CNC Setup

User's Guide

Version 1.0.0   4/30/2007

KCD software   Artistic AUTOMATION™

Table of Contents

Software License Agreement

Carefully read the following terms and conditions sign and return as accepting these terms and conditions.

This is a legal agreement between you, the end user (either an individual or an entity), and SA International, Inc. If you do not agree with the following, you should promptly return the package. Use of this program indicates your acceptance of the terms and conditions stated below.

The enclosed computer program "Software" is licensed, not sold, to you by SA International, Inc., for use on a non-exclusive, non-transferable basis, only under the following terms, and SA International, Inc., reserves any rights not expressly granted to you. You may not disclose to any third party any confidential information concerning the Software or SA International, Inc. or use such confidential information to the detriment of SA International, Inc..

1. License.

This software is protected by the United States Copyright Law and International Treaty Provisions. Therefore, you must treat the Software just as you would any other copyrighted material, such as a book. This license allows you to:

> (a) Make one copy of the Software in machine readable form; provided that such a copy of the original may be used solely for backup purposes. As an express condition of this License, you must reproduce on each copy the SA International, Inc., copyright notice and any other proprietary legends on the original copy supplied by SA International, Inc.

> (b) Transfer the Software and all rights under this License to another party together with a copy of this License and all written materials accompanying the Software provided you give SA International, Inc., written notice of the transfer and the other party reads and agrees to accept the terms and conditions of this License.

> (c) Use this Software on a single computer only, but may transfer it to another computer as long as it is used on only one computer at a time. "In Use" constitutes being loaded onto either temporary (i.e., RAM) or permanent memory (e.g., hard disk, CD-ROM or other storage device) of a computer.

2. Restrictions.

You may NOT distribute copies of the Software to others or electronically transfer the Software from one computer to another over a network. You may not de-compile, reverse engineer, disassemble or otherwise reduce the Software to a human perceivable form. You may not modify, adapt, transfer, rent, lease, loan, resell for profit, distribute, network or create derivative works based upon the software or any part thereof.

3. Termination.

This License is effective until terminated. This License will terminate immediately if you fail to comply with any of its provisions. Upon termination, you must return the Software, and all copies thereof, to SA International, Inc., and you may terminate this License at any time by doing so.

4. Export Law Assurances.

You agree that neither the Software nor any direct product thereof will be transferred or exported, directly or indirectly, into any country prohibited by the United States Export Administration Act or any international export laws and the restrictions and regulations thereunder, nor will it be used for any purposes prohibited by the Act or laws.

5. Warranty Disclaimer, Limitation of Remedies and Damages.

In no event will SA International, Inc., or Artistic Automation, Inc. be liable for any damages, including infringement, lost data, lost profits, cost of cover or other special, incidental, consequential or indirect damages arising from the use of the program however caused and on any theory of liability. This limitation will apply even if SA International, Inc., or an authorized dealer or distributor has been advised of the possibility of such damage. SA International, Inc. and Artistic Automation, Inc. MAKES NO WARRANTY, EXPRESS OR IMPLIED, WITH RESPECT TO THE SOFTWARE, AND DISCLAIMS, WITHOUT LIMITATION, ANY IMPLIED WARRANTY OF MERCHANTIBILITY OR FITNESS FOR A PARTICULAR PURPOSE. SA International, Inc. does not warrant any drivers for any CNC devices. These drivers are provided for our customers as a service only, and were developed using information provided to us at the time by the equipment manufacturers.

SA International, Inc. is not responsible for any typographical errors in the software or in the documentation.

6. General.

If you are a U.S. Government end-user, this License of the Software conveys only "RESTRICTED RIGHTS," and its use, disclosure, and duplication are subject to Federal Acquisition Regulations, 52.227-7013(c) (1) (ii). This License will be construed under the laws of the State of Pennsylvania, except for that body of law dealing with conflicts of law, if obtained in the U.S., or the laws of jurisdiction where obtained, if obtained outside the U.S. If any provision of this License is held by a court of competent jurisdiction to be contrary to law, that provision of this License will remain in full force and effect.

Windows is a registered trademark of Microsoft Corporation. The names of actual companies and products mentioned herein may be the trademarks and/or registered trademarks of their respective owners.

SA International, Inc.
International Plaza Two, Suite 625
Philadelphia, PA 19113-1518

1. CNC Setup Overview

CNC Setup is a companion application to KCDw's One Button Machining solution for nested-based manufacturing. CNC Setup provides a method for defining all of the information necessary to process the cabinet geometry and then create the nested output files that will be sent to your CNC machine. You provide information about your machine configuration, the tools and toolpath strategies you will use, and how the geometry created in KCDw is to be processed, and then all this information is saved so that it can be automatically used by KCDw to process parts.
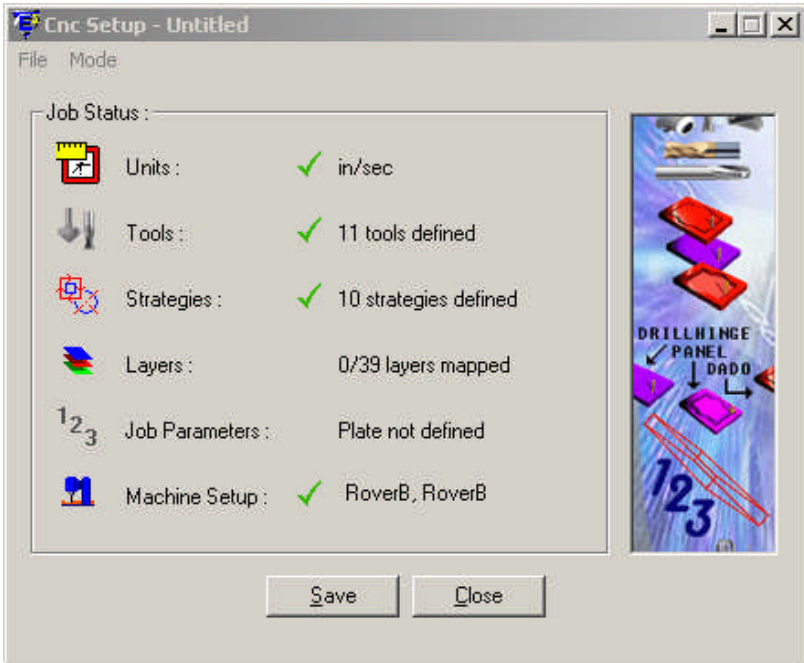
Summary and Navigation Dialog

Purpose of CNC Setup

The purpose of CNC Setup is to allow you to tell KCDw how you want to process the individual parts it creates from your design. You set parameters defining the following items.

- What tools you will use

- The methods for milling, drilling, and cutting your parts

- How the parts are to be nested
- The type, model, and configuration of your CNC machine

- How the machine output should be ordered

After you have entered all of the information about how your jobs will process, it is saved into a file on your computer that is used for the One-Button Machining in KCDw.

The Setup Process

CNC Setup provides two methods for working through the steps in the setup process, the **Wizard Mode** and the **Advanced Mode.** The Wizard method guides you through each of the steps sequentially, ensuring that nothing is left out. In the Advanced mode it is easier to move between each of the parameter topics. In both modes each of the necessary elements is checked, and any missing information is highlighted.

The dialog on the previous page shows the default dialog in the Advanced Mode. You can see that there are six different categories of information. These categories correspond to the steps presented in the Wizard Mode. The Wizard Mode is a good starting point, and it emphasizes all of the different types of information required for processing part information. After using the program a few times, you will likely feel comfortable with the needed information and then the Advanced Mode will be an easier way to get to the specific parameters you want to edit.

Following is the introductory screen in the Wizard Mode.

Organization of this Manual

After this introductory chapter, the chapters of this document are organized to first show how to navigate in the program, and then how to enter all of the detailed information necessary to successfully process a file.

The method for entering information into each of the dialogs is essentially the same in both the Advanced Mode and Wizard Mode, so after some explanation about how to move within and between the two modes they are treated as being the same.

2. Navigation

Wizard Mode

The Wizard Mode leads you through a step-by-step process for entering the required parameters. Within each step you are presented with the status of the information in that step, along with the ability to activate the data entry dialog so you can add or modify information as necessary.

Each of the wizard steps is largely self explanatory, and it would be redundant to explain each of the wizard screens here. There are a few elements of the dialogs that are worth noting.

- **Advanced Button –** At any time in the Wizard you can toggle over to the Advanced mode simply by clicking on the *Advanced* button.

- **Next/Back Buttons –** This allows you to move to the *Next* step after you have entered all of the information. If you want to move back to check information you have already entered, just click on *Back.*

  Note: If the *Next* button is not active this means that you need to enter additional information in order to complete the current step. When you are at the Layer Mapping step you must map all of the layers before the wizard will let you move on to the next step.

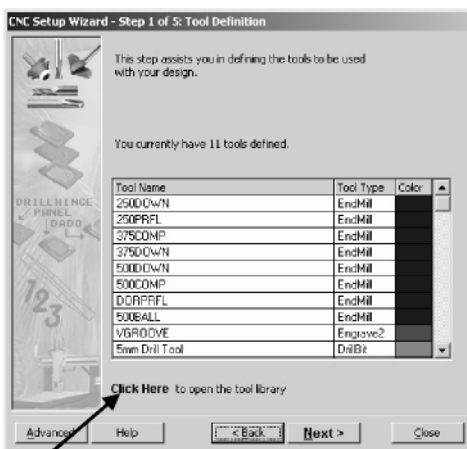- **Click Here –** When you see this text in the steps, you can click on the text in order to activate the specific dialog for the information required in that step. This is where you will enter the parameters for that topic.

Move to the next step

Click to toggle to Advanced Mode

Introductory Screen

Click on the 'Click Here' text to activate the dialog for each specific step.

Step 1 Screen

Advanced Mode

The Advanced Mode is likely how you will actually use CNC Setup on a regular basis. Its advantage is that it allows you to move between each of the data dialogs without having to do it sequentially, and without the limitation of having to complete one step before moving on to the next step.

The main dialog screen in the Advanced Mode provides a summary of the information that has been set up, and the information that still needs to be added. It also allows you to navigate between the different areas of the program. The dialog is shown below with notes that highlight key areas. A discussion of each item follows.
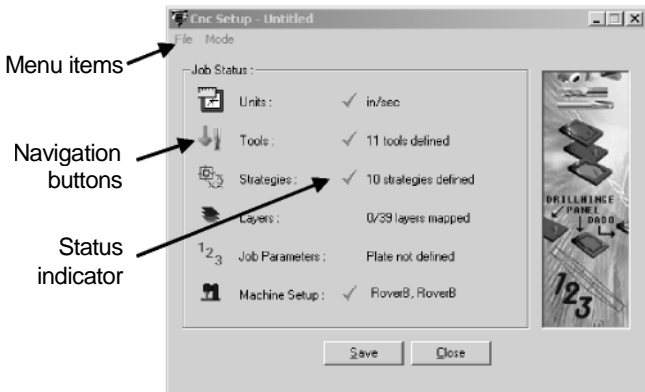
Advanced Mode Main Dialog

Menu items – The File and Mode menu items provide items for saving the settings file, opening an existing file, creating a new file, and switching between the advanced and wizard modes.
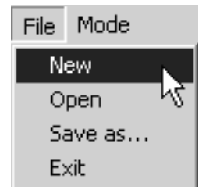
Navigation buttons – There are six different topics that are represented by the items in the table shown in this dialog. The icons on the left side are buttons that, when clicked, will open its respective topic area so that information may be reviewed and edited.

Status indicator – The green check marks indicate that the respective topic area has enough information to successfully process a job. It doesn't necessarily mean that additional information is not required. This status indicator is most meaningful in the bottom three categories. In the Layers category you won't see the check mark until all the layers have been

mapped. Job Parameters requires that the size of the material is defined, and the Machine Setup topic requires that an output device is selected.

Save and Close buttons – These buttons provide a quick way to save the current settings into the '.atp' settings file, and to exit CNC Setup. If you have edited the current settings but haven't yet saved the file, you will be asked to save the file before exiting the software.
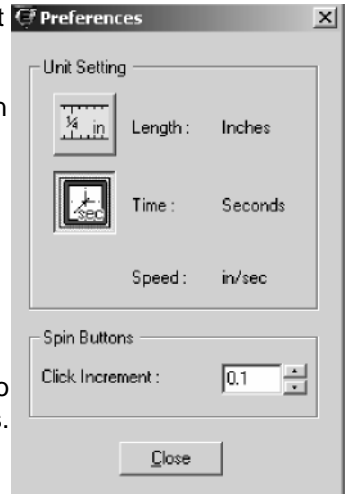
3. Preferences

Units

The preferences dialog is simple, but it is important because this is where you set the units you will use in all the other parameter dialogs. By clicking on the length icon you toggle between inches and millimeters, and by clicking on the time icon you toggle between seconds and minutes.

These units are used for defnining depths and feed rates for your strategies.

Note that these units do not apply to the machine setup parameters. Oftentimes a machine requires that output is sent in particular units, such as millimeters per minute, that are not convenient parameters to work in. Machine parameters and output units are defined separately.

Click Increment

This item is a preference for working in CNC Setup. The entry box for the Click Increment has a small up arrow and a small down arrow on its left side. When you click on these arrows, the number in the edit box will be increased or decreased by the click increment. When you change the click increment value, it affects the increment value for all of the edit boxes in the other dialogs. The default value is 0.10 and this value is likely appropriate for most work.

Preferences

4. Tools

The Tools section of CNC Setup allows you to define and name all of the different types and sizes of tools you will use to process your parts. This will likely include upcut, downcut and compression endmills, drills, and maybe a V-cut or engraving tool. CNC Setup is shipped with a small tool library that includes some of the tools you may use. It is likely that you will need to modify this library to fit your construction methods.

Tool Library

You can access the tool library from both the Wizard mode in Step 1 and the Advanced Mode by clicking on the tool icon. This activates a dialog that shows all of the tools in the library. These tools can be renamed, edited, or deleted. New tools can be added to the library as necessary.

Tool Table – The table in the left half of the dialog lists all of the tools that are currently in your tool library. The *Tool Name* can be whatever makes most sense to you. A descriptive name usually works best because it makes the tool more easily recognizable. The *Tool Type* is Endmill, DrillBit, Conic or Engrave2. Each of these tool types has a different set of dimension parameters, and is treated differently within the toolpath creation process. The *Color* of the tool is just a way to help you spot different types of

tools. Each of the tool types is assigned a different color.

Tool Drawing and Dimensions – The drawing of the tool on the right half of the dialog serves two purposes. First, it provides an easily recognizable drawing of the type of tool you have selected. Second, it shows the units and dimensions of the tool. When you are creating a new tool, or editing an existing tool, the dimensions are editable.

New Button – This button creates a new tool, adds it to the library, and lets you edit the name and all of the dimensions. The image below shows the dialog after a new tool has been added. All of the editable parameters are highlighted. The tool type is selected from the dropdown list that is displayed when you click on the down arrow in that cell.

Notice that the *Accept* button is not enabled in this image. This button is enabled after a tool name and reasonable dimensions have been entered.
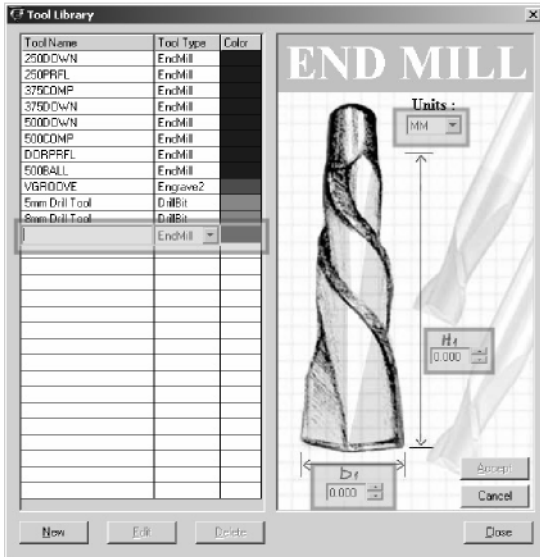
Edit Button – This button allows you to edit an existing tool in the library. Just click on the tool you would like to edit and then click on the Edit button. All of the editable parameters are displayed in the same form as with a new tool. After editing the parameters, click the *Accept* button to make the changes permanent, or click *Cancel* to abandon the changes.

Delete Button – Select the tool you would like to delete and then click the *Delete* button. You will be asked to confirm that you want to delete the tool before it is removed from the tool library.

Close Button – This closes the tool library dialog and returns you to your previous place in CNC Setup.

## Where is this information stored?

The tool library is stored in a file called 'ToolLibrary.ini' that is located in the main EnRoute program folder. It is a text file, so if you open it in a word processing application you can see how the information is stored. You probably won't ever have to work directly with this file, but it is always nice to know where information is stored. It is necessary for both CNC Setup and One Button Machining that this file is there, so please don't delete it.

5. Toolpath Strategies

Toolpath strategies are the different methods you use to rout, drill, and pocket your parts. In order to create a dado joint to fit a cabinet side to the back, you have to create the tab on one part and the dado on the other part, and probably drill some pilot holes for the screws to hold it together. The geometry for this joint has been defined in KCDw, now you need to define how the toolpaths will be created to properly cut it out.

Strategy Editor

The following graphic shows the Strategy Editor dialog with the primary areas of the dialog called out. This editor is used to create, edit and save all of the strategy types that are available in CNC Setup.
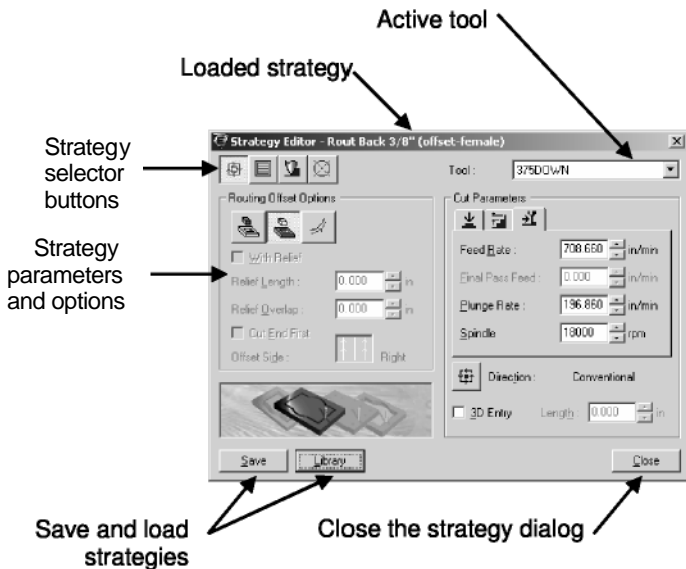
Strategy selector buttons – These buttons let you choose between the routing offset, fill, engrave, and drill strategy types. The appropriate options and parameters are displayed for the selected strategy type.

Strategy parameters and options – The two center regions of the dialog provide all of the available parameters for the selected strategy type. These parameters are discussed in detail later in

this chapter.

Save and Library buttons – These two buttons allow you to save the current strategy into the strategy library. After a named version of a strategy has been saved it is available to be used in the next step of the process, layer mapping. The library button lets you load a strategy from the library. You might use an existing strategy as a starting point for a new strategy, you might need to modify parameters to get the correct results, or you might just need to remind yourself how a strategy is defined.

Loaded strategy – The title bar along the top of the dialog displays the name of the currently loaded strategy. If no strategy has been loaded, this section is blank.

Active Tool – The active tool for a strategy is shown here. The dropdown provides access to all of the tools in your tool library. Just click on the down arrow to see the list of tools.

Note: *The active tool for a strategy must be selected first because many other parameters wil have constraints based on the geometry of the selected tool.*

Save Strategy

After entering all of the parameters for your strategy, it is important to save it so that it can be used in the layer mapping step. Click on the *Save* button, which activates the save dialog show below.
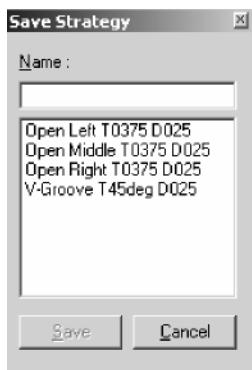

All that is required is to enter a name for your strategy and click on the *Save* button. Notice that the *Save* button is not enabled until a name has been entered in the top edit box. Click the *Cancel* button to choose not to save the strategy.

The strategy names in the center box are all the saved strategies of the same type in your library. If you are saving a routing offset

strategy, then all the saved routing offset strategies are listed.

Load Strategy

Loading a strategy from the strategy library is similar to saving.
Click on the *Library* button to activate the load dialog.
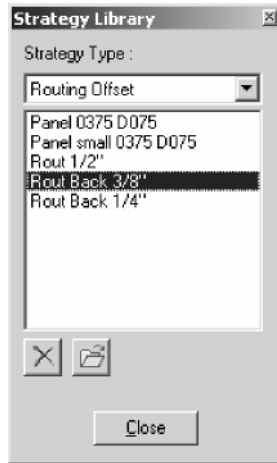
The top dropdown lets you choose what type of strategy you want to
load. Click on the dropdown button to choose from a list of the
available strategy types. If a particular strategy type, such as Drill, is
not in the list it means that no strategies of that type have been
saved in the library.

After choosing the strategy type, the center list shows all available
strategies. Click on the strategy you want to load and then press

the *Load from File* button   .

This dialog also lets you delete saved strategies. This lets you
clean up saved strategies that you have replaced, or that are no
longer needed. Click on the strategy in the list and then click on

the *Delete* button   .

Cut Parameters

The cut parameters define depths, number of passes and feed rates
for the strategy. These parameters are common to all of the
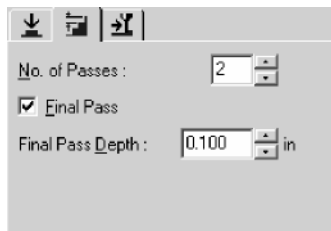different strategy types.

Depth

This parameter defines the finished depth of the toolpaths. You must define a depth other than 0.00 in order for the strategy to be valid. Notice on the depth tab that if the depth is defined as zero you see a red box around the icon. After you define a valid depth the red box goes away. This is intended as a reminder to define a depth for the strategy.

Final Depth : 0.500 in
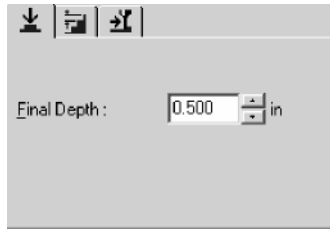
Passes

Because of the limitations of the tool you are using, or because of the density of the material, you may want the cutting to be performed in more than one pass. CNC Setup knows the geometry of the tool you have selected for the strategy, and will force more than one pass if the final depth is larger than the cutting length on the tool. If you have a final depth of 1.00 inch and you are using an end mill with a 0.75 inch cutting length, it is necessary for the strategy to use at least two passes.

No. of Passes : 2
☑ Final Pass
Final Pass Depth : 0.100 in

It may be desirable to define a *Final Pass Depth* that is different from the other pass depths in the strategy. This may be true because you want the final pass to cut less material in order to help maintain the vacuum hold-down of the part. Also, a smaller depth on the final pass can help reduce the deflection of the tool, helping to improve the quality of the cut.
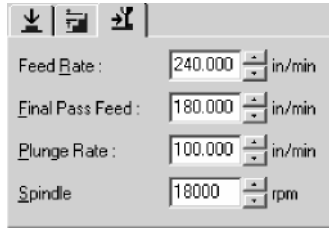
If you check the *Final Pass* checkbox, at least two passes will be required, and you need to enter a value for the *Final Pass Depth* that is greater than zero.

Feed Rates

These feed rates define how fast the tool will move cutting,

plunging into the material, and how fast the spindle will rotate the tool. The units are based on the active units you have defined.

If you have selected a final pass then you should also define a final pass feed rate. You may want this pass to cut a little slower than the other passes as a way to improve cut quality.

The actual feed rates that are appropriate for your work will be based on several factors that include the following,

- The material you are cutting.

- The size of the tool you are using. Larger diameter tools are stronger and can generally handle higher feed rates.

- Tool geometry. Two tools of the same diameter and style can have very different cutting geometry, and therefore, different cutting capabilities

- Your machine's capabilities. Size, weight, rigidity, and motor strength all play a part in your machines performance. The capability of the spindle on your machine also plays a part.

- Your edge finish requirements.

Cut Direction

The Cut Direction defines the direction of the toolpath relative to the direction that the spindle turns. The spindle turns the tool in a clockwise direction. A toolpath moving in the *Conventional* direction moves into the turning tool. In this case the tool is moving into the cut. It is commonly thought of as the normal direction of the cut, and for an external routing offset the toolpath will go in a counter-clockwise direction.

The *Climb* direction is the opposite of conventional. In this direction the toolpath is moving away from the turning tool. For certain material, such as solid wood, this may be the preferred toolpath direction. It can help reduce tearout of the wood. At the same time, it can also cause more chatter of the tool which reduces the cut quality.

The bottom line is that you need to experiment with the tools and material you use. If you see that the waste side of the cut has a better quality than the part side of the cut, reverse the direction of the toolpath.


Direction and 3D Entry parameters
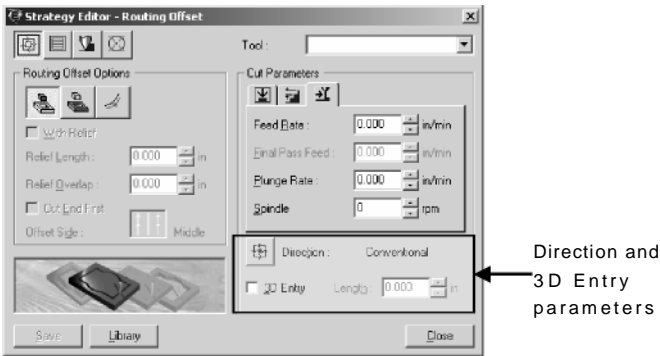
3D Entry

For Routing Offsets strategies you also have the option of defining a *3D Entry.* The toolpath will be created with an additional segment that starts just outside the part and moves into the part over a distance as specified by the *Length* parameter. It will also move down in the Z-axis, starting at the top of the material and moving down to the depth of the toolpath over the Length.

The 3D Entry is a way for the tool to cut its way into the material instead of drilling its way in. It can also reduce any marking of the edge of the part at the entry point.

3D Entry in Plan View
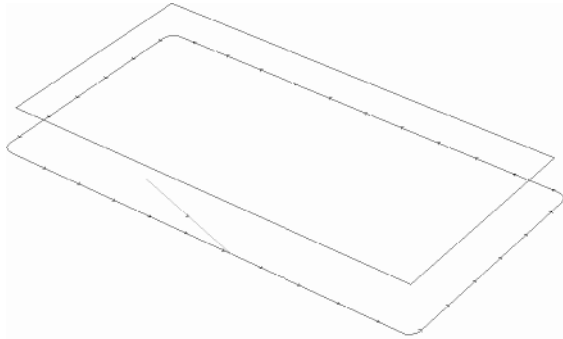
3D Entry in Perspective View

Routing Offset Strategy

Routing offsets are typically
used to accomplish three different types of cuts on a cabinet
part.

- External offset of the perimeter geometry to cut out a part.

- Internal offset of rectangular geometry to create a dado.

- Offset of open geometry to create the tab for a dado joint.

Of course, there are many other uses for routing offsets, and your
specific uses will be directed by your construction methods.
Essentially a routing offset creates a toolpath that is offset from the
original geometry based on the diameter of the tool.

The following graphic shows a depiction of the three types of
routing offsets. The lines that include small arrowheads are the
offset toolpaths. The solid lines that don't have any arrowheads
represent the original geometry used to create the toolpaths.
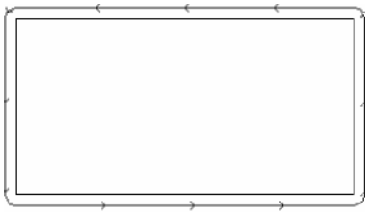
External (Male)
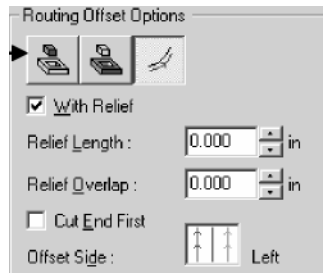routing offset
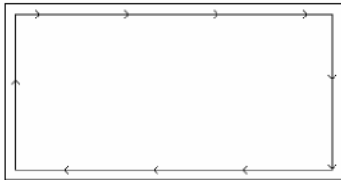
Internal (Female)
routing offset

Open contour
offset

Routing Offset Options

The options available for routing offsets are only relevant for open contour offsets. For both male and female offsets, these options are not enabled.

Choose from the male, female and open offset options

With Relief – The typical offset of an open contour starts at one end and moves to the other end of the contour. Because the spinning tool is trying to push material away from the cut, this can cause blowouts at the downstream end of the cut. The solution to this is to have the offset toolpath move close to the end, and then pick the tool up, move to the end of the contour and then move back into the material from the other direction. That way the tool is moving into the material from both ends. This is particularly an issue when cutting solid wood.

Relief Length – This defines how far away from the far end of the cut the toolpath stays before it picks up and moves to the end.

Relief Overlap – This defines how far the two toolpaths overlap as they approach the start of the relief point. With zero overlap, the toolpaths would start and stop at the same relief point. It is usually good to provide some overlap in order to minimize any marks left on the part from the tool.

Cut End First – This tells One Button Machining whether to cut the downstream toolpath first (checked) or the upstream toolpath first (unchecked).

Offset Side – If the original contour is constructed starting at the bottom of the page and moving up to the top of the page, a left offset would be on the left of the contour, and a right offset would be on the right. A middle offset is placed directly on top of the contour.

In order to select the desired option, click directly on the graphic in the dialog, on top of the line on the side you prefer. The text to the right of the icon tells you which option has been selected.

Fill Strategy

Fill strategies are used to remove material inside a boundary contour. Typically the depth of the fill is less than the full depth of the material. There are two types of fills, Hatch and Island. Both types accomplish the same thing, but one may be preferable based on the geometry being cut, or the material being cut.

The following graphic illustrates Hatch and Island fills.

Hatch fill
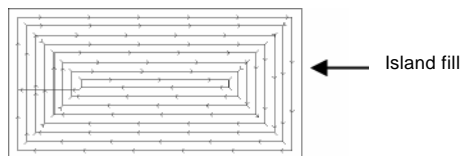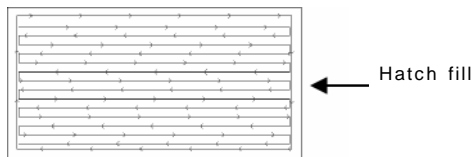
Island fill

A potential advantage of the island fill strategy in this type of application is that the hatch fill strategy is directional. The hatch angle is referenced from horizontal. Using a zero-degree hatch angle on the fill pattern for geometry that has a long x-dimension relative to the y-dimension works well. If the geometry is more vertical, and you apply the same strategy, it can create a situation where the toolpath movement is not very efficient because it introduces many more changes in direction. An island fill applied to these same two pieces of geometry creates an efficient toolpath for both. This concept is illustrated in the following graphic.
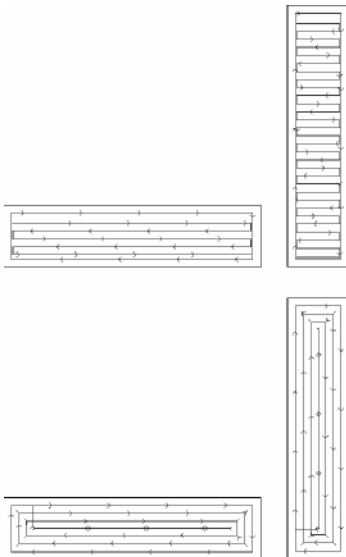
Here you can see that the hatch fill on the vertical part is much less efficient than the corresponding island fill on the same geometry below it. The directional nature of the hatch fill makes it a little harder to predict the results as it is applied to a range of geometric shapes.

Fill Options

The fill options that need to be defined for the fill strategy are pretty simple. First, choose between the Island and Hatch fill options. Then enter the appropriate overlap and angle parameters.
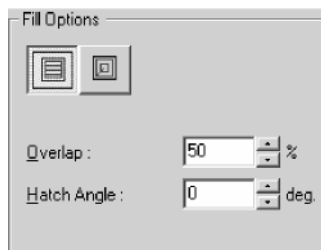
Overlap – This defines the spacing of adjacent toolpaths in the fill. In order to get full coverage of the area being filled, and to achieve a good quality on the fill surface, adjacent toolpaths in the fill should ensure that the diameter of the tool overlaps by some percentage of the tool diameter. It is typical to enter an overlap in the range of 40% to 60%. This also helps reduce the loading on the tool.

Hatch Angle – This parameter only applies to the Hatch Fill. The angle is referenced from horizontal. A hatch angle of either 0 degrees or 90 degrees is typical in this type of application where the geometry being cut is primarily rectangular.

Engrave Strategy

The Engrave Strategy is quite simple. Toolpaths are created using the depth and speed parameters that are directly on top of the geometry. This strategy is typically used in this application to create decorative cuts on doors or other portions of the design that will be used in a decorative way.

No additional options are required with this strategy.

Drill Strategy

Drills are primarily used to create the pilot holes and shelf holes on the parts. Drills are typically applied to the centers of circles in the geometry file. The drill strategy usually uses drill tools, but this is not required. End mills can also be used with the drill strategy as long as the tool is capable of drilling into the material you are using.
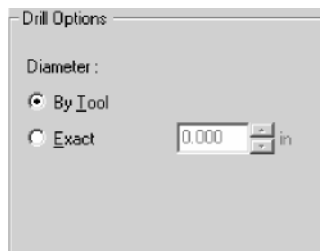
Diameter Option – This allows you to define whether the diameter of the tool defines the diameter of the hole, or whether the diameter is as defined by the dimension you provide. You can define a diameter that is larger than the tool's diameter, but in this application the tool's diameter will be used almost exclusively.

Where is this information stored?

The Strategy Library is stored in a file called 'StrategyTemplates.ini' that is located in the main EnRoute program folder. This file is a text file, so if you open it in a word processing application you can see how the information is stored. You probably won't ever have to work directly with this file, but it is always nice to know where information is stored. It is necessary for both CNC Setup and One Button Machining that this file is there, so please don't delete it. If you are backing up key information from your computer, this would be a good file to save.
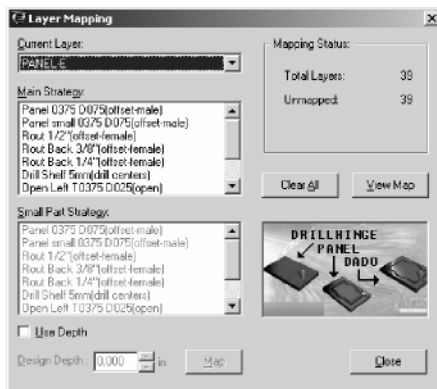
6. Layer Mapping

Here's where you define the connection between the strategies you have saved to your strategy library and the part geometry that is generated by KCDw. The geometry information is saved in a file format called 'DXF'. This refers to the structure of this file that conforms to a standard **D**rawing e**X**change **F**ormat defined by the Autodesk Corporation.

If you have experience using CAD software, you know that one way to separate different categories of information is to place them on different drawing layers. These digital layers are similar to drawing overlays that are registered correctly with each other.

This is how the different types of geometry are separated in the DXF drawings created by KCDw. Each drawing file represents one cabinet part, such as a right side, left side, back, top or bottom. Within a part drawing, the different types of geometry that correspond to the machining operations that will be performed are each on a different layer. As the part files are processed, a different toolpath strategy is applied to each drawing layer, and then the parts are assembled based on the material specified, nested to fit the specified material, and finally output to files to be sent to the machine.

The key step here is to define to correspondence between the drawing layers and the strategies. This is called **Layer Mapping.** KCDw has a standardized set of layer names that are defined with KCDw. These layer names are used by CNC Setup so that you know what layers need to be mapped.

The layer mapping dialog is accessed by clicking on the Layers icon from the Advanced dialog or as Step 3 in the Wizard.

The following sections describe each part of the Layer Mapping dialog and how it fits into the layer mapping process. The primary goal is to work through the list of layers, defining what strategy is to be assigned to each layer until all of the layers have been mapped.

Mapping Status – This panel shows the number of *Total Layers* that need to be mapped. In this example there are 39 layers to map. It also shows how many layers remain to be mapped. In this example none of the layers have been mapped so 39 remain *Unmapped.* This section lets you know how much mapping work remains to be done.

Current Layer – This dropdown lists the current layer to be mapped, and it also gives you access to the remaining list of layers that have not been mapped. As layers are mapped, they are removed from the dropdown list. Once this list is empty – you're done mapping.

Main Strategy – This is a list of all of the saved strategy templates in your Strategy Library. This is the primary strategy to assign to the current layer. The scroll bar on the right of the list lets you scroll through the list to find the correct strategy for the current layer.

Small Part Strategy – After you have selected the Main Strategy, you can *optionaly* assign a different strategy that will be applied to small parts. Small parts are parts that have an area smaller than a threshold that you can set. Defining a small part strategy is typically only necessary for routing offsets that will possibly be cutting out small parts. For these small parts you may want to define a strategy that uses more passes, or defines a very thin final pass to help ensure that the part is held in place by your vacuum hold-down system.

Use Depth and Design Depth – When geometry is created in the DXF files, it is typically placed at a Z-axis depth that is appropriate for the toolpath depth, using the design information within KCDw. If the Use Depth checkbox is checked, the actual depth of the geometry is used as the Final Depth in the applied strategy. This means that the strategy depth you defined in the template will be overridden by the geometry depth in the DXF file.

The advantage to this is that it allows you to use a strategy definition for more than just one type of geometry. If you are using a routing offset strategy to cut out parts that are both 0.50" thick and 0.75" thick, this option allows you to assign the same strategy to both layers. You know that One Button Machining will modify the strategy

depth based on the geometry depth, which works great. If you don't select the Use Depth option, the strategy depth that you defined will be used for that mapped layer.

If the Use Depth option is enabled, you may also *optionaly* define a Design Depth value. This concept is a little abstract, but as an illustration, there may be times when you know you are cutting a material that is nominally 0.75" thick. For some reason (maybe you are cutting a particular type of laminate) you decide to define a strategy with a final depth of 0.77" because you want to make sure that your parts all cut out. Your strategy is 0.02" deeper than the material you are designing for. Because you have the Use Depth option checked you know that if this strategy is used to cut geometry that is 0.50" deep, the depth of the strategy will be automatically adjusted. But, you still want to make sure that the actual cutting depth is 0.02 deeper than the geometry. By telling One Button Machining that your 0.77" deep strategy is actually meant to be used on 0.75" geometry (the Design Depth) you will ensure that this 0.02" difference will be maintained even when the depth is adjusted.

This Design Depth concept is definitely a finer point that is not typically necessary, but that may come in handy in specific situations.

Map Button – After selecting the Main Strategy, Small Part Strategy (optional), and Design Depth preference, click on the *Map Button* to map the current layer. The mapped layer is removed from the Current Layer list and the Unmapped value in the Mapping Status panel is decreased by one. Now just repeat the strategy selection and mapping process until they are all mapped.

Clear All – This button will un-map any mapped layers. Don't click on it unless you are ready to start the mapping process from the beginning.

View Map – This button allows you to view all of the mapped and unmapped layers in a tabular form. This is handy if you want to get an overall view of the mapping status.

The following graphic shows a layer map. Notice in this map that only one layer, Cutout Tool, is not mapped. If you want to make a change to a layer that has been mapped, just double-click on that layer in the table and you will be taken back to the mapping dialog with the clicked layer as the Current Layer. This makes editing mapped layers quite simple.

Layer Name	Main Strategy	Small Part Strategy	Use Depth	Design Dep
DRILLHINGE-E	Drill Shell 5mm	Drill Shell 5mm		
DRILLDRWFRONTPILOT-E	Drill Shell 5mm	Drill Shell 5mm		
SAW-E	Open Left T 0375 D025	Open Left T 0375 D025		
DRILLSHELF-E	Drill Shell 5mm	Drill Shell 5mm		
DRILLPILOT-E	Drill Shell 5mm	Drill Shell 5mm		
RABBETDRAWER-E	Open Left T 0375 D025	Open Left T 0375 D025		
ROUTDRAWER-E	Rout Back 1/4"	Rout Back 1/4"		
ROUTKICK-E	Rout Back 1/4"	Rout Back 1/4"		
DRILLDRAWER-E	Drill Shell 5mm	Drill Shell 5mm		
DRILLLEG-E	Drill Shell 5mm	Drill Shell 5mm		
DRILLPLATE-E	Drill Shell 5mm	Drill Shell 5mm		
ROUT	Rout 1/2"	Rout 1/2"		
BEADTOOL	Open Left T 0375 D025	Open Left T 0375 D025		
FRAMECLEANUPTOOL	Open Left T 0375 D025	Open Left T 0375 D025		
BEADCLEANUPTOOL	Open Left T 0375 D025	Open Left T 0375 D025		
RAISEDPANELTOOL	V-Groove T45deg D025	V-Groove T45deg D025		
PANELCLEANUPTOOL	Panel 0375 D075	Panel 0375 D075		
CUTOUTTOOL	Not Assigned	Not Assigned		
EDGETOOL	Open Left T 0375 D025	Open Left T 0375 D025		
FLATPANELTOOL	Panel 0375 D075	Panel 0375 D075		
GLASSPANELTOOL	Open Left T 0375 D025	Open Left T 0375 D025		
VGROOVETOOL	V-Groove T45deg D025	V-Groove T45deg D025		
WIDEGROOVETOOL	V-Groove T45deg D025	V-Groove T45deg D025		
DRILLPILOTTOOL	Drill Shell 5mm	Drill Shell 5mm		
PANEL-F	Panel 0375 D075	Panel 0375 D075		

After you are satisfied with all the selections in your layer map click *Close* to close the Layer Map table and then click on *Close* in the Layer Mapping dialog.

7. Job Parameters

The Job Parameters define how the job will be processed. Specifically, what size material will be used, how the part will be nested, and the ordering criteria for the output.
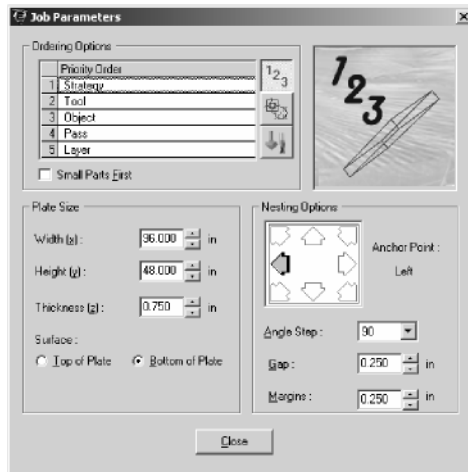
There are three panels in this dialog, each representing a different topic. These include Ordering Options, Plate Size, and Nesting Options. They are described below.

Plate Size

These dimensions define the Width, Height and Thickness of the material you will be cutting. The dialog above shows the parameters for 4-foot by 8-foot material that is 0.75" thick.

Note that the thickness parameter will be adjusted as necessary for the different thicknesses of parts you are processing.

Surface Option – This option gives you the choice between *Top of Plate* and *Bottom of Plate*. This refers to where you typically set the surface at your machine. If the machine is surfaced to the top of your machine's table, or the top of the spoil board, then select Bottom of Plate. If you set the machine surface at the top of the material to be cut, then select Top of Plate.

In the nested-based manufacturing market, most users surface their machine to the top of the spoil board and set this parameter to Bottom of Plate. This helps create consistent dado depths, which helps maintain the dimensional consistency of the finished cabinets.

Nesting Options

These parameters define how the parts will be nested on the material for cutting. Nesting is the process of fitting individual parts together on a sheet in an attempt to fit as many parts on a sheet as possible, and to minimize wasted material. As the parts are processed, nesting is completed automatically using these parameters as a guide.

One thing you will learn as you examine nested output is that invariably you will spot some situations where you could have fit some parts on a sheet more efficiently than the nester, or maybe even reduced the number of required sheets of material by rearranging the parts in some way. The trade-off is that the nesting is completed very quickly, and is generally quite efficient. This saves quite a bit of time on every job that would otherwise be required to manually place parts on every sheet.

Anchor Point – This tells the nester which edge, or which corner of the material, to prioritize when nesting. Typically you will select either the left edge or the bottom-left corner of the sheet. Just click on the arrow that represents the anchor point you prefer.

Angle Step – As the nester works to nest each part, it examines different positions and different rotations of the part. This angle can be anything from 0 to 180 degrees. Zero degrees tells the nester not to rotate any of the parts. 180 degrees maintains the directional nature of the part, but allows it to be flipped. Any angle in-between these two will allow the part to be rotated away from its default orientation. For this type of part processing, where the parts are mostly rectangles, if you want to allow part rotation it most often works best to use an angle step of 90 degrees.
Smaller steps don't typically increase the efficiency of the nest, but they do increase the time required to complete the nest.

Note: Part information generated by KCDw may disallow rotation of parts based on the material specified in the design. One Button Machining respects this setting for the parts, and only rotates parts that allow it.

Gap – This is the dimension that is maintained in between individual parts, taking into account the routing offset toolpaths that are used to cut the part out.

Margins – This is the dimension that is maintained around the perimeter of the sheet.

Note: Many users utilize Gap and Margin dimensions of 0.00. When parts are nested, One Button Machining automatically

ensures that

toolpaths do not overlap.

Ordering Options

Toolpath ordering defines how the toolpaths are created in the output files. It is important that certain operations are completed before others, and this is where that is defined. In the Ordering Options panel, there are three different lists that can be displayed based on the selected button on the right side of the panel. These include Priority Order, Strategy Order, and Tool Order. These are described below.

Priority Order

The Priority determines the order in which the toolpaths are ordered, using each of the five Priority options. These items may be placed in the desired order by clicking and dragging on the item to move it.
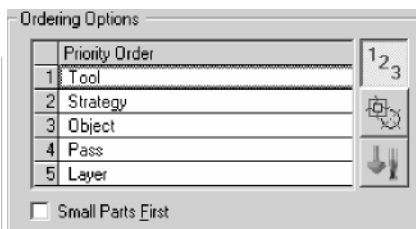
When the output is created, the toolpaths are ordered working from the bottom item in the list up to the top Priority item. For a given output file, the toolpaths are ordered five times; so, for the list in the dialog above they would be ordered first based on the order of the Layers, and then based on the Pass depths, then based on the sorted Objects, then based on the Strategy Order, and finally based on the Tool Order.

The result of this process is that the toolpaths will always be ordered to match the order of the top Priority item. They will be ordered to match the order of the second Priority item to the extent that it is consistent with the Tool Order, and so on down the Priority list. This means that the top items have the most impact on the order of the toolpaths.

The following simple example illustrates the concept of the **Priority Ordering** of toolpaths using numbers to represent toolpaths.

The following numbers are unordered. The numbers aren't repeated, and they are either big numbers or small.

5, 8 , 3, 9, 2 , 7, 1 , 4 , 6

The numbers are first ordered based on their numeric sequence.

1 , 2 , 3, 4 , 5, 6, 7, 8 , 9

Now, the numbers are ordered based on size, with the small numbers first, providing the following final order.

3, 5, 6, 7, 9, 1 , 2 , 4 , 8

You can see that by ordering by numeric sequence first and then size we get a specific order to the numbers. Within each of the two size groups, the numbers are in numeric sequence even though they are not in numeric sequence overall.

In the second option the numbers are first ordered based on their size.

5, 3, 9, 7, 6, 8 , 2 , 1 , 4 ,

Now, the numbers are ordered based on numeric sequence, providing the following final order.

1 , 2 , 3, 4 , 5, 6, 7, 8 , 9

You can see that by ordering by size first and then numeric sequence we get an order that is quite different than in the first option even though all we did was switch our ordering priorities.

Tool Order

The Tool Order gives the opportunity to define the order of the tools that are used in the current job. This order is used when the toolpaths are ordered using the Tool priority. CNC Setup automatically lists the tools that are utilized in the strategies included in the layer mapping.
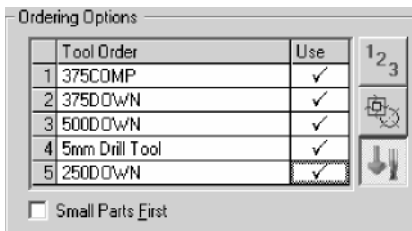
Tools can be included or excluded from the output by checking or un-checking the **Use** checkbox for each tool.

Strategy Order

The Strategy Order is similar to the Tool Order. The Strategy list is populated using the strategies from the layer mapping. This order is used when the toolpaths are ordered using the Strategy priority. A strategy can be excluded from the output by unchecking the **Use** checkbox for that strategy.
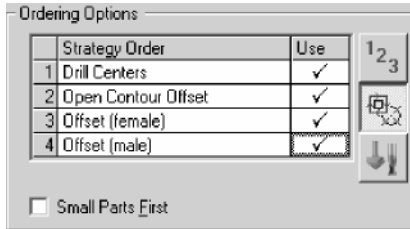

Small Parts First – This checkbox lets you ensure that the parts will be sorted so that any parts that are smaller in area than the small part threshold will be ordered before the larger parts. This option can be particularly useful if you are using a vacuum system to hold parts down. It allows the small parts to be cut out first, when the vacuum is most capable of holding them down.

Machine Setup

8. Machine Setup

This step must be completed in order for One Button Machining to know what machine to target when creating output. Machine output is pretty similar, in general, from machine to machine; however, output must be tailored to match the requirements of each manufacturer, and even the specific model from the manufacturer. CNC Setup and One Button Machining have post-processors to meet the needs of many different manufacturers.

In this step it is necessary to provide some specific setting and machine configuration information. After the machine setup step has been completed, it is only necessary to modify it if something about the machine configuration changes.
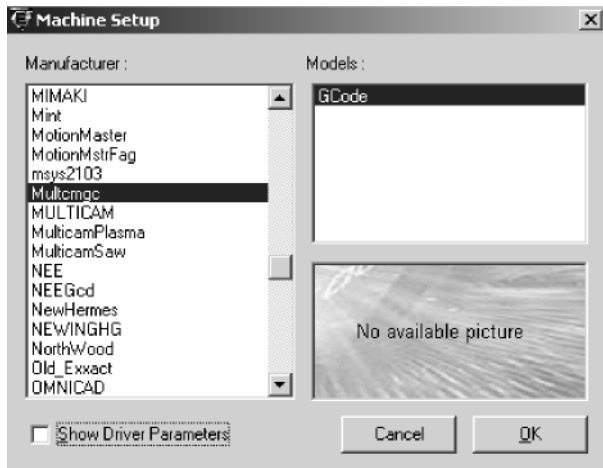
First select the manufacturer of your machine from the Manufacturer list. Then choose the machine model from the list of models for the post-processor you have selected.

Technically this is all that is required to select the post-processor you need to use. However, it is more likely that you will need to provide additional information about your machine configuration by setting additional driver parameters. To do this, click on the *Show Driver Parameters* checkbox. This will expand the dialog to provide access to driver parameters.

The expanded dialog is shown in the following graphic.

The expanded dialog shows six categories of parameters. The panel on the right side shows the parameters for the active category. The bottom three categories contain parameters that should only be changed by someone thoroughly familiar with driver setup[p].

Driver Parameters

Define the Width(X) and Height(Y) of your machine. You also specify the Z lift you would like to use. Z lift is the distance that the tool lifts above the top of the material as it moves from the end of one toolpath to the start of another.

Tool Changer

If your machine includes an automatic tool changer, this is where you define which tools are located in the tool changer positions. As the toolpaths are output, One Button Machining will load the appropriate tools from the tool changer as necessary.

If the *Tool Changer* checkbox is not checked, click on it to activate the tool changer list.

Click on the *Add Tool* or

the *Delete Tool* buttons as necessary in order to have the appropriate number of tool positions in the list. Each tool
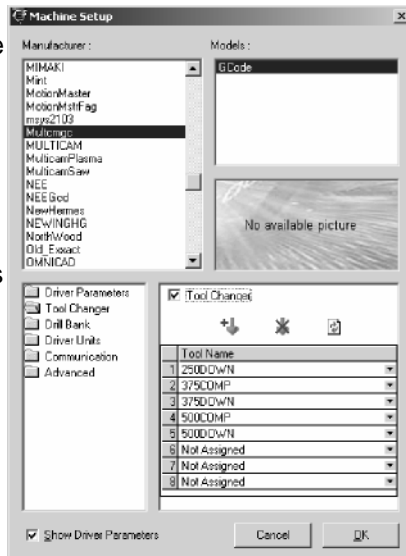
position contains a complete list of the tools in your tool library. Select the appropriate tool for each position that corresponds to your machine tool changer configuration.

You may also click on the Clear List icon in order to clear the tool list to make it easier to start the assignment process fresh.

Drill Bank

Configuring your drill bank (also referred to as a boring machine) is similar to setting up the tool changer. The primary difference is that you also now need to define the position of each drill tool relative to a drill bank origin. There are many drill bank configurations that are used by manufacturers, so unless you are certain of your configuration, it may be necessary to visit with the appropriate representative from your machine's manufacturer in order to get the specifications for the drill tool positions on your machine.
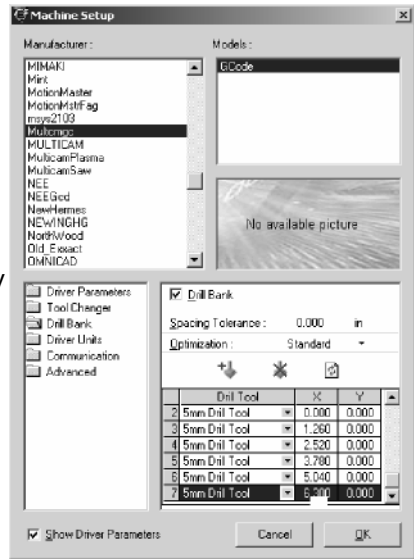
Spacing Tolerance – This value allows a small amount of variation between the precise tool spacing on your machine and the spacing of the drills in the parts you are processing. The standard 32mm (1.2598 in.) spacing of the drills for adjustable shelf holes may not exactly match the configuration on the machine, so this parameter allows you to specify a small tolerance value so that this does not get in the way of successfully processing your parts. A value of 0.01 inch is typically appropriate.

Optimization – This parameter is in place for future use, so it isn't necessary to change it.

Driver Units, Communication and Advanced Parameters

These parameters are typically pre-defined for your machine based on information from the manufacturer. It is rarely necessary to change these parameters, and should not be attempted without instruction from KCDw or SA International technical support.