EM69HP CNC
MACHINING CENTER
MACHINE OPERATIONS MANUAL

8029 S 200th Street, Kent, WA 98032 USA | www.rottlermfg.com | Ph: (253) 872-7050 | Fax: (253) 393-0230
Rev 092621

PARTS ORDERING

For optional equipment catalogs, please visit https://www.rottlermfg.com/documentation.php

For fastest service ordering parts or equipment, contact us via e-mail with the information below. For customers within the U.S., send emails to parts@rottlermfg.com, for customers outside of the U.S., use intlparts@rottlermfg.com

Have the following information on hand to expedite the ordering process:

1. Your name, business name, and contact number
2. Customer number, or your billing address if you do not have a customer number
3. Shipping address if different from the billing address
4. Machine model and serial number
5. Part number and description of the item(s) to order
6. Preferred method of shipment

For customers outside of the U.S. requiring faster service, contact your local distributor.

In some cases, you may be requested to send a photo of the part you are ordering if it is a replacement part or does not appear in our database.

If you are unsure which part you need to order, contact our service department, and ask to speak to one of our service consultants. They will assist you in determining which part(s) you require.

THERE IS A MINIMUM ORDER OF $25.00

MANUAL SECTIONS

INTRODUCTION

SAFETY

CONTROL DEFINITIONS

OPERATING INSTRUCTIONS

8029 S 200th Street, Kent, WA 98032 USA | www.rottlermfg.com | Ph: (253) 872-7050 | Fax: (253) 393-0230
Rev 092621

INTRODUCTION

Contents

Introduction

READ THE SAFETY CHAPTER BEFORE INSTALLING MACHINE. THOROUGHLY UNDERSTAND ALL SAFETY ISSUES BEFORE OPERATING MACHINE.

ATTENTION OWNER/BUSINESS MANAGER

To validate the warranty on your new Rottler machine, please be sure to sign and complete the "Installation Report" located in the Installation Chapter of this manual.

We suggest that the new user of the EM69HP read the CONTROL DEFINITIONS to get an idea how the machine operates.

The Operating Instructions chapter should be read in order to familiarize the user with the actual button pushing sequences required to carry out a job. These chapters in the manual should be considered an introduction. As the operators of the EM69HP series machines gain experience with using the different functions of the machine, complicated setups and programs will make more sense.

The rest of the manual contains information and part number reference on fixtures, cutting tools, and machine maintenance. The operator should read and become familiar with these areas as well.

Description

The Rottler EM69HP is a 5-axis CNC machine designed and developed specially for porting cylinder heads. The design of the machine allows the center of the "ball shaped" cutting tool to rotate about its own center on the 5th axis.

This unique design has many benefits to a performance engine builder of which the most important are ease of programming and the fastest and most accurate method to port cylinder heads.

Disclaimer

The EM69HP Manual (henceforth to be referred to as the “Manual”) is proprietary to Rottler Manufacturing LLC. (“Rottler Manufacturing”) and no ownership rights are hereby transferred. No part of the Manual shall be used, reproduced, translated, converted, adapted, stored in a retrieval system, communicated or transmitted by any means, for any commercial purpose, including without limitation, sale, resale, license, rental or lease, without the prior express written consent of Rottler Manufacturing.

Rottler Manufacturing does not make any representations, warranties or guarantees, express or implied, as to the accuracy or completeness of the Manual. Users must be aware that updates and amendments will be made from time to time to the Manual. It is the user’s responsibility to determine whether there have been any such updates or amendments. Neither Rottler Manufacturing nor any of its directors, officers, employees or agents shall not be liable in any manner whatsoever to any person for any loss, damage, injury, liability, cost or expense of any nature, including without limitation incidental, special, direct or consequential damages arising out of or in connection with the use of the Manual.

Rottler Manufacturing and its employees or representatives are not responsible for any information regarding final specifications of any workpiece that is created as a final product when using Rottler equipment. It is the responsibility of the end user of Rottler equipment to determine the final dimensions and finishes of the workpiece that they are working on. Any information regarding final dimensions and finishes that appears in any Rottler literature or that is expressed by anyone representing Rottler is to be regarded as general information to help with the demonstration of or for operator training of Rottler equipment.

Limited Warranty

Rottler Manufacturing Company Model EM69HP parts and equipment is warranted as to materials and workmanship. This limited warranty remains in effect for one year from the date of installation or two years from the date of the original shipment from Rottler or whichever date occurs first. This only applies is the machine is owned and operated by the original purchaser and is operated and maintained as per the instructions in the manual. A machine is warranted only if the Installation Report has been properly executed by a certified installation person and received by Rottler at the time of actual installation.

The products are warranted upon delivery to conform to their published specifications and to be free from defects in material and workmanship under normal use for a period of one year from shipment. Should a product not be as warranted, Rottler sole obligation shall be, at its option, to repair, correct or replace the product or to refund the amounts paid for the Product upon its return to a location designated by Rottler. No warranty shall extend to rapid wear Products (including tooling) or to Products which have been subject to misuse (including any use contrary to Rottler instructions), neglect, accident (including during shipment), improper handling or installation, or subject to any modification, repair or service not certified by Rottler. Rottler shall not be liable for any consequential, direct or indirect damages or for any other injury or loss. Buyer waives any right, beyond the foregoing warranty, to make a claim against Rottler. No warranty is provided for any Products not paid in full.

Merchandise cannot be returned to Rottler without prior approval. Customer must contact the Parts Department to get approval and to be issued a Return Goods Authorization number **(RGR#).** Merchandise authorized for return must be returned prepaid. If merchandise is returned with shipping charges collect, the actual amount of these charges may be deducted from any credit which may be due the customer. The **RGR #** assigned by the Parts Department should be written on the shipping label and must appear on a copy of the invoice(s) covering the original shipment. This invoice copy must be included in the box with the parts. Shipment must contain ONLY those items on the **RGR** as approved for return. Merchandise must be received within 10 days of the date of **RGR** or the **RGR** will be

canceled. All returned merchandise may be subject to a 20% restocking fee on under $1,000.00 amount or 10% on any items over $1,000.00. Parts or tooling over 30 days old are considered as customer property and can only be returned with prior approval from Rottler Corporation Management.

The issuance of a **RGR DOES NOT** guarantee credit - it is only authorization for the return of the goods. Credit for return merchandise is at the sole discretion of Rottler. Credit will be issued only after inspection of returned goods.

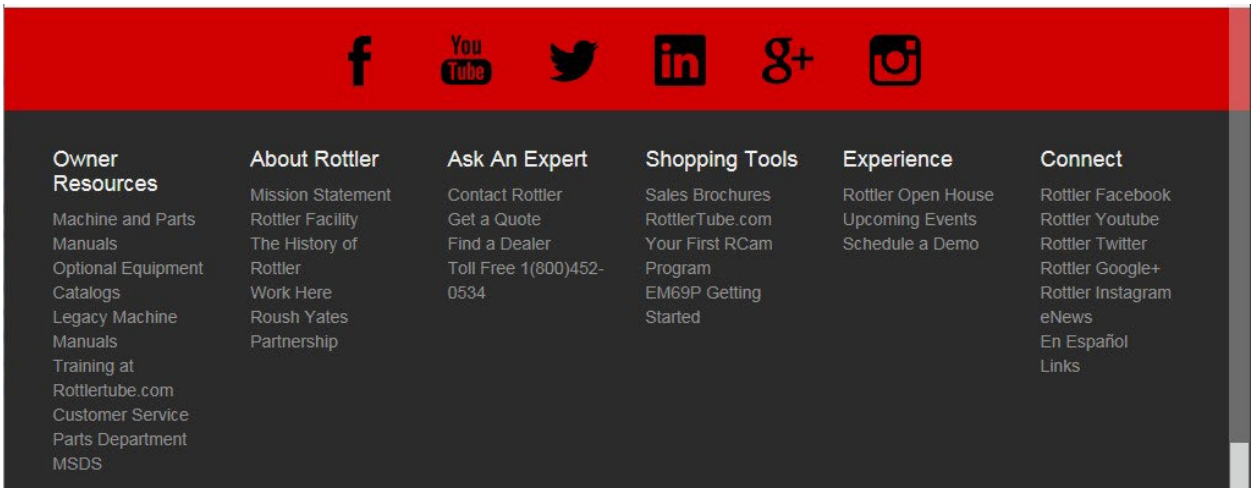
Tools proven to be defective within the warranty period will be repaired or replaced at the factory's option.

We accept no responsibility for defects caused by external damage, wear, abuse, or misuse, nor do we accept any obligation to provide compensation for direct or indirect costs in connection with cases covered by the warranty.

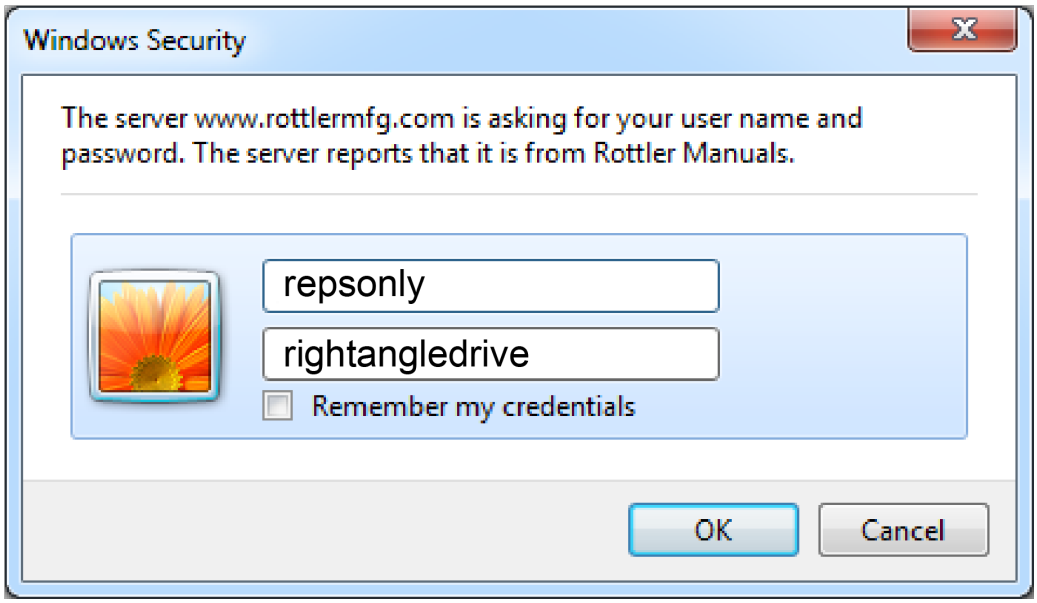
Online Documentation Access

Online documentation for machines and optional equipment can be accessed at the Rottler website. To access documentation open your browser and navigate to https://www.rottlermfg.com.

Scroll to the bottom of the page and under the Owner Resources title click the type of documentation you want to access.



If a log in window pops up asking for user name and password fill in the blanks as shown.



This page intentionally left blank

I

SAFETY

Contents

Safety Information

For Your Own Safety Read This Instruction Manual Before Operating This Machine.

This is the safety alert symbol. It is used to alert you to potential personal injury hazards. Obey all safety messages that follow this symbol to avoid possible injury or death.

DANGER indicates an imminently hazardous situation which, if not avoided, will result in death or serious injury.

WARNING indicates a potentially hazardous situation which, if not avoided, could result in serious injury.

CAUTION indicates a potentially hazardous situation which, if not avoided, may result in minor or moderate injury.

CAUTION used without the safety alert symbol indicates a potentially hazardous situation which, if not avoided, may result in property damage.

Safety Instructions for Machine Use WARNING This machine is capable of causing severe bodily injury

ONLY A QUALIFIED, EXPERIENCED OPERATOR SHOULD OPERATE THIS MACHINE. NEVER ALLOW UNSUPERVISED OR UNTRAINED PERSONNEL TO OPERATE THE MACHINE. Make sure any instructions you give in regards to machine operation are approved, correct, safe, and clearly understood. Untrained personal present a hazard to themselves and the machine. Improper operation will void the warranty.

KEEP GUARDS IN PLACE and in proper working order. If equipped with doors, they must be in the closed position when the machine is in operation.

KEEP WORK AREA CLEAN. Cluttered areas and benches invite accidents.

KEEP CHILDREN AND VISITORS AWAY. All children and visitors should be kept a safe distance from work area.

WEAR THE PROPER APPAREL. DO NOT wear loose clothing, gloves, rings, bracelets, or other jewelry which may get caught in moving parts. Non-Slip foot wear is recommended. Wear protective hair covering to contain long hair.

ALWAYS USE SAFETY GLASSES. Also use face or dust mask if cutting operation is dusty. Everyday eye glasses only have impact resistant lenses, they are NOT safety glasses.

DO NOT OVER-REACH. Keep proper footing and balance at all times.

USE THE RECOMMENDED ACCESSORIES. Consult the manual for recommended accessories. The use of improper accessories may cause risk of injury.

CHECK DAMAGED PARTS. Before further use of the machine, a guard or other part that is damaged should be checked to determine that it will operate properly and perform its intended function. Check for alignment of moving parts, breakage of parts, mounting, and other conditions that may affect its operation. A guard or other part that is damaged should be properly repaired or replaced.

NEVER OPERATE A MACHINE WHEN TIRED, OR UNDER THE INFLUENCE OF DRUGS OR ALCOHOL.
Full mental alertness is required at all times when running a machine.

IF AT ANY TIME YOU ARE EXPERIENCING DIFFICULTIES performing the intended operation, stop using the machine! Then contact our service department or ask a qualified expert how the operation should be performed.

DO NOT MODIFY OR ALTER THIS EQUIPMENT in any way. If modifications are deemed necessary, all such requests must be approved and/or handled by Rottler Manufacturing. Unauthorized modifications could cause injury and/or damage to machine and will void the warranty.

SAFETY DECALS SHOULD NEVER BE REMOVED. They are there to convey important safety information and warn of potential hazards.

ALL LOCAL SAFETY CODES AND REGULATIONS should be followed when installing this machine.

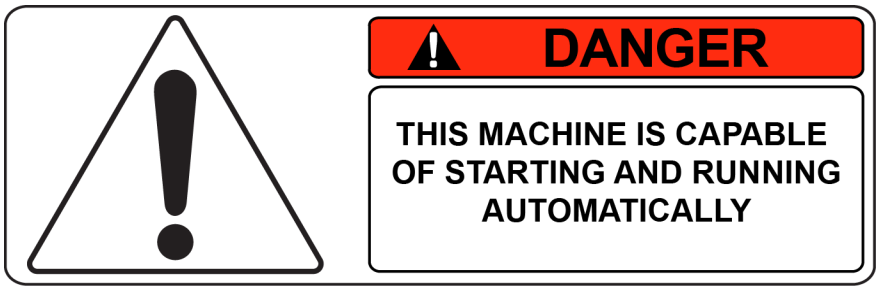
ONLY QUALIFIED PERSONAL should perform service on the electrical and control systems.

When boring the machine is capable of throwing metal chips over 10- feet from the cutting area. Always use the guards. Eye protection must be worn at all times by the operator and all other personnel in the area of the machine.

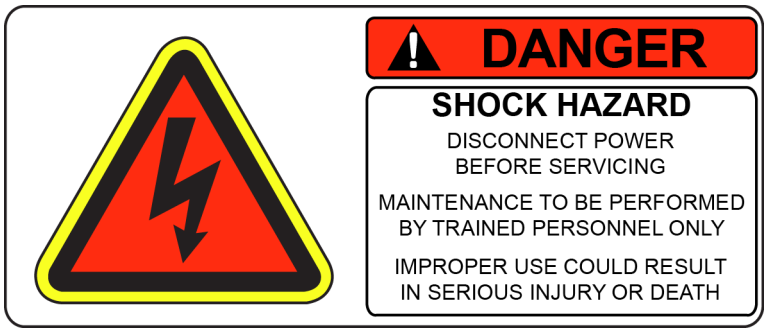
⚠ CAUTION No list of safety guidelines can be complete. Every piece of shop environment is different. Always consider safety first, as it applies to your individual working conditions. Use this and other machinery with caution and respect. Failure to follow guidelines could result in serious personal injury, damage to equipment or poor work results.

Electrical Power

THIS MACHINE IS AUTOMATICALLY CONTROLLED AND MAY START AT ANYTIME

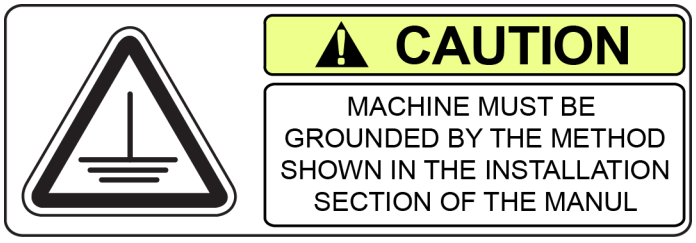


All electrical power should be removed from the machine before opening the rear electrical enclosure.



In the event of an electrical short, grounding reduces the risk of electric shock by providing a path of least resistance to disperse electric current.

Electrocution or a fire can result if the machine is not grounded correctly. Make sure the ground is connected in accordance with this manual. DO NOT operate the machine if it is not grounded.



⚠ CAUTION No single list of electrical guidelines can be comprehensive for all shop environments. Operating this machinery may require additional electrical upgrades specific to your shop environment. It is your responsibility to make sure your electrical system comply with all local codes and ordinances.

⚠ WARNING This machine operates under computerized control and, as is all computerized equipment, and is susceptible to extraneous electrical impulses internally for externally produced. The machine may make moves out of the operator control at any time. The operator should work in and around the machine with caution at all times.

The operator and nearby personnel should be familiar with the location and operation of the Emergency Stop Button.

Make sure all electrical equipment has the proper overload protection. This machine should have ***a fully isolated*** power supply to prevent damage and uncontrolled movement of the machine. If this machine is on the same power lines that are running to other electrical equipment (grinders, welders, and other AC motors) electrical noise can be induced into this machines electrical system. Electrical noise can cause the controller to see false signals to move. Not supplying a fully isolated supply to the machine may void factory warranty. Refer to the Power supply section located in the Installation section for voltage and amperage requirements of this machine.

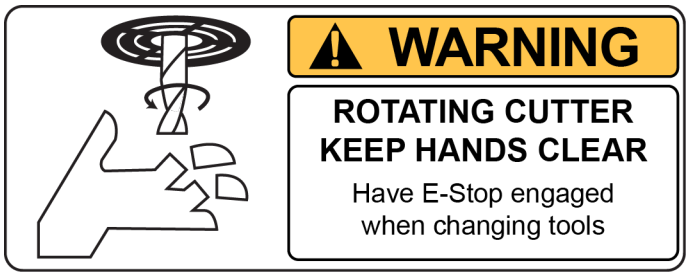
Machine Operator:

The operator of this machine should be a skilled machinist craftsman who is well versed in the caution, care, and knowledge required to safely operate metal cutting tools.

If the operator is not a skilled machinist he/she must pay strict attention to the Operating Instructions outlined in this manual, and get instruction from a qualified machinist in both production and operation of this machine.

This machine has the following areas of exposed moving parts that you must train yourself to respect and stay away from when they are in motion:

Cutting Tool Area – Any operation involving hands in the cutter head area, such as inspection or alignment of the cutter head or tools, changing Centering Fingers, tool insertion, and removal, cutter head changes, and size checking etc. requires the machine to be in Neutral.



Machining – Eye protection must be worn during all operations of the machine. Hands must be kept completely away from the cutter head. All chip guards must be in position during machine operations.

CAUTION **Work Loading and Unloading** – Carefully develop handling methods of loading and unloading work pieces so that no injury can result if hoist equipment or lift connection should fail. Periodically check lift components for damage that may cause failure.

CAUTION **Machine Maintenance** – Any machine adjustment, maintenance or parts replacement absolutely requires a complete power disconnection from the machine.

Emergency Procedure:

Assuming one of the following has occurred: tool bit set completely off size, work piece or spindle base not clamped, spindle is not properly centered, and these mistakes will become obvious the minute the cut starts

PRESS THE EMERGENCY STOP BUTTON (on the front control panel) **IMMEDIATELY!**

Find out what the problem is; return the spindle to its up position without causing more damage. To restart the machine, turn the Emergency Stop Button CW until the button pops out

Be alert to quickly stop the machine in the event of a serious disruption of the boring process either at the top or bottom of the bores.

"**REMEMBER**" metal cutting tools have the speed and torque to severely injure any part of the human body exposed to them.

Computer and Controller System Safety:

The computer and controller are located in the main rear electrical enclosure. This unit is a full computer, running Windows 7 64 Bit operating system. Contact the factory if more information on the computer system is required.

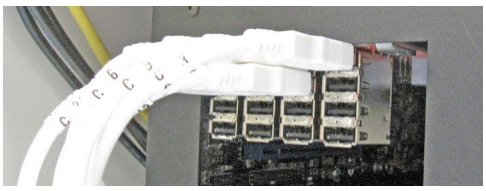
IMPORTANT The computer in this machine has the ability to connect to the World Wide Web via Ethernet or Wireless using a USB wireless (Wi-Fi) adapter. Updating the Rottler software should ONLY be done when directed to do so by a Rottler service technician. Updating Rottler Software when not directed by Rottler personnel will result in a non-operational machine.

The machine should be hooked up to the Internet anytime it is on. The software on the machine will automatically connect to our server to send back useful information on machine status.

Any "IT" personnel should **ALWAYS** get approval from Rottler before doing ANYTHING on the computer.

DANGER This machine is capable of causing severe injury or death. Doing any of the following without Rottler's direct consent may cause severe injury or death.

WARNING Do not attempt to install USB devices in the PCI ports. These ports have high voltage and any attempt to connect a USB device in these ports will result in destruction of that device. There is also the possibility of damage to the computer system of the machine.



IMPORTANT Downloading any program or changing any Rottler or Computer settings may cause the machine and/or software to become unstable. DO NOT install ANY screen saver, Anti-Virus, Spyware or any type of Security software on the computer. This could create a hazardous

environment for the operator and personnel around the machine. Performing any of the above will also result in the machine warranty being NULL and VOID.

IMPORTANT DO NOT connect any type of external hardware to the computer via USB or any other means. Do not install any type of Device Driver. This could create a hazardous
environment for the operator and personnel around the machine. Performing any of the above will also result in the machine warranty being NULL and VOID.

Electrical Safety Features Of Rottler DM Controlled Machines

All Rottler machines that use the DM operational control system are designed to comply with all applicable safety standards. This includes but is not limited to the following systems:

Thermal sensors in all motors and motor controls.

1. Current sensors in all motor control panels.
2. Electrical breakers to prevent voltage surges and spikes from reaching electrical system.
3. Electrical lockout on main electrical enclosure.
4. E-Stop that shuts down all operational systems in an event of an emergency.

All thermal and current limits for motors and motor controls are preset at the factory. In the event that any of those parameters are exceeded during operation of the machine, the machine control system will shut down the machine and a warning of the specific fault will appear on the control screen.

I

CONTROL DEFINITIONS

Contents

Control Definitions

The purpose of this chapter is to define the function of the buttons throughout the various screens. Certain button functions may not make sense right away in this chapter. As the operator reads through the Operating Instructions chapter of this manual, the function of these buttons will become clear.

Computer and Controller System Safety

The computer and controller are located in the main rear electrical enclosure. This unit is a full computer, running Windows 7 64 Bit operating system. Contact the factory if more information on the computer system is required.

IMPORTANT: The computer in this machine has the ability to connect to the World Wide Web via Ethernet or Wireless using a USB wireless adapter. This machine should NOT be connected to the Internet for any reason other than getting a software update. This should ONLY be done when directed to do so by a Rottler service technician. Updating Rottler Software when not directed by Rottler personnel will result in a non-operational machine.

Any “IT” personnel should ALWAYS get approval from Rottler before doing ANYTHING on the computer.

This machine is capable of causing severe injury or death. Doing any of the following without Rottler’s direct consent may cause severe injury or death.

Connecting to the Internet for any other reason will leave the machine vulnerable to viruses which could create a safety hazard and/or leave the machine inoperable.

Downloading ANY program from the Internet or by other means when not directed by Rottler is prohibited and will result in the machine warranty being NULL and VOID.

Downloading any program or changing any Rottler or Computer settings may cause the machine and/or software to become unstable. DO NOT install ANY screen saver, Anti-Virus, Spyware or any type of Security software on the computer. This could create a hazardous environment for the operator and personnel around the machine. Performing any of the above will also result in the machine warranty being NULL and VOID.

DO NOT connect any type of external hardware to the computer via USB or any other means. Do not install any type of Device Driver. This could create a hazardous environment for the operator and personnel around the machine. Performing any of the above will also result in the machine warranty being NULL and VOID.

Master Power On/Off Switch

This switch is located on the main electrical control enclosure on the back of the machine. The switch must be in the off position before opening the rear enclosure door.

When first applying power to the machine the computer will need to boot up. Be patient, it will take several minutes to complete booting. The Rottler program will not automatically start. Double tap the Rottler_WPF icon on the screen to start Rottler.

When turning the main power to the machine off there is a specific procedure to follow so as not to damage the computer. The computer must shut down its internal systems before main power is removed from it.

Press the “Start” button in the left-hand side of the Start Bar. This will bring up the “Start Menu”. Press the “Shutdown” line at the bottom of the Start Menu. This will bring up a Pop Up menu, make sure that “shut down computer” is selected and press “OK”.

This will shut down the computer. It is now OK to turn Main Power off to the machine.

Machine Startup

Start by double clicking the direct surface icon found on your desktop.

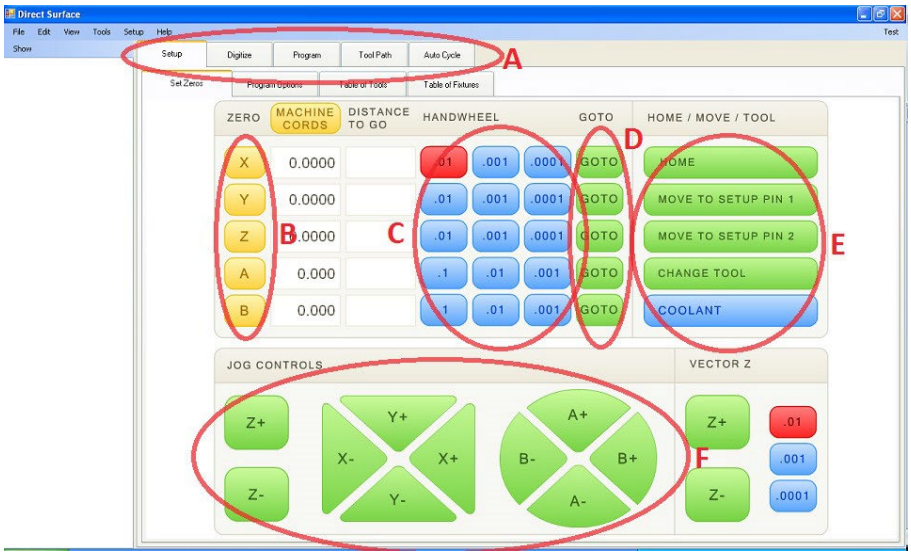
E-Stop

The E-stop is a safety tool that will prevent the machine from moving and allow you to immediately terminate any action that the machine is performing, in case there is an emergency. Once the Direct Surface program has opened, twist the red E-stop button until it pops out.

Homing the Machine

Every time a new session of Direct Surface has been opened the machine MUST be homed to ensure proper operation the machine. This allows the machine to accurately locate all travel limits and zeros. To home the machine you click the green “Home” button on the right side of the screen. The A and B axis will rotate to a specified ‘home’ location. Once this process is complete, a message saying “Homing Complete” will appear and you may click “OK.” If you receive a message that says “Error: Move Axis farther away from homing switch” The just use the green jog buttons to move the mentioned axis and press the “HOME” button again.

(You will see this message after all axes find their limits.)



Control

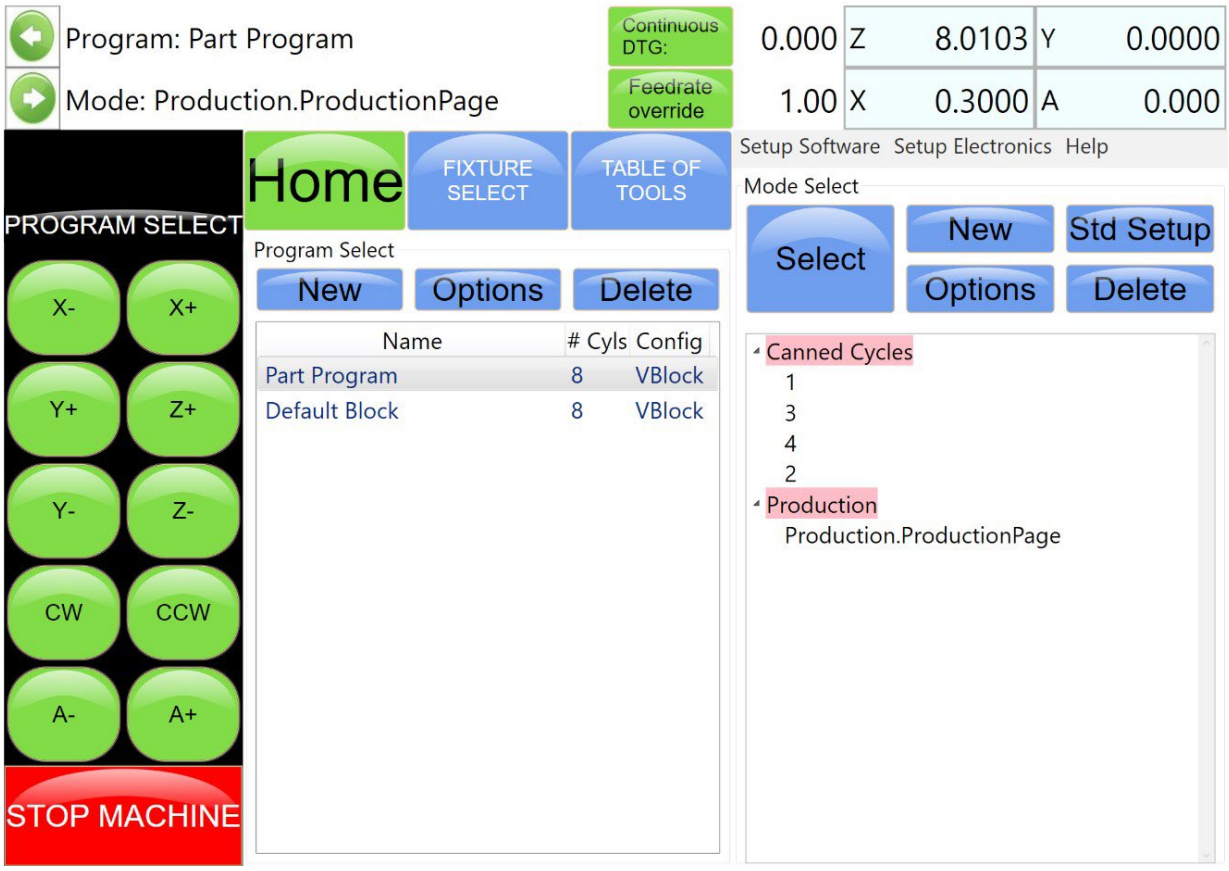
Viewing the control there are several tabs to click on (A) which include Setup, Digitize, Program, Tool path, and Auto Cycle. Each of these tabs will be used in order to guide you through the programming process. The yellow X, Y, Z, A, B buttons (B) allow you to zero you fixture coordinates. The blue “HANDWHEEL” buttons (C) are to select the increments you would like to move each axis by the hand wheel. The “GOTO” buttons (D) allow you to quickly move to a designated fixture location. The green buttons in E allow you to perform essential setup functions. The “JOG CONTROLS” (F) allow you to quickly move the machine in less precise increments to a desired location.

OPERATING INSTRUCTIONS

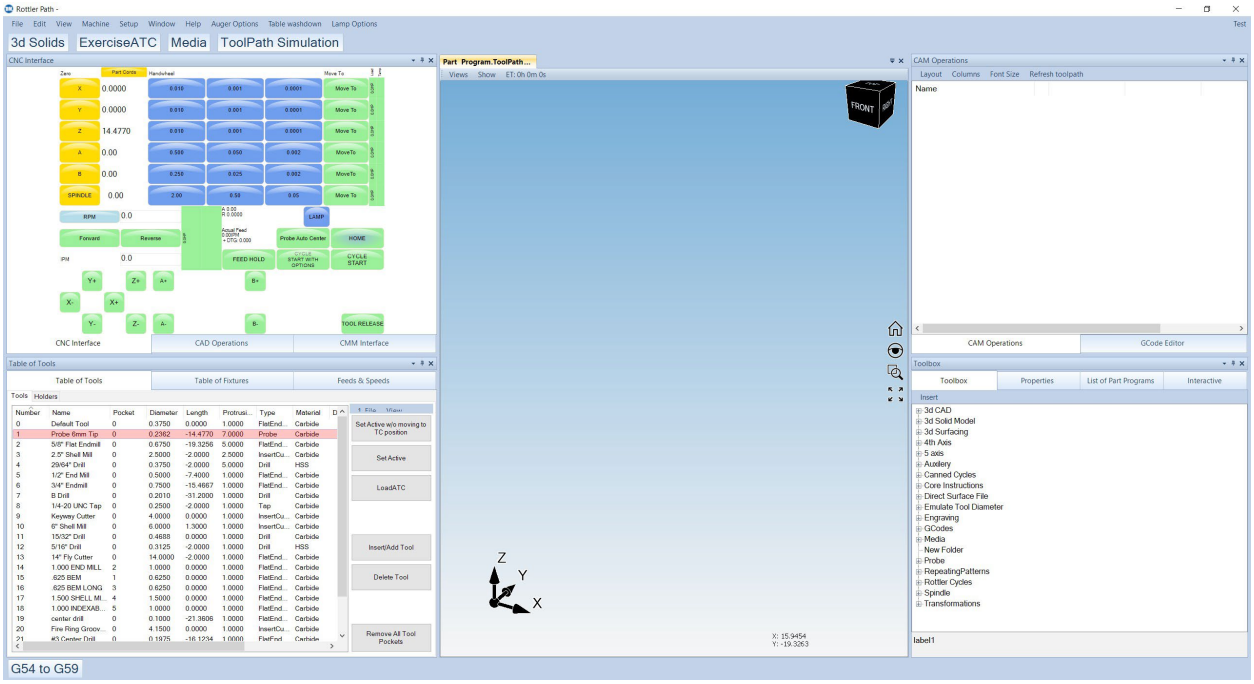
Contents

General Machine Information

The Rottler EM69HP utilizes Computerized Numeric Control (CNC). From within any of the Rottler software packages the CNC control interface can be used to move the machines axis in any of the labeled directions. Below are images of the main user interfaces found in the RPATH/4C software and the Rottler Block Software program packages.



Rottler Block Software User Interface



Rottler RPATH/4C User Interface

Homing

The EM69ATC ***MUST*** be homed anytime it is restarted after it has been shut down. The machine will automatically prompt the user to home the machine before being able to program and run any cycles. Homing is required so that the axes reference their current locations so that parts can be machined correctly in their respective coordinate locations.

Building Programs

NOTE: The instructions within this operator manual will cover the creation and use of block machining programs in the Rottler Block Software. For information regarding the creation and use of Rottler's Rpath/4C software packages consult the Rottler Introduction to Rpath/4C training manual that is supplied with the purchase of that software package.

Create a Block Program

Within the Block Software's main screen under the program select tab, select the "NEW" button to create a new engine block file. Enter the engine block information for name, number of cylinders and block configuration and press OK when finished. On the Right side of the screen under the mode select tab, press new and select the type of operation you wish to perform to create the operation program within the engine block file. For this example we will create a cylinder bore program.

Options

If you need to change the block configuration or name of a block that has already been created, use the Options button. This will bring up the same window as when the block was created.

Std (Standard) Setup

Pressing Std Setup will cause all of the available Modes to be inserted into the Modes area on the right hand side.

Select

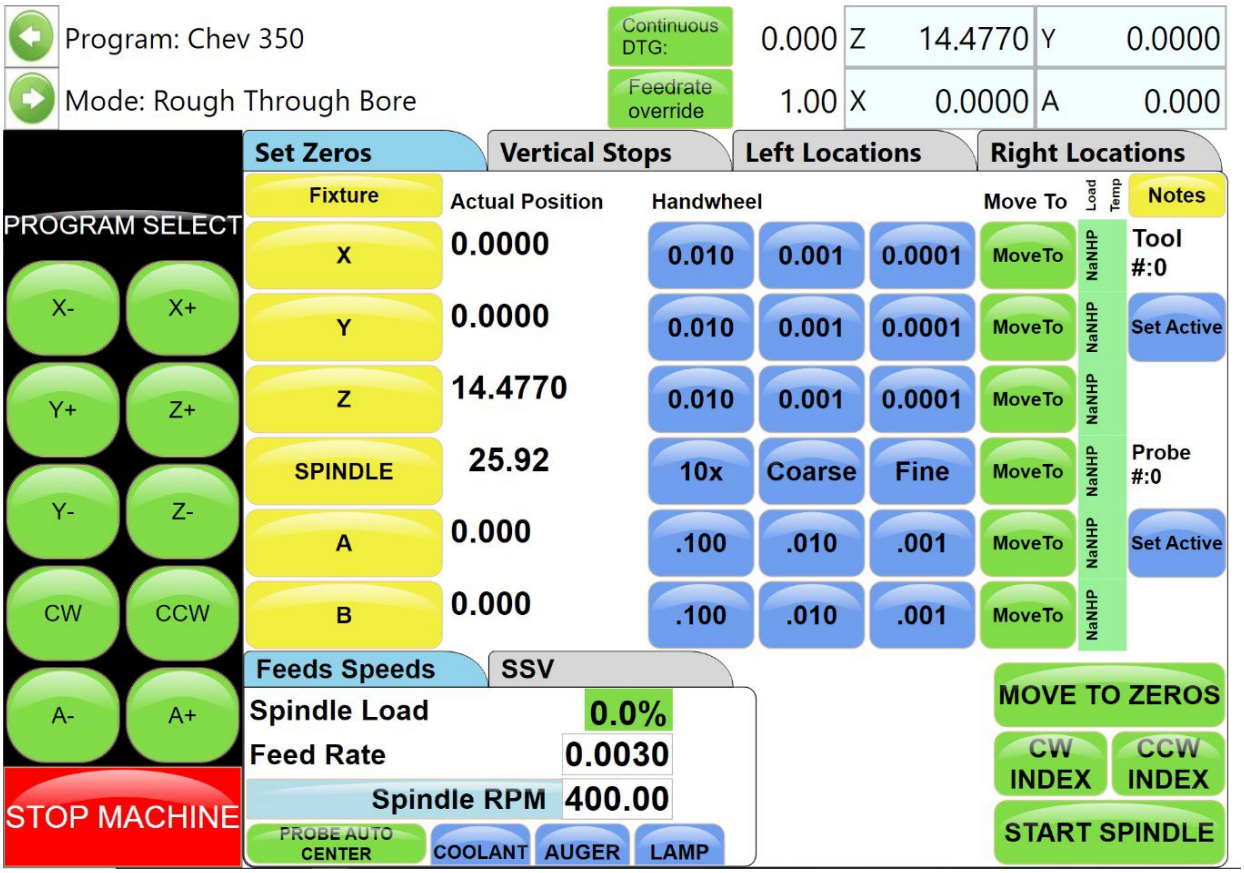
Pressing Select with a Mode highlighted will open the operations screens for using the program.

Options

Resetting the Options button with a Mode highlighted will open a window where you can change the mode name. There is also a check box to allow positive number to be entered into the program where they are normally forced to a negative value.

Cylinder Bore Mode 3 Axis

Select Cylinder Bore and then Rough Through Bore on the screen. This will bring up the boring program with the Set Zeros tab shown.

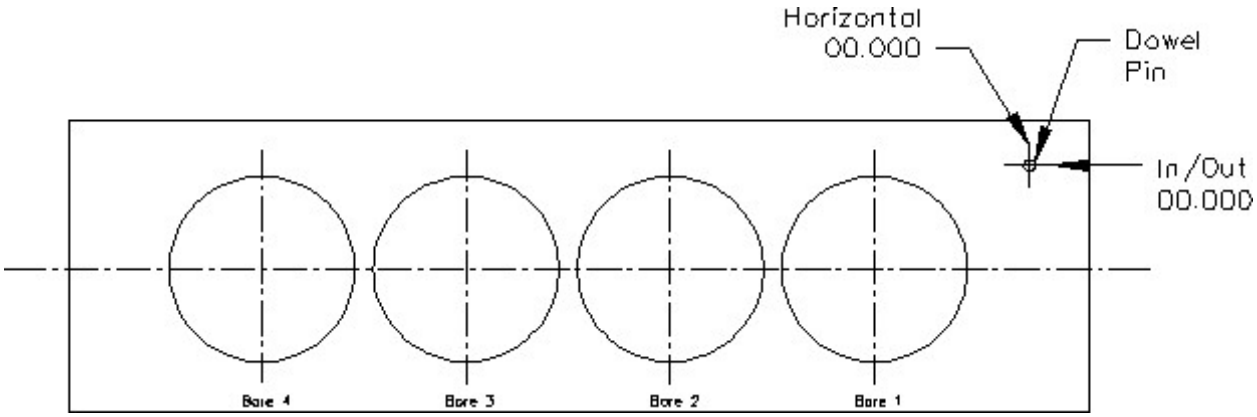


Setting Zeros

The purpose of setting zero points is to give the operator a specific point to build programs from. The machine also uses these zero points to run the operation. The zero points can be set at any point in the machines' travel. Every program will save it's individual zero positions, once the zero positions are set for an individual program then the operator need only to position the block in to be worked on in the same position each time to successfully run the operations.

X and Y Axis Zero's

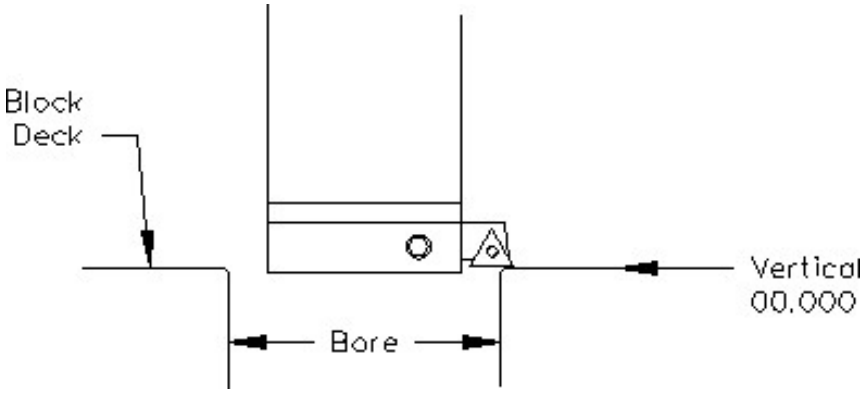
For this example, the Engine Block Dowel Pin will be our zero point for the **X-Axis** (Horizontal) and **Y-Axis** (In/Out) axis. Using either the touch trigger probe or a test indicator, find the center of the dowel pin. Without moving the machine, **double tap** on the **X and Y buttons** directly beneath the **Fixture button** on the **Set Zeros** screen. A window will pop up asking if you want to set the selected axis, **press yes to zero the axis.**



Vertical Zero

There are two different methods for setting the Z-axis zero for block machining

For this example, we will be using the block deck to zero our Z-axis (vertical axis). Insert a tool holder into the cutterhead you will be using to bore the block. Center the cutterhead over a cylinder. Using the Vertical Handwheel, bring the cutterhead down until the tool just touches the deck and press the Vertical Zero button. The display next to this button will go to zero. The Vertical zero has now been set.



The zeros points for all axis have now been set. All the numbers entered from this point on will reference these zero positions. You are finished with the Set Zeros screen, select the next Tab to the Right, Vertical Stops.

Blueprinting Method

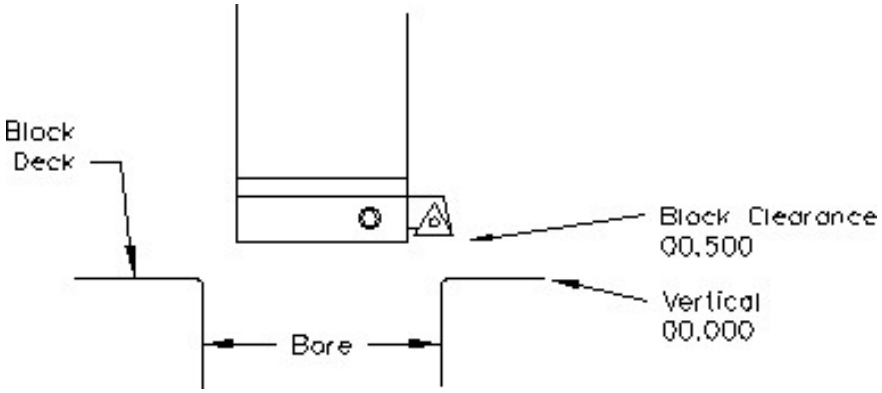
Even if you are not going to be boring a block to the blue print specifications it is recommended to have the Blueprint values entered. It will speed up the process of indicating and probing a block by giving the operator a close estimate of bore location.

Programming Vertical Stops

To build a program you must set the Vertical Stops for the program. This is done by filling out the boxes in the Vertical Stops Tab.

Block Clearance

This is the distance above the zero position or block deck allowing the cutterhead to move to the next bore unobstructed. If you are Blueprinting a block the number will be just enough to allow the cutterhead to clear the block deck. We recommend a .100” to .500” range for this value.

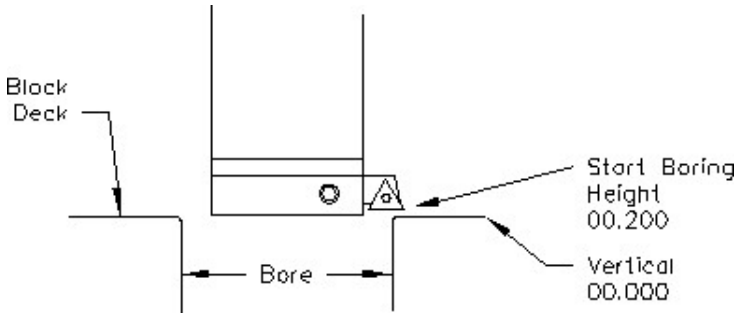


Centering Height

When Blueprinting this stop is not needed. It should be the same as the block Clearance Height.

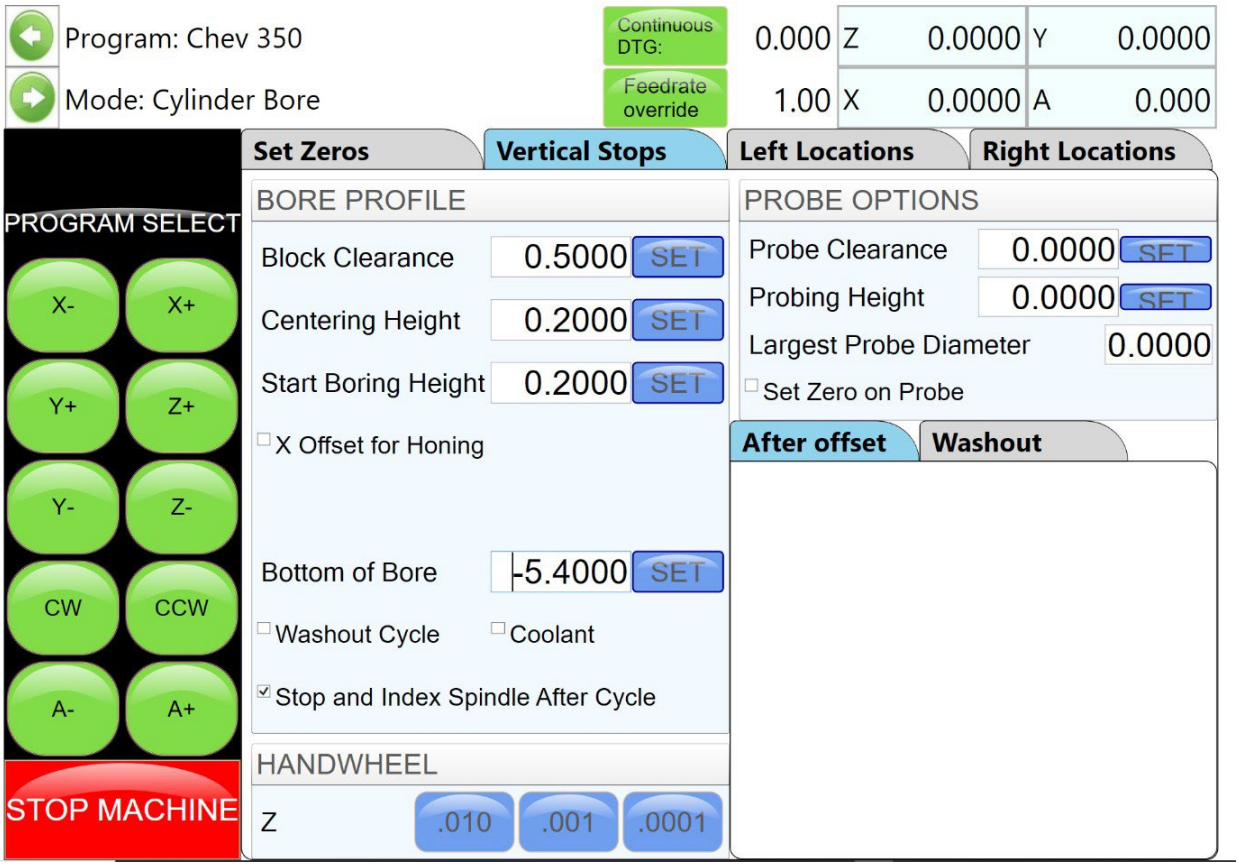
Start Boring Height

This is the distance above zero or the block deck where you want the cutterhead to start rotating and the downward feed to start. Generally, this is just a short distance above the block deck to minimize the amount of time the machine bores through air. We recommend .030” to .200” range for this value



Bottom of the Bore

This is the distance below zero or the Block deck where you want the machine to stop boring and retract out of the cylinder. When the spindle retracts it will then go back to the Block Clearance position. For blue printing you can use a OEM bore length specification.



X Offset for Honing

This feature is designed to offset the cutter at a certain height in the lower bore to cut out block web intrusions to make room for the honing process. Checking this box will add the offset parameters options to the Vertical Stops Tab.

Start Offset Height

This is the vertical depth at which the cutter will shift to the side to start cutting.

Horizontal Offset

This is the distance the cutter will offset from the bore center.

Change Speeds at Horizontal Offset

Often the clearance cut is much larger that the cut for the rest of the bore. For this you can check this box and enter a different RPM and Feed Rate. If a different speed and feed are not need do not check this box and the same feed and speed will be used that was used to bore the cylinder.

For each bank (of a V Block) you can select the direction required for the offset routine.

Washout Cycle

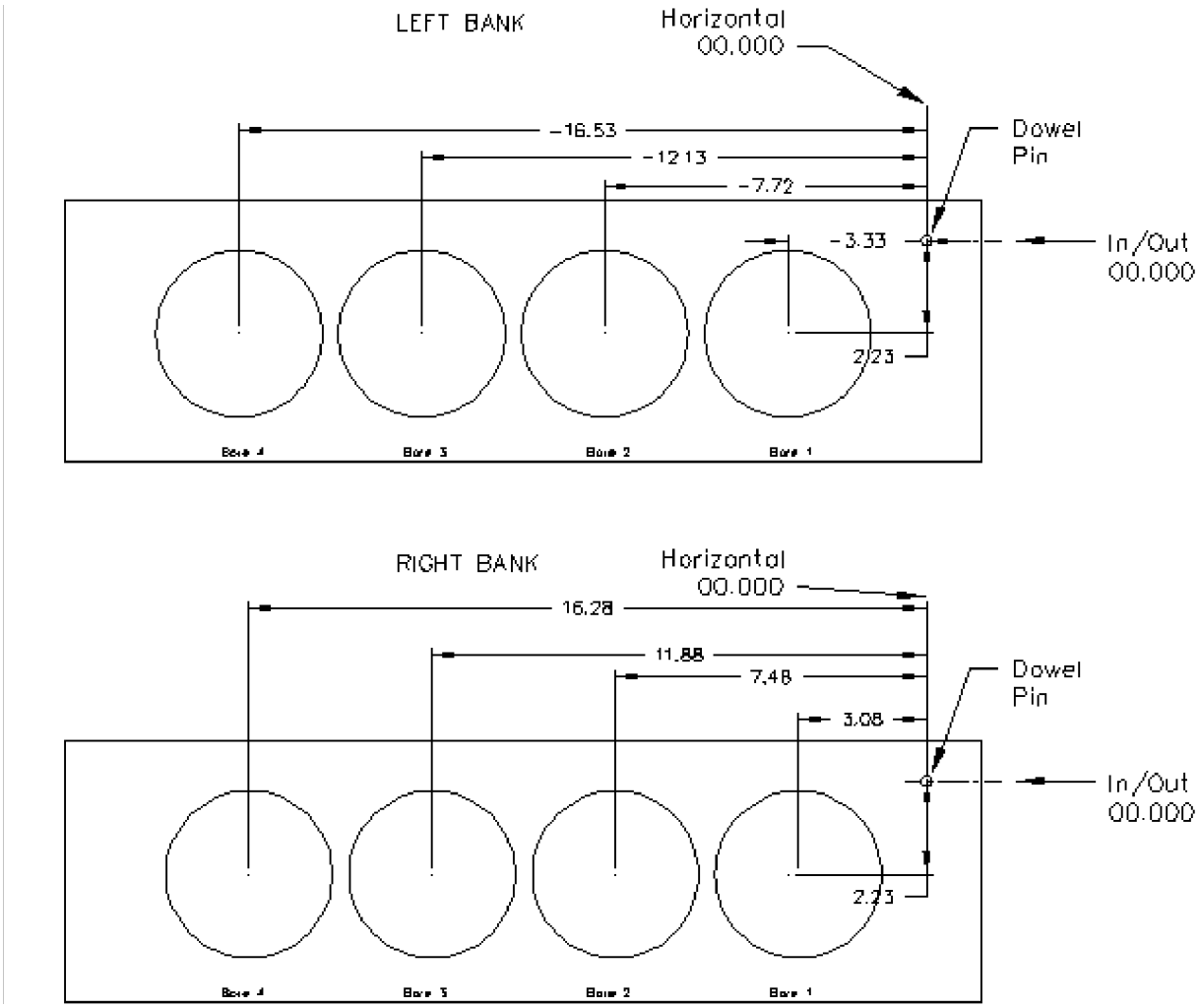
Checking this box will open another window on the right hand side of the screen. Here you can enter the RPM and number of revolutions that will be performed when the cutter reaches the Bottom of Bore position. This is used when a certain type of finish is required on a counter bore or the bottom of a sleeve cut. Typically the RPM is reduced during a washout cycle.

Stop and Index Spindle after Cycle

Checking this box will cause the spindle to be indexed to the three O'clock position after the cylinder has been bored but before it retracts. It will also offset to the left before the tool is retracted. This is the default setting. You would not want this check in an operation such as Lifter Boring.

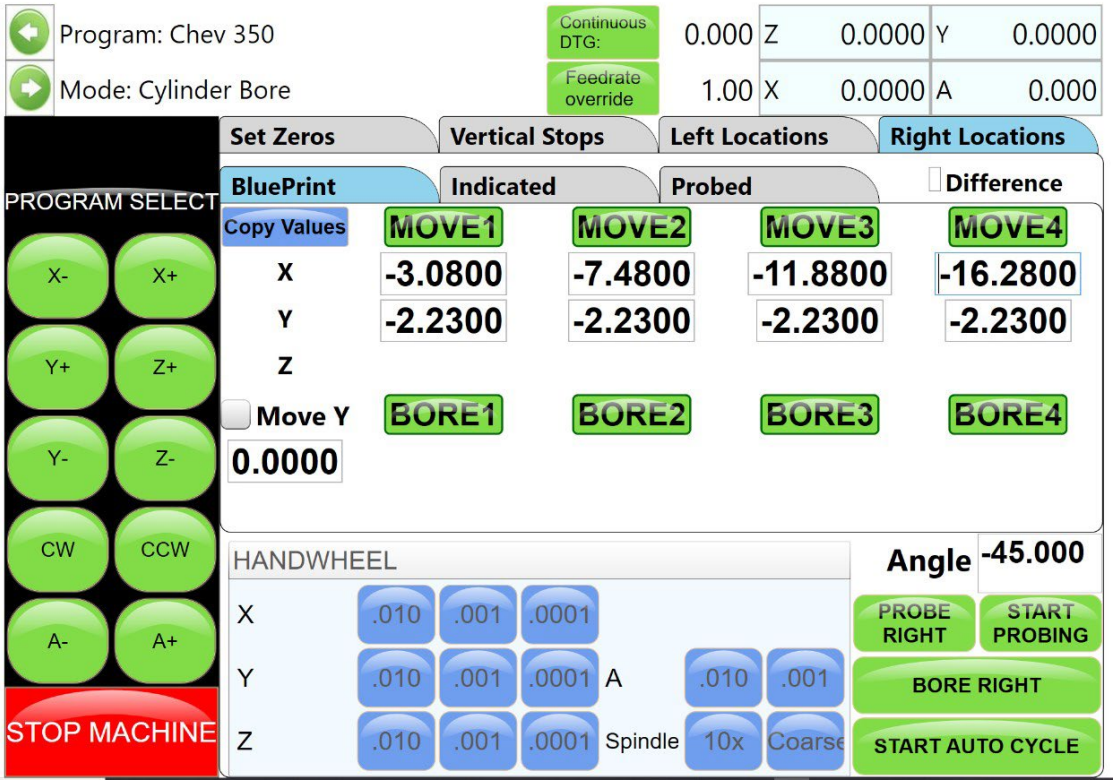
Bore Locations

To build a program you must set the X and Y axis locations for the individual cylinder bores. Since we previous set the program zero at the dowel pin location, we may now use the block blueprint dimensions to program the X and Y locations for the cylinders in reference to the dowel pin location.



Left Locations

Right Locations



Boring a Block

Once the Vertical Stops and Left/Right location stops have all been entered the Spindle RPM and Feed Rate need to be entered. This is done on the Set Zeros screen. Once this is done you can go to the Left and/ or Right Bore location screens and bore the cylinders.

Pressing the Bore Left for Bore Right buttons Will Bore all the cylinders that have Green bore button below them.

Pressing a Bore button once will turn that button Yellow. Any Yellow button will not be bored when the

Bore Left or Right button is pressed.

Double clicking any Bore button will turn all the Bore button yellow EXCEPT the one that was double click.

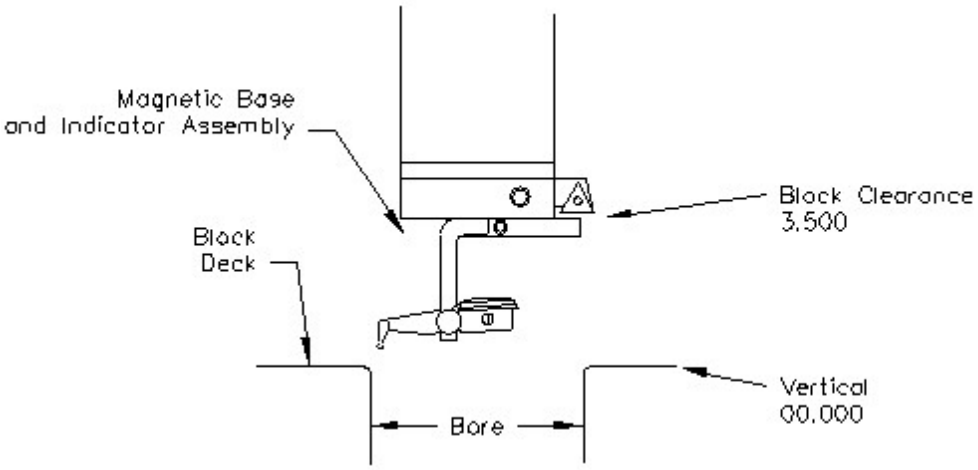
If your machine is equipped with a 4th axis fixture then the Start auto cycle button will bore both the left and right banks automatically and roll the 4th axis over in between the two banks.

Indicator Method

Sometimes it is necessary to us a dial indicator to find the bore locations of an engine block when creating the program. When this is required the programming is identical to the blueprinting method, with the only exception being that we will now take advantage of the centering height option. When using this method we want to program the machine so that it goes to the rough location of the bore, where we then can use a dial indicator to find the true center of the bore to be machined before saving the X,Y location.

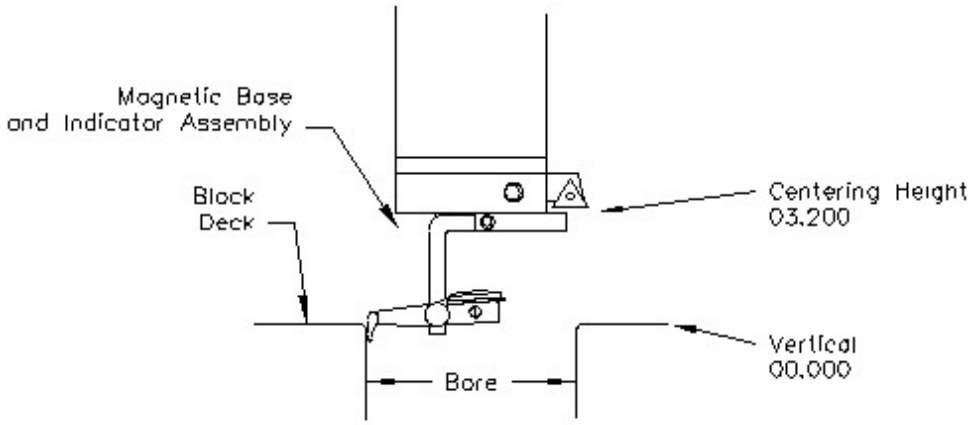
Block Clearance

This is the distance above the zero position or block deck allowing the cutterhead to move to the next bore unobstructed. When you are indicating the cylinders in you must have this stop set so the indicator will clear the block surface when traveling to the next cylinder.



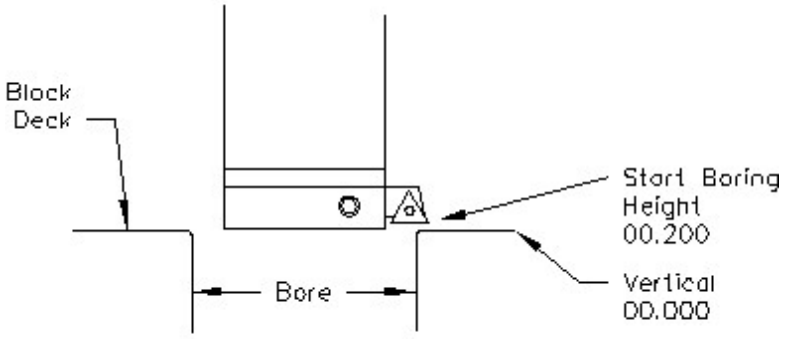
Centering Height

This is a distance above the vertical zero where you will be manually centering the block. The drawing below is a typical set up for manual centering or indicting a cylinder.



Start Boring Height

This is the distance above zero or the block deck where you want the cutterhead to start rotating and the downward feed to start. Generally this is just a short distance above the block deck to minimize the amount of time the machine bores through air.



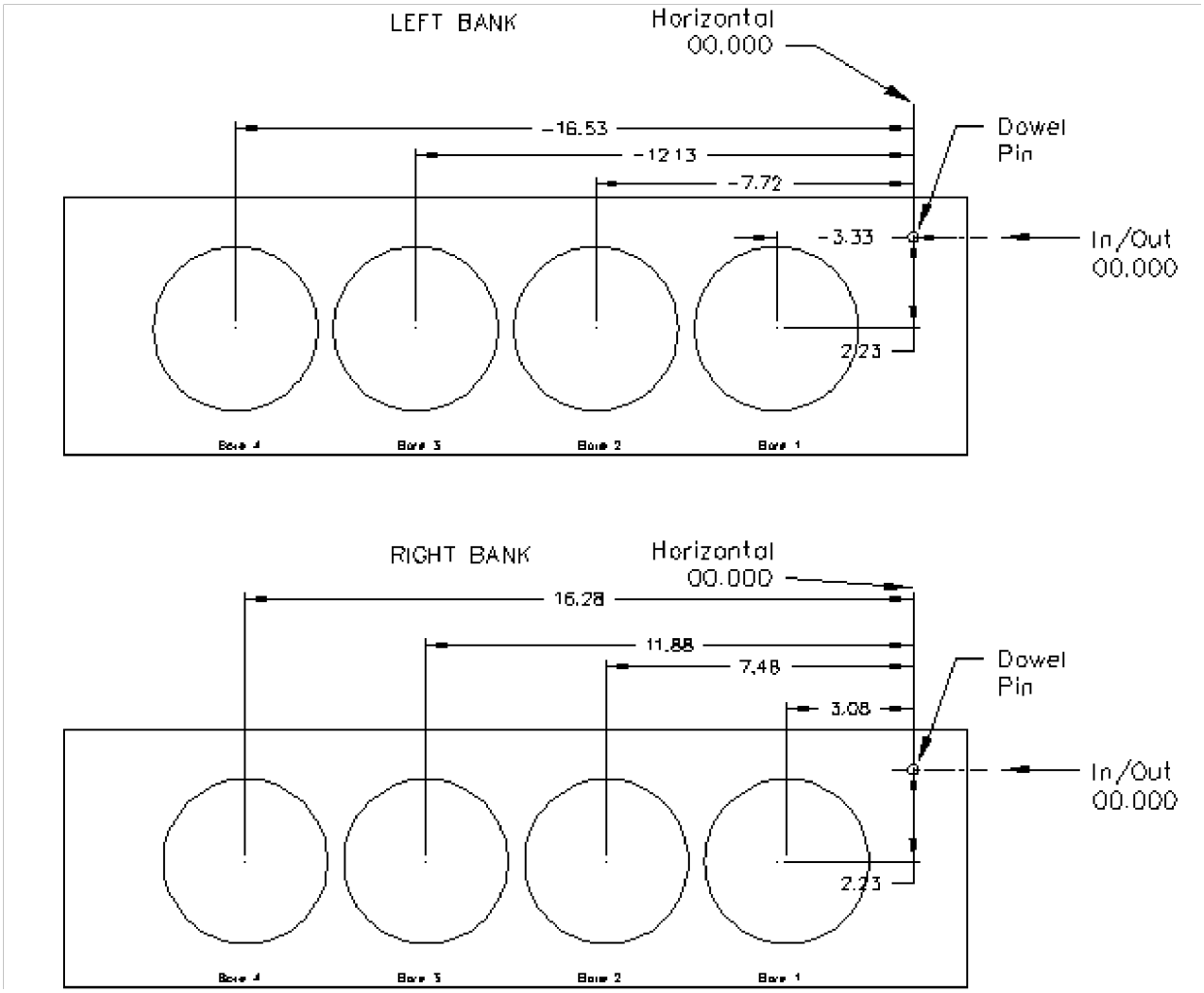
Bottom of the Bore

This is the distance below zero or the Block deck where you want the machine to stop boring and retract out of the cylinder. When the spindle retracts it will then go to the block Clearance position.

The Vertical stops have now been set. You are finished with the Vertical Stops screen, select Left and/or Right Locations.

Bore Locations

To build a program you must set the X and Y axis locations for the individual cylinder bores. Since we previous set the program zero at the dowel pin location, we may now use the block blueprint dimensions to program the rough X and Y locations for the cylinders in reference to the dowel pin location.



Select the left locations tab and then navigate to the indicated. If you have programmed the blueprint locations into this program, then press copy values and then blueprint. This will cause the values from the Blueprint page to be copied into the indicated page. This give you a starting point to indicate the individual cylinder from.

Press the Move 1 button. The machine will move to the first cylinder and stop at the centering position. Manually indicate the cylinder in using the X and Y axis handwheel options. Once the cylinder is centered press the Set 1 button. This will transfer the current position of the machine into the first set of Data Boxes. Repeat this process for all the cylinders that need to be indicated. Once the Left Locations have been indicated the same steps can be used to set the right locations.

Boring a Block

Once the Vertical Stops and Left/Right location stops have all been entered the Spindle RPM and Feed Rate need to be entered. This is done on the Set Zeros screen. Once this is done you can go to the Left and/ or Right Bore location screens and bore the cylinders.

Pressing the Bore Left for Bore Right buttons Will Bore all the cylinders that have Green bore button below them.

Pressing a Bore button once will turn that button Yellow. Any Yellow button will not be bored when the Bore Left or Right button is pressed.

Double clicking any Bore button will turn all the Bore button yellow EXCEPT the one that was double click.

If your machine is equipped with a 4th axis fixture then the Start auto cycle button will bore both the left and right banks automatically and roll the 4th axis over in between the two banks.

Probing Method

If your machine is equipped with a Renishaw touch trigger probe then we can use the machines probing routines to locate the block deck, cylinder locations, and set zero's automatically. When using this method we want to program the machine so that it goes to the rough location of the bore, where we then call the probing routine to probe the bore and record the calculated X,Y locations of the bore automatically.

Vertical Zero

If you have setup the table of tools and recorded tool length compensations values, then instead you can used the Probe Auto Center command and select the depth routine from within the pop up window to probe he block deck. This will feed the z-axis with down with the probe in the spindle until the probe finds the deck surface. After the cycle is complete the probe tip will be just touching the surface and you can double click the z axis button to set the vertical zero.

If you have not setup the table of tools then use the blueprinting methods strategy for setting you vertical zero by touching the cutter off of the top of the block deck surface.

Programming Vertical Stops

Block Clearance

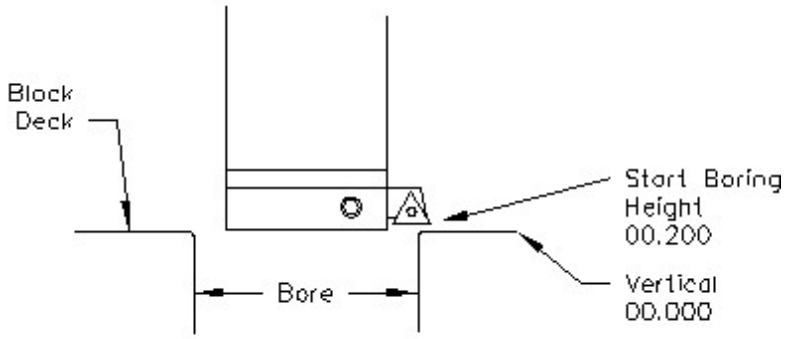
This is the distance above the zero position or block deck allowing the probe to move to the next bore unobstructed.

Centering Height

This stop is not used when you are using the probing feature. It is recommended that it be set to the same value as the Block Clearance.

Start Boring Height

This is the distance above zero or the block deck where you want the cutterhead to start rotating and the downward feed to start. Generally this is just a short distance above the block deck to minimize the amount of time the machine bores through air.



Bottom of the Bore

This is the distance below zero or the Block deck where you want the machine to stop boring and retract out of the cylinder. When the spindle retracts it will then go to the block Clearance position.

Probe Height

Using the handwheel bring the Probe down to the location in the cylinder you will be probing. Press the SET button next to Probe height. This will set the probing height position.

Using the handwheel move the probe up until it can safely move horizontal to the next cylinder. Press the SET button next to Probe Clearance. This will set the clearance height.

The Vertical stops have now been set. You are finished with the Vertical Stops screen, select Left and/or Right Locations.

Bore Locations

To build a program you must set the rough X,Y locations for the individual cylinder bores.

Select Left Locations and the Blueprint. Program the blueprint values (or close approximation) into the Horizontal and In/Out stops. Do the same for the Right Locations.

Select the Left Locations tab and then the probed tab. You can probe each cylinder individual by pressing the associated Probe button or you can probe the entire bank by pressing the Probe Left Button. **Probe Auto Center**

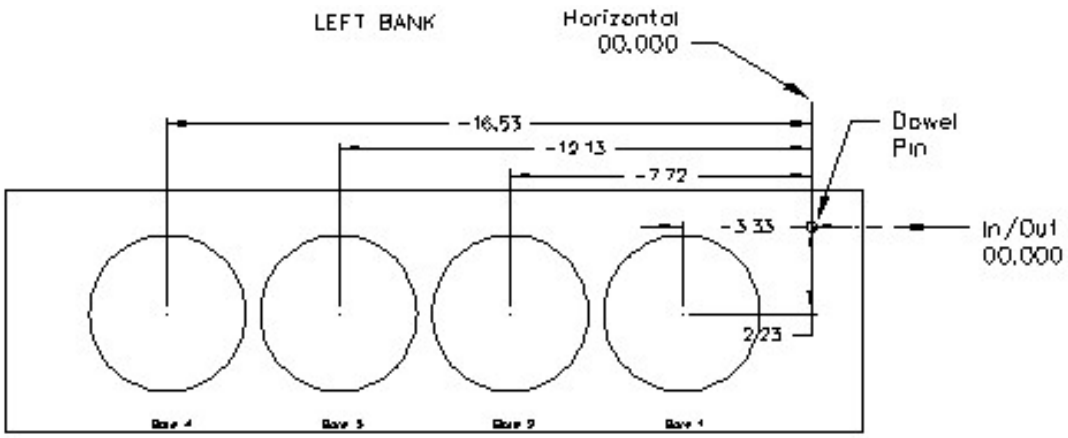
This feature is located on the Set Zero page. This allows easily find the center of a hole or cylinder. Roughly place the probe in the center of a cylinder. Press Probe Auto Center. The cylinder will be probed in 4 places, when finished the probe will move to the center of the probed cylinder. Pressing Horizontal and In/Out zero will then establish the center of that hole.

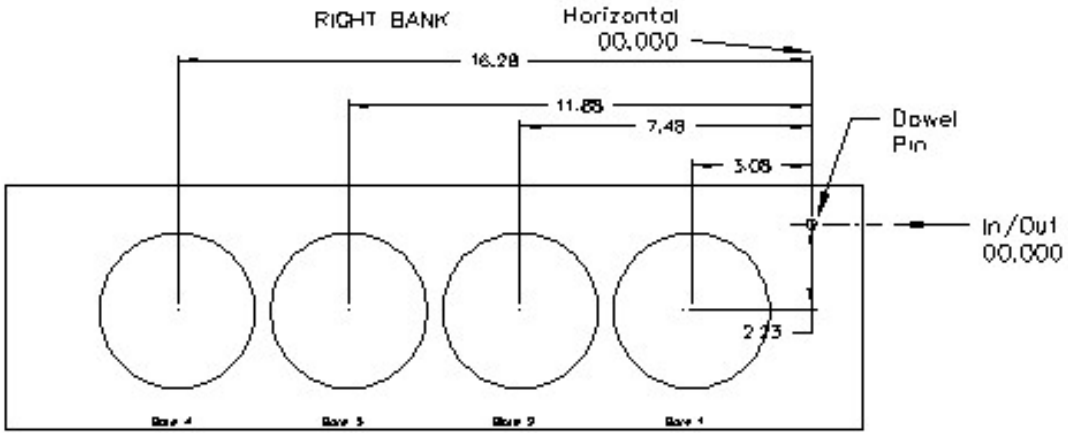
Automatic Probing Procedure

The probe will move to the center of the cylinder to be probed. It will then move to the right at a slow rate until the side of the cylinder is touched, it will then back off slightly and touch the same spot again to confirm position. The probe will then touch off the cylinder in three more spots and retract from cylinder.

As each cylinder is probed the Probed Diameter, Horizontal and In/Out positions will be placed into the Data Boxes for the corresponding cylinder.

Press the Right Locations tab and repeat the above procedure for the cylinders to be probed on the right bank..





Boring a Block

Once the Vertical, Horizontal and In/Out stops have all been entered the Spindle RPM and Feed Rate need to be entered. This is done on the Set Zeros screen. Once this is done you can go to the Left and/ or Right Bore location screens and bore the cylinders.

Pressing the Bore Left for Bore Right buttons Will Bore all the cylinders that have Green bore button below them.

Pressing a Bore button once will turn that button Yellow. Any Yellow button will not be bored when the Bore Left or Right button is pressed.

Double clicking any Bore button will turn all the Bore button yellow EXCEPT the one that was double click.

Cylinder Bore Mode 4th Axis

NOTE: The program with the 4th axis installed works basically the same as the 3 axis mode. ONLY the differences in operation and screens will be discussed here. Carefully read through the 3 Axis mode and then the 4th axis mode for operation and building programs.

Select Cylinder Bore and then Through Bore on the control panel. This will bring up the boring program with the Set Zeros tab shown.

Setting Zeros

The purpose of setting zero points is to give the operator a specific point to build programs from. The machine also uses these zero points to run the program from. The zero points can be set at any point

in the machines' travel. Each axis (except the Spindle rotation) will need to have a zero point set for the machine to operate from.

4th Axis (Rotational) Zero

The Zero position for the 4th (Rotational) Axis should be preset from the factory. If the zero needs to be reset use the following procedure.

There are three (3) flats cut onto the Head Stock Plate. Use the middle flat to set the rotational zero. Using an indicator off of the spindle indicate the middle flat to Zero all the way along it. Use the 4th Axis hand wheel to do this. When the middle flat is indicated in press the 4th Axis Zero button. You 4th (Rotational) Zero is set.

Finding the In/Out (Y) Axis Zero with 4th Axis

The Head Stock Plate has a hole in it next to the Middle Flat. This hole is centered on the center of the Main and Cam locator shafts.

Building Programs with the 4th Axis

Program are built the same as in the 3 Axis mode with the exception of setting the Angle for each Bank. The Left and the Right Locations page each have an Angle Data Box. Here you enter the angle of each bank from the 4th Axis (Rotational) zero position. The zero position is with the Cam and Crank Locators lined up vertically.

Example: On a Chevy 350 the Left bank would be positive 45 Degrees and the Right Bank would be a negative -45 Degrees.

Setting Vertical Clearance with 4th Axis

It is very important when setting your Vertical and Probe Clearance height that you be sure to account for the Roll Over of the block from bank to bank. When in an automatic program the block will roll from the Left Bank to the Right bank at the Left Bank Bore1 position. It will also rotate from the Bore1 position when going from Right Bank to Left.

Table of Tools for 3 and 4th Axis Bore Mode

NOTE: The Table of Tools is not needed to run the Rottler automatic programs. It is recommended that it not be used except by the advanced operator.

Building a Program with Table of Tools

Build the program as described above for 3 and 4 Axis programs using the same vertical zero locations.

Put the tools to be used into the Table of Tools as described in Chapter 2. In Bore mode you are not referencing another vertical location such as the Crank centerline so the Z Touch off Location will remain at zero.

Assigning Tools

Tools to be used in the boring operations are set on the Set Zeros page. To select a Tool, double click on Tool # on the right side of the screen. This will bring up the Table of Tools window. Highlight the tool you will be using, such as 2.9 Production Stub and select OK.

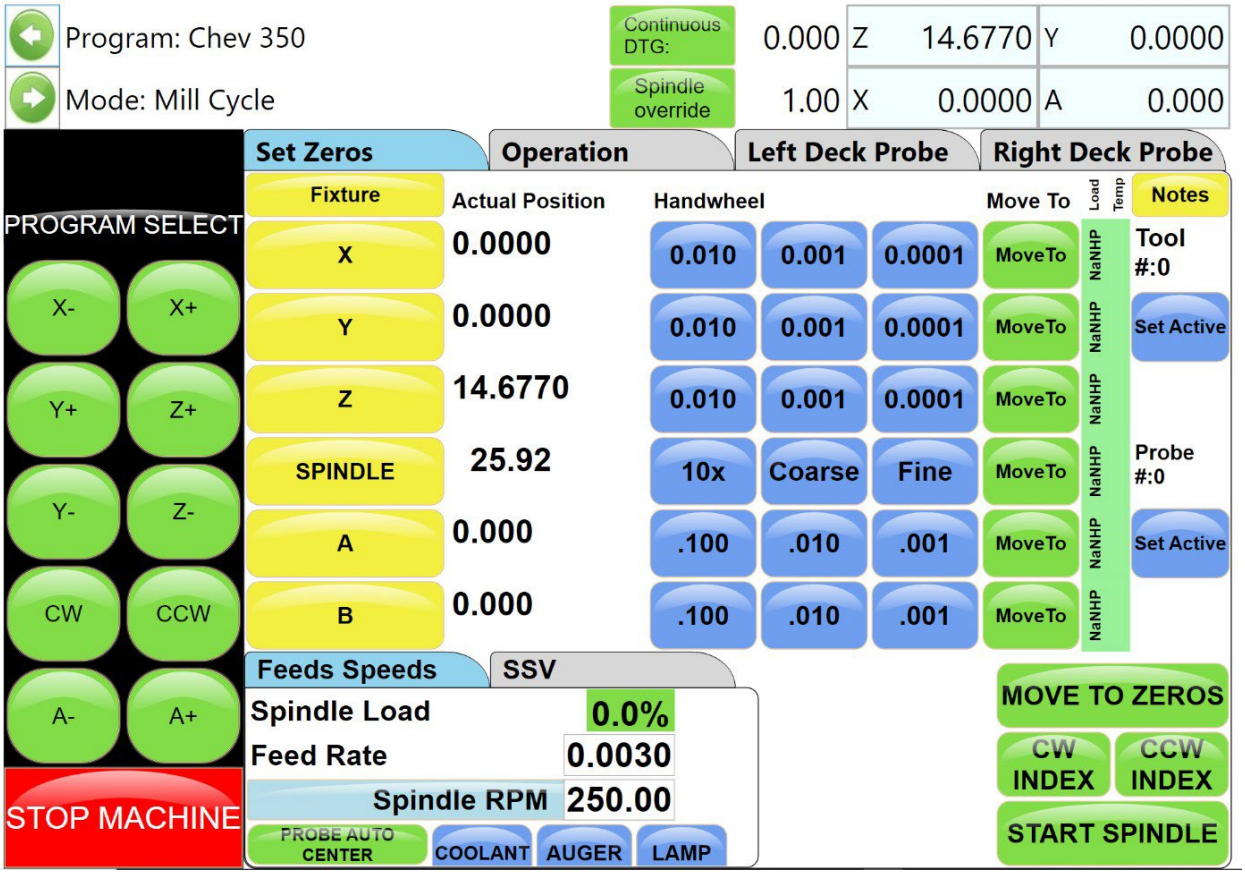
Do the Same to select the Probe you will be using, such as 100mm Probe.

NOTE: The Tool highlighted in red is the currently Active tool.

Mill Cycle 3 Axis

Mill Cycle

The mill cycle can be used for programming straight line toolpaths such as what is required for creating a surfacing/decking routine. From the mode select are in the main screen, select new and then select the mill cycle operation from the pop-up window. Once created, enter the mill cycle operations by selecting it in the mode select box.



Setting Zeros

The purpose of setting zero points is to give the operator a specific point to build programs from. The machine also uses these zero points to run the program from. The zero points can be set at any point in the machines' travel. Each axis will need to have a zero point set for the machine to operate from.

Horizontal Zero (X-axis)

For this example, we are going to set the Horizontal Zero (X-axis) at the point where the selected fly cutter is approximately ¼" passed the right hand side of the block to be machined.

In/Out Zero (Y-axis)

For this example we are going to set the In/Out Zero at the center line of the work piece.

Vertical Zero (Z-axis)

For this example the Vertical Zero will be at the deck height of the work piece.

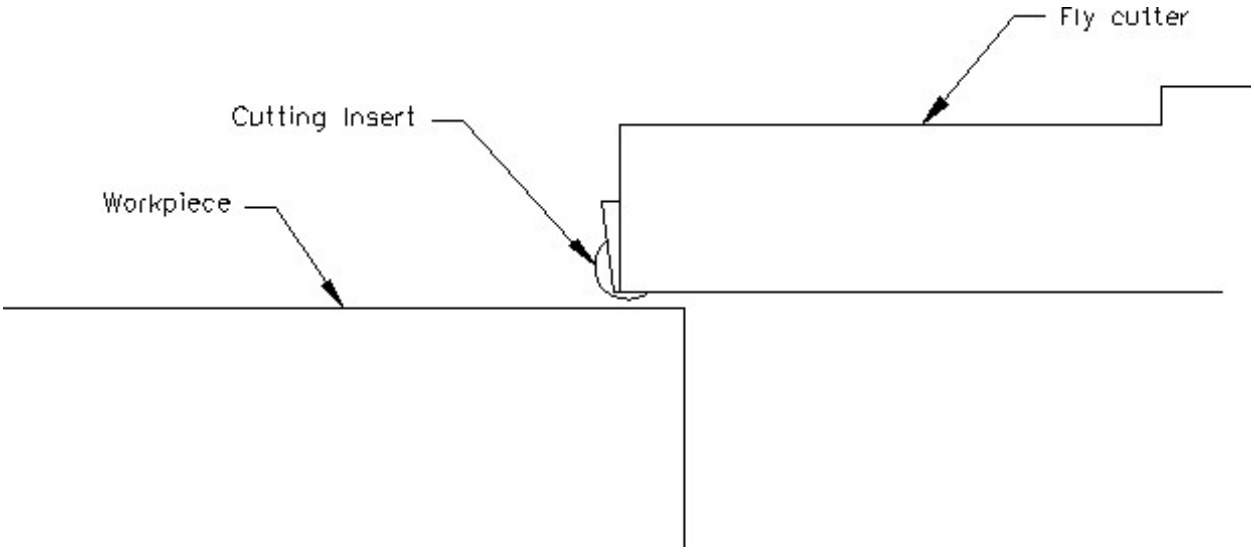
Feeds and Speeds (IPR/RPM)

The spindle RPM and Feed rate in IPR can be set in the lower left corner of the Set Zeros tab under the Feeds Speeds box. The RPM should be set based on the recommend SFM range for the insert being used, based on the material of the block being machined. Feed rate should be set based on the insert geometry, and surface finish requirements of the block to be machined.

Manual Procedure for Setting Vertical and Horizontal Zero's (Z-axis/X-axis)

Start the spindle. Select the .001" increment for the Z-axis handwheel and move the spindle down until you can hear or see the cutter just touching the block. Double tap the Z button to zero the Z-axis here.

Feed the cutter in the +X direction. When the cutter has cleared the block by approximately ¼" double tap the X button to zero the X axis.



Operation

Horizontal End

This is the programs ending location in the X axis. Since we are setting up on the right-hand side of the machine this number will be negative. To set this value the operator can either jog the machine to the left end of the block so the cutter to be used is passed the end of the block and then press SET to save the current location. Or if the length of the block is known then the operator can click the box next to the SET button and type in the length of the block plus the radius of the cutter to be used for the end location.

Amount Per Pass

This is the maximum depth of cut to be performed each time the cutter makes a pass across the block surface. This is used to set the depth for any roughing cuts that will be performed.

Vertical Start

This is the Vertical Position the machine will start cutting at. This value is usually Zero which is usually the starting Deck Height.

Vertical End

This is the Vertical Position the machine will stop cutting at. It is the Total amount of material you want to remove in the Milling process.

Copy Lowest/Copy Highest

These buttons are used in conjunction with the probe with mill for probing

Rough Settings

These values are used when taking multiple passes are necessary to remove material before finishing the block surface. Typically the rough settings will use more aggressive cutting parameters to remove material efficiently.

Rough Feed Rate

The desired roughing feed rate in IPR

Rough Spindle RPM

The desired roughing spindle speed in RPM

Finish Cut Settings

These are the cut parameters that the machine will automatically switch to when performing the final pass

Finish Amount

The amount to be removed on the last pass.

Finish Feed Rate

The desired Finish Feed Rate in IPR.

Finish RPM

The desired Finish Spindle RPM.

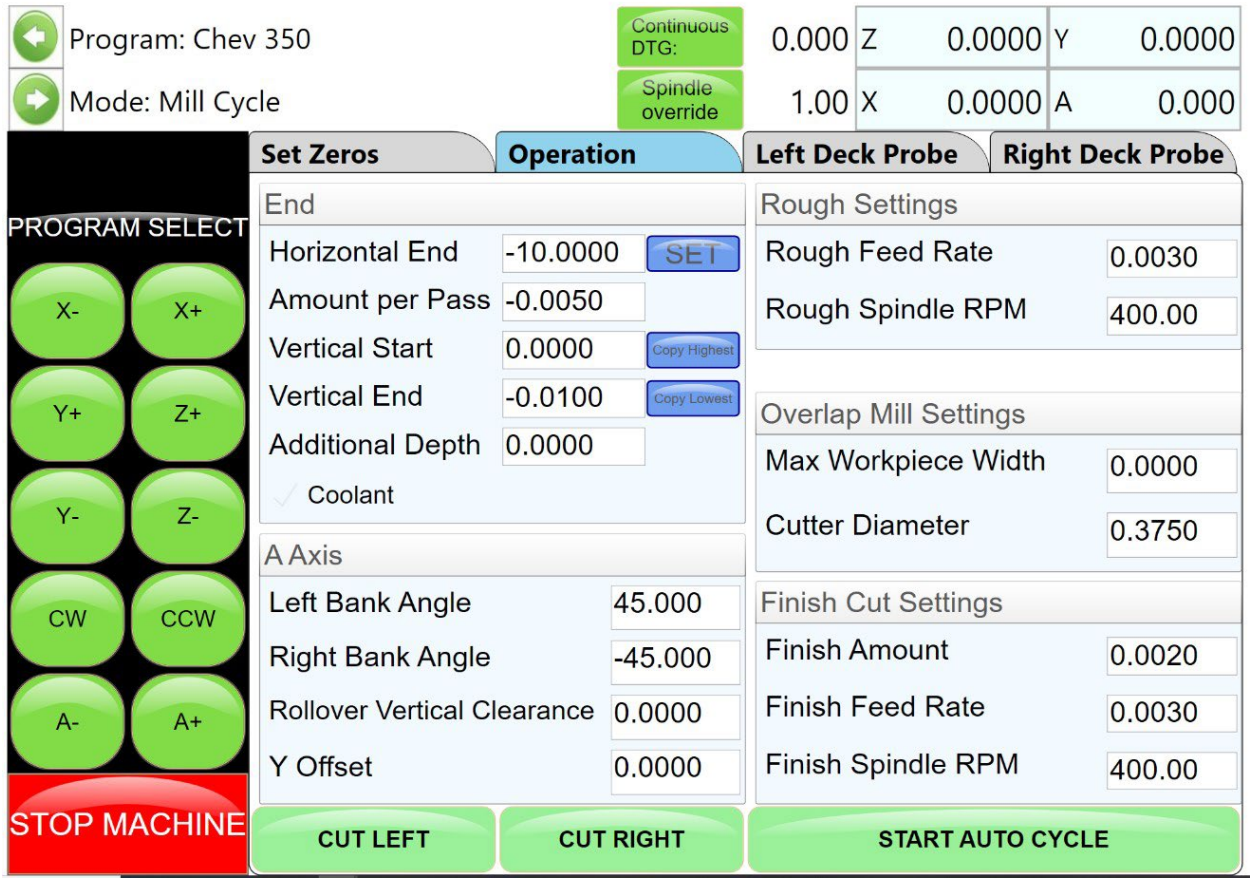
A-Axis

This is controls the 4th axis if machining a V-block.

Overlap Mill Settings

This is used if the cutter to be used cannot machine the entire width of the block in one pass.

NOTE: You do not need to have evenly divisible numbers in these sections. The computer will do the math to remove the maximum allowable material each pass while still using the specified finish cut settings.



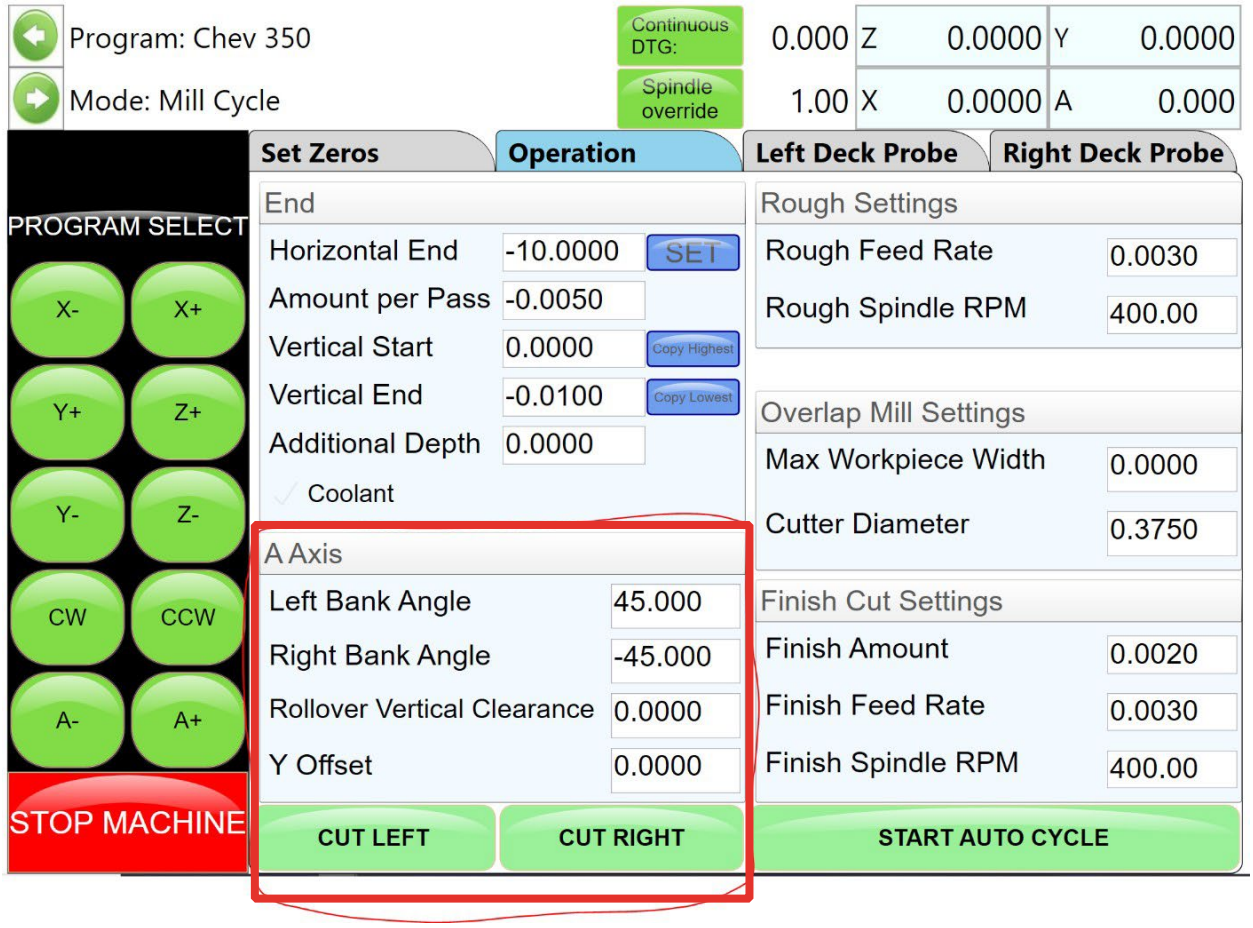
Start Auto Cycle

Pressing this button will start the machines automatic cycle. The cycle to be run is determined by the setting on this page. If you only require one pass to be made, do not enter any values into the Rough Setting, only the Finish Cut Settings.

Mill Mode 4th Axis

Program Additions to 3-axis Operation

The Program setup for a 4th Axis operation is largely the same as the programming for a 3-axis operation. The only difference is the addition of the A-Axis settings. When surfacing a V-block we first setup the surfacing operation on one deck and then fill out the A-axis settings to have the program repeat on the second deck.



4h Axis Angles

Left Bank Angle

Enter the angle of the Left Deck. This is the angle of the block in reference to the Cam and Crank bore being lined up Vertically.

Right Bank Angle

Enter the angle of the Right Deck. This is the angle of the block in reference to the Cam and Crank bore being lined up Vertically.

Rollover Vertical Clearance

Enter the value the Fly Cutter will have to move up vertically to clear the block when it rolls over from bank to bank. Make sure the block can completely rollover when in this position for safety

Cut Left and Cut Right

Pressing these buttons will cause the machine to run an automatic cycle (per the parameter defined in the Operations page) on the associated bank.

Start Auto Cycle

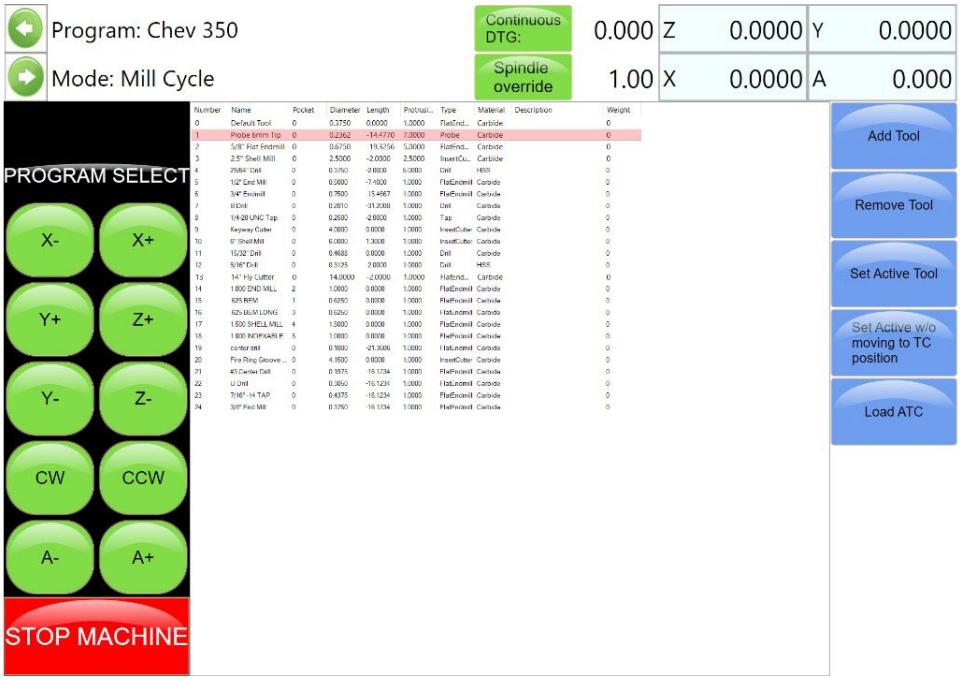
Pressing this button will start the machines automatic cycle. The cycle to be run is determined by the setting on this page. If you only require one pass to be made, do not enter any values into the Rough Setting, only the Finish Cut Settings.

Milling Using Automatic Deck Probing

The Rottler Milling program is set up to Automatically Probe the Deck height of a block and then Mill it to a set Deck Height. This can be done with both 3 and 4 axis operations.

Table of Tools for Milling

You MUST use the Table of Tools if you want to Automatically Probe the deck height and cut it to a set height. If you are unsure about how to setup the table of tools and set tool length compensation, then consult the Table of Tools section of the operator's manual before proceeding further in this section.

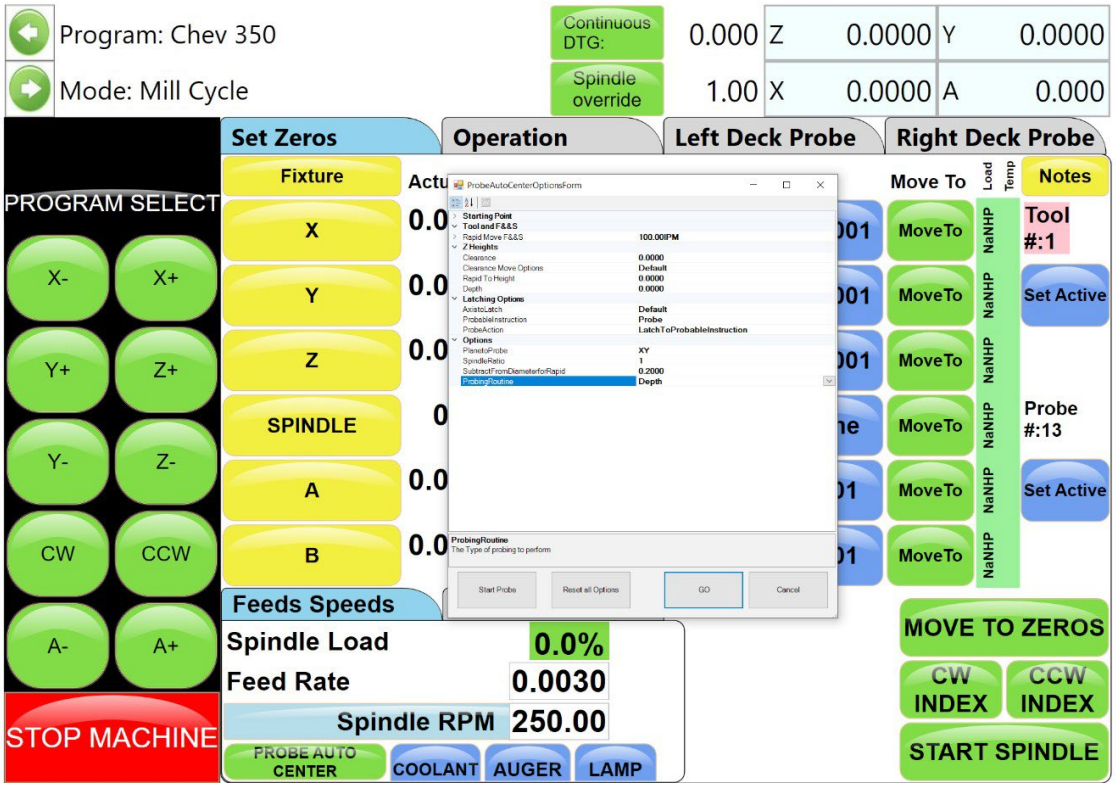


Building a Program Using the Probe and Table of Tools

Using the probe allows the programmer to setup different blocks that may require different tooling using only the probe to find locations and set zeros. Using the probe speeds up the setup time while also increasing the accuracy and precision of the blocks being processed. When surfacing we can use the probe to automatically find the lowest relative point of the surface and then use that gathered data to let the machine figure out how much material to remove.

Probe Auto Center

The probe auto center button can be found in the feeds and speeds tab within the set zeros page. Clicking this button will open the probe auto center form where the various probing routines can be called upon while jogging the machine. The Depth, Inside Diameter, and Outside diameter routines are the most used routines for performing engine work.



Probe Depth

Probes a surface in one point by moving down in the z-axis and stopping with the probe tip just touching the surface. When setting up a probing routine, you must run this operation and zero the z-axis while the probe tip is touching the surface to give the probe a starting reference.

Probe Inside Diameter

Probes 4 points 90 degrees offset from each other to find the center line of a circle in one plane.

Probe Outside Diameter

Probes 4 points 90 degrees offset from each other on the circumference of a circle to find the centerline in one plane.

Probing Engine Block Surfaces

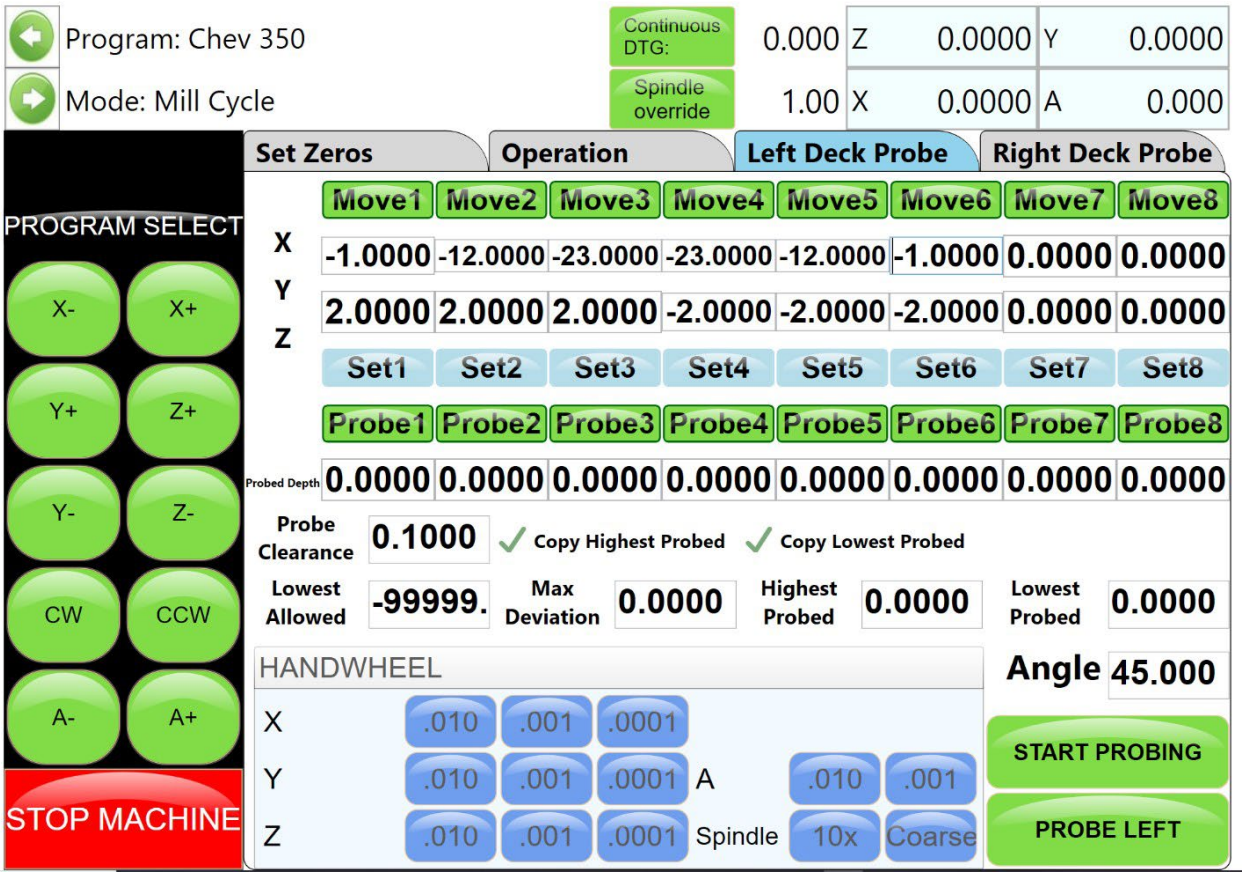
To probe the engine block deck surfaces we can use the left deck probe and right deck probe tabs from within the block software to define locations for the probe to move to and then run the depth probing routine

Left Deck Probe

Enter the positions you want the Probe to probe here. You can physically move the probe to the locations on the bank you want to probe and hit the set button also.

Right Deck Probe

Roll the block over to the Right Bank. Enter the positions you want the Probe to probe here. You can physically move the probe to the locations on the bank you want to probe and hit the set button also.



Probe Clearance

Position in the z-axis that the probe will return to when moving between points.

Lowest Allowed

Lowest Z-axis position that the machine will lower the probe to before stopping if it doesn't contact a surface.

Auto Probing

Press the Start Probing button. The machine will first probe each programmed location on the left bank and record the height. The spindle will move to Vertical Clearance height and the block will roll over to the right bank and probe the programmed locations and record them. The block will then roll back over to the Left bank and the spindle will move to the first Left location and stop.

Auto Milling

After the points have been probed the values gathered from the probe can be copied into the depth to cut within the operation tab.

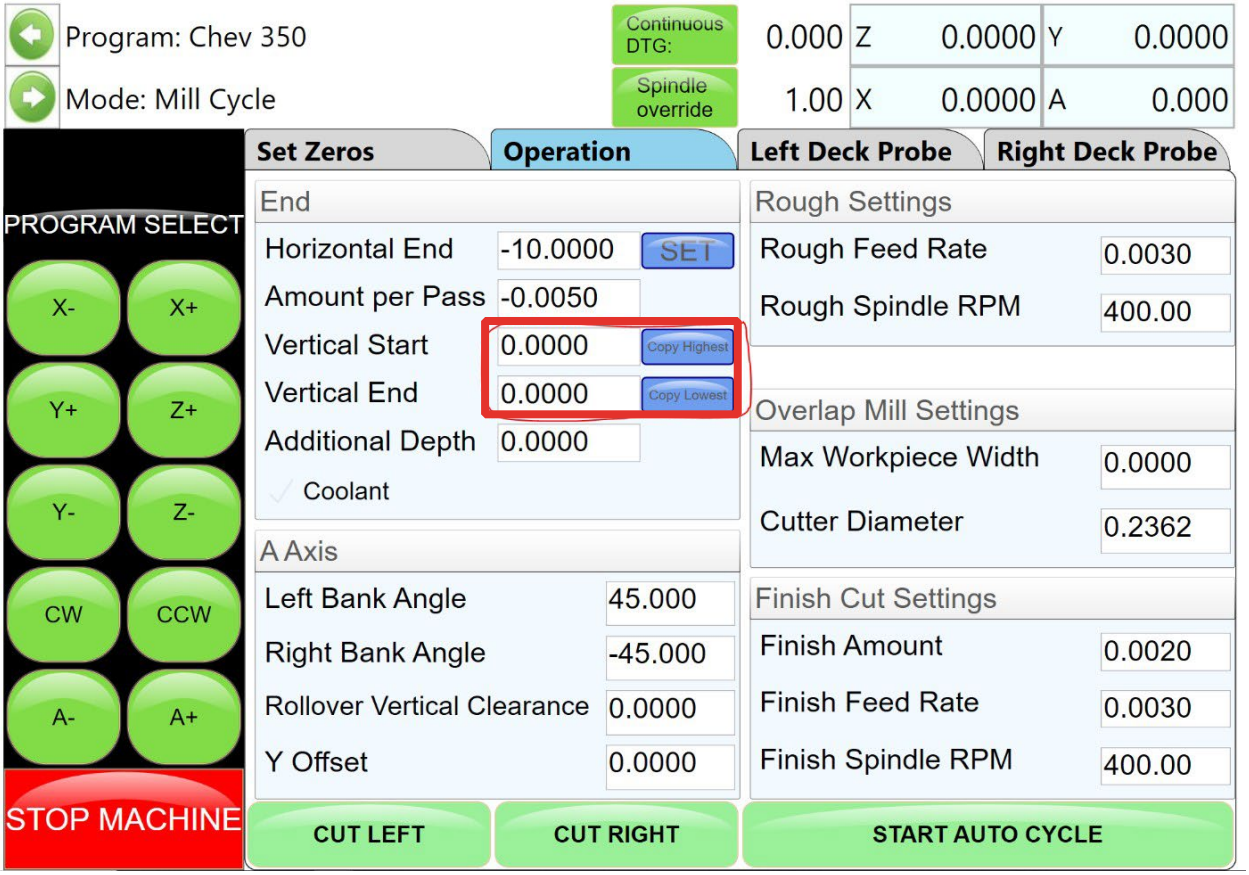
Vertical Start

Press Copy Highest next to Vertical Start. This will copy the Highest Probed point of either bank. This is the Height at which the Start Auto Cycle would start the first cutting pass.

Vertical End

Press copy lowest to copy the lowest point from the probed data into this box. This can be used to perform a minimum cleanup on the block, as the finish cut will end at the lowest probed points height. If

more material is to be removed, then the amount below the lowest point that the operator would like to cut can be entered manually in the box as done in previous methods.



Start Auto Cycle

Pressing this button will start the Auto Cycle for Both Banks. First the Left bank will be cut to the set parameters. The spindle will go to the Clearance Height and Roll over to the Right bank and cut it to the set parameters. The Spindle will again go to the Clearance Height and roll over to the Left bank. The machine will go idle at this point.

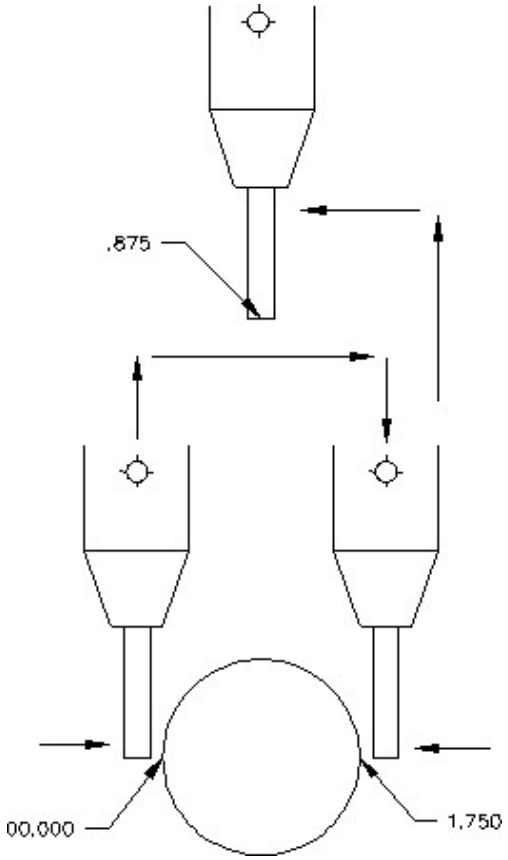
Lifter Bore Mode 3 Axis

Lifter Bore programs are built the same as described in the Bore Mode 3 Axis. The differences in locating the bores and tooling will be discussed in this section.

Y-axis Zero (In/Out)

The Y-axis zero position for lifters is the center line of the Cam Bore. An easy way to find the center of the cam line is to use the electronic probe. The following is an example of this procedure. Install the probe into the holder and the holder into the spindle. Bring the probe down until it is in the approximate center of the cam Bar Vertically. Press the Vertical Zero button now (this is only a temporary Vertical Zero position). Jog the Y-axis with the handwheel to bring the probe up to the Cam Bar until it lights. Press the Y-axis zero button here. Move the spindle up enough to clear the Cam Bar, move the probe to the other side of the Cam Bar. Bring the vertical down to the zero position. Hand wheel the probe into the Cam Bar until the light comes on. Note the Y-axis position reading. Divide this reading by two. Bring the spindle up until it can clear the Cam Bar. Use the Y-axis handwheel and move the Y-axis position until it matches the divided number. This is the center line of the Cam Bar. Press the Y-axis Zero button now. The Y-axis zero position has been set. The following illustration visual shows the above description.

Start Boring Height



Pay attention when setting this height, there are often protrusions in the casting that will not allow the end mill to travel unobstructed all the way to the start of the lifter bore. It is safest to set the Start Boring Height above the Deck.

Lifter Bore Angle

Rottler has specific Lifter Bore spacers that are installed on the Cam bar to set the correct angle for lifter boring when using the Performance Fixture.

Lifter Bore 4th Axis

Lifter Bore programs are built the same as described in the Bore Mode 4th Axis. Only the differences will be discussed in this section.

Start Boring Height

Pay particular attention when setting this height, there are often protrusions in the casting that will not allow the End Mill to travel unobstructed all the way to the start of the lifter bore. It is safest to set the Start Boring Height above the Deck.

Lifter Bore Angle

The angle for each bank is located on the associated Locations page. Press the angle numerical value and a pop-up will open so you can type in the Lifter Bore angle.

Calculate Y-axis zero

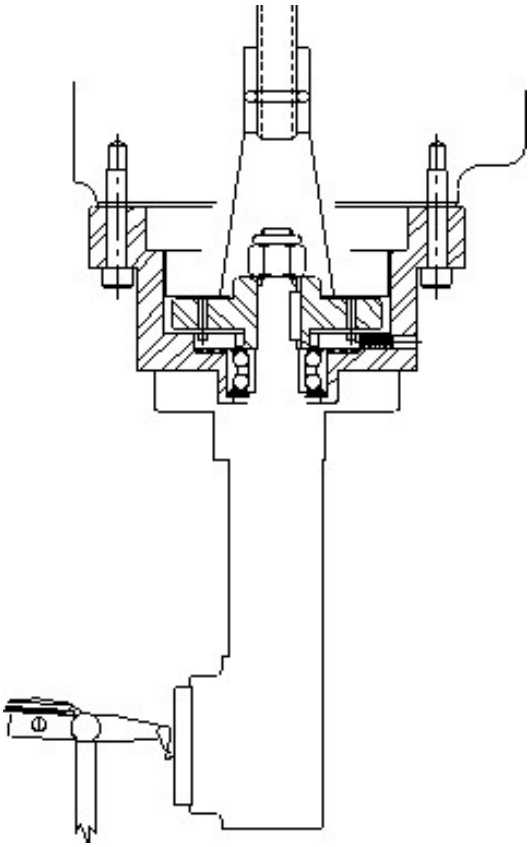
This button is located next to the Y-axis locations for each Bank. You must first have the correct angle entered in the angle data box. Then press the calculate In/Out button. A window will open where you

enter the center to center distance of the Cam to Crank bores. The In/Out locations will automatically be filled in.

Line Bore Mode

Create a Line Bore Operation in the mode select box from the main screen.

Mounting and Aligning the 90 Degree Head



Mount the 90-degree head onto the spindle and just snug mounting bolts. Use the following instructions to align the head. Mount a test indicator to the machine table or block. Align the indicator needle as shown in the figure below on the tool mounting surface.

Using the Y-axis handwheel increment move the indicator from one side to the other noting the amount of difference. Adjust the 90-degree drive until the variance across the face is less than .0005”. Tighten the mounting bolts for the head and check the surface again to be sure it did not shift when tightening the head.

Setting Zeros

The purpose of setting zero points is to give the operator a specific point to build programs from. The machine also uses these zero points to run the program from. The zero points can be set at any point in the machines’ travel. Each axis (except the Spindle rotation) will need to have a zero point set for the machine to operate from.

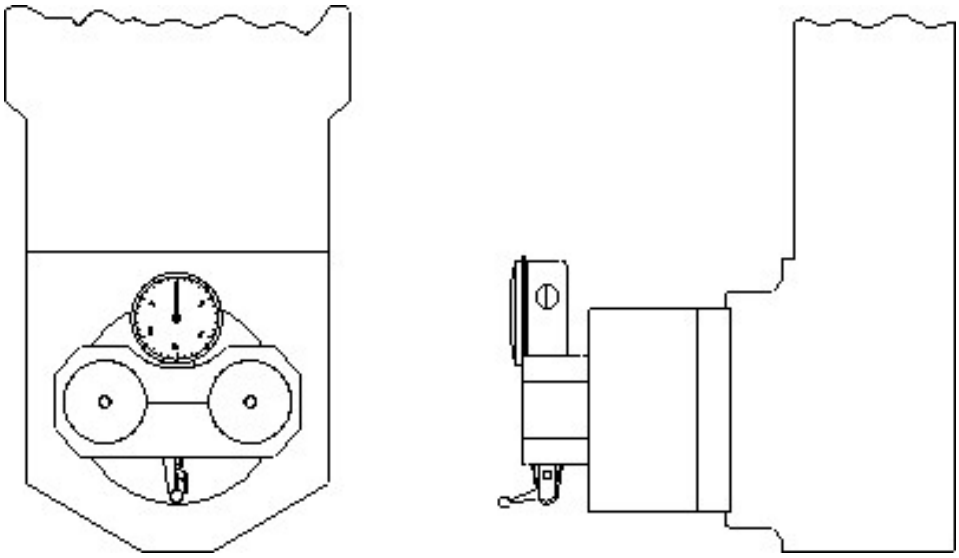
X-axis (Horizontal Zero)

The Horizontal should be set about .050” offset from the front of the first main to be bored, making sure that, that position will allow the head to travel up without interference. Bring the head down and roughly center it in front of the first main. It does not need to be

perfectly centered to set the horizontal zero. Double tap the X button from the set zeros tab to set the horizontal zero.

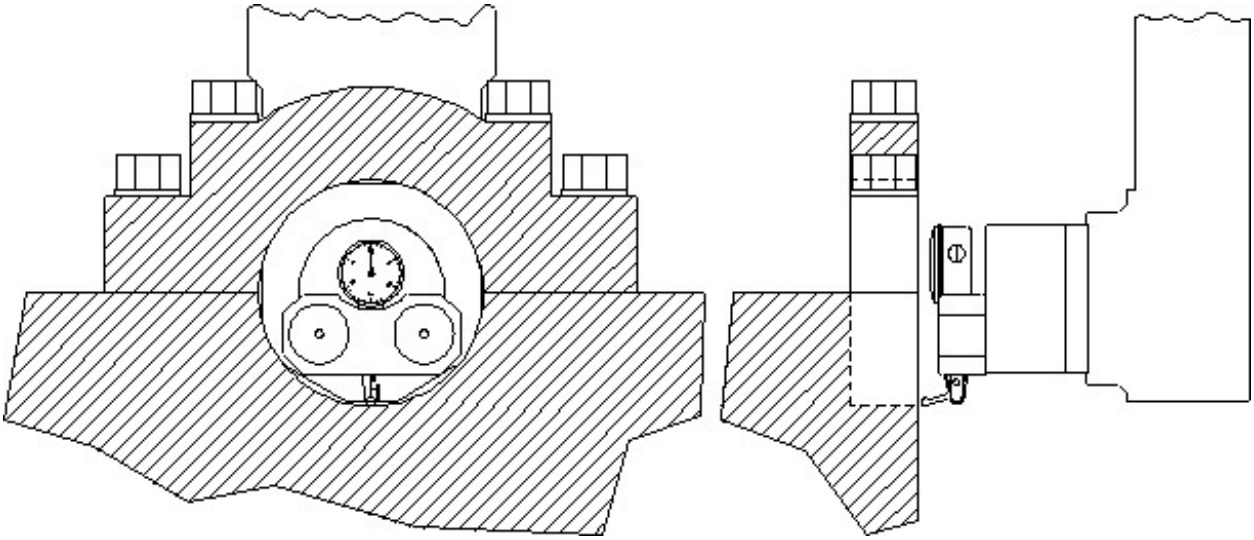
Y-axis (In/Out Zero)

Locate the supplied indicator and small magnetic base. Mount on cutterhead as shown below.



Using the X-axis handwheel increment option move the indicator inside the main bore, making sure the indicator is not touching the main bore at this point. You will be indicating both sides and the bottom of the saddle, generally the cap is not used while indicating the bore.

Physically move the indicator and mag base on the cutterhead until there is about .010” pressure on it. Start rotating the spindle CW and CCW watching he indicator. Move the machine in the Y-axis (In/Out) with the handwheel increment until the indicator readings are the same on both the sides and the saddle locations of the main. Once equal double tap the Y and Z buttons on the set zeros tab to set the zero locations for the program



Programming Vertical Stops

There are two (2) vertical stops used in the Line bore mode. These are Bore Centerline and Block Clearance.

Bore Centerline

The first vertical stop is on the main bore centerline. This is the same as the vertical zero that is set when indicating the block main in the previous section

Block Clearance

The block clearance height is the height that the spindle will retract up to before moving to the next main bore location. This height should be set somewhere above the block where the 90-degree head will not interfere with the block during horizontal movements.

Programming Horizontal Stops

The Horizontal Zero was set .050” before the first Main Bore. The first Horizontal stop will be 00.000. Measure the distance between each main and enter it into the corresponding stop number.

Programming Bore Length

Measure the length of each Main Bore and enter that value into the corresponding length box.

Running the Auto Cycle

You will need to set a Feed Rate and Spindle RPM on this screen to run an auto cycle. After this is done press the “Move to Zeros” button. The spindle will move up the Vertical Block Clearance distance if it is not already there. It will then move to the Horizontal and In/Out axis to the zero position. The vertical will then move down to the zero position and stop.

CAUTION: If you press the MOVE buttons or the Cycle Start button the machine will not move the In/Out axis to the zero position. You need to move the In/Out axis to the zero position manually before you press Cycle Start the machine will go idle at this time. Pressing the “Start Auto Cycle” button will cause the entire cycle to run. After a program has been completed the machine will move the spindle over to the first Main Bore at the Clearance Distance.

Thrust Cutting

Refer to Line Bore in this section for mounting the block and aligning the 90-degree head.

Note: It is important to read through the entire Thrust Bearing Cutting section before entering any values or starting the Auto Cycle. You will better understand how the program operates and how the values effect the operation of the Auto Cycle.

The Thrust Cutting program can cut a single or double thrust face using circular interpolation.

Select the Thrust Bearing Cutting button from the Main Menu. This will bring up the Thrust Bearing Cutting Bore Mode with the Set Zeros tab shown.

Setting Zeros

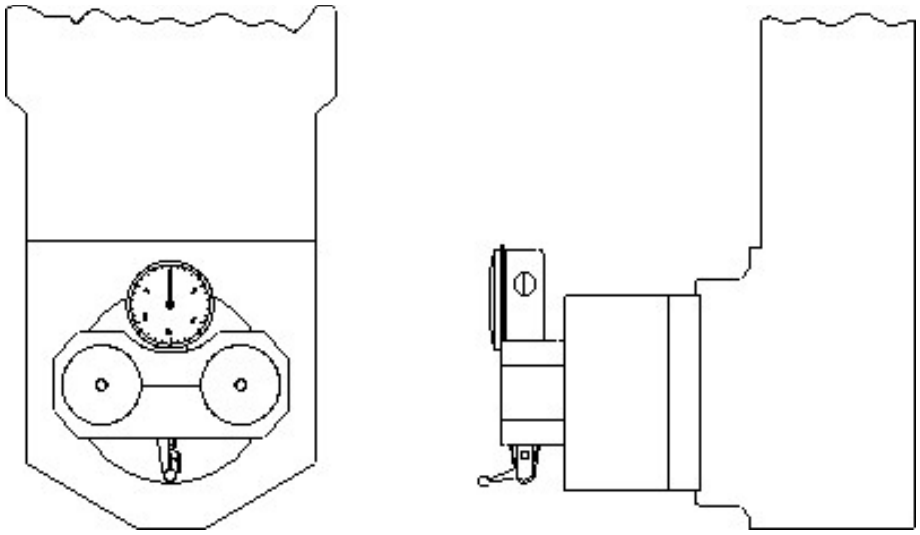
The purpose of setting zero points is to give the operator a specific point to build programs from. The machine also uses these zero points to run the program from. The zero points can be set at any point in the machines' travel. Each axis (except the Spindle rotation) will need to have a zero point set for the machine to operate from.

X-Axis (Horizontal Zero)

The Horizontal should be set with the cutter to be used just touching the thrust face. Use the handwheel increment buttons to jog the cutter into this locations and then double tap the X button to set the program zero.

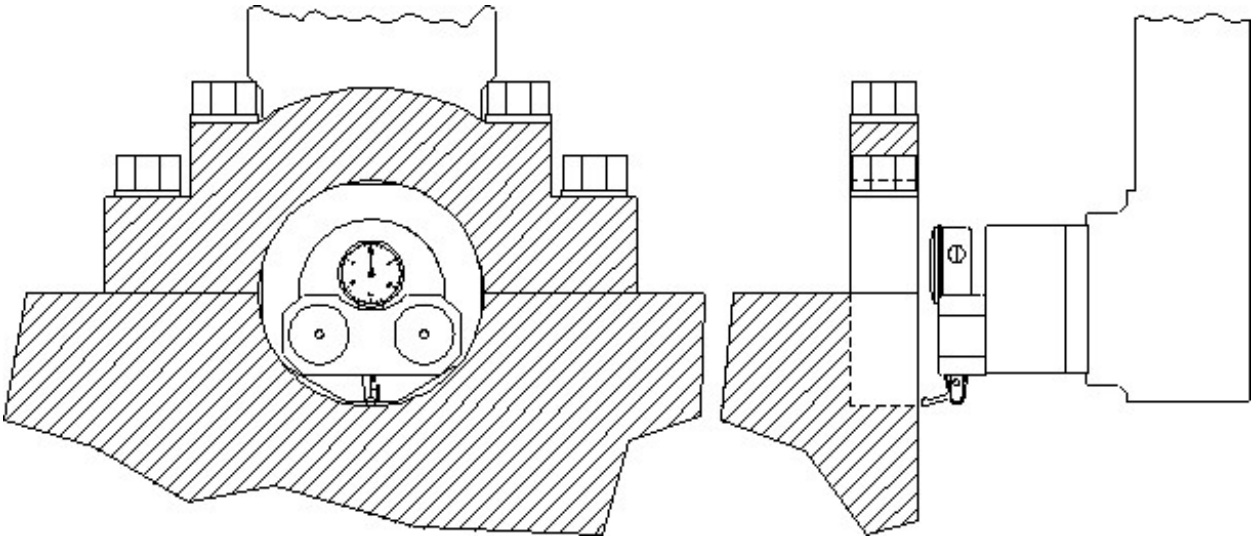
Y-axis (In/Out Zero)

Locate the supplied indicator and small magnetic base. Mount on cutterhead as shown below.



Using the X-axis handwheel increment option move the indicator inside the main bore, making sure the indicator is not touching the main bore at this point. You will be indicating both sides and the bottom of the saddle, generally the cap is not used while indicating the bore.

Physically move the indicator and mag base on the cutterhead until there is about .010” pressure on it. Start rotating the spindle CW and CCW watching he indicator. Move the machine in the Y-axis (In/Out) with the handwheel increment until the indicator readings are the same on both the sides and the saddle locations of the main. Once equal double tap the Y and Z buttons on the set zeros tab to set the zero locations for the program.



Dimensions & Auto Cycle

The program tab is where the critical dimensions are set for the thrust cutting process. Switch to this tab to enter the cutting dimensions and parameters.

Thrust Dimensions

Outside

This is the outside diameter of the thrust face to be machined.

Inside

This is the Inside diameter of the thrust face to be machined.

Cutter

This is the diameter of the cutting tool to be used.

Clearances

Z (Vertical)

This is the distance, from zero, the 90-degree head will have to travel up to clear the main caps on the block.

X (Horizontal)

This is the distance, from zero, the 90-degree head will have to travel to clear the main for the next vertical move.

Dimensions

Main Width

Width of the thrust face to be cut, this is the outside diameter minus the inside diameter.

Insert Width

This is the width of the cutting surface of the insert being used. This is used to calculate the step over required for interpolation.

Left Depth of Cut

This is the Inside diameter of the thrust face to be machined.

Right Depth of Cut

This is the diameter of the cutting tool to be used.

Cut Left Side

Cuts the left side thrust faces

Cut Right Side

Cuts the right side thrust faces

Cut Both Sides

Cuts both thrust faces.

Description and Running of the Auto Cycle

You will need to enter the Feed Rate and Spindle RPM the program will run at. There are no Move to buttons in this program. You **MUST** be at the zero positions when the Auto Cycle is started.

Start Auto Cycle

When you are at the corresponding zero positions press the Auto Cycle. The vertical feed will start at the programmed rate in an upward direction until the correct Outside diameter is reached. The circular interpolation will start at this point and go 360 degrees. It will then continue the circular interpolation back towards the center of the Main to clear the cutting tool from the thrust face. When the cutterhead is back at the center point (zero positions) of the Main, all motion will stop. The cutterhead will then rapid travel to the left taking the main width and the cutter diameter into account to reach the correct depth on the second thrust face. The same circular interpolation process will then be repeated for the second face. The cutterhead will then retract horizontally to the clearance distance then vertically to the block clearance distance.

When the program is running the “Start Auto Cycle” button will change to “Press to Pause”. If this button is pressed the machine will pause the program right where it is. At this point the screens are locked out from changing anything. The button will the change to “Press to Resume”. If you want to resume press the button and the program will continue from that point on. If you do not wish to continue press the “Stop” button. This will put the machine back in idle mode and changes can be made to the program.

Cam End Tunnel Boring

To bore the end tunnels on a Block refer to Block End Truing Fixture 650-3-30 when used with Cam Boring for setting up the block. Select a Cam bushing that will fit the existing Cam bore and place it in the Cam Spacer. Place the distributor end of the block facing up. You will need to be in the Bore Mode on the control panel. The Cam spacer placed in the center T-Slot should put the Cam tunnel in line with the Main bore.

Center the spindle over the Main bore using the electronic probe or magnetic base with indicator. Zero the X and Y axes.

Move the table the specified blue-print distance toward the Cam Tunnel. This distance should in the blue printing specifications for the block you are working with.

Check that you are on center of the cam bore with the electronic probe or indicator.

⚠ CAUTION ***Be very careful when correcting the existing Cam bore on the y-axis. This could cause the distributor gears to be damaged.***

Install the 650-2-3F cutterhead into the spindle.

Refer to the Bore Mode, programming Vertical Stops section earlier in this guide for guidance on setting up the vertical stops for this operation.

Note: It is important to bore the Cam End bores the full length of the cutterhead on both ends. If you do not you may have trouble getting the Cam Bar to bore the full length between Cam End Bores. Run the operation before proceeding.

Hint: It is helpful and more efficient to have three (3) tool holders set up for this procedure, two (2) of them for large material removal and one for a finish pass of .020” to .030”.

Note: To bore the oil groove in the Cam Bore, refer to the Cam Bore Oil Groove section in this chapter. This is a CNC operation.

Remove the block from the fixture, select a Cam bushing that will fit the bore that was made on the distributor end of the block.

Rotate the block so that the distributor end is now facing down. Tighten the block into the fixture. The Cam spacer will put the end bores in line.

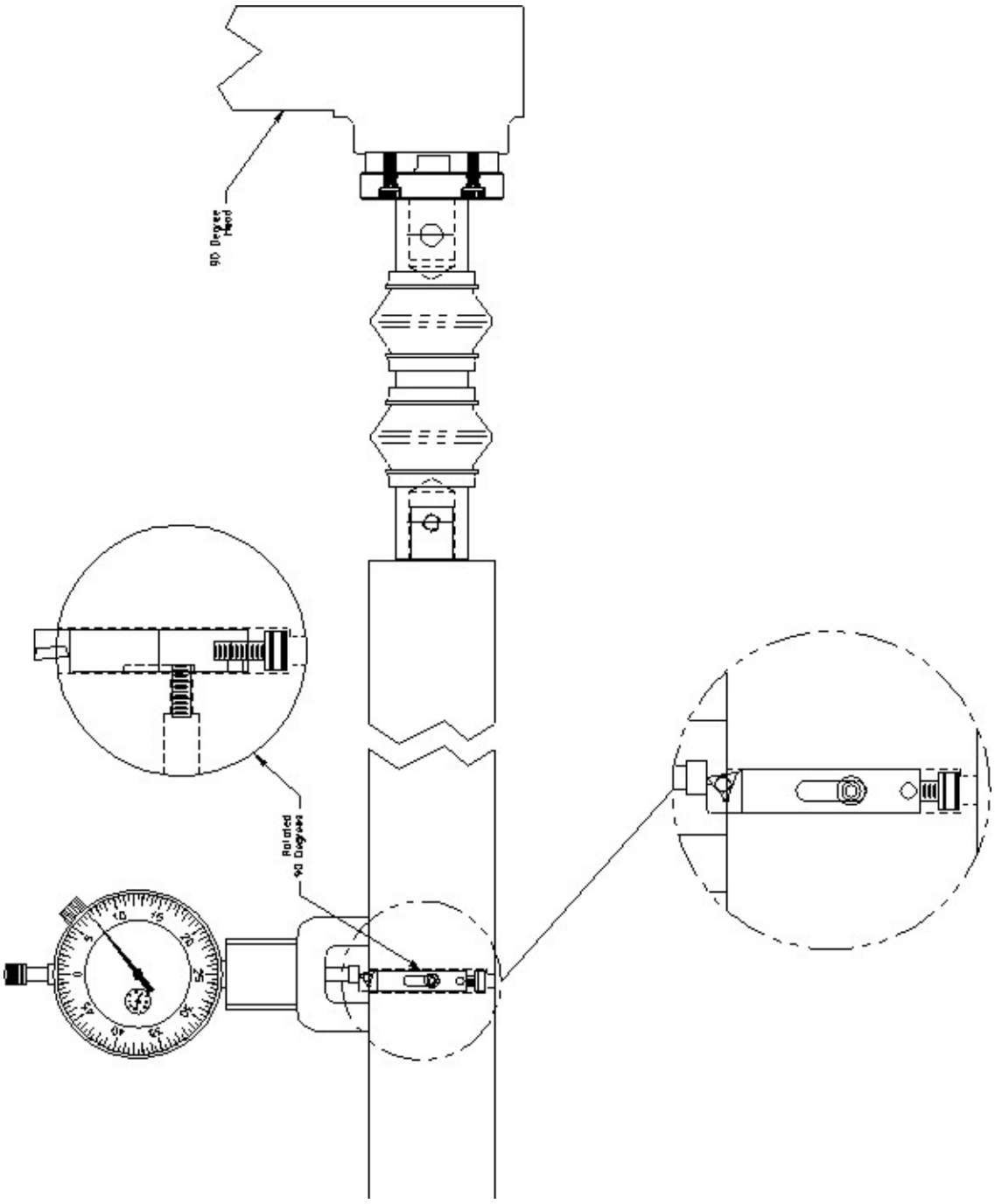
Press the move to zeros button.

Repeat the above process on this end of the block to finish boring the Cam End Bores.

Cam Tunnel Boring

To bore the center of the Cam tunnel, refer to Cam Tunnel Boring in the Block Mounting section of this chapter. Mount the block as shown.

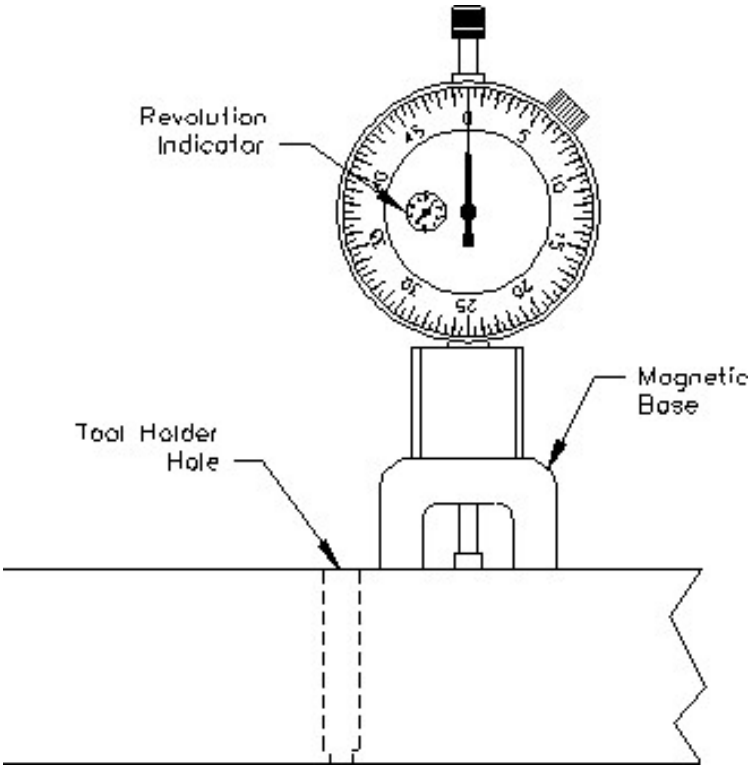
The following illustration shows the cutting tool and holder and how they are set inside the Cam Boring Bar.



Zeroing the Micrometer

Remove the magnet keepers from the bottom of the indicators magnetic base. These should be put back on when the magnet is not in use to keep the magnet strong.

Place the magnet on the smooth portion of the bar next to the tool holder hole. Set the zero on the indicators dial, noting the number of revolutions the dial has made.



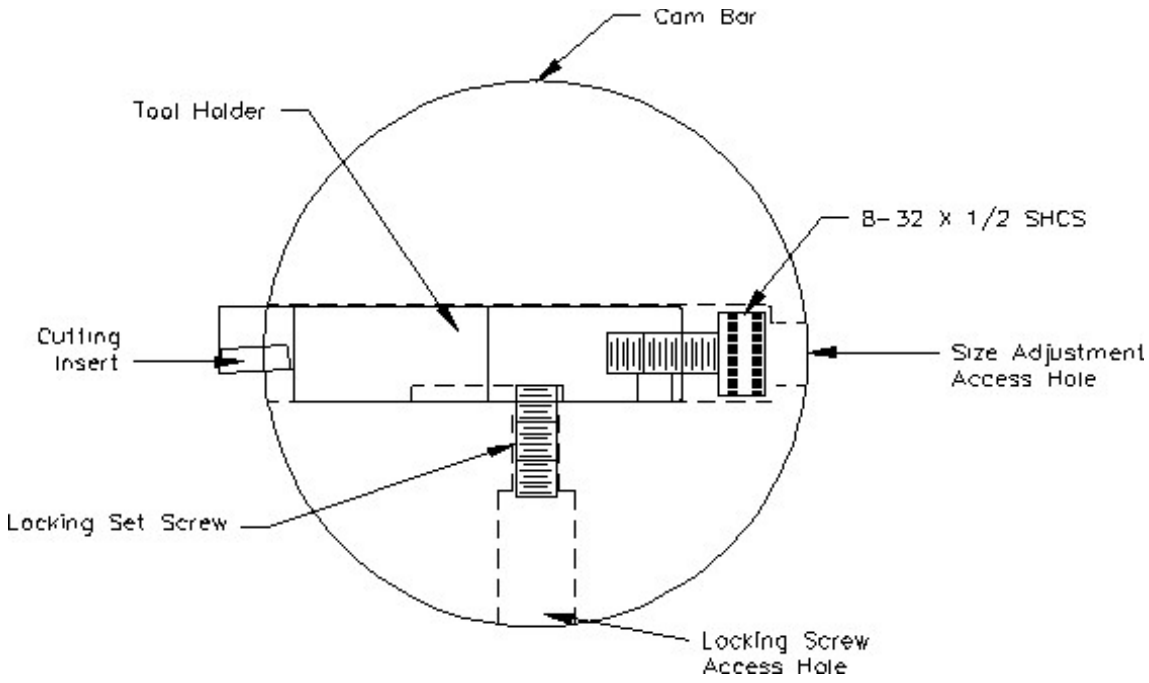
Setting Cutting Size

The diameter of the Cam Bar 650-2-32D is 1.7500”. The 8-32 X ½” socket head cap screw on the back of tool holder is used to adjust size. When the tool holder is inserted into the Cam Bar the cap screw goes against a ledge inside the Cam Bar. When the cap screw is turned in the size will get smaller. When the cap screw is turned out the size will get bigger.

⚠ CAUTION ***When adjusting the size on the tool holder, you must remember that the amount that will be taken off the diameter will be twice the reading on the dial indicator.***

When the dial indicator reads zero the bar will cut 1.7500”. Double the amount past zero on the dial indicator and add that to 1.7500” to determine the cut diameter.

Once the size has been set, lock the set screw in the Cam Bar to secure the tool.



Refer to the Line boring section of this chapter for mounting and alignment of the 90 degree head.

Select Line Bore Mode of operation.

Mount the dual flex coupling to the 90 degree head with the two (2) supplied socket head cap screws. Install one Cam Bearing Locator into the left side of the block.

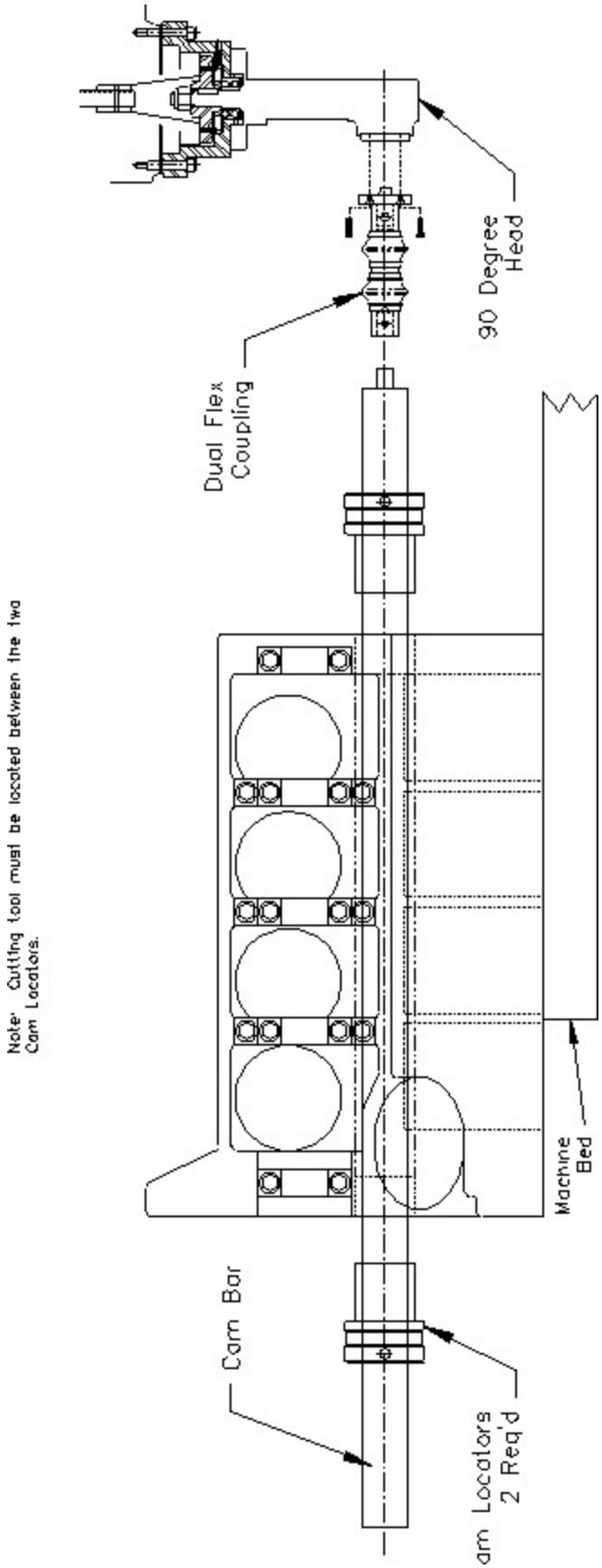
Slide the Cam Bar into the Cam Tunnel and then into the right side locator. Keep the end of the Cam Bar with the adapter on it to the right.

Slide the second locator onto the Cam Bar, then the locator into the Cam Bore. The cutting tool needs to be between the two (2) locators.

Bring the 90 degree head down and line up the end of the dual flex coupling with the adapter on the Cam Bar. This does not have to be a precise line up, the dual flex coupling will take care of any alignment variance. Tighten the socket head cap screw on the dual flex coupling on to the adapter on the Cam Bar.

Press the Vertical, Horizontal and In/Out zero buttons.

Final set up should look line the drawing on the following page. The mounting components are not shown on this drawing. Refer to the block mounting section of this chapter.


90 Degree Head
Dual Flex Coupling
Machine Bed
Cam Bar
am Locators 2 Req'd
Note: Cutting tool must be located between the two Cam Locators.

Setting Vertical Stops

Make sure the machine is at the zero positions as described previously.

When using the Line Bore Mode to do the Cam Tunnel boring the vertical stops described here will never change. They must be used to run the cycle without damaging parts.

Block Clearance: -.001

Block Center Line: 00.000

Setting Horizontal Stops

All of the Horizontal stops are to remain at 00.000 when using the Line Bore Mode to do Cam Tunnel boring. The only setting that gets changed on this screen is the Bore Length for Horizontal stop 1. This will be the distance between the two (2) end Cam bores that needs to be bored out.

Auto Cycle

You **DO NOT USE** the Auto Cycle when Cam Tunnel boring. The only items that get used on this screen are the Feed Rate and Spindle RPM.

Recommended feeds and speeds will be discussed later in this chapter.

Manual Bore

This screen is used to bore the Cam Tunnel. With the Horizontal and the In/Out axis at the zero position and the Vertical at or above the Block Clearance Height, Press the BORE1 button.

The spindle will do a rapid move down to the Block Center Line position (this is only .001 so will not notice the move). The spindle and Horizontal feed will start at the programmed speed. The machine will continue boring horizontally until the horizontal position set in the Bore Length is reached. The Vertical will retract .001 and the horizontal will retract back to the zero position.

Recommended Boring Procedure

The three (3) tool holders included in this package should be used as dedicated holders. Two of them set for roughing passes and the third set for a final finish pass.

It is recommended to set the first two tool holders for a .100” pass each, then set the third tool for the finish size.

Size is not critical on the first two passes, these tools can be set and not adjusted for each use. The third tool should be checked with the dial indicator for final size each time you use it.

Recommended Feed Rate: .001 - .003

Recommended Spindle RPM: 300 – 500

IMPORTANT: You should put a light coating of light weight oil on the Cam Bar to prevent it from seizing up as it goes through the Cam Locators. At higher spindle speeds the bar heats up more.

Circular Interpolation Tool Paths

Circular Interpolation is a common tool path that refers to using a tool smaller in diameter than the desired bore size and programming the machine to move in a circular move to mill the bore to the desired final diameter. Common applications include circular pockets, counter bores or semi-circular profiles such as internal or external radii or fillets. Both the RPATH and Rottler Block software's feature operations which can perform these various circular interpolation movements. The 2 main options for circular interpolation are as follows: Pocket, and Circular Move. Each of these are discussed in more detail below. For this lesson, the Rottler Block Software will be used in the sample images.

Objectives of this lesson:

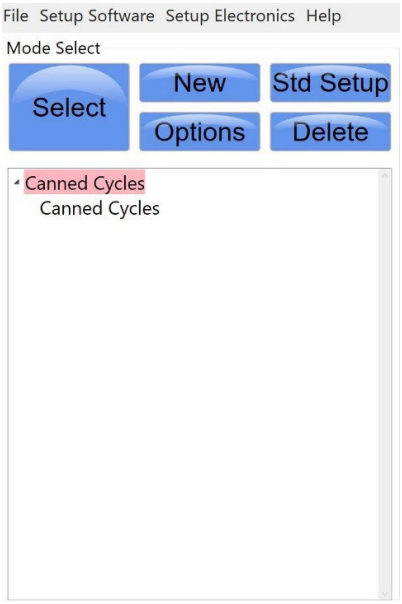
- Learn how to setup and program the 2 options for circular interpolation operations and the benefits for each method ○ Canned Cycle – Pocket
    - ○ Canned Cycle – Circular Move

*Note: All feeds, speeds, depth of cuts, and tool paths are for demonstration purposes only. The operator can change these to suit the machining philosophy of the shop and the tooling available. *

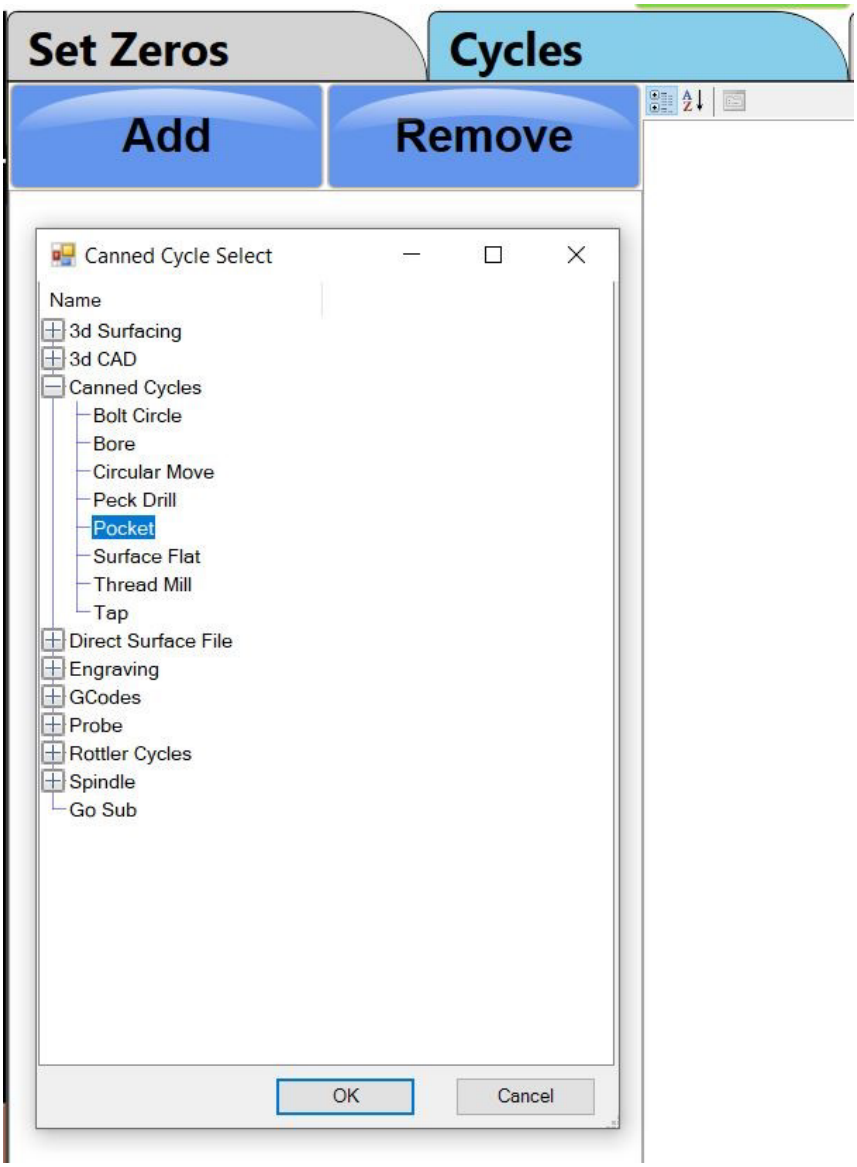
Canned Cycle – POCKET

The Pocket operation is a standard operation which can be used to remove material in a rectangular or circular recessed area such as a counter bore. This is typically used when the center of the area to be machined requires a significant amount of material removal.

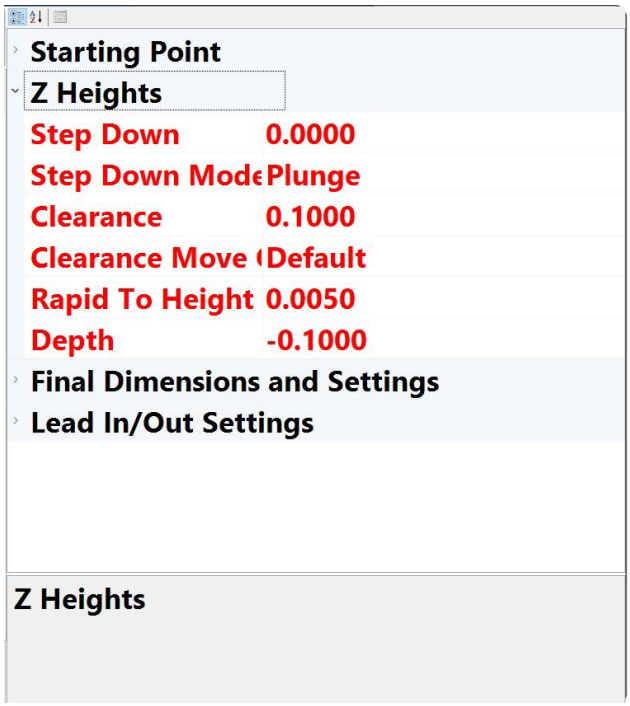
To add a canned cycle operation in the block software, select the program you wish to work form in the program selection area and then press NEW in the Mode Select area from the main screen. A pop-up window will appear, find and select canned cycle to create the mode in the block file.



Once created click on the Canned Cycles label to open the programming interface. Select the Cycles tab and you should see add and remove buttons on the left of the screen. From here select the add button and a window will appear where you can select the canned cycles drop down option which will allow you to select POCKET.



Press okay to confirm the selection. The pocket cycle will be added to the list and the properties for the pocket operation will appear in the right column of the screen in red text. These properties can be used to edit the type, size, and location for the pocket operation.



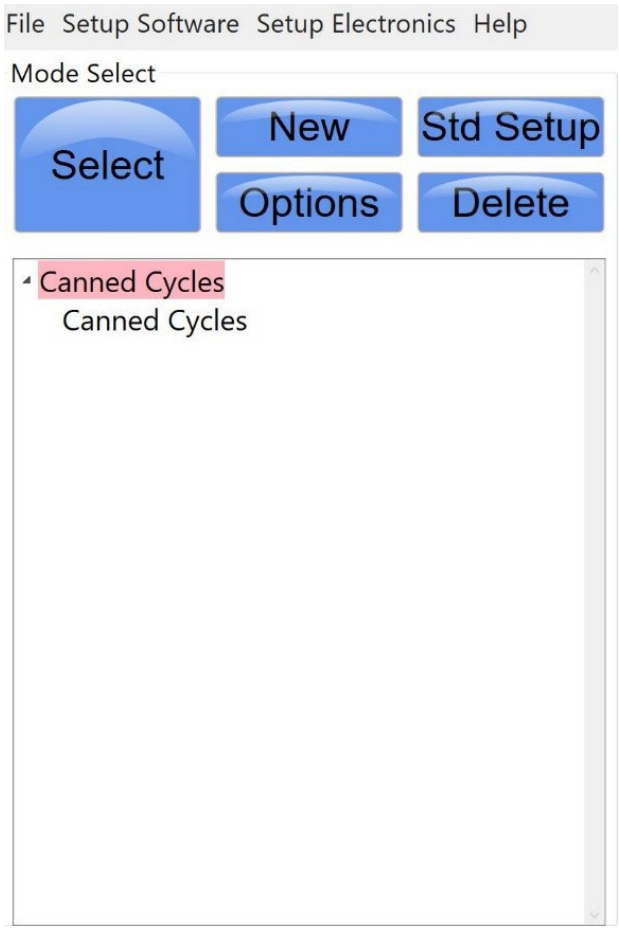
The Z Heights refer to the depth of the pocket and control where the cutter will start cutting as well as how much material to be removed per pass in the z-axis. You may also change the stepdown mode from plunge to ramp. Typically for pockets ramp is preferred.

The pocket dimensions and Pocket options are used to edit the dimensions of the pocket as well as the type. For circular interpolation change the Type of Pocket to Circular this will then use the Diameter setting for the pocket dimension. You may also change the max step out which controls the amount of material to be removed per pass radially. The cleanup option can be used to create a finish pass of a smaller amount than the standard step out.

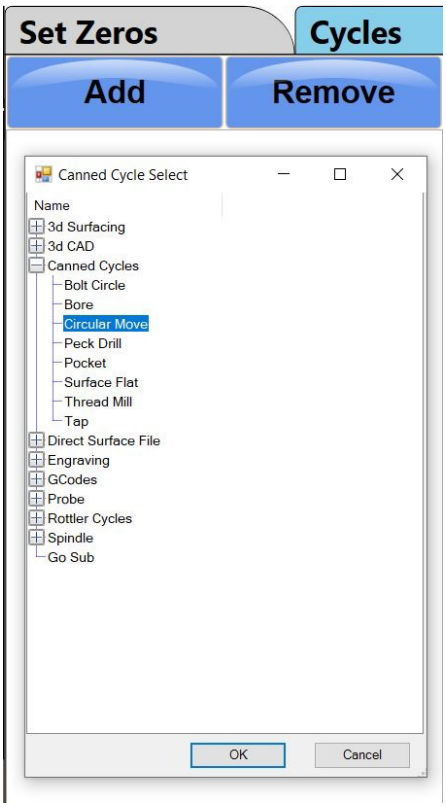
Canned Cycle – CIRCULAR MOVE

Circular move is a fixed diameter move that can be used to profile the ID or OD of a circular object. Circular move may also be used for creating a circular grove of a specific width. This is typically used to contour a single profile and clean up a surface or to create an o-ring groove.

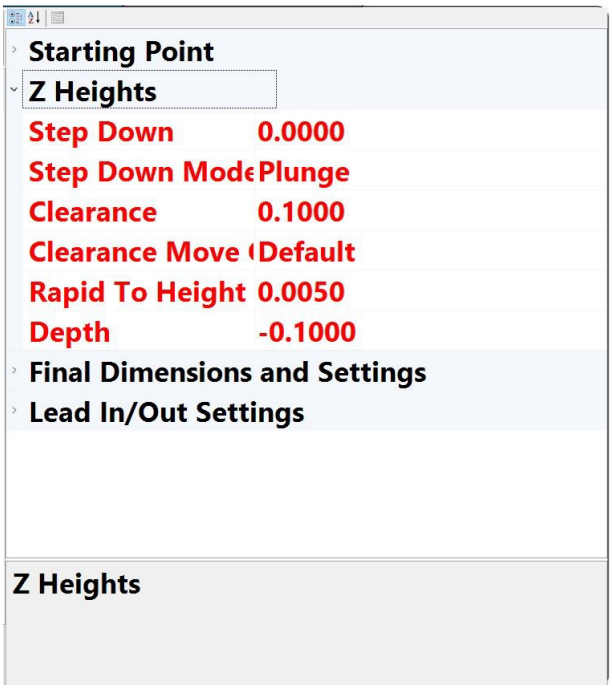
To add a canned cycle operation in the block software, select the program you wish to work form in the program selection area and then press NEW in the Mode Select area from the main screen. A pop-up window will appear, find and select canned cycle to create the mode in the block file.



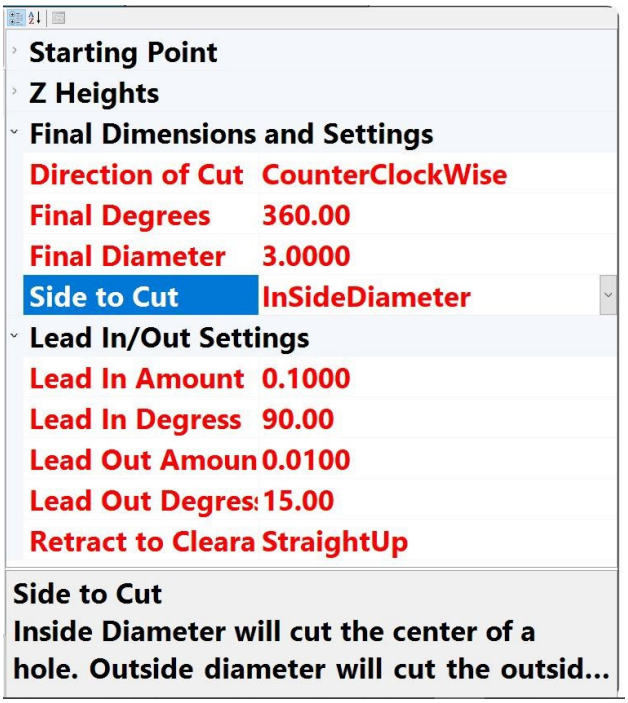
Once created click on the Canned Cycles label to open the programming interface. Select the Cycles tab and you should see add and remove buttons on the left of the screen. From here select the add button and a window will appear where you can select the canned cycles drop down option which will allow you to select CIRCULAR MOVE.



Press okay to confirm the selection. The circular move cycle will be added to the list and the properties for the circular move operation will appear in the right column of the screen in red text. These properties can be used to edit the type, size, and location for the circular move operation.



The Z Heights refer to the depth of the circular move and control where the cutter will start cutting as well as how much material to be removed per pass in the z-axis. You may also change the step-down mode from plunge to ramp. Typically for circular move plunge is preferred.



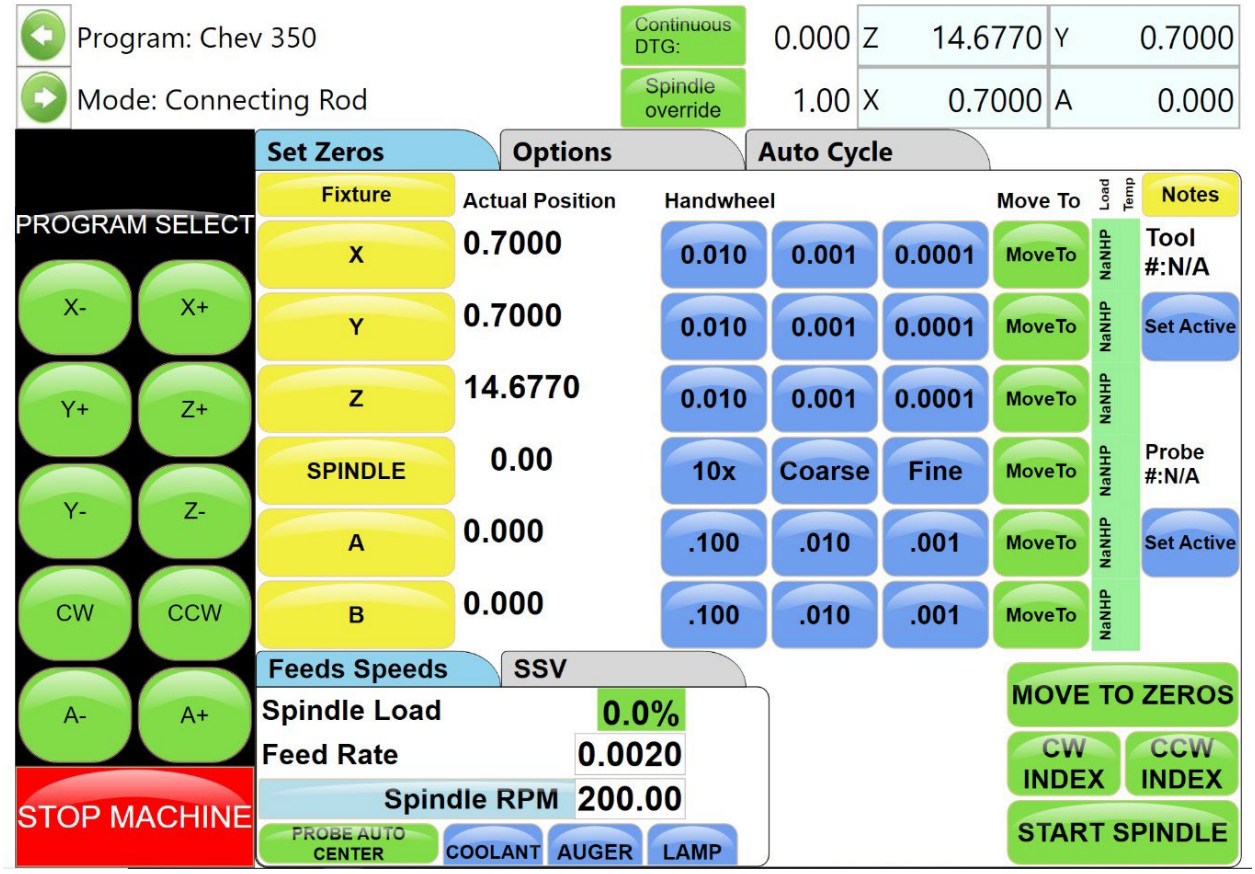
The Final dimensions and Settings are used to edit the dimensions of the operation as well as the type. For a full circle the final degrees will default to 360, this can be modified if less is to be

cut. The side to cut may also be changed to inside diameter, outside diameter, or no tool comp for grooving operations. The program will automatically account for the tool diameter so the final diameter should be set to the intended final dimension. Lead In/Out settings control the approach and exit of the cutter and can be modified should clearance be an issue.

Connecting Rod Boring

Main Screen

Select the Connecting Rod operation from within the mode select tab. This will add the operation and you may then open the connecting rod operation to bring up the standard set zeros page.



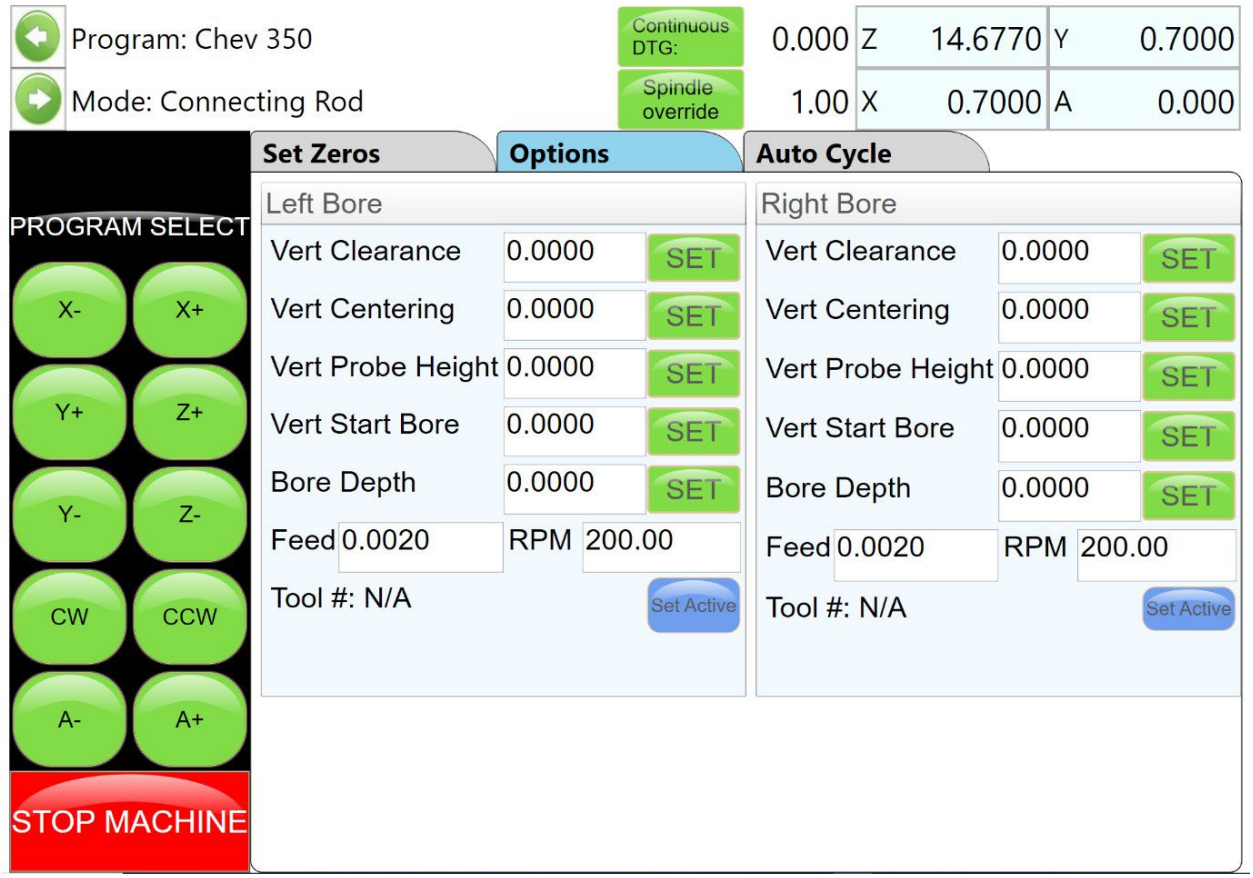
Setting Zeros

Using a dial indicator or the touch probe, find the center of the connecting rod large bore, typically this is done only on the connecting rod body side and not the cap side. Double tap the X and Y buttons to set the program zeros once the center is found.

Setting Vertical Zero

Using the electronic probe with the depth probing routine or the tool to be used. Touch off the top surface of the large bore on the connecting rod. Double tap the Z button to set the program vertical zero.

Program Options



This screen is used to input all the parameters for boring both the big and small end of the rod plus setting positions for centering and measuring.

Vert Clearance: This is the vertical height of the cutterhead with reference to vertical zero to where the cutterhead will move before any horizontal movements take place.

Vert Centering: This is the vertical height of the cutterhead with reference to vertical zero to where the cutterhead will move to allow the operator to center the machine with a dial indicator.

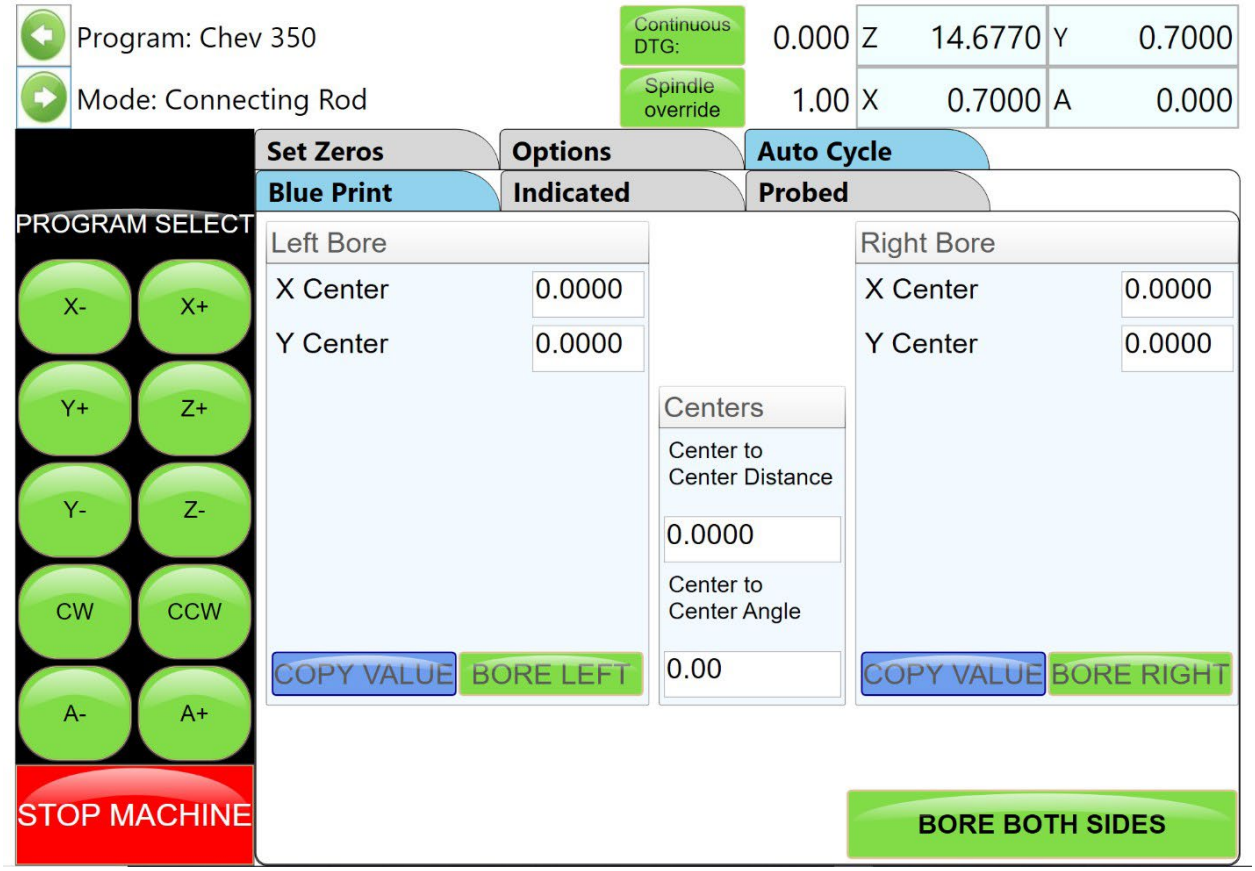
Vert Probe Height: This is the vertical height that the machine will move to before probing the bore to recenter the machine using the three-point probe routine.

Vert Start Bore: This is the vertical height of the cutterhead with reference to vertical zero to where the cutterhead will move to start boring the conrod. This is set about .040” (1mm) above the side of the connecting rod bore

Bore Depth: This is the vertical height of the cutterhead with reference to vertical zero to where the cutterhead will stop boring, index the boring tool and retract back to the clearance position after it’s finished cutting.

Tool #: Click the Tool word to bring up the tool select form and define the tool to be used for the individual bores.

Auto Cycle

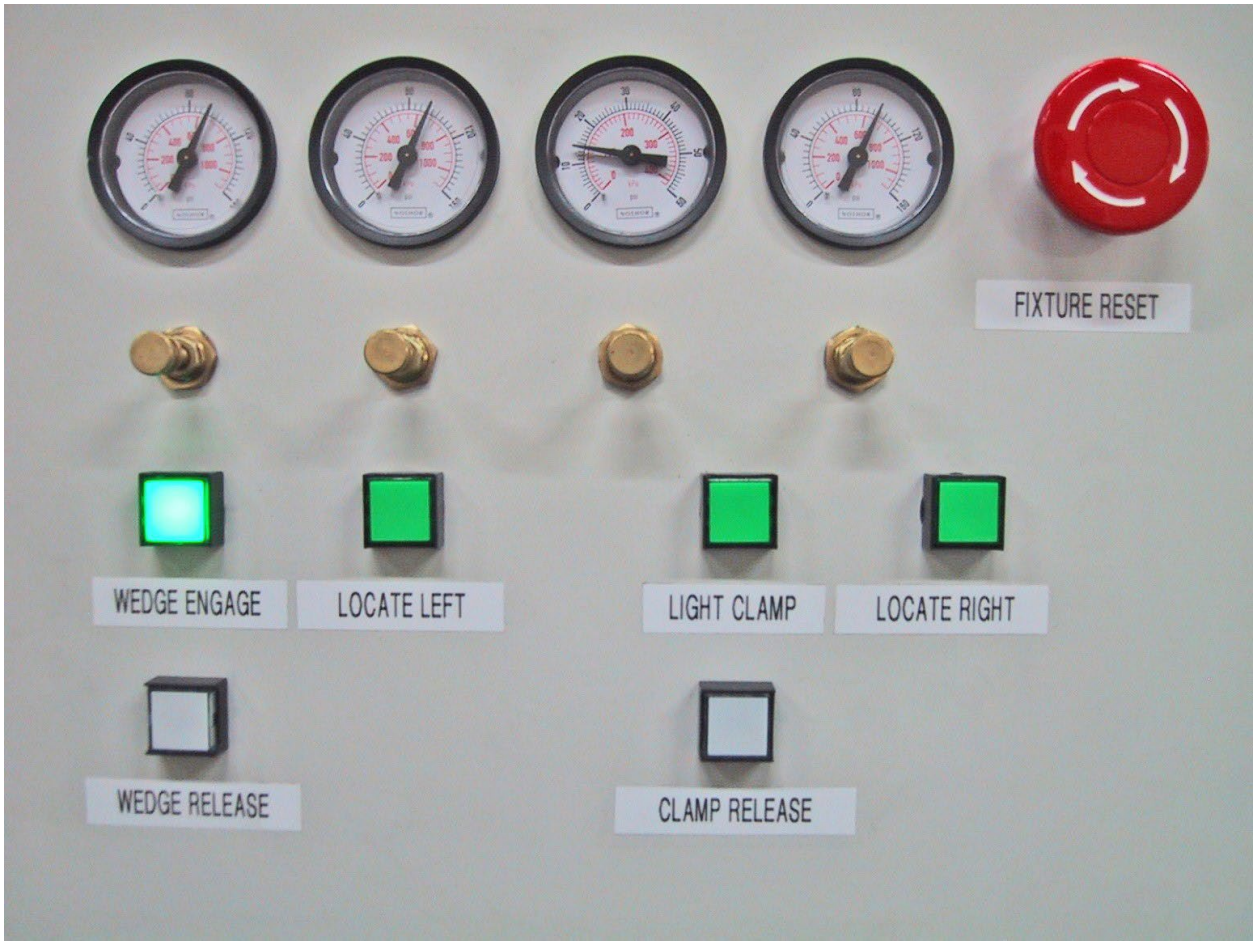


This screen is used to the connecting rod program. From the Auto Cycle tab the operator can use the Blueprint, Indicated or Probed tabs to machine using the respective methods as described in the 3 Axis Block Boring section.

Bore Both: If special cutterheads with big and small end tooling in one are being used, the block can be checked so that the machine will bore both big and small end in one cycle. Normally the big and small ends are bored with different cutterheads and this block will be unchecked.

Note: Do not stop an automatic cycle in mid cycle and then try to start it again. The CNC code running behind the Rottler screens use offsets in the controller. If the machine is stopped during an automatic cycle the machine must be shut down and restarted to clear the offsets. Otherwise the displayed position and actual position of the machine will not be correct.

Fixture Control Panel



Set Up Procedure For Conrod Fixture

Select the widest big and small end ball locators that will fit inside the conrod big and small end bores.

Press Locate Right and the right hand ball locator will move up and stop against it's end stop, then remove the air pressure supply to the conrod fixture so that the ball locator pivot arms may be moved manually.

Fit the selected ball locators to the right and left hand mounting positions. Be sure there are no chips and that the locators fix exactly in their mounting positions. Connect the air pressure to the fixture.

Press Locate Right Button, the right hand ball locator will move up and stop against it's end stop. Select the correct conrod support and place across the conrod fixture.

Place the conrod to be bored into the fixture so that the big end bore touches both the balls of the right hand ball locators.

Adjust the conrod rest so that the rod lies approximately horizontal.

Adjust the 3 big end support pads so that each support pad locates on the side of the big end and does not protrude into the big end bore. This will require removing and refitting the conrod to be sure the 3 support pads are correctly located and their hold down cap screws are tight.

Readjust the conrod support to allow the conrod to lie horizontal with no rock or tilt of the conrod on the 3 big end support pads.

Remove the conrod from the fixture.

Press locate left and the left hand small end ball locating device will lift up.

When the left hand ball locating device is at it's end of travel, place the conrod back in the fixture and adjust the left hand slide assembly so that both the left hand locating balls contact the bore inside the conrod small end.

Remove the conrod from the fixture.

Slide the left hand locating assembly approx 1/2" (12.7mm) to the right and lock both hold down handles securely, this will ensure that the small end ball locators contact the small end with some preload. Place the conrod in the fixture.

Position the clamp arms so that their feet are approx 1/8" (3mm) above the side of the big end, be sure that hey do nor protrude into the big end bore to be machined and adjust their travel limit stops and lock the lock nuts.

Press the Light Clamp button, this will places light clamping pressure on the clamp arms and lightly hold the conrod down against the 3 support pads under the big end of the conrod.

Press the Locate Left button, the small end ball locator will move up and contact the bore of the small end of the conrod and firmly press it against the big end and straighten the conrod along the center of the fixture.

Select a set of wedges that will allow the outside of the small end of the conrod to be supported during boring so that there is no chatter or vibration during boring.

Press Wedge Engage button, the wedges will be pressed against the outside of the small end.

The conrod is now ready to be bored.

Air Pressure Settings

Right Hand Side Air Gage:	Locate Right	100psi (6.5Bar)
Second from Right Hand Side:	Light Clamp	15psi (1.0Bar)
Second from Left Hand Side:	Locate Left	30psi (2.0Bar)
Left Hand Side Air Gage:	Wedge Engage	30psi (2.0Bar)

OPERATING INSTRUCTIONS

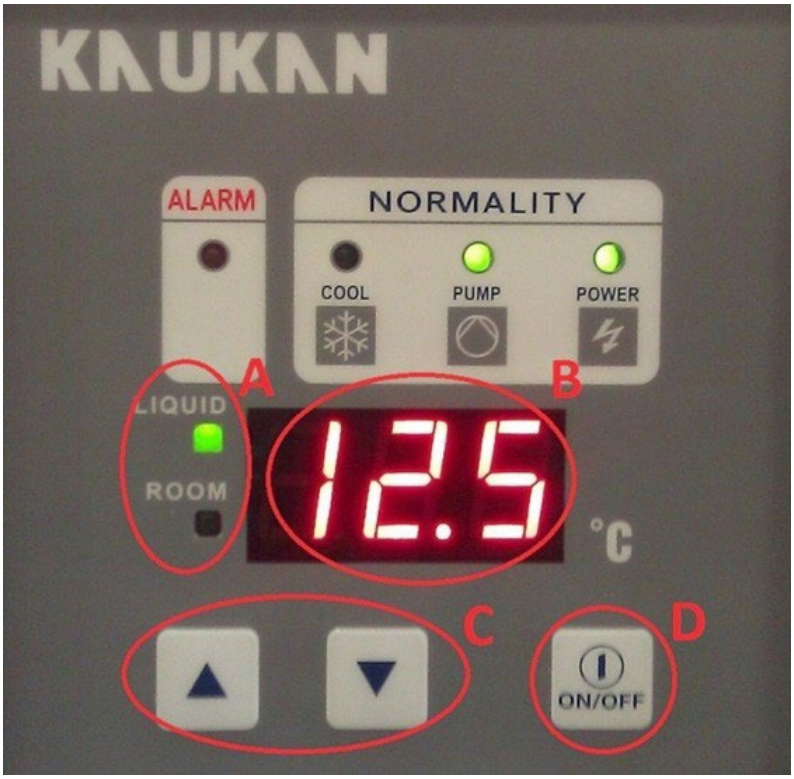
Spindle Chiller System

The spindle chiller system MUST be on whenever the machine is operating. The chiller is turned on separately from the machine. Before preceding you should check to make sure it is operating. The display should read a number representing a degree in Celsius. You can toggle between room and oil temperature by pressing the "ON/OFF" button.

To change the temperature settings of the chiller system, simply hold one of the "UP" or "DOWN" arrows on the control panel for 3 seconds. This will cause the number on the screen to change to a number "+15.0/-15.0." These numbers represent the desired temperature of the oil as compared to room temperature. The recommended settings are "-3.0" for room temperatures 20+ degrees Celsius and "+3.0" for room temperatures 19.5- degrees Celsius.

Chiller Control Panel

The green lights on the left hand side (A) indicate whether the number displayed is room or oil temperature. The number displayed (B) is a Celsius reading of temperature. The “UP” and “DOWN” arrows (C) allow you to change temperature settings in the oil chiller. The “ON/OFF” button (D) will toggle the temperature selection between oil and room temperature.



Spindle Warm Up/Break In

It is imperative that the spindle motor is warmed up before any cutting is done to prevent permanent damage to the internal components. There are two cycles that can be run to accomplish this. The first and most common cycle is the “Quick Warm Up” cycle. This cycle takes approximately 30 minutes to complete. The Quick Warm Up should be done any time the machine has sat in without the spindle running for 8 hours or more. The second cycle is the “Long Break In” cycle. This should be used any time the machine has been transported, the spindle has been left at a high angle overnight, or the spindle has not been operated in 7 days or more. This cycle takes approximately 2.5 hours. Both processes should be completed with the B axis at 0 degrees. There is no need to supervise the machine while it is operating in either of these modes.

⚠ CAUTION

Do not perform the Warm Up/Break In procedure with any tooling attached to the spindle.

MACHINE STARTUP

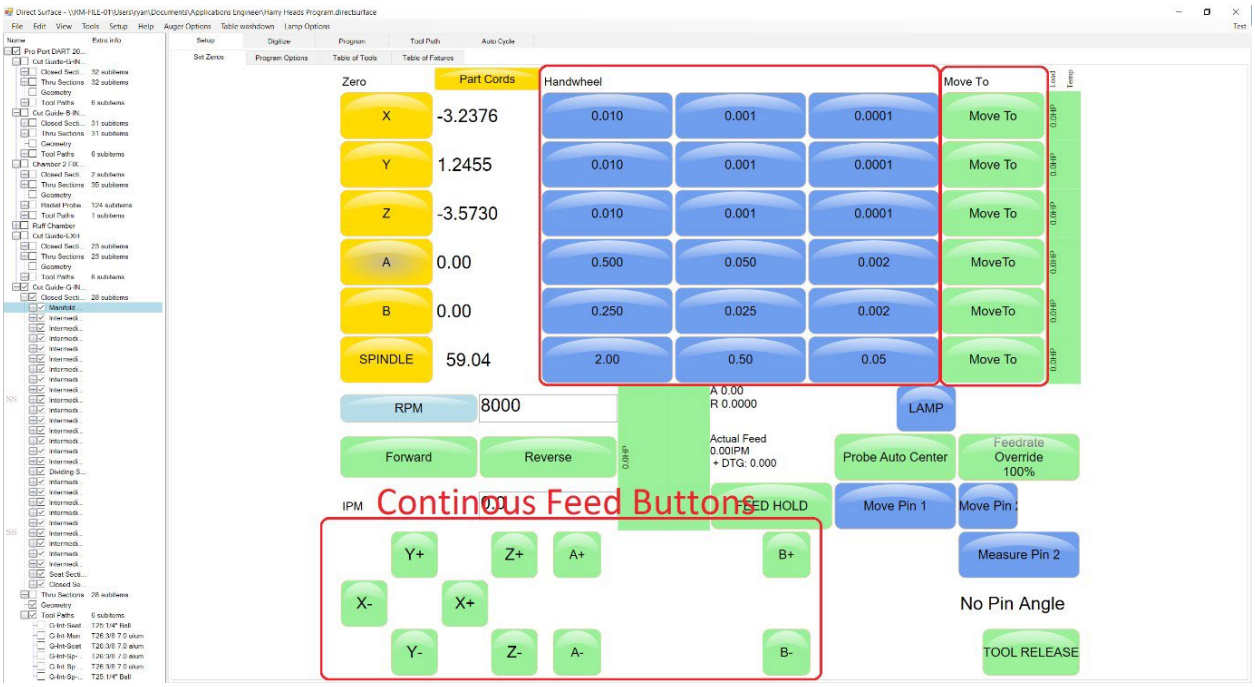
- Start by double clicking the direct surface icon found on your desktop.

- Before operating the check to see that the machine has the correct air pressure on the regulators
- Open the rear air controls enclosure to view regulator pressure.
- Manifold gauges should read; 100psi, 35psi, 12psi.
- Note the middle gauge is the one that regulates the pressure for the B axis. This setting is critical.
- Not having the specific air pressure to the machine will cause the B axis to fall when tilted at an angle and motor loses power, the support cylinder will not have the air to prevent from the motor from falling. If there is too much air in the system it will cause the head to rise up when the head is tilted and the motor loses power.
- If pressure is not to spec, follow the B-Axis air pressure adjustment procedure located in the maintenance section of this manual.
-

Home Machine

Once you are viewing the machine control interface Home the machine. Homing the machine makes the control reference the current machine position in order to complete future calculations for position movements. The machine will not run any auto cycles until this step is complete.

Once the machine has been homed the machine can be jogged around by using the axis control interface with either the feed buttons, move-to commands, or the handwheel increment selection buttons.



Control Interface

From the main screen we can see there are various tabs at the top of the control interface labeled **Setup, Digitize, Program, Tool path**, and **Auto Cycle**. Each of these tabs will be used in order to guide you through the programming process. You will start at the **Setup** tab and end with the **Auto Cycle** tab.

Setup Tab – Contains the Set Zeros, Program Options, Table of Tools, and Table of Fixtures Sub-tabs. This is where the operator can jog the machine, set program zeros, and record tool and fixture information

Digitize Tab – Contains a small window for machine control. This is where the operator can position the machine with the probe equipped and perform digitization of chambers and ports.

Program Tab – Contains the digitized information. This is where the operator can edit the digitized cross sections to manipulate port geometry.

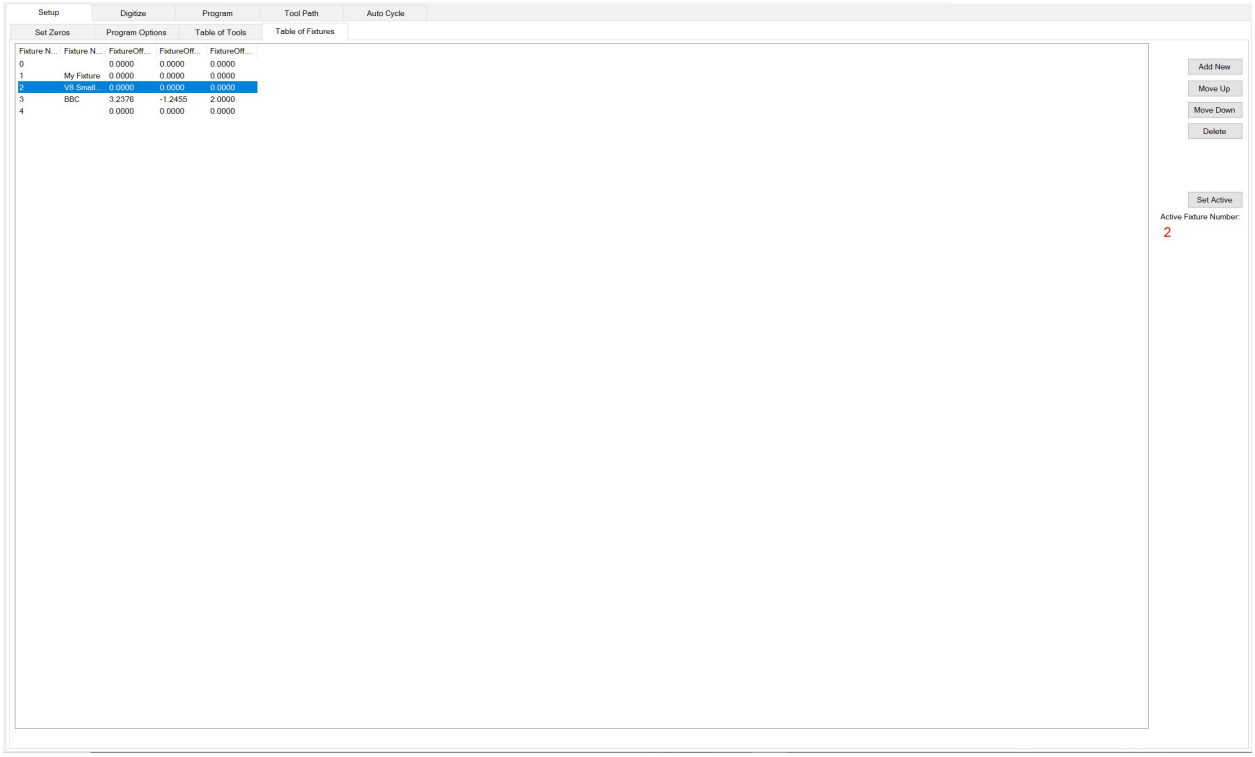
Tool Path Tab – Contains the automatically generated tool path that will be ran based on the port geometry from the program tab.

Auto Cycle Tab – This is where the operator selects which ports/chamber operations they wish to run and contains the cycle start button.

Each head will have its own specific offset or location within the machine work envelope. Notice the **"Table of Fixtures"** below. The table of fixtures stores the prescribed fixture name as well as it's fixture zero locations.

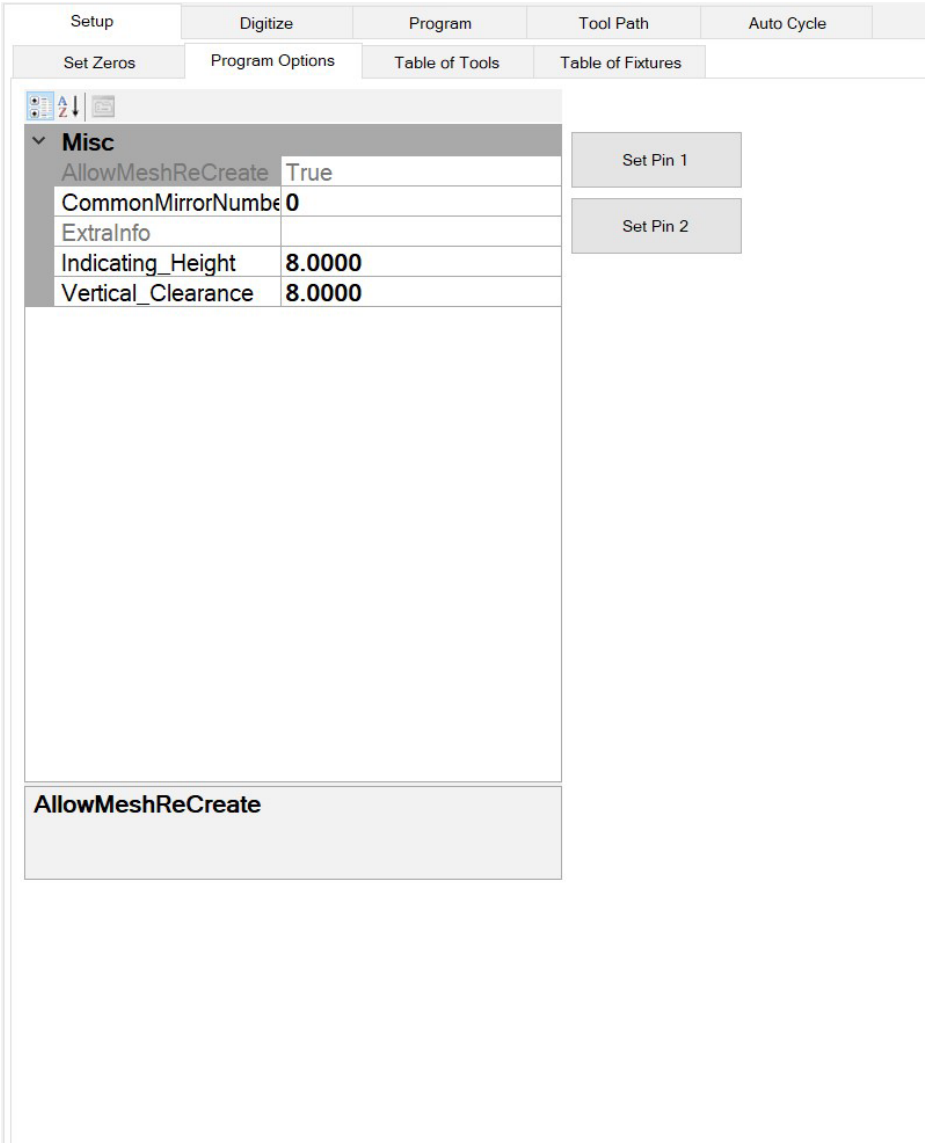
To choose the fixture to be used select the fixture from within the table of fixtures and it will be highlighted, then click on the **'Set Active'** button on the right side of the screen.

To add a new fixture simply click the 'Add New" button, then you can double click on the fixture number to change the name for the fixture. To set the zeros for a new fixture you can jog the machine and use the touch trigger probe to find the location you would like to use for the fixture offset. By double clicking on the axis buttons from within the CNC interface screen the machine will save the current location as the zero for that axis. The numbers next to the axis button are the part coordinates as reference from the current fixtures zero locations.



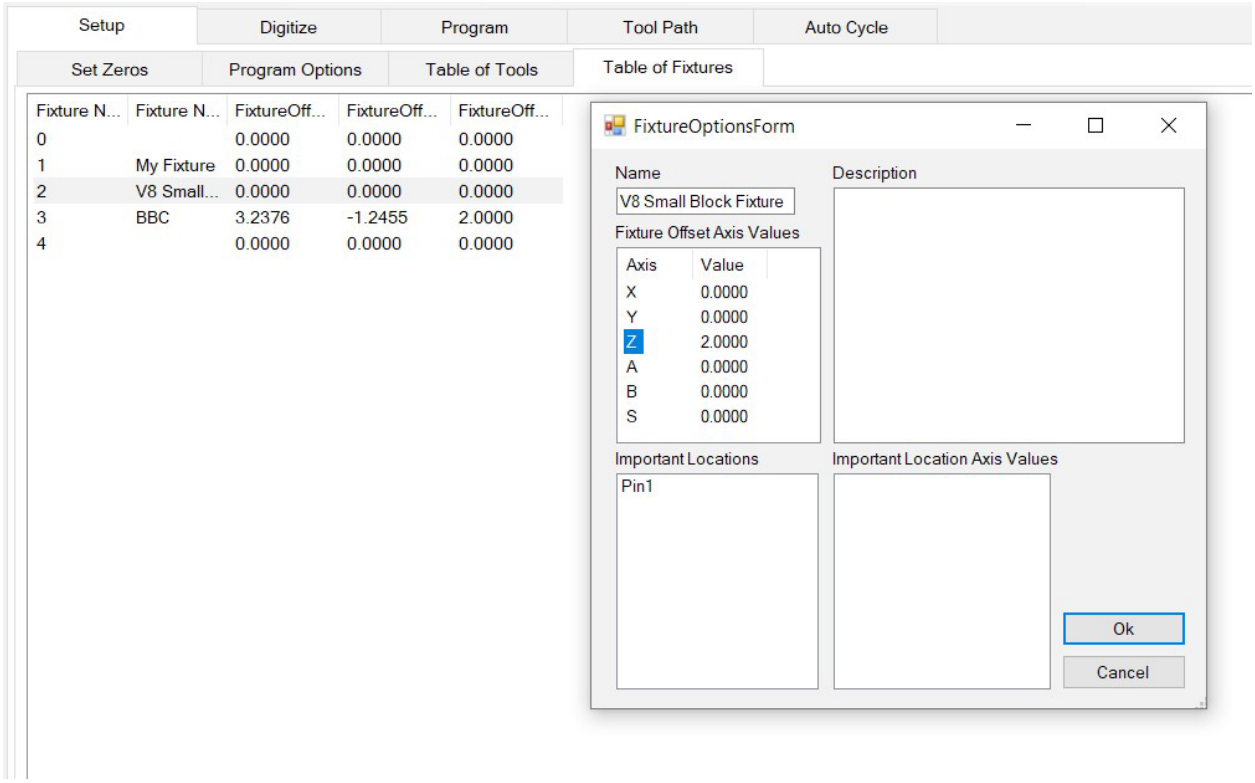
Locating Fixture

To locate a cylinder head fixture, the pins that were used for alignment earlier need to be located individually and set as a location within the program. Starting with the left pin, use a dial indicator or the touch trigger probe to locate the center of the pin. Within the **program options** sub tab, locate the Set **Pin 1** button then press the button. A window will come up asking if you are sure you want to set pin location and verifying that the right fixture number is selected. Repeat the process to locate pin 2 and once the center of the pin is located press the **Set Pin 2** button.



Pin 1 which is the left pin will be used as the fixture offset zero location for the current fixture that is being setup.

After both pins are located, the Z axis needs to be changed to 2.0000” since it’s the distance from what the fixture is rotating on the A axis. Go to the “Table of Fixtures” tab and click on the fixture number that is selected, there will be a window that come up indicating the position for the X, Y, and Z axis, double click on the Z axis and change it to 2.0000” when prompted to set the Z axis fixture offset.

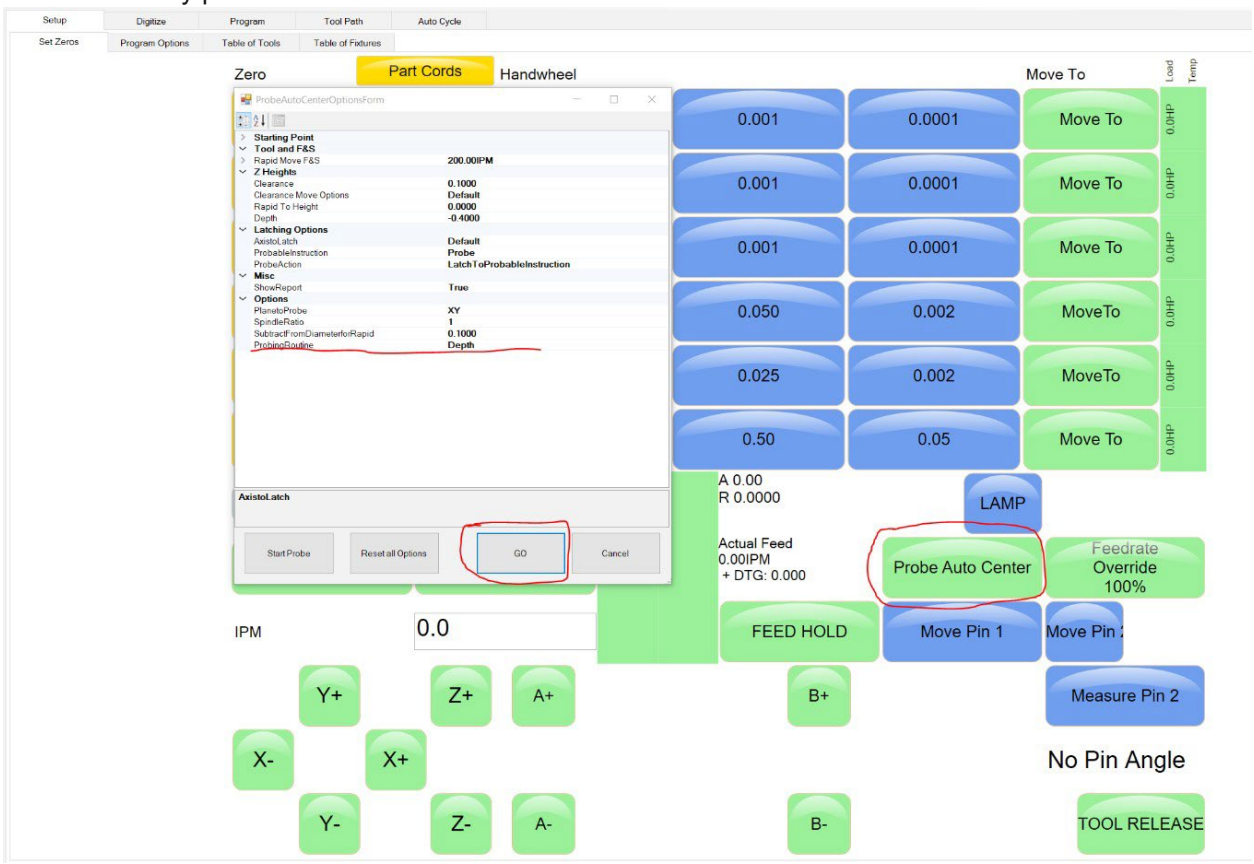

Setup
Digitize
Program
Tool Path
Auto Cycle
Set Zeros
Program Options
Table of Tools
Table of Fixtures
Fixture N...
Fixture N...
FixtureOff...
FixtureOff...
FixtureOff...
0 0.0000 0.0000 0.0000
1 My Fixture 0.0000 0.0000 0.0000
2 V8 Small... 0.0000 0.0000 0.0000
3 BBC 3.2376 -1.2455 2.0000
4 0.0000 0.0000 0.0000
FixtureOptionsForm
Name
V8 Small Block Fixture
Description
Fixture Offset Axis Values
Axis
Value
X 0.0000
Y 0.0000
Z 2.0000
A 0.0000
B 0.0000
S 0.0000
Important Locations
Pin1
Important Location Axis Values
Ok
Cancel

Defining Tool Length and Probe Length Offsets

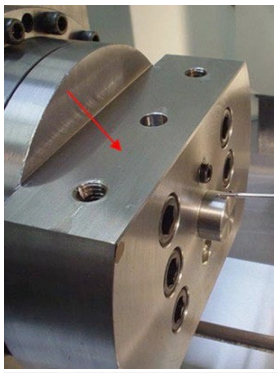
All tools will be touched off on the flat side of the 4th axis head stock, opposite to where the plate gets mounted. Note that the flats on the 4th axis have two different dimensions from the center of rotation. One flat is 2” from center the other flat is 2.5” from center. The 2” dimension will be positioned at A0.000. The 2.5” dimension will be positioned at A180.00. All tools and probes will be established from the 2.5” side.

Probe Length

To define probe height, position probe just over the flat of the 4th axis. Position the probe so its approximately 1 inch above the surface to be probed. From the Set Zeros sub tab press the Probe Auto Center button to bring up the probe options window. Make sure that the probing routine is set to depth and when ready press GO.



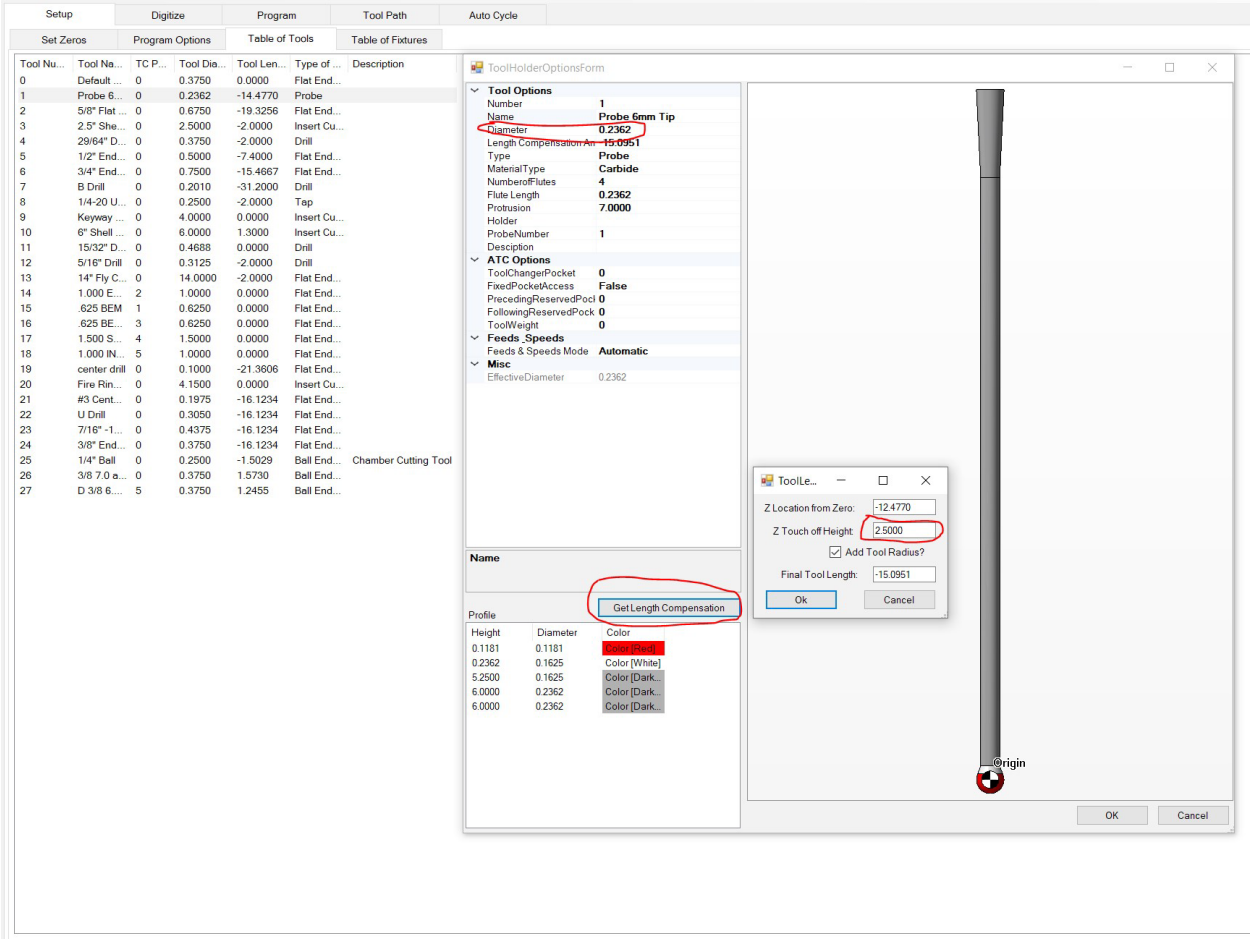
The probe will automatically move down until it touches the surface, move back up to a clearance plane, and then move back down until the probe tip is just touching the surface of the head stock.



Touch all cutters and probe off on the head of 4th axis at position 180 degrees as shown by the red arrow above. The dimension from the center of rotation to the flat positioned at 180 degrees is 2.5000”

Table of Tools

Open the table of tools and double click on the probe to bring up the Tool Holder Options Form. From this window you can edit the information that defines the probe. The two critical boxes that must be filled out are the Diameter and Length Compensation values. If using the standard 6mm probe then the diameter is .2362. This should be set from the factory. Pressing the get length compensation button will bring up the Tool Length Compensation Window.

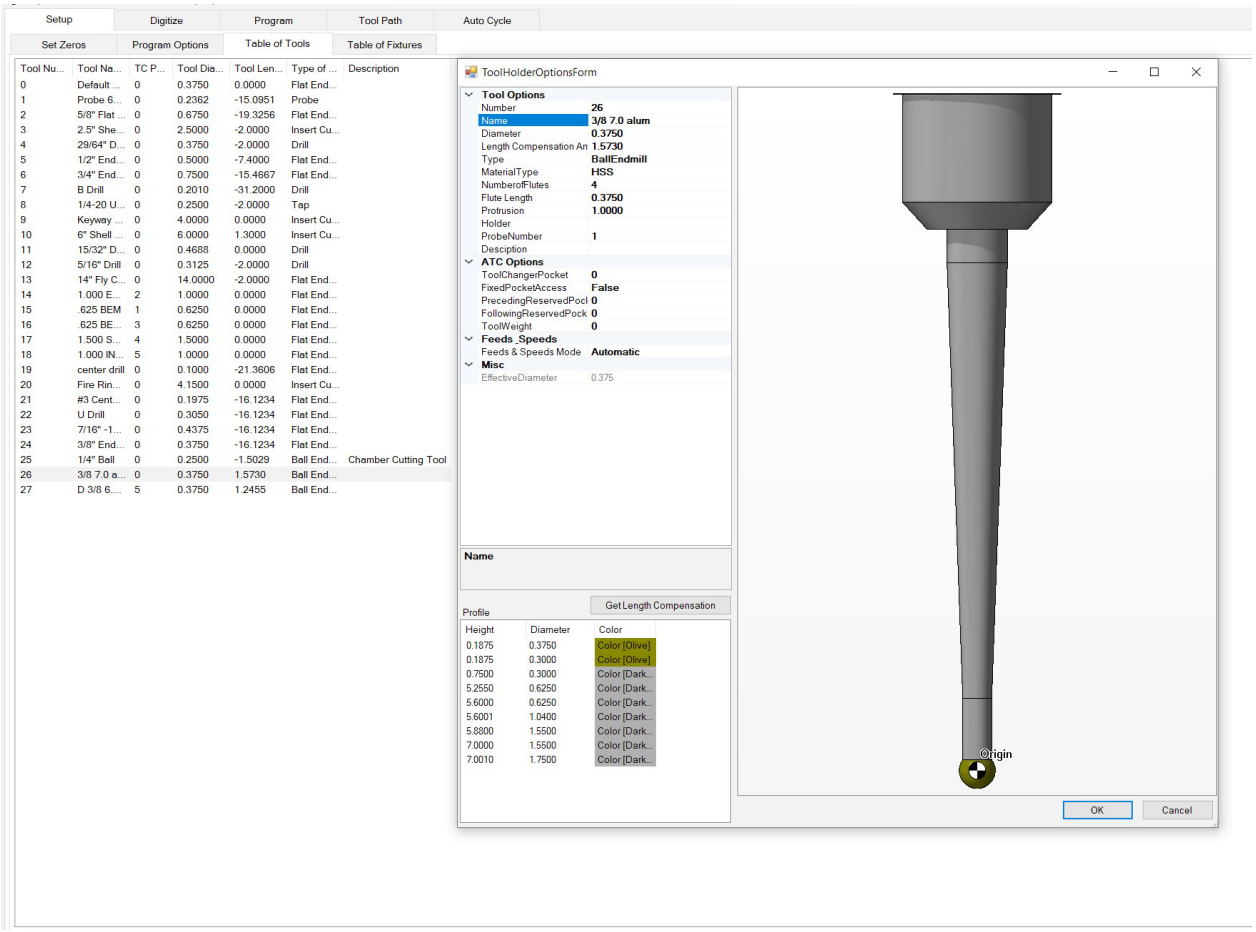


Note there is a value of 2.500” in the ‘Z Touch off Height’ field. Since we are using a probe and touching directly on the 4th axis flat to establish the height the value will be set at 2.500” to adjust the height offset number from the flat to the center of rotations(all probe and tool Z zeroes are established at the center of the 4th axis rotation).

Cutting tools are set up slightly different than the probe since we can’t use the probe auto center command. A .100” gage block is supplied for touching off hard tools such as the porting tools and any other carbide endmills the operator may wish to use.

To setup a tool we still need to touch off the top of the 4th axis spot on the 2.5” offset face as we did with the probe. The difference is we now have to bring the tool down manually until we can just slide the .100” gage block between the tool tip and the 4th axis surface.

With the tool loaded into the spindle jog the machine down until it is close to the touch off surface. Using the .001” increment on the z-axis handwheel setting, continue to move the cutter down as you slide the .100” block back and forth underneath the tool. Continue to do this until the cutter just touches the gage block and you can no longer slide the block back and forth. Once the tool is touching the gage block switch to the .0001” handwheel increment setting and jog the spindle up in the z axis just a few clicks. Move the tool back down in the z axis as you continue to slide block back and forth underneath the cutter. Continue doing this click by click until you feel the cutter just drag across the surface of the block. At this point open the Table of Tools, Select the current tool or create a new tool if necessary. Open the get length compensation tab for the tool. Press the Get length compensation, enter 2.5” for the Z touch off height and press OK.

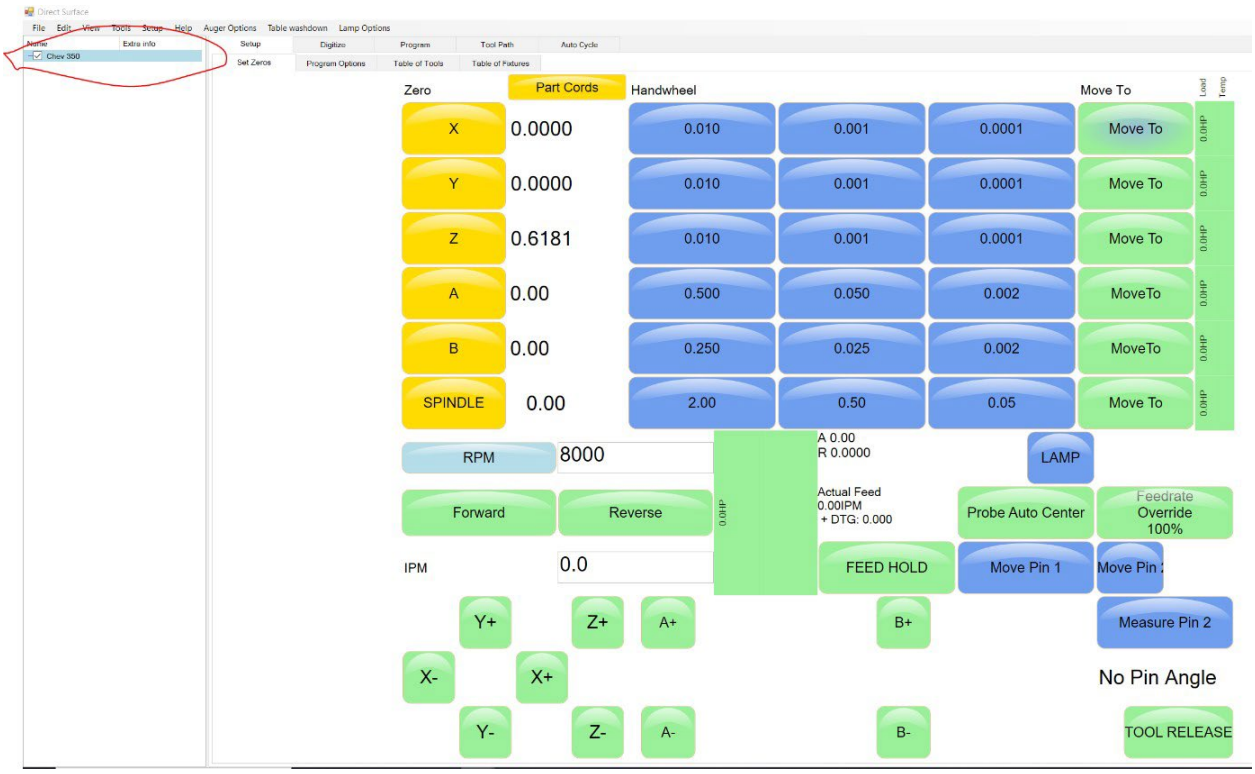


If you are setting up tools with ball ends such as the probe or porting tools then make sure that the Add Tool Radius box is clicked. This allows the machine to perform the correct position moves while cutting with multiple axes while making sure the tool doesn’t overshoot the desired dimensions.

DIGITIZING

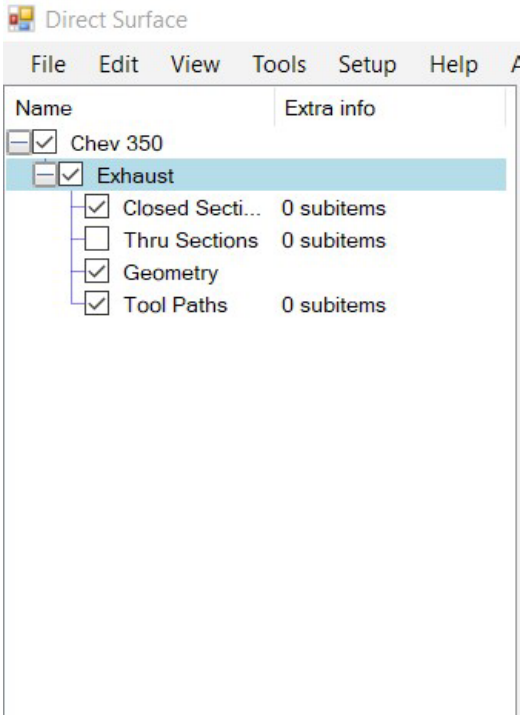
Starting a New Head

When you open a new file a new head program is automatically created. You can right click on the new head designation on the left side of the screen and press rename to name the head.



Creating a Port

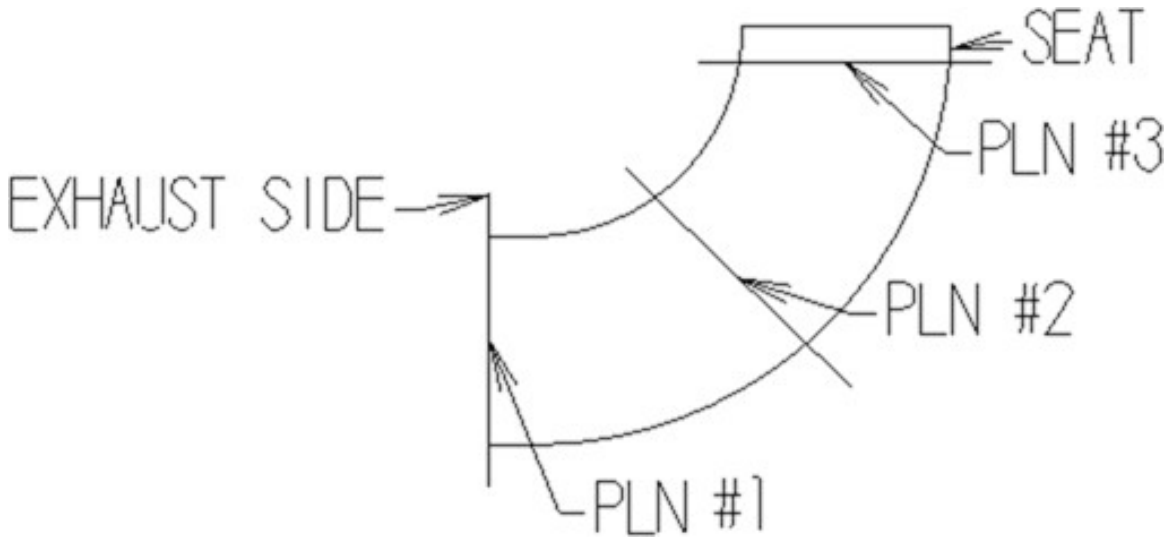
To add a port right-click on your newly named head and then click on ‘Add A New Port’. Click the (+) icon to expand the port information tree. You may also right click on the new port name to rename the port.



Digitizing a Port

- There are three planes that need to be defined by using the hand-wheel in order to create a road map for the automated digitizing process. It takes three points to define a plane. We will be using X,Y,Z, A, and B axes in order to define each of the three planes. We will start off by focusing on the exhaust port. The intake port(s) will be done in the exact same fashion.

- With regards to the exhaust port the three planes that we need to define occur at the exhaust manifold side, the middle of the port (where the major transition takes place), and lastly the valve seat area.
- The planes that we define are simply approximate positions. We are simply defining where we need the probe to move so that it can begin its automatic digitizing process. See the diagram below.
- Plane #1 will be digitized from the Manifold side. With the A axis oriented so that the manifold side is facing upward.
- Plane #2 can be digitized from either seat or exhaust side, whichever is easiest. The clearance available for the probe to reach into the port before shanking out is usually what determines the transition plane.
- Plane #3 will be digitized from the seat side with the A axis oriented so that the seat side is facing upward.



Creating Planes

Set the probe to active tool from within the table of tools

Navigate back to the Set Zeros tab and then right click on the Closed Sections in the Cylinder head information drop down list.

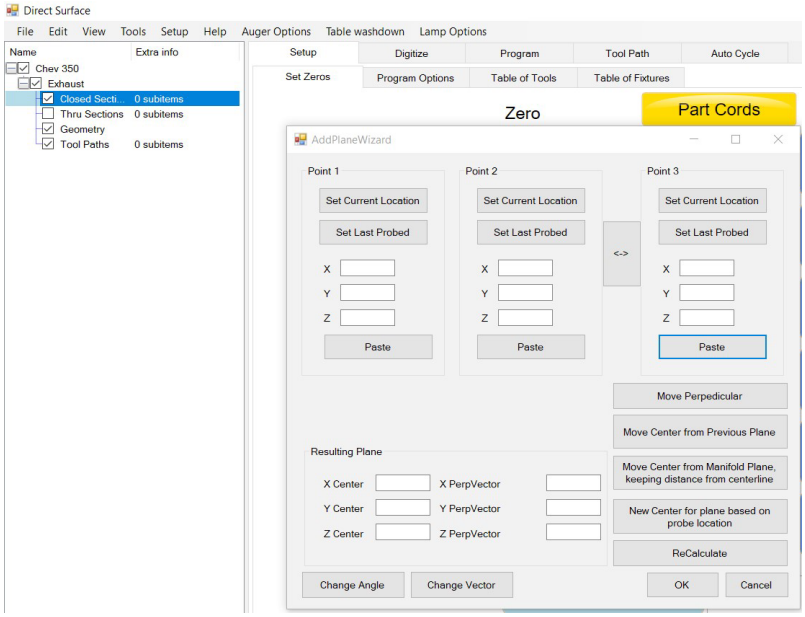
PLANE #1

First rotate the A-axis to get the manifold side flat and facing upward. To define plane #1 we will use 3 points that are spaced at approximately 120 degrees apart. You will pick these points up using the handwheel. The hand wheel can be move in X, Y, or Z to find the three points.

POINT #1 POINT #2 POINT #3

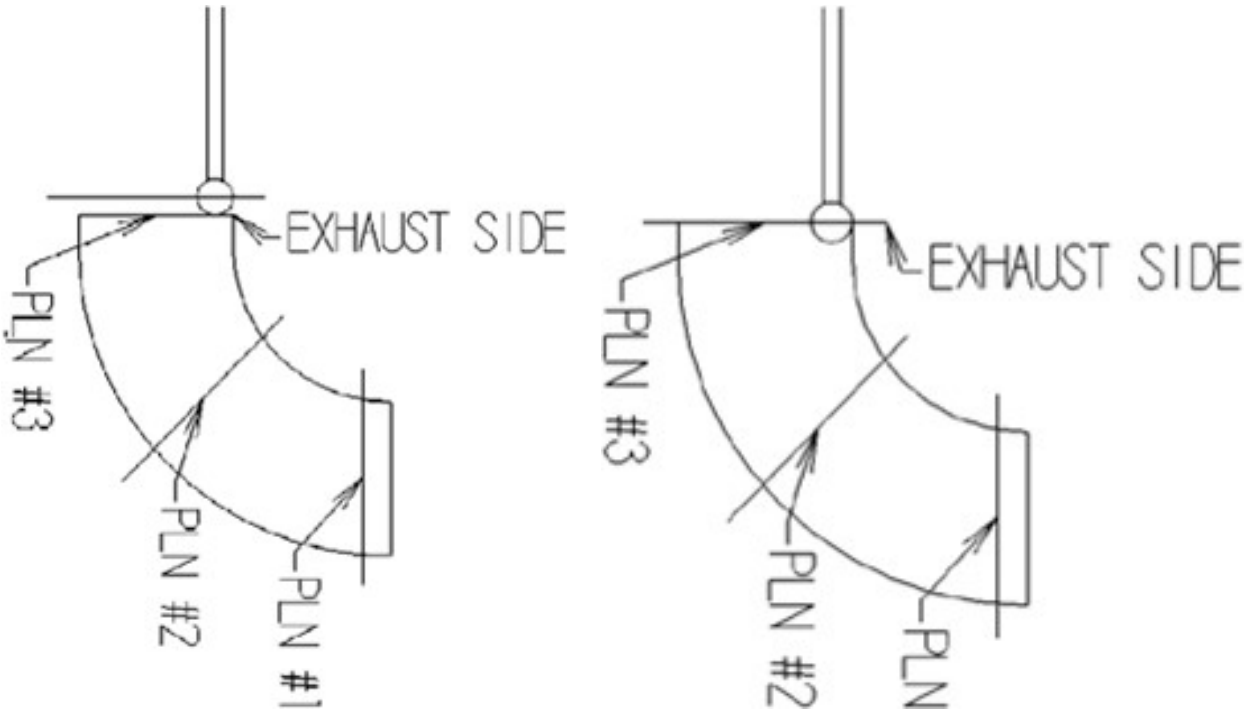
Right click on “Closed Sections” then click on **“Add New Plane for Probing”**. An ‘AddPlane Wizard’ will appear. Position your probe near where point #1 is from the above photos by using the hand-wheel. Using the .001” handwheel increments in the z-axis bring the probe down slowly until the lights on the probe change from green to red. Once the light turns red move the probe back up one handwheel click at a time until the light becomes green again.

In the Add Plane Wizard where it says Point 1 click the Set Last Probed button. This will collect the last probed X,Y,Z axis location and store the coordinates in the respective boxes. Repeat the above process at points 2 and 3 and use the set last probed button to fill out the X,Y, and Z boxes for both points within the Add Plane Wizard.

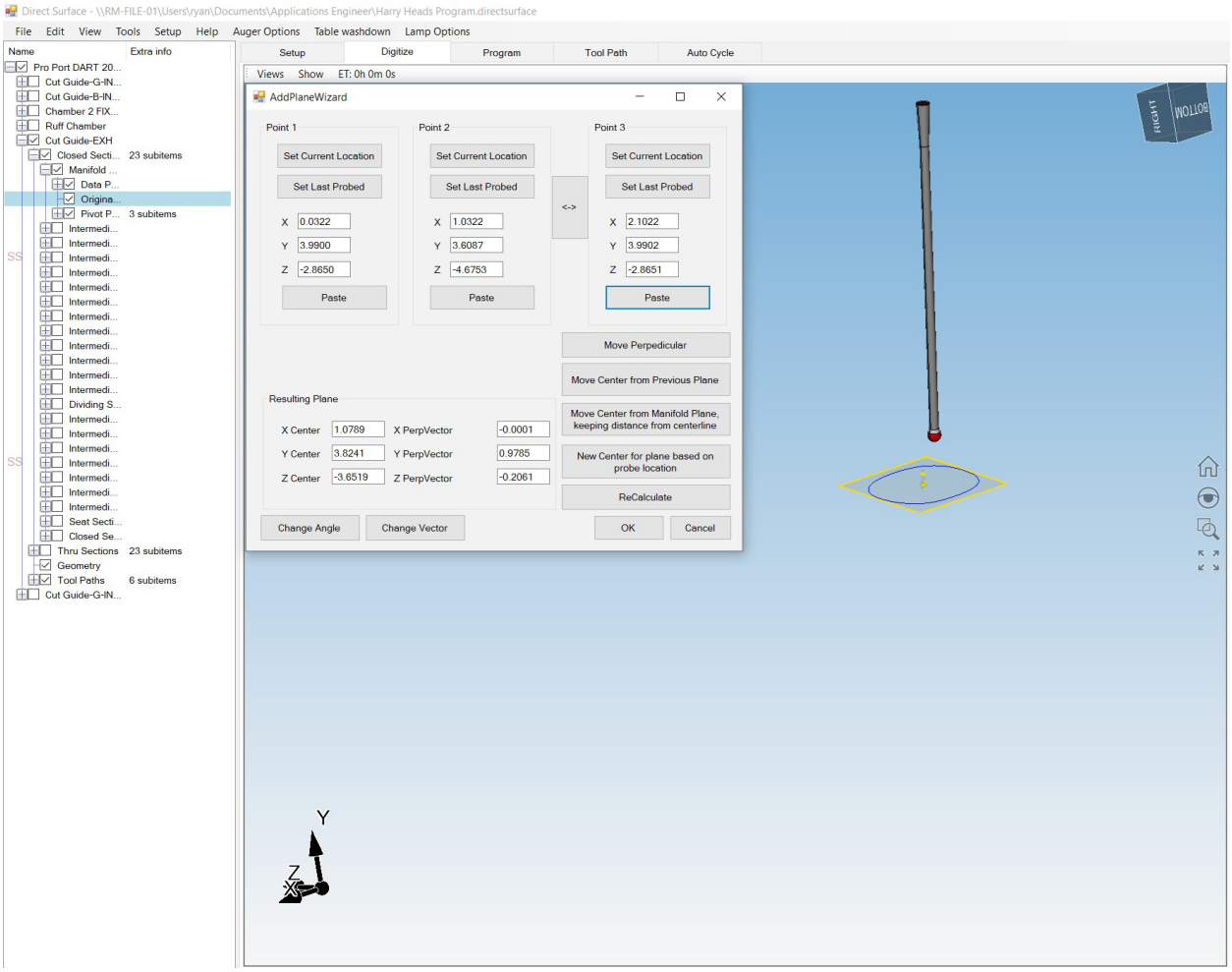


Once this plane has been created using the above method, we want to relocate the plane so that it’s at the very edge of the port. This is accomplished by moving the plane down by the radius of the probe tip which if using the 6mm probe is .118”

Original Plane Shifted Plane



To manipulate the plane, navigate to the digitize tab. Highlight the closed sections and in the viewing window click show and the planes to see the planes. Double click on the Original Plane in manifold plane tree to re-open the Add Plane Wizard.



Click on the Move Perpendicular button to open the Input Form. In the entry box type in -.118 and press OK to shift the plane down by the probe tip radius.

PLANE #2

To define plane #2 we will be using the side of the probe to pick up the three different points that will be making up the plane, instead of using the tip of the probe as used on the previous plane. So there will be no need to move the plane after it has been set. Right click on the closed sections option and select Add New Plane For Probing.

Hand wheel X, Y, Z, A, and B around until you can position the probe to the approximate points shown below. Points #1 and #2 can be picked up just to the left and to the right of the valve guide, being straight across from each other, as square as possible to the valve guide. (Make sure that only the ruby is touching the port and not the shank of the probe).

POINT #1 POINT #2 POINT #3 POINT #3 SIDE VIEW

Point #3 is the short turn radius point and can be the most difficult point of the 3 to define. It will need to be directly across from the 1st and 2nd point. We must be able to reach this point from both the manifold and seat sides so take some time and care when positioning this point.

The Figures below are shown as an example of how the probe must be able to reach Point #3 from both the manifold and seat side of the head. The pictures below show this plane being picked up from the seat side. As mentioned above you may choose which side you would like to first define this point. If the A-axis is not rotated enough you will not be able to get at this point from the other side. There has to be a "line of sight" to Point #3 from both directions.

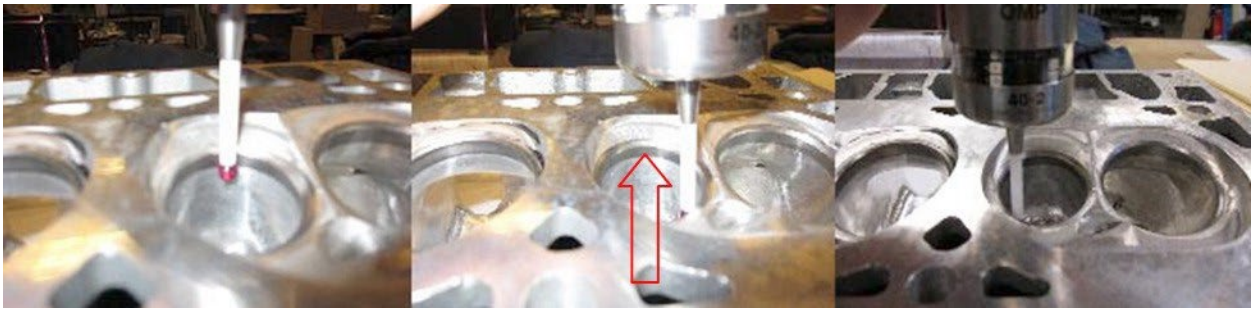
Sometimes it can take a couple of minutes to get your probe positioned where you like in which case the probe may shut off automatically. If this happens simply go into the table of tools and click on the 'Start Probe' button, then click on the 'Set Zeros' tab to get back to the control.

PLANE #3

Manually rotate the A-axis so that the seat side of the head is facing up. If you know the angle of the head then you can use the move-to command and enter the angle to have the deck of the head aligned with the horizontal plane.

Start by clicking on "Add New Plane for Probing" button, then use the "Add A Plane Wizard" to set plane #3. The three points to probe the plane will be as shown on the picture below, right after the valve seat insert finishes (see the red arrow in point 2 for clarity)

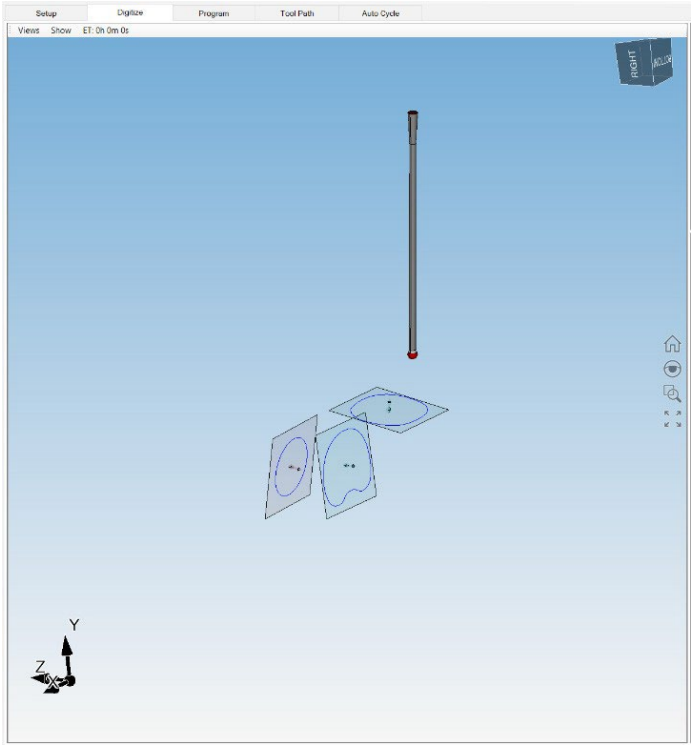
We will be using the side of the probe, and there will be no need to move the plane perpendicular when it's been set. Line up the center of the probe right between the end of the seat insert and the port. Set all three points as shown on the picture then click "OK" on the window to add the plane.



POINT #1 POINT #2 POINT #3

Use the methods learned for probing plane #2 to create plane #3 in the Add Plane Wizard. You may view the plane in the digitize tab to see that it's been created.

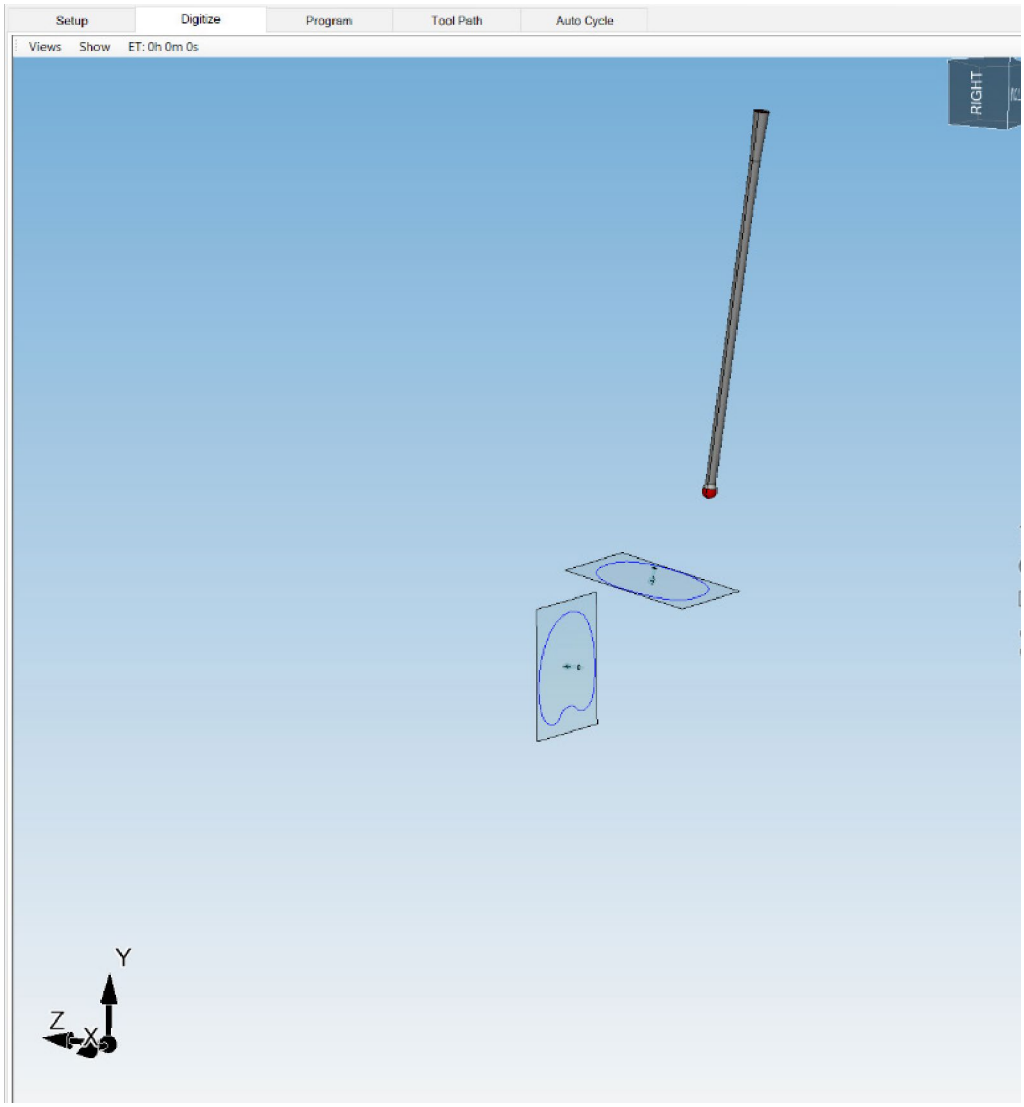
After Plane 3 has been created you should be able to view all three planes in the digitize tab by clicking the show planes option. The figure below shows an example of the three planes that were created for an exhaust port.



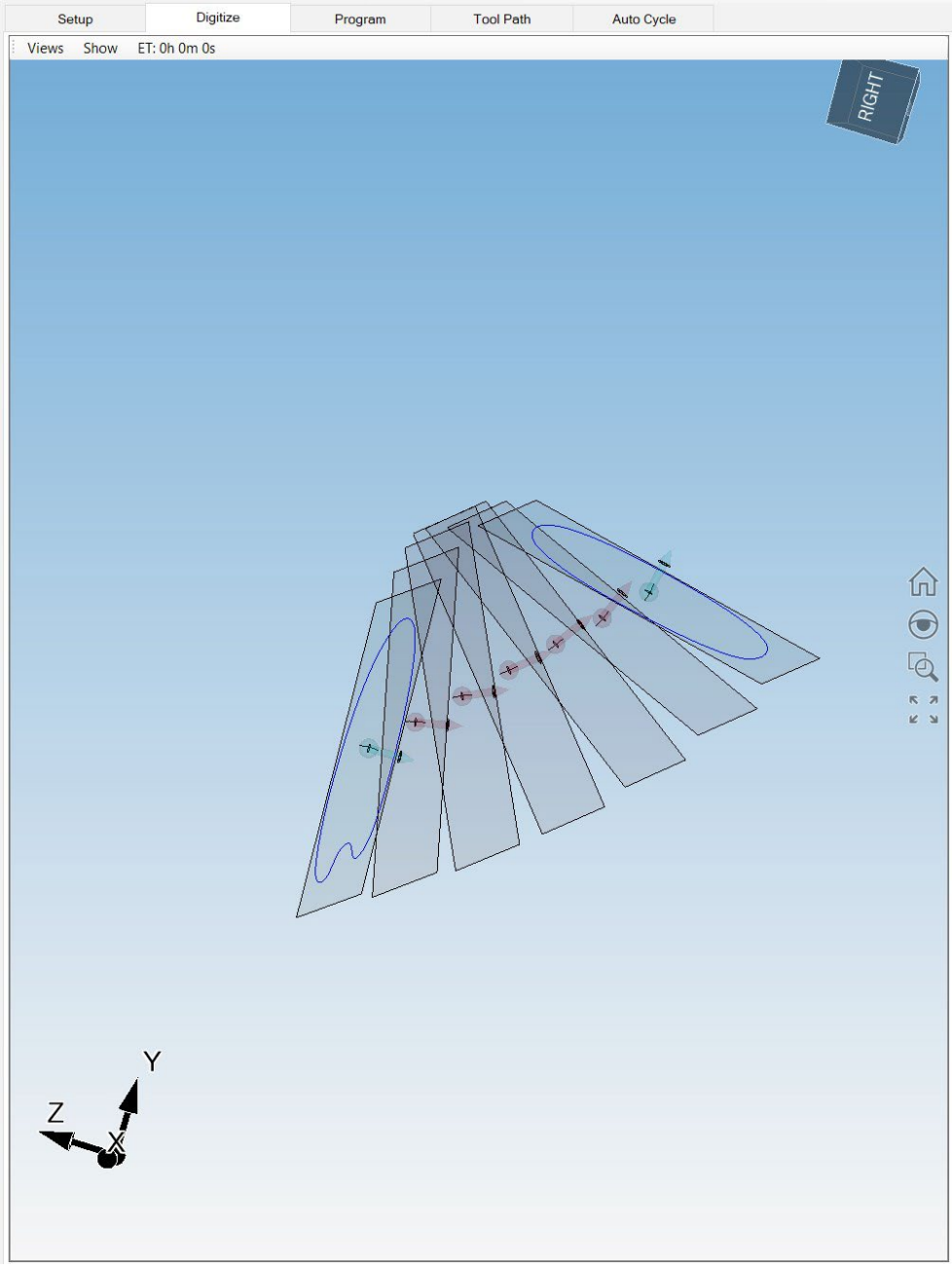
Creating Intersecting Planes

Now that we have all three planes defined for the port, it is now time to begin our automatic digitizing process. With the ‘Digitize’ tab still open make sure that all three of the planes have a check mark in each of the boxes. You should now be able to see the three planes that were created.

To create the intermediate planes, we need to isolate 2 of the 3 planes by unchecking one of the planes created. Uncheck Plane #3 in the cylinder head information tree on the left side of the screen so only the manifold plane and dividing line plane are shown in the graphics window.



Right click on the closed sections tab in the information tree and select create intermediate planes for probing. The system will then ask you to input the number of planes you wish to divide space between the manifold plane and dividing line plane into. This value will determine how many cross sections of the port the probe will digitize. Increasing the number of intermediate planes increases the resolution or accuracy of the digitized data. However, this has diminishing returns and the more intermediate planes that are used will increase the time it takes to digitize a port while only improving accuracy slightly. In the example below we are digitizing the exhaust port for a big block chevy head. We have chosen to use 5 additional planes between the manifold plane and dividing line plane. A typical rule of thumb is to choose a sufficient number of planes that will result in approximately a .500” step-down between the plane center lines.

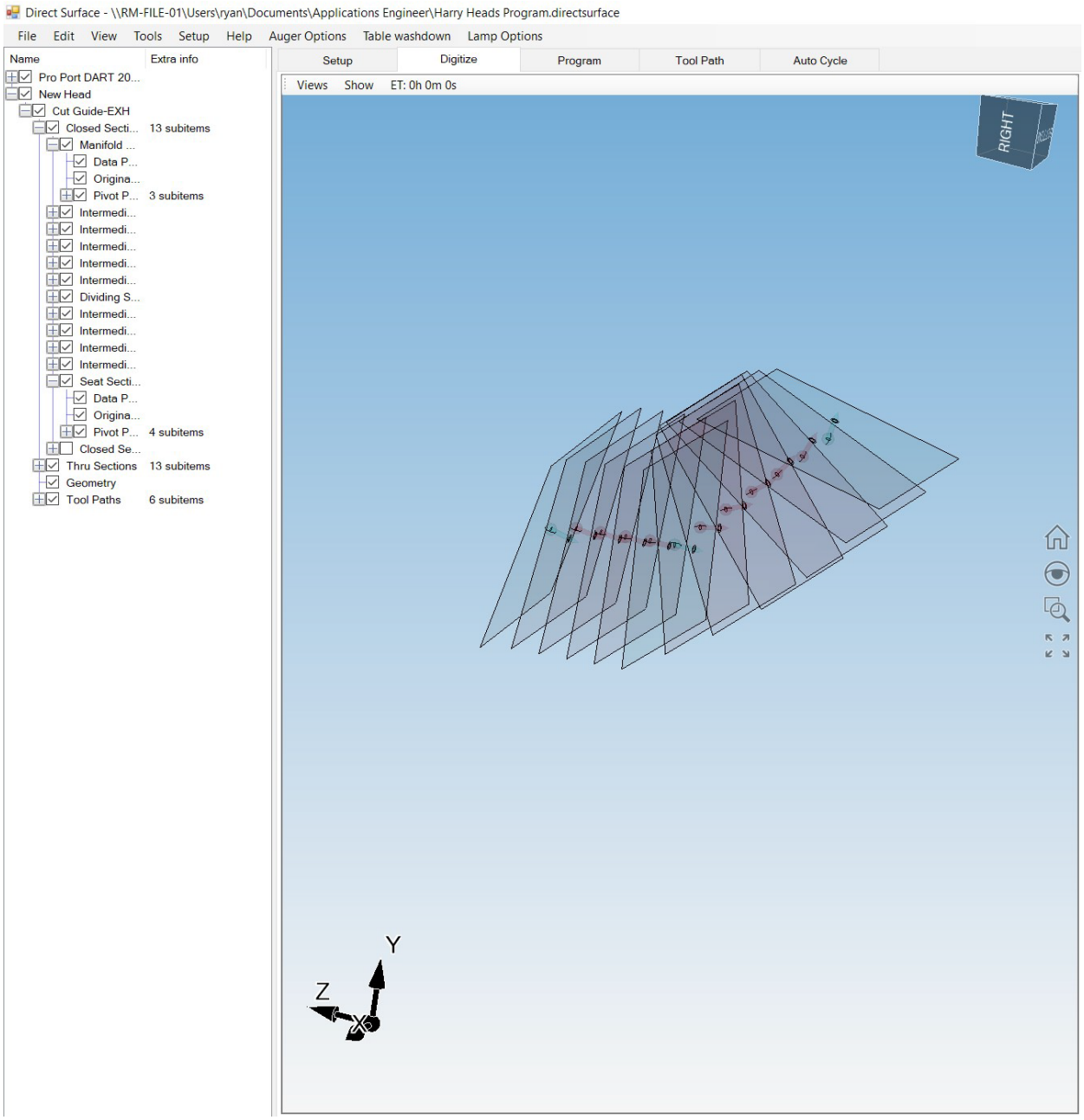


The figure above shows the manifold and divided plane being divided up by an additional 5 planes. Also note that there are 5 new closed sections listed. Each one of the closed sections represents a plane. You can turn these intermediate planes on and off by clocking the check mark box next to the closed section you wish to turn on or off.

Now we need to create planes for the divided line and the seat side planes. Repeat the process above by turning off all the closed sections except for the seat plane and dividing plane. Once complete right-click on the closed section line and select add intermediate planes. Once again for our example we are using 5 planes between the seat plane and dividing plane.

After creating intermediate planes between dividing plane and seat plane you will see that there are five new ‘Closed Sections’ in between the divided line plane and the seat side plane. Check all the

closed sections and select show planes in the graphics window to view all the planes for probing. When complete the graphics window should look like the figure below.

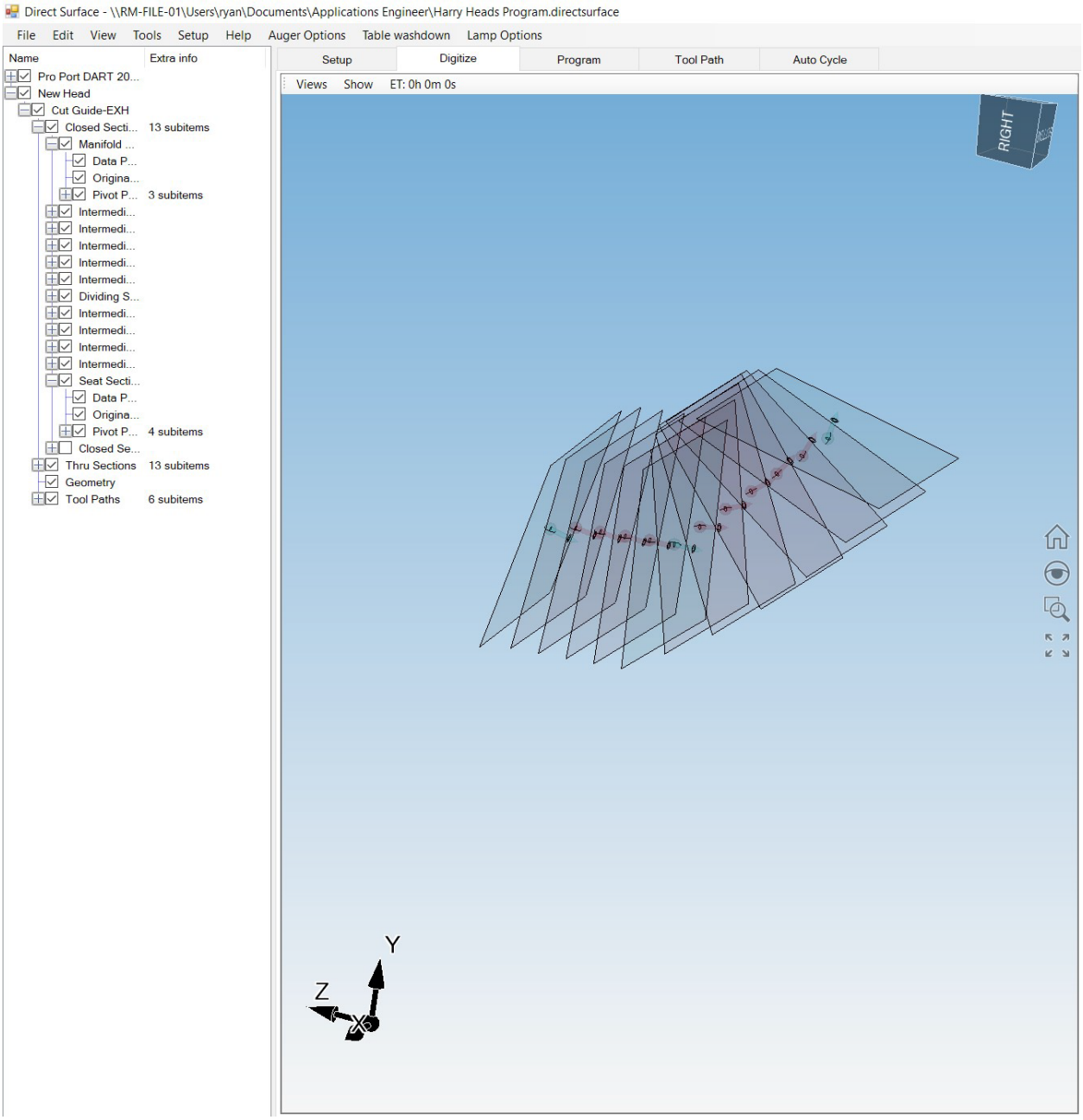


When we begin to digitize the probe will find the center of each of these planes. Then the probe will work its way to the wall of the port until it detects contact. Once contact has been made the probe will start finding points around the port within the plane in .065” increments. Once the probe has revolved around the port wall it will move back to center before proceeding down to the next plane and repeating the process. As the probe finds points it will populate them within the plane and draw a closed cross section representing the wall of the port.

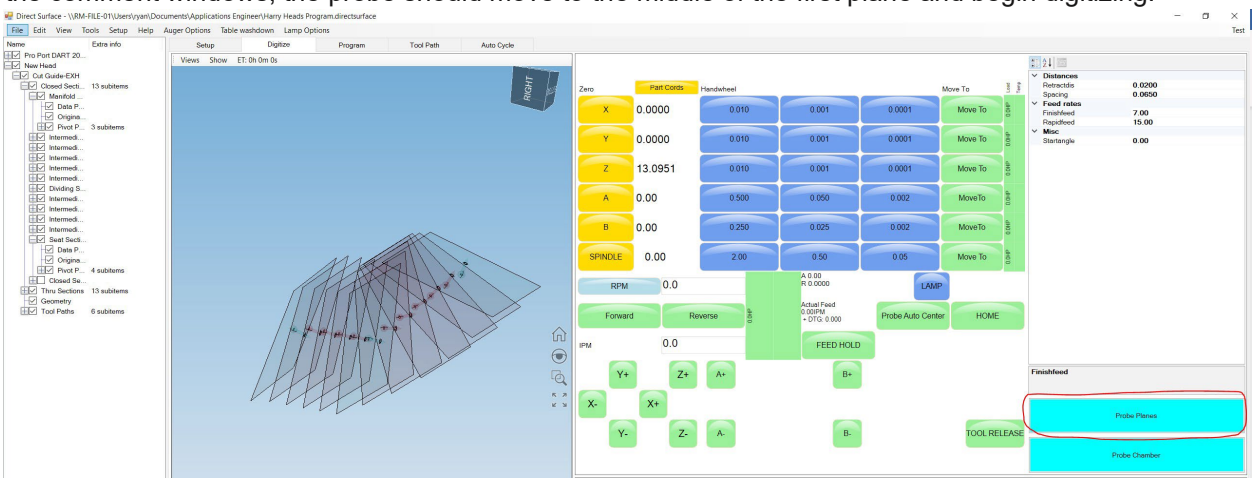
Probing Defined Planes

When you begin to digitize you will isolate each half of the port by checking the closed sections you wish to probe first. For example, you can start off by checking the ‘manifold side’ through the ‘divided side’ sections first, as shown below. This will digitize at the prescribed plane locations from the manifold plane through to the dividing line plane. Note that if you probe the dividing line plane from this side then it does not need to be probed again when you digitize the remaining planes from the seat side.

Notice that the only planes visible on the screen are the “manifold side”, “divided line”, and “closed section” intermediate planes. The planes that represent the other half of the port have been shut off. The planes that we have defined below are simply a road map for the probe to move to begin its digitizing process. The probe will find it way to the middle of each plane then at that level will start moving out to find its way to the edge of the port.



Click on **'Probe Planes'** from the CNC interface on the right of the screen to begin the digitizing process. Initially you may have to click **'Probe Planes'** twice as the first time turns probe on. Click "OK" through the comment windows, the probe should move to the middle of the first plane and begin digitizing.

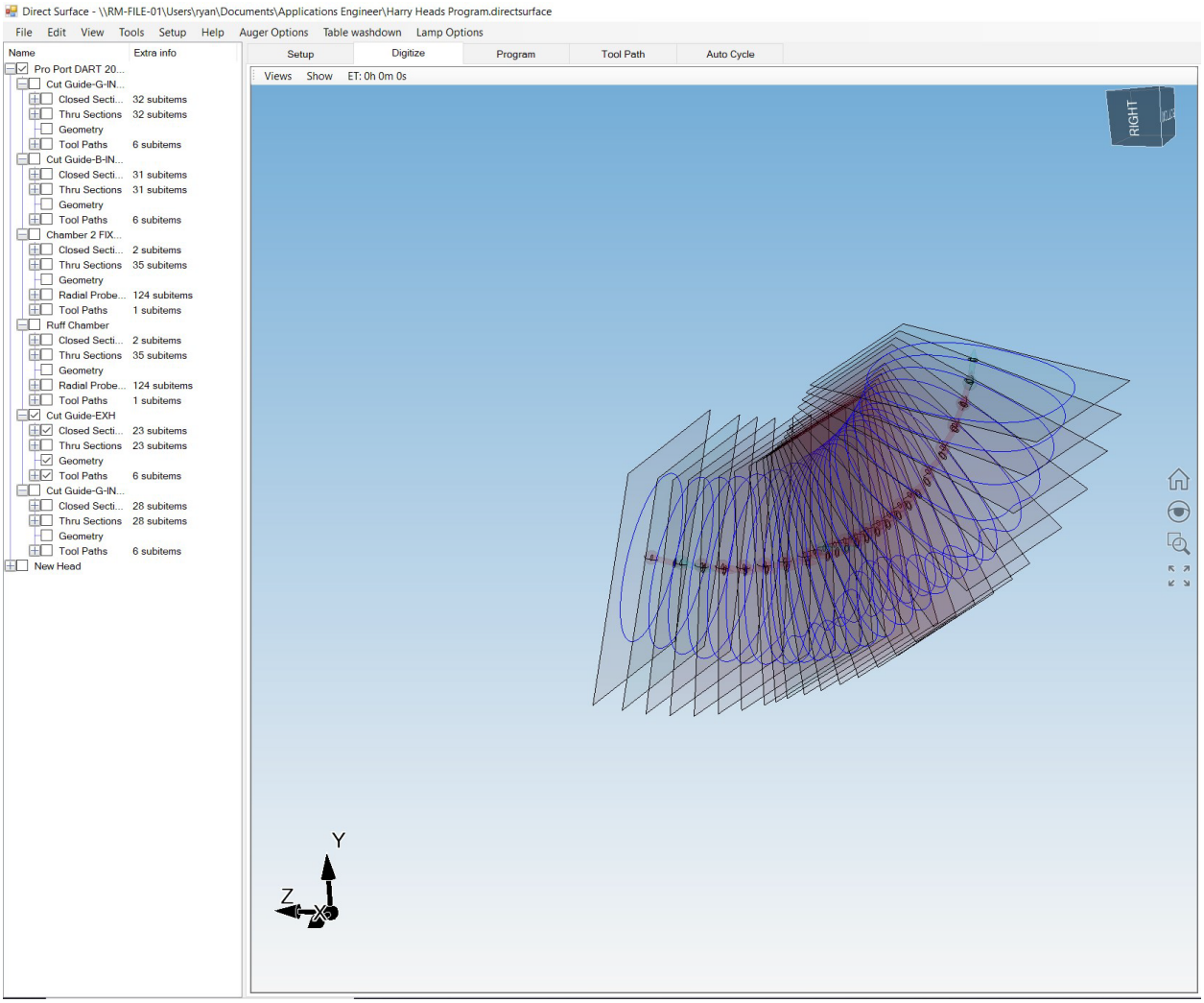


The software will tell you how much the 4th and 5th axis will rotate before it starts to move to the first Plane. This value is in absolute measurement scale from the origin location.

Once the machine finishes digitizing a single plane the software will again notify the operator of the next intended move to the proceeding plane. Check to make sure that the rotation seems reasonable and will not cause any collisions. Click OK to move to the next plane and to continue probing the port.

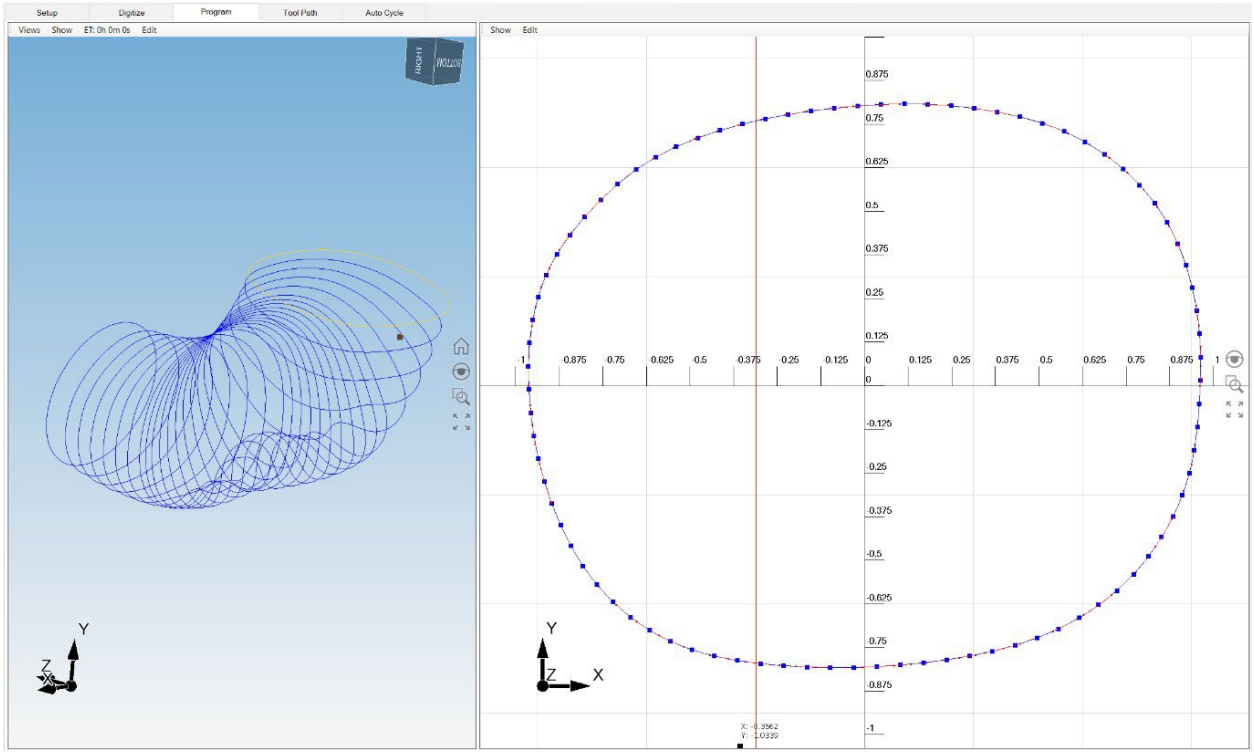
Repeat this process until you have made it through all the planes that you have checked. Once the machine has finished digitizing this section of the port, Rotate the head to the other side to be digitized and repeat the process above for the remaining planes.

After probing all the planes, you should now see all the cross sections at each plane have been defined. Check all the closed sections in the cylinder head information tree to view the probed sections in the graphics window. If done correctly then your graphics window should now look similar to the figure shown below.



Cross-Section Manipulation

Switching to the Program Tab will allow the operator to see the probe cross-sections in both the 3D graphics window as well as a 2D plot of the points probed within a selected plane. By clicking on a crosssection within the 3D graphics window the operator will see the corresponding shape drawn out in the 2D plot on the right of the screen. The square dots represent the probed data points and the connecting arcs between the parts are also shown.



The operator can click and drag on any of the individual square dots to move that point around on the 2D plot. The software will re-calculate the line of best fit to wherever the pint is re-positioned. Furthermore, by holding down the CTRL key on the keyboard, the operator can select any two points on the 2D plot. With two points selected, you can then right click anywhere on the screen and a window with a various set of editing tools will pop up. The tools available are listed below, it is recommended to use the “make line” and “make arc” functions to smooth or edit the port as necessary. When selected the operator can use the same click and drag method as used with single points to move and re-position the set of points between the two selected points. This will make the software re-calculate the line or arc of best fit between the two selected points while keeping the cross-section geometry smooth and continuous.

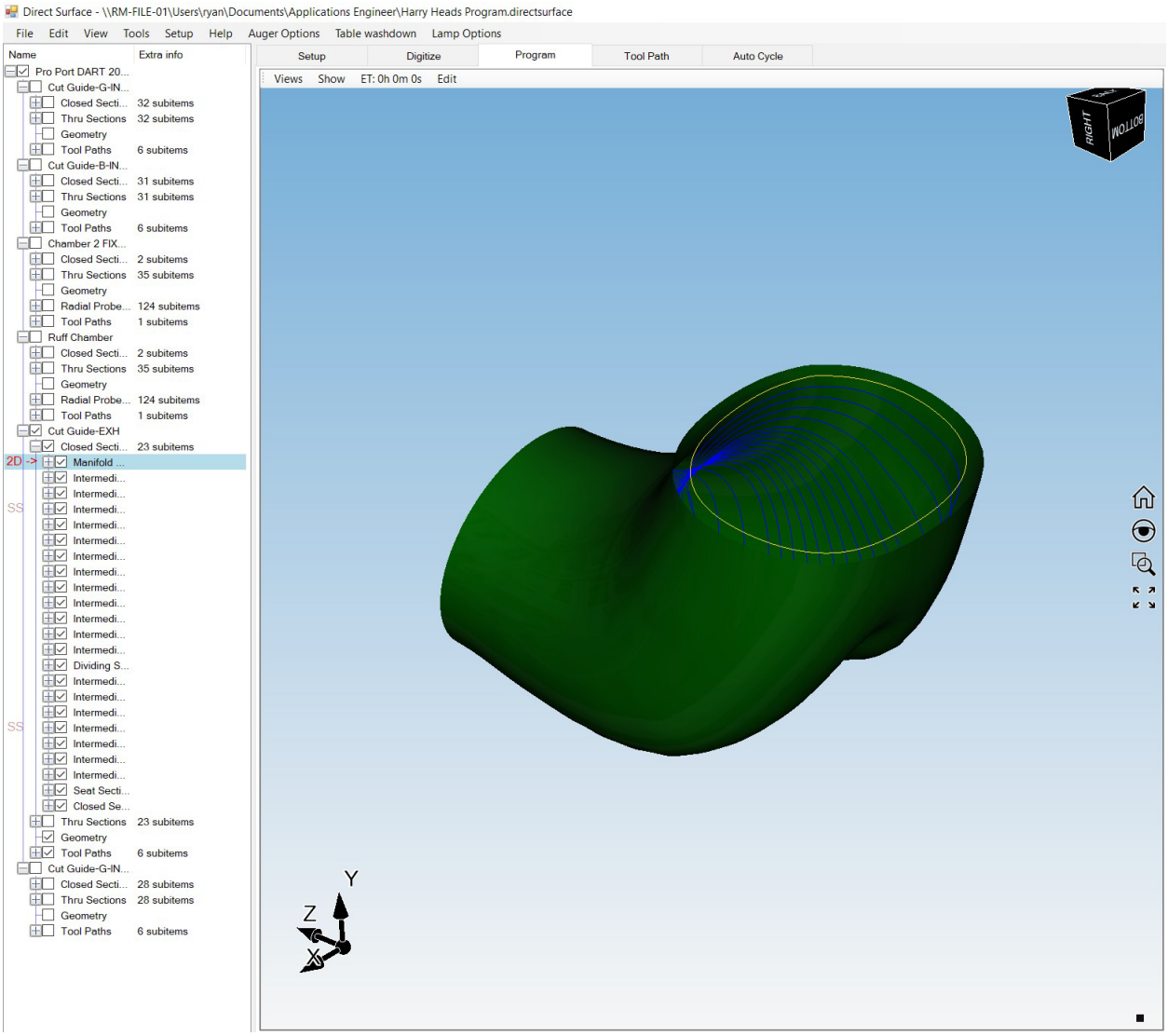
Make Straight Line: Re-arranges the points into a straight line between the two selected points with the location defined by where the operator clicks on the 2D plot.

Make Arc: Re-arranges the points into a smooth arc with a radius defined by where the operator clicks on the 2D plot.

Creating a Surface

After the digitized cross-sections have been manipulated to satisfactory geometry the next step is to create the port surface within the software. This process connects the cross-sections together to form a solid surface to which the program can then create tool path on. To accomplish this, right click on the port name in the cylinder information tree and select Create Surface. A series of windows will pop up, the first will ask the operator if a guide support exists, if it does then click “yes” if not click “no”. The second window will ask how many thru sections the program should use to “stitch” together the port surface, The default number is 72 and does not need to be changed so press “OK”.

The software will create a green surface that is offset from the blue cross-sections. The offset amount is the probe radius. This is because the cross-sections represent the probe center point when the tip contacted the port surface during digitizing.



Surface Manipulation

Before creating the toolpath, the operator should take some to inspecting the created surface to make sure it looks correct. If manipulation is required, then the previous method of editing the cross sections can still be used to manipulate any specific areas the operator desires. Editing the cross sections will automatically update the surface to the best fit available based on the edits. **Creating a Toolpath**

Once the surface has been manipulated to the operator's satisfaction, then the next step is to create the tool paths that will be used to machine the port. This is done by right clicking on the dividing line cross section in the cylinder head information tree and selecting one of the options for tool path creation that pop-up. The tool path options, and their descriptions are listed below;

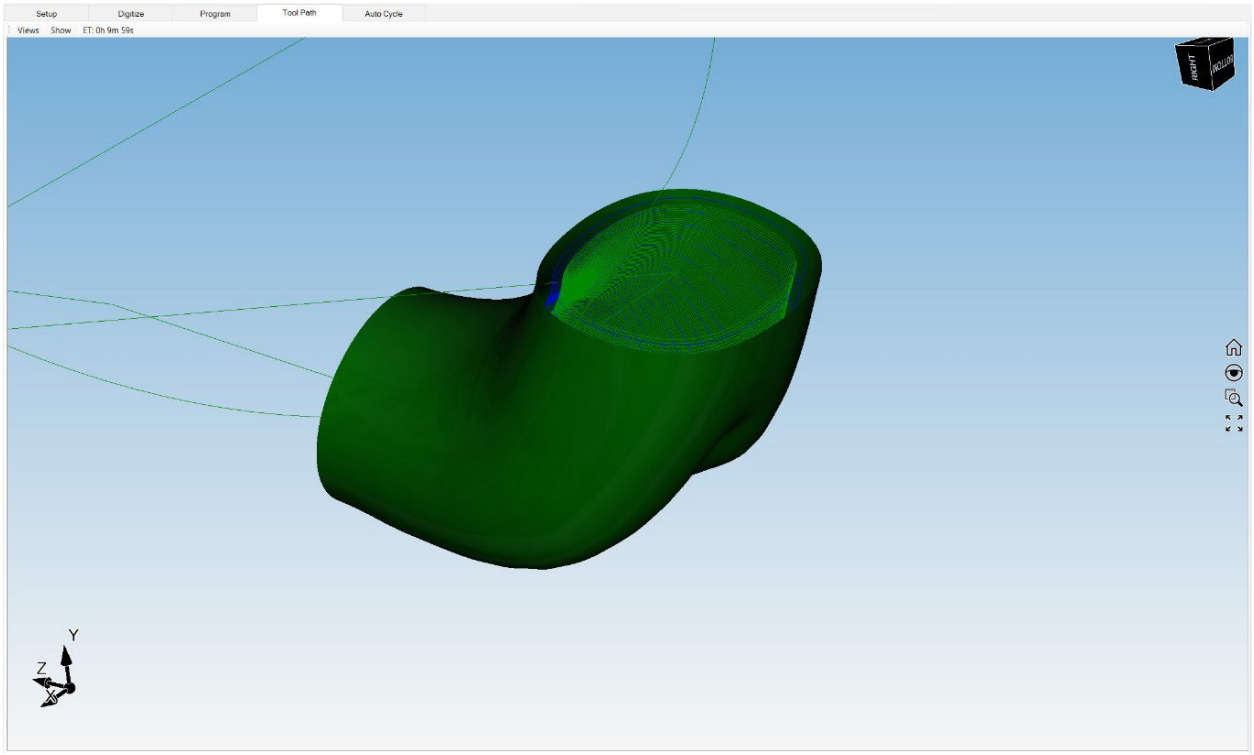
Make 2 Tool Paths: Creates two tool paths based on the two sections of planes above and below the dividing plane

Make 2 Dimple Tool Paths: Creates two dimpling operations on the two sections of planes above and below the dividing plane

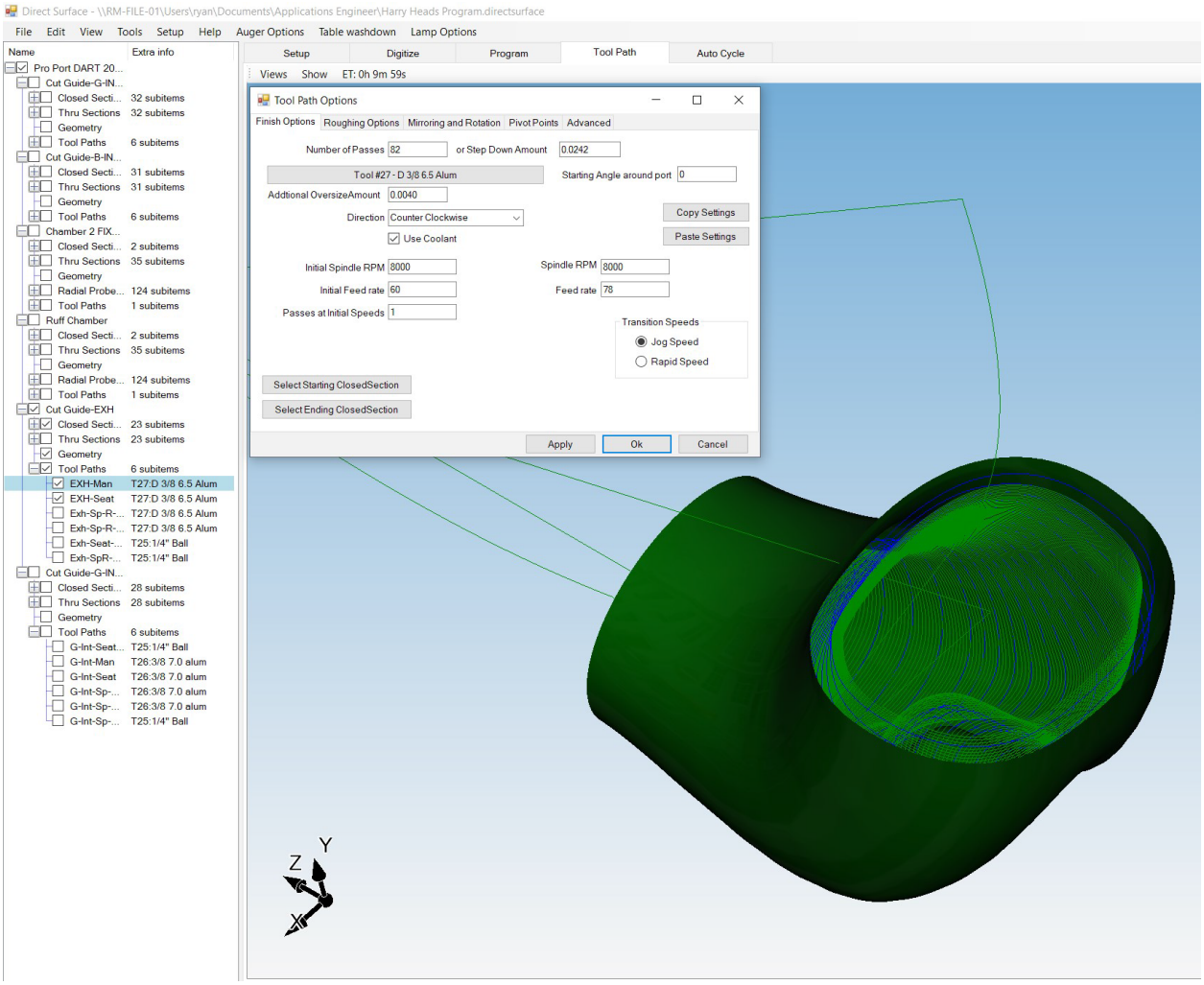
Make 2 Billet Roughing Tool Paths: Creates two roughing tool paths that are used to remove large amounts of material from the center of the port. This is used when creating billet heads or with original castings that are being significantly enlarged

Make 2 Spiral Roughing Tool Paths: Creates two roughing tool paths that are used to remove a relatively large amount of material from a port. This is used when significant enlargement of ports is to be done.

Once you select a tool path option, you can switch to the Tool Path tab and the software will automatically generate the toolpath. Once created you should see the toolpath in light green inside the port. The result should look like the figure below.

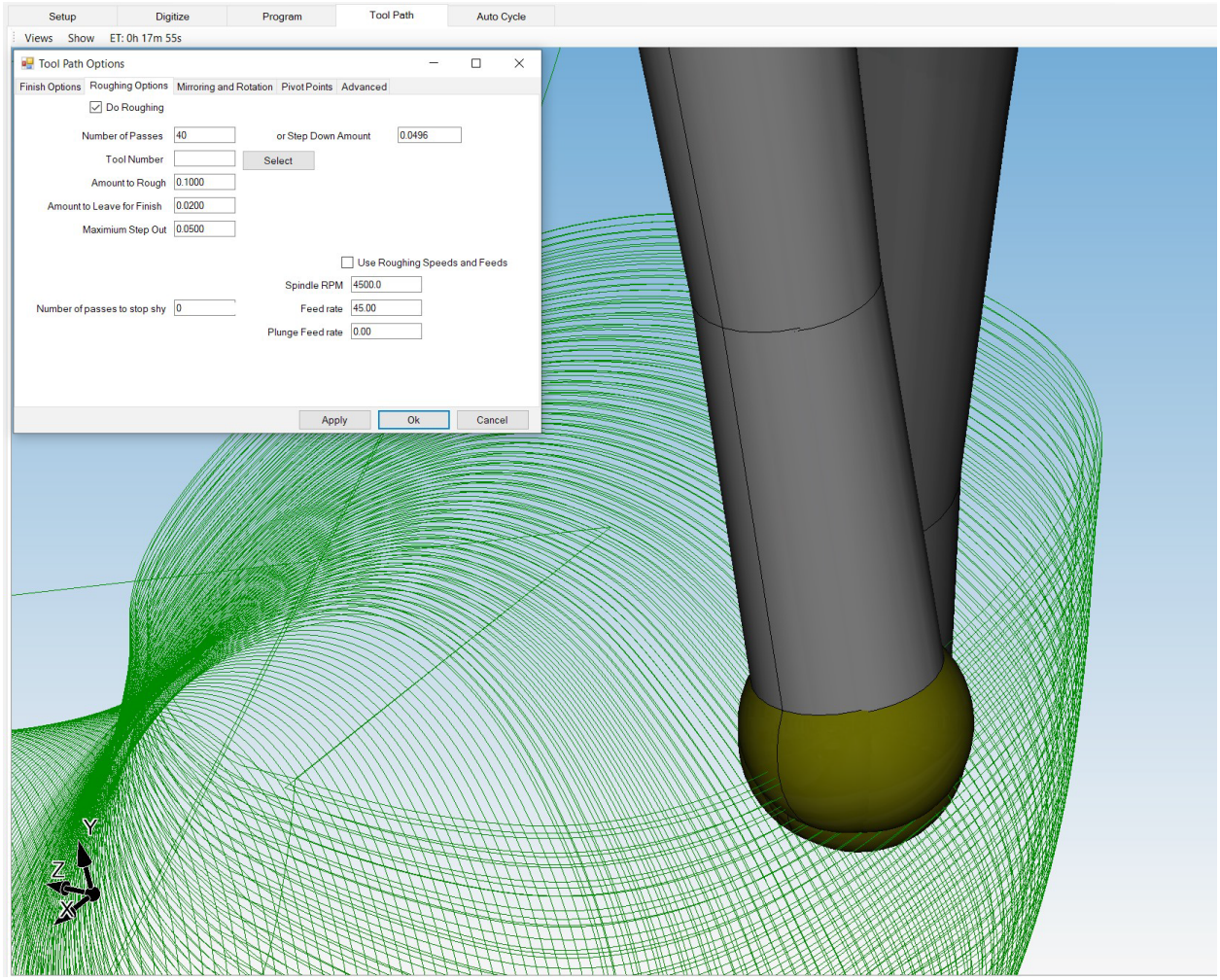


To edit the tool path parameters double click on the tool path you wish to edit from within the cylinder head information tree and the Tool Path Options window will pop-up. From this window you can change the number of passes, step-down increment, final port sizing, along with all other parameters that define the tool path. You may also edit the Feeds and Speeds for the tool path from this option window as well.



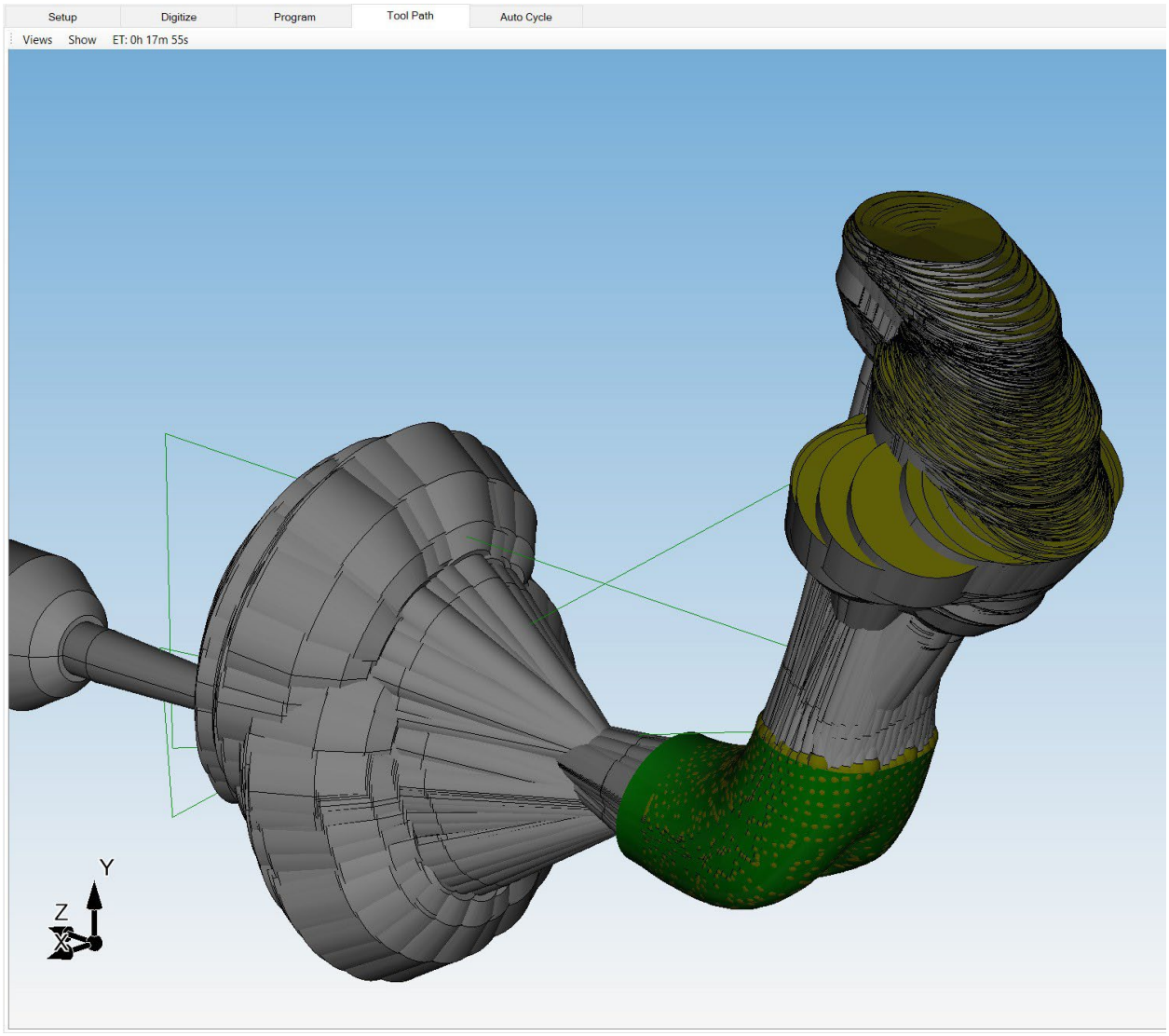
Creating a Roughing Routine for The Toolpath

To create a roughing routine, click on the Roughing Options tab within the tool path options window. Enter the number of roughing passes or a step-down amount that you would like the cutter to increment by as well as the amount to rough, amount to leave, and maximum step-out. Clicking apply will calculate a basic roughing tool path which will be offset inward from the original toolpath based on the parameters mentioned above.



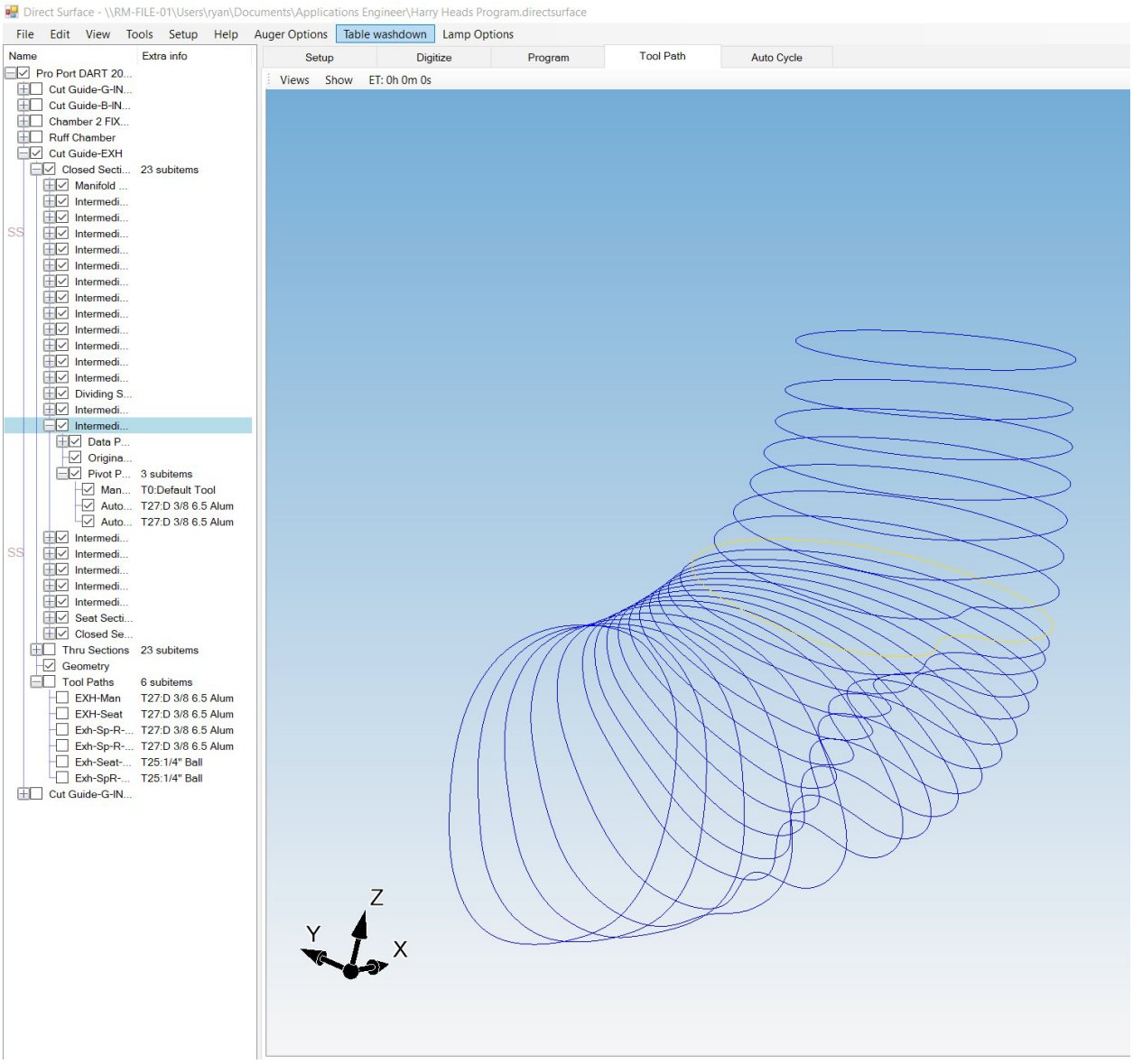
The roughing routine will appear in the 3D graphic window as shown in the figure above. You can hover over the tool path with the mouse to verify the tool positions to see where the cutter will be positioned during the process.

Next, click on ‘Show’ and be sure to have Solids and Tool checked. Next select the “Show toolpath on all points” option to present a simulation of all of the locations that the tool will be moved to during the porting process. Rotate the view to insure that at no point the tool will be outside of the port surfaces. This is a quick and effective way to verify that the program was written correctly.

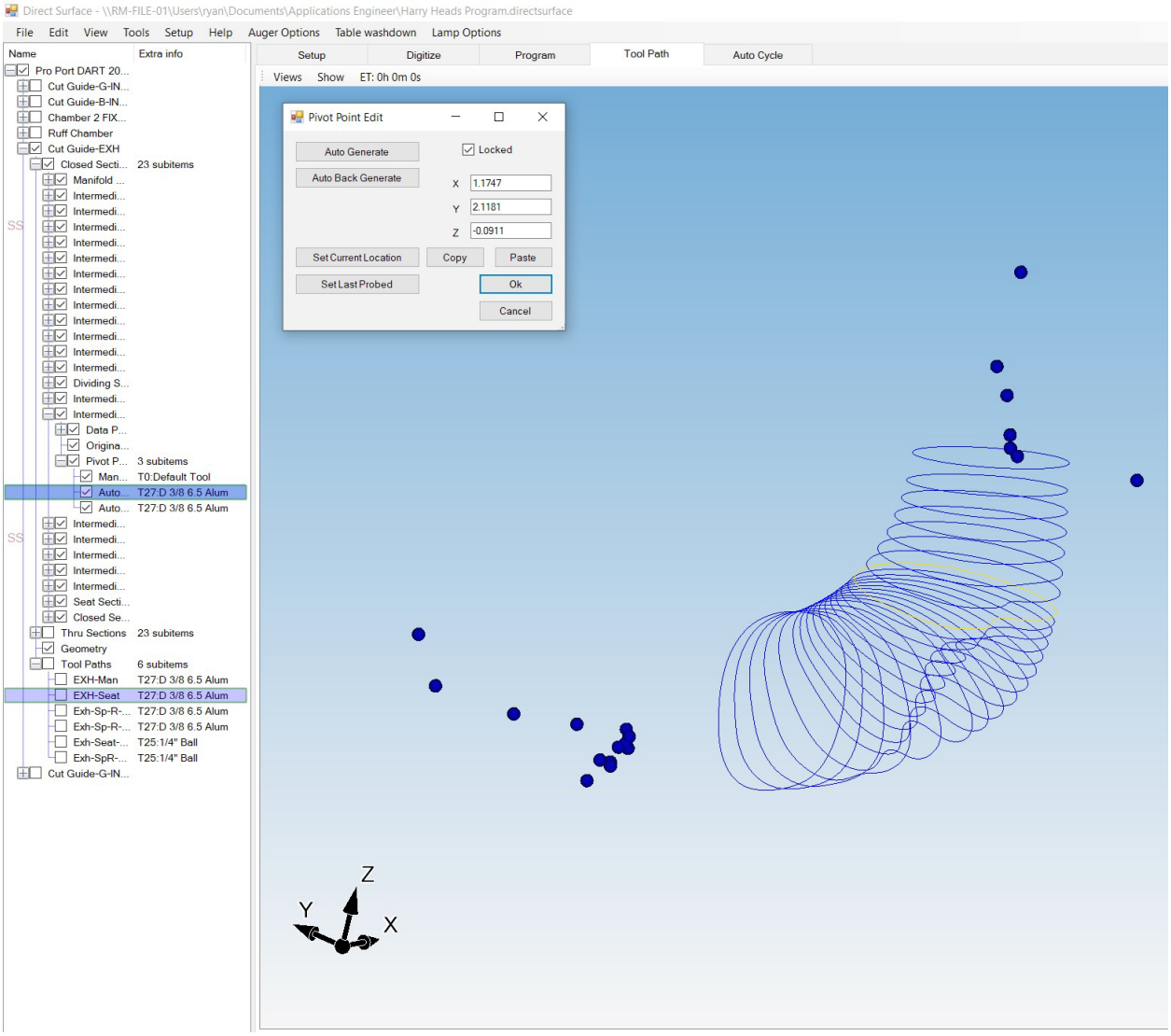


Moving Pivot Points

If during the tool path verification process, you notice the tool tip or shank are out of position at any moment, then it may be necessary to move pivot points for certain cross sections. This can be done by isolating the pivot point that corresponds to a specific cross-section, and then moving that pivot point to a different position in space to allow the tool a better angle to machine the port section at. To accomplish this task start by highlighting a cross section around the issue in the 3D graphics window.



Open the Closed Section that you want to manipulate the pivot point by clicking on the “+” sign. Then open the ‘Pivot Points’ for this cross section. Double click on ‘Automatically Create Pivot Point”. A box will pop up that will allow you to manipulate the X, Y, and Z points of this pivot point. After changing a pivot point be sure to put a check in the “Locked” box.

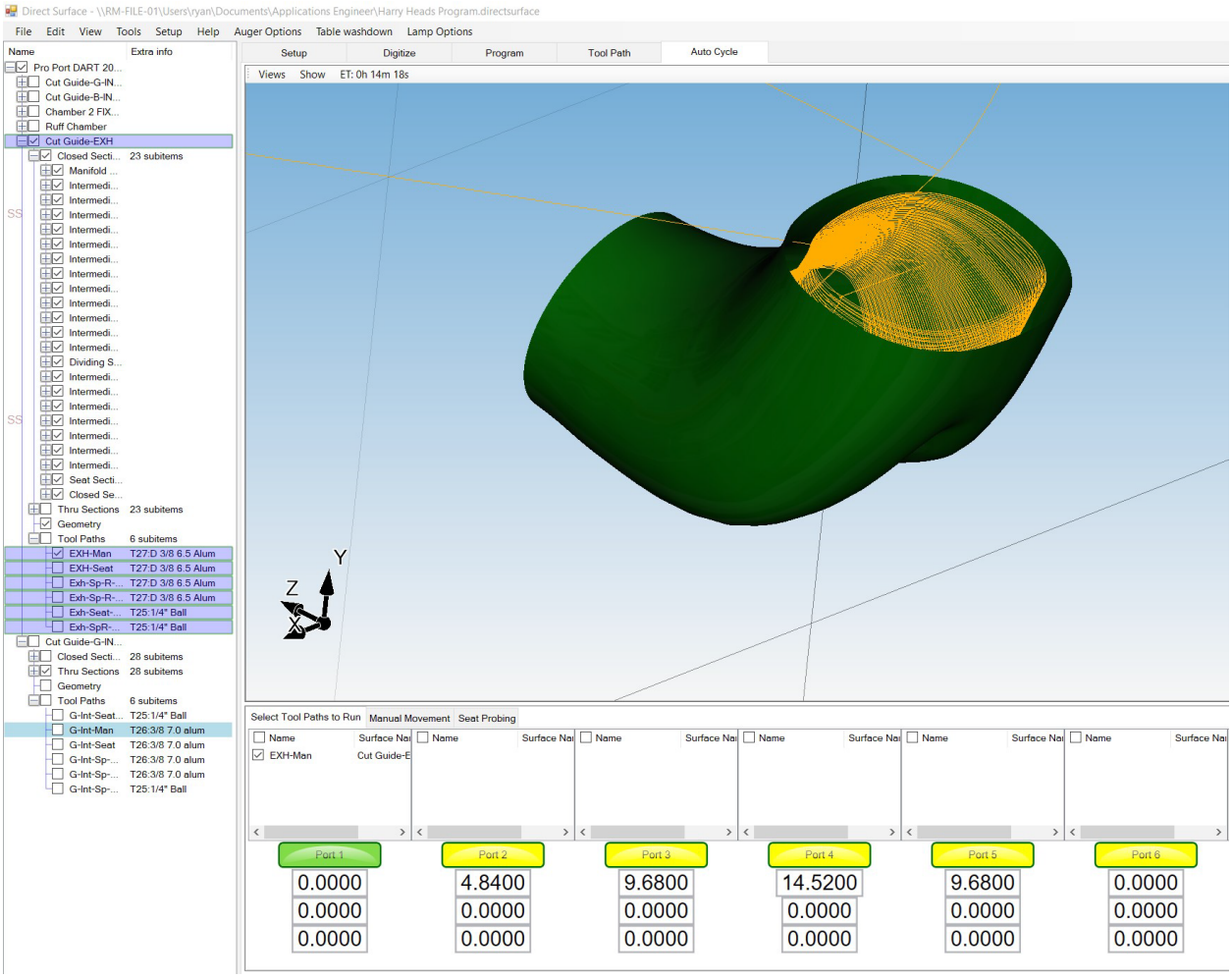


Make sure you click on ‘Show’ and ensure that the Pivot Points has a check by it. When you highlight the proper cross section you will see the pivot point show up as red. Depending on the direction(X, Y, or Z) you want to move the pivot point this will graphically show you where the pivot point is being moved to.

Keep in mind when moving pivot points that X,Y, and Z sit normally when the A axis is located at A zero. In other words when A is positioned at zero X positive is to the right, Y positive is facing to the back of the machine and Z positive is straight up. As the A axis rotates so do Y and Z. Let’s say for example the A axis rotates 90 degrees so that now the combustion chambers and fixture plate are facing the back of the machine. The Y and Z have rotated along with the head. In this instance the Z positive is now facing the back of the machine and the Y positive is facing toward the table. Think of it as fastening an X,Y,Z symbol on the top of the fixture plate when A axis is at zero and combustion chambers are facing up. As the head rotates around the X,Y,Z symbol rotates with the head. Once the pivot point has been moved the tool path automatically regenerates and you can simulate the cutter path to ensure that you have addressed tool interference. If not, the pivot point may need to be moved again.

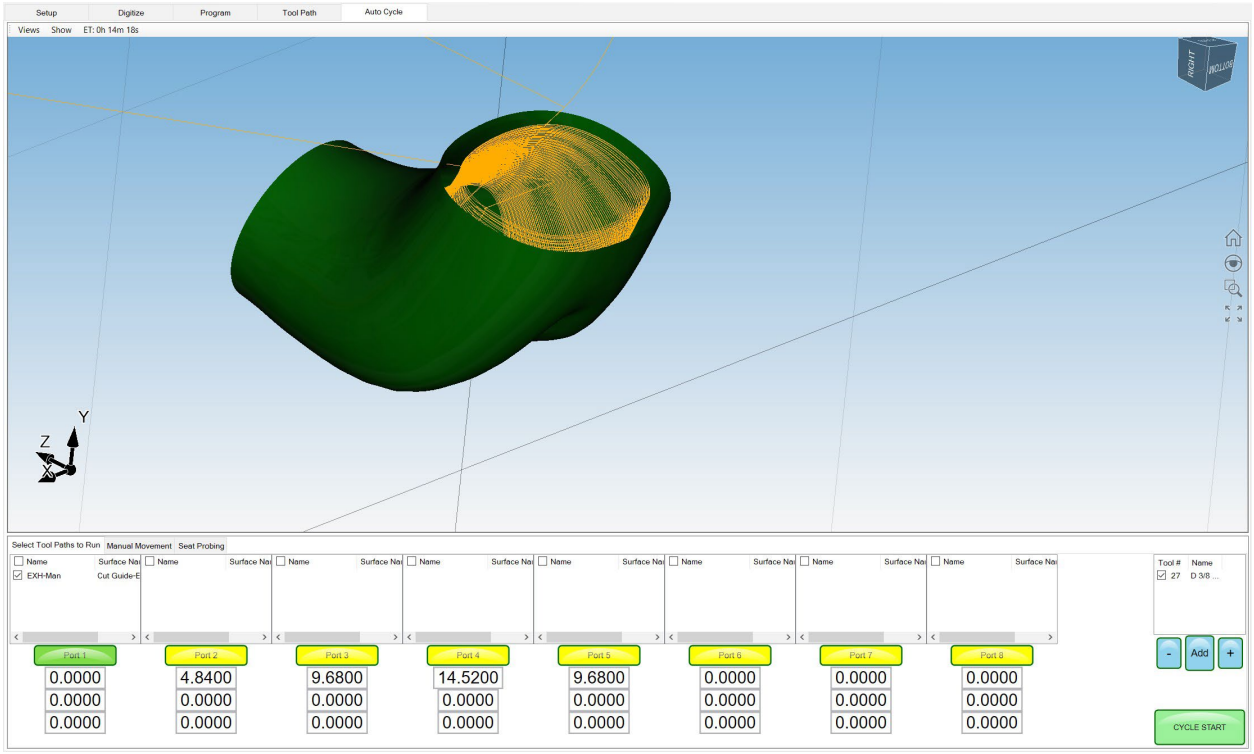
Loading and Running a Tool Path

Once you are satisfied with the simulation of the tool inside the port it is now time to run a tool path. Start off by clicking on the ‘Auto Cycle’ tab. Also, make sure that the ‘Select Tool Paths to Run’ tab is selected down at the bottom of the page.



Open the Tool Path in the cylinder head information tree. To add the tool-path you wish to run, drag and drop the tool path into the open box above the port number you want to perform the operation on. The tool path should appear in the graphics window, you can also select the show solid option to see the port surface in the 3D graphics window.

Notice that the port number is highlighted either green or yellow. Having the port lit up in green means that these are the ports that will be machined. In the example below Port 1 was used to digitize from. Whichever port was used to digitize will need to have the X, Y, Z location boxes set to zero. To set the location for the other ports you may type in the bore spacing in the top box.



The diagram below shows the actual tool path the cutter will follow. You will notice that there is a green arc that moves from the intake part of the port around to the seat side of the port. This represents the movement of the tool when it completes the first half of the port, then pulls out, then rotates around to the opposite side of the port to begin its operations where it will join in the middle of the port.

At this point to start the tool path cycle simply click on ‘Cycle Start. A message will come up verifying that you have the correct tool loaded up. If you do not have the correct tool click on ‘Change Tool’. If you do simply click ‘Okay’, The Program will start and beginning cutting the cylinder head ports.

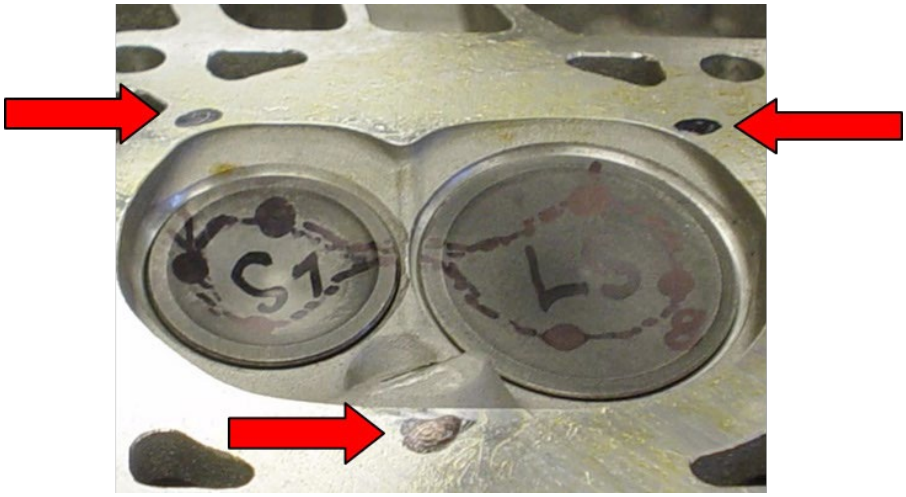
NOTE: It is very important to acknowledge that after clicking on OK that the machine will become active and will move quickly to its first position. Especially when you are first proving out a head the operator needs to keep their hand on the hand wheel and E-stop button. While the machine is in motion if you turn hand wheel counter clockwise it will slow the feed of the machine all the way down to zero inches per minute. This allows you to ease the cutter into the work and avoid potential crashing due to incorrect fixture or tooling offset or any other variable that could be overlooked.

Creating A Combustion Chamber

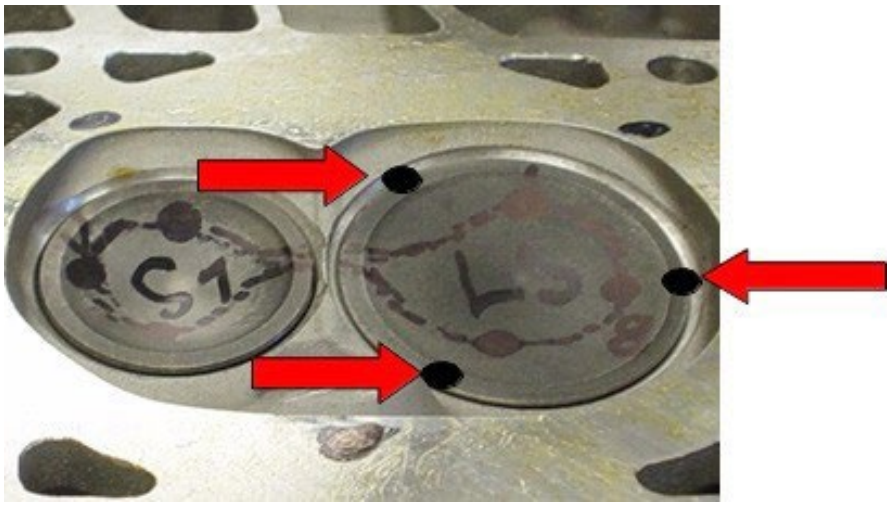
To create a combustion chamber right click on the head name then left click on ‘Add a New Chamber’ like shown below. You will see ‘Chamber’ appear just under the intake and exhaust ports. You can right clock on chamber and rename the chamber if you desire. **Adding Planes**

The first step in creating a combustion chamber is to establish two planes. The first plane will be established at the deck surface of the head. You will pick up three points on the deck to define this plane. The second plane we will define is the angle of one of the valves. Note that it does not matter which valve you will use. To define a plane on the valve you will need to collect three points. In the photo below there are arrow showing approximate positions to establish the two planes.

The three points that need to be collected to define the deck plane surface:



The three points that need to be collected to define the valve plane surface:



Defining the Planes

To define the two planes in the software, start by right clicking on ‘Add New Planes for Probing’. It will tell you that you are using a certain tool number to collect the points. Make sure that you have set the proper tool set as active. This can be found in the table of tools under the setup tab. Whatever probe you are using set this tool number to active.

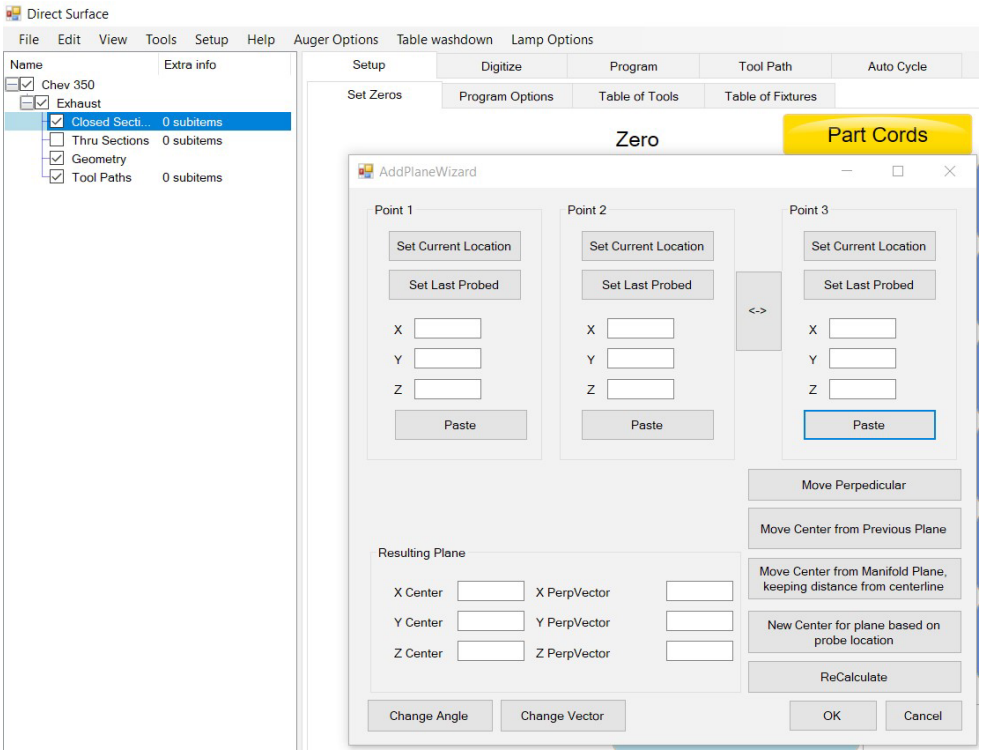
Defining Deck Plane

First Plane

After clicking on ‘OK’ an Add Plane Wizard will appear. You will enter three points into the wizard the same way we did when you defined the planes for the intake and exhaust ports. You will start by collecting 3 points on the deck surface like shown below.

Use the handwheel to bring the probe down in Z until the indicator light changes from green to red then move it back up in the positive Z direction until light changes back green. From within the Add Plane wizard click on ‘Set Last Probed’ to collect point’s one, two, and three. Remember, it takes three points to

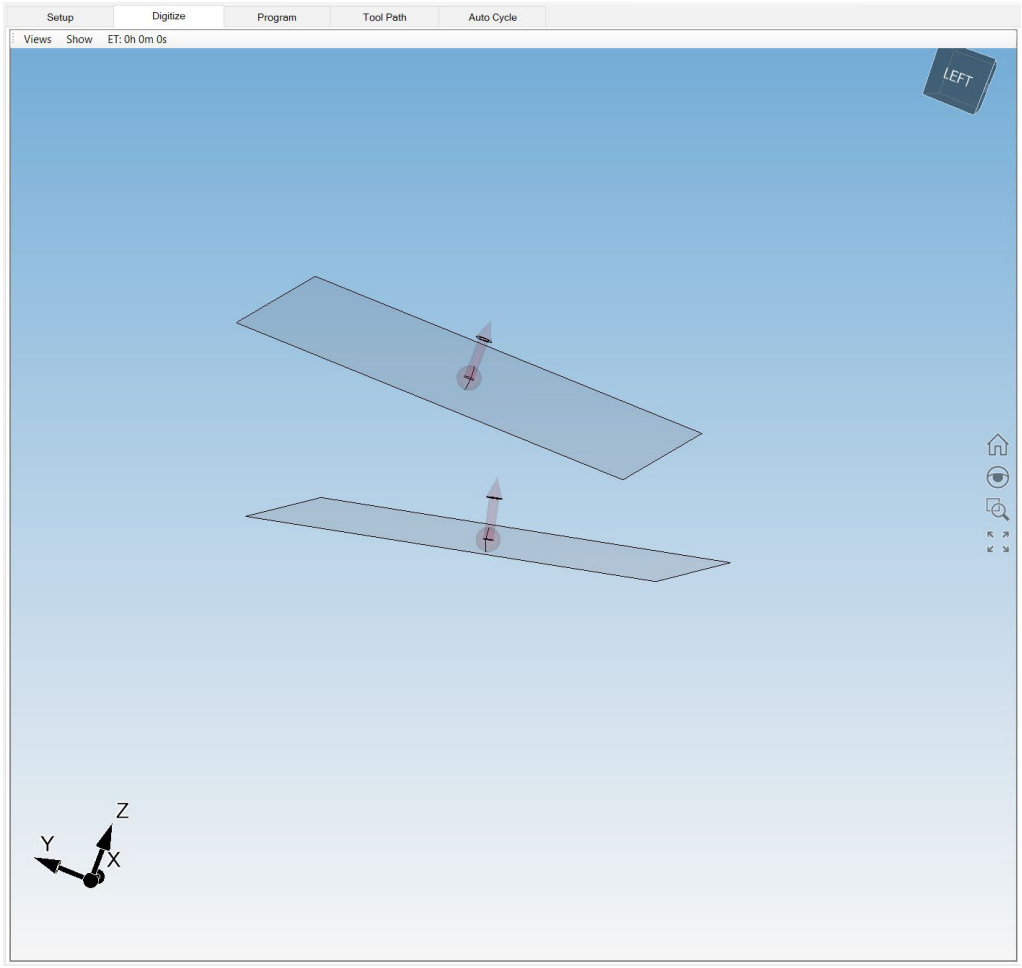
define a single plane. Note the position of the probe relative to the chamber's edge. The digitizing process begins in the center of the three points.



Second Plane

Just as you did with the deck plane you will again start by right clicking on 'Closed Sections Template' then left click on "Add New Planes for Probing." You should see a message appear that says "Adding new plane using tool #3" (If you are using tool #3 as the probe you want to use to digitize). You will repeat the same process that we used in creating the deck plane only this time you will use three points on the surface of the valve. See pictures below:

Once you have collected the three points you can verify that you have created the two planes by clicking on the ‘Digitize’ tab. If all you see is blank space ensure that you have clicked on ‘Show’ and that you have a check by ‘Planes’. You should then see your newly created planes in the graphics view below.

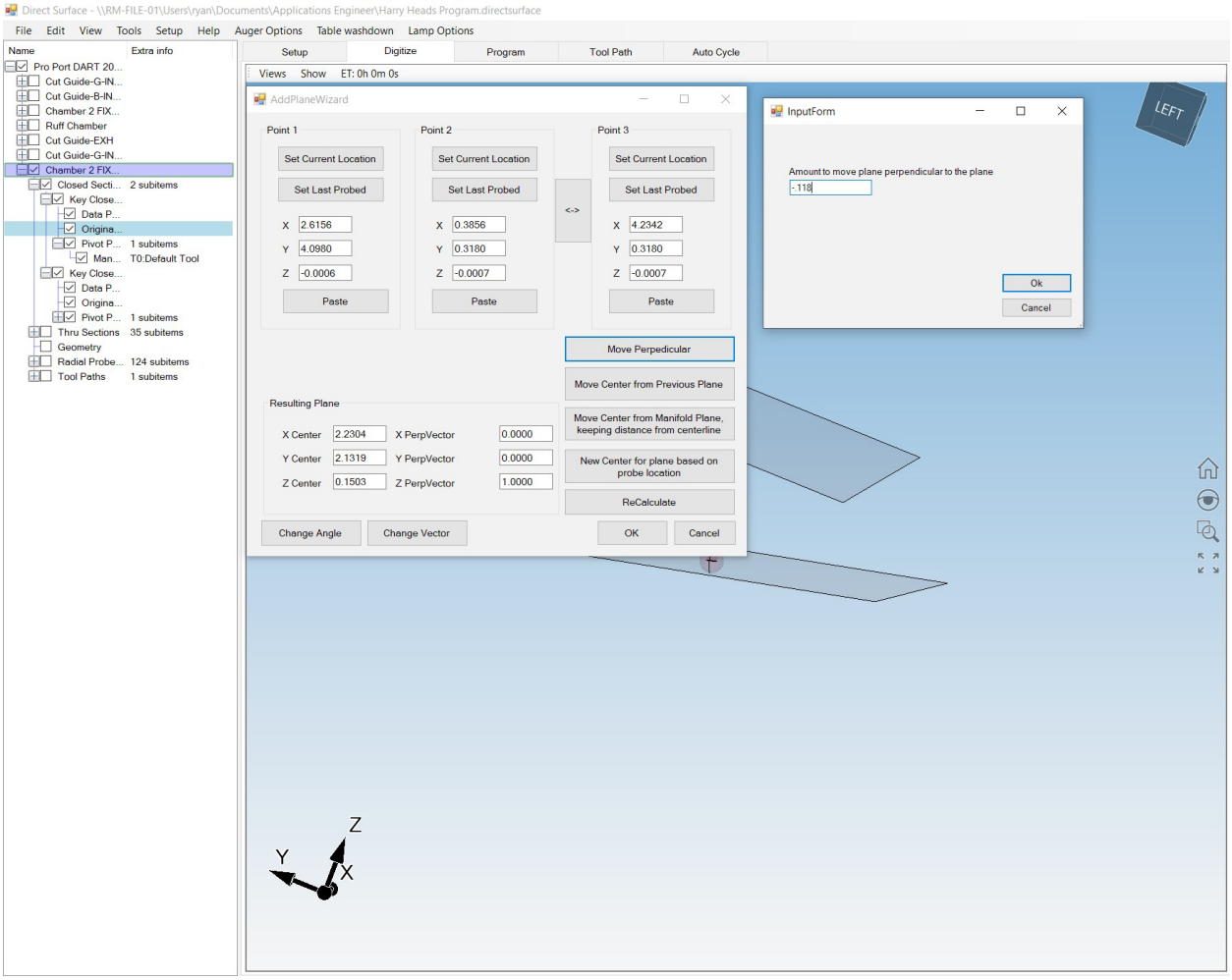


Moving Deck Surface

Before we begin digitizing for the cross section at the deck surface, we need to move the plane down toward the valves. You will notice that there is a red arrow at the center of the plane. This represents the positive direction. In the example above you will see the valve plane is on the opposite side of the red line. This means that you will move the plane negative (.118). Keep in mind that it will not always be a negative number. It can vary depending on how the software calculates the plane.

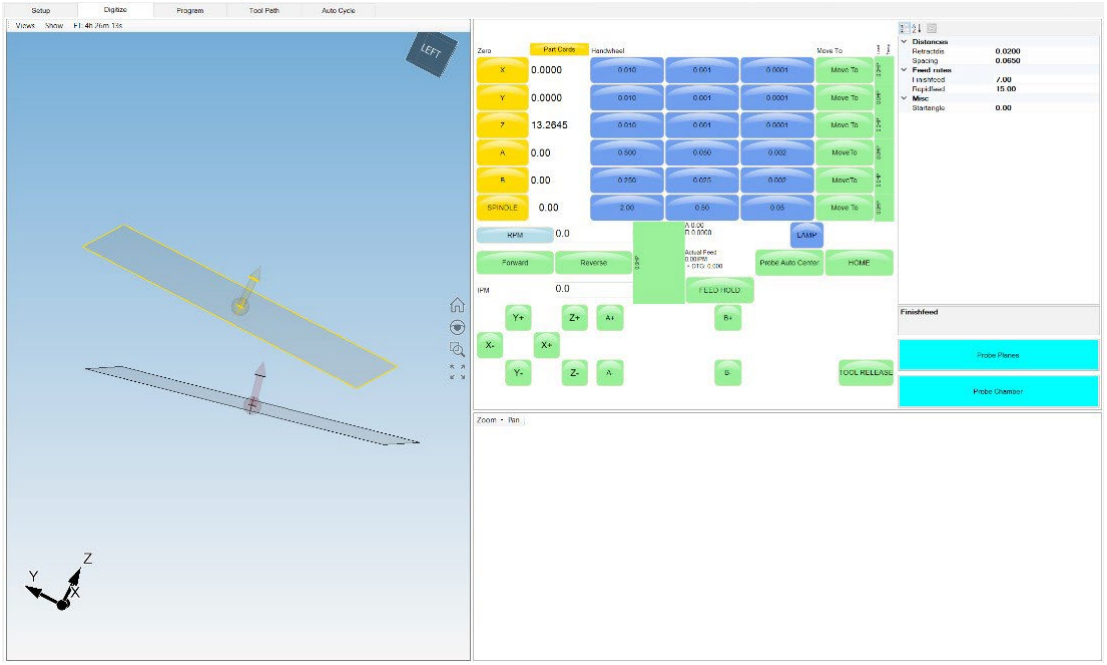
Another way to analyze the direction you want to move the plane is to click on show and check ‘Tool’ so that you can visually see the probe in the graphics view. Physically wheel your probe down so that the midpoint of the ruby is sitting at the deck surface. When you drop the plane down .118 you will see it move in the graphics. You can verify that the plane should be going through the tip of the probe exactly through the center of the ball. If it does not, you most likely moved the plane in the wrong direction. In this case you would have to double the .118 to .236 and change the sign.

To move the deck surface, open the deck surface plane that has been created. Next, double click on original plane. This will re-open the plane wizard. Next, click on move perpendicular. It is in this field where you will enter the value to move the plane. In this case it will have a value of (-) .118. This is approximately half the diameter of the probe tip. Actual tip size is .236 and half of that value is .118.

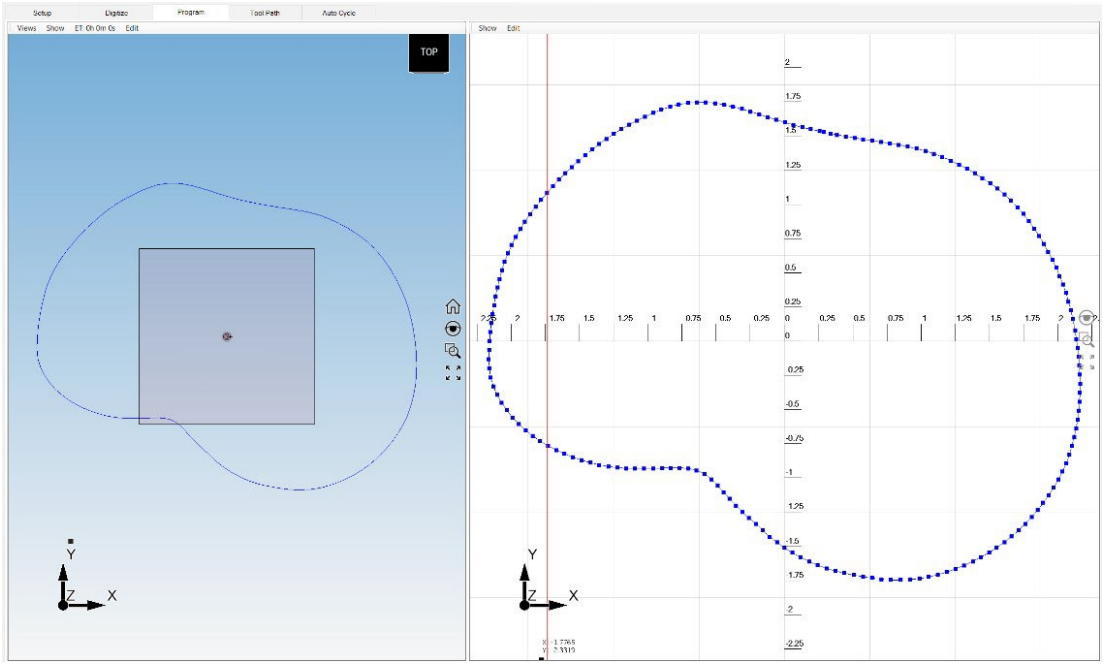


Create Cross Sections

After creating two planes you must now define the cross sections that are essentially the inner and outer borders in which the probe will be contained in. To define the cross section at the deck surface you will simply highlight the deck plane then click on probe planes.



Once the deck surface cross section has been probed automatically you should see something similar to what is shown below. Again, this is simply an outside border that the probe will move to when we begin digitizing the entire combustion chamber



Inside Border

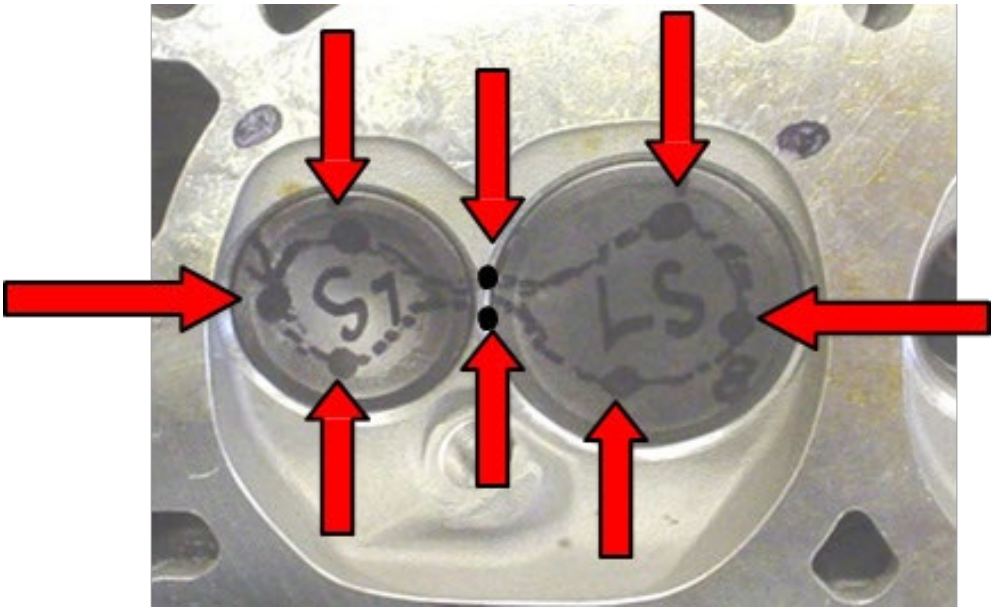
After the outside border is defined, the next step is to define the inside border for probing. The goal is to collect 8 data points in the bottom of the combustion chamber to define a rough shape that represents inner portion of the valves and chamber.

Due to the irregular shape of the combustion chamber and valves this process has to be done manually with the probe. Inside of the table of tools there is a button labeled ‘Start Probe’. Click on this button and

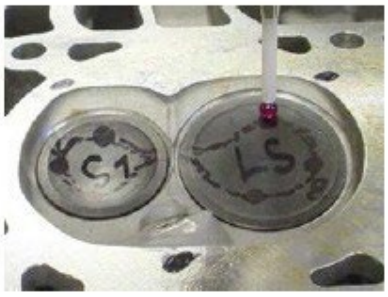
you will see the probe become activated. From the digitize tab we will jog the probe tip to the contact points as shown in the figure below. As we have done in previous procedures, you will move the probe to the point you would like to record, use the handwheel to move the probe down until the light on the probe turns red, and then back off until the probe light turns green using the .001" handwheel increment setting. The points shown in the figure below are arbitrary, you may select points in different locations for your application. The only requirement for the point selection is that the points should be offset from the edge of the valve and should be the rough representation of the combustion chamber outline.

NOTE: The example below shows a combustion chamber probing process for a 2 valve U.S. manufacturer V8 cylinder head. If you are attempting a chamber with multiple valves you will want to tweak the lower profile shape and points location to make a rough estimate of the outline of the cylinder heads combustion chamber, while still using points gathered from locations on the valves.

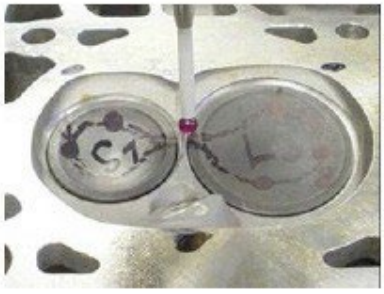
To record the points as you touch the probe to the individual locations click on 'Tools' label in the top right corner of the software screen by the 'File', 'Edit', and 'View' icons. From within the Tools drop down tab, select "Record Point" to store the current location of the probe. To speed up this process you may also use the "CTRL-R" keyboard shortcut to store the points as you go instead of navigating to the Tools tab each time.



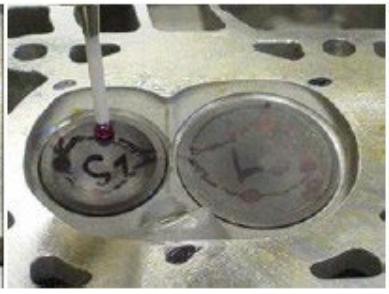
POINT 1



POINT 2



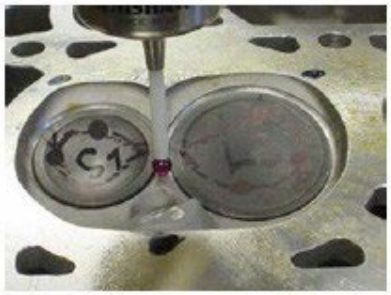
POINT 3



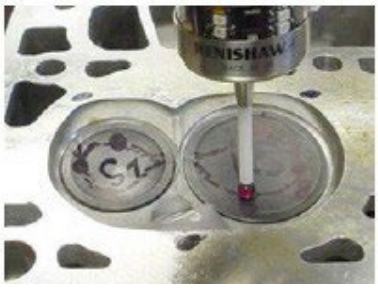
POINT 4

POINT 5

POINT6

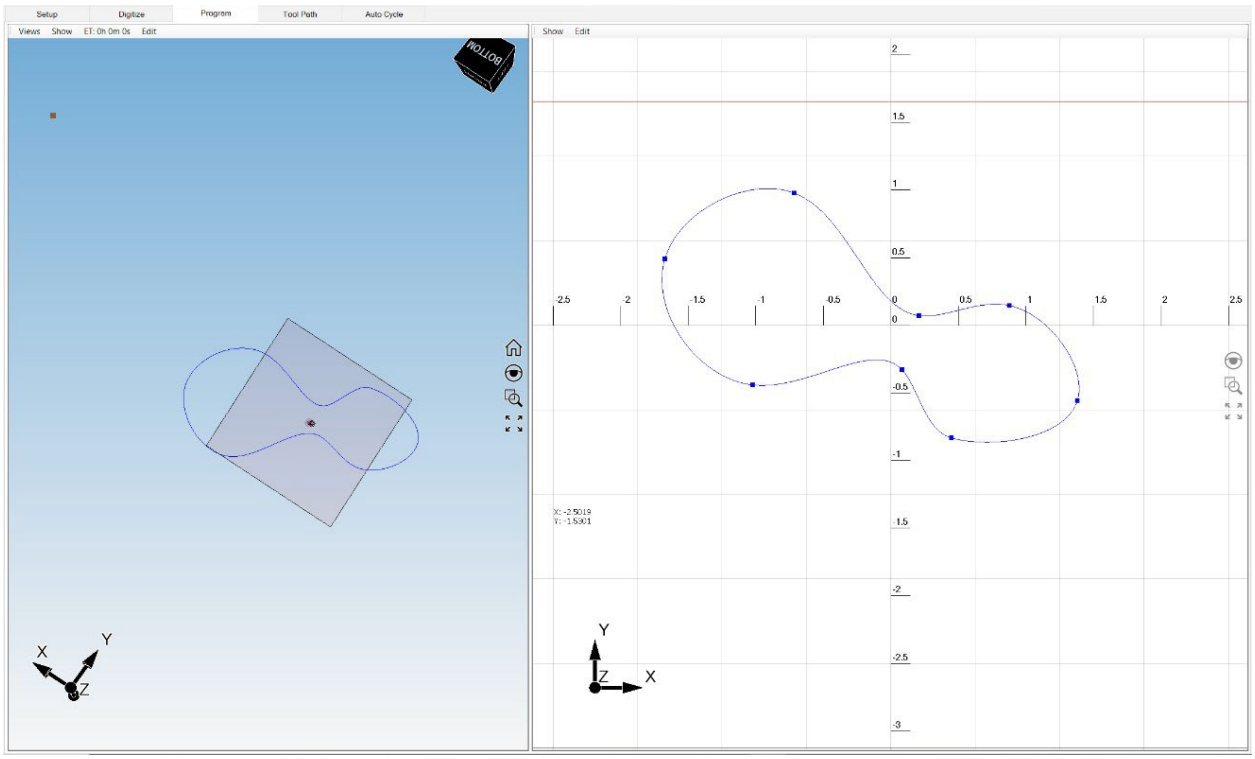


POINT 7

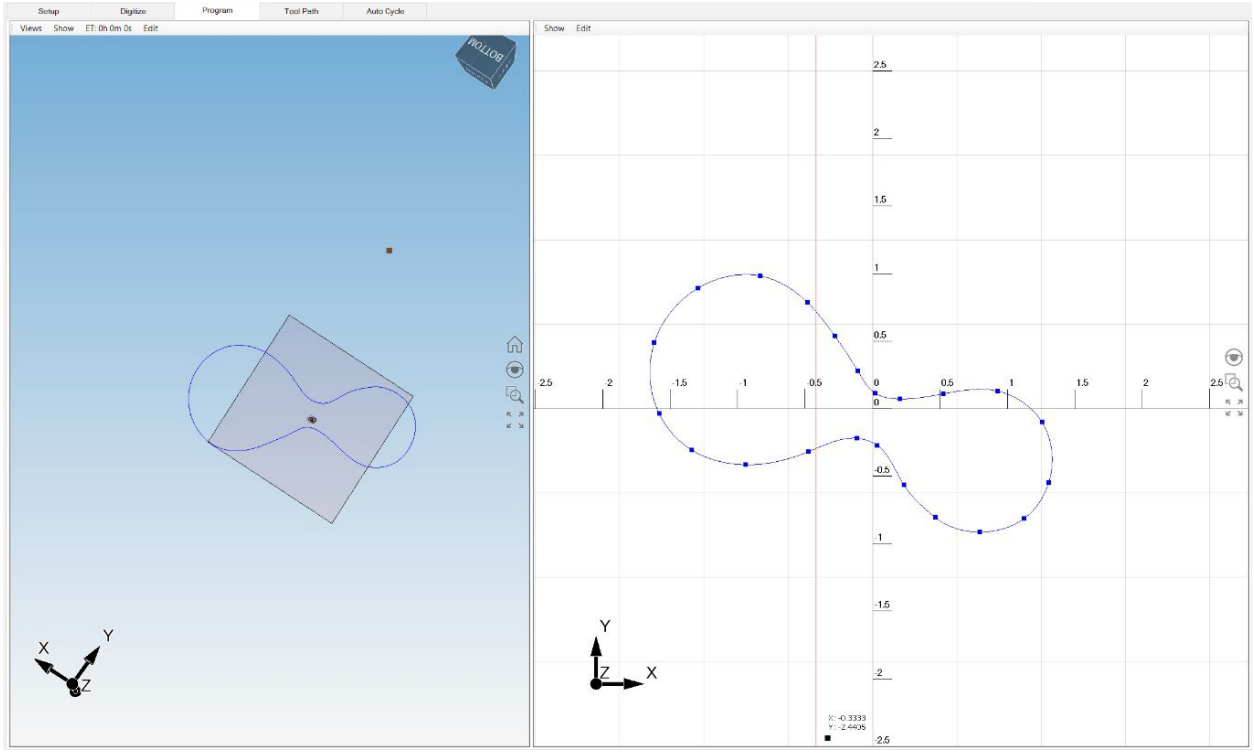


POINT 8

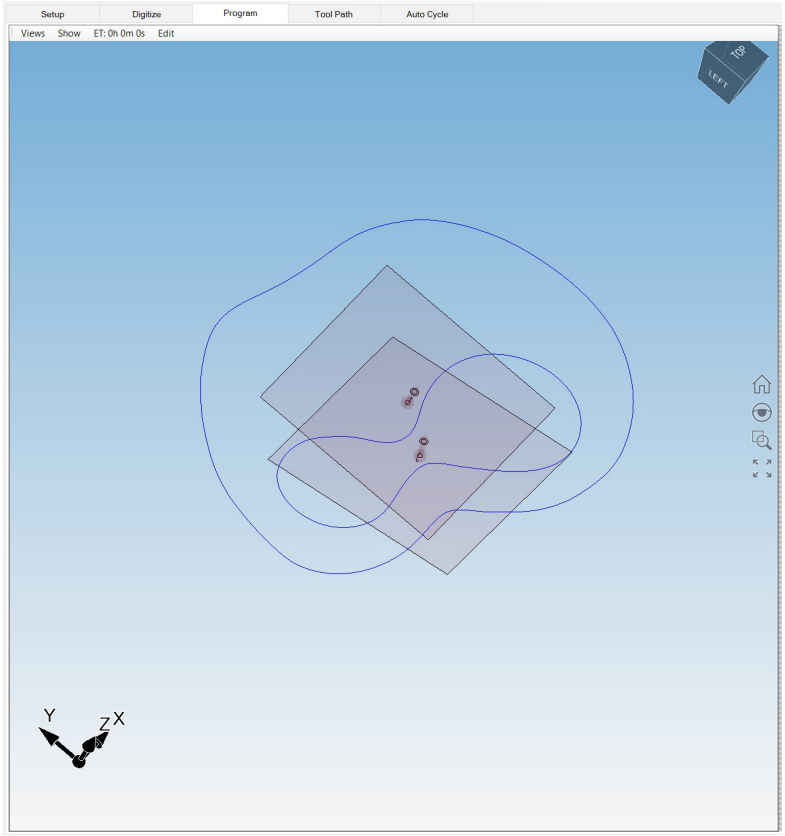
After you have recorded all eight points you should have something that looks like the picture below. To View the created lower boundary, switch to the Program Tab where you will see both the Plane and cross section in the 3D graphics window as well as the 2D Plot of the boundary.



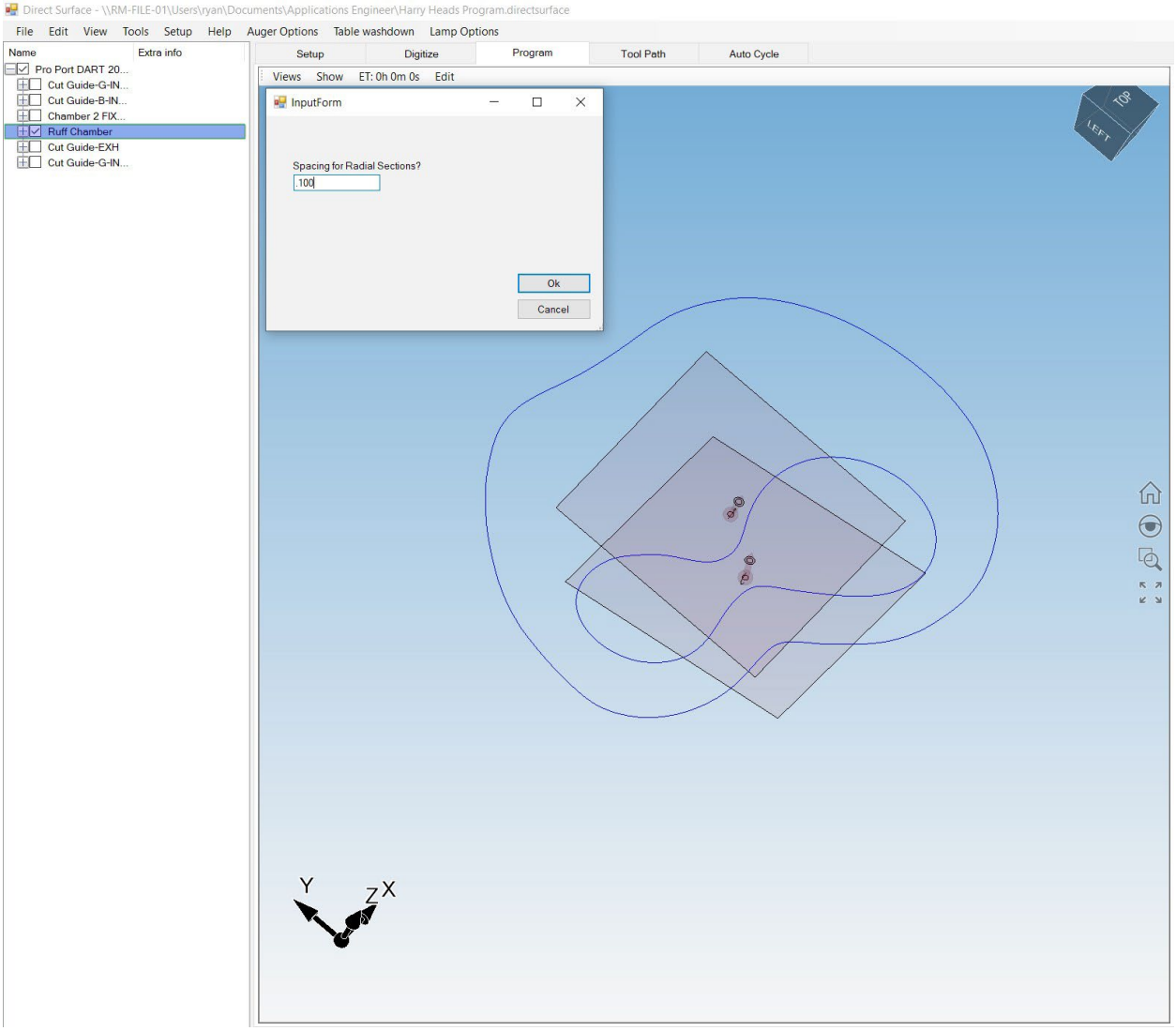
The next step is to add some more points to the cross-section to add control and smooth out the lines. To add points to the drawing, right click between any two points to create a point in-between them that will automatically cause the sketch to be re-drawn. A good rule of thumb is to add a point in-between any two points that are spread more than .5” apart based on the number line in the 2D plot. Once you have added the points, you can use the click and drag method to move the points around and smooth out the lower boundary. The figure below shows the added points and smoothing that was done to the sketch in the above picture.



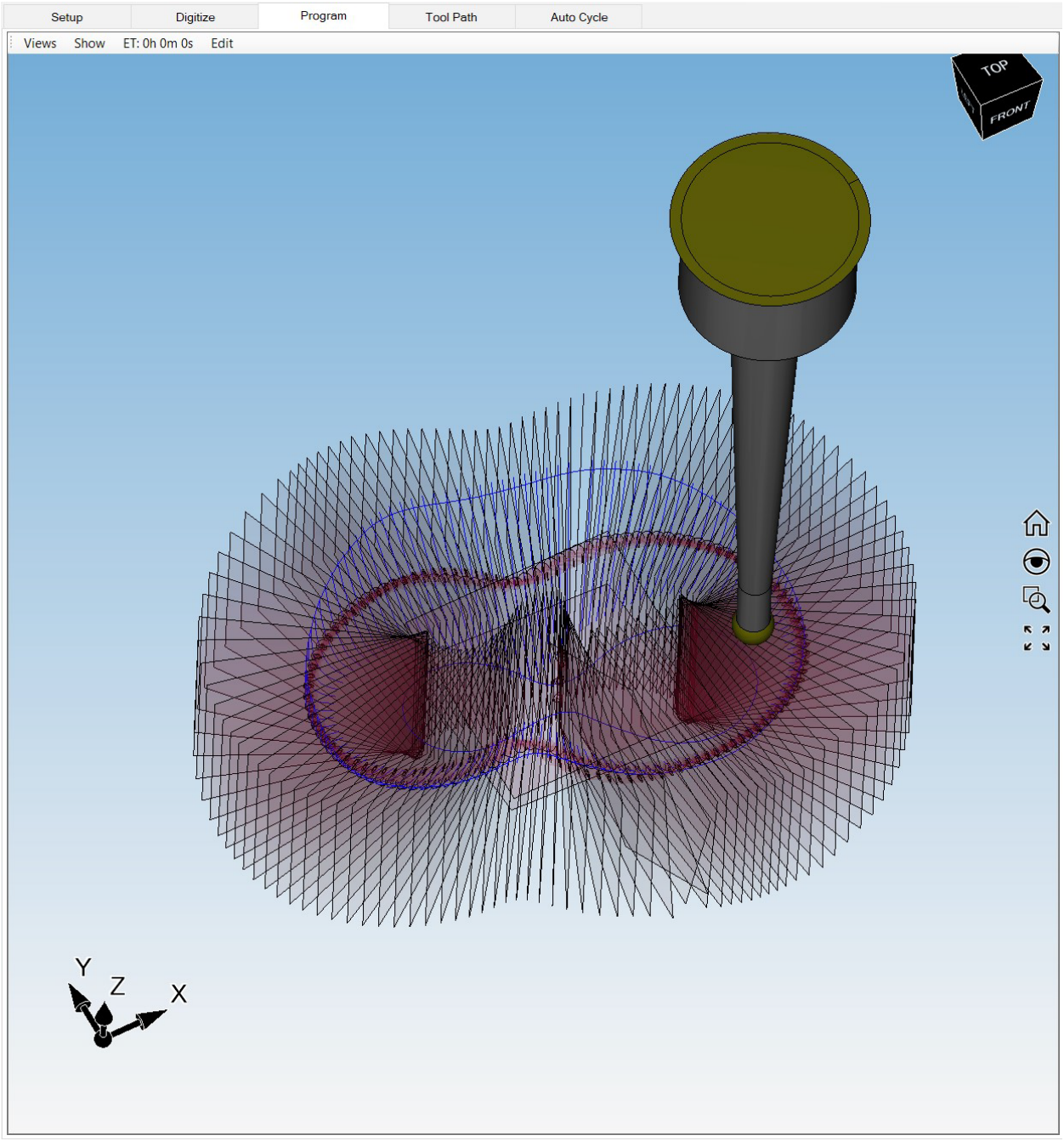
Now, check on the deck closed sections box so both the boundaries are displayed in the 3D graphics window. At this point we have created two boundaries. These containment boundaries define not only where the probe will digitize but also where the tool will cut when a tool path is created.



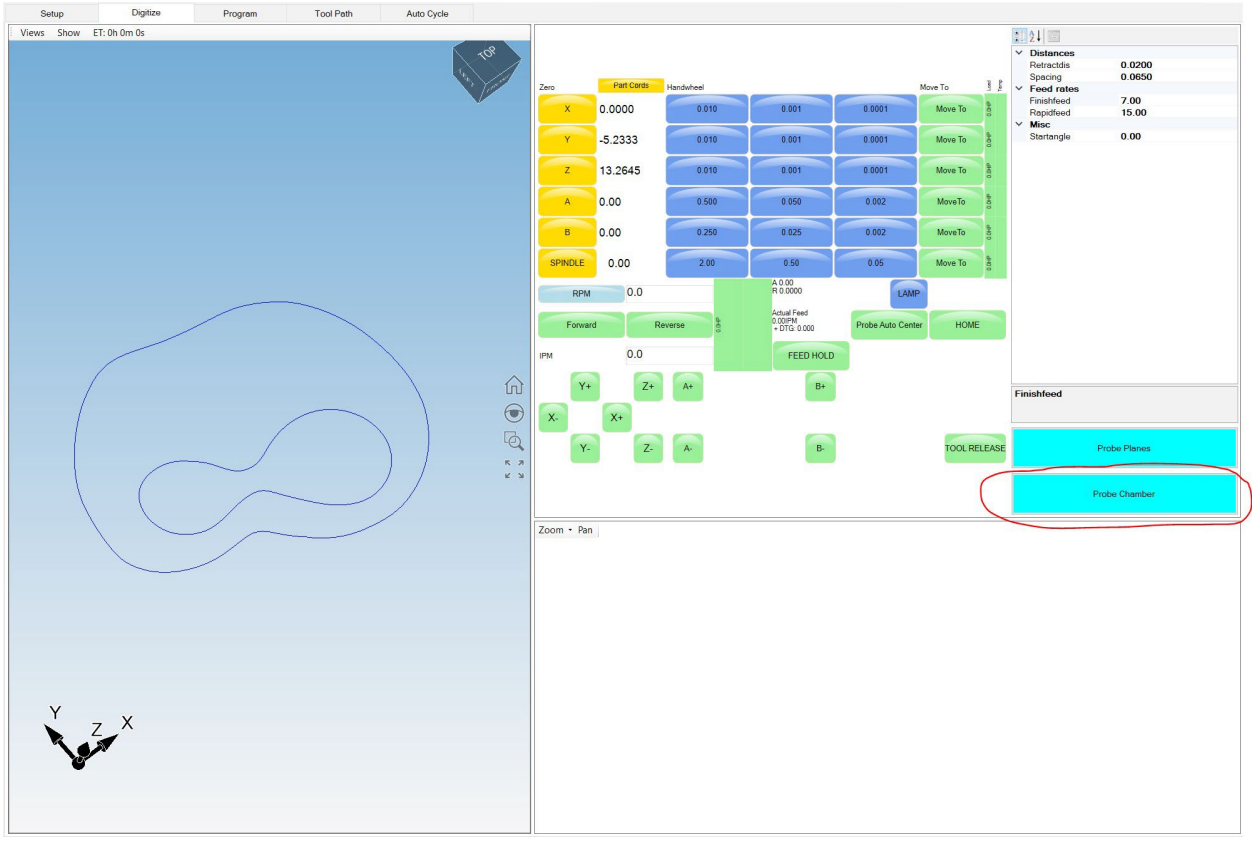
Now that the closed sections are complete and displayed. The next step is to create a set of radial lines that project out from the lower section to the upper section that the probe will used to follow and collect data points on. To accomplish this right click on the named chamber in the cylinder head information tree and select “Radial Operations” followed by“create radial template”. A window will pop up asking for a spacing for the radial sections, in this example we are using .100” spacing.



To view the radial sections that we created make sure that the “show planes” option is selected from within the show tab in the 3D graphics window. Then make sure that the “Radial Probed Planes” box is checked under the Cylinder head information tree. If done correctly then the 3D graphics window should look something like the image show below with all the probe planes visible.

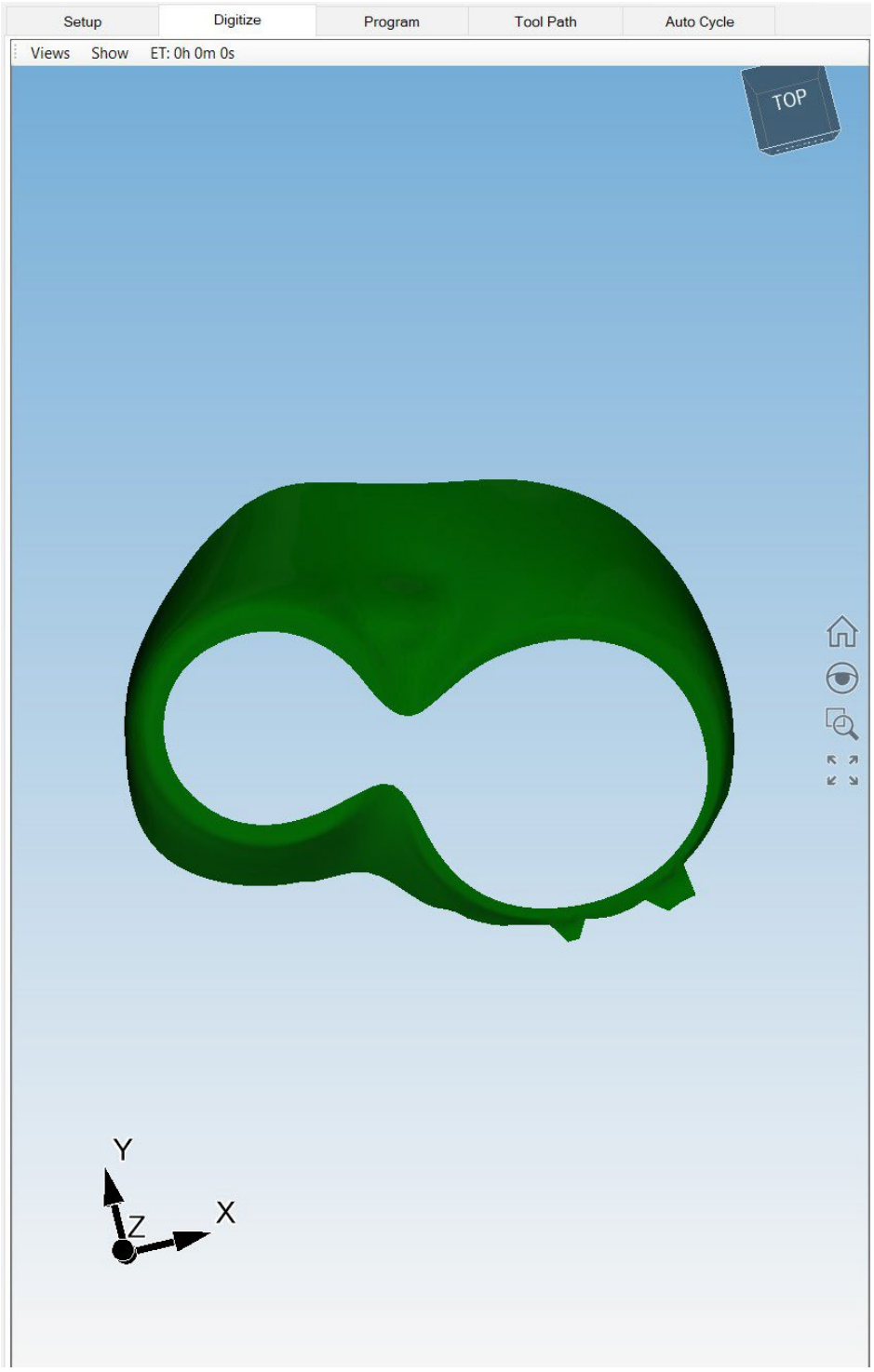


The next step is too active the automatic probing cycle. Make sure that the “Digitize” tab has been selected. Jog the probe so that the tip is positioned approximately 3 inches above the combustion chamber. When ready press the probe chamber button on the right of the screen to initiate the probing process.



The probe will start on the inside portion of the chamber and work its way up to the deck surface. The chamber will take some time to finish digitizing. If you choose to leave the machine unattended during this process, then make sure that the probe completes the first few radial lines and that the valves will stay secure during the whole process. Furthermore it is important that the spark plug holes have been plugged so that the probe tip will not error out inside of the hole.

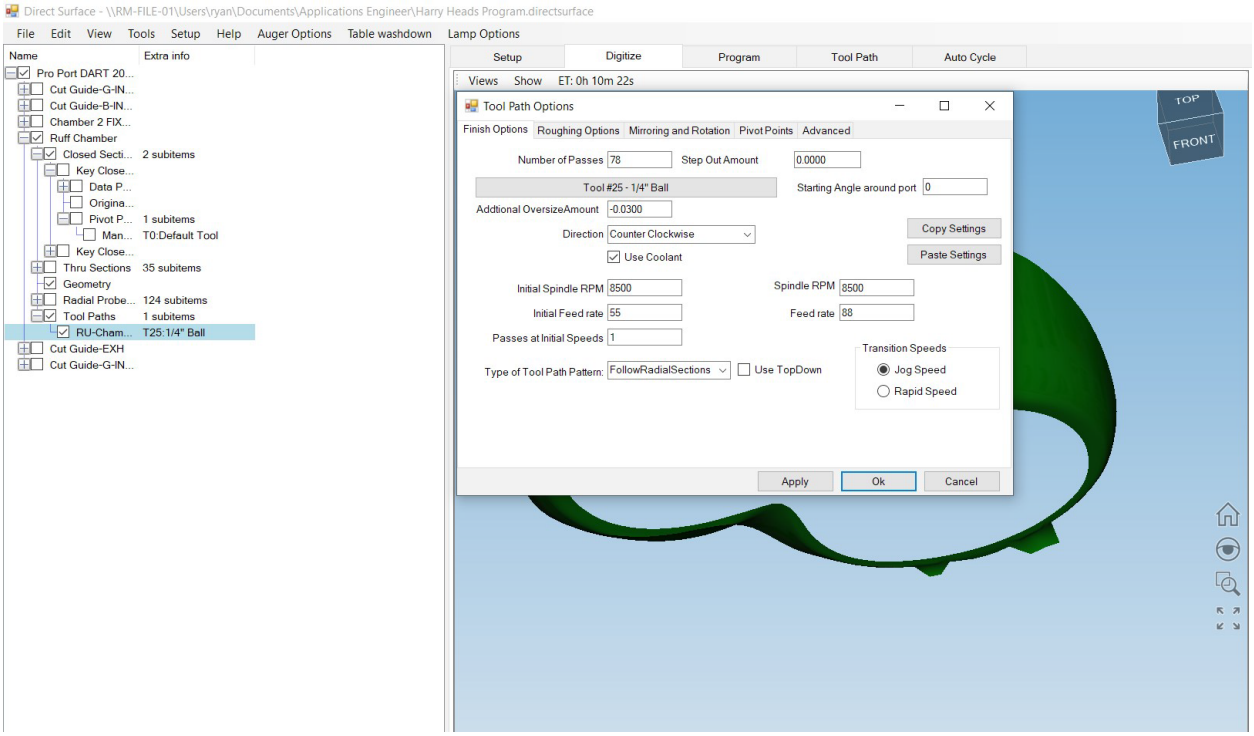
After probing completes, we can proceed to creating the surface. Similar to porting, we need to create this surface so that we can tell the machine where to create a toolpath. To create the surface, right click on the chamber name in the cylinder information tree and select “create surface”. To then view the surface, select “show solids” from within the show drop down tab in the 3D graphics window. When done the 3D graphics window should look something like the figure below.



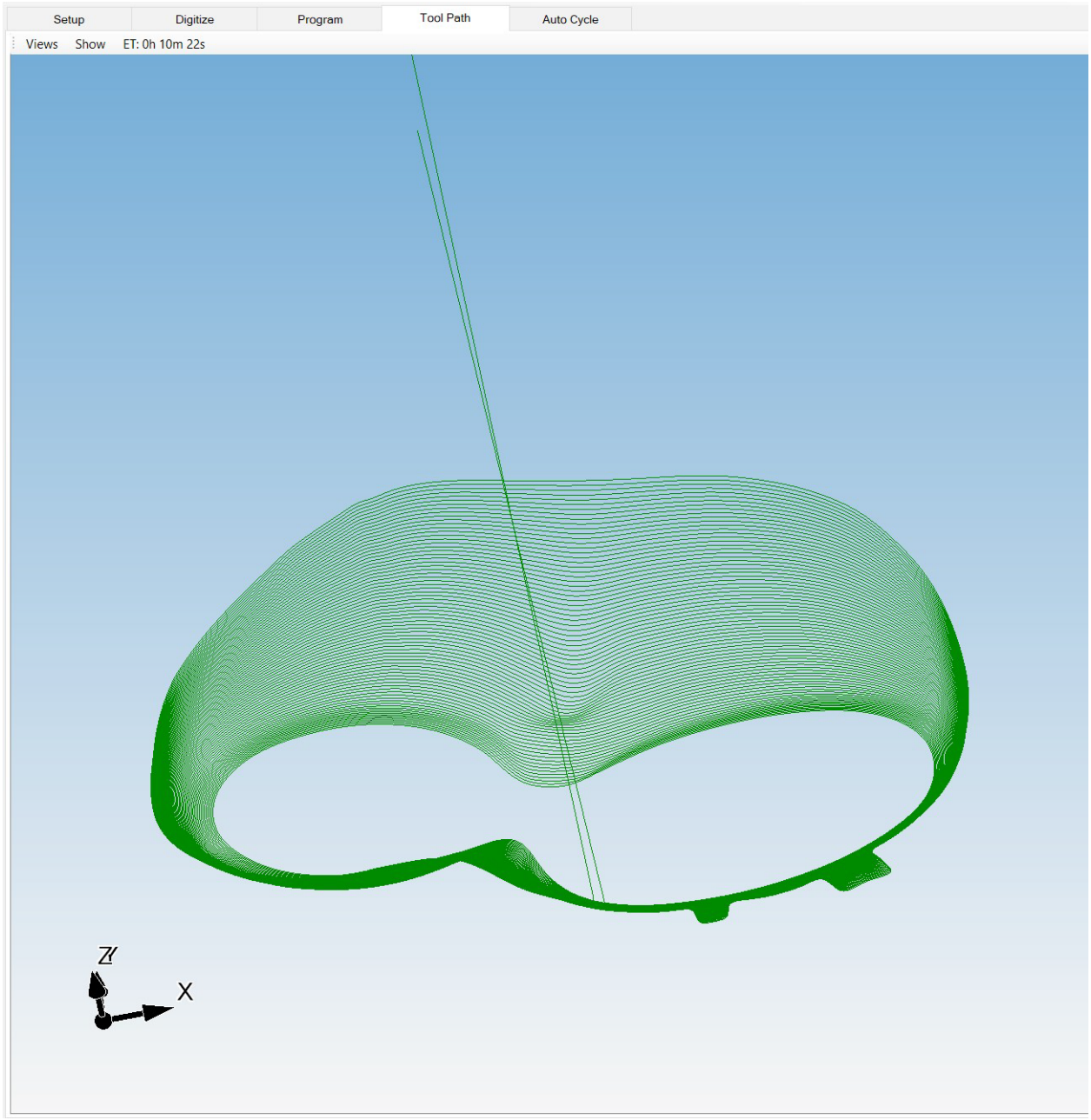
Creating a Tool Path

Once we have a surface created for the combustion chamber, it is then time to create the tool path to machine the combustion chamber surface. Under chamber in the cylinder open up “closed sections” and then open up the “key closed sections” which are two one for the top and the other for the bottom section. Under each of the “key closed sections” click and highlight the “pivot points” then press the delete key on the keyboard. It will ask if you are sure you want to delete it, chose “yes”.

To create a tool path for the chamber start by right clicking on ‘Tool Paths’ found under the chamber name as shown below. Double click on the Tool Path name that was created as shown below. A window will pop up which contains the machining parameters for the combustion chamber tool path. Enter your desired information here, an example is provided in the figure below.



Press “OK” when finished editing the machining parameters. To view the tool path after it generates, switch to the Tool Path tab. If done correctly the tool path should look something like the figure shown below.



The last step before running the tool path is to switch to the auto cycle tab and drag and drop the newley created tool path from the cylinder head information tree to the ports you wish to run the cycle on. After this is completed, make sure to turn the ports on by making them green as well as selecting the show tool path option in the 3D graphics window. Once complete the autocycle tab should look like the figure shown below. If everything looks correct then the operator may press the “cycle start” command to begin cutting the combustion chambers.

Mirroring

On some heads you will have to mirror the ports or chambers depending on the application that you are working on. If this is necessary then follow the instructions below to mirror the tool path to the correct location

Mirroring Tool Path

After you have all your tool paths created for the chamber and ports, with all feeds and speeds that you want to cut with, just simply right click on the “tool path” tab under either the port or chamber section and then click on “Duplicate All Tool paths as Mirrored Toolpath”

A window will come up and ask to input the offset to mirror around X, It’s the distance to the center of the chamber, for example in a SBC head the center of the chamber is 2.4 around the X axis. Which X is located off of the left dowel pin.

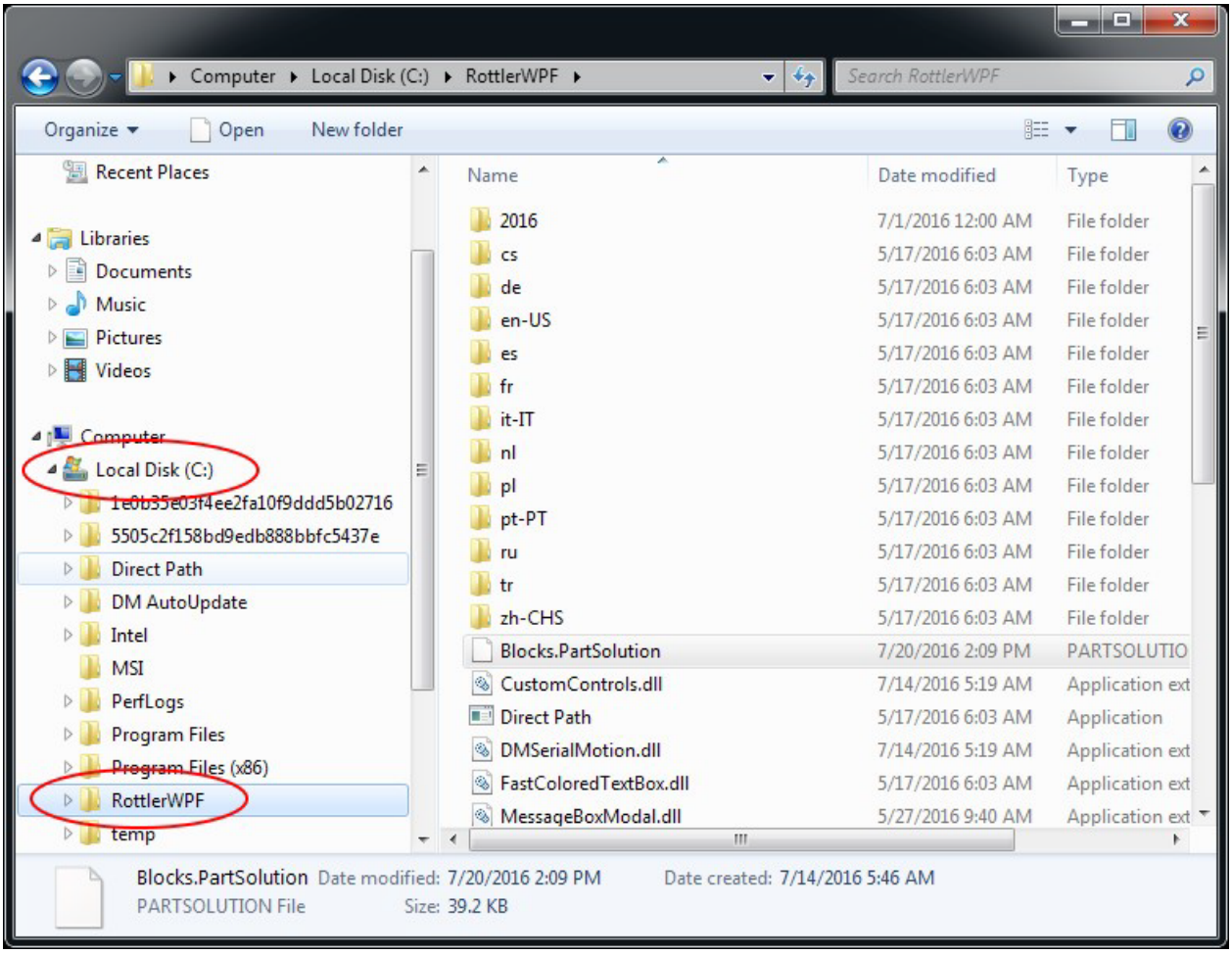
Do the same for the chamber tool path and then you will have the mirrored tool path created under your original tool path.

When setting up the tool paths on the desired cylinder, drag the mirrored tool path on the cylinders that are mirrored from the original digitized chamber and ports.

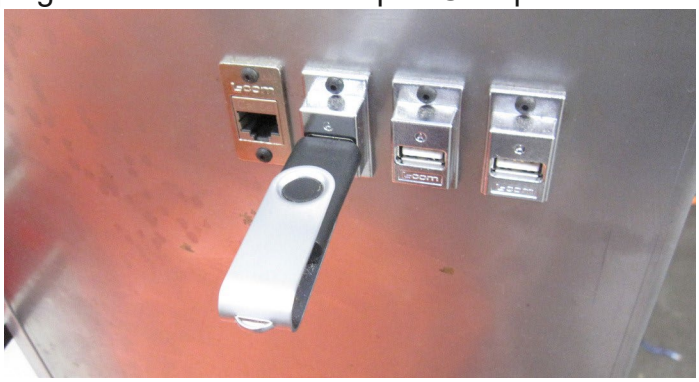
Backing Up and Restoring Block Profiles

This section will explain how to back up and restore the operator created block profiles for DM controlled machines for archival purposes or to transfer to a different machine.

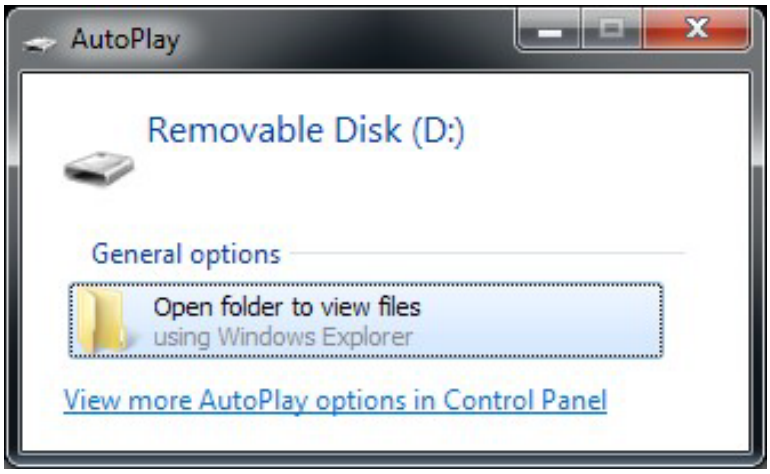
First step is to open your file bowser and locate the RottlerWPF file on the C disk drive.



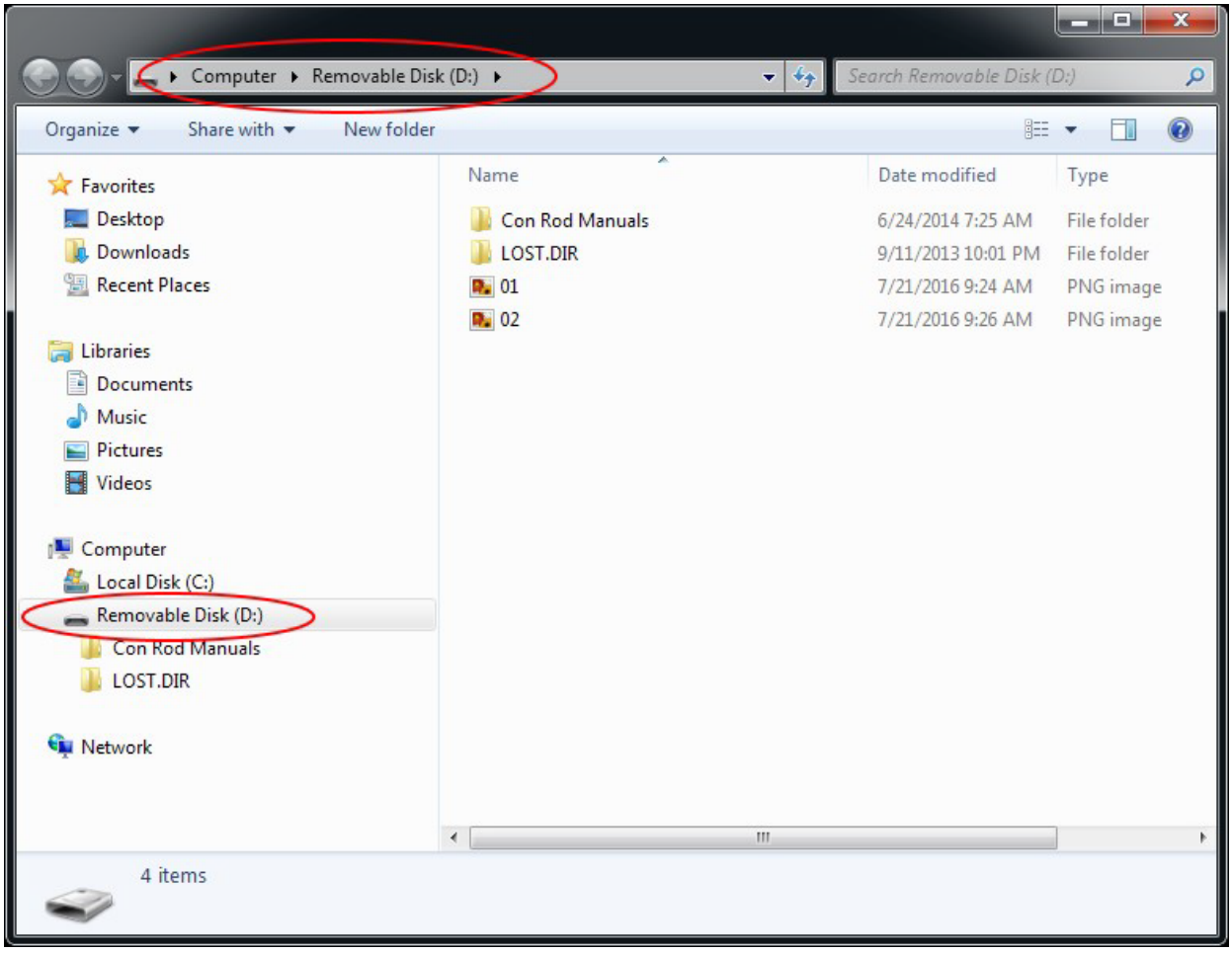
The next step is to plug in a flash drive to an open USB port



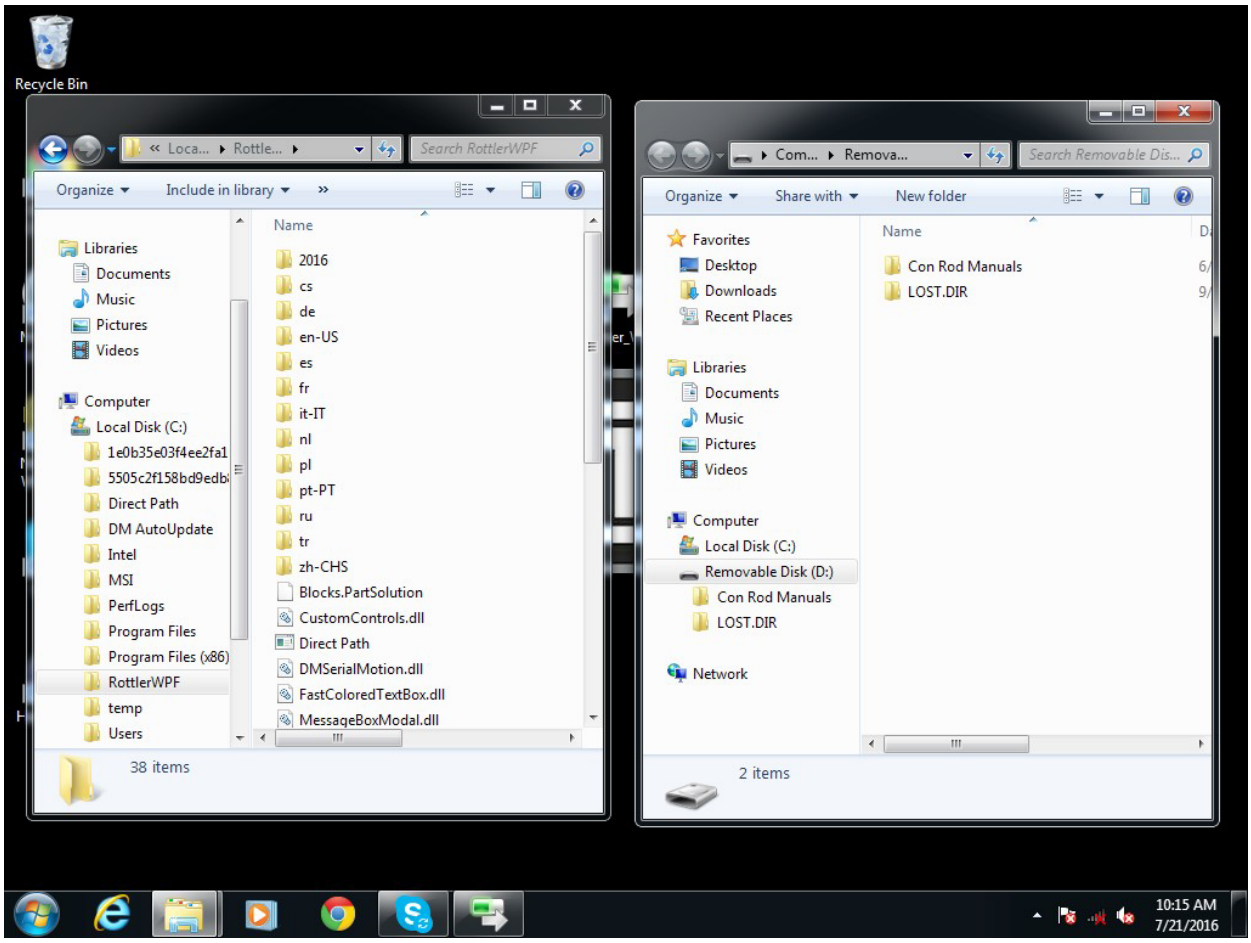
The following pop up box will appear on your screen.



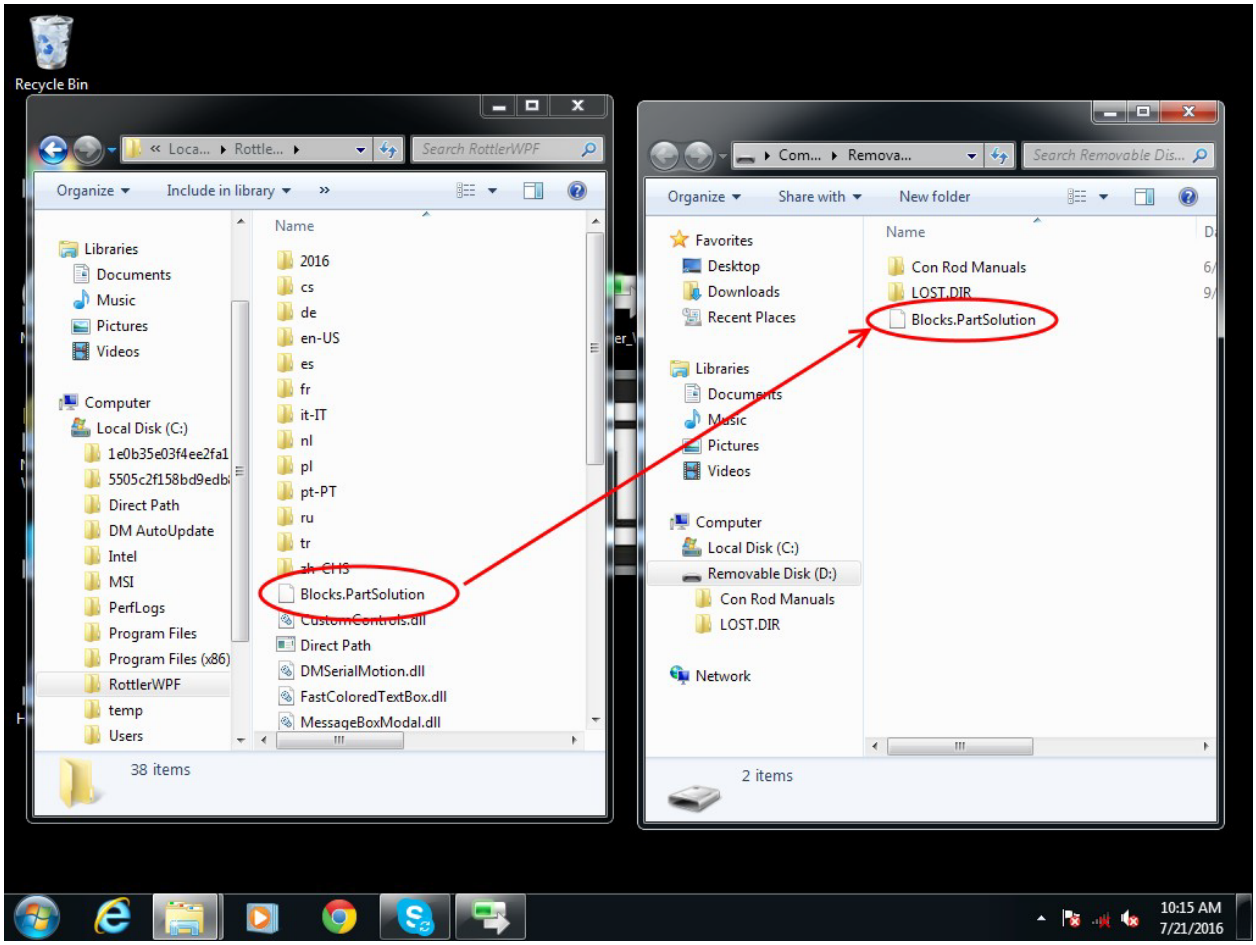
Click on the Open folder to view files option and the following screen will appear. This is the contents of the flash drive you just plugged in.



Next resize and arrange both file browsers so that they are side by side.



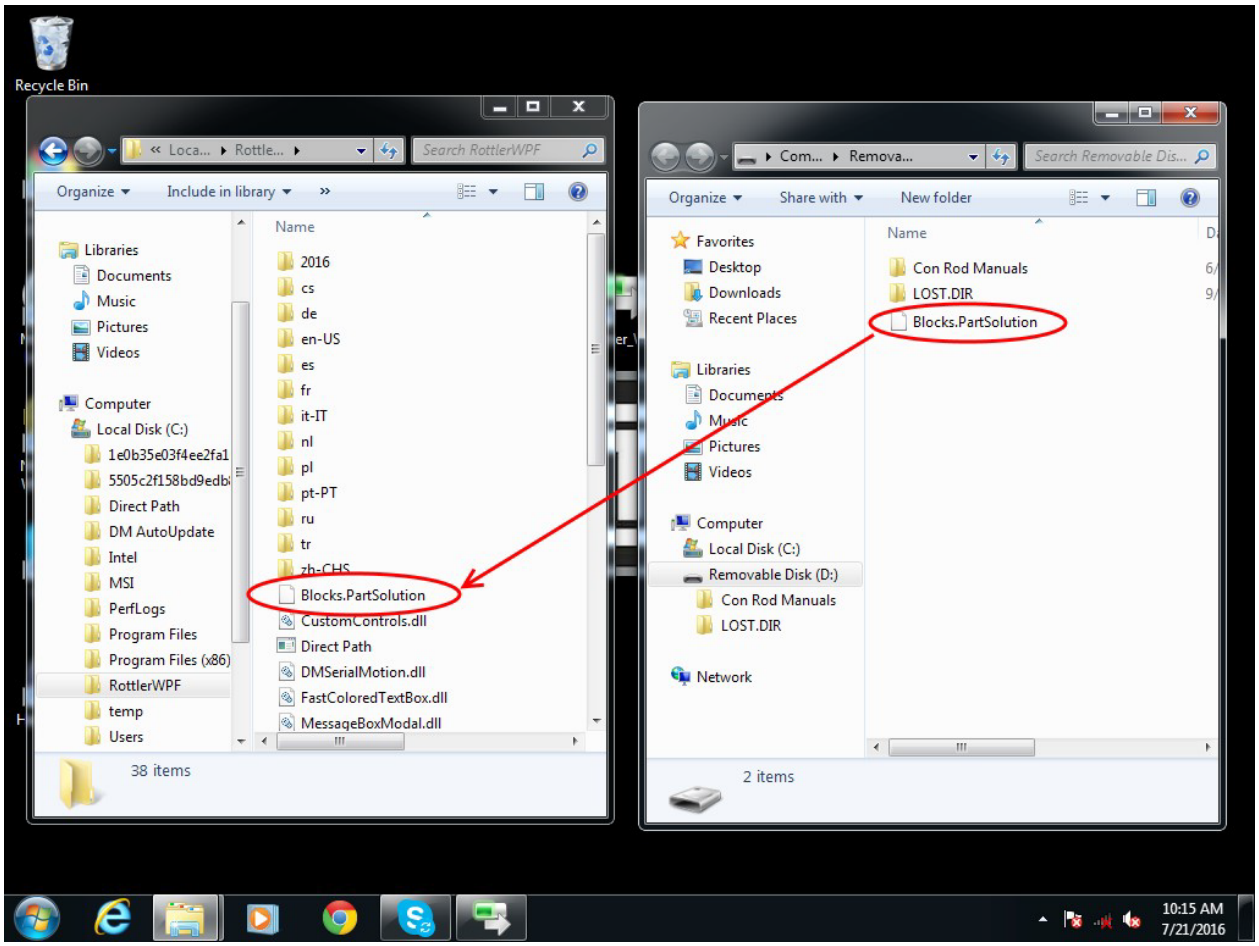
Block profiles are backed up each time the machine is run with the current profiles being shown in the RottlerWPF folder. All that needs to be done to back up the current profile is to simply drag it from the RottlerWPF folder to the flash drive folder. A copy of the file will be placed on the flash drive.



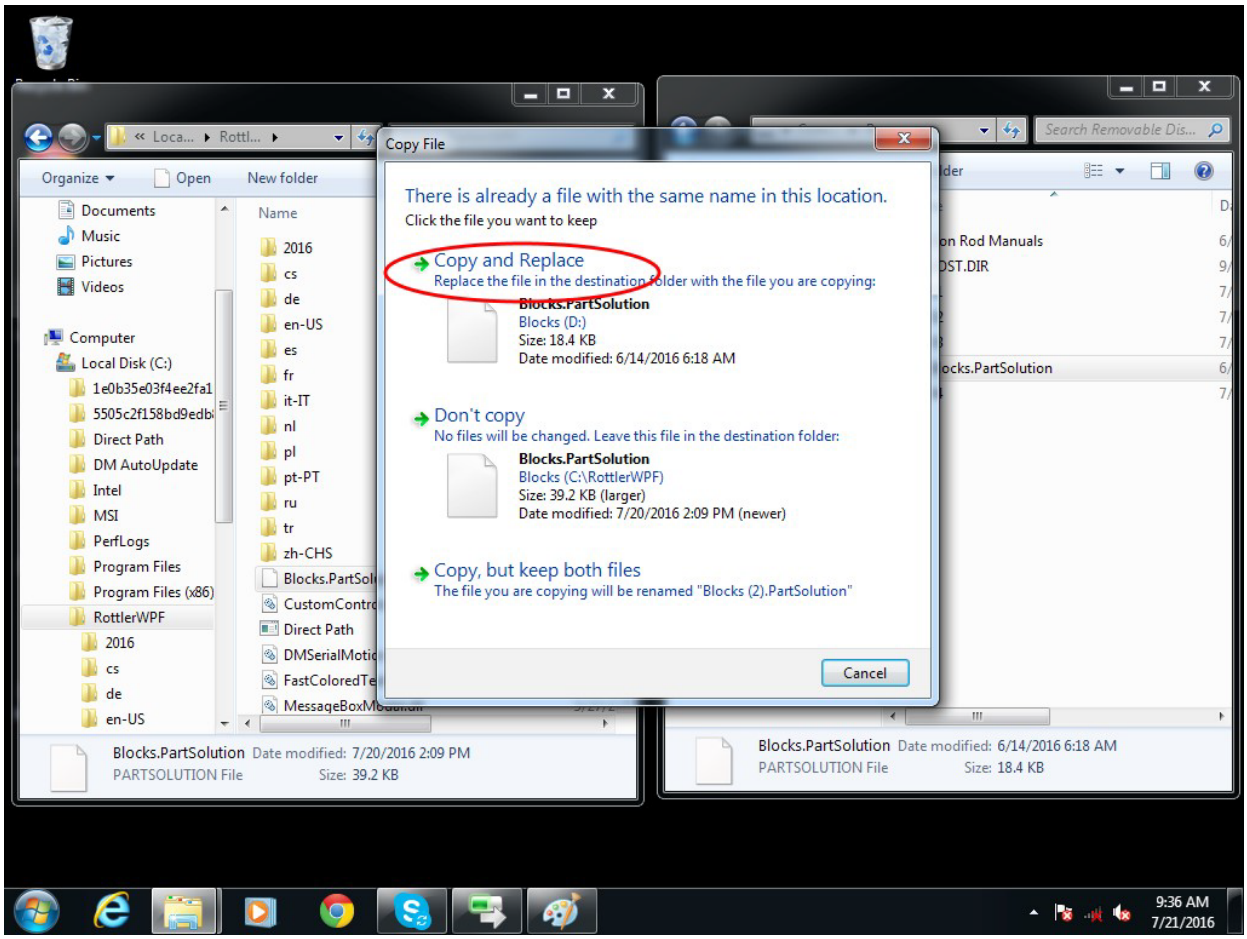
Backup is now complete. Close both file browser windows and remove the flash drive.

To restore or add block profiles go through the first 5 steps explained previously.

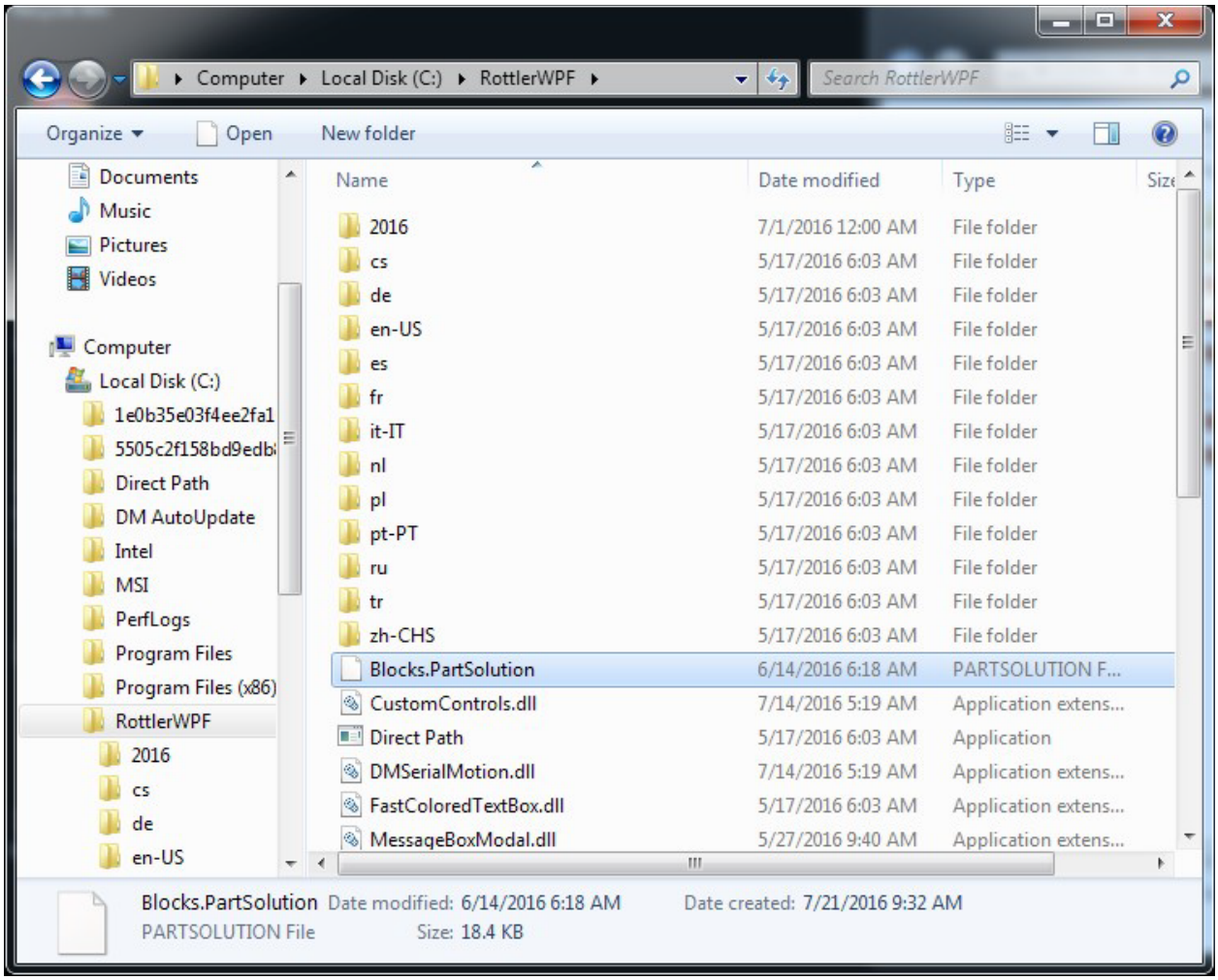
Highlight the block profiles file in the flash drive and drag it into the RottlerWPF folder on the local hard drive.



You will get a pop up window about there being a file of the same name in the destination folder. Click on the Copy and Replace option.



The archived block profiles will now be installed.



Close both browser windows and remove the flash drive. The restore process is now complete.

Using 3rd Party Tooling in Rottler Machines with CAT 40 Tooling.

When it is not convenient for the customer to order CAT 40 tooling from the factory or if the customer needs tooling that we don't stock, they may purchase tooling from 3rd party vendors such as MSC Metalworking.

Rottler uses a Parlec - A Style, CAT40 Taper, 5/8-11 Thread, 45 Degree Angle Radius, Standard Retention Knob with the following specifications: 1.68 Inch Overall Length, 0.281 Inch Coolant Hole Diameter, 0.74 Inch Knob Diameter, 0.12 Inch Flange Thickness, 0.64 Inch Knob to Flange Length, 0.635 Inch Pilot Diameter, Through Coolant.

The metric equivalent is a Parlec - A Style, BT40 Taper, M16 x 2
Thread, 45° Angle Radius, Standard Retention Knob 1.65 Inch Overall Length, 0.281 Inch Coolant Hole Diameter, 0.74 Inch Knob Diameter, 0.12 Inch Flange Thickness, 0.64 Inch Knob to Flange Length, 0.669 Inch Pilot Diameter, Through Coolant.