

Mastercam®

Getting Started Tutorials



Basic 2D Machining

mastercam x getting started tutorials
.....

Basic 2D Machining

September, 2007

Mastercam® X2 MR2 Basic 2D Machining

Date: September, 2007

Copyright © 2007 CNC Software, Inc.— All rights reserved.

First Printing: September, 2007

Software: Mastercam X2 MR2

ISBN: 1-883310-81-4

IMPORTANT NOTICE!

PLEASE READ THIS STATEMENT AND THE SOFTWARE LICENSE AGREEMENT COMPLETELY BEFORE USING THIS SOFTWARE.

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.

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. 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 reasonably 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 reasonable best efforts to ensure their compliance with such terms and conditions, including, without limitation, not knowingly permitting 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 in a manner not authorized by this Software License Agreement is using the Software, you shall immediately use reasonable best efforts to have such unauthorized use of the Software immediately cease. 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." YOUR SOLE REMEDY AND CNC'S SOLE OBLIGATION HEREUNDER SHALL BE, AT CNC'S SOLE OPTION, REPLACEMENT OF THE DEFECTIVE MEDIA OR REFUND OF THE PURCHASE PRICE OF THE SOFTWARE. ANY USE BY YOU OF THE SOFTWARE IS AT YOUR OWN RISK. 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. SHOULD THE SOFTWARE PROVE DEFECTIVE FOLLOWING ITS PURCHASE, YOU (AND NOT CNC, ITS DISTRIBUTOR, OR RESELLER) ASSUME THE ENTIRE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION AND ANY INCIDENTAL OR CONSEQUENTIAL DAMAGES.

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.**

Limitation of Liability

IN NO EVENT WILL CNC, OR ITS EMPLOYEES, SHAREHOLDERS, DISTRIBUTORS OR RESELLERS BE LIABLE TO YOU FOR ANY INDIRECT, INCIDENTAL, OR CONSEQUENTIAL DAMAGES (INCLUDING WITHOUT LIMITATION, SPECIAL, PUNITIVE OR EXEMPLARY DAMAGES FOR LOSS OF BUSINESS, LOSS OF PROFITS, BUSINESS INTERRUPTION, OR LOSS OF BUSINESS INFORMATION) ARISING OUT OF OR IN CONNECTION WITH THIS SOFTWARE LICENSE AGREEMENT OR THE SUBJECT MATTER HEREOF EVEN IF CNC HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. 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 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 as a result of any breach of this Software License Agreement by you or your employees, agents, resellers, dealers or sub-dealers and shall reimburse CNC 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 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.

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 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.

Contents

.....

Introduction	1
▶ Tutorial Goals	2
▶ Before You Begin	2
▶ If You Need More Help	3
▶ Additional Documentation	4
 1. Contouring Outside of Part	 5
▶ Lesson Goals	5
▶ Exercise 1: Assigning a Machine Definition	5
▶ Exercise 2: Creating a Toolpath on the Outside of the Part	6
 2. Pocketing the Large Interior Shape	 13
▶ Lesson Goals	13
▶ Exercise 1: Creating a Pocket Toolpath	13
▶ Exercise 2: Viewing Your Toolpath	17
 3. Pocketing the Inside Slot Shape	 19
▶ Lesson Goals	19
▶ Exercise 1: Creating a Pocket Toolpath on the Slot	19
 4. Drilling Holes	 25
▶ Lesson Goals	25
▶ Exercise 1: Drilling Four Holes	25
 5. Backplotting Toolpaths	 29
▶ Lesson Goals	29
▶ Exercise 1: Backplotting All Toolpaths	29
▶ Exercise 2: Verifying All Toolpaths	32
 6. Posting Toolpaths	 33
▶ Lesson Goals	33

- ▶ Exercise 1: Posting All Toolpath Operations.....33
- ▶ Post Processing Summary: Sending NC Files to Machine37

Conclusion37

INTRODUCTION

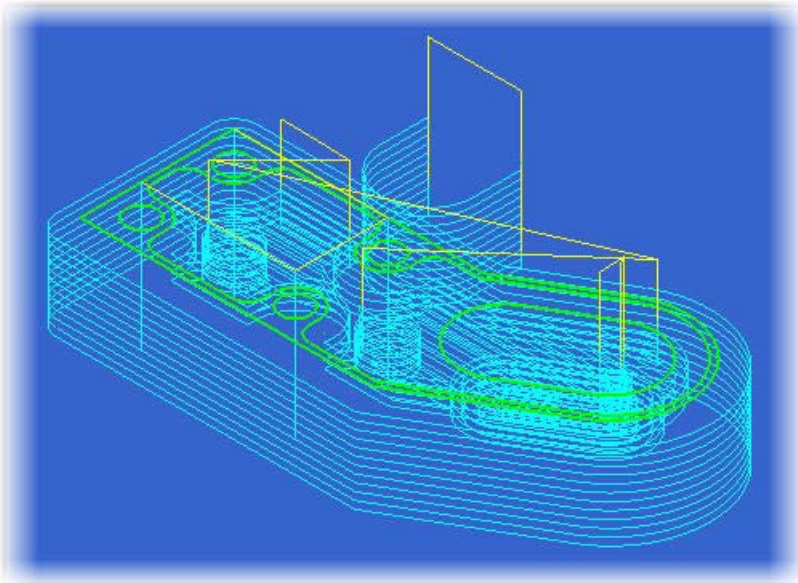
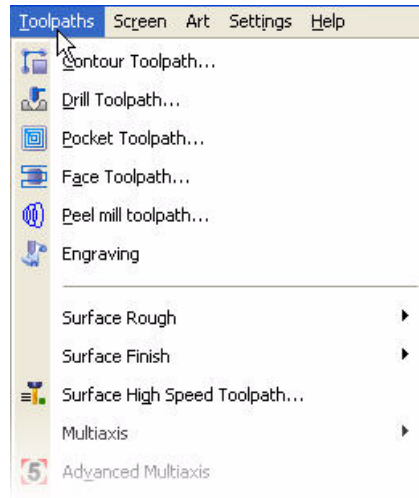
This tutorial focuses on applying several mill toolpaths to a part previously designed in Mastercam. The tutorial then guides you through the steps to take that toolpathing data and create code to machine the part.

Mastercam offers a variety of toolpath types that let you quickly build toolpaths for specific applications. You access these toolpaths, such as the mill toolpaths shown here, through the Toolpaths menu.

After you create a toolpath, you can use Mastercam's backplot feature to preview its operation on the screen. Once you are satisfied with it, post it from the Toolpath Manager to generate the NC code for a specific machine tool.

When you first begin this tutorial, you will be prompted to assign a machine definition to the part. This tutorial does not go into any depth on machine and control definitions. However, the Help and other documentation installed with Mastercam provide comprehensive information regarding these Mastercam features.

When you finish this tutorial, your part will look like this:



If you would like to design (create) the part before you begin this tutorial, please follow the "Basic 2D Design" tutorial procedures.

Tutorial Goals

- Open a part file, assign a default machine definition, and create a contour toolpath on the outside of the part (including chaining entities, choosing tooling, and setting machining values).
- Create two pocket toolpaths with helix entries on the inside of the part (including chaining entities, choosing tooling, and setting machining values).
- Create four drill holes with one toolpath (including selecting a drill point, choosing tooling, using tool tip compensation, and setting machining values).
- View all toolpaths in graphics window.
- View a specific toolpath by temporarily turning off the display of selected toolpaths.
- Backplot (view the path the tools take to cut the part) all toolpaths.
- Customize your backplot display.
- Simulate (verify) the machining of the part from a stock model display.
- Post all toolpath operations to an NC file, review/edit the code as necessary, and save the NC file.

Before You Begin

This is a module of the *Mastercam X Getting Started Tutorial Series*, which introduce basic Mastercam skills. Other tutorial series, which cover more advanced skills, are:

- *Focus Series*—This series provides more in-depth training on specific or advanced Mastercam features and functions.
- *Exploring Series*—This series explores the application of a single Mastercam product, such as Mill, Wire, or Art.

The Mastercam X tutorial series is in continual development, and we will add modules as we complete them. For information and availability, please contact your local Mastercam Reseller.

General Tutorial Requirements

Because each lesson in the tutorial builds on the mastery of preceding lesson's skills, we recommend that you complete them in order. In addition, the tutorials in this series have the following requirements:

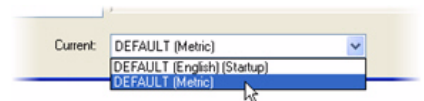
- You must be comfortable using the Windows™ operating system.
- You must have a seat of Mastercam X2 MR2 Design or higher to complete most of the tutorials in the *Getting Started* series. The tutorials cannot be used with Mastercam Demo and Mastercam Student versions. (Refer to the current *Mastercam Installation Guide* for license information. You can find this guide in the \Documentation folder of your Mastercam installation.)
- The *Basic 2D Machining* tutorial requires a seat of Mill Entry or Router Entry. Tutorials in other series may require higher level licenses.

- Part files may accompany a tutorial. They should be stored in a folder with the tutorial or in a location that you prefer.
- You must store tutorial Flash™ videos in the \Videos folder of your Mastercam installation. (Flash videos accompany several tutorial modules and are available for download from www.mastercam.com.)
- You must install Flash Player to display tutorial videos. You can download Flash Player from www.adobe.com/products/flashplayer.
- You must configure Mastercam to work in metric units. The next section includes instructions for setting Mastercam to metric.

Preparing for a Tutorial


Before you start a tutorial, be sure you have completed the following tasks:

- 1 Start Mastercam using your preferred method:
 - ♦ Double-click Mastercam's desktop icon.
 - Or
 - ♦ Launch Mastercam from the Windows Start menu.
- 2 Select the metric configuration file:
 - a Select **Settings, Configuration** from Mastercam's menu.
 - b Choose **DEFAULT (Metric)** from the **Current** drop-down list.
 - c Click **OK**.



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 bar 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 X2 Installation Guide*
- *Mastercam X2 Getting Started Guide*
- *Mastercam X2 Reference Guide*
- *Mastercam X2 Quick Reference Card*
- *Version 9 to X Function Map*

LESSON 1

Contouring Outside of Part

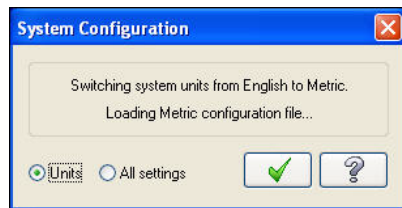
Lesson Goals

- Open a part file and assigning a machine definition.
- Create a contour toolpath (including chaining entities, choosing tooling, and setting machining values).

Exercise 1: Assigning a Machine Definition

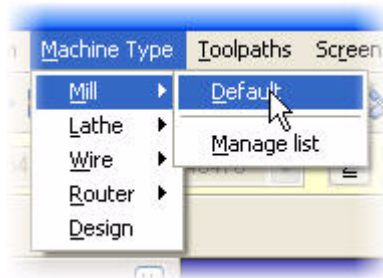
In this exercise, you open an existing part, save it under a new file name, and assign a machine definition to the part.

- 1 Open the tutorial part file
BASIC_2D_MACHINING_START.MCX.
- 2 Click **OK** if prompted to switch to a metric configuration.



- 3 Choose **Machine Type, Mill, Default** to open the default Mill machine definition.

In Mastercam, you select a machine definition before creating any toolpaths. The machine definition is a model of your machine tool's capabilities and features and acts like a template for setting up machining jobs.



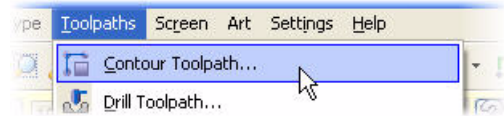
Note: Parts that have previously been saved with a machine definition will automatically load the associated machine definition.

- 4 Choose **File, Save As** and save the part under a different file name than the original file. This will protect the original tutorial file from being overwritten.

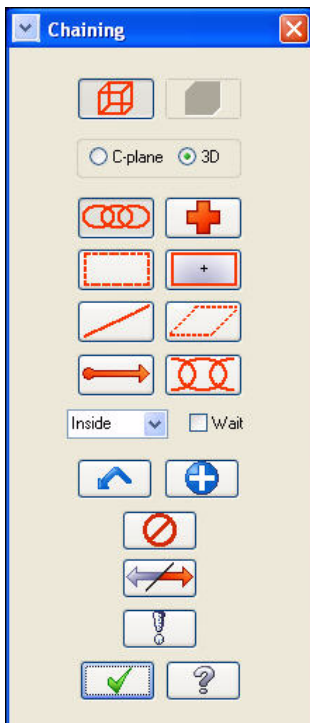
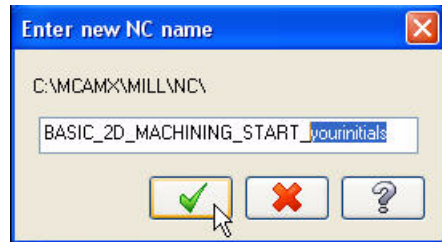
Exercise 2: Creating a Toolpath on the Outside of the Part

In this exercise, you create a contour toolpath on the outside of the tutorial part.

- 1 Choose **Toolpaths, Contour Toolpath**.



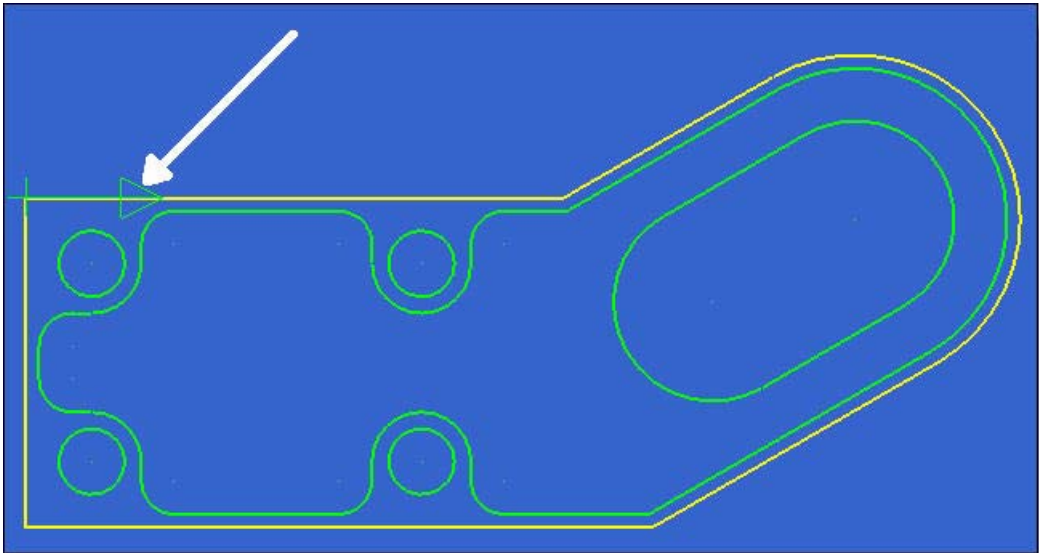
- 2 If prompted, click **OK** to confirm the default NC file name, or overwrite the file name and click **OK** to modify the default file name. The Chaining dialog box opens.



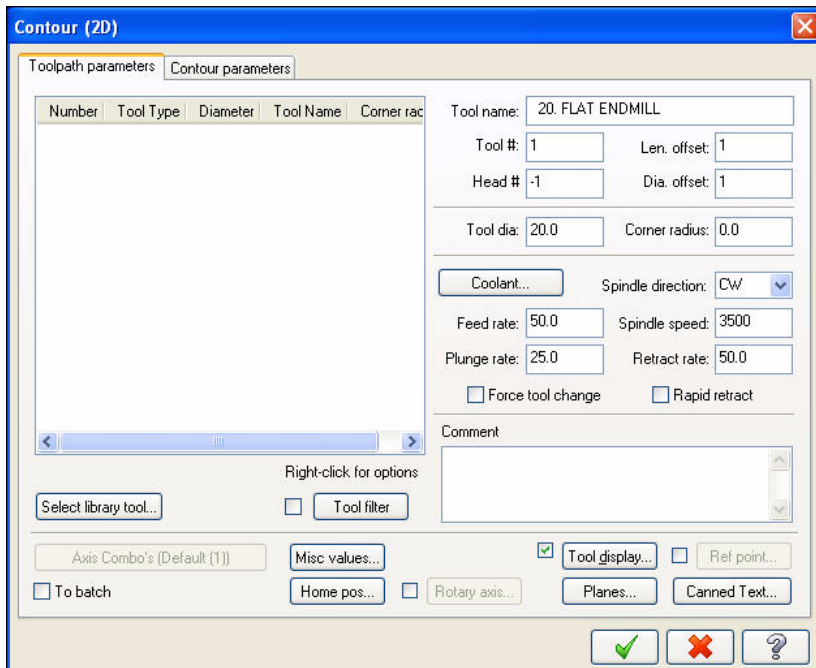
- 3 Click the outside contour of the part to chain it. The chaining arrow should point clockwise (as shown on the next page). If the chaining direction arrow is pointing counterclockwise, click the **Reverse** button on the Chaining dialog box.



Note the clockwise chaining direction.

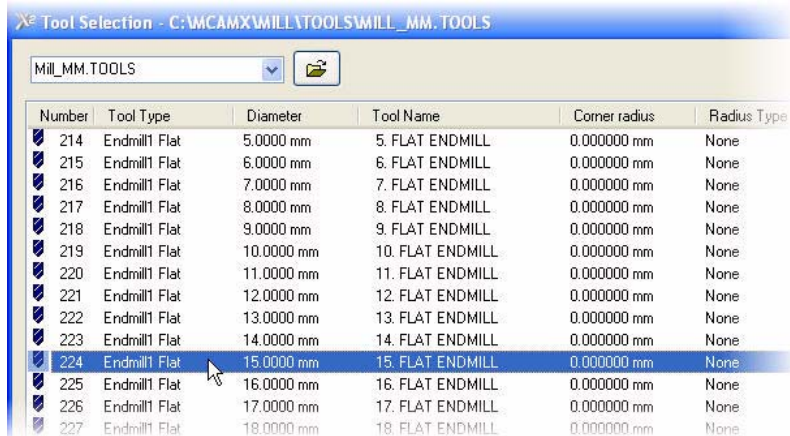


- 4 Click **OK** on the Chaining dialog box to chain the part. The Chaining dialog box closes and the Contour (2D) dialog box opens.

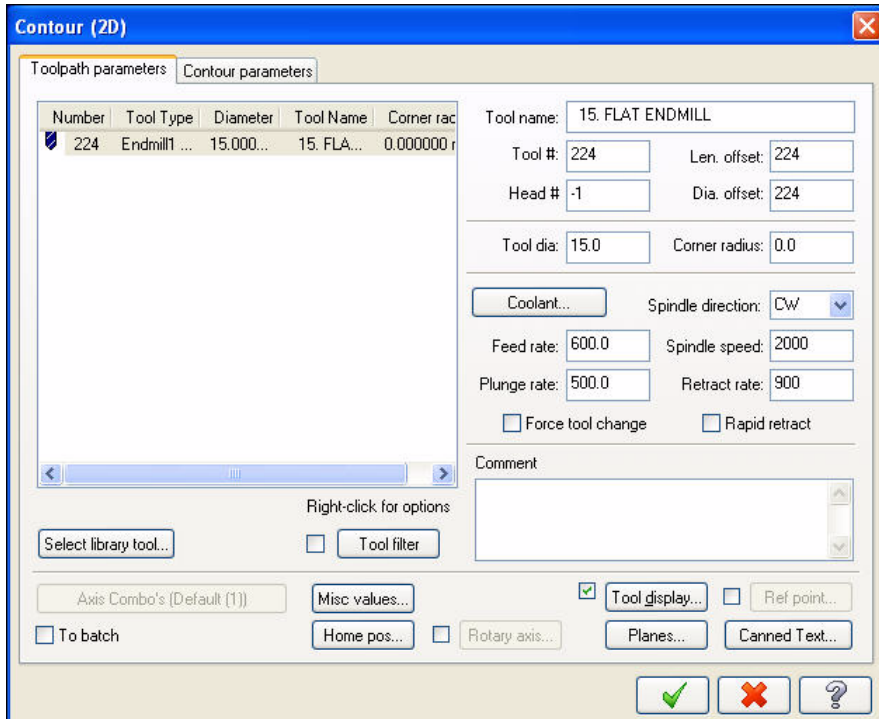


- 5 Click the **Select library tool** button. The default metric tool library opens.
- 6 Select the 15mm diameter flat endmill, and click **OK**.

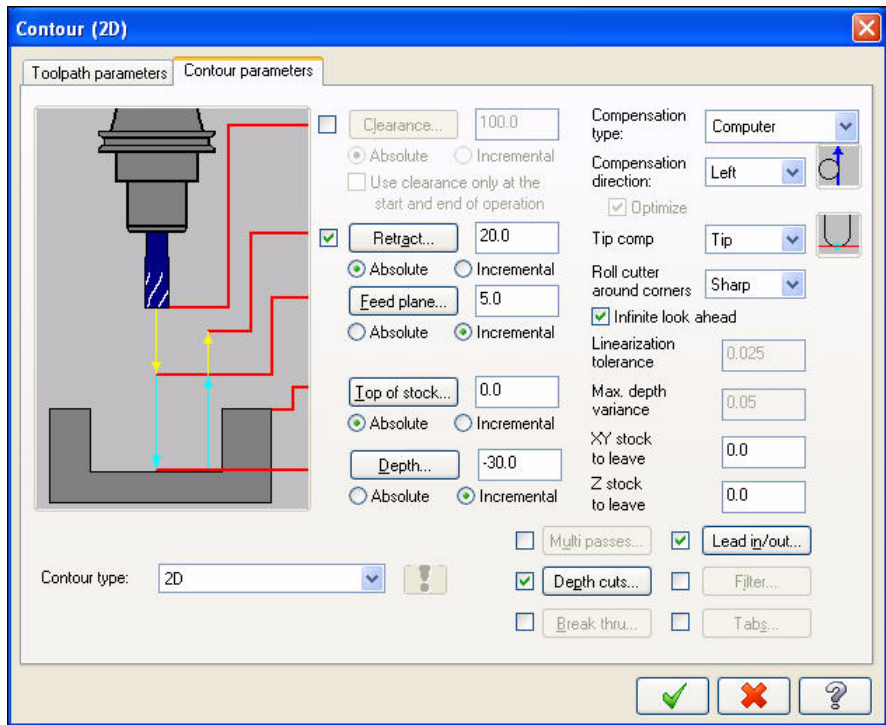
Select library tool...



- 7 Enter the values as shown here. (Do not click **OK** yet.)

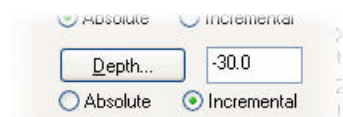


- 8** Click the **Contour parameters** tab and enter the values as shown here. Makes sure to check the **Depth cuts** check box to activate the button. (Do not click **OK** yet.)

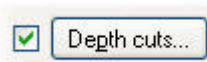


TIPS:

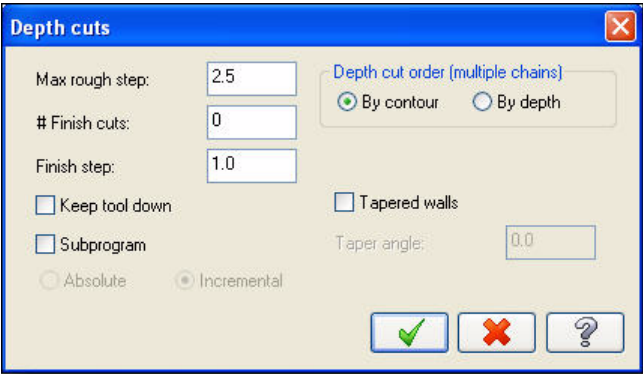
- The Depth value determines the final machining depth and the lowest depth that the tool descends into the stock. The default depth is the depth of the selected geometry. In this case, we have set the depth value at -30mm, that is 30mm below the top of the part.
- Mastercam uses lead in/out moves as a default. These moves are a combination of lines and arcs at the beginning and end of a 2D or 3D contour toolpath that control how the tool approaches and pulls away from the toolpath. Mastercam places entry and exit lines relative to the entry and exit arcs. If both an entry line and an entry arc are defined, the line is machined first. If both an exit line and an exit arc are defined, the arc is machined first. The lead in/out entry points may differ depending on where you click the entity to chain it.



- 9 Click the **Depth cuts** check box and button, enter the values as shown here, and click **OK**.



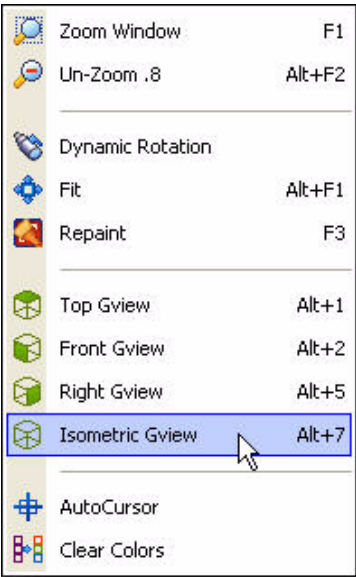
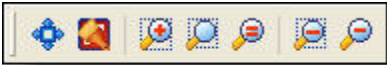
Contour depth cuts divide the total depth of a toolpath into smaller Z-axis cuts to reduce tool wear.



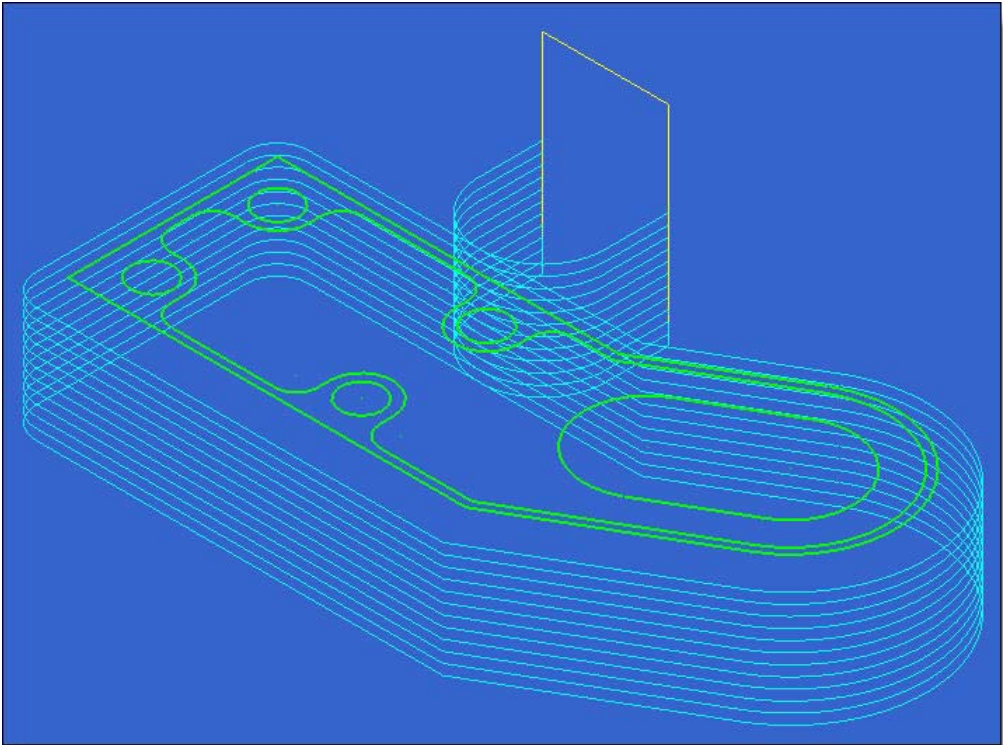
TIP: Depth cuts are Z-axis cuts that tools make in toolpaths. You can enter a maximum rough step and Mastercam divides the total depth into equal steps. Or you can enter the exact number of finish steps and the size of each finish step. Pocket depth cuts include the ability to use any island depths to set the depth cuts. For more details, please refer to the Mastercam Help.

- 10 Click **OK** to generate the toolpath.
- 11 Right-click in the graphics window and choose **Isometric Gview** from the menu to view the part and toolpath in the isometric view.

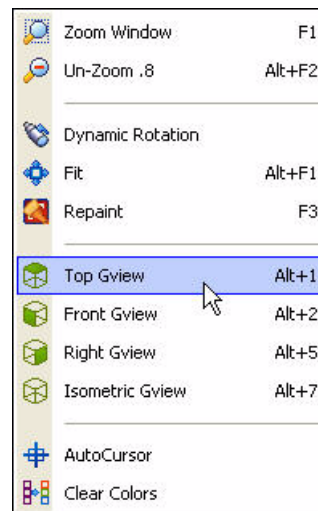
You may need to center the part in the graphics window to see it. The easiest way to do this is to use the graphics window right-click menu and select **Fit** to fit the part to the screen size [Alt+F1], then unzoom [Alt+F2]. You can also use the fit/zoom/unzoom buttons in the View Manipulation toolbar.



Your toolpath should look like this.



- 12** Right-click in the graphics window again and choose **Top Gview** from the menu to view the part and toolpath in the top view.



- 13** Choose **File, Save** or click the **Save** button to save your part.



LESSON 2

Pocketing the Large Interior Shape

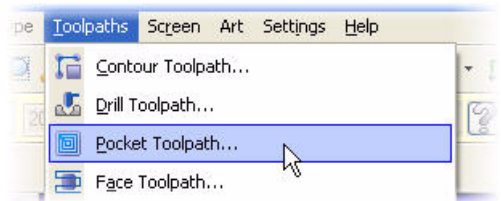
Lesson Goals

- Create a pocket toolpath (including chaining entities, choosing tooling, and setting machining values).
- View a specific toolpath by temporarily turning off the display of selected toolpaths.

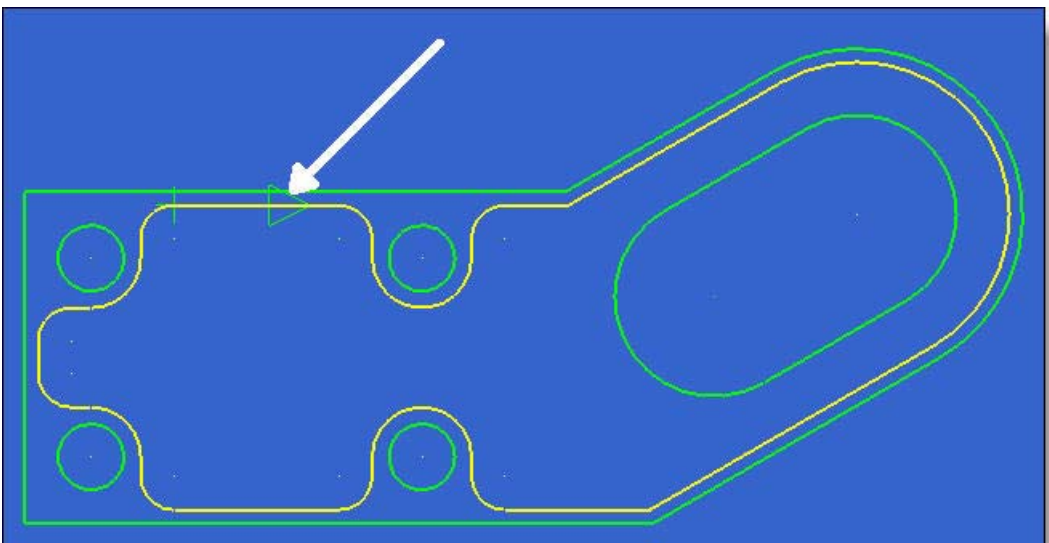
Exercise 1: Creating a Pocket Toolpath

In this exercise, you create a pocket toolpath with a helix entry on the large interior shape of the tutorial part.

- 1 Choose **Toolpaths, Pocket Toolpath**. The Chaining dialog box opens.

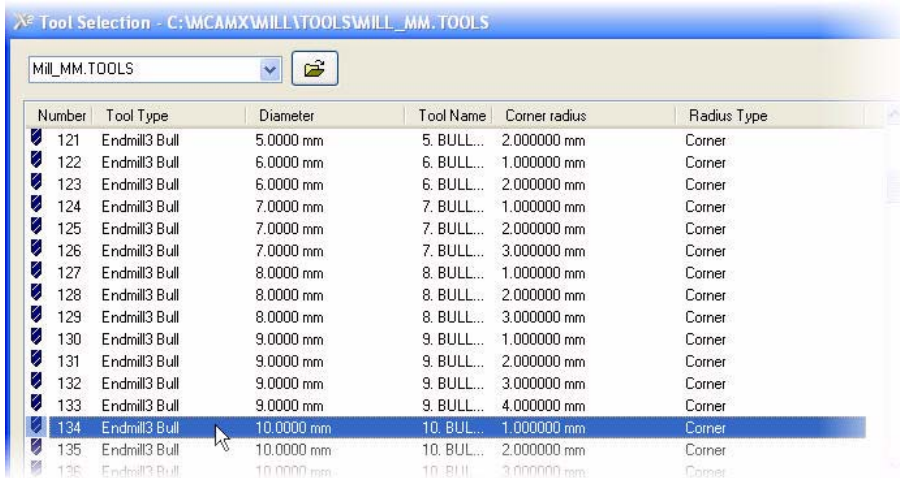


- 2 Click the large interior shape of the part to chain it. The chaining arrow should point clockwise.

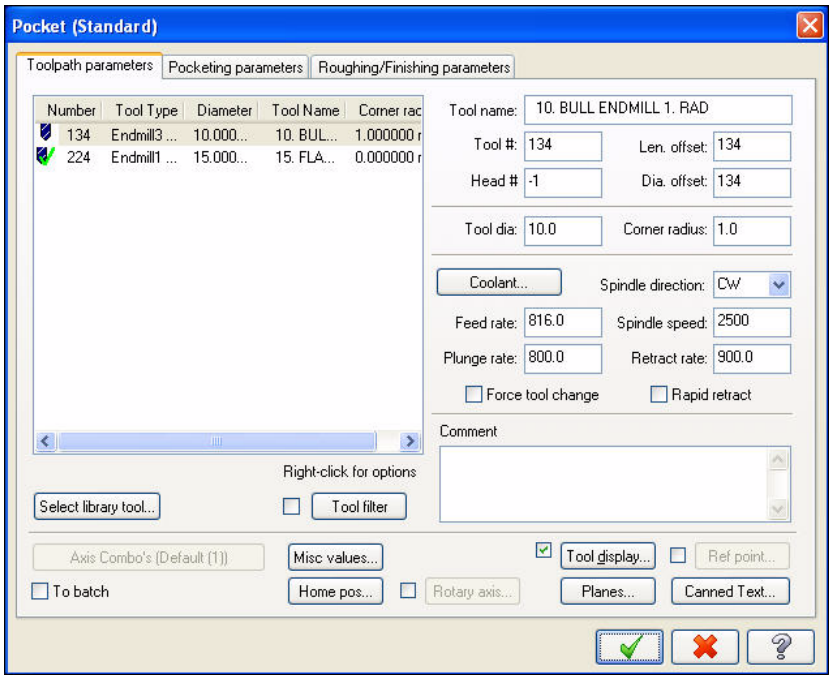


- 3 Click **OK** to chain the part. The Pocket (Standard) dialog box opens.
- 4 Click the **Select library tool** button, select the 10mm diameter bull endmill with the 1mm corner radius, and click **OK**.

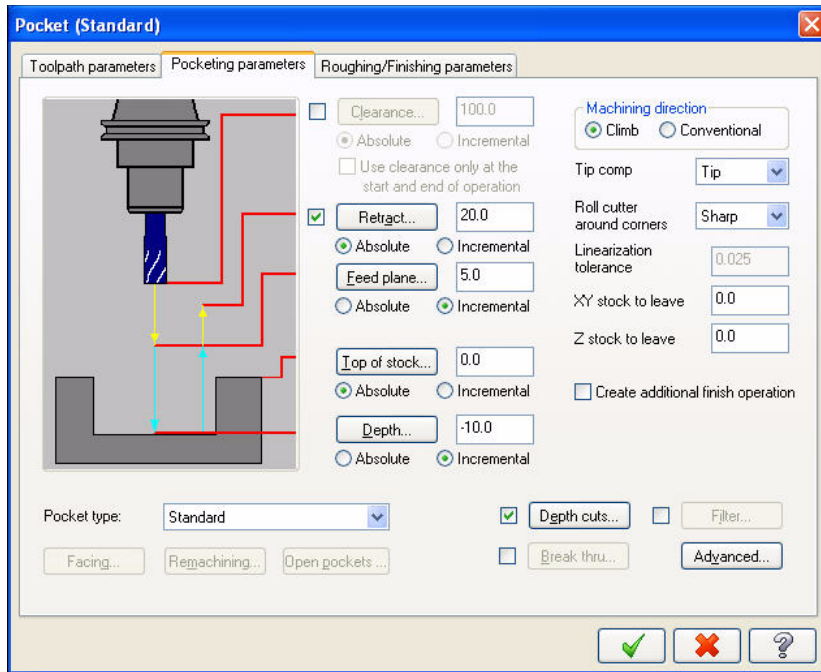
Select library tool...



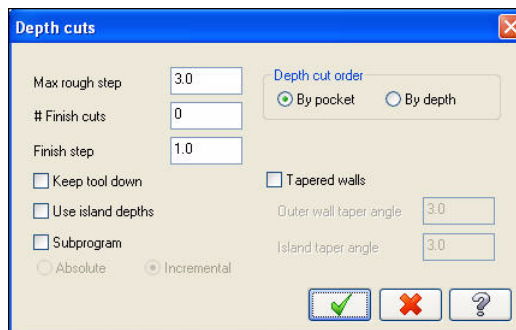
- 5 Enter the toolpath parameter values as shown here. (Do not click **OK** yet.)



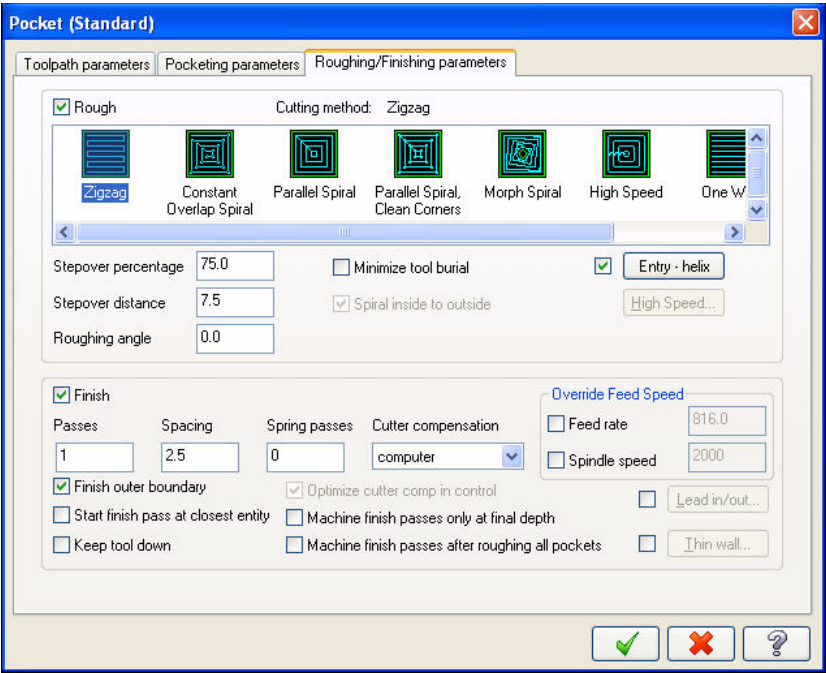
- 6 Click the **Pocketing parameters** tab and enter the values as shown here. (Do not click **OK** yet.)



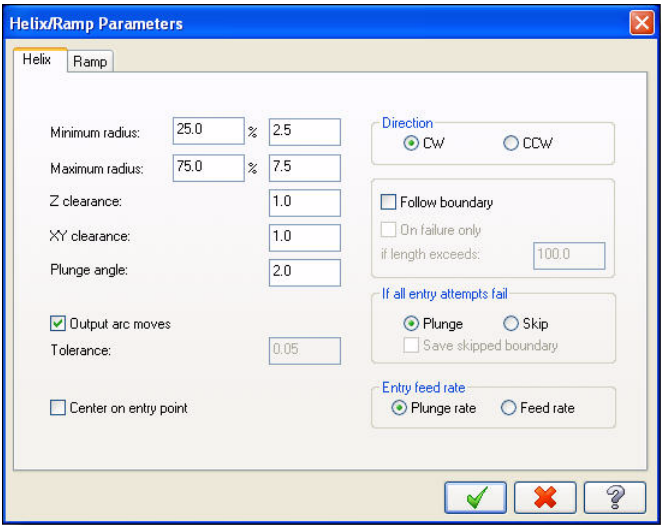
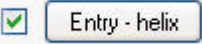
- 7 Click the **Depth cuts** check box and button, enter the depth cuts values as shown here, and click **OK**.



8 Click the **Roughing/Finishing parameters** tab and enter the values as shown here.



9 To add a helix ramp entry move to the pocket roughing operation, click the **Entry-Helix** check box and button, enter the helix values as shown here, and click **OK**.





TIP: An entry helix provides smooth tool motion when entering the material. It will always try to use the maximum possible radius to use more of the side of the endmill and reduce plunging. The direction of the entry helix usually matches the toolpath direction.

10 Click **OK** to generate the toolpath.

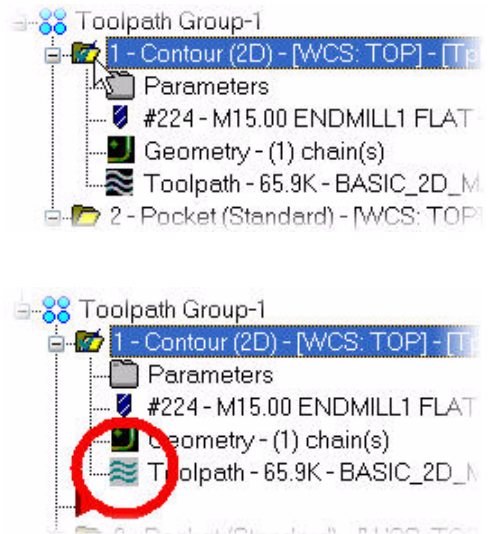
Exercise 2: Viewing Your Toolpath

In this exercise, you will temporarily turn off the display for the contour toolpath so that you can review only the pocket toolpath you just created.

- 1 In the Toolpath Manager, click the **Contour (2D)** toolpath operation to select it. Keep your cursor over the Toolpath Manager, and press **[T]** to toggle off its toolpath display in the graphics window.

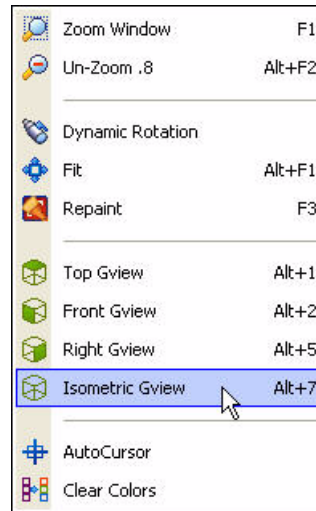
This function toggles the visibility of toolpaths on and off in the graphics window so that you can view only specific toolpaths.

The **Toolpath** icon changes to gray when the toolpath display is toggled off.

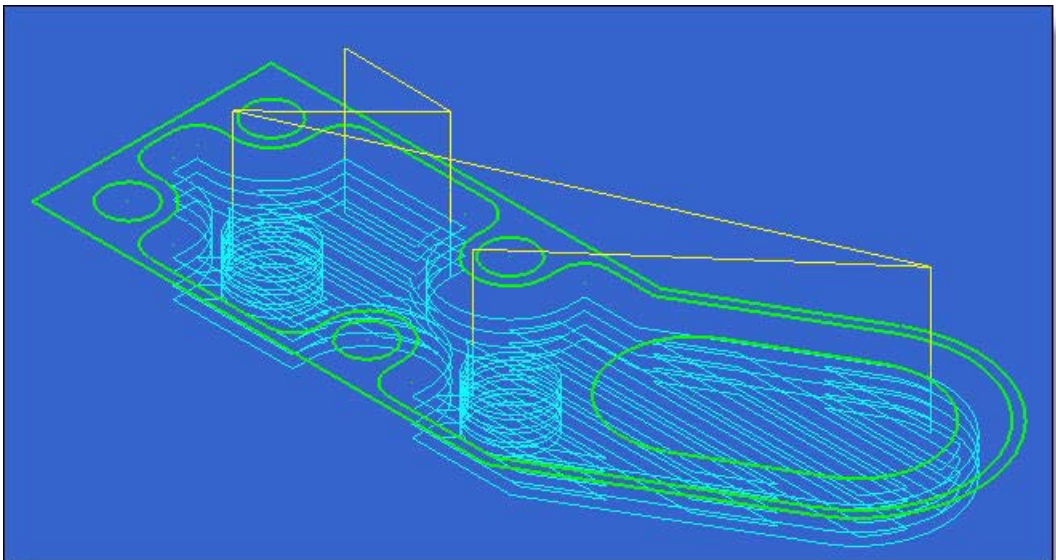


TIP: To toggle the display of all toolpaths in your graphics window, press the **[Alt]** key and the **[T]** key at the same time **[Alt+T]**. For this to work, you must place your cursor in the graphics window, not in the Toolpath Manager. This function does not change the display state of toolpaths (which individual toolpaths will display and which will not as described above). It will only toggle the visibility of toolpaths on and off in the graphics window.

- 2 Right-click in the graphics window to bring up the pop-up menu, and choose **Isometric Gview** to view the part and toolpath in the isometric view.



Your toolpath should look like this.



- 3 Right-click in the graphics window again to bring up the pop-up menu, and choose **Top Gview** to view the part and toolpath in the top view again.
- 4 Save your part.

LESSON 3

Pocketing the Inside Slot Shape

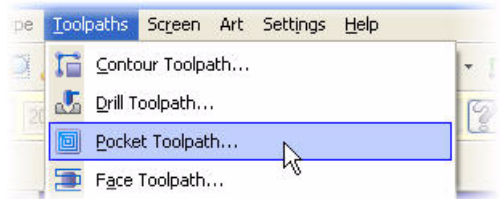
Lesson Goals

- Create a pocket toolpath (including chaining entities, choosing tooling, and setting machining values).
- View a specific toolpath by temporarily turning off the display of selected toolpaths.

Exercise 1: Creating a Pocket Toolpath on the Slot

In this exercise, you create a pocket toolpath with a helix entry on the inside slot shape of the tutorial part.

- 1 Choose **Toolpaths, Pocket Toolpath**. The Chaining dialog box opens.



- 2 If toolpaths are displayed in the graphics window, temporarily turn off the display by selecting all toolpaths in the Toolpath Manager and pressing [T].

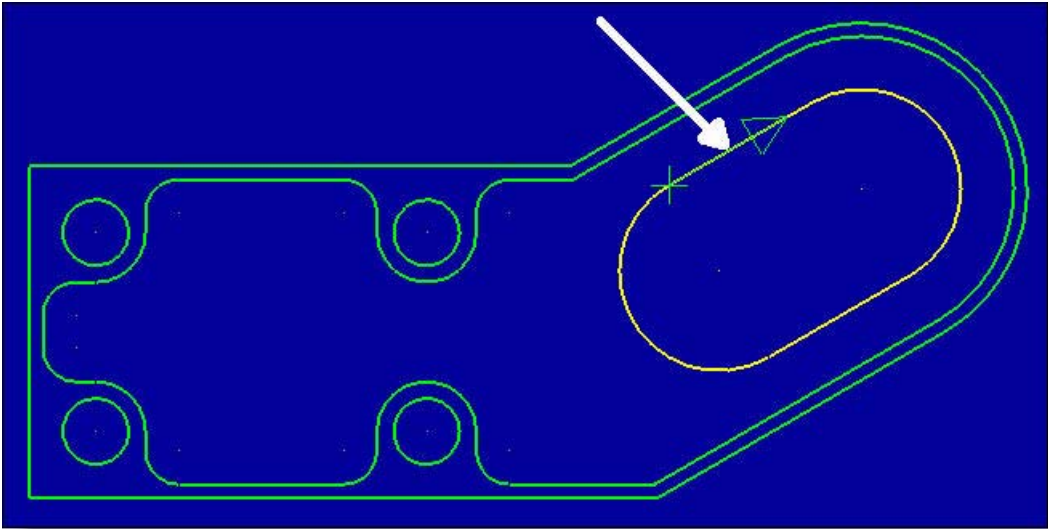
This will make it easier to select entities in the graphics window.



TIP: To select multiple items, hold down the [Ctrl] key and click the text or icon of operations. To deselect an item, hold down the [Ctrl] key and reselect that item or click a different item. If you do not want any items selected, click the insert arrow. You can also use the **Only display selected toolpaths** button (shown here) to toggle the display of selected toolpaths.

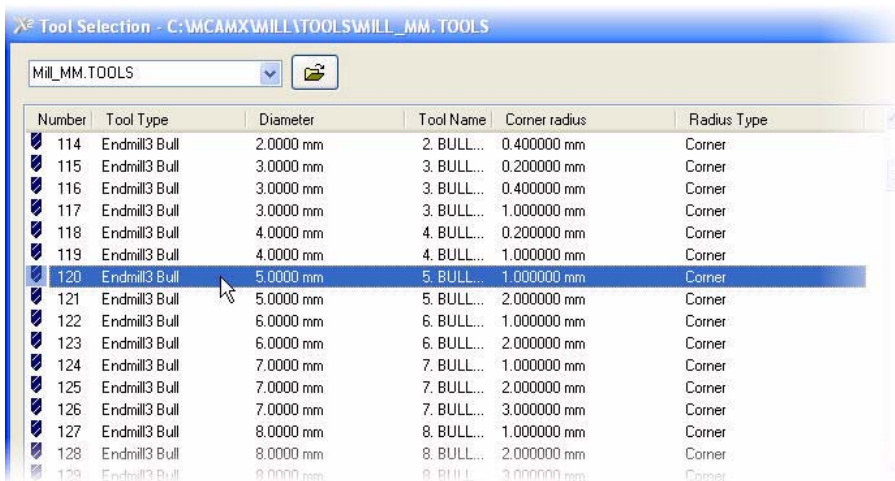


- 3 Click the inside slot shape of the part to chain it. The chaining arrow should point clockwise.

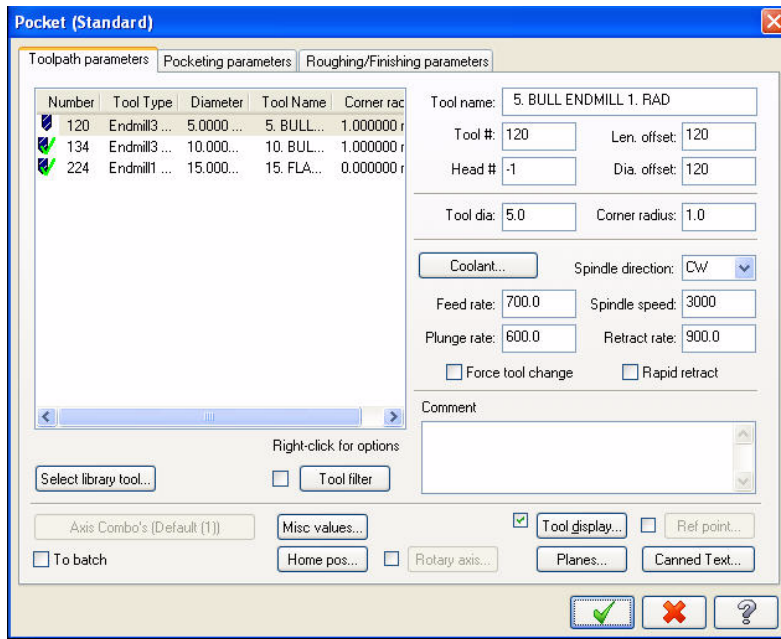


- 4 Click **OK** to chain the part. The Pocket (Standard) dialog box opens.
- 5 Click the **Select library tool** button, select the 5mm diameter bull endmill with the 1mm corner radius, and click **OK**.

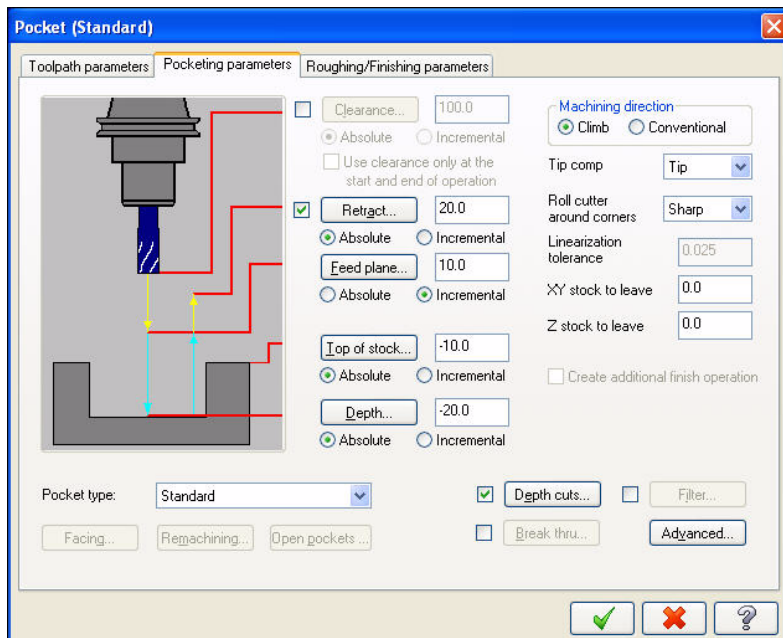
Select library tool...



6 Enter the values as shown here.



7 Click the **Pocketing parameters** tab and enter the values as shown here.





TIP: Many of the Mastercam dialog boxes retain their previous settings, saving you from having to reenter data, reselect function buttons, or reselect options in a drop-down list. The settings stay in their “last used” state for the remainder of the Mastercam session or until you change them.

- 8 Click the **Depth cuts** button, enter the depth cuts values as shown here, and click **OK**.



Depth cuts

Max rough step: 3.0
 # Finish cuts: 0
 Finish step: 1.0

Depth cut order:
☒ By pocket ☐ By depth

☐ Keep tool down ☐ Tapered walls
☐ Use island depths Outer wall taper angle: 3.0
☐ Subprogram Island taper angle: 3.0
☐ Absolute ☒ Incremental

Buttons: [OK] [Cancel] [Help]

- 9 Click the **Roughing/Finishing parameters** tab and enter the values as shown here.

Pocket (Standard)

Toolpath parameters | Pocketing parameters | **Roughing/Finishing parameters**

☒ **Rough** Cutting method: Parallel Spiral

Icons: Zigzag, Constant Overlap Spiral, **Parallel Spiral**, Parallel Spiral, Clean Corners, Morph Spiral, High Speed, One W

Stepover percentage: 75.0 ☐ Minimize tool burial ☒ Entry - helix
 Stepover distance: 3.75 ☒ Spiral inside to outside High Speed...
 Roughing angle: 0.0

☒ **Finish**

Passes: 1 Spacing: 2.5 Spring passes: 0 Cutter compensation: computer

☒ Finish outer boundary ☒ Optimize cutter comp in control ☐ Lead in/out...
☐ Start finish pass at closest entity ☐ Machine finish passes only at final depth ☐ Thin wall...
☐ Keep tool down ☐ Machine finish passes after roughing all pockets

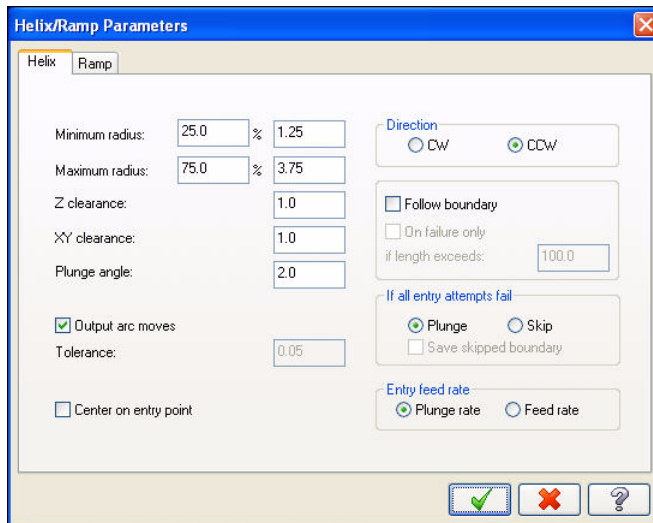
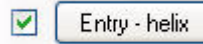
Override Feed Speed:
☐ Feed rate: 816.0
☐ Spindle speed: 2000

Buttons: [OK] [Cancel] [Help]



TIP: The **Parallel Spiral** cutting method roughs the pocket in a spiral that offsets the outer boundary by the stepover amount each time. This roughing toolpath does not add small cleanout moves in the corners of the pocket to remove more stock. This option does not guarantee cleanout, but for pocketing this type of slot shape, it will provide a clean cut and save time by eliminating the extra cleanout moves.

- 10** To add a helix ramp entry move to the pocket roughing operation, click the **Entry-Helix** button, enter the helix values as shown here, and click **OK**.



- 11** Click **OK** to generate the toolpath.

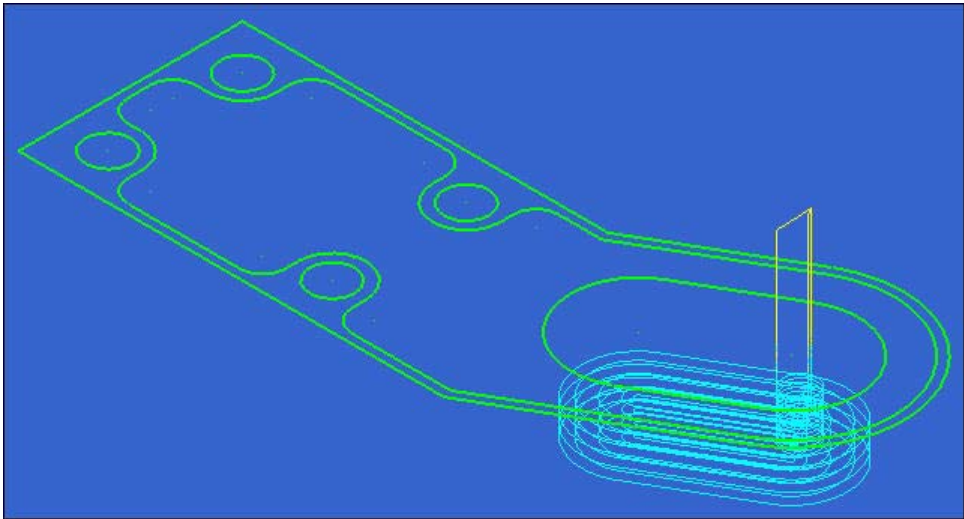
- 12** In the Toolpath Manager, make sure the current pocket toolpath is the only toolpath set to display in the graphics window.



TIP: Keep your cursor over the Toolpath Manager, and press **[T]** to toggle off selected toolpath displays in the graphics window.

- 13** Right-click in the graphics window to bring up the pop-up menu, and choose **Isometric Gview** to view the part and toolpath in the isometric view.

Your toolpath should look like this.



- 14** Right-click in the graphics window again to bring up the pop-up menu, and choose **Top Gview**.
- 15** In the Toolpath Manager, turn off the toolpath display for all toolpaths.
- 16** Save your part.