy is mutually exclusive, meaning you can only select one at a time (in other words, you can't have a hollow model with a hub: you have to choose only ONE of the four available center options).

2 - Center Cuts are completed FIRST

Next, please note that, unless "Mill Center" is selected, the chosen center option will be the FIRST thing cut during the toolpath. This can be surprising, but it's actually very helpful, since the model is best-supported at the start (the mill is cutting the selected center out of a thick block of wax while it is rigid. In addition, the "Cut out & Hollow" cut requires a change of cutters, from the 10-dgr cutter at the start, which clears away space for the hollowing cutter; to the hollowing cutter; back to the 10-dgr cutter to complete the remainder of the job. Running these cuts at the start of the mill job allows you to monitor the mill while it prompts you to change cutters, quickly completes the cut(s), then prompts you to change cutters back to complete the remainder of the model.



Cut out Center & Hollow come after cut-out curve and before flat cuts.



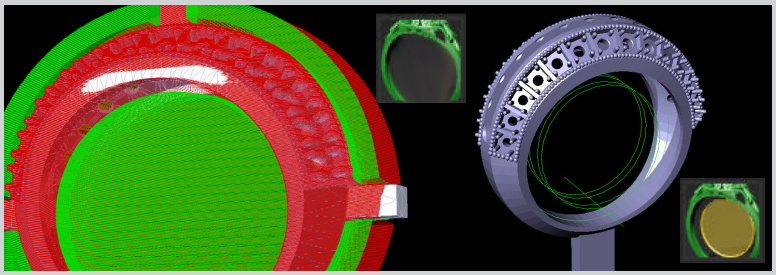
3 - Centers are ONLY offered with rings.

These center options are ONLY available when rings are being milled: NOT non-ring items.

Center Options for Dual-Sided or Base Clamp

Mill Center: Choose this option and the mill will cut away the center just like any other area of negative space in the model: milling away all of the wax, from both sides of the model (front and back), until it's gone.

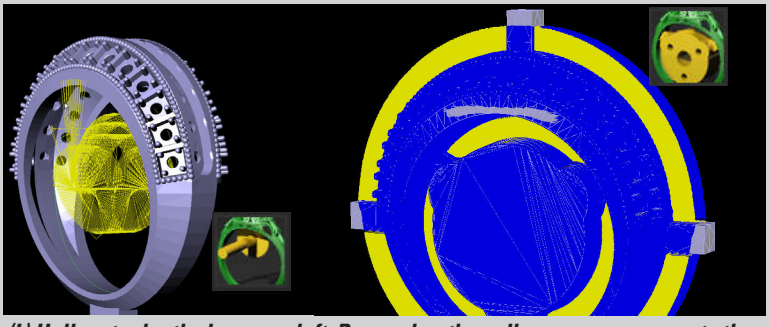
Cut Out Center: You'll notice that it takes a lot of time to "Mill" the entire center. To cut down on time, choose "Cut Out Center", the strategy during which the cutter will automatically follow a curve 2-mm inside the finger rail, from both the front and back of the model, and the center piece will simply fall out. The user must wait until this cut is completed, remove this piece from the model, and tell the mill to resume cutting.



(L) "Mill Center" clears finger hole with each pass and so takes longer. (R) "Cut out Center" clears out this wax first with one curve, leaving less wax to cut later.

Hollow: The mill will first run the "Cut out Center" cut to clear space for the hollowing cutter to do its job. Remove the center piece of wax then follow the directions in the Revo software to change cutters to the hollowing cutter. The mill will complete the hollowing cut before prompting you to change cutters back for the rest of the model.

Add Hub: No cutter change is required for this strategy. The mill begins by cutting the center of the hub from both sides (Front and Back) so that the model is well-supported during this step. It will then cut the remainder of the model, creating the rest of the hub during the typical Front and Back cuts as though it were just another part of the model.



(L) Hollow toolpath shown on left. Remember the yellow curve represents the toolpath calculator from the center of the disc cutter. (R) Hub is cut during the Front and Back cuts during any of the flat strategies.

About Each Center Option

1 - Mill Center This strategy requires the least user intervention during milling. When using either of the Dual-Sided (2- or 3-sides) strategies or the Base Clamp strategy, the mill simply cuts away the material at the finger hole during the flat cuts (front and back), leaving the finger hole clear so that the ring model is created with the finger size which was selected in Matrix. However, if the model was created "hollow" or "comfort-fit", these details will be missed when choosing this center strategy, so read on. NOTE: During the Rotary strategy, choosing this option causes the mill to simply "Ignore" the center of the ring. The user will have to ream out the finger size after the wax is created.

2 - Cut Out Center On either the Dual or Base Clamp fixtures, this center strategy causes the toolpath to start with a "follow curve" operation that traces a circle 2 mm inside the finger rail, cutting down to a depth of 1 mm past the center thickness of the model on both sides (front and back) in order to cut away the center of the ring from the rest of the model.

The mill then pauses for the user to remove this center piece from the model and tell the mill to continue with the job. This option saves time when milling the front and back of the ring, since the toolpath doesn't have to mill away all the wax in the center of the ring the way it does when Mill Center is selected. NOTE: This option is not available with the "Rotary" strategy.

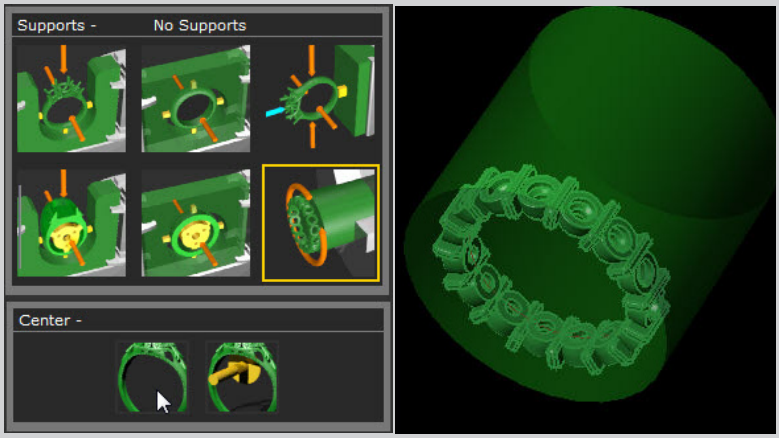
3 - Cut out and Hollow This option may be selected with any of the flat cuts OR the rotary cut on the 3-jaw chuck (as long as Hollow Tube wax is used in this fixture). When selected with any of the three flat strategies, the mill starts with a "Cut Out Center" cut as described above, to clear away space so that the disc cutter can access the center of the model and complete either a hollowing cut or a comfort-fit cut (or a "contour" cut, which leaves the inside of the finger rail flat rather than ever-so-slightly tapered, which results when using the tapered 10-degree cutter). On the Base Clamp and Dual Fixtures, this cut is performed with the Vertical Spindle, requiring the user to change cutters between the "Cut Out" curve and the Hollowing cut, and change back following the hollowing cut.

On the rotary fixture, this cut can ONLY be completed with a hollow tube of wax, which ensures the center of the ring is already cleared away and the hollowing cutter can easily access the area it needs to cut. In the case of the rotary fixture, the hollowing cutter is installed on the Horizontal spindle. NOTE: When choosing the Hollow option, the ring MUST be hollowed in the Matrix model, or this strategy will only chase the inside of the finger rail for a "contour" or smoothing cut. (In other words, the Hollow cutter cannot not hollow a ring if it isn't already hollow in the Matrix model.)

About Center Options: Rotary Strategy

Ignore: When the Rotary strategy is selected, "Ignore" center will not do anything about the center of the model. You will therefore need to "ream" the wax to the finger size manually.

Hollowing: This option may only be selected when the user will install a hollow wax tube into the Rotary fixture. In contrast to the flat fixtures, the mill software will prompt you to install the disc cutter in the Horizontal spindle (flat fixtures require hollowing cutter in Vertical spindle). This cut is best for a hollow or comfort-fit model, or to properly size the ring on this fixture.



4 - Add Hub On either of the Dual fixtures (Dual Sided, 2 or 3 sides; Base clamp) you may wish to add a rotary cut to the part in order to complete details on the model that would not be reached by the flat cuts alone. NOTE: The hub is ONLY available with the flat strategies (Base Clamp, 2- and 3-sided on dual), NOT the rotary strategy (3-Jaw Chuck). In addition to adding a hub to the center, a base piece is added at the bottom of the model to accommodate the "centering" method necessary to transfer this model from the Dual fixture to the Hub fixture.

Selecting this center strategy causes the mill job to begin with four cut-out curves and an arrow. One cut-out at the very center of the model provides space for the screw that attaches the model to the hub fixture. The next three cut-outs are smaller, and hold the three pins that help secure the model during milling. The arrow is milled into one side of the hub only and points up at the top of the model so the user will know which way to align the model when transferring it to the hub fixture (the arrow should be pointing up and visible to the user when the fixture is attached to the A axis prior to running the rotary portion of a hub job).

After following these cut-out curves (and requiring no user intervention), the mill will then return to cutting the model in its usual way. While completing the front and back cuts, the hub will be cut in the center of the model, along with a piece at the base of the model which assists in centering the model during the rotary strategy. Last of all, a "hub cal" cut will be made into the piece. After the user transfers the piece to the hub fixture, the next "hub cal" cut is made so the user can instruct the mill how to center the part (fully explained later). Then, the user instructs the mill to complete the rotary cut.

Using the Mill Builder ...

Screen 2: Wax Size & Type

By calculating the size of your model, the program will tell you the minimum wax size available for the fixture you selected.

The Wax Size Offered

The Matrix Mill C Builder can calculate the size of your part and the minimum required wax size from which it can be cut, based on the supports you chose in the previous step. Remember that each support strategy has an associated fixture. Each fixture holds different standard wax sizes, which are displayed and may be selected in this screen.

Ghost wax appears in selected size. Check it for fit.

To see in future pages, choose "Show Wax":

So, based on the size of your model and the fixture you selected with which to cut it, the smallest possible wax size will be recommended to you by the builder. This size is highlighted in the list, and a ghosted image of the wax block appears onscreen so you can visualize how it fits.

The builder-selected wax is the MINIMUM possible wax size with which you can cut this model. Smaller wax sizes will NOT be made available to you in this screen.

Choose ACTUAL Wax Size

If you are using the minimum wax size offered by the builder, verify your selection and proceed in the builder by clicking "Next". If you are using a larger size, make sure you select the ACTUAL wax block you are using.. This is VERY important so that the mill knows whether the cutter is long enough to reach all the details through the rough (uncut) wax, or whether it needs to automatically add "roughing passes" during toolpath creation. Roughing passes are multiple shallow cuts that cut more quickly (with lower part definition) in order to clear away wax so that, the next time through, the cutter can complete the details of the model without breaking (cutters can break when plunged too deeply into rough or uncut wax). More details on these in Chapter 3.

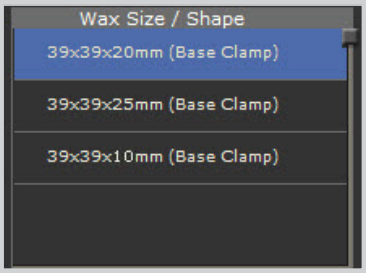
So, choose the wax you are ACTUALLY using for this job, and click Next to proceed in the builder.

Tips on Choosing a Wax Blank / Sizes Available

1 - Only stock wax size(s) are offered. The builder will only offer wax sizes that can work with your model: none that are smaller. The selected size is the smallest possible size you can use for this project. Even so, check to ensure the wax size is sufficient.

2 - Choose the Actual Wax Size being used. Choose the ACTUAL wax size you are using to mill the part, or the closest one to it, if you are not using the size recommended by the builder.

3 - What to do if the builder stops you: The builder will NOT let you use a wax size that is too small for your model. If you choose a strategy with waxes that are too small to accommodate your large model, you will need to go "Back" a step and choose a different strategy that CAN provide wax large enough for your model.



(Far Above, Left) Dual Fixture: Wax blocks of 46X42X5 mm, 10 mm, 15 mm, or 20 mm thick may be selected for use with the two-sided fixture (suitable for the two- or three-sided strategies).

(Far Above, Right) Rotary Fixture: The Three Jaw Chuck can accept wax tubes of 22, 26, 27, 33, and 49 mm in diameter.

(Above, Left) XL Fixture: Wax blocks of 95X95X10 mm, 15 mm, or 20 mm may be selected for use with the XL fixture (used for large parts or multiple parts that can be milled on two sides).

(Above, Right) Base Clamp: The base clamp can accept the 39 X 39 X 20 mm, the 39 X 39 X 25 mm or the 39x39x10mm grooved waxes.

Adding a New Wax Size to the Wax Library

Some milling jobs may require a wax that is outside the preset dimensions of waxes within the Revo C mill library. If you are trying to use a specific milling strategy and the part you are milling is just a bit larger than one of the standard Revo wax sizes, the software will indicate "No Wax Fits This Part". This warning will appear in the first screen and you can either choose to change strategies or proceed to the second screen and opt to add a new wax size to the Wax Library.

Click on the "Waxes" option to enter the Wax Library. A warning message will appear which essentially states that here you are not choosing a wax to use for a mill job, you are editing or creating definitions for a new wax piece to use for any mill job. Click OK and the Wax Library will open, displaying a list of available sizes. To add a new item, you can click ADD. However, It may be easier to edit an existing wax to your specific dimensions rather than adding a wax from scratch. To do so, choose to COPY. This can simplify the process. When you click COPY, a copy of the wax definition will appear at the bottom of the list and the Name field will be populated. Edit the text in the "Name" text box. Use the new wax measurements as the name. For instance, let's say the width for this wax needs to be expanded from 25mm to 30mm. Name it as such. The wax image will appear to the right with dimension windows open that allow change. In this example, the Thickness field would be changed to 30. Once all the values have been entered, simply click "SET". Your new wax block will now be listed in the wax library. This wax will be available now and for any future milling tasks that require a non-standard Gemvision wax size. Click the red X to close the library window.

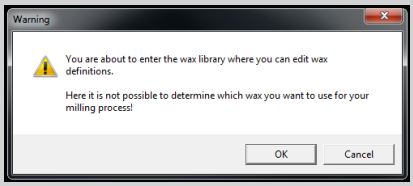
It is important to realize the allowable milling envelope in which you can safely mill before adding any new wax to the wax library. The software will limit the size you can enter into the library based on the clearance needed for any fixture movement during each specific strategy. In other words, if the wax size you are trying to create is going to cause the wax to collide with the table of the mill during the 180 degree rotation when using the XL fixture, the software will not allow you to add this size of to the wax library. Instead this value will be locked, or you will be limited in the value you may enter into the text box. If an unrealistic size wax is entered into a text box and you click Set, a dialog box will appear indicating the limitation of this value.

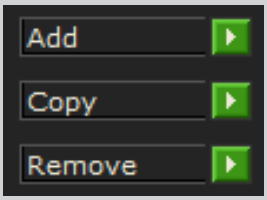
An Overview of Adding a Wax to the Wax Library

1: Object in Matrix
To open the Revo C builder in Matrix, you will need to first have an item in your viewport (any item will suffice). Then click on the Revo C builder button. You will NOT need to add this tem to the Model window.

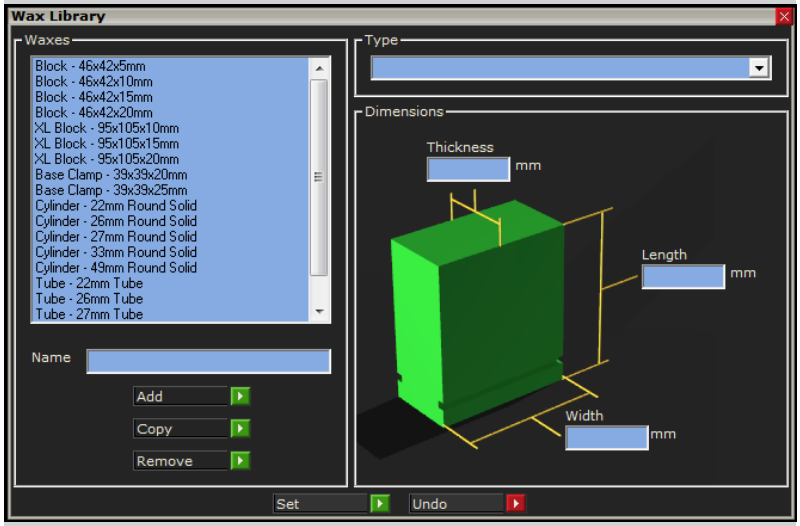
2: Warning
When you click Waxes a warning message will appear which essentially states that here you are not choosing a wax to use for a mill job, you are editing or creating definitions for a new wax piece to use for any mill job.

3: Add - Type
If you choose to add a wax, first choose the Type of wax from the drop down menu. Choose from the following options; Base Clamp, Cylinder, Tube, Block, XL Block. The image below will display the corresponding wax.





If no wax fits the part you are trying to mill, "Add "a wax and define every dimmension of the wax, or choose to "Copy" an exisitng wax and alter it's dimmensions.



4: Add - Name
Name the new wax according to the wax type and measurement. Best practices is to use the same naming convention so after the wax is named and the wax library is closed, the next time you reenter the library the new item will appear alphabetically alongside similar items.

5: Add - Dimensions
Within the preview image input a measurement for each of wax dimensions into the corresponding text box. After all of the values are entered click "SET". Your new wax block will now be listed in the wax library and available for milling. Click the red X to close the library.

6: Copy
To simplify the process choose an existing wax size from the list, which is similar in type and size to the wax you will be milling. Click Copy. Change the name to match the dimensions of the new wax size. Change any necessary dimensions. Click "SET".

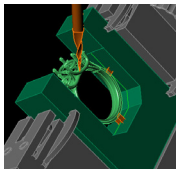
Using the Mill Builder ...

Screen 3: Flat Sides

JUST the Dual 3-Sided and Base Clamp fixtures offer options for reaching additional flat sides - 3 or 5 sides on the Dual, and 4 or 8 sides on the Base Clamp.

Additional Angles, 3-Sided Dual Fixture

Choosing the Three-Sided strategy on the Dual fixture offers an additional screen in the Mill Builder menu allowing you to choose between the typical "Two-Sided" cuts (Front and Back);"Three-Sided" cuts (Top, Front, and Back of model); or add an additional two cuts to that (at 45- and 135-degrees) for the "Five-Sided" strategy.



After top, front & back are cut, the Dual fixture rotates to the 2 new angles.

How these two additional cuts work is as follows. Think of how the Top-down cut is made during this strategy: by the A axis rotating the fixture to the 90-degree position. Similarly, the Front and Back cuts are made by the A axis rotating the fixture to the 180-degree and 0-degree positions. Angles that fall between these three cuts, or 45 and 135 degrees, are the angles of the two additional flat cuts.

Based on the needs of your model, select either 3-sides or 5-sides in this screen & click "Next".

Additional Angles, Base Clamp

Also recall how the base clamp works: by holding the model at its base, it first makes a top-down cut on the model using the horizontal spindle. It then comes in with the vertical spindle and cuts 1-mm of the model on the flat; rotates it 90 degrees and cuts 1-mm flat; and continues rotating it 90 degrees and cutting 1-mm on the flat until the entire model has been milled: all four sides.

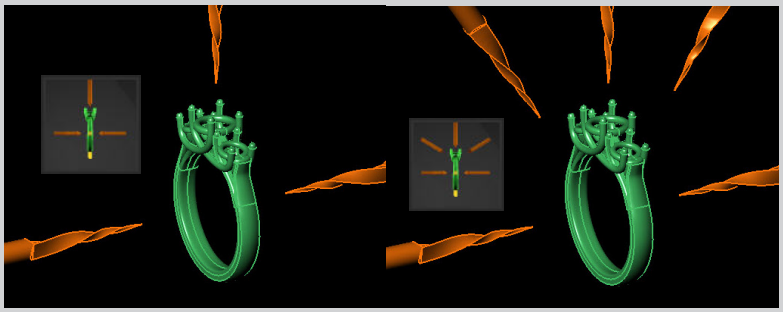
When choosing the "8-sided" option available on the additional builder page that appears, the model will be cut the same with regards to the top-down cut made by the horizontal spindle. The first cut will be done on the flat, as well. Then, the A axis will rotate the model 45 degrees (NOT 90), and make a flat cut at THIS angle. It will continue rotating the model 45 degrees until all EIGHT sides are cut.

Based on the needs of your model, select either 4-sides or 8-sides & click "Next".

Additional Angles, 3-Sided Dual Fixture

3-Sides: Choosing this option does the typical 3-sided cut, which approaches the model first from the top down, with the fixture rotated at 90 degrees on the A axis. The fixture is then rotated by the A axis to the 180- and 0-degree angles to cut the Front and Back of the model.

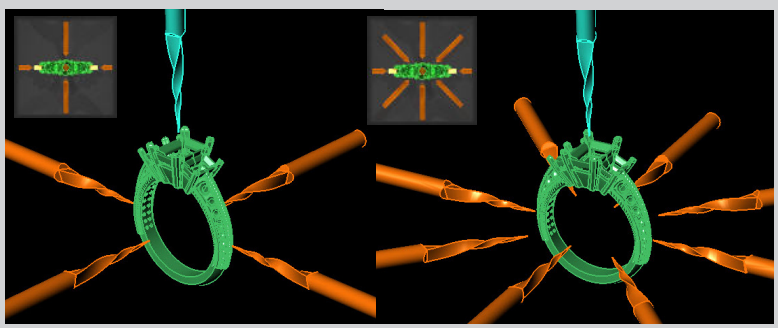
5-Sides: The top-down cut, at 90-degrees, is the same for this strategy. Then, instead of cutting at just 180- and 0-degrees, the A axis ALSO rotates the fixture to the 45- and 135-degree angles, to approach and better clean out the model from these sides, as well. Choose this option if your model requires it.



Additional Angles, Base Clamp

4-Sides: This option completes the typical base clamp cut. To begin, the top of the model is cut out with the Horizontal cutter. The Vertical cutter then completes 1-mm of one side; the A axis rotates 90 degrees, and the next 1-mm is cut. This process repeats until all 4 sides of the model are cut.

8-Sides: The top-down cut made by the horizontal cutter is the same; as is the front cut by the vertical cutter. Next, instead of rotating the fixture 90-degrees and completing 1-mm on one side of the model, the fixture is rotated 45-degrees and 1-mm is cut on this angle. This is repeated for all 8 sides.



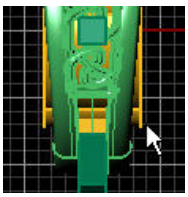
Either strategy may be chosen; however, the 5-sided or 8-sided strategy may cut additional parts of the model that were missed during the 3- or 4-sided cuts.

Using the Mill Builder ...

Screen 4: Center Supports

When the user selects the Hub Center to add a rotary pass to any flat job, a menu to build and input the hub appears.

If you selected the Hub center option in the first screen of the builder, the "Center Support Creator" screen appears next.



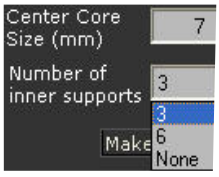
Make hub 1 mm thicker than narrowest part of ring.

1 - Center Core Size

In this screen, begin by selecting a "Center Core Size". This indicates the thickness of the hub. To size the hub, take a look at the model in the Side View viewport. The hub should be 1 mm thicker than the narrowest portion of the ring. So in the case of a 6 mm wide ring (at the thinnest part of the ring), as here, select 7 mm for the width of the hub.

2 - Number of Inner Supports

Using the drop-down menu, choose "3, 6, or None" for the number of inner supports, depending on the needs of the model. "3" will support most models best. Use "6" for more locations at which to support a very delicate or complex model. Or, you may model your own inner supports prior to entering the Mill C Builder and choose "None" in this screen. If you need to make your own supports, you must be CERTAIN to include your pre-made supports when entering the hub into the Preview Window.



Choose Center Core Size & supports before making hub.

3 - Make Hub & Input into "Hub" Window

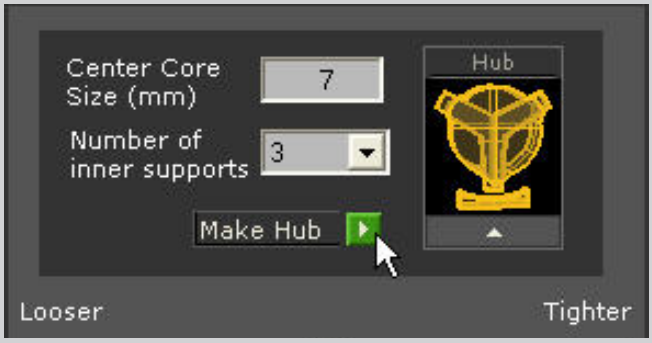
After indicating a center core size and a number of supports, click "Make Hub" and the hub will be added to the model. It remains selected until you click somewhere else in the viewports, so now is a great time to click on the "In" arrow beneath the "Hub" preview window to input it into this window. If you deselect the hub, MAKE CERTAIN you select the center hub, center supports ("Y"-shaped model or any you created yourself) AND the rectangular centering support at the base of the model. Use the "Select" option to automatically highlight all three portions of the hub.

Steps to Create Hub in this Menu

1 - Choose a Hub Thickness in mm. Examine the model in the Side View viewport. The hub should be more narrow than the wax blank size.

2 - Number of Inner Supports. Choose "3" center supports for most models. These affix the hub to the inside of the model. If you have a very complex or delicate model, choose "6". If you have modeled your own center supports in Matrix, choose "None".

3 - "Make Hub"; Input into Hub Window. Select Hub, "Y"-shaped supports (DON'T FORGET any supports you may have created yourself if you chose "None") AND the rectangular shape at the base of the model (used for centering later) and input into the Hub preview window.



Location Pin Size

Do NOT adjust this setting unless the three pin holes on the hub cut by the mill are routinely either too small to fit onto the hub fixture or so loose the model can be "wiggled" on the hub. The pin holes should fit VERY securely, but if you cannot get the model onto the hub, you may adjust this toward "Looser"; or, if the model is so loose on the pins that you can wiggle it, you may adjust this toward "Tighter". Otherwise, do NOT change it. Note: If hub fits loosely, it is likely you are milling with a broken cutter.

Using the Mill Builder ...

Screen 5: Cutters & DBT

In the final screen of the Revo Builder before Detailed Options, choose a cutter and assign the toolpaths step over. Then, create the .RVOC file and toolpath.

1 - Selecting Cutters

Check that the cutter selection in this screen is correct for the job you are working on. To review, the 10-degree cutter is the one you'll use for most jobs. If you will be using a disc or a hollowing cutter, this will likely be the Revo 12 mm hollowing cutter.

If you wish to change cutters, beware: only the 0.032 in. flat cutter (and the default 10 degree cutter) can cut rough wax. The other cutters (0.016 in. flat and 6 degree tapered) CANNOT cut rough wax and will break if you attempt to use them to do so. They should only be chosen as finishing cutters during any operations or milling passes you may add to the job in the Detailed Options screen (explained in a future chapter). So, for all but straight-sided models, select the 10 degree cutter. When straight sides are needed (and not too many tiny details, as it's a bigger cutter) select the 0.032 flat cutter.

2 - Distance Between Toolpaths

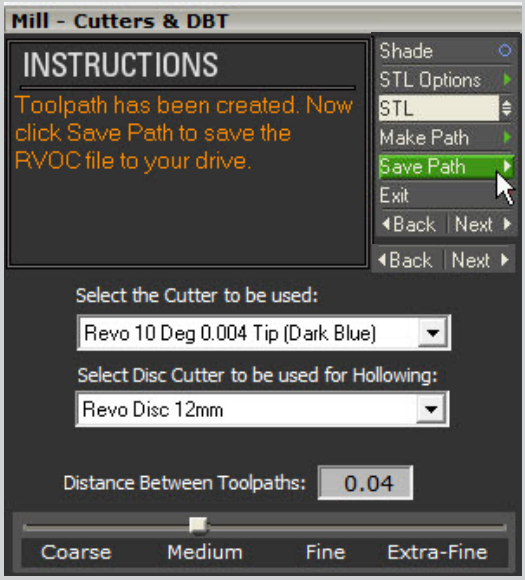
The Distance Between Toolpaths and Step Along Toolpaths - are controlled using the slider or by entering a value in the textbox. Click and drag the slider from Coarse (0.05) to Extra Fine (0.02) to control this value. The default value of 0.04 will suffice for most models.

Each time this curve reaches one side of your model, it must turn around and come back across the model. The length of the turn the cutter will make when it reaches the end of the model and turns around to come back over the model to continue the toolpath is referred to as the "Distance Between Toolpaths". This value is defined in the Revo C Builder, and assigned by the Coarse, Medium, Fine, and Extra Fine settings found in the "Cutters and DBT" screen in the builder.

The "Distance Between Toolpaths" is displayed in mm. So for example, on "Fine", the cutter only moves 0.03 mm when it turns around to make another pass.

Cutters & DBT Selection Menu.

Starting at the upper left-hand corner of this menu, ***1 - Click the drop-down menu to choose a cutter.*** The first cutter is the main cutter to be used for this job. See description of each cutter in Chapter 1. The second cutter represented here is the hollowing cutter (this option only appears if a hollowing cut is selected). Change it the same way: select cutter from dropdown menu. ***2 - Set Distance Between Toolpaths.*** The default value, 0.04, will be sufficient for nearly every job. However, if you wish to raise or lower the definition of this cut, click and drag this slider from Coarse to Extra-Fine. ***3 - Choose Options described below.***



Shade. Click to toggle from wireframe to shaded view of model.
Show Wax. Click to see ghosted wax from Wax Selection screen.
STL Options The STL or Mesh version of this model is used to create the toolpath. In 90 % of cases, the default options will do a great job. If your model demands more definition, check out the part of this section that instructs you how to change these values to improve cut definition.
Nurbs After creating the toolpath, you may wish to view the "Nurbs" version of the model, the "STL" view, or "None" (just the toolpath). Toggle this menu to choose one.
Make / Save Path. Click Make Path and the builder will create the STL version of the model and a toolpath for the model. Then, "Save Path" will appear. Click it to create and save the .RVOC file that directs the mill how to cut this model. Load this file into the Revo software.
Save Style. Click "Save Style" to save all the settings you just established for this model. Using "Load Style" in the first builder screen, you can load them for another model.

3 - STL Options

The mill must create a mesh version of your model "behind the scenes" in order to make this toolpath. How this mesh is made is defined by the STL options. Usually, the defaults will work just fine. However, if you wish to tweak them to add definition to your model, click the "STL Options" button. The best changes to make are to cut the "Minimum Edge Length" (0.01) and "Maximum Distance Edge to Srf" (0.02) in half, to 0.005 and 0.01, respectively. WARNING: Depending on your model, lowering these values MAY create a file that is too large for the mill software to handle!

4 - Make Path

Click "Make Path" to create the toolpath curve. This step does two things: it creates the STL (mesh version of the file) necessary to make the toolpath; and it creates the toolpath, or the curve that the cutter will follow when cutting the wax.

5 - Save Path

After the toolpath is made, the "Save Path" option will appear just beneath "Make Path" in the options menu. Clicking this option creates and saves the RVOC file, which contains the toolpath in a format the mill can understand. This is the file you must transfer to a jump drive to load onto the mill, so save it to a location on your computer that you can remember easily.

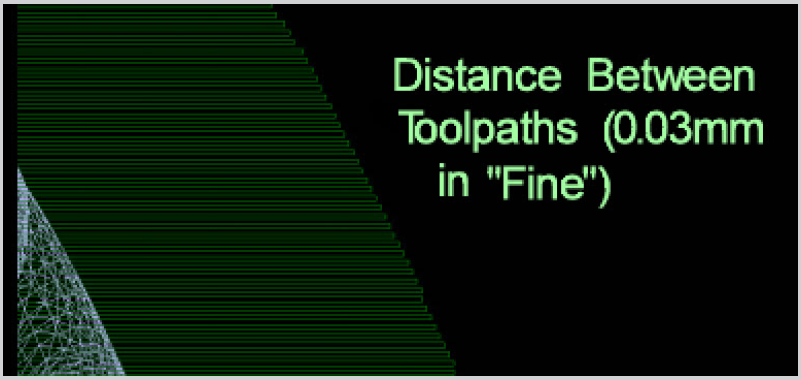
6 - After Creating the Toolpath

It is a good idea to inspect the toolpath and make certain it looks right and doesn't miss any of the key elements of your model. Do this in the "Detailed Options" screen by clicking "Next". Detailed Options is thoroughly explained in a future section of this manual.

About "Distance Between Toolpaths"

Coarse Through Extra Fine: Click and drag the slider from Coarse (0.05 mm value) to Extra-fine (0.02 mm value) to assign the "Distance Between Toolpaths" value for the job you're setting up.

Distance Between Toolpaths: The curve created by the builder runs back and forth over the model. When it reaches the end, it moves over this amount (0.03 mm on the default "Fine") and makes another pass over the model.

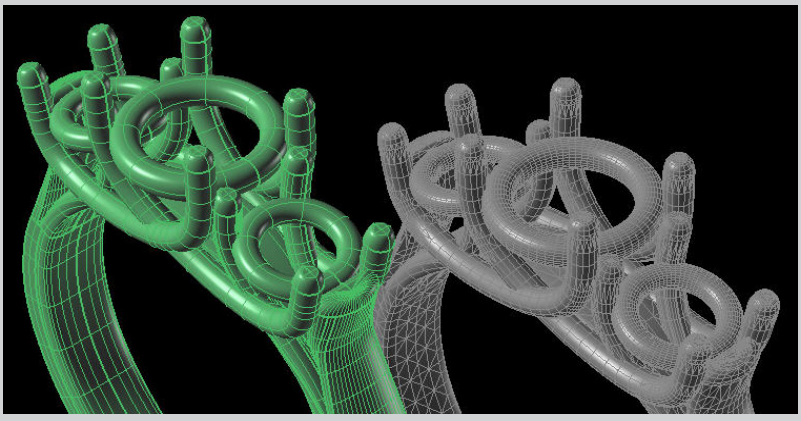


About "STL Options"

What is an STL? An STL is a mesh version of your model, which usually starts as a Nurbs model in Matrix.

How is it made? A mesh model is made by re-creating the Nurbs model with lots of tiny surfaces shaped like polygons (three or more sides). The nature of these polygons and how faithfully they re-create the Nurbs model is defined in the STL Options dialog.

Changing STL Options: Because re-creating rounded surfaces with a lot of tiny flat surfaces can sometimes make the resulting mesh look choppy or faceted, you may wish to change the STL Options. We recommend cutting one or both values - "Minimum Edge Length" and/or "Maximum Distance Edge to Surface" - in half. These values define the size of the polygons and the distance that the mesh surface may vary from the original Nurbs surface, respectively.



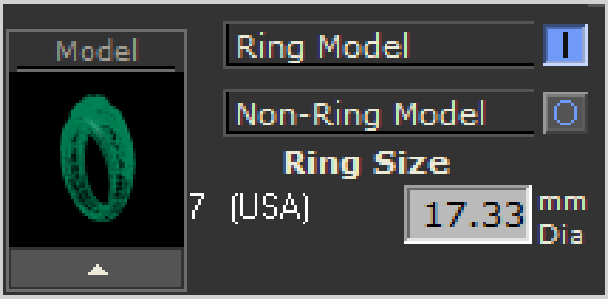
Nurbs (L) & STL (R) versions of the same ring. Tiny facets are not picked up by the cutter; larger ones can be, so change STL options if wax or STL shows large facets.

Tutorial 1 : Create a Toolpath for the 3-Jaw Chuck Fixture

Choose the 3-Jaw Chuck strategy for a ring model that requires just a rotary pass around the outside of the model to complete its details.

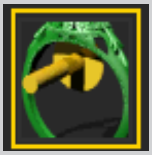
When using the 3-Jaw Chuck the ring rotates on the A axis while the vertical cutter mills the outer face of the ring. The 3-Jaw Chuck holds the wax tube for this strategy. In this tutorial, we will choose the "Hollow" Center option. This is only possible when using hollow tube wax. This strategy is ideal for this model due to the fact all of the detail is on the outer face of the ring in a location where the mill will reach with the vertical cutter. Although other strategies could work, this is the quickest way to mill this model making it the most efficient use of the mill's time. Make CERTAIN you have a Ring Rail prior to entering the builder. This cut requires no supports: the mill simply leaves 1.5 mm of wax between the edge of the model and the wax tube to affix it to the tube.

1 Prepare For Milling Verify the model includes a Ring Rail and the Ring Rail is the correct size. On a rotary toolpath, the ring rail determines the depth at which the cutter plunges down into the wax.



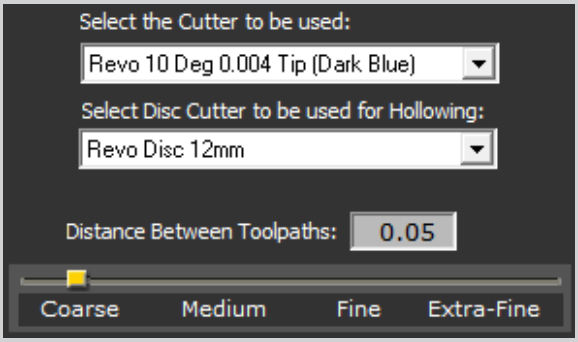
2 Mill Builder Enter the Mill C builder, select the model, and input it into the "Model" preview window. "Ring Model" should be selected automatically, with the correct Ring Size displayed. If it is not, select this option and manually input the Ring Size in mm. Or, return to Matrix and place a ring rail.

3 Support and Center Strategy In this case, we will choose the 3-Jaw Chuck fixture, because this is the most efficient use of mill time for a strategy that will mill all the details around the outside of this ring and allow the Hollowing cutter to remove the excess wax located on the inside of the ring. Click "Next".

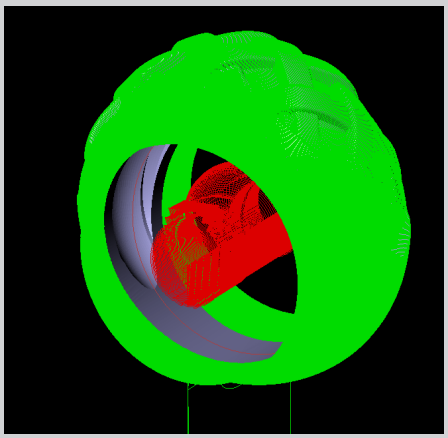


4 Choose Wax Size Choose the wax size you wish to use to cut the part. It is important to select the actual size you will use to cut the model. The mill will determine how deep the cutter can safely plunge into the wax based on the wax size you selected. Click "Next".

5 Cutter & DBT The 10-dgr tapered cutter and the Revo Disc 12mm cutter are ideal for this model. The Distance Between Toolpath or DBT determines the amount of step over as the toolpath moves back and forth across the model.



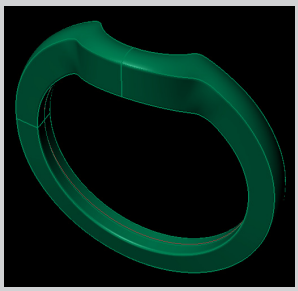
6 Make Paths Click "Make Path" to see the results. The toolpath will display on screen. Inspect the toolpath and verify all the detail on your model will be reached. The red toolpath is the hollowing toolpath and will be the first cut to take place while the wax is most rigid. The green toolpath is the rotary pass that will mill the outer face of the ring.



7 Save Path After inspecting the toolpath, choose "Save Path". Save this to the "RVO" folder found in the "My Documents > Matrix" folder, or to any location on your computer where later you can transfer the file to a thumb drive to load onto the Revo Mill's computer.

Tutorial 2: Create a Toolpath for the Dual Fixture

Choose this strategy for a ring model that requires just two cuts – front and back (2-sided) – on the Dual Fixture.

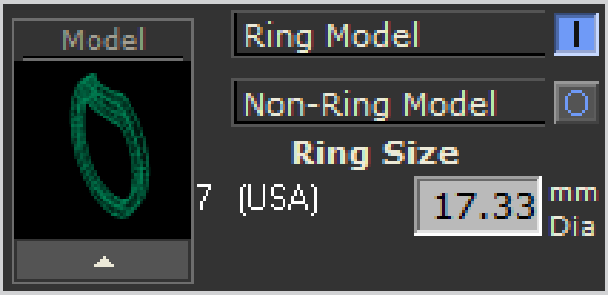


The Two-Sided Cut on the Dual fixture will be the most efficient strategy for this model. All of the details on this ring can be reached via the Front and Back cuts (as seen in the Through Finger view). The model will be supported by 4 small boxes that will be generated automatically and will be offset from the Ring Rail by 1.5mm.

This is why it is important to ALWAYS include a Ring Rail when milling a ring model. Take advantage of the automation while milling rings. In later demonstrations, when milling a non-ring item, you will be required to manually add supports to the model.

1 Prepare For Milling In preparation for milling, if a ring model does not have a Ring Rail, include one. The Revo C builder can automatically read the ring rail and understand where to put the automatically generated supports for any model.

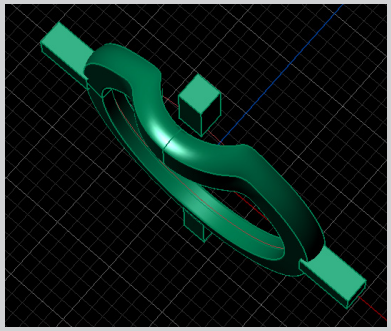
2 Mill Builder Enter the Mill C builder, select the model, and input it into the "Model" preview window. "Ring Model" should be selected automatically, with the correct Ring Size displayed. If it is not, select this option and manually input the Ring Size in mm. Or, return to Matrix and place a ring rail

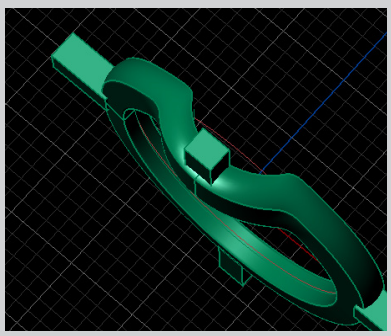


3 Supports In this case, we will choose the "Left, Right, Top and Bottom" supports, or the Dual Fixture, because all details in this model can be reached by the "Front" and "Back" cuts completed during this strategy (as seen in the Through Finger viewport). Other strategies, such as "Three-Sides on Dual" or "Base Clamp", complete additional cuts which – though not bad – take up additional time.

4 Center Strategy Next in this screen, choose the "Cut Out Center" strategy. This option traces a circle that is offset 2 mm to the inside of the finger rail. This saves time because, rather than having to mill out all that wax, it is eliminated quickly by the cut-out curve. The user removes it from the wax model on the mill, and the mill completes the cut: down one side of the ring and up the other, skipping all that time it would take to cut out the center hole of the ring. Click "Next".

5 Support Verification Make any required adjustments to the automated supports AFTER choosing the Support and Center Strategies. The Supports are offset to the outside of the Ring Rail by 1.5mm. Verify the supports intersect with the model. If an adjustment is necessary you may delete unnecessary supports or drag and drop a support to a new location as long as you keep the supports on the same side of the model at which they first appear. In this case, the top support will either need to be relocated to intersect with the model, or deleted. Verify the supports intersect with the model before clicking "Next".





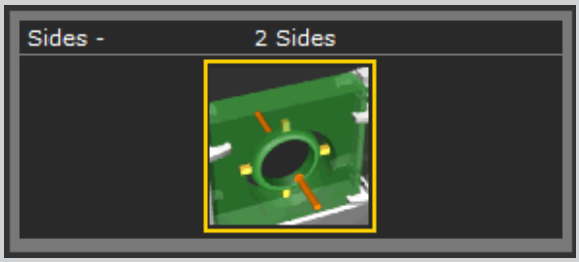
Above: The automated support does not intersect with the model and requires a slight adjustment.

Below: The model after the support has been adjusted.

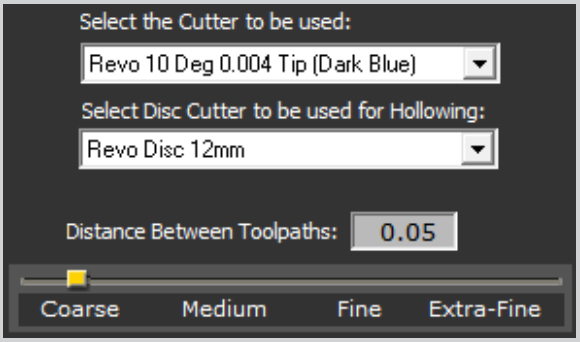
6 Choose Wax In the second screen, the smallest wax possible to complete the model will be suggested, however it is important to select the actual size wax you will be using. The software will automate any roughing passes required to avoid burying the cutter based on the wax size you select. Click "Next".

Toolpath for the Dual Fixture cont'd

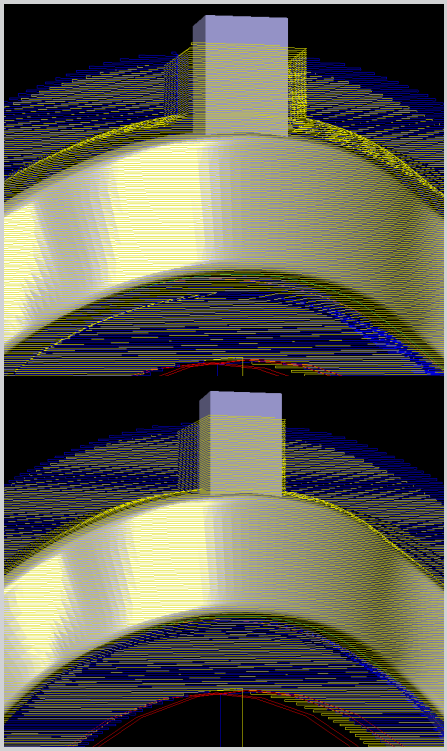
7 Approach Angle The only approach angle for the Dual 2 Sides is from the Front and Back. Click "Next".



8 Cutter and DBT Either the 10 Degree cutter or the 32 Thousandths cutter will suffice. Both toolpaths will take approximately the same amount of time if the DBT (step over) setting remains the same. However, the toolpaths will appear slightly different. The toolpath is calculated from the center of the cutter so if using the 32 Thousandths cutter the toolpath will appear to be offset from the model approximately .4mm or half the diameter of the cutter as it moves down the side of the model as seen in the Through Finger viewport after Clicking "Make Path".

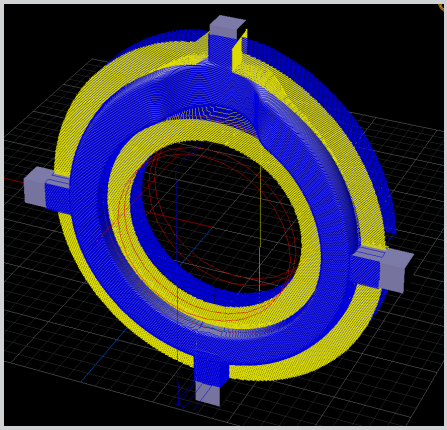


9 Make Path Click "Make Path" to see the results. The toolpath will display on screen. Inspect the toolpath and verify all the detail on your model will be reached. You should see the toolpath drape over your entire model. If the toolpath needs to be adjusted, refer to Chapter 3 - Detailed Options. In this example, the red toolpath represents the Cut Out Center. The Blue and Yellow toolpaths represent the cuts from the Front and Back sides. Take note, the toolpath drapes the model and the supports, however the supports extend out past the toolpath ensuring the model will be attached to the wax block after the excess was is milled away to reveal your model.



Above: Compare both toolpaths. The top toolpath is calculated using the 32 Thousandths Cutter. The toolpath appears offset from the model because the toolpath is calculated from the center of the Cutter. The toolpath calculated with the 10 degree cutter appears to drape closer to the model due to the fact the tip diameter is significantly smaller.

10 Save Path After inspecting the toolpath, choose "Save Path". Save this to the "RVO" folder found in the "My Documents > Matrix" folder, or to any location on your computer where later you can transfer the file to a thumb drive to load onto the Revo Mill's computer.



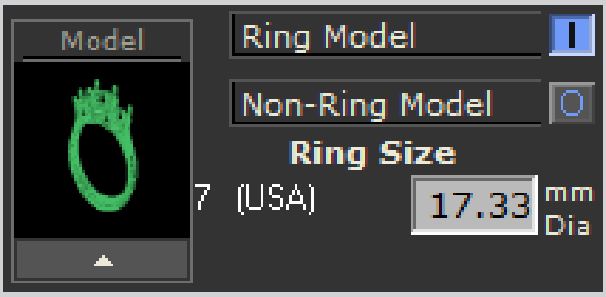
Tutorial 3 : Create a Toolpath for the Dual 3-Sided Fixture

Choose this strategy for a ring model that requires just three cuts – top, front, and back (3-sided)– on the Dual Fixture.

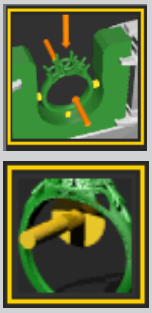
The Three-Sided Cut on the Dual fixture will be the most efficient strategy for this model. All of the details on this ring can be reached via the Top, Front and Back cuts (as seen in the Looking Down & Through Finger view). The model will be supported by 3 small boxes that will be generated automatically and will be offset from the Ring Rail by 1.5mm. This is why it is important to ALWAYS include a Ring Rail when milling a ring model. Take advantage of the automation while milling rings. In later demonstrations, when milling a non-ring item, you will be required to manually add supports to the model.

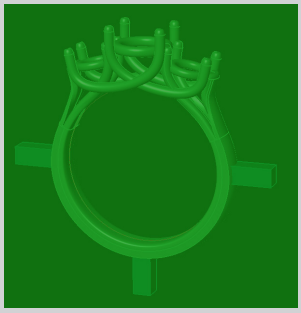
1 Prepare For Milling In preparation for milling, if a ring model does not have a Ring Rail, include one. The Revo C builder can automatically read the ring rail and understand where to put the automatically generated supports for any model.

2 Mill Builder Enter the Mill C builder, select the model, and input it into the "Model" preview window. "Ring Model" should be selected automatically, with the correct Ring Size displayed. If it is not, select this option and manually input the Ring Size in mm. Or, return to Matrix and place a ring rail



3 Supports Choose the support strategy ("Left, Right & Bottom") that corresponds to the Dual 3-Sided fixture. At that time, three supports will be added to the model by the builder. Dual 3-Sided is ideal because all of the details in this model can be reached by the "Top", Front", and "Back" cuts completed during this strategy (as seen in the Looking Down and Through Finger viewport).



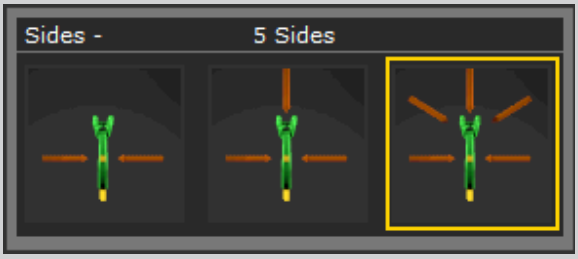


4 Center Strategy Next in this screen, choose the "Hollow" strategy. This option first traces a circle that is offset 2 mm to the inside of the finger rail. The mill will pause so the user can remove the plug of wax to allow the hollowing cutter to enter the wax and hollow the model.

5 Support Verification Make any required adjustments to the automated supports AFTER choosing the Support and Center Strategies. The Supports are offset to the outside of the Ring Rail by 1.5mm. Verify the supports intersect with the model. If an adjustment is necessary you may delete unnecessary supports or drag and drop a support to a new location as long as you keep the supports on the same side of the model at which they first appear. In this case, the automatically generated supports work well and no adjustments are necessary. Click "Next".

6 Choose Wax In the second screen, the smallest wax possible to complete the model will be suggested, however it is important to select the actual size wax you will be using. The software will automate any roughing passes required to avoid burying the cutter based on the wax size you select. Click "Next".

7 Approach Angle Analyze where the details are on the model and choose the appropriate approach. Clearly, the 2 sided option would not work well for this example. Choose either the 3-sided, or 5-sided approach. Take note, if you choose the 5 sided option, the mill will complete the traditional 3 sides (top, front, and back) and when approaching the model for the 4 & 5 sides, will analyze the geometry and mill ONLY the areas that could not be reached with the traditional 3 angles. This will save time, but also avoid over milling areas of the model. Best practices for milling are to avoid milling the same area more than once.



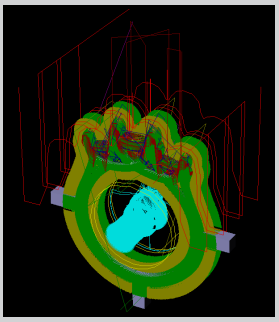
Create a Toolpath for the Dual Three-Sided Fixture cont'd

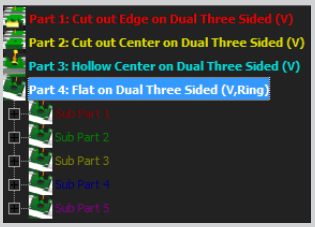
8 Cutter and DBT In this example, the 10 Degree cutter and the Revo Disc 12mm should be selected. The 10 degree will allow the cutter to mill the fine detail while the hollowing cutter will approach the model from the inside and remove any excess wax. With today's gold price, every dwt can add up.

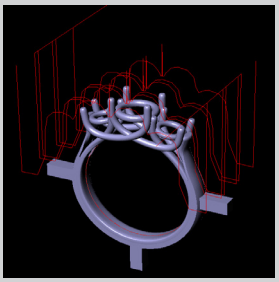
10 Detailed Options Each Part of the toolpath has been separated into a list format. A Part can consist of one approach angle, or multiple approach angles. Click on Part 1 to view only that portion of the toolpath. The list has been color coded to match the toolpaths on screen. In this example, Part 1 is red and is the "Cut Out Edge on Dual Three Sided (V)". During this Part, the cutter will follow the red curve to remove a large block of excess wax quickly and allow for the next cuts. The (V) means this is cut with the vertical cutter. Next, click Part 2. The light yellow curve is the "Cut Out Center" removing the plug in the center of the wax to allow the Hollowing cutter to enter the wax. Part 3 is the teal blue hollowing pass that will clear out any excess wax. Keep in mind the toolpath is calculated from the center of the 12mm cutter. Part 4 is white. You may notice, there is no white toolpath on screen. Anytime you see white in the toolpath list, it means this portion of the toolapth is comprised of more than one "Sub Part". Double Click Part 4 to expand the list into its Sub Parts. Five Sub Parts will display. The First Three are the traditional Top, Front, and Back cuts. Sub Part 4 & 5, are the 45 degree angle cuts that only mill the areas that could not be reached with the traditional top, front and back cuts.

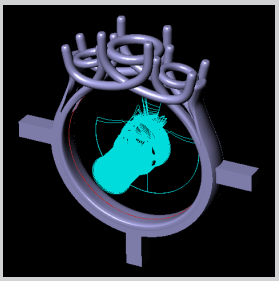
11 Save Path After inspecting the toolpath, choose "Save Path". Save this to the "RVO" folder found in the "My Documents > Matrix" folder, or to any location on your computer where later you can transfer the file to a thumb drive to load onto the Revo Mill's computer.

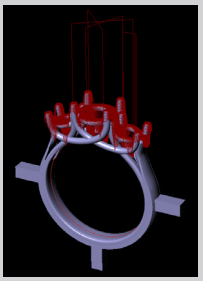
9 Make Path Click "Make Path" to see the results. The toolpath will display on screen. Inspect the toolpath and verify all the detail on your model will be reached. You should see the toolpath drape over your entire model. As the toolpaths become more complicated, it is easier to view the toolpath individually as opposed to trying to make sense out of overlapping toolpaths. To view these paths individually, Click "Next"

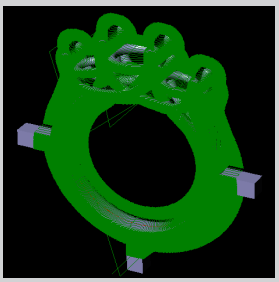


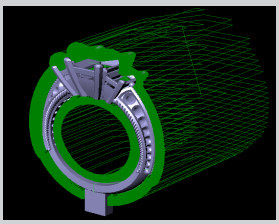


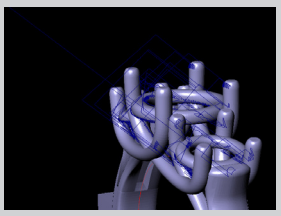


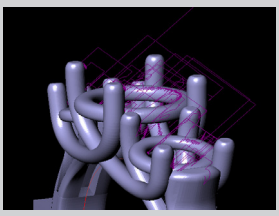






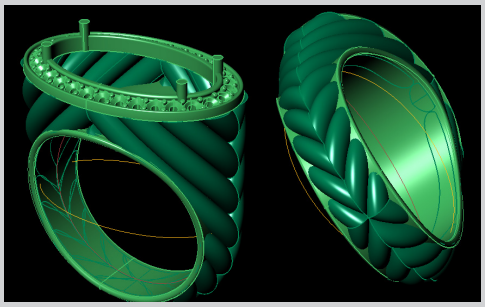






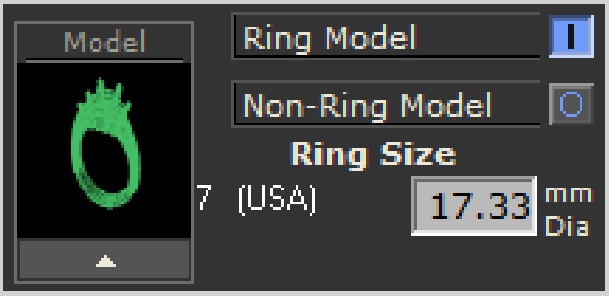
Tutorial 4 : Create a Toolpath for the Hub Fixture

Add a Hub to any Dual Fixture model or Base Clamp Model to add a Rotary Toolpath.

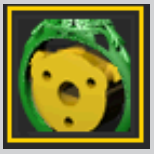


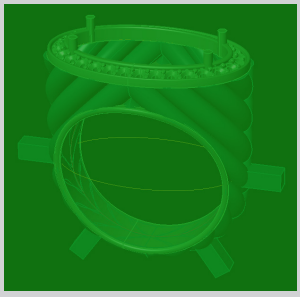
In the first screen of the Mill C Builder, the "hub center" can be added to any two- or three-sided cut on the Dual Fixture or any Base Clamp model, in order to add a rotary cut. This allows the mill to reach all the details around the outside of the model after milling the traditional flat cuts. The toolpath builder will add an extra screen wherein the user specifies the size for the "hub center": a piece of wax that secures the model to the hub fixture for the rotary cut. This also adds a base piece of wax that is cut last on any of the flat fixtures and first on the Hub fixture so the user can "Center" the model correctly on the "Hub" fixture.

1 Prepare For Milling In preparation for milling, if a ring model does not have a Ring Rail, include one. The Revo C builder can automatically read the ring rail and understand where to put the automatically generated supports for any model. The Ring Rail is also used to determine how deep the mill will plunge during the rotary cut. This model should be centered in the viewports because the wax is centered on the fixture, and a model that is not centered will recommend a larger wax than it actually needs.



2 Mill Builder Enter the Mill C builder, select the model, and input it into the "Model" preview window. "Ring Model" should be selected automatically, with the correct Ring Size displayed. If it is not, select this option and manually input the Ring Size in mm. Or, return to Matrix and place a ring rail



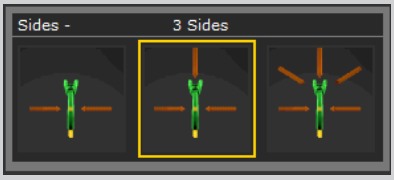


3 Supports Choose the support strategy ("Left, Right & Bottom") that corresponds to the Dual 3-Sided fixture. At that time, three supports will be added to the model by the builder.

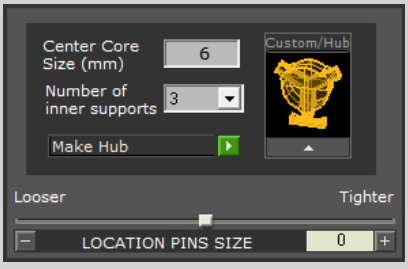
4 Center Strategy Next in this screen, choose the "Add Hub" strategy. The combination of the Dual 3-Sided and the Hub will reach all of the detail on this model. Take note, when the "Add Hub" Center option was selected, the bottom support was shifted to an angle and an additional support was added, making room for the Hub. Click "Next".

5 Choose Wax In the second screen, the smallest wax possible to complete the model will be suggested, however it is important to select the actual size wax you will be using. The software will automate any roughing passes required to avoid burying the cutter based on the wax size you select. Click "Next".

6 Approach Angle Analyze where the details are on the model and choose the appropriate approach. For this model, choose the 3-sided approach. Click "Next".

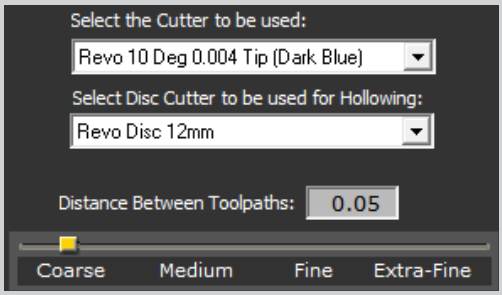


7 Center Support Creator The next step is to create a "Hub" that will appear in the center of the Ring Rail and be used to attach the model to the Hub fixture. Create a Hub that is smaller than the thickness of wax selected in Step 5. Choose the number of inner supports from the drop down list. The default number 3 is sufficient for most models. If you wish to create your own supports, choose None. Click "Make Hub" and the automated Hub will appear. The Hub is comprised of three parts, the inner supports, the hub and the small bar at the bottom used to center the model. Place all three items into the "Custom/Hub" window. Click "Next".

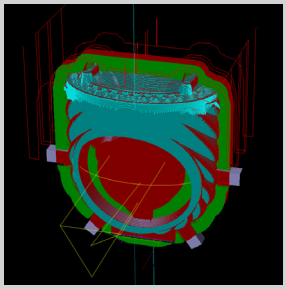


Create a Toolpath for the Hub Fixture cont'd

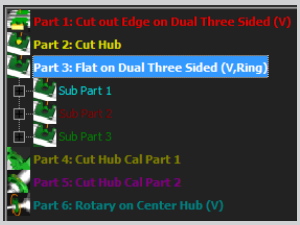
8 Cutter and DBT In this example, the 10 Degree cutter will suffice.

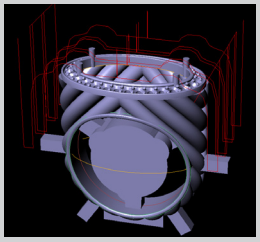


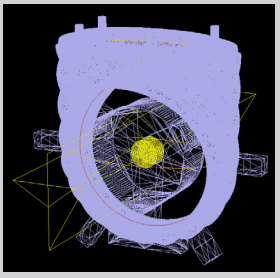
9 Make Path Click "Make Path" to see the results. The toolpath will display on screen. Inspect the toolpath and verify all the detail on your model will be reached. You should see the toolpath drape over your entire model. As the toolpaths become more complicated, it is easier to view the toolpath individually as opposed to trying to make sense out of overlapping toolpaths. To view these paths individually, Click "Next"

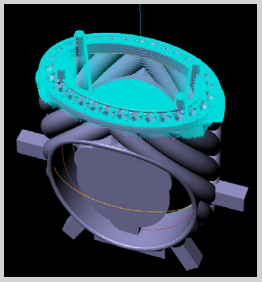


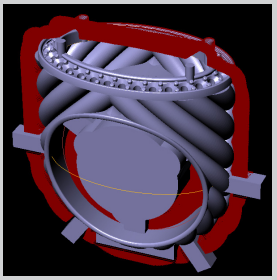
10 Detailed Options Each Part of the toolpath has been separated into a list format. A Part can consist of one approach angle, or multiple approach angles. Click on Part 1 to view only that portion of the toolpath. The list has been color coded to match the toolpaths on screen. In this example, Part 1 is red and is the "Cut Out Edge on Dual Three Sided (V)". During this Part, the cutter will follow the red curve to remove a large block of excess wax quickly and allow for the next cuts. The (V) means this is cut with the vertical cutter. Next, click Part 2. The light yellow curve is the "Cut Hub". During this approach, the three holes where the pins will mate are milled into the hub, the center hole, where the thumb screw will secure the model, and the arrow used for orienting the model will be inscribed onto the wax. Part 3 is white. You may notice, there is no white toolpath on screen. Anytime you see white in the toolpath list, it means this portion of the toolapth is comprised of more than one "Sub Part". Double Click Part 3 to expand this Part into its Sub Parts. Three Sub Parts will display. The three sub parts are the traditional Top, Front, and Back cuts. Part 4 & 5, are small toolpaths on the bottom of the ring shank. These cuts mill small lines used later to center the model onto the Hub fixture. Last but not least, Part 6 will finish the model. Take note, the support are milled off during this rotary cut.

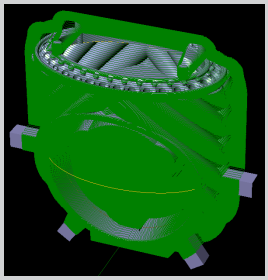


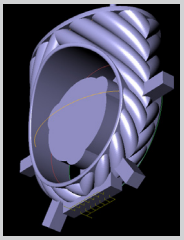


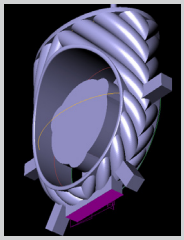


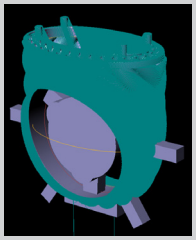












11 Save Path After inspecting the toolpath, choose "Save Path". Save this to the "RVO" folder found in the "My Documents > Matrix" folder, or to any location on your computer where later you can transfer the file to a thumb drive to load onto the Revo Mill's computer.

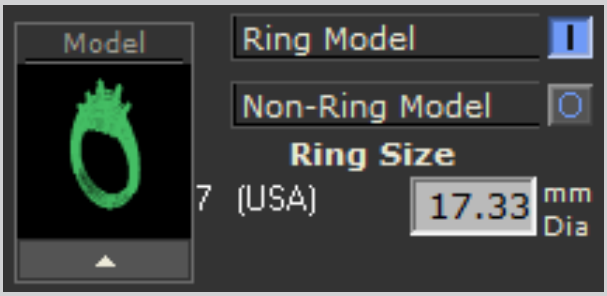
Tutorial 5 : Create a Toolpath for the Base Clamp Fixture

Create a toolpath for the Base Clamp fixture. The Base Clamp can mill detail from 9 sides.

The ring at left has details all around the outside, top of the shank, down the sides, and even at unique angles on the front and back of the model. The only place it lacks detail is at the very bottom, outside of the shank, where the sprue would usually be placed for casting. It is therefore the perfect model for use with the Base Clamp fixture. The Base Clamp can approach from 9 different angles ideal for cutting ring designs and this is why some people have been known to say, "Base Clamp, - for the WIN"!

1 Prepare For Milling In preparation for milling, if a ring model does not have a Ring Rail, include one. The Revo C builder can automatically read the ring rail and understand where to put the automatically generated supports for any model.

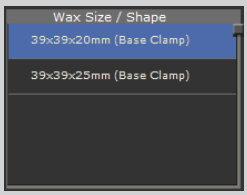
2 Mill Builder Enter the Mill C builder, select the model, and input it into the "Model" preview window. "Ring Model" should be selected automatically, with the correct Ring Size displayed. If it is not, select this option and manually input the Ring Size in mm. Or, return to Matrix and place a ring rail

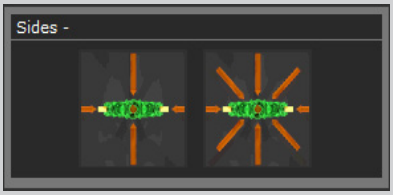


3 Supports . In this case, we will choose the "Bottom" support, which is the support strategy corresponding to the Base Clamp fixture. This strategy can reach the top-down (Looking Down view of the model), front and back (Through Finger view of the model) AND left and right (Side View of the model) sides of the ring. The Base Clamp has the additional benefit of adding four custom angles 45-degrees to the front, back, left & right.

4 Center Strategy Next in this screen, choose the "Cut Out Center" strategy. This option traces a circle that is offset 2 mm to the inside of the finger rail. This saves time, rather than having to mill out all that wax, it is eliminated quickly by the cut-out curve. The user removes it from the wax model on the mill, and the mill completes the cut: down one side of the ring and up the other, skipping all that time it would take to cut out the center hole of the ring. Click "Next".

5 Choose Wax In the second screen, the smallest wax possible to complete the model will be suggested, however it is important to select the actual size wax you will be using. The software will automate any roughing passes required to avoid burying the cutter based on the wax size you select. Click "Next".

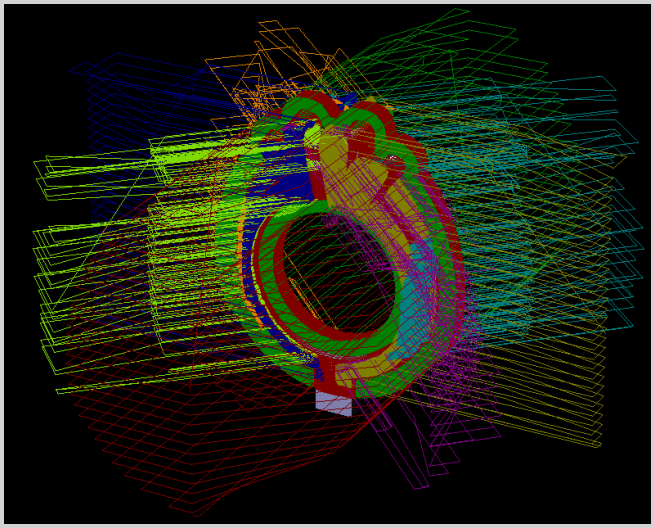




6 Approach Angle Analyze where the details are on the model and choose the appropriate approach. For this model becase of the angle of the detail on the knife edge of the ring's shank, choose 8 sided. Click "Next".

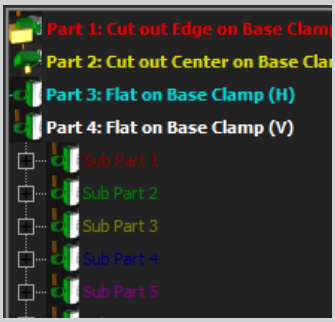
7 Cutter and DBT In this example, the 10 Degree cutter will suffice.

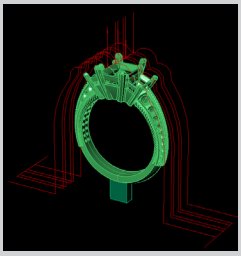
8 Make Path Click "Make Path" to see the results. The toolpath will display on screen. Inspect the toolpath and verify all the detail on your model will be reached. You should see the toolpath drape over your entire model. As the toolpaths become more complicated, it is easier to view each part of the toolpath individually as opposed to trying to make sense out of overlapping toolpaths. To view these paths individually, click "Next"

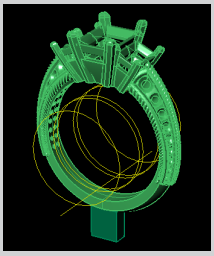


Create a Toolpath for the Base Clamp Fixture cont'd

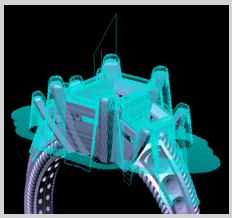
10 Detailed Options Each Part of the toolpath has been separated into a list format. A Part can consist of one approach angle, or multiple approach angles. Click on Part 1 to view only that portion of the toolpath. The list has been color coded to match the toolpaths on screen. In this example, Part 1 is red and is the "Cut Out Edge on Base Clamp". During this Part, the cutter will follow the red curve to remove a large block of excess wax quickly to allow for the next cuts. The (V) means this is cut with the vertical cutter. Next, click Part 2. The light yellow curve is the "Cut Out Center on Base Clamp" where the cutter traces a circle that is offset 2 mm to the inside of the finger rail and the mill pauses for the user to remove the wax plug from the center of the Ring Rail. Part 3 is teal in color and cuts the model from the top down as seen in the Looking Down viewport. This cutter approach uses the (H) Horizontal spindle. Part 4 is white. You may notice, there is no white toolpath on screen. Anytime you see white in the toolpath list, it means this portion of the toolapth is comprised of more than one "Sub Part". Double Click Part 4 to expand this Part into its Sub Parts. Eight Sub Parts will display. The eight sub parts approach the mill front and back (Through Finger view of the model) AND left and right (Side View of the model) sides of the ring. Four additional angles 45-degrees to the front, back, left & right are added.

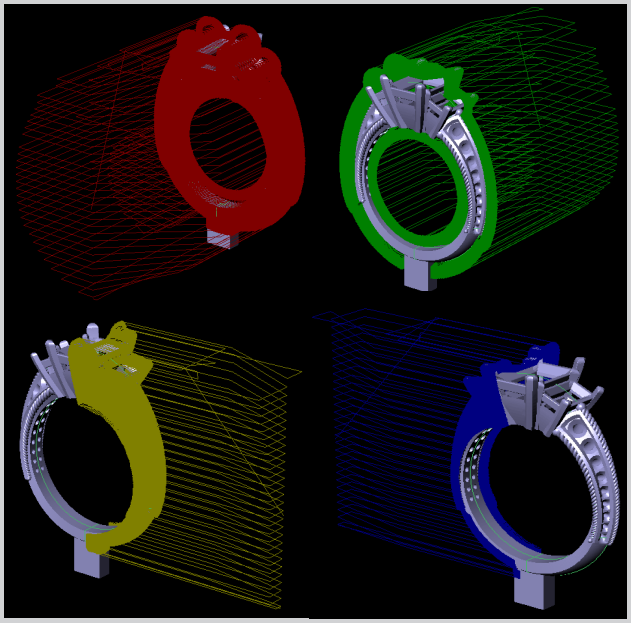






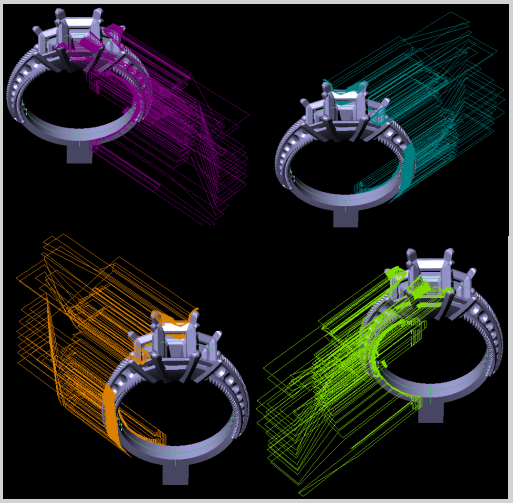
Note: During the Top Down cut it appear as if one toolpath shadows the other. This is what is known as a roughing pass, where the software analyzed how deep the toolpath was required to plunge into the wax and based on the cutter definition added an additional pass so the cutter would not be buried too deep jepordizing the safety of the cutter. The Revo 10 Deg .004 Tip (Dark Blue) will plunge 10mm deep into wax before a roughing pass will be added.





Pictured Above: the Vertical cutter approaches Front, Back, Left and Right. Pictured Right, the Vertical cutter approaches at the 45 degree angles, only milling the detail that could not be reached with one of the other approaches.

Note: When milling the 45 degree angles on the Base Clamp, the mill first analyzes all of the prior toolpath approaches. Then calculates the new paths limiting the cuts to details that could not be reached with one of the earlier approaches. This smart approach to milling will save time.



Using the Non-Ring Strategies ...

About Non-Ring Strategies

The non-ring strategies require YOU to place the supports where they belong. Read about each strategy, its sample models, and how to build supports in this section.

Overview of the Non-Ring Strategies

When selecting a "Non-Ring" model in the first screen of the Mill Builder, the builder will REQUIRE you to place supports into the Supports In-Box before it can proceed. You'll need to create these supports in Matrix FIRST. Their placement will be based on which of the non-ring strategies you wish to use, which in turn is based on... yep, you guessed it! ... Where the details are located on your model.

Similar to Ring Strategies: 2, 4 or 8 Sides

For most of the flat strategies (2-sided on Dual Fixture and 4- or 8-sided on Base Clamp), the support placement is very similar to their placement for Ring Strategies: for the 2-sided strategy, supports are placed on the top, bottom, left and right of the model as seen in the Looking Down Viewport; for Base Clamp, one support is placed on the bottom of the model as seen in the Through Finger viewport. For 3-sided and Rotary strategies, the support configurations differ from their Ring counterparts.

Three-Sided on Dual Fixture

For the three-sided strategy, two supports on the bottom of the model can abe sufficient. Make certain this is the option you want, as two sides of the model will as seen in the side view, will NOT be touched by the cutter.

Flat on Rotary Plus One

Other Non-Ring options CAN mill these sides, including Flat on Rotary Plus One, in which a wax tube holds onto the object at its base - usually with just two supports - and holds it flat on four or eight sides during milling while also allowing the horizontal cutter to perform a top-down cut (the "Plus One" part of this strategy.)

Rotary Plus One

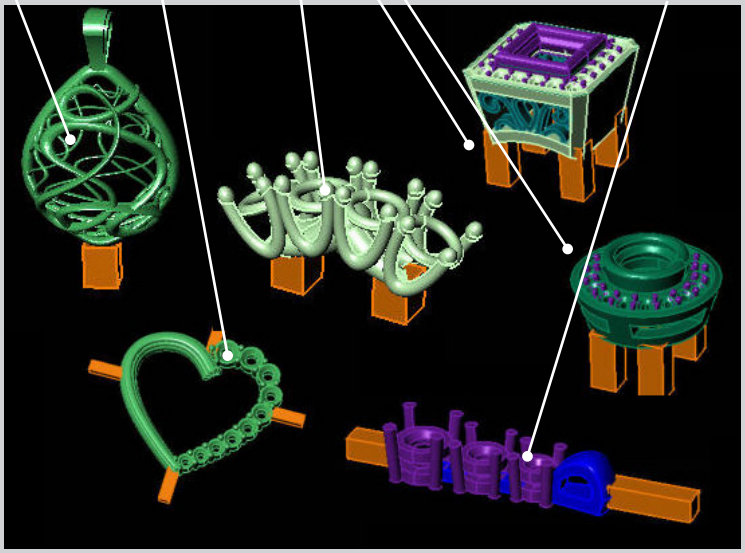
The same support strategy is used for "Rotary Plus One", which is identical to the previous strategy, except that it is selected for round parts: the mill rotates the part while cutting: it doesn't hold it flat on each side. This strategy also completes the top-down cut with the horizontal cutter.

Supporting the Non-Ring Strategies

Supports for the 2-sided Dual Fixture and 4- or 8-sided Base Clamp Fixture are the same as those for Rings: Top, Bottom, Left and Right (as seen in Looking Down) for the 2-sided fixture; and Bottom only for the 4- or 8-sided Base Clamp strategy.

For the 3-sided strategy, just two supports on the bottom of the model are needed.

For the Rotary fixture, two strategies require supporting the model via supports attached at the bottom, this strategy isno capable of milling the top of the model AND all around the outside of the model (while being held flat on the rotary - "Flat on Rotary + 1" or while being turned - "Rotary + 1"). Finally, the model may be cut on four flat angles on the 3-Jaw Chuck requiring just two supports on either end (Four Flats on Rotary).



Models above are, from clockwise starting in lower left: Two Sided on Dual; Base Clamp; Three Sided on Dual; Flat on Rotary Plus One; Rotary Plus One; Flat on Rotary.

Four Flats on Rotary

The final Non-Ring strategy of which the Rotary fixture is capable is known as "Four Flats on Rotary". For this strategy, the supports are placed on the left & right sides of the model, as seen in the Looking Down Viewport, and the model is held, by these two supports, to the wax tube while it is rotated 90 degrees and cut on the flat four times. In this way, the top, bottom, and two sides of the model can be milled; but NOT the two ends that are secured to the wax with supports. During the cut, the wax tube is milled away. The remaining tube of wax becomes additional support material around the model. This tube is secured at one end to the 3-Jaw Chuck which is attached to the A-axis.

Using the Non-Ring Strategies ...

How to Build Supports

Whether building your own supports for Ring models or placing supports where they are needed for Non-Ring models, here are the steps for creating supports.

Building Supports

The "Box" tool is most commonly used to build supports for ring and non-ring models. It is found in the Solid menu, and is very easy to use: start it up and draw a rectangle, corner to corner, at about the location for one support. (If this is hard to do where it is required, you can always draw it somewhere else and move it after.) This represents the length and width of the box. The next mouse-click will occur in a different viewport and sets the height of the box.

For most strategies, remember that supports should be no less than 2 mm X 2 mm thick, and at LEAST 3 mm long, if not more, so they can be sunk into the model to support it sufficiently AND their ends must not fall inside the automatic border placed around the model by the Mill Builder (which is drawn roughly 2mm away from the extents of the model). In the case of the Base Clamp, a single support should be drawn at the base of the model at 4 mm X 4 mm thick and at LEAST 3 mm long.

Locating Supports, Ring Models

Once you are familiar with the locations of the builder-created supports in each of the ring strategies, you may find that you'll come across a model where the details or design dictate the need for a new location for the supports. In that case, locate them as near as possible to the location expected by the builder - at the very least, on the same "side" of the model so the builder can recognize the strategy you're choosing, and as close as possible to the same size(s) described above. Note: Supports should ALWAYS be centered on the model on the side they're located so as not to cause flex: whether the model is a ring or not.

Locating Supports, Non-Ring Models

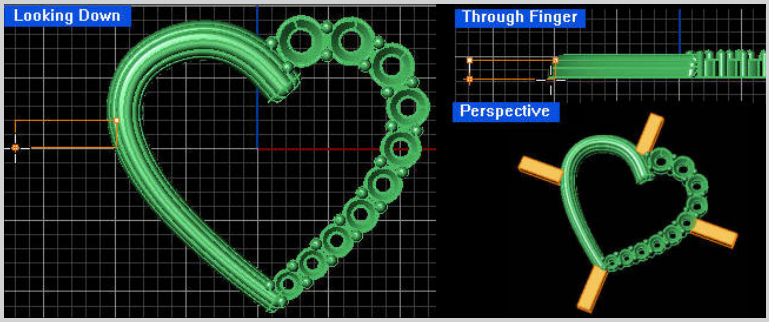
The location for each support will be based on where the details are NOT located on the model and should be correctly positioned for the chosen strategy, as shown in the following pages. The size of these supports should be comparable to those for Ring models, as described above.

Building & Locating Supports (Ring or Non-Ring)

1 - Use the Box Tool. The first two steps when using the Box tool: place two corners of the box, defining the length and width of one side. The final step defines the height or thickness of the box.

2 - The Proper Size. For most strategies, the box should be 2 X 2 mm thick and no less than 3 mm long. For the Base Clamp strategy, double the thickness to 4 X 4 mm. If necessary, scale the box so it is properly sized.

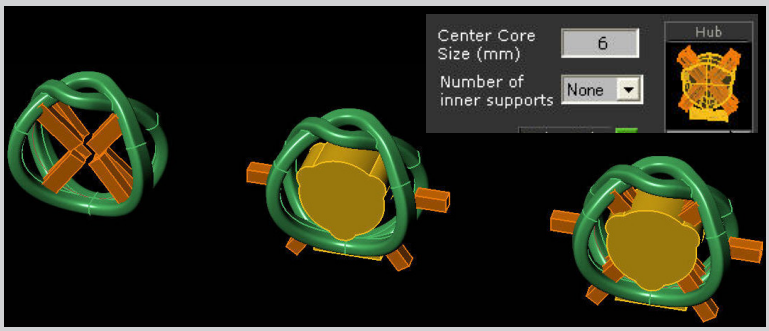
3 - Position Supports. It is usually easier to draw one support and then re-locate it to the proper position and copy, rotate & copy, or mirror it to the remaining required positions. Note: Supports must touch model!



(L) "Box" tool draws a rectangle first. Make this 2 mm X 3 mm or more. (Upper R) Make the box 2 mm tall, as well. (Lower R) Mirror, Rotate/Copy to place.

In case of exceptions: (Left-most illustration, below) For a design that has a split shank at the location where a support would usually be, create two supports - they may be slightly less than 2 mm X 2 mm thick - and position one on each side. If the ring is less than 2 mm thick, you may wish to place additional supports - still remaining faithful to the "side" of the ring (top, bottom, left or right, e.g.) on which they belong, to handle their smaller scale.

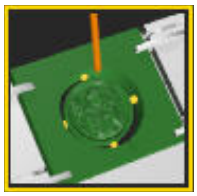
Finally, in the case of supporting the hub for ring models, if there is a split shank or other design elements interfering, you can also draw your own hub supports and input these into the Mill Builder, as shown below.



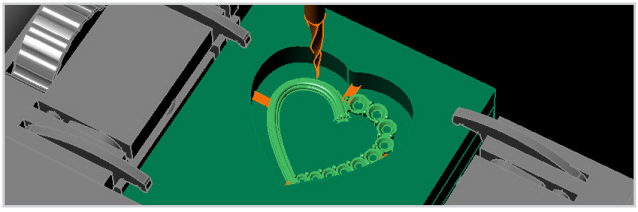
(L) User-created supports; (Center) Builder-created supports; (R) Input user-created internal supports together with Hub and Base piece into "Hub" in-box.

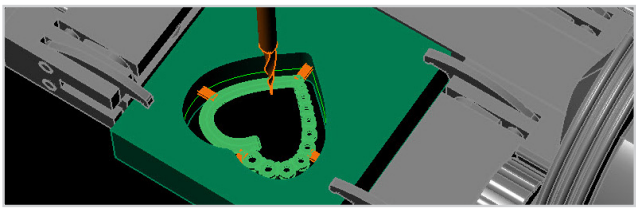
About the Non-Ring Strategies An Overview

Each fixture and its associated non-ring strategies (flat first; then rotary) are summarized in this section.



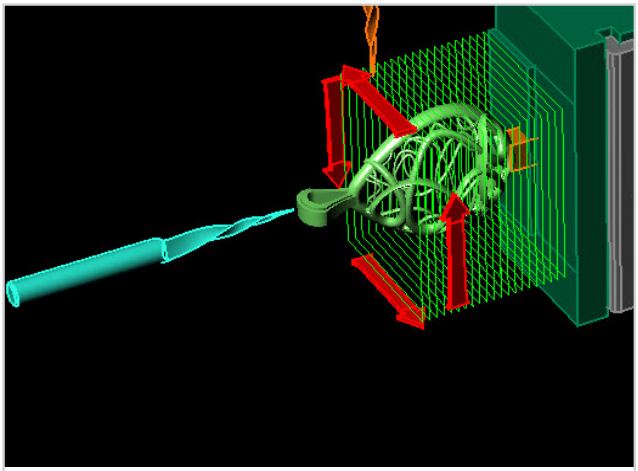
Two-Sided on Dual Fixture Just like its Ring Strategy counterpart, a two-sided part will be milled on the "top" and "bottom" of the model, as seen in the Looking down viewport.

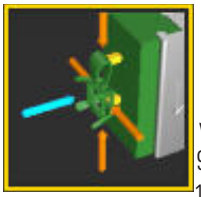




When to Choose this Strategy This strategy is best for "flat work", or models with details on the top and bottom and no details on the sides, where the mill cannot reach to create a "grooved" profile shape, for example.

Placing Supports Supports should be placed on the top, bottom, left and right of the part for best results. This ensures the model wil not flex while it is being milled.



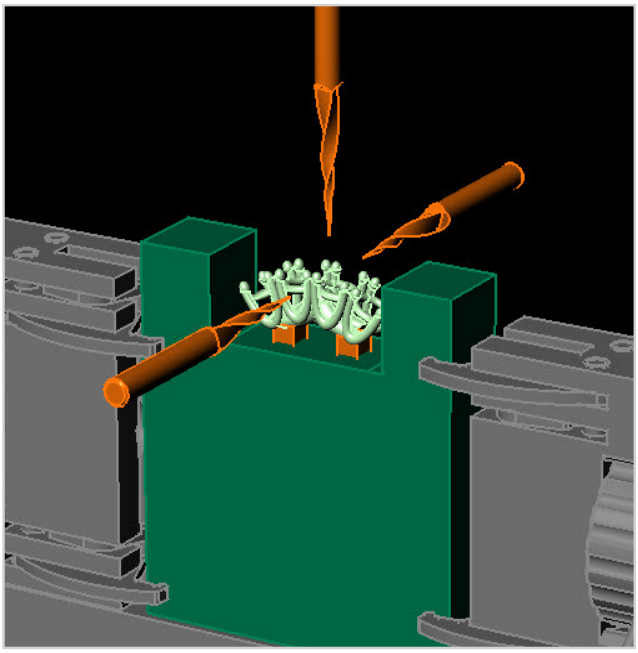


4- or 8-Sided Base Clamp Just like its Ring Strategy counterpart, a four-sided Base Clamp part will first be milled from the "top down" with the horizontal spindle, and then rotated in 90- or 45-degree increments on the A Axis, cutting 1-mm at a time, until it completes either 4 or 8 flat sides, respectively.

When to Choose this Strategy Choose this strategy when there are no details to mill on the very bottom, outside of the part, where the support is placed. Because this strategy is most time consuming, use Base Clamp when no other strategies can reach all the details on your model.

Placing Supports Locate a 4 X 4 mm thick support, no less than 3 mm long, at the very bottom of the model, centered on this side.

3-Sided on Dual Fixture This strategy can approach a model such as a head or a pendant from the top, front, and back of the model. It cannot complete details on the bottom of the model or on the two sides where the support wax is located, so this strategy is limiting; however, it is quicker than the Base Clamp and can be used when the model's details allow for it.

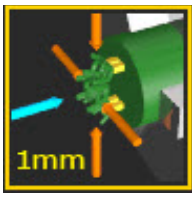


When to Choose this Strategy Choose this strategy for heads, crown assemblies, pendants or earrings that do not have details at the locations this strategy "misses" (shown at right), as it is a quicker strategy for the mill to complete than one of the Base Clamp or Flat on Rotary strategies.

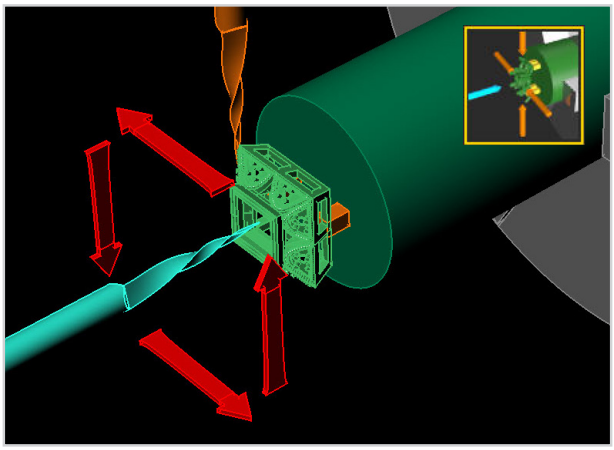
Placing Supports Two supports should be placed at the bottom of the model, sufficient in size to support it from the bottom. If necessary, two more may be placed, one at each side, as in the Three-Sided ring strategy.

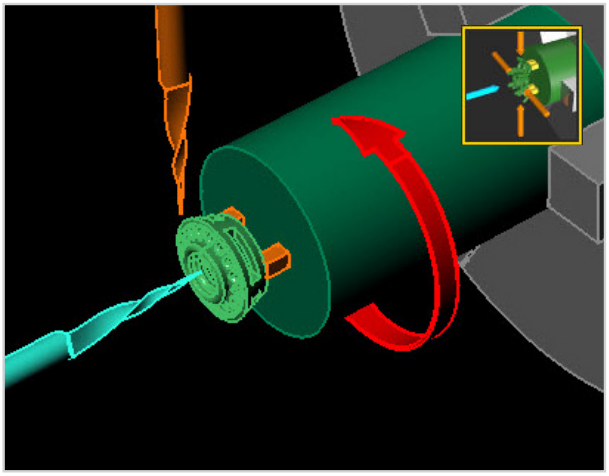
About Non-Ring Strategies An Overview, cont'd.

Each fixture and its associated non-ring strategies (flat first; then rotary) are summarized in this section.



Flat on Rotary Plus One This strategy is cut on the 3-jaw chuck by holding the model flat at four 90-degree angles while cutting each flat side (the "flat on rotary" part of the cut). Because this strategy starts with a top-down cut by the horizontal spindle, it can also mill the tops of models (the "Plus One" part of the cut). The "flat on rotary" cuts are performed when the A axis rotates the model so that one side is perpendicular to the vertical cutter; then holds it flat while completing all of one side; rotates the model 90 degrees and completes another flat cut; and repeats until all four flat sides have been cut. This cut can also be performed with "8 sides" as with the base clamp. Choose this option from the "Flat Sides" builder screen when the details on your model require it. It is also available with the "1 mm" strategy that the Base Clamp uses (select from first screen in builder). Choose this option for more delicate models.



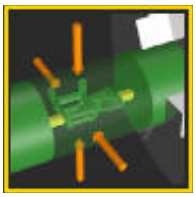


(Above) Four Sides on the rotary chuck plus the top-down cut is best for square-sided models; (Below) Rotary Chuck plus top turns model like a rotary ring; best for rounded models.

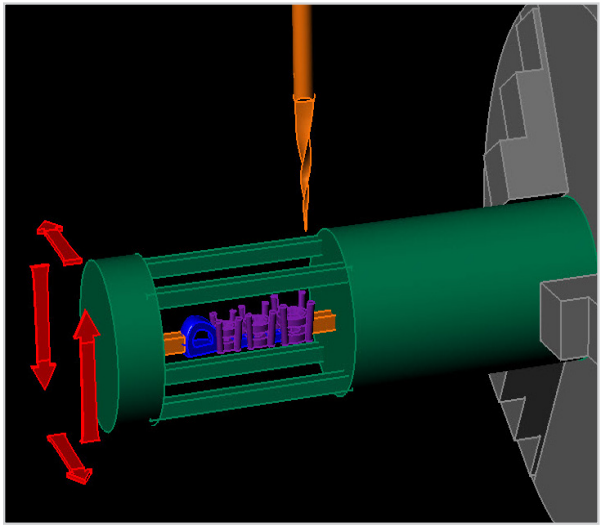
When to Choose this Strategy Choose this strategy for a head, pendant, or earring with no detail on the base (shown at right).

Placing Supports Place two or more supports on the bottom of the model to secure it to the wax tube.

Options In the "Flat Sides" builder screen during the "Flat on Rotary Plus One strategy, the option to select "Rotary" is offered. "Rotary" uses the same set-up and approach, but rotates the model on the A axis while cutting it with the vertical spindle the same way a regular rotary ring is cut. This should be selected when the head, pendant or bezel is round, oval, or rounded in shape - not flat on the sides - as shown above right.



Four Flats on Rotary The final strategy that can be run on the wax tube, Four Flats on Rotary, holds a model such as a pendant or a head assembly from two ends, which are the only parts of the model that won't be milled. This strategy rotates the model so that it is perpendicular to the vertical cutter and makes a flat cut. It then rotates the model 90-degrees and completes another flat cut, repeating these steps until all 4 flat cuts are completed. In the end, a "box" of support material is left behind - where the toolpath hasn't cut - to secure one end of the wax tube to the other.



Rotary fixture rotates model 90 degrees and stops and mills one flat section. This is repeated four times ("Four Flats" on Rotary). Four green wax strips left are created by the mill automatically to hold one end of support wax to the other end. User only creates orange supports.

When to Choose this Strategy Choose this strategy for a pendant or head assembly with no important details on its two ends.

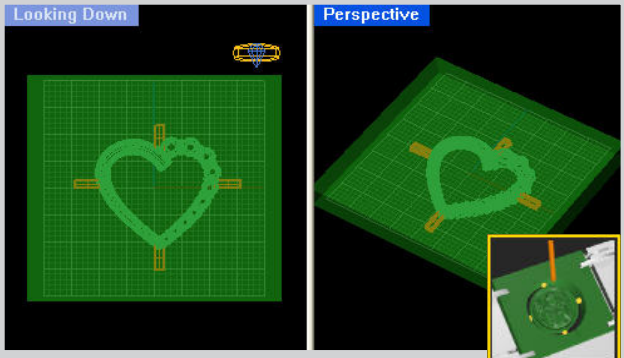
Placing Supports Place two or more sturdy supports on each end of the piece ONLY, as shown at right.

Proper Orientation of Non-Ring Models At-A-Glance

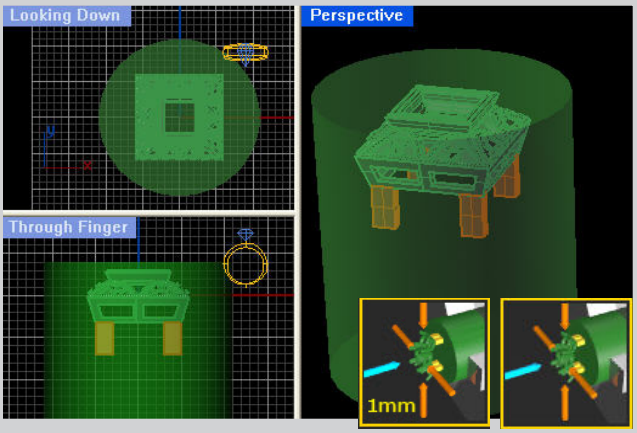
Each non-ring model must be oriented properly in the viewports or the Mill Builder may not find the proper wax to mill it.

About the Proper Orientation of Non-Ring Models

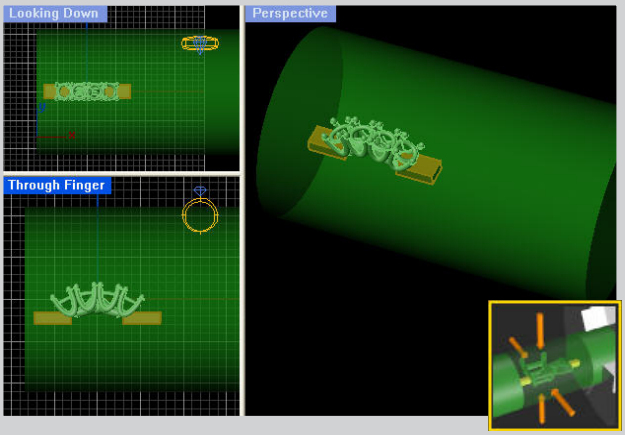
If a model being milled with any of the non-ring strategies is improperly oriented in the viewports, the builder will accept your model AND your supports, and it will even allow you to select a support strategy that corresponds to the selected supports. However, you'll know the model is not at the correct orientation when either the builder informs you that no wax can be found to fit the selected model or your model does not align with the wax preview. In contrast with the ring models, which orient the fixture to the location of the ring rail, the non-ring models can't find your model and instead have static locations for the fixtures in the viewports. These DIFFER based on the support strategy selected - not the fixture, meaning one fixture requires two or even three different orientations if it has two to three different associated support strategies - so pay close attention to the following illustrations to determine how to properly orient your model.



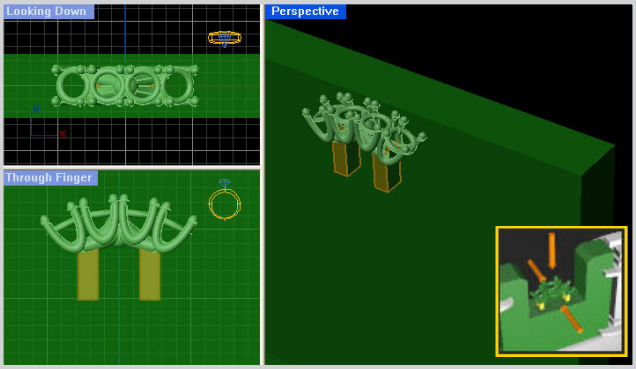
Two-Sided on Dual Fixture Orient this model flat in the Looking Down viewport centered on the Construction plane, with at least four supports (top, bottom, left and right when seen in this view) and the Mill Builder will suggest the proper wax size.



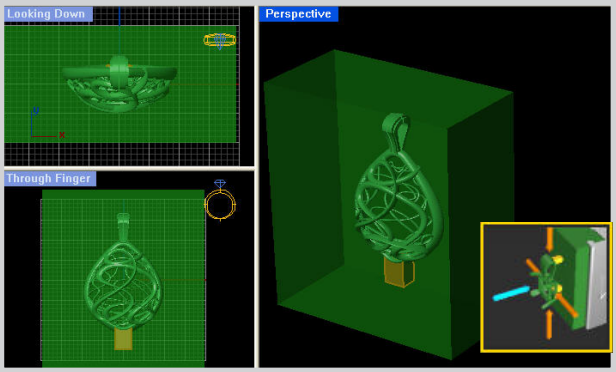
Rotary Plus One & Flat on Rotary Plus One Orient any Rotary Plus One model (cut while turning on rotary or while holding flat on rotary) straight up and down in the Through Finger viewport, with its supports beneath it. The toolpath will end 2 mm above the bottom of the supports, so make them long enough.



Four Flats on Rotary Orient this model flat in the Looking Down viewport and "long" in the Through Finger view, with one support on each end of the model as seen in this view.



Three-Sided Auto Flip Orient this model just as you would a ring, if only the head of the ring were being milled ("long way" visible in Through Finger and "top-down view" visible in Looking Down). Supports are only required beneath model- not at sides.



Base Clamp Orient this model just as you would a ring, with the bulk of it visible in the Through Finger viewport and a top-down view of the model visible in the Looking Down view.

Tutorial 6: Non-Ring Toolpath for the Dual Fixture: 2 sided

Choose this strategy for a non-ring model that requires just two cuts – front and back – on the Dual Fixture.

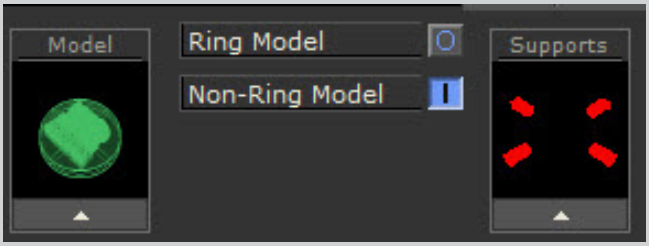
The Two-Sided Cut on the Dual fixture will be the most efficient strategy for this model. All of the details on this piece can be reached via the Front and Back cuts (as seen in the Through Finger view). The model will be supported by 4 supports that you will create with the Box tool in the Solids menu. These supports will then need to be added to the model prior to entering it into the Revo C builder. The supports are added separately and the mill builder will provide selections accordingly.

1 Prepare For Milling

If a model does not have a Ring Rail, the Revo C builder requires that you build the supports manually. Create one support using the Box tool in the Solids menu and measure approximately 2mm thick and extend out from the model at least 3mm or more. Don't forget to Job Bag it!

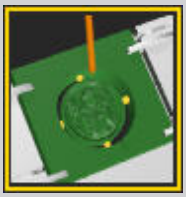
2 Mill Builder

Enter the Mill C builder, select just the model and input it into the "Model" preview window. "Non-Ring Model" will be selected automatically and a Supports window will appear. Select the supports you created and place them into the builder as well.



3 Supports

In this case, we will choose the "Left, Right, Top and Bottom" supports, or the Dual Fixture, because all details in this model can be reached by the "Front" and "Back" cuts completed during this strategy (as seen in the Through Finger viewport).

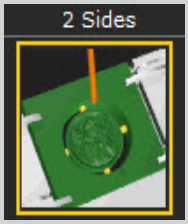


4 Click Next, Choose Wax

Click "Next" to advance the builder. In the second screen, you'll choose the wax size you wish to use to cut this part. Note: If the ACTUAL size of wax you'll use to cut this part is different from the size selected by the builder (smallest possible size is selected by the builder), select the actual size - rather than the size selected for you - to avoid negative results.

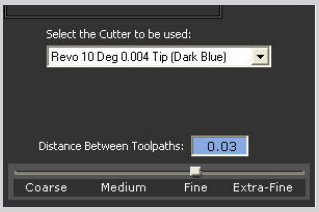
5 Click Next, Choose Angles

Click "Next" to proceed. For a two-sided flip on the dual fixture, only one angle strategy is offered - front and back (180 degrees). Click "Next" again.



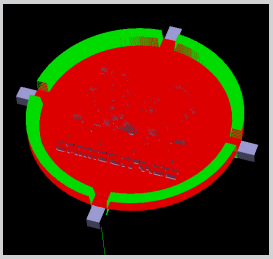
6 Choose Cutter and DBT

The default cutter, 10 dgr, is the cutter you'll use almost every single time in the Mill Builder. It is the cutter to use for this job, as well. "DBT" stands for "Distance Between Toolpaths"; or, how close the "passes" that the cutter makes over your part will be. The "Fine" setting, or 0.3 mm, will be sufficient resolution for almost all the jobs you will run. It is the value for this job, as well.



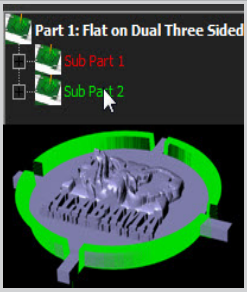
7 Make Path

Click "Make Path" and the Revo C builder will create the STL (or mesh version) and the corresponding toolpath. The toolpath curves will appear in the viewports.



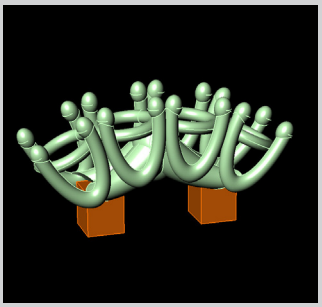
8 Detailed Options and Save

If you wish, click "Next to enter Detailed Options and inspect each part of your toolpath. Or, double-click a Part or Sub Part to see individual toolpaths. When satisfied, click "Save Path" to a chosen location on your computer.

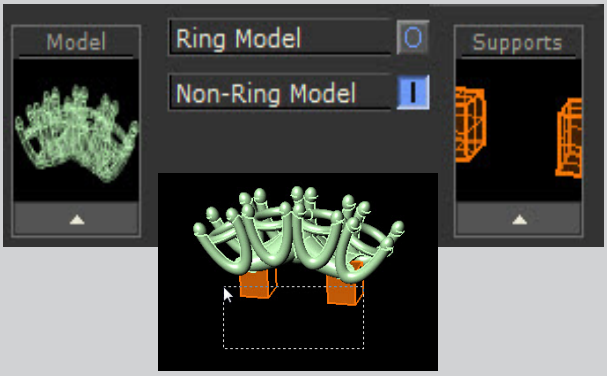


Tutorial 7 : Non-Ring Toolpath for the Dual fixture: 3 sided

Choose this strategy for a ring model that requires just three cuts – top, front, and back (3-sided)– on the Dual Fixture.



The Three-Sided Cut on the Dual fixture will be the most efficient strategy for this model. All of the details on this ring can be reached via the Top, Front and Back cuts (as seen in the Looking Down & Through Finger view). The model will be supported by supports that you create. How many and their placement depends on the type of model you will be milling. In this instance, two supports will suffice to support the head. Create one and place manually to one side - then mirror it to the other side.

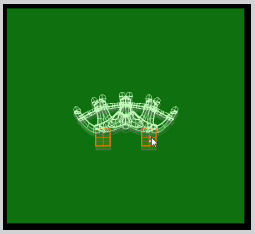


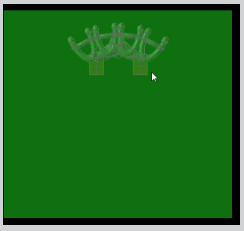
1 Prepare For Milling In preparation for milling a head piece like the one shown above, manually create supports using the Box tool in the Solids menu. Again, be sure the supports are 2x2mm thick and atleast 2mm in length (or longer). Create one support and mirror it to the other side. Job Bag these items.

2 Mill Builder Enter the Mill C builder, select the model, and input it into the "Model" preview window. "Non-Ring Model" should be selected automatically and the "Supports" window will appear. Select and place your supports into this window.

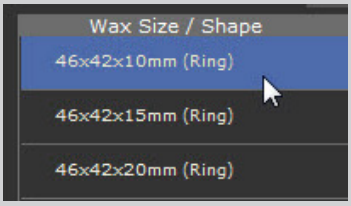
3 Supports Choose the support strategy ("Bottom (Top of Wax)") that corresponds to the Dual 3-Sided fixture. Dual 3-Sided is a good choice as all of the details in this model can be reached by the "Top", Front", and "Back" cuts completed during this strategy (as seen in the Looking Down and Through Finger viewport). Though the Revo C builder will also offer the 3-Jaw Chuck option with a tube wax, we will choose a Top, Front and Back approach for this tutorial.

4 Position Model It will be necessary to re-position your model so that it is near the top edge of the wax. This will ensure that it is in the most ideal location to be milled. Select it in either the Through Finger or Side view port and reposition but be sure the model is still well within the preview block of wax.

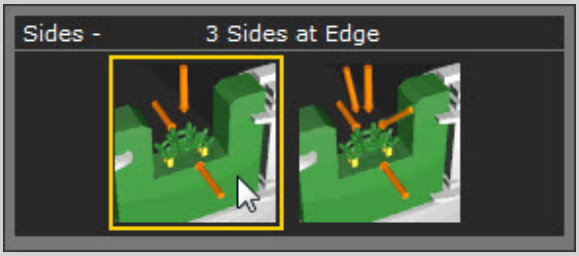




5 Click Next, Choose Wax Click "Next" to advance in the builder. The smallest wax possible to complete the model will be suggested, however it is important to select the actual size wax you will be using. The software will automate any roughing passes required to avoid burying the cutter based on the wax size you select. Click "Next".

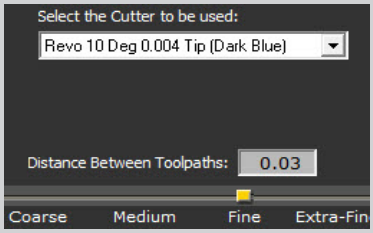


6 Click Next, Choose Angles Click next to proceed. Choose either the 3-sided, or 5-sided approach. Take note, if you choose the 5 sided option, the mill will complete the traditional 3 sides (top, front, and back) and when approaching the model for the 4 & 5 sides, will analyze the geometry and mill ONLY the areas that could not be reached with the traditional 3 angles. Click "Next".

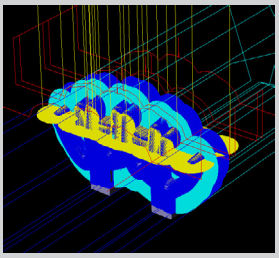


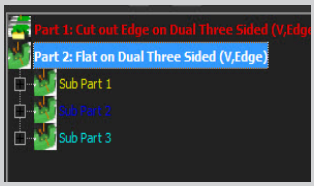
Toolpath for Dual fixture: 3 sided cont'd

7 Cutter and DBT The default cutter, 10 dgr, is the cutter you'll use almost every single time in the Mill Builder. It is the cutter to use for this job, as well. "DBT" stands for "Distance Between Toolpaths"; or, how close the "passes" that the cutter makes over your part will be. The "Fine" setting, or 0.3 mm, will be sufficient resolution for almost all the jobs you will run. It is the value for this job, as well.

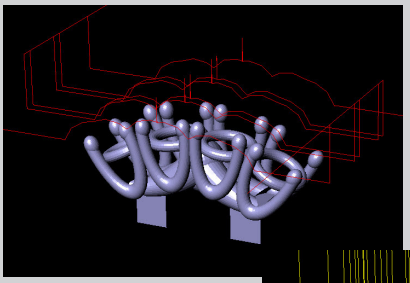


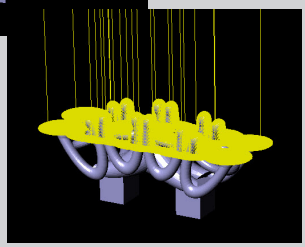
8 Make Path Click "Make Path" to see the results. The toolpath will display on screen. Inspect the toolpath and verify all the detail on your model will be reached. You should see the toolpath drape over your entire model. As the toolpaths become more complicated, it is easier to view the toolpath individually as opposed to trying to make sense out of overlapping toolpaths. To view these paths individually, Click "Next"





9 Detailed Options Each Part of the toolpath has been separated into a list format. A Part can consist of one approach angle, or multiple approach angles. Click on Part 1 to view only that portion of the toolpath. The list has been color coded to match the toolpaths on screen. In this example, Part 1 is red and is the "Cut Out Edge on Dual Three Sided (V)". During this Part, the cutter will follow the red curve to remove a large block of excess wax quickly and allow for the next cuts. The (V) means this is cut with the vertical cutter. Next, click Part 2. Part 2 is white. You may notice, there is no white toolpath on screen. Anytime you see white in the toolpath list, it means this portion of the toolapth is comprised of more than one "Sub Part".Next click Sub Part 1 of Part 2. Sub Parts 2 and 3 represent the Front and Back portions of the toolpath.

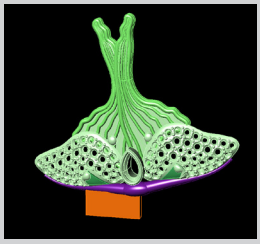




10 Save Path After inspecting the toolpath, choose "Save Path". Save this to the "RVO" folder found in the "My Documents > Matrix" folder, or to any location on your computer where later you can transfer the file to a thumb drive to load onto the Revo Mill's computer.

Tutorial 8 : Non-Ring Toolpath for the Base Clamp Fixture

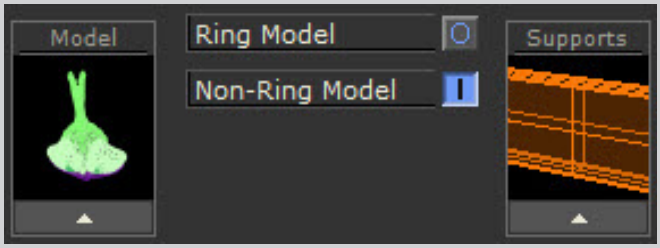
Create a toolpath for the Base Clamp fixture. The Base Clamp can mill detail from 9 sides.



The pendant at left has details all around the outside, top of the pendant, down the sides, and even at unique angles on the front and back of the model. The only place it lacks detail is at the very bottom, outside of the base, where the sprue would usually be placed for casting. It is therefore the perfect model for use with the Base Clamp fixture. The Base Clamp can approach from 9 different angles - ideal for cutting complex designs.

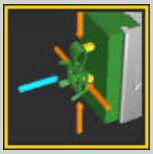
---

1 Prepare For Milling In preparation for milling, supports must always be manually created for any non-ring models. The support's dimensions are largely dependant on the model being milled. In this case, the support will need to be 2mm thick, 9mm wide and over 4mm in length.

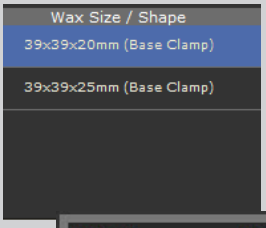


2 Mill Builder Enter the Mill C builder, select the model, and input it into the "Model" preview window. "Non-Ring Model" should be selected automatically, and the Supports window will appear. Place the lone support into the window.

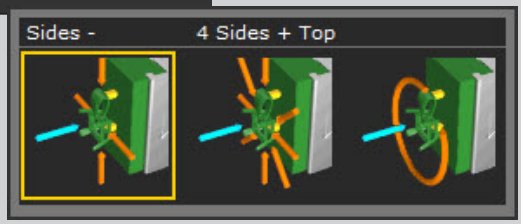
3 Supports . In this case, we will choose the "Bottom" support, which is the support strategy corresponding to the Base Clamp fixture. This strategy can reach the top-down (Looking Down view of the model), front and back (Through Finger view of the model) AND left and right (Side View of the model) sides of the model.The Base Clamp has the additional benefit of adding four custom angles 45-degrees to the front, back, left & right.

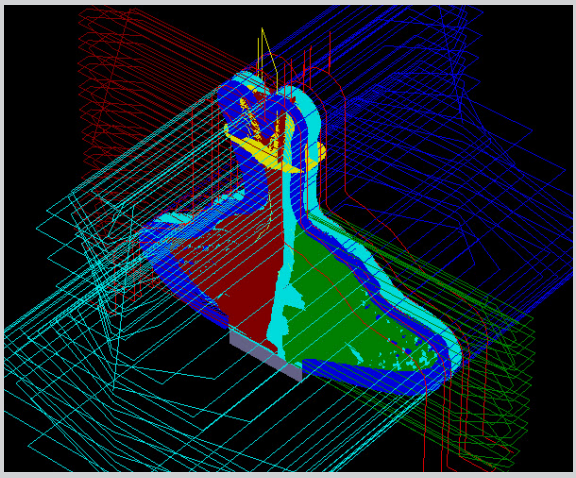


4 Choose Wax In the second screen, the smallest wax possible to complete the model will be suggested, however it is important to select the actual size wax you will be using. The software will automate any roughing passes required to avoid burying the cutter based on the wax size you select. Click "Next".



5 Approach Angle Analyze where the details are on the model and choose the appropriate approach. For this model because of the angle of the detail on the sides and top, 4 sides will be chosen. Take note, if you choose the 8 sided or Rotary options, the mill will complete the traditional 4 sides and when approaching the model for the additional sides, will analyze the geometry and mill ONLY the areas that could not be reached with the traditional angles. This will save time, but also avoid over milling areas of the model. Best practices for milling are to avoid milling the same area more than once. Click "Next"to proceed in the builder.



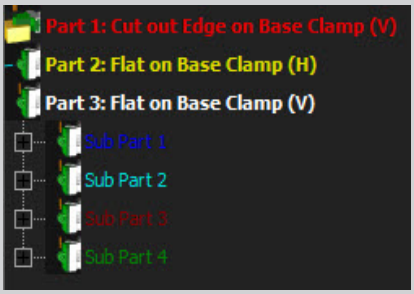


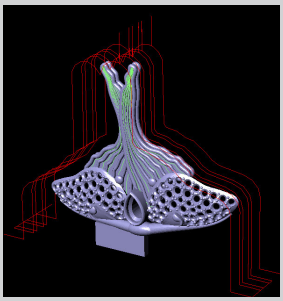
6 Cutter and DBT In this example, the 10 Degree cutter will be selected. Click "Next".

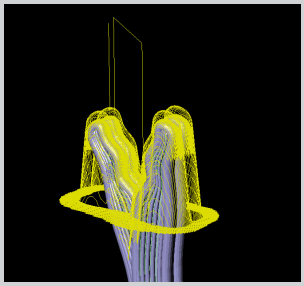
7 Make Path Click "Make Path" to see the results. The toolpath will display on screen. Inspect the toolpath and verify all the detail on your model will be reached. You should see the toolpath drape over your entire model. As the toolpaths become more complicated, it is easier to view each part of the toolpath individually as opposed to trying to make sense out of overlapping toolpaths. To view these paths individually, click "Next".

Non-Ring Toolpath for the Base Clamp Fixture cont'd

8 Detailed Options Each Part of the toolpath has been separated into a list format. A Part can consist of one approach angle, or multiple approach angles. Click on Part 1 to view only that portion of the toolpath. The list has been color coded to match the toolpaths on screen. In this example, Part 1 is red and is the "Cut Out Edge on Base Clamp". During this Part, the cutter will follow the red curve to remove a large block of excess wax quickly to allow for the next cuts. The (V) means this is cut with the vertical cutter. Next, click Part 2. The light yellow curve is the "Flat on Base Clamp" where the cutter cuts the model from the top down as seen in the Looking Down viewport. This cutter approach uses the (H) Horizontal spindle. and Part 4 is white. You may notice, there is no white toolpath on screen. Anytime you see white in the toolpath list, it means this portion of the toolapth is comprised of more than one "Sub Part". Double Click Part 4 to expand this Part into its Sub Parts. Four Sub Parts will display.





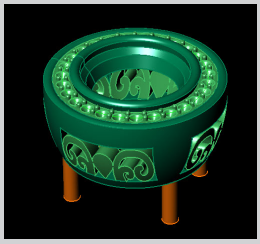


Note: During the Top Down cut it appear as if one toolpath shadows the other. This is what is known as a roughing pass, where the software analyzed how deep the toolpath was required to plunge into the wax and based on the cutter definition added an additional pass so the cutter would not be buried too deep jepordizing the safety of the cutter. The Revo 10 Deg .004 Tip (Dark Blue) will plunge 10mm deep into wax before a roughing pass will be added.

9 Save Path After inspecting the toolpath, choose "Save Path". Save this to the "RVO" folder found in the "My Documents > Matrix" folder, or to any location on your computer where later you can transfer the file to a thumb drive to load onto the Revo Mill's computer.

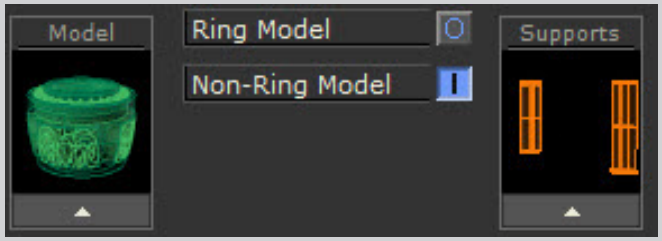
Tutorial 9 : Rotary Plus One for the 3-Jaw Chuck Fixture

Choose this strategy to complete a top-down cut plus a rotary cut on a non-ring object that is round.



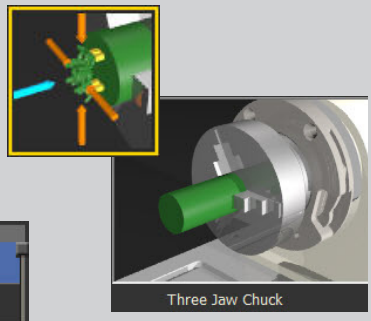
The "Rotary Plus One" strategy mills from the top-down with the Horizontal cutter and then completes a rotary cut (model is cut while it is being turned on the A axis) with the Vertical cutter. It is best used for circular non-ring models. For square models or eight-sided models, the Rotary Plus One option offers four- or eight-flat cuts on rotary, which hold the model flat while completing the cuts. But since our model is round, it will benefit best from a rotary cut – not four or eight flat cuts. Supports are located on the bottom of the model.

1 Prepare For Milling In preparation for milling, supports must always be manually created for any non-ring models. The support's dimensions are largely dependant on the model being milled. In this case, the support is created using the Cylinder tool in the Solids menu. It is placed at the base of the model, then quad flipped. Be sure it is centered on F4 with the Top "UP" in the Through Finger viewport - to minimize the amount of wax used. Then Job Bag it.



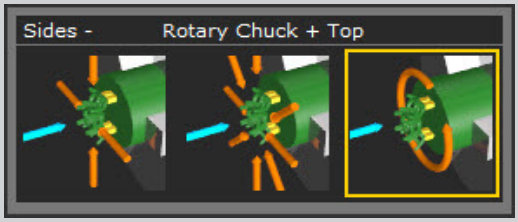
2 Mill Builder Enter the Mill C builder, select the model, and input it into the "Model" preview window. "Non-Ring Model" should be selected automatically, and the Supports window will appear. Place the four supports into the window.

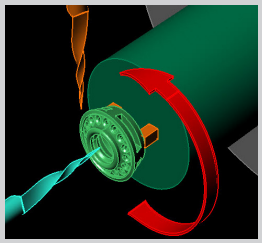
3 Supports The "Bottom" support corresponding to the 3-Jaw Chuck fixture should be automatically selected. If it is not, select it at this time. This strategy can reach the top-down (Looking Down view of the model), and all sides of the model depending on the Approach Angle selected.



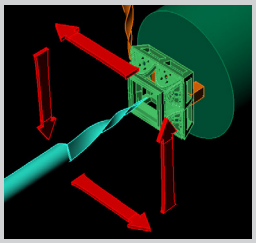
4 Choose Wax In the second screen, the smallest wax possible to complete the model will be suggested (in this case, the 22mm round tube wax), however it is important to select the actual size wax you will be using. Click "Next".

5 Approach Angle Analyze the shape of the model you will be milling as well as the location of the details. The option we'll use to cut this model will be "Rotary Chuck + Top", where the model is cut on the rotary (while the A axis is turning) during this cut, rather than being held flat on the A axis at 4- or 8-angles (the other two options shown in this screen). We're choosing this option because it is a round model, with details that can best be achieved with a rotary cut. If the detail on this model was flat, the appropriate approach angle should be selected. These cuts are performed when the A axis rotates the model so one side is perpendicular to the vertical cutter, then rotates the model 90 degrees to complete the next flat cut - and so on until the model is finished.



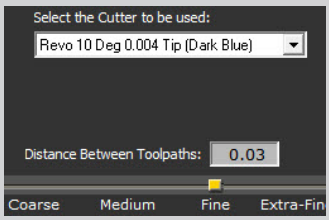


Rotary Chuck + Top



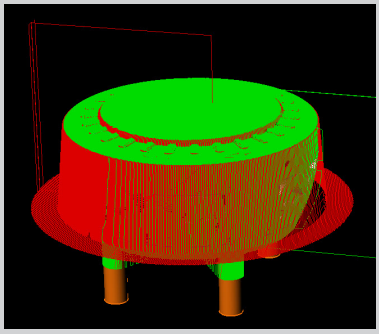
4 Sides Chuck + Rotary

6 Cutter and DBT In this example, the 10 Degree cutter on a setting of "Fine" will be selected. Click "Next".

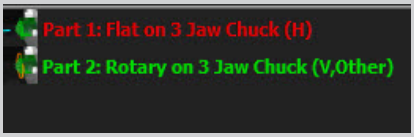


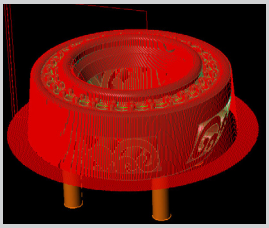
Rotary Plus One for the 3-Jaw Chuck Fixture cont'd

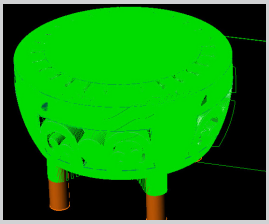
7 Make Path Click "Make Path" to see the results. The toolpath will display on screen. Inspect the toolpath and verify all the detail on your model will be reached. You should see the toolpath drape over your entire model. To view these paths individually, click "Next".



8 Detailed Options Each Part of the toolpath has been separated into a list format. A Part can consist of one approach angle, or multiple approach angles. Anytime you see white in the toolpath list, it means this portion of the toolapth is comprised of more than one "Sub Part". In this case, there are no WHITE Parts. Just the Part 1: Flat (or Top) cut and the Part 2: Rotary cut.



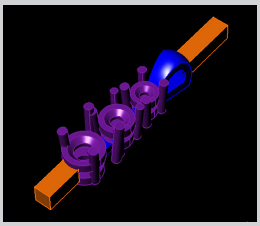




9 Save Path After inspecting the toolpath, choose "Save Path". Save this to the "RVO" folder found in the "My Documents > Matrix" folder, or to any location on your computer where later you can transfer the file to a thumb drive to load onto the Revo Mill's computer.

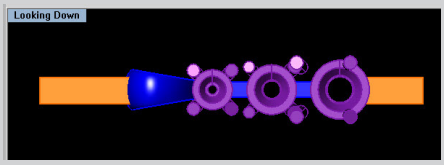
Tutorial 10 : 4 Flats Plus Rotary for the 3-Jaw Chuck Fixture

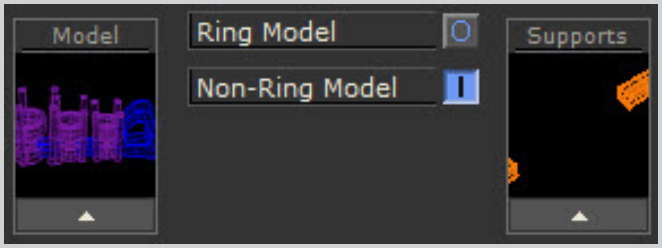
Choose this strategy to complete 4 sided non-ring object.



This strategy will hold the model flat on the 3-jaw chuck during milling; rotate it 90-degrees, and hold it flat for milling once more. This process is repeated for a total of four flat cuts ("Four Flats"). The user creates supports (Step 3) to hold the model to the wax at either end – the only place on the model that will not be milled and must be cleaned up later – and additional supporting material in the shape of a "wax box", holding one end of the wax tube to the other, is left behind by the mill when it is milling inside the border for each side of this piece.

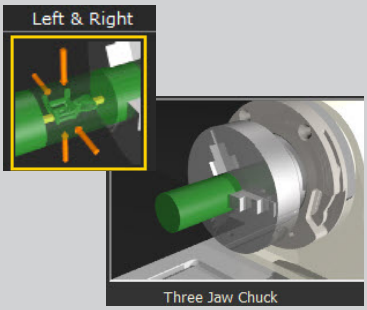
1 Prepare For Milling In preparation for milling, supports must always be manually created for any non-ring models. Using the Box tool found in the Solid menu, create two supports, 2x2 mm thick and at least 3mm long, on both sides of this piece. These affix it to the wax tube on either end. You will need to clean up this part of the model after cutting. Also, this model MUST be oriented the "long way" in the Looking Down viewport for this strategy to work correctly. Job Bag!





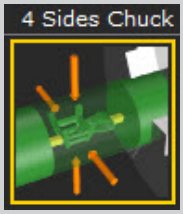
2 Mill Builder Enter the Mill C builder, select the model, and input it into the "Model" preview window. "Non-Ring Model" should be selected automatically, and the Supports window will appear. Place the two supports into the window.

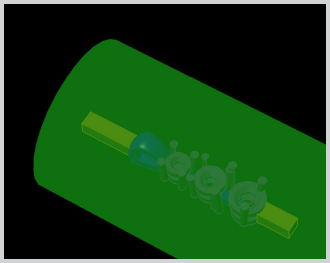
3 Supports The "Left and Right" support corresponding to the 3-Jaw Chuck fixture should be automatically selected. If it is not, select it at this time. This strategy can reach all four sides of the model.



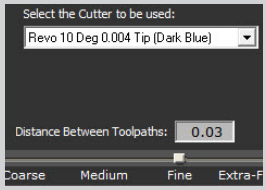
4 Choose Wax In the second screen, the smallest wax possible to complete the model will be suggested (in this case, the 22mm round tube wax), however it is important to select the actual size wax you will be using. Click "Next".

5 Approach Angle The option available to cut this model will be "4 Sides Chuck". In the orientation of this strategy, the A axis is at the right-hand side of the Looking Down viewport

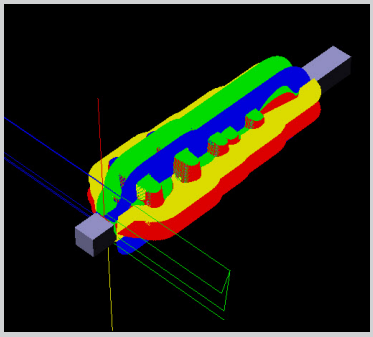




6 Cutter and DBT In this example, the 10 Degree cutter on a setting of "Fine" will be selected. Click "Next".

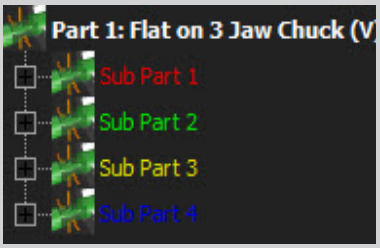


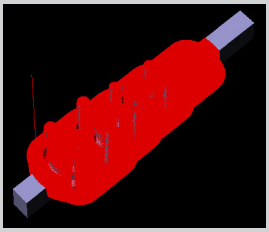
7 Make Path Click "Make Path" to see the results. The toolpath will display on screen. Inspect the toolpath and verify all the detail on your model will be reached. You should see the toolpath drape over your entire model. To view these paths individually, click "Next".

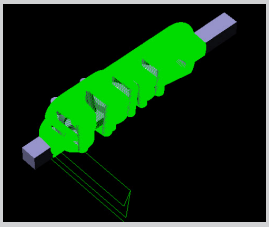


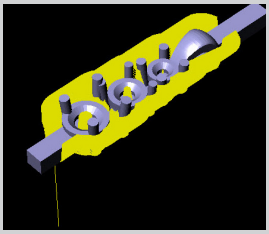
4 Flats Plus Rotary for the 3-Jaw Chuck Fixture cont'd

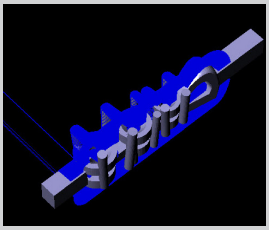
8 Detailed Options Each Part of the toolpath has been separated into a list format. A Part can consist of one approach angle, or multiple approach angles. Anytime you see white in the toolpath list, it means this portion of the toolapth is comprised of more than one "Sub Part". In this case, there are no WHITE Parts. Just the Part 1: Flat (or Top) cut and the Part 2: Rotary cut.











9 Save Path After inspecting the toolpath, choose "Save Path". Save this to the "RVO" folder found in the "My Documents > Matrix" folder, or to any location on your computer where later you can transfer the file to a thumb drive to load onto the Revo Mill's computer.

Milling Each Strategy ...

Steps for Each Strategy

Special steps in the Revo software on the mill are required for various strategies. These are explained below.

Using the 3-Jaw Chuck: Set Wax Edge

When loading ANY part (Ring or Non-Ring) that uses the 3-Jaw Chuck fixture, after set-up of the cutters, wax, and fixtures, the Revo software on the mill will prompt you to "Show Wax Edge". Click "Move", and the cutter location will move right to the furthest location possible for the loaded toolpath.

Of course, each time you install a wax tube, it will be a different length. So, after the mill has moved to the "minimum length" position, it will ask you to "Set". Now, it's up to you to "Jog" the mill; or, move the axes so that the cutters are positioned at the end of the current wax tube. This indicates to the mill where it should start the toolpath in order to correctly cut your model. Use the Jog Controls in the Revo software to do this. You'll want them on a "Slow" or "Medium-Slow" Setting so you don't crash the cutters into the wax or fixture, which will break them.

Once you have jogged the axes to the correct location, click "Set". The mill will return "Home" while saving the location of the Wax Edge in its memory, so it can start the toolpath in the right place. After this step, proceed with the instructions as usual when running a job: check the coolant levels and click "Start" to complete the model.

Centering the Hub

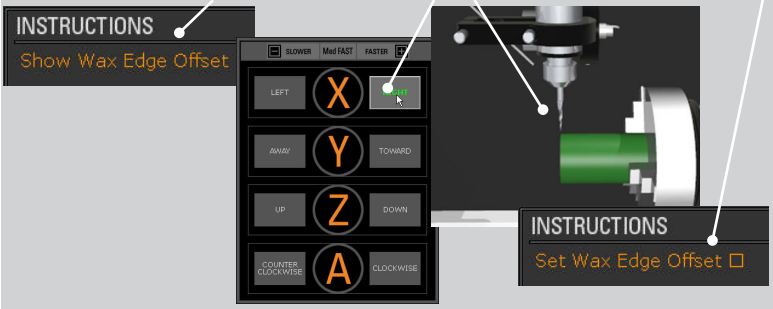
When the Hub Center is selected during a flat strategy, two additional cuts are run: the first right at the end of the selected flat strategy, and the second right after transferring the model to the Hub fixture. After the second special cut is completed, the mill will stop and the software will prompt you to inspect the location of these special cuts. You will select, onscreen, the location on the illustration where the horizontal and vertical cuts are most symmetrical. Click here, and the mill will automatically adjust for any inconsistencies between the flat and rotary cuts, centering the model in preparation for a successful rotary cut. You will then simply tell it to finish cutting the part.

Steps for Setting up 3-Jaw Chuck (for ANY Job)

1 - "Show Minimum Wax Length" When this prompt appears in the "Instructions" box in the mill software, click "Move" to find the furthest location needed to mill the chosen toolpath.

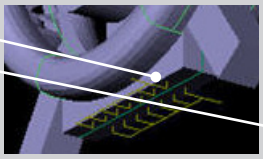
2 - Jog Mill to Edge of Wax Tube. It is now up to you to use the Jog Controls in the Revo Software to jog the Vertical cutter to the end of the wax tube, as close as possible, where the toolpath should start.

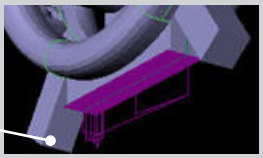
3 - "Set Wax Edge". The next Instructions prompt you to set the location where the mill will begin. ONLY click "Set" once the vertical cutter is located at the end of the wax tube (at the start of the toolpath).

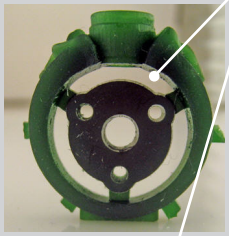


Centering a Model on the Hub Fixture

1 - "Hub Cal" Cuts Two special cuts are added to the Hub strategy: one completed on the flat and one completed on the rotary after the model is transferred from the flat fixture to the hub fixture.

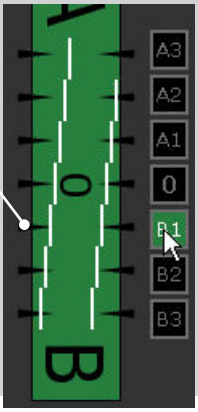






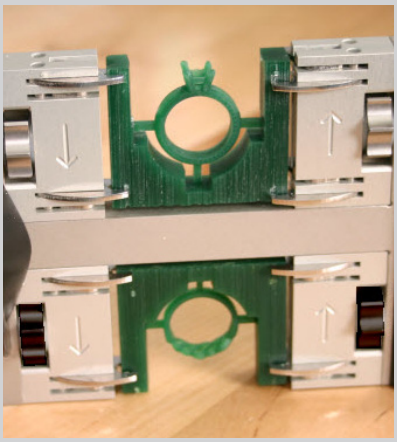
2 - Transfer Model to Hub Remove the model, with its wax hub intact, from the flat wax. Install it on the hub fixture by aligning the three pin holes with the three pins. The arrow should be pointing up and facing you. Attach cap, screw, and washers. Install hub fixture & complete second part of Cal cut on the rotary.

3 - Choose Position in Software In the mill software, click in the position on the diagram corresponding to the ACTUAL cuts in the model where one pair of horizontal cuts overlap the vertical cuts evenly. Revo will center & continue.



Tutorial 1: Two Jobs on the Dual Fixture

Learn to load two jobs onto the Dual Fixture - one in each position or the same file in both positions.



When loading a job onto the Dual Fixture, two possible load positions will appear: Load Revo File 1 and Load Revo File 2. Revo File 1 will be cut in Position 1 on the fixture, which is has the black thumb screws and is closest to the mill when the fixture is first installed (the mill is in the Home position). Revo File 2 will be cut in Position 2, which has silver thumb screws and is closest to the user when the fixture is first installed. -

Materials Needed

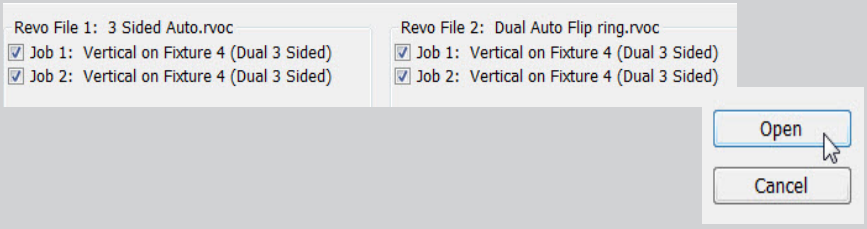
Matrix Mill C Builder:

- 1 or 2 Dual 3-Sided Fixture RVOC files

Revo:

- Dual Three-Sided Fixture

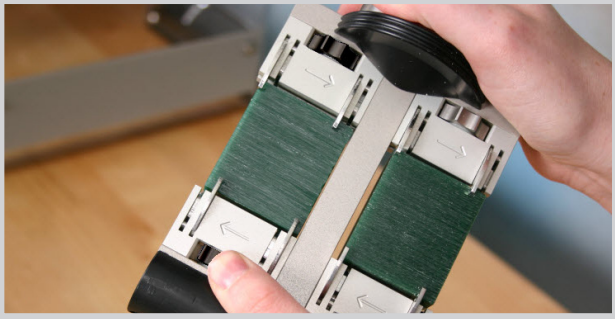
- Cutters, Waxes for each job

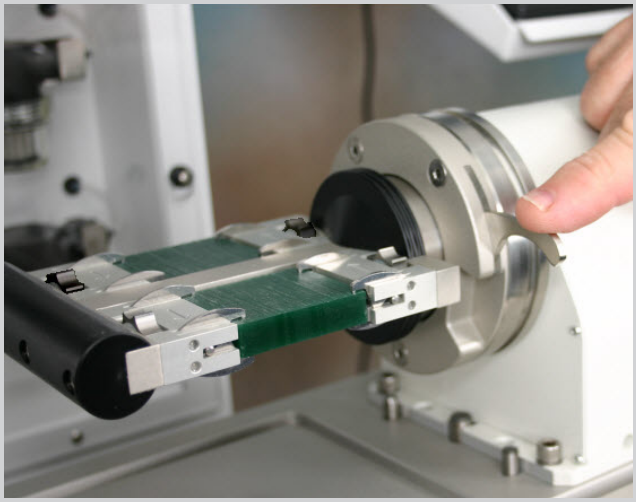


1 Load Job In the Revo software, begin by clicking "Load Job". Navigate to the RVOC file on your Revo computer that requires the Dual Three-Sided Fixture.

2 Choose File 1 or 2 Click on a Dual Three-Sided file to run and click *Load Revo File*. The file will automatically load into Revo File 1, however, two possible positions will appear below the file name: *Load Revo File 1* and *Load Revo File 2*.

3 Open Click *Load Revo File 2* to load the same file into Position2, or click on a different file you wish to load into Position 2. After choosing a position for each RVOC file, the job name will populate the "Revo File" area of the dialog. Choose either to run a file in Position 2 or no file for Position 2. Just be certain, if you choose NO file for the other position, that you install the wax in the correct side. With one or both positions populated, click "Open" to return to the Revo software.

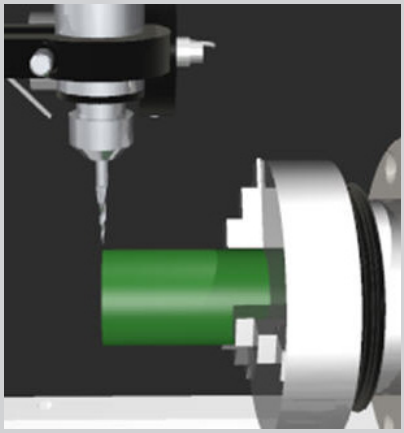




4 Final Step Install Dual 3-Sided Waxes, Fixture. When you are following the Instructions in the Revo software to load the waxes into the Dual Three-Sided fixture, be CERTAIN to install the correct-sized wax for each job in the correct side of the fixture: Position 1 has black thumb screws and Position 2 has silver clamp screws.

Tutorial 2: Set Wax Edge on Three-Jaw Chuck

Each and every time you run a job on the three-jaw chuck (ring OR non-ring), the software will prompt you to indicate the edge of the wax.



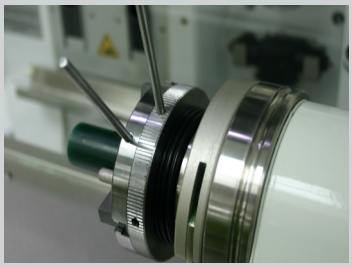
In order to indicate to the Revo software where the toolpath should start (i.e., where the end of the wax tube is located in space) for the three-jaw chuck fixture, the Revo software walks you through a few special steps when setting up this strategy. It will prompt you to align the tip of the cutter over the edge of the wax. Click "Set". Next, use the Jog Controls to place the spindle at the actual edge of the wax, and click "Set. The mill will start the toolpath at the location you indicated during "Set " > "Move".

Materials Needed

Matrix Mill C Builder:
.RVOC requiring Three-Jaw Chuck

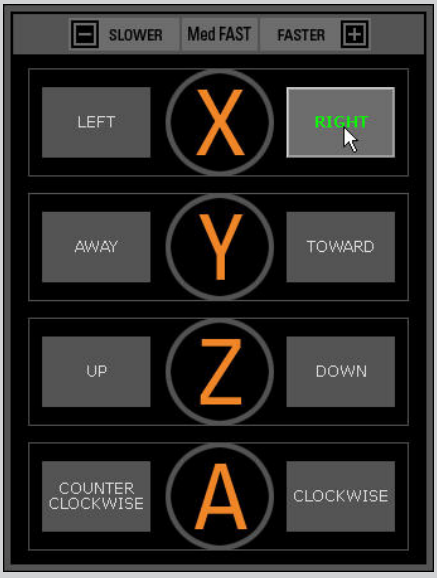
Revo Mill:
Three-Jaw Chuck Fixture
Correct Wax & Cutter for the Job

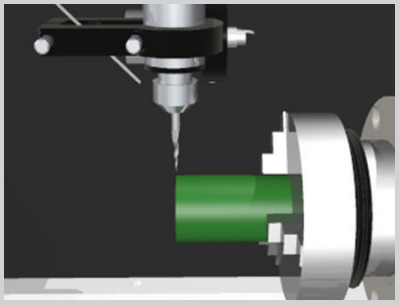
1 Load .RVOC requiring 3-Jaw Chuck Fixture Click "Load Job" and select an RVOC file requiring the three-jaw chuck fixture.



2 Install the 3-Jaw Chuck Follow the directions in the Revo software to install and measure the correct cutter(s), install the fixture on the A axis and load the correct wax size in the three-jaw chuck fixture (ONLY the wax tube may be used when a hollowing cutter is being used).

3 Show Minimum Wax Length > Move Click "Move" when the Minimum Wax Length instructions appear. Click "Move". The mill will move to the location where the smallest length of the wax could be used for the selected toolpath,





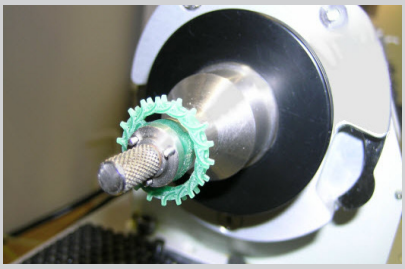
4 Jog the Mill In the Revo software, click on the Jog Control (shown at left) to manually jog (move) the mill into position so that the vertical cutter aligns with the end of the wax. Click "Slower" to set the mill to the slowest speed at which you can stand to move it.

5 Final Step: Set With the cutter aligned with the end of the wax tube, click "Set" in the Actions and the mill will return home, storing this value. Complete the instructions and Start this job, and it will start in the correct position at the end of the wax.

Tutorial 3: Centering the Hub

A special centering strategy appears in the Revo software after transferring a model to the hub fixture and completing a short pass.

When adding the Hub fixture, the mill completes a special "Hub Cal" pass at the end of the flat cuts on the Dual Fixture. (This is the toolpath referred to as "Hub Cal 1" in the Detailed Options screen in Matrix Mill C Builder.) Then, the mill software will prompt you to transfer the model to the hub fixture. After a short cut on the hub fixture ("Hub Cal 2" in Detailed Options), the mill will pause and the software will prompt you to inspect the model. Do NOT remove the model from the fixture! Simply inspect the base of the model, where the Hub Cal parts have cut, and examine the vertical and horizontal cuts that were made. It may help to dust the model with chalk dust to see these cuts better. Vertical lines running between the letters "A" and "B" on the base piece cross horizontal tic-marks on the left and right-hand sides of the base piece. A corresponding illustration appears in the Revo software. Examine your model to determine where the vertical lines cross the horizontal ticmarks evenly, so that each ticmark in a pair is covered equally by the horizontal line. Click and move the slider to that location on the diagram. The diagram will change to match your model. Cick OK and close the dialog and the mill will continue cutting. Congratulations: you have just centered the hub!

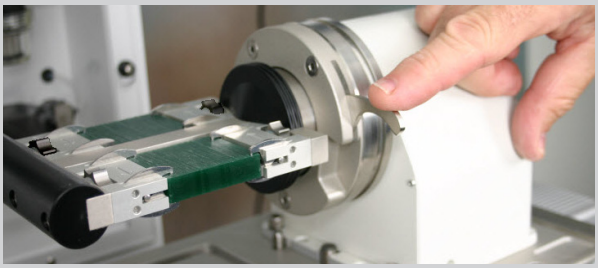


Materials Needed

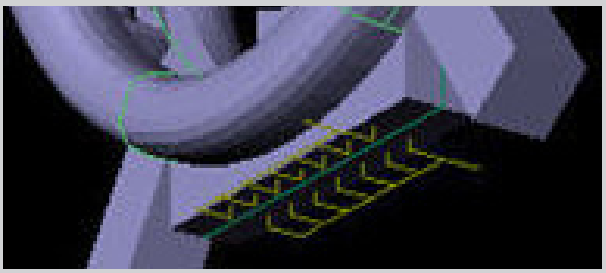
Matrix Mill C Builder:
- **RVOC file with Hub Center**

Revo:
- **Dual Three-Sided Fixture**
- **Hub Fixture**

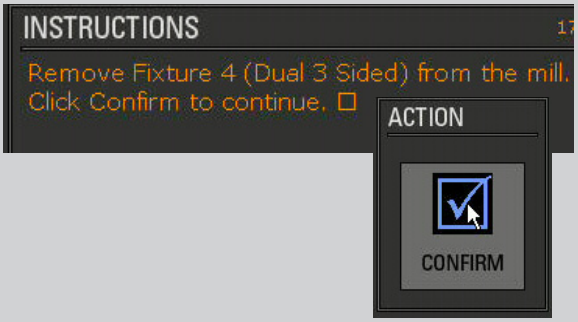
Optional:
- **Chalk Dust & a loupe to best view ticmarks on Hub Base Piece.**



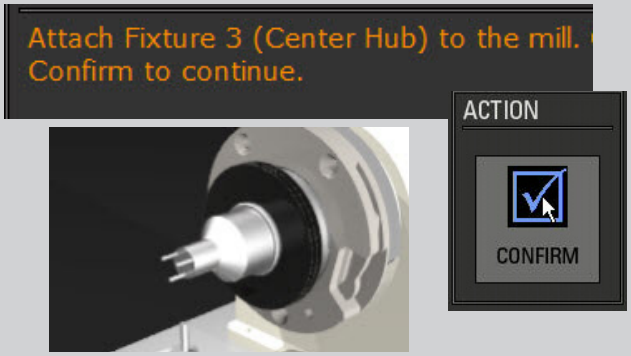
1 Dual 3-Sided Fixture Load a file cut on the Dual Three-Sided fixture with a Hub Center, to follow up the flat cut with a rotary cut on the Hub Fixture. Follow the prompts in the mill software to install and measure the correct Vertical cutter, load the wax into the Dual 3-Sided Fixture, and install the fixture on the mill.



2 Hub Cal Part 1 At the very end of the job on the Dual 3-sided fixture, the mill will cut a special toolpath into the base piece on the model (the base piece is added in the Matrix Mill C Builder in the Center Supports screen). This special cut, together with Hub Cal Part 2, will help the user tell the mill where the center of the model is when it is transferred to the hub fixture.



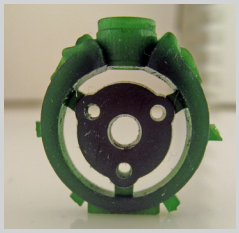
3 Remove Model from Dual With flat cuts completed on your model, you may now remove the fixture from the A axis and remove the wax from the fixture. GENTLY remove the model from the wax. Be CERTAIN you leave the hub and base piece intact. (If any supports remain, they will be removed by the mill during the rotary cut on the hub fixture.

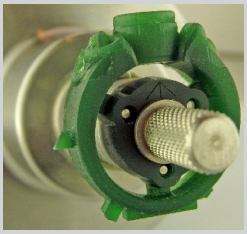


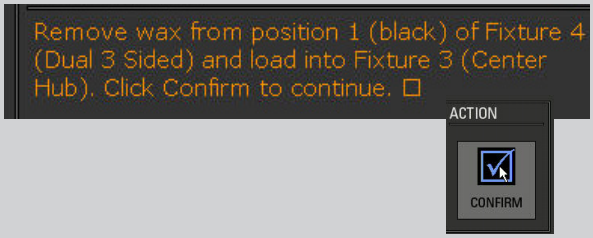
4 Attach Fixture 3 Next, attach Fixture 3 to the mill to prepare for the Rotary phase.

Centering the Hub cont'd.

A special centering strategy appears in the Revo software after transferring a model to the hub fixture and completing a short pass.

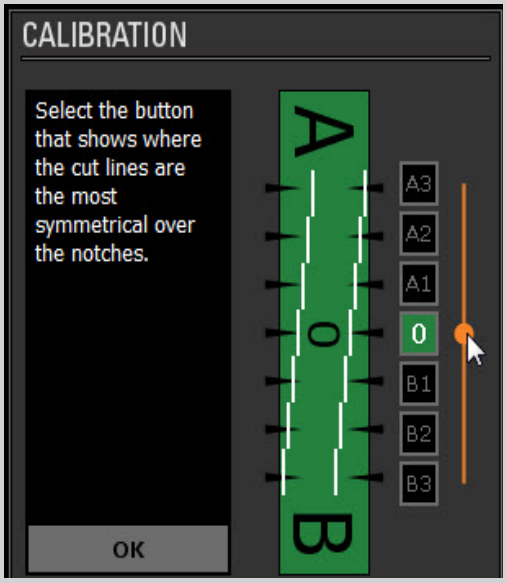


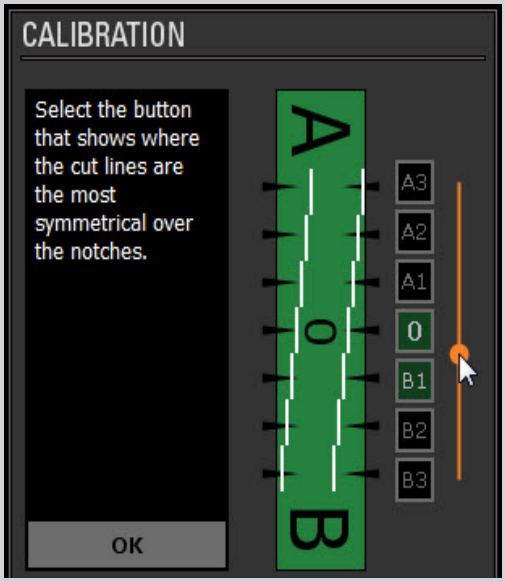




5 Install Model on Hub Install the model on the Hub fixture, taking care that the arrow on the center support piece is pointing upward and facing you when the Hub Fixture is installed on the A axis. After placing the model on the fixture, install, in this order, the cap, two washers, and screw to secure it in place.

6 Hub Cal Part 2 The first cut the mill runs on the Hub Fixture is a short pass completing the Hub Cal cuts. The software will now pause and prompt you to Move the model into the correct position to inspect the base piece on the model.



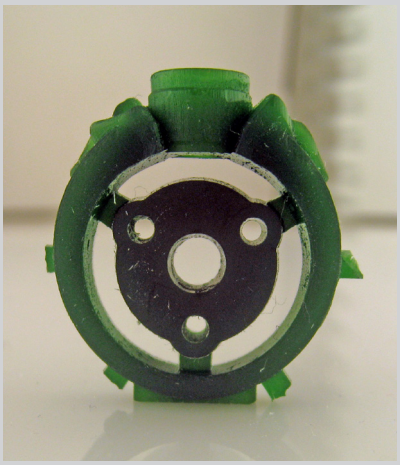


7 Inspect Base Piece Take a look at the base piece (far left illustration, above), where the vertical lines running down the model from A to B overlap the horizontal tic-marks on either side of the base piece. The vertical lines will overlap one pair of ticmarks evenly, so that the same amount of ticmark in that set (left and right = one set) is equally intersected by the horizontal cuts (outlined in red, above left). **Keep in mind that these marks are not ALWAYS perfectly symmetrical. The key then is to find the MOST SYMMETRICAL marks on the wax.

8 Final Step Click on Diagram to Match Model (Center illustration, above.) Click on the button in the diagram onscreen ("B1", above) to indicate the set of ticmarks where the vertical line intersects each ticmark in the set equally. The diagram will change to match your model (illustration on far right, above). Double-check that the diagram correctly matches the model and Close the dialog window. The final Load Part of this job will load, and all rotary cut(s) on the hub will be completed. NOTE: In the event the location where the ticmarks evenly overlap the vertical lines fall between two locations, use the slider bar and choose a location along the calibration where the ticmarks are evenly overlapping the vertical lines.

Tutorial 4: Running Part of a Job

Do you wish to load part - and not all - of a RVOC file? During "Load File", load one of the component "Jobs" in that file ONLY.



If you happen upon an occasion when you'll need to run part of a job, but not all, READ THESE TIPS THOROUGHLY before completing the steps below.

(1) Be certain you've ALREADY CUT all previous parts of the job. If the cut-out curves and other essential parts of the job are not run, this could damage your mill, your cutter, and/or your model.

(2) Make sure you haven't removed the fixture from the mill before loading the next part of the job when it takes place on the same fixture, unless of course the next part you're loading calls for another fixture (i.e. the rotary part of a dual three-sided job with a hub center.)

Materials Needed

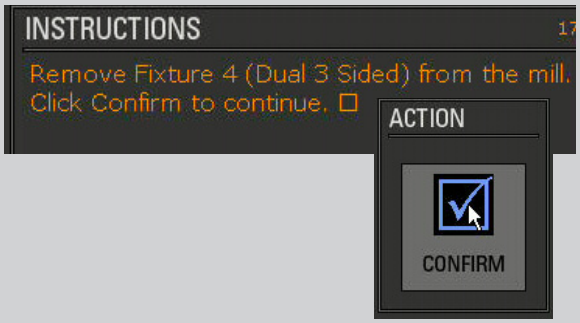
Matrix Mill C Builder:
- A RVOC file with more than one "Job" associated with it

Revo:
- The correct fixture, wax, & cutter(s) for the job

1 Load the Job After stopping the mill partway through a job, NOT removing the fixture (unless it calls for it) and returning to the mill later when it is safe to do so, click "Load Job" and return to the job you were running.

If the Dual 3-Sided part of the job was already run, deselect it. However, leave the first Job part on the Hub fixture (Hub Cal 2) selected: do NOT skip essential parts that weren't run yet!

2 Choose the Job Part to Run Notice that, upon selecting a part to load in the Revo software, the dialog populates at the bottom with each "Job part" associated with that model. Deselect any job part you do not need to run (BE CERTAIN that you do not de-select support cuts or cut-out curves if these have not been run yet.)



3 Final Step Complete Set-Up Install and measure cutters, load wax, install fixture, etc. in accordance with instructions. OR, if cutter, wax and fixture are already installed, "Skip" these steps: DO NOT remove wax and fixture if you are starting a "Job" part using the same fixture that was used before, as removing the model from the mill can cause it to be cut in a slightly different location than before.

Note: If using set line If your mill was shut down in the middle of a Job Part, not at the start or end of one, switch to the "G-Code" Run Mode click "Set Line" BEFORE clicking Start after Step 3. Type in the line of code on which this part of the job was stopped, minus 10 to back up a few steps before the interrupted line, and click "Set". NOW click Start to resume cutting this part.

Tutorial 5 : The Nudge Procedure

After attempting to find the Center of the Model along the X axis using the "Hub Calibration" method, if the calibration falls too far to the right or left making it impossible to effectively read the "Hub Calibration", use the "Nudge" procedure to manually align the model.

If after running the Hub Calibration, the position of the tic marks do not allow you to properly align the X axis on the Hub Fixture, use the Nudge procedure. In other words, if the vertical lines running down the model from A to B are shifted on the model to either the left or right hand side, too far to see where the horizontal pair of tic marks overlap (only one vertical line), you can use the Nudge procedure to manually realign the mill so the toolpath is centered over the model.

Materials Needed

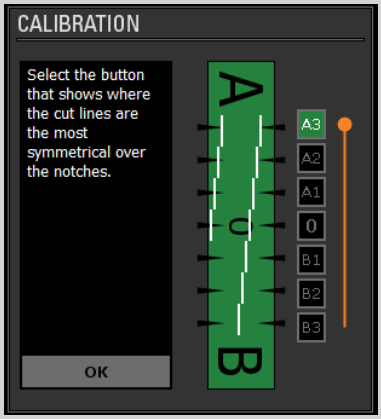
Revo Software:
- **Calibration menu > "Nudge"**

Optional but recommended:
- **Loupe or magnifying glass**
- **Flashlight**

Revo:
- **Hub Fixture**
- **A Dual + Hub wax model with Part 1 (Dual Fixture already complete)**
- **10 deg. Cutter (dark blue cap)**

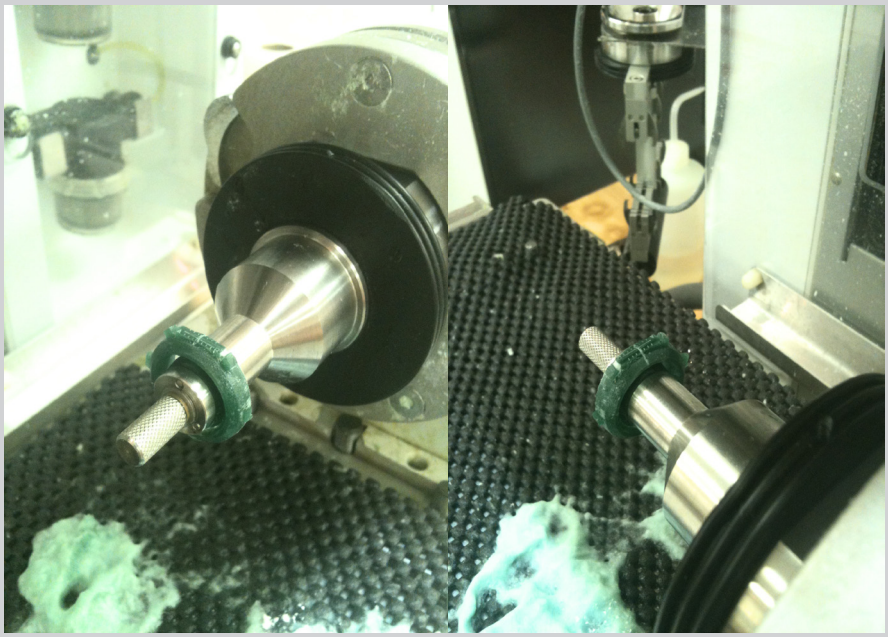
1 Click on the Diagram closest to the correct location

Inspect the base piece and choose the location on the "Hub Calibration" that is closest to being correct. This will be A3 if the vertical lines are shifted too far left or B3 if the vertical lines are too far right.



2 Run a small portion of the toolpath

The toolpath will automatically begin milling the rotary portion of the toolpath at the bottom of the ring shank. Allow the mill to pass back and forth across the ring approximately 3 times. Click "Stop". Using the Jog Controls, raise the spindle up (in the Z + direction), safely out of the way. Toggle off the spindle and Pump. After it is safe to do so, place a cap on the cutter.



3 Examine the Wax

Leave the wax on the fixture and the fixture attached to the mill and use a flashlight and loupe to closely examine the wax. Inspect both sides of the ring (left and right) where the first cut was made to see if the model has any cutter marks (gouges) on either side of the shank. If the model is scarred on one side, it means the toolpath is not centered over the model along the X axis. Determine which side of the model is scarred and approximately how deep the model is gouged.

The picture on the left shows one side of the wax without gouging. On the right, the cutter is gouging into the ring.

The Nudge Procedure cont'd.

4 Nudge From the "Calibration" drop-down menu at the top of the Revo interface select the "Nudge" option. The Nudge menu will appear with a separate jog control for the X axis. Using ONLY this control, move the X axis (work piece) left or right depending on which side of the model is scarred. If the scarring on the model occurs on the right-hand side of the model, Nudge Left. If the scarring occurs on the left-hand side of the model, Nudge Right. The distance of travel each time you click Right or Left is determined by the speed of the movement. The distance the mill is Nudged is as follows; Fast = .1 mm, Med Fast = .05mm, Med Slow = .01mm, Slow = .004mm.

5 Examine and Repeat Remove the cap from the cutter and click Start. The spindle and coolant will automatically turn on and the mill will begin where it left off. Allow the mill to pass across the ring and repeat Step 2-5 until upon examination, there are no gouges on either side of the model. The value nudged is stored internally within the software so that the next time you run a Dual plus Rotary toolpath, the mill should begin at the center of the model and the standard method for Centering the Hub using the Hub Calibration will be used.

The speed at which the jog control is set determines the incremental distance of the Nudge movement. The distance the mill is Nudged is as follows; Fast = .1 mm, Med Fast = .05mm, Med Slow = .01mm, Slow = .004mm.

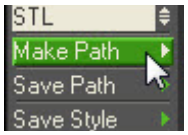
Chapter 3

Using Detailed Options

Examine parts of the toolpath one-by-one, edit various options within each part, and/or add or delete parts to enhance the outcome of your mill job.

Viewing Each Part of the Toolpath

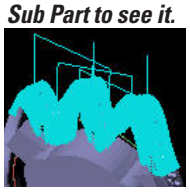
When the toolpath is completed upon clicking "Make Path", it can be a little daunting to make sense of all those colorful curves. If you'd like to see what each part of the toolpath is cutting, and check to make certain it's going to do a good job getting to every part of your model, click "Next" after making the toolpaths in the cutter selection screen to access the Detailed Options screen.



After "Make Path", Detailed Options displays each part of toolpath alone.

To use this screen for viewing parts of the toolpath, click once on each color coded "Part" to see ONLY that section of the toolpath. Certain parts of the toolpath, such as the "Flat" cuts, contain "Sub Parts" which further break down the path. Parts with more than one sub part are color coded white. Once color expanded they will display the color coded sub parts. To access the Sub Parts, double-click a white part to access each color coded sub part. Its sub-parts will all appear. Click on each of these once to view it alone.

In Detailed Options click on a Part or Sub Part to see it.



Analyzing Toolpaths

Each toolpath is calculated from the center of the cutter. Therefore, when viewing toolpaths it is imperative to take into consideration, which cutter was selected for each Part or Sub Part.

The 10 degree cutter (dark blue cap) is a tapered cutter with a tip diameter of .004". The toolpath is always calculated from the center of the cutter. In this toolpath example, the 10 degree cutter was used to create the rotary toolpath. Because the 10 degree cutter is tapered, you can see, as the cutter moves down the flat side of the ring, it actually steps away from the geometry to compensate for the taper of the cutter, leaving a 5 degree angle on the side face of the ring.

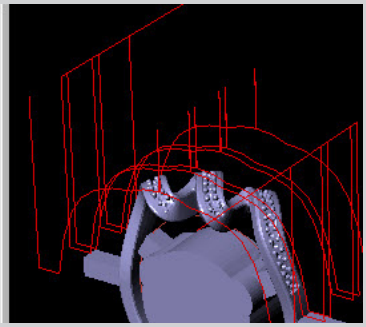
In comparison, a toolpath created using the .032" Straight cutter (black cap) is also pictured. Because this cutter is much larger in diameter, and because the toolpath is calculated from the center of the cutter, the toolpath does not drape the geometry in the same way. As you can see, as the toolpath moves vertically down the sides of the parts, the toolpath is offset from the geometry approximately. 4mm

Use Detailed Options to View Parts of Toolpaths

1 - Click on a "Part" to View it: Click each "Part" to see only that part of the toolpath in the viewports. If that "Part" Contains curves of more than one color and you wish to see each one alone, read on...

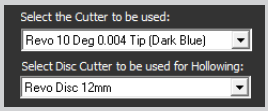
2 - Double-click a "Part" to see Sub Parts: If a "Part" is made up of more than one color, double-click it to expand it in the outline and see its component Sub-Parts. Click each one of these to see them separately.

3 - Double-click Part or Sub Part for details. Double-click a Part (with no Sub-Parts) or a Sub-Part to expand it and see each of its options. Click each one of these to view it, or read on to change an aspect of this path...

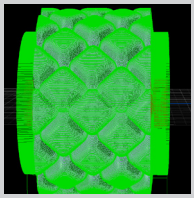


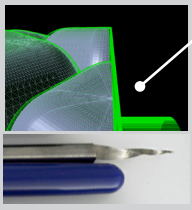
Use Detailed Options to Analyze Toolpaths

Click on each Part or Sub Part to analyze the toolpath, but remember to take into consideration the toolpath is calculated from the center of the cutter.

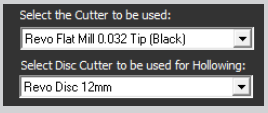


10 dgr cutter (Far view, Center; Close up, right)

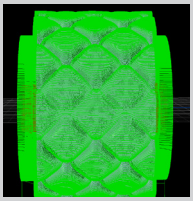


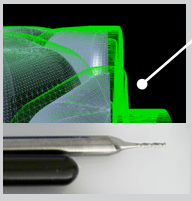


45 degree angle to allow for the angled 10 degree cutter.



032 cutter (Far view, Center; Close up, right)

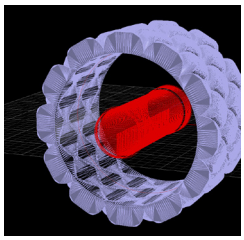




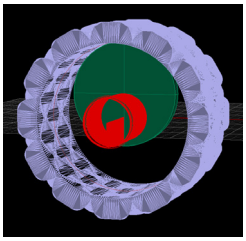
90 degree vertical with .4mm offset for straight 032 cutter.

from the part, or half the diameter of the cutter. This is most obvious on the edge of the model, where instead of moving at a diagonal away from the ring (as seen when using a tapered cutter), the toolpath moves at a 90 degree angle vertical and offset .4mm from the edge of the geometry. This is because the .032 " cutter is straight, with no taper.

To further drive this concept home, take a look at the Hollow toolpath. This toolpath was created using the 12mm Hollowing Cutter. As you can see below, the toolpath is offset approximately 6mm from the geometry. 6mm is the center of the 12mm Hollowing cutter.



12mm Hollowing Cutter Toolpath.



12mm Hollowing Cutter - offset from Center.

Editing a Part, Sub-Part

Various options are offered with each Part or Sub-Part which allow the user to change certain aspects of how that part of the toolpath is run. To view these options, double-click any Part (that has no sub-parts) or double-click any Sub Part, which causes the outline to "expand" so you can see all the aspects of this section of the toolpath only. Click on each of these elements to see what they're currently set to in the space beneath the menu. You can change them by using the menu controls offered in this space. Each of these Detailed Option controls is explained in full in this chapter.

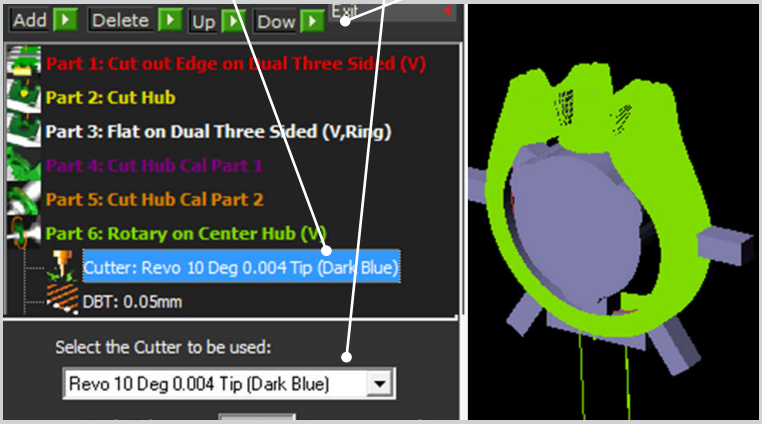
For instance, you may change: the cutter used for that portion of the toolpath, the angle at which the cutter approaches that portion of the toolpath, the depth to which the cutter sinks when cutting that portion of the toolpath, the "Distance Between Toolpaths" for that part of the cut, and even the size and shape of the area cut during that section of the toolpath (known as a "Border").

Use Detailed Options to Edit Parts of Toolpaths

1 - Click an option in a Part or Sub Part Click an option listed beneath a Part or Sub-Part and it will open beneath the Detailed Options menu so you can view and change its current settings.

2 - Use that option's menu to change it. Each option has a slightly different menu set-up which the user can access to change ONLY that part of the toolpath. These are explained in full in this chapter.

3 - Add or Move Toolpaths. There are also options to add and move toolpaths to complete difficult models. ONLY DO SO AT YOUR OWN RISK once you thoroughly understand these concepts.

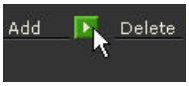


Topics Covered in this Chapter:

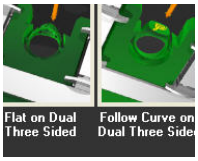
Adding & Moving Parts of Toolpaths

Explained last in this chapter, certain commands in this menu allow you to Add or Move parts of toolpaths for models that are especially difficult to mill.

The "Add" option opens an additional menu which allows the user to select which type of path to add. New toolpaths you'll learn about in this chapter - "Follow Curve" on Flat or Rotary - are offered here, along with the ability to add another pass of the types available with this strategy. Depending on which basic strategy you chose (Two sides on Dual, Three sides on Dual, Four or Eight sides on Base Clamp, Rotary pass on Three-Jaw Chuck, etc.), you will be offered different options in this menu to enhance your project.



Click "Add" for menu of more toolpaths to add to a job.



Options to "Add" vary based on fixture in use.

Another reason you may wish to add a part to a toolpath or edit an existing part is the ability to add a finishing pass to a part. A finishing pass is added after the wax has been mostly removed but further detail is required. This second, pass is run using the flat cutters 0.032", 10 -degree 0.016", or the 6-degree tapered cutter to reach additional details. It can applied to the entire model or JUST to a section of the model that requires further touch up. Limit where it cuts using the "Border" option.

You could use a flat cutter because they can mill truly straight sides on a model with lettering, straight-sided channels, bezels, etc.; while a tapered cutter, due to its shape, creates slightly-tapered sides. This taper may or may not be readily apparent on your model, depending on its design.

Choose the 0.032 " flat cutter only if the details are farther apart than this value. This cutter is the ONLY flat cutter supplied at Gemvision that can cut from the rough. If the details are closer than this, add a secondary pass with the 0.016" flat cutter. Or, if a miniscule taper won't affect your design & the details are even closer together than the 10-degree cutter can achieve, choose the 6-degree tapered cutter (not included with the mill). Because of its 3-degree taper on each side, vs. 5-degrees each for the 10-degree cutter, it can reach details that are closer together.

Each Part of the Detailed Options Menu

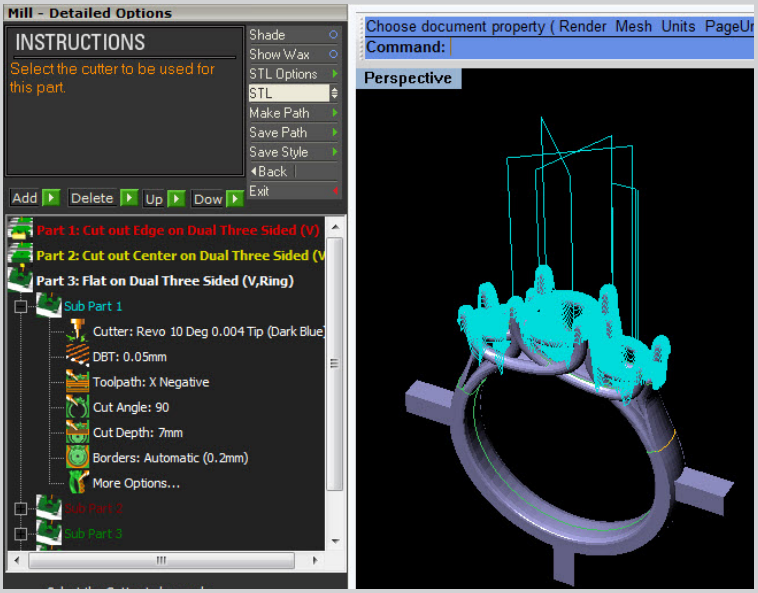
(Upper Left) As in other menus in Mill Builder, follow the Instructions to use the menu.

(Upper Right) Identical options as in the previous menu - for Shade, change STL Options, and "Make Path" / "Save Path" - are offered here.

(Main "Outline") Click each Part of outline to view associated toolpath in viewports, or double-click to expand and access Detailed Options.

(Below Main Outline) Space below "outline" displays and lets user edit detailed options.

(Above Main Outline) Add, Delete, and Up/ Down options are explained in this chapter.



Using Detailed Options...

Overview of Parts & Sub Parts

Learn the basic "Parts" and "Sub Parts" of each strategy. These can easily be edited or duplicated - with different options, such as changing a cutter - during Detailed Options.

Identifying Each Part & Sub Part

It is important to be able to recognize and identify each Part and Sub Part of a toolpath, so that you understand what the mill is doing during that section of the job. That way, you'll know how (and if) you wish to edit any aspects of that part of the toolpath in the Detailed Options screen. Also, when you "Add" a part to a toolpath, each of these parts are offered as options. Become familiar with them so you know when to add them.

A Summary of Each Type of Cut

Each strategy is really just a certain combination of the types of cuts of which the Revo C is capable. In Detailed Options, you are able to define certain aspects of each cut, and even ADD compatible cuts to the strategy you've selected to enhance the outcome of the model you're creating. Following is a short summary of each type of cut. This chapter further expands upon and illustrates each type.

Flat on Dual 3-Sided This cut might contain 2, 3, or 5 sub-parts depending on what kind of job you're running on the fixture: a 2-sided job (front and back of part - or, 180 and 0 degrees); a 3-sided job (top, front, and back of part - or, 90, 180, and 0 degrees); or a 5-sided job (top, front, back, and two angles - or, 90, 180, 0, 45, and 135 degrees). The Detailed Options screen allows you to edit ANY of these sub parts (cutter, DBT, depth or angle of cut) and ADD sub parts in this category with any of the above options defined.

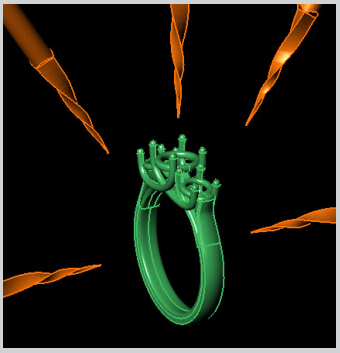
Rotary Cut This cut is run when the 3-jaw chuck or Hub center option is selected. It includes a default value for the "Border Distance" (or toolpath overcut) indicating the distance the cut will continue past the outside of the model, to leave the support material attaching it to the tube wax. It also includes a default Depth past the inside of the finger rail that it cuts in order to create azures, etc. This is called "Cut into Ring Rail" Either of these values may be edited.

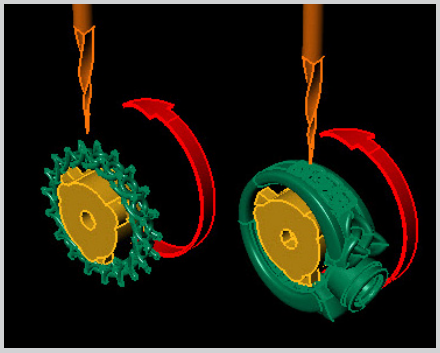
Whenever one of the two rotary fixtures (3-

Each Type of Cut on the Revo 540C

Flat on Dual 3-Sided (L, below) can be cut on the 90-degree (top-down) for a 3- or 5-sided cut; 180 and 0 degrees (2, 3, or 5-sided cut), or 45 and 135 (for a 5-sided cut). Edit cutter, DBT, Border, cut angle and depth.

Rotary Cut (Center & R) is available on the 3-Jaw Chuck and Hub Fixture. Remember that the supports are milled off when the Hub is added to a flat part. Edit cutter, DBT, border, and depth.



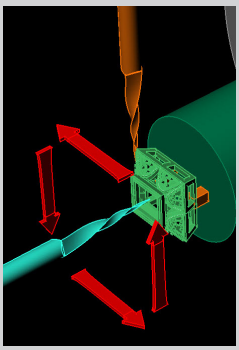


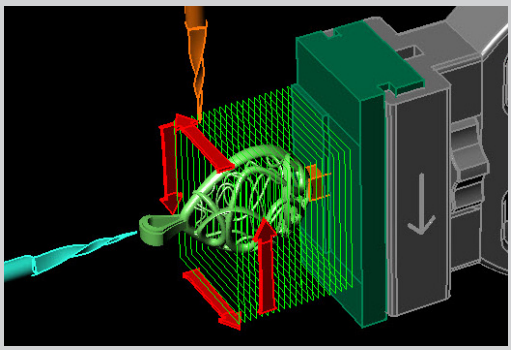
Each Type of Cut on the Revo 540C

Flat on Rotary Cut (V) takes place during the Four Flats on Rotary or Four Flats on Rotary Plus One cuts: the A axis rotates the model to a defined angle; then holds it flat while the Vertical cutter completes a pass.

Flat on Base Clamp (V) takes place when the A axis holds the model flat in 4- or 8-positions on the Base Clamp so the Vertical cutter can make its flat pass. Cutter, DBT, Cut Angle and Depth, and Border may all be edited.

Flat on Base Clamp or Rotary (H) is available during the Flat on Rotary Plus One, Rotary Plus One, or Base Clamp strategies. It takes place when the Horizontal cutter approaches for its top-down cut. ONLY cutter, cut depth, DBT, and border may be changed; NOT cut angle.





(L - R) Flat on Rotary Plus One; Base Clamp. Blue cutter indicates Horizontal cutter; red cutter indicates vertical cutter.

Jaw Chuck or Hub) is used, a rotary cut may be added to the path.

Flat on Rotary This is the cut used during the "Four Flat on Rotary", and "Flat on Rotary Plus One", in which the A axis rotates the model to a set angle and holds it flat there while it is being cut. In the case of these two strategies, the angles are 0, 90, 180, and 270 degrees. You may add a Flat on Rotary cut to ANY job that uses one of the two rotary fixtures (3-Jaw Chuck or Hub). The angle at which the A axis holds the fixture may be edited, as may the cutter depth and border (space within which the model is cut).

Flat on Base Clamp (V) This cut is made with the vertical cutter, and it is made when the A axis rotates the base clamp model to a flat angle, where it holds it while cutting a 1-mm wide piece of the model (0, 90, 180, and 270 degrees for the 4-sided cut and 0, 45, 90, 135, 180, 225, 270, and 315 for the 8-sided cut). These angles, the depth to which the cutter will plunge, and the cutter and DBT may all be edited; and, a Flat on Base Clamp cut may be added to any base clamp model.

Flat on Base Clamp (H) and Flat on Rotary (H) These are the cuts made by the Horizontal cutter to complete the top-down cut for a model held on the Base Clamp or 3-Jaw Chuck (for a Flat on Rotary Plus One or Rotary Plus One cut). User can control cutter, DBT, depth, and border of this cut ONLY (not angle).

Cut-Out Curves

The cut-out curves for each type of cut are unique in that the order cannot be edited: in order to work properly, they MUST take place in the default order. The cut out curve used to remove a large portion of excess wax in a short amount of time.

Only the Cutter and Offset may be edited in certain cut-out shapes. "Offset" indicates how far past the extents of the model the cut-out curve is drawn. The default is 2-mm past the edges of the part (3-sided and Base Clamp) and 2-mm past the inside of the part (Cut Out curve for center cut or hollowing cut).

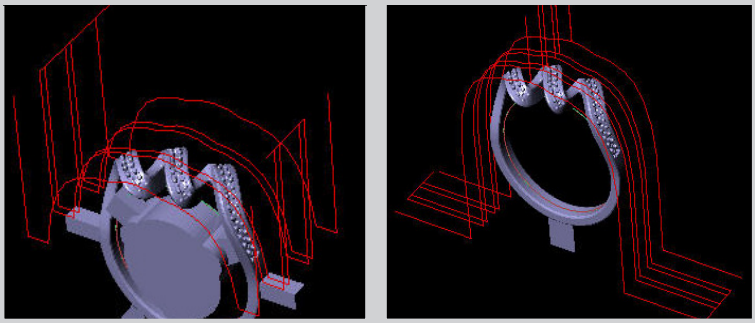
Examples of cut-out curves include the cut-out shape on any Base Clamp strategy and the Three-Sided strategy on the Dual Fixture, both of which are run in order to create a cut-out piece of wax you can remove so that less uncut wax needs to be cut when the model is being milled from the top down.

Other cut-out curves include the Center cut for the model, which removes the inside piece so it can be removed by the user and the entire inside of the model does not need to be milled away in a time-consuming process by the cutter during the Front and Back passes. This is also necessary to clear away wax for the Hollowing cutter, another cut which cannot be edited in its position in the outline.

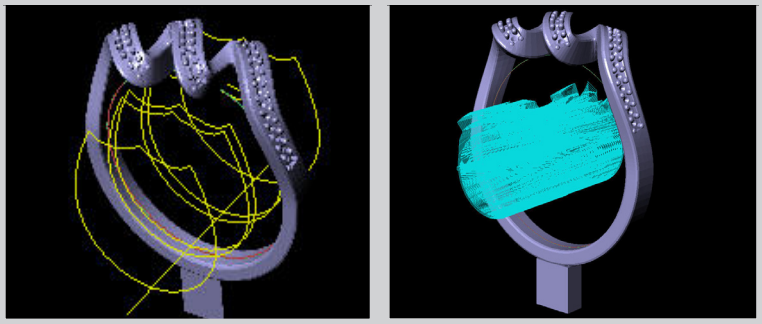
Finally, the Hub fixture contains three cut-out operations: one for milling out the arrow, pin-holes and center (screw) hole in the hub; one at the end of the flat milling process to create the first part of the Hub Cal cut; and one at the beginning of the rotary section of the cut to create the second part of the Hub Cal cut.

Cut-Out Curves for each strategy:

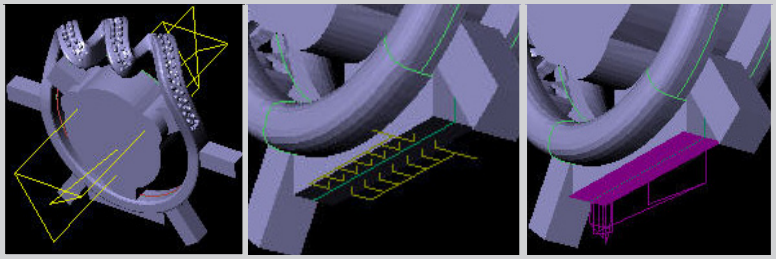
Three-Sided on Dual & Base Clamp: A cut-out curve is automatically created when either of these strategies are chosen. It is created by offsetting a curve in the shape of the part 2-mm away from the top of the part (or, the value you change this to in the "Offset" Option for this pass). Notice that these cuts only plunge down a few mm at a time, follow the assigned curve, and then plunge a few mm deeper than that to make the next cut. This continues to a depth of 1-mm past center on both sides of the model so the cut-away piece can easily be removed by the user. Remember that this cut is made so that the top-down cut on each of these strategies does not have to plunge through so much uncut wax.



Above: Cut-out Curves for 3-Sided on Dual (L) & Base Clamp (R); Offset = 2 mm from model; number of passes represented by multiple curves.



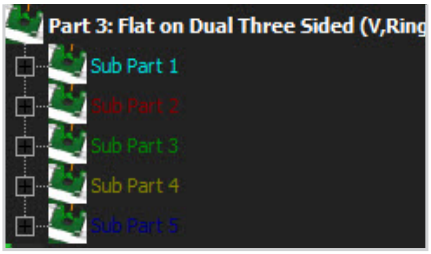
Above: Cut-out curves for Center & Hollowing; Center (L) is offset 2-mm. Hollowing (R) offset is automatic (size/shape of tool defines it).

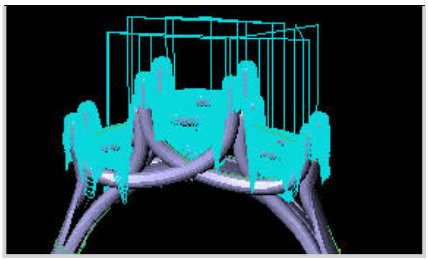


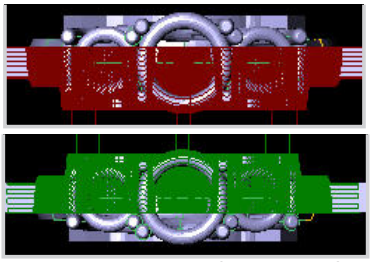
Above: Hub Cut-out Processes (L - R) are (1) Cut-Out Pin Holes, Screw Hole, & Arrow; (2) Hub Cal Part 1 (on Flat fixture); (3) Hub Cal Part 2 (on Rotary fixture).

At-A-Glance: Parts & Sub Parts for Each Strategy

Learn to identify and evaluate the Parts and Sub Parts of each strategy.



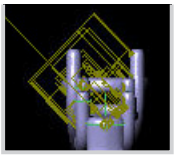


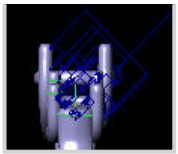


(Clockwise, from above) Sub Parts 1 - 5 in a five-sided job. Parts 2 & 3 (above, right) are the only parts in a 2-sided job; Parts 1 - 3 (above) are the only parts in a 3-sided job.

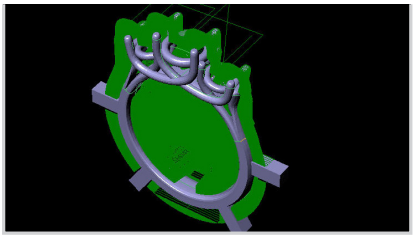
Flat on Dual Three-Sided This "part" is comprised of between two and five sub-parts in the Detailed Options "outline". Double-click it to see its subparts.

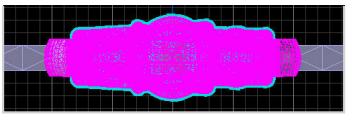
Two, Three, or Five Sub-Parts: When a two-sided cut is being run, you'll see just the two main cuts - "front" and "back". When a three-sided cut is selected you will see three sub-parts, adding the "top" cut before the two original cuts (front and back). Finally, two additional angles will be added if the "five-sides" custom angles were selected earlier on in the builder. Remember: These toolpaths ONLY mill what was missed by the other flat cuts.





Parts 4 - 5 (above), milled on 45-degree angles, cut only what was missed & appear only during a 5-sided job.



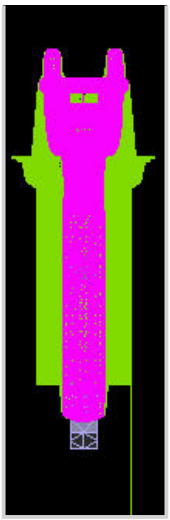


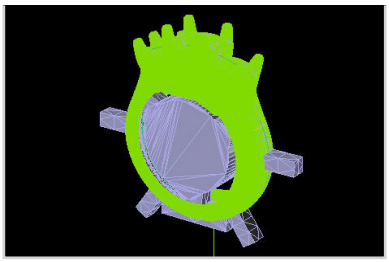
(Left) The border of the toolpath defaults to 2 mm past the model for front and back cuts; and (Above) 0.2 mm past outside of part for 90-, 45- and 135-degree cuts.

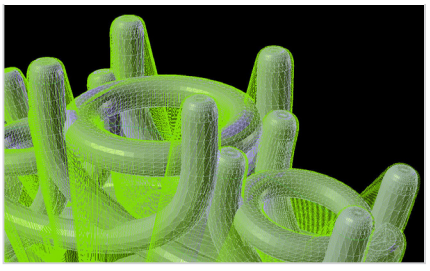
About Each Depth & Angle: Each of the two main flat cuts (front and back) are cut to a depth of 1 mm past the center of the model, as visible from the Looking Down (above) or Side View viewports. The top-down cut is cut to a depth of 1 mm inside the finger rail from the top-down. (You may wish to edit this option, as described in a future part of this chapter, if this depth does not cut all necessary details.) Also, the two additional angled cuts made when "5" custom angles are selected are each angled at 45 degrees. (If the angle or depth of these cuts is insufficient to mill all details, this is another aspect of the toolpath you may wish to edit).

About Each Border: The border inside which the "Front" and "Back" are cut extends to 2 mm past the outside of the part. This 2 mm gives the user some space to cut the model away from the supporting wax. If "Cut Center" is selected, the finger rail is also offset 2 mm to the inside and the space between these two borders is all that is cut. If this is too large or small, Detailed Options allows you to change it. For the 90- and 45-degree cuts, the border extends to just 0.2 mm past the outside of the part so it is as close as possible to the shape of the part and no extra time is spent cutting air.

Rotary The rotary part of a toolpath for either the 3-jaw chuck fixture or the hub fixture follows the outside of a model (model highlighted at right) at a width of 2 mm past the edges of the model. This space is known as the "Border Distance" so that a small amount of wax will be left to hold the model to the wax tube (shown in green toolpath at right).





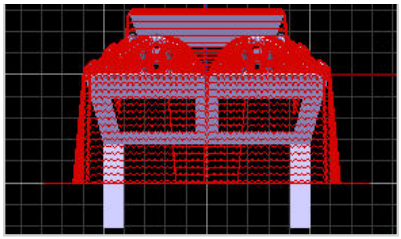


When Added to a Flat Cut: (Above) Remember that all supports are milled off during the rotary portion when a rotary cut on the Hub fixture is added to a flat cut.

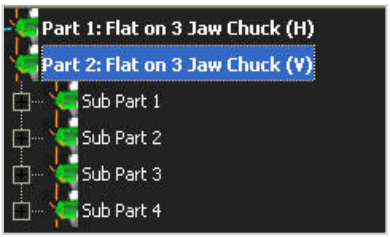
More about This Cut: (Above, Right) It is especially noticeable on the rotary that the cut tapers outward. This is because the 10 dgr cutter is tapered, and because the model is held perpendicular to the cutter - rather than the detail - during cutting. As described later in this chapter, you may use additional Detailed Options to fix this if it is compromises the design.

At-A-Glance: Parts & Sub Parts for Each Strategy

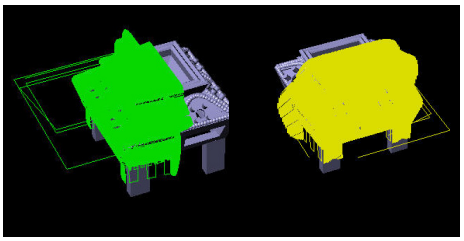
Learn to identify and evaluate the Parts and Sub Parts of each strategy.



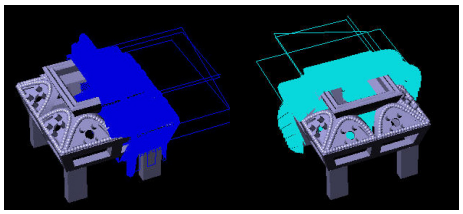
Flat on 3-Jaw Chuck (H) This cut is the initial cut made by the Horizontal cutter at the start of both the Rotary Plus One and Flat on Rotary Plus one toolpaths. This pass cuts down to 2 mm above the base of the model, so be sure and make supports long enough to accommodate it.

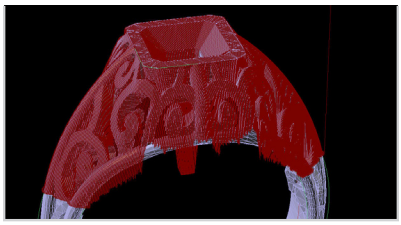


Flat on 3-Jaw Chuck (V) This is the cut performed during the Four Flats on Rotary and Flat on Rotary Plus One cuts. Double-click it to expand it into its four component parts. It holds the model flat on the rotary fixture, cuts it at a depth 1 mm past center on all four sides, with a border 0.2 mm beyond the part; rotates it 90 degrees, and repeats the cut until all 4 sides are complete.



(Above & Below) Sub Parts 1 - 4 for a "Flat on 3 Jaw Chuck" Part. Model is rotated 90 degrees and cut flat each time, for a total of four sides.

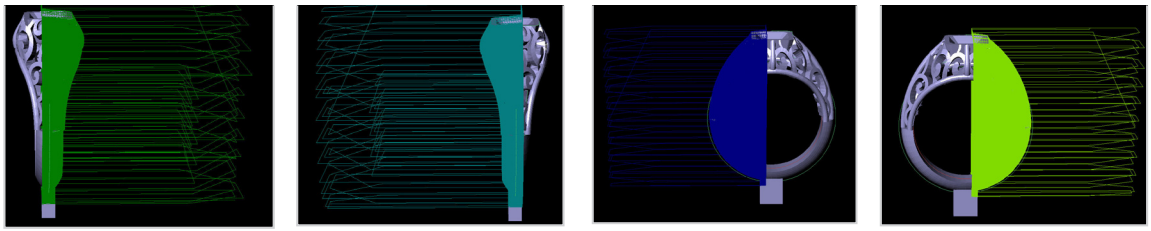




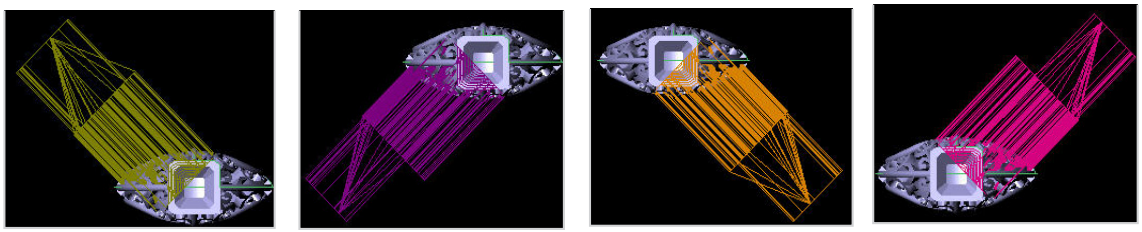
Flat on Base Clamp (H) This is the "top-down" cut on the base clamp, which is made with the Horizontal (H) cutter. It is completed to a set depth inside the finger rail, within a border of only 0.2 mm outside the extents of the part.

Flat on Base Clamp (V) This cut may be expanded to view each of its component cuts: four main cuts - front, back, left, right, and (if selected) four custom angle cuts. Double-click this "Part" to see the expanded list of sub-parts. You will see either four or eight parts. With the four additional angles selected, you will see a total of 8 sub-parts. Each of the sub-parts is illustrated below. Even so, remember the order in which these parts are cut: The mill cuts only ONE millimeter of the first toolpath; rotates the mill 90-degrees (for four sides) or 45-degrees (for eight sides) and cuts the next 1 mm of the part; rotates the mill and cuts the next 1 mm, and so on. So even though each toolpath curve looks like it's completed separately, don't be fooled. If each 45-degree angle does not get to the details you need, use the options "Cut Angle & Depth".

About Each Cut: Each Border of the cuts are identical to the top-down cut: 0.2 mm past the edges of the part. The Front and Back cuts are made at a depth of 1 mm past the center of the model, but the Left and Right cuts are made to 1 mm BEFORE the center of the model, so just keep this in mind. And of course, the additional 4 cuts are made on the 45. If any of these angles or depths don't reach the details you need, read on in this chapter to learn how to fix them.



(Above) The four main cuts in a four-sided base clamp cut: (L - R) Front, Back, Left and Right (not cut in this order: see above).
(Below) The additional four angles added when "8-sides" is selected in the "Custom Angles". Each is at a 45-degree angle to the part.



At a Glance: Which changes can you make to each Part or Sub Part?

Of all the "Default" toolpaths that combine to make a job, there are only certain Detailed Options that may be changed, based on the type of toolpath being run. Below, find a list of the different types of cuts and which Detailed Options you will be able to edit for each.

	Change Cutter	Offset	DBT	Cut Angle	Cut Depth	Borders	Cut New Only	Roughing
Cut out Edge	Y	Y	N	N	N	N	N	N
Cut out Center	Y	Y	N	N	N	N	N	N
Hollowing Cut	Y	N	N	N	N	N	N	N
Cut out Hub	Y	N	N	N	N	N	N	N
Hub Cal 1	Y	N	N	N	N	N	N	N
Hub Cal 2	Y	N	N	N	N	N	N	N
Flat on Base Clamp (H)	Y	N	Y	N	Y	Y	Y	Y
Flat on Base Clamp (V)	Y	N	Y	Y	Y	Y	Y	Y
Flat on Dual 3-Sided	Y	N	Y	Y	Y	Y	Y	Y
Flat on Rotary (H)	Y	N	Y	N	Y	Y	Y	Y
Flat on Rotary (V)	Y	N	Y	Y	Y	Y	Y	Y
Rotary Cut	Y	N	Y	Y	N	N	N	N

Options Available with Cut Out & Hub Cuts

Cut Out Edge cuts and Hub cuts WILL allow you to change the cutters, but it's NOT recommended to do so. Each of these processes is programmed to be completed with the 10-dgr cutter, and if they are changed to a different cutter, they might not produce the expected results. However, the option remains in order to minimize the amount of time you must intervene with the mill. Trial and error will determine whether or not you can live with the results, should the change be made.

Offset

The "Offset" value associated with these cuts indicates the distance from the extents of the model that the cut is offset, so it will not intrude upon the details of the model and yet will take away sufficient wax so that the main cut does not have to plunge too deep when cutting the model.

Distance Between Toolpaths

The detail on the model (coarse to fine, same as the settings in the Cutters & DBT menu) may be controlled individually for each part of a model. If you know there is a section of the model that won't require detail as fine as other parts, you may adjust this, but ONLY for the Parts and Sub Parts that cut model: not for those that cut the Edge Cuts, Hollowing, or Hub Cal cuts. Again: these types of cuts are pre-programmed to be created in a certain way, and we don't want to change that and risk unexpected results.

Cut Angle And Depth

Note that these can only be changed for cuts made with the VERTICAL cutter. The A axis can rotate the fixture to a flat angle, and hold it flat while the cutters complete the assigned cut. However, the Horizontal cutter can ONLY come in at one angle: the top-down, to complete its cut.

Borders

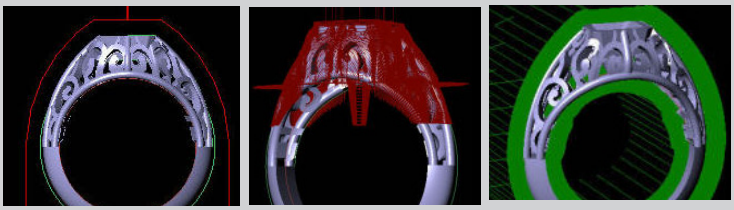
The distance past the extents of the model that each path will be cut is defined in this option. "Custom Borders", or, a closed curve the user draws to define this area, may also be input wherever the Borders option is available. This option is thoroughly explained later in this chapter.

Cut New Only

You'll recall how the 8-sided cut on the Base Clamp and 5-sided cut on the Dual fixture have the ability to ONLY cut the parts that were missed by previous cuts. This option is known as "Cut New Only", and you will be given the option to turn it on and off with each of these cuts.

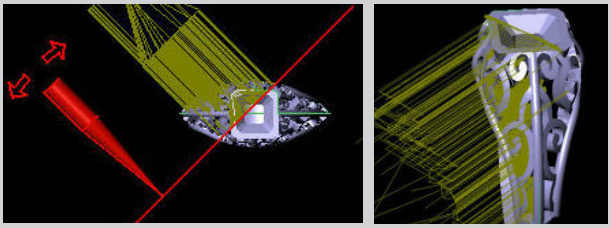
Roughing

The toolpath can account for the depth of the cutter and protect it from sinking too deep with each pass by creating Roughing passes automatically. You have the option to turn these on and off with each type of cut listed above.



(Top Row, Left) *Offset Value of 2 mm is the default for Cut-Out Edge and Cut-Out Center cuts.* ***(Center)*** *Roughing pass is visible as "ledge". It is an initial cut to protect cutter from sinking too deep into uncut wax.* ***(Right)*** *Border for flat cuts defaults to 2 mm past extents of parts; for Rotary, default is 0.2mm.*

(Bottom Row: Left) *Cut Depth & Angle defaults to a given value with each flat cut. Edit this easily by dragging a representative of the cutter & its depth, which is placed on screen when accessing this option (explained later in this chapter).* ***(Right)*** *"Cut New Only" is evident in 8-sides on Base Clamp or 5-sides on Dual: additional flat cuts ONLY cut what original flat cuts missed.*



Using Detailed Options...

Revo Options

The user can define the default settings for many aspects of the Toolpath including Mesh Setting, DBT, Border Offsets and Toolpath Depth.

Revo Options

The mill software always uses certain default settings when creating toolpaths in the Mill Builder software. And although a user can change these settings within each part or sub part of the Detailed Option section of the mill builder software, you may prefer the Mill Builder software always use your preferred settings. If so, on the first screen of the Mill Builder software, change these settings within the Revo Options menu. This menu allows you to choose the default settings for Mesh Settings, DBT, Border Offsets and Toolpath Depth.

Revo Options is located within the Toolpath Builder

Mesh Settings

When a toolpath is created the software must first analyze the geometry to determine the path it will take to mill the part. However, the toolpath does not understand Nurbs geometry and so when you create a toolpath, a few important things happen all at once, one of which, a Mesh is created from the Nurbs model. This Mesh is a series of polygons that attempt to define the surface of the Nurbs model from which the toolpath is calculated. Therefore the quality of this Mesh will determine the quality of the finished wax.

There are a variety of different settings that can be changed within the Mesh Settings menu which control the quality of the Mesh, however, for our purposes, we will be focusing on the setting Maximum Distance Edge to Surface. Therefore, for the following setting apply zero; Maximum Angle, Maximum Aspect Ratio, Minimum Edge Length, Maximum Edge Length, Minimum Initial Grid Quad, and Mesh Density. The only value we will assign is the Maximum Dist edge to srf. This will allow us to make changes to this setting and analyze only those changes. Assign a value of .002. This value determines the maximum distance the furthest edge of a polygon can vary from the true nurbs surface. A value of .002 will ensure the Mesh will be very similar to the original nurbs model. So why not make the value really small so the mesh is really fine? Well, it is important to take into consideration this settings will determine the overall file size and therefore can cause the mill and your computer to strain as they attempt to process the file. The finer the surface the larger the file size. Setting this value to .002, will be sufficient for most any instance without overly taxing the computers resources. After making any changes to this menu, Click Set to commit those changes.

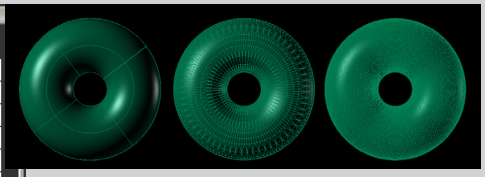
Each setting within this menu will help determine the quality of the mesh. And in an ideal world, you would take the time to mesh each individual part of the model, large flat items would be set to a higher value, while small curvy details would be set to a lower value. Each item would have a specific ideal set of values. If you would like to experiment with Mesh settings, do so within the Matrix interface and use the Mesh from Nurbs Object function located in the Utilities menu. Within the Polygon Mesh Options menu select Detailed Controls. Here you may easily make changes and view those changes on screen. Find a good default value for the Maximum Dist edge to srf, so you can eliminate the amount of time it would take to find the ideal setting for each and every part. Instead find a default setting which will work for any model you put to the test.

Comparing Maximum Distance Edge to Surface

1: Mesh from Nurbs
Within the Utilities menu in Matrix, choose Mesh from Nurbs Objects and enable the Detail Option menu to experiment with the quality of Mesh. Experiment with the Maximum Distance edge to srf setting.

2: Revo Options Menu
Within the first screen of the Toolpath Builder, click Revo Options to open the Mesh Setting Menu. Specify the default settings for the Mesh Settings, DBT, Tool Depth, and Border Offset.

3: Apply New Settings
If you determined a new preferred value for the Maximum Dist edge to srf setting, apply it now, otherwise, apply .002 which will work well for almost any model. Click Set to commit this change.



Pictured above are three torus. On the left see the Nurbs model. In the middle and right are meshes with different Maximum Dist edge to srf settings.; 1 and .2 respectively. The larger the value the more faceting that occurs. We recommend a value of .002 for this setting.

DBT, Tool Depth, and Border Offset

The Distance Between Toolpath Setting, or DBT determines the step over distance the mill will take as it passes back and forth across the model. The Default Distance Between Toolpaths setting is .04mm. In other words, as the mill approaches the edge and turn to come back the distance of the turn is .04mm. This means, no matter which cutter you are using, the DBT or step over between each pass will be set to .04mm. Which is why milling an item with the 10 degree cutter, which has a very tiny tip diameter and milling an item with the 32 thousandths cutter which is much larger, takes approximately the same amount of time. They both are covering the exact same distance to mill the item. This default setting works well for all of the Gemvision supplied cutters, however, you may find another setting you prefer to use depending on which cutter is selected because this will greatly effect the amount of time it takes to complete a toolpath and the quality of the finished part. The tighter the step over, the longer the mill will take to complete the job, but the finer the finish. Each cutter has a different tip size and therefore you can increase the DBT based on the cutter size. Experiment with this setting to find what works best for each of the cutters.

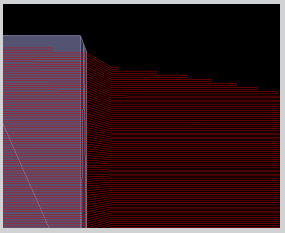
When the software creates a toolpath it is based on the are where the mill will cut and the depth the cutter will plunge. The software first analyzes the geometry and finds the center line of the model. It then sets the depth of the cutter based on this center line. The default setting for the Tool Depth is set to -1mm. This means when the tool approaches each part, it finds the center of the part and mills 1mm past the center line. For example, on the Dual fixture, the software will analyze the part and find the center of the model. The depth of cut will pass the center line by 1mm for each side, Front and Back. In the event the model has detail that can not be reached with this default setting, proceed to the Detailed Option section and manually change the mill depth, or instead, make the change to the Revo Options section of the builder and every toolpath will be calculated with this new setting. Keep in mind, the deeper the cut, the longer it will take to mill the item. In addition, the deeper the plunge, the more likely you are to add an automated roughing passes. The 10 degree cutter has been defined to allowed a 10mm deep cut before adding an automated roughing pass.

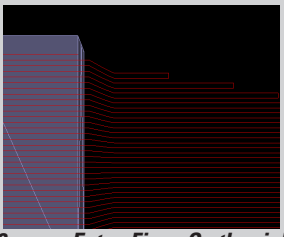
The default setting for the Border Offset is 2mm. This means the area or border where the mill will cut on any flat strategy is 2mm offset from the geometry. The larger the border, the longer it will take to mill. The tighter the border, the harder it will be to cut the model away from the support wax that is holding it. If you decide to change the Border Offset, keep in mind the supports you will be using need to extend past the border curve so the item you are milling is properly supported. If the border curve encompasses the support structure, the item will fall out of the wax in the middle of the milling process.

Defaults for DBT, Tool Depth, & Border Distance

1 : Distance Between Toolpath

The DBT determines the step over distance as the cutter mills back and forth across the surface of the model. This setting will determine the surface quality of the model as well as the time it take to mill the item.

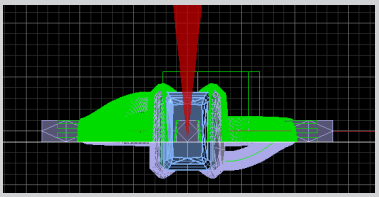


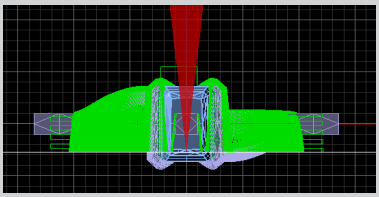


Pictured above left, the DBT is set to .02mm or Extra-Fine. On the right, the DBT is set to .05.

2 : Tool Depth

The DBT determines the step over distance as the cutter mills back and forth across the surface of the model. This setting will determine the surface quality of the model as well as the time it take to mill the item.

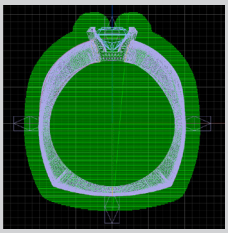


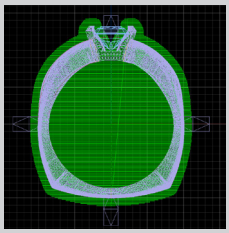


Pictured above left, the cutter at its default setting mills to a depth 1mm past the center line of the model. On the right, the mill depth has been adjusted in the Detailed Option section so the cutter will reach the inside of the shank on the right hand side of the model. On the left, the mill will not reach this part of the shank and a large portion of wax will remain on the model.

3 : Border Distance.

Set the Border Distance to determine the area in that will be milled. The software analyzes the geometry and creates a border that is offset from the edge of the model. The default setting for this offset is 2mm.





Pictured far left, the border is set to 2mm. Left, the border is set to 1mm. Make certain to leave enough room so a tool can be used to free the wax from the block which is supporting it.

Editing Parts & Sub Parts ...

Changing Cutters

Learn how and when to change the cutter for a certain Part or Sub Part.

Changing Cutters

In every Part and Sub Part, you have the ability to change cutters... however, you shouldn't change cutters for all of them. For instance, the Cut Out Curves (Cut Out Edge, Cut Out Center, Cut Out Supports, Hub Cal Parts 1 & 2) are all designed to be performed with the 10-degree cutter. If you change from this, you may get unexpected results. However, the ability to change cutters is contained in all of these options: that way, you can decide if the results are acceptable and, if they are, use the ability to change cutters to your advantage.

Having one cutter throughout the whole job eliminates the need for the mill to stop and prompt you to change cutters. This can work to your advantage, since you won't need to "babysit" the mill when running certain jobs: simply set it up and walk away.

Considerations when Changing Cutters: Why would you want to change cutters, anyway? Well, one good reason is when you wish to place straight sides on a part. The 10-degree cutter will add a slight taper (5-degrees on each side) on straight-sided models, which might compromise your design when creating lettering or prongs, straight-walled channels, straight-sided bezels, etc.

One way to remedy this is to change all the Parts and Sub Parts to the 0.032" straight cutter, and cut the part with a straight-sided cutter right out of the rough, uncut wax. Just remember: the 0.032" cutter is the ONLY other cutter, besides the 10-degree tapered cutter, that can cut rough wax.

Finishing Passes: Another way to remedy slightly-tapered sides on a model, and finish them off with straight sides, is to "Add" a toolpath, set up its options to re-create the toolpath(s) previously cut with the 10-degree cutter (a process explained later in this chapter), and choose for the new path(s) the 0.032" straight, 0.016" straight, or 6-degree tapered cutters.

Choosing New Cutters for Parts & Sub Parts

Choosing A Cutter Since the 10-degree tapered cutter is the default cutter for all jobs except for hollowing cuts, you'll want to be careful when changing it. In Detailed Options, you may select the "Cutter" option when the Part or Sub Part is expanded, and use the drop-down menu that appears at the bottom of the menu to select the new cutter for a particular part or sub part.

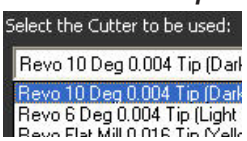
0.032" Flat is the ONLY OTHER CUTTER which can cut rough (uncut) wax. Since its tip size is much bigger (0.032 " vs. 10-dgr's 0.004"), ONLY select it as the cutter for an EXISTING part or sub-part when your details are farther apart than this distance.

0.016" Flat can ONLY work as a finishing cutter, which is used to re-cut a model with tapered sides so they will be straight. It CANNOT cut uncut wax. It can reach smaller spaces than the 0.032" cutter, at half its size, though it is still larger than the tip of the 10-degree cutter. ONLY select it when adding another pass to re-cut a part that's already been cut (explained later in this chapter).

6-degree tapered Is smaller still than the 10-degree cutter, with a tip of 0.003", and can therefore ONLY work as a finishing cutter to get into smaller areas than any other. It has a very slight taper at 3 degrees on each side (for a total of 6 degrees). Select it ONLY as a finishing cutter, when "Adding" another toolpath to re-cut the model.

(L - R) 6-degree tapered, 0.032" Flat, 10-dgr tapered, 0.016" Flat, 12-mm hollowing.

To change cutters, double-click the Part or Sub Part to expand it and choose the Cutter option.



Choose a new cutter from the dropdown that appears under the menu.

0.016" and 6-degree are finishing cutters ONLY, meaning they cannot cut rough (uncut) wax, and can only be used in this fashion, to "finish" or cut additional details on a model that was ALREADY cut with the 0.032" or 10-degree cutters. Remember: ONLY the 0.032" straight and 10-degree tapered cutters can cut rough wax.

Also note when you are changing cutters: each time you change a cutter in a Part, the mill software will need to stop and prompt you to switch to the new cutter. Therefore you may wish to think about this strategically, wherever possible, and put the same cutter on as many parts "in a row" as you can. That way, you'll spend less time trudging back and forth to the mill to change cutters.

Adding Custom Cutters: The Cutter Library

The location in the mill software to add custom cutters - or, cutters which can be purchased from sources other than Gemvision - is NOT found in the Detailed Options, even though we've saved it to talk about till now, since it is a rather advanced feature. The "Cutters" button is found in the first screen of the Mill Builder. You may click it at any time to add a new cutter which you may have purchased from sources other than Gemvision.

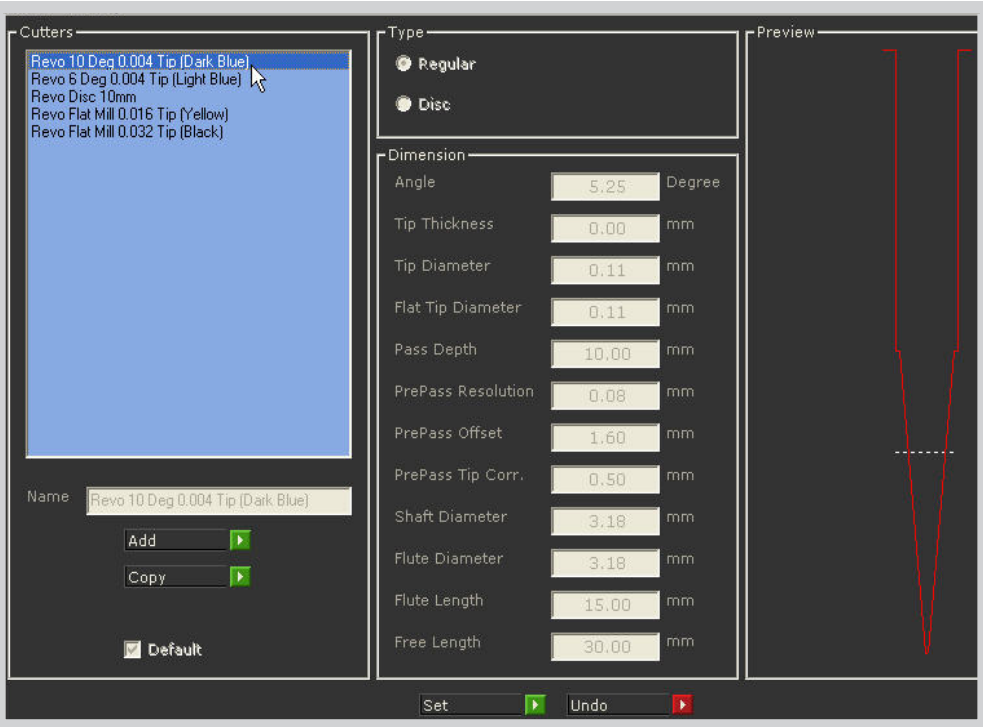
Remember how important it is to change cutters in the software if you are going to change them on the mill: it is JUST AS IMPORTANT to add any custom cutters that aren't in the software, using the method explained here. This is because the mill accounts for the cutter size, shape, and cut length (how deep the cutter can cut without breaking) when it is creating the toolpath, to ensure that the cutter won't gouge the part or be broken by making a pass that is too deep for it to perform.

Existing Cutters: Click an existing cutter to see its definition. You CANNOT edit an existing cutter: this prevents you from doing something you can't undo. This menu displays the name of the cutter & whether or not it is the default (left-hand column: 10 degree is the default cutter), the shape of the cutter (far right), as indicated by the cutter definitions in the center column.

Cutter Definitions: The center column contains the cutter definitions, including "Type" (Regular or Hollowing cutter), the Angle of a tapered cutter ("0" would be a straight cutter); the Tip Thickness, Tip Diameter, Flat Tip Diameter, Pass Depth (depth at which the cutter is capable of cutting without breaking: notice this is UNDERESTIMATED in the default cutters, to ensure they will not sink too deep and thereby be more prone to weakening and breaking), PrePass Resolution (when creating a roughing pass, this is the resolution that pass will be; if 0.03 is the resolution at which the model will be cut, 0.08, for the default 10-degree cutter, is the resolution of the roughing pass); PrePass Offset (or, how far offset from the ACTUAL cut the roughing pass will be offset, to ensure there is enough room for the cutter to get close enough to the model without hitting the wax that was cut away during the roughing pass); PrePass Tip Correction, Shaft Diameter (diameter of non-cutting portion of cutter), flute diameter (widest part of cutting portion of cutter); Flute Length (length of cutting portion of cutter), and Free Length (length of entire cutter, including non-cutting portion). Note that not all of these fields are available when defining every type of cutter. Those that are set to "0" have NO BEARING on the cutter definition (entering "0" is akin to leaving them blank, or saying to the mill "don't use this measurement when accounting for the shape and size of this cutter".

Add / Copy: The only way to edit the features of a cutter is by first clicking "Add" or "Copy". This is to prevent you from changing any of the existing Gemvision cutters. Upon clicking "Add", you will get a totally blank screen, and you will need to fill in all the values detailed above. Usually, at the time of purchasing your cutter, you'll be able to attain all these values. This helps to ensure that Revo has the correct cutter in mind when creating toolpaths with it. Click "Set" to save your changes. Alternately, you may select the cutter that is CLOSEST TO the custom cutter you have on-hand and, with that cutter selected, click "Copy". This will copy all those cutter definitions to a new cutter, while allowing you to re-name the cutter and edit all of the definitions to match the new cutter.

After Adding a Cutter: Once you've added your own cutter using Add or Copy, it will automatically appear in all of the Cutter menus in the Detailed Options portion of the builder, including the Cutters and DBT Screen and the Detailed Options screen for ALL of the Parts and Sub Parts.



Using the Cutter Library:
(Upper Left) Highlight a cutter to see its definitions. You CANNOT edit or remove Gemvision-created cutters! To start from an existing cutter, select "Copy" (bottom Left) and change the values. To "Add" a brand-new cutter of your own, select this option and the fields will be blank. You may Rename a new or copied cutter (bottom left), choose "Type: Regular or Disc" (middle column), and enter dimensions. An image of the cutter you have defined displays (far right). When you're done defining the cutter, click "Set". Use "Undo" to reverse your changes and "Remove" (appears below "Copy" for a non Gemvision cutter) to remove a new cutter entirely from the list.

Tutorial 1: Measure & Input a New Cutter

Use the Cutters Library described above to measure and input a new cutter (in the absence of cutter definitions from manufacturer).

Many cutters you'll be able to purchase and use outside of Gemvision's default cutters are made to a specific size and tolerance, and you'll be able to input these given the manufacturer's cutter definitions. However, in the case of the dremel cutter shown here (at left), you may wish to measure the cutter for yourself and input the measurements in the Cutters Library. This is an easy process: take the measurements, copy the existing cutter in the library that is closest to the definition of the cutter you are measuring (which re-names the cutter), and make sure to select the NEW cutter in the Cutter selection screen when defining the toolpath.

Materials Needed

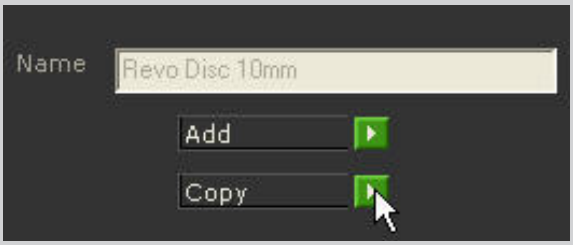
Included with Mill:
- Digital Calipers
Matrix Mill C Builder:
- "Cutters" Library
Not included with Mill:
- non Gemvision Cutter

Note: Early recipients of the Revo 540C may need to measure the dremel cutter included with the mill & input the correct Tip Thickness, as shown here, to ensure accuracy.

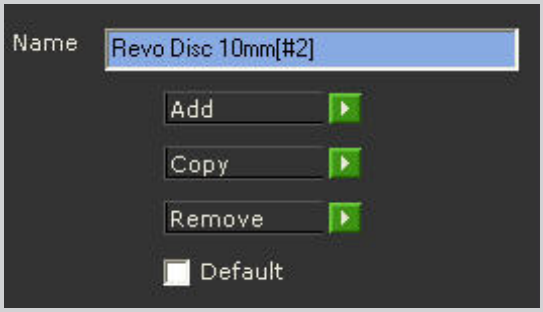
1 Measure the Cutter As shown above, use the digital calipers included with the mill to measure the dimensions on the non Gemvision cutter that differ from the cutter(s) included with the mill. Dremel cutter (circular or hollowing cutter) shown above.

Note: Hollowing cutters accompanying the mill will be standardized by Gemvision; however, early recipients of the Revo 540C who received a dremel cutter with their mill should follow these steps to ensure accuracy.

2 Enter the Cutters Library in Mill C Builder In the first screen of the Mill C Builder in Matrix, click the "Cutters" library to enter the library.

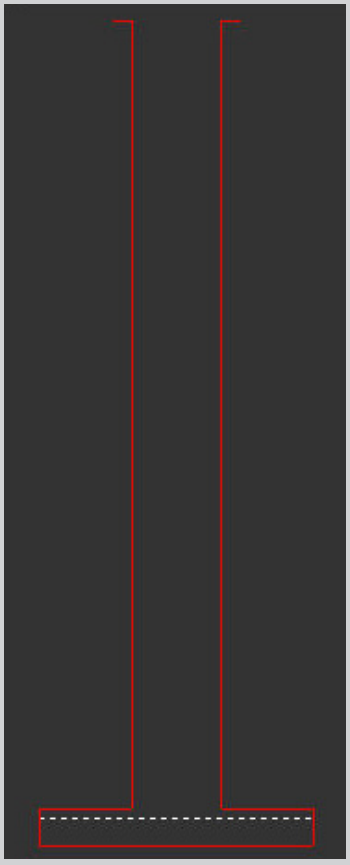


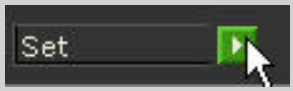
3 "Copy" a cutter similar to new cutter. Select the cutter whose type is closest to the one you are measuring and select "Copy". This will enter a new cutter into the Cutters library with editable definitions and a new name: "(selected cutter) #2" or next available number, usually.



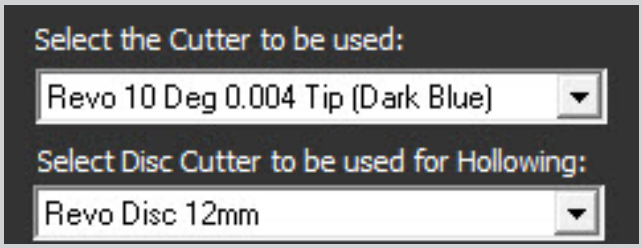
Measure & Input a New Cutter, cont'd.

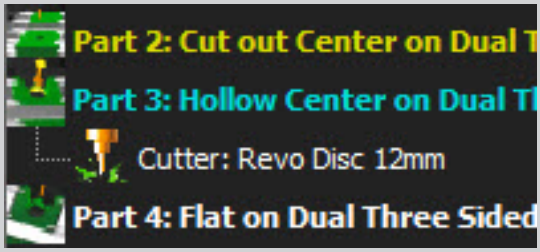
Use the Cutters Library described above to measure and input a new cutter (in the absence of cutter definitions from manufacturer).





4 Enter new value & "Set" it Enter the new value you measured into its corresponding position in the cutter definition. Click "Set" to save the cutter definition. Exit the library. Please notice something above: The "Pass Depth" is set to 1.0 mm while the Tip Thickness is set to 1.34 mm. Setting a smaller Pass Depth than Tip Thickness will allow the builder to ensure that the tip is not buried during cutting, and it's a good idea with each cutter you set up, to help prevent against cutter breakage.





5 Choose this cutter for the toolpath Complete toolpath set-up in the Matrix Mill C Builder as usual. Then, in the final screen of the Mill C builder when you are choosing a cutter, OR in the Detailed Options for the relevant part of the toolpath, be certain you choose the cutter with the new name.

Editing Parts & Sub Parts ...
DBT & "Toolpath" Options

The "Toolpath" options provides different methods for the mill to approach the cut.

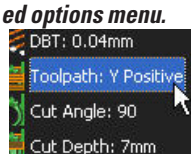
Changing "Distance Between Toolpaths"

Select this option, and a menu to enter a new value for Distance Between Toolpaths will appear. Type in a new value and click Set. The new value will be used just for this portion of the part. Why would you change this? If you wish to mill away a lot of wax more quickly, and you won't need to add a lot of detail (such as the flat back of a coin), set this value to a lower setting (0.08 instead of 0.04, e.g.).

"Toolpath" Options

How is the mill going to approach this cut: From the bottom of the model? the top? from left to right? right to left? will it include roughing passes? high detail? or will it just find the shortest route to the end?

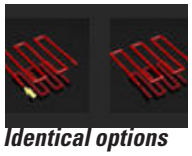
Choose toolpath. Name displays at top of menu & in detailed options menu.



All of these options are available by selecting the "Toolpath" option and choosing which of the 3 mutually-exclusive methods in this menu will be used to cut this particular portion of the model.

X or Y Positive or Negative: Select one of the top four buttons in this menu to indicate if the toolpath should run in the X direction, moving in the positive or negative directions in X (relative to the mill's spindle); or in the Y direction, moving in the positive or negative directions in Y.

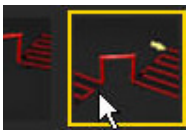
With Roughing: Select one of the four buttons in the middle row of this menu to automatically add roughing passes to the cut. These four methods correspond with the X or Y Positive or Negative methods defined above; however, they add a roughing pass to the start of the cut as defined by the cutter definition in the Cutters library (explained in a previous section in this chapter).



Identical options are available with roughing (above) and without it (below).

Spiral: Select the Spiral option on round coin style jewelry. This option is also ideal when a model has large flat areas where after the wax is complete, evidence of the toolpath approach can be seen in the wax. It can be difficult, or impossible to polish out these details. Changing to the Spiral toolpath can sometimes disguise any evidence of the toolpaths approach.

Y Positive or Negative "Shortest Route": The left two buttons in the final row in this menu allow the Mill C builder to find the shortest route over this model. When a typical Y cut is made for a ring that has the Center Cut Out center type selected (so the center of the model is already removed), the mill must cut up one side; then return to the bottom and cut down the other. With "Shortest Route" selected, it can cut across the piece quickly, skipping the center of the ring as it does so. Select "Positive" to move in the from the Y Positive direction and "Negative" to start from the Y negative direction.



Shortest Route (Above) and High Detail (Below) only available in Y +/-.

Y Positive or Negative "High Detail": To assign an area a cut that yields high detail, choose one of the two possible "High Detail" cuts in the Y (positive or negative) direction.

A Closer Look at the Toolpath Options

Below is a representation of the toolpath.

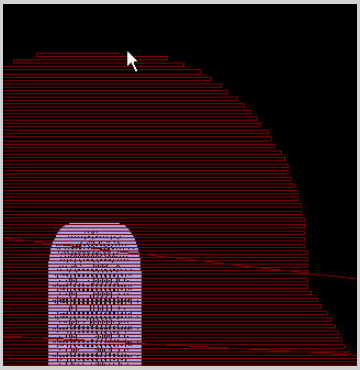
DBT: You'll recall that DBT assigns the value (in mm) between each curve as the toolpath takes a turn to make another pass over the part. It also assigns the "step along" value that defines each "point" along the toolpath that together define the path.

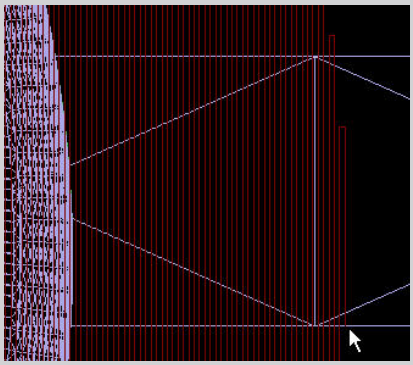
Spiral: Ideal when a model has large flat areas. The toolpath approach is less visible in the wax.

X & Y Positive & Negative: The direction that the cutter will move (relative to the model) to create the toolpath is defined by X or Y Positive or Negative.

Roughing: Select a "Roughing" pass and the mill will automatically account for the "Pass Depth" of the cutter assigned in the cutter library. This means that, to add additional roughing passes, you'd need to enter the library, "Copy" the cutter of your choice, lower the "Pass Depth" for the new cutter, and be certain you pick the new cutter for this section of the toolpath in Detailed Options.

High Detail: This is a special type of cut that runs only in the Y pos. or neg. direction.

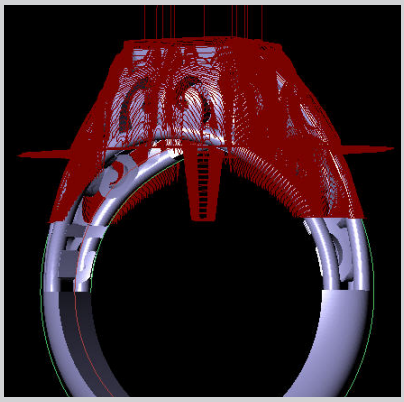




(L) X Negative toolpath runs in the Neg X direction (from top of model) (R) Y Negative toolpath runs in Neg Y direction (From side of model)

Tutorial 2: Edit Cutters to Add Roughing Passes

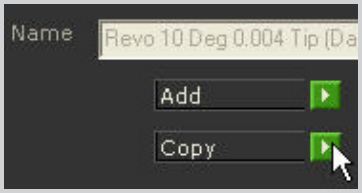
Add roughing passes to a part to ensure the cutter will not break, or to ensure maximum access to fine details (as when slowing feed rate).

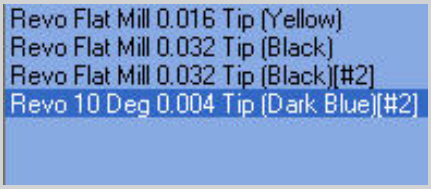


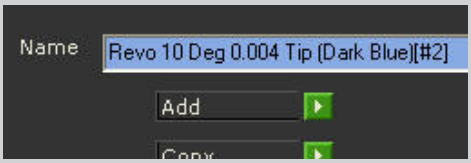
If you wish to add additional roughing passes (shown at left) to a part, you must access the Cutter definition via the "Cutters" library and adjust the allowable "Pass Depth" for the cutter you'll be using for this part of the toolpath. Why do this? Well, if you are afraid the cutter is cutting ALMOST too much wax and is in danger of breaking, or if you plan on running a certain part of the model at a slower spindle speed - and slowing down spindle speed in deep wax leaves the cutter in greater danger of breaking - then lower the pass depth and additional roughing passes will be added to ensure the cutter's safety.

Materials Needed

Matrix Mill C Builder:
- "Cutters" Library
- Cutter Selection (final Mill Builder screen or Detailed Options screen).

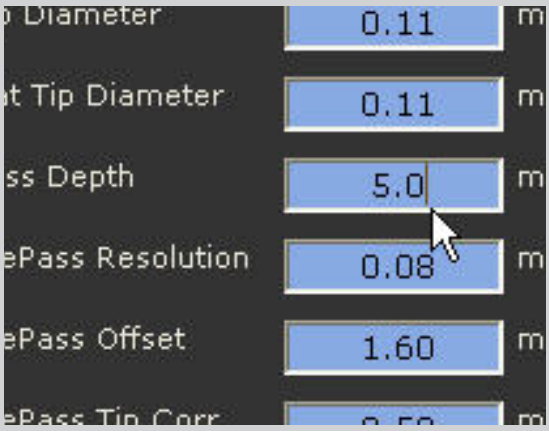


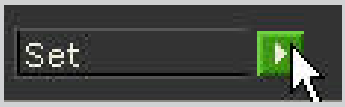




1 Enter the Cutters Library in Mill C Builder In the first screen of the Mill C Builder in Matrix, click the "Cutters" library to enter the library.

2 "Copy" the cutter Select the cutter which you will be using to complete the pass with additional roughing and select "Copy". This will enter a new cutter into the Cutters library with editable definitions and a new name ("selected cutter #2", usually).



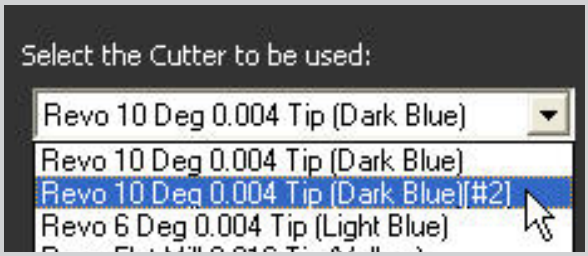


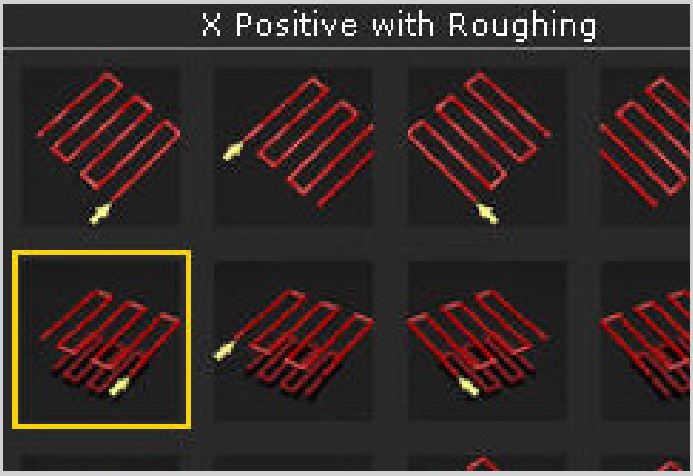
3 Enter lower "Pass Depth" value and Click "Set" Select a new, lower value for "Pass Depth" (cut the current value in half, e.g., or just lower it by a few mm). Remember: this value defines how far into rough wax the software thinks that the cutter can go (not how far it can actually go) so, when a "Roughing" toolpath is selected in the "Toolpath" options, this combined with the new cutter definition will cause the software to add roughing passes to clear away the right amount of uncut wax to approach your details with minimal danger to the cutter.

Note: to spend less time on this pass, you may also raise the resolution value, since it's a roughing pass only (cutting rough wax) and doesn't need to cut at a high resolution. Raise the value of "PrePass Resolution" (DBT for roughing passes) to do so.

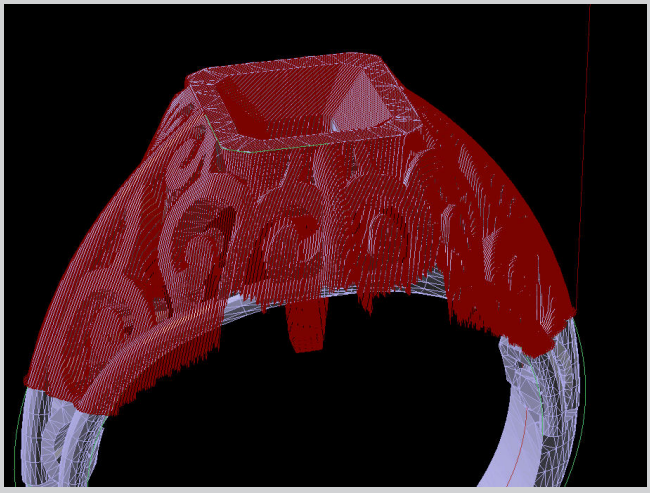
Edit Cutters to Add Roughing Passes cont'd.

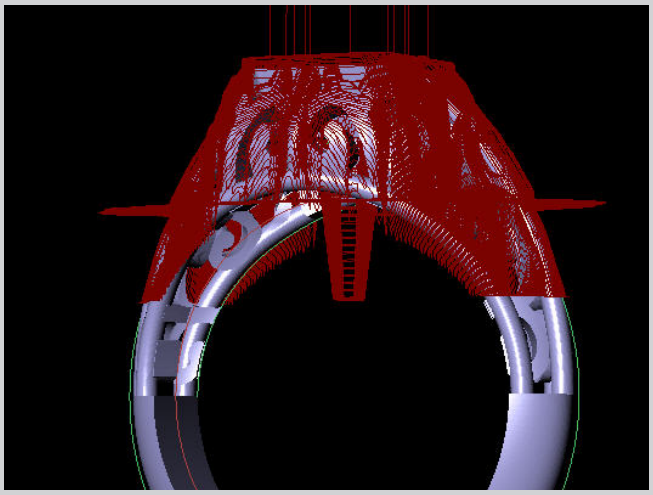
Add roughing passes to a part to ensure the cutter will not break, or to ensure maximum access to fine details (as when slowing feed rate).





4 Choose this cutter & roughing for the toolpath Complete toolpath set-up in the Matrix Mill C Builder as usual. Then, in the Detailed Options screen, select the new cutter you just defined AND be certain you select a Toolpath option with "Roughing" enabled.





(Above) Same toolpath with and without roughing passes, as added above. (L) Without roughing; (R) with roughing.

Editing Parts & Sub Parts ...

Cut Depth, Angle, & Borders

Learn about Detailed Options Cut Depth, Cut Angle, & Border for FLAT toolpaths

Cut Angle and Cut Depth may be adjusted for Flat cuts made on the typical flat fixtures (Dual Three-Sided and Base Clamp) as well as for flat cuts made on the Rotary fixtures (Three-Jaw Chuck and Hub), wherein the fixture is held flat at one angle on the A axis while the cut is made.

The method to adjust the Cut Angle, Depth, and Borders for any flat cuts, regardless of fixture, is the same. However, for a Rotary cut (during which the model is rotated while it is being cut) on either the Three-Jaw Chuck or Hub fixture, the method to adjust the Cut Angle differs. For a Rotary cut, only Cut Angle (not Depth or Border) can be adjusted. Read about how to adjust each of these options below.

Cut Depth, Flat Cut Only

When it is apparent that the default cut depth of 1-mm past the center of your model is not adequate to complete one half of the model - as in the case of an asymmetrical model or a model whose details continue past the "halfway point" of one side and are missed by the cut - it is necessary to adjust cut depth.

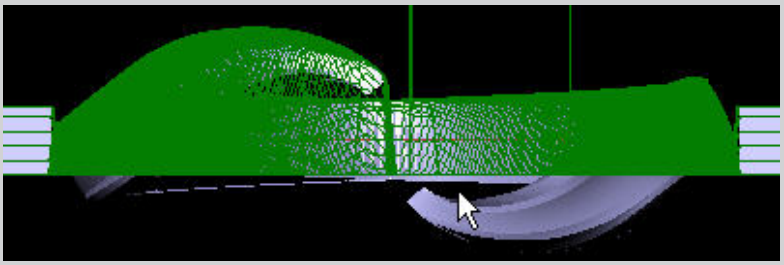
To do so, double-click on the Part or Sub Part to expand it, which displays all the detailed options associated with this part. Click on the "Cut Depth" option, and you will see a "cutter" with plane representing the Cut Depth (and Angle - more on that later) appear in the viewports. This cannot be selected in the traditional sense. Instead, to grab hold of it, click "Go" in the menu that opens underneath Detailed Options, and Viewport Control Handles will appear on the object.

Click and drag the "arrows" handle to change the depth of the plane. You will only be able to move this along one plane, to help ensure you cannot drag the cutter to an unrealistic location. Press Enter to keep the depth you've set. The handles will disappear and the depth will change in the Detailed Options menu. Note: You will need to do this for each like Sub Part, if it is necessary: there is no way to "Mirror" a cutting plane.

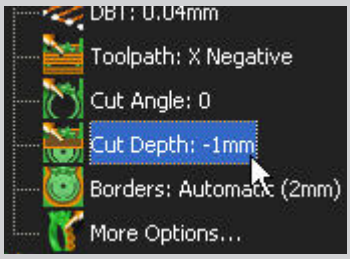
Click "Make Path" to create the toolpath with the new settings, and the new depth will be visible.

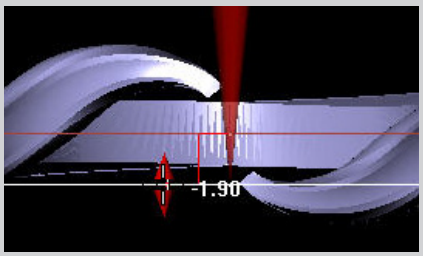
Adjusting Cut Depth, Flat Cut (on any fixture)

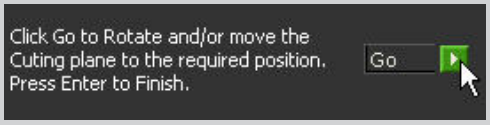
Cut Depth: Select the Cut Depth option for a Part or Sub Part for which, clearly, the toolpath does not sink in deep enough to reach all of the details necessary for this part (it defaults to 1-mm past center). Click "Go" to turn on Viewport Control Handles in "Cut Depth" Detailed Option. Make changes and press Enter. "Make Path" again to see changes. "Before" and "After" results, below.



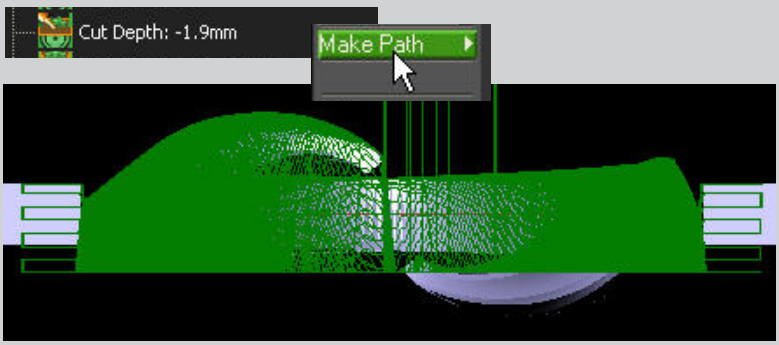
(Above) The default cut does not sink deep enough into the model to reach the details on the opposite side of the bypass shank.







(L) Select Cut Depth Option & click Go. (Above) Adjust Handles to set appropriate depth. Press "Enter" to end.



(Above The Cut Depth will change to reflect the new value. Click Make Path to see the difference in the toolpath. Now, it reaches all the way to the details on the other side of the ring.

Adjusting Cut Angle, Flat Cut

Although this is usually only done (and only recommended) when a toolpath is Added or during the additional cuts of a 5- or 8-sided job, (see later on in this chapter), it is possible to adjust the angle of a flat cut made with the Vertical cutter on the Dual fixture or Base Clamp in order to better approach the details being cut during that Part of Sub Part.

Pleas Note: Ideally the part being milled is only milled once. Milling the same part multiple times from slightly different angles can cause undesired results.

To adjust the cut angle, expand this Part or Sub Part and click "Go" to see Viewport Control Handles. The rotational handle can ONLY be rotated in the correct direction for this cut, so don't worry about rotating it the wrong way. Also, if it's rotated too far for the fixture to handle, a warning will appear and the angle will be re-set.

Rotate this object so it is at the best angle to approach the details perpendicular to the detail you wish to mill and press Enter to keep your changes. You will need to do this for EACH Part and Sub Part that is alike: there is no way to Mirror or Copy these values to other parts. For clarity, hold down Shift while clicking and dragging rotational handle to rotate in 5-degree increments.

Adjust Depth & Angle, Flat Cuts

You will notice that both options (Cut Depth and Cut Angle) offer both Viewport Control Handles. They are mutually exclusive. You may adjust them both while within either option and the value will be saved: there is no need to jump into the other option if you don't want to: you can save yourself a click.

Adjusting Borders, Flat Cuts

The Border is the space inside which the mill will cut during the given Part or Sub Part. On a Flat cut on the Dual or Base Clamp (the standard 2, 3, or 4 flat cuts), the border is 2 mm past the extents of the part. This leaves enough space between the model and the wax surrounding it for the cutter to safely make its pass and to remove the model from the wax after milling.

For a custom-angle cut (extra 2 angles on Dual or 4 angles on Base Clamp), this distance is 0.5 mm past the extents of the part. For a rotary cut, this value is 0.2 mm past the extents of the part.

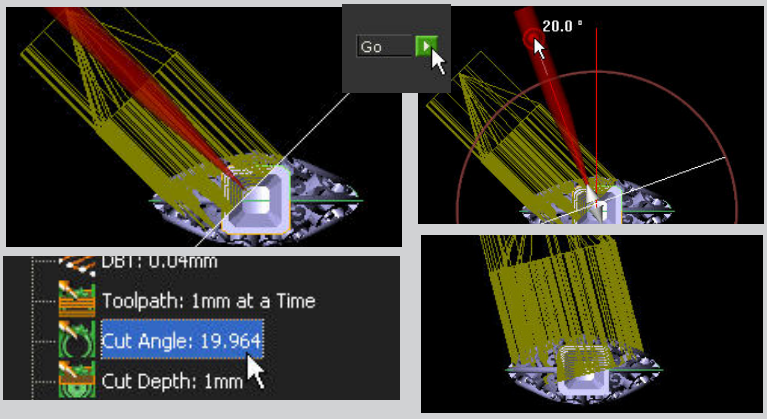
To change this value, expand the Part or Sub Part to change by double-clicking it and click on the "Borders" option in the Detailed Options that appear. In the menu beneath the Detailed Options outline, you'll see the options "Automatic Border" and "Manual Curve Border".

Automatic Border: This option automatically calculates the extent of the part and adds to that the value, in mm, that you input in the box beside it. To allow the builder to calculate the shape of the border, select this option and type a new value, in mm, into the box.

Manual Curve Border: This option allows you to input your own border curve into the In-Box that appears beside it. You may draw this curve by selecting any drawing tools you may need from the Rhino drop-down menus at the top of the

Adjusting Cut Angle, Flat Cut (on any fixture)

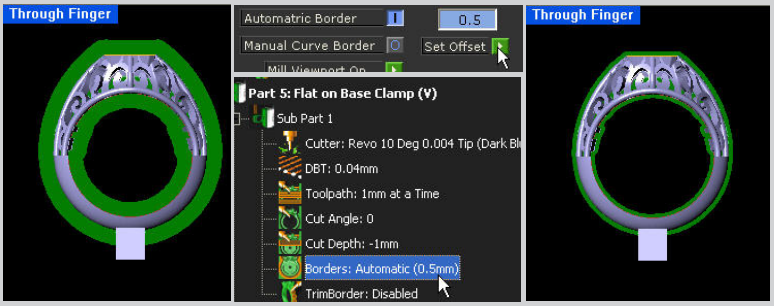
Cut Angle: Select the Cut Angle option for a Part or Sub Part you "Added", or for one of the additional angles cut during a 5- or 8-sided cut, as shown below. Use this cut when the default angle given can be improved to better approach the details on the model. Click "Go" to access the Viewport Control Handles and adjust the angle. Press Enter to keep changes, and Make Path. The results will be displayed in the new path.



(Upper Left) Is toolpath at best angle for cut? Original Cut Plane & cutter shown. (Upper Right) Cut Plane will be able to be rotated ONLY in view with rotational handle visible. (Lower Left) Press Enter to see changes. (Lower Right) After Make Path, new angle is apparent.

Automatic Border, Flat Cut (on any fixture)

Automatic Border: By default, each border is set to Automatic Border. This setting calculates the extents of the part and adds a set value to that, depending on which type of cut it is. This value is displayed in the menu beneath the Detailed Options outline. Change this value and, with Automatic Border selected, the new border will be calculated upon toolpath creation. (Click Set Border and Make Path to keep your changes). Lowering this value - when it is practical to do so - saves milling time.



(L) Before: Border at 2 mm. (Center) To Change Border. (R) After: Border at 0.5 mm.

screen. Draw this border planar in the viewport toward which the cutter points (visible during Cut Plane and Cut Angle options). It must be a closed curve.

If the view is not planar with one of the C Planes, which often happens, click "Show" in the Milling Viewport option to turn on this viewport. This option becomes available after choosing Manual Curve Border. This new, special viewport is planar with the CUTTING PLANE - the big plane out there that, along with the cutter, defines the angle and depth of this part or sub part of the toolpath. Now, it is easy to draw this curve planar against the grid, input it into the "Border" box, and this will define the extents of the cut. Close this viewport to proceed.

Set Border: In the event you choose Automatic but would like the border offset from the geometry a distance different than 2mm, type a new value into the text box and click "Set". Don't forget to use Make Path again so that your changes are reflected in the toolpath.

Cut Angle for Rotary Cuts

Rotary cuts do not allow the user to input a border curve; however, you can assign two angles between which the cut will be made. In this way, you can limit a rotary cut to ONLY cut the parts of the model that require this approach. This saves time and prevents parts of the model that do not need a rotary cut from being milled twice.

You may also wish to use this option when you have a especially fine details in one part of a rotary model and you wish to use the "High Detail" cut JUST for this portion of the rotary cut; not for other portions. Similarly, if you wish to use a special cutter in just one portion of the rotary cut, and you won't need to use it over all of the model during the rotary, this is how to set up that case, as well.

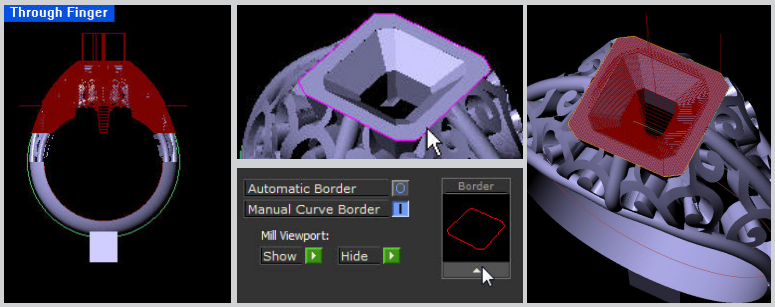
How to Assign a Rotary "Cut Angle": To use this feature, select this option from the Detailed Options list, opening up a menu under the Detailed Options outline. Click the "Line" button and a line will appear in the viewports, automatically extending from the center of the viewport in which it is correct to place this angle (this viewport may differ depending on whether you are milling a Ring or Non-Ring model and what Non-Ring approach you are using).

Click once in that viewport, on one side of the details you wish to cut using this strategy, to indicate the end of the line. Click the Line command again, and place this line on the opposite side of the area you wish to mill. Input one line as the "Start" angle and the other as the "End" angle. The mill runs in a clockwise direction, so keep this in mind while choosing your Start and End angles.

Remember that every time you make a change to Detailed Options, you must click "Make Path" again for your results to affect the toolpath. Check out your results and make sure they are correct.

Automatic Border, Flat Cut (on any fixture)

Manual Curve Border: Select this option when you wish to draw your own border beyond which this Part or Sub Part of the toolpath will not be cut. Use the Rhino dropdown menus at the top of the screen to draw the curve, or draw it before entering the mill builder, and make CERTAIN the curve is closed and planar. It should be drawn in the viewport at which the Cutter (visible in Cut Angle and Cut Depth) points. If this is not a planar viewport, click 'Show' under the Milling Viewport to turn on a viewport with a CPlane that is planar with the cutting depth. Draw the curve in this view, input it, and hide the Milling Viewport.



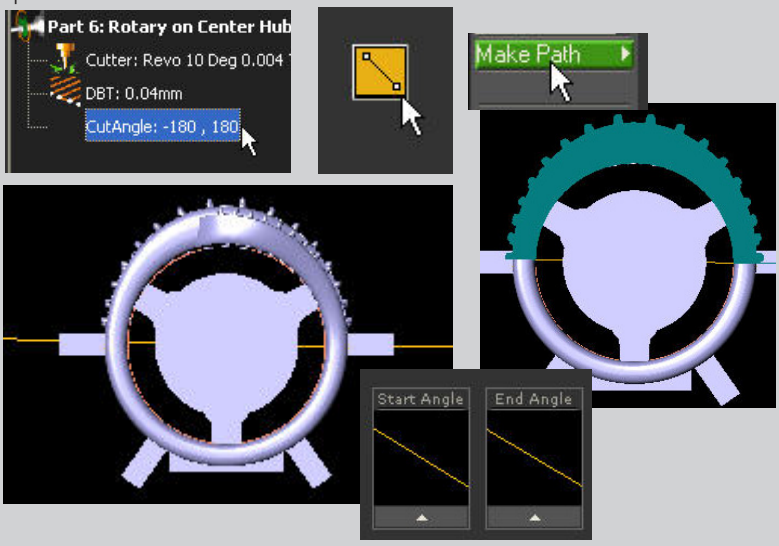
(L) Before: Toolpath can create a "mold line" or unsightly offset when parts of model are milled twice: once by Top pass & once by others. (Center) Input new border (Creation Curve for bezel). (Right) New path just cuts bezel from Top.

Setting Cut Angle, Rotary Cuts (Rotary Fixtures)

1 - Select CutAngle Option. Select this option and the Cut Angle menu appears underneath the Detailed Options outline.

2 - Click "Line". To assign a start and end angle, click "Line" and assign the end of the line for each angle. Mill works in clockwise direction.

3 - Input "Start" and "End". Input each line in the correct box to indicate the "Start" and "End" angles. "Make Path" to see results.



Editing Parts & Sub Parts ...

"More Options" for Borders

The following detailed options are described below, along with how & when to change them: Trim Border, Fill Holes, Border Softening, and Protect Finger.

About "Trim Border"

Only turned on by default for the additional 4 sides of an 8-sided cut and the additional 2 sides of a 5-sided cut, "TrimBorder" allows the current cut to account for the areas that were already cut and not re-cut them. This is an easy feature to change: simply toggle On or Off the "Cut New Only" setting found in the More Options menu.

The "Trim Border Tolerance" defines the depth of the cut that the "Cut New" feature will NOT re-cut, to avoid gouging the model. So if the model is already cut at a depth equal to or greater than this value, it will not be cut again by this toolpath. If it's cut less than this value, it will be re-cut during this pass. So, raise this value if the toolpath is over cutting an area that was already cut, causing gouging in the model, or lower this value if it is not cutting parts that should be cut and were missed by other parts of the toolpath. The default value is 0.2 mm. Type in a value, in mm, for this Tolerance and click "Set" to assign it.

Please note that it is a BAD idea to change this for the basic cuts: these being front, back, and top (Dual), or front, back, left, right and top (Base Clamp or Four-Flats on Rotary). ONLY use it for additional angles (see the "Adding Options" section of this chapter) you wish to add beyond the basic ones listed above.

When adding additional angles to a strategy, this option will save you lots of time and helps prevent gouging your model, which can happen when parts are cut more than once from different directions.

About Hole Filling

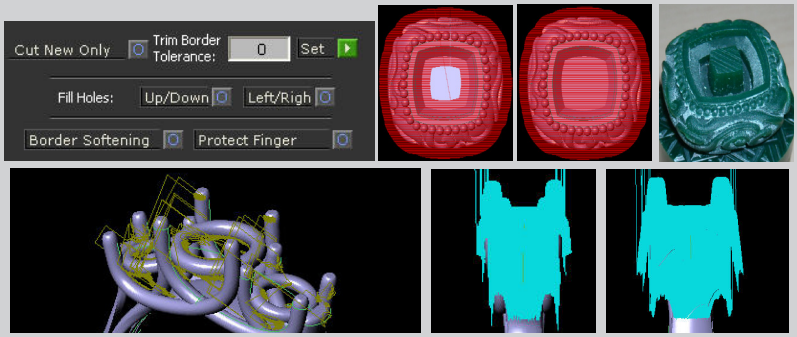
The next option in the "More Options" menu that opens beneath the Detailed Options menu when this selection is made in the outline is "Hole Filling". This feature should be turned on when there are holes in the model that are large enough so that the builder treats them as "borders" of the model. You'll recall that a border is offset from the model the distance set in the "Borders" option (defaults are 2 mm for basic flat cuts, 0.1 mm for custom-angled cuts, etc.). Therefore, if a hole is so large that it is treated like a border, a piece of wax will be left inside the hole that could break the cutter or damage the model. To keep this from happening, select "Fill Holes" during a toolpath that passes over such a hole. This feature will analyze the model from top to bottom (select "Up /Down" option) from "Left to Right" (select this option), or both (select both options) to find and fill these holes.

More Border Options...

Trim Border Cut ONLY the parts of the model that were NOT already cut by other Parts or Sub Parts at a depth defined by the Tolerance value or deeper. This option ("Cut New Only") is turned on by default for the additional angles on 5-sided and 8-sided cuts.

Hole Filling When a model has a hole so large it is treated like a "Border" (the toolpath is offset from it the "Border" distance), select "Fill Holes" to protect cutter & model. Choose "Up / Down" &/or "Left / Right" to search for holes in one or both directions.

Border Softening Where two or more cuts overlap, a line may be visible at their borders where the cutter pulls back to exit the ring. Choose this option and the cutter will take a curving path away from the model, softening the border's appearance.



(Below Left) Trim Border ON; (Above, Ctr) Hole Filling OFF; (Above, Ctr) Hole Filling ON; (Above, Right) Wax Fill Holes; (Below, Ctr) Border Softening Off; (Below, Right) Border Soft. ON.

Border Softening

When the same section of the model is milled from the top-down and from one or more sides, so that the cuts overlap one another, occasionally this overlap might be visible right at the borders of the cuts, where the cutter pulls straight up and away and exits the ring. If this creates a visible line in your model, select Border Softening. Rather than pulling straight out and away from the model at the border, the cutter follows a curving path away from the model before exiting the border, in effect "softening" the edges of the border. When two or more borders that overlap are "softened" in this way, that visible line is diminished.

Protect Finger

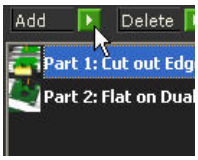
To save milling time, select this option where available and the cut will stop at the finger rail, to prevent the mill from "cutting air" inside the finger hole.

Adding Parts & Sub Parts ...

Add, Delete & Move

Learn how to Add Parts and Sub Parts to any toolpath.

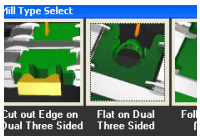
Adding a Part or Sub Part



Click "Add" to add Parts or Sub Parts

"Add" allows you to add a Part or Sub Part in order to mill additional details on the model that wouldn't otherwise be milled. Depending on which fixture and strategies you've selected, you will be offered different options during the "Add" command. These will all be covered in this section.

For instance: you can "Add" a Part or Sub Part that is just like one you're already running during this job. But this also means that, if you haven't selected a Hollow center, you won't be offered the option to add a Hollow center to your model later by adding another toolpath. Likewise, if you've chosen a two- or three-sided model on the Dual Fixture, you won't be offered any of the Rotary options unless you've also chosen the Hub center in the first screen of the builder.



Types of parts available to "Add" limited by strategy & fixture chosen.

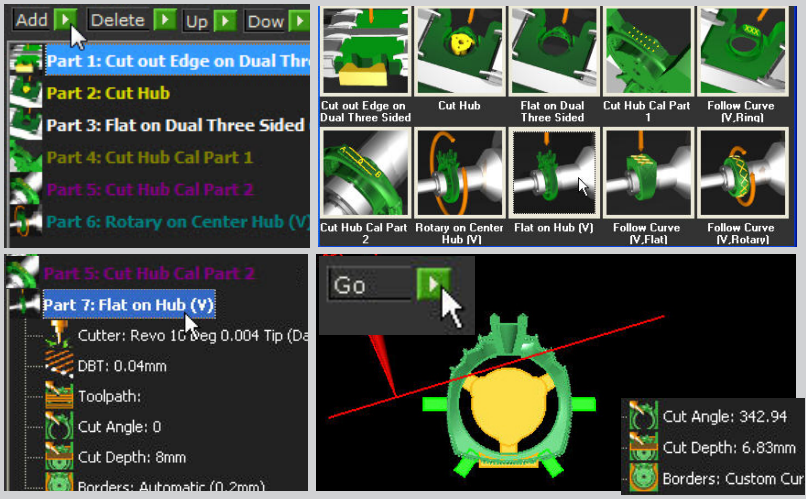
The options you will be offered include all the types of cuts already being run in your toolpath, including all of the cut-out curves, hollowing cuts, hub cal, etc; as well as three new kinds of cuts we'll be exploring in the remainder of this chapter: three "Follow Curve" cuts, which are cuts that simply move the cutter at a set depth along a curve on the model to create an engraving effect.

These are "Follow Curve on Flat" (traces a curve that lies flat on one flat angle in a model; this may be done with the Vertical or Horizontal cutter depending on the necessary approach for the model being cut), "Follow Curve on Rotary" (traces a curve that runs around the rotary portion of the model, so that the model is turned on the A axis while the curve is being cut), and "Follow Curve on Flat on Rotary" which holds the model on the rotary fixture flat on the A axis, at an angle indicated by the user, to trace a curve on the model at that angle.

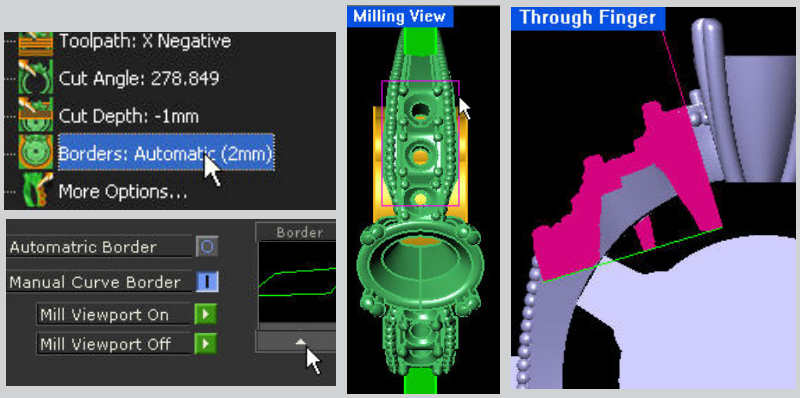
Using "Add" with other Detailed Options

Add: Click this option to see a menu of all cuts that may be made with the selected strategy - including any cut-out, hollowing, and hub cuts that are ALREADY supported by the toolpath (i.e., they MUST have been selected earlier-on in the builder. If the cut need is not available, click "Back" and use different builder options.

Choose an Option: Click on the option from this menu that you wish to Add. It will be automatically added to the location in the Detailed Options outline that the software determines is best for it: i.e. with like cuts ONLY. You will NOT be able to move a Part; you can move a Sub Part.



(Upper Left) Choose "Add". (Upper Right) Select a new Part or Sub Part - "Flat on Hub (V)" selected here and it will be added to the toolpath (Lower Left). You may NOT Move a Part: just a Sub Part. (Lower Right) Set a new cut angle.



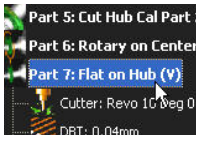
(Left) To limit what is cut, select the "Borders" option and turn on the Milling Viewport. (Center) Draw a Rectangle planar in this view using the Rhino Dropdown menus. (Right) After: Only new toolpath shown.

Uses for "Add"

"Add" identical cuts with a smaller cutter for "finishing passes".

Use "Add" for placing finishing passes (an identical cut made with a smaller cutter that cannot cut from rough wax). Finishing passes FOLLOW a cut made by a larger cutter which can cut from rough wax. Or, use "Add" for additional cutting angles for models with details that can best be achieved from angles other than the default cutting angles.

Order of Operations during "Add"



Builder knows where to Add each Part or Sub Part. Only Sub-Parts can be re-sequenced.

The toolpaths you add "know" where to come out in the Detailed Options outline. For instance, if you choose to re-cut the Hollowing pass - say, with a different, non Gemvision cutter for finishing hollow work - the software "knows" to add this part AFTER the existing Hollowing cut - not at the end of the entire job. Also, if you are cutting a 3-sided piece on the dual with the hub center, you will have the option to Add a flat cut (at any angle you choose) or a rotary cut (on the hub). If you choose to add a flat cut, the builder will (very intelligently, might I add) add the new cut on as a Sub Part at the end of all the flat parts; conversely, it will add a new Rotary cut on the Hub onto the end of the entire job as a new "Part".

However, keep in mind that you may ONLY change the order of Sub-Parts within a single Part: NOT the order of the Parts overall. This is because the software knows when it has to cut each part of the model: i.e., it will not let you "Move" a rotary part up before a flat part during a Dual Fixture Plus Hub Center strategy, since this is illogical: rotary must be cut AFTER flat, not the other way around. However, be VERY careful about the order of operations for any Sub Parts you've moved before using "Make Path" and "Save Path": do NOT, for instance, move a cut made with a finishing cutter up before a cut made with a cutter that can cut from rough wax.

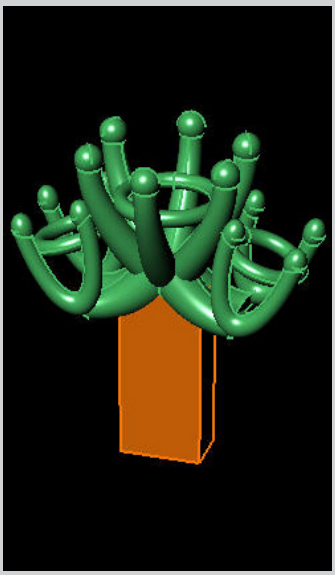
In the example above, at right, you can see when it would be logical (and perfectly fine) to change the order of sub parts: In this example, a 3-sided non-ring model on the Dual fixture has very fine details, and would be structurally stronger if the angled cuts (45- and 135-degrees) were made before the traditional flat cuts (0- and 180-degrees). In this case, you may rearrange the order of the Sub Parts so that the angled cuts are made first, while the model is the most stable. Just make certain the cutter is not sinking too deep into rough (uncut) wax: once you begin moving these operations around, it is up to you to verify that the proper amount of wax has been cleared away so that the cutter will not be broken when it sinks in to make the next cut.

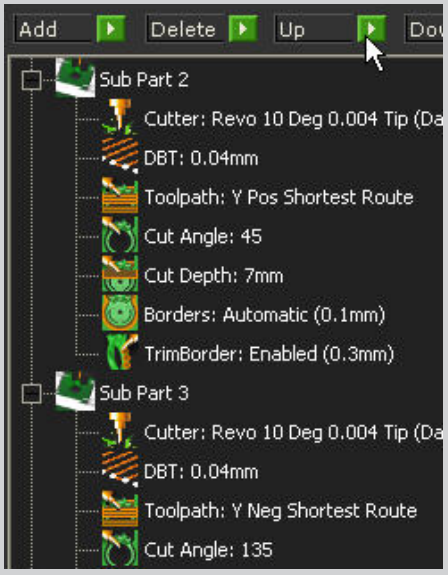
Using "Delete" & "Move". When to use Move?

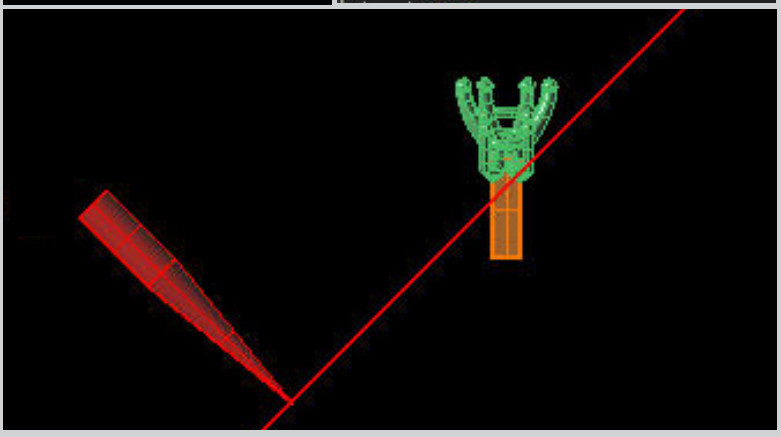
Delete: If you need to delete a Part or Sub Part, select it and click Delete. If you delete one too many, there is no "Undo" so be careful: you will need to "Add" that part of the operation back in and set up the options again.

Move: With the Part or Sub Part to move selected, click "Up" or "Down" to move it up or down in the list. You will NOT be able to move operations out of their logical order: for instance, you will not be able to place a Rotary cut on the Hub BEFORE a flat cut on the Dual Fixture.

In the Example Below: (Left) In this Three-Sided model on the Dual fixture, the model might become too weak during the 0- and 180-degree cuts, causing it to flex during the 45- and 135-degree cuts. Therefore, using the "Up" button (Right), Sub Parts 4 and 5 (traditionally 45- and 135-degrees) were moved up in the order to become Sub Parts 2 and 3. That way, the 0- and 180-degree cuts will be made last so that the model will have more structural integrity when the angled cuts are made, and it will be less likely to break.

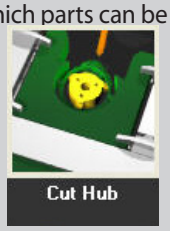


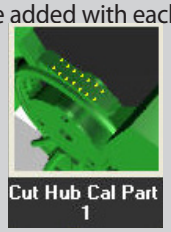




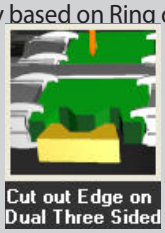
At-A-Glance: Adding Parts

Which parts can be added with each strategy? (Note: Icons may vary based on Ring or Non-Ring Model & fixture chosen.)

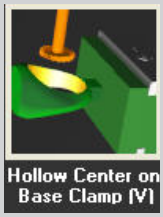




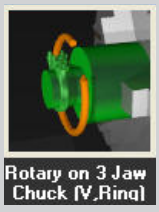
Hub Parts Parts representing the Hub cuts (center holes, Hub Cal Parts 1 & 2) appear ONLY during a Ring model with a Hub Center selected (the Hub cannot be added to Non-Ring Models) and may be added if they are mistakenly deleted, or if you wish to re-cut them for any reason. "Add" will only sequence them with like parts (you may not re-sequence them).

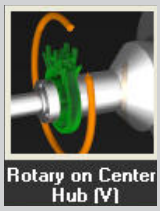


Cut-Out Parts Cut-Out Edge parts appear ONLY when the fixtures / strategies for which they are needed are chosen earlier in the menu: with the 3-sided cut on the Dual Fixture and when the Base Clamp fixture is selected, respectively. When the Cut Out Center cut appears, the icon will reflect WHICHEVER fixture is being used (base clamp above, right).

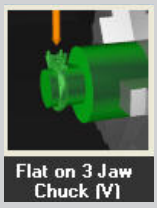


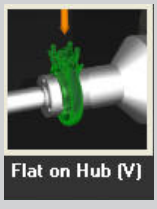
Hollowing Parts Parts representing the hollowing cuts will vary based on the fixture selected: 3-jaw chuck with hollow tube wax above, left; Base Clamp above, right. Note also that a Cut Out Center option (center, above) will also appear when the Hollowing cutter is used with either of the flat fixtures. Be CERTAIN the Cut Out Center option precedes the Hollow Center option (don't inadvertently delete it.)

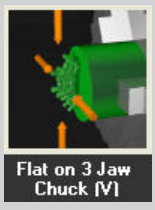




Rotary & Follow Curve on Rotary Notice that the icon will differ based on which rotary fixture (3-jaw chuck or hub) is selected, as well as whether the model is a ring or a non-ring model. Rotary cuts, as you know, are made while the model is turning on the A axis. The option to Add one will appear whenever the 3-Jaw Chuck or Hub fixtures are selected. They can ONLY be added AFTER an existing rotary cut, and CANNOT be moved in the sequence. "Follow Curve on Rotary" also appears under the same circumstances. Input a curve that proceeds around the outside of the ring or non-ring model and can be cut while the model is turning on the A axis.





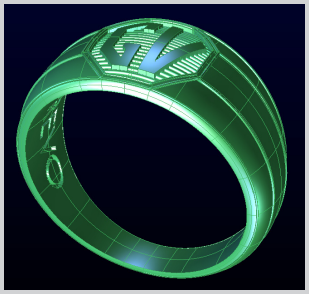


Flat on Rotary Cuts Whenever the 3-jaw chuck or Hub fixtures are chosen, the option to hold the model flat on that fixture and mill it at a chosen angle will be offered. These options are also available with curves you may input (Follow Curve on Flat, V). If you are milling a non-ring model ONLY, the option to complete a top-down cut (H cutter) will also be offered, as well the option to follow a curve with the Horizontal cutter (center and right, second row).

Flat Cuts When adding a flat cut to the Dual Fixture (above left), you may add it at any angle of which the mill is capable (safety features in the software will not allow you to enter a Cutting Plane at an angle the mill cannot physically reproduce). Flat cuts on the Base Clamp (above, center and right) are offered for the Horizontal and Vertical cutters. The angles may ONLY be adjusted on the Vertical base clamp cuts.

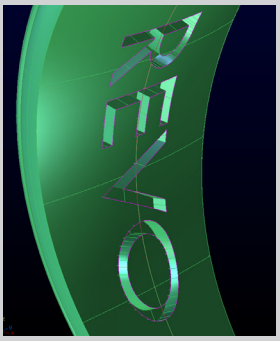
Tutorial 3: Borders, Depth, Angle & Protect Finger Rail

Explore the advantages of using Detailed Options to add Manual Curve Borders. Learn how to disable the Protect Finger Rail setting to mill on the inside of this rings shank and to do so we will need to explore Cutter Depths and Angles.

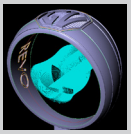


Evidence of how a part was milled can sometimes be seen in the finished wax. By simply adding a Manual Border Curve, you can improve the quality of the finished wax and avoid any unsightly lines, similar to the look of a mold line present when injecting a wax into a two part rubber mold. These "mold line' can be seen when the values entered into the wax calibration block are slightly amiss. Using a Manual Border will avoid any evidence of this offset. We will also explore how to use the Base Clamp strategy and disable the Protect Finger Rail Option to allow the cutter to delve inside of the shank of the ring to place an hallmark or special message for your customer.

1 Prepare the Model Verify the model includes a ring rail and it is the proper size. In preparation for creating the toolpath, also include a closed curve that encompasses only the GV emblem on the top of the ring. This curve will be used to define the millable area. This curve can be drawn from scratch, or better yet extracted from the emblem itself. Verify it is a closed curve.



2 Prepare the Model continued Create additional curves that outline each letter on of the text on the inside of the shank. To do so, use the Extract Surface tool to isolate the inner surface of the ring's shank. Next run the Duplicate Border tool located within the Curve Menu. This will create a curve on every open surface, thus creating an outline of the text. Within the Utilities menu, Group these curves. The curves created in Step 1 & 2 will be used to define the area in which the mill will be allowed to cut the model. Although it is easiest to draw these closed curves while in Matrix prior to entering the Toolpath Builder, if you forget to do so, you can access the needed tools using the Rhino drop-down menus after you have entered the Toolpath Builder.

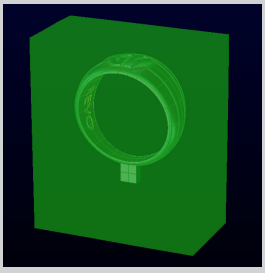


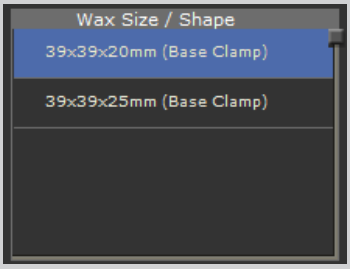
3 Model In Enter the Toolpath Builder and input the model into the "Model" preview window. "Ring Model" should be selected automatically, with the correct Ring Size displayed. If it is not, select this option and manually input the Ring Size in mm. Or, return to Matrix and place the appropriate size ring rail. Click "Next".

4 Support Strategy . In this case, we choose the "Bottom" support, which is the support strategy corresponding to the Base Clamp fixture. This strategy can reach the top-down (Looking Down view of the model), front and back (Through Finger view of the model) AND left and right (Side View of the model) sides of the ring. The Base Clamp has the additional benefit of adding four custom angles on the 45-degree slant, in addition to the front, back, left & right.

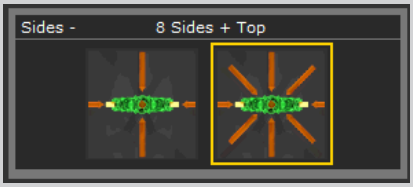
5 Center Choose the Hollow option to bring the ring up to the appropriate finger size and remove extra wax that remained on the inside the shank from the angle on the 10 degree cutter when it approached from the front and back sides. Click "Next".

6 Wax Size The software will recommend the smallest possible wax to use for the model. Here choose the actual size of the wax block you'll be using to cut this part. Click "Next".



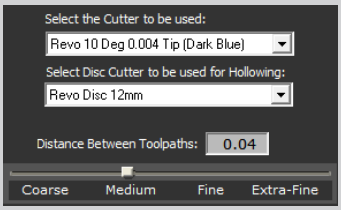


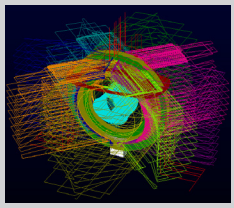
7 Approach Angle Choose the 8-sided approach needed to mill the text on the inside of the shank. Click "Next".



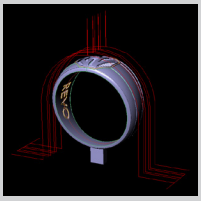
Borders, Depth, Angle & Protect Finger Rail cont'd.

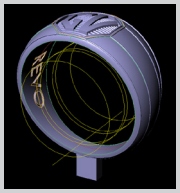
8 Cutter and DBT Choose the 10 degree cutter and the 12mm hollowing cutter. Define the value for the DBT. Click "Next".

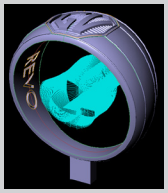




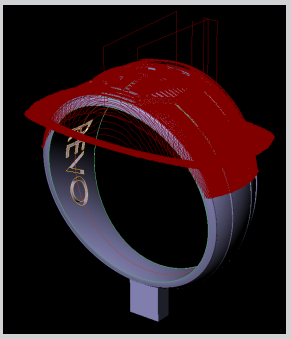
9 Detailed Options Click "Make Paths" to view the toolpaths. The color coded toolpaths will display in list form along the left-hand side of the screen. White represents a Part that is comprised of more than one sub part. Begin at the top of the list and view each toolpath. Part 1 is the Cut Out Edge Curve where the cutter follows the curve to remove large chunks of wax very quickly. Part 2, Cut Out Center, removes the plug on the interior of the ring to allow the Hollowing cutter to enter the wax. Part 3, Hollow Center brings the ring up to finger size.

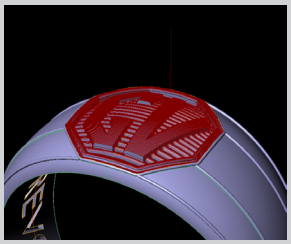






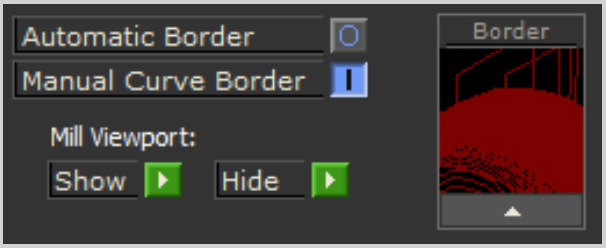
10 Detailed Options con't - Borders Part 4, Flat on Base Clamp with the Horizontal cutter completes a roughing pass first, then mills the emblem and all of the detail it can reach on the top of the ring. However, the only detail we need to reach during this pass is the GV emblem. To isolate this area use a Manual Curve Border. Double click on Part 4 to expand the Detailed Options available for this Part. Click on Borders to display the Border menu options. Toggle the Automatic Border option to Manual Curve Border. Insert the hexagonal closed curve created in Step 1 into the Border Window. The toolpath will disappear. To view the results of this change, Click Make Paths. This new path (pictured far right) will take less time and will produce better results avoiding any unsightly "mold lines".



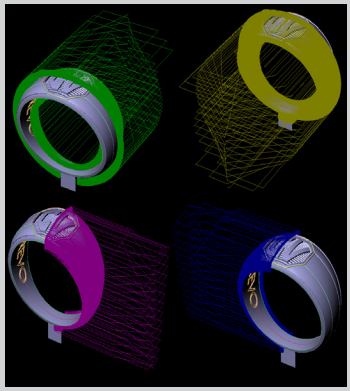


Manual Borders Curves MUST be Closed Curves.

Pictured above, this ring was milled using the default borders and mill depths. However, the inside of the head is the only detail that will not be cut with another approach angle. Using a Manual Border Curve for the inside of the head would have eliminated the mold line seen in the picture on the right.

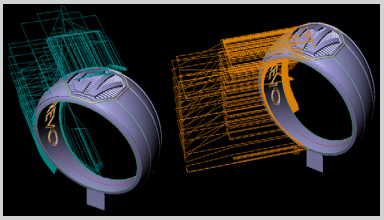


11 Detailed Options con't Sub Parts 1, 2, 3, & 4 are the traditional Front, Back, Left and Right approaches. Which are needed to mill the detail on the body of the ring,



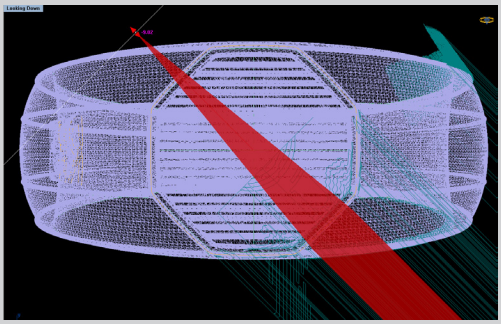
The Front, Back, Left and Right toolpaths will mill the detail on the ring. Two additional 45 degree angled cuts will be responsible for milling the text on the inside of the shank.

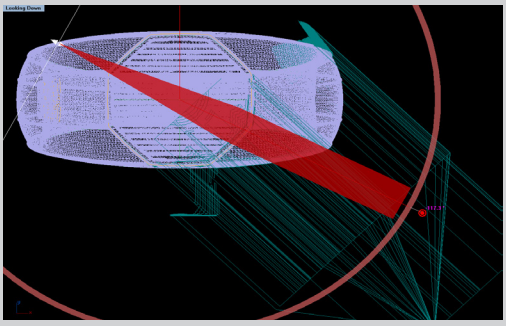
12 Detailed Options con't Sub Parts 5 & 6 approach the model at 45 degree angles on the left hand side of the ring as seen in the Looking Down viewport. These toolpaths are unnecessary. Click on Sub Part 5 and it's text will become highlighted. Click Delete (located above the list of toolpaths) to remove this path. The toolpath list will compress and the toolpaths on screen will disappear. Toolpath 6 just became toolpath 5. Delete this path . Part 5 should now be comprised of 6 sub parts. Click "Make Path" to view the changes.



Borders, Depth, Angle & Protect Finger Rail cont'd.

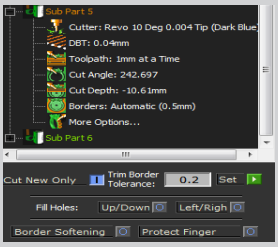
11 Detailed Options con't' - Depth Sub Parts 7 & 8, which are now Sub Parts 5 & 6 show the 45 degree cuts that approach from the right hand side as seen in the Looking Down viewport. These toolpaths are intended to mill the text on the inside of the ring shank. However, upon inspection, the toolpaths do not delve deep enough to reach the text. Double click on Sub Part 5 to expand the detailed options available for this sub part. Click on Cut Depth or Cut Angle (both options lead to the same place) to display the depth at which the cutter will plunge. Click "Go" to activate the Viewport Control Handles. A arrow, which controls the depth of the cutter and a doughnut, which controls the approach angle with appear. Hover the mouse over the arrow until it highlights red and click and drag the arrow until it goes deep enough to reach the text. It is best to make this change in the viewport you can view the doughnut shaped viewport control handle and easiest to gauge the depth of the text when viewing the model in wireframe.



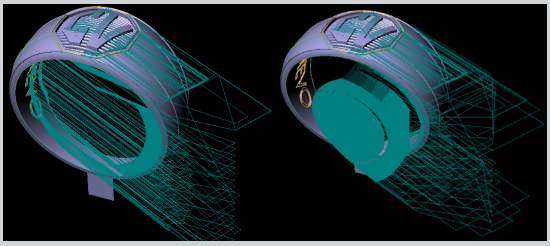


12 Detailed Options con't' - Angle Next, adjust the angle at which the cutter is approaching. Keep in mind, clearance around the right hand side of the shank is needed for the spindle and cutter. Make this adjustment using the doughnut shaped, angle viewport control handle. Is is best to approach as close to perpendicular to the detail as possible. After making this adjustment, it may be necessary to readjust the depth of the cutter. Click "Make Paths" to view the change.

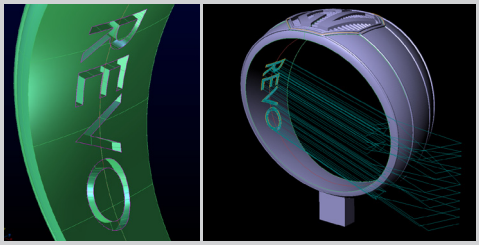
13 Detailed Options con't - Protect Finger Notice the new toolpath on screen appears to have a force field protecting the area in which we are trying to mill. This is evidence of the Protect Finger option which is intended to keep the geometry out of harms way from the side of the cutter. Click 'More Options" and toggle off the Protect Finger setting. Click Make Paths again to view these changes.

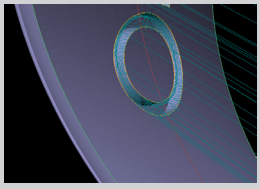


14 Detailed Options con't - Borders Although this path is better, it can still be improved. Remember in Step 10 when we confined the area in which the toolpath can cut? Let's apply the same concept here and ONLY mill inside the Text. This will save a lot of time and will give great results to the final wax. Click "Borders". Toggle "Manual Curve Border " to the on position. Input the text curves ,which were extracted in Step 2, into the Borders window. Click "Make Paths".



15 Detailed Options con't Repeat Steps 11-15 in Sub Part 6. Click Make Paths and once again verify the toolpaths. If everything checks out, click "Save Path". Save this to the "RVO" folder found in the "My Documents > Matrix" folder, or to any location on your computer where later you can transfer the file to a thumb drive to load onto the Revo Mill's computer.





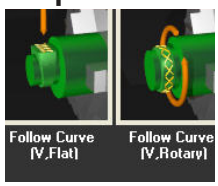
Note, if one border curve is within another border curve, the toolpath is created between the two curves.

Adding Parts & Sub Parts ...

"Follow Curve" Options

Three "Follow Curve" options add engraving-style cuts to any job.

About the "Follow Curve" Options



Choose one of the Follow Curve cuts to add engraving.

Three types of Detailed Options allow the user to add an engraving-style cut to a model. They work by accepting a curve for the cutter to follow and a distance beneath that curve for the cutter to sink into. If you consider the 10-degree cutter, you'll recall that it gets wider the farther up the tip you go. Without regard for the shape of the cutter (i.e. this cut is designed to cut into the surface the depth you tell it, without any "safety" features usually in place to prevent the cutter from gouging the surface), this cut allows you to sink the cutter into the part the depth needed to end up with the width of cut you want.

These cuts are especially useful for creating engraved lettering or designs, and can be performed one of three ways: during the flat side of a job on the Dual or Base Clamp fixtures (V and H spindles on the Base Clamp); during a rotary job, when the model is being turned on the A axis while the engraving cut is being made; and during a "flat on rotary" job, when the model is being held still on a user-selected angle on the A axis while the curve is cut (V or H spindles - H spindle only available during a non-ring model on the 3-jaw chuck).

Cutter will follow any curves at depth selected to "engrave" a pattern.

Adding a "Follow Curve" Cut

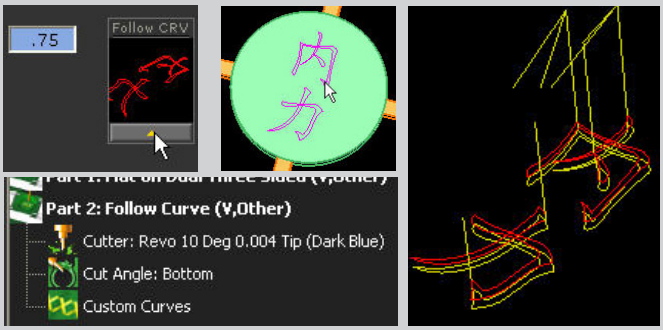
It is very easy to add a "Follow Curve" cut to an existing toolpath. For the selected strategy, enter the Detailed Options screen and click "Add". Select the "Follow Curve" cut available with the chosen strategy. It will be added where it makes the most sense in the part: a "Follow Curve on Flat" cut will be added following the other flat cuts on the selected (Dual, Base Clamp) fixture; while a "Follow Curve on Rotary" or "Follow Curve on Flat on Rotary" cut will be added following the other rotary

Three Types of "Follow Curve" Cuts

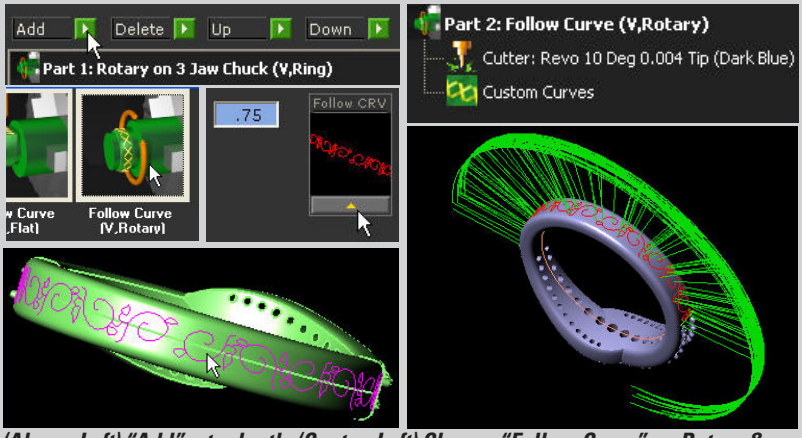
Follow Curve on Flat: Available for either the Dual or the Base Clamp Fixture, select this option and input the curve. Choose a depth beneath the curve to sink the cutter to when making the cut. Make CERTAIN you input the correct Cutting Angle in the Detailed Options for this cut. (See Note below.)

(Below, Left) Click "Add". (Right) choose "Follow Curve, V". (Far below, Left) For "Follow Curve Side", choose side of model on which curves lie: "Bottom", here.

(Below, Center and Left) Select curves and input them. Indicate a distance BENEATH the curve to cut (0.75 mm here). (Right) After "Make Path" (model hidden).



Follow Curve on Rotary: Available for either the 3-jaw chuck or the hub fixture, select this option, input the curve and indicate a depth for the cutter to sink to when following the curve to make this cut. The A axis will turn while the cut is made, so no "Angle" is offered.



(Above, Left) "Add" a toolpath. (Center, Left) Choose "Follow Curve" on Rotary & input curves selected (Below, Left). (Right) After "Make Path".

cuts - usually as the final option in the toolpath.

Related options appear beneath this cut. It is important to make certain that the Cut Angle (available for "Follow Curve on Flat" and "Follow Curve on Flat on Rotary") is set correctly. No "Cut Depth" will be available; "Cut Depth" will not be affected by inputting the Cutting Plane for the angle, as "Depth" is set when inputting the curves. To input the curves, select this option and select the curves on screen. Place them in the "In" box and type a depth beneath the curve for the cutter to sink while it is following the curve into the "Drop Depth Below Curve" text box.

Choosing a Cutter

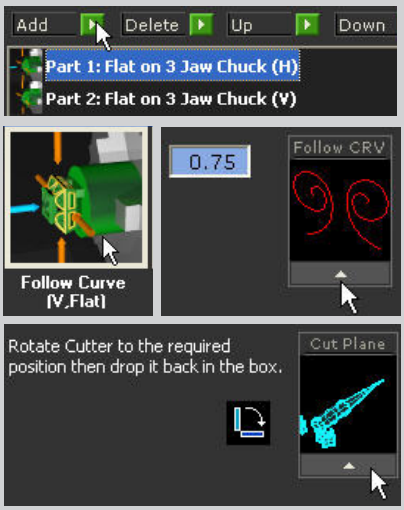
This depth will determine the width of the cut: the farther below the surface it sinks (for the tapered 10-degree cutter), the wider the cut. For the straight (0.032" cutter), the depth doesn't matter. Consider cut length when you are choosing this value, and do not sink a cutter that is too narrow (0.016" or 6-degree tapered) farther below the surface than its cut length can handle. It is best to do these cuts with the other two, more sturdy cutters.

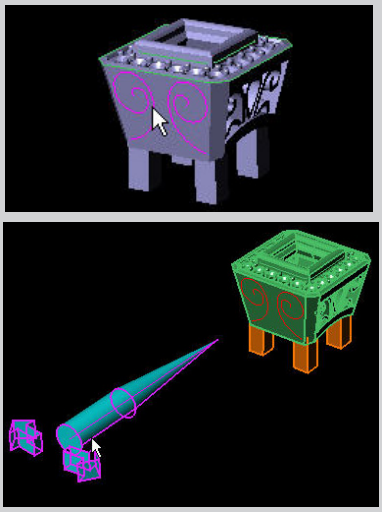
Some users even install an old, broken cutter, with a wider end on it, during this part to get a cut of the desired width. To do so, change the cutter (in Matrix) so the mill will pause before completing this cut and prompt you to change cutters. When prompted to by the mill, you would install the broken cutter, measure it as usual, and complete the cut using it, rather than the cutter you chose in Matrix.

Again: remember that this path does not consider the geometry of the cutter, like other passes do; it simply sinks the cutter currently installed on the mill below the surface of the wax to the depth indicated, following the shape of the curve you input into the Detailed Options menu for this cut.

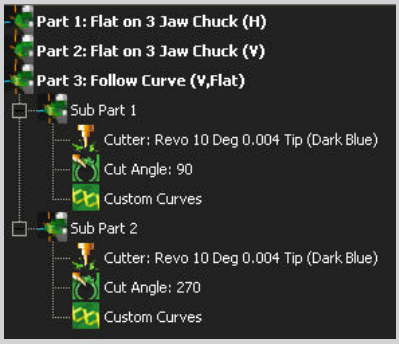
Three Types of "Follow Curve" Cuts, cont'd.

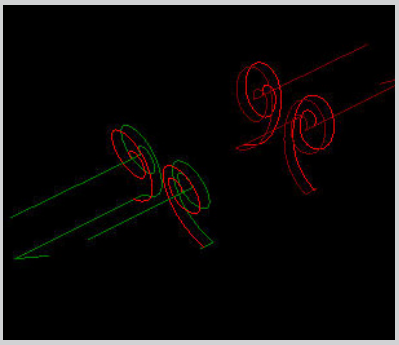
Follow Curve on Flat on Rotary: Also available with either rotary fixture, select this option, input the curve, and indicate a depth for the cutter to sink to when following the curve. Make CERTAIN you set the cut angle at which the A axis will hold the fixture still while making this cut. (See Note below.)





(Above, Left) Click "Add". (Center, Left) Choose "Follow Curve" on Flat, Vertical cutter. Input selected curves ON CORRECT SIDE (Above, right) and indicate a depth below curve (Center). Select Cutter Angle option and rotate cutter (Right) to correct angle. Input cutting angle. (Below, Left) Repeat all steps for opposite side of head: Cutting Angle 270, below. (Right) After "Make Path" (model hidden).





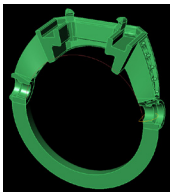
Note: When using the Horizontal Cutter "Follow Curve on Flat (H)" for the Base Clamp and "Follow Curve on Flat on Rotary (H)" for the 4-sides plus top strategy (used for non-ring models on the 3-jaw chuck) , you will NOT be able to set a Cut Angle. Only one angle - horizontal - is possible, and this is set automatically.

Adding Parts & Sub Parts...

Split a Ring to Hollow

Use the Split strategy to mill a ring model in two halves. Next, melt the halves together and finish milling the ring on the Hub fixture.

Prepare the Model in Matrix



One half of the model with all exposed surfaces closed.

Use the tools in Matrix to divide the model into two halves as seen in the Looking Down or Side viewports. Using the Plane & Cube cutter found in the Transform menu place a plane so it divides the ring along the X axis. Toggle Off all gemstone in the Layers menu. Split the ring with the Plane. Select the entire ring and run the Cap Planar function within the Solid menu. After this process is complete, when the two halves are placed back together, the flat surfaces will mate perfectly together to complete the model. Take note to close any open surfaces that may not have capped when using the cap planar command. All of the exposed interior parts should be capped so when the cutter approaches it see's a surface. Use the Group command to Group both halves. This will make it easier to select each half or the ring later when asked to do so. Make sure to include the appropriate size ring rail. You are now ready to enter into the Mill Builder.

Creating the Toolpath

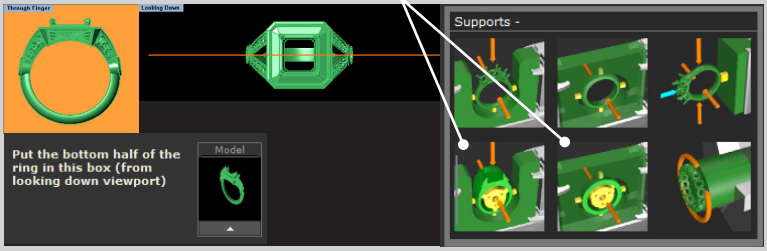
On the first screen, input both halves of the ring into the model window. Choose one of the two Split Ring strategies depending on where the detail on the model is located. Upon placing the model into the model window, the on screen supports will appear to support the top half of the ring, as seen in the Looking Down viewport. Follow the on screen instructions and place the bottom half of the ring into the lower Model window. The on screen suggested wax size will shift to the top half of the model. Click Next. In the next few screens choose the wax size, approach angle, create a hub and create the toolpath. Take note of Part 2: Flat on Dual Three Sided(V,Ring). Zoom in to see a 2 X .5 mm flange has been added to the outside of each half of the model.

Create the Split Ring toolpath in the Mill Builder

1: Prepare Model
In Matrix,prepare the model by splitting it into two halves as seen in the Looking Down viewport. Close any open surfaces the cutter will see when it approaches the model on the Dual fixture.

2: Split Strategy
Place the model into the Model window and choose one of the Split strategies. A second model window appears. Place only the bottom half of the ring into the lower model window.

3: Make the Toolpath
Proceed through the remaining screens in the Mill Builder. Choose the wax size, approach angle, create the hub and make the toolpath. Run the toolpaths in the two sides of the Dual fixture.



Mill the Split Ring toolpath on the Revo C

1: Dual Fixture
Run both halves of the model in Side 1 and Side 2 of the Dual Fixture. Rinse both halves.

2: Melt the Halves
Place both halves onto the Hub Fixture. Use a wax pen and melt the flanges together.

3: Hub Fixture
Mill only the rotary toolpath on the Hub fixture to complete the ring design.

Milling the Toolpath on the Revo 540C

Save the toolpath and transfer it to the Revo Mill. Click the Load File Action button. Load the first toolpath into Revo File 1, Side 1 on the Dual fixture. Load the second half of the toolpath into Revo File 2, Side 2 of the Dual fixture. Run the Toolpath. Thoroughly clean both halves and free them from the wax blocks. Slide both halves of the ring onto the Hub Fixture. Use a wax pen and melt to two halves together. Take care not to drip any wax down the side faces of the Ring. The rotary path will only clean the outer face of the ring design. Once again, load the toolpath onto the Revo C mill. Uncheck all load parts up until Job 3: Vertical on Fixture 3 (Center Hub). Run the final Rotary toolpath and marvel at the results.

Adding Parts & Sub Parts ...

Saving & Loading Styles

After making a toolpath, you may "Save" a style by choosing this option in the Detailed Options screen. "Load" it in the future to re-use all Mill Builder settings.

Saving & Loading Styles

If you wish to re-use any of the Builder settings established in the Mill C Builder for a future project, including the Detailed Options, the "Save Style" and "Load Style" functionality in the Detailed Options screen of the Mill C Builder will help you do just that.

When you've set up a project just the way you want it, click the "Save Style" option in the Options list at the top, right-hand corner of the Detailed Options screen. Click this to save ALL the builder settings.

To load these settings for a new project, enter the Mill Builder, input the new model into the Model in-box FIRST, and then select the "Load Styles" command in the first screen of the builder. The builder menu will be populated with all the options you established when the Style was saved.

Uses for Style Files

There are two main uses for Style Files. First, when you've run a project on your mill and decided you'd like to try it again with some different options, you can load the Style File for the SAME model and make changes to it to see if they improve the results in the wax model.

Next, you can input a similar model and use the same settings for it. File type when saving a Revo C style is ".RVSC". This is the only file type in which you can save a Revo C Style file using the "Save Style" button.

Just take care to go through each builder screen thoroughly and make certain the same settings work for the new model. You can edit them if they do not; however, "Save and Load Styles" gives you a shortcut to a great starting point for adding like options to similar models.

File Types for Styles: .RVSC & .RVOC

The file type when saving a Revo C style is ".RVSC". This is the only file type in which you can save a Revo C Style file using the "Save Style" button.

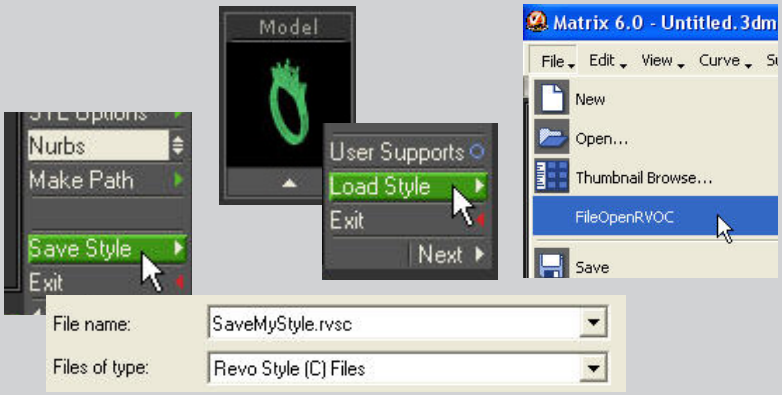
Steps for Saving & Loading Revo C Style Files

1 - Set Builder Options
Begin by making sure all the Builder Options are the way you want them for this project.

2 - Click "Save Style".
In the final screen of the Mill C Builder (Detailed Options), click "Save Style". Navigate to a location on your computer to save the style file (.RVSC) and type a file name. Click "Save".

3 - "Load Style".
To retrieve these settings, input the same model or a new model into the first screen of the Builder and click "Load Style" in this screen. This loads the Style file, which populates all the builder settings with the settings which it had when the RVSC file was saved. IMPORTANT: Double-check these to make certain they work for the new model.

4 - Or, "Open .RVOC"
Alternately, select "Open RVOC" from the gray "File" dropdown menu in Matrix. This both opens the .3dm file in Matrix AND populates the Mill C builder when you click Load Style on the first page of the Mill Builder Style file settings. You can Job Bag or Save your Matrix Model (it isn't lost, don't worry) and / or tweak your settings in the Revo C builder.



(L) Choose "Save Style" to create an "RVSC" file that saves all the settings used to create the RVOC file in the builder. (Center) AFTER inputting a new (or the same) model, you may "Load" the style in the first screen of the builder. (R) Click "File" dropdown menu and choose "Open RVOC" to open both 3DM in Matrix and Style file in Mill C Builder.

However, the .RVOC file that the software creates when you click "Save Path" actually CONTAINS the Style File. To load a style from a .RVOC file, click the "File" dropdown menu in Matrix (from among the gray dropdown menus along the top of the screen) and choose "Open RVOC. This will load the Matrix file (.3dm) on screen and will populate the Mill C builder with the Style File - or, all the same settings you established in order to save the RVOC file in the first place.

This is also a very handy tool when you've forgotten to Job Bag or save your Matrix file, since you can now retrieve the 3DM from the RVOC file.

Editing Detailed Options

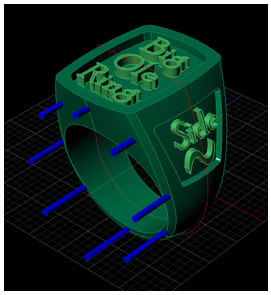
"Inside and Outside Curves"

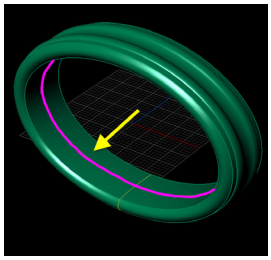
Ensure details on your model are cut perpendicular to the surface.

Cleaner Cuts

Another selection in Detailed Options for a Rotary tool path allows the user to add defining curves that can better guide the cutters as they carve the surface of the model. These work especially well for items like Signet rings with sharp corners or side detail in their design.

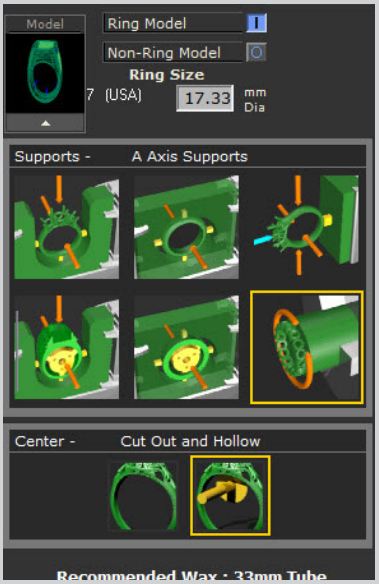
It also offers inside borders for items such as bangles that don't possess ring rail curves that would normally help to define cutter depth.



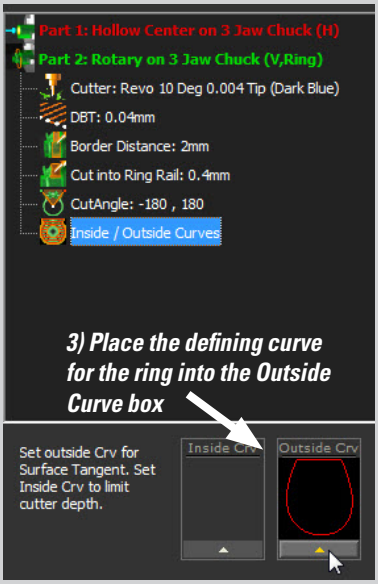


Adding Curves to the Tool Path ...1-2-3

1) When creating the Tool Path, select the Rotary Strategy with Center Cut Out and Hollow.



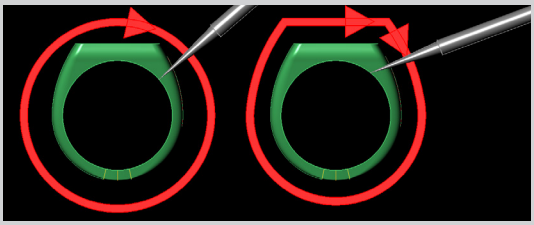
2) Click Next to reach the Parts list, then double click Part 2 to access the Inside /Outside detailed option



3) Place the defining curve for the ring into the Outside Curve box

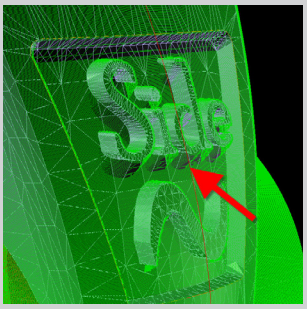
Outside Curves

As shown in this model, when milled with a standard rotary toolpath, the cutter mills the surface as the object slowly rotates on the hub, the center of rotation being the center of the ring rail. While this typically produces excellent surfaces, it may result in shadowing effects on items that have non radial detail. As seen in the images, the side and top detail display shadowed effects.

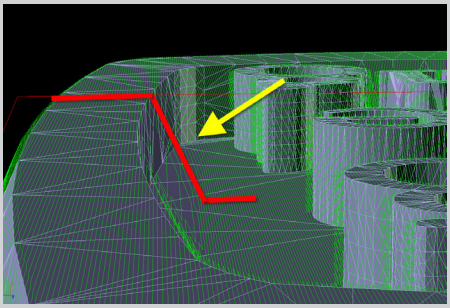


With the addition of an outside curve drawn to mimic the shape of the ring, this curve becomes the guide for the cutter during the rotary tool path. Add the curve to the detailed options under Inside/Outside Curve, Create Path, then see the vast difference this feature brings to the rings surface.

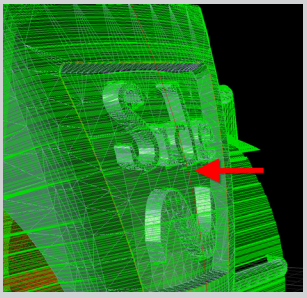
As the cutter approaches the surface perpendicular to the outside curve, it is able to navigate steep corners and sharp edges far more effectively.

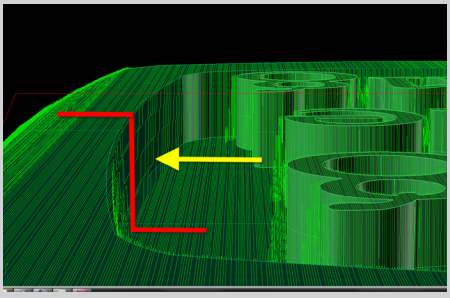


The base of the letters angle or 'shadow' because of the angle of the cutter to the surface.



In this example the inside wall of the top (flat) surface angled as well, where a straighter edge would be preferred.

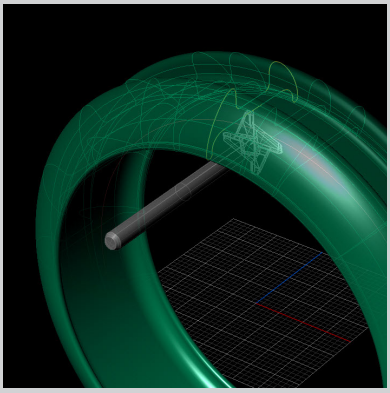




After Adding the Outside Curve to define the shape of the signet ring, the cutter follows perpendicular to the outside curve to achieve cleaner, straighter edges.

Tutorial 1: Milling a Bangle Bracelet

Explore the considerations and limitations of milling a large bangle bracelet on the Revo 540 C mill



Begin with a Bangle, built with consideration for cutter dimensions - to ensure an efficient and effective milling process. Step through the strategies and wax options - using various toolpaths and creative uses of mill fixtures to achieve a finished wax model requiring minimal clean up.

Materials Needed

Matrix Menu

- Solids: Cylinder

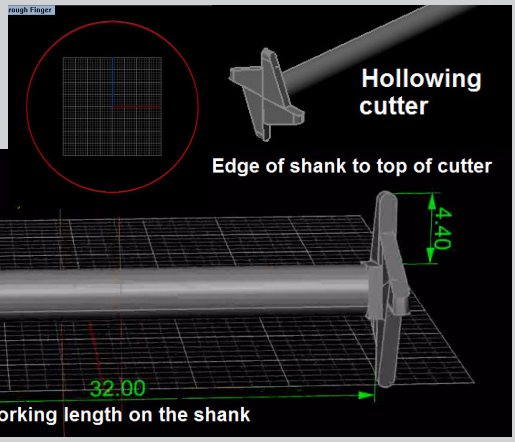
Matrix Mill C Builder:

- "Cutters" Library

- Cutter Selection (final Mill Builder screen or Detailed Options screen).

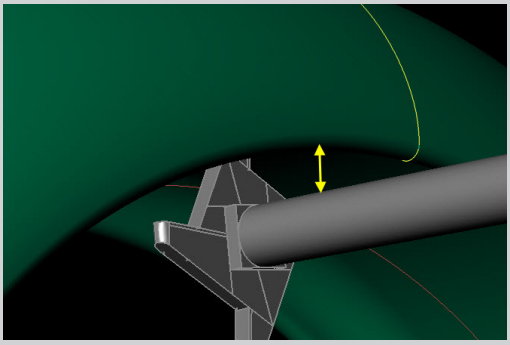
1 Building the Bangle

For this tutorial, the diameter of the bangle is set for 70mm and is created in the Through Finger viewport using the Circle tool. For this bangle , you may prefer to build your own profile to create the surface. However, the cutter dimensions will need to be considered when determining the size of the profile.

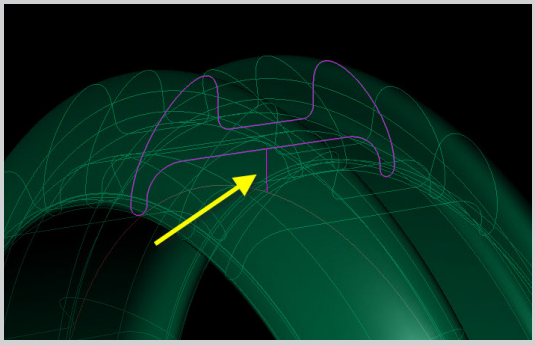


2 Cutter Dimensions

As you draw your profile, the largest width measurement will be limited by the dimension of the cutter. For a bangle that will be carved by a hollowing cutter, be aware that the "working" length of its shank is 32mm. This dictates the width limitations of the bangle. The hollowing depth is also determined by the distance from the edge of the shank to the outer tip of the cutter (4.4mm). The profile is shown highlighted here. Note the curve underneath.

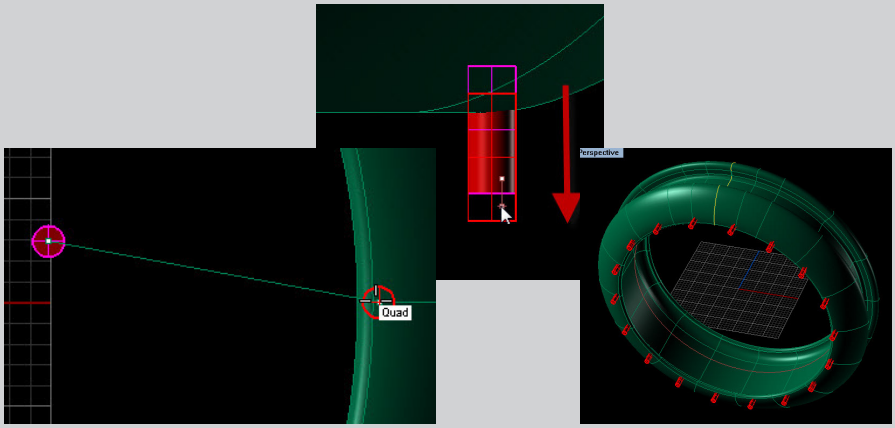


This straight curve was snapped to the top quad point of the rail, then draw at 3.5mm length as a guideline to ID the distance needed between the base of the hollowed profile and the rail.



3 Supports Added

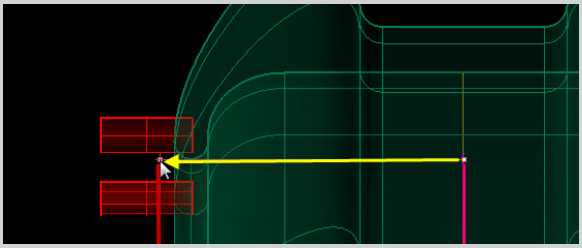
This process remains a manual one as model designs and support needs vary so dramatically. For this example a single cylinder is created using the Cylinder Tool in the Solids menu. It measures 1.5mm in diameter and 4mm long. It is snapped to the Quad Point of the rail curve, then adjust the support slightly away from the curve and out from the surface edge. Then polar array the cutter x 16. Supports should be created and placed at the front of the model. Or on the LEFT side - as seen from the Side view.



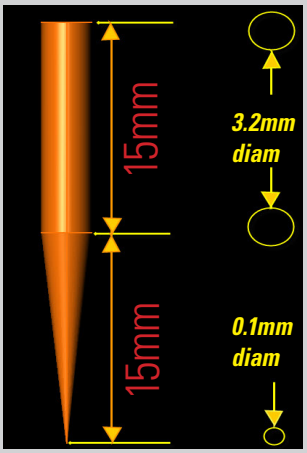
Milling a Bangle Bracelet cont'd

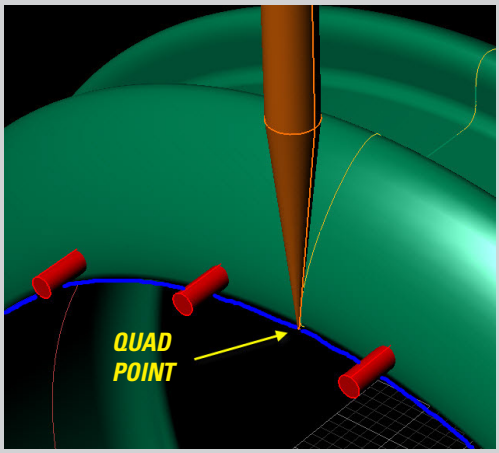
4 Add a Curve for Clean up with Toolpath

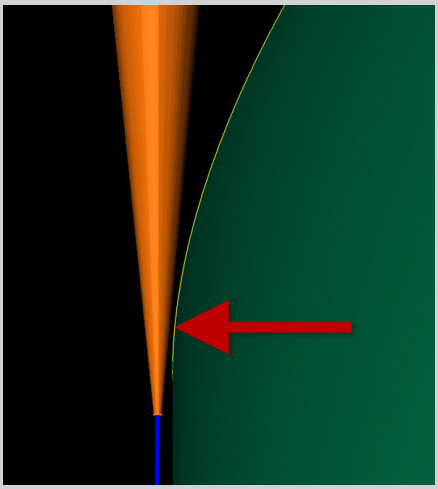
A clean way to remove the supports at the end of the milling job, is to create and place a curve in just the right location. The easiest way to accomplish this is to make a copy of the rail curve and place it in the same vicinity as the supports. Change the curve color to User 03.



NOTE: The location of this curve may be precisely placed with the use of a cutter model. A replica of the 10 degree cutter may be created by drawing three different circles in the dimensions and distances shown to the right. Loft the curves together and cap the ends, then position the cutter on the curve. The tip of this model should be snapped to the top quad point of the curve, then lined up against the surface of the band.

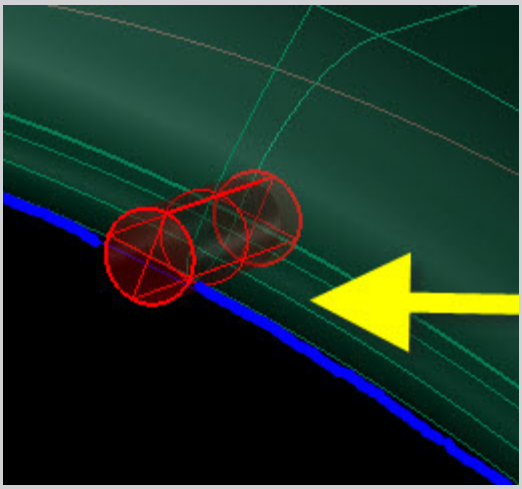






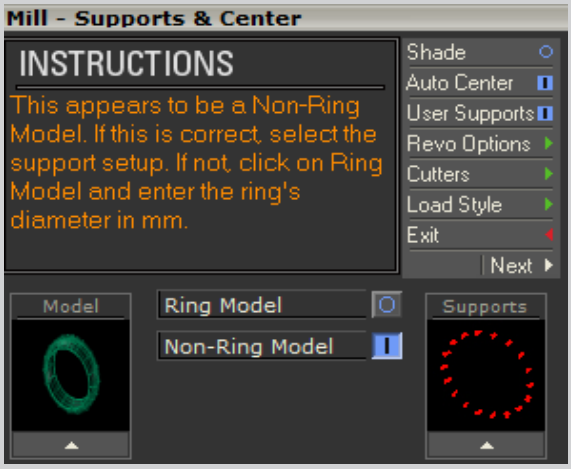
Then In the Side View, adjust the curve AND cutter together so that the edge of the cutter rests very close to the edge of the band's surface. Because the supports rest just above the curve, they will be cut off during the last portion of the milling process. Job Bag it.

With the curve in place, the 'mock cutter' is deleted and the toolpath may be created.



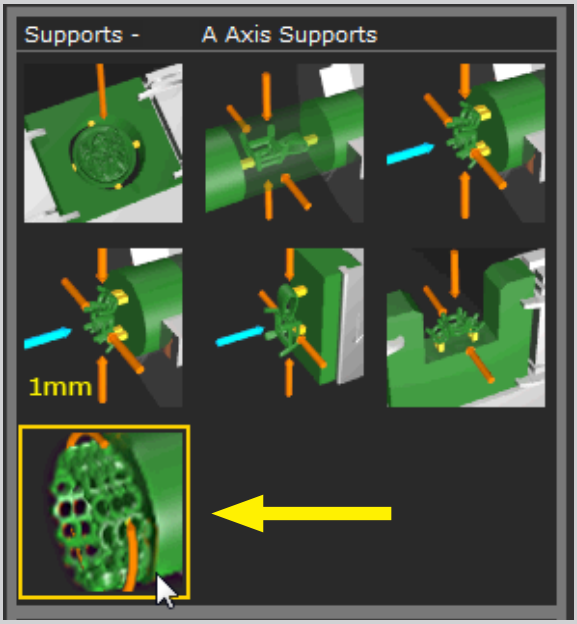
5 Place model into the Revo C Mill builder

When you do so, choose the Non-Ring Model option. Then select the supports and place them into the Supports window.

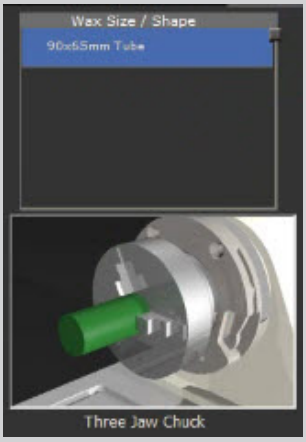


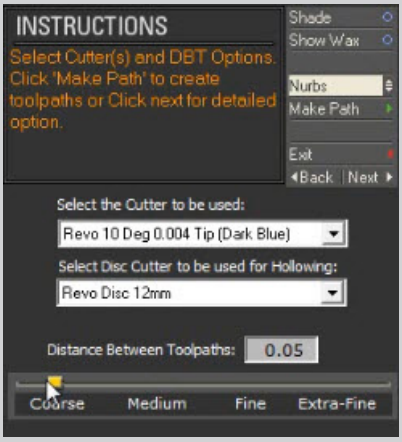
Milling a Bangle Bracelet cont'd

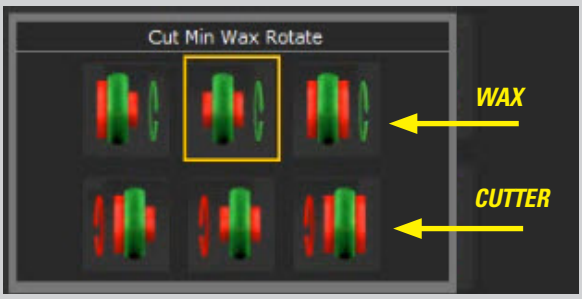
6 **Select from Supports Menu** From the A Axis Supports menu, select the Bangle support option.



7 **Verify Wax & Cutters and Set DBT** The wax size should populate as a 90x65mm Tube, requiring the Three Jaw Chuck fixture. Click Next. Two cutters will be required (10 Degree and 12mm Disc). For a simpler design with less exterior detail, setting the Distance Between Toolpaths (DBT) at .05 or 'Coarse' can eliminate 30% off the total milling time.



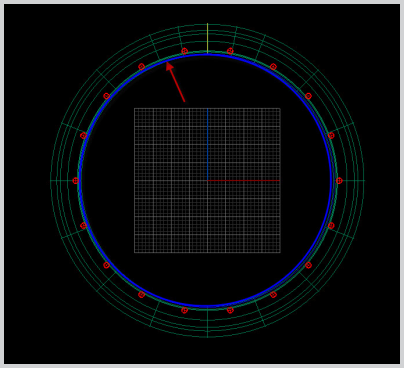


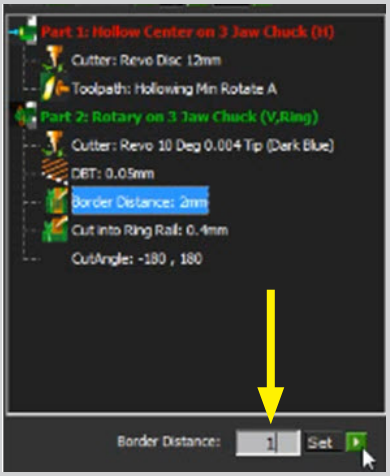


8 **Hollowing Options** For the Hollowing step, there are six options to choose from. The arrows by the graphics signify which object is in motion. The upper selections indicate the WAX moving while the lower choices indicate the CUTTER in motion. Using the lower options for a large bangle such as this might cause a moving cutter to run into the table of the mill. The upper options which signify the WAX moving will accommodate very large pieces (up to 150mm in diameter) and are the better choice for this project.

What does the RED color signify? Larger areas of red indicate longer milling time to accommodate more detail. Smaller areas indicate little or no detail. This bangle has minimal detail on BOTH sides, therefore the Min Wax Rotate option is selected.

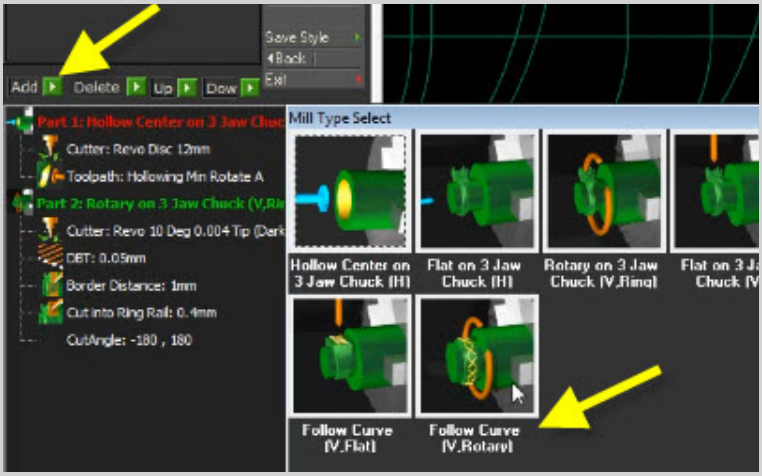
9 **Border Distance** Typically the Revo would mill 2mm beyond the border of the object. In this case however, a 'cut off curve' has been established to clean off the supports at the end of the milling process. With this in place, you can save milling time selecting the Border Distance option in **Part 2** and changing to distance from 2mm to 1mm. Click Set.

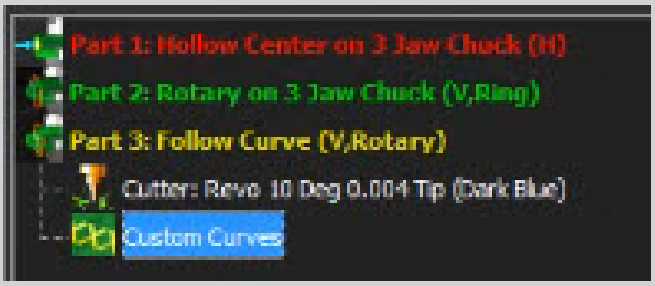




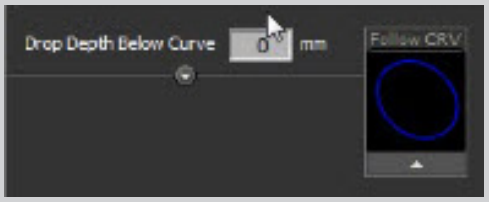
Milling a Bangle Bracelet cont'd

10 Add Follow Curve The last step of your mill strategy is to add the defining curve that will guide the cutter to remove the supports for the bangle. Click the Add button and choose *Follow Curve (Vertical Rotary).* **Part 3** is generated for the tool path. Add the defining curve by clicking on the Custom Curves option. Select the cut off curve from the view port and insert it into the window. The *Drop Depth* below curve must remain at a '0' value so that the cutter does not infringe on the bangle's surface.

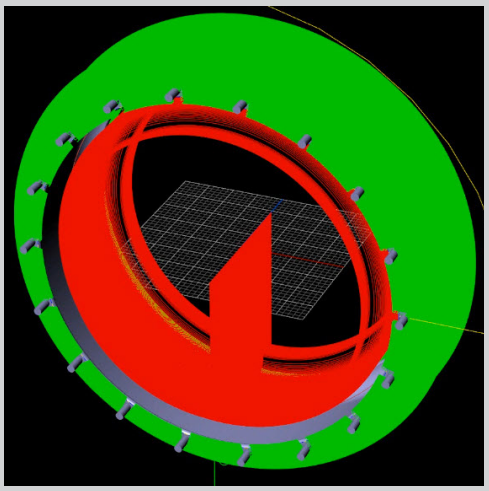




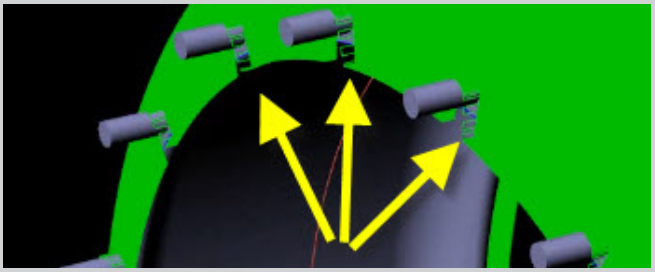
NOTE: This technique can also be used for oval and non-circular bangles - as long as you have a way to trace the 'edge' curve.



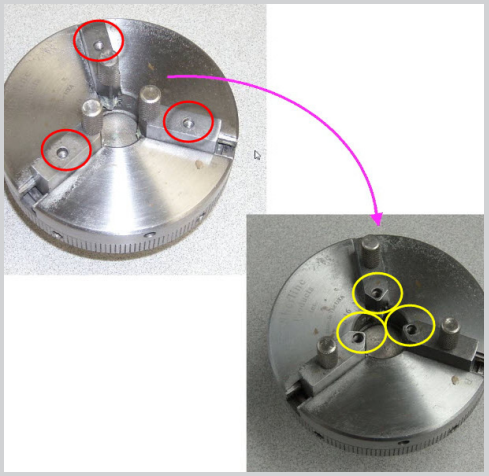
Once these steps are complete, click Make Path.



11 Check Tool Path To the right is an example of the generated toolpath from the RevoC Mill builder. You will want to ensure that there are 'shadowed' areas below each of the support. This will signal that the supports will remain in tact throughout the milling process until the Part 3: Follow Curve removes them.

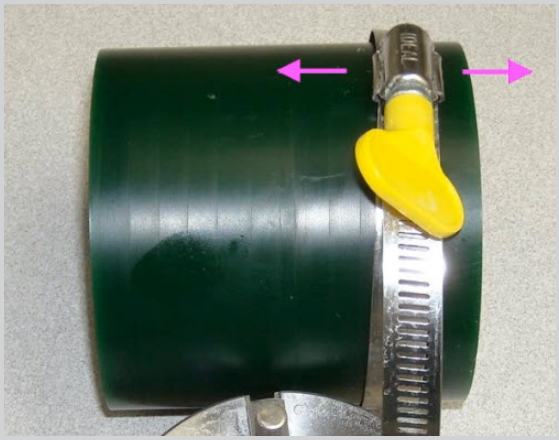


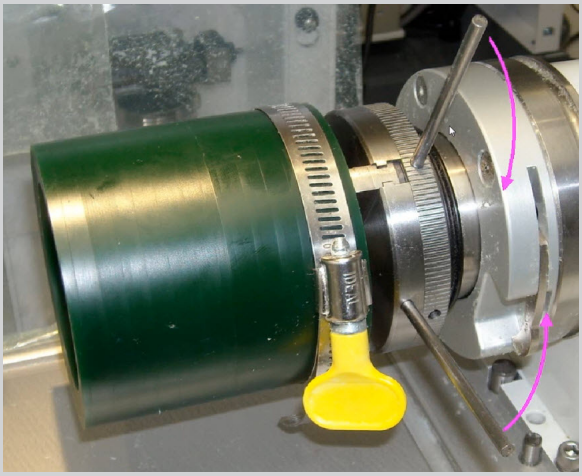
12 Setting up the 3 Jaw Chuck in order to use the 3-Jaw Chuck on a large scale bangle of this kind, a modification must be made. The 3 clamps of the fixture must be changed. The 3 screws that hold the clamps in place must be moved from their inner threaded positions to their outer threaded positions. This allows the 3-Jaw Chuck to grasp the large, tubular wax from the inside of the piece.



Milling a Bangle Bracelet cont'd

13 Adding a Clamp The next step is to add a pipe clamp to the wax. The item pictured here is a standard clamp for an electrical vent - found in most hardware stores. Automotive clamps work fine as well as long as they can manage a 95mm diameter area. The purpose of the clamp is to provide a balanced mechanical support to the 3-Jaw Chuck. Without the clamp on the outside of the wax, you may run the risk of cracking or splitting the tube. Be sure it's attached near the end and simply finger tightened.



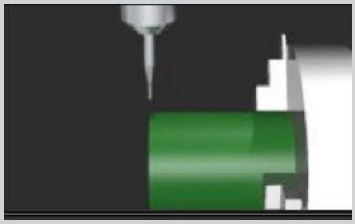


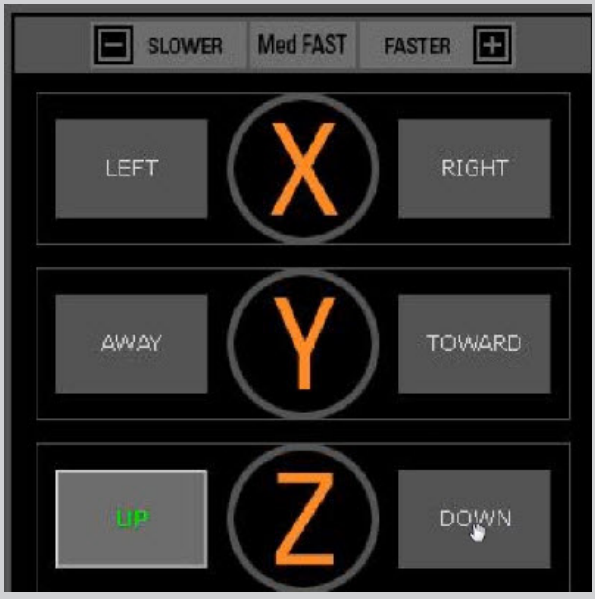
14 Following the Steps As you load the job file and begin to step through the instructions, follow them up through Step 21 and attach the 3-Jaw Chuck fixture.

Attach the wax to the fixture and use the dowel pins to tighten. Note the direction of the arrows to tighten the pins. It is different than normal since you are adjusting to the *inside* of the wax rather than the *outside*. Be sure that it is nice and tight.

Again, because the band has been attached to the outside of the wax, it will protect it from breaking.

15 Adjust the Cutter Position Now you may use the DOWN, LEFT and RIGHT jog controls to move the cutter into position so that it rests at the top edge of the wax. Click Set and the mill will automatically move LEFT - back into the Start position which is Step 26. The first phase is the hollowing phase and will be accomplished with the 12mm disc cutter. NOTE: Should you receive a message that the model is too large the problem is the wax you are using is too long. You will need to use the Windows Task Manager to exit from the Revo software, then trim our wax and reload the tool path.



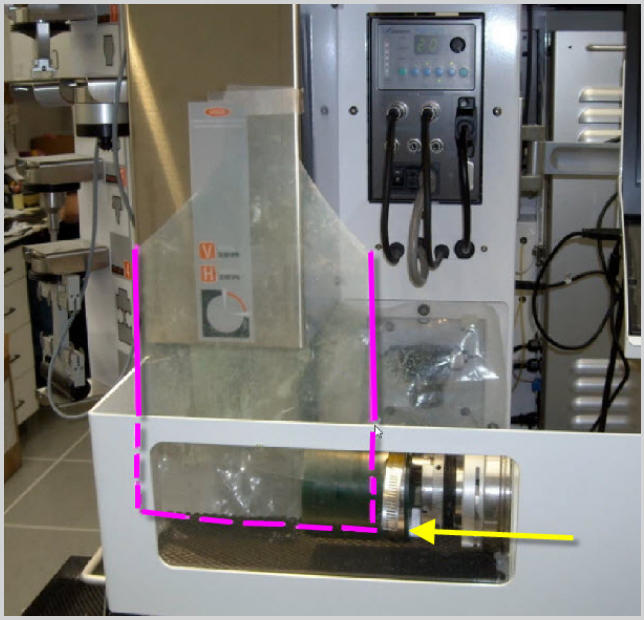


Milling a Bangle Bracelet cont'd

ABOUT COOLANT: When working with a large model like a bangle, it is important that you turn the coolant pulse rate and duration rate all the way up to ensure you have enough lubrication.

With a high rate of coolant applied to this large piece, a great deal of splash over can occur. We strongly recommend you attach some plastic to the outer wall of the steel shroud to prevent this. See image to the right.

Note that the plastic spray shield hangs down inside of the splash guard. This will prevent a great deal of liquid from spilling out onto your work room floor.



Gemvision

REVO 540C

Be sure to visit
www.gemvisionforum.com
to connect online with hundreds of other users and Gemvision to share information and resources. You'll also have access to tutorials, tips, news, and announcements direct from Gemvision as soon as they become available!

REVO TECHNICAL SUPPORT:

toll free call:
888.357.6272
email:
revo@gemvision.com

Gemvision

Price $50.00