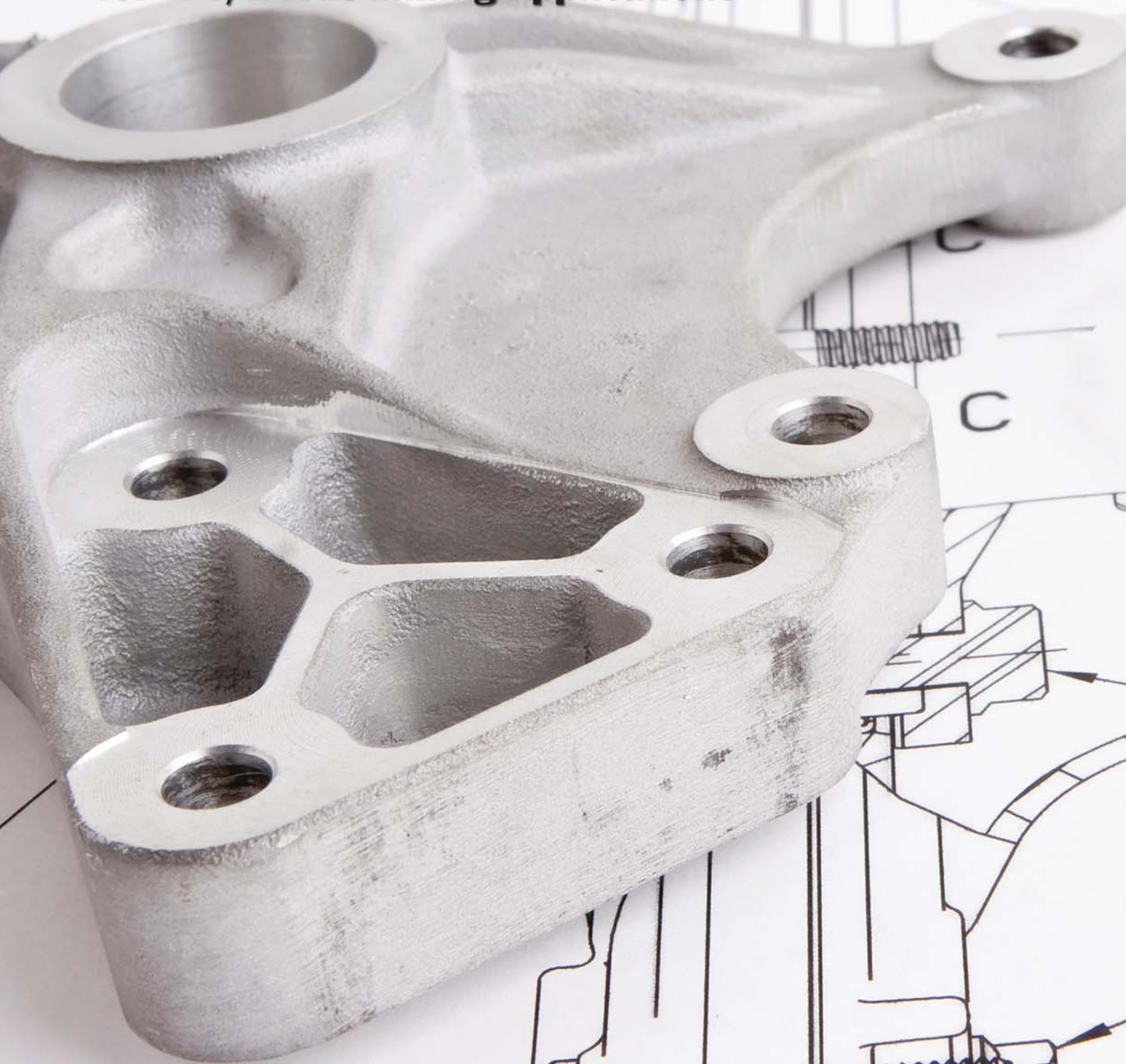


Exploring Mastercam X3

for 2-1/2 Axis Milling Applications



Peter Smid

Disclaimer

In-House Solutions, Inc. (the *Publisher*)

and

Peter Smid (the *Author*)

provide this publication in the form of 'as is', without warranty of any kind, either expressed or implied, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose.

The author may make improvements or changes in this publication and/or any electronic files (if available), or in the program examples used in this publication, at any time and without notice.

Neither the *Publisher* nor the *Author* assumes any responsibility for any error that may appear in the publication.

Use of names of companies and products in this publication does not reflect an endorsement by either those companies or by the *Publisher* or the *Author*.

Published by:

In-House Solutions, Cambridge, ON, Canada

**Head Office:
In-House Solutions, Inc.
240 Holiday Inn Drive, Unit A
Cambridge, ON N3C 3X4
Canada**

Tel: (519) 658-1471

Fax: (519) 658-1335

**www.inhousesolutions.com
www.emastercam.com**

books@inhousesolutions.com

**This book or parts thereof may not be reproduced,
stored in a retrieval system
or transmitted in any form
without the permission of the publisher**

**Cover Design: In-House Solutions, Inc.
Coordinators: Kim Faustini, Brett Thomas, Tyler Yanta**

1 2 3 4 5 6 7 8 9 10

Copyright © 2008 Peter Smid - All rights reserved

Printed in Canada

Introduction

Mastercam has been the leader in CAM software since 1985. Even today, it retains this number one market position with over 135 000 seats worldwide. Developed by CNC Software, Inc., of Tolland, CT, USA, it has become the de facto standard software of its kind.

The purpose of this *Exploring Mastercam X3 for 2-1/2 axis Milling Applications* handbook is to provide a connection of specific Mastercam features to standard CNC programming. Mastercam is highly customizable and its user has many ways to influence the final program output. Although much of the output format is handled by customized machine and control definitions, as well as properly configured post processor, it is important to understand various settings in Mastercam functions, many that significantly influence the final program output.

It has been said that to successfully use a computer software to develop a CNC program, one has to understand manual methods of CNC programming. This handbook makes the job a bit easier, as it explains some basic CNC concepts behind various Mastercam functions.

This handbook does not duplicate what you can find in standard Mastercam documentation - it assumes the user is familiar with Mastercam on a very basic level. The handbook is intended for those CNC programmers who want to understand in reasonable depth the various Mastercam settings, the logic behind them, and how they relate to CNC program output.

Exploring Mastercam X3 for 2-1/2 axis Milling Applications is a significant update to the previous version for Mastercam X2. Not only it contains all new features of X3 as they relate to 2-1/2 axis, such as 2D High speed and Feature Based Machining, it also has 27% more overall content. Most previous topics have been expanded and new topics added.

Overall, keep in mind that Mastercam has taken manual CNC programming concepts and turned them into a computerized method of programming. Knowledge of manual CNC methods and applications is absolutely critical to successful Mastercam applications.

Peter Smid
December 2008

psmid@nas.net
(905) 546-1875

About the Author

Peter Smid has been using and teaching Mastercam at all levels since its beginning. He has many years of practical industrial experience with CNC and CAD/CAM applications. He is a professional consultant, educator and speaker, with many years of practical, hands-on experience, in the industrial and educational fields. During his career, he has gathered extensive experience with CNC and Mastercam applications. He consults to manufacturing industry and educational institutions on practical use of Computerized Numerical Control technology, part programming, Mastercam, advanced machining, tooling, setup, and many other related fields. His comprehensive industrial background in CNC programming, machining and company oriented training has assisted several hundred companies to benefit from his wide-ranging knowledge.

Mr. Smid's long time association with advanced manufacturing companies and CNC machinery and software vendors, as well as his affiliation with a number of Community and Technical College industrial technology programs and machine shop skills training, have enabled him to broaden his professional and consulting skills in the areas of CNC and Mastercam training, computer applications and needs analysis, software evaluation, system benchmarking, programming, hardware selection, software customizing, and operations management.

Over the years, Mr. Smid has developed and delivered hundreds of customized educational programs to thousands of instructors and students at colleges and universities across United States, Canada and Europe, as well as to a large number of manufacturing companies and private sector organizations and individuals.

He has actively participated in many industrial trade shows, conferences, workshops and various seminars, including submission of papers, delivering presentations and a number of speaking engagements to professional organizations. He is also the author of technical articles, and since year 2004 has a monthly CNC related column in *ShopTalk Magazine* (www.shoptalkmag.com), a machine shop focused magazine that is distributed in several states of the Great Lakes region. He is also the author of many in-house publications on the subject of CNC applications. During his many years as a professional in the CNC industrial and educational field, he has developed tens of thousands of pages of high quality training materials.

Peter Smid has also written the predecessor to this publication:

Exploring Mastercam X2 for 2-1/2 axis Milling Applications - ISBN13: 9781894487894

Books for both versions are available from:

<http://www.inhousesolutions.com>

<http://www.emastercam.com>

<http://volumesdirect.com/detail.aspx?ID=3716>

Peter Smid is also the author of three hardcover CNC books (and working on the next one):

CNC Programming Handbook, A Comprehensive Guide to Practical CNC Programming
Third Edition - ISBN (978-0-8311-)3347-4

CNC Programming Techniques, An Insider's Guide to Effective Methods and Applications
ISBN-10 (0-8311-)3185-3 or ISBN-13 (978-0-8311-)3185-2

Fanuc CNC Custom Macros, Practical Resources for Fanuc Custom Macro B Users
ISBN-10 (0-8311-)3157-8 or ISBN-13 (978-0-8311-)3157-9

All three hardcover books have been published by *Industrial Press, Inc., New York, NY*, and are also available as eBooks - www.industrialpress.com.

TABLE OF CONTENTS

TABLE OF CONTENTS	<i>i - xii</i>
--------------------------------	-----------------------

Section 1

INTRODUCTION TO MASTERCAM X3	13
MACHINING APPLICATIONS	13
2 and 2-1/2 Axis Capabilities	13
Lathe Capabilities	13
SOFTWARE SPECIFICATIONS	14
Operating System	14
Graphic Requirements	14
Overall Specifications	14
<i>Mastercam X3 - Features</i>	
POINTING DEVICE	15
Mouse	15
<i>Left Mouse Button</i>	
<i>Right Mouse Button</i>	
<i>Scroll Wheel</i>	
GRAPHICAL INTERFACE	15

Section 2

MASTERCAM WORKFLOW	17
Program Development Process	17
Engineering Drawing	17
Material Selection	18
<i>Material Libraries</i>	
Part Setup	18
Tooling Selection	19
Cutting Data	19
Toolpath Geometry	19
Toolpath Verification	19
Program Output	20
Communications	20
Summary of Mastercam Workflow	20

Section 3

WORKING WITH MASTERCAM	21
SELECTING MENU ITEMS	21
Drop Down Menus (Pull Down Menus)	21
Toolbars	22

Status Bar Area	23
<i>Status Bar Settings</i>	23
SYSTEM CONFIGURATION	27
Start / Exit	28
C-hooks	29
Tool Tips and Learning Mode	29
OTHER SETTINGS	30
File Extensions	30
Project Manager	31
Screen	31
Backplot Interface	32
Verify Interface	32
VIEW MENU	32
FILE MENU	33
New	33
Open	33
File Merge / Pattern	34
Edit / Open External	34
Save / Save As	34
Save Some	35
Print	35
Print Preview	35
Import / Export Directory	35
Change Recognition	35
Tracking	35
SCREEN MENU	36
Clear Colors	36
Screen Statistics	36
Display Entity Endpoints	36
Blank / Un-Blank Entity	37
Hide Entity / Unhide Some	37
Screen Grid Settings	37
Shade Settings	37
Regenerate Display List	37
Combine Views	37
Geometry Attributes	38
Toggle Auto Highlighting	38
Copy Screen Image to Clipboard	38
HELP MENU	38
 Section 4	
MACHINE - CONTROL DEFINITIONS	39
FEATURES	40
TYPICAL CONFIGURATION	41

Machine Definition Manager	42
Control Definition Manager	43
Common Settings	43
FINAL COMMENTS	46
Section 5	
OPERATIONS MANAGER	47
OPERATIONS MANAGER WINDOW	47
Operation Descriptions	48
Toolpath Geometry Associativity	50
Display	50
PROPERTIES	50
Files Tab	51
Tool Settings Tab	51
Stock Setup Tab	54
<i>Rectangular Shape</i>	
<i>Display</i>	
<i>Cylindrical Shape</i>	
<i>Stock Origin</i>	
<i>Select Corners</i>	
<i>Bounding Box</i>	
<i>All Entities</i>	
<i>Safety Zone</i>	
OVERALL CONSIDERATIONS	62
Section 6	
MATERIAL MANAGER	63
TOOL SETTINGS - MATERIAL	63
Material Library	64
Cutting Speed (CS)	65
Feed per Tooth	66
Tool Materials	66
Section 7	
TOOL MANAGER	67
TOOLPATH GEOMETRY	67
TOOL MANAGER	67
Tool Library Selection	68
Filtering Tools	69
<i>Tool Types</i>	
<i>Range of Tools</i>	
Edit Tool	72
Create Tool	77

Section 8

TOOLPATH GEOMETRY	79
CREATE MENU	79
LIVE ENTITIES	80
CREATE POINT	81
Position	81
Dynamic	81
Node Points	81
Segment	81
Endpoints	82
Small Arcs	82
CREATE LINE	82
Endpoint	82
<i>Fast Point</i>	
<i>Line Between Two Points</i>	
<i>Multiline</i>	
<i>Polar Line</i>	
<i>Vertical - Horizontal Line</i>	
<i>Tangent Line</i>	
Closest	85
Bisect	85
Perpendicular	85
Parallel	86
Tangent Through Point	86
CREATE ARC	86
Circle Edge Point	86
Circle Center Point	87
Arc Polar	87
Arc Polar Endpoints	87
Arc Endpoints	87
Arc 3 Points	87
Arc Tangent	87
CREATE FILLET	88
Fillet Entities	88
Fillet Chains	89
CREATE CHAMFER	89
Chamfer Entities	90
Chamfer Chains	91
SPLINE	91
RECTANGLE	91
RECTANGULAR SHAPES	92
POLYGON	93
ELLIPSE	93
SPIRAL	94

HELIX	95
LETTERS	96
BOUNDING BOX	96
BOLT CIRCLE	97
Section 9	
EDITING GEOMETRY	99
CREATE FILLET - CREATE CHAMFER	99
EDIT MENU	100
Trimming Methods	101
Break	102
XFORM MENU	102
Translate	103
Mirror	104
Rotate	105
Scale	106
Move to Origin	107
Offset and Offset Contour	107
Offset	
Offset Contour	
Project	108
Rectangular Array	108
XFORM GROUP AND RESULT	110
Group	110
Result	110
Group Manager	110
Section 10	
SELECTING ENTITIES	111
SELECTION OPTIONS	111
General Selection Toolbar	111
All and Only Selection Modes	
Submenu In-Out-Intersect	
SELECTION METHODS	113
Chain	113
Window	113
Polygon	114
Single	114
Area	114
Vector	114
CHAINING METHODS	115
BRANCH POINT	117
CHAINING OPTIONS - MASKING	117

Quick Masks	118
GEOMETRY INTEGRITY	118
Dirty Geometry	118
<i>Red Arrow</i>	
<i>Possible Solutions</i>	
Section 11	
DRILLING TOOLPATH	123
SELECTION OF POINTS	123
Sorting	124
DRILL TOOLPATH	125
Toolpath Parameters	125
Simple drill - no peck	126
PROGRAM STRUCTURE - DRILLING	126
Cycle - Default Settings	127
<i>Description of Fixed Cycles</i>	
G98 and G99 Selection	128
Z-depth Control - Drills	129
Z-depth Control - Spot Drills and Countersinks	132
COLORS AND LEVELS	133
DRILL POINT MANAGER	134
DWELL IN FIXED CYCLES	134
EDITING POINTS	134
Top of Stock Setting	136
SUBPROGRAM OPTION	137
DRILL PATTERNS	139
Grid	140
Bolt Circle	140
Section 12	
CONTOUR TOOLPATH	141
TOOLPATH FOR CONTOURING	141
Toolpath Geometry and Planes	143
TOOLING SELECTION	143
TOOLPATH PARAMETERS	144
Coolant	146
Spindle and Feedrates	146
<i>Spindle direction</i>	
<i>Spindle speed</i>	
<i>Feed rate</i>	
<i>Plunge rate</i>	
<i>Retract rate</i>	

<i>Force tool change</i>	
Feed & Speed Calculator	148
Comment	148
<i>Additional Comments - Manual Entry</i>	
Canned Text	150
CONTOUR PARAMETERS	151
Absolute - Incremental	151
General Data Input	152
Clearance and Retract	152
Feed Plane - Top of Stock - Depth	153
Feed Plane	156
Top Of Stock	156
Depth	157
Other Contour Parameters	157
CUTTER RADIUS COMPENSATION	158
Basic Rules	158
Commands	158
Direction of Cut	158
Machining Modes	159
Restrictions	159
Benefits	160
CUTTER RADIUS COMPENSATION IN MASTERCAM	160
Compensation Type	161
<i>General Approach</i>	
<i>Computer Option</i>	
<i>Control Option</i>	
<i>Wear Option</i>	
<i>Reverse Wear Option</i>	
<i>Off Option</i>	
<i>Stock Allowance</i>	
<i>Overview</i>	
CONTOUR TYPES	167
2D chamfer	167
Ramp	167
Remachining	168
Oscillate	168
Multi Passes	169
What Is A Finish Pass ?	169
XY-axes - Stock to Leave	170
Depth Cuts	170
<i>Keep Tool Down</i>	
<i>Z-axis - Stock to Leave</i>	
Examples and Illustrations	171
<i>Multi passes</i>	
<i>Depth cuts</i>	
Break Thru	172

Section 13

LEAD-IN AND LEAD-OUT	173
LEAD MOTIONS	173
General Settings	174
Line Settings	175
<i>Ramp Height</i>	
Arc Settings	175
<i>Radius</i>	
<i>Helix height</i>	
EFFECT OF LEAD SETTINGS	176
OTHER SETTINGS	178
Gouge check entry/exit motion	178
Overlap	179
Use Entry Point	179
Use Exit Point	179
Use Point Depth	180
Enter on first depth cut only	180
Exit on last depth cut only	181
Plunge after first move	181
Retract before last move	182
Override feedrate	182
Adjustments	182

Section 14

POCKET TOOLPATH	183
POCKET GEOMETRY	183
Pocket Evaluation	184
Tool Selection	184
Summary	184
TOOLPATH PARAMETERS	185
POCKETING PARAMETERS	185
Depth Cuts Distribution	186
<i>Other settings</i>	
Pocket Type	187
Stock Allowance Options	187
Pockets with Islands	188
ROUGHING/FINISHING PARAMETERS	188
Cutting Method	189
Stepover - Width of Cut	190
Other Rough Pocket Settings	190
<i>Entry - helix</i>	
Finish Pocket Settings	192
Remachining	193
Open Pockets	194

Section 15

FACE TOOLPATH	195
USING CONTOUR TOOLPATH	195
Stock Setup	196
Toolpath and Contour Parameters	196
USING FACE TOOLPATH	197

Section 16

SPECIAL TOOLPATHS	199
ENGRAVING	199
CIRCLE PATHS	199
Circmill	199
Thread Mill	200
Auto Drill	203
Start Hole	203
Slot Mill	203
Helix Bore	207
TRANSFORM	207
Translate	207
Rotate	208
<i>Example</i>	
Mirror	210
NESTING	211
MANUAL ENTRY	211
POINT	212
TRIM	212
2D HIGH SPEED	213
HSM - Brief Background	213
2D HIGH SPEED MENU	214
Core Mill	215
Peel Mill	215
Blend Mill	215
Area Mill	216
Rest Mill	216

Section 17

CHECKING FOR ACCURACY	217
ANALYZE MENU	217
Entity Properties	218
Position	219
Distance	219
Area / Volume	220
<i>Properties of a Circle</i>	

Chain 223

Contour 224

Angle 225

Dynamic 225

Toolpath Analysis 225

BACKPLOT 226

 Details Tab 226

 Info Tab 227

VERIFY 227

 Display Control 228

Moves / step

Moves / refresh

Speed vs. Quality Slider

Update after each toolpath

 Stop Options 229

Stop on collision

Stop on tool change

Stop after each operation

 Icon Section 230

Verbose

Other Functions

 Configuration Options 232

Stock

Boundaries

Initial stock size source

Cylinder axis

Translucent stock

Miscellaneous options

Tool profile

Tolerances

Set Colors

CONFIGURATION FILE 236

Section 18

VIEWS, PLANES AND WCS..... 237

GRAPHICS WINDOW 237

COORDINATE SYSTEM 238

VIEWS 239

 Gview 239

 Cplane 239

 WCS 239

 Seven Standard Views 239

WORK COORDINATE SYSTEM - WCS 240

 Work Offset Numbers 240

 Application Example 241

WCS AND TOOL PLANES 244

Planes Button	245
Section 19	
CUSTOMIZING MASTERCAM	247
PURPOSE OF CUSTOMIZATION	247
CUSTOMIZATION OPTIONS	247
Appearance	248
<i>Toolbars</i>	
<i>Colors and Other Attributes</i>	
Defaults	249
<i>Operation Defaults</i>	
CUSTOMIZING TOOLBARS	250
New Toolbar Development	251
<i>Summary</i>	
SETTINGS MENU	253
Key Mapping	253
Macro Manager and VBScript	254
Section 20	
POST PROCESSING	255
WHAT IS POST PROCESSING?	255
DEFAULT POST PROCESSOR	256
CUSTOMIZING POST PROCESSOR - Evaluation	257
Initial Steps	257
Sample Program	257
<i>What to include ?</i>	
Post Development Test	259
CUSTOMIZING POST PROCESSOR - Identifying Changes	262
Changes Related to Contents	263
Changes Related to Appearance	263
Changes Related to Structure	264
<i>Breaking Arcs</i>	
<i>Arc Tolerances</i>	
<i>Arc Control Restrictions</i>	
<i>Enable / Disable Program Number</i>	
<i>Enable / Disable Sequence Numbers</i>	
CUSTOMIZING G-CODES AND M-CODES	267
FORMAT STATEMENTS	268
Adding a New Format Statement	271
COMMENTS IN POST PROCESSOR	271
PROGRAM OUTPUT - Customized Post	272
SUGGESTIONS FOR POST PROCESSING	274

Section 21

FEATURE BASED MACHINING	275
BASIC CONCEPTS	275
FBM IN MASTERCAM X3	275
Part Definition	276
Automated Tasks	276
FBM DRILL	276
FBM Drill - Selections	277
FBM Drill - Toolpaths	278
<i>Automatic Program Generation</i>	
<i>Z-depth for Spot Drill</i>	
<i>Z-depth for Drill</i>	
<i>Z-depth for Counterbore</i>	
Conclusion	280
FBM MILL	281
FBM MILL - Comments	283
<i>Mill and Drill Together</i>	
MODIFICATION OF DEFAULTS	284
Drill	284
Mill	285

Section 22

PROGRAMMING REFERENCES	287
SPEEDS AND FEEDS - BASIC CHARTS	287
Descriptions and Tables	288
Surface Speed	289
CENTER DRILL DATA	289
SPOT DRILL DATA	290
DRILLING DATA	291
Through Holes	292
Blind Holes	293
DWELL TIME CALCULATION	293
PREPARATORY COMMANDS - G-CODES	294
MISCELLANEOUS FUNCTIONS - M-CODES	294
INDEX	297

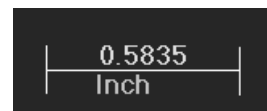
18

VIEWS, PLANES AND WCS

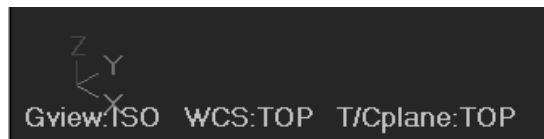
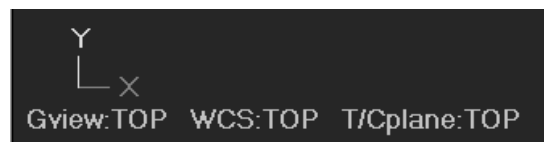
GRAPHICS WINDOW

When you open Mastercam, the large screen area that appears is used for graphics work. This is called the *Graphics Window* and it represents the main Mastercam workspace. Many previous examples have used this area without any special mention. The *Graphics Window* shows more than just the toolpath geometry and pop-up windows - it also shows certain status displays of its own. Although it appears empty at a casual view, there are two sections at the screen bottom that should be the focus of some interest.

The *lower right corner* shows an indicator of the dimensional units currently active (Inch or Metric) as well as the scale represented by the displayed horizontal line. In the illustration, the horizontal line represents 0.5835 units, in inches. The purpose of this information is to provide a visual idea of the part size.



The *lower left corner* is more comprehensive and is directly related to the subject of this chapter. Its purpose is to show the axis orientation with the currently active graphic view (*Gview*). Each axis has a different color, for better distinction - X-axis is red, Y-axis is green, and Z-axis is blue. Mastercam uses an interesting name for the axis orientation icon - '*gnomon*'.



Below the axis icon is shown the current status of aligned views:

- **Gview** ... View selected for best visual appearance of geometry
- **WCS** ... Work Coordinate System currently active
- **Cplane** ... Plane in which toolpath geometry will be defined (created)
- **Tplane** ... Toolpath cutting plane (perpendicular to the tool axis)

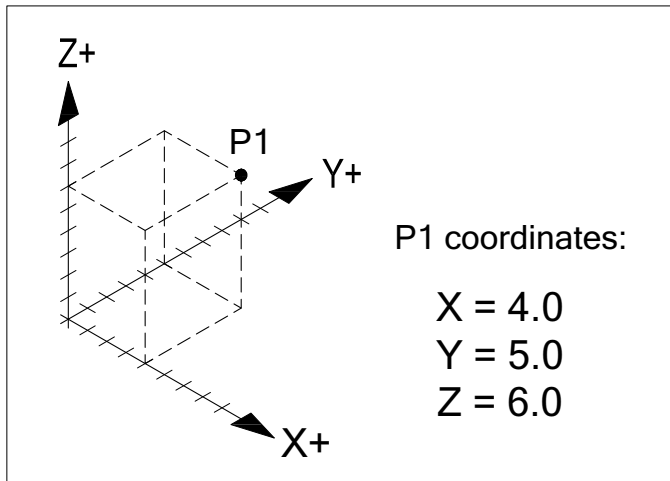
A couple notes - The *Gview* (Graphic View) is strictly a true view for visual purposes only and should not be confused with *Cplane* (Construction Plane) or *Tplane* (Tool Plane), which are used for geometry creation purposes. Mastercam sometimes refers to a plane as a view, which can lead to some confusion. Also note that there will be no *Tplane* displayed if Mastercam is in *Design* mode.

In 2-1/2 axis machining, tool plane (*Tplane*) is hardly used. It does have its advantages for machines with four axes, such as XYZ and A or XYZ and B.

In order to understand Mastercam views and planes, it is important that you understand at least the very basics of *Rectangular Coordinate System* - also known as the *Cartesian Coordinate System*. In brief, the coordinate system defines a part location in space.

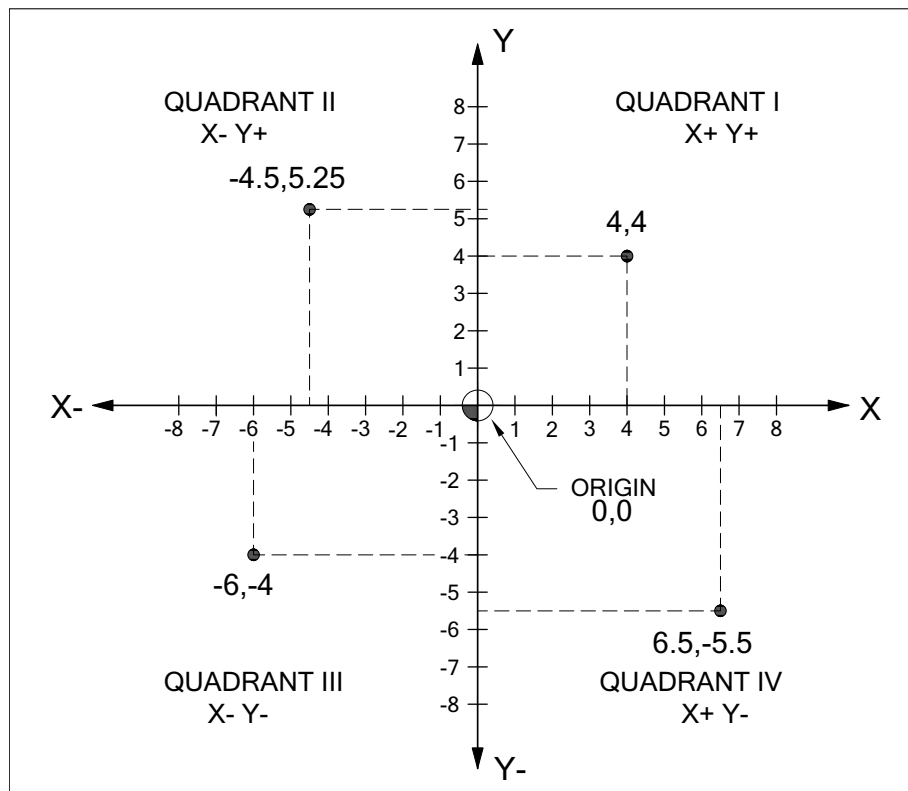
COORDINATE SYSTEM

The *Cartesian Coordinate System* is named after a French philosopher and mathematician René Descartes (1596-1650). His greatest achievement in mathematics was to describe geometrical relationships using algebra. **In CNC, we use this method to define a point.**



Based on the intersection of two perpendicular lines (called axes) that are oriented orthogonally (*i.e.*, only *vertically* or *horizontally*), any point can be defined on a *plane* (2D or two-dimensional) or in *space* (3D or three-dimensional). The point of intersection is called **origin**.

All modern CAD, CAM and CNC technology is based on this almost four hundred year old concept.



Both illustrations on this page show definition of a point within the *Cartesian Coordinate System* (also known as the *Rectangular Coordinate System*). Each point in this system of coordinates is defined by its distance from origin and located within one of four quadrants; each coordinate (XYZ) is always measured along an axis. Note *positive* and *negative* direction of each axis and the distribution of quadrants.

VIEWS

Gview

Graphic View - **Gview** - allows viewing or looking at a particular geometry in a convenient way. Geometry defined in any *Cplane* will always be based on the current construction plane (*Cplane*) - not the current graphic view (*Gview*).

Select **Gview** for viewing - select **Cplane** for construction

Cplane















Cplane means construction plane. The word *construction* refers to development of any toolpath geometry in the form of points, lines, arcs, etc. For 2-1/2 axis machining, the *Cplane* is almost always set to **TOP**, although the *Gview* mode can be set as required.

WCS

WCS is an abbreviation for *Work Coordinate System*. In CNC programming, *Work Coordinate System* is a method of changing part origin in the same program. This method will be shown later in this chapter, using a typical example.

Seven Standard Views

There are seven predefined (standard) views in Mastercam, accessible from the status bar and toolbars. The two illustrations shown below the difference between the *Gview* (left) and *Cplane* (right) menu selections:

 Top Gview	Alt+1	 Top (WCS)
 Left Gview	Alt+6	 Front (WCS)
 Back Gview	Alt+3	 Back (WCS)
 Front Gview	Alt+2	 Bottom (WCS)
 Right Gview	Alt+5	 Right (WCS)
 Bottom Gview	Alt+4	 Left (WCS)
 Isometric Gview	Alt+7	 Isometric (WCS)

As both selections are independent from each other, it is possible to combine them to provide convenient and efficient working environment. For example, you can use *Gview* set to *Isometric* while the geometry construction takes place in *Cplane TOP*.

Any view can be saved in the MCX file using a meaningful name, to make later selections easier. User generated views can also be named and saved within the MCX file.

Gview can be selected from the status bar, pull-down menu *View > Standard Views*, or toolbar *Graphics Views*. *Cplane* can also be selected in a similar way.

WORK COORDINATE SYSTEM - WCS

The WCS - *Work Coordinate System* - is used to represent origin and orientation of a part located on the machine for a specific setup. If a move to another face of the part is based on a new setup, a new WCS is used. If a move to another face of the part is to be interpreted as a rotary or index move on the machine, the original WCS should be maintained, and a new Toolplane should be used.

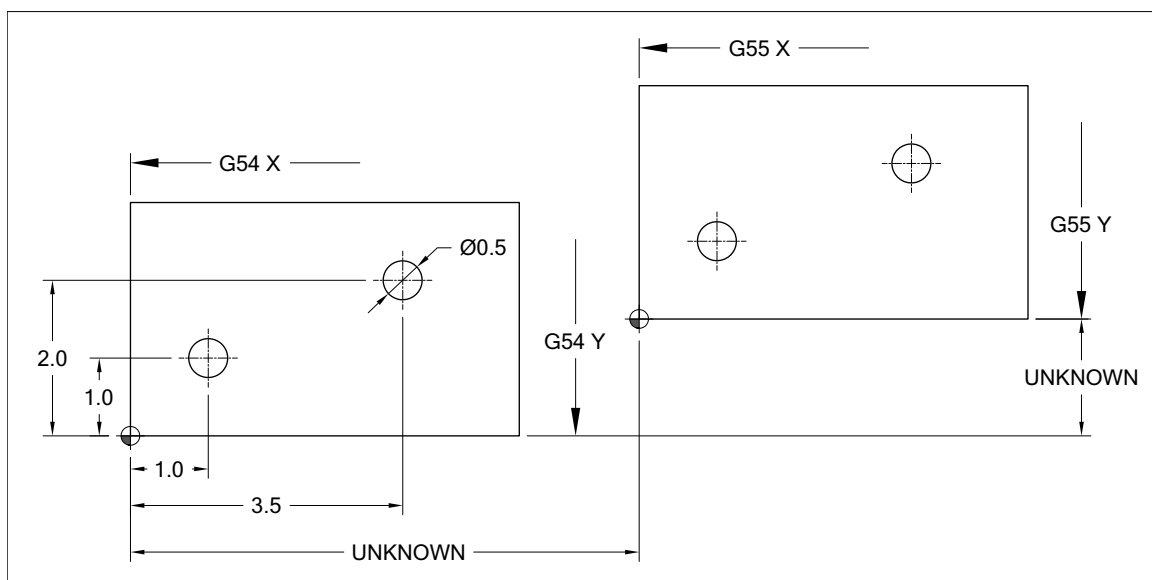
Shifting the coordinate axes allows you to align the coordinate system and origin to the part rather than manipulation of part geometry. WCS should only be used to reflect different physical setups on the machine.

The work coordinate system should not be used for rotating coordinate system for horizontal milling, for example. Although not applicable to a 3-axis machining center, it is a good example where a suitable machine definition should be used instead.

Work Offset Numbers

An example in the illustration is a typical application of changing work offset for two identical parts set on the machine table. On most Fanuc and similar controls, there are six standard work offsets - G54-G59. On Fadal, the format uses E1, E2, etc. Proper output will be determined by the post processor.

Each work offset (fixture offset) can be used to set part zero for up to six parts or more - up to additional 48 offsets on Fanuc, if the extra option is available.



At the machine, the CNC operator will set G54 to the origin of the left part and G55 to the origin of the right part. Note that during programming process, the XY dimensions between the two origins are not known.

The output will use the same XY coordinates with *two different* work offsets. The coordinates can be stored as a subprogram or be part of the main program. Mastercam requires a generic format that is common to all machines and controls, and the specific code is configured in the post processor.

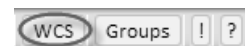
To illustrate this concept, the following description will be applied to the example drawing illustrated on the previous page.

Application Example

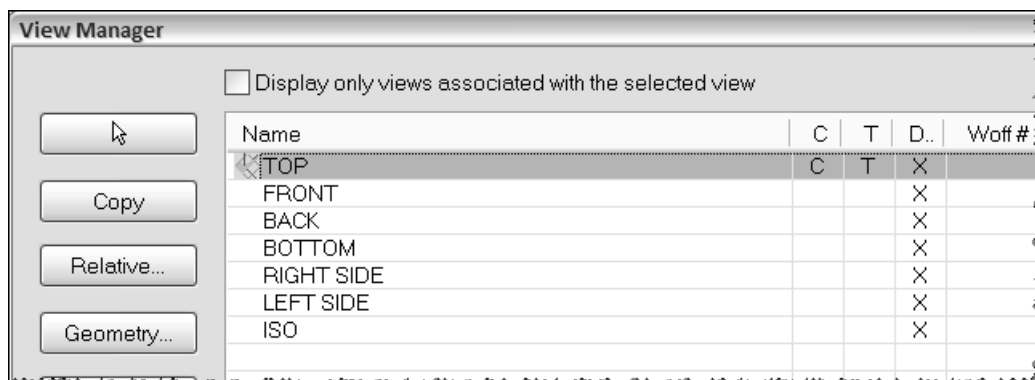
There are two identical parts set on the machine table. Each is mounted in a machine vise, and part origin (X0Y0) is at the lower corner of each plate. G54 command is required for the left part, G55 command for the right part. During programming, the distance between the respective part origins is *unknown*.

In Mastercam, each part origin has to be assigned a different work offset. During toolpath development, the generic offset numbers will be set, to be output as G54 and G55 respectively, by any Fanuc type post processor. Physical setup of G54/G55 origins takes place at the CNC machine.

Start from the status bar at bottom of the screen display and select WCS as the selection:

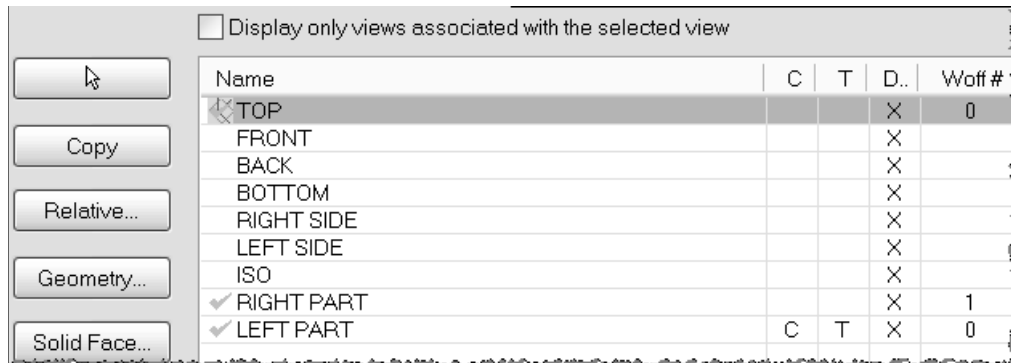


Click on the *WCS* item in the *Status Bar* area, then select *View Manager*, in order to open the default dialogue box:



A new view for each vise (part origin) will be defined next. In 2-1/2 axis machining, *TOP* view is the standard working view, and the two new views will be defined by copying the current top view, using the *Copy* function at left of the view names.

Each new view should have a unique name for better organization. For this example, the new views will be called *LEFT PART* and *RIGHT PART*. Generic offset number for each vise (part origin) also have to be entered as attributes - *0* for the left part, and *1* for the right part (see the next page):

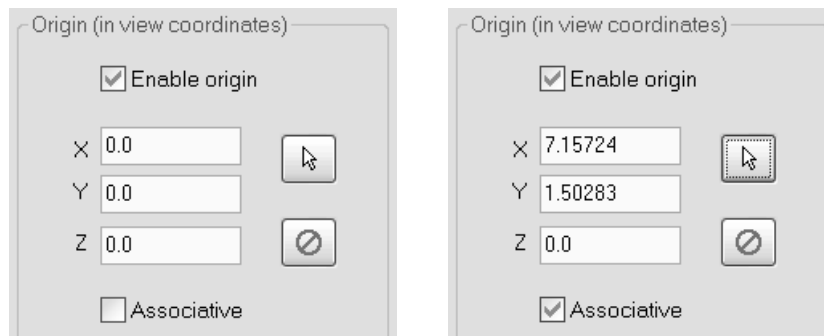


As each view has to have its origin defined, the last step is to use the small *Select* button and pick the point of origin on each vise. In the example, point of origin will be the lower left corner of each part. Mastercam will determine the actual location of the two origins as drawn.



Keep in mind that the position of parts in Mastercam does not necessarily reflect their actual position on the machine

For the example, you can check Mastercam settings of the two origins in view coordinates:



LEFT PART

RIGHT PART

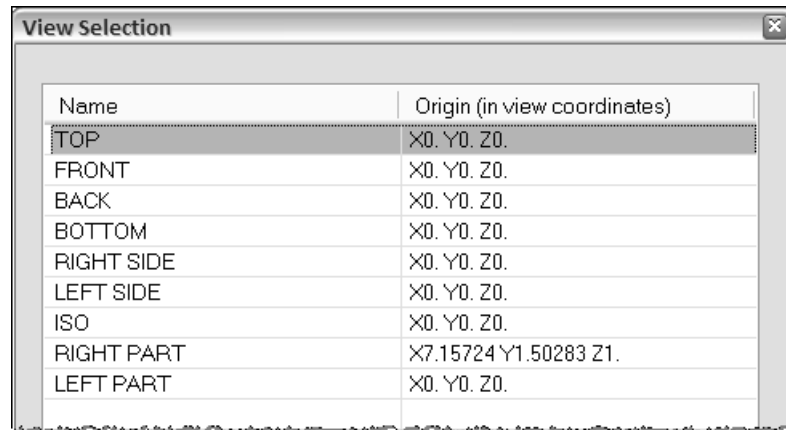
So far, only the necessary settings have been done. The actual application of each work offset takes place during toolpath development.

A spot drill is required for a set of two holes located on two parts. Although the machining is identical for both parts, **two operations** have to be created in order to apply the offsets.

From the *Planes* button located on the status bar, select *Named Views* to see the current *View Selection* (see next page).



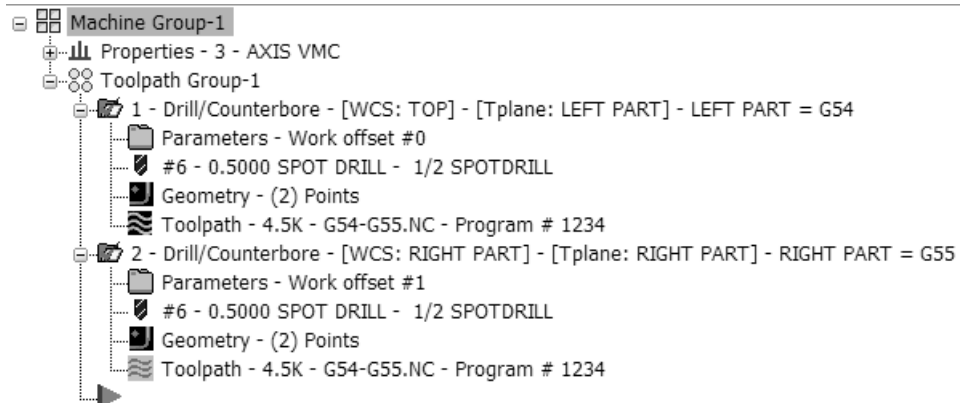
Select the view you want to define a machining operation for, such as LEFT PART. Define toolpath for the two holes on the left part with all parameters. For the right part, select view RIGHT PART and define toolpath for the other two holes, again, with all parameters.



Name	Origin (in view coordinates)
TOP	X0. Y0. Z0.
FRONT	X0. Y0. Z0.
BACK	X0. Y0. Z0.
BOTTOM	X0. Y0. Z0.
RIGHT SIDE	X0. Y0. Z0.
LEFT SIDE	X0. Y0. Z0.
ISO	X0. Y0. Z0.
RIGHT PART	X7.15724 Y1.50283 Z1.
LEFT PART	X0. Y0. Z0.

Using the *MPMASTER.PST* as the post processor (or any other properly configured post processor), the output will be as desired. Note that absolute coordinates for both sets of holes are identical. Also note they are under two different work offsets - G54 for the left part, and G55 for the right part. For more than two parts located on the machine table, the process itself does not change - only more work offsets will be defined.

In the following illustration is the listing of both spot drilling operations as they appear in the *Operations Manager*.



The following program listing may be somewhat different, depending on the post processor but the critical XY coordinates and the plane commands should be the same. Comments have been removed for better clarity:

```
O1234 (G54-G55)
(** GENERAL COMMENTS REMOVED **)
N1 G17 G20 G40 G80
(LEFT PART = G54)
N2 T6 M06
N3 G00 G90 G54 X1. Y1. S1200 M03
N4 G43 H6 Z.1 M08
N5 G99 G82 Z-.14 R.1 P.2 F8.
N6 X3.5 Y2.
N7 G80
```

```

(RIGHT PART = G55)
N8 G55 X1. Y1. Z.1
N9 G99 G82 Z-.14 R.1 P.2 F8.
N10 X3.5 Y2.
N11 G80
N12 M09
N13 M05
N14 G91 G28 Z0.
N15 G90
N16 M30
%
```

In blocks N3 and N8, as well as in blocks N6 and N10, the same coordinates are used - they refer to the XY location of the first hole in each setup. Since the two parts are identical, the XY locations will also be identical, but within two work offsets.

WCS AND TOOL PLANES

This chapter was focused on the WCS settings with their typical application for 2-1/2 axis work. In order to understand some basic items used in the example, the most important one is assignment of the generic offset numbers.

Default value of -1 can be considered something of an auto mode. In this case, Mastercam will try to make an 'intelligent' choice. Of course this choice may not be your choice, so it is much better to input a generic value that will be interpreted by the post and outputs proper work offset:

- -1 = Auto
- 0 = G54 for Fanuc or E1 for Fadal
- 1 = G55 for Fanuc or E2 for Fadal
- 2 = G56 for Fanuc ...
- 3 = G57 for Fanuc
- 4 = G58 for Fanuc
- 5 = G59 for Fanuc

In order to make these offsets work, it is important to set the type of offset in the control definition dialogue box, under the *Work System* heading.

Work offset auto selection is shown as the most common:



WCS is not suitable for all coordinate system changes, so Mastercam also provides *Tool Planes* for special setting applications. When deciding whether to use *WCS* or *Toolplane*, understand a few simple rules:

- ***Tool Plane* is defined as a cutting plane for a given toolpath and is commonly used to switch between typical four-sided tombstone sides on a horizontal machining center:**

***Tool planes FRONT, RIGHT SIDE, BACK, and LEFT SIDE
will be used in relationship to Work Coordinates System (WCS)***

- ***WCS* should always be used for toolpaths applied in a single setup**
- **For multiple fixtures (work offsets) needed in a single setup, use *WCS***
- **Rotary and 5-axis applications have their own requirements (not covered in this book)**

Planes Button

When you select *Operations Manager*, you can change an existing work offset number for any selected operation. For example, you may want to change the current G55 offset for the right part to a new G58 offset. That means changing the generic Mastercam number from the current setting of 1 to a new setting of 4.

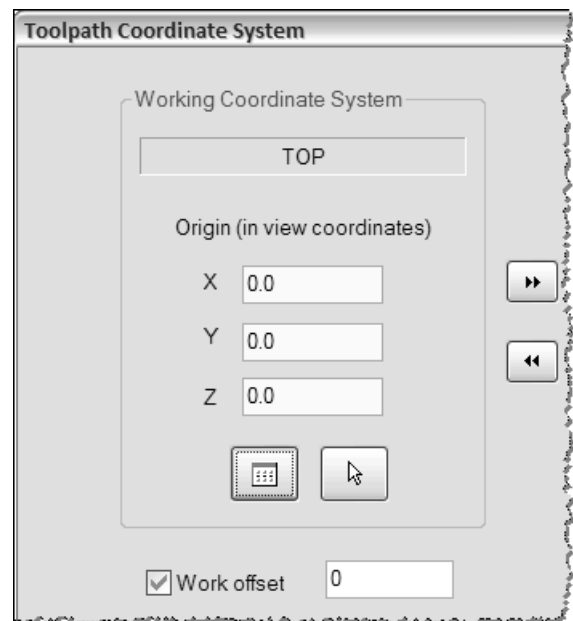
This can be easily accomplished from the *Planes* button, located on the *Toolpath parameters* page (tab) of the operations *Parameters*.



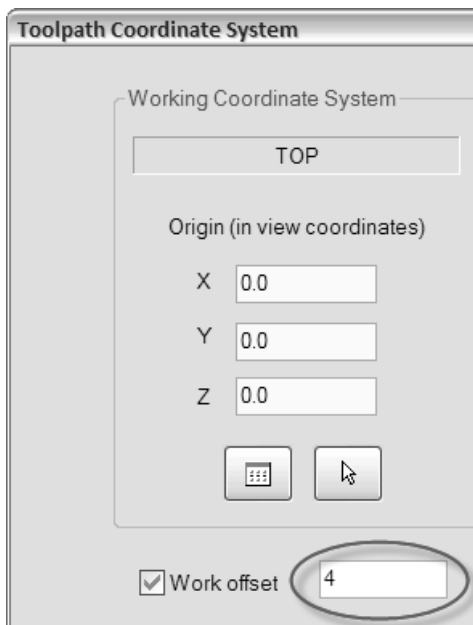
Mastercam will open *Toolpath Coordinate System* dialogue box that has two or three windows, depending on the active offset:

The first window - as loaded (right):

Note that the setting is exactly the same as defined through the view manager - nothing has been typed in and a new selection had been made.

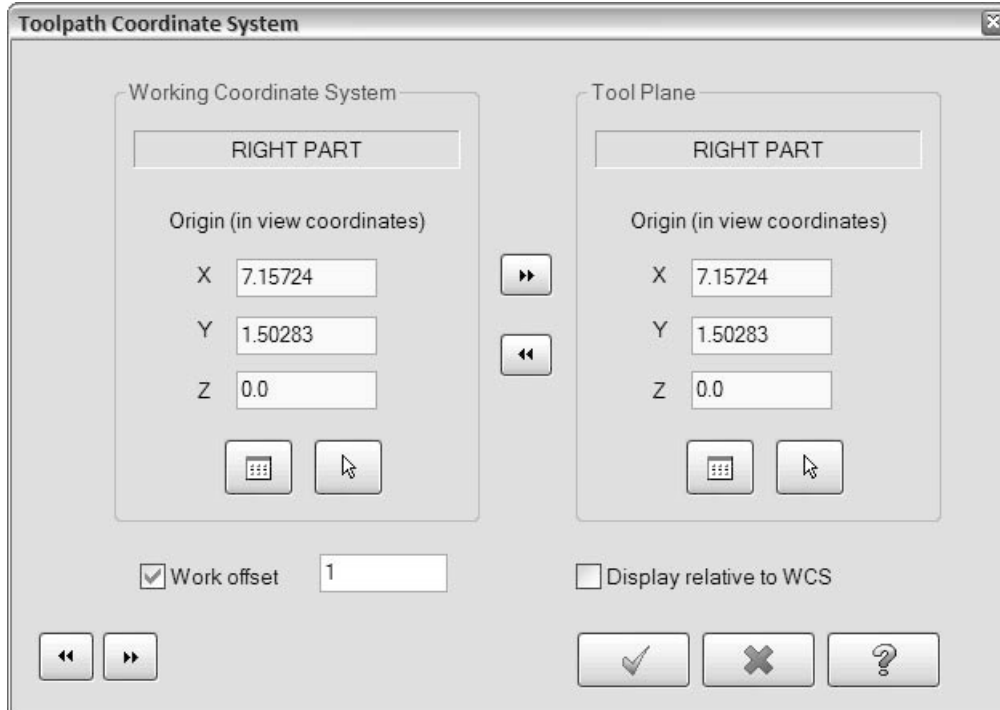


The first window - after change:



This is the window where you will change the work offset number from 1 to 4.

There are two other windows that do not normally change:



Once the change has been made (work offset 4), the operation has to be regenerated, and a new settings post processed (with a new comment).

The following excerpt shows the change:

```
O1235 (G54 AND G58)
(** GENERAL COMMENTS REMOVED **)
N1 G17 G20 G40 G80
(LLEFT PART = G54)
N2 T6 M06
N3 G00 G90 G54 X1. Y1. S1200 M03
N4 G43 H6 Z.1 M08
N5 G99 G82 Z-.14 R.1 P.2 F8.
N6 X3.5 Y2.
N7 G80
(RIGHT PART = G58)
N8 G58 X1. Y1. Z.1
N9 G99 G82 Z-.14 R.1 P.2 F8.
N10 X3.5 Y2.
N11 G80
N12 M09
N13 M05
N14 G91 G28 Z0.
N15 G90
N16 M30
%
```

Other settings and/or changes can be made from the *Planes* button
The one shown is most common for 2-1/2 axis mill work