NC-CAM 7
User's Guide

third edition

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C O N T E N T S

I N T R O D U T I O N

This manual is the User's Guide for the NC-CAM 7 Drill and Router Programming System for personal computers running Microsoft Windows NT, Windows 95, Windows-98 Windows 2000 and Windows XP Professional. NC-CAM 7 provides a mission-specific workstation approach to printed circuit board drill and router program engineering. By concentrating exclusively on the drill and rout processes, this system is able to offer a much more highly evolved set of problem-solving tools than are available on general-purpose CAM systems. It does this while maintaining the ease of use typical of Windows programs.

While NC-CAM is an integrated drill and router programming system, it is also made available with only the drill or only the rout functions enabled. This User's Guide covers all NC-CAM functions for both drill and rout programming, however it is organized to allow you to easily distinguish which chapters apply to the options you have installed.

What's in this manual

This book is divided the following parts:

- **Chapter 1, "NC-CAM: An overview,"** describes the basic functions NC-CAM performs.
- **Chapter 2, "System walkthrough,"** gives a quick tour through the features of NC-CAM 7 CAD, RoutEdit, and DrillEdit.
- **Chapter 3, "Using the CAD and RoutEdit modules,"** gives a step-by-step tutorial of rout programming.
- **Chapter 4, "Using the DrillEdit module,"** gives a step-by-step tutorial of drill programming.
- **Chapter 5, "CAD command reference,"** documents each of the commands in the CAD module.
- **Chapter 6, "RoutEdit command reference,"** documents each of the commands in the RoutEdit module.
- **Chapter 7, "DrillEdit command reference,"** documents each of the commands in the DrillEdit module.
- **Appendix A, "In case of difficulty,"** addresses potential installation and configuration problems.
- **Appendix B, "Frequently-asked questions,"** is a collection of the questions and answers which occur most often with new NC-CAM users.

Typefaces and icons in this book

The different typefaces in this manual are used as follows:

`Monospace` This typeface represents text as it appears on the screen, and anything you must type.

Italics Italics are used for emphasis and to introduce new terms.

KEYCAP This typeface indicates a key on your keyboard. It usually indicates a specific key you should press; for example, "Press **ESC** to cancel a menu."

Menu commands reached by a sequence of selections, for example closing a file, will be shown in italic type, separated by forward slashes. Hence, the main menu command for closing a file will be shown as ***F**ile* | ***C**lose*. Note that the "hot keys" used to select the choices from the keyboard (i.e. **ALT+F** and **ALT+C**) are shown in boldface.

The light bulb icon is used to point out special tips and ideas for using NC-CAM more efficiently.

The mouse icon is used to indicate functions which *require* the use of the mouse.

The lightning icon is used to point out tips for more efficient drilling and routing.

The "attention" arrow is used to draw your attention to especially important points.

Support Equipment

Here's what you need to run NC-CAM:

Computer. NC-CAM is designed to run on PC-compatible computers, running the Microsoft Windows-NT (WinNT), Windows-95 (Win95), Windows-98 (Win98) or Windows 2000 operating system.

Memory. To run NC-CAM, you'll need at least 32 megabytes of RAM. This figure should be considered to be minimum values only:

Disk space. Installing NC-CAM requires that your computer have at least eleven megabytes of free disk space available. The performance of Windows will suffer any time less than 40 megabytes are free *after* installing NC-CAM.

Printer port. NC-CAM is a secured application, and its authorization lock must be installed on an IBM-compatible printer port. (The port remains usable for driving a printer).

A three-buttoned mouse make running CAD much, much faster

Mouse. Your computer must have a two or three-button mouse. A three-button mouse is *highly* recommended for users who will regularly be creating new CAD drawings for routing.

How to contact FASTechnologies

FASTechnologies offers several types of technical support, both directly from FASTechnologies in the USA, and from its dealers overseas. In order to receive technical support, you must register your copy of NC-CAM with FASTechnologies. Be sure to contact FASTechnologies for you temporary authorization codes as soon as you install NC-CAM.

Resources in your package

This package contains several resources to help you:

This manual provides information on every aspect of NC-CAM. Use it as your main information source.

NC-CAM features context-sensitive help. You may press the F1 function key at any time. If NC-CAM can determine the exact function you're trying to use, it will pop up help on that topic. Otherwise, pressing F1 will give you the help index.

Many common questions are answered in Appendix A, “In case of difficulty,” Appendix B, “Frequently Asked Questions,” and the README.TXT file located in the program directory.

FASTechnologies resources

FASTechnologies has offers real-time technical support via telephone, email & fax. New users are automatically entitled to free technical support for a period of ninety days following installation. Technical support may be reached in the USA at 763-763-0611, and the lines are staffed from 08:00 to 16:00 hours US central time (usually the same as Greenwich Mean Time minus 6 hours). If you're calling from outside the US, please leave a FAX number for your reply.

Technical support is also available via FAX, at 949-203-6483, and email, support@fastec.com Please include your product serial number and revision (as shown on your ***F**ile | **A**bout menu)* on all faxed and emailed technical support inquires.

C H A P T E R 1

NC-CAM: An overview

This chapter introduces you to some of the basic procedures involved in using NC-CAM, and what its capabilities are. There's a special section for those who have used earlier versions of NC-CAM. The chapter also describes:

- how to start NC-CAM
- how to enter security codes
- the parts of the NC-CAM 7 desktop
- how layers and modules are related
- how to get onscreen help
- how to exit NC-CAM

After reading this chapter, you should be able to run NC-CAM, navigate through its menus and modules, obtain help on its functions, and exit NC-CAM. You should also have a basic understanding of the functioning of NC-CAM's layers and modules.

NC-CAM's capabilities

NC-CAM was designed specifically to work with CNC drill data, CAD drawings for rout programming, and CNC rout programs, in either metric or inch measurement units. It allows you to import data from a variety of sources, graphically edit in CAD, drilling, and routing modes, and output production-ready CNC programs. Unlike general-purpose artwork CAM systems, the needs of the drill and rout processes are given top priority at every stage of the operation of NC-CAM. The jobs you'll make with NC-CAM are the highest-quality CNC programs available. The system's CNC focus, and the resulting high quality output, begin with the way NC-CAM stores your coordinates.

In NC-CAM, all coordinate data is stored in integer form at a resolution of one nanometer (0.000001 millimeter, or about 0.00000004 inch). If this resolution sounds incredibly small, it is: One nanometer is roughly the width of five hydrogen atoms lined up side-by-side. Why such extreme resolution? Because in printed circuit board manufacturing, there is often the need to switch between metric and inch measurement units. In the floating-point databases often used in other CAM systems, switching units is just one of many ways that data can be corrupted by math round-offs.

Nanometer resolution means no round-offs

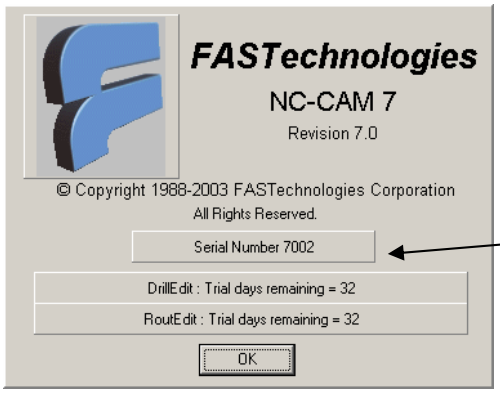
Because NC-CAM always stores your coordinates in nanometers, it is not possible to cause NC-CAM to destructively round off coordinates. When you switch between inch and metric units, your coordinates are not actually changed at all, they're just displayed (and output) in the units of your choice. Thus, you may freely switch between inch and metric with no accuracy penalty of any kind.

When you start up NC-CAM, you'll notice that the work area shown on-screen is square. This is because NC-CAM's nanometer database has a maximum range of 2.4 meters (about +/-42 inches). Limiting the work area this way allows NC-CAM to use very fast database and display math, and it is adequate for almost all known PWB fabrication needs.

The NC-CAM icon

When you're ready to begin working with NC-CAM, find and select the NC-CAM icon. The first time NC-CAM is executed, you may see the "Lock Menu," as shown below. (If you don't see the Lock Menu, but instead see NC-CAM's splash screen, just skip the next few paragraphs).

The first time you run NC-CAM in a new installation, or if you're in an evaluation period which has expired, you'll see this Authorization dialog



Check for the serial number

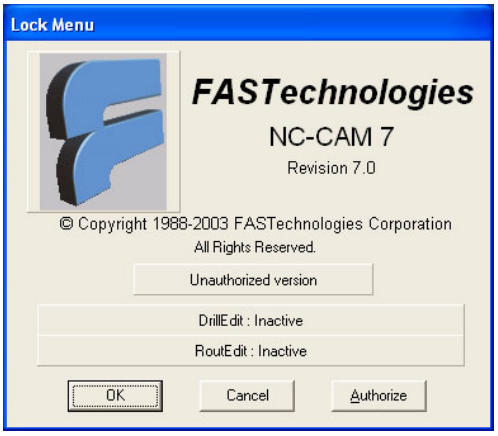
Check your serial number

No serial number means lock isn't working

If you do get the "Lock Menu" when you run NC-CAM, the first thing to look for is the serial number. If you don't see a serial number, you may see a message saying "Unauthorized version." The "Unauthorized version" message means that NC-CAM cannot communicate with its printer port software lock. If this happens on your machine, you should double-check your NC-CAM installation instructions, and perhaps consult Appendix A, *"In case of difficulty."* Of course, you may also contact FASTechnologies Tech Support (see page 5). In any event, you will not be able to proceed until you get NC-CAM to display your lock's serial number.

Your new NCCAM 7 supports the use of the product over the network. You may install the software on as many computers as you wish, but the software can only be run on one machine at a time for each software license that you own. The software lock needs to only be installed on one computer. Any other computers on the same network can then share the license when it is not in use.

If you try to run the software on more machines than you have a license for you will get the software will not start, and you will get the following screen.



If you are sharing a license(s) of NCCAM 7 on several computers, it is a good idea to get in the habit of exiting the software when you are not actually using it so that others will be able to use the software when they need it.

Authorization codes

When you see the authorization menu, you will need to enter the authorization codes which your NC-CAM software needs in order to run. If you received your NC-CAM 7 package directly from FASTechnologies, or if it came from a dealer, just contact FASTechnologies, and we will FAX or Email the codes to you.

First check your date and time

Before you enter any authorization codes into NC-CAM, it's very important that you check your PC's time and date, to make sure they are correct. The NC-CAM lock has a built-in clock, which must agree with the PC's clock within reasonable limits. So make sure your PC is set to the correct date and time (including the a.m. or p.m. setting) before entering your authorization codes.

Check the serial numbers &, then enter your authorization codes

When you receive authorization codes from FASTechnologies, they will refer to a specific NC-CAM serial number. Just in case your company owns more than one FASTechnologies product, you should check to see that the serial number supplied with the authorization codes matches the lock serial number displayed on your authorization menu. If there's a mismatch, contact FASTechnologies with the serial number shown on the menu, so we can correctly register which software you're using with which software lock. We may also need to supply you with new authorization codes for the appropriate lock serial number.

NC-CAM software security codes come in two types; temporary and permanent. Temporary codes consist of both a number of days and a ten-digit code number. The temporary codes are issued when you first install a new copy of NC-CAM. Later, after any evaluation period is completed, FASTechnologies will supply you with permanent authorization codes.

Note that there are three different authorization codes used in NC-CAM, one for each of the three modules; drill, rout, and CAD. To enter the codes for a module, use the **TAB** key or your mouse to select the menu boxes adjacent to the name of the module. If you're entering a temporary code, first enter the number of days in the “days” column. If you're entering a permanent code, just leave the “days” column blank.

Enter the ten-digit authorization code in the “code” column. You may enter codes for all three modules at once. When you're done entering your codes and you've verified them, press the OK button.

Insure your lock!

New users of NC-CAM often ask one or two questions about the software lock. As this is an important topic, we'll address the questions here in the introductory section of the manual.

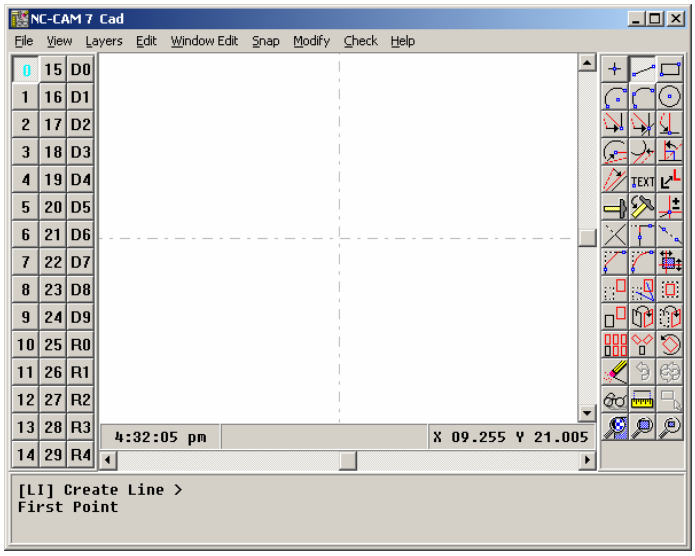
Question #1, “Why is this software locked?” The answer to this question is simple: If FASTechnologies did not lock its software, many companies would run more copies of the program than they've paid for, and some companies would copy it but wouldn't pay for it at all

*Your lock **is** your NC-CAM system!*

Question #2, “What happens if I lose my software lock?” This answer is also simple: **A replacement NC-CAM lock will cost you over ten times its weight in gold!** If you lose your lock, first file an insurance claim, then call FASTechnologies. We'll exchange your lock for free if it should fail, but if it turns up missing, you'll have to buy a new copy of the product. Your lock ***is*** your NC-CAM system. We strongly recommend that you take reasonable precautions against the loss or theft of your software lock. You should also have the software lock explicitly listed ***at full value*** in your business insurance policies.

Add the software lock to your business insurance!

The NC-CAM screen



edit drill, rout or CAD by just clicking the layer buttons

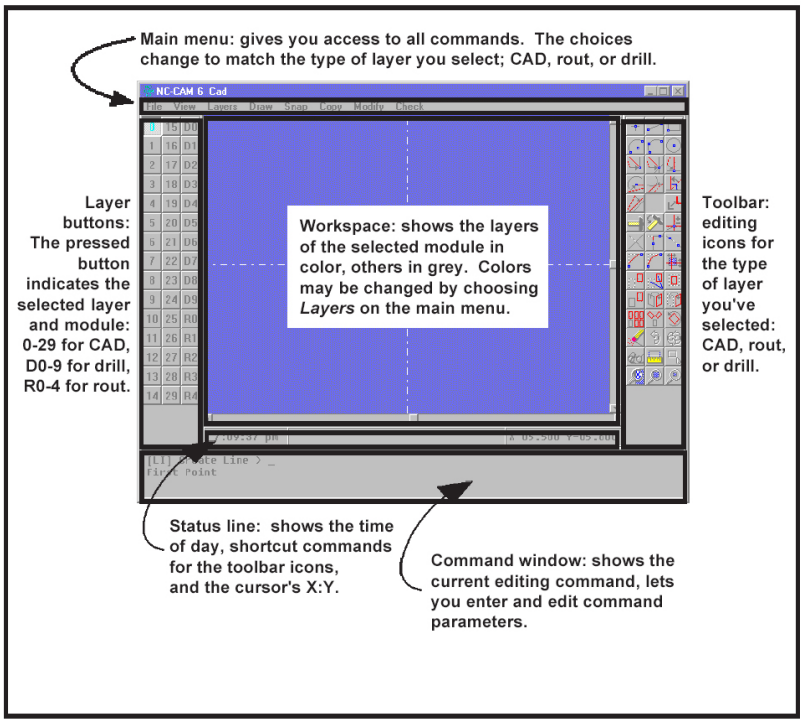
Editing command are accessed by icons, pulldown menus, or two-letter shortcuts

Once your software is authorized, you'll see the NC-CAM screen shown above. The display will look familiar to those who have experience with NC-CAM 5, because the NC-CAM 7 display resembles the CAD display in NC-CAM 5.

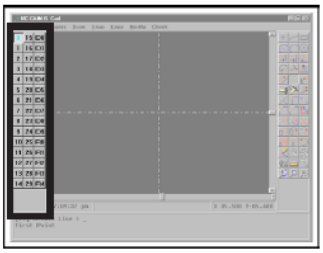
In NC-CAM 7, most of the editing functions for CAD, rout, and drill are available in three different ways: By selecting an icon on the *toolbar* (to the right), by pulling down a choice from the *main menu* (at the top), or by typing a two-letter shortcut into the *command window* (at the bottom). This choice of three ways to enter commands was extremely popular in the NC-CAM 5 CAD editor, which is why the CAD interface model has been extended to both rout and drill editing in NC-CAM 7.

NC-CAM's desktop

The figure below describes NC-CAM's desktop. Each part of the desktop is described in more detail in the following pages.



The layer buttons



These are the most important buttons in NC-CAM. When you left-click a layer button, you select two things. First, the pressed layer button determines the *selected module* (drill, rout, or CAD). The selected module determines the type of data you can edit. Secondly, the layer button determines the *selected layer*. The *selected layer* is the layer you're allowed to edit.

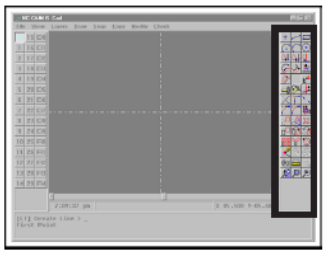
The layer buttons (on the left side of the screen) control both the display in the *workspace* and the editing mode. When you left-click on a layer button, you have made that layer the *selected layer*. When a layer is the *selected layer*, you may create, alter, and delete individual drawing items on that layer.

The first thirty layers, numbered zero through twenty-nine, are CAD drawing layers. When you select one of the CAD layers, the CAD tools will appear on the *toolbar*, and the CAD choices will appear on the *main menu*.

The layers numbered "D0" through "D9" are drill layers. When a drill layer is the selected layer, the drill tools and choices will appear on the *toolbar* and the *main menu*.

The layers numbered "R0" through "R4" are rout layers. When a rout layer is selected, the rout tools will appear in the *toolbar*, and the rout choices will appear in the *main menu*.

The toolbar



The toolbar lets you select individual editing functions by clicking icons. The icons will change when you select CAD, rout, or drill layers on the *layer buttons*. If you want to know the name of an icon's function, place your mouse over the icon and hold it there. The button's description will appear in the *status line.*

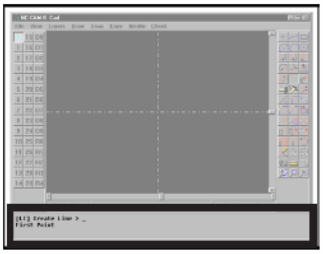
Toolbar functions may also be selected by choosing a menu item from the *main menu*, or by typing the function's two-letter shortcut into the *command window*.

Some of the buttons on the toolbar are modal. This means that when you press a button, it stays down, and the function is repeated until another button is selected. The modal buttons generally create or change drawing items.

Many buttons on the toolbar stay down until you press another

If you're ever in doubt about which function is currently in effect, check the *command window* at the bottom of the screen. The currently-selected command will always be shown there, next to the "**>**" prompt

The command window



The command window shows the current editing command. There's also a command prompt "**>**" where you can type in keyboard commands and coordinates.

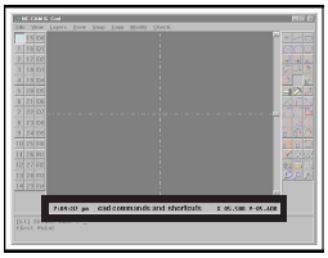
Any time NC-CAM is expecting a coordinate, you may enter an X:Y coordinate pair on your keyboard. For instance, if you want to draw a line from X=1,Y=1 to X=2,Y=2, first you would select the command Create Line. Then you would type `1,1` and press **ENTER**. Then you would type `2,2` and again press **ENTER.** Hand-entering coordinates allows you to duplicate the coordinates exactly as they exist on a blueprint.

Note that when a command is already selected, you may type in a different a new two-letter command. This will cancel the current command and activate the new one.

The parameters for the advanced functions are show in blue. Just click on them to edit them.

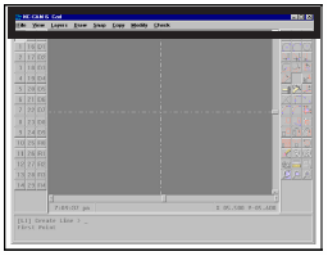
Some functions, such as the [MC] Matrix Copy command have several parameters. In the case of Matrix Copy, there are horizontal and vertical steps, and horizontal and vertical quantities. Parameters like these are also displayed in the command window, with their names in black, and their values in blue. Any number shown in blue in the command window may be clicked with the mouse and then changed with the keyboard.

The status line



The status line contains several pieces of information. First, there is the system clock, which shows the system time. Then there's the hint area, which shows the function and two-letter shortcut for the toolbar buttons when you move the mouse over them. Finally, the coordinate display on the right side of the status line shows the X and Y location of the mouse cursor as you move it through the workspace area.

The main menu

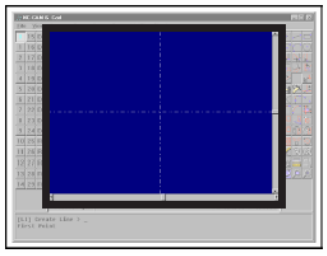


This drop-down style menu gives you access to all the functions of the selected module. The selected module is determined by which layer button is currently depressed. Many of the functions on the main menu are not available from the toolbar or with two-letter command window shortcuts.

In order to get context-sensitive help on a main menu choice, you must first highlight it. (This is most easily done by using the keyboard arrow keys.) When a main menu choice is highlighted, you get help by pressing function key **F1**.

Note that the system-wide file management functions are always accessible on the ***F**ile* pulldown, regardless of which module (DrillEdit, RoutEdit, or CAD) is selected.

The workspace



The workspace area is where all the features contained in NC-CAM's database are displayed. The workspace area is 84 inches square, with 42 inches to travel in the positive direction, and 42 inches in the negative direction. Note that the orientations of the X and Y axes in the workspace are user-settable, with the ***V**iew* | ***V**ersion* function. This is especially useful in cases where the CNC machines are set up in one of the seven nonstandard axis versions.

The scrollbars at the side and bottom of the area determine which part of the workspace you are currently viewing. You may pan the screen by using the scrollbars, or by using the "magnifying glass" icons on the toolbar. You may also simply drag the workspace view with your mouse by first holding the **ALT** key, and then dragging in the workspace while holding down your left mouse button.

*Drag the workspace display by holding the **ALT** key*

DrillEdit, RoutEdit, and CAD layers

NC-CAM 7 consists of three different modules, DrillEdit, RoutEdit, and CAD, which may be purchased separately or in any combination. In NC-CAM, each module uses its own layers. There are thirty CAD layers, nine DrillEdit layers, and five RoutEdit layers.

When you select a layer by clicking on its button, you're also automatically selecting the editing module for that layer. Click a CAD layer, and the CAD module is automatically selected. When a drill layer is selected, the DrillEdit module is selected, and when a rout layer is selected, the RoutEdit module is selected. You can always tell which module is selected by looking at NC-CAM's title bar: It'll say "DrillEdit," "RoutEdit," or "CAD."

The workspace displays the data on all layers, but the data in the unselected modules is shown in grey. The data in the selected module is shown in different colors, one for each layer in that module.

New things go to the layer that's selected on the layer buttons

When you create new things, whether they're CAD lines, drill holes, or rout cuts, they go to the selected layer. When you choose edit functions which change things, they generally change things only on the selected layer. So, where making and changing individual things are concerned, the *selected layer* acts like it's the *only* layer.

If you can see something, you can snap to it

The snap functions allow you to use existing points, lines, arcs, and circles to define positions when you're creating or changing things. You can snap to anything you have displayed in the workspace. For example, you can snap a drill hole to the end of a CAD line, etcetera.

Windows operations can affect all layers in the selected module

By default, window operations like delete, move, copy, and so on, affect all of the layers in the selected module. This way, you can make window changes to all of the layers in CAD, drill, or rout at the same time. (It's not possible to change data of all three types at once, however).

Shutting layers off

For special occasions, it is possible to "shut off" individual layers in the selected module, so that window operations like delete, move, and copy will not affect those layers. If you want to shut off just one layer, do this by right-clicking its layer button, to get the following dialog.

When you right-click a layer button, you'll see the dialog shown above. To shut off the layer, click off the checkbox labeled "**ACT.**," and then press OK. That layer is now inactive, and it won't be affected by window edits.

You may hide a layer by clicking the checkbox labeled "**DIS.**," which makes the layer invisible. Doing this automatically shuts off the checkbox labeled "**ACT.**," so you won't accidentally window-change data you can't see.

When you want to check on all of the layers in NC-CAM at once, you may select ***L**ayers* | ***L**ayer Control* button on the main menu. This brings up a large dialog containing "**ACT.**" and "**DIS.**" buttons for all of the layers at once.

The button labeled "**C**" pops up the layer's color dialog. The foreground color of each layer can be set independently in this dialog. There's only one system-wide setting for the workspace background color, however. The background color setting is found by selecting ***F**ile* | *Options*.

Layer modes

Layers have four different modes; disabled, displayed, active, and selected.

The first mode, *disabled,* means not displayed and not active. When a layer is disabled, it is hidden, and it can't be changed. The items on disabled layers remain exactly the same, unless you select ***F**ile* | ***N**ew*, which erases all layers.

The second mode, *displayed,* means displayed but not active. The layer is shown on the screen, in color if the layer belongs to the selected module, or in grey if it's not in the selected module. When a layer is only displayed but not active, you can snap to the items on the layer, but the layer cannot be changed.

The third mode is *active*. When a layer is active, it's always also displayed. Items on an active layer can be used in snaps, and they can also be changed with any of the window editing commands. Window editing commands include delete, move, scale, copy, and so on.

The fourth mode is *selected*, indicated by the layer button. Only one layer button can be selected, and the selected layer is always both displayed and active. Newly-created entities are created on the selected layer. In CAD, when you do a Create Line, or any other operation that adds an entity to the database, will be placed on the selected layer. On drill or rout layers, anything that creates database entities (such as AutoRout or Drill Hole) will create those entities on the selected layer. In addition to this, the selected layer also determines the selected module, DrillEdit, RoutEdit, or CAD.

Layer colors

You can change the color in which each layer is shown by clicking on the button simply labeled "**C**" in either the individual layer's control dialog, or on the main menu ***L**ayers* | ***L**ayer Control* dialog. To get an individual layer's control dialog, you must right-click on its layer button.

C H A P T E R 2

System walkthrough

Introduction

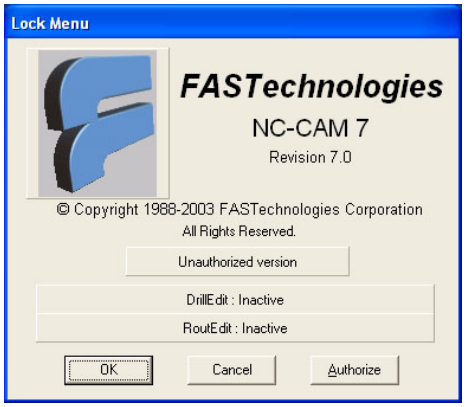
This system walkthrough assumes that the shipping defaults for NC-CAM 7 are still in effect. For example, the input data is formatted assuming that the program is running in Inch mode and not in Metric mode.

Note that this system walkthrough will highlight features of all three modules of NC-CAM. Therefore, if your system is not authorized to run one of the modules, you will want to skip over the sections applying to the module(s) you don't have.

Starting NC-CAM

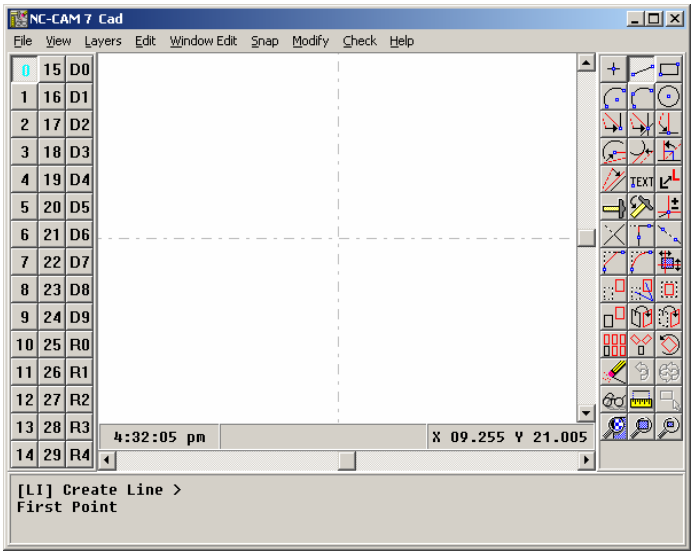
Find and select the NC-CAM 7 icon.

After a few seconds, the program will display a dialog. If your copy of NC-CAM has already been authorized to run, this dialog will be a splash screen, which displays how many trial days are remaining.



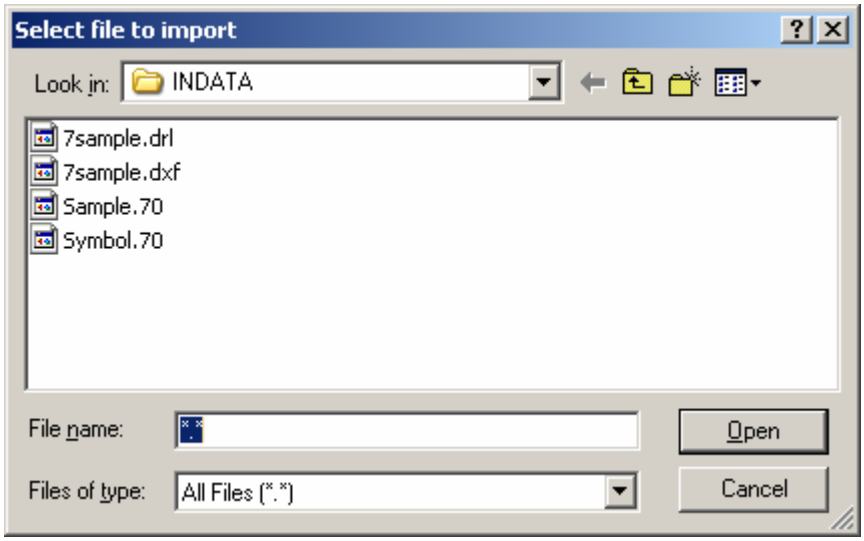
Otherwise, a lock dialog will be displayed, where you will need to enter security codes that you have received from FASTechnologies. For more help on entering security codes, see Chapter 1. When the software is authorized, you can click on OK to continue. The NC-CAM 7 desktop will be displayed, with the words "NC-CAM 7" and a module, "DrillEdit", or "RoutEdit" displayed at the top.

NC-CAM 7 remembers which layer you had selected the last time you ran it, and starts up with the same layer selected.

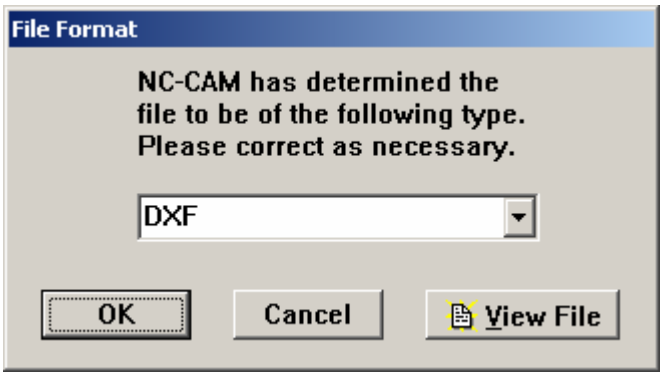


Loading CAD data

This walkthrough takes you through many stages of what would have to be accomplished to take a CAD drawing for a rout program from a customer and produce a production-ready CNC program for routing. The first step in this process is to load the CAD drawing into the CAD module, which can be accomplished by choosing the ***File*** | ***Import*** menu item from the main menu. The familiar "open" dialog will be displayed, with the title "Select file to import."

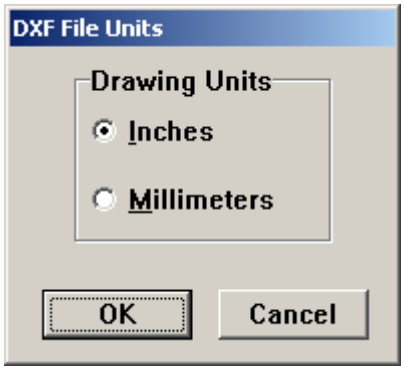


Select the file "7SAMPLE.DXF" for importing by clicking on the name of the file, and then clicking on the Open (or OK) button. A "File Format" dialog appears, to verify the format of the file that you have selected.

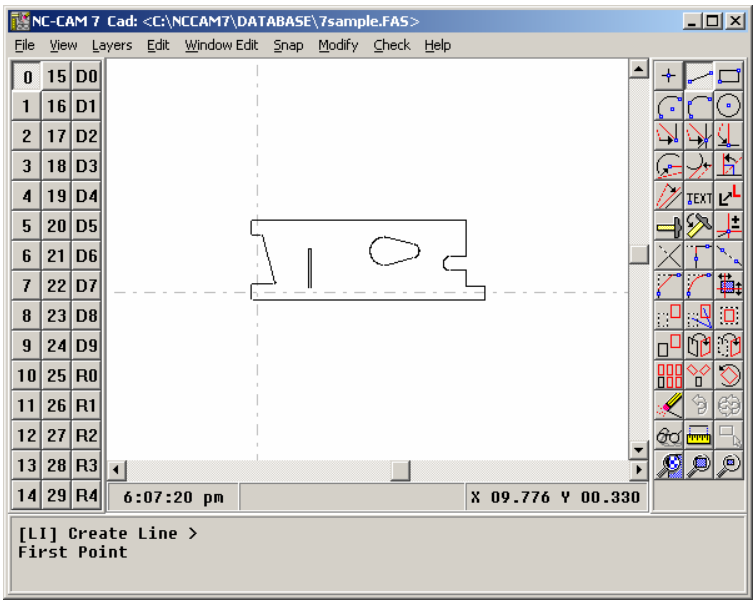


NC-CAM has identified that this is a DXF file, so you can click on OK to proceed with importing the file.

The "DXF File Units" dialog appears next. (This is needed since the unit flags in DXF files are often misleading). The sample file uses inch units, so you can again click on OK.



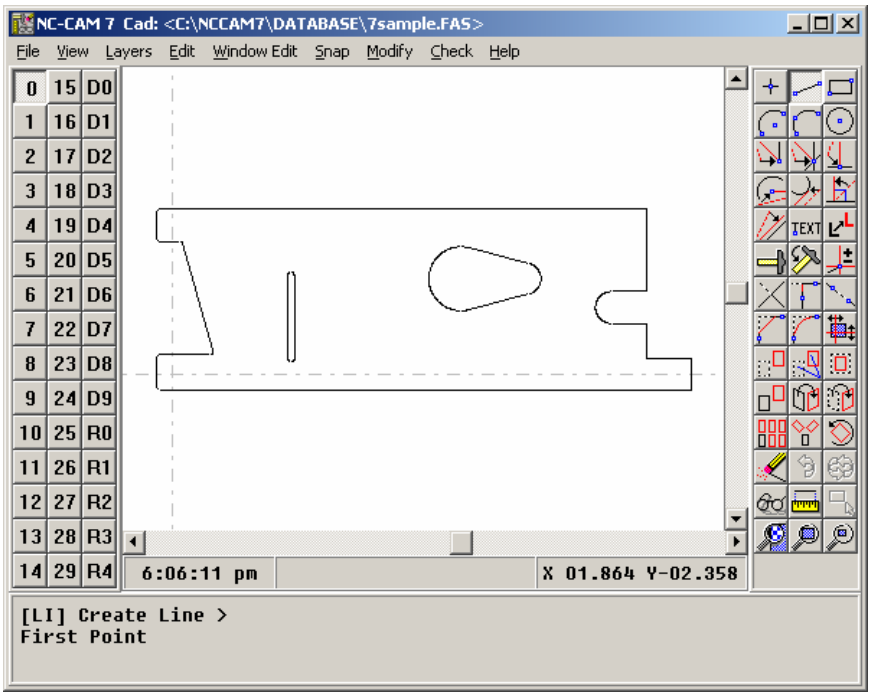
As shown below, a CAD drawing should now appear in the work area.



As you can see, the sample DXF file contains a board outline on only one CAD layer. When you import DXF files having multiple layers, you'll see the data on the different layers displayed in different colors.

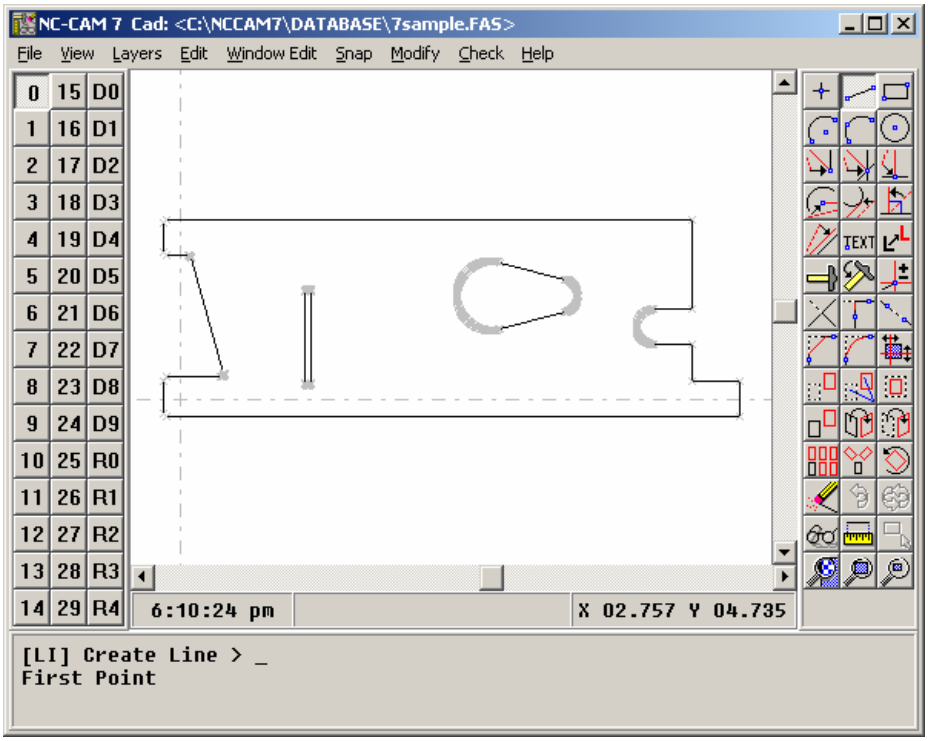
The NC-CAM desktop is resizable, to allow you to take advantage of as much of your computer's screen as possible. If you need to view another application while running NC-CAM, simply resize NC-CAM down to its smallest size by dragging one of the corners of the window towards the opposite corner. The program will shrink down to a fixed minimum size.

In order to examine the data in fine detail, it makes sense to use as much of your computer's screen as possible. To do this, click on the “Maximize” button at the upper right corner of the desktop. Once the desktop is as large as you desire, select *View* | *Zoom All* from the main menu.



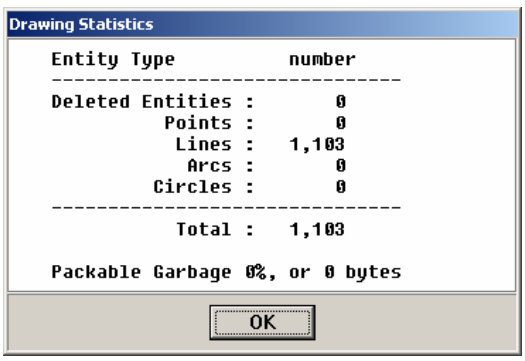
The ***View*** | *Zoom **All*** function is available in every NC-CAM module. Choosing it will cause NC-CAM to display the work area in the highest resolution possible. In other words, it will make the drawing as large as possible, so that you can see the entire drawing in fine detail. Investing in a high-end monitor and video card can make a huge difference in the amount of detail that can be seen in your drawings. A display that can reach a resolution of 1024 by 768 pixels provides much more detail than one that can display only 640 by 480 pixels. It should be noted that NC-CAM 7.0 uses only the Windows standard 16 colors, so setting your computer to video modes with more colors will not enhance the NC-CAM display.

The 7SAMPLE.DXF drawing has a problem very common to customer-supplied rout-path drawings. Select ***View*** | ***C**onst Points* from the main menu to see "X" symbols at the ends of every entity, and at the centers of arcs and circles.



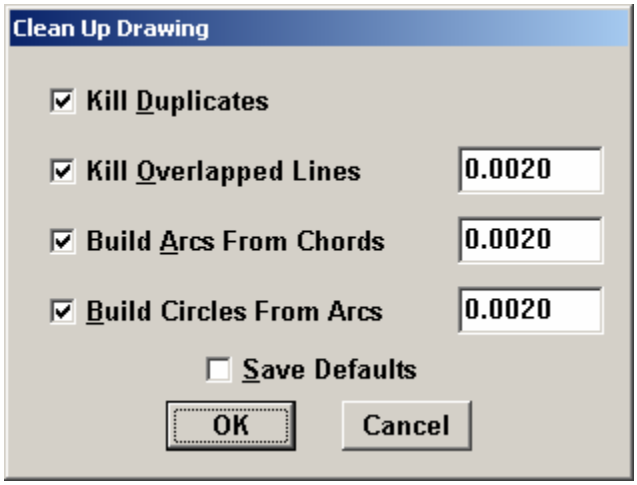
The arcs in this drawing are not true arcs, but are instead drawn with hundreds of small lines. These are known as "chorded arcs," meaning that straight-line chord segments are used in place of true arcs.

To see how bad the drawing data is, select ***File*** | ***G**et Info* from the main menu. A dialog labeled "Drawing Statistics" will appear.



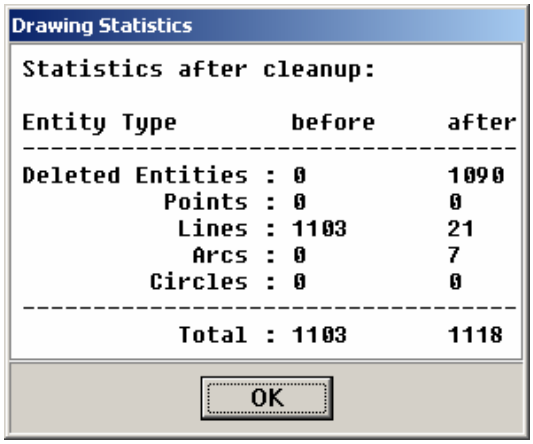
Note that NC-CAM reports that the drawing is made of 1,103 lines, and no arcs. Click on OK to continue.

In order to prepare this drawing for rout processing, you will need to create true arcs in the place of the chorded arcs. In most cases, NC-CAM can accomplish this task with terrific ease. Select ***M**odify* | ***A**utomatic Cleanup* from the main menu, and the "Clean Up Drawing" dialog will appear.

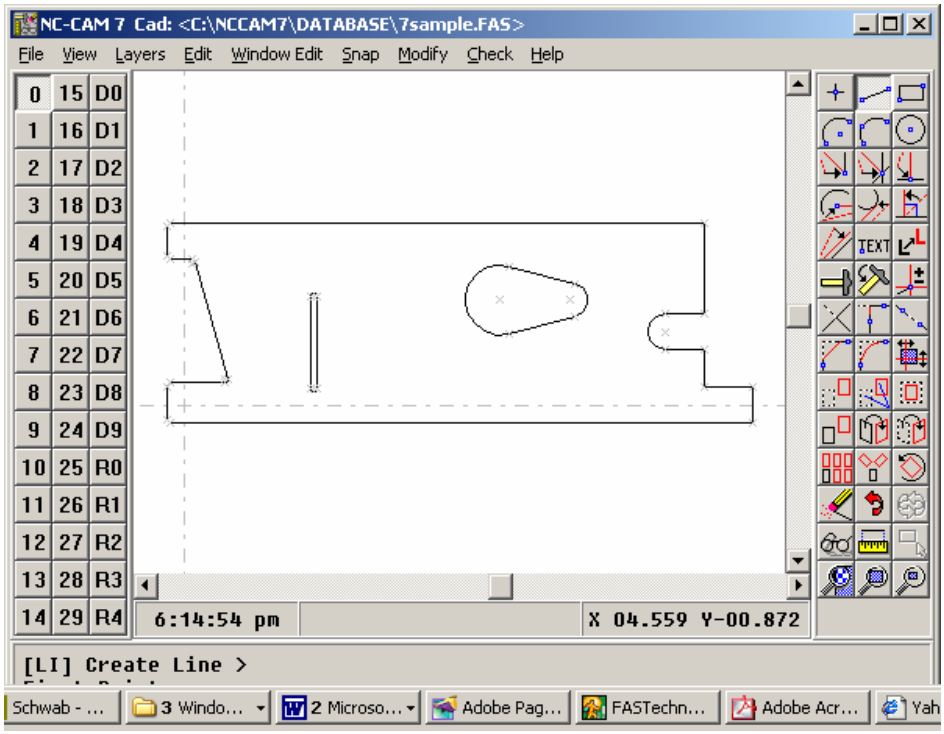


Enable each of the cleanup tasks with the checkbox next to each name. Three of the tasks each require a parameter, which is a measure of how much the task is allowed to change the drawing. The parameters should all be set to 0.0020 inches, for now. Click on OK to continue with Automatic Cleanup.

Once the Automatic Cleanup tasks have completed, a "Drawing Statistics" dialog will be displayed.



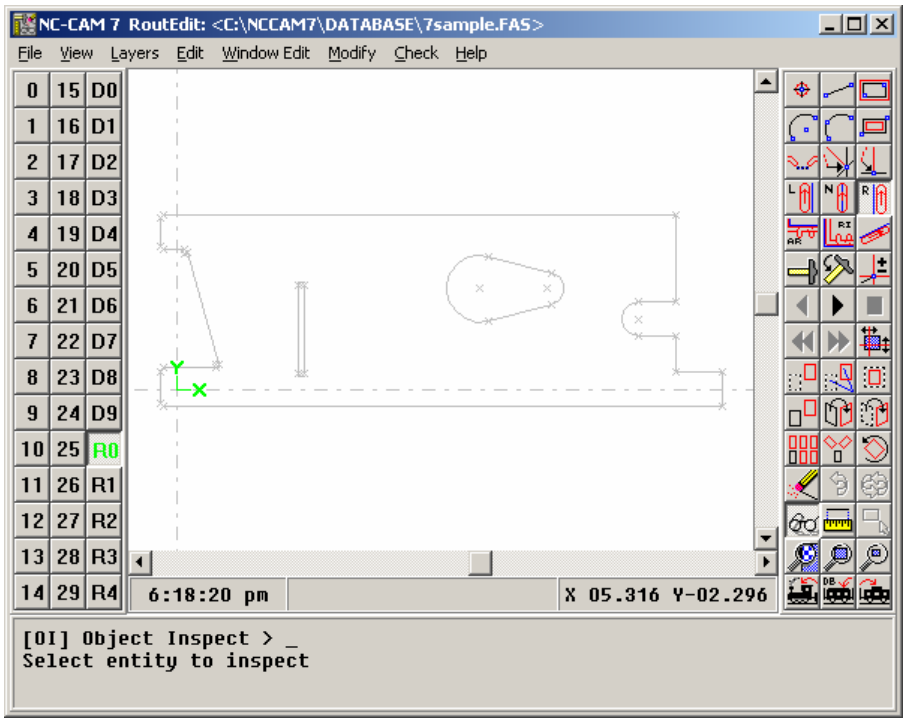
Note that 1090 entities have been deleted! Although the drawing looks the same, there are now only 21 lines and 7 arcs. Working with 28 entities is significantly easier than working with 1,103. Also, the arcs are now true mathematical arcs, and could be edited as such. Click on OK to exit this dialog. Select ***V**iew* | ***C**onst Points* to turn off the construction point display.



NC-CAM saves all changes that you make, so that you can easily "un-do" any mistakes. The advantages of being able to revert your work back to a stage before you made any mistakes are obvious. The disadvantage is that if you modify many entities, such as by doing a Matrix Copy of your entire drawing, and then undo your work, many "deleted" entities have been stored in memory, and will remain there as long as you edit the drawing. Select ***F**ile* | *Pac**k** Database* from the main menu to eliminate all of these deleted entities from the database. Remember, though: After you pack the database, you will not be able to undo any previous changes!

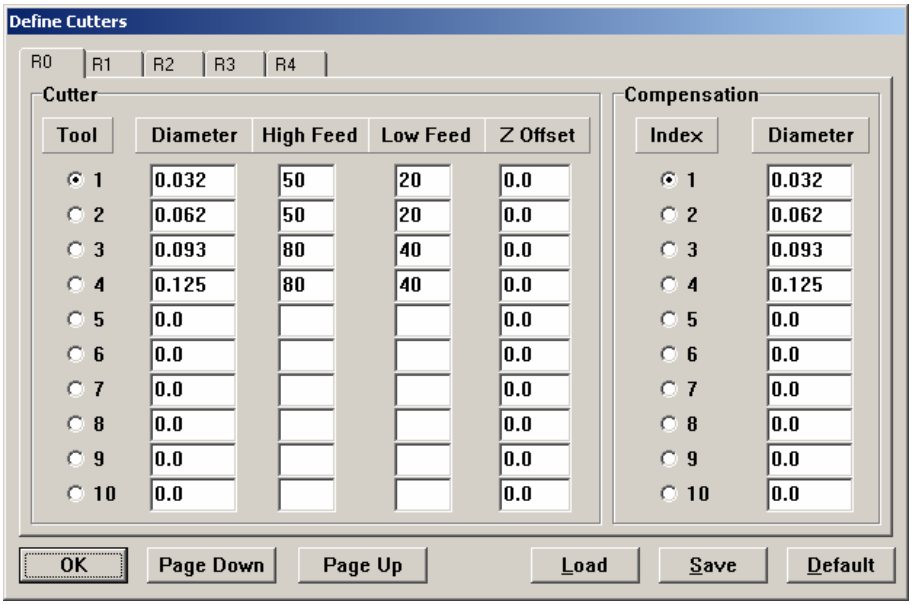
Routing from CAD data

Select the first rout layer, “R0” by clicking on the R0 layer button.



The title bar now says “RoutEdit,” and the buttons in the toolbar have changed to allow rout editing. In this section, we’re going to rout the board outline first, and then we'll create the internal routs. When this is done, we’ll edit the sequence to make the internal cuts first, to show you how to re-sequence rout cuts in NC-CAM.

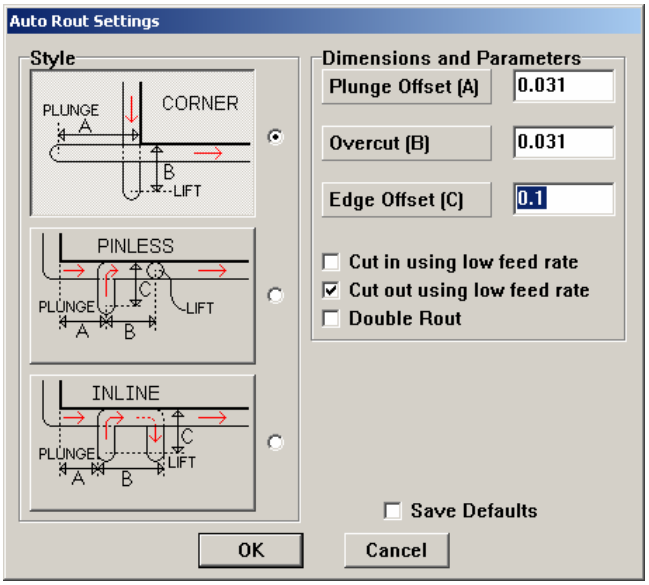
Usually, it makes sense to rout the outside of the board with a large cutter. To select the cutter, select ***M**odify* | *C**u**tter Config* from the main menu.



There is one set of tools for each RoutEdit layer. Since R0 is the layer we're working on, leave the R0 tab (at the top of the dialog) selected.

As you can see, default values are already in effect. Tool 1 should have a diameter of 0.032 inches. Tool 2 should be 0.062, Tool 3 should be 0.093, and Tool 4 should be 0.125 inches. An 0.062 cutter will work fairly well, so Select Tool 2 in the left-hand side of the dialog. Note that the compensation jumps to Tool 2, as well. Click on OK to exit this dialog.

Almost all routing is done with right compensation, so select ***M**odify* | *Compensatio**n*** | ***R**ight* from the main menu. For the outside of the panel, you can use the Auto Rout Outside function to generate a good rout path. Select ***E**dit* | ***A**uto Rout Outside* from the main menu. You will almost always want to change the parameters for Auto Rout, so select ***M**odify* | ***A**uto Rout Config* from the main menu, and a dialog labeled "Auto Rout Settings" will appear.



Click on the top drawing, which has the text "CORNER" in it. It should stay down, to indicate that it is selected. For this rout, only the A and B parameters are used, as can be seen in the selected diagram. Set Plunge Offset (A) to a value of 0.031 inches. Set Overcut (B) to 0.031 inches. Since the C value is not used, you can leave the Edge Offset (C) value as it is. Click on OK to continue with routing.

Most NC-CAM commands that take coordinates as input can take either a mouse click (or mouse drag), or they can take typed values in the command window. In this example, you will select the bottom right edge to begin the Auto Rout. If you were to click on it with the mouse, you would want to click at the top end of that line. To make it easy to communicate exactly where clicks should be made, this text will refer to coordinates by the value that you can type to specify them. Enter the coordinate `8.1,0.26` by typing the keys **EIGHT PERIOD ONE COMMA ZERO PERIOD TWO SIX ENTER**.

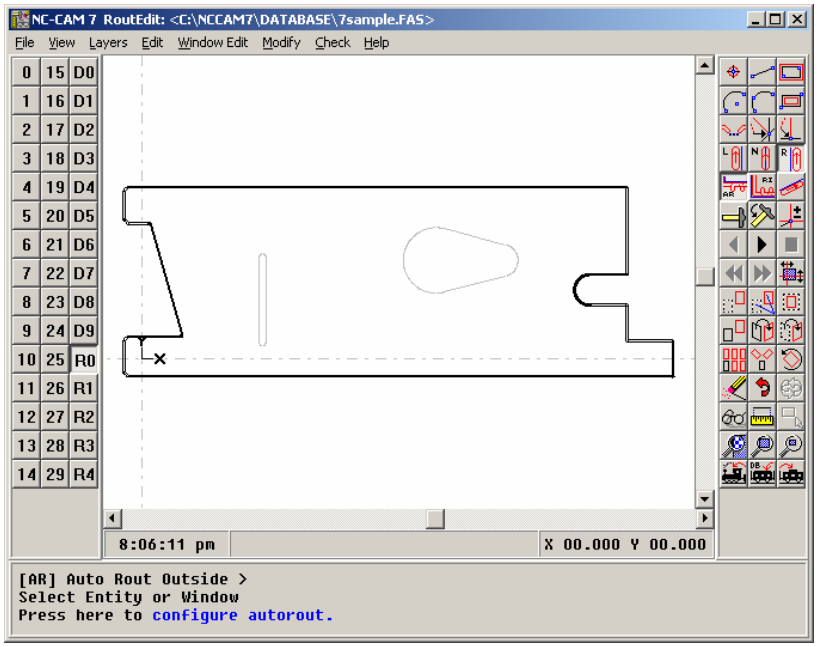
NC-CAM Rout traces the outside edge of the board, creating compensated cuts that exactly follow the outside edge of the board, as can be seen in the drawing above. It Auto Routs the Outside Edge, creating both line cuts and arc cuts, until it reaches the starting point. It uses the options specified in the Auto Rout Config dialog box to generate the appropriate Plunge Offset and Overcut.

Click in the workspace with the mouse or hit **ENTER** to complete the rout program. Auto Rout Outside remains active to allow you to rout additional features. The command will remain active until you select another command.

Routing internal cuts

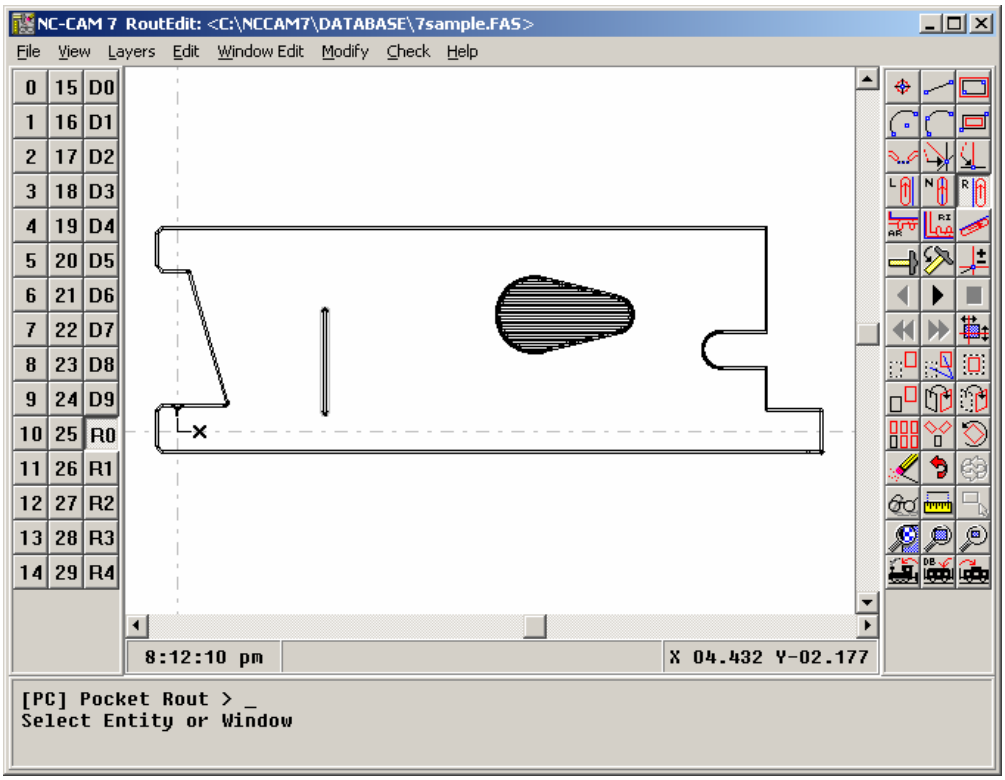
The internal cuts should be routed with a smaller cutter. The pocket could potentially be routed with the current cutter, but some shops prefer to do all of their internal routing with one cutter radius, if possible. In fact, pocketing should be done with the largest cutter that will fit through all of the pathways in the polygon, to save time. For this example, you will make all of the internal cuts with one cutter. Select ***M**odify* | ***C**utter Config* from the main menu. The "Define Cutters" dialog will appear again, with R0 still selected. The 0.032 cutter should be selected, so click next to Tool 1 in the right-hand side. Click on OK to continue.

Select ***E**dit* | ***F**ollow Profile* from the main menu. Enter the coordinates **1.9,0.9** in the command window, and then hit **ENTER**.



At this point, the right-hand line in the slot will be displayed with a dashed line, to indicate that it has been selected. The command window will display instructions telling you to "Select Direction With Both Buttons." In this case, you will use the mouse to select which direction the cutter should follow. With right compensation selected, the cutter should follow the inside of the slot by moving in a clock-wise direction. In order to inform NC-CAM of the direction you wish the cutter to follow, click on the bottom arc of the slot with both buttons (left and right buttons on a three-button mouse.) Check the slot to make sure that it has been routed correctly, with the rout path completely inside the original CAD drawing of the slot.

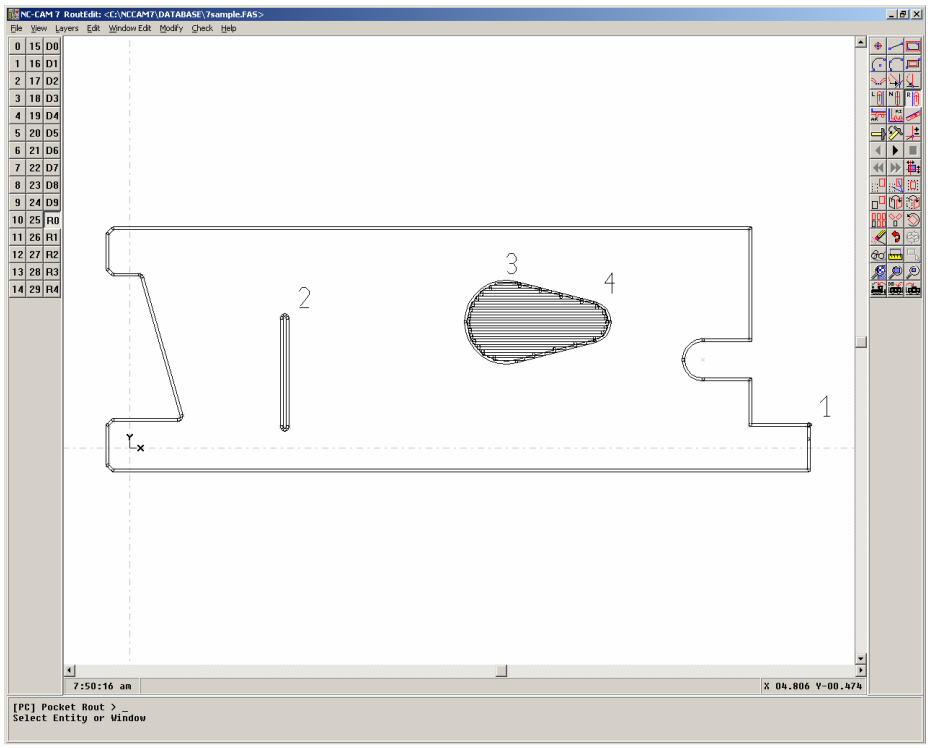
Select ***E**dit* | *Poc**k**et Rout* from the main menu. Use the mouse to select one of the lines of the internal cutout toward the center of the drawing and then the entire pocket will be filled with a rout path.



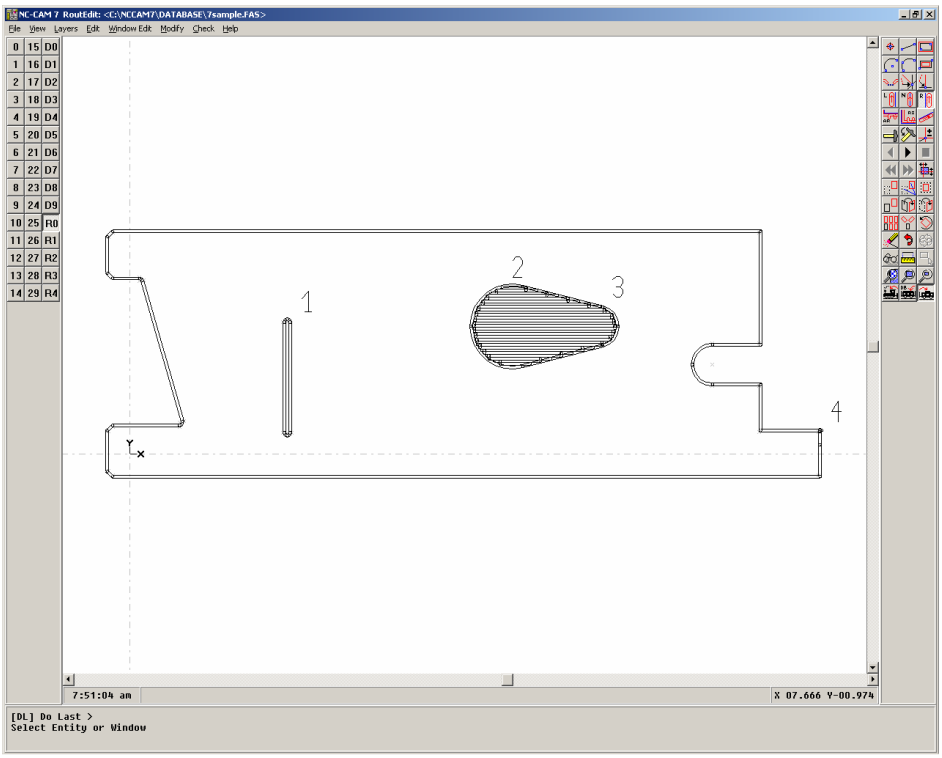
The rout path crosses left-to-right and right-to-left, moving up or down when the cutter encounters another line in the polygon. Then, the path goes once around the outside of the pocket, to ensure a smooth edge.

Changing rout sequence order

In order to rout the board in the proper order, you will have to re-order the cuts so that the internal cuts are made first. To see the order displayed, select ***View*** | *Seq**u**ence Number* from the main menu.



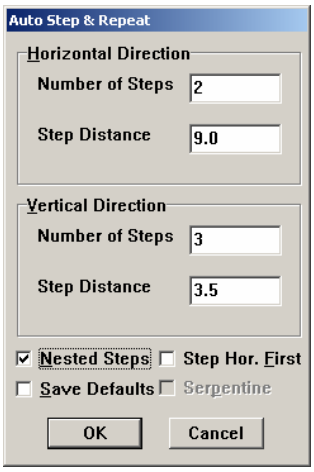
Next to each cut, a number is displayed, indicating the order that the cuts will be made. Select ***M**odify* | *Do **L**ast* from the main menu. Click on the outside cut next to the "1", making it the last cut.



Notice that the sequence number next to the internal cuts are lower numbers than the outside cut. Turn off the sequence number display by selecting ***V**iew* | *Se**q**uence Number* again.

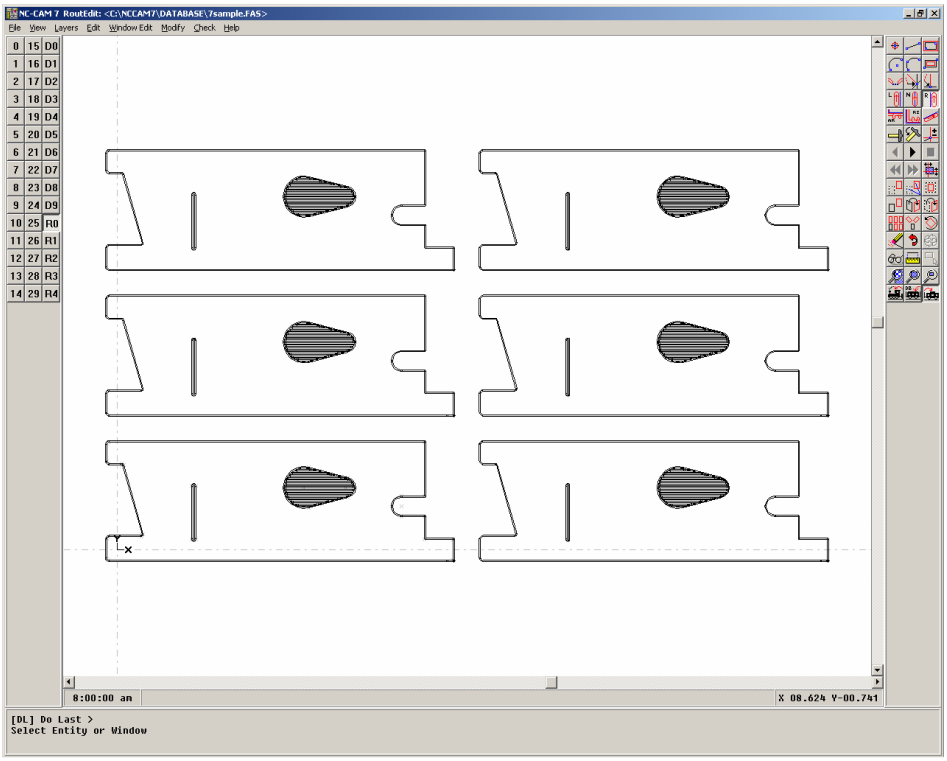
Step & repeats in rout programs

To get the maximum usage out of a panel, a board is often stepped and repeated many times. In order to easily generate a step and repeat, select ***M**odify* | ***A**uto Step & Repeat* from the main menu. In order to Select All of the entities for stepping and repeating, type **SA** at the command window. Hit **ENTER** to confirm the selection of all of the entities. A dialog labeled "Auto Step & Repeat" will appear.



In the "X Direction" section, enter a value of 2 for Number of Steps, and a Step Distance of 9.0. In the "Y Direction" section, enter 3 for Number of Steps, and 3.5 for Step Distance. Make sure that "Nested Steps" is selected, and then click on OK.

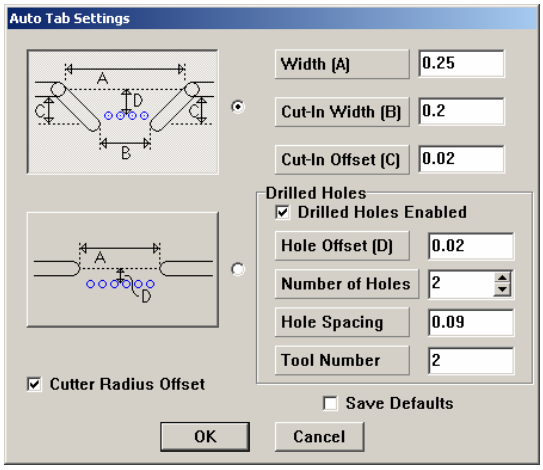
By default, stepped parts are displayed with dashed lines. In order to turn this off, select ***View*** | *Dash **S**tepped Parts* from the main menu. To view the entire panel area, select ***View*** | *Zoom **A**ll* from the main menu.



One of the most powerful features of NC-CAM's RoutEdit is the ability to modify a board after it has been stepped and repeated. In the following section, you will add break-away tabs to the board, and NC-CAM will automatically modify all of the boards in the step and repeat pattern.

Creating tabs

From the main menu, select ***E**dit* | *Auto **T**ab*. Select ***M**odify* | *Auto **T**ab Config* from the main menu to modify the settings for the tabs. A dialog labeled "Auto Tab Settings" will appear. Select the topmost drawing by clicking on it. Select a Width (A) of 0.25 inches. Enter a Cut-In Width (B) of 0.2 inches, and a Cut-In Offset (C) of 0.02 inches. Make sure that both the Cutter Radius Offset and the Drilled Holes Enabled features are enabled. Enter a Hole Offset (D) of 0.01 inches, 2 for the Number Of Holes, a Hole Spacing of 0.09 inches, and 2 for the Tool Number.



Click on OK to continue with these values.

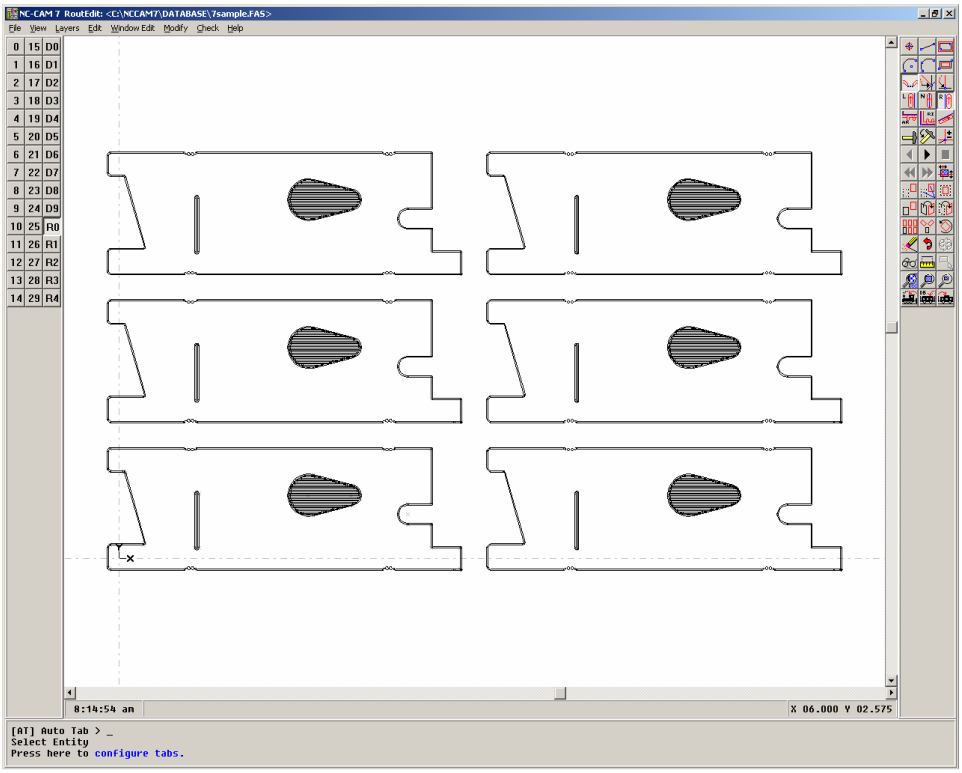
To specify the location of a break-away tab, you must first select the cut that a break-away tab will be inserted into. That is, you will select the cut which will be interrupted by the tab. Then, you specify the location along that cut where the break-away tab will be inserted.

To select the bottom line by clicking on it, or enter **1.7,-0.25** in the command window. To specify the location, enter **1.7,-0.25** again.

Select the right half of the bottom line by typing **6.4,-0.25** and then specify the location by entering **6.4,-0.25** again.

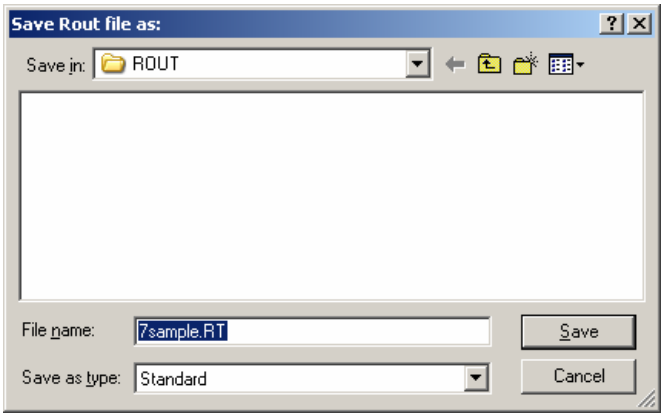
Select the top line with **1.7,2.6** and then the location with **1.7,2.6** again.

Select the right half of the top line with **6.4,2.6** and then specify the location by typing **6.4,2.6** again.

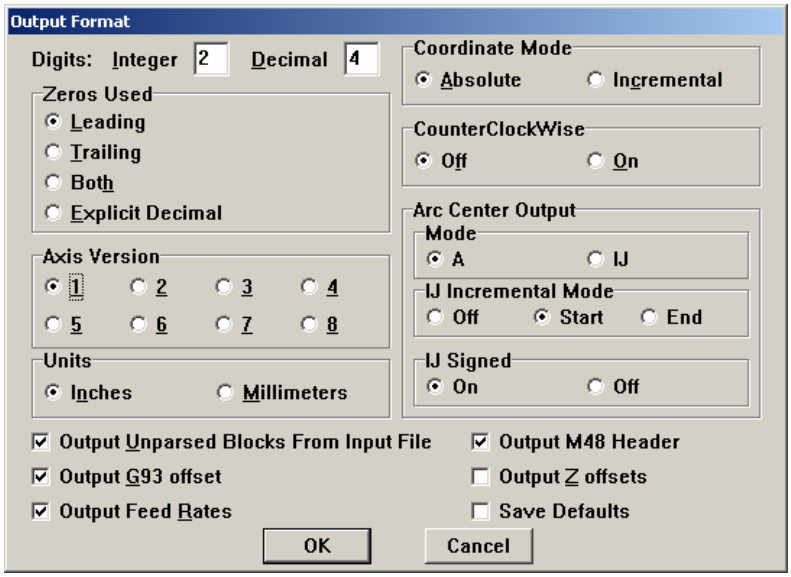


Output rout

In order to save the rout data that you have created, select ***F**ile* | ***M**ake Rout Tape* from the main menu. A dialog labeled “Save Rout file as:” will appear.



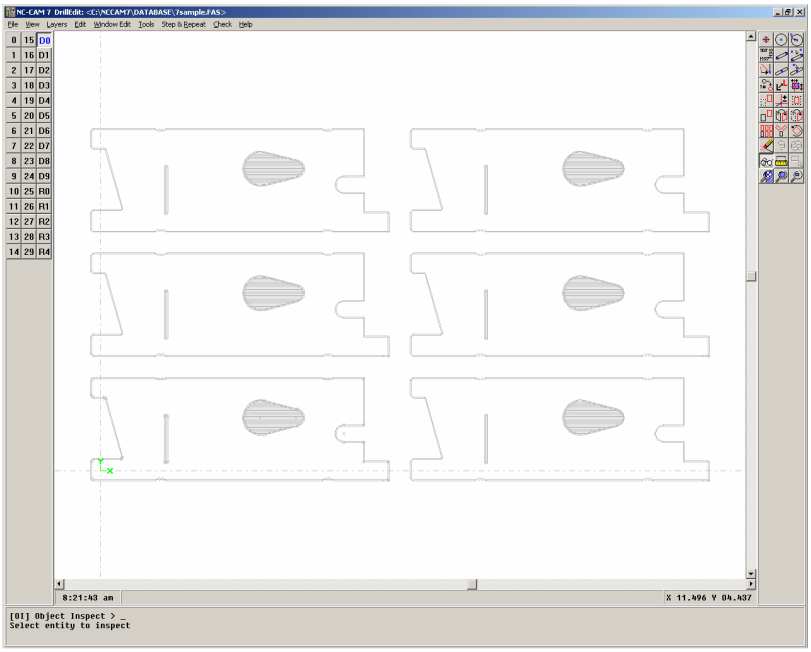
Enter the file name “7SAMPLE.RT” and then click on OK. An “Output Format” dialog will appear.



All of the default values for this dialog will be fine, so click on OK to finish saving the file.

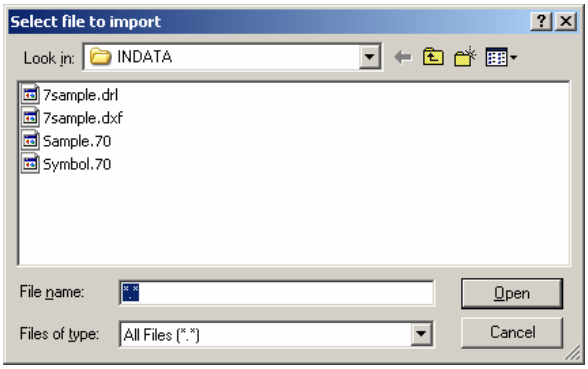
Loading drill data

Select the first drill layer, “D0” by clicking on the D0 Layer Button.



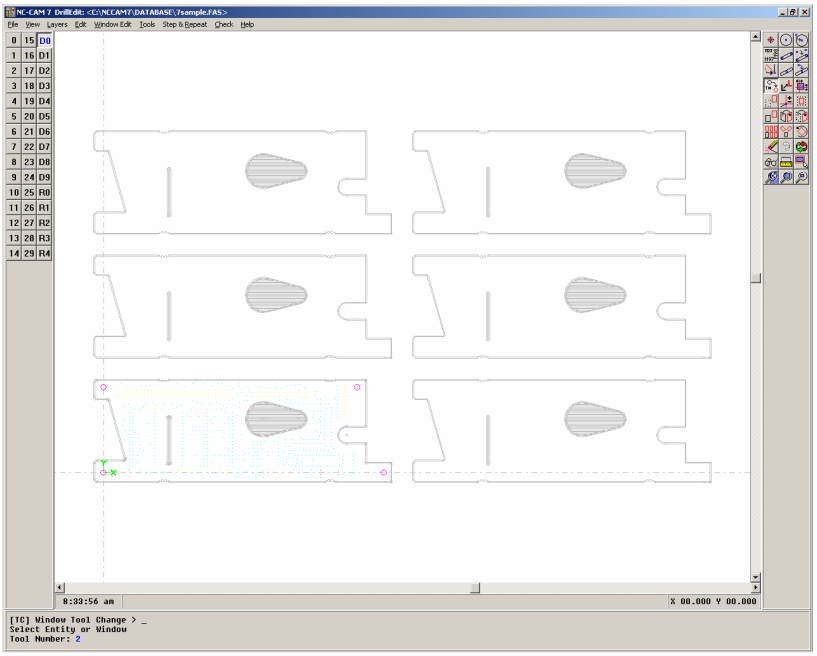
The title bar now says “DrillEdit,” and the buttons in the toolbar have changed to allow drill editing.

Load the sample drill file into NC-CAM by selecting ***File*** | ***Import*** from the main menu, which causes the “Select file to import” dialog to appear.



Click on the name "7SAMPLE.DRL" and then click on OK to select the file. The "File Format" dialog appears, with the type "Excellon Drill/Rout" selected, so click on OK to continue. A dialog appears with the title "Excellon-to-Drill," with several parameters available.

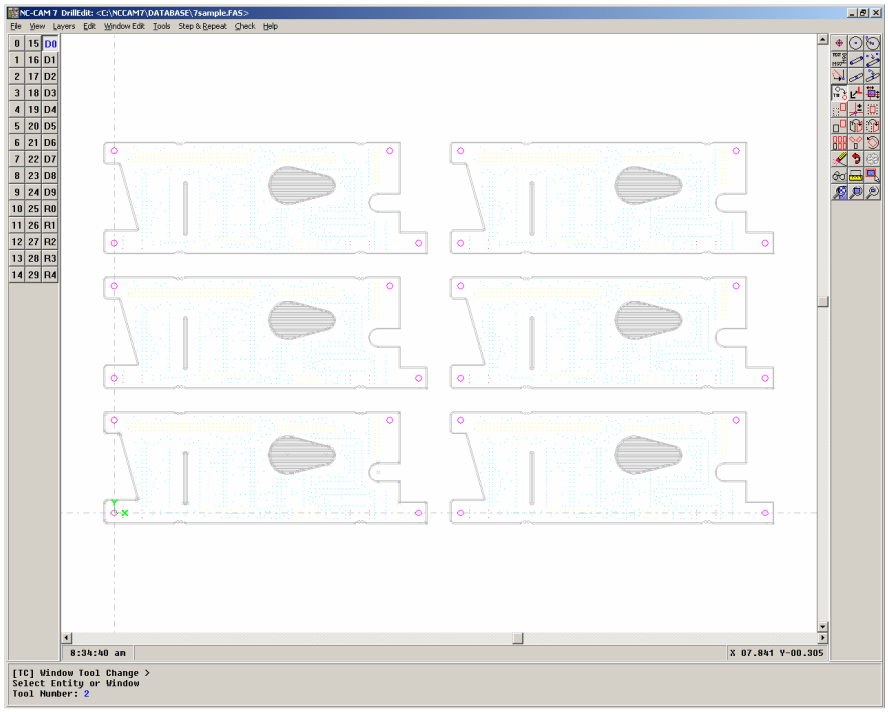
The sample file fits these parameters, so simply click on OK. The Import Results window pops up to show information on the file you just read in. This is just for your information, so click OK. NC-CAM will ask if this is the main part. Click on Yes to continue. NC-CAM will then ask what layer to place the data on with a dialog labeled "Select Layer For Data" with Layer D0 already selected, so click on OK.



As you can see above, NC-CAM will display the drill data in the work area.

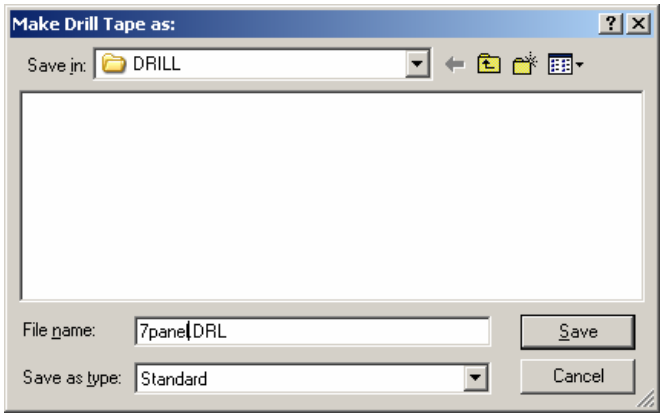
Step and repeats in drill programs

The step & repeat information for your drill data will typically mirror the step & repeat information for your rout data, since the boards will physically be layed out in the same configuration. Therefore, select *Step & **R**epeat | **A**uto Step & Repeat* from the main menu. To Select All of the drill data, type **SA** in the command window, and then hit **ENTER**. The "Auto Step & Repeat" dialog will appear, with the same values that you entered while editing the rout data. Click on OK to step & repeat the drill data in the same way.

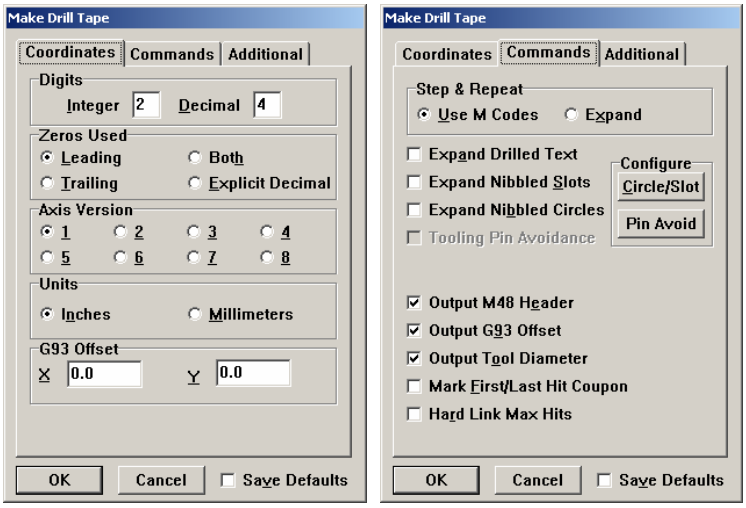


Output drill

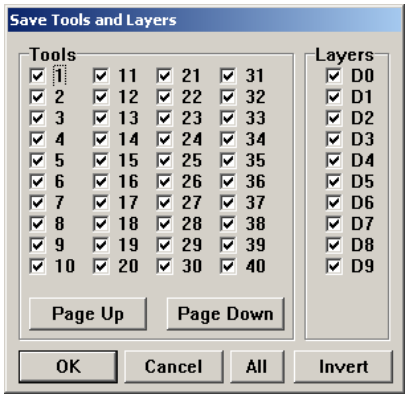
In order to save the drill data, select ***F**ile | **M**ake Drill Tape* from the main menu. A dialog labeled "Make Drill Tape as:" will appear, in which you should enter the name "7PANEL.DRL" and then click on OK.



Another dialog, labeled “Make Drill Tape” will appear.



You’ll want to pay close attention to the values that you set in this dialog. For now, change the Optimizer value to “OptiScan!” and turn off the checkboxes labeled “Mark First Tooling Hits” and “Mark Last Tooling Hits”, and then click on OK.



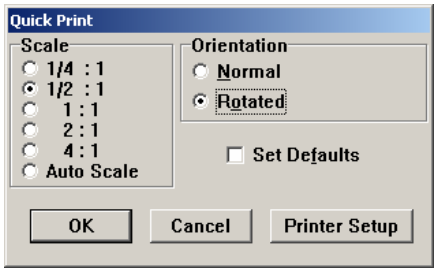
A dialog will appear, labeled “Save Tools and Layers” with all of the tools selected, which is fine, so click on OK to finish making a drill tape.

A window will pop up asking if you want to print out a tool table of the drill data in this file. Click Yes or No. You now have a completed Excellon drill program in your \NCCAM7\DRILL directory.

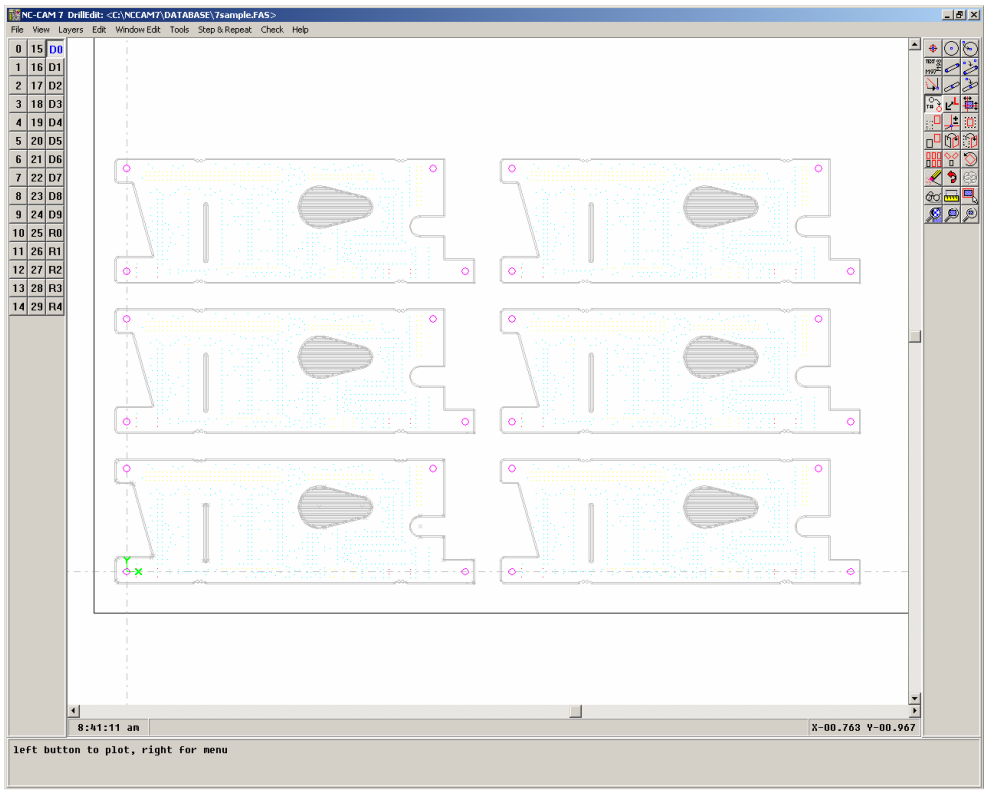
Printing

Be aware that NC-CAM will print to the active printer, in the same manner as all of your Windows applications. To change the active printer, select ***F**ile* | ***P**rinter Setup* from the main menu.

Select ***F**ile* | ***Q**uick Print* from the main menu, and a “Quick Print” dialog will appear.



Select a scale of 1/2:1 for the printing, and specify that the data be Rotated. Click on OK, and the dialog will close, and you will be prompted to specify the lower left-hand corner of where the page will be placed, relative to the work area.



Click on the lower left-hand most edge of the displayed data, and Quick Print will print to your active printer.

The Plot Configuration window will pop up, allowing you to add 2 lines of text to your drawing. Type in the text (i.e. your name, a part number, etc.) and click the Plot format block option. Click OK to send the screen image to the printer. If you do not want to add text, simply click OK.

C H A P T E R 3

Using the CAD and RoutEdit modules

This chapter provides a step-by-step introduction to NC-CAM's rout programming system, the CAD and RoutEdit modules. Through the course of the tutorial, you will learn how to:

- build routable CAD drawings from paper drawings
- build and output rout programs from your CAD drawings

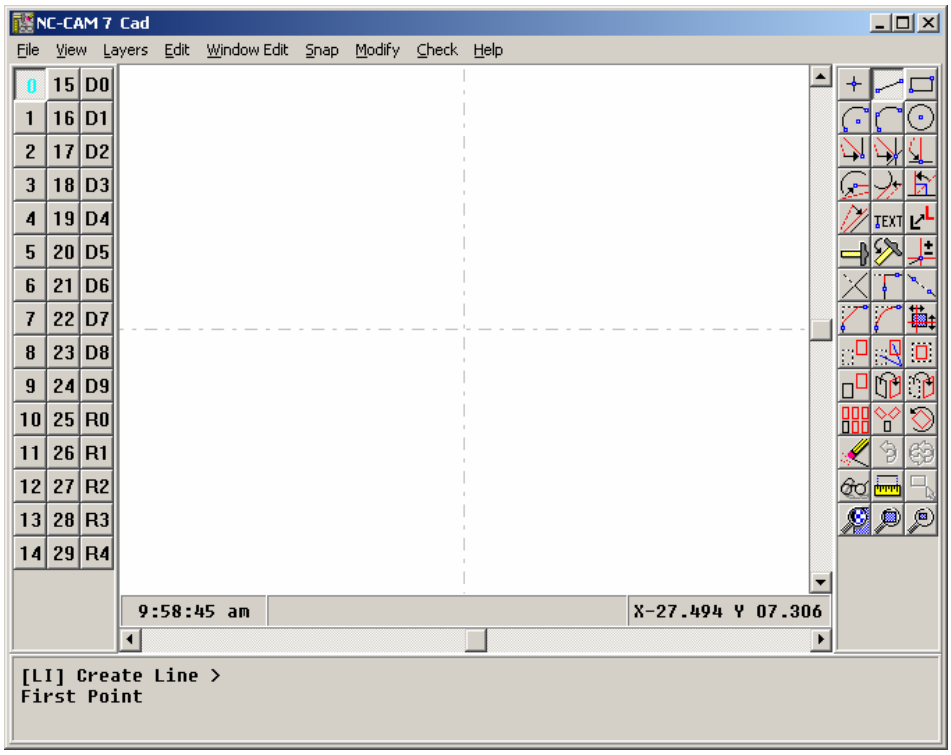
After reading this chapter, you should be able to access the CAD and RoutEdit modules, and use their basic features to create rout programs.

Your first CAD drawing

Generally speaking, there are two ways you'll begin when it's time to make a CNC router program. You'll be supplied with either a paper drawing of the customer's board profile or panel, or a data file (photoplotter, HPGL, or even an old rout program) *containing* the board profile. NC-CAM is very well equipped to handle each of these possibilities. This chapter will cover both situations.

Finding your way around

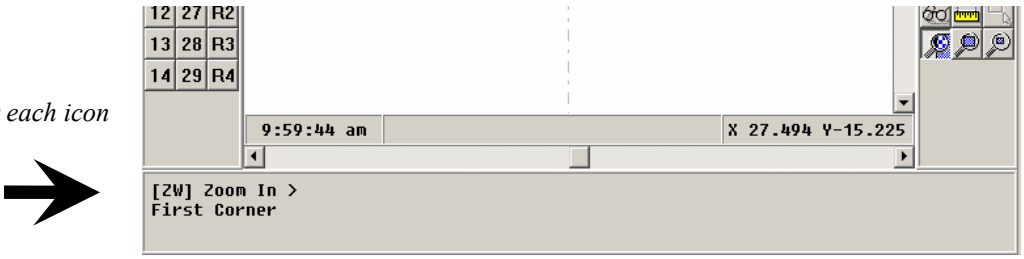
To begin your CAD session, select CAD layer zero on your layer buttons. The CAD layers are numbered 0 through 29. When a CAD layer is selected, you'll see the word "CAD" in the program's title bar, and you'll see the CAD toolbar at the right side of your screen.



In CAD, the majority of the work you'll be doing involves editing drawing entities. To do this, you need ready access to a large number of special editing functions. Most of these editing functions are made available to you in the icons on the right side of the screen.

To help you learn what these icons indicate, there is an automatic *fly-by* help built in to the CAD editor's menu system. To get a look at the fly-by help, position your cursor over one of the icons *without pressing any mouse buttons*.

Fly-by help for each icon



As you can see, the icon beneath the cursor is the button for the **ZW** *Zoom In* function. The two letters "ZW" in front of the function's name indicate the shortcut keyboard command for this function. Experienced users of older NC-CAM revisions will recognize **ZW** as the same command used by Generic CADD and NC-CAM 5 for the Zoom Window function. This consistency allows all experienced users of any revision of NC-CAM's CAD module with very little adjustment.

You should take the time to place the mouse over each of the buttons, and read the fly-by help for each of them. If you're an experienced 2D CAD user, you'll find that all of the major functions you're going to need for building printed circuit board profiles are available on these icons.

If you select an operation and press the **F1** Key, the NC-CAM Online Help will be opened to the entry for this function. This allows you to see a detailed description of each command. You can also access a complete directory of Help Topics for the features and operations of NC-CAM.

Drawing in CAD

Drawing the basic entities in the CAD module involves first selecting what you want to draw, then entering the points needed to draw it. There are four types of entities used in the CAD module.

CAD Entities

Points: Points are shown on-screen as small plus "+" signs. They are defined by a single XY coordinate. The point shortcut command is **PO**.

Lines: On-screen, lines just look like thin, straight line segments. They are defined at their two endpoints by two XY coordinates. In CAD editing for board profiles, the lines (and arcs and circles) have no width attribute: They are regarded as being infinitely thin. The shortcut command for *Create Line* is **LI**.

Arcs: On-screen, arcs are shown as semicircles. They are defined by three XY coordinates; one at each endpoint, and one at the center. The shortcut commands for making arcs are **A2** and **A3**.

Circles: A circle is defined by its radius and an XY center coordinate. The circle shortcut command is **C2**.

Every drawing you'll ever create or import into CAD will be represented by combinations of these four types of entities. The small number of data types allows the CAD editor to provide you a very predictable, uncomplicated way of working

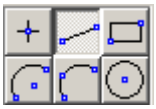
with the drawings to make rout profiles, regardless of where or how the drawings were made.

Coordinates and dimensions

In using CAD to make profile drawings of printed circuit boards, you're called upon to produce very exacting drawings, with precise coordinates and dimensions. Because of this, you cannot avoid using the keyboard quite a lot, since entering accurate values is the only way to produce accurate drawings. *The most important part of learning to use a CAD editor for PCB work is learning when to use the keyboard, and when to use the mouse*. In general, using the mouse is much faster than using the keyboard, so the technique you'll learn is to construct your drawings by snapping and trimming lines to other lines you've already created, thus avoiding as much keyboard entry as possible.

To enter a coordinate with the keyboard, you type the X value, a comma, then the Y value, and press **ENTER**. So, to enter the coordinate X 12.34 Y-4.56 you'd type in `12.34,-4.56` and press **ENTER**. The values are entered with decimal points and minus signs as needed.

Modal functions



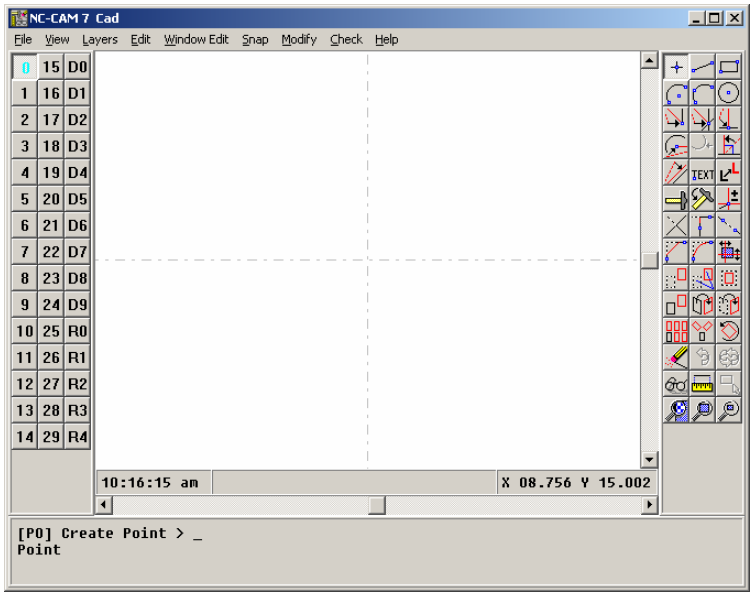
The button that stays depressed determines which function is selected.

When you select a drawing function in the CAD module, the toolbar button corresponding to the function will remain depressed. That's because the drawing and editing function buttons are *modal*. When a toolbar button is "modal," it remains selected until you select a different toolbar button. Note that in NC-CAM, the main drawing and editing functions are generally modal, while the modifier buttons (like snaps) are not modal.

Sometimes you may forget which function you've got selected, and by clicking the mouse you'll start to draw a feature you really don't want. Any time you want to abort a drawing function in NC-CAM, press your **Esc** key. Pressing the **Esc** key will free you from whatever function you may be "stuck in".

Making points

Let's start our CAD drawing by creating some points. To create a point, select the point icon (at the top of the menu), or type **PO**. You'll see that the prompt at the bottom of the screen now indicates you're about to create a point.

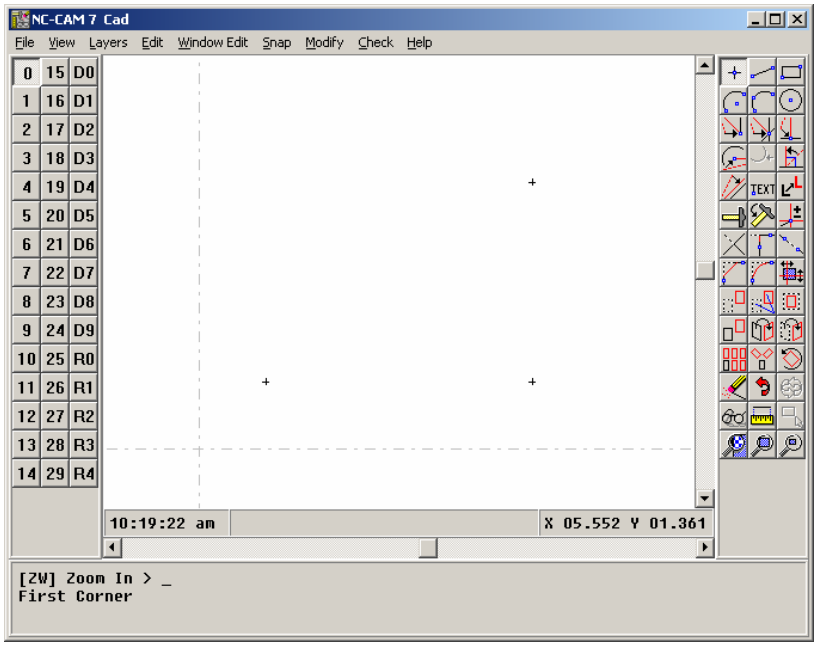


CAD allows you to combine its functions in a variety of ways. If you ever lose track of what function or combination of functions you've selected, look to the text prompts at the bottom of the screen.

Begin this drawing by creating a point at X 1.0" Y 1.0". Do this by typing `1,1` and pressing **ENTER**. You'll see a plus "+" sign appear near the intersection of the two dashed zero-zero indicator lines. Make another point by typing `5,1` and again pressing **ENTER**. (The **PO** point function is modal, so you don't need to reselect it to make a second point). Make a third point by typing `5,4` followed by **ENTER**.

Zoom All

To get a better look at the three points you've made, select Zoom All either by pressing the button, or by typing the two-letter shortcut **ZA**.



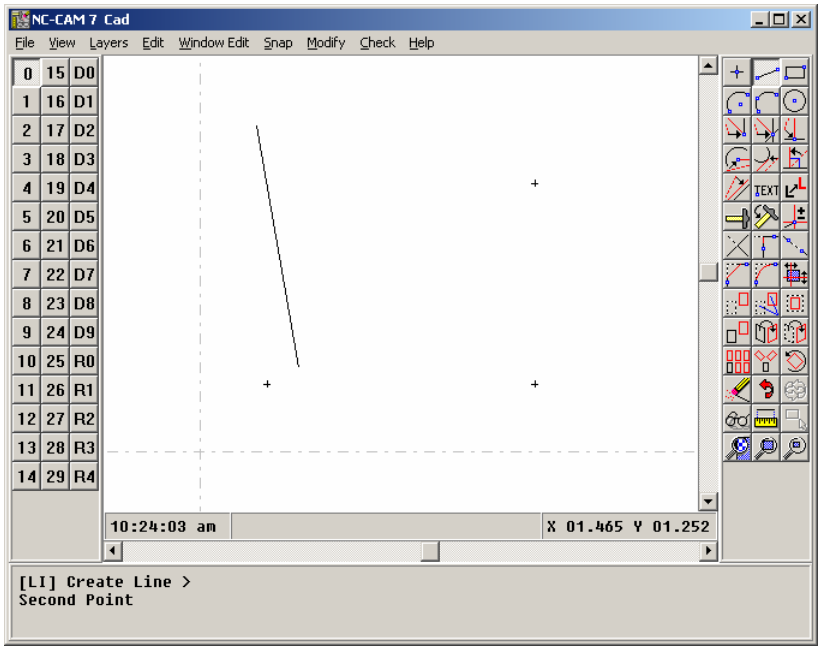
As you can see, the **ZA** *Zoom All* function re-sizes the screen to zoom in as far as possible, while still showing all of the entities in your drawing. The other zoom commands are:

ZB *Zoom Back:* Zooms out by a factor of two (makes things smaller)

ZW *Zoom Window*: Zooms the screen in on a user-defined window.

Making lines

The command to make a line is **LI.** You may select the line function by typing **LI** or pressing the line button. Start your first line by typing its endpoint, `1,4` then press **ENTER**. When you move your mouse around, you'll see a rubber-banded line is attached to your cursor.



One of the ways you can avoid entering coordinates on the keyboard is to *snap* to existing points in your drawing. Now move your cursor close to the lower-left point, as shown in the figure above.

Snaps

With the cursor positioned as shown on the preceding page, type **NP** on your keyboard. **NP** stands for "nearest point," and it causes your cursor to snap to the nearest point, and it simulates a center mouse button click there. As you can see, your line has snapped to *precisely* the same coordinate as the first point you created.

If you're using a three-buttoned mouse, you can access the **NP** command by pressing your center mouse button. It's a real time-saver!

CAD offers several more snap commands. They are:

SC *Snap Closest:* Acts exactly like the near point snap, except that it waits for you to click your left mouse button, allowing you to move the mouse close to the endpoint you wish to snap to.

SI *Snap Intersection:* Asks you to select two entities, then snaps to their intersection point. (If the entities intersect in more than one place, snaps to the intersection closest to the mouse when you pick the second entity).

SM *Snap Midpoint:* Snaps to the midpoint of the line or arc you select.

SN *Snap Center:* Snaps to the center of the circle or arc you select next.

ST *Snap Tangent:* Snaps to the edge of the arc or circle you select next. Doesn't establish the line end immediately: It keeps moving the snapped end to keep the line tangent to the arc or circle as you move your mouse.

SP *Snap Perpendicular:* Snaps to the entity you select next. Like the snap tangent function, this does not establish the line end: It keeps moving the line end to keep the line from your mouse perpendicular to the entity you selected. (Works on arcs and circles, too!)

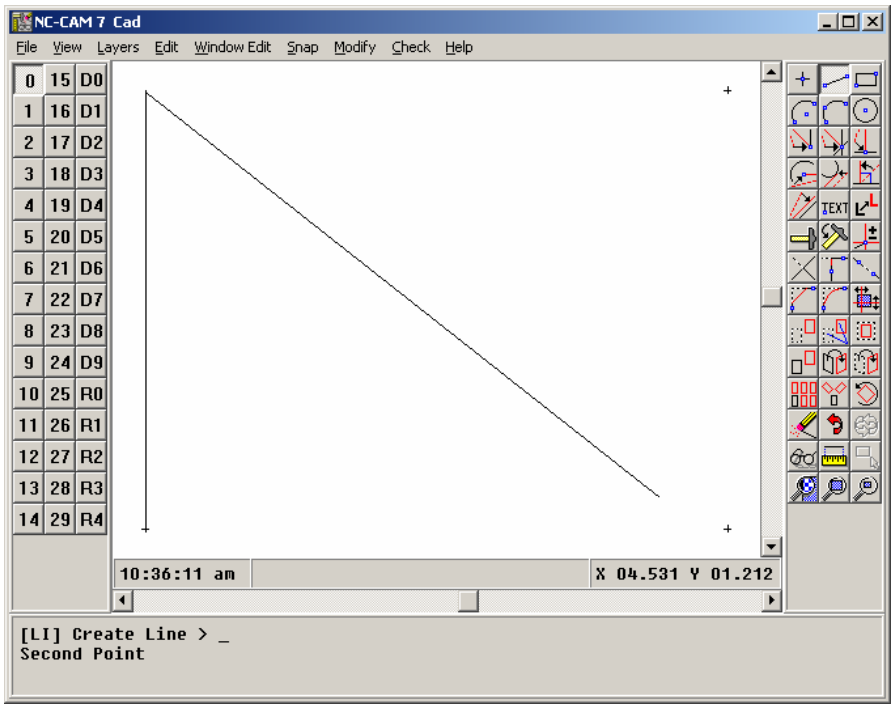
The snap commands save a tremendous amount of time, make it possible to avoid entering redundant coordinates, as well as performing complex trigonometric functions.

To complete this part of the drawing, move your mouse close to the top end of the line you've made. Type **NP** on your keyboard. This will snap the cursor to the top of the line, and begin drawing a new line (since you did not select any other modal command).

Now, let's see what happens when you make a mistake while drawing in CAD. Assuming that our drawing is supposed to end up as a plain rectangle, it would be a mistake to connect a line from the upper-left corner to the lower-right. Go ahead and move your mouse to the lower-right corner, and click your left mouse button.

Undo...Redo

The line you've just made doesn't belong in this drawing. If you've taken the time to look at the buttons in the CAD menu, you've probably seen the "pencil eraser" icon. Yes, you could use the eraser to just erase the errant line you just made, but there's a more powerful tool you could use. For almost any error you make in your CAD drawing, you can undo the mistake by selecting **OO**, the Undo command.



Place your mouse over the "reverse arrow" icon, and you'll notice that the help prompt at the bottom-right corner of the screen says, "Undo make line." If you press the button, the bad line you've made will disappear.

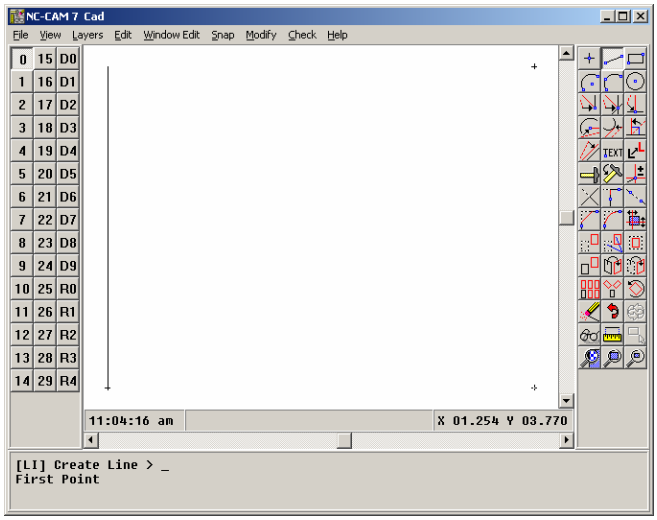
Redo

If you press the "Undo" button again, you'll erase the line at the left side of the drawing. Now you've undone something you actually wanted to keep! Never fear, just move your mouse to the **UU** *Redo* button (just to the right of the Undo button), and read the help prompt.

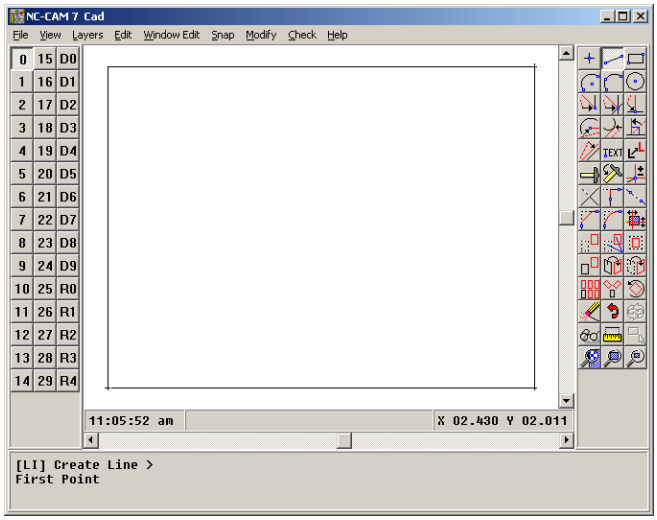
As you can see, the Redo function lets you put back changes you've reversed with the Undo function. You can think of the Undo as acting somewhat like a “rewind,” and Redo as acting like a “replay” function.

More snaps

To continue your drawing, select the **SC** *Snap Closest* function. This behaves much like the **NP** *Near Point* function you've already used, except that it waits for you to reposition your mouse and click your left mouse button before it snaps to the closest point.



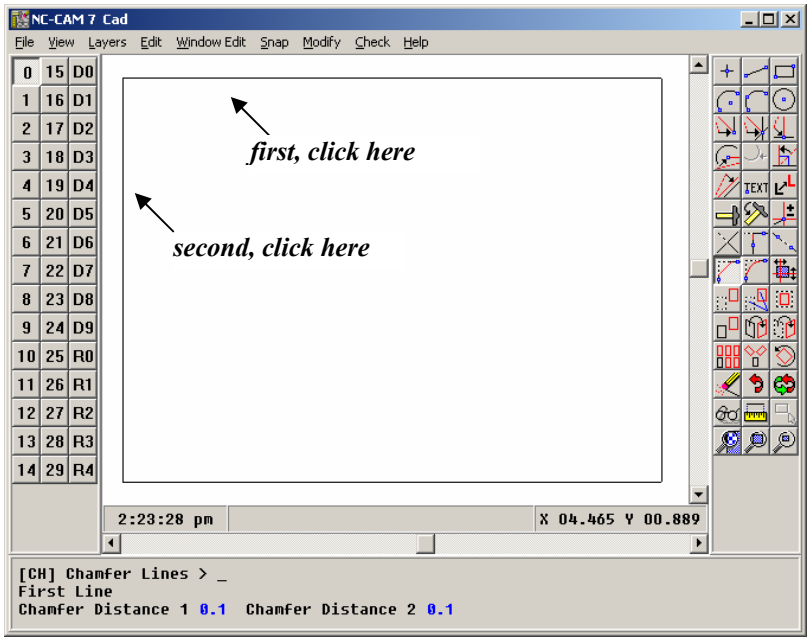
After selecting **SC** *Snap Closest*, place your mouse near the top of the line at the left side of your screen, and click your left mouse button. You'll see the starting end of your line snap to the top of the other line. Now select *Snap Closest* again, position your mouse near the point at the upper-right corner of your screen, and again click your left mouse button. You'll see that the line end has snapped to the point at coordinate X5Y4.



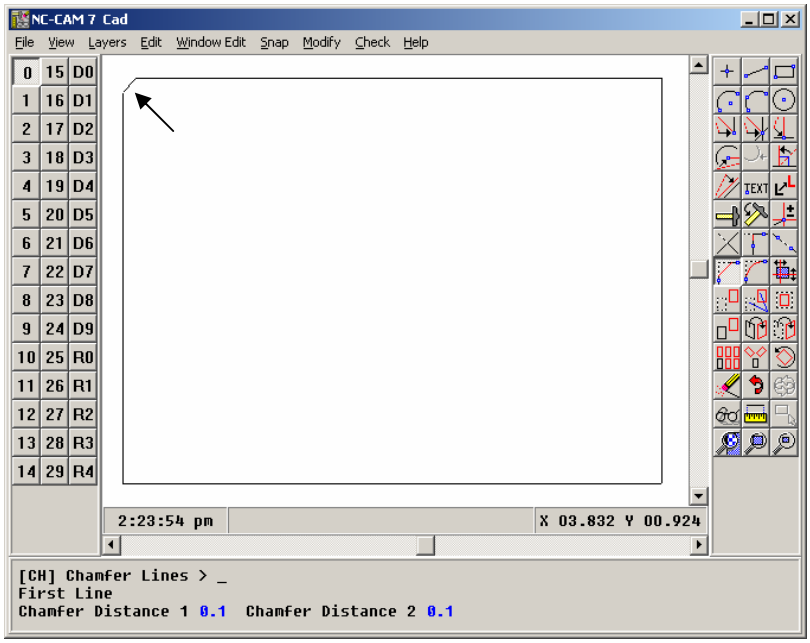
Use **LI** *Create Line* and **SC** *Snap Closest* to add two more lines, completing a rectangle as shown above. If you make any mistakes, “undo” them by using the **OO** *Undo* command.

Chamfer

Often times, you'll need to chamfer square corners in a drawing. To chamfer a corner, type **CH** or select the *Chamfer* icon on the menu.



After you select **CH** *Chamfer*, you must identify the intersection where you want to make the chamfer. To identify the upper-left intersection, click first near the center of the top line. You'll see the line become highlighted. Then, click near the center of the line at the left side of the screen. You'll see the lines highlight as you select them, then you'll see the chamfer made automatically.



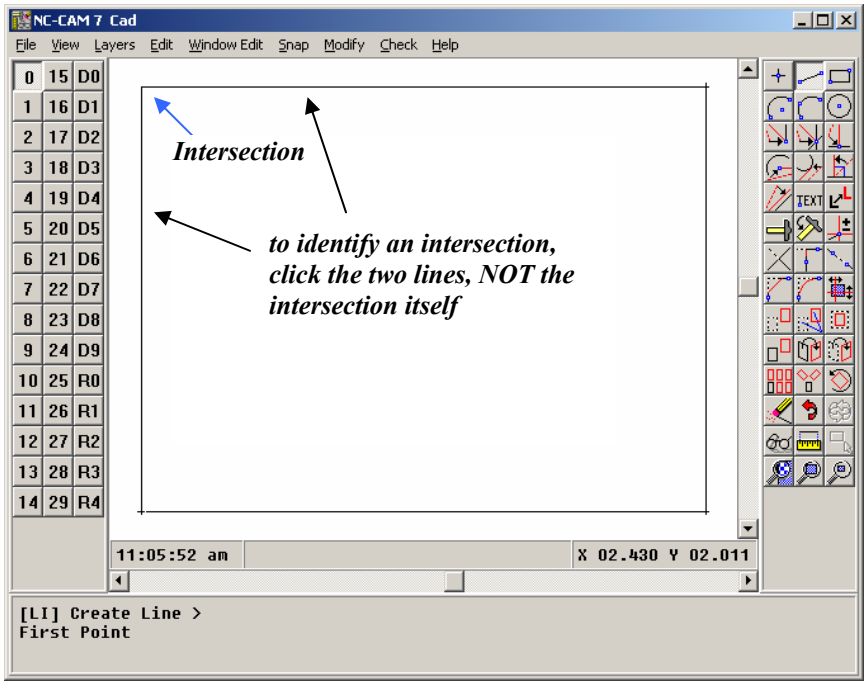
Here you can see the 0.1" chamfer made at the upper-left corner of the rectangle. There's an important lesson to be learned in the way you indicated the intersection at the upper-left corner to the CAD program: You selected the *entities* that intersected, *not the intersection itself*. Many of the edits and snaps you'll want to perform in CAD require that you select intersections, so we'll examine this further.

Selecting intersections

Some of the most powerful editing functions you can perform in CAD require that you select an *intersection*. The **CH** *Chamfer* function shown in the preceding pages gives one example of a function that "happens" at an intersection. Before we examine other functions which act on intersections, we'll focus on this idea of selecting an intersection in CAD. First, there's the definition of an intersection:

> **INTERSECTION: The point where two entities either meet, or *would* meet if they were extended. (CAD makes no distinction between actual and would-be intersections).**

Any time you need to select an intersection in CAD, select the two entities which intersect, not the intersection point itself.



In this example, assume you want to select the intersection in the upper-left corner of the drawing. To do this, you would first select the top line, then the left line. You would *not* click near the intersection, because CAD needs to find the entities before it can compute their intersection. The next pages will introduce you to several functions that deal with intersections. In each case, when CAD needs to know which intersection you want, you *must* identify it by clicking separately on *each* of the two intersecting entities.

More about Chamfer

When you select **CH** *Chamfer*, you'll see something new at the bottom of your screen. The extra text below the command line describes the current distance settings for the chamfer function.

Assume that you need to make an asymmetric chamfer which is 0.25" on one side, and 0.5" on the other. Begin by selecting the **CH** *Chamfer* function. The text at the bottom of your screen will show:

```
[CH] Chamfer Lines >
First Line
Chamfer Distance 1 0.1  Chamfer Distance 2 0.1
```

To change the chamfer settings, select *Chamfer Distance 1*, which is now set to "0.1." You may select the *Chamfer Distance 1* field by clicking on it with your mouse. After you've selected the field, type `0.25` then press **ENTER**.

```
[CH] Chamfer Lines >
First Line
Chamfer Distance 1   0.25 hamfer Distance 2 0.0
```

Select the *Chamfer Distance 2* field, and set it to one half inch by typing `0.5` and pressing **ENTER**.

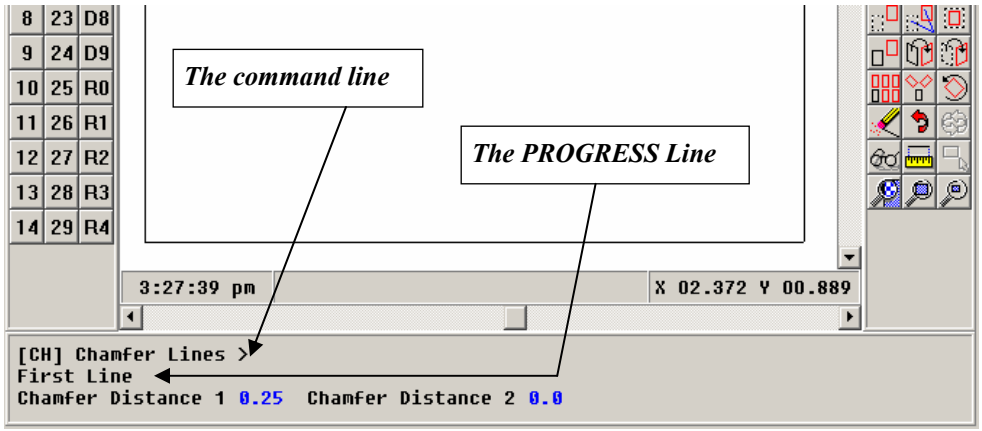
Now, chamfer the upper-right corner of the rectangle, by clicking on the right-hand line, then the top line. You'll see that the short side of the chamfer is on the line you clicked first; the right-hand line. This is Chamfer Distance 1. The long side of the chamfer, Chamfer Distance 2, is on the line you clicked last; the top line.

Points vs. Entities

In CAD, sometimes you need to enter points, and at other times you need to select entities. How do you know what CAD is expecting at any given time? The answer is usually found in the *progress prompt* at the bottom of the screen.

When you're creating something new in CAD, you need to define a point, or possibly several points. You may define points by clicking your left mouse button, typing in coordinates, or snapping to entities. When you're editing or snapping, you need to select entities that are already present in your drawing. The progress prompt usually indicates what CAD is expecting you to do next.

when in doubt, check the command line and the progress prompt



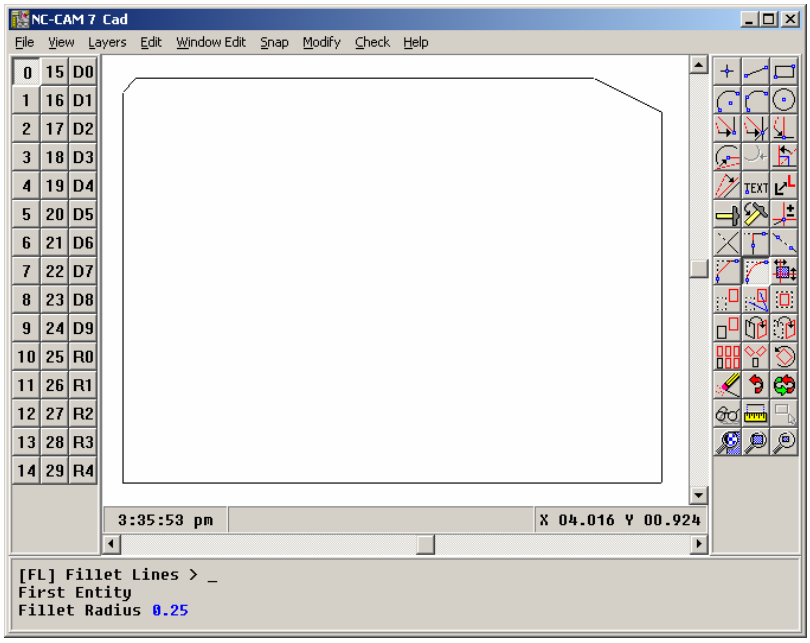
As shown in the illustration, the top two lines of the text box are called the *Command Line*, and the *Progress Prompt*. When you type coordinates and commands in CAD, these will appear on the Command Line. If you've selected one or more functions (like **LI**, then **SC**), these commands will appear to the left of the "**>**" prompt on the Command Line, as a reminder to tell you what you've entered.

To abort any function: right mouse button or the ESC key

When you select a command that requires several steps to complete, the step that's needed next will usually be indicated in the Progress Prompt. In the case of **CH** *Chamfer*, CAD needs to "know" which two lines you want to chamfer. So, after you've selected **CH** *Chamfer*, the Progress Prompt says "First Line." This means that, in order to proceed with the chamfer, CAD needs for you to select a line.

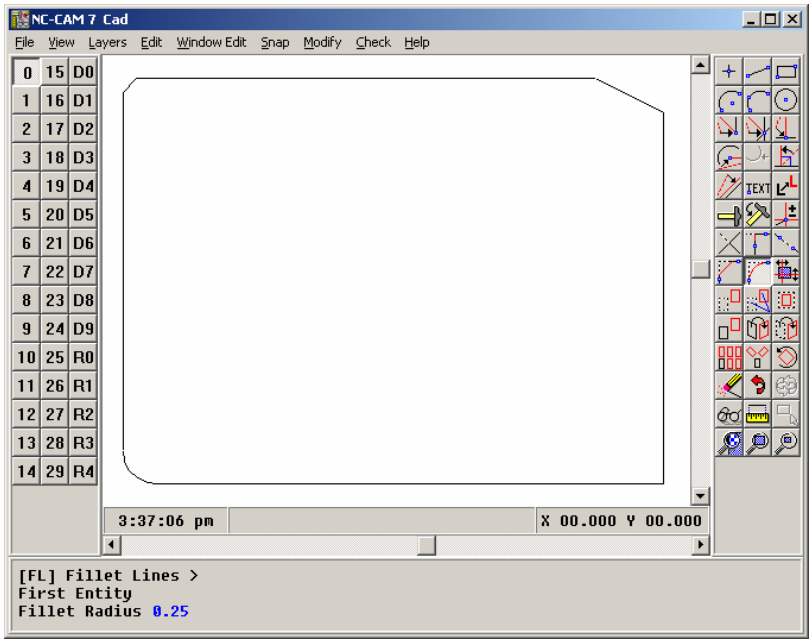
Fillet

Radiused corners are another common feature of routed PCB profiles. In CAD, a radiused corner is called a *fillet*. CAD will automatically make a fillet between any two lines (or arcs) with the **FL** *Fillet* command.



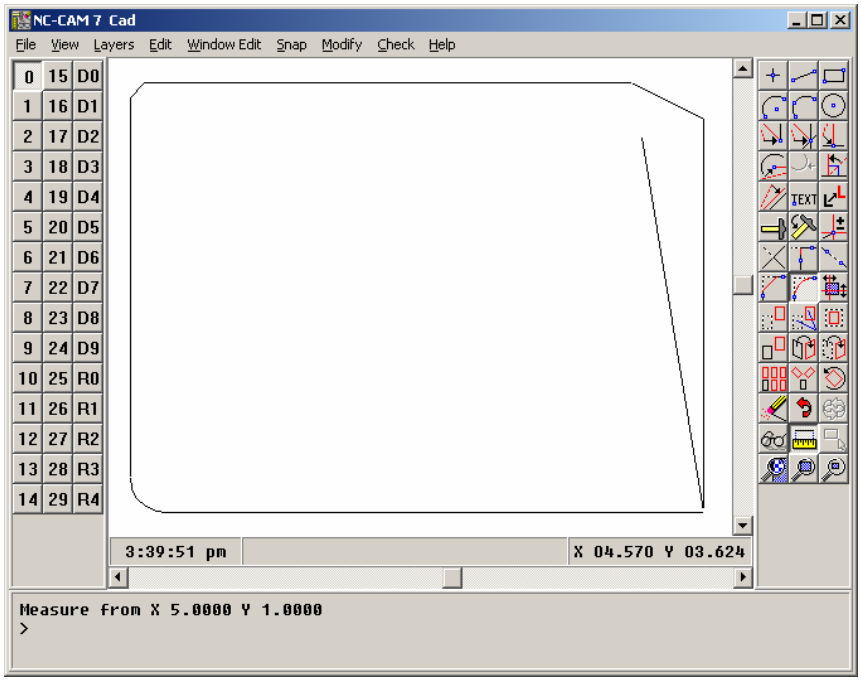
When you select the **FL** *Fillet* command, you'll see the fillet parameters appear at the bottom of your screen. Fillet works exactly like the Chamfer function, inasmuch as it requires you to select the two entities you wish to join with a radiused corner. In this case, select the bottom and left-hand lines to fillet the bottom-left corner of the rectangle.

Measuring

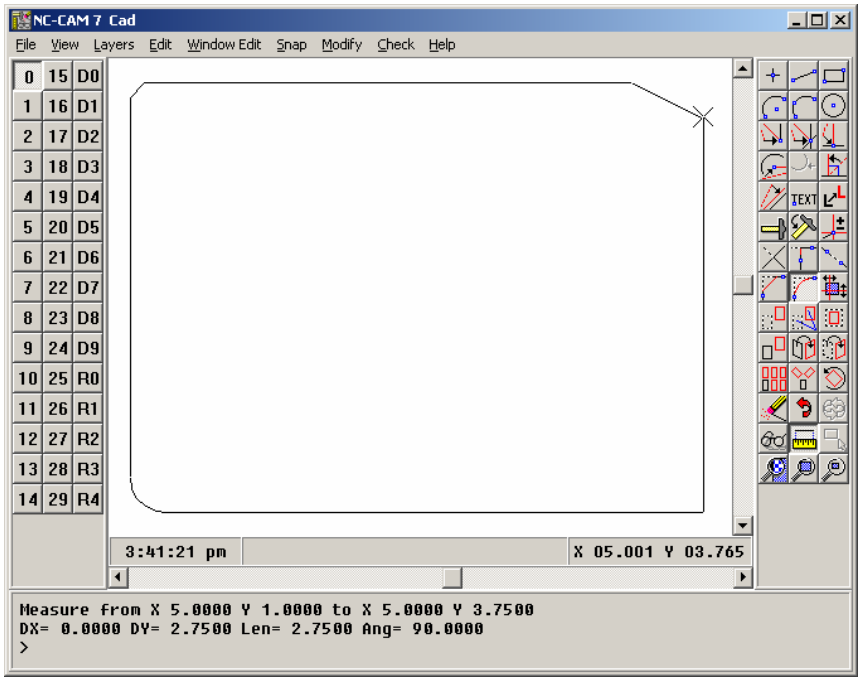


If you've followed the steps on the preceding pages closely, your drawing should match the above illustration.

There are two inspection functions in CAD; **ME** *Measure*, and **OI** *Object Inspect*. Select the **ME** *Measure* function, and click your mouse in the lower-right corner, as shown. The cursor will snap to the corner, and attach a rubber-banded line to it. The end of the rubber-banded line attached to the corner indicates the point you're measuring *from*.



Now, position your cursor as shown above, and click the left mouse button. The right-hand line will change color, and measurement values will appear at the bottom of the screen.



```
Measure from X 5.0000 Y 1.0000 to X 5.0000 Y 3.7500_
DX= 0.0000 DY= 2.7500 Len= 2.7500 Ang= 90.0000
>
```

The measurement values given are:

Measure from-to: Gives the start and end points of your measurement.

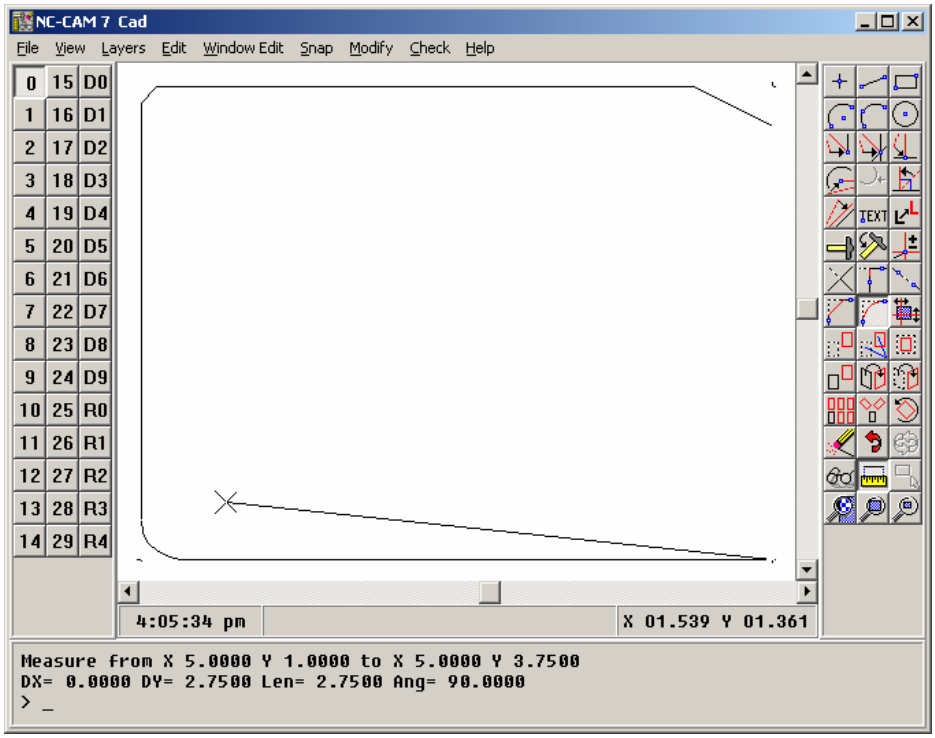
DX: The from-to distance in the X direction.

DY: The from-to distance in the Y direction.

Len: The from-to length

Ang: The from-to angle, measured from the positive horizontal axis.

You'll notice that there's still a rubber-banded line attached to your cursor, and there is also a large "X" under the center of the cursor. This is the other part of the measure function; the ability to measure angles.



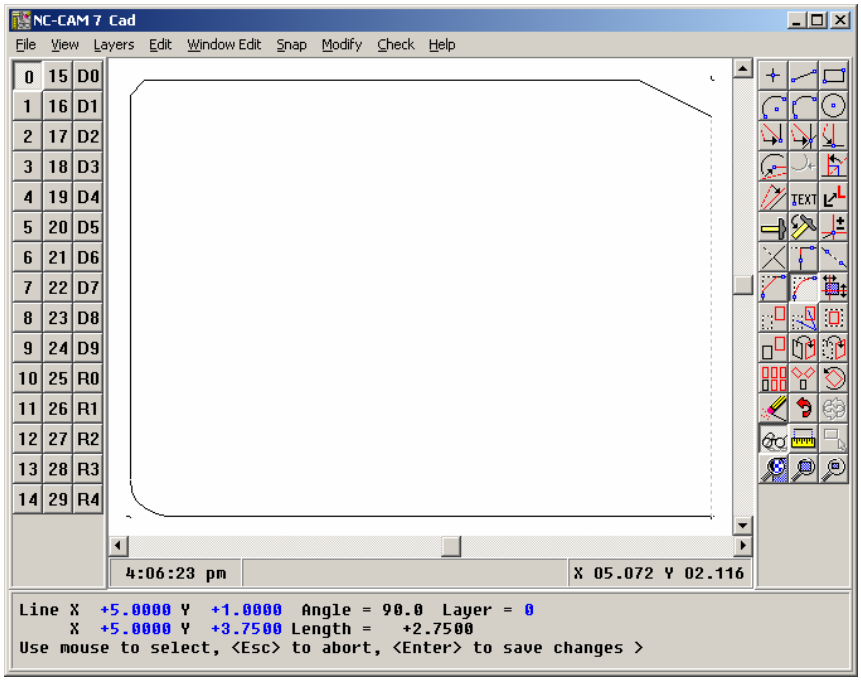
Position your cursor as shown, and click the left mouse button. You'll see the "X" snap to the left end of the bottom line, and the measurement report at the bottom of the screen will update to show you the statistics for the bottom line.

The arc between the two measurement lines indicates the angle that's being reported by the text "Ang=180.0000."

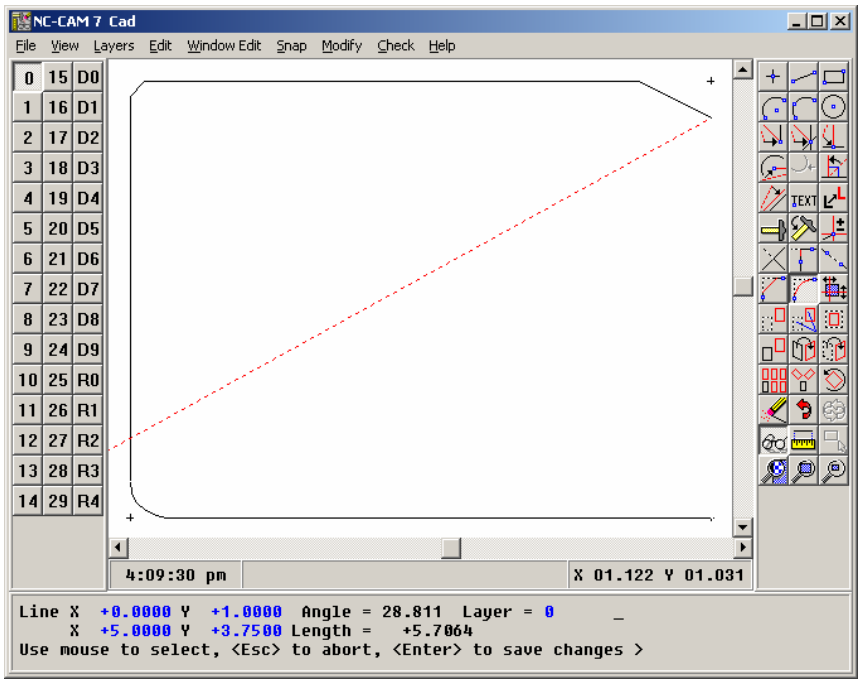
To take the **ME** *Measure* function a step further, click your mouse on the "X" at the left end of the bottom line. The "X" will re-attach itself to your cursor, allowing you to snap it to another point in your drawing for another measurement. Go ahead and test measuring to various endpoints in the drawing. When you want to exit the measure function, press your *right-hand* mouse button, or the **Esc** key.

Inspect

In addition to the measurement capability described in the preceding pages, CAD offers a powerful **OI** *Object Inspect* function. Object Inspect allows you to select an object, view all of its parameters, and *change them as desired*. Try selecting **OI** *Object Inspect*, and then click on the right-hand line.



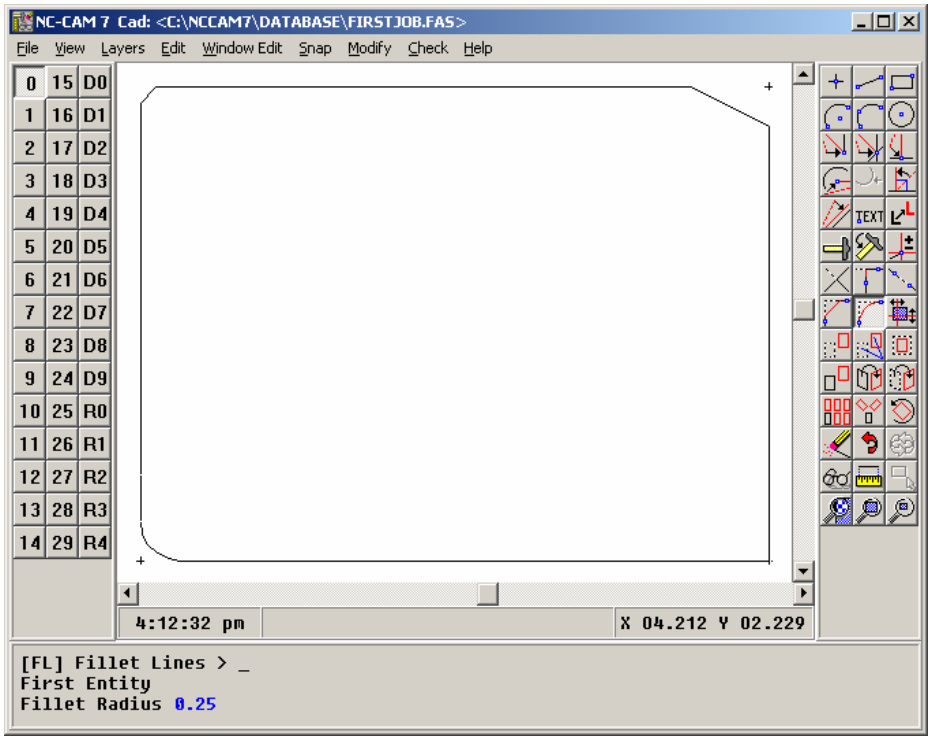
As you can see, the line's endpoint coordinates are now available in editable fields at the bottom of the screen. You can use this function to verify the values an entity has, or change them by clicking on any of them with the mouse. Try clicking on the upper-left X value, and typing `0` then **ENTER**.



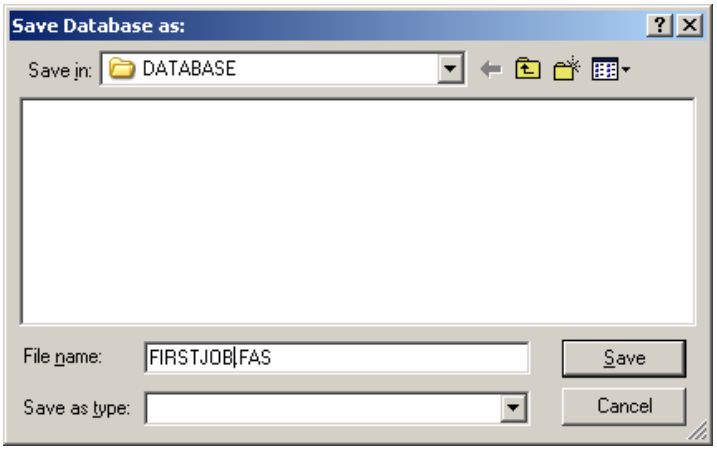
Here's the line with one of its X ordinates set to zero. If you wanted to keep the changed line, you would press **ENTER** (don't do it). To abandon the change, press **ESC.** The line will be restored to its original location.

Saving your work

If you've followed all of the steps in this chapter, your drawing should match the illustration below.



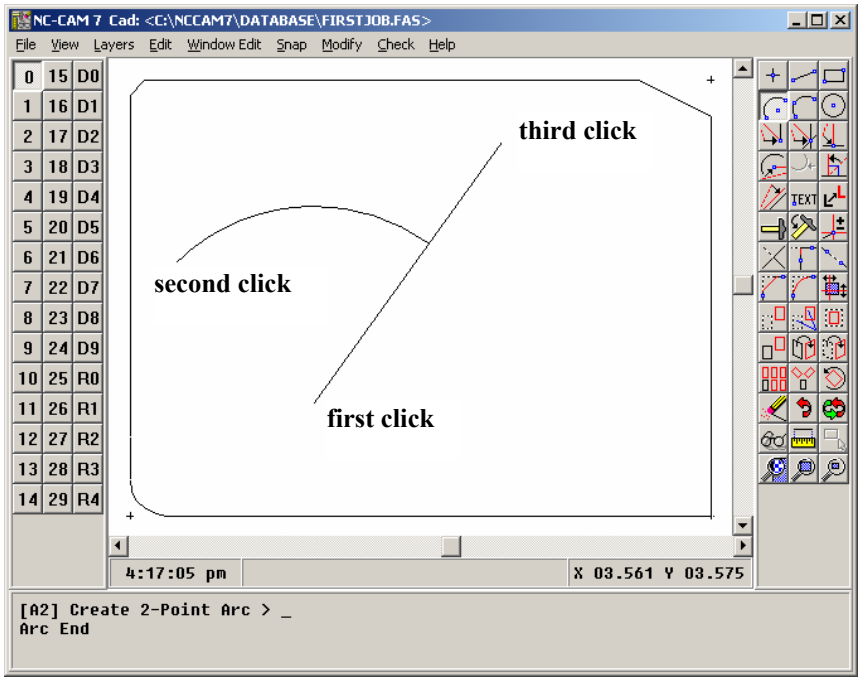
To save your drawing, select ***File/Save As...*** from the main menu. You'll see a dialog box appear, with the cursor blinking in the file name field.



Type a name for your file, for example `FIRSTJOB.FAS` and press **ENTER**. Your file will be saved to the default job directory, \NCCAM6\DATABASE. Note that when you save a job in NC-CAM 7, you're saving all layers of the CAD, Rout, and Drill data, to an NC-CAM 7 database file. If you want to save your CAD drawings to another format (such as DXF), select ***File/Export.***

Arcs

There are two functions for creating arcs in CAD; the two-point arc, and the three-point arc. We'll look at the more commonly used function, **A2** *Create 2-Point Arc*, so type **A2** or click the 2-point arc button.



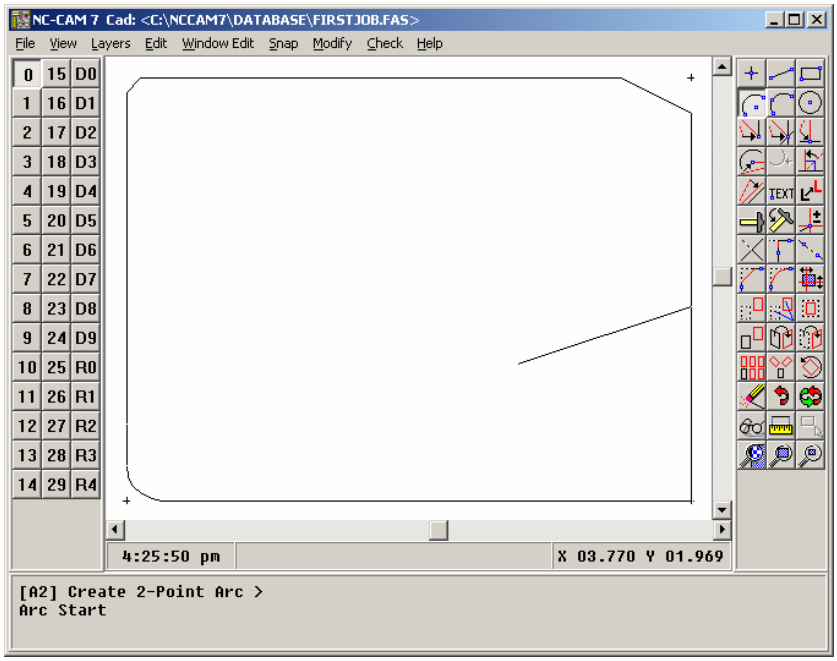
Just to get a look at how a two-point arc is created, use the mouse to click the arc center, arc start, and arc end as shown. As you can see, when you're making a two-point arc the first point establishes the arc's center. The second point sets the arc start *and defines the arc's radius*. The third point determines the arc's endpoint indirectly, by combining the arc's radius and the angle of the third point you enter.

Use **OO** *Undo* to erase the arc you just made. Next, we'll combine the **A2** *Create 2-Point Arc* command with other commands, to create an arc cut-out on the right-hand line.

Snaps with other functions

Snap functions can create accurate points quickly, when you'd otherwise have to compute them and type them in. You may use a snap function any time CAD is waiting for you to enter a point. To see this, begin by selecting the **A2** *Create 2-Point Arc* function.

To use a snap with the **A2** function, select **SM** *Snap Midpoint*. Next, click your mouse near the center of the right-hand line (as shown). The Snap Midpoint function will compute the exact coordinate of the line's midpoint, and snap the center of your arc to that location. Now you're ready to create the arc's start point. In this case, we'd like the arc to have a radius of one-half inch. How can you create the start point and know that it's set to a half-inch radius, when you don't know the coordinate of the center point? By using CAD's relative mode.



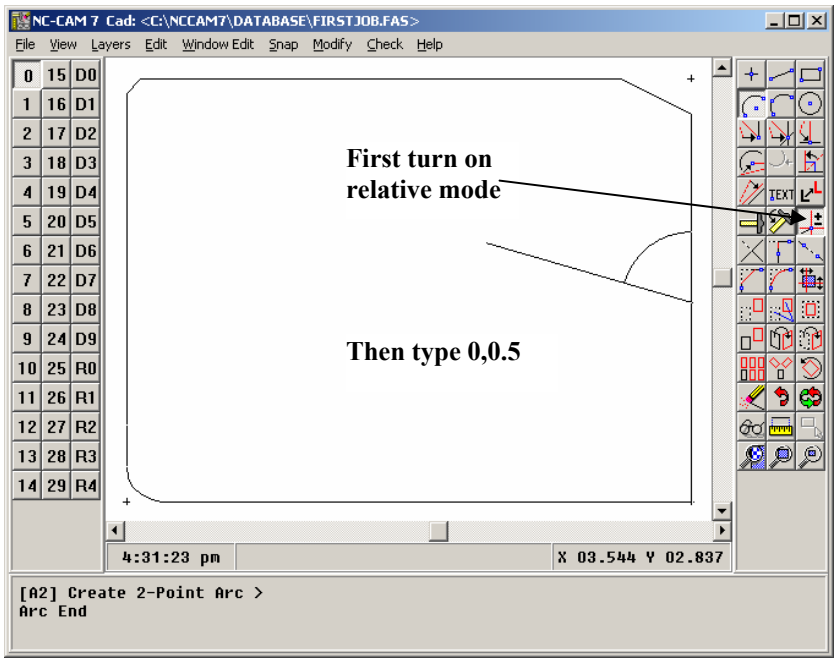
Relative mode

CAD drawings tend to be dimensioned arbitrarily. There's usually enough information in the dimensions for you to duplicate the drawing, but they're often given as distances from other features. The coordinates you've entered into CAD thus far have been absolute distances from your drawing's zero point.

For the half-inch arc you're making, it is convenient to enter the start point as a distance away from the arc's center point. To do this, you may toggle CAD's coordinate entry mode from absolute (the default) to relative mode. Type **MR**, or press the *Relative Mode* button on the menu.

When you turn on Relative Mode, notice that the button stays depressed. This is because Relative Mode is a *modal* function: It stays turned on until you shut it off. While the Relative Mode button is depressed, each coordinate you type will be taken as an offset from the previous coordinate. This is true whether you snapped to the previous coordinate, or you typed it in.

When you want to type in a coordinate as an offset, use relative mode

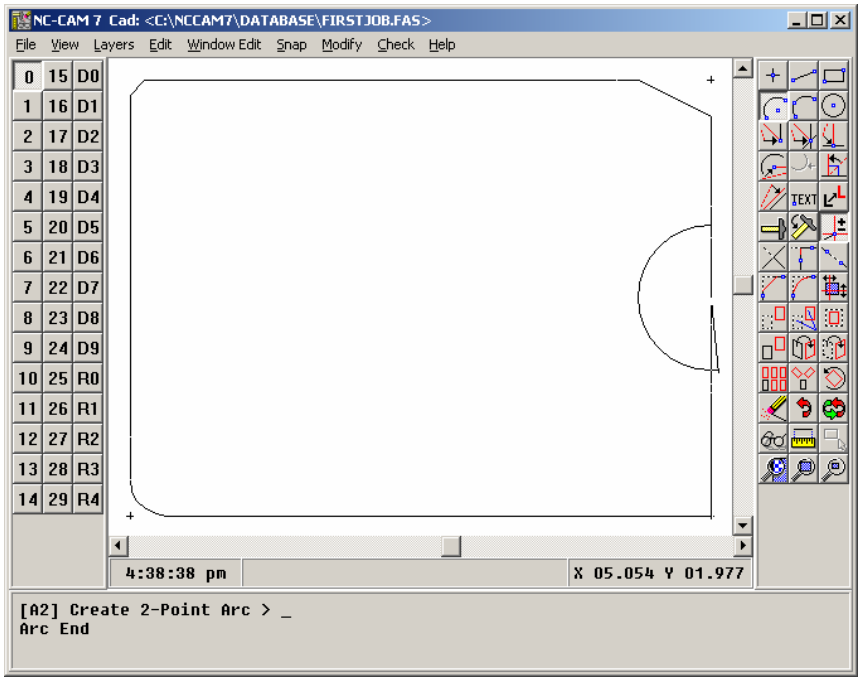


To set the "last point" you can do an NP (near point snap) to the point, then press Esc. The next coordinate you type will be relative to the point you snapped.

With the **MR** button toggled on, type `0,0.5` and press **ENTER**. This will set your arc's start point one-half inch above the center point. Now shut off Relative Mode by typing **MO** (for *Manual-Origin*), or press the Relative Mode button. The button will pop up to indicate that Relative Mode is now off.

Trims

Trims are fast; much faster then typing in coordinates



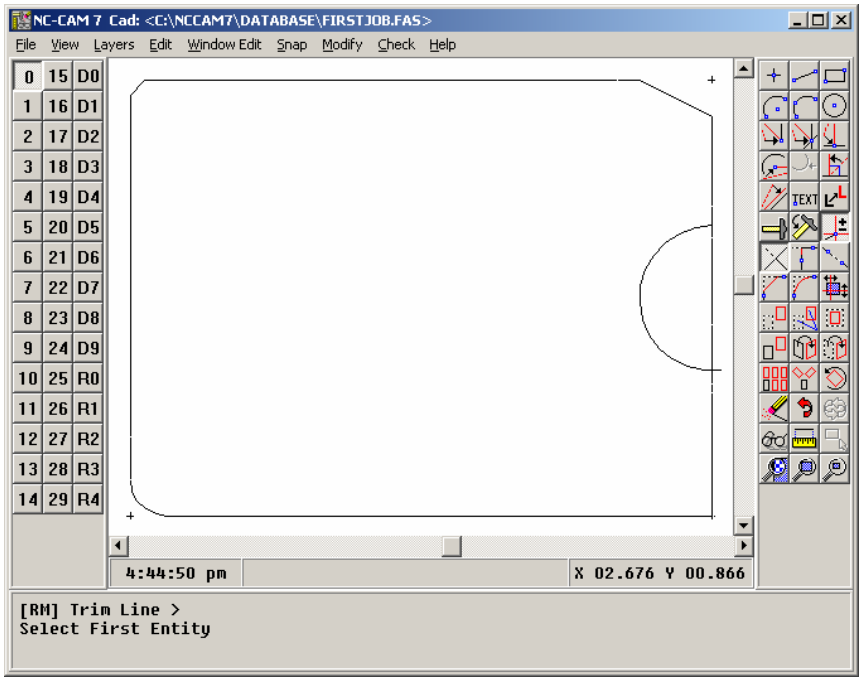
Complete your arc by using the mouse to drag the arc down, and then past the right of the vertical line as shown. Click your left mouse button to complete drawing the arc.

With the arc drawn as shown, you'll need to trim the arc's end point to stop at the vertical line. This idea of trimming lines to other lines is used often in CAD, because it helps you make absolutely accurate features without having to calculate coordinates. Once you've created an accurate feature in CAD, the Trim functions allow you to use the feature like a knife, to cut new lines to precisely where they intersect the existing feature. The trim functions cut (or extend) lines and arcs. You can use lines, arcs, or circles as the "knives" for cutting lines and arcs.

As with intersections, the trim functions work whether the two entities actually intersect or not. If two lines (or arcs) *could* intersect if they were extended, the trim functions work.

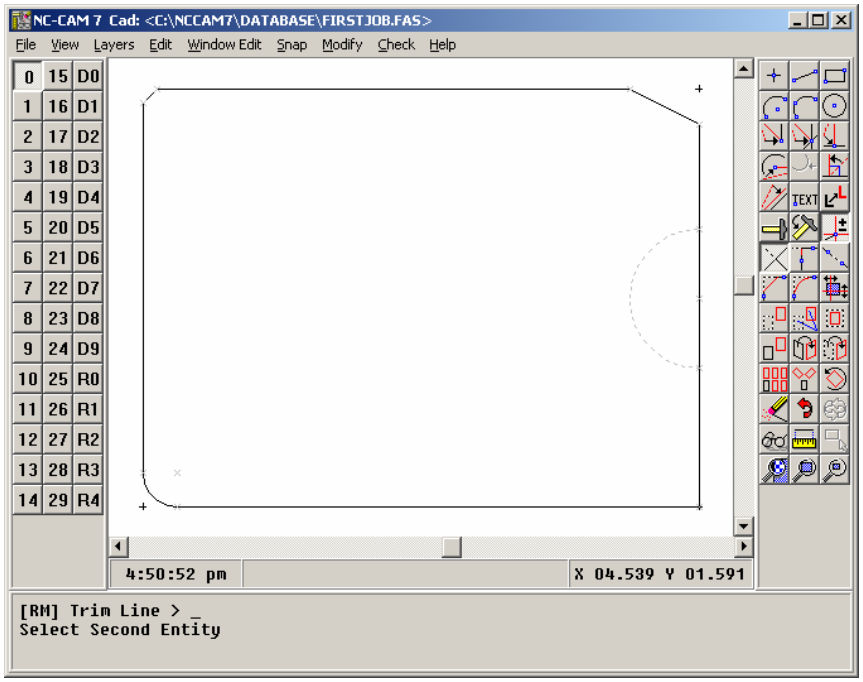
There's one major idea to keep in mind when you use the trim functions: *Click the side you want to keep.* If you're trimming a line that crosses something else, you've got to click the side of the line you want to keep, not the side you want to throw away.

*Click the part of the arc you want to **keep***



To trim the bottom of the arc to the vertical line, select **RM** *Trim Line*, position your mouse as shown, and click the left mouse button. This selects *what* you want to trim, and *which* side of it you want to keep. Next, click on the line as shown below.

the line you select will act like a knife, trimming the arc.

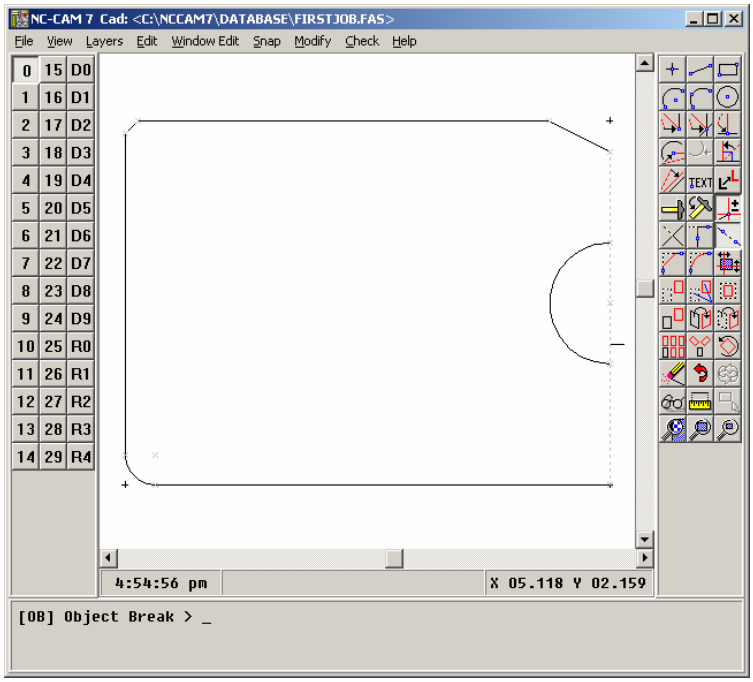


If you click at exactly the two points indicated, the trim function will neatly trim your arc to the point where it intersects the vertical line. If anything went wrong, use the **OO** *Undo* function to reverse the mistake, and try again, paying close attention to your mouse positions.

Breaking lines

To complete the sample board outline, you'll need to break the right-hand line at the top and bottom of the arc.

Break lets you break one object into two



Select the **OB** *Object Break* function, and click with your mouse on the right-hand line. Remember, when you're selecting an entity, put the cursor *on* the entity, *away* from any other entity.

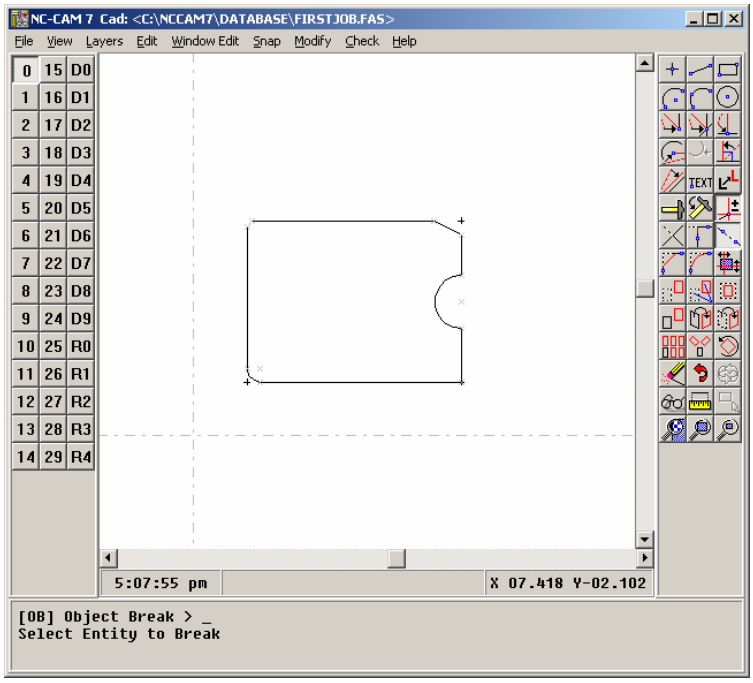
To break things precisely, use a snap function

When you've selected the vertical line for the Object Break operation, the line will highlight, and you'll have a rubber-banded indicator line attached to your cursor. This line indicates the point where you'll break the right-hand line if you click your left mouse button. In this case, you want to break the line exactly where it meets the arc, so you'll use the **SC** *Snap Closest* function. Select Snap Closest, position your mouse as shown in the illustration, and click your left mouse button. If you now move your mouse up and down, you'll see that the right-hand line will be broken exactly where it meets the bottom of the arc.

To complete the break operation, again select **SC** *Snap Closest*, place the cursor near the top of the arc, and click your left mouse button. You'll see the line is neatly broken into two lines which end exactly at the top and bottom of the arc.

Check your profiles with test path before routing.

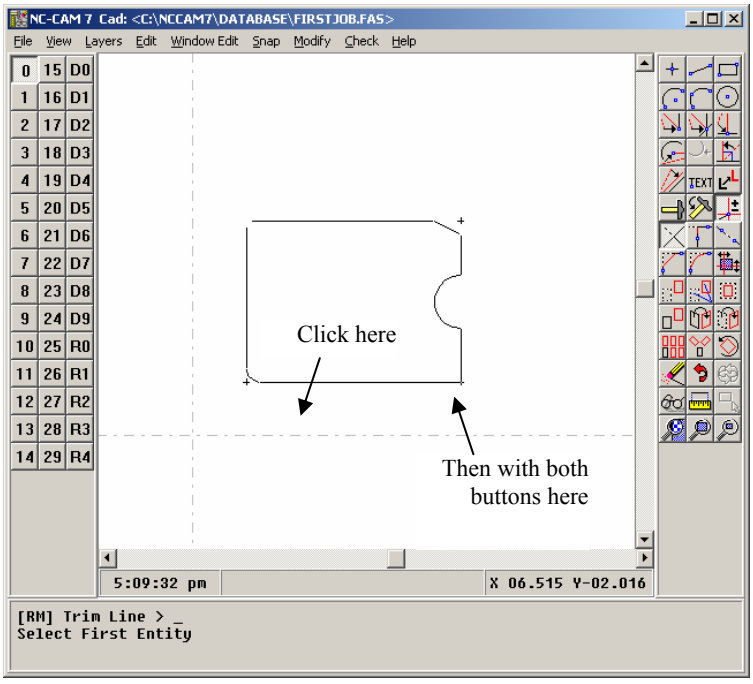
*The profile-follower goes towards the line end you're **closed** to when you click both buttons.*



Select the **ZB** *Zoom Back* function to get a good overall look at your drawing. If you've followed all of the instructions in this chapter, your drawing should match the above illustration. If not, you'll want to review your steps now.

This would be a good time to save the job to disk again, using the ***F**ile/**S**ave* main menu function.

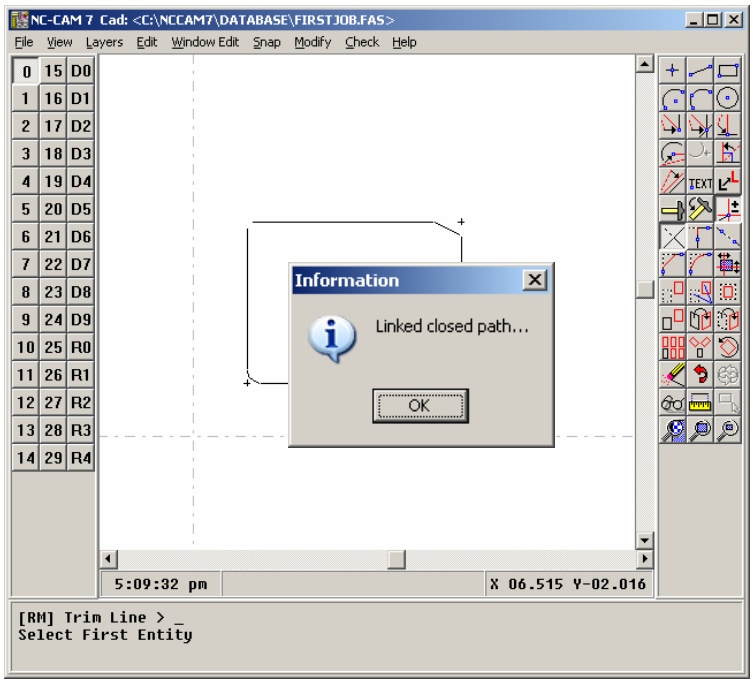
Regardless of how you've built your drawing, it's a good idea to check the outlines before you try to use them to build a rout program. It's not possible to create a good rout program unless the lines you're planning to rout are correctly trimmed (or snapped) to each other. The main-menu function to check the profile is *Chec**k**/**P**ath*.



When you select *Check/Path*, you'll see this display. The procedure used here is the same one you'll use when you're creating a rout program.

The general idea is that you first select the line where you want the test to begin. Then, you move your mouse to the end of the line you want the test to go towards. When you've got the mouse close to the proper line end, you simultaneously press, *hold*, and then release *both* the left and right mouse buttons. The profile-follower will chase the part edge, checking each intersection along the way. The profile-follower will stop when the path is completed, where there is a gap, or where there is more than one path to follow.

When test path can follow the profile, it's ready for routing.



When you press, hold, and release both buttons, the profile-follower should completely follow the outline you've made, and this message should appear. If it does, congratulations are in order: You've successfully made your first board profile! Click OK to exit the Test Path function.

On the other hand, many first-time users do not succeed at building a perfect profile on the first try. The two most common problems are duplicated lines and gaps.

Fix multi-paths with erase.

Duplicated lines may be difficult to detect, because if one line is directly on top of another, there's no way to see the problem on the screen. When you get the error message "Stopped due to multiple paths" but you cannot see more than two lines meeting at the trouble spot, you've probably got overlapping lines there. The easiest way to eliminate overlapping lines is to use **WE** *Window Erase* to erase one of the lines, then **RD** *View Refresh* to see what might have been underneath the line you've erased. The erased line may be easily replaced by using the **NP** *Near Point* snap function.

Fix gaps with Join.

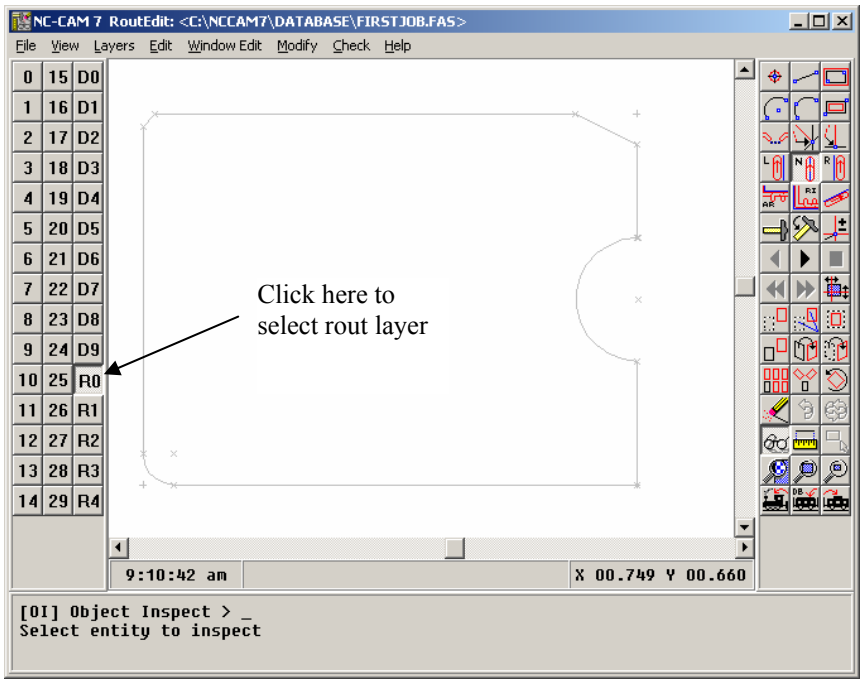
Gaps are most easily cured with the **JO** *Join Lines* function. The Join Lines function prompts you to select two intersecting entities, and it then trims the two so that they meet perfectly. Once you've corrected any problems in your profile, test it again with the *Check/Path* function. *Do not proceed* to the next section until your drawing passes the test.

Your first rout program

With your board profile complete, it's time to call up the Rout module. Select the first rout layer, "R0" by clicking the R0 layer button. This automatically selects the rout module, and the routing tools will be displayed on the toolbar.

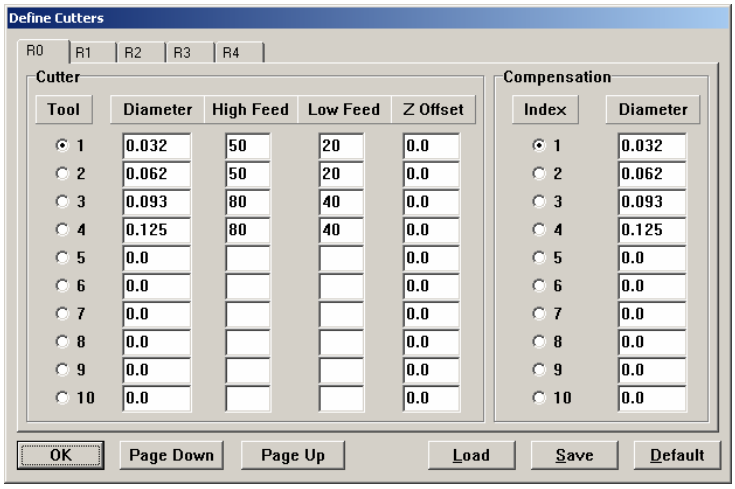
In the rout module, you'll be able to create, view, and modify rout paths using the specialized routing toolbar. For this exercise, we're going to quickly create a simple external rout of this board.

To build a rout program first you select a routing layer.



Cutter selection

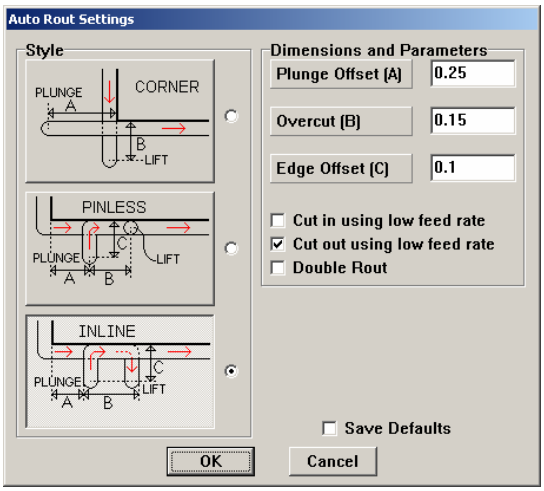
To define cutter diameters and select cutters, select ***Modify|Cutter Config*** from the main menu.



Because NC-CAM 7 allows you to work on as many as five rout programs at once, there are also five tool tables. When you select ***Modify|Cutter Config***, you must then select which of the five tables you wish to use. In this case, choose the tab for layer R0. For this rout, you'll want to select tool number three, which defaults to an 0.093 cutter size. Note that when you press the radio button for tool three, the compensation automatically switches to index number three. Click OK to proceed with this cutter.

Auto Rout

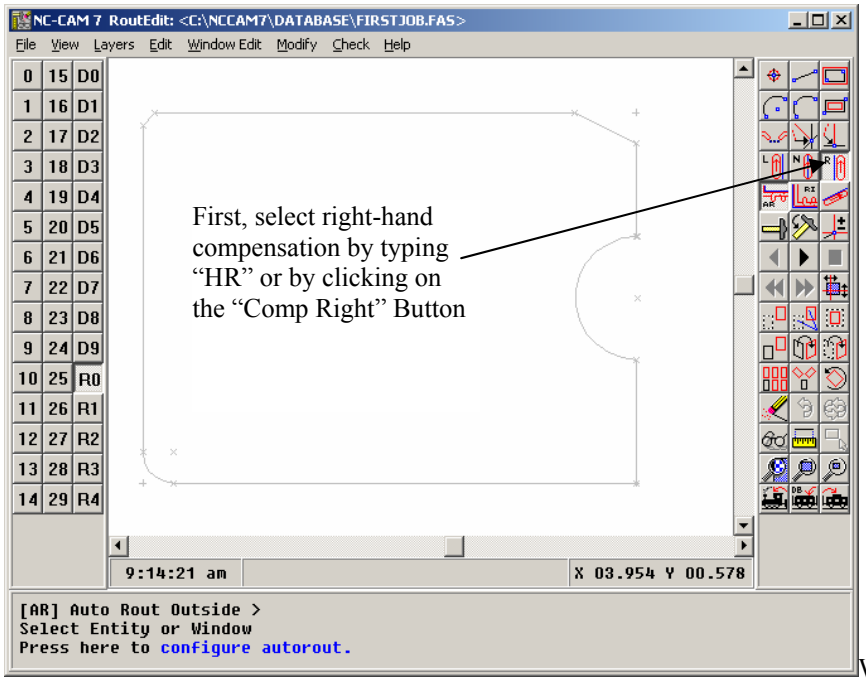
To rout the outline you've created, select ***Edit|Auto Rout Outside***. The Auto Rout function offers several different cut-in/cut-out styles, which you may choose by selecting ***Modify|Auto Rout Config***, or by clicking on the words "configure autorout" in the Command Window.



Auto Rout offers three cut-in/cut-out styles; Corner, Inline and Pinless

Choose the Inline cut-in/cut-out style by pressing the large button at the lower-left corner of the dialog. Then press **OK** to proceed.

Choosing compensation

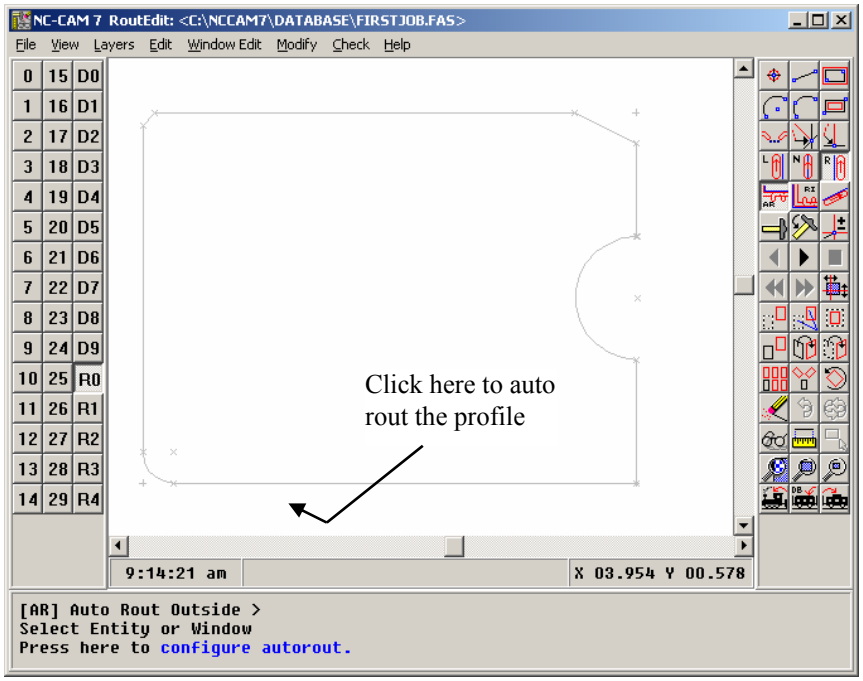


For most routing, you'll want to select the right-hand compensation

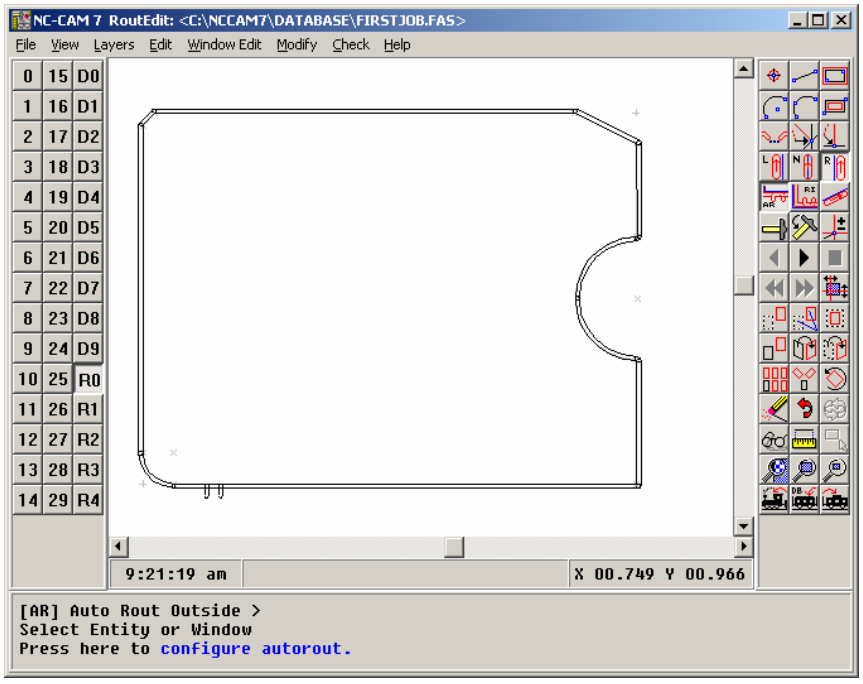
When you're creating external routs, you'll ordinarily want to use the standard "comp right, counter-clockwise" technique. To use this approach, you'll need to select **HR** Comp Right before you create the rout. (It is also possible to make NC-CAM perform external routs using left-hand compensation and a counter-clockwise approach. This is useful in certain cases where internal pinning is supplied, and two counter-clockwise passes can be used for high accuracy.)

For now, select the right-hand compensation as shown.

AutoRout Outside automatically picks the direction, based on your choice of right-hand or left-hand compensation.



Once you've selected the Auto Rout Outside function and the right-hand compensation button is depressed, routing the profile is as simple as clicking your mouse near the line where you want the plunge and lift to occur, and then pressing enter to confirm the selected line. When you have multiple items to rout, you must select them in the sequence in which you want them routed on the machine.

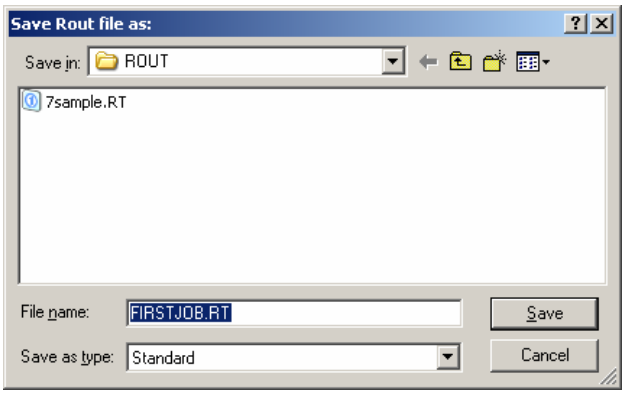


When you click on the line, NC-CAM's Rout module analyzes the board outline and creates the routing information automatically. The cutter path is shown graphically atop the CAD outline, exactly simulating a CNC rout controller.

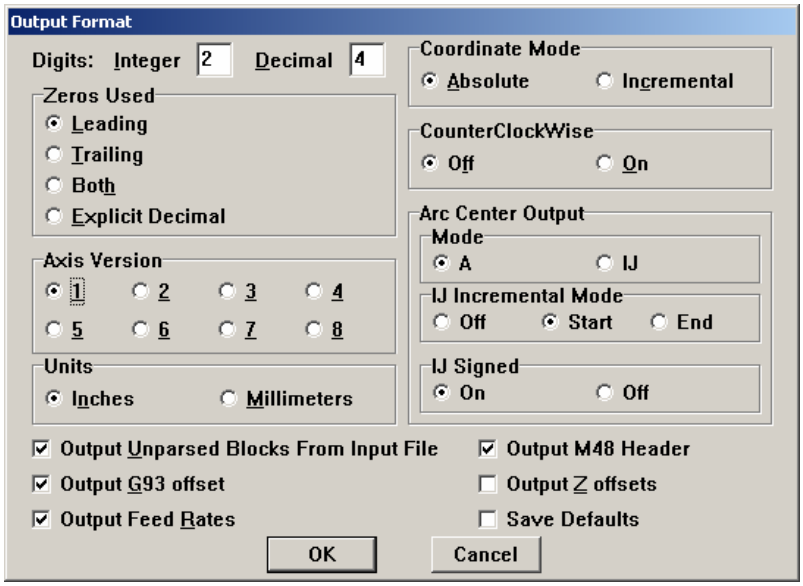
Select ***File|Save*** to save the job as an NC-CAM 7 database, including both the CAD and Rout layers.

Rout program output

When you've finished creating a rout in NC-CAM, you will undoubtedly want to output it in a CNC format so you can run it on a routing machine. To do this, select ***File|Make Rout Tape***.



After you've chosen the directory and filename for your rout program, press **ENTER** or click Save. The format dialog appears. This is to let you select the format for your router program. Choose the format you'd like, and press **OK**.



When you press OK, the rout layer will be saved as a CNC router program. Congratulations! You have now completed your first rout program, using the CAD and Rout modules. Your output rout program is a regular ASCII DOS file, ready to send to a CNC routing machine.

C H A P T E R 4

Using the DrillEdit module

This chapter provides a step-by-step introduction to NC-CAM's "DrillEdit" drill programming capabilities. Through the course of the tutorial, you will learn how to:

- import customer data files
- take measurements in the drill pattern
- sort tool tables
- edit holes individually and in windows
- check for and correct spacing violations
- step-and-repeat a board into a panel
- optimize and output a finished drill program

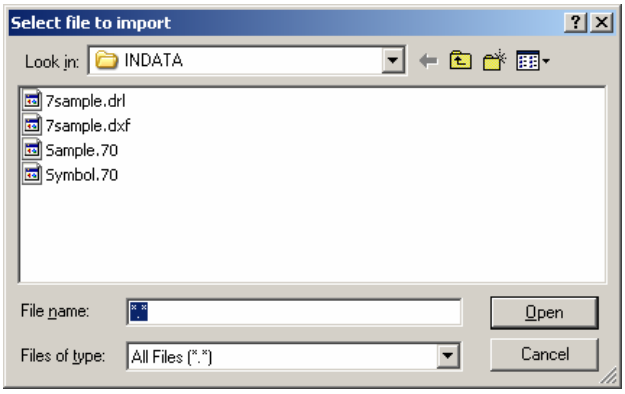
After reading this chapter, you should be able to use NC-CAM's DrillEdit module to process drill programs. You should also have an understanding of how DrillEdit handles tool tables, inch

Your first drill program

To select the DrillEdit module, press the "D0" button on the layer menu (on the left side of the screen). This will make layer D0 the selected layer, and the DrillEdit module's toolbar and main menu will be displayed. Why would a drill editor have "layers?" Because you will sometimes have to work with more than one drill job at a time. Consider the case of a first-drill (pre-plating) and second-drill (post-plating) process. With NC-CAM, you can bring in the data for *both* of these drill processes at once, by using *layer D0* for the first-drill data and *layer D1* for the second-drill data. You can use additional layers for blind and buried vias, etcetera.

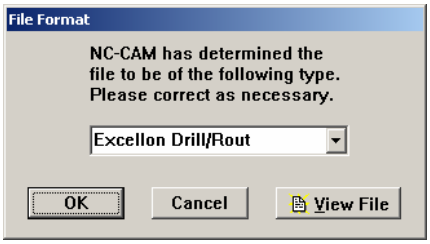
The advantage here is that you can make sure the several drill programs match perfectly, long before you ever drill them in production. You can check to see that the two programs don't conflict. You can use the *same* step-and-repeat data for both files. In short, you can use NC-CAM to be absolutely certain that the programs are correct: And being absolutely certain is what NC-CAM is all about.

Using the DrillEdit module is not at all like hand-editing a drill program in a text editor. Apart from the obvious difference that DrillEdit shows you the drill program as graphics instead of text, DrillEdit is a genuine CAD program, with a CAD database. Begin your exploration by selecting ***File/Import***. This will give you a look at the *file import* dialog box.



This is the menu you'll be using to read all of your drill files into DrillEdit. We'll get into the details of how to use the file import dialog in later chapters. For now, just select the file `SAMPLE.70`, either by typing the name in, or by double-clicking on the name with your left mouse button.

Any time you select ***File/Import*** in any of NC-CAM's three modules, NC-CAM will try to automatically figure out what type of file you've selected. NC-CAM is a very flexible program. It allows you to read in data from a variety of sources, not just CNC drill files. When you select ***File/Import***, NC-CAM's import routines each take a quick look at the file to see if they can tell what it is. They "bid" for the opportunity to decode the file. Whichever routine best "understands" the file is given the job of reading the file in. What this means is that you don't have to tell NC-CAM what kind of file you're trying to read. You just click on the file name, and NC-CAM figures out how to read the file.



The File Format dialog tells you which data format NC-CAM thinks the file is. If you want to look at the file in text mode to verify the format, press the **VIEW FILE** button. In this case, the SAMPLE.70 file is in the Excellon format, so just press **OK**.

Excellon-to-Drill dialog

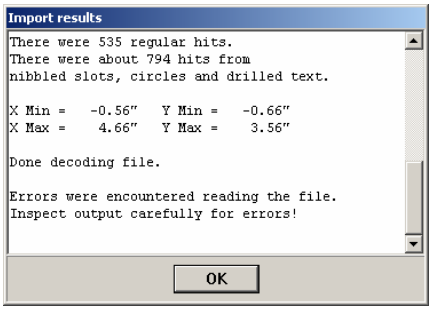
After you press **OK** on the File Format dialog, the DrillEdit module will bring up the Excellon-to-Drill dialog, as shown below.

The settings here are just like the settings on a drilling machine, and they have the same effect. Before you choose the settings for this file, you might want to take a look at it. To do this, you would select *View File*, either by typing **V** on your keyboard, or by pressing the button with your mouse.

Note that no matter what settings you choose in the Excellon-to-Drill dialog, if the drill program you're reading contains format control commands in an M48 header, the commands in the drill program will override your menu settings. (This is also exactly the way an actual drilling machine would behave). Press **ENTER** or Click OK to read the drill file.

Import Results Dialog

This window shows information on the file, including the number of drill hits and size of the part. Click OK to proceed.



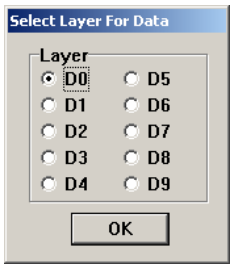
Is this the main part?

The next dialog you'll see asks the question, “Is this the main part?” Why would the DrillEdit module want to know if the file you're about to read in contains the “main part?” For several reasons. First, understand that DrillEdit allows you to read *several* files into the same job. These may include a part, some coupons, panel tooling holes, and so on. So, DrillEdit needs to know which file contains the “main part” so that it knows which file to pay the most attention to.

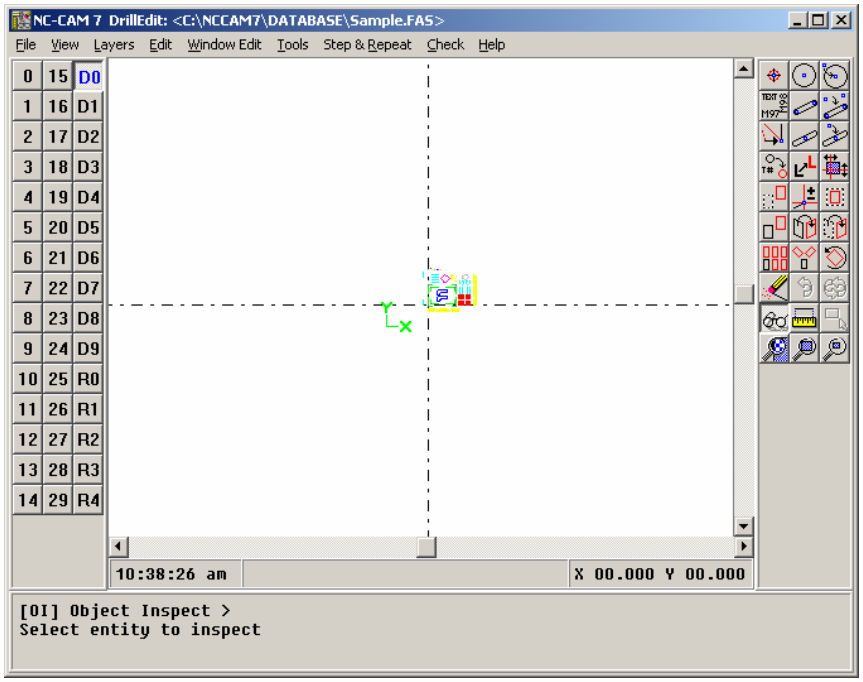
When you're importing the main part, DrillEdit automatically positions it in the workspace with no offset, so that the coordinates you'll see on-screen are the same as the coordinates in the original file (you can always change these later). DrillEdit also automatically accepts any G93 offset (if there are any) which the "main part" file contains. Any time you're working with a simple job having only one file, you'll answer "Yes" to this question. For now, press **Y** on your keyboard, or press the *Yes* button with your mouse.

Select Layer for Data Dialog

This window allows you to place the drill data on any one of 10 different drill layers. The selection defaults to Layer D0. This will work for our purposes, so simply click on OK.



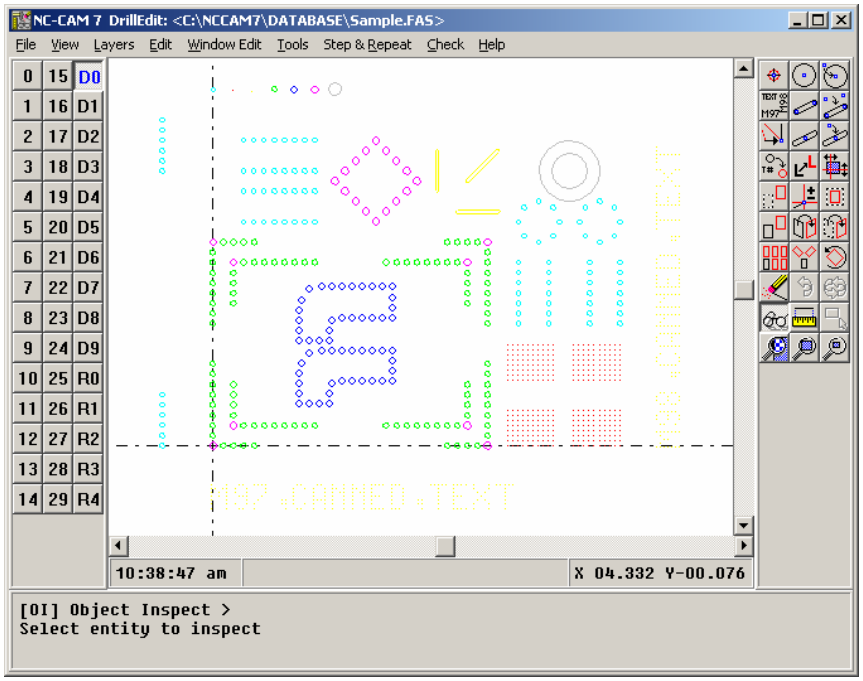
*Before using **View/Zoom All**.*



After you click OK on the Select Layer window, you'll see the drill data appear on the screen. The location of the data relative to the zero-zero lines is determined by the X:Y values of the coordinates in the drill file.

At this point, the screen is "zoomed out" so that the visible area is about 46 inches (1.2 meters) wide. The drill data will look very small. To get a better look at this job, first select ***View/Zoom All***.

*after using **View/Zoom All**.*



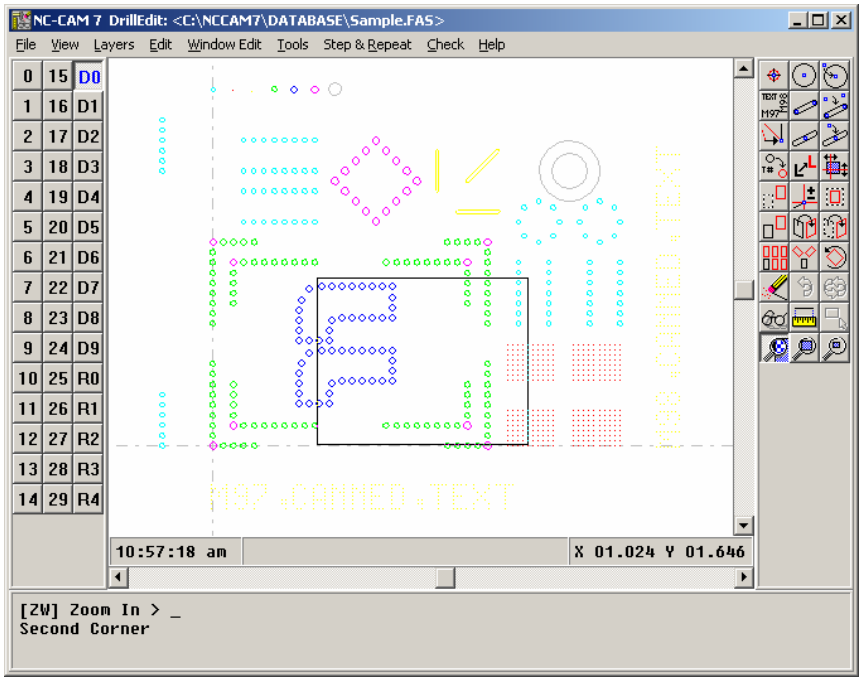
The Zoom All function will automatically resize the screen to center your view, and magnify it as well. (Zoom All is a function you'll use frequently.)

Now you're actually looking at the drill data taken from the SAMPLE.70 file. Note that we don't say, "you're looking at the data *in* the file," because you are not. When you read a drill program into the DrillEdit module of NC-CAM, it isn't a drill program any more. It is a 2D CAD database of drill holes with a tool table. You can inspect any hole by just clicking your left mouse button on it. You can make needed changes by clicking on holes or windows of holes. You can control the sequence in which holes are drilled by placing windows around them. You can create nibbled slots, add holes, change hole sizes, check for double hits, make coupons... in short you can do almost anything you'd ever want to do with drill data, and you can do it quickly, all because your drill data is no longer a text file of XY coordinates, it's in a special-purpose 2D CAD database.

Zoom and Pan

To zoom in, first type ZW, then click+drag the area you want to have fill the workspace

In the next couple of pages, you're going to need to get a closer look at part of this drill pattern. To zoom in, you may type **ZW**, click the left "magnifying glass" icon, or select ***View|Zoom In*** from the main menu.



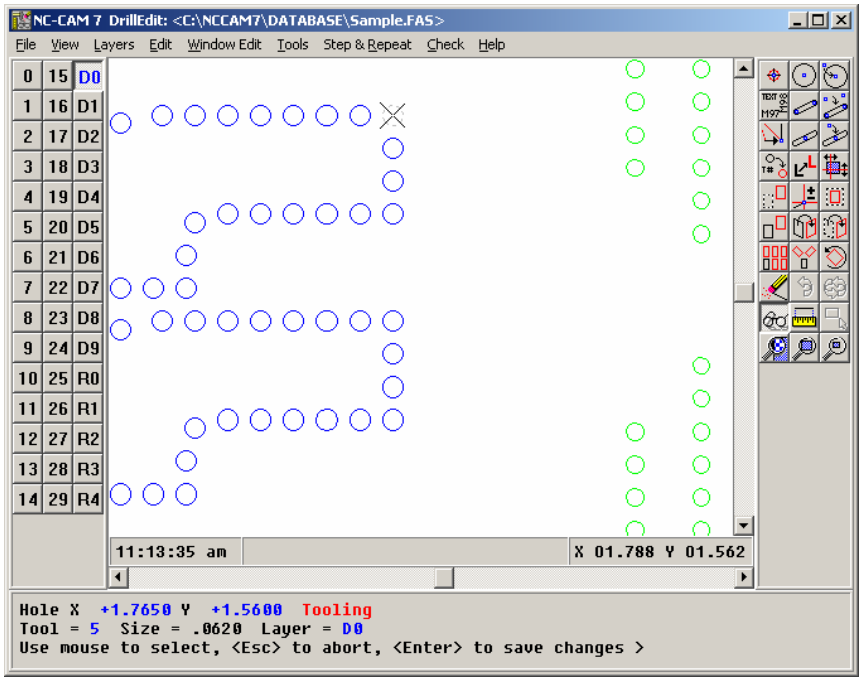
With Zoom Window, the area you select will be magnified to fill the workspace. Note that when you're "zoomed in," you can drag (pan) the screen around by holding down the **ALT** key while you click+drag the workspace with the mouse.

Changing units

If you read the introductory text on page 10, you know that NC-CAM can be easily switched between inch and metric units. How easily? Try selecting ***View/Units/Millimeters***, then click on another hole to inspect it. You'll see that the units for the drill diameter and the hole location are all now in millimeters. This chapter is written assuming you're using the "inches" display mode, so it would be best if you reselect ***View/Units/Inches***, for now.

Inspecting holes

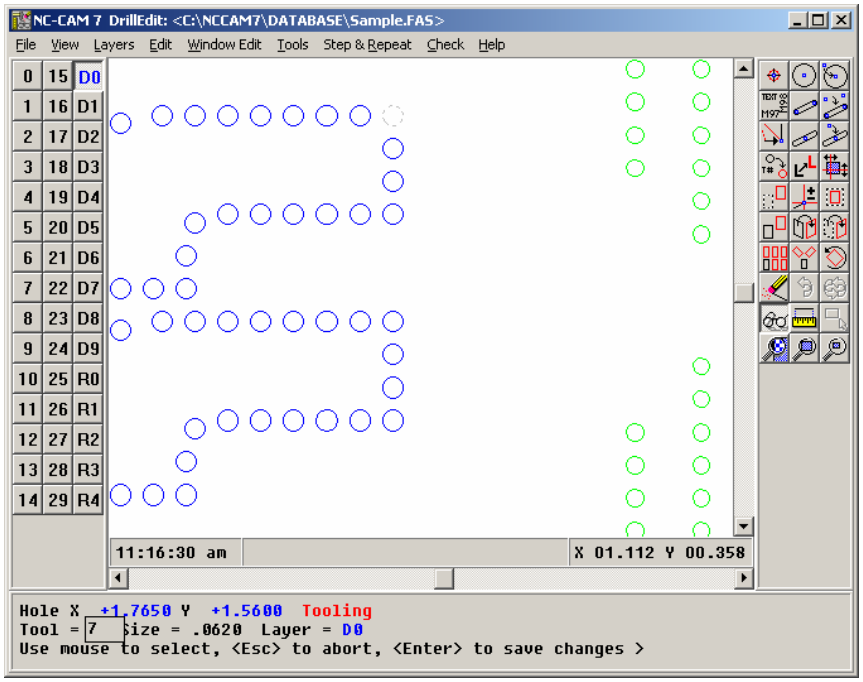
The (OI) Object Inspect function lets you inspect and edit



To inspect a hole, place the point of the arrow cursor close to the hole, and click your left mouse button. You'll see an "X" shape snap to the hole. This indicates which exact hole you're currently locked on to. You'll also notice that the command window shows the X and Y ordinates of the hole, its tool number and diameter, and which Layer the hole is located on (D0). The X and Y ordinates, the tool number, and drill layer are displayed in blue, because they are editable.

To alter a hole's location or tool number, you just use your **TAB** key to select the appropriate blue number in the command window, type in the new value, and then press **ENTER** twice.

To change a hole's tool, use OI Object Inspect, click on the hole, and type the new tool number into the command windows



In this case, press the **TAB** key three times, to highlight the tool number. Type in **7** for the new tool number, and press **ENTER** only once. At this point, you'll see the

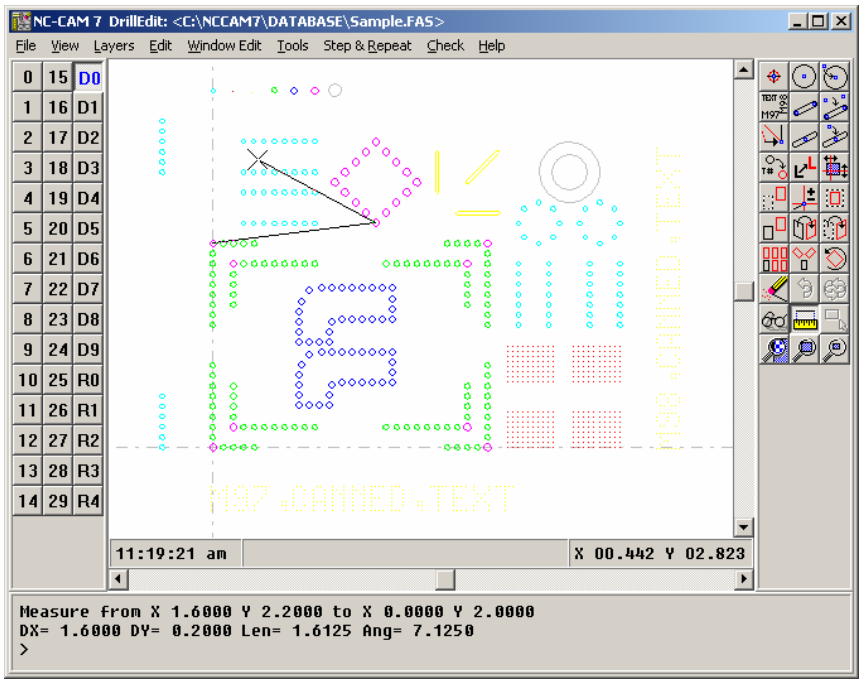
hole change to the diameter of the tool number seven, 0.126", and the hole will change to red in color. If you want the hole to be permanently changed to the new tool number, you may press **ENTER** a second time. If you want to return the hole to its original tool number, you may press **ESC**. In this case, press Enter to change this hole to tool number seven.

Measuring

In preparing a drill job for production, many times you'll need to measure the distance between two holes. First, though, select ZA (zoom all) to show all of the drill data in this job.

To measure, first select the [ME] Measure icon on the toolbar. Next, click on the hole you wish to measure *from*. You'll see a rubber-banded line attached to your cursor. Next, select the hole you wish to measure *to*. In the command window, you'll see the X:Y coordinates of both of the holes, as well as three distances: The "Manhattan" X and Y distances, as well as the straight-line distance between the two holes.

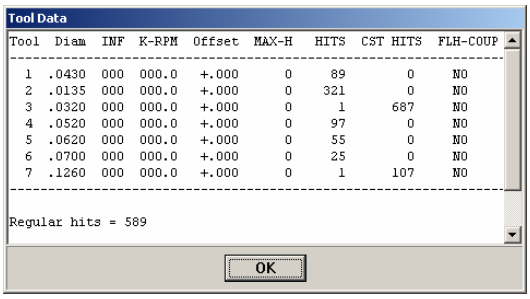
The ME measure function measures distance, X:Y difference and angles



You may now measure from the first hole to any other hole by just clicking on the other holes. All measurements will be made from the first hole you selected. To get out of this measurement mode, you may either press your right mouse button or the **Esc** key.

Tool tables

As we said earlier, this 2D drill database contains holes and a tool table. So what does the tool table contain? To find out, select ***Tools/Show Tool Table***, and you'll see this display:



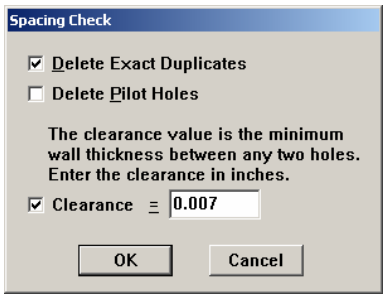
As you can see, the tool table lists the tools by number, with the diameters and feed and speed parameters, if any, from the input file's M48 header. The hit counts for regular holes and nibbled circle/slot/drilled text holes are listed separately. The last column is headed with the abbreviation "FLH-COUP," which stands for first-hit/last-hit coupon. Click OK to close the dialog.

As is often the case with the drill files you'll receive from your customers, the tool sizes are not in any particular order. You may sort the tool table by selecting ***Tools|Sort Small Tool First*** from the main menu. When you do, the screen will repaint, and you may notice that the hole colors have changed to reflect the new tool assignments. Select ***Tools|Show Tool Table*** a second time, and you'll see that the tools are now in a smallest-to-largest sequence.

Spacing check

With the pervasive use of CAD printed circuit board design software (especially by inexperienced operators), much of the drill data you'll receive will have multiple hits at the same location. It is also common to have double hits at *almost* the same position. These "near double hits" result in a great deal of tool breakage on the drilling machines, and a lot of scrapped panels. DrillEdit's spacing check function allows you to absolutely prevent these problems *before* they cause scrap. Once the problems are identified, you can use Point Edit to correct them. (Remember that we changed a hole from tool number five to tool number seven a few pages ago. That's going to cause a spacing violation in this job).

To use the spacing check function, select ***File/Spacing Check*** from the main menu.

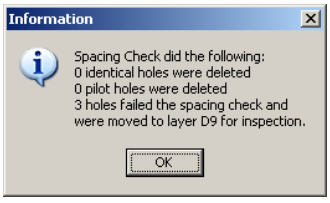


You'll notice some options in the Spacing Check dialog. First, there's a checkbox for ***Delete Exact Duplicates***. This choice does what it says: If any hits in the job are duplicated exactly (same tool, same exact coordinate). they will be deleted automatically.

The ***Delete Pilot Holes*** choice deletes small hole(s) when they are found at exactly the same coordinate as larger holes.

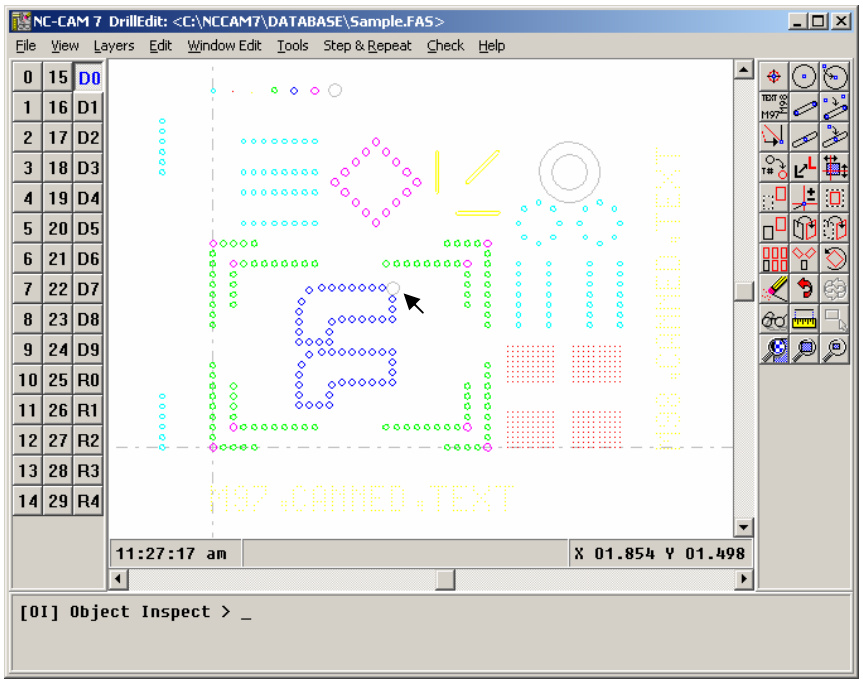
Finally, the ***Clearance*** choice actually compares every hole to every other hole, to verify that there no two hole edges are closer than the distance you've specified. In this case, set the clearance value to 0.007 inch, make sure the ***Clearance*** checkbox is checked, and then press ***OK***.

Spacing check will very quickly perform the tests you requested. When the tests are done, you'll see the results in a dialog.



In this case, the Spacing Check function has detected three holes which violated the specified minimum hole wall thickness of 0.007 inch. Holes violating spacing check

are not automatically changed in any way (because NC-CAM can't just guess *how* to fix the violation). Instead, they are just moved to layer D9, so that you can correct the problem after consulting with your customer or his documentation.



The cursor is pointing to the three holes which are closer than 0.007 inch, as shown by Spacing Check. To correct this violation, repeat the steps on the previous pages, using Object Inspect to find the one large hole, and change it to tool number five.

Window editing

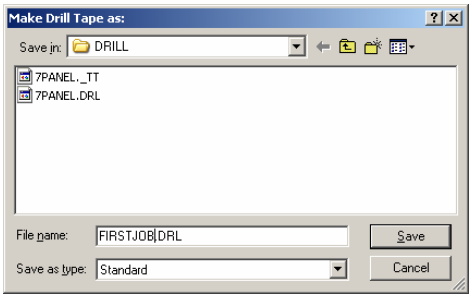
To change the three holes back to layer one, select [LO] Layer Operation from the toolbar. The Layer Operation choice may be used to move holes from one layer to another.

```
[WM] Window Move > _
Select Entity or Window
Target Layer: D0
```

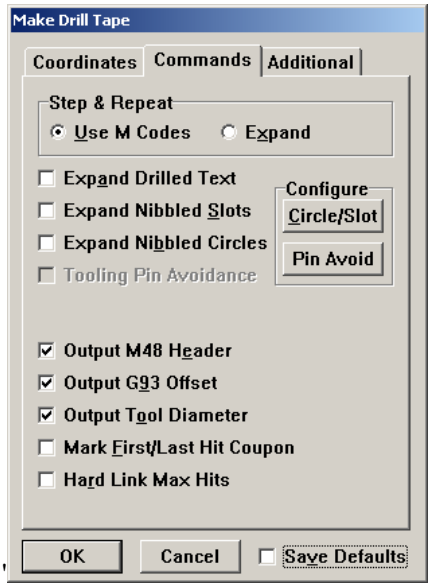
As you'll see in the command window, you may select any drill layer as the "target" of the layer operation. In this case, you want to move the three holes to layer D0, so just leave the default setting alone. Next, drag a window around the three holes, and they'll be moved back to layer D0. Note that it doesn't matter if you also include other holes in the window: They're already on layer D0, so nothing will happen to them if they're also in the window when you do this layer operation.

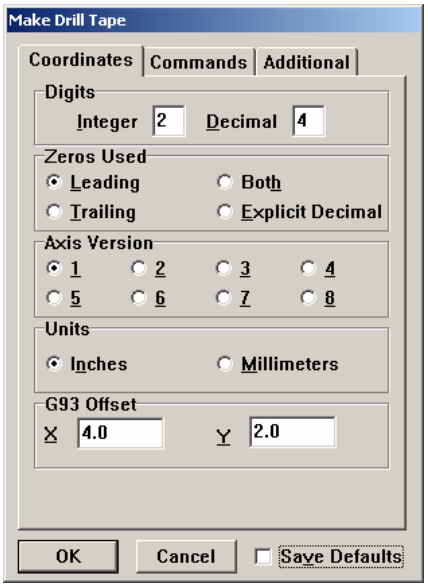
Making an output drill program

To create an output drill program, begin by selecting ***File/Make Drill Tape***. You'll first be asked to select a name for your new output drill program.



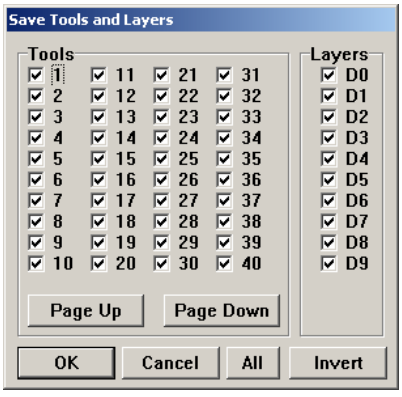
For the moment, you may name the job FIRSTJOB.DRL. Type the file name in and either press Enter or click Save. Next you'll see the imposing-looking *Make Drill Tape* menu.





'The *Make Drill Tape* menu offers control over most aspects of the CNC drill program you'll output. There are many choices available (these are discussed in *Chapter 7, "DrillEdit Command Reference"),* but for the moment, the default values will work fine. Just press *OK*.

After you press *OK*, the Active Control Menu pops up automatically. This is to remind you that only the *active* tools, layers, and types will be output to your drill program. This is where you may control the output of multiple files for special purposes. Make sure all of the checkboxes are turned on, and press *OK*.



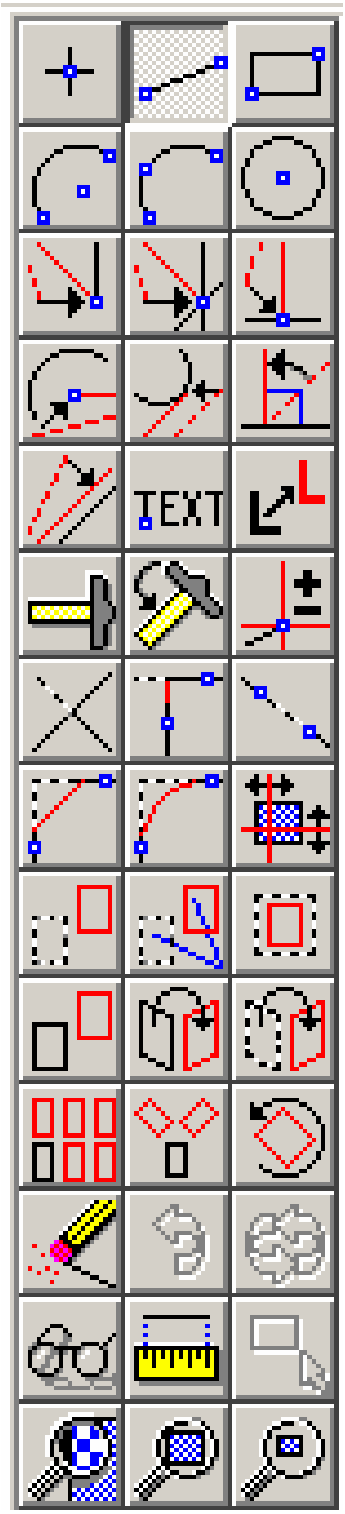
At this point, there's a lot to watch. You'll see NC-CAM go through the steps of optimizing and outputting your file.

When the output is completed, you'll be prompted with a menu asking if you want to print out the "new" tool list. What would be "new" about the tool list? Possibly quite a bit. As you'll learn in Chapter 7, it is possible for DrillEdit to automatically enforce maximum hit counts, and automatically average the number of hits each tool is actually used for. It's also possible to output the nibbled slots and circles as individual hits, instead of G84 and G85 commands. There can be some tremendous quality advantages in this approach. In any case, it's quite possible that the output file will have a different tool list than you had in the editor, and that's why you're given the option of printing out the new list.

C H A P T E R 5

CAD command reference

This chapter provides a command-by-command reference to the NC-CAM 7 CAD functions.



The functions, in the order presented on the CAD button menu are:

PO *Create Point*
LI *Create Line*
RE *Create Rectangle*

A2 *Create 2-Point Arc*
A3 *Create 3-Point Arc*
C2 *Create Circle*

SC *Snap Closest Endpoint*
SI *Snap Intersection*
SM *Snap Midpoint*

SN *Snap Arc-Circle Center*
ST *Snap Tangent*
SP *Snap Perpendicular*

SL *Snap Parallel*

LO *Layer Operation*

OR *Turn on Ortho Snap*
OA *Ortho Angle*
MR *Relative Mode*

RM *Trim Line*
JO *Join Lines*
OB *Object Break*

CH *Chamfer Lines*
FL *Fillet Lines*
DO *Drawing Origin*

WM *Window Move*
WS *Window Stretch*
SW *Window Scale*

WC *Window Copy*
WI *Window Mirror*
WF *Window Flip*

MC *Matrix Copy*
RC *Radial Copy*
RO *Rotate*

WE *Window Erase*
OO *Undo*
UU *Redo*

OI *Object Inspect*
ME *Measure*
SA *Select All*

ZW *Zoom In*
ZA *Zoom All*
ZB *Zoom Out*

[PO] Create Point

The *Make Point* function allows you to make a reference point, which is displayed in your drawing as a small plus "+" sign. The points you make with **PO** are primarily useful for tagging locations you will need later, in the course of constructing your drawing.

One good example of a case where you might use a point would be to tag the zero location of your drawing when you want to use **RO** *Rotate* to rotate the entire drawing to work on an off-axis section. This happens often in the flex circuit business, where the drawings will often have entire sections which are dimensioned off-axis. In this case, having a point at the zero location gives you an easy reference to rotate the drawing around.

One case where you might encounter a lot of points is when you import a Gerber photoplot file. Because CAD's import filter has no provision for entering an aperture list for the Gerber data, any "D03" flash commands in the Gerber file will be converted to points in the CAD database.

[LI] Create Line

The *Line* function is undoubtedly the one you'll use most often in CAD. In CAD, lines are actually line segments, with no thickness, defined by their two endpoints. When you create a line, you'll be prompted to enter the first endpoint. After you do, you'll notice a "rubber-banded" line connecting the first endpoint you made to the cursor. When you define the second endpoint, the rubber-banded line will change to a line entity in the CAD database.

There's one key modifier you'll often use when making lines; the *orthogonal mode*. The button for the *orthogonal mode* is a drafting "T-square" symbol, for good reason. The *orthogonal mode* forces your line to be square to the axes, or to the angle you've entered with the **OA** *Ortho Angle* function. The *ortho mode* is available instantly without toggling on the **OA** button, though: Any time you want the *ortho mode* turned on for just one mouse click, it's easier to just press and hold the **CTRL** key on your keyboard. This works both ways, i.e., if the *ortho mode* button is turned on, pressing **CTRL** will temporarily release the *ortho mode*.

[RE] Create Rectangle

The **RE** *Rectangle* function allows you to create an axis-aligned rectangle by placing its two diagonally-opposite corners. Once you've created a rectangle, it isn't a rectangle any longer, though. The **RE** *Rectangle* function actually creates four independent line entities in the shape of a rectangle. The only respect in which a rectangle is still treated as a rectangle after it's made is by the **OO** *Undo* and **UU** *Redo* functions.

As with the **LI** *Line* function, after you enter the first coordinate of a rectangle, you'll see a rubber-banded rectangle attached to your cursor.

[A2] Create 2-Point Arc

The **A2** *Two-Point Arc* function creates an arc, by defining the arc's center point, then a startpoint, and finally its endpoint. The centerpoint is like the place you'd anchor a compass if you were drawing an arc on a drafting table. Once the centerpoint is defined, you'll get a rubber-banded straight line attached to your cursor, until you define the start point.

A *two-point arc*'s start point defines both the angle from the center to the start, as well as fixing the arc's radius. After the start point is defined, the rubber-banded arc you'll see attached to your cursor has a radius that's fixed at the distance from the center point. When you're defining the end point, all you're actually setting is the ending angle of the arc, even if you use a snap function to enter it.

[A3] Create 3-Point Arc

The **A3** *Three-Point Arc* function allows you to create an arc by defining one endpoint, then a point along the arc, and finally the other endpoint. This means that the radius is not actually set until the third and final point has been entered. The resulting rubber-band effect is quite unusual, as the arc's center and radius change fluidly as you move the mouse while defining the second end point.

[C2] Create Circle

The **C2** *Create Circle* command will add a circle to the CAD database, by defining the circles' center point, and secondly, a point on the edge of the circle. Once you define a center point, you'll get a rubber-banded circle that is attached to your cursor, until you define a point on the circle.

[SC] Snap Closest Endpoint

The **SC** *Snap Closest* function is used when you're entering a coordinate and you want it to be exactly the same as an existing coordinate in the drawing. To use snap closest, you simply select the function, then move the cursor close to the point you wish to snap to, and press your left mouse button.

There's a similar function that's actually used more often than **SC** by experienced CAD users, and that is the **NP** *Near Point* function. Unlike the **SC** function, **NP** does not require that you click your left mouse button. Instead, it snaps immediately when you enter **NP** on the keyboard. This immediate snap, and the fact that you don't have to move your mouse over to the menu and then back to the desired point, save a considerable amount of time. Obviously, there's no point in having a menu button for **NP**, because you'd have to have your mouse on the menu and not near the snap point in order to press the button! An alternate way to select **NP** is to click the middle mouse button, if you have a three-button mouse.

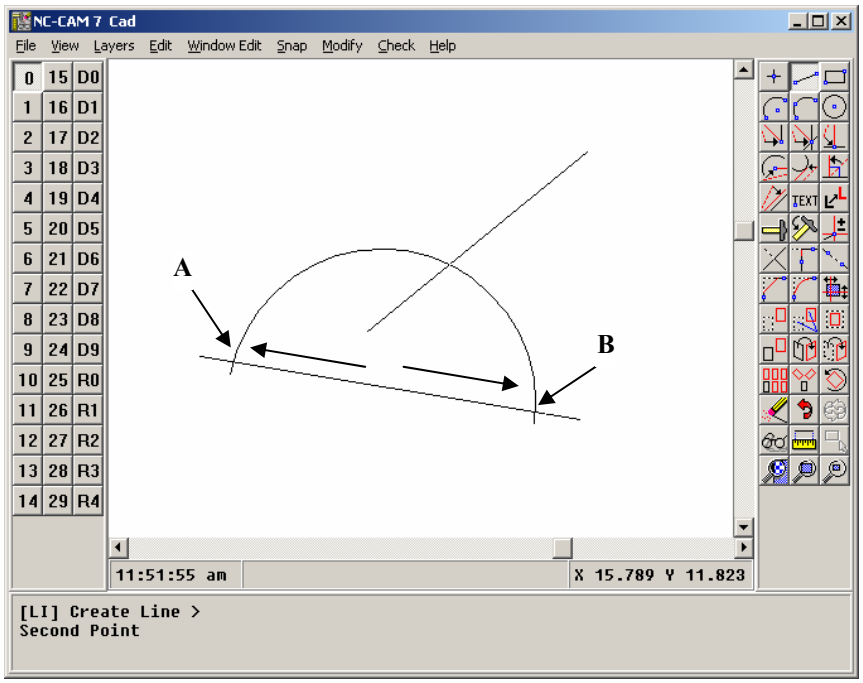
There's one aspect of **SC** and **NP** that's not obvious to first-time users: There are more snappable points than you can ordinarily see on the screen. This is because both arcs and circles have construction points at their centers. These points become visible only when you turn on the ***V**iew*|***C**onst Points* function, but they may always be used for snapping.

If you have a 3-button mouse on your computer, the middle button works as an **NP** snap. Simply move the cursor near a point and press the middle mouse button once to snap to the point.

[SI] Snap Intersection

The **SI** *Snap Intersection* function is used when you're entering a coordinate, and you want it to be at precisely the point where two existing entities intersect. The process of using snap intersection, as described in the progress prompt, is to select the two intersecting entities, one at a time. CAD then computes the location of the intersection of the two entities, and enters the needed coordinate at that location.

An interesting nuance of the *Snap Intersection* function occurs when there are actually two places where the entities you've chosen intersect. This is possible any time one or both of the intersecting entities are circular. In this instance, the intersection you'll snap to is the one closest to your mouse when you select the second entity.



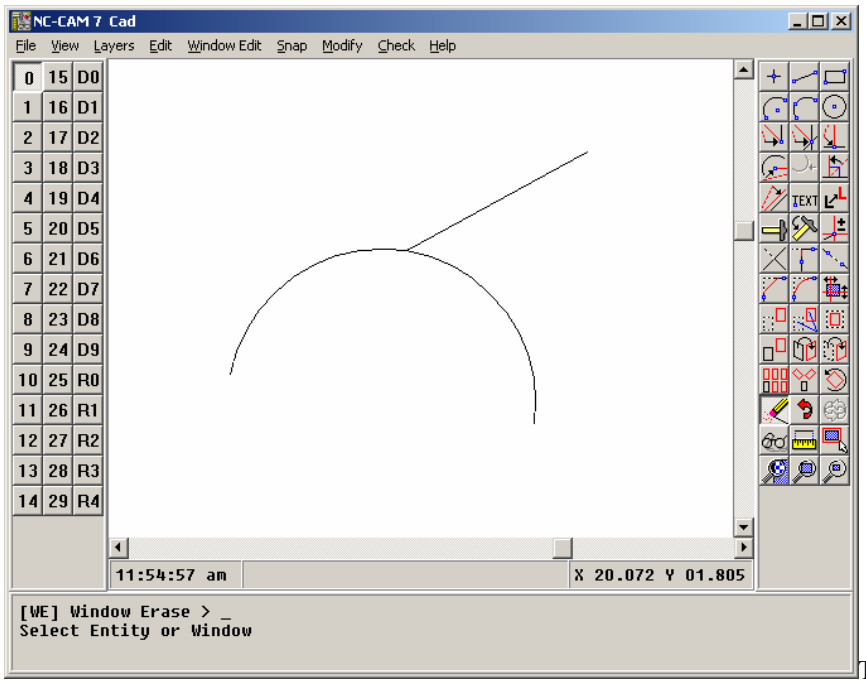
In the example shown above, to snap to intersection "A," you must select the line to the left-hand side, which is closer to the "A" intersection than it is to the "B" intersection.

Note that the snap intersection function does not require that the two entities actually intersect; only that they could intersect if they were long enough to touch each other.

[SM] Snap Midpoint

The **SM** *Snap Midpoint* function snaps to the midpoint of the line or arc you select next. The line midpoint chosen is exactly halfway between its two endpoints.

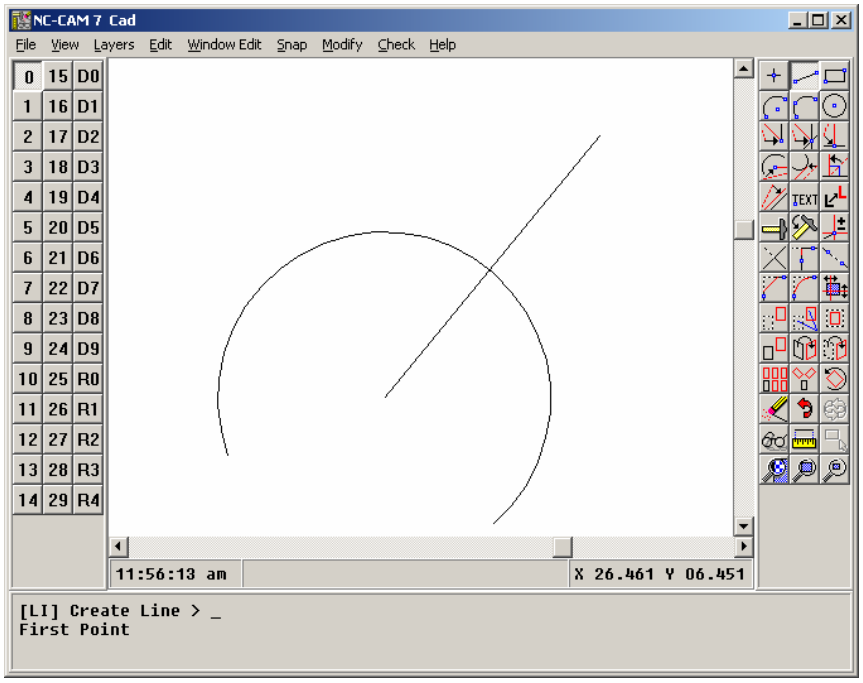
If you select an arc with the *Snap Midpoint* function, the coordinate you'll snap to is the arc's midpoint, not its center.



The midpoint of an arc is a point along the arc, such that the point bisects the arc as shown above.

[SN] Snap Arc-Circle Center

The **SN** *Snap Arc Center* command snaps the coordinate to the center of the arc or circle you choose.



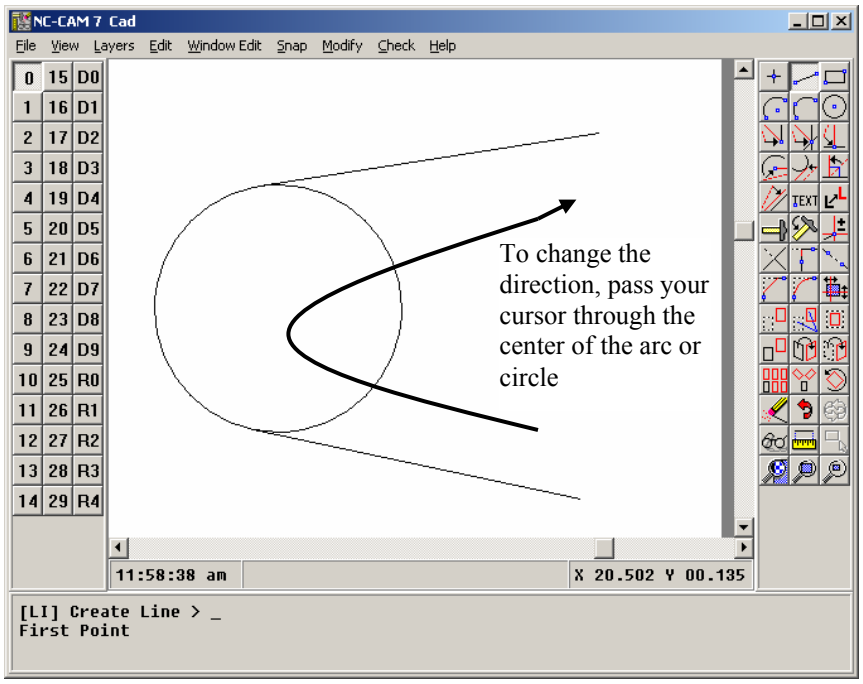
In the illustration above, one line end has been snapped to the center of the arc.

Note that it is not possible to snap to an arc center which falls outside of the 84 inch (2.14 meter) working area of the CAD database.

[ST] Snap Tangent

The **ST** *Snap Tangent* function performs a fairly complex function. First off, **ST** works only when you're creating a line, not an arc or a circle. Secondly, the snap is tricky because the point you're creating with the snap isn't actually defined until you create the second line end. When you select snap tangent and click on an arc or circle, the rubber-banded line you'll see constantly moves to remain tangent to the arc or circle, until you select the second line end.

There is another important issue when you're using snap tangent: There are actually two lines that would be tangent to an arc or circle from any given point outside the arc or circle. In other words, your rubber-banded tangent line may be coming off of the selected circular entity either clockwise or counterclockwise.



If you want to reverse the direction of the snap tangent function, you must pass your cursor through the center of the circular object, and out the other side.

[SP] Snap Perpendicular

The **SP** *Snap Perpendicular* function acts to alter the behavior of the rubber-banded line you're creating. Snap perpendicular also works only when you're creating lines, not arcs or circles.

One surprising behavior of the snap perpendicular function is that you may snap perpendicular to a line, point, circle, or arc. When you're snapping perpendicular to an arc or circle, the line you make will end at the circular object, aligned with its center.

Note that like the snap intersection function, it is not necessary that the point you're trying to create with the snapped end actually touch the entity it's perpendicular to.

[SL] Snap Parallel

The **SL** *Snap Parallel* function, like the **SP** *Snap Perpendicular* function, works only when creating lines, not arcs or circles. The function can be used in two different ways. Both methods require that a line is already present to apply the snap to. Both methods also require that you are currently using **LI** *Create Line*.

The first method for using **SL** *Snap Parallel*, is to pick the **SL** *Snap Parallel* button first. In the user panel, you will see the parallel offset. This is the distance from the selected entity where the line will be placed. Next, pick the line to which you wish to snap. Then you must pick the first point of the line you are creating. The point you pick will determine the first point in the following way: The point forms a perpendicular line to the selected entity. The point on this line at the specified offset is the first point of the line created.

You must now select the second point. The second point is picked in a similar fashion to the first point.

The second method for using **SL** *Snap Parallel*, is to select the first point before using *Snap Parallel*. Using this method, the parallel offset is determined by the first point, not a value specified in the user panel.

After selecting the first point, press the **SL** *Snap Parallel* button. You must then select a line entity to which to apply the snap. After selecting the entity, you must select the second point of the line in the same fashion as in the first method.

[LO] Layer Operation

The **LO** *Layer Operation* function allows you to move entities from one layer to another. This function may be used in the following two ways:

1) When this button is pressed by itself, the function acts like **WE** *Window Erase*, with one difference. The selected entities are not erased, they are merely moved to the layer specified in the user panel. The selection of entities, however, is exactly the same as **WE** *Window Erase*. You can use *Layer Operation* to move either single entities or windowed groups of entities. This window function does not, however, allow you to select multiple entities using the shift-click approach described in the **WM** *Window Move* function: When you select an entity by clicking on it, or a group of entities with a click-drag window, they are moved immediately.

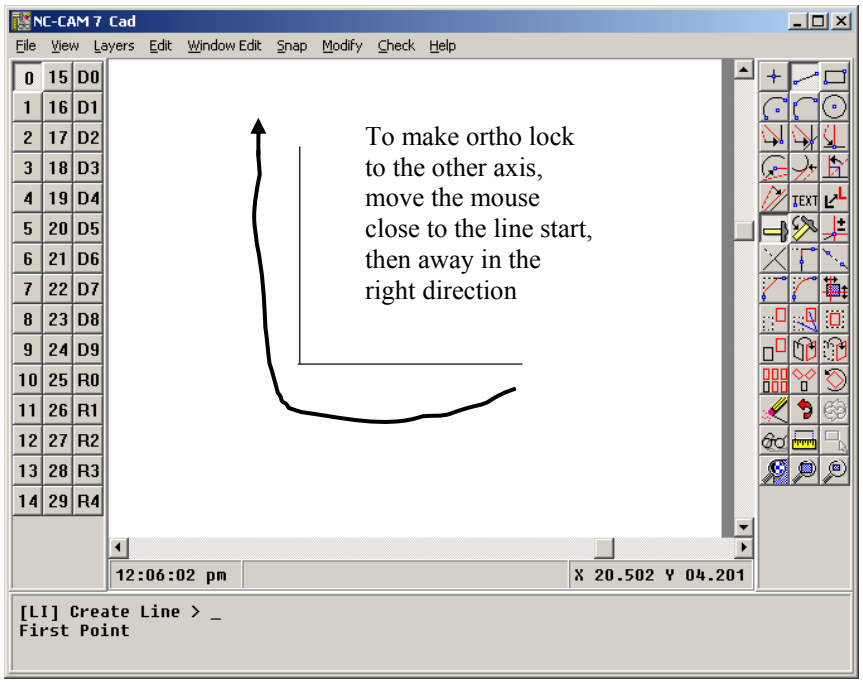
2) The *Layer Operation* function may be applied to any window edit function as well. To do so, first pick the operation you wish to perform. Then select *Layer Operation*. The result of the operation, instead of landing on the construction layer, will be placed on the layer selected in the user panel instead. Notice that doing a *Window Move* still requires you to select the offset when using this function.

[OR] Turn on Ortho Snap

OR *Ortho Mode* applies a constraint to the cursor motion once a point has been entered. The constraint acts to force the line you're creating to be square to the axes, or to the angle you've entered with the **OA** *Ortho Angle* function described on the following page.

When it's turned on, the *Ortho Mode* will force newly-created lines square to the ortho angle, even if you snap the second end of a line you're creating to some other object.

In one respect, ortho mode acts a bit like the **ST** *Snap Tangent* function described earlier: Once it's locked on to an axis, say the "X" axis for example, it stays locked on to that axis no matter how far you move your mouse along the other axis.

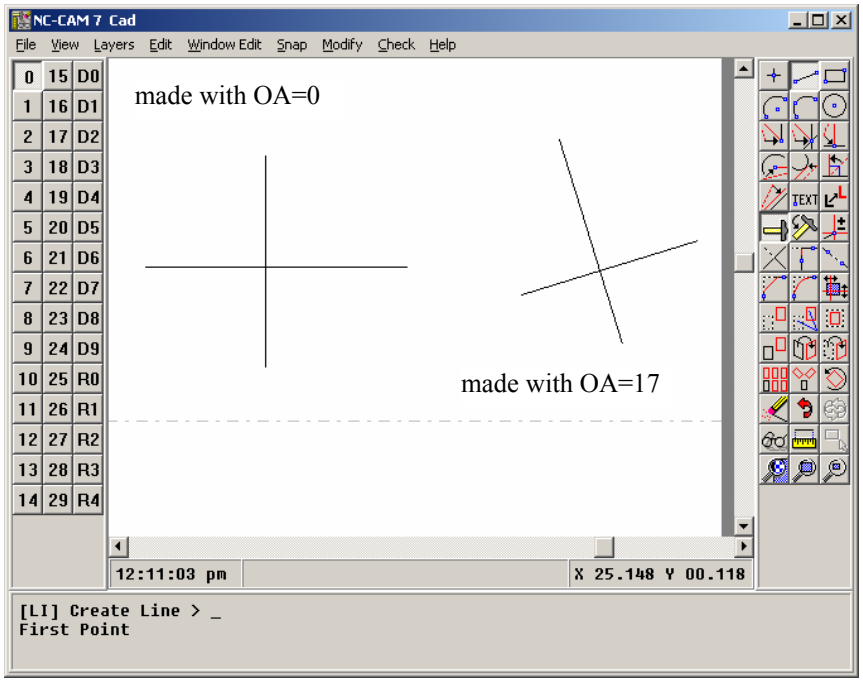


To get the line to swap from being locked on one axis to being locked on the other, you must move your cursor very close to the start point of the line.

One final note about *Ortho Mode*: You may instantly reverse *Ortho Mode*'s on/off state by holding down the **CTRL** key. This is much quicker than actually pushing the button on the menu.

[OA] Ortho Angle

The **OA** *Ortho Angle* function allows you to set the angle of the axes used by the **OR** *Ortho Mode* to any angle you wish. This is incredibly handy for creating lines which have a known angle, and one endpoint you can create with a snap or a keyboarded coordinate.



The angles you specify with the ortho angle function are measured in degrees, and the positive angles run counterclockwise with respect to the right-hand horizontal axis (usually the X+ axis).

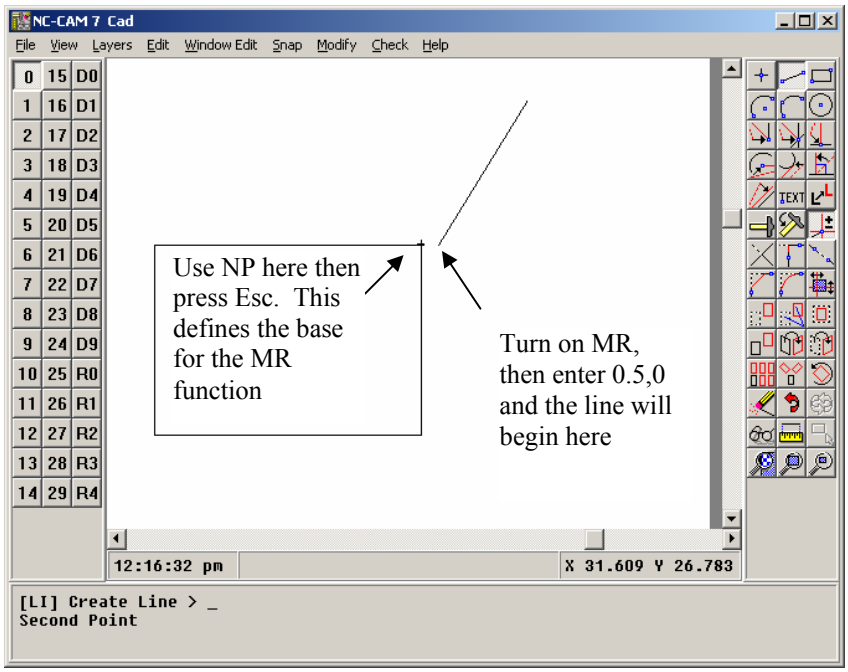
Using the ortho angle function automatically turns on ortho mode, as you'll see when the ortho mode button automatically flips to the "on" state when you enter a new ortho angle.

The only catch with using ortho angle is that it's persistent: Once you set it to an odd angle, that angle remains the effective ortho angle until you select **OA** a second time, and set the angle back to zero.

[MR] Relative Mode

The **MR** *Relative Mode* toggle affects the way coordinates you enter are interpreted. Ordinarily, all of the coordinates you enter are relative to the zero point of your drawing. Sometimes, however, it's advantageous for you to be able to enter a coordinate as a distance from the previous coordinate. That's what the **MR** *Relative Mode* toggle allows you to do.

Assume for a moment that you have a dimension on a drawing that's given as being one-half inch to the right of the upper-right corner of a rectangle. You could either inspect the rectangle's upper right corner, write down its coordinate, add half an inch in the X axis, and type in the resulting coordinate, or you could use the **MR** mode to enter the point.



As shown above, if the point you most recently created is not the point you've got to enter a coordinate relative to, just use **NP** or **SC** to snap the start of a new line to your desired reference point, then press **Esc**. Turn on **MR**, and the numbers you enter will be relative to that reference point.

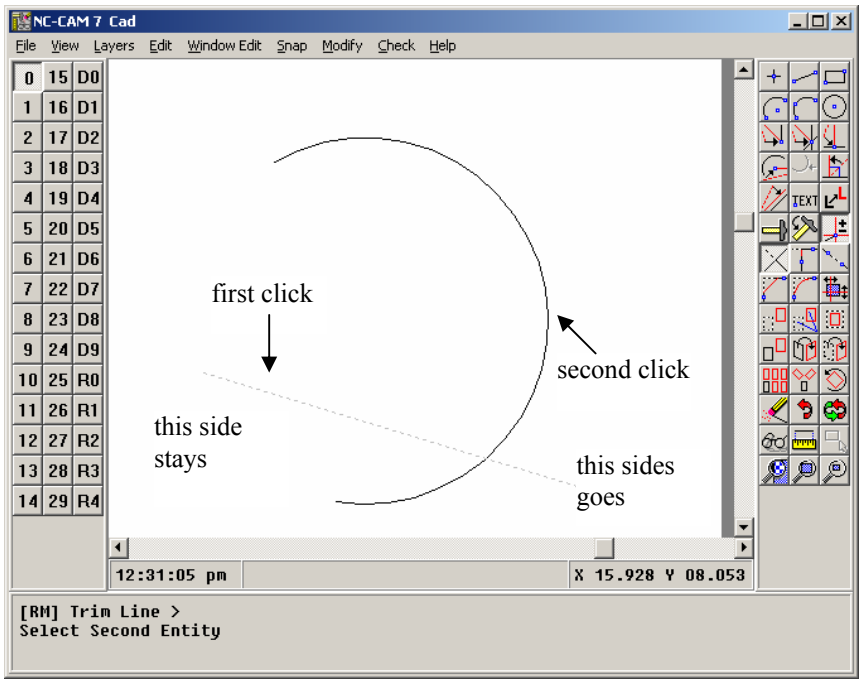
There's one added point worthy of special mention here: It's best to make sure you turn off the **MR** *Relative Mode* as soon as you're done using it. Forgetting that you're in relative mode can lead to some frustration when you later enter coordinates, and discover that they're not ending up where you intended them to be.

[RM] Trim Line

The **RM** *Trim* function allows you to extend or trim a line or arc to the point where it does (or would) intersect another linear entity. This function saves a huge amount of time when you're creating CAD drawings, because it allows you to avoid entering coordinates repeatedly.

When you select **RM** *Trim*, you must first choose the entity you want to change. Second, you select the entity you want to use as a knife, to cut the first entity at the point where the two intersect.

One aspect of the **RM** function that confuses some first-time users is that when you're using it to trim off the end of an entity, it keeps the side of the entity you click on first.



In this case, the left side of the line will be kept, and the right side will be trimmed off. As with the **SI** *Snap Intersection* function, if two intersections are possible, the one used is the one closest to your cursor when you select the second entity.

There's one added point worthy of special mention here: It's best to make sure you turn off the **MR** *Relative Mode* as soon as you're done using it. Forgetting that you're in relative mode can lead to some frustration when you later enter coordinates, and discover that they're not ending up where you intended them to be.

[OB] Object Break

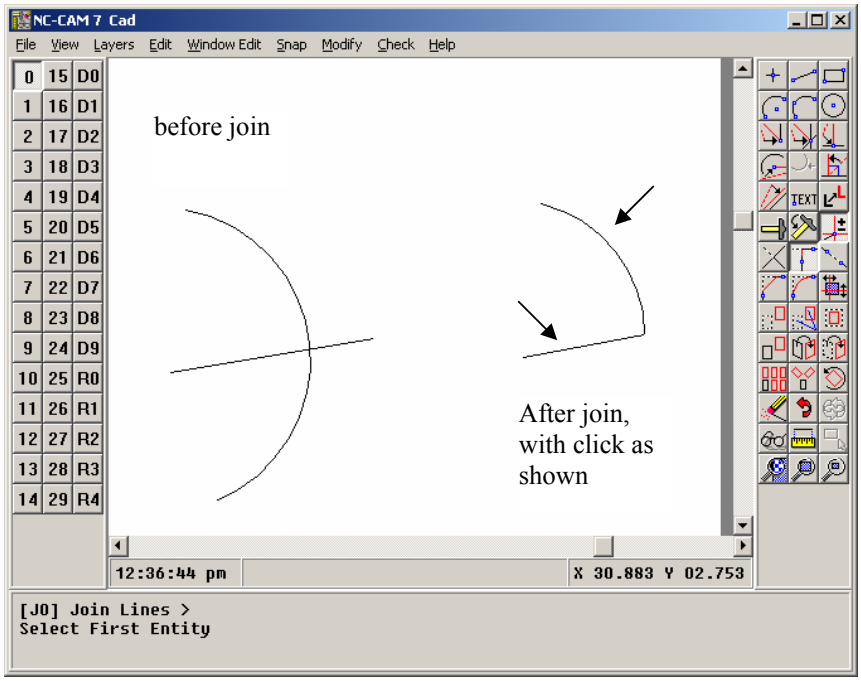
OB *Object Break* allows you to break lines, arcs, and circles. When you break a line or an arc, the result is two lines or arcs. When you break a circle, it becomes an arc.

When you're using the object break function, rubber-banded indicator lines connecting to your cursor will show where the object will be broken when you enter the next coordinate. You may break objects at precise locations by using snap functions to determine the break points.

One special note about using break on arcs and circles: If you need to break a circular entity at precise locations, it's generally faster to just break the object casually and then trim its ends than it would be to use the snap intersection function during the break command itself.

[JO] Join Lines

The **JO** *Join* function acts just like **RM** *Trim*, except that it trims both of the entities you select, keeping the side of each entity that you clicked on.



The **JO** *Join* function is very useful for cleaning up badly made intersections in customer-supplied drawings.

[WM] Window Move

The **WM** *Window Move* command allows you to click on an entity, or click-drag a window around a group of entities, and then move them.

The process of establishing the from/to positions for the entities to move is exactly the same as creating a line. This means you can use the same tricks you'd use in making a line to determine the exact positions the window moves from and to.

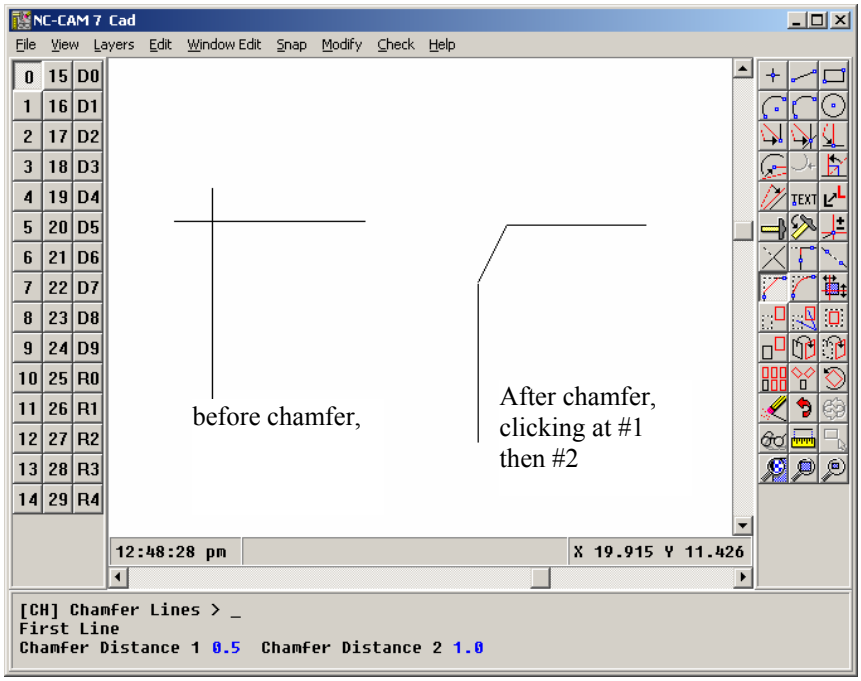
As with many CAD functions, there's a hidden feature in the window move, as well as all of the other window functions which follow: When you select a window full of objects, they become highlighted on the screen. If you want to add selected entities to the group to be changed, hold down your **SHIFT** key and click on the entities you want to add. You'll see them become highlighted, indicating that they're now included in the group you windowed!

This "shift-click" function also works in reverse: If your window accidentally selected an entity you did not want to include in the group, use shift-click to deselect it. You'll see the entity un-highlight, to indicate that it is no longer a member of the selected group.

[CH] Chamfer Lines

The **CH** *Chamfer* command creates a perfect chamfer (or bevel, if you prefer) at the intersection of the two lines you select. As with the **JO** *Join* command, the parts of the lines to be kept will be the parts you click.

The chamfer function has two settings; **CA** *Chamfer Distance 1*, and **CB** *Chamfer Distance 2*. These control how far back the two lines are trimmed before the joining chamfer is created.



In this example, chamfer distance 1 was one-half inch, and chamfer distance 2 was one inch.

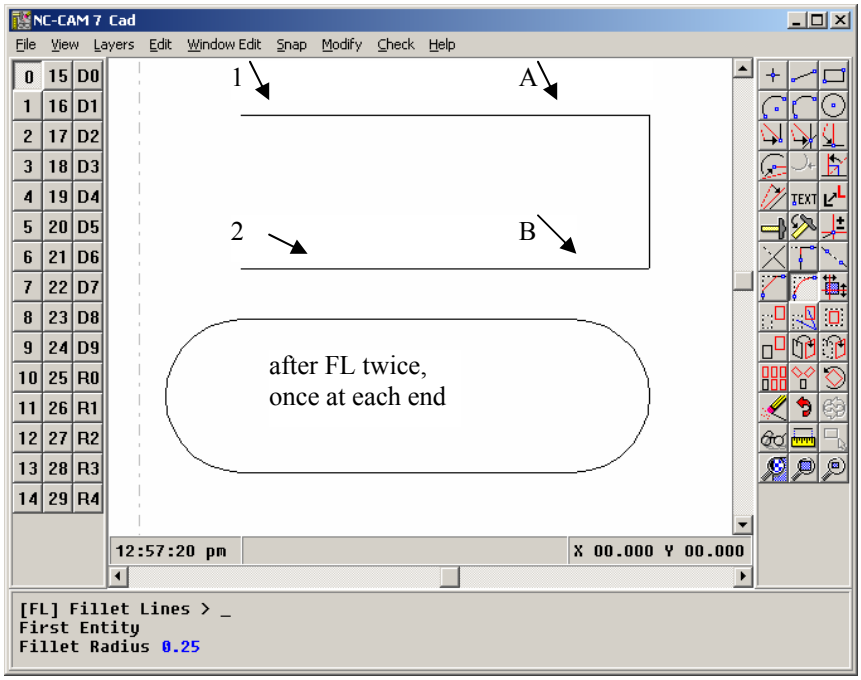
[DO] Drawing Origin

The **DO** *Drawing Origin* command moves all of the data in your drawing relative to the drawing origin. This is useful for temporarily setting the drawing's origin during editing, as well as finally establishing the zero point for routing.

[FL] Fillet Lines

The **FL** *Fillet* function acts like the **JO** *Join* function, but it creates a radius between the two entities. The radius is set with the **FR** *Fillet Radius* function. As with the join and chamfer functions, when the line or arc entities will be trimmed by the function, the side kept is the side you clicked on.

Two special cases for the **FL** function arise when you use choose lines which are parallel and end square to each other. In this case, the setting of the fillet radius is ignored, and a perfect 180 degree fillet is created between the two parallel lines.



In this illustration, **FL** has been used twice: Once with the clicks at 1 & 2, and again with the clicks at A & B. As shown above, filleting the open end of a rectangle puts a half circle across the end. Filleting the closed end of a rectangle also makes a half circle, but it deletes the joining line and backs up the arc to the point where it would have been tangent to the joining line.

[WS] Window Stretch

The **WS** *Window Stretch* function is used to "lengthen" a board profile by moving an edge in or out. When this function is activated, you will be asked to select an entity or select a group of entities using the click-drag window method to stretch.

When the group of entities is selected, there will be two types of highlighted entities on the screen. The first type will be the entities highlighted in white. These entities will simply be moved to their new location.

The second type of entity that will appear will be red highlighted entities. These are the entities that will be stretched. The endpoint of these entities that falls in the selected window will be moved to its new location, while the endpoint that fell out of the selected window will stay anchored.

After the entities have been selected, you will be asked to specify the move vector. The process of establishing the from/to positions for the entities to move is exactly the same as creating a line. This means you can use the same tricks you'd use in making a line to determine the exact positions the window moves from and to. The function works the same as **WM** *Window Move*.

[SW] Window Scale

The **SW** *Window Scale* function allows you to stretch or shrink the windowed entities by the factor you enter, referenced from a point you select.

Window Scale can alter the windowed group differently in the horizontal and vertical directions, if you wish. If you do scale a window asymmetrically, the circles and arcs receive special treatment: Circles remain circular when scaled asymmetrically, but the radius of each circle will be altered by the product of the two scale factors. This

means that if you scale a window by 2.0 in the horizontal axis, and 0.3 in the vertical axis, the radii of any circles you've selected will be scaled by 2.0 times 0.3, resulting in a scale of 0.6:1.

Arcs, on the other hand, actually are scaled asymmetrically. The treatment given to the arcs follows this procedure: First each arc has an added point computed for it, the point being the arc's midpoint (see the **SM** function for a definition of this). Then, the arc's midpoint, start point, and end point are scaled according to the two different scale factors you've selected. Finally, the arcs are reconstructed, using the math of the **A3** *Three-Point Arc* function to reestablish each arc using the three scaled points. This rather complex process does allow for asymmetrical scaling of arcs, albeit with arc centers that move as necessary to maintain equal start and end radii.

[WC] Window Copy

The **WC** *Window Copy* function allows you to copy all of the selected entities. After you select the entity or entities to be copied, you define the point to copy from, and the point to copy to. The "from" and "to" points may be entered as you'd enter any coordinate, but usually you'll want to use a snap or key in the values.

[WI] Window Mirror

The **WI** *Window Mirror* function copies all of the selected entities, mirroring around an arbitrary axis line. This powerful function is very handy for creating copies of individual entities at odd angles.

It's probably best for you to experiment with the **WI** *Window Mirror* function a bit before you use it in a live drawing. The mirror around an arbitrary line tends to be a little confusing, until you realize that the arbitrary line acts just like a "fold" in your drawing. A reasonable metaphor for the function would be that the windowed entities are "folded" around your arbitrary line.

[WF] Window Flip

The **WF** *Window Flip* function moves all of the selected entities, mirroring around an arbitrary axis line. This powerful function is very handy for moving entities to odd angles.

It's probably best for you to experiment with the **WF** *Window Flip* function a bit before you use it in a live drawing. The mirror around an arbitrary line tends to be a little confusing, until you realize that the arbitrary line acts just like a "fold" in your drawing. A reasonable metaphor for the function would be that the windowed entities are "folded" around your arbitrary line.

[MC] Matrix Copy

The **MC** *Matrix Copy* function produces from one to ninety-nine copies of the selected entities in the X and Y direction, evenly distributing the copies in an array with spacing you specify.

[RC] Radial Copy

The **RC** *Radial Copy* function produces from one to ninety-nine copies of the selected entities, evenly distributing the copies across the angle you specify, around the center point you specify. The icon for this function depicts a radial copy with a quantity of three, an angle of 360 degrees, and a center point located just above the bottom rectangle.

deletes. This replaces the **OE** *Object Erase* function GCADD users will be familiar with.

[OO] Undo

The **OO** *Undo* function allows you to reverse the effects of your editing, one step at a time, back to the beginning of your session. If you place your cursor over the *Undo* icon, the help text in the Status Window will describe the function you're about to reverse.

Note that when you pack the database (with the ***F**ile/Pac**k** Database* function), the *Undo* function's memory is erased.

[UU] Redo

The **UU** *Redo* function reverses the effect of the **OO** *Undo* function, one step at a time. This button will be "grayed out" until you've pressed the **OO** *Undo* button. As with the **OO** *Undo* icon, if you place your cursor on the **UU** *Redo* icon, the help text will describe the function you're about to *Redo* if you press the button.

[OI] Object Inspect

The **OI** *Object Inspect* function allows you to select any entity in the drawing, and view the coordinates of the points making up the entity. Further, once you've selected an entity with **OI** *Object Inspect*, you can modify the entity by clicking on any of its defining values, and typing in new values. When you do this, the highlighted entity will instantly change on-screen, and turn red in color. If you want to restore the entity to its original condition, press **ESC**. To keep any change you've made, press **ENTER**.

[ME] Measure

The **ME** *Measure* function allows you to measure from any construction point in your drawing to any other. Further, you may measure angles by defining a baseline with your first two clicks, and an angle with the third mouse click.

To reverse the angle reading in the measure mode, click your mouse near the apex of the angle.

[SA] Select All

The **SA** *Select All* button is unavailable until you've selected any of the window functions. With a window function selected, **SA** will automatically select all of the entities in your drawing.

There's a special capability available with the **SA** *Select All* function. If you want to select everything in your drawing except what's inside a window, follow these steps: First, select the window function you want. Second, window the group you want excluded from the window operation, and they'll highlight. Last, hold the **SHIFT** key and press the **SA** *Select All* button. This will reverse the selections you've made.

You may then proceed with the remaining steps of the window function you originally selected.

[ZW] Zoom In

The **ZW** *Zoom Window* function allows you to window a portion of the screen you want to zoom in on. The creation of this window is identical to the process of creating a rectangle. After the zoom region is specified, the work area is redrawn, focusing in on the area you selected.

The counterpart to this function is the **ZB** *Zoom Back* function, which can be used to decrease the magnification.

A useful new feature of NC-CAM is the ability to pan with a simple **ALT**-click-drag combination. If the display was magnified to show the upper-left-hand portion of your drawing, and you wanted to "slide" the view over towards the upper-right-hand side of the drawing, the new pan feature can be used. Hold down the **ALT** key, left-click with the mouse on the right-hand side of the drawing, and drag that location to the left-hand side of the drawing. In essence, the new pan feature "picks up and drags the work area," just as you would move a sheet of paper to center in front of you the area that you wanted to work on.

To repeatedly pick up and move the drawing in this manner, keep holding down the **ALT** key, let go of the left mouse button, drag the mouse to the opposite side of the drawing, left-click the mouse, and drag back. Advanced users of NC-CAM will learn the click-drag, click-drag, click-drag motion and grow to use it more often.

[ZA] Zoom All

The **ZA** *Zoom All* function redraws the screen at a scale which will show all of the entities in your drawing.

[ZB] Zoom Out

The **ZB** *Zoom Back* function reduces the magnification of the screen by a factor of two.

The counterpart to this function is the **ZW** *Zoom Window* function, which can be used to increase the magnification.

[PA] Pan

The **PA** *Pan* function, although now mostly obsolete, is included to keep backwards compatibility with NC-CAM 5. This function is replaced by the new scheme whereby you hold down the **ALT** key, and depress the left mouse button to pan the screen.

With this pan function, however, you will be asked to select a new center for the screen. When you click the mouse, the location you select will be the new center of the visible viewing area, with the same zoom factor.

[OE] Delete Entity(s)

OE *Delete Entity(s)* is the same function as **WE** *Window Erase*.

The **WE** *Window Erase* function erases single entities, or windowed groups of entities. This is the only window function which does not allow you to select multiple entities with the shift-click approach described in the **WM** *Window Move* function: When you select an entity by clicking on it, or a group of entities with a click-drag window, they are deleted immediately.

Because the windowing functions in NC-CAM's CAD editor also allow you to select an individual entity by doing a simple click (instead of a click-drag), this function is also used for individual deletes. This replaces the **OE** *Object Erase* function GCADD users will be familiar with.

[FR] Fillet Radius

The **FR** *Fillet Radius* function allows you to set the radius that will be used by the **FL** *Fillet Lines* function. The units are entered in the current (Metric/Inch) mode.

[NX] Number in X

The **NX** *Number in X* function sets the number of copies made in the X direction when using the **MC** *Matrix Copy* function.

[NY] Number in Y

The **NY** *Number in Y* function sets the number of copies made in the Y direction when using the **MC** *Matrix Copy* function.

[XS] X Increment

The **XS** *X Increment* function sets the distance between copies made in the X direction when using the **MC** *Matrix Copy* function. You can not reach this command directly, it is only a part of the **MC** *Matrix Copy* command.

[YS] Y Increment

The **YS** *Y Increment* function sets the distance between copies made in the Y direction when using the **MC** *Matrix Copy* function. You can not reach this command directly, it is only a part of the **MC** *Matrix Copy* command.

[TD] Total Degrees

The **TD** *Total Degrees* function sets the total degrees used during a **RC** *Radius Copy*.

[NC] Number of Copies

The **NC** *Number of Copies* function sets the number of copies made during a **RC** *Radius Copy*.

[RD] Redraw

The **RD** *Redraw* function redraws the workspace area. This is especially useful if you have deleted several entities, and the deleted entities have "covered" part or all of some entities that remain.

[NP] Snap Near Point

The **NP** *Snap Near Point* function, although only available from the keyboard, is one of the most useful snap functions in the CAD snap set. It allows the user to make "perfect" drawings where there are no gaps left in the corners of a board profile that might be left if you use freehand drawing.

If you have a 3-button mouse on your computer, the middle button works as an **NP** snap. Simple move the cursor near a point and press the middle mouse button once to snap to the point.

[CA] Chamfer Distance 1

This function sets the first chamfer distance used in **CH** *Chamfer Lines*.

[CB] Chamfer Distance 2

This function sets the second chamfer distance used in **CH** *Chamfer Lines*.

[MO] Absolute Mode

This command turns off relative mode which can be used to enter coordinates in the user panel. This function turns off the relative mode regardless of the state of the **MR** *Relative Mode* button.

[OC] Copy

The **OC** *Copy* function in the same as the **WC** *Window Copy* function.

The **WC** *Window Copy* function allows you to copy all of the selected entities. After you select the entity or entities to be copied, you define the point to copy from, and the point to copy to. The "from" and "to" points may be entered as you'd enter any coordinate, but usually you'll want to use a snap or key in the values.

[WO] Window Delete Outside

The **WO** *Window Delete Outside* function behaves much like the **WE** *Window Erase* function, with the exception that the entities deleted are the entities outside of the click-dragged window, not the entities on the inside.

[LS] Target Layer

The **LS** Target Layer function allows you to set the layer where **LO** *Layer Operation* functions will land.

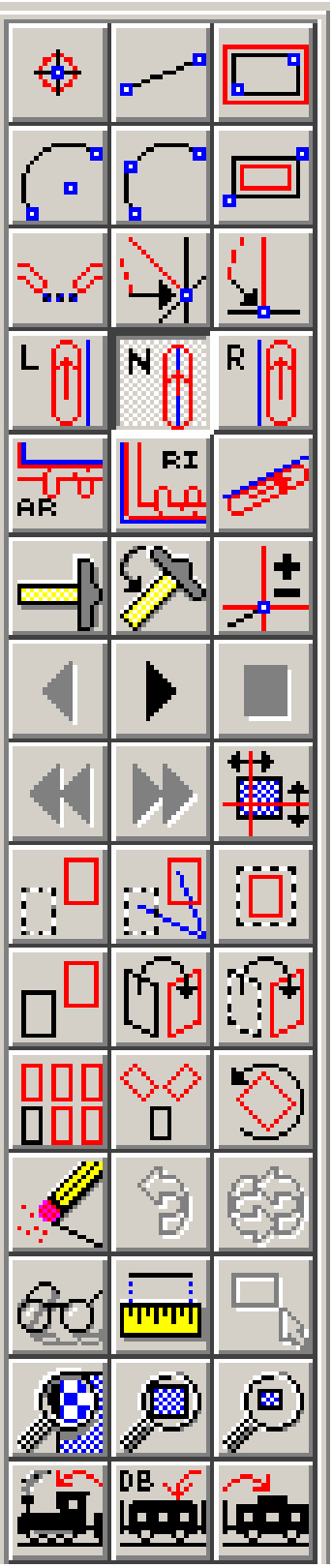
[TP] Test Path

The **TP** *Test Path* allows you to test a board profile in CAD before going over to Rout and using **FP** *Follow Profile* or **AR** *Auto Rout* to rout the profile. This helps find problems such as unlinked paths and paths with multiple entities connected to one point.

C H A P T E R 6

RoutEdit command reference

This chapter provides a command-by-command reference to the NC-CAM 7 RoutEdit functions.



DH *Drill Hole*
LI *Rout Line*
RS *Rout Outside Rectangle*

A2 *Create 2-Point Routed Arc*
A3 *Create 3-Point Routed Arc*
IR *Rout Inside Rectangle*

AT *Auto Tab*
SI *Snap Intersection*
SM *Snap Midpoint*

HL *Comp Left*
HN *Comp None*
HR *Comp Right*

AR *Auto Rout Outside*
RI *Auto Rout Inside*
FP *Follow Profile*

OR *Turn on Ortho Snap*
OA *Ortho Angle*
MR *Relative Mode*

PR *Previous Cut*
PL *Next Cut*
EN *End Single Step*

RW *Previous Sequence*
FF *Next Sequence*
DO *Drawing Origin*

WM *Window Move*
WS *Window Stretch*
SW *Window Scale*

WC *Window Copy*
WI *Window Mirror*
WF *Window Flip*

MC *Matrix Copy*
RC *Radial Copy*
RO *Rotate*

WE *Window Erase*
OO *Undo*
UU *Redo*

OI *Object Inspect*
ME *Measure*
SA *Select All*

ZW *Zoom In*
ZA *Zoom All*
ZB *Zoom Out*

DF *Do First*
DB *Do Before*
DL *Do Last*

[DH] Drill Hole

The **DH** *Drill Hole* function allows you to drill a single hole in a specified location. It is very similar to the **PO** *Create Point* function in CAD.

This function is primarily used in DrillEdit, however, it is also available in RoutEdit for creating holes that will be stored at output time as G05 holes.

[LI] Rout Line

The **LI** *Rout Line* function can be used to create Rout programs one cut at a time. This is useful if you are creating simple square routs or for adding a quick internal cut. The function works much the same way that **LI** *Create Line* in CAD works, with one notable exception. When the command is used, you will be prompted for the first point. After you do, you will notice a "rubber-banded" line from the first point to the cursor. When you define the second point, you will notice that a linear cut is added to the database.

In addition, you will also notice that there is now a rubber-banded line from the last point you created to the cursor. Once again, when you define another point, another linear cut is added to the database. With this command, you can create consecutive cuts until you reach the lift point of the cutter. At this point, you will need to exit the consecutive cut mode by right-clicking with the mouse, or hitting **ESC**.

Should you wish to use *Rout Line* in conjunction with **A2** *Rout 2-Point Arc* or **A3** *Rout 3-Point Arc*, switching to either of these two commands will not shut off the rubber band. Instead, you can continue creating cuts without inserting a lift-plunge in the database.

Additionally, should you with to continue with this function after already hitting **ESC**, you can use **NP** (*Snap Near Point*) or **SC** (*Snap Closest*) to "attach" to the rout profile and continue adding cuts.

As in CAD, you may use ortho mode by holding down the **CTRL** key.

[RS] Rout Outside Rectangle

The **RS** *Rout Outside Rectangle* function is used for adding simple square profiles to the rout database. The direction of travel is determined by the current compensation which is set using **HL** *Comp Left*, **HN** *Comp None*, **HR** *Comp Right*.

The **RS** *Rout Outside Rectangle* works very similar to the **RE** *Create Rectangle* command.

When the **RS** *Rout Outside Rectangle* function is selected, you will be prompted to select the first corner of the rectangle. Note that this also determines the plunge point for the router bit. Once the first corner is selected, a rubber-banded rectangle will be shown on screen from the first corner to the current cursor position. You will be prompted for the second corner. Once this corner is selected, the routed rectangle will be added to the rout database and shown on screen.

[A2] Create 2-Point Routed Arc

The **A2** *Two-Point Routed Arc* function creates a circular cut as defined by an arc. The arc is defined in a slightly different order than in CAD, so that the user may continue defining attached arcs without a lift-plunge being inserted in the database. An arc in rout is defined by first choosing the start point, then the arc's center point, and finally its endpoint. The center point is like the place you'd anchor a compass if you were drawing an arc on a drafting table. Once the start point is defined, you'll get a rubber-banded straight line attached to your cursor, until you define the center point.

A two-point arc's center point defines both the angle from the start to the center, as well as fixing the arc's radius. After the center point is defined, the rubber-banded arc you'll see attached to your cursor has a radius that's fixed at the distance from the center point. When you're defining the end point, all you're actually setting is the ending angle of the arc, even if you use a snap function to enter it.

In addition, you will also notice that there is now a rubber-banded line from the last point you created to the cursor. Once again, when you define another center and end point, another circular cut is added to the database. With this command, you can create consecutive cuts until you reach the lift point of the cutter.

Should you wish to use **A2** *Create 2-Point Routed Arc* in conjunction with **LI** *Rout Line* or **A3** *Create 3-Point Routed Arc*, switching to either of these two commands will not shut off the rubber band. Instead, you can continue creating cuts without inserting a lift-plunge in the database.

Additionally, should you with to continue with this function after already hitting **Esc**, you can use **NP** (*Snap Near Point*) or **SC** (*Snap Closest*) to "attach" to the rout profile and continue adding cuts.

[A3] Create 3-Point Arc

The **A3** *Create Three-Point Routed Arc* function creates a circular cut as defined by an arc. The arc is made by selecting one endpoint, then a point along the arc, and finally the other endpoint. This means that the radius is not actually set until the third and final point has been entered. The resulting rubber-band effect is quite unusual, as the arc's center and radius change fluidly as you move the mouse while defining the second end point.

In addition, you will also notice that there is now a rubber-banded line from the last point you created to the cursor. Once again, when you define another second and end point, another circular cut is added to the database. With this command, you can create consecutive cuts until you reach the lift point of the cutter.

You may switch to creating Routed Lines or 2-Point arcs, and can also resume drawing 3-Point Arcs as explained in **A2** *Create 2-Point Routed Arc*.

[IR] Rout Inside Rectangle

The **IR** *Rout Inside Rectangle* function is the same as the **RS** *Rout Outside Rectangle* function except that this function is meant for creating cutouts inside a board profile. The direction of travel is determined by the current compensation which is set using **HL** *Comp Left*, **HN** *Comp None*, **HR** *Comp Right*.

The **IR** *Rout Inside Rectangle* works very similar to the **RE** *Create Rectangle* command in CAD.

When the **IR** *Create Inside Rectangle* function is selected, you will be prompted to select the first corner of the rectangle. Note that this is also determines the plunge point for the router bit. Once the first corner is selected, a rubber-banded rectangle will be shown on screen from the first corner to the current cursor position. You will be prompted for the second corner. Once this corner is selected, the routed rectangle will be added to the rout database and shown on screen.

[AT] Auto Tab

The **AT** *Auto Tab* function is a method for adding quick breakaway tabs to any rout profile. This function should be selected after the board outline has already been routed using **FP** *Follow Profile* or **AR** *Auto Rout Outside*.

When the **AT** *Auto Tab* function is selected, you will notice the "Configure Tabs" prompt in the Command Window. By selecting this prompt with your mouse, you can define the tab style of your breakaway tabs. See the help section for *Configuring Tabs*.

When the correct style tab has been configured, you will be prompted for the edge of the board on which you wish to place the breakaway tab. The selected entity will be highlighted in white and a rubber-banded line will appear from the current cursor position perpendicular to the selected rout cut. At this point you can specify the position of the tab in the following ways:

1) You may press the cursor in the workspace at the perpendicular location where you wish to place the tab.

2) You may key the perpendicular coordinates in the user panel.

3) You may select the **SM** *Snap Midpoint* function to place the tab in the middle of the currently selected rout cut.

4) You may select the **SI** *Snap Intersection* function to place the tab at the intersection of the currently selected rout entity and a CAD entity of your choice. This is useful if the location is specified by a line in the cad drawing.

5) You may key in an X or Y aligned coordinate by keying in **X=1.100**, or **Y=2.45**, or something similar.

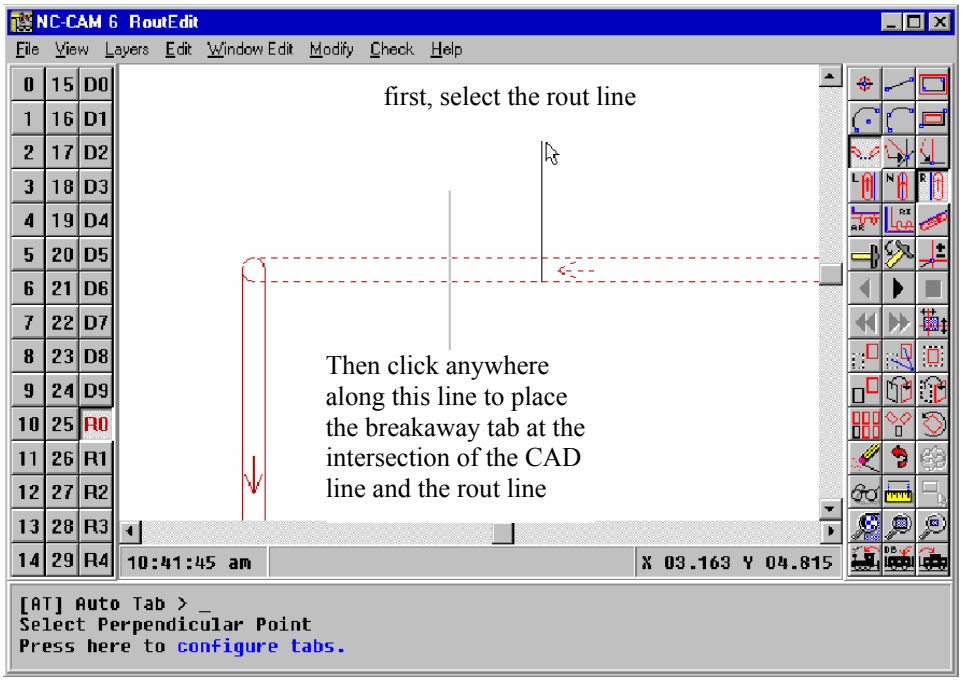
The tab will now be shown as it is added to the database.

[SI] Snap Intersection

The **SI** *Snap Intersection* function is used when you're entering a coordinate, and you want it to be at precisely the point where two existing entities intersect. The process of using snap intersection, as described in the progress prompt, is to select the two intersecting entities, one at a time. Rout then computes the location of the intersection of the two entities, and enters the needed coordinate at that location.

In RoutEdit, the **SI** *Snap Intersection* command is used for placing breakaway tabs at positions determined by intersecting CAD lines. This is particularly useful if you want to place a tab at a location dimensioned in your CAD data. First select the AT Auto Tab function and click on the rout segment that you wish to break. This segment will highlight as a dotted entity. Select the SI command and you will be prompted in the Command Window to pick the Select Entity - Intersect. Now you can click on the CAD line which locates the tab on the routed segment. The tab will appear where this CAD line intersects the rout.

This snap can be used on routed arcs as well as straight cuts.

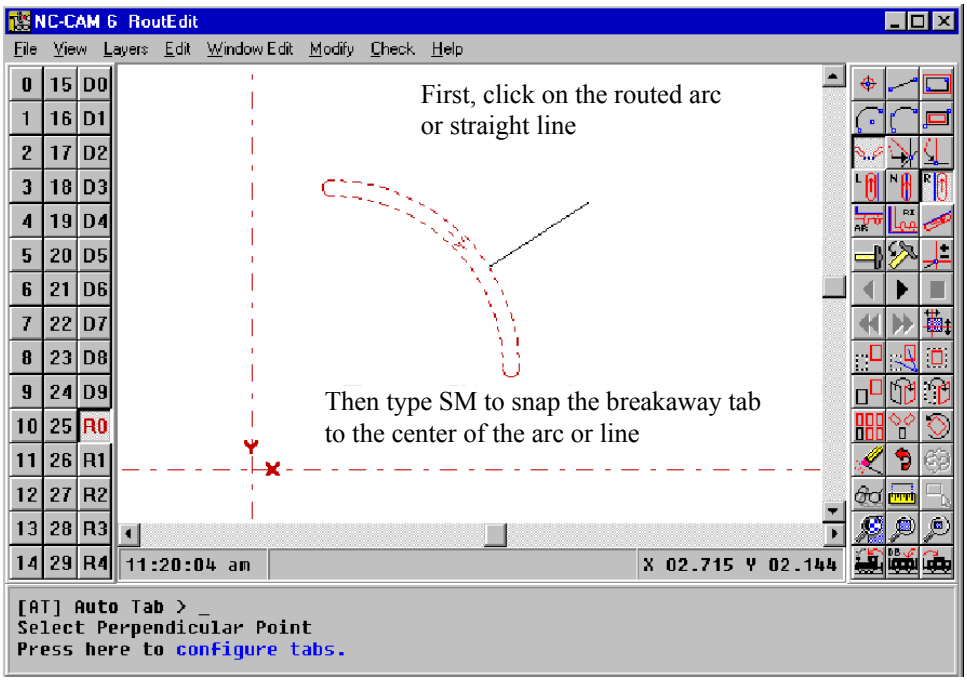


Note that the snap intersection function does not require that the two entities actually intersect; only that they could intersect if they were long enough to touch each other.

[SM] Snap Midpoint

The **SM** *Snap Midpoint* function snaps to the midpoint of the line or arc you select next. The line midpoint chosen is exactly halfway between its two endpoints.

If you select an arc with the *Snap Midpoint* function, the coordinate you'll snap to is the arc's midpoint, not its center.



The midpoint of an arc is a point along the arc, such that the point bisects the arc as shown above.

In RouteEdit, the **SM** *Snap Midpoint* command is used for placing breakaway tabs at the midpoint of the routed segment. First select the **AT** *Auto Tab* function and click

on the rout segment that you wish to break. This segment will highlight as a dotted entity and a rubber-banded line will appear perpendicular to the selected entity. Now simple type **SM** and the breakaway will be added at the midpoint of the routed segment.

This snap can be used on routed arcs as well as straight cuts.

[HL] Comp Left

The **HL** *Comp Left* function sets the current comp type to left compensation. All entities added after this selection will be comped to the left.

[HN] Comp None

The **HN** *Comp None* function sets the current comp type to no compensation. All entities added after this selection will have no compensation.

[HR] Comp Right

The **HR** *Comp Right* function sets the current comp type to right compensation. All entities added after this selection will be comped to the right.

[AR] Auto Rout Outside

The **AR** *Auto Rout Outside* function is used to create a rout path using data that exists in the CAD database. When this function is selected, you must have the CAD data you wish to rout currently displayed on the screen.

The compensation is selected by using the **HL** *Comp Left*, **HN** *Comp None*, **HR** *Comp Right*.

When you select this function, you will be prompted to select an entity to chase. You may select any entity in the CAD database.

Once you have selected the appropriate entity, **AR** *Auto Rout Outside* will follow all the entities connected end to end until one of three things happens:

1) The chaser finds no more connected entities. It terminates abnormally, giving you an error indicating that **AR** *Auto Rout Outside* could not link a closed path.

2) The chaser loops back upon itself. It terminates normally, showing you the closed path it has created.

3) The chaser connects to more than one entity, and can't figure out which way to go. In this case, you will be prompted for which branch to follow. You have two options here. You can stop by selecting "Stop Here" or you can select OK and select the direction in which you wish to continue. Notice that the path already visited is highlighted in white, and the undetermined directions are highlighted in red. Choose one of the red entities or press **Esc** to stop at the current point. Notice that if you stop before completing the path, the error mentioned in 1) will occur.

Once the **AR** *Auto Rout Outside* function terminates, the path will be added to the Rout database, and shown on-screen.

[FP] Follow Profile

The **FP** *Follow Profile* function is used to create a rout path using data that exists in the CAD database. When this function is selected, you must have the CAD data you wish to follow currently displayed on the screen.

The compensation is selected by using the **HL** *Comp Left*, **HN** *Comp None*, **HR** *Comp Right*.

When you select this function, you will be prompted to select an entity to chase. You may select any entity in the CAD database. Once the entity is selected, you will be asked to select the direction in which to Rout. This is done by selecting the end of the entity towards which the cutter will travel. To select this end, press both the left and right mouse buttons at the same time. You may also change the entity you selected by simply clicking on another entity.

Once you have selected the appropriate entity and the direction for travel, **FP** *Follow Profile* will follow all the entities connected end to end until one of three things happens:

1) The chaser finds no more connected entities. It terminates normally, showing you the path it has created.

2) The chaser loops back upon itself. It terminates normally, showing you the closed path it has created.

3) The chaser connects to more than one entity, and can't figure out which way to go. In this case, you will be prompted for which branch to follow. You have two options here. You can stop by selecting "Stop Here" or you can select OK and select the direction in which you wish to continue. Notice that the path already visited is highlighted in white, and the undetermined directions are highlighted in red. Choose one of the red entities or press **Esc** to stop at the current point.

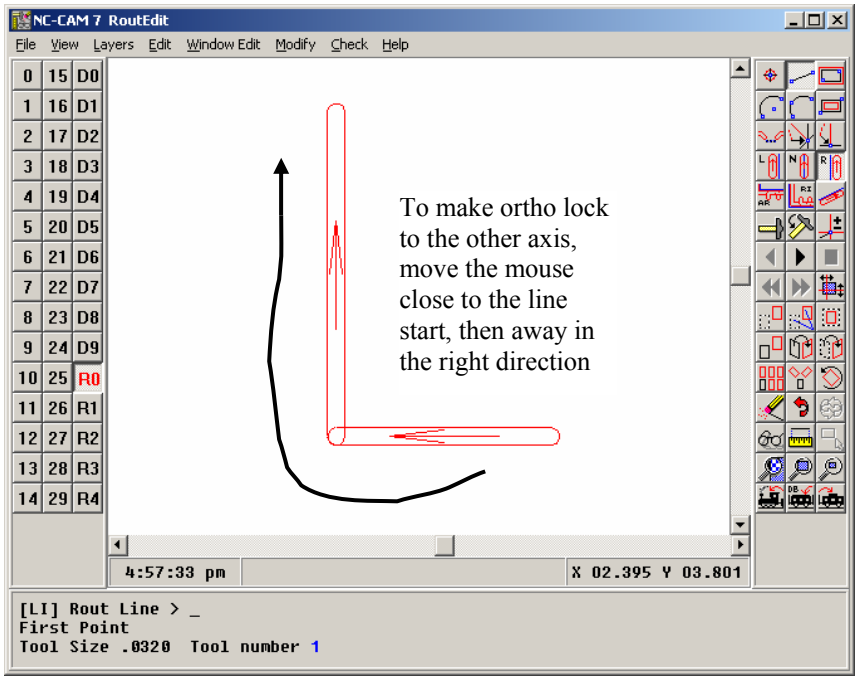
Once the *Follow Profile* function terminates, the path will be added to the Rout database, and shown on-screen.

[OR] Turn on Ortho Snap

OR *Ortho Mode* applies a constraint to the cursor motion once a point has been entered. The constraint acts to force the rout line you're creating to be square to the axes, or to the angle you've entered with the **OA** *Ortho Angle* function described on the following page.

When it's turned on, the *Ortho Mode* will force newly-created rout lines square to the ortho angle, even if you snap the second end of a line you're creating to some other object.

In one respect, ortho mode acts a bit like the **ST** *Snap Tangent* CAD function described earlier: Once it's locked on to an axis, say the "X" axis for example, it stays locked on to that axis no matter how far you move your mouse along the other axis.

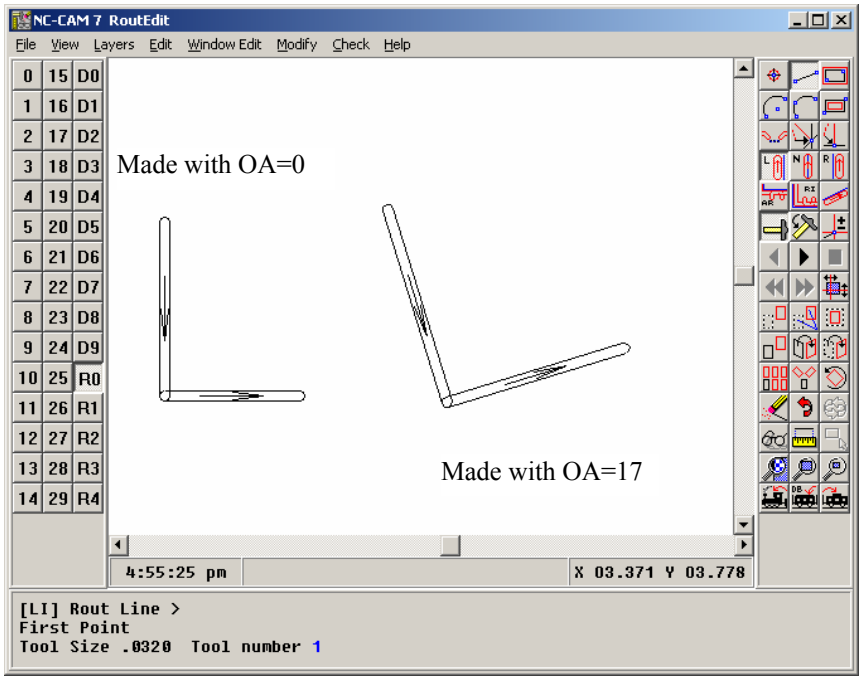


To get the line to swap from being locked on one axis to being locked on the other, you must move your cursor very close to the start point of the line.

One final note about *Ortho Mode*: You may instantly reverse *Ortho Mode*'s on/off state by holding down the **CTRL** key. This is much quicker than actually pushing the button on the menu.

[OA] Ortho Angle

The **OA** *Ortho Angle* function allows you to set the angle of the axes used by the **OR** *Ortho Mode* to any angle you wish. This is incredibly handy for routing lines which have a known angle, and one endpoint you can create with a snap or a keyboarded coordinate.



The angles you specify with the ortho angle function are measured in degrees, and the positive angles run counterclockwise with respect to the right-hand horizontal axis (usually the X+ axis).

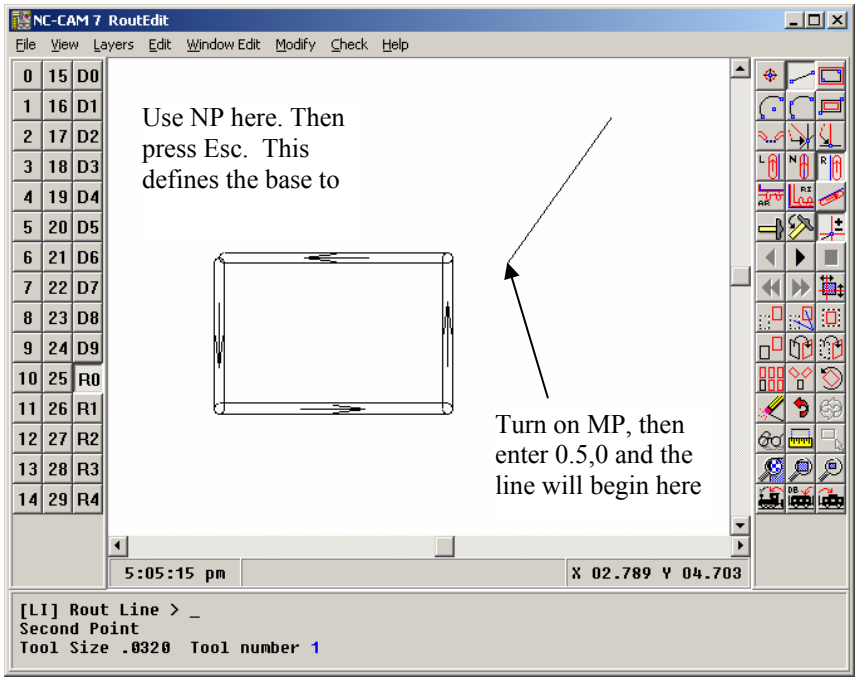
Using the ortho angle function automatically turns on ortho mode, as you'll see when the ortho mode button automatically flips to the "on" state when you enter a new ortho angle.

The only catch with using ortho angle is that it's persistent: Once you set it to an odd angle, that angle remains the effective ortho angle until you select **OA** a second time, and set the angle back to zero.

[MR] Relative Mode

The **MR** *Relative Mode* toggle affects the way coordinates you enter are interpreted. Ordinarily, all of the coordinates you enter are relative to the zero point of your drawing. Sometimes, however, it's advantageous for you to be able to enter a coordinate as a distance from the previous coordinate. That's what the **MR** *Relative Mode* toggle allows you to do.

Assume for a moment that you have a dimension on a drawing that's given as being one-half inch to the right of the upper-right corner of a rectangle. You could either inspect the rectangle's upper right corner, write down its coordinate, add half an inch in the X axis, and type in the resulting coordinate, or you could use the **MR** mode to enter the point.



As shown above, if the point you most recently created is not the point you've got to enter a coordinate relative to, just use **NP** or **SC** to snap the start of a new line to your desired reference point, then press **Esc**. Turn on **MR**, and the numbers you enter will be relative to that reference point.

There's one added point worthy of special mention here: It's best to make sure you turn off the **MR** *Relative Mode* as soon as you're done using it. Forgetting that you're in relative mode can lead to some frustration when you later enter coordinates, and discover that they're not ending up where you intended them to be.

[PR] Previous Cut

The **PR** *Previous Cut* function is active only during single stepping. It displays the previous cut.

[PL] Next Cut

The **PL** *Next Cut* function is active only during single stepping. It displays the next cut in the Rout database.

[EN] End Single Step

The **EN** *End Single Step* function terminates the single step process and redisplays the database.

This function is only active during the single step process.

[RW] Previous Sequence

The **RW** *Previous Sequence* function is active only during the single step process. It rewinds the single step process to the previous head up.

[FF] Next Sequence

The **FF** *Next Sequence* function allows you to fast forward the single step process to the next head lift in the database.

[DO] Drawing Origin

The **DO** *Drawing Origin* command moves all of the data in your drawing relative to the drawing origin. This is useful for temporarily setting the drawing's origin during editing, as well as finally establishing the zero point for routing.

[WM] Window Move

The **WM** *Window Move* command allows you to click on an entity, or click-drag a window around a group of entities, and then move them.

The process of establishing the from/to positions for the entities to move is exactly the same as creating a line. This means you can use the same tricks you'd use in making a line to determine the exact positions the window moves from and to.

As with many Rout functions, there's a hidden feature in the window move, as well as all of the other window functions which follow: When you select a window full of objects, they become highlighted on the screen. If you want to add selected entities to the group to be changed, hold down your **SHIFT** key and click on the entities you want to add. You'll see them become highlighted, indicating that they're now included in the group you windowed!

This "shift-click" function also works in reverse: If your window accidentally selected an entity you did not want to include in the group, use shift-click to deselect it. You'll see the entity un-highlight, to indicate that it is no longer a member of the selected group.

[WS] Window Stretch

The **WS** *Window Stretch* function is used to "lengthen" a board profile by moving an edge in or out. When this function is activated, you will be asked to select an entity or select a group of entities using the click-drag window method to stretch.

When the group of entities is selected, there will be two types of highlighted entities on the screen. The first type will be the entities highlighted in white. These entities will simply be moved to their new location.

The second type of entity that will appear will be red highlighted entities. These are the entities that will be stretched. The endpoint of these entities that falls in the selected window will be moved to its new location, while the endpoint that fell out of the selected window will stay anchored.

After the entities have been selected, you will be asked to specify the move vector. The process of establishing the from/to positions for the entities to move is exactly the same as creating a line. This means you can use the same tricks you'd use in making a line to determine the exact positions the window moves from and to. The function works the same as **WM** *Window Move*.

[SW] Window Scale

The **SW** *Window Scale* function allows you to stretch or shrink the windowed entities by the factor you enter, referenced from a point you select.

Window Scale can alter the windowed group differently in the horizontal and vertical directions, if you wish. If you do scale a window asymmetrically, the circles and arcs receive special treatment: Circles remain circular when scaled asymmetrically, but the radius of each circle will be altered by the product of the two scale factors. This means that if you scale a window by 2.0 in the horizontal axis, and 0.3 in the vertical axis, the radii of any circles you've selected will be scaled by 2.0 times 0.3, resulting in a scale of 0.6:1.

Arcs, on the other hand, actually are scaled asymmetrically. The treatment given to the arcs follows this procedure: First each arc has an added point computed for it, the point being the arc's midpoint (see the **SN** function for a definition of this). Then, the arc's midpoint, start point, and end point are scaled according to the two different scale factors you've selected. Finally, the arcs are reconstructed, using the math of the **A3** *Three-Point Arc* function to reestablish each arc using the three scaled points. This rather complex process does allow for asymmetrical scaling of arcs, albeit with arc centers that move as necessary to maintain equal start and end radii.

[WC] Window Copy

The **WC** *Window Copy* function allows you to copy all of the selected entities. After you select the entity or entities to be copied, you define the point to copy from, and the point to copy to. The "from" and "to" points may be entered as you'd enter any coordinate, but usually you'll want to use a snap or key in the values.

[WI] Window Mirror

The **WI** *Window Mirror* function copies all of the selected entities, mirroring around an arbitrary axis line. This powerful function is very handy for creating copies of individual entities at odd angles.

It's probably best for you to experiment with the **WI** *Window Mirror* function a bit before you use it in a live drawing. The mirror around an arbitrary line tends to be a little confusing, until you realize that the arbitrary line acts just like a "fold" in your drawing. A reasonable metaphor for the function would be that the windowed entities are "folded" around your arbitrary line.

[WF] Window Flip

The **WF** *Window Flip* function moves all of the selected entities, mirroring around an arbitrary axis line. This powerful function is very handy for moving entities to odd angles.

It's probably best for you to experiment with the **WF** *Window Flip* function a bit before you use it in a live drawing. The mirror around an arbitrary line tends to be a little confusing, until you realize that the arbitrary line acts just like a "fold" in your drawing. A reasonable metaphor for the function would be that the windowed entities are "folded" around your arbitrary line.

[MC] Matrix Copy

The **MC** *Matrix Copy* function produces from one to ninety-nine copies of the selected entities in the X and Y direction, evenly distributing the copies with spacing you specify.

[RC] Radial Copy

The **RC** *Radial Copy* function produces from one to ninety-nine copies of the selected entities, evenly distributing the copies across the angle you specify, around the center point you specify. The icon for this function depicts a radial copy with a quantity of three, an angle of 360 degrees, and a center point located just above the bottom rectangle.

[RO] Rotate

The **RO** *Rotate* function simply rotates all of the selected entities around a point. The function is extremely useful for working on sections of drawings which are dimensioned off-axis. By using the **RO** function, you can rotate your entire drawing to place the needed section on-axis, do the editing you need to do, and then use **RO** a second time to rotate the entire drawing back to its original alignment.

[WE] Window Erase

The **WE** *Window Erase* function erases single entities, or windowed groups of entities. This is the only window function which does not allow you to select multiple entities with the shift-click approach described in the **WM** *Window Move* function: When you select an entity by clicking on it, or a group of entities with a click-drag window, they are deleted immediately.

Because the windowing functions in NC-CAM's CAD editor also allow you to select an individual entity by doing a simple click (instead of a click-drag), this function is also used for individual deletes. This replaces the **OE** *Object Erase* function GCADD users will be familiar with.

[OO] Undo

The **OO** *Undo* function allows you to reverse the effects of your editing, one step at a time, back to the beginning of your session. If you place your cursor over the *Undo* icon, the help text in the Status Window will describe the function you're about to reverse.

Note that when you pack the database (with the ***F**ile/Pac**k** Database* function), the *Undo* function's memory is erased.

[UU] Redo

The **UU** *Redo* function reverses the effect of the **OO** *Undo* function, one step at a time. This button will be "grayed out" until you've pressed the **OO** *Undo* button. As with the **OO** *Undo* icon, if you place your cursor on the **UU** *Redo* icon, the help text will describe the function you're about to *Redo* if you press the button.

[OI] Object Inspect

The **OI** *Object Inspect* function allows you to select any entity in the drawing, and view the coordinates of the points making up the entity. Further, once you've selected an entity with **OI** *Object Inspect*, you can modify the entity by clicking on any of its defining values, and typing in new values. When you do this, the highlighted entity will instantly change on-screen, and turn red in color. If you want to restore the entity to its original condition, press **ESC**. To keep any change you've made, press **ENTER**.

[ME] Measure

The **ME** *Measure* function allows you to measure from any construction point in your drawing to any other. Further, you may measure angles by defining a baseline with your first two clicks, and an angle with the third mouse click.

Once you've clicked on three points, you may "pick up" the endpoints by clicking on them a second time. To reverse the angle reading in the measure mode, click your mouse near the apex of the angle.

[SA] Select All

The **SA** *Select All* button is unavailable until you've selected any of the window functions. With a window function selected, **SA** will automatically select all of the entities in your drawing.

There's a special capability available with the **SA** *Select All* function. If you want to select everything in your drawing except what's inside a window, follow these steps: First, select the window function you want. Second, window the group you want excluded from the window operation, and they'll highlight. Last, hold the **SHIFT** key and press the **SA** *Select All* button. This will reverse the selections you've made. You may then proceed with the remaining steps of the window function you originally selected.

[ZW] Zoom In

The **ZW** *Zoom Window* function allows you to window a portion of the screen you want to zoom in on. The creation of this window is identical to the process of creating a rectangle. After the zoom region is specified, the work area is redrawn, focusing in on the area you selected.

The counterpart to this function is the **ZB** *Zoom Back* function, which can be used to decrease the magnification.

A useful new feature of NC-CAM is the ability to pan with a simple **ALT**-click-drag combination. If the display was magnified to show the upper-left-hand portion of your drawing, and you wanted to "slide" the view over towards the upper-right-hand side of the drawing, the new pan feature can be used. Hold down the **ALT** key, left-click with the mouse on the right-hand side of the drawing, and drag that location to the left-hand side of the drawing. In essence, the new pan feature "picks up and drags the work area," just as you would move a sheet of paper to center in front of you the area that you wanted to work on.

To repeatedly pick up and move the drawing in this manner, keep holding down the **ALT** key, let go of the left mouse button, drag the mouse to the opposite side of the drawing, left-click the mouse, and drag back. Advanced users of NC-CAM will learn the click-drag, click-drag, click-drag motion and grow to use it more often.

[ZA] Zoom All

The **ZA** *Zoom All* function redraws the screen at a scale which will show all of the entities in your drawing.

[ZB] Zoom Out

The **ZB** *Zoom Back* function reduces the magnification of the screen by a factor of two.

The counterpart to this function is the **ZW** *Zoom Window* function, which can be used to increase the magnification.

[DF] Do First

The **DF** *Do First* function allows you to control the sequence of the cuts in the rout database. This function, with the help of ***V**iew*|*Se**q**uence Number*, allows you complete control over the sequence that the router performs its cuts.

When this function is selected, you are prompted for either an entity of a window containing several entities. When you select the entity or group of entities, they will immediately be moved to the front of the output rout file. With the sequence numbers showing, you will see the selected entity or group of entities moved to number 1, etc..

[DB] Do Before

The **DB** *Do Before* function is similar to the **DF** *Do First* and **DL** *Do Last* functions, with the exception that this function allows you to shift-click select and unselect group items, and select where in the rout sequence the selected items are placed.

When you select the **DB** *Do Before* function, it will prompt you to select either an entity or a group of entities. You may either click on an entity to select just that entity, or you may click-drag a window around a group of entities. After you select an entity or group of entities, you can use shift-click to include or exclude cuts from the group you have selected, in a similar fashion to CAD window functions.

After you have selected a group of entities to operate on, you will be prompted to select an entity to place the group before.

The most useful aspect of this function is to do internal cutouts before cutting the outline of the board loose. To do this, select all the internal cutouts, then click on the board outline, and all the internal cutouts will now be routed before the board is cut loose. You can use Single Step to verify the order visually.

[DL] Do Last

The **DL** *Do Last* function allows you to control the sequence of the cuts in the rout database. This function, with the help of ***V**iew|**Seq**uence Number*, allows you complete control over the sequence that the router performs its cuts.

When this function is selected, you are prompted for either an entity of a window containing several entities. When you select the entity or group of entities, they will immediately be moved to the end of the output rout file. With the sequence numbers showing, you will see the selected entity or group of entities moved to the last number.

[PA] Pan

The **PA** *Pan* function, although now mostly obsolete, is included to keep backwards compatibility with NC-CAM 5. This function is replaced by the new scheme whereby you hold down the **ALT** key, and depress the left mouse button to pan the screen.

With this pan function, however, you will be asked to select a new center for the screen. When you click the mouse, the location you select will be the new center of the visible viewing area, with the same zoom factor.

[OE] Delete Entity(s)

OE *Delete Entity(s)* is the same function as **WE** *Window Erase*.

The **WE** *Window Erase* function erases single entities, or windowed groups of entities. This is the only window function which does not allow you to select multiple entities with the shift-click approach described in the **WM** *Window Move* function: When you select an entity by clicking on it, or a group of entities with a click-drag window, they are deleted immediately.

Because the windowing functions in NC-CAM's CAD editor also allow you to select an individual entity by doing a simple click (instead of a click-drag), this function is also used for individual deletes. This replaces the **OE** *Object Erase* function GCADD users will be familiar with. [NX] Number in X

The **NX** *Number in X* function sets the number of copies made in the X direction when using the **MC** *Matrix Copy* function.

[NY] Number in Y

The **NY** *Number in Y* function sets the number of copies made in the Y direction when using the **MC** *Matrix Copy* function.

[XS] X Increment

The **XS** *X Increment* function sets the distance between copies made in the X direction when using the **MC** *Matrix Copy* function. You can not reach this command directly, it is only a part of the **MC** *Matrix Copy* command.

[YS] Y Increment

The **YS** *Y Increment* function sets the distance between copies made in the Y direction when using the **MC** *Matrix Copy* function. You can not reach this command directly, it is only a part of the **MC** *Matrix Copy* command.

[TD] Total Degrees

The **TD** *Total Degrees* function sets the total degrees used during a **RC** *Radius Copy*.

[NC] Number of Copies

The **NC** *Number of Copies* function sets the number of copies made during a **RC** *Radius Copy*.

[RD] Redraw

The **RD** *Redraw* function redraws the workspace area. This is especially useful if you have deleted several entities, and the deleted entities have "covered" part or all of some entities that remain.

[NP] Snap Near Point

The **NP** *Snap Near Point* function, although only available from the keyboard, is one of the most useful snap functions in the CAD snap set. It allows the user to make "perfect" drawings where there are no gaps left in the corners of a board profile that might be left if you use freehand drawing.

If you have a 3-button mouse on your computer, the middle button works as an **NP** *Snap*. Simply move the cursor near a point and press the middle mouse button once to snap to the point.

[MO] Absolute Mode

This command turns off relative mode which can be used to enter coordinates in the user panel. This function turns off the relative mode regardless of the state of the **MR** *Relative Mode* button.

[OC] Copy

The **OC** *Copy* function in the same as the **WC** *Window Copy* function.

The **WC** *Window Copy* function allows you to copy all of the selected entities. After you select the entity or entities to be copied, you define the point to copy from, and the point to copy to. The "from" and "to" points may be entered as you'd enter any coordinate, but usually you'll want to use a snap or key in the values.

[WO] Window Delete Outside

The **WO** *Window Delete Outside* function behaves much like the **WE** *Window Erase* function, with the exception that the entities deleted are the entities outside of the click-dragged window, not the entities on the inside.

[CT] Configure Tabs

The **CT** *Configure Tabs* function allows you to precisely configure the way that a breakaway tab is inserted into a rout program when using the **AT** *Auto Tab* function.

The large buttons on the left side of the configuration box describe the available tab types. One of the tab styles features cut-ins, commonly used to avoid having remaining material from the breakaway tab stick out over the edge of the profile. Either tab style may also include drilled holes, to assist in breakability.

The breakaway holes are added to the rout database. They are output as a G05 command at the end of the rout file.

[AU] Configure Auto Rout

The **AU** *Configure Auto Rout* function allows you to choose between the different *Auto Rout* styles, and set specific distances as they relate to the *Auto Rout* plunge styles. The diagrams included on the large buttons on the left side of the dialog box describe the plunge style, and show how the specifications apply to that style. The style is selected by pressing the appropriate button down.

[LF] Link Sequence First

The **LF** *Link Sequence First* function is the same as the **DF** *Do First* function. It allows you to control the sequence of the cuts in the rout database. This function, with the help of ***V****iew*|*Se****q****uence Number*, allows you complete control over the sequence that the router performs it's cuts.

When this function is selected, you are prompted for either an entity of a window containing several entities. When you select the entity or group of entities, they will immediately be moved to the front of the output rout file. With the sequence numbers showing, you will see the selected entity or group of entities moved to number 1, etc..

[EX] Reverse Rout Path

The **EX** *Reverse Rout Path* function allows you to change the order of the cuts in a specific link to be reversed. Notice that this function does not change the compensation type, so the path made by the cutter switches to the other side of the profile edge if the compensation is set to anything but NONE.

This function is useful for taking internal cutouts that use no compensation and reversing their order so that the smooth edge of a one pass routed slot can be changed.

[SR] Step & Repeat

The **SR** *Step & Repeat* function allows you to automatically create a Step & Repeat pattern for panelization. The inputs for this function are the number of steps in the X and Y direction, and the distance between the parts.

When this function is selected, you will be asked to select an entity or a window. Typically you would select the entire part, including internal cutouts. Once the part is selected, you may shift-click to include or exclude different cuts, in the same fashion as the window operations in CAD.

Once you have selected the appropriate cuts for the Step & Repeat pattern, you will be asked to confirm the operation by either hitting **ENTER** or clicking in the workspace. When you confirm the operation, you will see the dialog box that allows you to set the matrix parameters for the Step & Repeat pattern.

The NESTED STEPS checkbox allows you to specify the format of the Step & Repeat list. If you select nested steps, the resulting steplist will contain an M01, the list of steps in the X direction, another M01, and a list of steps in the Y direction. If you deselect this checkbox, the step list will explicitly list all the steps, with only one M01.

When OK is selected, the Step & Repeat list is added to the RoutEdit database and the steps are shown on-screen.

[SE] Step & Repeat Edit

The **SE** *Step & Repeat Edit* function allows you to edit a Step & Repeat List already in the database. After you have finished the edit, the changes will immediately show up on the screen. You will note that the separate nesting levels are represented by using multiple edit windows separated by a vertical bar.

[SD] Step & Repeat Delete

The **SD** *Step & Repeat Delete* function allows the user to delete an existing Step & Repeat pattern from the database.

When the function is selected, it will ask for either an entity or a group of entities which may be selected using the click-drag method. Once the entity or group of entities are selected, all the Step & Repeat information associated with those entities is immediately deleted.

The results of the delete will be immediately displayed.

[CS] Create Step & Repeat

The **CS** *Create Step & Repeat* allows you to interactively create a Step & Repeat list in an edit window.

When the function is selected, like **SR** *Step & Repeat*, the function will ask you for an entity or group of entities to which to apply the Step and Repeat list. Once you have selected the entities, you may add or delete entities by shift-clicking on entities.

Once you have established the group of entities to which you wish to apply the function, you need to confirm the operation by clicking in the workspace or pressing **ENTER**. Once you have confirmed the operation, an edit screen will appear where you can enter the Step & Repeat list. Once the Step & Repeat is entered, and the OK button is selected, the Step & Repeat list is applied to all the selected entities, and displayed in the workspace.

[AS] Add to Step & Repeat

The **AS** *Add to Step & Repeat* function is used to add a cut or set of cuts to an existing Step and Repeat list. This function is especially useful when adding an internal cut to a part that has already been panelized using Step & Repeats.

When the function is selected, it will ask you to select either an entity or group of entities by clicking on an entity or click-dragging a window around a group of entities. You may add or subtract entities from the selected group by shift-clicking on the entities. Once you have selected the group on which to apply the operation, you must select a stepped entity which has the appropriate Step & Repeat list to copy.

[XP] Expand Step & Repeat

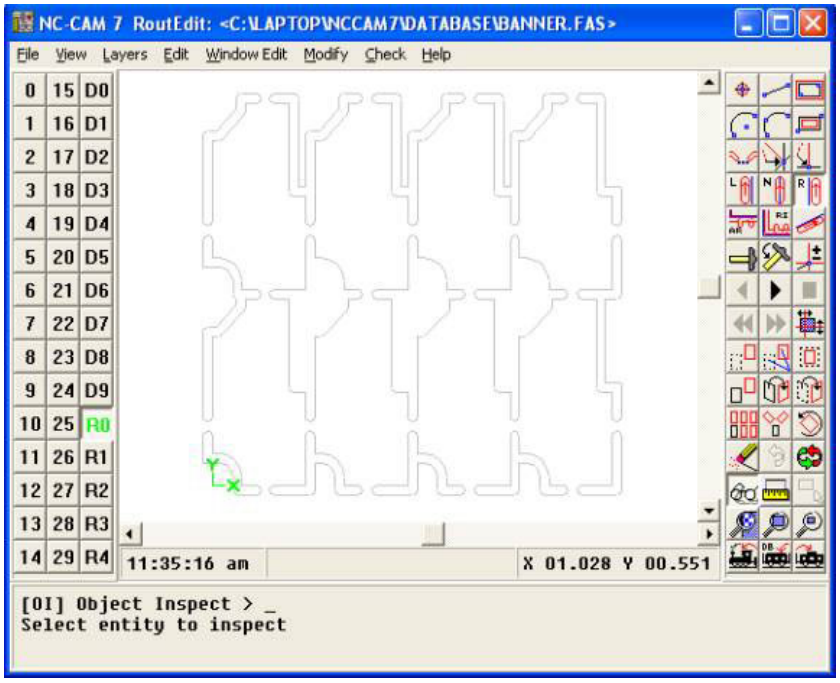
The **XP** *Expand Step & Repeat* function allows you to expand an entity or group of entities that are affected by a Step & Repeat list into the database. This function is useful when there is a need to modify one part in a panel. Since you cannot modify one part that is stepped without affecting all the other parts, you must expand the Step & Repeat list so that each part is routed individually.

When this function is activated, you will be asked to select an entity or group of entities that are affected by a Step & Repeat list. They may be selected by clicking on an entity, or selecting a group of entities by click-dragging a window around them. The steps are immediately expanded and displayed on the screen.

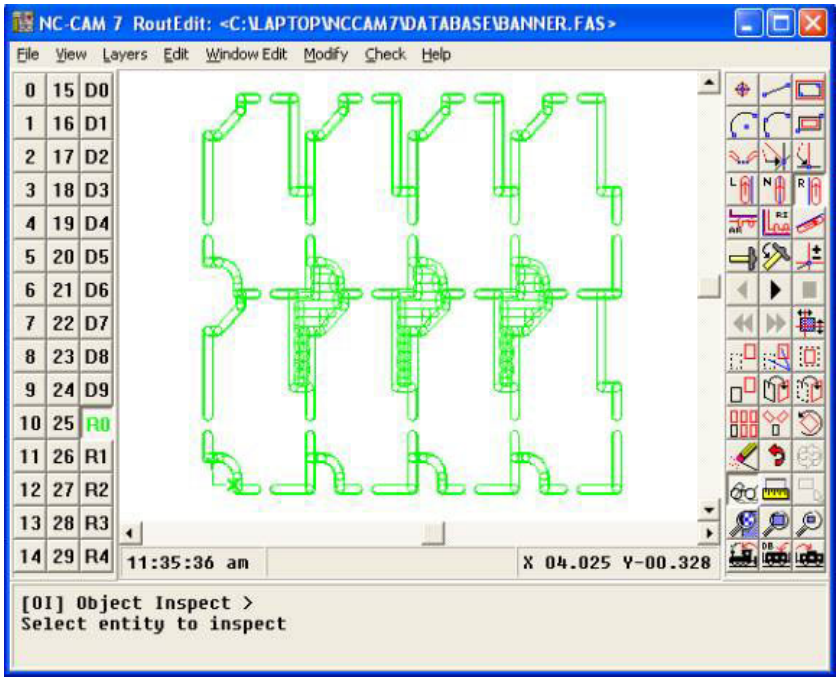
[RT] Full Panel Autorout

The **RT** *Full Panel Autorout* function is used to do pocket routing and create rout programs when the outline of the cutter path is fully defined in a CAD file. *Full Panel Autorout* uses the currently selected cutter and works on all selected active CAD layers. For each closed polygon that *Full Panel Autorout* finds, it creates a fully defined rout path to rout that polygon. If the polygon contains a large enough region in the middle, two passes will be created. The first pass will mill out the middle of all of the polygons. The second pass will cut the outside edges of the polygons.

As an example of what *Full Panel Autorout* is capable of, the following figure illustrates a CAD drawing that fully defines the rout pathways for a board.



After executing *Full Panel Autorout*, the workspace would display the fully defined rout path, as in the following figure.



In order for *Full Panel Autorout* to work, the CAD drawings must follow several rules. The drawing cannot contain multiple paths that the algorithm would have to chose from. The polygons must all be closed off, that is, each polygon must be connected back on itself. Each entity, line or arc, that is in a polygon must be connected at both ends to other entities, with no gaps between them. A polygon cannot cross over itself, for example, in a bowtie shape, or figure-eight. All pathways in the polygon must be at least as large as the cutter diameter that will be used. If any of these rules are broken, the output of *Full Panel Autorout* is undefined. Fortunately, like all NC-CAM 7 commands, *Full Panel Autorout* can be undone simply by executing **OO** *Undo*.

Removing all of the additional information in a CAD drawing before executing *Full Panel Autorout* will yield the best results. Specifically, dimension lines and text must be erased, along with all other data that is not meant to be routed. If a drawing contains multiple pathways or is not fully connected, or has chorded arcs, executing an *Automatic Cleanup* before attempting a *Full Panel Autorout* might take care of all or most of these problems.

[PC] Pocket Rout

The **PC** *Pocket Rout* function allows you to "hog-out" an area of circuit board as described by a closed path in CAD.

When this function is selected, you will be asked to select a CAD entity. The cad entity can be any part of a closed polygon in CAD. Once the CAD entity is selected, the hogged out section of panel will be added to the rout database, and displayed in the workspace.

The *Pocket Rout* function behaves exactly like **RT** *Full Panel Autorout* in all other respects. The most useful aspect of *Pocket Rout* is that extraneous data can still be a part of the CAD drawing, as long as an entity from a polygon which follows all of the rules listed in **RT** *Full Panel Autorout* is selected. In this manner, single polygons can be selected for automatic routing, one at a time.

[GC] G32/G33 Circle

The **GC** *G32/G33 Circle* command will add a routed circle to the rout database, by defining the circle's center point, and secondly, a point on the edge of the circle. Once you define a center point, you'll get a rubber-banded circle that is attached to your cursor, until you define the point on the circle.

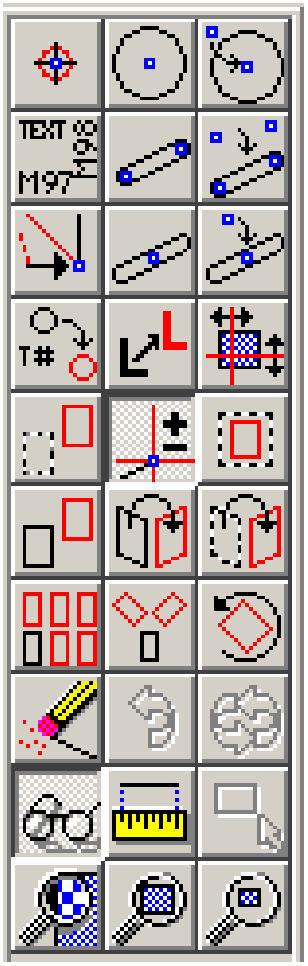
Notice that the routed circle you define is defined just like a G32/G33 command. The radius of the circle specifies the outside edge of the routed circle, not the center.

The direction of the routed circle is determined by the current compensation type. Usually for this function, you want to have the comp type set to Right Compensation, since this produces the cleanest routed edge.

C H A P T E R 7

DrillEdit command reference

This chapter provides a command-by-command reference to the NC-CAM 7 DrillEdit functions.



The functions, in the order presented on the DrillEdit button menu are:

DH *Drill Hole*
- **DC** *Drill Circle*
  - **HC** *Convert Hole To Circle*

DT *Drill Text*
- **DS** *Drill Slot*
  - **CD** *Convert Holes to Slot*

SC *Snap Closest Endpoint*
- **SH** *Single Hole Slot*
  - **HT** *Convert Single Hole To Slot*

TC Window Tool Change
- **LO** *Layer Operation*
  - **DO** *Drawing Origin*

WM *Window Move*
- **MR** *Relative Mode*
  - **SW** *Window Scale*

WC *Window Copy*
- **WI** *Window Mirror*
  - **WF** *Window Flip*

MC *Matrix Copy*
- **RC** *Radial Copy*
  - **RO** *Rotate*

WE *Window Erase*
- **OO** *Undo*
  - **UU** *Redo*

OI *Object Inspect*
- **ME** *Measure*
  - **SA** *Select All*

ZW *Zoom In*
- **ZA** *Zoom All*
  - **ZB** *Zoom Out*

[DH] Drill Hole

The **DH** *Drill Hole* function allows you to drill a single hole in a specified location. It is very similar to the **PO** *Create Point* function in CAD.

This function is primarily used in DrillEdit, however, it is also available in RoutEdit for creating holes that will be stored at output time as G05 holes.

[DC] Drill Circle

The **DC** *Drill Circle* function creates a drilled circle (G84) in the drill database, by defining the circles' center point, and secondly, a point on the edge of the circle. Once you define a center point, you'll get a rubber-banded circle that is attached to your cursor, until you define the point on the circle.

Notice that the drilled circle you define is defined just like a G84 command. The radius of the circle specifies the outside edge of the drilled circle, not the center.

[HC] Convert Hole To Circle

The **HC** *Convert Hole to Circle* function is used to convert a hole in the drill database to a Drilled Circle. When you select the **HC** *Convert Hole to Circle* function, you will notice the blue words "Configure Circle" appear in the user panel. This is where you select the radius of the circle. When you click on the text "Configure Circle", a dialog box will appear with the current radius value.

The **HC** *Convert Hole to Circle* function acts like a snap command. When you select the function, you will need to select the hole which to convert. By clicking in the workspace, you will select the closest active hole and convert it to a circle.

[DT] Drill Text

The **DT** *Drill Text* function creates a drilled text command (M97/M98) in the drill database, by defining the lower left corner of the drilled text's first letter. (Note: if you are adding vertical text, the point selected actually refers to the lower right of the drilled text, due to the 90 degree rotation.) Once the location is selected (by either clicking in the workspace with the mouse or typing the coordinates in the user panel), a dialog will appear asking for the text and the orientation of the text. After selecting OK, the text will appear in the workspace.

Note: The text does not need to be entered with commas separating the words instead of spaces, as is necessary in an Excellon drill program. DrillEdit automatically substitutes the spaces with commas at output time.

[DS] Drill Slot

The **DS** *Drill Slot* function allows you to add a drilled slot (G85) command to the drill database. The slot may optionally be expanded at output time.

When you create a slot, you will be prompted to enter the first endpoint. After you do, you'll notice a "rubber-banded" line connecting the first endpoint to the cursor. When you define the second endpoint, the rubber-banded line will change to a drilled slot in the drill database.

Note that in order to use this function, you must know both endpoints of the slot. If you have a center point, angle and length specification for the slot, use the **SH** *Single Hole Slot* function instead.

[CD] Convert Holes to Slot

The **CD** *Convert Holes to Slot* (Convert Double) function is used to create a slot in the drill database whose length and orientation is specified by two existing holes in the database. When this function is selected, you will simply be prompted for the two holes that specify the slot. Select the two holes that define the slot, and the two holes will be deleted from the database, replaced by a slot.

[SC] Snap Closest Endpoint

The **SC** *Snap Closest* function is used when you're entering a coordinate and you want it to be exactly the same as an existing coordinate in the drawing. To use snap closest, you simply select the function, then move the cursor close to the point you wish to snap to, and press your left mouse button.

There's a similar function that's actually used more often than **SC** by experienced CAD users, and that is the **NP** *Near Point* function. Unlike the **SC** function, **NP** does not require that you click your left mouse button. Instead, it snaps immediately when you enter **NP** on the keyboard. This immediate snap, and the fact that you don't have to move your mouse over to the menu and then back to the desired point, save a considerable amount of time. Obviously, there's no point in having a menu button for **NP**, because you'd have to have your mouse on the menu and not near the snap point in order to press the button! An alternate way to select **NP** is to click the middle mouse button, if you have a three-button mouse.

There's one aspect of **SC** and **NP** that's not obvious to first-time users: There are more snappable points than you can ordinarily see on the screen. This is because both arcs and circles have construction points at their centers. These points become visible only when you turn on the ***V**iew*|***C**onst Points* function, but they may always be used for snapping.

If you have a 3-button mouse on your computer, the middle button works as an **SP** *Snap*. Simply move the cursor near a point and press the middle mouse button once to snap to the point.

[SH] Single Hole Slot

The **SH** *Single Hole Slot* function is used to add a drilled slot (G85) command to the database. To add a slot to the database using this function, you must know the center point of the slot, it's orientation (in degrees) and the length of the slot. If the slot specification you have specifies two endpoints instead, use the **DS** *Drill Slot* command instead.

When you select the **SH** *Single Hole Slot* function, you will notice a blue "Configure Slots" prompt in the user panel. Use your mouse to click on this prompt to enter the orientation and the length of the slot. Once you have entered the orientation and length, select OK.

You will then be prompted for the center point of the slot. You may either key in the coordinates in the user panel or select the location using the cursor in the workspace. Once the point is selected, the drilled slot will appear as it is added to the drill database.

[HT] Convert Single Hole To Slot

The **HT** *Convert Single Hole To Slot* function is used to add a drilled slot (G85) command to the database, using an existing hole as the center location. To add a slot to the database using this function, you must know the slot's orientation (in degrees) and the length of the slot. If the slot specification you have specifies two endpoints instead, use the **CD** *Convert Two Holes to Slot* command instead.

When you select the **HT** *Convert Single Hole To Slot* function, you will notice a blue "Configure Slots" prompt in the user panel. Use your mouse to click on this prompt to enter the orientation and the length of the slot. Once you have entered the orientation and length, select OK.

You will then be prompted for the holes specifying the center point of the slot. Use the cursor to select the drilled hole you wish to convert. Once the hole is selected, the hole will be deleted, and a drilled slot will appear as it is added to the drill database.

[TC] Window Tool Change

The **TC** *Window Tool Change* function allows you to select multiple drilled holes and change them all to one tool number.

When this function is selected, it will ask for a new tool number. Once the tool number is selected, you may select any entity or group of entities which will be immediately changed, and the display will show the altered holes. Selection of entities is identical to the selection method for *Window Erase*.

[LO] Layer Operation

The **LO** *Layer Operation* function allows you to move entities from one layer to another. This function may be used in the following two ways:

1) When this button is pressed by itself, the function acts like **WE** *Window Erase*, with one difference. The selected entities are not erased, they are merely moved to the layer specified in the user panel. The selection of entities, however, is exactly the same as **WE** *Window Erase*. You can use *Layer Operation* to move either single entities or windowed groups of entities. This window function does not, however, allow you to select multiple entities using the shift-click approach described in the **WM** *Window Move* function: When you select an entity by clicking on it, or a group of entities with a click-drag window, they are moved immediately.

2) The *Layer Operation* function may be applied to any window edit function as well. To do so, first pick the operation you wish to perform. Then select *Layer Operation*. The result of the operation, instead of landing on the construction layer, will be placed on the layer selected in the user panel instead. Notice that doing a *Window Move* still requires you to select the offset when using this function.

[DO] Drawing Origin

The **DO** *Drawing Origin* command moves all of the data in your drawing relative to the drawing origin. This is useful for temporarily setting the drawing's origin during editing, as well as finally establishing the zero point for routing.

[WM] Window Move

The **WM** *Window Move* command allows you to click on an entity, or click-drag a window around a group of entities, and then move them.

The process of establishing the from/to positions for the entities to move is exactly the same as creating a line. This means you can use the same tricks you'd use in making a line to determine the exact positions the window moves from and to.

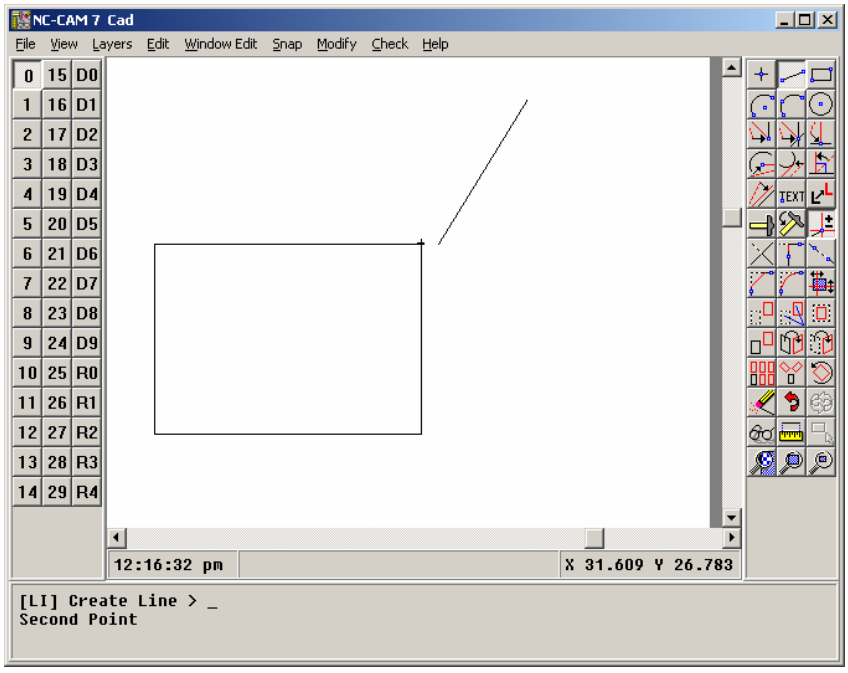
As with many CAD functions, there's a hidden feature in the window move, as well as all of the other window functions which follow: When you select a window full of objects, they become highlighted on the screen. If you want to add selected entities to the group to be changed, hold down your **SHIFT** key and click on the entities you want to add. You'll see them become highlighted, indicating that they're now included in the group you windowed!

This "shift-click" function also works in reverse: If your window accidentally selected an entity you did not want to include in the group, use shift-click to deselect it. You'll see the entity un-highlight, to indicate that it is no longer a member of the selected group.

[MR] Relative Mode

The **MR** *Relative Mode* toggle affects the way coordinates you enter are interpreted. Ordinarily, all of the coordinates you enter are relative to the zero point of your drawing. Sometimes, however, it's advantageous for you to be able to enter a coordinate as a distance from the previous coordinate. That's what the **MR** *Relative Mode* toggle allows you to do.

Assume for a moment that you have a dimension on a drawing that's given as being one-half inch to the right of the upper-right corner of a rectangle. You could either inspect the rectangle's upper right corner, write down its coordinate, add half an inch in the X axis, and type in the resulting coordinate, or you could use the **MR** mode to enter the point.



As shown above, if the point you most recently created is not the point you've got to enter a coordinate relative to, just use **NP** or **SC** to snap the start of a new line to your desired reference point, then press **Esc**. Turn on **MR**, and the numbers you enter will be relative to that reference point.

There's one added point worthy of special mention here: It's best to make sure you turn off the **MR** *Relative Mode* as soon as you're done using it. Forgetting that you're in relative mode can lead to some frustration when you later enter coordinates, and discover that they're not ending up where you intended them to be.

[SW] Window Scale

The **SW** *Window Scale* function allows you to stretch or shrink the windowed entities by the factor you enter, referenced from a point you select.

Window Scale can alter the windowed group differently in the horizontal and vertical directions, if you wish. If you do scale a window asymmetrically, the circles and arcs receive special treatment: Circles remain circular when scaled asymmetrically, but the radius of each circle will be altered by the product of the two scale factors. This means that if you scale a window by 2.0 in the horizontal axis, and 0.3 in the vertical axis, the radii of any circles you've selected will be scaled by 2.0 times 0.3, resulting in a scale of 0.6:1.

Arcs, on the other hand, actually are scaled asymmetrically. The treatment given to the arcs follows this procedure: First each arc has an added point computed for it, the point being the arc's midpoint (see the **SN** function for a definition of this). Then, the arc's midpoint, start point, and end point are scaled according to the two different scale factors you've selected. Finally, the arcs are reconstructed, using the math of the **A3** *Three-Point Arc* function to reestablish each arc using the three scaled points. This rather complex process does allow for asymmetrical scaling of arcs, albeit with arc centers that move as necessary to maintain equal start and end radii.

[WC] Window Copy

The **WC** *Window Copy* function allows you to copy all of the selected entities. After you select the entity or entities to be copied, you define the point to copy from, and the point to copy to. The "from" and "to" points may be entered as you'd enter any coordinate, but usually you'll want to use a snap or key in the values.

[WI] Window Mirror

The **WI** *Window Mirror* function copies all of the selected entities, mirroring around an arbitrary axis line. This powerful function is very handy for creating copies of individual entities at odd angles.

It's probably best for you to experiment with the **WI** *Window Mirror* function a bit before you use it in a live drawing. The mirror around an arbitrary line tends to be a little confusing, until you realize that the arbitrary line acts just like a "fold" in your drawing. A reasonable metaphor for the function would be that the windowed entities are "folded" around your arbitrary line.

[WF] Window Flip

The **WF** *Window Flip* function moves all of the selected entities, mirroring around an arbitrary axis line. This powerful function is very handy for moving entities to odd angles.

It's probably best for you to experiment with the **WF** *Window Flip* function a bit before you use it in a live drawing. The mirror around an arbitrary line tends to be a little confusing, until you realize that the arbitrary line acts just like a "fold" in your drawing. A reasonable metaphor for the function would be that the windowed entities are "folded" around your arbitrary line.

[MC] Matrix Copy

The **MC** *Matrix Copy* function produces from one to ninety-nine copies of the selected entities in the X and Y direction, evenly distributing the copies with spacing you specify.

[RC] Radial Copy

The **RC** *Radial Copy* function produces from one to ninety-nine copies of the selected entities, evenly distributing the copies across the angle you specify, around the center point you specify. The icon for this function depicts a radial copy with a quantity of three, an angle of 360 degrees, and a center point located just above the bottom rectangle.

[RO] Rotate

The **RO** *Rotate* function simply rotates all of the selected entities around a point. The function is extremely useful for working on sections of drawings which are dimensioned off-axis. By using the **RO** function, you can rotate your entire drawing to place the needed section on-axis, do the editing you need to do, and then use **RO** a second time to rotate the entire drawing back to its original alignment.

[WE] Window Erase

The **WE** *Window Erase* function erases single entities, or windowed groups of entities. This is the only window function which does not allow you to select multiple entities with the shift-click approach described in the **WM** *Window Move* function: When you select an entity by clicking on it, or a group of entities with a click-drag window, they are deleted immediately.

Because the windowing functions in NC-CAM's CAD editor also allow you to select an individual entity by doing a simple click (instead of a click-drag), this function is also used for individual deletes. This replaces the **OE** *Object Erase* function GCADD users will be familiar with.

[OO] Undo

The **OO** *Undo* function allows you to reverse the effects of your editing, one step at a time, back to the beginning of your session. If you place your cursor over the *Undo* icon, the help text in the Status Window will describe the function you're about to reverse.

Note that when you pack the database (with the ***F**ile/Pac**k** Database* function), the *Undo* function's memory is erased.

[UU] Redo

The **UU** *Redo* function reverses the effect of the **OO** *Undo* function, one step at a time. This button will be "grayed out" until you've pressed the **OO** *Undo* button. As with the **OO** *Undo* icon, if you place your cursor on the **UU** *Redo* icon, the help text will describe the function you're about to *Redo* if you press the button.

[OI] Object Inspect

The **OI** *Object Inspect* function allows you to select any entity in the drawing, and view the coordinates of the points making up the entity. Further, once you've selected an entity with **OI** *Object Inspect*, you can modify the entity by clicking on any of its defining values, and typing in new values. When you do this, the highlighted entity will instantly change on-screen, and turn red in color. If you want to restore the entity to its original condition, press **ESC**. To keep any change you've made, press **ENTER**.

[ME] Measure

The **ME** *Measure* function allows you to measure from any construction point in your drawing to any other. Further, you may measure angles by defining a baseline with your first two clicks, and an angle with the third mouse click.

Once you've clicked on three points, you may "pick up" the endpoints by clicking on them a second time. To reverse the angle reading in the measure mode, click your mouse near the apex of the angle.

[SA] Select All

The **SA** *Select All* button is unavailable until you've selected any of the window functions. With a window function selected, **SA** will automatically select all of the entities in your drawing.

There's a special capability available with the **SA** *Select All* function. If you want to select everything in your drawing except what's inside a window, follow these steps: First, select the window function you want. Second, window the group you want excluded from the window operation, and they'll highlight. Last, hold the **SHIFT** key and press the **SA** *Select All* button. This will reverse the selections you've made. You may then proceed with the remaining steps of the window function you originally selected.

[ZW] Zoom In

The **ZW** *Zoom Window* function allows you to window a portion of the screen you want to zoom in on. The creation of this window is identical to the process of creating a rectangle. After the zoom region is specified, the work area is redrawn, focusing in on the area you selected.

The counterpart to this function is the **ZB** *Zoom Back* function, which can be used to decrease the magnification.

A useful new feature of NC-CAM is the ability to pan with a simple **ALT**-click-drag combination. If the display was magnified to show the upper-left-hand portion of your drawing, and you wanted to "slide" the view over towards the upper-right-hand side of the drawing, the new pan feature can be used. Hold down the **ALT** key, left-click with the mouse on the right-hand side of the drawing, and drag that location to the left-hand side of the drawing. In essence, the new pan feature "picks up and drags the work area," just as you would move a sheet of paper to center in front of you the area that you wanted to work on.

To repeatedly pick up and move the drawing in this manner, keep holding down the **ALT** key, let go of the left mouse button, drag the mouse to the opposite side of the drawing, left-click the mouse, and drag back. Advanced users of NC-CAM will learn the click-drag, click-drag, click-drag motion and grow to use it more often.

[ZA] Zoom All

The **ZA** *Zoom All* function redraws the screen at a scale which will show all of the entities in your drawing.

[ZB] Zoom Out

The **ZB** *Zoom Back* function reduces the magnification of the screen by a factor of two.

The counterpart to this function is the **ZW** *Zoom Window* function, which can be used to increase the magnification.

[PA] Pan

The **PA** *Pan* function, although now mostly obsolete, is included to keep backwards compatibility with NC-CAM 5. This function is replaced by the new scheme whereby you hold down the **ALT** key, and depress the left mouse button to pan the screen.

With this pan function, however, you will be asked to select a new center for the screen. When you click the mouse, the location you select will be the new center of the visible viewing area, with the same zoom factor.

[OE] Delete Entity(s)

OE *Delete Entity(s)* is the same function as **WE** *Window Erase*.

The **WE** *Window Erase* function erases single entities, or windowed groups of entities. This is the only window function which does not allow you to select multiple entities with the shift-click approach described in the **WM** *Window Move* function: When you select an entity by clicking on it, or a group of entities with a click-drag window, they are deleted immediately.

Because the windowing functions in NC-CAM's CAD editor also allow you to select an individual entity by doing a simple click (instead of a click-drag), this function is also used for individual deletes. This replaces the **OE** *Object Erase* function GCADD users will be familiar with.

[NX] Number in X

The **NX** *Number in X* function sets the number of copies made in the X direction when using the **MC** *Matrix Copy* function.

[NY] Number in Y

The **NY** *Number in Y* function sets the number of copies made in the Y direction when using the **MC** *Matrix Copy* function.

[XS] X Increment

The **XS** *X Increment* function sets the distance between copies made in the X direction when using the **MC** *Matrix Copy* function. You can not reach this command directly, it is only a part of the **MC** *Matrix Copy* command.

[YS] Y Increment

The **YS** *Y Increment* function sets the distance between copies made in the Y direction when using the **MC** *Matrix Copy* function. You can not reach this command directly, it is only a part of the **MC** *Matrix Copy* command.

[TD] Total Degrees

The **TD** *Total Degrees* function sets the total degrees used during a **RC** *Radius Copy*.

[NC] Number of Copies

The **NC** *Number of Copies* function sets the number of copies made during a **RC** *Radius Copy*.

[RD] Redraw

The **RD** *Redraw* function redraws the workspace area. This is especially useful if you have deleted several entities, and the deleted entities have "covered" part or all of some entities that remain.

[NP] Snap Near Point

The **NP** *Snap Near Point* function, although only available from the keyboard, is one of the most useful snap functions in the CAD snap set. It allows the user to make "perfect" drawings where there are no gaps left in the corners of a board profile that might be left if you use freehand drawing.

The **NP** *Snap Near Point* function requires only that you move the mouse near the point you want to snap on and then type **NP**. If you have a 3-button mouse, clicking on the middle button does the same thing as type **NP**.

[MO] Absolute Mode

This command turns off relative mode which can be used to enter coordinates in the user panel. This function turns off the relative mode regardless of the state of the **MR** *Relative Mode* button.

[OC] Copy

The **OC** *Copy* function in the same as the **WC** *Window Copy* function.

The **WC** *Window Copy* function allows you to copy all of the selected entities. After you select the entity or entities to be copied, you define the point to copy from, and the point to copy to. The "from" and "to" points may be entered as you'd enter any coordinate, but usually you'll want to use a snap or key in the values.

[WO] Window Delete Outside

The **WO** *Window Delete Outside* function behaves much like the **WE** *Window Erase* function, with the exception that the entities deleted are the entities outside of the click-dragged window, not the entities on the inside.

[LR] Target Layer

The **LR** Target Layer function allows you to set the layer where **LO** *Layer Operation* functions will land.

[SR] Step & Repeat

The **SR** *Step & Repeat* function allows you to automatically create a Step & Repeat pattern for panelization. The inputs for this function are the number of steps in the X and Y direction, and the distance between the parts.

When this function is selected, you will be asked to select an entity or a window. Typically you would select the entire part. Once the part is selected, you may shift-click to include or exclude different holes, in the same fashion as the window operations in CAD.

Once you have selected the appropriate holes for the Step & Repeat pattern, you will be asked to confirm the operation by either hitting **ENTER** or clicking in the workspace. When you confirm the operation, you will see the dialog box that allows you to set the matrix parameters for the Step & Repeat pattern.

The NESTED STEPS checkbox allows you to specify the format of the Step & Repeat list. If you select nested steps, the resulting steplist will contain an M01, the list of steps in the X direction, another M01, and a list of steps in the Y direction. If you deselect this checkbox, the step list will explicitly list all the steps, with only one M01.

When OK is selected, the Step & Repeat list is added to the DrillEdit database and the steps are shown on-screen.

[SE] Step & Repeat Edit

The **SE** *Step & Repeat Edit* function allows you to edit a Step & Repeat List already in the database. After you have finished the edit, the changes will immediately show up on the screen. You will note that the separate nesting levels are represented by using multiple edit windows separated by a vertical bar.

[SD] Step & Repeat Delete

The **SD** *Step & Repeat Delete* function allows the user to delete an existing Step & Repeat pattern from the database.

When the function is selected, it will ask for either an entity or a group of entities which may be selected using the click-drag method. Once the entity or group of entities are selected, all the Step & Repeat information associated with those entities is immediately deleted.

The results of the delete will be immediately displayed.

[CS] Create Step & Repeat

The **CS** *Create Step & Repeat* allows you to interactively create a Step & Repeat list in an edit window.

When the function is selected, like **SR** *Step & Repeat*, the function will ask you for an entity or group of entities to which to apply the Step and Repeat list. Once you have selected the entities, you may add or delete entities by shift-clicking on entities.

Once you have established the group of entities to which you wish to apply the function, you need to confirm the operation by clicking in the workspace or pressing **ENTER**. Once you have confirmed the operation, an edit screen will appear where you can enter the Step & Repeat list. Once the Step & Repeat is entered, and the OK button is selected, the Step & Repeat list is applied to all the selected entities, and displayed in the workspace.

[AS] Add to Step & Repeat

The **AS** *Add to Step & Repeat* function is used to add a hole or set of holes to an existing Step and Repeat list. This function is especially useful when adding holes to a part that has already been panelized using Step & Repeats.

When the function is selected, it will ask you to select either an entity or group of entities by clicking on an entity or click-dragging a window around a group of entities. You may add or subtract entities from the selected group by shift-clicking on the entities. Once you have selected the group on which to apply the operation, you must select a stepped entity which has the appropriate Step & Repeat list to copy.

[XP] Expand Step & Repeat

The **XP** *Expand Step & Repeat* function allows you to expand an entity or group of entities that are affected by a Step & Repeat list into the database. This function is useful when there is a need to modify one part in a panel. Since you cannot modify one part that is stepped without affecting all the other parts, you must expand the Step & Repeat list so that each part is routed individually.

When this function is activated, you will be asked to select an entity or group of entities that are affected by a Step & Repeat list. They may be selected by clicking on an entity, or selecting a group of entities by click-dragging a window around them. The steps are immediately expanded and displayed on the screen.

[HS] Configure Single Hole Slot

This function configures what a slot will look like when you create a slot using **SH** *Single Hole Slot*. When this function is selected a dialog will appear with the angle of the slot, and the length of the slot.

[CE] Configure Single Hole Circle

This function describes what a drilled circle created by **HC** *Convert Hole to Circle* will look like. The only thing to configure here is the radius of the drilled circle.

A P P E N D I X S

Appendix A
In case of difficulty

This appendix provides information on solving problems one might encounter when installing or configuring NC-CAM.

READ.ME for NC-CAM 7

Be sure to read the READ.ME that is included with your NC-CAM distribution!

IMPORTANT INSTRUCTIONS for first time users:

You must install lock drivers to allow this software to talk to the software security lock. If you fail to do this, the software will not run. The instructions follow below:

Windows NT Users only:

1. Under the Microsoft Windows NT Main group, double click on "Command Prompt".

2. Change drive to the drive containing the NC-CAM 7 files. Change directory to WIN_NT subdirectory. The current path should now be something like C:\NCCAM6\WIN_NT>

3. Type "INSTALL.BAT" at the command prompt. There are two command line options: (See the INSTALL.BAT file for examples.) 1. /q Quiet mode. Normal dialogs described below are not displayed. Error messages are displayed. 2. /pxxx Path, where xxx is the path of files to be installed. Specify the path of files to be installed. Otherwise, files will be copied from the default directory.

4. A window with the title bar "Sentinel Driver Setup Program" is displayed.

5. Select "Functions" and then "Install Sentinel Driver" from the menu bar.

6. A dialog box with the default path for the NT driver is displayed. Change the drive letter if necessary and click "OK".

7. The Sentinel Driver and associated files are copied to the hard disk. One of the DLLs, SNTI386.DLL, SNTMIPS.DLL, SNTALPHA.DLL, or SNTPPC.DLL and SENTTEMP.HLP are copied to \%SYSTEMROOT%\SYSTEM32. SENTTEMP.SYS is copied to the file \%SYSTEMROOT%\SYSTEM32\DRIVERS\SENTINEL.SYS. %SYSTEMROOT% is the directory where Microsoft Windows NT has been installed.

8. If the driver installation is successful, a dialog box with the message "Sentinel Driver Files Copied Successfully" is displayed.

9. When complete, a dialog box with the message "Driver Installed! Restart your system" is displayed.

10. Click "OK" to continue.

11. Restart your computer.

Windows 95/98/2000:

1. Start Windows. Select “Run” from the Taskbar and run the file SENTW95.EXE in the \NCCAM6\WIN_95 (98 or 2000) subdirectory on the drive where NC-CAM 7 is installed. There are two command line options: 1. /q Quiet mode. Normal dialogs described below are not displayed. Error messages are displayed. 2. /pxxx Path, where xxx is the path of files to be installed. Specify the path of files to be installed. Otherwise, files will be copied from the default directory.

2. Select “Install Sentinel Driver” from the “Functions” menu.

3. Click “OK” when the “Driver installed! Restart your system.” message appears. Restart Windows.

4. The following files have been created on your hard disk:
WINDOWS\SYSTEM\SENTINEL.VXD
WINDOWS\SYSTEM\RNBOSENT\SENTW95.EXE
WINDOWS\SYSTEM\RNBOSENT\SENTW95.DLL
WINDOWS\SYSTEM\RNBOSENT\SENTW95.HLP
WINDOWS\SYSTEM\RNBOSENT\SENTINEL.SAV

General equipment requirements

NC-CAM 7 is a Windows Native 32-bit program. You need to have either Windows NT (3.5 and up) Windows 95, 98 or 2000 to run this software.

The recommended system(s) for NC-CAM are as follows:

Windows with the latest Service Packs installed. (Please see the note below on service packs). A Pentium 90 or faster. 32 megs of RAM.

The minimum platform for Windows is a 486-DX2-66 with 32 megs of RAM.

Notice that the above requirements are the requirements that Microsoft sets for a machine as an **absolute** minimum to run the operating system. We do NOT recommend attempting to run NC-CAM on a system configured as such, since the speed will greatly suffer.

Please use Microsoft’s Web Site, http://www.microsoft.com, to download the latest service packs. These service packs fix operating system bugs, which may cause an application to behave incorrectly. When you call for Tech Support from FASTechnologies, we will ask you to install the latest service packs in order to aid us in correctly diagnosing your problem.

Lock authorization

Please contact FASTechnologies for authorization codes for this product.

Appendix B
Frequently-asked questions

This appendix provides a list of frequently-asked questions and their answers.

Q: Why doesn't NC-CAM recognize the lock?

Plug your FASTechnologies lock into your printer port first, before any locks for other software products.

Q: How do I make sure my drill and rout output register to each other?

The best way to output drill and rout programs so that they register together is to set the zero points you use in Drill, Rout, and CAD to the zero point of your drill/routing machines. This eliminates the need for G93 offsets and simplifies programming and troubleshooting of finished files.

For this example, start in the lower left corner of the panel:

1) Program drill hits using the default position of the X-Y origin as the zero-zero of the print data.

2) Panelize the data to the desired number up using Auto Step & Repeat.

3) Draw a single board image in CAD

4) Move the origins of the Drill and CAD functions to the drill/routing machine zero-zero. For drill and rout, move Job Datum to zero and then set G93 offset to 0,0 by using Panel Origin command. For CAD, move Drawing Origin to the machine zero. At this point the CAD outline should register to the drill data matching the information on the engineering drawing. It should look like a finished PCB.

5) Use DXF CAD outline of board to program the rout for a single board. The co-ordinates of the rout data will be absolute when measured from the machine origin point with no offset.

6) Use Auto Step and Repeat to multiply the rout images.

7) Output drill file using Make Drill Tape command with Output G93 Offset turned off.

8) There are options in the view menu to allow viewing of drill, rout, and cad together in the Rout and CAD portions of the software.

Q: How do I read old Excellon Rout programs into CAD?

You can Import an Excellon rout file into the CAD portion of NC-CAM 7 to create a DXF file of the board outline. The Excellon data is automatically compensated and changed into an editable DXF file.

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