Mach3 Addons

For Mill

www.NewfangledSolutions.com

User's Manual

Page Intentionally Left Blank

Page Intentionally Left Blank

Table of Contents

Appendix 1 - Operations Screen Definitions

Introduction:

This manual has been assembled in an effort to provide the user with some basic information about the layout, functionality, and overall use of the Mach3 Addons from Newfangled Solutions LLC. It is intended to provide the beginning user with some very basic information that will hopefully help decrease the time necessary to go from "Start to Part". The format this manual follows is simple - we begin with some very basic user interface items, we give some information about each of the wizard operations included, and we give some basic suggestions for material settings. Each wizard operation then has its own chapter where all terms used in that operation are defined.

Basic Operational Flow:
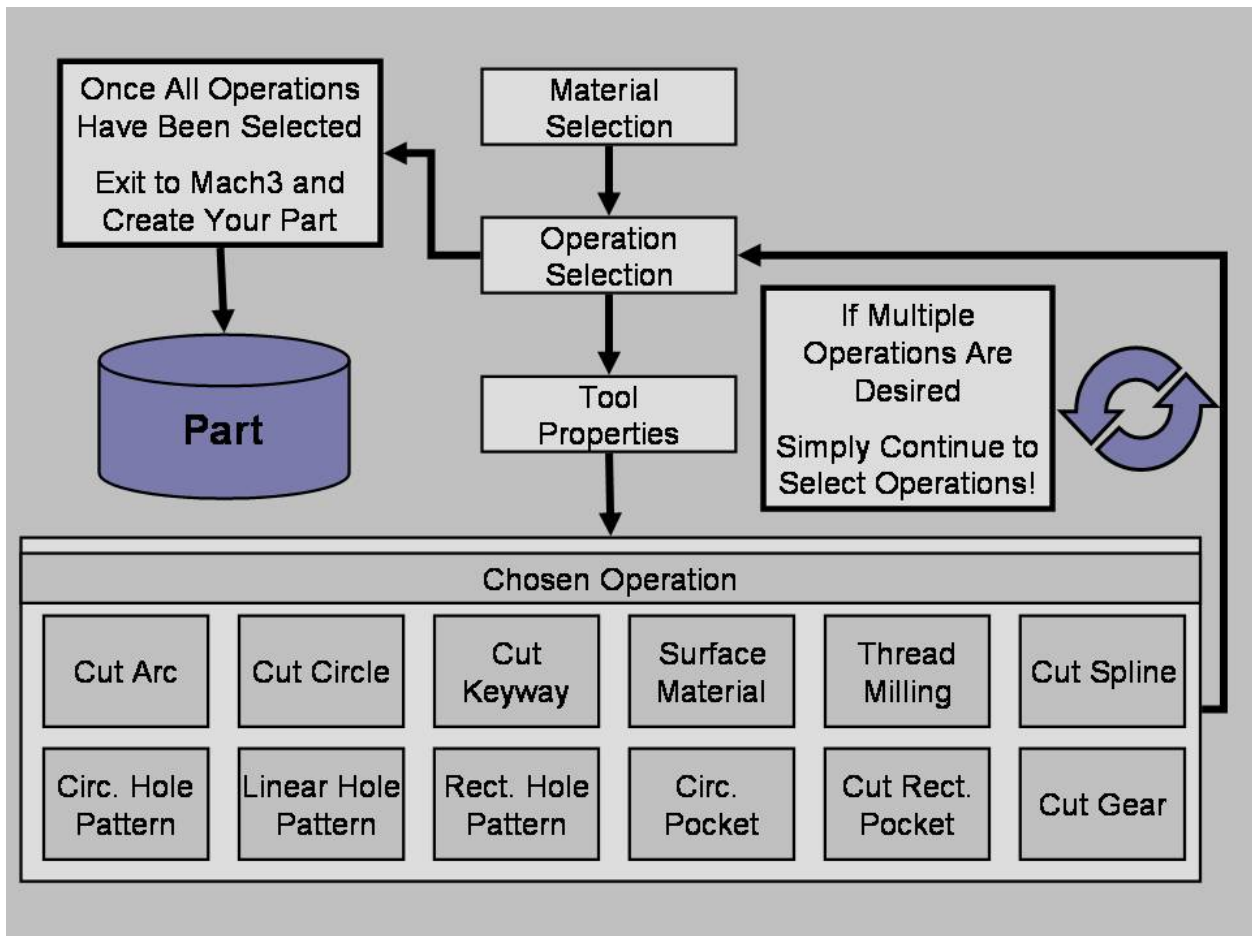
Throughout Mach3 Addons for Mill, the user will be required to enter information. It is this information that will be used for all of the built in functions of the software. Through the course of conducting a selected operation, the user will progress from the initial Material Selection Screen, to the Operation Selection Screen, to the Tool Properties Screen, to the Operation Specific Screen, where the user can **Post Code**. At this point the user will have completed an operation, and could exit to Mach3 to run that operation. However, the user can also continue to select operations allowing the production of more complex parts from a single G-code File. Each operation is clearly differentiated in the created G-code, making this an effective teaching tool, or just easier to modify.

Interface Basics:

Essential values used throughout the Mach3 Addons package are displayed to the user as a Digital Read Out (DRO - See Figure 1 below). At different points, the user will need to provide information necessary for an operation to be completed. When entering such values in this screenset, you must select the DRO you wish to use by positioning the mouse over it and pressing the left mouse button (Left Clicking). You will notice the DRO color change upon selection. After entering the desired value, the user must then press the Enter key.

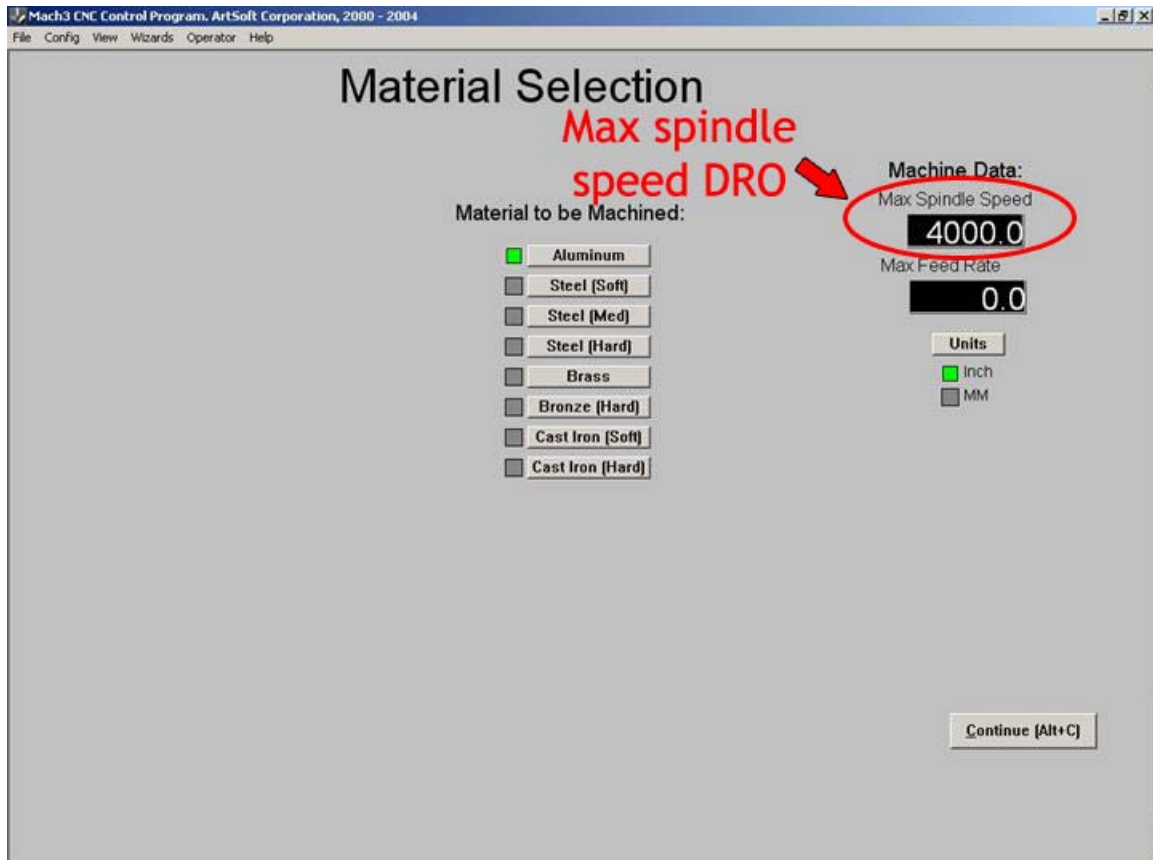**Figure 1: Digital Read Out Example**

If your process requires a finish pass, we recommend leaving yourself some room. Create pockets, holes or inside cuts **slightly undersize**, make your surfacing or outside cuts **slightly oversize** and then simply run the wizard a second time, changing the required settings to your desired finish values. Since your settings should be saved, it's quite simple!

Included Operations:
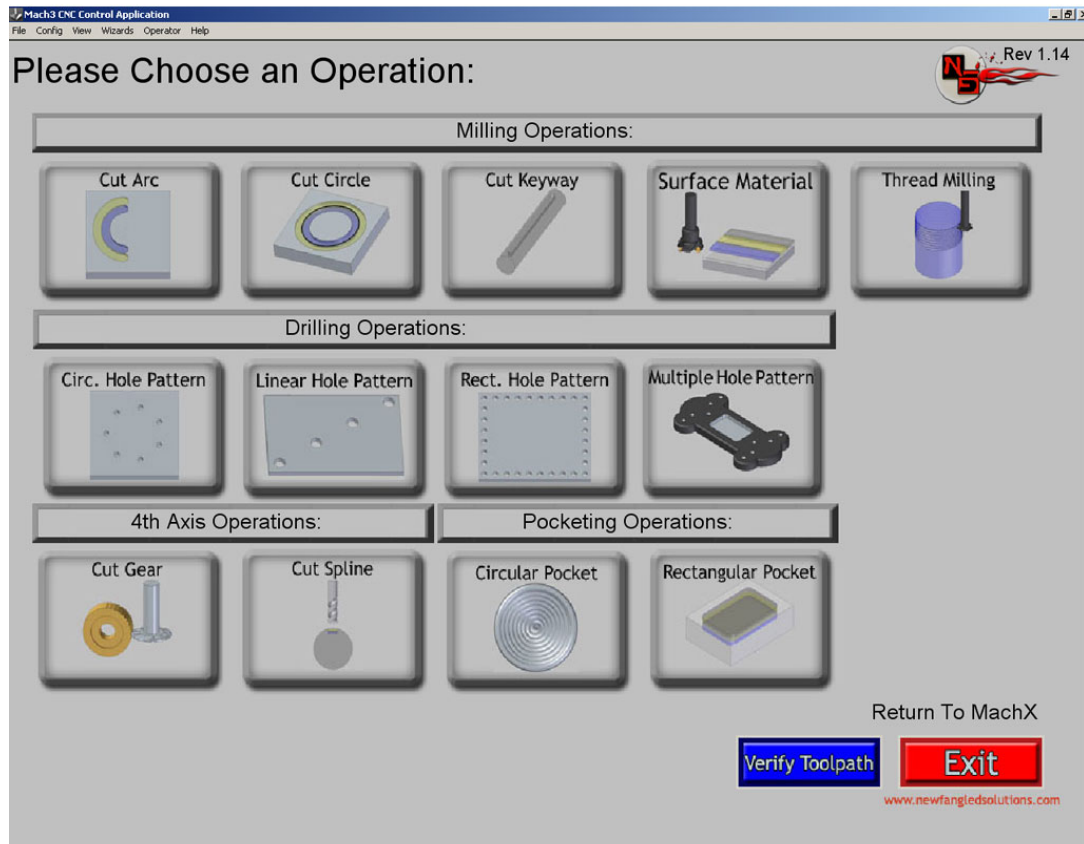
Milling Operations
Cut Arc – Commonly used for fillets, or milling curves
Cut Circle – Commonly used for cutting circles, or circular groves for oil rings etc.
Cut Keyway – Commonly used to mill Keyways in round sock
Surface Material – Commonly used to initially face material before milling
Thread Milling – Commonly used to create internal or external threads on various parts

Hole Patterning Operations
Circular Hole Pattern – Commonly used to position holes along a circular pattern
Linear Hole Pattern - Commonly used to position holes in a linear pattern
Rectangular Hole Pattern - Commonly used to position holes along a rectangular pattern
Multiple Hole Pattern – Commonly used to "drill" in up to 20 predetermined locations

Pocketing Operations
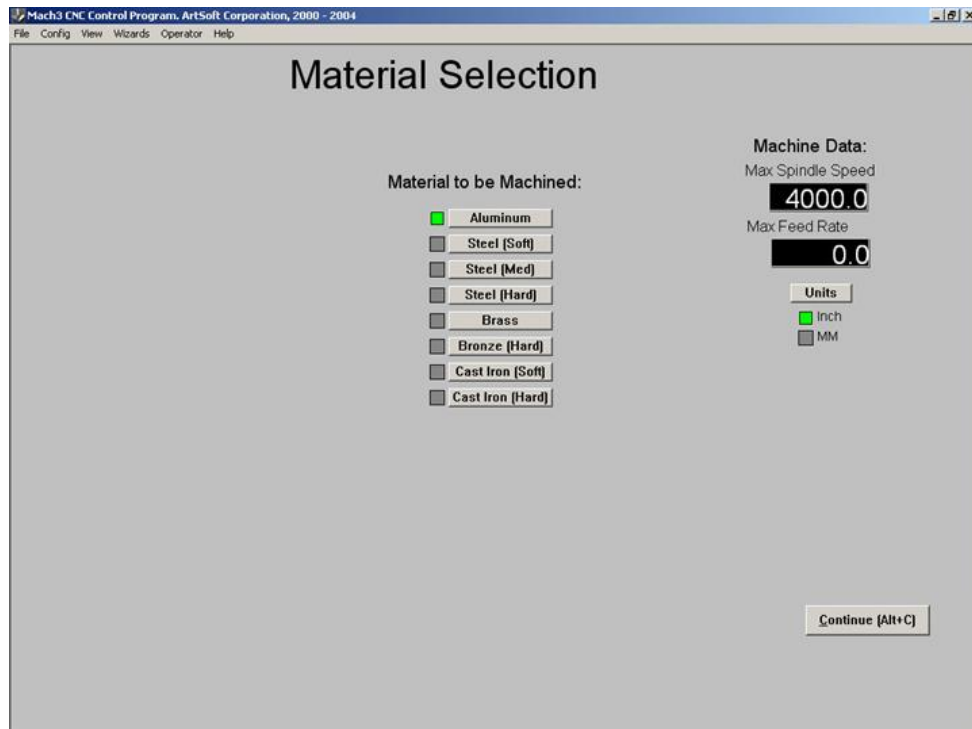**Circular Pocket** – Commonly used to create a circular recessed pocket
Rectangular Pocket - Commonly used to create a rectangular recessed pocket with corner radius of the tool diameter

4th Axis Operations
Cut Gear – Commonly used to cut gears using a gear cutting tool
Cut Spline - Commonly used to create a spline using a spline cutting tool

Material Selection

Machine Data: Max Spindle Speed

Maximum Spindle RPM for the Mill (One Time Entry for Most Users)

Machine Data: Max Feed Rate

Maximum Feed Rate for the Mill (One Time Entry for Most Users)

Current Material Options

Aluminum

Appropriate for general milling, also appropriate for soft metals, wood, and some plastics such as PTFE, PVC, and UHMW

Steel (Soft)

Appropriate for Common Steels such as Hot and Cold Rolled 1018

Steel (Medium)

Appropriate for Harder Steels such as 4140, O1, D2, or other Tool Steels

Steel (Hard)

Appropriate for Hardened Tool Steels, and Stainless Steels

Brass

Appropriate for Brass, medium metals, and many plastics susceptible to melting during cutting such as Polypropylene, Polyethylene, and Acrylic

Bronze (Hard)

Appropriate for Bronze and Coppers

Cast Iron (Soft)

Cast Iron (Hard)

Tool Properties

Calculate Speed and Feed

Select this button to calculate the Speed and Feed values based on user entered information.

Carbide

Carbide Tipped Tooling

Carbide (TiN)

Titanium Nitride Coated Carbide Tooling

Chip Load per Tooth

Desired chip load per cutting tip (Generally accepted as the depth of cut desired for each tip of the cutting tool)

Flood

User selected Flood coolant option

HSS

High Speed Steel

HSS (TiN)

Titanium Nitride Coated High Speed Steel

Mist

User selected Mist coolant option

Number of Cutting Tips

Number of cutting tips on tool to be used for the selected operation

Overrides: Feed %
>   Feed Rate override allowing user to customize output values to their specific equipment. Entered value of 100% or less will decrease Feed Rate to the entered percentage of the otherwise calculated value. (Calculated Feed * override% = new Feed Therefore entering 80% will change a calculated Feed Rate of 14.4 in/min to a value of 11.5 in/Min)

Overrides: Spindle %
>   Spindle speed override allowing user to customize output values to their specific equipment. Entered value of 100% or less will decrease spindle RPM to the entered percentage of the otherwise calculated value. (Calculated RPM * override% = new RPM Therefore entering 80% will change a calculated spindle RPM of 4000 to a value of 3200)

Percent Plunge Feed Rate
>   Plunge Feed Rate override allowing user to customize output values to their specific needs. Entered value of 100% or less will decrease Plunge Feed Rate to the entered percentage of Feed Rate value. (Calculated Feed Rate * override% = new Plunge Feed Rate) Therefore entering 50% will change the Plunge Feed Rate from the Calculated Feed Rate of 14.4 in./min. to a value of 7.2 in./min.)

Special SS Units: Ft./Min.
>   User selected Units for specialized tooling

Special SS Units: Meters/Min.
>   User selected Units for specialized tooling

Speed and Feed: Feed Rate
>   Calculated Feed Rate determined by user entered values

Speed and Feed: Spindle RPM
>   Calculated Spindle RPM (Revolutions per Minute) determined by user entered values

Spindle Direction: CCW (M4)
>   Spindle turns in a clockwise direction

Spindle Direction: CW (M3)
>   Spindle turns in a clockwise direction

Surface Speed for Special Tool
>   User entered speed override enabling the use of specific tool speeds necessary for specialized tooling

Tool #
>   User assigned tool number

Tool Diameter
>   Diameter of tool to be used for the selected operation

Tool Units: Inches
>   User selected Units for general tooling

Tool Units: Millimeters
>   User selected Units for general tooling

Navigating through Operations:
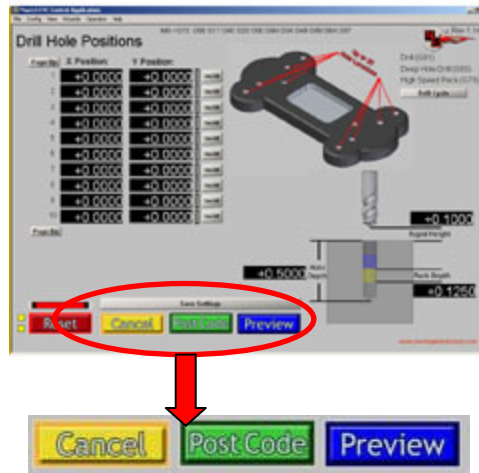
Cancel
> Returns user to Select Operation screen

Post Code
> Appends G code from selected operation to Parent G code file

Preview
> Gives user a view of the toolpath generated from the G code created by the selected operation
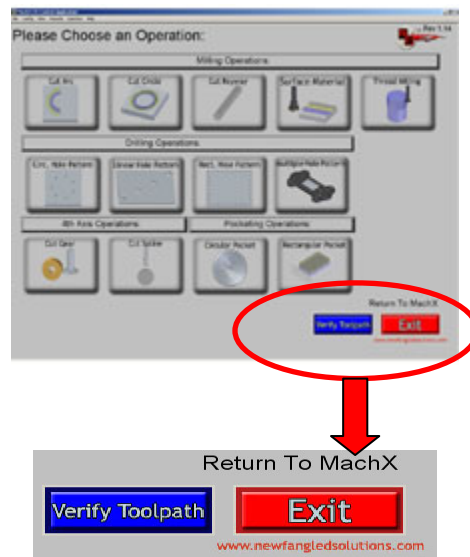
Verify Toolpath
> Gives User a view of the toolpath(s) created by the G code contained in the parent G code file.

Exit
> Returns user to the Run Program screen of Mach3 and loads the parent G code file.

Appendix 1 - Operations Screen Definitions

Milling Operations

Cut Arc

Arc Radius

The radius of the arc to be cut.

Arc Sweep

A value in Degrees, which represents how much of an arc is desired.

Cut Side (Inside)

This selection causes the tool to cut on the inside of the user specified Arc. This makes the outside radius of the cutout the user specified value.
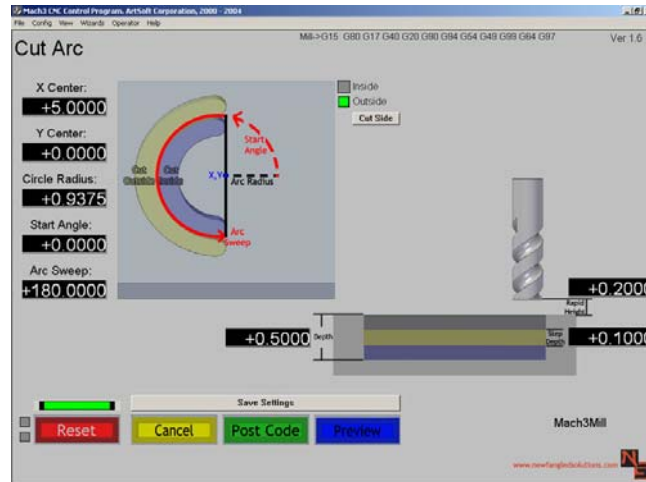
Cut Side (Outside)

This selection causes the tool to cut on the outside of the user specified Arc. This makes the inside radius of the cutout the user specified value.

Rapid Height

Distance between the horizontal surface of the part and the tip of the tool.

Start Angle

The angle created between the defined X axis and the line intersecting the user specified circle where the first hole will be placed.

Step Depth

Depth of material removed per tool pass.

Total Depth

Final depth of cut. (Total amount of material removed after completing all steps or pecks.)

X Center

X value of the Center point (X,Y) of the Arc to be cut.

Y Center

Y value of the Center point (X,Y) of the Arc to be cut.

Cut Circle
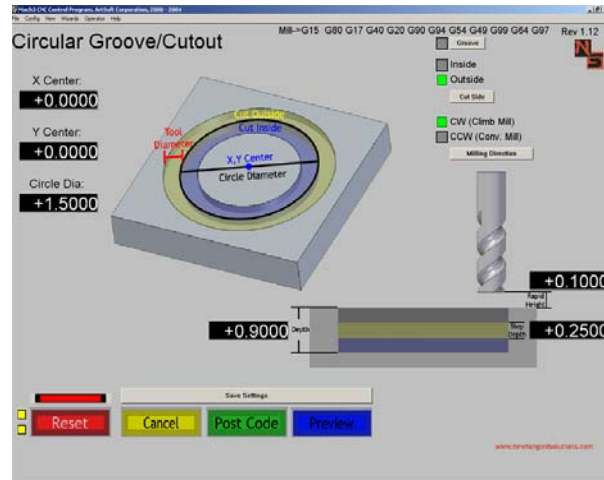
% Stepover
> A percentage of the tool diameter representing the amount of overlap between subsequent tool passes.

CCW Milling Direction (Conventional Mill)
> Counter clockwise tool rotation utilized in conventional milling.

Circle Diameter
> Diameter of the Circle desired. This circle should represent the finished diameter desired, and the appropriate Cut Inside or Cut Outside selection should be made to create the desired result.

Cut Side (Inside)
> This selection causes the tool to cut on the inside of the user specified circle. This makes the outside diameter of the cutout the user specified value.

Cut Side (Outside)
> This selection causes the tool to cut on the outside of the user specified circle. This makes the inside diameter of the cutout the user specified value.

CW Milling Direction (Climb Mill)
> Clockwise tool rotation utilized in climb milling.

Groove
> This selection will make the initial plunge occur in line with the desired grove, resulting in a clean circular groove.

Rapid Height
> Distance between the horizontal surface of the part and the tip of the tool.

Step Depth
> Depth of material removed per tool pass.

Tool Diameter
> User specified Tool Diameter (will represent the width of material removed by this operation).
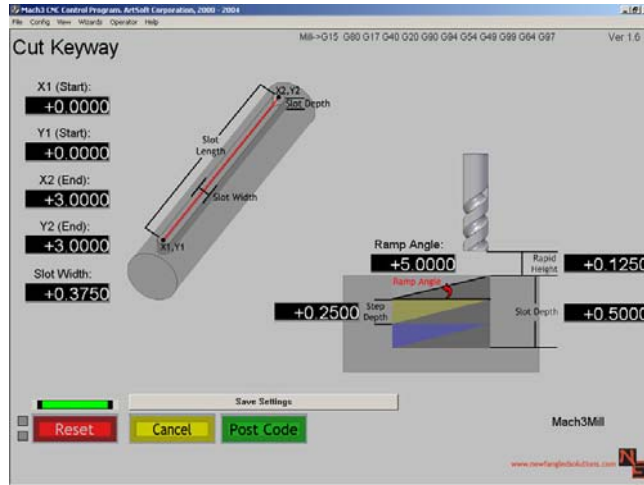
Total Depth
> Final depth of cut. (Total amount of material removed after completing all steps or pecks.)

X Center
> X value of the Center point (X,Y) of the circle to be cut.

Y Center
> Y value of the Center point (X,Y) of the circle to be cut.

Cut Keyway

Ramp Angle

Angle from horizontal which the tool will initially follow at the start of each Step.

Rapid Height

Distance between the horizontal surface of the part and the tip of the tool.

Slot Depth

Distance from material surface to bottom of cut. Represents the amount of material removed.

Slot Width

Width of the material removed.

Step Depth

Depth of material removed per tool pass.

X1 (Start)

Position along the X-axis of the start of the slot.
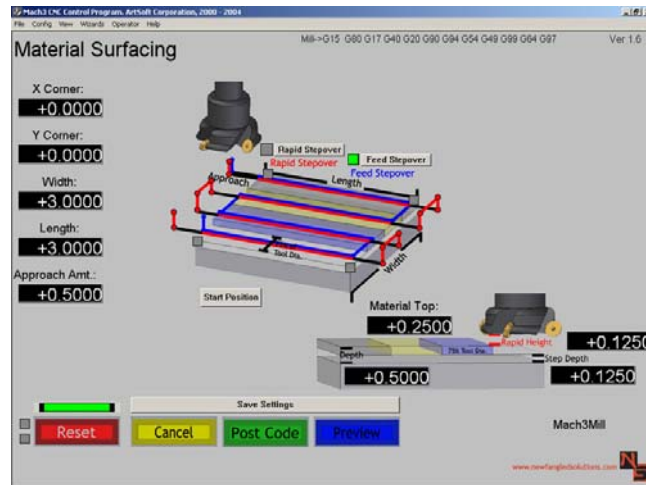
X2 (End)

Position along the Y-axis of the start of the slot.

Y1 (Start)

Position along the X-axis of the end of the slot.

Y2 (End)

Position along the Y-axis of the end of the slot.

Surface Material

Approach Amount

The horizontal distance between the leading edge of the tool and the part surface at the start of the initial surface pass. This value will also be used as the distance the tool will travel after completing a surface pass prior to the rapid move when utilizing the Rapid Stepover option.

Depth

Distance from material surface to bottom of cut. Represents the amount of material removed.

Feed Stepover

Surfacing method where the tool remains in contact with the part surface until each Step has been completed.

Length

The measurement of the extent of something along its greatest dimension.

Material Surface

Location of the material surface.

Rapid Height

Distance between the horizontal surface of the part and the tip of the tool.

Rapid Stepover

Surfacing method where the tool continues for the specified Approach Distance beyond the part prior to making rapid moves which set up the next stepover pass.

Step Depth

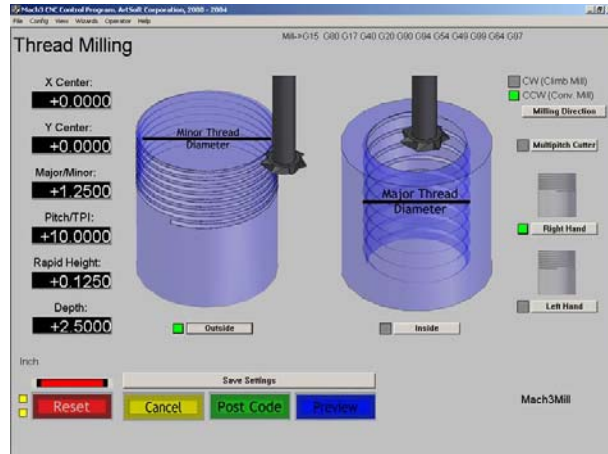Depth of material removed per tool pass.

Width

The measurement of the extent of something along its minor dimension.

X Corner

Location of the X-axis Zero corner.

Y Corner

Location of the Y-axis Zero corner.

Thread Milling

CCW Milling Direction (Conventional Mill)
> Counter clockwise tool rotation utilized in conventional milling.

CW Milling Direction (Climb Mill)
> Clockwise tool rotation utilized in climb milling.

Depth
> Distance from material surface to bottom of cut. Represents the amount of material removed.

Inside Thread
> Selection enabling internal threading.

Left Hand Thread
> Selection enabling the creation of left handed threads. Thread is cut in a clockwise direction.

Major Thread Diameter
> Diameter of the inside thread.

Minor Thread Diameter
> Diameter of the outside thread.

Multipitch Cutter
> Selection allowing the use of multipitch cutters for threading.

Outside Thread
> Selection that allows the creation of Outside threads.

Pitch/TPI
> The thread pitch is the distance between threads expressed in millimeters (measured along the length of the part). TPI is simply a count of the number of threads per inch measured along the length of the part.

Rapid Height
> Distance between the horizontal surface of the part and the tip of the tool.

Right Hand Thread
> Selection enabling the creation of right handed threads. Thread is cut in a counter clockwise direction.

X Center
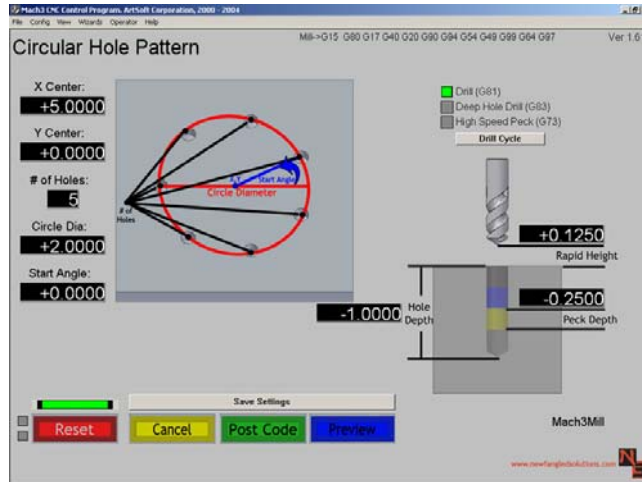> X value of the Center point (X,Y) of the thread to be cut.

Y Center
> Y value of the Center point (X,Y) of the thread to be cut.

Hole Patterning Operations

Circular Hole Pattern

# of Holes
>   Number of Holes you wish to include in the circular pattern.

Circle Diameter
>   Diameter of the circle the holes will be placed along. This circle should represent a circle which passes through the center of the holes desired.

Drill Cycle - Deep Hole Drill (G83)
>   Drill cycle with full retract clearing steps to remove chip buildup inside the hole being drilled.

Drill Cycle - Drill (G81)
>   Drill cycle without clearing steps. Commonly used for shallow holes where chip buildup is a minimal issue, or for spot drilling holes.

Drill Cycle - High Speed Peck (G73)
>   Drill Cycle with numerous clearing steps. Will only break chips, will **Not** do a full retract.

Hole Depth
>   Distance from material surface to bottom of cut. Represents the amount of material removed.

Peck Depth
>   Depth tool will travel before breaking or clearing chips.

Rapid Height
>   Distance between the horizontal surface of the part and the tip of the tool.
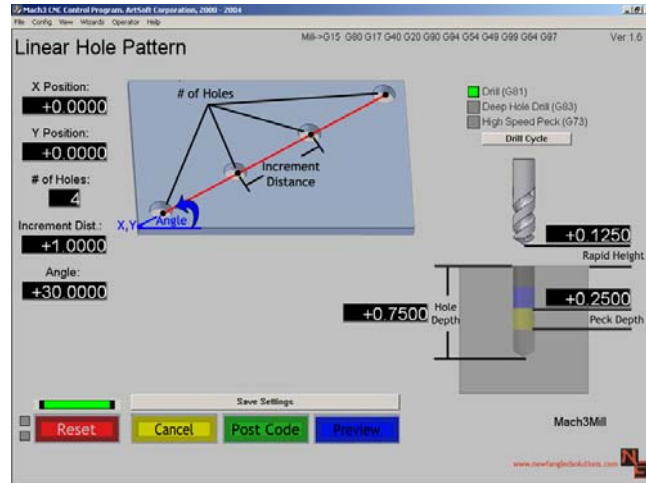
Start Angle
>   The angle created between the defined X axis and the line intersecting the start of the user specified Arc.

X Center
>   X value of the Center point (X,Y) of the pattern to be cut.

Y Center
>   Y value of the Center point (X,Y) of the pattern to be cut.

Linear Hole Pattern

# of Holes

   Number of holes you wish to include in the linear pattern.

Angle

   The angle created between the defined X axis and the desired line the hole(s) will be placed
   along.

Depth

   Distance from material surface to bottom of cut. Represents the amount of material
   removed.

Drill Cycle - Deep Hole Drill (G83)

   Drill cycle with clearing steps to remove chip buildup inside the hole being drilled.

Drill Cycle - Drill (G81)

   Drill cycle without clearing steps. Commonly used for shallow holes where chip buildup is a
   minimal issue.

Drill Cycle - High Speed Peck (G73)

   Drill Cycle with numerous clearing steps. Will only break chips, will **Not** do a full retract.

Increment Distance

   Linear distance between desired hole centers.

Peck Depth

   Depth of tool travel before clearing created chips.
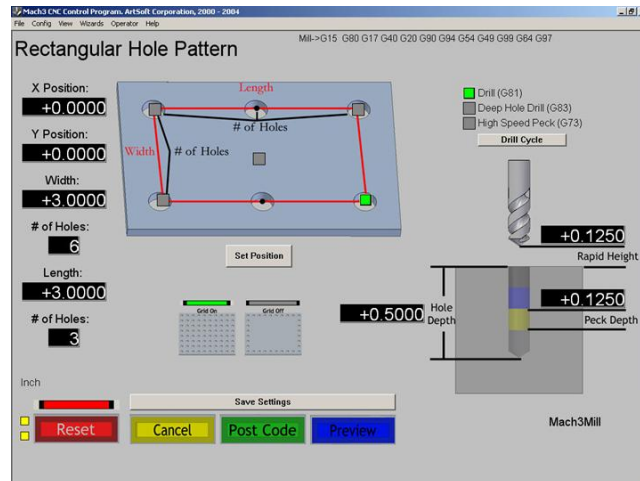
Rapid Height

   Distance between the horizontal surface of the part and the tip of the tool.

X Position

   Location of Starting point along the X-axis.

Y Position

   Location of Starting point along the Y-axis.

Rectangular Hole Pattern

# of Holes (L)

Number of Holes you wish the make along the length of the part.

# of Holes (W)

Number of Holes you wish the make along the width of the part.

Drill Cycle - Deep Hole Drill (G83)

Drill cycle with clearing steps to remove chip buildup inside the hole being drilled.

Drill Cycle - Drill (G81)

Drill cycle without clearing steps. Commonly used for shallow holes where chip builup is a minimal issue.

Drill Cycle - High Speed Peck (G73)

Drill Cycle with numerous clearing steps. Will only break chips, will **Not** do a full retract.

Grid Off

Selection that results in a rectangular pattern of holes around the perimeter of the rectangle.

Grid On

Selection that results in a rectangular matrix of holes throughout the entire rectangle.

Hole Depth

Distance from material surface to bottom of cut. Represents the amount of material removed.

Length

The measurement of the extent of something along its greatest dimension.

Peck Depth

Depth of tool travel before clearing created chips.

Rapid Height

Distance between the horizontal surface of the part and the tip of the tool.

Width

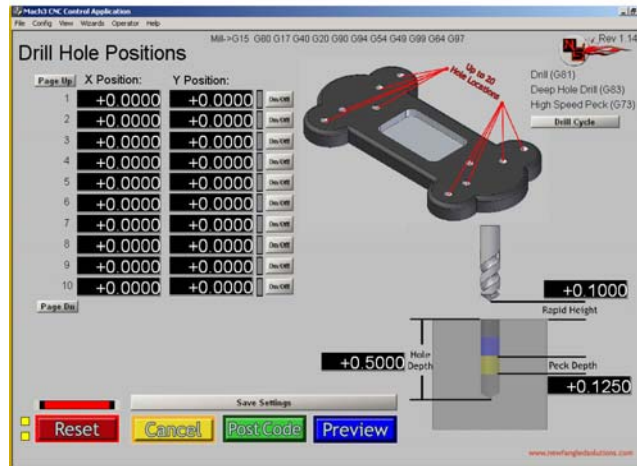The measurement of the extent of something along its minor dimension.

X Position

Location of Set Position point along the X-axis.

Y Position

Location of Set Position point along the Y-axis.

Multiple Hole Position

Drill Cycle - Deep Hole Drill (G83)

Drill cycle with clearing steps to remove chip buildup inside the hole being drilled.

Drill Cycle - Drill (G81)

Drill cycle without clearing steps. Commonly used for shallow holes where chip builup is a minimal issue.

Drill Cycle - High Speed Peck (G73)

Drill Cycle with numerous clearing steps. Will only break chips, will **Not** do a full retract.

Hole Depth

Distance from material surface to bottom of cut. Represents the amount of material removed.

Page Down

Allows user to enter values for hole locations 11-20

Page Up

AllowsUser to enter values for hole locations 1-10

Peck Depth

Depth of tool travel before clearing created chips.

Rapid Height

Distance between the horizontal surface of the part and the tip of the tool.
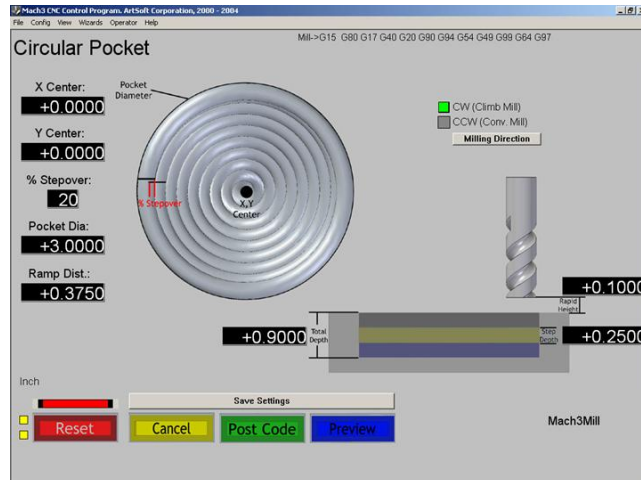
X Position

Location of Hole Position along the X-axis (Up to 20 locations possible per repetition)

Y Position

Location of Hole Position along the Y-axis. (Up to 20 locations possible per repetition)

Pocketing Operations

Circular Pocket

CCW Milling Direction (Conventional Mill)
>   Counter clockwise tool rotation utilized in conventional milling.

CW Milling Direction (Climb Mill)
>   Clockwise tool rotation utilized in climb milling.

Pocket Diameter
>   Outside diameter of the circle created.

Ramp Distance
>   This is the distance to ramp out of the cut. It is like a pull off for each pass.

Rapid Height
>   Distance between the horizontal surface of the part and the tip of the tool.

Step Depth
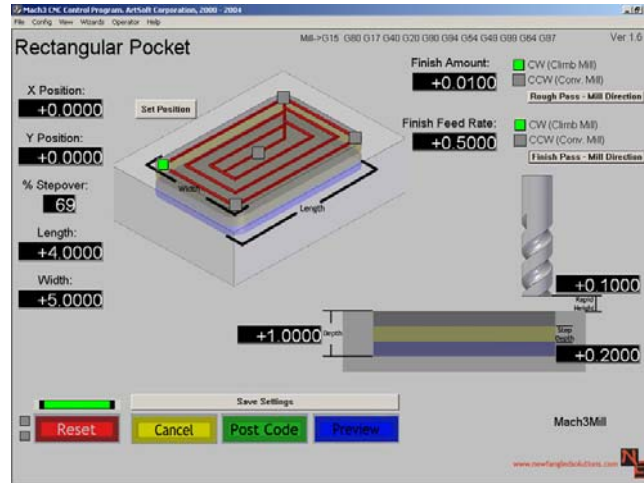>   Depth of material removed per tool pass.

Total Depth
>   Final depth of cut. (Total amount of material removed after completing all steps or pecks.)

X Center
>   X value of the Center point (X,Y) of the circular pocket to be cut.

Y Center
>   Y value of the Center point (X,Y) of the circular pocket to be cut.

Rectangular Pocket

% Stepover

A percentage of the tool diameter representing the amount of overlap between subsequent tool passes.

Length

The measurement of the extent of something along its greatest dimension.

Rapid Height

Distance between the horizontal surface of the part and the tip of the tool.

Step Depth

Depth of material removed per tool pass.

Depth

Distance from material surface to bottom of cut. Represents the amount of material removed.

Width

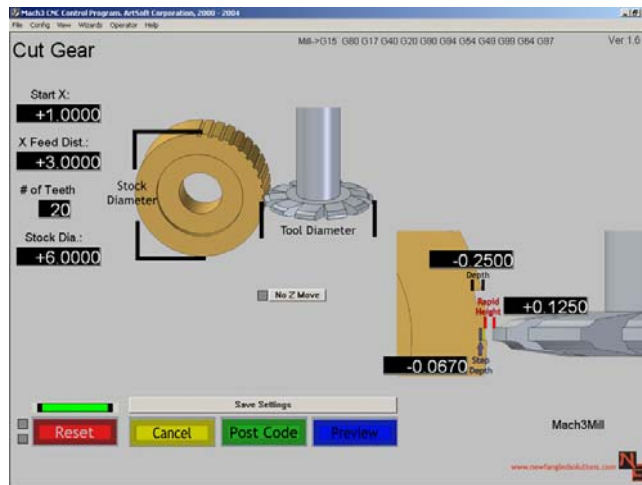The measurement of the extent of something along its minor dimension.

X Position

Location of Set Position point along the X-axis.

Y Position

Location of Set Position point along the Y-axis.

4th Axis Operations

Cut Gear

# of Teeth

Number of teeth the finished part must contain.

Depth

Distance from material surface to bottom of cut. Represents the amount of material removed.

No Z Move

Disable upward tool travel.

Rapid Height

Distance between the horizontal surface of the part and the tip of the tool.

Starting X

Location along the X-Axis where cut should begin.

Step Depth

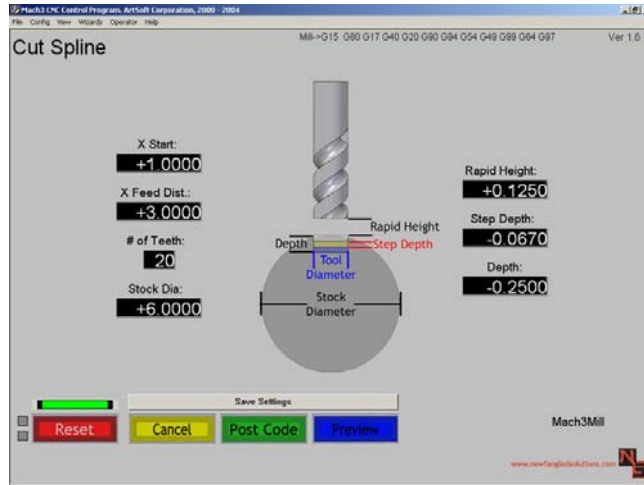Depth of material removed per tool pass.

Stock Diameter

Diameter of initial stock.

Tool Diameter

Previously specified Tool Diameter.

X Feed Distance

Length of cut.

Cut Spline

of Teeth
Number of teeth the finished part must contain.
Depth
Distance from material surface to bottom of cut. Represents the amount of material removed.
Rapid Height
Distance between the horizontal surface of the part and the tip of the tool.
Starting X
Location along the X-Axis where cut should begin.
Step Depth
Depth of material removed per tool pass.
Stock Diameter
Diameter of initial stock.
Tool Diameter
Previously specified Tool Diameter (will represent the width of material removed by this operation)
X Feed Distance
Length of cut.