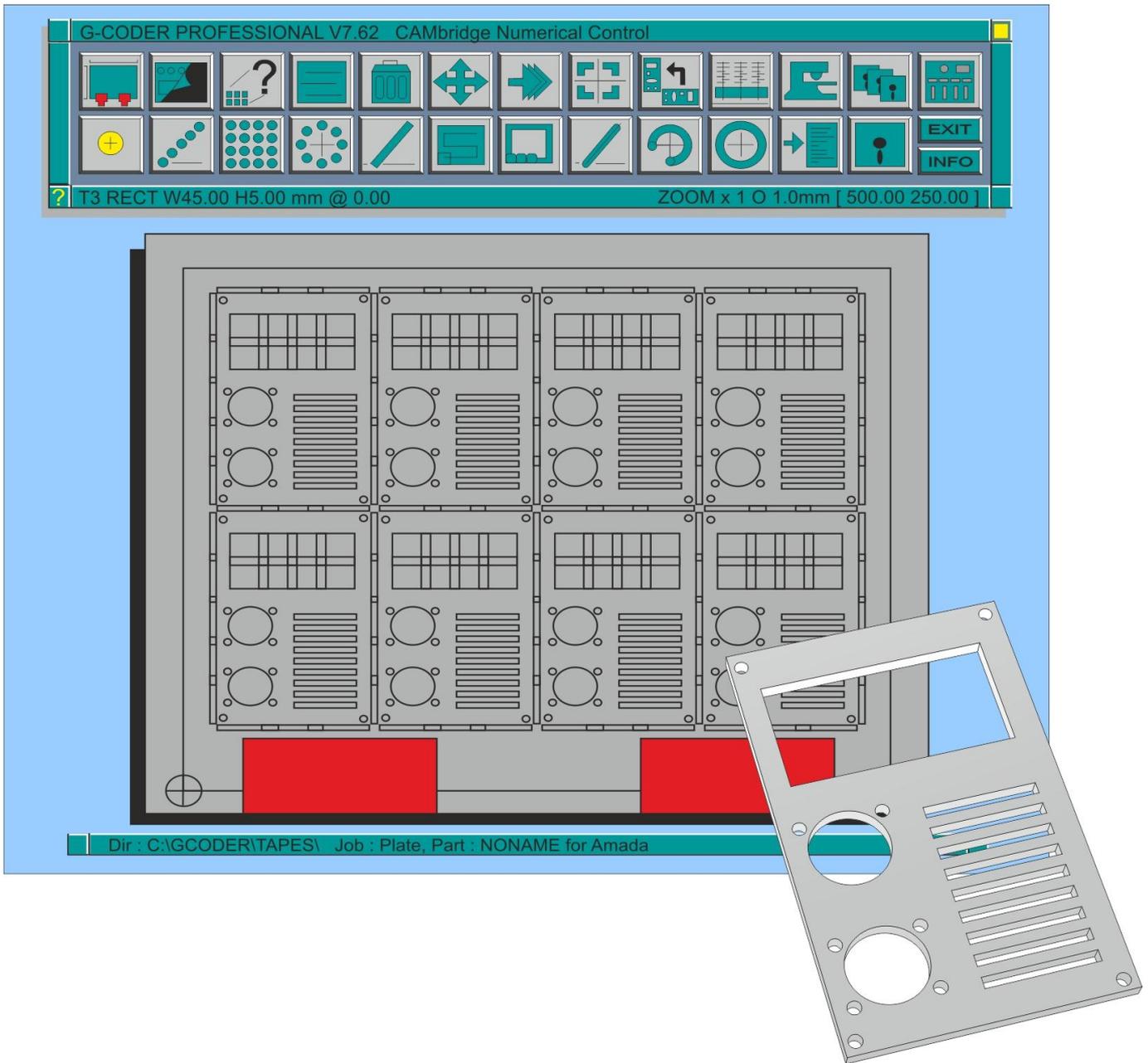


G-CODER Professional Reference Manual



Cambridge
Numerical Control



Driving Productivity Forward

Units 8 & 9 Royce Court
Burrell Road
St. Ives Cambs
PE27 3NE
Tel +44 (0)1480 468639
Fax +44 (0)1480 301577
Specialists in CAD/CAM & DNC
Established since 1981

Contents

SCREEN LAYOUT	5
HOW TO USE THE ICONS AND POP UP WINDOWS	6
SHEET CONVENTIONS	8
PUNCHING ON THE SHEET	8
CALCULATING CO-ORDINATE ENTRY VALUES	8
LIST OF ACTION KEYSTROKES	10
SCREEN DISPLAY	10
PUNCH SELECTING	10
PUNCH ACTION POSITIONING	10
PUNCH MODE	11
MIRROR COMMAND	11
MOVE COMMAND	11
TOOL PRIORITIES	12
SELECTING PUNCHES (QUERY, DELETE, MOVE, COPY, MIRROR AND ROTATE).....	13
THE ICON BOX.....	14
SHEET PARAMETERS	14
Comments Button	15
Printing Button	15
REDRAW	16
Redraw Mode : FULL SPEED / SLOW / SINGLE STEP	16
By Tool Priority : ON / OFF	16
DXF Geometry : ON / OFF	16
ZOOM IN / OUT	16
PAN	17
QUERY PUNCH	18
Punch Mode	18
Split row / grid	19
Split line	19
Geometry Mode	19
SORT	20
Manual	20
Next Nearest	20
Sort in X.....	20
Sort in Y.....	20
Sort by AI Angle.....	20
DELETE.....	21
MOVE (AND ROTATE)	21
COPY AND MULTI-PART	23
Creating a multi-part	23
Editing punches within the Multi-Part.....	23
Editing the multi-part itself (number of copies, pitch etc.)	24
Deleting a multi-part without deleting the original punches.	24
Adding a new punch to any existing multi-part	24
Redrawing of multi-parts.....	24
Post processing of multi-parts.	24
Addendum for Amada users	25

G-CODER Professional Reference Manual

MIRROR	26
Example (Corner Notching).....	26
ROTATE	30
Normal stations	31
Auto-index stations.....	33
TURRET	35
Turret Layout	35
Turret Station Size Checking	35
Change Button	36
Standard Tool Types.....	38
Special Tool Types.....	42
SWAP Button	44
SAVE Button	44
LOAD Button.....	44
MACHINE TYPE.....	45
PARTS	46
Material Type and Thickness	46
Part Examples.....	47
Append Part	48
Save Part	48
Save As Part	48
Delete Part File	49
Directory.....	49
Nest New Parts / Repeat Last Nest.....	49
PUNCH OPERATIONS	50
PUNCH MODE AND GEOMETRY MODE	50
CHANGING BETWEEN PUNCH MODE AND DXF GEOMETRY MODE.....	50
PUNCHING DXF GEOMETRY.....	51
AUTO-INDEX STATIONS	51
AUTO-PUNCH	52
SINGLE PUNCH.....	55
Punch	55
Move	55
Ref.P	55
Prog Stop	56
Auto-index.....	56
Geometry Mode	56
PUNCH ROW	58
Auto-index.....	59
Geometry Mode	59
PUNCH GRID	60
Auto-index.....	61
Geometry Mode	61
PUNCH ARC OR BOLTHOLE.....	62
Auto-index.....	63
Punching a complete arc/circle with Auto-index	63
Geometry Mode	63
PUNCH LINE	64
Auto-index.....	65
Geometry Mode	66
CUT-OUT TO SCRAP	67
Geometry Mode	67
PUNCH CUT-OUT (INSIDE OR OUTSIDE)	69
Geometry Mode	70
NIBBLE LINE	71
Pitch.....	71
Compensation LEFT/RIGHT : LEFT/ONLINE/RIGHT	72
Length : INCREASE/EQUAL/DECREASE	72

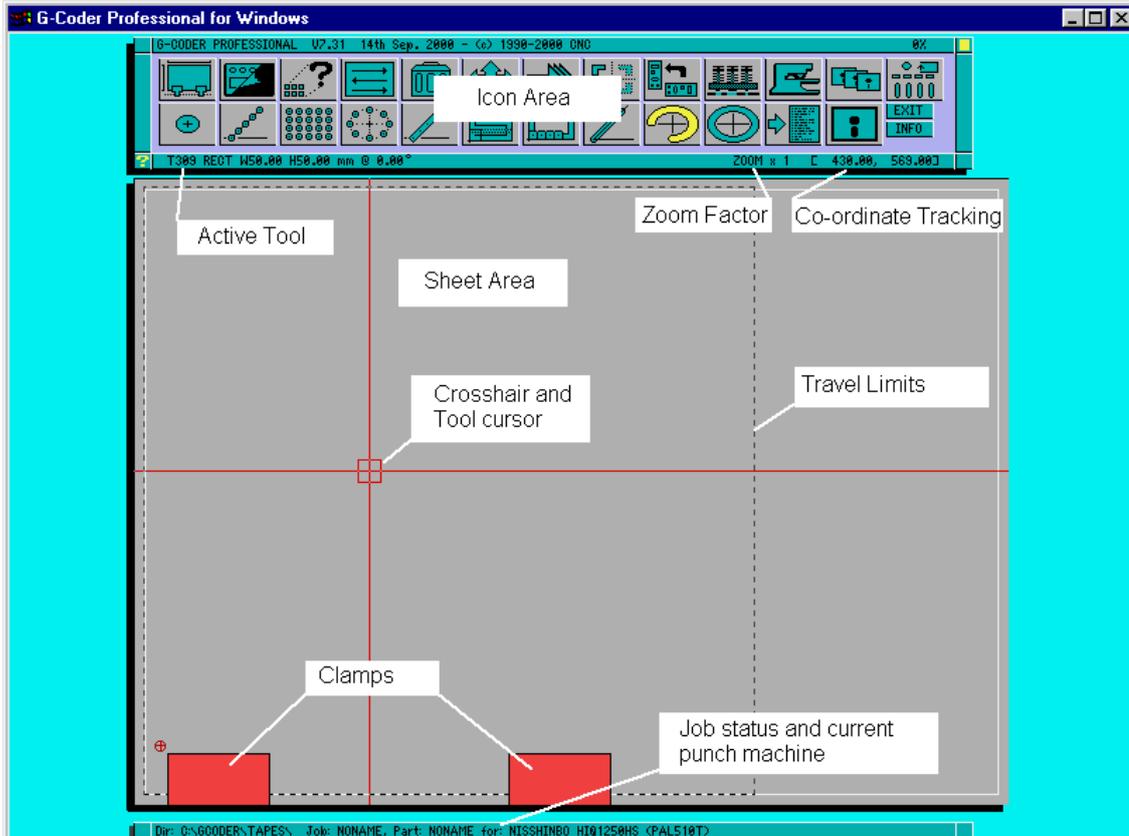
G-CODER Professional Reference Manual

Tagging ON/OFF : NONE/DEFAULT/VALUE	72
Geometry Mode	72
Splines/ Polylines and Ellipses.....	72
NIBBLE ARC.....	73
Start Angle :	74
End Angle :	74
Direction : CLOCKWISE/ANTI-CLOCKWISE	74
Radius :	75
Compensation : INSIDE/ONLINE/OUTSIDE	75
Length INCREASE/DECREASE : INCREASE/EQUAL/DECREASE	75
Hints	75
Geometry Mode	76
Nibble arcs with rectangular tool in Auto-index station	77
CIRCULAR CUT-OUT	78
Geometry Mode	80
UNDO – PRESS U ON SHEET	81
POST PROCESSOR.....	82
Program number	82
Debug Lines.....	82
Line Numbers / Block numbers	82
Decimal points.....	82
Sort by Tool	83
SHEET RE-POSITIONING.....	83
FILE ICON.....	88
Load JOB File	88
Save JOB File	89
Save as JOB File	89
Delete JOB File.....	89
New JOB	89
Load GXF File	90
Save GXF File	90
Save as GXF File	90
Delete GXF File.....	90
Directory.....	90
Read DXF File – read drawing files created by a CAD system	90
Write DXF	90
AUTOMATIC DISSIMILAR PARTS NESTING	92
Quantity.....	92
Angle Inc / End Angle / Mirror	92
Priority.....	92
Filler Parts.....	93
Tool Dia	93
Part Gap	93
Dir.	93
Efficy.....	94
Refining the nest	94
Saving and post processing the final nest	94

G-CODER Professional Reference Manual

Screen Layout

The G-CODER Professional screen looks like the one below and can be split into three sections; the icon area and status information, the sheet area and the machine information line showing current directory and job.



The icon area contains a number of icons and two buttons (Exit and Info). To select any of the icons, simply move the mouse arrow cursor over the top of the icon and click either mouse button. For example, move the arrow cursor onto the bottom left icon (single punch), click the mouse button and the icon is now highlighted. Notice that the status line indicates the current tool station in use and the tool type and size in that turret.

The sheet area is where the punches will be displayed. If you move the mouse cursor into this area you will see that it changes from an arrow cursor into a cross-hair cursor. The co-ordinate values at the right of the status information line reflect the current X and Y co-ordinate values of the cross-hair position. If you hold down the shift key and click either mouse button in this area you will perform a punch on the sheet.

Superimposed on the cursor is the current tool shape drawn to scale and at the current angle. This helps you to position punches by eye when, for example, knocking out the inside of an irregular shape.

Finally, the job and machine information line at the bottom of the screen gives you the name of the current directory, the current job (NONAME if you have just come into the program), the name of the current part and the machine type, with its associated controller in brackets.

G-CODER Professional Reference Manual

How to use the icons and pop up windows

The icons can be split into two categories: those that set up a punch action (mostly the bottom line of icons) and those which perform an immediate action (such as change sheet size). Some of these icons require that you enter information into a pop up window and this can be done in a number of ways. Let us look at the sheet parameters icon (top left). Move the cursor over the icon and click either button and a window like the one shown below will appear:

SHEET, CLAMPS AND DATUM	
Sheet Units :	MM <input checked="" type="checkbox"/>
Sheet Material :	
Sheet Thickness :	0.00
Sheet Width :	1000.00
Sheet Height :	500.00
Clamp 1 (X) :	150.00
Clamp 2 (X) :	750.00
Clamp Width :	200.00
Clamp Height :	80.00
Clamp Backoff :	2.00
Virtual Datum X :	0.00
Virtual Datum Y :	0.00
Default Tag :	0.20
Nudge factor :	0.10
Move Rotate Angle :	90.00

The numbers in this window can be changed by clicking the tip of the arrow cursor on any of the numbers (such as sheet width) and then typing in a new value and pressing the enter key. Alternatively, if the change you want to make is a small one (eg change sheet size from 1000 to 1005) you can click on either of the up or down arrows to the right of the number and the value will increase or decrease with every click of the mouse button. Those entries (such as sheet units) which have a switch symbol to the right of them can only be changed by clicking on the actual switch symbol.

Once you have completed the changes that you wish to make, move the arrow cursor over the OK button and click either mouse button. If you do not wish to keep the changes or just popped up the box to have a look at the values, then click over the CANCEL button or press the escape (ESC) key.

Some of the pop up windows contain a list of items which can be selected by clicking either mouse button on the actual text. For example, select the turret icon (top row, fourth from the right) and you will see a list of your turret stations and the tools currently in each station, similar to the one shown.

TURRET	
T1	(180) [50] RECT W40.00 x H40.00mm @ 0.00°
T2	(25) [5] ROUND D5.00mm
T3	(25) [10] ROUND D4.20mm
T4	(25) [50] ROUND D15.00mm
T5	(50) [20] SPECIAL COLLAR-12 mm @ 0.00°
T6	(25)
T7	(25) [10] ROUND D6.70mm
T8	(25)
T9	(50) [20] SPECIAL COLLAR-50 mm @ 0.00°
T10	(25)
T11	(25)
T12	(25) [15] ROUND D11.20mm
T13	(50) [25] RECT W20.00 x H20.00mm @ 0.00°
T14	(25) [15] ROUND D14.00mm
T15	(25) [12] ROUND D16.00mm
T16	(25) [12] ROUND D20.00mm
T17	(50) [12] RECT W20.00 x H20.00mm @ 0.00°
T18	(25)
T19	(25)
T20	(25)

Current Station Priority : 50

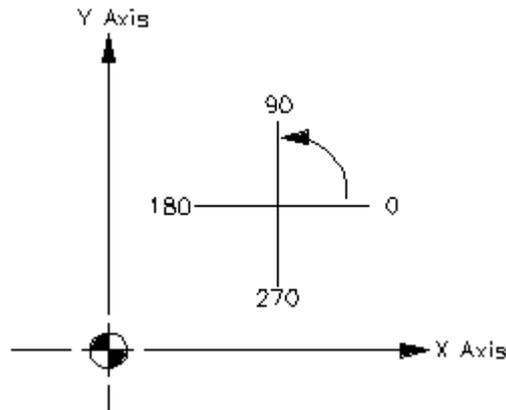
G-CODER Professional Reference Manual

Notice that there is a scroll bar to the right of the turret descriptions which has two arrows in it. If you click on the up arrow you will move up the turret list and the down arrow will move you down the list; the indicator between the two arrows lets you know approximately how far down the list you are. When you can see the turret station you wish to use, simply click anywhere on the text of that turret and it will be highlighted. You have now changed tool and all you need to do is click on the OK button to confirm this and close the window.

This method of changing values and selecting items in pop up windows is used extensively in G-CODER Professional and once you are accustomed to using it, you will find it quick and easy.

Sheet Conventions

The diagram below shows how G-CODER Professional defines its angles and X,Y values. No matter which machine you are programming for, this will remain the same.



The reason for this convention is that it makes it much easier to change programs between machines with different axis directions (for example from Pulsa to Proteus). The only difference you will notice is that if you are programming a Pulsa, Titan or Achilles, then the clamps will appear at the top of the sheet; all other machines will have the clamps at the bottom of the sheet. When G-CODER Professional produces the 'G' code program it will automatically make allowances for datum position.

Punching on the sheet

Once you have selected a punch icon (bottom row) and typed in the necessary details, you can place the punch action on the sheet in a number of ways. If you move the cursor across the sheet, you will see the current tool shape displayed on the cursor; this helps you to line up the punch with other punches. The co-ordinates of the cursor are displayed on the right of the status line below the icons. By holding down the Shift key and clicking the left mouse button the current punch action will be punched at the current cursor position. Alternatively, you can press the space bar and an absolute co-ordinate (G90) pop up window will appear; type in the XY co-ordinates and the punch will be performed at that position. If you hold down the Shift key and then press the space bar, a relative co-ordinates (G91) pop up window will appear. The values you give will then be relative to the last punch.

Calculating co-ordinate entry values

When using the spacebar or Shift spacebar to enter punch operations co-ordinates, you can also use the following functions:

+	Addition
-	Subtraction
/	Division
*	Multiplication

Brackets can also be used in a calculation for example:

$100+10*(6/2)$ will be calculated as 330

The order of evaluation is $() / * + -$

As well as supporting the 4 basic functions, there are 4 special codes that can be used in a calculation; these are as follows:

R	- Radius of Tool
D	- Dia of Tool

G-CODER Professional Reference Manual

W - Width of Tool

H - Height of Tool

These special codes are set by the current tool value when you enter the co-ordinate entry window and can then be used in a calculation.

For example if you are using a 45 x 5 rectangular tool the W code would be set to 45 and the H code to 5 so they could be included in a calculation as follows:

$100+10+W$ will be calculated as 155

This avoids having to use a pocket calculator to add or subtract drawing values to find the position of a punch operation. In the following example, an equation has been entered and on pressing the ENTER key it will be evaluated and the result displayed. If you make a mistake when entering values or functions use the BACKSPACE key to delete and then type the correction.

ABSOLUTE COORDS		
X Coord :	125+15+35-10	↕
Y Coord :	-100.00	↕
?		

List of Action keystrokes

The following keystrokes are valid in G-Coder Professional - please note that you must have the cursor on the sheet for them to have any effect. Some commands require a mouse button to be clicked whilst one or two keys are held down on the keyboard. For example Ctrl + Left Click means press and hold down the Ctrl key and then click the left mouse button.

Screen display

Ctrl + Left Click	-	Zoom in by factor of 2
Ctrl + Right Click	-	Zoom out by factor of 2
Ctrl + Alt + Left Click	-	Pan across the sheet at same magnification
F5	-	Draw bounds of punches on sheet
F6	-	Recalc bounds of punches on sheet
T	-	Toggle tool display on or off at cursor
U	-	Undo last action

Punch Selecting

Left Click	-	Select punch action
Right Click	-	De-select punch action
Alt + Left Click	-	Start a selection window (left click again will select everything in window) (right click again will de-select)

Punch action positionig

Spacebar	-	Enter absolute co-ordinates
Any Number key or +- Enter key	-	Enter abs co-ordinates (no spacebar needed) Enter abs co-ordinates
Shift + Spacebar	-	Enter relative co-ordinates
Shift + Click	-	Punch the current punch on the sheet (at cursor position in punch mode) (snap to nearest geometry in geometry mode)
C	-	Snap punch to corner of given (rectangular) tool (must be in single punch icon first)
A	-	Punch all marked rads with current tool. The rads must have been marked first using the Query Icon. (Must be in Geometry Mode)

G-CODER Professional Reference Manual

Punch Mode

- G - Toggle to geometry mode
(Indicated by tool and cursor drawn in RED)
- P - Toggle to punch mode
(Indicated by tool and cursor drawn in BLACK)

Mirror Command

- X - Default to X mirror (line is drawn up sheet)
- Y - Default to Y mirror (line is drawn across sheet)

Move Command

- Page Up - Rotate highlighted part Anti Clockwise
- Page Down - Rotate highlighted part Clockwise
(rotate angle is determined by the sheet params)
- Cursor arrow - Move selected punches in arrow direction
by nudge value set in the Sheet parameters.
One key press is equal to move of nudge value.

Tool Priorities

If you want to make sure that some tools are punched first (eg all small tools) and some tools are punched last (eg slitting tools), then you can set the priority levels on these tools. The priorities range from 1 (tool punched first) to 100 (tool punched last) and each new tool added to the turret has a default priority of 50.

To change the tool priorities, select the turret icon and the turret number that you want and then click on the priority window below the turret numbers. You can then enter any value from 1 to 100 for the currently selected tool. It is a good idea if you set the priority of the first tool to 5, the second to 10 and so on. This way you will be able to insert a tool between any two tools without having to renumber all the priorities.

If you want to confirm the order in which the tools will be punched, select the redraw icon then make sure the by tool priority switch is ON and click OK. Click on the redraw mode switch so that it displays SLOW, then click OK. This order is the same as G-CODER Professional will use to punch the actual program, provided that you have the Sort By Tool switch ON in the post processor window.

Don't forget to reset the redraw mode to FULL SPEED when you are satisfied that the punching order is correct.

Note : Setting a tool priority to a value of 90 or above means that it will be ignored when optimising the punching order using the Sort Icon.

Selecting punches (Query, Delete, Move, Copy, Mirror and Rotate)

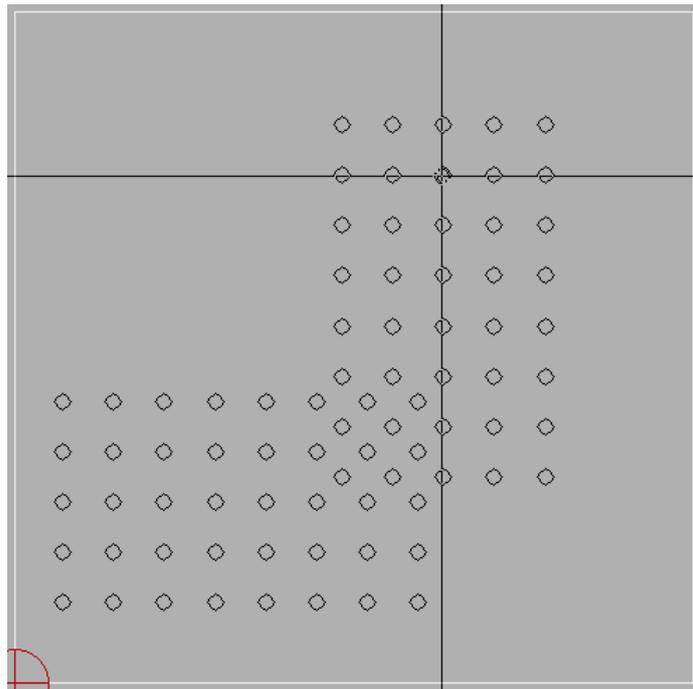
Each of these commands require that you select one or more punch operations. First select the icon (eg delete) then place the cross hair cursor over each item to be selected and click a mouse button. The left button selects a punch operation and the right button de-selects it. Once a punch operation has been selected it will be highlighted in green.

If a number of punch operations overlap a question mark will be displayed (?) and each punch operation will be highlighted in turn. Click the left button to select the punch and the right button to ignore it. Continue with this until the question mark is replaced by the normal cross hair cursor.

Most of these commands (except Query) have an ALL button in the pop up window which allows you to select everything on the sheet in one go. You can then de-select any punches that you want to keep before clicking on the OK button to perform the delete.

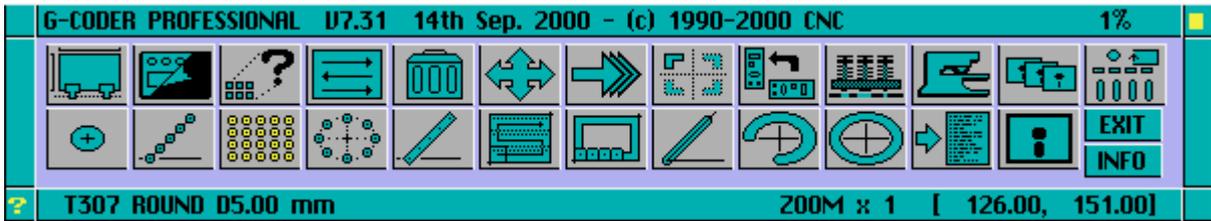
To select a number of punch operations it is easier to use the window select mode. To do this position the cross hair cursor to one side of punches to be selected, press the ALT key and left mouse button. Move the mouse so that all the punches to be selected are within the window and then click the left button to select. This method can also be used for de-selecting by clicking the right mouse button.

The selection mechanism works by placing a rectangle around each punch operation (which is not drawn on the screen). If you place the cursor anywhere within this rectangle then you will select the punch. This method does mean that some very small single punches (Eg 3mm round punch) will have small bounding rectangles (try zooming in to select them) and large grids will have very large bounding rectangles. For example if you have two grids overlapping each other as below but only want to select the right hand grid place the cursor as shown.



The ICON Box

The Icon Boxes allow you to create and manipulate programs for your punch press.



The main menu window contains Icons which represent, in picture form, the main functions of G-CODER Professional.

To perform a function the particular icon must be selected by moving the arrow over the top of the icon and clicking either of the mouse buttons.

Sheet Parameters



This icon allows you to set the sheet units, material type/thickness, size, clamps, datum, tag value and nudge factor.

SHEET, CLAMPS AND DATUM	
Sheet Units :	MM <input checked="" type="checkbox"/>
Sheet Material :	
Sheet Thickness :	0.00
Sheet Width :	1000.00
Sheet Height :	500.00
Clamp 1 (X) :	150.00
Clamp 2 (X) :	750.00
Clamp Width :	200.00
Clamp Height :	80.00
Clamp Backoff :	2.00
Virtual Datum X :	0.00
Virtual Datum Y :	0.00
Default Tag :	0.20
Nudge factor :	0.10
Move Rotate Angle :	90.00

Buttons: OK, CANCEL, COMMENTS, PRINT

All the boxes in this window have a default value which is held in a configuration file for your punch press. The configuration file can be edited using G-CODER to change these default values to what you most commonly use. This is normally done for you during installation of the system.

When you start a new job this will be the first icon you choose. The sheet units and width/ height are self explanatory; simply type in the values you want.

The material type and thickness are required by some machine tools (Trumpf, Nisshinbo for example) and are useful if you are saving jobs as Part Files for future use. If you start a new job and set the material and thickness you are warned if you try and Append a Part file of dissimilar material or thickness. This is to prevent you from producing parts from the wrong material specification.

Select the boxes for clamp information and type in your replacements to set up the clamps and a reasonable safe zone size. G-CODER Professional will then warn you if any punches are in these clamp zones.

Redraw



This icon will allow you to redraw the sheet and all punches on it at full speed, slow speed or single step.



Redraw Mode : FULL SPEED / SLOW / SINGLE STEP

It is useful to be able to slow down the punch speed so that you can remind yourself of the order in which you made the punches. If you click on the redraw mode switch and set it to slow, as the job is being redrawn you can vary the speed by entering a number (1=slowest and 9=fastest). If you press the escape key whilst a slow or single step redraw is in progress then the rest of the punches are displayed at full speed. If you press the spacebar during either fast or slow redraw you can revert to single step mode, pressing a number will then revert back to slow mode. In single step mode a punch will be displayed each time you click the mouse button. If you hold down the mouse button and move the mouse it will display continuously until you stop moving the mouse. If you press T in the middle of slow or single step redraw all of the current tool will be displayed at full speed and then G-CODER Professional will put you back into single step mode. This is useful for ignoring one or more tools.

By Tool Priority : ON / OFF

The post processor allows you to sort your output by tool priority and it is useful to be able to redraw in the same order. If you click on the Prioritised switch (set it to ON) then the punches will be displayed in this order each time you do a redraw (regardless of speed setting).

Punch Style : OUTLINE / SOLID / CENTRE PT

Punch Style refers to the way that punches are drawn on the sheet. Normally you will work in outline mode but this can be changed to solid mode to give a more representative image of the sheet of material after punching. Centre point is useful if you job contains many special or multi-punch tools as it only displays a centre point for each punch and not the entire tool.

DXF Geometry : ON / OFF

The redraw window has a switch to allow you to turn off the drawing of the DXF geometry. This can be useful when you have finished punching the geometry and want to speed up the drawing of the punches on the sheet.

HINT : To see clearly what you have punched it may help to temporarily change the Punch Style to Solid. However don't leave it set to this as it will slow down the drawing speed.

Zoom In / Out

To zoom in, position cross hair cursor at centre of the area of interest and whilst holding down the Ctrl key click the left mouse button. If you wish to zoom in further to see more detail just keep clicking the left mouse button.

To zoom out click the right hand mouse button whilst holding down the Ctrl key. As you zoom in and out the scale or zoom factor is displayed on the status line beneath the icons.

G-CODER Professional Reference Manual

Using zoom you can quickly magnify an area examine it, then zoom out, move across the sheet and zoom in on a different area. If you are having problems selecting very small punches you can zoom in select and then zoom out.

Pan

If you wish to move across the sheet without zooming out first, you can use the PAN command. Position the cross hair cursor at the edge of the sheet and then whilst holding down the Ctrl and Alt keys click the left mouse button. The sheet will move across to allow you to see punches which were previously outside the viewing area. Using PAN you can view around an area that you have just zoomed and avoid zooming out moving and then zooming in again.

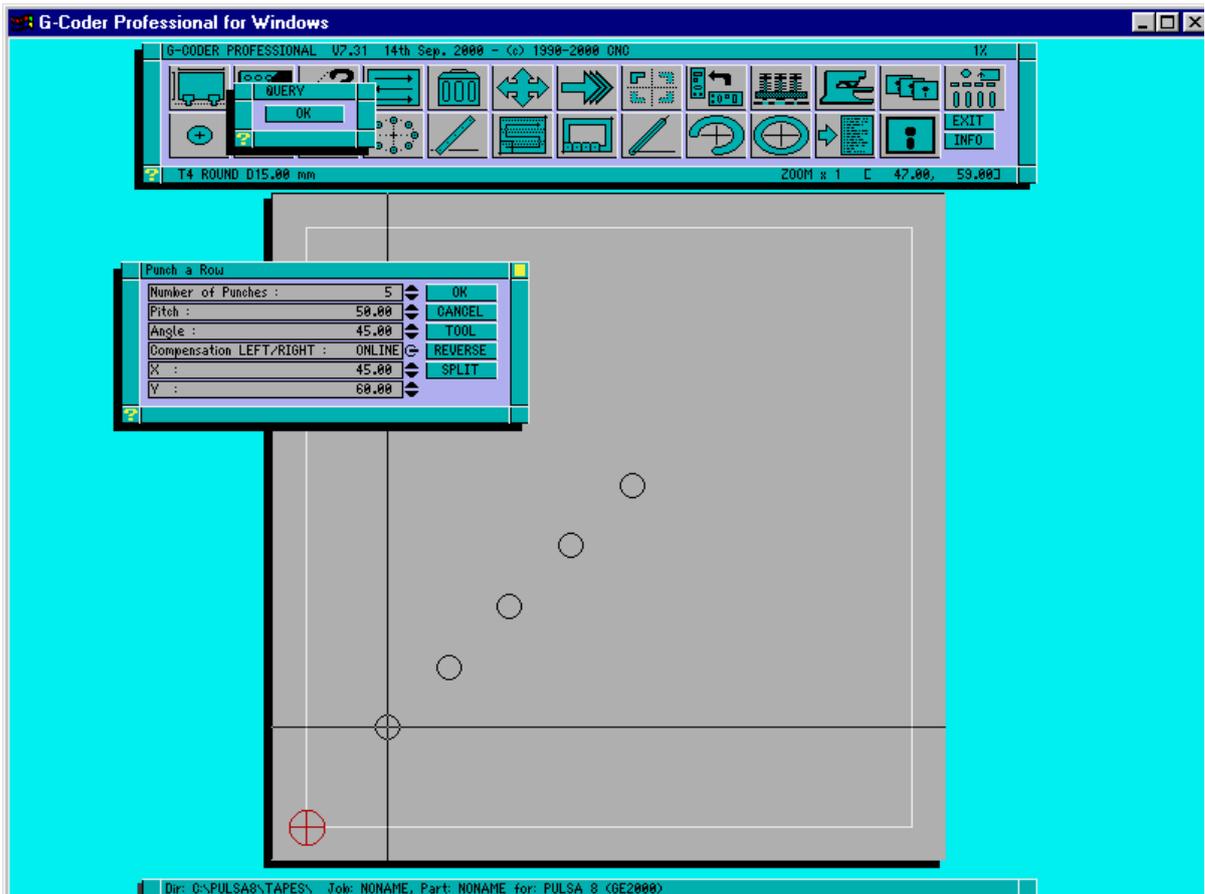
Query Punch



Punch Mode

This icon allows you to display and alter the details of any punch operation on the sheet and is one of the most powerful (and useful) commands in G-CODER Professional.

If you have punched a row and want to change the number of punches in the row you select the query icon and then select the row. Query will display all the relevant details of that row in a pop up window. You can then change any of these details, click on the OK button in the window and see the row change immediately. Query will even let you change the tool that was used for the row and its position on the sheet, all in one go.



Punch a Row, Nibble a Line and Punch a Line all have the additional button to REVERSE the direction of the punch operation. If you click on this button the direction of angle will reverse and the compensation will change automatically to the opposite side. This feature is useful for doing a limited amount of manual optimization. Punch a Row and Punch Grid also display a further button SPLIT which allows you to split rows and grids into individual hits.

HINT : If you have made a mistake it is often quicker to use QUERY to correct the error than to delete and re-punch.

G-CODER Professional Reference Manual

Split row / grid

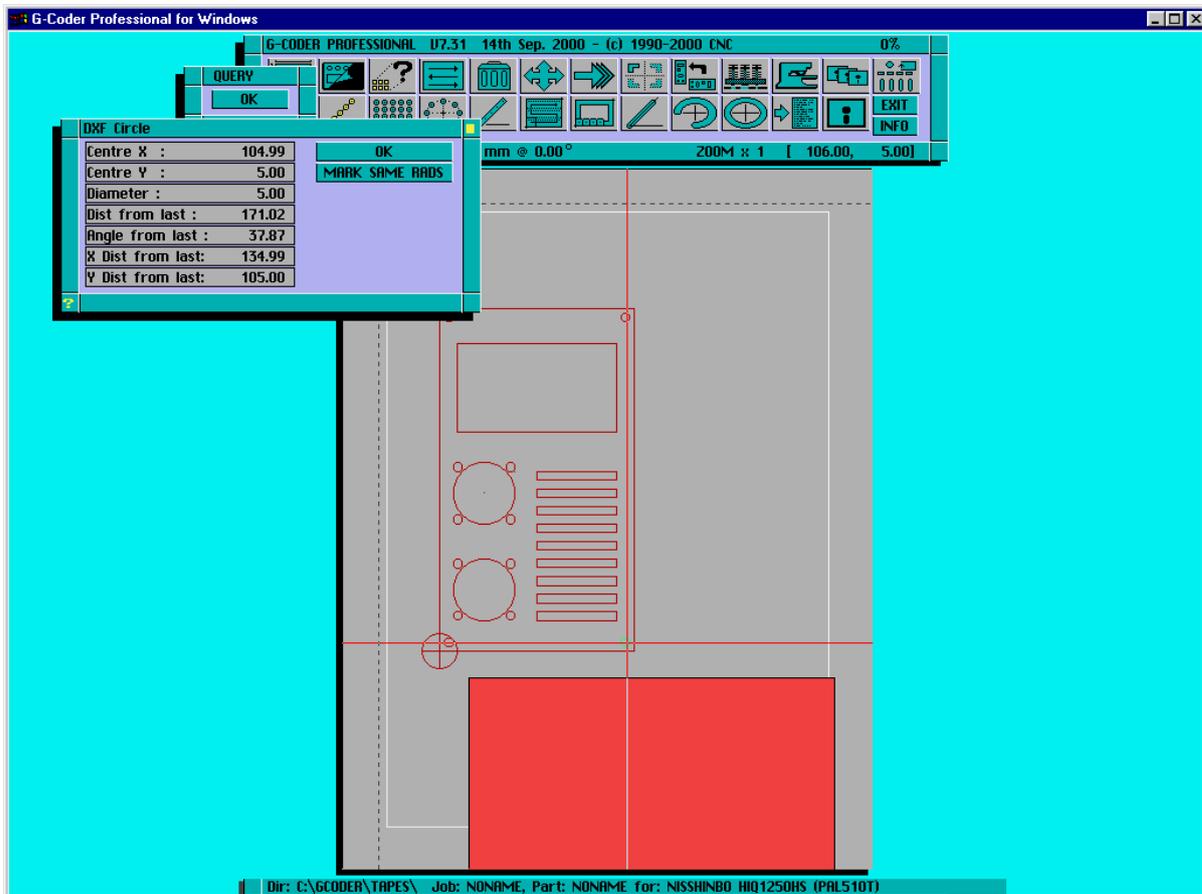
If you Query a grid or a row of discrete holes (not a punch line) then you will see a new SPLIT button displayed. Clicking this button will split the grid or row into individual punches. You can then remove single holes from the grid or row.

Split line

If you Query a punched line then clicking the SPLIT button will split the line at the point of Query into two lines. This is extremely useful if you want to punch a line bigger than the travel limits of the machine and then split it in the middle for post processing.

Geometry Mode

Once you have read in a DXF file it may be necessary for you to check the sizes of holes, cutouts etc. in the drawing. You can do this by selecting QUERY ICON and then selecting the geometry item (Make sure you are in Geometry Mode first by pressing the G key). A screen like the one below will be displayed :



You cannot modify the information in the window (unlike Punch Query), it is for information only.

The first few boxes give all the relevant information on the geometry item which allows you to find the diameter of any round holes on the sheet.

The last four boxes give you the distance, angle, distance in X and distance in Y of the item from the last geometry item queried. This is extremely useful when you are trying to find the radius of a bolt hole or the pitch of a grid.

Sort



This Icon allows you to sort and optimise the order of punching using four methods.



Manual

There are times when you wish to punch a number of holes in a certain way for such reasons as to avoid deforming the material. The Manual option allows you to use the mouse to select the order of punching. As you select punches they are marked in green so you can see which you have selected.

HINT : Use the Redraw Icon to set Redraw Mode to slow speed to see the order of punching.

Next Nearest

This option will sort the order of punches starting from the bottom left hand corner into a next nearest order. The results may look a little odd as the direction may tend to wander around the sheet, however this is normally the quickest and best method of sorting. After sorting, a box will be displayed showing the before and after path length with a factor representing any improvement in punching speed.

Sort in X

This option is similar to Next Nearest but will give preference to punching in the 'X' axis direction. This method uses a ranging technique over the sheet and if there are a lot of punches can take several minutes to perform.

Sort in Y

This option will do the same as Sort in X except that preference is given to punching in the 'Y' axis direction.

Sort by AI Angle

If this switch is set to YES (this is the default) then any Auto-index tools will be sorted by angle to minimise tool turning starting at 360 degrees and working back to 0. This means that the horizontal slits are done last to maintain maximum sheet strength.

Note : If there are tools that you don't want to be sorted then using the turret icon set their priority numbers to 90 or greater and they will be ignored during the Sort.

Delete



This icon allows you to delete any of the punch operations or the DXF geometry on the sheet. You can press G to swap to geometry mode in the middle of selecting punches and P will get you back to punch mode.

Once you have selected the items you want to delete, click on the OK button and they will be deleted, clicking on the CANCEL button will exit from the delete command without deleting anything. If you are unsure about how to select a punch operation then see the section "Selecting punch operations" in this manual.

HINT : Save your job prior to deleting a number of punches just in case you have selected something you did not want to delete. You can then always reload the job and try again.

Move (and rotate)



This icon allows you to select a number of punch operations/ dxf geometry items (or all of them) and move them across the sheet.

Once you have selected the punches you want to move, select the OK button from the pop up window. You will now see a cross displayed on the first item you selected; this is the reference point that you will be moving, all other punches are moved relative to this point. If you move the cursor a line will be drawn from the reference point to the current position.

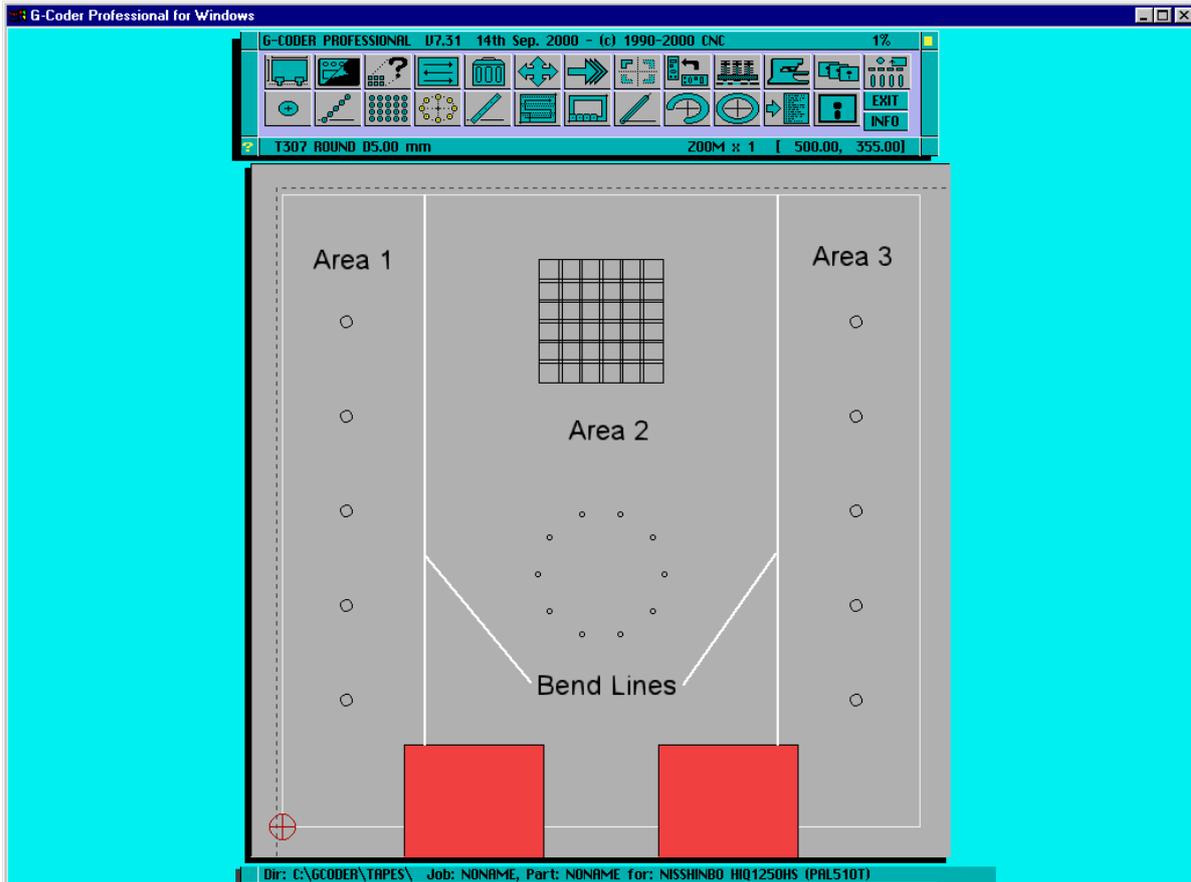
Pressing either mouse button will move all the selected punches to the current position. You can continue moving the mouse and clicking a button until you are happy with the new position. Alternatively you may want to move the punches relative to the current position or to a new absolute position. To do this use the space or shift space bar and enter the X and Y values.

You may also use the nudge facility to move punches a set amount relative to the current position. Move the cursor onto the sheet and press one of the four cursor keys. Each time you press a key the selected punches are moved up, down, left or right (depending on the cursor key pressed) by the nudge amount set in the sheet parameters window.

If you wish to rotate the punches around the cursor then you can press PgUp or PgDn to rotate by 90 degrees (or the angle specified in the sheet parameters). *WARNING* : once you have rotated the part you cannot cancel this rotation.

At any time you can cancel the move by clicking on the CANCEL button in the pop up window on top of the icons. Clicking on the OK button will move all the punches and exit from the move command.

G-CODER Professional Reference Manual



EXAMPLE : In this example you can position all the punches on the sheet ignoring the bending allowances (BA). Using the MOVE ICON you can then select all punches in Area 2 and move them by pressing the shift and space bar and entering the BA value. All punches in Area 3 can then be selected and moved in the same way by twice the BA value.

This method saves having to add the BA value to your dimensions and thus reduces the risk of making a mistake.

EXAMPLE USING NUDGE : Click on the DATUM ICON and enter the bend allowance (BA) into the nudge factor box then click on OK. Using the MOVE ICON select all of the punches in areas 2 and 3 (not forgetting to click on OK), move the cursor back onto the sheet and press the right cursor key once then click on OK. This will move all of the punches to the right by the bend allowance. Next select all of the punches in area 3 and press the right arrow once more and click on OK.

Copy and Multi-Part



This command allows you to create, edit and post process multi-parts on the sheet.. This saves space in the job file and, on some machine controls, produces much smaller programs. You can have upto 10 multi-parts in a single job.

MULTI-PART			
Relative Pitch in X :	105.00	↕	OK
Relative Pitch in Y :	105.00	↕	CANCEL
Fit Multi-part to sheet :	YES	☑	
Number of Copies in X :	1	↕	
Number of Copies in Y :	1	↕	
Starting Corner :	BOTTOM-LEFT	⏴	
Punch as :	SINGLE PUNCHES	⏴	
Copy By Tool Priorities :	NO	☑	
Slitting Into Clamps Only :	NO	☑	

Creating a multi-part

Click on the COPY icon and select the punches to be multi-parted in the normal way, then click the OK button followed by the MULTI button. Next fill in the part size in X and Y (allowing for any slitting tools) and enter the number of copies in X and Y or alternatively use the Fit to Sheet switch which will calculate the number of parts. Click on the 'Punch as' switch until 'MULTI-PARTS' is displayed.

Finally click on the OK button to see the multi-part.

When you click OK again to accept the Multi-Part you will notice that the original part is drawn in black (as normal) and the copies are drawn in Dark Green.

The next time you click on the MULTI button the number of parts in X and Y will displayed for the last copy.

Editing punches within the Multi-Part.

When you edit a punch within a multi-part (change its position, change its tool, add tags etc.) you must select the original black punch. Clicking on one of the Dark Green copies will have no effect (as these punches do not actually exist in the database).

You can use any of the following to edit punches within a multi-part:

QUERY After you OK your change, the original and all the copies of that punch will be redrawn.

SORT Sort will sort the order of the punches within the multi-parts (or parts).

DELETE You can delete any number of punches out of a multi-part and all the copies are removed. If you delete all the punches within a given multi-part the multi-part itself is deleted.

MOVE You can move a single punch or a series of punches or a whole Multi-Part in one go. Each time you click on the sheet (or type in the new coordinates) the Multi-Part is redrawn.

ROTATE You can rotate single punches or an entire multi-part.

The following commands do not change any multi-parts selected but simply make a copy of the original punches selected:

G-CODER Professional Reference Manual

COPY The part selected is simply copied without the multi-part copies.

MIRROR The part selected is mirrored without the multi-part copies.

If you want to multi-part the copies you must create a new multi-part as described previously.

Editing the multi-part itself (number of copies, pitch etc.)

You can edit any of the multi-parts in the job by using the COPY/MULTI-PART icon. This is very useful if you have changed a job from one machine to another and the travel limits of the new machine are different.

Simply do the following;

Click on the COPY icon and select any punch within the multi-part that you want to edit. Next click on the OK button and you will see a new button M-EDIT appear.



Click on this button to edit the details of the multi-part and you will see a window like the following:



Once you have changed the details of the multi-part such as pitch in X, number of parts etc. click the OK button to see the changes to the multi-part.

Deleting a multi-part without deleting the original punches.

Follow the instructions above for editing a multi part upto clicking on the M-EDIT button. You will notice that the multi-part window contains a DELETE button; click this button and the multi-part will be deleted leaving the original punches.

Adding a new punch to any existing multi-part

If you want to add a new punch to an existing multi-part then you must first delete the multi-part (but not the punches themselves) and re-create the multi-part with the new punch.

Redrawing of multi-parts.

G-CODER Professional can redraw multi parts either by tool priority or in the original punch order in slow or single step mode. Select the REDRAW icon and click on the 'Redraw Mode' switch to set it to slow or single step. This will then redraw the whole sheet (including the Dark Green copies) in the order that it would be punched. Note that if you have more than one multi-part using the same tool (or tools) then all the punches for a given tool are done first, rather than the multi-part being punched all in one go.

Post processing of multi-parts.

G-CODER Professional Reference Manual

The post processor can produce macros or subroutines for some machine controls (check with CNC if you are unsure about your machine) which reduces the size of program considerably for big multi-parts. It does this in two ways:

If you set 'Sort by tool priority' to ON in the post processor window (which is the default and most often used) then G-CODER Professional will create a macro for each tool in each multi-part.

For example, suppose you use the same 5 tools (T1 to T5) in 3 different multi-parts (MP1 to MP3) and T1 is highest priority through to T5 as lowest. You will end up with 15 macros calls in the following order:

```
T1 in MP1 followed by T1 in MP2 followed by T1 in MP3
T2 in MP1 followed by T2 in MP2 followed by T2 in MP3
.
.
T5 in MP1 followed by T5 in MP2 followed by T5 in MP3
```

If you are unsure of the punch order then use the REDRAW icon to check.

If you set 'Sort by tool priority' to OFF in the post processor window then the post processor will create a single macro for each multi-part with tool changes in the macro itself.

Please note that you can change the order of punching within a multi-part using the SORT icon.

Addendum for Amada users

The post processor in G-CODER Professional can support macro numbers U1 to U89. By default it uses U1 to U5 which are re-used (several times) if need be. Although you may not be used to this approach it should work on any Amada without any problems. This is a deliberate design decision as these are the only macro numbers supported on some of the older Amada controls. If you need to change this then add the line: `MAX_MACRO_NUM S/E;` to your config file, where S is the start number and E is the end.

For example `MAX_MACRO_NUM 6/89;`

Mirror



This operation will allow you to select any number of punches and mirror them in either the 'X' or 'Y' axis.

After clicking on the Mirror Icon you can use the ALL button to select all the punches on the sheet or pick the punches individually using the cursor or window method (Refer to section Selecting Punches). Once you have selected the punches you want to mirror click on the OK button. You will now see a line appear across the sheet to represent the position about which the punches will be mirrored. You can change this line to mirror in 'X' or 'Y' by pressing the 'X' key or 'Y' key on the keyboard. The mirror line can be positioned using the mouse or if you press the Spacebar you can enter the exact position about which to mirror the punches.

After punches are mirrored you are asked if you wish to delete the original component. This is useful if you want to mirror a complete component over the top of itself (sometimes you may want to do this if the burrs are on the wrong side of the job).

At any time you can cancel the mirror by clicking on the CANCEL button in the pop up window. Clicking on the OK button will mirror all the selected punches.

Example (Corner Notching)

In the following example you will see that we can take a bottom left corner notch and mirror it in 'Y' to create the top left. Then we can select both of these and mirror them in 'X' to produce the top right and bottom right corners.

To produce the top left corner:

Select the Mirror Icon.

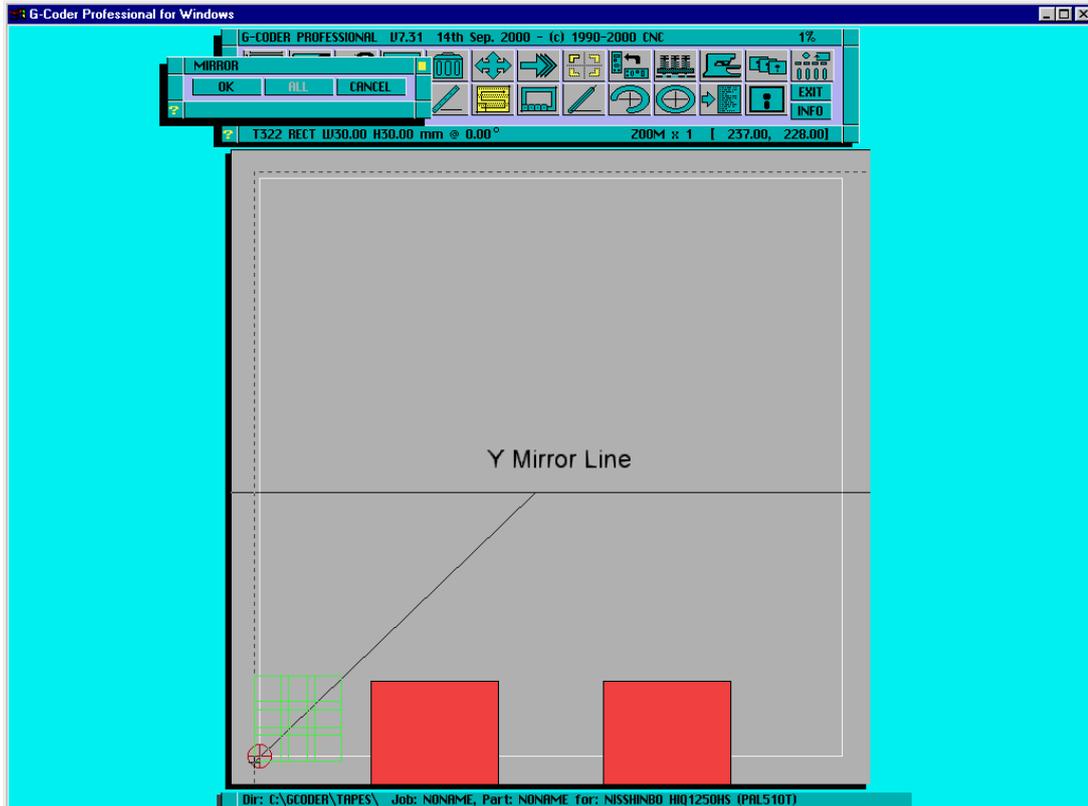
Move the cursor over the bottom left corner notch and click with the left mouse button. It will be redrawn in green to confirm that it has been selected.

Move the cursor and select the OK button.

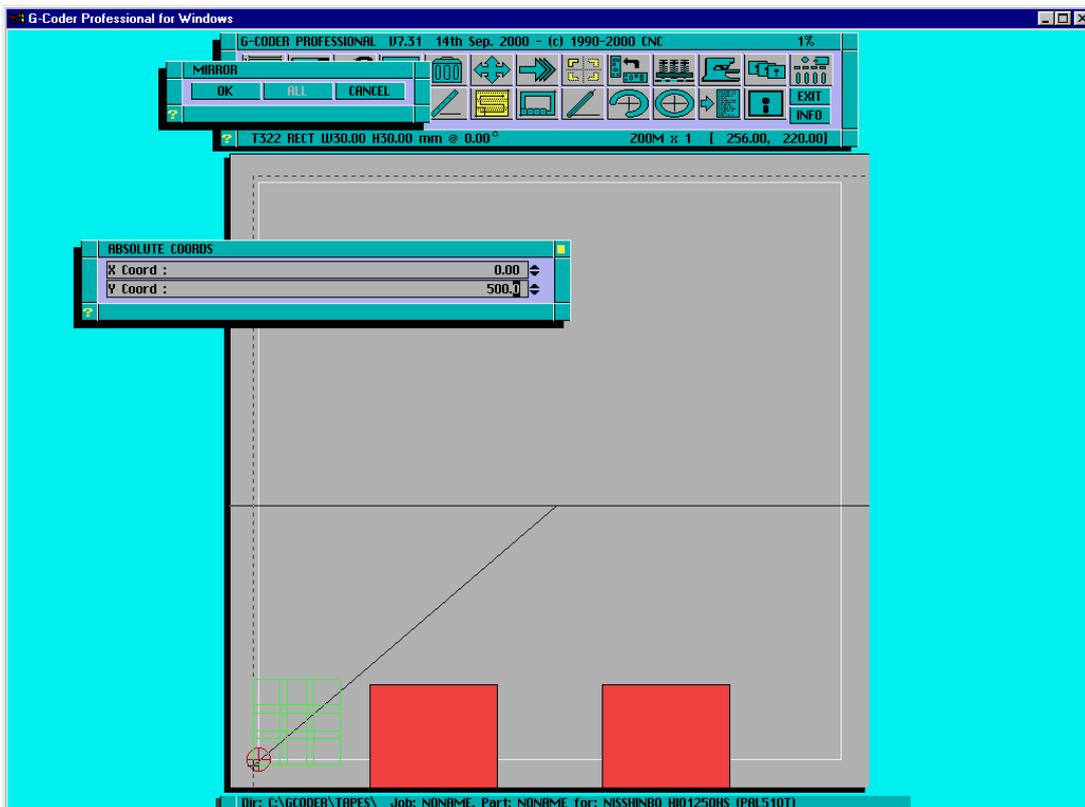
Press the Y key to select mirror in Y axis.

A mirror line will appear across the sheet as shown.

G-CODER Professional Reference Manual

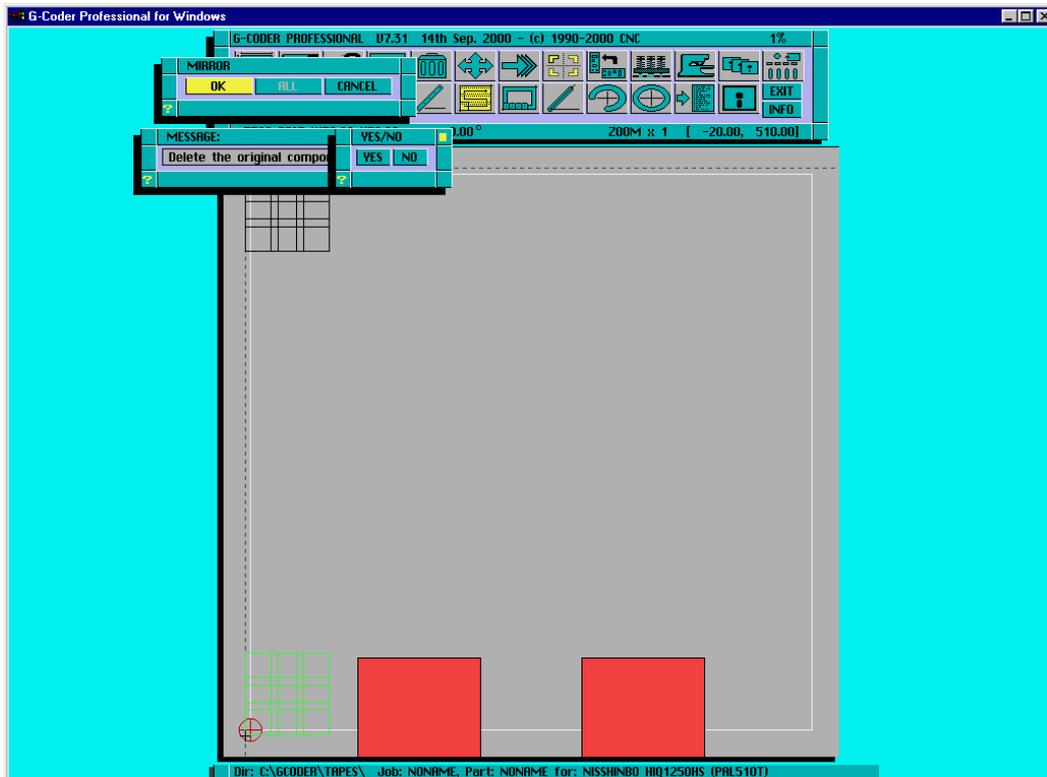


You can now position the mirror line by either moving the mouse so that the co-ordinate tracking values in the status line show a value of 500.00 (half of the sheet height) in the 'Y' axis or by pressing the Spacebar and entering a value of 500.00 into the 'Y' co-ordinate box. Please note that the 'X' value is ignored during a mirror in 'Y'.



G-CODER Professional Reference Manual

The bottom left corner notch will now be mirrored to produce the top left corner. You can then select the OK button to accept or CANCEL to undo. You are then asked if you want to delete the original component, in this case click on the No button to retain the original corner notch.



NOTE : If you redraw the job in Slow Mode you will notice that the direction of punching on the top left corner notch has also been mirrored so it will punch from the outside of the sheet.

To produce the top right and bottom right corner:

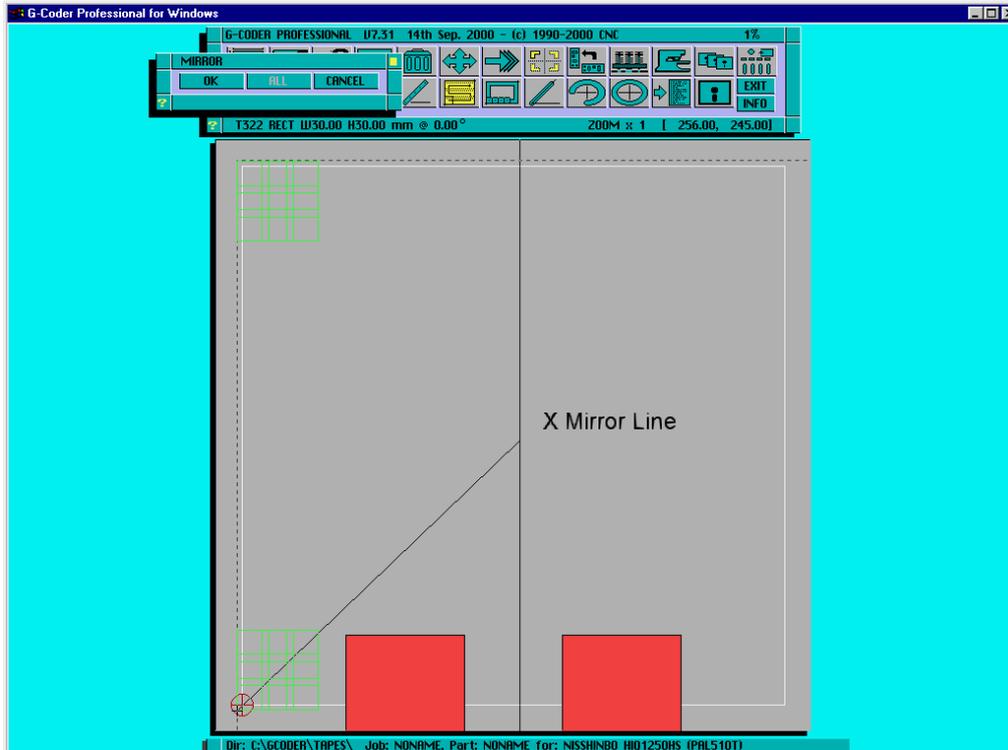
Select the Mirror Icon.

Using the mouse select the bottom left and top left corner notches. They will be redrawn in green to confirm that they have been selected. Please note that the order of selecting the punches is important as it affects the order of punching .

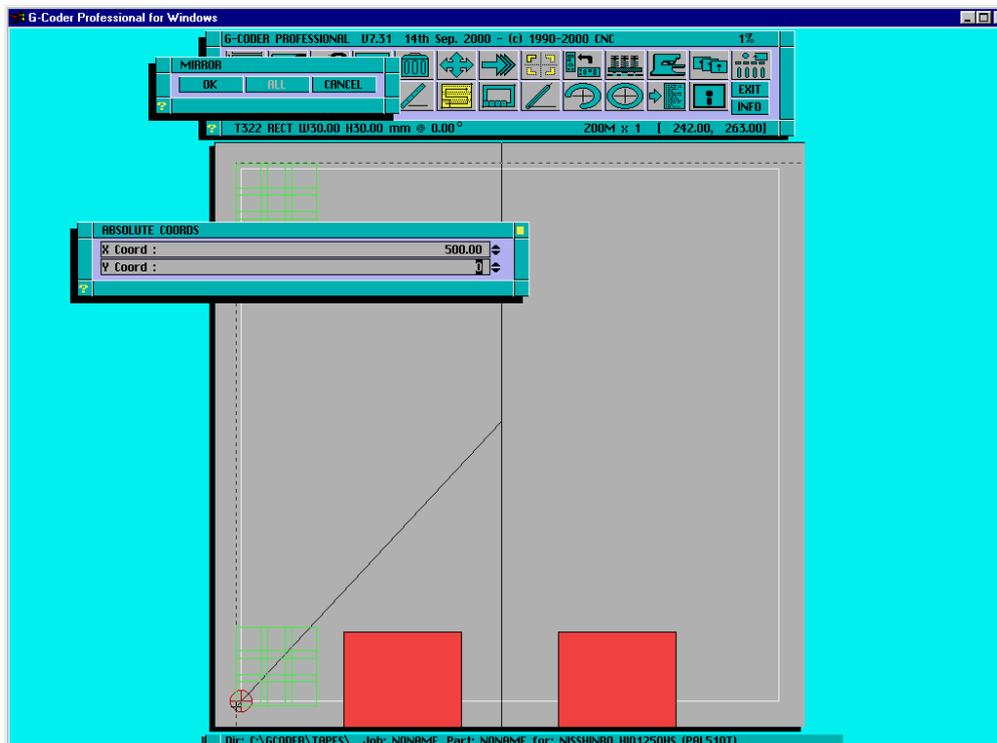
Move the cursor and select the OK button.

A mirror line will appear across the sheet (default mirror is in the 'X' axis) as shown:

G-CODER Professional Reference Manual

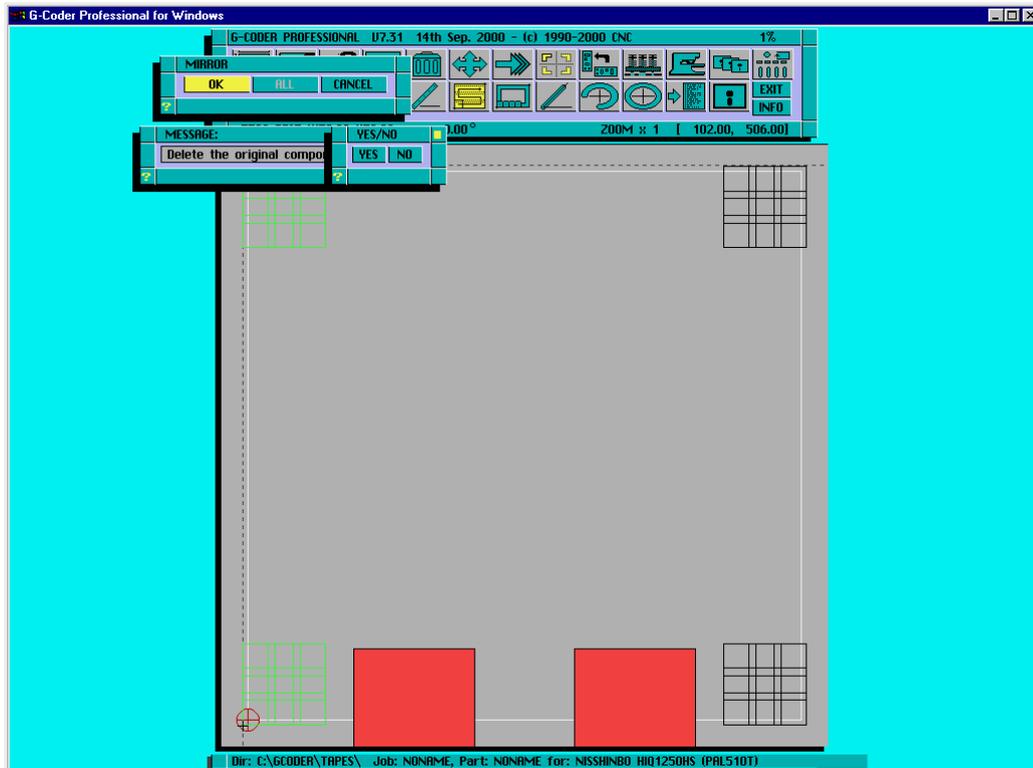


You can now position the mirror line by either moving the mouse so that the co-ordinate tracking values in the status line show a value of 500.00 (half of the sheet width) in the 'X' axis or by pressing the Spacebar and entering a value of 500.00 into the 'X' co-ordinate box. Please note that the 'Y' value is ignored during a mirror in 'X'.

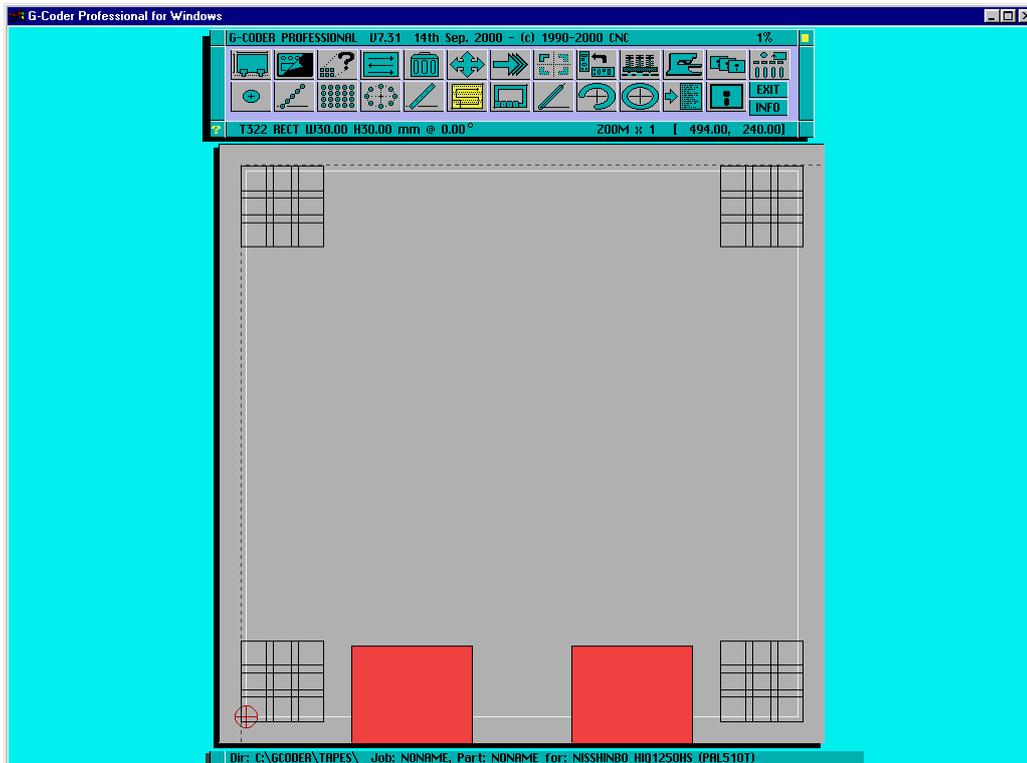


The bottom left and top left corner notches will now be mirrored to produce the top right and bottom right corners. You can then select the OK button to accept or CANCEL to undo. You are then asked if you want to delete the original component, in this case click on the No button to retain the original corner notch.

G-CODER Professional Reference Manual



The corner notches are now finished and should look like the following:



Rotate



G-CODER Professional Reference Manual

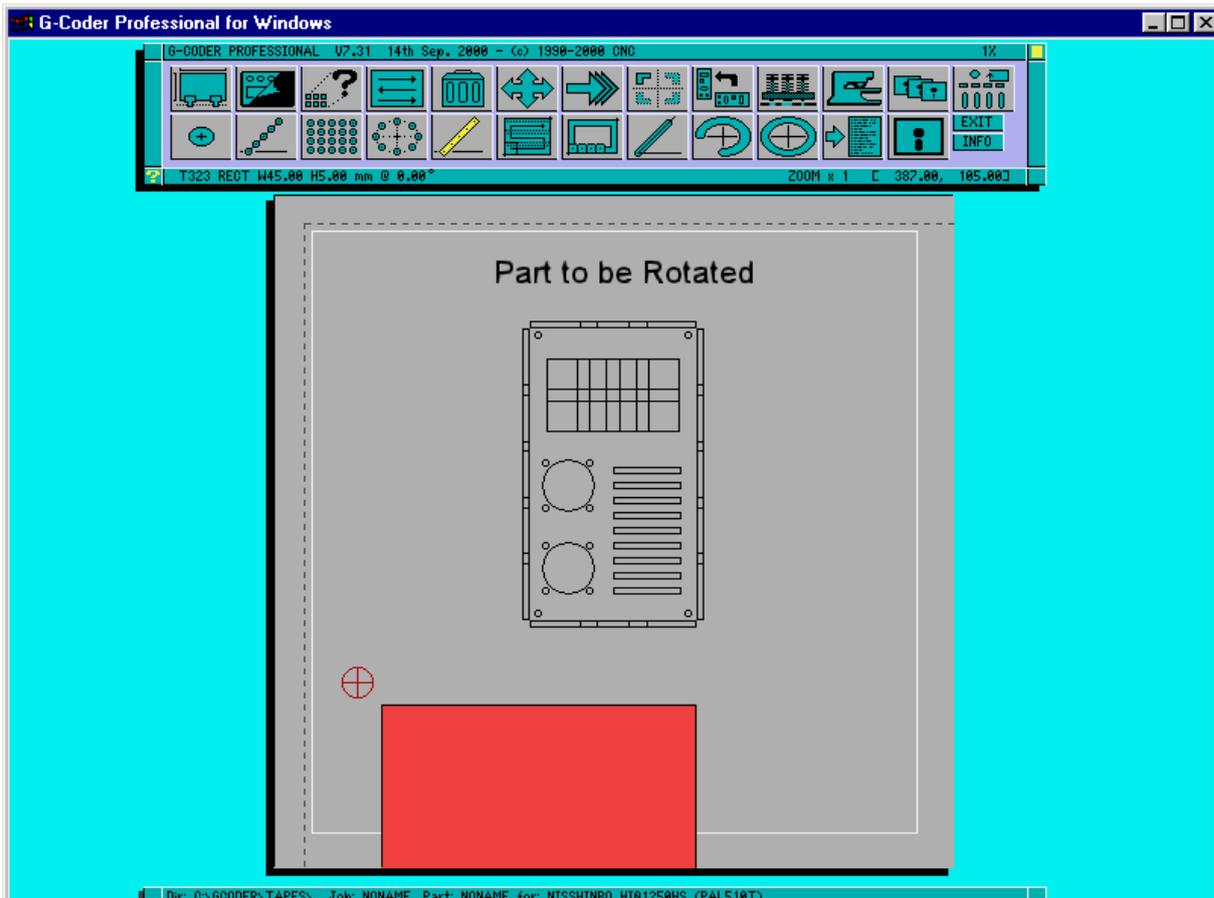
This icon allows you to rotate a number of punches, a complete component or DXF geometry through any angle. Useful if you can fit another row of components on a sheet by rotating and then copying.

Normal stations

If the punch is round it can be rotated through any angle. If a punch is not round but say rectangular ie 45mm x 5 mm slitting tool then when it is rotated the punch angle will remain as set in the turret as shown in the following example. Please note that if the punch is square it can be rotated through 90 degree increments without adding a new tool to the turret.

Example

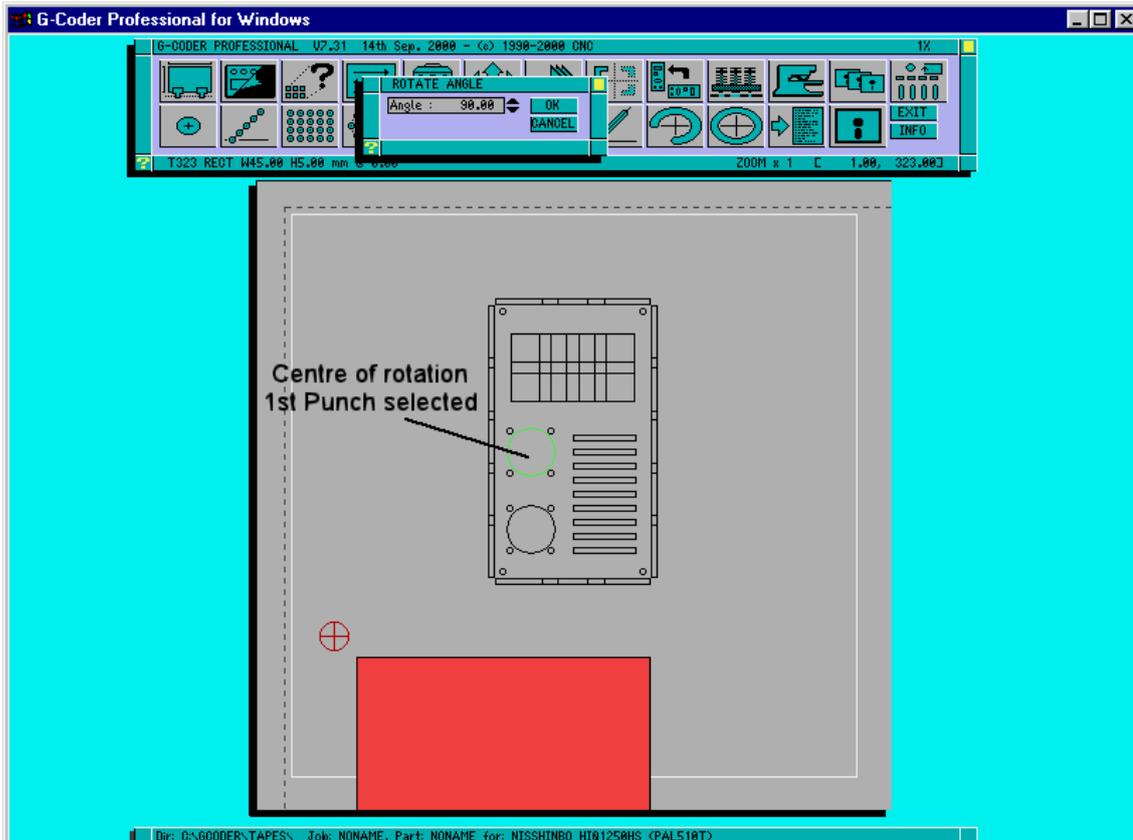
Select the Rotate Icon.



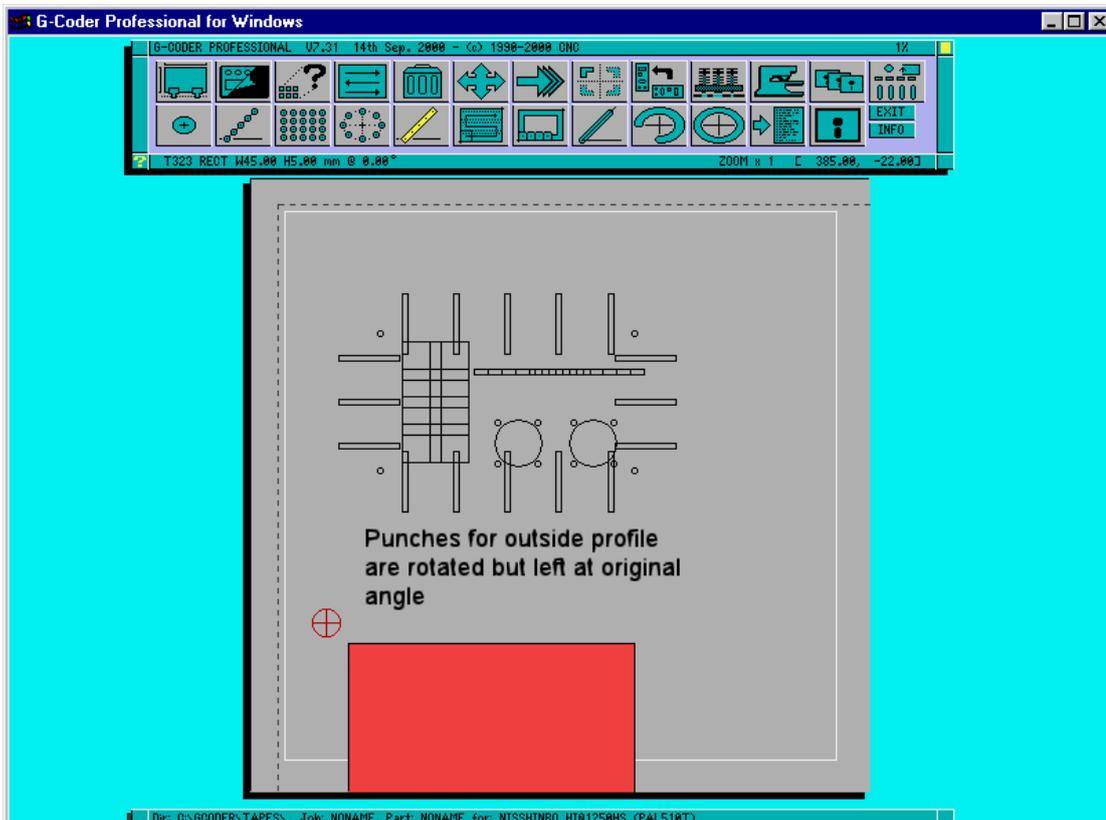
Select the large hole and then select rest of the punches using the window method. The first punch selected will be the centre of rotation.

G-CODER Professional Reference Manual

Click on the OK button



Enter 90 as the angle of rotation and then select OK. The part will be rotated through 90 degrees as shown.



G-CODER Professional Reference Manual

The punch lines for the outside shape of the component are rotated but the tools are left at the original tool angle. To remedy this use the Query Icon to select the punch and then click on the TOOL button. This will display the tool turret from which you can either select an appropriate tool or add a new tool at the required angle.

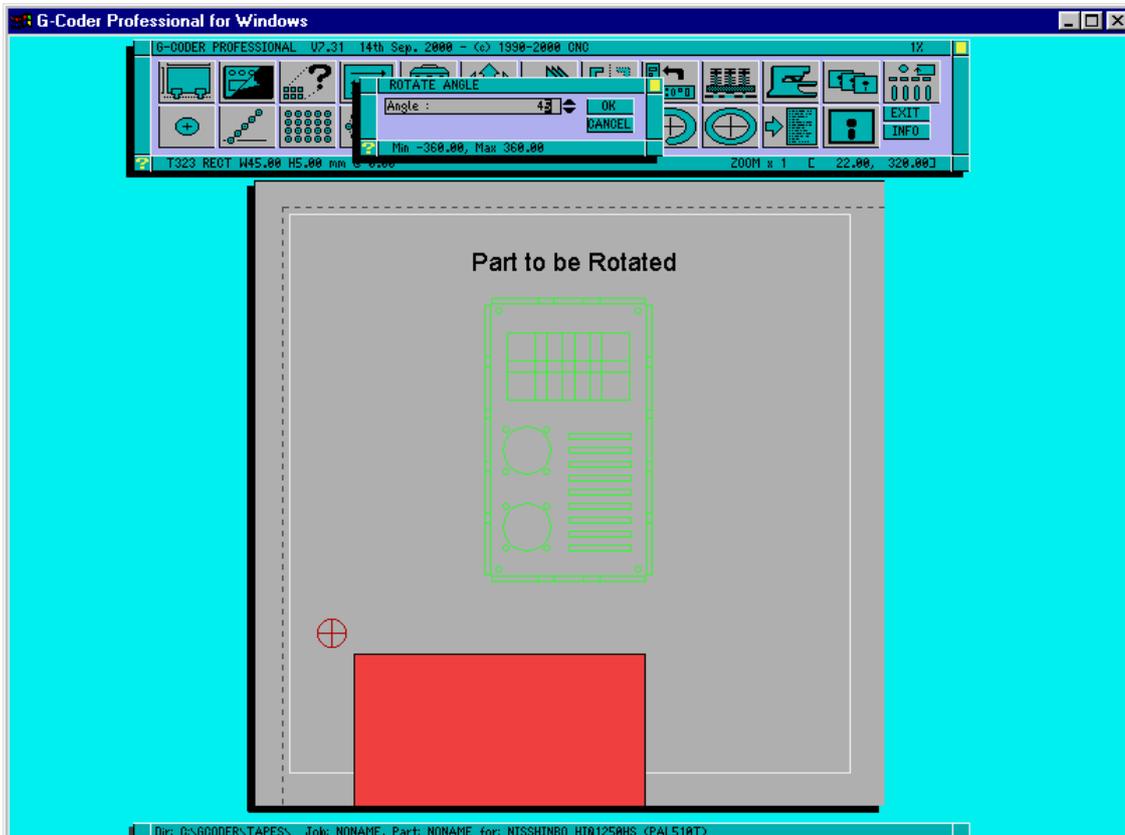
WARNING Do not alter the angle of a tool if it is being used at a different angle somewhere else in the same job.

Auto-index stations

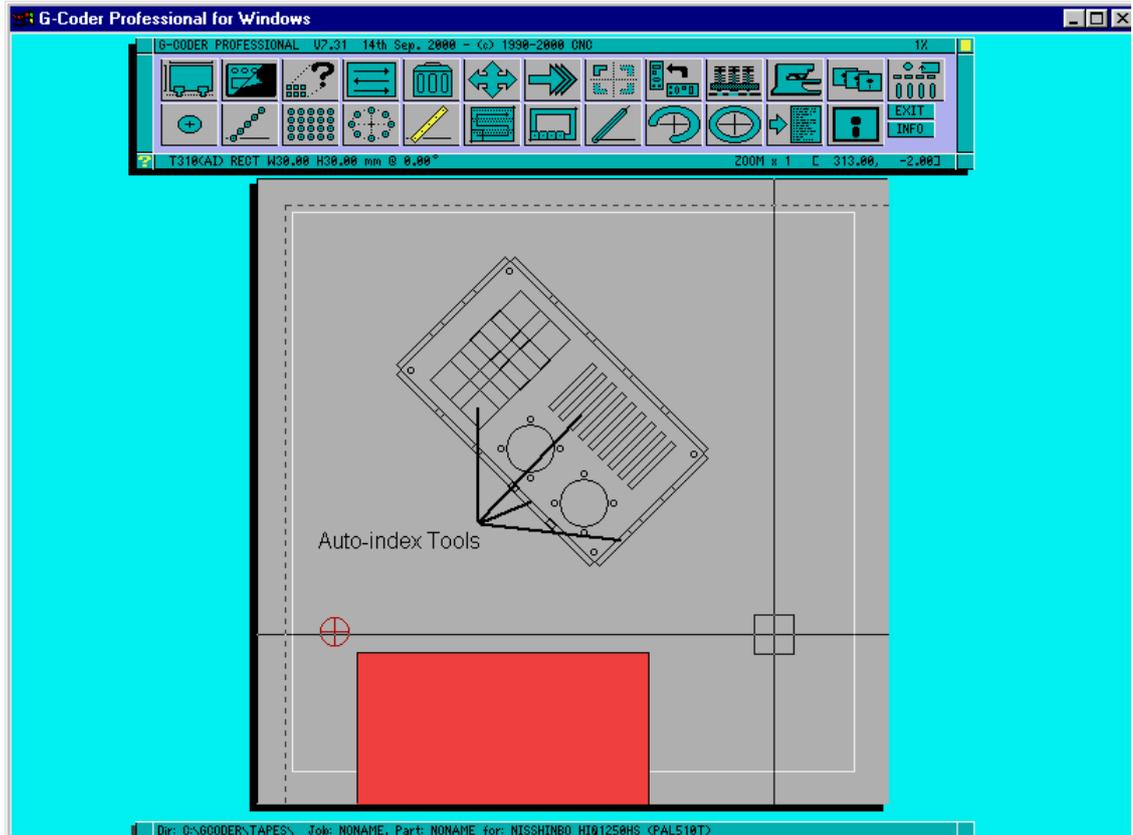
If the punch you are rotating is in an auto-index station then it can be rotated to any angle.

Example

The method is exactly the same as with Normal stations but any tool in an auto-index station will be rotated to the new angle.



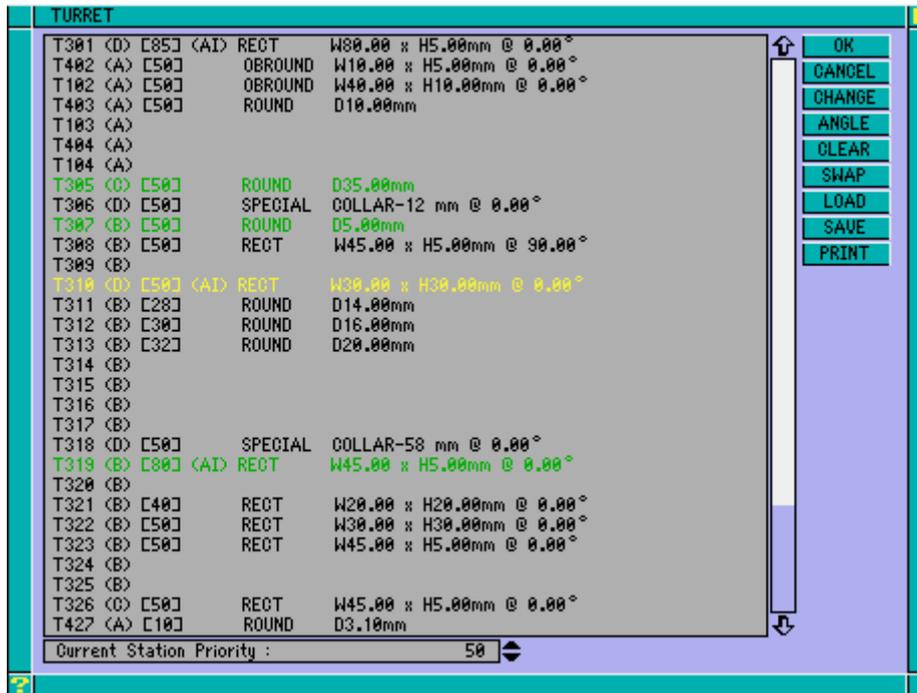
G-CODER Professional Reference Manual



Turret



This icon allows you to change the tools in your turret and add, delete or edit tools in your tool library.



Turret Layout

Your turret numbers are displayed down the left hand side, followed by the station size, the tool priority, and (AI) if it is an Auto-index station, then the description. The current tool is the one highlighted in Yellow and can be changed simply by clicking anywhere on the text of another turret. The current tool is also displayed on the left hand side of the status line below the icons. All the tools being used in the current job are marked in green. Tools marked in black have been placed in the turret but are not being used in the current job.

Turret Station Size Checking

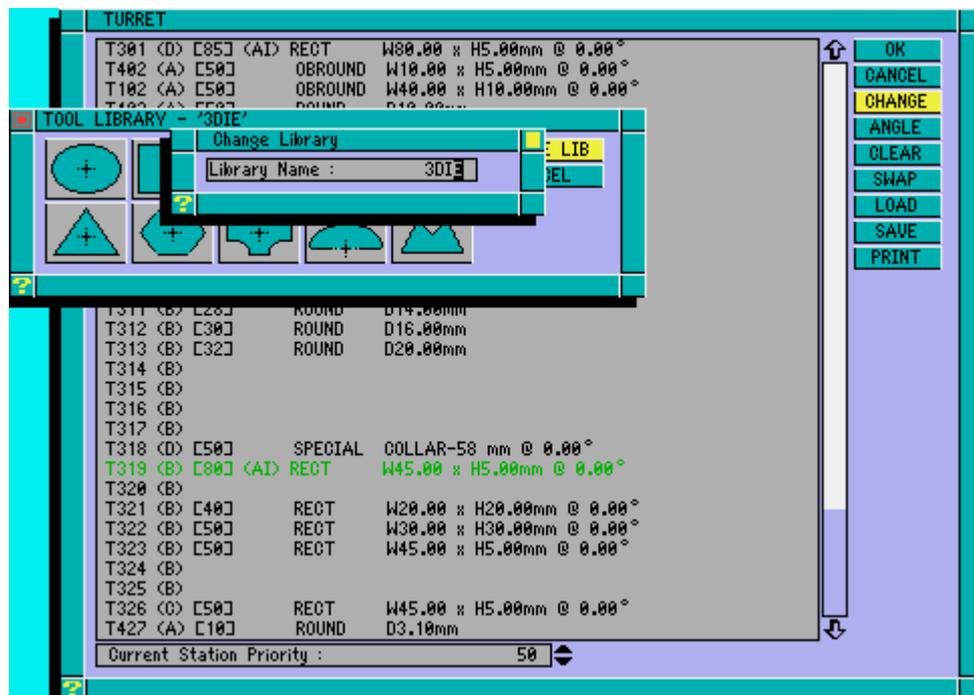
The turret window contains the station size for each turret. On some machines this is displayed as A,B,C,D or E; on other machines it is SMALL, MEDIUM and LARGE. Multi-tool stations are generally displayed as M-T.

Whenever you use the CHANGE button to load a new tool into the turret, G-CODER Professional will check the station size against the tool size. If these do not match, G-CODER Professional will issue a warning, but will still allow you to place the tool into the turret.

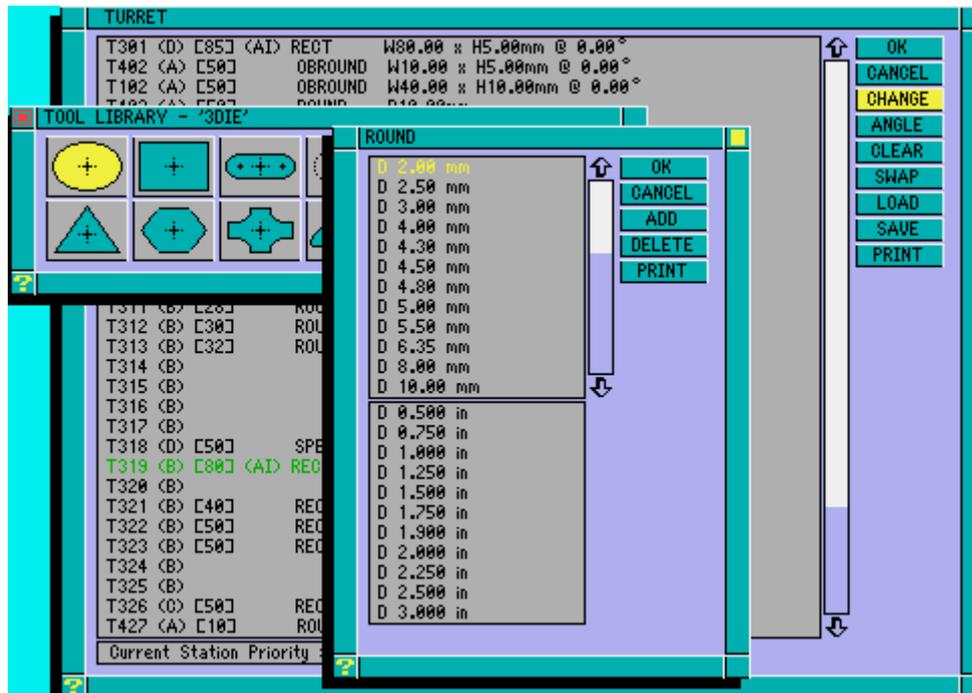
Change Button

You can change the tool in any given turret by clicking on the CHANGE button. This will then display another pop up window with the standard tool shapes. There is a button for CHANGE LIB, by clicking on this button you can type in the name of the Tool Library you wish to make active. Multiple tool libraries are useful as you could setup separate tool libraries for tools that have different die clearances. If for example you named a library 3DIE, then by changing to it, only the tools with a die of .3 clearance will be displayed. The default tool library is called 'STANDARD'. If you try to change to a tool library that does not exist then you will be prompted if you wish it to be created. If you click on the Yes button then the tools from the 'STANDARD' tool library will be copied and you can then Add or Delete tools from it in the normal way.

Once you have selected a tool library or used the default 'STANDARD' library then you can select the tool shape required. If for example you selected the round tool then all your tools of that shape will be displayed in a window similar to the following example:



G-CODER Professional Reference Manual



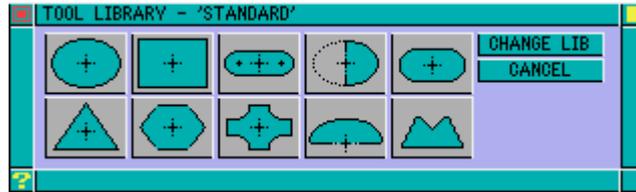
Select a tool from either the metric or inch tool library, click on the OK button and the tool will be placed into the current turret. If you press escape or click on the CANCEL button the current turret remains unchanged. If the tool you want is not in the list you can click on the ADD button and enter the dimensions of the tool to add. Similarly you can delete tools that you no longer use.

It is a good idea to add all the tools you use regularly as soon as you start using G-CODER Professional.

The tool is placed in the turret at an angle of 0 degrees. If you want to change this then click on the ANGLE button and enter the tool angle. If you wish to clear a turret completely then click on the CLEAR button and the turret will then be cleared. G-CODER Professional will not allow you to clear a turret which is being used in the current job.

Standard Tool Types

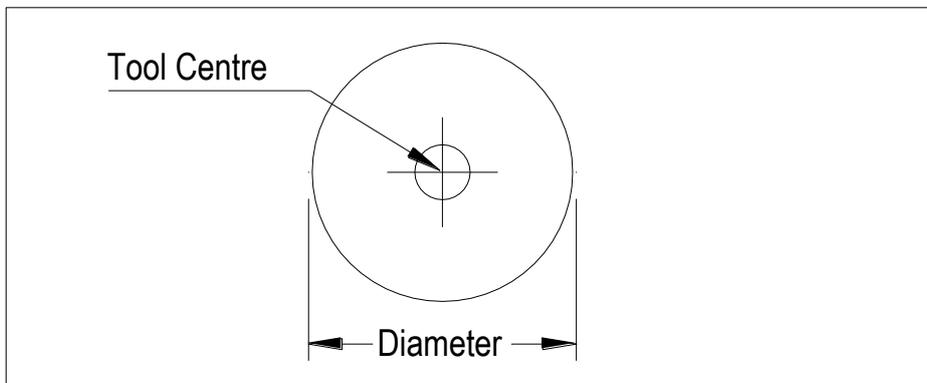
When the Change button is selected the following window is displayed. From this window you can select the shape of tool you wish to place in the turret.



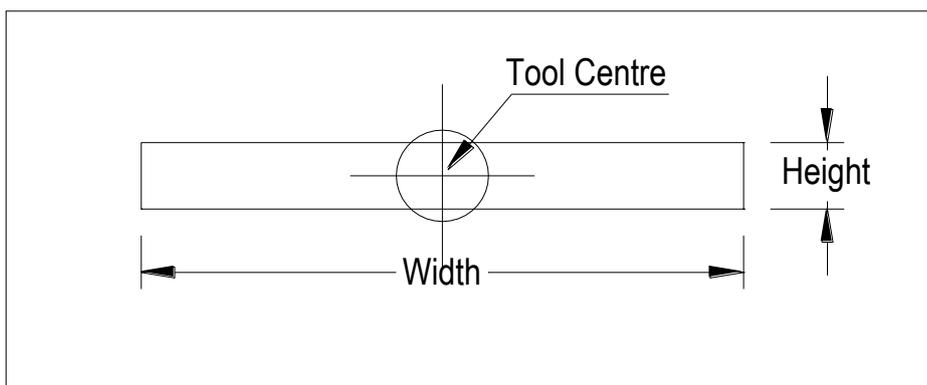
If the tool is not listed in the library then you can use the ADD button to enter the tool information. When entering the tool dimensions it is important to always enter the largest edge of the tool as the WIDTH field, otherwise problems will be experienced when using that tool. For example to define a 45 x 5 slitting tool it must always be defined as WIDTH 45, HEIGHT 5 and its ANGLE set to either 0 or 90 degrees in the TURRET window.



Round Tool – Specify Diameter

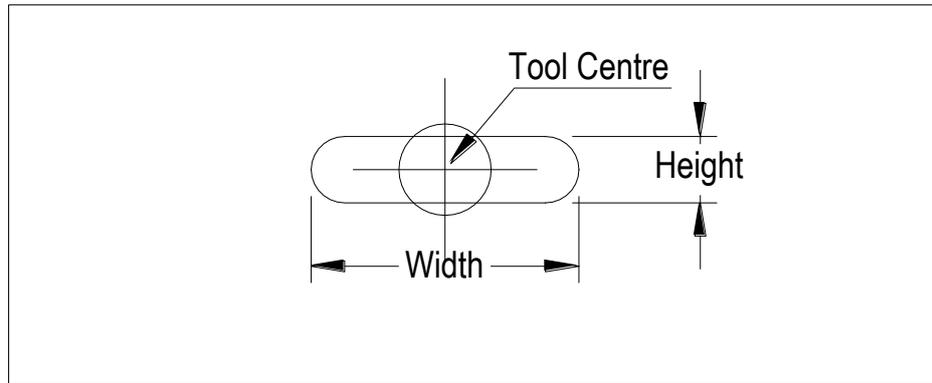


Rectangular Tool – Specify the Width and Height

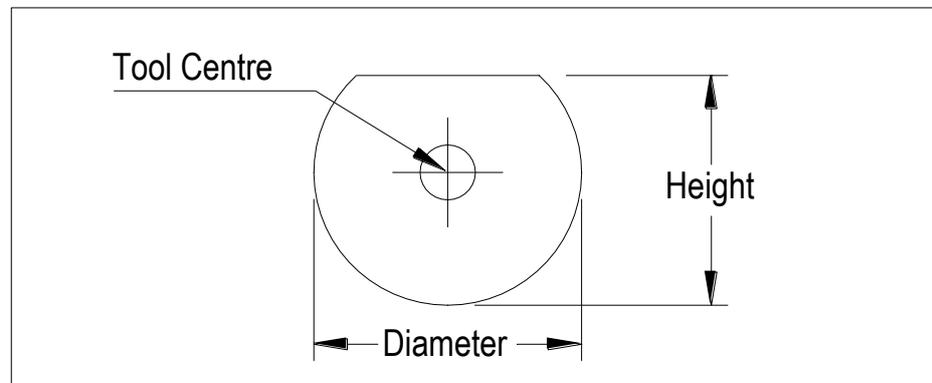




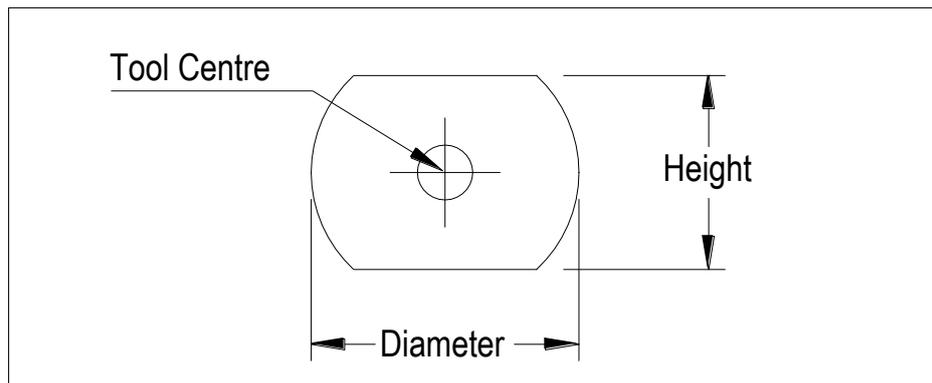
Obround – Specify the Width and Height



'D' Shape – Specify the Diameter and Height



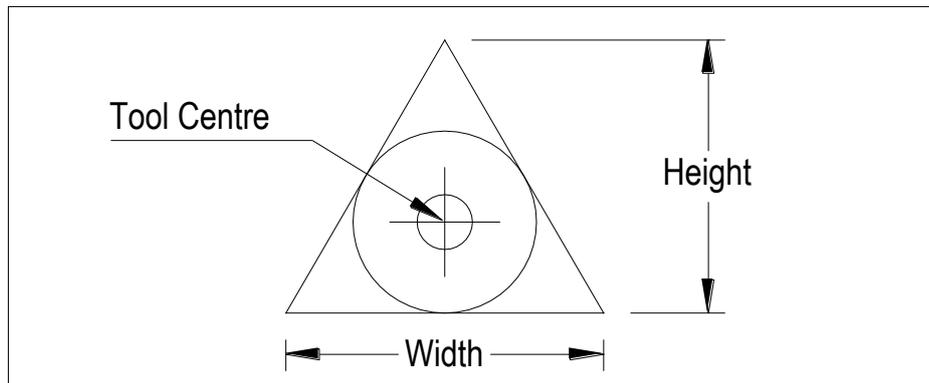
Double 'D' Shape - Specify the Diameter and Height



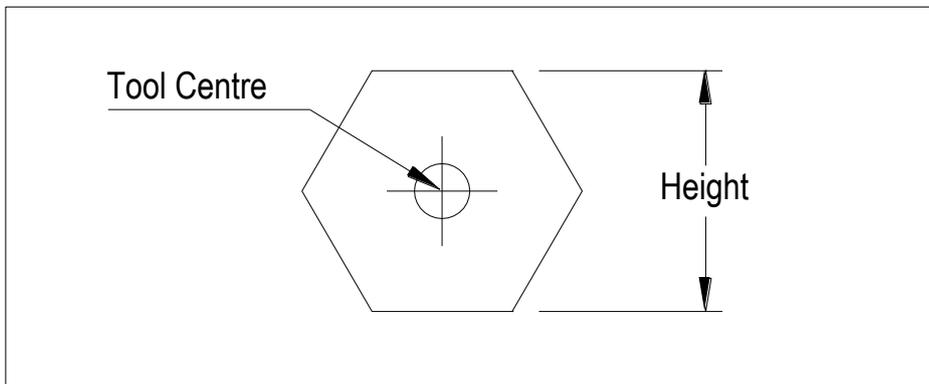


Triangular – Specify the Width and Height

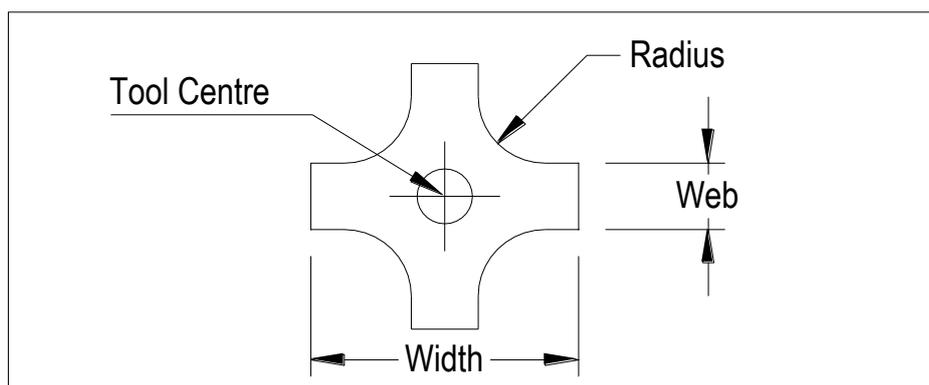
Centre of tool is at the centre of an inscribed circle.



Hexagonal – Specify Height

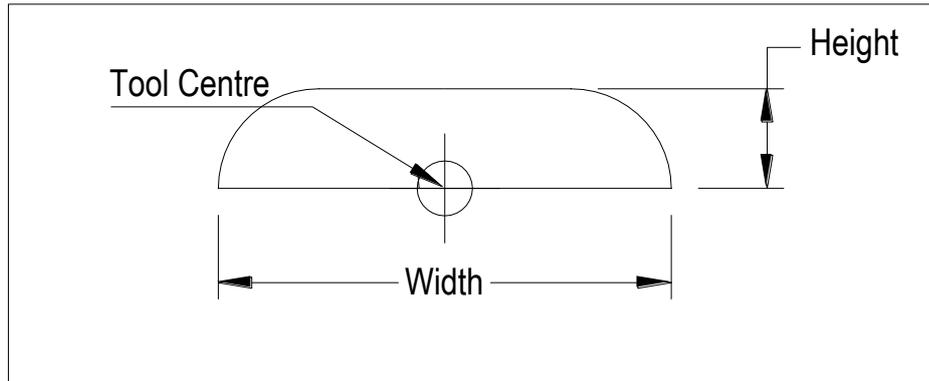


Crucifix – Specify Width, Web and Radius





Louvre – Specify Width and Height

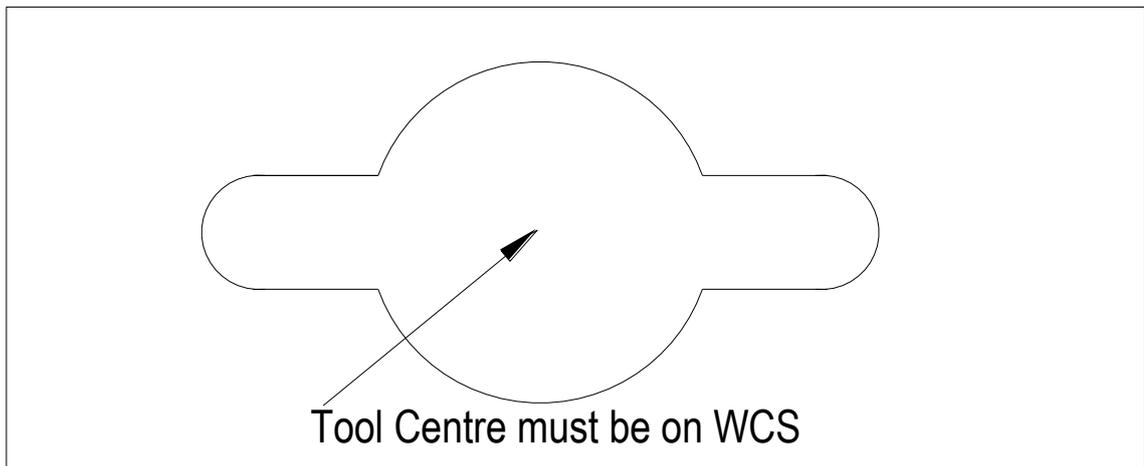


Special Tool Types

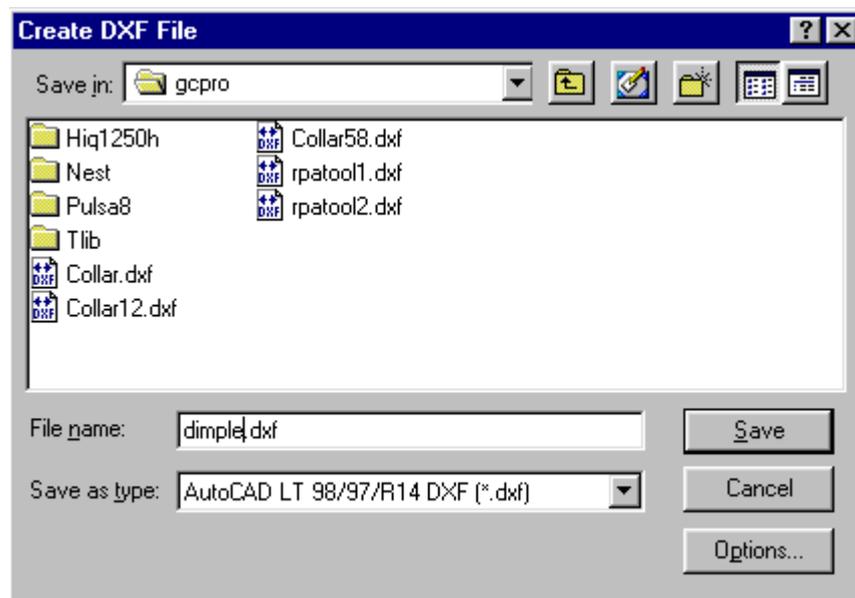


Special – Draw tool in AutoCAD

If the tool you wish to define does not conform to the standard tool types then it must be drawn as a special using a CAD system such as AUTOCAD. It is important to draw the tool with its centre on the World Co-ordinate System (WCS). Once drawn the tool has to be EXPORTED into the GCPRO directory as a DXF file.

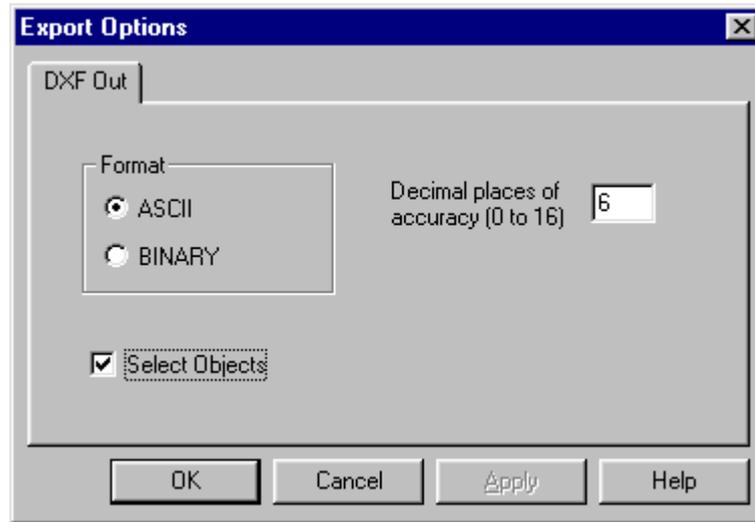


To export a DXF file from AUTOCAD type in DXFOUT at the command line. The Create DXF File window will appear, set the Save in box to GCPRO and then type in a File name eg dimple.dxf (only use a filename with a max of 8 characters)



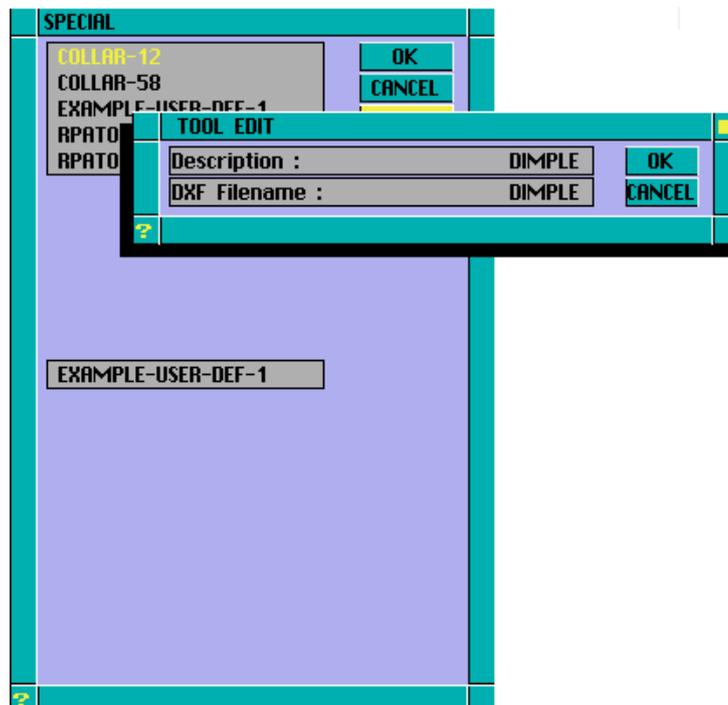
G-CODER Professional Reference Manual

Now click on the Options button and the Export Options window will appear, click on the Select Objects box and then click on the OK



The Command line at the bottom of the screen will now ask you to select objects. Type in ALL at the command line and press the ENTER key. Autocad will respond by telling you the number of objects selected, to complete the command press the ENTER key. A DXF file of the tool will now be written to the GCPRO directory.

To load the Special tool into the tool library and the Turret click on the Special Tool Button and then click on the ADD button. Type in a description of the tool eg DIMPLE and press the ENTER key. Type in the DXF Filename of the tool (must be exactly the same as the filename you used when you did the DXFOUT from AutoCAD) and press the ENTER key.



NOTE : Its usually best to use the same name for Description as the DXF Filename to avoid confusion at a later date.

G-CODER Professional Reference Manual

SWAP Button

This button allows you to swap two tools in the turret even if these tools are used in the current job.

Click on the first station to swap, then click the SWAP button, click on the second station then click the SWAP button for the second time. The two tools will be swapped in the turret and all punches will be changed from one turret station to the other.

If you click on any other button in the turret before clicking SWAP for the second time, the swap function is cancelled.

SAVE Button

G-CODER Professional now allows you to save standard turret configurations for each punch press. These can then be re-loaded at any time from any directory.

Click this button to save the current turret configuration. G-CODER Professional prompts you for a turret name and saves the file (please note any other standard configuration with the same name will be overwritten). If you have more than one punch press, then this turret configuration is unique to the current punch press.

LOAD Button

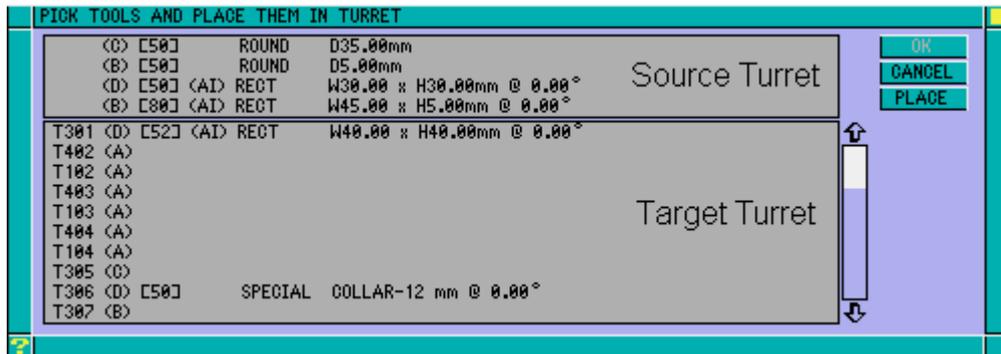
This button is only available when you do not have any punches on the sheet, as it overwrites the current turret configuration. Click this button and you will see a list of the standard turrets for the current punch press. Choose a name and click OK; the turret will be loaded and displayed immediately ready for use.

Machine Type



This icon only functions if you have more than one type of punch press and allows you to select the current machine or change jobs from one punch press to another. The current machine type is displayed in the Machine Information Line at the bottom of the screen.

If punches have been punched on the sheet or you have loaded a previously programmed job then selecting this icon will allow you to change it to a different machine. A window will appear like the one below showing the source and target turrets. The source turret is for the current machine and the target turret is for the machine that you wish to change the job to.



You now have to copy the tools from the source turret to the target turret. Start by pointing the cursor at the first tool in the source turret and click the mouse button. Then point to the turret number in the target turret that you wish to place the tool and click the mouse button again. Now click on the PLACE button to copy the tool to the target turret. As each tool is copied they are marked in red. Continue selecting and placing until all the tools have been placed into the target turret. You can now select OK and the program will be converted to the new machine.

You will have to take a number of things into consideration when changing from one machine to another such as travel limits, number and sizes of turret, clamp positions. However once a job has been converted to a new machine it can be changed to accommodate most of these differences. For example if the job required a re-position move because the target machine has a smaller travel limit in X then the post processor could automatically make the necessary allowances.

Parts



This icon allows you to save standard parts so that you can later append them to your current job to make use of any scrap material. It can also help to reduce stocking levels of standard parts by manually nesting of dissimilar parts on a sheet.

To create a part, start a new job and then program the job as normal making sure that you have only the one part on the sheet that you want to save. Click the sheet parameters icon and fill in the material type and thickness then click OK. Click the Parts icon to see the following window.



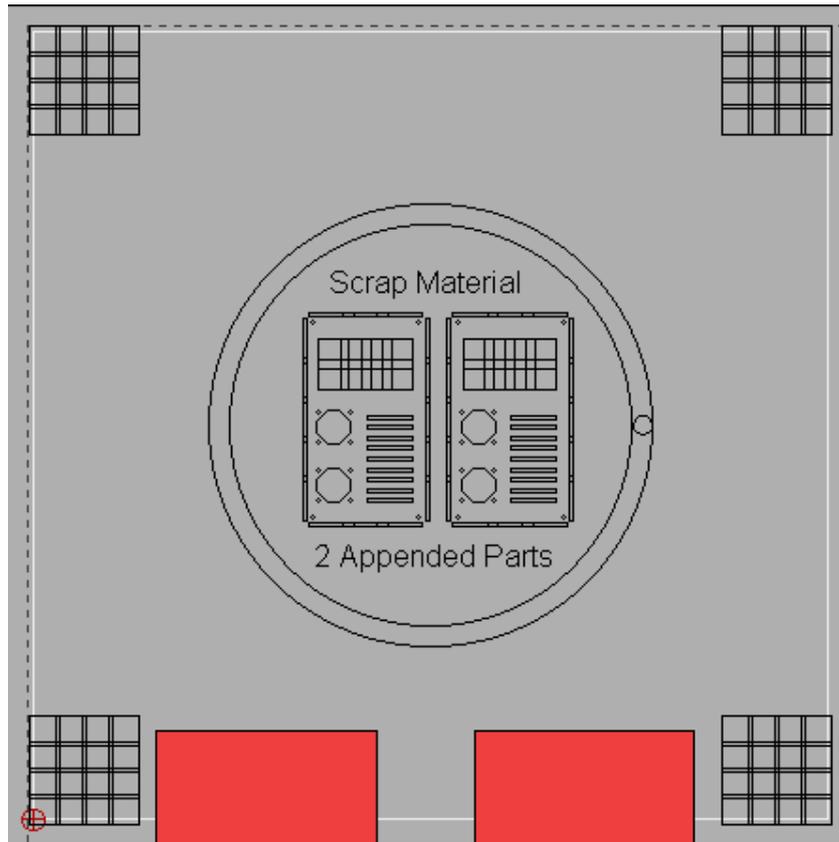
You can use the DIRECTORY button to change directory (you may want to save parts in a separate directory from your jobs) then click SAVE AS PART. Type in a name for the part (this can be the same name as a job file and will not overwrite the job) then click OK. Any geometry on the sheet will be saved with the part file. It is essential that any parts you want to nest with automatic nesting are saved with the geometry.

NOTE : All part files use the file extension .PRT

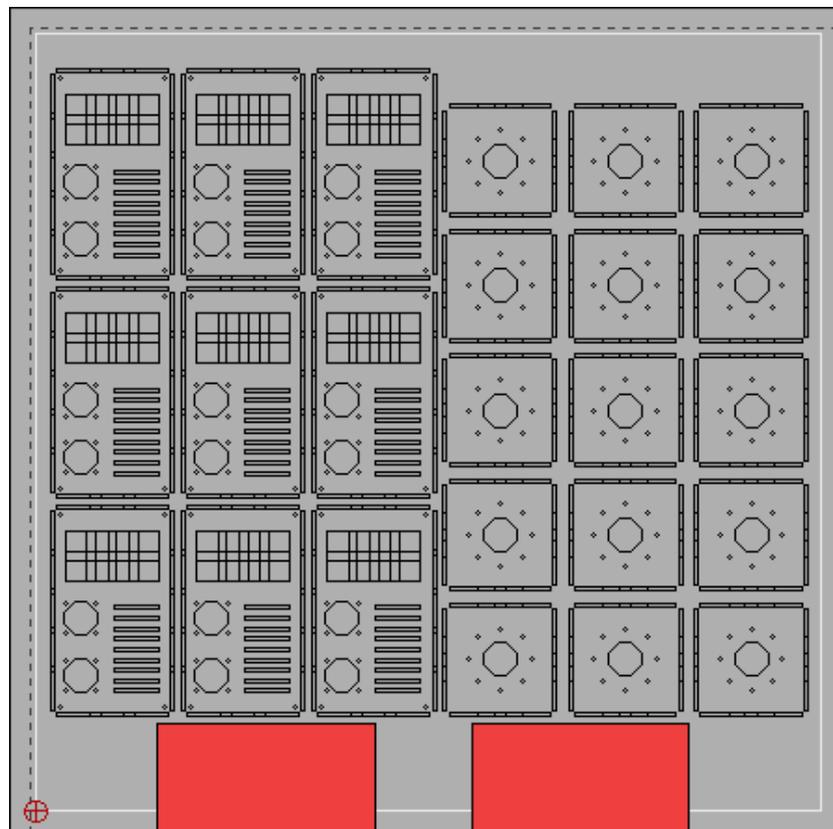
Material Type and Thickness

Make sure that you specify the material type and thickness in the sheet parameters window. When a part is saved this information will be stored with the part file. Later if you append a part in another job it will check that its material and thickness properties are the same, if not a warning is issued which you can override if required.

Part Examples



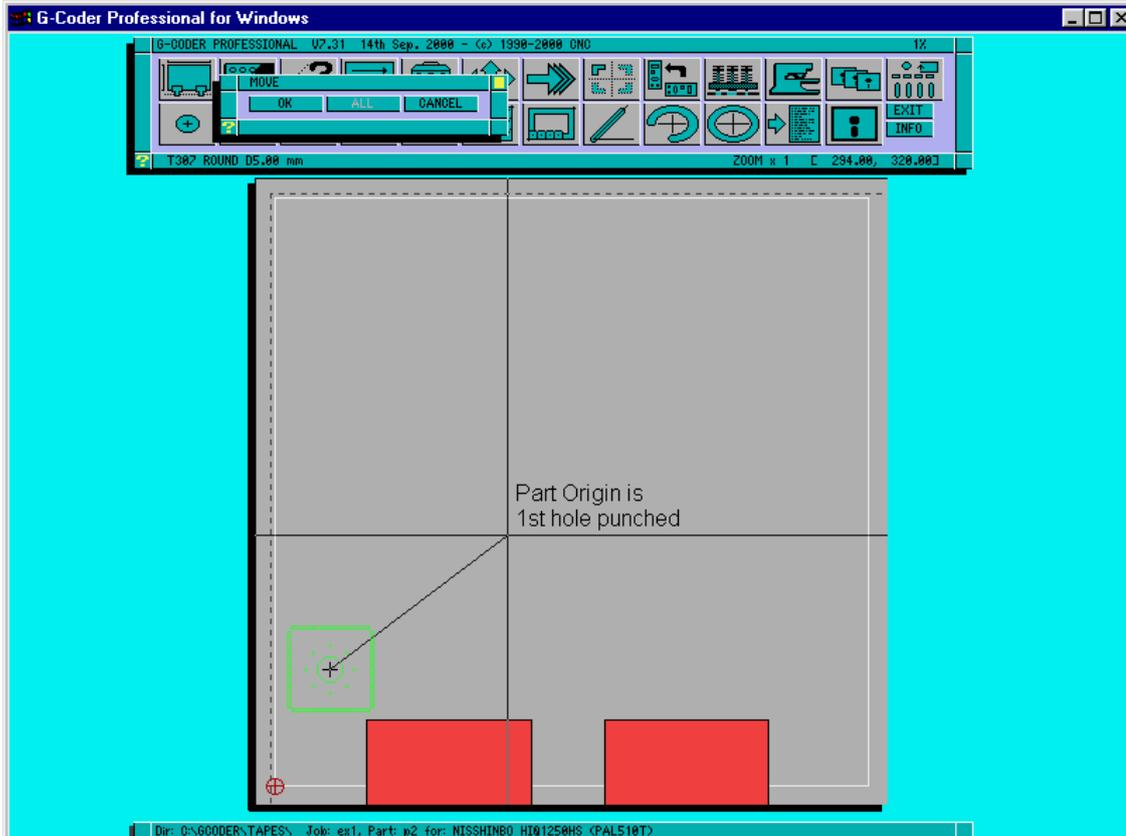
NOTE : In this example of using up the scrap material you must make sure that the tool that is nibbling the circular cutout has a priority level set so that the appended parts are punched first.



G-CODER Professional Reference Manual

Append Part

Use the Append option to load a previous defined part into your current job. Select the part required by moving the cursor over its title and then clicking a mouse button. When loading the part, the turret is checked to see if all the tools used on that part are available, if some tools are not in the turret then a list of tools requiring placing will be displayed. Click on each tool in turn and then select an appropriate tool station in the machine turret list.



If you have set the material type and thickness under the Sheet Parameters Icon then you will be warned if you try to load a part of dissimilar material specification. You can override this warning if you consider the difference in material specification not important.

NOTE : If there are not enough available tool stations in your turret a error message will be displayed. When placing tools you must ensure that the station is large enough to accept the tool.

After all the tools required for the appended part are in the turret then the part will be drawn and you can either use the cursor, spacebar or shift spacebar to position the part on the sheet.

Save Part

Use SAVE to save any changes you have made to a Part once you have used SAVE AS to give the part a name. If the Part already exists on disk in the current directory a message to OVERWRITE will be displayed. Click on YES if you wish to save changes to same Partname or click on NO to abort operation.

Save As Part

G-CODER Professional Reference Manual

This button is used to save a new Part to disk or to save a modified Part to a new Partname. If you have to program a similar Part to a previously programmed Part then load the Part and make the necessary edits and then use SAVE AS to save as a new Part.

The process of creating a Part file is the same as creating a Job. The first punch in the part will become its origin when it is loaded.

After you have clicked on the SAVE AS button type 8 characters to describe your Part and then press ENTER. Click on the OK button to accept Partname or CANCEL to abort.

Delete Part File

Use this option to delete Parts from your disk. Point and click at the Part you wish to delete and then click on OK. Once you have deleted the Parts you no longer require, click on CANCEL to exit.



NOTE : Once a Part is deleted it is removed from the hard disk. Your only hope of retrieving a deleted Part is either from a back-up or by using a utility specifically for unerasing files. Please use extreme caution when deleting Parts.

Directory

Please refer to the explanation of Directory under the FILE ICON heading.

Nest New Parts / Repeat Last Nest

These buttons allow you to nest dissimilar parts on a new sheet - please refer to the section Dissimilar Parts nesting at the back of this manual.

Punch Operations

Most of the icons on the bottom row are punch operation icons and are mutually exclusive (you can only use one at a time). If you click on the single punch icon it will be highlighted and stay highlighted until you click on one of the other punch operation icons. All of the punch operations pop up windows in which you enter details such as length of line and so on. Once you have entered these details and clicked on the OK button, the current operation will be punched each time you enter coordinates on the sheet. This is very useful when punching the same pattern across a sheet.

Each time you enter details into one of the punch windows and click on OK these values are saved and will be redisplayed whenever you click on the same icon again.

Some of the punch operations will only work with certain tool types. For example you can only punch a rectangular cutout with a rectangular tool. If you have entered all the details into the window and the current tool is incorrect you will get an "Invalid Tool Type" error message and then you can select the TOOL button in the pop up window to change the current tool. G-CODER Professional will not allow you to click on OK until you have a valid tool. Note that the current tool is always displayed in the status line on the left hand side.

Punch Mode and Geometry Mode

G-CODER Professional uses two methods to enable you to place punches on the sheet. These are referred to as Punch Mode and Geometry Mode.

Changing between Punch Mode and DXF Geometry Mode

Once you have a DXF file displayed (in red) on the screen you can then use the Move, Copy, Query, Mirror or Rotate commands to manipulate the geometry. To do this you must change to Geometry Mode by moving onto the sheet and pressing the G key. The cursor changes colour to Red to match the colour of the geometry. Once in Geometry Mode you can only select geometry.

To get back to Punch Mode, which allows you to select punches once more, move onto the sheet and press the P key.

HINT : You can change between Geometry and Punch Mode at any time by pressing the G and P keys, even in the middle of a command such as Move, Copy etc.

REMEMBER : **G for Geometry Mode**
P for Punch Mode

G-CODER Professional Reference Manual

Punching DXF Geometry

Once you have read a DXF file into G-CODER Professional then you can punch the geometry without the need to key in coordinates. To do this you must first make sure that you are in Geometry Mode (Red Cursor) then you must select a tool from the turret and a Punch Icon (such as single punch) in exactly the same way as Punch Mode. The major difference is that you do not have to fill in all of the information in the Punch Icon Window and you do not have to type in the start coordinates of the Punch Operation. The way lines and arcs were originally drawn (the direction) is also irrelevant as G-CODER Professional will always start at the end of the line or arc nearest the cursor.

Once you have filled in the relevant information in the Punch Icon Window and OK'd it you simply select the geometry that you want to punch by holding down the shift key and clicking the mouse button next to it. G-CODER Professional will snap to the nearest valid geometry item and punch it with the current tool.

Auto-index stations

If the current tool selected is in an auto-index station then several of the punch operations require additional information about the angle of the tool. The punch operations that currently support auto-indexing are Single Punch, Punch Row, Punch Grid, Punch Bolthole and Punch Line. You will find an explanation on the effect of auto-indexing on each of these punch actions in the following descriptions.

Auto-Punch



Before using the Auto-Punch feature you must load DXF Geometry onto your sheet using the Read DXF Button in the File Icon window.

To use the auto-punch feature, proceed as follows:

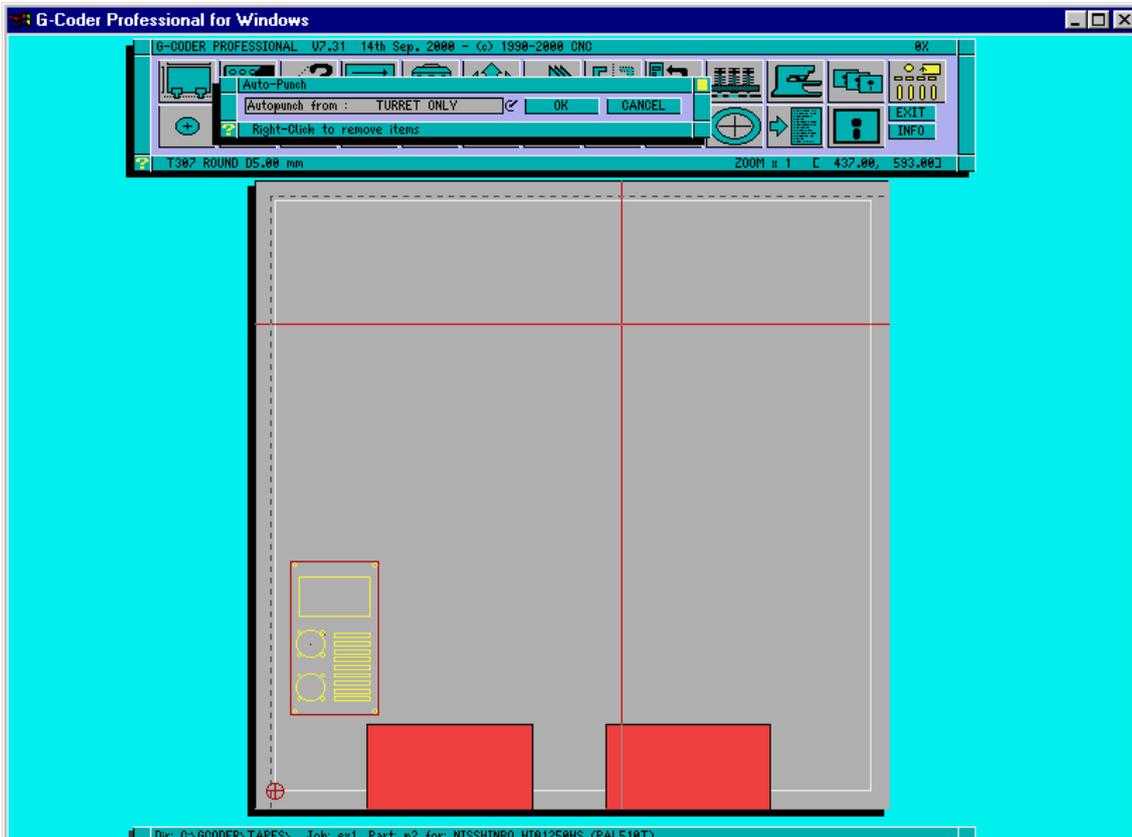
Read in a DXF file that you want to punch.

Go into the turret and LOAD your standard turret file (if required).

Click the Auto-punch icon (top right).

All Round holes (including two joined semi-circles), rectangular holes and obround holes will be highlighted in yellow. These are the shapes that the auto-punch has recognised.

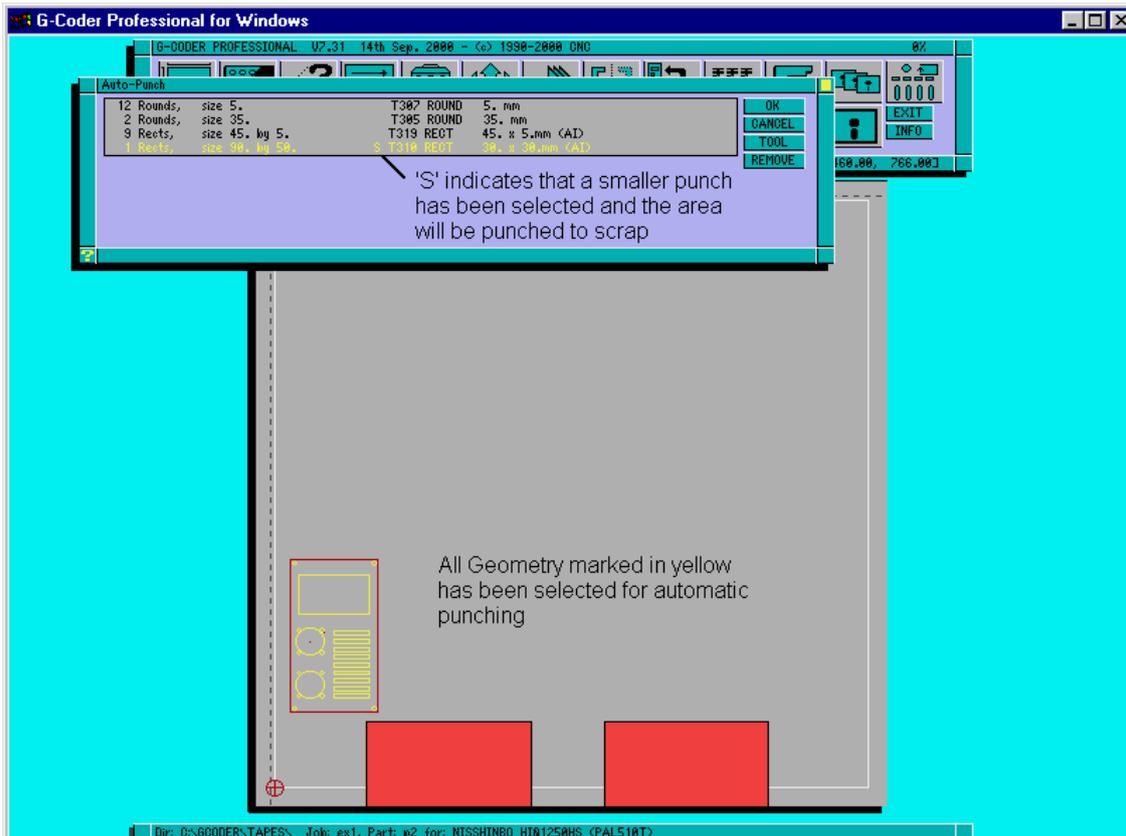
The following window will be displayed:



You may now use right mouse click to remove any shapes that you do not want to punch or use ALT-click to drag out a window and right click to remove all shapes within the window. These shapes will then be redrawn in red.

Click on the Autopunch switch in the window to choose tools from the turret only, or from the turret and the tool library.

Click OK and you will see the following window:



All the shapes of different sizes/angles are displayed in a list sorted from smallest to largest. Each shape has a tool description beside it which gives the turret number and tool description. If this description says NO TOOL then the auto-punch routine has not allocated a tool for this shape.

Click on the OK button and all of the shapes which had a tool description are punched. Any shapes that did not have a tool allocated are left highlighted in yellow.

Auto-punch from turret only will only use the tools in the current turret to punch the shapes.

Auto-punch from turret and library will check the turret first for the correct tool. If one does not exist, the tool library is searched for the correct tool which is then loaded into the first available turret station of the correct size.

You may change the tools used in the shape window at any time by clicking one of the following buttons:

REMOVER Removes the tool allocated to a shape (the shape will not be punched).

TOOL This will then pop up the turret and allow you to manually change the tool used for the current shape. G-CODER Professional will only allocate this tool if it is appropriate for the shape (see list below).

Auto-punch will match the following shapes automatically with the following tools:-

Shape	Tool
Round	Round of same size Round of smaller size (punch to scrap)
Square/Rectangle	Square/Rectangle of same size at same angle (AI at any angle) Rectangle of same height as shape (punch as a line)

G-CODER Professional Reference Manual

Rectangle of smaller size (punch to scrap)

Obround

Obround of same size at same angle

(AI tool at any angle)

Obround of same height as shape (punch as line)

Round of same height as shape (punch as nibble line)

In addition you may manually change the tool as long as it conforms to the above or to one of the following:

Shape

Tool

Round

Round of larger size than shape (punch as single punch)

Any other type of tool (punch as single punch)

Square/Rectangle

Square/Rectangle where either width or height is bigger than the shape size (punch as single punch)

Obround

Obround/Round where either width or height is bigger than the shape size (punch as single punch)

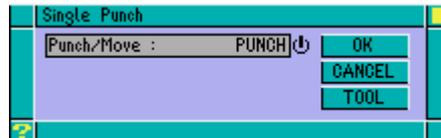
Any other combinations are invalid.

Single Punch



Punch

Choose this icon whenever you want to perform a single punch on the sheet. Click on the switch to set to either Punch, Move, Ref Point or Prog Stop. When set to punch you can then use the shift key and mouse button to punch a hole on the sheet or alternatively if you press the space bar a window will appear into which you may enter its absolute values in X and Y . Once in single punch mode you can then press shift and space bar together to punch a hole relative to the last hole punched.



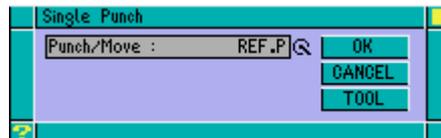
Move

When set to Move then the punch press will move to the position but will not punch. These moves will be drawn in the current punch shape in blue. You can edit these as you would a normal punch.



Ref.P

The Reference Point can be used as a datum point for Mirror, Rotate and other commands.



G-CODER Professional Reference Manual

Prog Stop

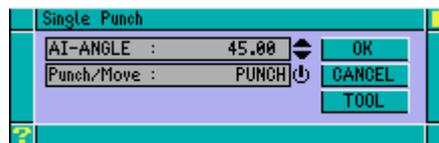
At any point in a job you can use this switch to add a Program Stop (M00) into your program, which will be drawn in the current punch shape in blue. Program stop will cause the punch press to move to the coordinates given first, then stop. You can edit these as you would a normal punch.

Note : Be careful if you use the Sort Icon to optimise the order of punching your program stop may appear at a different position in your program.



Auto-index

If the turret selected is an auto-index station then a window as shown will be displayed allowing you to enter the angle of the tool. Please note that the angle of the tool is always assumed to be set at 0 degrees in the turret. All angles are defined as detailed under the heading Sheet Conventions



Geometry Mode

You can pick any geometry item in single punch and it will be punched as follows:

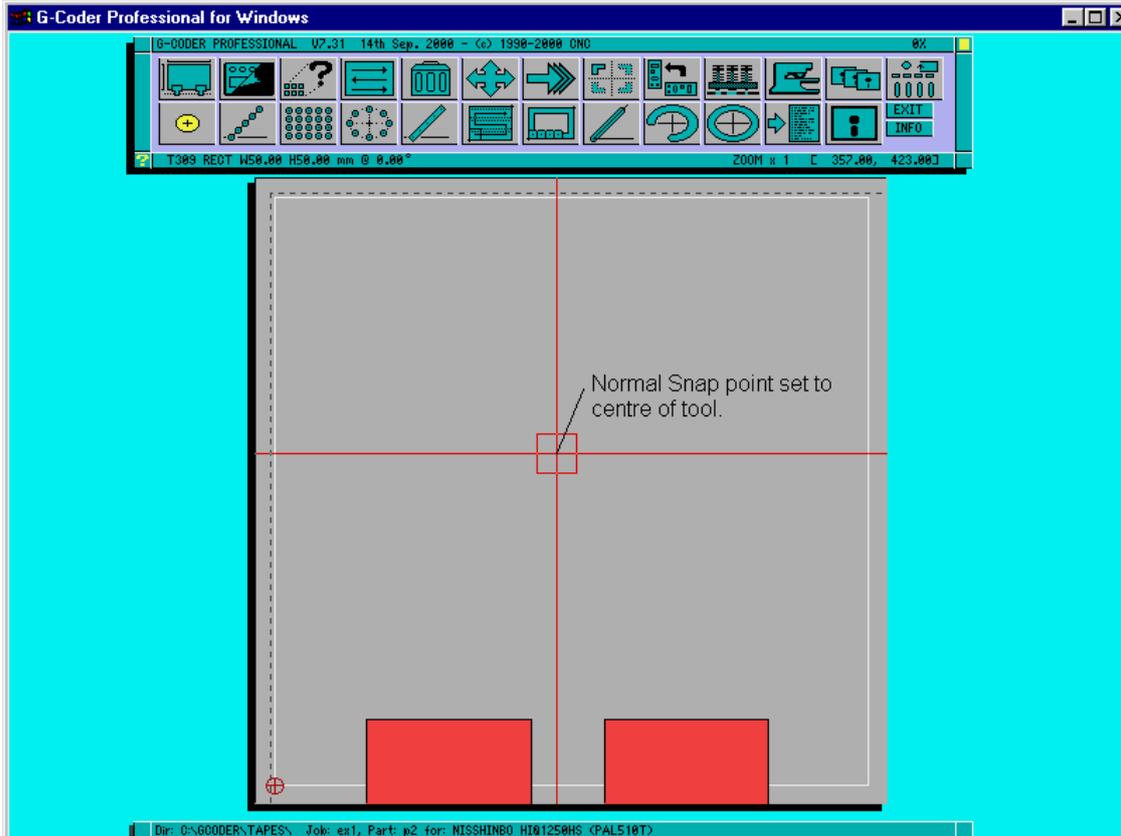
- POINTS - punch point at centre of punch
- LINES - punch the nearest endpoint to the cursor
- ARCS - punch the centre of the arc
- CIRCLES - punch the centre of the circle.

HINT : Using the QUERY ICON you can select a round hole and then click on the MARK SAME RADS button. This will mark all holes of the same size in yellow. Then by selecting the right size tool from the turret, single punch mode and pressing the A key all the holes of the same radius will be punched automatically.

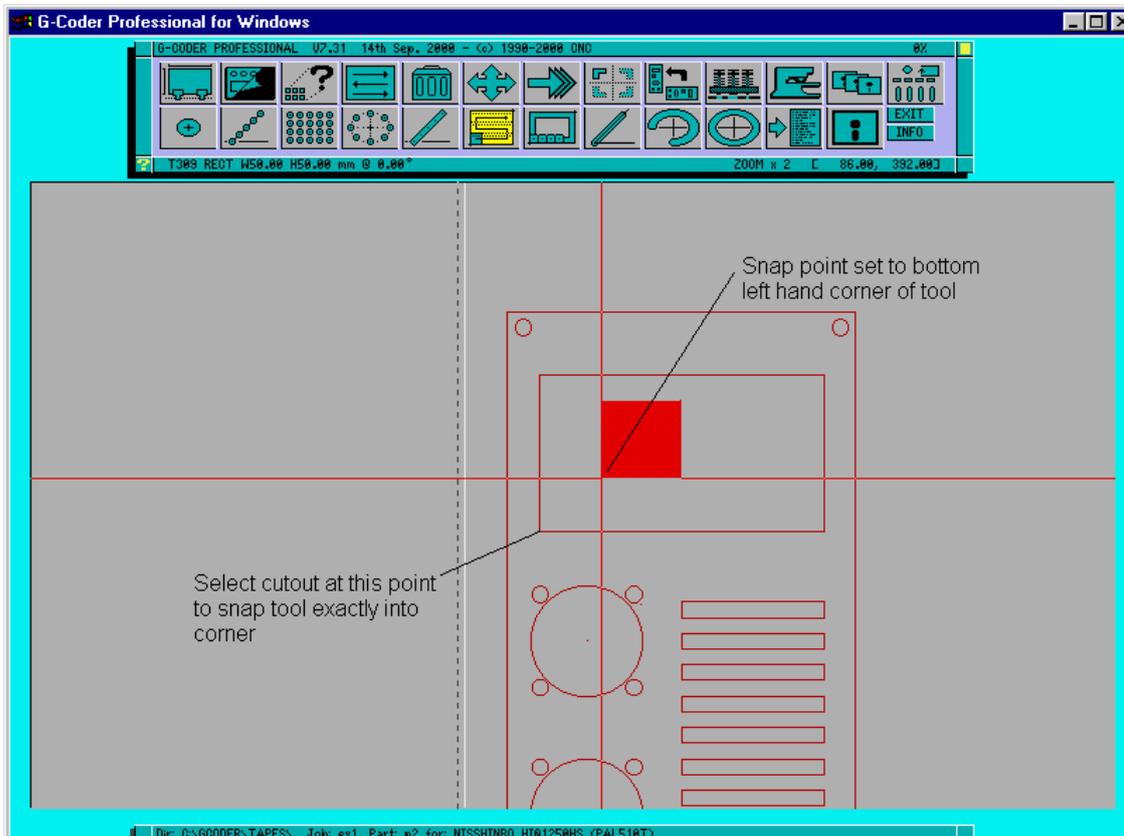
If you are in single punch mode and the current tool is rectangular the snap point of the tool can be moved from its centre to either one of its four corners by pressing the C key.

G-CODER Professional Reference Manual

Snap point set to centre of tool.



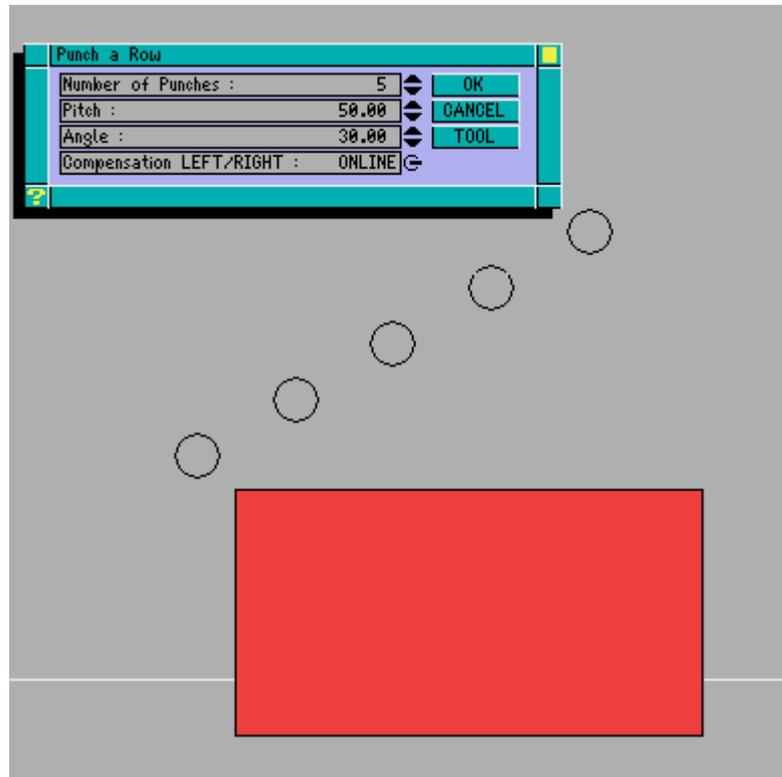
Snap point set to bottom left corner of tool so it will snap exactly into corner of component.



Punch Row



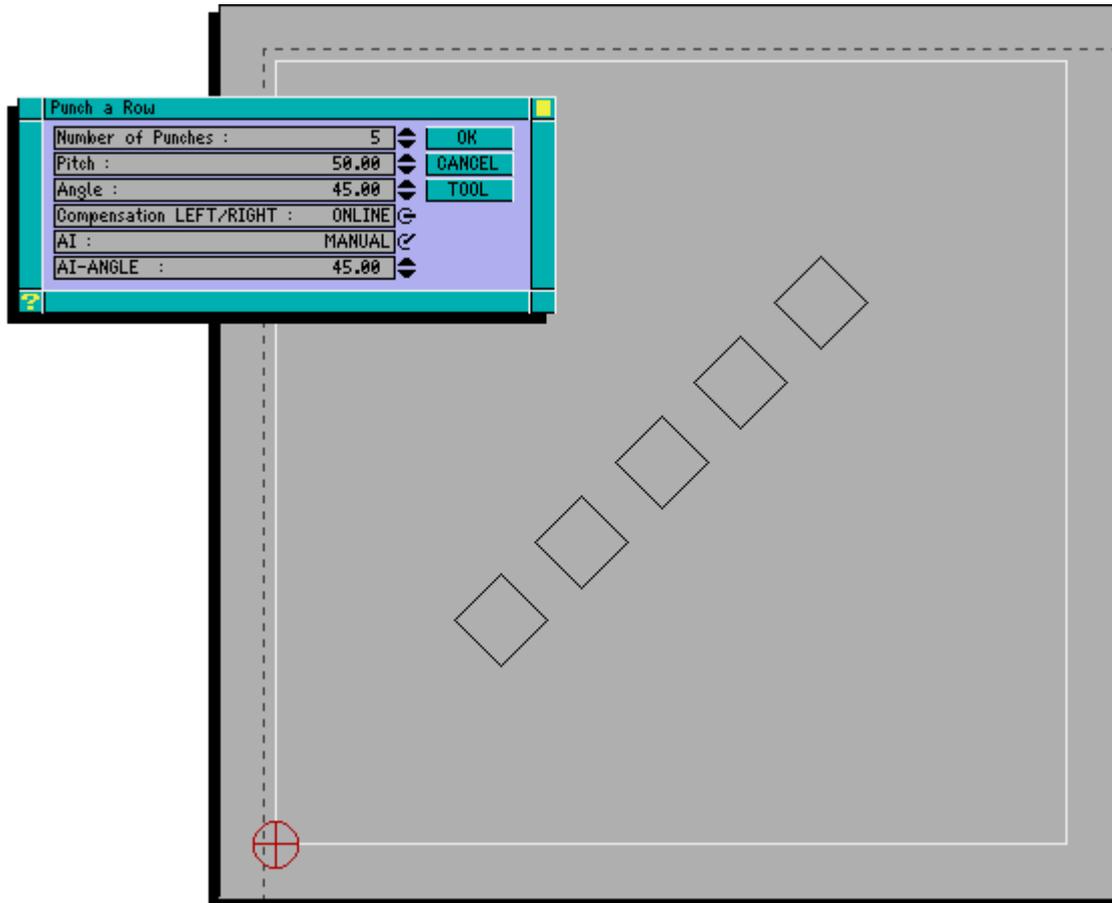
Choose this icon whenever you want to punch a row of evenly spaced holes at any angle. This operation will automatically compensate to the left or right of the line.



Hint : If you want to punch a row but exclude some punches, punch the row then Query it and use the SPLIT button to turn the row into single punches. You can then delete the holes you do not want.

Auto-index

If the turret selected is an auto-index station then the window will contain the additional lines for AI and AI-ANGLE.



If the AI switch is set to MANUAL then the angle of the tool is controlled by entering its angle into the AI-ANGLE box. If the AI switch is set to AUTO then the angle of the tool is automatically set to the angle of the row.

Geometry Mode

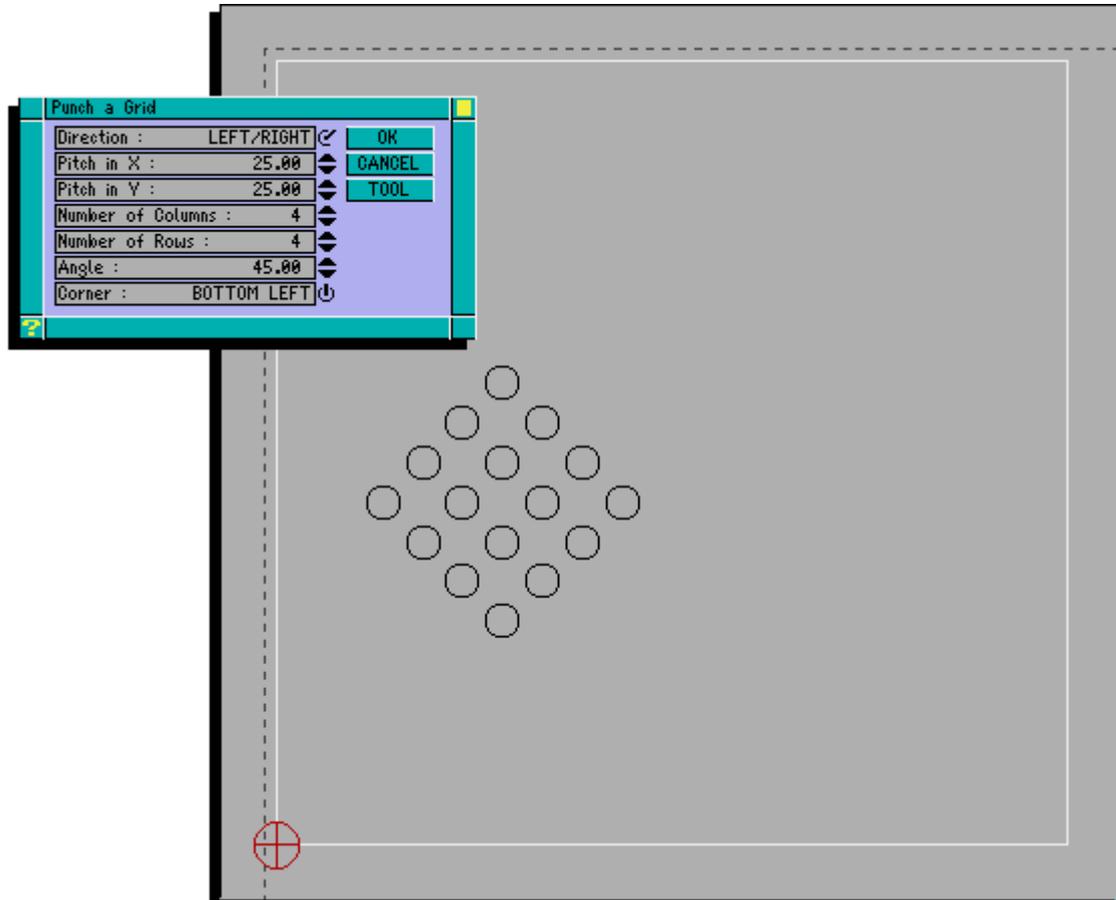
You must fill in all of the information in the Punch Row Window and then select any valid geometry item as the start of the row.

HINT : Use the Query Icon to Calculate the pitch and angle of holes in a row before filling in the Punch Window information.

Punch Grid



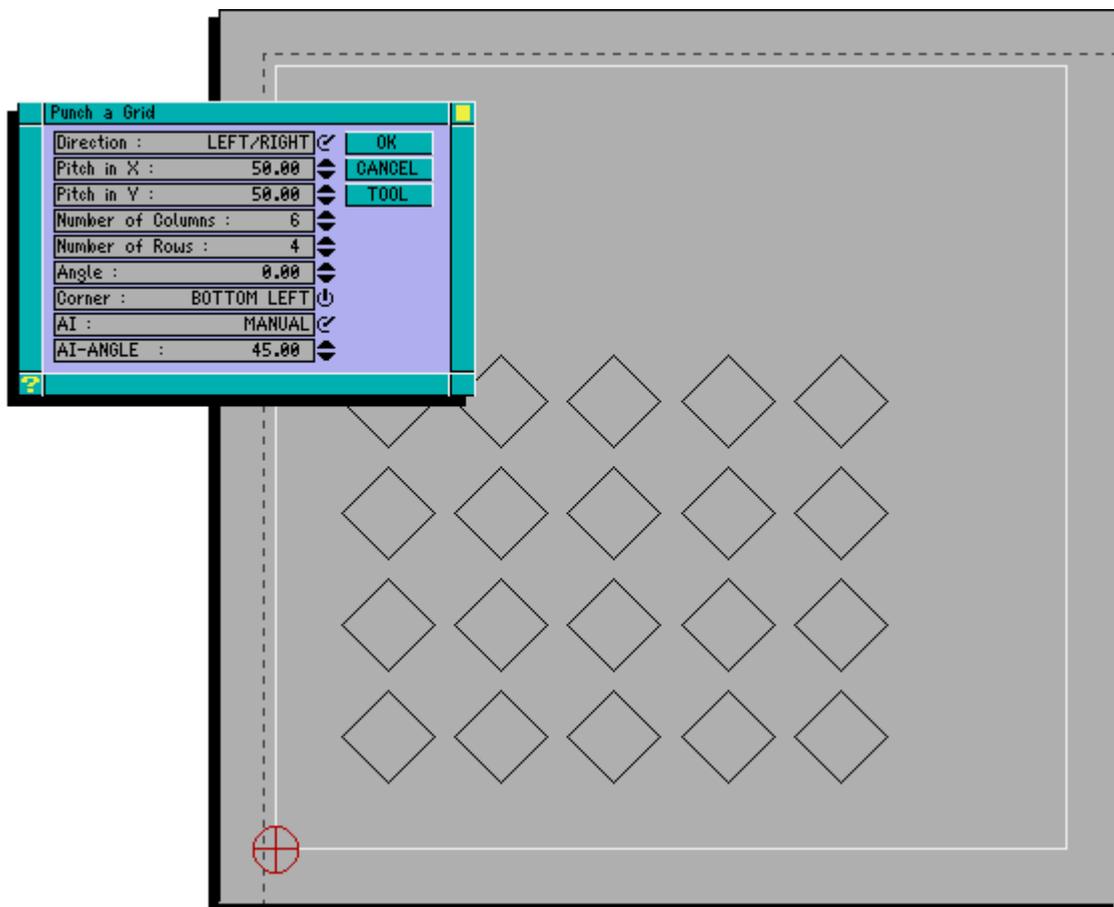
Choose this icon whenever you want to punch a grid of holes evenly spaced in X and Y. You can also choose whether to punch the rows first or the columns first and the angle of the grid.



If you want to confirm the order of the punches in the grid select the redraw icon and set the speed to slow.

Auto-index

If the turret selected is an auto-index station then the window will contain the following additional lines for AI and AI-ANGLE.



If the AI switch is set to MANUAL then the angle of the tool is controlled by entering its angle into the AI-ANGLE box. If the AI switch is set to AUTO then the angle of the tool is automatically set to the angle of the grid.

Geometry Mode

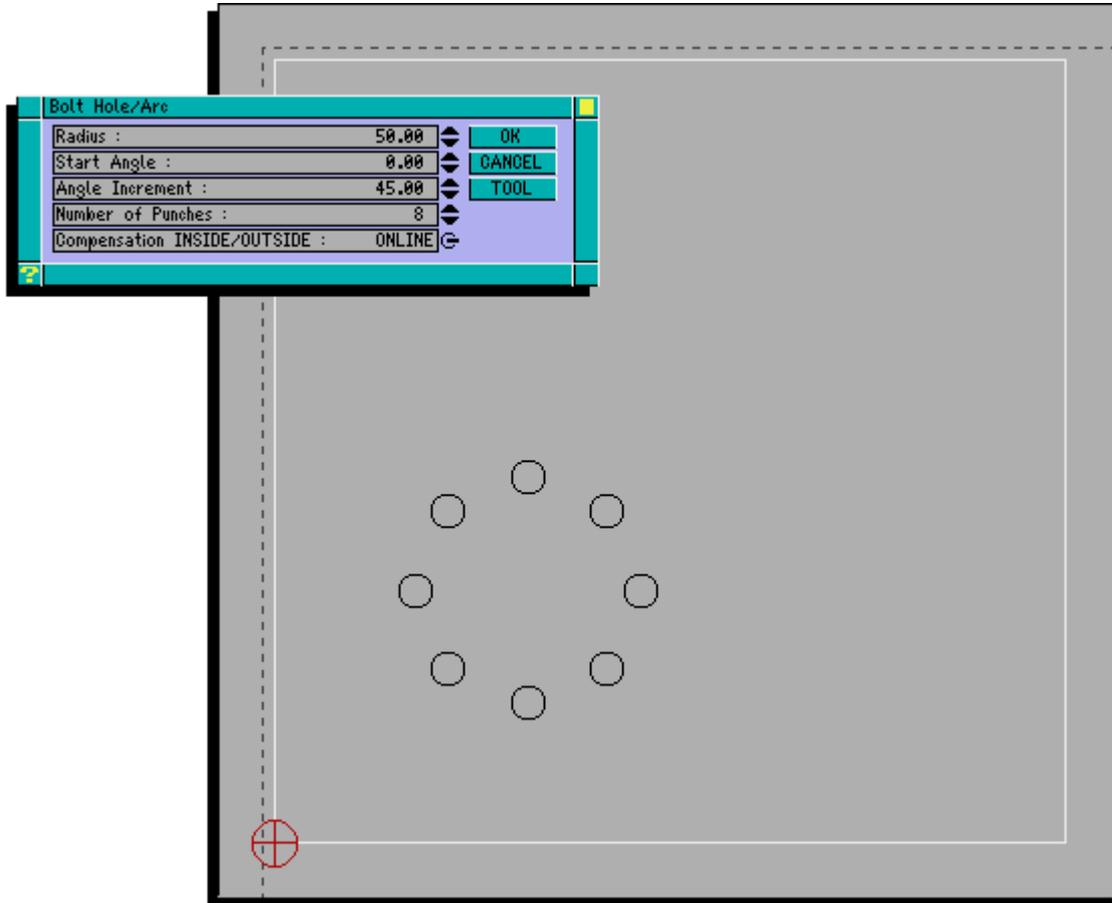
You must fill in all of the information in this window and then select any geometry item as the start of the grid.

HINT : Use the Query Icon to Calculate the pitch and angle of holes in a grid before filling in the Punch Window information.

Punch Arc or Bolthole



Choose this icon whenever you want to punch a series of holes around an arc. This operation can automatically compensate the punch centres to lie inside or outside of the radius.

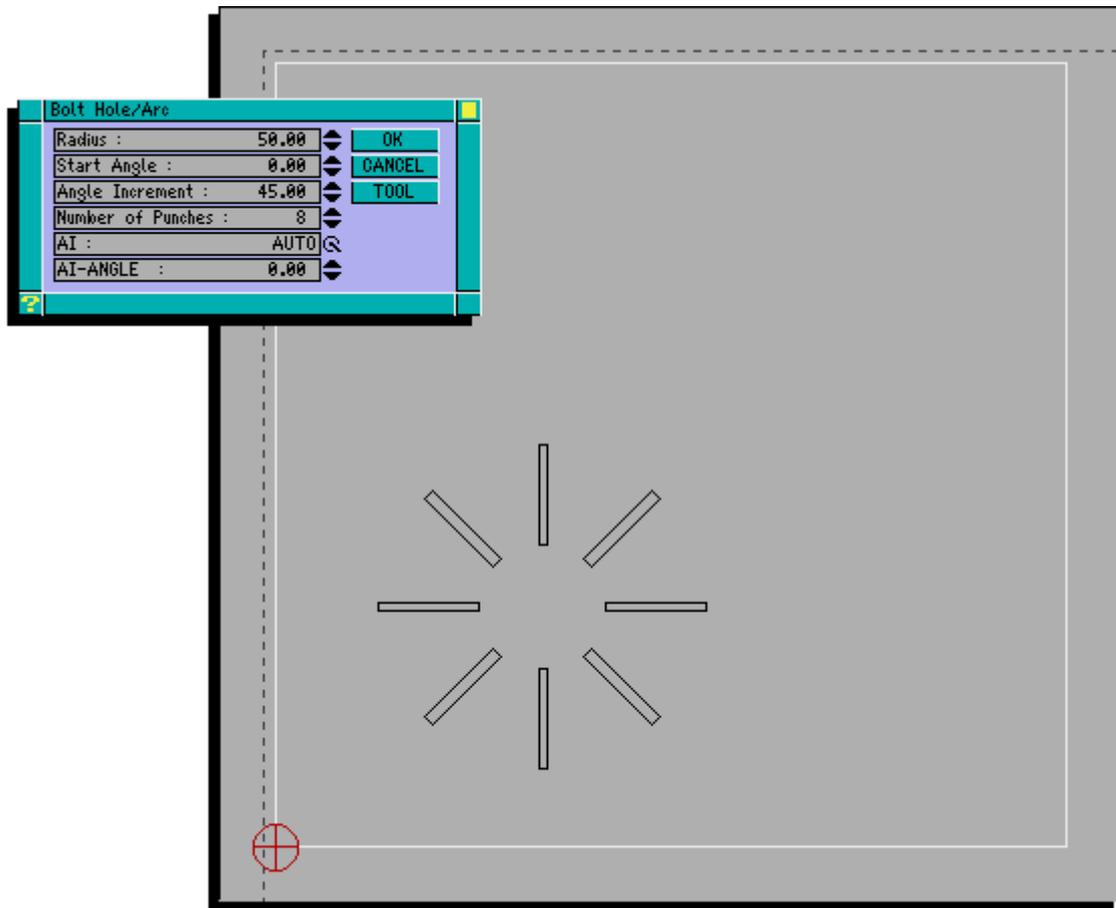


If you want to punch a complete bolt hole then you must calculate the number of punches. For example if you want a complete bolt hole starting at 0 degrees at 30 degree intervals then the number of punches is 12 (360 divided by 30).

Alternatively if you want 15 punches evenly spaced around a bolt hole then the angle increment is 24 (360 divided by 15).

Auto-index

If the turret selected is an auto-index station then the window will contain the following additional lines for AI and AI-ANGLE



If the AI switch is set to MANUAL then the angle of the tool is controlled by entering its angle into the AI-ANGLE box. If the AI switch is set to AUTO then the angle of the tool is initially set to the AI-ANGLE and will then rotate at the angle increment of the bolt hole as shown in the example.

Punching a complete arc/circle with Auto-index

You can use an auto-index station with a square tool to punch a complete arc or circle. You must first calculate the pitch and number of holes for the arc and the start angle. Fill in these details in the punch window and set the AI entry to AUTO and set the AI-ANGLE entry to the start angle of the arc or circle.

For example a complete circle of internal radius 50 with a 10 square tool would need 32 hits at 11.25 degrees.

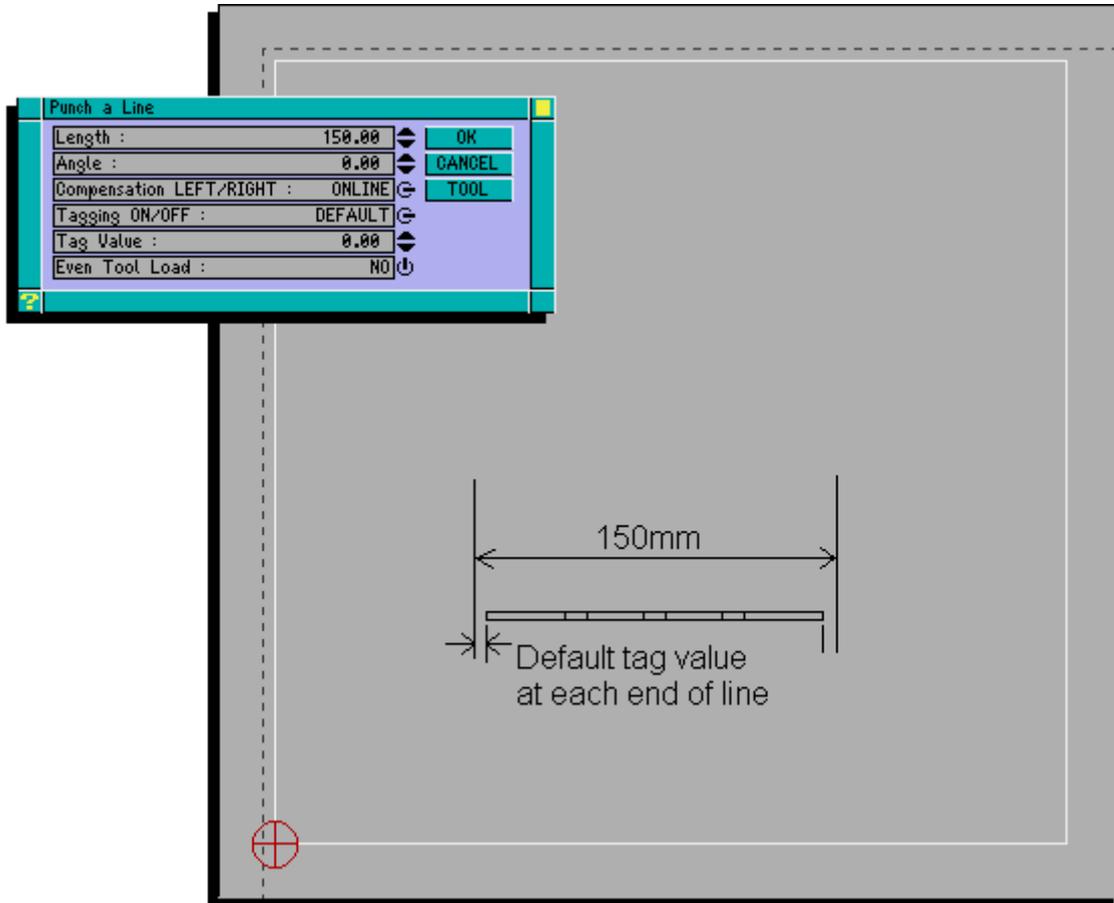
Geometry Mode

You must fill in all of the information in this window and then select any geometry item as the centre of the bolt hole or arc. Use the Query Icon to calculate the radius, start angle of the first hole of the arc and angle increment.

Punch Line



Choose this icon whenever you want to punch out a complete line with a rectangular tool of a given length for operations such as slitting between components.

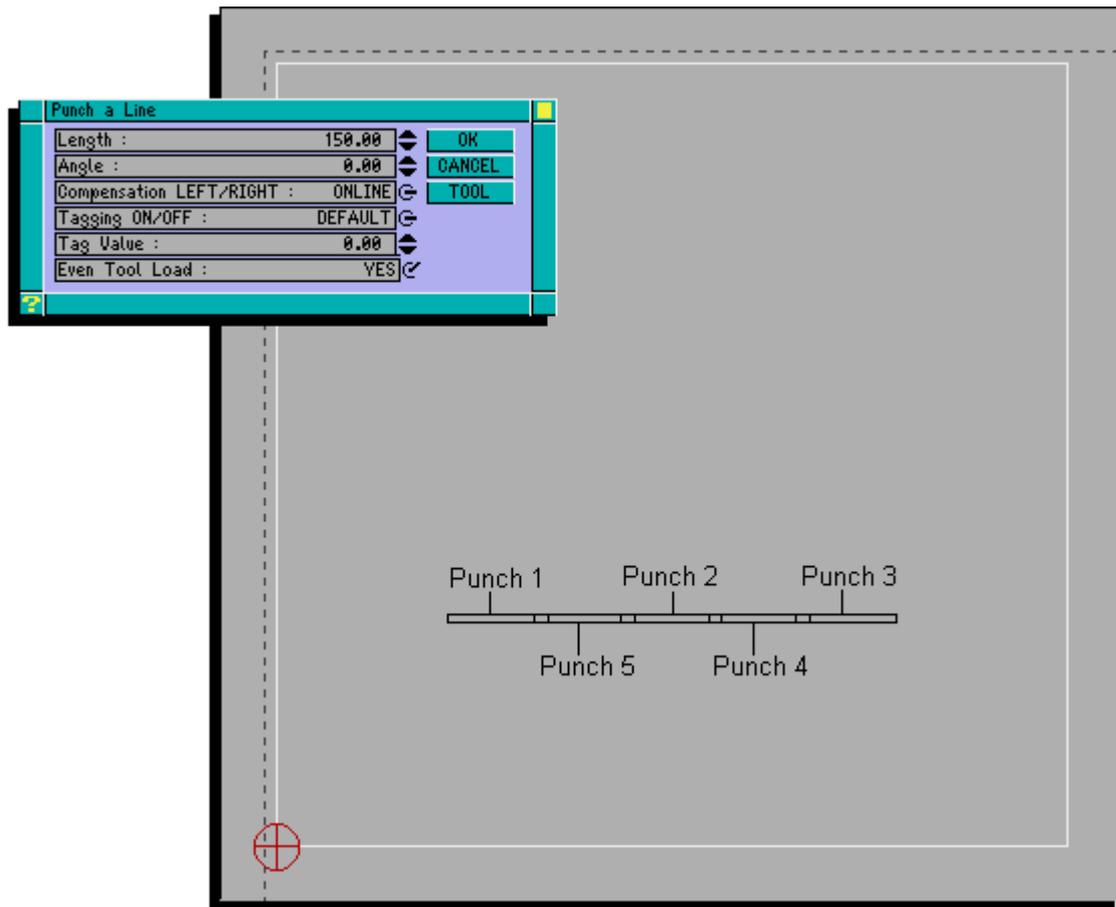


Punch line is a very useful operation as you only have to specify the start point of the line and the length and G-CODER Professional will work out all the tool centres for you. It will also apply a tag value to each end of the line and online/left/right compensation.

If you require a tag at both ends of the line then you can set TAGGING to DEFAULT and this will create a tag as specified in the sheet parameters. If you punch all of your lines that require tags in this way then you can change the DEFAULT TAG VALUE in the sheet parameters and all of the tags on the sheet will change. This is useful when you are punching a new type of material and are not sure about the tag size.

If you enter a line length shorter than the tool width then G-CODER Professional will punch a single hit compensated to the start of the line and will put a tag at the beginning of the line (if required).

The even tool load option is useful when punching stainless steel or tough materials and helps to reduce the wear on the punch by evening the tool load forces. This is achieved by punching every other punch and then reversing the direction to remove the material in between as shown in the following job.



Auto-index

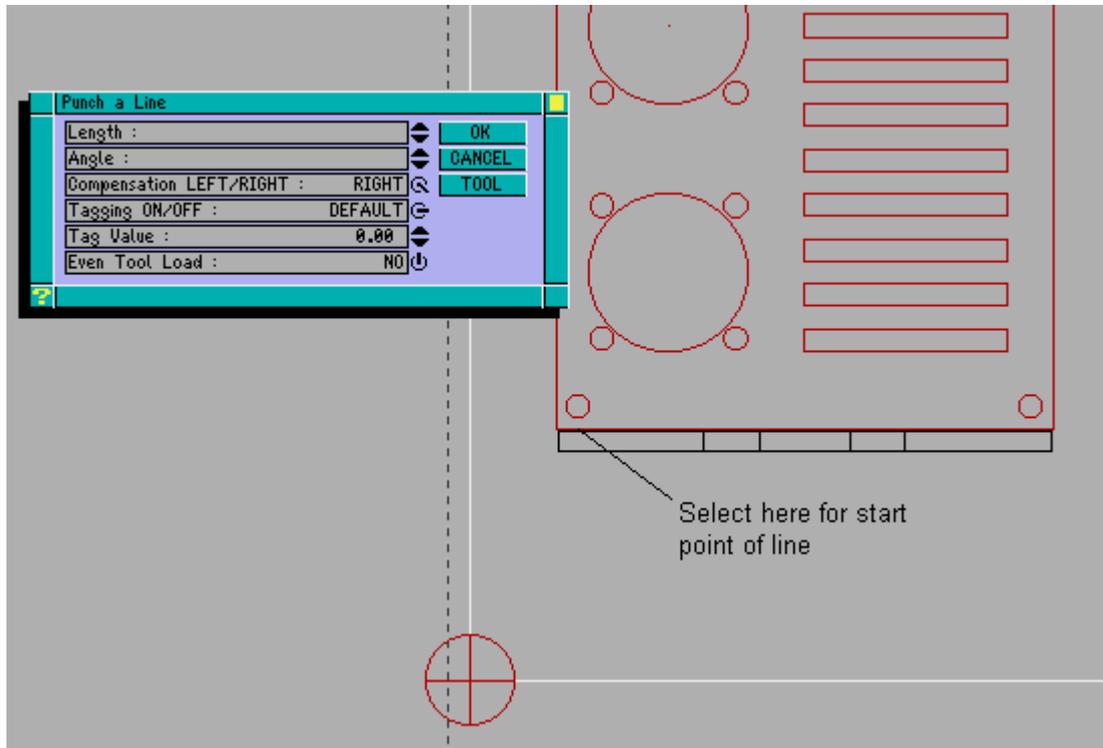
If the turret selected is an auto-index station then the angle of the tool is automatically set to the angle of the line.

WARNING : G-CODER Professional always uses the width of the tool to calculate the step in between punches. So make sure that you enter rectangular tools in the tool library as largest dim then smallest dim. For example width 45 by height 5 and not width 5 by height 45.

G-CODER Professional Reference Manual

Geometry Mode

You only need to fill in the Compensation, Tag boxes and Even Tool Load for Punch Line. When you click on a line (which is the only valid geometry item for this punch operation) G-CODER Professional will automatically pick the nearest end point of the line as the start point and will calculate the length and angle of the line. Note that the direction of punching is determined by which end of the line you select.

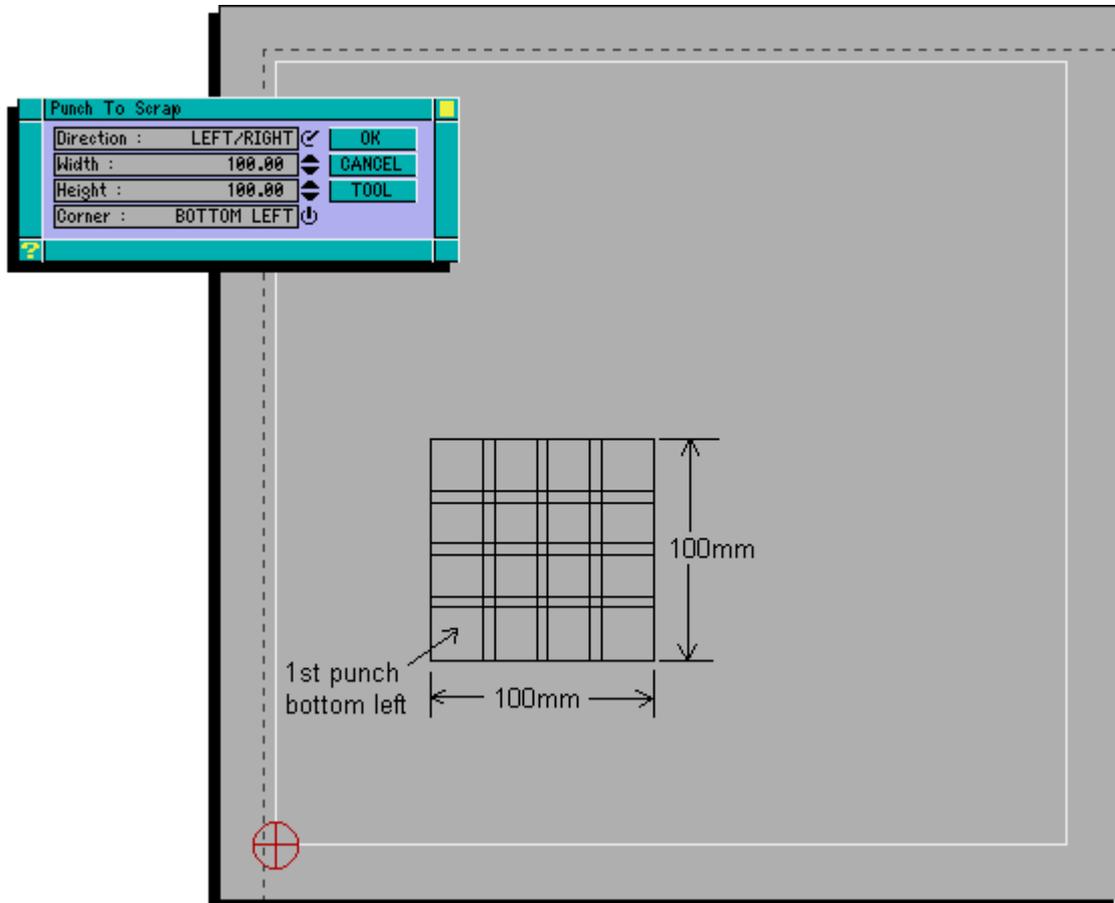


HINT : If you want to punch around the outside of a component, set the Punch parameters once and set the compensation to LEFT. Start in the bottom left of the component and pick the lines in a clockwise manner. This way you will not have to go back into the Punch Operation Window to change the compensation which slows you down.

Cut-out to scrap



Choose this icon whenever you want to punch a cutout completely to scrap. You can use either a square or rectangular shaped punch.

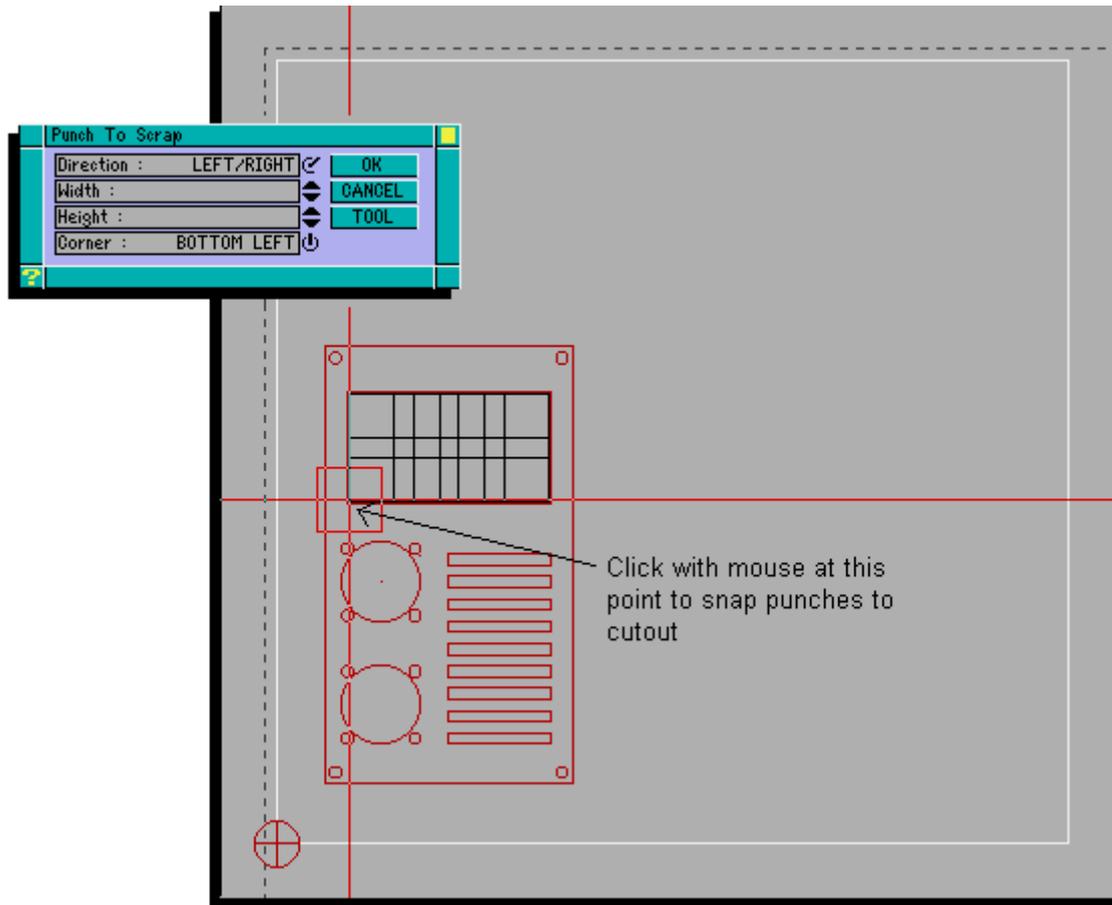


All you have to do is enter the overall size of the cutout in X and Y along with the direction (left to right or up/down) and either of the four corners as a start point. G-CODER Professional will then calculate all the punch centres for you and punch a cutout whenever you click on the sheet or press the space bar. You can then modify any of the parameters including the tool using the query icon.

Geometry Mode

You only need to fill in the Start Corner and Direction for this Punch Operations.

Pick the corner of any two lines (which must join up and be at 90 degrees to one another) and G-CODER Professional will calculate the width and height of the cutout. The start corner will be what you entered in the window regardless of the corner you pick on screen.



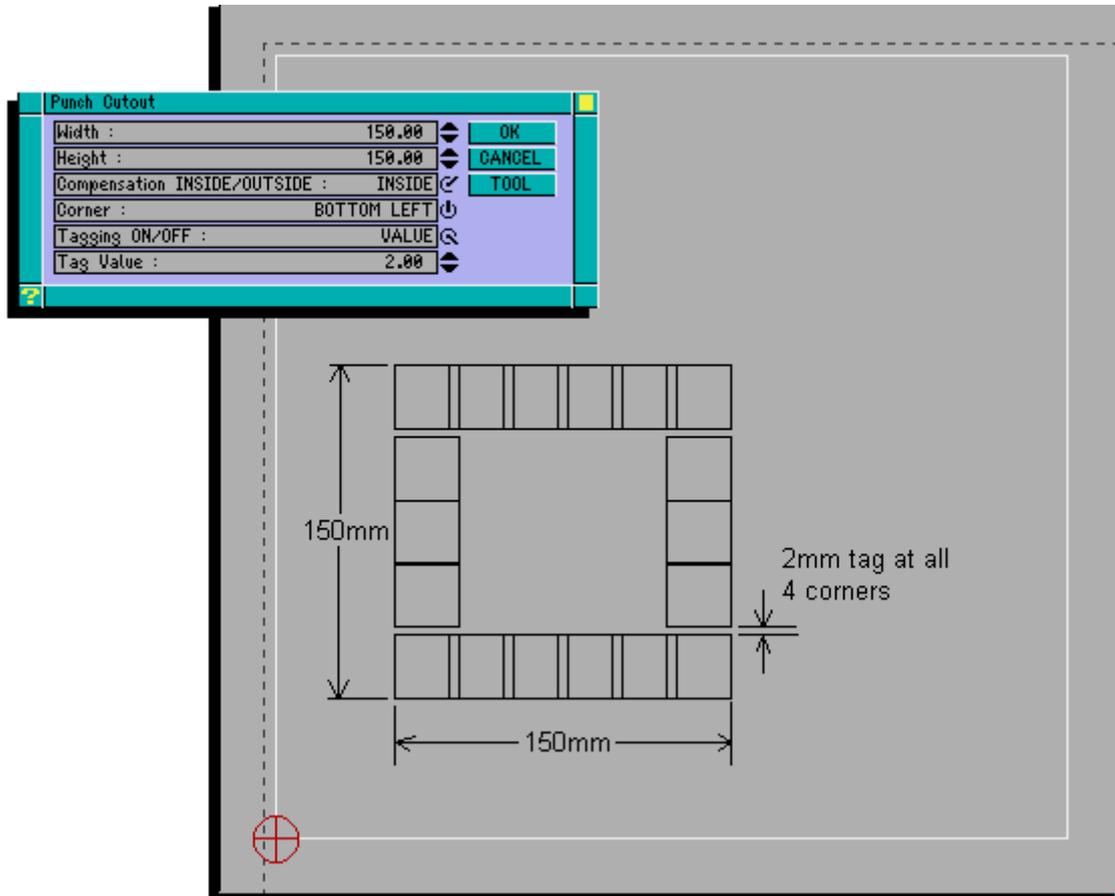
WARNING : If the draughtsman has broken any of the lines in an aperture then G-CODER Professional will only take the lengths of the two nearest lines into account.

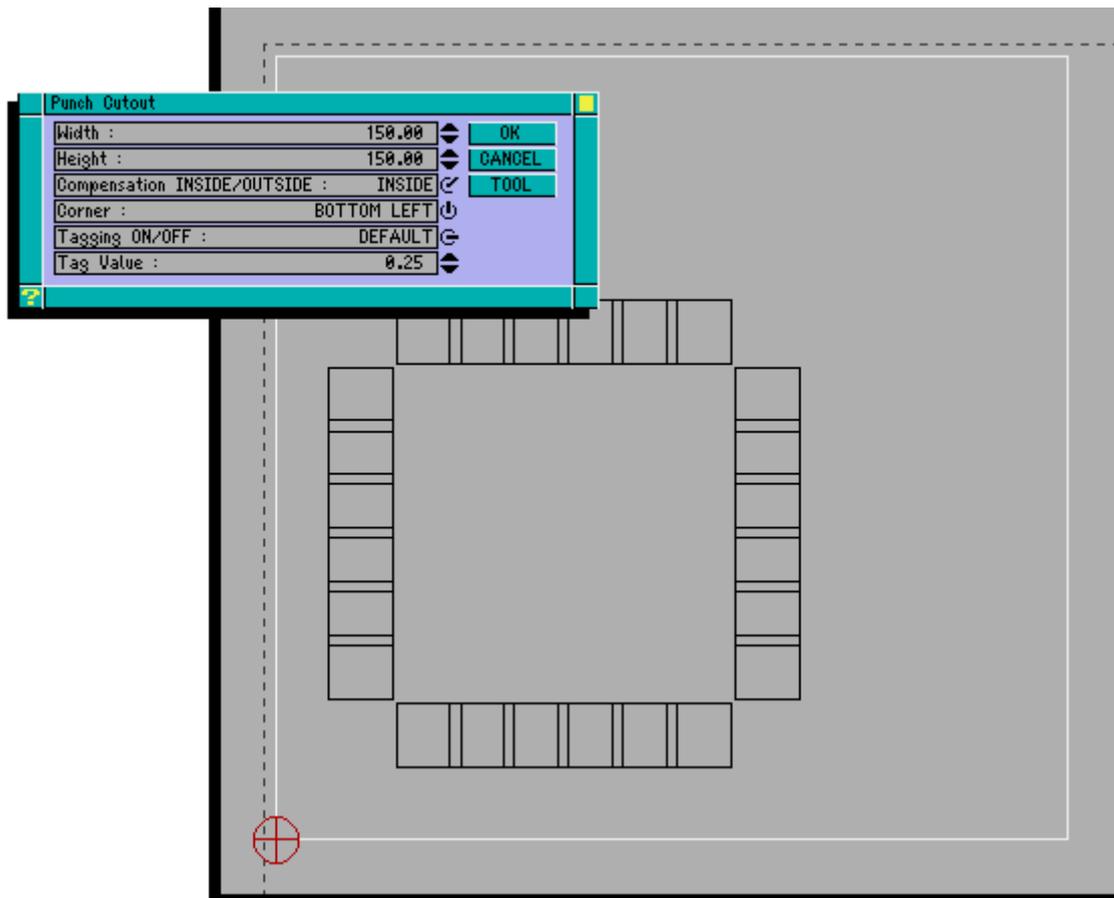
Punch Cut-out (Inside or outside)



Choose this icon whenever you want to punch a cutout leaving either the inside or outside of the cutout held by tags.

Example of inside cutout





Once you have entered the details of the cutout you can place it on the sheet in the normal way. G-CODER Professional will calculate the tool centres for you and leave the corners tagged ready for removal. If you turn tagging off then G-CODER Professional will insert a program stop (M00) into the post processed program for you.

Geometry Mode

You only need to fill in the Start Corner and Direction for this Punch Operations.

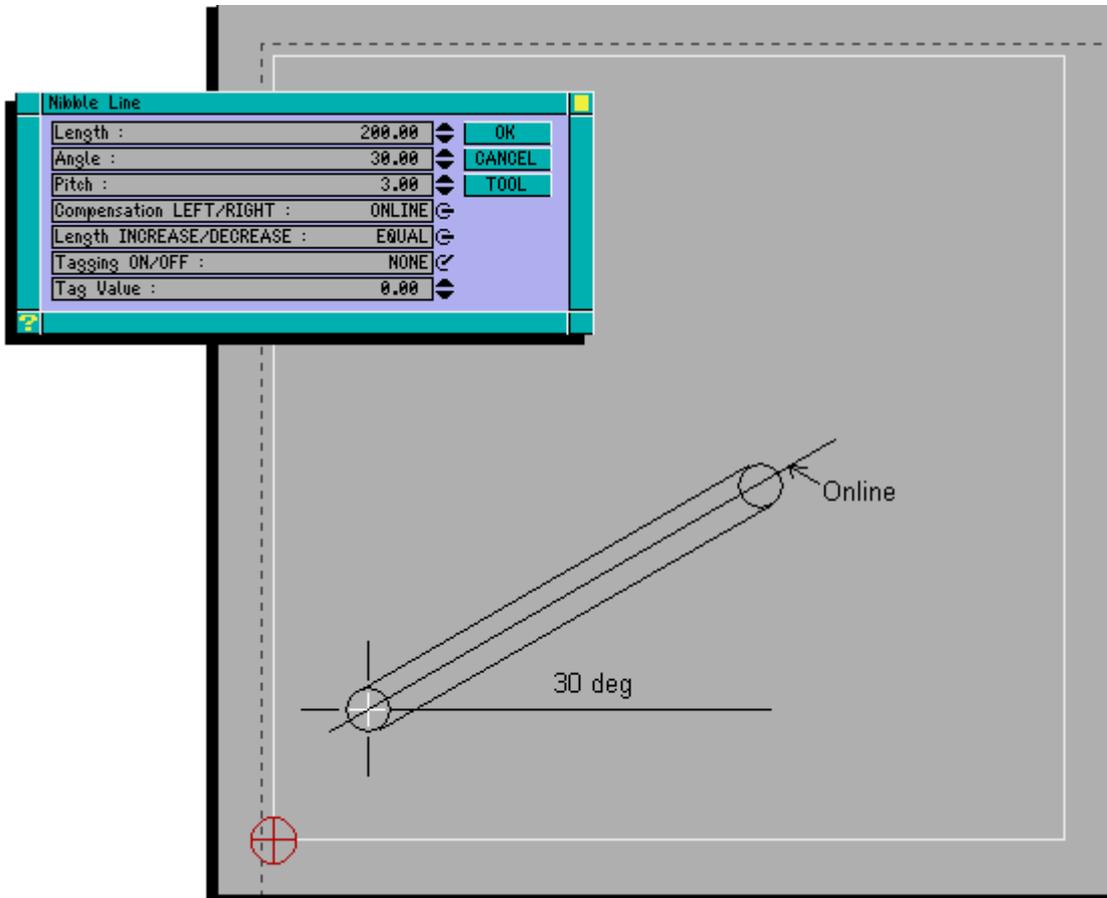
Pick the corner of any two lines (which must join up and be at 90 degrees to one another) and G-CODER Professional will calculate the width and height of the cutout. The start corner will be what you entered in the window regardless of the corner you pick on screen.

WARNING : If the draughtsman has broken any of the lines in an aperture then G-CODER Professional will only take the lengths of the two nearest lines into account.

Nibble Line



Choose this icon whenever you want to nibble a line of given length.



This operation will nibble a line of a fixed length from the start point you give it. Compensation can be applied left/right and to the ends of the line automatically. If you want to nibble point to point you will have to calculate the length and angle for example if you want to nibble from 100,100 to 253,312 then the distance is 261.44 (square root of $((253-100)^2 + (312-100)^2)$) and the angle is 54.18 degrees (arctan of $((312-100)/(253-100))$).

Pitch

This value is for the nibble increment (the distance between individual hits) and should never be set to less than the material thickness. Note : The larger you set this value then rougher the finish will be.

Compensation LEFT/RIGHT : LEFT/ONLINE/RIGHT

This switch allows you to offset the line by the radius either to the left , right or online for no offset. This avoids you having to calculate what the offset position would be.

Length : INCREASE/EQUAL/DECREASE

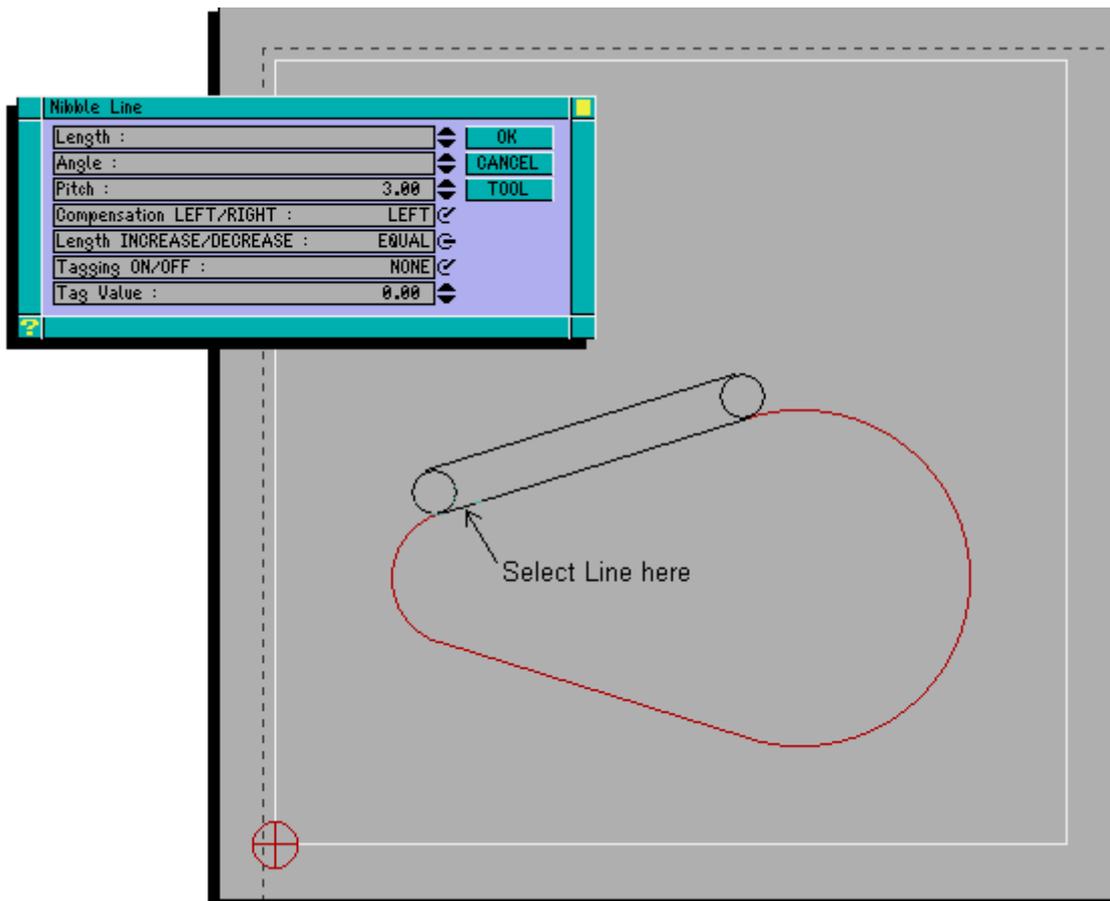
This box allows you to increase or decrease the length of a line by adding or subtracting the tool radius from the ends of the line. The box is normally set to EQUAL and means that the length of the line is from the centre of the first punch to the centre of the last punch.

Tagging ON/OFF : NONE/DEFAULT/VALUE

This box allows you to apply tags to both ends of the line and works in the same way as tags on punch lines.

Geometry Mode

You only need to fill in the Pitch, Compensation, Length INCREASE/DECREASE, Tagging and Tag Value boxes for Nibble Line. When you click on a line (which is the only valid geometry item for these punch operations) G-CODER Professional will automatically pick the nearest end point of the line as the start point and will calculate the length and angle of the line.



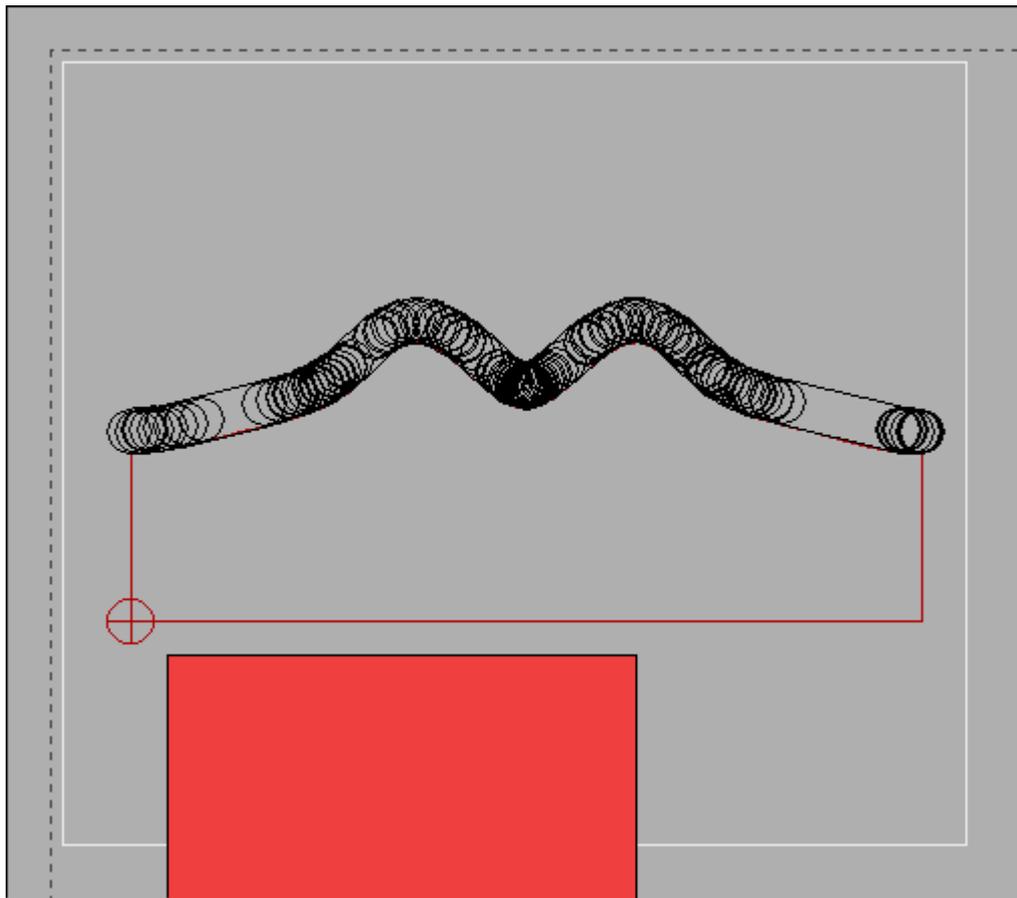
Splines/ Polylines and Ellipses

G-CODER Professional Reference Manual

Read DXF has been modified substantially to allow Gcpro to read splines and polylines, useful for creating parts for ducting work such as lobster backs. This means that you can read smooth curve information from the DXF file and then punch it in Gcpro. Ellipses can also be read as long as they are converted to polylines before being exported to DXF.

For all users of AutoCad and AutoCad LT then all you have to do is make sure that when you export the DXF file choose "R2/R12 DXF file format" when exporting. Users of other systems should find that there is an option to split or explode the Ellipses. AutoCad users no longer need to explode rectangles or polylines before sending to G-Coder Professional as AutoCad outputs these as polylines in R12 DXF files.

Once the DXF file is displayed in Gcpro you can use Move/ Copy/ Delete etc. as you would for any other geometry item and Query will allow you to split Polylines into individual line segments. This is only necessary if you want to split a rectangle into 4 lines. Do not do this for smooth curves. Autopunch will recognise rectangles whether split or not.

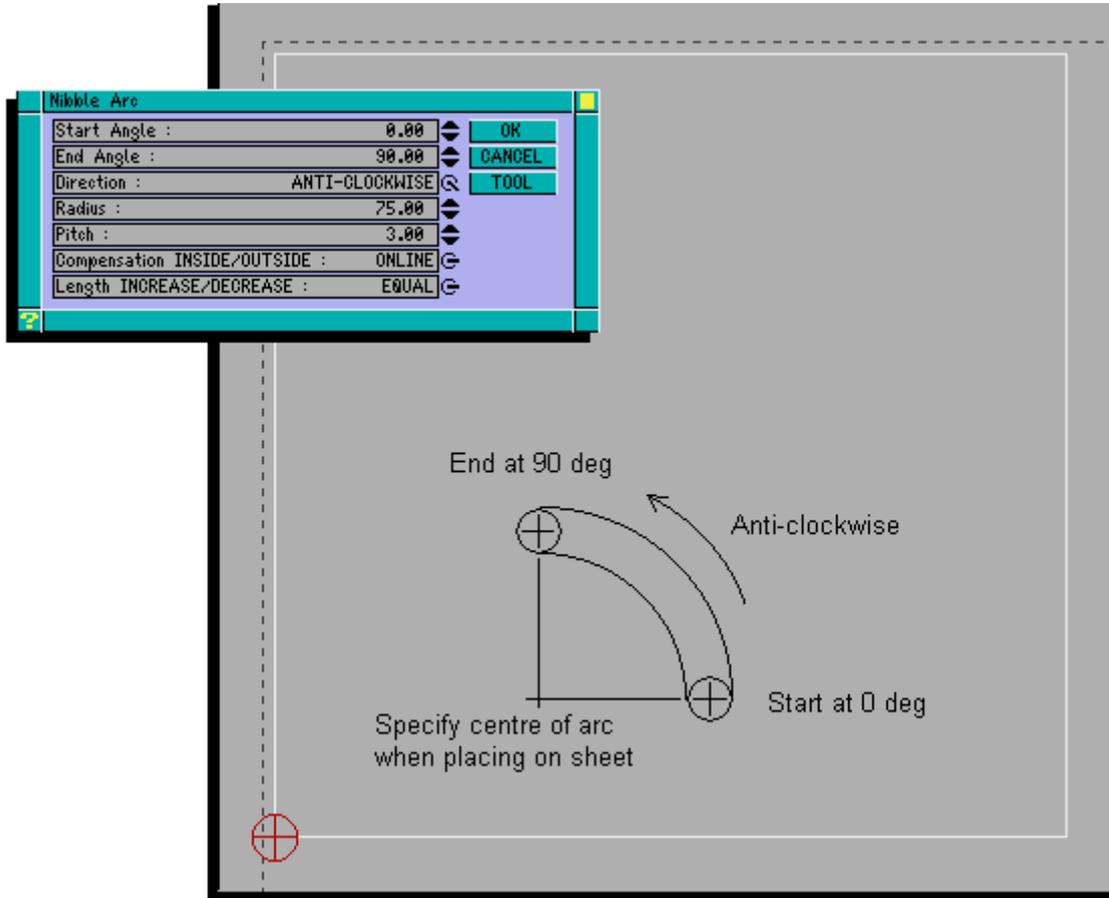


To punch a smooth curve or ellipse, choose nibble line with a round tool, set the length comp to equal and the left/right comp and choose a nibble inc. In geometry mode shift and click the curve and Gcpro will punch the curve. Each line segment output will always be larger than a nibble increment to avoid machine problems and longer nibbles will be output where the curve angle does not deviate more than 0.5 of a degree. This is to save machine memory.

Nibble Arc



Choose this icon whenever you want to nibble an arc of a given radius.



This operation will nibble an arc given the centre point, the radius, the start and end angle of the nibble and the direction. Compensation can be applied inside/outside and to the length of the arc automatically.

Start Angle :

Enter the start angle for the arc, refer to heading Sheet Conventions for a description of angle directions.

End Angle :

Enter the end angle for the arc.

Direction : CLOCKWISE/ANTI-CLOCKWISE

This switch allows you to specify the direction of the arc.

G-CODER Professional Reference Manual

Radius :

Enter the radius of the arc without adding or subtracting punch diameter as this can be applied using the Compensation switch.

Compensation : INSIDE/ONLINE/OUTSIDE

Use this switch to add or subtract the tool diameter to the radius value.

Length INCREASE/DECREASE : INCREASE/EQUAL/DECREASE

This box allows you to increase or decrease the length of an arc by adding or subtracting the tool radius from its ends. The box is normally set to EQUAL .

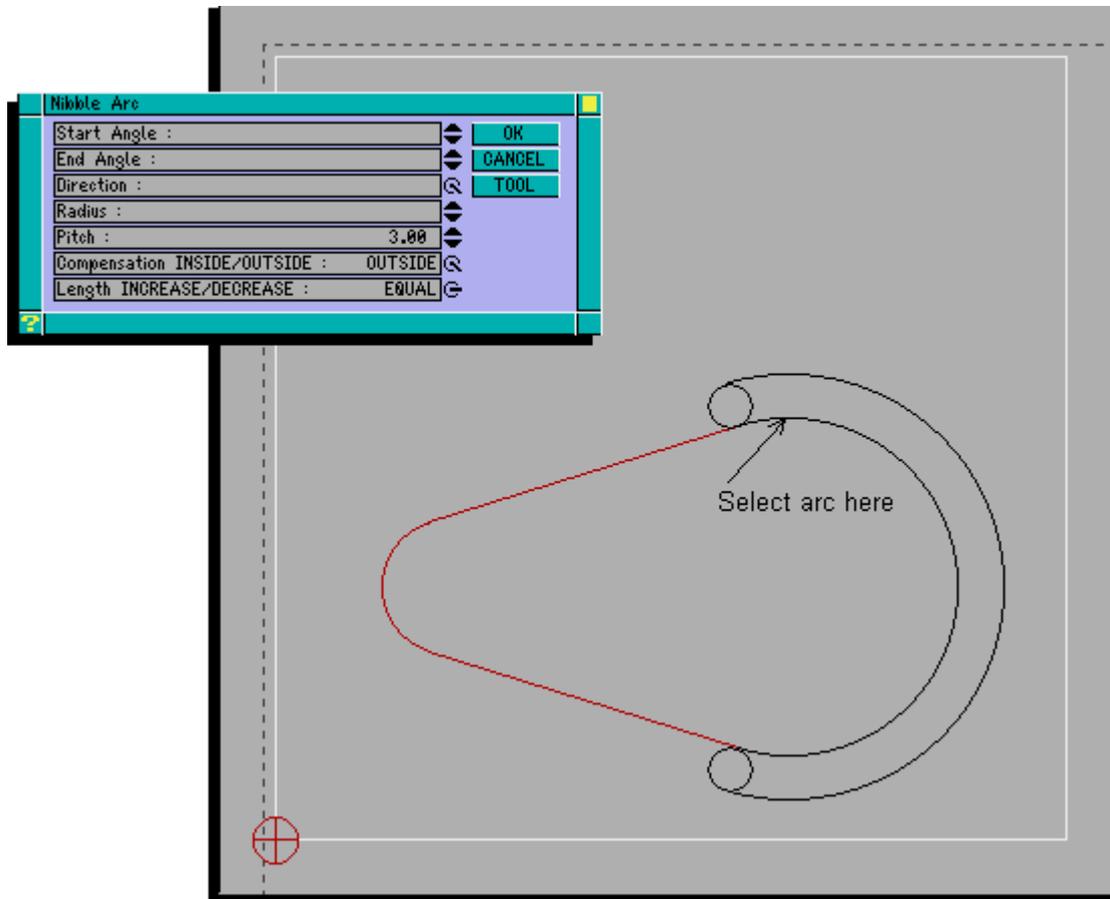
Hints

If you wish to nibble a very shallow arc of large radius then you can still do this by entering the radius in the pop up window. Once you have clicked on the OK button then use the space bar to enter the centre of the arc. In this case G-CODER Professional will not move to the centre of the arc on the machine as it would overtravel.

For example suppose that you wanted to nibble an arc of radius 2000 starting at 80 degrees and continuing on to 100 degrees on a sheet of size 1000 by 500mm. The centre of the arc is 400,-1800. Click on the nibble arc icon and fill in the start angle, end angle, radius, pitch and any compensation then click on OK. Move the cursor onto the sheet and press space bar, enter 400 for X and -1800 for Y then click on OK. You will see a very shallow nibble.

Geometry Mode

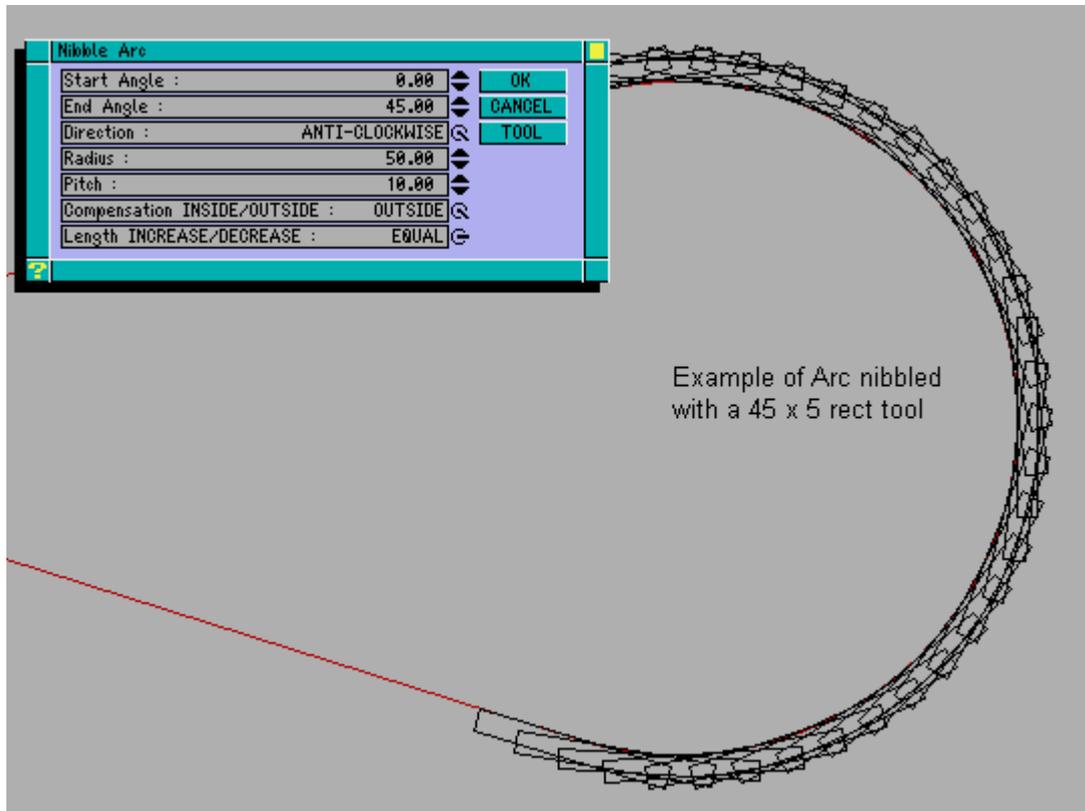
You are only required to fill in the information in this window for Pitch, Compensation (INSIDE / ONLINE / OUTSIDE) and LENGTH compensation (EQUAL / INCREASE / DECREASE). The Start Angle, End Angle, Direction and Radius values are all provided when the Arc is selected.



G-CODER Professional Reference Manual

Nibble arcs with rectangular tool in Auto-index station

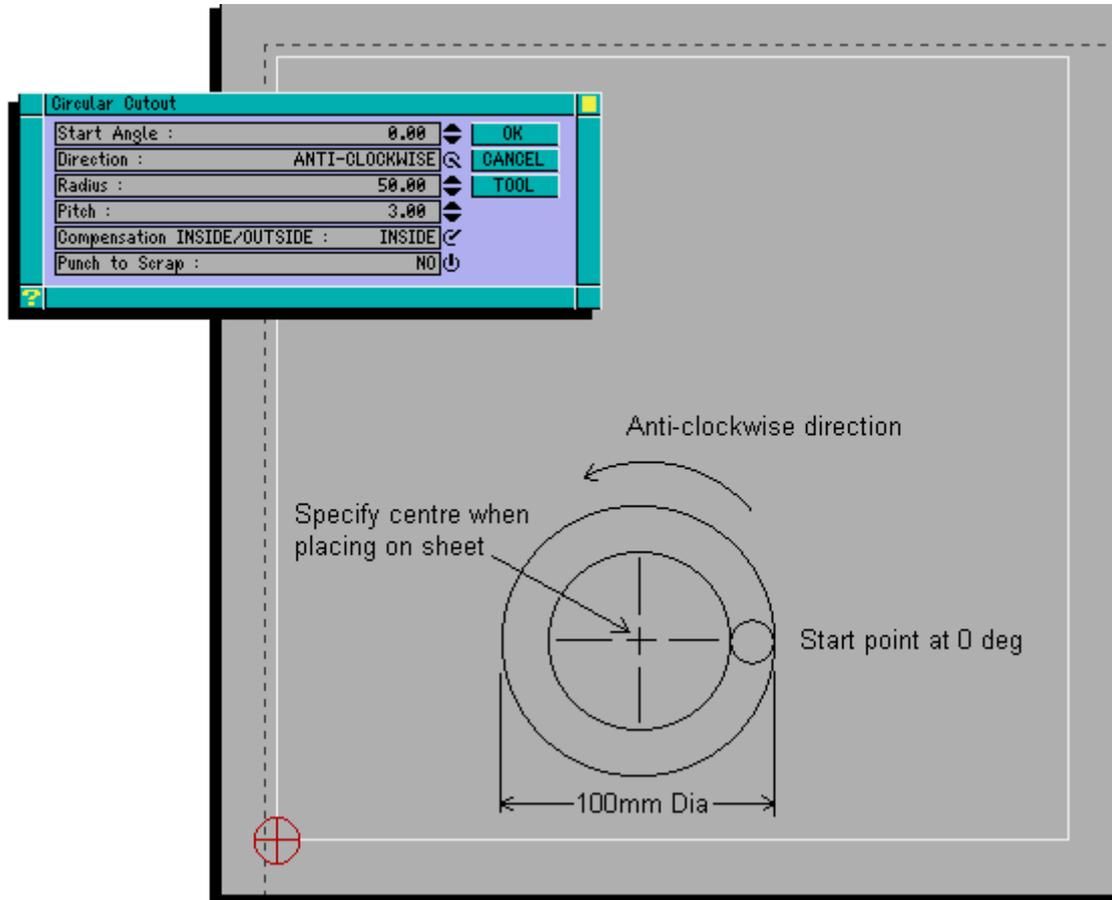
Arcs can be nibbled with rectangular tools to give a better edge finish to the component. Choose a rectangular tool then click on the nibble arc icon and set the inside/outside comp and length comp to decrease. Set the nibble increment to be less than the tool width (ie 40 by 5 tool can use upto 39 mm nibble) and then punch on the sheet. The post processor will automatically output single hits if the radius of the arc is large to avoid accuracy problems on the machine.



Circular Cut-out



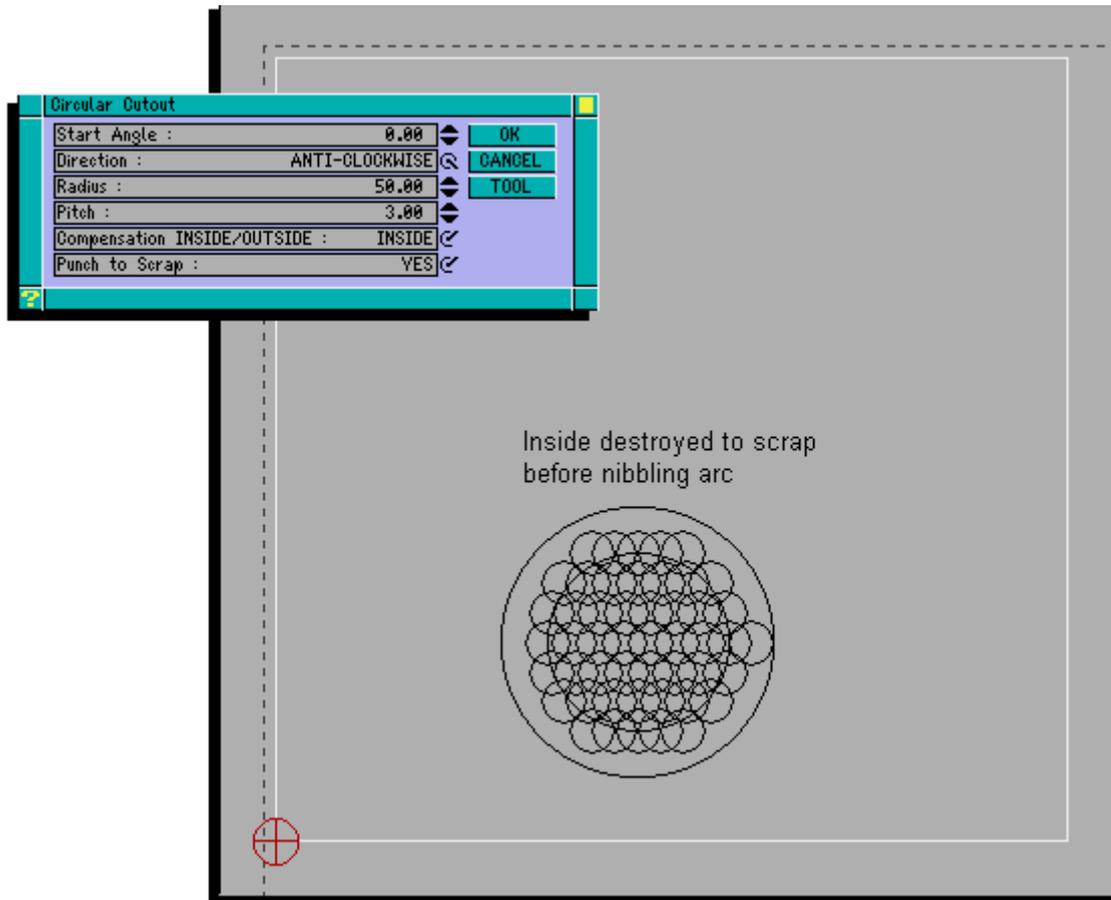
Choose this icon whenever you want to nibble a complete circular cutout.



This operation will nibble a circular cutout of a given radius with inside/outside compensation (as in nibble arc) and then stop the machine (insert a M00 code) so that the scrap can be removed.

G-CODER Professional Reference Manual

By selecting the Punch to Scrap switch the inside of the circle can be automatically punched to scrap as shown below and G-CODER Professional will not output an M00.



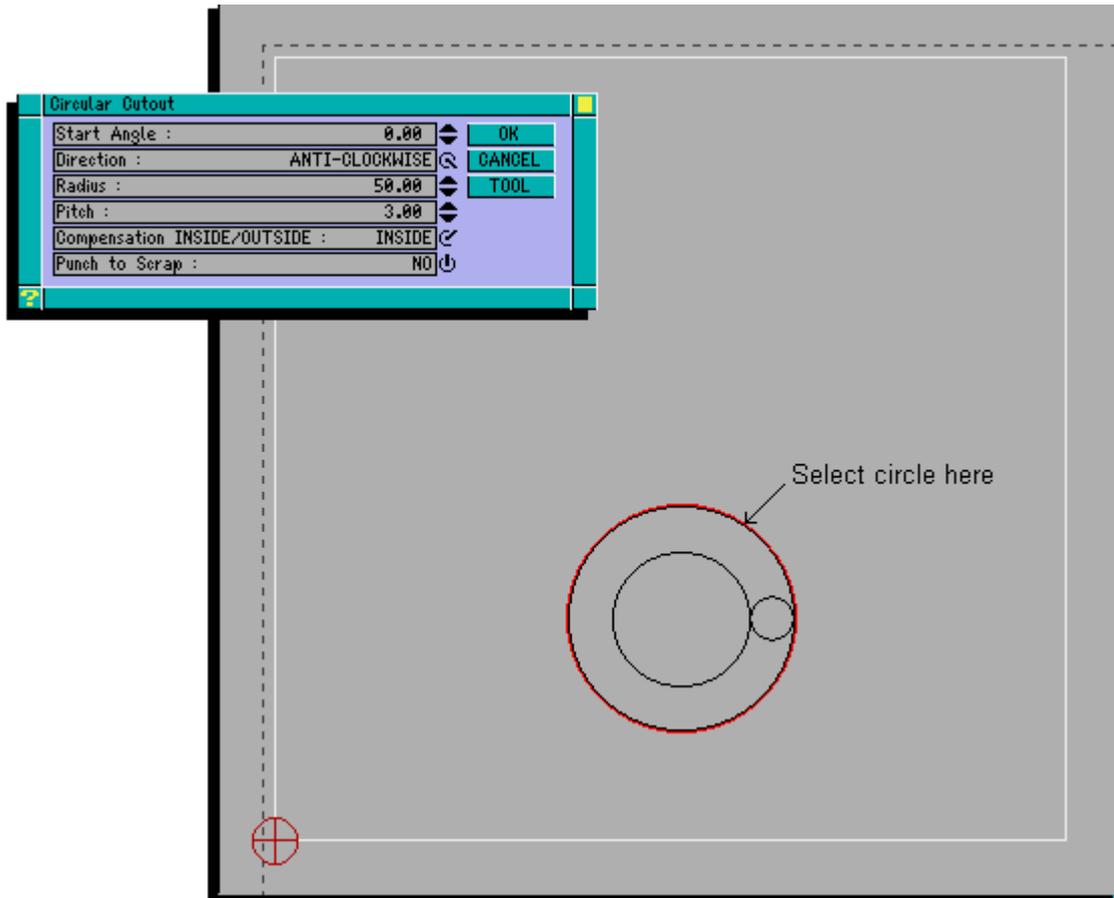
G-CODER Professional Reference Manual

Geometry Mode

You only need to fill in the Direction, Pitch and Compensation boxes for this Punch Operation. When you click on an arc or circle (which are the only valid geometry items for these punch operations), G-CODER Professional will automatically calculate the radius and start angle.

A M00 code will be automatically inserted into the 'G' code program after the circular cutout if you have not set punch to scrap on. This will stop the machine so the operator can safely remove the round blank of material formed by the circular cutout.

Note : You can nibble a circular cutout even if you have only picked an arc. This is because some CAD systems break circles up into two semi-circular arcs when they create a DXF file.



G-CODER Professional Reference Manual

Undo – press U on sheet

G-Coder Professional has a multi-level undo facility. This allows you to undo the last few commands. Simply press the U key with the cursor on the sheet and the last punch or move etc. will be undone.

This only works when there are no other windows open on the sheet which means that it is not possible to press U in the middle of another command such as Move, Copy or Mirror. You would have to cancel or complete the current operation then press U to undo.

Some operations cannot be undone for obvious reasons these include :-

- Save Job
- New Job
- Change Machine
- Automatic Nesting

Post Processor



Choose this icon when you have completed your job (and saved it) and want to produce a program for the machine control.



Once you have selected this icon the post processor window will be displayed. The first thing you will notice is the machine type and the controller type in the top line of the window. The items in the windows can be explained as follows :

Program number

This is the number (or name if your control allows it) that will appear at the top of the program. For example if you type 00001 and your controller is a Fagor then the program number will appear as %00001 at the top of the program. Some controllers do not require program numbers (Westinghouse, AI Flex); in this case the number or name you enter will appear in the comments at the top of the program. If you save your job file as a number (no text characters) then G-CODER Professional will automatically insert this into the program number for you.

Debug Lines

If you set this switch on, the post processor will put G-CODER comment lines (which do not get sent to the machine) in the program describing the punch operation, followed by the actual G-Codes for the same operation. Normally you would not use this switch, but if you have a problem and want to check the G-Codes this can make it much easier to find a given punch operation.

Line Numbers / Block numbers

Set this switch to ON if you want block numbers (N1 etc) to appear in the program. Some controls (Eg Fagor) insist that you have block numbers. The block numbers are always output in 1's as some controls insist on this.

Decimal points

Set this switch to ON if you want the program to use decimal point programming. This option is used mainly for the older Amada controllers which allow non decimal point programming and, generally speaking, is ignored for other controllers as they all require decimal points.

G-CODER Professional Reference Manual

Sort by Tool

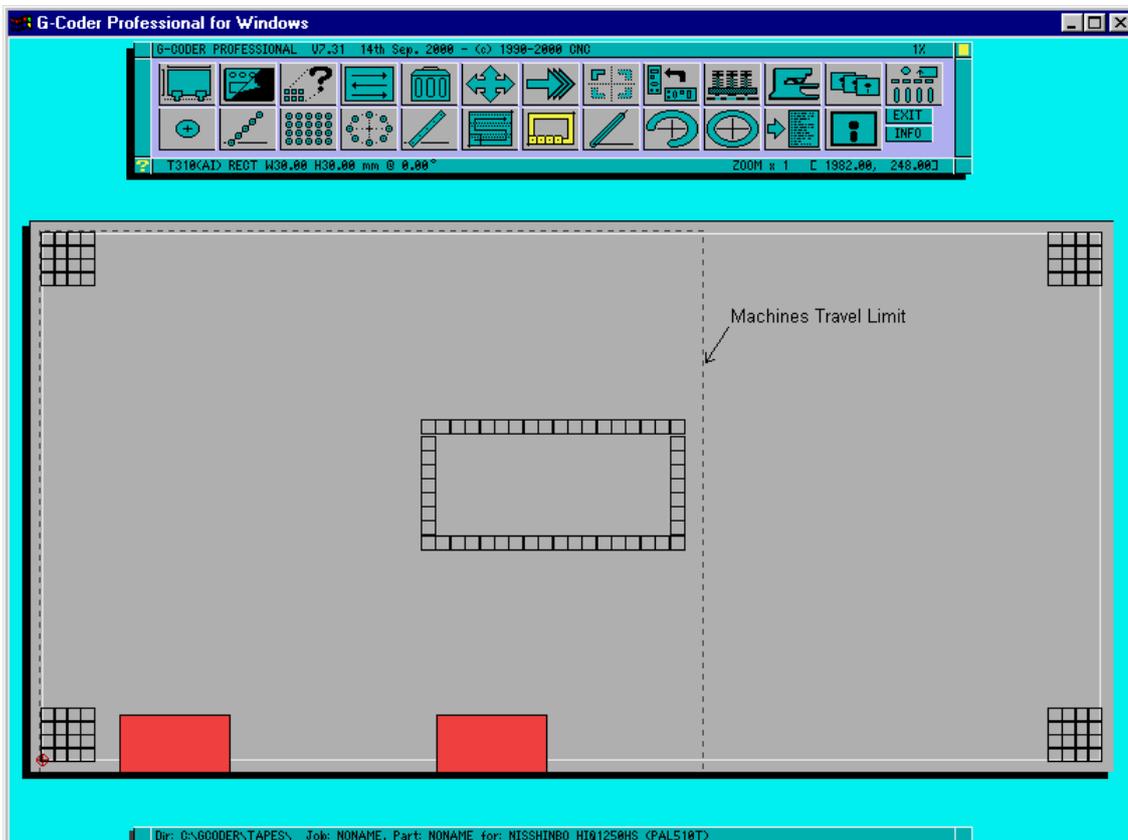
This switch is normally ON and makes sure that the program is created in order of tool priority. If you set this switch to OFF then G-CODER Professional will output the program in the physical order that you punched the job on the sheet and could mean many unnecessary tool changes.

Once you have entered the relevant information click on the OK button and the program will be created. This usually takes only a few seconds and then you can use G-CODER's communications to send the program to the machine control.

The program created by the post processor uses information from a configuration file which can be customised to your requirements for program startup and sheet unload. This should have been done for you already. If you do wish to make any further changes then contact your supplier.

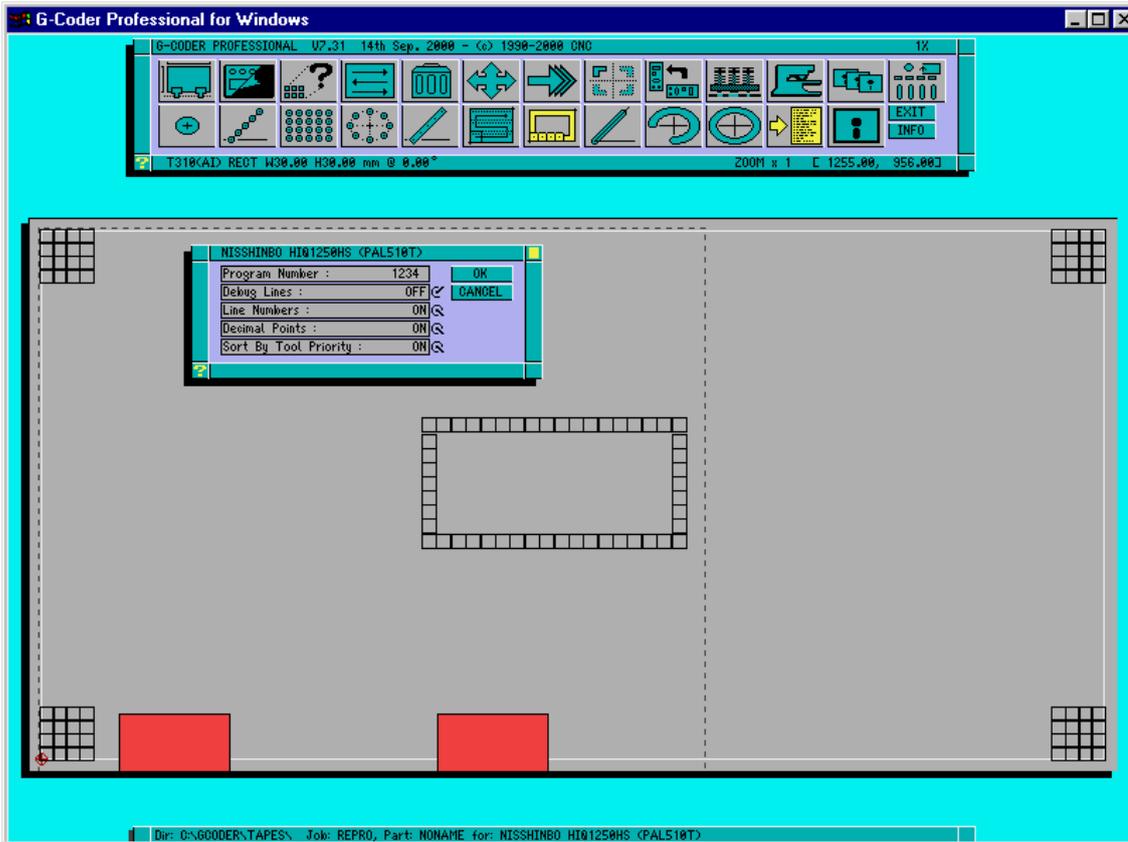
Sheet re-positioning

The following job is of a typical component which would require a reposition as the corner notches on the right hand edge are outside the travel limits of the machine.



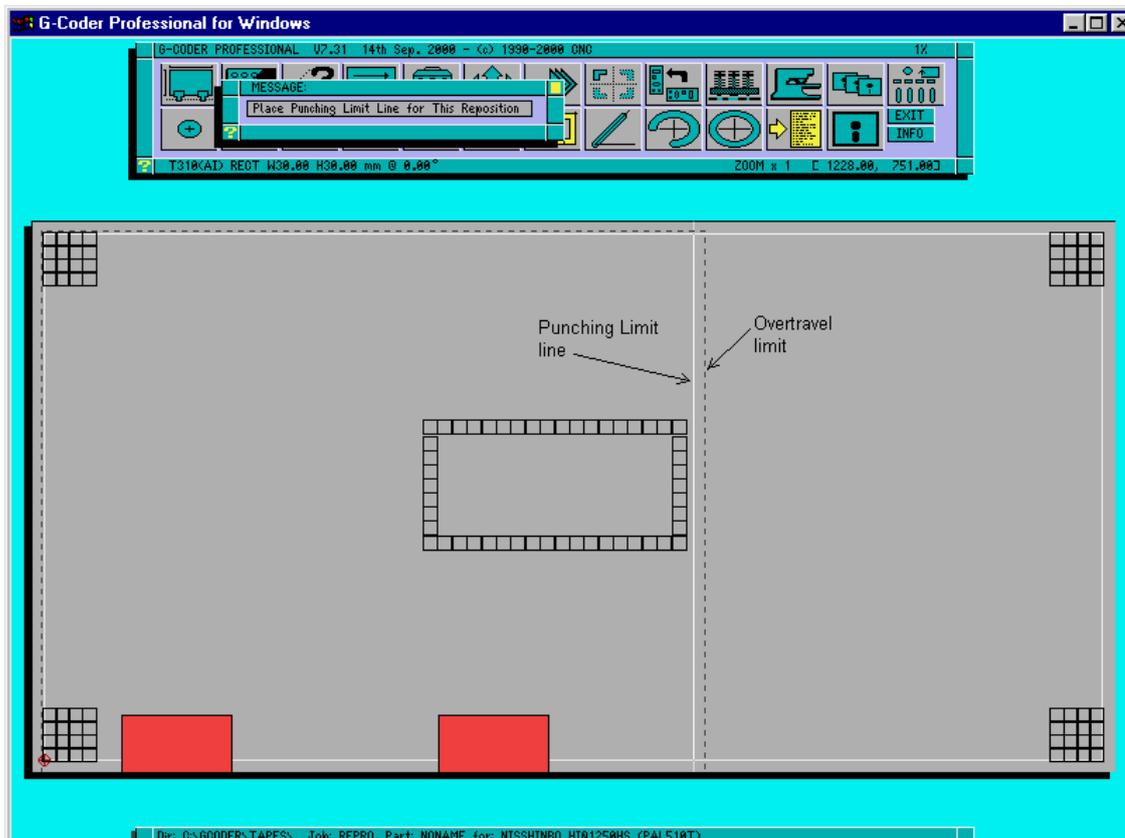
G-CODER Professional Reference Manual

When you select the Post Processor Icon a window will appear into which you are required to enter a program number as shown.



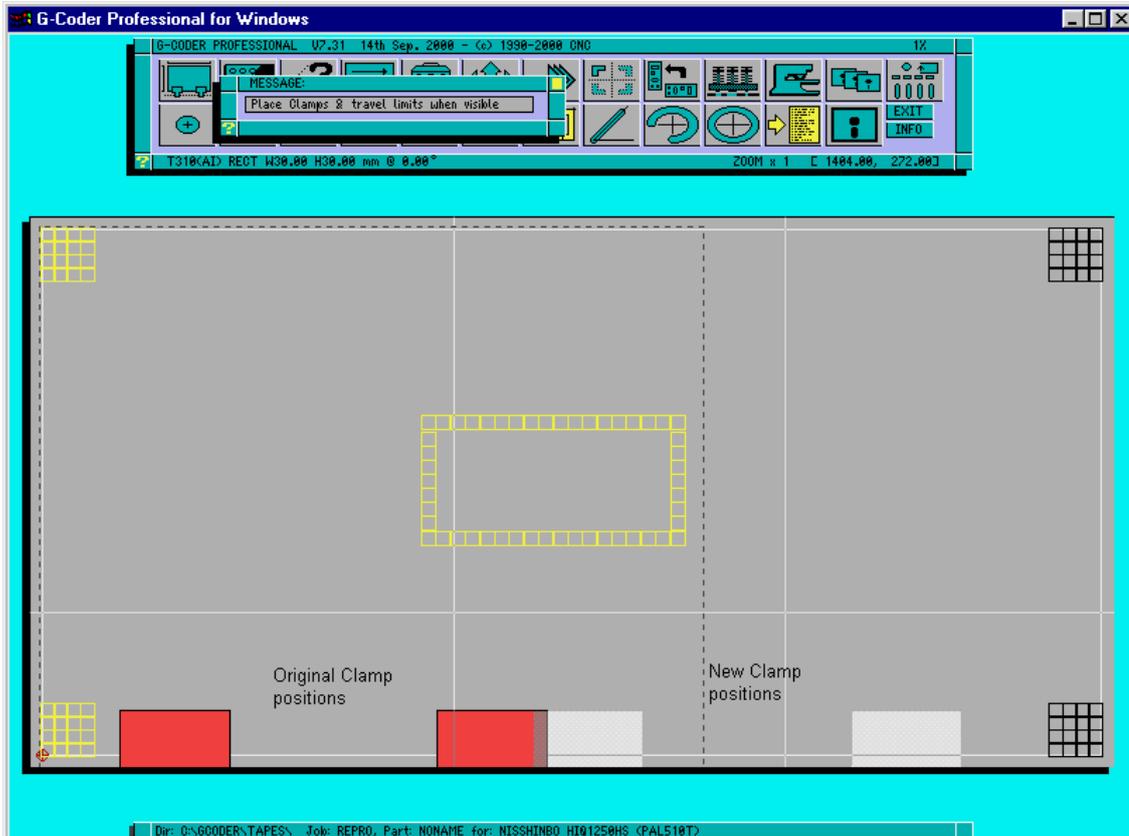
G-CODER Professional Reference Manual

In this example the left hand corner notches and the aperture can be punched as they are within the travel limits (indicated by dashed line). When you click on the OK button a Punching Limit Line will appear on the sheet. Using the mouse you can then position this line on the sheet to indicate how much to punch before performing a reposition.



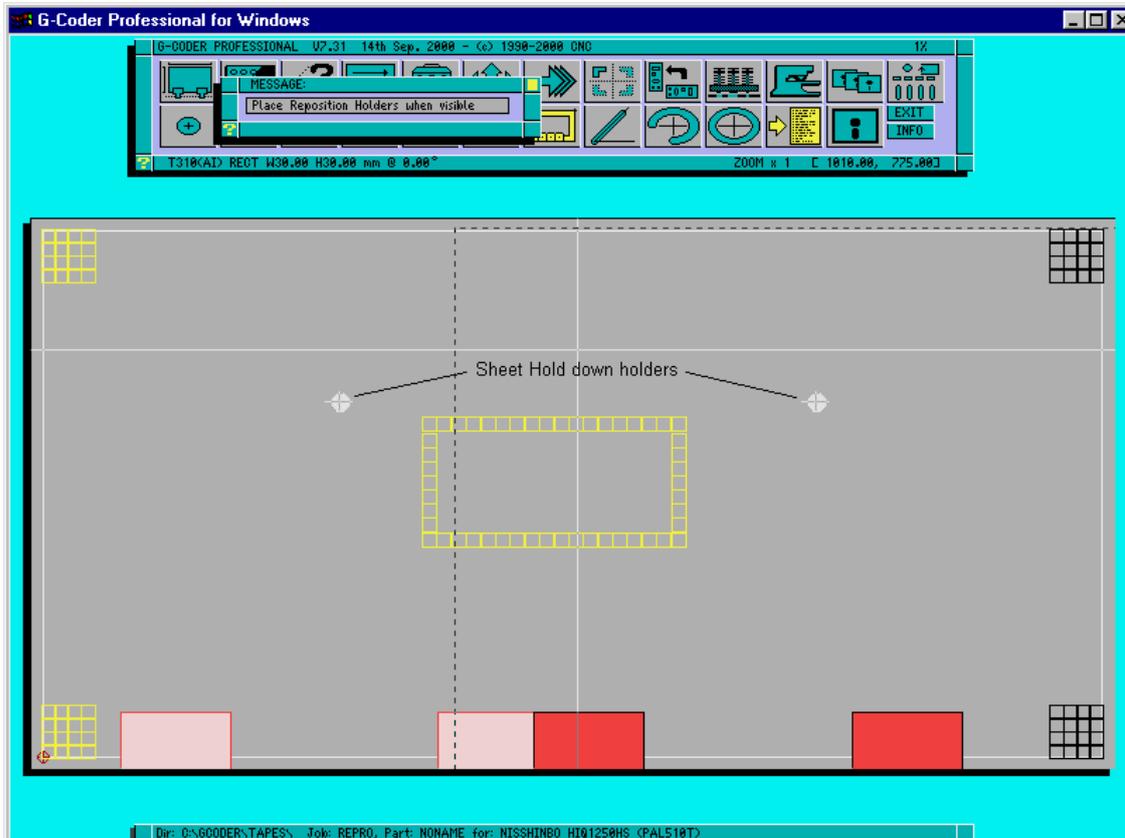
G-CODER Professional Reference Manual

As the punches are converted into 'G' codes they will change colour to yellow. After the group of punches are processed a message is displayed requesting that you move the clamps to the next suitable position. Whilst moving the mouse you will see that a white image of the clamps when they are in a valid position and a black image when they are in an invalid position. When in a suitable position click the mouse button.



G-CODER Professional Reference Manual

G-CODER Professional will now request you to position the work holders in a suitable position. Note that the work holders are displayed in white while in a valid position and black when in an invalid position. When you are satisfied with the work holder position click the mouse button. The rest of the punches will then be marked in yellow and the 'G' code program written to disk.



File icon



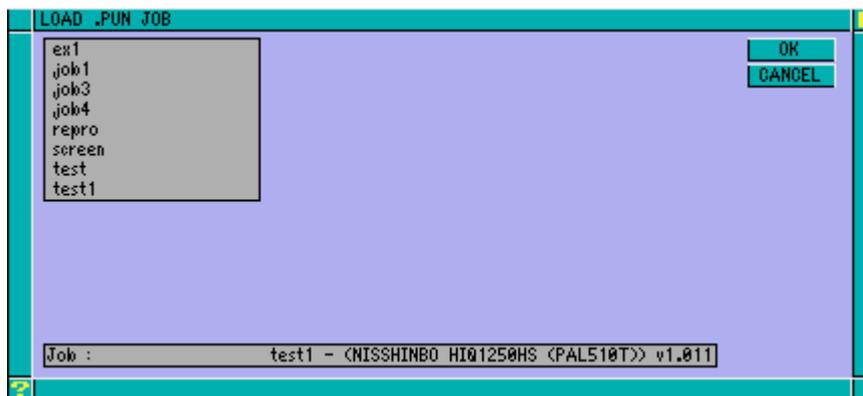
Choose this icon whenever you want to load a job from disk or save the current job or start a new job from scratch. You can also delete jobs from this menu.



Load JOB File

Click on this button if you wish to load an existing job from disk. You may need to use the DIRECTORY option first to select the correct directory from which to load your job.

A load window similar to the example below will be displayed.



The line at the bottom of the window contains the name of the selected job and the machine for which it was originally programmed. To load a job, position cursor over job name and then click mouse button. On clicking on the OK button the select job will be loaded in G-CODER Professional.

NOTE : You may have to click on the scroll bars to scroll through the list of jobs to reach the one you require.

G-CODER Professional Reference Manual

Save JOB File

Use SAVE to save any changes you have made to a job. If the job already exists on disk in the current directory a message to OVERWRITE will be displayed. Click on YES if you wish to save changes to same job name or click on NO to abort operation.

Save as JOB File

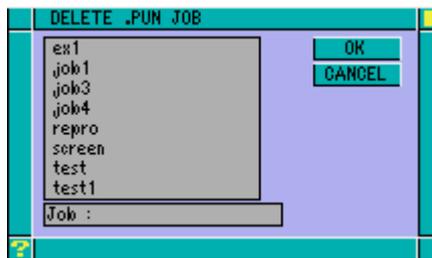
This button is used to save a new job to disk or to save a modified job to a new job name. If you have a similar job to a previously programmed job then load the job and make the necessary edits and then use SAVE AS to save as a new job.

G-Coder Professional has been able to read long file names since the release of 7.05 but could not save long file names. New to version 7.30 and above is the ability to save long file names. If you are still using the DOS version of G-Coder or are using the Westinghouse or CNC7000 communications software you will still have to limit yourself to 8 characters, so be careful. If you have not got the latest Windows G-Coder (V5.02) then call us for an update.

After you have clicked on the SAVE AS button type 8 characters to describe your job and then press ENTER. Click on the OK button to accept job name or CANCEL to abort.

Delete JOB File

Use this option to delete jobs from your disk. Point and click at the job you wish to delete and then click on OK. Once you have deleted the jobs you no longer require, click on CANCEL to exit.



NOTE : Once a job is deleted it is removed from the hard disk. Your only hope of retrieving a deleted job is either from a back-up or by using a utility specifically for unerasing files. Please use extreme caution when deleting jobs.

New JOB

Click this button if you wish to start a new job. If you have a job loaded you will be asked if you are sure. Click on YES to start new job or NO to abort.

NOTE : Make absolutely sure that you have saved any changes to current job as selecting this and clicking on YES could result in you losing your edits.

G-CODER Professional Reference Manual

Load GXF File

GXF stands for the **G-CODER EXCHANGE FORMAT** file format. This is a special format which allows 'G' code programs to be reversed back into G-CODER Professional. This can be useful if you already have a number of existing punch programs that you wish to load into the G-CODER Professional system. To use this option you need to purchase the G-CODER reverse engineering module for your punch press.

Save GXF File

This button will save the current job in the GXF format. This option has been included to allow other people the ability to interface their software to G-CODER Professional. More technical details on the GXF format is available from Cambridge Numerical Control.

Save as GXF File

This button is used to save a new GXF file to disk or to save a modified file to a new filename. After you have clicked on the SAVE AS GXF FILE button type 8 characters to describe your GXF file and then press ENTER. Click on the OK button to accept the name or CANCEL to abort.

Delete GXF File

Use this option to delete GXF files from your disk. Point and click at the job you wish to delete and then click on OK. Once you have deleted the file you no longer require, click on CANCEL to exit.

Directory

Use this option to change the current directory.

Using sub-directories allows you to group related programs together under common headings. This makes locating programs easier and helps in the management of the G-CODER Professional system. Naming of sub-directories is down to the individual user, but we would suggest naming them logically such as BOXES, COVERS etc. To change the current directory click on its name, click on the CHANGE button and then click on OK.

NOTE : If you wish to create new directories then use the DIRECTORY MAKE NEW option in G-CODER.

Read DXF File – read drawing files created by a CAD system

Selecting this option will display a list of DXF files in the current directory (use the DIRECTORY button to change directory first if necessary). Select a file from the list and click on OK. You must know the scale of the drawing as G-CODER Professional will ask you for a scale factor next. For example if the file was drawn 1:1 then enter 1. If the drawing was drawn half size then enter 2 to scale the drawing back up to full size.

The DXF drawing will then be displayed on the screen and you will be able to move the cursor across the sheet and click to move the geometry into place. Notice that the DXF geometry is displayed in Red to distinguish it from the punches which are displayed in Black.

You may append (or add) as many DXF files to the current job as you like using this command.

Write DXF

G-CODER Professional Reference Manual

Once a punch job has been created you can create a DXF file by selecting the WRITE DXF FILE button from the File Window. This file can then be loaded into your CAD system to produce a working drawing or to add dimensions, notes etc before plotting.

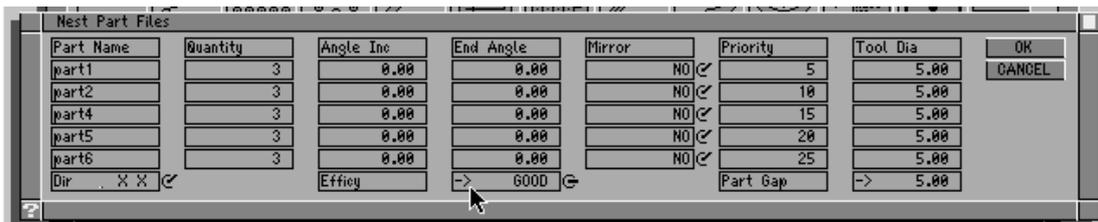
Automatic Dissimilar Parts Nesting

G-CODER Professional includes automatic nesting, which allows you to fill a sheet with a number of dissimilar parts.

If you program a job using DXF geometry and then save it as a part, then the geometry is saved to the part file as well as the punches. This facility is essential as only part files containing DXF geometry can be nested using the dissimilar parts nesting.

To produce a nested sheet:

- Program any parts that you want to nest using DXF geometry and save each as a part file. The DXF geometry for each part must contain a single exterior profile in order for the nesting to work properly.
- Set the sheet size using the Sheet icon.
- Click on the Part icon, then click on the NEST NEW PARTS button.
- The list of part files is displayed - click on each file you wish to nest with the left button (you may use the right button to de-select a file). Click the OK button.
- If any tools need to be placed from the part files into the current turret, then the Place Tools window is displayed (for each part if necessary) and you place the tools in the normal way.
- The following window is displayed:



This window displays each part to be nested and parameters for that part. The parameters can be explained as follows:

Quantity

Enter the number of parts that you want in this box. The quantity can be different for each part.

Angle Inc / End Angle / Mirror

These parameters control the degree of rotation and mirroring allowed for each part. For example, if you set Angle Inc to 90 and End Angle to 180 then the nesting will try to fit the part on the sheet at 0 degrees (its original orientation), 90 degrees and 180 degrees. If you set Angle Inc to 45 and End Angle to 180 then the nesting will try to fit the part at 0 degrees, 45 degrees, 90 degrees, 135 degrees and 180 degrees.

If, in addition, you set Mirror to YES, then the nesting will attempt to fit the part as above but will also try mirroring each combination to get a better fit. The more options you give the nester, the more combinations it will try and the longer the nesting will take.

WARNING: G-CODER Professional will not allow you to rotate a part if it contains tools (other than rounds) which are not in an auto-index station. If this is the case you will not be able to click into the angle boxes for that part but you will still be able to nest the part at its original orientation (0 degrees).

Priority

G-CODER Professional Reference Manual

G-CODER Professional assigns a priority to each part. You may change this priority to make sure that some parts are placed before others. For example if you wanted PART2 to be placed before PART1 then set its priority to be higher (a lower number). For example you could set PART1 to 10 and PART2 to 5 and PART2 would be nested first.

Filler Parts

One of the most useful features of the nester is the ability to fill whatever is left of the sheet (after nesting the other parts in the normal way) with a given part. This part is called a filler part. Set the priority to 0 for any part that you want to be a filler part. You may have more than one filler part simply by setting the priority of each filler part to 0.

Tool Dia

This value should be set to maximum width of the tool(s) used to punch the external profile of the part. If you use a 20 square tool to punch the corners of a part then set this value to 20 for that part to avoid it overlapping other parts.

Part Gap

This value is used to determine how much material to leave between the punched parts on the sheet.

Dir.

This determines the direction of the nest.

XX is the default and will nest the parts starting in the bottom left corner of the sheet and progressing along the width of the sheet (X axis). This will most often result in the best nest.

YY will start bottom left and progress up the sheet.

XY will weave the nest from bottom left corner to top right corner.

G-CODER Professional Reference Manual

Efficy

This determines the Efficiency of the nest by setting a tolerance which is used when trying to fit parts together. Leave this set to FAIR or GOOD while you determine the best combinations of Angles etc., then use BEST to get the most parts out of a sheet.

Once you have filled in these details, click OK and the nesting will display a message window giving the nesting progress. When the nest has finished, press Esc to return to G-CODER Professional and the nest will be displayed.

Refining the nest

The best approach when nesting is to sort out the priority of parts first and then, without changing any angles, just try the nest by clicking OK. You can then determine the best combination of angles for the parts (a rectangular part would normally only be rotated by 90 degrees and not mirrored for example) and go back to the Part Icon and click the REPEAT LAST NEST button. This opens up the same window as last time with the same parts and priorities etc. You may then change one or more of the parameters and click OK to try the nest again. This way you can refine the nest until you get what you want.

Saving and post processing the final nest

Once you are happy with the nest, you can save it as a JOB file and post process it. There is nothing special about these JOB files - they are exactly the same as a normal job. This means you can still delete, move or copy parts on the nested sheet and you can even append a part to the sheet manually.