

HEIDI^(C)

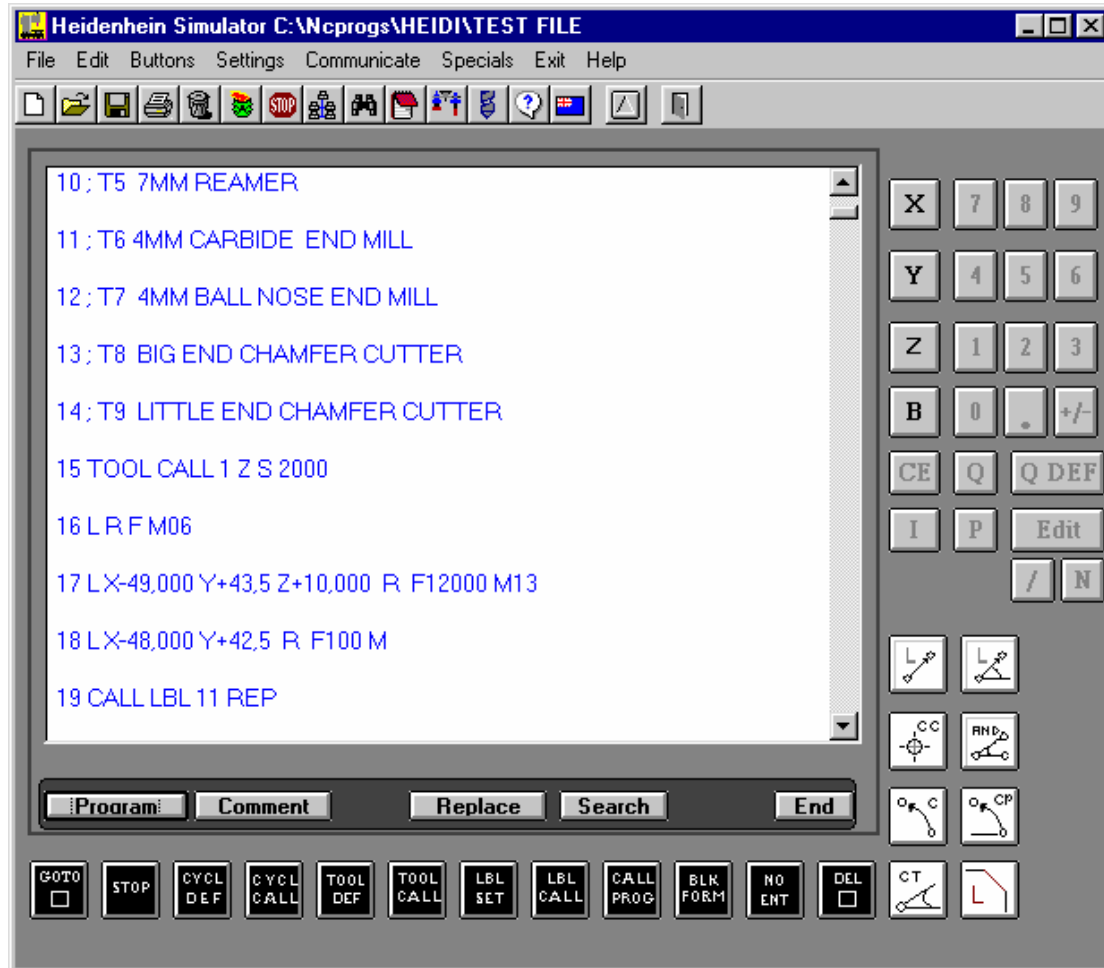
**HEIDENHAIN LANGUAGE
EDITOR for Windows**

USERS MANUAL

Copyright (C) 1990-2006

Congratulations! We are excited about your decision to take a closer look at our Heidenhain Editing software HEIDI, with in built compatibility to suit the dialogue programming found in Heidenhain CNC Controls.

Heidenhain is a registered trade name from Germany.



We have spent many long hours to try and ensure that HEIDI will run trouble free. Should you encounter any problems with HEIDI please send a description of your problem to support@shera.com.au. The directors and staff will not be held responsible for any omissions or errors in the HEIDI software however we will do our best in trying to resolve any issues you may discover.

SHERA BONNET & ASSOCIATES Pty Ltd.
4 Baird Court,
West Pennant Hills
New South Wales
AUSTRALIA 2125

Ph ++61-2-9872-1255 Fax ++61-2-9872-2782

Email support@shera.com.au

Web www.shera.com.au

INDEX

A

ASCII	
Characters	28
AUTO TOOL CHANGE	
Cycle Definition	18

B

B Hot Key	10
BLOCK FORM	10

C

C Hot Key	7
CABLE	
Diagram	30
CALL PROG	
Sub Program calling	21
CALL PROGRAM	
Program Number	21
CARRIAGE RETURNS	26, 28
CHAMFER	14
CIRCLE	7
Centre	12
Polar	13
Tangent	13
CIRCULAR POCKET MILLING	
Cycle	16
COLOUR	
Text and Background colours	33
COMMENT	7
Entering comments into your program	21
COMMUNICATIONS	
Advanced	25
Baud Rate	24
Character delay between	25
Data Bits	24
Handshake	24
Ignore Character	26
Ignoring Characters	
When receiving	27
New Style	24
Old Style	30
Parity	24
Port	24
Stop Bits	24
Time out	27
CYCLE CALL	19
CYCLE DEF	15, 19
CYCLES	
Auto Tool Change	18
Circular Pocket Milling	16
Datum Shift	17
Dwell	17
Mirro Image	17
Peck Drilling	15
Pocket Milling	16, 18
Program Call	18
Rigid Tapping	18
Rotation	17

INDEX

Scaling	18
Slot Milling	16
Tapping	15
D	
D Hot Key	7
DATUM SHIFT	
Cycle Definition	17
DEFAULTS	
Hedi.INI file	35
DIRECTORY	
Default Folder	32
DRILLING	
Cycle	15
DWELL	
Cycle definition	17
E	
EIA	27
ELECTRICAL	
Interference	31
EQUATIONS	23
ESC	
Button	7
F	
F11	19
F4	
Incremental	10
F5	15
F6	19
FILE	
New	6, 10
Open	6
Save As	6
FILENAMES	
Default extensions	27
FOLDER	
Default Directory	32
FONT	
Size	34
FORCING	
Upper case Characaters	27
FOURTH AXIS	
Setting axis name	32
FUNCTION KEYS	7
G	
G Hot Key	7, 14
GOTO	7, 14
H	
HEIDI.INI	
Defaults	35
Hot Keys	7
I	
ICONS	9
ISO	27

INDEX

L

L Hot Key	7, 11
LABEL CALL	
Label Number	20
Number of Repeats	20
LABEL NUMBER	
Label Call dialogue.....	20
Label Set dialogue	20
LABEL SET	
Label Number	20
LABELS REPEATING	
Label Call dialogue.....	20
LINE FEEDS	26, 28
LINEAR	7, 11

M

M CODES	
Cycle Call	19
M00	
Stop Command	14
MIRROR IMAGE	
Cycle Definition	17

N

NEW	
Program	6
NO ENTER	21

P

P Hot Key	7
PAPER TAPE	
Leaders & Trailers	25
PATH.....	29
POCKET MILLING	
Cycle.....	16, 18
PRINT	
Number of lines to print.....	34
PROGRAM	7, 10
ID Character	27
PROGRAM CALL.....	18
PROGRAM NUMBER	
Call Program Dialogue	21

Q

Q DEFINITIONS	23
Q Hot Key.....	7
QUICK START.....	6

R

R Hot Key.....	7, 12
RECEIVE	30
REPLACE	
Search and Replace.....	21
RIGID TAPPING	
Cycle Definition	18
ROTATION	
Cycle Definition	17
ROUNDING.....	7

INDEX

S

S Hot Key7, 23

SCALING

 Cycle Definition' 18

SEARCH

 Search and Replace.....21

 Searching for particular cycles22

SEND.....30

 Program Repeatedly27

SETTING32

 Font size.....34

 Fourth Axis.....32

Program Name

 Length of name32

 Text.....33

SLOT MILLING

 Cycle.....16

STOP

 M00.....7, 14, 23

T

TAPPING

 Cycle.....15

TOOL CALL

 Spindle Speed20

 Tool Number.....20

 Working Axis20

TOOL DEF19

 Tool Length19

 Tool Number.....19

W

WAIT

 For machine to send start signal26

WORKING AXIS

 Tool Call20

X

X Hot Key.....7, 10

Y


Y Hot Key.....7, 10

Z

Z Hot Key7, 10


QUICK START

HEIDI makes it very easy to get started on the Editing and creation of Heidenhain Programs provided that you already have a basic knowledge of the Windows operating system.

- To Start a **NEW PROGRAM** you can either click on the  button at the lower left hand corner of the Program screen

Or


Select **FILE NEW** from the pull down text menu at the top left hand corner of the HEIDI screen.

Or click the  icon on the HEIDI toolbar

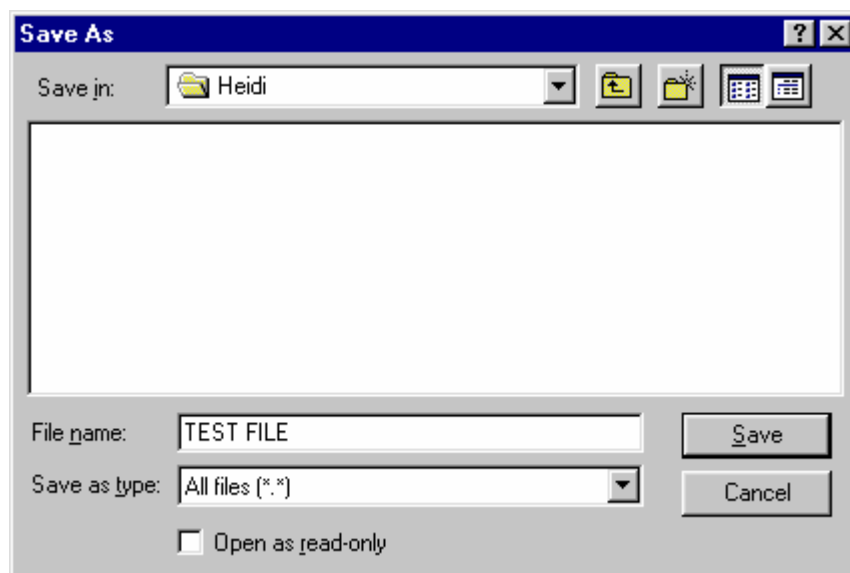
- To **EDIT** a program which you know already exists on your hard disk or a network drive.

Select **FILE OPEN** from the pull down text menu at the top left hand corner of the HEIDI screen.

You may also use the keyboard and type **Ctrl'O'** to open an existing file

Or click the  icon on the HEIDI toolbar

- To **Save** your first file to disk after creating it in HEIDI select **FILE SAVE AS** from the pull down text menu at the top left hand corner of the HEIDI screen. You will be presented with a window where you can type in the Filename of your First program.



Click **Save** in this window to save your file.

HOT KEYS

SPEED is important to all programmers and accordingly we have added a number of single keystrokes to HEIDI so you do not need to click with the mouse every time you wish to enter or modify program data.

- C** The 'C' key will activate the dialog for the CIRCLE command.
- D** The 'D' key will activate the dialog to enter a program COMMENT
- G** The 'G' key will activate the GOTO function.
- L** The 'L' key will activate the dialog box for LINEAR programming.
- P** The 'P' key will activate the dialog for Program Number input.
- Q** The 'Q' key will activate the dialog for Q Definitions
- R** The 'R' key will activate the dialog for ROUND CORNERS.
- S** The 'S' key will activate the dialog for the STOP command.
- X** The 'X' key will activate the dialog for Axis motion either LINEAR or Non Linear.
- Y** The 'Y' key will activate the dialog for Axis motion either LINEAR or Non Linear.
- Z** The 'Z' key will activate the dialog for Axis motion either LINEAR or Non Linear.

- ESC** The Escape button will cancel the input of your current dialogue.
- F2** Will open the Edit dialogue when a block of your program is Highlighted

F2 will also act as the OK button when you are finished editing a particular block of your program
- F4** Toggles Incremental on and off in dialogues
- F5** Cycle Definition
- F6** Cycle Call
- F7** Label Set
- F8** Label Call
- F11** Tool Definition
- F12** Tool Call
- Ctrl F12** Tool Change

HOT KEYS

Ctrl A	Save the current Heidenhain program with the option of providing a new name.
Ctrl B	Move to the Top of your program
Ctrl E	Move to the End of your program
Ctrl F	Find a particular part of the program. Also Search and replace
Ctrl N	Starts a new Heidenhain Program
Ctrl O	Opens an existing Heidenhain Program
Ctrl M	Merge another Heidenhain program into the program which is currently open in Heidi
Ctrl P	Print your Heidenhain Program
Ctrl R	Renumber the program blocks
Ctrl S	Save the current Heidenhain Program
Ctrl T	Display temporary data
Ctrl U	UnDelete the last block which was deleted

ICONS



- | | | |
|-----------------------------|-------------------------|-------------------------|
| File New <i>Ctrl N</i> | File Open <i>Ctrl O</i> | File Save <i>Ctrl S</i> |
| Print Program <i>Ctrl P</i> | Delete Block <i>Del</i> | Undelete <i>Ctrl U</i> |
| Insert STOP M00 | Send / Receive | Search <i>Ctrl F</i> |
| Windows Notepad | Insert Toolchange | Special Drilling |
| Help | About Heidi | Screen Large/small |
| Exit Heidi | | |



- | | | |
|----------------|--|----------------|
| Linear | | Linear Polar |
| Circle Centre | | Rounding |
| Circle | | Circular Polar |
| Circle Tangent | | Chamfer |

DIALOGUES



Click the File New icon or Program button to display the Program Number Dialogue or press Ctrl N on your keyboard



PROGRAM BEGIN / END
PROGRAM NUMBER
1234
Units
 MM INCH
Cancel OK



Click this button to display the **BLOCK FORM** Dialogue

BLOCK FORM
Minimum Values Maximum Values AXIS
X X X
Y Y Y
Z Z Z
OK Cancel



Click any of these keys to open the Single axis dialogue. You can also press the keyboard characters X, Y, Z, B

F4
INCREMENTAL
Toggle On/Off

LINEAR
X 100
Y
Z
B
Radius
 R R0
 R+ R-
 Linear Move
Feed F MAX F-None
M OK Cancel

DIALOGUES



Click this Icon to open the **LINEAR** dialogue or press the **L** key on your keyboard

LINEAR

X 100

Y 200

Z 300

B 400

Feed 1234

M 03

Radius

R R0

RR RL

Linear Move

F-MAX F-None

OK Cancel



This button opens the **LINEAR POLAR** Dialogue

LINEAR POLAR

Polar Radius

Polar Angle

Feedrate

F-Max

F-None

OK Cancel

DIALOGUES



This Icon opens the **CIRCLE CENTRE** Dialogue

CIRCLE CENTRE

Centre in X

Centre in Y

Centre in Z

OK Cancel



Corner rounding can be entered using this icon or press the **R** key on your keyboard

ROUNDING

Radius

Feed

OK Cancel



This icon will open the **CIRCLE** Dialogue or press the **C** key on your keyboard

CIRCULAR

X

Y

Z

B

Feed F-Max F-None

M

Radius

R

R0

RR

RL

CCW CW

DR plus DR minus

OK Cancel

DIALOGUES



Click this button to open the **CIRCULAR POLAR** dialogue

CIRCULAR POLAR

Optional

F-Max F-None

CCW CW
 DR Plus DR Minus



Click this button to open the **CIRCLE TANGENT** dialogue

CIRCLE TANGENT -CT-

F-Max F-None

Radius

R
 R0
 RR
 RL

DIALOGUES



Click on this button to open the **CHAMFER** Dialogue

CHAMFER

X

Y

Z

B

Feed

M

Radius

R R0

RR RL

Linear Move

F MAX F-None

OK Cancel



Click on this button so that you may enter a line number to jump to in your CNC Program. You can also use the **G** hot key on your keyboard. Type your number and press enter to find the line.

GOTO

Enter a Block Number



Click on this button or press the **S** stop hot key on your keyboard. You can then enter your required stop code. The default is STOP M

STOP

Stop Code

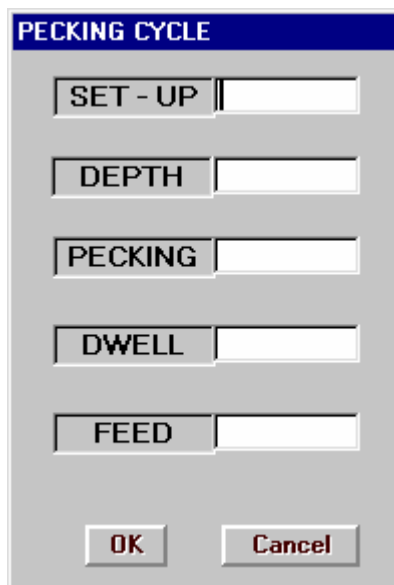
OK Cancel

DIALOGUES

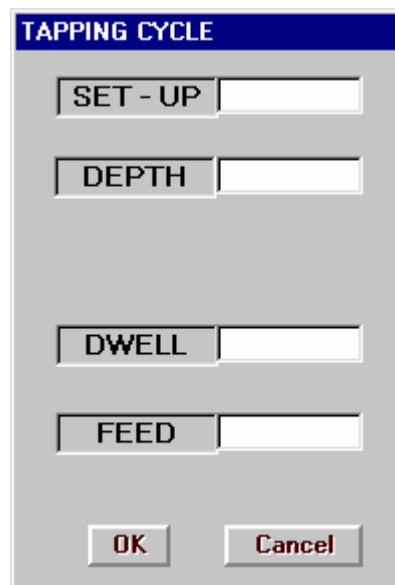


Clicking on this button will show you a dialogue from which you can select the type of Cycle you wish to use in your Heidenhain program. Scroll to the Cycle of your choice and then double click the selection . You can also use the Cycle Definition hot key **F5**

Note at this time we have implemented the most commonly used cycles into Heidi. Should you require any other cycle from your machine please send a request together with a sample of that cycle from your machine to www.support@shera.com.au



PECK DRILLING CYLCE



TAPPING CYCLE

DIALOGUES

SLOT MILLING DEFINITION

SET - UP	<input type="text"/>	1ST SIDE	<input type="text"/>	<input checked="" type="radio"/> X <input type="radio"/> Y <input type="radio"/> Z
DEPTH	<input type="text"/>	2ND SIDE	<input type="text"/>	<input type="radio"/> X <input checked="" type="radio"/> Y <input type="radio"/> Z
PECKING	<input type="text"/>	FEED RATE	<input type="text"/>	
PECK FEED	<input type="text"/>			

SLOT MILLING CYCLE

POCKET MILLING DEFINITION

SET - UP	<input type="text"/>	1ST SIDE	<input type="text"/>	<input checked="" type="radio"/> X <input type="radio"/> Y <input type="radio"/> Z
DEPTH	<input type="text"/>	2ND SIDE	<input type="text"/>	<input type="radio"/> X <input checked="" type="radio"/> Y <input type="radio"/> Z
PECKING	<input type="text"/>	FEED RATE	<input type="text"/>	Direction <input checked="" type="radio"/> DR+ <input type="radio"/> DR-
PECK FEED	<input type="text"/>			

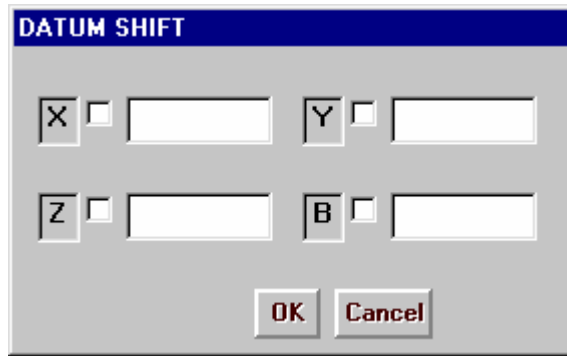
POCKET MILLING CYCLE

CIRCULAR POCKET DEFINITION

SET - UP	<input type="text"/>	RADIUS	<input type="text"/>	
DEPTH	<input type="text"/>			
PECKING	<input type="text"/>	FEED RATE	<input type="text"/>	Direction <input checked="" type="radio"/> DR+ <input type="radio"/> DR-
PECK FEED	<input type="text"/>			

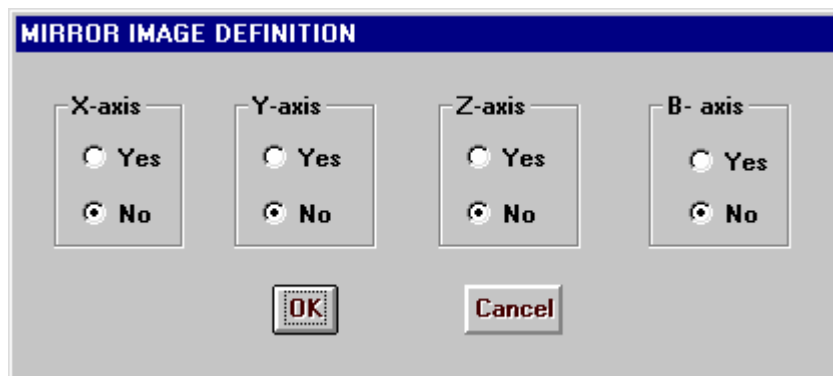
CIRCULAR POCKET MILLING CYCLE

DIALOGUES



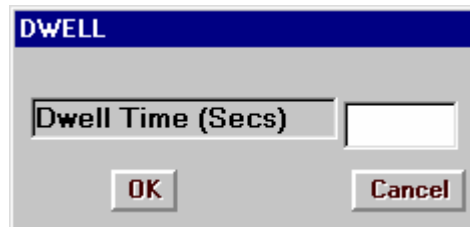
The DATUM SHIFT dialog box has a blue title bar with the text "DATUM SHIFT". Below the title bar, there are four input fields arranged in a 2x2 grid. Each input field is preceded by a letter in a small box: 'X', 'Y', 'Z', and 'B'. To the right of each letter is a small square checkbox. Below the input fields are two buttons: "OK" and "Cancel".

DATUM SHIFT



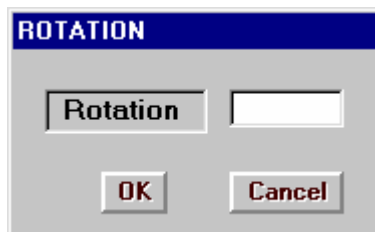
The MIRROR IMAGE DEFINITION dialog box has a blue title bar with the text "MIRROR IMAGE DEFINITION". Below the title bar, there are four columns, each representing an axis: "X-axis", "Y-axis", "Z-axis", and "B-axis". Each column contains two radio buttons: "Yes" and "No". The "No" radio button in each column is selected. Below the columns are two buttons: "OK" and "Cancel".

MIRROR IMAGE



The DWELL dialog box has a blue title bar with the text "DWELL". Below the title bar, there is a text label "Dwell Time (Secs)" followed by an empty input field. Below the input field are two buttons: "OK" and "Cancel".

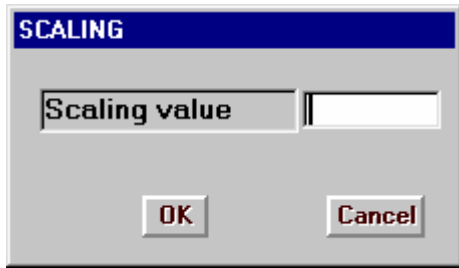
DWELL



The ROTATION dialog box has a blue title bar with the text "ROTATION". Below the title bar, there is a text label "Rotation" followed by an empty input field. Below the input field are two buttons: "OK" and "Cancel".

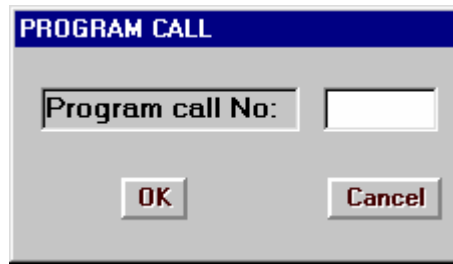
ROTATION

DIALOGUES



A dialog box titled "SCALING" with a blue header bar. It contains a text input field labeled "Scaling value" and two buttons at the bottom: "OK" and "Cancel".

SCALING



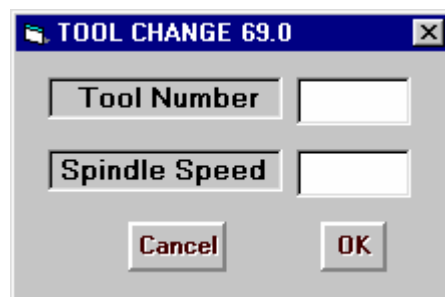
A dialog box titled "PROGRAM CALL" with a blue header bar. It contains a text input field labeled "Program call No:" and two buttons at the bottom: "OK" and "Cancel".

PROGRAM CALL



A dialog box titled "RIGID TAPPING CYCLE" with a blue header bar. It contains three text input fields labeled "SET - UP", "DEPTH", and "PITCH", and two buttons at the bottom: "OK" and "Cancel".

RIGID TAPPING



A dialog box titled "TOOL CHANGE 69.0" with a blue header bar and a close button (X) in the top right corner. It contains two text input fields labeled "Tool Number" and "Spindle Speed", and two buttons at the bottom: "Cancel" and "OK".

DIALOGUES



The **CYCLE CALL** button will initiate the following dialogue. Here you can enter any M Code which you may wish to use with your cycle. You can also use the Cycle Definition hot key **F6**

A dialog box titled 'CYCLE CALL' with a blue header. It contains a text input field labeled 'M-CODE' with a cursor. Below the input field are two buttons: 'OK' and 'Cancel'.

The **TOOL DEF** button will open a new dialogue where you may enter the Tool Number, Tool Length and Tool Radius for the Tool you wish to define to your Heidenhain.. You can also use the Tool Definition hot key **F11**

A dialog box titled 'TOOL DEF' with a blue header. It contains three text input fields labeled 'Tool Number', 'Tool Length', and 'Tool Radius'. To the right of the 'Tool Length' and 'Tool Radius' fields are buttons labeled 'OK' and 'Cancel' respectively.

DIALOGUES



Tools can be recalled in your Heidenhain program by clicking the **TOOL CALL** Button in Heidi. Here you may enter the necessary data to inform the Heidenhain of which Speed and Feed Rate you are going to require when this tool is cutting. You must also define which Axis is the primary axis of the Spindle so that the Heidenhain knows which way your machine is configured. Typically Z is the most common “Working Axis”

TOOL CALL

Tool Number 1

Spindle Speed 4500

Working Axis

X-axis

Y-axis

Z-axis

OK

Cancel



The **LABEL SET** button will enable you to define part of your program as a repeatable macro. These Labels can then be recalled at anytime later in your program. Enter a suitable Label number. Setting the Label number to Zero will indicate to the Heidenhain that you have reached the end of the program macro.

LABEL SET

LABEL No

OK

Cancel



Labels can be recalled multiple times to save the programmer the need to write the same data over and over. Enter the Label number to be recalled and the number of time you wish to repeat the running of that label.

LABEL CALL

Label Number

No of Repeats

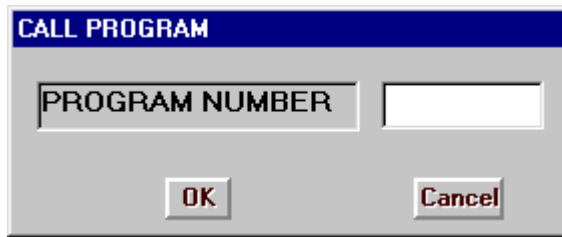
OK

Cancel

DIALOGUES



The **CALL PROGRAM** Button will allow you to enter the number of a sub program call in your CNC program.



The **NO ENTER** button has no use in the HEIDI software at this point in time.



The **DELETE** button will delete the currently highlighted block in your program. If a Block has been deleted you will see the **Un-Delete** button in the lower right hand corner of your Heidi screen.

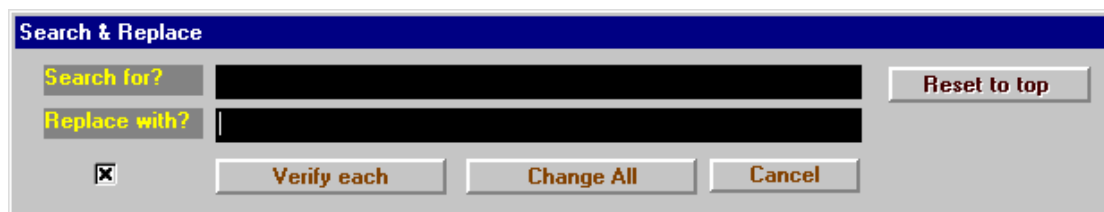
Click this button to place the most recently deleted block back into your program.



The **Comment** Button will allow you to enter a new comment into your Program. The Dialogue will also be displayed if you double click on an existing comment in your CNC program.



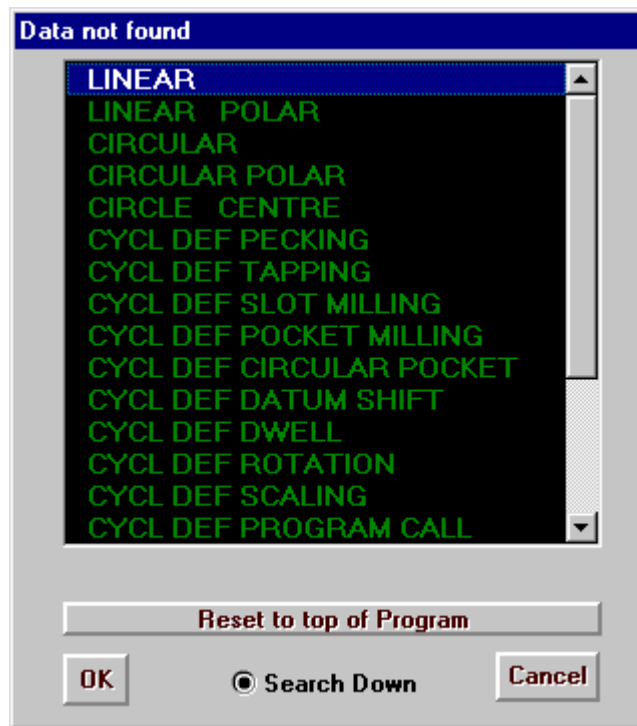
The **Replace** button offers you the opportunity to search through your program and change existing values to new values. You can step through your program block by block and verify each item found which matches your search criteria or have the software run through your program and change all matched values automatically. If you make the Search and Replace values identical then you can use this dialogue to search for particular items in your program.



DIALOGUES

Search

The **Search** button allows you to search through your program for particular types of program data. Scroll through the list of search criteria and then click on the cycle you wish to search for. The next cycle found will be highlighted on the Heidi Program screen.



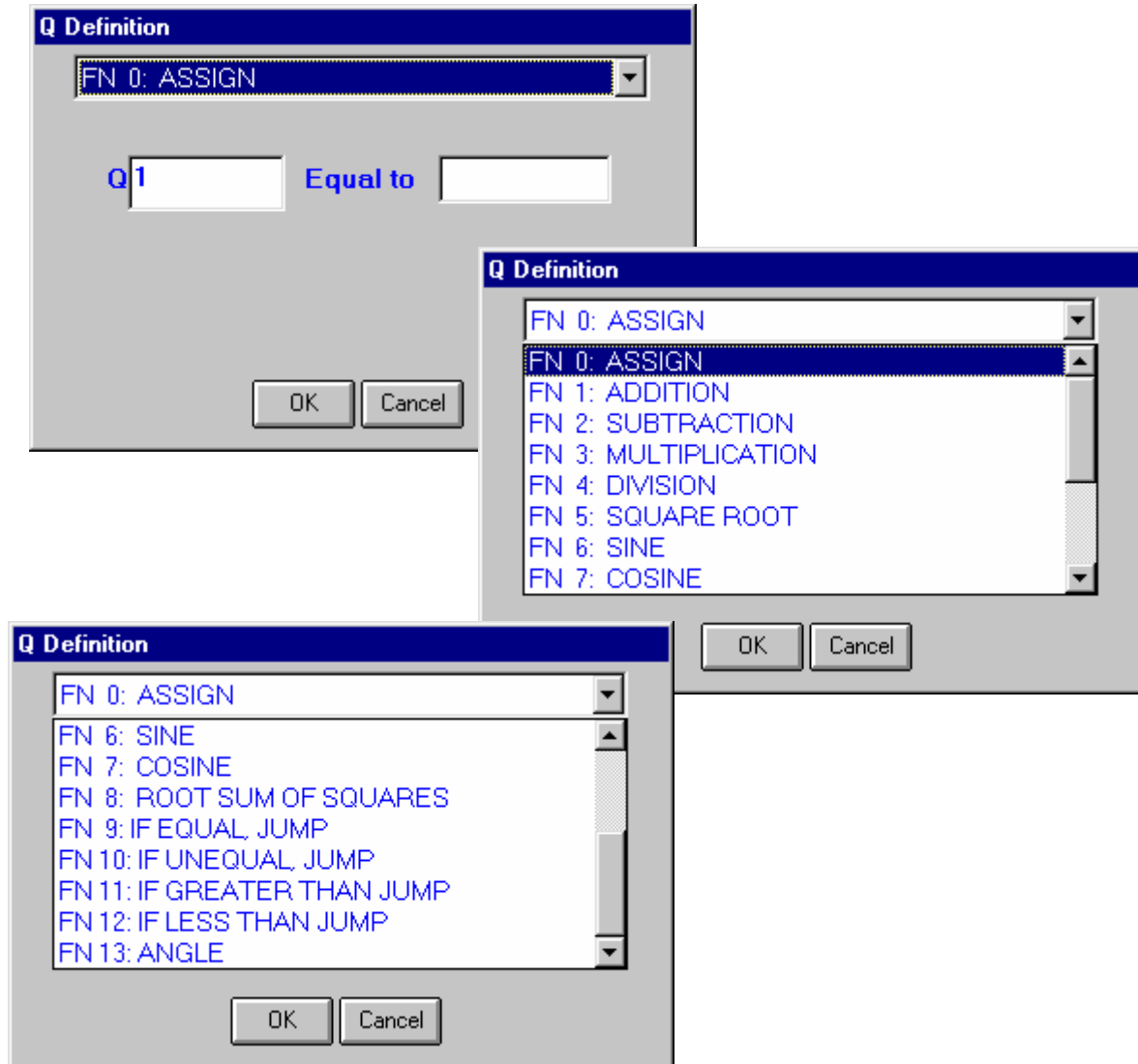
Begin **End**

The **Begin** and **End** buttons will force the highlighted block to be either the first line or last line in your program.

Q DEFINITIONS

Q DEF

Clicking the **Q DEFINITION** button will open the dialogue for you to make use of Variable programming in your CNC Program. This dialogue will then offer you a drop down list for the various types of mathematical expressions used in variable programming.



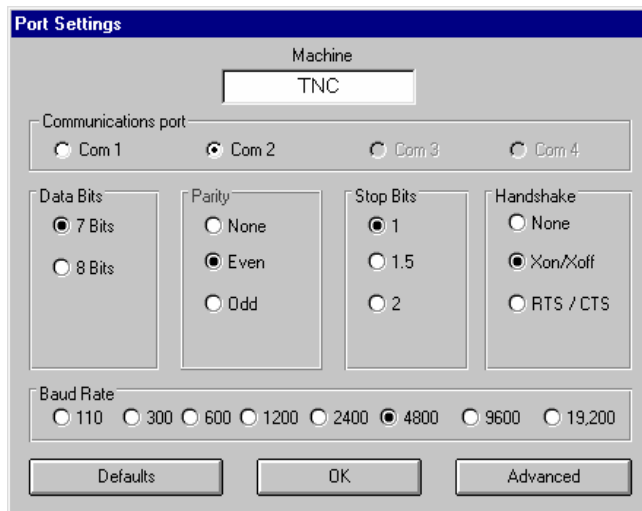
From the Above dialogues you can see the 13 different types of mathematical operators which can be used within your Q Definitions. Select one type of QDEF from the drop down and you will be presented with the appropriate input fields for you to enter your equation.

COMMUNICATIONS

To Send your program file to the CNC Control you must first setup your communication parameters.

Click on the **Communicate** Pull down text menu

Click on *Communications Setup (New Style)*



Here you can configure the parameters required to enable two way communications on your Heidenhain.

Click the correct Radio buttons (the little round buttons) to setup the desired values

Communications Port

Com1 is normally a 9 Pin Male connector at the back of your computer. Com 2 is normally a 25 pin male connector. On newer computers you will find both Com1 and Com2 are both usually 9 pin connectors

Com 3&4 are currently not yet supported.

Data Bits

Normally use 7 Bits for ISO and 8 Bits for EIA codes.

Parity

Normally use Even for ISO and None for EIA codes..

STOP BITS

Normally use 1 stop Bits.

HANDSHAKE

Normally use XON/XOFF. Use RTS/CTS for EIA code
We do not recommend the use of NONE.

BAUD RATE

Select the recommended setting for your machine. If there is not one for your machine then we suggest you start with 2400 Baud and work your way up until you have unstable transmissions. Once you establish this ceiling change back to a lower setting.

NOTE:

Some machines require a small delay in between each character sent to the machine. There is more information about setting these delays in the subsequent advanced settings section of this manual.

COMMUNICATIONS

Click the **ADVANCED** button to enable more communications setup options.

- **Number of characters for Paper Tape Leader** *Default = 300*
- **Number of characters for Paper Tape Trailer** *Default = 300*

Enter the number of Nulls to be punched before and after the main body of your CNC Program.

Note: there are 10 nulls per one inch of Paper Tape in the old way of thinking.

- **Delay Between Characters 1/1000 Sec** *Default = 0*

Enter the number of Milliseconds delay between sending each program character to your CNC.

Effective only when Sending data from your PC

COMMUNICATIONS

- **Character used to ignore rest of block** *Default = ^*

Blocks starting with this character will not be transmitted from your PC to your CNC.

If the character is part way through your Block of CNC Code only the portion to the left of the character will be transmitted.

- **Wait for XON before sending data to your machine.** *Default = YES*

Heidi for windows will wait until it sees a start signal from your CNC before it sends data. Your screen will show a message similar To the one below.



- **Send Carriage Returns to the Machine** *Default = YES*

Carriage returns will not be filtered out of your program when transmitting data to your CNC.

- **Send Line Feeds to the Machine** *Default = YES*

Carriage returns will not be filtered out of your program when transmitting data to your CNC if this option is set to YES.

- **Send Carriage Returns before 1st Char** *Default = NO*

Carriage returns will not be transmitted to your CNC prior to the first character in your file. This function allows us to filter out unwanted characters before the Valid CNC code.

- **Send Line Feeds before 1st Char** *Default = NO*

Same as for Carriage returns

COMMUNICATIONS

- **Ignore incoming Characters untils 1st CR** *Default = NO*

When receiving data from your CNC Heidi will ignore any characters until it sees a carriage return (Ascii 13).

IE: A carriage return character is the first valid program character if set to "YES".
- **Timeout when receiving (in seconds)** *Default = 10*

Heidi will wait for this amount of time for additional characters when receiving data from your CNC. If No characters are received within this amount of time then the Receive function will be terminated.
- **Filename Extension (.NC)** *Default = .NC*

Enter your preferred filename extension here.
- **Program ID Character** *Default =*

Heidi will filter out all characters in your file when transmitting a program to your CNC until it sees this particular character.
- **ISO / EIA** *Default = ISO*

This option allows you to use one of the two most widely used CNC Tape Codes. Not normally used with Heidenhain controls

ISO = Even Parity and 7 Data Bits. **EIA** = No Parity and 8 Data Bits
- **MACHINE TYPE** *Default = MILL*

Selection of one of the described machine types will change the type of Bitmap displayed in the machine selection Panel. This option is not currently supported. The function was designed for use in our Mycoms software.
- **FORCE UPPER CASE CHARACTERS**

When Checked this option ensures that all characters are converted to Upper Case when sending data to the CNC. Many CNC's, especially Fanuc do not like lower case characters.
- **SEND PROGRAM REPEATEDLY**

Normally left Un-checked this option will re-queue a program (when Checked) to send to the CNC again as soon as it finishes it's current transmission to your CNC. This will save the CNC operator walking back to the PC to prepare the program a second time if he has trouble during a transfer from PC to CNC.

COMMUNICATIONS

- **CONVERT LF TO CR ON RECEIVE**

Normally left Un-checked this option will ensure additional Line Feeds are inserted into your program while it is being received from your CNC.

Note: Line feeds are only added when this option box is Checked

- **SEND SPACES TO MACHINE**

Normally left Un-checked this option will ensure that spaces within your file will be transmitted. Many CNC's do not require spaces between words. Fanuc's for example will automatically place spaces between each word of CNC code.. Sending spaces can mean wasted time transmitting these un-wanted codes.

Note: Ensure this box is checked if the spaces **MUST** be sent to your CNC.

- **SPECIAL LEADER CHARACTERS** *Default =*
- **SPECIAL TRAILER CHARACTERS** *Default = 37,*

Click on this field to enable a look up list of ASCII characters as per the next diagram.

Double Click on the Lookup list for the desired character, it's ASCII decimal value will be inserted into the Leader or trailer field.

Click the *Esc* button to clear the lookup list from the screen.

Note: Each decimal value **MUST** be separated by a Comma.

Port - Advanced Settings

No of Characters for Paper Tape Leader	<input type="text" value="300"/>	<ul style="list-style-type: none"> 0 NULL 1 SOH 2 STX 3 ETX (Ctrl 'C') 4 EOT 5 ENQ 6 ACK 7 BEL 8 BS (Backspace) 9 HT 10 LF (Line Feed) 11 VT 12 FF (Form Feed) 13 CR (Carriage return) 14 SO 15 SI 16 DLE
No of Characters for Paper Tape Trailer	<input type="text" value="300"/>	
Delay between Characters 1/1000 Sec	<input type="text" value="0"/>	
Character used to Ignore rest of Block	<input type="text" value="^"/>	
Wait for Xon before sending file to M/c	<input type="text" value="YES"/>	
Send Carriage Returns to Machine	<input type="text" value="YES"/>	
Send Line Feeds to Machine	<input type="text" value="YES"/>	
Send Carriage Return before 1st Char	<input type="text" value="NO"/>	
Send Line Feeds Before 1st Char	<input type="text" value="NO"/>	
Ignore Incoming Characters until 1st 'CR'	<input type="text" value="YES"/>	
Timeout when Receiving (in Seconds)	<input type="text" value="10"/>	
Special Leader Characters	<input type="text" value="10."/>	
Special Trailer Characters	<input type="text" value="37,14."/>	
File name Extension (.NC1)	<input type="text" value=".NC"/>	
Program 'ID' Character	<input type="text"/>	

Send PATH

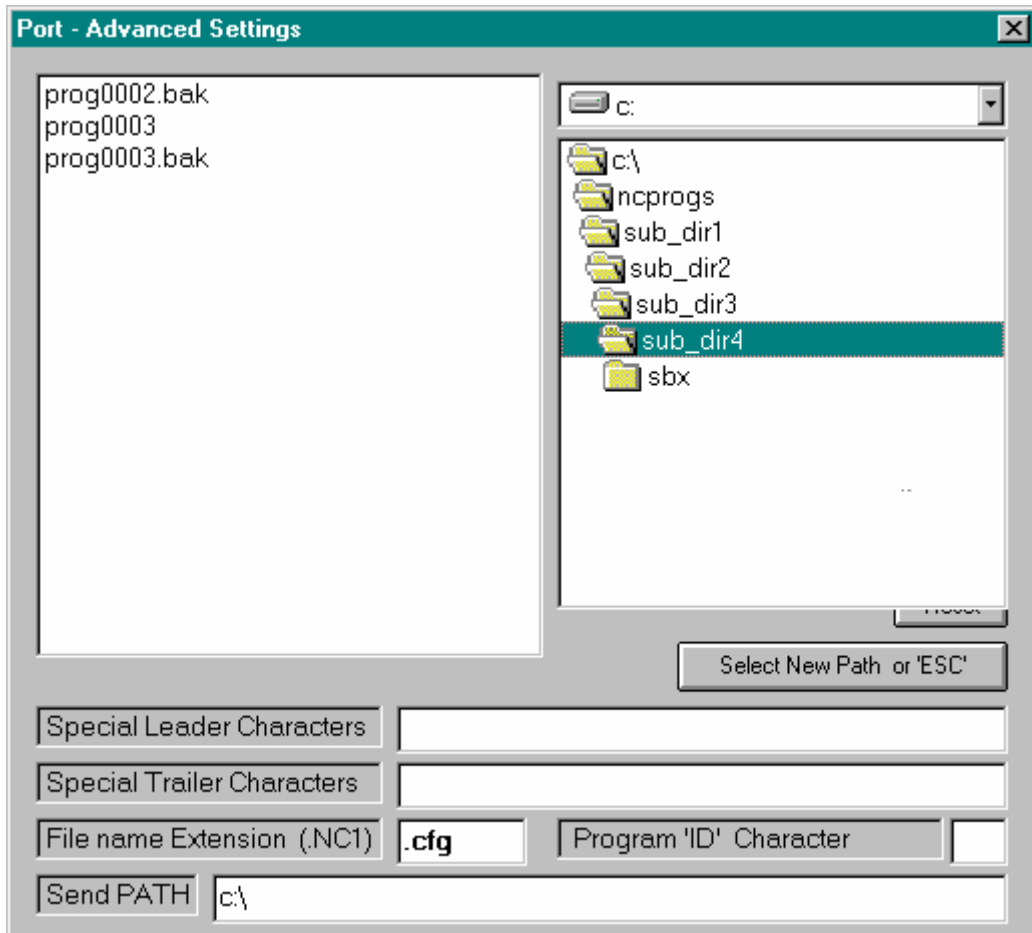
COMMUNICATIONS

- **SEND PATH**


Click on this field to enable a path selection box to appear as shown below.

You can change the Active drive for your current CNC's default path when searching for files. Then click down until you find the required folder for the default files. Heidi will use these defaults each time you start Heidi

Click **Select New Path** to set your new defaults or click the **ESC** button to exit from the drive and path selections.

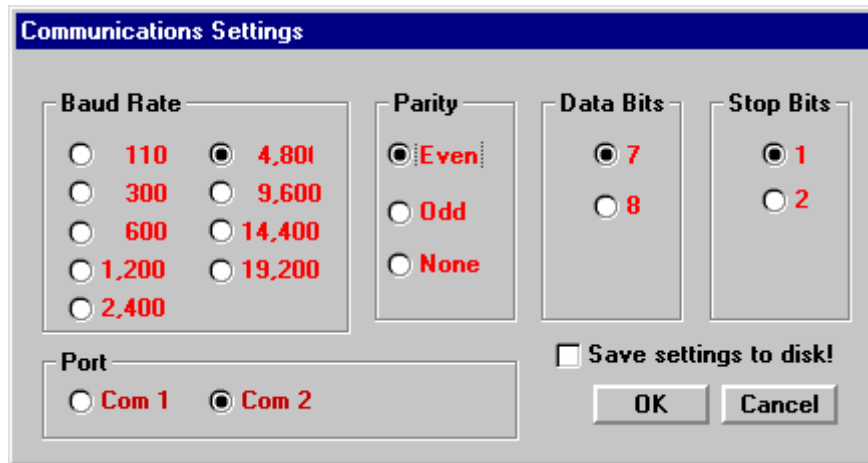


COMMUNICATIONS

Once you have configured your machine and Heidi you will then be able to send and receive files between your PC and Heidenhain using the  icon

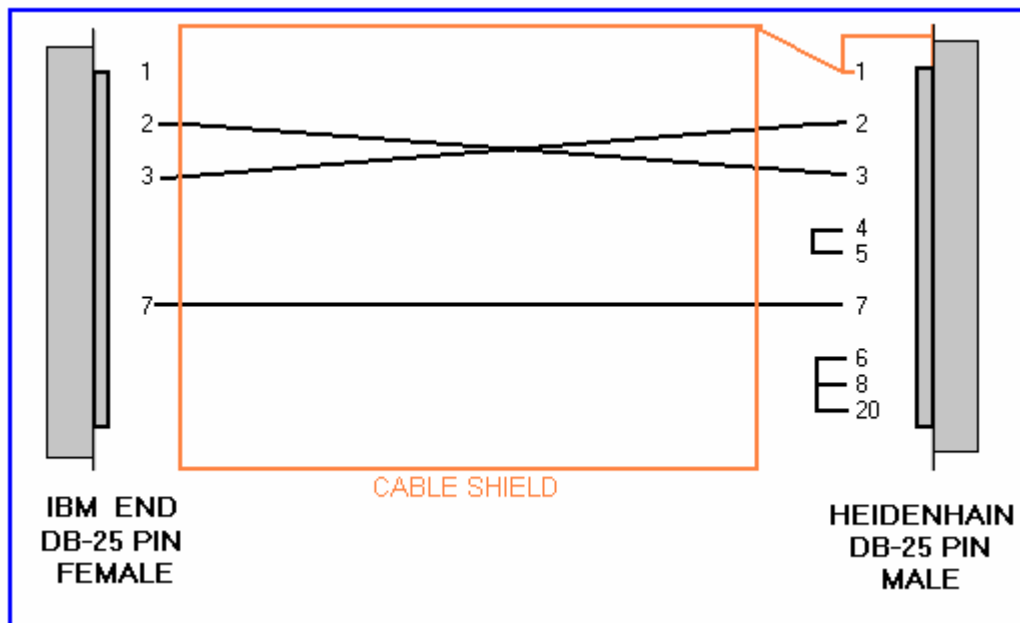
Heidi also offers an older style of communications protocol. We decided to leave this protocol in Heidi, however the previous screens are the preferred method.

To initiate the Old style communications settings click on *Communicate* pull down menu and then *Old Style communications* and then *Settings(Old Style)*. As you will see from the diagram below the old style system offers far less options for configuration.



NOTE:

The old Style communications protocol only sends data to the Heidenhain. It will not store programs from the machine.



CABLE DIAGRAM

COMMUNICATIONS

SPECIAL NOTE:

Lightning strikes can cause severe damage to machine tool communications equipment. Please ensure that you take suitable precautions to protect your machine tools and computers by adequate shielding and if possible either optical isolation or total disconnection during electrical storms.

Old Style Communications settings can be saved by clicking on the *Settings* and then *General* pull down text menus. You will be presented with the following dialogue

General Settings

Default Program Directory
C:\NCPROGS\SMR

Program Name
 4 Characters long 16 Characters long
 10 Characters long
 Alpha characters Can be used

Axis movement blocks
 Force X-Y-Z or 4th axis moves to L'inear moves
 Fourth Axis = B Fourth Axis = C

Generic

Program Text Color on background

Program background Color

Program Text Font Size: 9

Reload from existing Disk settings
 Reset to Heidi default values
 Save current settings to disk

OK

Click onto the *Save Settings to Disk* checkbox and then click Ok to update the Heidi.ini file

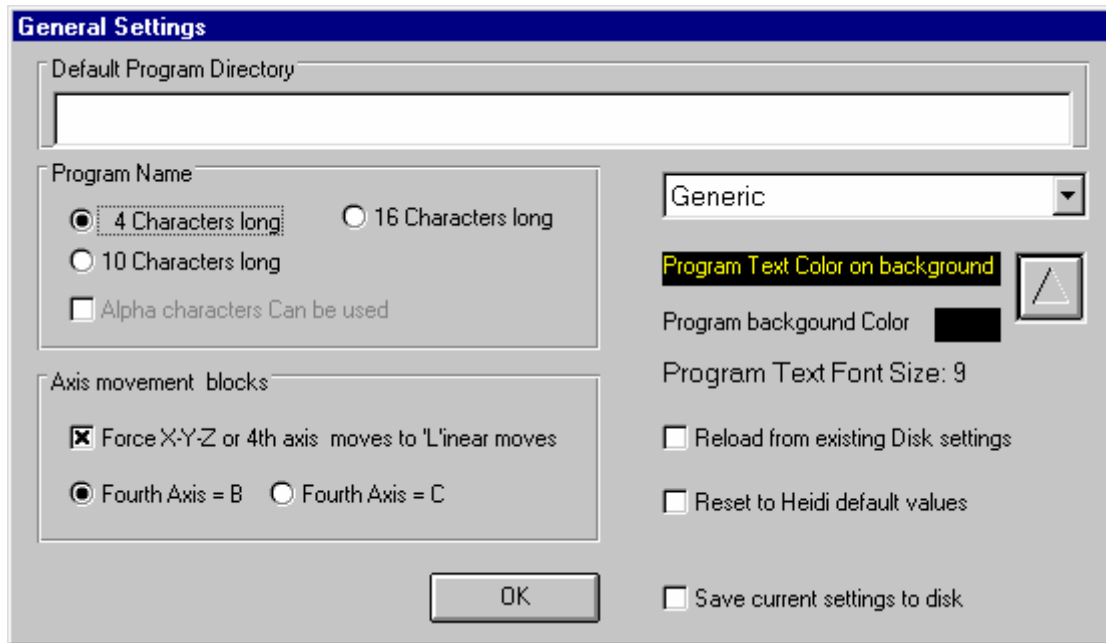
SETTINGS

Heidi offers several settings which can allow you to configure the way the software looks and operates.

Click on the *Settings* pull down text menu, then choose one of the Settings types.

Settings

General




- **Default Program Directory** Type in here the folder that you wish Heidi to open first when looking for your CNC programs.
- **Program Name** this options will limit the way in which the user enters The program number.
 - 4 Characters in length
 - 10 Characters in length
 - 16 Characters in length

Please note that Heidi currently only supports the use of numeric values in the BEGIN PGM and END PGM blocks.

- **Axis movement blocks**
 - Force X-Y-Z or 4th axis moves as linear, will ensure that the blocks are created with an **L** rather than just a simple X, Y, Z movement
 - Fourth Axis = B allows the user to configure which Letter is produced when programming his 4th Axis. Either **B** or **C**

SETTINGS

- **Generic** Pull down menu allows you to choose which type of Tool change codes you require when using the Toolchange button 

Please contact us at support@shera.com.au if you require a special tool change command to be added to Heidi.

- Program Text colour on Background *Displays the current settings*
- Program background colour “ “ “ “
- Program Text font size “ “ “ “

- **Reload from Existing Disk settings** Resets your current Heidi settings to the values which were used the last time you saved the settings to disk.
- **Reset to Heidi Default values** Resets all values to the system defaults
- **Save current settings to Disk** When this box is ticked Heidi will write all current values to the file C:\Windows\Heidi.ini. This will include settings for the old style communications. New style communication settings will be written to a separate file when working in the new style communication configuration pages.

Click the Ok button when this checkbox is ticked to save the new settings!

Settings

Colours

Program Text colour

Program Text Background

Clicking either of the colour menus will present you with the windows colour pallet where you may select a colour for either the Text in your program or the background colour of the text.



SETTINGS

Settings

Goto Options

This function is currently turned on all the time. IE the option cannot be set.

Settings

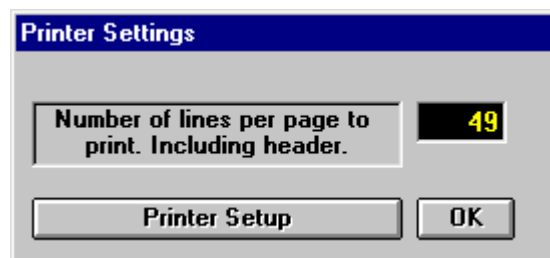
Font Size

Here you may choose what size of font your program should be displayed in. The current sizes are 8,10,12,14 points.

File

Page Setup

It is often necessary to be able to adjust the number of lines to be printed when creating hard copies of your CNC program. The page setup menu allows you adjust the number of lines as required. The value for this dialogue is stored in the file C:\windows\Heidi.ini



To save the values you can either edit the parameter in the Heidi.ini file or use the Settings General menu to save the system settings as described above.

DEFAULTS

The default start up parameters for Heidi can be found in the file c:\windows\Heidi.ini. If this INI file cannot be found when the software is loading then a new INI file will be built with the following defaults.

START_DIR	C:\HEIDI	defines the default directory to find programs
LENGTHNAM	8	Maximum number of characters in a program name this is for the BEGIN PGM & END PGM blocks
COMS_PORT	2	Communications port for machine communications (Old style communications)
BAUD_RATE	4800	Baud rate for communications (Old style communications)
PARITY	E	Parity for communications (Even, None or Odd parity) (Old style communications)
DATA_BITS	7	No of Data bits for old style communications (Old style communications)
STOP_BITS	1	No of Stop bits for old style communications (Old style communications)
TRAILER	3,37,	Special output codes for terminating send to machine these are in decimal form. EG Decimal 37 = %
FONT_SIZE	12	The size of font to be used for your program text
FORECOLOR	14671839	The colour of the program text
BACKCOLOR	8421376	The background colour for the program text
FOURTHAXI	B	The letter to be used when programming a 4 th Axis
PRINTLINE	49	The number of lines to be printed on hard copies
WINDOWBIG	1	The size of the default window on start up 0= Small window, 1= Large window
LINEAR_ON	1	Linear flag can be forced on in Linear and Non Linear Moves. Setting LINEAR_ON to 0= leave the program block as it was when you selected it Setting to 1= Make sure the movement is Linear

NOTES:

The HEIDI.INI file can be modified using a normal text editor or the windows Notepad.