

Dolphin PartMaster Turning

Copyright © 2000-2005 Dolphin CADCAM Ltd.

This document is copyrighted and all rights are reserved. This document may not, in whole or in part, be copied or reproduced in any form, or by any means without the prior written consent of Dolphin CadCam Ltd.

Dolphin CadCam Ltd. Makes no representations or warranties with respect to the contents hereof and specifically disclaims any warranties of fitness for any particular purpose. The information in this documentation and the software programmes to which it refers are subject to change without notice.

Dolphin CadCam Ltd. assumes no responsibility for any errors that may appear in this document.

Dolphin CadCam Ltd.

7, South Dean Road,

Kilmarnock. KA3 7RE

Scotland.

Tel +44 (0)1563 543989

Fax +44 (0)1563 530874

Email info@dolphin.gb.com

Revision – March 2005 – 8-1024-1a

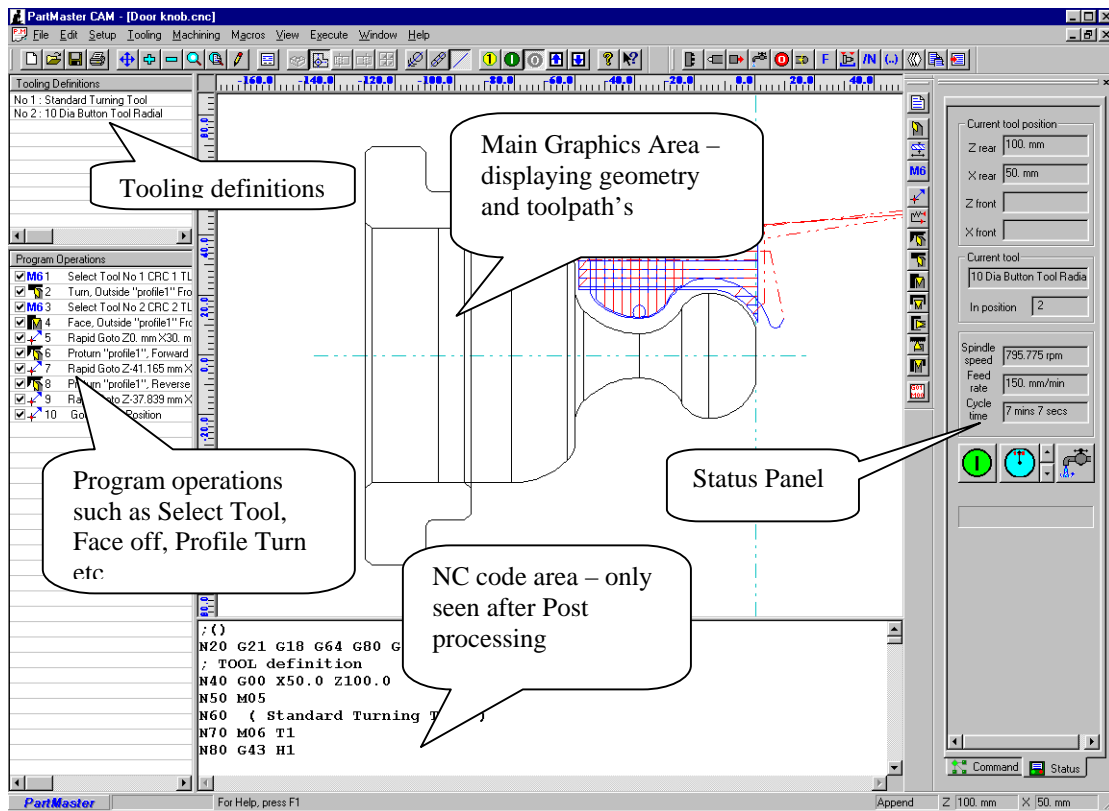
Table Of Contents

Dolphin PartMaster Turning.....	1
Table Of Contents.....	2
Screen Layout.....	5
Program Operations	6
Program Simulation	7
Status Panel	8
Tooling Definitions Panel.....	9
Top Toolbar	10
Operations Toolbar - Turning	10
Miscellaneous Toolbar - Turning	10
Turning Menus.....	11
Graphic Display - Turning.....	12
Toolpath Display	12
Geometry Display	13
Setup - Billet - Turning.....	14
CREATING PROGRAMS	15
EXECUTING PROGRAMS.....	16
Execute - Run Program	17
Execute - Stop Program	17
Execute - Single Step	17
Execute - Pause Mode	17
Execute - Slower.....	18
Execute - Faster	18
Execute - Post Process	19
File Menu	20
File - New.....	20
File - Open	20
File - Save.....	21
File - Close.....	22
File - Save As	22
File - Import.....	23
File - Export	24
File - Print	24
File - Print preview	25
File - Send NC Program to... ..	26
File - Send as Mail	26
Setup Menu - Turning.....	27
Setup - Machine Type - Turning.....	27

Setup - Machine Type - Program ID.....	28
Setup - Machine Type Turning - Tool Change	29
Setup - Machine Type - Vector Tolerance.....	30
Setup - Machine Type - Origin.....	31
Setup - Machine Type - Extra Info.....	32
Setup - Fixture Offsets - Turning	32
Setup - Reset - Turning	33
Setup - Billet - Turning.....	33
Setup - Load Setup - Turning	34
Setup - Save Setup - Turning.....	34
TOOLING.....	35
Tooling - Define Turning Tool.....	36
Tooling - Definitions.....	37
Drill.....	39
Tooling - Load Toolfile	43
Tooling Menu - Turning	43
Tooling - Save Toolfile.....	43
Tooling - Select Turning Tool	44
MACHINING	45
Machining - Drill.....	46
Machining - Goto	48
Machining - Groove/Trepan.....	49
Machining Menu - Turning.....	52
Machining - Profile Turn	53
Machining - Profile Turn - Options.....	56
Machining - Threadcut.....	58
Machining - Turn/Face.....	60
Machining - Partoff	64
MISCELLANEOUS FUNCTIONS.....	65
Barfeed	66
Block Skip	66
Chuck.....	67
Coolant	67
Feedrate	68
Insert.....	68
PP Function	68
Tailstock.....	69
Stop	70
Spindle.....	70
Part Catcher.....	71

Help Menu	71
Help - About DCAM	72
Help - What's New	72
View Menu	73
View - Cycle Time Data	73
View - Editor Font	73
View - Geometry Info	74
View - NC Program	74
View - Notes	75
View - Options	76
View - Preferences	77
View - Preferences - Angles	78
View - Preferences - Colors	78
View - Preferences - Distance	79
View - Preferences - Feed Per Min	79
View - Preferences - Feed Per Revolution	80
View - Preferences - Folders	80
View - Preferences - Fonts	81
View - Preferences - Options	81
View - Preferences - Retract	82
View - Preferences - Simulation	82
View - Preferences - Spindle	83
View - Preferences - Tabs	83
View - Preferences - Tuning	84
View - Preferences - User Name	85
View - Preferences - Z Planes	85
View - Redraw	86
View - Rulers	86
View - Split	87
View - Toolbars	87
View - Zoom	88
View - Zoom - Cursor	88
View - Zoom - Cutter Path	88
View - Zoom - Double	88
View - Zoom - Extent	88
View - Zoom - Half	89
View - Zoom - Previous	89
View - Zoom - Selection	89
Window Menu	89

Screen Layout



Depending on your screen resolution the above layout may be different on your PC, don't forget all toolbars are dockable in your preferred locations.

All of the individual panes of the above display screen can be re-sized by dragging the bordering bars. Toolbars can be switched on and off by means of the commands under the **View - Toolbars** sub-menu

Main Menu

The main menu consists of a series of 9 menus, which control all aspects of Dolphin CAM.

Toolbars

The toolbars contain buttons, which offer quick access to the main functions. All of these operations are also available via the menus. See Toolbars

Tooling Definitions area

This area is used to display all the tools that you have defined within the program, either by means of the Tooling Menu or by the Tool Definition buttons within the Operations Toolbar.

- Program Operations area** This area is used to display all the machining operations that you have defined by means of the Machining Menu or the by the buttons in the Operations Toolbar.
- Graphic Display area** This area is used to display the geometry that you have created or imported. The cutter paths and machining sequences that you define in Dolphin CAM are also displayed here.
- Edit Area** This area is used to display the NC code generated when the program is executed via one of the post processors.
- Status Panel** The Status panel contains a constantly updated display of various parameters for the current program.
- Status Bar** The Status bar shows information about the button or menu field, which lies under the cursor.

Program Operations

Program Operations	
<input checked="" type="checkbox"/> M6	1 Select Tool No 1 CRC 1 TLC 1, Spin 2500.0000 Ch
<input checked="" type="checkbox"/>	2 Turn, Outside "PartProfile" From Z-190. mm X100. n
<input checked="" type="checkbox"/>	3 Face, Outside "" From Z-10. mm X0. mm To Z0. mm
<input checked="" type="checkbox"/> M6	4 Select Tool No 4 CRC 4 TLC 4, Spin 2500.0000 Ch
<input checked="" type="checkbox"/>	5 Groove, Outside From Z-50. mm X20. mm To Z-40. r
<input checked="" type="checkbox"/>	6 Rapid Goto Z-130. mm X60. mm (Abs)
<input checked="" type="checkbox"/>	7 Groove, Outside From Z-130. mm X58.146 mm To Z
<input checked="" type="checkbox"/> M6	8 Select Tool No 5 CRC 5 TLC 5, Spin 2500.0000 Ch
<input checked="" type="checkbox"/>	9 Proturm "PartProfile", Forward Outside
<input checked="" type="checkbox"/>	10 Rapid Goto Z0. mm X10. mm (Inc)
<input checked="" type="checkbox"/> M6	11 Select Tool No 3 CRC 3 TLC 3, Spin 2500.0000 C
<input checked="" type="checkbox"/>	12 Threadcut Cycle

As each operation in the machining sequence is defined it is added to the bottom of the list in the **Program Operations** Pane. You can re-set the size of this pane to the full window size by means of the **Operations** command in the **Window Menu**. The **Tile** command in the same menu will set all the panes within the display window back to the default configuration.

Activate/De-activate operation

Each operation is preceded by a box, which contains either a tick or a cross. When the operation is first defined the box is ticked, showing that the operation is activated. If you wish to de-activate the operation, simply click on the box with the left hand mouse button. Only those operations that are activated will be carried out when the program is executed for a simulation or to produce NC code via the post processor.

Edit an operation

To edit an existing operation, double click on the entry with the left hand mouse button. The Dialog that defines the operation will be displayed showing all the values and parameters that

you originally set. Make any necessary alterations and then click on OK. Note that the alterations will NOT be carried out until the program has been executed after the alteration.

Errors within a sequence

If the operation icon is followed by a question mark, it means that Partmaster has found a problem within the operation during execution. Use the **Edit** procedure as shown above, to correct the error.

Re-order operation sequence

To re-order the operations simply click on the text part of the entry and, holding down the mouse button, drag the operation to the required position within the sequence. Upon release of the mouse button the operation will be placed in the new position within the list and the entries will be automatically re-numbered.

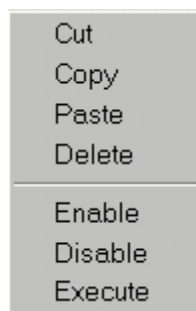
Delete operation sequence

To delete an operation from the sequence simply click on the text part of the entry and hit the delete key on the keyboard or select **Delete** from the **Edit Menu**.

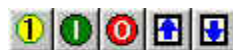
Insert operation into sequence

A new operation is always added to the bottom of the sequence. Define the operation as required and then use Re-Order as described above to move the operation to the required position.




Block **Copy, Cut & Paste** operations can also be performed on multiple entries in the program operations list. Any selection of items in the list can be made by holding down the **Control** key whilst clicking with the left hand mouse key on the desired items. A sequential group of items may be selected by clicking on the first & last items whilst holding down the **Shift** key. By clicking with the right hand mouse button a new menu is displayed with the following options








Program Simulation



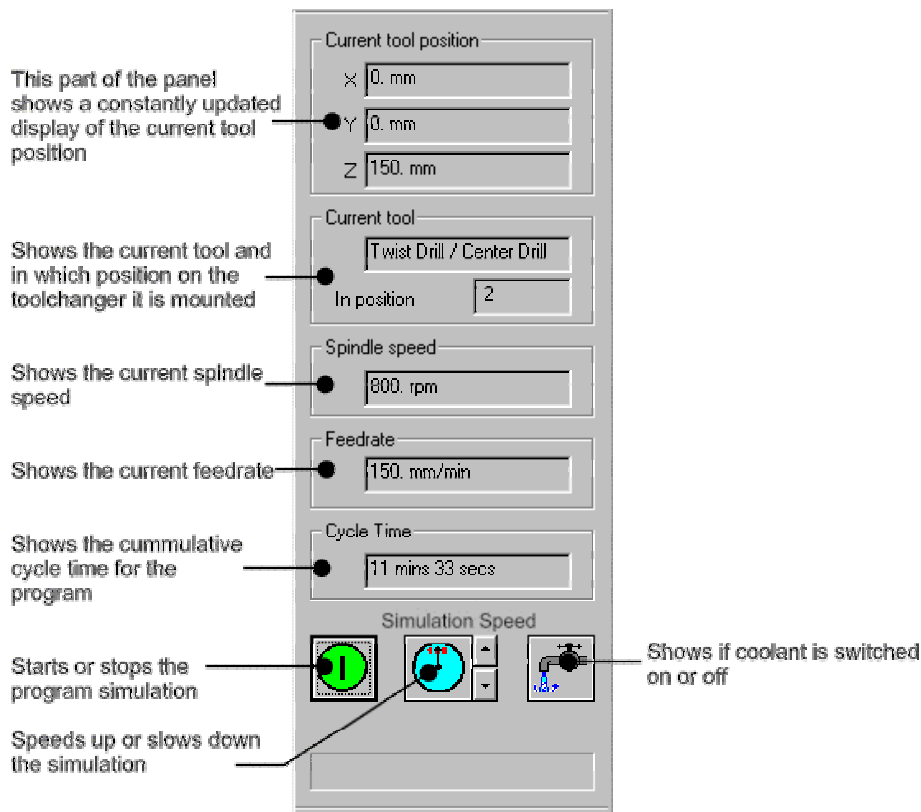
This group of buttons controls the simulation of the program that you have defined.

	This button sets the simulation to single step mode. Every time you press the Execute button, only the next operation in the sequence will be carried out. Note that a Tool Definition or a Tool Select command is counted as an operation. This button is the same as the menu field Execute - Single Step
	Starts the execution of the program simulation. The  button on

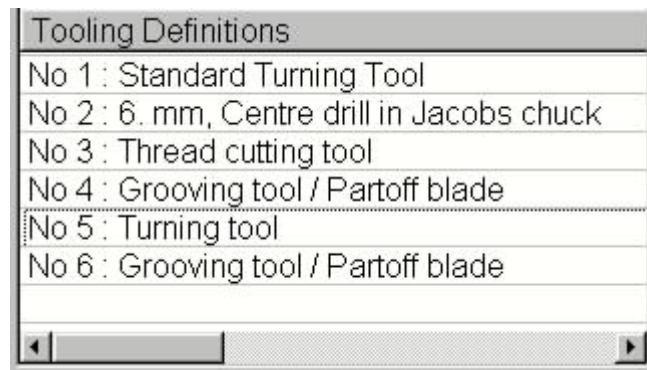
	the Status Panel is the same as this. This button is the same as the menu field Execute - Run Program
	Stops the execution of the program simulation. Note that this button is only available if a simulation is actually running. The  button on the Status Panel is the same as this. The menu field Execute - Stop Program has the same effect as these buttons
	Pressing this button when a program simulation is running will have the effect of speeding up the simulation. This button is the same as the menu field Execute - Faster . The Simulation Speed control on the Status Panel has the same effect as this button.
	Pressing this button when a program simulation is running will have the effect of slowing the simulation down. This button is the same as the menu field Execute - Slower . The Simulation Speed control on the Status Panel has the same effect as this button.

See also: Status Panel & Execute Menu

Status Panel



Tooling Definitions Panel



As each tool is defined it is added to the bottom of the list in the **Tooling Definitions** Pane. You can re-set the size of this pane to the full window size by means of the **Tooling** command in the **Window Menu**. The **Tile** command in the same menu will set all the panes within the display window back to the default configuration.

Edit a Tool Definition

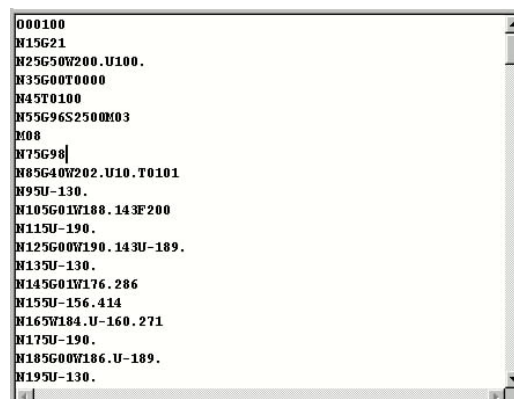
To edit an existing definition, double click on the entry with the left hand mouse button. The Dialog that defines the tool will be displayed showing all the values and parameters that you originally set. Make any necessary alterations and then click on OK. Note that the alterations will NOT be carried out until the program has been executed after the alteration.

Delete Tool Definition

To delete an operation from the sequence simply click on the text part of the entry and hit the delete key on the keyboard or select **Delete** from the **Edit Menu**.

See Also: Define Turning Tool & Define Drilling Tool

NC Code Area



The screenshot shows a window displaying a list of NC code lines. The code is as follows:

```
O00100
N15G21
N25G50W200.U100.
N35G00T0000
N45T0100
N55G96S2500M03
M08
N75G98|
N85G40W202.U10.T0101
N95U-130.
N105G01W188.143F200
N115U-190.
N125G00W190.143U-189.
N135U-130.
N145G01W176.286
N155U-156.414
N165W184.U-160.271
N175U-190.
N185G00W186.U-189.
N195U-130.
```

This area of the screen displays the NC code which is generated by means of the **Execute - Post Process** command. **See Post Process**. The font that is used for the text display may be changed by means of the **View - Editor Font** command. **See View - Editor Font**

The display of this area can be switched on and off by means of the **View - NC Program** command in the **View** menu

The code displayed can be edited by means of a full screen editor, which is active when the screen cursor is clicked within the area.



USE WITH CARE

If the NC code is edited in this manner; please remember that it no longer matches the defined program and the post-processor that was used. If the program is re-run in the future the NC code generated will **NOT** be the same as the edited version.

Top Toolbar



This Toolbar contains all the major global operations necessary Open, Save and print programs and also options for how the screen will presented and how to draw the tools, and execute (run) a program

Operations Toolbar - Turning



This Toolbar contains all the major operations necessary to define an NC program. The buttons are laid out in a logical progression:

i.e. at the left hand end (top) is the Setup button. This operation must be carried out before any of the other commands are made available. The second button enables you to define a tool, the third button enables you to select a tool. No machining operations will become available to you until you have performed these two operations in the correct order - you cannot select a tool if you haven't defined one.

The button at the extreme right (bottom) calls the post processor which will convert and output the program in the appropriate NC code for the chosen machine.

Miscellaneous Toolbar - Turning

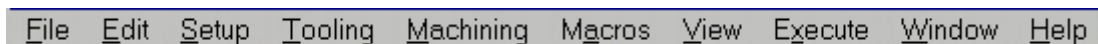


This Toolbar contains many secondary and optional operations necessary to define an NC program. .

NOTE: The actual output of many of the miscellaneous operations are completely dependent upon the post processor settings!

Turning Menu

The main turning menu consists of a series of 9 menus, which control all aspects of Dolphin CAM turning



FILE	This menu contains commands concerned with loading and saving programs, printing and page setup
EDIT	This menu contains commands to cut, copy and paste information and also enables you to undo unwanted operations or mistakes and to edit the postprocessor
SETUP	This menu enables you to set up the main machine parameters including the tool change position, safe position, tool fixture positions etc.
TOOLING	This menu contains all the option necessary to define and select tools for the machining sequences
MACHINING	This menu contains all the options necessary to define machining sequences - area clearance cycles for turning and facing, boring, profile turning, drilling cycles etc.
MACROS	This menu contains commands to control the generation and execution of Visual Basic Scripts (Macros)
VIEW	The view menu has two main functions: it allows you to alter which screen elements are displayed (toolbars etc.) and how the cursor appears, secondly it controls the display of geometry and toolpath simulation for the program that you have defined
EXECUTE	This menu contains the options available for simulating the program that you have defined.
WINDOW	This menu enables you to select which of the various frames are displayed in the program window.
HELP	Offers access to this help file and information about Dolphin CAM (Version No. etc:).

Graphic Display - Turning


This part of the screen is used to display the geometry which defines the workpiece, together with the raw material blank if defined. It is also used to simulate the cutter path when the program is executed.

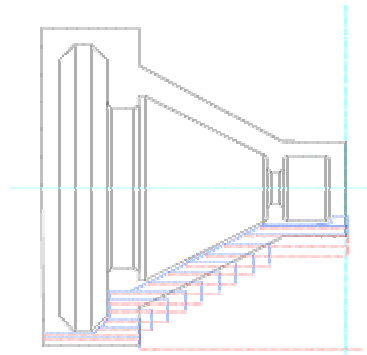
The display options can be accessed via the **Options** field on the **View** menu or via the  button on the main toolbar.

Toolpath Display


The toolpath and machining that you define can be displayed in three different ways as follows:

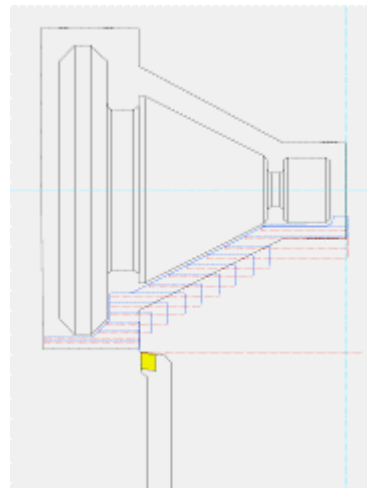
The toolpath is shown as the centre line only of the cutter tip radius. Cutting moves at feedrate are shown as blue lines, retract and approach moves at rapid speed are shown as dotted red lines.

The  button on the main toolbar is a shortcut for this option.




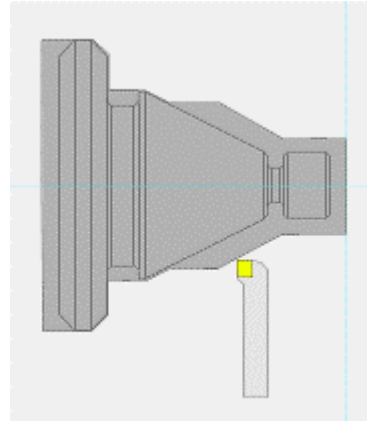
A full simulation of the cutter path with an animated cutter is shown. Note that full tool geometry can only be shown if the tool geometry is defined within a tool database. Cutting moves at feedrate are shown as blue lines, retract and approach moves at rapid speed are shown as dotted red lines.

The  button on the main toolbar is a shortcut for this option.




The toolpath is shown as full simulation of the cutter. The blank material is shown as a filled body which is removed as the cutter path is simulated. This option is the nearest to a full visual re-creation of the turning process.

The  button on the main toolbar is a shortcut for this option.



Geometry Display

In the turning module the view that can be displayed is limited to the plan view shown by the  icon on the top toolbar.

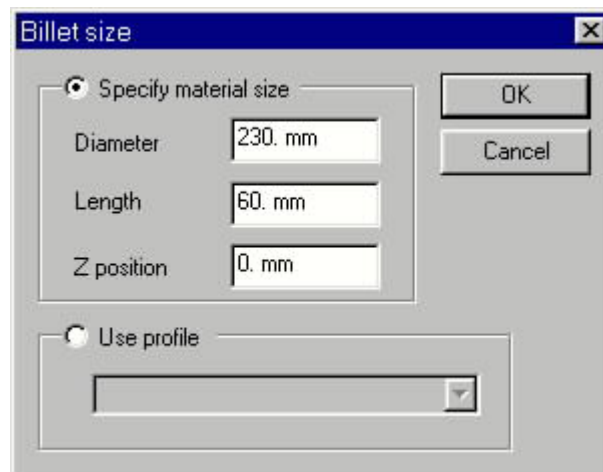
Sub-options are available as follows:

- 1 The geometry can be shown at the work surface only or with depth (if defined): meaningless within the Turning module, but one of the two must be chosen
- 2 The names of geometric entities can be displayed
- 3 The span numbers of elements within geometric entities can be displayed
- 4 The direction of spans within geometric entities can be displayed
- 5 Enhance Profiles: This option adds view lines across the profiles which produce a correct side view of the workpiece.

All of these options can be accessed via the **Options** field on the **View** menu or via the  button on the main toolbar.

Setup - Billet - Turning


This command enables you to define a billet (blank) for turning. The command is not available if the Machine Type is set to a milling machine.

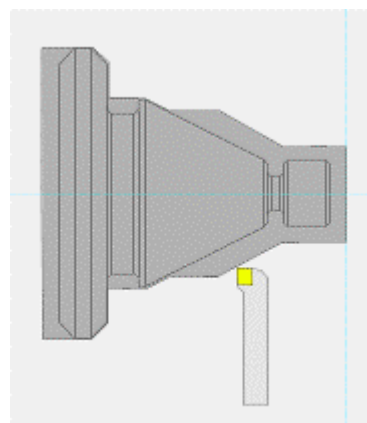



A non-cylindrical billet such as a casting can be defined by means of a profile which you can choose from within the list of defined geometric elements in the lower part of the dialog. A billet which is defined by a profile can be accessed in the Face/Turn dialog and used as an outer limit to the machining, thus reducing unwanted air cutting.

To view the Billet the Tool drawing option must be set to this option.

The toolpath is shown as full simulation of the cutter. The blank material is shown as a filled body which is removed as the cutter path is simulated. This option is the nearest to a full visual re-creation of the turning process.

The  button on the main toolbar is a shortcut for this option.



Having set the tool drawing option you must re-draw  the screen to see the Billet.

CREATING PROGRAMS

Creating programs within PartMaster CAM is a sequential process, below are listed the steps required.

- 1) Define the tool you wish to use and set the tool type, tip radius and angles, maximum cutting depth etc.
- 2) Select the tool for use - in effect invoke a tool change sequence, here we set the spindle speed, federate etc.
- 3) Choose the machining option required, this could be for instance facing off the bar, performing a roughing cut or cutting a groove.
- 4) View the tool graphics on screen and check that the operations you have defined are correct.
- 5) Invoke the post processor to convert the graphics you see on screen into machine tool language, most commonly Gcodes.
- 6) Send the program to the machine tool controller via the RS232 port, or perhaps your machine might have a floppy disk fitted.


The actual program to cut even the simplest of components might have many operations, for instance:-

- 1) Define a facing off tool, tool number 1
- 2) Select tool 1 and set the speeds/feeds
- 3) Use the Face/Turn macro to perform the operation
- 4) Define a roughing tool, tool number 2
- 5) Select tool 2 and set the speeds/feeds
- 6) Use the Face/Turn macro to rough out from the stock to the profile leaving a finishing allowance.
- 7) Define a finishing tool, tool number 3
- 8) Select tool 3 and set the speeds/feeds
- 9) Use the Profile turning operation to perform the finishing cut.
- 10) Define a parting off blade, tool number 4
- 11) Select tool 4 and set the speed/feeds.
- 12) Invoke the part catcher to catch the part before parting off.
- 13) Part off the component.

For some examples use File > Open and navigate to the Examples Folder.


EXECUTING PROGRAMS

Execute - Run Program

This command starts the execution of the program simulation. The  button on the Status Panel is a short cut for this command. When a program is executing the progress can be checked by looking at the Operations Window, the command being processed will have it's tick mark highlighted in blue. On simple programs this may happen so quickly that it cannot be seen.

Execute - Stop Program

The stop program buttons  on the top toolbar and on the status panel are shortcuts for this command.

Stops the execution of the program simulation. Note that this button is only available if a simulation is actually running. The  button on the Status Panel is the same as this command. The **Continue Program** command on the Execute menu can be used to continue execution of a program that has been stopped.

Execute - Single Step

This button sets the simulation to single step mode. Every time you press the Execute button, only the next operation in the sequence will be carried out. Note that a Tool Definition or a Tool Select command is counted as an operation.

The single step button  on the top toolbar is a shortcut for this command.

If you want to move the tool WITHIN AN OPERATION from point to point, select the **Execute - Pause Mode**

Execute - Pause Mode

This button sets the simulation to pause mode. Every time you press the Execute button on the Status Panel, the tool is moved to the next geometry position within the operation.



Press this button to start the pause mode simulation.




The button will turn RED and Flash, press the button to see the next tool movement. Only press this button once and wait for the tool get to it's new position before pressing again. You can't "store" hits.

Note: This is not the same as Single Step mode which, upon activation of the Execute button, will carry out a complete single operation. See Execute - Single Step.

To switch off Pause mode use the Execute menu item and untick Pause Mode, the system will revert to full run mode.

Execute - Slower

The slower button  on the top toolbar and the Simulation Speed control on the status panel are shortcuts for this command.

Selecting this command when a program simulation is running will have the effect of slowing down the simulation



You may also use this button on the Status Panel

Execute - Faster

The faster button  on the top toolbar and the Simulation Speed control on the status panel are shortcuts for this command.

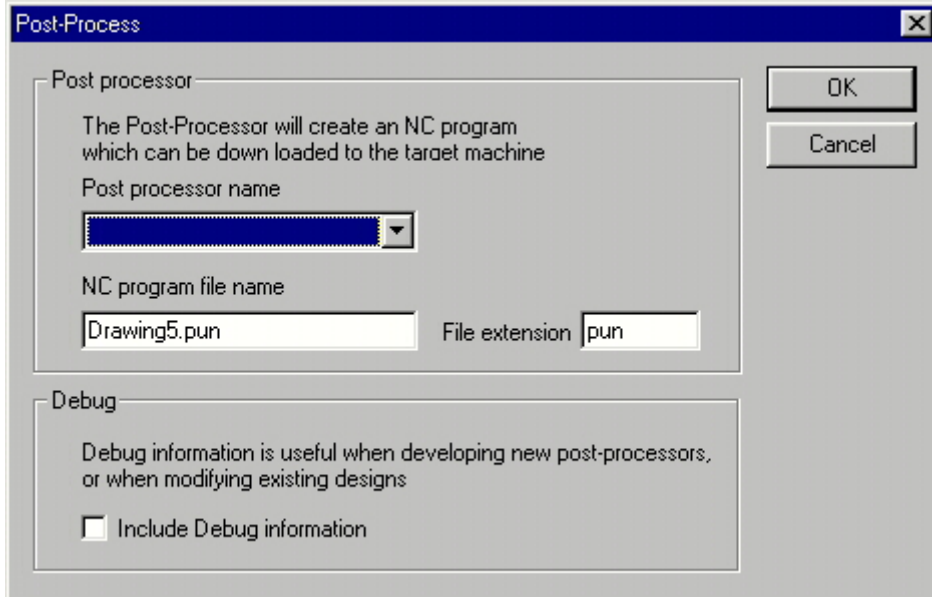
Selecting this command when a program simulation is running will have the effect of speeding up the simulation.



You may also use this button on the Status Panel

Execute - Post Process

The post process button  on the operations toolbar is a shortcut for this command.



The dialog box, titled "Post-Process", contains the following elements:

- Post processor** section:
 - Text: "The Post-Processor will create an NC program which can be down loaded to the target machine"
 - Label: "Post processor name" above a dropdown menu.
 - Label: "NC program file name" above a text input field containing "Drawing5.pun".
 - Label: "File extension" above a text input field containing "pun".
- Debug** section:
 - Text: "Debug information is useful when developing new post-processors, or when modifying existing designs"
 - Checkbox: "Include Debug information" (unchecked).
- Buttons: "OK" and "Cancel" on the right side.

When you have completed and debugged you program, you will then need to produce a file with the NC code necessary for the machine tool. This is accomplished by means of a post processor. The required post processor can be selected by means of the dialog shown above.

File Menu

<u>N</u> ew	Ctrl+N
<u>O</u> pen...	Ctrl+O
<u>S</u> ave	Ctrl+S
<u>C</u> lose	
Save <u>A</u> s...	
<u>I</u> mport	
<u>E</u> xport	
<u>P</u> rint..	Ctrl+P
Print <u>P</u> review	
Send NC program to ...	
<u>1</u> Area Clear.cnc	
<u>2</u> Area Clear with Taper.cnc	
<u>E</u> xit	

This menu contains commands to do with opening, closing, saving and printing files.

File - New

The  button on the Top Toolbar is a shortcut for this command

The File - New command opens a new program. If a program is already open you will be asked if you wish to save it before the new program is started.

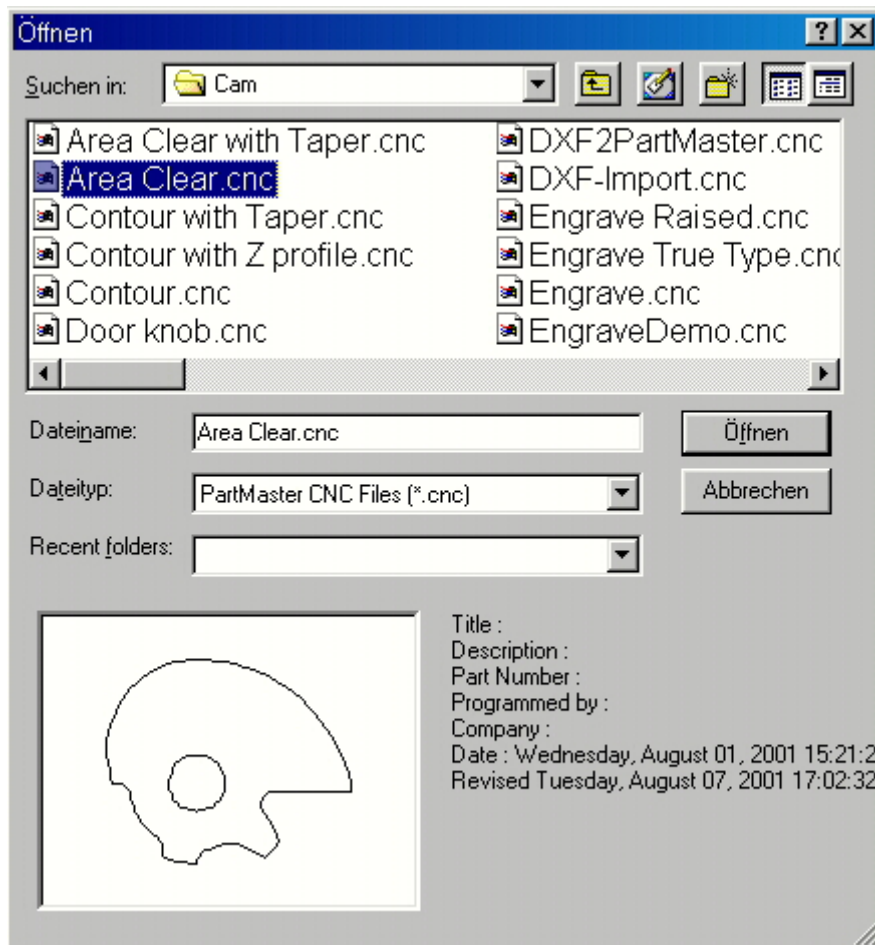
File - Open

The  button on the Top Toolbar is a shortcut for this command

The File - Open command is used to load an existing PartMaster program in a new window. There is no restriction on how many programs that you may load. The program that is the currently active program may be selected by means of the **Window Menu** or by clicking anywhere within the display if more than one program is displayed. How the various programs

are displayed is controlled by the options on the Window menu. A preview panel on the dialog will show the geometry of the currently selected file as a help in selecting the correct program.

The following dialog is displayed:

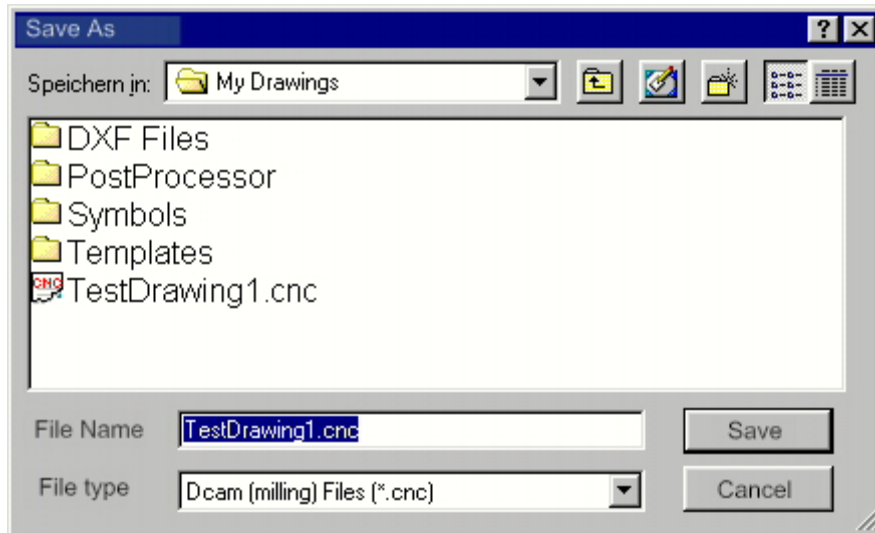


The dialog will automatically be set to the **My Documents\PartMaster\Drawings** folder. If the drawing that you wish to load is not in this folder, use the arrow button to open the complete directory, where you can select the folder that you want. Select the file name that you wish to open and then click on the Open button. Under the Windows XP operating system you will also see an extra toolbar at the left which contains entries such as My Recent Documents etc.

File - Save

The  button on the Top Toolbar is a shortcut for this command

The File - Save command saves the current program to the name and directory previously assigned without further comment. If the program has not been saved previously, the File - Save command is equivalent to the **File - Save As** command which will ask you to choose a folder and enter a name for the drawing. See File - Save As



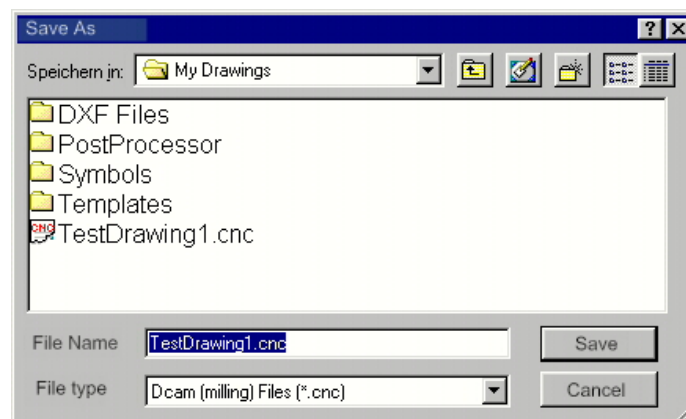
File - Close

The **File - Close** command closes the currently selected program file. If the program has not been saved previously or it has been altered since it was opened, you will be asked if you wish to save the file before it is closed. Note that this operation does **NOT** exit Partmaster - the program will remain open and active. If any other programs are currently loaded, the next one on the list will become the active program.

File - Save As

The File - Save As command is used to save the current drawing to a name or folder different to that which is currently assigned.

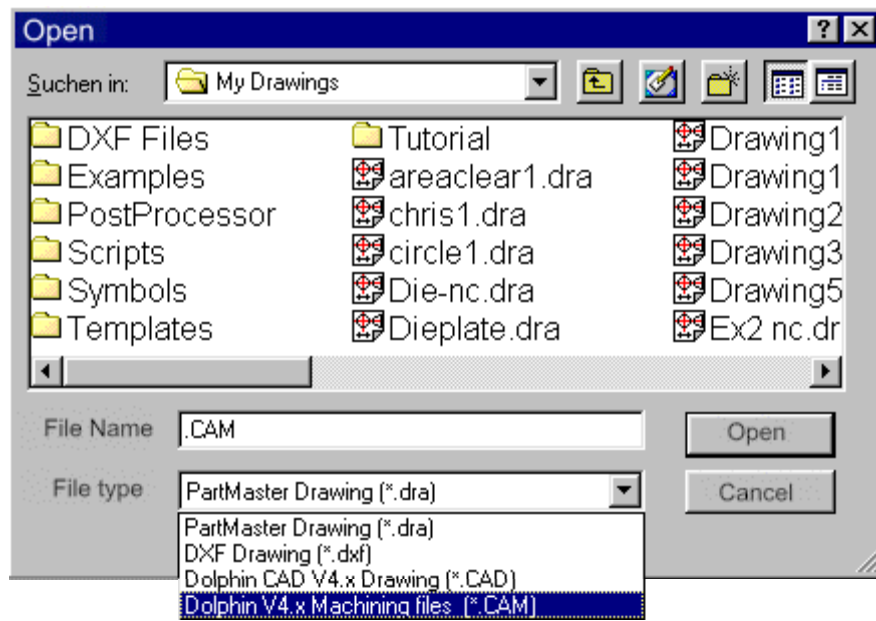
The following dialog will be displayed:



File - Import

The File - Import command is used to load an existing program or drawing. The file types currently available are PartMaster drawings, Dolphin CAD version 4 drawings, DXF drawings and machining programs written in Dolphin CAM version 4.

The following dialog is displayed:



The dialog will automatically be set to the **My Drawings** folder. If the file that you wish to load is not in this folder, use the mouse to open the complete directory, where you can select the folder that you want. Select the file name that you wish to open and then click on the **Open** button.

DXF IMPORT - When you import a DXF file, the elements within the drawing will be handled in the following manner:

- The drawing is scanned for POLYLINE features which have been imported as "Contours" on the layer "PartMaster", if none are found, the system attempts to create "Contours" from the available geometry. This means that if the drawing has been prepared beforehand by using POLYLINE's to identify the geometry to be imported, then the system will import only the prepared geometry, otherwise it will attempt to import all available geometry. (Maybe including the drawing frame and other undesired features).
- It is useful to note that if the export software that produced the DXF file has the option to select **Version 12 DXF** then this is the recommended choice as this option will export any NURBS as basic elements (lines, arcs etc) which can be handled within Dolphin CAM for machining.
- All suitable geometry is imported and assigned names based on the drawing name, e.g. Drawing1_CURVE001, Drawing1_CURVE002 etc. This allows geometry to be imported from multiple drawings without creating a naming conflict.

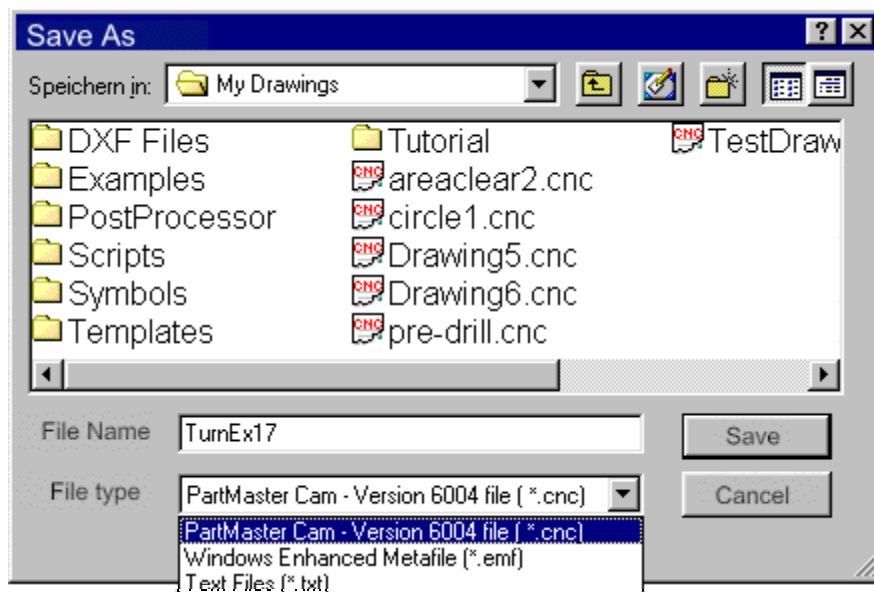
If complex drawings (containing text, help lines, frames etc.) are imported with no attempt to simplify the geometry held within them, the import routine may well find that the geometry cannot be resolved in a satisfactory manner. In these cases the

recommended procedure is to open the DXF file within the Dolphin CAD system and simplify the drawing as required.

File - Export

The File - Export command is used to save the current program in a range of external formats (other than PartMaster). The file types currently available are PartMaster CAM files version 6004, Windows enhanced Metafile, and Text Files

The following dialog is displayed:

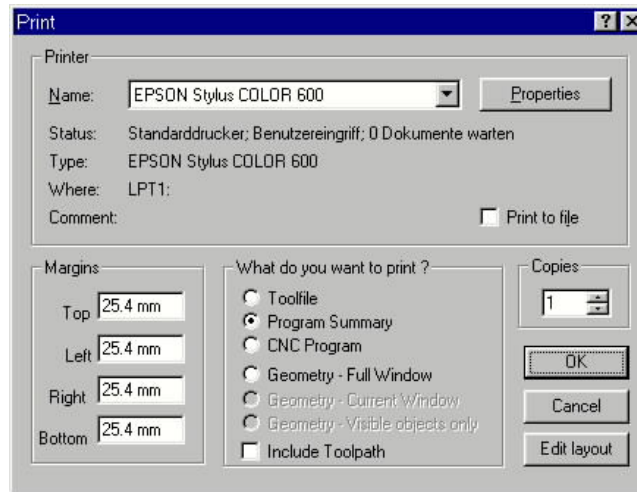


The dialog will automatically be set to the **My Drawings** folder. If this is not the folder where you wish to save the data, use the mouse to open the complete directory, where you wish to save the data. Input the file name that you wish to use and then click on the Save button.

File - Print

The  button on the Top Toolbar is a shortcut for this command

The File - Print command sends the selected output to the printer. A dialog will be opened where you can select the printer to be used (from the installed printers), the print parameters to be used and also what you want to send to the printer.

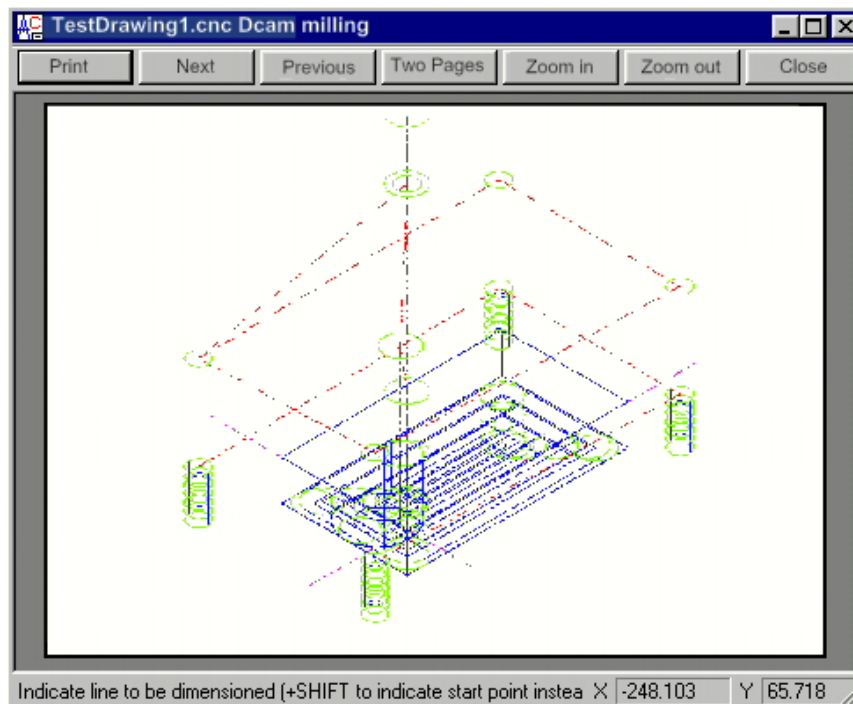


What do you want to print: With this field you can select what you wish to print out

Edit Layout: This button enables you to set exactly what will printed as part of the document that you have selected.

File - Print preview

The Print Preview command opens a new window which will display how the program will look when it is printed with the current printer settings. The display will change according to the data that you have selected to print in the **Print** command.



When you wish to revert to the normal Dolphin CAM window, click on the Close button.

File - Send NC Program to...

The File - Send NC Program to command is used to send the current NC program (after it has been post processed) to a machine tool controller. The target is set to that specified in the selected cfg program. The cfg (configuration program) is set within the DComms module and specifies the target port (COM1, COM2 etc.), the baud rate, data bits, stops bits and handshaking protocol to be used in the data transfer.

The following dialog is displayed:



The dialog will automatically be set to the **Applications Data** folder. If the drawing that you wish to load is not in this folder, use the arrow button to open the complete directory, where you can select the folder that you want. Select the configuration program that you wish to use and then click on the Open button.

For further information please see the PartMaster Communications (Dcomms) program and Help files.

File - Send as Mail

The File - Send as Mail option is used to send the current CAM file *.cnc to your resident Email program such as Outlook or Outlook Express.

Setup Menu - Turning

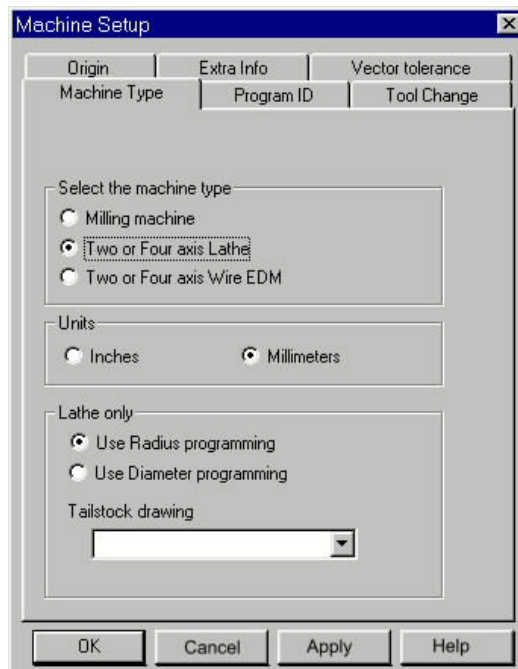


This menu contains commands which enable you to define the machine type (mill, lathe, edm), to define any fixture offsets (milling & edm only), and to define a billet (raw material block). You can also save these definitions and reload a stored setup if required.

Setup - Machine Type - Turning

The  button on the **Operations Toolbar** is a shortcut for this command.

Setup is the first command which must be carried out when beginning a new program - when you enter PartMaster or when you activate the File - New command most other menu commands and buttons will be disabled. The Setup dialog is used to set up the basic parameters which will apply to the new program. The Setup dialog will be displayed:



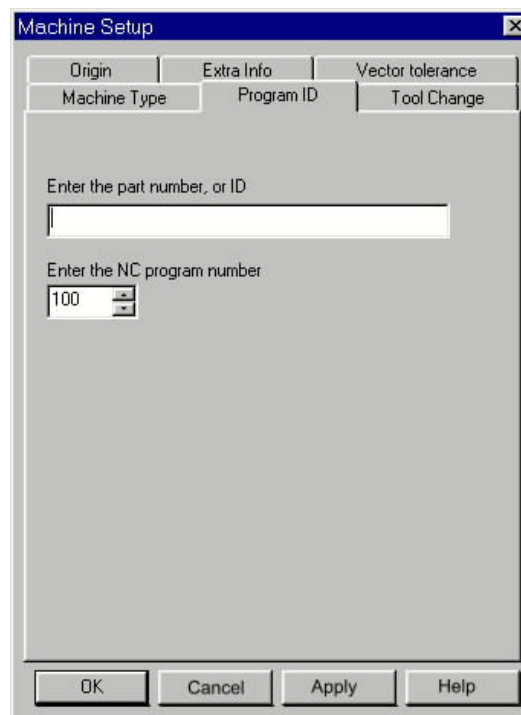
As you can see the dialog has 6 folders, the first one which will be displayed automatically is the one headed **Machine Type**. In this page you must select the type of machine to be used and the units that will be used within the program (Millimetres will be pre selected).

If you select a **Two or Four Axis Lathe** as the machine type then the bottom part of the dialog will have a panel where you must select either Radius or Diameter programming, together with the name of the drawing which defines the tailstock (if required).

NOTE ! The Radius and Diameter programming mode controls only how data entered within the CAM is expressed, it does NOT control what to output to the machine tool controller. This is handled by the Post-processor

If you enter values and the click on OK button the dialog will be closed. If you wish to enter values on various pages and activate them whilst remaining within the dialog, use the APPLY button.

Setup - Machine Type - Program ID



In this page you can enter a part number or ID of the program in order to identify the job more easily. This name or number can be output with the NC code as a comment if the post processor is set up to do this.

The NC program number is the program number that will be output at the start of the NC code.


Neither of these names or numbers are connected with the file name under which the program will be stored on the hard disk.

Note: If you enter values and the click on OK button the dialog will be closed. If you wish to enter values on various pages and activate them whilst remaining within the dialog, use the APPLY button.

Setup - Machine Type Turning - Tool Change

The screenshot shows a 'Machine Setup' dialog box with the 'Tool Change' tab selected. The dialog contains several input fields and checkboxes. The 'Rear Turret Tool change / Safe (Home) position' field has 'Z' set to '100. mm' and 'X' set to '100. mm'. The 'Front Turret Tool change / Safe (Home) position' field also has 'Z' set to '100. mm' and 'X' set to '100. mm'. The 'Spindle speed' field is labeled 'Specify the maximum spindle speed for this machine. (Limit used when turning at constant surface speed)' and has a value of '3500. rpm'. The 'Clearance distance in canned cycles' field is labeled 'The Z height at which the tool switches from Rapid traverse to the programmed Feedrate' and has a value of '1. mm'. At the bottom, there are two unchecked checkboxes: 'Stop spindle before tool change' and 'Switch off coolant before tool change'. The dialog has buttons for 'OK', 'Cancel', 'Apply', and 'Help'.

Tool Change Position/Safe (Home Position)

This field defines the position to which the machine will go in order to carry out a manual tool change. If the machine has an auto tool changer then this position will probably be ignored by the postprocessor **BUT** it is still sensible to enter figures that are approximately correct because they will be used in the graphic display. The Safe (Home) Position defines a position to which the tool can be sent when you wish to ensure that it is clear of the job. This function can be accessed via the **Goto** function 

Two sets of figures can be input, one for the rear turret and one for the front turret - fill in one or both of these depending upon the configuration of the lathe.

Spindle Speed

This field is used to specify a maximum spindle speed for the machine when you are programming using constant surface speeds. Otherwise, if the tool approaches the centre line of the job, an improbably high spindle speed may be attempted.

Clearance Distance for Canned Cycles

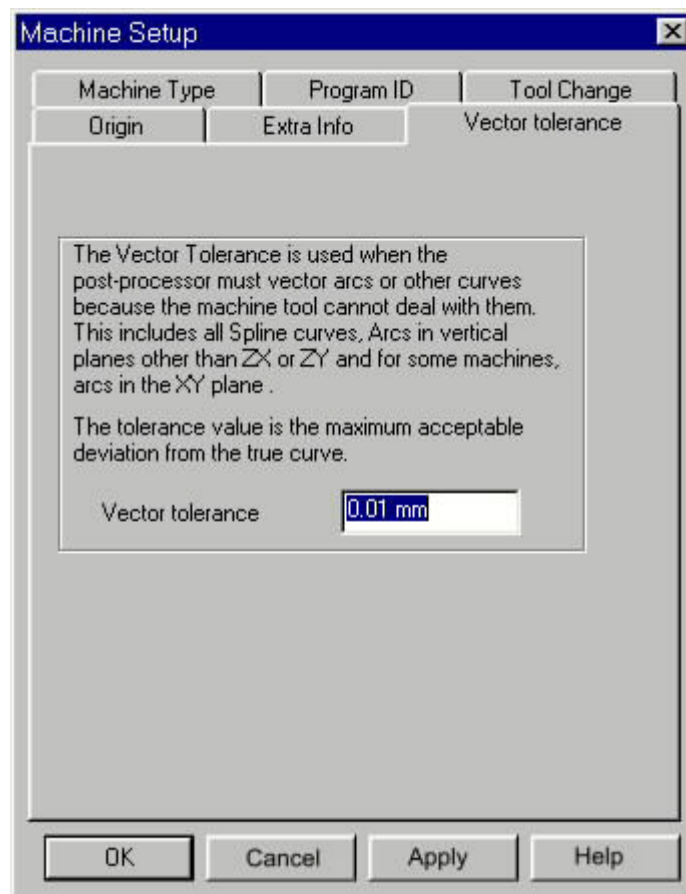
This field is used to set the distance away from the workpiece at which the feedrate will change from rapid to the programmed rate when the tool approaches the job at the start of a canned cycle.

Tick Boxes

The tick boxes at the bottom of the page allow you set a spindle stop and to switch the coolant off at every tool change. Whether these are necessary depends upon the machine tool controller. Many machines will automatically carry out these steps as soon as a tool change is commanded, others require a specific command.

Note: If you enter values and the click on OK button the dialog will be closed. If you wish to enter values on various pages and activate them whilst remaining within the dialog, use the APPLY button.

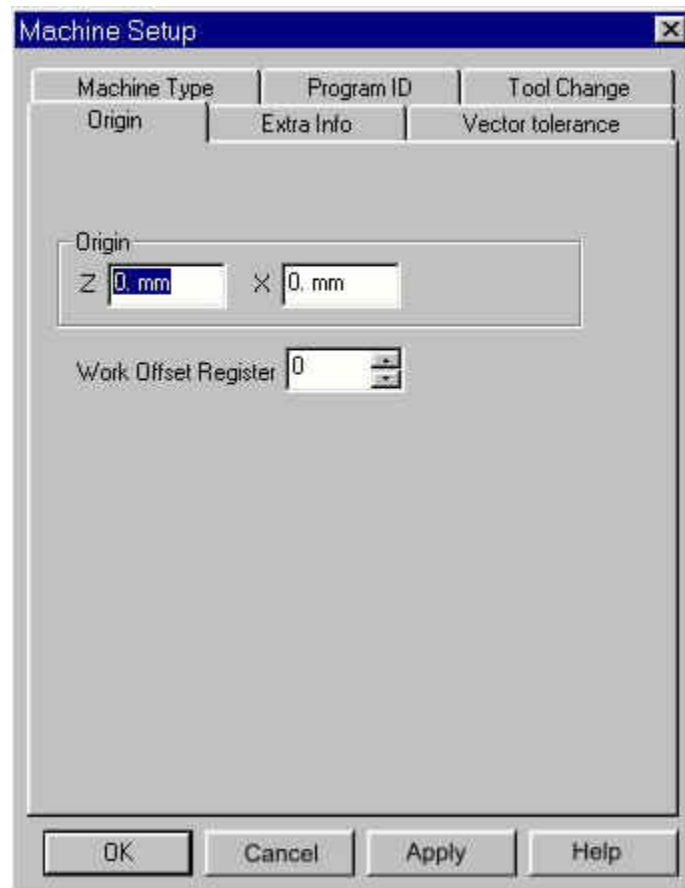
Setup - Machine Type - Vector Tolerance



Vector Tolerance: The Vector Tolerance is used by PartMaster to turn arcs and other curves such as splines, which the machine tool controller cannot handle, into a series of small linear moves approximating the shapes defined. The smaller the vectors the more nearly they approach the perfect form. Using an extremely small vector tolerance will produce a shape which requires little finishing but which may have many thousands of NC blocks. It is easily possible to exceed the size of a single program which is allowed by the NC controller - in this case it may be necessary to machine the form in DNC mode by drip feeding the controller from the computer. The Vector Tolerance sets the maximum value by which the linear moves (i.e. the vertices) may deviate from the perfect curve.

Note: If you enter values and the click on OK button the dialog will be closed. If you wish to enter values on various pages and activate them whilst remaining within the dialog, use the APPLY button.

Setup - Machine Type - Origin



The image shows a 'Machine Setup' dialog box with a blue title bar and a close button. It has three tabs: 'Machine Type', 'Program ID', and 'Tool Change'. The 'Machine Type' tab is selected and contains three sub-sections: 'Origin', 'Extra Info', and 'Vector tolerance'. The 'Origin' section has two input fields: 'Z' with a value of '0. mm' and 'X' with a value of '0. mm'. The 'Work Offset Register' section has a dropdown menu showing '0'. At the bottom are four buttons: 'OK', 'Cancel', 'Apply', and 'Help'.

Origin

The origin is used to define the machine tool zero point relative to the geometry origin and will typically appear in a G92 block. For lathes the only sensible input here is an X value which will have the effect of moving the machining along the X-axis.

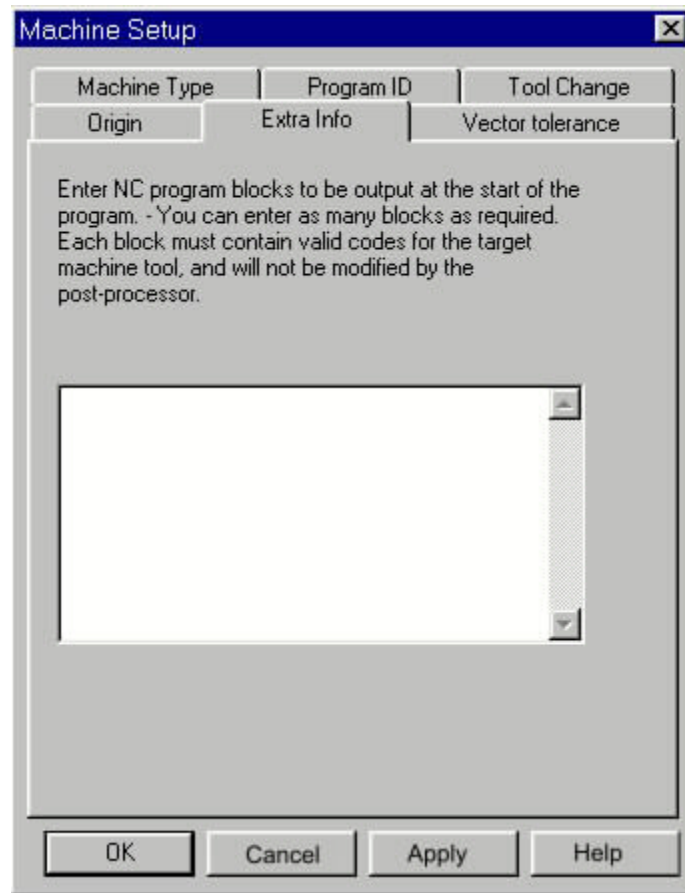
Work Offset Register

The work offset register is typically used by the post processor to create a G54, G55..... block. Offset register 1 = G54, 2 = G55 etc. The exact output depends upon the post processor settings. This function is normally used on milling machines where several workpieces may be mounted at different positions on the machine table.

Note: If you enter values and the click on OK button the dialog will be closed. If you wish to enter values on various pages and activate them whilst remaining within the dialog, use the APPLY button.

This command is primarily for Milling.

Setup - Machine Type - Extra Info



Extra Info

The Extra Info command allows you to define NC blocks which are output at the start of the program after the program number but before any machining instructions. Note that these entries will NOT be acted upon by the post processor so they must contain valid NC code for the controller in question.

Note: If you enter values and the click on OK button the dialog will be closed. If you wish to enter values on various pages and activate them whilst remaining within the dialog, use the APPLY button.

Setup - Fixture Offsets - Turning

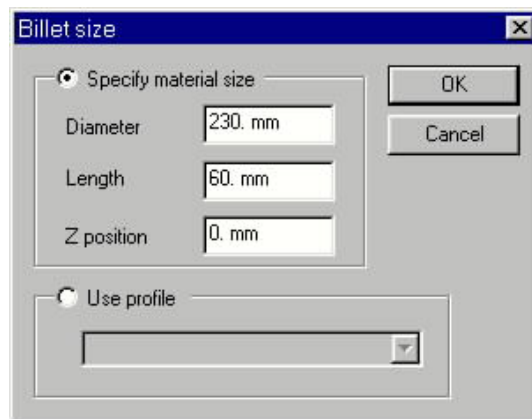
NOTE: This command is meaningless within the Turning module and applies only to milling operations.

Setup - Reset - Turning

This command enables you to selectively delete parts of an existing program. Upon choosing this command you will be asked consecutively if you wish to delete all the tools from the program, all the machining commands from the program or all the geometry from the program

Setup - Billet - Turning


This command enables you to define a billet (blank) for turning. The command is not available if the Machine Type is set to a milling machine.

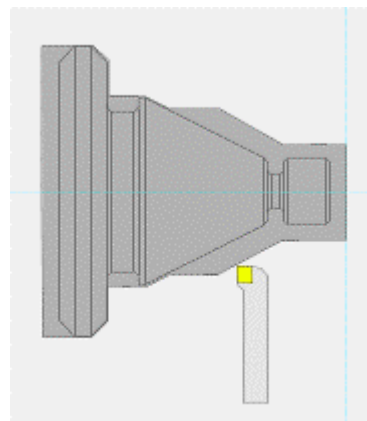



A non-cylindrical billet such as a casting can be defined by means of a profile which you can choose from within the list of defined geometric elements in the lower part of the dialog. A billet which is defined by a profile can be accessed in the Face/Turn dialog and used as an outer limit to the machining, thus reducing unwanted air cutting.

To view the Billet the Tool drawing option must be set to this option.

The toolpath is shown as full simulation of the cutter. The blank material is shown as a filled body which is removed as the cutter path is simulated. This option is the nearest to a full visual re-creation of the turning process.

The  button on the main toolbar is a shortcut for this option.



Having set the tool drawing option you must re-draw  the screen to see the Billet.

Setup - Load Setup - Turning

This command enables you to re-load a set of Setup parameters previously stored with the **Setup - Save Setup** command. The standard File Open dialog is displayed with the default folder set to **Application Data** and the file type set to ***.mcn** (machine tool information file).

Setup - Save Setup - Turning

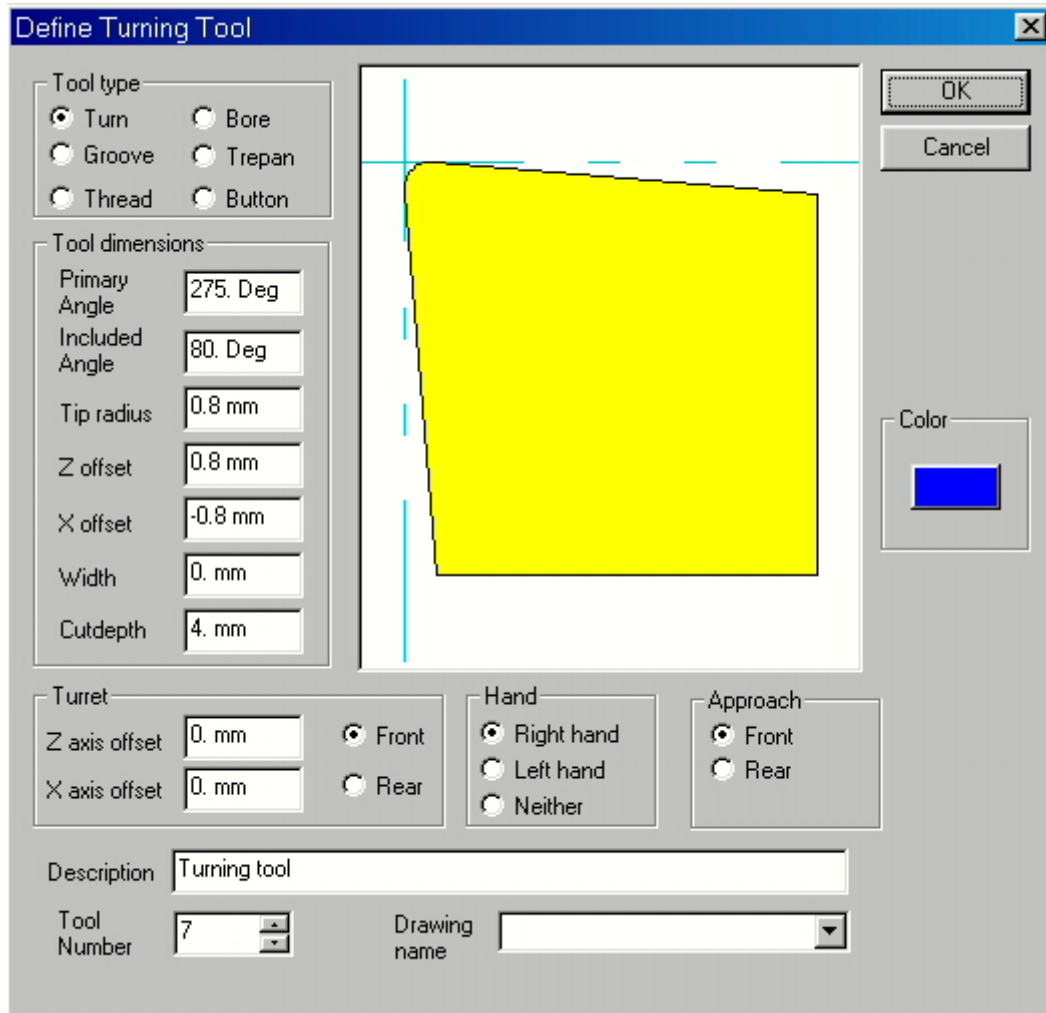
This command enables you to save a set of Setup parameters previously defined with the commands on the Setup menu. In addition to the normal setup parameters, this command will also save all the cutters that have been defined within the program.


The standard File Save dialog is displayed with the default folder set to **Application Data** and the file type set to ***.mcn** (machine tool information file).

TOOLING

Tooling - Define Turning Tool

The  button on the Operations Toolbar is a shortcut for this command



The **Define Tool** dialog allows you to define turning tools of six different kinds as shown above. **Drilling tools** (drill, reamer, tap, End mill etc.) are defined in a separate dialog which is accessed via the  button on **Operations Toolbar**. See **Define Drill Tool**

Tool Dimensions: The fields that are applicable here depend upon the type of tool that is chosen.

The **Cut Depth** field specifies the actual cutting depth which the cutter is capable of. If no **Cutdepth** is entered into any of the machining operations which use this tool, the cutting depth will default to this value. The **Cutdepth** value may be overwritten in the machining dialogs.

Turret: The Turret may be either Front or Rear mounted. The Z and X-axis offsets are used for those NC machines which require co-ordinate data referred to some datum other than the tool datum, e.g. a turret datum point. In this case the Z and X tool offsets must be entered into the Define Tool dialogue. The tool offsets are the distances in Z and X from the tool datum to the turret datum. For lathes with two turrets mounted on a single cross slide, this feature can be used to refer all tools on both turrets to a single datum point. On lathes that expect the tool

datum co-ordinates together with Z and X tool offsets, the offset registers are programmed in the **Tooling - Select Tool** dialogue. See **Tooling - Select Tool**

Hand: This defines in which direction the cutter approaches the job, i.e. they cut from **Right** to left or from **Left** to right along the Z axis. **Neither** means that it is irrelevant and applies to symmetrical tools such as thread cutters.

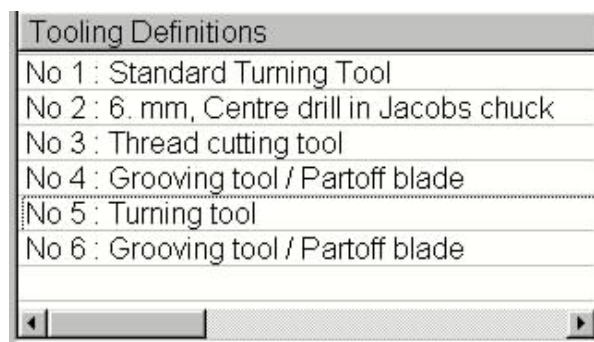
Approach: Defines if the tool approaches the job from the front or the rear - it has NO connection with the position of the turret - a tool mounted on a rear turret may nevertheless approach from the front.

The **Description** field will contain a short description of the tool type chosen. You can edit or delete this description and write in your own description as desired.

The **Tool Number** field defines the number of the tool in the program. Tools do not have to be defined or selected in any particular order. The length and offset register numbers are set in the **Select Tool** dialog. See **Select Tool**

The **Drawing Name** field allows you to access a PartMaster CAD drawing which defines the tool geometry. This is necessary if you wish the on-screen simulation to be done with a full drawing of the tool (including shank etc.), otherwise only the tool tip geometry as defined in the **Tool Dimensions** fields can be shown. **Click here for further details about preparing a drawing which defines the tool geometry** The drawings are stored in a sub-folder of the PartMaster System called **Application Data**.

Tooling - Definitions



As each tool is defined it is added to the bottom of the list in the **Tooling Definitions** Pane. You can re-set the size of this pane to the full window size by means of the **Tooling** command in the **Window Menu**. The **Tile** command in the same menu will set all the panes within the display window back to the default configuration.

Edit a Tool Definition

To edit an existing definition, double click on the entry with the left hand mouse button. The Dialog that defines the tool will be displayed showing all the values and parameters that you originally set. Make any necessary alterations and then click on OK. Note that the alterations will NOT be carried out until the program has been executed after the alteration.

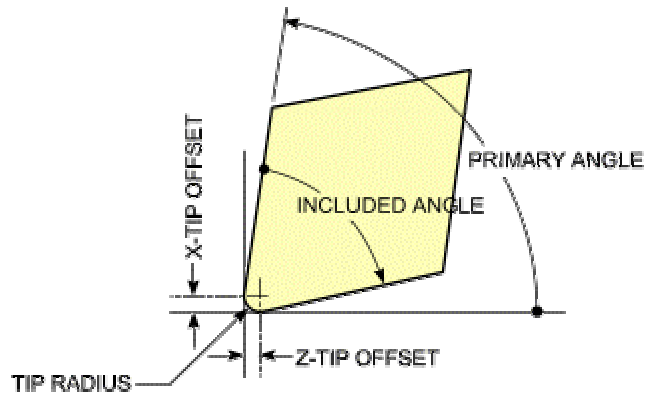
Delete Tool Definition

To delete an operation from the sequence simply click on the text part of the entry and hit the delete key on the keyboard or select **Delete** from the **Edit Menu**.

See Also: Define Turning Tool & Define Milling Tool

Bore Tool

A Bore tool is used for internal turning operations. The tool dimensions are as follows:

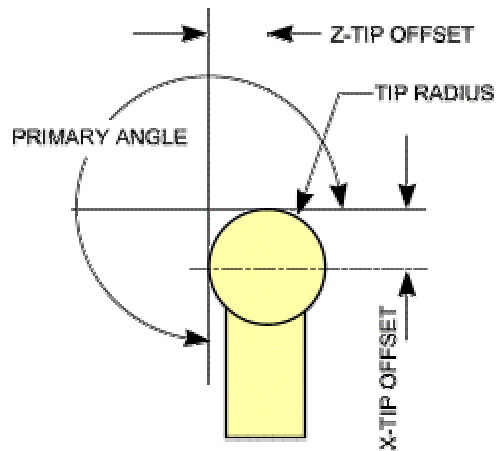


- PRIMARY ANGLE** Measured anti-clockwise from the Z-axis round to the primary cutting face
- INCLUDED ANGLE** Measured anti-clockwise from the primary to the secondary cutting face
- TIP RADIUS** The radius of the tool tip
- Z TIP OFFSET** The distance from Z zero to the centre of the tip radius
- X TIP OFFSET** The distance from X zero to the centre of the tip radius

NOTE: Regardless if the tool is mounted on a front or rear turret or approaches the job from left or right, the tool geometry is always defined in the quadrant shown above. PartMaster will make any necessary adjustments for other quadrants.

Button Tool

This option defines a tool which may be used for all Turn, Bore, Groove or Trepan operations



- PRIMARY ANGLE** Anti-clockwise from the Z-axis to the primary cutting face - normally 270 degrees

INCLUDED ANGLE Not-Applicable

TIP RADIUS The radius of the tool tip

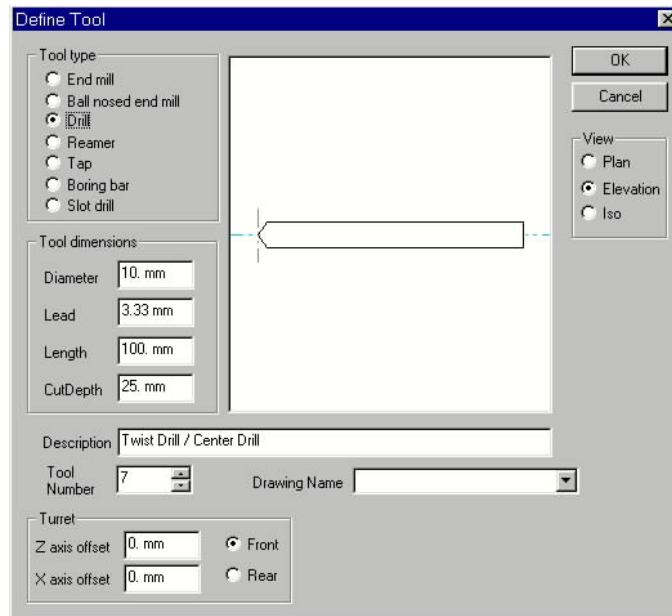
Z TIP OFFSET The distance from Z zero to the centre of the tip radius

X TIP OFFSET The distance from X zero to the centre of the tip radius

NOTE: Regardless if the tool is mounted on a front or rear turret or approaches the job from left or right, the tool geometry is always defined in the quadrant shown above. PartMaster will make any necessary adjustments for other quadrants.

Drill

The  button on the Operations Toolbar is a shortcut for this command



The **Define Drill Tool** dialog allows you to define turning tools of seven different kinds as shown above.

Tool Dimensions: The fields that are applicable here depend upon the type of tool that is chosen. All these tools are intended for axial operations along the machining centre line and thus the geometry is reduced to a diameter and length. The **Lead** is the distance from the point of a drill to the position where it reaches full diameter. This figure is automatically calculated and entered as one third of the diameter if the tool type is specified as **Drill**. All hole depths in drilling cycles will be adjusted by this amount unless the option is suppressed in the cycle definition dialog. **See Machining - Drill**

The **Cut Depth** field specifies the actual cutting depth which the cutter is capable of. For tools of this type it is intended for guidance only and will be ignored by the machining cycle.

The **Description** field will contain a short description of the tool type chosen. You can edit or delete this description and write in your own description as desired.

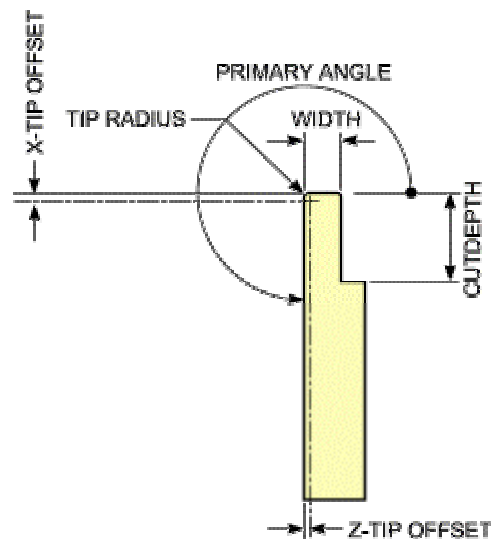
The **Tool Number** field defines the number of the tool in the program. Tools do not have to be defined or selected in any particular order. The length and offset register numbers are set in the **Tooling - Select Tool** dialog. See **Tooling - Select Tool**

The **Drawing Name** field allows you to access a PartMaster CAD drawing which defines the tool geometry. This is necessary if you wish the on-screen simulation to be done with a full drawing of the tool (including shank etc.), otherwise only the tool tip geometry as defined in the **Tool Dimensions** fields can be shown.

Turret: The Turret may be either Front or Rear mounted. The Z and X-axis offsets are used for those NC machines which require co-ordinate data referred to some datum other than the tool datum, e.g. a turret datum point. In this case the Z and X tool offsets must be entered into the Define Tool dialogue. The tool offsets are the distances in Z and X from the tool datum, to the turret datum. For lathes with two turrets mounted on a single cross slide, this feature can be used to refer all tools on both turrets to a single datum point. On lathes that expect the tool datum co-ordinates together with Z and X tool offsets, the offset registers are programmed in the **Tooling - Select Tool** dialogue. See **Tooling - Select Tool**

Groove Tool

A Groove tool is used for radial plunge operations on the outside diameter of the workpiece. The tool types that are defined with this definition include part-off tools as well as groove tools (with or without corner rads) and round button type tools. The tool dimensions are as follows:



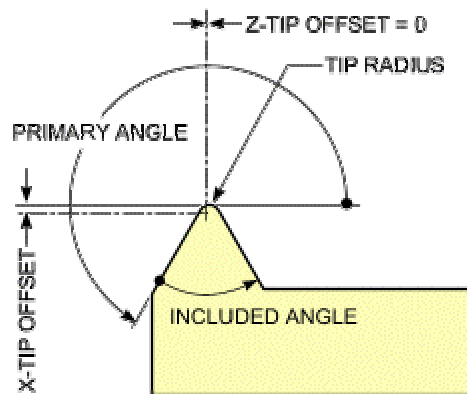
- PRIMARY ANGLE** Anti-clockwise from the Z-axis to the primary cutting face - usually 270 deg.
- INCLUDED ANGLE** Anti-clockwise from the primary to the secondary cutting face - usually 0 deg.
- TIP RADIUS** The radius of the tool tip if any
- Z TIP OFFSET** The distance from Z zero to the centre of the tip radius
- X TIP OFFSET** The distance from X zero to the centre of the tip radius

NOTE: If the **Cycle Type** on the machining dialog is set to **Plunge Cutting** the **Cutdepth** parameter is used only for the graphic display - it will not be checked by PartMaster

NOTE: Regardless if the tool is mounted on a front or rear turret or approaches the job from left or right, the tool geometry is always defined in the quadrant shown above. PartMaster will make any necessary adjustments for other quadrants.

Thread Tool

This option defines tools used for either internal or external threading operations. The tool dimensions are as follows:



PRIMARY ANGLE Anti-clockwise from the Z-axis to the primary cutting face

INCLUDED ANGLE Anti-clockwise from the primary to the secondary cutting face

TIP RADIUS The radius of the tool tip if any

Z TIP OFFSET The distance from Z zero to the centre of the tip radius, normally 0

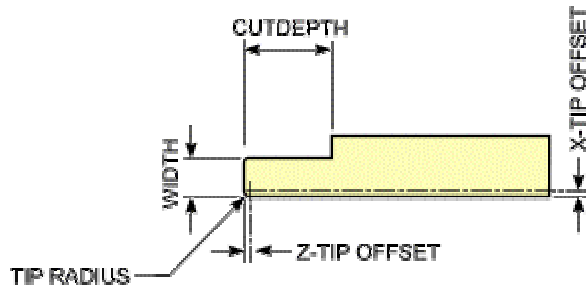
X TIP OFFSET The distance from X zero to the centre of the tip radius

NOTE: Because the tool is symmetrical about the X axis, the Z axis tip offset is zero. Either edge can be used as the primary edge, provided that the primary and secondary angles are consistent. Normally the primary edge is the one that is cutting when the tool move towards the chuck in the Z axis.

NOTE: Regardless if the tool is mounted on a front or rear turret or approaches the job from left or right, the tool geometry is always defined in the quadrant shown above. PartMaster will make any necessary adjustments for other quadrants.

Trepan Tool

A Trepan tool is used for axial plunge operations on the face of the workpiece. The tool types that are defined with this definition include trepan tools (with or without corner radi) and round button type tools. The tool dimensions are as follows:



PRIMARY ANGLE Anti-clockwise from the Z-axis to the primary cutting face - usually 0 deg.

INCLUDED ANGLE Anti-clockwise from the primary to the secondary cutting face - usually 0 deg.

TIP RADIUS The radius of the tool tip if any

Z TIP OFFSET The distance from Z zero to the centre of the tip radius

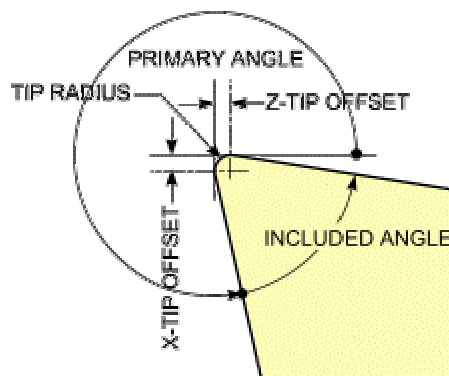
X TIP OFFSET The distance from X zero to the centre of the tip radius

NOTE: If the **Cycle Type** on the machining dialog is set to **Plunge Cutting** the **Cutdepth** parameter is used only for the graphic display - it will not be checked by PartMaster

NOTE: Regardless if the tool is mounted on a front or rear turret or approaches the job from left or right, the tool geometry is always defined in the quadrant shown above. PartMaster will make any necessary adjustments for other quadrants.

Turn Tool

A Turn tool is used for general facing and turning operations. The tool dimensions are as follows:



PRIMARY ANGLE Measured anti-clockwise from the Z-axis round to the primary cutting face

INCLUDED ANGLE Measured anti-clockwise from the primary to the secondary cutting face

TIP RADIUS The radius of the tool tip

Z TIP OFFSET The distance from Z zero to the centre of the tip radius

X TIP OFFSET The distance from X zero to the centre of the tip radius

NOTE: Regardless if the tool is mounted on a front or rear turret or approaches the job from left or right, the tool geometry is always defined in the quadrant shown above. PartMaster will make any necessary adjustments for other quadrants.

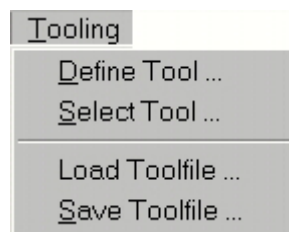
Tooling - Load Toolfile

This command allows you to re-load a toolfile which has previously been saved with the **Tooling - Save toolfile** command.

This is particularly useful for turning if you have tool turrets which are always loaded with a range of standard tools, or for milling when a tool changer or carousel has a range of standard cutters in pre-defined positions.

The **Open** dialog is opened with the file name extension ***.tlf** and the default folder set to **Application Data**.

Tooling Menu - Turning



This menu contains commands which enable you define and select tools

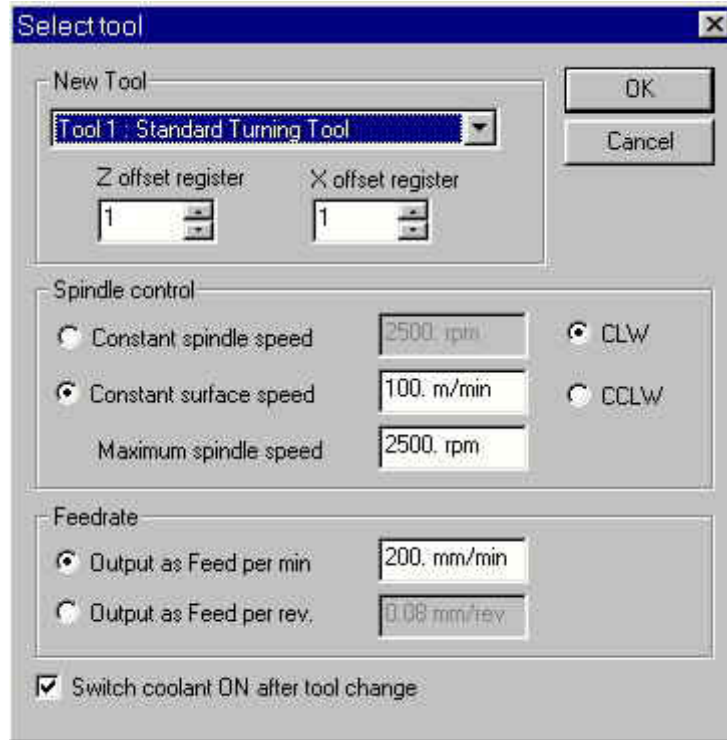
Tooling - Save Toolfile

All the tool definitions in the current program can be saved to a toolfile which can then be re-loaded into another program for future use. This is particularly useful for turning if you have tool turrets which are always loaded with a range of standard tools, or for milling when a tool changer or carousel has a range of standard cutters in pre-defined positions.

The **Save** dialog is opened with the file name extension ***.tlf** and the default folder set to **Application Data**

Tooling - Select Turning Tool

The  button on the Operations Toolbar is a shortcut for this command



The drop down list at the top of the dialog shows all the available tools which have been defined.

The **Z-Offset Register** and **X-Offset Register** fields refer to the offset registers of the NC machine, they may or may not be the same as the tool number.


Spindle Control: This part of the dialog controls the spindle speeds. You may set a **Constant Spindle Speed** (conventional) or you can set a **Constant Surface Speed**. If you set a Constant Surface Speed the **Maximum Spindle Speed** field allows you to set a maximum in case the machine tries to reach some dangerously high speed when the cutter approaches the centre line of the workpiece.

Spindle speed may also be set independently of a tool change by means of the menu field

Machining - Spindle or via the  button on the **Miscellaneous Toolbar**.

Feedrate: may be output as Feed per minute (conventional) or as a Feed per Revolution

Coolant may be switched ON automatically after the tool change. Note that you may switch the coolant on and off independently of a tool change by means of the menu field **Machining**

- **Coolant** or via the  button on the **Miscellaneous Toolbar**.

The **Select Tool** dialogue may also used to set a new spindle speed and direction, a new cutting feedrate and to switch coolant ON or OFF. A tool change sequence is initiated only if the tool number entered is different to the currently selected tool, otherwise only the new spindle speed, feedrate and coolant settings are sent to the post-processor.

MACHINING

Machining - Drill

The  button on the Operations Toolbar is a shortcut for this command

Although entered on the Menu as **Drill**, Canned Cycles include all the various types of drilling, reaming, boring and tapping cycles. Although they are called canned cycles the way that they are output to the NC machine is dependant upon the post processor setup. This means that it is possible to define a cycle which the NC machine does not have built in and have it output as a series of individual movement commands. It is however usual to use the built in canned cycles on the NC machine wherever possible in order to keep the program length to a minimum.

CYCLE TYPE	DESCRIPTION
DRILL	Feed to depth, retract at rapid. - Can be used with tools defined as either MILL or DRILL.
DEEP DRILL	Drill hole in a series of pecks, retracting tool completely between each peck.
PECK DRILL	Drill hole in a series of pecks, retracting tool slightly between each peck. The number of pecks is entered in the dialogue. PartMaster calculates the depth of each peck
REAM	Feed to depth, retract at feedrate. Can only be used with REAM type tools
BORE	Feed to depth, spindle stop, retract at feedrate, spindle start. Can only be used with DRILL tools.
TAP	Feed to depth, reverse spindle, retract at feedrate, reverse spindle. Can only be used with TAP type tools.

Each of the cycles can be modified according to which options are ticked in the dialogue box.

OPTION	DESCRIPTION
Dwell before retract	Use drill with dwell canned cycle, the dwell time must also be entered in the dialogue.
Do not reverse tap	For Tap cycle only: Do not reverse tap when retracting - usually because the Tap holder reverses automatically.
Rapid Infeed for Deep Drill	If a deep drill cycle is selected, ticking this options causes the tool to descend at rapid feedrate, to a point just above the previous depth drilled, after retracting to clear swarf from the hole being drilled.
Ignore tool's lead value	The distance from the point of a drill to the point where it reaches full diameter is called the lead (see Tooling - Define Drill). Any drill depth is normally adjusted so that the hole depth applies at the full diameter not the point of the drill. If this option is ticked, this calculation is ignored and the tip of the drill is sunk to the defined depth.

The **Pattern Name** field is greyed out for Turning as all these operations are performed on the workpiece centreline.

Z-PLANES	DESCRIPTION
FEED CHANGE PLANE	Defines a position above the Work Surface to which the tool moves at Rapid Speed before changing to the programmed feedrate for the cycle.
WORK SURFACE	Defines the Z axis position at which the cycle starts relative to the workpiece zero
DEPTH OF HOLES	Defines the depth of the cycle as measured from the Work Surface . Depths are always positive into the job.

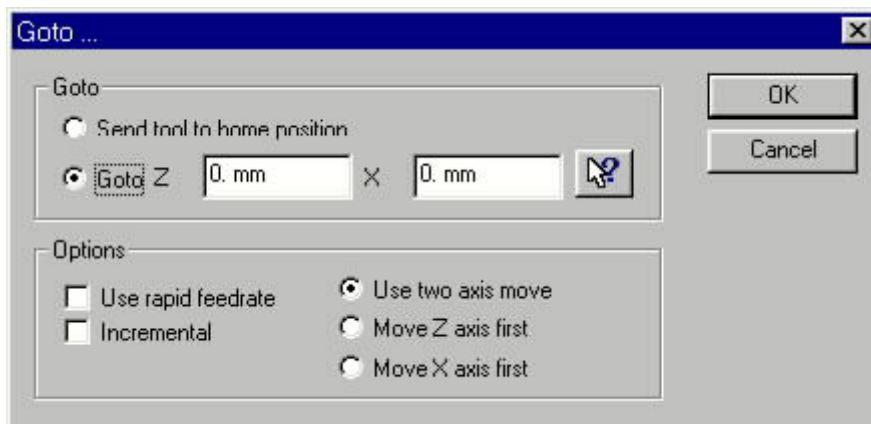
WORK SURFACE = 0
DEPTH = 20

WORK SURFACE = 15
DEPTH = 20

Machining - Goto

The  button on the Operations Toolbar is a shortcut for this command


Goto is the only command (apart from Digitise) provided for driving the tool directly. All other commands drive the tool with respect to some piece of pre-defined geometry.



Goto

Send Tool to Home Position: This command sends the tool to the Safe (Home) position as defined on the **Tool change** Page of the **Setup** dialog.

See **Setup - Machine Type - Tool change**

Goto: Sends the tool to a specific XZ co-ordinate. A click on the  button enables you to use the cursor to pick a position on the screen directly without having to know the exact co-ordinates. The dialog will be temporarily removed and the cursor will be placed in the display screen. Move the cursor to the required position and click with the left hand mouse button. The dialog will re-appear with the co-ordinates of the indicated position entered in the appropriate fields.

Options:

The Options tick boxes allow you to alter the way that the move is executed. The Goto move may be made at rapid feedrate if the **Rapid Feedrate** box is activated, otherwise it will be made at the current feedrate. If the **Incremental** tick box is activated the XZ co-ordinates will be taken as an incremental move from the current position. The other three options specify how the move is to be made - in two axes simultaneously or Z followed by X or X followed by Z. These options can be important in moving the tool clear from the job before the final position is reached.

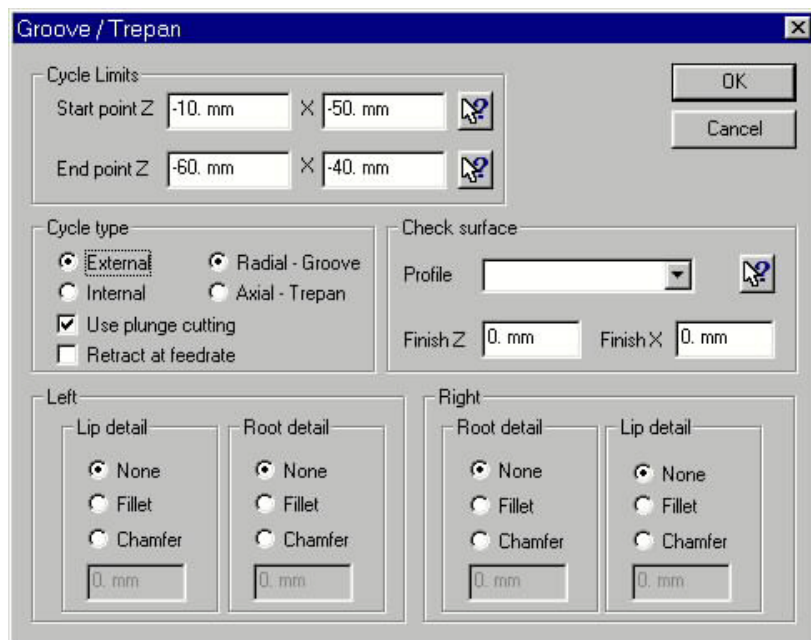


Caution is needed with the Goto command as the move will NOT be checked for collisions!

Machining - Groove/Trepan


The  and the  buttons on the Operations Toolbar are shortcuts for this command


This command enables you to define a cycle to machine grooves in the diameter of the job (**Groove**) or in the face of the job (**Trepan**). The basic cycle which is defined by the co-ordinates of two opposite corners can be modified so that a profile (usually the finished part) is used as a limit to the machining. The **Cutdepth** specifies the in-feed between successive passes of the cutter. The program will default to the **Cutdepth** value specified in the tooling Definition. See **Tooling - Tool Definition**



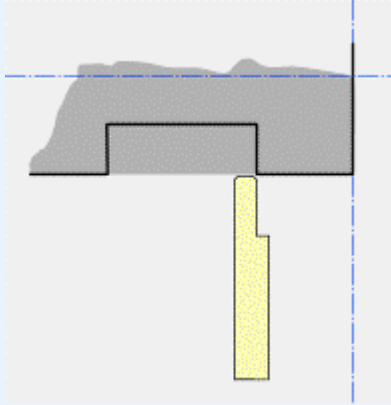
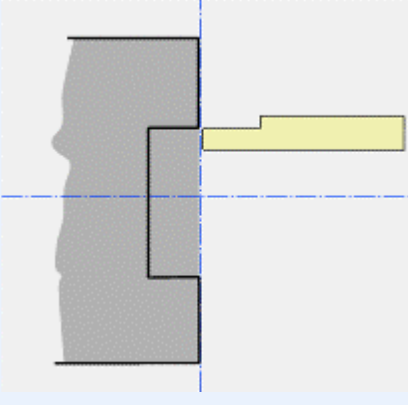
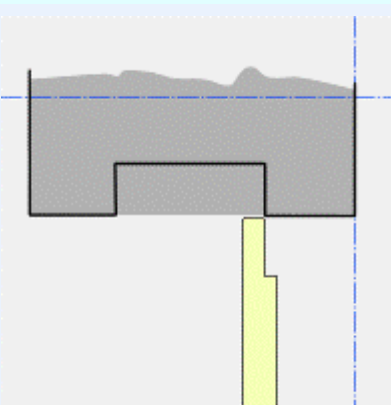
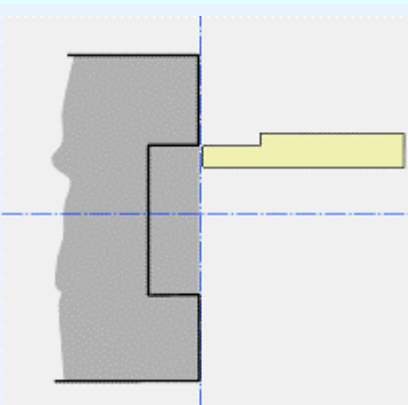
CYCLE LIMITS

These fields enable you to define two diagonally opposite corners of the area to be covered by the cycle. The two corners may be given in any order - top right/bottom left - bottom right/top left - top left/bottom right - bottom left/top right. If the cycle is not modified in any other way a simple area clearance will be performed between these limits. The extent of the cycle may be modified by a limiting (protected) profile - normally that of the finished part. If the groove is defined by simple ZX co-ordinates the bottom part of the dialog can be used to define lip or root fillets or radii.

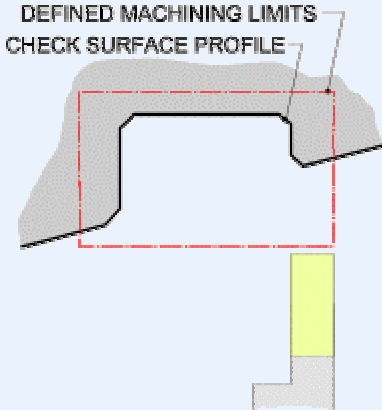

The  button enables you to use the cursor to pick a position on the screen directly without having to know the exact co-ordinates. If you click on the button, the dialog will be temporarily removed and the cursor will be placed in the display screen. Move the cursor to the required position and click with the left hand mouse button. The dialog will re-appear with the co-ordinates of

the indicated position entered in the appropriate fields. If you hold down the **Shift** key when you click on the screen position, the cursor will change to a  and PartMaster will pick up the exact co-ordinates of the nearest geometry point, either of the blank or the part profile

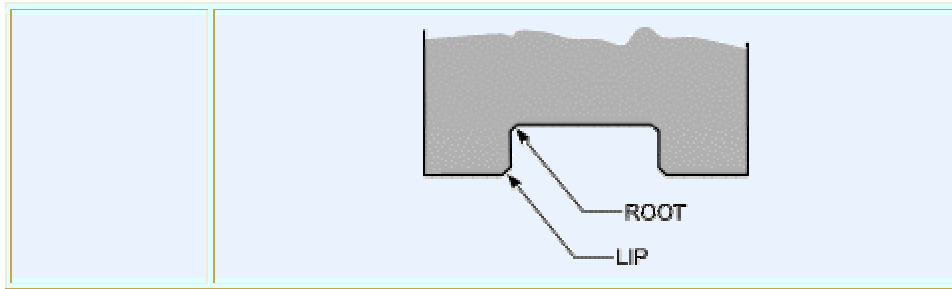
The cycle can be modified according to which options are ticked in the dialogue box.

CYCLE TYPE	DESCRIPTION	
External	The cycle will be carried out on the external diameter of the workpiece. i.e. the feed will be from outside towards the centre line.	
Internal	The cycle will be carried out on an internal diameter of the workpiece. i.e. the feed will be from the centre line towards the outside.	
Radial - Groove Axial - Trepan	<p>The cycle will consist of a series of cuts along the Z-axis with the feed in direction in the X-axis</p>  <p style="text-align: center;">GROOVE</p>	<p>The cycle will consist of a series of cuts along the X-axis with the feed in direction in the Z-axis.</p>  <p style="text-align: center;">TREPAN</p>
Use Plunge Cutting	<p>If this option is chosen the cutting cycle is changed so that it consists of a series of plunge cutting moves rather than a series of axis parallel cutting moves.</p>  <p style="text-align: center;">GROOVE - PLUNGE</p>  <p style="text-align: center;">TREPAN - PLUNGE</p>	
Retract at	At the end of each cutting pass the tool is retracted and moved to the	

Feedrate	start of the next pass. This retract move is normally made at rapid feedrate but if you select this option it will be made at the programmed cutting feedrate instead.
-----------------	--

<p>CHECK SURFACE PROFILE</p>	<p>CHECK SURFACE PROFILE: A profile name entered here will be used as a limiting (protected) profile to amend the machining as defined by the simple XZ co-ordinates (see above). Any profile may be chosen as a check surface and this means that a groove or trepan cycle does not have to be applied to a regular rectangular form - any shape can be machined provided that the cutter tip geometry is suitable for the job. The area will first be semi-roughed out and then finished with a bi-directional profiling operation.</p>  <p>The  button enables you to use the cursor to pick either of the above profiles directly from the screen. If you click on the button, the dialog will be temporarily removed and the cursor will be placed in the display screen. Move the cursor over the required profile and click with the left hand mouse button. The dialog will re-appear with the name of the profile entered in the appropriate field.</p>
<p>FINISH ALLOWANCE</p>	<p>The Finish fields allow you to define a different finishing allowance for both the Z and X axes. This is the amount of material that will be left on the part profile for finishing operations.</p>

<p>LEFT - RIGHT LIP OR ROOT FILLET OR CHAMFER</p>	<p>If you have defined the groove or trepanning cycle with simple ZX co-ordinates with no reference to a check surface profile, these fields allow you to define a fillet radius or chamfer at the lip or root of the detail at either or both the right and left hand side.</p>
--	--



Machining Menu - Turning



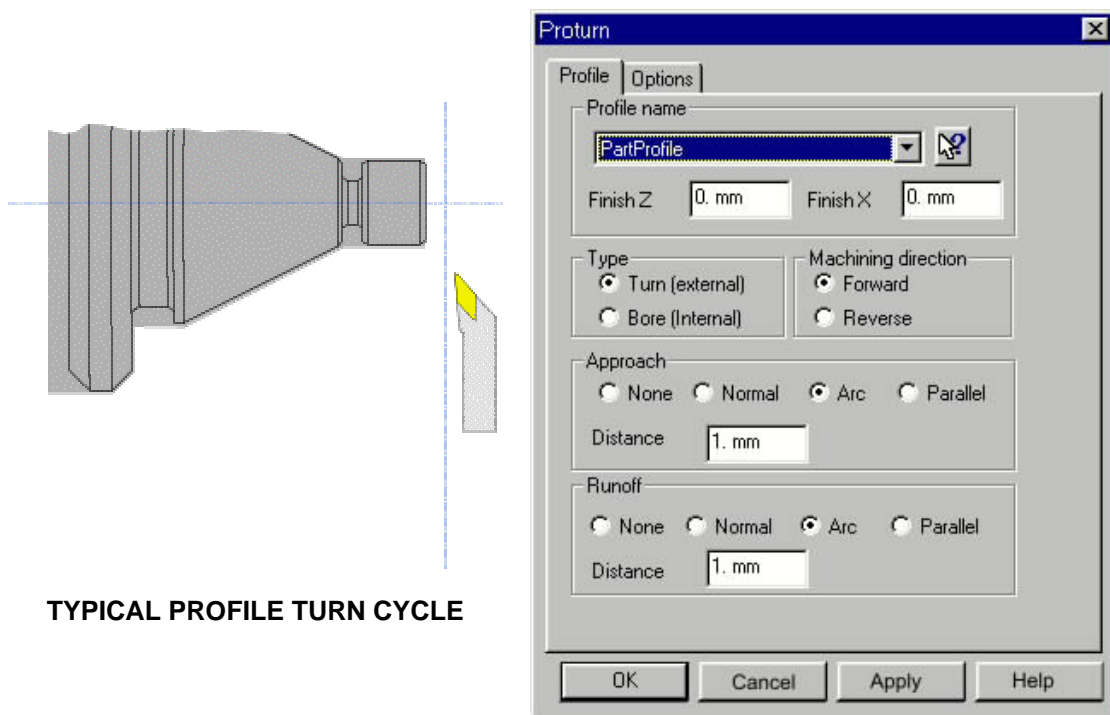
This menu contains commands which enable you define machining commands.

NOTE: The actual output of many of the miscellaneous operations in the lower part of the menu are completely dependant upon the post processor settings!


Machining - Profile Turn

The  button on the Operations Toolbar is a shortcut for this command

This command enables you to drive the cutter along a defined profile. It is usually used for finishing operations.



To see details of the options on page two (Options) of this dialog, click on the appropriate part of the graphic above.

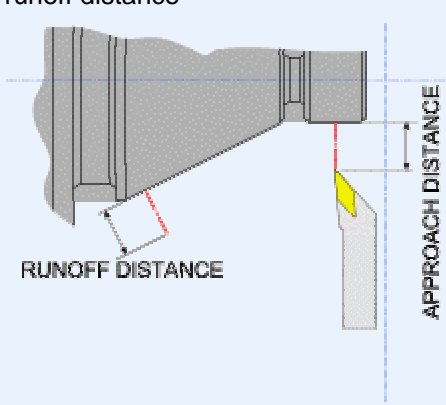
PROFILE NAME	<p>This field allows you to enter the name of the profile that is to be machined. The drop-down list will show all the profiles that are currently defined within the job.</p> <p>The  button enables you to use the cursor to pick the profile directly from the screen. If you click on the button, the dialog will be temporarily removed and the cursor will be placed in the display screen. Move the cursor over the required profile and click with the left hand mouse button. The dialog will re-appear with the name of the profile entered in the appropriate field.</p>
---------------------	--

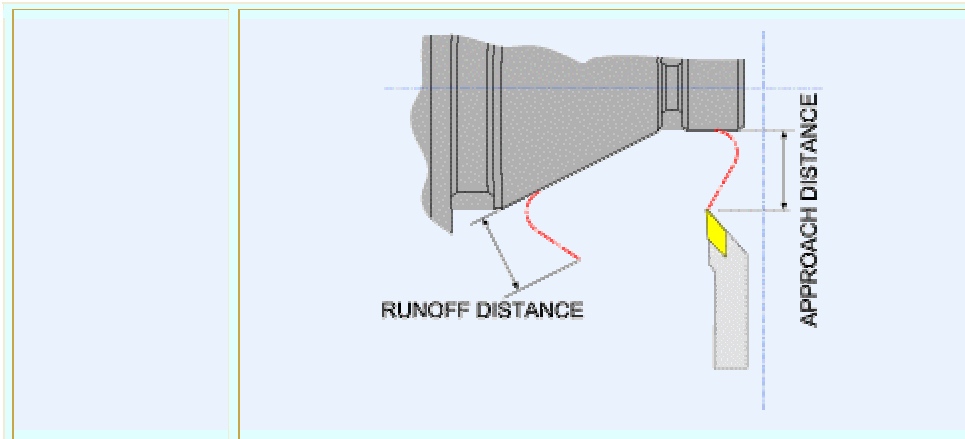
The cycle can be modified according to which options are ticked in the dialogue box.

CYCLE TYPE	DESCRIPTION
External	The cycle will be carried out on the external diameter of the workpiece. i.e. the feed will be from outside towards the centre line.
Internal	The cycle will be carried out on an internal diameter of the workpiece. i.e. the feed will be from the centre line towards the outside.
Finish	The Finish fields allow you to define a different finishing allowance for both the Z and X axes. This is the amount of material that will be left on the part profile for further finishing operations.

MACHINING DIRECTION	The machining direction may be specified as either Forward or Reverse . This refers to the direction of the profile as defined by the geometry.
----------------------------	---

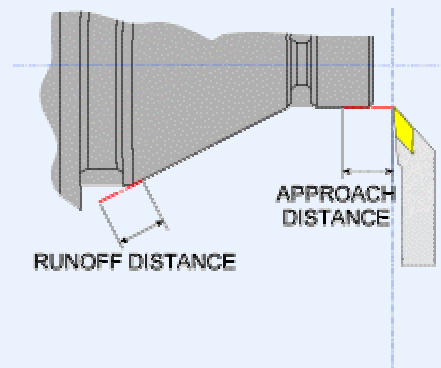
Approach and **Runoff** determine how the tool will attach to the shape to be machined, the point at which the tool will begin the cutting process and the point at which the tool will retract. There are 4 options :

TYPE	DESCRIPTION
NONE	No approach / runoff - the tool simply attaches to the profile or retracts from the profile at the point specified by the start and end points as defined on the Options page of the dialog.
NORMAL	The tool descends (approach) or retracts (runoff) from a point on a line at right angles to the first span to be machined. The length of this line is specified by the approach / runoff distance 
ARC	An ARC approach produces a line followed by an arc which is tangential to the first span to be machined. A Runoff move produces an arc followed by a line.



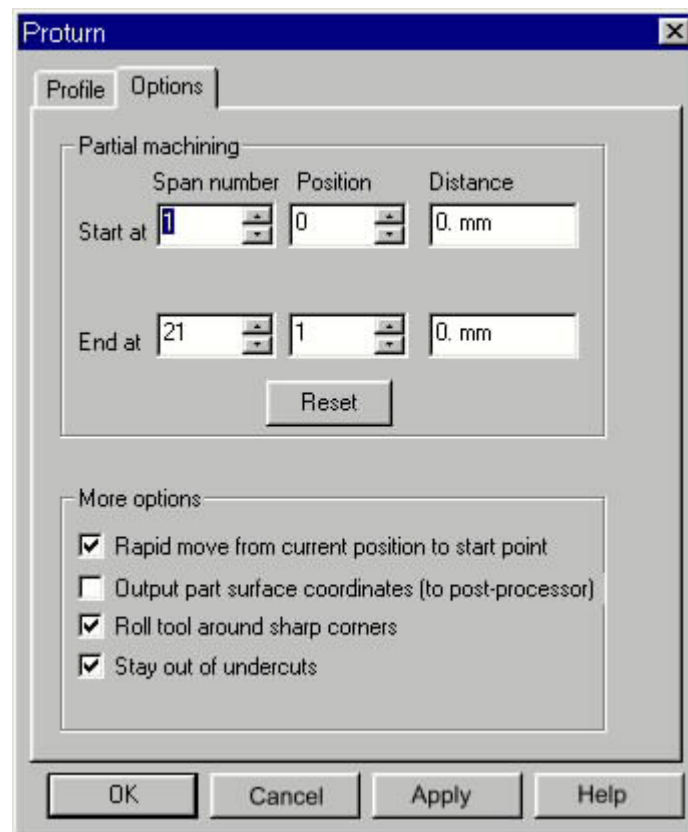
PARALLEL

The approach / runoff move is made by extending the first / last span to be machined by an amount equal to the approach / runoff distance specified.



Machining - Profile Turn - Options

The second page of the **Profile Turn** dialog enables you to set further options to the standard cycle.



PARTIAL MACHINING


These fields allow you to select any part of the profile for machining as the start and end points of the cycle.

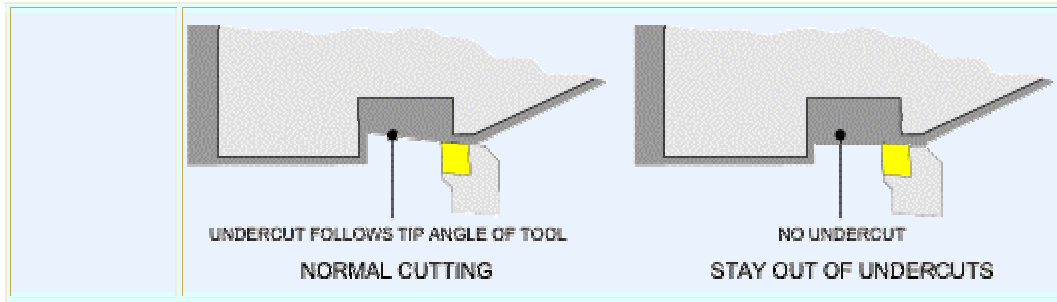
NOTE: The simple ZX position defined by say Span 1, 100% is identical to that of Span 2, 0%. This does **NOT** mean that these two positions are treated identically within the cycle. If you define an Approach or Runoff at this position, how this is carried out will depend upon the span geometry. For example, an

Approach position defined at the end of span 1 will not be the same as that defined by the start of span 2.

These fields will always have the start and end points of the full profile as default values. i.e. if you change nothing, the whole profile will be profiled (or at least as much as the cutter geometry allows).

Span number	The span number refers to the number of the element within the profile. Spans are numbered sequentially in the definition direction starting with one. To display the span numbers within a profile on screen, use the View - Options - Span Numbers command
Position	This field defines (in %) the distance along the selected span. The start of the span is 0% (0), the middle of the span is 50% (0.5) and the end of the span is 100% (1).
Distance	<p>This field defines a distance along the span from the Position defined by the above field. i.e. if you select span 2, 0.5, 3mm as the start position of the profiling operation, machining will begin at a position which is 3mm past the midpoint of span 2 as seen in the defined direction. If you wished to start machining at a point exactly 15mm along span 3 of the profile you could set the parameters as follows: Span 3, position 0, distance 15</p> <p>NOTE: Theoretically it is possible to define a distance here which will take the cutter beyond the end of the span. This will be collision checked against the profile and only carried out if the position does not interfere with other elements of the profile, otherwise machining will start (or end) at the end of the span.</p>

FURTHER OPTIONS	DESCRIPTION
Rapid Move from Current Position	This option will move the tool from its current position to the start position of the cycle (as modified by any Approach moves) at rapid feedrate, otherwise it will be moved at the currently defined feedrate.
Output Part Surface Coordinates	<p>Using this option will cause PartMaster to bypass its normal offsetting routines and output the finish part surface co-ordinates together with cutter radius compensation details, instead of outputting a corrected toolpath. (Finish allowance is still considered).</p> <p> This means that the offsetting routine is performed by the NC machine. There is no guarantee that the target machine tool can successfully cut the part</p>
Roll Tool around Sharp Corners	PartMaster can machine external sharp corners either by rolling the tool around the corner or by extending the tool path to the intersection point. It defaults to rolling around external sharp corners.
Stay out of Undercuts	Depending upon the tool tip definition, the cutter may be able to partially machine re-entrant areas of the workpiece. Normally it will do this - if you do NOT wish it do so, tick this box




Machining - Threadcut


The  button on the Operations Toolbar is a shortcut for this command

This command enables you to define threading cycles, both external (male) and internal (female).

Thread cutting cycle [X]

Cycle Limits

Start point Z × 

End point Z × 

Cutting direction

External Internal

Feedrate

Feedrate Lead Pitch TPI

Feed angle No. of Starts

Options


Thread depth Thread Width


Runoff distance

Roughing cuts Finishing cuts

CYCLE LIMITS

These fields enable you to define two diagonally opposite corners of the area to be covered by the cycle. The two corners may be given in any order - top right/bottom left - bottom right/top left - top left/bottom right - bottom left/top right.

The  button enables you to use the cursor to pick a position on the screen directly without having to know the exact co-

ordinates. If you click on the button, the dialog will be temporarily removed and the cursor will be placed in the display screen. Move the cursor to the required position and click with the left hand mouse button. The dialog will re-appear with the co-ordinates of the indicated position entered in the appropriate fields. The dialog will re-appear with the co-ordinates of the indicated position entered in the appropriate fields. If you hold down the **Shift** key when you click on the screen position, the cursor will change to a  and PartMaster will pick up the exact co-ordinates of the nearest geometry point, either of the blank or the part profile

CYCLE TYPE	DESCRIPTION
External	The cycle will be carried out on the external diameter of the workpiece. i.e. the feed will be from outside towards the centre line.
Internal	The cycle will be carried out on an internal diameter of the workpiece. i.e. the feed will be from the centre line towards the outside.

FEEDRATE	DESCRIPTION
Feedrate	The Feedrate for the cycle is entered in this field. The option boxes select which type of feedrate is being specified. Choose from Lead, Pitch or TPI
Feed Angle	This field specifies at which angle the feed-in between passes is made. It usually matches the flank angle of the thread but does not have to. Feed angle is measured positive anti-clockwise from a line parallel to the X axis. A feed angle of 0.0 will cause the tool to cut evenly on both flanks. Setting the feed angle to half the thread included angle will cause the tool to cut on one flank only.
No. of Starts	This specifies the number of starts that the thread has. If more than one start is specified, each start will be completely machines (roughing & finishing cuts) before the next is started.

The cycle can be modified according to which options are ticked in the dialogue box.

OPTION	DESCRIPTION
Thread Depth	Thread depth is the total depth of the thread including roughing and finishing cuts. It is measured radially and is always positive.
Thread Width	.
Runoff Distance	Runoff Distance: If a non zero runoff distance is entered the tool will be retracted in the X axis while continuing to cut the thread in the Z axis. This value should usually be about the same value as the thread depth.
Roughing Cuts	Number of roughing cuts - Number of finishing cuts: The number of roughing cuts and the number of finishing cuts

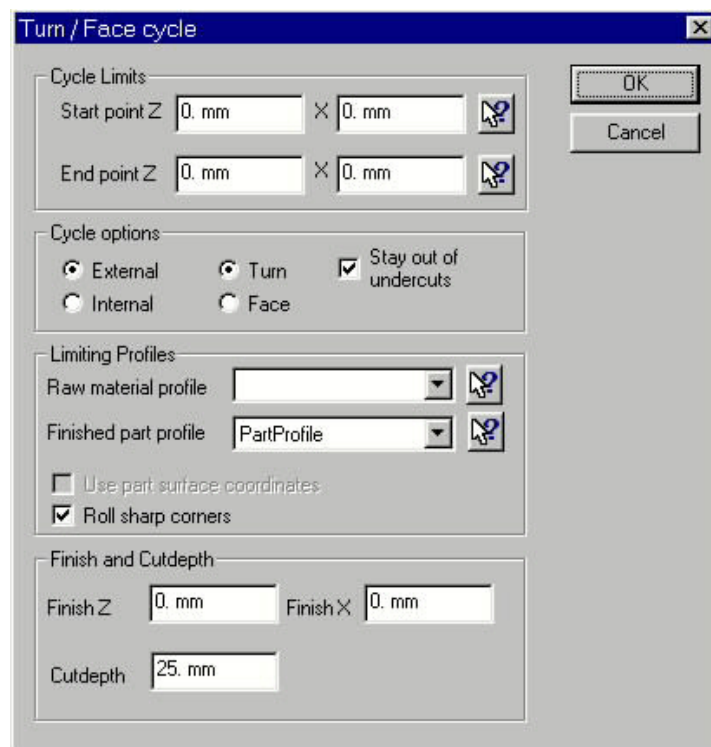
Finishing Cuts

together determine the total number of passes used to cut the thread. PartMaster increments the depth uniformly for each roughing cut and each finishing cut. The increase in depth for all the finishing cuts together is equal to the increase in depth for one roughing cut. i.e. if the thread depth was 3mm and you specified 5 roughing cuts and 2 finishing cuts, each roughing cut would be 0.5mm deep and each finishing cut 0.25 deep. When specifying multi-start threads, each thread start will be completely machined before the next thread is started.

Machining - Turn/Face

The  and the  buttons on the Operations Toolbar are shortcuts for this command



This command enables you to define general turning and facing cycles. The basic cycle which is defined by the co-ordinates of two opposite corners can be modified so that a profile (usually the finished part) is used as a limit to the machining. If a blank or billet is defined by means of another profile then machining can automatically be restricted to the area between this and the finished part profile.



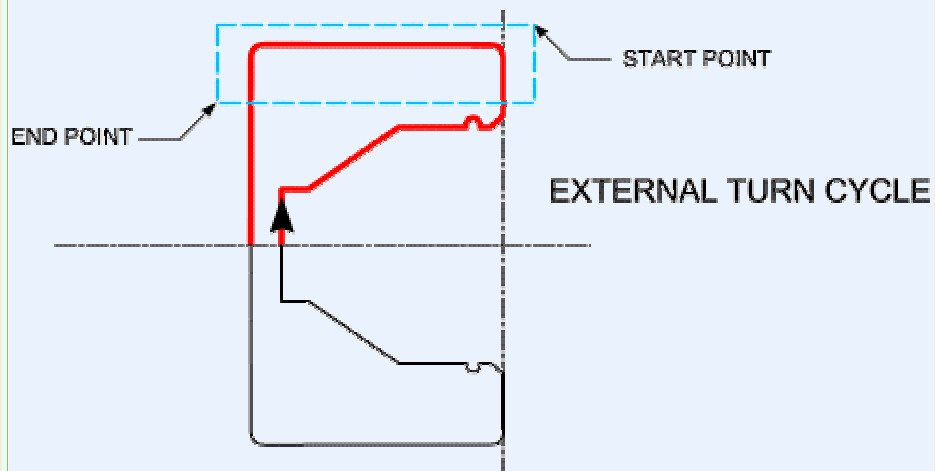
CYCLE LIMITS

These fields enable you to define two diagonally opposite corners of the area to be covered by the cycle. The two corners may be given in any order - top right/bottom left - bottom right/top left - top left/bottom right - bottom left/top right. If the cycle is not modified in any other way a simple area clearance will be performed between these limits. The extent of the cycle may be modified by a limiting (protected) profile - normally that of the finished part. Note that although the defined limits may cross over the centre line of the job, machining will finish at the centre line (as modified by any offsets). A blank or

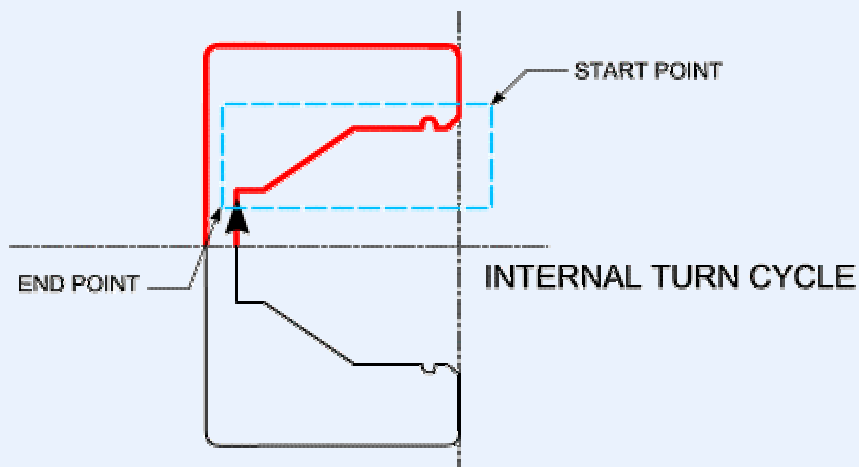
billet may also be used to limit the machining if it has been defined by a profile definition.

The  button enables you to use the cursor to pick a position on the screen directly without having to know the exact co-ordinates. If you click on the button, the dialog will be temporarily removed and the cursor will be placed in the display screen. Move the cursor to the required position and click with the left hand mouse button. The dialog will re-appear with the co-ordinates of the indicated position entered in the appropriate fields. If you hold down the **Shift** key when you click on the screen position, the cursor will change to a  and PartMaster will pick up the exact co-ordinates of the nearest geometry point, either of the blank or the part profile.

- When specifying the limits of a turn cycle, the coordinates you enter represent the diagonal of a rectangular shape.



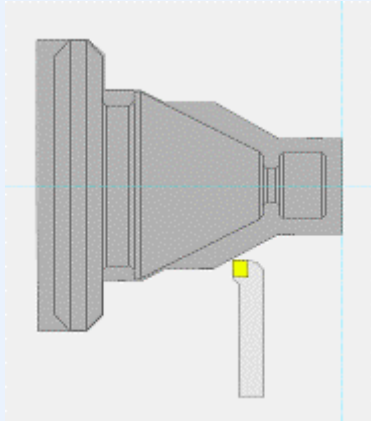
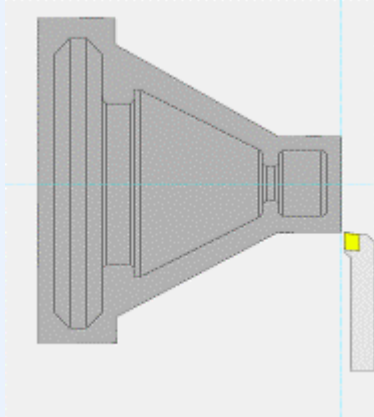
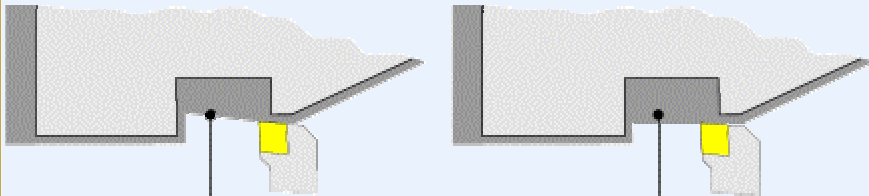
- Make sure that either the start or end point is actually in fresh air.
- Make sure that the start point is more than (Tip Radius + Finish Allowance) away from the finish part profile in both Z and X
- It does not matter which diagonal the start/end points define, Dcam will do its best to make sense of it.



- If you do not enter a valid Profile name in the "Finished Part Profile", then the cycle generated will be a simple TURN or FACE cycle within the limits you specify.

- If you enter valid Profile names for both the "Finished Part Profile", and the "Raw Material", then Dcam will assume that the "Raw material" profile represents a casting / forging and will assume that everything outside of it is fresh air. In the examples folder see the job 'Machined form casting'.

The cycle can be modified according to which options are ticked in the dialogue box.

OPTION	DESCRIPTION	
External	The cycle will be carried out on the external diameter of the workpiece. i.e. the feed will be from outside towards the centre line.	
Internal	The cycle will be carried out on an internal diameter of the workpiece. i.e. the feed will be from the centre line towards the outside.	
Turn/Face	<p>The cycle will consist of a series of cuts along the Z-axis with the feed in direction in the X-axis</p>  <p style="text-align: center;">TURN</p>	<p>The cycle will consist of a series of cuts along the X-axis with the feed in direction in the Z-axis.</p>  <p style="text-align: center;">FACE</p>
	<p>Stay Out of Undercuts Depending upon the tool tip definition, the cutter may be able to partially machine re-entrant areas of the workpiece. Normally it will do this - if you do NOT wish it do so, tick this box</p>  <p style="text-align: center;"> UNDERCUT FOLLOWS TIP ANGLE OF TOOL NO UNDERCUT NORMAL CUTTING STAY OUT OF UNDERCUTS </p>	

LIMITING PROFILES


RAW MATERIAL PROFILE

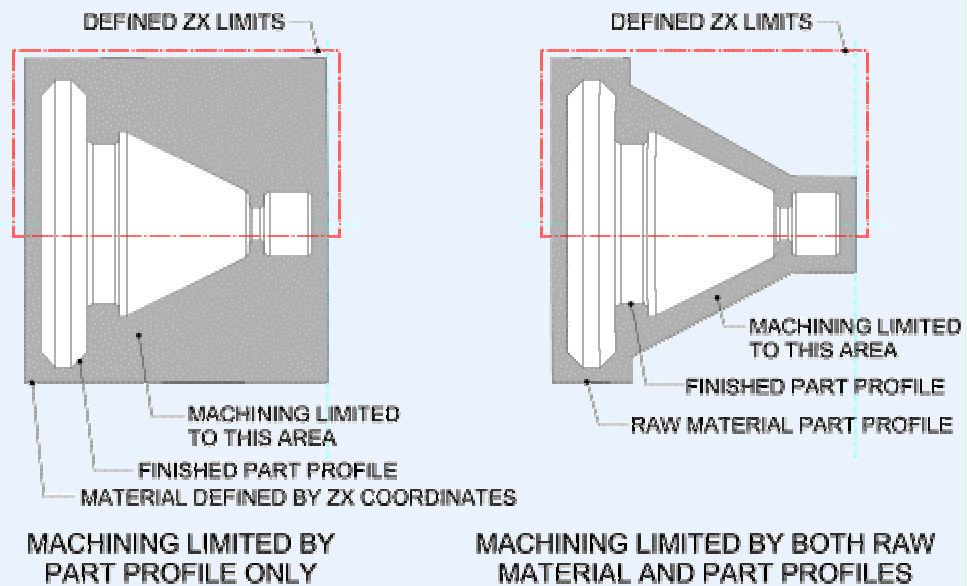
If the raw material (blank or billet) is defined by a profile, it can be entered here and will be used as a limit to the machining. This will avoid air cutting when a shaped casting is used.

FINISHED PART PROFILE

A profile name entered here will be used as a limiting (protected) profile to amend the machining as defined by the simple XZ co-ordinates (see above).



The  button enables you to use the cursor to pick either of the above profiles directly from the screen. If you click on the button, the dialog will be temporarily removed and the cursor will be placed in the display screen. Move the cursor over the required profile and click with the left hand mouse button. The dialog will re-appear with the name of the profile entered in the appropriate field.



Roll sharp corners: PartMaster can machine external sharp corners either by rolling the tool around the corner or by extending the tool path to the intersection point. It defaults to rolling around external sharp corners

FINISH & CUTDEPTH

The **Finish** fields allow you to define a different finishing allowance for both the Z and X axes. This is the amount of material that will be left on the part profile for finishing operations.

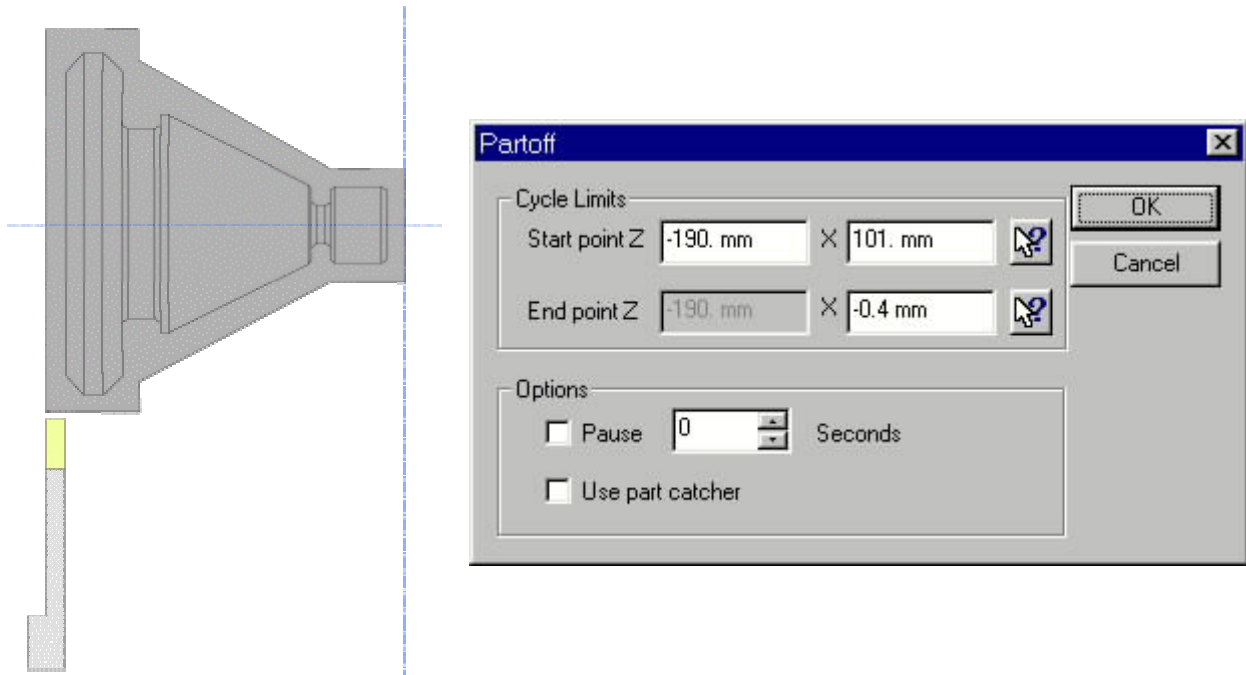
The **Cutdepth** specifies the in-feed between successive passes of the cutter. If no depth is entered here then the program will default to the cutdepth value specified in the tooling Definition.

See **Tooling - Tool Definition** 

Machining - Partoff



The  button on the Operations Toolbar is a shortcut for this command

This command enables you to define a partoff cycle to separate the finished part from the billet.



CYCLE LIMITS

These fields enable you to define two corners to define the position at which the partoff will take place. Note that for the second corner, only an X-axis value is required as a partoff cycle only takes place at one Z-position.

The  button enables you to use the cursor to pick a position on the screen directly without having to know the exact co-ordinates. If you click on the button, the dialog will be temporarily removed and the cursor will be placed in the display screen. Move the cursor to the required position and click with the left hand mouse button. The dialog will re-appear with the co-ordinates of the indicated position entered in the appropriate fields. If you hold down the **Shift** key when you click on the screen position, the cursor will change to a  and PartMaster will pick up the exact co-ordinates of the nearest geometry point, either of the blank or the part profile.

The cycle can be modified according to which options are ticked in the dialogue box.

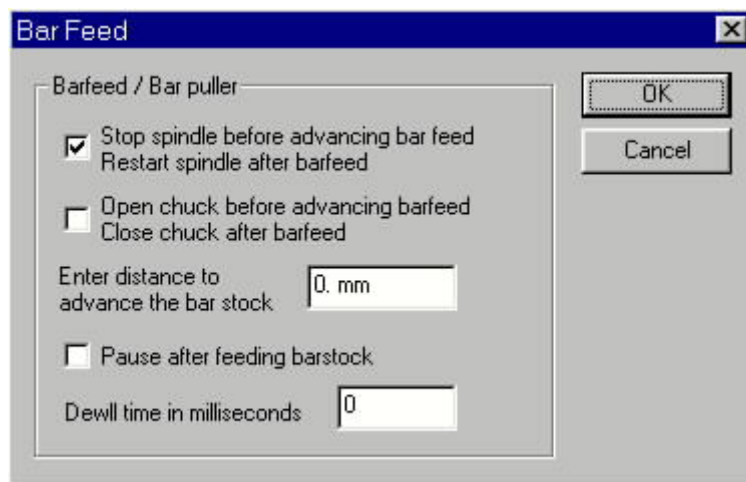
OPTION	DESCRIPTION
Pause	Pause: This enables you to program a pause in seconds when the tool reaches the greatest depth of the cycle.
Use Part Catcher	If a part catcher is used to catch the part that has been parted off, then click this tick box to send an appropriate command to the post processor. The Part Catcher can also be programmed separately See Part Catcher

MISCELLANEOUS FUNCTIONS

Barfeed

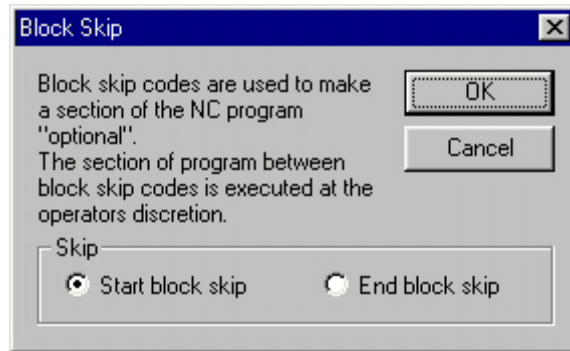
The  button on the Miscellaneous Operations Toolbar is a shortcut for this command

This option allows you to send a command to the post processor to move activate an automatic (powered) barstock feeder. Options allow you to specify the advance feed distance of the feeder as well as options to stop and start the spindle before operating the feeder and to pause the feeder for a given number of seconds. Which of these options are required will depend upon the implementation of the bar feed device for the lathe in question.



Block Skip

The  button on the Miscellaneous Operations Toolbar is a shortcut for this command

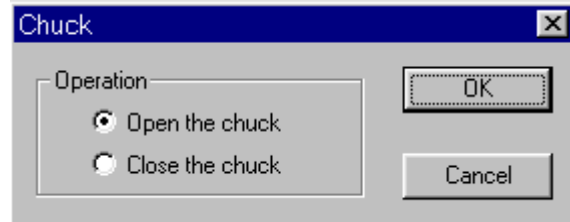


With this command you can specify an segment of the program which will be enclosed within "optional" markers. These commands will only be carried out if the NC machine is set to read the optional blocks

Chuck

The  button on the Miscellaneous Operations Toolbar is a shortcut for this command

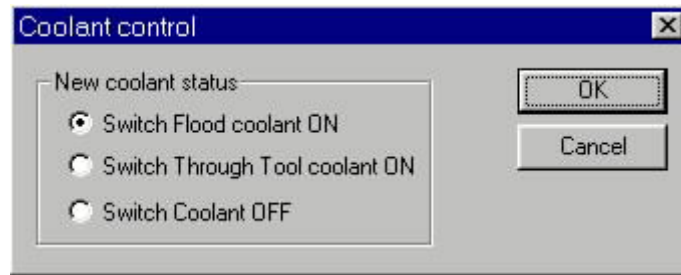
This option allows you to send a command to the post processor to either open or close an automatic (powered) chuck



Coolant

The  button on the Miscellaneous Operations Toolbar is a shortcut for this command

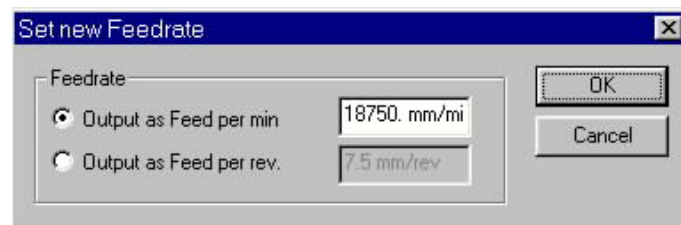
This option allows you to switch the flood coolant flow on the machine on or off as well as a enabling you to control through the tool coolant for machines that are fitted with this option.



Feedrate

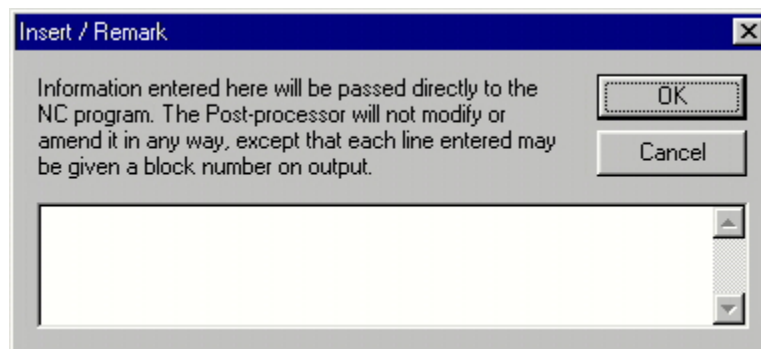
The  button on the Miscellaneous Operations Toolbar is a shortcut for this command

This option allows you to reset the cutting feedrate at any time during the program.



Insert

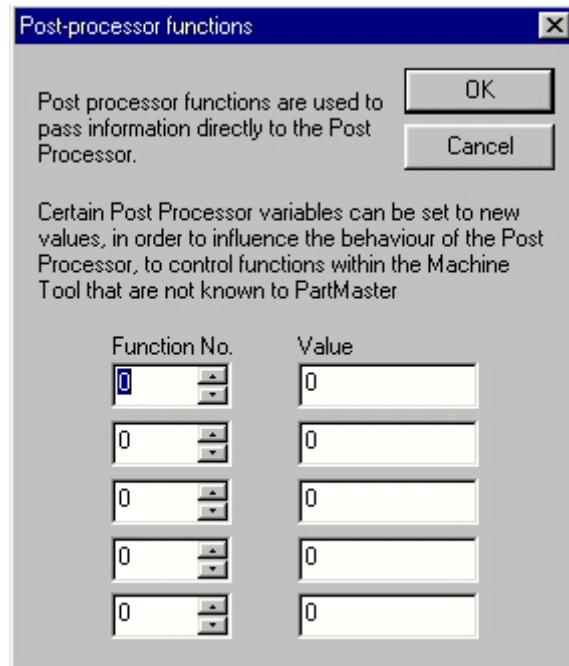
This command can also be accessed via the  button on the Miscellaneous Operations Toolbar



With this command you can specify a remark or other text which will be inserted into the NC code. See also post-processing.

PP Function

This command can also be accessed via the PP Function button  on the Miscellaneous Operations Toolbar

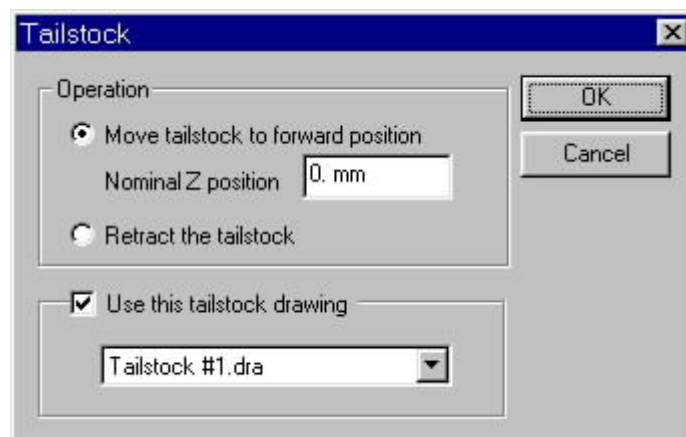


With this command you can address a function that has been built into the post processor. The function is called by number and a parameter can also be passed. It is generally used to active special commands and features that are not available within the normal PartMaster machining commands

Tailstock

The  button on the Miscellaneous Operations Toolbar is a shortcut for this command

This option allows you to send a command to the post processor to move an automatic (powered) tailstock to its forward or rearwards position. The Z-axis co-ordinate of the forward position can be input into the field provided.

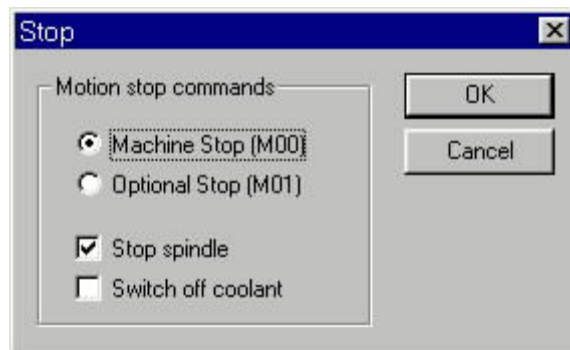


If a tailstock drawing is defined, this can be accessed and will be used in the graphic simulation on screen.

Stop

The  button on the Miscellaneous Operations Toolbar is a shortcut for this command

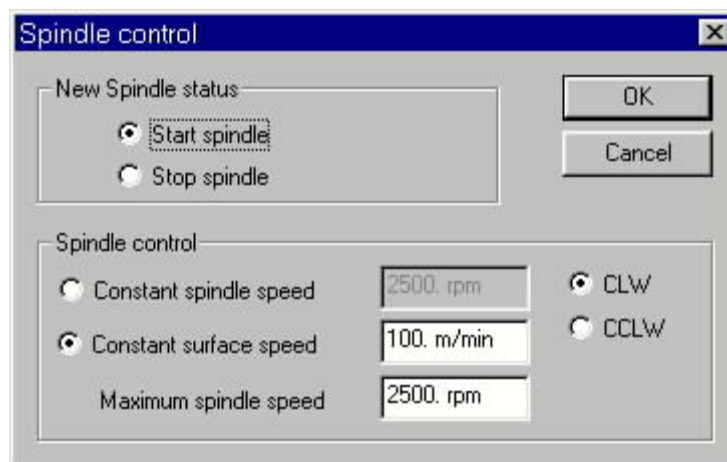
This option allows you to switch the NC machine off, either with a direct or optional stop command. Additional options allow you to switch both the spindle and coolant off as well, if necessary. **See also Coolant and Spindle**



Spindle

The  button on the Miscellaneous Operations Toolbar is a shortcut for this command

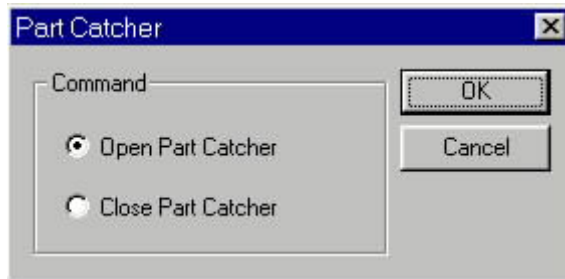
This option allows you to stop and start the spindle at any time as well as allowing you to enter a new spindle speed.



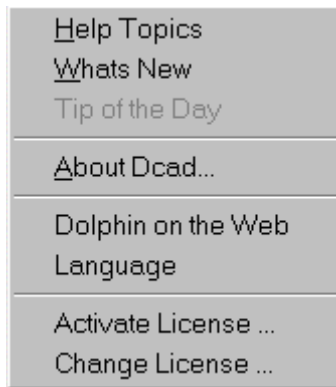
Part Catcher

The  button on the Miscellaneous Operations Toolbar is a shortcut for this command

This option allows you to open or close a part catcher which is normally used in conjunction with a Partoff operation. **See Machining - Partoff**



Help Menu




This menu contains commands to open this help file and to show information about the version number of DCAM.

Help Topics: This field of the help menu accesses this help file.

What's New: Opens a file which gives information about the latest release of the software.

Tip of the Day: Opens a dialog which gives helpful tips about Partmaster CAM

About DCAD: Opens a Dialog with three tabs which show information about DCAD - Version

No. Serial No. etc. This function can also be accessed via the  button on the top toolbar

Dolphin on the Web: Will open the Internet Explorer and access the Dolphin Web Site. To be successful you must have a valid Internet connection.

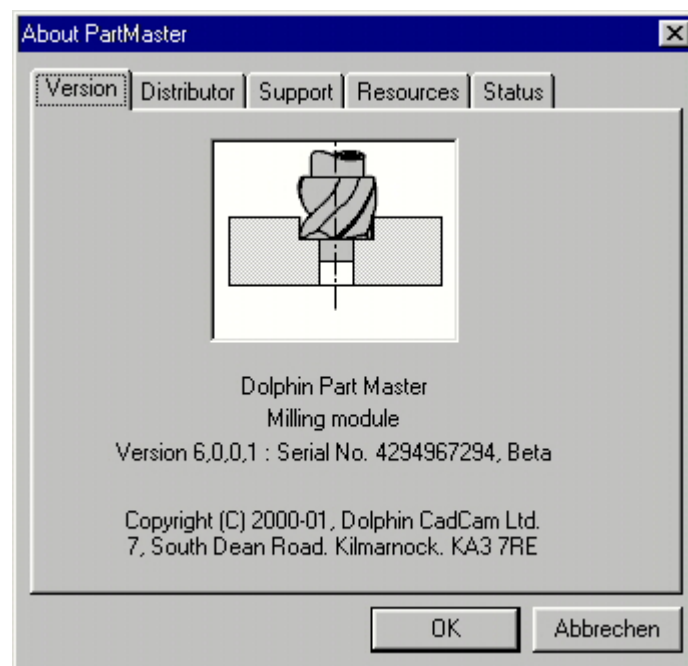
Language: Opens a dialog in which you can select the required language from the pull-down list. At the moment English is the only valid choice.

Activate License ... This command gives you access to the Wizard which will take you through the process of activating your users license. To do this you will need to acquire an activation code from Dolphin. All Dolphin products may be downloaded from the Dolphin website and will be complete in every detail except that the final output functions will be restricted. To turn your software into a full working product you need only obtain the activation code from Dolphin and then follow this procedure.

Change License ... This command gives you access to the wizard which will take you through the steps necessary to change or update your license. An update code will be required which you can obtain from Dolphin

Help - About DCAM

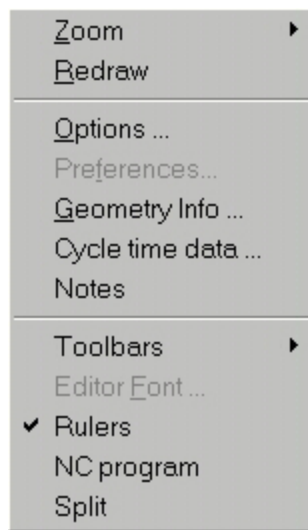
This command displays a dialog which contains information about the version number and other information about Dcam.



Help - What's New

Opens a help file which documents all the various alterations and fixes of the various software releases

View Menu



This menu contains commands to do with displaying the geometry and toolpath on screen as well as enabling you to set default values for a wide range of options..

View - Cycle Time Data

This command opens the a dialog where data concerning machining times for the currently defined machining cycles is displayed.

The screenshot shows a dialog box titled "Cycle time Estimate data" with a table containing the following data:

Operation	Rapid	Feed	Time	Total Time
M6 1 Select Tool No 1 CRC 1 TLC ...	0. mm	0. mm	0 mins 0 secs	0 mins 0 secs
2 Area Clear "con1", At Z 20. m...	573.862 mm	1495.545 mm	10 mins 29 s...	10 mins 29 secs

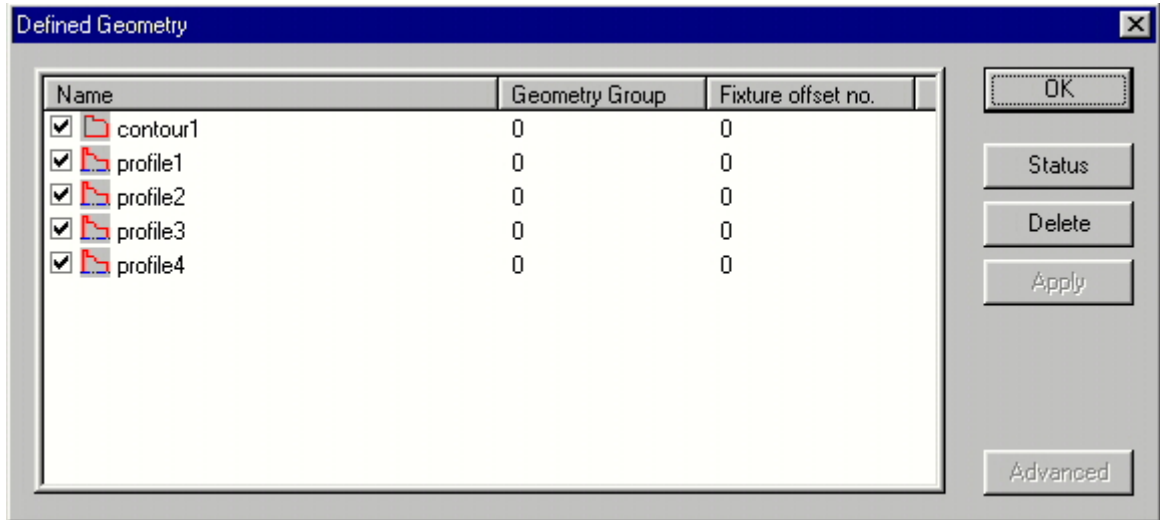
Buttons: OK, Cancel, Print

View - Editor Font

This command enables you to select the font that will be used to display the NC code generated by the post processor. This text is displayed in the Edit NC Code area of the screen. The standard Windows Font dialog will be opened which will display all the fonts currently installed on your computer - select the required font, style, size and colour.

View - Geometry Info

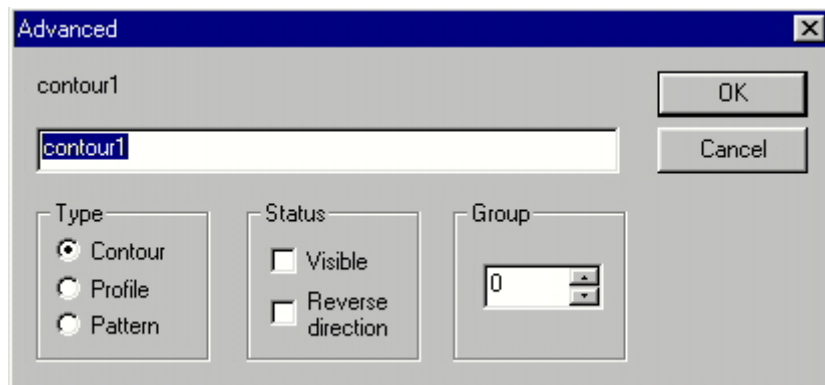
This command opens the a dialog where a list of all the currently defined geometry is displayed.



Geometry can be activated or deactivated by means of the click boxes provided. If it is deactivated it will not be displayed on screen.

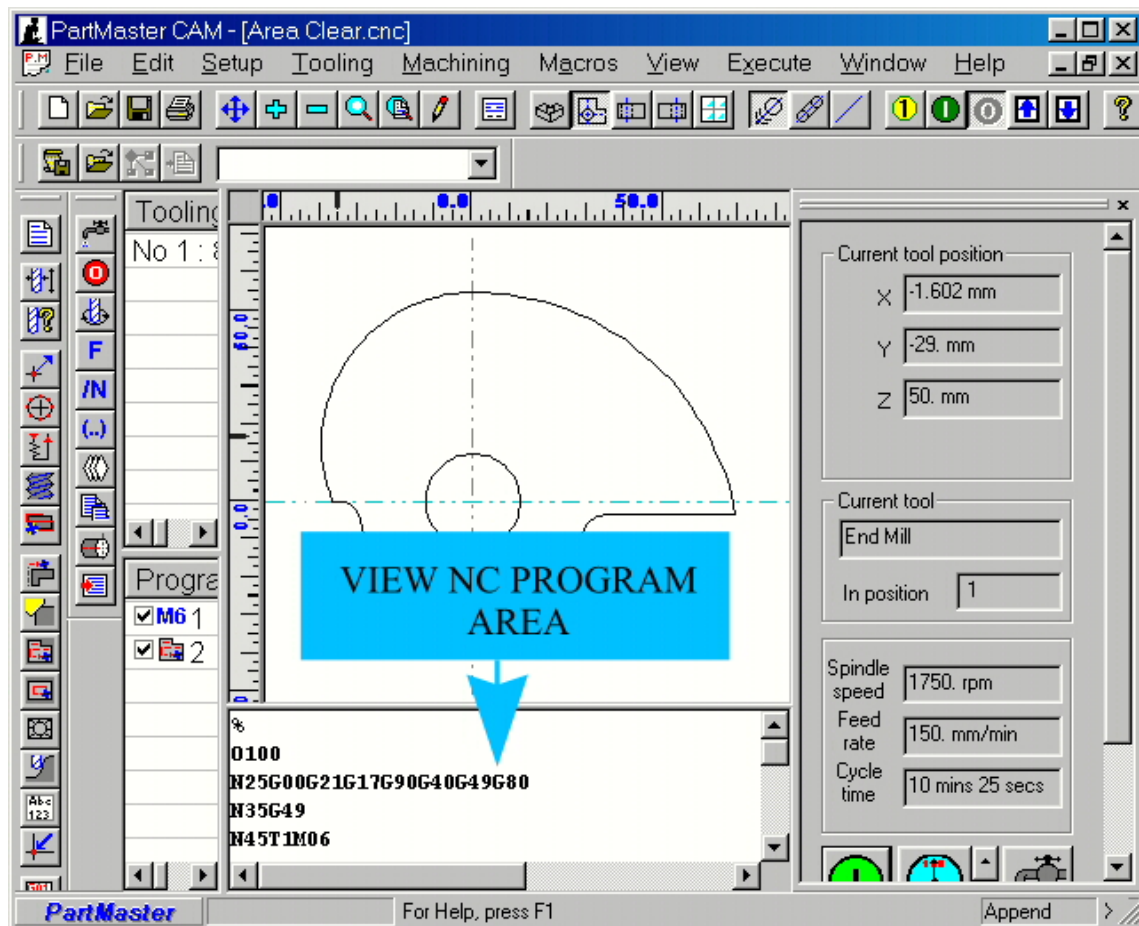
A geometric element can be deleted by clicking on it to highlight it and then hitting the **Delete** button.

You can edit the parameters of a geometric element by double clicking on it with the left hand mouse button. A new dialog displaying the parameters is displayed:



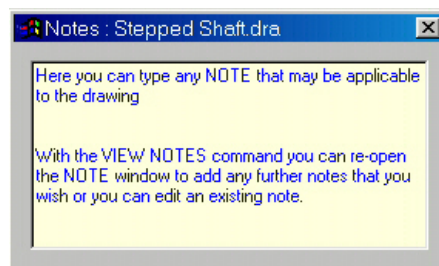
View - NC Program

This command opens a window beneath the normal graphic window where the current NC program (after post processing) is displayed This window will also be opened automatically if you run a post process operation on the current program.



View - Notes

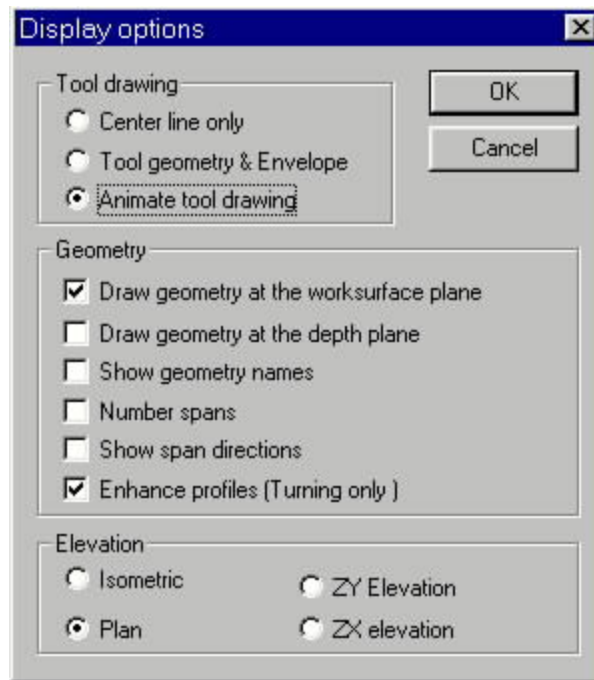
The **View - Notes** command opens a window in which you can add any notes that you wish to the program. When you have finished the note, hit the ESC key or the close icon on the window. You will be asked if you wish to save the note. If an existing note has been saved you can re-open it with the View - Notes command. The note can be edited or you can add further information as required. The View Note window is "**sticky**" i.e. it will remain displayed at all times irrespective of any other operations that you perform until you close it. The View-Notes command may also be used to provide on-line help in carrying out various operations. Some preliminary versions of these tutorials may be found in the **How Do I ...** folder. If the Notes dialog is visible when the program document is closed, it will automatically be re-displayed when that program is next opened. The dialog may be resized (by dragging its edges with the mouse) and can be moved around the screen by dragging its title bar.



View - Options

The  button on the Top Toolbar is a shortcut for this command, or right click when the cursor is in the graphics area.




This command opens the Display Options Dialog where you can set a wide range of options for displaying both the geometry and toolpath's and cutting sequences.



A general description is given below but the details will vary depending upon whether the Milling or Turning module is being used.

Tool Drawing

The toolpath and machining that you define can be displayed in three different ways as follows:

- 1 The toolpath is shown as a centre line only. The  button on the main toolbar is a shortcut for this option
- 2 The toolpath is shown as a centre line with an animated cutter. The  button on the main toolbar is a shortcut for this option.
- 3 A full simulation of the cutter path with an animated cutter and shading of the machined part. Note that full tool geometry can only be shown if the tool geometry is defined within a tool database. The  button on the main toolbar is a shortcut for this option





Geometry

Geometry display options are available as follows:

- 1 The geometry can be shown at the work surface only or with depth (if defined)

- 2 The names of geometric entities can be displayed
- 3 The span numbers of elements within geometric entities can be displayed
- 4 The direction of spans within geometric entities can be displayed

The geometry that you have defined or imported can be displayed in any one of four view directions as follows:

- 1 Isometric. The  button on the main toolbar is a shortcut for this option.
- 2 Plan view (XY). The  button on the main toolbar is a shortcut for this option
- 3 Front View (XZ). The  button on the main toolbar is a shortcut for this option
- 4 Side View (YZ). The  button on the main toolbar is a shortcut for this option

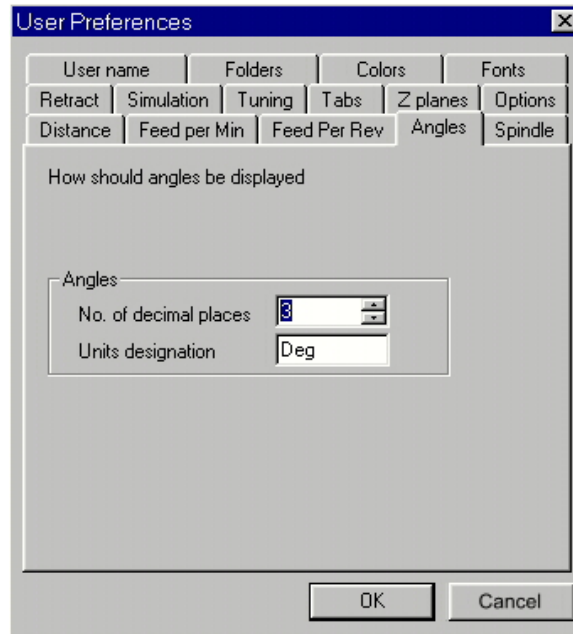
View - Preferences

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



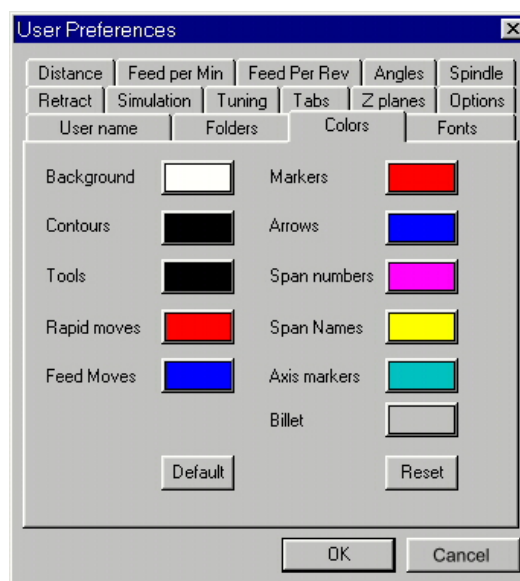
View - Preferences - Angles

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



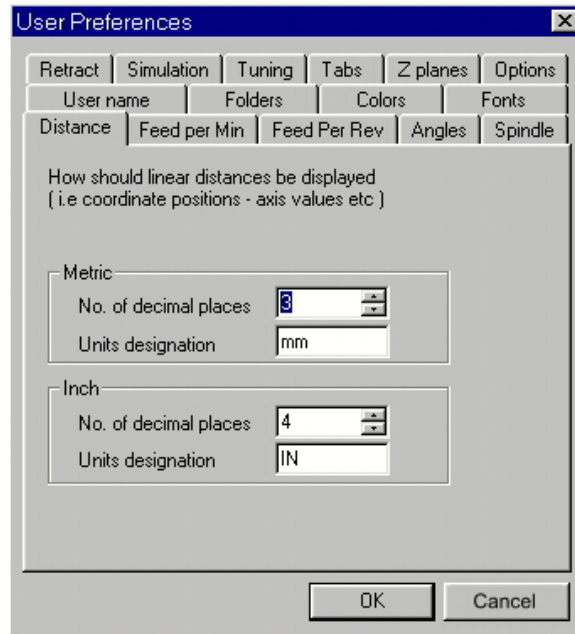
View - Preferences - Colours

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



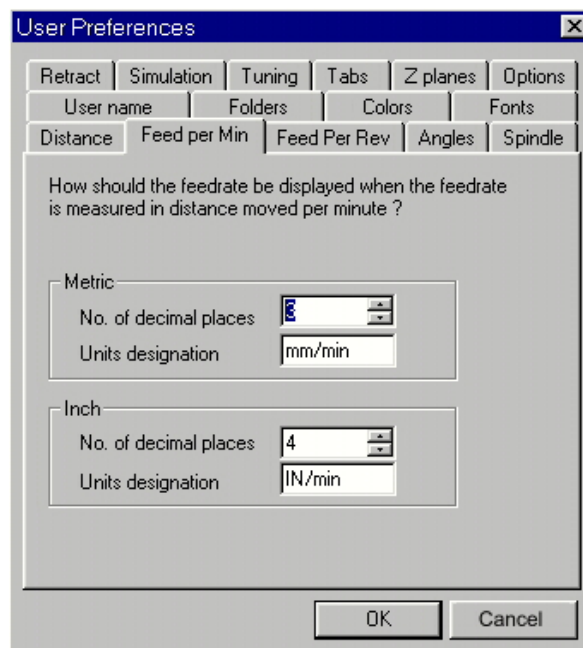
View - Preferences - Distance

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



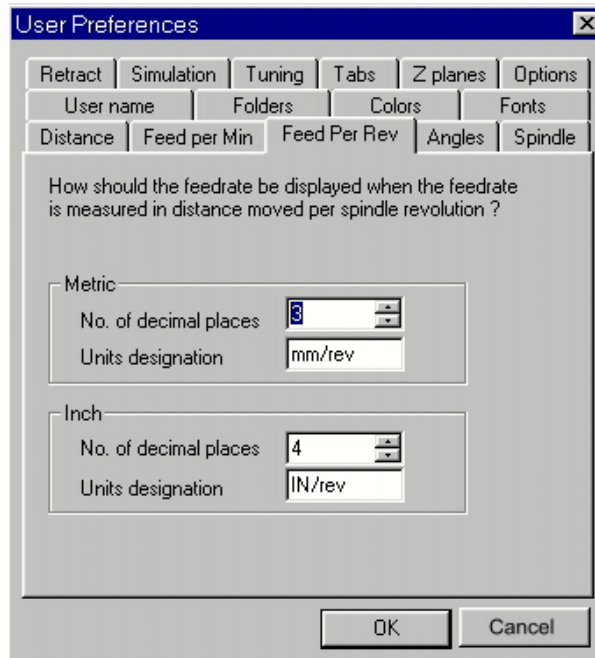
View - Preferences - Feed Per Min

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



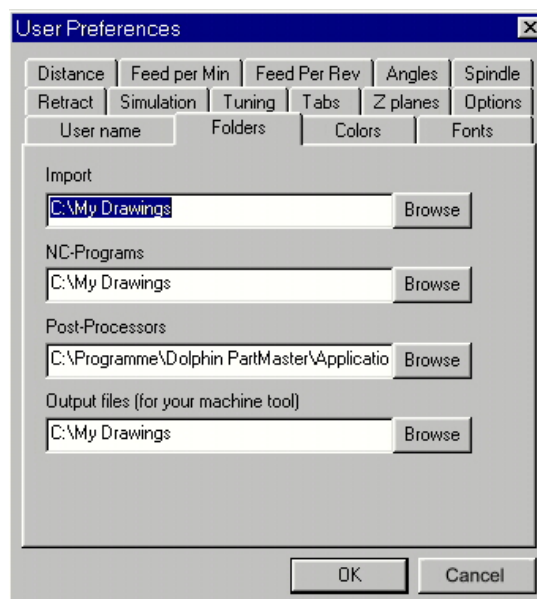
View - Preferences - Feed Per Revolution

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



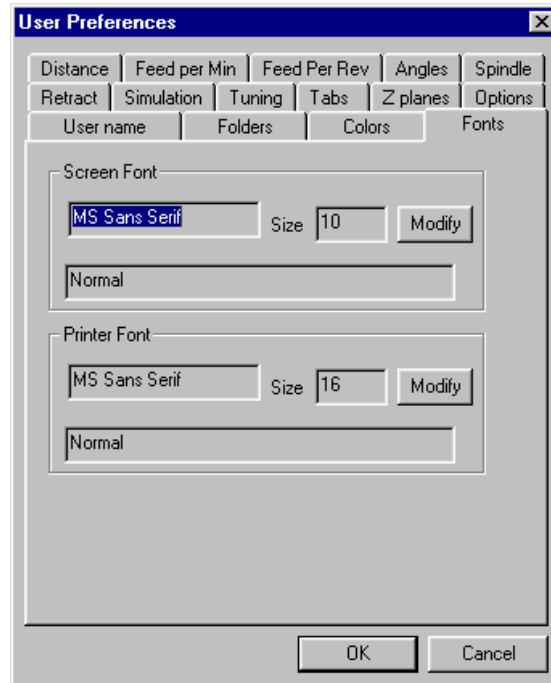
View - Preferences - Folders

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



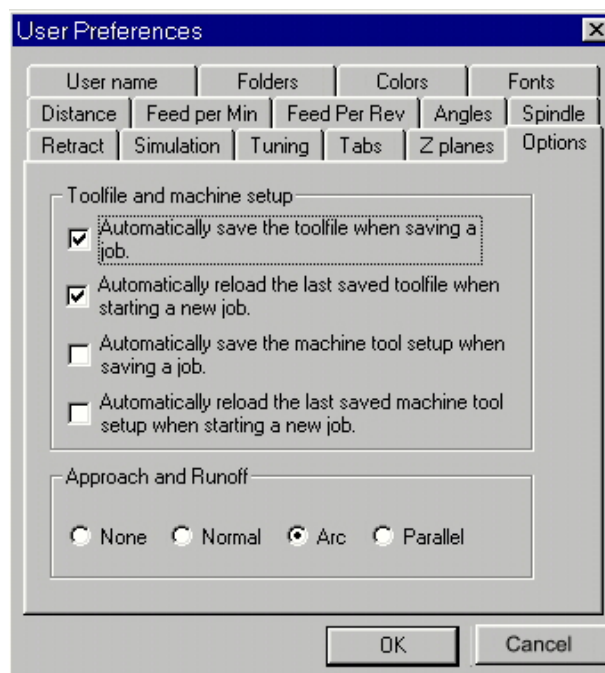
View - Preferences - Fonts

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



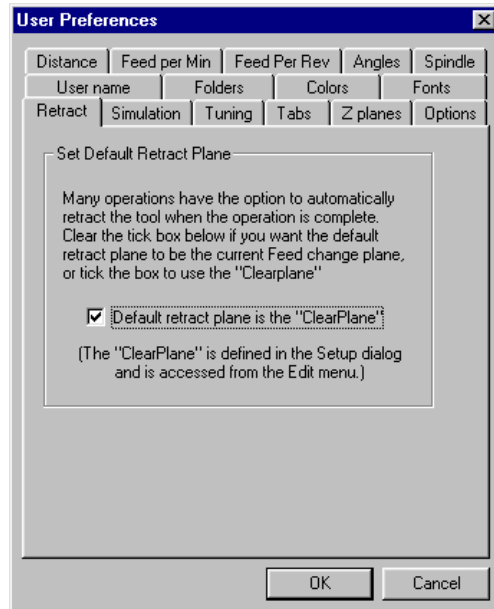
View - Preferences - Options

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



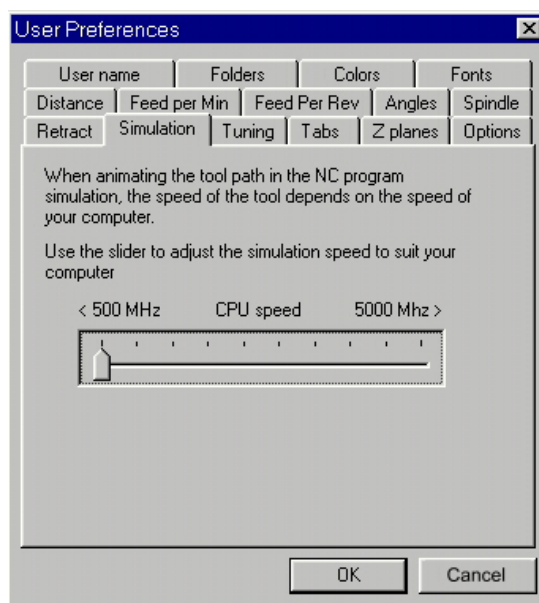
View - Preferences - Retract

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



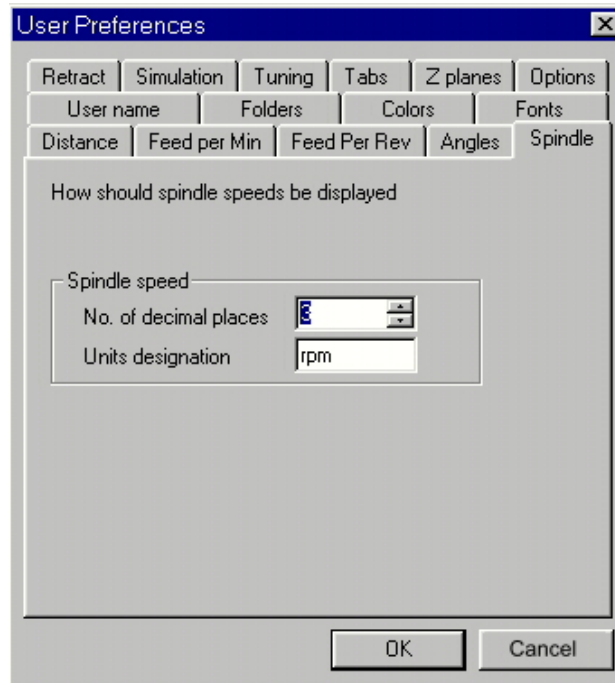
View - Preferences - Simulation

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



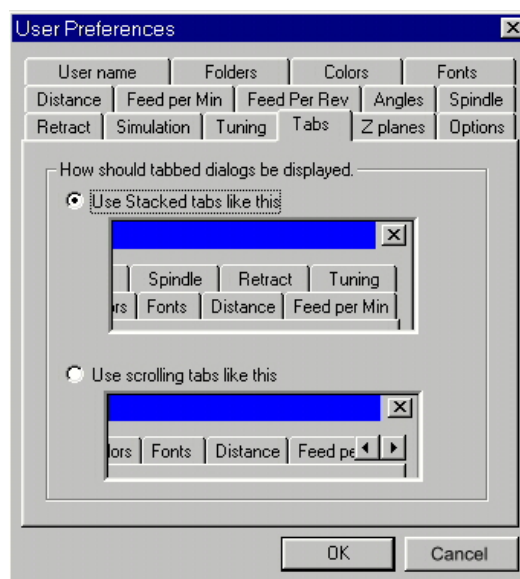
View - Preferences - Spindle

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



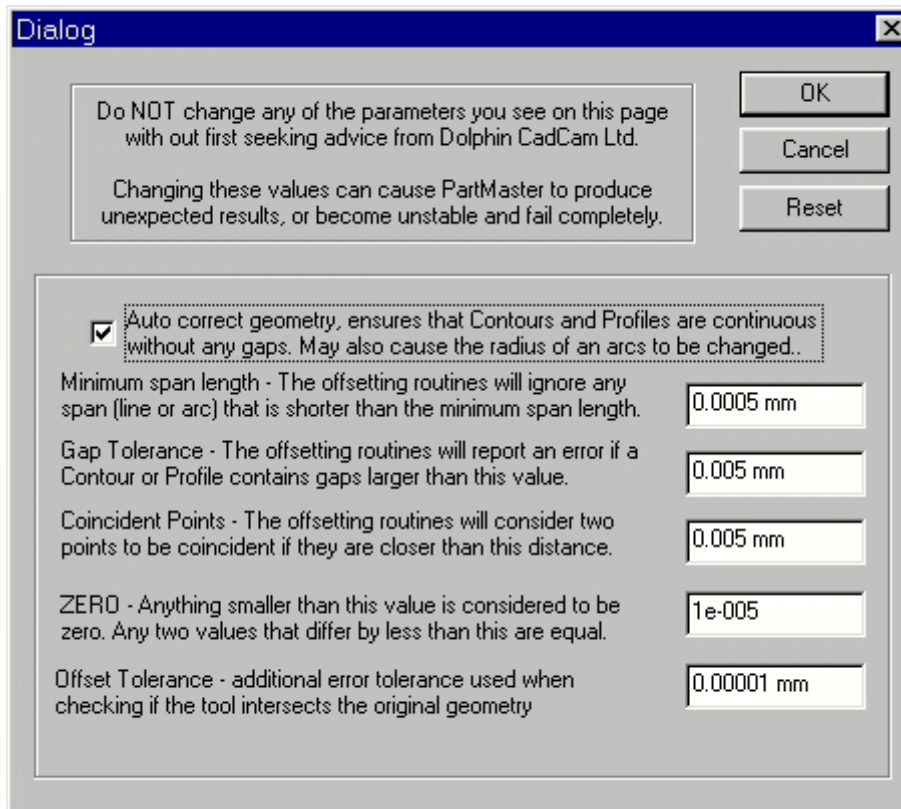
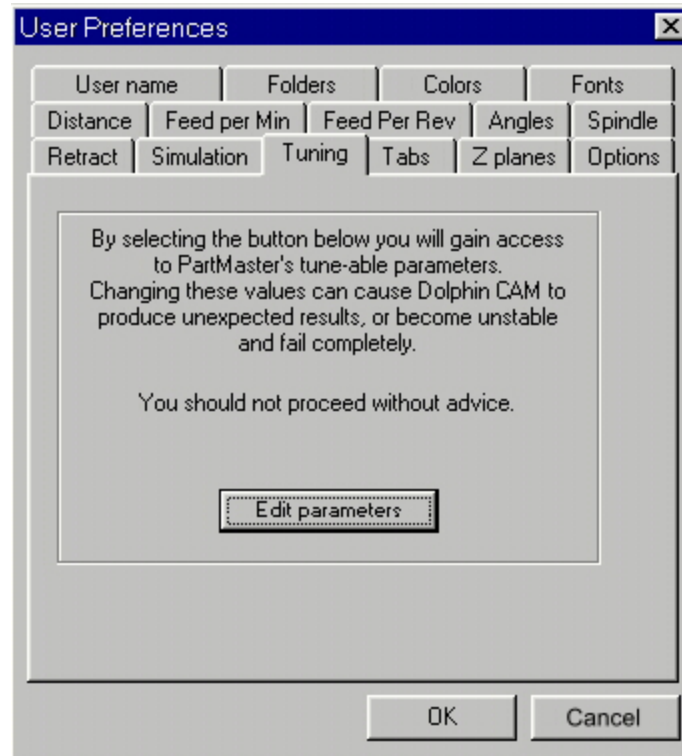
View - Preferences - Tabs

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



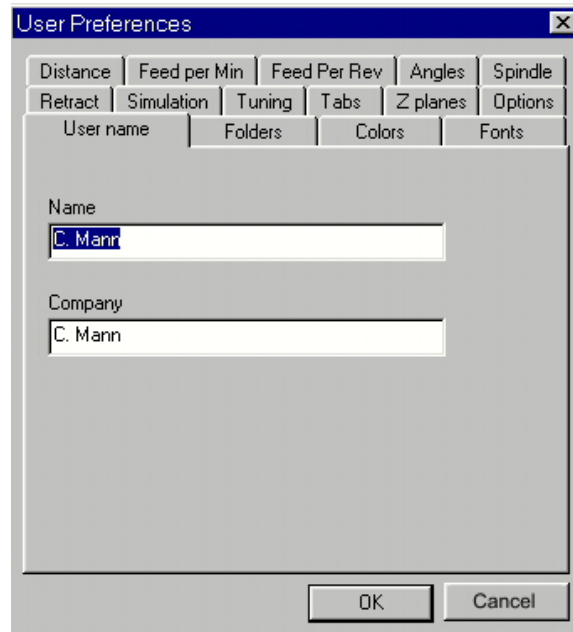
View - Preferences - Tuning

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



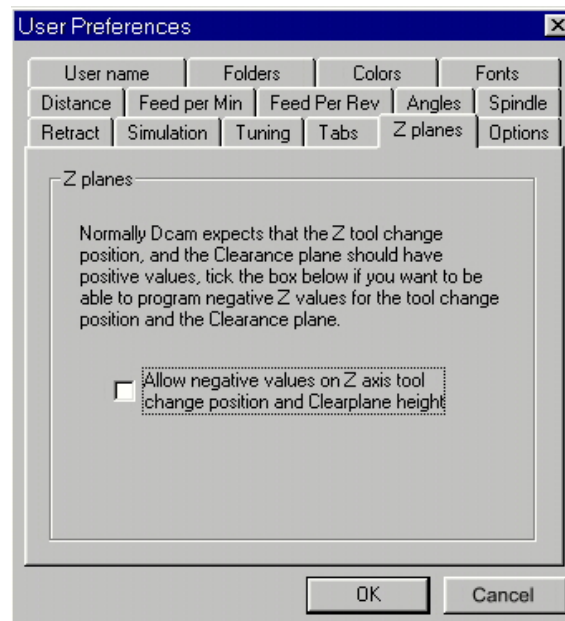
View - Preferences - User Name

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.




View - Preferences - Z Planes

This command opens the User Preferences Dialog where you can set a default values for a wide range of options.



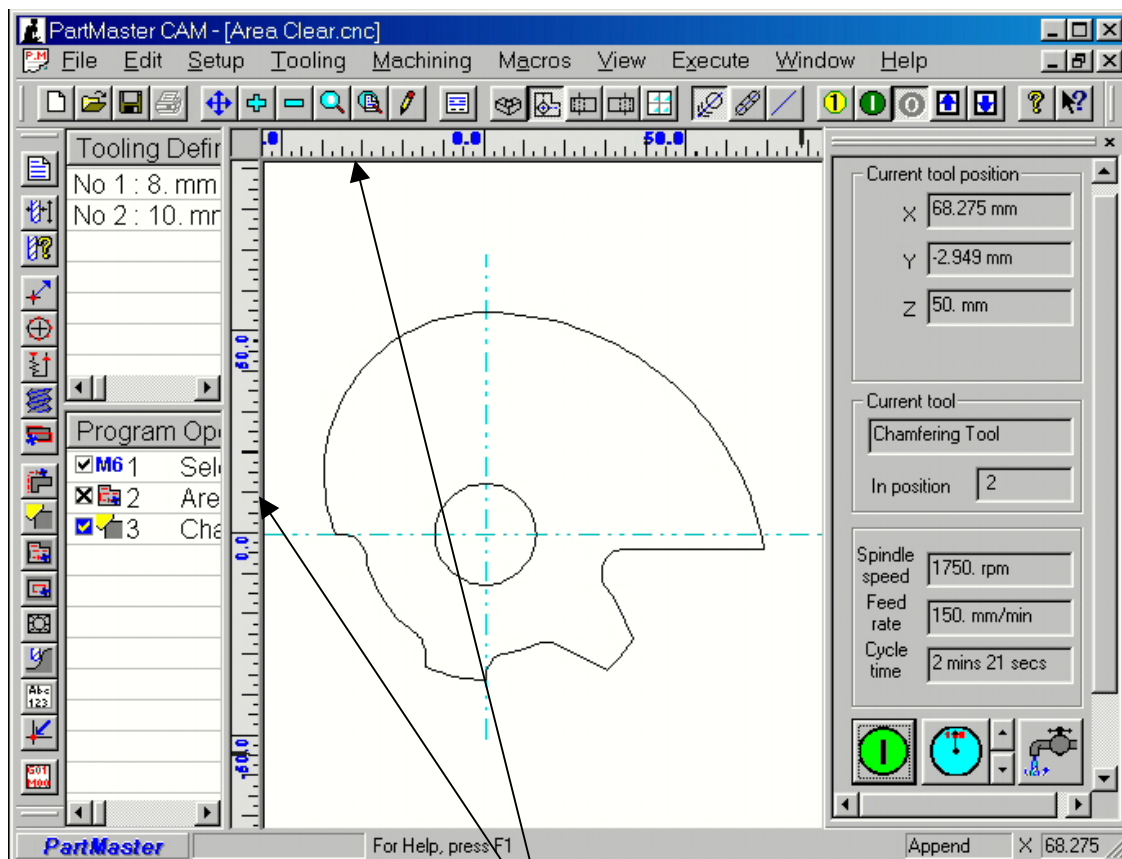
View - Redraw

The  button on the Top Toolbar is a shortcut for this command

The View - Redraw command redraws the current graphic screen, removing any toolpath's that are displayed. Note that the toolpath's are NOT deleted, they are simply removed from the visual display.

View - Rulers

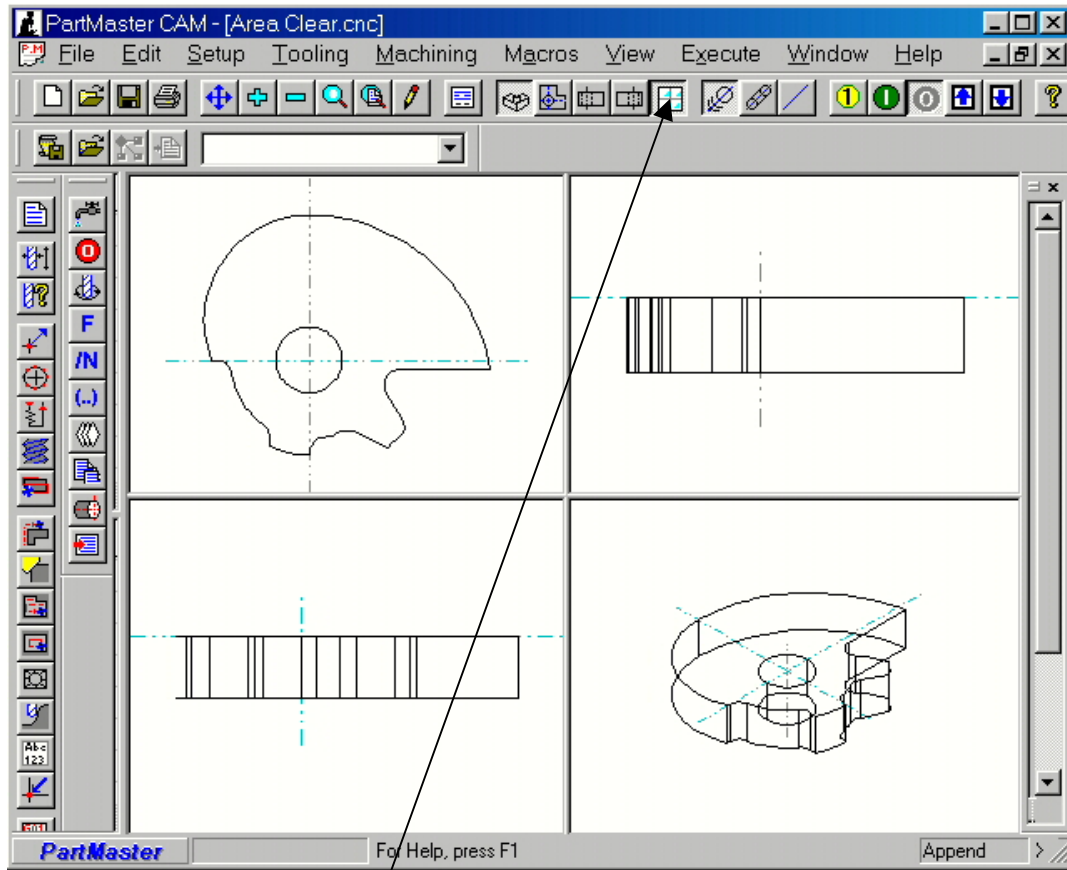
The **View - Rulers** command displays both horizontal and vertical rulers along the top & left hand side of the display screen id a single plan view is selected. The origin of the scales is placed at the program origin point. Movement of the cursor is shown by dynamic movement of cross hairs on the scales.



The screen Rulers

View - Split

This command splits the screen into four smaller windows containing the four views which are normally available. The active screen is chosen by clicking within one of the four windows with the left hand mouse button. Program execution and simulation is only shown within the currently selected window.



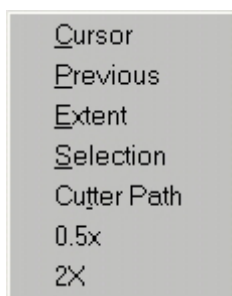
This is the SPLIT VIEW icon

View - Toolbars

This command opens the View - Toolbars sub-menu which you can use to switch the display of all the screen toolbars on or off.

- ✓ Files Toolbar
- ✓ Machining commands
- ✓ Misc. commands
- ✓ Browser Toolbar
- ✓ Macro Toolbar
- ✓ Machine Status
- ✓ Status Bar

View - Zoom



This sub-menu contains commands to zoom the screen display:

View - Zoom - Cursor

The  button on the Top Toolbar is a shortcut for this command

The geometry displayed on screen is redrawn so that the area defined by the cursor just fits inside the screen limits. Any toolpath's that are displayed will be removed from the screen. Note that the toolpath's are NOT deleted, they are simply removed from the visual display.

Click on the icon and then move the cursor into the graphic display area. move to one corner of the area that you wish to display. Click and hold down the left hand mouse button. Drag the cursor to the diagonally opposite corner of the desired area. A rubber band box will be drawn to shown the area defined. When the box is at the desired position release the mouse cursor. The area of the screen defined by the box will be redrawn so that it just fits within the display area.

View - Zoom - Cutter Path


The screen is redrawn so that the cutter path is just fitted within the screen extents..

View - Zoom - Double

The  button on the Top Toolbar is a shortcut for this command

The geometry displayed on screen is doubled in size. Any toolpath's that are displayed will be removed from the screen. Note that the toolpath's are NOT deleted, they are simply removed from the visual display.

View - Zoom - Extent

The  button on the Top Toolbar is a shortcut for this command

The geometry displayed on screen is redrawn so that all the defined elements just fit inside the screen limits. Any toolpath's that are displayed will be removed from the screen. Note that the toolpath's are NOT deleted, they are simply removed from the visual display.

View - Zoom - Half

The  button on the Top Toolbar is a shortcut for this command

The geometry displayed on screen is halved in size. Any toolpath's that are displayed will be removed from the screen. Note that the toolpath's are NOT deleted, they are simply removed from the visual display.

View - Zoom - Previous

The screen is redrawn to the last selected zoom option.

View - Zoom - Selection

The screen is redrawn so that the currently selected geometry (contour etc.) is just fitted within the screen extents..

Window Menu



This menu contains commands which control how the various programs that you have loaded will be displayed.

All the programs that you have currently loaded are listed at the bottom of the menu. Click on any one of these names with the left hand mouse button to select it as the currently active program.

