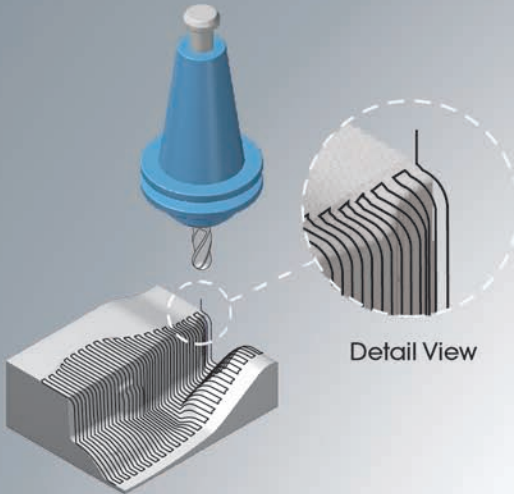
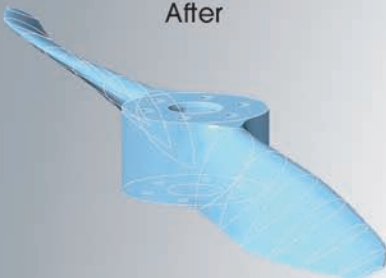


Before



Detail View

After



X³

Handbook Volume 2

MILL Level 2 & 3

Mastercam[®]

When Second Best Won't Cut It[®]

Mastercam Handbook Volume 2 for Mastercam X3

Date: September 16, 2008

Copyright © 1984 - 2008 In-House Solutions Inc. - All rights reserved.

Software: Mastercam X3 Mill Level 3 & Solids

Authors: In House Solutions

ISBN: 978-1-926566-09-2

Notice

In-House Solutions Inc. reserves the right to make improvements to this manual at any time and without notice.

Disclaimer Of All Warranties And Liability

In-House Solutions Inc. makes no warranties, either express or implied, with respect to this manual or with respect to the software described in this manual, its quality, performance, merchantability, or fitness for any particular purpose. In-House Solutions Inc. manual is sold or licensed "as is." The entire risk as to its quality and performance is with the buyer. Should the manual prove defective following its purchase, the buyer (and not In-House Solutions Inc., its distributor, or its retailer) assumes the entire cost of all necessary servicing, repair, of correction and any incidental or consequential damages. In no event will In-House Solutions Inc. be liable for direct, indirect, or consequential damages resulting from any defect in the manual, even if In-House Solutions Inc. has been advised of the possibility of such damages. Some jurisdictions do not allow the exclusion or limitation of implied warranties or liability for incidental or consequential damages, so the above limitation or exclusion may not apply to you.

Copyrights

This manual is protected under the copyright laws of Canada and the United States. All rights are reserved. This document may not, in whole or part, be copied, photocopied, reproduced, translated or reduced to any electronic medium or machine readable form without prior consent, in writing, from In-House Solutions Inc.

Trademarks

Mastercam is a registered trademark of CNC Software, Inc.

Microsoft, the Microsoft logo, MS, and MS-DOS are registered trademarks of Microsoft Corporation; Windows2000, Windows XP, are registered trademarks of Microsoft Corporation.

This document complies with Mastercam-X3 as of Sept, 2008. Requires Mastercam Mill Level 3 and Solids.

Contents

Introduction

Chapter 1	3D Workspace
Chapter 2	Splines
Chapter 3	Surface Modeling
Chapter 4	Surface Rough Machining
Chapter 5	Surface Finish Machining
Chapter 6	Surface High Speed Toolpaths
Chapter 7	Mold Project
Chapter 8	Prototype Project
	Index
App.	CNC Programming Planning Sheet
App.	Drill Chart

Introduction

Congratulations on your purchase of the Mastercam Handbook Volume 2. This book was developed and tested over several years as part of a course to teach machinists how to program CNC machine tools with Mastercam. It assumes you have a working knowledge of machining and CNC's, including tooling, work-holding, and common G&M codes. You should have already completed the Mastercam Handbook Volume 1 or have equivalent training or experience.

The Handbook can be used as the primary resource for a Mastercam class, as a self-study guide, or a shop reference. Mastercam help functions and information on the student CD supplement the material in this book.

A unique feature of this book is the comprehensive, top-down approach it takes to learning. Concepts and essential knowledge are included along with practical applications. This approach means you not only learn how to use Mastercam, but why things are work as they do.

By understanding exactly what you are doing and why each step of the way, you learn to recognize the best approach to problems. Not only will you be better prepared to work effectively, you will gain a solid foundation of knowledge that will help you continue to learn and adapt as technology changes.

Specific recommendations about how to best use Mastercam are included. There are often many ways to accomplish any task. However, you will learn faster and understand the overall picture of what you are doing if you first master fundamentals and standard practices.

In any case where information in this book conflicts with your machine manuals or the methods used at your company or school, ignore the suggestions in this book and use the information in the manuals and established procedures at your facility.

Never operate a CNC Machine without having read and understood the operator and programmer manual, and having received safety and operator training by a qualified person on that machine.



Warning

Contents

This book is divided into ten chapters. Each covers a specific knowledge area. Following is a breakdown of the chapters and what you will find in each:

Chapter 1: 3D Workspace explains the Mastercam 3D work environment.

Chapter 2: Splines shows how to create and machine splines. Spline modeling and machining concepts are an essential foundation for surface modeling and machining.

Chapter 3: Surface Modeling shows how create and modify all types of surfaces in Mastercam.

Chapter 4: Surface Rough Machining shows how remove excess material from a surface or solid model in preparation for finish machining.

Chapter 5: Surface Finish Machining shows how to finish machine surface and solid models.

Chapter 6: Surface High Speed Toolpaths introduces concepts and principles of high speed machining and how to use Mastercam's advanced surface high speed rough and finish toolpaths.

Chapter 7: Mold Project introduces basic mold concepts and terminology. A simple two piece working plastic injection mold is made starting with prints.

Chapter 8: Prototype Project is an assignment that starts with a complex CAD surface model. The model is prepared for machining and then rough and finished machined.

App. A: NC Program Planning Sheet is a template used document the machining process.

App. B: Drill Chart

Icons are used to alert, inform, and enhance your learning experience. The following icons are found in the margins of the book:

Icons

Remember reminds you of important information that will help you work safely and productively.



Remember

Tips are suggestions from experienced CAD/CAM users that will guide your learning and use of Mastercam.



Tip

Step by Step are detailed instructions on how to use a specific function or perform a task.



Step by Step

On The CD alerts you that a file exists on the CD included with this manual that may be necessary for accomplishing a task.



On The CD

Try It assigns a task you should be able to successfully complete before proceeding further.



Try It

Warning is used to emphasize situations that can cause damage to machines, property, bodily injury or death. Machining can be dangerous. Take these warnings seriously and do not proceed unless you are certain your methods and setup are completely safe.



Warning

In Depth are notes of interest that deepen your understanding and knowledge of a topic.



In Depth

Power User denotes tips that are likely only applicable to the highest level users of Mastercam.



Power User

Conventions

Key words and Mastercam menu items are shown in **bold** the first time they are used. Columns on the outside edges of each page and note pages at the end of each chapter provide ample space for taking notes.

Useful tips, recommended settings, best practices, and detailed instruction on the most important features are included when possible.

Extra credit exercises are included on the student CD in PDF format. These will help build your skill to a higher level.

The following terms are used throughout this book.

Left Click means to click once on the left mouse button.

Click means the same as left click.

Right Click means to click once on the right mouse button.

Scroll means to roll the mouse scroll wheel, or move the scroll index in a list.

Options are Mastercam functions selected from the main menu.

Enter means to select the <**Enter**> key on your computer keyboard.

Press means to press on a keyboard key.

Choose means to select a menu option or button.

Open/Close means to open or close a dialog or information box.

Dialog Box is a window that opens to allow input of information and setting of defaults.

Drop Down/Fly Out Menus are menus that expand down, left, right, or up, to reveal more menu lists.

A **Function** is the same as a menu option or command.

Help means the Mastercam help files loaded with your software.

4 Surface Rough Machining

This chapter shows how to remove excess material from complex multi-surface models in preparation for finish machining operations. Upon completion of this chapter, you should be able to do the following:

Objectives

- Know common 3D Machining definitions.
- Understand the importance of careful organization and planning for 3D machining jobs.
- Understand factors related to tool selection and speeds and feeds for surface machining.
- Understand the importance of surface data integrity.
- Understand the difference between programming the Tip or Center of the tool.
- Understand how different cutter shapes affect coordinate data output.
- Know how to prepare a surface model for machining, including the use of tool containment boundaries, plunge points, and runoff surfaces.
- Be able to successfully select and use all Mastercam Surface Roughing functions.

Surface roughing functions are used to remove excess stock material from a part before finish machining. The objective is to clear out excess material as quickly and efficiently as possible. This may require the use of one, or many, roughing functions.

Introduction

In most cases, after roughing the part should have a uniform amount of stock remaining all over, allowing the finish pass to remove a constant thickness of material. This produces a more precise finished part and reduces the possibility of tool breakage.

Mastercam offers several surface roughing strategies. Each works in a distinct and specific way, and is best suited to certain shapes.

Many factors are involved with selecting and creating efficient and effective 3D tool paths, including the shape of the part, available fixturing and tools, and CNC machine limitations. Because of the large variation in these factors, it is difficult to establish set strategies and rules. Yet it is helpful to discuss some overall goals and parameters to provide a reference for decision-making.

Remember though that every situation is unique. Surface roughing can be among the most challenging of all CNC Programming operations, and it requires knowledge, skill, and creativity.

Whatever tactics and strategies you use, always keep in mind the overall goal: to produce a part that meets or exceeds all customer's requirements for dimensional accuracy, surface finish, delivery date, and cost without risking injury or damaging the machine, tools, or equipment.

3D Machining Definitions

You should be familiar with the following terms and definitions related to 3D surface machining.

Term	Definition
Ball Nose Mill	A full radius cutter (cutting end is spherical).
Boss	A shape, or part of a shape, that protrudes up from the part.
Bull Nose	Similar to a flat end mill but with a corner radius.
Cavity	The female shape of a mold. Plastic injection molds often have both a cavity and core. Hot plastic is injected at very high pressure into the space between the cavity and the core of a mold to create a part.
Core	The male shape of a mold.
Check Surfaces	Surfaces "tagged" to be avoided by a roughing or finish 3D tool path.
Chordal Deviation	How accurately a surface or spline follows the specified path.
Cut Tolerance	How accurately the tool path follows the surface shape.
Draft	Taper angle on the sides of a part, typically from 1-5 degrees. Most molds have draft on the near vertical walls so the part can be easily separated from the mold.
Drive Surfaces	The surfaces that are being machined.
End Mill	A milling tool with a flat bottom.

High Speed Machining	A set of functions that automatically adjusts cutting feed, smoothes transitions between paths, reduce code file size, or otherwise improves machine speed and performance.
Runoff Surface	A surface or surfaces not part of the actual part, but used to force the tool to cut a floor or extended area beyond the part shape.
Scallop Height	The height of the cusps left between surface cutting paths.
Stepover	How far the cutter moves between cutting passes, similar to pocket and stepover distance. Stepover may be expressed as a fixed distance or as a scallop height.
Stock Allowance	Also known as Stock to Leave. The amount of stock left on the part after the machining operation. Both positive and negative stock allowances may be used. Positive values leave additional stock on the part. Negative values over-cut the part by the amount specified.
Tapered End Mill	A flat or bull end mill with tapered flutes; usually from 1-5 degrees. These are used to machine a constant draft angle on the walls of a part.
Tip Comp	Determines if the NC code coordinates represent the tip or center of a bull or ball mill. Tip is by far the most common setting.
Tool Center Boundary	These are “containment” profiles (lines, arcs, splines) used to contain 3D tool paths.

**Organization
And Planning**

Before programming a surface part, take time to organize and plan your work. The importance of planning cannot be overstated. Unless the model is extremely simple, you are not going to be able to “wing it” in the 3D surface machining world.

Check the part print carefully. Pay particular attention to dimensions and surface finish requirements. If a 3D CAD data file is provided, use Shading to help identify all part features. Also, use **Analyze, Surfaces** to determine critical design parameters, such as depths and minimum internal radii. You may even want to create some “trial” tool paths before settling on an operation sequence, or even before quoting the job.

Use a written process plan. A sample NC Program Planning Sheet is included in Appendix A. A Word® file of this sheet is included on your student CD. The process plan should include important machining parameters such as tool lengths, diameters, speeds and feeds. Check to see if the tooling selected is readily available. Special tools, such as extended length cutters, may have to be special ordered and can affect part delivery dates.

NC Program Planning Sheet												
Part Number: _____			Part Name: _____				Date: _____					
Page _____ of _____												
OP NO 1	CUT TYPE	TOOL #	TOOL TYPE	DIA.	LENGTH	FLUTE LEN	CORNER RAD	SPEED	FEED	STEP OVER	STEP DOWN	STOCK LEFT
OP NO 2	CUT TYPE	TOOL #	TOOL TYPE	DIA.	LENGTH	FLUTE LEN	CORNER RAD	SPEED	FEED	STEP OVER	STEP DOWN	STOCK LEFT
OP NO 3	CUT TYPE	TOOL #	TOOL TYPE	DIA.	LENGTH	FLUTE LEN	CORNER RAD	SPEED	FEED	STEP OVER	STEP DOWN	STOCK LEFT
OP NO 4	CUT TYPE	TOOL #	TOOL TYPE	DIA.	LENGTH	FLUTE LEN	CORNER RAD	SPEED	FEED	STEP OVER	STEP DOWN	STOCK LEFT
OP NO 5	CUT TYPE	TOOL #	TOOL TYPE	DIA.	LENGTH	FLUTE LEN	CORNER RAD	SPEED	FEED	STEP OVER	STEP DOWN	STOCK LEFT
OP NO 6	CUT TYPE	TOOL #	TOOL TYPE	DIA.	LENGTH	FLUTE LEN	CORNER RAD	SPEED	FEED	STEP OVER	STEP DOWN	STOCK LEFT

Pay particular attention to how the part is to be held, and how the datum will be set for each operation.

Do not begin programming until you have a clear and detailed plan that you are confident will work. If the plan is not clear, or if the setup appears shaky, keep planning and consider seeking help from others.

Fight the urge to start programming without a clear, workable plan. Time spent understanding the tasks and processes required to accomplish it is never wasted.

What is a waste is finding out, far along in the machining process, that the plan will not work, the fixturing is not acceptable, or there is some other problem that requires starting over. While it is not possible to foresee all problems every time, good planning will eliminate many of them.

Surface models can be confusing. Anything that reduces screen clutter, such as the use of colors, layers, and the Hide function, will make your job easier, more effective, and reduce the possibility for error.

Force yourself to take regular breaks. The brain needs breaks from time to time to work effectively. You will be surprised how often creative solutions “pop” into your mind when doing something completely unrelated to programming, such as taking a walk or eating a meal.

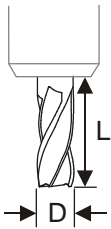
Finally, know when to stop for the day. Few things are more frustrating, or more expensive, than scrapping a part on the last operation because you failed to do something simple or obvious because you were tired.

Tool Selection

Many of the same principles of tool selection for 2D machining also apply to 3D Machining. Here are some general guidelines:

Select the largest diameter tool possible. While there is a practical limit to this rule, it is a good goal to try for. To reduce chatter, try not to finish any surface with a cutter that is the same size as the feature being machined. For example, if the smallest internal radius of the part is .25, consider roughing with the next smaller size cutter (7/16 diameter).

One contradiction to this rule is if the setup is not highly rigid, or if the part itself is very thin and thus prone to deflection by the cutting action of the tool. In this case, you may wish to use a smaller tool to reduce tool pressure.



Tool length/diameter ratio should be kept as small as possible. A good way to calculate this is to divide the length of the tool extending from the holder by its diameter. The resulting tool length/diameter ratio should be less than 2.5. Larger values greatly increase the chance for the tool to chatter, which can greatly degrade surface finish and dimensional accuracy.

When speaking about tool length, we mean the part of the tool extending past the support end of the holder. To keep the unsupported length of the cutter as short as possible, you should set the cutter as far up in the holder or collet as possible, making sure the flutes do not extend past the end of the collet.



**Insert Type
Ball Mill**

For tool holders with a set screw you may want to grind a flat area on the tool for the set screw to engage to keep the tool from spinning in the holder.

Quality tools work better and last longer. Carbide, cobalt or insert-type cutters are more rigid than High Speed Steel, but they cost more. However, when you consider their longer cutting life they can be a better choice than High Speed Steel.

Rigid tools create better cuts. Obviously, a four-flute cutter is more rigid and thus less likely to flex and chatter than a two-flute cutter.

Some materials like aluminum require more chip clearance. Without sufficient clearance, chips may re-circulate in a cavity, even with a strong coolant stream. This recirculation can be heard as a “growling” noise similar to rocks being tumbled, and results in a poor surface finish.

Try using a two- or three- flute cutter instead. While a good compromise of rigidity and chip clearance, three-flute cutters are more expensive and difficult to find than two or four flute types.

Consider using a tapered end mill extension. These allow smaller tools to reach deeper into a cavity while retaining good rigidity.

Consider using a roughing cutter. Roughing end mills are more expensive than standard cutters, but can dramatically reduce production time. Besides being able to achieve much higher cutting feed rates they produce a small, granular chip that evacuates from the work area better.

Consider using a Drill or Plunge Cutter. Plunge roughing removes material using a series of drilling operations. Conventional twist drills, insert drills, or a special tool designed just for this type of roughing operation can be used.

Speeds and feeds are based on many factors including tool and part geometry and material, setup and machine rigidity, coolant, and more. Surface speeds and feeds for 3D work are often different from those for 2D milling. For example, if the tool machines in a descending path, it is common to reduce speed by about 20%. Ball mill chip loads are usually reduced by 20% below those for end mill cutters.

A common problem, especially with roughing operations, is selection of overly conservative speeds and feeds. Time is money. Overly conservative feeds, in addition to dramatically increasing run time and part cost, may actually reduce cutter life and part quality. If the cutter is not taking a “bite” out of the material, it is rubbing against the part causing excessive heat and tool wear.

Certainly, you should not take unnecessary risks. Complex parts, especially molds and aerospace components, are expensive. Not only



End Mill Extension



Roughing End Mill

Cutting Speeds & Feeds

Time = \$

are material and tools expensive, a significant investment in time and money may have been expended to prepare it for surface machining. A broken tool could scrap the part.

Yet, it is also important not to be too conservative. Some novices, having broken a tool, become gun-shy and resort to using overly conservative speeds and feeds. Excessive and habitual caution wastes significant amounts of time and money.

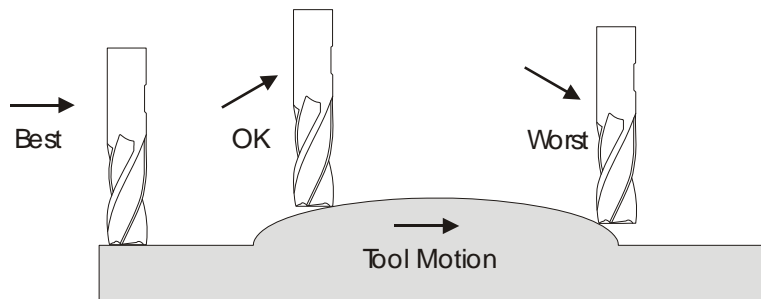
If you are new to surface machining, ask the advice of a trusted and experienced programmer. Use other resources, such as tooling sales engineers and manufacturer's data for information.

A good tool sales representative often has received substantial training on the use of tools and speeds and feeds and can provide sound technical advice. They are also aware of the latest developments in tooling. Some tooling companies offer training seminars on tooling selection that can help increase your knowledge.

3D Cutting Motion

One consideration when selecting a cutting strategy is how well tools perform while moving across 3D shapes. In general:

- Climb cutting is usually preferred over Conventional milling.
- Tools cut best during X-Y only motion.
- Tools cut OK when climbing a wall.
- Tools cut worst when descending a wall.

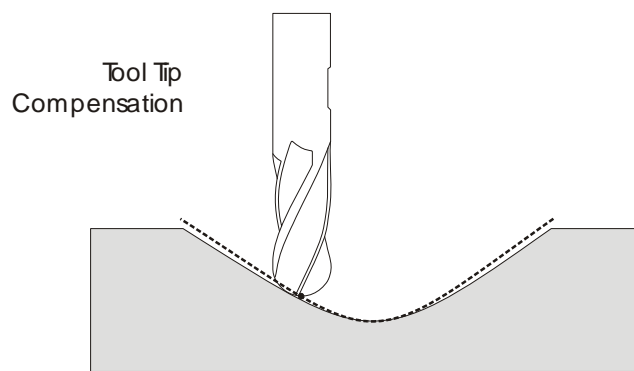
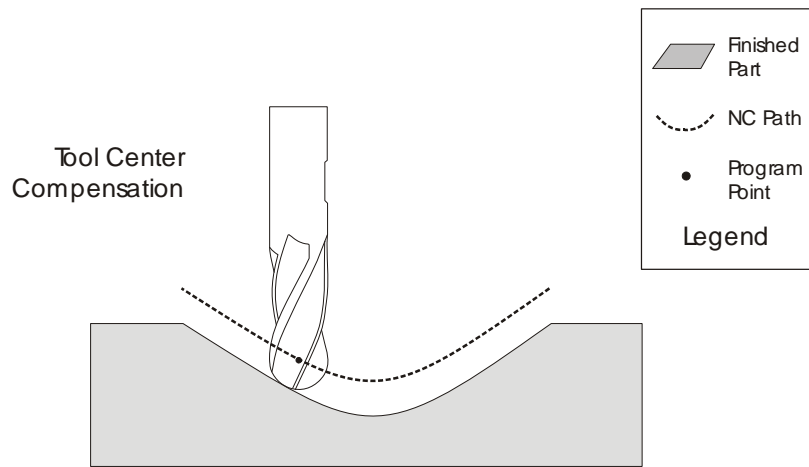


3D Cutting Motion

Tool paths can be created for either the **Center** of the tool, or the **Tip**. While either setting keeps the tool edge on the surface, the Tip method is the most common setting, since it allows the operator to touch off the tool as in 2D machining.

Tool Tip Compensation

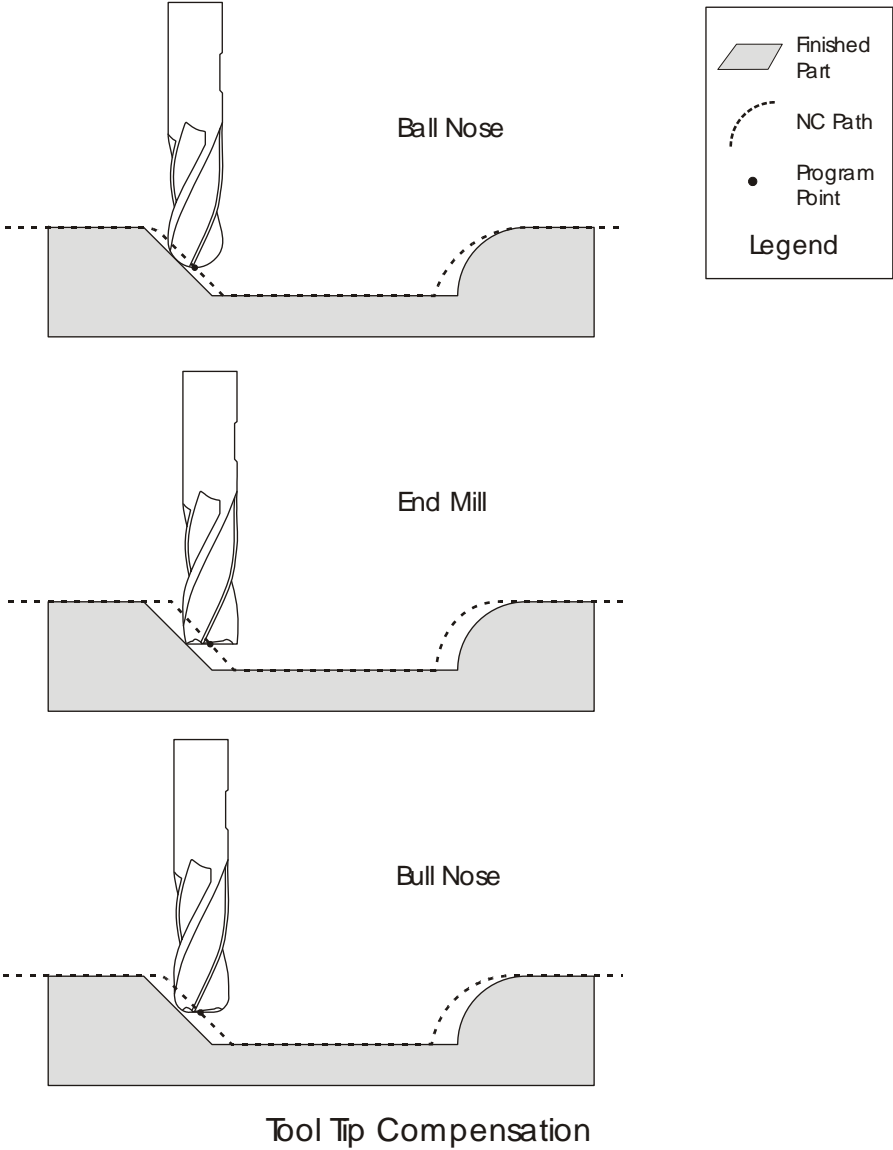
If Center is used, the operator must subtract the tool radius from the height offset register.



Tool Tip vs Tool Center Compensation

Effect of Tool Shape

The diagrams below illustrate tool tip compensation for ball, flat, and bull nose cutters. To keep the edge of the tool on the finish surface the coordinate information changes constantly. This path is affected by the part geometry and this size and shape of the tool.

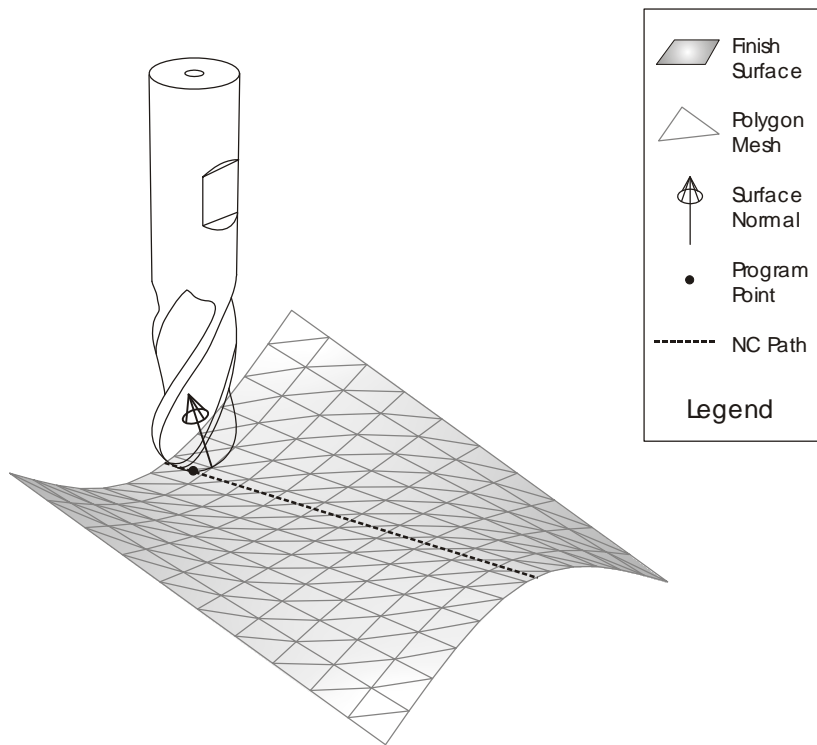


To create a 3D surface tool path, Mastercam first creates a polygon mesh across the surfaces. This mesh, composed of many small triangles, provides the information needed to calculate the surface normal.

**Tool
Surface
Compensation**

This is used to calculate the tool offset at any point along the surfaces, taking into account the tool shape, stock allowance, interference with surface and adjacent surfaces, and gaps between surfaces.

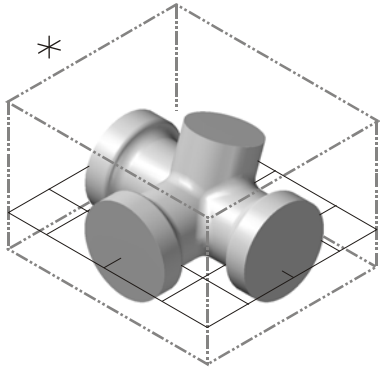
Considering the complexity of these processes, it is easy to understand why surface tool paths require so much computer processing power!



3D Surface Compensation

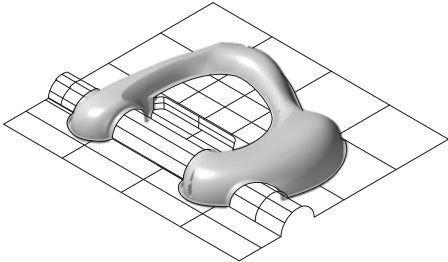
Surface Machining Terms

The following diagram shows geometric features used for surface roughing and finishing tool paths. Not all these features are available for every tool path type.

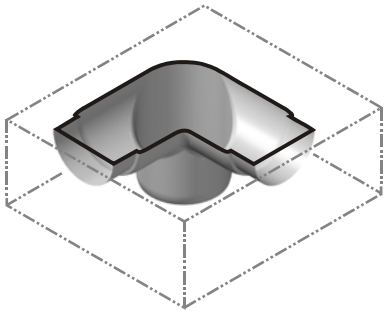


Check Surfaces Used to Protect Areas Machined Later

	Drive Surfaces
	Check Surfaces
	Runoff Surfaces
	Containment Boundary
	Stock Boundary
	Start Point
Legend	



Runoff Surfaces to Expand Toolpath Beyond Part Boundary



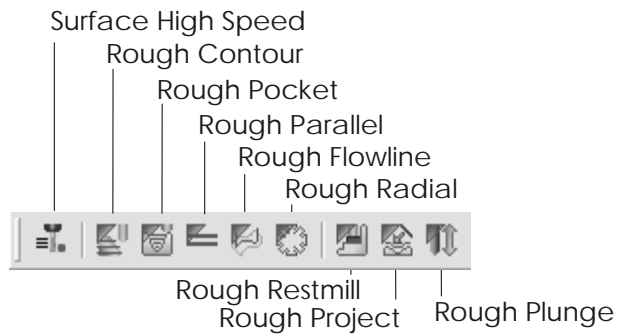
Containment Boundary Required for Rough Pocket

Term	Definition
Drive Surfaces	<p>Drive surfaces are the surfaces to be machined. Both negative and positive stock allowances are allowed on drive surfaces.</p> <p>Complete solids or selected solid faces can be selected as drive entities. In fact, you can have both solid and surface entities selected in the same surface machining operation in Mastercam.</p>
Check Surfaces	<p>Check surfaces create walls that keep the tool path away from other areas of the part. For example, you can cap the tops of bosses that will be finish machined later. Both negative and positive stock allowances are allowed on drive surfaces.</p>
Runoff Surfaces	<p>Runoff surfaces are selected as part of the drive surfaces set. Runoffs are used to extend the tool path beyond the part boundary. Another use is to force the tool path.</p> <p>While used mostly for surface finish operations, runoff surfaces have some applications in roughing.</p>
Containment Boundary	<p>A wireframe profile that acts like a fence to limit the XY extends of the tool path.</p> <p>Tool path can be compensated to the inside, centerline, or outside of the containment boundary.</p> <p>Some rough operations require a containment boundary. Others may process faster or produce better results with it.</p> <p>Containment boundaries can be made up of 2D and 3D wireframe entities. They do not have to form a closed boundary. Multiple containment boundaries can be selected in the same operation.</p>
Start Point	<p>An approximate starting point for the tool path. This point can be located outside the stock boundary for the tool plunge away from the rough stock.</p>
Stock Boundary	<p>Stock definition in the Operations Manager Stock Setup dialog box. Rough tool paths are usually expanded to the limits of the stock material.</p>

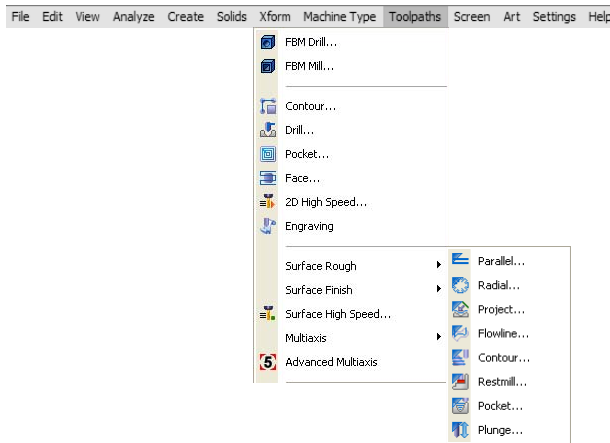
Rough Tool path Types

The following surface rough tool paths are supported in Mastercam. These can be selected from the Toolpaths, Surface Rough drop down menu or the Surface Rough Toolpaths toolbar.

A brief description of each is given here, with more detailed descriptions to follow. Functions are covered in an order best suited for training, not necessarily in the order they appear in the menus.



Surface Rough Toolpaths Toolbar



Surface Rough Flyout Menu

Term	Definition
Surface High Speed	High Speed roughing attempts to maximize cutting feed rates and reduce tool breakage.
Rough Contour	Rough Contour tool paths perform multiple cuts at constant Z levels. Since the tool does not climb or plunge along the surfaces, this function allows high material removal rates.
Rough Pocket	Rough Pocket tool paths create pocket boundaries from selected surfaces similar to 2D pocket machining.
Rough Parallel	Rough Parallel tool paths are used to rough a single boss or cavity. The resulting tool paths, when viewed from the top of the part, form parallel paths.
Rough Flowline	Rough Flowline tool paths follow the natural contours of the surface as seen by the U or V curves.
Rough Radial	Rough Radial tool paths are used for round parts. When viewed from the top, the tool paths are aligned like the spokes on a wheel.
Rough Restmill	Rough Restmill tool paths only machine areas where stock is left by previous operations.
Rough Project	Rough Project tool paths create tool path by projecting curves, points, or other tool paths.
Rough Plunge	Rough Plunge tool paths machine surfaces using drilling motions.

Before selecting a rough tool path, it is suggested you do the following:

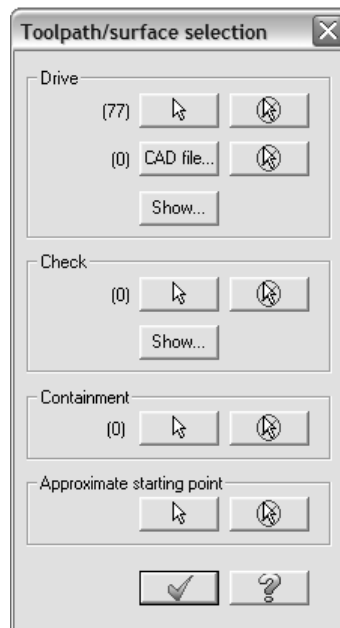
- Prepare a written process plan that includes operation description, tool size and type, speeds and feeds, stepover, stepdown, tolerances, and stock allowances for each machining operation.
- Segregate drive and check surfaces and containment boundaries by color or level.
- Use Analyze Chain to check all containment boundaries.
- Use Analyze Test Surfaces to confirm that there are no problems with the drive or check surfaces.
- Save a backup of your work.

**Before
You
Begin**

**Tool path/
Surface
Selection**

After selecting a rough tool path type, Mastercam will prompt to **Select Drive Surfaces**. Use the General Selection toolbar to select the drive surfaces.

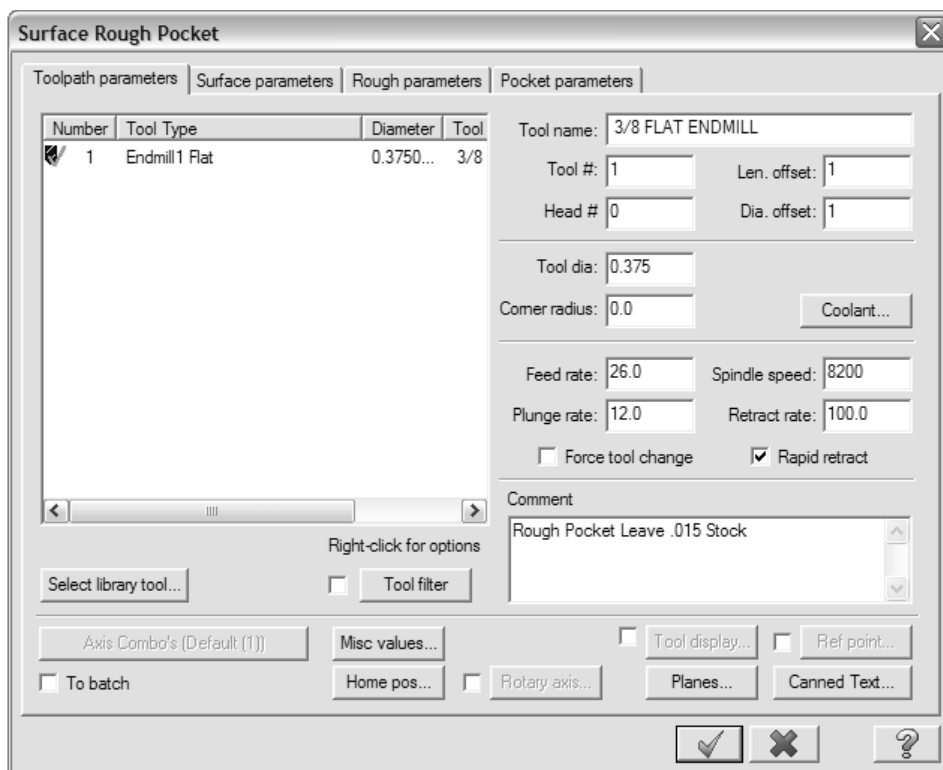
The Toolpath/surface selection dialog box controls selection of geometry for the surface machining operation. Not all selections are available for every tool path type, and are dimmed if the function does not need or support it.



Item	Definition
Drive	Select, unselect, or show drive surfaces. The number of entities selected appears in parenthesis. CAD imports an STL file for the drive geometry.
Check	Select, unselect, or show check surfaces.
Contain	Select or unselect wireframe containment geometry.
Approximate starting point	Select or unselect an approximate starting point.

After closing the Toolpath/surface selection toolbox, the **Tool parameters** page is presented. This page is identical to that used for 2D machining operations. Refer to the help menus or the **Mastercam Handbook, Volume 1** for a complete explanation of these settings.

Tool Parameters



Checking **To batch** uses Mastercam’s batch processing function. Calculation intensive tool paths can be set to start at a specified time. For example, you could set Mastercam to calculate tool paths overnight.

With the increased speed of modern PC computers, the need to batch process tool paths has decreased.



In Depth