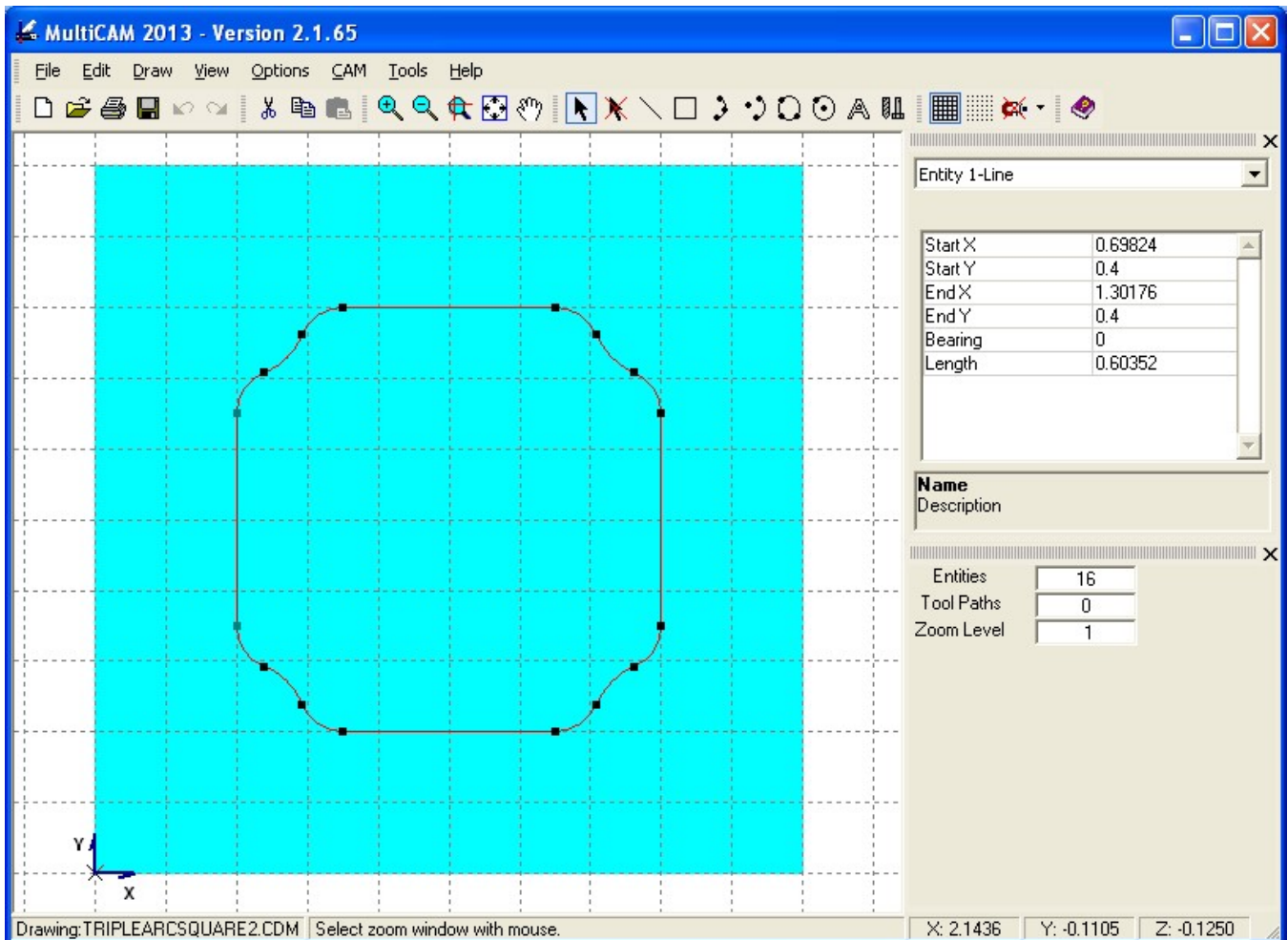# MultiCAM™ 2013

Computer Aided Manufacturing Software

For Microsoft Windows® Based PC's
Version 2.1.xx

Contents Quick Jump

Installing *MultiCAM™ 2013*

** Exit ALL programs before installing **

From the Windows Desktop:

1) For *Windows XP™*, *Vista™* and future *Windows* releases, log in as an **Administrator**. If you are logged in as a **Standard User**, some files may not copy correctly.

2) Insert the *MultiCAM™ 2013* installation CD. Depending on your PC's *AutoPlay* settings, the following screen may appear.
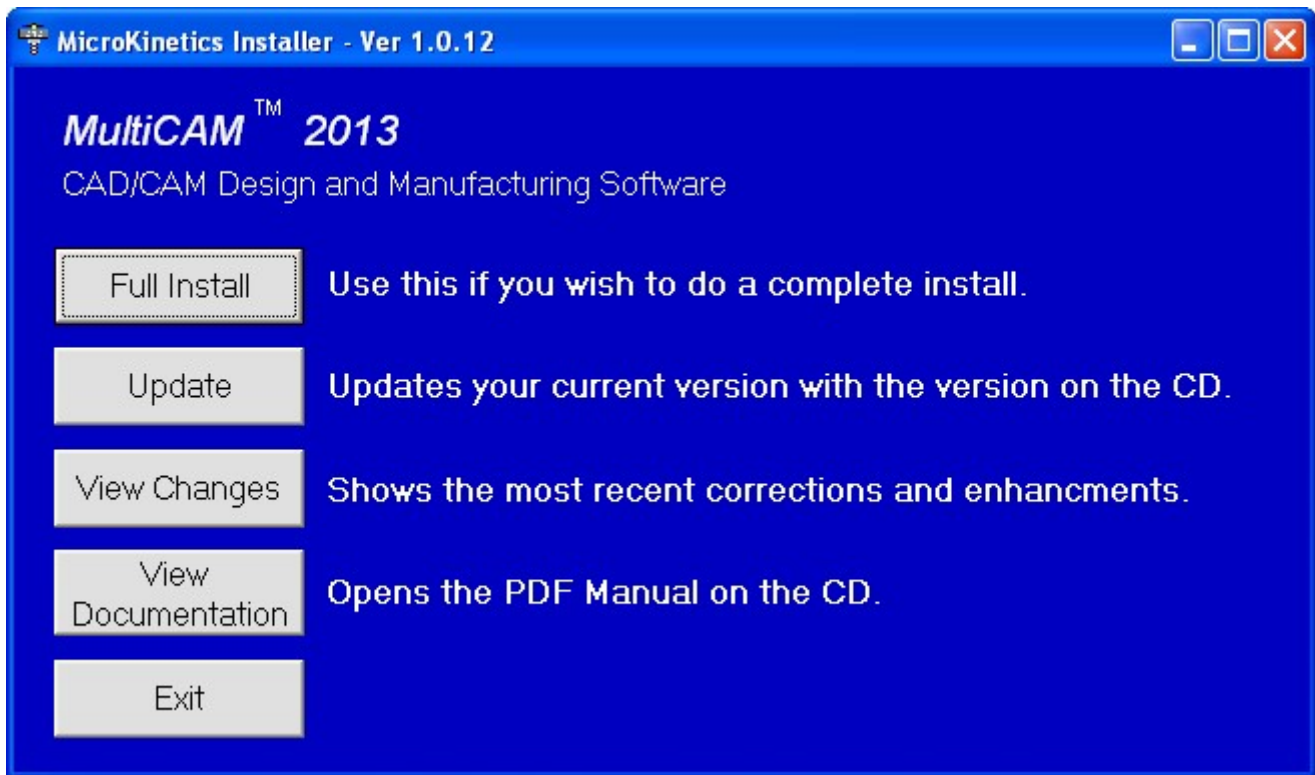

If you see an *AutoPlay* screen with a list of choices, choose *Run MK_install.exe* to see this screen.

If nothing happens when you insert the installation CD, open *Windows Explorer* manually and double-click *MK_Install.exe* to see the this screen. Alternatively, you can double-click *Mcfw_Install.exe* or *Mcfw_Update.exe* and continue to **Step 4**.

Note: If you are installing the latest update from the *MicroKinetics* website, navigate to http://www.microkinetics.com/mcfw and follow the instructions.

3) You may view the documentation or install the software. If you choose *Full Install* or *Update*, continue to **Step 4**. Otherwise press the *Exit* button to close the installation program.

4) The following screen (or a similar screen if **Update** was selected) will appear:
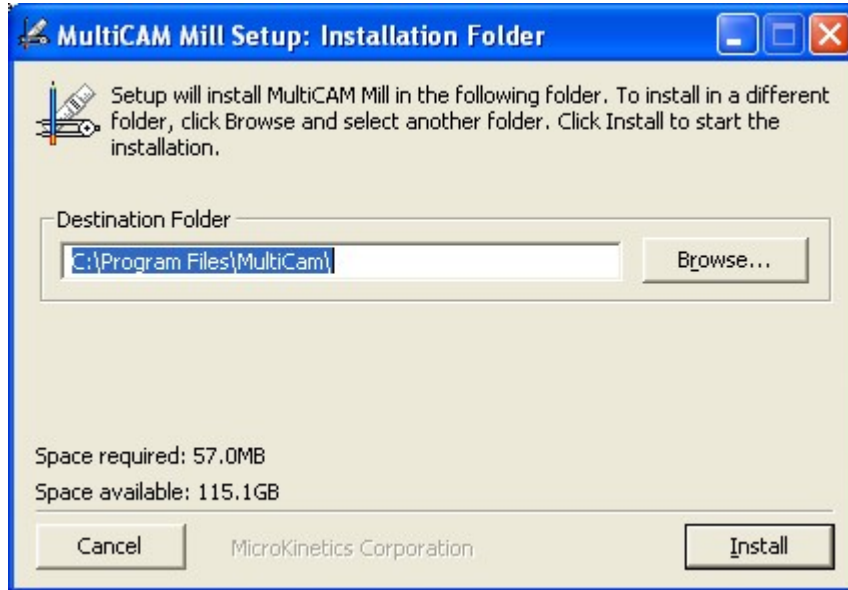
You may accept the default Destination Folder, or select a different one.

Click the **Install** button. This starts the auto-installation process.

5) After the installation process is completed, the following screen will appear:
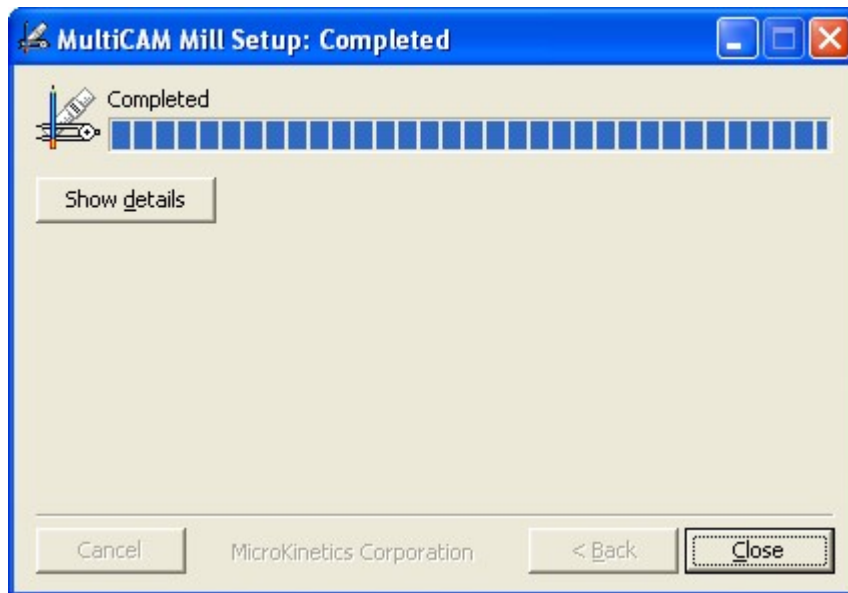
MultiCAM™ 2013 is now installed on your hard drive.

NOTE: If you experience any difficulties or errors when installing, write down detailed information regarding the problem, including any cryptic error information, before calling Tech Support. This will enable us to solve your problem as efficiently as possible.

Selecting Pull Down Menus

Many of the menu items are self explanatory because they follow established standards. For most users, commands such as *File | Open* and *File | Exit* do not need explanation. This help document covers commands specific to *MultiCAM™ 2013* and how to use them to operate the software successfully.
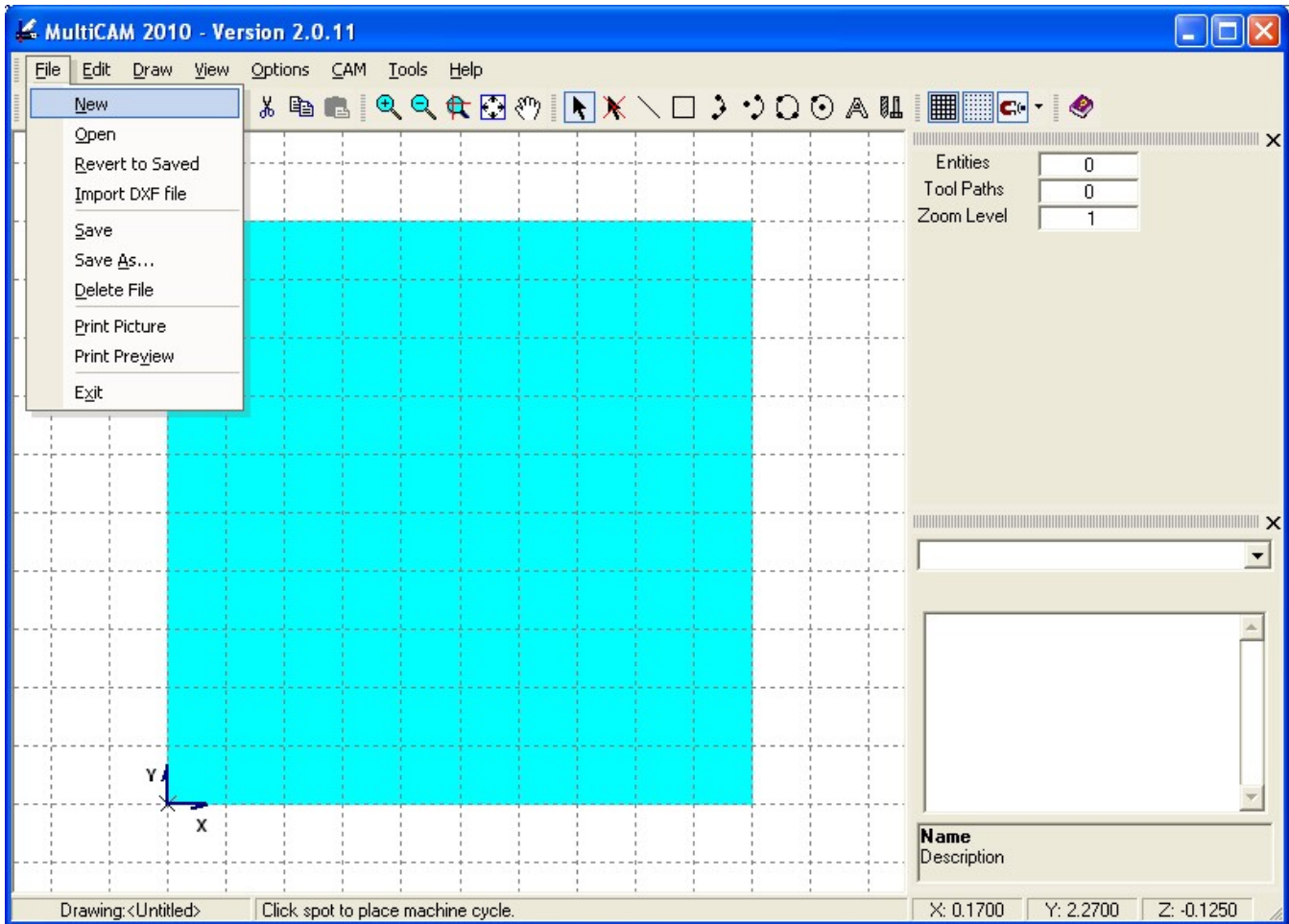
Here's the tool bar. Placing the mouse pointer over these will provide a 'mouse over tip' that gives an indication of what each icon does.

Menu Reference

File Menu

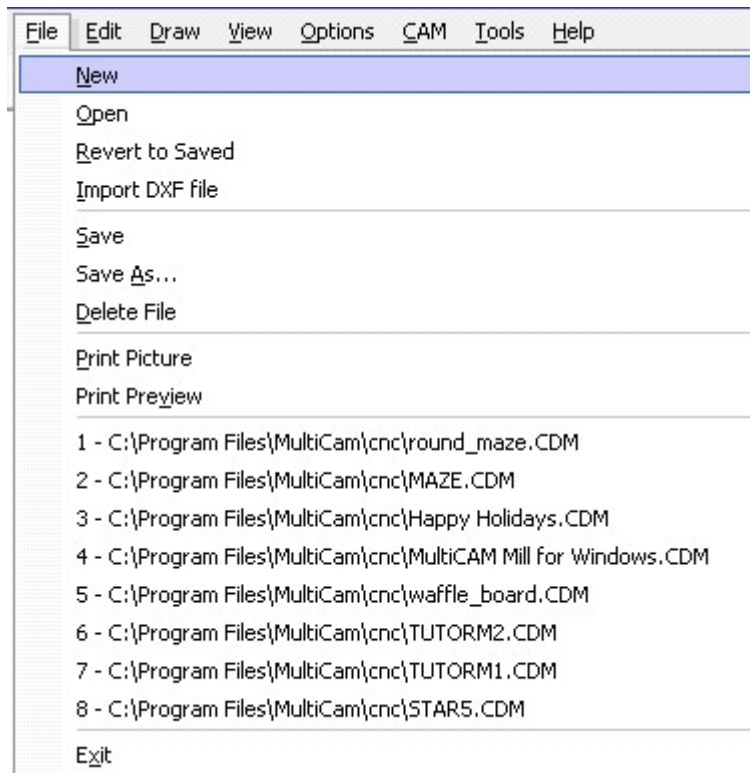| File	Edit	Draw	View	Options	CAM	Tools	Help

New

Open

Revert to Saved

Import DXF file

Save

Save As...

Delete File

Print Picture

Print Preview

1 - C:\Program Files\MultiCam\cnc\round_maze.CDM

2 - C:\Program Files\MultiCam\cnc\MAZE.CDM

3 - C:\Program Files\MultiCam\cnc\Happy Holidays.CDM

4 - C:\Program Files\MultiCam\cnc\MultiCAM Mill for Windows.CDM

5 - C:\Program Files\MultiCam\cnc\waffle_board.CDM

6 - C:\Program Files\MultiCam\cnc\TUTORM2.CDM

7 - C:\Program Files\MultiCam\cnc\TUTORM1.CDM

8 - C:\Program Files\MultiCam\cnc\STAR5.CDM

Exit

File- *New*

Clears the part program in memory and establishes the standard defaults for a new drawing.

The icon in the Toolbar also corresponds to this menu command.

File- *Open*

Displays the Open File Dialog Window which displays all drawing files in the current directory.

The icon in the Toolbar also corresponds to this menu command.

File- *Revert to Saved*

Reverts to the last saved version of the current drawing file.

File- *Import DXF file*

This menu command allows you to import industry standard **.DXF** files up to **Version 15 (AutoCAD 2000 format)**.

Only the following entities are supported for conversion to G-code:

- POLYLINE
- LINE
- ARC
- CIRCLE
- POINT

NOTE: *AutoCAD* does true circular interpolation, whereas *Corel Draw* breaks circles and arcs into chords and is therefore not as smooth and continuous when machining. Also, this function does not accept **.DXF** files that are in 3D format or that have tables enabled.

When this function is selected, an **Open File** dialog box will appear. Select the **.DXF** file of your choice and click 'Open'. A new window will appear with a large viewport (image below). This is the **DXF viewing conversion window**. At the bottom of the window you will notice two command buttons. Select **Import to MCAM** to load the drawing in *MultiCAM™ 2013*.
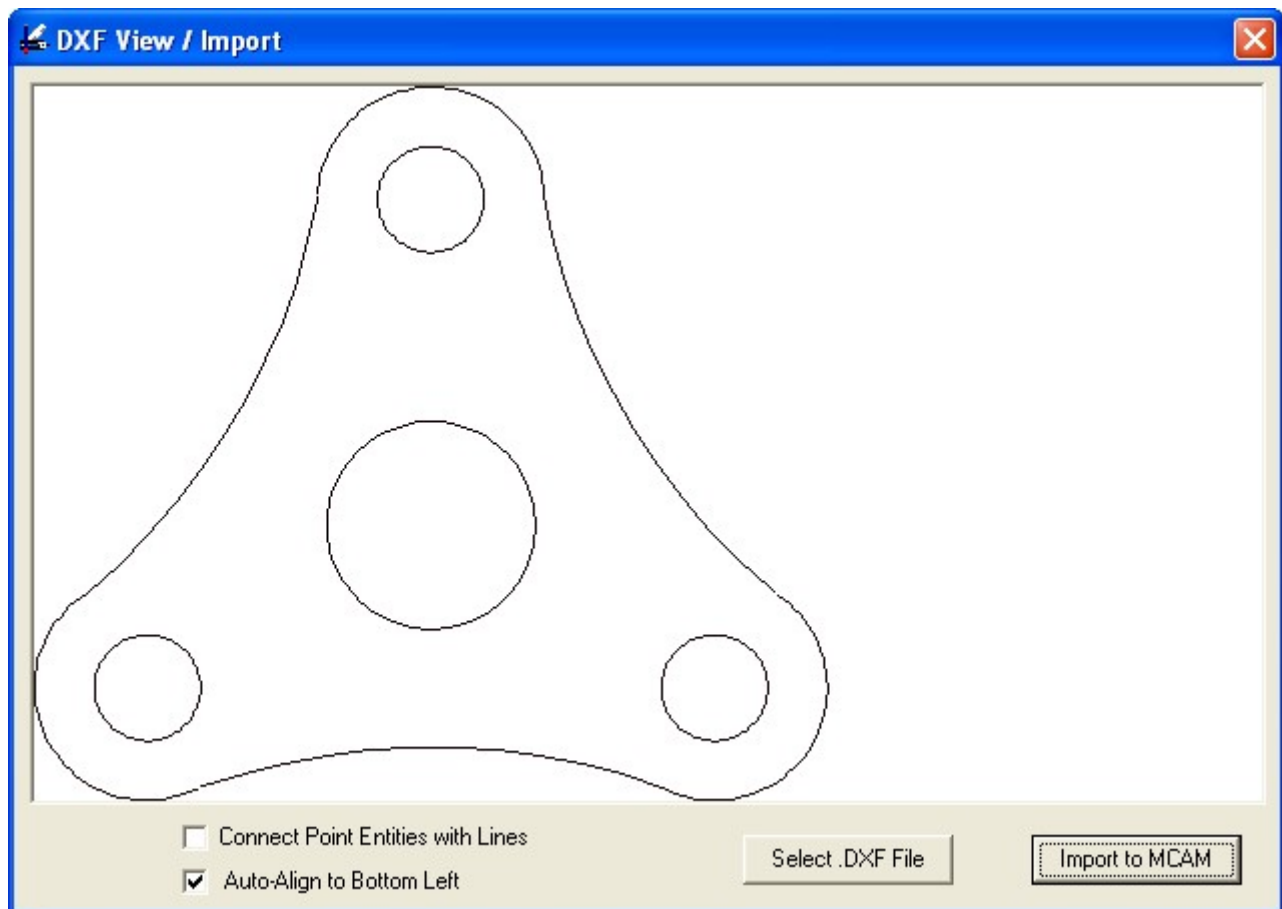
File- *Save*

Quickly saves the current drawing to the current file folder. The extension **.CDM** is automatically assigned.   (If tool paths have been placed, those are saved to a parallel file that has the same base filename with the extension **.TPM**).

The 💾 icon in the Toolbar also corresponds to this menu command.

File- *Save As*

Displays the Save File Dialog Window, which Saves the current drawing to disk allowing the user to rename the file. The file extension **.CDM** is automatically assigned. .   (If tool paths have been placed, those are saved to a parallel file that has the same base filename with the extension **.TPM**).

File- *Delete File*

Delete a drawing/tool path file.

File- *Print Picture*

Prints the current viewport.

The 🖨 icon in the Toolbar also corresponds to this menu command.

File- *Print Preview*

Graphically previews the current viewport before printing.
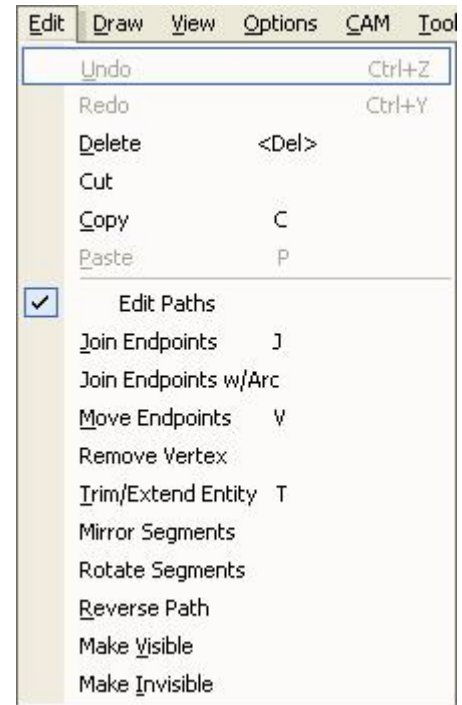

File- *1-9*

Most recently used (MRU) list. Up to 9 recent drawing files are displayed here. Clicking one of these entries will open the corresponding drawing file.

File- *Exit*

Exits **MultiCAM™ 2013** and returns to the Windows desktop.

Edit Menu

Edit	Draw	View	Options	CAM	Tool
Undo					Ctrl+Z
Redo					Ctrl+Y
Delete			<Del>		
Cut					
Copy			C		
Paste			P		
Edit Paths					
Flip Arc			F		
Rev. Arc Dir			D		
Join Endpoints			J		
Join Endpoints w/Arc					
Add Fillet					
Move Endpoints			V		
Break Segment			B		
Remove Vertex					
Change Radius					
Move Profile			M		
Set Origin					
Trim/Extend Entity			T		
Enter Position			E		
Entity Info					Ctrl+I
Scale Drawing			S		
Mirror Segments					
Rotate Segments					
Reverse Path					

Edit	Draw	View	Options	CAM	Tool
Undo					Ctrl+Z
Redo					Ctrl+Y
Delete			<Del>		
Cut					
Copy			C		
Paste			P		
✓ Edit Paths					
Join Endpoints			J		
Join Endpoints w/Arc					
Move Endpoints			V		
Remove Vertex					
Trim/Extend Entity			T		
Mirror Segments					
Rotate Segments					
Reverse Path					
Make Visible					
Make Invisible					

Edit - *Undo - <Ctrl> Z*

Reverses the last change made to the drawing file.

The ↶ icon in the Toolbar, and **<Ctrl> Z** on the keyboard also correspond to this menu command.

Edit - *Redo - <Ctrl> Y*

Brings back what was changed with most **Undo** command.

The ↷ icon in the Toolbar, and **<Ctrl> Y** on the keyboard also correspond to this menu command.

> **Note:** *Undo* and *Redo* only apply to drawing entities. They will not undo or redo any changes to path segments placed. Use the CAM menu to modify path segments.

Edit - *Delete*

Deletes the currently selected drawing entities or path segments (depending on the value of the **Edit Paths** checkbox).

If no drawing entities or drawing segments are selected, clicking in the edit window will delete the nearest drawing entity or path segment.

Pressing the **<Del>** key on the keyboard also corresponds to this menu command.

Edit - *Cut*

Cuts the currently selected drawing entities or path segments (depending on the value of the **Edit Paths** checkbox) and places them onto the clipboard.

The ✂ icon in the Toolbar also corresponds to this menu command.

Edit - *Copy*

Copies the currently selected drawing entities or path segments (depending on the value of the **Edit Paths** checkbox) onto the clipboard.

The ▤ icon in the Toolbar, and the **C** key on the keyboard also correspond to this menu command.

Edit - *Paste*

Pastes the current drawing entities or path segments from the clipboard to the current cursor location.

The ▤ icon in the Toolbar, and the **P** key on the keyboard also corresponds to this menu command.

Edit - *Edit Paths*

This is a checkbox field. If checked, the edit menu commands work only on *Path Segments*. If unchecked, the menu works with *Drawing Entities*.

> **Note:** The edit menu changes appearance depending on the value of the checkbox. There are significantly more functions available for *Drawing Entities*.

> **Note:** For most of the following functions, one or more mouse clicks or mouse drags are required to complete the function. Instructions will be given for each step in the **Status Line** at the bottom of the screen. Pressing **<Esc>** at any time before the function is complete will cancel the operation.

Edit - *Flip Arc*

After selecting this option, click near the arc you want to flip. The arc will be converted to its mirror image along an axis between the endpoints of the arc.

The **F** key on the keyboard also corresponds to this menu command.

Edit - *Reverse Arc Direction*

After selecting this option, click near the arc you want to reverse. The arc will be converted to its complement, that is, an arc which if added to the first arc, would complete a circle.

The **D** key on the keyboard also corresponds to this menu command.

Edit - *Join Endpoints*

This option will draw a line between two endpoints. After selecting this option, click near the first endpoint, then click near the second endpoint. A line will be drawn connecting these two endpoints.

The **J** key on the keyboard also corresponds to this menu command.

Edit - *Join Endpoints w/Arc*

This option will draw an arc between two endpoints. After selecting this option, click near the first endpoint, then click near the second endpoint, then again at a third point on the desired arc. An arc will be drawn connecting these two endpoints.

Edit - *Add Fillet*

This option adds a *Fillet*, or arc between two joined line segments. The user selects 2 joined line segments and the radius of an arc. An arc of the specified radius is placed, connecting the 2 line segments and the line segments are trimmed as needed.

Edit - *Move Endpoints*

This option will move one or more endpoints. After selecting this option, click near an endpoint and drag it to a new location. All segments or paths with that endpoint will be changed to reflect the new endpoint.

The **V** key on the keyboard also corresponds to this menu command.

Edit - *Break Segment*

This function will break one drawing segment into two segments at its midpoint. After selecting this option, click near the segment you want to break.

The **B** key on the keyboard also corresponds to this menu command.

Edit - *Remove Vertex*

This function will merge two drawing segments into one segment by deleting the joining vertex. After selecting this option, click near the vertex you want to delete.

Edit - *Change Radius*

Changes the radius of an arc. After selecting this option, click near the arc you want to modify. An input box will appear with the current radius of the circle. Enter the new radius and click **<Accept>**.

Edit - *Move Profile*

Moves all drawing entities relative to the current (X, Y) position. After selecting this option, click on a position relative to the current (X, Y).

The **M** key on the keyboard also corresponds to this menu command.

> **Note:** The current (X, Y) position is indicated by a crosshairs or X on the drawing screen.

Edit - *Set Origin*

Similar to **Move Profile**, except movements are relative to the origin (0, 0), and the movement of the entities will appear to be reversed since we are moving the origin and not the segments.

Edit - *Trim/Extend Entity*

Trims or extends a drawing entity where it intersects (or would intersect) another entity. After selecting this option, click near the drawing entity you want to trim or extend. Then click near the trimming or extending entity.

If the entities intersect, the first entity will be trimmed at the intersection point. If the entities do not intersect, then the first entity will be extended to intersect the second entity if possible.

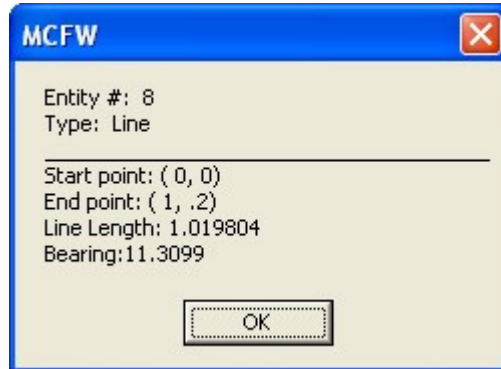The **T** key on the keyboard also corresponds to this menu command.

Edit - *Enter Position*

Moves the cursor to a specific position.

The **E** key on the keyboard also corresponds to this menu command.

Edit - *Entity Info - <Ctrl> I*

Displays information about a specific entity. After selecting this option, click near a drawing entity. A screen similar to the following will appear:

```
MCFW                                    ☒

  Entity #: 8
  Type: Line
  _____
  Start point: ( 0, 0)
  End point: ( 1, .2)
  Line Length: 1.019804
  Bearing:11.3099

                  [    OK    ]
```

<**Ctrl**> **I** on the keyboard also corresponds to this menu command.

Edit - *Scale Drawing*

This function will scale all drawing entities (or selected entities). After selecting this option the following screen will appear:

```
Scale Drawing Multiplier

   Enter Value: [1.0000    ]

   ☐  Scale Selected Entities Only ?

      [ Cancel ]        [   OK   ]
```
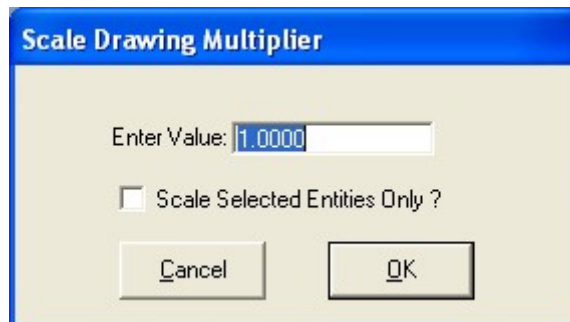
Enter the scaling value. If the "*Scale Selected Entities Only ?*" checkbox is checked, then only selected entities will be modified. If it is not checked, all drawing entities will be modified. Press *<OK>* to continue or *<Cancel>* to quit.
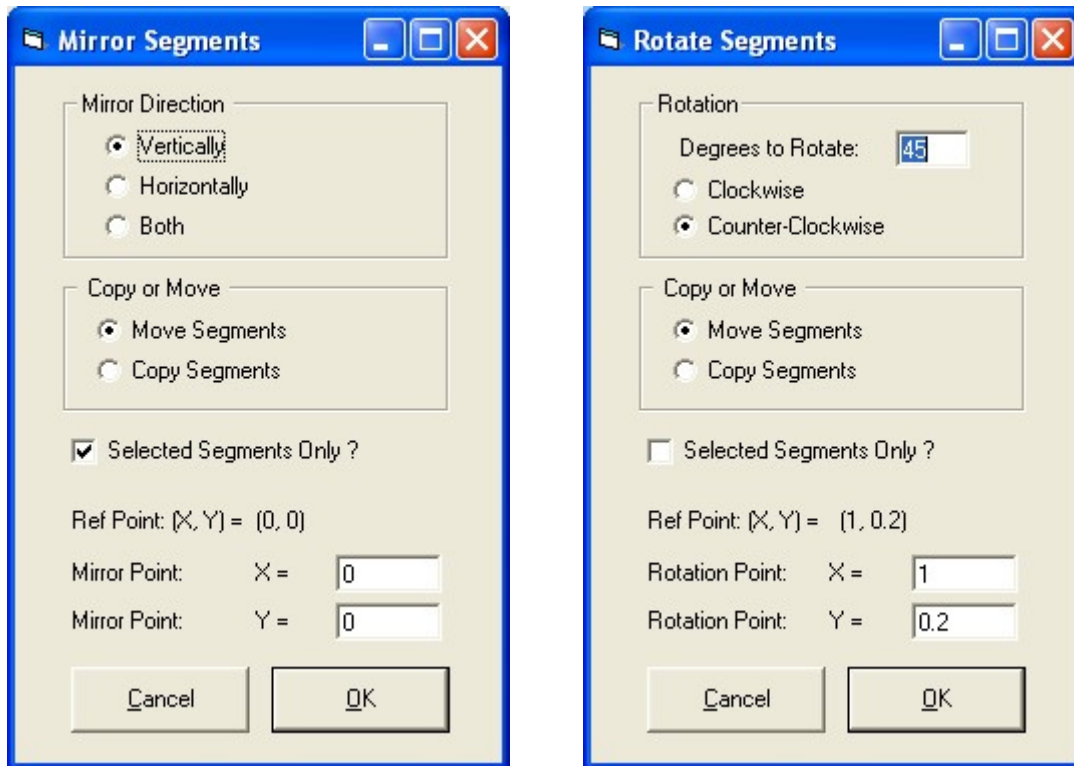
The **S** key on the keyboard also corresponds to this menu command.

Edit - *Mirror Segments*

This option brings up an input box allowing you to mirror segments on either the X or Y axis or both. Select the options you want and press **<OK>**.

Edit - *Rotate Segments*

This option brings up an input box allowing you to rotate segments about any (X, Y) point. Select the options you want and press **<OK>**.

Note that when mirroring or rotating entities, you may choose to have the new entities replace the source entities or keep them by selecting the move segments or copy segments option.

Edit - *Reverse Path*

Reverse the direction of the selected path or drawing segment. After selecting this option, click near the path segment you want to reverse.

Edit - *Make Visible* (Paths Segments Only)

Make any selected paths visible again. Only selected path segments will become visible again. If you want to make all path segments visible, select all path segments using <Ctrl> A then select **Edit | Make Visible**.

Edit - *Make Invisible* (Paths Segments Only)

Make any selected path segments invisible. This option does not delete any paths, it only makes them temporarily invisible so the user can more easily see other drawing entities.

Draw Menu
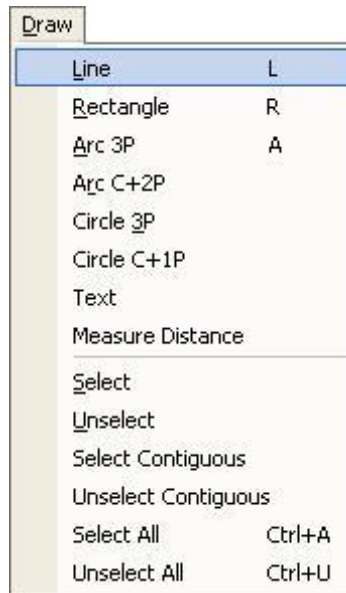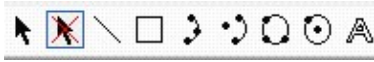
Draw - *Line*

Sets drawing mode to *Line*. When in line mode, clicking and dragging anywhere inside the drawing area will draw a line from the current (X, Y) position to the cursor, and change the current (X, Y) position to to cursor position.

This allows for an extremely quick method of drawing contiguous line segments.

Pressing the *L* key, or clicking on the ╲ icon in the Toolbar will also set the drawing mode to *Line*.

> **Note:** You can change the current (X, Y) position, and thus the starting point of the next line, by positioning the cursor to where you want the current position to be, and *right-clicking* the mouse.

Draw - *Rectangle*

Sets drawing mode to *Rectangle*. When in rectangle mode, clicking and dragging anywhere inside the drawing area will draw a rectangle from the current (X, Y) position to the cursor. The current (X, Y) position will not be changed.

Pressing the *R* key, or clicking on the □ icon in the Toolbar will also set the drawing mode to *Rectangle*.

Draw - *Arc 3P*

Sets drawing mode to *3-Point Arc* mode. When in *3-Point Arc* mode, the current (X, Y) position is considered the starting point of the arc. Clicking once sets the endpoint of the arc. Clicking again and dragging anywhere inside the drawing area will draw an arc from the starting point to the arc endpoint through the current cursor position.

Pressing the *A* key, or clicking on the ⌇ icon in the Toolbar will also set the drawing mode to *3-Point Arc*.

Draw - *Arc C+2P*

Sets drawing mode to *Center + 2 Point Arc* mode. When in *Center + 2 Point Arc* mode, the current (X, Y) position is considered the starting point of the arc. Clicking once sets the endpoint of the arc. Clicking again and dragging anywhere inside the drawing area will draw an arc from the starting point to the arc endpoint with a center point at the current cursor position.

Clicking on the ⌇ icon in the Toolbar will also set the drawing mode to *Center + 2 Point Arc*.

Draw - *Circle 3P*

Sets drawing mode to *Circle 3-Point* mode. When in *Circle 3-Point* mode, the current (X, Y) position is considered the starting point of the circle. Clicking once sets a second point of the circle. Clicking again and dragging anywhere inside the drawing area will draw a circle going through the first two points and the current cursor position.

Clicking on the ⌇ icon in the Toolbar will also set the drawing mode to *Circle 3-Point*.

Draw - *Circle C+1P*

Sets drawing mode to *Circle Center + 1 Point* mode. When in *Circle Center + 1 Point* mode, the current (X, Y) position is considered the center point of the circle. Clicking once will draw a circle going through the second point with a center at the current (X, Y) position.

Clicking on the ⌇ icon in the Toolbar will also set the drawing mode to *Circle Center + 1 Point*.

Draw - *Text*

This function will add text to your drawing. After selecting this option the following screen will appear:
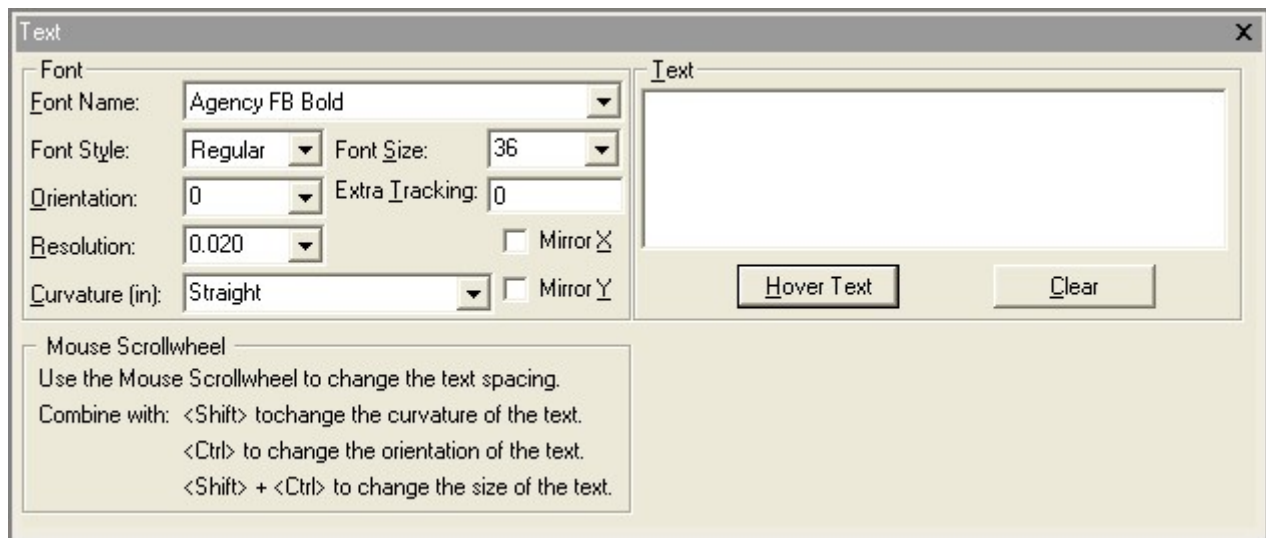
Select the text properties you want on the left side of the screen. Enter the text to place in the textbox on the right of the screen.

Press **<*Hover Text*>** to generate a floating box with the specified text. You can now drag this floating box anywhere in the drawing area. When you're satisfied with the placement, left click the mouse to place the text.

While hovering, you can use the scroll wheel on the mouse to change certain attributes of the text "on the fly". The following functions are available:

>   **<scroll>** - change the text spacing (tracking)
>   **<Shift> <scroll>** - change the text curvature
>   **<Ctrl> <scroll>** - change the text orientation
>   **<Shift> <Ctrl> <scroll>** - change the text size

If you wish to cancel a *Hover* in process press the **<*Clear*>** button. To close the **Text** window, press the **X** in the upper right corner of the screen.

The 🅐 icon in the Toolbar also corresponds to this menu command.

Draw - *Measure Distance*

This function allows you to measure the straight-line distance between two points. After selecting this option click the starting point and drag the mouse anywhere in the drawing area. While the left mouse button is down, the coordinates of the starting point, and the endpoint, plus the straight-line distance are shown in the status line at the bottom of the screen.

When the left mouse button is released, the status line returns to its normal state.

Draw - *Select*

Sets the drawing mode to *Select*. When in this mode, left clicking the mouse will select the nearest drawing segment or path segment (depending on the value of the *Edit Paths* checkbox).

Clicking on the ➤ icon in the Toolbar will also set *Select* mode.

> **Note:** Drawing segments are usually *blue* when not selected. Path segments are normally *green*. Selected segments turn *red* to indicate they are selected.

Draw - *Unselect*

Sets the drawing mode to *Unselect*. When in this mode, left clicking the mouse will deselect the nearest drawing segment or path segment (depending on the value of the *Edit Paths* checkbox).

Clicking on the ➤ icon in the Toolbar will also set *Unselect* mode.

Draw - *Select Contiguous*

Sets the drawing mode to *Select Contiguous*. When in this mode, left clicking the mouse will select the nearest drawing segment or path segment plus any segments which are contiguous.

Draw - *Unselect Contiguous*

Sets the drawing mode to *Unselect Contiguous*. When in this mode, left clicking the mouse will deselect the nearest drawing segment or path segments plus any segments which are contiguous.

Draw - *Select All - <Ctrl> A*

Selects all drawing segments or path segments (depending on the value of the *Edit Paths* checkbox).
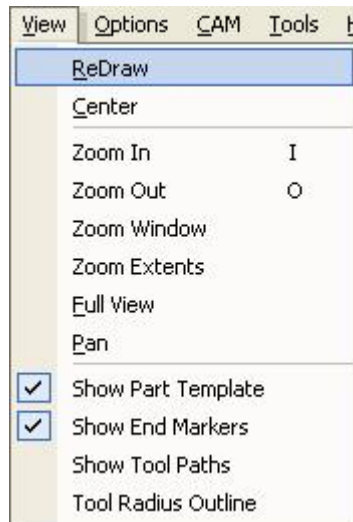
<Ctrl> A on the keyboard also corresponds to this menu command.

Draw - *Unselect All - <Ctrl> U*

Deselects all drawing segments or path segments (depending on the value of the *Edit Paths* checkbox).

<Ctrl> U on the keyboard also corresponds to this menu command.

View Menu

```
View  Options  CAM  Tools  H
      ReDraw
      Center

      Zoom In           I
      Zoom Out          O
      Zoom Window
      Zoom Extents
      Full View
      Pan
  ✓   Show Part Template
  ✓   Show End Markers
      Show Tool Paths
      Tool Radius Outline
```

View - *ReDraw*

Redraws the currently selected drawing area. This will often clean up the drawing.

View - *Center*

Centers the origin (0, 0) within the drawing area.

View - *Zoom In*

Zooms into (magnifies) the drawing area.

Pressing *I* on the keyboard, or clicking the 🔍 icon in the Toolbar will also zoom in.

View - *Zoom Out*

Zooms out of (demagnifies) the drawing area.

Pressing *O* on the keyboard, or clicking on the 🔍 icon in the Toolbar will also zoom out.

View - *Zoom Window*

Places the zoom mode to ***Window Zoom***. When in this mode, clicking and dragging the mouse will select a zoom area or window. When the mouse button is released, the drawing area will zoom to the selected zoom window.

Clicking on the 🔍 icon in the Toolbar will also activate ***Window Zoom*** mode.

View - *Zoom Extents*

Zooms and Pans so that the drawing area provides the best fit for the drawing.

If no *Path Segments* or *Drawing Entities* exist, the drawing area will remain unchanged.

Clicking on the ⊡ icon in the Toolbar will also zoom extents.

View - *Full View*

If any *Path Segments* or *Drawing Entities* exist, then **View | Full View** will behave exactly like **View | Zoom Extents**.

If no *Path Segments* or *Drawing Entities* exist, the drawing will zoom to the part template.

View - *Pan*

Toggles **Pan** mode within the drawing area. Pan mode is indicated by a cursor shaped like a hand. When the cursor appears as an arrow pointer, pan mode is turned off.

When in pan mode, clicking and dragging the mouse will pan the drawing area real-time. To stop panning, release the mouse button. To turn **Pan** mode off, press the **Pan** icon on the toolbar.

Clicking on the ✋ icon in the Toolbar will also toggle **Pan** mode.

View - *Show Part Template*

This is a checkbox field. If checked, a blue-green rectangle will appear indicating the material dimensions specified in the options menu.

View - *Show End Markers*

This is a checkbox field. If checked, small dots will be drawn on the endpoints of each drawing entity.

View - *Show Tool Paths*

This is a checkbox field. If checked tool path segments are drawn in green.

View - *Tool Radius Outline*

This is a checkbox field. If checked, a spiral path (using circles) is drawn for all tool path segments. The circles drawn have the same diameter as the current tool. This graphically shows the tool path and all material that is cut.

If this checkbox is not checked, only the center of the tool is drawn with an arrow indicating the direction of tool movement.
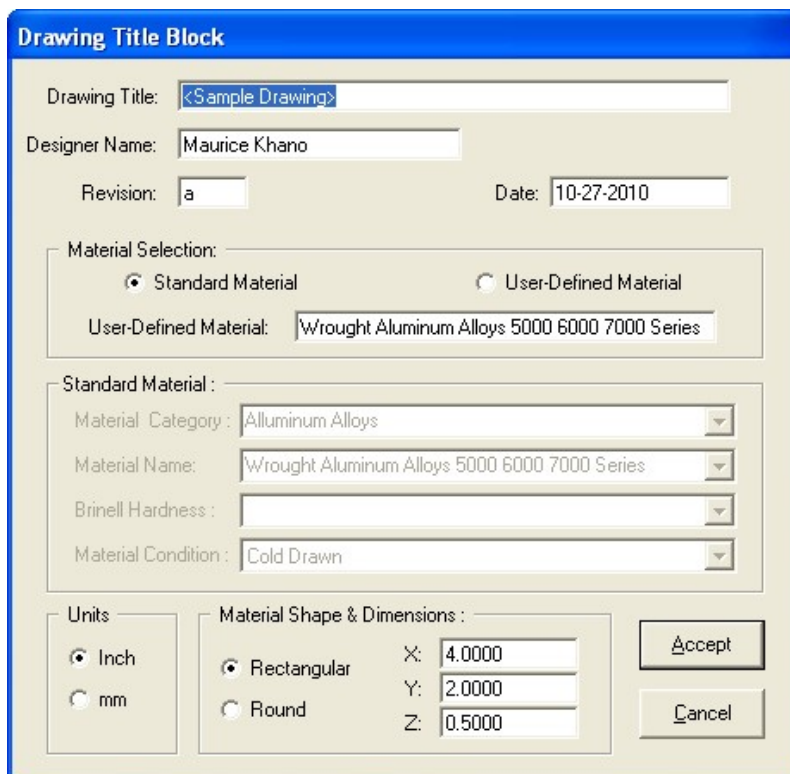
Options Menu
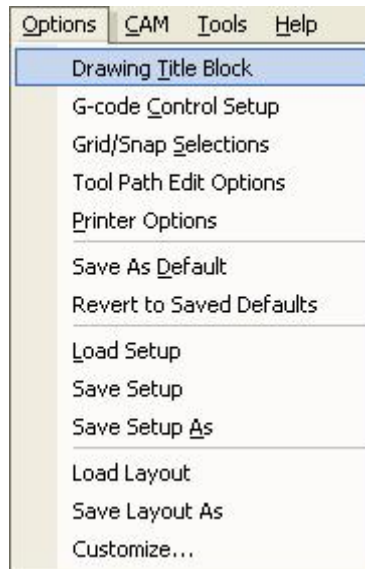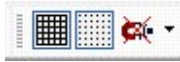
Options - *Drawing Title Block*

Displays the input screen shown below.

The fields in the top one-third of the screen are included as comments when the drawing is converted to a G-code file.

The *Cutting Speed* section is used by *Tool Selection & Definition* screen to recommend spindle and cutting speeds for each tool. See **CAM | Select Cutting Tool** for more information.

The bottom section is used to accurately display the part dimensions in the drawing area.

Options - *G-Code Control Setup*

Displays the following screen. These values are used when creating a G-code program.

Start Pos: The initial tool position set in the resulting G-code file.

Tool Change: The location the machine is directed to go to just before each tool change command in the resulting G-code file.
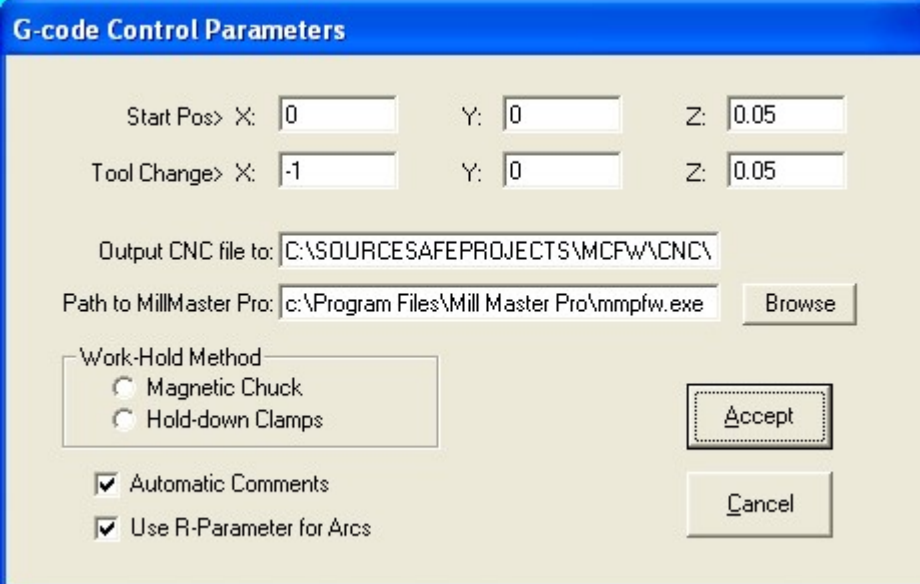
Output CNC file to: The default subdirectory to save generated G-code files.

Path to MillMaster Pro: The directory and filename of the **MillMaster Pro™** executable file.

Work-Hold Method: For reference purposes only and places an appropriate configuration command for the controller identifying the way the part is to be held into the machine.

Automatic comments: When checked, automatic comments are added that describe what the program is doing in each section of the code. When unchecked, this will reduce file size.

Use R-Parameter for Arcs: If checked**, G02/G03** commands generated for **MillMaster Pro™** or **TurnMaster Pro™** will use the **R** (Radius) parameter instead the **I/J/K** parameters.

```
G-code Control Parameters

    Start Pos> X: [0        ]   Y: [0        ]   Z: [0.05     ]
 Tool Change> X: [-1       ]   Y: [0        ]   Z: [0.05     ]

    Output CNC file to: [C:\SOURCESAFEPROJECTS\MCFW\CNC\        ]
 Path to MillMaster Pro: [c:\Program Files\Mill Master Pro\mmpfw.exe]   [ Browse ]

  ┌─ Work-Hold Method ──────────┐
  │  ○ Magnetic Chuck            │        ┌──────────┐
  │  ○ Hold-down Clamps          │        │  Accept  │
  └──────────────────────────────┘        └──────────┘

  ☑ Automatic Comments                    ┌──────────┐
  ☑ Use R-Parameter for Arcs              │  Cancel  │
                                          └──────────┘
```

Options - *Grid/Snap Selections*

Displays the following screen:
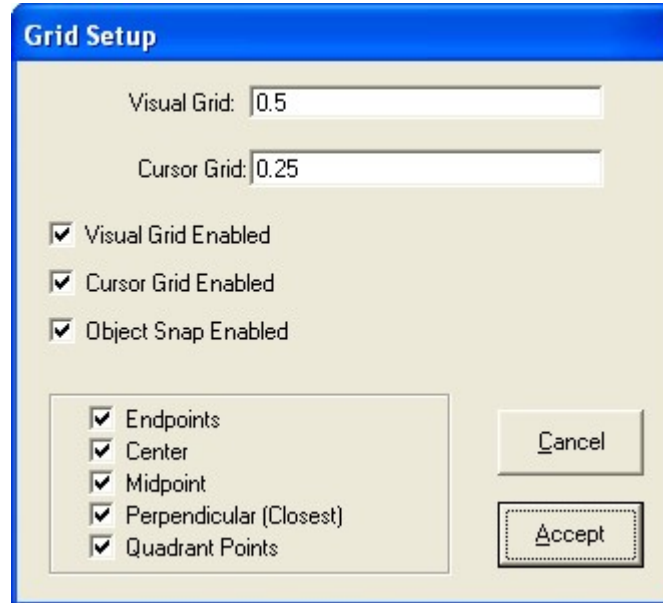
The *Visual Grid* is displayed in the drawing area. It does not affect the cursor position or cursor placement.

Clicking on the [icon] icon in the Toolbar, or pressing the **<G>** key on the keyboard will toggle the *Visual Grid* On/Off..

The *Cursor Grid* (if checked) will round the cursor's (X, Y) position to the nearest grid point.

Clicking on the [icon] icon in the Toolbar will toggle the *Cursorl Grid* On/Off..

If *Object Snap Enabled* is checked, then the cursor will try to "snap" to the closest *Endpoint, Center Point, Midpoint, Perpendicular,* or *Quadrant Point* as specified in the snap checkboxes.

Clicking on the [icon] icon in the Toolbar will toggle the *Object Snap* On/Off..

Note: The five snap checkboxes are only enabled if *Object Snap Enabled* is checked.

Options - *Tool Path Edit Options*

Displays the following screen.

The option selected describes how the **Select**, **Unselect**, **Reverse Path Direction**, and **Trim Segment** functions will behave if multiple paths at different Z-levels are found.
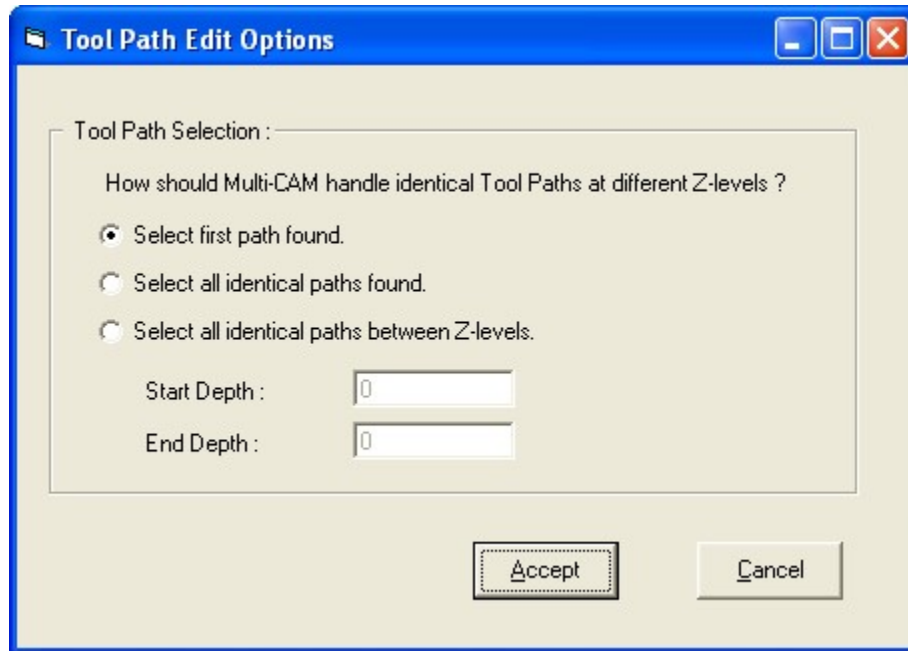
Options - *Printer Options*

Allows you to change the printer settings.

Options - *Save As Default*

Changes the default setup file to be the current setup file **(.MSU)**. This information is stored in the startup file: **MCAM.DEF**

Options - *Revert to Saved Defaults*

Reloads the current setup file **(.MSU)** as stored in **MCAM.DEF**, abandoning all changes to the setup file since the last save.

Note: The file **MCAM.DEF** contains the name of the startup file **(.MSU)**. *MultiCAM* always reads **MCAM.DEF** to determine which startup file to read.

Options - *Load Setup*

Loads a different setup file **(.MSU)**.

This changes the setup for the current session but does not change the default setup file. Thus the next time *MultiCAM* starts, it will reload the default setup file.

To change the default setup file, use **Options | Save As Default** as described above.

Options - *Save Setup*

Saves the current setup file **(.MSU)**, including any changes made since the last save.

Options - *Save Setup As*

Saves the current setup file **(.MSU)** using a different filename.

Options - *Load Layout*

Loads a different layout file (**.TB3**).

The layout file stores information about the screen layout, including toolbars, property windows, etc.

During normal operation, *MultiCAM* stores the current screen layout in the file **Layout.TB3**, and reloads it the next time *MultiCAM* starts up. The default layout shipped with *MultiCAM* is **Default.TB3**.

Normally, there is no need to load or save other layout files, but you can save specific screen configurations as special **.TB3** files and reload them later. If you do this, however, use filenames other than **Layout.TB3** and **Default.TB3**, as using these filenames may cause unexpected results.

Options - *Save Layout As*

Saves the current layout using a different filename.

Options - *Customize*

Allows you to customize some of the screen layout settings.

CAM Menu

CAM   Tools   Help
Undo Last Place
Select Cutting Tool
Pocketing/Profiling Specs
Verify/Accept Pocket
Verify/Accept Island
Rough Cut Pocket
Place Finish Path
Place One Path Segment
Quick Place Selected Segs
Edit Path/Cycle
Clear All Paths
Make CNC Mill File   Ctrl+M
Make CNC Lathe File   Ctrl+L
View CNC File
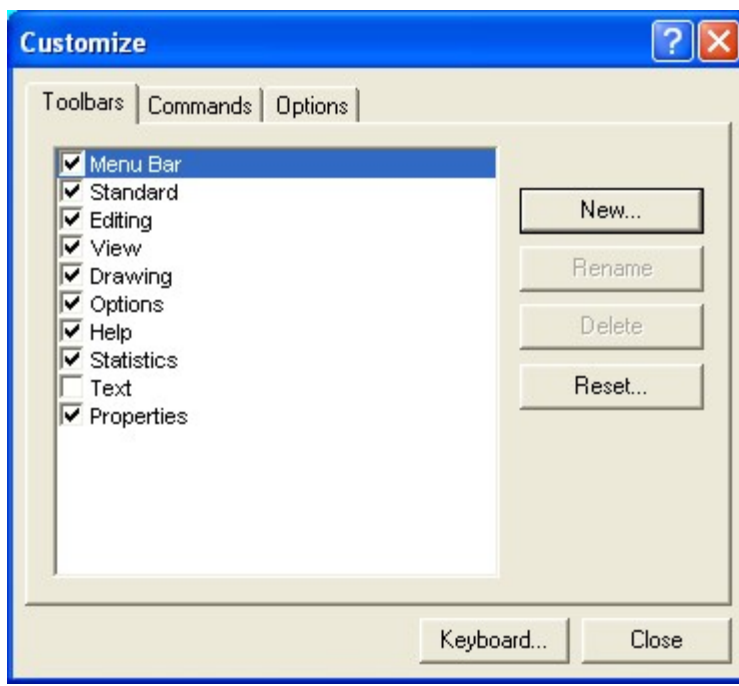
CAM - *Undo Last Place*

Undoes the last placement of one or more *Path Segments*.

CAM - *Select Cutting Tool*

Displays the **Tool Select & Definition** window shown below.

You may add up to 99 tools. Use the scrollbar on the right hand side of the screen to view additional tools.

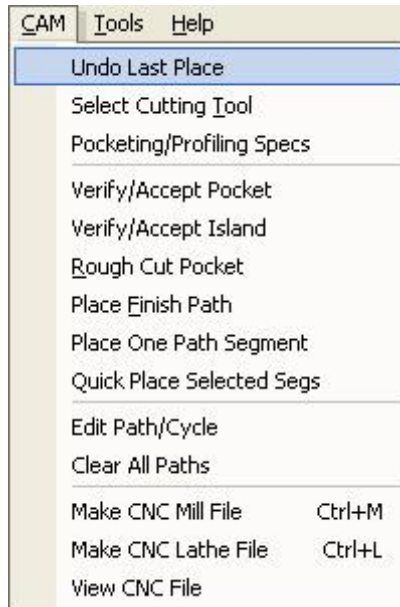The *Tool Library* button gives you access to standard tools which can used as templates for tools added to your list. First position your cursor to an empty tool or one you want to override. Press the *Tool Library* button. On the *Tool Library* screen, select the standard tool to use as a template. Click **Copy to Tool Chest**. Now, complete the tool entry items on the **Select & Definition** screen.

The Speeds and Feeds section will set spindle speed and feed rate when a tool change is required in a generated G-code file. You may enter any values you wish, or you may accept the recommended speeds by pressing the **Use Recommended Spindle Speed** button.

Notes: The *Tool Material*, and *Tool Type* are selected on this screen. The material to be cut is entered in the *Drawing and Title Block* screen.
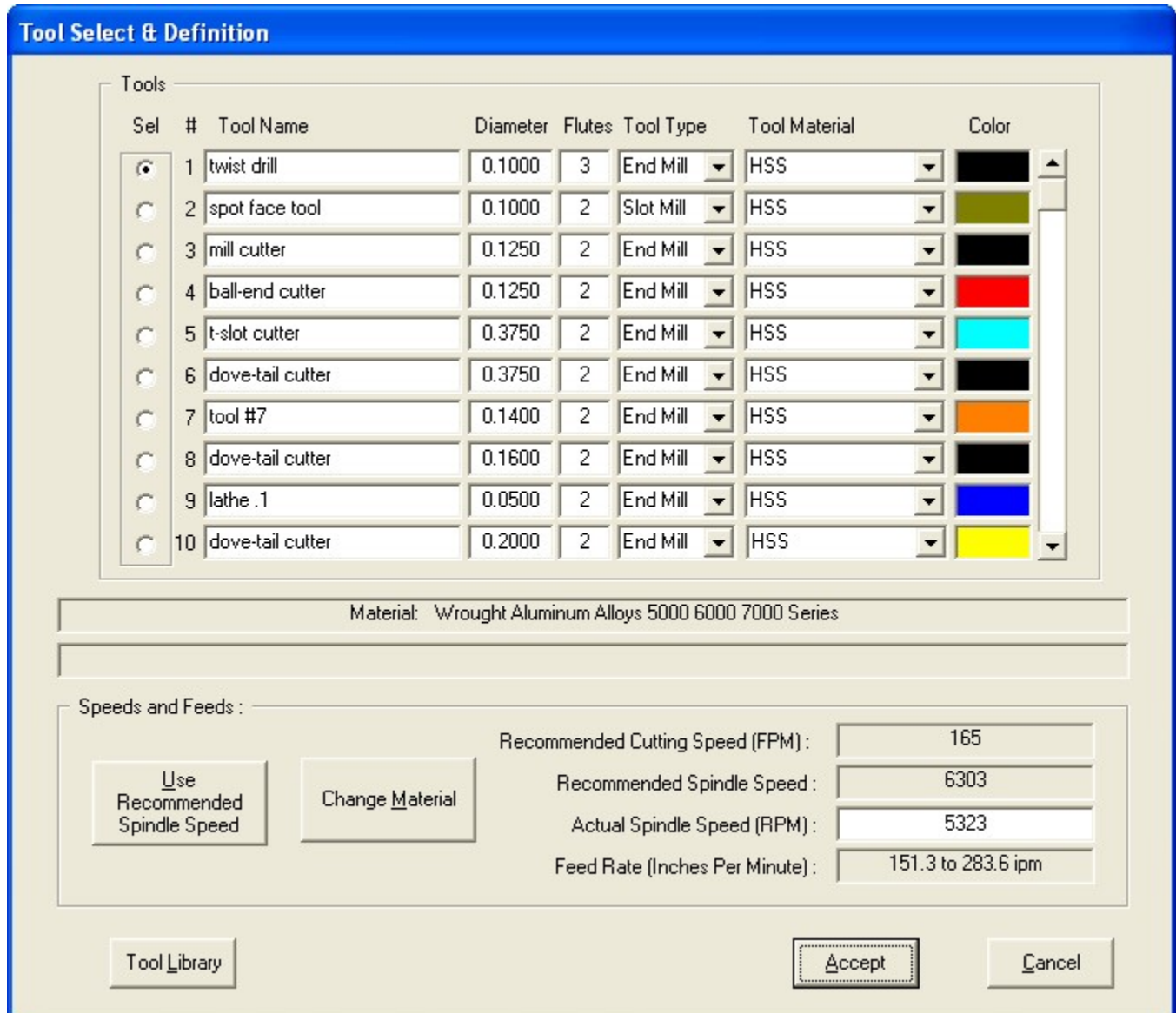
See **Options | Drawing Title Block** for more information.

Not all combinations of materials and tool types are available.

These are recommended speeds only. You should independently verify and test all speeds before using them to machine a part.

If the **Sel** radio button is on for a particular tool, that tool becomes the selected tool. This tool will be used when placing path segments until the tool is changed. The color of that tool will be the color of any paths placed using that tool.

Clicking on the ⬚ icon in the Toolbar will also bring up the *Tool Select & Definition* window.

Tool Select & Definition

Tools

Sel	#	Tool Name	Diameter	Flutes	Tool Type	Tool Material	Color
⦿	1	twist drill	0.1000	3	End Mill ▾	HSS ▾	⬛ ▲
○	2	spot face tool	0.1000	2	Slot Mill ▾	HSS ▾	🟫
○	3	mill cutter	0.1250	2	End Mill ▾	HSS ▾	⬛
○	4	ball-end cutter	0.1250	2	End Mill ▾	HSS ▾	🟥
○	5	t-slot cutter	0.3750	2	End Mill ▾	HSS ▾	🟦
○	6	dove-tail cutter	0.3750	2	End Mill ▾	HSS ▾	⬛
○	7	tool #7	0.1400	2	End Mill ▾	HSS ▾	🟧
○	8	dove-tail cutter	0.1600	2	End Mill ▾	HSS ▾	⬛
○	9	lathe .1	0.0500	2	End Mill ▾	HSS ▾	🟦
○	10	dove-tail cutter	0.2000	2	End Mill ▾	HSS ▾	🟨 ▼

Material: Wrought Aluminum Alloys 5000 6000 7000 Series

Speeds and Feeds :

Use Recommended Spindle Speed	**Change Material**

Recommended Cutting Speed (FPM) : 165

Recommended Spindle Speed : 6303

Actual Spindle Speed (RPM) : 5323

Feed Rate (Inches Per Minute) : 151.3 to 283.6 ipm

Tool Library   Accept   Cancel

CAM - *Pocketing Profiling Specs*

Displays the **Pocketing & Profiling Parameters** window. You should set all pocketing parameters before you attempt to rough cut a pocket.

Fields are given for cutting depths, rough cut clearance, and feed rates during pocketing.

Separate sections are available for **Milling Direction**, **Pocketing Style** and **Step Over Amount.**

Various **Lead In/Out Path** parameters are available.

If checked, **Apply Gouge Avoidance** will perform an extra pass, checking for tool paths that might collide with the part.

Connect Pocket Paths will leave the tool in the material when transitioning to the next pocket path. If not checked, the tool will be lifted between pocketing paths.

Lathe Style Pocketing is used for creating CNC files for lathes using *TurnMaster Pro*.   This checkbox is enabled only if the **Pocketing Style** is *Horizontal Only*.

Aperture (tolerance) is the distance used for determining pocketing closure. If the endpoint of one segment is this distance or less from the start point of the next segment, then they are treated as the same point.

CAM - *Verify/Accept Pocket*

Verifies the legality of a pocket.

The user must first select all segments that make up a pocket (and deselect any other segments).   The segments making up the pocket to be verified should be *red*.

If the pocket is valid, the *red (selected)* segments turn *green (selected and verified)*.

CAM - *Verify/Accept Island*

Verifies the legality of an island.

The user must first select all segments that make up an island (and deselect any other segments).   The segments making up the island to be verified should be *red*.

If the island is valid, the *red (selected)* segments turn *green (selected and verified)*.

CAM - *Rough Cut Pocket*
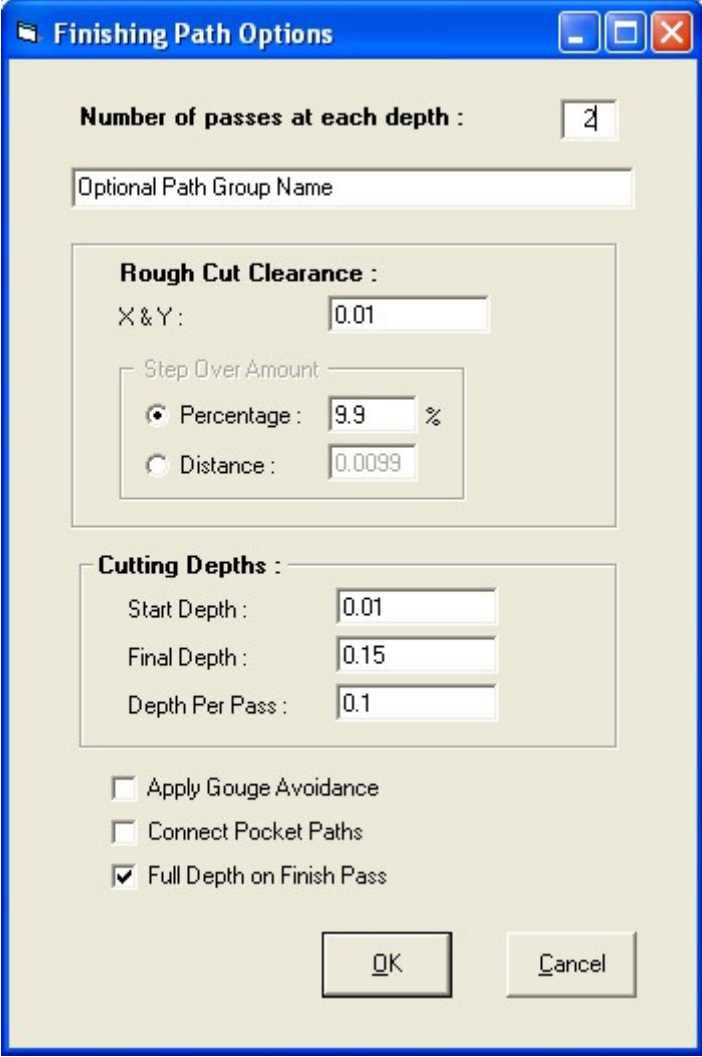
Rough cuts a pocket. The pocket should be verified (green) before selecting this option.

The pocket will be drawn using the parameters entered in **CAM | Pocketing & Profiling Specs**.

If the results of the pocket are not as desired, select **CAM | Undo Last Place** to remove the just placed segments, select **CAM | Pocketing & Profiling Specs**, to change the way the pocket is milled, and rough cut the pocket again.

CAM - *Place Finish Path*

This places a finished path on or next to a set of contiguous drawing entities.

Finishing Path Options

Number of passes at each depth : 2

Optional Path Group Name

Rough Cut Clearance :

X & Y : 0.01

Step Over Amount

⦿ Percentage : 9.9 %

◯ Distance : 0.0099

Cutting Depths :

Start Depth : 0.01

Final Depth : 0.15

Depth Per Pass : 0.1

☐ Apply Gouge Avoidance
☐ Connect Pocket Paths
☑ Full Depth on Finish Pass

OK   Cancel

Enter the **Total Number of Passes**. The **X & Y Rough Cut Clearances** entry will be enabled if the Total Number of Passes is 2 or more. The **Step Over Amount** data fields will be enabled if the Total Number of Passes is 3 or more.

After pressing the <**OK**> button on this screen, you will be prompted (via the status line) to mouse-click three times. These are:

1) **Click near the starting point.** (Where to begin milling.)
2) **Click to indicate direction.** (The direction of milling.)
3) **Click to indicate finish reference.** (Should the finish path be to the left, right, or directly on the drawing entity.)

CAM - *Place One Path Segment*

Similar to **CAM | Place Finish Path**, except that only one path segment will be drawn.

CAM - *Quick Place Selected Segs*

Automatically place *Path Segments* directly over any selected (*red* or *green*) drawing entities. Placement order and direction of the *Path Segments* are based the order and direction that the *Drawing Entities* were placed.

CAM - *Edit Path Cycle*

Displays the *Verify/Edit Path Segments* window.

Use this window to manually edit or delete any path segments that were not generated as expected.

You can manually change the segment type and/or coordinates of any segment.

You can navigate through the paths or path groups using the *Prev* and *Next* buttons. A new path group is generated for each *place* generated by the user.

You can *Insert* or *Delete* paths, and *Delete Path Groups* using the associated buttons.

To apply the changes made on the screen to a segment, click *Apply*. To accept your changes and exit, click *Accept*. To exit click *Exit*.

CAM - *Clear All Paths*

Deletes all path segments.

CAM - *Make CNC Mill File - <Ctrl> M*

Generates a **CNC Mill File** for use with *MillMaster™ 2013*.

If your drawing file (.CDM) has not been saved, you will be prompted to save it.

Your generated file will be named the same as your *.CDM* file only with a *.CNC* extension, and it will be stored in *<MultiCAM's default directory>\CNC*.

A screen will be displayed similar to the following:

```
View File - C:\sourcesafeprojects\MCFW\CNC\Test Mill File.cnc

/Paxmill Post version 1.0
/*********************************************************************
/*   MultiCAM Mill (tm)  Version 2.0.1
/*       RS 274-D CNC CODE
/*  {File Name}:  Test Mill File.CNC
/*  {Drawing Title}:  Test Drawing    {Rev}: a
/*  {Designer}:  MicroKinetics Inc.
/*  {Material}:  Shock Resisting S1, S2, S5, S6, S7  {Fixture}:  CHUCK   {Units}:  INCH
/*  {SLENGTH}: 2.0  {SWIDTH}: 2.0  {SHEIGHT}: 0.5
/*  Setup==>  {X Start}: 0.0   {Y Start}: 0.0   {Z Start}: 0.05
/*         {X Origin Zero}:   1
/*         {Y Origin Zero}:   1
/*         {Z Origin Zero}:  -1
/*  {Creation Date}: 04-21-2009   {Creation Time}: 15:27:50
/*  Created for MillMaster Pro (tm)
/*********************************************************************
G70
G90
G75
/
/--------------------------------------------------------
/    << Tool #2 code >>
```

[Transfer to MillMaster Pro]   [Close]

You can transfer this file directly to *MillMaster* or simply *Close* the window.

CAM - *Make CNC Lathe File - <Ctrl> L*

Generates a **CNC Lathe File** for use with *TurnMaster™ 2013*.

See **CAM | Make CNC Mill File** above for details.

CAM - *View CNC File*

Allows you to view and/or transfer a *.CNC* file.

Tools Menu

Tools | Help
Add Machine Cycle
Add Bolt Hole Circle

Apply **G**ouge Avoidance
Cleanup Drawing
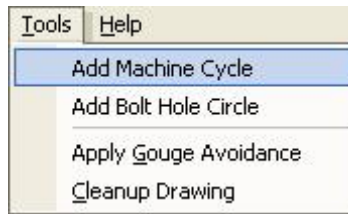
Tools - *Add Machine Cycle*

Add a canned machine cycle to the part. The following screen will display:

Cycle Select & Definition

Optional Path Group or Process Name

Canned cycles

G79=Deep Channel
G81=Drill Hole
G82=Spot Face
G83=Deep Hole w/Chip Clearing
G85=Boring Hole
G87=Deep Hole w/Chip Break
G89=Boring w/Dwell

Start Depth: 0

Final Depth: 0.10000

Incr Depth: 0.02000

Dwell (sec): 1

Feed Rate: 8

☐ Centers of All Selected Circles
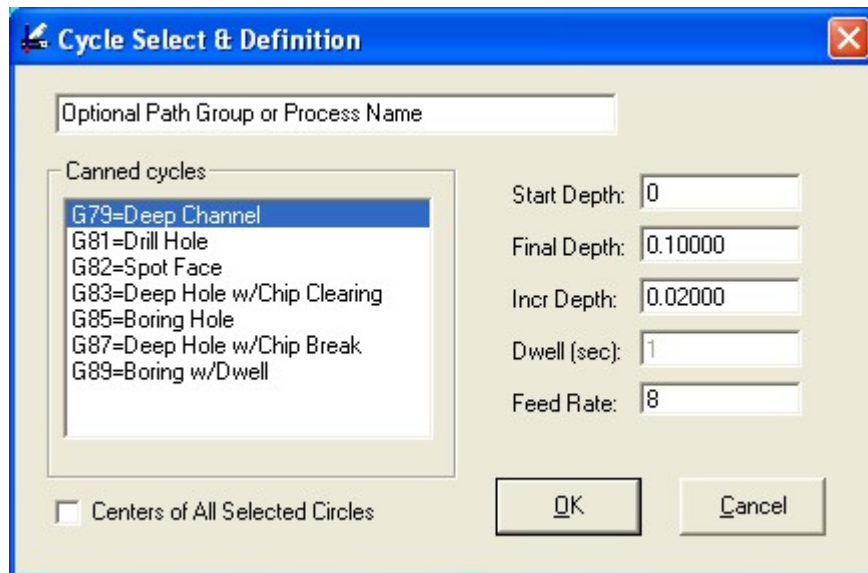
OK   Cancel

Select the details of the machine cycle and press ***OK***, or press ***Cancel*** to quit.

If the ***Centers of All Selected Circles*** checkbox is checked, then a machine cycle will be immediately placed at the center of any currently selected circles. You will not be prompted to click to place a cycle.

If the ***Centers of All Selected Circles*** checkbox is not checked, you will be prompted as follows:

Click spot to place machine cycle.

Follow the instructions to place the machine cycle.

Tools - *Add Bolt Hole Circle*

Compute **Bolt Hole Circle**. The following screen will display:

Bolt Hole Circle

Input Values :

Number of Bolt Holes | 9
Bolt Circle Diameter | 2
Hole Diameter | 0.1
Degrees Offset for Hole #1 | 0
Center X Coordinate | 0
Center Y Coordinate | 0

☑ Display Coordinates on Diagram ?

Diagram labels:
(0.1736, 0.9848)
(-0.5, 0.866)
(0.766, 0.6428)
(-0.9397, 0.342)
(0, 0)
(1, 0)
(-0.9397, -0.342)
(0.766, -0.6428)
(-0.5, -0.866)
(0.1736, -0.9848)

Listbox:
1 - X=1, Y=0
2 - X=0.766, Y=0.6428
3 - X=0.1736, Y=0.9848
4 - X=-0.5, Y=0.866
5 - X=-0.9397, Y=0.342
6 - X=-0.9397, Y=-0.342
7 - X=-0.5, Y=-0.866
8 - X=0.1736, Y=-0.9848
9 - X=0.766, Y=-0.6428

Results :

Dist Btwn Hole Centers: 0.684067
Dist Btwn Hole Edges: 0.584067
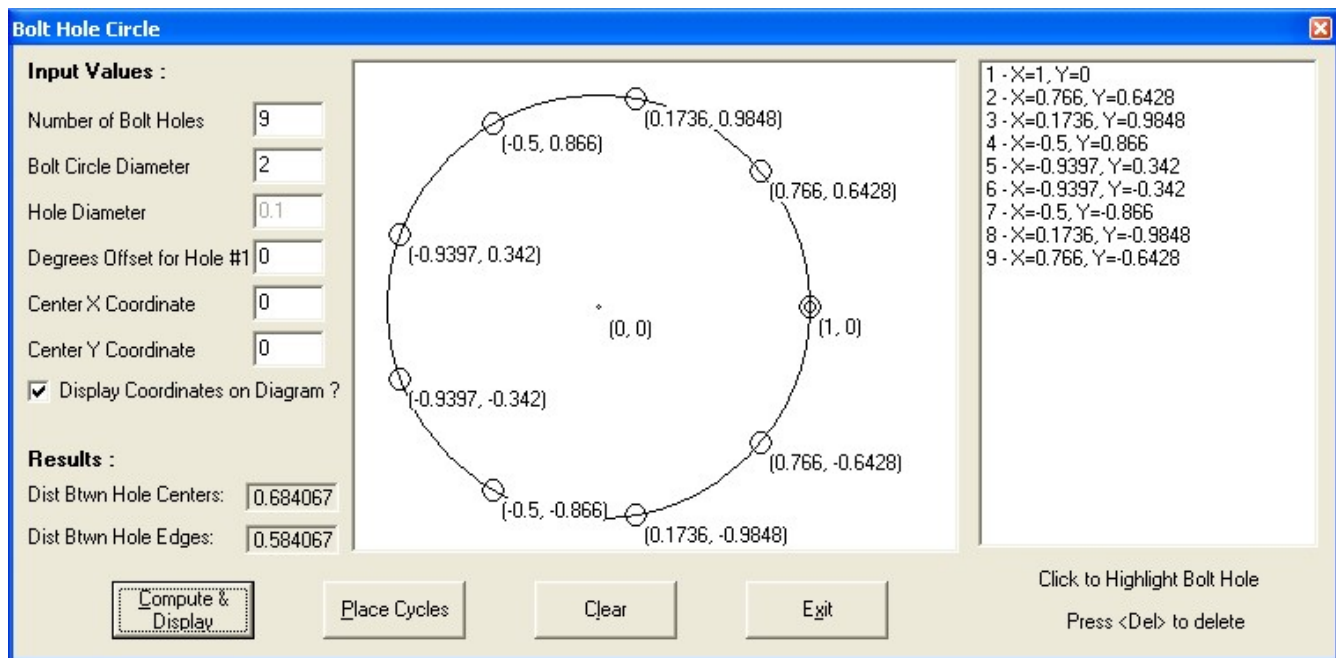
Buttons: Compute & Display | Place Cycles | Clear | Exit

Click to Highlight Bolt Hole
Press <Del> to delete

Enter the desired parameters and click **Compute & Display**.

You will see the details of the generated cycles in a listbox on the right side of the screen. You can highlight and/or delete individual cycles by clicking in this listbox.

If you want to place these cycles in the drawing, click **Place Cycles**.

Tools - *Apply Gouge Avoidance*

This option will perform a one-time extra pass, for the entire drawing, checking for tool paths that might collide with the part.

Tools - *Cleanup Drawing*

Displays the following screen:

Gives the user the option to cleanup any
Duplicate Segments or **Zero Length Segments**
that are found,

Check the desired checkboxes and click **Cleanup** or
click **Cancel** to exit without performing any cleanup.

Cleanup Drawing

Type of Segment | Occurences
☑ Duplicate Segments | 1
☐ Zero Length Segments | 0

Buttons: Cancel | Cleanup
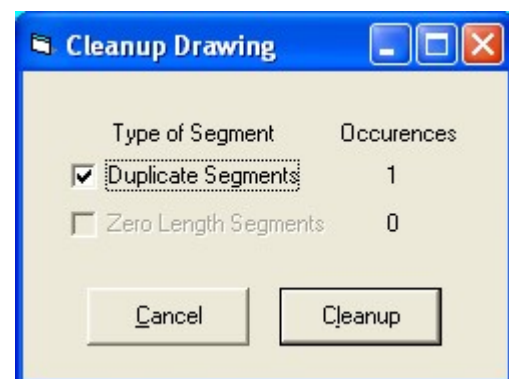
Help Menu

Help - *Contents*

Displays this **Help File** (*MCFW_Manual.PDF*), starting at Page 2, **Contents Quick Jump**

Clicking on the icon in the Toolbar, or pressing <**F1**> on the keyboard will also bring up the ***Help Contents*** window.

Help - *Search for Help on*

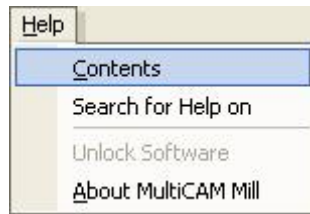Displays this **Help File** (*MCFW_Manual.PDF*), starting at Page 1, **MultiCAM™ 2013 Title Page**, with the Find textbox available in the upper-left of the screen.

Help - *Unlock Software*

If the software is locked, the following screen will appear. If the software is already unlocked, this option will be grayed out in the Help menu.

Contact MicroKinetics to get your **Decode Key**. This will enable all the features of ***MultiCAM***.

Help - *About MultiCAM Mill*

Displays the ***About MultiCAM™ 2013*** window.

Tutorial - Creating a Drawing

Select *File | New* or click on the new document icon to clear the memory of any previous entries and start fresh.

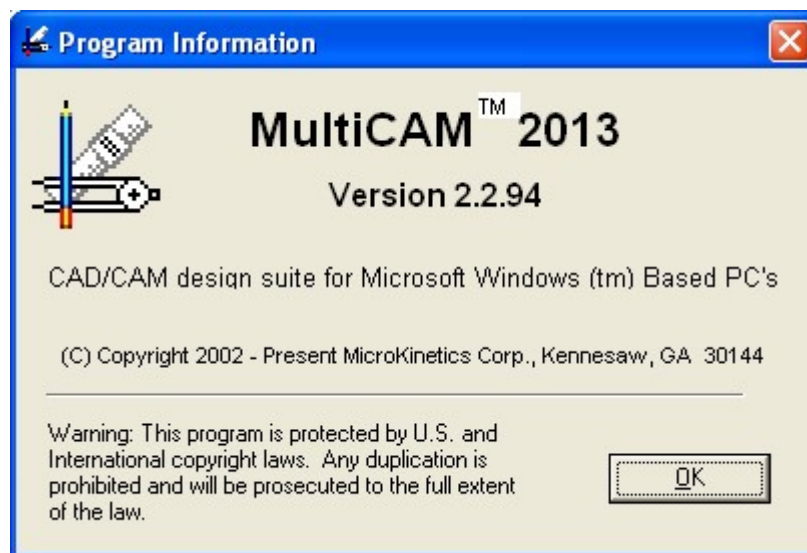Choosing the Line icon or selecting *Draw | Line* allows you to draw straight lines. The line is drawn from the X marker to the mouse pointer. Use the right mouse button to position the X marker and the left mouse button to draw. While holding the left mouse button down you'll see the line forming but it won't actually be drawn until you release the button.

You may also draw arcs, circles, and rectangles by clicking on the appropriate icons (or selecting them from the draw menu), positioning the X marker and then the mouse pointer as desired to achieve the shape you are drawing. Some entities will require multiple X markers to define the shape.

If you make a mistake you can erase it by selecting *Edit | Undo* (or clicking on the undo icon). A redo icon and menu selection are also available to reverse an undo.
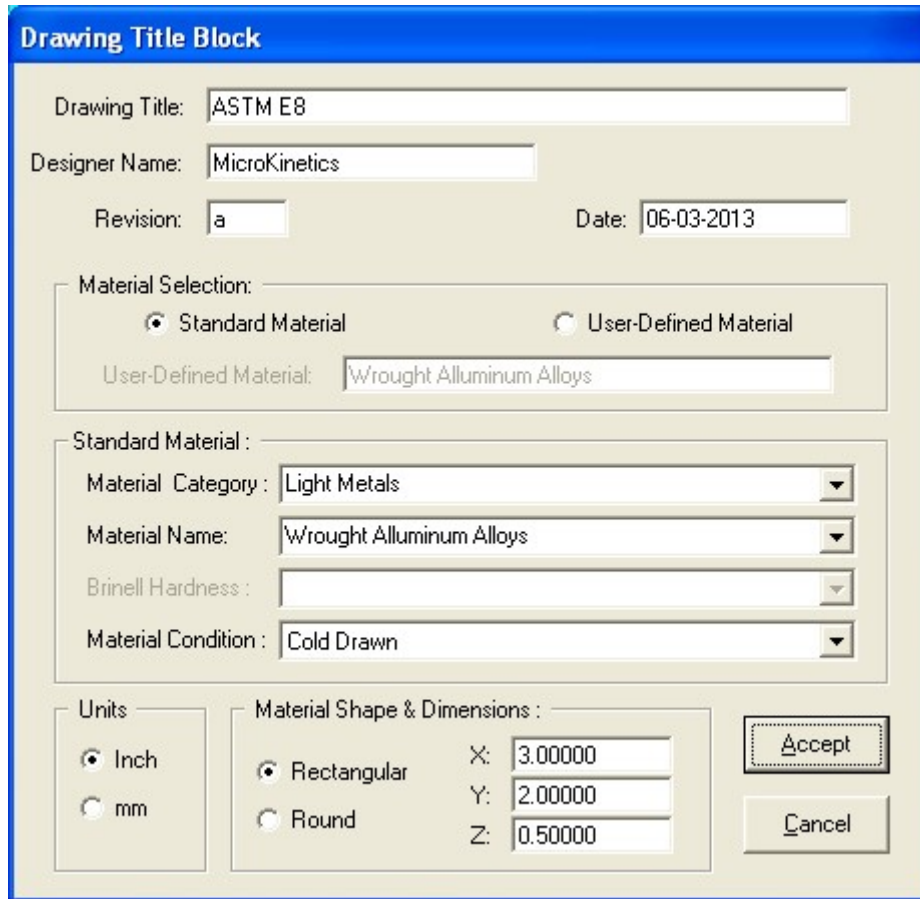
Another way to create a drawing is to import an existing DXF file. A DXF file can be created in AutoCAD or other CAD software. Choose *File | Import DXF File* to do this. You will need to have created a DXF file containing the appropriate portions of the original drawing.
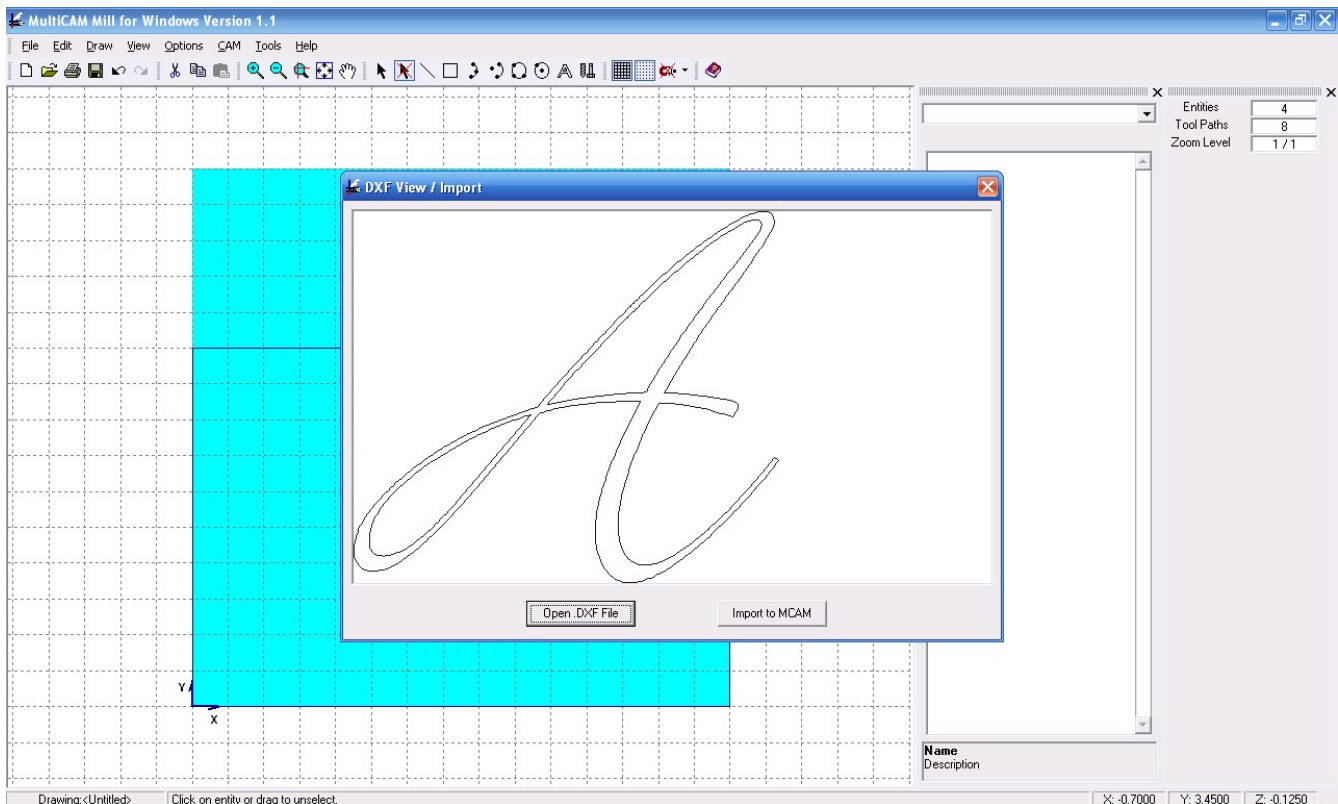
Once you have a drawing on your screen, it is important to save it, if you haven't done so. It is also important to clean up the drawing from any duplicate entities before you start to create a tool path.

The Title Block

Set up the drawing title block first and fill in the information about the design. This information will be helpful when machining the part as it will be placed in the header of the G-code file.

Drawing Title Block

Drawing Title: ASTM E8

Designer Name: MicroKinetics

Revision: a   Date: 06-03-2013

Material Selection:
- ● Standard Material   ○ User-Defined Material

User-Defined Material: Wrought Alluminum Alloys

Standard Material :

Material Category : Light Metals

Material Name: Wrought Alluminum Alloys

Brinell Hardness :

Material Condition : Cold Drawn

Units
- ● Inch
- ○ mm

Material Shape & Dimensions :
- ● Rectangular
- ○ Round

X: 3.00000
Y: 2.00000
Z: 0.50000

Accept

Cancel

Selecting a Cutting Tool

In deciding on a cutting tool you must take into consideration the material you will be machining. The tool diameter, the length of the tool, and the number of flutes (cutting edges per revolution) are all important.   In the **Option | Drawing Title Block** you will find a place to select the material. This selection comes into play when choosing a cutting tool.

Tool Select & Definition

Tools

Sel	#	Tool Name	Diameter	Flutes	Tool Type	Tool Material	Color
●	1	twist drill	0.1000	3	Drill ▾	HSS ▾	
○	2	spot face tool	0.2000	12	Face Mill ▾	Coated Carbide ▾	
○	3	mill cutter	0.3000	6	End Mill ▾	HSS ▾	
○	4	ball-end cutter	0.0300	2	End Mill ▾	Uncoated Carbide ▾	
○	5	t-slot cutter	0.0200	2	Drill ▾	HSS ▾	
○	6		0.0000	0	▾	▾	
○	7		0.0000	0	▾	▾	
○	8		0.0000	0	▾	▾	
○	9		0.0000	0	▾	▾	
○	10	dove-tail cutter	1.0000	2	Face Mill ▾	Coated Carbide ▾	

Material: Wrought Alluminum Alloys

Speeds and Feeds :

Recommended Cutting Speed (FPM) :	390
Recommended Spindle Speed :	14897
Actual Spindle Speed (RPM) :	1681
Feed Rate (Inches Per Minute) :	715.1 to 1385.4 ipm

Use Recommended Spindle Speed   **Change Material**

Tool Library   **Accept**   **Cancel**

** If you have already selected the material in the previous step, the recommended speeds may be displayed depending on if there is an entry in the database for that material.

Click on the tools icon or select *CAM | Select Cutting Tools* to access the tool chest. Choose the cutting tool you will be using or set up a new one for your work. Note that the speeds and feeds as well as the RPM of the spindle may be suggested as in the screen above. You can accept the defaults or enter your own values.

Each tool has certain parameters you need to specify. The tool diameter is the most important of these, because this will affect the actual tool path. Also of importance is the type of tool (steel vs. carbide and the number of flutes). These affect the feed rates and spindle speeds recommended by the system.

If you will be using a few tools frequently, it is probably a good idea for you to take the time to set up each cutting tool in the tool chest and physically number these tools or tool holders to correspond.

Tool Path Specifications

The next step before we place a tool path command is to select the specifications that these tool paths will adopt. These are located in the **CAM | Pocketing & Profiling Parameters**.

Pocketing & Profiling Parameters

Cutting Depths :

Start Depth :	0.01
Final Depth :	0.15
Depth Per Pass :	0.1

Rough Cut Clearance :

X & Y :	0.01
Z :	0

Feed Rates

Z-Axis Plunge :	1
Rough Cut :	8
Finishing Cut :	6

Recommended : 715.1 to 1385.4
using Tool # 1
Wrought Alluminum Alloys

Milling Direction
- ○ Conventional Milling only
- ● Climb Milling only
- ○ Bidirectional

Pocketing Style
- ● Horizontal Only
- ○ Vertical Only
- ○ Parallel Spiral

Spiral Start Distance from Pocket Center : 0

Step Over Amount
- ● Percentage : 9.9 %
- ○ Distance : 0.0099

Lead In/Out Paths
- ☐ Lead In/Out Paths
- ○ Lines   ○ Arcs

Entry/Exit Angle :	30
Entry/Exit Length :	0.1

- ☐ Apply Gouge Avoidance
- ☐ Connect Pocket Paths
- ☐ Lathe Style Pocketing
- ☐ Lathe (Work from Inside Out)
- ☐ Auto Place Finish Path

Aperture (tolerance)   0.001

[Accept]   [Cancel]

The final depth is the depth of the cut. The start depth is where in the up/down axis to begin. If the depth per pass is less than the depth, it will automatically know to subdivide the cuts and create multiple passes to achieve the desired depth.

The rough cut clearance determines how far away to stay from the finished features. The X and Y values are for side to side wall clearance. The Z value is how far to remain off the bottom. This is only for the pocketing feature. Placing a finish path does not use these parameters.

The rough cut feed rate is used during pocketing while the finishing feed rate is used during the profiling or finishing cuts.

The milling direction is used for pocketing and can specify conventional, climb, or bidirectional cutting. Conventional mode removes material such that the cutting edges of the tool attack the material in the same direction. Climb milling moves the cutter such that it is cutting to the rear of the movement.
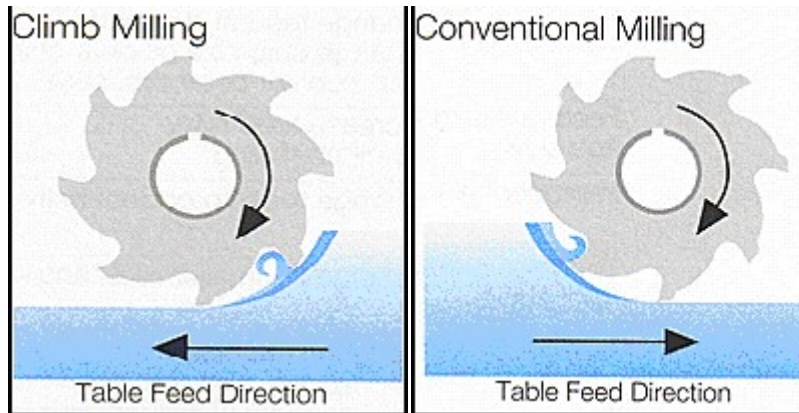
Climb milling can be used in most milling applications, It is especially important when machining titanium, cobalt and nickel based alloys.

Conventional milling is recommended for milling castings or forgings with very rough surfaces.

In most cases you will use bidirectional to save time.

The pocketing pattern can be horizontal, where each stroke is left to right or right to left, or vertical where each stroke is front to back or back to front, or spiral where the software will calculate concentric patterns that follow the shape of the contour.

The step over amount determines how much material will be removed per stroke and defaults to 50% of the cutter diameter. You may specify this in terms of a percentage of the tool diameter or in terms of a specific linear amount.
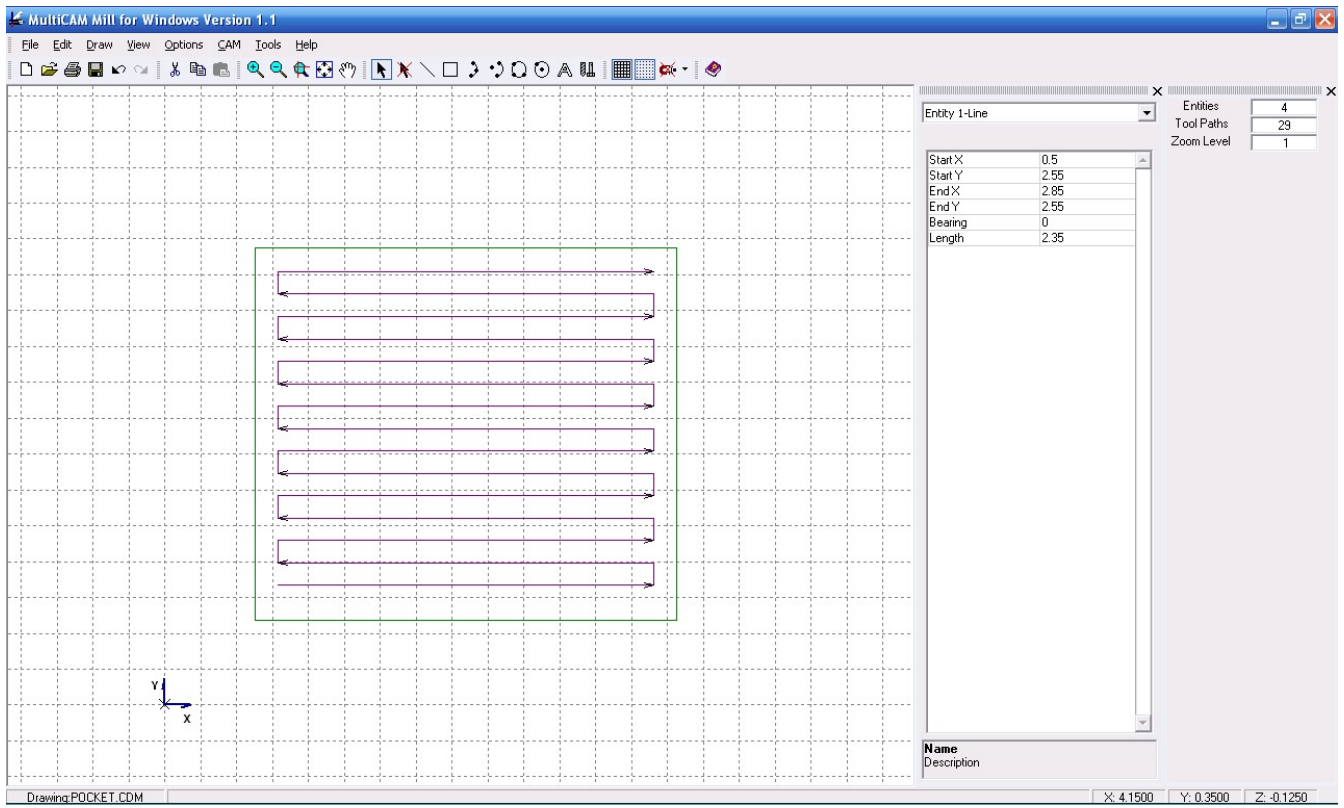
Lead in / Lead out Paths

Lead in and lead out are used whenever cutting a finish path and you want to avoid having a gouge in the side of the finish profile due to tool deflection during plunge. The lead paths can be either a straight line or an arc.

Choose the angle and length of the lead in/out command to be added. Generally, an angle in the range of 10 to 30 degrees and a length that is roughly equivalent to the cutter diameter are recommended values depending on your design requirements.
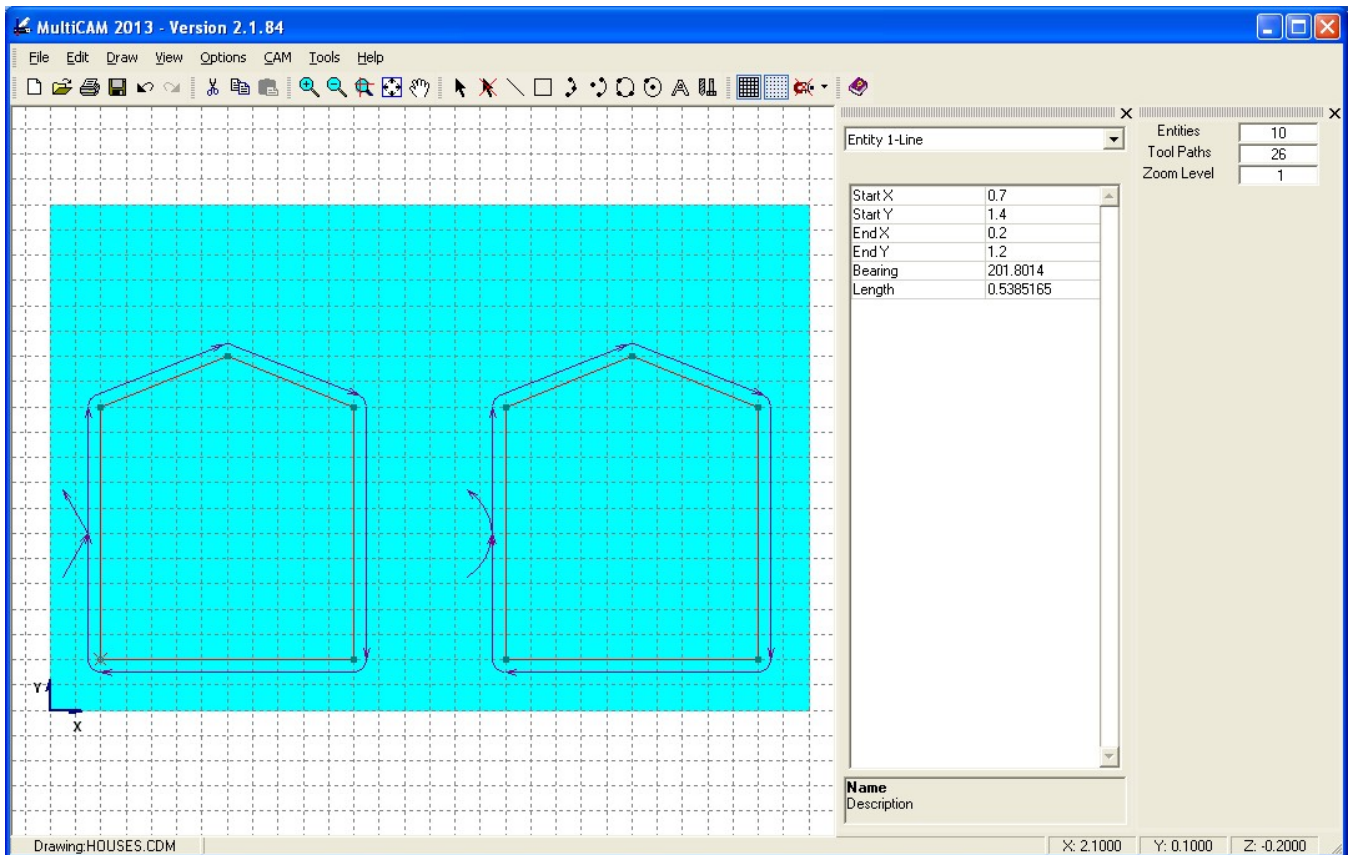
Gouge Avoidance

A gouge is an overlap in the tool path. This occurs when the tool is attempting to pass through a constriction that is smaller that the diameter of the cutter. Having the gouge avoidance checked cleans up any gouges in the tool path automatically. You may choose to leave this unchecked and perform the gouge removal with a second command: **Tools | Apply Gouge Avoidance**. This can be a useful diagnostic tool.

Connect Pocketing Paths

This option, if checked, will automatically link the pocketing tool paths such that within the same pocket the tool is not raised and lowered unnecessarily.
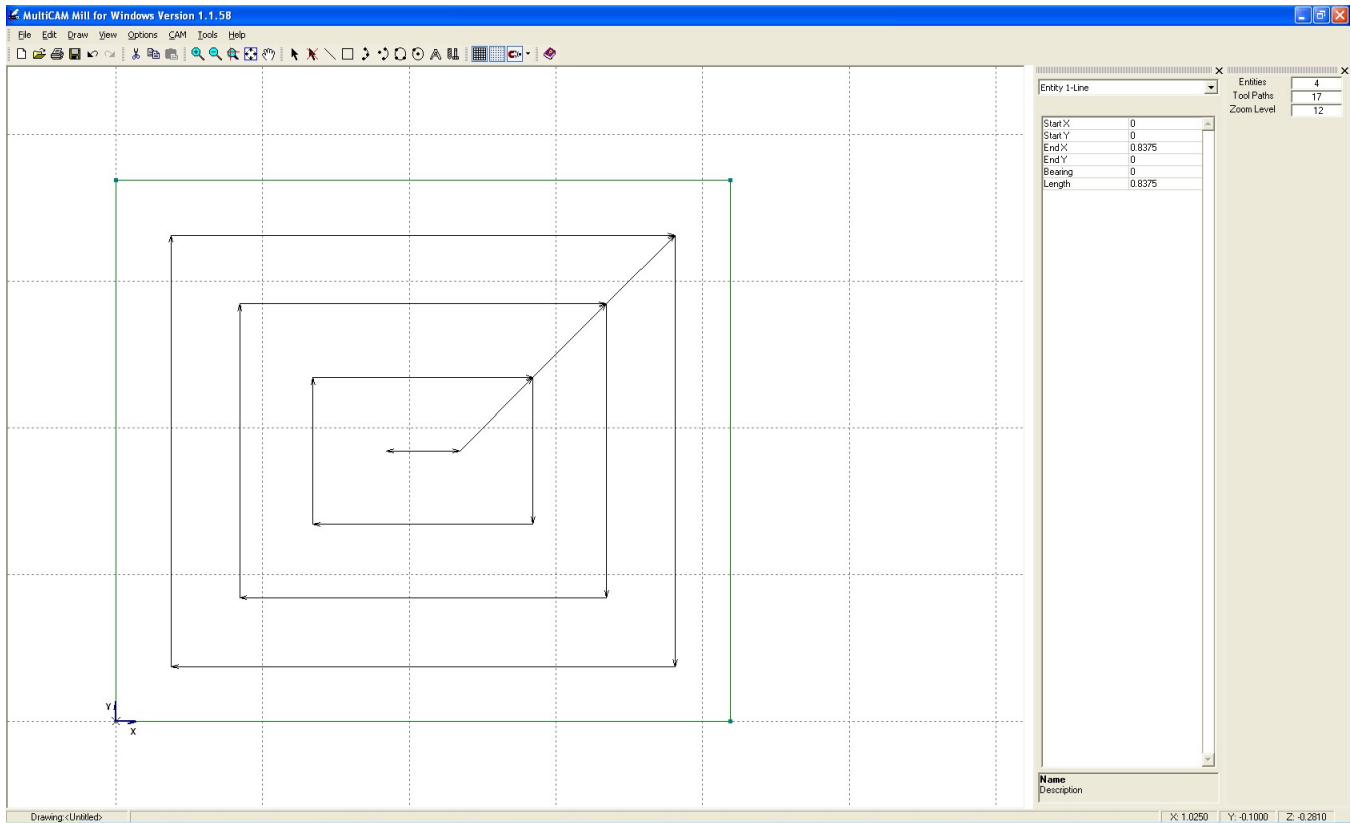
Aperture (Tolerance)

This value can be used to allow the software to be more forgiving in what it considers a match. A line segment that is supposed to start where another ends may be off by this amount and still be considered close enough.

Creating Tool Paths

Select **CAM | Place Finish Path**. The status line will prompt you to click three times.

First, decide where you want the cutting tool to start. Click on the starting line. Next, click some distance down the line in the direction you want the initial tool cut. Finally, click on one side or the other next to the line, or directly on top of the line.

This will place the tool path. If there are other segments connected, the path will continue until the end is reached or the geometry looped back to the start point of the initial segment.

Making a CNC File

If you are satisfied with the tool path, you can create a CNC file by selecting **CAM | Make CNC Mill File** or **CAM | Make CNC Lathe File** . This will create a G-code file and display its contents in a new window. You may click **Transfer to MillMaster Pro** (or **Transfer to TurnMaster Pro**) to load this program into *MillMaster Pro* (or **TurnMaster Pro**)**.**

Machining a Pocket

To machine a pocket, select the segments that comprise the pocket and select **CAM | Verify/Accept Pocket**. This will change the color of the red segments to green. If you have any islands you want to avoid, you would select the segments for each island and select **CAM | Verify/Accept Island**. Then select **CAM | Rough Cut Pocket**. This will place the pocket paths based on parameters you previously set in the *Pocketing & Profiling Parameters* screen.

Drilling Holes

In order to drill a hole, you simply select *Tools | Add Machine Cycle.* The *Cycle Select & Definition* screen will appear. Choose the type of machine cycle you desire and enter any details. Click **OK**.
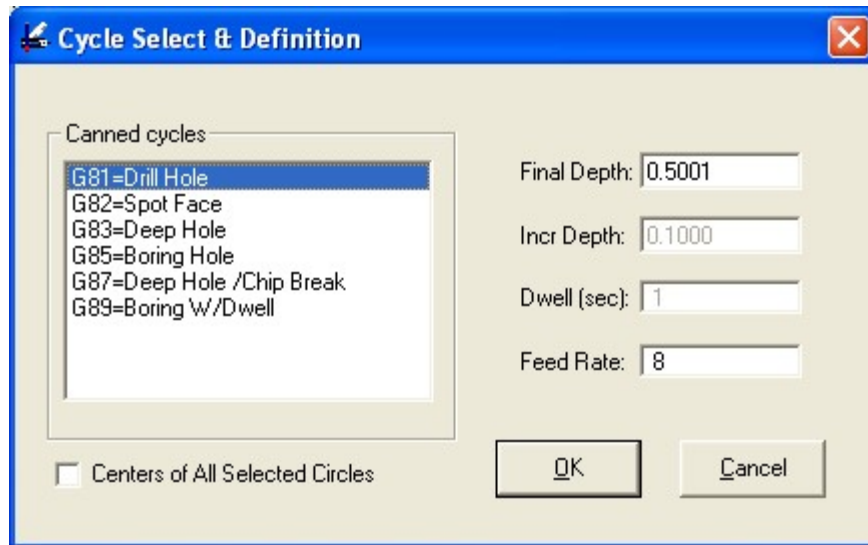
The status line will now prompt you to: *Click spot to place machine cycle.*

You may click and release anywhere in the drawing area to place the canned cycle. Or you can click and drag to the desired location. All cursor and object snap options will function as normal. Release the mouse button when you are happy with the placement.

Bolt hole circles can be place using a special utility. Select *Tools | Add Bolt Hole Circle*.

This will allow you to place any number of holes along a specified diameter and centered on any point.

Holes can be deleted from the pattern if not needed by selecting their coordinate entry lines (the graphic will highlight the hole) and pressing the delete key.
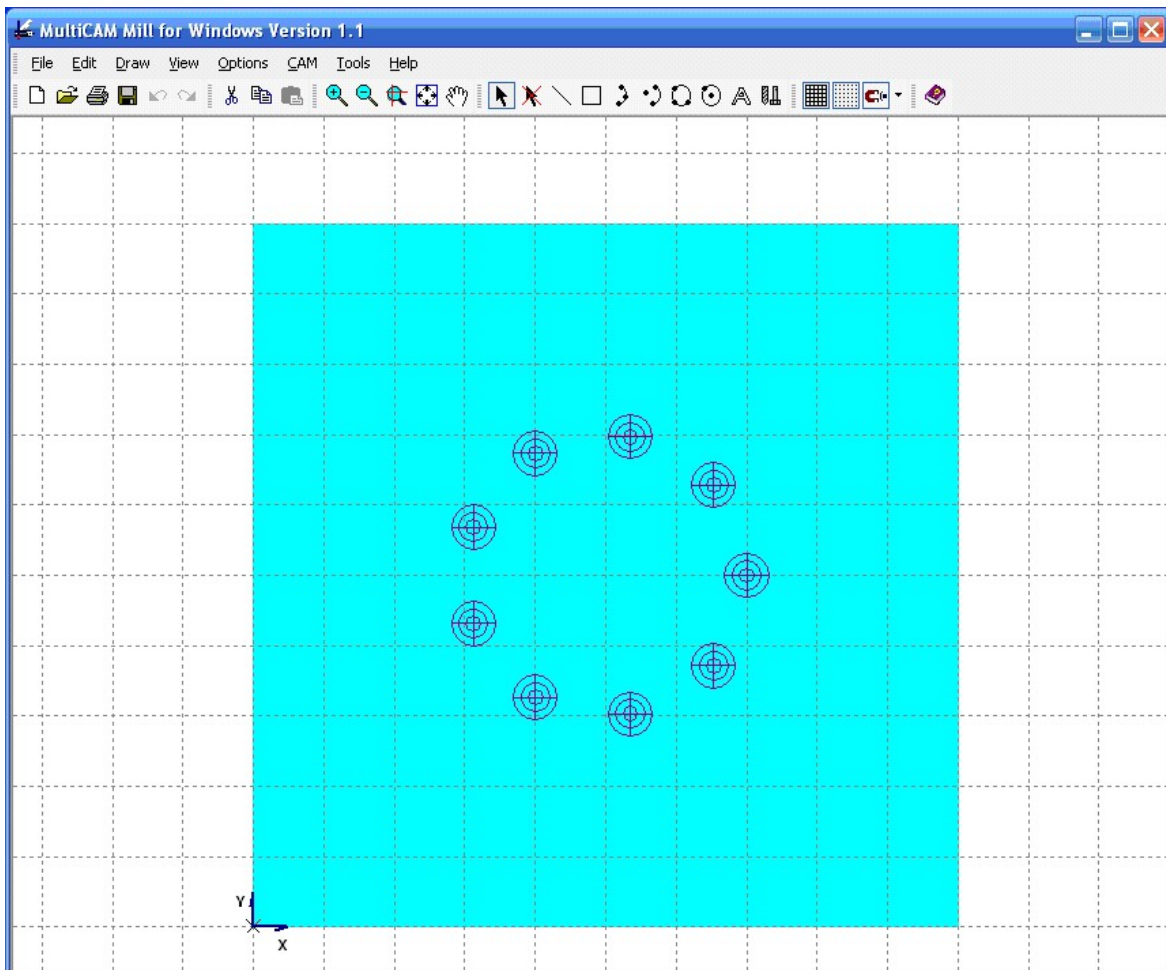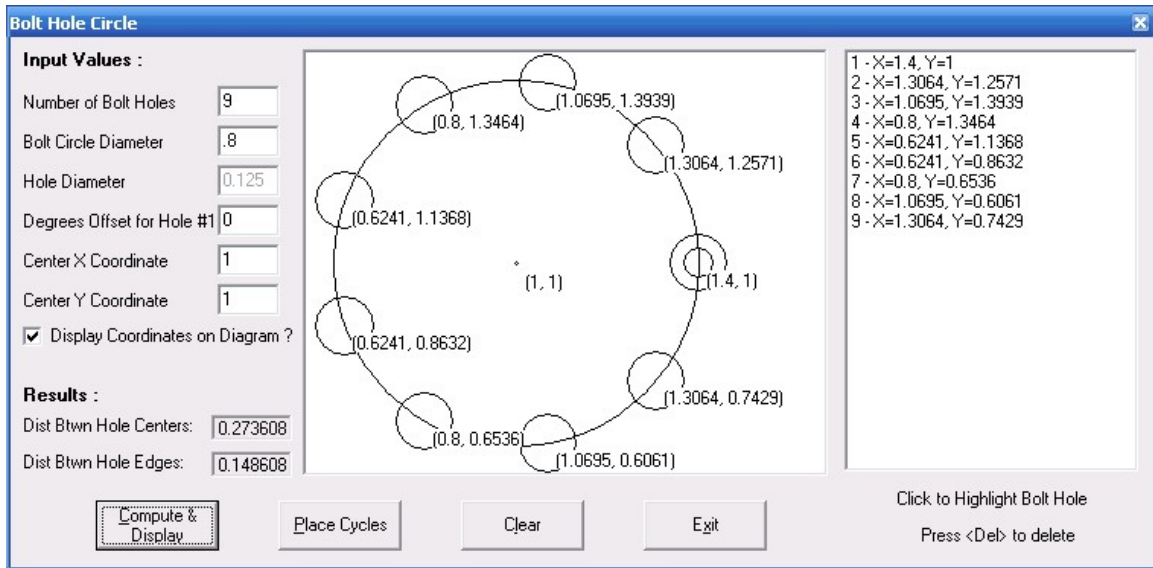
Advanced Drawing Features

Using the Visual Grid

The visual grid is like graph paper. You get horizontal and vertical lines to help guide your drawing. You can turn this ON or OFF by clicking on the appropriate icon or selecting **Options | Grid/Snap Selections** and checking the visual grid checkbox. The value in the entry for the visual grid is the distance between the lines.

Using the Cursor Snap Grid

The cursor grid restricts the cursor to specified points. You can turn this option ON or OFF by clicking on the appropriate icon or selecting **Options | Grid/Snap Selections** and checking the cursor grid checkbox. The value in the entry for the cursor grid is the distance between the snap points along each axis.

Using the Object Snap Mode

The object snap mode allows you to snap to points on objects in the drawing. These include endpoints, middle points, perpendicular points as well as circle center and quadrant points. This can be very useful in getting new drawing entities to start and end on features of existing ones.

Select **Options | Grid/Snap Selections** to enter the desired settings.

Displaying the End Markers

When many objects are close together it can be hard to tell where a line or arc starts and ends. Selecting *View | Show* **End Markers** places a marker at the beginning and end of each line segment or arc.

Displaying the Tool Path

When placing a tool path you'll want to have this option checked so you can see the tool paths. Tool paths are what ultimately will be the source for your CNC program.

Displaying Tool Radius Outline

Tool radius outline can aid in visually verifying what will be machined and what will not. This will give you a visual representation of the diameter of the cutting tool. Turn this on or off by checking this item.
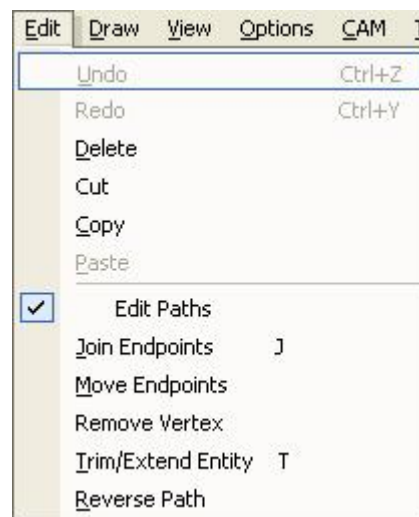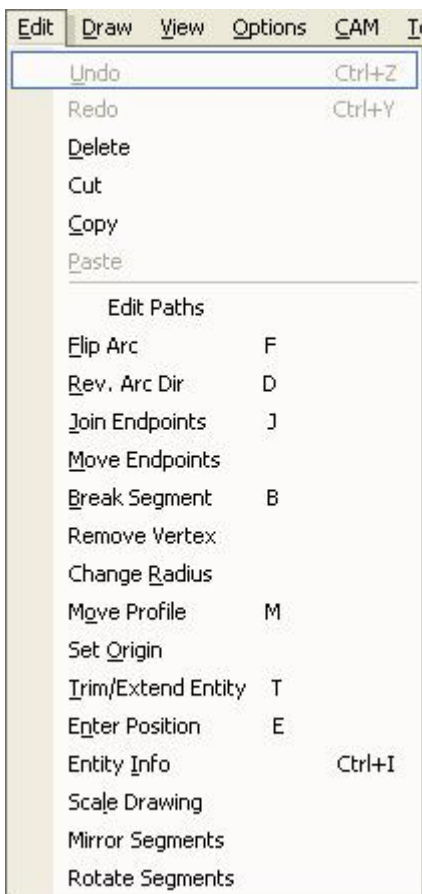
Using the Drawing Editing Features

Most of the editing functions are self explanatory with the following added notes.

The **Edit | Trim/Extend Entity** function can truncate or extend a line or arc to meet another object. If the object does not cross the one to be trimmed, the software will automatically assume the extend mode.

Holding the <Shift> key while drawing or editing can turn on/off the object snap mode.

When copying and pasting objects, the reference point is the X marker for both operations.

If you select several objects ahead of time and then copy or delete, all selected segments will be operated on.

Edit	Draw	View	Options	CAM	To
Undo				Ctrl+Z	
Redo				Ctrl+Y	
Delete					
Cut					
Copy					
Paste					
Edit Paths					
Flip Arc		F			
Rev. Arc Dir		D			
Join Endpoints		J			
Move Endpoints					
Break Segment		B			
Remove Vertex					
Change Radius					
Move Profile		M			
Set Origin					
Trim/Extend Entity		T			
Enter Position		E			
Entity Info				Ctrl+I	
Scale Drawing					
Mirror Segments					
Rotate Segments					

Edit	Draw	View	Options	CAM	To
Undo				Ctrl+Z	
Redo				Ctrl+Y	
Delete					
Cut					
Copy					
Paste					
✓ Edit Paths					
Join Endpoints		J			
Move Endpoints					
Remove Vertex					
Trim/Extend Entity		T			
Reverse Path					

Using the Path Editing Mode

You can edit the paths instead of the drawing objects by enabling the **Edit Paths** mode in the edit menu. The commands displayed in the editing menu will be those applicable to path segments.

Selecting Entities

Choose *Select* mode (the arrow icon), then click on each item or click and drag to select multiple objects. All entities that are even partially in the dashed rectangle will be selected. To select only those items that are totally within the rectangle, hold the shift key simultaneously.

Unselecting Entities

Choose the *Unselect* mode (the arrow icon with a red X), then click on each item or click and drag to unselect multiple objects. All entities that are even partially in the dashed rectangle will be unselected. To unselect only those items that are totally within the rectangle, hold the shift key simultaneously.

Note: When placing a tool path, if the path will come to a "fork in the road" you can pre-select the segments of interest and it will follow the selected portion.

Placing Text

Select the **Draw | Text** command or click on the text icon. The font selections will be those available on your computer. Most fonts are made up of straight line segments and frequently contain some with zero length. After placing the text you should use the **Tools | Cleanup Drawing** command to automatically delete these. Text may come out small in size compared to your machining needs, choose a 72-point size and/or scale the drawing afterwards using the *Edit | Scale Profile* command.
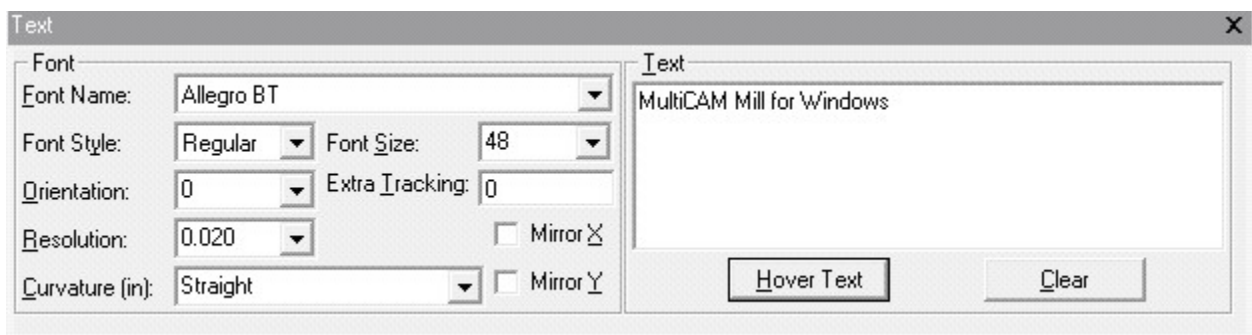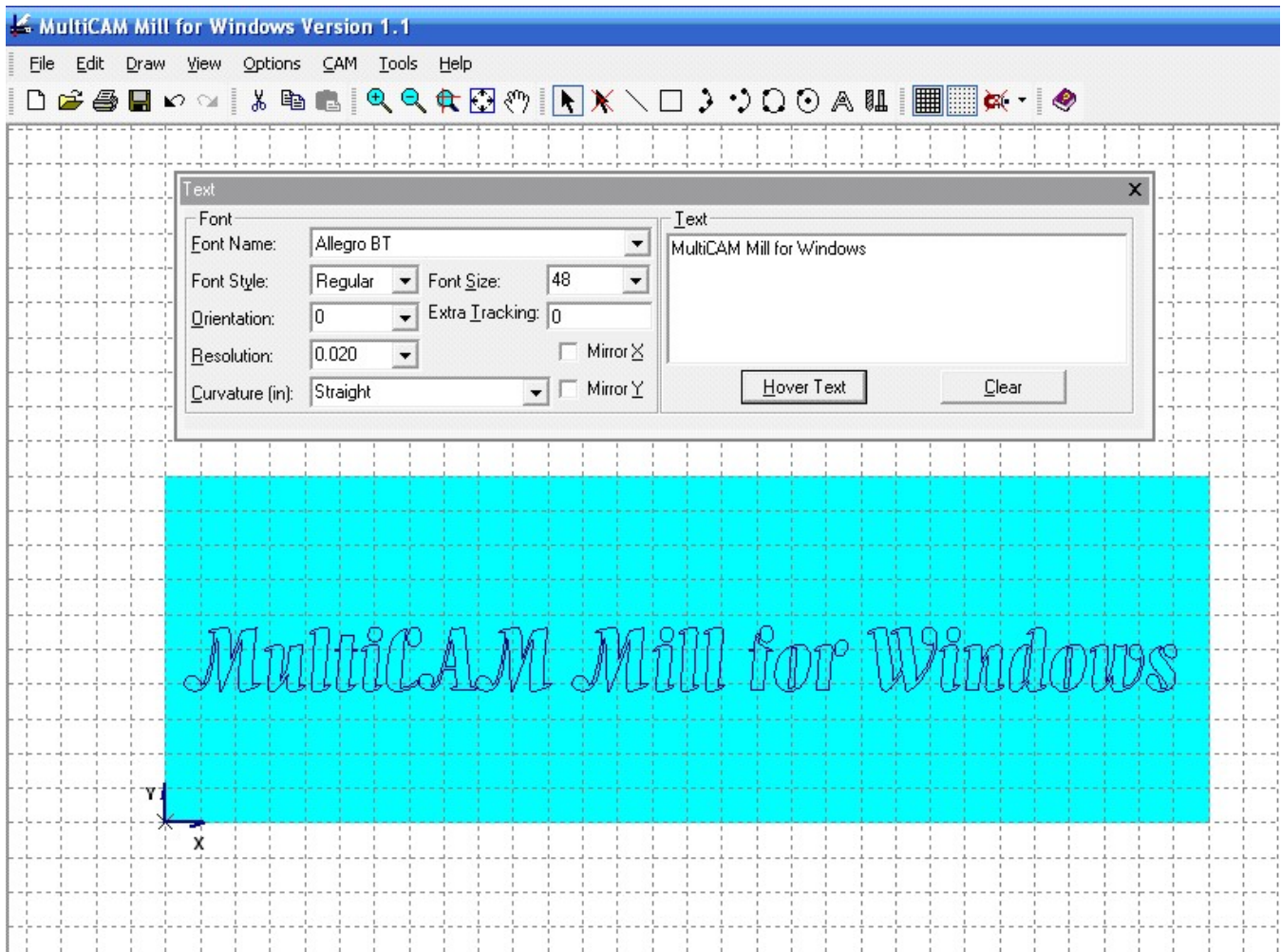
Once you have the text placed the way you want it, it is easy to select the entire drawing area and select **CAM | Quick Place Selected Segs** to create the tool path.

Generally you'll need a very fine endmill for most engraving work. This may be in the 0.010 to .050 range for most applications.

Editing Segments in the Entities Window

There are two ways to edit entities. You can use the commands available in the edit menu. Alternatively, you can select entities and adjust them numerically in the properties window. Numerical adjustments allow a high degree of precision.
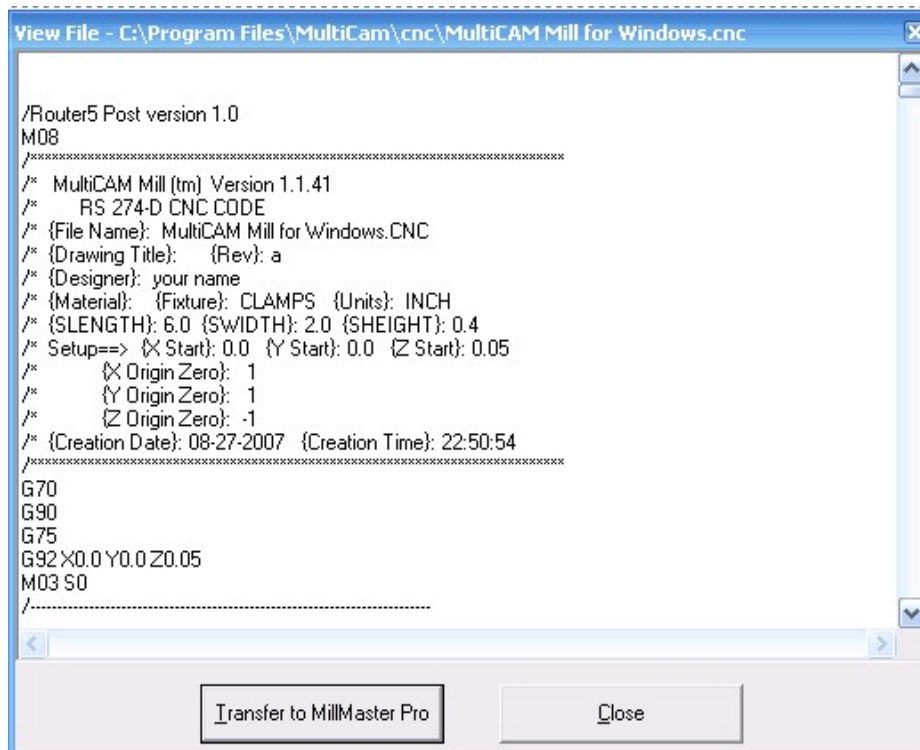
Editing Path segments

There are two ways to edit tool paths. You can use the commands available in the edit menu while the edit paths item in the edit menu is checked. Alternatively, you can select the *CAM | Edit Path/Cycle* menu command.

Creating CNC Files

Once you have the tool paths and any cycles completed, select *CAM | Make CNC Mill File* or *CAM | Make CNC Lathe File.*

The G-code program will be created and displayed. Click on **Transfer to MillMaster Pro** (or **Transfer to MillMaster Pro**) and Viola!

```
View File - C:\Program Files\MultiCam\cnc\MultiCAM Mill for Windows.cnc

/Router5 Post version 1.0
M08
/*****************************************************************
/*  MultiCAM Mill (tm)  Version 1.1.41
/*      RS 274-D CNC CODE
/* {File Name}:  MultiCAM Mill for Windows.CNC
/* {Drawing Title}:     {Rev}: a
/* {Designer}:  your name
/* {Material}:    {Fixture}: CLAMPS   {Units}:  INCH
/* {SLENGTH}: 6.0  {SWIDTH}: 2.0  {SHEIGHT}: 0.4
/* Setup==>  {X Start}: 0.0  {Y Start}: 0.0  {Z Start}: 0.05
/*         {X Origin Zero}:  1
/*         {Y Origin Zero}:  1
/*         {Z Origin Zero}:  -1
/* {Creation Date}: 08-27-2007  {Creation Time}: 22:50:54
/*****************************************************************
G70
G90
G75
G92 X0.0 Y0.0 Z0.05
M03 S0
/-------------------------------------------------

        [Transfer to MillMaster Pro]        [Close]
```
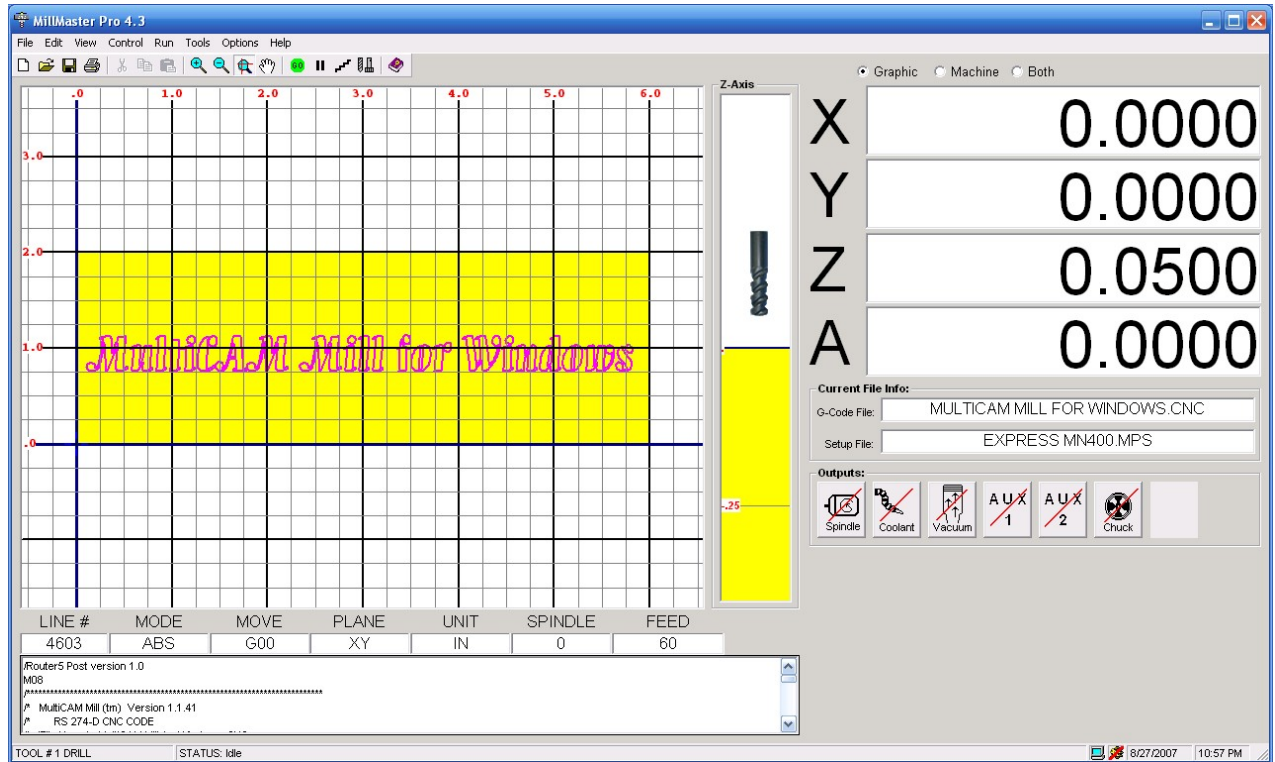
Appendix A - Working with Lathes

Although **MultiCAM**™ **2013** was designed to work with with mills and **MillMaster Pro**™, you can also design simple lathe parts and send the resulting part programs to **TurnMaster Pro**™.

First, some background information:

MultiCAM™ **2013** uses the familiar (X, Y) coordinate plane, with a Z-axis for tool depth. For lathes, the horizontal axis is Z, the vertical axis is X, and there is no axis for tool depth. So what you see on MultiCAM's X-axis will be converted to the Z-axis in **TurnMaster Pro**™ and what you see on on MultiCAM's Y-axis wil be converted to the X-axis. Don't be concerned about this conversion; simply design your part as it will look on the lathe.

Since the horizontal axis on a lathe usually starts at 0, and moves negative as the tool moves left, it is advisable to set your part size with a negative width. Also, since you are working with only the top half of the part for a lathe, the height should be set to the **radius** (or ½ of the diameter) of the part. What you will see in **MultiCAM**™ **2013** is only the top half of the part.

For example if you had lathe stock of 4" width and 2" diameter (1" radius), you may set up your material dimensions as follows:

Remember the X and Y axes are the horizontal and vertical axes and will be mapped to your lathe's Z and X axes respectively when you create the CNC lathe file. The Z-axis you see on the screen is not applicable to lathes and will be ignored.

With these parameters, you should see the following blank part. Note that the X-axis (Z-axis on the lathe) starts at 0, and moves negative as the tool moves left.
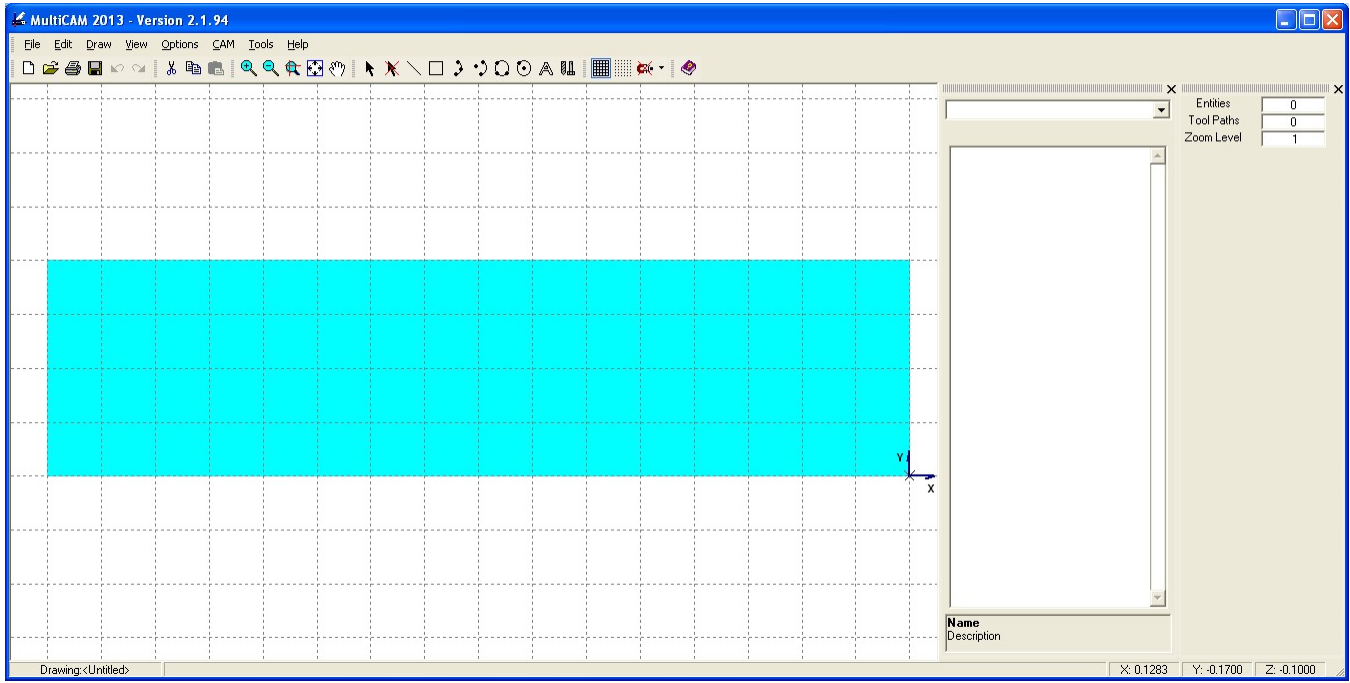
Now you can begin designing your part.

Suppose your part design looks as follows:
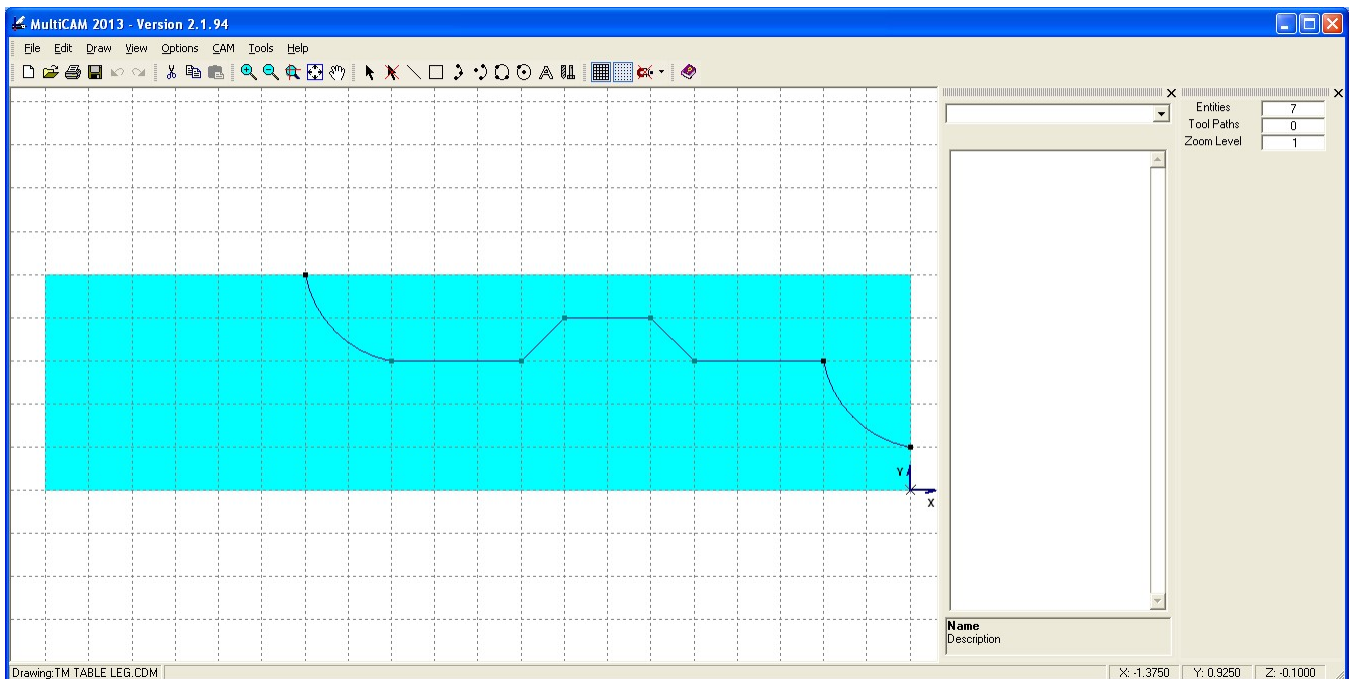
To rough cut the part, do the following:

- close the pocket by drawing lines from X, Y points (0, 0.2) to (0, 1) and from (0, 1) to (-2.8, 1).

- set the **Pocketing/Profiling Specs** from the **CAM** dropdown menu.

- Before clicking the **Lathe Style Pocketing** checkbox, be sure **Final Depth** and **Depth per Pass** have the same values. These values are ignored during pocket placement. When placing finishing paths however, if these values are different, extra path segments may be drawn.

- Be sure the **Pocketing Style** is **Horizontal Only** and the **Lathe Style Pocketing** checkbox is checked.

- make sure the *Edit Paths* option is NOT checked in the **Edit** dropdown menu.

- click **Select All** line segments from the **Draw** dropdown menu.

- click **Verify/Accept Pocket** from the **CAM** dropdown menu.

- click **Rough Cut Pocket** from the **CAM** dropdown menu

Now you can add the finishing cut. Do the following:

- Remove the lines from (X, Y) points (0, 0.2) to (0, 1) and (0, 1) to (-2.8, 1).

- Click **Place Finish Path** from the **CAM** dropdown menu.

- Successively click near the start point (0, 0.2), then second point (-0.4, 0.6), and a finish reference point.

To create a lathe CNC file and export the file to **TurnMaster Pro**™ do the following:

- Create a lathe CNC file by clicking **Make CNC Lathe File** from the **CAM** menu.
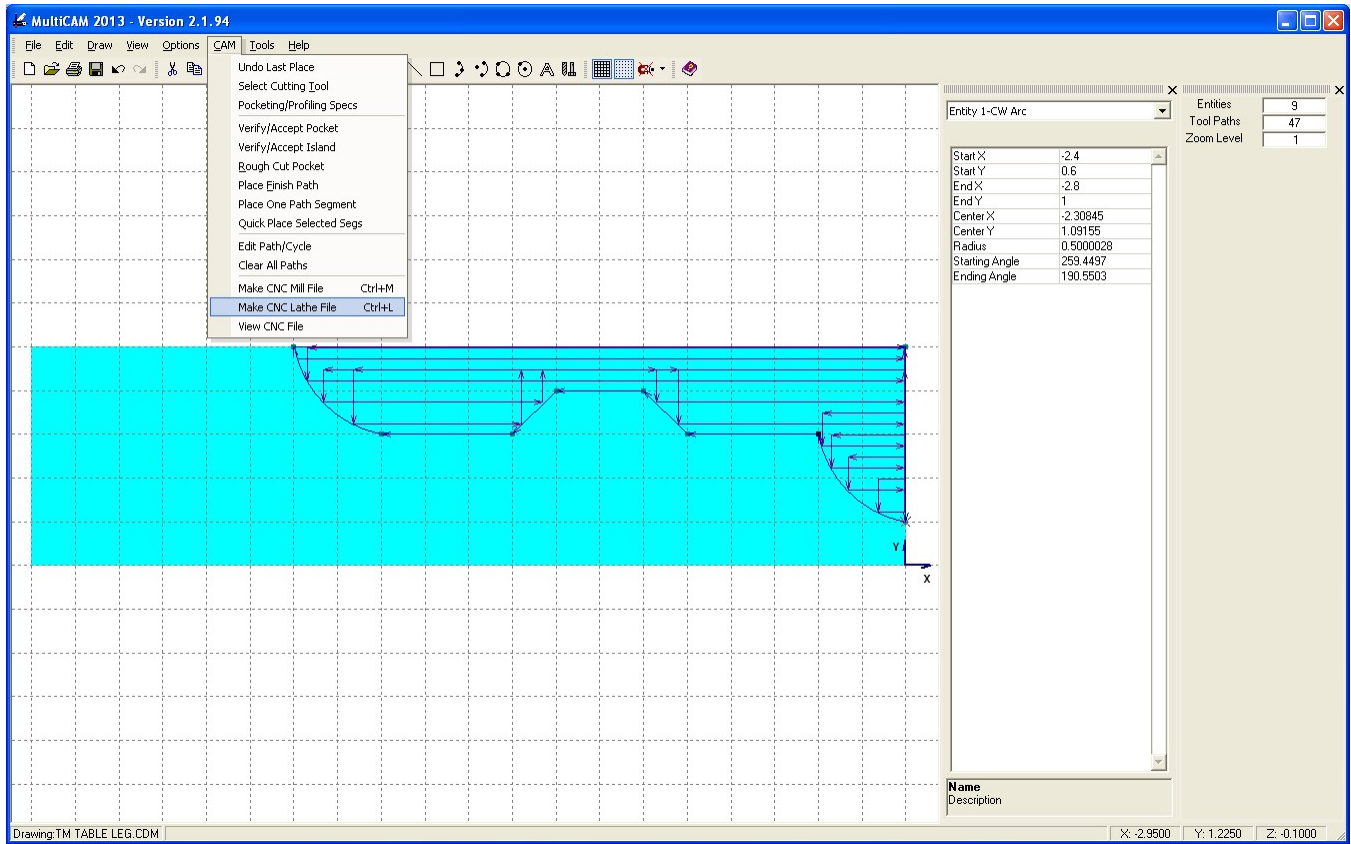
Click **Transfer to TurnMaster Pro** from the **View File** screen.

```
View File - C:\sourcesafeprojects\MCFW\CNC\TM Table Leg.cnc          [x]

/********************************************************************
/*   MultiCAM Mill (tm)  Version 2.1.94
/*      RS 274-D CNC CODE
/*  {File Name}:  TM Table Leg.CNC
/*  {Drawing Title}:     {Rev}: a
/*  {Designer}:  your name
/*  {Material}:
/*  {Material Shape}:  Rectangular
/*  {Fixture}:  CLAMPS   {Units}:  INCH
/*  {PART LENGTH}: 4.0  {O.D.}: 2.0  {I.D.}:  0.0
/*  {{Z ORIGIN ZERO}: -1
/*  Setup==>  {X Start}: 1.0   {Z Start}: 0.0
/*  {Creation Date}: 08-12-2013  {Creation Time}: 12:56:26
/*  Created for TurnMaster Pro (tm) 2012
/********************************************************************
G70
G90
G75
/
/---------------------------------------------------------
/    << Tool #1 code >>
/---------------------------------------------------------
M06 T1
M03 S0
G00 X0.94655 Z-2.78176

/    << Group #1  >>
G01 Z-0.005 F60
G00 X0.99655
G00 Z-2.73861
```

[Transfer to TurnMaster Pro]   [Close]

In **TurnMaster Pro**™ be sure you run your program in graphics mode before attempting any machining. You may also want to run your program at least once in *Single-Step* mode to verify the order and accuracy of the machining.

There may be some fine-tuning needed before the G-Code program runs as expected.

Here's how this part turned out.
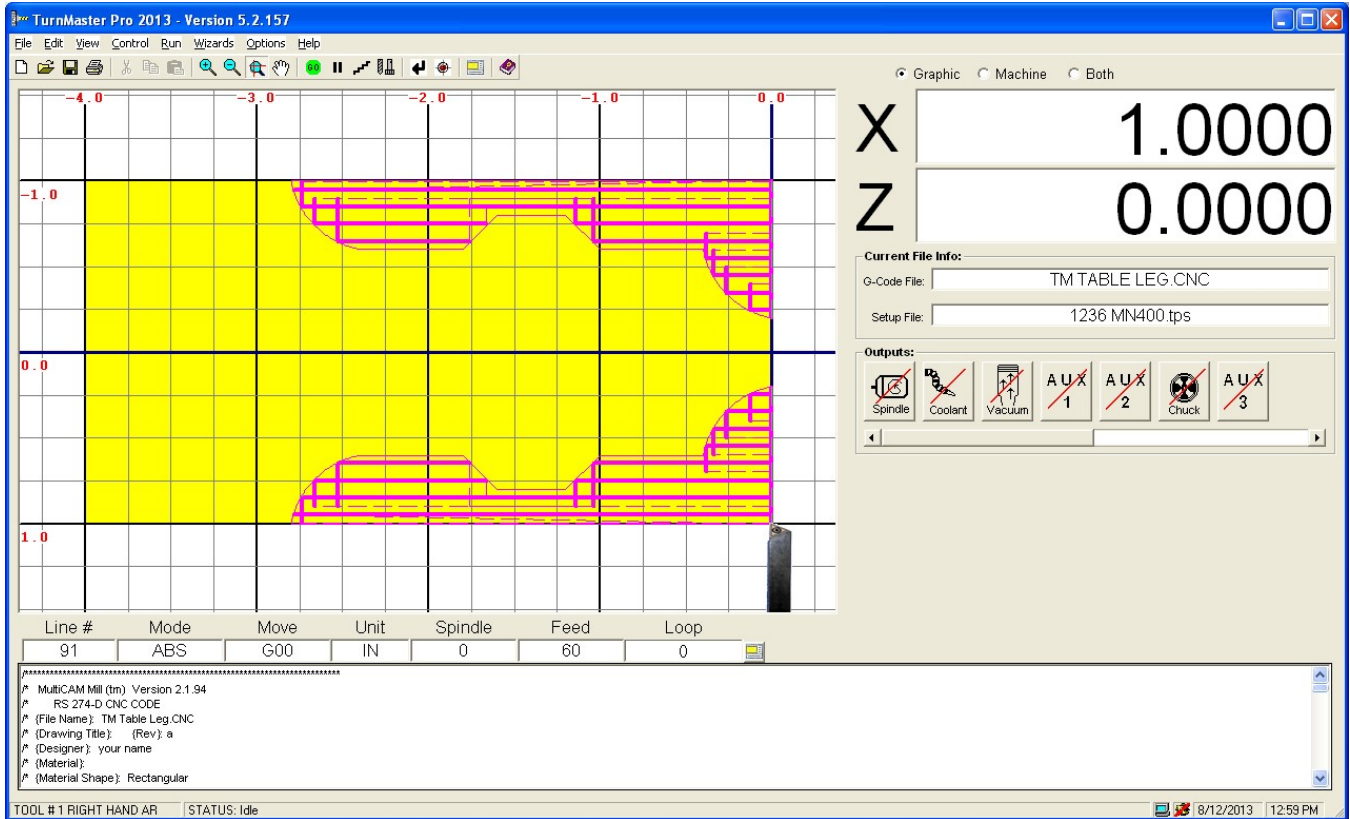
```
/**********************************
/*   MultiCAM Mill (tm)  Ver 2.1.94
/*        RS 274-D CNC CODE
/* {File Name}:  TM Table Leg.CNC
/* {Drawing Title}:      {Rev}: a
/* {Designer}:  your name
/* {Material}:
/* {Material Shape}:  Rectangular
/* {Fixture}: CLAMPS  {Units}: INCH
/* {PART LENGTH}: 4.0  {O.D.}: 2.0
/* {I.D.}:  0.0
/* {{Z ORIGIN ZERO}: -1
/* Setup==>  {X Start}: 1.0
/*           {Z Start}: 0.0
/* {Creation Date}: 08-12-2013
/* {Creation Time}: 12:56:26
/* Created for TurnMaster Pro (tm)
2012
/**********************************
G70
G90
G75
/
/-----------------------------------
-----------------------------------
/     << Tool #1 code >>
/-----------------------------------
-----------------------------------
M06 T1
M03 S0
G00 X0.94655 Z-2.78176

/     << Group #1  >>
G01 Z-0.005 F60
G00 X0.99655
G00 Z-2.73861
G01 X0.84655
G01 Z-0.005
G00 X0.89655
G00 Z-2.66349
G01 X0.74655
G01 Z-1.66052
G00 X0.89655
G00 Z-1.13948
G01 X0.74655
G01 Z-0.005
G00 X0.89655
G00 Z-2.5254
G01 X0.64655
G01 Z-1.76052
G00 X0.89655
G00 Z-1.03948
G01 X0.64655
G01 Z-0.005
G00 X0.69655
G00 Z-0.38176
G01 X0.54655
G01 Z-0.005
```

```
G00 X0.59655
G00 Z-0.33861
G01 X0.44655
G01 Z-0.005
G00 X0.49655
G00 Z-0.26349
G01 X0.34655
G01 Z-0.005
G00 X0.39655
G00 Z-0.12539
G01 X0.24655
G01 Z-0.005
G00 X0.99655

/     << Group #2  >>
/-----------------------------------
-----------------------------------
/     << Tool #2 code >>
/-----------------------------------
-----------------------------------
M06 T2
G00 X0.2 Z0.0
G02 X0.6 Z-0.4 R0.5
G01 Z-1.0
G01 X0.8 Z-1.2
G01 Z-1.6
G01 X0.6 Z-1.8
G01 Z-2.4
G02 X1.0 Z-2.8 R0.5
G01 Z0.0
G01 X0.2 Z0.0
/-----------------------------------
-----------------------------------
/     << Reset to first tool used.
Tool #1 >>
/-----------------------------------
-----------------------------------
M05
M06 T1
/
M05
M25
G74
M02
```

Appendix B - Single-Stroke Fonts in MultiCAM

Most *TrueType* fonts supplied with **Windows** are based on character outlines. They include a set of equations for drawing lines and curves to create an outline shape or *glyph*. After drawing the outline of a character, it is *filled in* for display purposes. This works well when displaying characters on a CRT screen.

For example:

A A

 However, when engraving text where the desired width of each character stroke is the same as the width of the engraving tool, it would be preferable to have a *single-stroke font*. These types of fonts may also be referred to as *stroke-based fonts, single-line fonts, stick-figure fonts, machining fonts,* or *engraving fonts*.

These fonts are clearly preferable due to the reduced number of machining moves as well as the cleanliness of the output. For example:

ABCDEOS abcdeos 2468
ABCDEOS abcdeos 2468

In order for the *MultiCAM™ 2013* `Draw|Text` function to create g-code for *single-stroke fonts*, one or more of these fonts must be installed on the PC.

Windows usually comes preloaded with quite a few fonts. Unfortunately, none of these are *single-stroke fonts.* There are literally thousands of fonts you can purchase or download for free via the web. Suprisingly, however, *single-stroke fonts*, are somewhat hard to find. We found some free fonts at:

   http://www.featurecam.com/general/support/engrave_fonts.asp
   http://www.mr-clipart.com/int/freestuff.php

We also found some packages for sale. The best approach is to do some web searches using some of the terms already mentioned, to see the option best suited for your application.

Installing Fonts:

Once you have found an appropriate *single-stroke font*, you will need to install it.

TrueType font files have a **.TTF** extension.

If you downloaded the font, then the file may be in a temporary download directory, or in a temporary directory you specified. If you purchased a CD, then the vendor will tell you where the file(s) is.

- Click **Start | Settings | Control Panel** *(Windows XP)*
     or **Start | Control Panel** *(Windows Vista)*
- Click **Fonts**
- Click **File | Install New Font.**
- Select file location and the font to install, and press **OK**
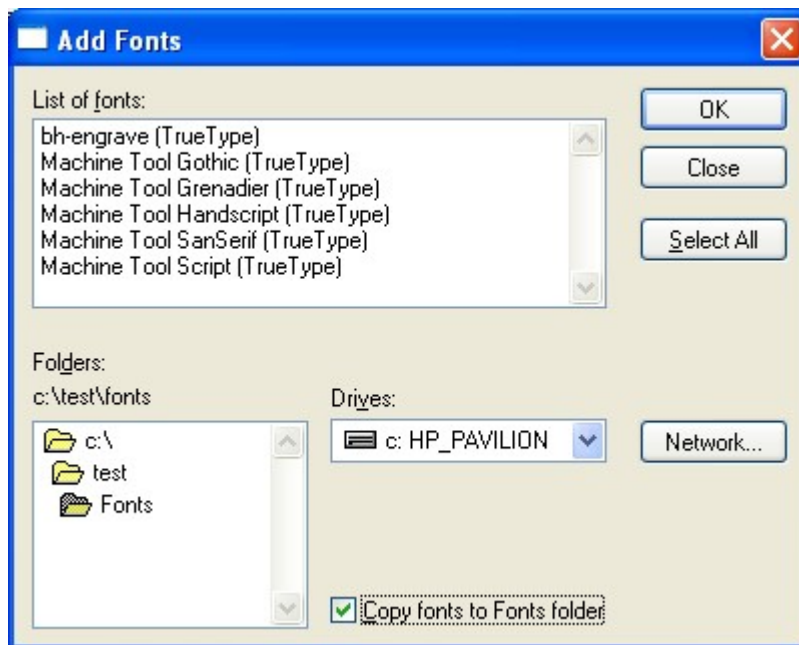
The installed font should now be available to the ***MultiCAM Mill***™ `Draw|Text` function.
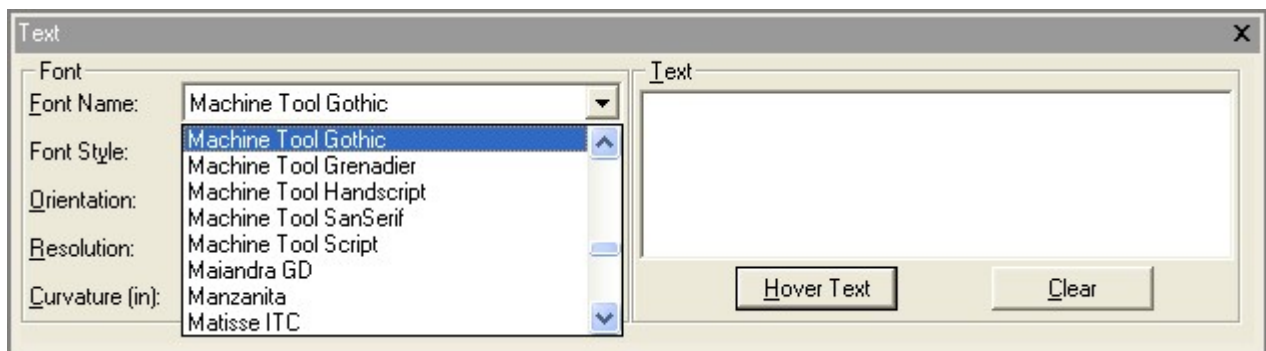
Appendix C - Glossary of Hotkeys

As in most *Windows®* applications hotkeys play an important role in improving the user experience and increasing productivity. Many common menu selections can be performed with hotkeys. Here is a list of available hotkeys with a brief description of their usage.

Hotkey	Description	Menu Selection
\<Ctrl\> Z	Undo last edit, place or delete.	**Edit | Undo**
\<Ctrl\> Y	Redo last undo.	**Edit | Redo**
\<Ctrl\> I	Display segment entity information.	**Edit | Entity Info**
\<Ctrl\> A	Select all segment entities.	**Draw | Select All**
\<Ctrl\> U	Unselect all segment entities.	**Draw | Unselect All**
\<Ctrl\> M	Make .CNC file for *MillMaster*.	**CAM | Make CNC Mill File**
\<Ctrl\> L	Make .CNC file for *TurnMaster*.	**CAM | Make CNC Lathe File**
\<F1\>	Display help window.	Help Contents on Toolbar
\<F3\>	Repeat last edit.	
\<F12\>	Repeat last edit. (same as **\<F3\>**)	
G	Toggle visual grid mode.	VGrid on Toolbar
\<Del\>	Delete selected entries or enter Delete mode.	**Edit | Delete**
C	Copy a segment.	**Edit | Copy**
P	Paste the last copied segment.	**Edit | Paste**
F	Flip arc (mirror image).	**Edit | Flip Arc**
D	Reverse arc direction (complete the circle).	**Edit | Reverse Arc Dir**
J	Join endpoints.	**Edit | Join Endpoints**
V	Move endpoints.	**Edit | Move Endpoints**
B	Break segment.	**Edit | Break Segment**
M	Move profile.	**Edit | Move Profile**
T	Trim or extend segment.	**Edit | Trim/Extend Entity**
E	Position cursor to exact location.	**Edit | Enter Position**
S	Scale entire drawing.	**Edit | Scale Drawing**
L	Draw line.	**Draw | Line**
R	Draw rectangle.	**Draw | Rectangle**
A	Draw 3-point arc.	**Draw | Arc 3P**
I	Zoom drawing in.	**View | Zoom In**
O	Zoom drawing out.	**View | Zoom Out**