

What's New in Mastercam X3



Mastercam® X3

What's New in Mastercam X3

July 2008

Be sure you have the latest information!

Information might have been changed or added since this document was published. The latest version of this document is installed with Mastercam or can be obtained from your local Reseller. The ReadMe file ([ReadMe.htm](#)) includes the latest information about new features and enhancements.

Mastercam® X3 WHAT'S NEW IN MASTERCAM X3

Date: July 2008

Copyright © 2008 CNC Software, Inc.— All rights reserved.

First Printing: July 2008

Software: Mastercam X3

ISBN: 1-883310-72-5

IMPORTANT NOTICE!

PLEASE READ THIS STATEMENT AND THE SOFTWARE LICENSE AGREEMENT COMPLETELY BEFORE USING THIS SOFTWARE. **YOU MUST ACCEPT THE TERMS OF THIS LICENSE BEFORE YOU CAN OPERATE THE SOFTWARE PROGRAM.**

BY CONTINUING TO USE THIS SOFTWARE, YOU (EITHER AN INDIVIDUAL OR A SINGLE ENTITY) INDICATE YOUR INTENTION TO BE BOUND BY AND ACCEPT THE TERMS AND CONDITIONS OF THIS SOFTWARE LICENSE. IF YOU DO NOT AGREE TO THESE TERMS AND CONDITIONS, YOU MAY NOT ACCESS OR OTHERWISE USE THIS SOFTWARE AND WILL IN FACT BE PROHIBITED FROM DOING SO. THIS COMPUTER SOFTWARE MAY BE USED ONLY PURSUANT TO THE TERMS AND CONDITIONS SET FORTH BELOW, AND SOLELY IN CONJUNCTION WITH THE ACCOMPANYING SECURITY MECHANISM (UNLESS OTHERWISE SPECIFIED IN THE "EXCEPTIONS TO SECURITY MECHANISM REQUIREMENTS" SECTION OF SUCH TERMS AND CONDITIONS) WHICH MUST BE PRESENT ON YOUR COMPUTER (OR NETWORK AS APPLICABLE) AT ALL TIMES DURING SUCH USE.

Please Note: Your company may have additional restrictions and guidelines in place regarding the use of this Software that are in addition to those found within this Software License. If you have any questions or concerns as to the existence of any such additional restrictions or guidelines, you should consult with your supervisor or your IT Department before using this Software.

Read This If You Have Received This Software From A Third Party: If you received this Software from someone other than an authorized CNC Software, Inc. reseller, you do **not** have a legal software license. (For example, if you acquired this Software simply by buying a machine or Software from the former owner or from an auction, you do **not** have a valid license.) You should contact CNC Software, Inc. (860-875-5006) in order to obtain a new and valid license. **Use of this Software without a valid software license is unlawful, a violation of the Copyright Act and may expose you to criminal liability under 17 United States Code Sec. 506, including fines and damages.**

Software License

If you have paid a license fee, CNC Software, Inc. ("CNC") a Connecticut corporation with its principal place of business at 671 Old Post Road, Tolland, Connecticut, 06084 hereby grants to you a non-exclusive, non-transferable license (the "License") to use this software program and its accompanying documentation (and, if applicable, to permit your authorized employees to use them), solely in accordance with the terms and conditions of this Software License Agreement. You may use the Software solely for your internal business purposes and solely in conjunction with the accompanying hardware or software device, method, scheme or other security measure provided by CNC which allows a user to access the Software and prevents unauthorized access to the Software (the "Security Mechanism"). The Software, any updates to the Software through purchase or due to enrollment in an authorized software Maintenance program (including any that you download through the Internet), and the Documentation in printed or electronic form shall hereinafter collectively be referred to as the "Software" and are all governed by this License.

Restrictions

You may not use the Software without a Security Mechanism provided by CNC or CNC's suppliers. When CNC or CNC's resellers provide you with a single-user Security Mechanism, the Software may only be used (in executable code form only) on a single computer to which the Security Mechanism is physically attached. In the event CNC or CNC's resellers provide you with a multiple-user Security Mechanism for use over an internal network (a "Network Security Mechanism"), the Software may be used: (a) in

executable code form only; (b) only on end-user computers that are connected to the internal network to which the Network Security Mechanism is attached; and (c) only by the number of users and accessed by the number of end-user computers for which licenses were purchased and as further allowed by the Network Security Mechanism. You may physically transfer the Software from one computer equipped with a single-user Security Mechanism to another only if the Security Mechanism is included in the transfer and is installed with the new computer.

You shall not: (a) copy (except as provided below), adapt, modify the Software; (b) publish, display, disclose or create a derivative work from the Software or any part thereof; (c) de-compile or translate, disassemble, create or attempt to create, by reverse engineering or otherwise, the source code form of the Software from the executable code of the Software; (d) remove any proprietary notices, labels or marks from the Software; (e) sell, rent, lease, distribute or otherwise transfer or provide all or any part of the Software to any person or entity without the prior written consent of CNC; (f) use the Software to provide outsourcing, service bureau, time sharing or other services to any third party; or (g) sublicense, assign, delegate or otherwise transfer your rights in the Software, under the Software License Agreement or any of the related rights or obligations for any reason without the prior written consent of CNC. You shall not circumvent, bypass, modify, reverse engineer, disassemble, disable, alter, enhance or replicate the function of the Security Mechanism in any manner whatsoever. Any attempt to do so shall result in automatic termination of this License without prejudice to all other legal rights and remedies of CNC.

Copying Restrictions

You may make one (1) copy of the Software for backup or archival purposes, provided that you reproduce all proprietary notices of CNC on any such copy.

Non Transferable

You may not transfer or assign the Software or this Software License Agreement or any rights or obligations hereunder. Any attempt to do so will automatically terminate this License without the need for notice. This termination is without prejudice to all other legal rights and remedies of CNC.

Intellectual Property Rights

The Software is and includes intellectual property of CNC. All associated intellectual property rights, including, without limitation, worldwide patent, trademark, copyright and trade secret rights, are reserved by CNC. CNC retains all right, title and interest in and copyrights to the Software, regardless of the form or media in or on which the original or other copies may subsequently exist. This Software License Agreement shall not constitute a sale of the Software and no title or proprietary rights to the Software are transferred to you hereby. You acknowledge that the Software is a unique, confidential and valuable asset of CNC, and CNC shall have the right to seek all equitable and legal redress which may be available to it for the breach or threatened breach of this Software License Agreement including, without limitation, injunctive relief. Unauthorized copying of the Software or failure to comply with the above restrictions shall result in automatic termination of this License and this Software License Agreement without prejudice to all other legal rights and remedies of CNC.

Confidentiality

You acknowledge that the Software contains proprietary trade secrets of CNC and you hereby agree to maintain the confidentiality of the Software using at least as great a degree of care as you use to maintain the confidentiality of your own most confidential information. You agree to promptly communicate the terms and conditions of this Software License Agreement to those persons employed by you who come into contact with the Software, and to use your best efforts to ensure their compliance with such terms and conditions, including, without limitation, absolutely preventing such persons to use any portion of the Software for the purpose of deriving the source code of the Software or defeating the Security Mechanism.

Enforcement Obligations

In the event you become aware that any person or entity in your employ or under your control is using the Software in a manner not authorized by this Software License Agreement, you shall immediately take all steps necessary to stop such unauthorized use of

the Software. You shall promptly notify CNC in writing of any unauthorized use of the Software of which you become aware.

Limited Warranties

CNC WARRANTS THAT THE MEDIA ON WHICH THE SOFTWARE IS DISTRIBUTED WILL BE FREE OF DEFECTS IN MATERIAL OR WORKMANSHIP FOR A PERIOD OF THIRTY (30) DAYS AFTER PURCHASE. THE FOREGOING LIMITED WARRANTY EXCLUDES DEFECTS ARISING OUT OF ACCIDENT, NEGLIGENCE, MISUSE, FAILURE OF ELECTRIC POWER AND CAUSES OTHER THAN ORDINARY AND AUTHORIZED USE. EXCEPT FOR THE FOREGOING LIMITED WARRANTY, THE SOFTWARE IS PROVIDED "AS IS, WITH ALL FAULTS." THIS LIMITED WARRANTY IS THE ONLY WARRANTY PROVIDED BY CNC REGARDING THE SOFTWARE. TO THE MAXIMUM EXTENT PERMITTED BY LAW, CNC DISCLAIMS ALL OTHER WARRANTIES OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, WITHOUT LIMITATION, IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. CNC IS NOT OBLIGATED TO PROVIDE ANY UPDATES TO THE SOFTWARE.

Sole and Exclusive Remedy for Breach of Warranty: YOUR SOLE EXCLUSIVE REMEDY AND CNC'S SOLE OBLIGATION HEREUNDER SHALL BE, AT CNC'S SOLE OPTION (i) REPLACEMENT OF THE DEFECTIVE MEDIA; OR (ii) REFUND OF THE PURCHASE PRICE OF THE SOFTWARE. CNC SHALL HAVE NO OTHER OBLIGATION OR LIABILITY TO YOU ARISING FROM OR RELATED TO THIS LICENSE OR YOUR USE OF THE SOFTWARE. ANY USE BY YOU OF THE SOFTWARE IS AT YOUR OWN RISK.

Note on Documentation: While CNC makes every effort to ensure that its Documentation for the Software is accurate and up-to-date, it cannot guarantee the Documentation at all times represents the latest operation and functionality of the Software. The content of all documentation, in electronic or printed form, for the Software is provided for informational purposes only. The content of the Documentation may be changed without notice to you. **CNC expressly disclaims any warranty or representation that the Documentation is an accurate and/or current reflection of the Software's operation and performance.**

Disclaimer of Consequential Damages and Limitation of Liability

IN NO EVENT WHATSOEVER WILL CNC, OR ITS EMPLOYEES, SHAREHOLDERS, DISTRIBUTORS OR RESELLERS BE LIABLE TO YOU FOR ANY INDIRECT, INCIDENTAL, OR CONSEQUENTIAL DAMAGES ARISING OUT OF OR IN CONNECTION WITH THIS SOFTWARE LICENSE AGREEMENT OR THE USE OF THE SOFTWARE, EVEN IF CNC HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. WITHOUT LIMITING THE FOREGOING, CONSEQUENTIAL DAMAGES SHALL INCLUDE, WITHOUT LIMITATION, SPECIAL, PUNITIVE OR EXEMPLARY DAMAGES, DAMAGES FOR LOSS OF BUSINESS, LOSS OF PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION, AND LOSS OF OR DAMAGE TO ANY PRODUCTS THAT THE SOFTWARE IS USED IN CONJUNCTION WITH.

CNC'S ENTIRE LIABILITY WITH RESPECT TO ITS OBLIGATIONS UNDER THIS SOFTWARE LICENSE AGREEMENT OR OTHERWISE SHALL NOT EXCEED THE AMOUNT OF THE LICENSE FEE PAID BY YOU FOR THE SOFTWARE. SOME JURISDICTIONS DO NOT ALLOW THE EXCLUSION OR LIMITATION OF IMPLIED WARRANTIES OR LIABILITY FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES, SO THE ABOVE LIMITATIONS OR EXCLUSIONS MAY NOT APPLY TO YOU.

Indemnification

You shall indemnify and hold harmless CNC, its officers, directors, employees, resellers and agents (the "Indemnified Parties") from and against all losses, settlements, claims, actions, suits, proceedings, judgments, awards, damages, liabilities, costs and expenses including, without limitation, reasonable attorneys' fees (collectively "Losses") which arise out of or are related to any breach of this Software License Agreement by you or your employees, agents, resellers, dealers or sub-dealers, and shall reimburse the Indemnified Parties for any and all legal, accounting and other fees, costs and expenses reasonably incurred by any of them in connection with investigating, mitigating or defending any such Losses.

- Educational Pricing** If this Software was obtained through or in accordance with a CNC "Educational Pricing" plan, option, grant, schedule or program, it may not be used by anyone, including you, to conduct any computer aided design, computer aided drafting, computer aided machining, or training activities that, directly or indirectly, generate or otherwise result in monetary revenues for the benefit of any individual or any entity, other than the school that originally received this Software.
- Termination** This Software License Agreement is effective until terminated. You may terminate this Software License Agreement at any time by returning to CNC all copies of the Software under your control and by returning the Security Mechanism to CNC. CNC may terminate this Software License Agreement if CNC determines, in its sole discretion, that you have violated the terms of this Software License Agreement. Upon termination of this Software License Agreement, you agree to immediately return to CNC all copies of the Software, return the Security Mechanism to CNC, and certify to CNC in writing that all known copies, including backup copies, have been returned. All provisions relating to confidentiality, proprietary rights, indemnification and non-disclosure shall survive the termination of this Software License Agreement. **You may not transfer this Software to the purchaser of any equipment on which the Software may be resident. You may not transfer this Software via liquidation, bankruptcy, auction, close of business, or any other method that does not involve an authorized Mastercam reseller. This License is for you alone.** In the event you breach the provisions of this Section, CNC shall be entitled to liquidated damages in the amount of Fifteen Thousand Dollars (\$15,000), plus its reasonable attorney's fees and court costs.
- General** This Software License Agreement shall be governed by and construed in accordance with the laws of the state of Connecticut, USA without regard for Connecticut's conflicts of law principles. The sole and exclusive jurisdiction and venue for any litigation arising from or related to this Software License Agreement or the subject matter hereof shall be in an appropriate state or federal court located in Hartford, Connecticut. You hereby submit to the personal jurisdiction of the US District Court for the District of Connecticut and the Superior Court of the State of Connecticut. This Software License Agreement shall constitute the entire agreement between you and CNC with respect to the subject matter hereof. Any waiver or modification of this Software License Agreement shall be valid only if it is in writing and signed by both parties hereto. If any part of this Agreement is found invalid or unenforceable by a court of competent jurisdiction, the remainder of this Agreement shall be interpreted so as to reasonably affect the intention of the parties.
- Attorneys Fees** In the event that CNC is required to take legal action to enforce its rights under this License Agreement and obtains a ruling or settlement in its favor, you shall be liable to CNC for its reasonable attorney's fee incurred in investigating and prosecuting its claim.
- U.S. Government Restricted Rights** The Software provided hereunder is a "commercial item," as that term is defined in 48 C.F.R. 2.101, consisting of "commercial computer software" and "commercial computer software documentation," as such terms are used in 48 C.F.R. 12.212. Consistent with 48 C.F.R. 12.212 and 48 C.F.R. 227.7202-1 through 227.7202-4, the Software made available to the United States of America, its agencies and/or instrumentalities, is provided with only those rights set forth in this Agreement. Use, duplication or disclosure of the Software by the government is subject to the restrictions as set forth in subparagraph (c) (1) and (2) of the Commercial Computer Software-Restricted Rights clause at 48 C.F.R. 52.227-19, as amended, or any successor regulations thereto.
- Export Restrictions** You represent and warrant that you will not, without obtaining prior written authorization from CNC and, if required, of the Bureau of Export Administration of the United States Department of Commerce or other relevant agency of the United States Government, export or re-export, directly or indirectly, the Software from the United States to (i) any country destination or entity to which export is restricted by the Export Administration Regulations of the United States Department of Commerce; (ii) any country or entity subject to sanctions administered by the Office of Foreign Assets Control, United States Department of the Treasury; or (iii) such other countries or entities to which export is

VIII • WHAT'S NEW IN MASTERCAM X3

restricted by any other United States government agency. You further agree that you are solely responsible for compliance with any import laws and regulations of the country of destination of a permitted export or re-export, and any other import requirement related to a permitted export or re-export.

Exceptions to Security Mechanism Requirements

CNC software programs Mastercam Design LT and Mastercam Demo as well as printed and electronic documentation do not require the use of Security Mechanisms, and the provisions in this Software License Agreement relating to Security Mechanisms do not apply to your use of such programs, provided, however, that such provisions shall apply to your use of all other Software and documentation provided hereunder.

Survival

All provisions of this Software License Agreement relating to confidentiality, non-disclosure, CNC's proprietary rights, disclaimers, and limits of liability, or indemnification by Customer shall survive termination of this License for any reason.

Reservation of Rights

All rights not expressly granted are reserved by CNC.

Trademarks

Mastercam® is a registered trademark of CNC.

Windows, Windows 2000, Windows XP, and Windows Vista are registered trademarks of Microsoft Corporation. Mastercam Verify is created in conjunction with MachineWorks Ltd. Mastercam Backplot includes Copyrighted intellectual property rights owned by NWD.

Printed in the United States of America.

Printed on recycled paper.

Contents

.....

Introduction	1
▶ Before You Begin.....	2
▶ If You Need More Help.....	2
▶ Additional Documentation.....	3
General System Enhancements	5
▶ Importing SolidWorks History.....	5
▶ <i>Tutorial: Importing SolidWorks history</i>	5
▶ System Configuration Changes	7
▶ Translator Updates.....	8
▶ Zip2Go Improvements	9
▶ Engraving, Nesting, and Rast2Vec Licensing Changes.....	11
Design Enhancements	13
▶ Swept Surface Improvements	13
▶ <i>Tutorial: Creating a swept surface with sharp corners</i>	13
▶ Xform Improvements	15
▶ <i>Tutorial: Using Xform Move to Origin</i>	15
▶ <i>Tutorial: Placing multiple copies using Xform Drag</i>	17
▶ <i>Tutorial: Using Xform Move on a solid model</i>	20
▶ Solids Layout	23
▶ General Selection - Last.....	24
▶ Select by Depth	24
▶ Line Perpendicular.....	25
▶ Shortcut Key for Editing Text	25
General Mill Enhancements	27
▶ New Toolbar Buttons on Tree-Style Dialogs	27
▶ Improved Support for STL Entities.....	27
▶ Prompt When Deleting Operations.....	28
▶ Verify Improvements.....	28

Feature Based Machining	31
▶ FBM Mill.....	32
▶ <i>Tutorial: Creating an FBM Mill toolpath</i>	33
▶ FBM Drill.....	44
▶ <i>Tutorial: Creating an FBM Drill toolpath</i>	45
▶ Using SolidWorks Hole Wizard Data.....	54
▶ <i>Tutorial: Importing SolidWorks Hole Wizard data</i>	54
Mill Level 1 Enhancements	59
▶ 2D High Speed Toolpaths	59
▶ New Cutting Method for Facing Toolpaths	64
Mill Level 3 Enhancements	65
▶ Formula Files.....	65
▶ New Climb/Conventional Options for HST Toolpaths	65
▶ HST Tool Holders.....	66
▶ Multiaxis Edge Blending	66
▶ <i>Tutorial: Adding edge blending to a multiaxis toolpath</i>	66
Router Enhancements	71
▶ Single File ATP Processing.....	71
▶ Nesting Block Drill Operations within Machine Limits.....	72
▶ Creating Separate Block Drill Operations.....	72
▶ Engraving Toolpath Multiple Entry Point Options.....	73
Lathe Enhancements	75
▶ Steady Rest C-Hook	75
▶ Tool Catalog and Library Changes	76
▶ VTL Machine Definition Changes.....	76
Wire Enhancements	79
▶ Multiple 4-Axis Wirepaths in One Operation.....	79
▶ Setting UV Tab Widths in 4-Axis Wirepaths	80
▶ Changes to the Planes (WCS) Page in Wirepath Parameters..	81
▶ Autosync Rails.....	82

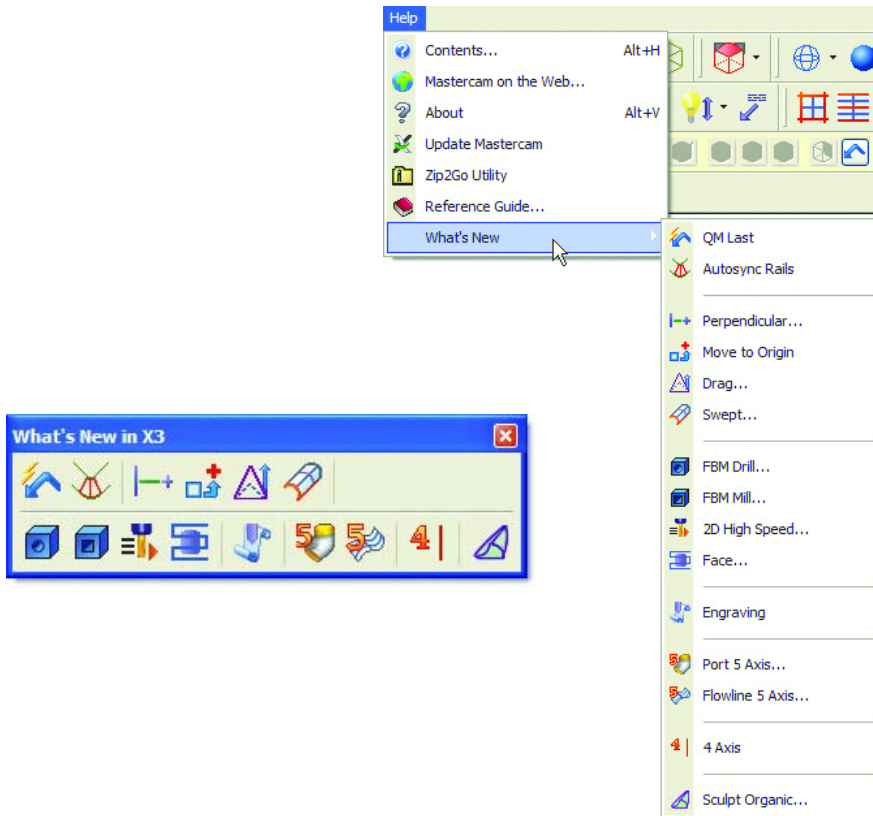
Art Enhancements..... 83

- ▶ Bézier Curve Dynamic Cross Sections..... 83
- ▶ New Sculpt Organic Surface..... 84
- ▶ *Tutorial: Creating a Sculpt Organic surface* 85
- ▶ Art Manager Improvements..... 91
- ▶ Vertex Buffers..... 93
- ▶ Art Shading and Material Mapping Improvements 94

Introduction

Welcome to Mastercam X3! The new Mastercam X3 suite of CAD/CAM products is designed with real-world programming needs at the forefront, making your work easier and faster than ever before. The improvements in Mastercam X3 are focused on providing speed and efficiency for your machining jobs. And because Mastercam can grow with your needs, you have the long term confidence, experience, and support from the #1 CAD/CAM developer in the world.

You can quickly access some of the new and exciting features of Mastercam X3 through the **What's New** toolbar and menu. Go to **Help, What's New** in the Mastercam menu to view some of the new functions that Mastercam X3 offers.



Notes:

- If you migrate your toolbar settings from a previous version of Mastercam, the What's New in X3 toolbar will not be available.
 - Some of the options in the What's New toolbar and menu are available only if you have certain products enabled on your Mastercam SIM.
-

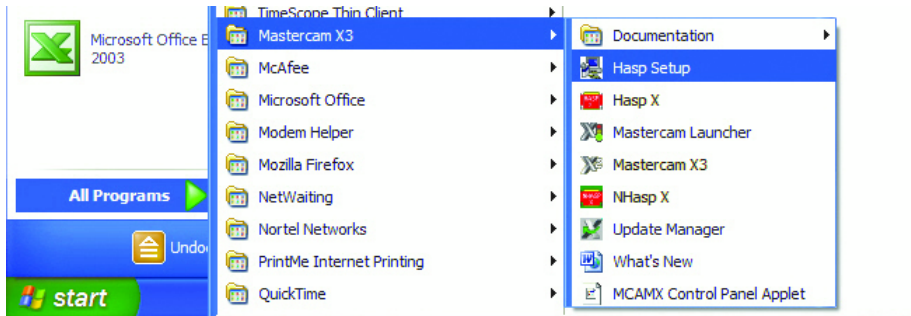
Before You Begin

What's New in Mastercam X3 outlines the new features in each product level of Mastercam. The title of each chapter corresponds to the Mastercam product that contains the features. For example, the Mill Level 1 Enhancements chapter discusses new functionality that you can access if you are a Mastercam X3 Mill Level 1 or higher customer.

Tutorial Requirements


Many of the chapters in this book contain short tutorials to let you try out the new features in Mastercam X3. To complete these tutorials, you should meet the following requirements:

- You must be comfortable using the Windows® operating system.
- You must have installed your Mastercam X3 products. All of the tutorial files are located in the `.. \Documentation\ExampleParts` folder of your Mastercam installation.
- Start Mastercam using your preferred method:
 - ♦ Double-click Mastercam's desktop icon.
 - Or
 - ♦ Launch Mastercam from the Windows Start menu.



If You Need More Help

There are many ways to get help with Mastercam, including the following:

- *Mastercam Help*—Access Mastercam Help by selecting **Help, Contents** from Mastercam's menu or by pressing **[Alt+H]** on your keyboard. Also, most dialog boxes and ribbon bars feature a Help button  that opens Mastercam Help directly to related information.
- *Online help*—You can search for information or ask questions on the Mastercam Web forum, located at www.emastercam.com. You can also find a wealth of information, including many videos, at www.mastercam.com and www.mastercamedu.com.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.

- *Technical Support*—CNC Software's Technical Support department (860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Documentation feedback*—For questions about this or other Mastercam documentation, contact the Technical Documentation department by e-mail at techdocs@mastercam.com.

Additional Documentation

You can find more information on using Mastercam in the following materials, located in the \Documentation folder of your Mastercam installation:

- *Mastercam X3 Installation Guide*
- *Mastercam X3 Transition Guide*
- *Mastercam X3 Getting Started Guide*
- *Mastercam X3 Reference Guide*
- *Mastercam X3 Quick Reference Card*
- *Mastercam X3 Wire Getting Started Guide*
- *Version 9 to X Function Map*

General System Enhancements

No matter what level of Mastercam X3 you have, you can take advantage of these new features to make your programming faster and simpler. Mastercam X3 makes it easier than ever to work with files from other software packages and configure your system to your specific needs.

Importing SolidWorks History

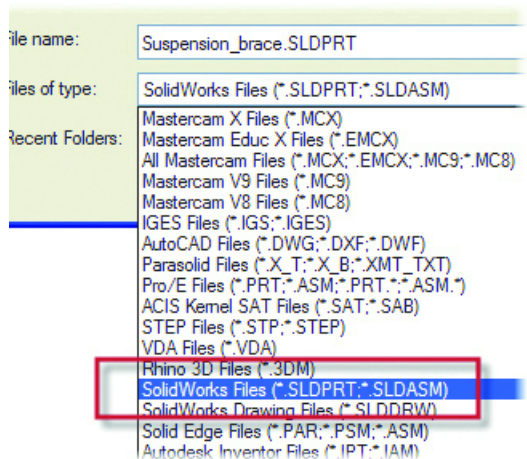
Mastercam X3 can import SolidWorks® solid history data directly into the Mastercam Solids Manager. When you open a SolidWorks part file, click **Options** and select the **Import Solids History** check box. When you click **Open**, Mastercam connects to SolidWorks and reads in the complete solid history. You can then edit the solid using the Mastercam Solids Manager.



IMPORTANT: SolidWorks must be installed on the same system as Mastercam for the history to be imported.

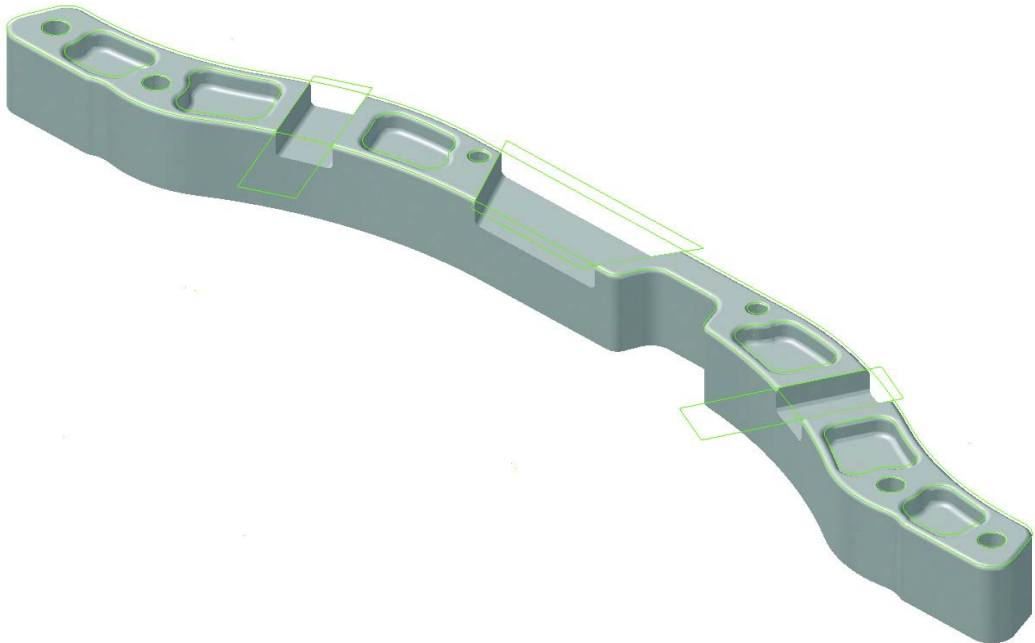
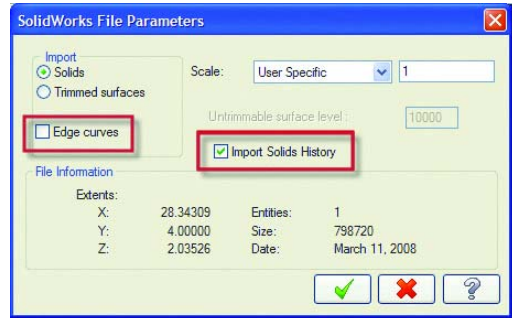
► Importing SolidWorks history

- 1 Start Mastercam X3.
- 2 Choose **File, Open** from the Mastercam menu.
- 3 Select **.SLDPRT** in the **Files of type** drop-down list.
- 4 Click **Options**.



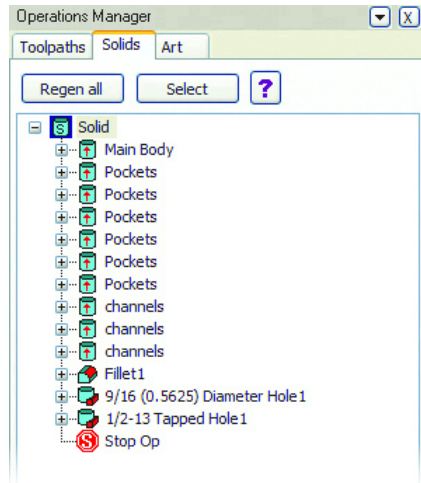
6 • WHAT'S NEW IN MASTERCAM X3

- 5 Deselect **Edge curves** to make it easier to see the solid geometry.
- 6 Select **Import Solids History**.
- 7 Choose **OK**.
- 8 Go to ..\Documentation\ExampleParts and select `SUSPENSION_BRACE.SLDPRT`.
- 9 Click **Open**. Mastercam connects to SolidWorks to read in the history data and opens the SolidWorks file in Mastercam.
- 10 Right-click in the graphics window and select **Isometric Gview** to orient the part in the isometric graphics view.
- 11 Right-click in the graphics window again and choose **Fit** to see the whole part in the graphics window.
- 12 Press **[Alt+S]** to turn on shading. The part should look like the following picture.



- 13** Go to the Solids Manager and expand the solid by clicking the + in front of the Solid icon. All of the history displays.

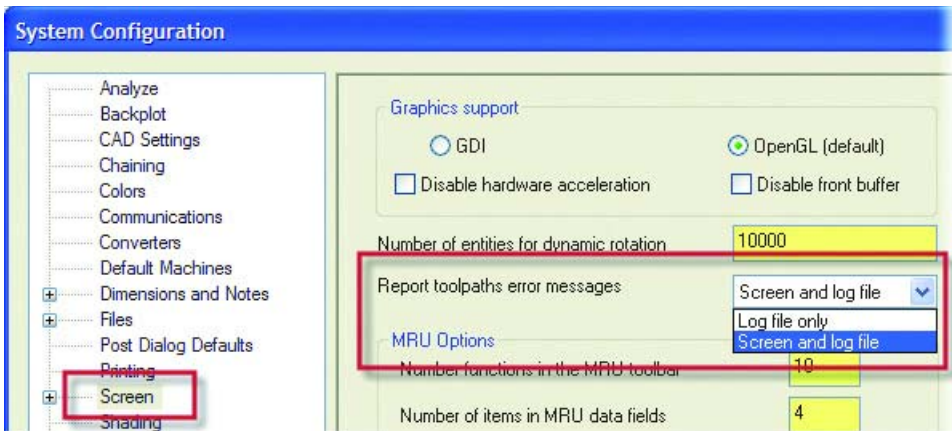
You can now make any necessary edits to the solid, such as suppressing some of the channels and pockets before machining.



System Configuration Changes

Silent Mode

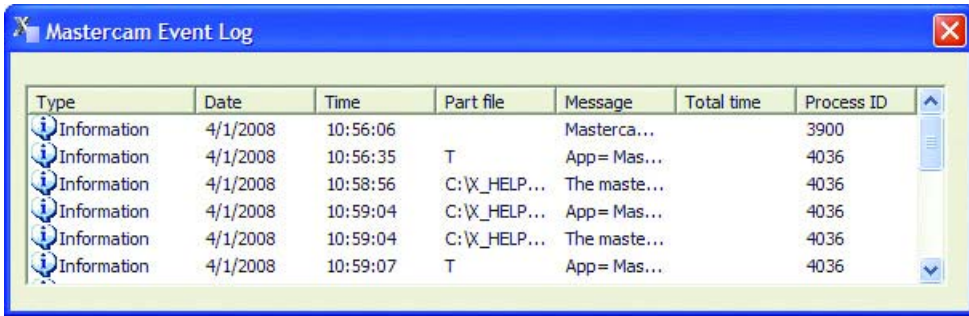
A new option on the Screen page of the System Configuration dialog box forces Mastercam to send error messages to a log file rather than to the screen when you are trying to operate Mastercam in an unattended mode.



This allows other toolpaths to continue to process even if one of them has problems and is unable to complete. Prior to this option, Mastercam would open a warning dialog box and wait until the operator clicked the OK button, possibly losing processing time while the user was away from the machine.

This information is reported in a log file that opens every time Mastercam starts and can be accessed from an icon on the Windows® Taskbar.



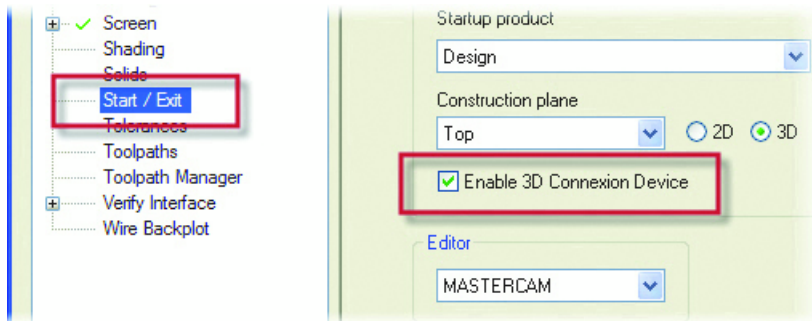


Placing Configuration Files on a Network

Configuration files (*.CONFIG) can now be accessed from any drive on your network. You are not restricted to the ..\Config directory.

3D Connexion Device Support

You can enable or disable your 3D Connexion® navigation device using a new switch on the Start/Exit page of the System Configuration dialog box.



Translator Updates

The following list shows which file formats are supported by Mastercam X3's file translators.

- Parasolids® - 19
- ACIS - R18
- AutoCAD® DWG, DXF, DWF - 2008
- Autodesk® Inventor IPT, IAM, IDW - 2008
- CATIA® V5 - R18
- KeyCreator® - V6
- Pro/E® - Wildfire 4

- Rhino[®] - 4
- Solid Edge[®] - V20
- SolidWorks - 2008
- SpaceClaim - 2008

Zip2Go Improvements

Mastercam's Zip2Go utility gathers and compresses the open (current) Mastercam part data into a .Z2G file. Zip2Go scans the machine groups in the Toolpath Manager of your current file and captures certain information such as your Mastercam configuration, machine definition, post files, and machine group default and operation files. That file can then be opened and viewed with most zip/unzip programs such as WinZip[®]. This is especially useful if you are trying to share file information with Technical Support contacts.

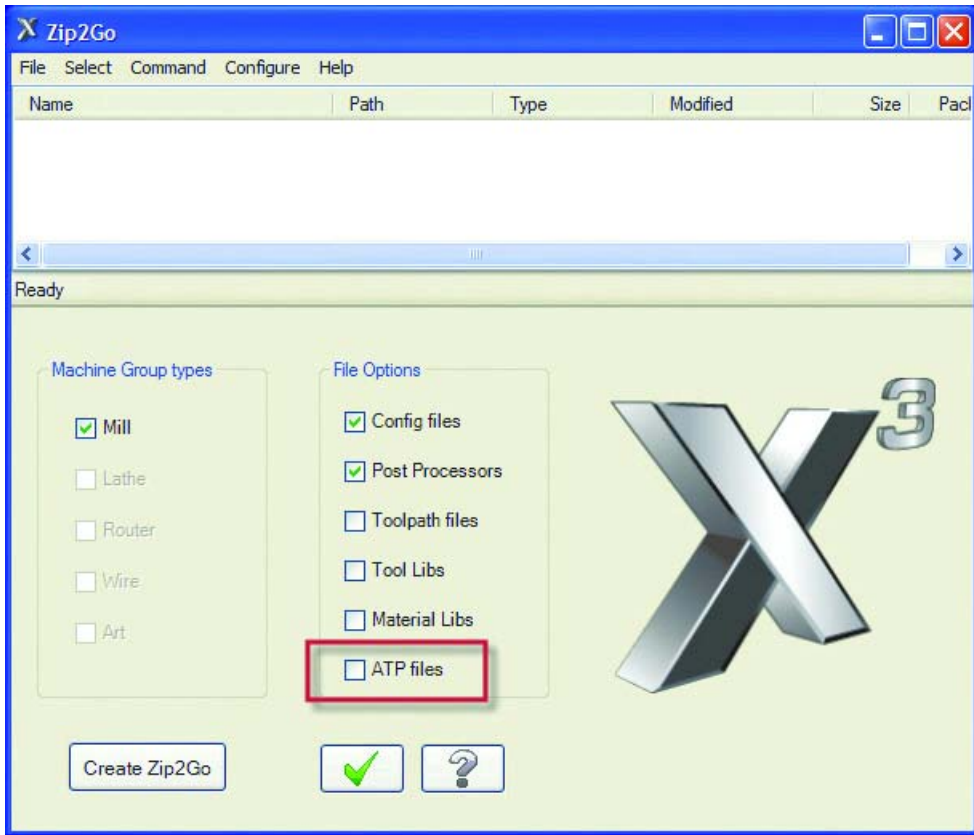
Note: Zip2Go only compresses what is saved on your hard drive. If there are unsaved changes in your file, they are not included in the .Z2G file.

Mastercam X3 includes the following Zip2Go enhancements:

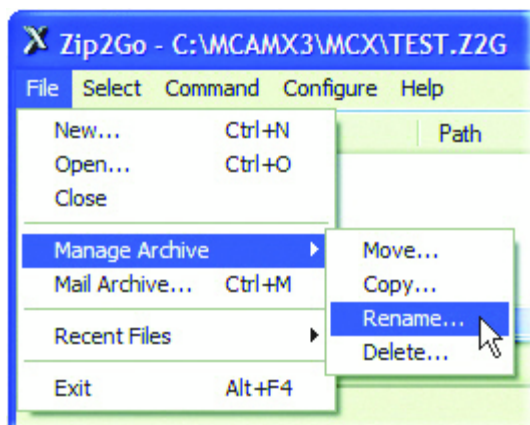
- **.DEFAULTS and .OPERATIONS files** - All files of these types that are listed in the Machine Group Properties are now automatically included in the Zip2Go package.
- **Wire power libraries** - All power libraries that are referenced in any Wire operation are included in the Zip2Go package. You do not have to select the **Tool/Power Libs** option. If you select this option, you will also get the power libraries listed in the Group Properties for each Wire Machine Group.
- **Custom Mill and Lathe tools** - Zip2Go now checks every Mill/Router/Lathe operation for any custom tools that are used (custom tools that are stored in an external MCX file) and automatically includes them in the Zip2Go package. You do not have to select the **Tool/Power Libs** option.
- **STL files** - All Mill/Router operations are checked for the use of an STL file. If there are surface toolpaths that reference an STL file in the Geometry (CAD file) branch of the operation, that STL file is automatically included in the .Z2G file.
- **Report file changes** - The specific version of Mastercam that created the .Z2G file is now included in the Zip2Go_(filename).TXT report file.

```
Zip2Go_(TEST_ZIP2GO).TXT
12/19/2007 11:05:28 AM
Version: X2 MR2 SP0
C:\MCAMX2-MR2\MCX\TEST_ZIP2GO.MCX
```

- **ATP files** - There is a new file option for ATP (Automatic Toolpathing) files. If selected, all the ATP-related files from the last ATP session are included in the Zip2Go file.

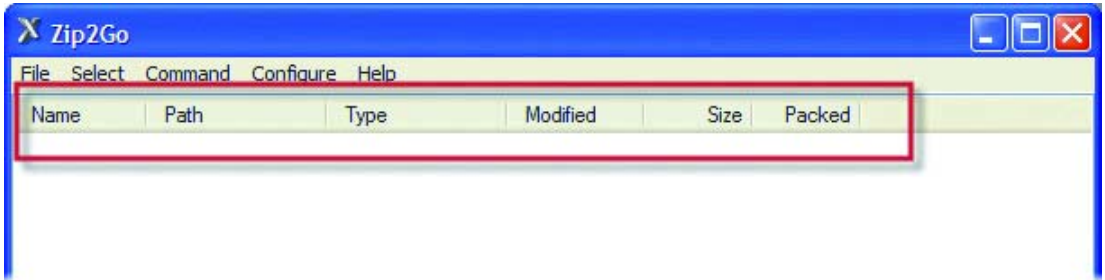


- **File renaming** - There is now a **Rename** command on the **File, Manage Archive** menu for easier file renaming.



- **Drag and drop** - You can now add files to the .z2G file by dragging and dropping them onto the file list area of the Zip2Go dialog.

- **Full file name display** - Zip2Go now truncates the path\file name shown on the title bar to display as much of the filename.Z2G as possible. How much of the file name displays depends on the current size (width) of the Zip2Go dialog.
- **Modal layout** - Zip2Go remembers the layout (order and width) of the columns between sessions.



- **Standard interface** - Zip2Go now uses the Mastercam standard File Open and Save As dialogs.

Engraving, Nesting, and Rast2Vec Licensing Changes

The Engraving, Nesting, and Rast2Vec add-ons are now included in some of the Mastercam X3 product suite.

Table 1: Engraving, Nesting, and Rast2Vec Licensing Changes

Product:	Now included in:
Engraving	Mill Level 1, Mill Level 2, Mill Level 3, Router, Router Pro, Router Plus
Nesting (TrueShape)	Router, Router Pro, Router Plus. Also available as an add-on to Router Entry, all Mill levels, and Design.
Rast2Vec	Mill Level 1, Mill Level 2, Mill Level 3, Router, Router Pro, Router Plus. Also available as an add-on to Mill Entry and Router Entry.

Design Enhancements

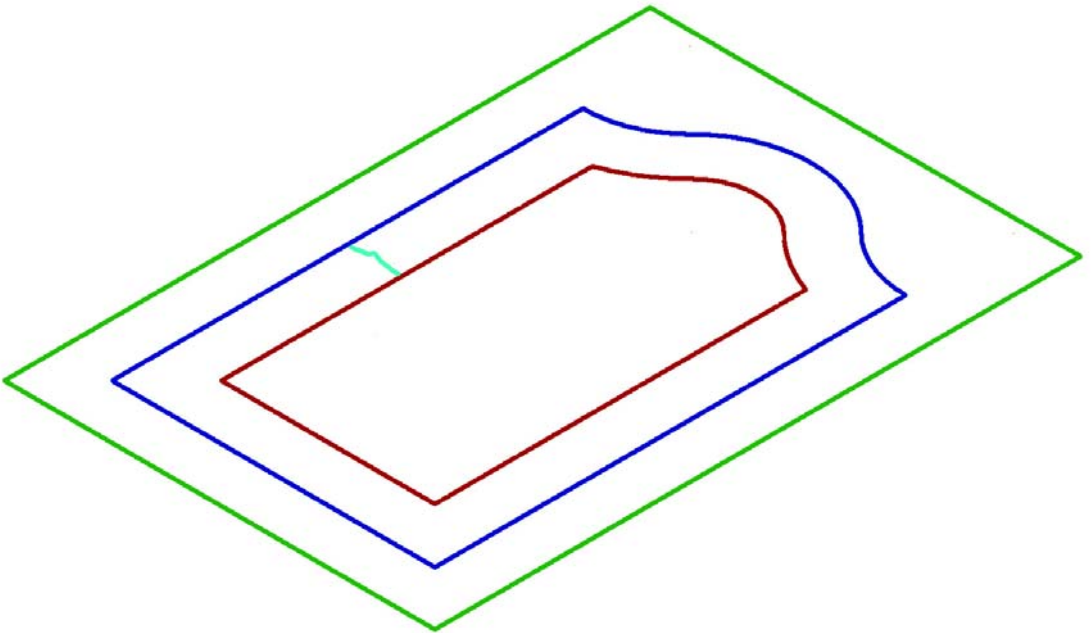
Mastercam X3 Design includes many new features that save you time when drawing and editing your parts.

Swept Surface Improvements

Mastercam X3 now permits sharp corners in the rails. You will no longer receive an error message if the rails contain sharp corners. Mastercam now builds these types of surfaces, but it splits the surfaces into multiple pieces wherever necessary so that individual surfaces do not have sharp corners in them. This will help users who are producing surfaces for door or window frame shapes.

► **Creating a swept surface with sharp corners**

- 1 From the Mastercam menu, choose **File, Open**.
- 2 Go to `..\Documentation\ExampleParts` and select `SHARP_CORNERS_DOOR.MCX`

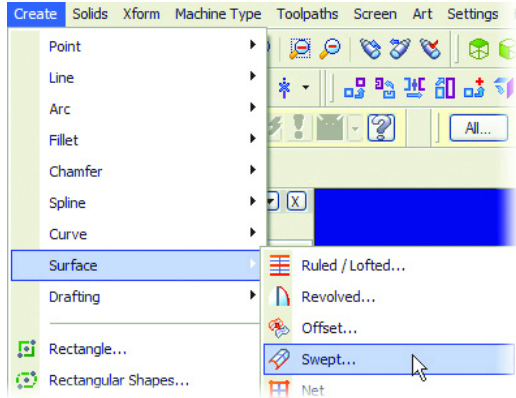


You will sweep the across contour around the doorframe to create a surface.

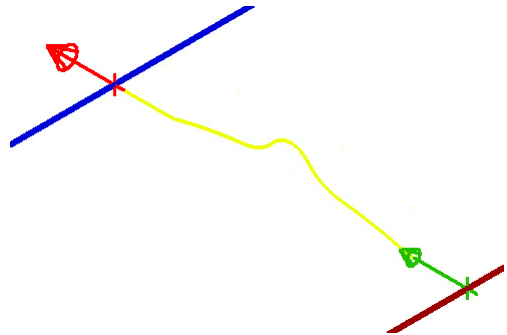
- 3 Choose **Create, Surface, Swept** from the Mastercam menu.



*Note: You could also select the **Create Swept Surfaces** button from the toolbar.*



- 4 Click the **Two Rails** button on the Swept Surfaces ribbon bar. This surface creation method allows the two along curves of the doorframe to control the across curve.
- 5 Define the across contour by selecting the chain that runs across the doorframe. The chain can point in either direction.
- 6 Click **OK** on the Chaining dialog box.



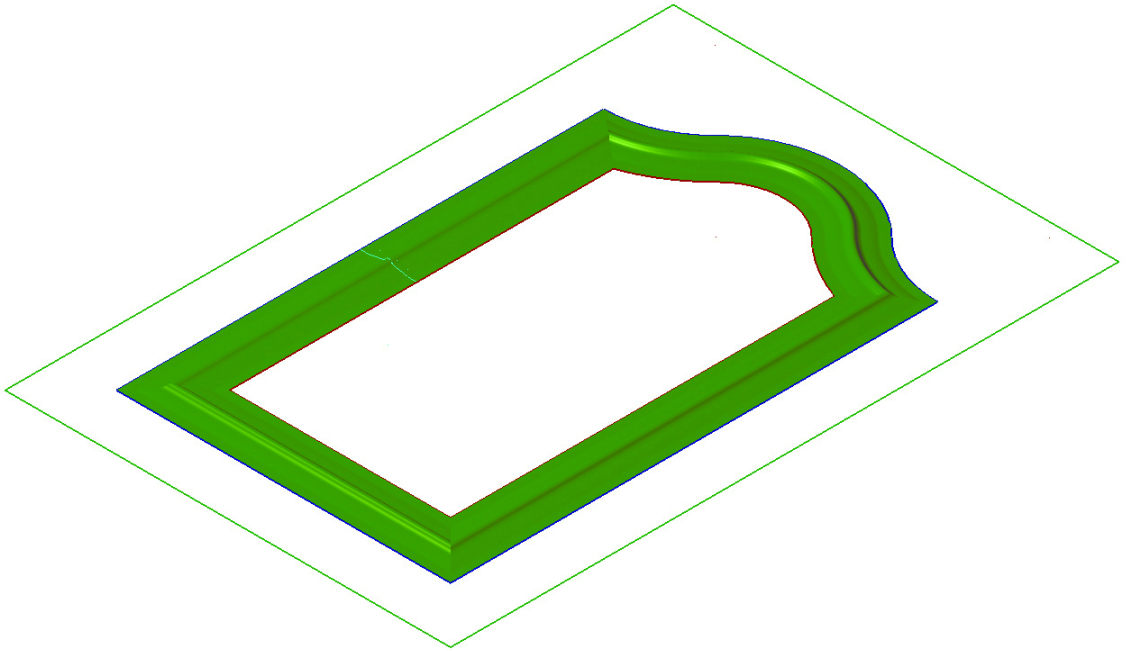
- 7 Chain the blue outer doorframe as the first along curve. You can place the start point anywhere along the chain.
- 8 Click the **End** button on the Chaining dialog box to complete the first along curve.
- 9 Chain the brown inner doorframe. Like the outer frame, you can place the start point anywhere along the chain.



Five swept surfaces are generated. In previous versions of Mastercam, you would have received a warning that the along curves had sharp corners and no surfaces would have been created.

- 10 Press **[Alt + S]** to shade the new surfaces.

- 11 Click **OK** on the Swept Surfaces ribbon bar to accept the results.



Xform Improvements

Mastercam's Xform functionality has been streamlined to make part creation and setup faster and easier.

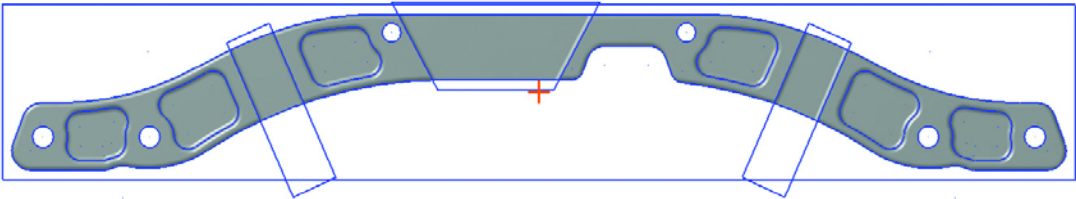
Xform Move to Origin

This new function helps with automating part setup by moving all visible geometry to the current Work Coordinate System (WCS) origin. Once the function is started, you are prompted for an AutoCursor position. Once the position is selected, all of the geometry moves to the origin relative to that position.

► Using Xform Move to Origin

- 1 From the Mastercam menu, choose **File, Open**.

- 2 Go to ..\Documentation\ExampleParts and select SUSPENSION_BRACE_ORIGIN.MCX.

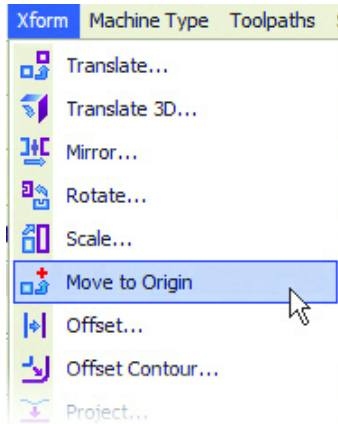


The origin for this part needs to be changed before machining.

- 3 Choose **Xform, Move to Origin** from the Mastercam menu.



*Note: You could also select the **Xform Move to Origin** button from the toolbar.*



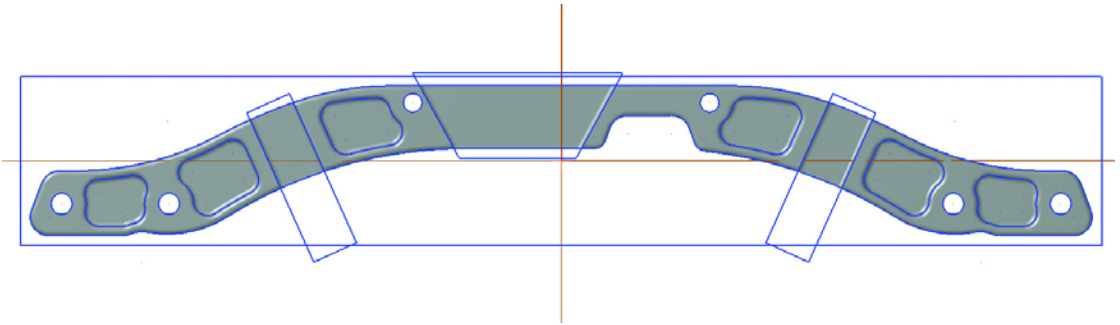
- 4 Select the orange point in the center of the part.

Mastercam moves all the entities to the current WCS origin relative to the selected point.

- 5 Press **[F9]** to display the system origin.



- Right-click in the graphics window and choose **Fit** to show the whole part in the graphics window.



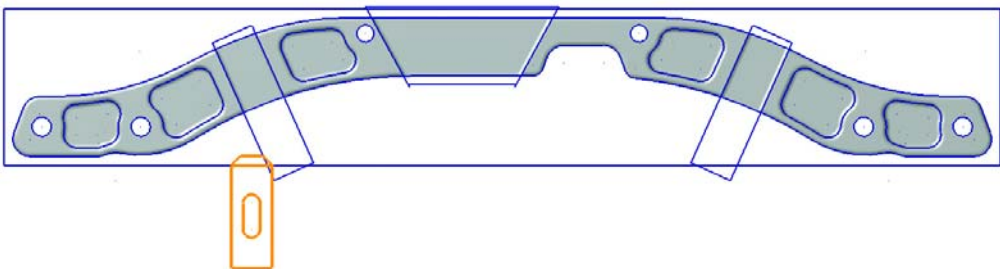
The part is now ready to be machined.

Xform Drag

The Xform Drag function in Mastercam X3 has been improved to reduce mouse clicks. You can now select the **Multiple** button on the Xform Drag ribbon bar to place multiple copies of the geometry without having to press [Enter] after each location to fix the live entities.

► Placing multiple copies using Xform Drag

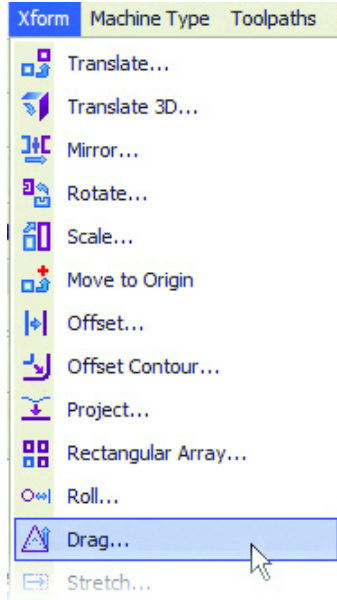
- From the Mastercam menu, choose **File, Open**.
- Go to `..\Documentation\ExampleParts` and select `SUSPENSION_BRACE_DRAG.MCX`



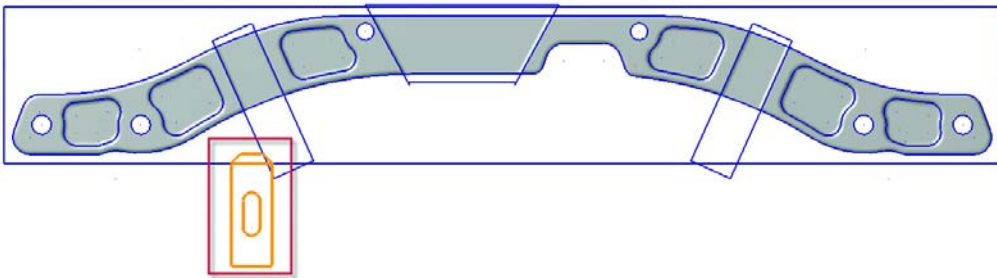
- 3 Choose **Xform, Drag** from the Mastercam menu.



*Note: You could also select the **Xform Drag** button from the toolbar.*



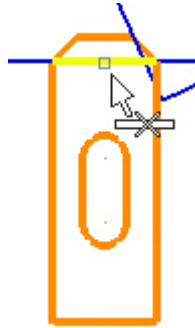
- 4 Use Window selection to select the orange geometry and press **[Enter]**.



- 5 Select the **Multiple** button on the Drag ribbon bar to place multiple copies of the geometry.



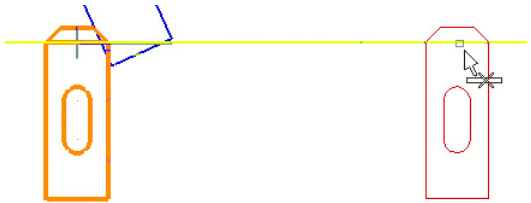
- 6 Select the midpoint of the top straight line as the translation base point.



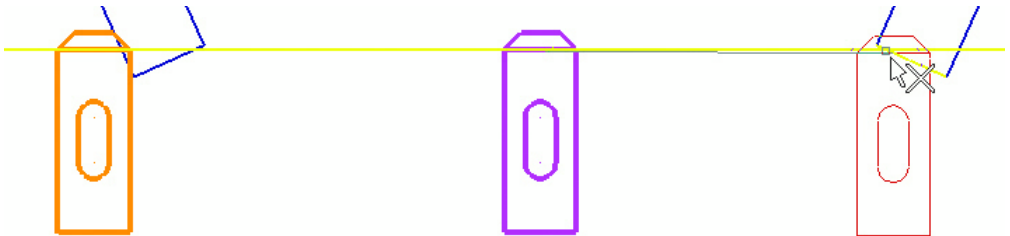
Notes:

- The Visual Cues icon changes depending on what type of geometry you are near. This icon indicates the midpoint of a line.
- If the midpoint icon does not display, right-click in the graphics window and choose **AutoCursor**. Select the **Midpoint** check box and click **OK** to have AutoCursor detect midpoints.

- 7 Drag your mouse to the right and click once when you get to the midpoint of the lower blue line. This places your first copy.



- 8 Drag your mouse to the right again until you get to the intersection of the lower blue line and the right rectangle. Click again to place a second copy.



Note: The Visual Cues icon changes to indicate a geometry intersection.

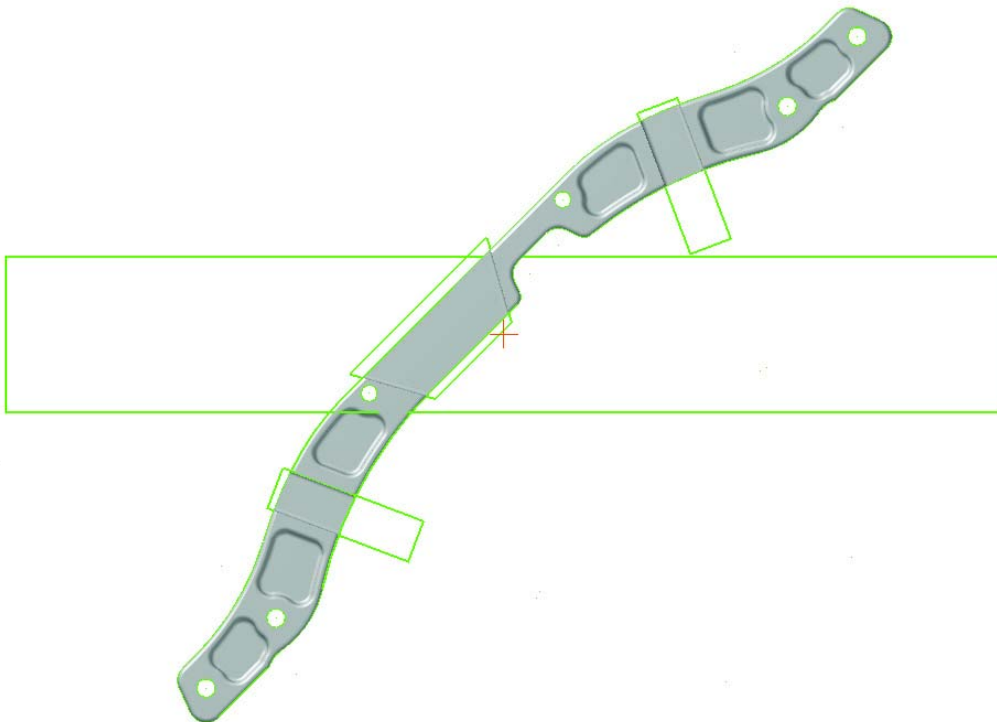
- 9 Continue to click around the part to place copies in other areas of the part.
- 10 Press **[Esc]** or click the **OK** button on the Drag ribbon bar to complete the process.

Xform Move for Solids

Mastercam X3 improves the logic for when a solid is moved and the underlying geometry is not included in the selection. Mastercam can move the associated wireframe geometry, even if it is on a level not being displayed, or make a copy of the geometry. This helps you to manage the wireframe data required for solids.

► Using Xform Move on a solid model

- 1 From the Mastercam menu, choose **File, Open**.
- 2 Go to `..\Documentation\ExampleParts` and select `SUSPENSION_BRACE_MOVE.MCX`.

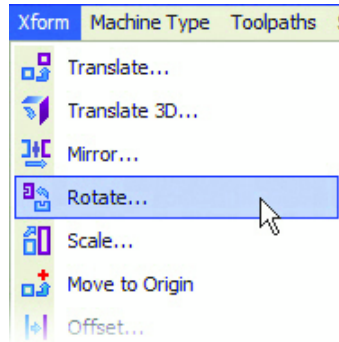


- 3 Choose **Xform**, **Rotate** from the Mastercam menu.

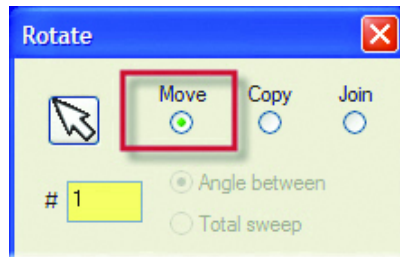


*Note: You could also select the **Xform Rotate** button from the toolbar.*

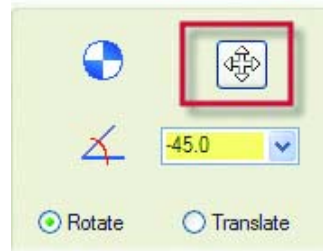
- 4 Click anywhere on the gray solid to select it for rotation and press **[Enter]**.



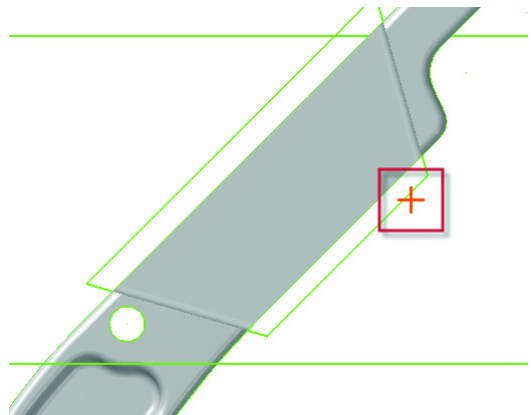
- 5 Select the **Move** radio button at the top of the Rotate dialog box.



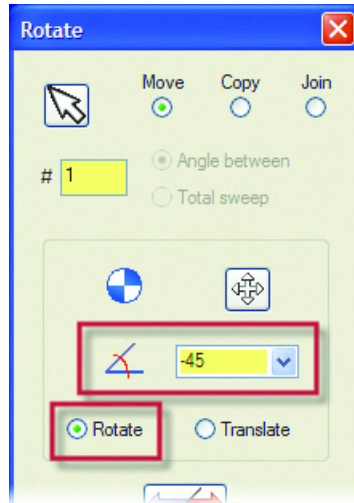
- 6 Click the button with the four-headed arrow to select the rotation center point in the graphics window.



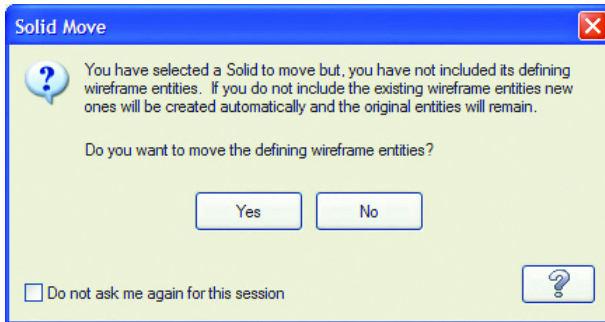
- 7 Select the orange point near the center of the part.



- 8 Enter **-45** for the rotation angle and make sure the **Rotate** option is selected.

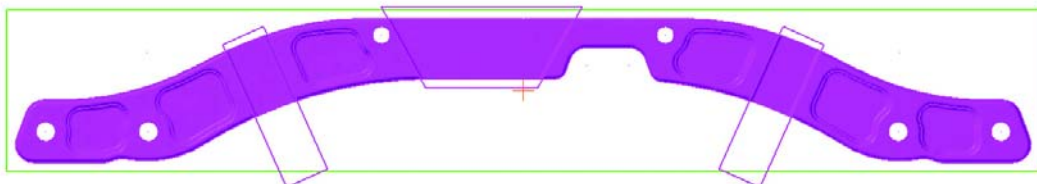


- 9 Click **OK**. Mastercam displays a message about moving the original wireframe entities along with the solid model.



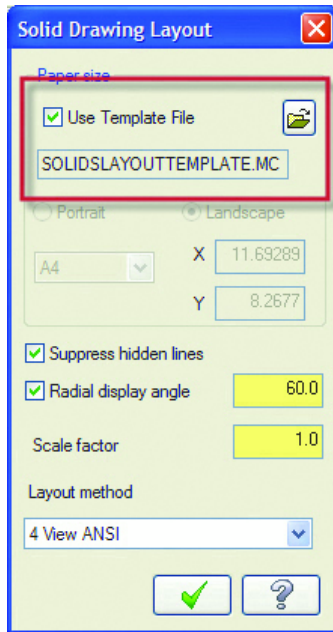
- 10 Click **Yes** to move both the solid and the defining wireframe entities. The results are shown in the following picture.

*Note: If you choose **No**, Mastercam makes a copy of all the wireframe entities in the new location.*

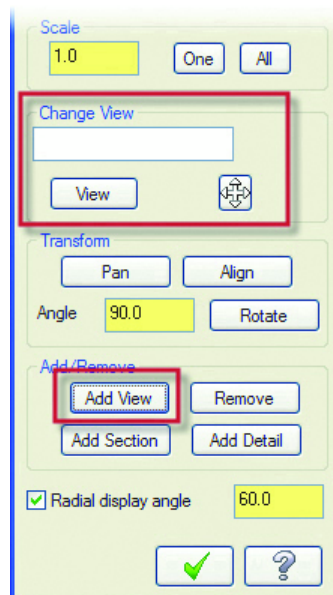


Solids Layout

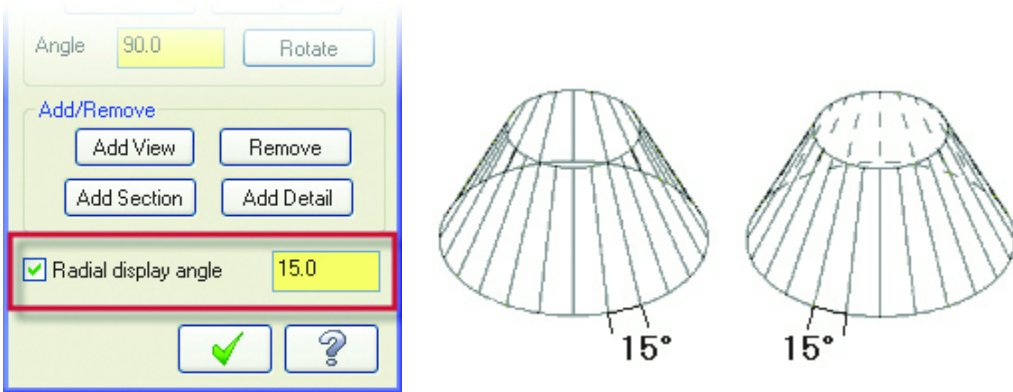
Mastercam X3 includes simpler Solids Layout dialogs for easier use. In addition, you can select a template file to add borders or other entities to the drawing layout.



References to view numbers have been replaced with named views for easier selection, such as on the Solid Drawing Layout Parameters dialog box.



The new **Radial display angle** field adds or removes radial display lines in solid layout views that represent the surfaces of closed, circular solids (cones, cylinders, spheres, tori). If deselected, Mastercam removes all radial lines from your solid layout. When selected, the angle you set defines the position and number of radial lines that are created. In the following picture, the Radial display angle for both the solid's wireframe and hidden line display is 15 degrees.



General Selection - Last

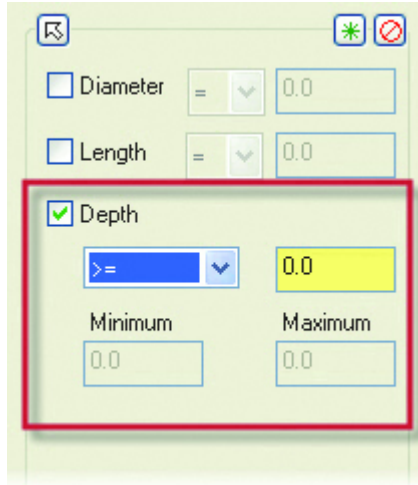
You often need to perform sequential functions on geometry selected with General Selection. Unfortunately, after the function is performed, the list of selected entities is cleared. This forces you to do the selection again for the next function, which can take a bit of effort on complex parts.

Mastercam already has a **Last** function when the Solids option is used in General Selection. This new addition makes the Last functionality available for all General Selection. It is now available on the General Selection toolbar, regardless of the selected entity type, and in the QuickMask toolbar.



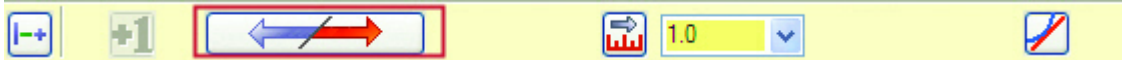
Select by Depth

This capability allows you to focus selection to a specific Z depth or to a zone defined by minimum and maximum Z values. This option applies to all points, lines, arcs, and 2D splines and allows you to set conditions equal to, greater than, greater than and equal to, less than, or less than and equal to a specific Z value. It has been added to the Miscellaneous section of the All and Only options on the General Selection toolbar.



Line Perpendicular

This ribbon bar now includes a **Flip** button for easier line construction.



Shortcut Key for Editing Text

Mastercam X3 adds a new shortcut key for editing text when placing a dimension. The letter **J** as found in **Adjust Text** is now the shortcut key.

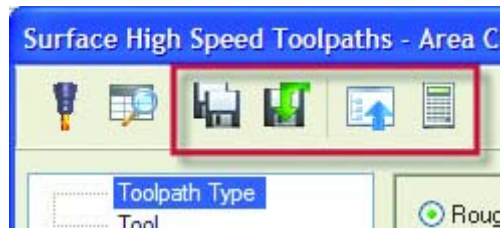
General Mill Enhancements

The new features in all Mastercam X3 Mill products help you produce more parts in less time by improving efficiency and reducing programming time.

New Toolbar Buttons on Tree-Style Dialogs

To make it easier to access some common toolpath functions, four toolbar buttons have been added to the toolpaths that use tree-style dialogs, such as FBM Drill and Mill, 2D High Speed, Surface High Speed, and Saw toolpaths (Router only).

These functions could previously be found only on the right-click menu of the Tool page. Some buttons are unavailable for certain toolpath types.



Save parameters to defaults - Makes the current toolpath settings the default for future toolpaths of this type. The current values are saved to the .DEFAULTS file specified in the machine group properties.



Reload parameters from defaults - Overwrites the current values with the values from your .DEFAULTS file. These are read from the .DEFAULTS file specified in the machine group properties.



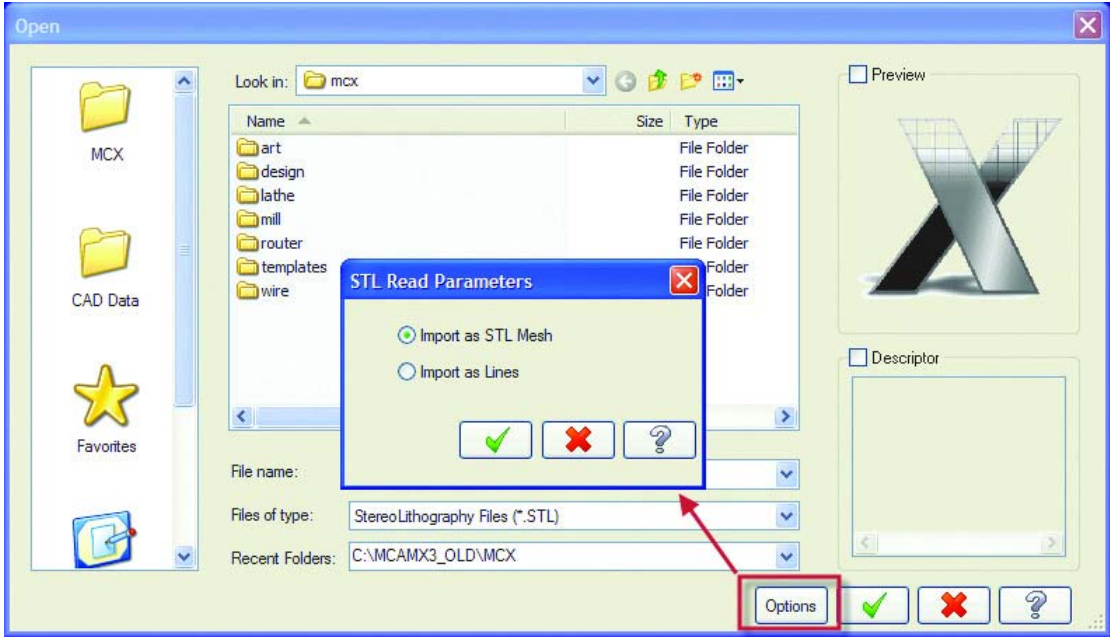
Import operations - Imports one or more operations from a library and automatically applies them to the current geometry selection.



Feed and speed calculator - Calculates feeds and speeds including surface speed or feed per tooth.

Improved Support for STL Entities

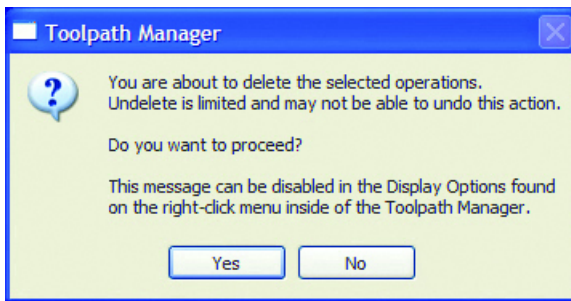
New database support has been added to Mastercam that allows an STL model to be loaded into Mastercam, shaded, transformed, and used for machining. On the File Open dialog, when an STL file type is selected, an **Options** button allows you to import the model as lines or as the new STL Mesh format.



Once imported as an STL Mesh, the information is saved inside the MCX file just like all other entities (surfaces, solids, wireframe, text, dimensions). Any time an option in a toolpath offers the use of a CAD file, the STL Mesh option is also available.

Prompt When Deleting Operations

Since the undelete capabilities in the Toolpath Manager are restricted, Mastercam X3 adds an option to prompt you when selected operations are deleted. This option can be turned off in the **Display options** found on the right-click menu of the Toolpath Manager.

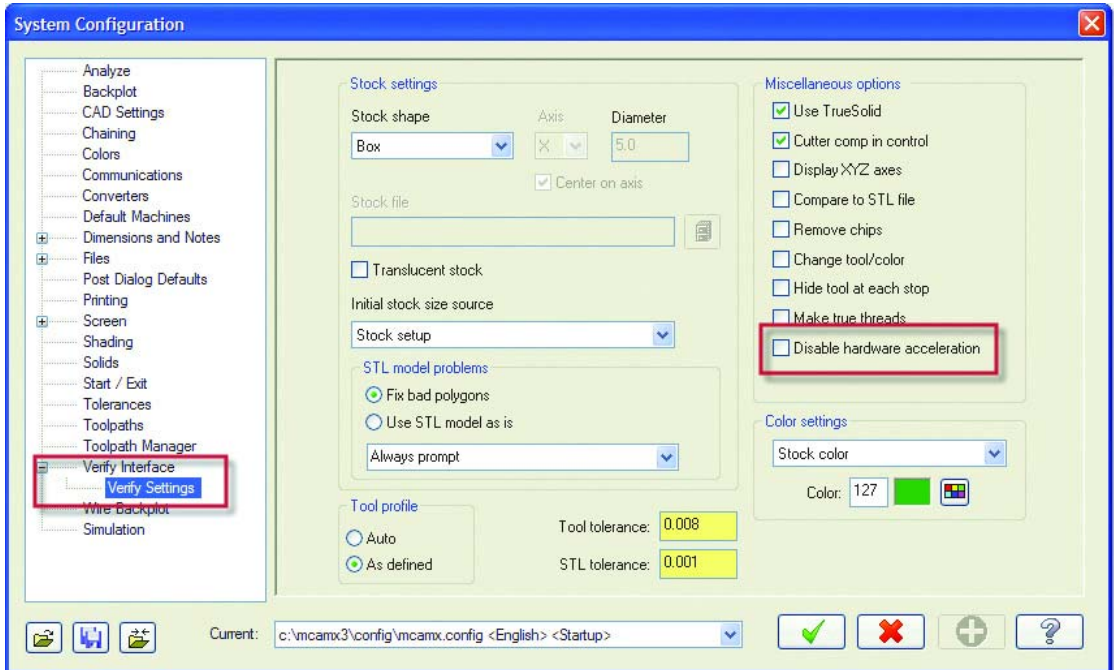


Verify Improvements

Mastercam X3's solid model toolpath verification makes it faster and easier to simulate your toolpath motion.

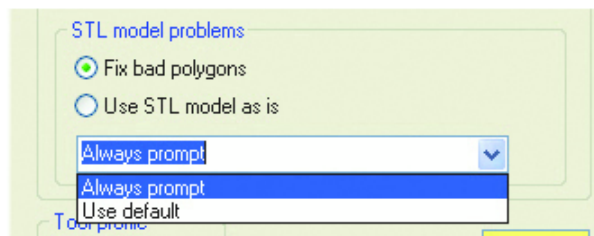
Speed Improvements

Mastercam X3 activates an OpenGL[®] option that greatly enhances the speed of Verify. The **Disable hardware acceleration** option on the Verify Settings page of the System Configuration dialog box disables this new feature if it causes system problems.



STL File Handling

The Verify Settings page of the System Configuration dialog box has some new controls for handling problems with STL stock models. You can use the model as is or “waterfall” it to fix the problem areas. These options also set the default state of the prompt that displays when Mastercam finds an issue in the STL model. You can set it to always show the prompt dialog box or skip the prompt and just apply the default behavior.

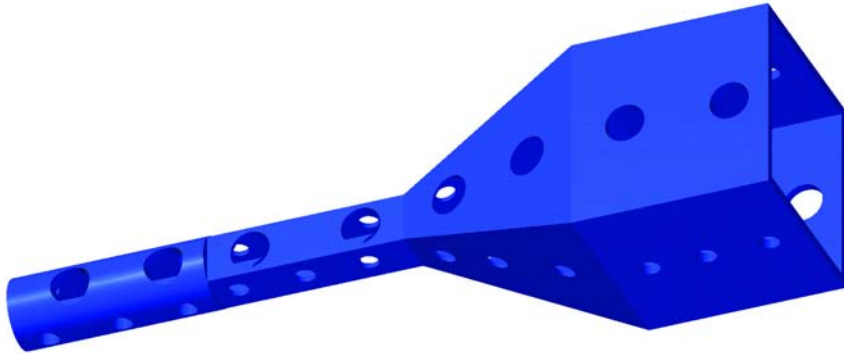


TrueSolid 5-Axis

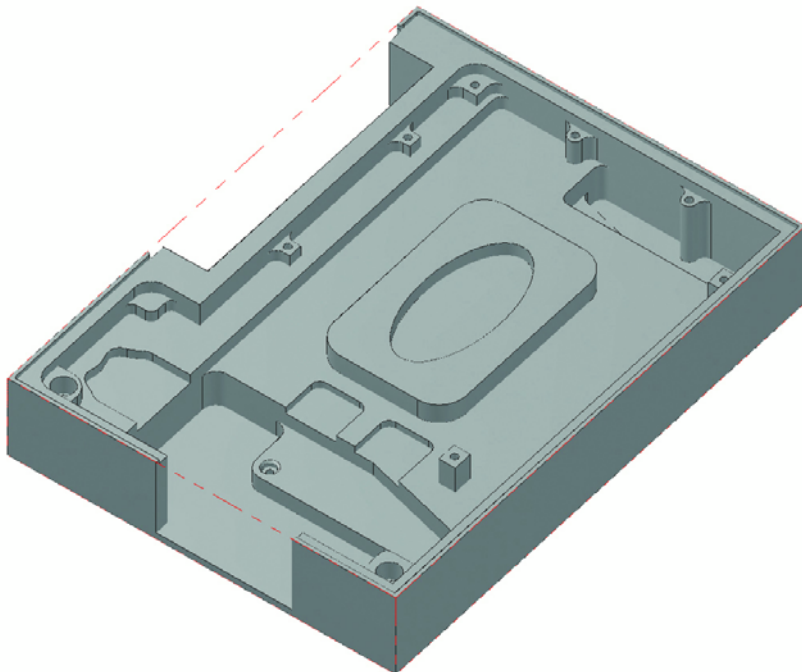
This function no longer requires a separate SIM code to function. It is available with any Multiaxis SIM code. Maintenance is not required.

Feature Based Machining

One of the highlights of Mastercam X3 is the introduction of Feature Based Machining (FBM). FBM, which is available with Mastercam Mill Level 1 or Mastercam Router and higher, eliminates the manual processes involved in identifying features for programming milling and drilling operations on solid parts.



Features define a solid part's topology. They describe the part's physical properties including its faces and the shape, size, and location of holes, slots, pockets, bosses and other characteristics. FBM analyzes the part for specific feature types and automatically creates the individual toolpaths needed to machine the selected features.



Mastercam X3 includes two types of FBM toolpaths - FBM Mill and FBM Drill. Both types use the information derived from the part's features in combination with the machine group's stock definition.

With this information and the guidelines you define for the FBM operation type (Mill or Drill), Mastercam performs the following tasks:

- Detects all features for the selected FBM operation type, based on your selection criteria. You can then review all of the identified machining features, and selectively modify or remove features from the list before generating toolpaths.
- Intelligently designs an effective machining strategy.
- Selects the appropriate tools, either from your preferred tools list or from libraries you specify.
- Creates and assigns boundaries needed to drive or constrain tools.
- Automatically generates all of the toolpaths necessary to machine the features.

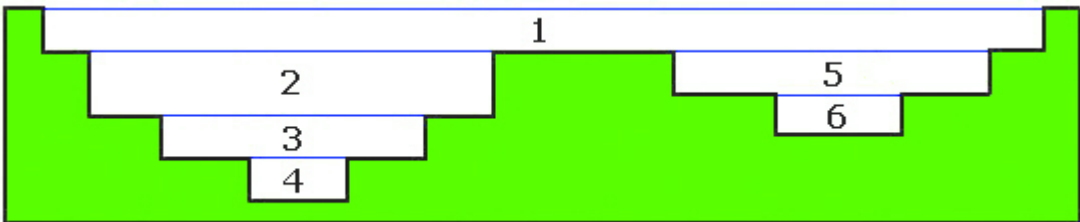
You can easily change parameters in the FBM operation and regenerate all of the toolpaths. You can also customize the generated toolpaths before posting. Associativity is maintained between the FBM parent operation and the generated toolpaths. FBM Drill also maintains associativity with the solid model. Mastercam notifies you when any of these associative entities have been modified, marking them “dirty.”

Note: For the latest information on the FBM toolpaths, please see the Mastercam Help.

FBM Mill

With a single operation, FBM Mill analyzes a solid part, detects all machining features in a specified plane, and automatically generates all of the 2D milling toolpaths necessary to completely machine the selected features.

FBM Mill machines closed, open, nested, and through pockets. For complex nested pockets, Mastercam creates a separate zone for each depth and creates the boundaries required to machine it. In the following example, six zones were created and machined separately within a single nested pocket.



From a common set of parameters, FBM Mill creates the following types of 2D toolpaths:

- facing if stock in the Z axis is above the top of the part (optional)
- roughing and restmill for each zone

- separate finish toolpaths for walls and floors in each zone (optional)
- outside contours if the stock extends beyond the part in the X and Y axes (optional)

You can control the tool selection by using tools already in the .MCX file, pre-defining up to ten preferred tools per operation type, choosing a tool library, or allowing Mastercam to create tools, when necessary.

FBM Mill Requirements

- Mastercam Solids, and Mastercam Mill Level 1 or Mastercam Router and higher
- Mill or Router machine definition (select from the **Machine Type** menu)
- Stock model (define in the **Machine Group Properties, Stock Setup** tab)
- At least one solid part model in the current MCX file with one or more open, closed, nested, or through pockets with flat floors and straight walls (90 degrees)

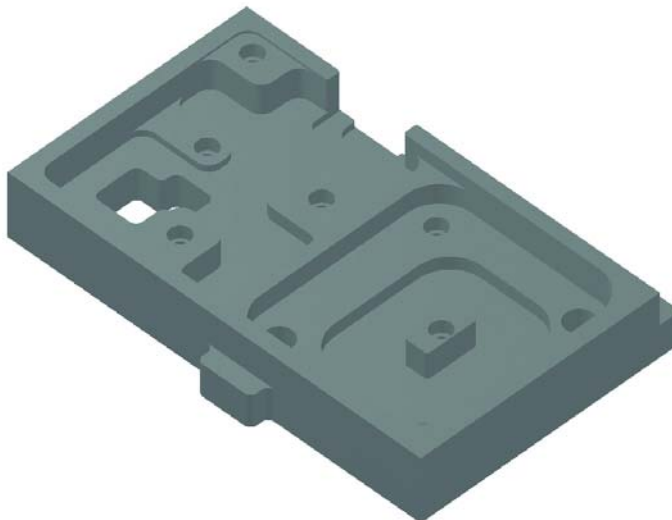


TIP: If you customize FBM-generated toolpaths, do so only when you are sure that you will not regenerate the FBM parent operation. Any customization is overwritten when the FBM-generated toolpaths are recreated.

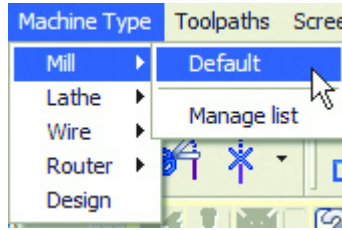
► Creating an FBM Mill toolpath

Setting Up Your Part

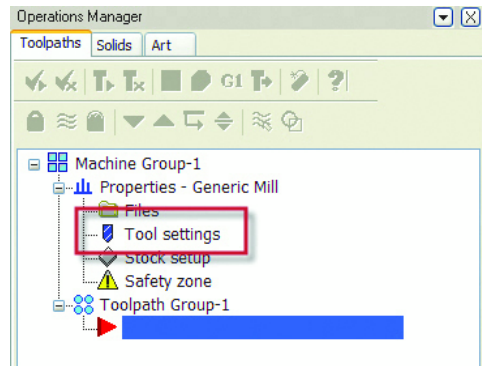
- 1 From the Mastercam menu, choose **File, Open**.
- 2 Go to `..\Documentation\ExampleParts` and select `FBM_MILL_X3.MCX`.



- 3 Choose **Machine Type, Mill, Default** from the Mastercam main menu to use the default Mill machine definition.

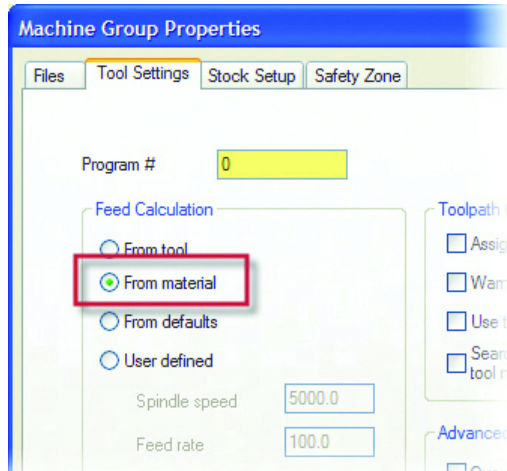


- 4 In the Toolpath Manager, click the **Tool settings** icon in the Machine Group Properties tree control.



- 5 In the Feed Calculation section, select **From material** to calculate each tool's feeds and speeds based on the part's material.

Selecting this option assures that the feeds and speeds will be appropriate to the type of material you plan to machine. The material set for this part is Aluminum 2024, which is shown at the bottom of the page. If the material changes, the feeds and speeds update accordingly.



*Note: If you select the **From tool** option and you have not set feeds and speeds for the tools in your tool library, a warning message displays after the FBM toolpaths are generated.*

- 6 Click the **Stock Setup** tab at the top of the Machine Group Properties dialog box.

FBM Mill requires a defined stock model before it can analyze the part.

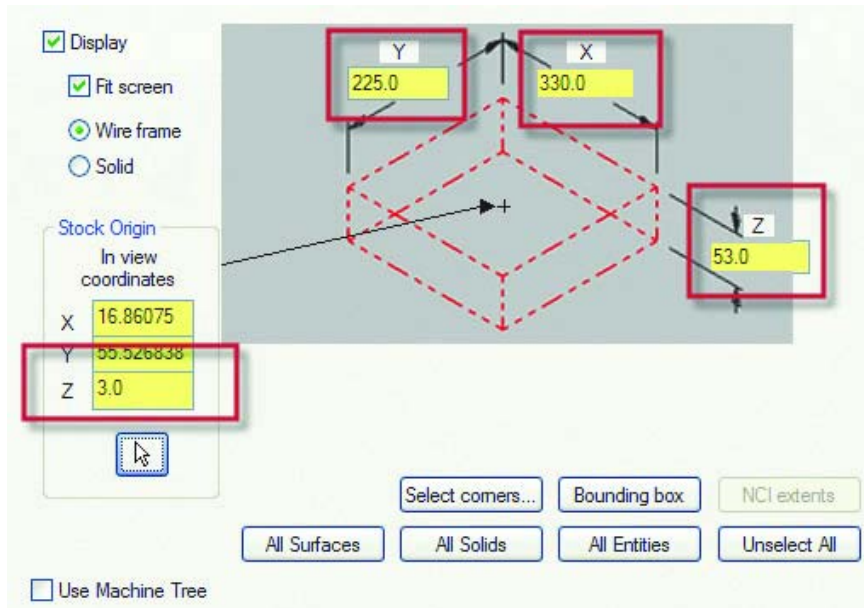
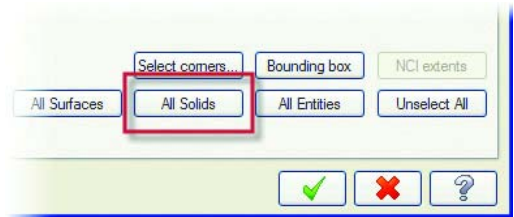
- 7 Click the **All Solids** button to calculate the stock dimensions based on all visible solids in the part.

This updates the XYZ coordinates for the stock dimensions and stock origin.

- 8 Adjust the stock dimensions to the following values:

- ♦ X = 225.0
- ♦ Y = 330.0
- ♦ Z = 53.0
- ♦ Stock Origin Z = 3.0

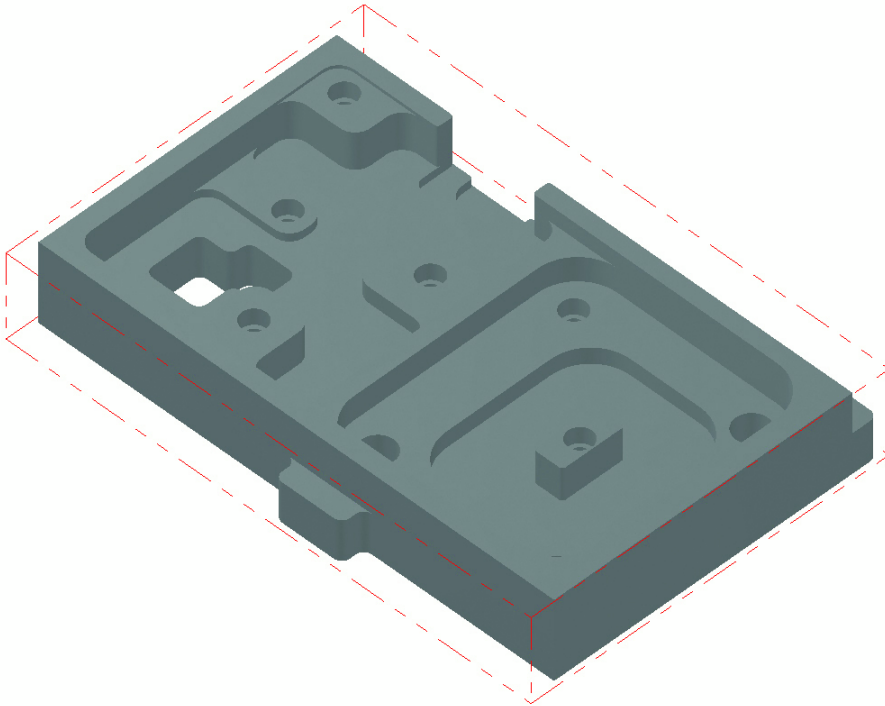
This adds some stock above the part that can be removed during a facing operation.



- 9 Click **OK** to close the Machine Group Properties dialog box.

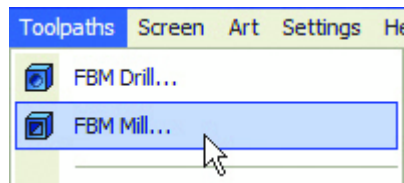


The stock boundaries display around the part in the graphics window.

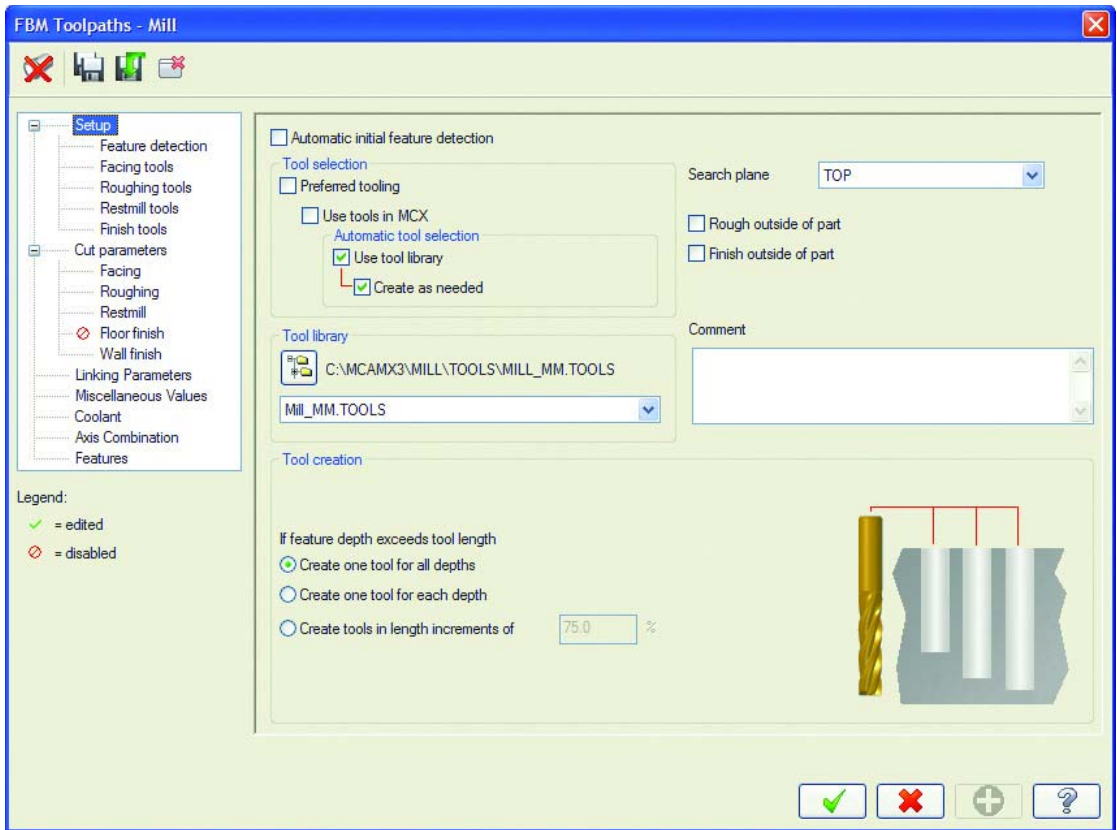


Entering FBM Mill Parameters

- 1 Choose **Toolpaths**, **FBM Mill** from the Mastercam menu.

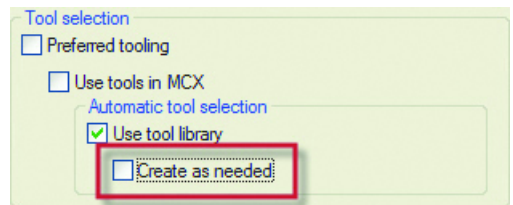


The FBM Mill dialog box displays.



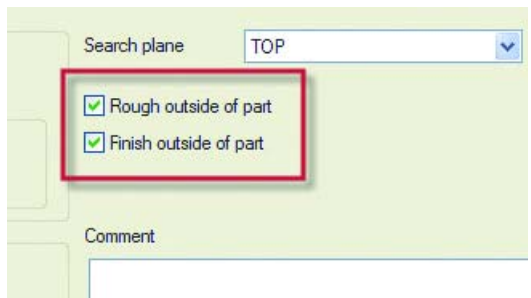
- 2** On the **Setup** page, deselect **Create as needed** in the Automatic tool selection section.

FBM will use only tools in the selected tool library. It will not create additional tools when generating the toolpaths.

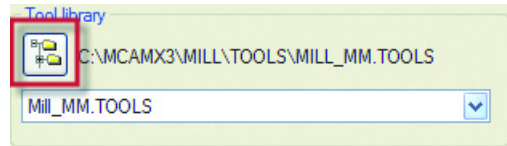


- 3** Also on the **Setup** page, select **Rough outside of part** and **Finish outside of part**.

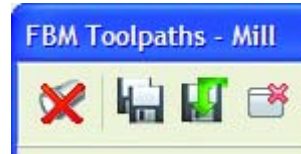
These options add 2D roughing and finishing operations to cut the part's outside profile.



Note: You can select a different tool library for the automatic tool selection by clicking the folder button on the Setup page. But your tutorial results will vary from what is shown in this book.



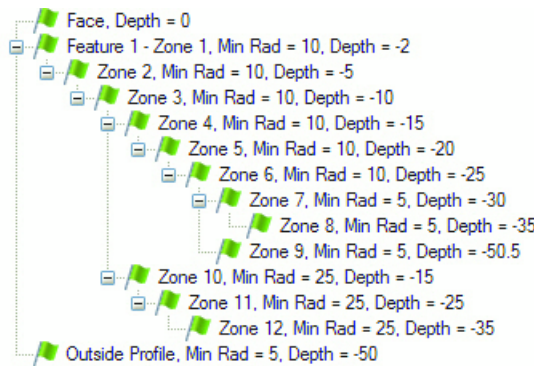
4 Click the **Detect** button at the top of the FBM Mill dialog box. This analyzes the solid model and finds all the machinable features based on the FBM Mill parameters.



TIP: The red X over the Detect button indicates that the FBM operation does not include the most recent set of features on the part. Once the part is re-analyzed, the red X is removed.



The results are listed on the Features page. The Features and Zones are listed in a tree-style format starting with **Face** and ending with **Outside Profile**.












A *Feature* can be something as simple as a Facing operation, which is automatically recognized when there is extra stock in the Z axis above the model. Or it can be a complex feature consisting of a pocket with any combination of bosses, open pockets, benches, or nested pockets.

FBM Mill then breaks complex features down into simpler *Zones* that are recognized as discrete subsets of the larger Features. The placement of features and zones in the features list matches the part topology from the top down.

The icons in the Features tree are designed to help you understand how FBM Mill will interact with the part topology to machine the part. The green flag next to each zone indicates that Mastercam can successfully generate toolpaths to completely machine the area.

Legend:

-  = edited
-  = disabled
-  = Internal sharp corner
-  = Fillet / tool mismatch
-  = Rough tool doesn't fit
-  = Finish tool burial
-  = Restmill uncut stock
-  = Zone can be machined
-  = Feature suppressed

- 5 Click **OK** to accept your FBM Mill parameters.

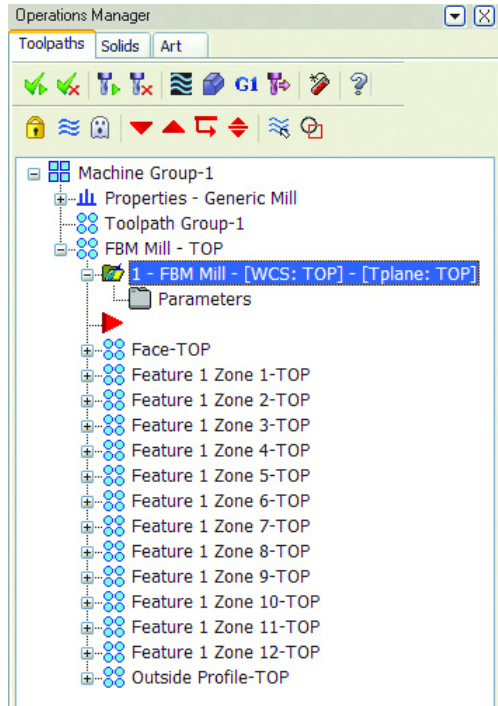
Mastercam generates the toolpaths necessary to machine the part. The toolpaths will take several minutes to process.



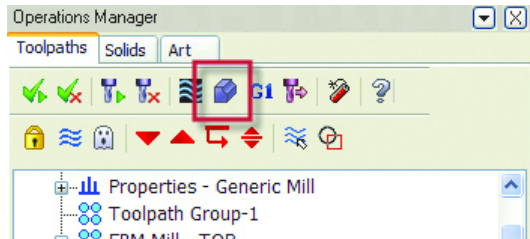
Verifying the FBM Mill Toolpaths

- 1 Click on the - sign in front of each resulting toolpath group in the Toolpath Manager to collapse each group.

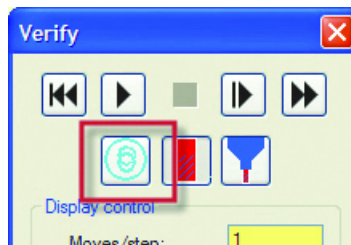
You can see that FBM Mill created 14 toolpath groups - one for each feature and zone.



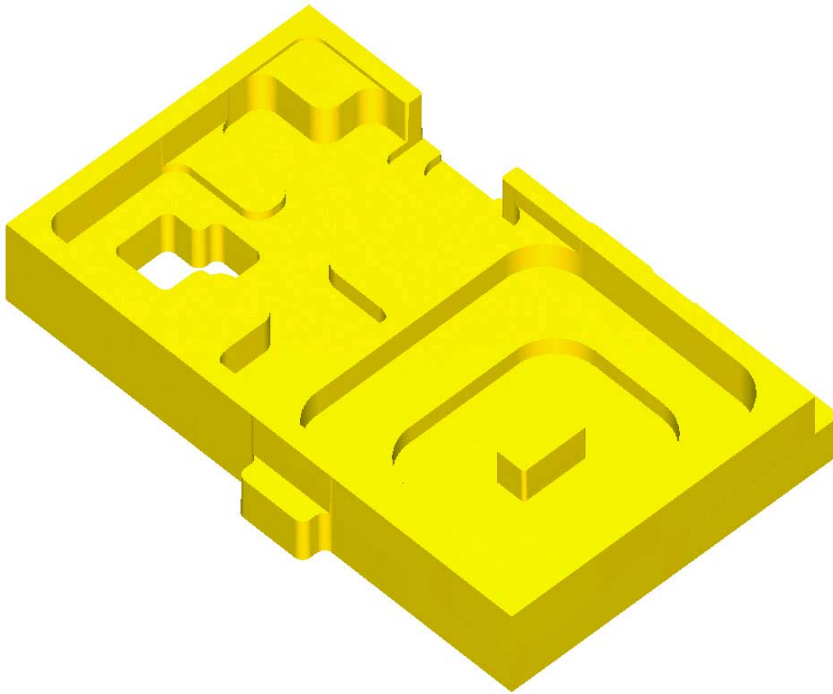
- 2 Select all toolpaths and choose **Verify** to confirm that all material has been removed.



- 3 Choose **Turbo** at the top of the Verify dialog box to run the solid model verification as quickly as possible.



- 4 Click **Machine** to run the verification. All material is removed and the completed part displays in the graphics window.



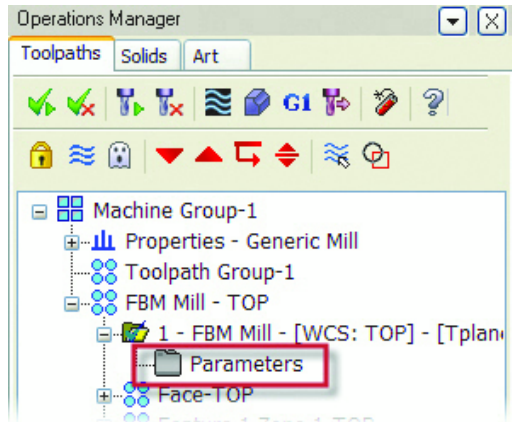
- 5 Choose **OK** to exit Verify.



Changing Your FBM Mill Results

You may need to make changes to your FBM Mill toolpath results based on changes in your part or machining environment. For example, you may need to disable the outside profile pass on the part due to a change in fixturing.

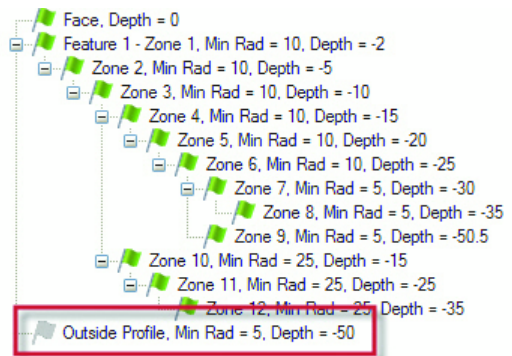
- 1 Click the **Parameters** icon for the FBM Mill toolpath in the Toolpath Manager.



- 2 On the **Setup** page, deselect the **Rough outside of part** and **Finish outside of part** check boxes.

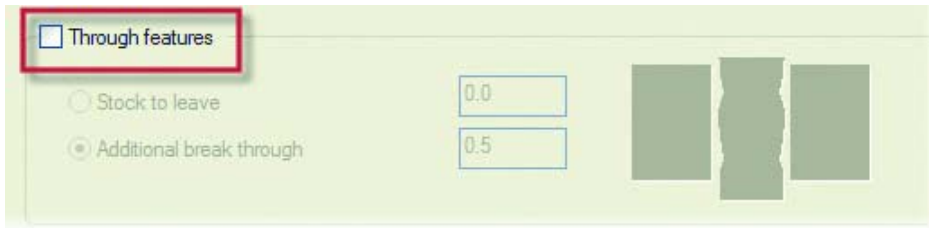


- 3 Select the **Features** page and notice that the Outside Profile feature has been suppressed.
- The feature is recognized by FBM Mill, but will not be machined.

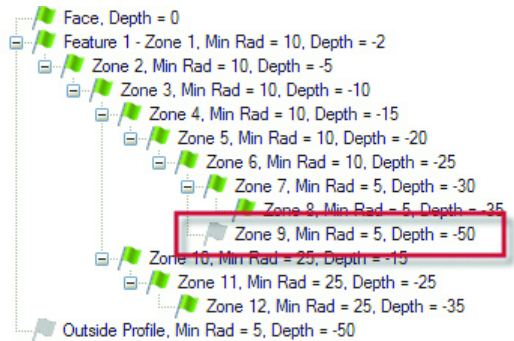


- 4 On the **Feature detection** page, deselect the **Through features** check box.

The cut outs could be machined at a later time on a Wire EDM machine.



- 5 Go back to the **Features** page and notice that Zone 9 is now also suppressed.



- 6 Select **Zone 10** and press [**Delete**].

This removes Zone 10 and both zones below it (Zones 11 and 12) from the features list. You may need to remove zones or features if you want to generate toolpaths for only certain areas of your part.

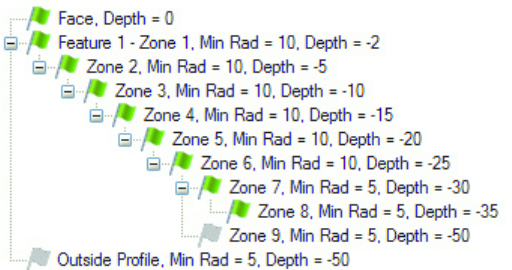
- 7 Click **OK** to close the FBM Mill dialog box.

- 8 Regenerate only the FBM Mill toolpath.

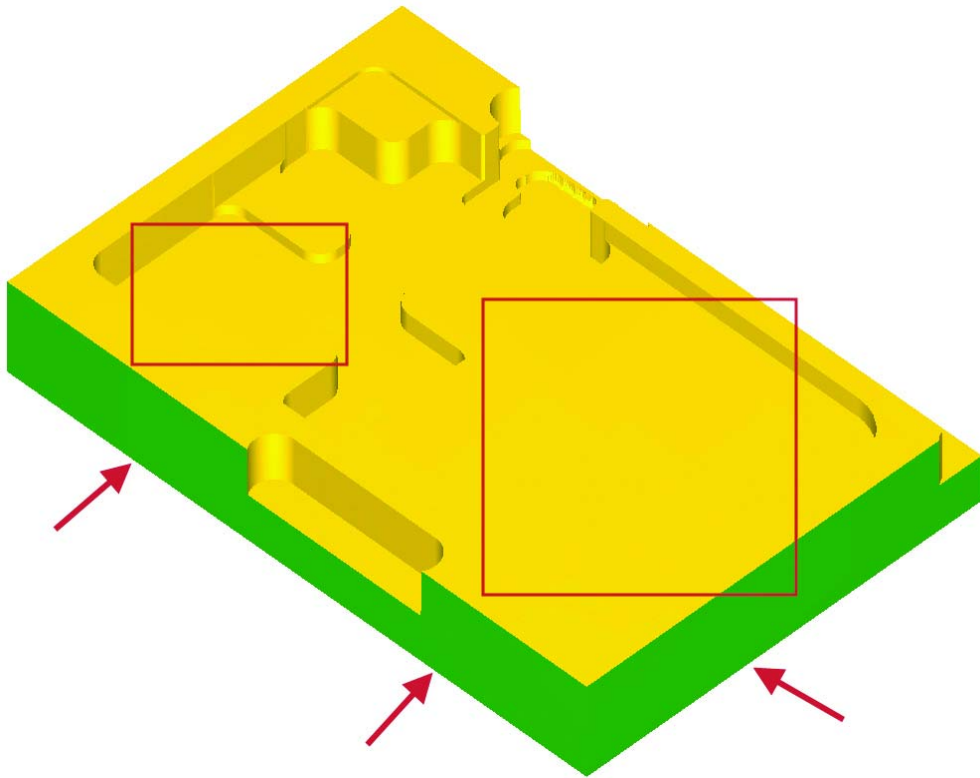
FBM Mill generates toolpaths to machine only the Face and Zones 1 through 8.

- 9 Select all toolpaths and click **Verify** in the Toolpath Manager to see the updated results.

- 10 Click **Machine** to run the solid model verification. The updated part displays in the graphics window. Notice the areas that



were not machined due to the modified parameters.



- 11 Click **OK** to exit Verify and return to the Toolpath Manager.

FBM Drill

FBM Drill is a robust feature based machining strategy. Within a single FBM Drill operation, you perform the following basic functions:

- detect holes in a solid based on specified criteria.
- review the detected features list and edit or delete features.
- preview toolpath operations and make additional changes before they are generated.
- automatically generate a complete series of drill operations for the selected features.

Hole detection can include blind, through, co-axial, and split holes. FBM Drill also reads hole data from solids created with the SolidWorks® Hole Wizard®. You can choose to group operations by tool or by plane. Use other FBM Drill options to define deep drilling, spot drilling, and pre-drilling operation parameters. You control the tool selection by using tools already in the MCX file, choosing a tool library, and allowing Mastercam to create tools, when necessary.

FBM Drill Requirements

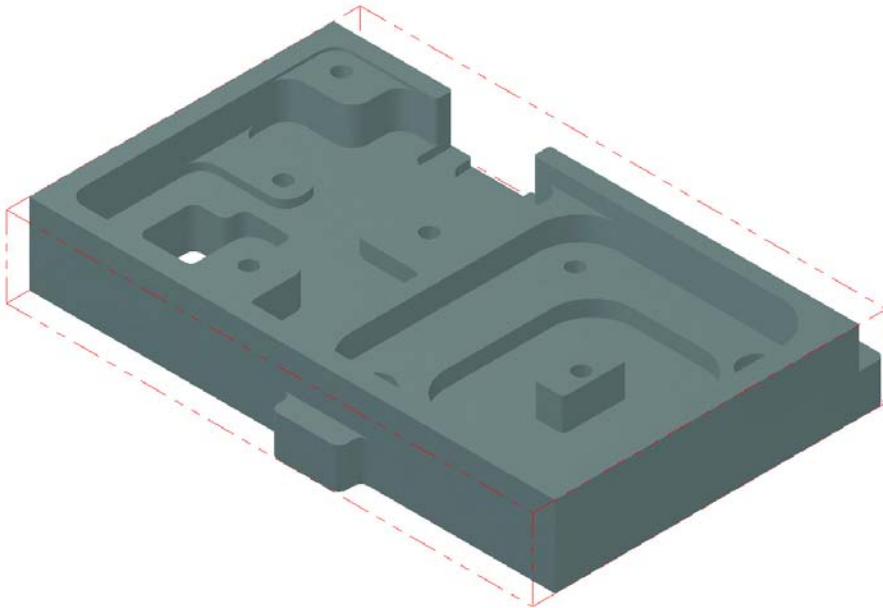
- Mastercam Solids, and Mastercam Mill Level 1 or Mastercam Router and higher
- Mill or Router machine definition (select from the **Machine Type** menu)
- At least one solid part model in the current .MCX file with one or more holes



TIP: If you customize FBM-generated toolpaths, do so only when you are sure that you will not regenerate the FBM parent operation. Any customization is overwritten when the FBM-generated toolpaths are recreated.

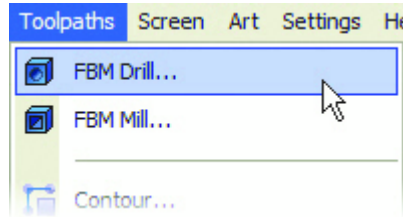
► Creating an FBM Drill toolpath

- 1 From the Mastercam menu, choose **File, Open**.
- 2 Go to `..\Documentation\ExampleParts` and select `FBM_DRILL_X3.MCX`.

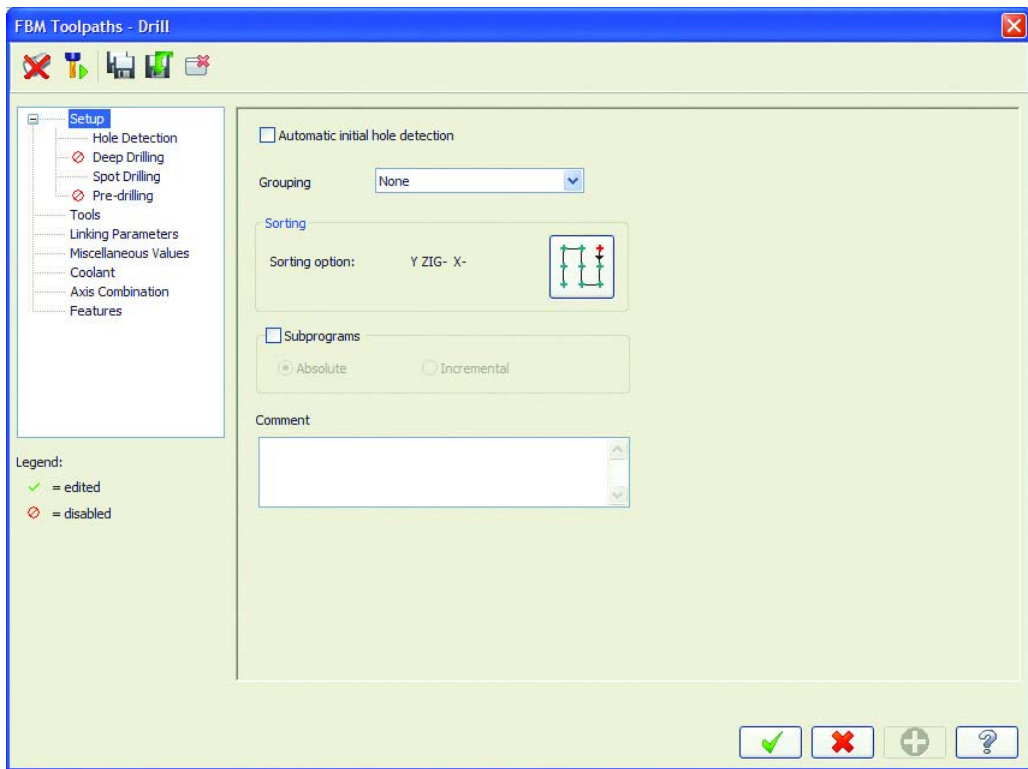


Note: This file contains FBM Mill toolpaths that remove the necessary material prior to drilling holes.

- 3 Choose **Toolpaths**, **FBM Drill** from the Mastercam menu.



Mastercam displays the FBM Drill dialog box.



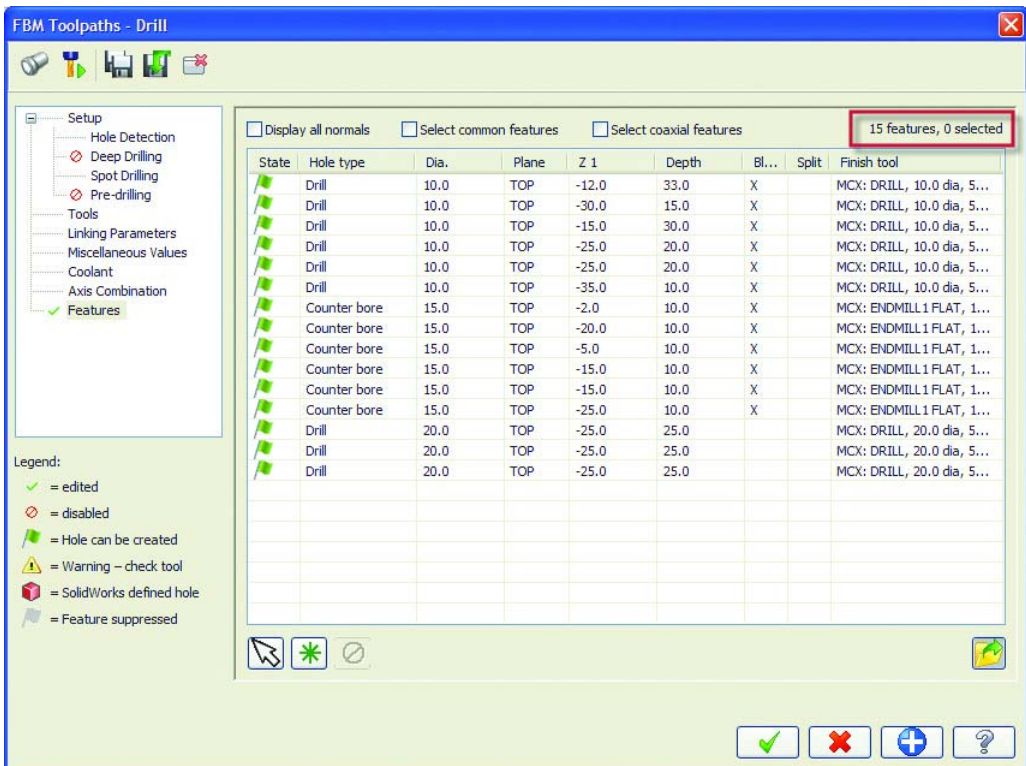
- 4 Click the **Detect** button at the top of the FBM Mill dialog box. This scans the solid model and finds all drill holes that match the default operation parameters.





TIP: The red X over the Detect button indicates that the FBM operation does not include the most recent set of features on the part. Once the part is re-analyzed, the red X is removed.

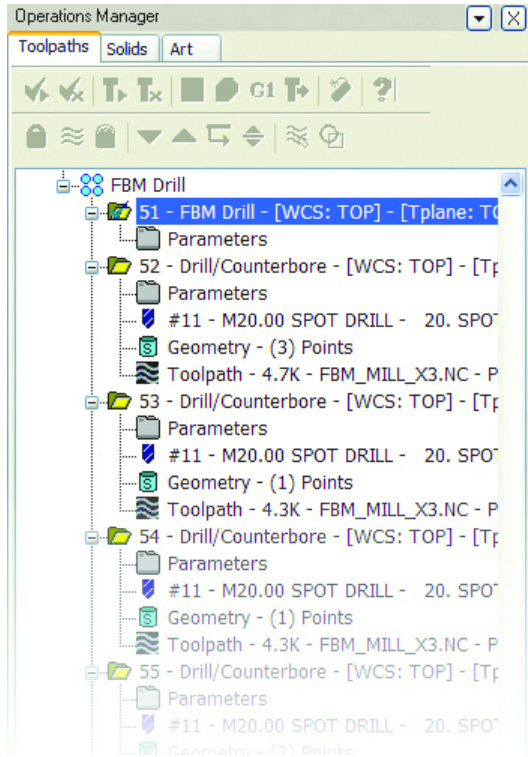
The results are listed on the Features page. FBM Drill detected 15 features at various depths. The green flag icon in front of each hole indicates that the current FBM Drill settings can create all the operations required to completely machine each feature.



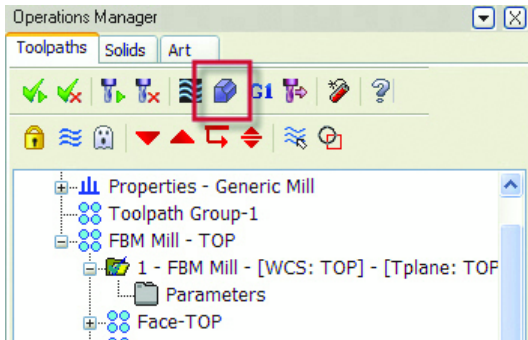
5 Click **OK** to generate the toolpaths required to machine the holes.



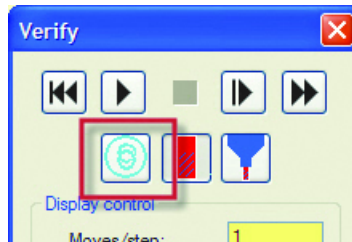
FBM Drill generates the necessary drill toolpaths in the Toolpath Manager.



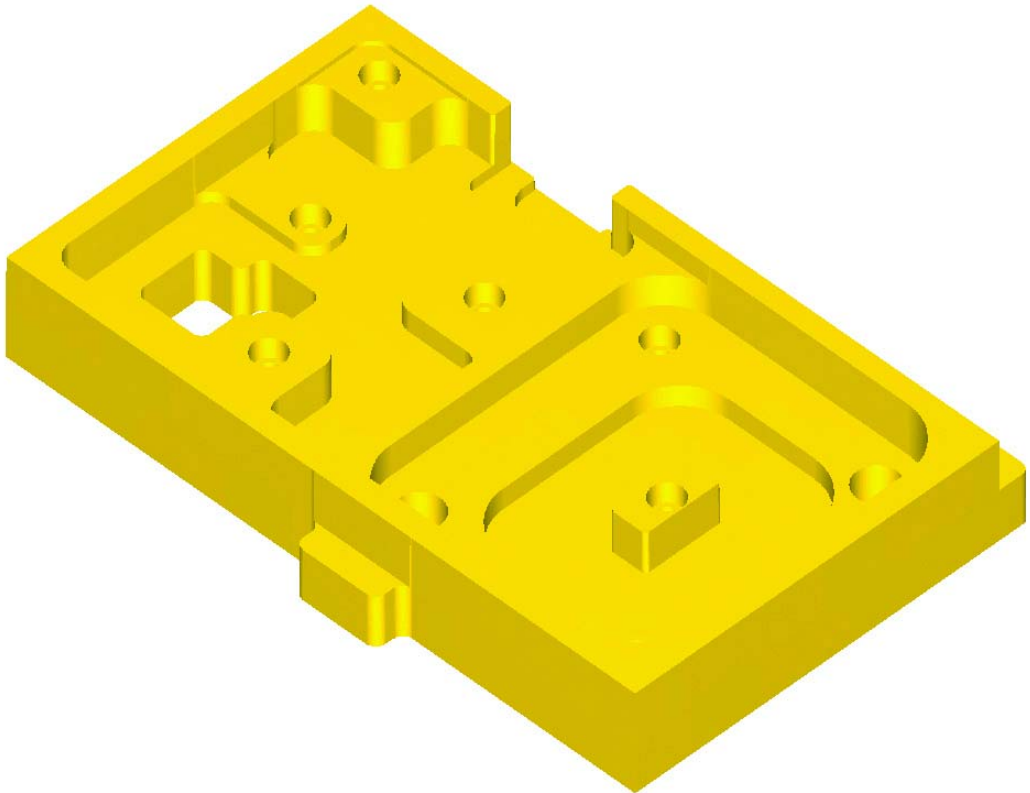
- 6 Select all the toolpaths in the Toolpath Manager (including the Mill toolpaths) and click **Verify** to confirm that all the holes were successfully drilled.



- 7 Choose **Turbo** at the top of the Verify dialog box to run the solid model verification as quickly as possible.



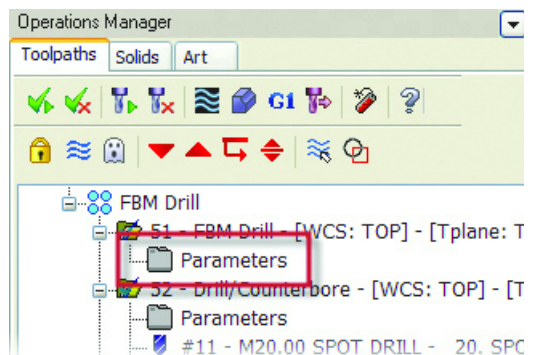
- 8 Click **Machine** to run the verification. The completed part displays in the graphics window.



- 9 Click **OK** to exit Verify.

Changing Your FBM Drill Results

- 1 Click the **Parameters** icon for the FBM Drill operation in the Toolpath Manager.



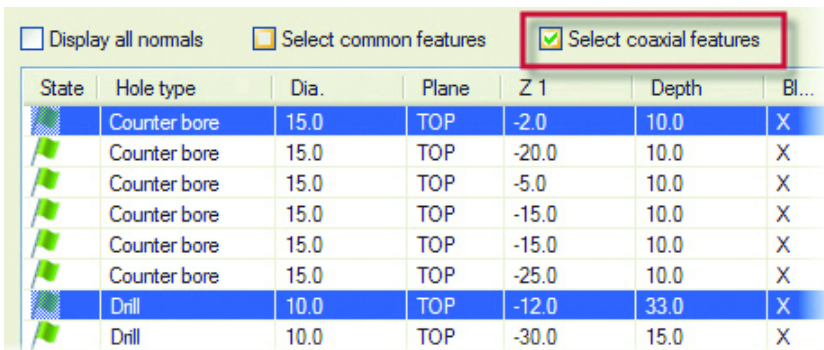
- 2 On the **Features** page, click the **Hole type** column heading to sort the features list alphabetically by hole type.



You can sort the holes by clicking on any of the column headings in the features list.

- 3 Click the **Select coaxial features** check box and select the first counterbore in the list.

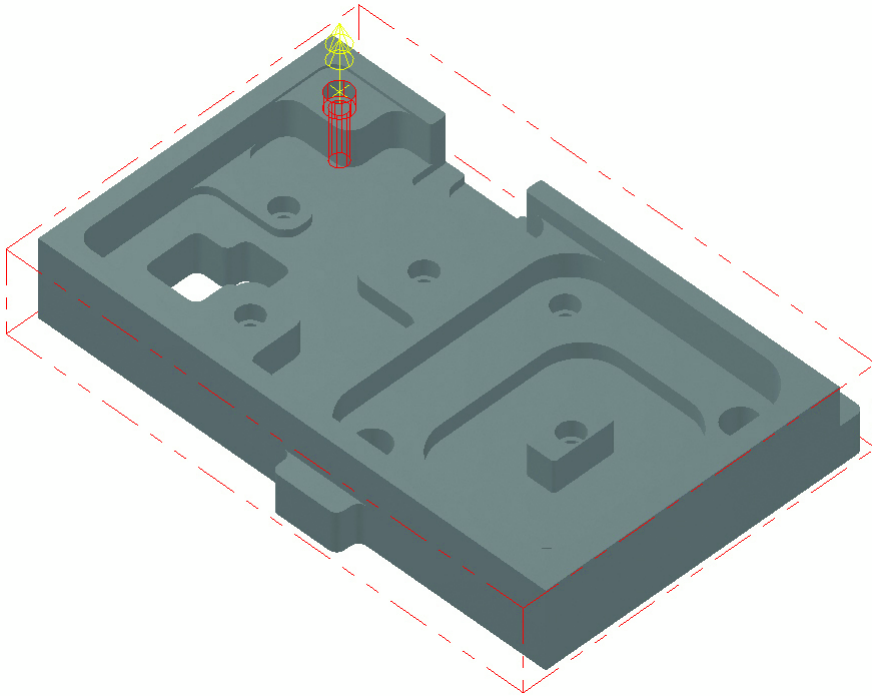
This check box automatically selects all co-axial hole features in the list, which are multiple holes that share a common axis. When you click on the counterbore, the Drill hole that lies below the Counterbore hole is also selected.



- 4 Click the **Hide dialog** button at the top of the FBM Drill dialog box to minimize the dialog box and confirm which holes are selected.



The selected holes and their common axis are highlighted on the part.



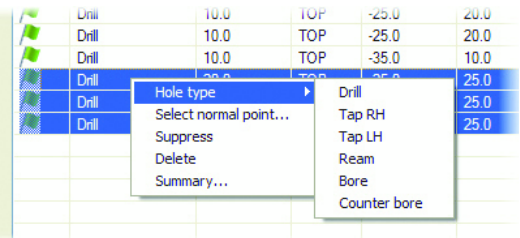
- 5 Press **[Esc]** to redisplay the dialog box.
- 6 Deselect the **Select coaxial features** check box.
- 7 Click the **Select common features** check box at the top of the features list.

8 Click on one of the **Drill** holes with a 20.0 mm diameter. The two similar holes are highlighted.

Display all normals Select common features Select coaxial features 15 features, 3 selected

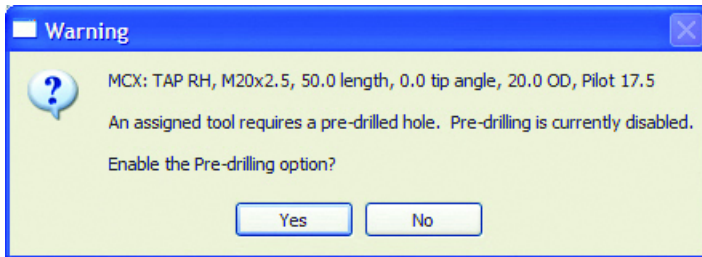
State	Hole type	Dia.	Plane	Z 1	Depth	Bl...	Split	Finish tool
	Counter bore	15.0	TOP	-2.0	10.0	X		MCX: ENDMILL1 FLAT, ...
	Counter bore	15.0	TOP	-20.0	10.0	X		MCX: ENDMILL1 FLAT, ...
	Counter bore	15.0	TOP	-5.0	10.0	X		MCX: ENDMILL1 FLAT, ...
	Counter bore	15.0	TOP	-15.0	10.0	X		MCX: ENDMILL1 FLAT, ...
	Counter bore	15.0	TOP	-15.0	10.0	X		MCX: ENDMILL1 FLAT, ...
	Counter bore	15.0	TOP	-25.0	10.0	X		MCX: ENDMILL1 FLAT, ...
	Drill	10.0	TOP	-12.0	33.0	X		MCX: DRILL, 10.0 dia, 50...
	Drill	10.0	TOP	-30.0	15.0	X		MCX: DRILL, 10.0 dia, 50...
	Drill	10.0	TOP	-15.0	30.0	X		MCX: DRILL, 10.0 dia, 50...
	Drill	10.0	TOP	-25.0	20.0	X		MCX: DRILL, 10.0 dia, 50...
	Drill	10.0	TOP	-25.0	20.0	X		MCX: DRILL, 10.0 dia, 50...
	Drill	10.0	TOP	-35.0	10.0	X		MCX: DRILL, 10.0 dia, 50...
	Drill	20.0	TOP	-25.0	25.0			MCX: DRILL, 20.0 dia, 50...
	Drill	20.0	TOP	-25.0	25.0			MCX: DRILL, 20.0 dia, 50...
	Drill	20.0	TOP	-25.0	25.0			MCX: DRILL, 20.0 dia, 50...

9 Right-click in the **Hole type** column for one of the highlighted drill holes and choose **Hole type**. Mastercam shows a list of alternate drill cycles you can apply to the holes.



10 Select **Tap RH** from the list.

Mastercam warns you that this hole type change requires a pilot hole so the tap can fit into the hole. To address this, FBM Drill can automatically turn on the Pre-drilling option for you.



11 Click **Yes** to enable pre-drilling for the selected holes.

The three holes are converted to Tap RH holes.

Drill	10.0	TOP	-25.0	20.0	X	MCX: DRILL, 10.0 dia, 50...
Drill	10.0	TOP	-35.0	10.0	X	MCX: DRILL, 10.0 dia, 50...
Tap RH	20.0	TOP	-25.0	25.0		LIB: TAP RH, M20x2.5, 5...
Tap RH	20.0	TOP	-25.0	25.0		MCX: TAP RH, M20x2.5, ...
Tap RH	20.0	TOP	-25.0	25.0		MCX: TAP RH, M20x2.5, ...

- Click the **Preview toolpath operations** button at the top of the FBM Drill dialog box to see a list of the drill toolpaths that FBM Drill will generate.



- Click **OK** to close the Toolpath Operations Preview dialog box.
- Click **OK** to close the FBM Drill dialog box.

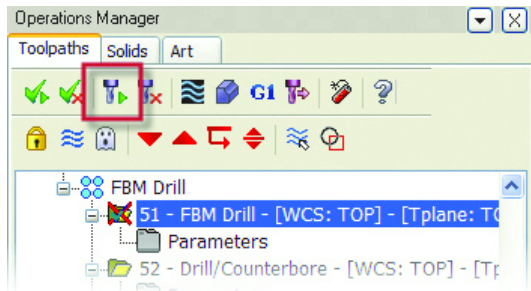
The FBM Drill operation is marked as dirty in the Toolpath Manager.

Verifying Your FBM Drill Toolpaths

- Click **Regenerate all selected operations** in the Toolpath Manager.

FBM Drill generates new drill toolpaths based on the updated parameters.

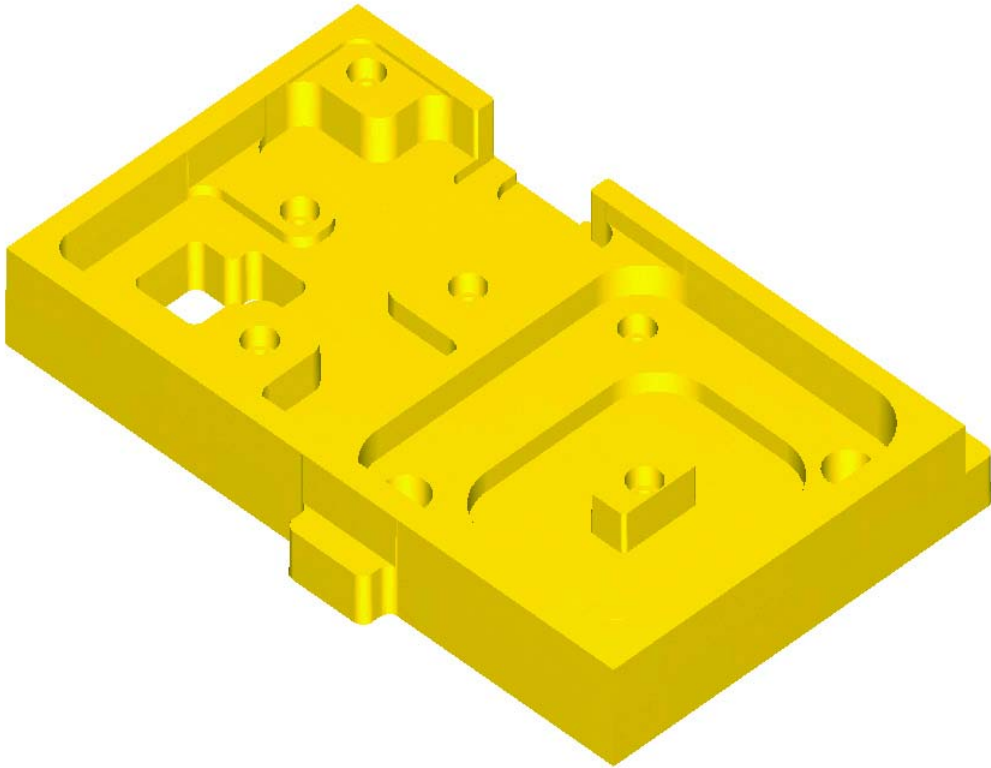
- Select all the toolpaths in the Toolpath Manager (including the Mill toolpaths) and choose **Verify** to confirm that all the holes were successfully drilled.



- Click **Machine** to run the verification.

The completed part should look like the following picture.





4 Click **OK** to exit Verify.

Using SolidWorks Hole Wizard Data

FBM Drill also reads hole data from solids created with the SolidWorks Hole Wizard[®]. If you are working with an imported SolidWorks part, and you have SolidWorks installed on your local PC, you can open the model's .SLDPRT file and update the features list with imported SolidWorks Hole Wizard data.



IMPORTANT: SolidWorks must be installed on the same system as Mastercam for the Hole Wizard data to be imported.

► **Importing SolidWorks Hole Wizard data**

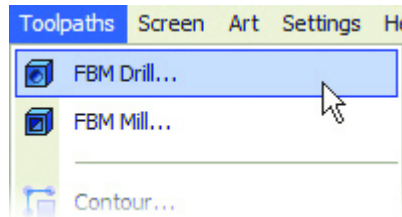
- 1 From the Mastercam menu, choose **File, Open.**

- 2 Go to ..\Documentation\ExampleParts and select HOLE_WIZARD.MCX.

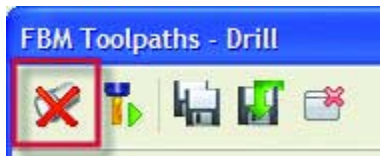


- 3 Choose **Toolpaths, FBM Drill** from the Mastercam menu.

Mastercam displays the FBM Drill dialog box.



- 4 Click the **Detect** button at the top of the FBM Mill dialog box. This scans the solid model and finds all drill holes that match the default operation parameters.



TIP: The red X over the Detect button indicates that the FBM operation does not include the most recent set of features on the part. Once the part is re-analyzed, the red X is removed.

The results are listed on the Features page. FBM Drill detected 12 features at various depths. The green flag icon in front of each hole indicates that the current FBM Drill settings can create all the operations required to completely machine each feature.

Display all normals Select common features Select coaxial features 12 features, 0 selected

State	Hole type	Dia.	Plane	Z 1	Depth	Bl...	Split	Finish tool
	Drill	0.4219	TOP	-0.04405	1.95595			MCX: DRILL, 27/64, 2.0 l...
	Drill	0.4219	TOP	-0.04405	1.95595			MCX: DRILL, 27/64, 2.0 l...
	Drill	0.4687	TOP	-0.17675634	1.82324366			MCX: DRILL, 29/64, 2.0 l...
	Drill	0.4687	TOP	-0.17675634	1.82324366			MCX: DRILL, 29/64, 2.0 l...
	Drill	0.4687	TOP	-0.17675634	1.82324366			MCX: DRILL, 29/64, 2.0 l...
	Drill	0.4687	TOP	-0.17675634	1.82324366			MCX: DRILL, 29/64, 2.0 l...
	Chamfer (100.0)	0.89	TOP	0.04615048	0.41954982			MCX: CSINK, #1, 10.0 le...
	Chamfer (100.0)	0.89	TOP	0.04615048	0.41954982			MCX: CSINK, #1, 10.0 le...
	Chamfer (100.0)	0.89	TOP	0.04615048	0.41954982			MCX: CSINK, #1, 10.0 le...
	Chamfer (100.0)	0.89	TOP	0.04615048	0.41954982			MCX: CSINK, #1, 10.0 le...
	Chamfer (90.0)	0.51	TOP	0.245	0.5			MCX: CSINK, #1, 2.0 len...
	Chamfer (90.0)	0.51	TOP	0.245	0.5			MCX: CSINK, #1, 2.0 len...

5 Click the **Select SolidWorks file** button in the lower right corner below the features list.

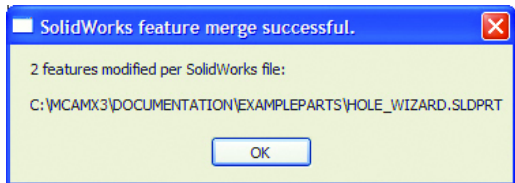


6 In the Open dialog box, go to
 ..\Documentation\ExampleParts.

7 Select **.SLDPRT** in the **Files of Type** drop-down list.

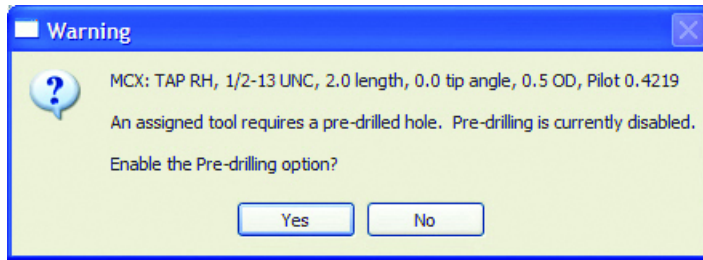
8 Select **HOLE_WIZARD.SLDPRT** and click **OK**.

Mastercam connects to SolidWorks on your computer and reads the Hole Wizard data from the selected part. Mastercam then displays a message that the SolidWorks information was successfully merged into the features list and two features were affected.



9 Click **OK** to continue.

Mastercam warns you that the new hole type for the edited features requires a pilot hole. To address this, FBM Drill can automatically turn on the Pre-drilling option for you and generate these drill cycles.



- 10 Click **Yes** to enable pre-drilling for the selected holes.

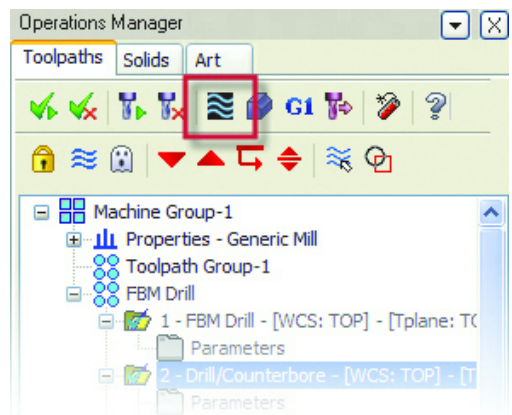
The red cube icon in the features list indicates that the information for these two holes was imported from SolidWorks.

State	Hole type	Dia.	Plane	Z 1	Depth	Bl...	Split	Finish tool
	Tap RH	0.5	TOP	-0.005	1.995			MCX: TAP RH, 1/2-13 U...
	Tap RH	0.5	TOP	-0.005	1.995			MCX: TAP RH, 1/2-13 U...
	Drill	0.4687	TOP	-0.17675634	1.82324366			MCX: DRILL, 29/64, 2.01...
	Drill	0.4687	TOP	-0.17675634	1.82324366			MCX: DRILL, 29/64, 2.01...

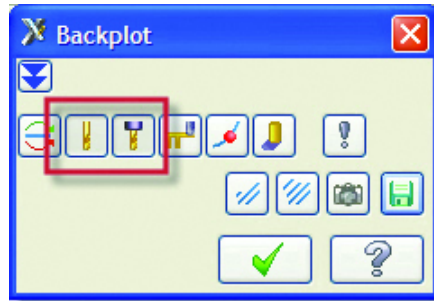
- 11 Click **OK** to generate toolpaths to machine the holes.

Verifying the SolidWorks Hole Wizard Results

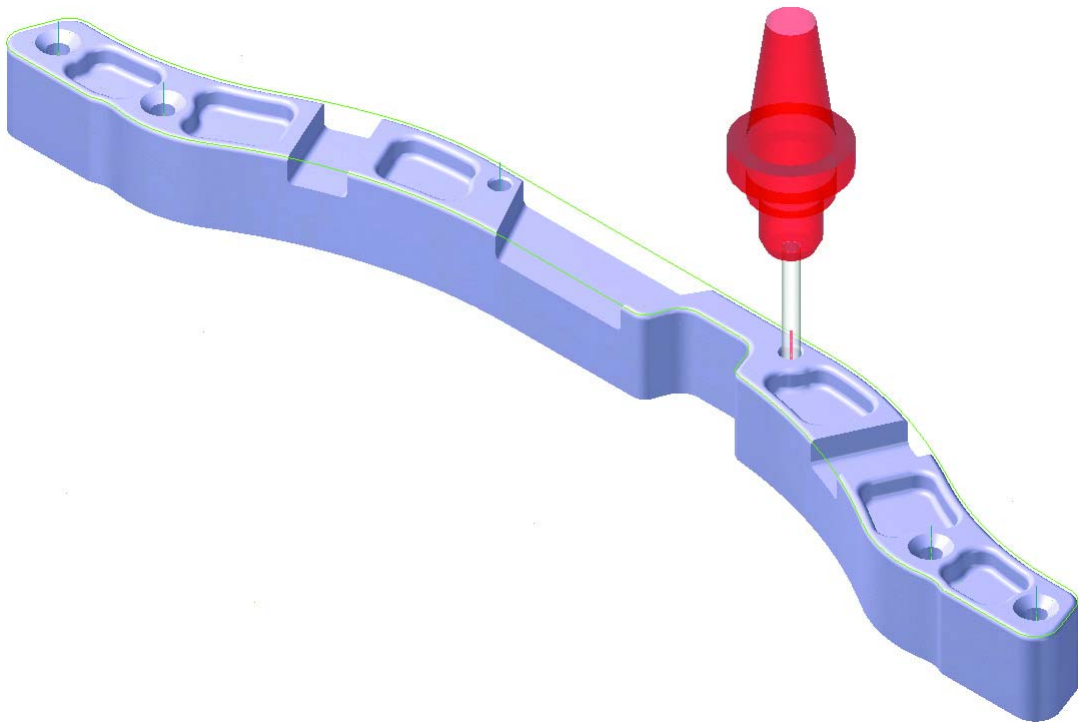
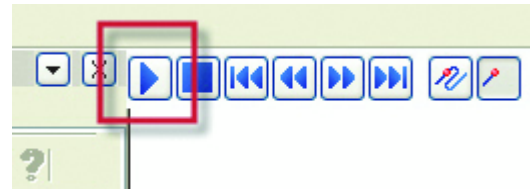
- 1 Select all the toolpaths in the Toolpath Manager and click the **Backplot** button.
- 2 Right-click in the graphics window and choose **Isometric Gview** to make it easier to see the tool motion.



- 3 Select the **Display tool** and **Display holder** buttons so you can see the tools and holders during the backplot.



- 4 Press the **Play** button on the Backplot VCR bar above the graphics window to run the backplot.



- 5 Click **OK** to exit Backplot.

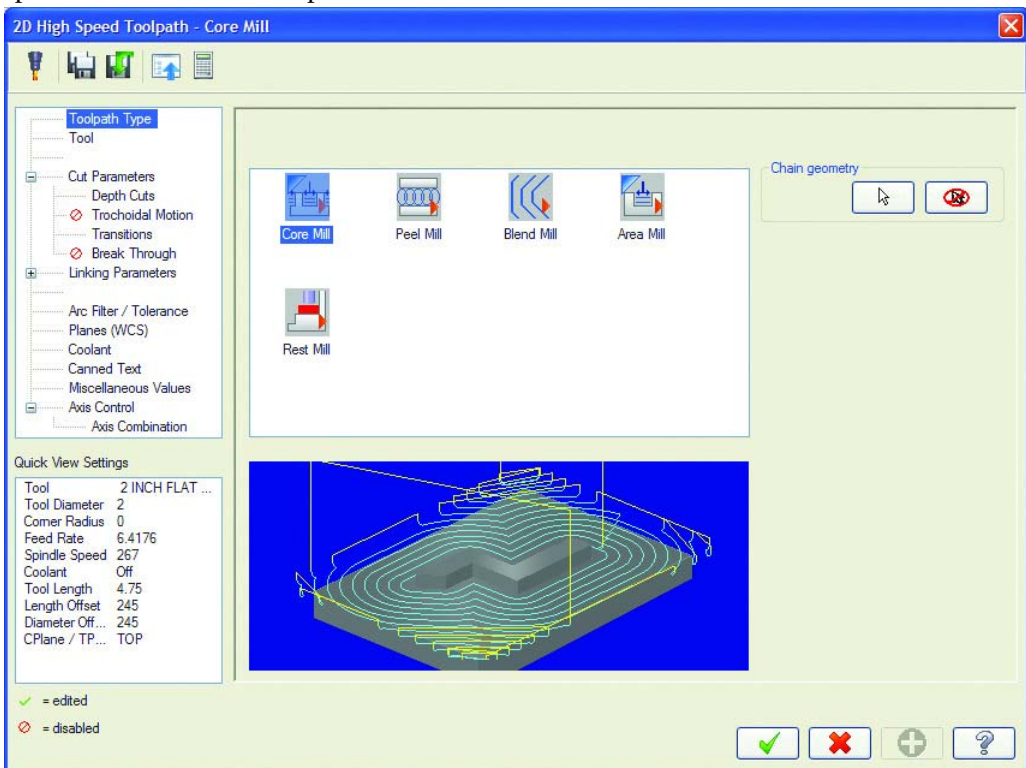
Mill Level 1 Enhancements

Mill Level 1 in Mastercam X3 features new toolpaths and machining strategies to make machining your 2D parts faster and more accurate.

Note: Router customers can also take advantage of these Mastercam X3 enhancements.

2D High Speed Toolpaths

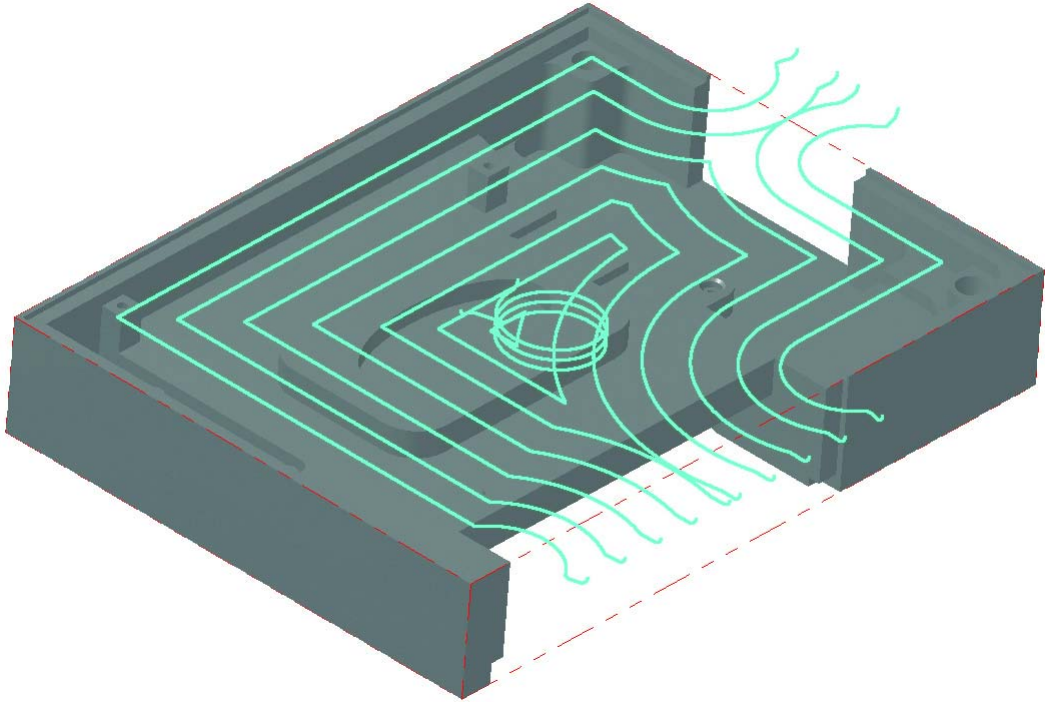
Mastercam X3 marks the introduction of a suite of new 2D toolpaths called 2D High Speed. The five toolpaths — Peel Mill, Core Mill, Area Mill, Rest Mill, and Blend Mill — are optimized for high speed machining and hard milling. They can make your current processes more efficient and automated, and minimize programming time and cycle times. These toolpaths utilize a new interface that will be expanded to include all toolpaths in the future.



Core Mill

The Core Mill toolpath requires less CAD knowledge and model preparation from the programmer. Core Mill automatically machines standing bosses and cores in one easy step, generating smooth, free-flowing motion.

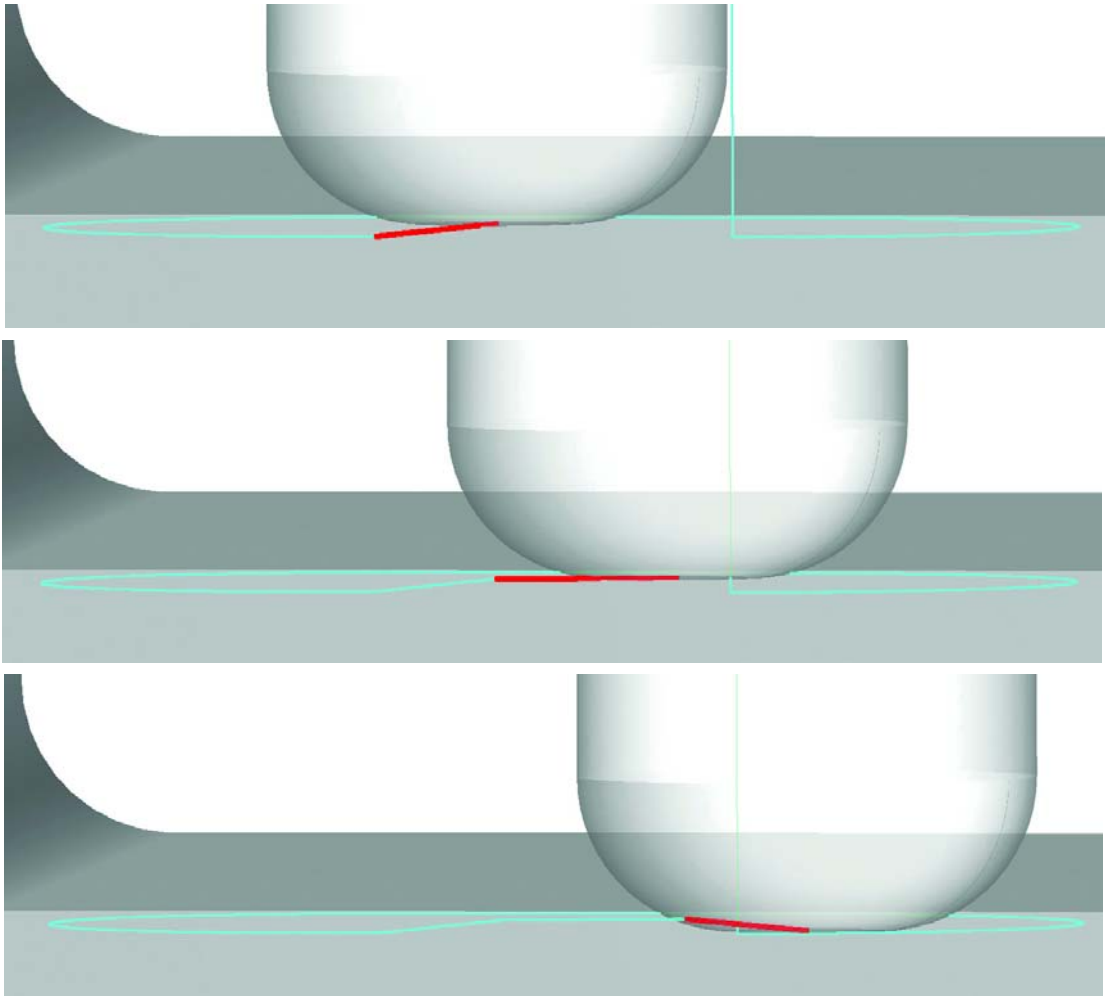
This toolpath type requires two boundaries. The outside chain represents the stock boundary, allowing the toolpath to move freely outside this area, and the inner chain defines the limit of the toolpath. The toolpath starts from the outside, and each pass slowly takes on the final shape of the part as it approaches the final pass. Climb or conventional motion is maintained, and the transitions between cuts utilize high speed-friendly methods.



Peel Mill

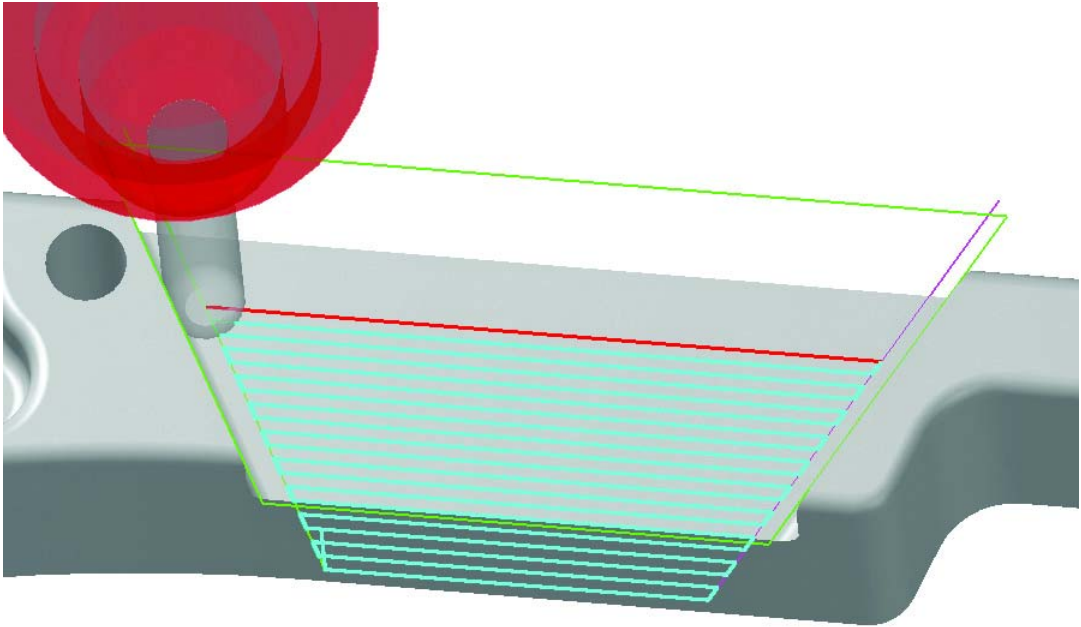
The Peel Mill toolpath was introduced in Mastercam X2 MR2 and has been enhanced in Mastercam X3. Peel Mill efficiently machines open slots and channels using the full depth of the cutting tool.

Mastercam X3 introduces microlifts for Peel Mill. Microlifts raise the cutting tool a very small distance to avoid dragging the tool across the floor of the part on the back passes, which could affect the floor finish and generate unwanted heat. The following pictures show the progression of the microlift moves in a side view.



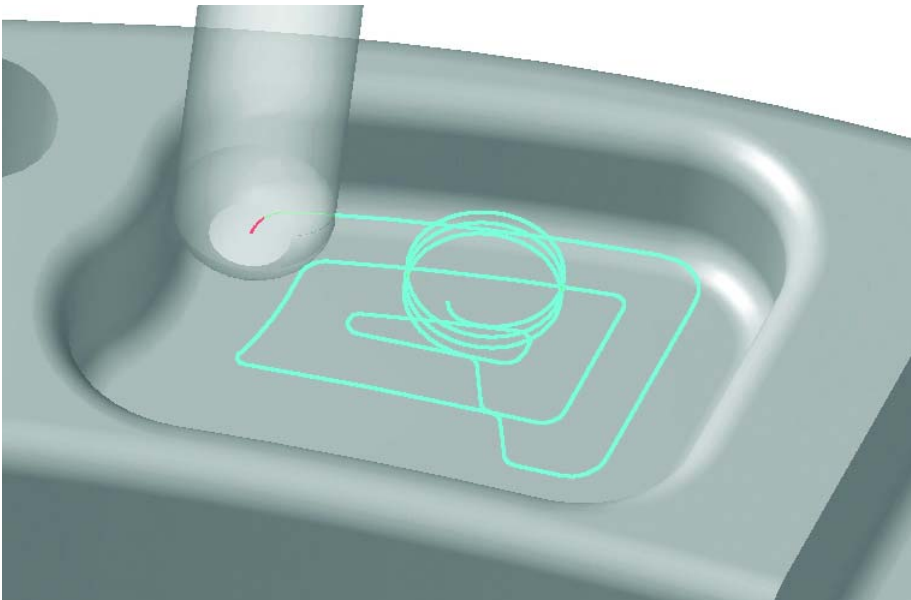
Blend Mill

Mastercam X3's Blend Mill toolpath morphs between two curves, generating motion previously unavailable in 2D toolpaths. Blend Mill toolpaths generate fewer retracts and toolpath operations on these difficult-to-machine features. Blend Mill also uses the tool's full depth and more of the flute length, resulting in less cycle time and even tool wear, as shown in the following picture.



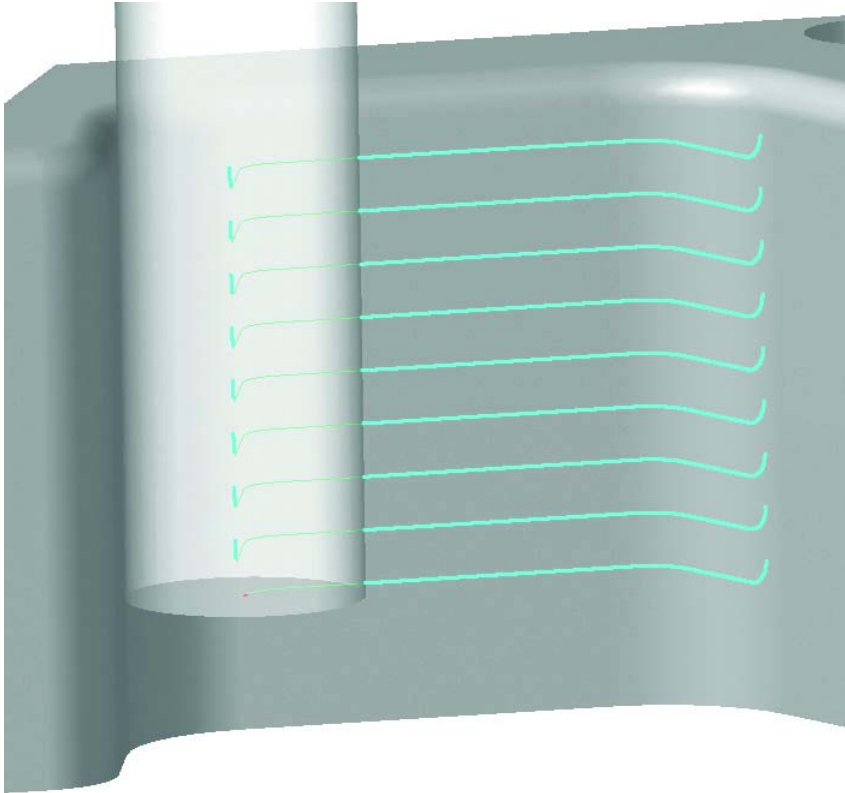
Area Mill

Area Mill produces smooth, clean motion when machining pockets. You can control the smoothness settings to create the most efficient path for your tools to take, avoiding sharp corners and direction changes. Smooth helical entries and tangent stepovers also create efficient motion for machine tools.



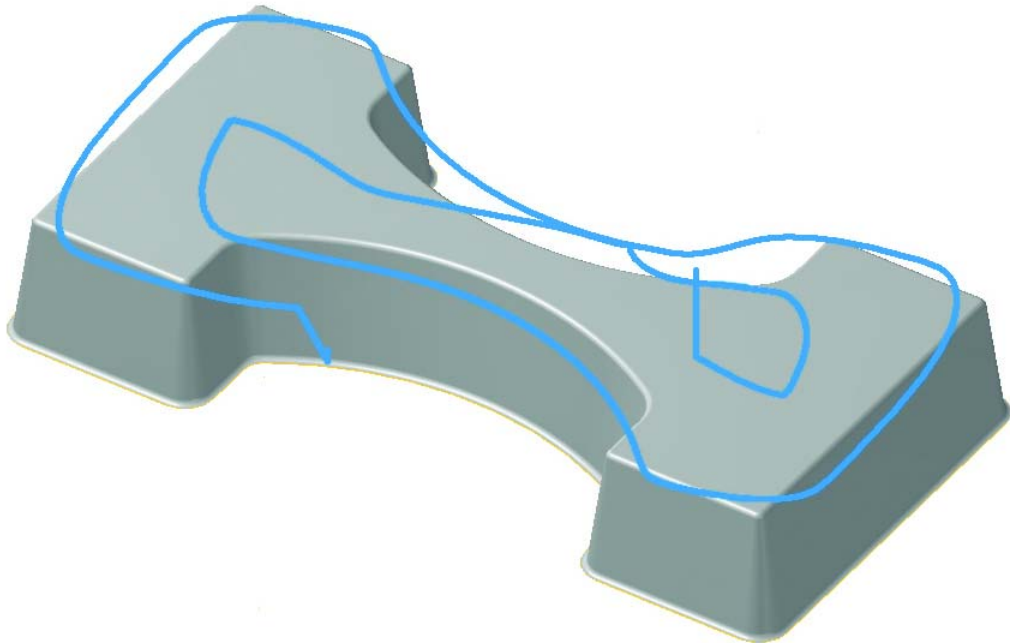
Rest Mill

Rest Mill targets material that previous toolpaths left behind. Rest Mill ensures that the tool safely approaches leftover material by using either helical entry or vertical arcs. Rest Mill motion can also be smoothed, resulting in longer tool life and less wear on machine tools.



New Cutting Method for Facing Toolpaths

The Face toolpath has been modified to include an additional cutting method called *controlled engagement*. When you select the **Controlled engagement** cutting method on the Facing parameters tab, several options in the dialog are disabled, and only those parameters appropriate for controlled engagement are available. This style attempts to maintain a constant amount of tool engagement within a limit for the maximum engagement amount.



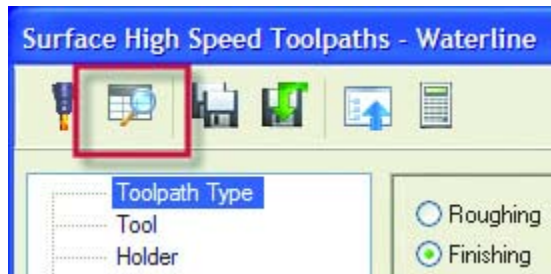
Mill Level 3 Enhancements

Mastercam X3 offers several enhancements for Mill Level 3 users including speed improvements for multiaxis toolpaths and additional controls for Surface High Speed toolpaths (HST). These new features save you time and effort by making your programming time as efficient as possible.

Note: Router Pro customers can also take advantage of these Mastercam X3 enhancements.

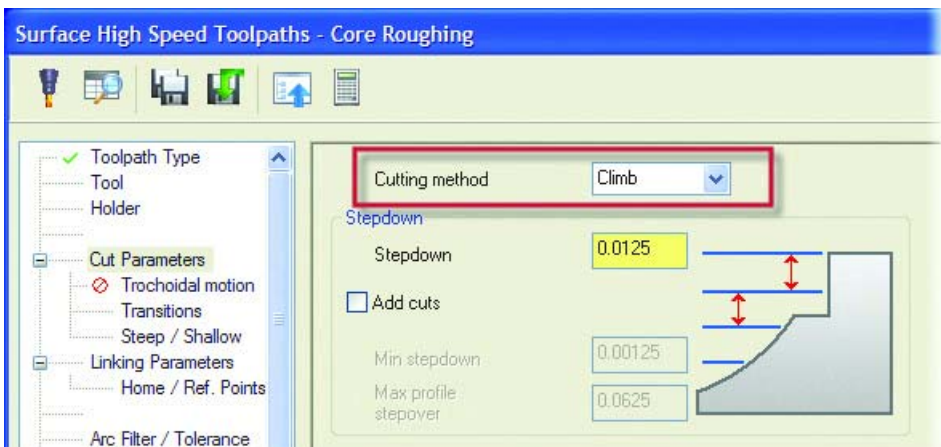
Formula Files

The Surface High Speed toolpaths (HST) can now be controlled by utilizing a new .FORMULA file located in the . . \Mill\Formula directory. This XML-based file contains formulas used to derive the values in the HST toolpaths. This gives you the ability to set intelligent values for different cutting conditions (for example, material hardness). A new button located on the HST toolpaths interface lets you select one of the supplied .FORMULA files.



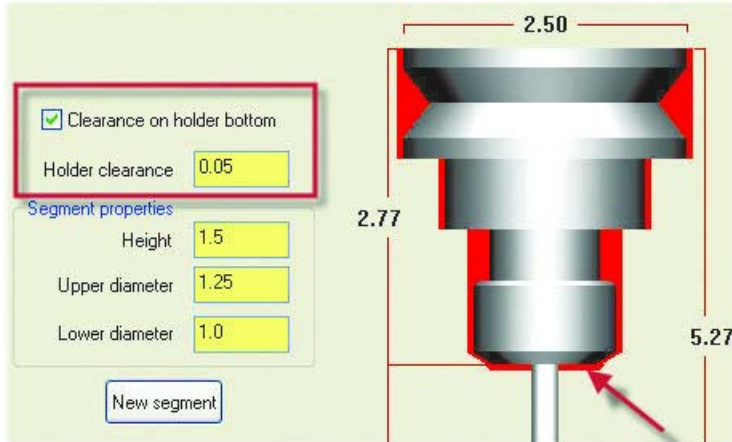
New Climb/Conventional Options for HST Toolpaths

Options for controlling climb or conventional cutting have been added to all constant-Z HST toolpaths, which include Core Roughing, Area Clearance, Waterline, Rest Roughing, and Horizontal Area.



HST Tool Holders

An option for clearance below the holder has been added. This allows the toolpath to avoid collisions between the holder and the stock in the XY direction and now the Z direction as well. You have the option to check for clearance below the holder or to just check against the XY direction.



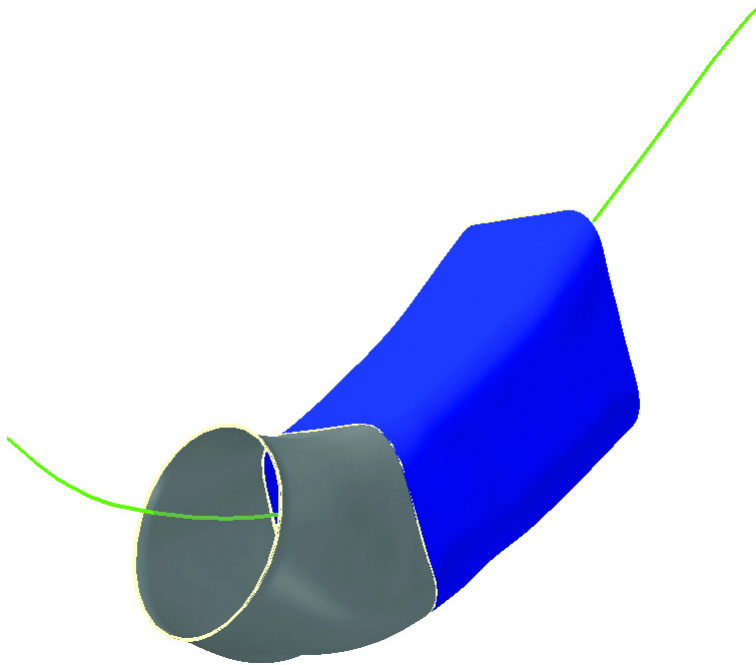
Multiaxis Edge Blending

The Port5ax and Flow5ax toolpaths include a new feature that allows an overlap/blend to be added at the beginning and end of a toolpath. This feature is very useful in creating better finishes where toolpaths may need to meet from opposite directions. A new tab page called **Edges** has been added that contains several options for controlling the start and end of a port or flowline multiaxis toolpath. Options include skipping the first pass, last pass, or both, and blending the entry, exit, or both.

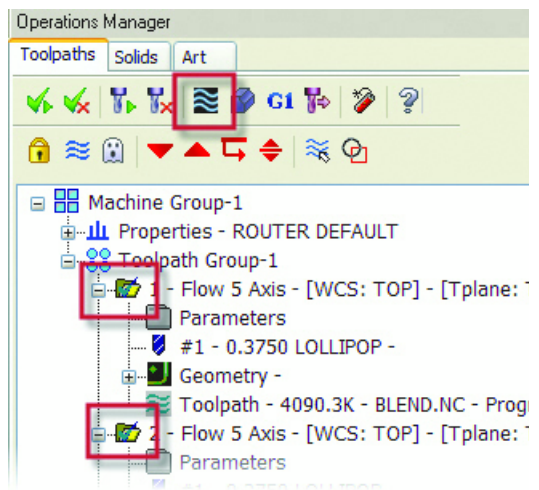
► Adding edge blending to a multiaxis toolpath

- 1 From the Mastercam menu, choose **File, Open.**

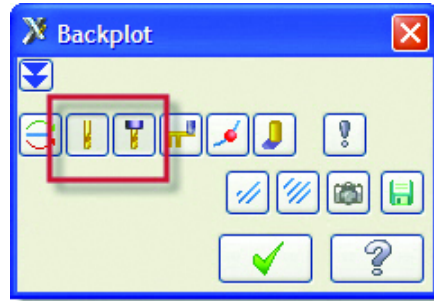
- 2 Go to ..\Documentation\ExampleParts and select BLEND_START.MCX



- 3 If shading is on, press **[Alt+S]** to turn off shading. This makes it easier to see the flowline 5-axis toolpaths.
- 4 Select both of the toolpaths in the Toolpath Manager and click the **Backplot** button.



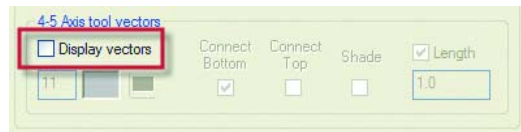
5 Select the **Display tool** and **Display holder** buttons so you can see the tool and holder during the backplot.



6 Click the **Options** button and deselect the **Simulate Rotary Axis** option in the lower left corner so the part stays in place during the backplot.



7 Deselect the **Display vectors** option in the 4-5 Axis tool vectors section to simplify the backplot display.

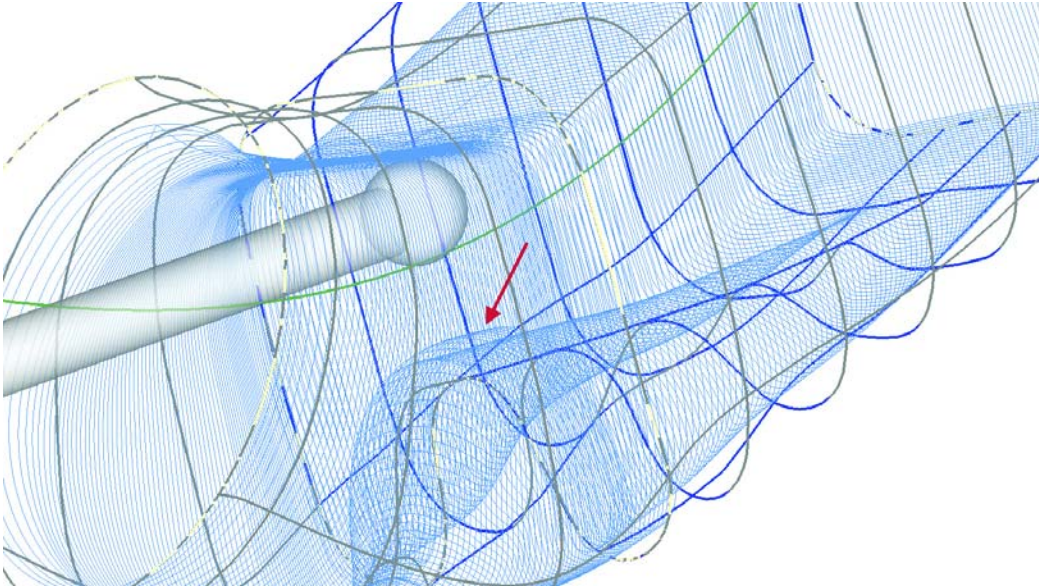


8 Click **OK** to close the Backplot Options dialog box.

9 Press the **Play** button on the Backplot VCR bar above the graphics window to run the backplot.



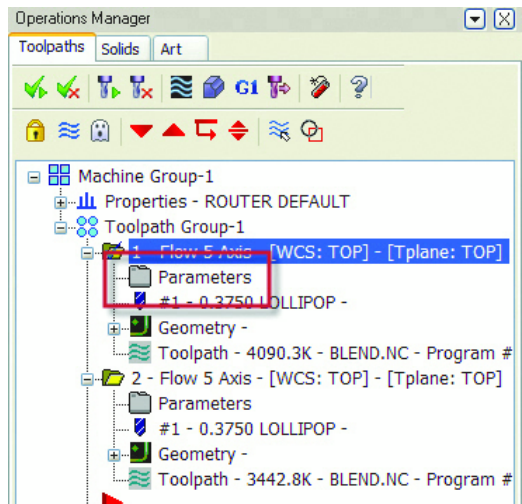
10 When the backplot is done, press **[F1]** and zoom in on the area where the two toolpaths overlap. This section where the surface edge boundaries meet could have blending problems.



- 11 Click **OK** to complete the backplot.

Adding an Exit Blend

- 1 Click the **Parameters** icon for the first Flow 5-Axis toolpath in the Toolpath Manager.



2 Select the **Edges** tab and set the following options:

- ◆ **Add Exit Blend**
- ◆ **Taper Distance = 0.1**
- ◆ **Taper Angle = 20**

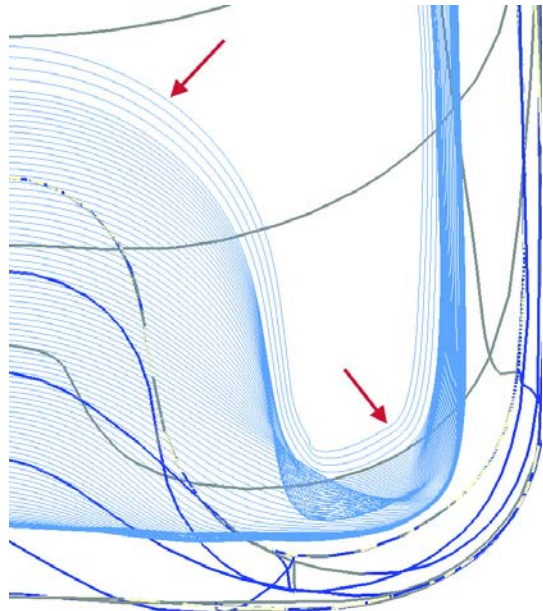
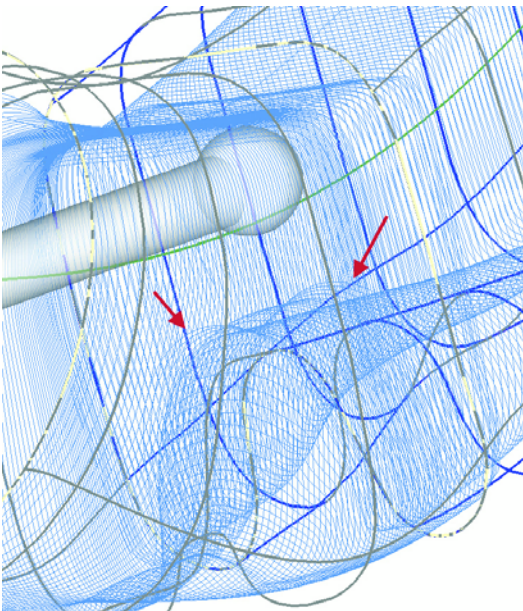
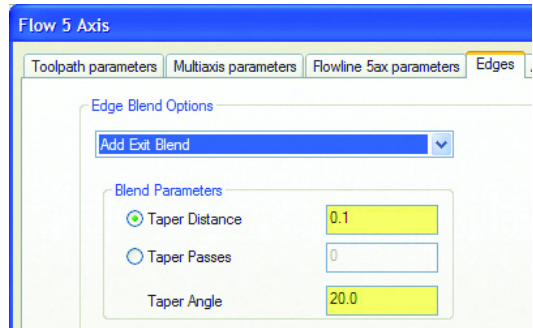
This will create a 20-degree exit blend defined at 0.1 inches past the final toolpath pass.

3 Click **OK** to complete the toolpath changes.

4 Make the same changes to the second toolpath. Using the same blend settings creates a smooth transition on the surface edge where the two toolpaths meet.

5 Regenerate both toolpaths and backplot again.

The generated motion creates a tapered cone effect that moves the tool away from the surface edge as it tapers 0.1 inches past it.

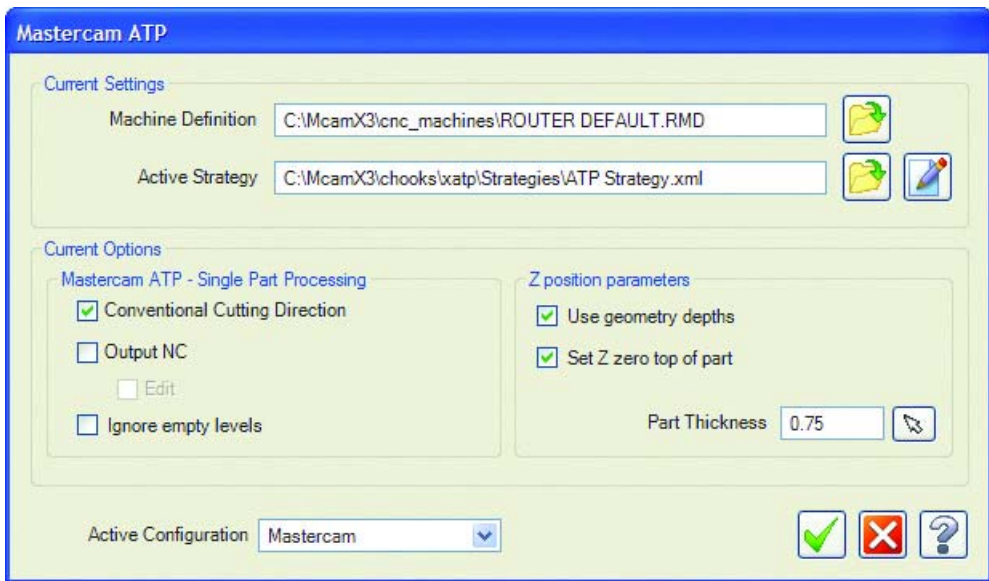


Router Enhancements

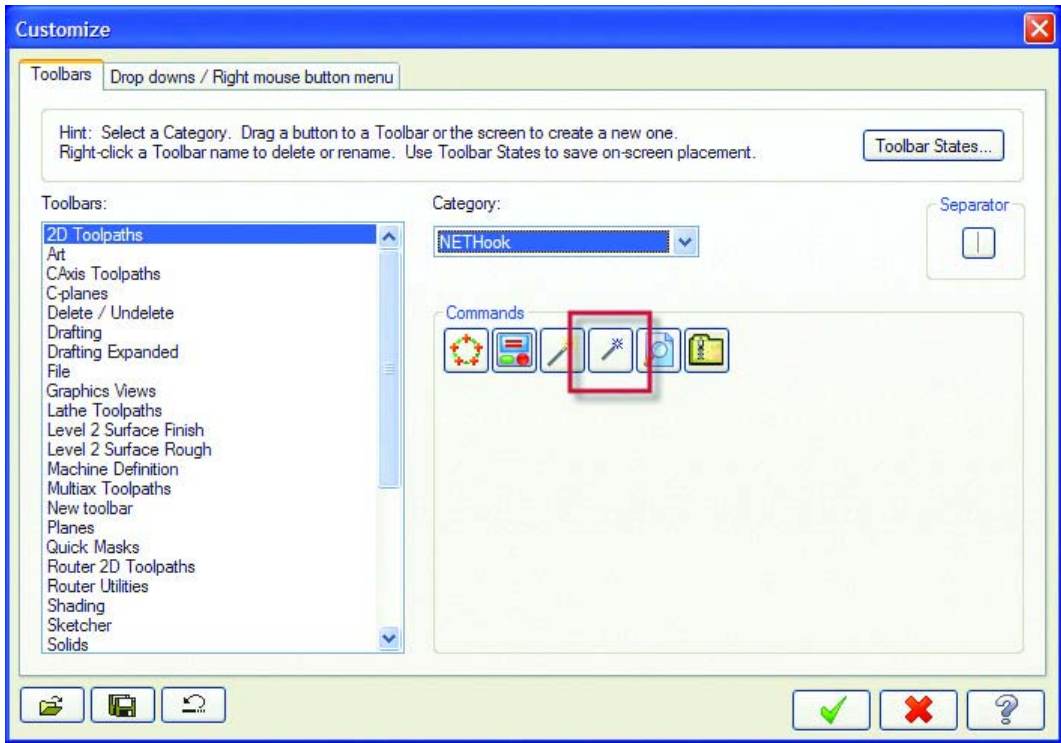
Mastercam X3 Router provides new functions that give you additional control over your router machining jobs.

Single File ATP Processing

You can now create or open a layered (multi-level) geometry file, select a machine definition, and select an Automatic Toolpathing (ATP) strategy to automatically chain the geometry and create toolpaths using the new Single File ATP option. The main dialog box provides easy access to commonly used parameters such as climb or conventional cutting motion, part thickness, and more.



You can access this new function by adding the **X ATP Single Part Processing** button to your toolbar from the Settings, Customize dialog box.



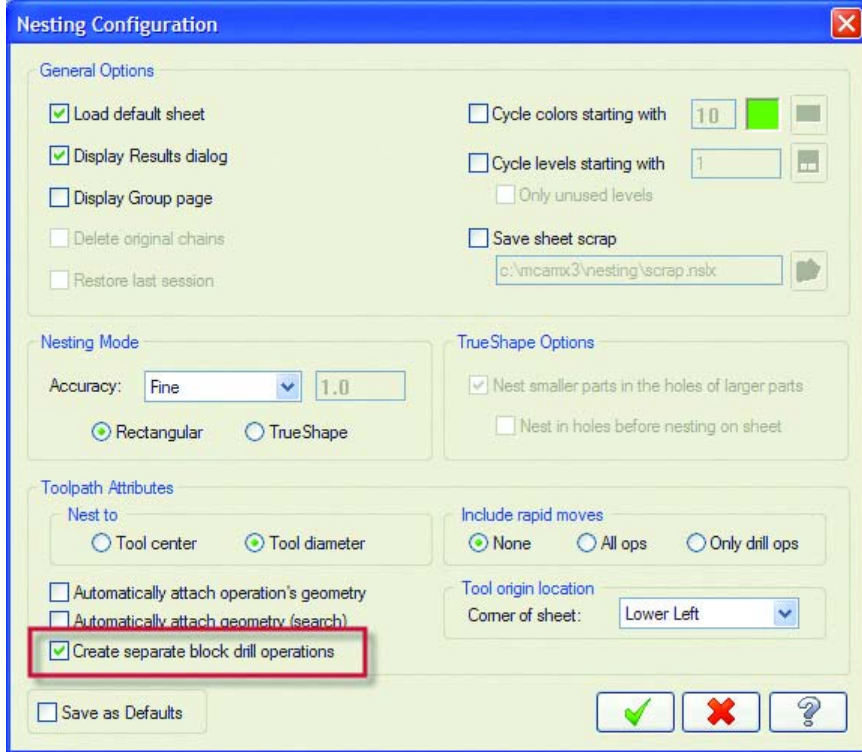
Nesting Block Drill Operations within Machine Limits

In Mastercam X3, you can nest block drill toolpaths within a restricted area on a sheet to ensure that they are placed within the machine travel limits. If you specify machine axis limits for the X and Y axes, and the proper offset for the drill block component in your machine definition, Mastercam positions block-drilled parts in a confined area of the sheet, ensuring that all drilling is done within the travel limits of the machine.

Creating Separate Block Drill Operations

Generating separate block drill operations for each sheet of material can provide more efficient sorting by optimizing the block drilling motion for all parts as a group rather than treating them individually. In addition, this option offers improved material usage by allowing block-drilled parts to be rotated. The option to create separate block drill operations is available in the Nesting

Configuration dialog box. This dialog box displays during manual nesting or in the Automatic Toolpathing (ATP) function.

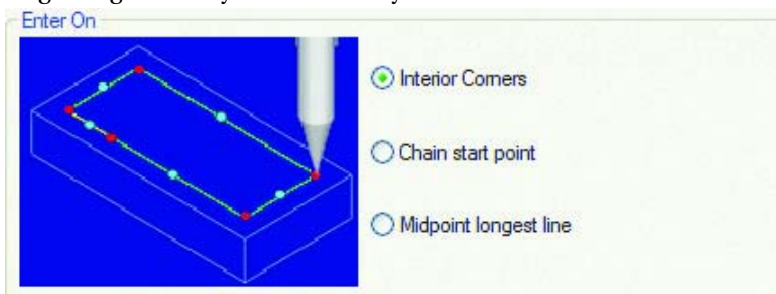


Engraving Toolpath Multiple Entry Point Options

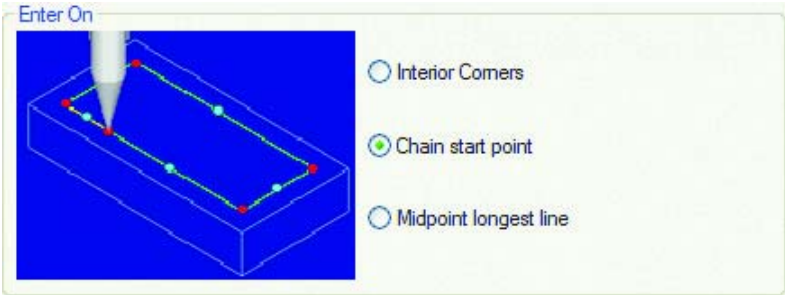
You can now direct where the tool enters the stock on an Engraving toolpath. Previously, the Engraving toolpath directed the tool to enter the operation at the first interior corner of the chained geometry.

On the third page of the Engraving parameters, there are three new radio buttons in the **Enter On** group box:

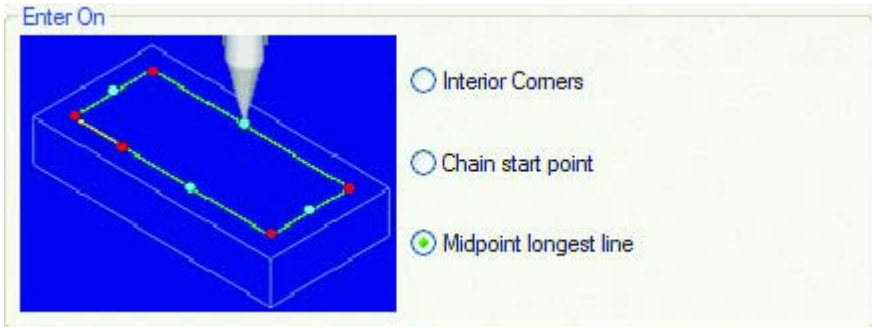
- **Interior corners** - This is the default option. It enters the part on an interior corner. Engraving currently uses this entry method.



- **Chain start point** - The toolpath enters on the beginning of the selected chain geometry. If you want to start somewhere else, you have to break the line and chain from the break. The example shows breaking the long line on the bottom into two pieces.



- **Midpoint longest line** - The toolpath begins at the midpoint of the longest line in the selected chain. Note that the bottom line in the example is broken so the longest line is the upper line.

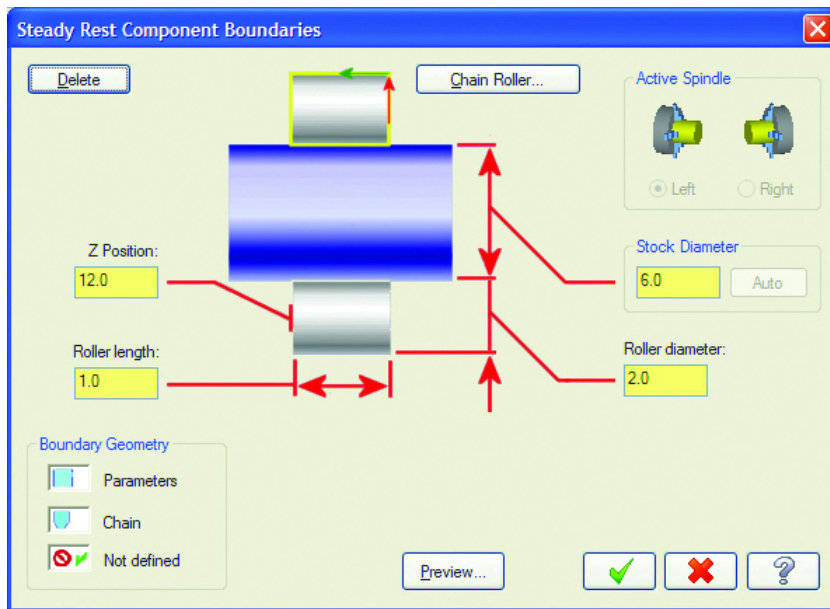


Lathe Enhancements

Mastercam X3 Lathe provides the latest tool and machine information to make your lathe jobs more efficient and accurate.

Steady Rest C-Hook

This new Lathe add-on defines the boundaries for an existing steady rest machine component for Lathe machine definitions. You can create the steady rest component boundaries by either chaining existing geometry for customized steady rest designs, or by entering parameters to define the size and position for generic roller-type steady rest components.



To access this function, choose **Settings, Run User Application** from the Mastercam menu and open the `Steadyrest.dll` file.

Notes:

- Stock must already be defined in the Stock Setup tab of the Machine Group Properties dialog box for at least one of the spindles.
 - The active lathe machine definition must be configured with a steady rest component. If there is no steady rest component already defined and you try to use this C-Hook, Mastercam displays a warning.
-

Tool Catalog and Library Changes

Mastercam X3 includes updated Sandvik® and Kennametal® tool catalogs. A new VTL right turret tool library was also added with some basic tooling correctly oriented.

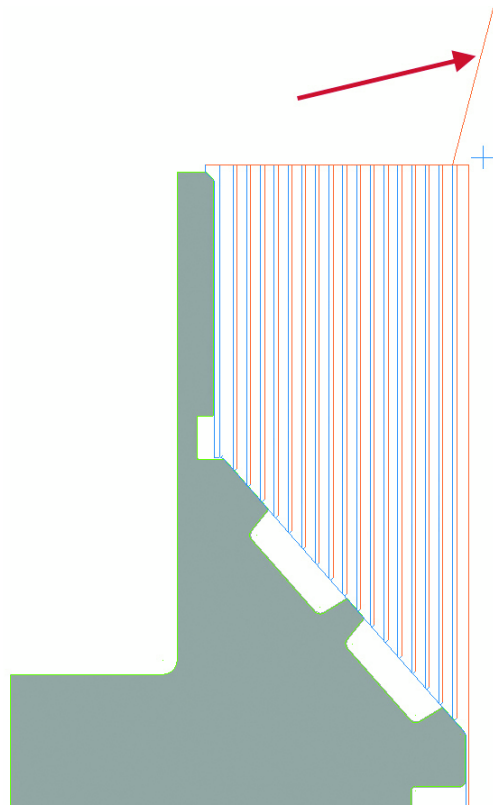
VTL Machine Definition Changes

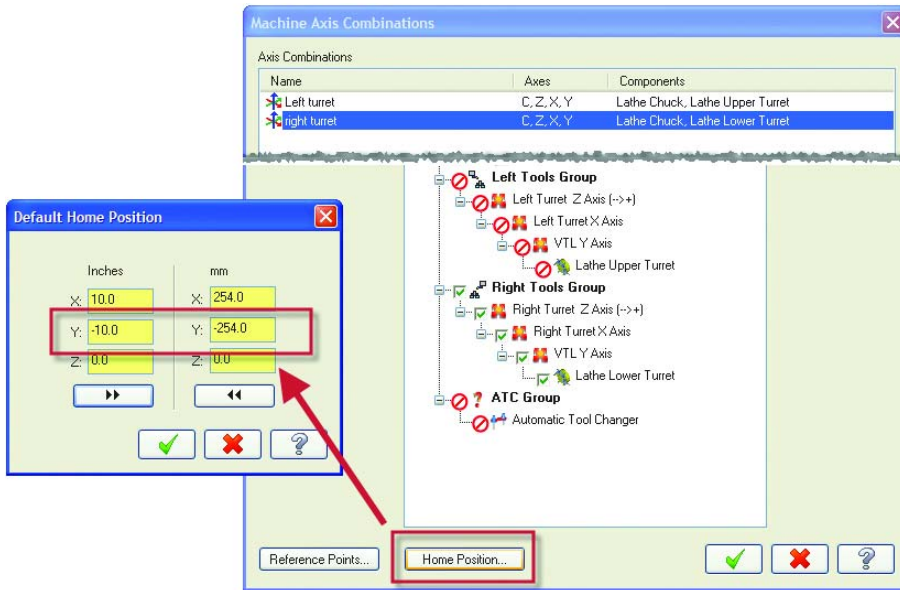
The VTL machine and component definition files used in previous versions were removed in Mastercam X3 and new files were substituted:

Table 2: Mastercam X3 VTL Machine and Component Definition Changes

Removed files	Added files
LATHE 2 AXIS VTL.LMD	VTL RIGHT TURRET.LMD
LATHE 2 AXIS VTL MM.LMD	VTL RIGHT TURRET MM.LMD
LATHE VTL MULTI AXIS.LMD	VTL MULTI TURRET.LMD
LATHE VTL MULTI AXIS MM.LMD	VTL MULTI TURRET MM.LMD
LATHE VTL COMPONENTS.GMD	VTL COMPONENTS.GMD
LATHE VTL COMPONENTS MM.GMD	VTL COMPONENTS MM.GMD

Each of the new VTL machine definitions includes several improvements. Right turrets with a Y axis were changed to a negative home position to show the correct toolpath approach and retract.

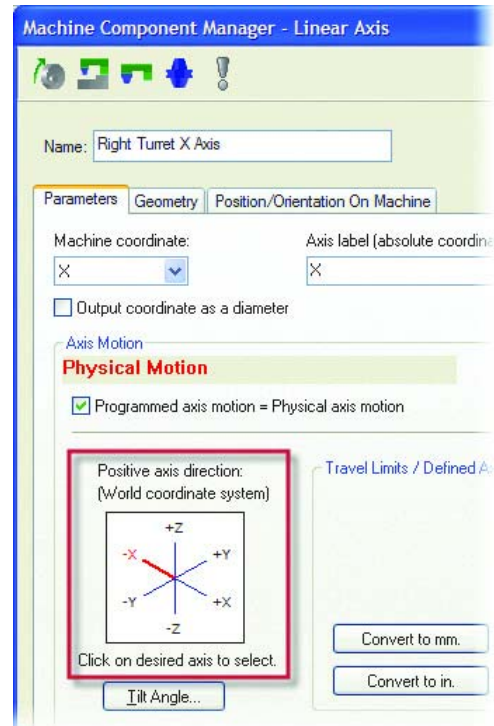
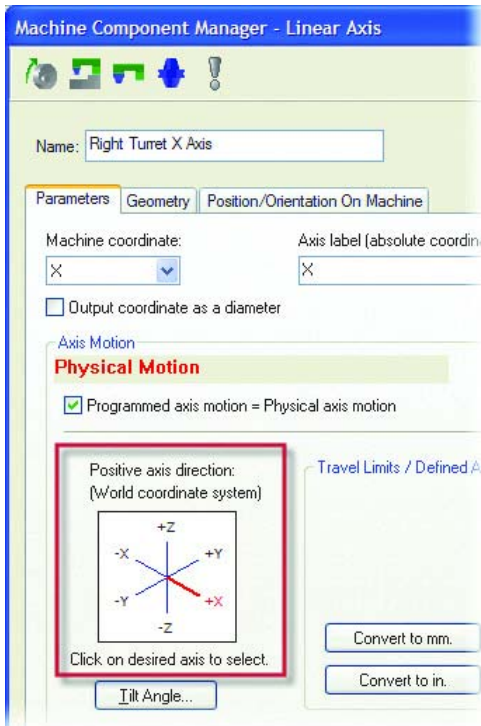




The positive axis direction for the X axis right turret was updated. This change provides the correct coordinates when backplotting.

Right turret X axis setting in X2 MR2:

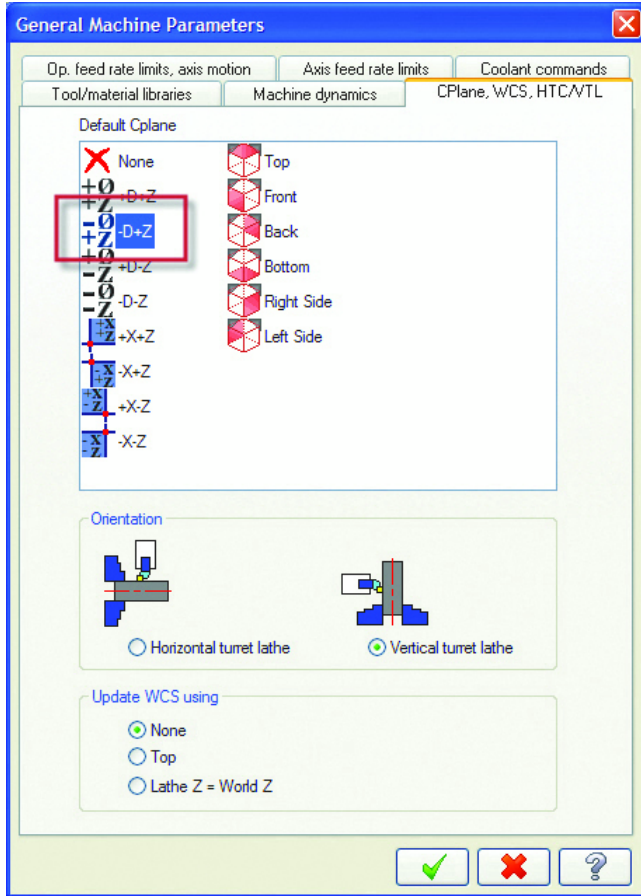
Right turret X axis setting in X3:



Note: This adjustment may require a post processor change. You may need to change `dia_mult` to -2:

`dia_mult : 2` #Multiplier for output on X axis (Neg. switches sign of X)

The default construction plane (Cplane) selection has changed to -D+Z.

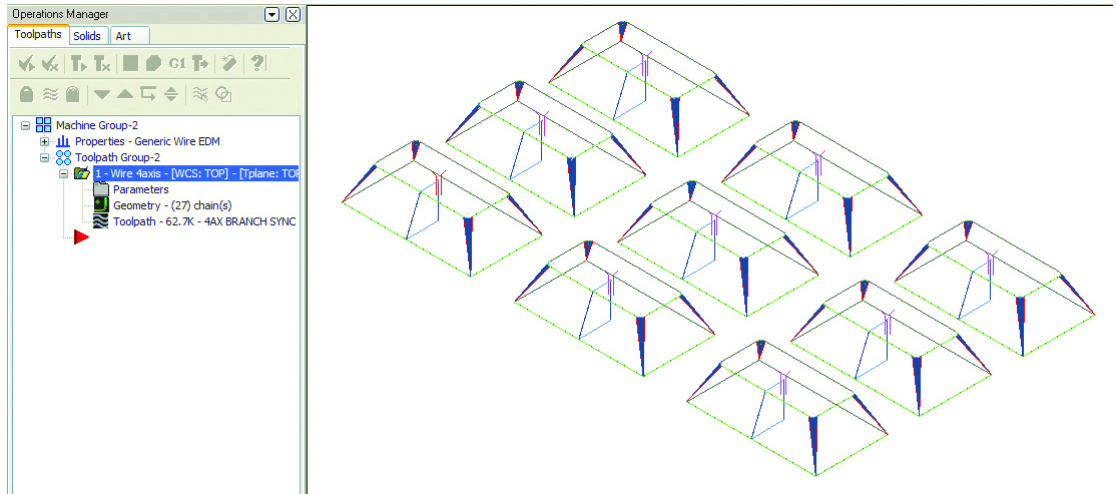


Wire Enhancements

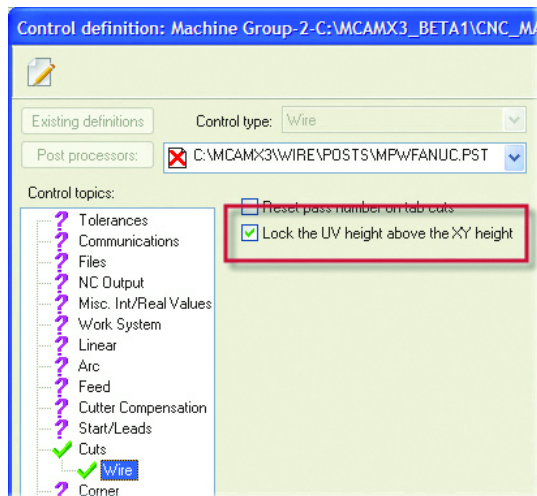
Mastercam X3 Wire simplifies your wire machining processes by offering improved part selection and additional wirepath parameters.

Multiple 4-Axis Wirepaths in One Operation

Mastercam can now support multiple 4-axis profiles in the same toolpath operation.



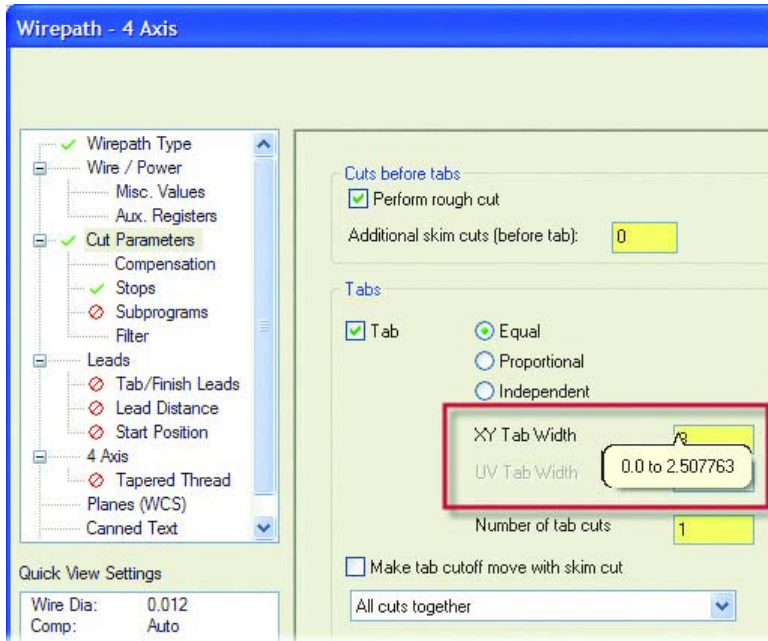
Window chaining has been added to help you select multiple profiles. Mastercam looks for the two chains with the closest endpoint to each thread point in the window. If you window chain multiple 4-axis parts that do not have corresponding thread points, Mastercam attempts to match the XYs with the correct UVs, but the start point on each chain may not be correct. To ensure that the lower contour (XY) is located first, always have a thread point and select the **Lock the UV height above the XY height** option in the Control Definition (Cuts, Wire page).



If the chain order of multiple 4-axis profiles in one window does not give the desired results, you can still use window chaining and/or regular chaining to build up the selections of XY/UV chain sets. You can chain each set individually to place all the profiles in a single operation.

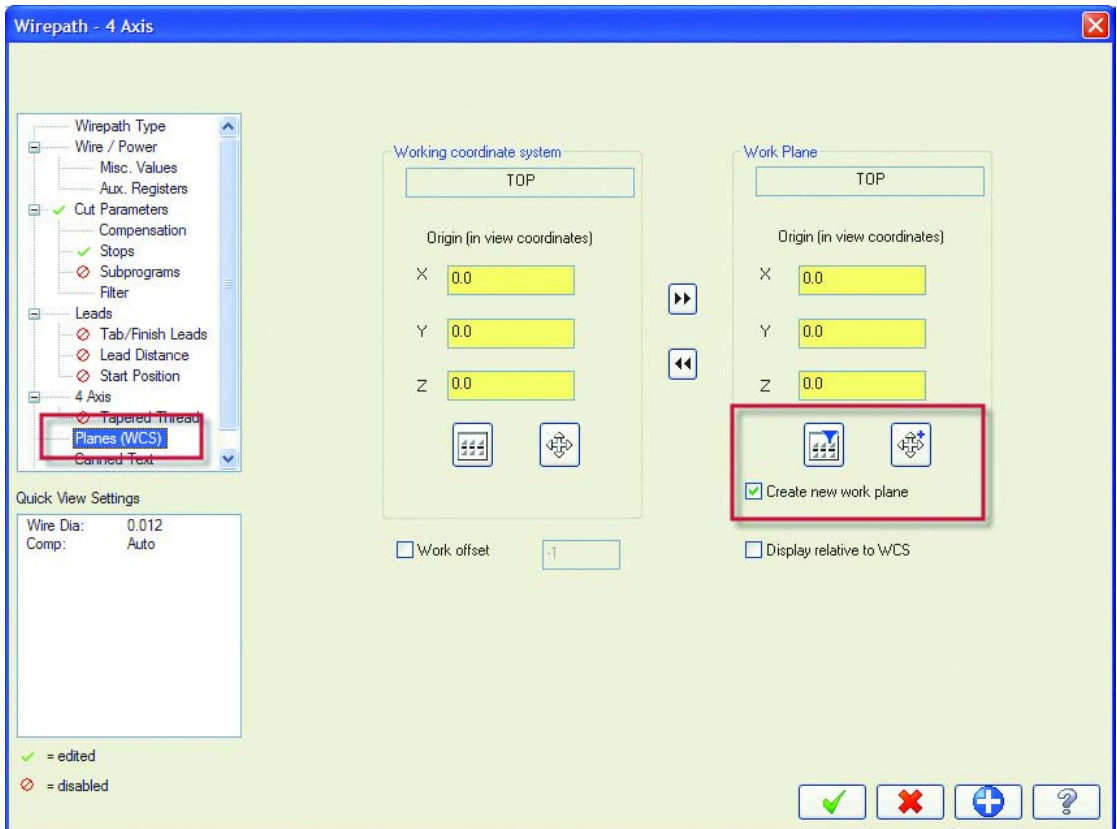
Setting UV Tab Widths in 4-Axis Wirepaths

For all tab cases in a 4-axis wirepath, the tab width cannot exceed the width of the last entity in chains of the respective (XY and/or UV) planes. If you try to enter a value that is too large, Mastercam displays the allowed range.



Changes to the Planes (WCS) Page in Wirepath Parameters

These improvements allow you to create new work planes and work plane origins (separate from the WCS origin) while programming wirepaths.



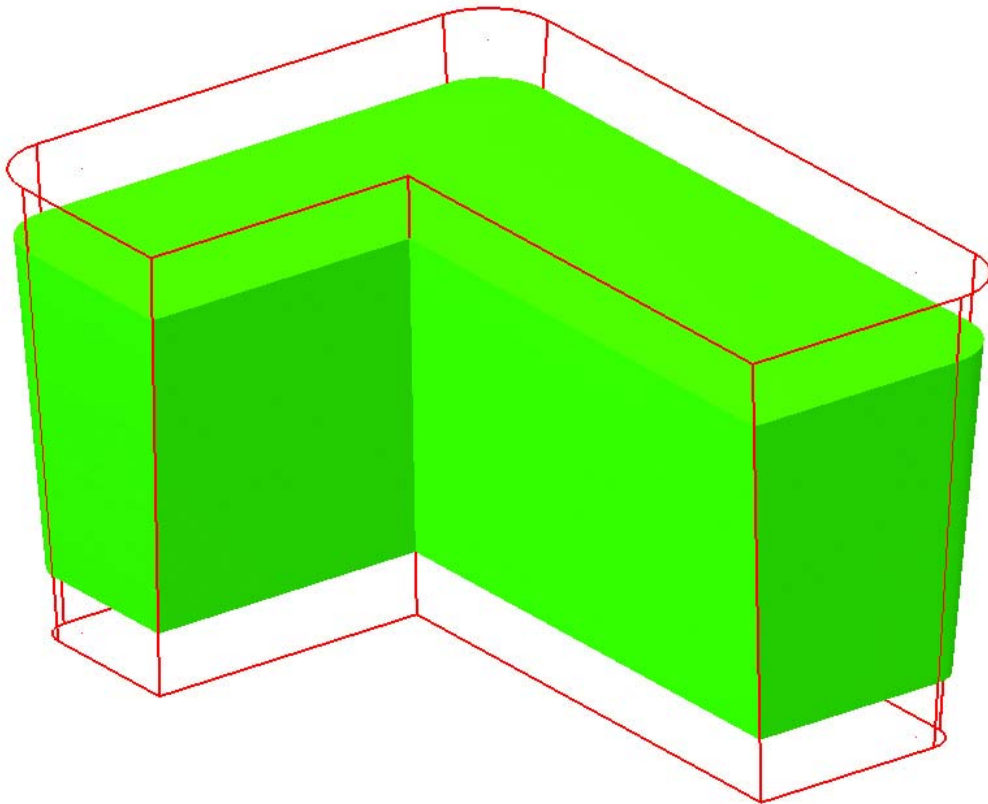
When you select the **Create new work plane** option, the Select Origin button icon will have a blue '+' sign. This means that when you click the Select Origin button, a new work plane (parallel to the current WCS view) will be automatically created for you. You can then set a new work plane origin that will be used for the new wirepath without affecting the WCS origin.

The blue triangle on the Select View button indicates that clicking this button will display only views parallel to the current WCS view.

Autosync Rails

This new feature on the Create menu creates geometry to assist with 4-axis wirepaths on solid models. Designed for parts where the upper and lower profiles (rails) need to be synchronized, Autosync Rails creates the rails at specific Z levels and adds proper branch lines to connect these profiles.

The following picture shows an example of how the upper and lower rails are projected as wireframe contours with branch lines to create sync points (nodes) where the upper and lower rail vectors are not perpendicular.

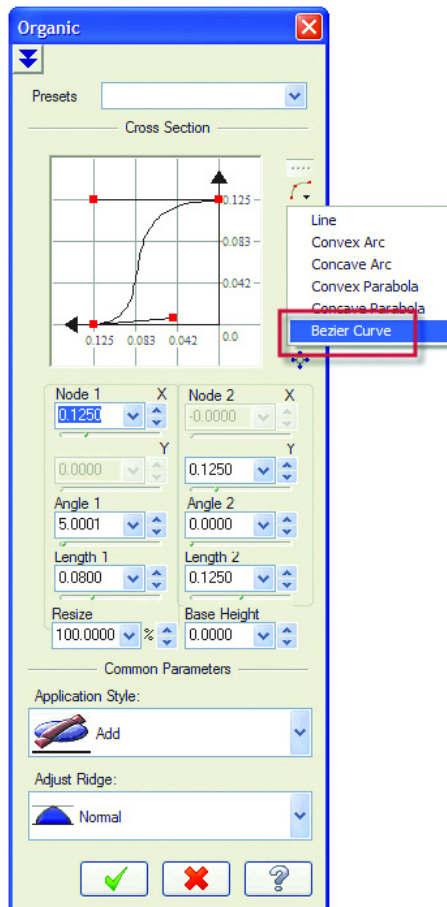


Art Enhancements

Mastercam X3 Art contains new features for drawing more complex surface types and improved functionality for organizing and displaying your work.

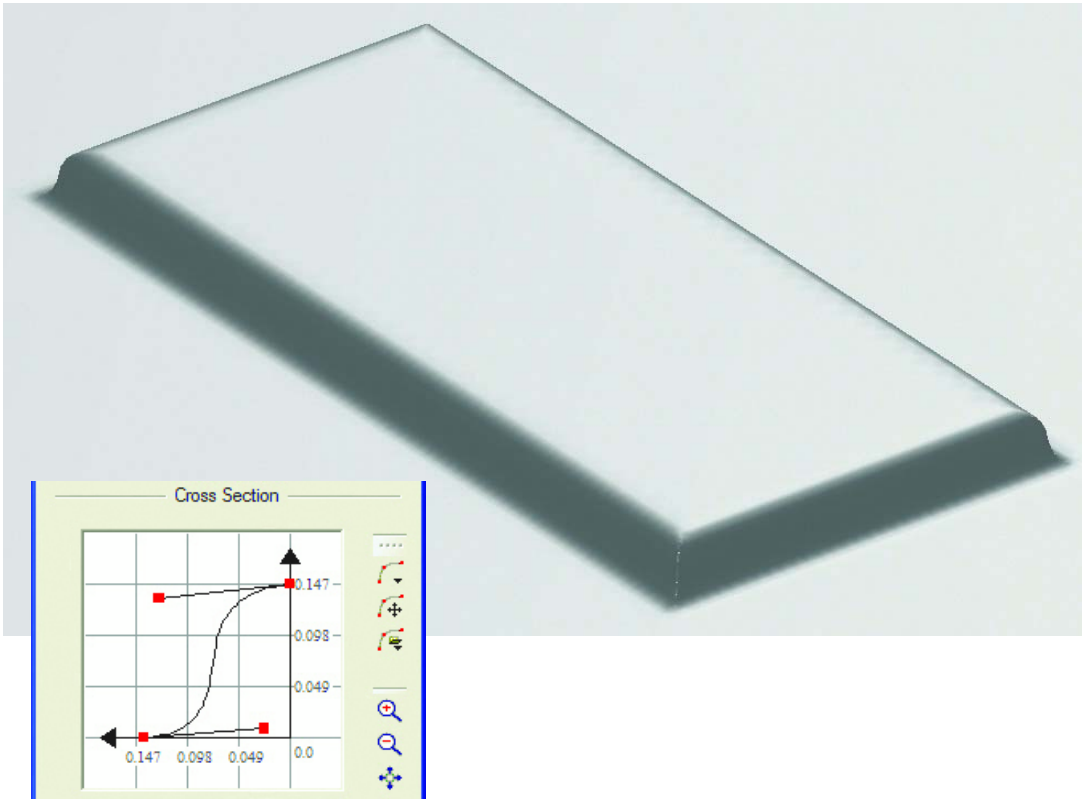
Bézier Curve Dynamic Cross Sections

Bézier Curve has been added to the dynamic cross section library, allowing you to create a new category of complex cross sections. The Bézier Curve cross section has beginning and ending node points that can be dragged dynamically to adjust the length, as well as two control handles that can be dragged and stretched to flex the curve. Many simple and complex shapes can be created with this new cross section. To create these shapes previously, the cross sections had to be drawn manually in Mastercam.



With the current dynamic cross sections, there has always been a sharp definition of the edge of each Art Surface Operation (ASO). With the Bézier Curve dynamic cross section, it is easy to make a

smooth transition. The picture below shows a Roman Ogee shape that is easily created with the Bézier Curve cross section.



New Sculpt Organic Surface

This new surface allows Art users to create more sophisticated surfaces and have more control over those surfaces. It drives an organic surface through two open contours. It also provides the ability to dynamically adjust not only the shape of the surface between these two drive curves (cross section) but also the shape of the surface along the length of the drive curves (ridge profile). You can also dynamically adjust where the drive curves intersect as they build the surface from the edges towards the center (ridge path).

The easy-to-use interface lets you control all characteristics of the complex 3D surface operation with 2D dynamic dialogs. From these dialogs, you can manipulate drive splines by adjusting a pre-defined Bézier Curve or by selecting geometry you create in the Mastercam graphics window. If desired, the Sculpt Organic surface accepts 3D drive curves, allowing you even greater control and precision.

► Creating a Sculpt Organic surface

Create the Cross Section

- 1 From the Mastercam menu, choose **File, Open**.
- 2 Go to ..\Documentation\ExampleParts and select FISH_SCULPT_ORGANIC.MCX.
- 3 Select **Art, Create Art Surface Operation, Sculpt Organic** from the Mastercam menu.
- 4 Select the two outside drive curves. Start each selection from the head to make sure that the chains run in the same direction.



IMPORTANT: Always chain Sculpt Organic drive curves in the same direction.



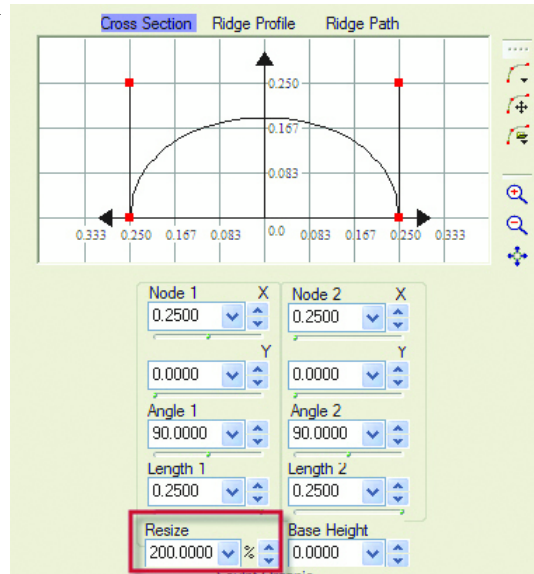
- 5 Click **OK** to complete the chaining.

The **Sculpt Organic** dialog box displays with **Cross Section** tab active.

The Cross Section tab controls the shape of the cross section created between the drive curves.

6 Enter **200** in the Resize field.

Resizing scales the cross section, changing the height of the surface.



7 Click the **Zoom to Fit** button to the right of the cross section area so the entire cross section displays in the grid area.



8 Use one of two methods to shape the cross section:

- ◆ Click and drag the curve's red endpoints and control points to shape the cross section visually.
- ◆ Enter the values below into the parameter fields.

Node 1: X	0.2500	Node 2: X	0.2500
Y	0.0000	Y	0.0000
Angle 1	21.500	Angle 2	20.000
Length 1	0.2000	Length 2	0.2225

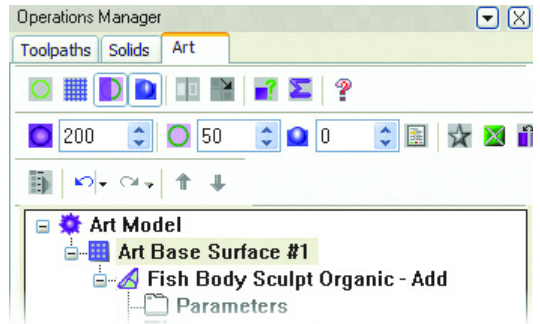
9 Click **OK** to save your modifications.



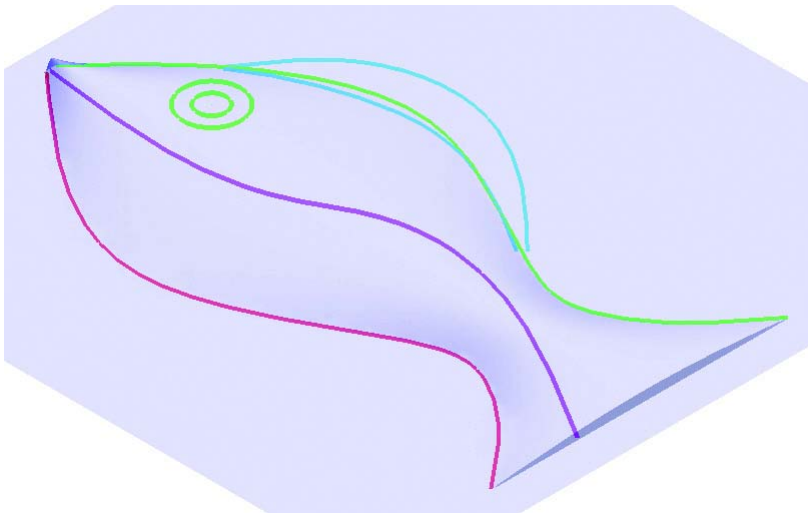
- 10** Name the surface **Fish Body** and click **OK**.

By default, Art prompts you to name new surface operations.

The new Art surface operation displays in the Art Manager.



- 11** Right-click in the graphics window and select **Isometric Gview** to orient the part in the isometric graphics view.
- 12** Press **[Alt + S]** to toggle Art model shading. The new Sculpt Organic surface should look like the illustration.



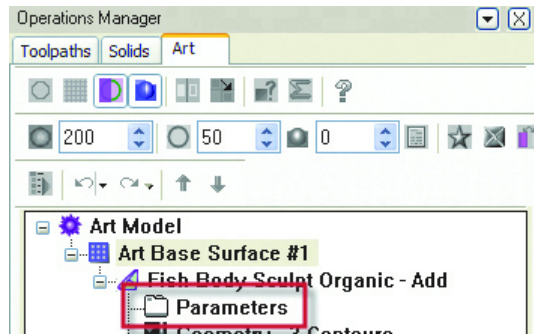
Add a Ridge Profile

The ridge profile defines the shape of the fish from the nose to the tail.

- 1 In the Art Manager, double-click the **Parameters** icon under the Fish Body Art surface operation.

The Sculpt Organic dialog box opens.

- 2 Select the second tab, **Ridge Profile**.



- 3 Click the **Dynamic Cross section** icon.

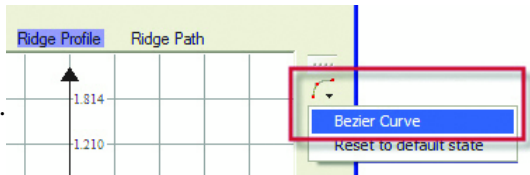
- 4 Select **Bézier Curve**.

The default Bézier curve displays in the grid.

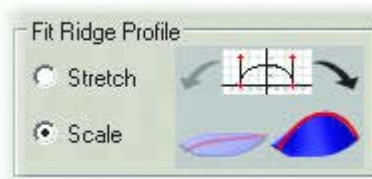
- 5 Use one of two methods to modify the ridge profile:

- ◆ Click and drag the curve's endpoints and control points to shape the ridge profile visually.
- ◆ Enter the values below into the parameter fields.

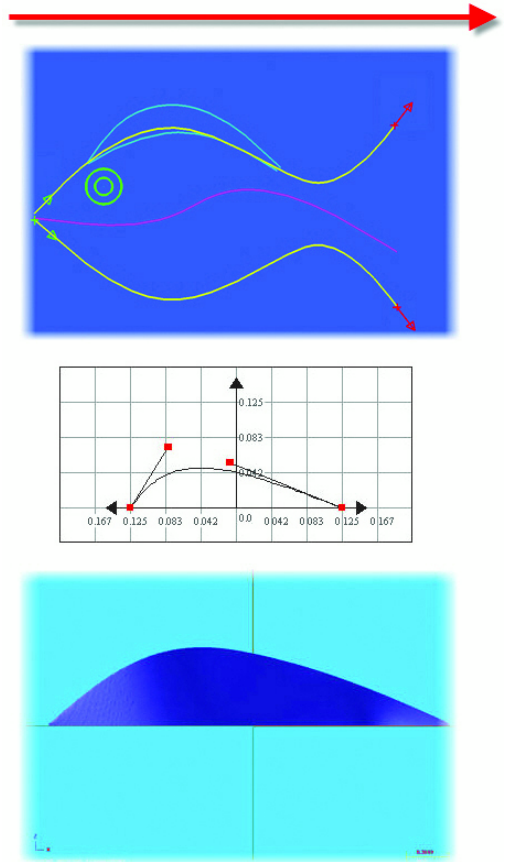
Angle 1 58.750 **Angle 2** 22.000
Length 1 0.085 **Length 2** 0.145



- 6 Select the **Scale** option in the Fit Ridge Profile section of the dialog.



The ridge profile follows the drive curve chaining direction.



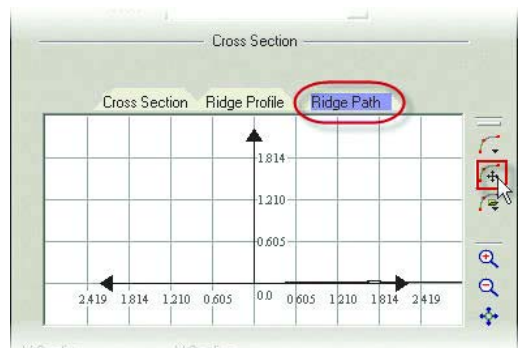
Add a Ridge Path

Use a ridge path to give the fish visual movement. If not specifically selected, Art automatically creates a ridge path at the center of the two drive curves.

- 1 Select the third tab, **Ridge Path**.

The default ridge path is represented by a flat line on the X axis to the right of the origin.

- 2 Click the **Use Mastercam to Pick Cross-section** icon.

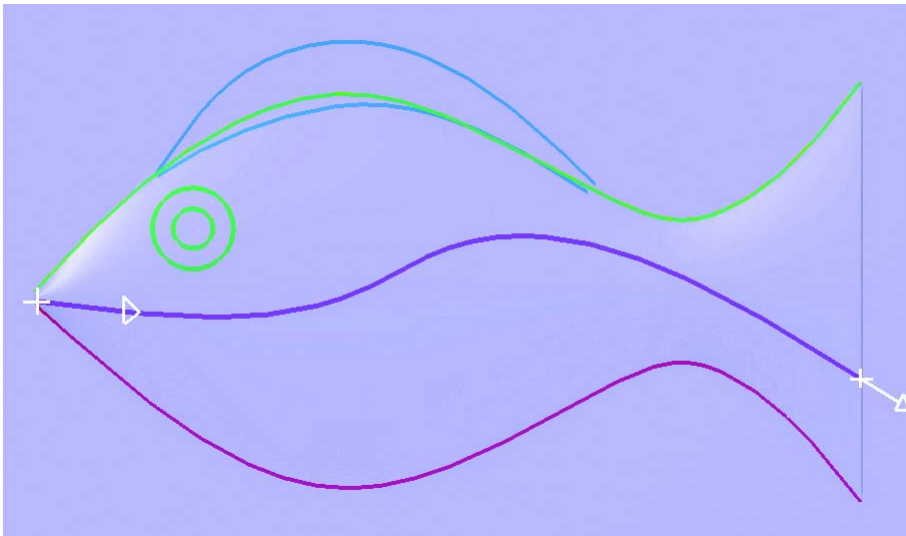


- 3 Right-click in the graphics window and select **Top Gview** to make it easier to select the geometry.

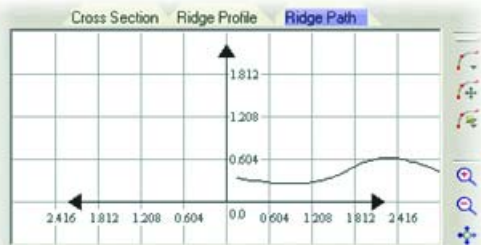
*Note: You may need to right-click in the graphics window and select **Repaint** to see the wireframe geometry on top of the Sculpt Organic surface.*

- 4 From the end closest to the nose, select the center curve.

Remember that all drive curves in a Sculpt Organic surface must be chained in the same direction.



The new ridge path displays in the grid.



- 5 Click **OK** to save your modifications. The modified surface should look like the following illustration.

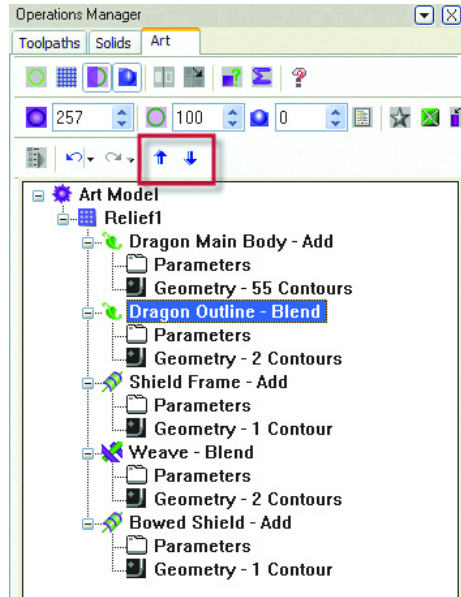


Art Manager Improvements

Moving Art Surface Operations

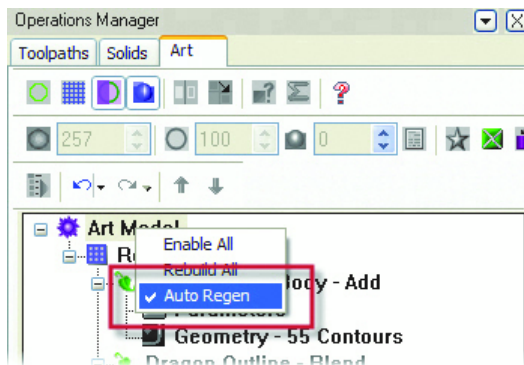
Use the new Up and Down arrows in the Art Manager to move Art Surface Operations (ASOs) within the Art Manager list. Regenerate the list when you have your ASOs in the correct position. You can use this new function to help keep large projects organized and for situations when the order of the ASOs changes the appearance of your part.

Note: The arrow icons are disabled until there are at least two ASOs in the Art Manager.



Auto Regeneration

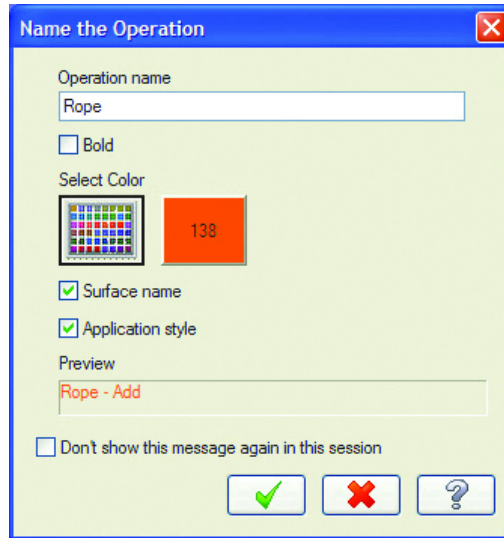
Mastercam X3 Art includes the ability to have any edited operations automatically regenerate in the Art Manager. Right-click on the Art Model in the Art Manager tree and select **Auto Regen**. When selected, this option regenerates all items automatically. If deselected, edited items require you to click the Regenerate button.



Labeling Art Surface Operations

Advanced Art users know that naming ASOs as you create them is a best practice that greatly increases efficiency in editing, because it indicates which ASO creates which shape in the model. When you create a new ASO in Mastercam X3 Art, you are immediately prompted to name it. If you enter **Rope** as the name, the Art Manager displays the ASO name as **Rope-Add**. If you cancel the dialog box, the ASO is labeled using the standard naming, which is just the operation type.

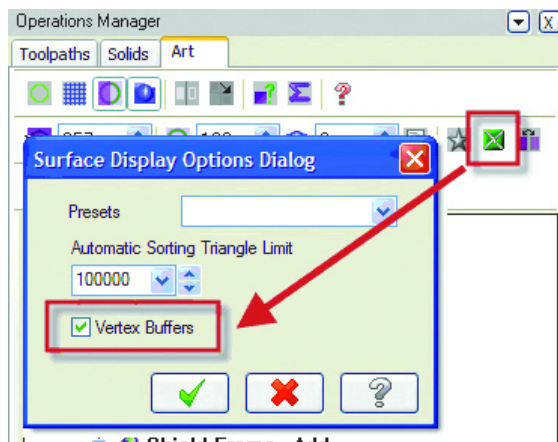
When you enter the ASO name, you can also select a color or use bold text for the name in the Art Manager. This makes it much easier to manage complex Art projects with many ASOs.



Vertex Buffers

If you have an OpenGL® graphics card, selecting the **Vertex Buffers** check box on the Surface Display Options dialog box increases your speed when working on large Art files. To see speed improvement, you need a project with at least 500,000 triangles where you need to rotate the ABS.

Note: Transparency must be turned off for this function to work correctly.



Art Shading and Material Mapping Improvements

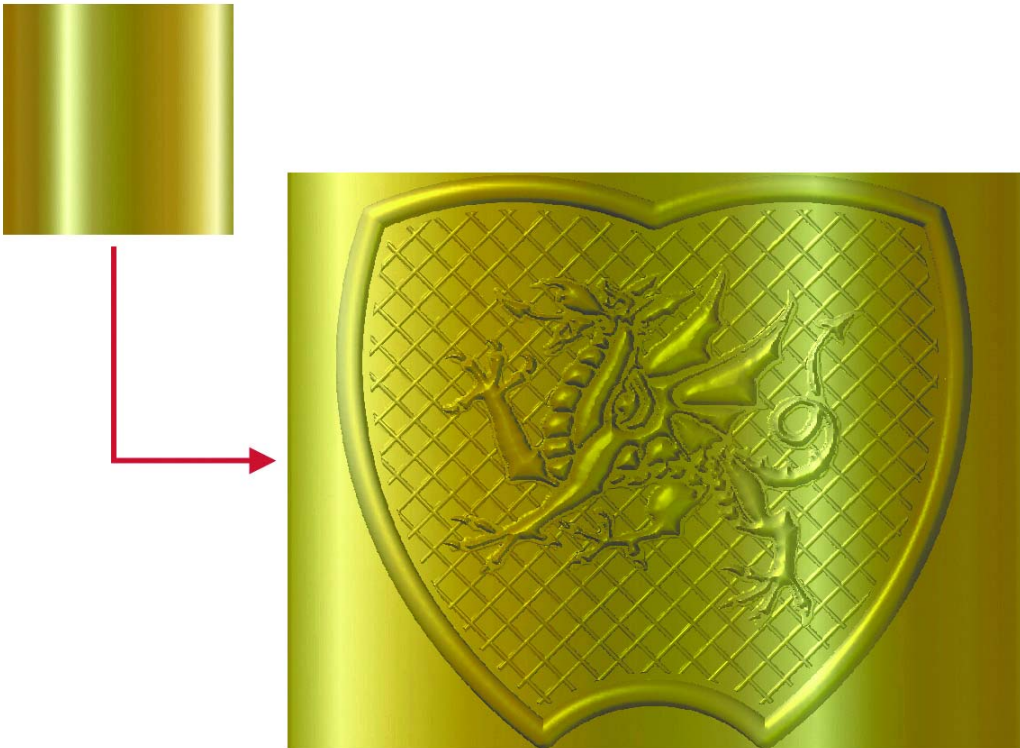
You can now map materials to the Art surface, which allows you to show the Art model in the material specified by the customer. If you need an additional material, you can add a square BMP file (16x16 pixels, 32x32, 64x64, 128x128) to the `..\Art\Art material bitmaps` folder. You can also simultaneously blend, show material, add reflection, adjust color and opacity, and see them all applied to the model in real time.

The material bitmap function actually maps the bitmap onto the Art surface, so you see all of the detail from the material bitmap on the Art surface. This is different from the way reflection works. The reflection bitmap is an invisible dome over the Art surface that reflects against the surface, although the full detail of the reflection bitmap is never realized on the Art surface.

The following picture shows a part with no material or reflection applied.



In the following picture, the Material function mapped the selected bitmap file over the Art surface. Notice that you can see the vertical bars from the original bitmap. The Material function overlays the selected bitmap, retaining all of its detail.



The next example uses Reflection. Material has been turned off, and you can see how the vertical bars from the original bitmap are not visible. The Reflection function reflects the bitmap onto the surface, whereas the Material function maps it.





cnc software, inc.

671 Old Post Road
Tolland, CT 06084 USA
www.mastercam.com

Printed in the USA
Mastercam X3 What's New in Mastercam X3 1-883310-72-5