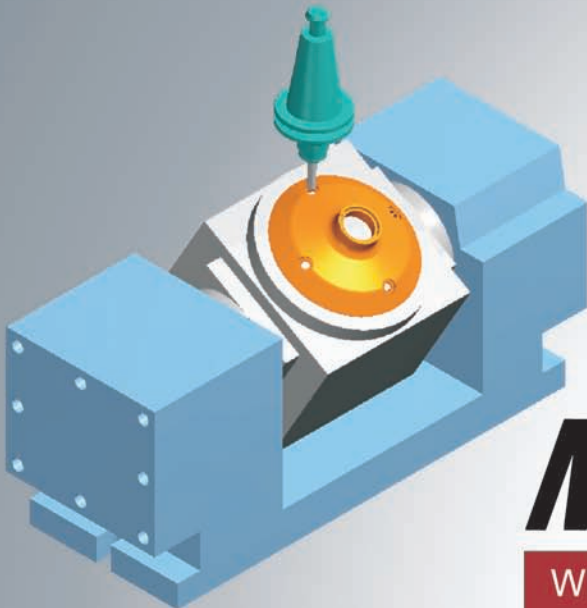


X³

Handbook

Volume 3

Multiaxis Machining



Mastercam®

When Second Best Won't Cut It.®

Mastercam Handbook Volume 3 for Mastercam X3

Date: September 16, 2008

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

Software: Mastercam X3 Mill Level 3 & Multiaxis

Authors: In House Solutions

ISBN: 978-1-926566-10-8

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 September 2008. Requires Mastercam Mill Level 3 and Multiaxis.

Contents

Introduction

Chapter 1	Introduction to Multiaxis Machining
Chapter 2	Multiaxis Curve Machining
Chapter 3	Filter and Entry/Exit
Chapter 4	Multiaxis Drilling
Chapter 5	Swarf Machining
Chapter 6	Multiaxis Flowline Machining
Chapter 7	Multiaxis Multi-surface Machining
Chapter 8	Rotary 4-Axis Machining
Chapter 9	Advanced Multiaxis Toolpaths
Chapter 10	Multiaxis Post Processors
	Index
App. A	Drill Chart

Introduction

Congratulations on your purchase of the Mastercam Handbook Volume 3. 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, common G&M codes. You should have already completed the Mastercam Handbook Volume 1 and 2 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 work as they do.

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 on that machine by a qualified person.



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: Introduction to Multiaxis Machining provides an overview of 5-axis toolpath types and differences between 3D and Multi-axis setups.

Chapter 2: Multiaxis Curve Machining shows how to machine 3D contours.

Chapter 3: Filter and Entry/Exit shows how to filter Multi-axis toolpaths and create entry and exit moves.

Chapter 4: Multiaxis Drilling shows how drill normal to a surface.

Chapter 5: Swarf Machining shows how to machine complex shapes using the side of the cutter.

Chapter 6: Multiaxis Flowline Machining shows how to multi-axis machine with paths that follow the natural contour of surfaces.

Chapter 7: Multiaxis Multi-surface Machining shows how to multi-axis machine across landscapes of many surfaces.

Chapter 8: Rotary 4-Axis Machining shows how to machine using a simultaneous rotary table.

Chapter 9: Advanced Multiaxis Toolpaths shows how to use Mastercam's new Advanced Multiaxis functions.

Chapter 10: Multiaxis Post Processors shows how to configure a generic multiaxis post processor to support different machine configurations.

App. A: 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 of interest only to the highest level users of Mastercam.



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.

An accompanying CD ROM contains files needed to complete all exercises in this handbook, as well as movie (.WMV) files to demonstrate certain topics and examples.

You will need Windows Media Player with the appropriate CODEC, an Audio Card, and speakers or headset to play these movies. These files provide detailed step-by-step instructions on how to use Mastercam.

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

6 Multiaxis Flowline Machining

The purpose of this chapter is to introduce multiaxis flowline tool paths.

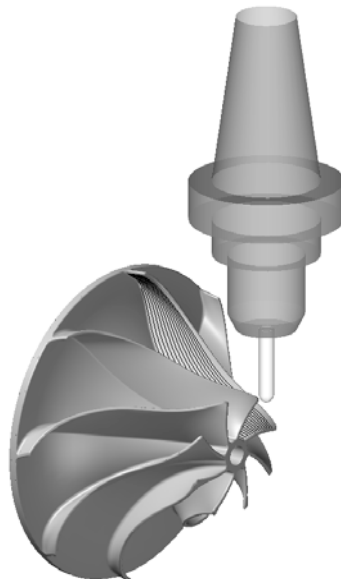
Objectives

Upon completion of this chapter, you should be able to do the following:

- Understand concepts related to multiaxis flowline toolpaths.
- Understand the advantages of flowline over other Multiaxis toolpaths, such as Swarf or Multisurf.
- Know how to control the tool axis and tip for multiaxis flowline.

Flowline tool paths are a particularly useful and efficient machining strategy. These paths follow the natural contour of the surface, allow scallop height control, and leave an excellent surface finish. Machining times can be less than with other strategies, and the NC file is generally smaller than other tool path types. For all these reasons, flowline is often the first choice to consider when machining surfaces.

Introduction



Impeller.MCX



A multiaxis flowline tool path, like three axes flowline paths, is usually limited to single surface machining.

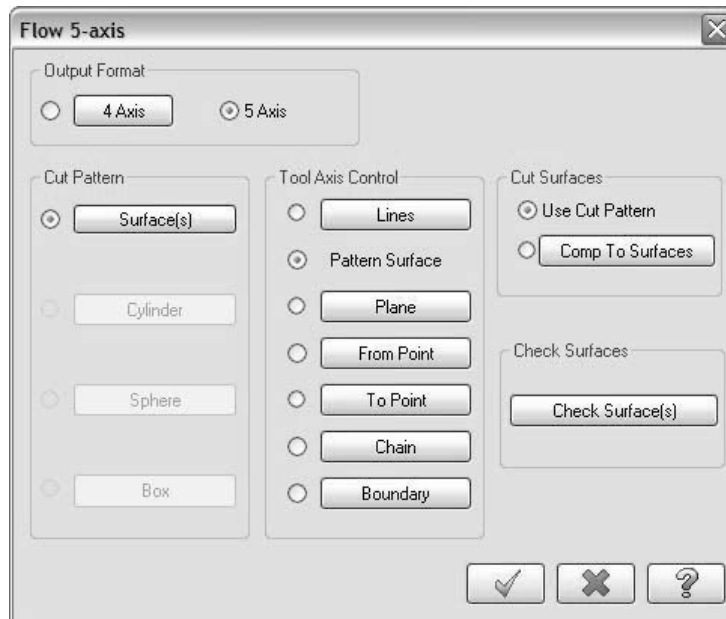
Multisurface flowline may be possible if the surfaces lie in a well-ordered row. For surfaces to form a row, adjacent surface edges must be equal in length and tangent. Surfaces usually form a well-ordered row if the shape could be constructed as a single surface.

5 Axis Flowline

Select **Toolpaths, Multiaxis, 5 Axis Flowline Toolpath** or the 5 Axis Flowline icon from the Multiaxis Toolpath tool bar.



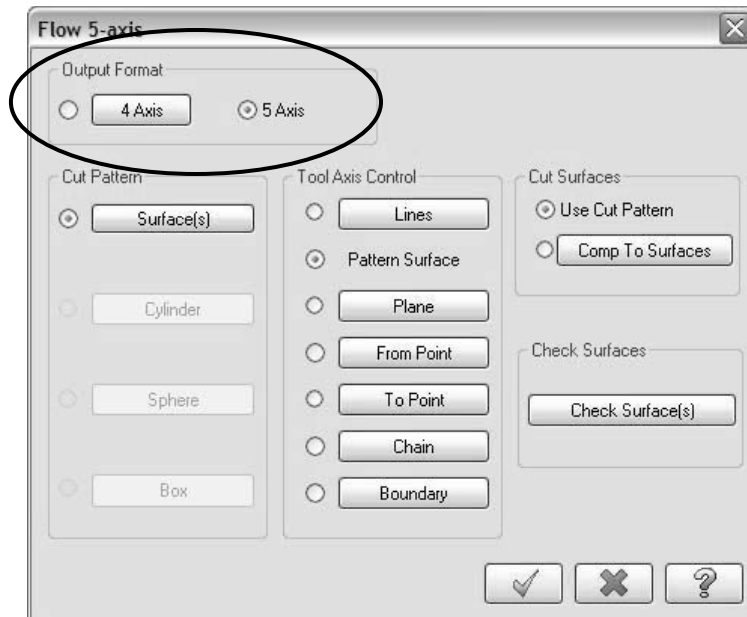
The Flow 5-axis Swarf dialog box controls the output format, tool axis control, and more.



Item	Definition
Output Format	Determines if the output code is 4-axis (one rotary axis), or full 5-axis (two rotary axes).
Cut Pattern	Only the Surface(s) option is available for the 5-Axis Flowline tool path.
Tool Axis Control	When the Point entity type is selected, determines how the tilt of the tool is controlled as it moves along the trajectory.
Cut Surfaces	Determines which surfaces and method will be used to compensate the tip of the cutter.
Check Surfaces	Allows selection of surfaces for the tool to avoid.

Output Format allows selection of either 4 or 5-axis output. Notice that 3-axis format is not allowed.

Output Format



Cut Pattern determines the step along pass as defined by the UV lines of the surface. Only the **Surface(s)** option is available for Flowline. The other cut patterns are available for 5 Axis Multisurface Toolpath, which is covered in the next chapter.



Flowline works for multiple surfaces, if these surfaces lie in a well ordered row. How do you determine if the surfaces meet this requirement?

The simple answer is, if the surfaces look like they lie in a well ordered row, try using Flowline. If it fails, do one of the following, in the order listed:

- Increase the edge tolerance (discussed later).
- Re-create the shape as a single surface.
- Use one of the other surface machining strategies discussed in this handbook.

Flowline machining is one of the few Mastercam tool paths that requires attention to the direction of surface normals. Flowline uses the normals to determine which side of the surface to machine.

Change Normals



The cutting side of the surface is easily flipped during toolpath creation for single surface machining. For multi-surface Flowline, it is best to set all the normals in the correct direction before creating toolpaths.

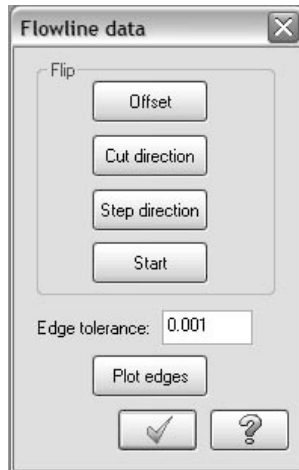
The easiest way to check normals is to turn shading off (**Alt-S**). Observe the UV lines of the surfaces. The side the normal points away from is shown as the colored side of the surface. The other side is a dull grey color.

Reverse the normal using **Edit, Change Normal**. Click on the Flip button on the Change Normal ribbon bar to change the normal direction.



After selecting the surfaces to machine, the Tool path/surface selection dialog appears. It includes a **Flowline** data button. Click on this button to display the flowline data dialog box to set up the flowline tool path.

Flowline Data

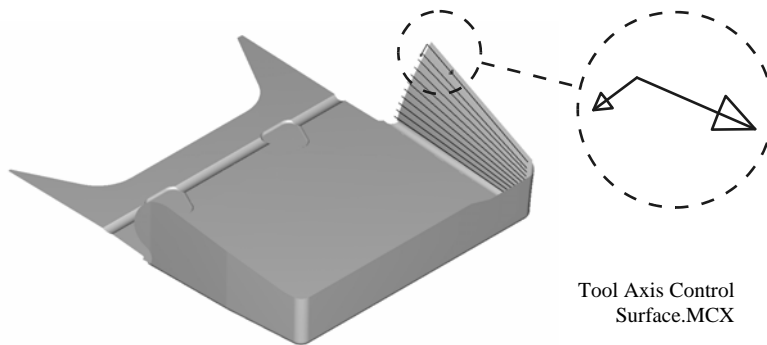


Item	Definition
Offset	Which side of the surface(s) to machine. Default is the surface normal side.
Cut direction	Machine in U or V curve direction.
Step direction	Sets steper direction; left to right or right to left.
Start	Sets which side of the surface to start machining from.
Edge tolerance	Determines which edges are considered shared when creating flowline tool path along multiple surfaces.
Plot edges	Color codes surface edges to show which Mastercam sees as shared or not shared. Used to help troubleshoot problems when flowline is across multiple surfaces. Blue: Free edges Cyan: Partially shared edges. Red: Fully shared edge Red points: Touch point Light blue points: Touch point projected onto adjacent surface.

**Flowline
Visual Cues**

Offset and **Cut Direction** are indicated by grey curves. Grey toolpath indicators show which side will be machined. Toggle the offset by clicking on this menu item once. A set of arrows indicates the selection.

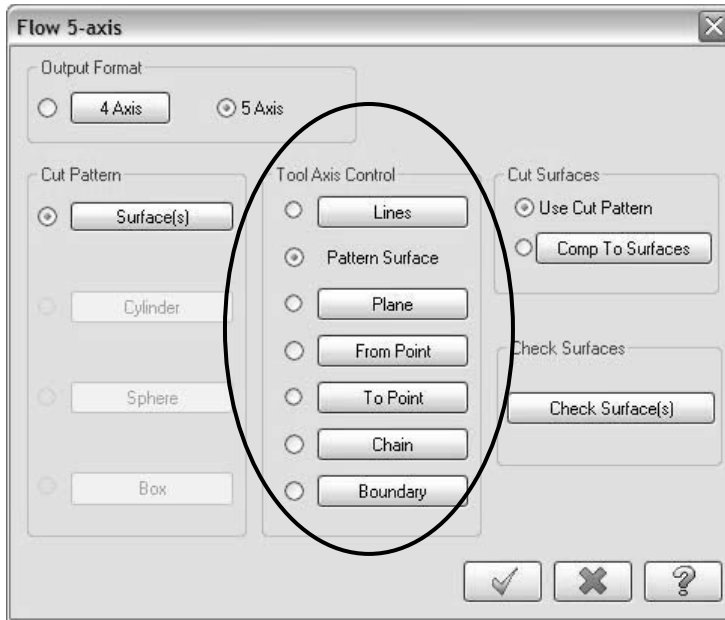
Step dir and **Start** are indicated by a set of arrows. The location of the arrow indicates where the toolpath starts. The large arrow indicates the direction of the toolpaths along or across the surface. The small arrow indicates the step direction of the toolpath.



Tool Axis Control
Surface.MCX

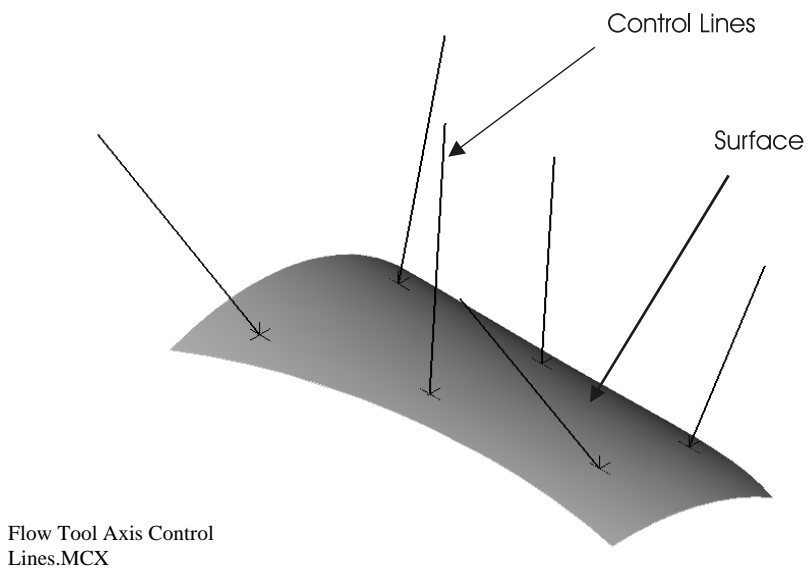
The **Tool Axis Control** group sets the rotary axis angles of the tool.

Tool Axis Control



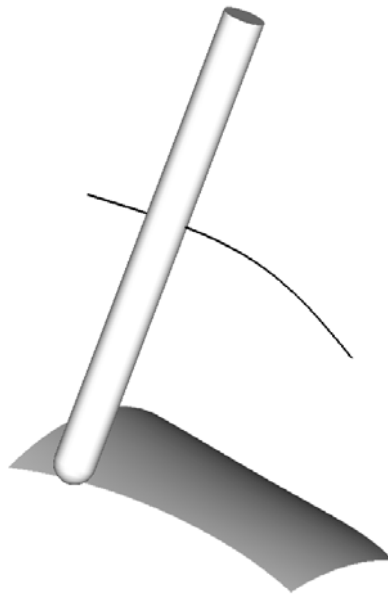
Item	Definition
Lines	Sets the tool axis normal to lines spaced along the surface.
Pattern Surface	Controls the tool axis normal to the flowline surface.
Plane	Keeps the tool axis normal to a plane. The tool moves in a 3-axis mode in relation to the plane.
From Point	The selection orients the tool axis so it remains aligned with a control point. From Point and To Point are detailed in Chapter 2.
To Point	Similar to From Point, except the projection is from the tool to the point, rather than from the point.
Chain	Controls the tool axis using control curves
Boundary	Contains the shank of the tool within a closed profile

Lines sets the tool axis normal to lines spaced along the surface. Lines point in the direction of the tool axis as the tool passes the line. The tool axis is influenced by the nearest control line and gradually changes as it moves across the cut surface(s).



Tool Axis Control Lines

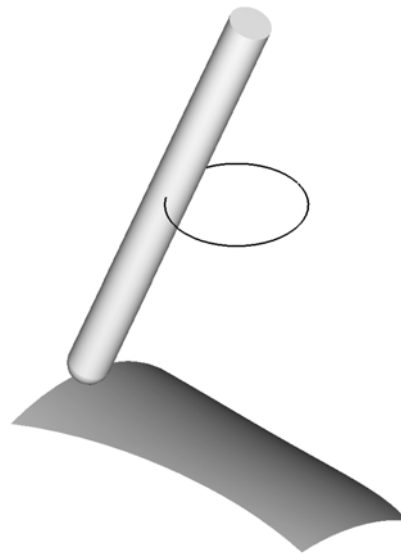
Chain controls the tool axis using control curves. As the tool moves along its path, the shank of the cutter passes through the control curves. Use Chain when the controlling curves are open. Use Boundary when the curves are closed.



Flow Tool Axis Control
Chain.MCX

Control Chain

Boundary contains the shank of the tool within a closed profile. This method can be used to thread the tool through a confined window. An example application would be for porting engine cylinder heads.

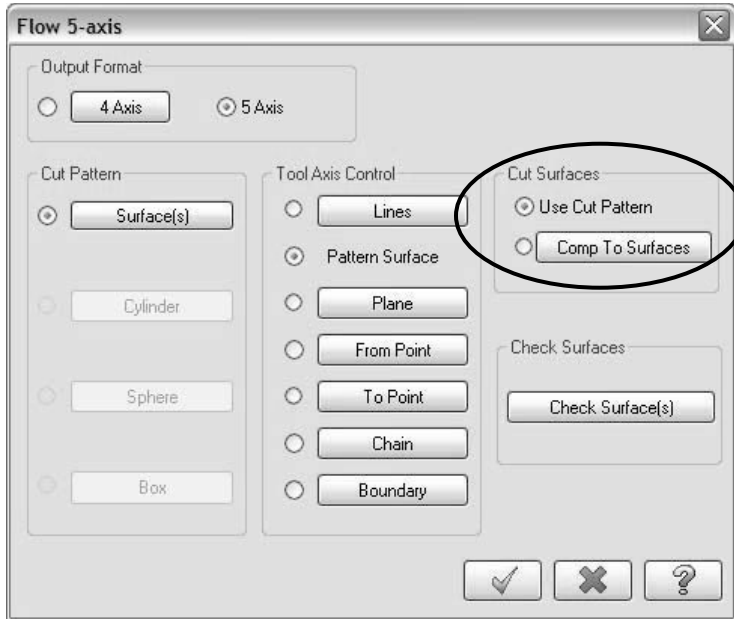


Flow Tool Axis Control
Boundary.MCX

Control Boundary

Cut Surfaces determines which surfaces and method will be used to compensate the tip of the cutter.

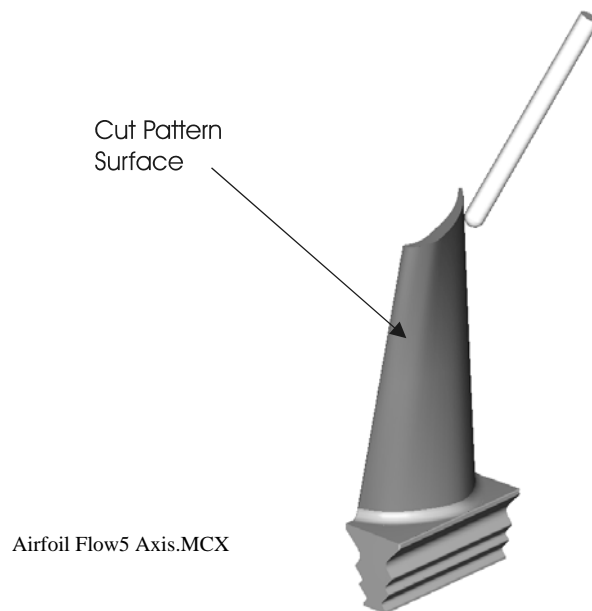
Cut Surfaces



Item	Definition
Use Cut Pattern	Use the UV lines of the cut pattern surface(s).
Comp to Surfaces	Allows tip compensation to be applied to different surfaces than the pattern surfaces.

Use Cut Pattern compensates the tool to the UV lines of the cut pattern surface(s). Use this method when the UV lines flow in the same direction you want the toolpath to flow.

In the example below, the turbine blade consists of a single lofted surface. If it did not, the programmer might choose to re-create the wing section of the blade as a single surface to allow using Flowline.



Cut Surfaces - Use Cut Pattern



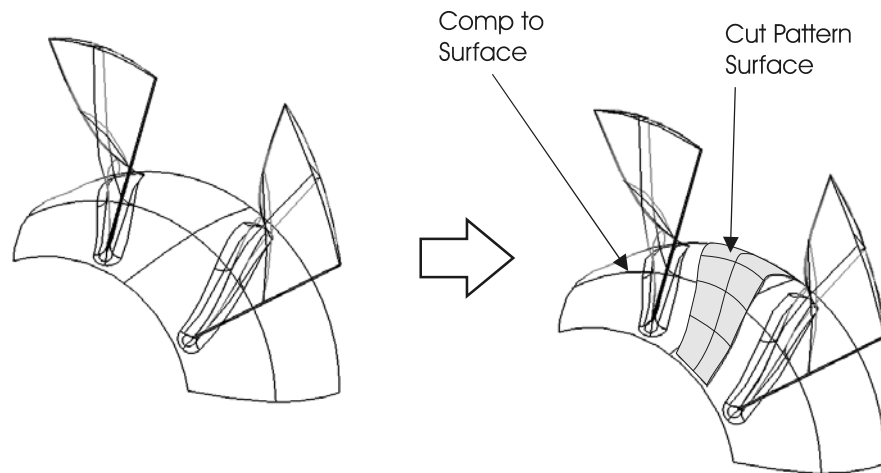
Another advantage of flowline is that it allows undercutting of surfaces using a lollipop cutter. Just be sure to define the shank of the lollipop cutter long enough to clear all potential interferences.

Comp to Surfaces compensates the tool to surfaces other than the pattern surfaces. This method uses the entire tool tip diameter and shank for gouge checking, so undercutting is not possible.

In some cases, it is helpful to cut pattern surfaces with UV lines as desired to control the flow of the toolpath.

In the example below, the hub of the turbine wheel is a revolved surface whose UV lines flow in a radial or axial pattern. The manufacturer required full length toolpath scallops to follow the root of the blades.

To accomplish this, a Coons surface was created by offsetting and projecting curves onto the turbine hub. This surface was used as the cut pattern surface. The original revolved hub surface is used as the tool comp surface.



Cut Pattern Surface.MCX

User Created Cut Pattern Surface

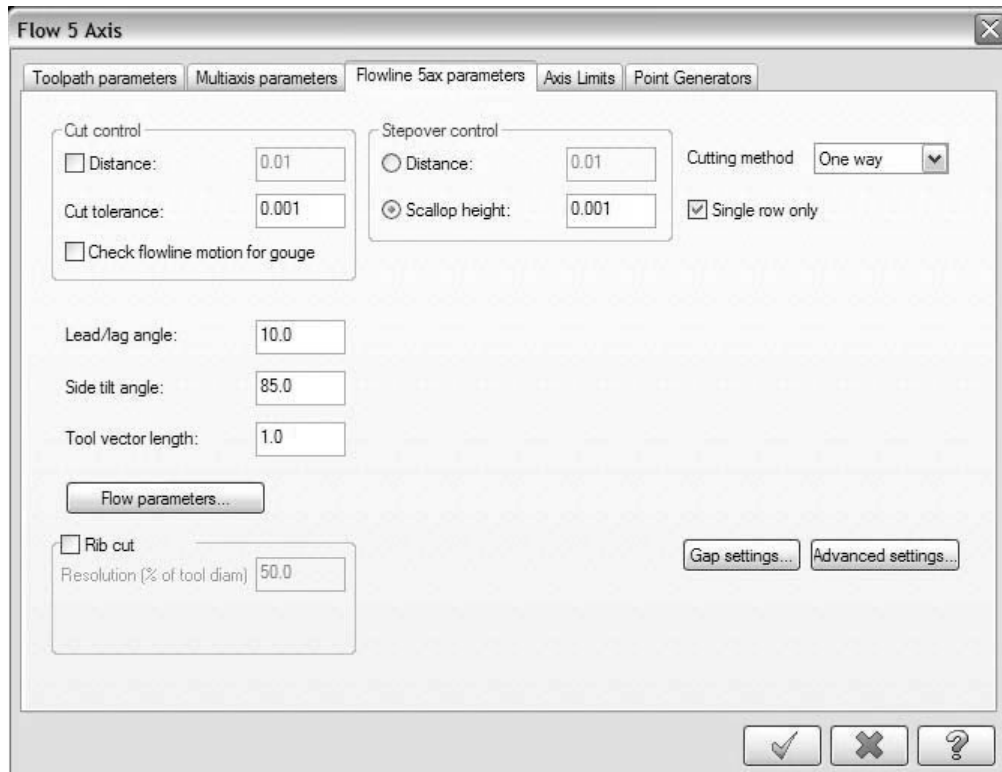
Cut pattern surfaces are powerful. They allow the programmer to construct a surface whose UV lines control the shape of the tool path. Look closely at the previous example. See how the programmer cleverly constructed a cut pattern surface between the blades that allows using a very efficient Flowline to rough machine this area while avoiding any collision problems.

**Flow
5 Axis
Parameters**

You should already be familiar with most of the settings on the Flow 5 Axis parameters dialog box tab.

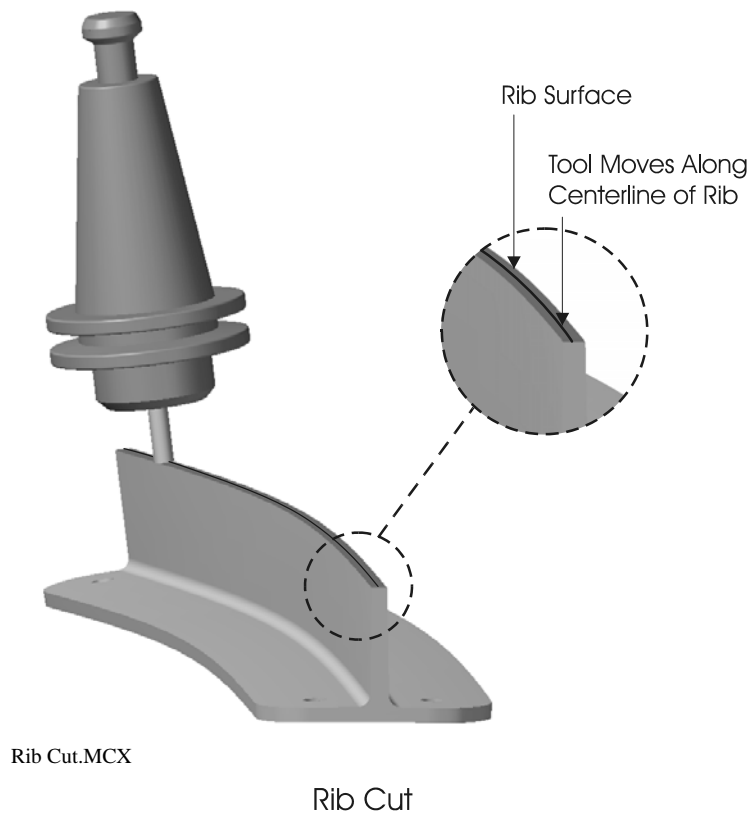
One way, zig-zag, and spiral cutting methods are supported.

Clicking on **Flow parameters** brings up the Flowline: menu, which was discussed on page 4 of this chapter.



Item	Definition
Cut Control	Controls the tolerance of the tool path along the cut, either as a fixed Distance , or as a Cut tolerance (Chordal Deviation).
Check flowline motion for gouge	Deletes tool motion that could gouge the surface. Turn this setting off to speed calculation time if there is no chance of gouging.
Lead/lag angle	Sets the tilt of the tool forward (lag) or backward (lead) in relation to the cut direction.
Side tilt angle	Sets the cant of the tool, left or right, in relation to the cut direction.
Tool vector length	Sets display length of tool vectors during backplot. This setting has no effect on the NC code file.
Flow Parameters	Opens the Flowline data dialog box to allow changing the offset, cut direction, step direction or start point.
Rib cut	Creates a single pass flowline down the centerline of a the drive surfaces. Used to face cut support ribs typical of aerospace work.
Stepover control	Sets the spacing between tool paths as either a fixed Distance , or a desired finish Scallop Height .
Cutting method	Selects the cutting method for the tool path: One way , ZigZag , or Spiral .
Single row only	Select this option to create a Mastercam version 8-style tool path. See Help for more information.
Gap settings	Sets tool motion at gaps in the surfaces or curves.
Advanced settings	Sets how sharp corners and hidden faces are handled. There is usually no need to adjust these settings. See the Mastercam Help for more information.

The illustration below shows the rib cut option. The tool makes a single pass, down the centerline of the rib surface(s). Adjust the Resolution setting to smooth the tool path.



The Axis Limits and Point Generators dialog box tabs are identical to those already covered in this book.