Park Industries®

VEKTOR

Operation and Maintenance Manual

VK10000

Issue 03 Build 02

Park Industries®

Precision. Performance. Peace of Mind.

VK10002C

Copyright

© 2011, Park Industries
All Rights Reserved
Printed in U.S.A.

Revision History

Issue	Build	Date	Reason for Change
2	1	June 2011	Changes to electric cabinet, coolant screens
3	1	June 2012	Desiccant filter added to machine. SN: 159150
3	2	August 2014	Change to auto lubricator.

Trademark Information

Park Industries is a registered trademark of Park Industries.
VEKTOR is a registered trademark of Park Industries.
Precision, Performance, Peace of Mind. is a registered trademark of Park Industries.

Disclaimer of Liability

The information in this document is subject to change without notice. Park Industries, makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Park Industries, shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

Please do not attempt to operate or repair this equipment without adequate training.

This Park Industries product has been built to the high standards of Park Industries. Any use, operation, or repair in contravention of this document is at your own risk. By acceptance of this equipment you hereby assume all liability consequent to your use or misuse. Park Industries assumes no liability for incidental, special or consequential damage of any kind. Equipment specification, applications and options are subject to change at the sole discretion of Park Industries, without notice.

Park Industries
PO Box 188
St. Cloud, MN 56302-0188
Telephone: 320-251-5077 800-328-2309
Fax: 320-251-8126
Website: www.parkindustries.com
E-mail: park@parkindustries.com

Safety Precautions

When operating or working near this equipment certain personal safety precautions are required:

<u>EYE PROTECTION:</u> While processing your material on this machine it is possible for material or tooling chips to become airborne. It is therefore required that anyone in the vicinity of the machine wear Safety Glasses or Goggles (Z87 or Z87.1) at all times.

<u>EAR PROTECTION:</u> Prolonged exposure to stone working equipment, while it is processing material, can lead to a loss of hearing. It is therefore required that anyone in the vicinity of the machine, while it is in operation, wear some form of hearing protection (ear plugs, ear muffs, etc with an NRR of 20 or higher).

<u>FOOT PROTECTION:</u> Due to the weight of the material that is being processed, especially during loading and unloading, it is also recommended that Steel Toe Safety Shoes (ASTM F2412-05 or F2413-05) be worn while using this machine.

Safety Precautions

It is the customer's responsibility to make sure all machine operators are trained and well-versed in both the operation of the machine as well as all safety processes and procedures.

PERSONAL SAFTEY

This machine, manufactured by Park Industries, is designed for cutting and processing natural stone. Only trained personnel should operate this machine, and only for the purpose for which this machine has been designed. **Modification of the machine, misuse or failure to take proper safety precautions may result in damage to the machine, personal injury or death.**

PERSONNEL QUALIFICATION

This Operation Guide is intended for use by personnel experienced in the use of automated equipment. Operators and maintenance personnel should be experienced in the use of electro-mechanical, hydraulic assemblies and components, and familiar with accepted industrial maintenance practices. All operating personnel must be **trained by a Park Industries employee** in the use of this machine and read this Operation Guide to insure the safe and proper operation of the machine.

SAFETY COMPLIANCE LIABILITY

State and local laws related to safety in the workplace apply primarily to the responsibilities of the employer, and not the equipment manufacturer. Park Industries agrees to cooperate with the buyer in finding feasible answers to compliance problems. However, because Park Industries has little control over factors which may significantly affect the environment in which this equipment is installed, Park Industries does not warrant this equipment to be in compliance with OSHA, 29 CFR 1910 or state or local laws or regulations. It is the buyer's responsibility to assure compliance with these laws and regulations at the point of installation. For information on OSHA compliance see: http://www.osha.gov/

GENERAL PRECAUTIONS

Must be followed when operating the machine:

- Never modify the machine or removal or attempt to defeat any of its safety features.
- Be sure all personnel are clear of the machine before starting.
- A shock hazard may exist if equipment is not properly grounded.
- Stop the machinery before oiling, wiping, repairing or performing any service that requires the removal of guards.
- Electrical Enclosures - Dangerous voltages are present within the electrical enclosures. DO NOT operate this equipment with electrical enclosure covers open or removed.
- Protective Clothing - it is the responsibility of the user to use the proper protective clothing, including wearing eye protection.
- **Never allow untrained personnel to operate the machine.**

This page intentionally left blank.

Table of Contents

Contents Page

Preface

Safety Precautions iv
About this Manual xi

Section 1
Description and Application

Introduction 1-1
Definition of Common Terms 1-2
Component Identification
- Complete Machine 1-3
- Control Station 1-4
- Pendent and Safety Switch 1-5
- Air 1-6
- Water 1-7
- Gantry (Y Axis) 1-9
- Cross Travel (X Axis) 1-11
- Rise & Fall (Z Axis) 1-13
- Lubrication 1-18
- Electrical 1-19
- Spindle Cover 1-26
- Tool Holder and Tool Height Measuring Assembly 1-27
- Air Conditioner Unit 1-28

Machine Specifications 1-30
Connectivity Requirements 1-31

Section 2
MasterCAM® Setup

General 2-1
CAD Principles and Operations 2-2
CAD Screens and Definitions 2-3
Help Screens 2-4
Customize Screen Configuration 2-5
Customize Toolbars 2-9
Create Tool Library 2-12
Creating Default Tools 2-18
Changing Default Posting Settings 2-35
Setting the Default Machine 2-39

Table of Contents (cont'd)

Contents Page

Section 3
MasterCAM® Operation

General 3-1
CAD Principles and Operations 3-2
CAD Screens and Definitions 3-3
Starting MasterCAM® 3-4
Managing Tool Bars and Operations Manager Window 3-5
Point of Origin 3-6
Swept 2D and Contour Example
- Introduction 3-7
- 3D Translate 3.8
- Contour 1 Create Toolpath Line 3-13
- Contour 1 Defining Across and Along Contour 3-17
- Contour 1 Extending the Toolpath 3-24
- Contour 2 Copy Contour 1 3-34
- Contour 2 Editing Toolpaths 3-35
- Display Stock 3-39
- Verify Operation 3-42

Pocket and Facing Example
- Introduction 3-45
- Stock Setup 3.47
- Create Facing Toolpath 3-49
- Creating First Pocket Toolpath 3-56
- Creating Second Toolpath 3-62
- Verify Operation 3-66

Post 3-68
Nesting Parts 3-71

Section 4
Machine Operation

General 4-1
Powering On the VEKTOR 4-3
Spindle Warm Up 4-5
Powering Off the VEKTOR 4-6
Part Processing
- Loading Material 4-8
- Loading Programs 4-9
- Setting Profile Offsets 4-12
- Running Part Programs 4-16
- Restart in Cut - Profile Mode 4-19

Table of Contents (cont'd)

Section 4 (cont'd)
Machine Operation

How to Calibrate the Setup Tool Length 4-24
How to Calculate Tool Length 4-28
Storing Tool Length 4-31
Using G-Codes to Change a Tool 4-33
Using G-Codes to Change a Tool's Position 4-35
Manually Changing a Tool 4-39
Tool Rack Setup 4-40
Verifying Alarms 4-47
Park Industries Generated Alarms 4-50
Input/Output Screens 4-51

Section 5
Maintenance

General 5-1
Lubrication Warnings 5-4
Spindle and Retention Know Warning 5-5
Routine Lubrication Schedule 5-6
Maintenance Tasks 5-7
Maintenance Procedures 5-31
Maintenance Log 5-41
Maintenance Log Masters 5-46

Section 6
General Information

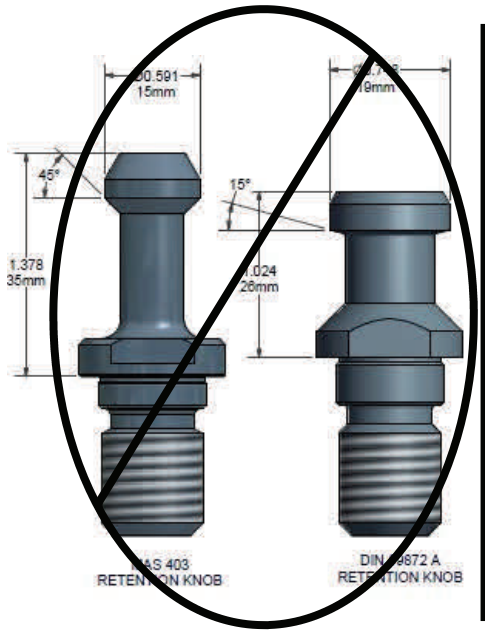
General 6-1
Parts Order Form 6-2
Parts by Phone or Online 6-3
Notes 6-4
Quick Reference Sheet.. 6-11
Technical Drawings 6-12

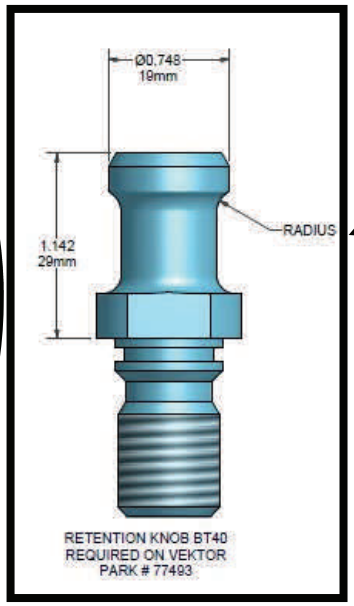
Warning

Your VEKTOR has a high speed spindle. To prolong spindle life, purchase and use only balanced tools and tool holders.

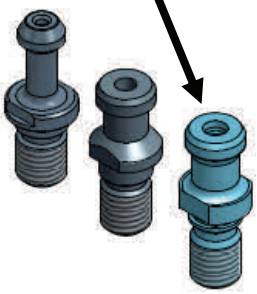
Warning

Damage to the spindle will occur if you don't use Park Industries part # 77493 retention knob with the BT40 tool holder on the VEKTOR.





This is the correct retention knob for the BT40 tool holder.



About This Manual

This user manual describes Park Industries VEKTOR CNC Stone Profiler System. This manual also provides instructions necessary to operate and maintain the equipment.

<u>WARNING</u>: If you have not read the Operation and Maintenance Manual and feel you were not properly trained DO NOT ATTEMPT TO OPERATE THIS MACHINE. Serious injury or death may result.

It is the customer's responsibility to make sure all machine operators are trained and well-versed in both the operation of the machine as well as all safety processes and procedures.

Section 1: Description and Application

Contents Page

Introduction 1-1
Definition of Common Terms 1-2
Component Identification
- Complete Machine 1-3
- Control Station 1-4
- Pendent and Safety Switch 1-5
- Air 1-6
- Water 1-7
- Gantry (Y Axis) 1-9
- Cross Travel (X Axis) 1-11
- Rise & Fall (Z Axis) 1-13
- Lubrication 1-18
- Electrical 1-19
- Spindle Cover 1-26
- Tool Holder and Tool Height Measuring Assembly 1-27
- Air Conditioner Unit 1-28

Machine Specifications 1-30
Connectivity Requirements 1-31

Introduction

This section describes the VEKTOR CNC Stone Profiler, includes pictures of complete mechanical components.

Common Terms used throughout the manual

Alarms - Information about an illegal program or system information

CAD - Computer Aided Drafting

CAM - Computer Aided Manufacturing

CNC - Computer Numeric Control

Control Station - Operator station where the machine is ran from

Cutting Bed - Area where slabs are put onto to be cut and processed

Feed Rate Override - Allows full operator control over the speed of any axis during Rapid and Cutting moves

Material - Limestone, Sandstone, Bluestone, Granite and Marble Slabs

Pendant - Manual control unit connected to the machine via a cable used for moving the machine around and setting offsets

Program Zero - Starting location of the part

Spindle Assembly - Contains the spindle

Spindle Speed Override - Allow full operator control over the speed of the spindle

X Axis - Cross travel motion that moves the spindle, left and right on the bridge

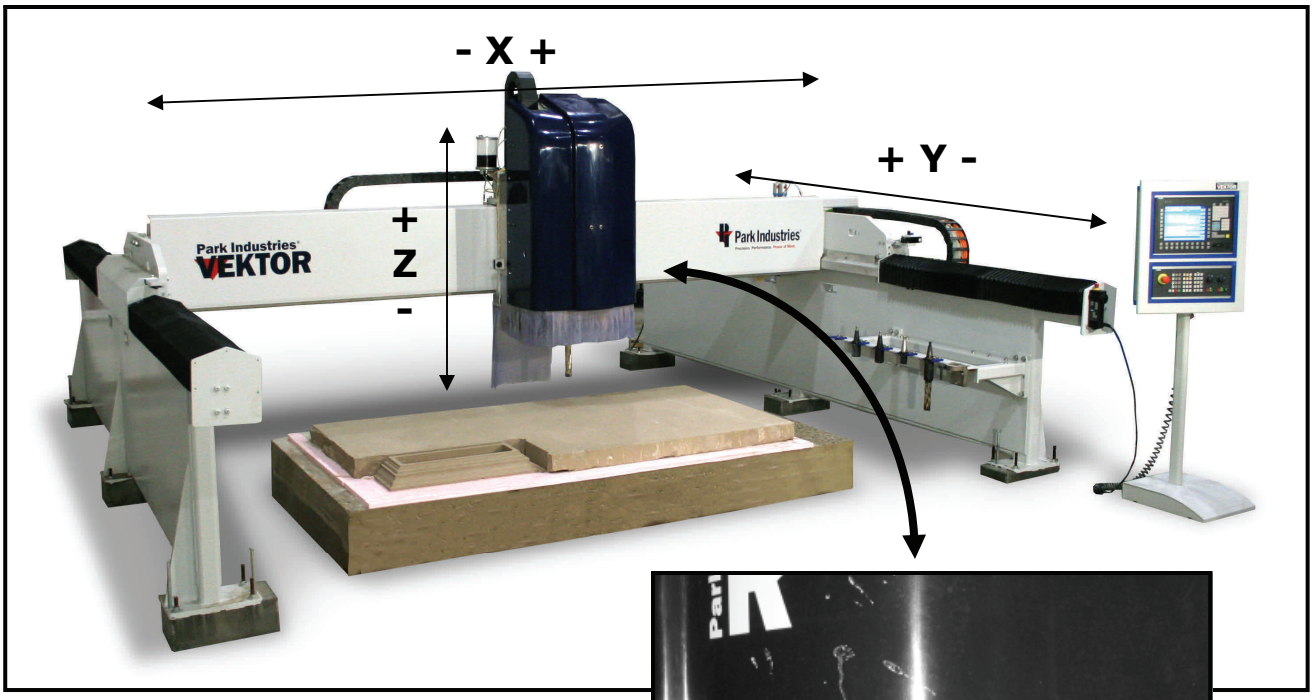
Y Axis - Gantry travel motion that moves the spindle, toward and away on the bridge rails

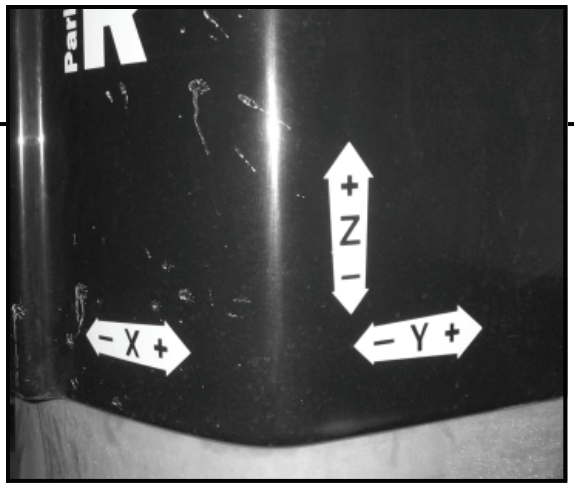
Z Axis - Up and Down motion that moves the spindle on the bridge assembly

Component Identification

Complete Machine

This machine picture shows the complete final assembly of the X, Y, and Z axis.



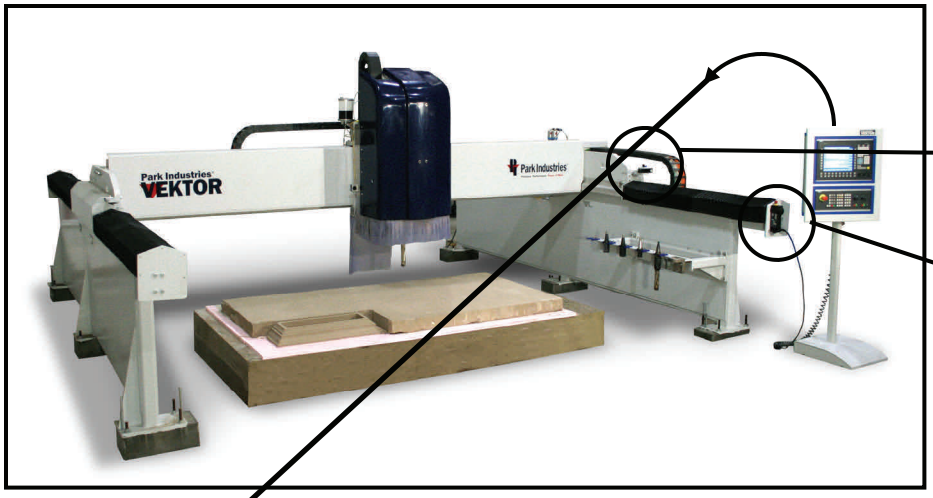


Axis Labels on Spindle Cover

Component Identification (Cont'd)

Control Station

The Control Station is a part of the machine that an operator uses to manually move the machine, process part programs and be able to run the machine.



Control Station

Operator Panel

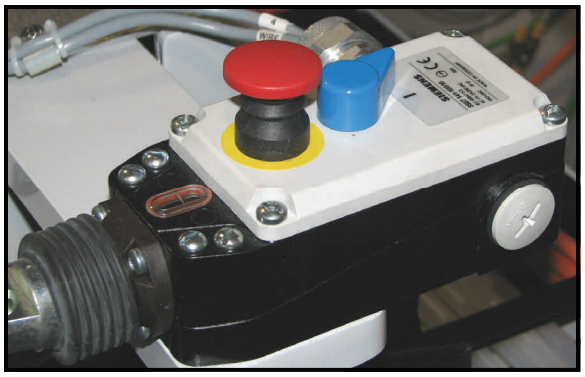
Control Panel

Component Identification (Cont'd)

Pendant and Safety Switch

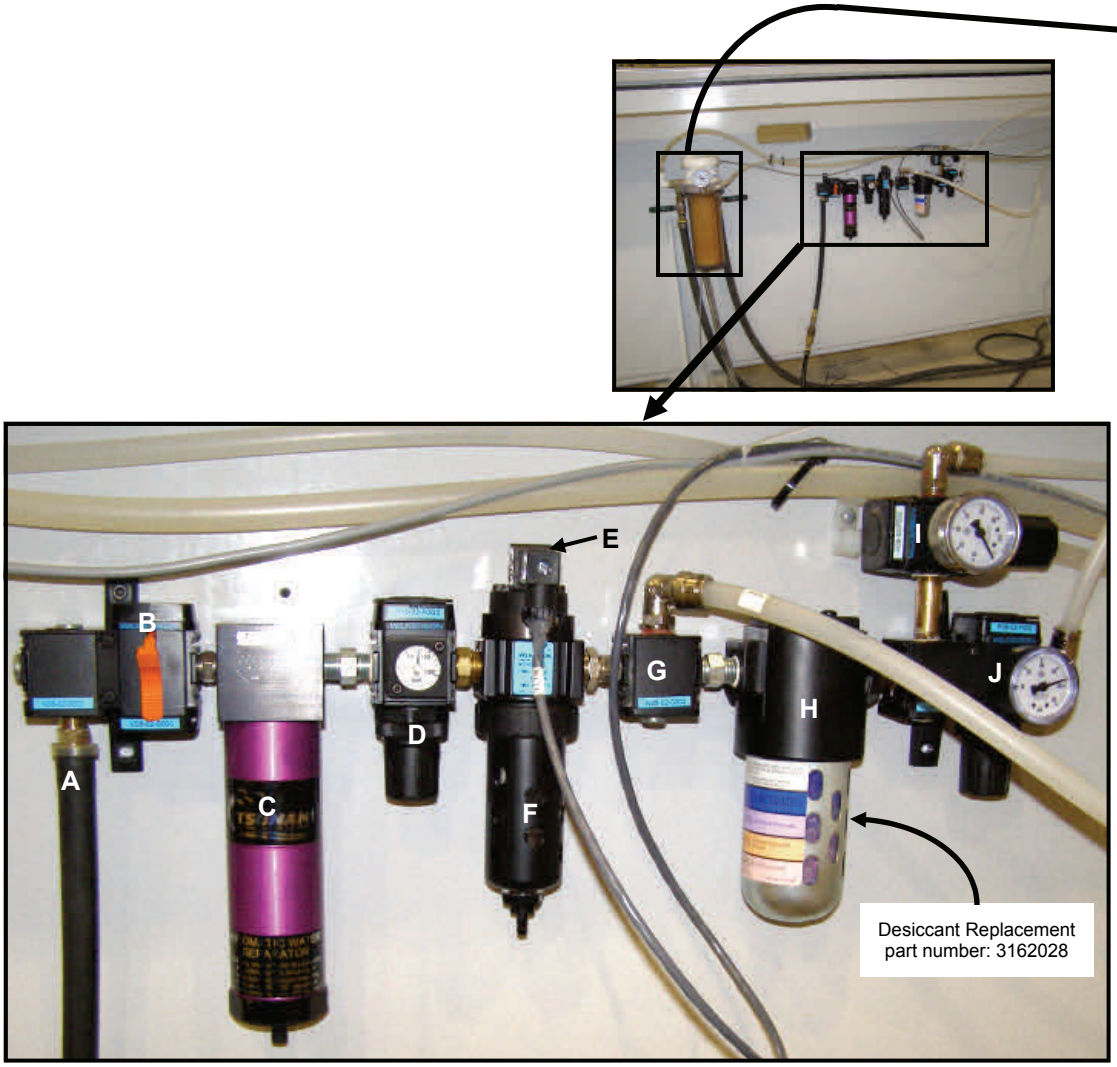
The Pendant is a part of the machine that works in conjunction with the Control Station. An operator can also use this device to manually move the machine and also set the Program Zero when processing part programs.

Safety Pull Cord Switch



Component Identification

Air Components

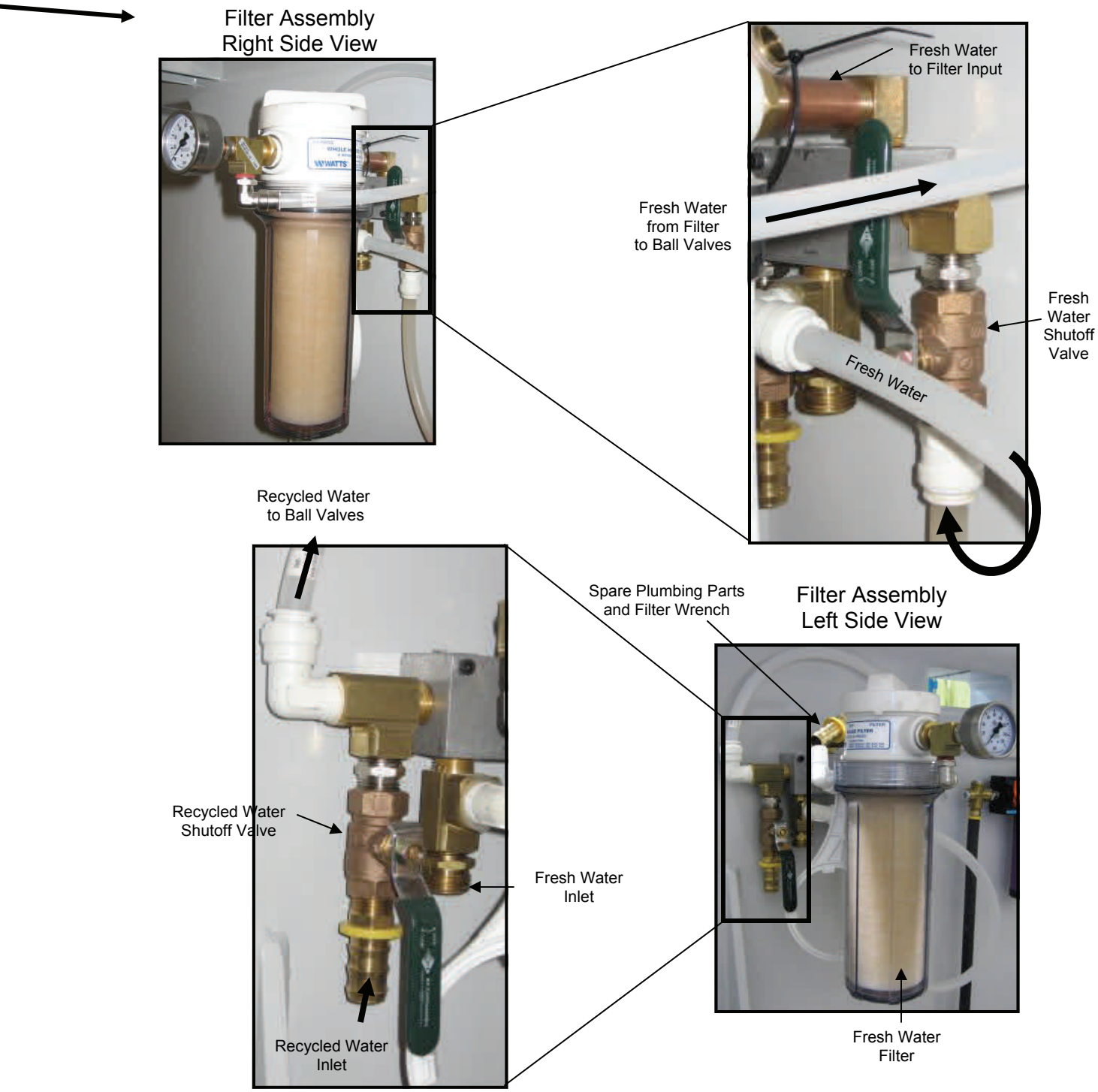


A	Site Air Supply
B	Air Shutoff
C	Tsunami air dryer
D	Regulator and pressure gauge for main air supply (set to 90 PSI)
E	Electronic Differential Pressure Indicator
F	Coalescing Filter
G	Diverter
H	Desiccant Air Dryer
I	Regulator and pressure gauge for spindle air cooling supply (set at 5 PSI)
J	Regulator and pressure gauge for spindle air purge (set to supply 30 PSI at spindle)

Component Identification

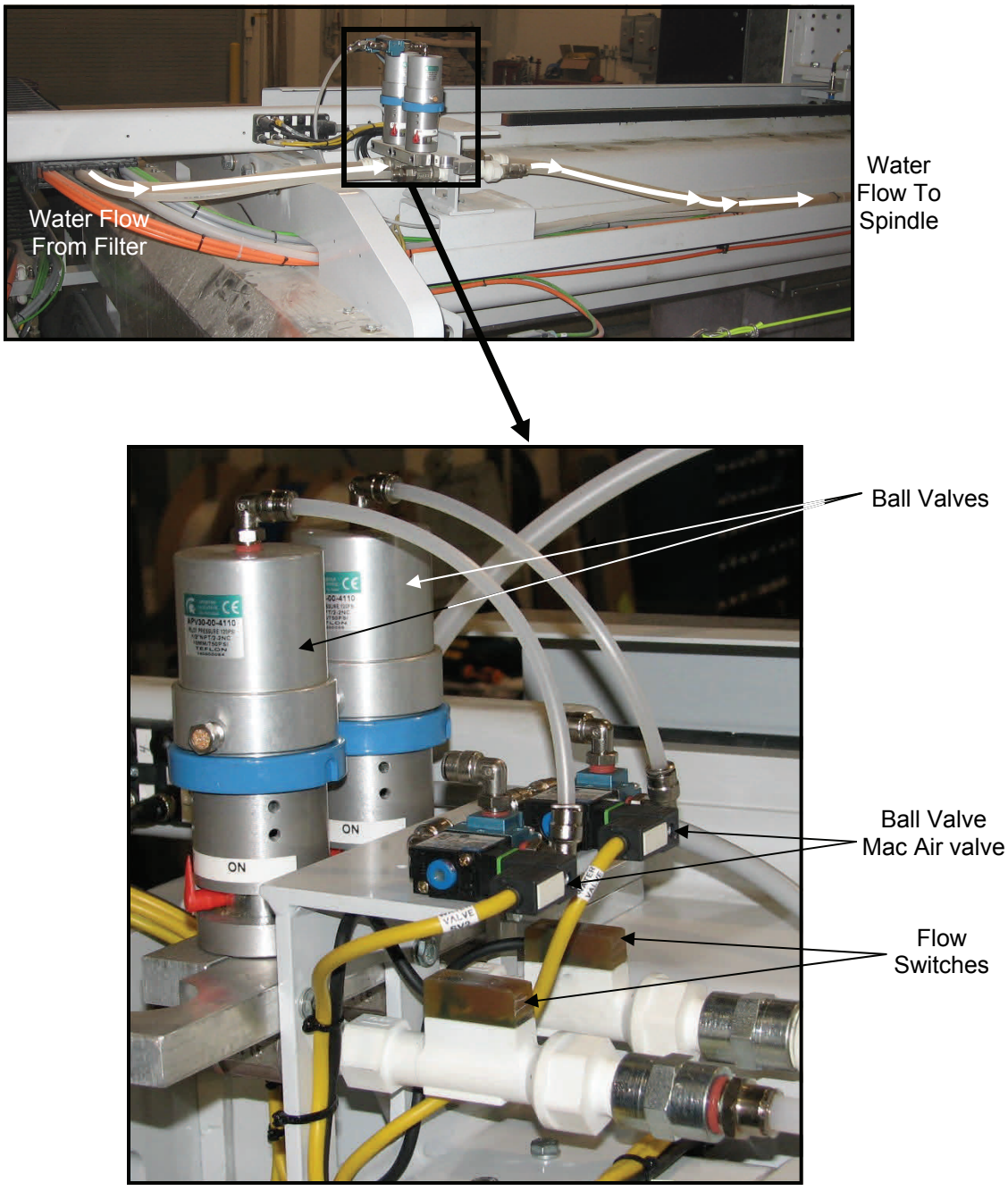
Water System Components

Note: The VEKTOR filter is for emergency situations only. It is NOT intended to totally filter site water.



Component Identification

Water System Components (Cont'd)



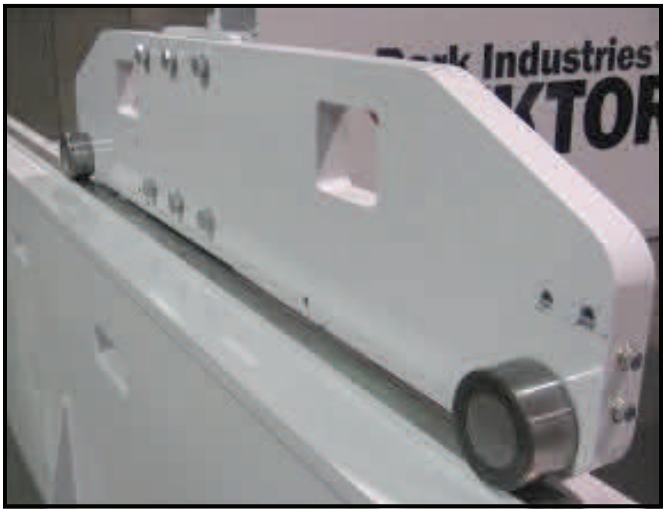
Component Identification

Gantry (Y Axis) Components

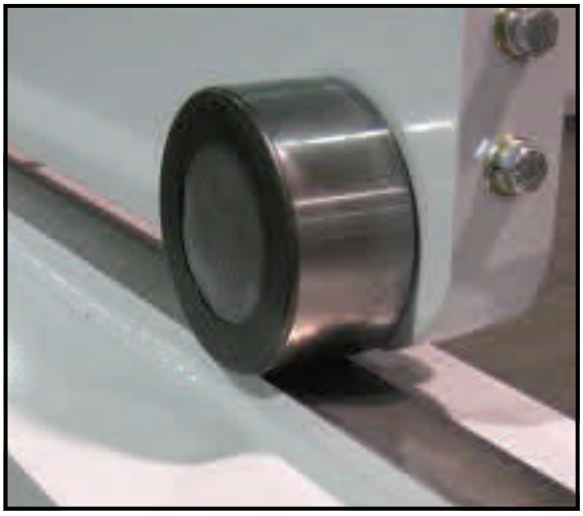
NOTE

Some images may show machines with guarding removed for visual clarity. Never operate any machinery without safety equipment in place and in working order.

Away Side End Truck



Away Side Flat Rail and Roller



Toward Side End Truck

Toward Side Round Rail and Roller

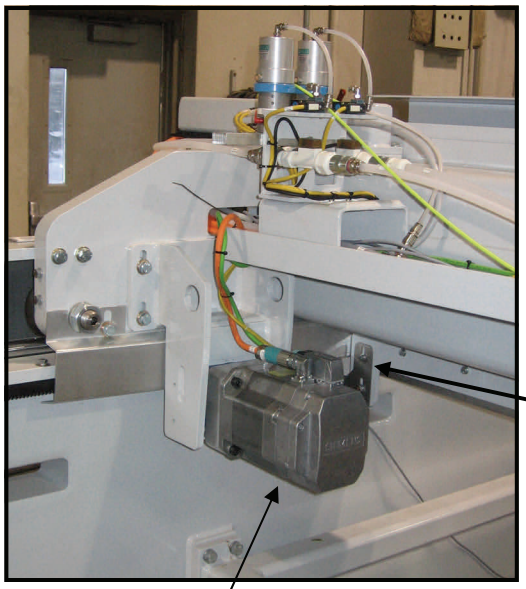
Component Identification

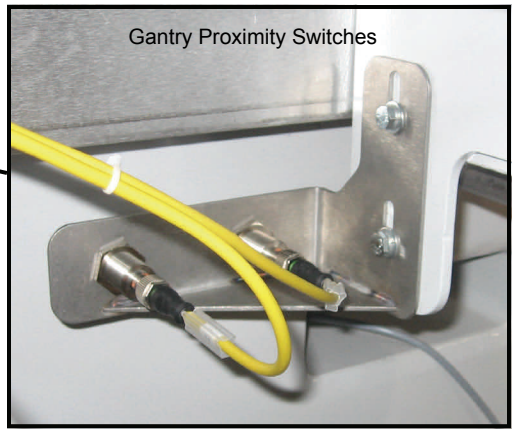
Gantry (Y Axis) Components (cont'd)

NOTE

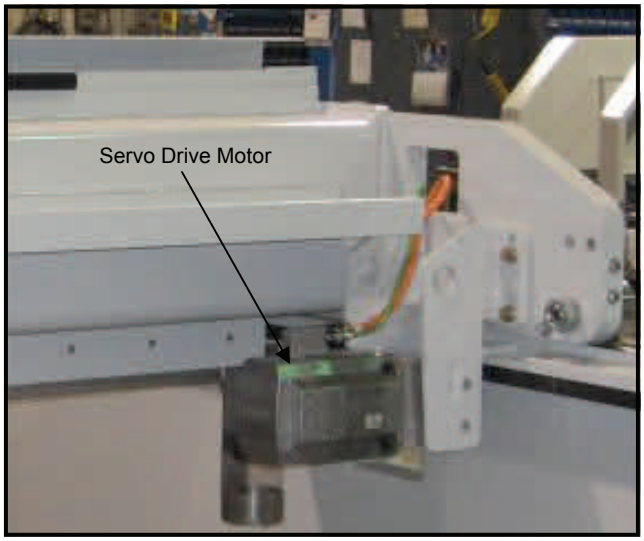
Some images may show machines with guarding removed for visual clarity. Never operate any machinery without safety equipment in place and in working order.

Toward Side Gantry Drive Area

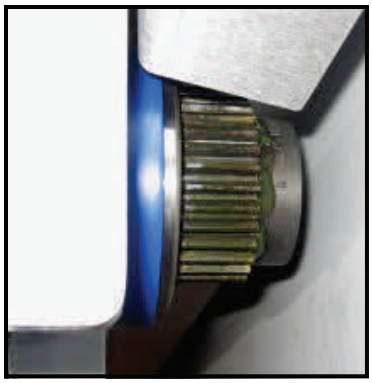




Away Side Gantry Drive Area



Close Up View of a Drive Motor Gear



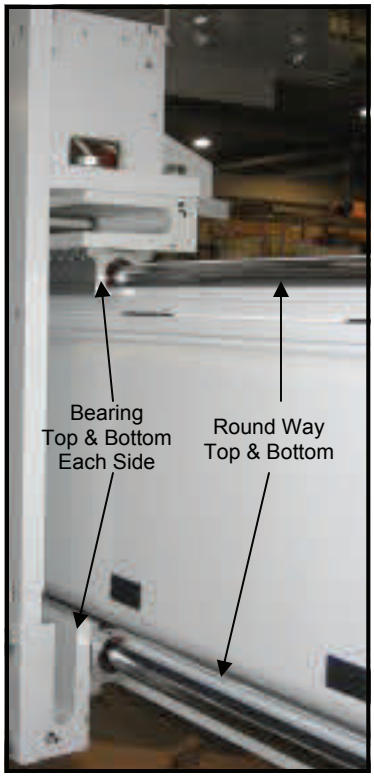
Component Identification

Cross Travel (X Axis) Components

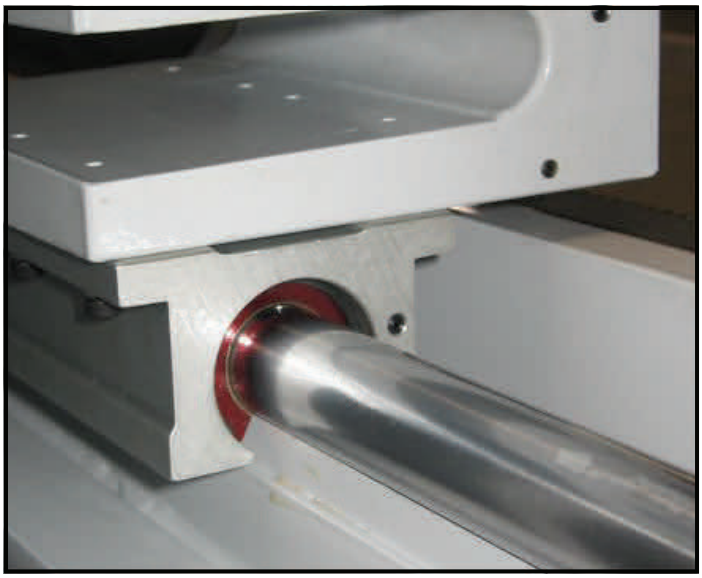
NOTE

Some images may show machines with guarding removed for visual clarity. Never operate any machinery without safety equipment in place and in working order.

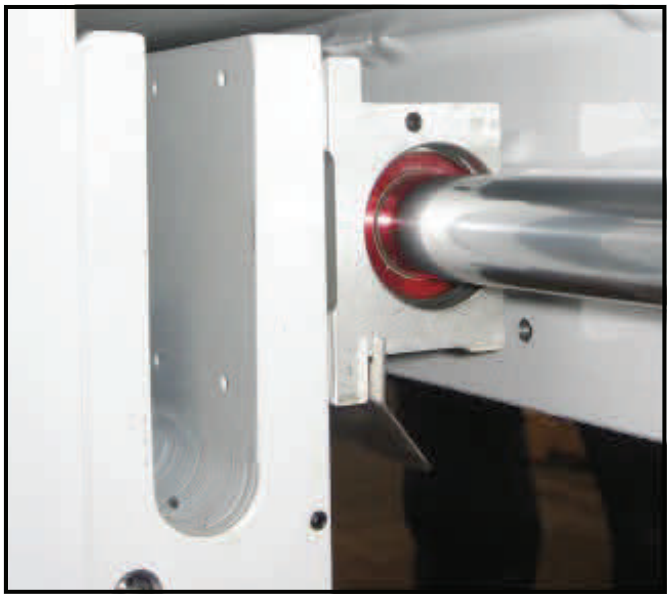
Cross Travel
Bearings and Rails



Close Up View of a Upper Bearing and Rail



Close Up View of a Lower Bearing and Rail

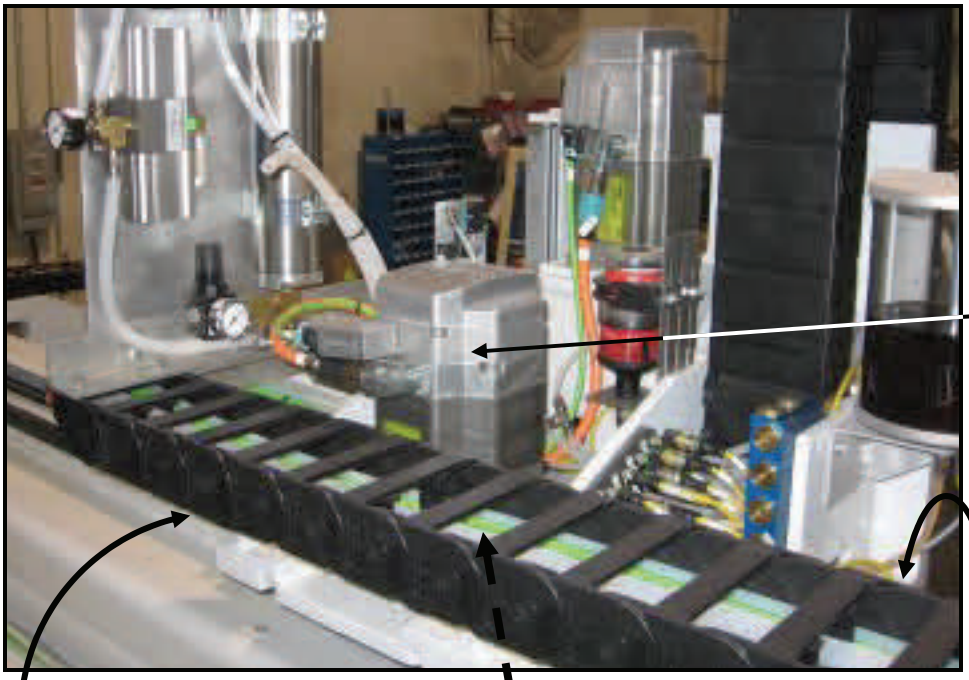


Component Identification

Cross Travel (X Axis) Components (cont'd)

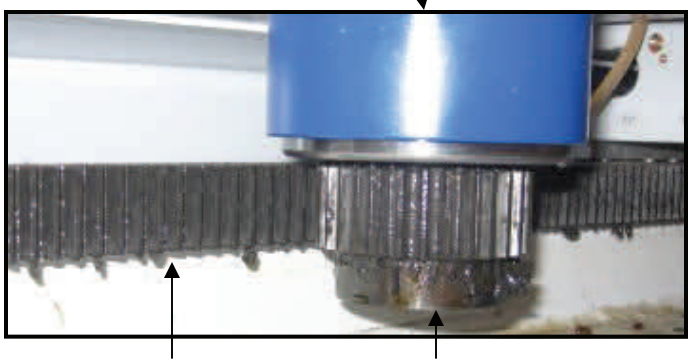
NOTE

Some images may show machines with guarding removed for visual clarity. Never operate any machinery without safety equipment in place and in working order.



X Axis Servo Drive Motor

X- Proximity Switch



X Axis Rack

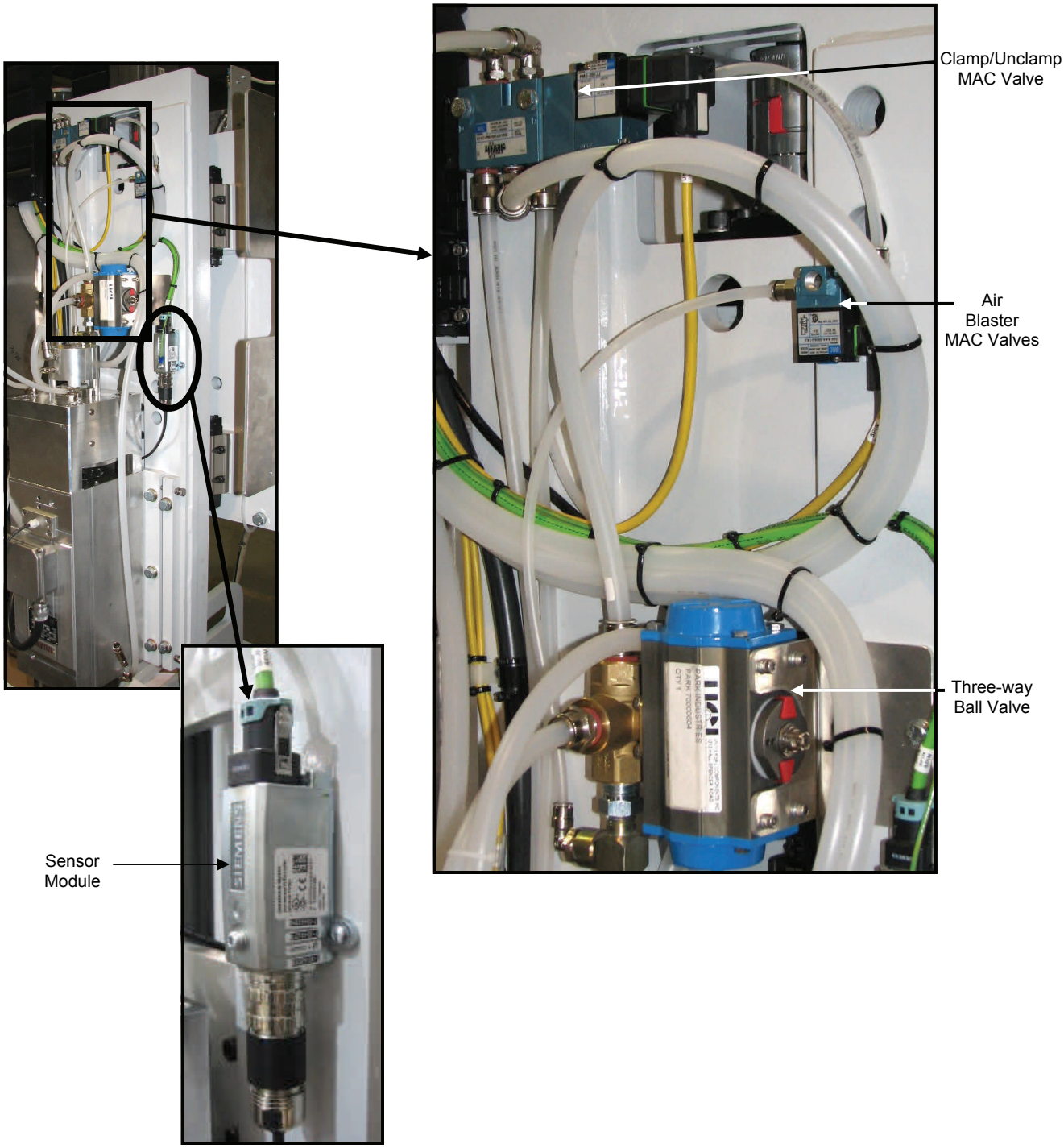
X Axis Drive Gear

X+ Proximity Switch

Component Identification

Rise and Fall (Z Axis) Components

> **NOTE**
>
> ***Some images may show machines with guarding removed for visual clarity. Never operate any machinery without safety equipment in place and in working order.***



Component Identification

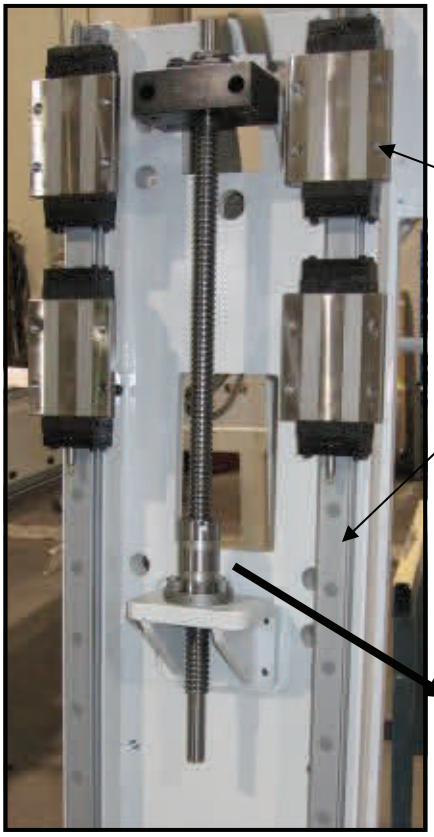
Rise and Fall (Z Axis) Components (cont'd)

NOTE

Some images may show machines with guarding removed for visual clarity. Never operate any machinery without safety equipment in place and in working order.

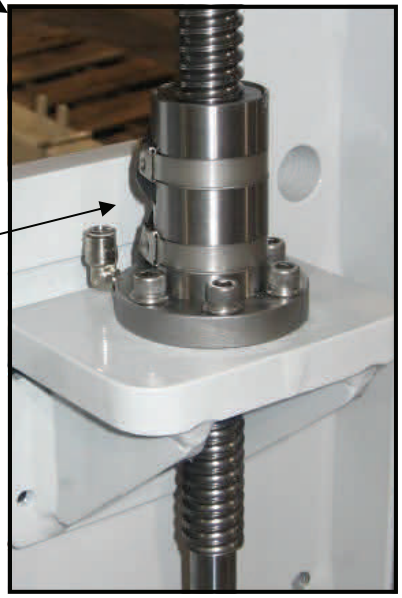
NOTE: These photos were taken during different stages of assembly. Once fully assembled, these components are difficult to see.

Back Side of Spindle Mounting Weldment



THK Bearings (4)
&
Rails (2)

Rise & Fall
Ball Screw

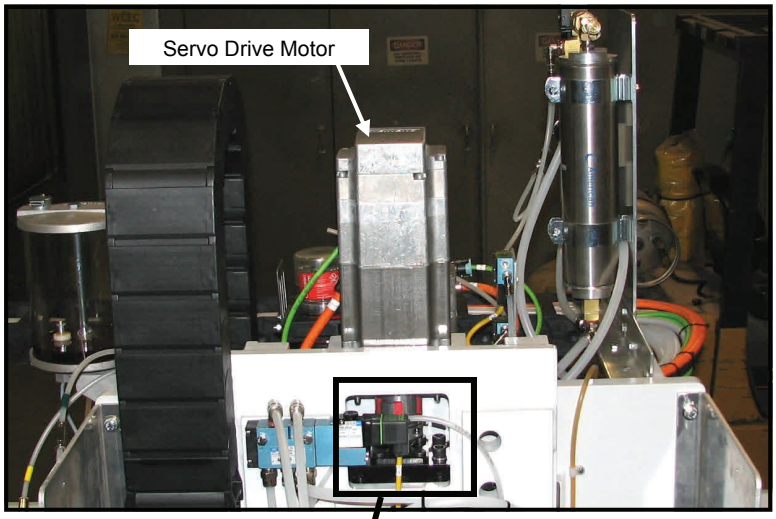


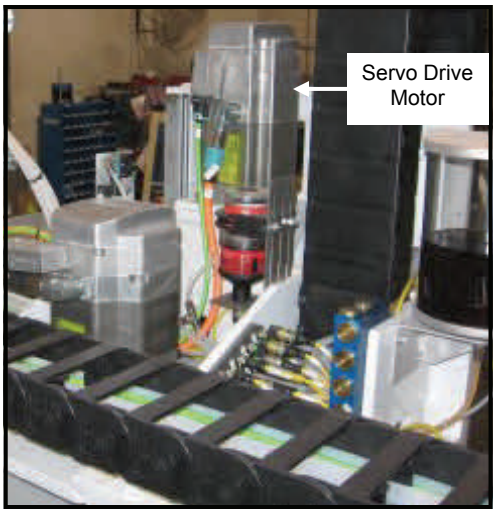
Component Identification

Rise and Fall (Z Axis) Components (cont'd)

NOTE

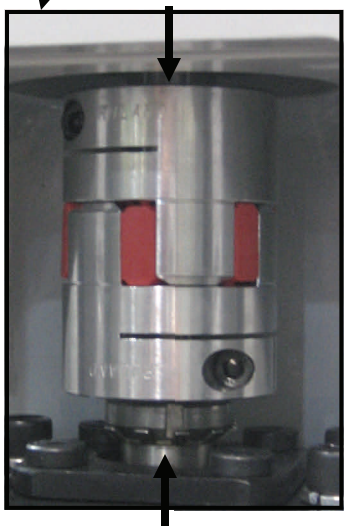
Some images may show machines with guarding removed for visual clarity. Never operate any machinery without safety equipment in place and in working order.





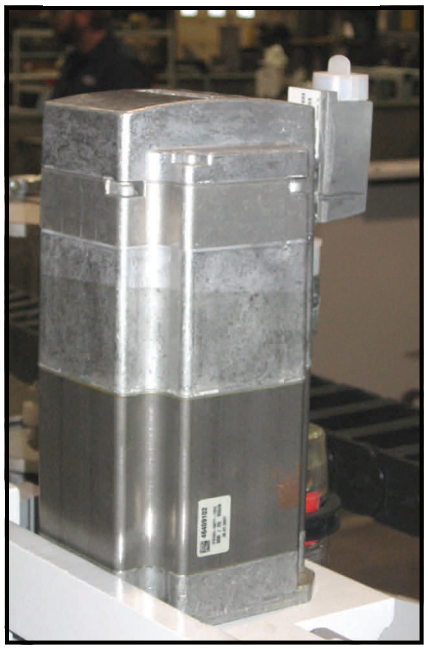
From Servo Drive Motor Shaft

Servo Drive Motor to R & F Threaded Shaft Coupler



To R & F Threaded Shaft

Servo Drive Motor

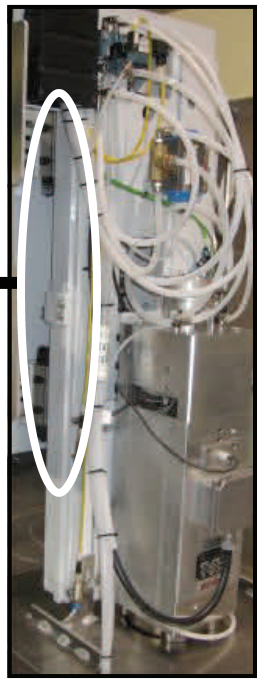


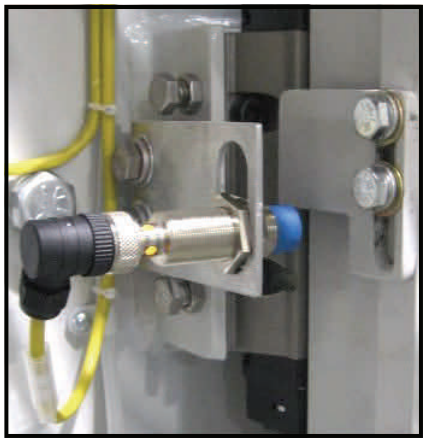
Component Identification

Rise and Fall (Z Axis) Components (cont'd)

Some images may show machines with guarding removed for visual clarity. Never operate any machinery without safety equipment in place and in working order.

NOTE: These photos were taken during different stages of assembly. Once fully assembled, the upper proximity switch is difficult to see.

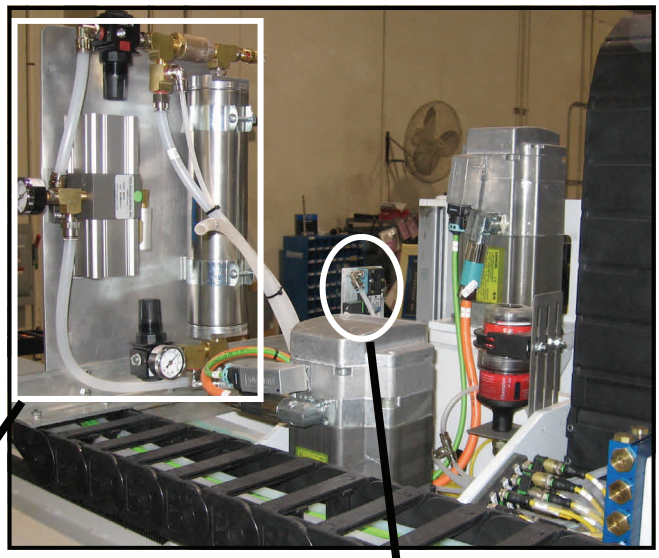




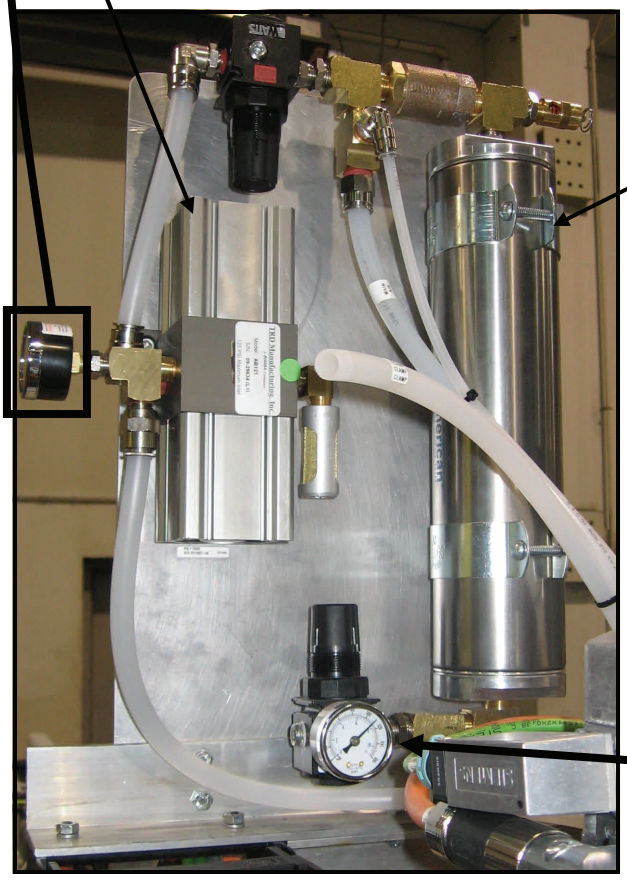
Component Identification

Rise and Fall (Z Axis) Components (cont'd)

Set to 115 PSI



Intensifier



Accumulator Tank

Water Blaster MAC Valve

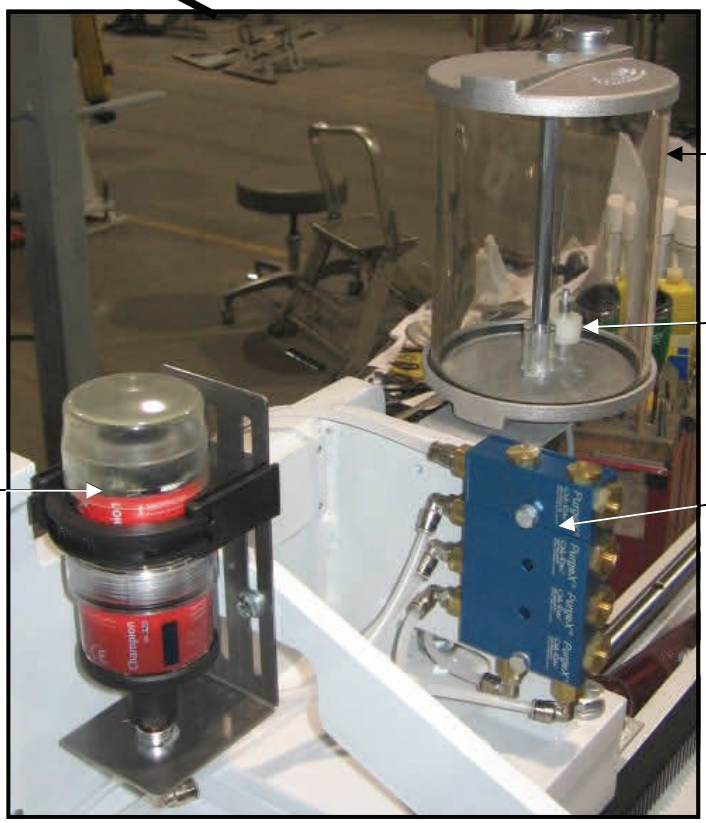
Oiler MAC Valve

Set to 100PSI

Component Identification

Lubrication Components

NOTE: These photos were taken during different stages of assembly. Once fully assembled, these components are difficult to see.



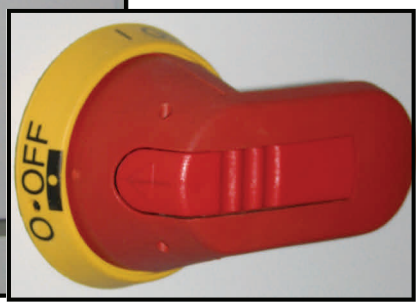
Component Identification

Electrical Components

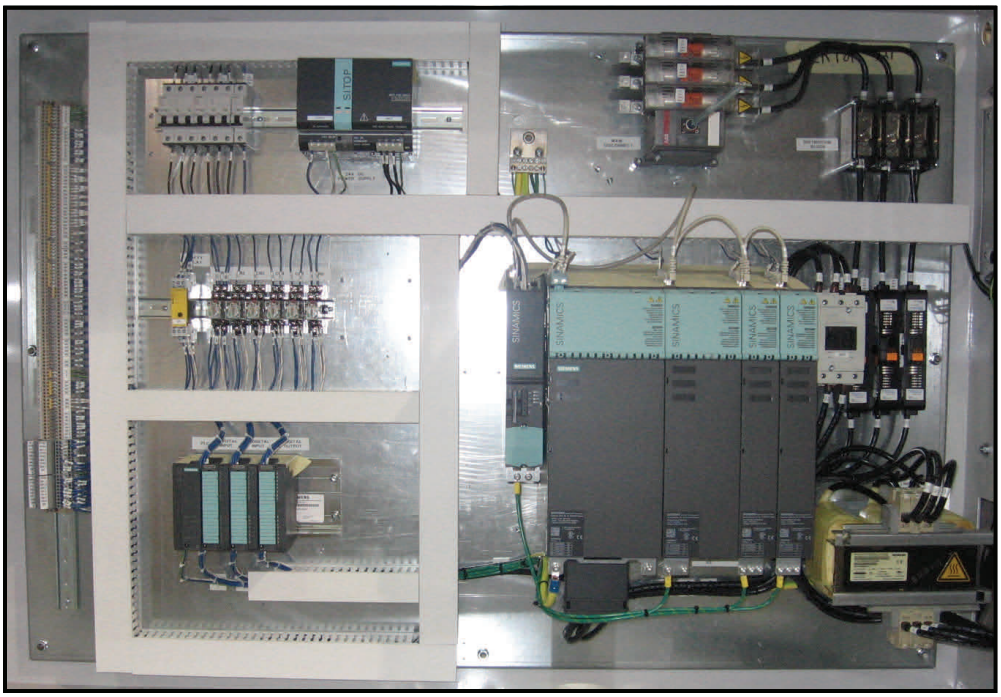
Electrical Cabinet, Doors Closed

ON Position

OFF Position

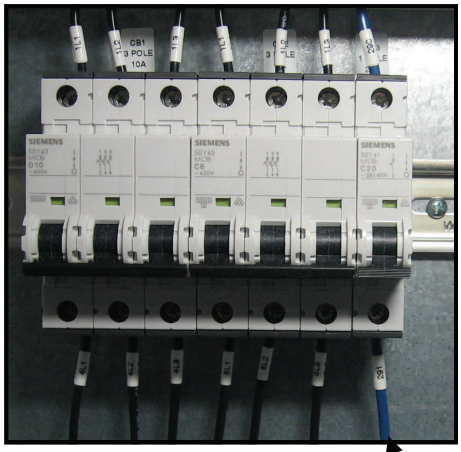


Electrical Cabinet, Doors Open (individual components photos on the next pages)



Component Identification

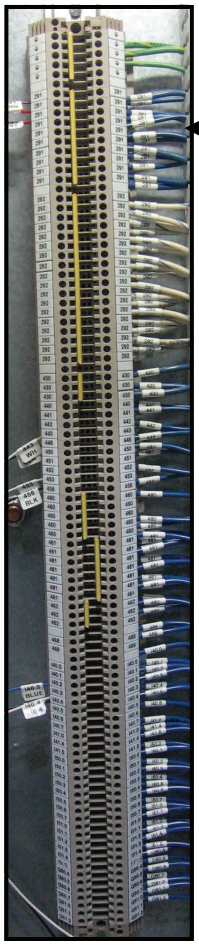
Electrical Components (cont'd)

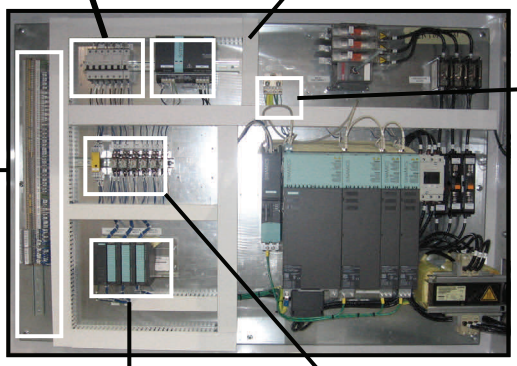


Circuit Breaker Fuse Bank

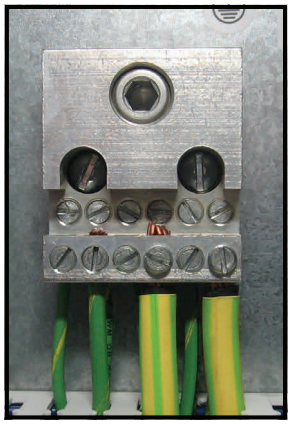
480 VAC to 24 VDC Power Supply

Input/Output Terminal Strip

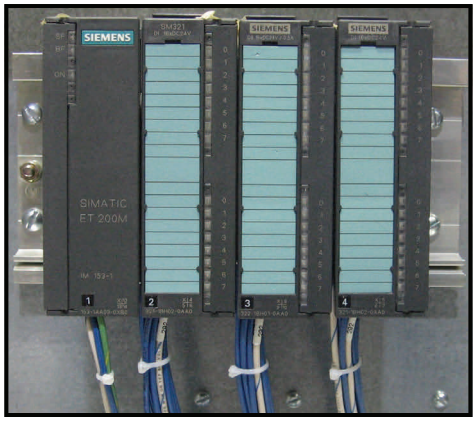




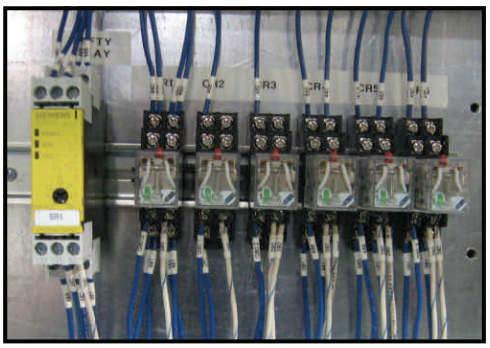
Ground Termination Block



I/O Rack



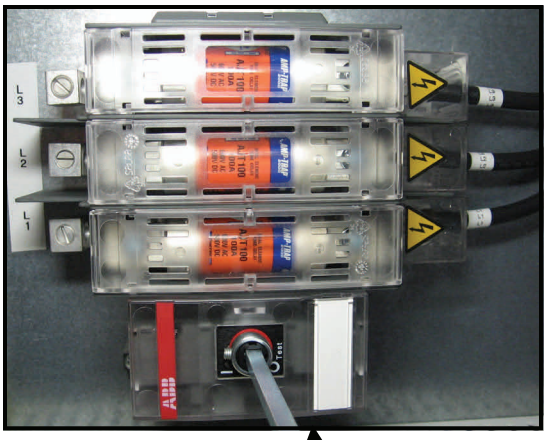
Relays



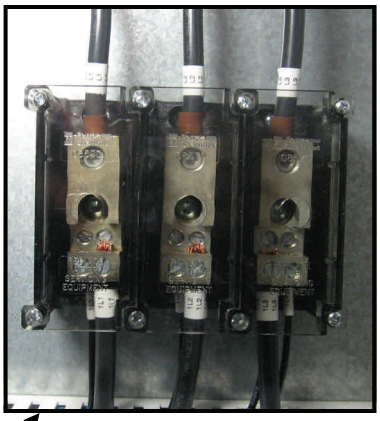
Safety Relay

Component Identification

Electrical Components (cont'd)

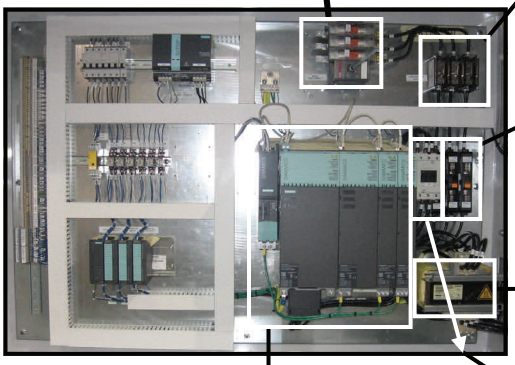


Main Power Disconnect

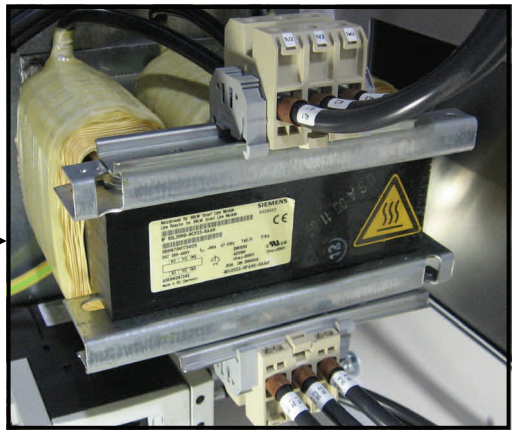


AC Power Distribution Block

AC Line Fuses



Line Reactor



Smart Line Power Module

Spindle Drive

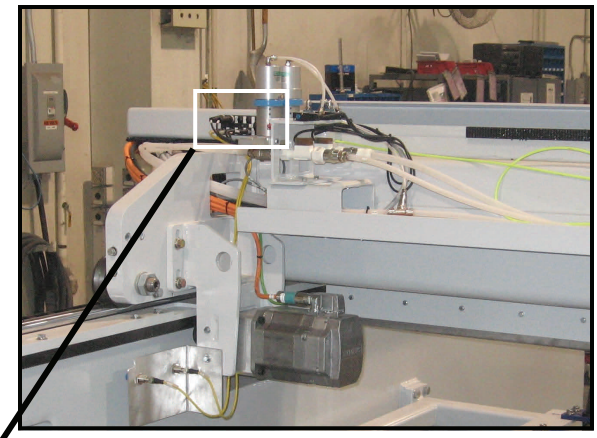
R & F and Cross Travel Drive

Y Master & Slave Drive

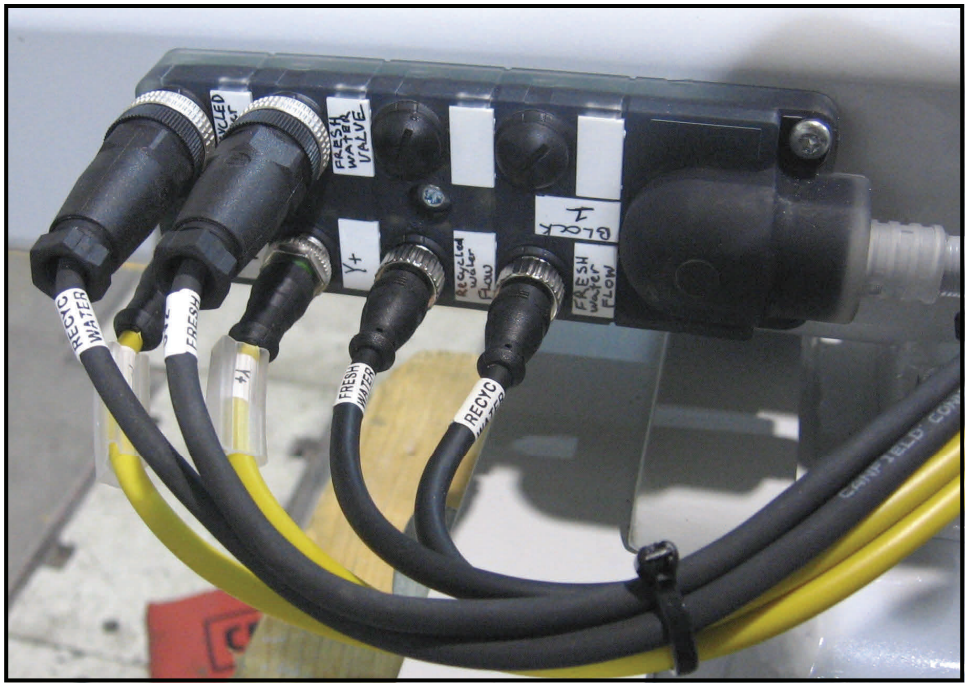
Contactor

Component Identification

Wiring Block 1



Wiring Block 1

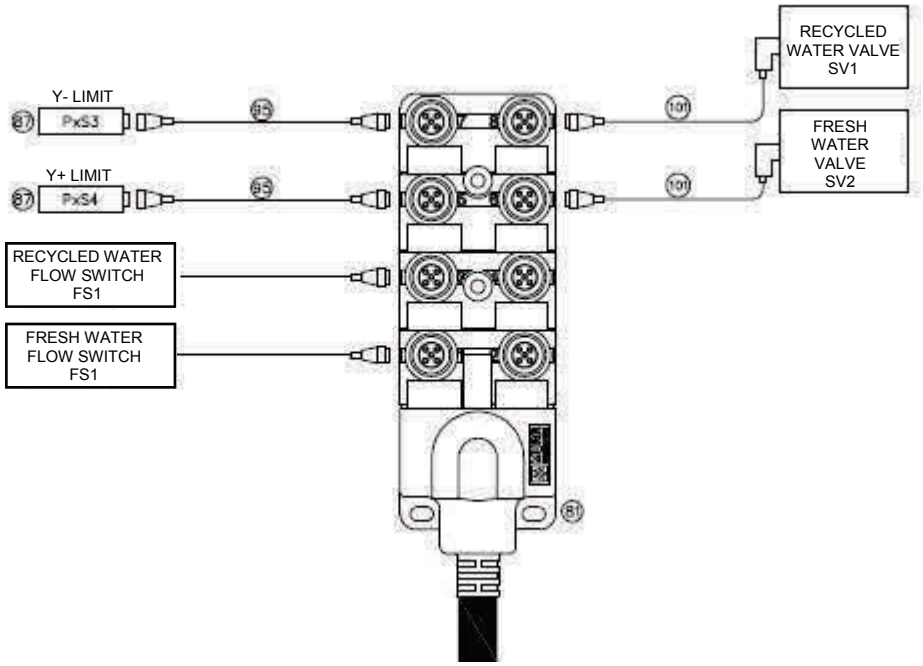


See opposite page for connector labeling.

Component Identification

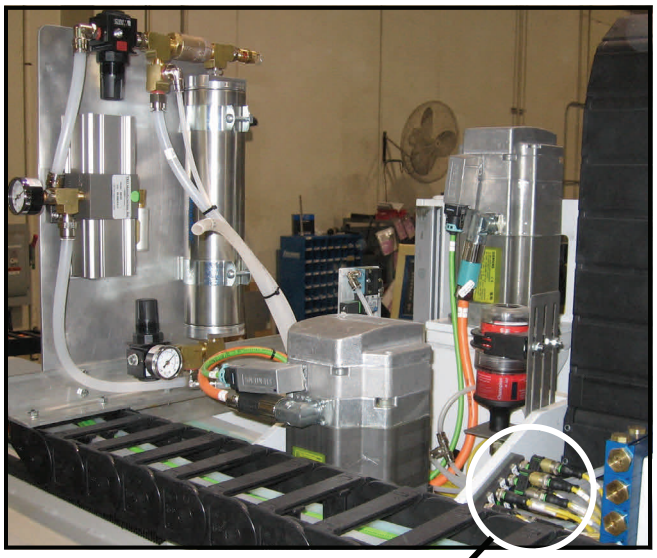
Wiring Block 1 (cont'd)

Wiring Block 1

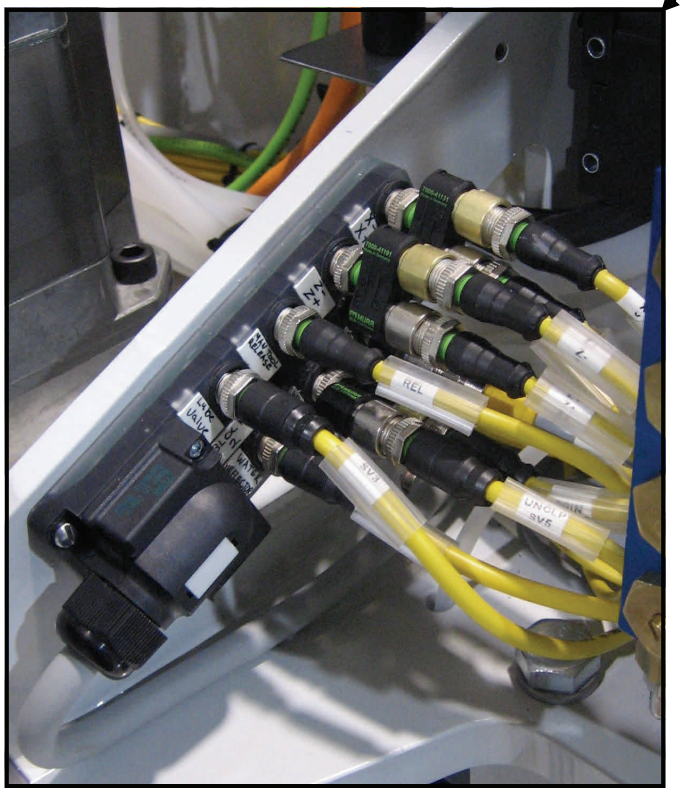


Component Identification

Wire Block 2



Wiring Block 2

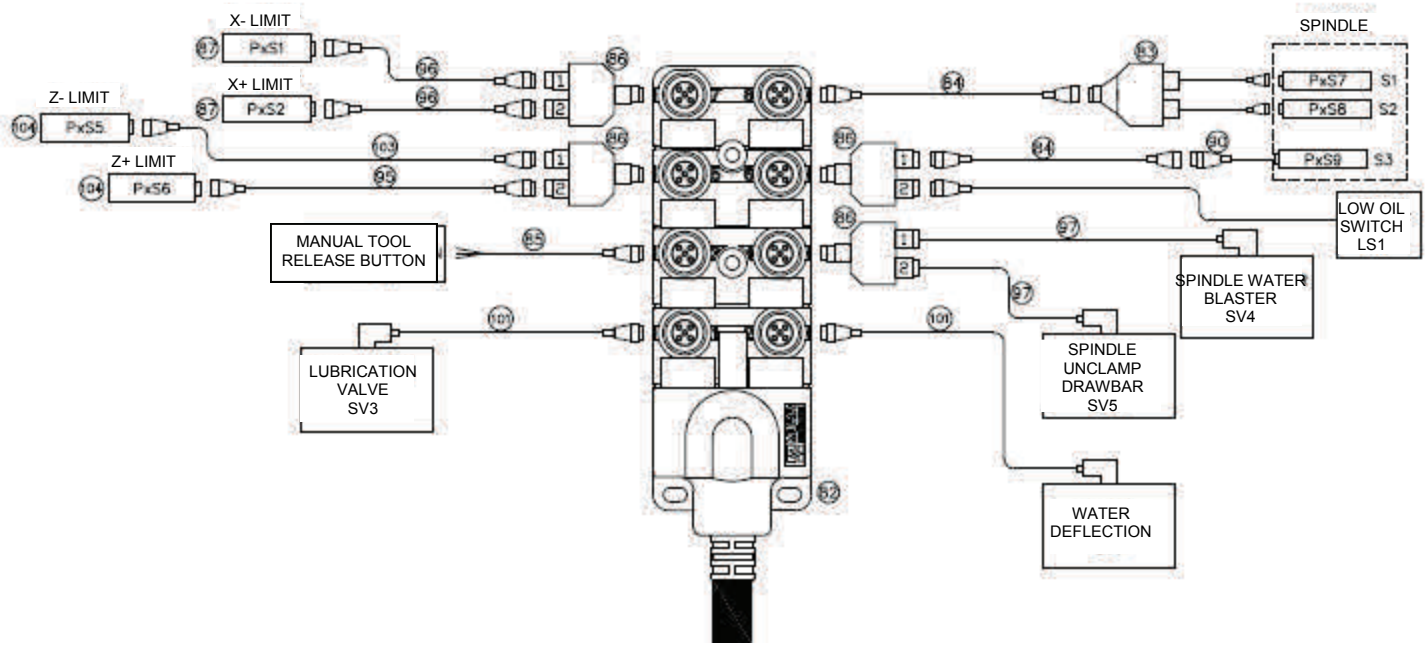


See previous page for connector labeling.

Component Identification

Wiring Block 2 (cont'd)

Wiring Block 2

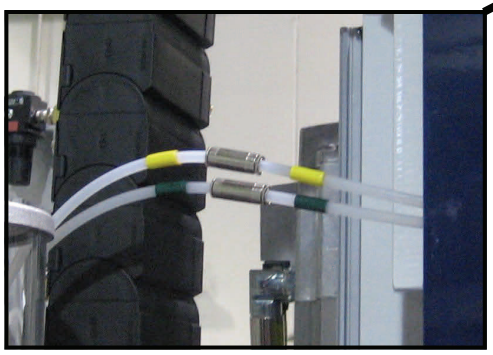


Component Identification

Spindle Cover

Spindle Hood Front View

Air Line Disconnects



Manual Spindle
Release Button
(Left Side)

Water Shroud

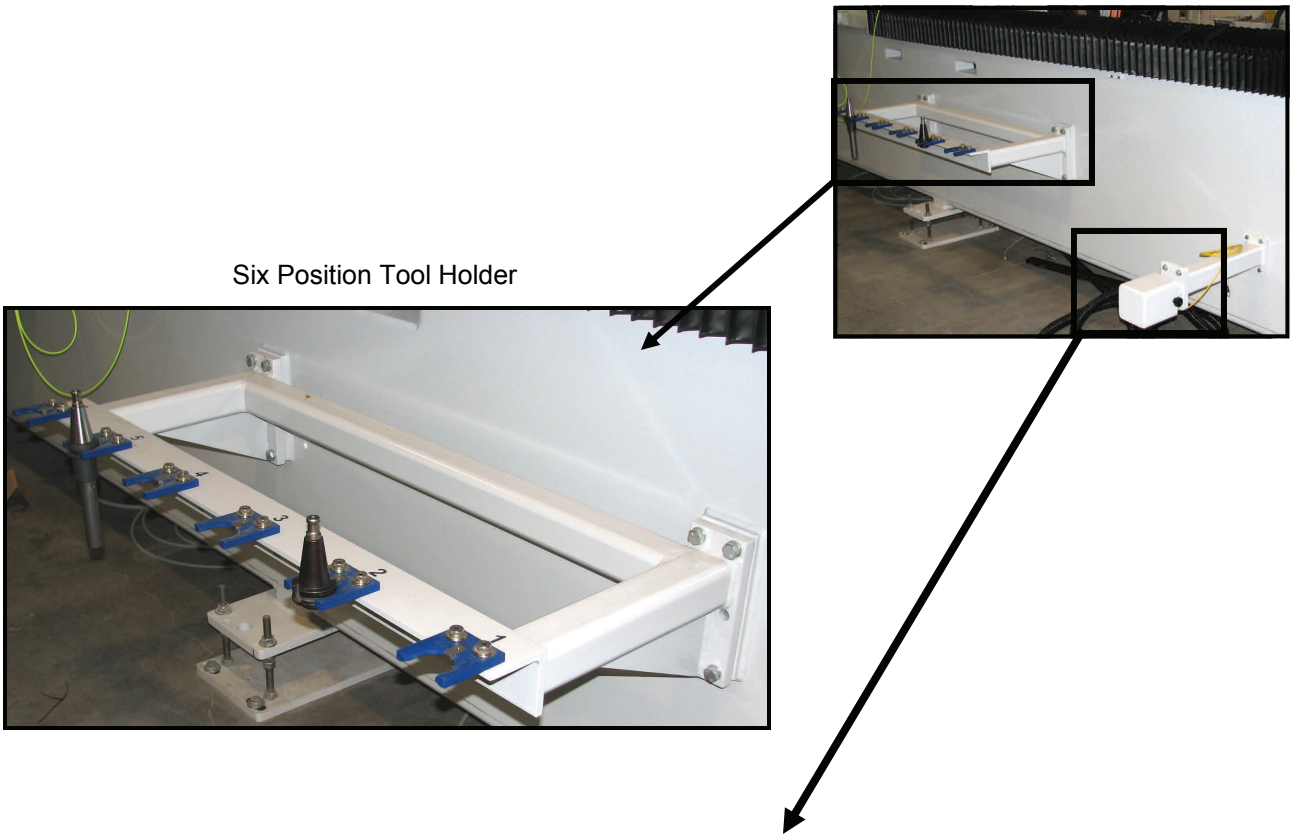
Spindle Hood Rear View

Water Shroud
Extend/Retract
Air Cylinders

Water Shroud

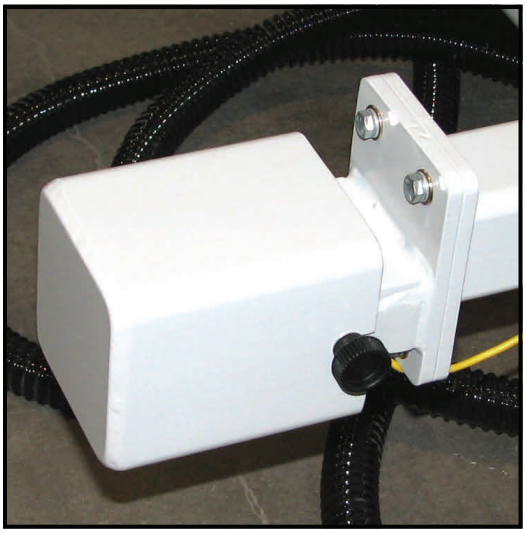
Component Identification

Tool Holder and Tool Height Measuring Device

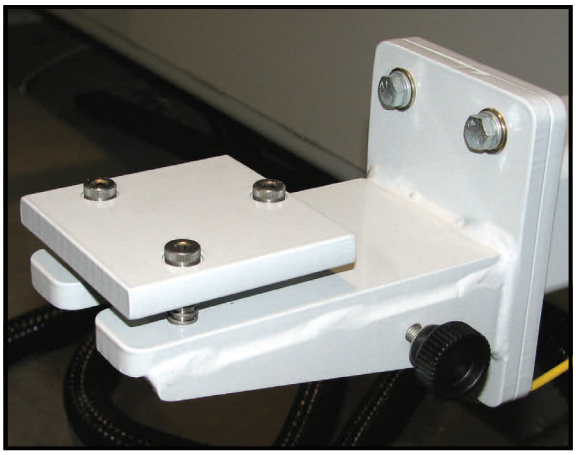


Six Position Tool Holder

Tool Height Measuring Assembly (Covered)



Tool Height Measuring Assembly (Uncovered)



Component Identification

Air Conditioner Unit

Outside Electrical Cabinet
(Front Side of Unit)

Inside Electrical Cabinet
(Back Side of Unit)

Digital Readout

This page intentionally left blank.

Machine Specification

3 CNC controlled axes

X – Axis (Cross travel) - 13'2" of movement, 600IPM maximum rapid traverse, rack and pinion drive guided dual round linear ways with precision slide bearings, SIEMENS AC servo motor with low backlash gearing and position feedback

Y – Axis (Gantry) - 12'6" of movement (11' cutting slab bed), 22'6" of movement (20' cutting slab bed), 600IPM maximum rapid traverse, rack and pinion drive guided by hardened flange/v-groove wheels on precision track, SIEMENS AC servo motor with low backlash gearing and position feedback to prevent "skewing" during travel

Z - Axis (Spindle Rise and Fall) - 18" of movement, zero backlash ball screw drive, 500IPM maximum rapid traverse, maximum precision ball screw drive guided by dual THK linear rails with precision THK bearings, SIEMENS AC servo motor with position feedback

Spindle Drive System

12.5 HP VEM spindle
Variable speed 1000 to 9000 RPM
BT 40 quick-change tooling
Through spindle water feed, External water ring
Retractable water shroud to reduce water and stone spray

Power

480 Volt AC, 3 Phase, 80 AMP service (Standard machine, no transformer required)
208 Volt AC, 3 Phase, 150 AMP service (Transformer required)
240 Volt AC, 3 Phase, 150 AMP service (Transformer required)
600 Volt AC, 3 Phase, 60 AMP service (Transformer required)

Water

Dual Water Feed - (two 3/4 inch ID water lines minimum)
- 5 GPM, 45 PSI, filtered to 50 micron (spindle center water feed)
- 5 GPM, 45PSI, filtered to 50 micron (water ring)

Single Water Feed - (one 3/4 inch ID water line minimum)
- 10 GPM, 45 PSI, filtered to 50 micron

Air

1/2 inch ID line, 100 PSI pressure, 20 SCFM volume (continuous)

Controls

SIEMENS 840 Di CNC control system
Standard (3) axis "Teach in" pendant (X,Y and Z axes)
Low water safety, Over amp safety, Arbor Amp Following, Water control solenoid
MasterCAM CAD/CAM software (Park Industries supplied computer)

Machine Weight

Weight 16,000 lbs

Includes the following

24' of gantry track, gear rack, flexible cable carrier, wire and hose

Connectivity Requirements

The following computer and internet connections are required for efficient VEKTOR operation and to allow Park Industries Customer Service individuals access to your machine for maintenance purposes. If you have any questions about connectivity requirements, please contact Park Industries at 1-800-328-2309 and ask for your Installation Coordinator

Internet Connection:

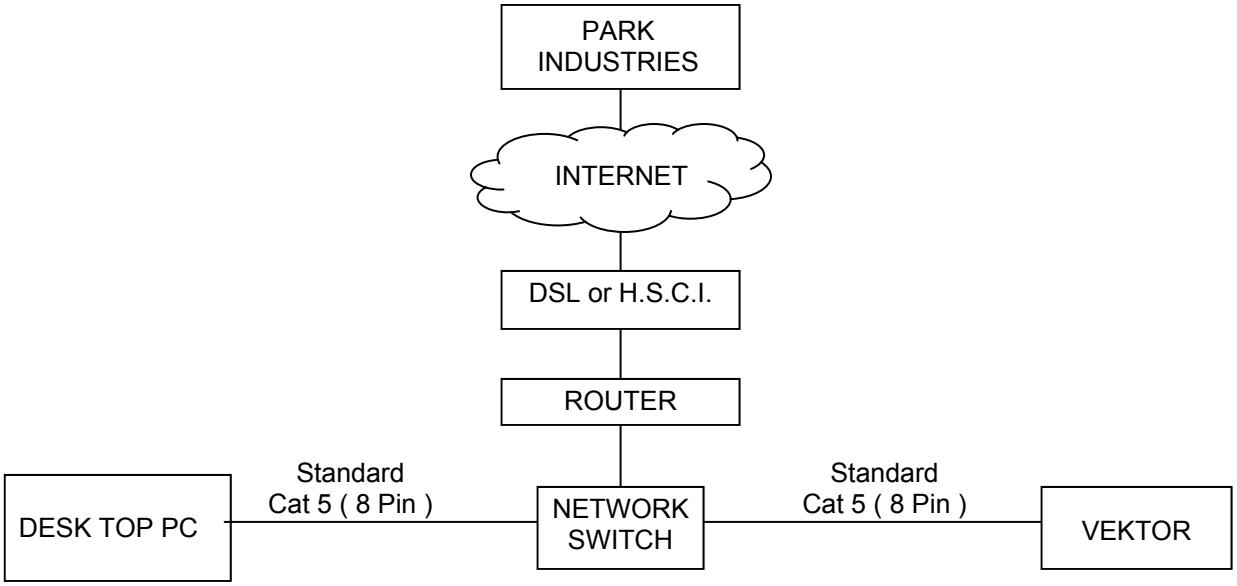
DSL phone line or High speed cable internet is required with the ability to do network address translation (NAT) at the router.

Other info:

Windows XP.
Ports 5635 & 5636 need to be open. Park Industries sets up PC ANYWHERE with a secure logon. (IP address at Park is 66.173.42.130)

CAT5 network cable hook-up:

This is required for the computer included with your purchase for remote office programming. Cable to reach back right corner of machine plus 10 feet as viewed from loading side of table.



Section 2: MasterCAM Setup

Contents Page

General 2-1
CAD Principles and Operations 2-2
CAD Screens and Definitions 2-3
Help Screens 2-4
Customize Screen Configuration 2-5
Customize Toolbars 2-9
Create Tool Library 2-12
Creating Default Tools 2-18
Changing Default Posting Settings 2-35
Setting the Default Machine 2-39

General

This section will describe the use of the Mastercam® Software Package which is the CAD/CAM software chosen to run the VEKTOR CNC Profiler. It describes how to set up Mastercam® for use, how to customize screens, toolbars, the actual tools used to cut stone for day to day operation, and posting settings.

CAD Principles and Operations

This Section describes how to:

- Use the Menu and Status Bars
- Customize the screen and toolbars for ease
- Customize Tooling needed to run production

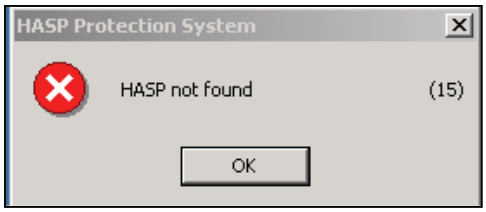
What is CAD?

CAD stands for Computer Aided Drafting. CAD is software that was developed to make the drafting or drawing process more efficient and flexible.

The examples in the next chapter will demonstrate the use of common patterns that include straight lines and arcs to draw simple shapes.

Licensing

Mastercam® uses two types of licensing: Single user license or Network license. The single user license uses a hardware device called a HASP which is attached to a computers Parallel Port or a USB Port. When the HASP is not connected an error will be generated as shown below.

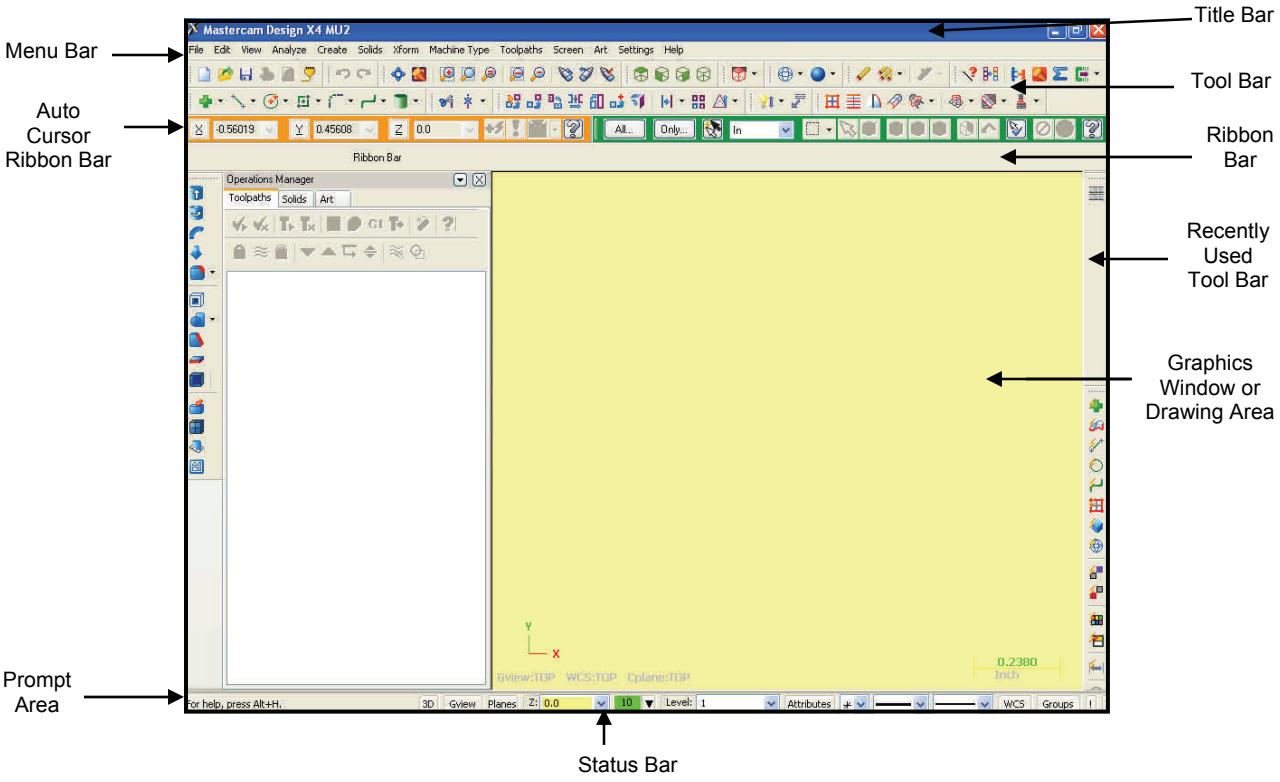


If a network license is being used, contact the system administrator if any of the following errors are displayed.

- Error checking out a router license. No licenses have been purchased for this product.
- Active NetHASP server not found.
- All available licenses are in use.

CAD Screens and Definitions

Before drawing can be started, the understanding and use of the CAD options are essential. The main screen is where all of the drawing will be done and where most of the options are displayed. The screen below is the way Mastercam® appears before any modifications are made. For ease, the programmer may want to personalize it to fit his/her style.



Menu Bar - Pull-down menus listing commands.

Tool Bar - Each of the buttons in the toolbar is a shortcut for commands in the Menu Bar. If the cursor is positioned on top of the button it will define what the button does.

Auto Cursor Ribbon Bar - Shows actual coordinates of the position where the mouse pointer is located.

Ribbon Bar - Prompts you with icons for distances, OK or Help.

Recently Used Toolbar - Shows icons previously used for each time different command is selected.

Graphics Window - The drawing area where parts and tool paths will be displayed.

Prompt Area - This space tells you what information needs to be entered. What is typed will also be displayed here.

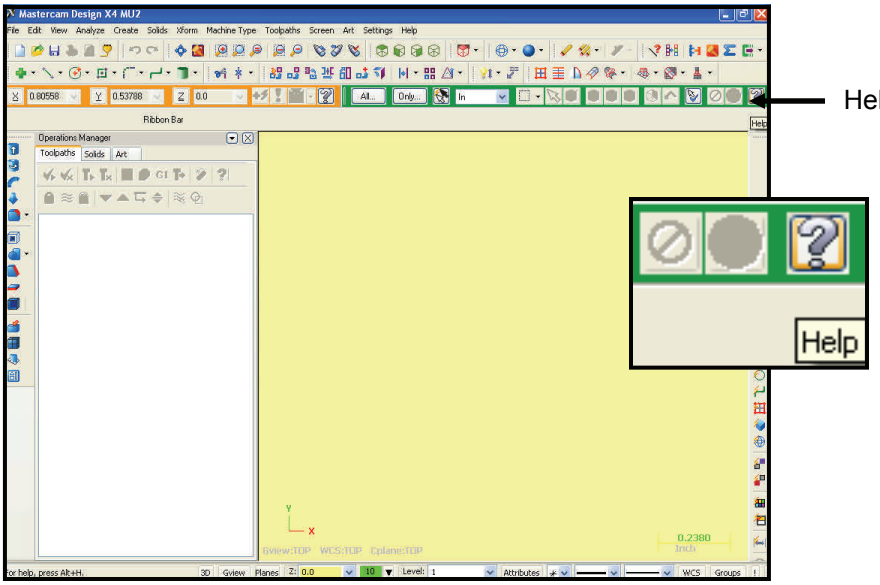
Status Bar - Shows information about lines, coordinate system type information.

Help Screens

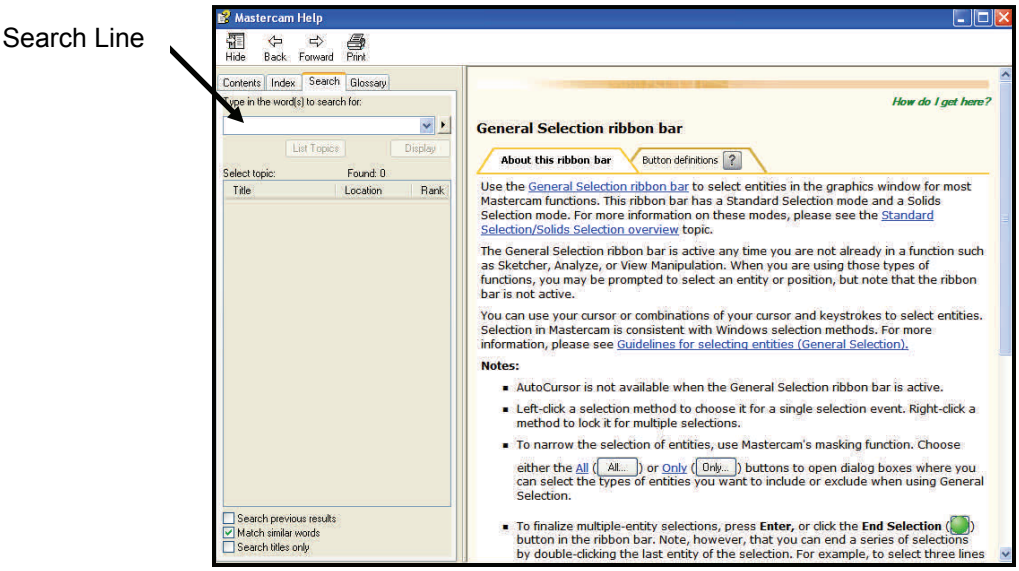
The CAD/CAM software has extensive help screens to explain all areas of the CAD/CAM screen. Definitions are given for such things as tool bar, drawing process, and file process.

How to obtain information from the help menu to draw a “Polar” line entity, for example.

1. From the Main Screen, select the from the top of the screen.



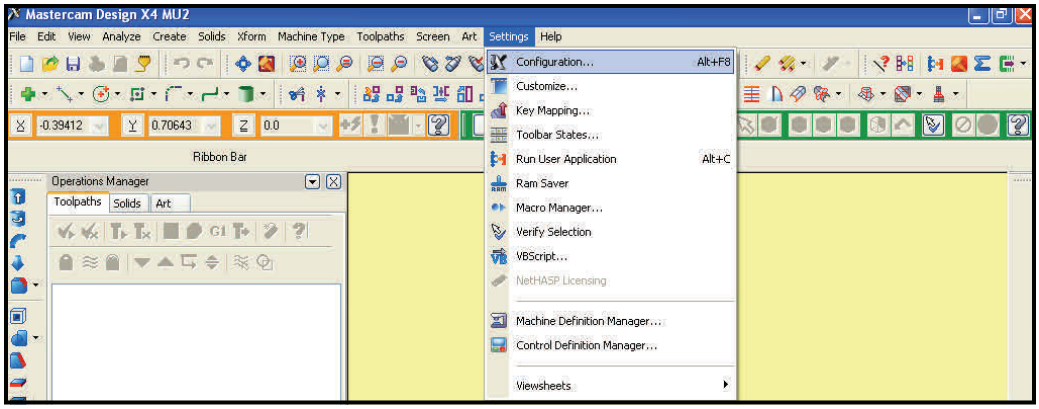
2. After pressing the Help button, the “Mastercam® Help” dialog box will be displayed. Click on the “Search” tab. Use the search line to entry a word or subject for which help is required and then click on the arrow on the right side of the entry line.



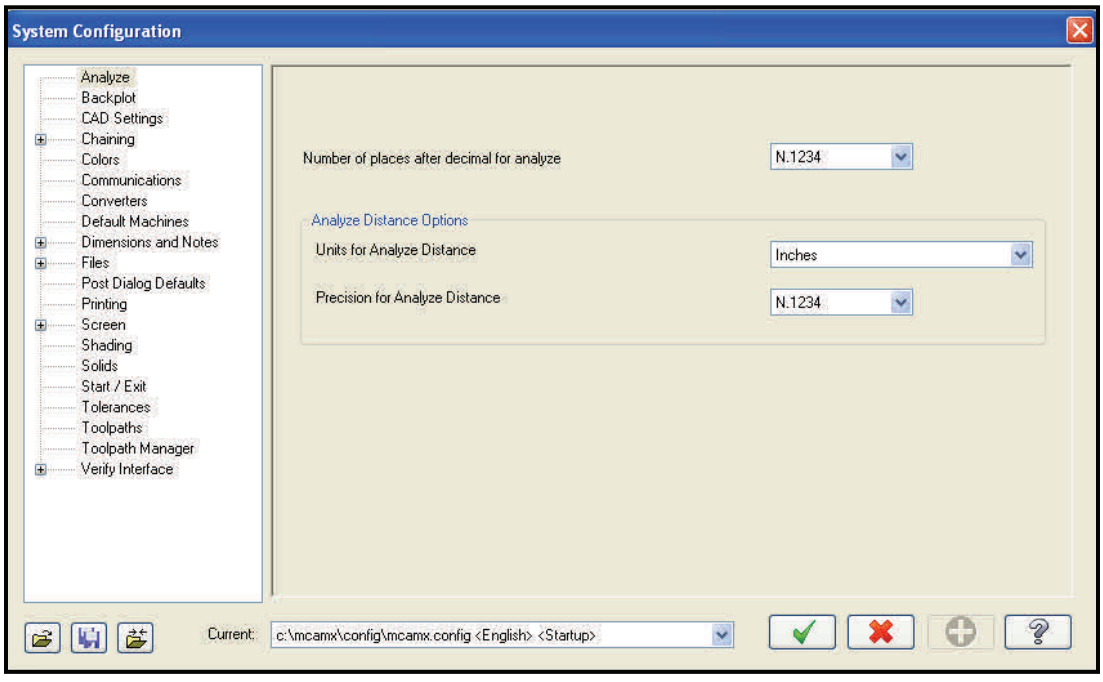
Customize Screen Configuration

Mastercam® gives the option to personalize items within the software to make it easier on the programmer. All programmers are different and want their own style and this is why Mastercam® has the opportunity to customize the screens appearance. Below are steps shown to personalize useful items, these are items found to fit Park Industries needs.

1. From the Main Screen, select “Settings” from the Menu Bar at the top of the screen. Then choose “Configuration” as shown below.



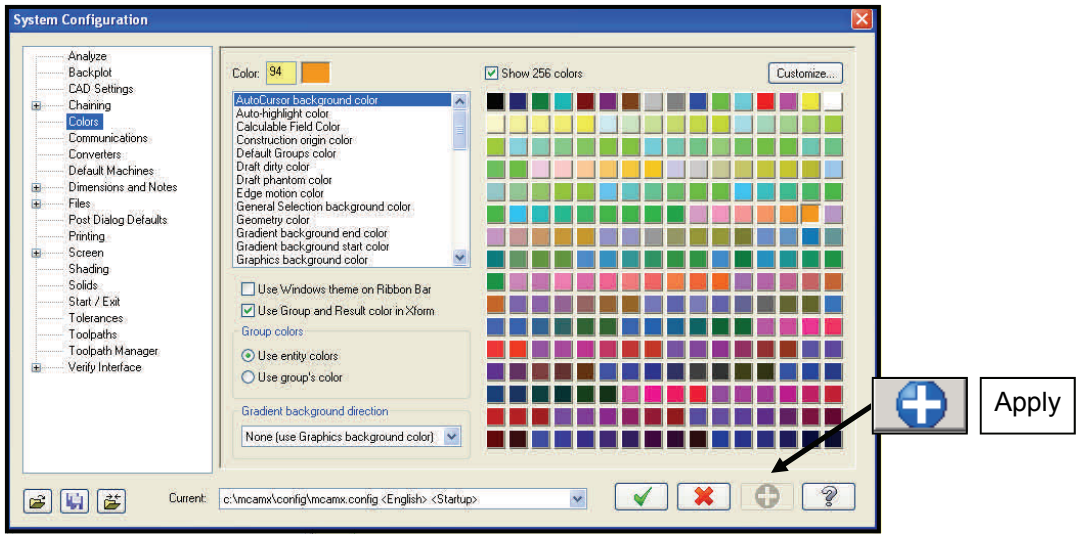
2. Once “Configuration” has been chosen the following screen “System Configuration” is displayed. This is where the user can customize the different “Topics” that pertain to drawing in Mastercam®



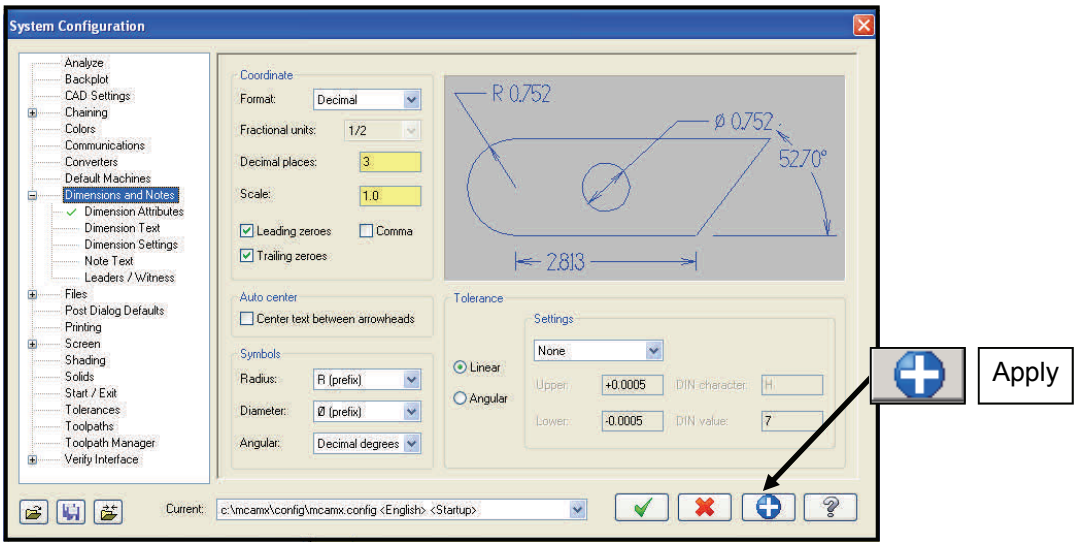
Customize Screen Configuration (Cont'd)

3. On the left side of the "System Configuration" screen there are many different topics to choose from. Out of the many topics, most will stay standard as they came when the software was loaded.

   A. Choose "Colors" as shown below. Check to see if "Geometry Color" = 10, "Graphics Background Color" = 0 and "System Origin Color" = 6, "AutoCursor background color" = 94, "General Selection background color" = 128, make changes if necessary. Once finished, click on the blue + symbol for Apply.

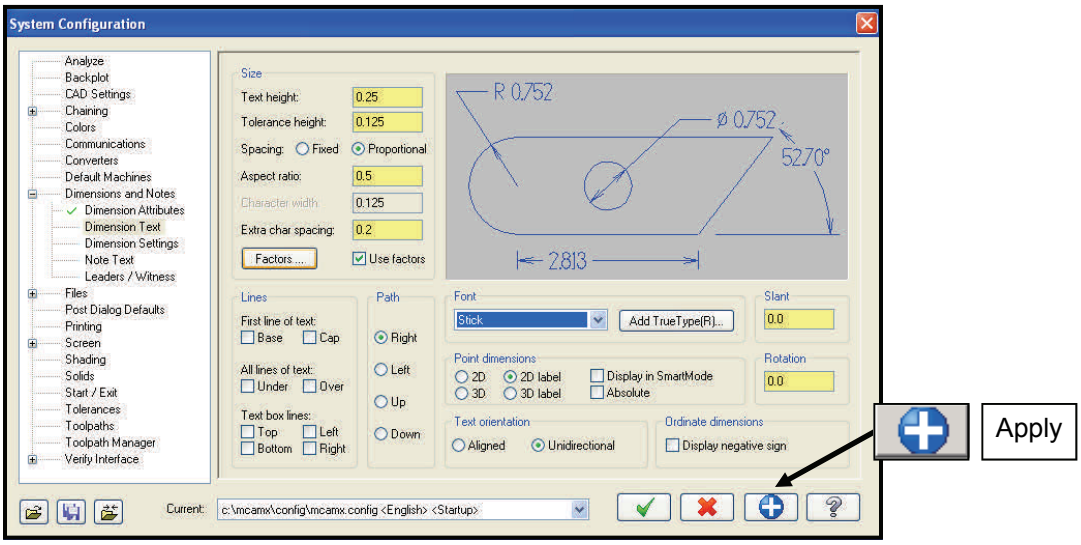


   B. NOTE: Dimensions and Notes line has additional sub topics. Click the + symbol next to the title to list the other topics.
   Choose "Dimension Attributes" as shown below. Choose the type of "Format" that you want to use generally "Decimal or Fractional". Once the "Format" has been chosen then for "Decimal Places" pick 2 = .01 or 3 = .010, for "Fractional Units" pick 1/64 = 5/64 or 1/32 = 8/32, these are how all dimensions will be displayed on the screen when using "Dimension". Once finished, click on the blue + symbol for Apply!

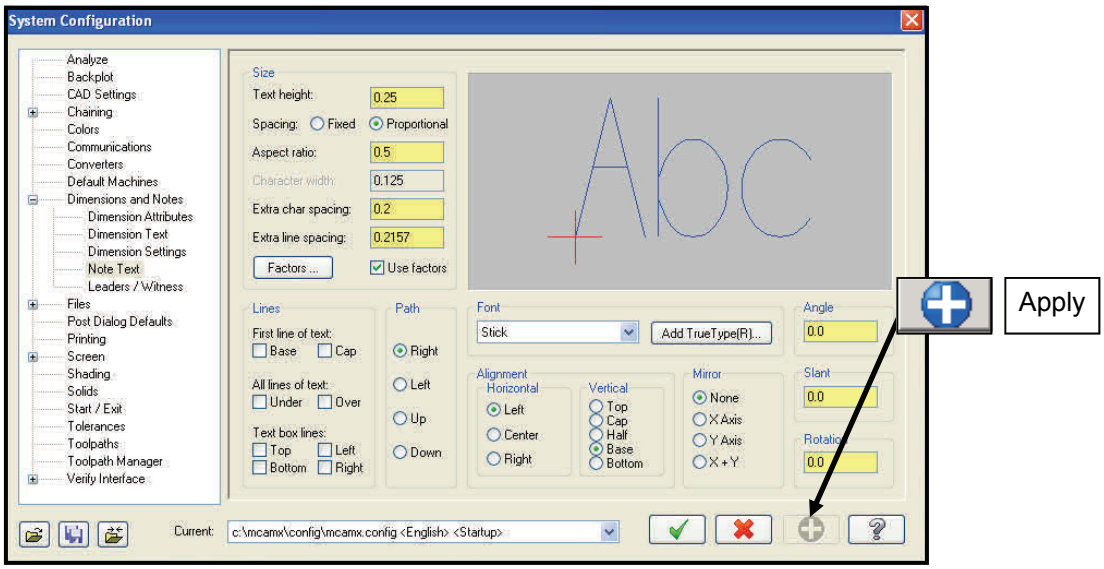


Customize Screen Configuration (Cont'd)

C. Choose "Dimension Text" as shown below. Change "Text Height" from .25" to .50" to make it easier to view. Once finished, click on the blue + symbol for Apply.

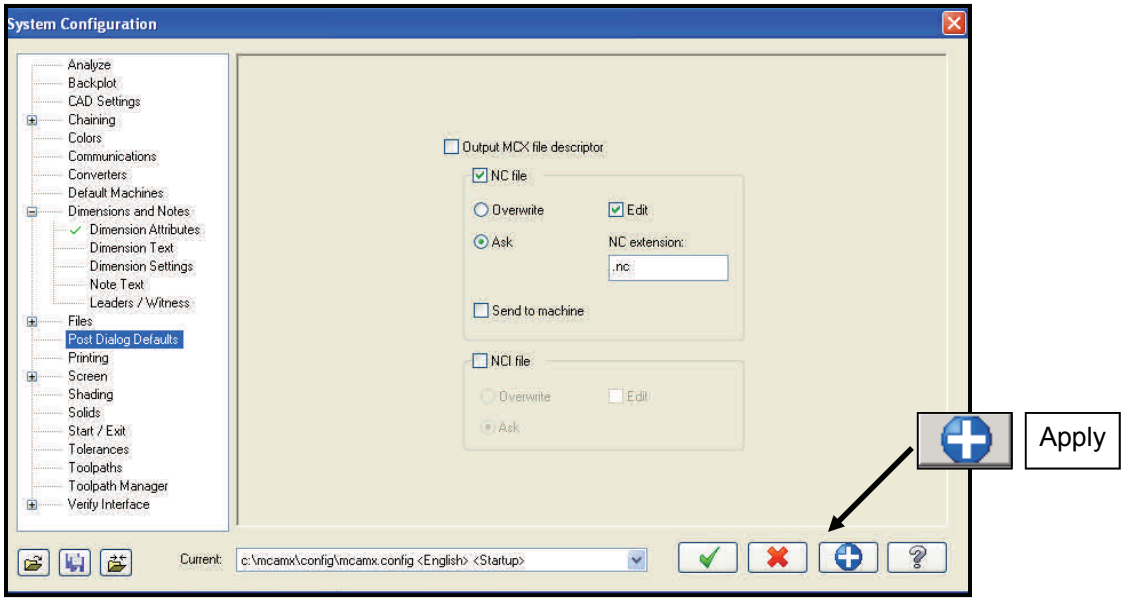


D. Choose "Note Text" as shown below. Change "Text Height" from .25" to .50" to make it easier to view. Once finished, click on the blue + symbol for Apply.

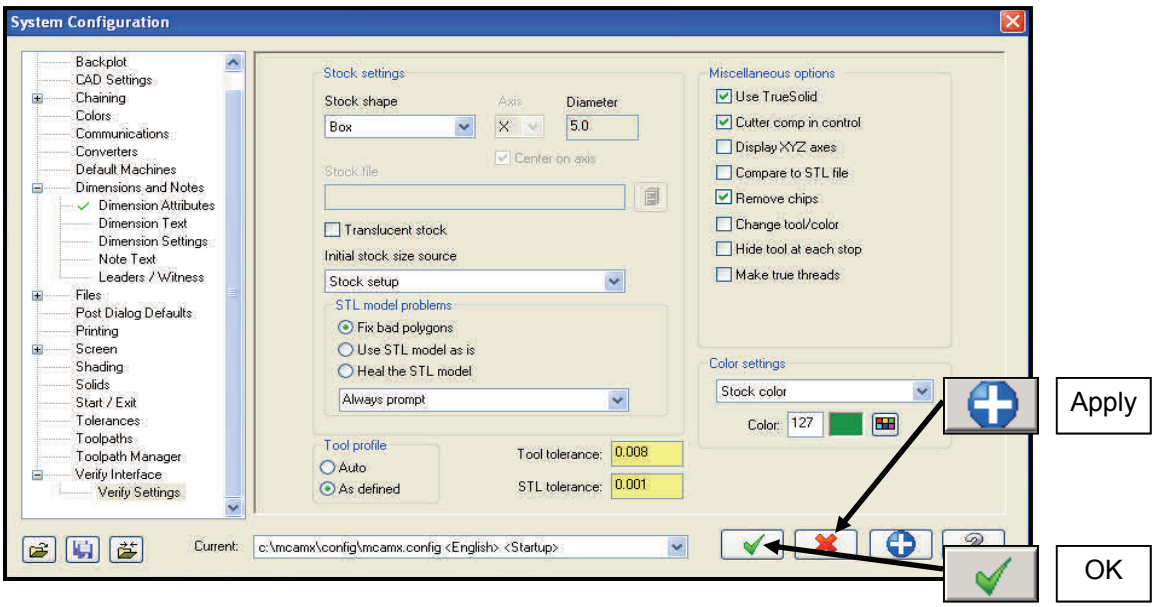


Customize Screen Configuration (Cont'd)

E. Choose "Post Processing" as shown below. Change "NC Extension" from .nc to .txt , this makes it possible for the machine to read. Once finished, click on the blue + symbol for Apply.



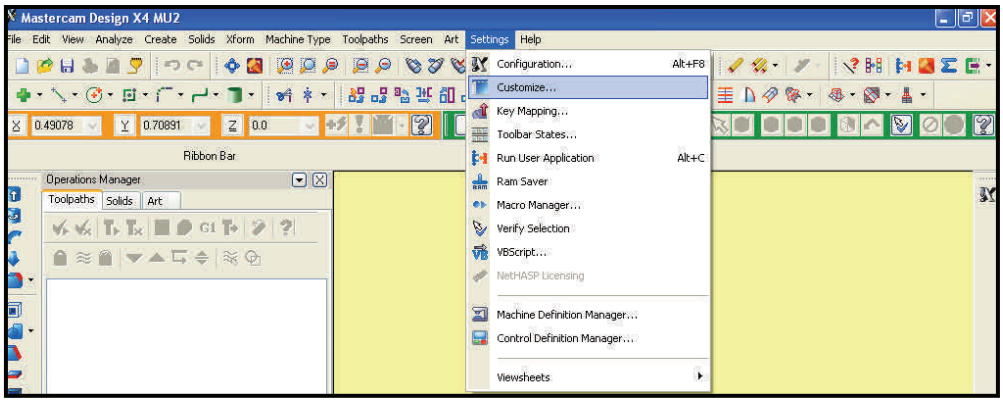
F. Expand the "Verify Interface" (click + box) and the select "Verify Settings" as shown below. Click on "Display XYZ axes" and also "Remove Chips" so both have check marks in both boxes. Once finished, click on the blue + symbol for Apply then the green check mark for OK.



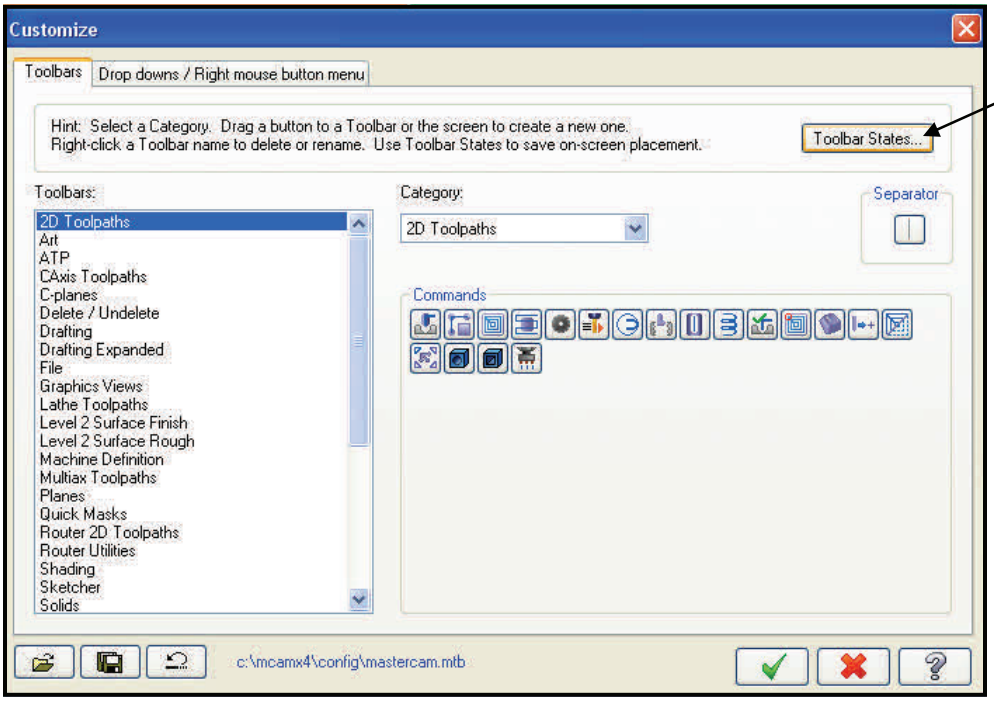
Customize Toolbars

Mastercam® provides the option to personalize items within the software to make it easier on the programmer, since all programmers are different and want their own style. This is the reason Mastercam® has the ability to customize screen appearance. Below are a few items that Park Industries has customized to fit our needs.

1. From the main screen, select “Settings” from the Menu Bar at the top of the screen. Then choose “Customize” as shown below.



2. Once “Customize” has been chosen the following screen will display. Click on “Toolbar States”. This is where the user can choose the different items displayed in the toolbar.



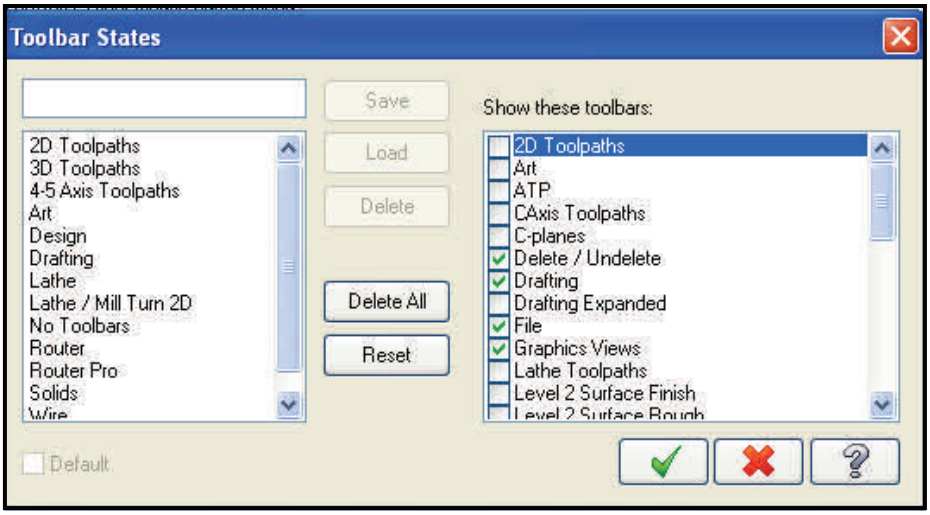
Toolbar States

Customize Toolbars (Cont'd)

3. From the "Toolbar States Screen", click on/off the white boxes within the "Show these toolbars" area. This is where the user decides which toolbars are displayed on the screen. To save individual preferences enter a name then press "Save", if one already exists then click on that name after making changes then press "Save". Once the main toolbars have been chosen then click on the green check mark button for OK to exit.

   The toolbars that we have chosen for our standards are:

   2D Toolpaths
   C-Planes
   Delete / Undelete
   Drafting
   File
   Graphics View
   Lathe Toolpaths (Optional)
   Planes
   Sketcher
   Trim / Break
   Undo / Redo
   Utilities
   View Manipulation
   WireFrame Toolpaths
   Xform

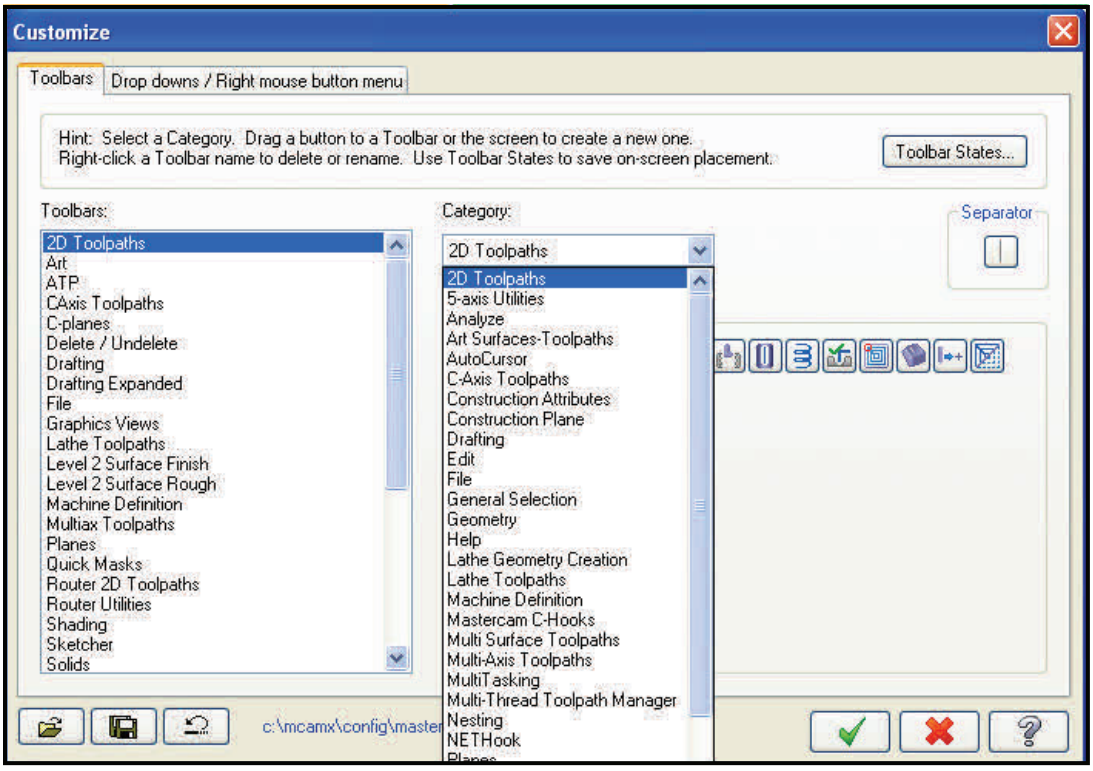


Customize Toolbars (Cont'd)

4. Once the "Toolbar States" have been chosen then each of those can be modified. To customize the toolbars that have been chosen, icons and separators that are unwanted can be clicked on and dragged to the main screen, they will be deleted automatically from the toolbar.

   To add icons to the existing toolbars, choose a "Category" type by clicking on it (use pull down when needed). Next click on the icon to be added and then drag it to the selected toolbar. Once all changes have been made click on the green check mark for OK and all changes are made to the file name that is to the left of the green check mark

   Note: For Mastercam® to start up with all the necessary changes made, all changes must be saved to: C:\mcamx\config\mastercam.mtb.



Create Tool Library

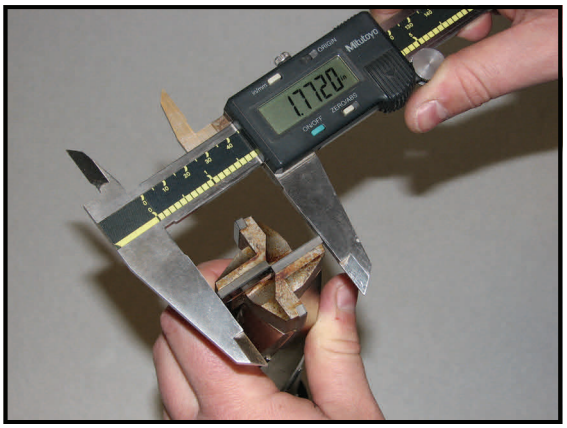
In order to program parts in Mastercam®, and run the parts on the VEKTOR, a tool library needs to be created. The tool library will contain information for the tool (in this case a 1 3/4" pcd mill bit is shown), such as: tool lengths, tool diameter, tool pocket location numbers, feed rates, RPMs, tool name and other tool related information.

This procedure requires two steps:

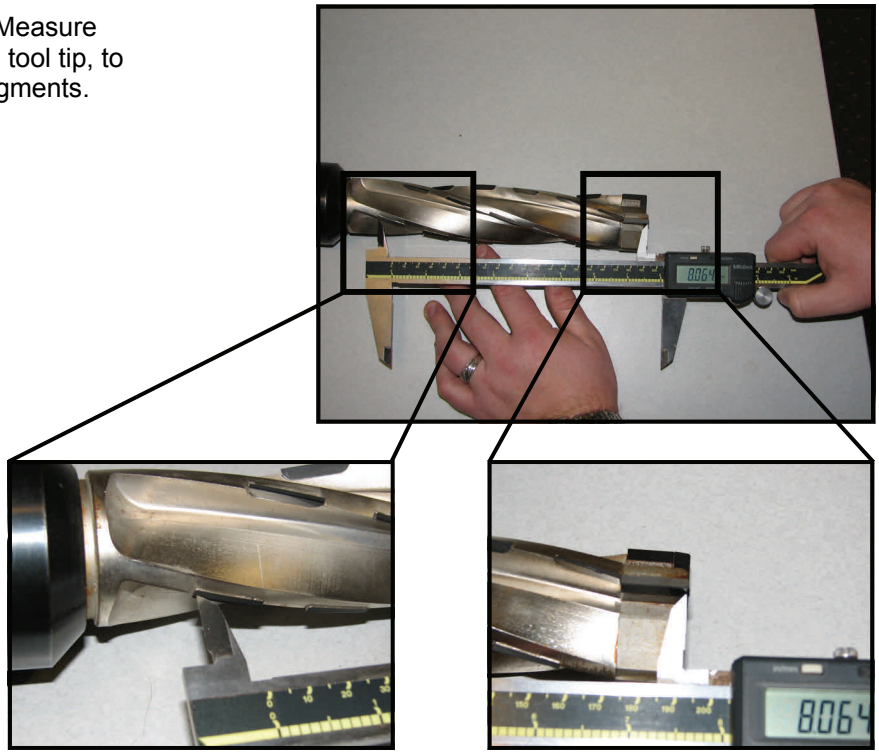
- Measure the lengths and diameter of the tool.
- Enter the measured lengths and diameter into the tool manager, along with the remaining required information.

Gathering Measurements

1. **Tool Diameter** – Measure the diameter of the tool across the PCD segments. Rotate the caliper, until the largest value is returned from the caliper. Write this diameter value down.

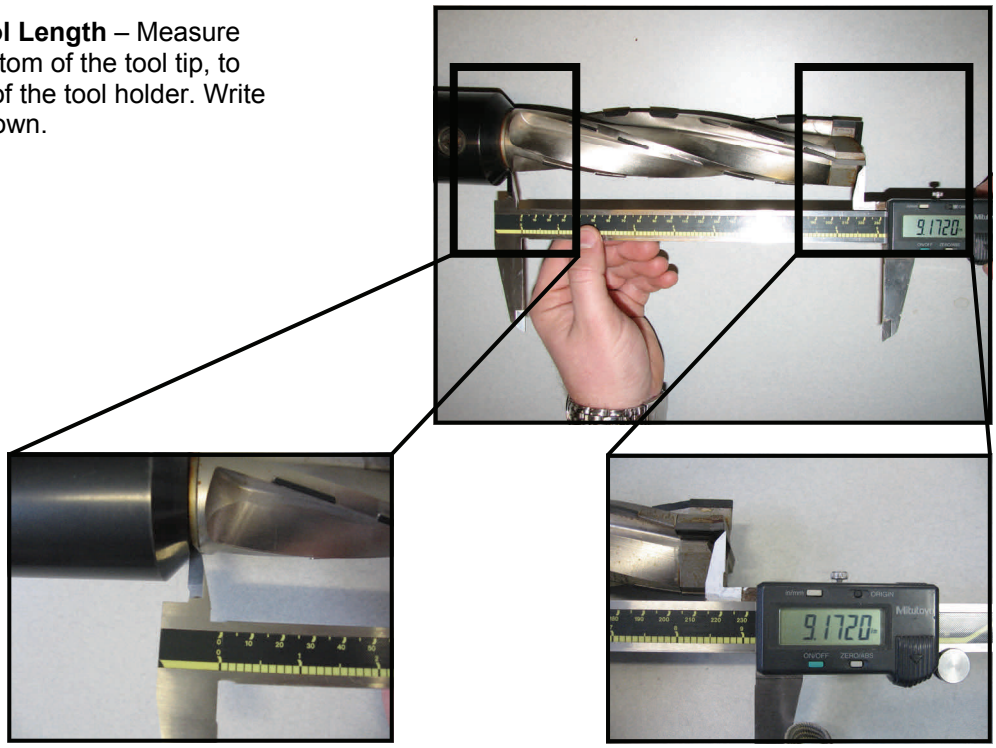


2. **Tool Flute Length** – Measure from the bottom of the tool tip, to the top of the PCD segments. Write this value down.



Create Tool Library

3. **Overall Tool Length** – Measure from the bottom of the tool tip, to the bottom of the tool holder. Write this value down.

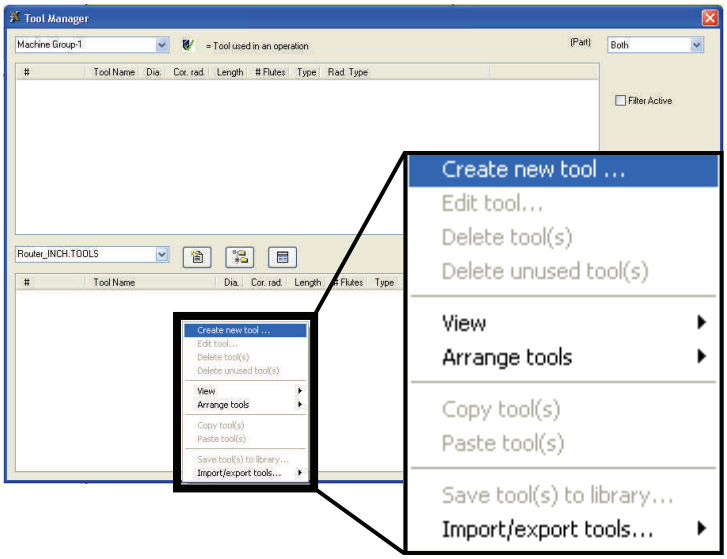


Creating tool in Mastercam®

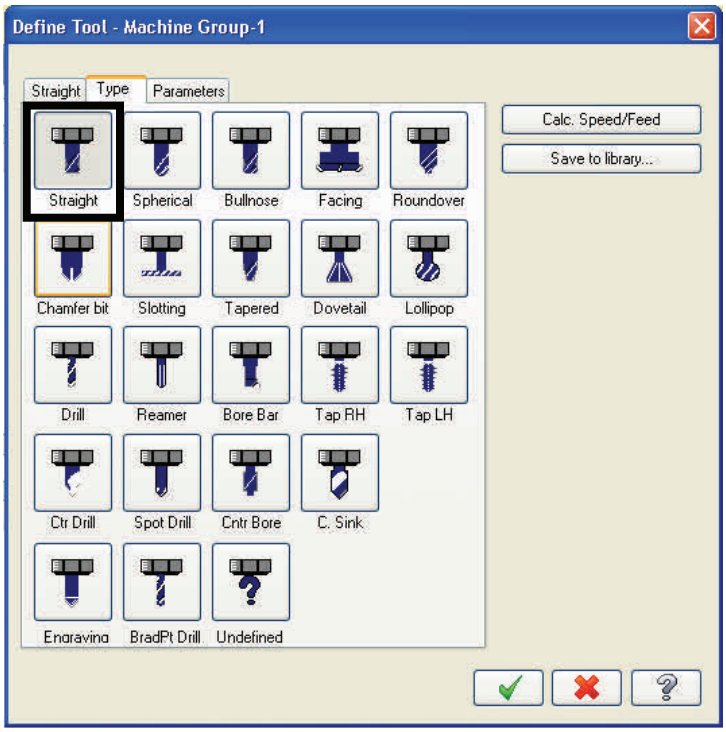
1. Within Mastercam® go to the Toolpaths pull down menu, then left click on Tool Manager.

Create Tool Library

2. Right click with the mouse in the bottom white box and left click on Create new tool.

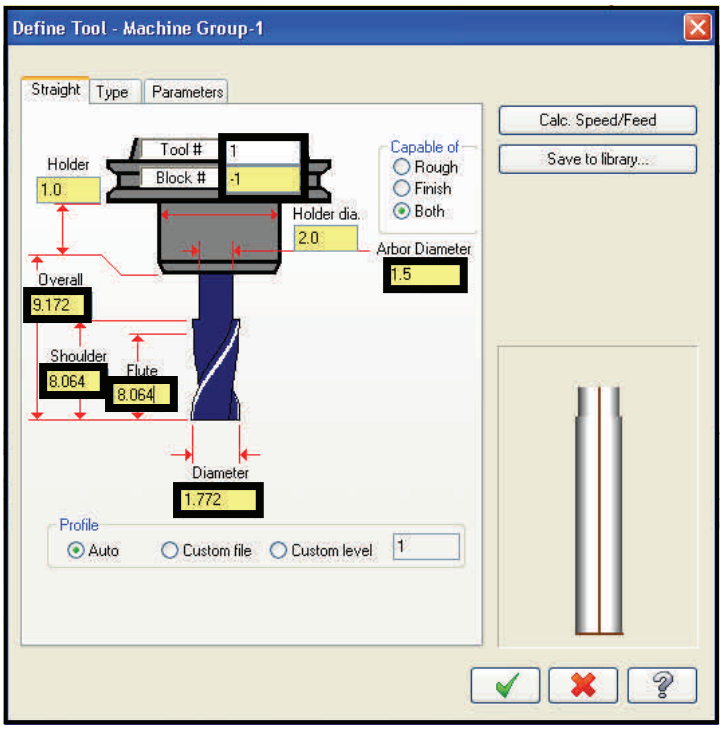


3. Left click on the picture of the “Straight” mill bit, to select the tool type.

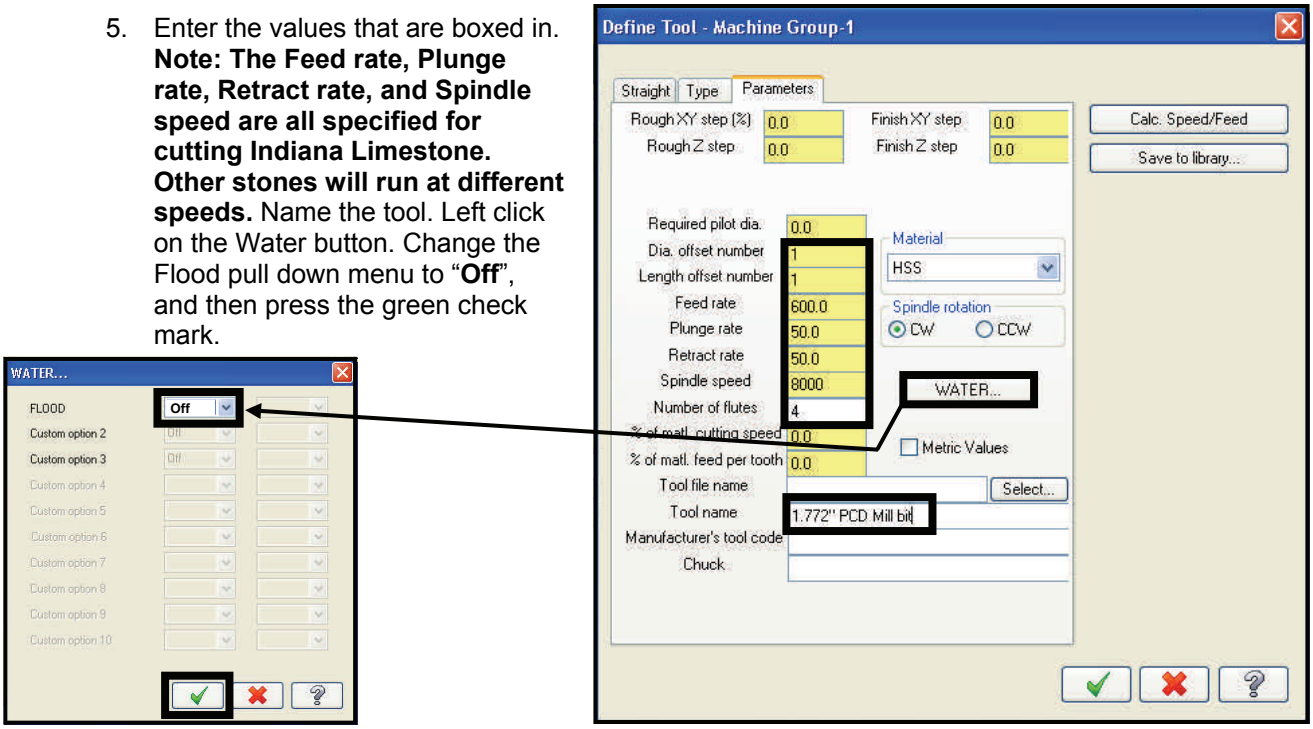


Create Tool Library

4. Enter the values that are boxed in. Match the Shoulder with the Flute length. The Arbor Diameter from a Terminator 1 3/4" PCD mill bit is 1.5". The Tool # refers to which tool pocket location (1-6) the tool will be located on the VEKTOR. The holder and holder diameter are optional to run this tool with Mastercam®. The remaining options are all default from an original install of Mastercam® and do not need to be set.

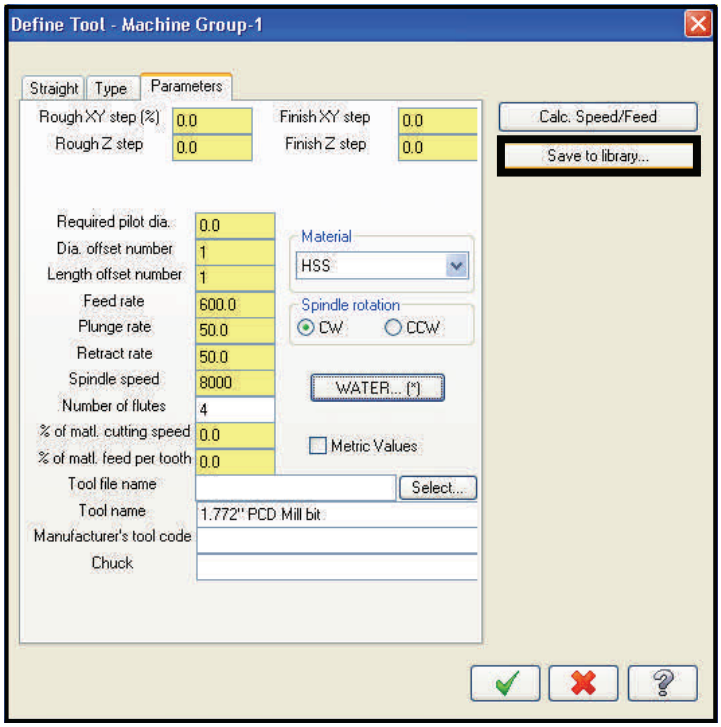


5. Enter the values that are boxed in. **Note: The Feed rate, Plunge rate, Retract rate, and Spindle speed are all specified for cutting Indiana Limestone. Other stones will run at different speeds.** Name the tool. Left click on the Water button. Change the Flood pull down menu to "**Off**", and then press the green check mark.

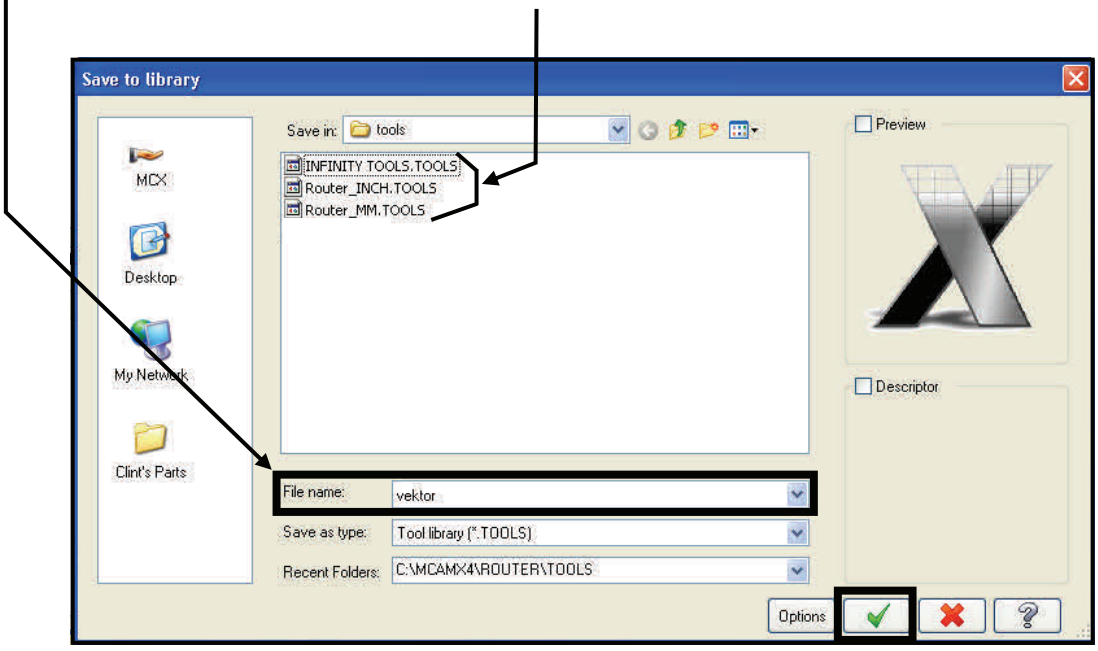


Create Tool Library

6. Left click the Save to Library button.

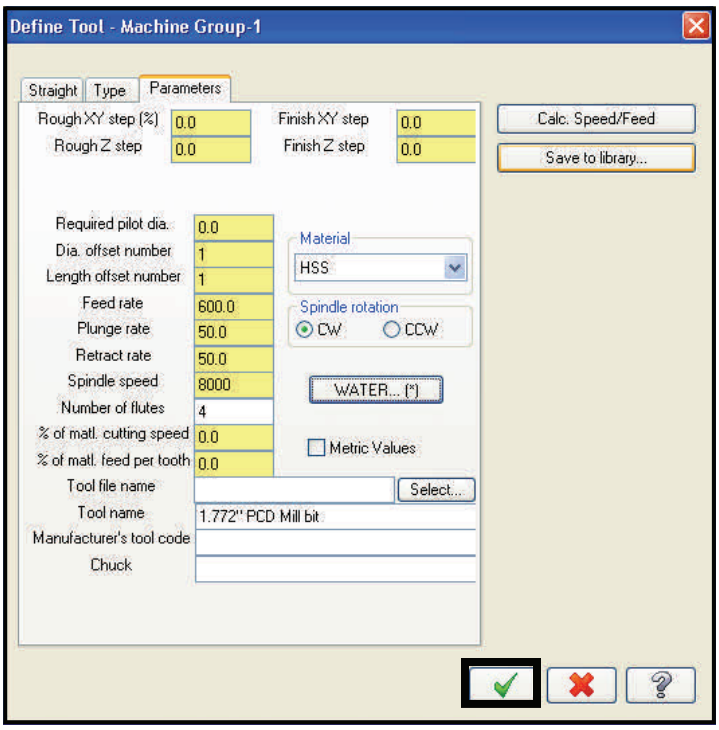


7. Left click in the “File name:” box, and type in VEKTOR to create a new tool library for this machine. This library will show up in the white portion of the below screen, the next time a tool is saved. Press the green check box to finish.
8 When another tool is created and saved, the programmer can left click on the desired library name that the tool is to be saved to, and then left click the green check box.

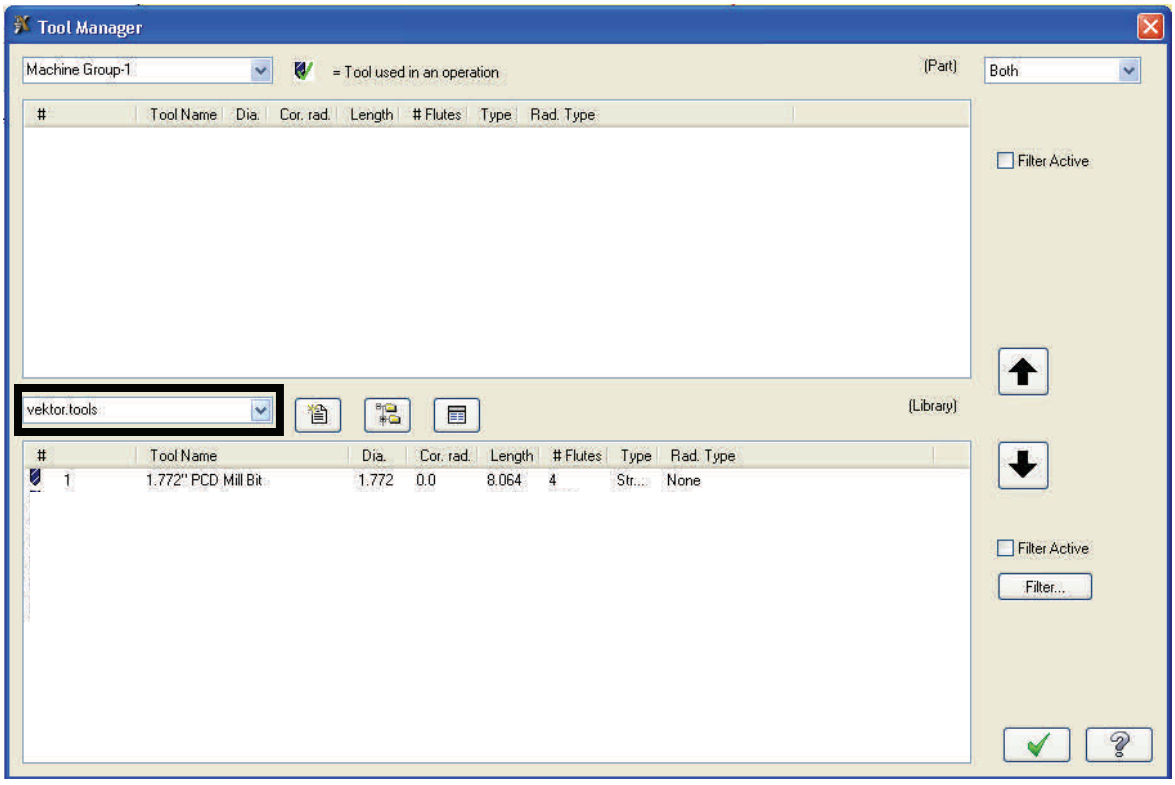


Create Tool Library

9. Left click the green check box on the Define Tool screen.
   Left click the green check box on the Tool manager screen.
   Go back to the Toolpaths pull down menu and left click on Tool manager..



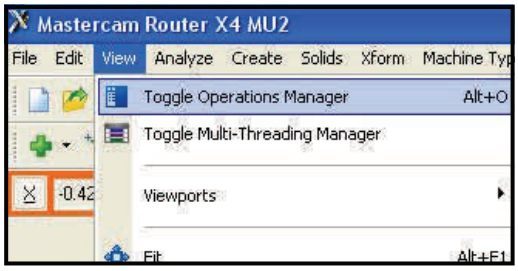
10. Left click the pull down to select the VEKTOR tools library. The newly created tool should show up, as below.

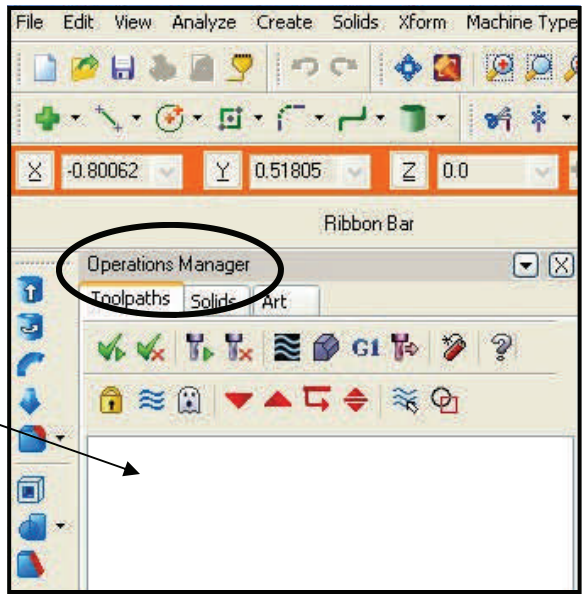


Creating Default Toolpaths

The previous section covered setting up a tool for use of the Router which consist of a milling bit. When a default tool is made, all information pertaining to that tool and process is saved in each of the type of cuts used, for example “Router - Swept 2D, Contour or Pocketing”. The default information will be used every time a toolpath is created. After picking the entities for a toolpath, a normal toolpath parameters box is displayed with the tool made as a default, with the “Water On”, the “Depth of Cuts” preset and “Keep tool down” chosen.

1. If the “Operations Manager” window is not displayed, click “Toggle Operations Manager” found in the “View” pull down menu.





The “Operations Manager” shown here does NOT have a machine group or any operations displayed. If a group appears in this area and does NOT have the word “VEKTOR” in the title, go to the next step.

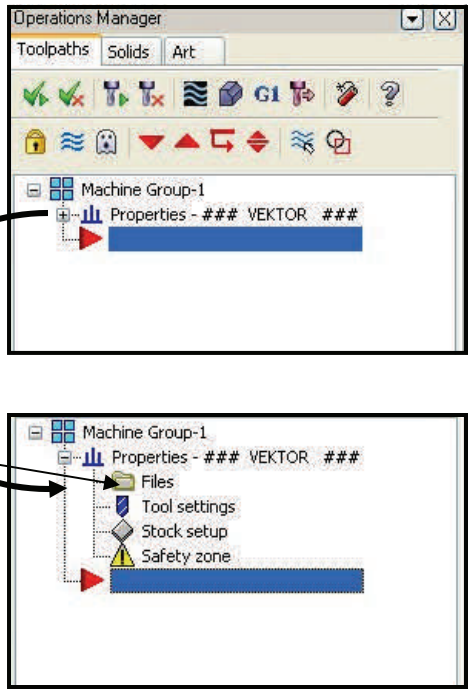
2. To load the machine group select “Machine Type” from the menu bar then “Router” and then “Default”.

Creating Default Toolpaths (cont'd)

3. Verify that a Machine Group with word "VEKTOR" after the Properties line now appear in the Operations Manger window.

4. Expand the "Properties" entry. (Single click + box.)

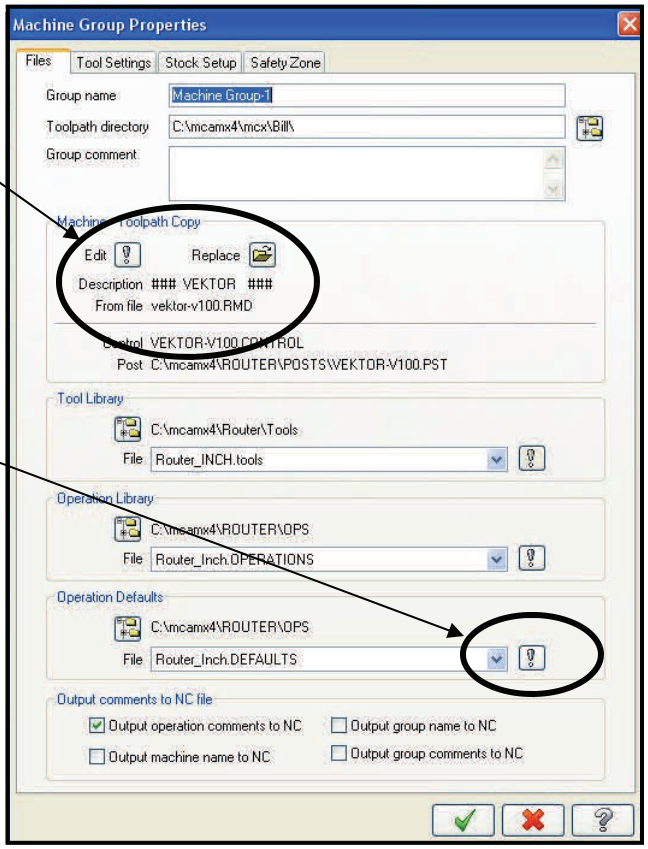
5. Open the "Files" folder> (Double click on the folder.)



Opening the "Files" folder launches the Machine Group Properties window.

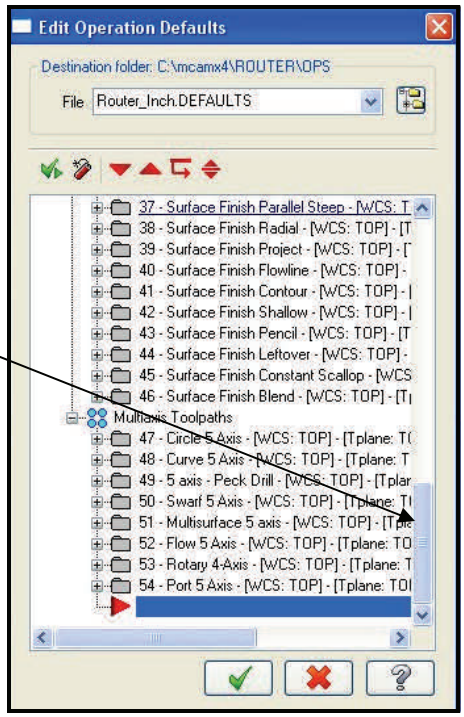
6. Verify that the word "VEKTOR" appears in the description line.

7. To edit operations click the open editor icon.



Creating Default Toolpaths – Contour (Ramp) (cont'd)

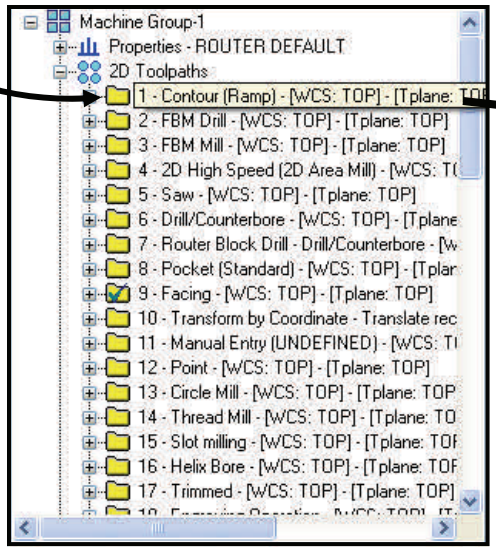
At this point, the Edit Operation Defaults window is displayed. This window, which lists all the tool operations, always opens displaying the last operation in the list.

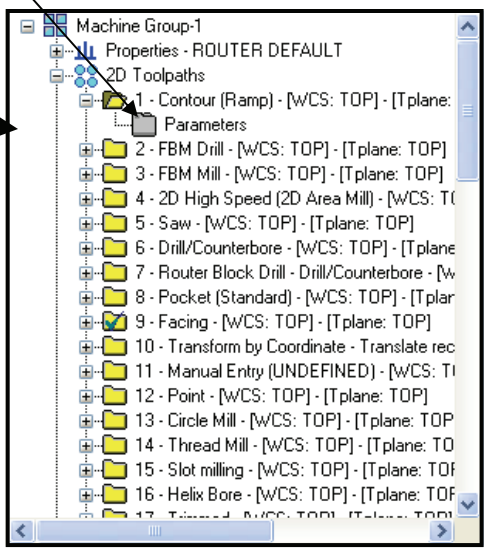


8. Use the scroll bar to display the first operation in the list.

9 Locate first operation - Contour (Ramp) Open the Contour (Ramp) folder (Double click on the folder.)

10. Open the "Parameters" folder (Double click on the folder.)





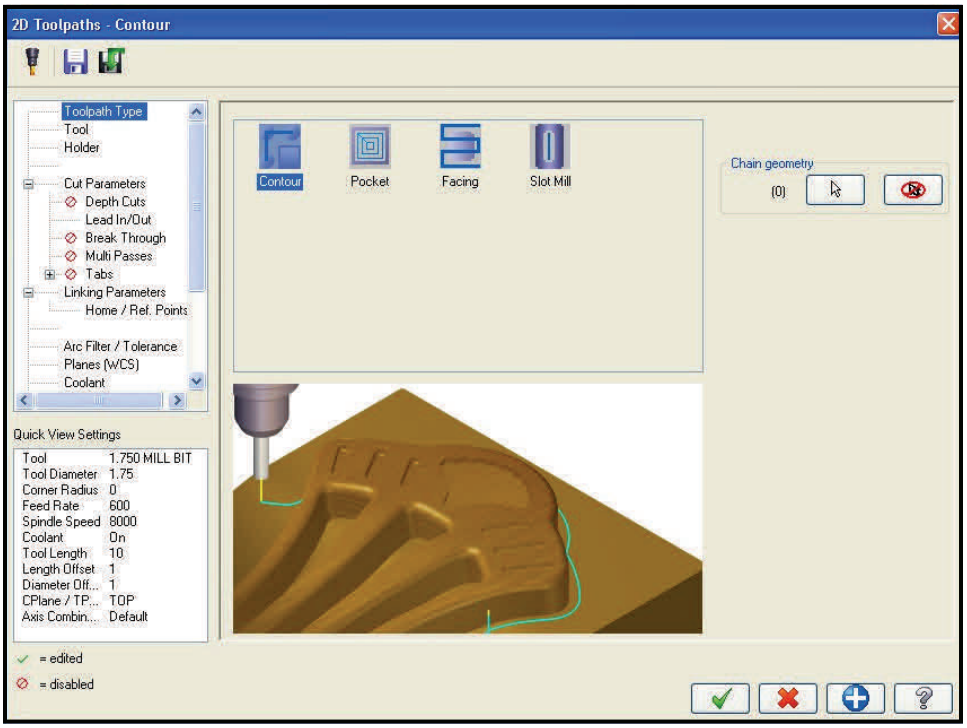
Creating Default Toolpaths – Contour (Ramp) (cont'd)

The next three pages illustrate the six Contour toolpath parameter setup screens.

Toolpath Type Screen

Verify the Contour icon is highlighted.

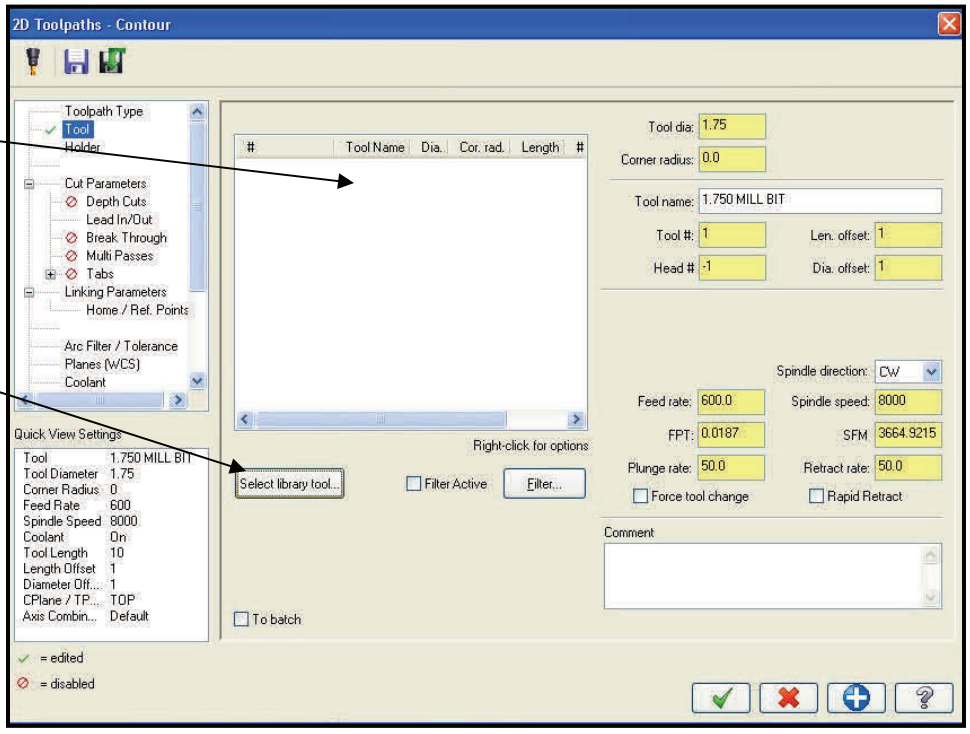
Click "Tool" for the next screen.



Tool Screen (without a tool loaded)

The first time this window is opened the tool list will be empty. A tool must be loaded from the library.

Click on the "Select library tool" to display the Tool Selection window.

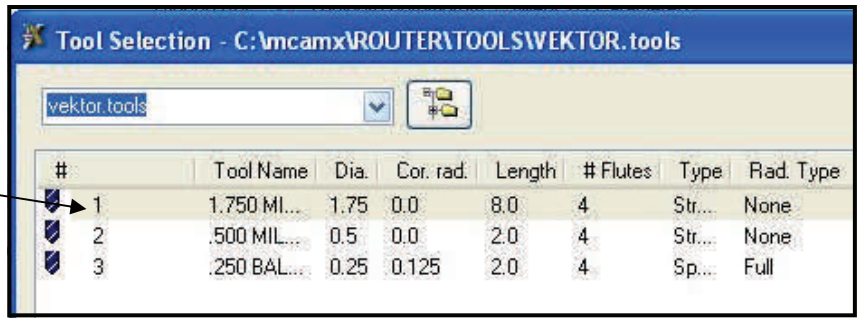


Creating Default Toolpaths – Contour (Ramp) (cont'd)

Tool Screen

Select "vektor tools" in the pull down menu to display a list of VEKTOR tools.

From the vektor tool list highlight/select tool # 1.

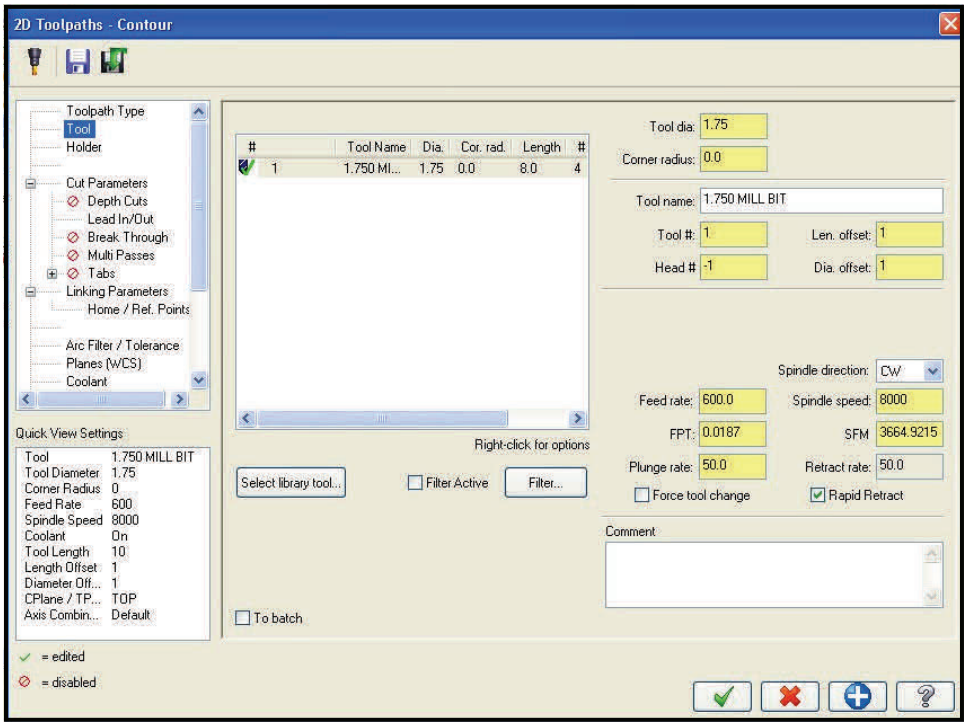


On the Tool Selection window click.

Tool Screen (with a tool loaded)

Enter the parameter values shown on this screen.

Click "Cut Parameters" to display the next screen.

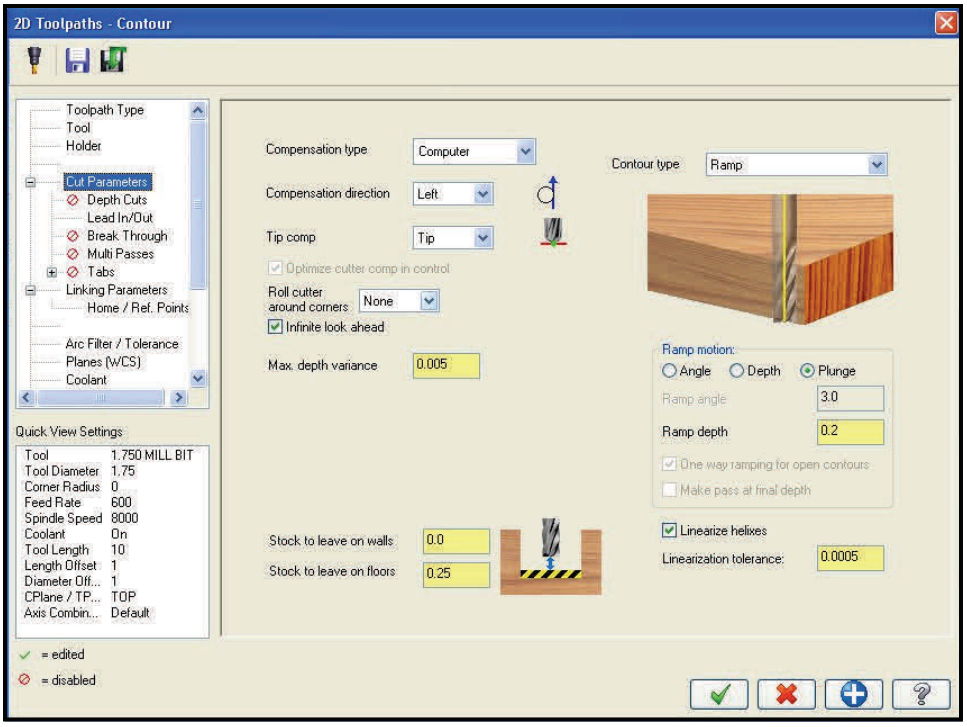


Creating Default Toolpaths – Contour (Ramp) (cont'd)

Cut Parameters Screen

Enter the parameter values shown on this screen.

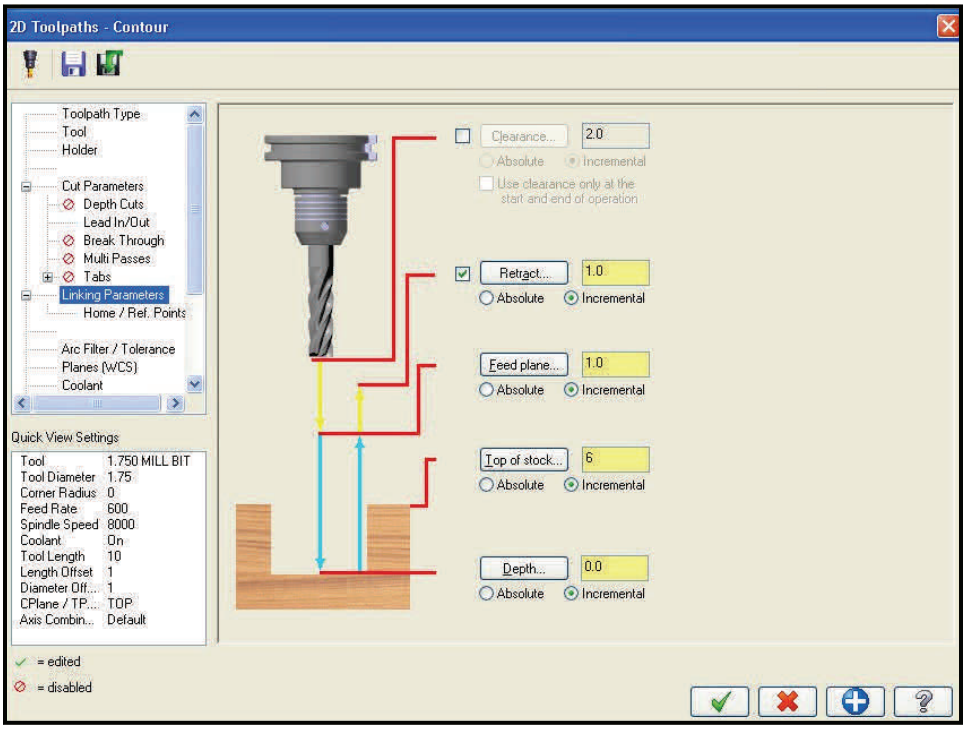
Click "Linking Parameters" to display the next screen.



Linking Parameters Screen

Enter the parameter values shown on this screen.

Click "Coolant" to display the next screen.

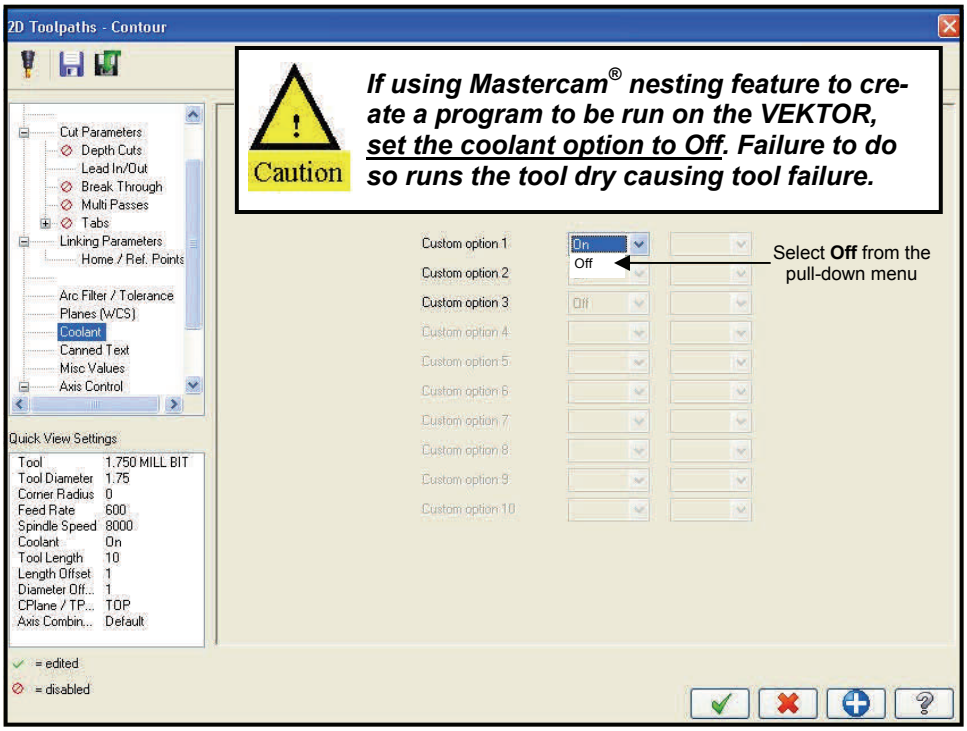


Creating Default Toolpaths – Contour (Ramp) (cont'd)

Coolant Screen

Set the option to **Off**.

Click green check mark when done.



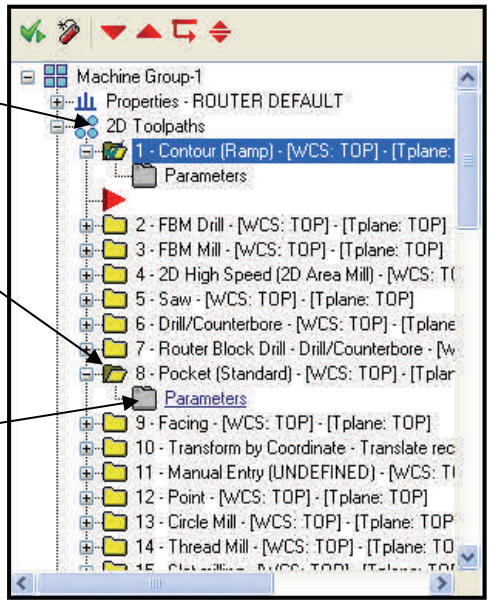
Creating Default Toolpaths – Pocket (Standard) (cont'd)

Repeat the same steps of the next toolpath.

Locate Pocket (Standard) in the 2D Toolpaths.

Open the Pocket (Standard) folder (Double click on the folder).

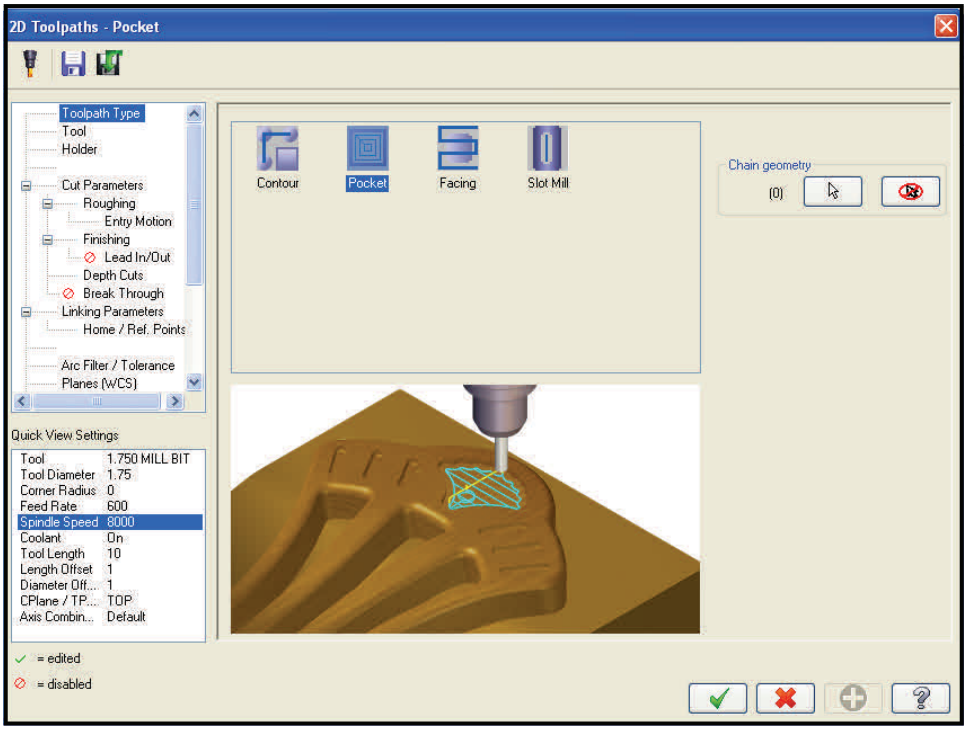
Open the "Parameters" folder (Double click on the folder).



Toolpath Type Screen

Verify the Pocket icon is highlighted.

Click "Tool" for the next screen.



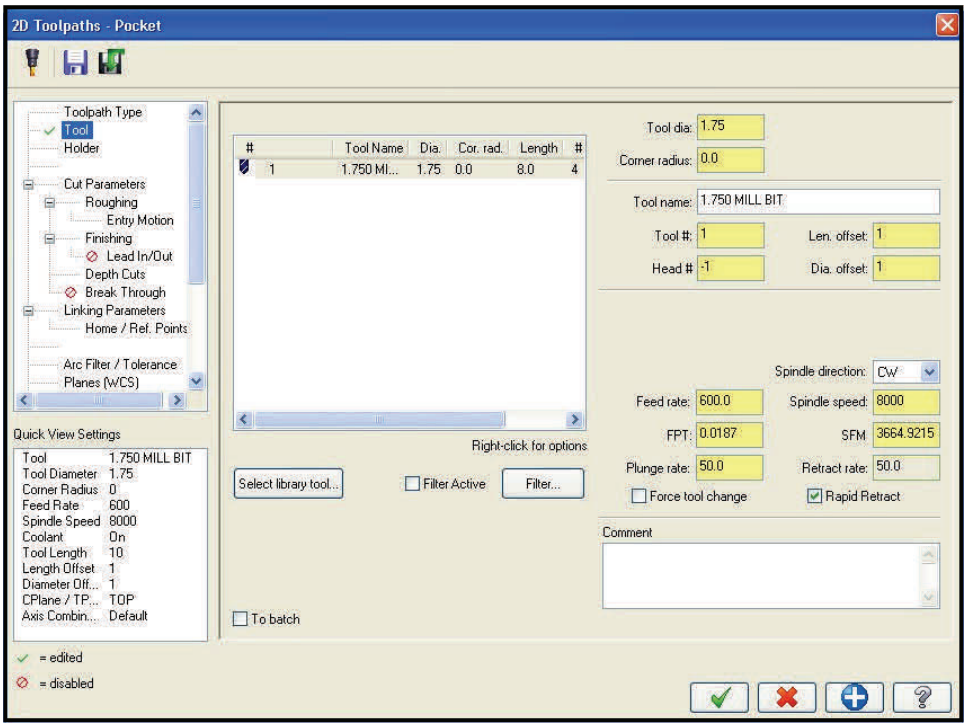
Creating Default Toolpaths – Pocket (Standard) (cont'd)

The next three pages illustrate the six Pocket toolpath parameter setup screens.

Tool Screen

Enter the parameter values shown on this screen.

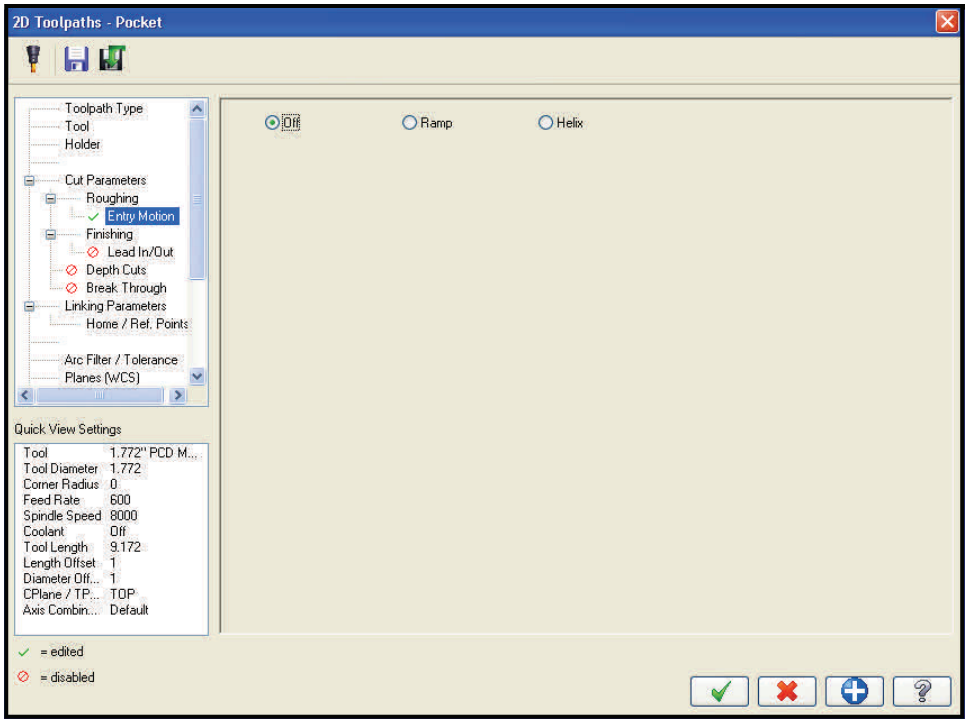
Click "Cut Parameters" to display the next screen.



Cut Parameters Screen

Enter the parameter values shown on this screen.

Click "Roughing" to display the next screen.

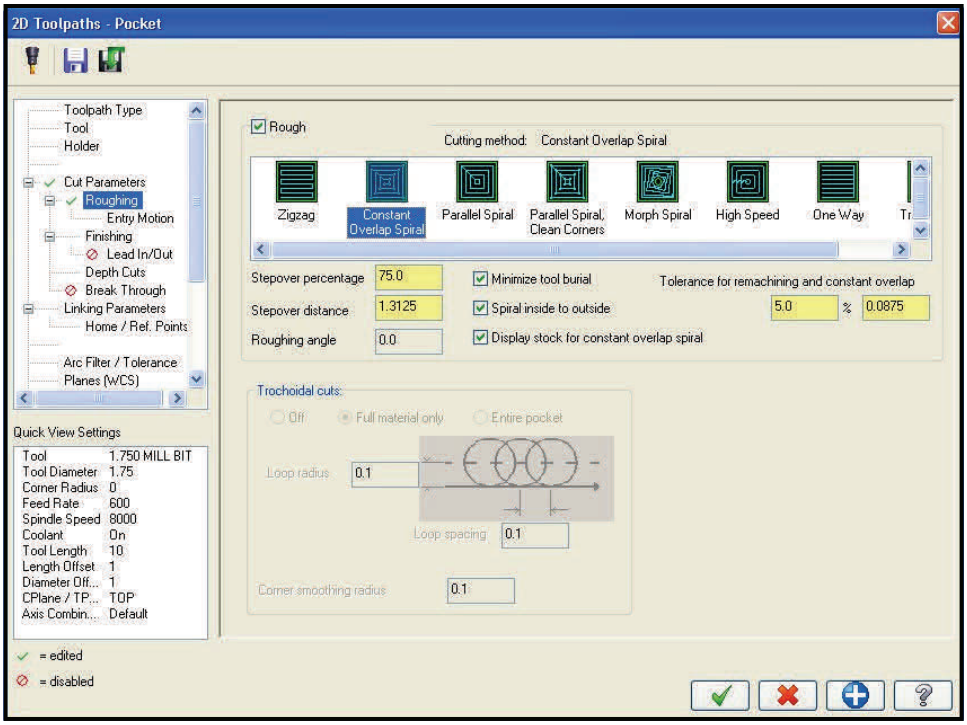


Creating Default Toolpaths – Pocket (Standard) (cont'd)

Roughing Screen

Enter the parameter values shown on this screen.

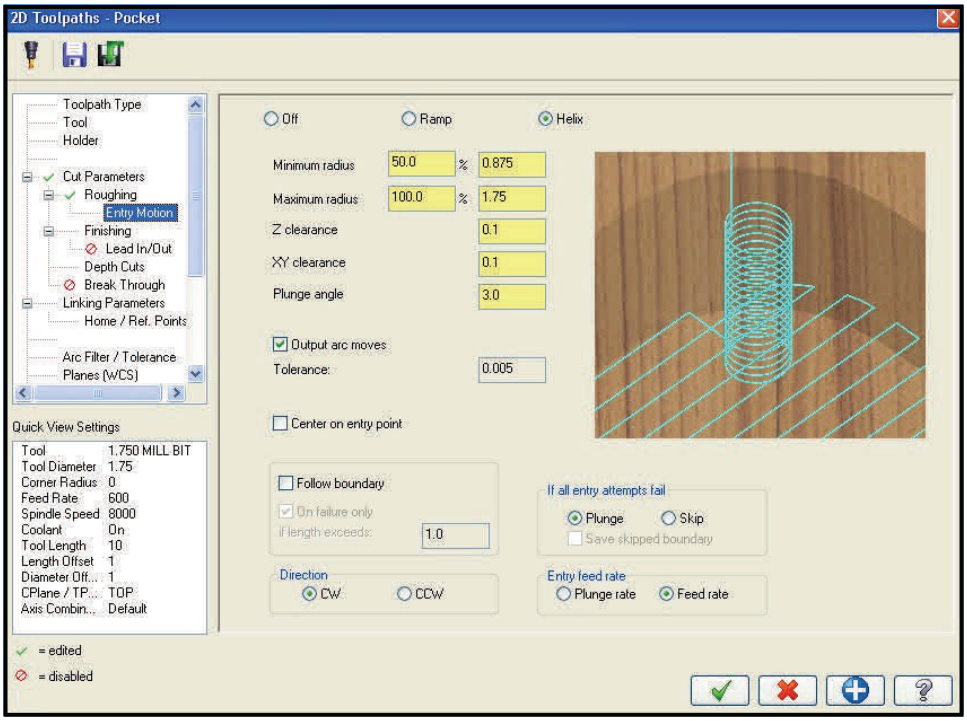
Click "Entry Motion" to display the next screen.



Entry Motion Screen

Enter the parameter values shown on this screen.

Click "Depth Cuts" to display the next screen.

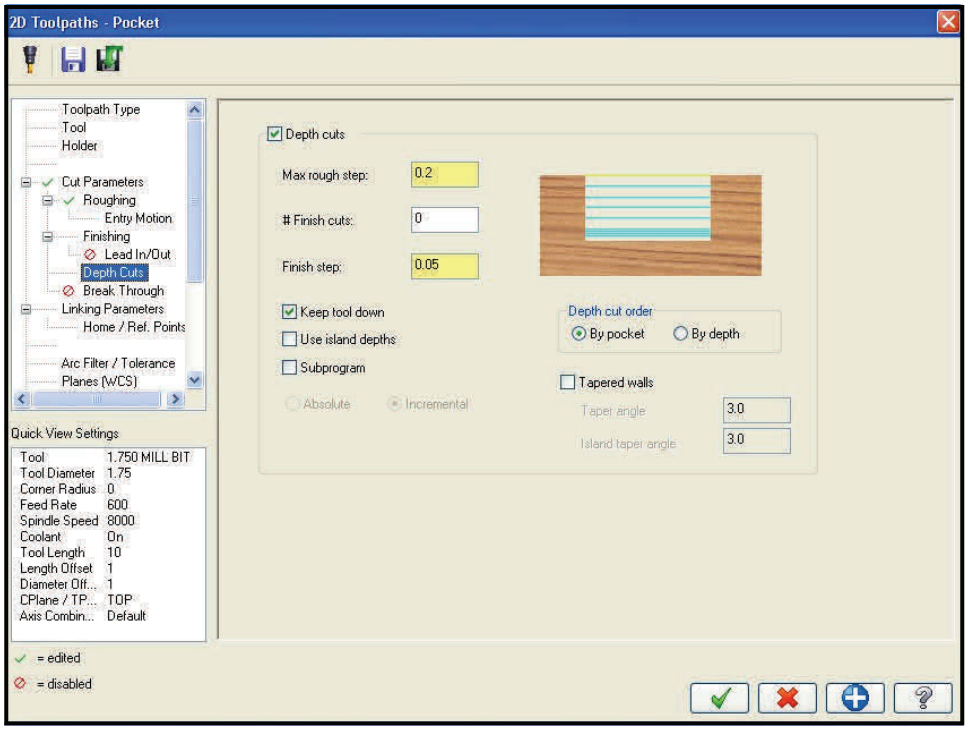


Creating Default Toolpaths – Pocket (Standard) (cont'd)

Depth Cuts Screen

Enter the parameter values shown on this screen.

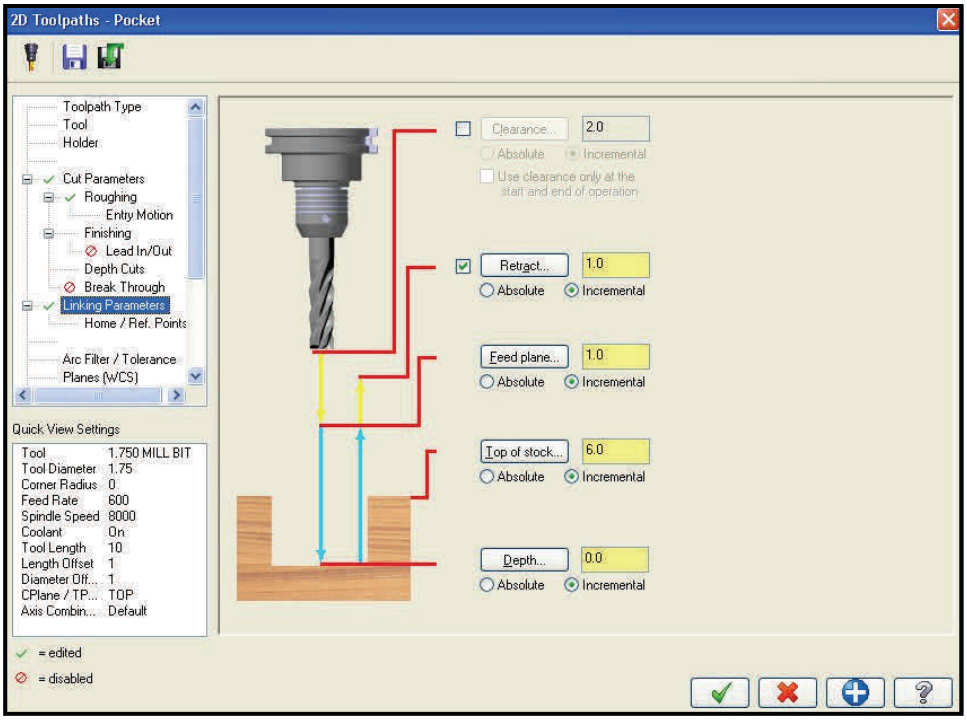
Click "Linking Parameters" to display the next screen.



Linking Parameters Screen

Enter the parameter values shown on this screen.

Click "Coolant" to display the next screen.

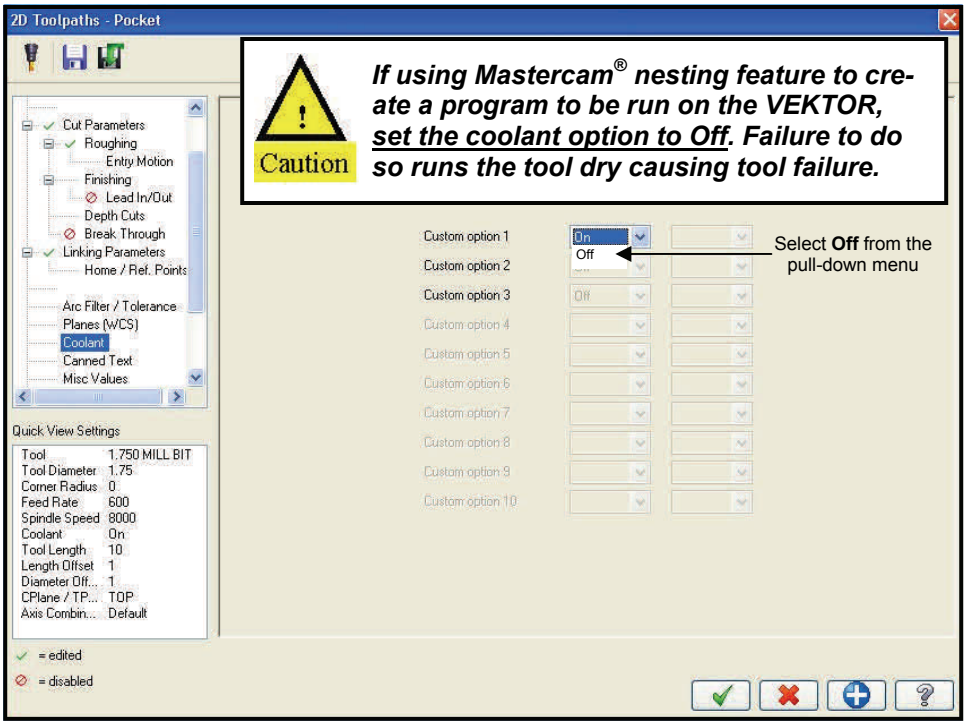


Creating Default Toolpaths – Pocket (Standard) (cont'd)

Coolant Screen

Set the option to **Off**.

Click green check mark when done.



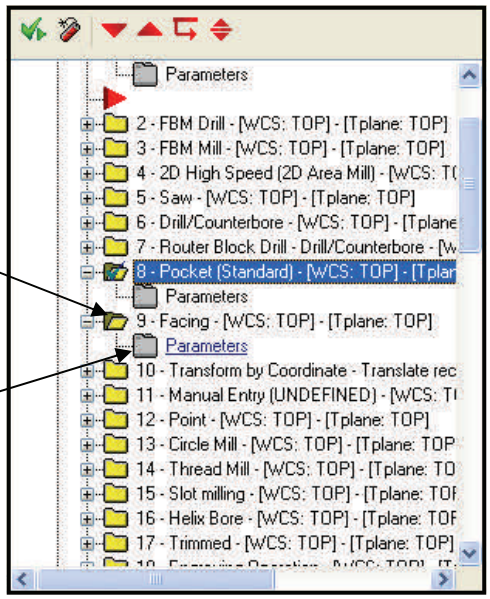
Creating Default Toolpaths – Facing (cont'd)

Repeat the same steps of the next toolpath.

Locate Facing (WCS: Top) in the 2D Toolpaths.

Open the Facing (WCS: Top) folder (Double click on the folder).

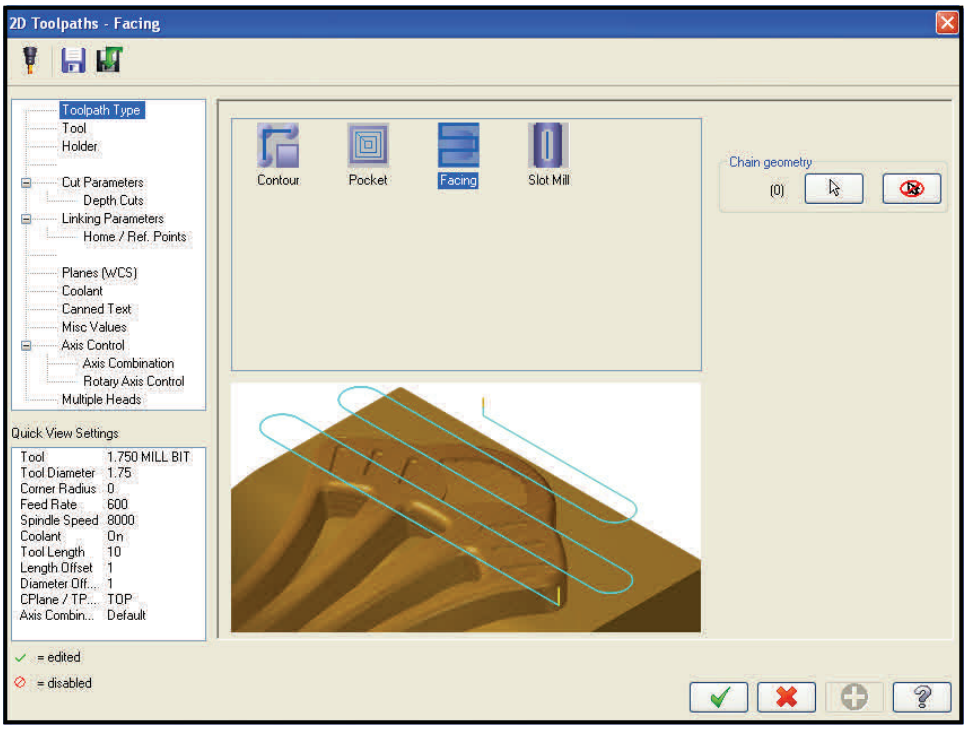
Open the "Parameters" folder (Double click on the folder).



Toolpath Type Screen

Verify the Facing icon is highlighted.

Click "Tool" for the next screen.

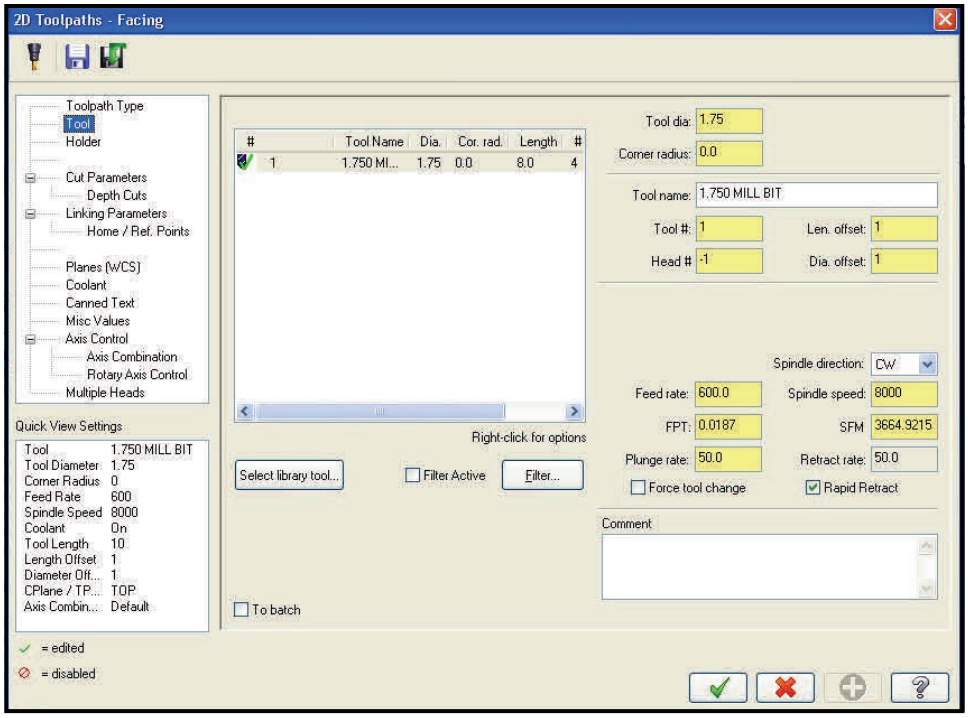


Creating Default Toolpaths – Facing (cont'd)

Tool Screen

Enter the parameter values shown on this screen.

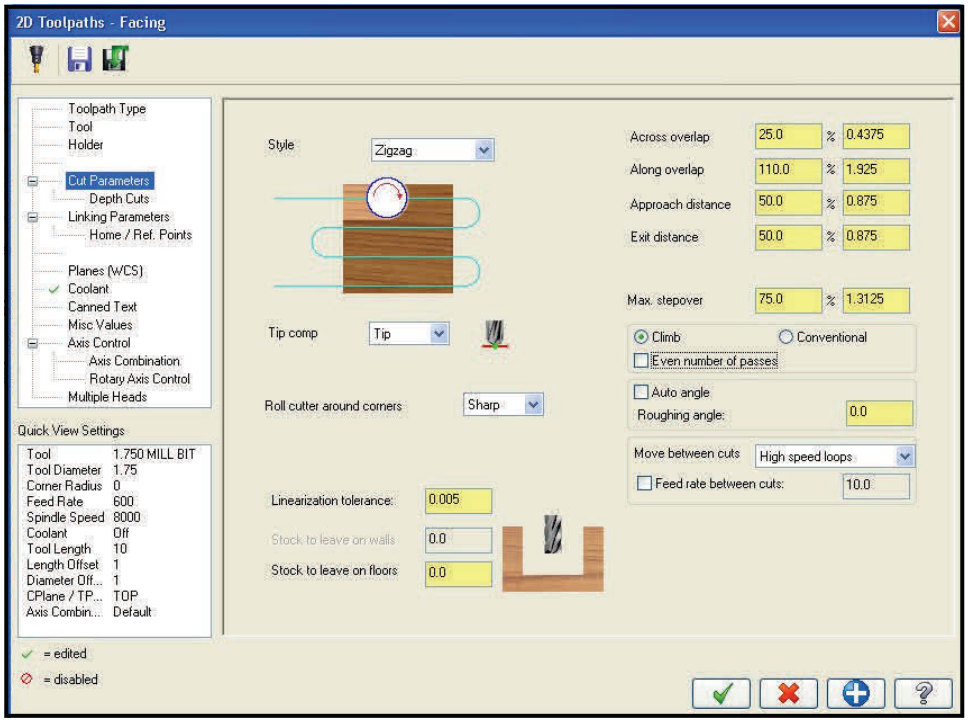
Click "Cut Parameters" to display the next screen.



Cut Parameters Screen

Enter the parameter values shown on this screen.

Click "Depth Cuts" to display the next screen.

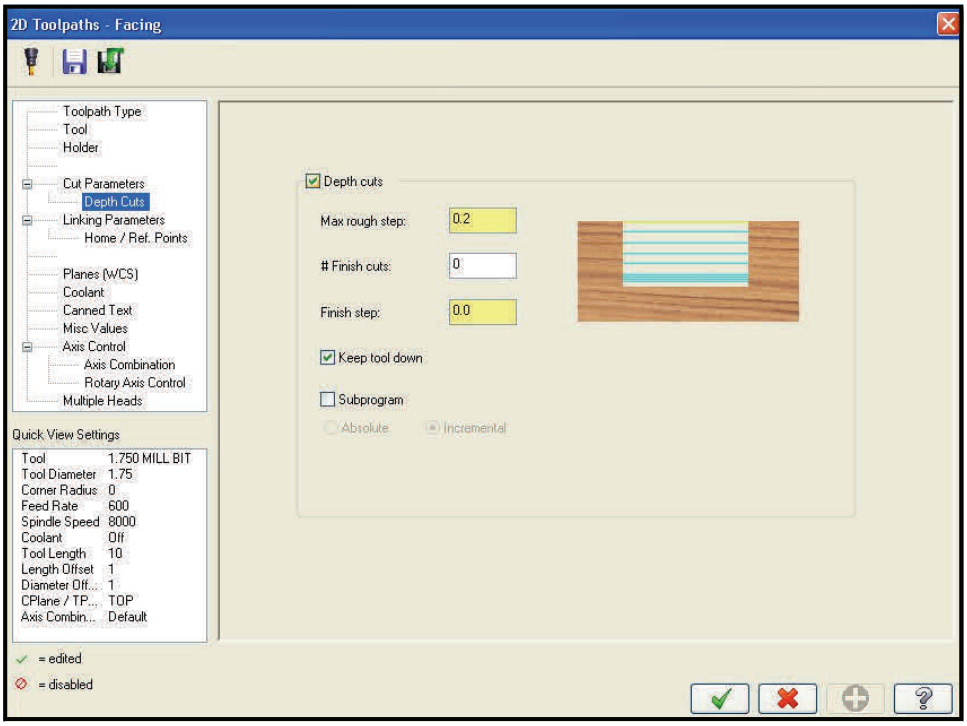


Creating Default Toolpaths – Facing (cont'd)

Depth Cuts Screen

Enter the parameter values shown on this screen.

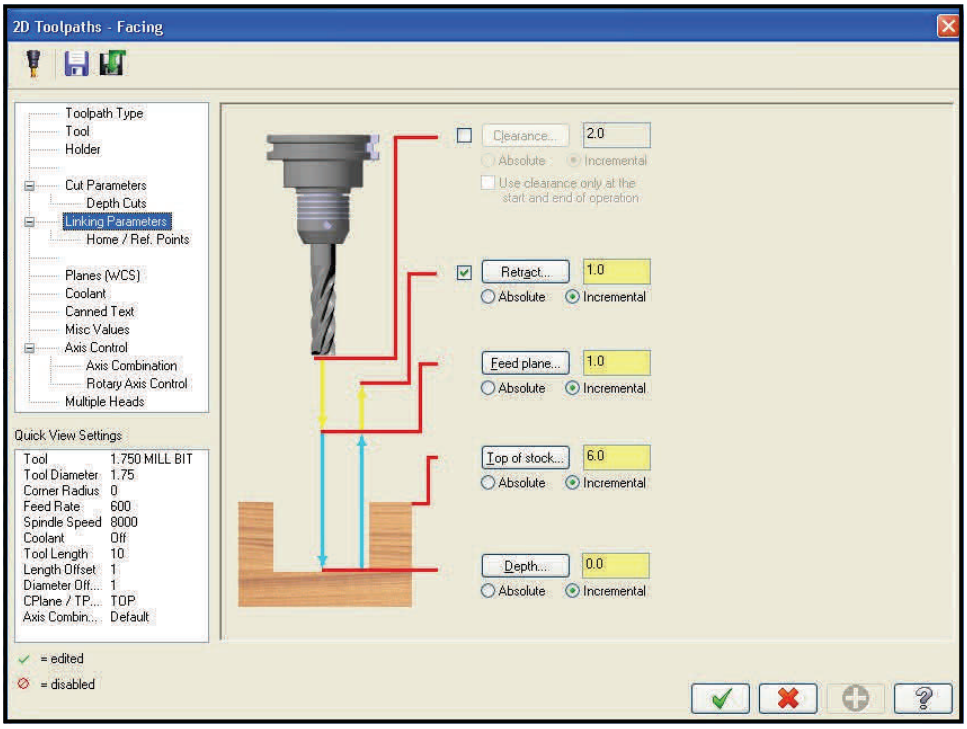
Click "Linking Parameters" to display the next screen.



Linking Parameters Screen

Enter the parameter values shown on this screen.

Click "Coolant" to display the next screen.

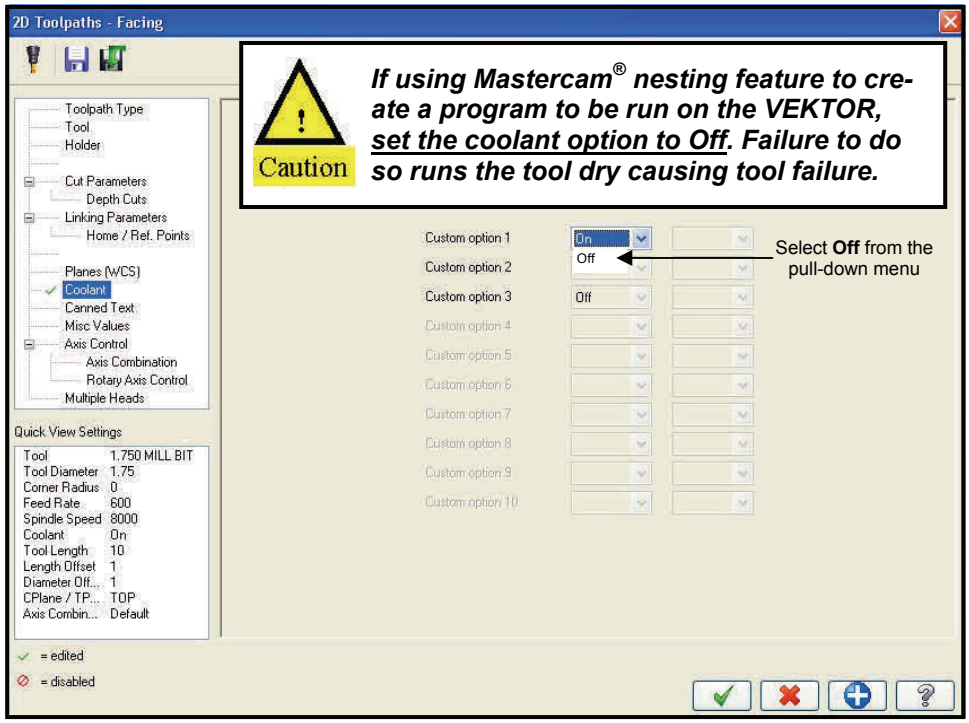


Creating Default Toolpaths – Facing & Swept 2D (cont'd)

Coolant Screen

Set the option to **Off**.

Click green check mark when done.

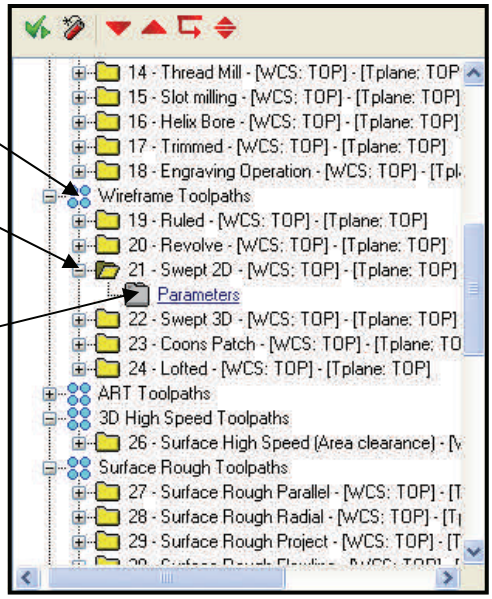


Repeat the same steps of the next toolpath.

Locate Swept 2D (WCS: Top) in the Wireframe Toolpath.

Open the Swept 2D (WCS: Top) folder (Double click on the folder).

Open the "Parameters" folder (Double click on the folder).

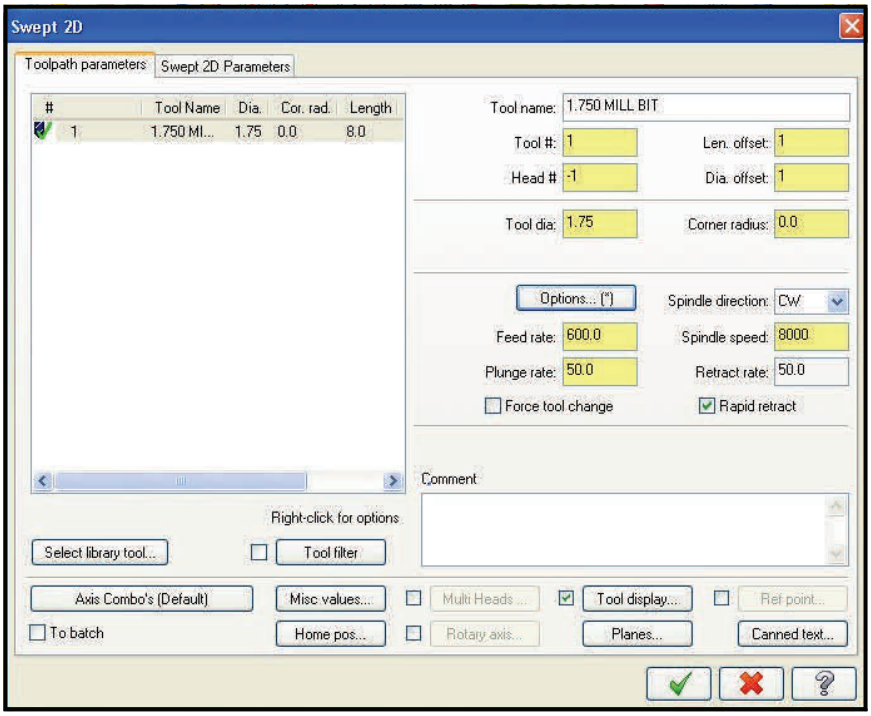


Creating Default Toolpaths – Swept 2D (cont'd)

Toolpath Parameters Screen

Enter the parameter values shown on this screen.

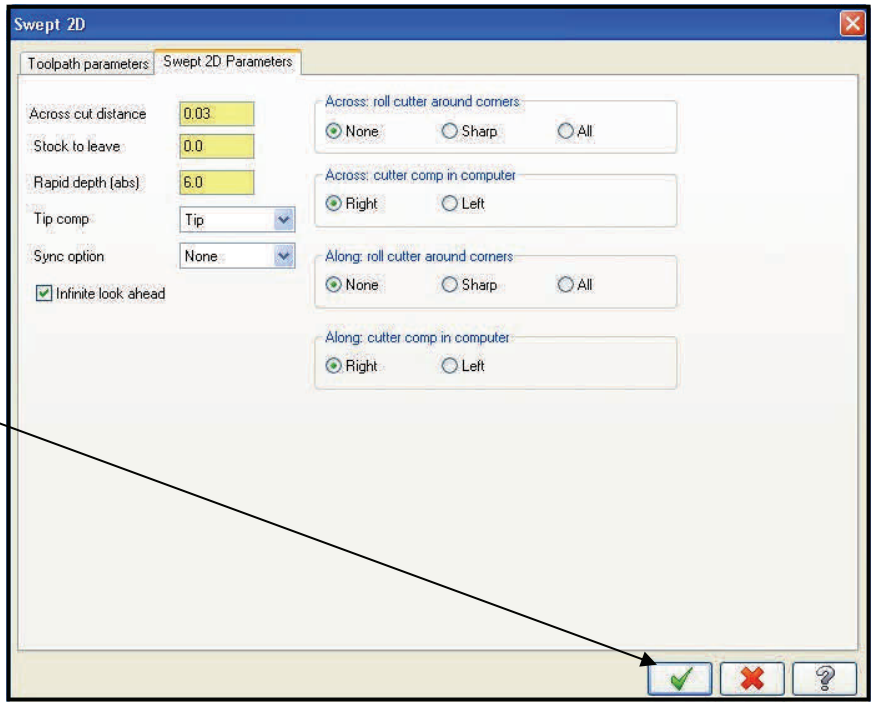
Click Swept 2D Parameters tab to display the next screen.



Swept 2D Parameters Screen

Enter the parameter values shown on this screen.

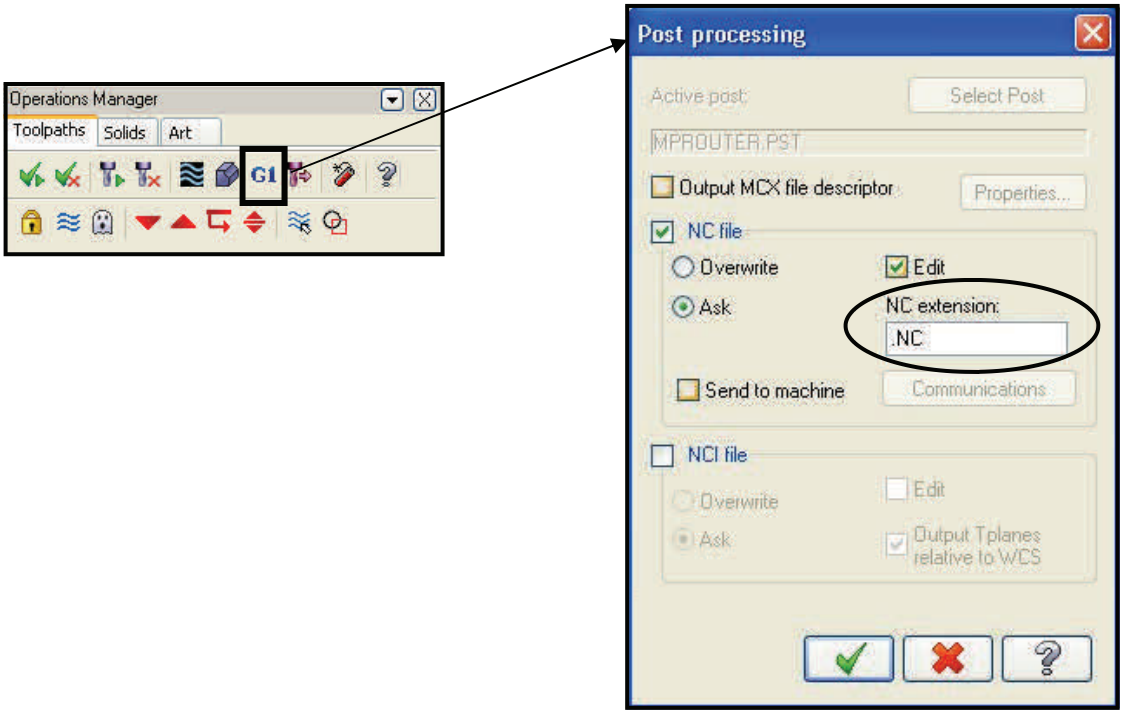
Click green check mark.



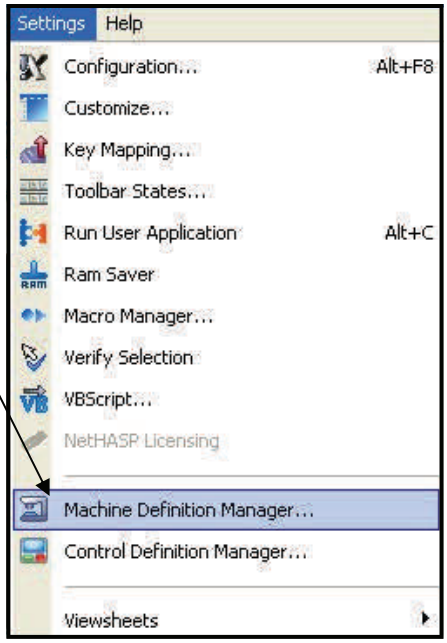
Changing Default Posting Settings

Posting or generating the G code is done by clicking the G1 button in the Operations Mangers menu. This launches the Post processing window.

G code files are save as .txt files. When Mastercam® is installed the default NC extension is not set to .txt (see below example). This procedure lists the steps to change the default NC extension to .txt.

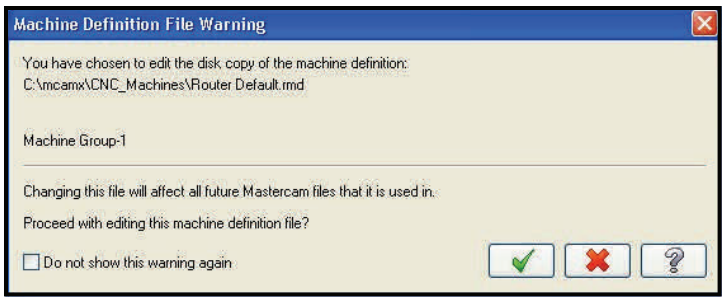


1. From the Menu Bar select Settings→Machine Definition Manager

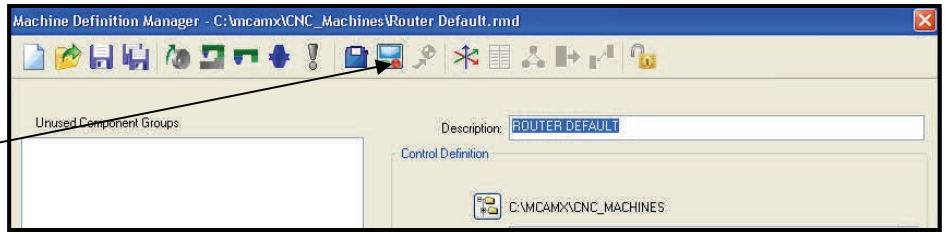


Changing Default Posting Settings (cont'd)

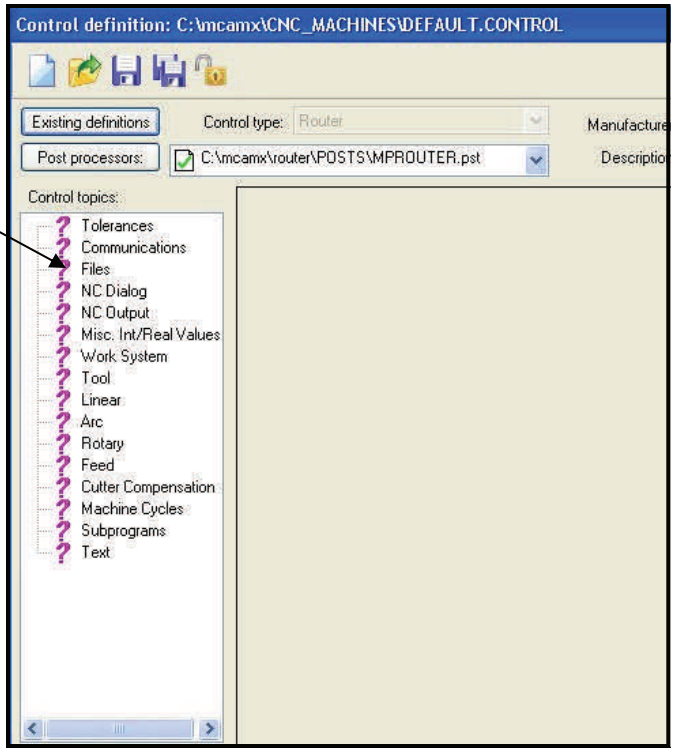
2. On this warming dialog box left click



3. Single left click "Edit the Control Definition" icon.



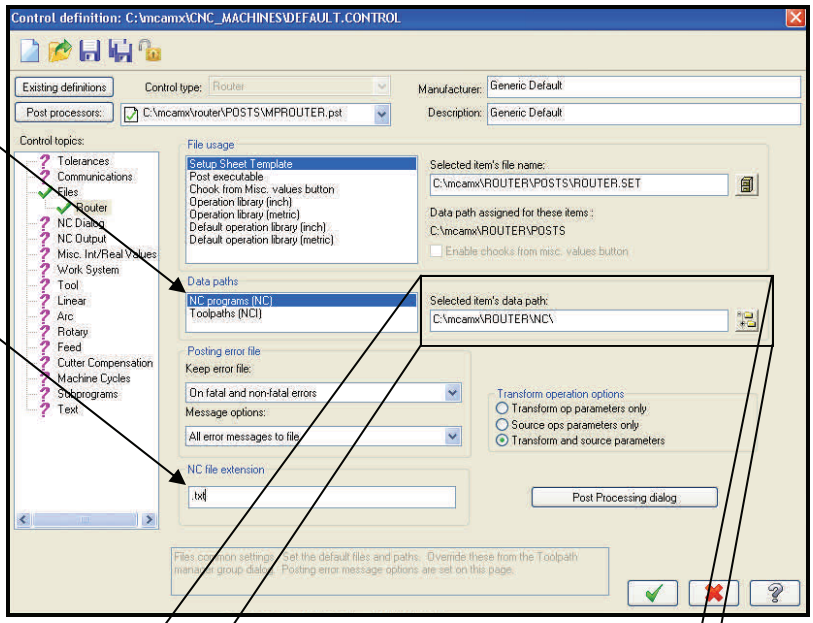
4. On the Control Definition screen single click on "Files"



Changing Default Posting Settings (cont'd))

5. Verify "NC programs (NC)" is highlight. If not, single click on the entry.

6. Change NC File Extension entry to .txt



During a Post operation, this field determines where the file is stored. In most shops, the files are stored on the company server, or mapped machine drive. Defining the path in this field eliminates the step of selecting the path each time a file is posted.

7. Click the "Browse For Folder" icon to open the browse dialog box.
8. Find and select the default location to store Posted files.
9. When the desire path has been selected, click 'OK".

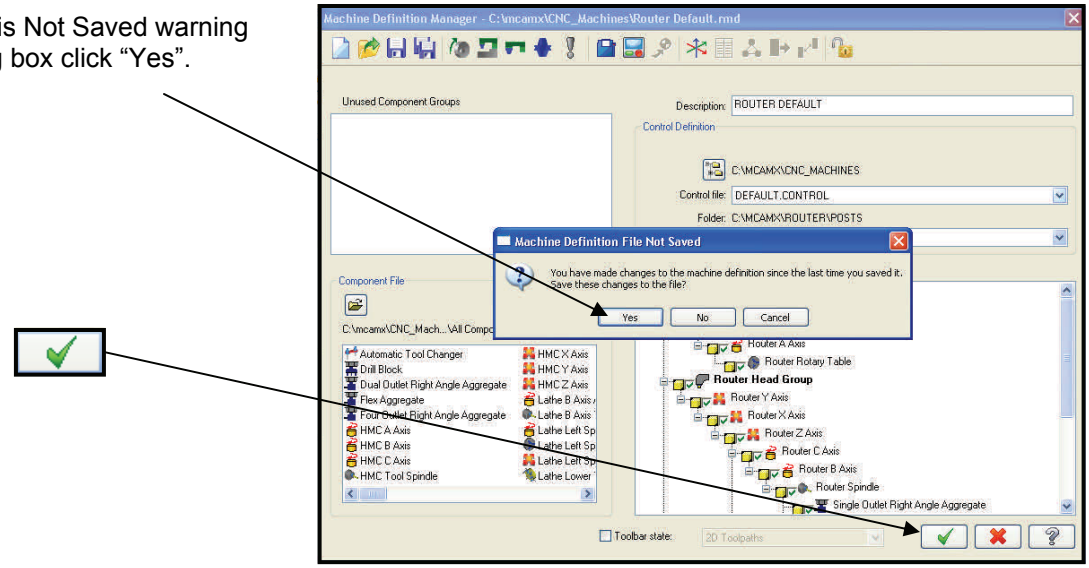
10. Click

11. When this warning dialog box appears, click "Yes".

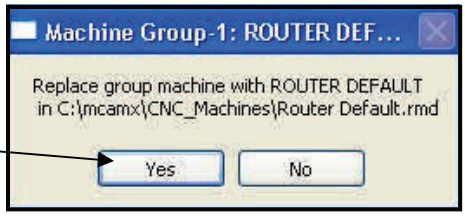
Changing Default Posting Settings (cont'd)

12. On this Not Saved warning dialog box click "Yes".

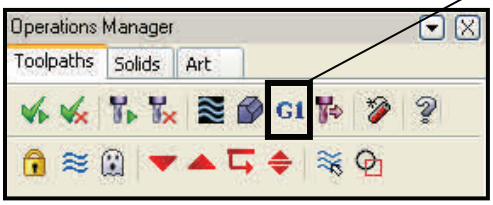
13. Click

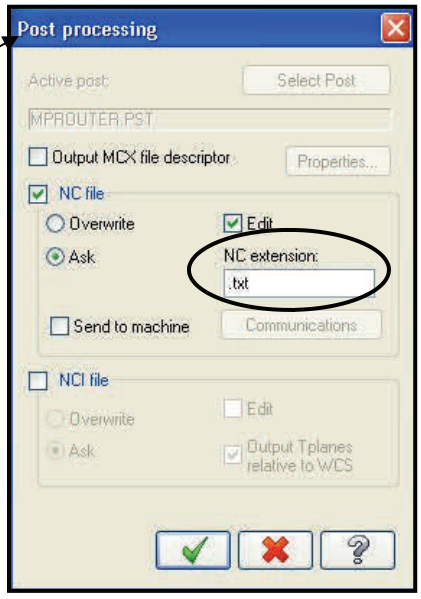


14. On the Replace Group Machine warning dialog box click "Yes".



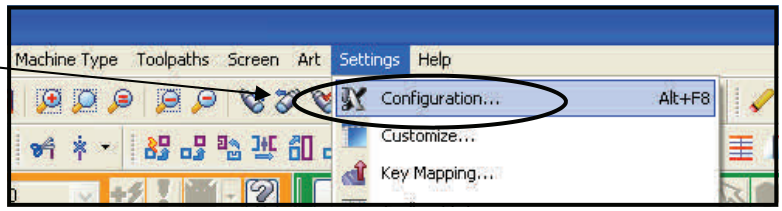
Verify NC Extension on the Post Processing window now read .txt. Click G1 button on the Operations Manage menu to display the Post Processing window.



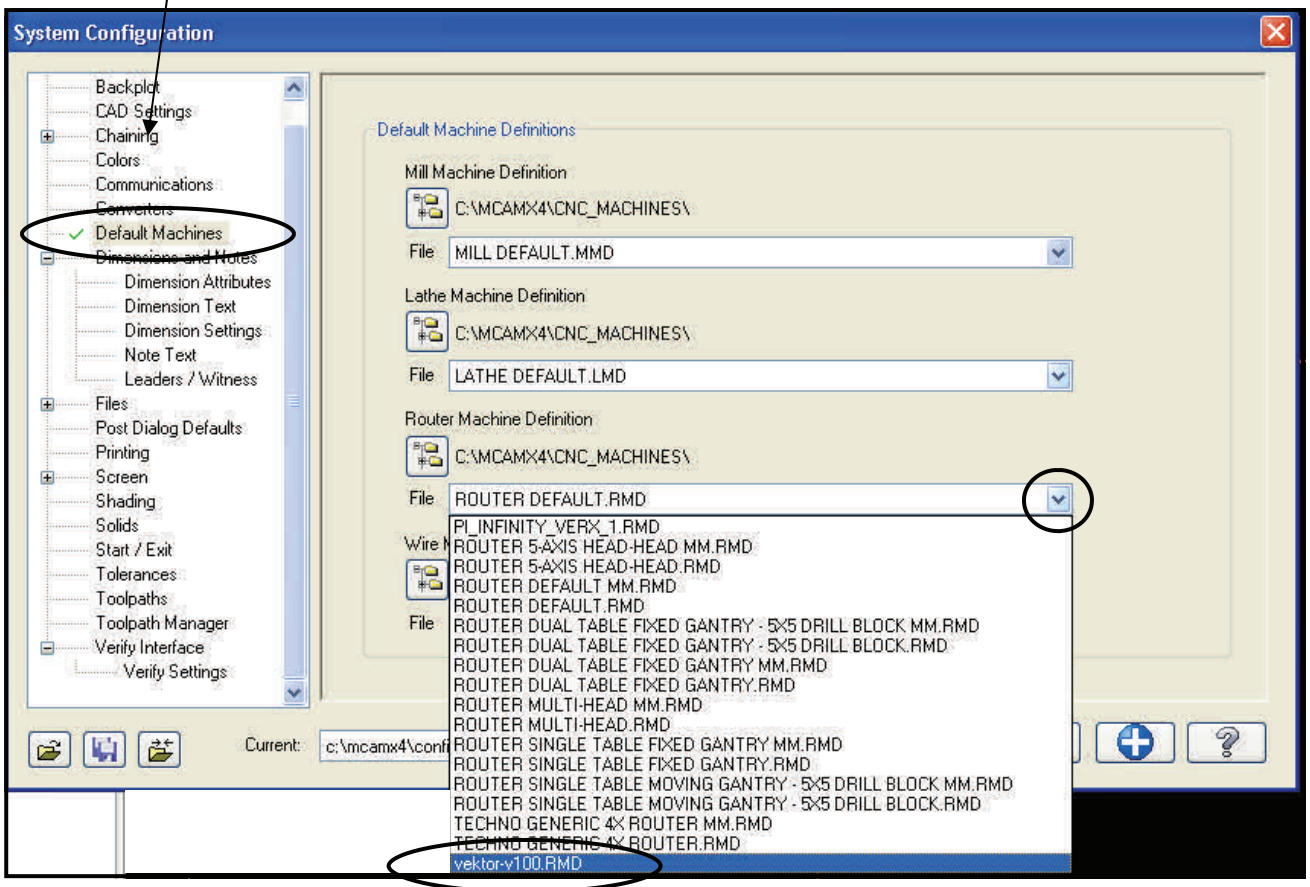


Setting the Default Machine

1. Go to “Settings”, then “Configuration”.

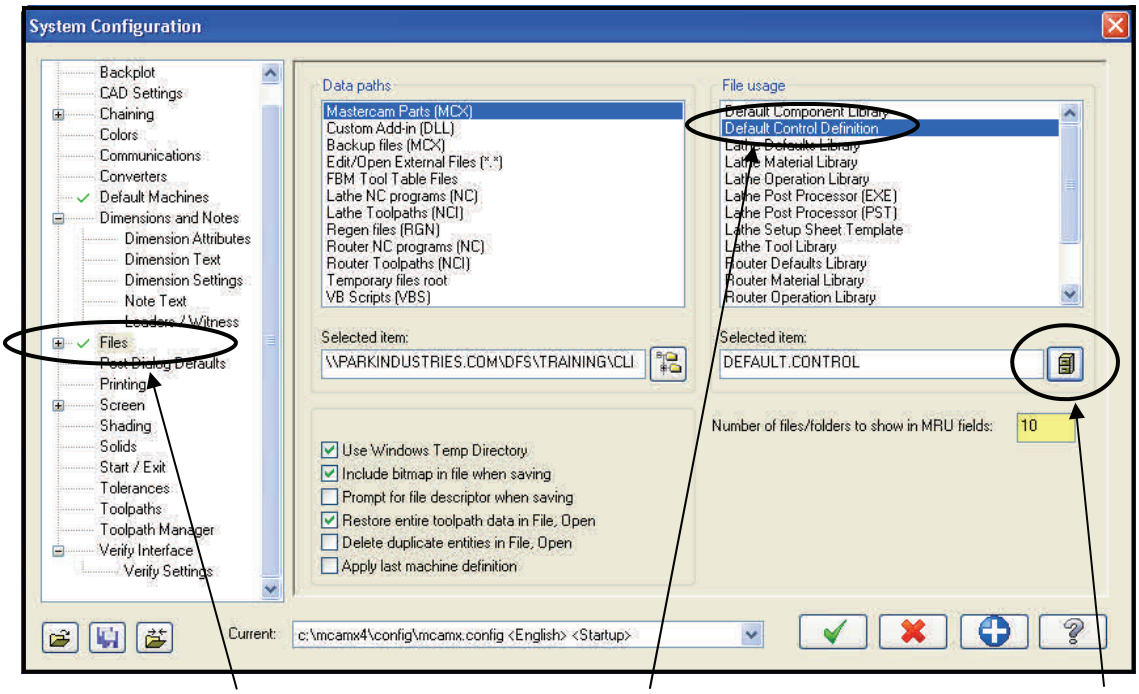


2. Left click on “Default Machines”.

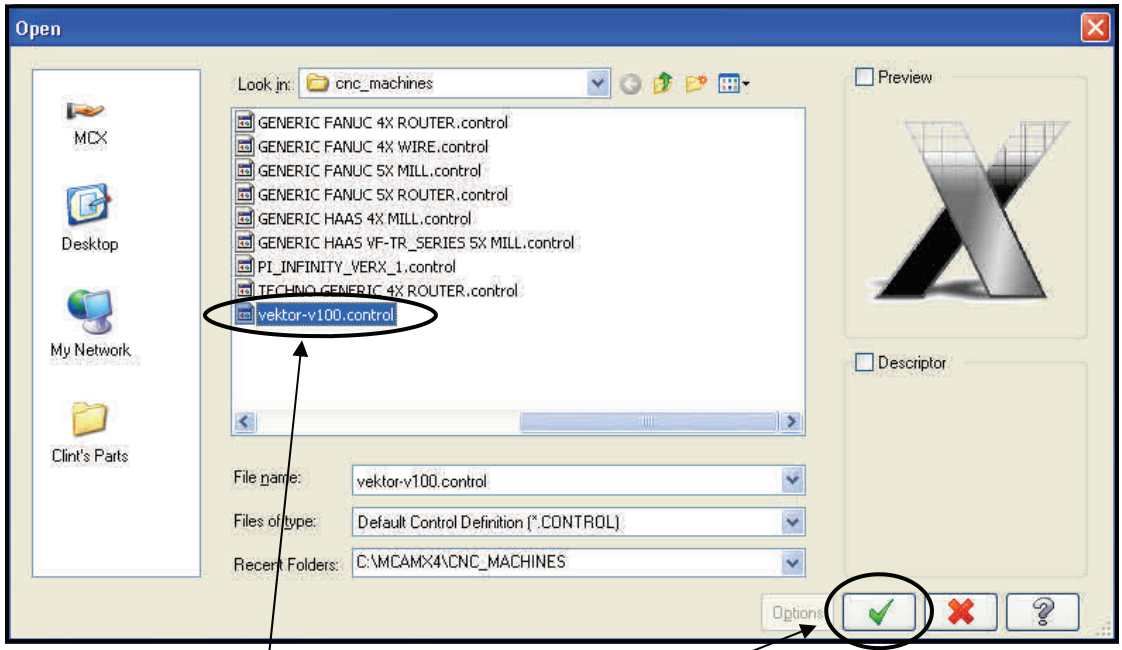


3. Left click on the blue pull down arrow and click on “vector -v100.RMD”.

Setting the Default Machine (cont'd)



4. Left click on "Files", then left click on "Default Control Definition", then left click on the "Select" button.

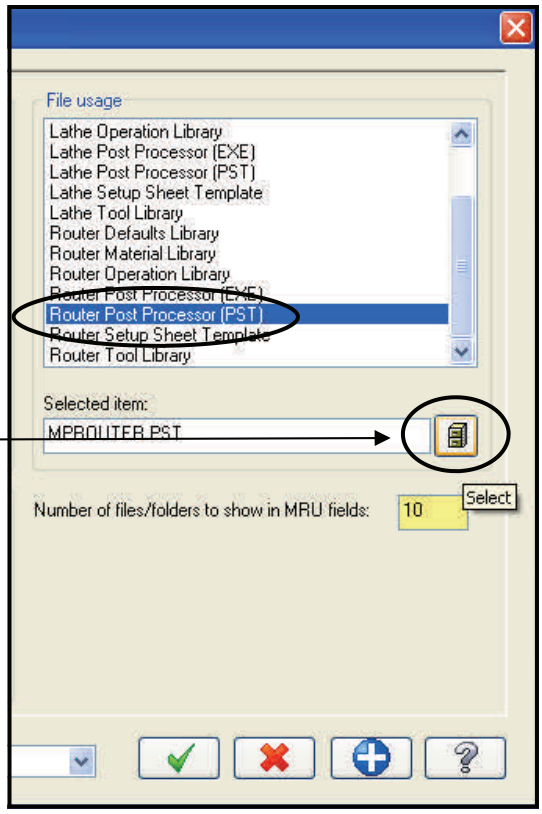


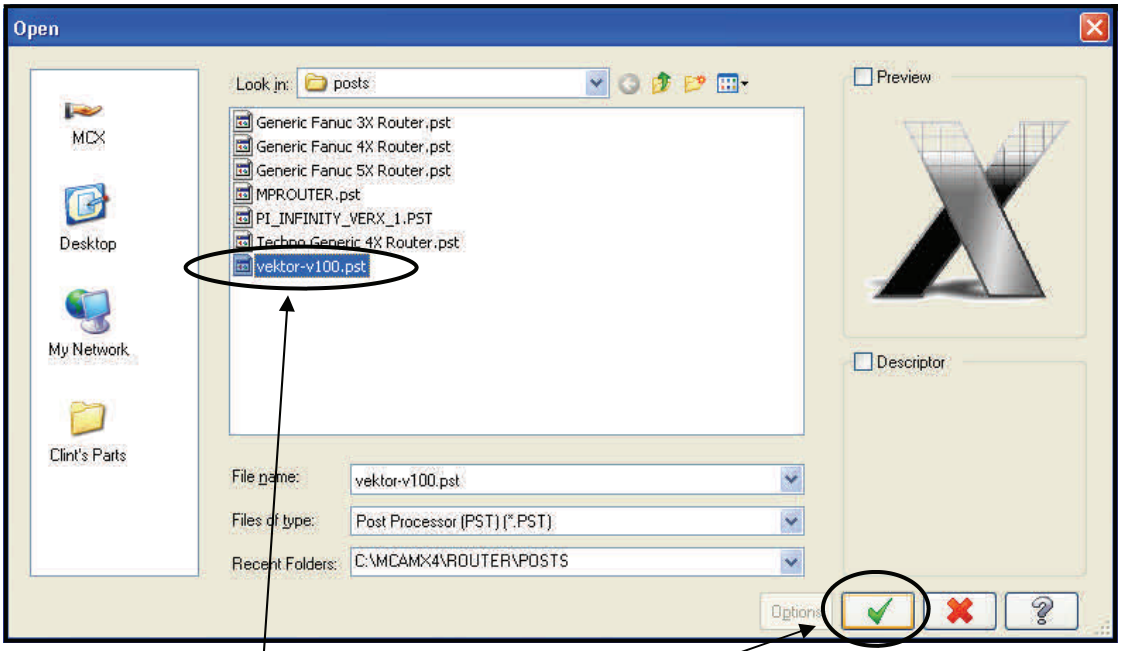
5. Left click on "vector-v100.control", then left click on "OK".

Setting the Default Machine (cont'd)

6. Left click on "Router Post Processor (PST)"

7. Left click on "Select"

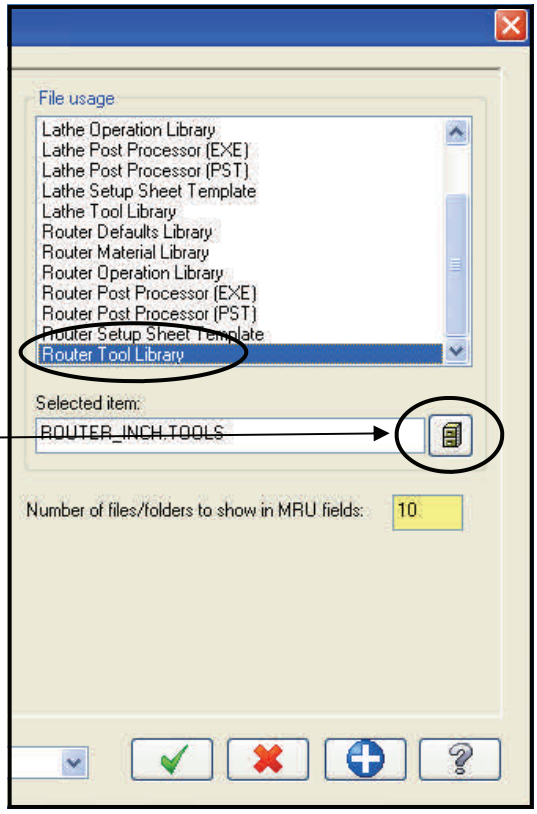




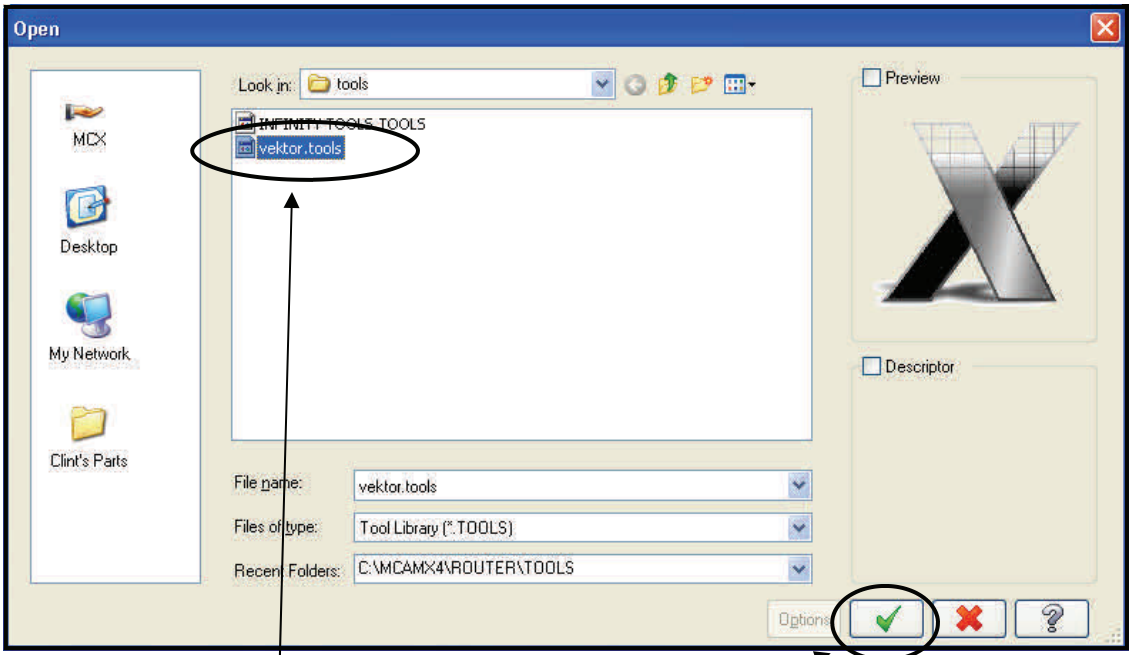
8. Left click on "vector-v100.pst", then left click on "OK".

Setting the Default Machine (cont'd)

9. Left click on "Router Tool Library"

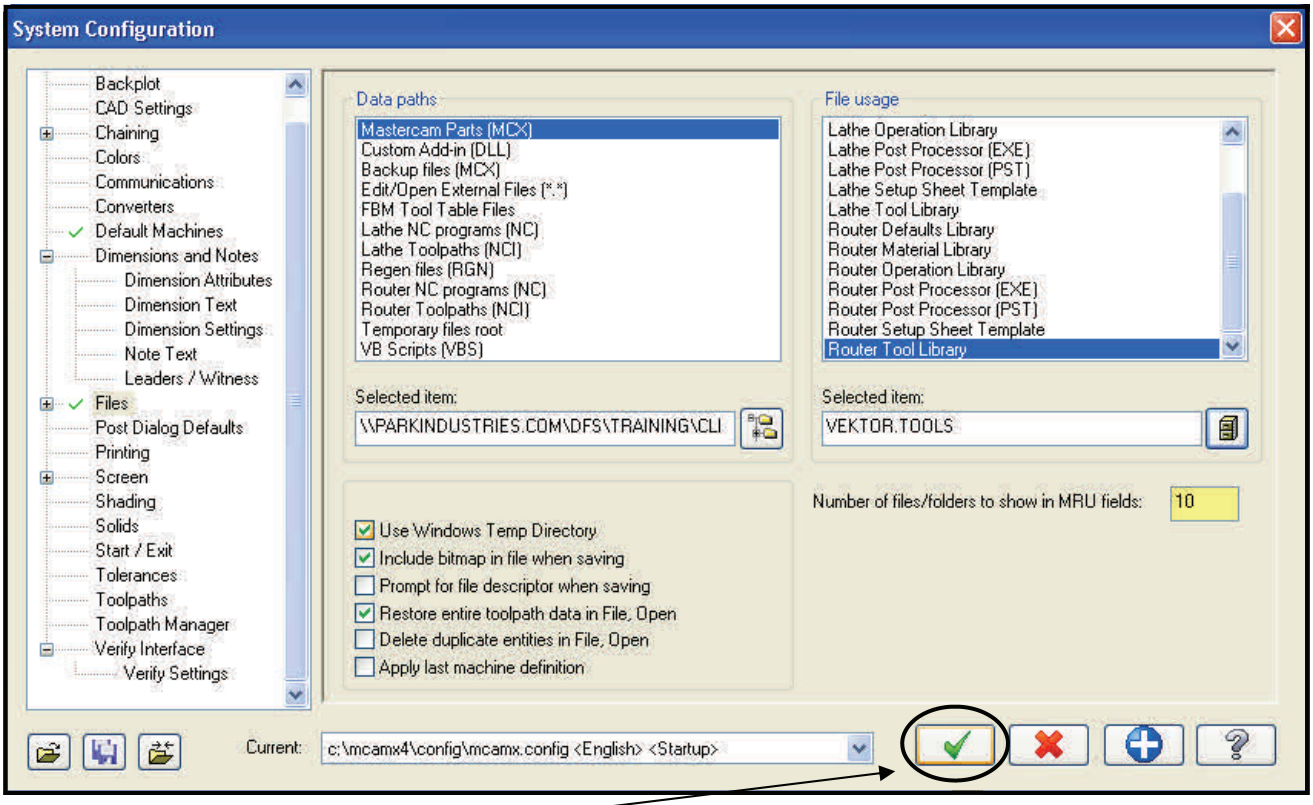


10. Left click on "Select"

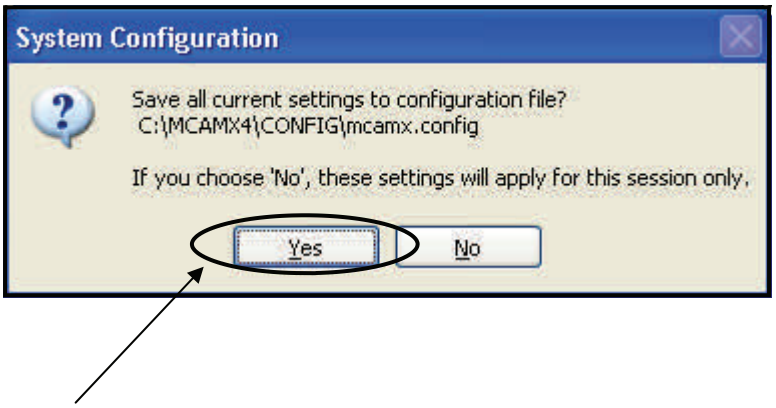


11. Left click on "vector.tools", then left click on "OK".

Setting the Default Machine (cont'd)

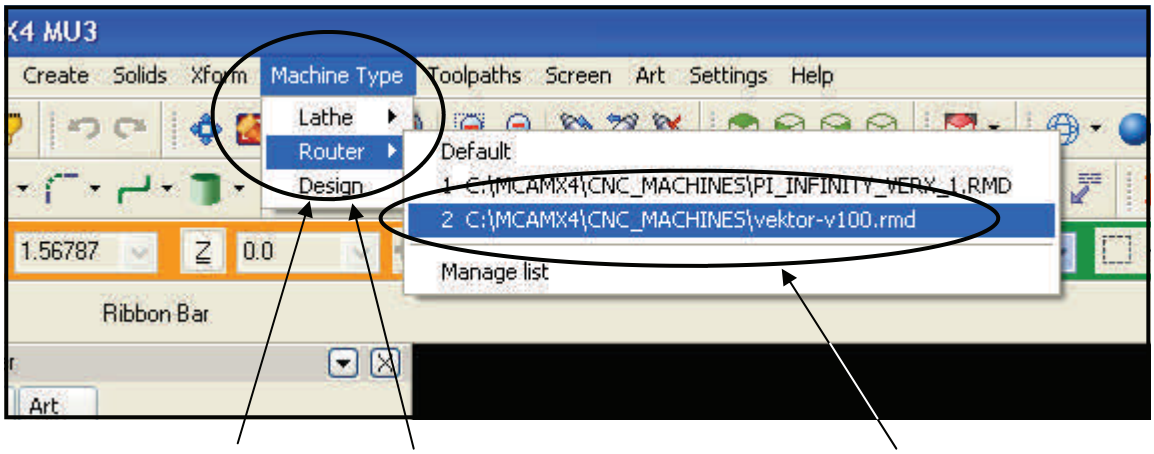


12. Left on "OK".



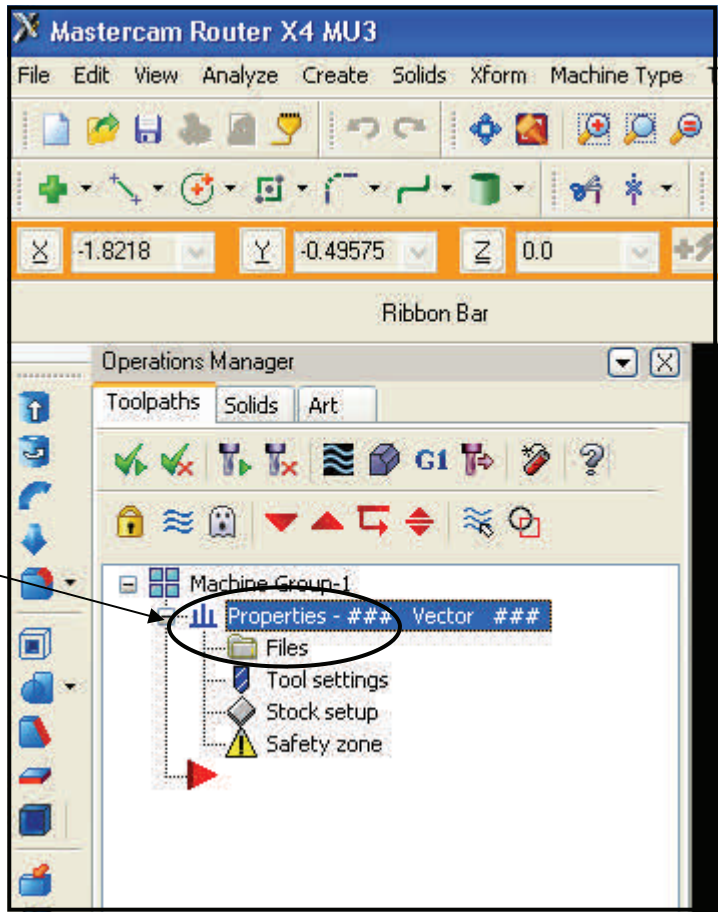
13. Left click on "Yes" to save all the changes that were previously completed in the configuration file.

Setting the Default Machine (cont'd)

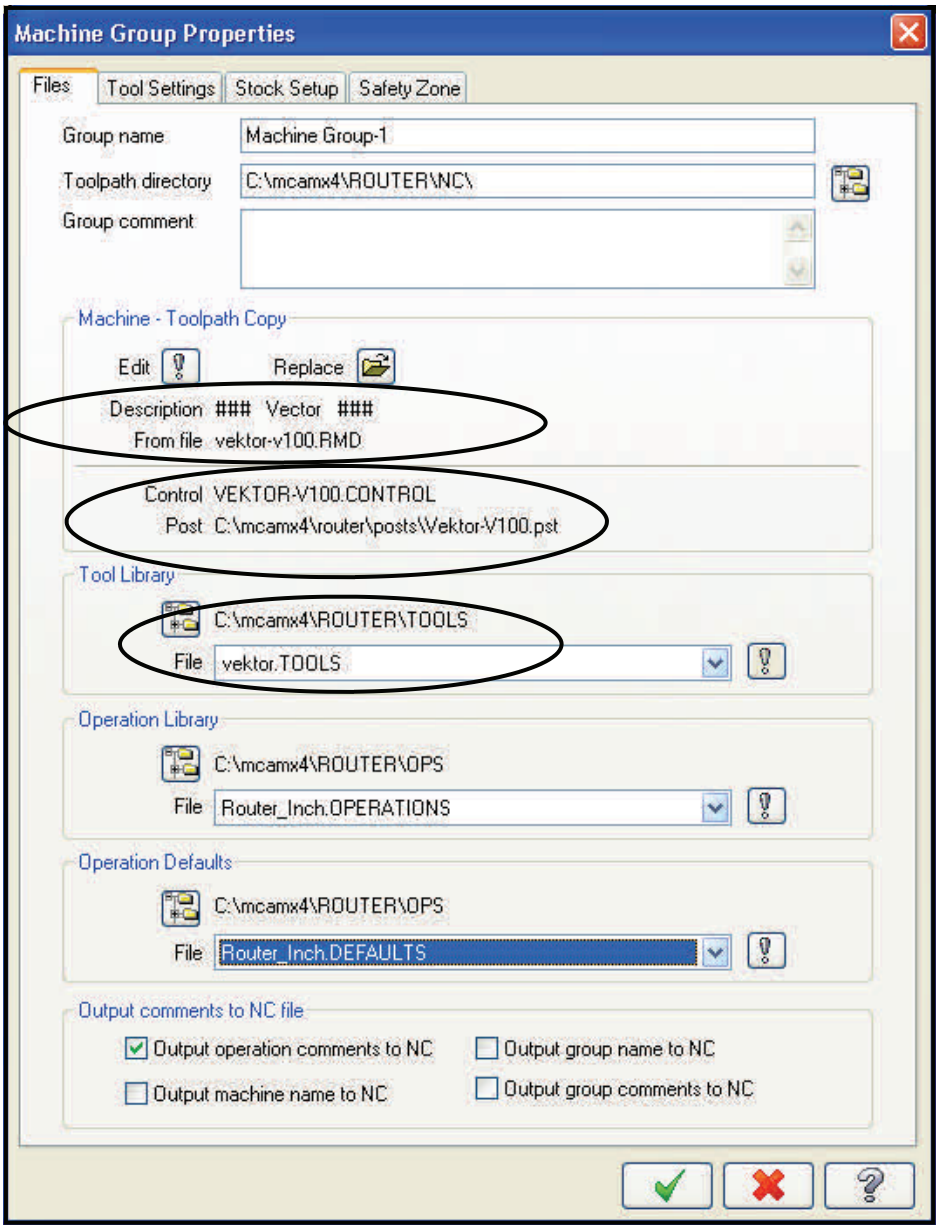


14. Go to "Machine Type" then "Router", then left click on "C:\MCAMX4\CNC_MACHINES\vektor-v100.rmd" to select the correct machine group and post settings.

15. Left click on "Files"



Setting the Default Machine (cont'd)



16. Verify that all the information that is circled matches the programmers PC.

Section 3: MasterCAM Operation

Contents Page

General 3-1
CAD Principles and Operations 3-2
CAD Screens and Definitions 3-3
Starting Mastercam® 3-4
Managing Tool Bars and Operations Manager Window 3-5
Point of Origin 3-6
Swept 2D and Contour Example
- Introduction 3-7
- 3D Translate 3.8
- Contour 1 Create Toolpath Line 3-13
- Contour 1 Defining Across and Along Contour 3-17
- Contour 1 Extending the Toolpath 3-24
- Contour 2 Copy Contour 1 3-34
- Contour 2 Editing Toolpaths 3-35
- Display Stock 3-39
- Verify Operation 3-42

Pocket and Facing Example
- Introduction 3-45
- Stock Setup 3.47
- Create Facing Toolpath 3-49
- Creating First Pocket Toolpath 3-56
- Creating Second Toolpath 3-62
- Verify Operation 3-66

Post 3-68
Nesting Parts 3-71

General

This section describes the use of the Mastercam® Software Package which is the CAD/CAM software chosen to run the VEKTOR CNC Profiler. This section shows how to draw a shape using lines and arcs necessary to produce a profile, how to build various toolpath and create the code needed by the machine to cut the part.

CAD Principles and Operations

This Section describes how to:

- Create lines and arc's using Mastercam® X CAD/CAM Software to make profiles
- Save Part Drawings
- Creating a Tool Path

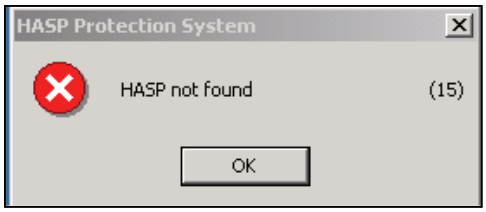
What is CAD?

CAD stands for Computer Aided Drafting. CAD is software that was developed to make the drafting or drawing process more efficient and flexible.

The examples in this chapter will demonstrate the use of common patterns that include straight lines and arcs to draw simple shapes.

Licensing

Mastercam® uses two types of licensing: Single user license or Network license. The single user license uses a hardware device called a HASP which is attached to a computers Parallel Port or a USB Port. When the HASP is not connected an error will be generated as shown below.

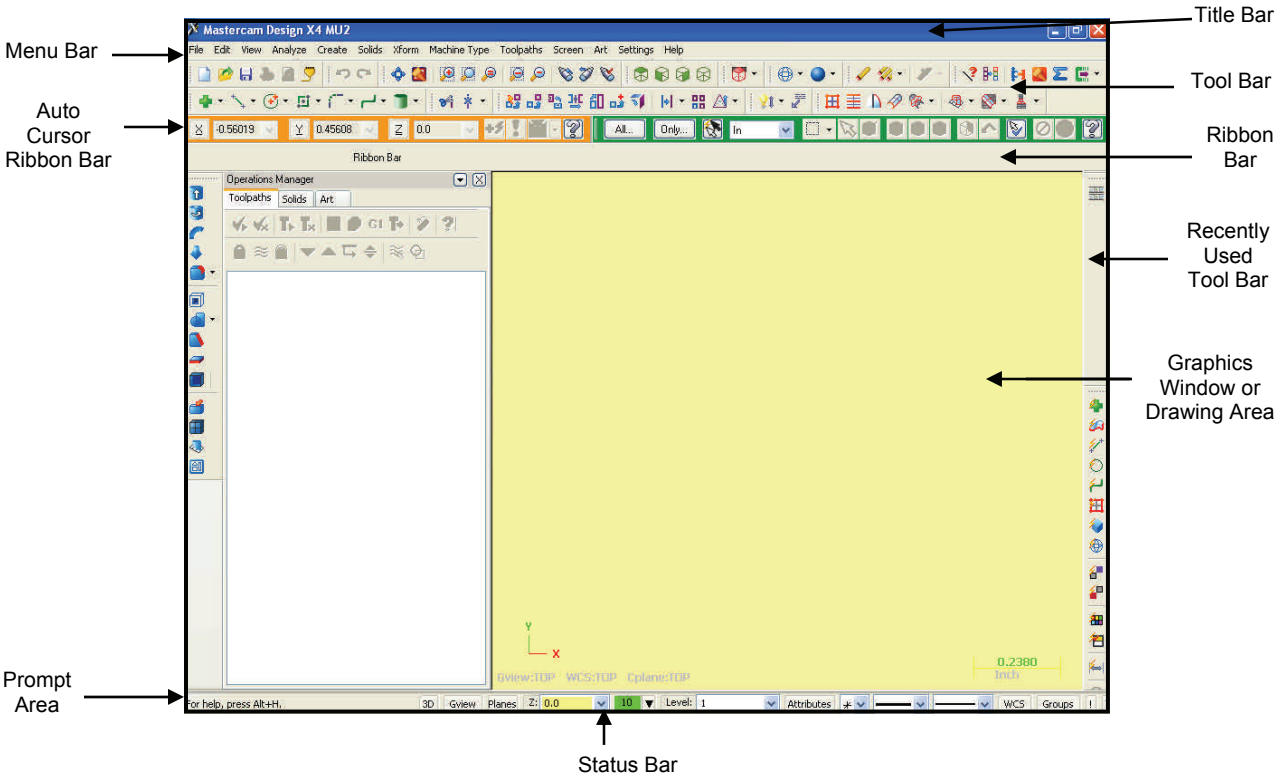


If a network license is being used, contact the system administrator if any of the following errors are displayed.

- Error checking out a router license. No licenses have been purchased for this product.
- Active NetHASP server not found.
- All available licenses are in use.

CAD Screens and Definitions

Before drawing can be started, the understanding and use of the CAD options are essential. The main screen is where all of the drawing will be done and where most of the options are displayed. The screen below is the way Mastercam® appears before any modifications are made. For ease, the programmer may want to personalize it to fit his/her style.



Menu Bar - Pull-down menus listing commands.

Tool Bar - Each of the buttons in the toolbar is a shortcut for commands in the Menu Bar. If the cursor is positioned on top of the button it will define what the button does.

Auto Cursor Ribbon Bar - Shows actual coordinates of the position where the mouse pointer is located.

Ribbon Bar - Prompts you with icons for distances, OK or Help.

Recently Used Toolbar - Shows icons previously used for each time different command is selected.

Graphics Window - The drawing area where parts and tool paths will be displayed.

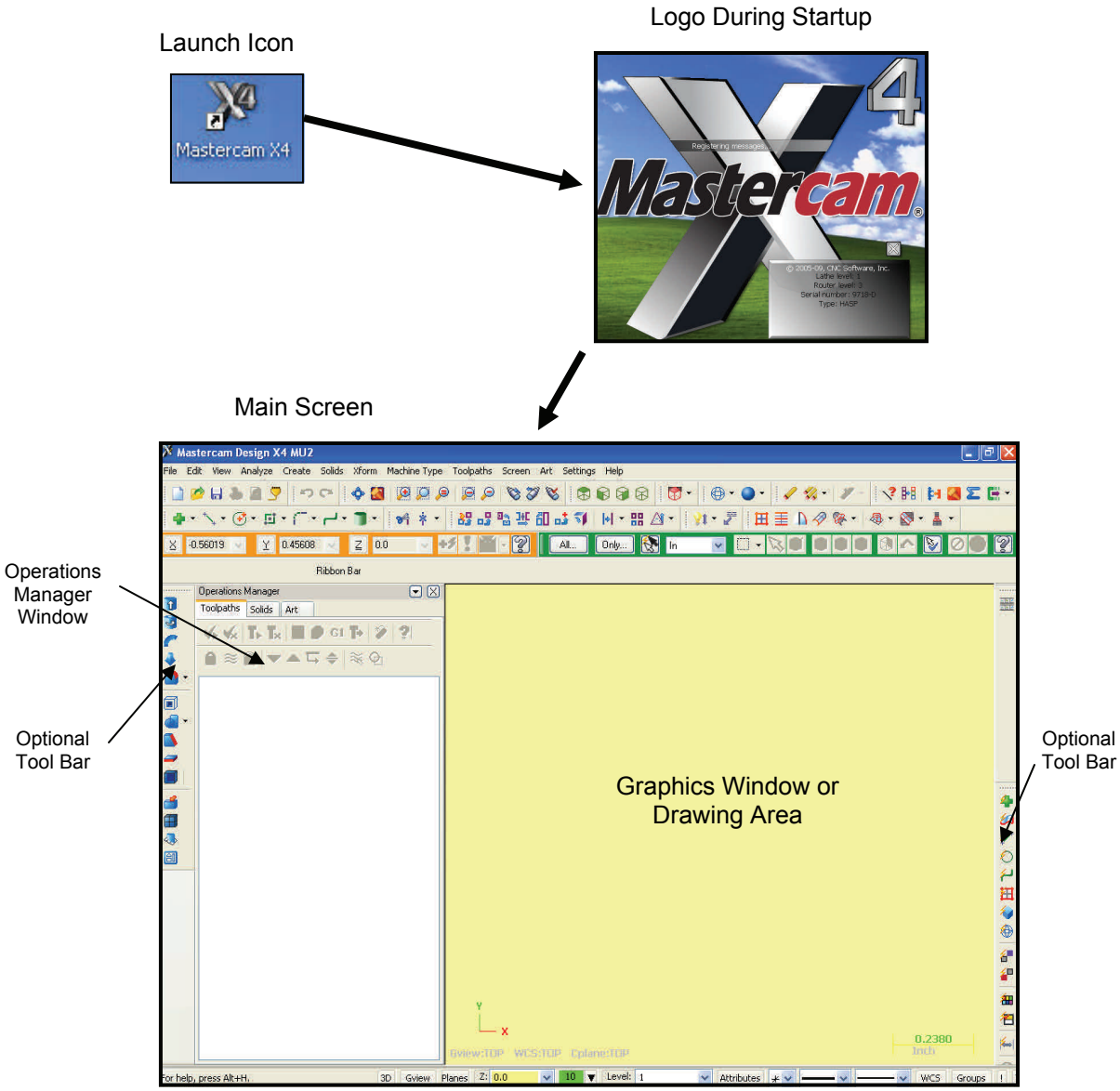
Prompt Area - This space tells you what information needs to be entered. What is typed will also be displayed here.

Status Bar - Shows information about lines, coordinate system type information.

Starting Mastercam®

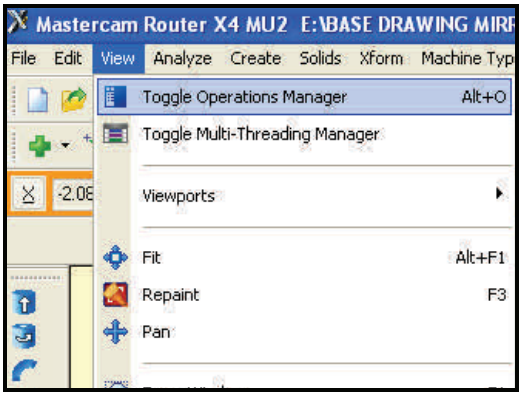
To start Mastercam® X4, double click the icon on the computer's desktop. The Mastercam® logo is displayed as the program loads. After a few seconds, the main screen is displayed. The graphics window is where all of the drawing is done.

NOTE: The graphics area's color is normally black. The color shown below was selected to improve the clarity of the screen shots used in this manual. The number of tool bars (top and sides) varies according to individual preference. Also, the Operations Manager window may not be open the first time the program is launched. Information on managing tool bars and viewing the Operations Manager window is shown on the next page.



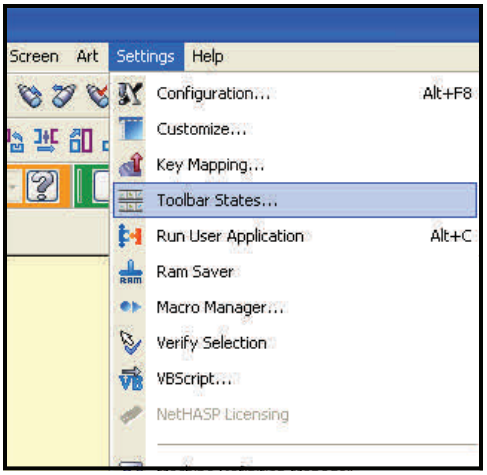
Managing Tool Bars and Operations Manager Window

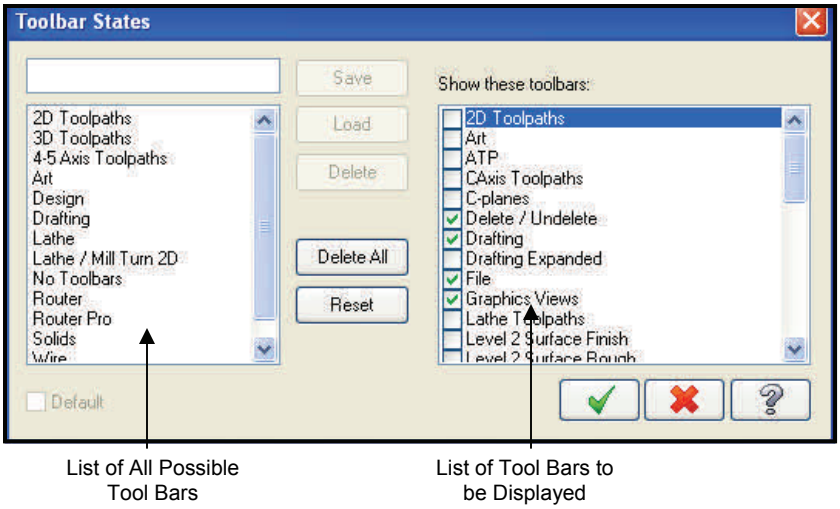
Use the “Toggle Operations Manager” command located in the “View” menu to display or remove that window from the screen.



To enable or disable tool bars, select the “Toolbar States” command from the Settings menu. This command launches the Toolbar State dialog window which allows the user to determine which tool bars are displayed. For most people displaying the following tool bars is a good starting point.

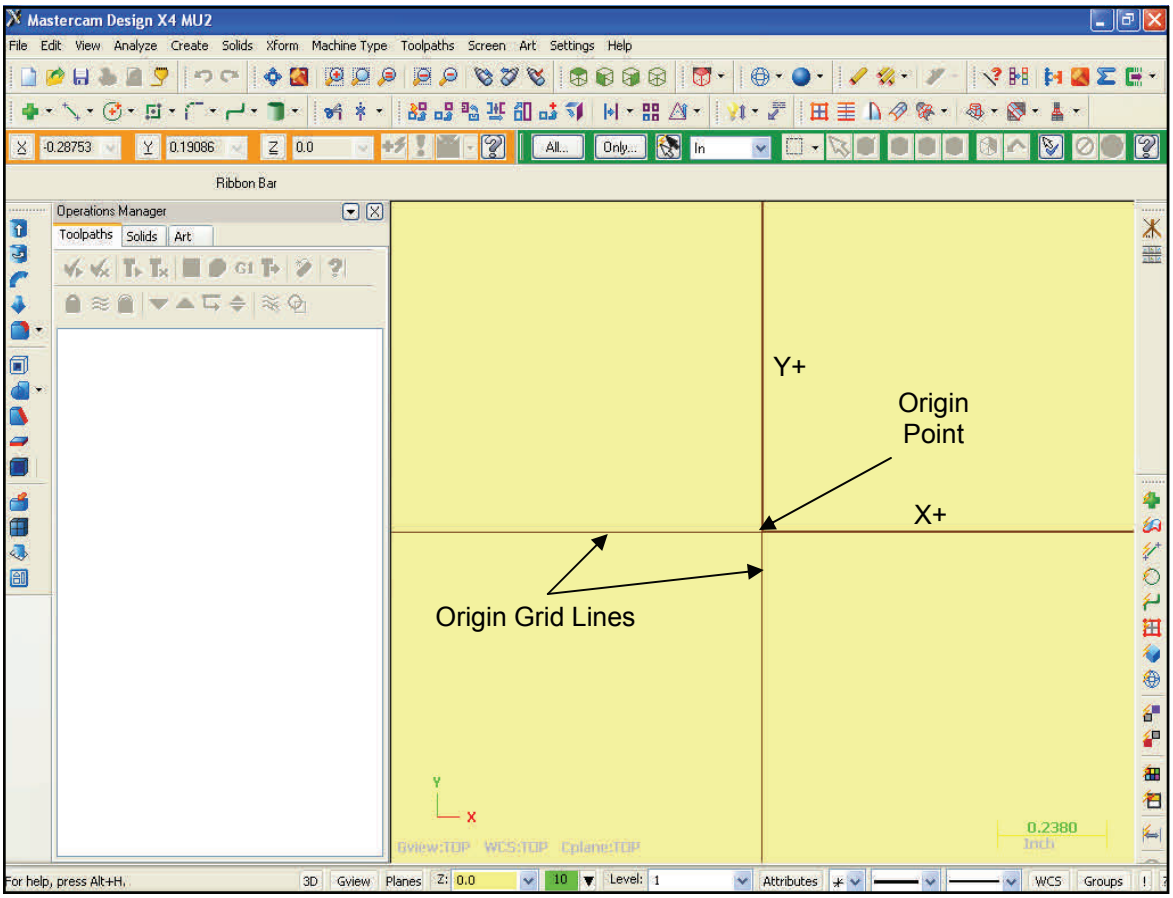
- Delete/Undelete
- Drafting
- File
- Graphic View
- Planes
- Sketcher
- Trim/Break
- Undo/Redo
- Utilities
- View Manipulation
- Xform





Point of Origin

Once the main screen is displayed, pressing the F9 key toggles the origin grid in the graphics window OFF to ON or ON to OFF. It's easier to start a drawing at the point of origin.



For the machine to work properly, use the upper right quadrant of the graphics window when drawing all parts for the VEKTOR. The machine part program references from the point of origin of the drawing. All programming can be made out in space but before the G-Code is created for the machine to run, move the part drawing to the origin in the drawing. When setting the machine up to cut the parts, create a work coordinate which is in relationship to the origin of the part drawing.

Note: If the part drawing is not moved to the origin, the only way to locate the slab is to measure the distance from the origin to where the corner of the part is in the drawing, if the machine is setup this way then the slab may not even be on the cutting bed under the machines rails. **This way is not recommended always reference the part program at the point of origin.**

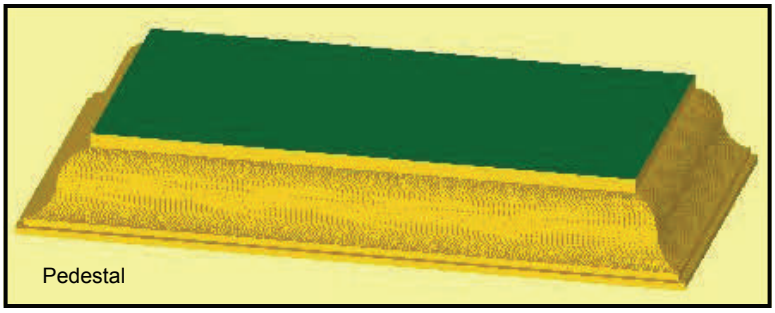
Swept 2D and Contour Example

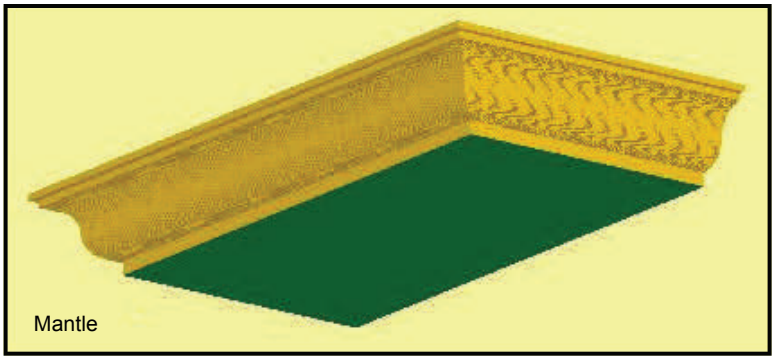
Introduction

The first example in this section is the fireplace mantle or pedestal (depending which side is up) profile. The below figures illustrate the final product after executing the Mastercam® verification operation. The three sides are profiled while the back is straight cut with no profile.

A swept 2D cutting cycle is used to profile the three sides. A contour cutting cycle is used to cut the back and remove the piece from the stock or slab.

This section illustrates the steps to create the toolpath operations and code that a CNC machine uses to cut the part.

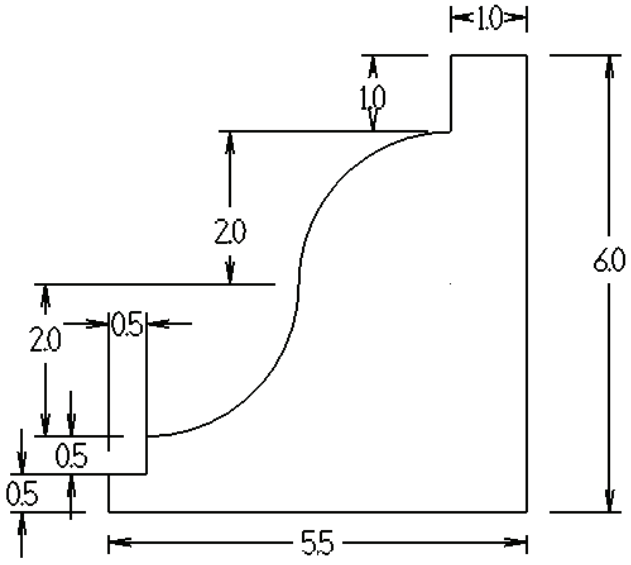

Pedestal


Mantle

The major Mastercam® commands or tools used to accomplish this job are:

- 3D Translate
- Translate
- Offset
- Swept 2D Toolpath
- Extend Line
- Contour
- Stock Setup
- Verify

The initial step to produce the part is to create a two dimensional (2D) drawing (using basic Mastercam® tools) of the profile found in that product. The below drawing shows the dimensions of the base profile.



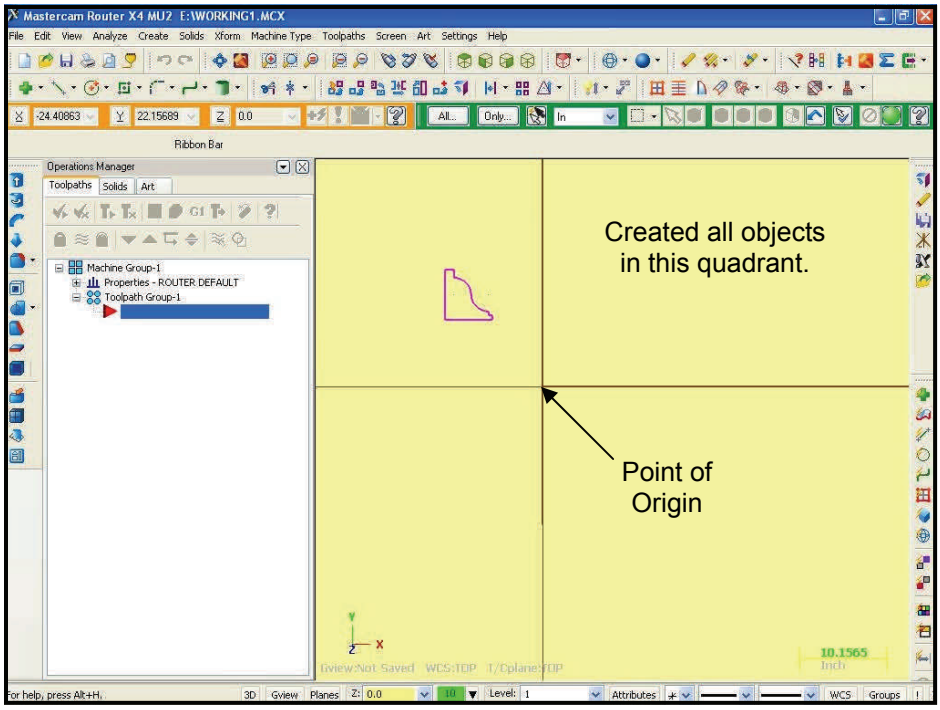
Swept 2D and Contour Example

Task: 3D Translate and Move

The translate and move task consists of three parts.:

- 2D drawing is translated to a 3 dimensional (3D) view
- Create object representing dimensions of the final product
- Moving the 3D view to the final product drawing

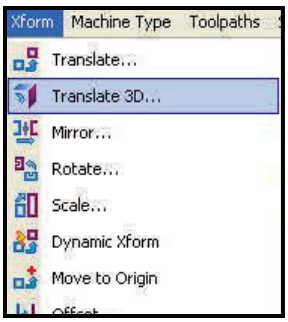
In the drawing area of below screen view, the base profile has been moved from the point of origin quadrant to a location in the upper left quadrant. The actual location is not that important, just as long as it is not in the upper right quadrant. The lower left corner of the final product must always start at the point of origin and be created in the upper right quadrant.



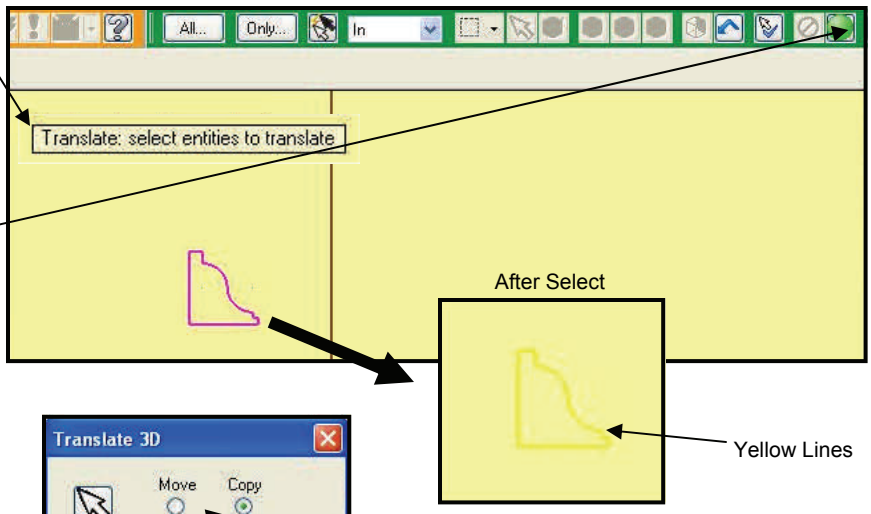
Swept 2D and Contour Example

Task: 3D Translate and Move (cont'd)

1. To start the translate process choose "Xform" from the menu bar and then click on "Translate 3D".



2. The next step is to select the entities to translate. Using one of the many select object tools, select the entire profile (Selected lines are yellow.) Click on the green end button to end the selection process.

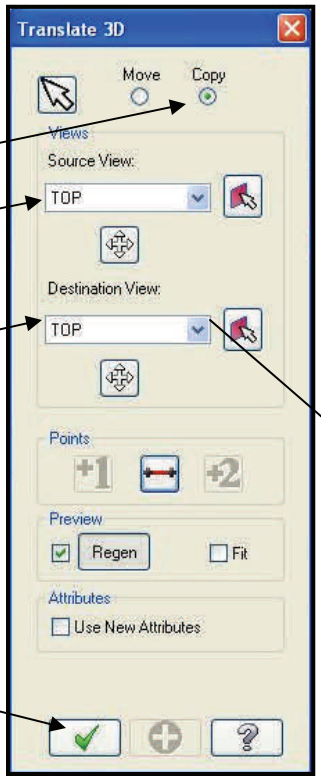


3. Once the selection process is done, the Translate 3D menu is displayed.

   Check "Copy"

   Source View - Top

   Destination View - select Left Side from the pull down menu.



Destination View Pull Down Menu

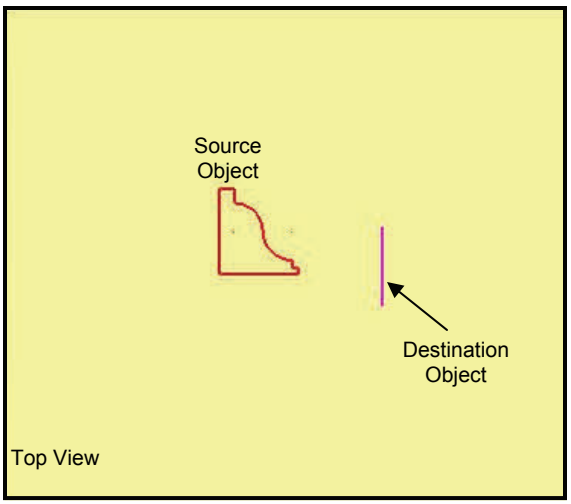
4. When done, click the green check mark.

Swept 2D and Contour Example

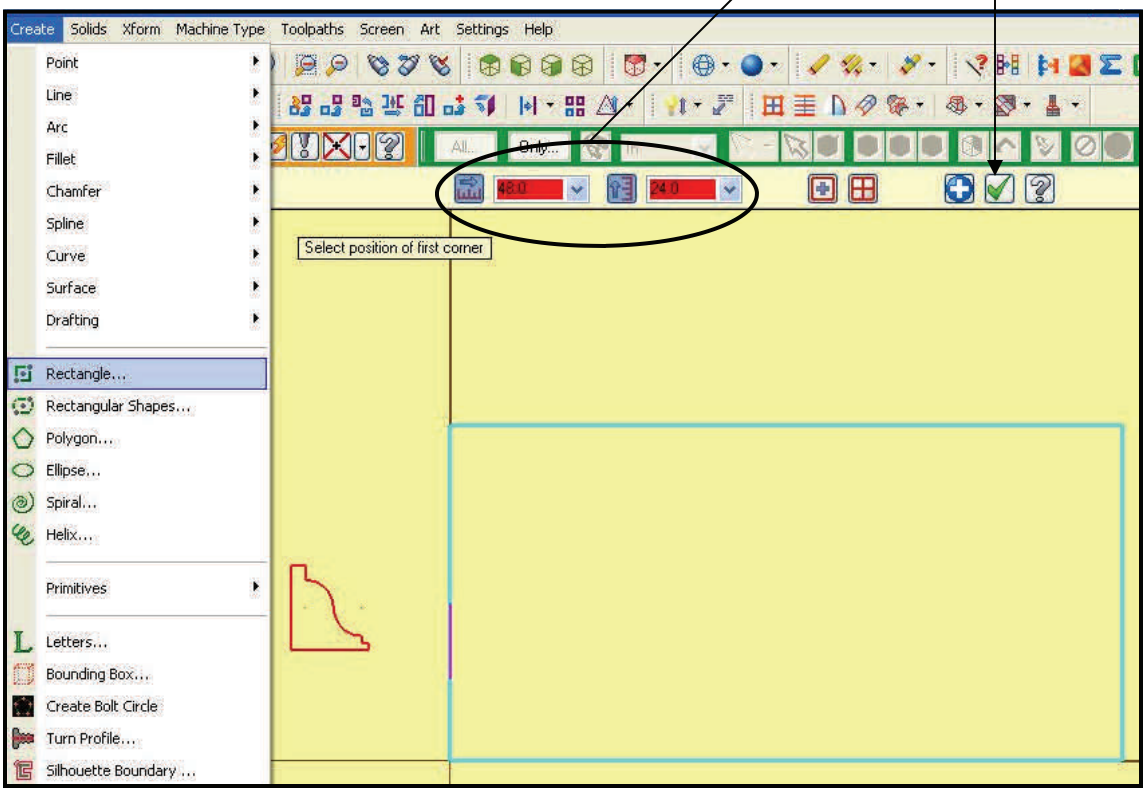
Task: 3D Translate and Move (cont'd)

This screen view illustrates the drawing area after the translate 3D command was executed. Remember the drawing area is still in Top View thus the translated object appears as a straight line. Before going to the Isometric view, an object the size of the final product has to be created. It is easier to draw this object in the Top View.

The finished product is rectangular in shape with the profile on three sides. The size of the rectangle is 48" by 24".



5. To create a rectangle:
   - choose "Create" from the menu bar and then click on "Rectangle".
   - Starting at the origin create a rectangular shape object in the upper right quadrant.
   - Enter dimensions:X 48.0 and Y 24.0
   - Click either direction button
   - Click green check Mark.



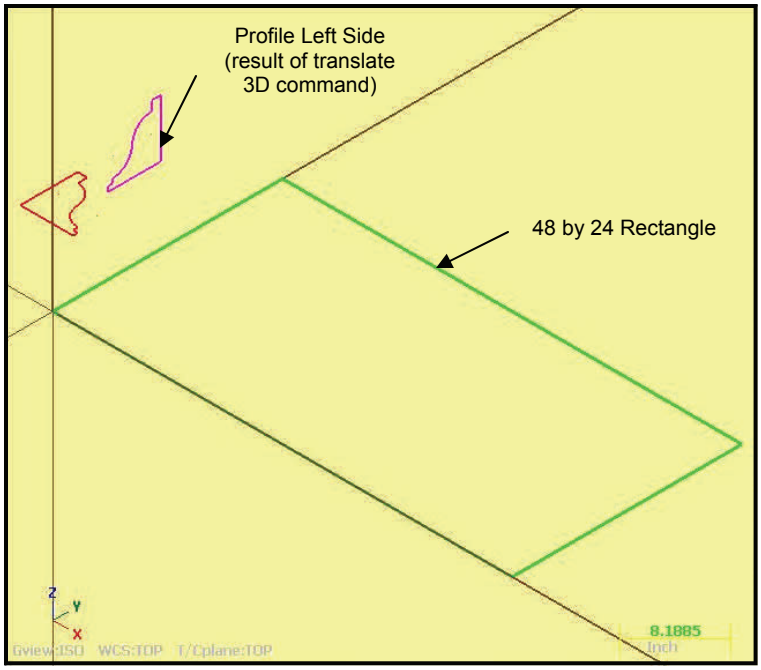
Swept 2D and Contour Example

Task: 3D Translate and Move (cont'd)

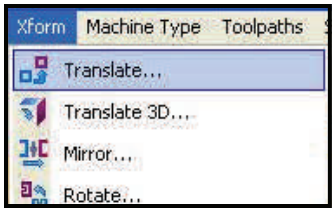
Switch to Isometric View

This screen view illustrates the drawing area after the view was changed from Top View to Isometric View.

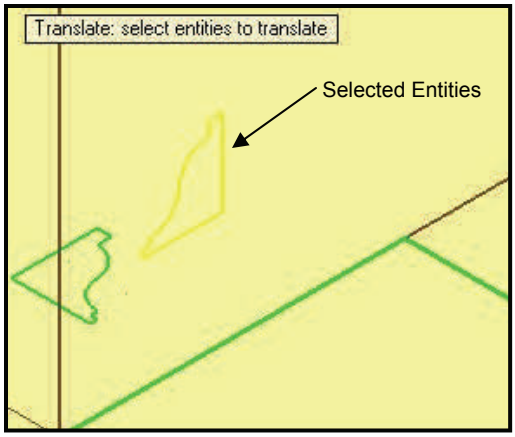
The next task is to move the left side profile to the center of the front line of the rectangle. The Translate command is used to make this move.



6. Choose "Xform" from the menu bar and then click on "Translate".



7. Using one of the many select object tools, select the entire profile (Selected lines are yellow.)



8. Click on the green end button to end the selection process.

Swept 2D and Contour Example

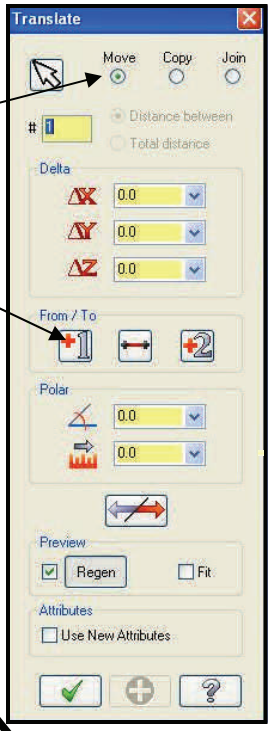
Task: 3D Translate and Move (cont'd)

9. Once the selection process is done, the Translate menu is displayed.
   Check "Move"
   Click "Select From point" button.

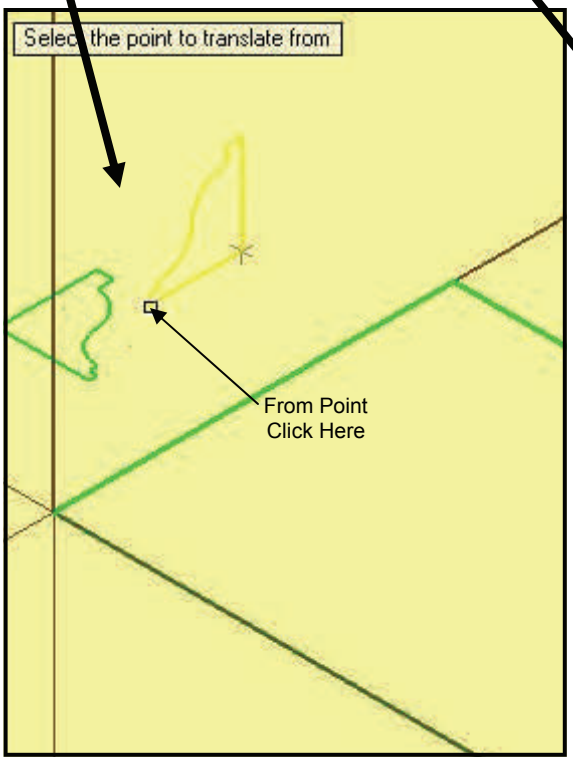
10. Click on the point to translate from.

11. Click on the point to translate to.

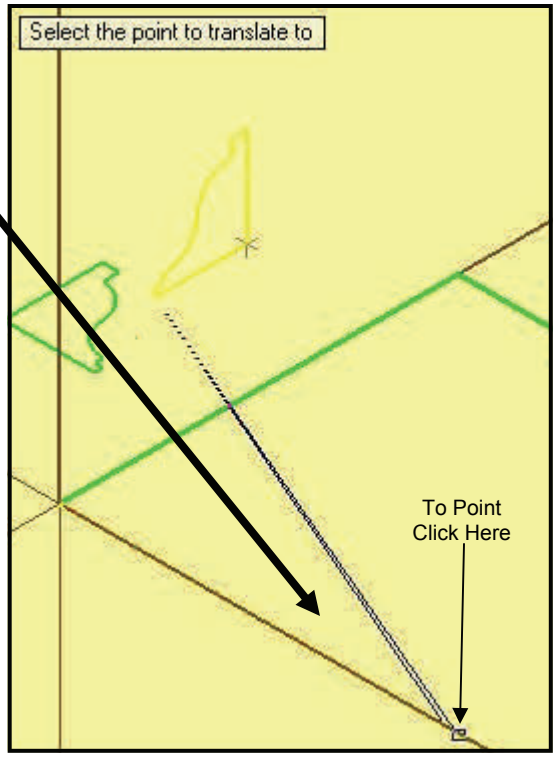
    See top of next page for results.



Step 10



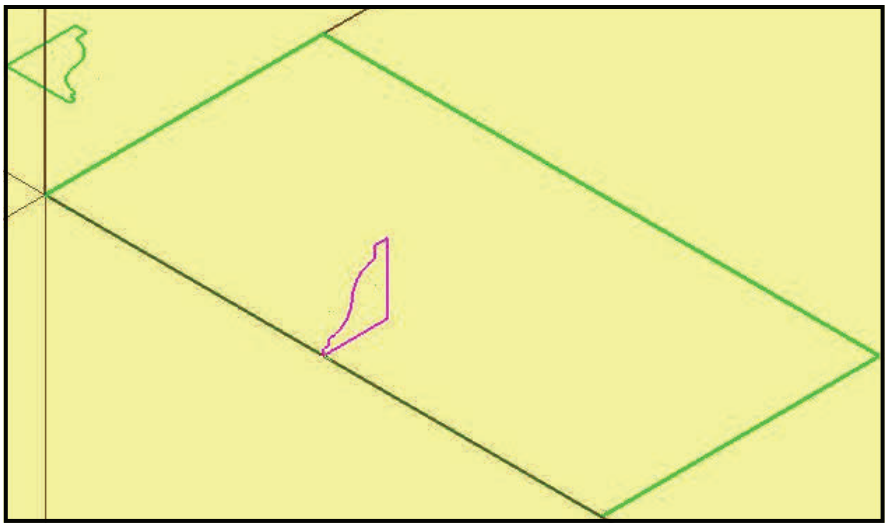
Step 11



Swept 2D and Contour Example

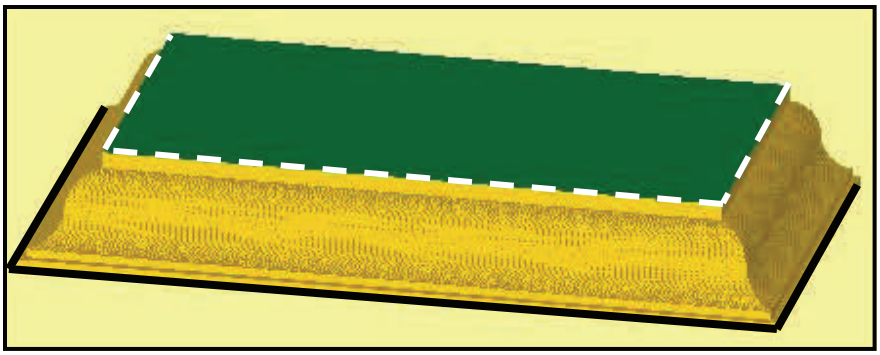
Task: Swept 2D - Create Toolpath Line

This screen view illustrates the drawing area after the translate command was executed. The profile has been moved to center point of the front line of the 48” by 24” rectangle. The next task is to draw a line or lines the tool will follow. For this explanation, the line or lines are called contour tool line.



Cutting is always done from top of stone to bottom of stone. For this example, the white dashed line on the below drawing is the Swept 2D contour tool line. Since the profile is on only three sides, the line appears on three sides. The offset command is used to create and place the contour tool line in the correct position.

The offset command selects three sides of the base rectangle (solid black lines on below drawing) copies the three and then offsets them to new the position. The steps to execute the offset command are illustrated on following pages.



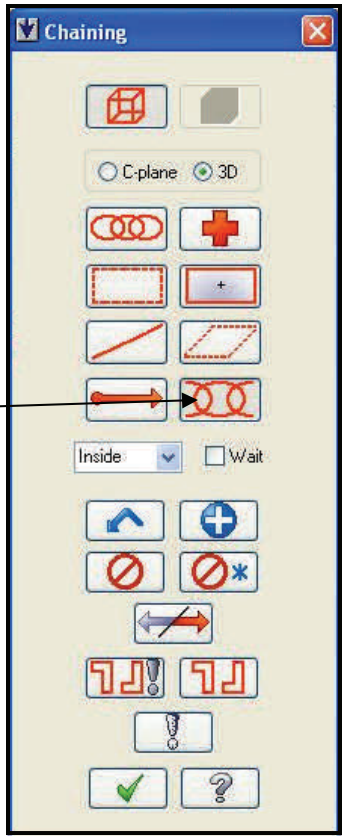
Swept 2D and Contour Example

Task: Swept 2D - Create Toolpath Line (cont'd))

Steps to Offset a Contour

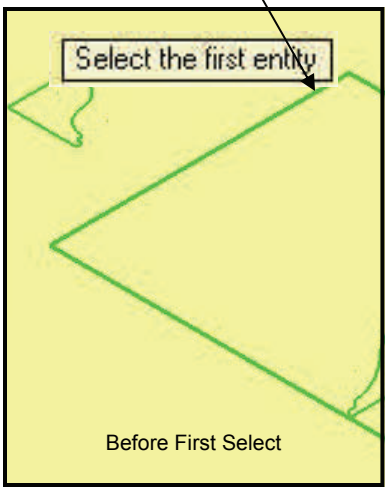
1. Select "Offset Contour" command

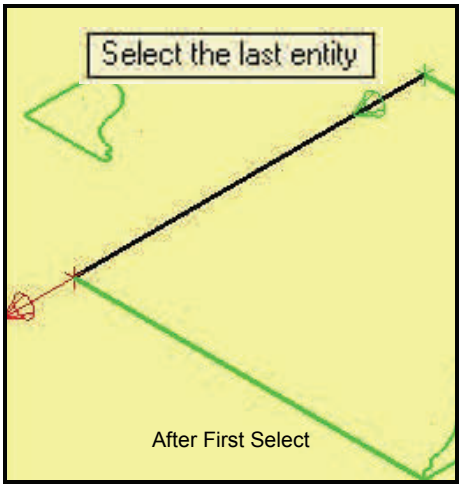
2. Click "Partial" chain



3. The message box displays:
   "Select the first entity"

   Single click here





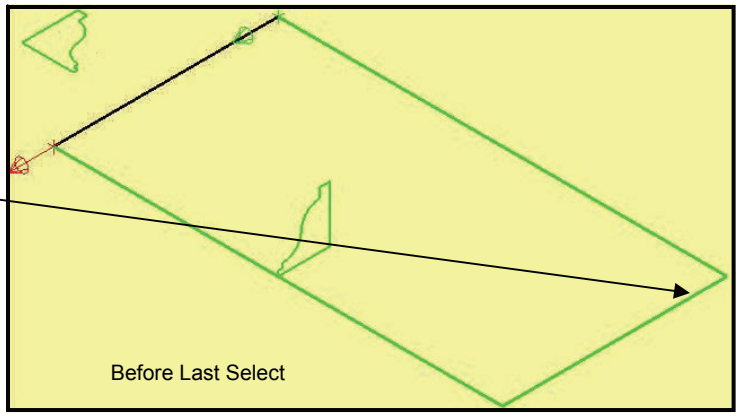
Swept 2D and Contour Example

Task: Swept 2D - Create Toolpath Line (cont'd)

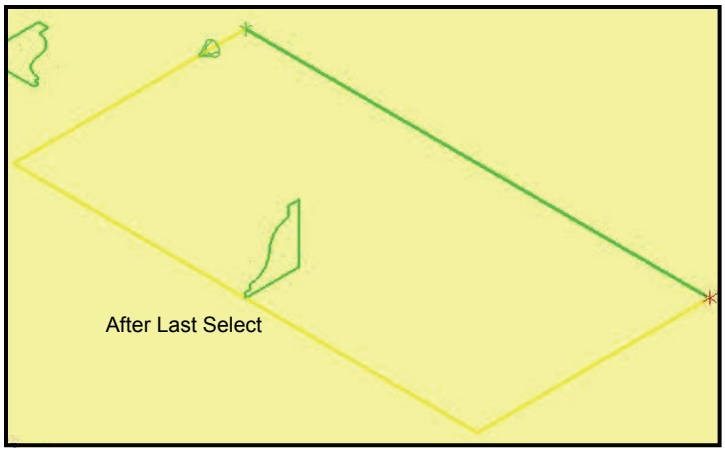
Steps to Offset a Contour (cont'd)

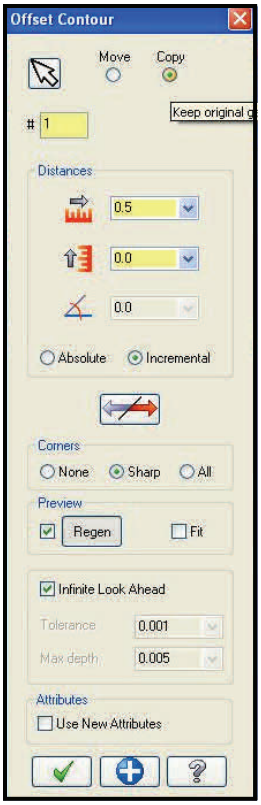
4. The message box displays:
   "Select the last entity"

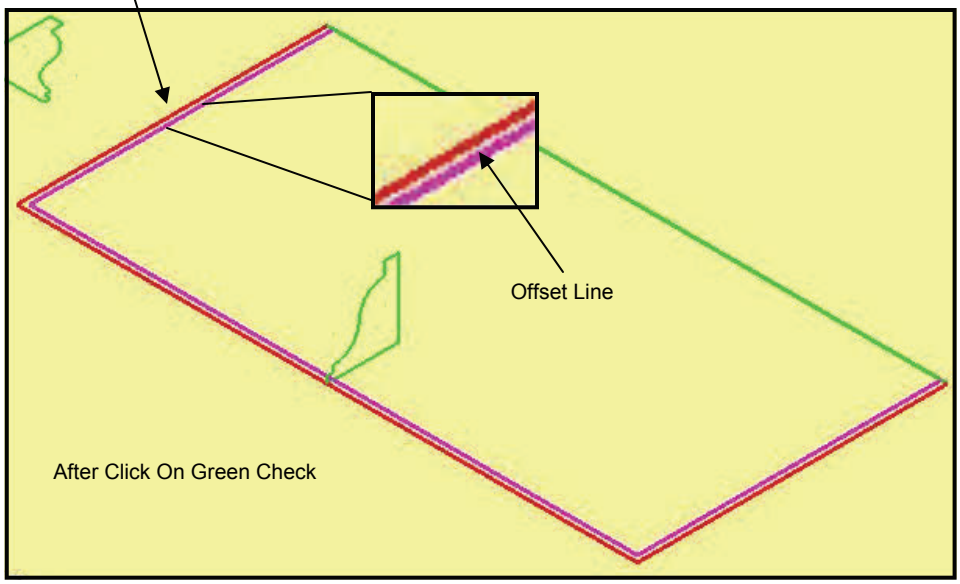
   Single click here



5. Click the green check mark at the bottom of the Chain window. The following happens:
   - ends the selection process
   - launches Offset Contour window
   - creates offset lines in drawing area







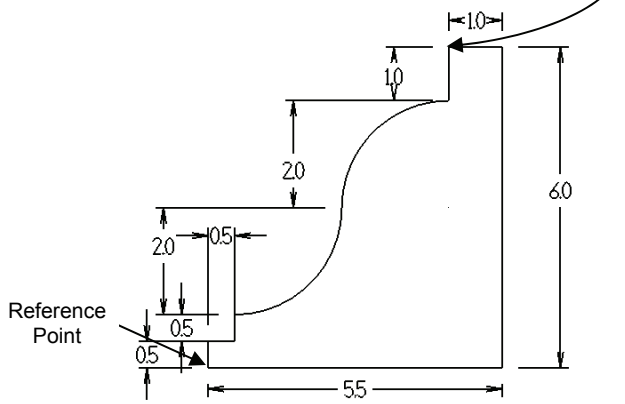
Swept 2D and Contour Example

Task: Swept 2D - Create Toolpath Line (cont'd)

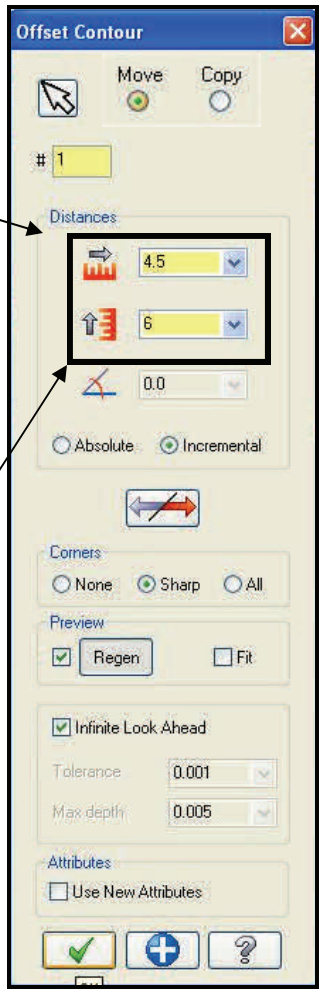
Steps to Offset a Contour (cont'd)

At this time the message box displays:

The Offset Contour window contains a Distance dialog box which determines how to move the offset line. For this example move the offset to this point.

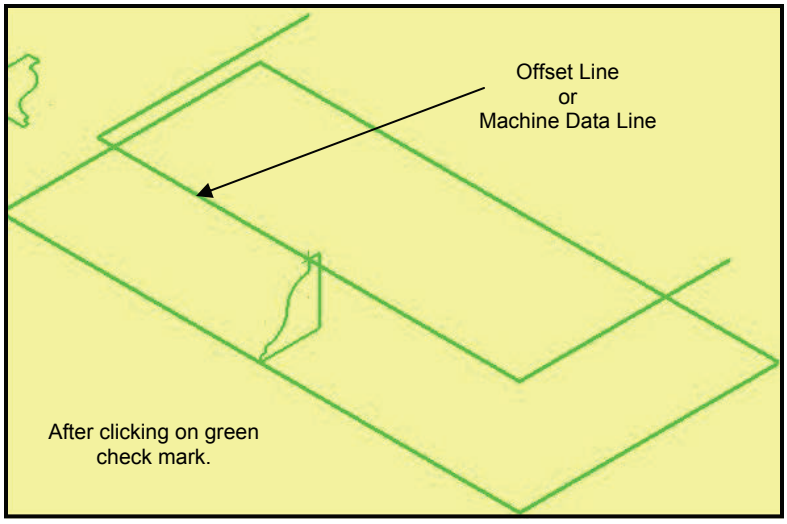


That point is 4.5 inches over and 6 inches up from the reference point. These are the offset values entered in the Distance dialog area.



6. On the Offset Contour window:
   - check Move radial button
   - enter the offset values

7. Click the green check mark at the bottom of the Offset Contour window. The following happens:
   - moves offset line
   - close Offset Contour window



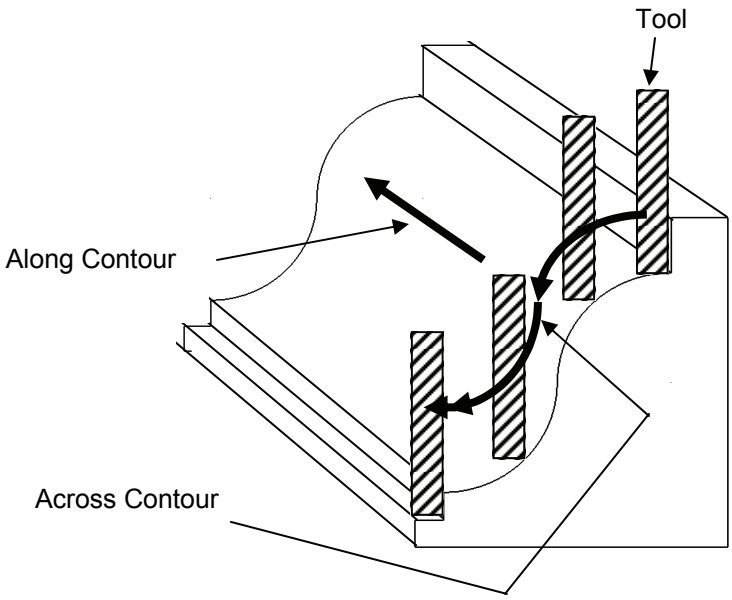
Swept 2D and Contour Example

Task: Swept 2D - Defining Across and Along Contour

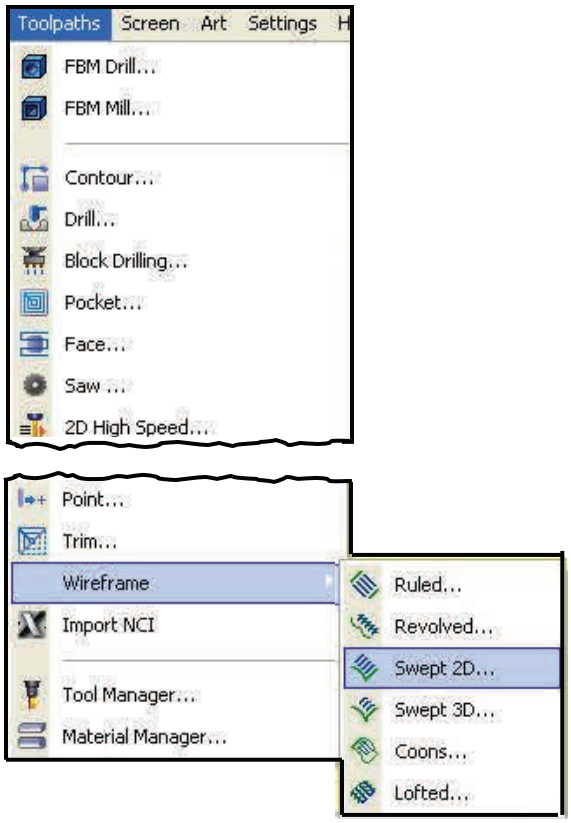
A swept 2D cutting cycle performs a cut across and along a boundary to produce the profiled part. This drawing illustrates the two contours for this example. Cutting is always done from top of stone to bottom of stone.

For this job, the stock material and the finished product is the same height - six (6) inches. Therefore any part of the profile at the six inch height does NOT have to be cut.

The next task is to create the actual tool path. These steps are illustrated on the next pages.



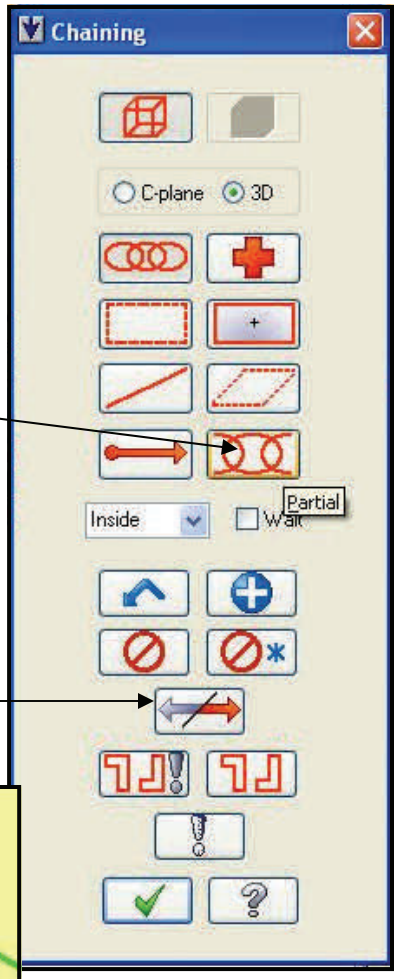
1. Choose “Toolpaths” from the menu bar, then “Wireframe” and then “Swept 2D”.



Swept 2D and Contour Example

Task: Swept 2D - Defining Across and Along Contour (cont'd)

2. When this dialog box appears, click on the green check mark. Verify the Chaining window is displayed.

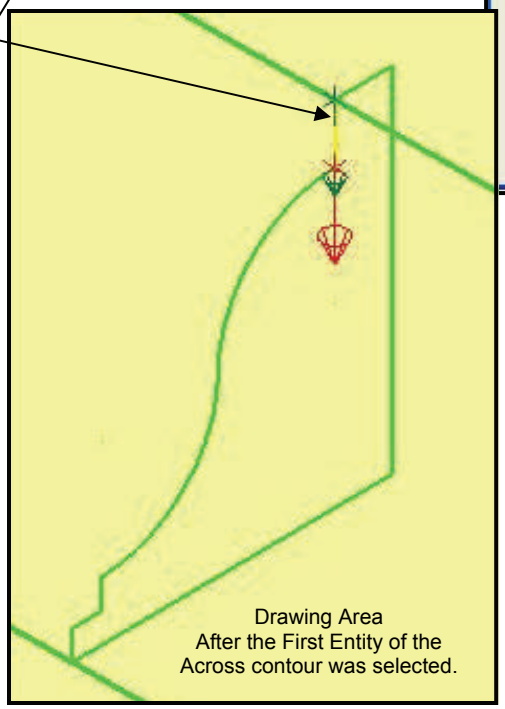


3. Click on the "Partial" chain icon.

At this time the message box displays:

4. Zoom in on the profile part in the drawing area and single click here

A direction arrow pointing down appears indicating the direction of the contour. Click the Reverse direction button on the Chaining window if the arrow is point up.



Drawing Area
After the First Entity of the
Across contour was selected.

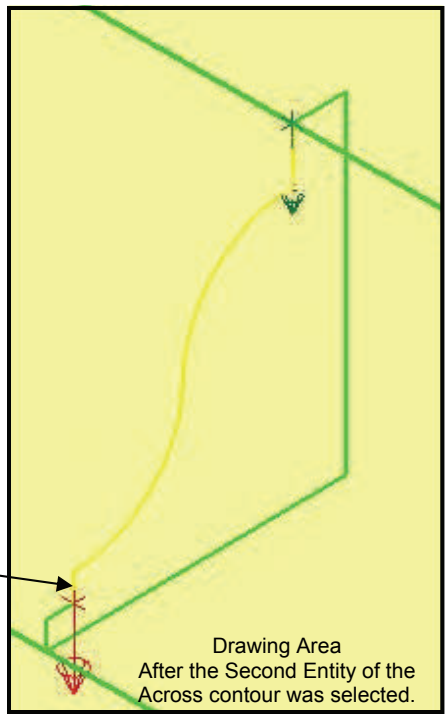
Swept 2D and Contour Example

Task: Swept 2D - Defining Across and Along Contour (cont'd)

At this time the message box displays:

5. Zoom in on the profile part in the drawing area and single click here

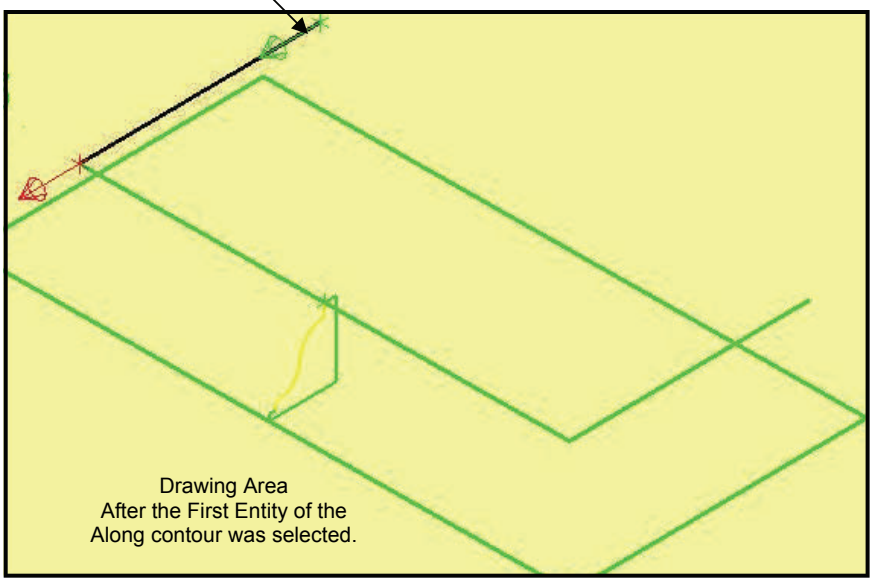
A direction arrow pointing down appears indicating the direction of the contour.



At this time the message box displays:

6. Zoom out so that all three contour tool lines are visible and then single click here

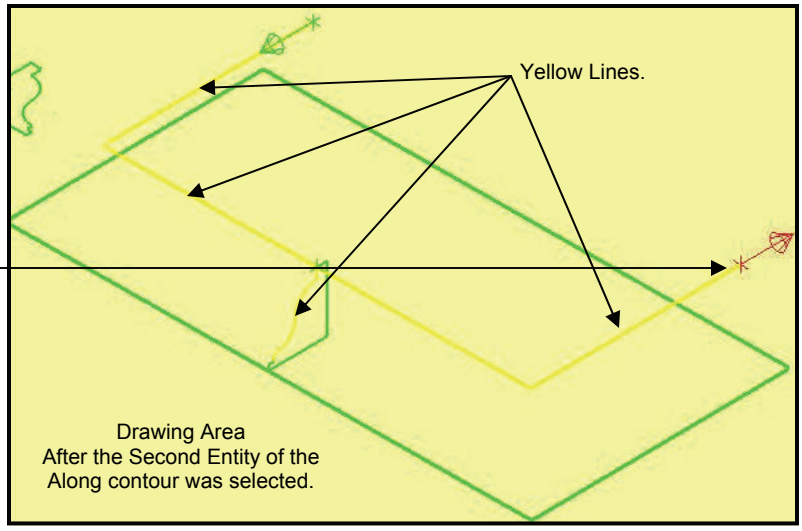
A direction arrow pointing the direction of the contour appears. Click the Reverse direction button on the Chaining window if arrow is not point toward the front.



Swept 2D and Contour Example

Task: Swept 2D - Defining Across and Along Contour (cont'd)

At this time the message box displays:



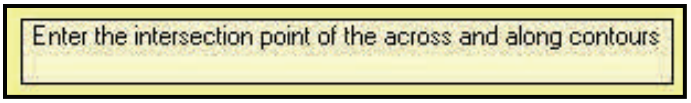
7. To select the last entity single click here

Check the directional arrows and verify that all selected contours are yellow.

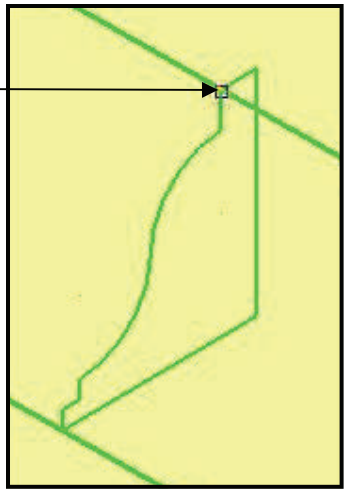
At this time the message box displays:

8. On the Chaining window click

At this time the message box displays:



9. Zoom in on the profile part in the drawing area and single click here

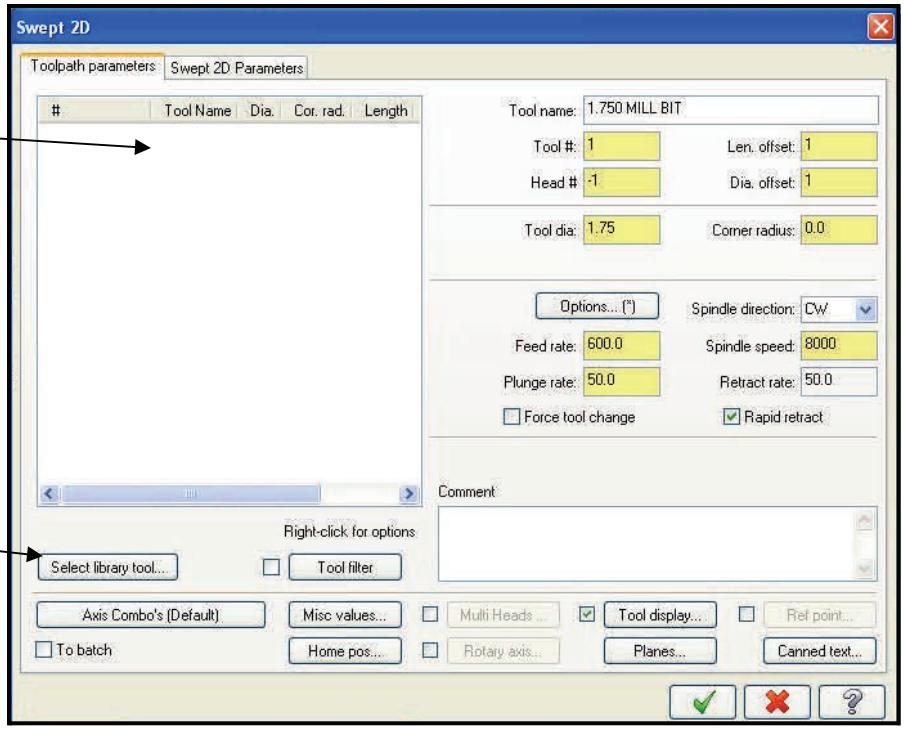


Clicking the intersection ends the selection part and launches the Swept 2D window shown at the top of the next page.

Swept 2D and Contour Example

Task: Swept 2D Setting Toolpath Parameters (cont'd)

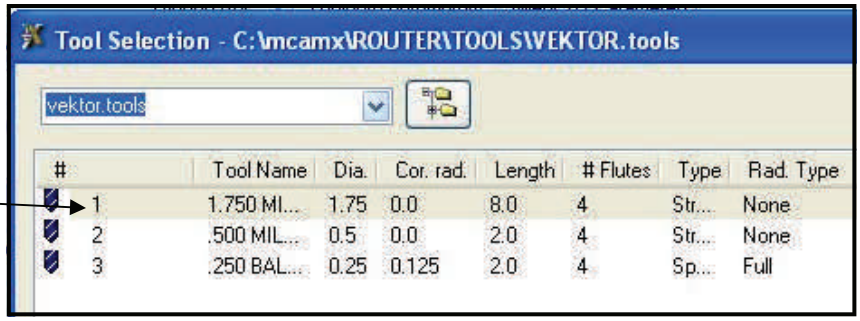
The Swept 2D window opens with the "Toolpath parameters" tab open. The first time this window is opened the tool list will be empty. A tool must be loaded from the library.



1. Click on the "Select library tool" to display the Tool Selection window.

2. Select "vektor tools" in the pull down menu to display a list of VEKTOR tools.

3. From the vektor tool list highlight/select tool # 1.



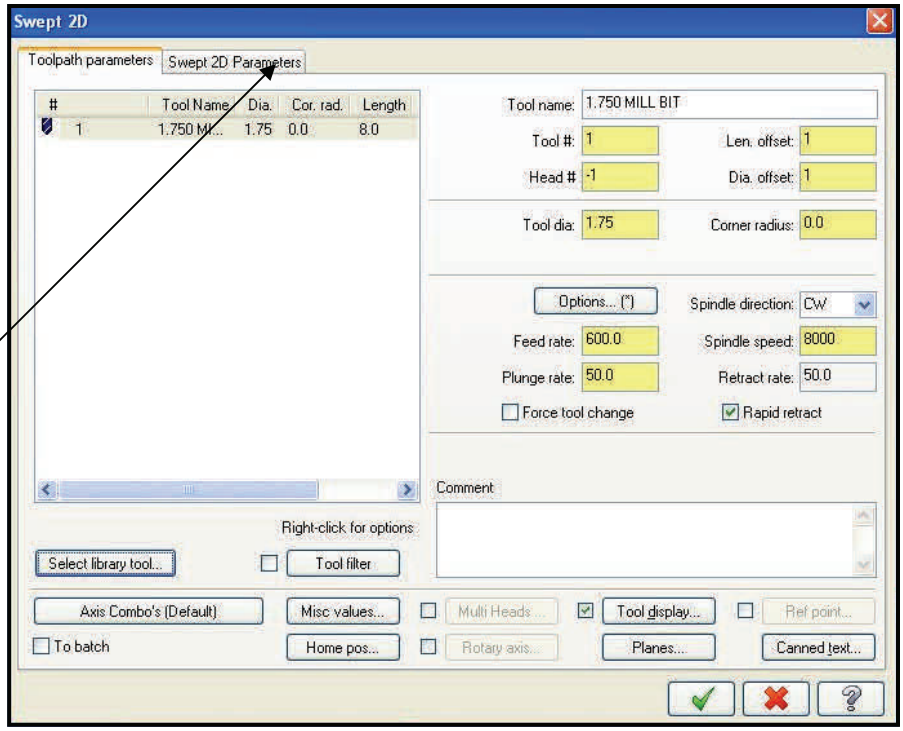
4. On the Tool Selection window click

Swept 2D and Contour Example

Task: Swept 2D Setting Toolpath Parameters (cont'd)

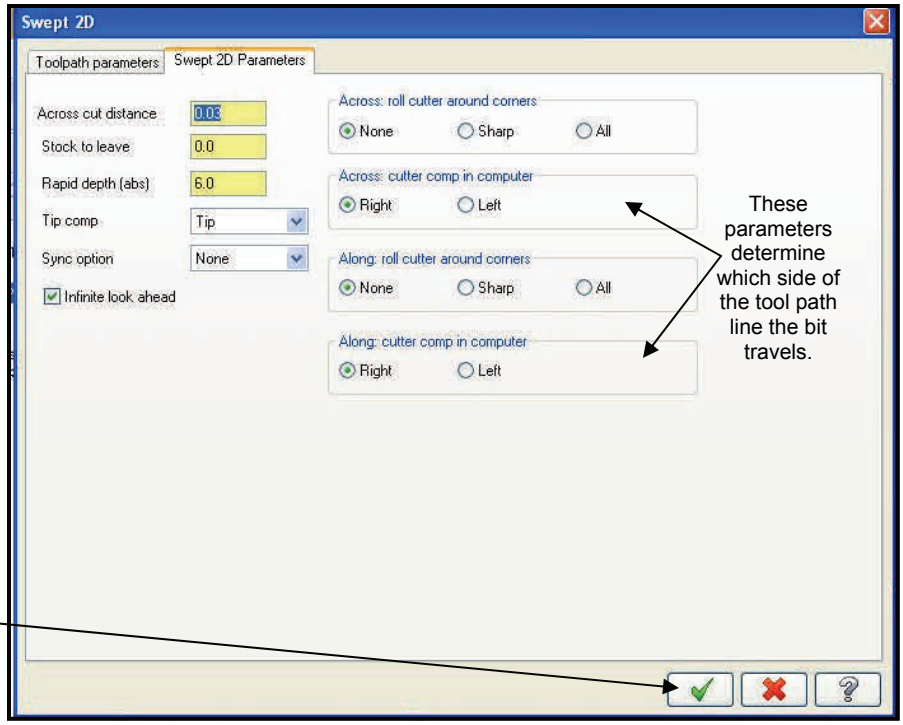
5. Verify the tool appears and the default values are correct for the stone type being cut.

6. Click the "Swept 2D Parameters" tab.



7. Check the following:
   - Across cut distance is set to 0.03.
   - Rapid depth is set to height of material plus 1 inch. For this example that is seven (7) inches.

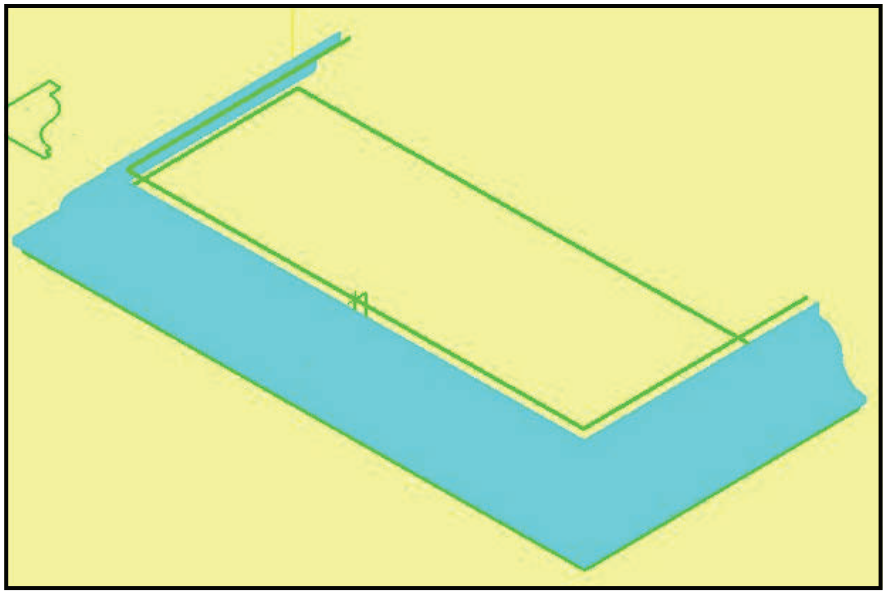
8. At the bottom of this window click the green check mark.



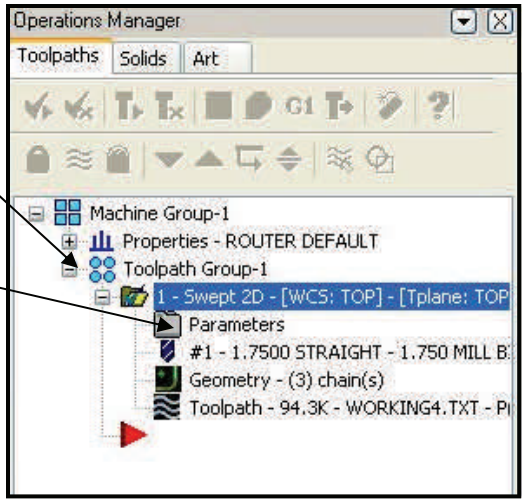
Swept 2D and Contour Example

Task: Swept 2D Setting Toolpath Parameters (cont'd)

The drawing area now displays the toolpath applied to profile. Verify the path is on the outside of the toolpath line. If not, change the cutter comp parameter on the Swept 2D Parameter tab.



The toolpath also appears in the Toolpaths tab of the Operations Manager window. To edit the parameters just double click on the Parameters folder.



= Backplot

= Verify

= Post operations

= High Speed

= Delete operations

= Help

= Select operations

= Regenerate operations

= Toggle Options

= Move arrow

Swept 2D and Contour Example

Task: Extending the Toolpath

The next task is to extend the toolpath on the each side. Figure A illustrates (top view) where the tool is stopping during the Swept 2D cutting on the two sides. Notice that the center of the tool is even with the back line/edge of the profile. To reduce possible breakouts and chipping at the corners it is recommended to extend the tool path beyond the corner. In Figure B, the toolpath has been extended a distance equal to the tool's diameter. Extending the toolpath one half (½) the tool's diameter is the minimum distance that should be used.

The below steps illustrate how to extend the two side toolpath lines and then regenerate the toolpath.

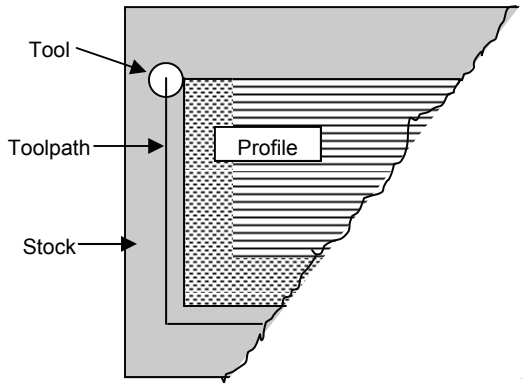


Figure A

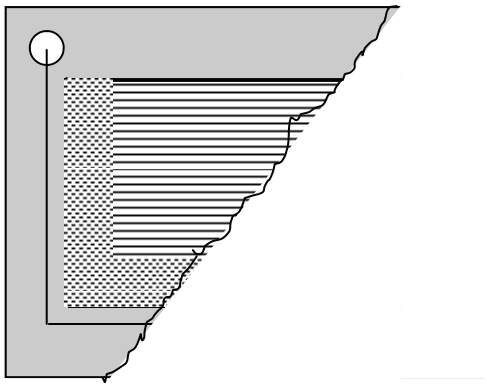
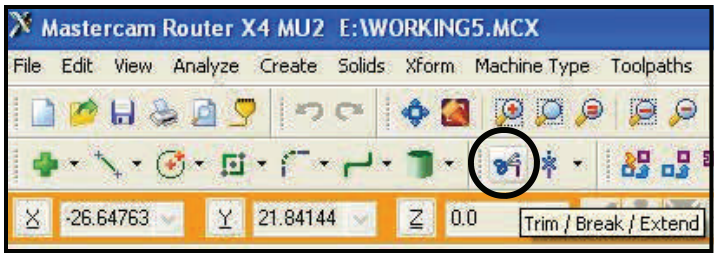
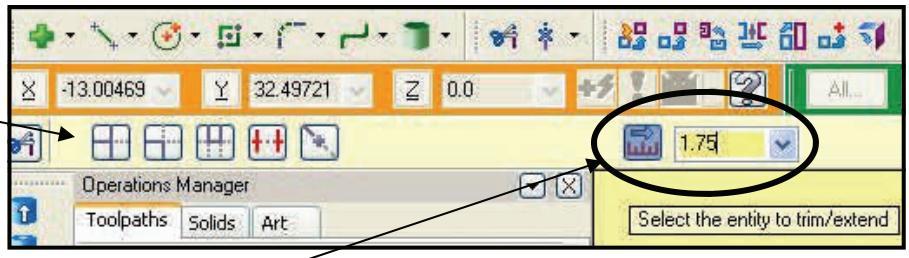


Figure B

1. Start the line extend by clicking on the "Trim/Break/Extend" icon located in the tool bar.



2. Clicking the icon launches icons related to the command and a distance value entry box in the ribbon bar.
3. Enter the distance the line/lines are to be extended. 1.75" was used for this example.



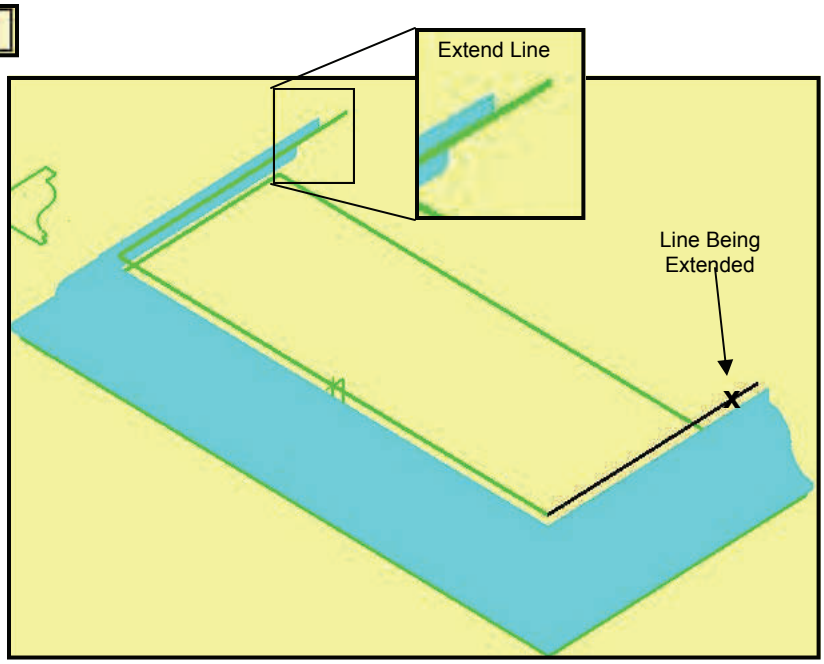
Swept 2D and Contour Example

Task: Extending the Toolpath (cont'd)

The current message is::

Select the entity to trim/extend

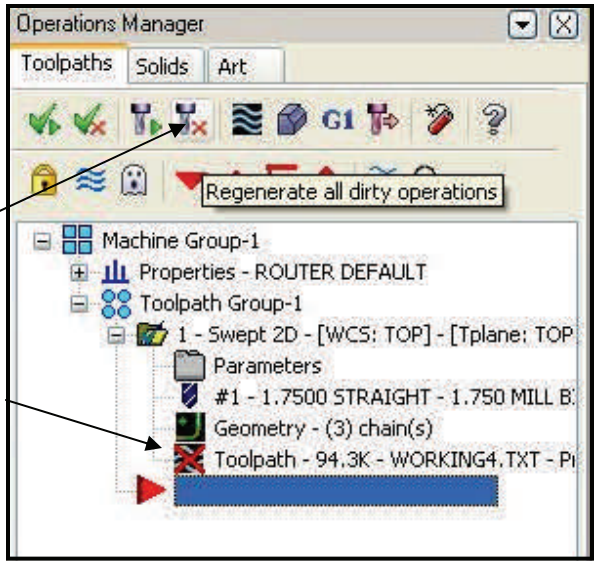
4. Click on the end of the line that is to be extended.



5. When both lines have been extended, click the green check mark in the ribbon bar.

When the lines are extended, the toolpath was changed. The software now marks the tool-path as dirty by placing a red X over the toolpath icon in the Toolpaths Tab.

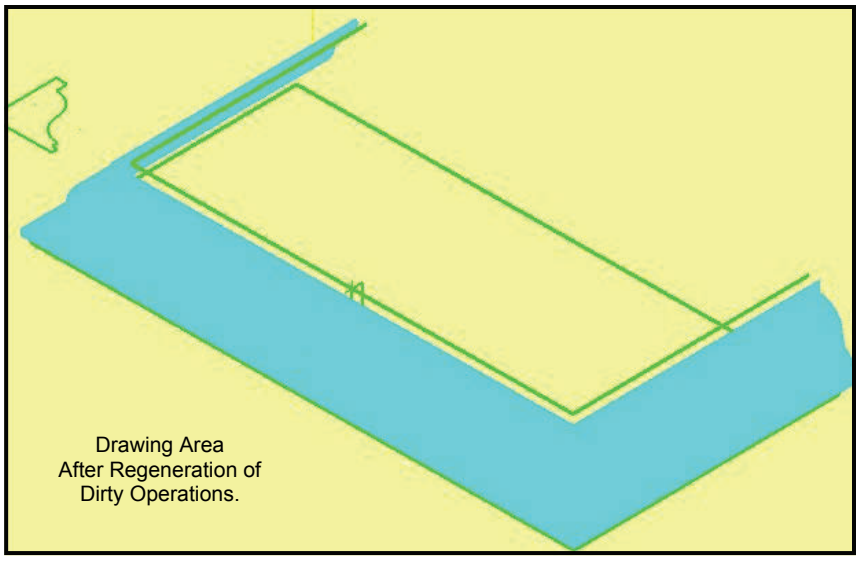
6. To regenerate all dirty operations, click here.



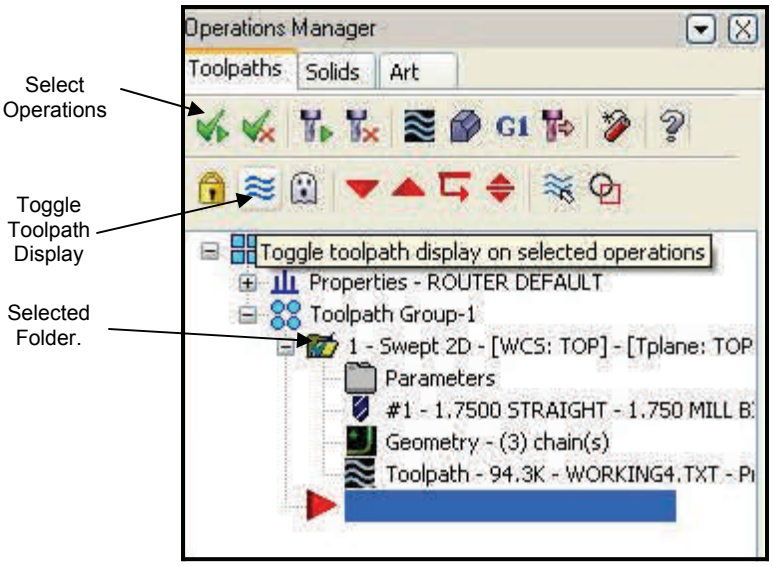
Swept 2D and Contour Example

Task: Extending the Toolpath (cont'd)

After the "Regenerate all dirty operations" command the new toolpath is displayed as shown below.



In the Operation Manager window's tool bar, the Toggle Toolpath Display icon switches the toolpath on all selected operations from off-to-on or on-to-off. An operation is selected if there is a check mark over the operation's main folder.

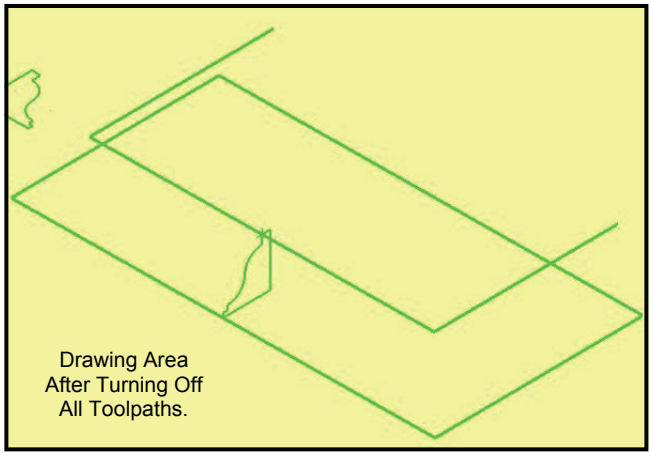


Swept 2D and Contour Example

Task: Contour 1 - Create Toolpath Line (cont'd)

The Swept 2D operation has completed three sides of the profile. It is now time to create a Contour toolpath to remove the part from the stock or slab. The Contour cutting cycle allows the removal of material along the tool boundary. In most case two Contour operations are used. The first cuts at a high speed to within ½ to ¼ inch from the bottom of the stone. The second cuts at a lower speed ⅛ inch below the bottom of the stock.

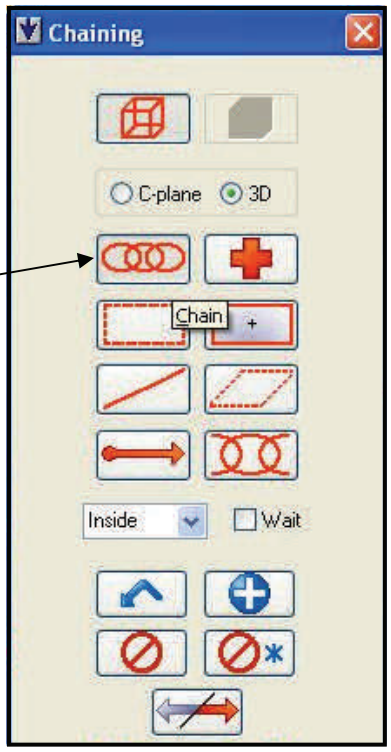
1. Toggle all toolpaths to off.



2. Launch the Contour command.

Swept 2D and Contour Example

Task: Contour 1 - Create Toolpath Line (cont'd)



At this time the Chaining widow is displayed.

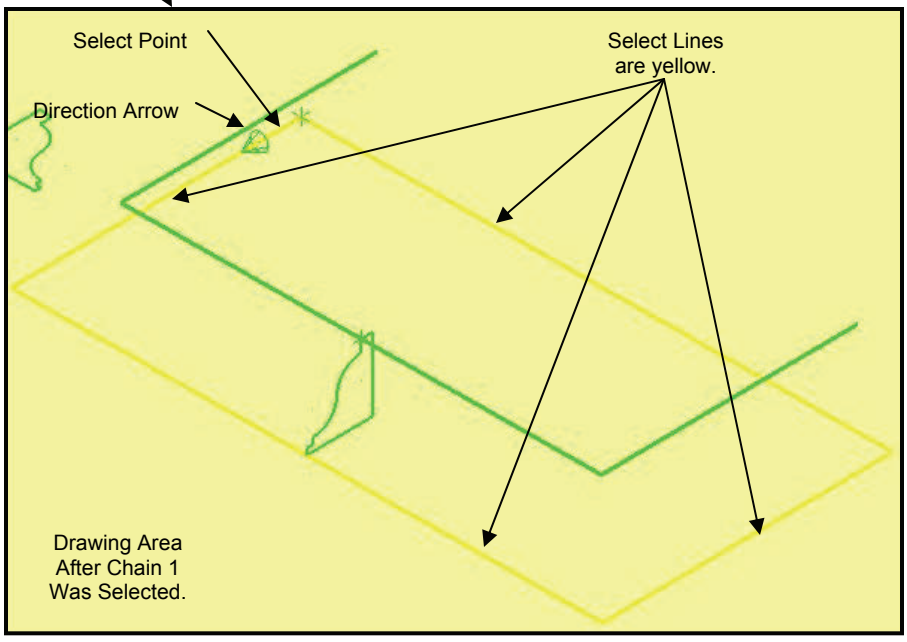
The current message is::

Select Contour chain 1

3. Click the chain button.

4. Since this is a chain only one line has to been selected. The software selects all lines in that chain. Click on a line in the chain.

Verify a direction arrow appears and the lines are selected (yellow).

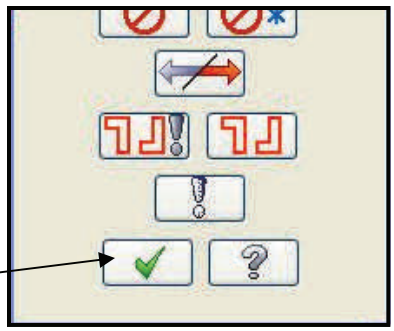


Swept 2D and Contour Example

Task: Contour 1 - Create Toolpath Line (cont'd)

The current message is:

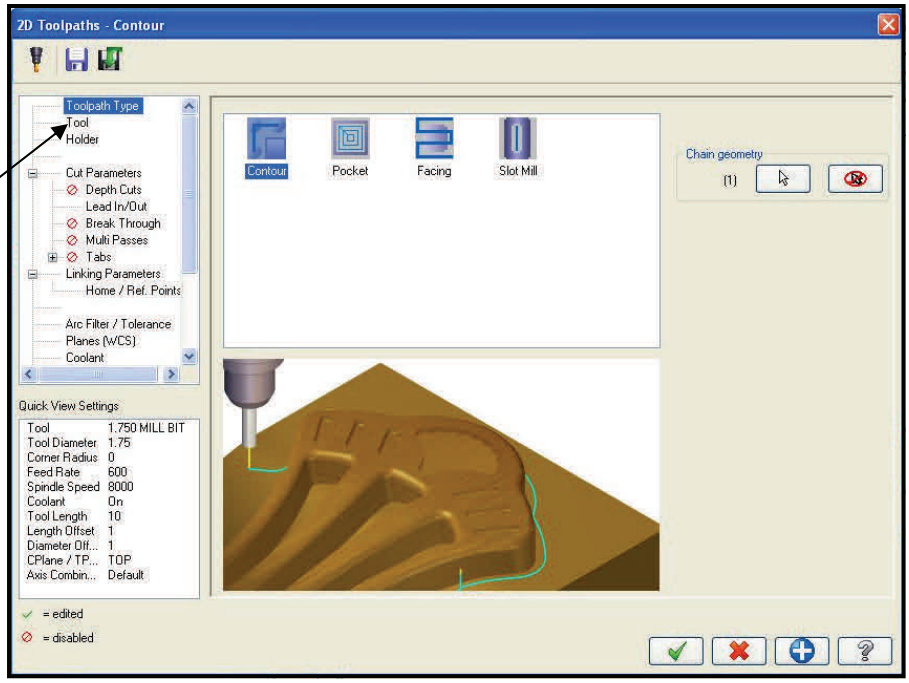
In this example, there was only one chain, thus the chaining window can be closed.



5. Click the green check mark to end the chain operation and launch the 2D Toolpaths Contour window.

6. Verify the Contour symbol is high-lighted.

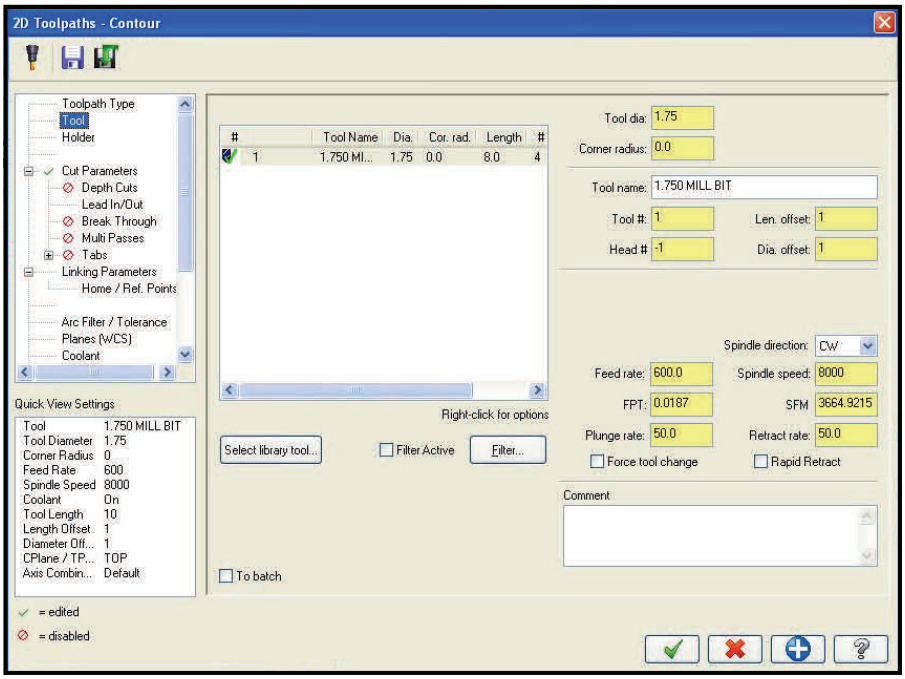
7. Click Tools to display the tools screen.



Swept 2D and Contour Example

Task: Contour 1 - Create Toolpath Line (cont'd)

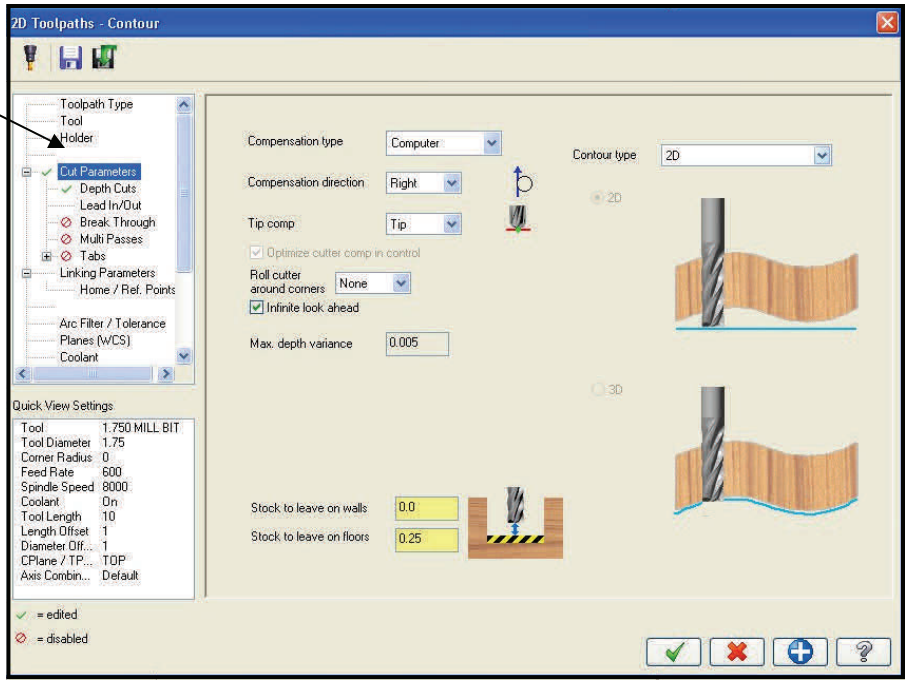
8. Verify the correct tool is loaded. In this example the Contour operation is using the same tool used by the Swept 2D operation, thus it appeared when the window was opened. If different tool is required or no tool was listed, click on the "Select library tool" button.



9. Click here to display the Cut Parameters.

10. For this operation set or check:
    - Contour type set to 2D
    - Compensation direction set to Right
    - Stock to leave on floors set to 0.25

NOTE: Always press the Enter key after changing the "Stock to leave on floors" value.

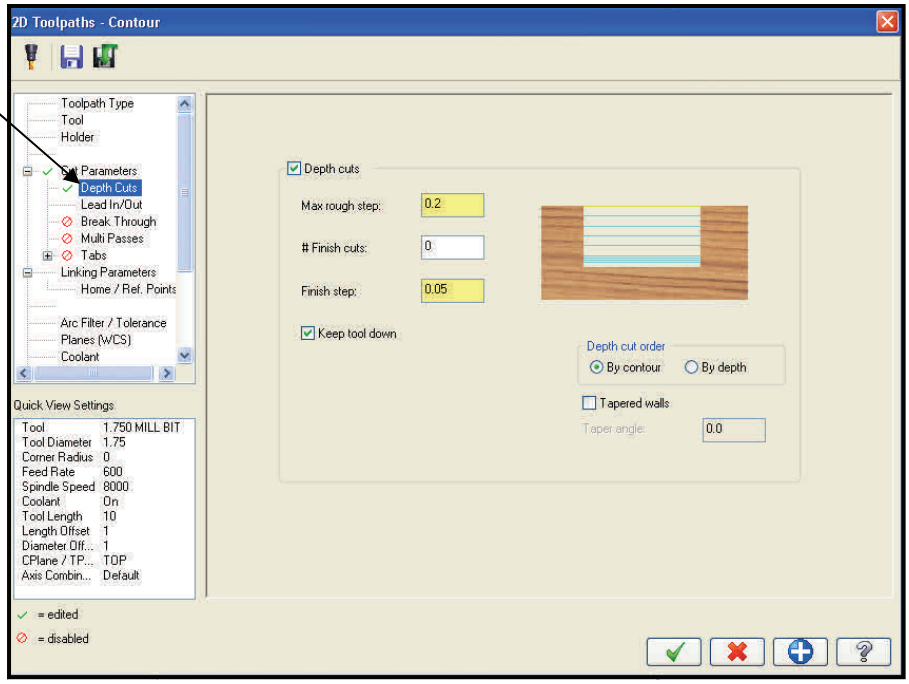


Swept 2D and Contour Example

Task: Contour 1 - Create Toolpath Line (cont'd)

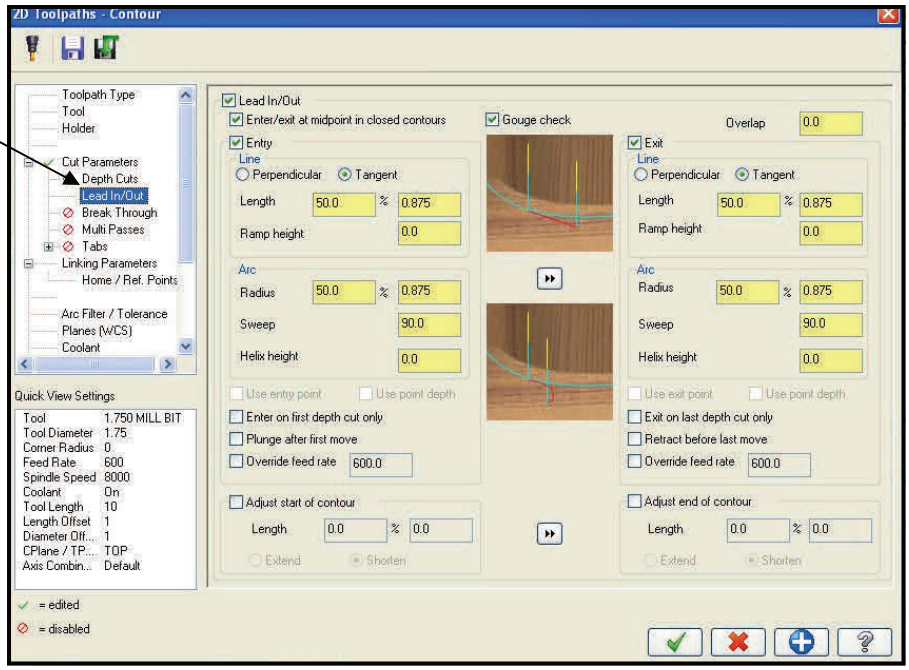
11. Click here to display the Depth Cuts.

12. For this operation set or check:
    - Depth cuts is checked
    - Max rough step is set to 0.2
    - Keep tool down is checked



13. Click here to display the Lead In/Out screen.

14. For this operation set or check all the default values are good.



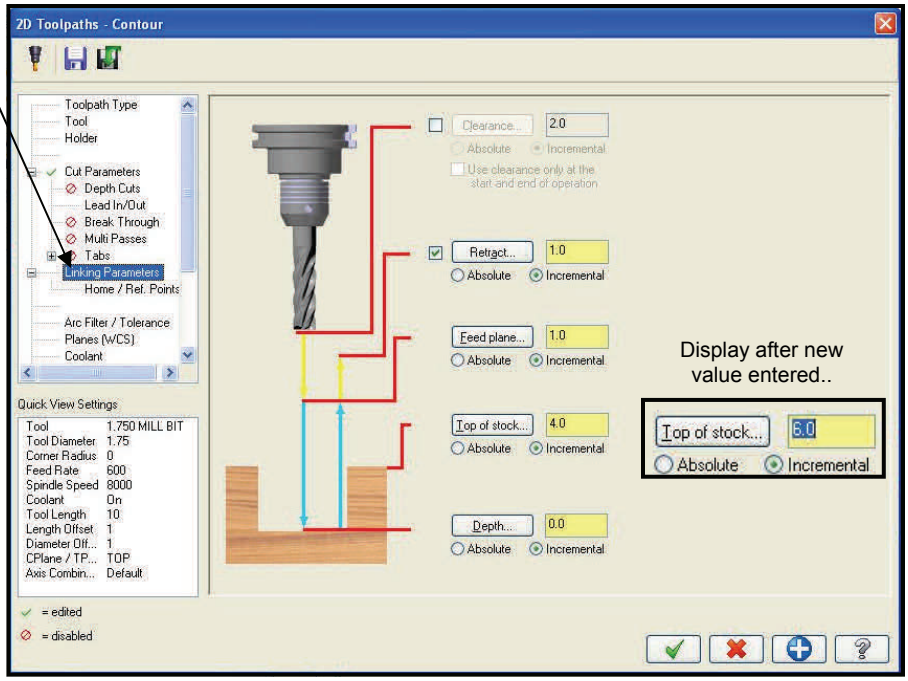
Swept 2D and Contour Example

Task: Contour 1 - Create Toolpath Line (cont'd)

15. Click here to display the Linking Parameters Screen

16. For this operation set or check:
    - Incremental is check for all conditions
    - Top of stock is set to six (6)
    - Depth is set to 0.0

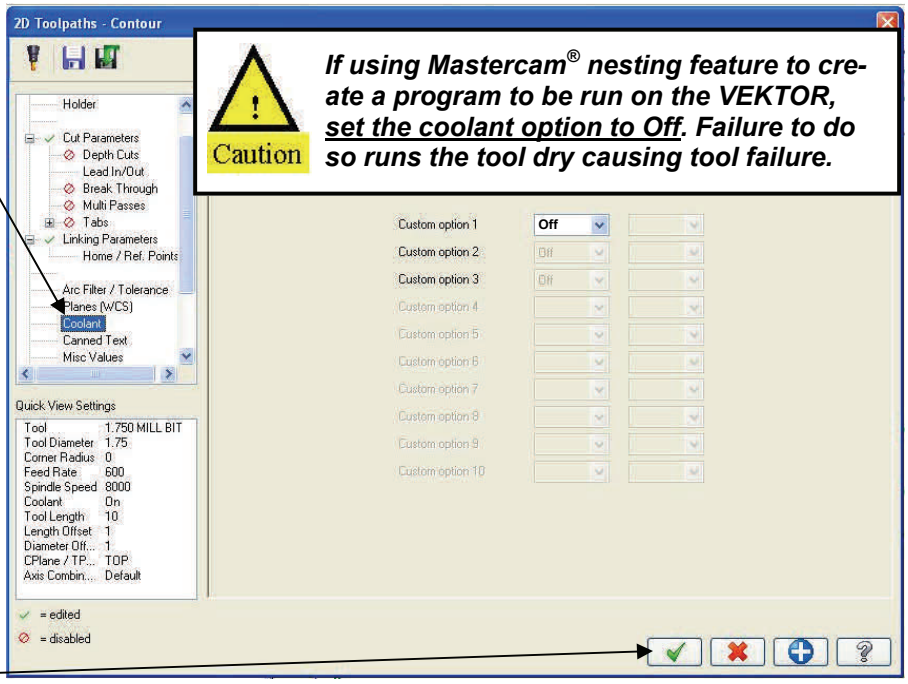
NOTE: Always press the Enter key after changing the "Top of stock" value.



17. Click here to display the Coolant Screen

18. Verify option is set to the **Off** state.

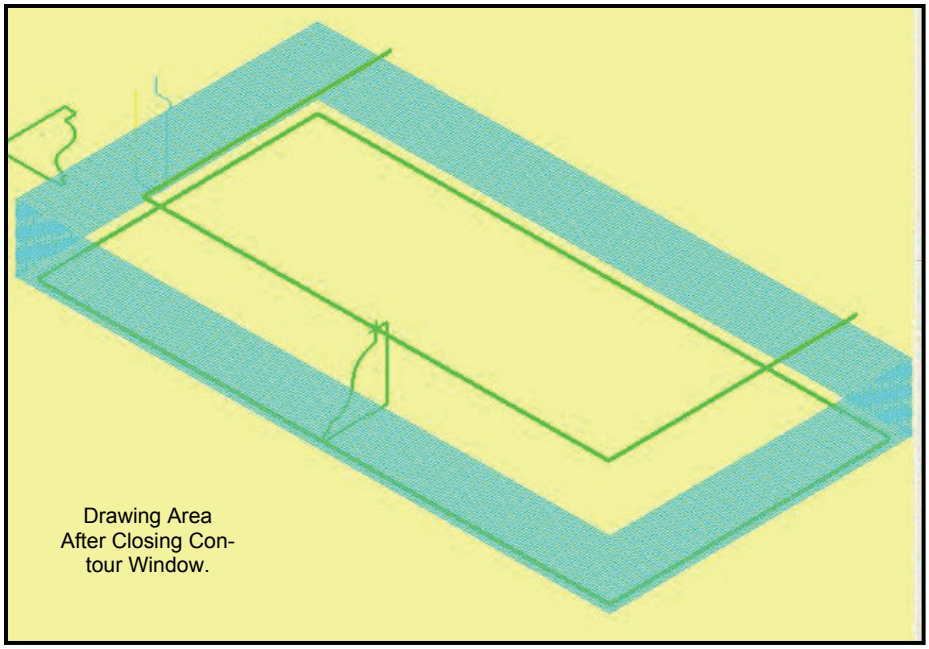
19. Click on the green check mark..



Swept 2D and Contour Example

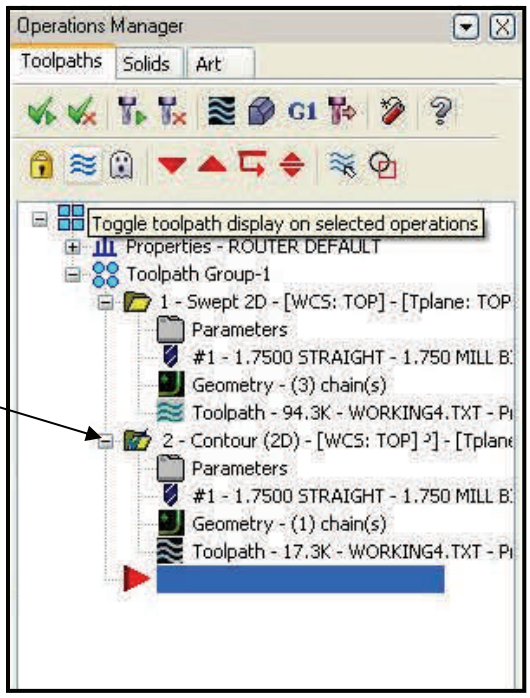
Task: Contour 1 - Create Toolpath Line (cont'd)

The drawing area now displays the toolpath applied to profile. Verify the path is on the outside of the toolpath line. If not, change the cutter comp parameter on the Contour Parameter tab.



Drawing Area After Closing Contour Window.

The Contour toolpath also appears in the Toolpaths tab of the Operations Manager window. To edit the parameters just double click on the Parameters folder.

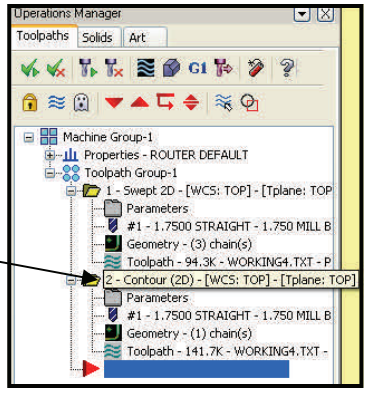


Swept 2D and Contour Example

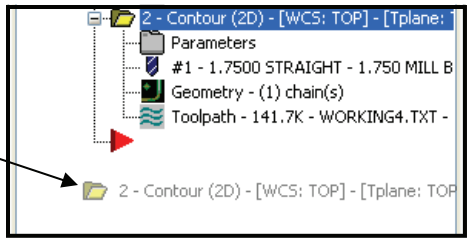
Task: Contour 2

For the most part, Contour two is the same as Contour one. A short cut is to just copy the first contour and insert the copy after the first one. Once copied the parameters can be changed.

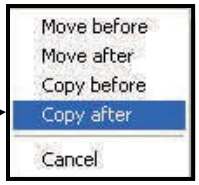
1. Position the cursor over the word Contour.
   A highlight box appears around the text when the cursor is over the line.



2. Depress and hold down the right mouse button, drag down to a point below the insertion arrow and then release the button.



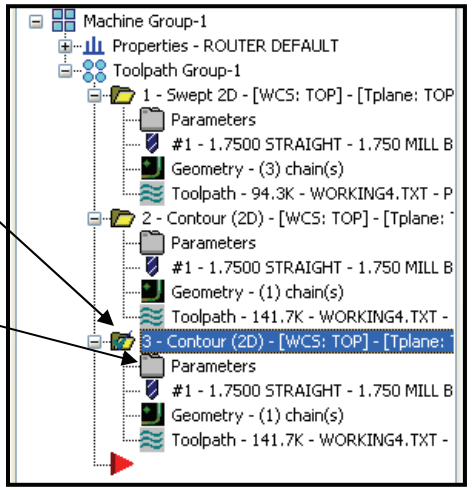
A pop-up menu appears when the mouse button is released.



3. Click (left button) “Copy after”

4. Verify new Contour was created.

5. Double click on Parameters folder to open the parameter screen.

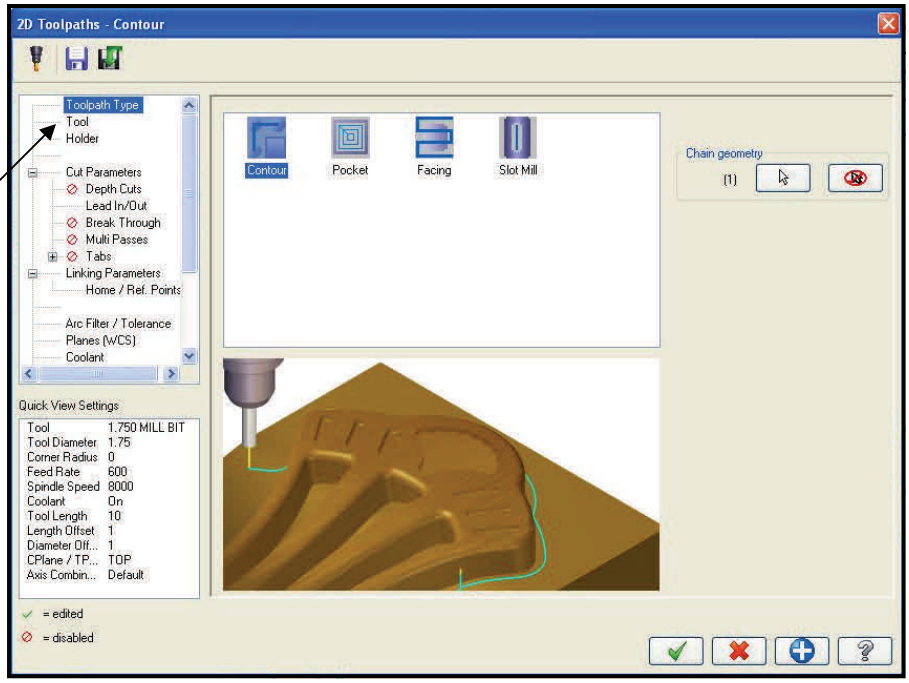


Swept 2D and Contour Example

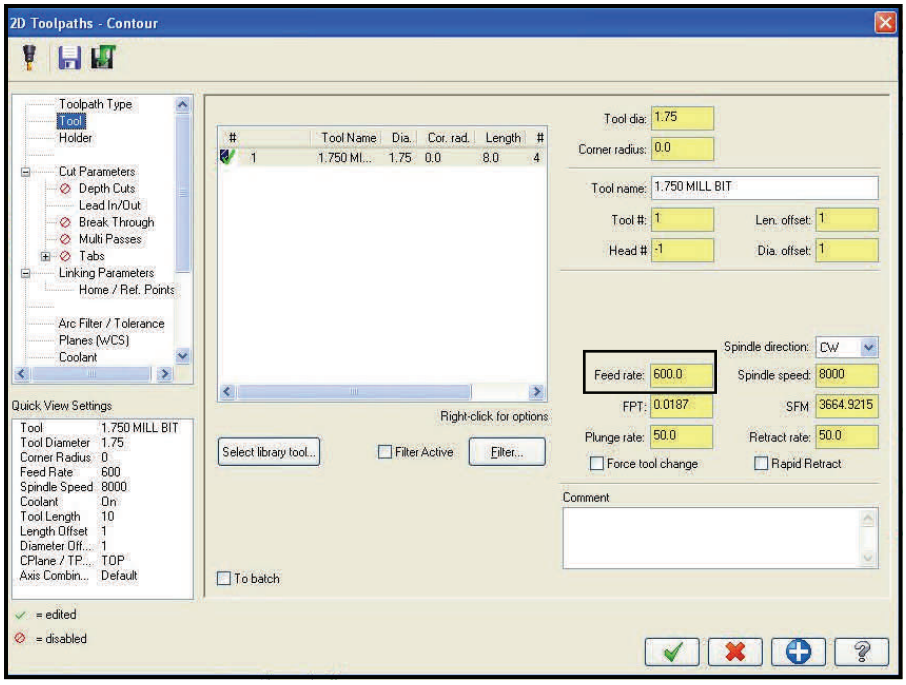
Task: Contour 2

6. Verify the Contour symbol is highlighted.

7. Click Tools to display the tools screen.



8. Verify the correct tool is loaded. In this example the Contour operation is using the same tool used by the last Contour operation, thus it appeared when the window was opened. If different tool is required or no tool was listed, click on the “Select library tool” button. **Slow the Feed Rate down to 50 inches per minute**.



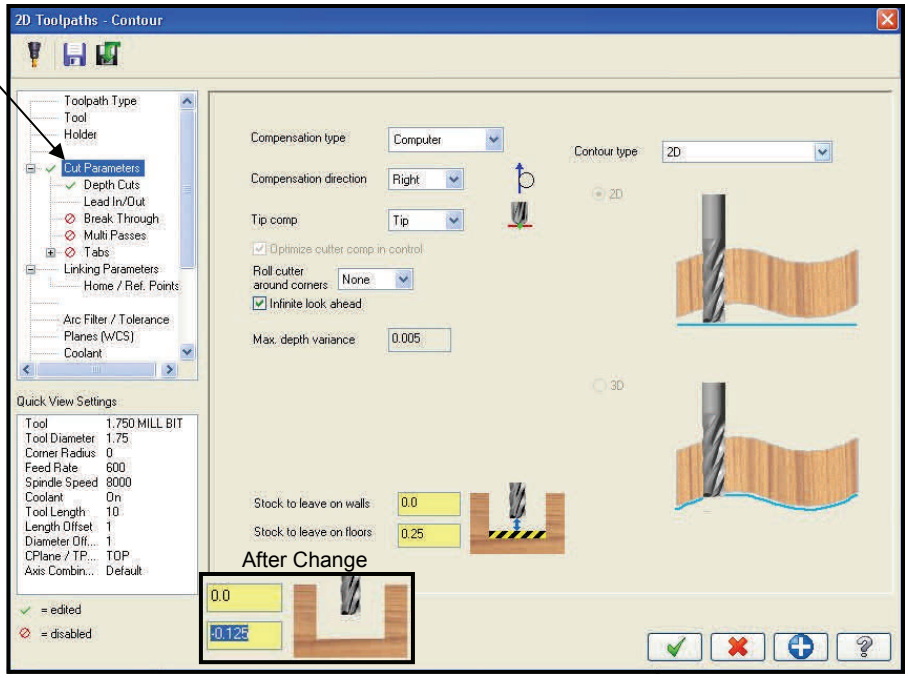
Swept 2D and Contour Example

Task: Contour 2

9. Click here to display the Cut Parameters.

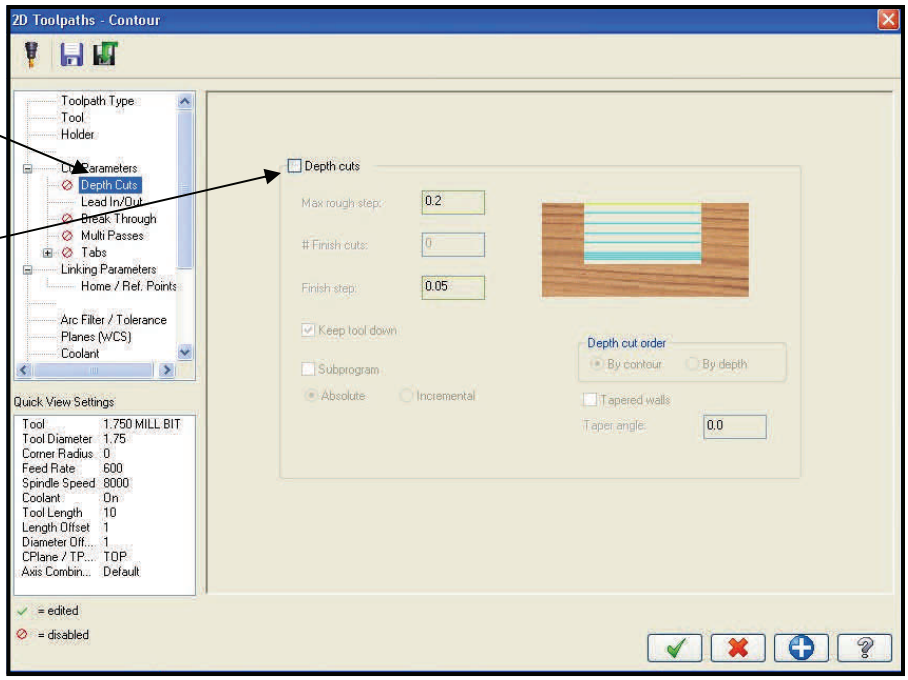
10. For this operation set or check:
    - Contour type set to 2D
    - Compensation direction to Right
    - Stock to leave on floors set to negative 0.125

NOTE: Always press the Enter key after changing the “Stock to leave on floors” value.



11. Click here to display the Depth Cuts.

12. For this operation set or check:
    - Depth cuts is **NOT** checked

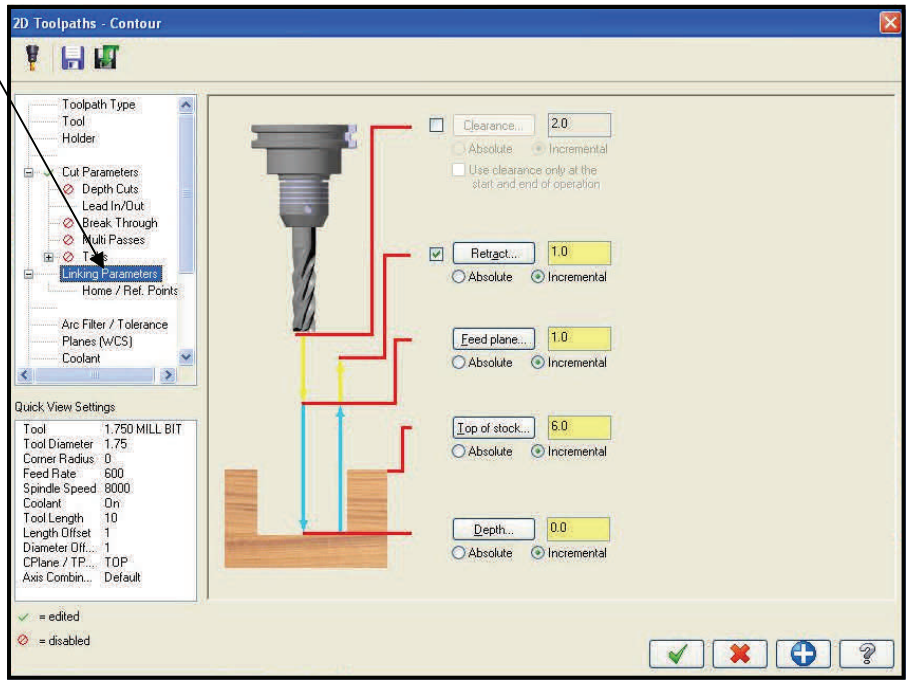


Swept 2D and Contour Example

Task: Contour 2

13. Click here to display the Linking Parameters Screen.

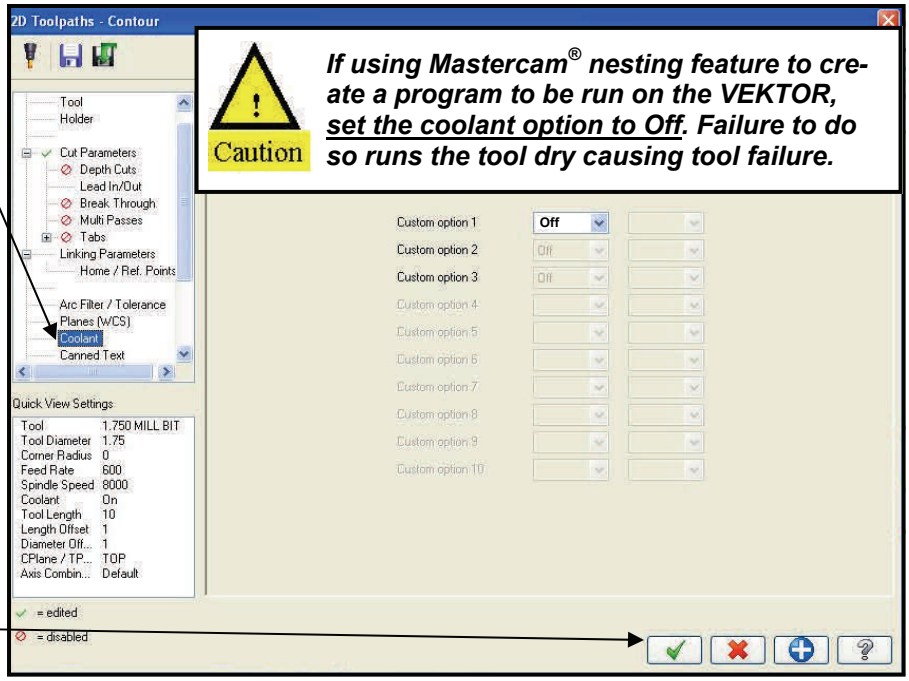
14. Nothing to change on this page, use Contour 1's values. Verify nothing has changed.



15. Click here to display the Coolant Screen.

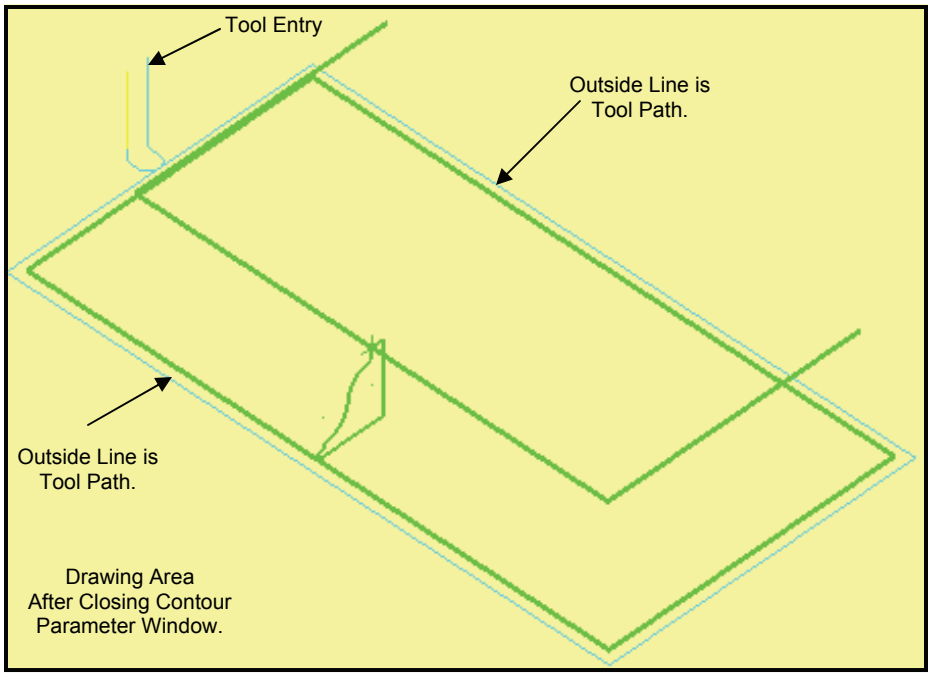
16. Verify option is set to the **Off** state.

17. Click on the green check mark.



Swept 2D and Contour Example

Task: Contour 2

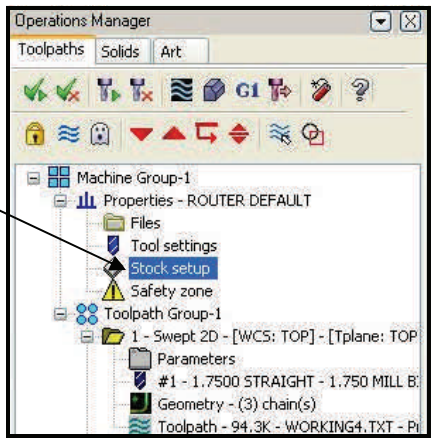


Swept 2D and Contour Example

Task: Display Stock

1. Left click “Stock setup”

This opens the Machine Group Properties window and selects the Stock Setup tab.



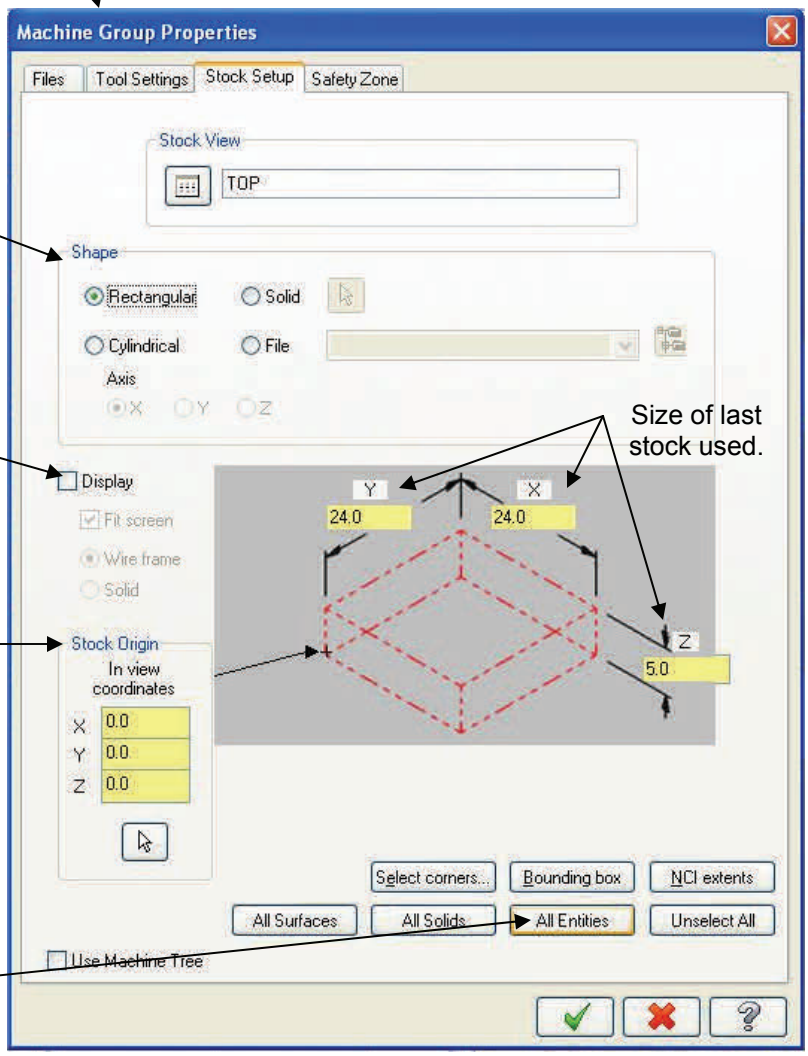
Determines Stock’s Shape

If this box is checked, the stock is displayed in the drawing area. If not checked, the stock is NOT displayed.

Determines distance between the stocks origin (lower left corner) and the point of origin of the profile. The X and Y values are normally negative, which leaves some stock to the left and bottom of the profile. Z is always zero.

Loads the X, Y, and Z dimensions from the drawing area.

Size of last stock used.

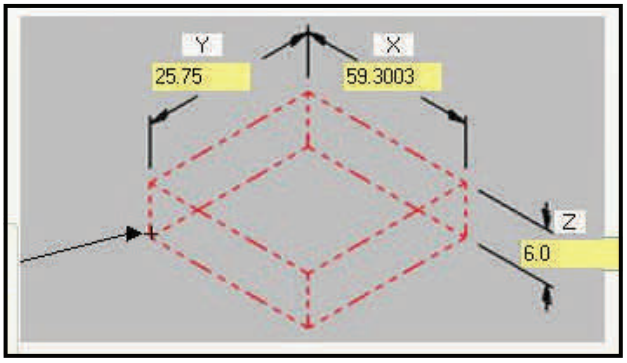


Swept 2D and Contour Example

Task: Display Stock

2. Check these boxes or buttons: Display, Fit Screen and Wire Frame

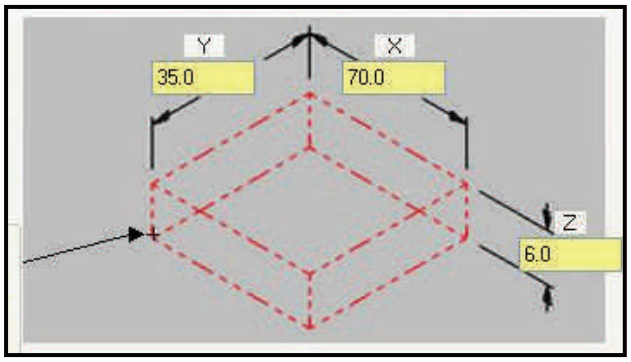
3. Click



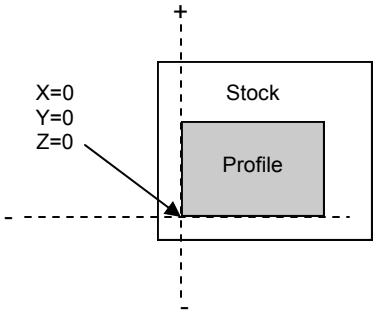
The Y, X and Z entries now display values from the drawing.

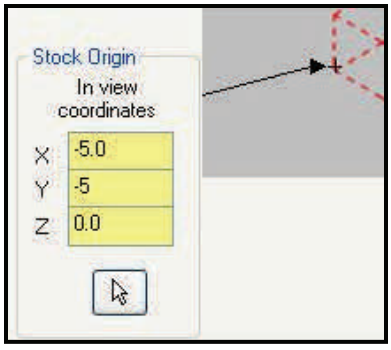
At this time the operator would change the Y, X and Z values to size of the slab being placed in the cutting area. For this example ten (10) inches are added to the Y and X dimensions. The Z dimension is correct.

4. Enter the Y and X dimensions of the slab.



Stock Origin is the lower left corner of the stock or slab. The stock origin values are referenced to the Mastercam® drawing point of origin coordinates which are X=0, Y=0 and Z=0. In most case some stock should be left between the edges of the profile and the slab/stock. This places the stock's lower left corner in the negative quadrant.





5. Enter X and Y Stock Origin values. For this example negative 5.0 has been entered for both.

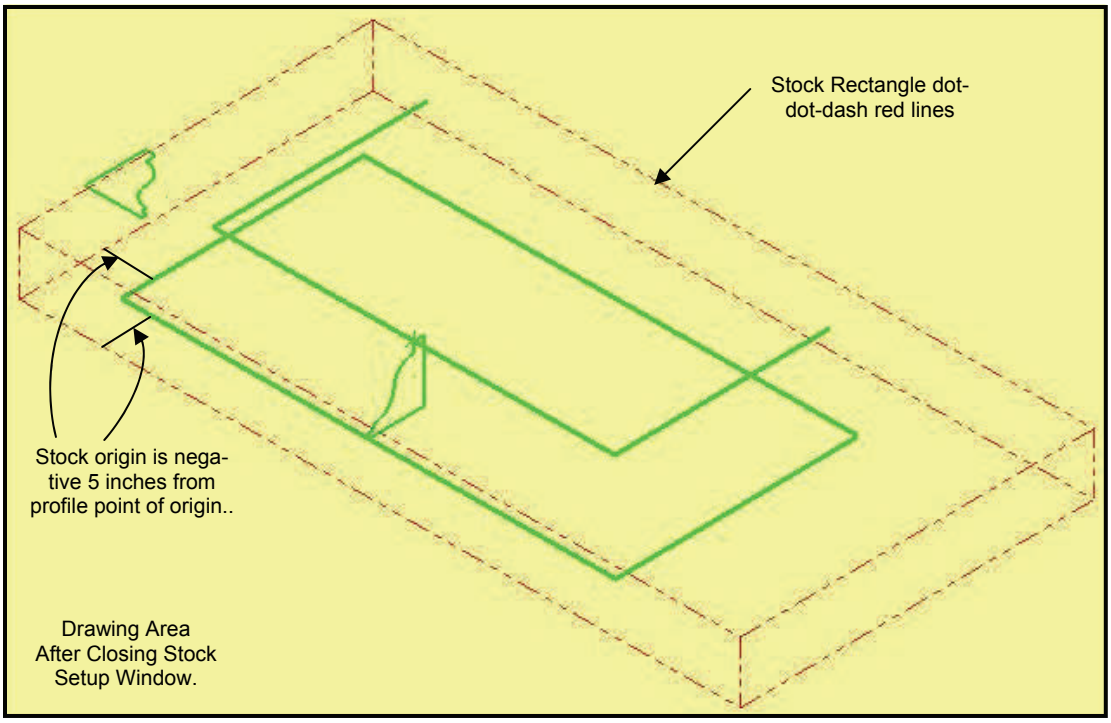
6. Click the green check mark.

Swept 2D and Contour Example

Task: Display Stock

After closing the Stock Setup window the drawing area should appear as shown below. The stock is in wire frame format using red dot-dot-dash line style. Also notice that the stock's origin is to the right and below the profile's point of origin.

The next task is run the verify command.

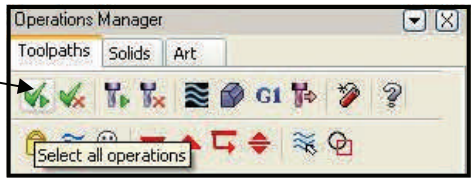


Swept 2D and Contour Example

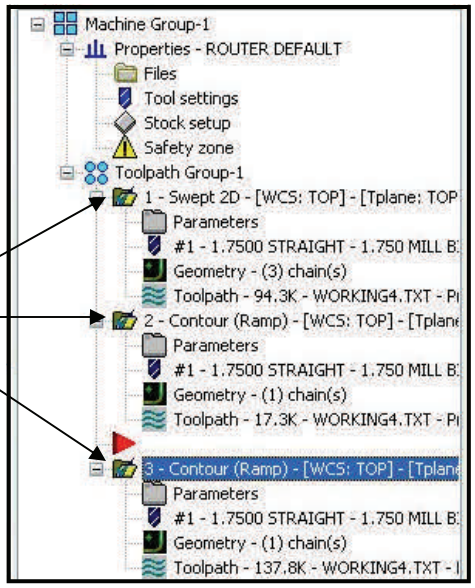
Task: Verify Operation

Verify is the process of running the toolpath as an actual machine would to make sure there are no tool crashes and that the profile cuts are in the correct position. The verification uses the stock material and runs only the selected operations.

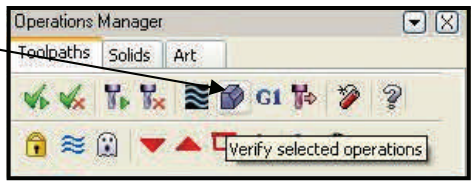
1. Click “Select all operations” button.



2.. Verify that all operations have been selected. Selected operations have check mark on top of the operation folder.



3. Click “Verify selected operations” button.

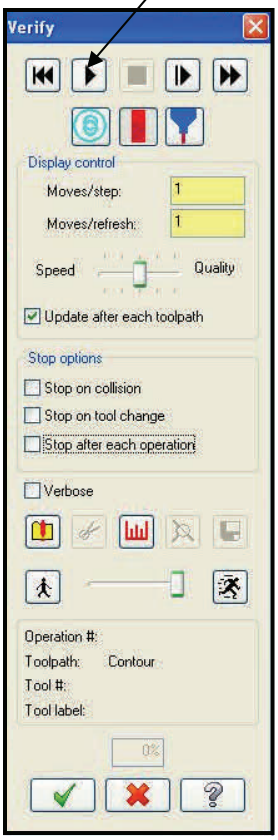


Swept 2D and Contour Example

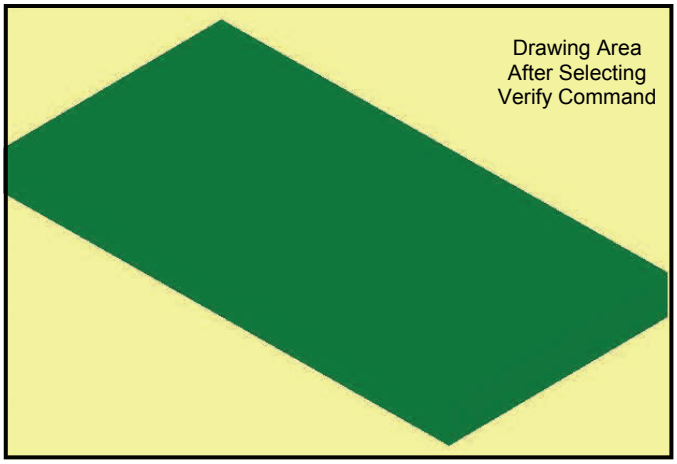
Task: Verify Operation (cont'd)

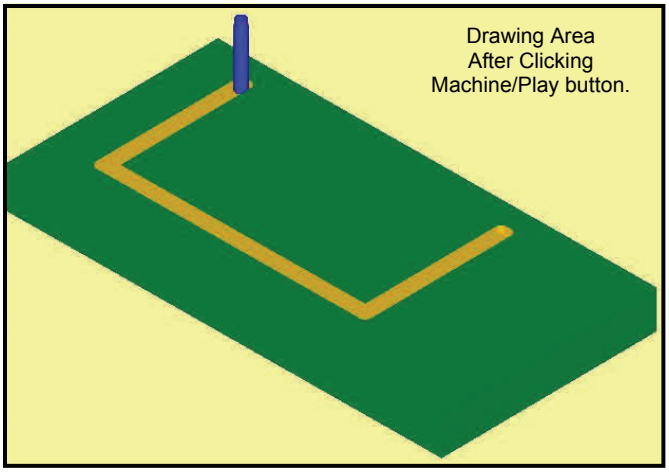
At this time, the red wireframe is now a green solid rectangle and the Verify window is overlaid on the screen.

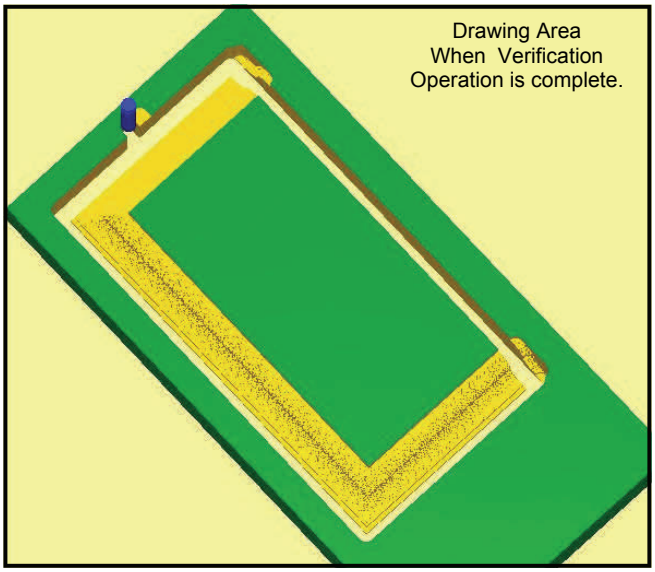
To start the verify process, click the "Machine/Play" button



When complete click the green check mark button to close the Verify window and return to the normal drawing area.







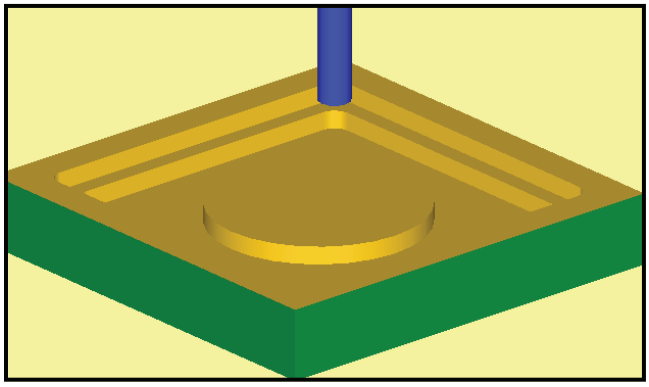
This page intentionally left blank.

Facing and Pocket Example

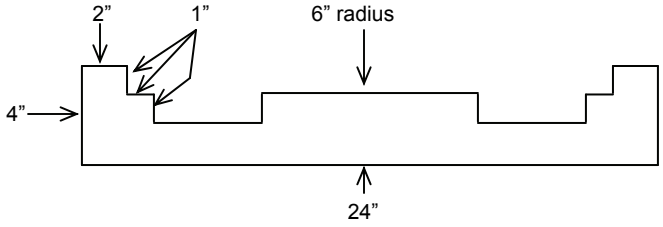
Introduction

This example illustrates the steps to create a facing and pocket tool path.

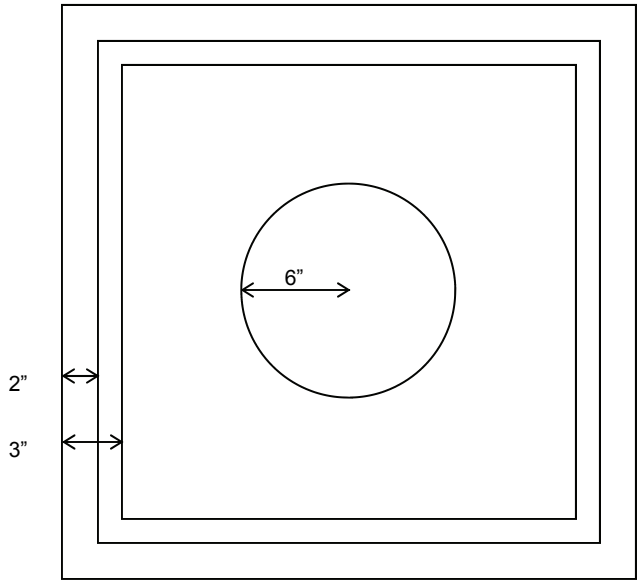
This figure illustrates the shape of the final product that is being created in this example.



A basic cross sectional drawing gives the dimensions. (not drawn to scale)



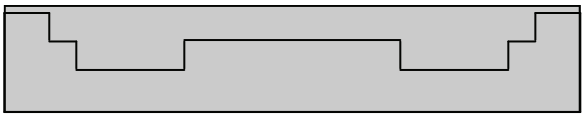
Top view drawing - The outside square has 24 inch sides. The second square is offset 2 inches from the outside square. The third square is off set 3 inches from the outside square. This circle has a 6 inch radius centered in the squares.



Facing and Pocket Example

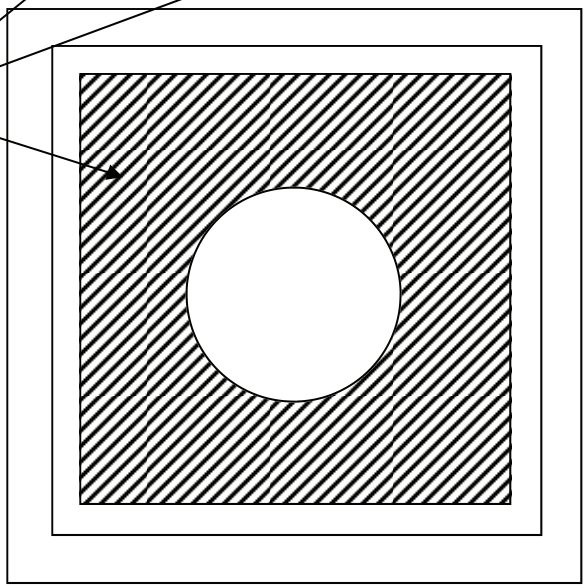
Introduction

The stock material (gray shaded area) being used for this profile is 4½ high. As seen on the last page the finished product is only 4 inches high. The extra ½ inch is removed using a facing toolpath.



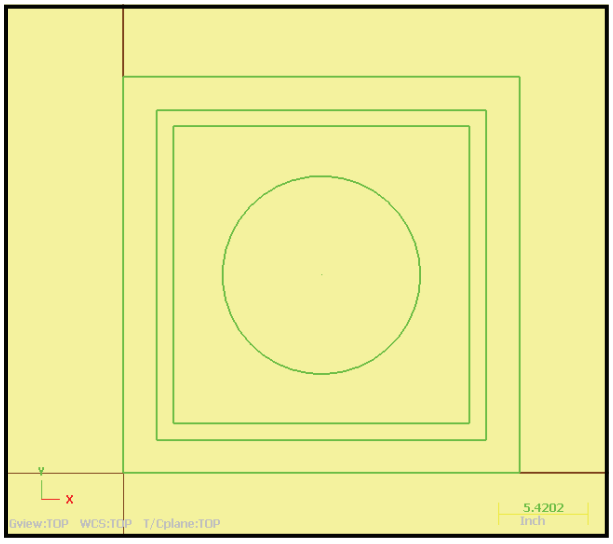
The second toolpath is a pocket which removes one inch of material for this area.

The third toolpath is also a pocket with removes one inch of material between the inside square and the circle.



The first step is to create a top view drawing in Mastercam®. The basic commands are:

- Rectangle 24 by 24 inches
- Offset (copy) rectangle to the inside by 2 inches
- Trim ends of offset to create second square
- Offset (copy) outside rectangle to inside by 3 inches
- Trim ends of offset to create third square
- Draw lines corner to corner to create the letter “X” in the third square
- Draw a 6 inch radius circle with using intersection of above lines as the center



Facing and Pocket Example

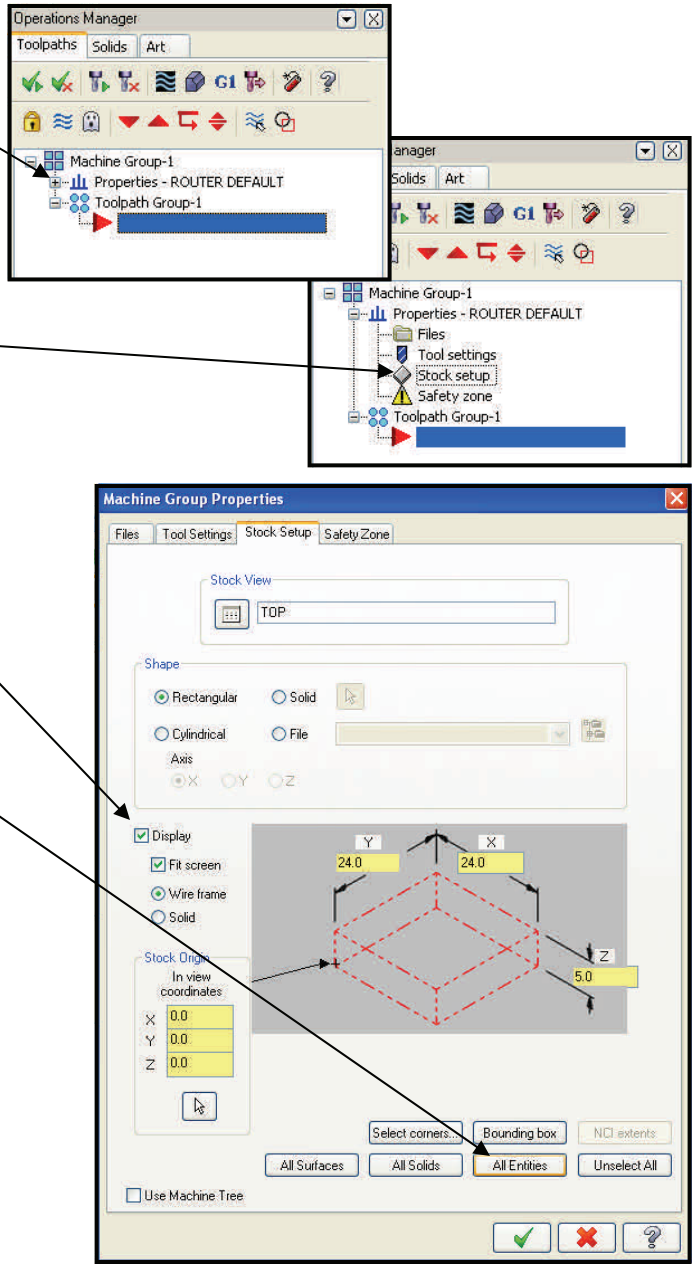
Task: Stock Setup

To view the stock material open (click) the Properties - ROUTER DEFAULT folder.

Click here to open the stock setup screen

Verify the “Display” is checked.

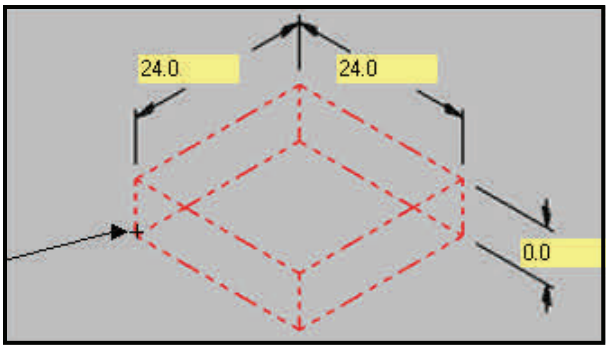
Click on the “All Entries” button. This loads the Y, X and Z dimensions from the current drawing. (See next page)



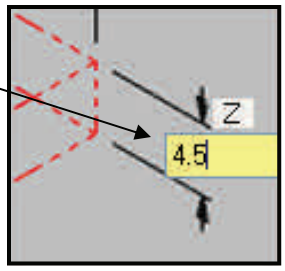
Facing and Pocket Example

Task: Stock Setup (cont'd)

The dimensions of the outside square are used, thus X and Y are both 24. Since this is top view drawing there is no Z value, thus 0.0 was loaded.



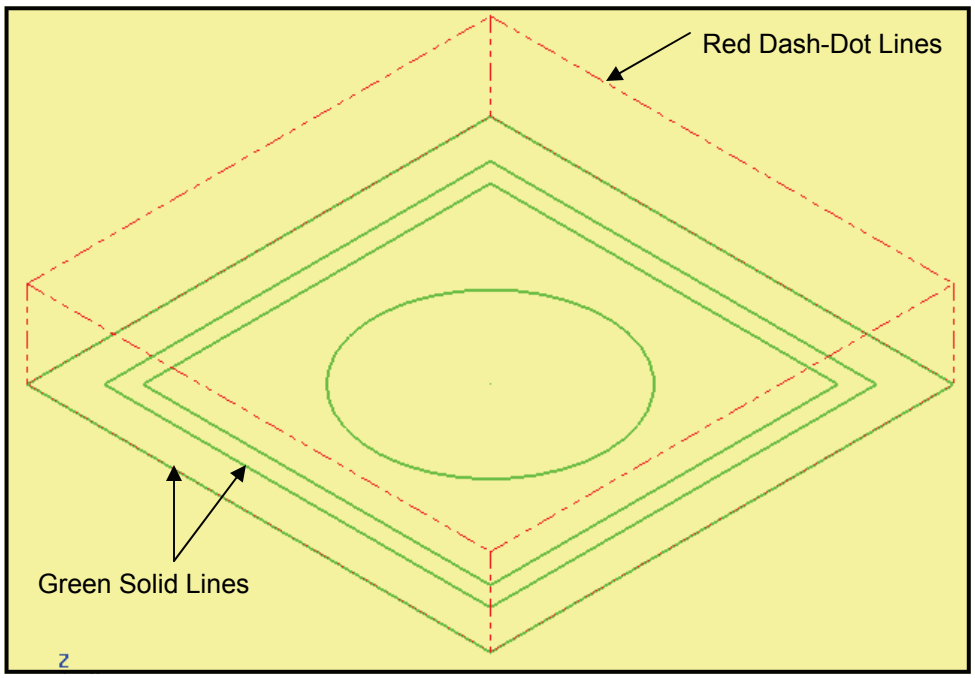
In this example the stock stone is 4.5 inches tall, thus 4.5 is enter via the key-board as the Z dimension.



When done, press

Change to Isometric view (right click on the drawing area, then select).

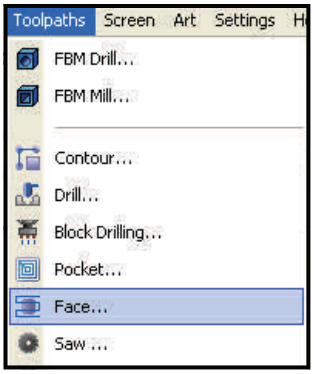
The drawing area should appear as shown below. The stock is shown in red dash-dot lines and the drawing is shown in solid green lines.



Facing and Pocket Example

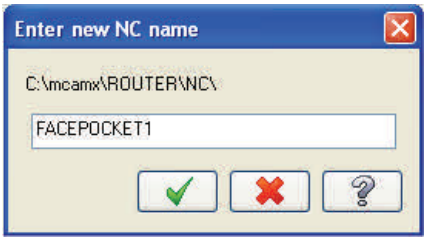
Task: Creating the Facing Toolpath

1. Select “Toolpaths” then “Face”

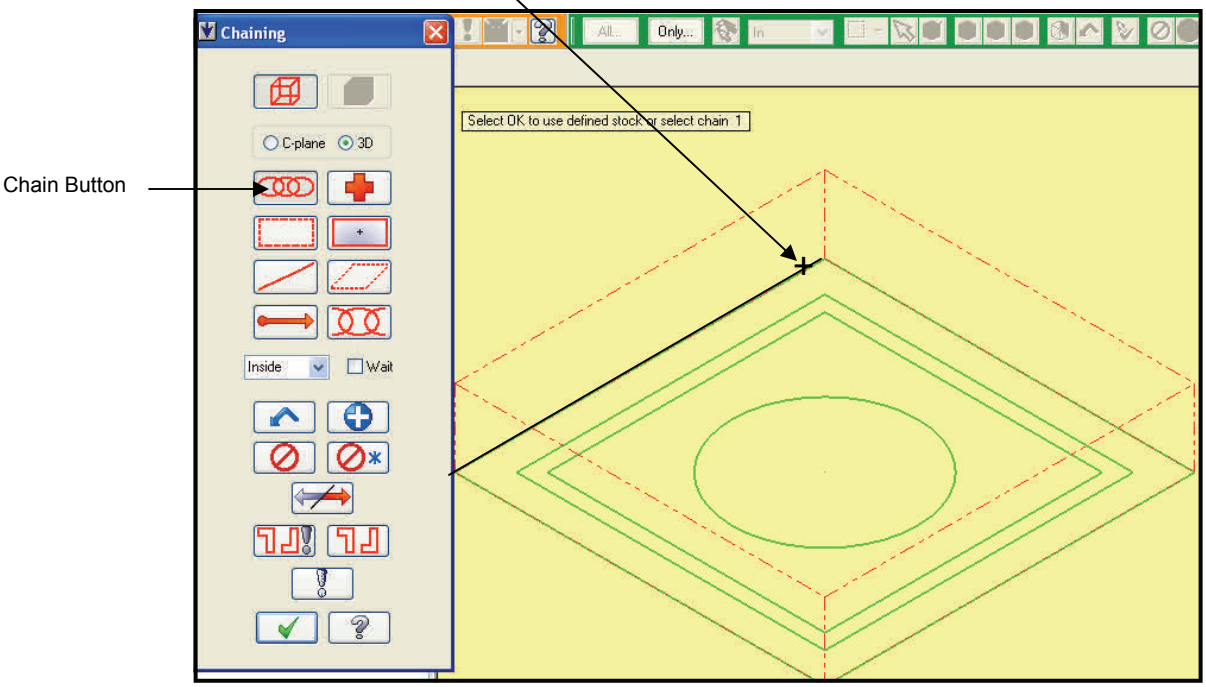


2. When asked for a new NC name, click on the green check mark.

3. On the Chaining window, click the “Chain” button.



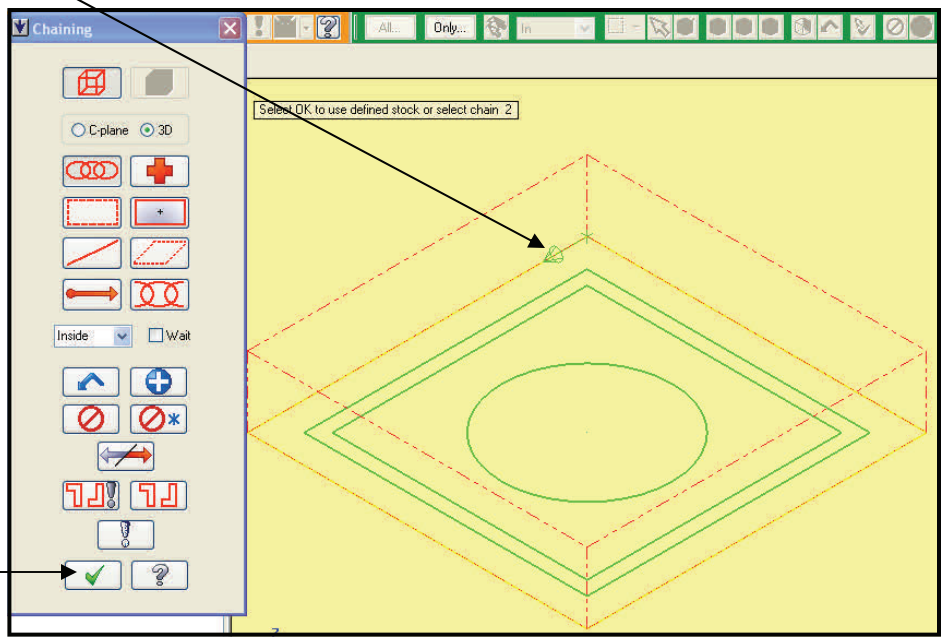
4. Click on the line where the chain should start.



Facing and Pocket Example

Task: Creating the Facing Toolpath (cont'd)

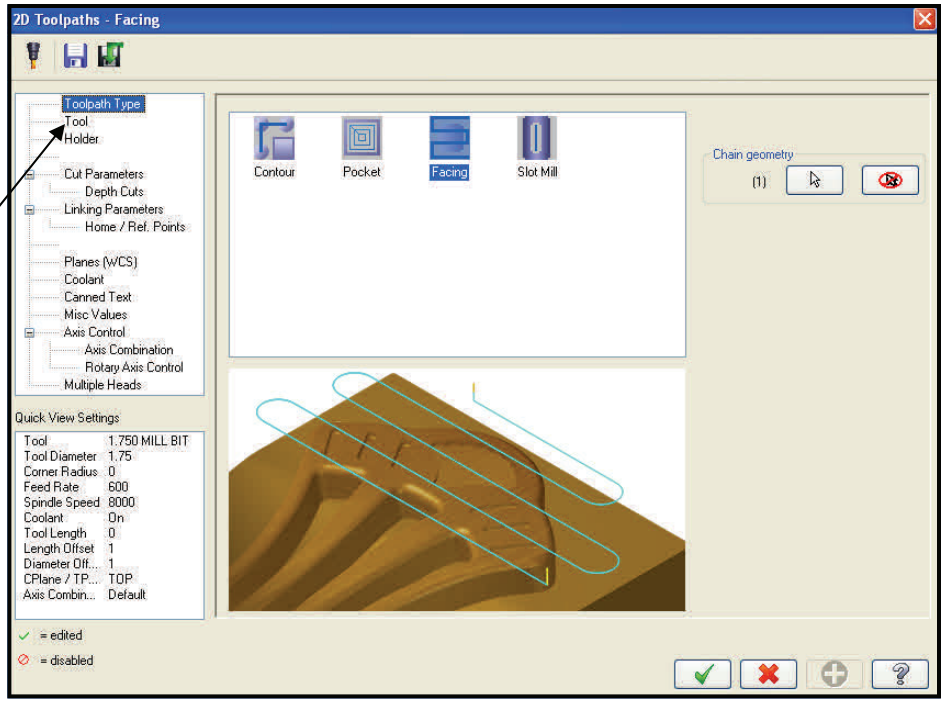
5 The arrow indicates the direction of the chain. All selected lines are yellow.



6. Click green check mark.

7. Verify the 2D Toolpaths - Facing screen appears.

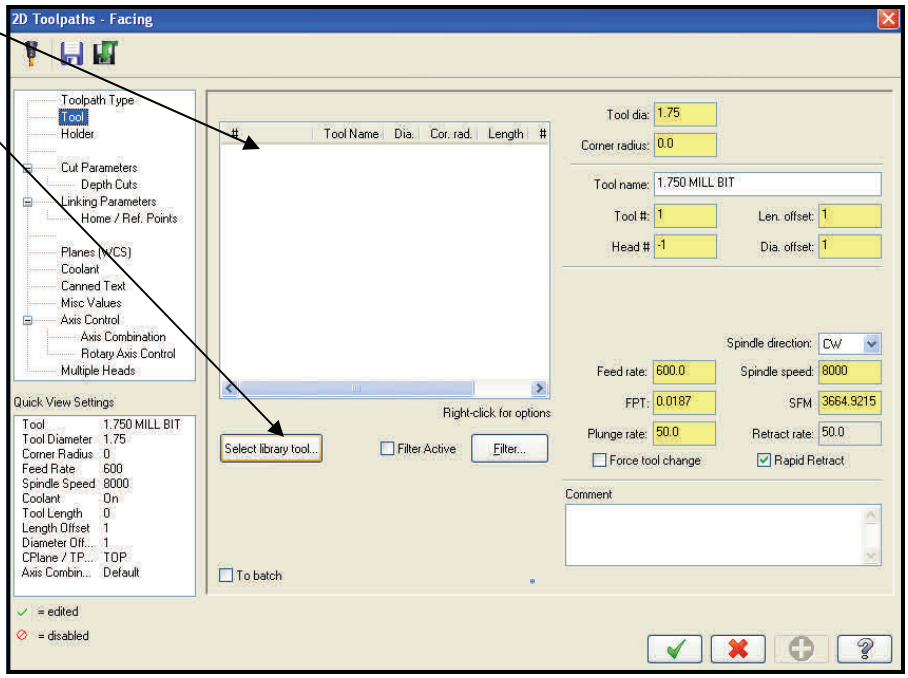
8. Click on "Tool"



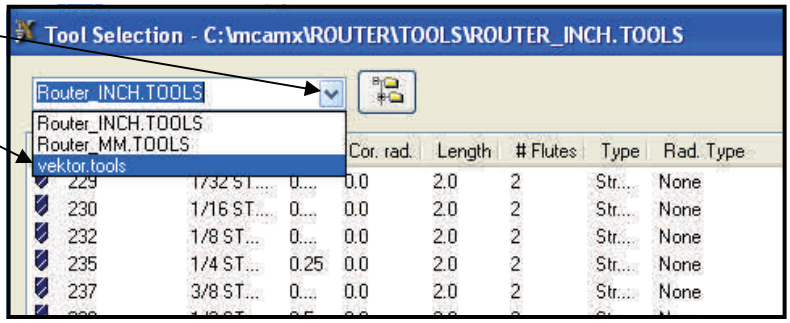
Facing and Pocket Example

Task: Creating the Facing Toolpath (cont'd)

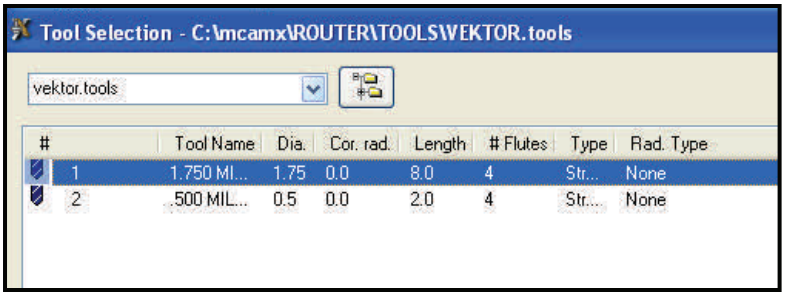
9 If no tool is selected, click "Select Tool Library . ." button



10. From the pull down menu, select "vektor tools".



11. Select desired tool. In this example, tool # 1 a 1.750 MILL tool has been selected.



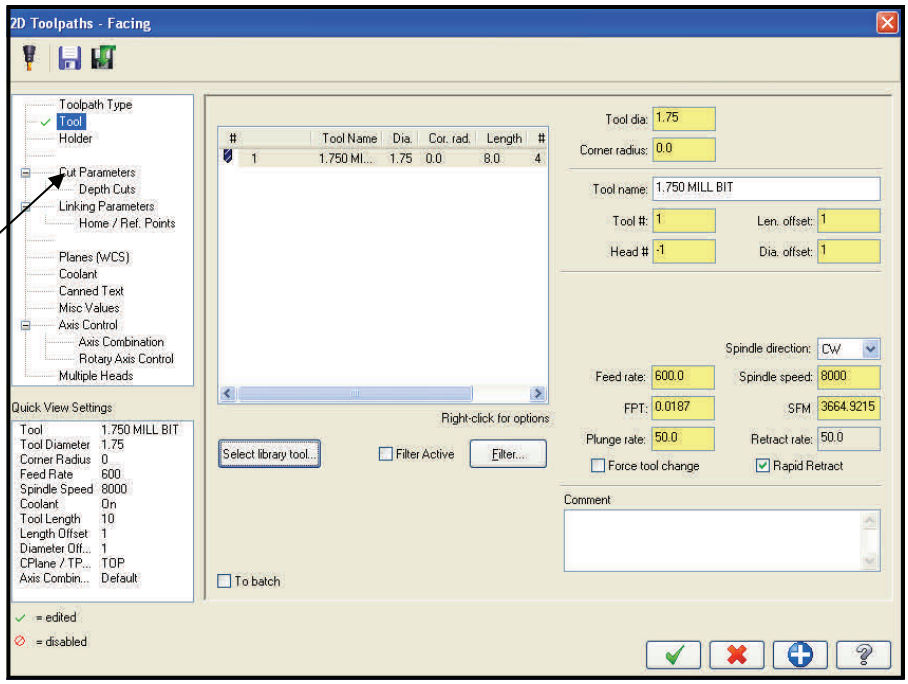
12. Click

Facing and Pocket Example

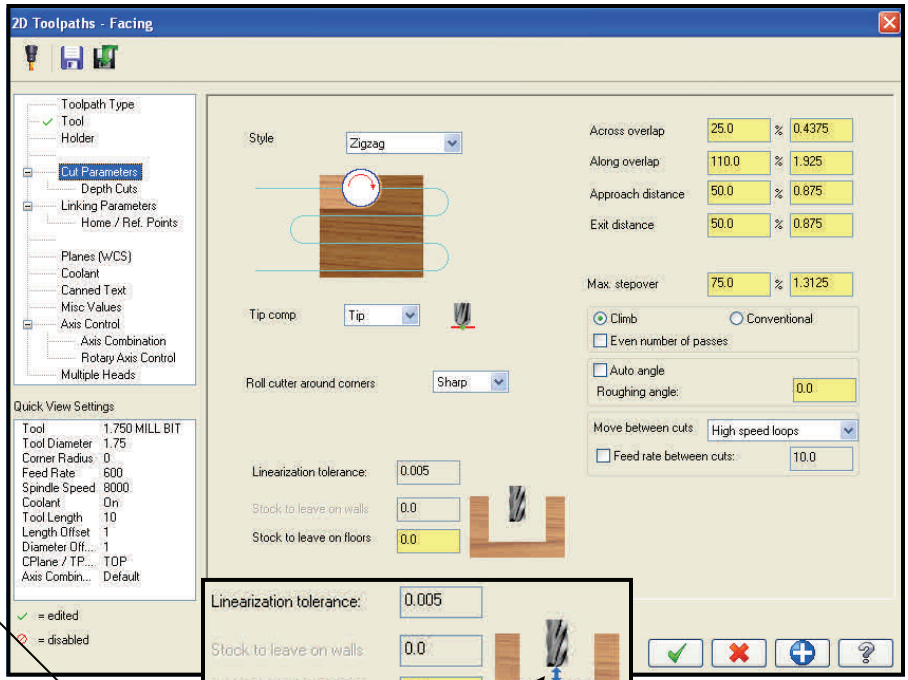
Task: Creating the Facing Toolpath (cont'd)

13 Verify the tool and the default loaded and are correct.

14. Click Cut Parameters.



15. This screen displays the cut parameters. In this example, ZigZag has been selected as the Style. For most operations the default values shown can be used.



16. Set the "Stock to leave on floors" to four (4).

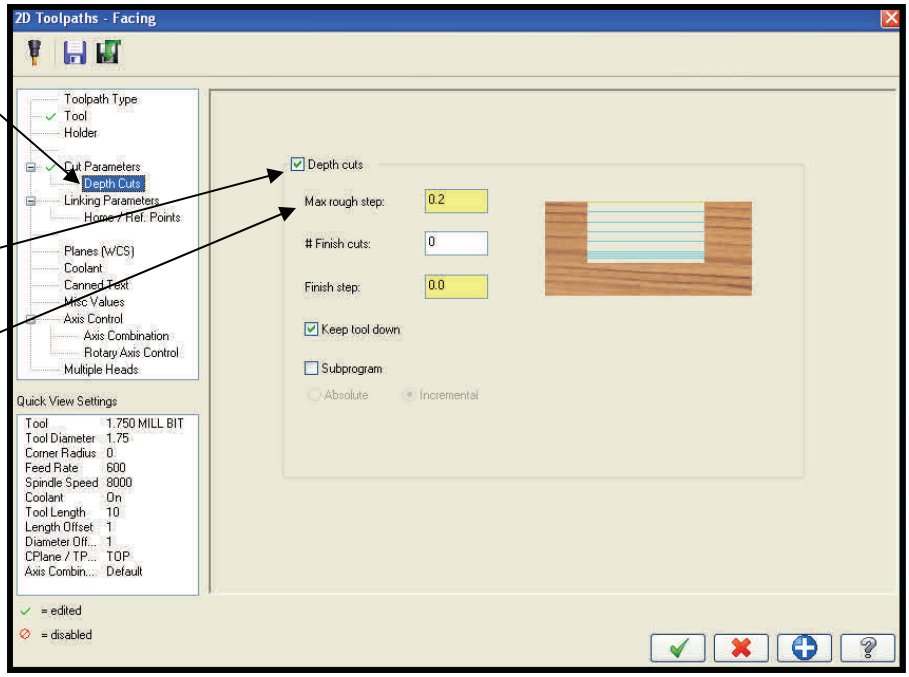
Note: Always press "Enter" on the keyboard after the changing the value.

Facing and Pocket Example

Task: Creating the Facing Toolpath (cont'd)

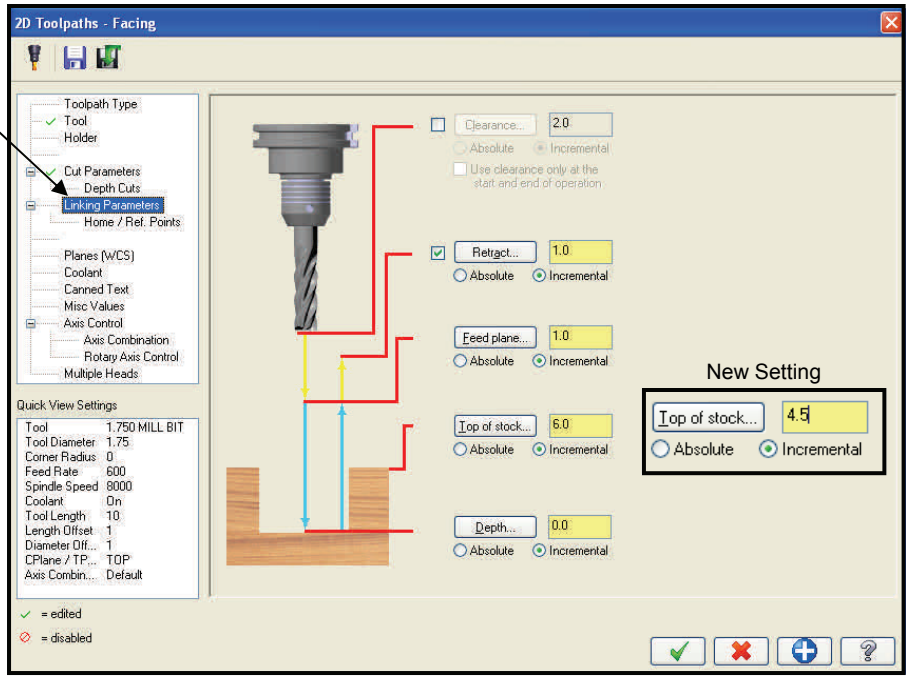
17. Click Depth Cuts.

18. Verify "Depth cuts" is checked and "Max rough step" is set to 0.2.



19. Click Linking Parameters

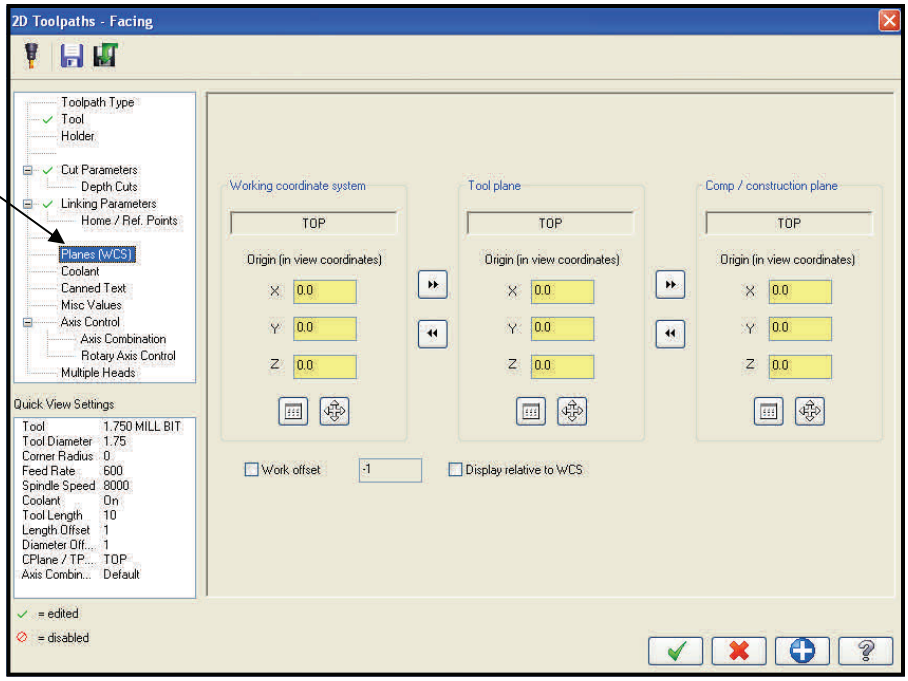
20. Set "Top of stock" to height of slab. In this example it is four point five (4.5). Leave "Depth" at zero on this screen. The value entered on the Cut Parameters screen is used to determine the depth. "Retract" and Feel plane" should be set to one (1) and Incremental selected for all.



Facing and Pocket Example

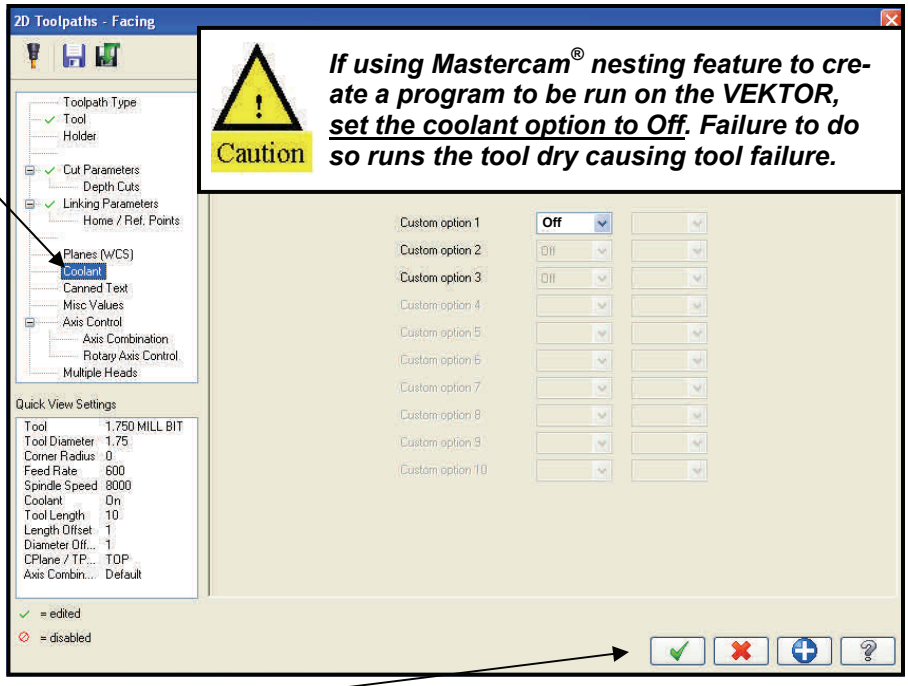
Task: Creating the Facing Toolpath (cont'd)

21. To display this screen click "Planes".



22. Verify "TOP" appears in all three planes.

23. To display this screen click "Coolant".



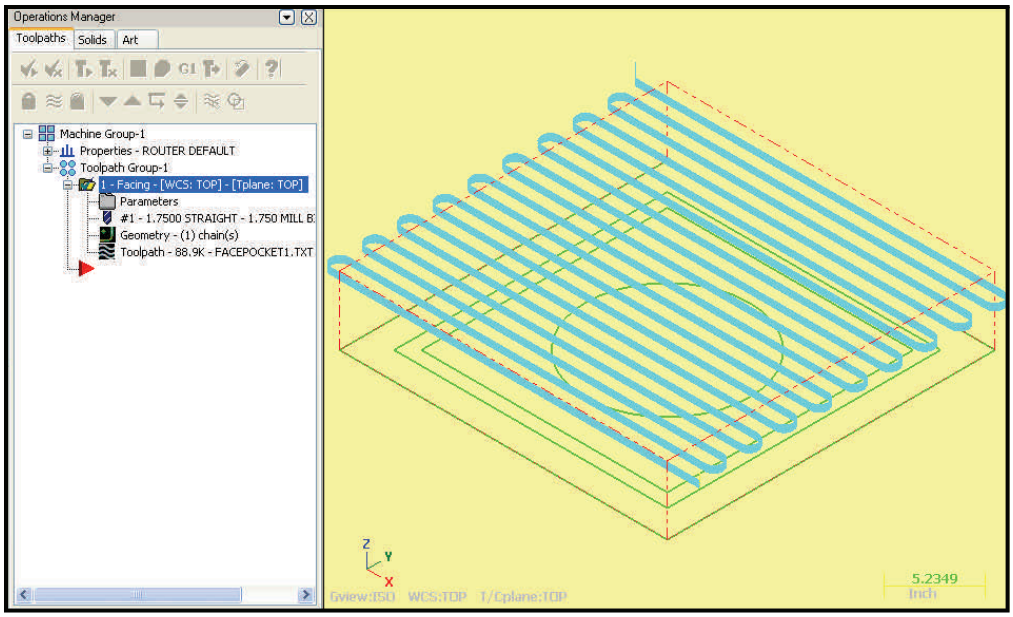
24. Verify the option is set to the **Off** state.

25. Click

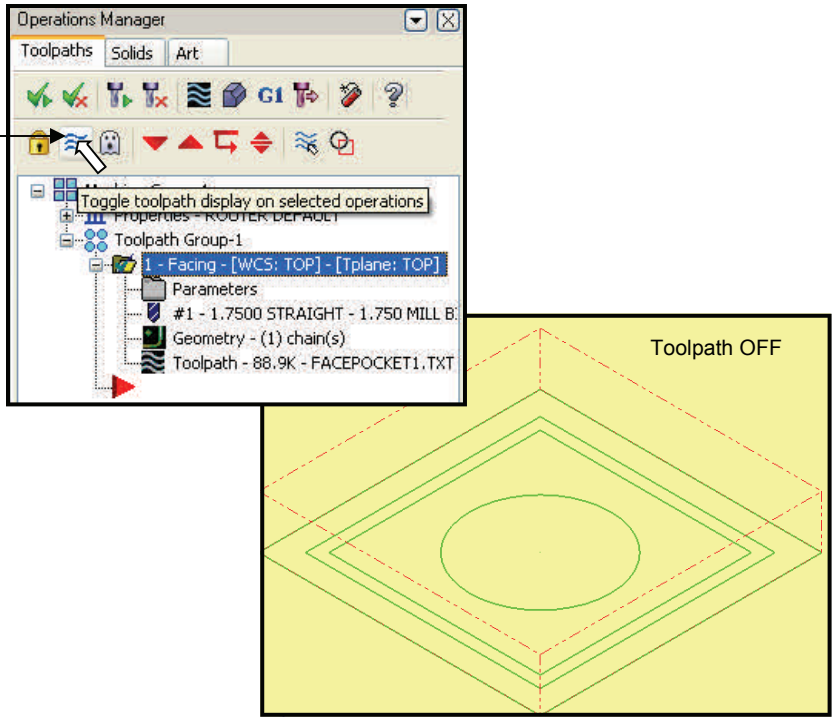
Facing and Pocket Example

Task: Creating the Facing Toolpath (cont'd)

26. The drawing area now displays the tool path for the facing operation. At this point the stock material has a height of four (4) inches. The next operation is to cut a pocket one (1) inch from the sides and one (1) inch deep.



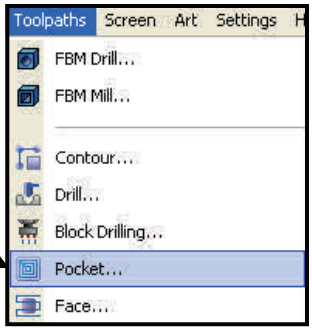
27. To remove the toolpath display, click the "Toolpath Toggle" button.



Facing and Pocket Example

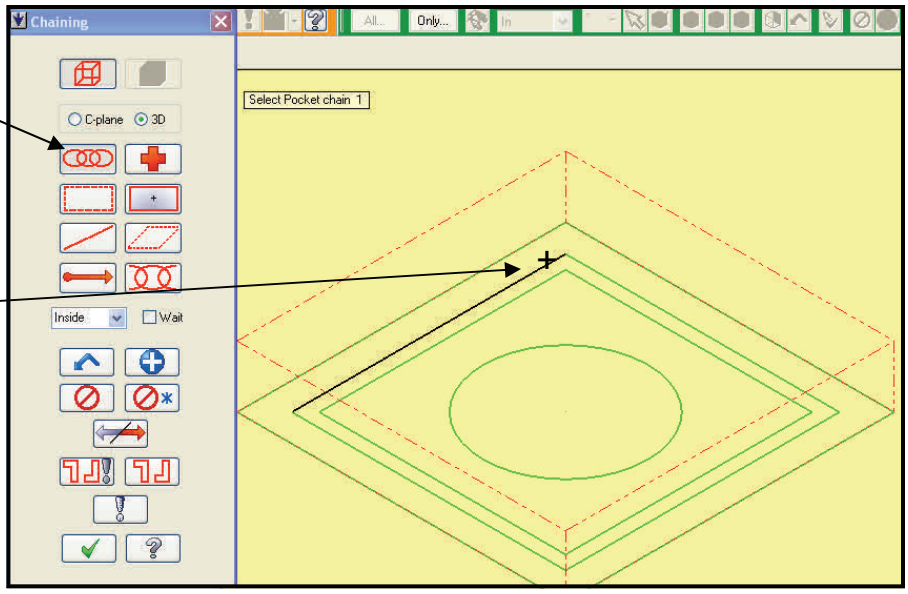
Task: Creating the First Pocket

1. The first pocket is one (1) inch from the sides and one (1) inch deep. To start the operation, select “Pocket” from the “Toolpahts” pull down menu.

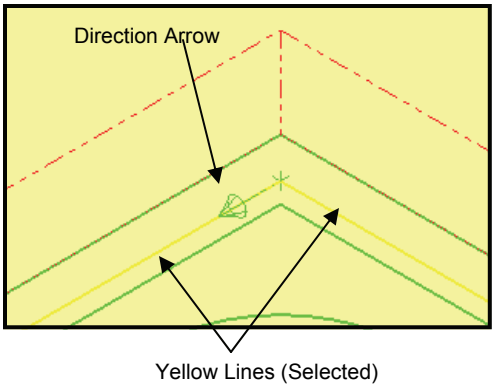


2. Click “Chains”.

3. Click where the chain should start.



4. Verify the arrow indicating direction appears and all lines are yellow.



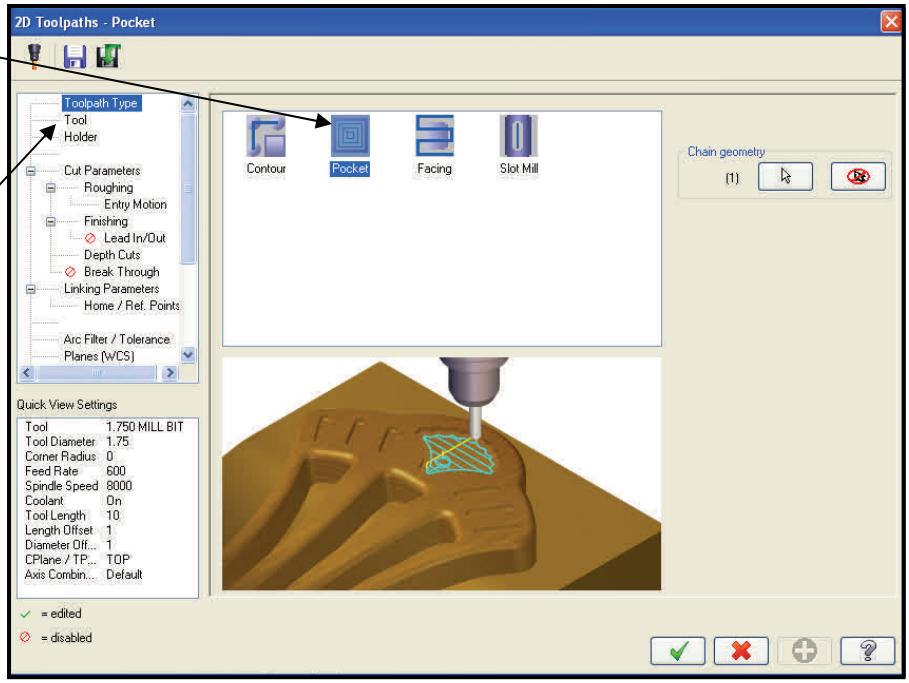
5. Click

Facing and Pocket Example

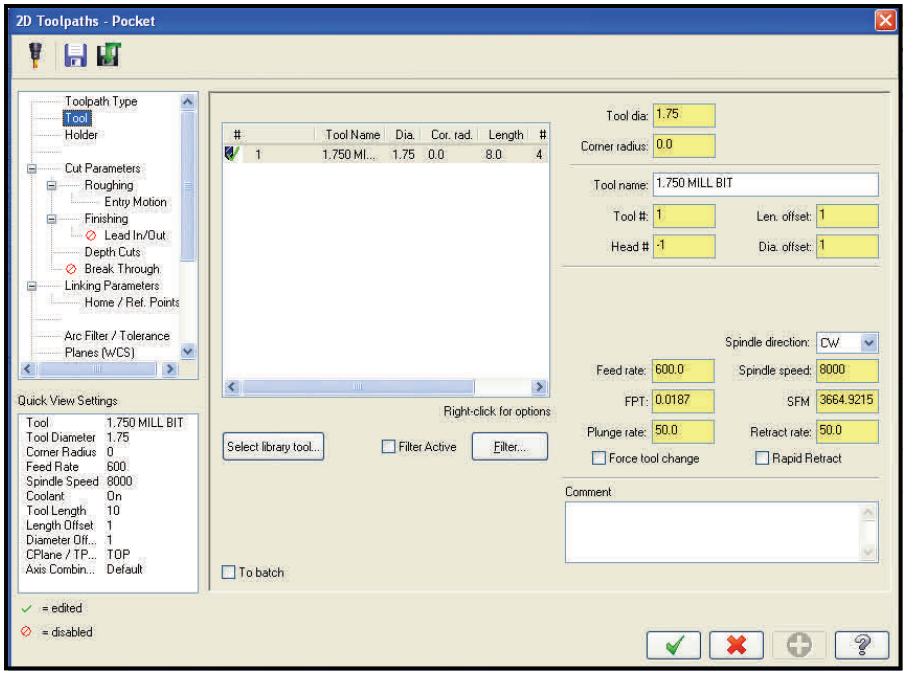
Task: Creating the First Pocket (cont'd)

6. Verify "Pocket" is selected.

7. Click "Tool".



8 The tool screen displays the tool used for the Facing operation. For this example the same tool is used for the Pocket operation. Verify all setting are correct.



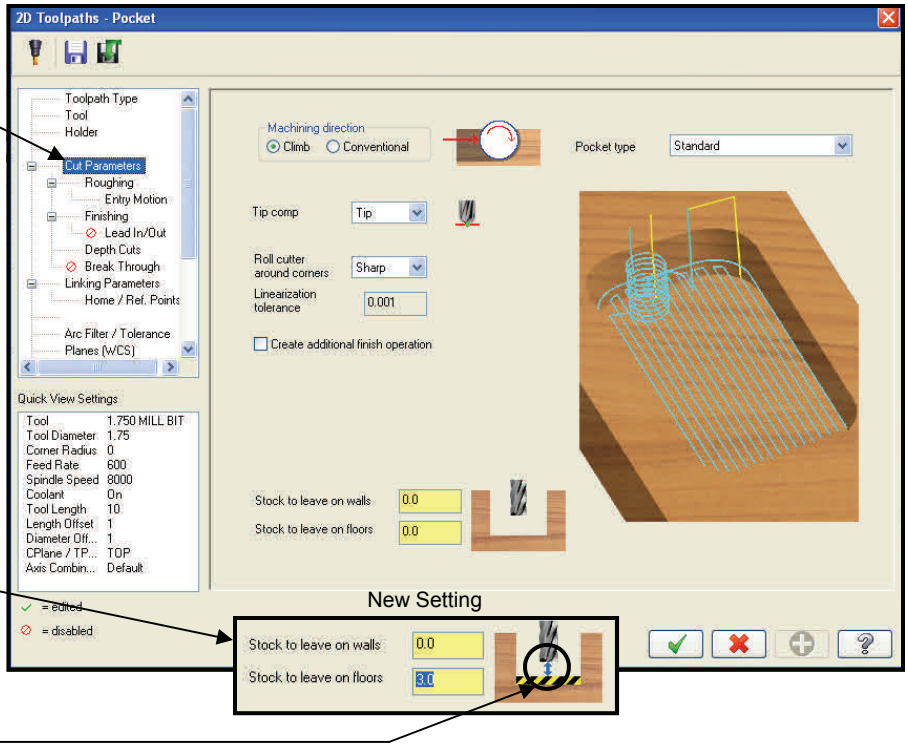
Facing and Pocket Example

Task: Creating the First Pocket (cont'd)

9. To display this screen click.

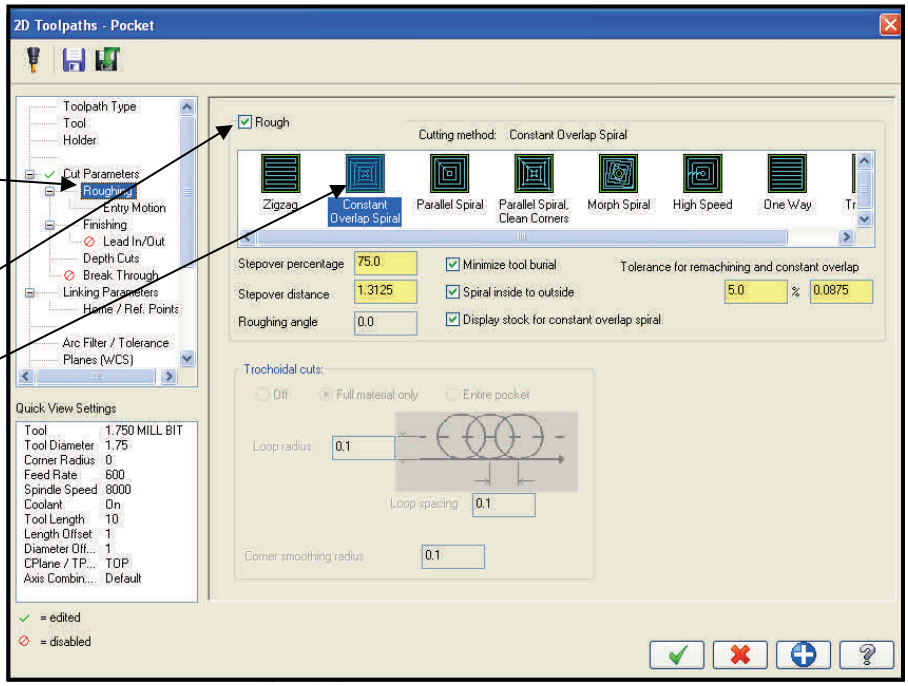
10. The default settings are OK for this example. Only the "Stock to leave on floors" must be entered.

11. Key-in three (3) for the new "Stock to leave on floors value". Press the "Enter" key to ensure the value is saved.



12. To display this screen click.

13. Verify "Rough" is checked. For this example "Constant Overlap Spiral" was selected as the cutting method and none of the default settings were changed.

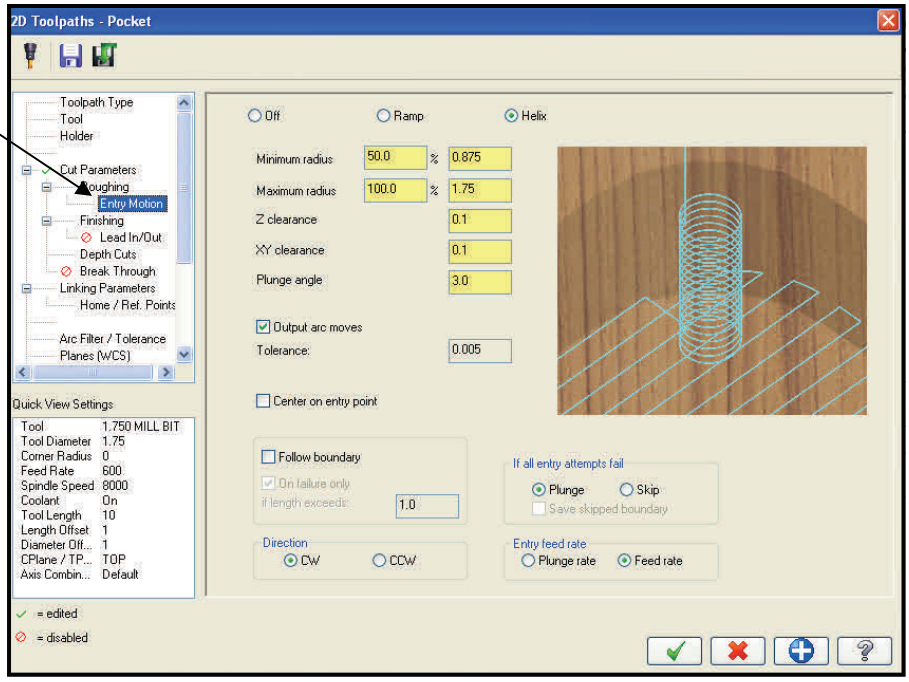


Facing and Pocket Example

Task: Creating the First Pocket (cont'd)

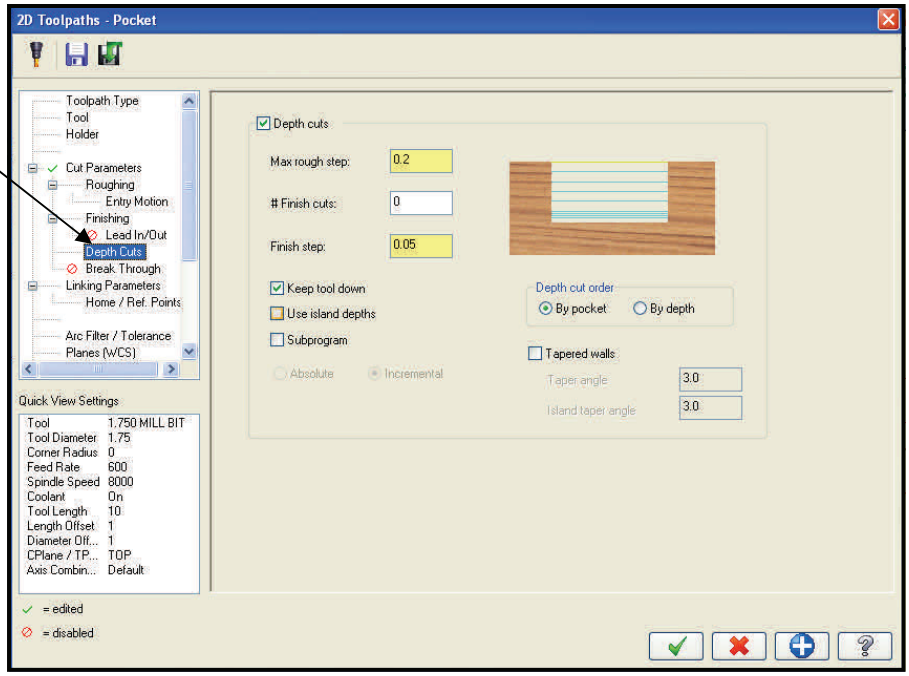
14. To display this screen click.

15. For this example the default settings shown are OK.



16. To display this screen click.

17. Verify the "Depth cut" is checked and "Max rough steps" is set to 0.2.

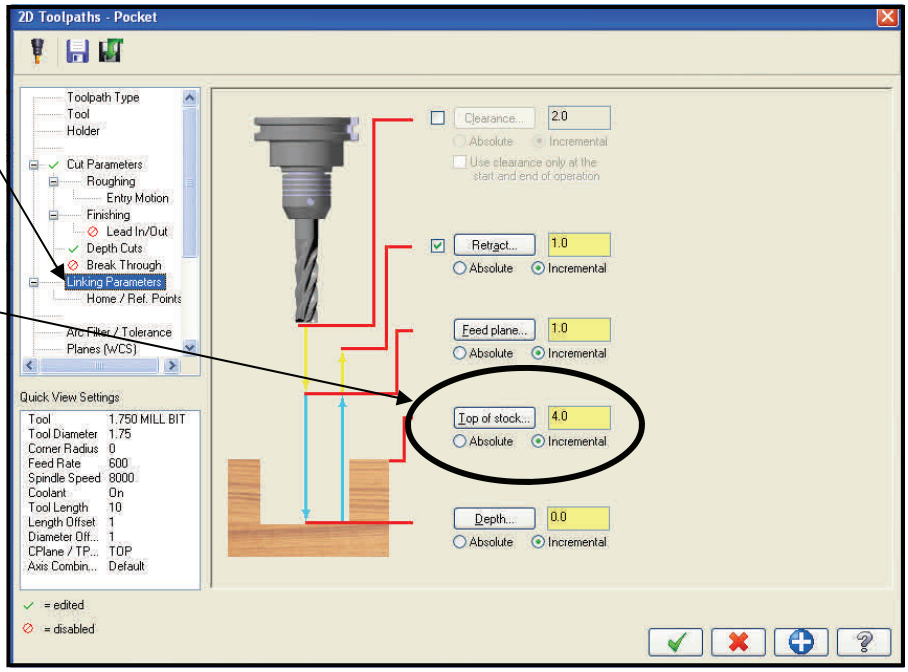


Facing and Pocket Example

Task: Creating the First Pocket (cont'd)

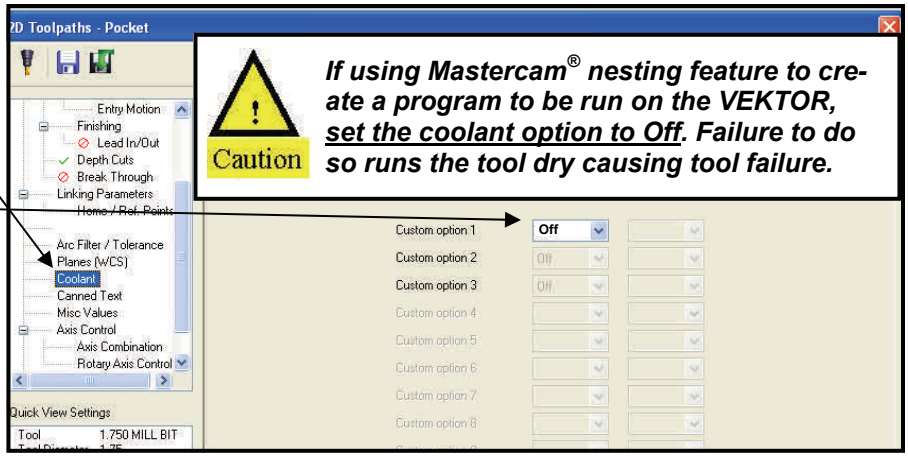
18. To display this screen click.

19. The default settings are OK for this example. Only the "Top of stock" has changed from the facing operation. The new value is four (4).



20. To display this screen click.

21. Verify "**Off**" appears.

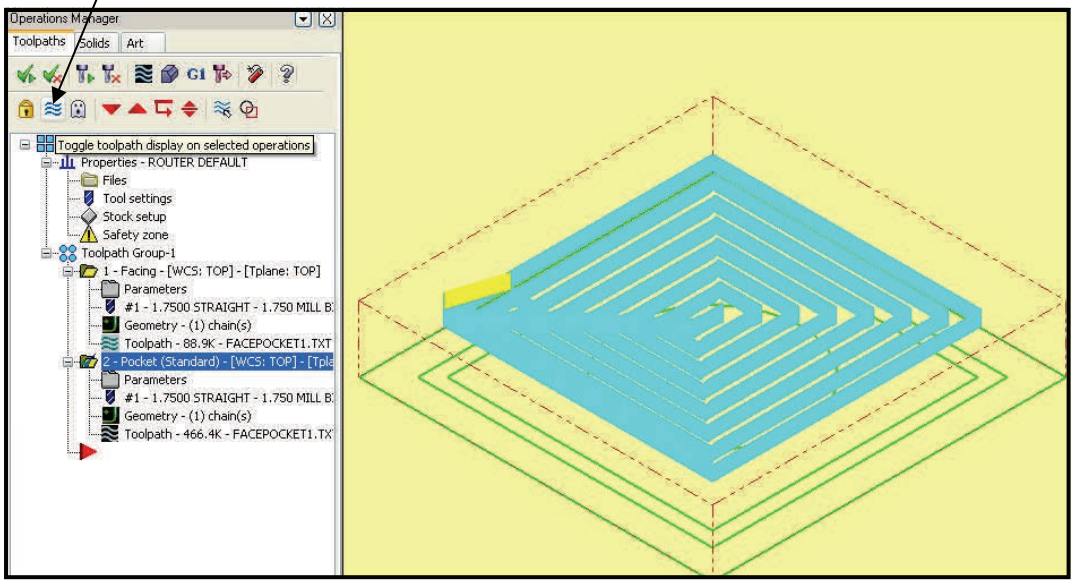


22. Click

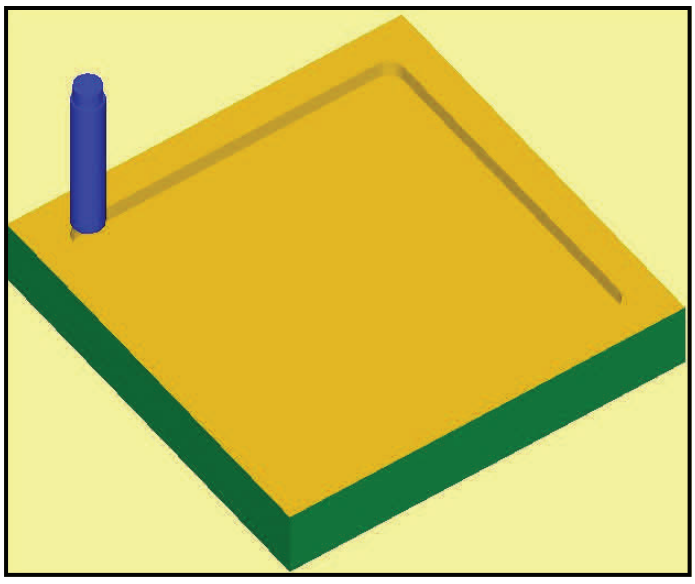
Facing and Pocket Example

Task: Creating the First Pocket (cont'd)

23. The drawing area now displays the tool path for the first pocket operation. This is one (1) inch deep and one (1) inch from the sides. To remove the toolpath display, click the "Toolpath Toggle" button.



This figure illustrates how the part would after the toolpath was executed.

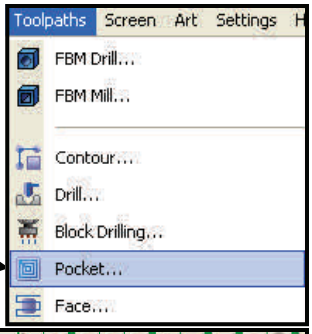


Facing and Pocket Example

Task: Creating the Second Pocket

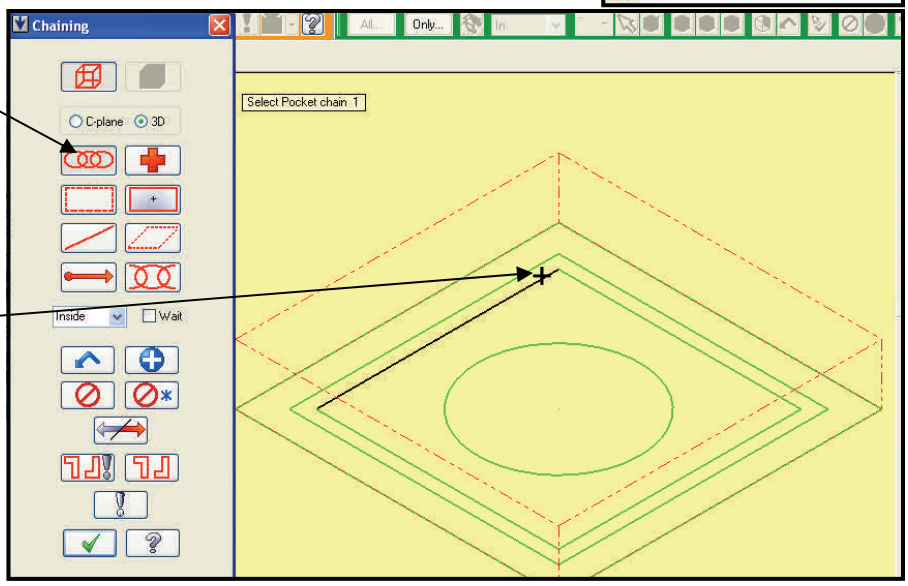
The second pocket is one (1) deep and removes the material between the outside edge of the circle edged created by pocket one. Since the last toolpath was a pocket the majority of setup screen require no changes.

1. To start the operation, select “Pocket” from the “Toolpahts” pull down menu.

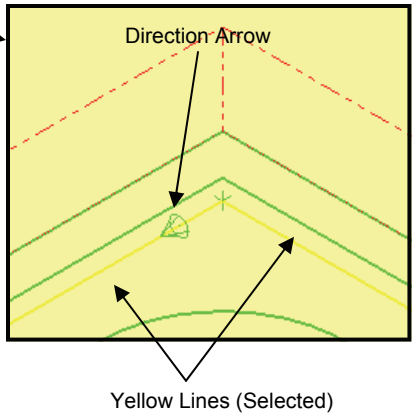


2. Click “Chains”.

3. Click where the chain should start.



4. Verify the arrow indicating direction appears and all lines are yellow.

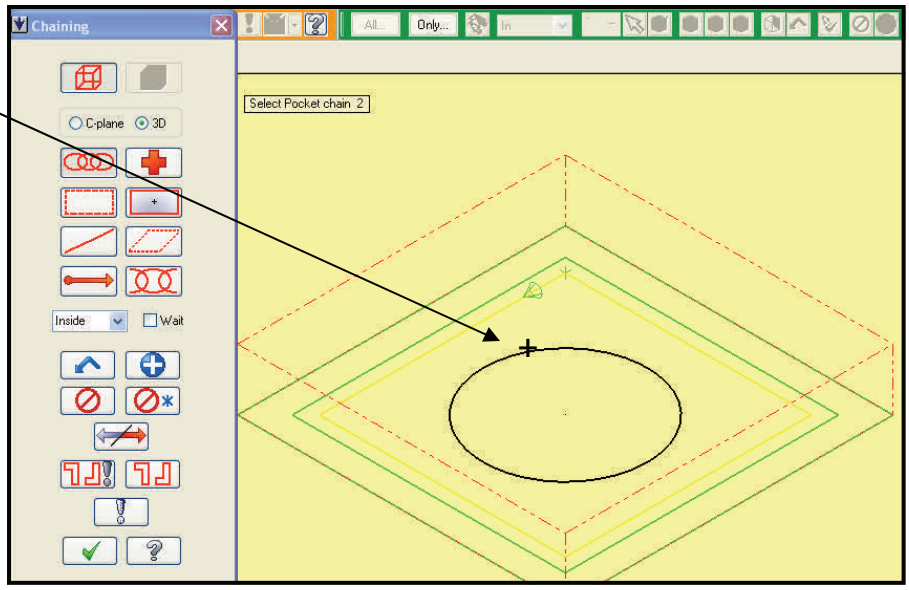


5. Click

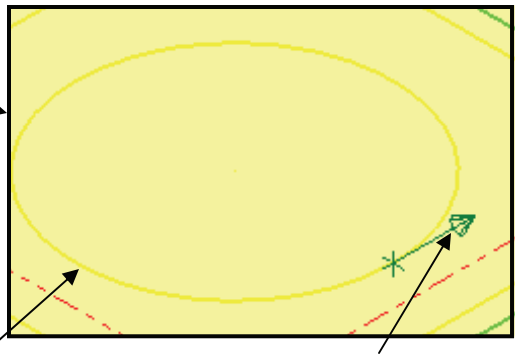
Facing and Pocket Example

Task: Creating the Second Pocket (cont'd

6. Select chain two by clicking where the chain should start.

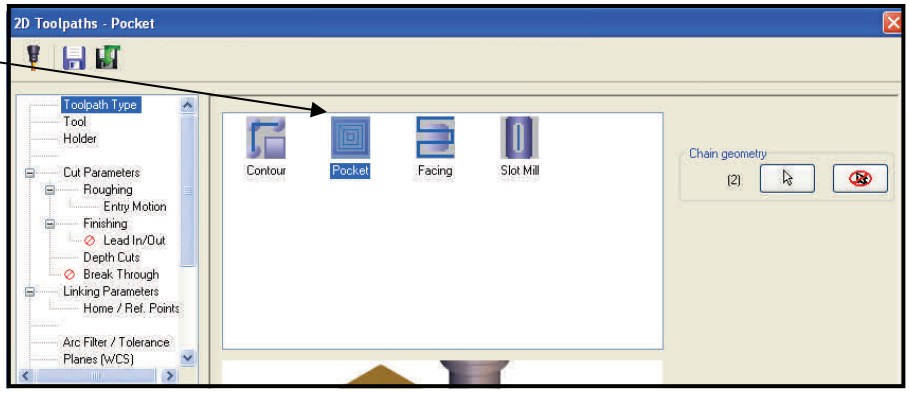


7. Verify the arrow indicating direction appears and all lines are yellow.



8. Click

9. Verify "Pocket" is selected.



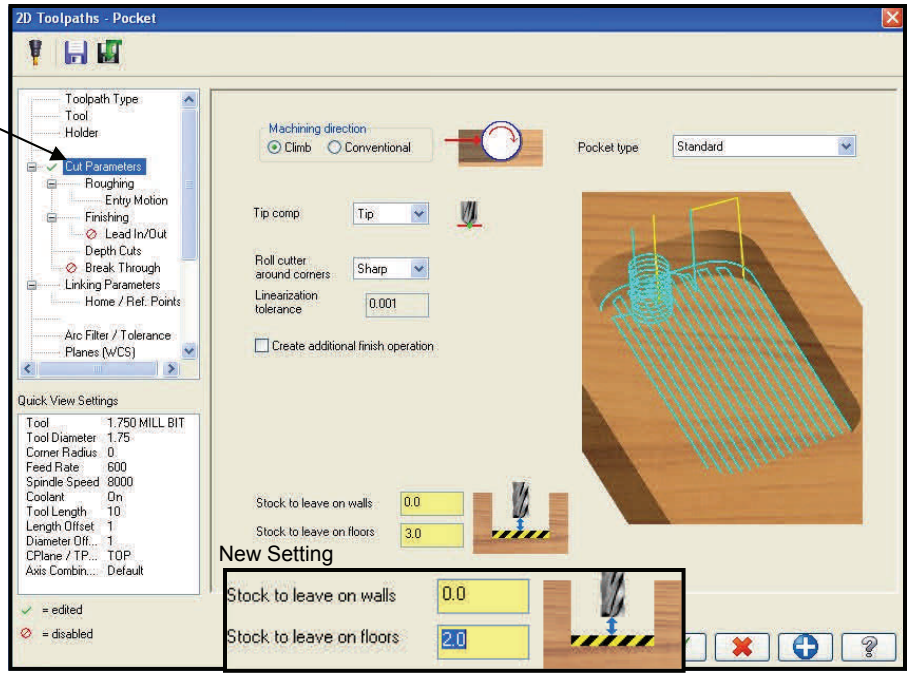
Facing and Pocket Example

Task: Creating the Second Pocket (cont'd)

Since the last toolpath was a pocket, only the Cutting Parameter and Linking Parameters screens have parameters that must be changed.

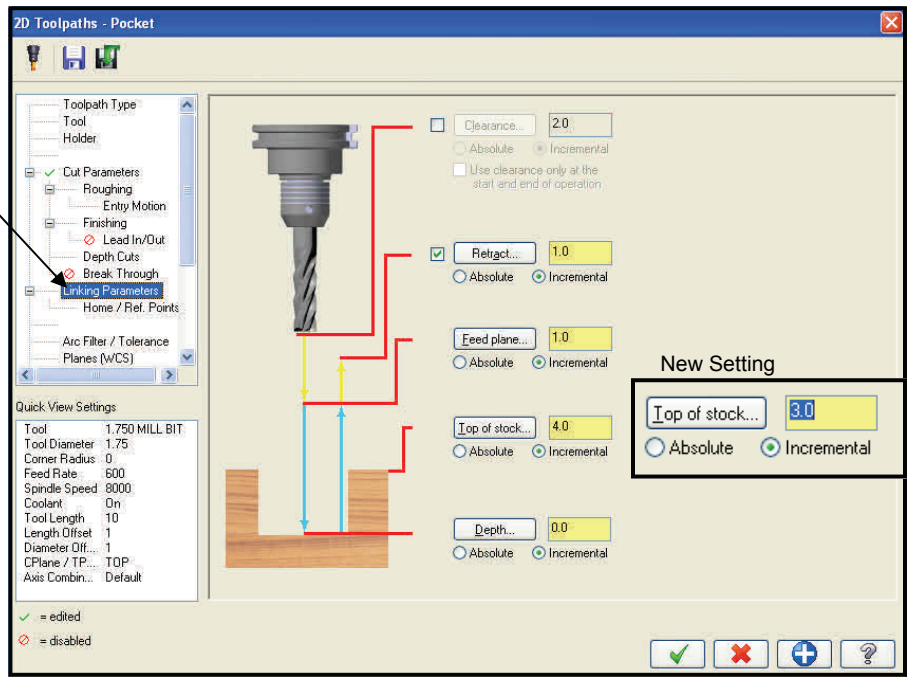
10. To display this screen click.

11. On this screen only the "Stock to leave on floors" parameter has to be changed. For this example, one (1) more inch needs to be re-move. This leaves two (2) inch of stock. Remember to press "Enter" key after changing the value.



12. To display this screen click

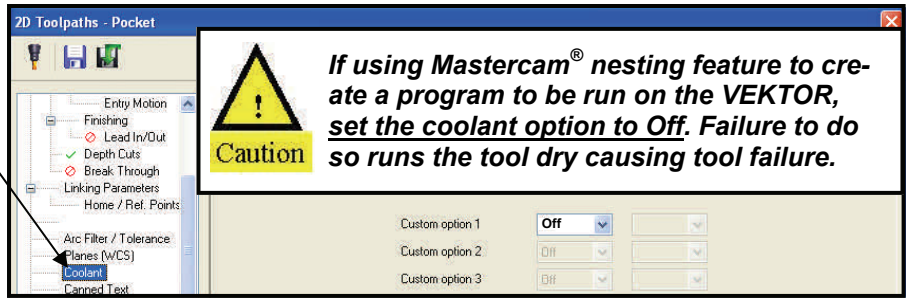
13. Since the last pocket removed one (1) inch of material, there are only three (3) inches left. Thus the new "Top of stock" parameter is set to three (3).



Facing and Pocket Example

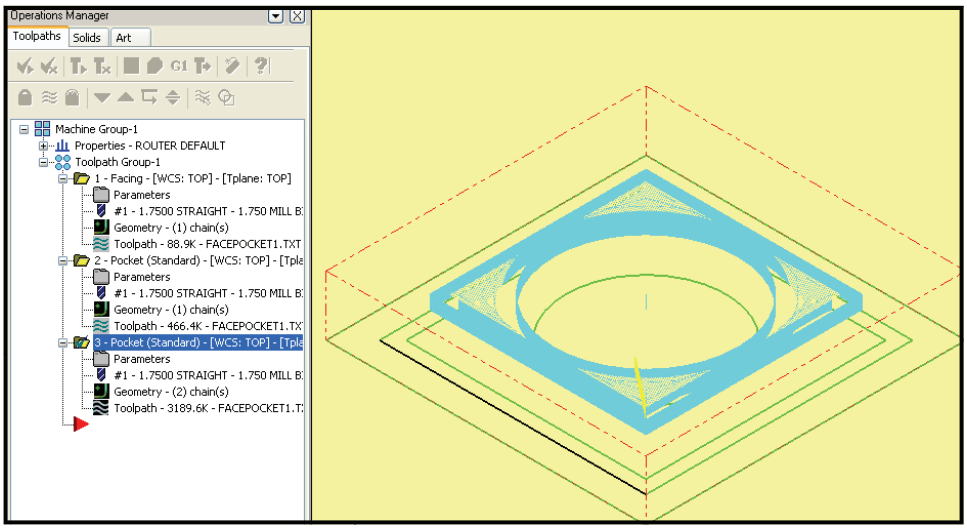
Task: Creating the Second Pocket (cont'd)

14. To display this screen click.
15. It is good idea to always check if the coolant parameter is set to "**Off**".

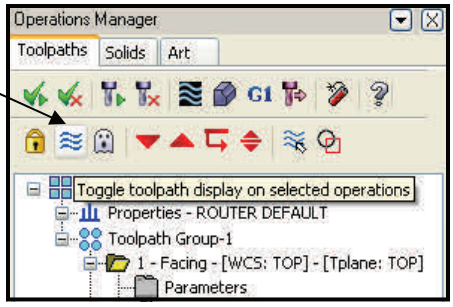


16. Click

The drawing area now displays the tool path for the second pocket operation. This is one (1) inch deep and removes the material between the outside edge of the circle and the edge created by pocket one.



17. Remove the toolpath display, by clicking the "Toolpath Toggle" button.

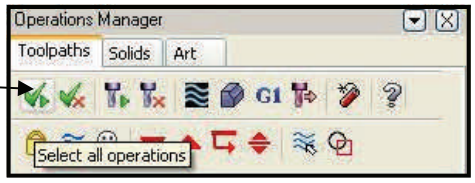


Facing and Pocket Example

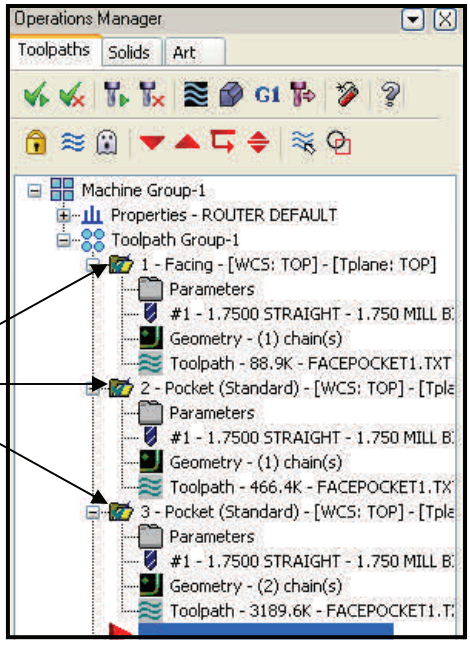
Task: Verify Command

Verify is the process of running the toolpath as an actual machine would to make sure there are no tool crashes and that the profile cuts are in the correct position. The verification uses the stock material and runs only the selected operations.

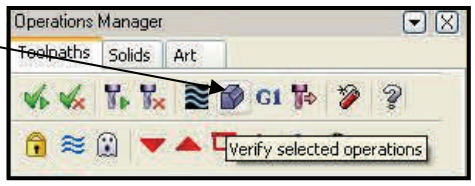
1. Click “Select all operations” button.



2.. Verify that all operations have been selected. Selected operations have check mark on top of the operation folder.



3. Click “Verify selected operations” button.

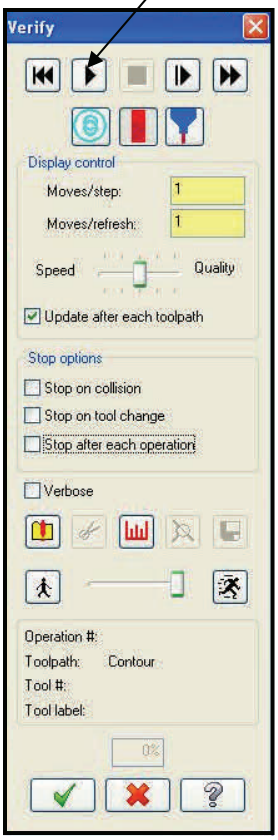


Facing and Pocket Example

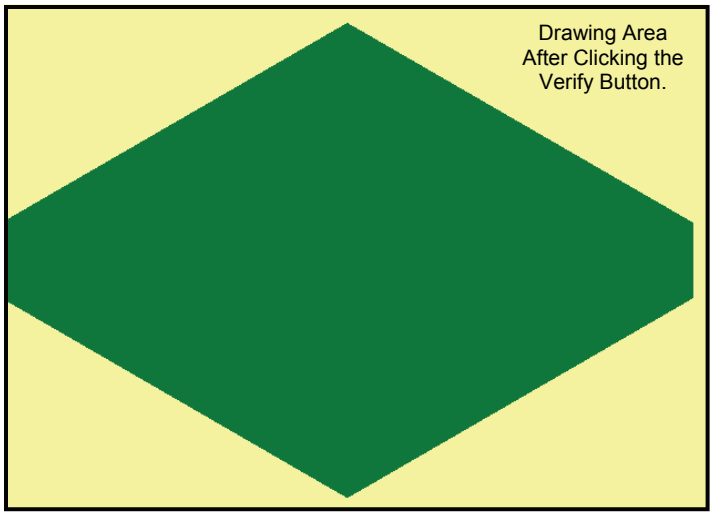
Task: Verify Command

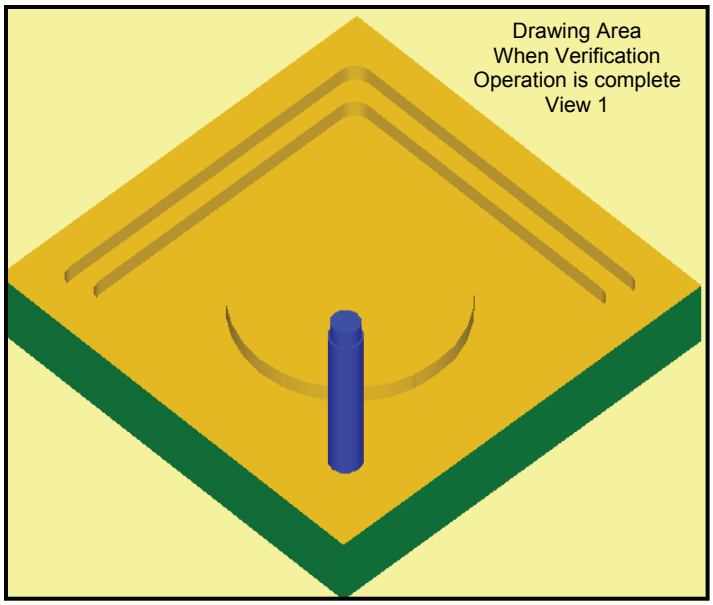
At this time, the red wireframe is now a green solid rectangle and the Verify window is overlaid on the screen.

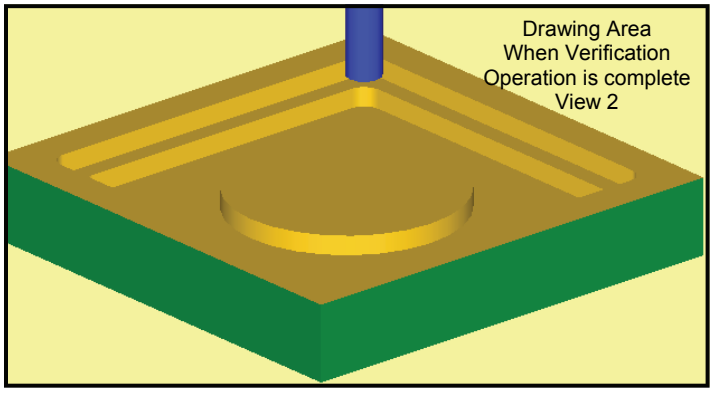
To start the verify process, click the “Machine/Play” button.



When complete click the green check mark button to close the Verify window and return to the normal drawing area.



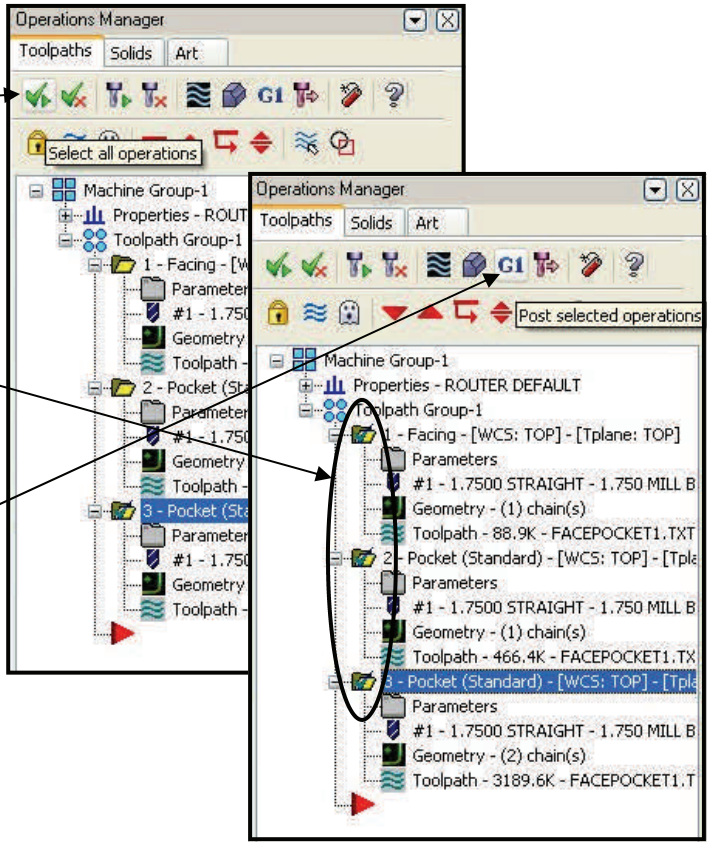




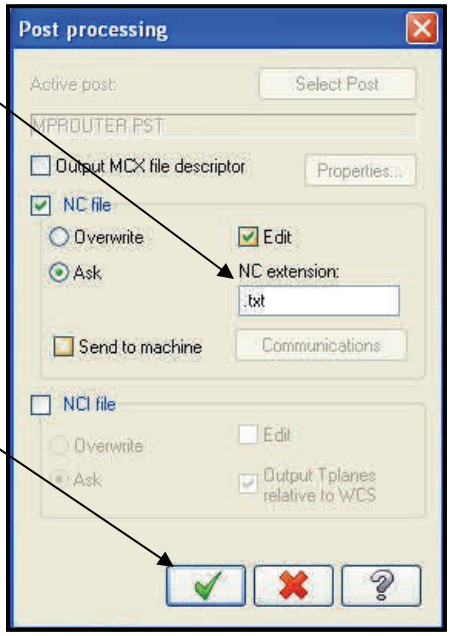
Posting

After the tool path has been created and verified, the next step is to create the G-Code that makes the VEKTOR run. This process is call Posting.

1. Click “Select all operations” icon.
2. Verify the folders for all operations have check marks over them.
3. Click “Post selected operations” icon (G1).



4. Verify the “NC extension” field is set to: .txt
5. Click the green check mark.



Posting (cont'd)

At this point, the standard Windows "Save As" dialog box is displayed (see below). Before the green check mark is clicked the following fields must be check for accuracy.

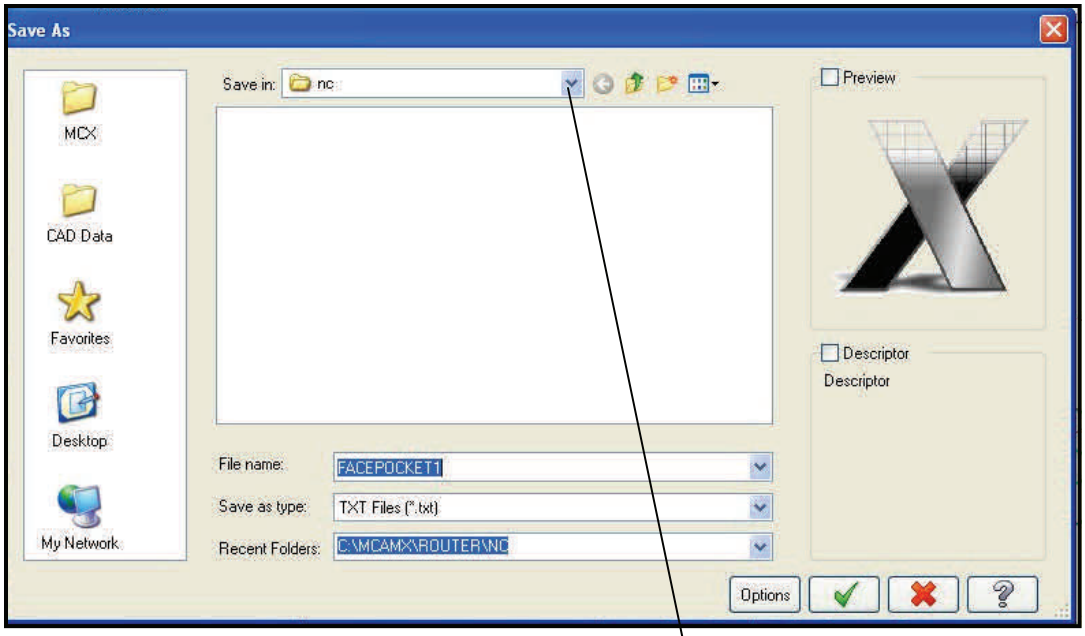
- "Save in" - detestation/location where this file is to be stored
- "File name" - name given to this file, unique and reflective of job/customer
- "Save as type" - format used to create this file, this must be TXT Files (*.txt)

The "Save in" and "Save as type" fields are loaded from default settings entered via the Control Definition screen and should NOT have to changed on this screen. If these fields are consistently incorrect, use the Change Default NC Extension and Save In procedure to edit the default values.

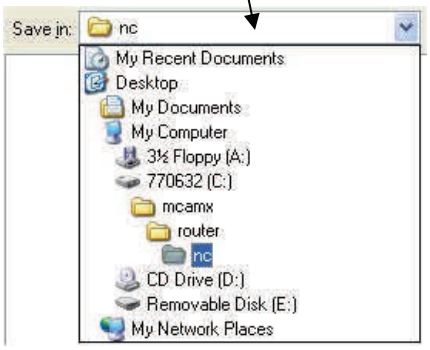
6. Verify "Save in" and "Save as type" fields are set to correct values.

7. Verify the "File name" is correct. .

8. Click the green check mark.



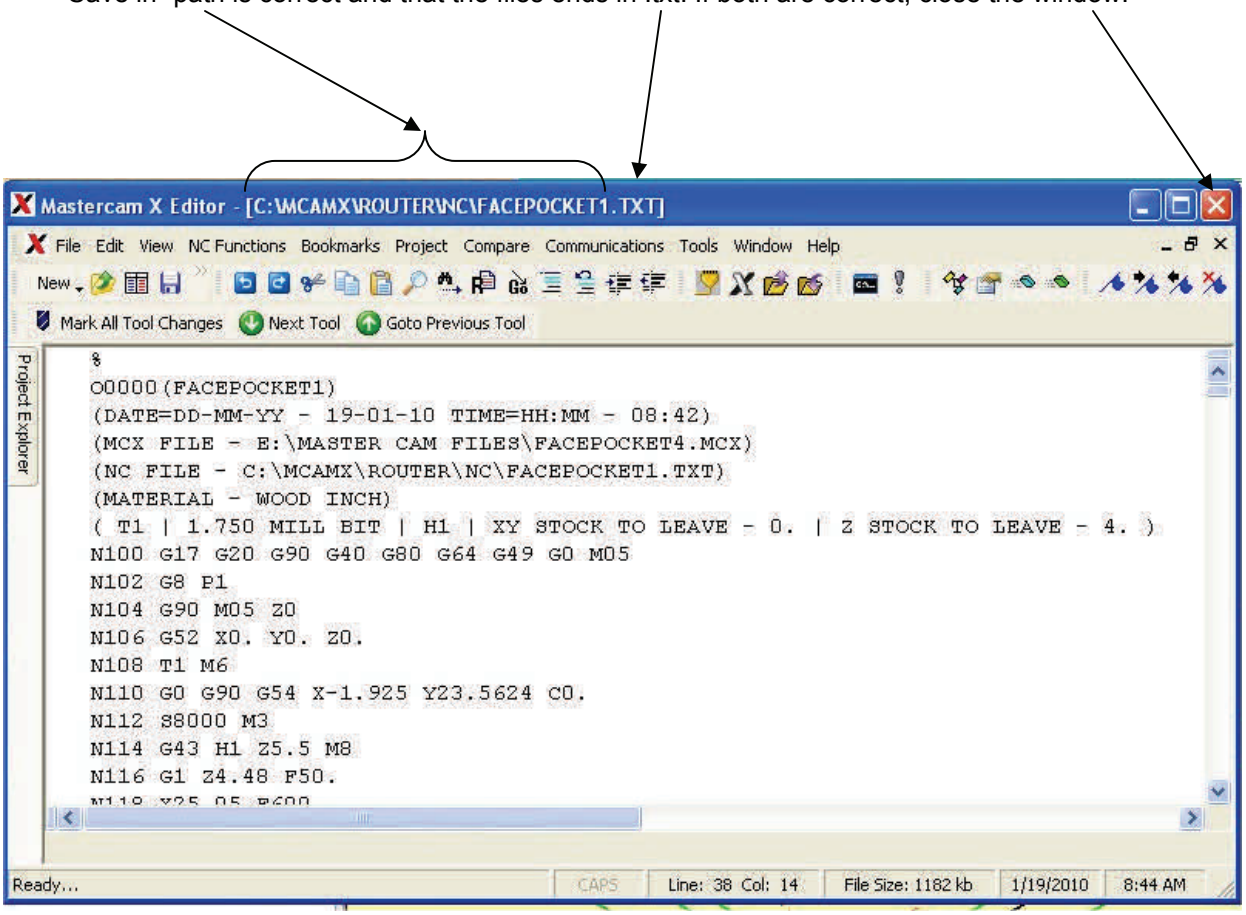
The "Save in" pull-down menu displays the file structure for the current computer. If the default values have been set, the pull-down menu is not needed.



Posting

The below progress message box appears as the file is being posted.

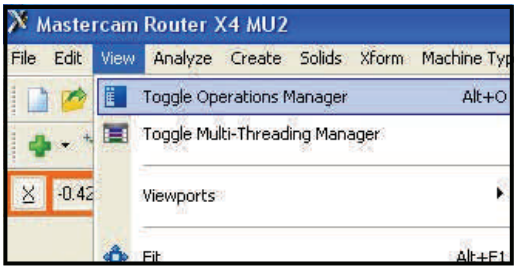
When the process is done, the below window opens and displays the G code. Verify the "Save in" path is correct and that the files ends in .txt. If both are correct, close the window.

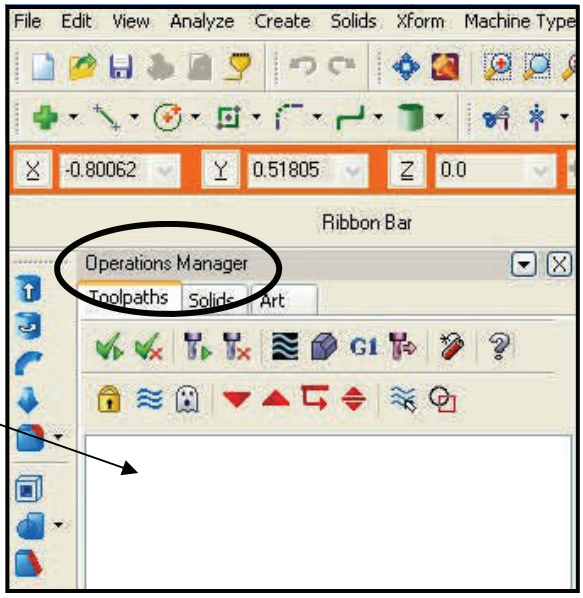


Nesting Example

Nesting is the process of placing more than one part on the stock material or slab and creating the G-code to profile the entire job.

1. If the “Operations Manager” window is not displayed, click “Toggle Operations Manager” found in the “View” pull down menu.



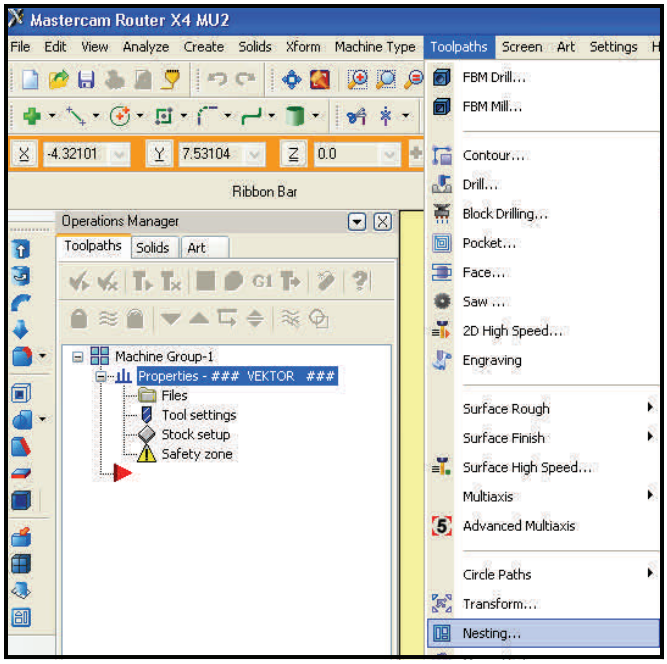


The “Operations Manager” shown here does NOT have a machine group or an operations displayed. If a group appears in this area and does NOT have the word “VEKTOR” in the title, exit Mastercam, launch/open the program again and if necessary repeat step 1.

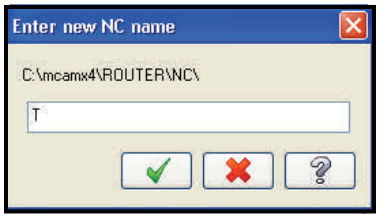
2. To load the machine group select “Machine Type” from the menu bar then “Router” and then “Default”.

Nesting Example (cont'd)

3. To start the nesting operation, select "Toolpaths" from the menu bar and then "Nesting".

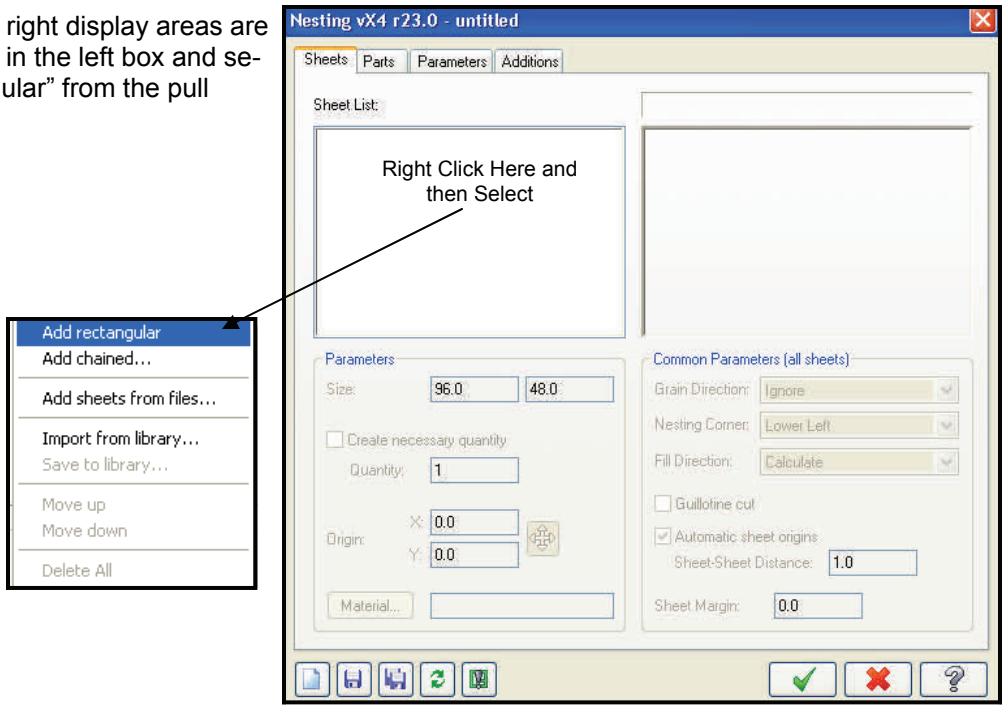


The below dialog box appears after "Nesting" is clicked.



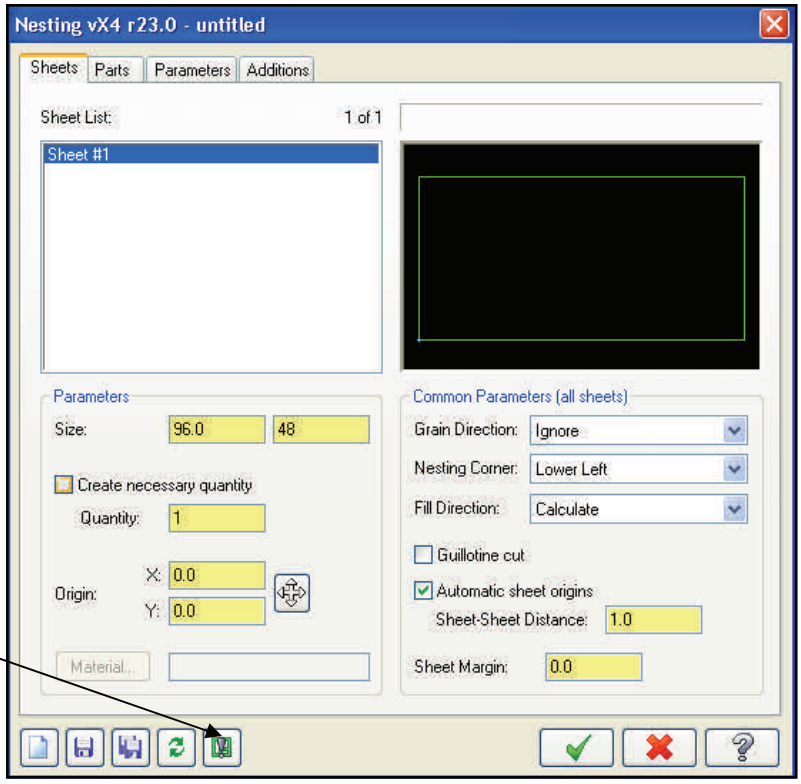
4. Click the green check mark to display the Nesting window.

If the top left and right display areas are empty, right click in the left box and select "Add rectangular" from the pull down menu.



Nesting Example (cont'd)

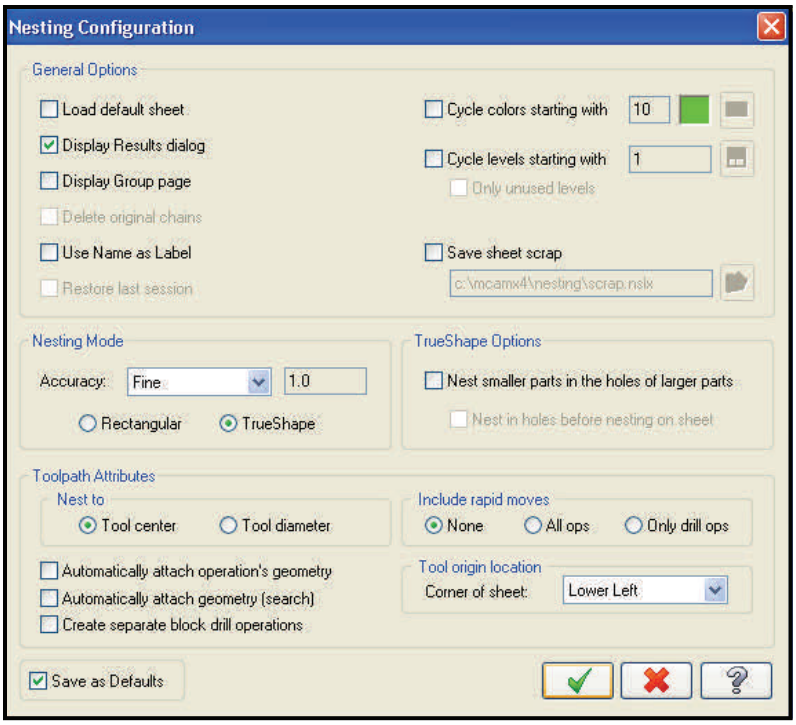
5. On the Sheets tab, set the following:
   - Size: for this example 96 by 48
   - Quantity: for this example 1
   - Fill Direction: Calculate
   - Check "Automatic sheet origins" option
   - Verify the other parameters are set as shown.



6. Before leaving this screen click on the "Config" icon to display the Nesting Configuration window.

NOTE:

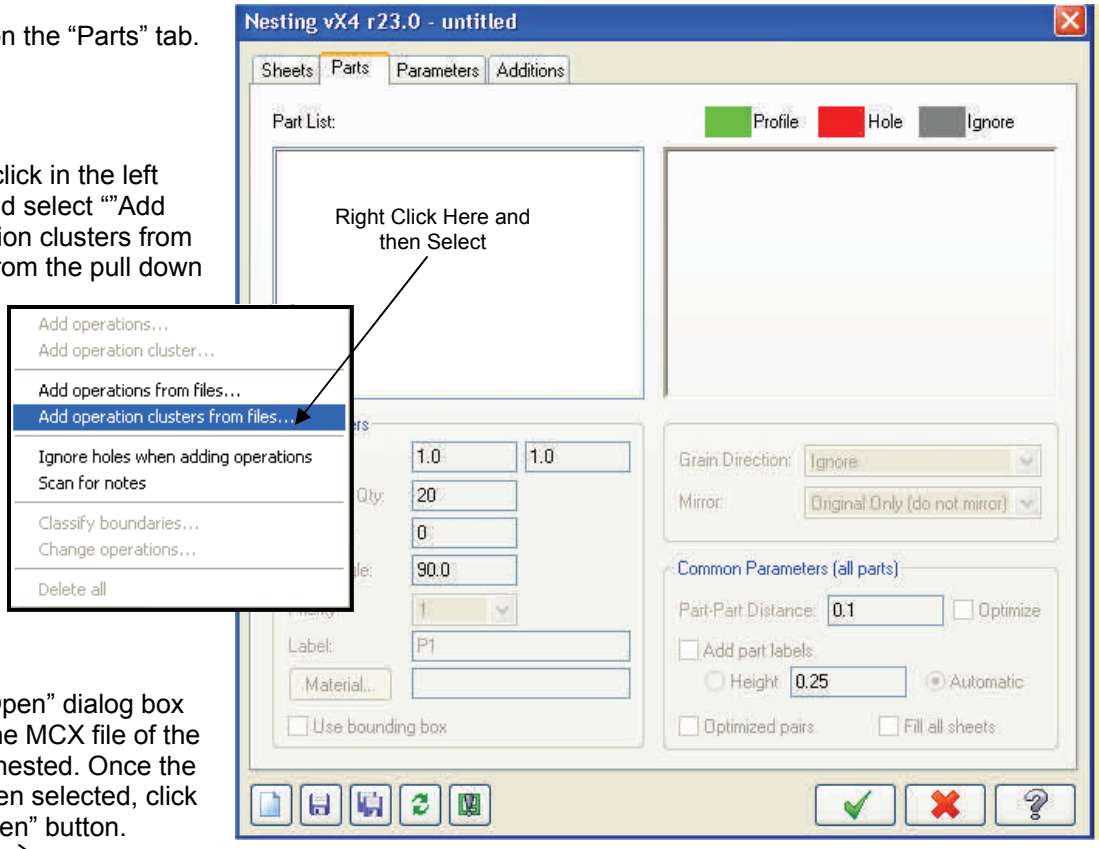
If "Tool center" and "Save as Defaults" boxes are NOT checked, check them. Check the green check mark on this screen and the Nesting screen and then exit Mastercam®. If asked to save values, answer "Yes". Restart Mastercam® and repeat the steps to this point.



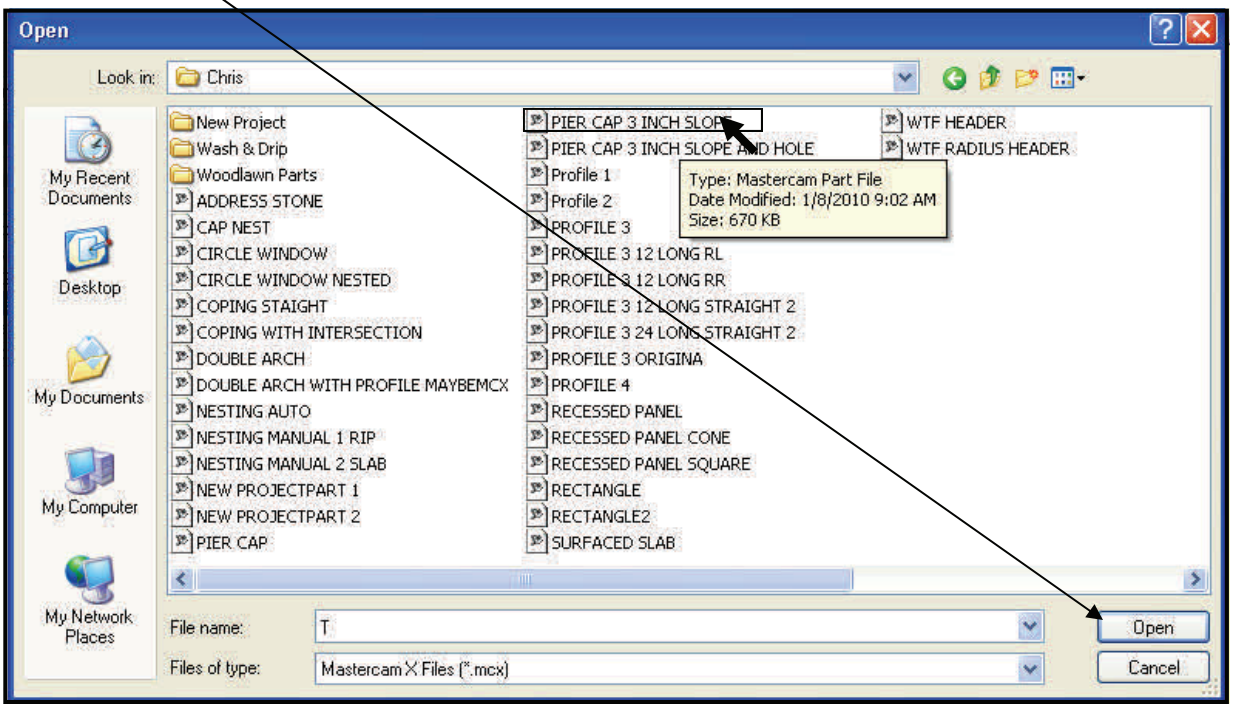
Nesting Example (cont'd)

7. Click on the "Parts" tab.

8. Right click in the left box and select ""Add operation clusters from files" from the pull down menu.



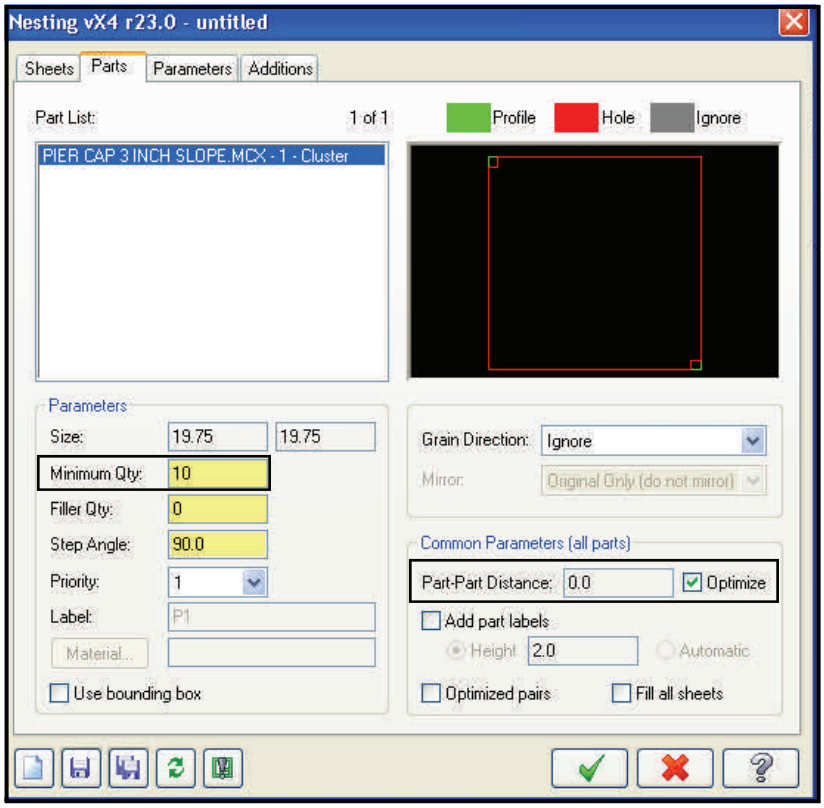
Use the "Open" dialog box to locate the MCX file of the part to be nested. Once the file has been selected, click on the "Open" button.



Nesting Example (cont'd)

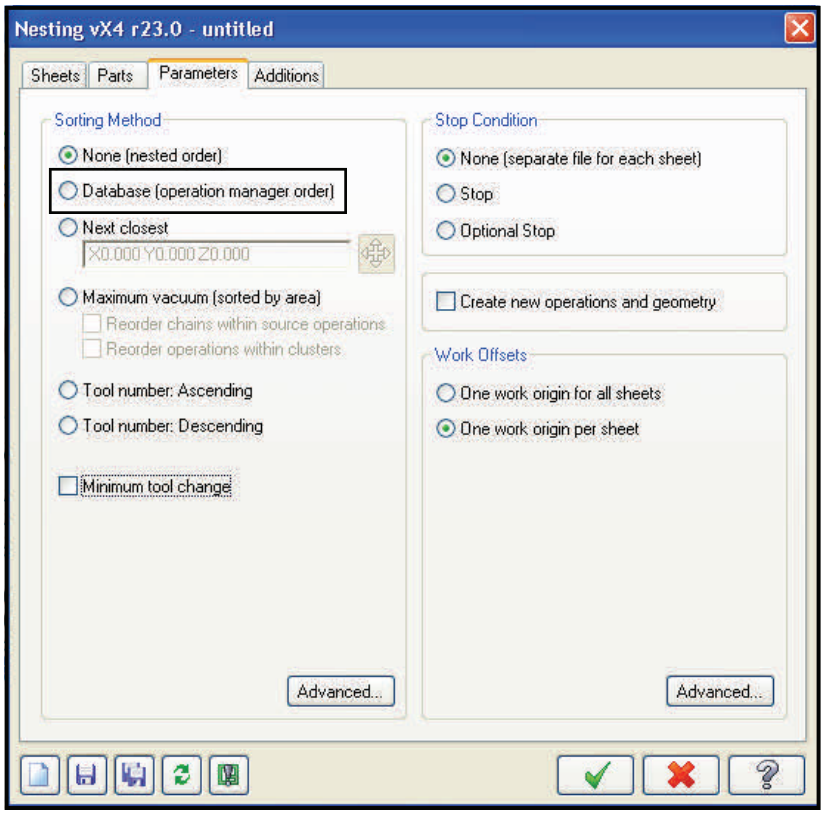
9. On the "Parts" tab, set the following:
   - Quantity: for this example 10
   - **Uncheck "Optimize"**
   - **Part-Part Distance "0.1"**
   - Verify the other parameters are set as shown.

10. Click "Parameters" tab.



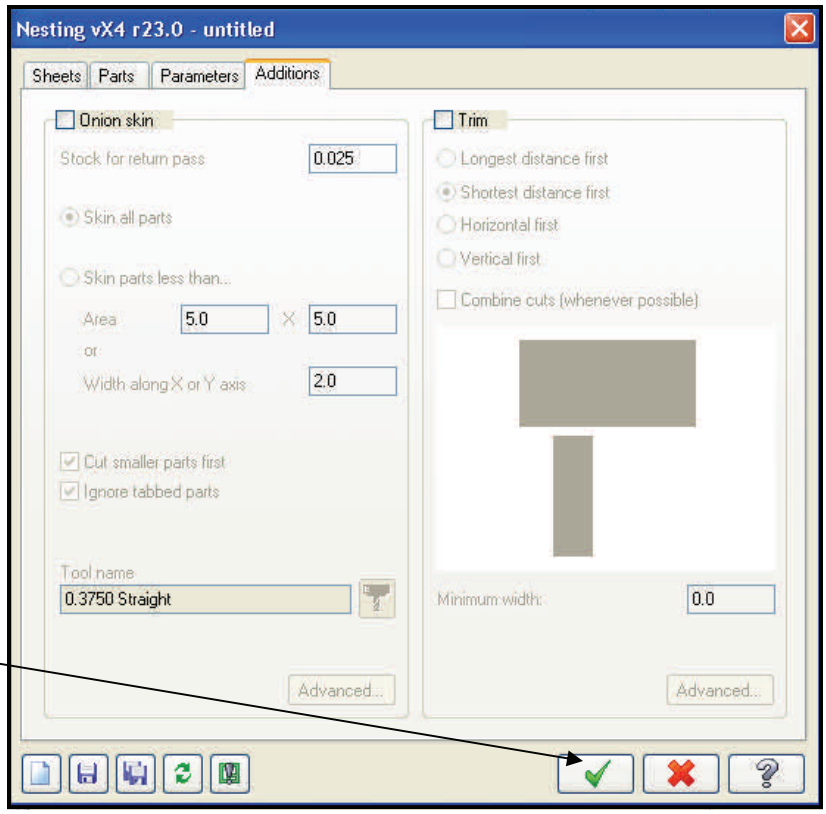
11. On the "Parameters" tab, set the following:
   - Sorting Method: "Database (operation manager order)"
   - Stop Condition: "None"
   - Work Offsets: "One work origin per sheet"

12. Click the "Additions" tab.



Nesting Example (cont'd)

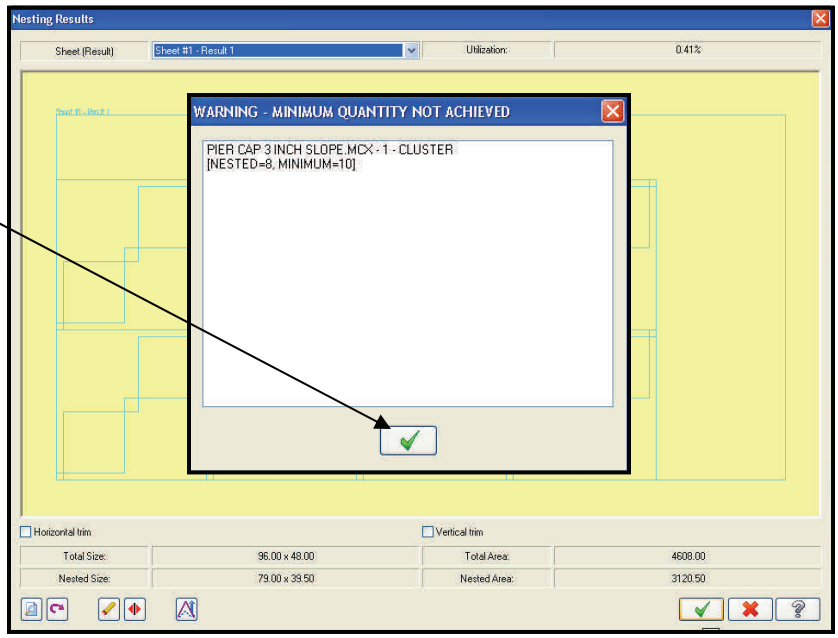
13. On the "Additions" tab, none of the features are selected.



14. Click the green check mark. This generates the "Nesting Results" window shown below.

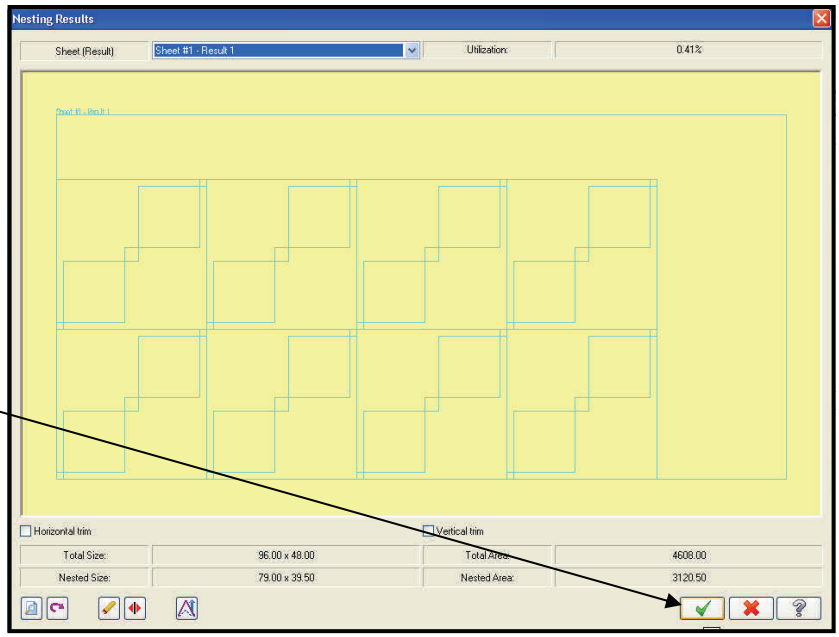
In this example a "Warning" dialog box appears over the results window. The message indicates that only eight of the ten parts were nested. For this example, that is OK.

15. Click the green check mark to continue.



Nesting Example (cont'd)

16. Verify the Nesting Results screen displays the desired results.

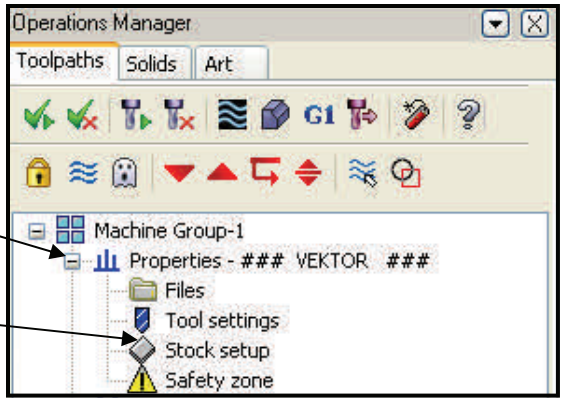


17. If everything appears to as expected click the green check mark to start the posting.

During the posting process the per cent complete information appears in the lower left corner of the screen. This process can take a fair amount of time to complete. Be patient.

18. Expand the Properties entry

19. Open the "Stock setup" folder.

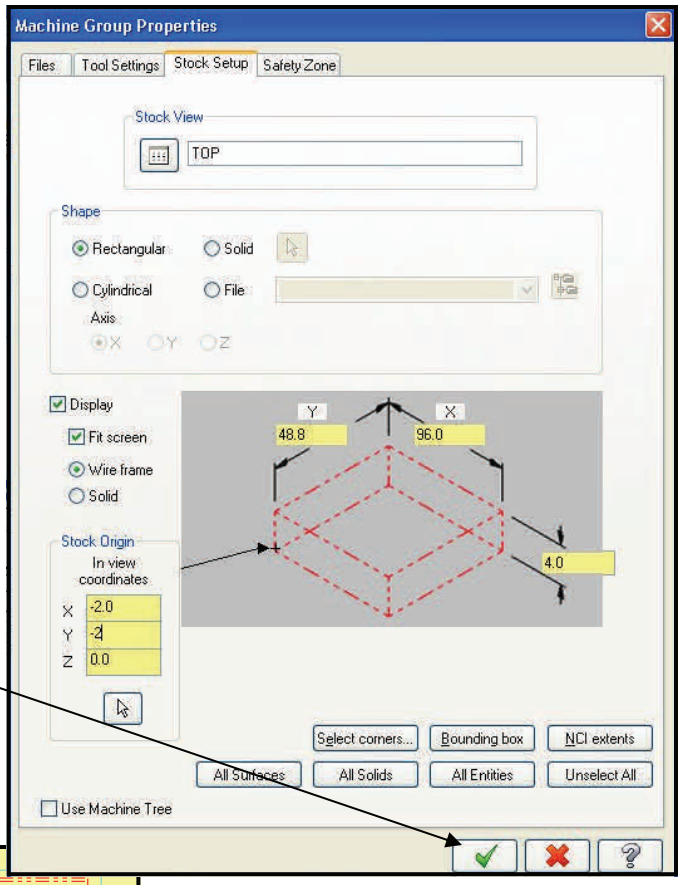


Nesting Example (cont'd)

20. Ensure "Display" is checked.

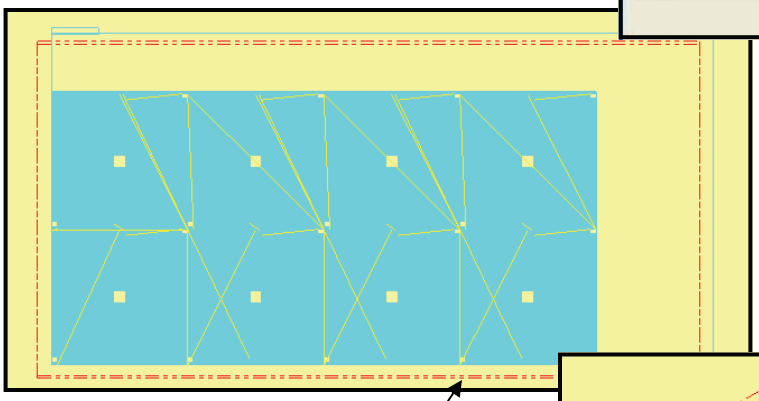
    Enter the following dimensions :
    Y = 48
    X = 96
    Z = 4

    Set "Stock origin" as follows:
    X = -2.0
    Y = -2.0
    Z = 0

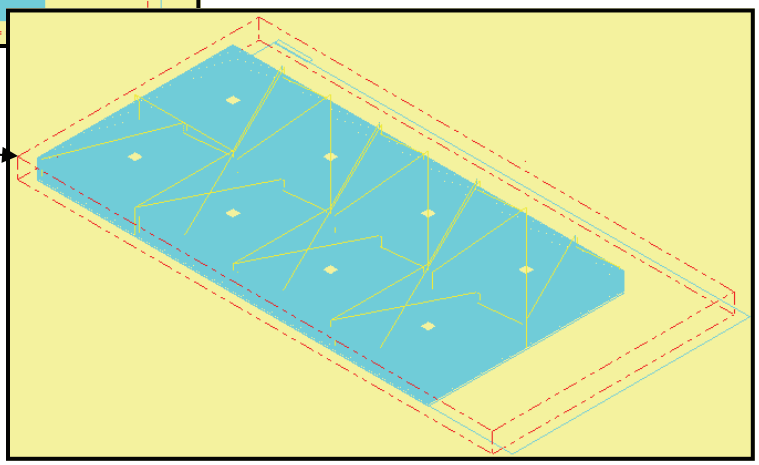


21. Click the green heck mark.

Top View



Isometric View

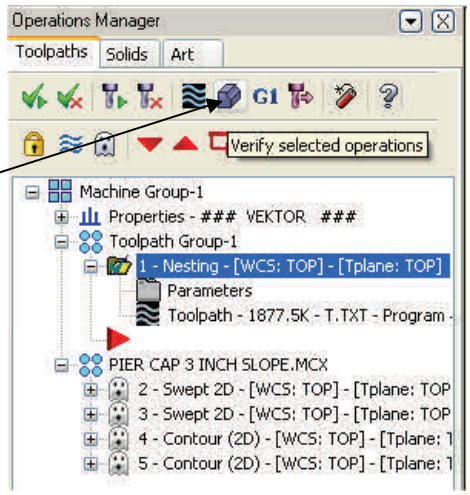


Stock
(red dot-dot-dash Line)

Nesting Example (cont'd)

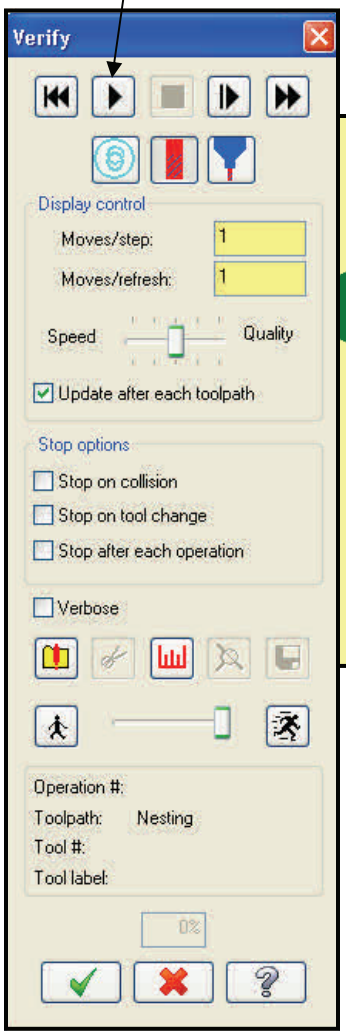
22. Verify that only the "Nesting" toolpath is selected (check mark over the folder).

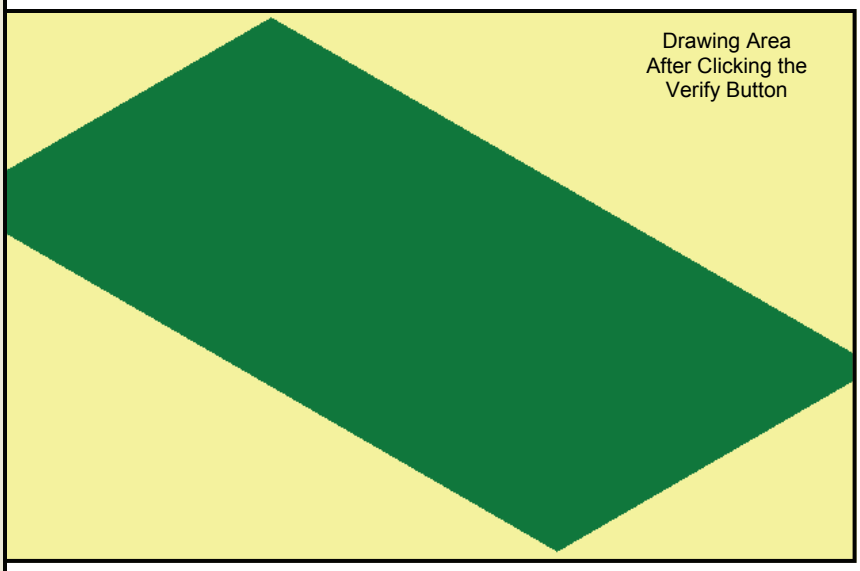
23. Click the "Verify" icon to start the verify all selected operations.



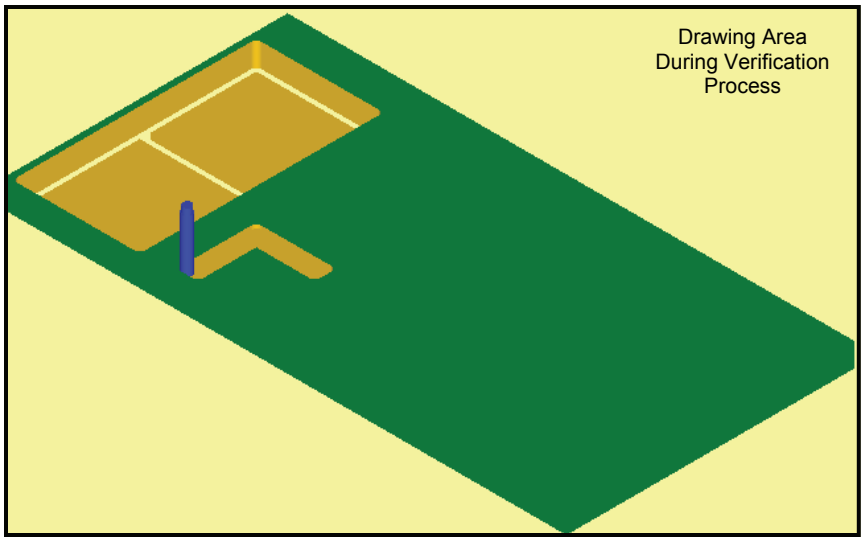
At this time, the red wireframe is now a green solid rectangle and the Verify window is overlaid on the screen.

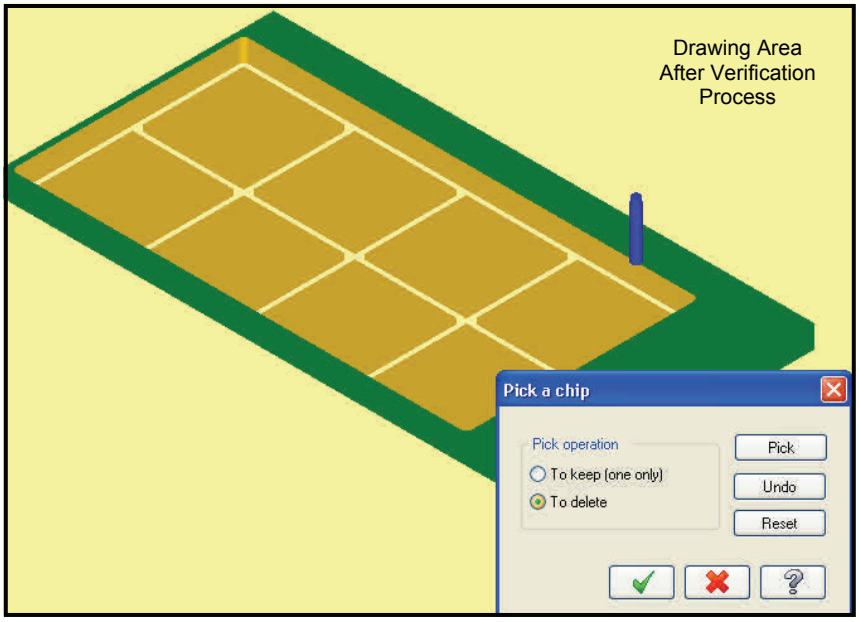
24. To start the verify process, click the "Machine/Play" button.

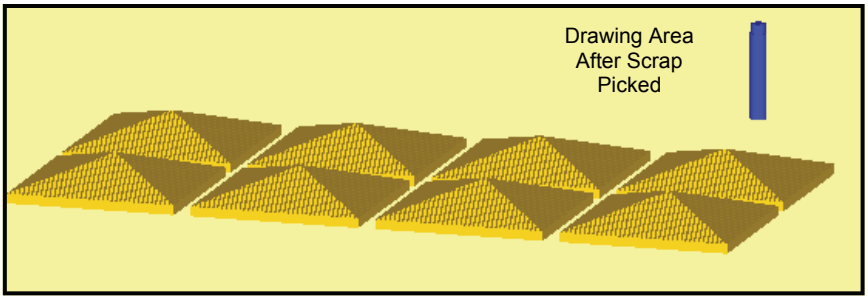




Nesting Example (cont'd)







This page intentionally left blank.

Section 4: Machine Operation

Contents Page

General 4-1
Powering On the VEKTOR 4-3
Spindle Warm Up 4-5
Powering Off the VEKTOR 4-6
Part Processing
- Loading Material 4-8
- Loading Programs 4-9
- Setting Profile Offsets 4-12
- Running Part Programs 4-16
- Restart in Cut - Profile Mode 4-19

How to Calibrate the Setup Tool Length 4-24
How to Calculate Tool Length 4-28
Storing Tool Length 4-31
Using G-Codes to Change a Tool 4-33
Using G-Codes to Change a Tool's Position 4-35
Manually Changing a Tool 4-39
Tool Rack Setup 4-40
Verifying Alarms 4-47
Park Industries Generated Alarms 4-50
Input/Output Screens 4-51

General

This section will describe the use of the VEKTOR control. Using the control for managing programs, for setting up "Profiling or Slab Cutting". Using the control for manually moving the machine and changing the tool. This Section describes how to:

- Powering On the VEKTOR Machine
- Warm Up the Spindle
- Powering Off the VEKTOR Machine
- Part Processing
- Restart in Cut in Profile Mode
- How to Calibrate the Setup Tool Length
- How to Calculate Tool Length
- Storing Tool Length
- Using G Codes to Change a Tool
- Manually Changing a Tool
- Verifying Alarms

NOTE

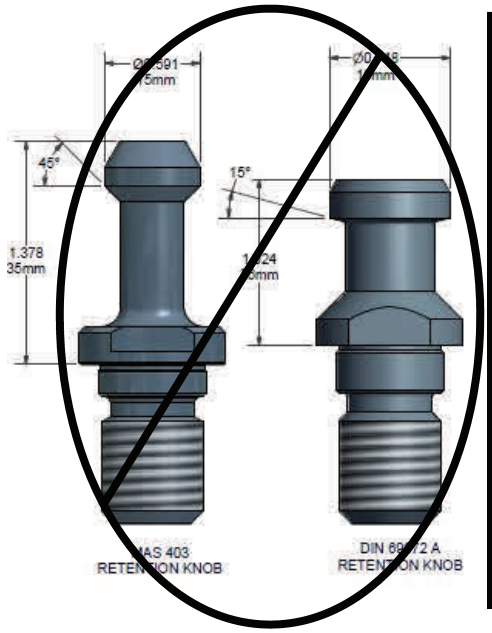
It is the customer's responsibility to make sure all machine operators are trained and well-versed in both the operation of the machine as well as all safety processes and procedures.

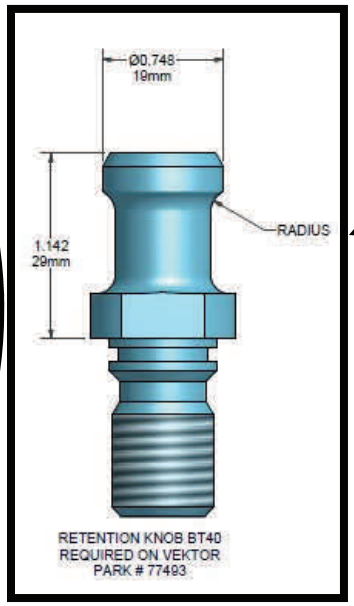
Warning

Your VEKTOR has a high speed spindle. To prolong spindle life, purchase and use only balanced tools and tool holders.

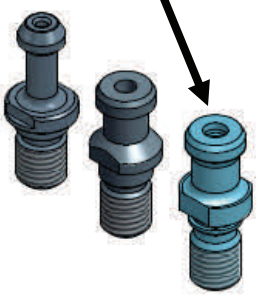
Warning

Damage to the spindle will occur if you don't use Park Industries part # 77943 retention knob with the BT40 tool holder on the VEKTOR.





This is the correct retention knob for the BT40 tool holder.

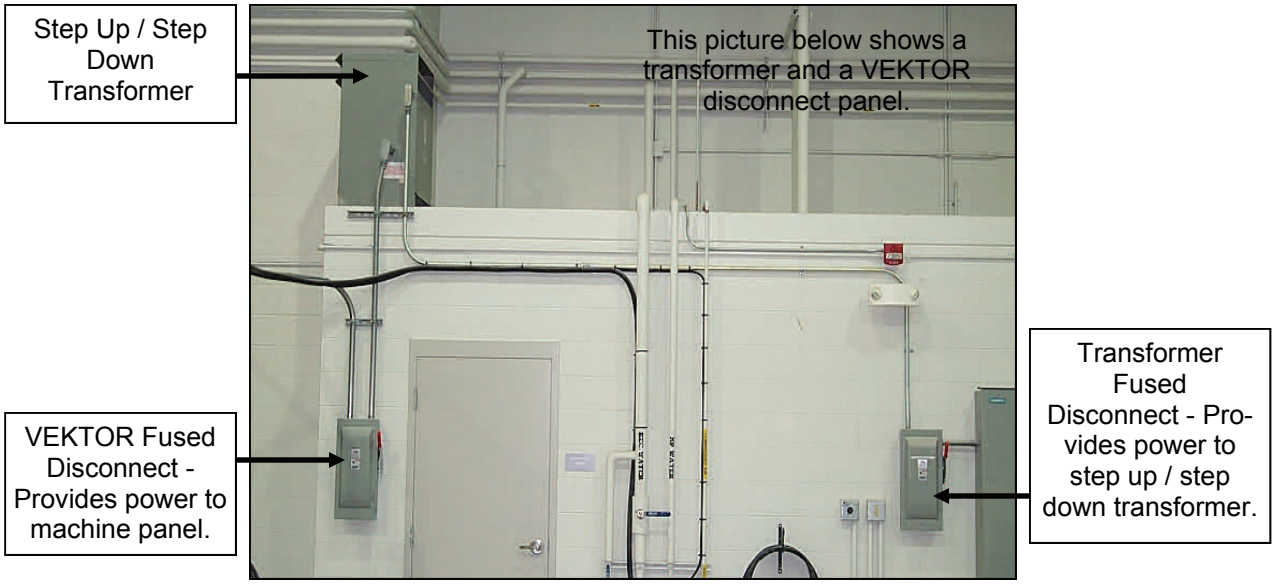


Powering On the VEKTOR

1. **System with NO Transformer** (480VAC 3-Phase power) This system has a single disconnect which controls the main power to the machine. To power on this system locate the disconnect panel switch and turn on the power to the VEKTOR.

   System with Transformer (208. 240. or 600VAC 3-Phase power) In this system there are two fused disconnects: one which controls power to a step up or step down transformer, a second which controls the power going to the machine. Steps to power on this system are:

   - Turn on the power to the transformer at the transformer disconnect.
   - Turn on the power to the VEKTOR at the main disconnect.

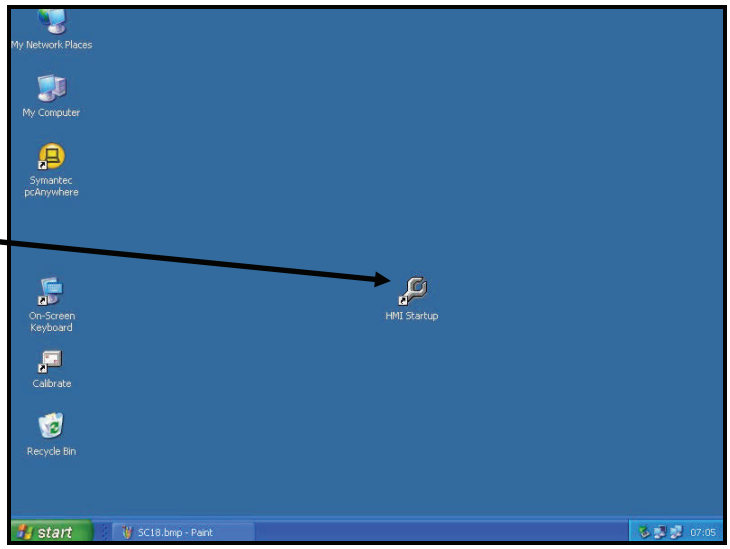


2. Once power has been applied, it will begin the startup process of the Siemens control. The front panel portion of the control runs on Windows XP Professional®.

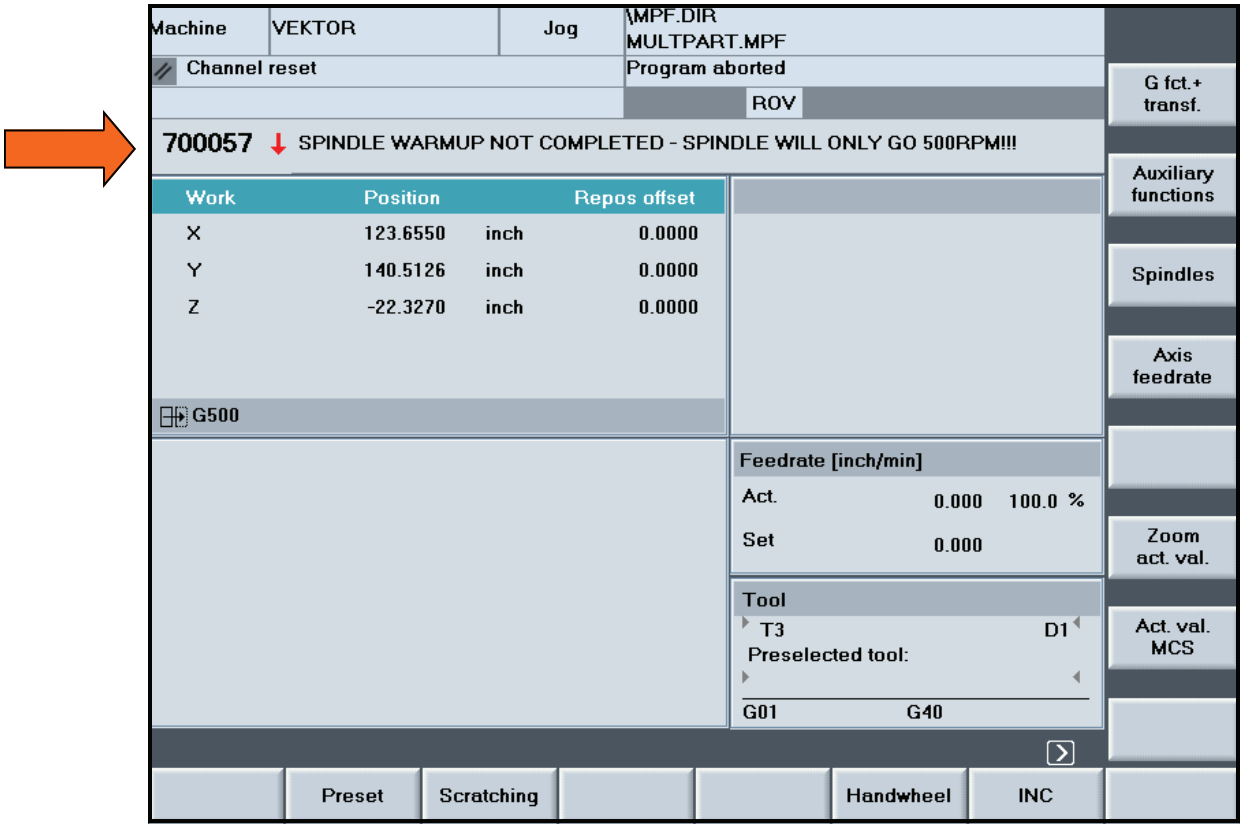
Powering On the VEKTOR (Cont'd)

3. After the control is fully booted up, double click the "HMI Startup" icon on the desktop to start the Siemens software.

"HMI Startup" Icon



4. Beware that there are many programs running in the background for the software to run correctly, so be patient and let all the screens load by themselves. When the software is fully loaded and started the screen shown below will appear. NOTICE the "Spindle Warmup Not Complete" message. The next action is to perform the spindle warm up steps listed on the next page.



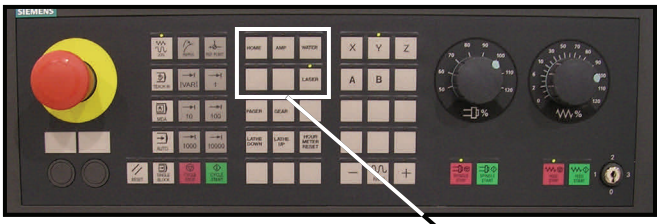
Spindle Warm Up

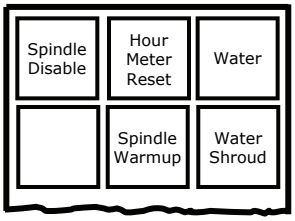
The spindle warm up routine must be run each time power is applied to the VEKTOR. The process, which takes seven (7) minutes, steps up the RPMs until 7000 RPMs is reached. The spindle warm up routine is run to extend spindle life due to bearing failure. The procedure is also recommended any time the spindle has been idle for an extended period of time. This time is depended on shop temperature.

1. Turn system air and water on to the machine.

2. Press the Spindle Warmup button on the Control Panel

Control Panel

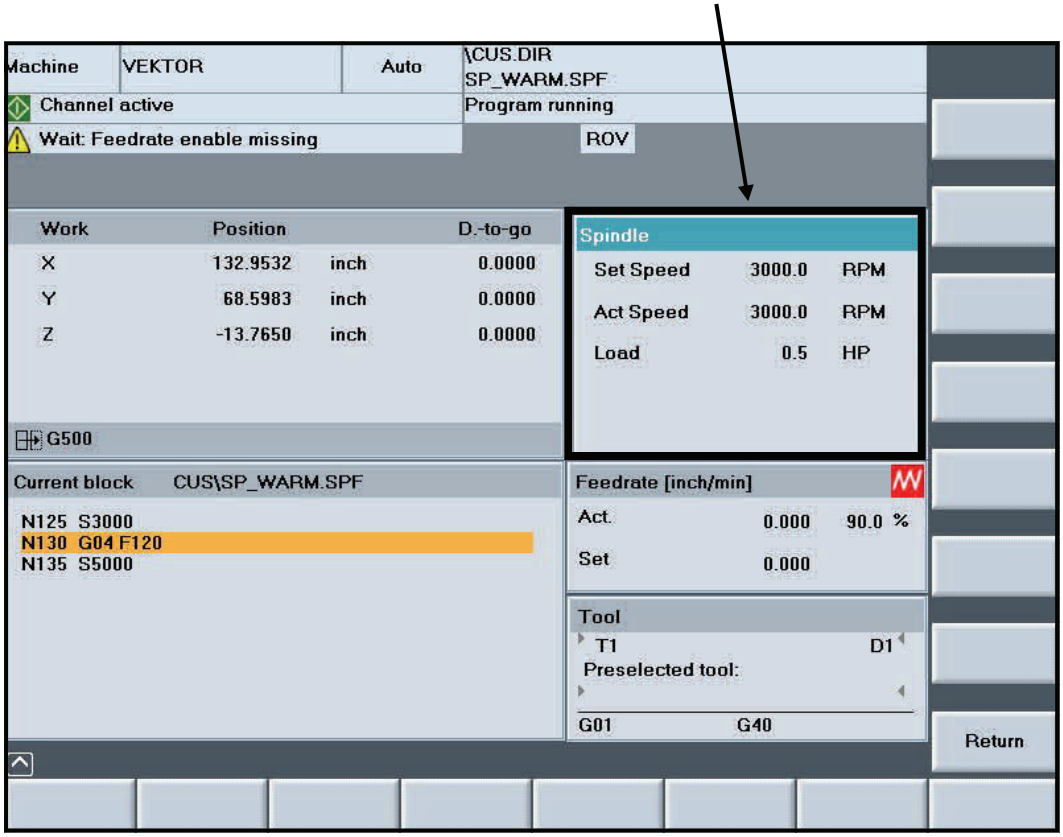




The table to the right shows the spindle RPM and the time in seconds that the machine's spindle will run during the warm up procedure.

RPM	Time (Seconds)
1,500	120
3,000	120
5,000	120
7,000	60

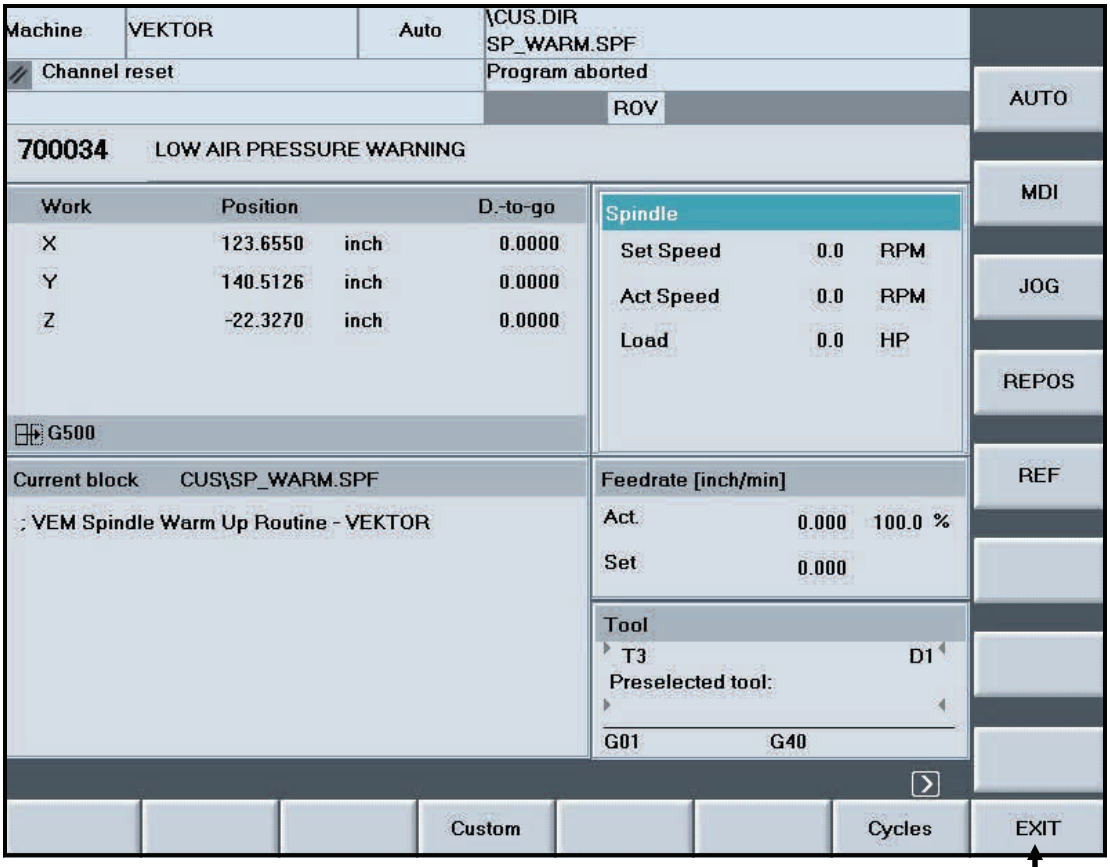
This area of the screen can be used to monitor the spindle RPM during the warm up routine



Powering Off the VEKTOR

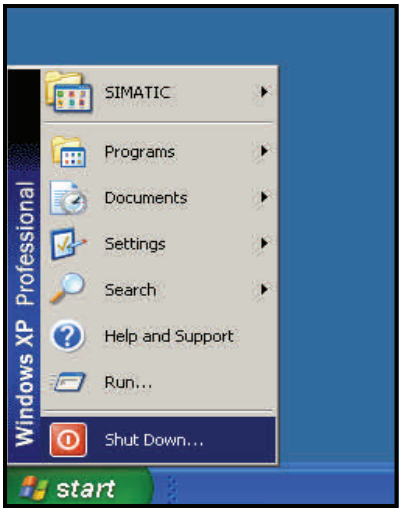
At the end of the day or when the VEKTOR is not in use for an extended period of time, the machine can be powered down. There is no problem keeping the machine on when not in use, but it is a waste of energy. Another good reason to power off the VEKTOR is the potential of an unexpected power loss due to weather related condition (lightening strikes, ice storm, tornado, hurricane etc). Since this system is very sensitive, it maybe worth shutting the machine down if any of these weather conditions are predicated. Follow these steps to power down the VEKTOR

1. E-Stop machine an to shut off the servo drives.
2. Press the “Menu Select” soft key. Verify the screen shown below appears.
3. Press the “Right Arrow” soft key to highlight the “Exit” soft key (lower right corner of screen)
4. Press the “Exit” key.



Powering Off the VEKTOR (Cont'd)

5. The Siemens HMI software will automatically shut down all of the running programs and come back to the normal Windows desktop. At this time use the mouse and click on the "Start" button in the bottom left hand corner and choose "Shutdown".



6. When the Shut Down Windows dialog box appears, verify "Shutdown" is selected and then press "OK".

7. Once the computer totally shut downs (black screen), turn off power at all disconnects or circuit breakers.

8. Turn water and air off to the machine.

Part Processing

This section will describe all items that must be done in order from start to finish to get a program running on the VEKTOR.

The following list the step require to process a part:

- Loading Material
- Loading Part Programs that have been Posted from the Mastercam® software
- Set Part Offsets (Work Coordinates)
- Running the Part Program (Automatic Mode)

Loading Material

The best way to load materials on the VEKTOR is to use an overhead crane. Although not all places have overhead cranes they must then resort to the use of a forklift. When using either device, the slab must be orientated on the “Cutting Bed” the way the Mastercam® drawing is orientated.

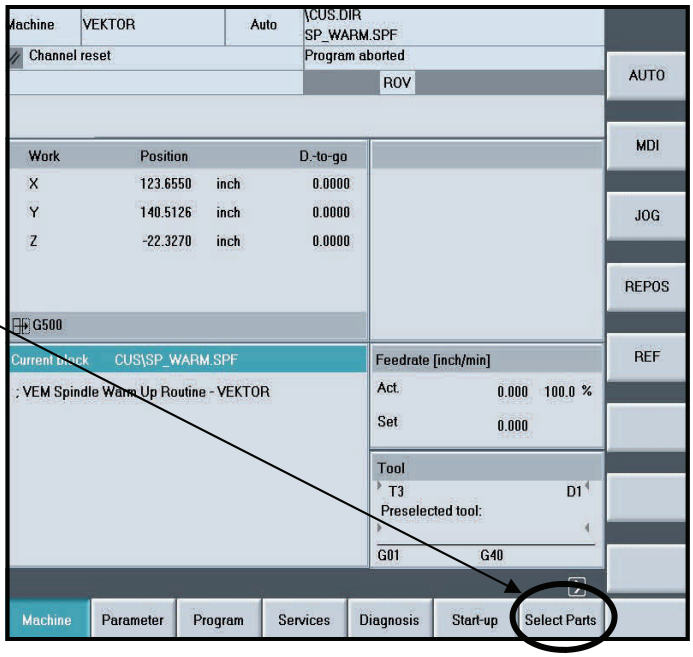
When using the milling bit Park Industries recommends setting the entire slab down on a 4’ by 8’ sheet of Owens Corning Formular Insulated Sheathing either 3/4” or 1” preferably or a similar type material. This material works the best when multiple items of different shapes are nest with in a slab of material. Using this type of material keeps the parts from falling through and also seems to help the parts from moving around when they are being cut.

When setting the slab on either type of material, the slab needs to be positioned so it is parallel to the X or Y axis travel. The axis to align the slab to depends axis the parts were nested to in the Mastercam file. Use a large pry bar to move the slab and to get it lined up.

Loading Part Programs

1. Press the “Menu Select” soft key on the Operator Panel.

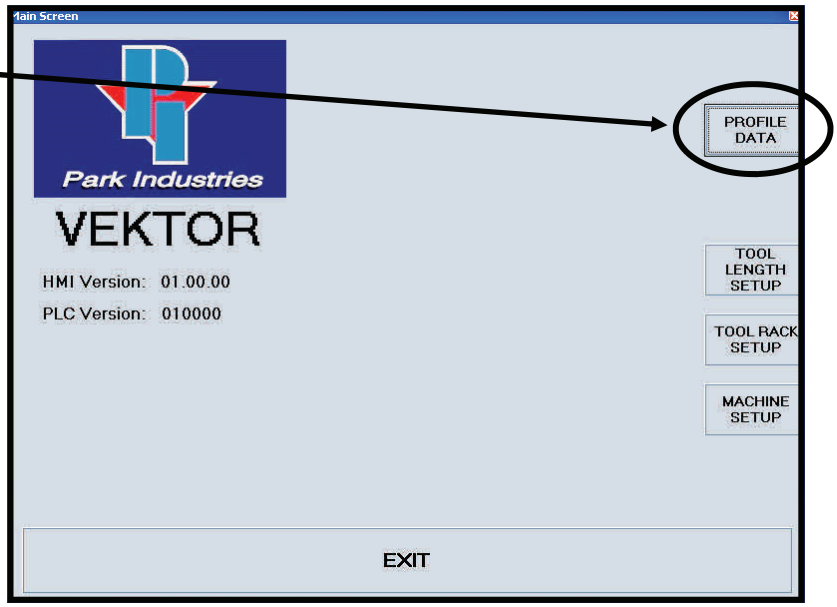
2. Verify the screen shown below appears.



3. Press the “Select Parts” soft key on the main row of keys shown below. This key loads programs created by Mastercam® software.

4. Verify the screen shown below is displayed. This screen is called the “Main” screen.

5. Press the “PROFILE DATA” soft key



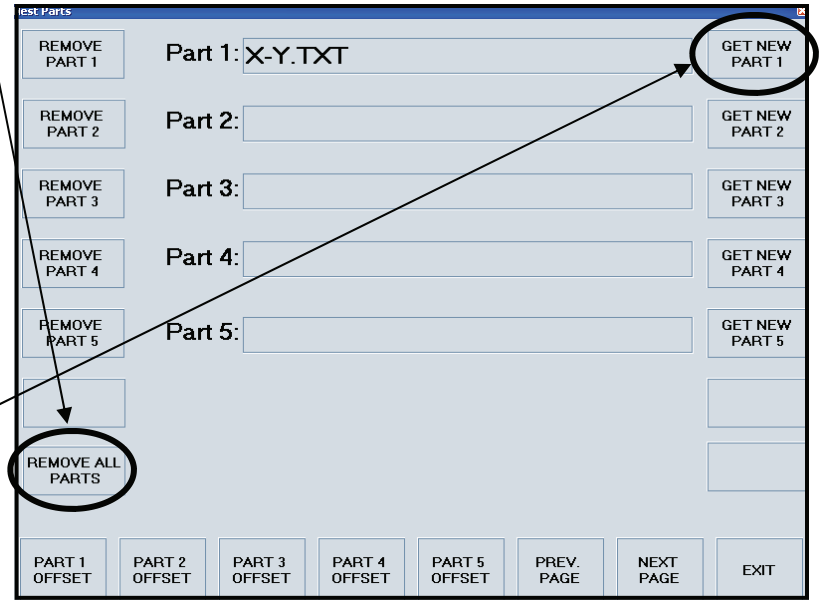
Steps Continued on Next Page

Loading Part Programs (Cont'd)

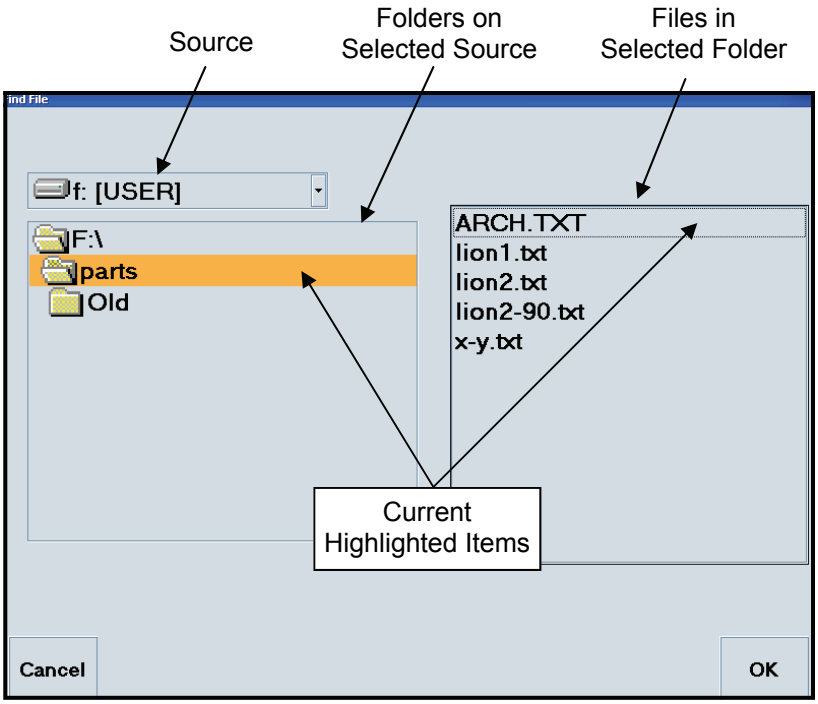
6. Verify the "Nest Parts" screen shown below appears. This is where the part program (created in Mastercam®) and the offset for that program are selected. Fifteen different parts can be selected by using the "Next Page" or "Previous Page" keys.

7. Press "Remove All Parts" to clear any unknown programs that maybe be loaded on another page.

8. Press "Get Part 1" to select the file name that needs to be cut on the first piece or slab of stone.



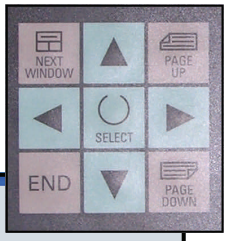
9. Verify the "Find File" screen appears. The screen is used to select the location (source & folder) of the desired .txt file. In this example the screen lists the files found in the "parts" folder on the "F" drive. This is the typical default values.



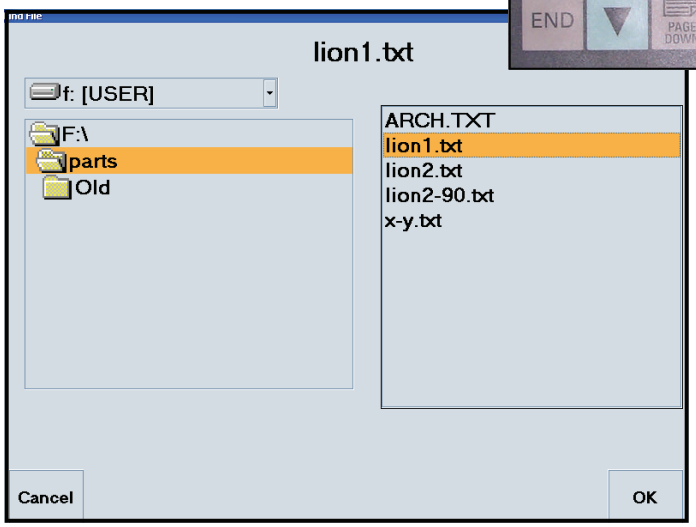
Steps Continued on Next Page

Loading Part Programs (Cont'd)

10. Use the blue arrow keys to move the highlighted line up or down to choose the correct file name. In this example "lion1.txt" has been selected.

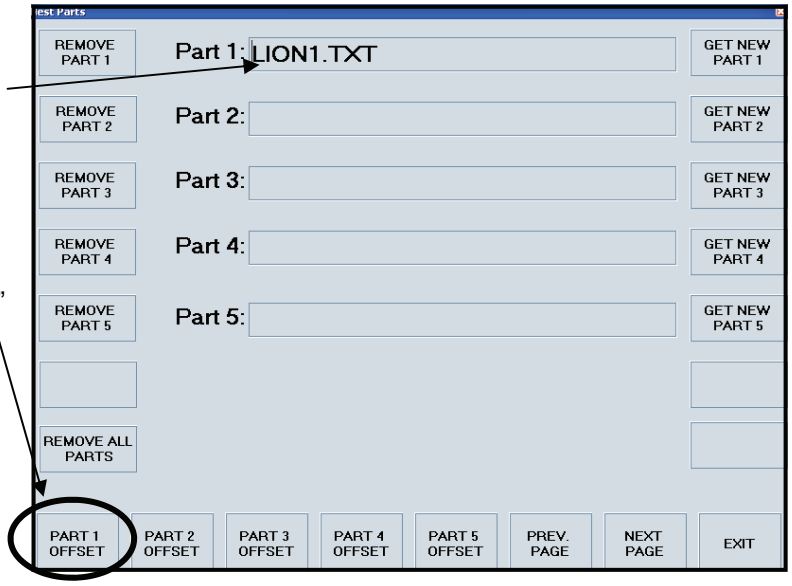


11. Press the "OK" soft key to select the file and exit this screen.



12. Verify the "Nest Parts" screen is displayed with the selected file name displayed in the correct part entry box.

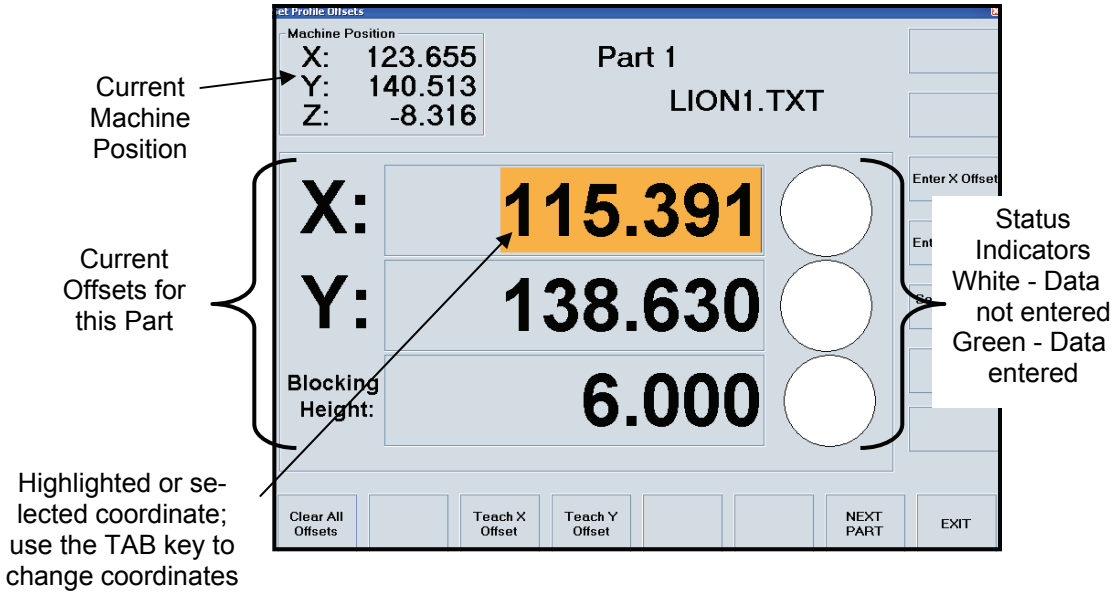
13. Each part program has it's own part offset. Press the "Part 1 Offset" soft key, to match with Part Program 1.



Steps Continued on Next Page

Setting Profile Offsets

14. Verify the "Set Profile Offsets" screen is displayed. The screen allows the user to set the origin offsets (Work Coordinates) for each part.



How to Use This Screen - Before the X and Y coordinate values can be entered they must be determined. In other words the physical point (X and Y coordinates) on the stone which the *Mastercam*® program uses as the point of origin must be located. Note: The point of origin is the lower left corner of the part. Either the operator knows the coordinate or he/she must find them.

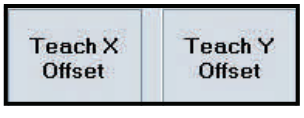
In a shop where the slab is always positioned at the same place on the cutting surface, the machine operator knows or learns the X and Y coordinates to be used as the origin. If the current X, Y, and Z positions are correct, press EXIT. If new known values need to be entered, follow these steps:

- use the "TAB" key to select the coordinate entry box
- use the numeric key pad to enter the value
- press the appropriate "Enter Offset" button (status button turns green)

If the coordinates must be found, use the pendent to move the milling bit over the desired origin point on the stone. There are two methods to enter the actual coordinates:

Pendant:

- turn the axis switch to five (5)
- **DO NOT** hold down the activation switch
- press the F1 button for the X axis and then F2 for Y axis (status buttons turn green)
- turn the axis switch to zero (0)



Screen:

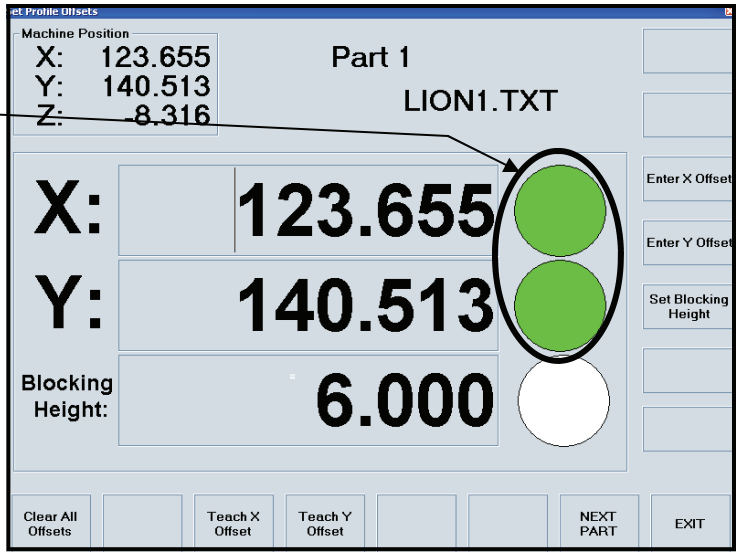
- press the Teach X and Teach Y Offset buttons (status buttons turn green)

Steps Continued on Next Page

Setting Profile Offsets (Cont'd)

15. **<u>If the current X and Y positions are correct, go to step 22</u>**.
<u>If new X and Y positions must be captured</u>. use the pendent to place the milling bit above the point of origin. Using either the pendent buttons or "Teach Offset" buttons, transfer the machine position to current offset area. (See bottom of previous page for steps to transfer the data.)

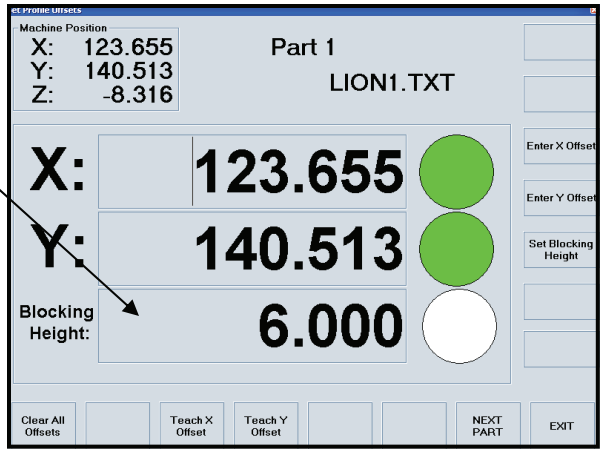
16. Verify that both the X and Y status circles are green.



<u>NOTE</u>:
If the current blocking height is correct, go to step 22.
If a new value has to be entered, continue with the next step.

17. With a standard measuring tape, measure and record the distance between the bottom of the slab and the top of the cutting surface.

18. Press the "TAB " key until the Blocking Height entry area is highlighted.

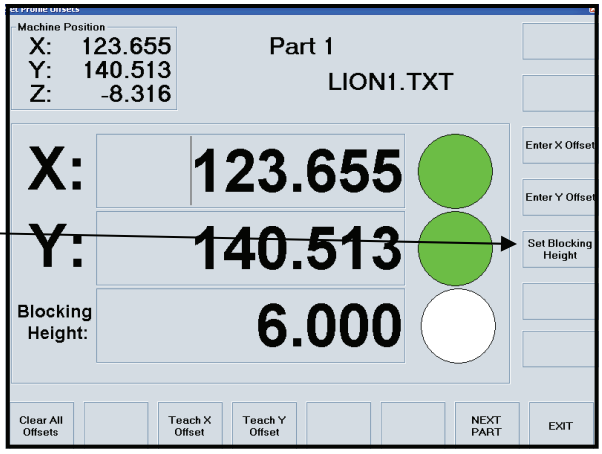


Steps Continued on Next Page

Setting Profile Offsets (Cont'd)

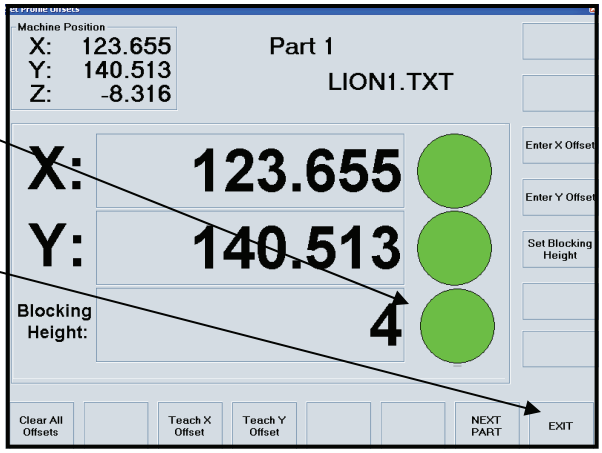
19. Using the numeric key pad, enter the value recorded in step 17. Verify the numbers are changing on the screen.

20. Press the "Set Blocking Height" soft button.



21. Verify the Status circle is green.

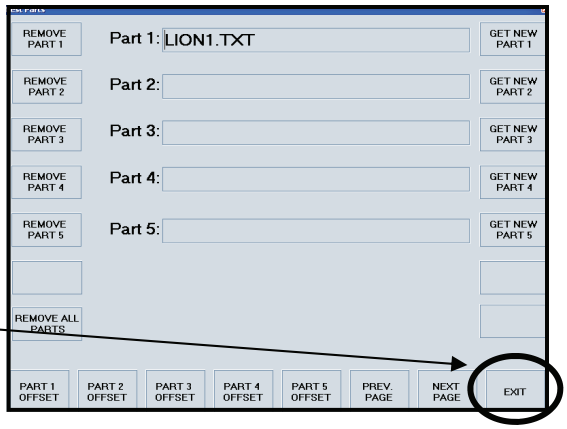
22. Press the "EXIT" soft button.



Steps Continued on Next Page

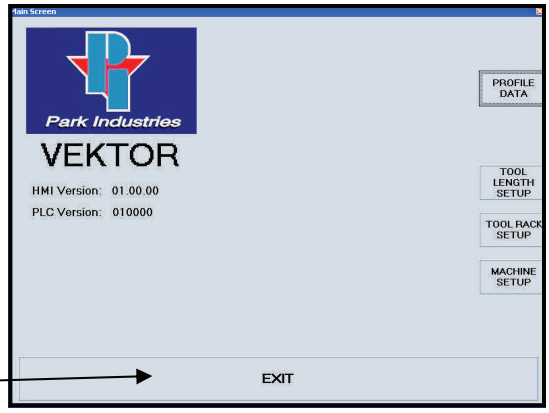
Setting Profile Offsets (Cont'd)

23. Verify the "Nest Part" screen is displayed.



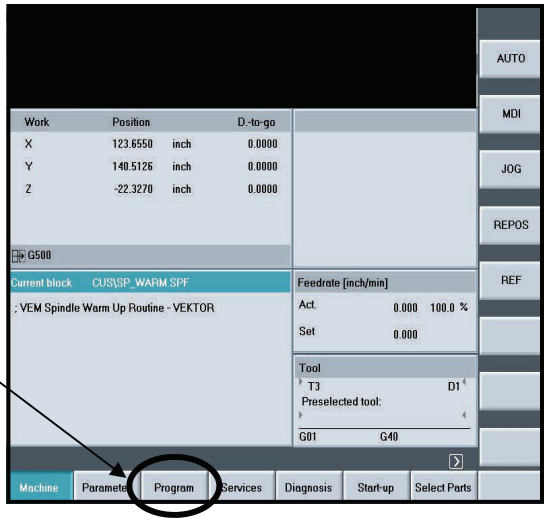
24. Press the "EXIT" button.

25. Verify the "Main Screen" is displayed.



26. Press "EXIT" soft key.

27. Press the "Program" soft key.



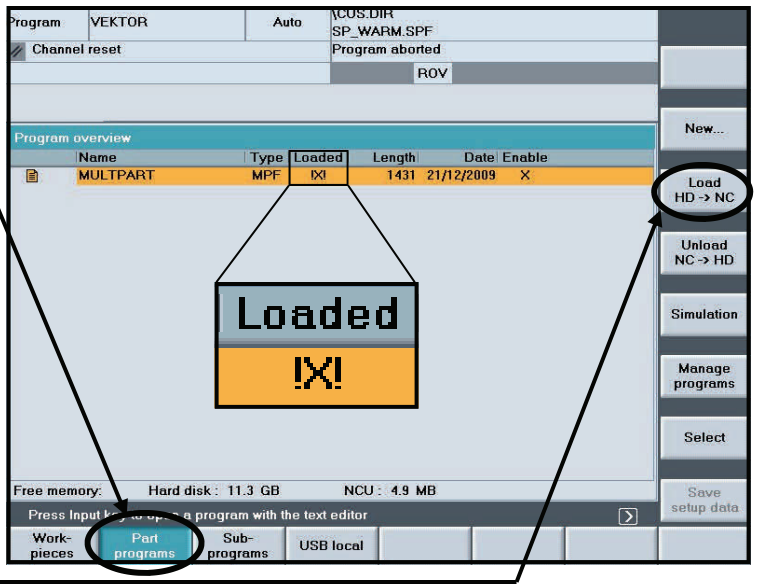
Steps Continued on Next Page

Running Part Programs

28. Verify the Program screen is displayed.

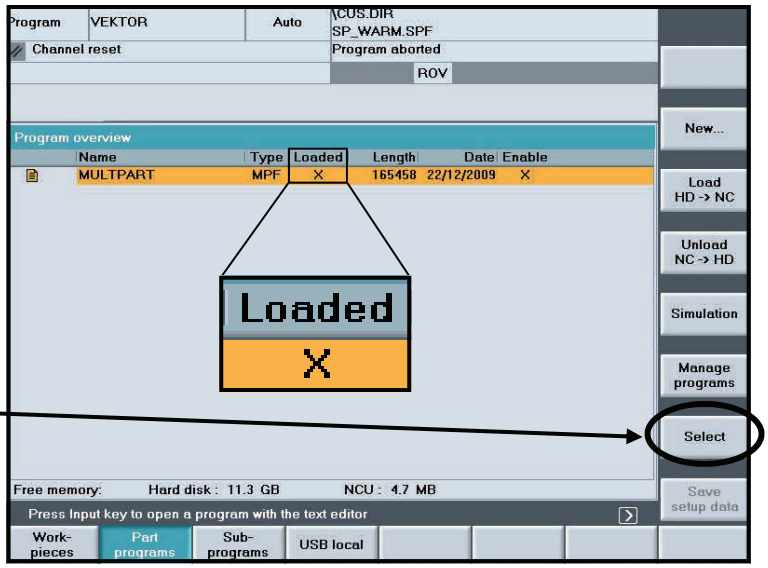
29. If “Part program” is NOT selected (gray letters on a blue background), press the “Part program” soft key.

    NOTE: !X! under the Loaded column means program has NOT been loaded.



30. Press the “Load HD→NC” soft key.

31. Verify !X! has changed to just plain X.



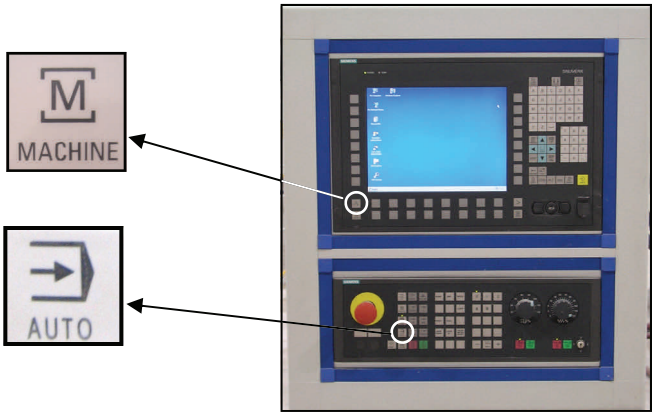
32. Press the “Select” soft key.

Steps Continued on Next Page

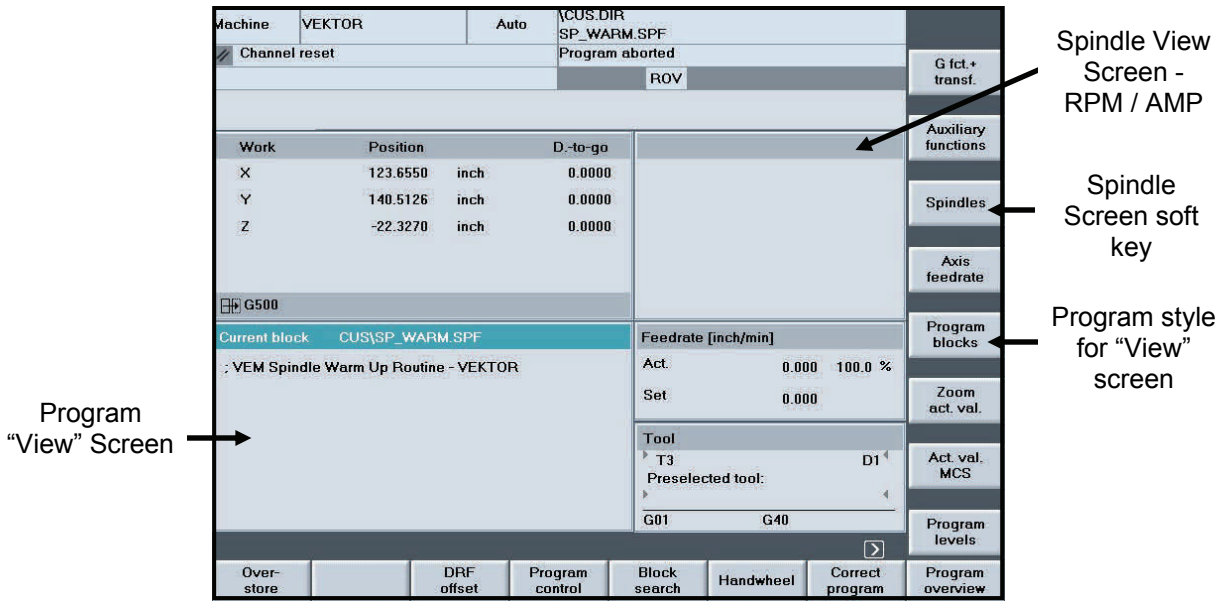
Running Part Programs (Cont'd)

33. Press the "Machine" soft key.

34. Press the "Auto" soft key.



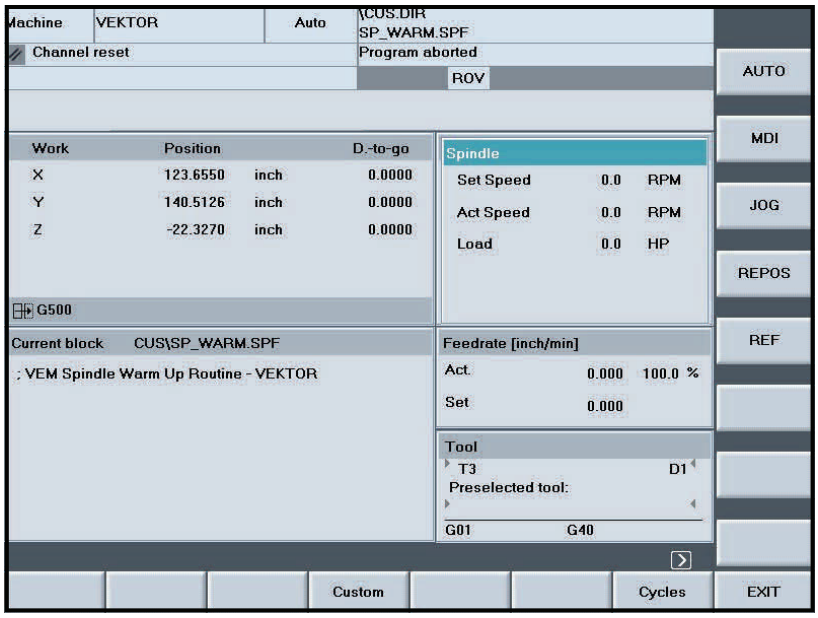
35. Verify the screen shown below is displayed.
    This is the normal "Automatic" machine screen.



Steps Continued on Next Page

Running Part Programs (Cont'd)

36. Press the "Spindles" soft key and also "Program Blocks" to see the screen as shown below



37. Initially set the Spindle Feedrate to 1 or 2%. This is done to ease the tool into the stone.

38. Press the green "Cycle Start" button on the Control Panel to start the program.

39. Raise the Water Shroud.

40. Verify the spindle has moved to correct X and Y position and has started the Z down movement.

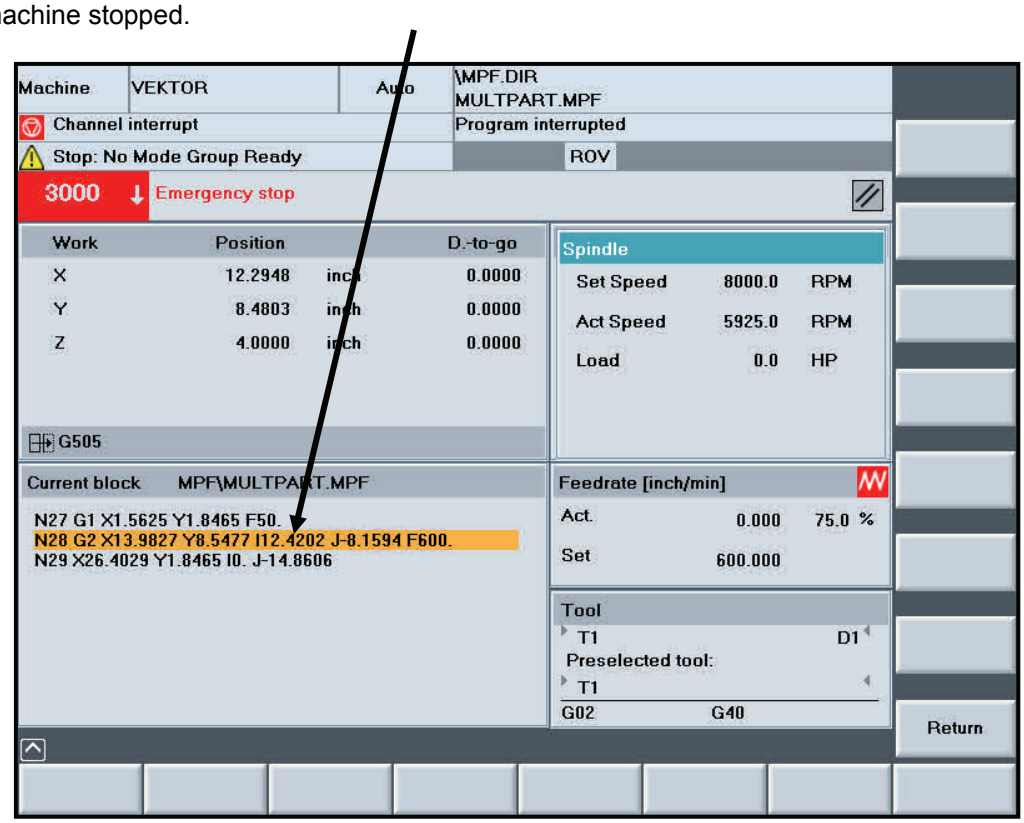
41. When the tool is in the stone, rotate the Spindle Feedrate to 100% or the desired speed for this type of operation.

42 Lower the Water Shroud.

Restart a Cut in Profile Mode

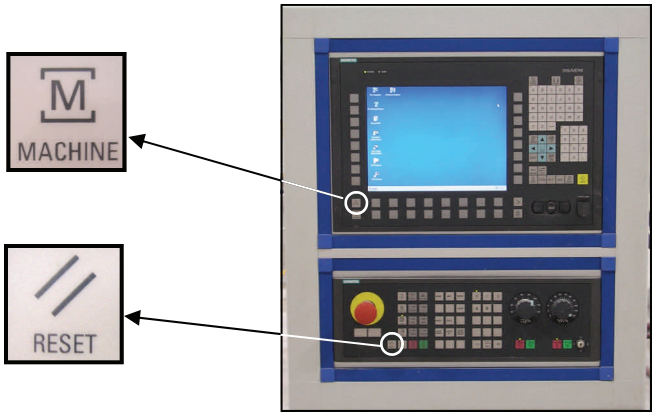
"Restart in Cut" is used to start the machine close to the area, when a program has stopped due to an alarm or the operator has stopped it intentionally. When a condition stops the machine during the cutting of a part, an alarm message is displayed at the top of the screen and the program line number highlighted in the program area. With the use of the "Restart" function the machine can be quickly started where it stopped at, avoiding the necessity to cut the area that has already been done.

1. After an alarm has occurred ***"DO NOT PRESS RESET"***! **Record the line number that the machine has stopped on**. For this example line "N28" is the line being executed when the machine stopped.



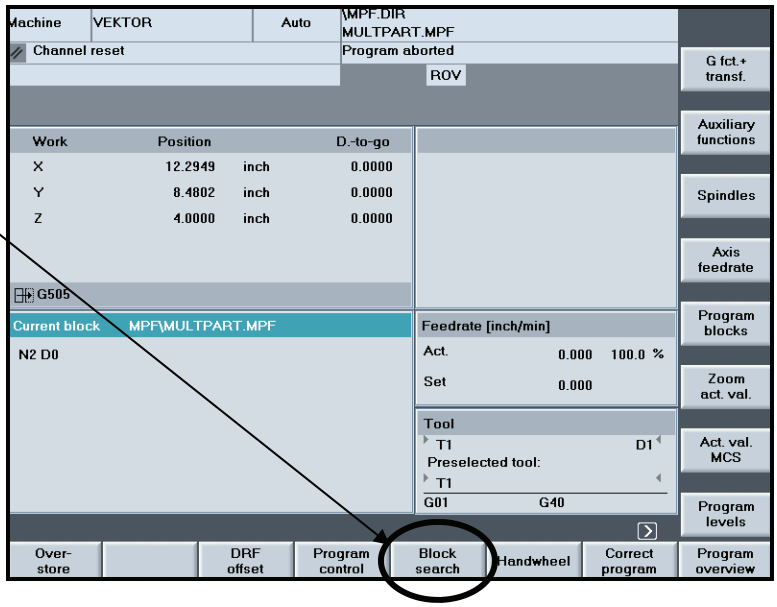
2. Correct the condition which caused the alarm and then press "Reset" button. To troubleshoot alarms refer to the Verifying Alarms part of this section.

3. Press "Machine" button the console.



Restart a Cut in Profile Mode (cont'd)

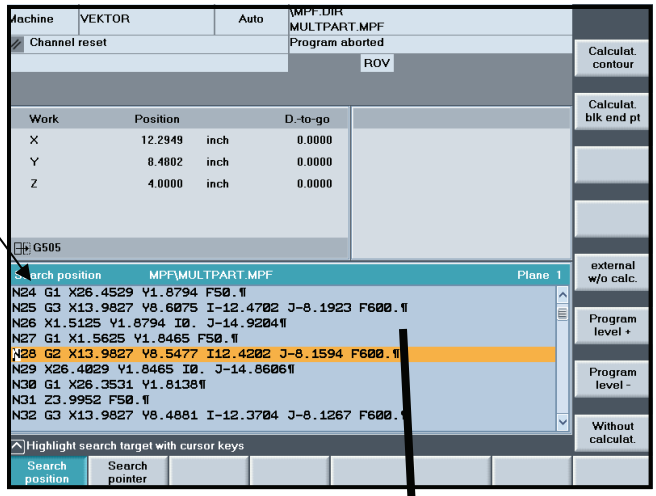
4. Press the "Block Search" soft key



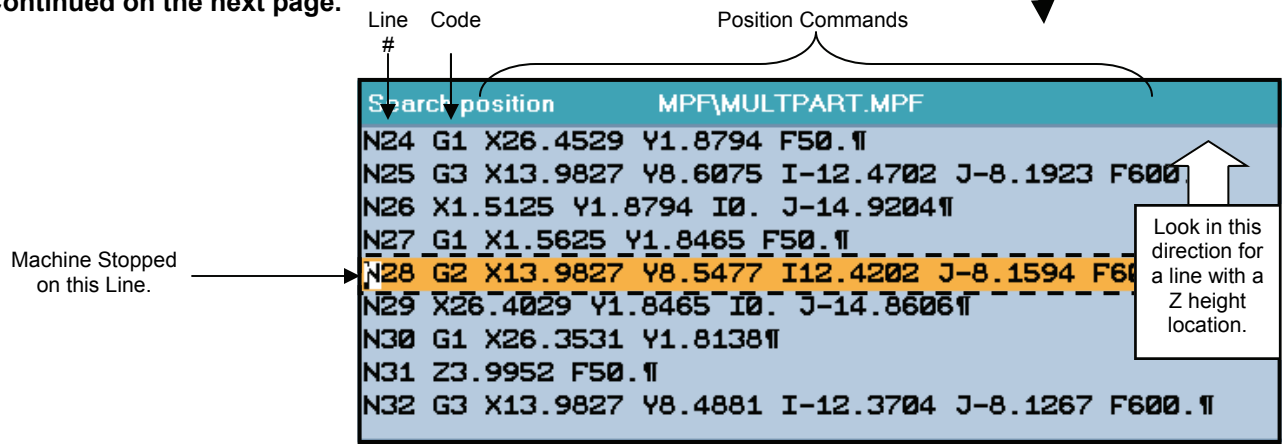
Pressing the Block Search soft key displays nine (9) lines of code, four (4) before and four (4) after the line the program stopped on.

To successfully restart the program, the first code line with a Z height position before the failed code line must be found.

5. Visually search for a Z position command. In this example, lines N28, N27, N26, N25 and N24 do NOT have a Z position command. Go to step 8 if a Z position command is found.

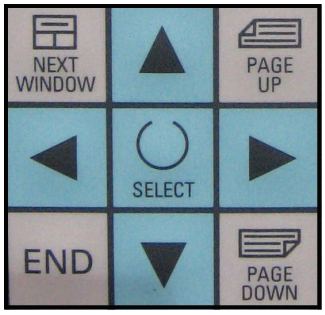


Continued on the next page.



Restart in Cut in Profile Mode (cont'd)

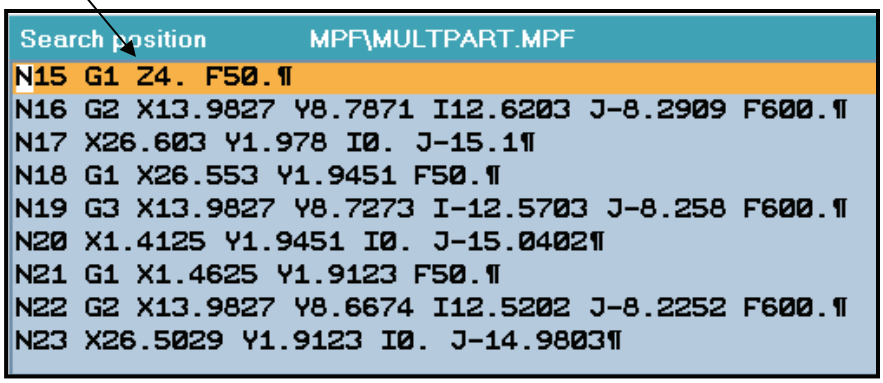
6. Press the "PAGE UP" button once to display nine (9) lines of code.



7. Visually search for a Z height position command.

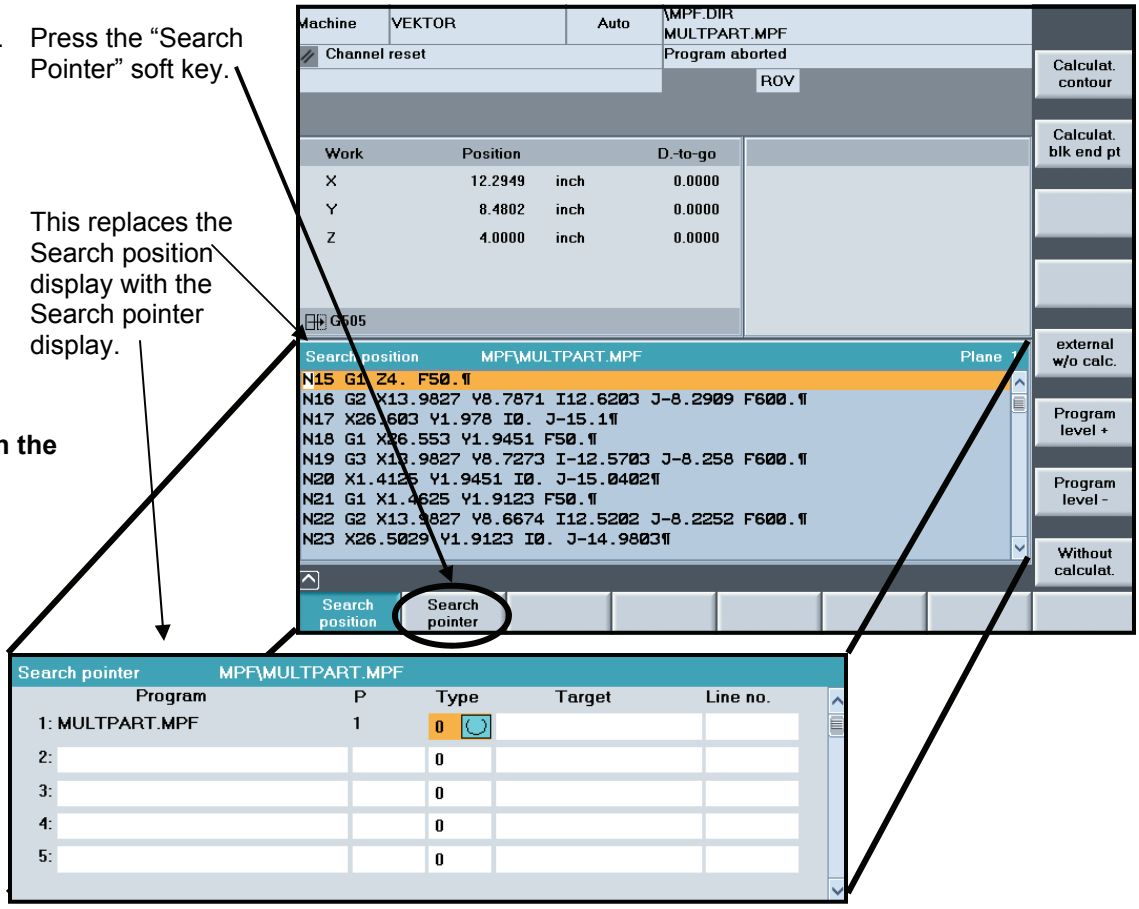
NOTE: Repeat steps 6 and 7 until a Z height position command is found.
In this example line N15 has Z4.

8. Record the "N" number.



9. Press the "Search Pointer" soft key.

This replaces the Search position display with the Search pointer display.

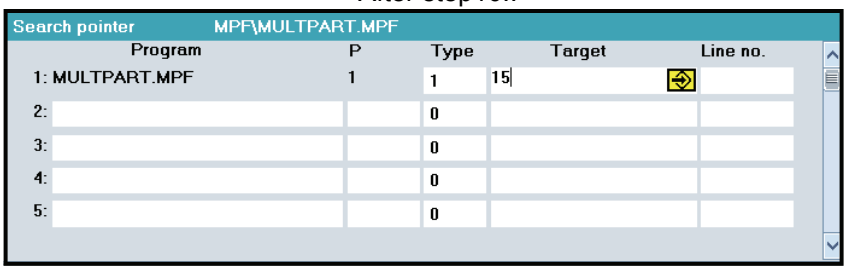


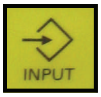
Continued on the next page.

Restart in Cut in Profile Mode (cont'd)

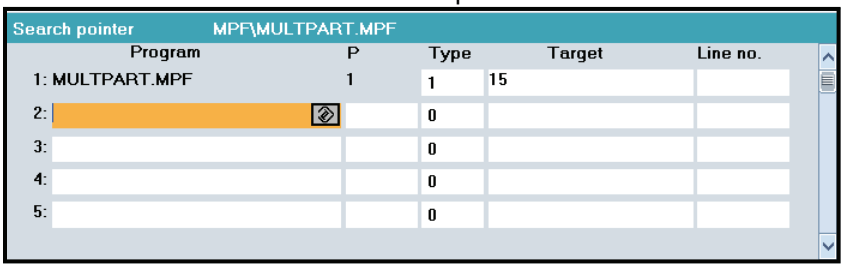
10. Using the numeric key pad enter the following:
    - "Type" - 1 (always set to 1)
    - "Target" - line number recorded in step 8 (the letter "N" is not entered)

After step10..



11. Press 

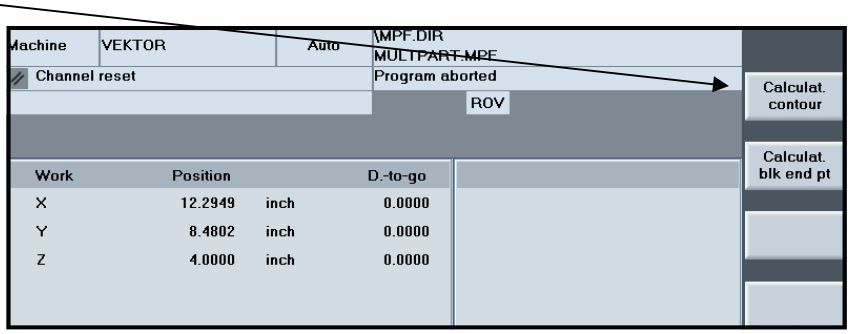
After step11..



12 Press the "Calculat contour" soft key

.

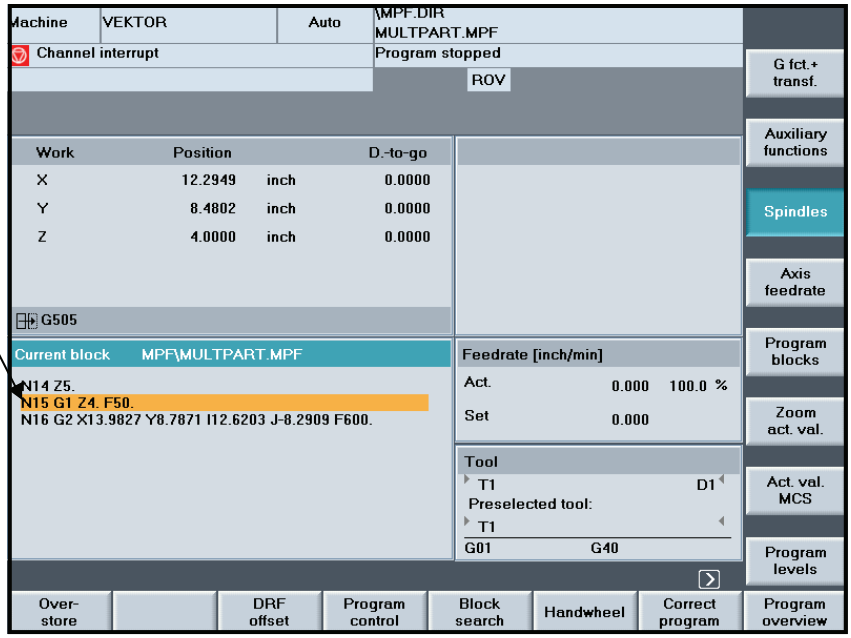
Continued on the next page.



Restart in Cut in Profile Mode (cont'd)

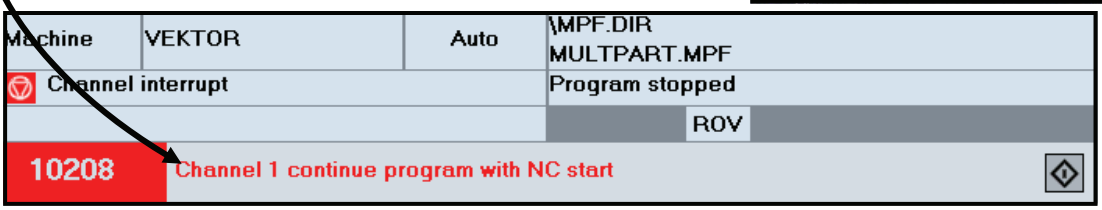
13. The main screen is displayed again.

    The line number entered in step 10 is highlighted in the Current block section of the screen.



14. Press the "Cycle Start" pushbutton on the control panel. A little subprogram will run – this puts the machine into the correct X & Y position, and starts the spindle. Once the spindle is started, the machine will Z down to a starting position, (slightly above the cut position) and then stop.

15. The operator is prompted to press the "Cycle Start" pushbutton on the control panel. The program starts at the block number entered in step 10.



How to Calibrate the Setup Tool Length

This procedure uses the Park Industries setup tool supplied with the VEKTOR, to calibrate the Tool Length Setup program. The procedure would be used if the Tool Length Measuring device ever got moved, service work done, or bumped into.

1. Locate the Park Industries setup tool.

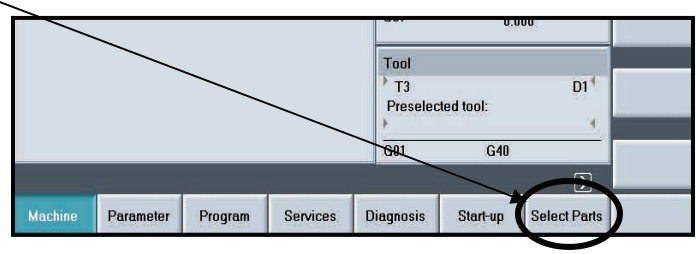
2. Record below the length stamped on the side on the setup tool.

   _____

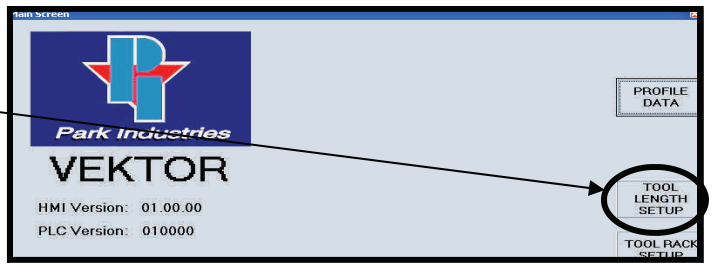
3. Load the set up tool using the G Code or Manual Tool Change method listed in this section.

4. Press the “Menu Select” key.

5. Press “Select Parts”

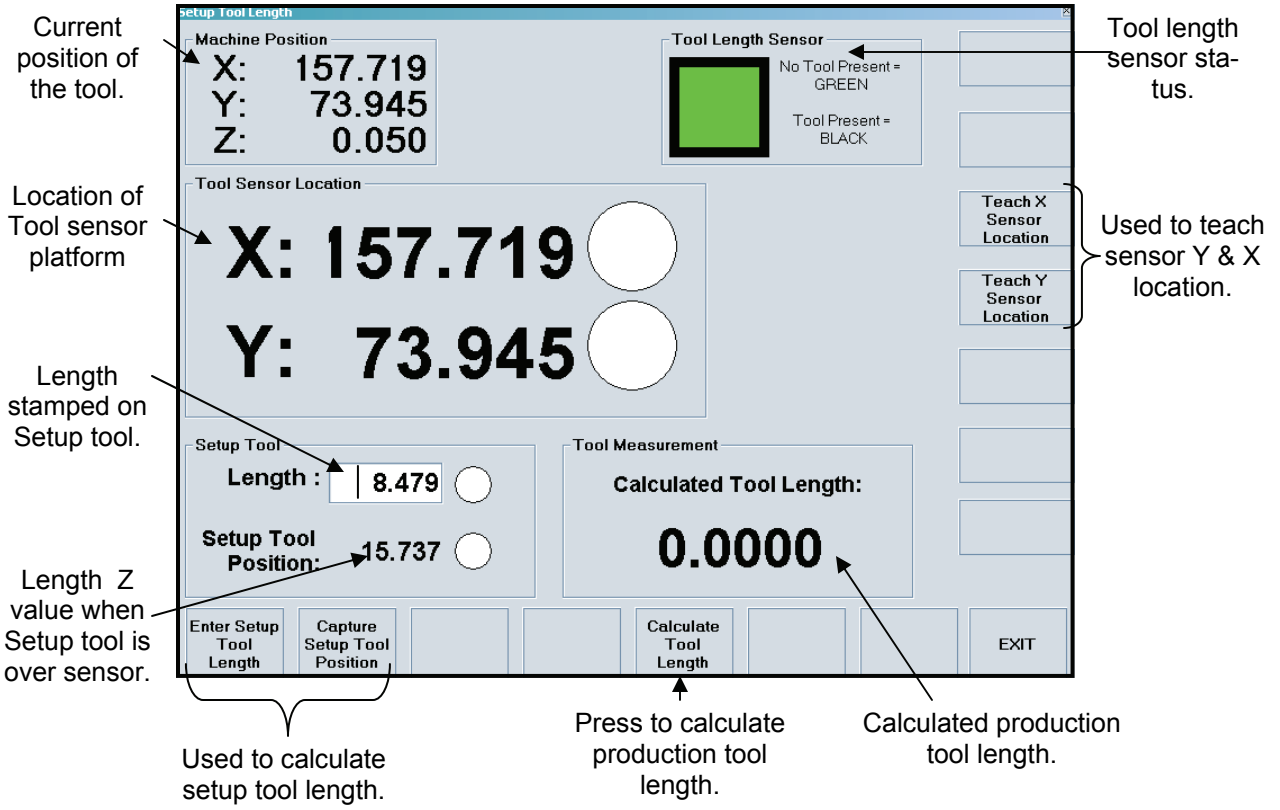


6. Press “Tool Length Setup” soft button.

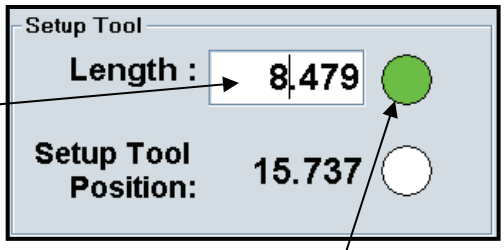


How to Calibrate the Setup Tool Length (cont'd)

Brief Explanation of Tool Length Setup Screen Buttons and Displays (steps continue at bottom of page)



7. Verify the cursor in the length entry box.



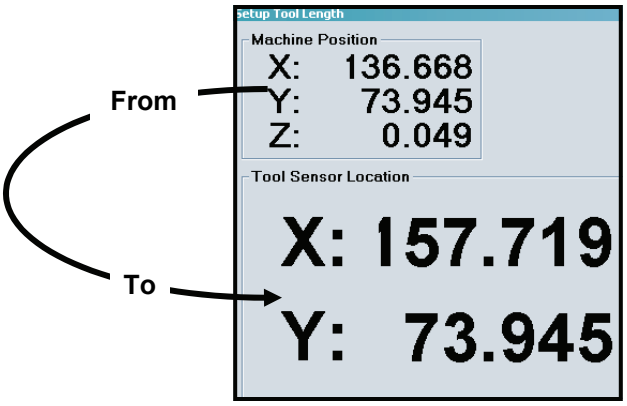
8. Using the keyboard, enter the tool length stamped on the Park Industries setup tool. This is the number recorded in step 2.

9. Press "Enter Setup Tool Length" soft button. The circle to the right of the entry box changes from white to green.

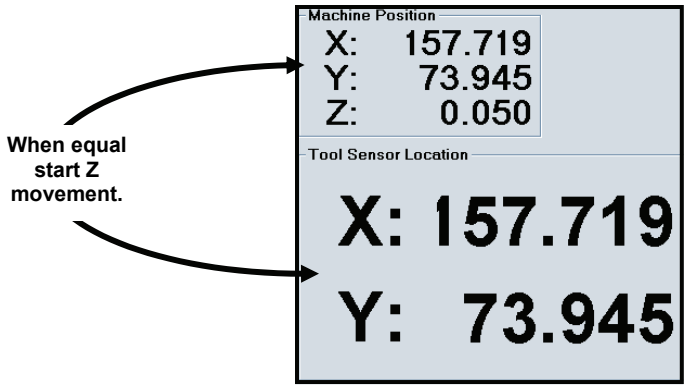
How to Calibrate the Setup Tool Length (cont'd)

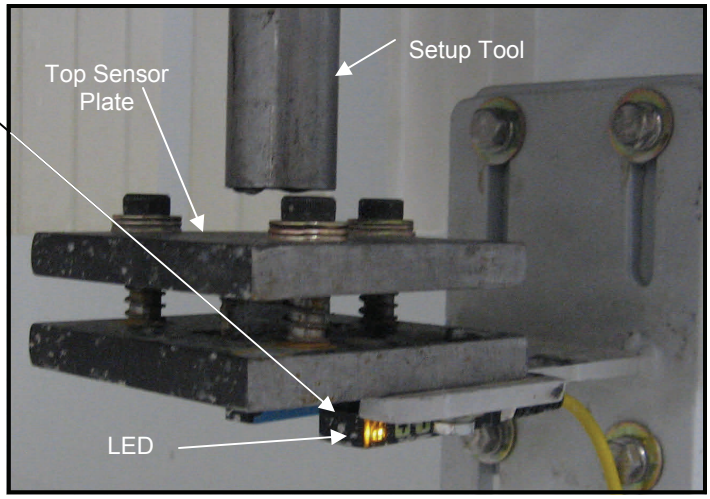
10. Use the pendent to move the tool from the X and Y "Machine Position" to the "Tool Sensor Location"

CAUTION: As tool approaches the senor, verify the tool clears the senor plate and <u>does not crash</u> into the side.



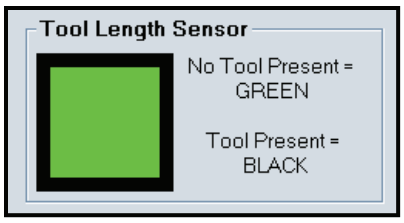
11. When "Tool Sensor Location" and "Machine Position" X and Y locations are equal, use the pendent (F1 or F2 key) to lower (Z) the tool toward the sensor plate. **<u>Stop when the LED on the bottom side of the sensor plate goes off.</u>**





How to Calibrate the Setup Tool Length (cont'd)

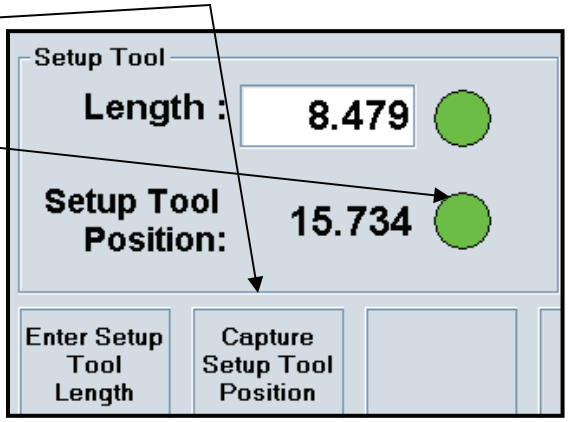
12. At the console, verify that the "Tool Length Sensor" toggles between green and black when the Z axis is moved up and down. **Stop when on the click which toggles the LED from green to black.**



13. Press the "Capture Setup Tool Position" soft key.

    Verify the circle to the right of the entry box changes from white to green.

    NOTE: The Setup Tool Position value is the sum of the current Z position and setup tool Length number.



14. Press the "Exit" soft key which is located in the lower right corner of the screen.

15. Using the pendent or console, move the spindle to the Z upper limit.

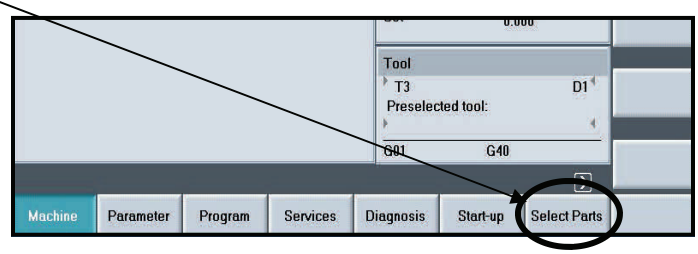
How to Calculate Tool Length

The tool length needs to be calculated and entered into the VEKTOR tool library. The length could be found using many different measuring devices. The tool length is needed for precision toolpaths in the stone. The VEKTOR's built-in tool measuring device is used to find the tool length.

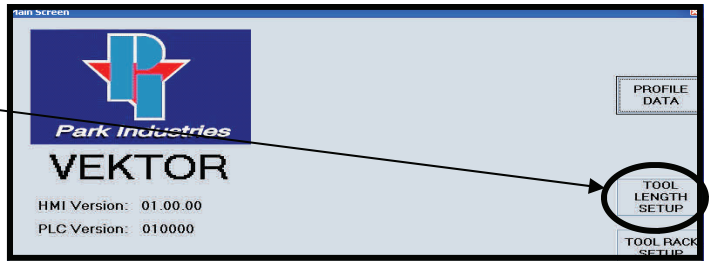
1. Load the set up tool using the G Code or Manual Tool Change method listed in this section.

2. Press the "Menu Select" key.

3. Press "Select Parts".



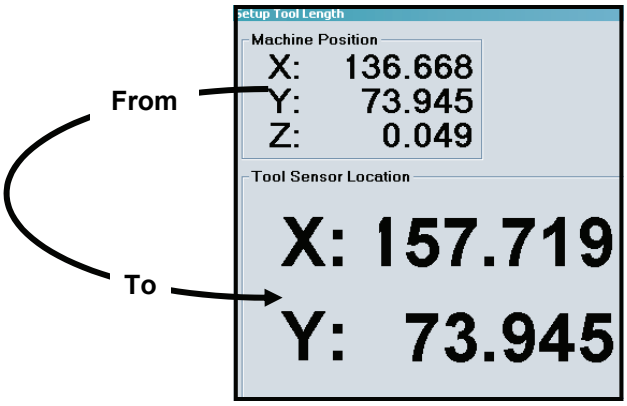
4. Press "Tool Length Setup" soft



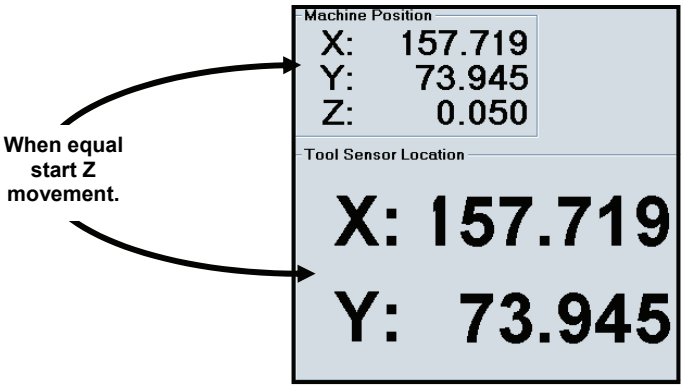
How to Calculate Tool Length (cont'd)

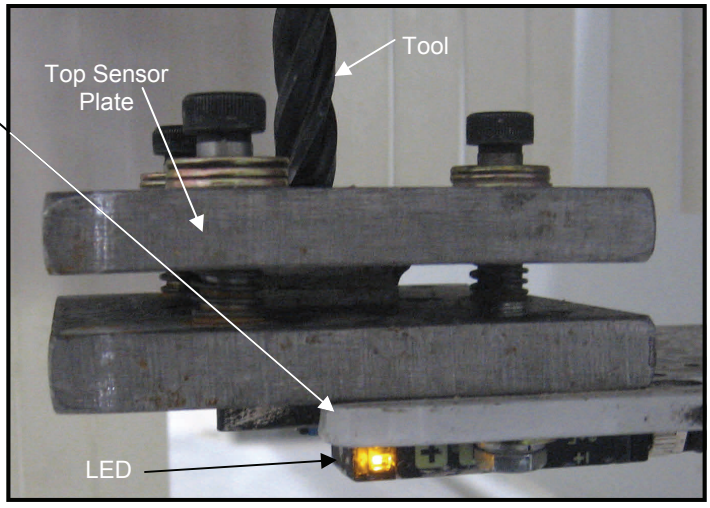
5. Use the pendent to move the tool from the X and Y "Machine Position" to the "Tool Sensor Location"

CAUTION: As tool approaches the senor, verify the tool clears the senor plate and <u>does not crash</u> into the side.



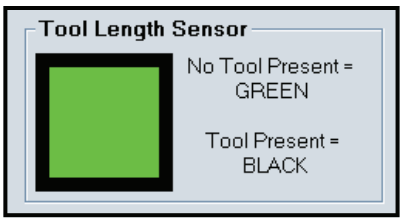
6. When "Tool Sensor Location" and "Machine Position" X and Y locations are equal, use the pendent (F1 or F2 key) to lower (Z) the tool toward the sensor plate. **<u>Stop when the LED on the bottom side of the sensor plate goes off.</u>**





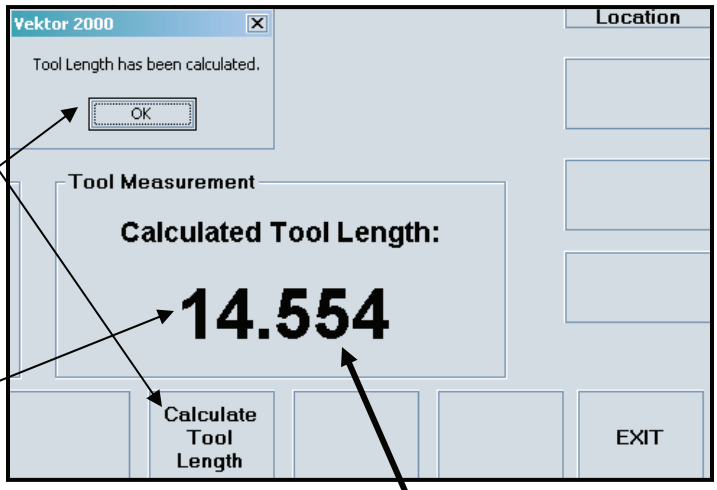
How to Calculate Tool Length (cont'd)

7. At the console, verify that the "Tool Length Sensor" toggles between green and black when the Z axis is moved up and down.
   Stop when on the click which toggles the LED from green to black.



8. Press the "Calculate Tool Position" soft key.

9. A dialog box stating the tool length has been calculated appears to indicate the process is done. Click OK button.



NOTE: The Calculated Tool Length number must be recorded. The next procedure illustrates how to store this number on the Parameter page.

RECORD the tool number and Calculated Tool Length number before leaving this screen.

10. Record the number now.

11. Press the "Exit" soft key which is located in the lower right corner of the screen.

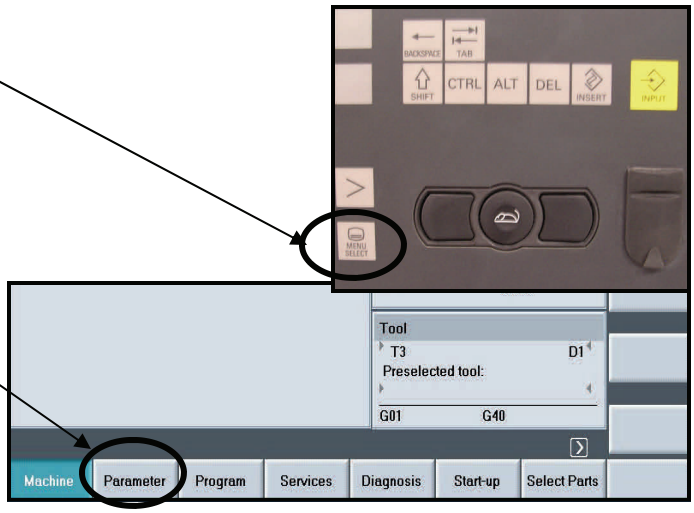
12. Using the pendent or console, move the spindle to the Z upper limit.

Storing Tool Length

Once the tool length has been calculated, it must be entered into the VEKTOR tool library to ensure precision toolpaths in the stone. Use the “How to Calculate Tool Length” procedure to

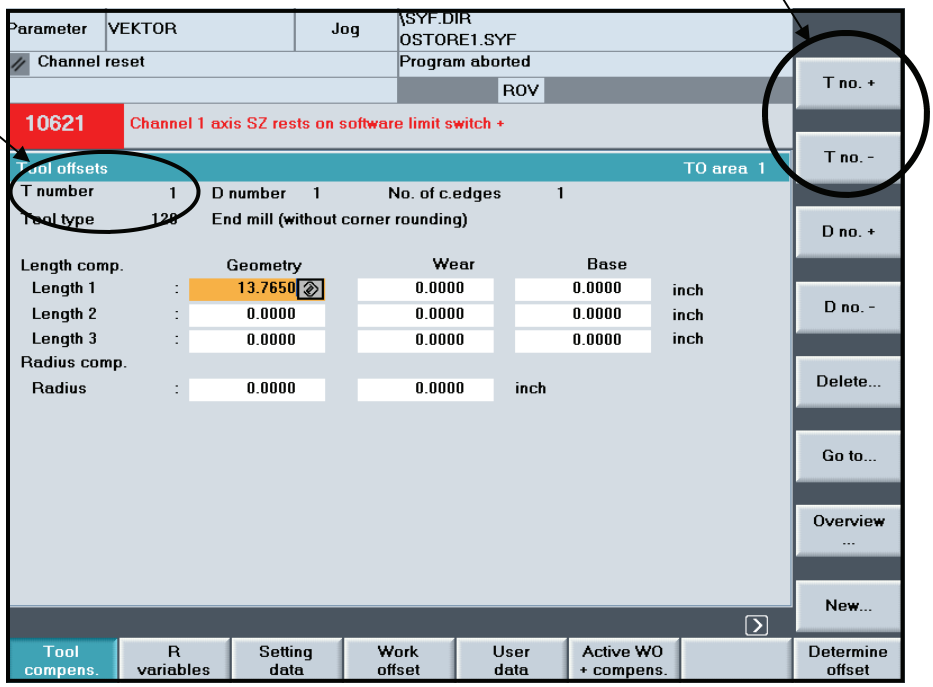
1. Press the “Menu Select” key.

2. Press “Parameter”



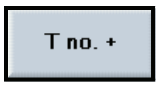
3. Verify the Parameter screen is displayed.
   At this time parameters for tool T1 are displayed.
   The “T no.+” and T no.-” keys are used to select other tool numbers.

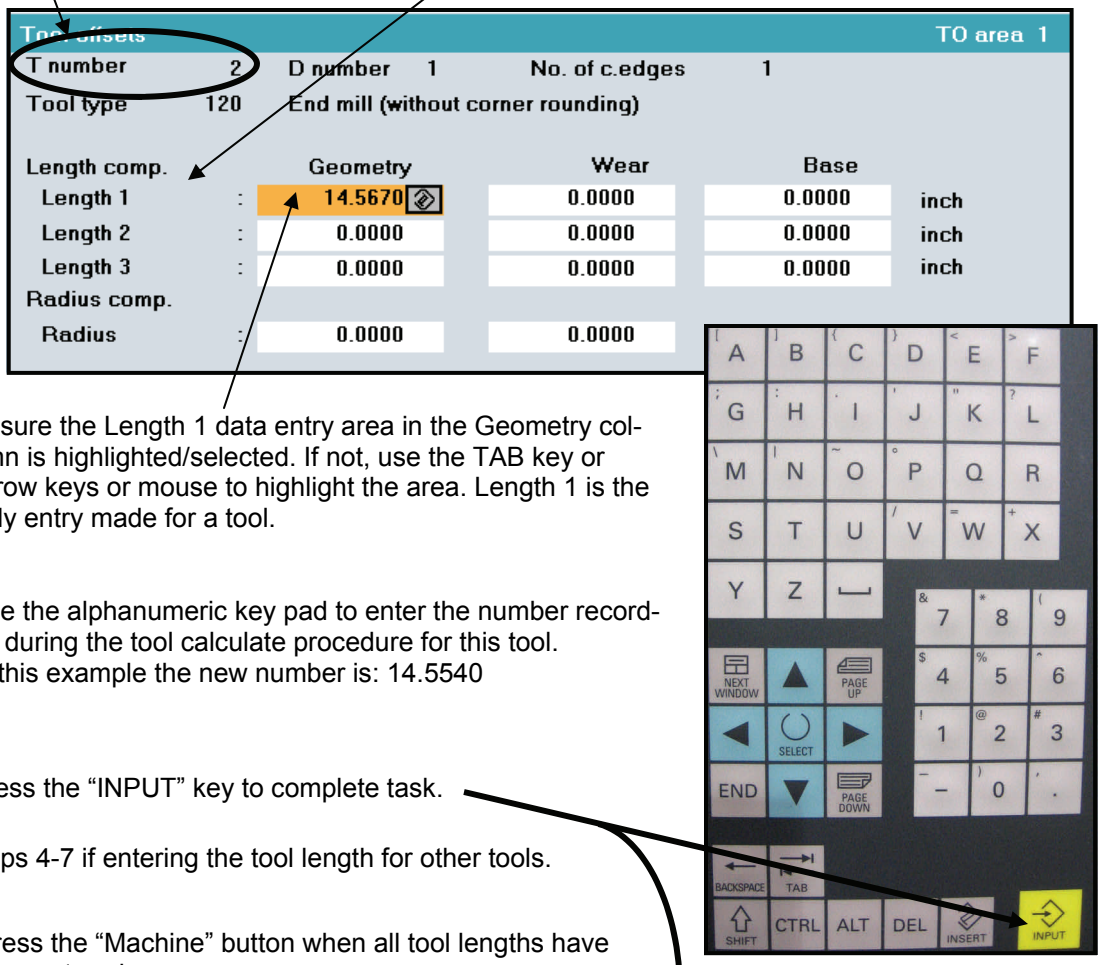
Current T number being displayed.



Storing Tool Length (cont'd)

4. Press "T no.+" or "T no.-" keys until the parameters of the desired tool are displayed. In this example the length of tool 2 is being changed.





5. Ensure the Length 1 data entry area in the Geometry column is highlighted/selected. If not, use the TAB key or Arrow keys or mouse to highlight the area. Length 1 is the only entry made for a tool.

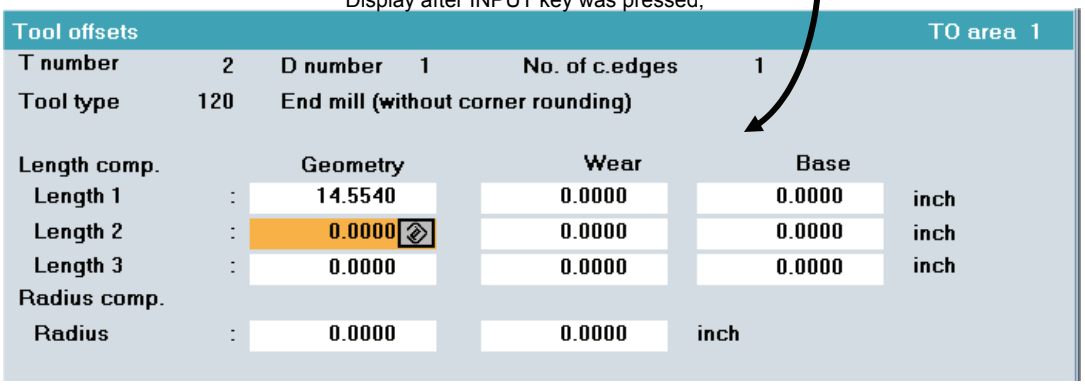
6. Use the alphanumeric key pad to enter the number recorded during the tool calculate procedure for this tool.
In this example the new number is: 14.5540

7. Press the "INPUT" key to complete task.

Repeat steps 4-7 if entering the tool length for other tools.

8. Press the "Machine" button when all tool lengths have been entered.

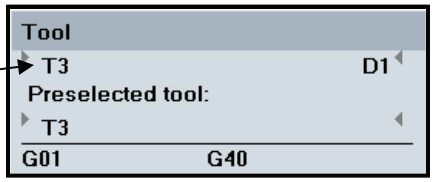
Display after INPUT key was pressed,



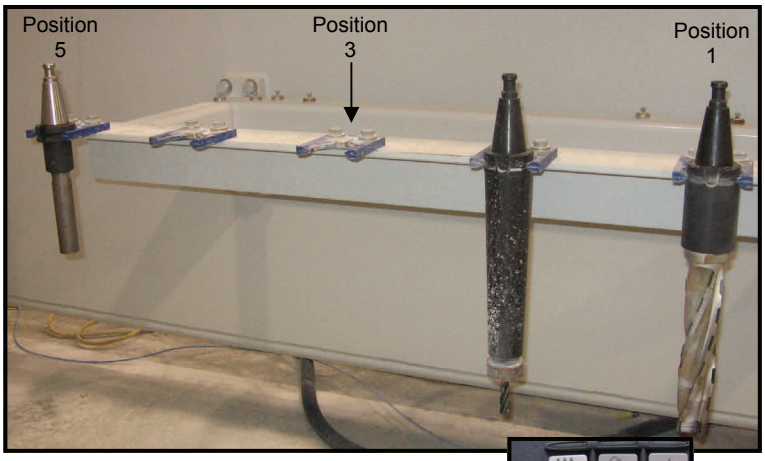
Using G Codes to Change a Tool

Using the manual method to change a VEKTOR tool may be cumbersome for just one person. The G-code method is used to change a tool by typing the correct information into the console, and then allowing the machine to execute the tool change.

1. The “Tool” display (located in the lower right corner on the screen) lists the tool currently loaded. In this example it is T3.

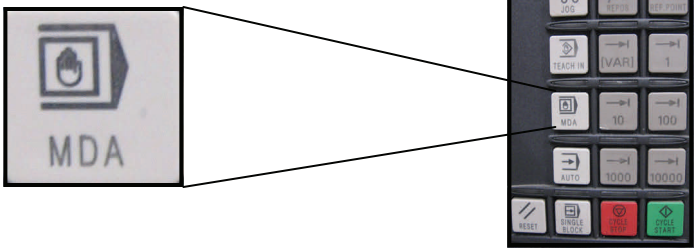


2. Verify that the position of the tool listed in step 1 is open in the tool rack. (Six position tool racks are found on all newer VEKTOR units)

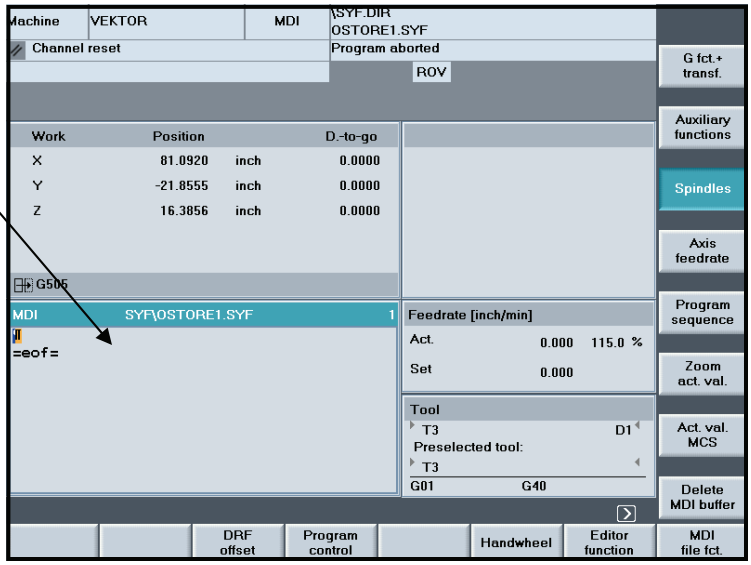


3. Ensure the tool to be loaded in correctly positioned in the tool rack. In this example it is position 5.

4. Press “MDA” button

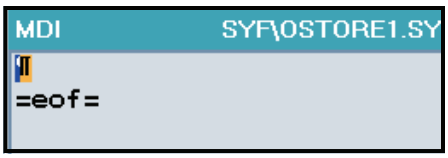


5. Verify this screen appears. The G codes are displayed in this area.



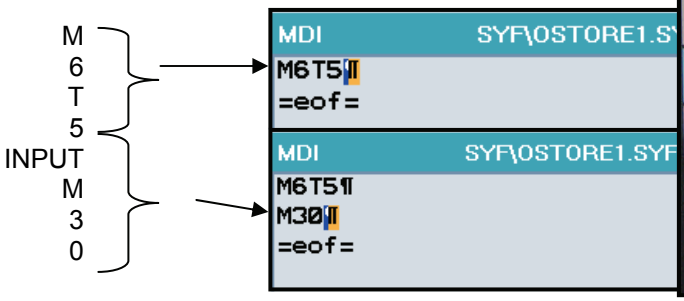
Using G Codes to Change a Tool (cont'd)

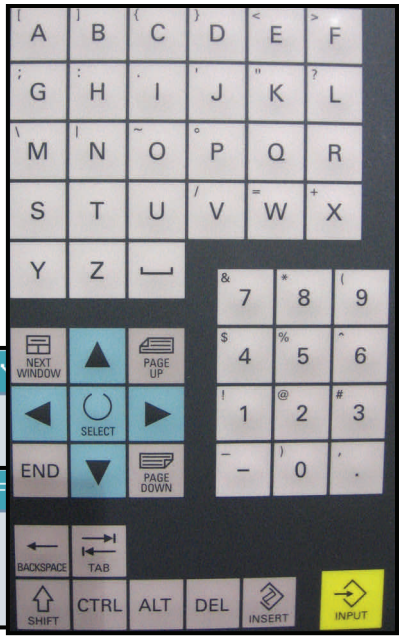
6. Use the DEL or BACKSPACE keys on the alphanumeric pad to remove any code from a previous session. Insure the cursor is at that upper left corner before entering any data. Do not remove/erase the =eof= entry.



7. In this example only two lines of code are entered. They are:
   M6T5
   M30
   M6 is a tool change code and T5 is tool #
   M30 is end of program

8. Use the alphanumeric key pad to enter the two lines of code. Actual keystrokes are:

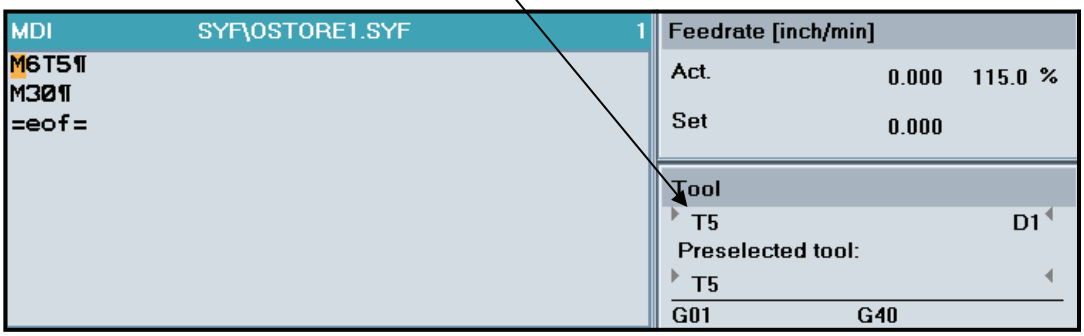




9. Press "Cycle Start"

The VEKTOR now automatically unloads tool T3 and loads tool T5. This takes about 2 minutes.

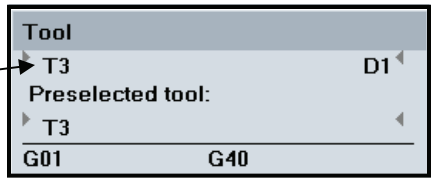
10. Verify the desired tool has been loaded.
11. Press "JOG" to exit MDA mode.



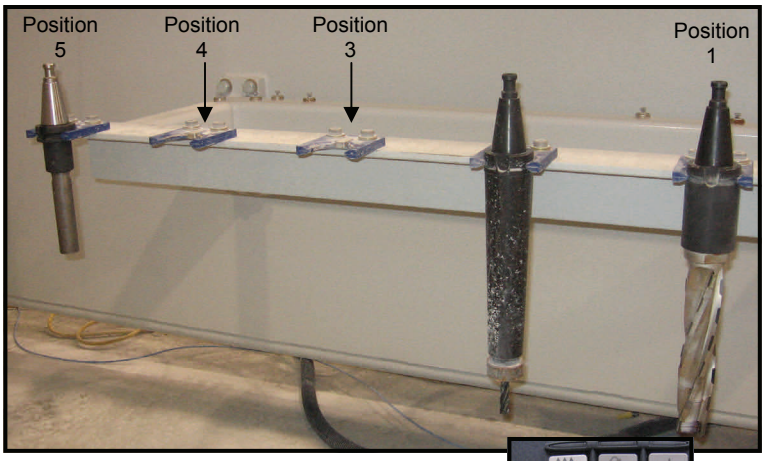
Using G Codes to Change a Tool's Position

A simple G-Code command can be used to change the tool number/rack position of the currently loaded tool. The command changes the "T" number of the currently loaded tool in the software. When the next tool exchange occurs, the loaded tool is placed in a new tool rack position.

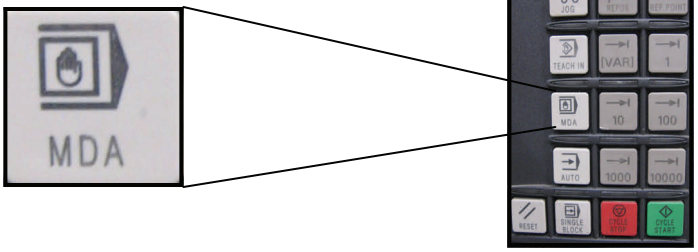
1. The "Tool" display (located in the lower right corner on the screen) lists the tool currently loaded. In this example it is T3.



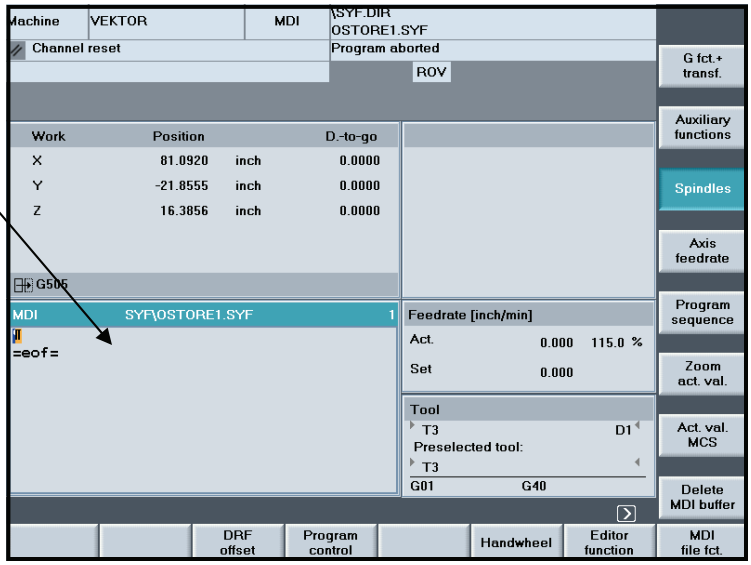
2. Verify that the new tool rack position is open. In this example, currently loaded tool (#3) will be changed to tool #4. (Six position tool racks are found on all newer VEKTOR units)



3. Press "MDA" button

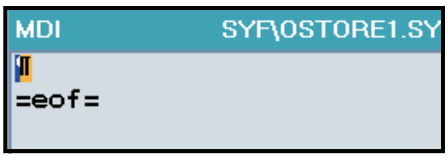


4. Verify this screen appears. The G codes are displayed in this area.



Using G Codes to Change a Tool (cont'd)

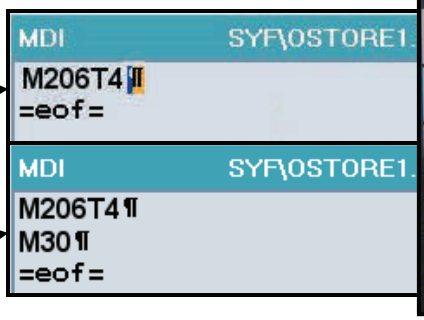
5. Use the DEL or BACKSPACE keys on the alphanumeric pad to remove any code from a previous session. Insure the cursor is at that upper left corner before entering any data. Do not remove/erase the =eof= entry.

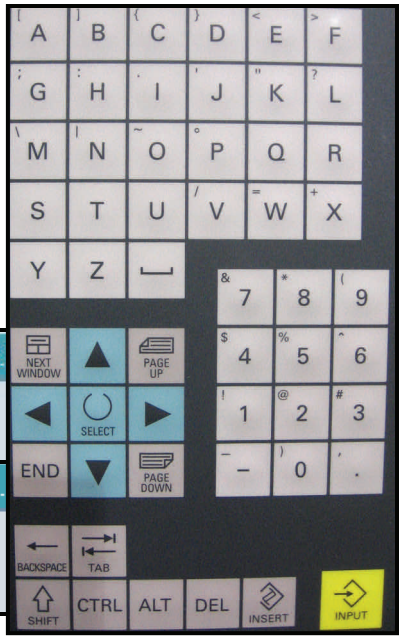


6. In this example only two lines of code are entered. They are:
   M206T4
   M30
   M206 is a change tool number code and T4 is tool #
   M30 is end of program

7. Use the alphanumeric key pad to enter the two lines of code. Actual keystrokes are:

M
2
0
6
T
4
INPUT
M
3
0



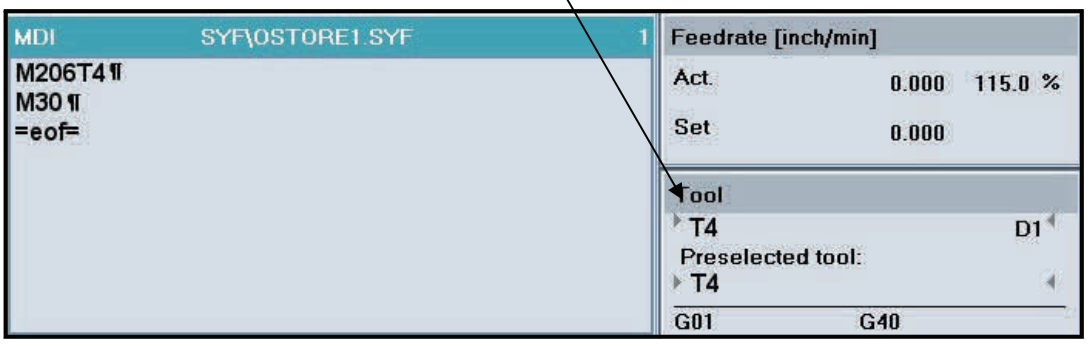


8. Press "Cycle Start"

The VEKTOR <u>DOES NOT</u> unload the current tool. The software has changed the position number of the current tool. On the next tool change the current tool will unload to the new physical position in the tool holder.

9. Verify the tool position changed to the new desired position.

10. Press "JOG" to exit MDA mode.

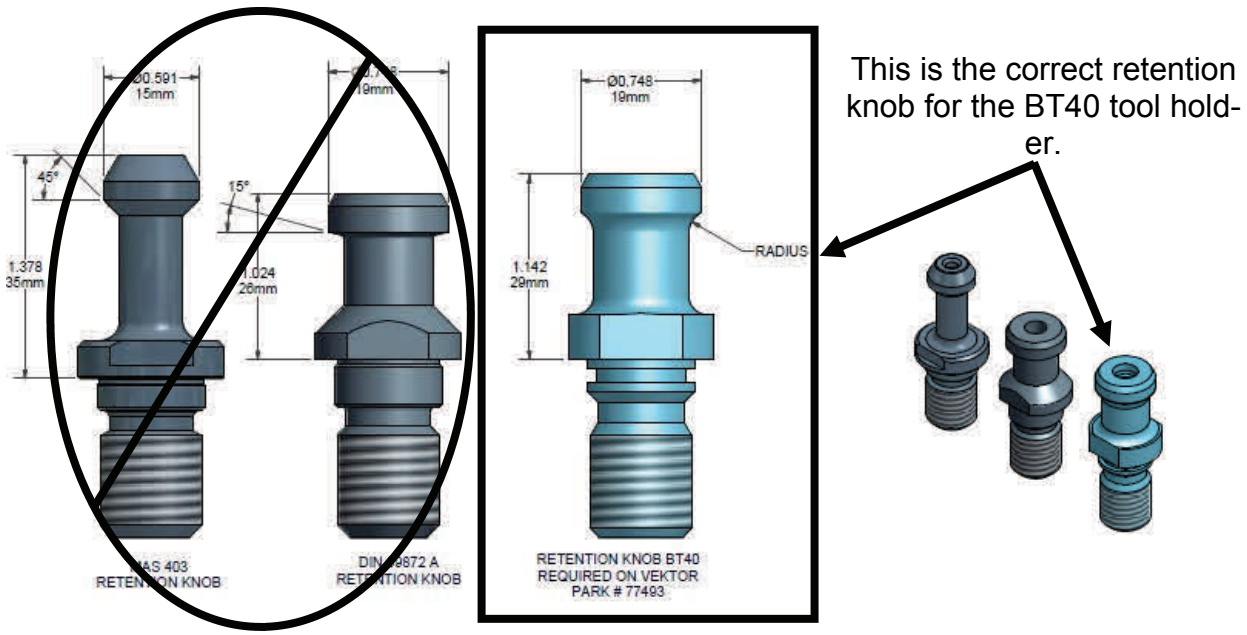


Warning

Your VEKTOR has a high speed spindle. To prolong spindle life, purchase and use only balanced tools and tool holders.

Warning

Damage to the spindle will occur if you don't use Park Industries part # 77943 retention knob with the BT40 tool holder on the VEKTOR.



This is the correct retention knob for the BT40 tool holder.

Spindle Gripper Components and Operation

Below is a brief explanation of the gripper parts and how they function to secure a tool to the spindle. The drawings used are not to scale and do not show all components.

Figure 1 illustrates the gripper assembly's basic parts. The gripper is a tube about one inch in diameter. The bass bushing/O-ring assembly is free to move inside the tube, but is pined in a way that it does not fall out. The spring pushes the bushing and O-ring downward.

Figure 2 illustrates the inside shape of the tapered portion of the spindle. Note the cavity located between the tapered portion and straight upper part. When extended the gripper moves into this cavity and the balls are able to move outward allowing the retention knob to be inserted.

Figure 3 illustrates the gripper position when the spindle has been extended. Notice that the balls can now be pushed outward into the larger cavity to allow the head of retention knob to be inserted. As the retention knob is pushed upward, it will engage the O-ring/bass bushing assembly which is currently being pushed downward by the spring.

Figure 4 illustrates the position of a properly inserted retention knob. When manually loading a tool, it is very important that the operator pushes the tool upward until resistance is felt and then continue to push upward until the tool bottoms out before releasing the manual tool change button. By pushing until a bottom is reached (spring compressed), the balls are allowed to grip the neck portion of the knob when the gripper is retracted. If the retention knob is NOT fully seated, the balls can be damaged during retraction or the tool may fly out during operation.

Figure 5 illustrates the gripper and retention knob after the spindle has been retracted. During retraction, the straight walls of the spindle body force the ball inward gripping the neck of the retention tool. The spring (now compressed) is pushing the brass collar and O-ring against the top of the retention tool ensuring a water tight connection.

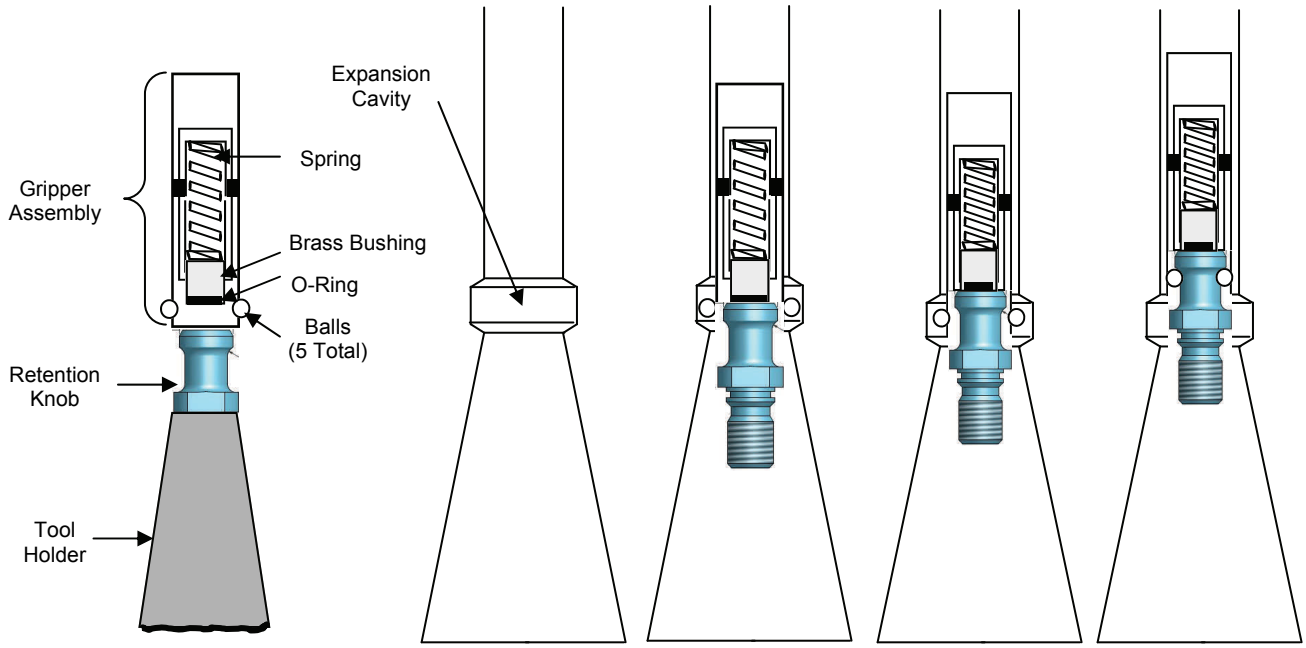


Figure 1 Figure 2 Figure 3 Figure 4 Figure 5

Manual Tool Change

NOTE ***It is the customer's responsibility to make sure all machine operators are trained and well-versed in both the operation of the machine as well as all safety processes and procedures.***

Caution **Failure to properly load a tool may result in damage to the machine, personal injury or death.**

The steps to manually change a tool are not difficult, **but extreme spindle damage** will occur if the operator does not fully seat the tool into the spindle.

To prevent spindle damage, the tool must be completely seated in the spindle before the spindle release button is released. Before loading a tool the first time carefully read the below information.

To Unload a Tool:

1. press & hold the spindle release button,
2. remove tool from spindle
3. then release button.

To Load a Tool:

1. press & hold the spindle release button,
2. insert tool into spindle*
3. then release button.

* As the tool is inserted into the spindle, resistance is encountered when the tool is about ¼ inch from the drive dogs. See Figure 1. **Push through the resistance** ensuring the spindle's drive dogs align with the tool's drive-dog slots and the gap between the bottom of the spindle and the top of the tool is about ¼ inch before releasing the spindle release button. See Figure 2.

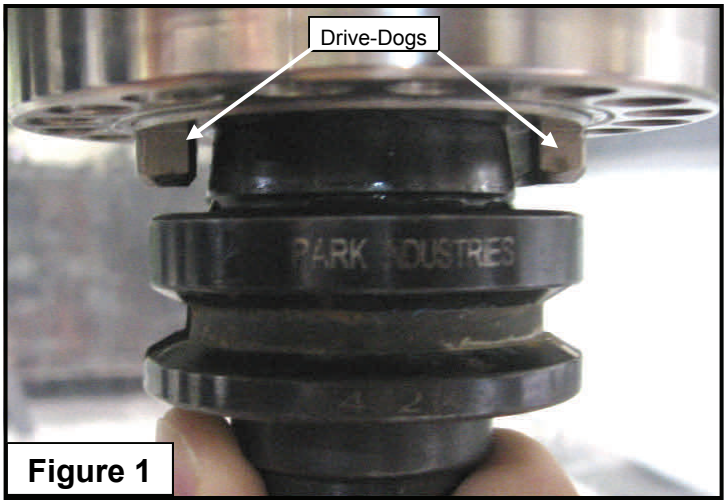


Figure 1

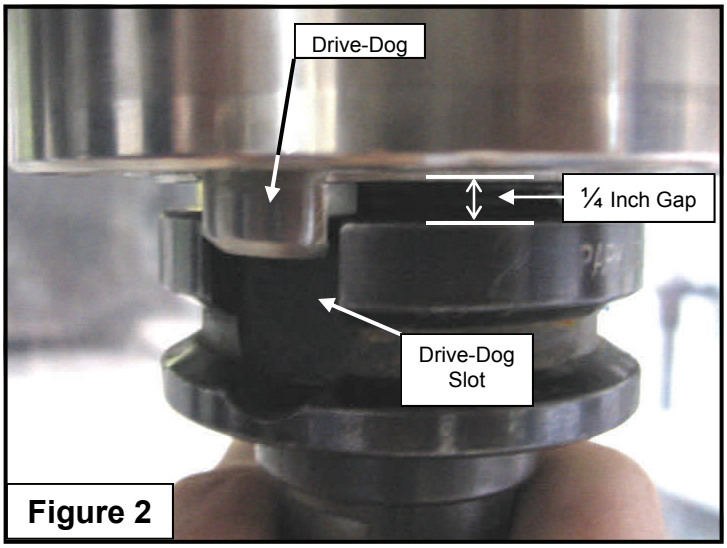


Figure 2

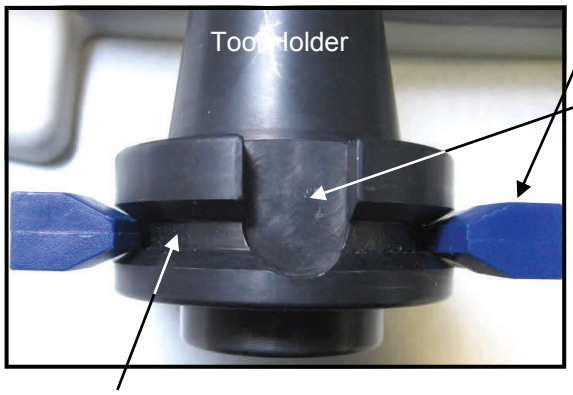
Tool Rack Setup

This procedure determines and sets the X, Y and Z coordinates and spindle offset for each tool rack assembly tool position. There are three parts to the setup:

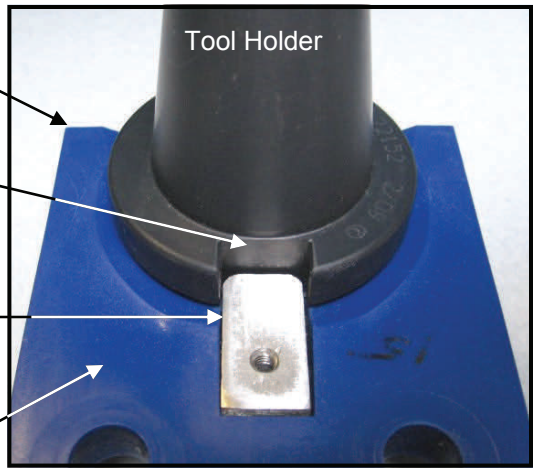
- X & Y Position
- Z Position
- Spindle Orientation

The below components are referenced in this procedure. Use the below photos to identify these items.

- Gripper Plate
- Gripper Plate Alignment Pin
- Gripper Plate Fingers
- Tool Holder
- Tool Holder Alignment Notch



Tool Holder Alignment Grove



The following items are needed to complete the setup steps:

- Dial Indicator
- Pointed Alignment Tool
- Tool Rack Jig
- Tool Length Setup Tool

Tool Rack Jig

Dial Indicator

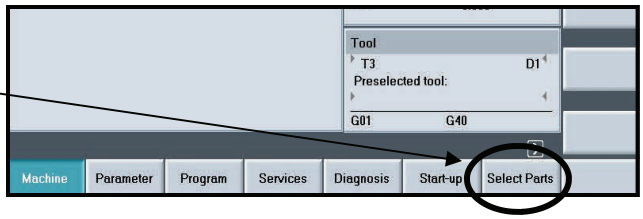
Pointed Alignment Tool

Tool Rack Setup (cont'd)

Accessing the Setup ToolRack Screen

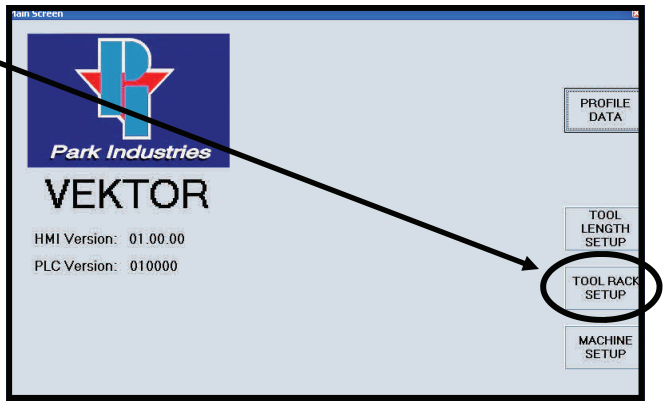
1. Press the "Menu Select" soft key on the Operator Panel.

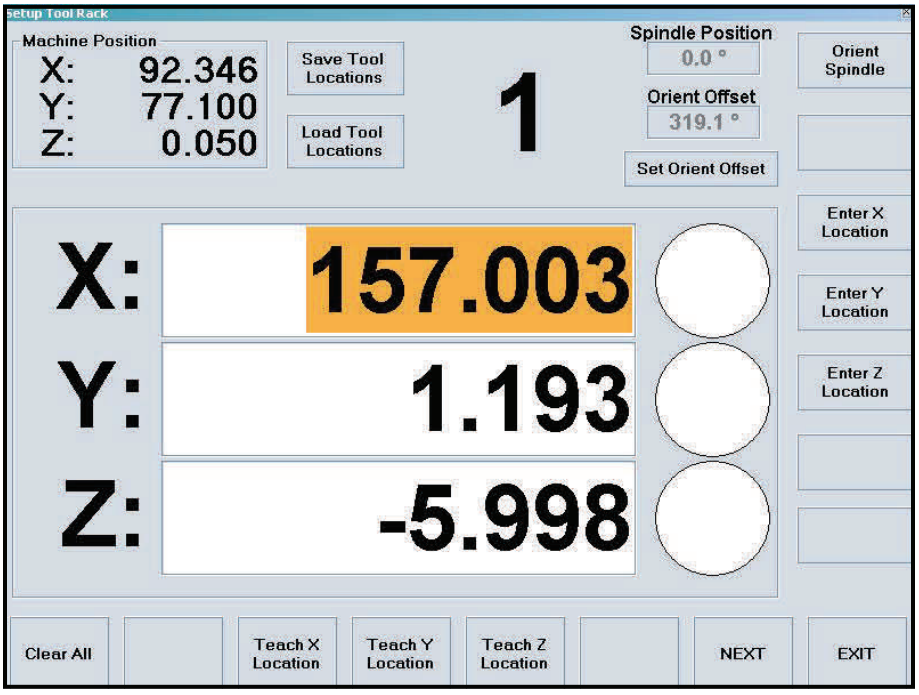
2. Press the "Select Parts" soft key on the Machine Screen.



3. Press the "Tool Rack Setup" soft key

4. Verify the below Setup Tool-Rack screen is displayed. See next page for more information about this screen.





Tool Rack Setup (Cont'd)

How to use this screen - This screen is use to store the X, Y, and Z positions and spindle orientation of each tool holder position. In other words there are six (6) screens, one for each tool holder position. The large number displayed at the top indicates which tool holder position's data is displayed.

Once a physical position (X, Y or Z) has been established during the setup procedure, there are three ways to enter the data.

Pendant:

- turn the axis switch to five (5)
- **DO NOT** hold down the activation switch
- press the F1 button for the X axis, F2 for Y axis, F3 for Z axis (status indicator turn green)
- turn the axis switch to zero (0)

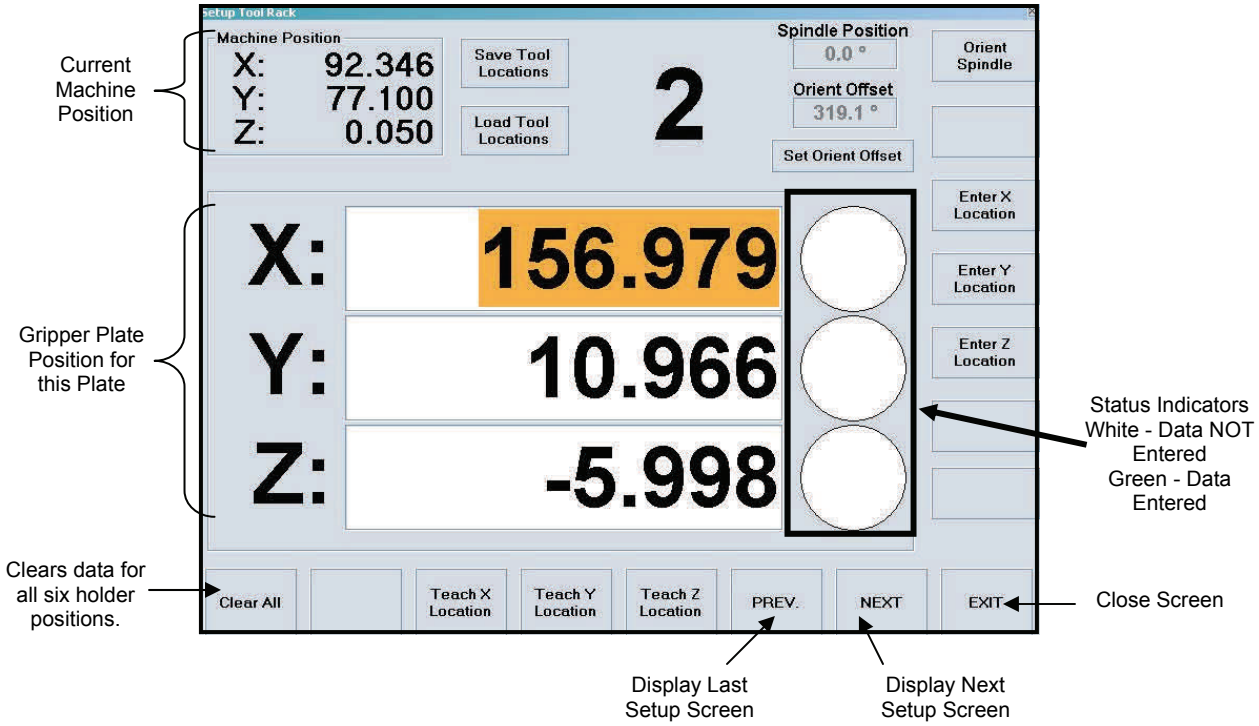
Screen:

- press a Teach Location button for the axis being entered (status indicator turn green)

Key Pad:

- use the "TAB" key to select the coordinate entry box
- use the numeric key pad to enter the value
- press the appropriate "Enter Location" button (status indicator turns green)

Save/Load Locations buttons writes/reads the gripper plate data to/from a location on the system's hard drive.

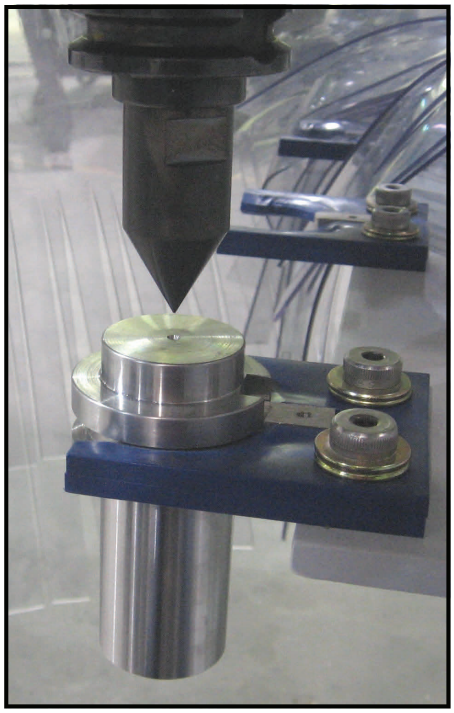


Tool Rack Setup (Cont'd)

X & Y Position Setup

5. Load the pointed alignment tool into the spindle.

6. At tool rack position to be setup, insert the tool rack jig into the gripper plate. Ensure the jip is fully seated. Always start at position one.

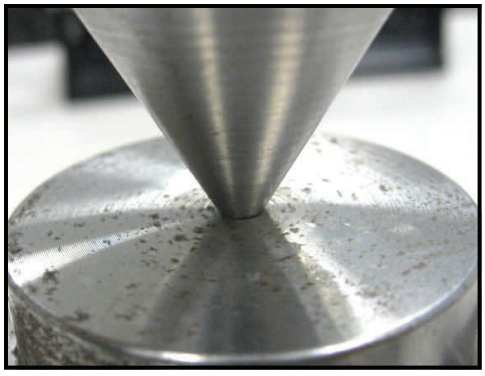


7 **Coarse Movement** - Using the pendent (F2 or F3 speed), position the tool over the jig. Stop Z down movement when the tool is about ½ inch above the jig.

8. **Fine Movement** - Switch to F1 speed on the pendent. Slowly move the tool on the X, Y and Z axis until the pointed tool is perfectly center in the jig's hole.

9. Verify Tool Setup Screen's number (upper middle) matches the tool holder position being setup.
If not, use the "NEXT" or "PREV." soft key to select the correct Tool Setup Screen.

10. Using either the Pendent buttons or the "Teach Location" soft keys enter the X and Y positions. The status indicator circle changes from white to green to indicate the values have been read.



11. Repeat steps 6 through 10 until all six positions have been setup.

NOTE: At the sixth position, DO NOT change the Y axis position.

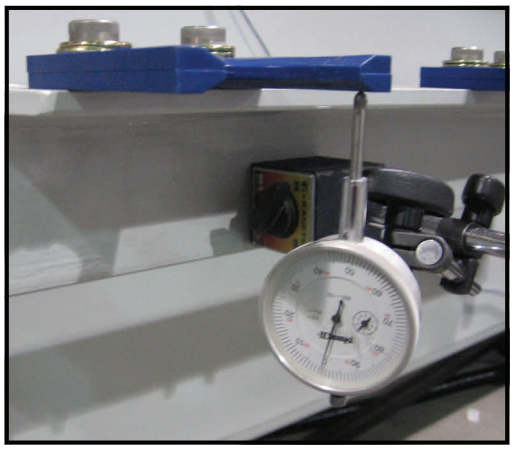
Tool Rack Setup (Cont'd)

Z Position Setup

12. Remove the pointed tool.

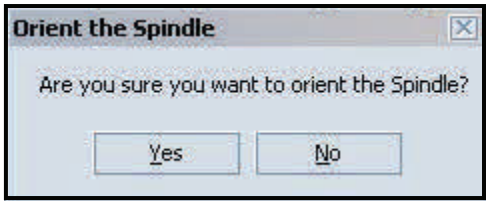
13. Remove the tool rack jig from position six (6).

14. Load the Park Industries tool length setup tool.



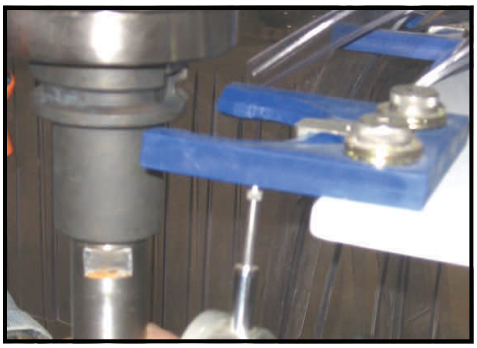
15. Attach a dial indicator to the tool rack frame.
    Position the probe as shown in the photo and zero the indicator.

16. On the Setup Tool Rack screen press
    Press yes to orient the spindle
    Verify the spindle rotates.
    When motions stops go to the next step.



Tool Rack Setup (Cont'd)

Z Position Setup (cont'd)

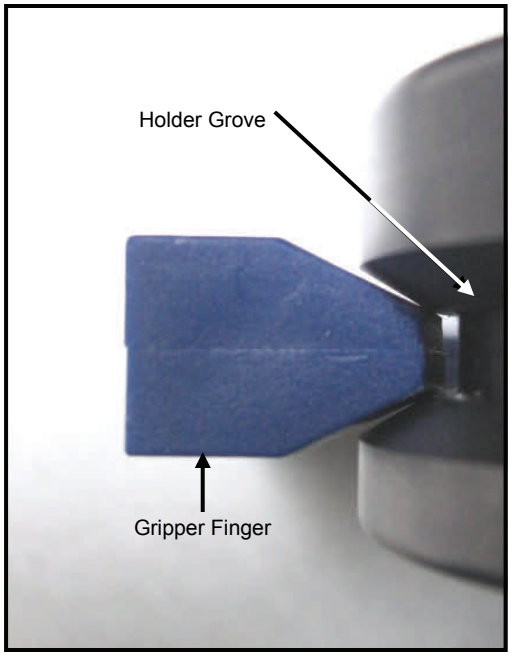


NOTE: DO NOT move on the Y axis during any part of this setup procedure.

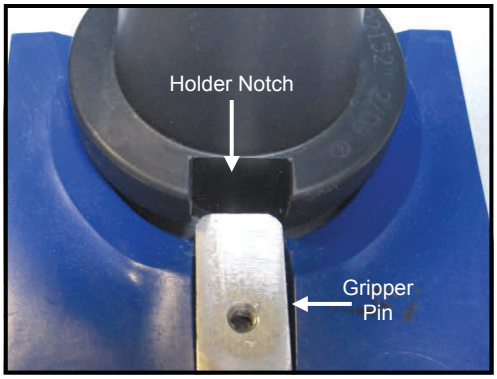
17. **Coarse Movement** - Using the pendent (F2 or F3 speed), move the tool on the X and Z axis until the edge of the tool holder is about ½ inch from the gripper plate and the tool holder groves closely align with the gripper plate fingers.

In the next step, the tool holder is moved into the gripper plate. During this movement watch for two key alignments:

- ensure the tool holder groves align with the gripper plates fingers (+ or - Z axis)
- tool holder alignment notch aligns with the gripper plate's alignment pin (rotational)



18. **Fine Movement** - Switch to F1 speed on the pendent. Below is a sequence of steps used by equipment installers. Read all five steps before starting.
    A. Slowly move in the plus X direction.
    B. Stop when the dial indicator detects (moves off of zero) gripper tool deflection.
    C. Slowly move on the Z axis until the dial indicator reads zero again.
    D. Check the alignment of the holder's notch and gripper's pin.
    E. Repeat until the current X axis Machine Position equals the X position of the Tool Sensor Position set in the last section.



19. Verify Tool Setup Screen number 6 (six) is currently displayed. If not, use the "NEXT" or "PREV." soft key select the number 6 (six) Tool Setup screen.

20. Use either the screen or pendent to load the current Z position. Status circle changes from white to green to indicate the value has been read.

NOTE: The Z position has been entered on all six tool holder positions.

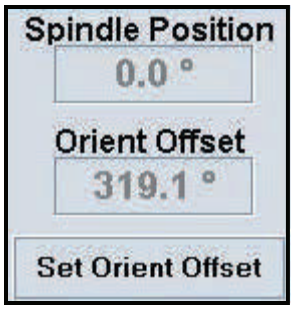
NOTE: *DO NOT* change the current X, Y, or Z position of the tool.

Tool Rack Setup (Cont'd)

Spindle Orientation

This part of the procedure determines an offset value which is added to the spindle's orientation point to ensure a tool holder's notch always aligns with gripper plate's pin during a tool change operation.

The "Spindle Position" readout displays the current spindle position. The readout is used to find the tool holder's center point. As the spindle is rotated by hand, the degree number displayed does change.



21. Grasp the tool and rotate clockwise.

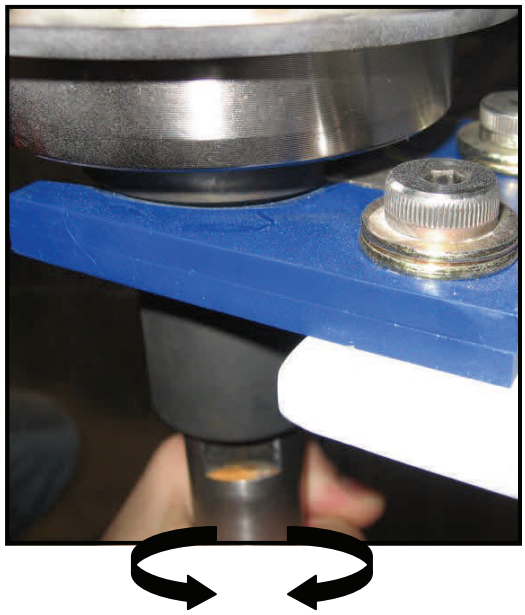
22. Record the "Spindle Position" value. _____

23. Grasp the tool and rotate counter clockwise.

24. Record the "Spindle Position" value._____

25. Add the two positions together and then divide by two (2). _____

26. Rotate the tool/spindle until the 'Spindle Position" readout equals the number from the last step within XXX amount.



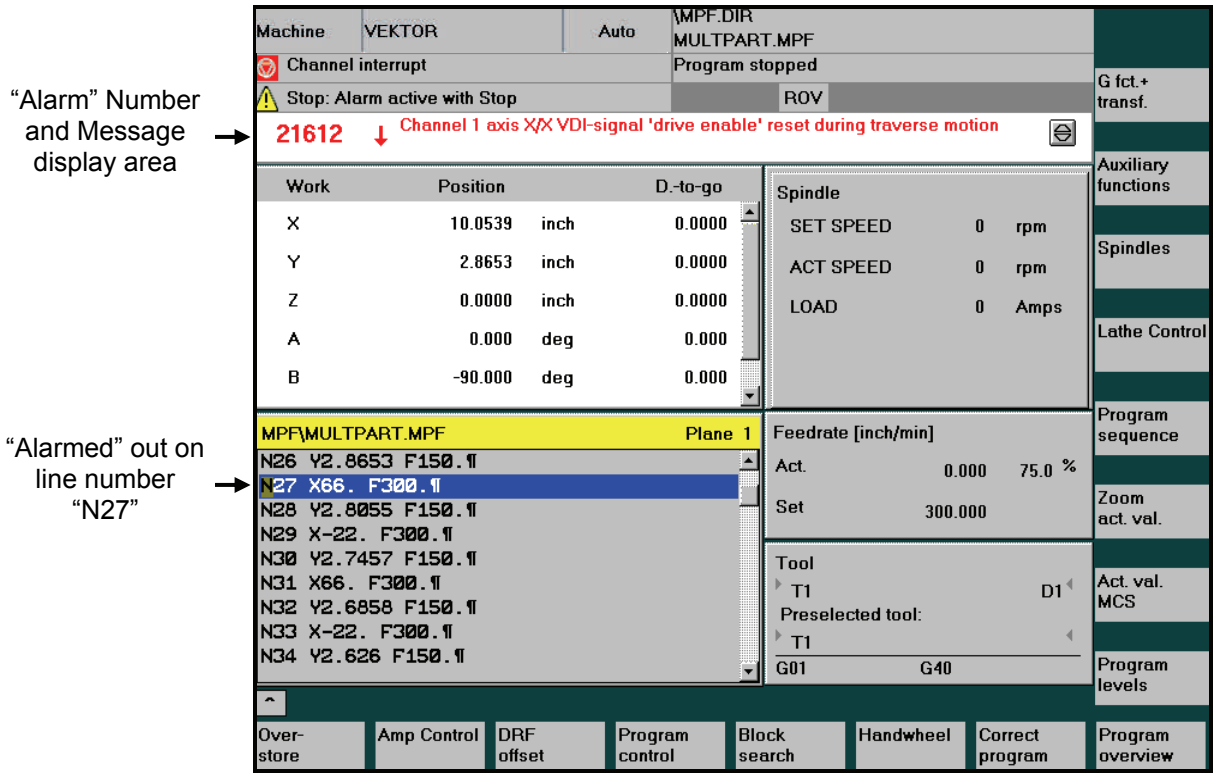
27. Use the Operator Panel's mouse to position the cursor over the "Set Orient Offset" button and then press the left button click.

NOTE: The Orientation Offset has been entered on all six tool holder positions.

28. Press the "Exit" soft key.

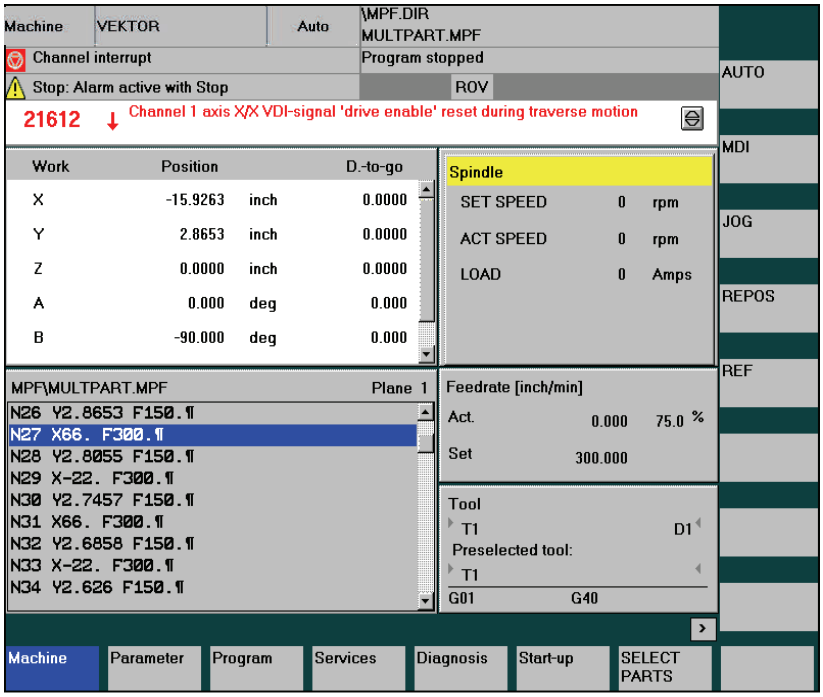
Verifying Alarms

There are many things that could go wrong when running a CNC machine. The VEKTOR uses an elaborate Siemens control that controls many axes at the same time and also watches over simple items such as the water flow switch and emergency stops in the plc. At any point of time, something could happen and an alarm message is displayed on the top half of the screen as shown below. The alarms will stay displayed untill the cause is resolved. The picture also displays the line number that the program stop on, which was line “N27”. With this line number the program can be started close to the area that it had faulted out at (refer to the ”Restart in Cut” section to start the program over without loosing much time). After an alarm has occurred ***“DO NOT PRESS RESET”***, even though alarm is saved in the “Alarm Log”. It is easiest to look at the “Alarm” page which shows current alarms. Once the “Reset” button is pressed the machine clears the existing alarm and also resets the program to line N1.

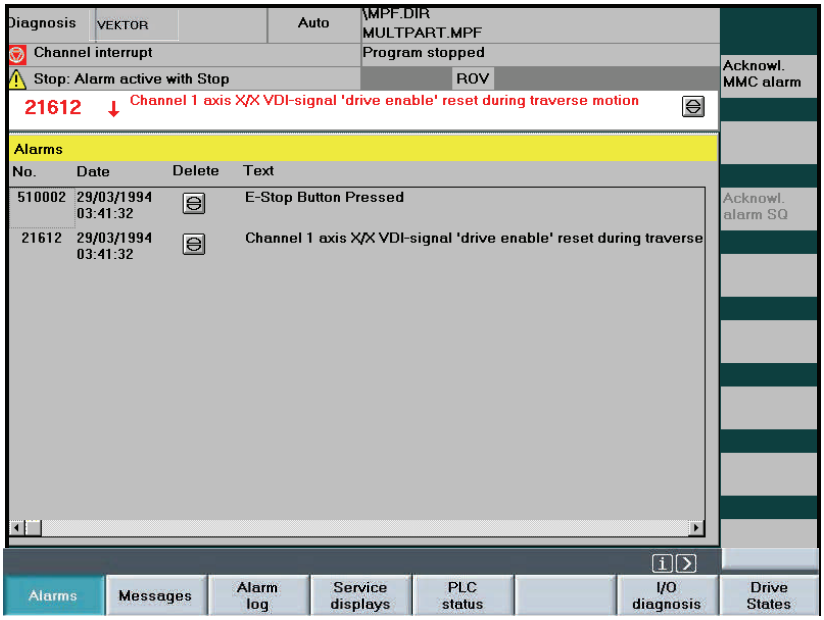


Verifying Alarms (Cont'd)

The Siemens control is quite unique, because if something happens to the machine an alarm maybe displayed but it may not be the cause of the alarm. To look at the "Alarm" page, cause press the "Menu Select" soft key to display the main page and then press the "Diagnosis" soft key at the bottom of the screen.

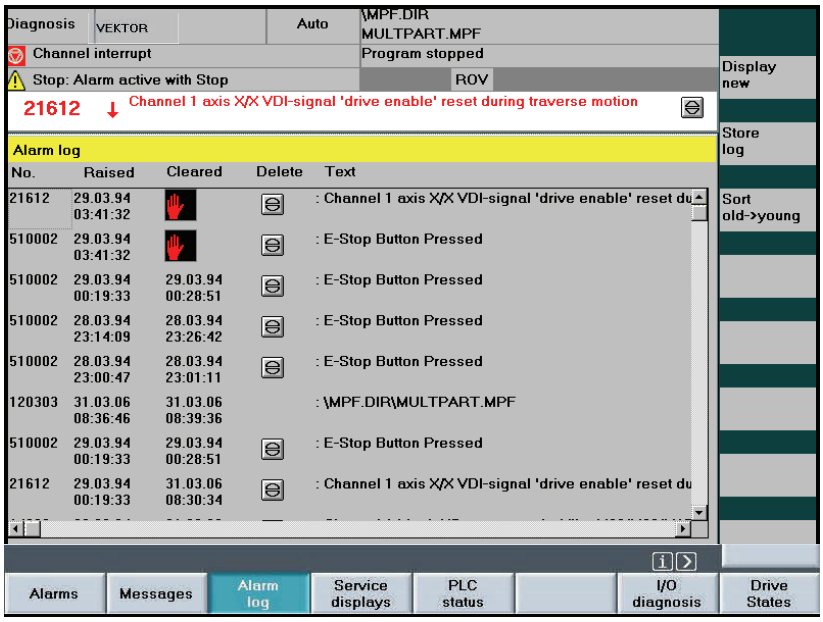


In the "Diagnosis" screen press "Alarms". The first alarm that is displayed at the top of the page is the root cause of the rest of the alarms. So because of the 510002 Alarm (E-Stop Pressed) this has caused an alarm of 21612 which is a signal alarm on the X axis which was trying to move.

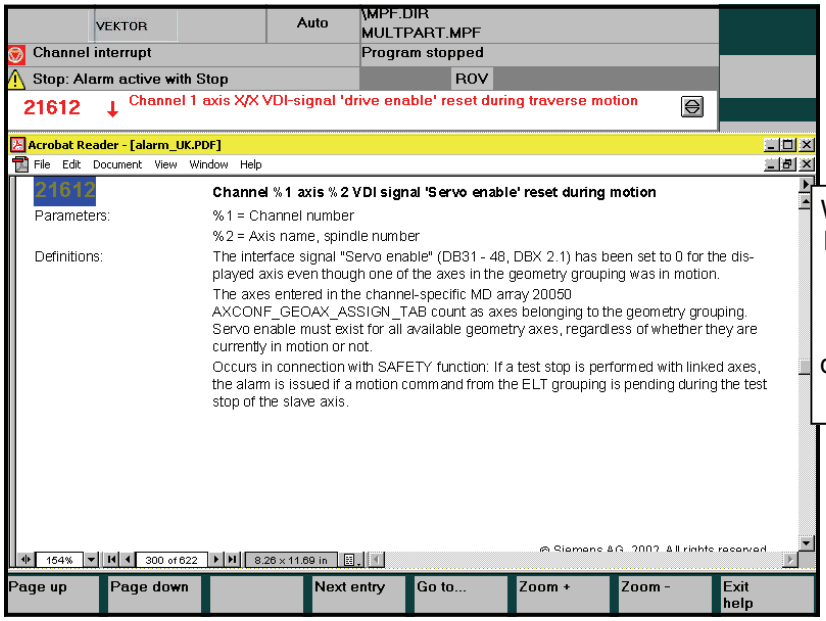


Verifying Alarms (Cont'd)

To see all of the alarms that have occurred press the "Alarm Log" soft key. Once pressed , a list of all of the alarms is displayed, but it will show the very last alarm thru the first that is listed in the log. You may change the list around by using the "Sort" soft key. It is easiest to see the last alarm that has occurred, first on the page for troubleshooting purposes.



To get an explanation for a Siemens alarms while in either the "Alarms" or the "Alarm Log" screen use the blue arrow keys. Use the keys to move up or down to choose the alarm number and then press the "i -HELP" soft key on the "Operator Panel" for information dealing with this alarm. Once finished with "HELP" press the "Exit Help" soft key at the bottom of the page.



When the "HELP" button is pressed for a specific alarm, a SIEMENS description will be displayed

Park Industries Generated Alarms

These are the Park Industries Inc. generated alarms for the VEKTOR. This list does not include CNC alarms generated by Siemens control.

510000 “ALARM : TOOL CHANGE - SPINDLE DID NOT ORIENT"
510002 “ALARM : TOOL CHANGE ALARM - DRAW BAR DID NOT UNCLAMP"
510003 “ALARM : TOOL CHANGE ALARM - SPINDLE DID NOT RELEASE THE TOOL"
510004 “ALARM : TOOL CHANGE ALARM - SPINDLE DID NOT CLAMP ON A TOOL"
510005 “ALARM : SPINDLE FRESH WATER FLOW ALARM"
510006 “ALARM : SPINDLE RECYCLED WATER FLOW ALARM"
510007 “ALARM : BLADE WATER FLOW ALARM"
510008 “ALARM : E-STOP: CONSOLE PANEL"
510009 “ALARM : E-STOP: PENDANT"
510010 “ALARM : SPINDLE DRIVE FAULT - LOOK AT THE DISPLAY ON THE DRIVE"
510013 “ALARM : SPINDLE FAULT - NO TOOL DETECTED"
510014 “ALARM : SPINDLE DRIVE ENABLE MISSING"
510015 “ALARM : E-STOP: BRIDGE PULL CORD"
510300 “ALARM : X AXIS IS NOT REFERENCED"
510301 “ALARM : Y TOWARD AXIS IS NOT REFERENCED"
510302 “ALARM : Y AWAY AXIS IS NOT REFERENCED"
510304 “ALARM : SPINDLE Z AXIS IS NOT REFERENCED"
700000 “PLC FAULT: OB82 I/O POINT FAULT"
700001 “PLC STOPPED: OB86 RACK FAULT"
700031 “WARNING: AUTO GREASER IS LOW, PLEASE REFILL"
700032 “AIR CONDITIONER FAULT"
700034 “LOW AIR PRESSURE WARNING"
700035 “MOTION DISABLED DUE TO SPINDLE DISABLE SWITCH"
700036 “AIR FILTER FLOW ALARM : MANUAL SPINDLE-STARTS AND PROCESS CYCLE-STARTS ARE DISABLED"
700037 “WARNING : AIR FILTER RESTRICTION - LESS THAN 1 HOUR OF OPERATION RE-MAINING!"
700038 “WARNING : AIR FILTER RESTRICTION - LESS THAN 30 MINUTES OF OPERATION REMAINING!"
700039 “WARNING : AIR FILTER RESTRICTION - LESS THAN 10 MINUTES OF OPERATION REMAINING!"
700042 “INTERNAL SPINDLE PROX. MALFUNCTION DETECTED, USE INPUT DIAGNOSTICS"
700050 “MDA/AUTO MODE REQUEST DENIED. SWITCH PENDANT SELECTOR SWITCH TO 0"
700053 “MACHINE IS IN FEEDHOLD - CHECK SPEED POT AND FEED START PB"
700054 “MACHINE IS NOT IN JOG MODE"
700055 “SPINDLE STOPPED - CHECK PUSHBUTTON ON CONTROL PANEL"
700056 “SPINDLE WARMUP REQUEST DENIED - SWITCH PENDANT SELECTOR SWITCH TO 0"
700057 “SPINDLE WARMUP NOT COMPLETED - SPINDLE WILL ONLY GO 500RPM!!!"

Input & Output Screens

The Inputs & Outputs screen is a diagnostic tool used by the Park Service personal to troubleshoot potential problems within the machine. This screen is accessed by pressing the “I/O diagnosis” button on the Diagnostic screen.

The status area contains three columns. The left column lists the system variable. The center column indicates the variables current status zero (0) for OFF state, one (1) for ON state. The third column is the number assigned to the variable.

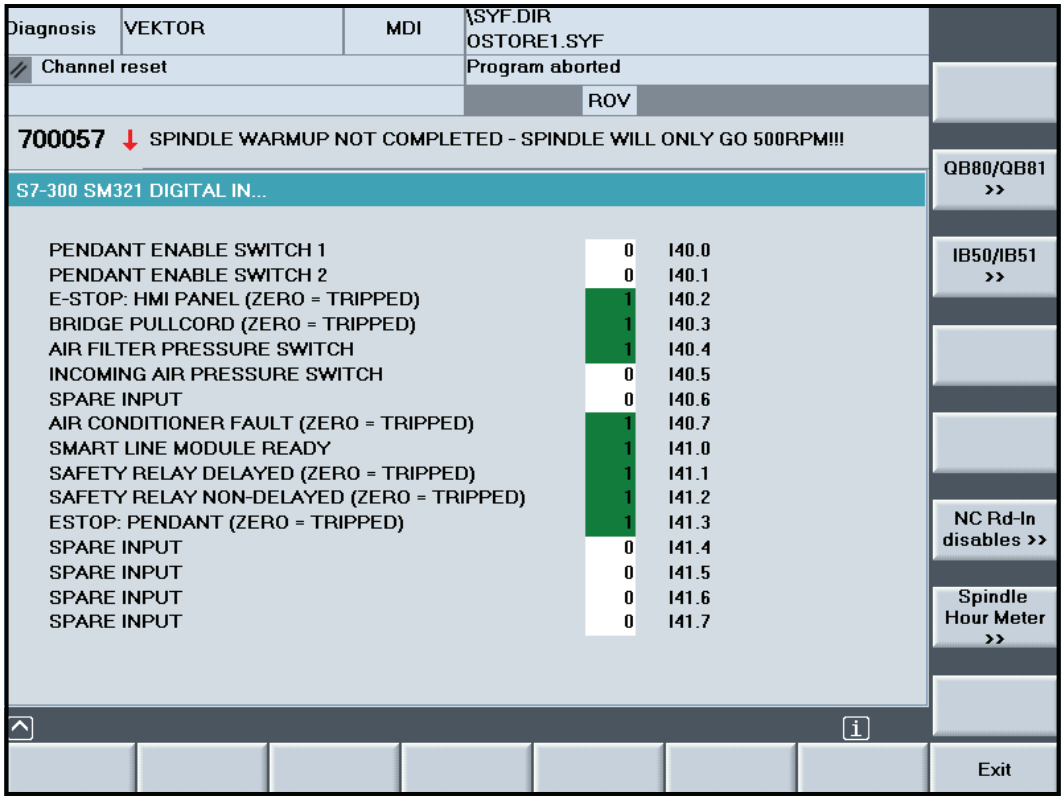
Variable	Status	Number
PENDANT ENABLE SWITCH 1	0	I40.0
PENDANT ENABLE SWITCH 2	0	I40.1
E-STOP: HMI PANEL (ZERO = TRIPPED)	1	I40.2

In most cases, the values on this screen are used only by a Park Industries Technician. If there are any questions or concerns, please contact Customer Service at 1-800-785-3391.

There are two input pages and one output page. The first screen displayed lists the status for IB40 and IB41 (inputs) .The select buttons (upper right side of the screen) determine which system variables are displayed. IB50/IB51 and QB80/QB81 variables are shown on the next page.

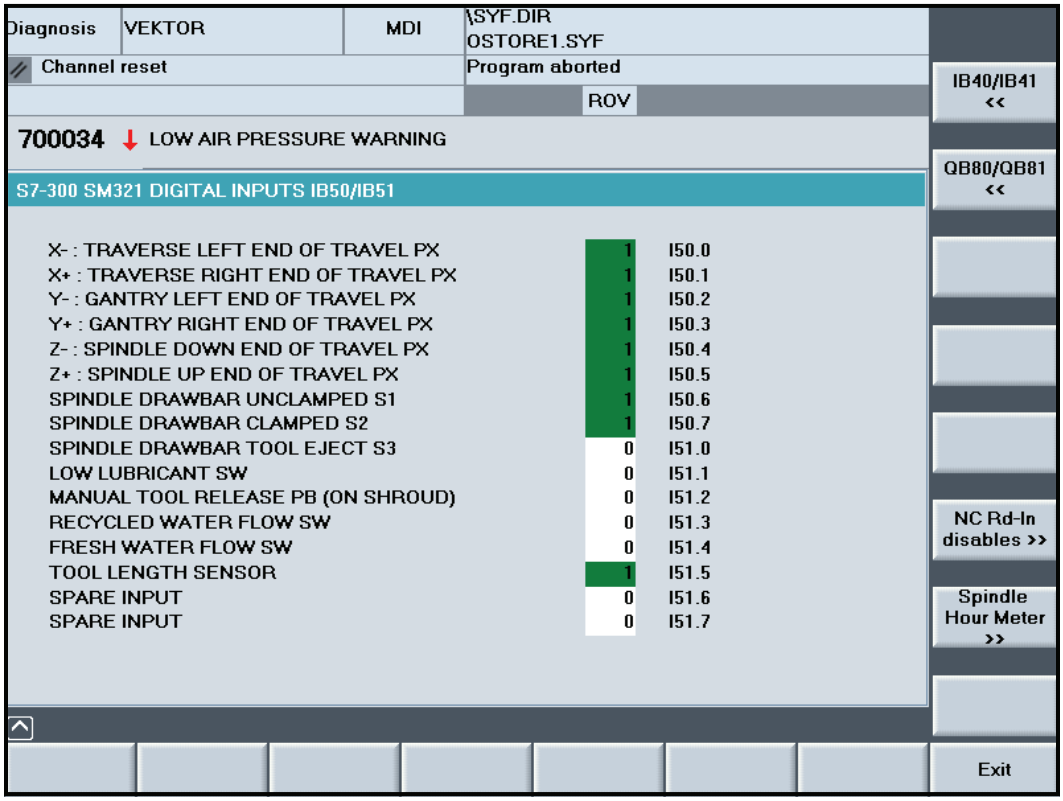
Press the :Exit” key located in the lower left corner to return to the Diagnostic screen.

IB40/IB41 Screen

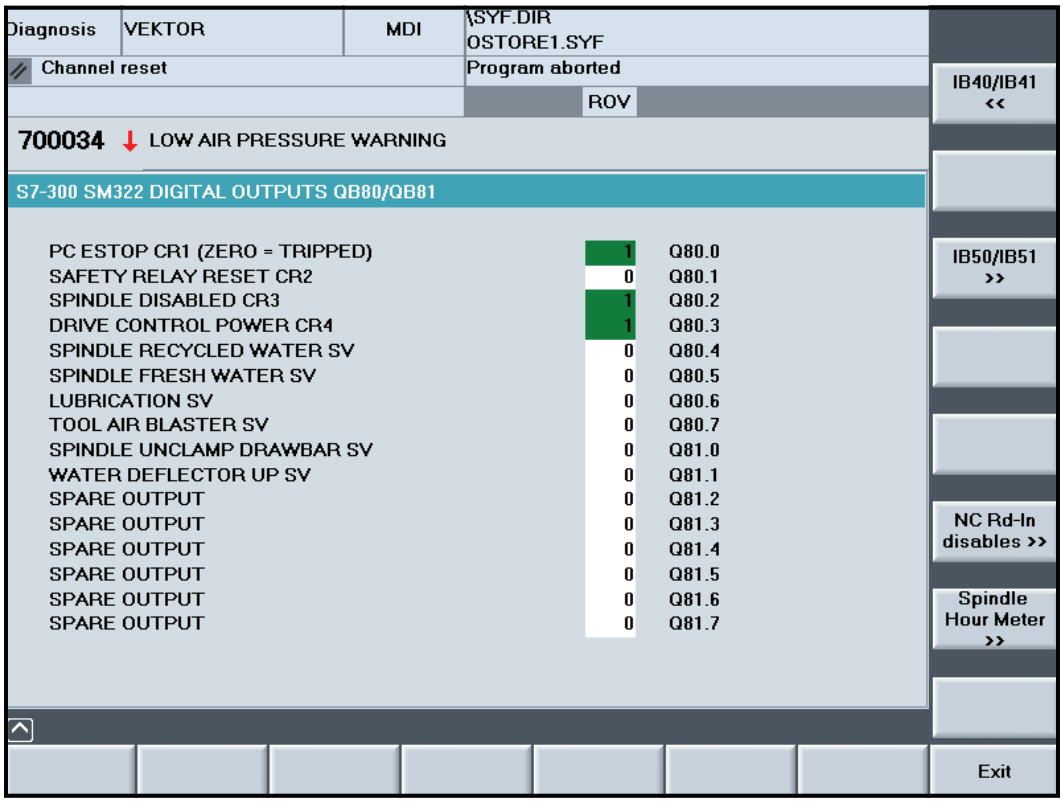


Input & Output Screens

IB50/IB51 Screen



QB80/QB81 Screen



This page intentionally left blank.

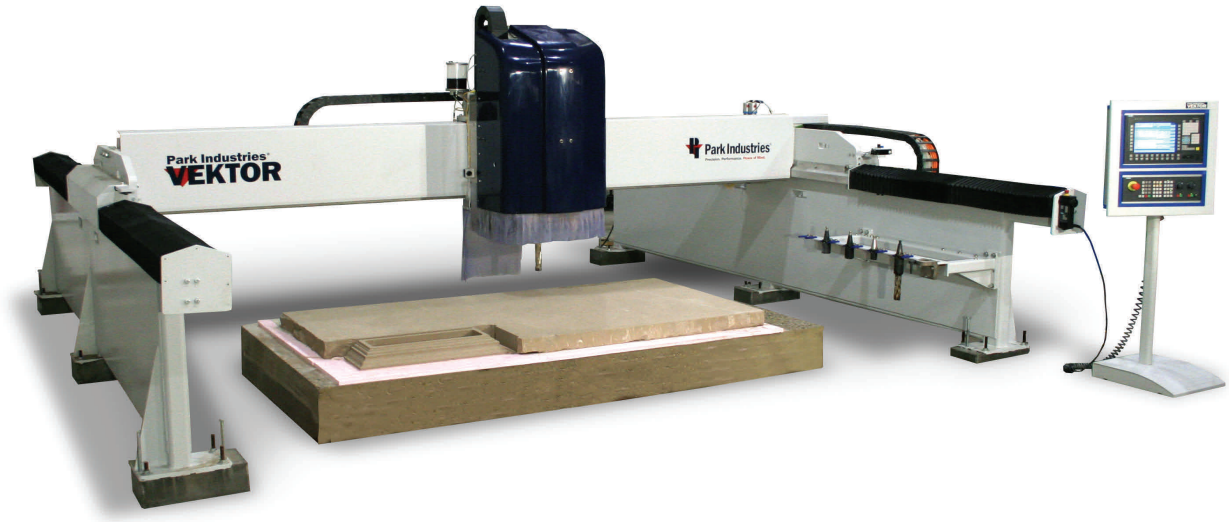
Section 5: Maintenance

Contents Page

General 5-1
Lubrication Warnings 5-4
Spindle and Retention Know Warning 5-5
Routine Lubrication Schedule 5-6
Maintenance Tasks 5-7
Maintenance Procedures 5-31
Maintenance Log 5-41
Maintenance Log Masters 5-46

General

This section presents preventative maintenance procedures for the VEKTOR CNC Stone Profiler. The corrective maintenance items documented in this section are those that can be performed by an individual with average mechanical abilities. If for any reason the person working on the machine is not comfortable with the procedures, please contact Park Industries Customer Service at 1-800-785-3391. The preventative maintenance items listed in this section must be performed to keep the machine performing at peak efficiency, with a minimal amount of repairs and downtime.



NOTE

It is the customer's responsibility to make sure all machine operators are trained and well-versed in both the operation of the machine as well as all safety processes and procedures.

Caution

<u>WARNING</u>: If you have not read the Operation and Maintenance Manual and feel you were not properly trained DO NOT ATTEMPT TO OPERATE THIS MACHINE. Serious injury or death may result.

READ FIRST!

SAFETY INFORMATION

Warning: If you have not read the Operation and Maintenance Manual and feel you were not properly trained DO NOT ATTEMPT TO OPERATE THIS MACHINE.

Before Performing Maintenance or Repairs on This Machine, Verify:

1. All workers have proper eye, ear and foot protection.
2. Follow Machine Shutdown Procedures.
3. Ensure Lock-Out / Tag-Out has been performed on the machine (Be sure to follow the Lock-Out / Tag-Out procedures established by your company).
4. All water supply valves to the machine must be off.
5. All air supply valves to the machine must be closed.
6. No hazardous voltages are present inside the electrical cabinet. Before opening the cabinet, make sure that the power is off at the Main AC disconnect and proper lock- out/tag-out procedures followed. *Only a qualified technician, electrician or electrical maintenance person familiar with the hazards involved should be working inside this cabinet when under power.*
7. You are trained in both the operation and maintenance of the machine.
8. The work area around the machine is clear of all foreign objects.

Before Putting the Machine Back Into Operation, Verify:

1. All workers have proper eye, ear and foot protection.
2. Safety covers and guards are properly installed.
3. Electrical enclosure covers are closed and sealed for water tight operation.
4. The operator is trained and well-versed in both the operation of the machine as well as all safety processes and procedures.
5. Work area around the machine is clear of foreign objects.
6. Everything appears normal and there are no foreign wires, oil leaks, or loose fittings.
7. Follow Machine start-up procedures.

Diagnostic Procedures:

Some diagnostic procedures may require that the machine be powered up. In these cases, it is critical to closely follow the instructions in your Operators Manual, or those given by an authorized Park Industries Service Technician. Failure to do so could result in injury to yourself or damage to your equipment.

If you have any questions about any procedures or process steps, please contact a customer service representative at Park Industries before you begin work on the machine.
Customer service (Toll-Free, 24/7): 1-800-785-3391.

Recommended Lubrication

Greases, Oils, Rack Lube

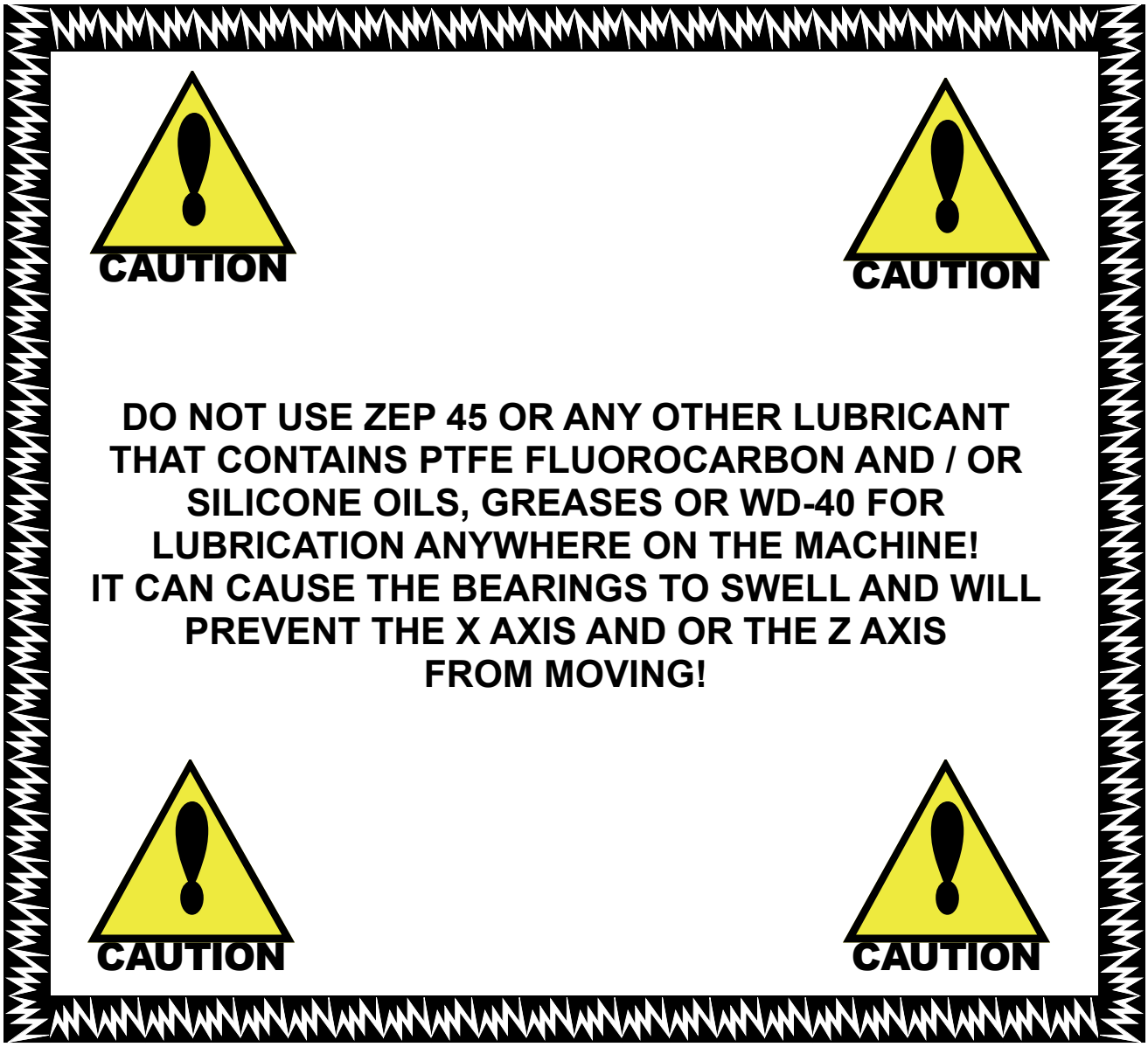
Below is a few products that are used within the machines that we manufacture. Not all products shown on this page will be used in / on your machine. Refer to the Routine Lubrication Schedule for the correct types to use. Also below are the listed Park Industries part numbers for each of the products.

Mobil Vactra #2
Way Oil
Part # 3403002-4

MetaFlux
Gleitmentall
Part # 3400037

Mobil Mobilgrease XHP222 Grease Part # 3400008

Kluber Microlube® GB0 Grease Part # 3400050



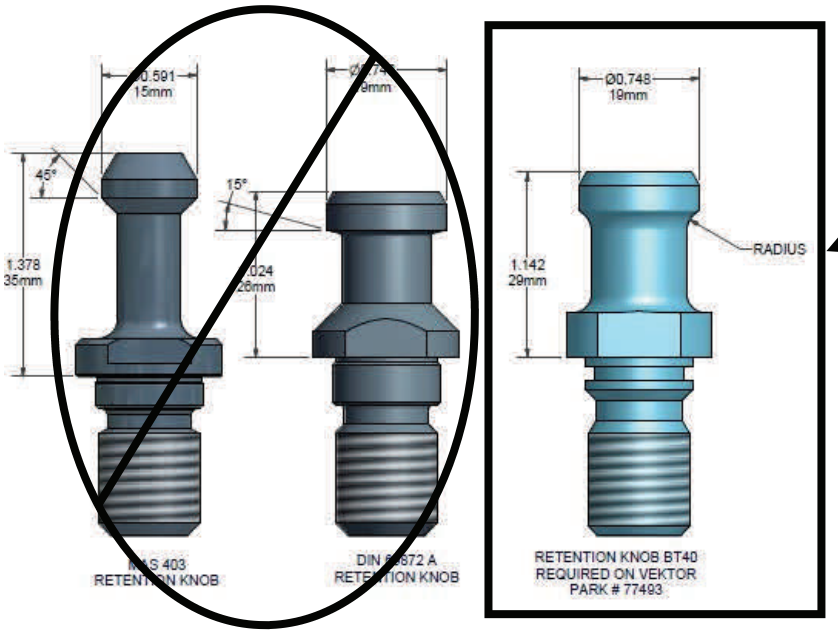
WARNING: If you have not read the Operation and Maintenance Manual and feel you were not properly trained DO NOT ATTEMPT TO OPERATE THIS MACHINE. Serious injury or death may result.

Warning

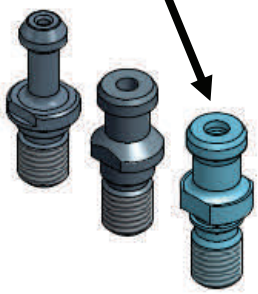
Your VEKTOR has a high speed spindle. To prolong spindle life, purchase and use only balanced tools and tool holders.

Warning

Damage to the spindle will occur if you don't use Park Industries part # 77943 retention knob with the BT40 tool holder on the VEKTOR.



This is the correct retention knob for the BT40 tool holder.



Routine Lubrication Schedule

Follow the procedures for daily, weekly, and monthly service tasks and take the appropriate maintenance or corrective action immediately on any item that does not meet the inspection criteria.

Every 8 Hours or Daily Service Tasks

- Flush the work table, gantry accordion covers, and rails with water to remove mud.
- Check the oil reservoir level and refill as needed with **Mobil Vactra #2** way oil. (page 5-7)
- Inspect the Tsunami air/water seperator. Clean or replace if necessary. (page 5-8)
- Check the oil overflow jar at away end of the bridge. (page 5-9)

Every 40 Hours or Weekly Service Tasks

- Check level of grease in FAG lubricator and replace grease cartridge and battery pack as necessary. (page 5-10) Function display on the drive unit indicates the following:
  - Green flash (every 8 seconds) - proper operation
  - Red flash (every 8 seconds) - malfunction
  - Green/Red flash (every 3 seconds) - lubricant canister empty
  - Permanently Red - discharge in process
- Blow dust from the air conditioner radiator holding the air hose 4 to 6 inches away from the upper grill. (page 5-11)
- Open the toward side gantry accordion covers. Wipe clean and inspect the round roller and round way. Wipe the round way with a light coat of **Mobil Vectra #2** way oil. (page 5-12)
- Open the away side gantry accordion covers. Wipe clean and inspect the flat roller and flat way. Wipe the flat way with a light coat of **Mobil Vectra #2** way oil. (page 5-12)
- Wipe clean and inspect the upper and lower round ways and bearings on the cross travel. Inspect the bearings for seal and retainer damage. (page 5-13)
- Clean, inspect and lubricate spindle cone and O ring. (page 5-15)
- Perform daily service tasks.

Every 80 Hours or Each Two Weeks Service Tasks

- Clean, inspect and spray Y Axis (Gantry) rack and pinion drive assemblies. (page 5-18)
- Clean, inspect and spray X Axis (Cross Travel) rack and pinion drive assemblies. (page 5-19)
- Perform daily service tasks.

Every 120 Hours or Monthly Service Tasks

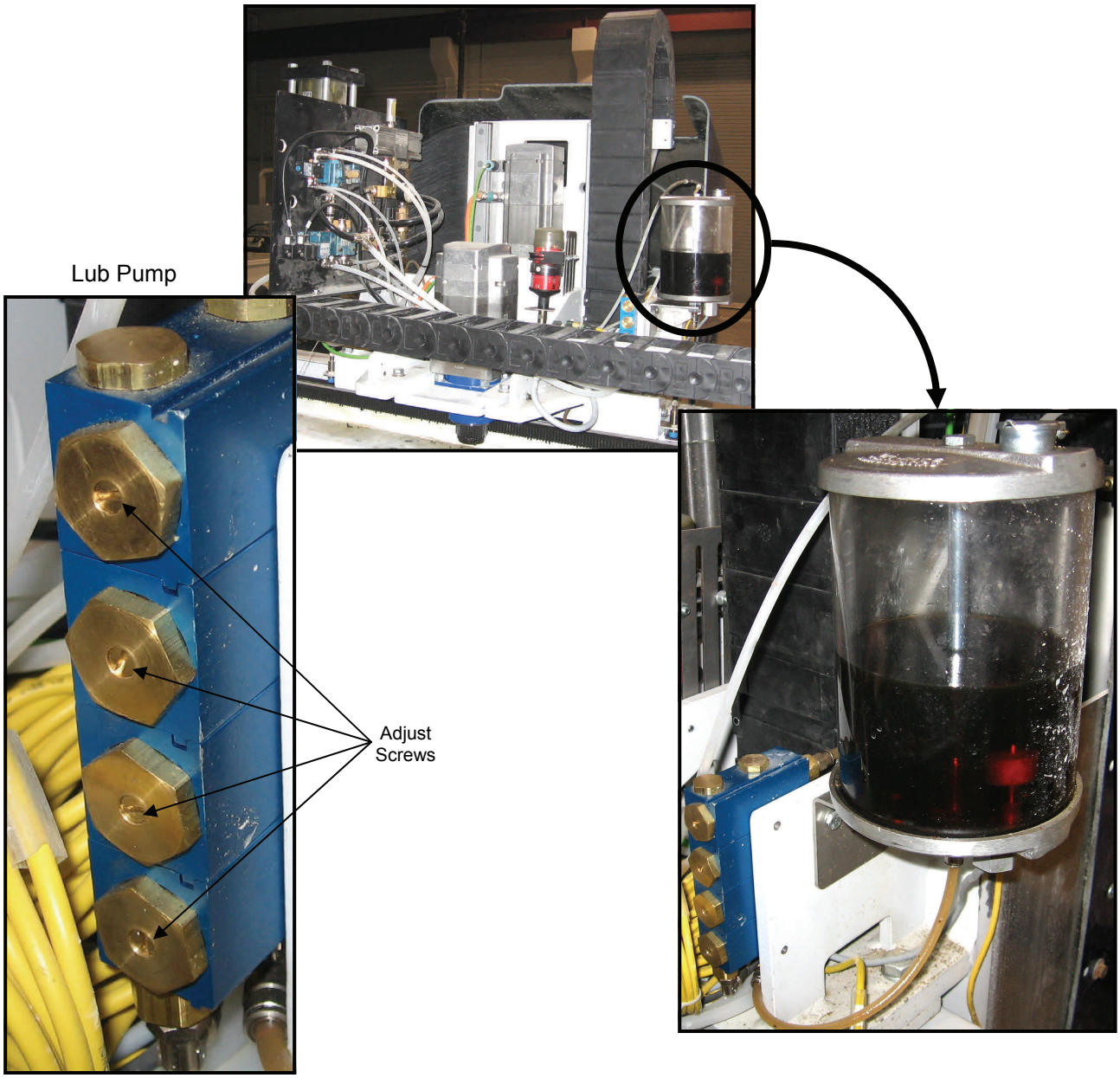
- Clean and inspect the both Z Axis (rise and fall) rails. Lubricate all four Z Axis (rise and fall) bearings with one pump of **Mobil XHP222** grease. (page 5-20)
- Lubricate the screw-base bearing with two pump of **Mobil XHP222** grease. (page 5-21)
- Replace the water filter. (pages 5-23)
- Remove and lubricate gripper balls. (page 5-25)
- Grease the spindle rotary union with one pump of **Mobil XHP222** grease. (page 5-30)
- Perform weekly and daily service tasks.

To prevent personal injury, remove power at the Main AC disconnect before performing maintenance procedures. Follow the lock-out/tag-out procedures established in your company.

Every 8 Hours or Daily Maintenance

Way Oil Reservoir

Check the oil reservoir level each day, located on the back of the cross travel assembly. Fill as necessary using **"Mobil Vactra #2"**.



The automatic cross travel lub. pump releases a fixed amount of oil to each cross travel bearing (four total) for an amount of X axis movement. For the VEKTOR, the distance is one hundred twenty (120) inches of X axis movement. The amount of oil is determined by the lub pump screws. During final assembly of the VEKTOR, the screws are turned clockwise until the screw bottoms-out and then counter clockwise one-half (½) turn. Increasing the counter clockwise turns, increases the amount of oil pumped to the bearings.

Every 8 Hours or Daily Maintenance

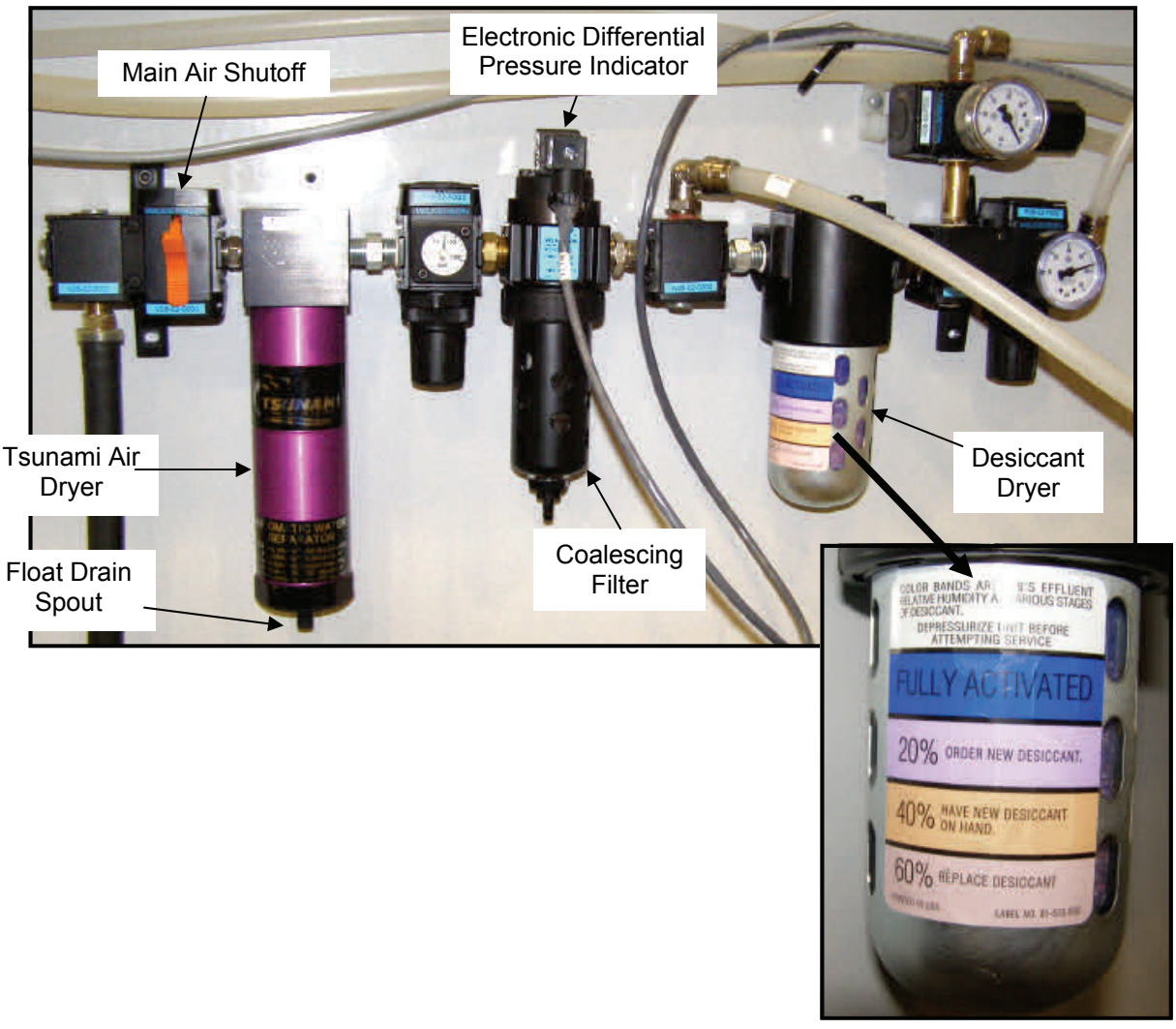
Air Supply/System Checks

Perform the following three air supply/system checks:

Tsunami Filter - Place your hand under the bottom of the filter. If air is coming out of the float drain spout, maintenance is required. Follow the procedure shown on page 5-31.

Coalescing Filter - This filter has an electronic indicator which signals the controller when the pressure differential across the filter is 10 PSI. A message indicating the filter must be replace within one (1) hour appears on the main screen. Replacement steps are shown on page 5-34.

Desiccant Dryer - Check the silica gel color. Service is required when the desiccant color or moisture indicator has changed from Blue (meaning dry) to Pink (meaning wet). When the color change occurs follow the steps shown on page 5-35 to change the silica gel.



Every 8 Hours or Daily Maintenance

Check Oil Tray Catch Jar

Visually check the oil level of the oil tray catch jar. Empty at the ¾ full level. **This oil can NOT be reused in the oil reservoir.**

Every 40 Hours or Weekly Maintenance

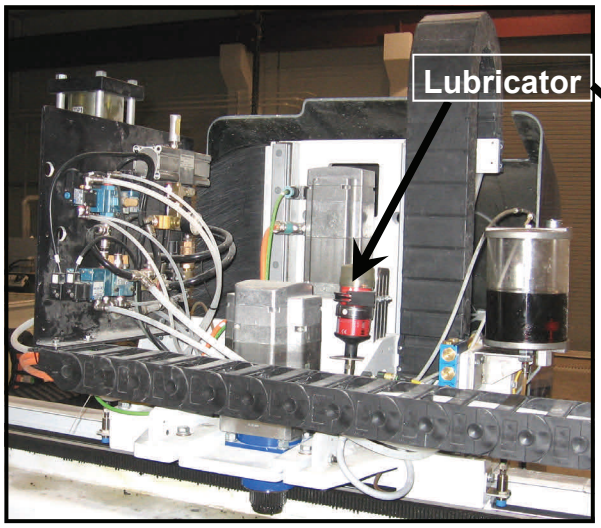
Check Grease Level In FAG Lubricator

Your machine is equipped with a self-contained and a self-powered FAG lubricator for the spindle ball-screw. Check the level of grease in the FAG lubricator each week. The lubricator should last approximately six months, after which you will need to replace the grease cartridge and battery pack. Please note, the lubricator is located on the back of the cross travel assembly.

LEDs and a LCD display displays the FAG lubricator status as follows:

Description	Frequency	Status	LCD Screen Displays
Green LED Flash	every 7 sec.	Normal operation	Time remaining and volume
Red LED Double Flash	every 8 sec.	Malfunction/high pressure warning *	OL
Green LEDS Flash	every 2 sec.	Lubricant discharging	ru
Red LED Double Flash	every 2 sec	**Empty reservoir, change	LC

* Indicates the line to the bearing may be blocked.
** This occurs when the unit is first activated.



NOTE:
Replacement steps are listed on page 5-36.

LEDs are positioned at four locations around the sides of the drive.

Every 40 Hours or Weekly Maintenance

Blow Dust From Air Conditioner Radiator

For a general inspection remove the front covers, use pressurized air to blow off the front cover **(USE A FACE MASK IF NECESSARY),** fan and the complete condenser coil area. To prevent damage to the compressor fins hold the air hose 4 to 6 inches away from the fins.

Check inside the electrical cabinet to make sure the air conditioner is holding temperature approximately 75 to 85 degrees Fahrenheit.

Caution	**To prevent personal injury, remove power at the Main AC disconnect before performing maintenance procedures. Follow the lock-out/tag-out procedures established in your company.**

Blow out fins on the Front Cover.

Blow out the Fan and Compressor Fins.

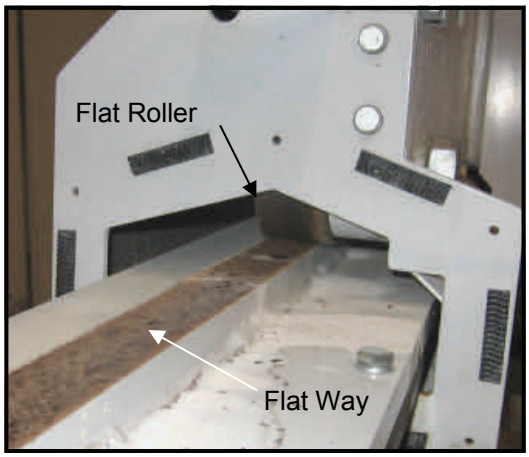
Every 40 Hours or Weekly Maintenance

Inspect, Clean and Oil Y Axis (Gantry) Ways And Rollers

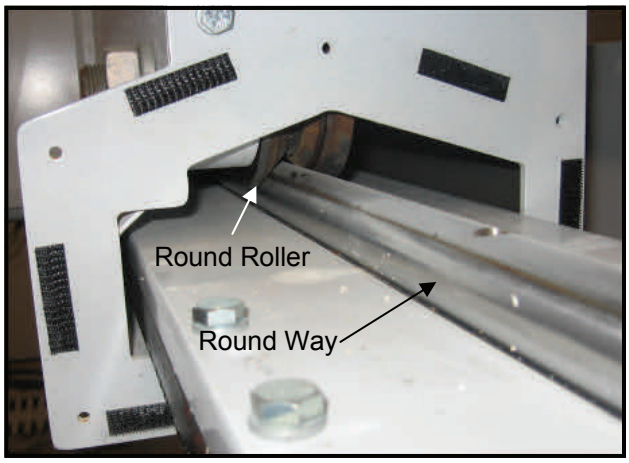
Wipe clean and inspect the round roller and round way on the toward side, and the flat roller and flat way on the away side. Wipe the round way on the toward side, and the flat way on the away side, with a light coat of **Mobile Vectra #2** way oil.

Note: To properly clean these components follow these steps:

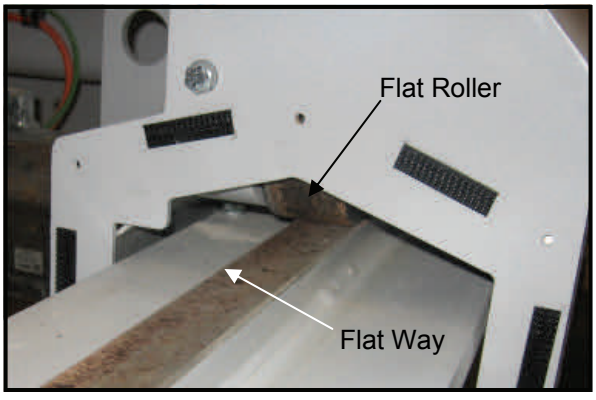
- move the bridge to +Y soft stop
- **E-Stop** the machine and then open the front accordion covers on both sides of the bridge
- inspect, clean and oil the exposed rollers and ways
- move the bridge to -Y soft stop
- **E-Stop** the machine and then open the front accordion covers on both sides of the bridge
- inspect, clean and oil the exposed rollers and ways



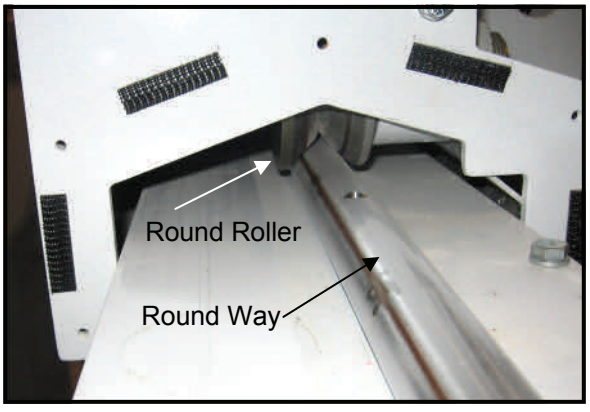
Away Side Front Roller and Way



Toward Side Front Roller and Way



Away Side Rear Roller & Way



Toward Side Rear Roller & Way

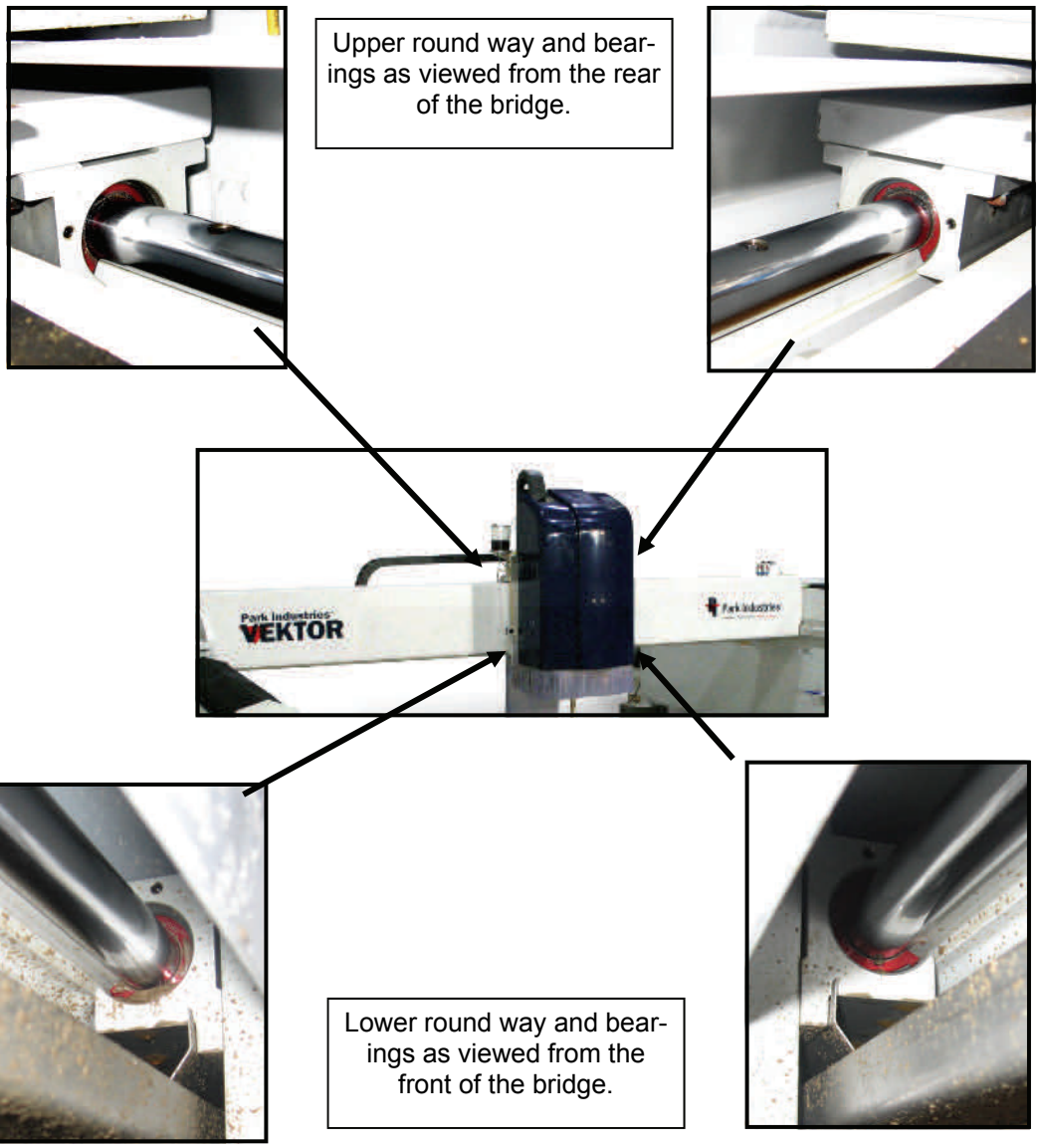
Every 40 Hours or Weekly Maintenance

Wipe Clean And Inspect X Axis (Cross Travel) Way And Bearings

Movement of the spindle assembly on the X Axis travel is accomplished by a upper way with two bearings and a lower way with two bearings. Wipe clean and inspect the round ways and bearings. Inspect the bearings for seal and retainer damage.

Note: To properly clean these components follow these steps:

- move the spindle to -X soft stop
- **E-Stop** the machine
- Inspect and clean exposed upper and lower ways
- move the bridge to +X soft stop
- **E-Stop** the machine
- Inspect and clean the portion of the ways not cleaned above



Spindle Gripper Components and Operation

Below is a brief explanation of the gripper parts and how they function to secure a tool to the spindle. The drawings used are not to scale and do not show all components.

Figure 1 illustrates the gripper assembly's basic parts. The gripper is a tube about one inch in diameter. The brass bushing/O-ring assembly is free to move inside the tube, but is pinned in a way that it does not fall out. The spring pushes the bushing and O-ring downward.

Figure 2 illustrates the inside shape of the tapered portion of the spindle. Note the cavity located between the tapered portion and straight upper part. When extended the gripper moves into this cavity and the balls are able to move outward allowing the retention knob to be inserted.

Figure 3 illustrates the gripper position when the spindle has been extended. Notice that the balls can now be pushed outward into the larger cavity to allow the head of retention knob to be inserted. As the retention knob is pushed upward, it will engage the O-ring/brass bushing assembly which is currently being pushed downward by the spring.

Figure 4 illustrates the position of a properly inserted retention knob. When manually loading a tool, it is very important that the operator pushes the tool upward until resistance is felt and then continue to push upward until the tool bottoms out before releasing the manual tool change button. By pushing until a bottom is reached (spring compressed), the balls are allowed to grip the neck portion of the knob when the gripper is retracted. If the retention knob is NOT fully seated, the balls can be damaged during retraction or the tool may fly out during operation.

Figure 5 illustrates the gripper and retention knob after the spindle has been retracted. During retraction, the straight walls of spindle body force the ball inward gripping the neck of the retention tool. The spring (now compressed) is pushing the brass collar and O-ring against the top of the retention tool ensuring a water tight connection.

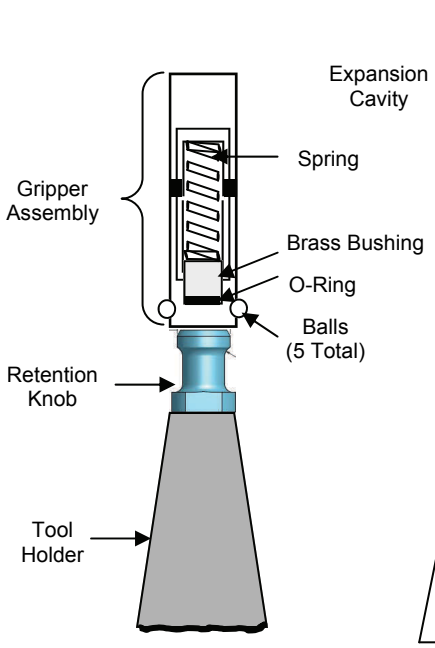


Figure 1

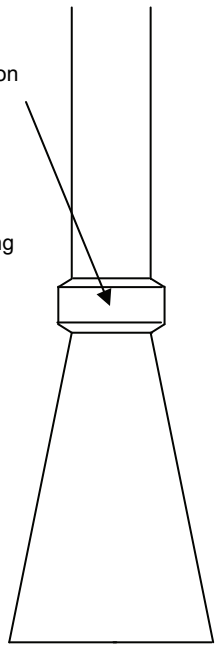


Figure 2

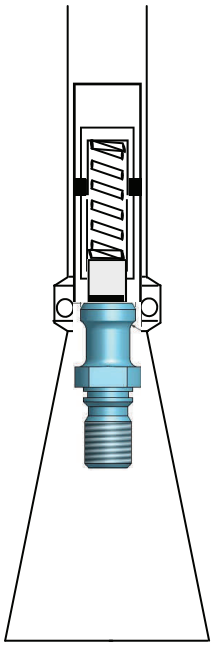


Figure 3

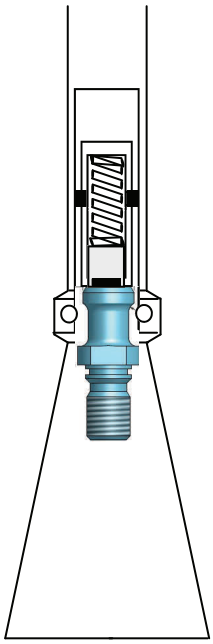


Figure 4

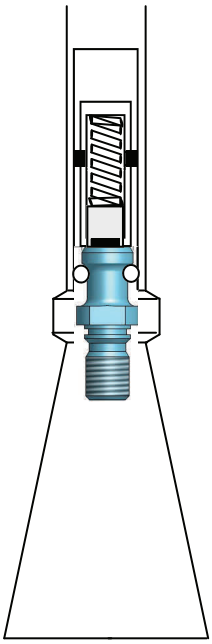


Figure 5

Every 40 Hours or Weekly Maintenance

Clean, Inspect and Lubricate Spindle Bushing Slides and O ring

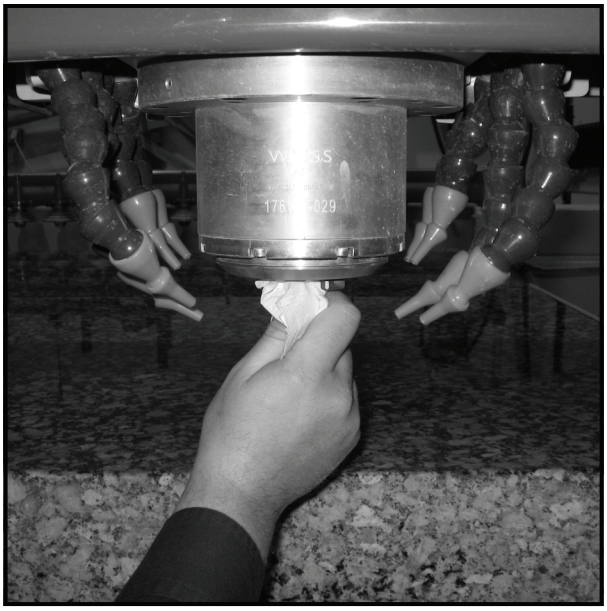
This preventative maintenance procedure covers three tasks:

- Clean the tapered lower portion
- Inspect the O-ring and brass bushing
- Apply lubricate to the cavity

For additional information on the individual components found in the gripper assembly and how these components function see the explanation on the opposite page.

1. Dampen a clean shop towel with denatured alcohol, wipe down tool holders, ensure they are clean and dry

2. Wipe the taper portion of the spindle with clean shop towel dampened with denatured alcohol.
   Never use air in the spindle taper.

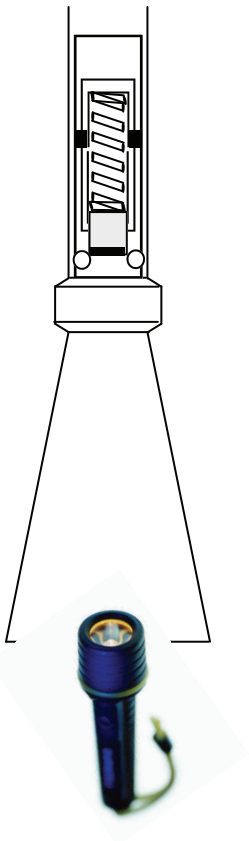


Continued on next page.

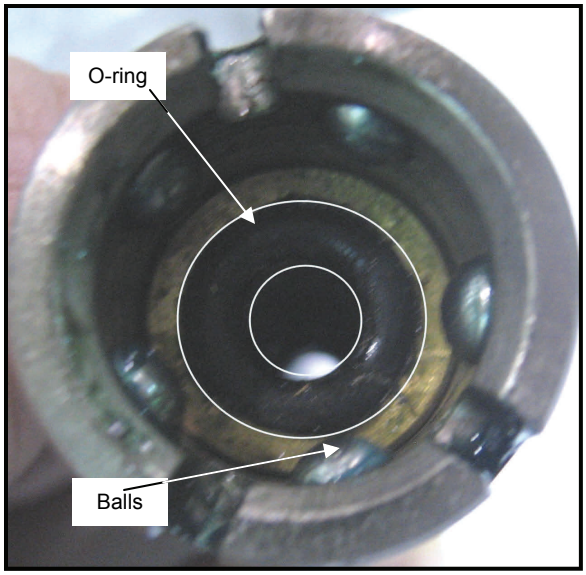
Every 40 Hours or Weekly Maintenance

Clean, Inspect and Lubricate Spindle Bushing Slides and O ring (cont'd)

3. Using a flashlight , visually inspect the O-ring and look for any missing or damaged balls.



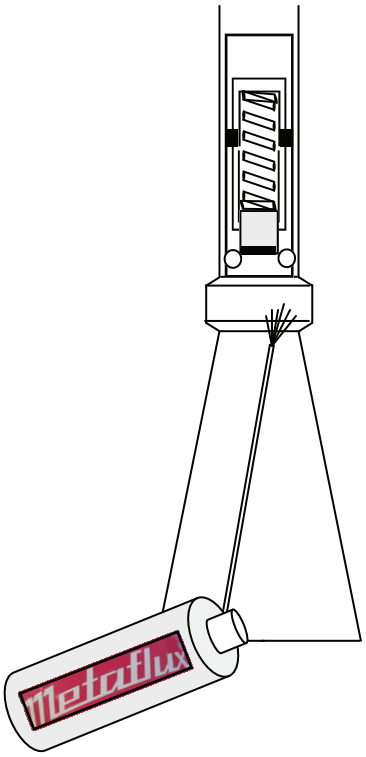
VEKTOR Gripper Assembly



Every 40 Hours or Weekly Maintenance

Clean, Inspect and Lubricate Spindle Bushing Slides and O ring (cont'd)

4. In the clamped position (manual tool button NOT pressed) spray Metaflux grease into the cavity area. The goal here is to ensure there is grease in cavity so the balls can move out when the gripper is extended and in when the gripper is being retracted. Wipe excess grease from spindle taper .

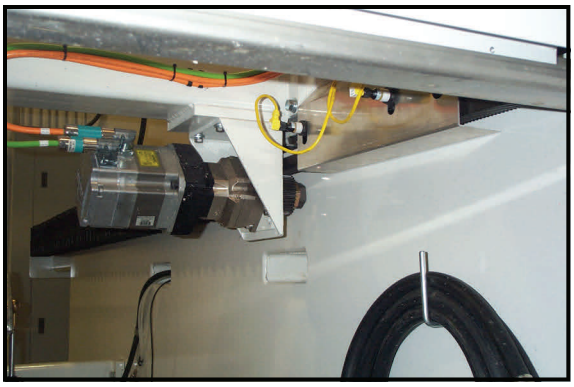


Clean the spindle with a clean shop rag and denatured alcohol and let the alcohol dry completely. **NEVER USE AIR TO DRY THE SPINDLE!!** In unclamped position (manual tool change button is pushed on the side of the fiberglass shroud) spray Metaflux grease for 3 seconds into gripper fingers. Actuate drawbar 20 times to circulate grease.

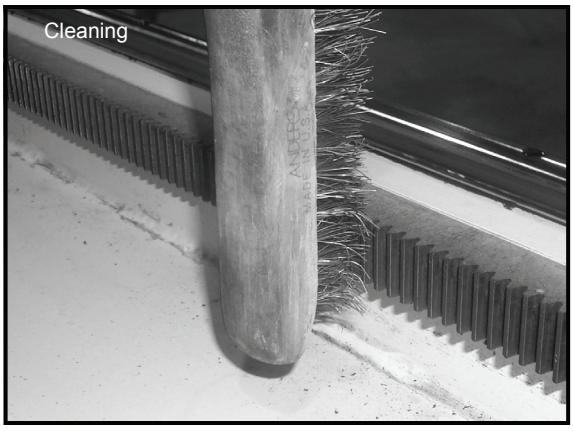
Every 80 Hours or Every Two Weeks Maintenance

Clean, Inspect, And Spray Y Axis (Gantry) Rack And Pinion

On both the toward and away side, clean and inspect the rack and pinion drive assemblies and then coat with a light layer of **Kluber Microlube® GB 0** grease. Use a very short bristle brush to apply the grease.



Gantry gear rack and pinion drive assemblies are located on each side of the machine.

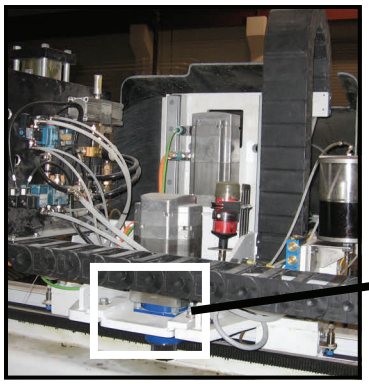


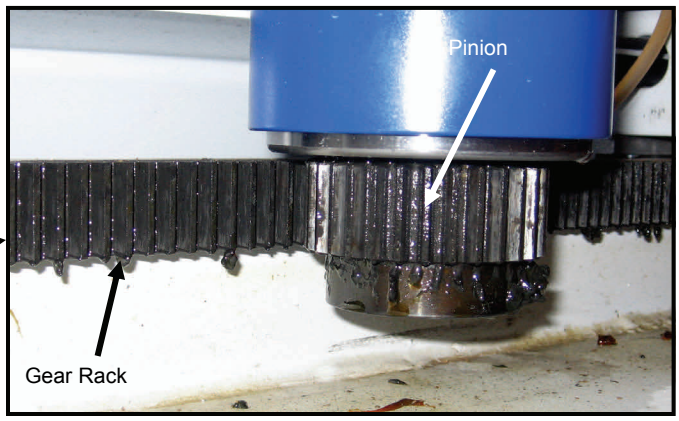
Caution	**Replace all protective covers before returning power.**

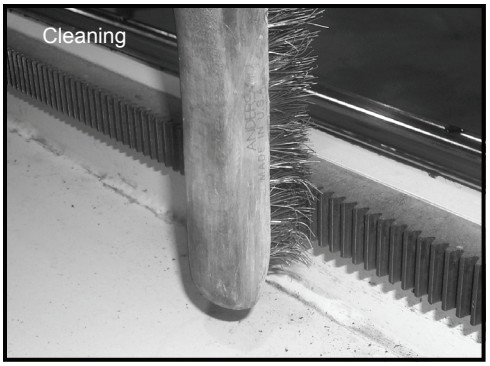
Every 80 Hours or Every Two Weeks Maintenance

Clean, Inspect, And Spray X Axis (Cross Travel) Rack And Pinion

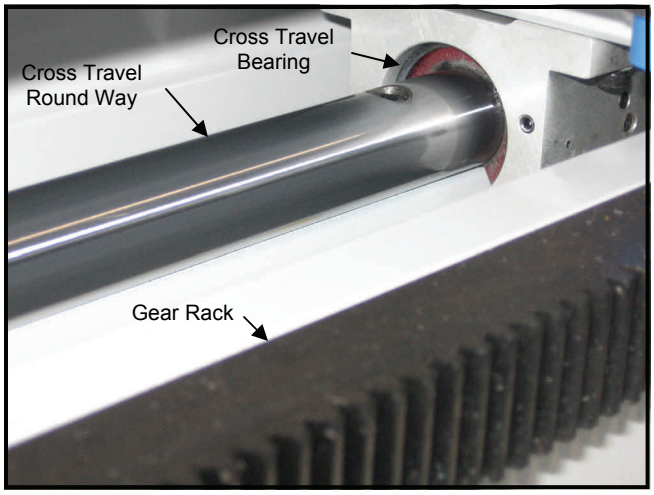
X Axis (Cross Travel) gear rack and pinion drive assembly can be accessed from the back of the machine. Clean and inspect the rack and pinion drive assemblies and then coat with a light layer of **Kluber Microlube® GB 0** grease. Use a very short bristle brush to apply the grease. When done cleaning and lubing the rack, wipe the upper round way with a light coat of **Mobile Vectra #2** way oil.







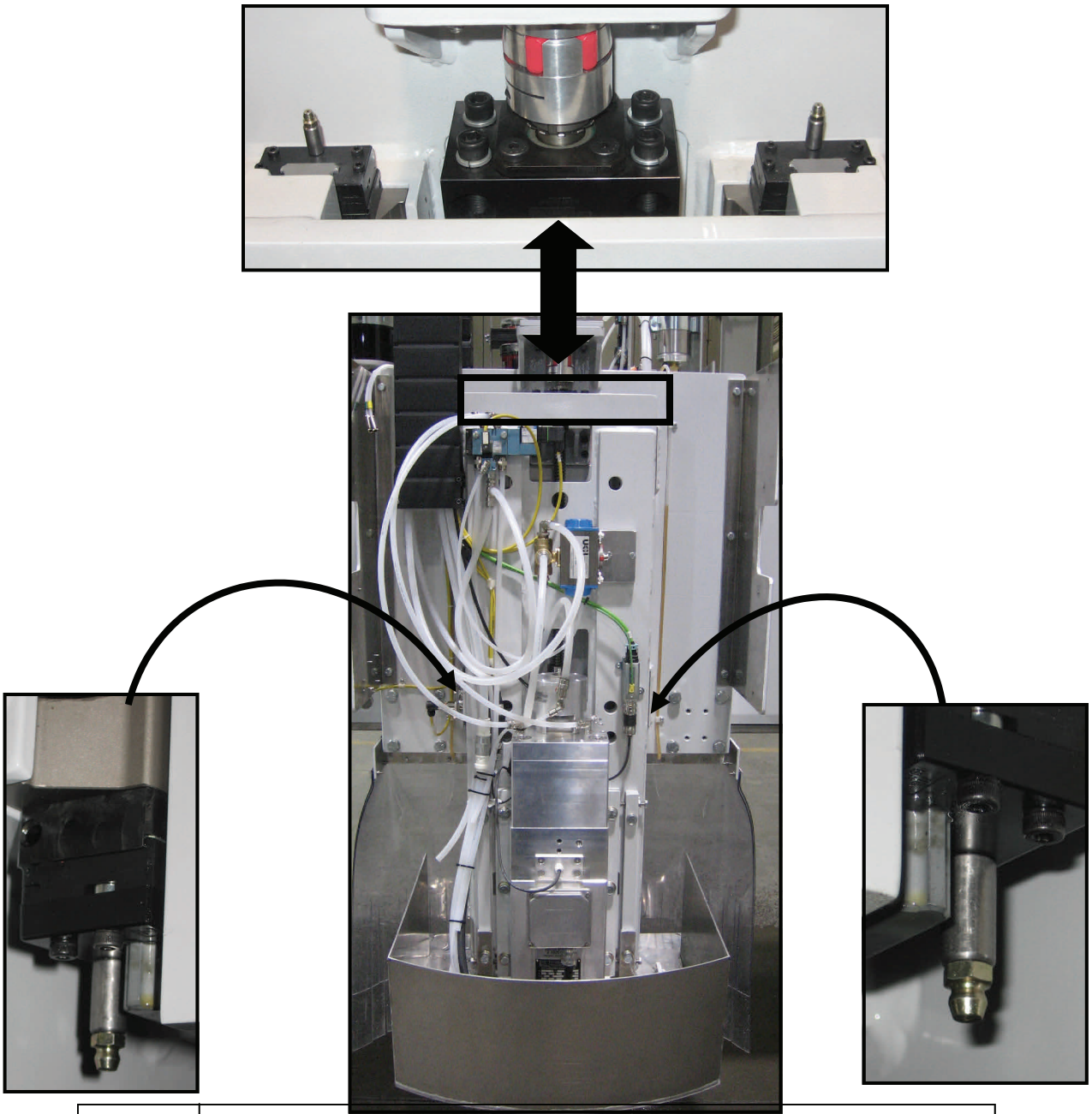
Caution	**When cleaning, DO NOT allow any particles to remain on the cross travel round ways.**
Caution	**When lubricating, DO NOT allow any grease to remain on the cross travel round ways. When done, wipe the ways with a rag coated with way oil, leaving a light coat of oil.**



Every 120 Hours or Monthly Maintenance

Wipe Clean And Inspect Z Axis (Rise and Fall) Rails And Bearings

Remove the blue spindle cover to access the Z Axis (rise and fall) bearings. Wipe clean and inspect both rails and all four slide bearings. Grease all four rise and fall rail slide bearings once a month with one pump of **Mobile XHP222** grease.



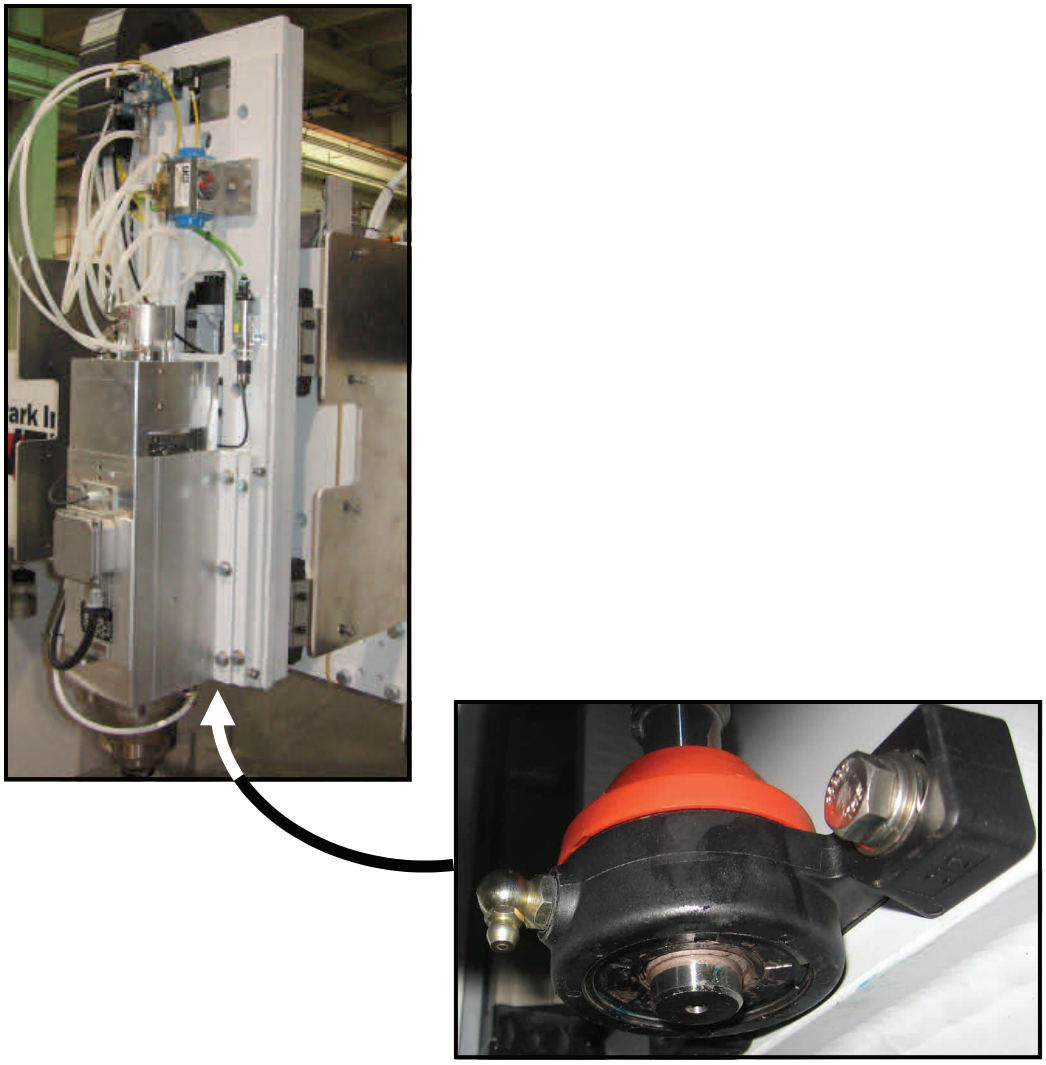
Caution	**Replace all protective covers before returning power.**

Every 120 Hours or Monthly Maintenance

Grease Screw-Base Bearing

Grease the rise and fall turn screw-base bearing, once per month, with one pump of **Mobile XHP222** grease. Please note, the screw-base bearing is located under the blue spindle cover, between the rise and fall ways, near the bottom.

After the spindle cover is remove, move the spindle assembly to the upper limit.



Caution	**Replace all protective covers before returning power.**

Every 120 Hours or Monthly Maintenance

Change Water Filter

The following information was taken from Park Technical Bulletin 10020A.

Spindle Water Filtration Recommendation

Spindles are highly engineered to provide a useful life for many years. Like other major components (e.g. car engine), routine care and maintenance are essential to prolonging the life of the spindle. Eventually, the spindle will need to be replaced but to prolong its life, it is critical that you adhere to the below specification guidelines.

Feed your CNC machine with clean water and air. Park Industries requires that water filtered to 50 microns be supplied to the spindle. In addition, the dewpoint of the air should be no higher than 37.4 degrees Fahrenheit.

The CNC has filters that are designed to act as a safety device to prevent damage that may occur in the event that the air and water filter systems fail. It is a good idea to catch impurities or contaminants upstream at the source before they get to your CNC machine. If filters are plugged more often than expected, evaluate your air and water systems upstream to find the source of the problem. **The machine filters are not intended to be part of the shop air and water system but rather a safety net in the event of a failure upstream.**

It’s good practice to change the filters on your machine, in accordance with the maintenance manual, to ensure that the safeguard filters are in good condition to catch the large particles that can potentially damage your spindle.

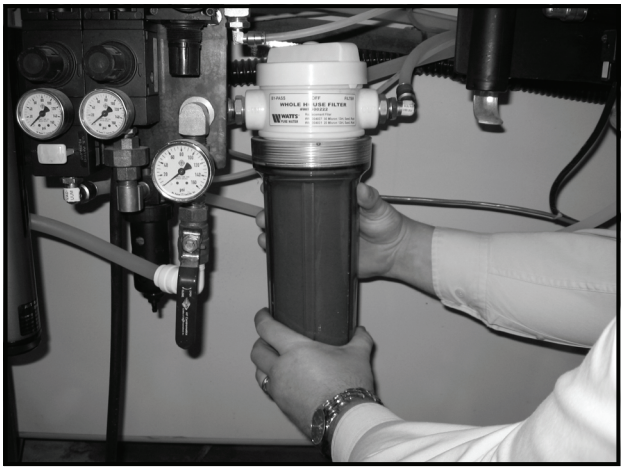
You have made a substantial investment in your CNC machine and we want to help you utilize this machine to its ultimate capabilities. Give it the care it deserves.

Every 120 Hours or Monthly Maintenance

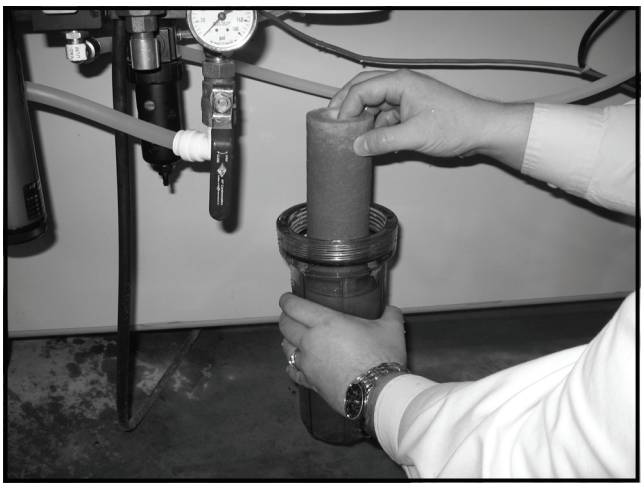
Change Water Filter (cont'd)

The following information was taken from Park Technical Bulletin 10020A.

1. Changing the water filter monthly is an important part of maintenance. To start, turn off water supply at the source. Grasp the filter bowl and unscrew it counter-clockwise.



2. Lower the filter bowl and pull out the old filter. Discard the old filter.



3. Rinse the filter bowl out with clean water and be sure to get out any contaminates. Wipe filter bowl with a clean shop towel. Inspect o-ring at the top of the filter bowl and replace if necessary.

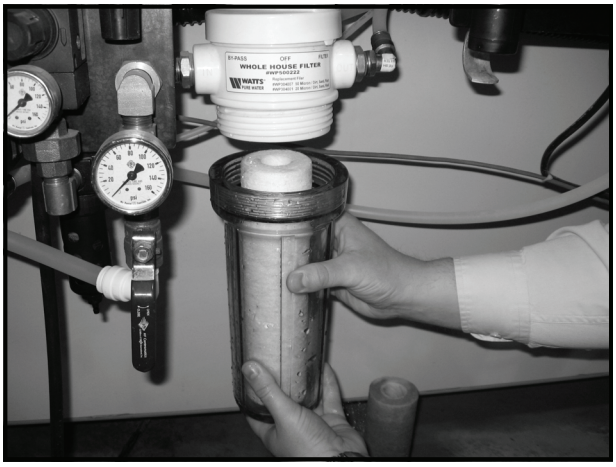
Every 120 Hours or Monthly Maintenance

Change Water Filter (cont'd)

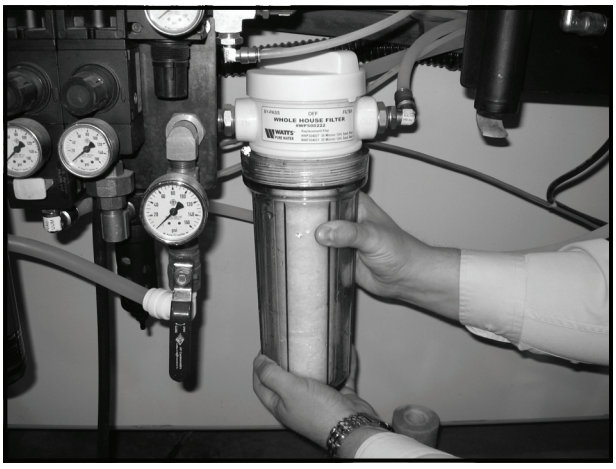
The following information was taken from Park Technical Bulletin 10020A.

4. Place new filter element (Park part #70001172) into the filter bowl. Be sure to place new filter onto alignment spud that protrudes from the bottom of the filter bowl.

5. Line up filter bowl with the threads of the filter housing.



6 Tighten filter bowl on hand tight only. Never use any kind of wrench to tighten the filter bowl onto the filter housing.



Every 120 Hours or Monthly Maintenance

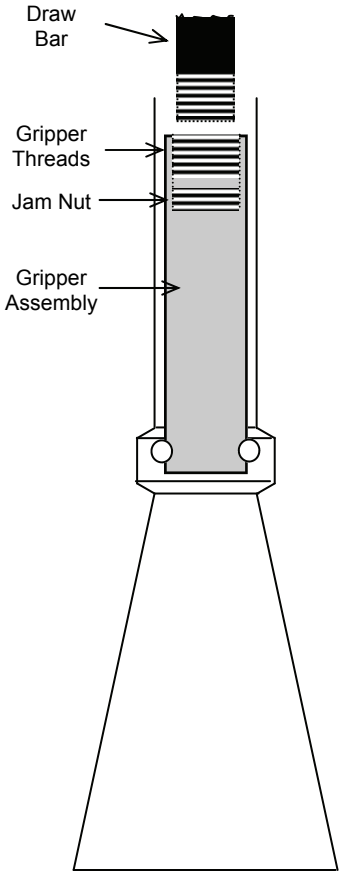
Spindle Gripper Removal/Replacement

The purpose of the maintenance task is to prolong spindle life and ensure proper tool clamping. The gripper assembly is cleaned, inspected and lubricated before being replaced. This procedure lists the steps to remove and replace the spindle gripper assembly.

The three components referenced in this procedure are:

- Draw Bar - threaded to accept gripper, has a 5mm hex center
- Gripper - threads at top end, notches at bottom (shown below)
- Jam Nut - inside the gripper has a 6mm hex center

The gripper assembly screws onto the main spindle shaft. To ensure that gripper assembly does not loosen over time, a jam nut inside the gripper locks the assembly to the spindle shaft.

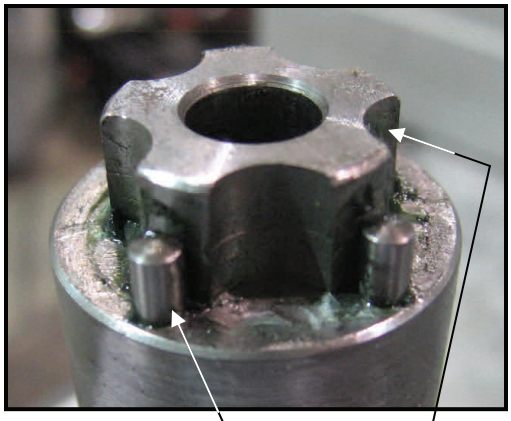


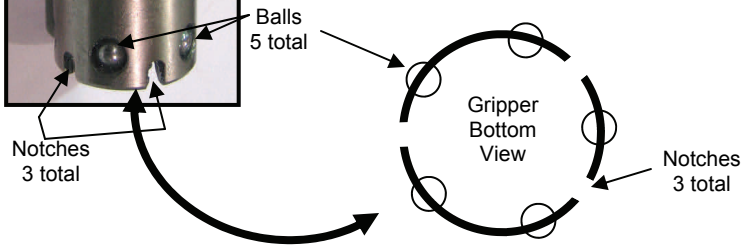
The following items are needed to remove the gripper:

- Gripper removal tool (shown below)
- 6mm Allen/hex wrench at least 10 inches long
- 5mm Allen/hex wrench at least 10 inches long
- Dead blow hammer
- Large towel or shop cloth (catch any balls that fall out)
- Metaflux grease

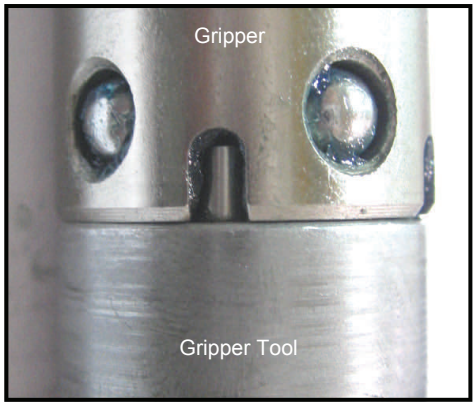
Gripper

Gripper Tool





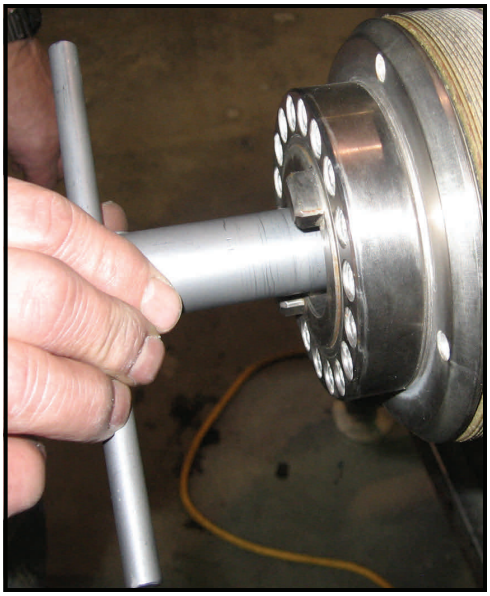
Gripper Tool
Inserted into Gripper



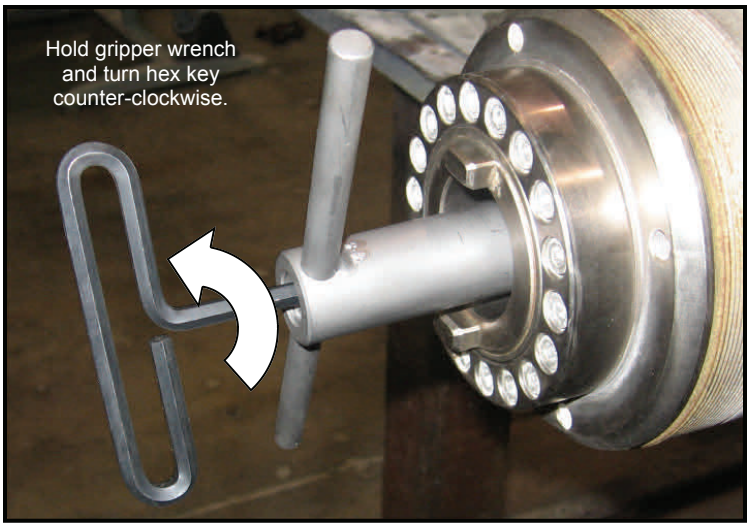
Every 120 Hours or Monthly Maintenance

Spindle Gripper Removal/Replacement (cont'd)

1. Move the spindle to a suitable work area.
2. If a tool is loaded, remove at this time.
3. Remove the blue spindle cover.
4. Spread a towel or shop cloth on the floor or cutting surface directly below the spindle.
5. **Press and hold manual tool change button until the gripper is removed.**
6. Insert the gripper tool and rotate until the three pin align with the gripper notches.



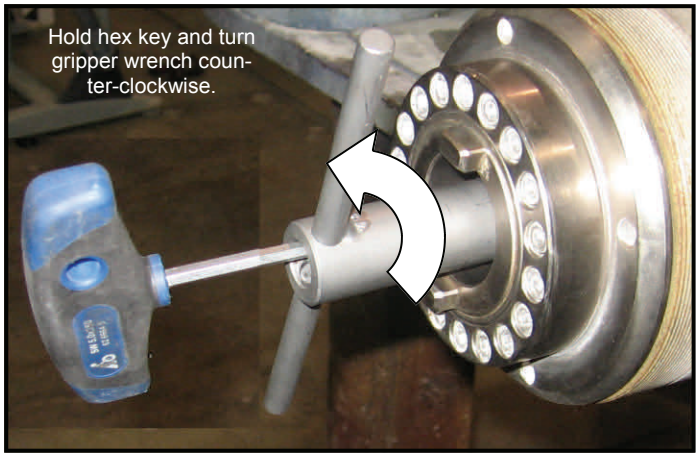
7. Insert the 6mm hex key wrench through the center of the gripper tool into the jam nut.
8. While holding the gripper tool, turn the 6mm hex key wrench counter-clockwise loosen the jam nut.
   NOTE: The jam nut bottoms out at about 1½ turns of the 6mm hex key wrench.
9. When the jam nut bottoms out, remove the 6mm hex key wrench.



Every 120 Hours or Monthly Maintenance

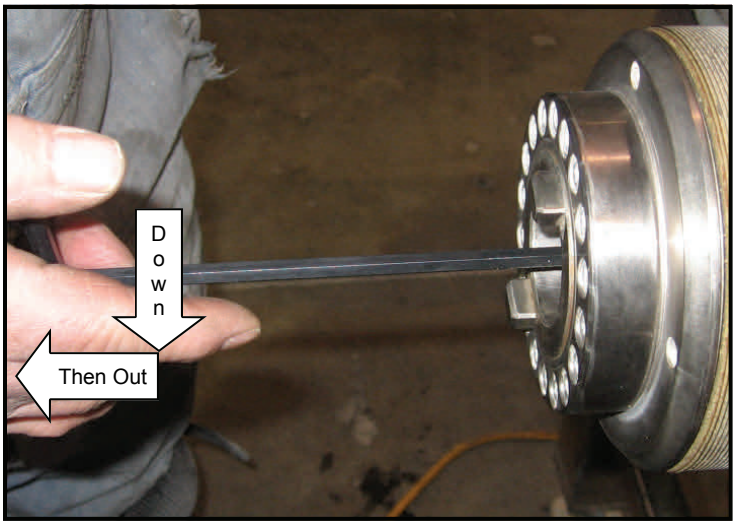
Spindle Gripper Removal/Replacement (cont'd)

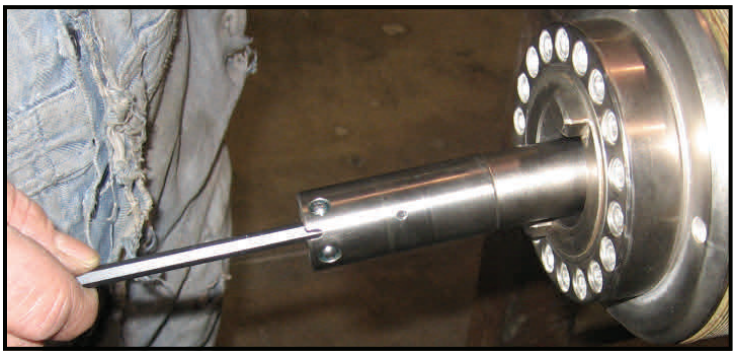
10. Insert the 5mm hex key wrench through the center of the gripper tool into the center of the draw bar.
11. While holding the 5mm hex key wrench, turn the gripper tool counter-clockwise to loosen the gripper assembly. NOTE: May have to tap the tool's handle with mallet to free the gripper assembly.
12. Continue to turn hold the hex key and turn the gripper tool counter-clockwise for about 16 or 17 turns.
13. Remove the 5mm hex key and gripper tool.



In the next step, the 6mm key is used to pull the gripper from the spindle body. The task is to insert the key into jam nut, create a friction between the key and nut and then pull key and gripper from the spindle body.

14. Insert the 6mm hex key into the gripper stopping at the jam nut. Push and hold the key's end slightly downward. At the same time pull the key away from the spindle body. Note: It may take a few attempts until the gripper slides out.
15. **It is now OK to release the manual tool change button.**

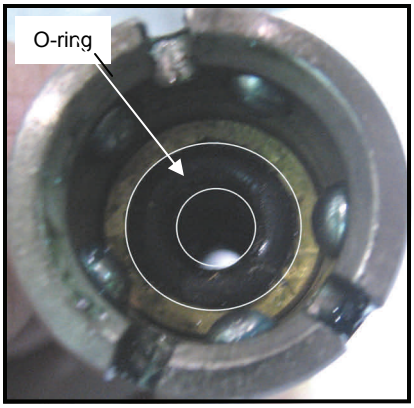




Every 120 Hours or Monthly Maintenance

Spindle Gripper Removal/Replacement (cont'd)

16. Inspect the balls. Check for flat surfaces and pit marks. Replace any damaged balls. Note: A small amount of grease around each ball will help hold them in place. Also inspect the O-ring.

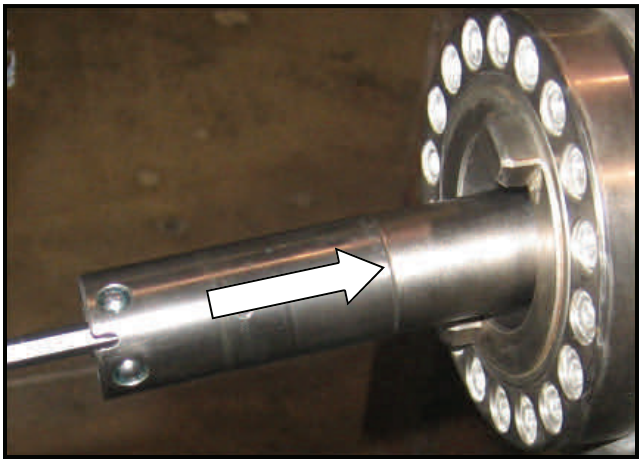


17. Insert the 6mm hex key into gripper stopping at the jam nut.

18. Coat the gripper with a liberal coat of Metaflux.

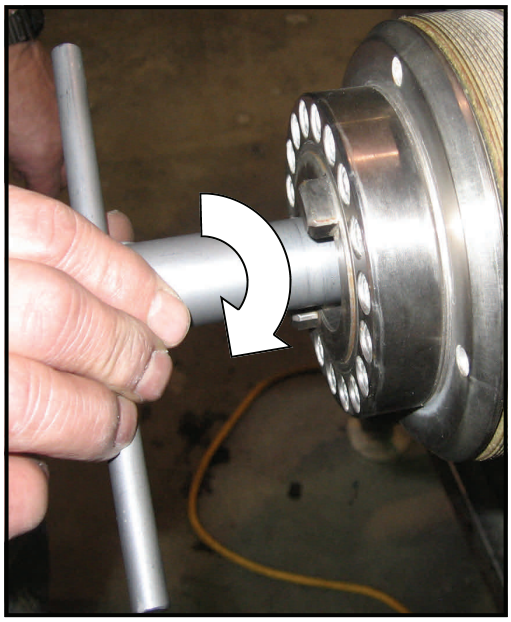
19. **Press and hold manual tool change button until the gripper is replaced.**

20. With 6mm hex key still inserted in the gripper, slide the gripper assembly into the spindle body until it bottoms out.

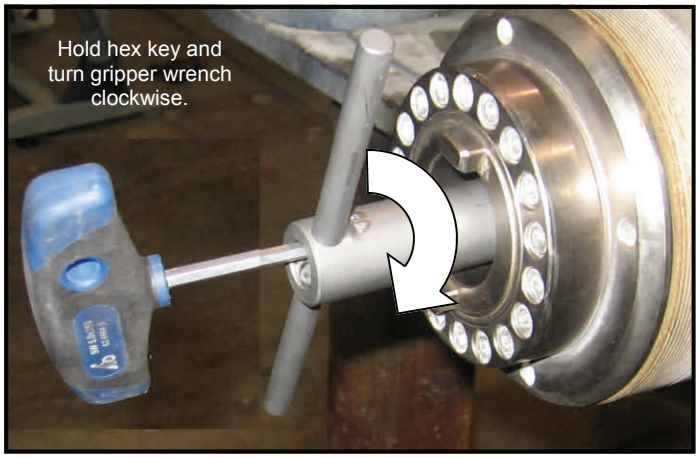


Every 120 Hours or Monthly Maintenance

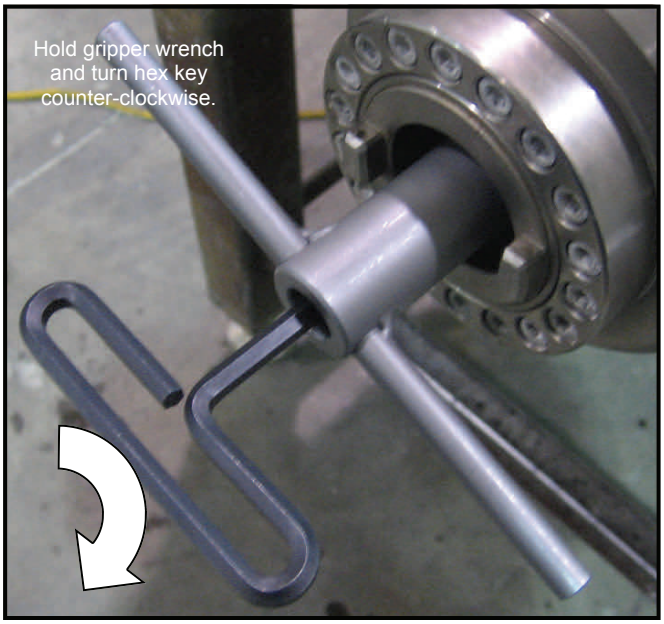
Spindle Gripper Removal/Replacement (cont'd)



21. Remove the 6mm hex key and insert the gripper tool. Rotate until the three pins align with the gripper notches.
22. Continue to turn the gripper tool clockwise for about 16 or 17 turn or until tight.



23. Insert the 5mm hex key wrench through the center of the gripper tool into the center of the draw bar.
24. While holding the 5mm hex key wrench, turn the gripper tool clockwise to tighten the gripper assembly.
25. Remove the 5mm hex key and gripper tool.

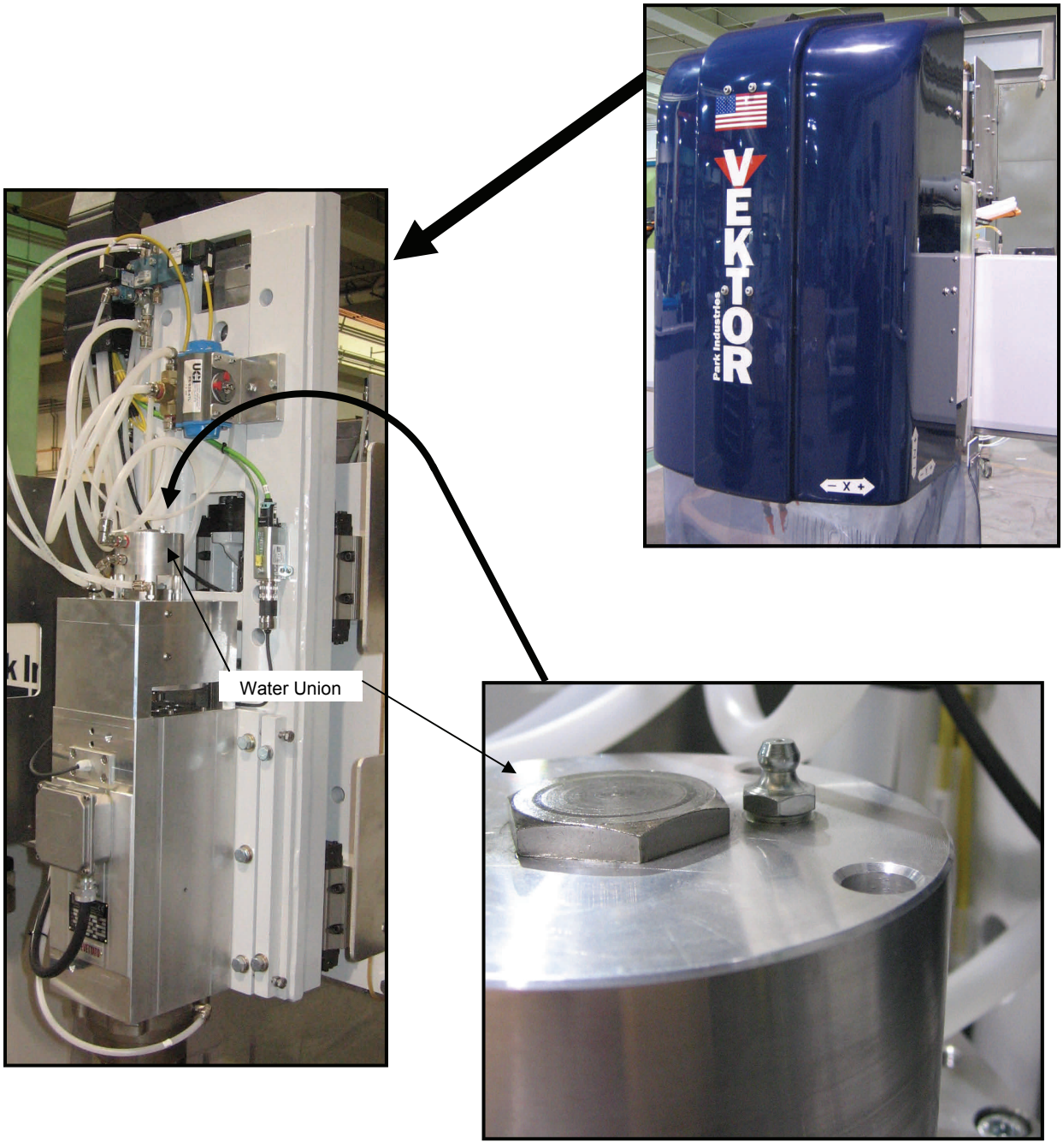


26. Insert the 6mm hex key wrench through the center of the gripper tool into the jam nut.
27. While holding the gripper tool, turn the 6mm hex key wrench clockwise to tighten the jam nut.
    NOTE: The jam nut bottoms out at about 1½ turns of the 6mm hex key wrench.
28. **It is now OK to release the manual tool change button.**

Every 120 Hours or Monthly Maintenance

Grease the Spindle Rotary Union

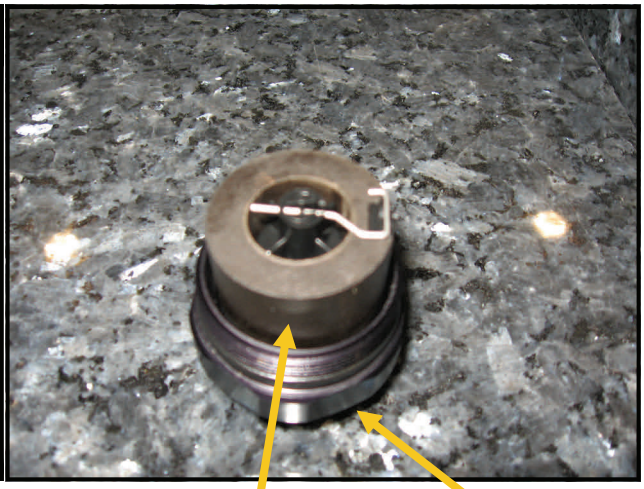
Remove the blue spindle cover to access the spindle rotary union. Grease the bearing once a month with one pump of **Mobil XHP222** grease.



Replacing the Tsunami Float Drain

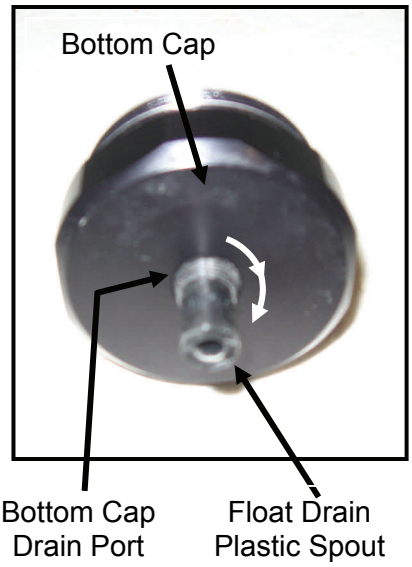
Replacing the Float Drain is simple and can be done with a minimum of tools. Care should be taken when handling the Float Drain as it is plastic and comprised of many small parts.

1. **Remove the Bottom Cap from the Main Body -** Grasp the main body with one hand and with the other hand turn the bottom cap counter clockwise. This will separate the Bottom Cap from the main body (the Float Drain is attached to the Bottom Cap)



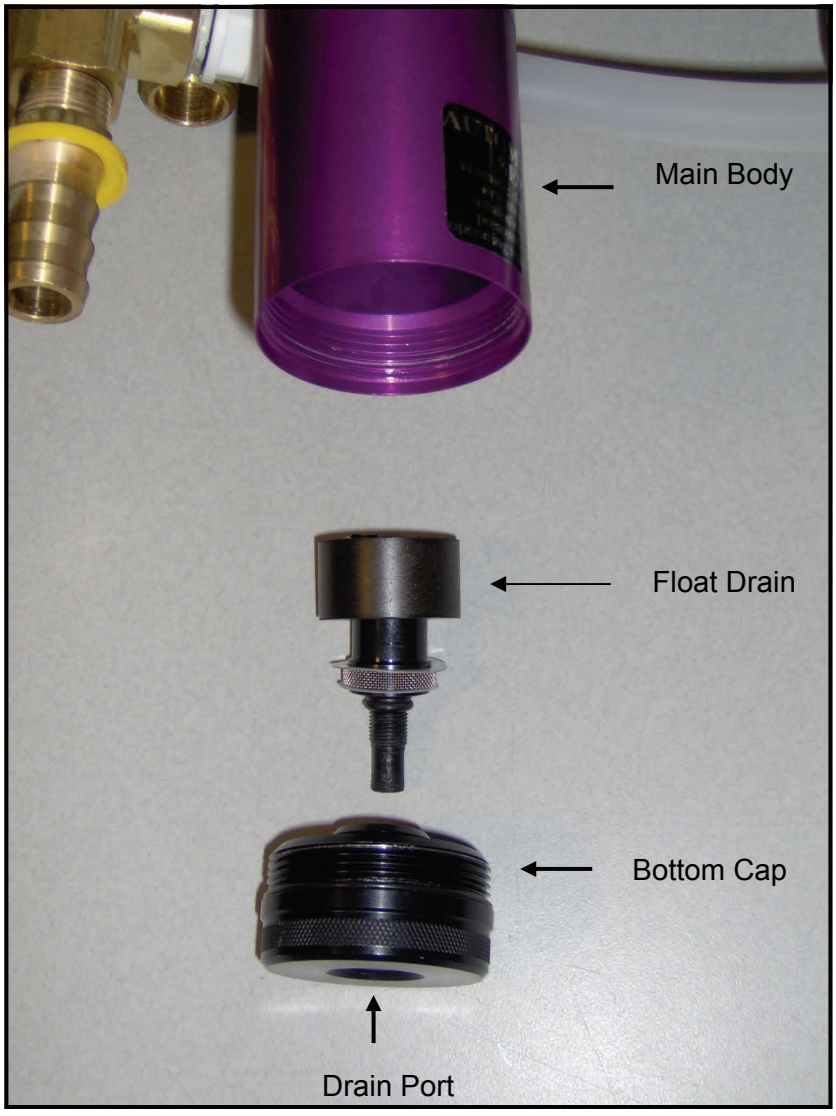
2. **Locating the Float Drain Spout.** ***Important: Do not attempt to unscrew the Float Drain using the portion normally found inside the body. -*** The Float Drain extends through the Bottom Cap and emerges recessed in the 1/2" NPT drain port. In this view you can see the plastic spout shown inside the port (hole in cap).

3. **Unscrew Float Drain -** Using a tool (pliers) grasp the Spout and unscrew the Float Drain down into this side of the Bottom Cap. Turn to the right (clockwise) to loosen. Remove the old Float Drain and discard.



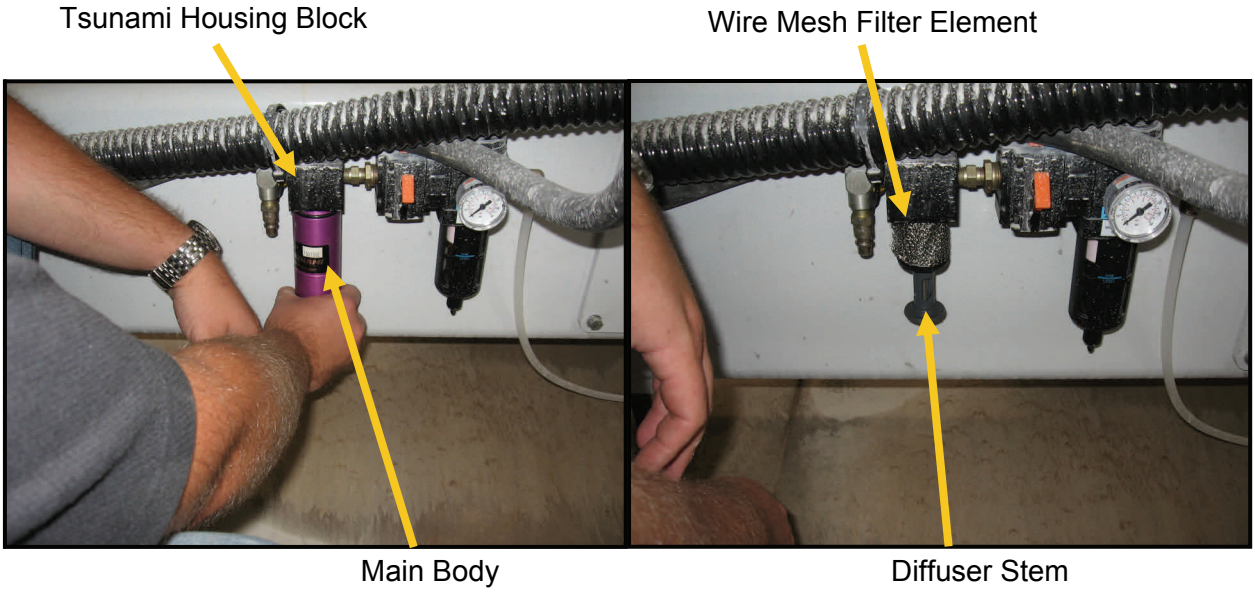
Replacing the Tsunami Float Drain (cont'd)

4. **Install New Float Drain -** Grasp the new Float Drain firmly and start threading into the into the Bottom Cap. The O-ring on the Float Drain will snug inside the countersink of the Bottom Cap. Finger-tighten only. Then reinstall Bottom Cap to Body.

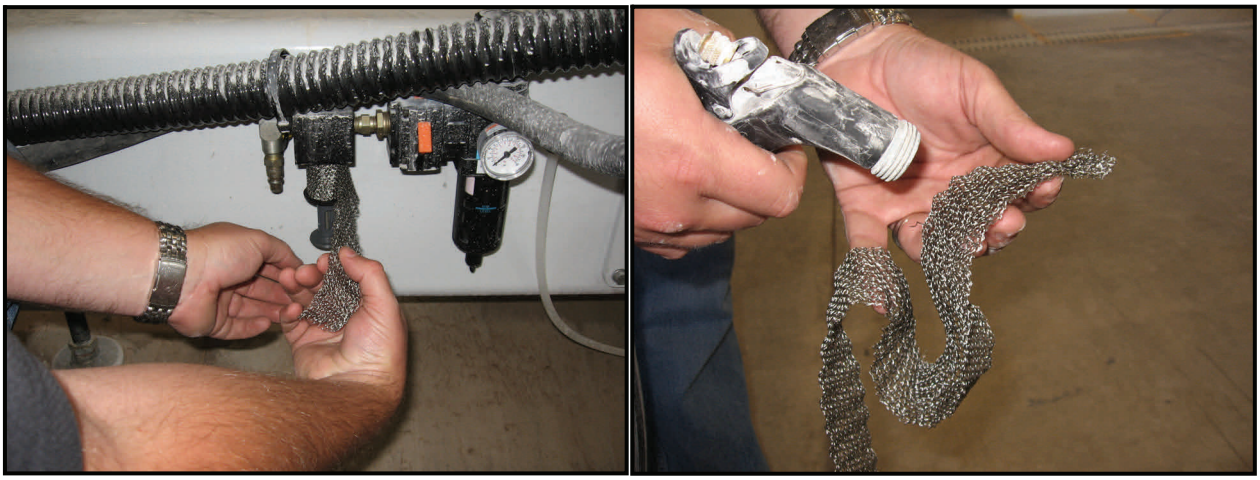


Replacing the Tsunami Float Drain (cont'd)

5. While replacing the Float drain it is a good idea to clean the Wire Mesh Filter that is located inside the Main Body. To do this unscrew the Main Body counter clockwise to remove it from the Tsunami Housing Block. The Wire Mesh Filter is wrapped around a Diffuser Stem inside the Tsunami.



6. Unwrap the wire mesh and wash out the filter element. To re-install simply do the reverse. Re-wind the filter element back onto the Diffuser Stem and screw the Main Body back on to the Tsunami Housing Block.

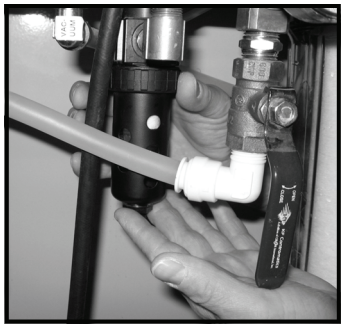


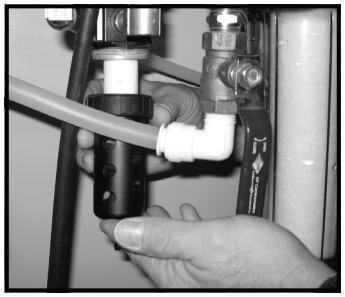
Replacing the Coalescing Filter

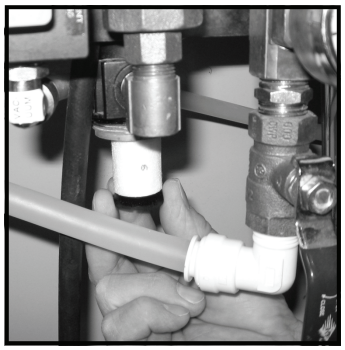
Below is an illustrate parts break down of the Wilkerson filter found on the VETKOR.
To replace the filter:

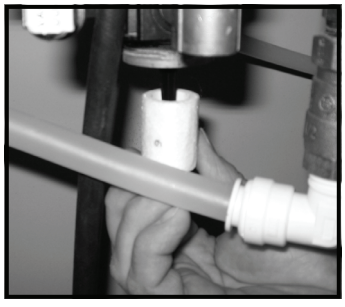
- Turn retaining collar counter clock wise and remove retaining collar, bowl and bowl guard.
- Unscrew the baffle
- Slide the element off the filter holder

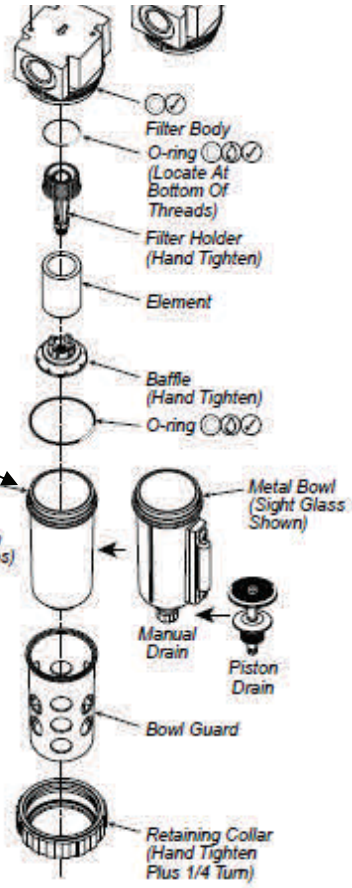
Reverse the steps to reassemble, only hand tighten the baffle, bowl and collar.









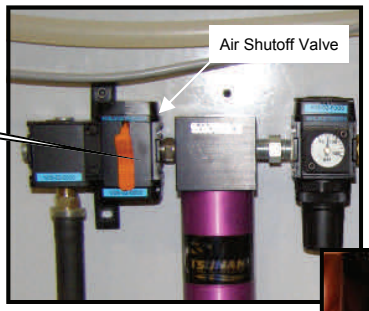


To clean Poly-carbonate Bowl use mild soap and water. **DO NOT** use cleaning agents.

Replacing the Silica Gel Desiccant

Replacement desiccant part number is 3162028.

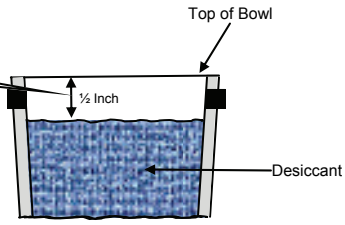
1. Turn off and depressurize the air lines.



2. Remove bowl guard as follows:
   - release lock (pull down) and then
   - in one motion rotate guard and pull down.

3. Remove bowl as follows: in one motion slowly rotate bowl and then pull straight down. The bowl is ***NOT*** screwed into the top housing.

4. Pour out old desiccant, open new contain and refill bowl. Tap the bowl to settle desiccant. Desiccant should be ½ inch from top of bowl.

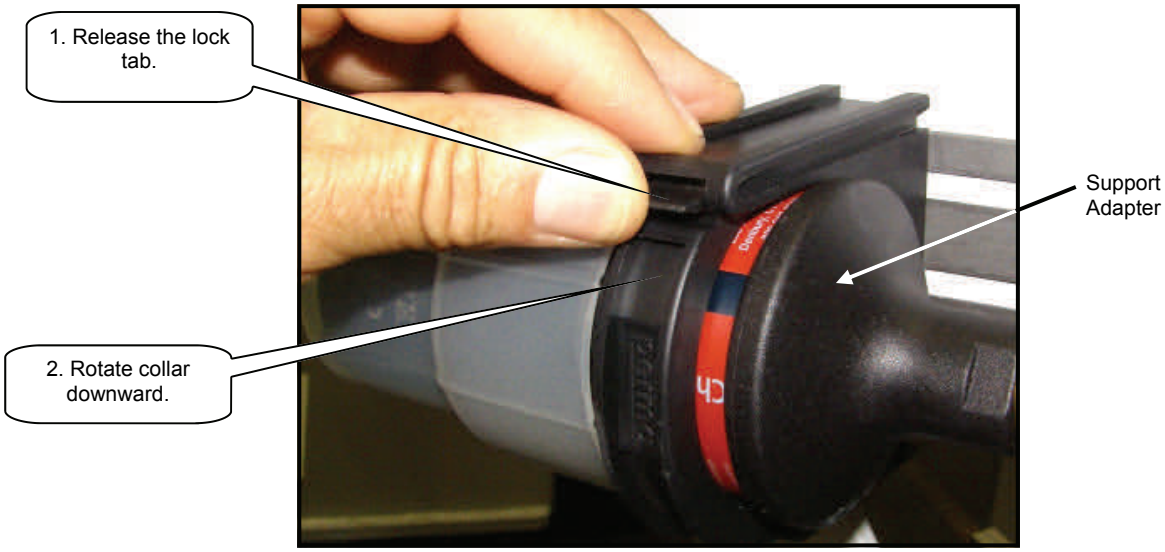


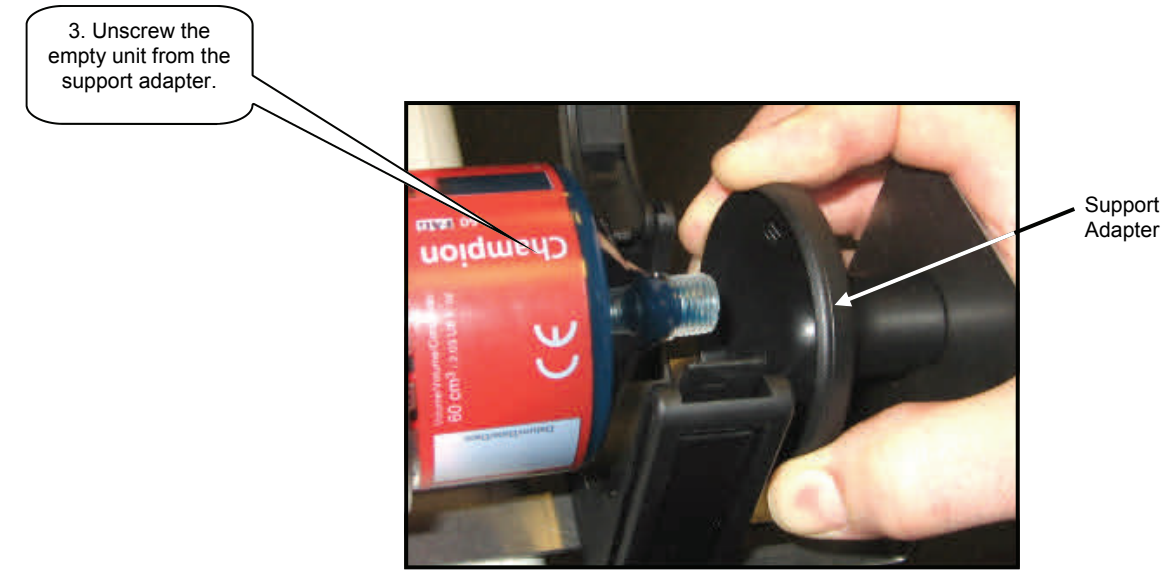
5. Replace the bowl and bowl cover.

6. Restore air to the VEKTOR.

Replacing the FAG Lube Canister/Reservoir

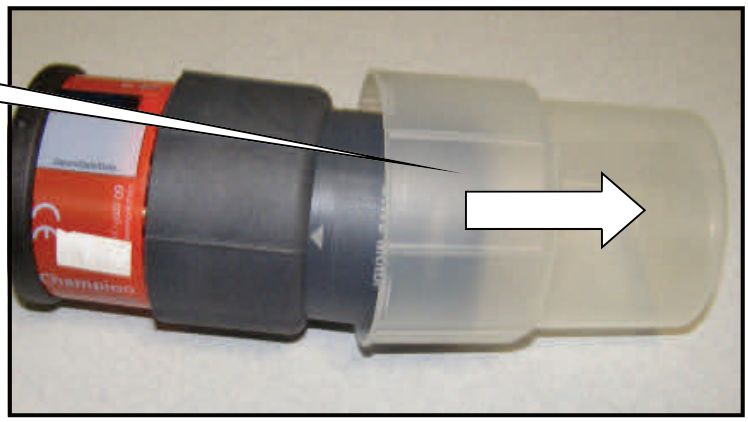
This procedure illustrates the steps to replace the FAG Lube Canister/Reservoir. The part number for the replacement kit is 3162038-1. The kit contains a lube canister/reservoir and battery pack.
NOTE: The battery pack is <u>NOT</u> sold separately.



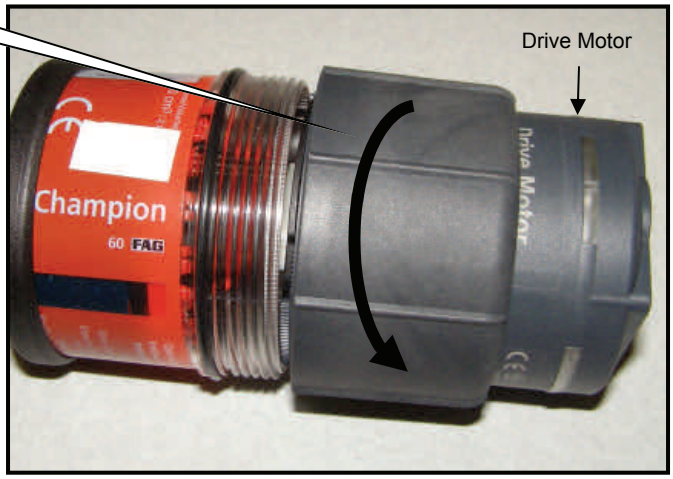


Replacing the FAG Lube Canister/Reservoir (cont'd)

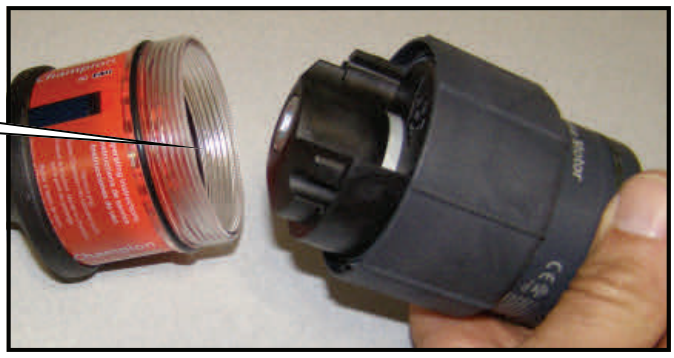
4. Remove outer cover. Pull straight off, do NOT rotate.



5. Unscrew collar. Do NOT rotate drive motor.

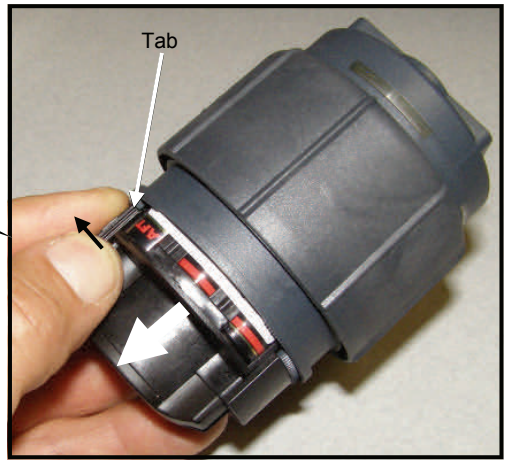


6. Separate drive motor from grease canister.



Replacing the FAG Lube Canister/Reservoir (cont'd)

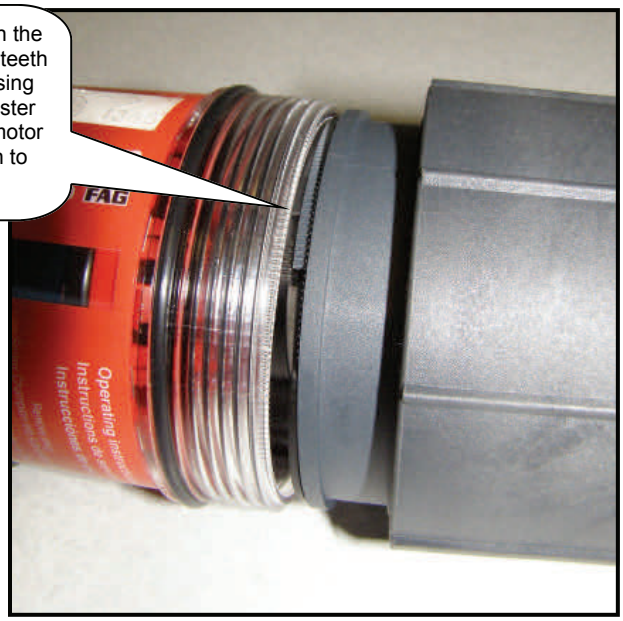
7. To remove battery, push locking tab away from battery and then tilt drive motor.



8. Install new battery pack.

9. Rotate tab to lock battery in place.

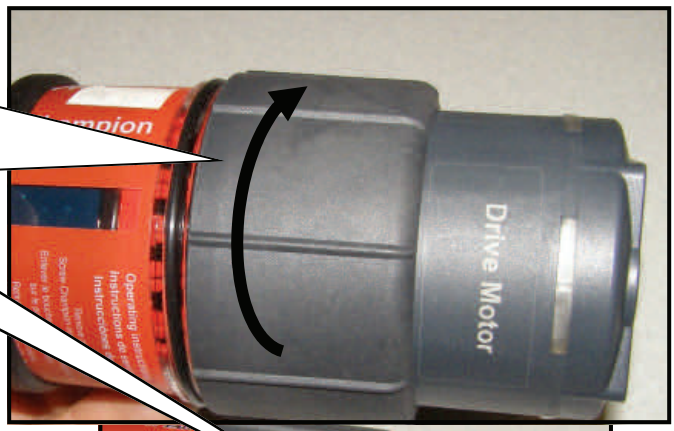
10. Install motor drive on the new canister. Insure the teeth on the drive motor housing mess/lock with the canister teeth. Rotate the drive motor or canister just enough to mess the teeth.

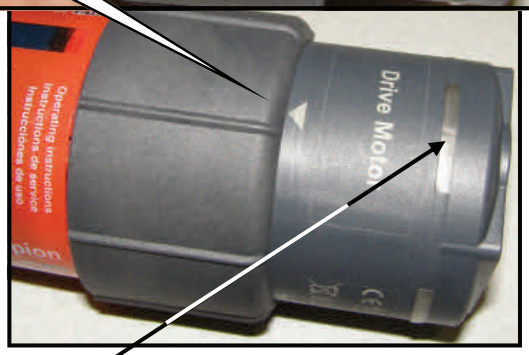


Replacing the FAG Lube Canister/Reservoir (cont'd)

11. Keeping the motor and canister coupled together, tighten the drive motor collar onto the canister. **Do NOT rotate drive motor or canister**.

When tight, the arrow head on the drive motor is fully exposed.



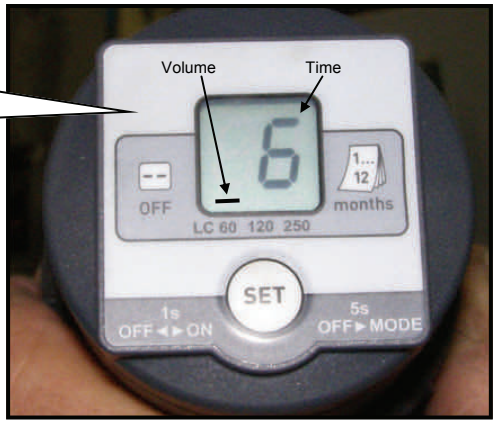


LEDs

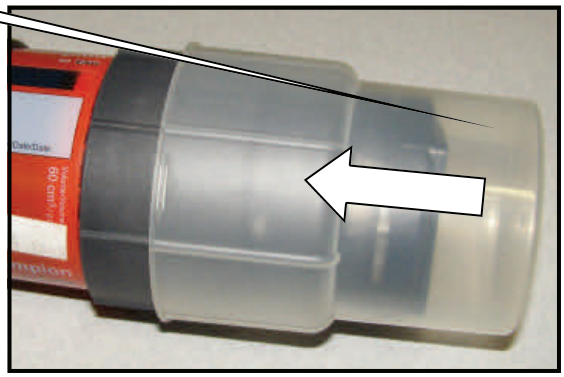
12. Verify the following:

- LCD displays screen time and volume.
- green LEDs flash every 7 seconds. LEDs are on side of drive motor (4 places)

If these conditions do not exist, all Park Customer Support at 1-800-785-3991.

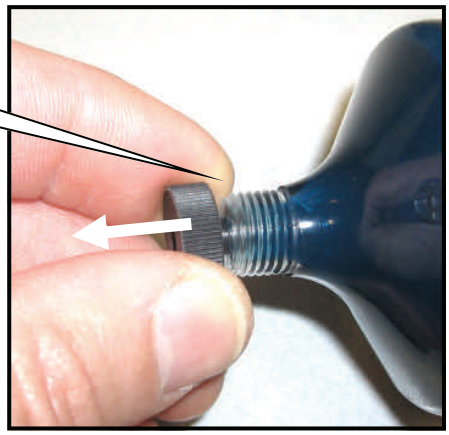


13. Replace cover.



Replacing the FAG Lube Canister/ Reservoir (cont'd)

14. Remove shipping plug. (pull straight out).



15. Screw the unit into the adapter.

16. Close the locking ring.

Filter Replacement Log

Company: _____
Model: _____
Serial Number: _____

REPLACMENT DATE	REPLACMENT DATE	REPLACMENT DATE	REPLACMENT DATE	REPLACMENT DATE

VEKTOR Every 40 Hours or Weekly Preventative Maintenance Log (Sheet 1 of 2)

Month _____ Year _____

Week (Date)	Check the level of grease in the FAG lubricator. See page 5-10.	Blow dust from the air conditioner radiator See page 5-11	On toward side gantry clean and inspect the round roller and round way. Wipe the round way with a light coat of oil. See page 5-12.	On away side gantry clean and inspect the flat roller and flat way. Wipe the flat way with a light coat of oil See page 5-12	Clean and inspect the upper and lower cross travel round ways and bearings Inspect the bearings for seal and retainer damage See page 5-13.
1st	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
2nd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
3rd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
4th	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
5th	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:

VEKTOR Every 40 Hours or Weekly Preventative Maintenance Log (Sheet 2 of 2)

Month _____ Year _____

Week (Date)	Clean, inspect and lubricate spindle bushing sides and O-Ring See page 5-15.				
1st	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
2nd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
3rd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
4th	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
5th	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:

VEKTOR Every 80 Hours or Two Weeks Preventative Maintenance Log

Month _____ Year _____

Week (Date)	Clean, inspect and spray Y Axis (Gantry) rack and pinion drive assemblies See page 5-18.	Clean, inspect and spray X Axis (Cross Travel) rack and pinion drive assemblies See page 19.			
1st	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
2nd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
3rd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:

VEKTOR Every 120 Hours or Monthly Preventative Maintenance Log (Sheet 1 of 1)

ACTION	MONTH: _____	MONTH: _____	MONTH: _____
Clean and inspect the Z Axis rails. Lubricate all four Z Axis bearings. See page 5-20	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:
Lubricate the screw-base bearing. See page 5-21.	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:
Replace the water filter. See page 5-23.	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:
Remove, inspect and lubricate the gripper assembly See page 5-25.	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:
Grease spindle rotary union See page 5-30	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:
	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:

NOTE:

The data on the opposite page is the master for the Filter Replacement Log. Use the page to make additional log sheets.

Filter Replacement Log

Company: _____
Model: _____
Serial Number: _____

REPLACMENT DATE	REPLACMENT DATE	REPLACMENT DATE	REPLACMENT DATE	REPLACMENT DATE

NOTE:

The data on the opposite page is the master for page one of the Weekly or Every 40 hours maintenance check list sheet. Use the page to make additional log sheets.

VEKTOR Every 40 Hours or Weekly Preventative Maintenance Log (Sheet 1 of 2)

Month _____ Year _____

Week (Date)	Check the level of grease in the FAG lubricator. See page 5-10.	Blow dust from the air conditioner radiator See page 5-11	On toward side gantry clean and inspect the round roller and round way. Wipe the round way with a light coat of oil. See page 5-12.	On away side gantry clean and inspect the flat roller and flat way. Wipe the flat way with a light coat of oil See page 5-12	Clean and inspect the upper and lower cross travel round ways and bearings Inspect the bearings for seal and retainer damage See page 5-13.
1st	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
2nd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
3rd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
4th	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
5th	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:

NOTE:

The data on the opposite page is the master for page two of the weekly or every 40 hours maintenance check list sheet. Use the page to make additional log sheets.

VEKTOR Every 40 Hours or Weekly Preventative Maintenance Log (Sheet 2 of 2)

Month _____ Year _____

Week (Date)	Clean, inspect and lubricate spindle bushing sides and O-Ring See page 5-15.				
1st	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
2nd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
3rd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
4th	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
5th	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:

NOTE:

The data on the opposite page is the master for page of the every two week or 80 hours maintenance check list sheet. Use the page to make additional log sheets.

VEKTOR Every 80 Hours or Every two Weeks Preventative Maintenance Log

Month _____ Year _____

Week (Date)	Clean, inspect and spray Y Axis (Gantry) rack and pinion drive assemblies See page 5-18.	Clean, inspect and spray X Axis (Cross Travel) rack and pinion drive assemblies See page 19.			
1st	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
2nd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:
3rd	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:	Yes/No Name: _____ Note:

NOTE:

The data on the opposite page is the master for the Monthly maintenance check list sheet. Use the page to make additional log sheets.

VEKTOR Every 120 Hours or Monthly Preventative Maintenance Log (Sheet 1 of 1)

ACTION	MONTH: _____	MONTH: _____	MONTH: _____
Clean and inspect the Z Axis rails. Lubricate all four Z Axis bearings. See page 5-20.	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:
Lubricate the screw-base bearing. See page 5-21.	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:
Replace the water filter. See page 5-23.	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:
Remove, inspect and lubricate the gripper assembly See page 5-25.	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:
Grease spindle rotary union See page 5-30	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:
	Name Date: Status/Notes:	Name Date: Status/Notes:	Name Date: Status/Notes:

Section 6: General Information

Contents Page

General 6-1
Parts Order Form 6-2
Parts by Phone or Online 6-3
Notes 6-4
Quick Reference Sheet.. 6-11
Technical Drawings 6-12

General

This section contains parts ordering information and Park Technical Bulletins. Technical drawings related to your actual VEKTOR CNC Stone Profiler are found at the end of this section.
.

PARTS ORDER FORM

PO Box 188
St. Cloud, MN 56302

Park Industries®

Telephone: 800-785-3391
Fax: 800-500-5231

BILLING AND SHIPPING INFORMATION

Bill to:
Company: _____
Address: _____
City, State, Zip: _____
Phone: _____
Fax: _____
Authorized By: _____

Ship to (if different):
Company: _____
Address: _____
City, State, Zip: _____
Phone: _____
Fax: _____
Authorized By: _____

SHIPPING INSTRUCTIONS

- ❒ UPS EARLY AM (Next Day by 8:30 am)
- ❒ UPS NEXT DAY (By 10:30 am)
- ❒ UPS NEXT DAY SAVER (By 3:00 pm)
- ❒ UPS 2nd DAY AM (2nd Day by 10:30 am)
- ❒ UPS 2nd DAY AIR (2nd Day by 5:00 pm)
- ❒ UPS 3 DAY (3rd Day by 5:00 pm)
- ❒ UPS GROUND (Varies by location 3-5 days)

BILLING INSTRUCTIONS

- ❒ Invoice — approved credit
- ❒ COD
- ❒ Prepayment
- ❒ Mastercard ❒ Visa

Account Number:_____
Expiration Date:_____

ORDER BLANK

Qty.	Part #	Description	Unit Price	Total

FAX YOUR ORDER TO US 1-800-500-5231

Order your parts by telephone:

1-800-785-3391

This is a direct line to the
Customer Service Department

OR

Order your parts by e-mail:

CustomerService@parkindustries.com

G10014C

Notes:

Notes:

Notes:

Notes:

Notes:

Notes:

Notes:

Quick Reference

Decimal/Metric Converter

INCHES		mm	INCHES		mm
1/64	0.015	0.396	33/64	0.515	13.096
1/32	0.031	0.793	17/32	0.531	13.493
3/64	0.046	1.190	35/64	0.546	13.890
1/16	0.062	1.587	9/16	0.562	14.287
5/54	0.078	1.984	37/64	0.578	14.684
3/32	0.093	2.381	19/32	0.593	15.081
7/64	0.109	2.778	39/64	0.609	15.478
1/8	0.125	3.175	5/8	0.625	15.875
9/64	0.140	3.571	41/64	0.640	16.271
5/32	0.156	3.968	21/32	0.656	16.668
11/64	0.171	4.365	43/64	0.671	17.065
3/16	0.187	4.762	11/16	0.687	17.462
13/64	0.203	5.159	45/64	0.703	17.859
7/32	0.218	5.556	23/32	0.718	18.256
15/64	0.234	5.953	47/64	0.734	18.653
1/4	0.250	6.350	3/4	0.750	19.050
17/64	0.265	6.746	49/64	0.765	19.446
9/32	0.281	7.143	25/32	0.781	19.843
19/64	0.296	7.540	51/64	0.796	20.240
5/16	0.312	7.937	13/16	0.812	20.637
21/64	0.328	8.334	53/64	0.828	21.034
11/32	0.343	8.731	27/32	0.843	21.431
23/64	0.359	9.123	55/64	0.859	21.828
3/8	0.375	9.525	7/8	0.875	22.225
25/64	0.390	9.921	57/64	0.890	22.621
13/32	0.406	10.318	29/32	0.906	23.018
27/64	0.421	10.715	59/64	0.921	23.415
7/16	0.437	11.112	15/16	0.937	23.812
29/64	0.453	11.509	61/64	0.953	24.209
15/32	0.468	11.906	31/32	0.968	24.606
31/64	0.484	12.303	63/64	0.984	25.003
1/2	0.500	12.700	1	1.000	25.400

Technical Drawings:

This page left open intentionally for future additional information.