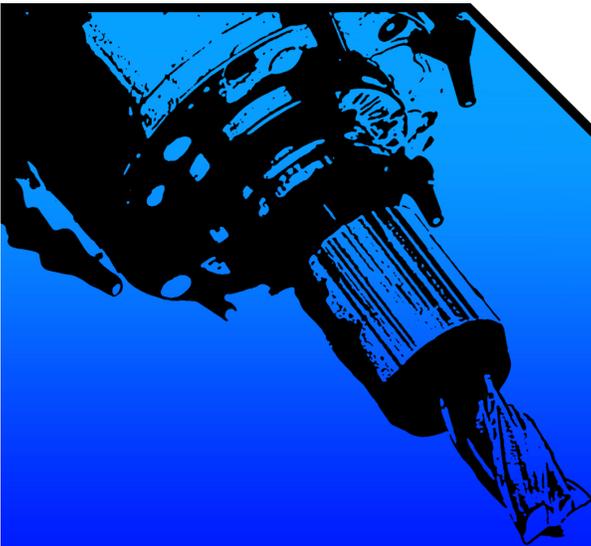


Getting Started MANUAL

N 6 9 6 X 4 . 4 2 3 8 Y 2 . 6 0 6 6
N 6 9 7 X 4 . 4 1 9 8 Y 2 . 6 0 5 3
N 6 9 8 X 4 . 4 0 6 8 Y 2 . 6 0 1 2
N 6 9 9 X 4 . 4 0 3 2 Y 2 . 6 0 0 1
N 7 0 0 X 4 . 3 8 9 2 Y 2 . 5 9 5 7
N 7 0 1 X 4 . 3 8 6 3 Y 2 . 5 9 4 7
N 7 0 2 X 4 . 3 7 5 3 Y 2 . 5 9 1 3
N 7 0 3 X 4 . 3 7 3 2 Y 2 . 5 9 0 6
N 7 0 4 X 4 . 3 6 5 3 Y 2 . 5 8 8 1
N 7 0 5 X 4 . 3 6 4 Y 2 . 5 8 7 7
N 7 0 6 X 4 . 3 6 0 1 Y 2 . 5 8 6 5
N 7 0 7 X 4 . 3 5 9 2 Y 2 . 5 8 6 2
N 7 0 8 X 4 . 3 5 6 8 Y 2 . 5 8 5 4
N 7 0 9 X 4 . 3 3 4 8 Y 2 . 5 7 8 5
N 7 1 0 X 3 . 1 5 6 5 Y 2 . 2 0 7
N 7 1 1 X 3 . 1 4 3 2 Y 2 . 2 0 2 8
N 7 1 2 X 3 . 1 4 1 9 Y 2 . 2 0 2 4
N 7 1 3 X 3 . 1 4 0 5 Y 2 . 2 0 1 9
N 7 1 4 X 3 . 1 3 9 5 Y 2 . 2 0 1 6
N 7 1 5 X 3 . 1 3 8 6 Y 2 . 2 0 1 8
N 7 1 6 X 3 . 1 3 7 8 Y 2 . 2 0 1 2
N 7 1 7 X 3 . 1 3 6 9 Y 2 . 2 0 0 6
N 7 1 8 X 3 . 1 3 6 0 Y 2 . 2 0 0 1
N 7 1 9 X 3 . 1 3 5 1 Y 2 . 1 9 9 5
N 7 2 0 X 3 . 1 1 1 4 Y 2 . 1 9 2 8

BobCAD-CAM
V23



Mill | Lathe

Getting Started Manual – Version 23 Mill & Lathe
January 2010

Copyright © 2010 by BobCAD-CAM Inc., All rights reserved. No part of this work may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying and recording, or by any information storage or retrieval system without prior written permission of BobCAD-CAM Inc. unless such copying is expressly permitted by federal copyright law. This document may be downloaded from www.bobcad.com and printed for personal use. Address inquires to Documentation Department, BobCAD-CAM, Inc., 28200 US HWY 19 N., Suite E, Clearwater, Florida 33761

AutoDesk™, ACIS™, Rhinoceros™, ParaSolid™, Predator CNC Editor™, Predator Virtual CNC™, and SolidWorks™ are all trademarked by their respective companies.

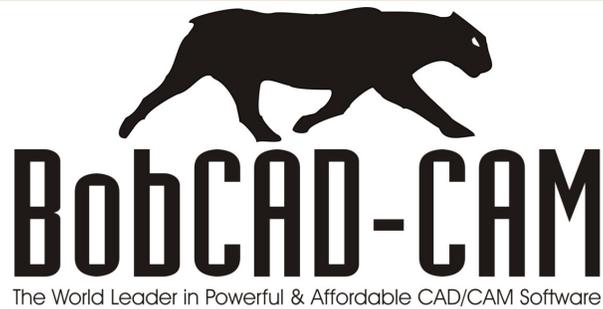


TABLE OF CONTENTS

About this Manual	5
Chapter 1. Installation & Registration	7
1.1 Introduction	8
System Requirements	8
How to Check the System Requirements	9
Updating Your Computer	10
Installation	15
Registering Your Software	16
Hardlock Security Device	16
Online Registration	17
Register By Phone	18
Register By Fax	18
Vista Permissions	19
Re-Key	21
Re-Key Procedure	22
Machine Post Processor Information	23
Chapter 2. Version 23 Mill	27
2.1 Mill Software Setup	28
Setup for Plasma, Laser & WaterJet	58
2.2 Mill Lesson	60
Drawing the Part	60
Machining the Part	65
3D BobART Example	73
Chapter 3. Version 23 Lathe	85
3.1 Lathe Software Setup	86
3.2 Lathe Lesson	99
Drawing the Part	99
Machining the Part	102

Chapter 4. Posting & Communications	107
4.1 Posting & Communications	108
Saving Posted Programs	109
File Formats and Extensions	110
Establishing Communications	110
Communication Settings	111
Communication Hardware	114
Transferring Files Via RS 232	115

About this Manual

This Manual was created with the purpose of helping you get started with your new BobCAD-CAM software. The following Version 23 products are included in this guide:

- **Mill (includes Routers)**
- **Lathe**

The first chapter of this document is dedicated to providing you valuable information and instruction on the installation and registration aspects of your software, which includes all add-on modules. The following chapters are dedicated to helping you set up your new software for both Milling and Turning. There are added lessons which have been added to help you get started for either product.

BobCAD-CAM offers a 2 Book Advanced Training set for CAD Design and Milling and Training Professor Video Series for each major product module. These important training products can be purchased by contacting us directly.

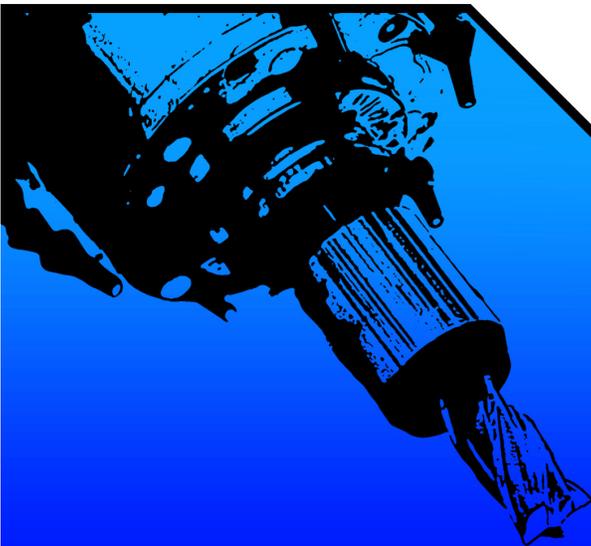
Toll Free: 877-262-2231 x123
International: 727-442-3554 x123
Web: www.bobcad.com

Thank you for choosing BobCAD-CAM as your manufacturing solution!

Chapter 1

Installation & Registration

N 6 9 6 X 4 . 4 2 3 8 Y 2 . 6 0 6 6
N 6 9 7 X 4 . 4 1 9 8 Y 2 . 6 0 5 3
N 6 9 8 X 4 . 4 0 6 8 Y 2 . 6 0 1 2
N 6 9 9 X 4 . 4 0 3 2 Y 2 . 6 0 0 1
N 7 0 0 X 4 . 3 8 9 2 Y 2 . 5 9 5 7
N 7 0 1 X 4 . 3 8 6 3 Y 2 . 5 9 4 7
N 7 0 2 X 4 . 3 7 5 3 Y 2 . 5 9 1 3
N 7 0 3 X 4 . 3 7 3 2 Y 2 . 5 9 0 6
N 7 0 4 X 4 . 3 6 5 3 Y 2 . 5 8 8 1
N 7 0 5 X 4 . 3 6 4 Y 2 . 5 8 7 7
N 7 0 6 X 4 . 3 6 0 1 Y 2 . 5 8 6 5
N 7 0 7 X 4 . 3 5 9 2 Y 2 . 5 8 6 2
N 7 0 8 X 4 . 3 5 6 8 Y 2 . 5 8 5 4
N 7 0 9 X 4 . 3 3 4 8 Y 2 . 5 7 8 5
N 7 1 0 X 3 . 1 5 6 5 Y 2 . 2 0 7
N 7 1 1 X 3 . 1 4 3 2 Y 2 . 2 0 2 8
N 7 1 2 X 3 . 1 4 1 9 Y 2 . 2 0 2 4
N 7 1 3 X 3 . 1 4 0 5 Y 2 . 2 0 1 9
N 7 1 4 X 3 . 1 3 9 5 Y 2 . 2 0 1 6
N 7 1 5 X 3 . 1 3 8 6 Y 2 . 1 9 9 8
N 7 1 6 X 3 . 1 3 7 6 Y 2 . 1 9 9 2
N 7 1 7 X 3 . 1 3 6 7 Y 2 . 1 9 7 1
N 7 1 8 X 3 . 1 3 5 8 Y 2 . 1 9 6 4
N 7 1 9 X 3 . 1 3 4 9 Y 2 . 1 9 5 6
N 7 2 0 X 3 . 1 1 1 4 Y 2 . 1 9 2 8



1.1 Introduction

Congratulations on choosing BobCAD-CAM for your manufacturing needs. This guide is designed to help you through the software installation process, walk you through registering your new product and give you some basics on how to get started in using the software.

We are always interested in your feedback regarding features and functionality that you would like to see developed into this CAD/CAM product so that we can continue to remain the world leader in powerful and affordable CAD/CAM systems. If you have any difficulties in using this guide, please feel free to contact us directly at one of the numbers provided here. Thank you again for choosing BobCAD-CAM!

BobCAD-CAM, Inc.
28200 U.S. Highway 19, N.
Clearwater, Florida 33761
Toll Free: 877-262-2231
Phone: 727-442-3554
Technical Support: 727-489-0003
www.bobcad.com

System Requirements

The following are system requirements for using your new software on **Windows XP or Vista**.

Optimum System Requirements:

2.0 GHz Processor
2 GB RAM
2 GB Available Hard Disk (Used for Paging File)
512 MB Graphics Adapter that supports OpenGL 1.1
For Windows Vista stations we suggest an ATI or NVIDIA graphics cards.

Minimum System Requirements:

2 GHz Processor 1 GB RAM
2 GB Available Hard Disk (used for Paging File)
256 MB Graphics Adapter that supports OpenGL 1.1
For Windows Vista stations we suggest an ATI or NVIDIA graphics cards.

NOTE: The minimum configuration will allow the system to run, but there will be some delay in the execution of commands. The BobCAD-CAM V23 system is a solids and surface modeling system. If you are using a computer that has less RAM than the minimum requirements listed above, you may experience delays in executing functions that relate to rendering models, generating toolpath and G-Code programs. By upgrading your computer to the recommended requirements above, you will be able to execute those functions more efficiently.

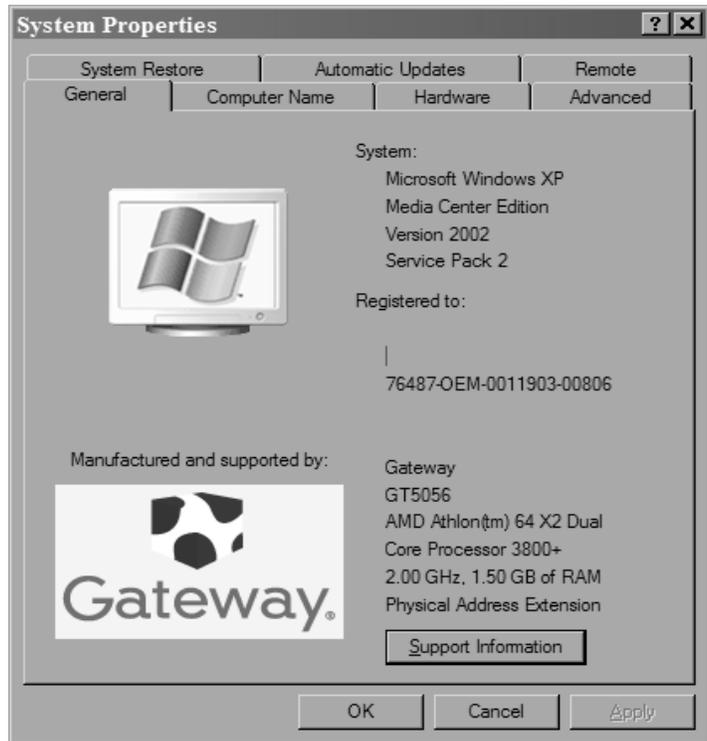
How to Check The System Requirements

To check and see if your computer meets the minimum system requirements you can check by completing the following steps.

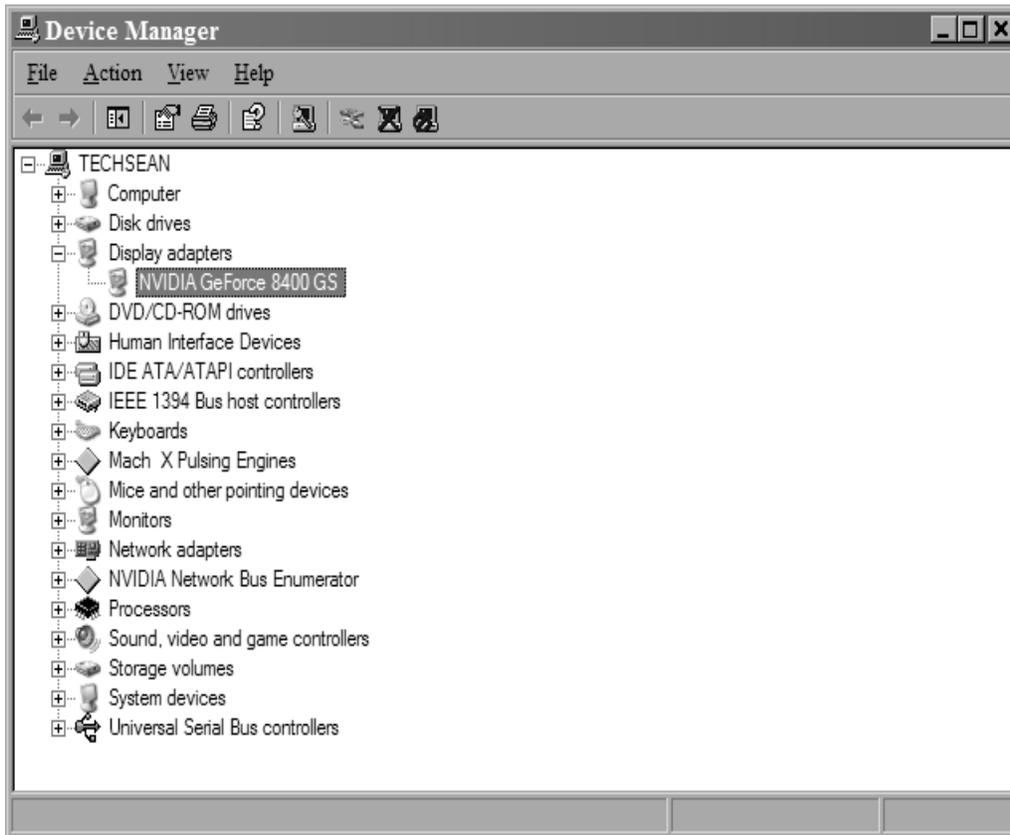
1. Click on the windows Start Menu.



2. Right click on My Computer (XP) or Computer (Vista & 7) and click on Properties.
3. This will open the general tab you can gather most of the system information here, the display adapter can be checked in the next step.



4. Click the Hardware Tab and then click on “Device Manager.”



5. Click the plus symbol next to the Display adapter. This will display the brand and model of graphics card. You will need to find the specs for the card on the manufacturer's website.

Updating Your Computer

If you feel that your computer is maintained, and does not require these steps you may skip this process. *it is highly recommended that you follow this process if you encounter any software performance problems, if the software will not work at all or if you get any errors during installation.

Updating and maintaining your computer is necessary. If your computer's software is not up to date you may experience problems, such as poor performance or even system instability. Your computer will need to have the following updates loaded. The latest available graphics card driver.

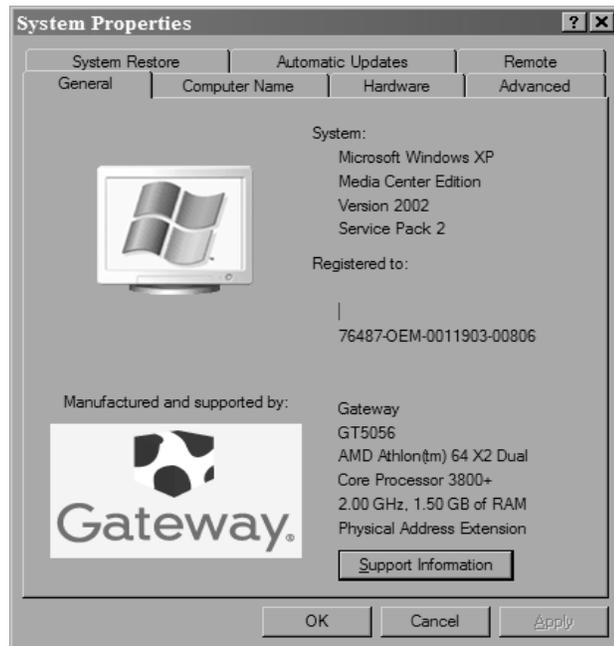
To update Video Card Driver, you must first know what Video Card you have, and if it is up to date.

Follow these steps:

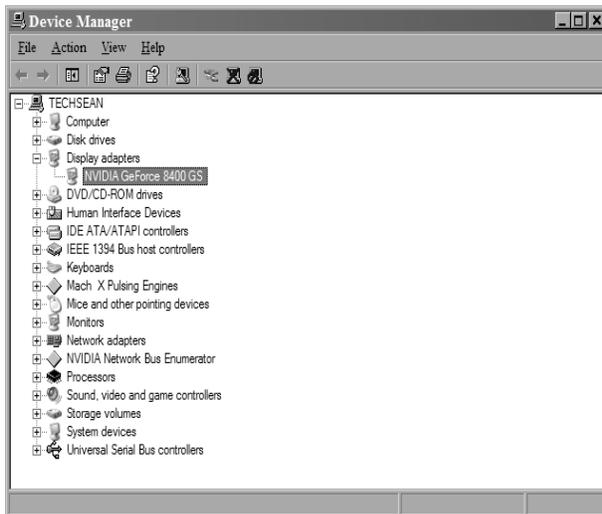
1. Click on the windows Start Menu.



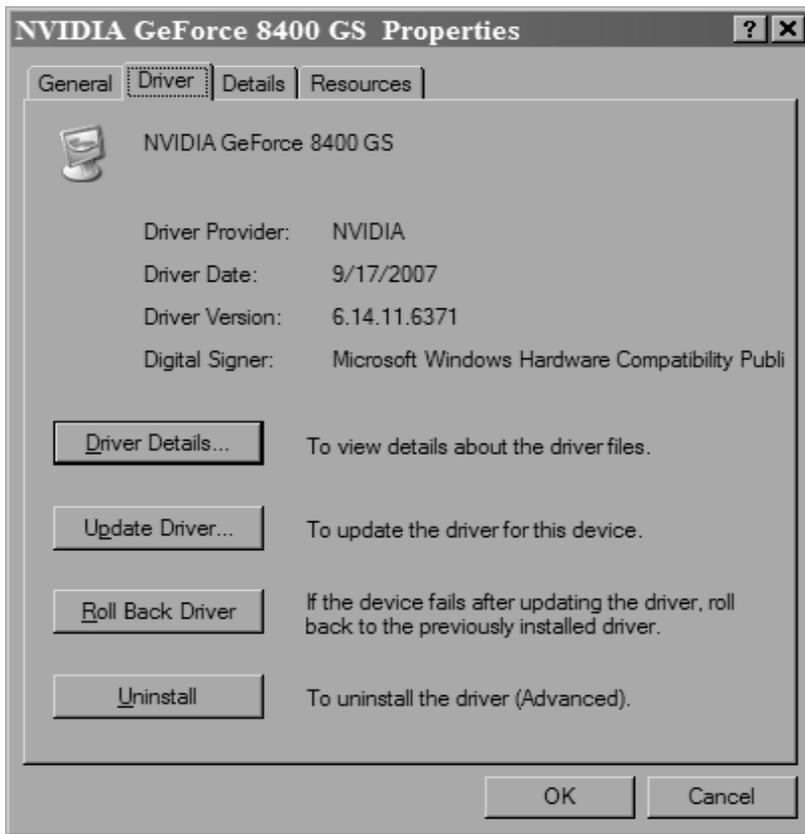
2. Right click on My Computer (XP) or Computer (Vista & 7) and click on Properties.



3. Click the Hardware Tab and then click on “Device Manager.”



- Right click on the display adapter name, then left click properties and click on the Driver Tab. You will need the graphics card name, driver date and version.



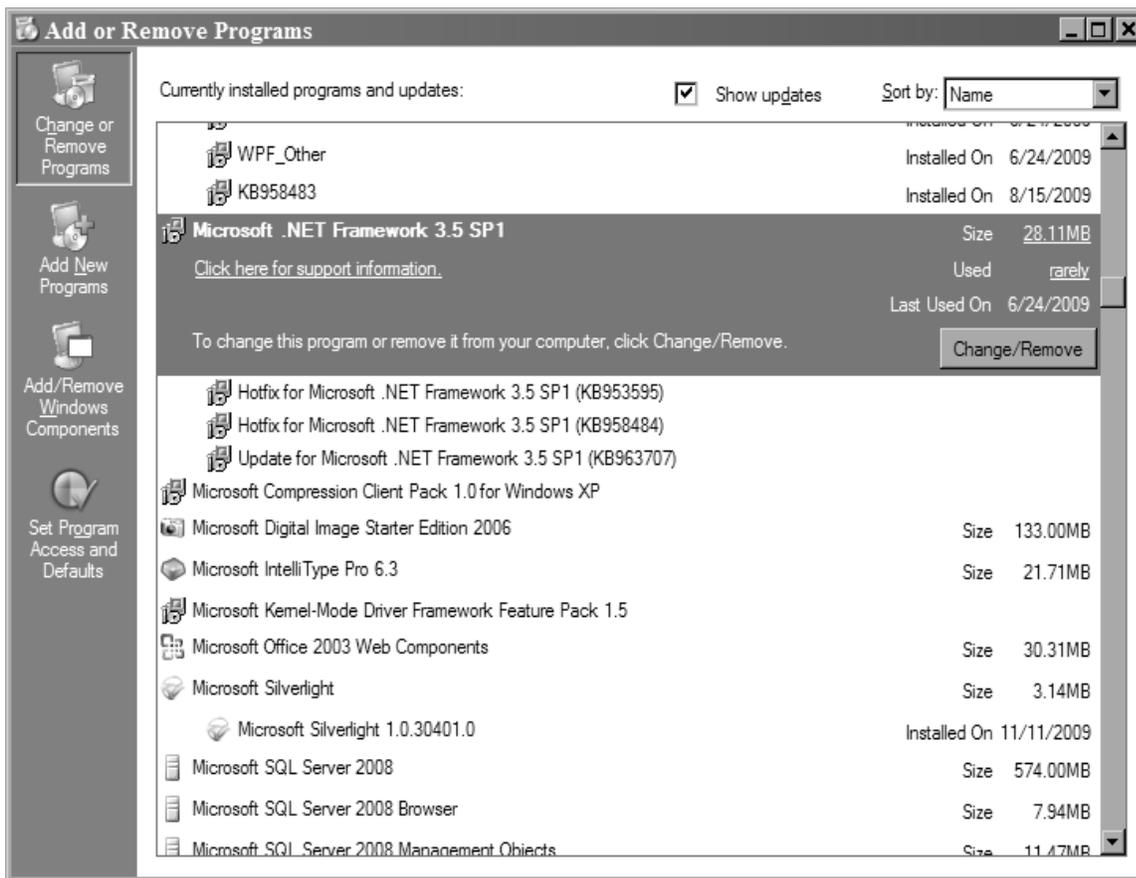
- Locate the manufactures website and follow their instructions for obtaining an update if available. This can usually can be downloaded directly from their website.
**The most Common manufacturer websites are Nvidia.com and ATI.com. For integrated video cards (not recommended), you should be able to get the latest driver version on the Hardware portion of the Windows Update Page or the computer manufactures website.
-

The latest Microsoft .Net Framework update

First check what version is installed, The latest as of 12/14/09 is .Net Framework Version 3.5 SP1.

To check the version:

1. Click on the windows start button.
2. Then double click on control panel.
3. Double click Add/Remove Programs.
4. Check the box that says show updates.
5. Find the latest version number of the .Net Frameworks.

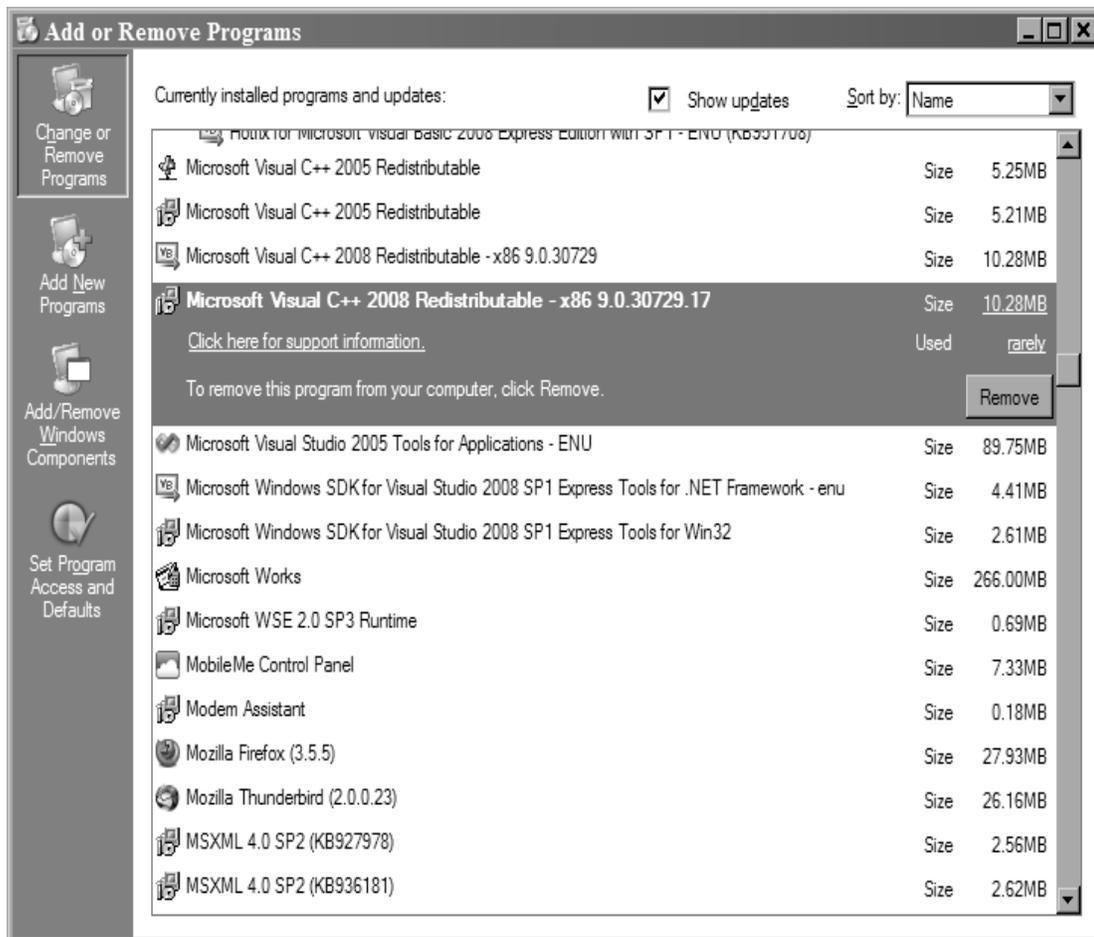


NOTE: If you regularly update your computer, the latest version should already be installed.

BobCAD keeps a copy of the required Microsoft .Net Framework available for download on the web. The file can be located at www.bobcad.com/updates. Click on the .Net framework link, save the file to your computer and then double click the file to run the installation.

The Latest Microsoft Visual C++ Redistributable pack

1. First check what version is installed, The latest as of 12/14/09 is Microsoft Visual C++ 2008. To check the version do the following:
 - a. Click on the windows start button.
 - b. Then double click on control panel.
 - c. Double click Add/Remove Programs.
 - d. Check the box that says show updates.
 - e. Find the latest version number of the Microsoft Visual C++ Redistributable .



NOTE: If you regularly update your computer, the latest version should already be installed.

BobCAD keeps a copy of the required Microsoft Visual C++ Redistributable available for download on the web the file can be located at www.bobcad.com/updates. Click on the Visual C++ Redistributable link and save the file to your computer. Then double click the file and run the installation.

Installation

Follow these simple loading instructions to install your new software.

1. Insert the BobCAD-CAM V23 installation CD into the CD Drive of the computer.
2. The installation wizard will automatically start the installation process. Follow the instructions given in the installation wizard to complete the installation.



3. Upon successful installation a dialog will appear giving you an option to preview the Getting Started Videos included with the installation.
4. If you wish to view the videos, check the box labeled "Yes" and click Finish. Otherwise, click the Finish button.
5. Now that the installation is complete, the Installation CD may be removed from the CD Drive of the computer. Now continue with the next step in opening your software.

NOTE: The full installation of BobCAD-CAM V23 includes a 5 day fully-functional period. This 5 day period will begin when the software is initially opened on the computer. Before your 5 days is up, please register your software so that you do not experience any delays.

Registering Your Software

There are 3 different methods of BobCAD-CAM software licensing. They are:

1. **Hardlock Security Device:** If you purchased the software with a hardlock security device (also known as a key), you will not need to contact BobCAD-CAM. You will want to review the section below called, “Hardlock Security Device” for step by step instructions on how to start up your new software. You do not need to insert the hardlock prior to installation. However before using the software you will want to insert the hardlock into your USB drive.
2. **Online/Internet Registration:** This means that if your computer is connected to the internet, you can register and un-lock your software online rather than call or fax BobCAD-CAM. For more information you will want to refer to the section called, “Online Registration.”
3. **Phone Registration:** This method means that you can phone us directly during business hours and register your software. In this case we will provide you with a pass-code right over the phone that you will use for un-locking your product. If this is how you will register your product, please go to that section of this document and follow the step by step instruction. If this is the method you will be using, you may also fax in your registration form to BobCAD-CAM after having filled it out completely.

The following sections will help you register your software.

NOTE: BobCAD-CAM V23 will run in a full functional mode for a period of 5 days after it is first launched (opened). After the 5 day period has lapsed, the system will revert to **demo mode** until registered.

Demo mode has the following restrictions:

- Machine communication (DNC< etc.) will not run.
- Only the first 20 lines of any program will be posted.
- Saving to external file formats (formats other than .bbcd) will be disabled.
- .bbcd files created in Demo mode can not be loaded directly into BobCAD-CAM in licensed mode. (Files created in “Licensed” mode can be viewed in Demo mode, but if the file is saved while in Demo mode it can not be re-opened in Licensed mode.)
- 3D toolpaths are not able to be computed while in Demo mode.

Hardlock Security Device

Hardlock Security Device Licensing: If you have purchased your software product with a hardlock security device as the method of software activation and licensing, you will want to read this section.

**As a special note, you may also want to review the section titled, “What is a Re-Key” in this manual.*

To use the software with the hardlock security device method of licensing, follow these instructions:

1. Close the BobCAD-CAM V23 application if it is open.

2. Insert the hardlock into an available USB port on the computer.
3. Open the BobCAD-CAM V23 Software. The software will automatically read the hardlock and authorize the modules that are licensed for the product .

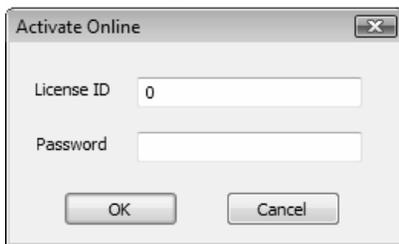
To verify which modules are licensed for a particular seat, click on Help in the main menu and then click on License. A dialog box will appear. In this dialog a check mark will be displayed in the boxes for the module(s) that have been purchased.

Online Registration

The online registration method requires that you have access to the original invoice as well as an active internet connection on the computer that is being registered. Follow the steps below to register your software online.

1. Open the BobCAD-CAM V23 Software.
2. Click on the Help menu and then click on “Activate Online” and the Online Activation Dialog will appear (see diag. A-2).

Diag. A-2



3. Input the License ID and the Password provided on your original invoice (see Diag. A-3).

Diag. A-3

		P.O. NO.
PRODUCT ID	PRODUCT NAME	
012310111	Bob CAD-CAM V	
022310111	Bob CAD-CAM V	
032310111	Bob CAD-CAM V	
License Information		
License Num	License Password	
13356	F6PKE2EYX2	

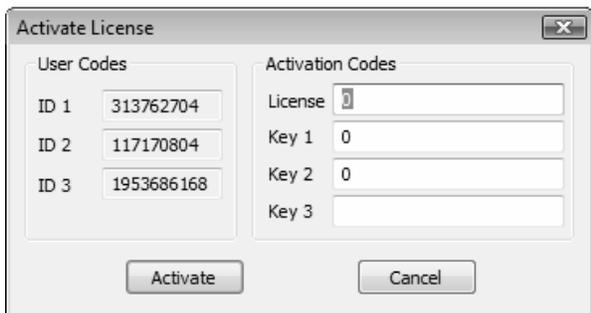
4. Then click OK. The software will then retrieve the necessary codes to authorize the software. If an error occurs during this procedure, please contact BobCAD-CAM directly at 877-262-2231 x110 for U.S. customers and 727-442-3554 x110 for International customers.

Phone Registration

You can register your BobCAD-CAM software by phone. Follow these steps:

1. Open the BobCAD-CAM V23 software.
2. Click on the Help menu and then click on “Activate License” and the Activate License dialog will appear. (see Diag. B-2).

Diag. B-2



3. Call the BobCAD-CAM Registration department (877-262-2231 for U.S. customers and 727-442-3554 for International customers).
 4. The Registration department will ask for the numbers given in the User Codes fields.
 5. After providing the User Codes that you are asked for, you will be given numbers to input into the Activation fields.
 6. Click Activate to complete the authorization of the software.
-

Fax Registration

You can register your BobCAD-CAM software by fax. Follow these steps:

1. Open the BobCAD-CAM V23 software.
2. Click on the Help menu and then click on “Activate License” and the Activate License dialog will appear. (see Diag. C-2)

Diag. C-2



3. Write the User Codes down onto the Fax Registration form. Do not close your BobCAD-CAM software. If you close the software for any reason, the numbers will change and you will have to start this process over again.
-

4. Fax the form to BobCAD-CAM Registration at 727-442-1773.
 5. Leave the BobCAD-CAM V23 software open until you receive the return fax. Then enter the codes into the correct fields and click “Activate.”
-

Vista Operating System and Setting Permissions

If you have Vista it is highly recommended to use Vista Home Premium or higher, for performance reasons.

Before upgrading any existing Microsoft Windows 98/2000/XP machine to Microsoft Windows Vista, it is highly recommended to consult a computer repair technician.

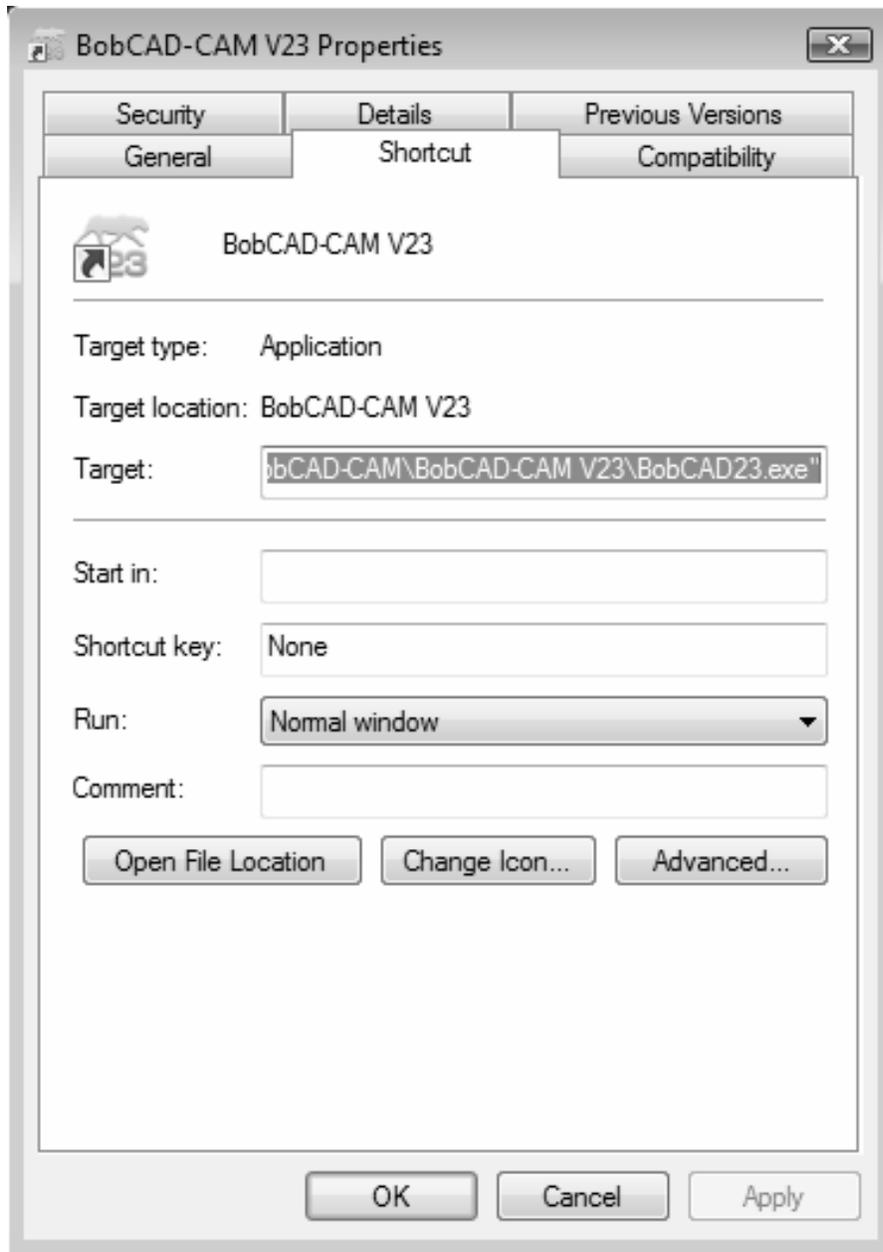
To ensure proper operation when using the BobCAD-CAM software on a Vista computer under a normal user account, modifying the user permissions is necessary. The steps for modifying the required permissions can be found on the BobCAD-CAM Support website at <http://www.bobcadsupport.com/> under the *Vista & Re-Key Information* heading.

Vista Permissions

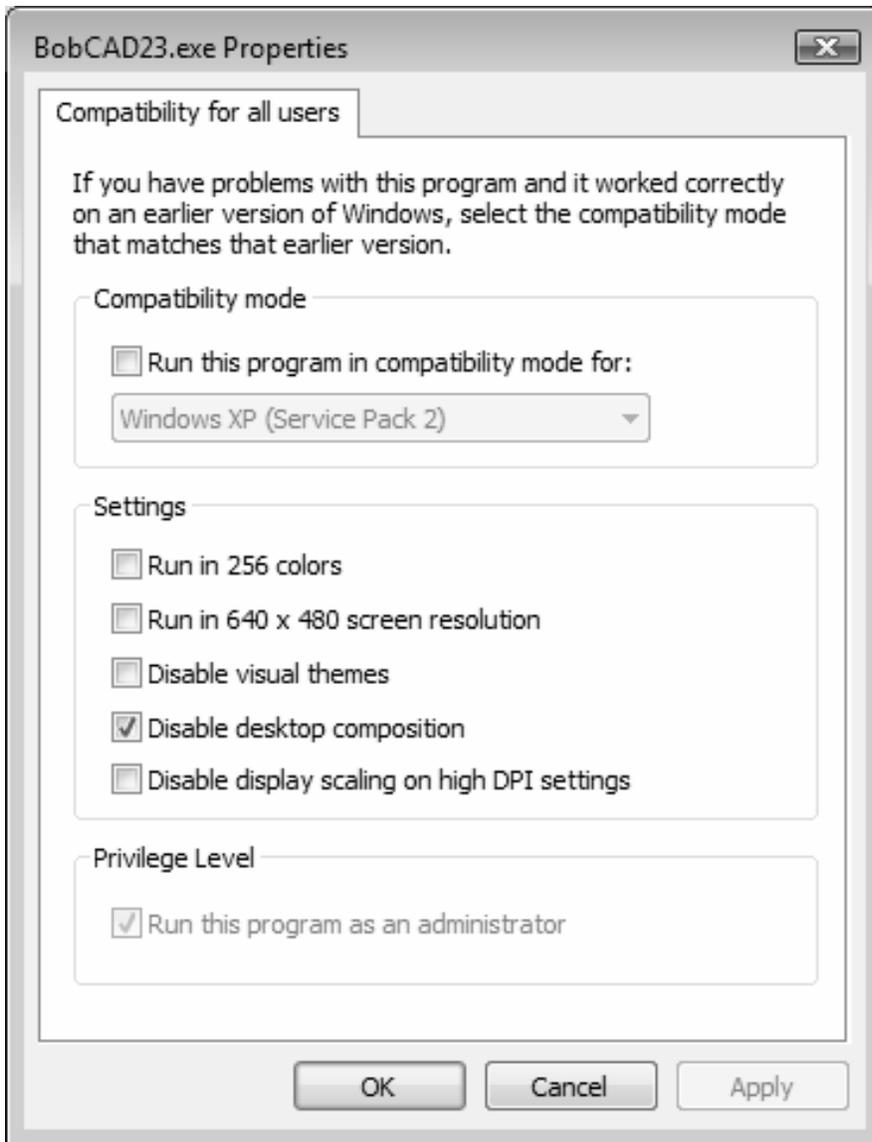
If you have a **Vista** Operating System and you purchased the software with a hardware lock and experience a message that you are running in “**Demo Mode**” when you try to launch the program you will need to set the Vista permissions in order to correct this. This procedure is how to fix the problem.

If you are not the administrator on the computer that your BobCAD-CAM V23 software is installed on, you will need to have the system administrator log into the computer and perform these steps for you.

1. Close all open applications as a precaution.
2. Open the application **Compatibility-Properties** dialog.
 - a. Right Click on the BobCAD-CAM V23 icon located on the Desktop.
 - b. Left click on the Properties option in the pop-up menu that appears. The BobCAD-CAM V23 Properties dialog will appear.



- c. Then left click on the **Compatibility** tab at the top of the dialog. If the **Compatibility** tab is not present, do the following:
- d. Click on the **Open File Location** button to open the BobCAD-CAM V23 directory.
- e. Right click on the BobCADV23.exe file and click on **Properties** in the pop-up menu. You should now be able to see the **Compatibility** tab.



3. Click on the **Show settings for all users** button.
 4. Check the **Disable desktop composition** option and the **Run this program as an administrator** option.
 5. Click on the OK buttons in the dialogs. This will apply the changes and close the dialogs.
-

What is a Re-Key

If you have BobCAD-CAM Version 23 with a hardlock and you purchase an add-on software product at a later date, you will need to perform what is known as a Re-Key operation. The reason for this is that the software tag and password that is a part of the licensing changes when you add a new product module. An example of this would be the addition of the BobART or Lathe Module to the Milling system. **There are 2 ways that a Re-Key can be accomplished, manually or by using the internet.**

The Re-Key Procedure

Therefore, once again when purchasing new modules or new software from BobCAD-CAM Inc., it is required that an update be applied to your hardlock in order to unlock the new modules. Choose which Re-Key method you would like to use and follow each step to success.

A. Online Re-Key Update

1. Make sure that the computer being used to update the license has an active internet connection. If it does not, either change computers or go to method B for updating the hardlock manually.
2. Insert your hardlock into a working USB port on your computer. Make sure that the hardlock that you insert is the hardlock that you have purchased the upgrade for.
3. Launch the BobCAD-CAM software.
4. Click the Help menu and then click Upgrade License.
5. Click the Update Online button.

B. Manual Re-Key Update

1. Launch the BobCAD-CAM software.
2. Click the Help menu and then click Upgrade License. A dialog will appear that shows a Serial Number, Tag and Password.
3. Copy the Serial Number, Tag and Password to a piece of paper, or to a text file that can be transferred to another computer.
4. Next compose an email to keys@bobcad.com. In this email include your name and phone number, so that you can be contacted if necessary, and the Serial Number, Tag and Password that you copied down from the previous step.
5. An email response will be sent back to you that is not time sensitive, so you may continue working on other projects or close the software.
6. When the response email is sent back to you it will include an attached file with the .bcllic extension. Save this file to a floppy disk or flash drive (also called a memory stick). Now insert the flash drive or floppy disk on the computer that has BobCAD-CAM loaded on it.
7. Launch the BobCAD-CAM software.
8. Click the Help menu and then click Upgrade License.
9. Click the Browse button on the dialog (this is the button with 3 dots [...])
10. Navigate to the floppy drive or flash drive that contains the .bcllic file attachment you received.
11. Then click on the file that you received. The files are named so that they are easy to find and identify (Update_00000000-00000000-0000000000.bcllic). Your Customer ID is the first set of numbers and the Serial Number for the key that is being updated is the second set of numbers. **(NOTE: An update file will only work for one key. If you have multiple keys, multiple files will have to be supplied.)**
12. Next click Open. This will load the file into the Upgrade License dialog.
13. Finally click Update. This will read the file into the key and authorize the new products.

NOTE: After clicking Update if everything went through successfully a dialog will appear saying: "Update Successful"

Troubleshooting Potential Re-Key Problems

There are two possible errors with this process. If either error is displayed follow the steps included for each to get this matter resolved quickly.

1. Not Authorized to Update:

If this error occurs it means that our servers were unable to retrieve the proper Tag and Password for your hardlock. This can occur due to firewall interference.

In order to have this resolved quickly:

- a. Write down the Serial Number, Tag and Password or Copy and Paste the values into a text file.
- b. Then email those values to keys@bobcad.com. A new license file will be emailed promptly.

2. Not a Valid License File:

- a. This error is produced when trying to update a hardlock using an older build of the BobCAD-CAM software. Users must be on the most recent update of the software in order to apply updates to keys. In order to get the most recent update, navigate to www.bobcad.com/updates and download the most recent revision for your software. Once the update has been applied, please repeat the steps for updating your license online.

Following the steps supplied in this document should have you quickly on your way to using your recently purchased software modules, but if after following the steps an error still occurs, please contact technical support by calling 727-489-0003.

Machine Post Processor Information

The BobCAD-CAM Version 23 system includes many post processors plus the ability to customize existing post processor configurations.

Available Post Processors

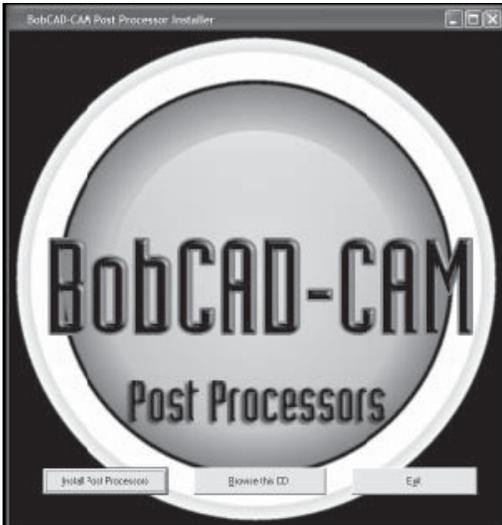
BobCAD-CAM, Inc. will post all of the currently available post processors as they become available on its <http://www.bobcad.com/> website. To access these, click on the Support link on the main page, then choose the appropriate Post Processor link for your CNC controller. From there, the available post processors are sorted alphabetically by make and model. Locate the necessary post processor, download it and save it onto the computer that BobCAD-CAM V23 is installed on, then double-click on it to run it. Follow the instructions displayed in the post installation wizard and the post will install and be ready for immediate use.

BobCAD-CAM V23 ships with a second CD that also contains all of the post processors available.

See the steps outlined below for installing from the CD.

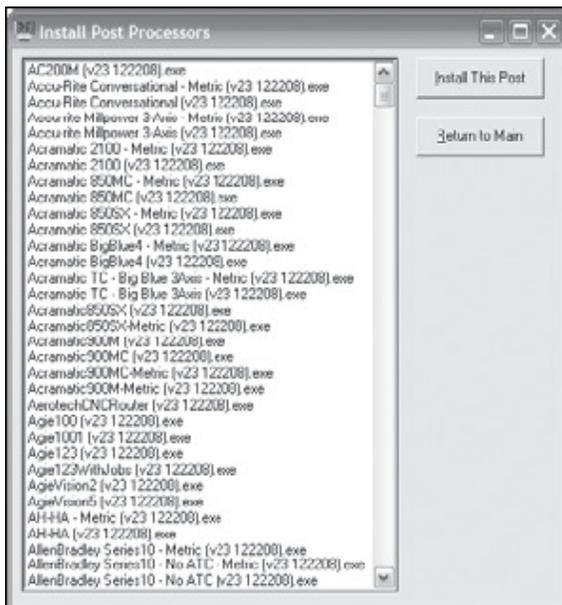
Installing Post Processors from the CD

BobCAD-CAM software comes with a Post Processor CD that offers hundreds of current posts that you can load directly from.



Here is the procedure:

1. Place the included Post Processor CD into the CD or DVD drive on the computer. The box illustrated below should automatically appear. If it does not, double-click on the CD drive icon in My Computer (Computer for Vista users) and then locate and double-click on PostProcessorSetup.exe to launch the post installer. Then, click on the button labeled Install Post Processors.
2. Choose the appropriate controller make and model for the machine. Click on Install This Post from the buttons on the right of the box and that post will install. Repeat this step for each post processor needed. Simply close the application when finished.



Post Modification Requests

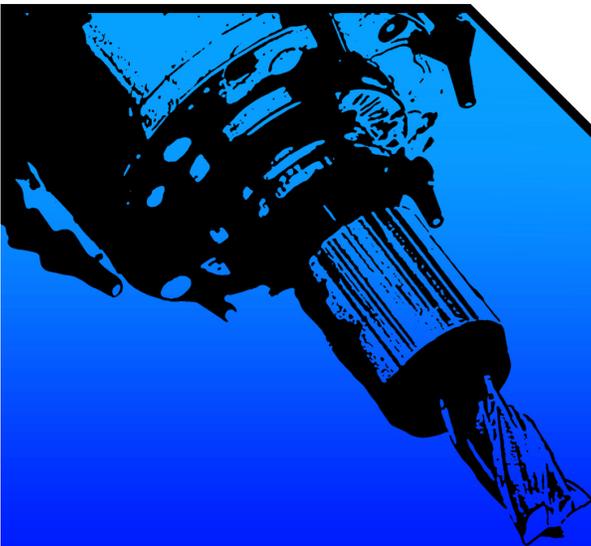
A Post Processor is required in order to translate machine toolpath information into a formatted numeric language (G-Code) that your machine control can identify with and read in. BobCAD-CAM is responsible for the creation of post processors for the most current versions of our CAD/CAM products. BobCAD-CAM supplies a library of default machine post processors that are available on CD along with your software product as well as under the support section of our website at www.bobcad.com. These generic post processors should not require additional modifications. If you are seeking additional modifications to these posts, **please review your technical support membership level benefits for post related guidelines** before requesting modifications to existing post processors. For new post processor requests you can contact the BobCAD-CAM Technical Support unit at 727-489-0003 or email them directly at support@bobcad.com.

After having reviewed your support level post modification benefits, if you qualify for post modifications, you can request a modification to your Post Processor by contacting technical support at 727-489-0003. You can email support@bobcad.com and ask for a Post Processor Modification Request Form to be sent to you as well. Again, all Post Processor requests are serviced based on your support level benefits. If you would like to upgrade your support level benefits please contact BobCAD-CAM directly at 877-262-2231. Thank you.

Chapter 2

Version 23 Mill

N 6 9 6 X 4 . 4 2 3 8 Y 2 . 6 0 6 6
N 6 9 7 X 4 . 4 1 9 8 Y 2 . 6 0 5 3
N 6 9 8 X 4 . 4 0 6 8 Y 2 . 6 0 1 2
N 6 9 9 X 4 . 4 0 3 2 Y 2 . 6 0 0 1
N 7 0 0 X 4 . 3 8 9 2 Y 2 . 5 9 5 7
N 7 0 1 X 4 . 3 8 6 3 Y 2 . 5 9 4 7
N 7 0 2 X 4 . 3 7 5 3 Y 2 . 5 9 1 3
N 7 0 3 X 4 . 3 7 3 2 Y 2 . 5 9 0 6
N 7 0 4 X 4 . 3 6 5 3 Y 2 . 5 8 8 1
N 7 0 5 X 4 . 3 6 4 Y 2 . 5 8 7 7
N 7 0 6 X 4 . 3 6 0 1 Y 2 . 5 8 6 5
N 7 0 7 X 4 . 3 5 9 2 Y 2 . 5 8 6 2
N 7 0 8 X 4 . 3 5 6 8 Y 2 . 5 8 5 4
N 7 0 9 X 4 . 3 3 4 8 Y 2 . 5 7 8 5
N 7 1 0 X 3 . 1 5 6 5 Y 2 . 2 0 7
N 7 1 1 X 3 . 1 4 3 2 Y 2 . 2 0 2 8
N 7 1 2 X 3 . 1 4 1 9 Y 2 . 2 0 2 4
N 7 1 3 X 3 . 1 4 0 5 Y 2 . 2 0 1 9
N 7 1 4 X 3 . 1 3 9 5 Y 2 . 2 0 1 6
N 7 1 5 X 3 . 1 3 8 6 Y 2 . 1 9 9 8
N 7 1 6 X 3 . 1 3 7 6 Y 2 . 1 9 9 2
N 7 1 7 X 3 . 1 3 6 7 Y 2 . 1 9 7 1
N 7 1 8 X 3 . 1 3 5 8 Y 2 . 1 9 6 4
N 7 1 9 X 3 . 1 3 4 9 Y 2 . 1 9 5 6
N 7 2 0 X 3 . 1 1 1 4 Y 2 . 1 9 2 8



Chapter 2. Version 23 Mill

This chapter is dedicated to teaching fundamental aspects of how to setup your software for Milling. This would also apply to CNC Routing, Laser, Plasma and WaterJet.

2.1 Software Setup

BobCAD-CAM's milling features provide the user with a number of options to aid machining with a mill. There are several aspects that the user should be familiar with prior to machining parts. This chapter will assist you in setting up your software for production.

First it is important to introduce you to:

- The Tool Library
- Stock Selection and Modification
- Cutting Conditions
- Tool Patterns
- Drilling Operations
- The Software Machine File

You should be able to set up these system attributes for your specific needs when completing the contents of this chapter. The following setup information will also apply to CNC Routers, Plasma, Laser and WaterJet machines.

Setting Up the Tool Library & Cutting Conditions

Let's take a look at Milling Tools first and how the software utilizes tools for machining operations and then how to modify them and set up the software for milling. Therefore we want to review the following topics:

Milling Tools
Milling Tool Dialog
Assigned Tool Dialog
Tool Pattern Program Dialog
The Machining Order Dialog
Tool Pattern Global Dialog
The Cutting Conditions Global Dialog

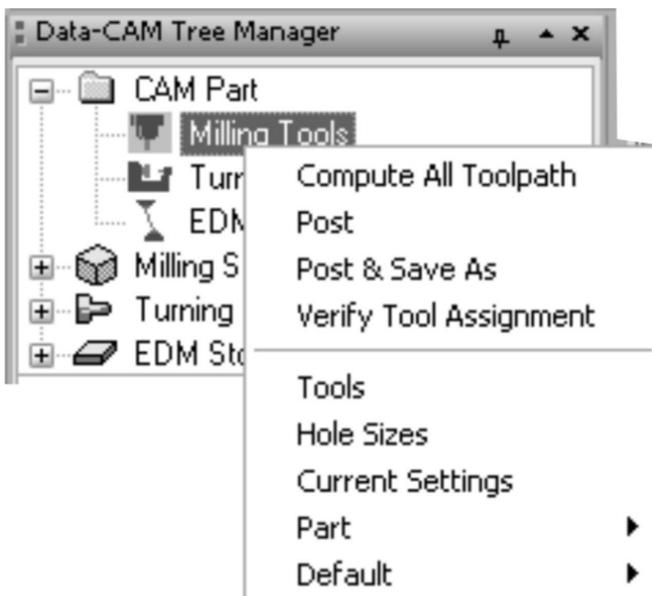
Milling Tools

Milling Tools contains all of the settings relating to the tool library and the general machine settings, including post processing. The user should take the time to review this before machining parts. Milling Tools at the CAM Part level of the tree defines the machine settings and cutting conditions for milling in BobCAD-CAM. It is simpler to describe the items available in the order they appear.

To gain access to these settings, first click the + icon next to CAM Part at the top of the tree to expand it.



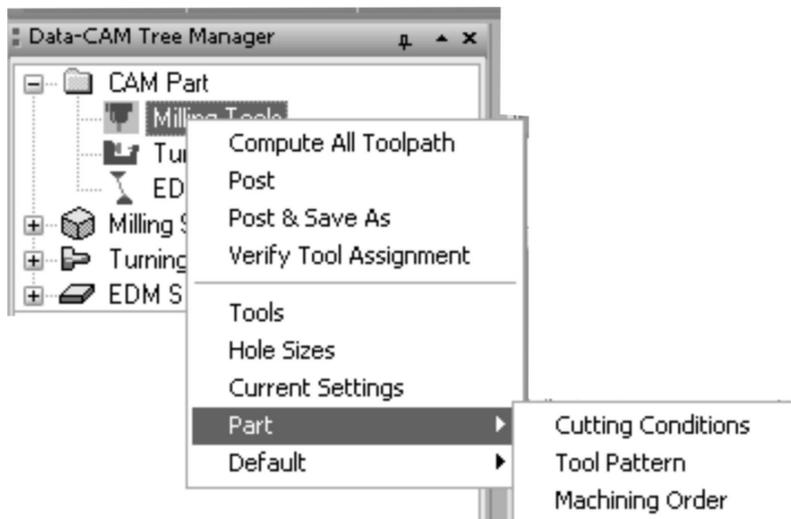
Right click on Milling Tools to display the context menu.



Let's take a closer look at the items in this new menu.

- **Compute All Toolpath.** This item in the menu will cause every feature that is already in the tree to re-compute their toolpaths in the order that they are listed. Depending on the number, type, and complexity of those features, the re-compute could take some time.
- **Post.** After all toolpath in the tree is computed, click this item to post process a completed NC program. Only those features in the tree under Milling Stock that are not marked not to post will be included.
- **Post & Save As.** This item operates the same as Post above, but also automatically opens a Save As dialog to prompt the user where to save the finished NC code.
- **Verify Tool Assignment.** This tool displays the Assigned Tools dialog box containing all of the tools used in the NC program. The tool numbers used inside BobCAD-CAM may not match the tools as they are listed on the machine, so this item is provided so that the user can reassign those numbers to reflect what is actually used in the shop.

- **Tools.** This item opens BobCAD-CAM's milling tool library for editing with the Milling Tool Menu dialog box.
- **Hole Sizes.** This item opens BobCAD-CAM's tap editing dialog to permit the user to adjust the different tools to use.
- **Current Settings.** This item opens the Milling Settings dialog to edit and set the machine defaults for a number of settings.
- **Part.** The items under Part in the menu apply only to the currently active part. These settings have no effect on other parts that may be loaded into the system.



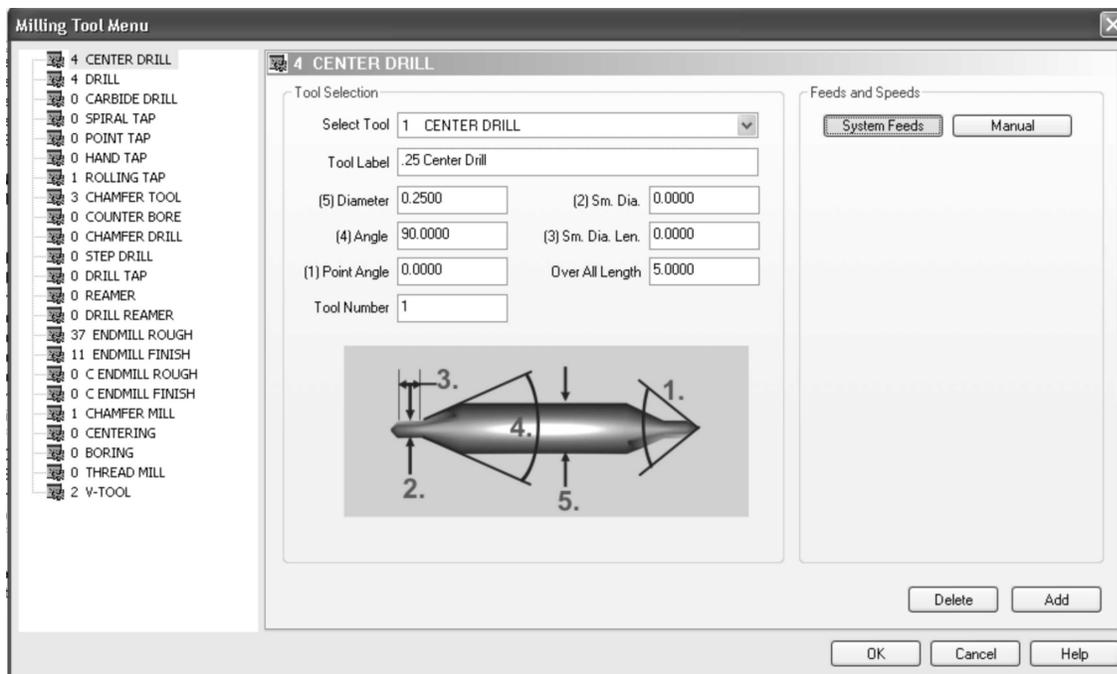
- **Part Cutting Conditions.** This item displays the Cutting Conditions Program dialog with the currently selected stock material listed. This dialog allows the user to set various parameters related to the stock material.
- **Part Tool Pattern.** This item displays the Tool Pattern Program dialog to permit the user to re-order the basic operations used in each feature type.
- **Part Machining Order.** Displays the Machining Order dialog to permit the user to set the feature machining priorities / tool-use optimization.
- **Default.** The items under Default do not apply to the current part, nor to any part that has already been created. The settings here are available to replace the current system defaults and will apply to all parts created thereafter.
- **Default Cutting Conditions.** This item displays the Cutting Conditions Global dialog with all available stock materials listed. This dialog allows the user to set various parameters related to the stock material.
- **Default Tool Pattern.** This item displays the Tool Pattern Global dialog to permit the user to re-order the basic operations used in each feature type.
- **Default Machining Order.** Displays the Machining Order dialog to permit the user to set the feature machining priorities / tool-use optimization.

Milling Tool Dialog

The **Milling Tool** dialog allows the user access to BobCAD-CAM's milling tool library. From here the user may add tools, delete tools, and switch individual tools from using the system-generated feeds to manually input feeds.

There are two important notes when using tools:

1. When using manually input feeds, the user must take into consideration which material is being used for the part. The system-generated feeds automatically take this into account.
2. If any milling feature specifies a tool that is not already in the tool library, the system will automatically add it and initialize it to system values for use. The tool that is added may be edited later if this is not desirable.



On the left side of the dialog is a list of various tool types that BobCAD-CAM currently supports. The right side contains the editing fields for each type of tool.

To Add a new Tool Definition:

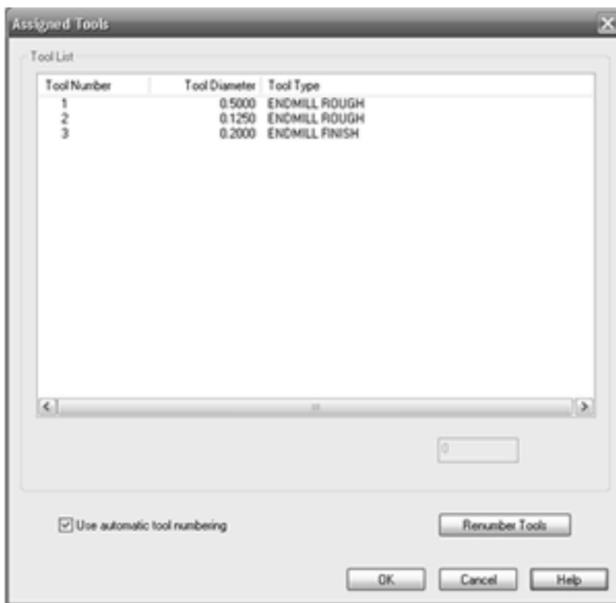
1. Click on the CAM Tree tab in the Data-CAM Tree Manager.
2. Click the + icon next to CAM Part at the top of the tree to expand it if it is not already.
3. Right click on Milling Tools in the tree and choose Tools.
4. In the list on the left side of the dialog, click on the type of tool that is to be added.
5. Click the Add button. This will add a new tool with default values.
6. Input the values for the tool. These values will vary depending on the type of tool the user is adding.
7. After all values are input, click OK. This will save the tool to the library for later use.
8. Repeat the steps 1 through 7 until all of the necessary tools have been added to the library.

To Delete a Tool Definition:

1. Click on the CAM Tree tab in the Data-CAM Tree Manager.
2. Click the + icon next to CAM Part at the top of the tree to expand it if it is not already.
3. Right click on Milling Tools in the tree and choose Tools.
4. In the list on the left side of the dialog, click on the type of tool that is to be removed.
5. In the Select Tool field, select the tool that is to be deleted.
6. Click the Delete button.
7. The selected tool will be removed from the library.
8. To remove more tools, repeat steps 4-6 for all tools to be deleted, then click OK. To cancel ALL changes made in this dialog, click Cancel.

Assigned Tools Dialog

The **Assigned Tools** dialog box permits the user to verify the tool number assignments in the posted program before the code is even post processed.



Let's take a closer look at each aspect of this dialog box:

- **Tool Number.** This field displays the assigned tool number for each tool that will be output in the finished NC code.
- **Tool Diameter.** This field displays the diameter of each tool, mainly to aid the user in telling the tools apart.
- **Tool Type.** Displays the type of each tool to aid the user in telling the tools apart.
- **Number field.** This field has no label attached. If the **Use automatic tool numbering** is not set below, the user may enter a number in this field to change the assigned tool number for the selected tool.
- **Use automatic tool numbering.** On by default, this checkbox numbers all tools that will be used in the order that they will appear in the finished program.

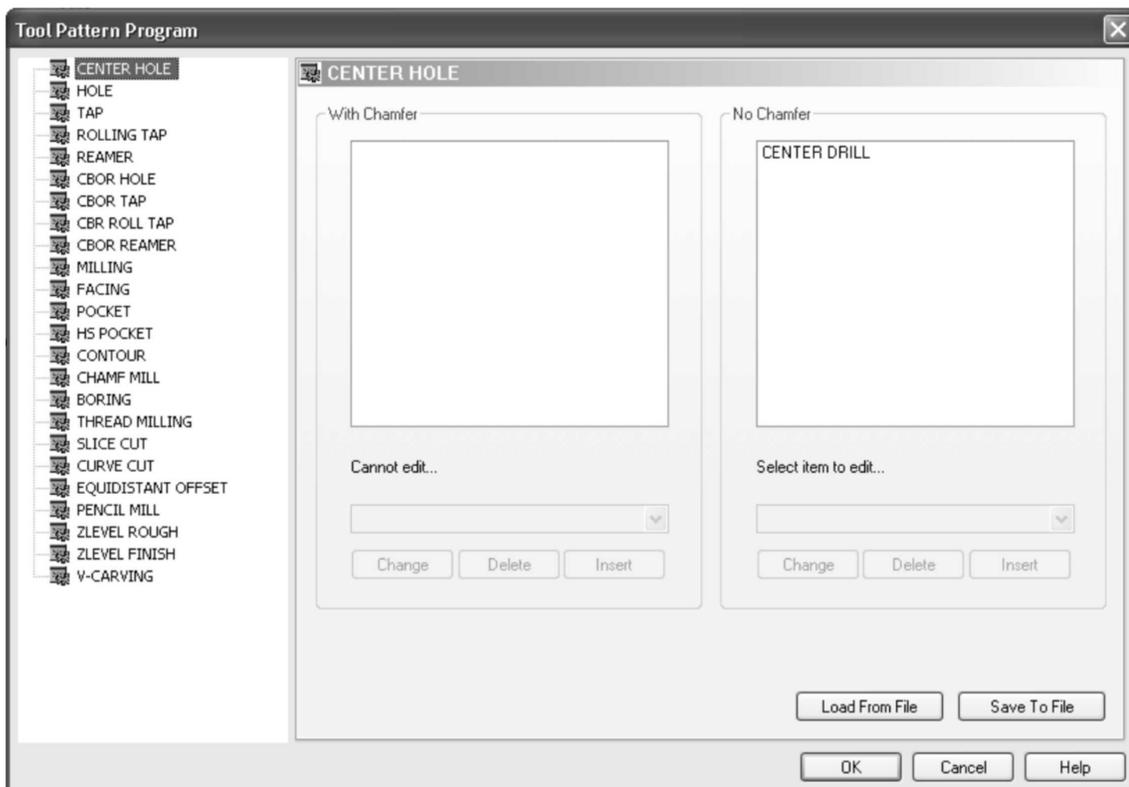
- **Renumber Tools.** If **Use automatic tool numbering** above is set, this button will cause BobCAD-CAM to renumber all of the tools in sequential order.
- **OK.** Click this to have BobCAD-CAM accept any changes made to this dialog.
- **Cancel.** Click this to have BobCAD-CAM discard any changes made and to close this dialog.
- **Help.** Display this page.

Note: To change a single tool number to a manually-assigned number, uncheck **Use automatic tool numbering** and click on the tool number in the list above. The number field will be unlocked and the user may type in a new number. If the number does not appear immediately, click on another number and the new number will be applied. All changes will apply when the user clicks OK even if they are not displayed in the list. To renumber tools after post processing, revisit this box and make the required changes. Afterward, right click on Milling Tools and choose Post. The NC code will be updated with the new tool numbers.

Tool Pattern Program Dialog

The **Tool Pattern Program** dialog serves as the list governing the order of operations in all features that BobCAD-CAM supports. Any changes applied here affect the current part only, and will not be applied to any other parts.

The dialog is split into two parts: the left side lists the features supported in the system and the right side displays the operations that make up that feature by default. The right side is set up in two halves: the operations list for the feature with a chamfer and without. Following the illustration below is a list of steps for editing the order of operations.



- **Operations list.** This box lists the operations that are currently set up for the feature highlighted on the left.
- **Select item to edit...** This drop-down box displays a list of supported operations that may be used.
- **Change.** Changes the item highlighted in the operations list to the one selected in the drop-down list above.
- **Delete.** Removes the item selected in the operations list.
- **Insert.** Inserts the item selected in the drop-down list above the operation highlighted in the operations list.
- **Load From File.** Loads an entire tool pattern configuration from a previously saved .tpatt file.
- **Save To File.** Saves the entire tool pattern configuration to a new .tpatt file.
- **OK.** Click this to have BobCAD-CAM accept the changes made.
- **Cancel.** Click this to have BobCAD-CAM discard all changes made in this dialog and close it.
- **Help.** Display this help page.

Add an item:

1. Select the feature to be edited from the list on the left of the dialog.
2. In the operations list, highlight the item that should appear below the new operation.
3. In the Select item to edit... list, choose the new item to add.
4. Click Insert.
5. The new operation will be added to this feature. When this feature is added to the tree under Milling Stock, the new operation will appear in the tree with it.

Change an item:

1. Select the feature to be edited from the list on the left of the dialog.
2. In the operations list, highlight the item that should be changed.
3. In the Select item to edit... list, choose the new item to change the highlighted item to.
4. Click Change.
5. The highlighted operation will be changed in this feature type to the new one. When this feature is added to the tree under Milling Stock, this operation will appear in the tree with it.

Remove an item:

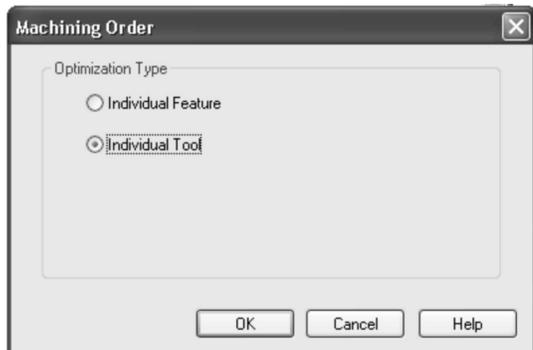
1. Select the feature to be edited from the list on the left of the dialog.
2. In the operations list, highlight the item that should be deleted.
3. Click Delete.
4. The highlighted operation will be removed from this feature type entirely. When this feature is added to the tree under Milling Stock, this operation will no longer appear in the tree with it.

Note: In the CBores Hole (the Counter Bore Hole feature in Milling Stock) in the list to the left, the default tool for the counter bore operation is a counter bore mill, not a counter boring tool. This was originally done to illustrate the ability to use different tools for the same operation. If a strict counter boring tool is desired, change the CBores Mill tool in this pattern to CBores Tool instead.

The Machining Order Dialog

The **Machine Order** dialog allows the user to specify how the tool changes in the program are optimized. The tool ordering can be a little confusing at first; see the examples below.

If the Machine Order dialog is accessed through **Default** in the **Milling Tools** context menu, the setting here will affect all new parts created after it was changed. If accessed through **Part** in the same menu, it will affect only the current part. To change it for the current part in any case, it must be changed through **Part**.



The following two examples center drill, drill, chamfer, and tap two different sized holes. The center drill and the chamfer tools are the same size, but the drills and taps used are different.

Individual Feature

Center drill hole 1
Drill hole 1
Chamfer hole 1
Tap hole 1

Center drill hole 2
Drill hole 2
Chamfer hole 2
Tap hole 2

Individual Tool

Center drill both holes
Drill hole 1
Chamfer hole 1
Tap hole 1

Drill hole 2
Chamfer hole 2
Tap hole 2

In these examples, though two of the tools are the same, BobCAD-CAM will use the same tools for as long as that can be used. With Individual Tool set in the tapping examples above, both holes are center drilled one after the other without a tool change. Since the next step in the features is to drill the holes out and the drills are differing sizes, BobCAD-CAM continues finishing the first hole feature from that point in the same way that the Individual Feature setting would, then returns to drill the second feature and complete it. Each step in both examples requires a tool change, and these can be minimized when using Individual Tool.

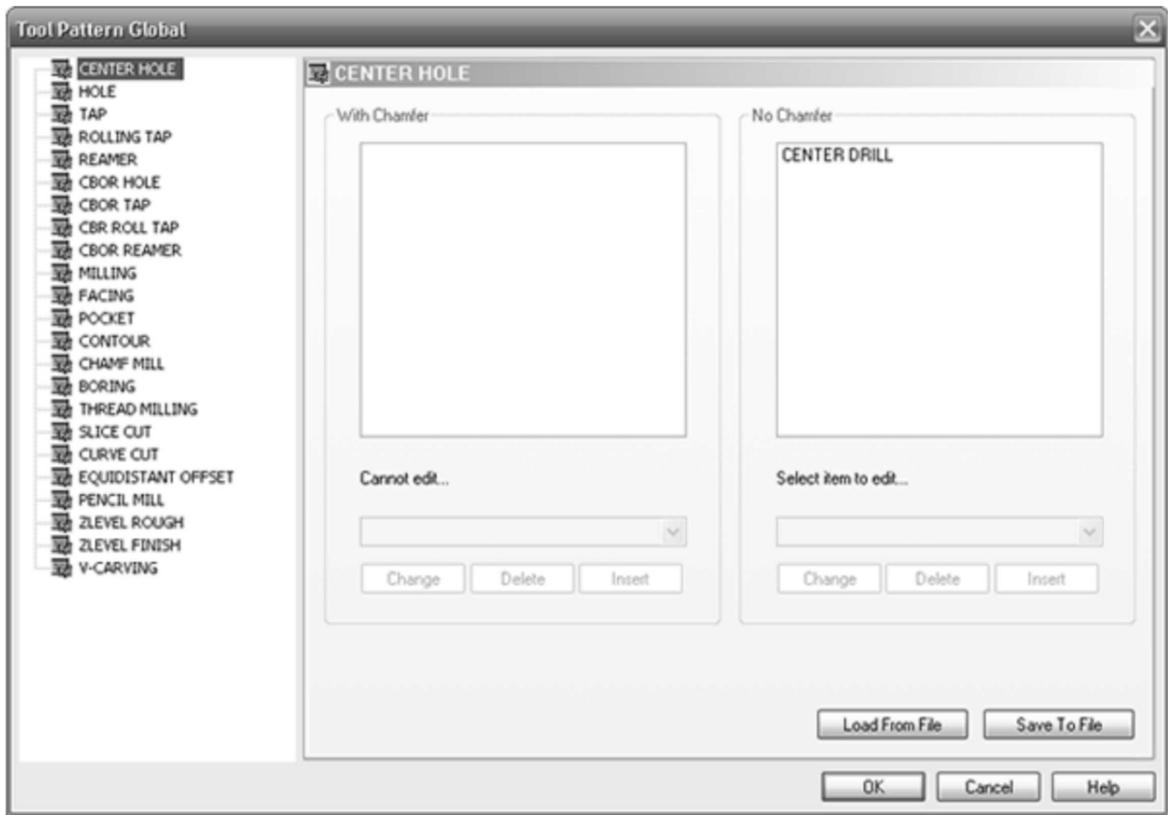
There are times when Individual Feature should be used instead. These are normally when the stock left from one feature must be completely removed in order to machine an adjacent feature on the part. Even if the tools used are the same, BobCAD-CAM will change tools in the post processed output until the first feature is completed before moving on to the next.

Tool Pattern Global Dialog

The **Tool Pattern Global** dialog serves as the master list governing the order of operations in all features that BobCAD-CAM supports.

The dialog is split into two parts: the left side lists the features supported in the system and the right side displays the operations that make up that feature by default. The right side is set up in two halves: the operations list for the feature with a chamfer and without.

Following the illustration below is a list of steps for editing the order of operations.



- **Operations list.** This box lists the operations that are currently set up for the feature highlighted on the left.
- **Select item to edit...** This drop-down box displays a list of supported operations that may be used.
- **Change.** Changes the item highlighted in the operations list to the one selected in the drop-down list above.
- **Delete.** Removes the item selected in the operations list.
- **Insert.** Inserts the item selected in the drop-down list above the operation highlighted in the operations list.
- **Load From File.** Loads an entire tool pattern configuration from a previously saved .tpatt file.
- **Save To File.** Saves the entire tool pattern configuration to a new .tpatt file.
- **OK.** Click this to have BobCAD-CAM accept the changes made.
- **Cancel.** Click this to have BobCAD-CAM discard all changes made in this dialog and close it.
- **Help.** Display this help page.

Add an item:

1. Select the feature to be edited from the list on the left of the dialog.
2. In the operations list, highlight the item that should appear below the new operation.
3. In the Select item to edit... list, choose the new item to add.
4. Click **Insert**.
5. The new operation will be added to this feature. When this feature is added to the tree under **Milling Stock**, the new operation will appear in the tree with it.

Change an item:

1. Select the feature to be edited from the list on the left of the dialog.
2. In the operations list, highlight the item that should be changed.
3. In the **Select item to edit...** list, choose the new item to change the highlighted item to.
4. Click **Change**.
5. The highlighted operation will be changed in this feature type to the new one. When this feature is added to the tree under **Milling Stock**, this operation will appear in the tree with it.

Remove an item:

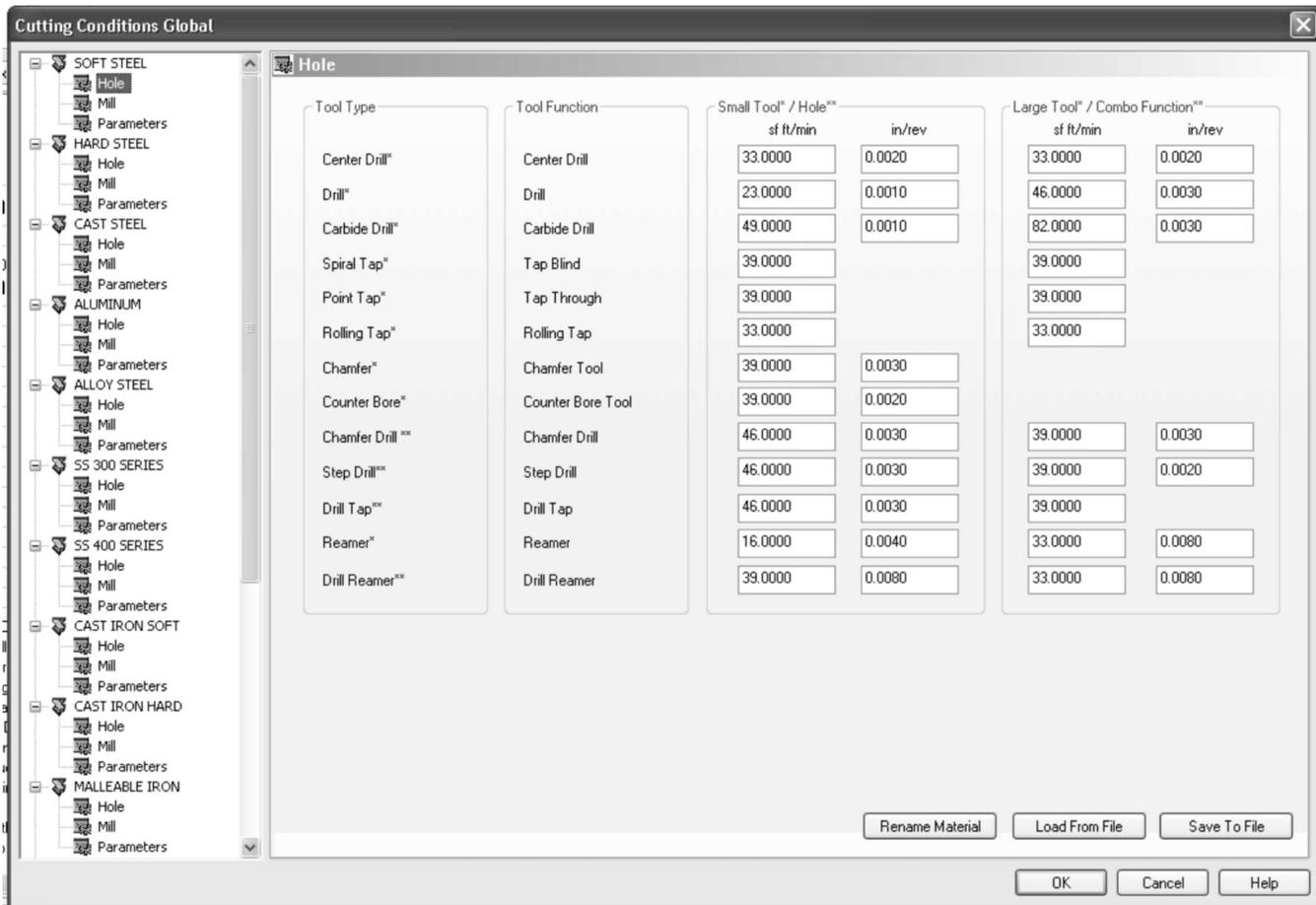
1. Select the feature to be edited from the list on the left of the dialog.
2. In the operations list, highlight the item that should be deleted.
3. Click **Delete**.
4. The highlighted operation will be removed from this feature type entirely. When this feature is added to the tree under **Milling Stock**, this operation will no longer appear in the tree with it.

Note: In the CBores Hole (the Counter Bore Hole feature in Milling Stock) in the list to the left, the default tool for the counter bore operation is a counter bore mill, not a counter boring tool. This was originally done to illustrate the ability to use different tools for the same operation. If a strict counter boring tool is desired, change the CBores Mill tool in this pattern to CBores Tool instead.

The Cutting Conditions Global Dialog

The **Cutting Conditions Global** dialog edits the conditions for all of the available materials in the library. The dialog is split into two sides. On the left side of the dialog is a tree view list of the materials currently available in the materials library. The available conditions in the library are Mill, Hole, and Parameters. The editing fields are listed in the right side of the dialog. These inputs fields will change depending upon which condition is selected in the tree on the left. Let's go ahead and take a closer look at each of these categories.

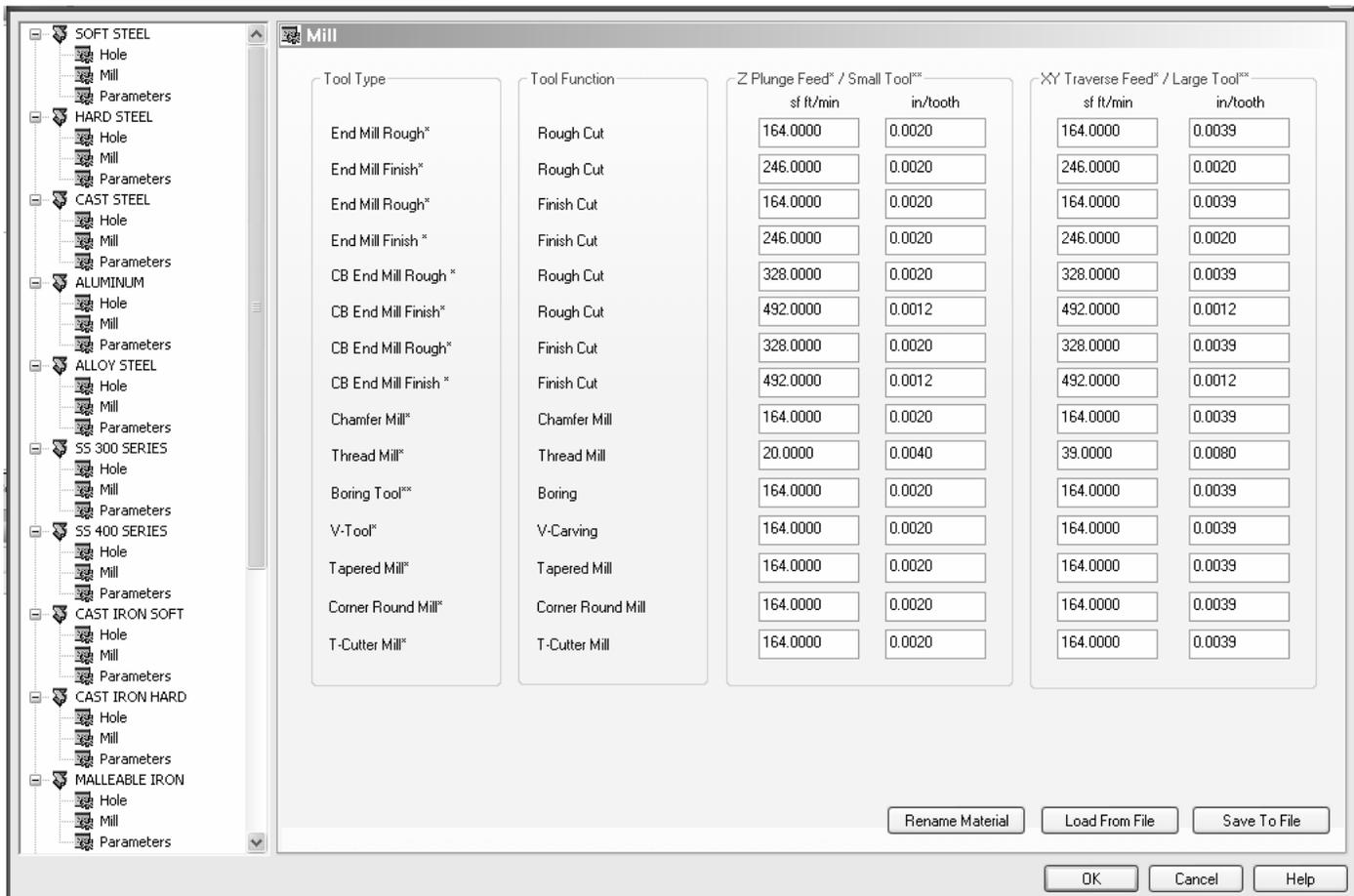
The Hole Category



- **Tool Type:** The items in this field cannot be modified by the user. This is a list of the tools supported for drilling features in the software.
- **Tool Function:** The items in this field cannot be modified by the user. This is a list of the operations that each tool is used for.
- **Small Tool*/Hole**:** This field contains a list of the feeds specified for the individual tools. The feed rates that are specified in this field are used when the hole size is less than or equal to 0.118 inches (3 mm). The input boxes in this field allow the user to specify both the surface feet per minute (sf ft/min) and units per revolution (in/rev or mm/rev) for each tool.

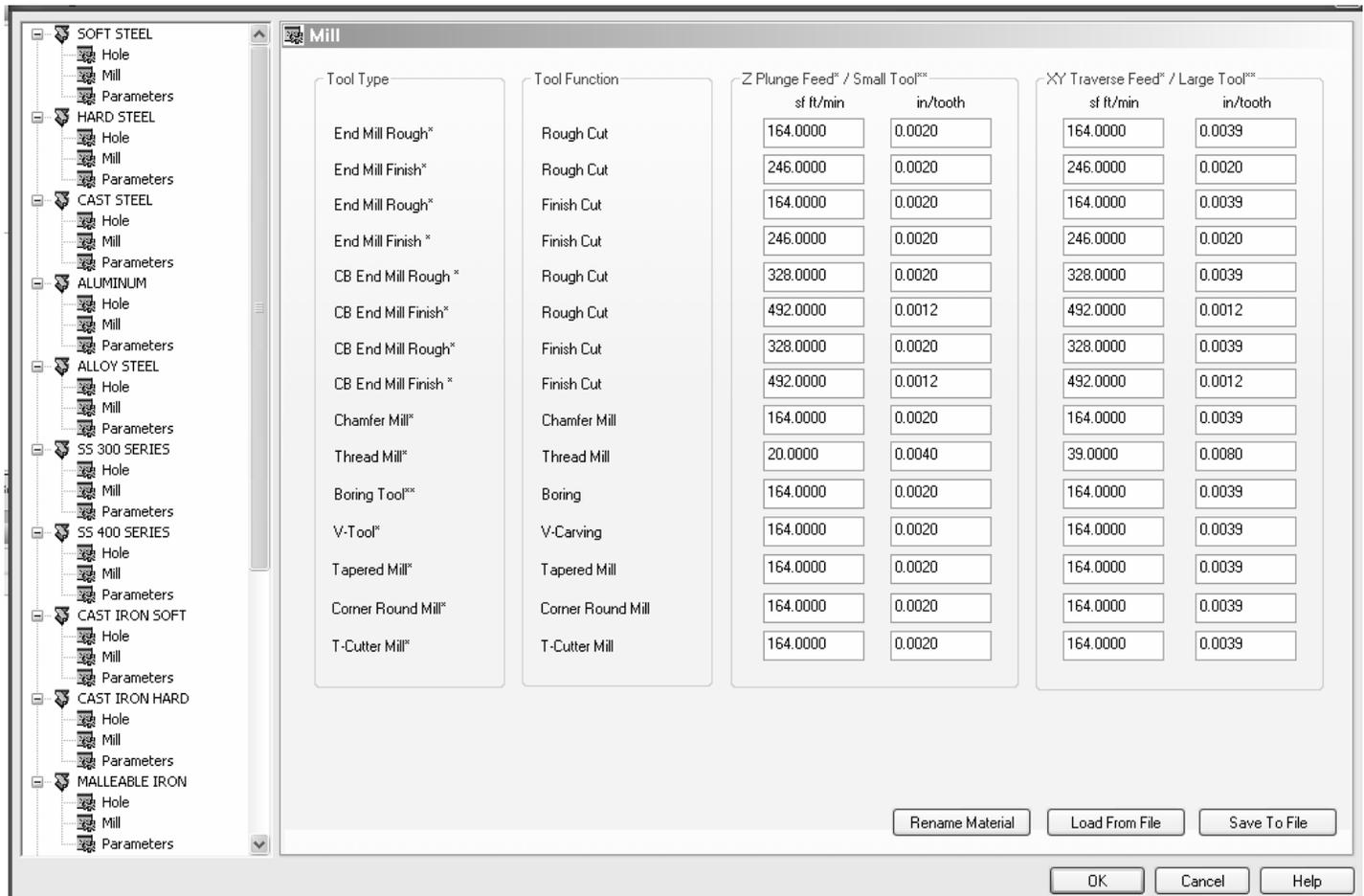
- Large Tool*/Combo Function**:** This field contains a list of the feed rates specified for the individual tools. The feed rates that are specified in this field are used when the hole size is greater than 0.118 inches (3 mm). The input boxes in this field allow the user to specify both the surface feet per minute (sf ft/min) and units per revolution (in/rev or mm/rev) for each tool.

The Mill Category



- Tool Type:** The items in this field cannot be modified by the user. This is a list of the tools supported for milling features in the software.
- Tool Function:** The items in this field cannot be modified by the user. This is a list of the operations that each tool is used for.
- Z Plunge Feed*/Small Tool**:** These fields are used to calculate the feed rates for vertical plunge moves for the selected material. The input fields allow the user to specify the surface feed in units per minute as well as the chipload per tooth.
- XY Traverse Feed*/Large Tool**:** These fields are used to calculate the feed rates for the X/Y movements for the selected material. The input fields allow the user to specify the surface feed in units per minute as well as the chipload per tooth.

The Parameters Category



Center Drill

- **Dwell Seconds:** When this option is selected the value in the input box will reflect a specific measure of time. This measure of time will be applied to the **Center Drill** operations that are output by the CAM system.
- **Dwell Revolutions:** When this option is selected the value in the input box will reflect revolutions the tool must complete during the dwell.

Drill

- **Drill Step Ratio:** This parameter governs when the system will automatically apply a peck drilling cycle. By default, the value is set to 300 percent, meaning that if the depth of the hole is three times greater than the diameter of the drill the system will automatically output a G73 or G83 depending on which option is selected and which is supported in the post processor.
- **Drill Step Increment Ratio:** For a peck drill, this determines the peck depth that the drill will plunge into the hole to start the next peck rather than retracting to the clearance plane. This field is given as a percentage of the diameter of the tool.

- **Length Through Cut:** This value is used to define the amount, or distance that the tool will travel through the material after it has reached the bottom of the stock in 'through hole' machining. This distance does not include the tool point. This value only has effect in hole making jobs that have been set to **Through** in the **Parameters** page of the feature's edit dialog.
- **Peck Retract Amount:** For a peck drill, this determines the distance that the drill will retract in the hole to start the next peck rather than retracting to the clearance plane.
- **Add To Drill Depth Blind Tap:** This value is used to add an extra constant amount to all drill depths when doing blind hole tapping. The need for this value usually arises when a machine does not have rigid tapping, or when some older machines, in the process of reversing the tap direction, make extra revolutions before stopping. This value acts as a safety so as not to break taps.
- **Stop Before Hole:** This value defines the distance above the work piece at which the tool movement will switch from rapid feed rate to cutting feed rate. This value is used for all hole making functions (drilling, tapping, etc.).
- **Dwell Seconds:** When this option is selected, the value in the input box will display and allow the user to modify the dwell time in seconds. This measure of time will be applied to the drill operations that are output by the CAM system.
- **Dwell Revolutions:** When this option is selected the value in the input box will display and allow the user to modify the dwell time in # of full revolutions of the spindle rather than a set amount of time.

Ream

- **Ream Primary Hole Depth:** This value is used to automatically calculate the drilling depth in reaming jobs with blind holes when the entered drilling depth is less than or equal to the reaming depth. This value is used in place of the **Inef. Length** field in the assigned reamer tool if this tool parameter is set to 0.0. The addition drilling amount will be equal to the reamer diameter multiplied by the entered value.
- **Reamer Finish Allowance 1 Large:** The system will allow the user to use one or two finishing steps between the drill and reamer operations of the reaming job. These are normally performed with end mills. This field is the value used for the first step of a two step finishing process if the reamer O.D. is greater than 0.118. The end mill that will automatically be selected will have a diameter of the reamer diameter minus the **Reamer Finish Allowance 1 Large**.
- **Reamer Finish Allowance 2 Large:** The system will allow the user to use one or two finishing steps between the drill and reamer operations of the reaming job. These are normally performed with end mills. This field is the value used for the second step of a two step finishing process if the reamer O.D. is greater than 0.118. The end mill that will automatically be selected will have a diameter of the reamer diameter minus the **Reamer Finish Allowance 2 Large**.
- **Reamer Finish Allowance 1 Small:** The system will allow the user to use one or two finishing steps between the drill and reamer operations of the reaming job. These are normally performed with end mills. This field is the value used for the first step of a two step finishing process if the reamer O.D. is less than or equal to 0.118. The end mill that will automatically be selected will have a diameter of the reamer diameter minus the **Reamer Finish Allowance 1 Small**.

- **Reamer Finish Allowance 2 Small:** The system will allow the user to use one or two finishing steps between the drill and reamer operations of the reaming job. These are normally performed with end mills. This field is the value used for the second step of a two step finishing process if the reamer O.D. is less than or equal to 0.118. The end mill that will automatically be selected will have a diameter of the reamer diameter minus the **Reamer Finish Allowance 2 Small**.
- **Dwell Seconds:** When this option is selected, the value in the input box will display and allow the user to modify the dwell time in seconds. This measure of time will be applied to the ream operations that are output by the CAM system.
- **Dwell Revolutions:** When this option is selected the value in the input box will display and allow the user to modify the dwell time in # of full revolutions of the spindle rather than a set amount of time.

Chamfer

- **Chamfer Cutting Position:** This value defines the default tool position for chamfer milling operations. The distance is used to determine which part of the angular cutting edge of the chamfer milling tool will be cutting the chamfer. The distance from the bottom of the chamfer to the bottom of the tool is the value that is needed.
- **Dwell Seconds:** When this option is selected, the value in the input box will display and allow the user to modify the dwell time in seconds. This measure of time will be applied to the chamfer operations that are output by the CAM system.
- **Dwell Revolutions:** When this option is selected the value in the input box will display and allow the user to modify the dwell time in # of full revolutions of the spindle rather than a set amount of time.

Counter Bore

- **Counter Bore Mill Finish Allowance Small:** This field is used to help the system select an end mill that will be used to counter bore milling operation. If the counter bore diameter is less than or equal to 0.5, then the selected end mill will be less than or equal to the counter bore diameter minus the value in **Counter Bore Mill Finish Allowance Small**.
- **Counter Bore Mill Finish Allowance Large:** This field is used to help the system select an end mill that will be used to counter bore milling operation. If the counter bore diameter is greater than 0.5, then the selected end mill will be less than or equal to the counter bore diameter minus the value in **Counter Bore Mill Finish Allowance Large**.
- **Dwell Seconds:** When this option is selected, the value in the input box will display and allow the user to modify the dwell time in seconds. This measure of time will be applied to the counter boring operations that are output by the CAM System.
- **Dwell Revolutions:** When this option is selected the value in the input box will display and allow the user to modify the dwell time in # of full revolutions of the spindle rather than a set amount of time.

Tap

- **Tap Primary Hole Depth:** This value defines the default number of ineffective threads that are to be used to automatically calculate the drilling depths for blind hole tapping when the entered drilling depth is less than or equal to the tap depth. If the tapping tool that is automatically assigned to the tap job has a value greater than 0.0 for **Inef. Length**, then the value in the tool definition will be used in place of this value. This value is used if the **Inef. Length** value in the tool definition is set to 0.0.

Bore

- **Bore Finish Allowance 1 Small:** The system will allow the user to use one or two finishing steps between the drill and bore operations of the boring job. These are normally performed with end mills. This field is the value used for the first step of a two step finishing process if the bore O.D. is less than or equal to 0.118. The end mill that will automatically be selected will have a diameter of the bore diameter minus the **Bore Finish Allowance 1 Small**.
- **Bore Finish Allowance 2 Small:** The system will allow the user to use one or two finishing steps between the drill and bore operations of the boring job. These are normally performed with end mills. This field is the value used for the second step of a two step finishing process if the bore O.D. is less than or equal to 0.118. The end mill that will automatically be selected will have a diameter of the bore diameter minus the **Bore Finish Allowance 2 Small**.
- **Bore Finish Allowance 1 Large:** The system will allow the user to use one or two finishing steps between the drill and bore operations of the boring job. These are normally performed with end mills. This field is the value used for the first step of a two step finishing process if the bore O.D. is greater than 0.118. The end mill that will automatically be selected will have a diameter of the bore diameter minus the **Bore Finish Allowance 1 Large**.
- **Bore Finish Allowance 2 Large:** The system will allow the user to use one or two finishing steps between the drill and bore operations of the boring job. These are normally performed with end mills. This field is the value used for the second step of a two step finishing process if the bore O.D. is greater than 0.118. The end mill that will automatically be selected will have a diameter of the bore diameter minus the **Bore Finish Allowance 2 Large**.
- **Dwell Seconds:** When this option is selected, the value in the input box will display and allow the user to modify the dwell time in seconds. This measure of time will be applied to the boring operations that are output by the CAM System.
- **Dwell Revolutions:** When this option is selected the value in the input box will display and allow the user to modify the dwell time in # of full revolutions of the spindle rather than a set amount of time.

Rename Material: When this button is selected the Rename Material dialog will appear in the foreground. This allows the user to specify the name of material as it appears in the Cutting Conditions file and as a result in the software. After inputting a new name in the Rename Material dialog clicking OK will close the dialog and assign the input name to the material. If the user clicks Cancel in the Rename Material dialog, the dialog will close without making the change to the Cutting Conditions.

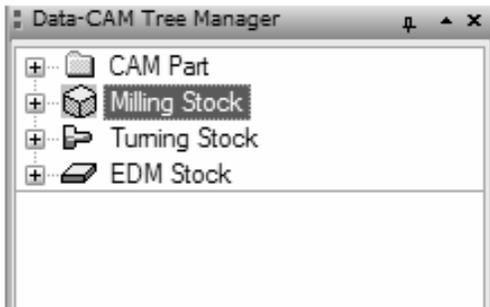


Load From File: When this button is selected a File Open dialog will appear. This allows the user to select and load a Cutting Condition File (.ccond) that was previously saved. After loading the .ccond file the materials and settings will reflect the values that were saved.

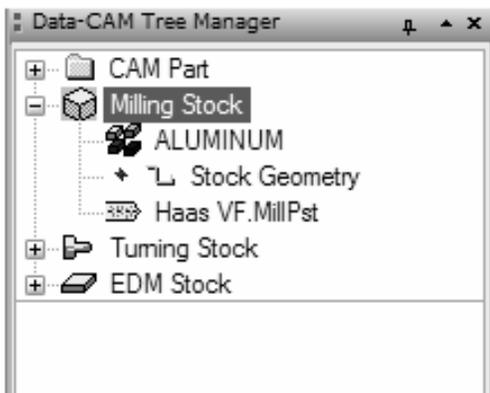
Save To File: When this button is selected a File Save dialog will appear. This allows the user to save the material names and settings to a file that can later be loaded for a different machine or loaded on a different computer.

Milling Stock

Milling Stock allows the user to change the material to be machined and also serves as the placeholder for all milling features that have been inserted into the tree.



Click on the small + symbol next to **Milling Stock** to expand it and display these items:



Let's take a closer look at the tree items:

- **Stock Material.** Right-click on this item and choose Edit to display the Material Selection dialog and quick-change the current stock material.
- **Stock Geometry.** By default, BobCAD-CAM will assume a rectangular stock just large enough to contain any generated toolpath.

TIP – How to change stock geometry:

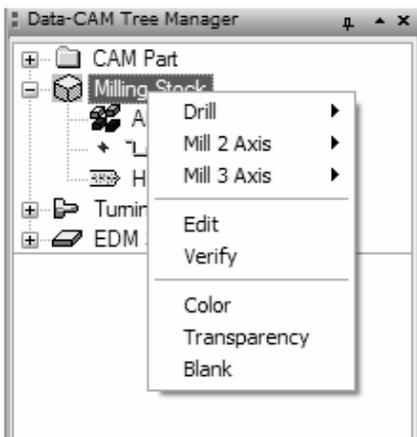
BobCAD-CAM automatically assumes a rectangular stock geometry when toolpath is generated. The default stock is automatically resized in order to encompass all of the generated toolpath. However, occasionally the user will require a different shape for the stock.

To define a new stock shape:

1. Draw the 2D shape of the milling stock to be used.
2. Right click on Stock Geometry under Milling Stock and choose Re/Select.
3. While holding the Shift key on the keyboard, click on the wireframe shape that represents the outer boundary of the stock.
4. Right click anywhere in the workspace and choose OK from the menu that will appear.
5. The stock will be automatically drawn with the 2D shape selected. To change the depth, see the Stock dialog.

- **Current Post Processor.** This item displays the name of the post processor that is currently loaded for milling parts. Right-click on it and choose Edit to display the Set Post Processor dialog, which allows the user to quick-change the current post processor without editing the general machine setup.

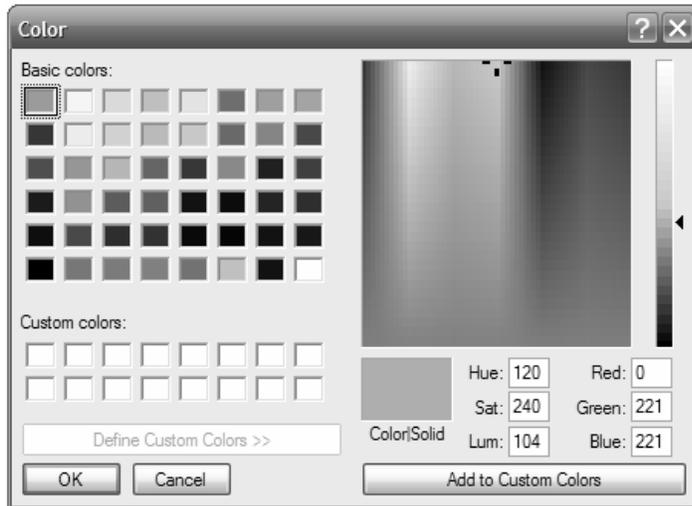
Milling Stock also has a right-click context menu. Right click on it to display the different options below:



The following is a list of the options found in this menu:

- **Drill:** This option allows the user to add a new hole-making feature to the bottom of the CAM tree.
- **Mill 2 Axis:** This option allows the user to add a new "2D" milling feature to the bottom of the CAM tree.
- **Mill 3 Axis:** This option allows the user to add a new "3D" milling feature to the bottom of the CAM tree.
- **Edit:** This option allows the user to modify the parameters of the stock.
- **Verify:** When this option is selected the verification window will be opened for the user to simulate and verify the calculated toolpath.
- **Color:** This function displays the Color dialog to allow the user to modify the color of the stock displayed in the workspace.

The **Color** dialog allows the user to choose nearly any color that the computer is capable of displaying for the stock as it is defined in the workspace.



In the box are several options:

- **Basic Colors.** These comprise the pre-defined colors that ship with BobCAD-CAM. These colors cannot be edited. To use them, simply click on one of them and click the **OK** button to accept the new color setting.
- **Custom Colors.** When the user drags the cursor around the "rainbow" color picker box and clicks on **Add to Custom Colors** below it, the chosen color will be added to the **Custom colors** fields. BobCAD-CAM will remember these color choices.
- **OK.** Click this button to have BobCAD-CAM accept whatever changes have been made in this dialog.
- **Cancel.** Click this button to discard any changes to this dialog and close it.
- **Color/Solid.** This field displays the chosen color.
- **Hue.** The **Hue** of the chosen color. (*For an explanation of HSV/HSL colorspaces, see <http://www.wikipedia.com> and search for "HSV color" in the box provided. This is an external website; BobCAD-CAM cannot be responsible in any way for the content displayed.*)
- **Sat.** The **Saturation** of the chosen color.
- **Lum.** The **Luminance** of the chosen color.
- **Red.** This field displays the red component number of the chosen R/G/B color. (*For an explanation of RGB colorspaces, see <http://www.wikipedia.com> and search for "RGB color" in the box provided. This is an external website; BobCAD-CAM cannot be responsible in any way for the content displayed.*)
- **Green.** This field displays the green component number of the chosen R/G/B color.
- **Blue.** This field displays the blue component number of the chosen R/G/B color.
- **Add to Custom Colors.** This button adds the color chosen in the color picker above to the **Custom colors** section. If there is no custom color box chosen, it will add the color as a new color. If there is one selected, it will change that color to the one chosen instead.

- **Transparency:** This function displays the Stock Transparency dialog to allow the user to change the opacity of the defined stock displayed in the workspace.

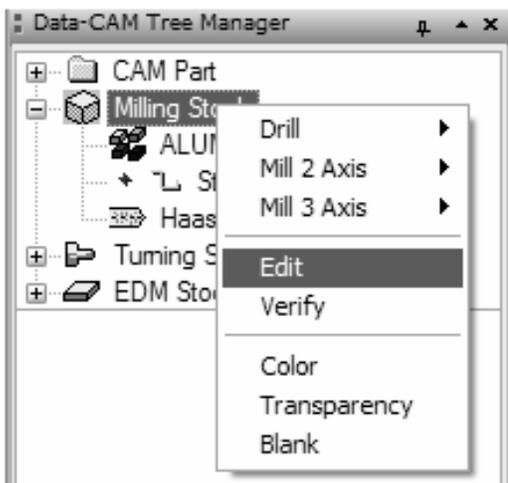
The **Stock Transparency** dialog adjusts the transparency of the stock as it is drawn in the workspace.



- **Low** - the left end of the slider minimizes the transparency of the stock so that it appears more opaque.
 - **High** - the right end of the slider maximizes the transparency of the stock so that it appears more see-through.
 - **OK** - the OK button instructs BobCAD-CAM to accept the new setting and closes the dialog.
- **Blank:** This option allows the user to hide or show the defined stock in the workspace.

The Stock Dialog

The **Stock** dialog provides the user with a place to change all of the settings used by BobCAD-CAM for the current part's stock. To display the dialog, right click on Milling Stock and choose Edit from the context menu:

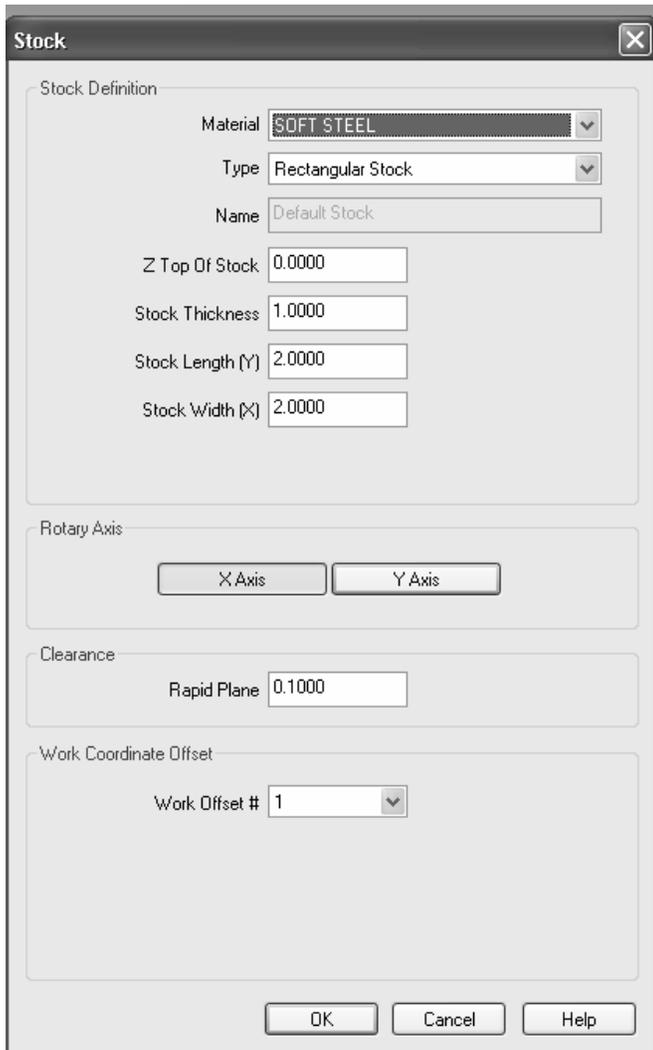


BobCAD-CAM supports two different types of stock for milling, rectangular and cylindrical. If the Stock Geometry has been set for this stock, many of the fields detailed below will not be available because the geometry chosen will dictate the shape of the stock.

Both stock shapes use different parameters and are detailed separately below:

Rectangular Stock:

The **Rectangular Stock** shape is the standard cube of stock most often used in 2 and 3 axis milling.



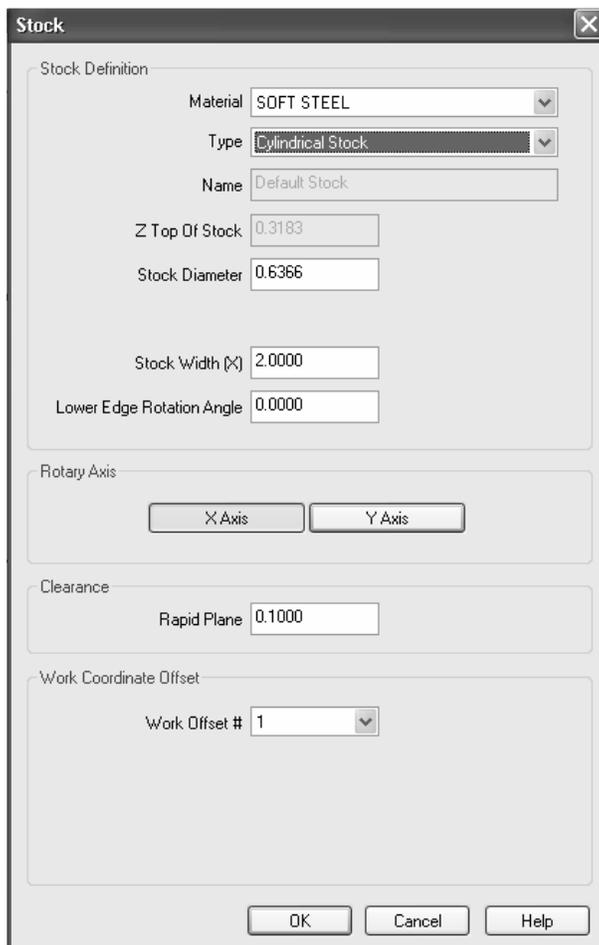
Let's take a closer look at this dialog box:

- **Material.** This is the type of material comprising the stock.
- **Name.** Displays the name of the stock. Currently only one stock is supported at a time so the name field is not editable.
- **Z Top Of Stock.** Sets the Z-axis of the top surface of the material.
- **Stock Thickness.** Set the thickness of the stock. This is always a positive value and extends downward from the **Z Top Of Stock** setting above.
- **Stock Length (Y).** This sets the size of the stock rectangle in the Y axis.
- **Stock Width (X).** This sets the size of the stock rectangle in the X axis.
- **Clearance**

- **Rapid Plane.** This value sets the global retract plane for the entire stock. It is taken into account in every feature under this stock in the tree.
- **Work Offset #.** If the stock is set up on to use a specific work offset on the machine, use this field to choose the offset that is output into the posted program.
- **OK.** Click OK to have BobCAD-CAM accept the changes to the stock.
- **Cancel.** Click Cancel to have BobCAD-CAM discard any changes made and close this dialog.
- **Help.** Display this help page.

Cylindrical Stock:

Cylindrical Stock is used in 4 axis parts where the Y axis length of the toolpath is converted into the A rotary axis, and 4th axis indexed parts that change rotations only once at the beginning of a feature. The post processor must be set up to use a rotary axis and cylindrically-shaped stock in order for this stock type to be valid.



- **Material.** The type of material comprising the stock.
- **Name.** Displays the name of the stock. Currently only one stock is supported at a time so the name field is not editable.
- **Z Top Of Stock.** Sets the Z-axis of the top surface of the material.

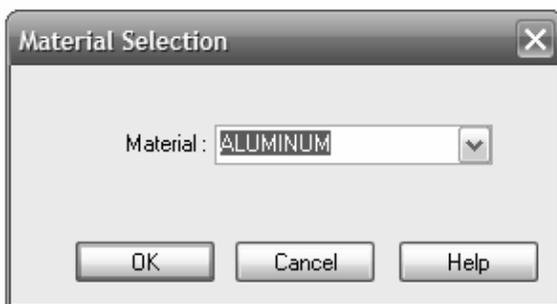
- **Stock Diameter.** This allows the user to set the maximum diameter of the milling stock.
- **Stock Width (X).** This item permits the user to set the length of the stock along the X axis.
- **Lower Edge Rotation Angle.** If the stock is to begin at a rotary angle other than 0, set this field to that starting angle.
- **Clearance**
- **Rapid Plane.** This value sets the global retract plane for the entire stock. It is taken into account in every feature under this stock in the tree.
- **Work Offset #.** If the stock is set up on to use a specific work offset on the machine, use this field to choose the offset that is output into the posted program.
- **OK.** Click **OK** to have BobCAD-CAM accept the changes to the stock.
- **Cancel.** Click **Cancel** to have BobCAD-CAM discard any changes made and close this dialog.
- **Help.** Display this help page.

Selecting Material from the Tree

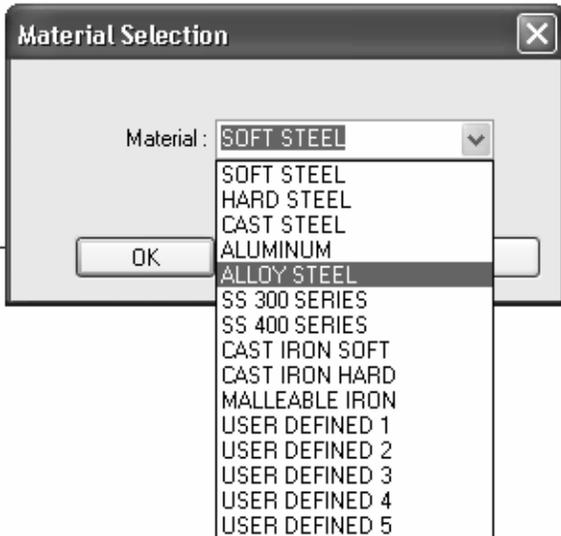
The **Material Selection** dialog allows the user to quickly change the material used for the stock without the need to go through the Stock editing dialog. To display this dialog, right click on the material name in the tree:



The Material Selection dialog will appear:



To change the material, click on the arrow to the right of the material to open the drop-down box.



Select the material to use, then click OK and BobCAD-CAM will accept the material change.

Click the Cancel button to close the dialog without changing the material.

Using the Tool Page within a Feature

The **Tool** page within each feature allows the user to utilize system generated tools as well as manually generated tools and tools stored in the tool library. The user is also provided with the means to verify and/or adjust the feeds and speeds for the tool within the feature.

Tool Selection

Tool Type: ENDMILL ROUGH

(1) Diameter: 0.1250 (2) Flute Length: 0.5000

Corner Radius: 0.0000 Number of Flutes: 2

Tool Number: 2 Over All Length: 5.0000

Feeds and Speeds

S F P M: 49.0000

Spindle Speed: 1497.3297

Z - S F P M: 49.0000

Z - Speed: 1497.3297

Feed per Tooth: 0.0020

Cutting Feed: 5.9893

Z - Feed per Tooth: 0.0012

Z - Feed: 3.5936

Spindle Speed %: 100

Cutting Feed %: 100

Arc Slowdown %: 100

Tool Selection

- **System Tools:** By default all features within the software use a system tool. When using the system tool the user is required to input the *Diameter* of the tool and the *Corner Radius* of the tool. Based on these inputs the software will attempt to find a tool in the library that matches the values that the user has input. If a tool cannot be found the system will add a new tool to the library with the Diameter and Corner Radius specified by the user. All other values will be assigned default values.
- **Manual Tools:** Clicking on this button allows the user to specify all of the parameters for the tool that they are creating. After creation the tool will be stored to the Tool Library in the next available slot for that tool type.
- **Select Tool:** When the Manual Tool button is selected the Select Tool option will become available to the user. When this button is clicked the Select Tool dialog will appear allowing the user to select a tool from the tool library. The tools displayed are only those tools that are available for that operation.
- **Tool Type:** This field is determined by the system or by the selected tool. If a Tool Description was input for a tool in the library it will also appear in this field (i.e. "**ENDMILL ROUGH - My Tool Description**").
- **Diameter:** This field allows the user to specify the overall diameter of the fluted portion of the tool.
- **Corner Radius:** This field is used to indicate whether the tool is a BullNose, Ball or Flat Endmill. A Ball Endmill is determined by setting this field to half of the input diameter. Flat Endmill is determined by leaving this field set to 0. BullNose is determined by setting this field to something greater than 0 and less than half of the tool diameter.
- **Tool Number:** This field is automatically set when using System Tool. When using a Manual Tool this field is determined by the user. When using Select Tool this field is determined by the value previously setup in the tool library.
- **Flute Length:** This field indicates the distance from the tip of the tool to the end of the flute or beginning of the tool shank.
- **Number of Flutes:** This field indicates the total number of flutes on the tool.
- **Overall Length:** This field indicates the total length of the tool from end to end.

Feeds and Speeds

- **System Auto:** When this option is selected the feeds and speeds will be calculated using the Cutting Condition database as well as the Tool *Diameter* and *Number of Flutes*.
- **Manual:** When this option is selected the *Spindle Speed*, *Z-Speed*, *Cutting Feed* and *Z-Feed* input fields will be made available for the user to input values. The input values will be output in the program for the specified tool.
- **SFPM:** This field indicates the Surface Feed Per Minute for X/Y and X/Y/Z tool motion as specified within the Cutting Conditions table.
- **Spindle Speed:** This field indicates the rotation speed of the spindle for X/Y and X/Y/Z tool motion.
- **Z-SFPM:** This field indicates the Surface Feed Per Minute for Z Plunge moves, as specified within the *Cutting Conditions* table.
- **Z-Speed:** This field indicates the rotation speed of the spindle for Z plunge moves.
- **Feed per Tooth:** This field displays the chipload per tooth for X/Y and X/Y/Z tool motion as specified within the *Cutting Conditions* table.

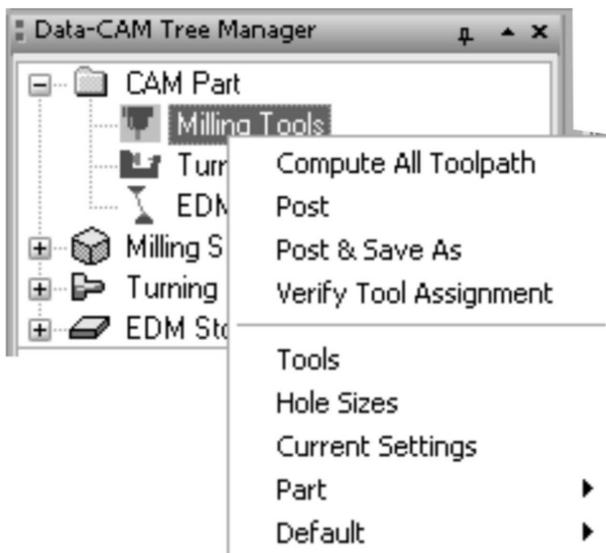
- **Cutting Feed:** This field displays the feedrate for X/Y and X/Y/Z tool motion for the selected tool.
 - **Z-Feed per Tooth:** This field displays the chipload per tooth for Z Plunge moves as specified within the *Cutting Conditions* table.
 - **Z-Feed:** This field displays the feedrate for Z plunge moves for the selected tool.
 - **Spindle Speed %:** This field is used to override the system generated spindle speeds for the selected tool. The user is permitted to edit this field when using *System Tool* only.
 - **Cutting Feed %:** This field is used to override the system generated feedrates for the selected tool. The user is permitted to edit this field when using *System Tool* only.
 - **Arc Slowdown %:** This field is used to control the feedrate of the tool as it makes circular movements and is relative to the *Cutting Feed* value. Values less than 100 in this field will cause the system to output a lower feedrate for arcs, while a value that is greater than 100 will cause the system to output a higher feedrate for arcs.
-

Milling Settings

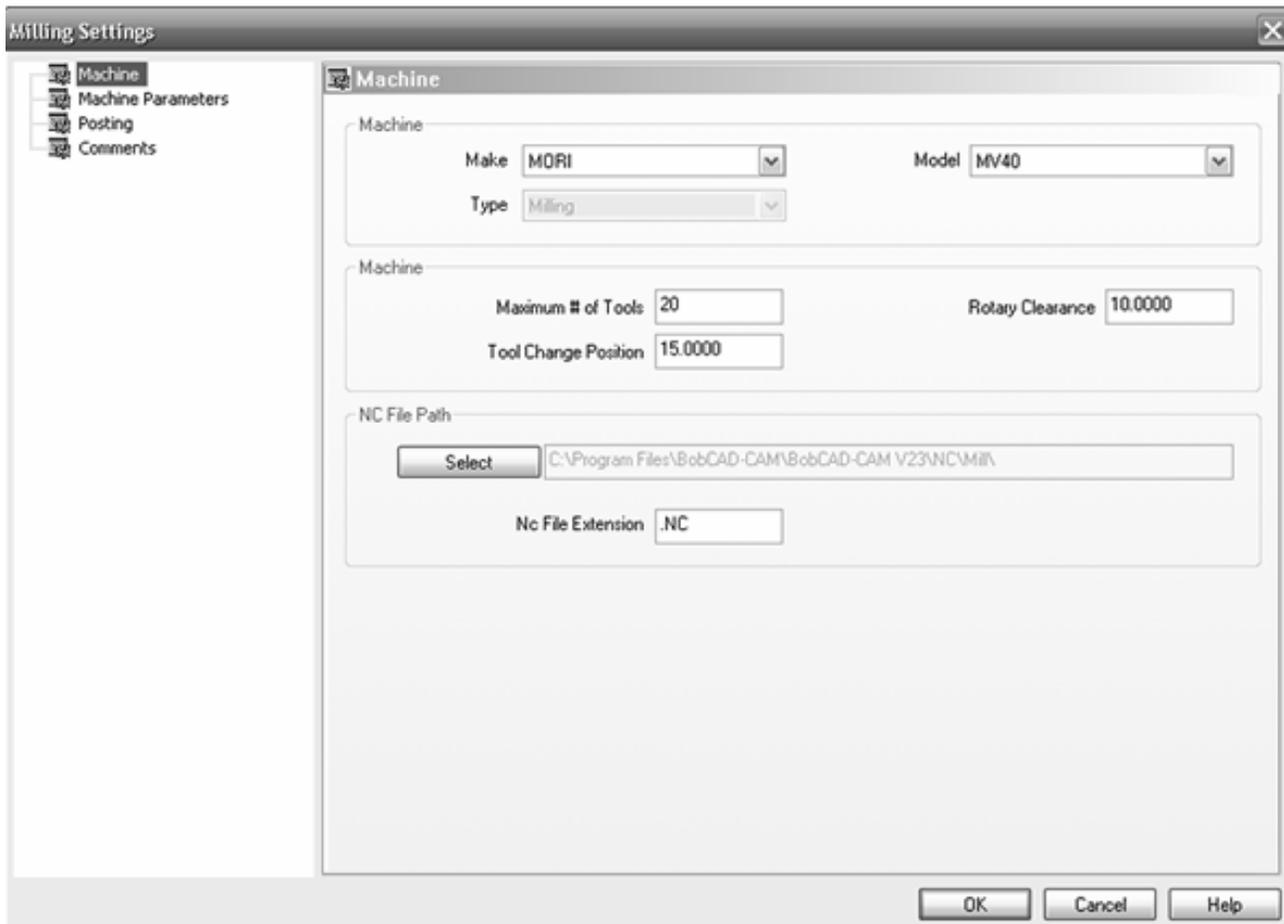
The **Milling Settings** dialog permits the user to control many of the options that may be present on the machine and customize program generation for the individual shop.

The settings in this dialog apply to the chosen machine only and are saved when the user clicks on OK below. Setting up this box to reflect what the machine is capable of is an important part of properly setting up BobCAD-CAM to work correctly.

To access this you will place your cursor on Milling Tools and Right-Click your mouse. Then click on Current Settings.



The Machine Page



Machine

- **Make:** Sets the make of the machine to narrow the machine choices to include only those produced by that OEM.
- **Model:** Sets the model of the machine in use and sets the post processor to use in the **Posting** page below.
- **Type:** Sets the type of machine that this is. The available choices are:
 - a. **Milling**
 - b. **Laser**
 - c. **Plasma**
 - d. **Waterjet**

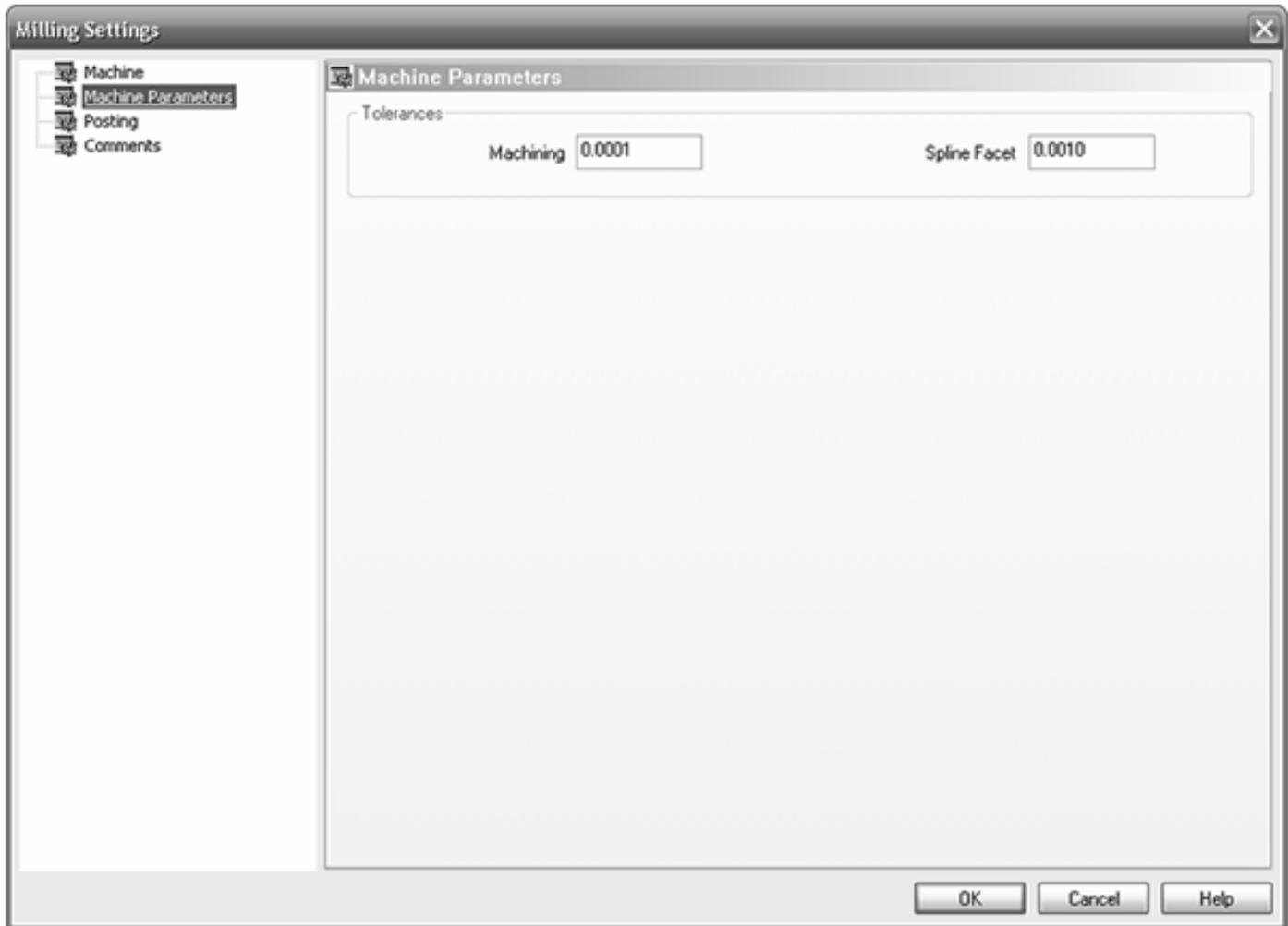
Machine

- **Maximum # of Tools:** Sets the maximum number of tools the machine can use in a single program.
- **Tool Change Position:** Sets the position BobCAD-CAM will rapid to before issuing a tool change command.
- **Rotary Clearance:** For use with machines with an indexing table, this setting sets the distance away to rapid the tool to allow for clearance when changing position.

NC File Path

- **Select:** Sets the folder that BobCAD-CAM will save the finished NC program to.
- **NC File Extension:** Sets the file extension that BobCAD-CAM will automatically add to the finished NC program file.

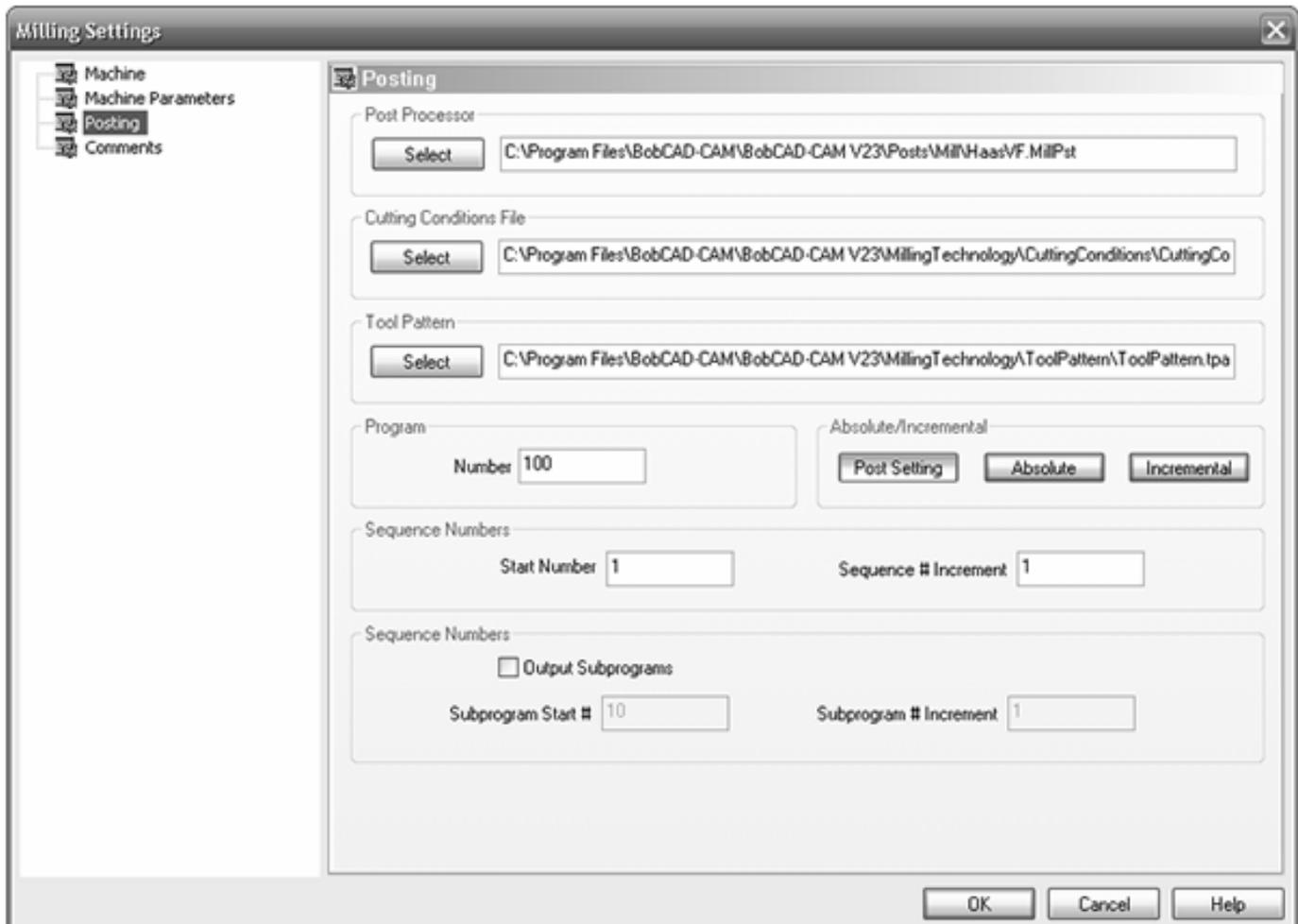
The Machine Parameters Page



Tolerances

- **Machining:** This sets the default tolerance to achieve in the NC program and toolpath. A larger number will result in less NC code but will also cause a poorer finish on the part. A smaller setting will improve the part, but may cause BobCAD-CAM to generate much more NC code.
- **Spline Facet:** When BobCAD-CAM breaks up spline entities into lines and arcs for machining, this setting governs how far away the resulting entities may be from the mathematically correct spline. A larger number results in less code but a much less accurate part, and a smaller number will produce a much better part but may generate more code than desired.

The Posting Page



Post Processor

- **Select:** Click this button to choose a new .MillPst post processor file to override the default post processor used for the current machine.

Cutting Conditions File

- **Select:** Click this button to choose a new cutting conditions (.ccond) file to override the default conditions assigned for use with the current machine.

Tool Pattern

- **Select:** Click this button to choose a new tool pattern (.tpatt) file to override the default tool patterns used for the current machine.

Program

- **Number:** Change this number to assign a different NC program number in the posted NC program.

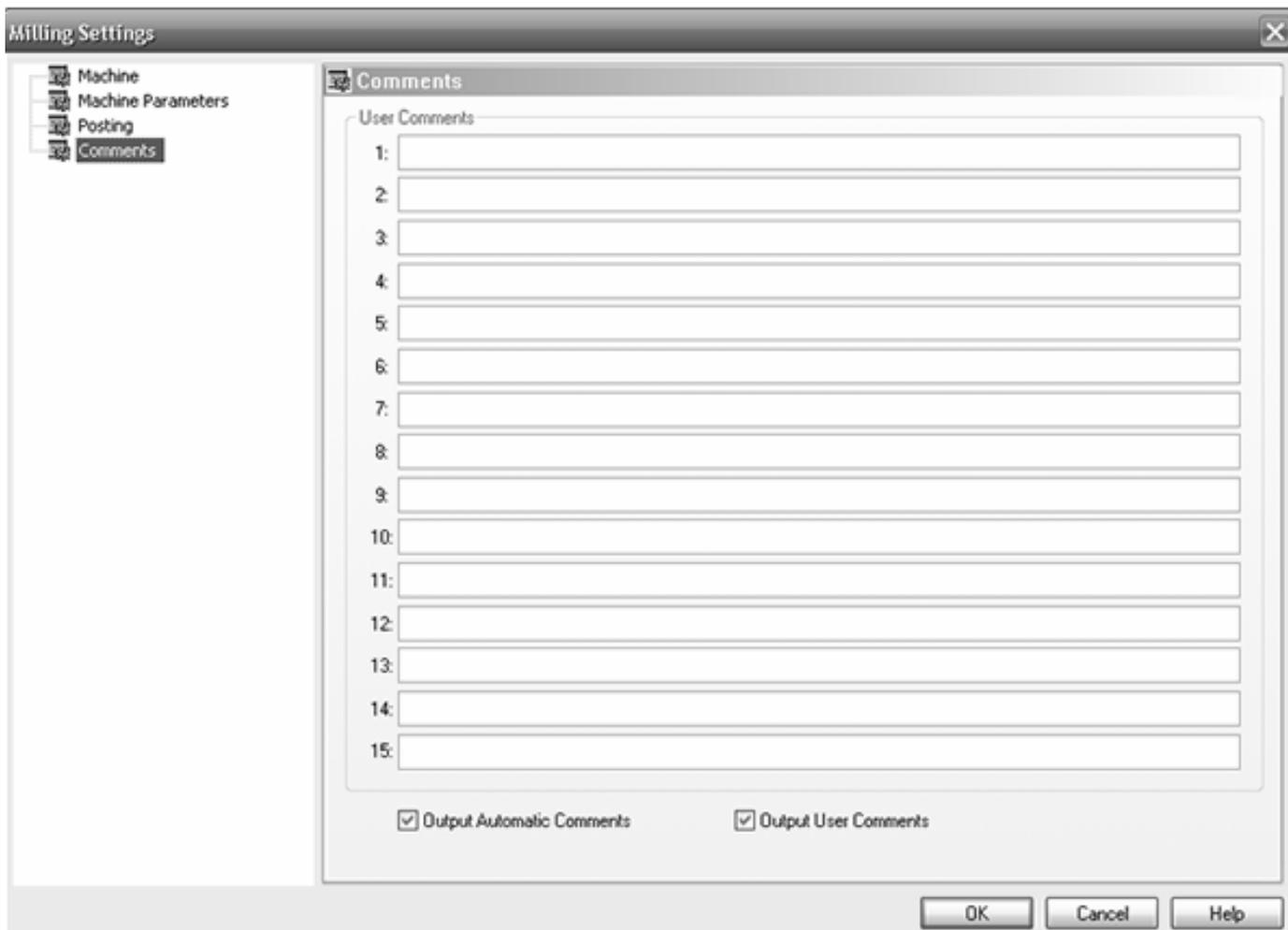
Sequence Numbers

- **Start Number:** The number in this field designates the starting line number for events in the NC program if the machine is configured to output them.
- **Sequence # Increment:** This sets the number to be added to each subsequent line number for the next line.

Sequence Numbers

- **Output Subprograms:** A check in this box will allow BobCAD-CAM to generate repetitive subprograms in the posted program, if the post processor is set up to do so, to save on machine memory.
- **Subprogram Start #:** This sets the first subprogram number to be used in the finished output.
- **Subprogram # Increment:** If more than one subprogram is to be generated, this number sets the number to be added to the previous subprogram number for the next number.

The Comments Page



User Comments

- **#1-#15:** This field allows the user to add up to additional comments to be displayed in the program at various points. **Note** that the post processor configuration file must be set up to specify where each comment should appear in the posted output.
- **Output Automatic Comments:** If this is checked, the system will output the normal feature comments into the program.
- **Output User Comments:** If this button is set, the comments entered into this page will be included in the posted program.

Choosing a Post Processor

The **Post Processor** item under the individual stock types in the CAM tree displays the post configuration file that has been selected for that stock. The post processor defines the "language" of the posted NC program.

You can change the post processor at the stock or by using the following method:

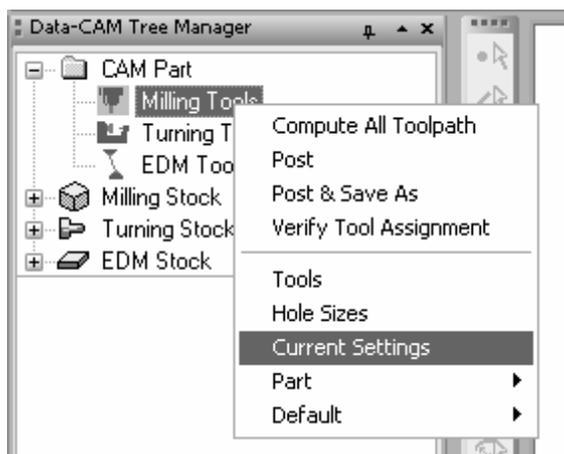
1. Click on the CAM Tree tab of the Data-CAM Tree Manager.
2. Click on the + icon next to the desired stock to open it and display the items underneath it in the tree.
3. Right click on the name of the current post processor file.
4. Click on the Edit item in the pop-up menu. The Set Post Processor dialog will appear.
5. Click on the Select button. An Open dialog box will appear.
6. Select the path and post processor configuration file to use for this stock, then click OK.

NOTE: The post processor configuration files must have an extension of .MillPst for milling machines, .LathePst for lathes, and .EDMPst for wire EDM machines.

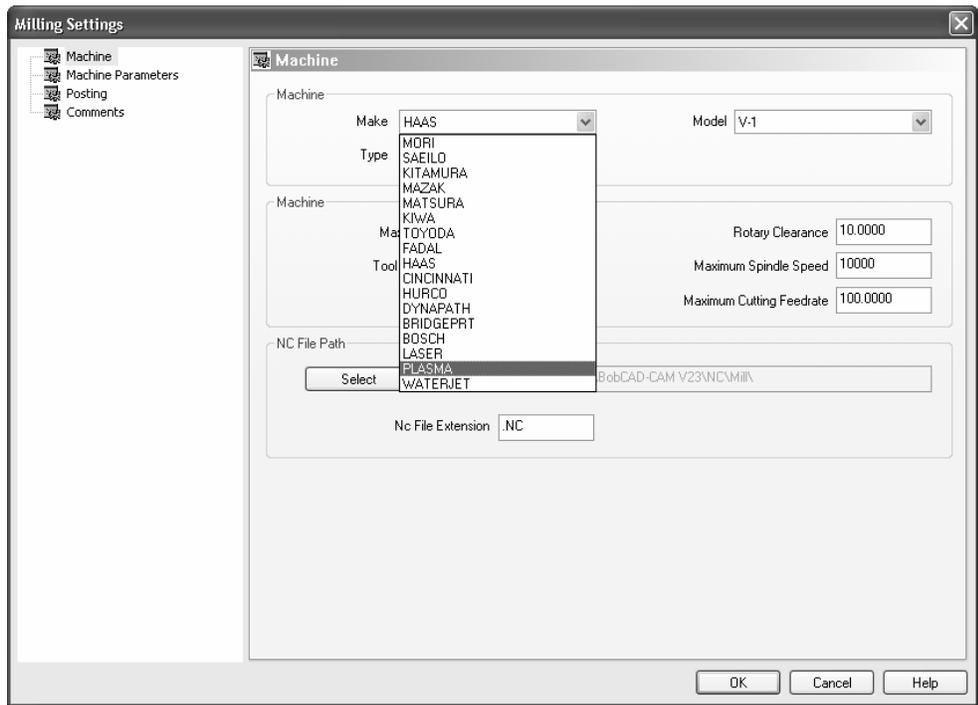
Setup for Plasma, Laser and WaterJet

Changing your machine configuration to a Laser, waterjet or Plasma machine is simple. Here is how to do this:

1. Click the CAM Tree tab of the Data-CAM Tree-Manager to access the CAM tree.
2. Now click the small plus (+) symbol next to CAM Part at the top of the tree. Now you will see Milling Tools listed under CAM PART.
3. Place your cursor on Milling Tools and right-click your mouse and choose Current Settings from the pop-up menu.

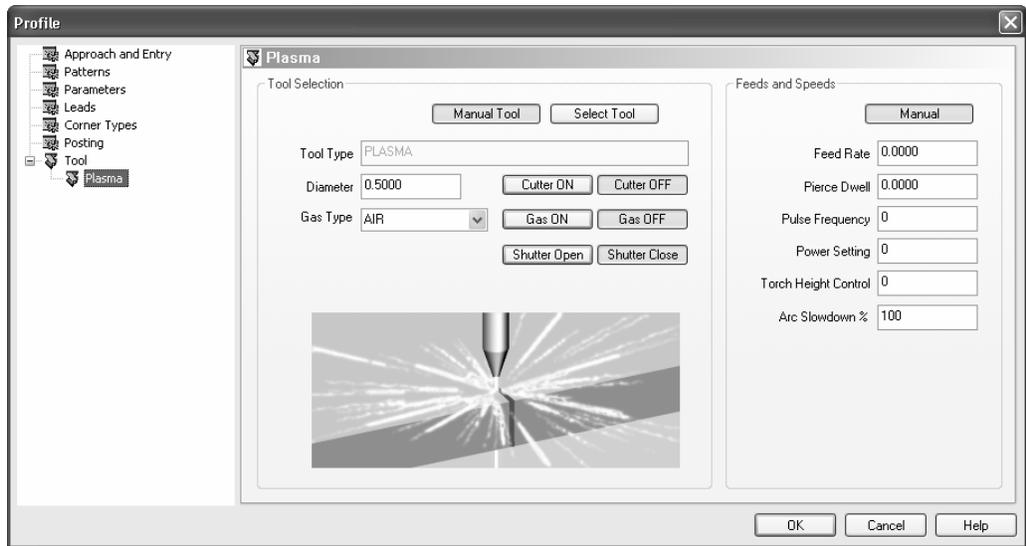


4. This brings up the Milling Settings dialog.



The options in the left side of the box are Machine, Machine Parameters, Posting and Comments. Make sure you select “Machine” at the top. You will see the word “Make” in the Machine category. Click the drop down menu and choose Plasma or Laser or Waterjet, whichever you need from the drop down menu and then you can click the OK button to set the software machining feature dialog boxes for that type of machine.

5. Now when you select Plasma as an example, the only 2 Axis option that you will find in the tree will be Profile. The Profile Edit dialog will give you the available parameters you need when you are ready to edit the profile operation when loaded into the tree.



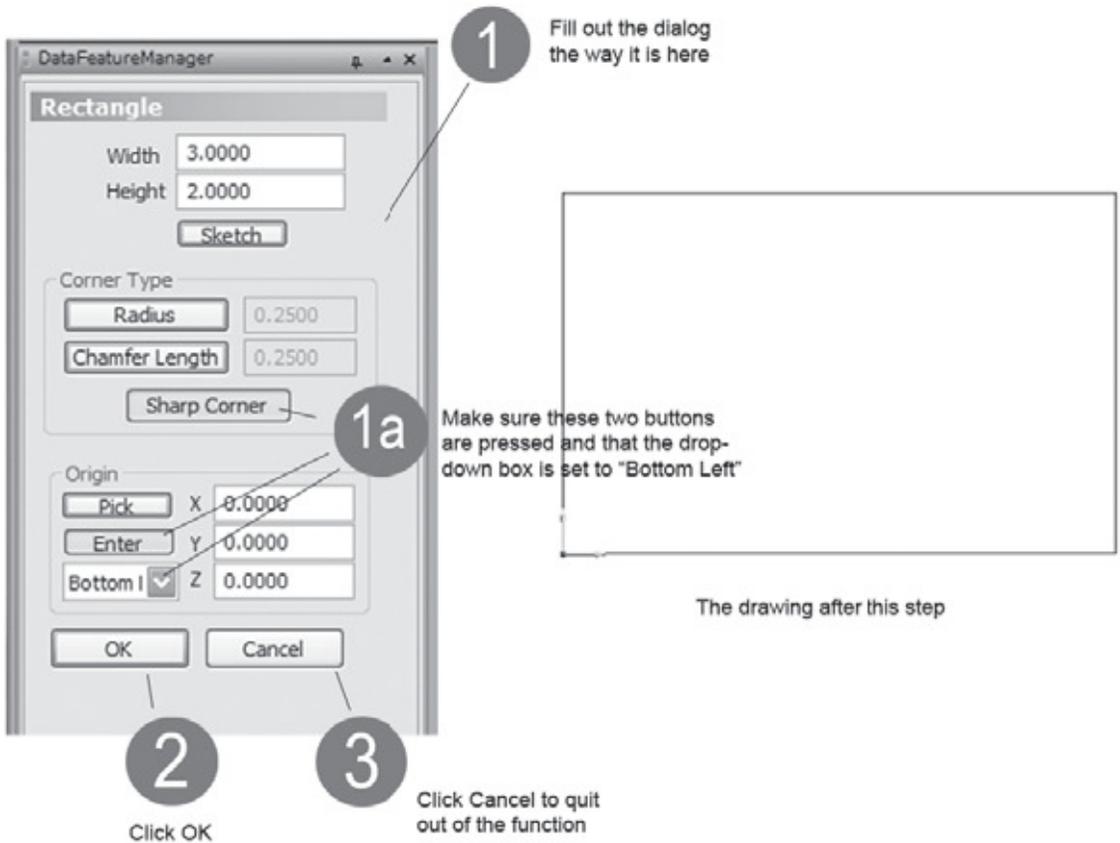
2.2 Mill Lesson

BobCAD-CAM can draw with wire frames and with solids and surfaces. Generally speaking, wire frames consist of lines, arcs, points, and a few special items like fillets, gears, and ellipses. Solids have 5 different primitive types: cube, sphere, cone, cylinder, and torus. Nearly any combination is possible, and many are often combined to produce unusually shaped surfaces. See the Help system in the software for a full explanation of each of these. All of the functions used in this section are available in all levels of BobCAD-CAM.

Drawing The Part

Do the following lesson and learn more about how to get started in making basic drawings.

1. Begin with an empty drawing by using any one of these 3 methods:
 - e. Click on File in the main menu and then New OR
 - f. Click on the New File icon in the toolbar OR
 - g. Type CTRL+N to start a new drawing.
2. Click on Other from the main menu, and then click on Rectangle.

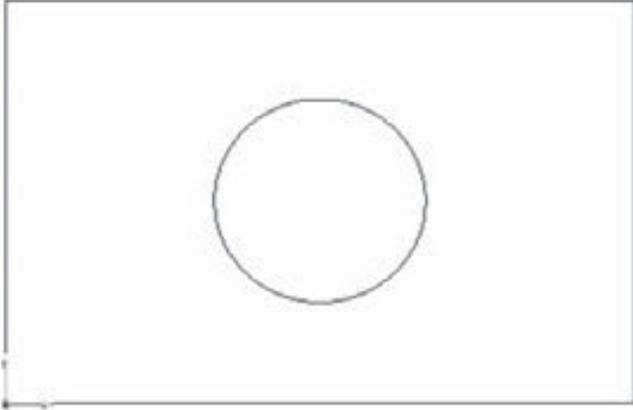


3. Click on **Arcs** from the main menu, then **Coordinates**.



1

Fill out the dialog as it appears here



The drawing should appear like this

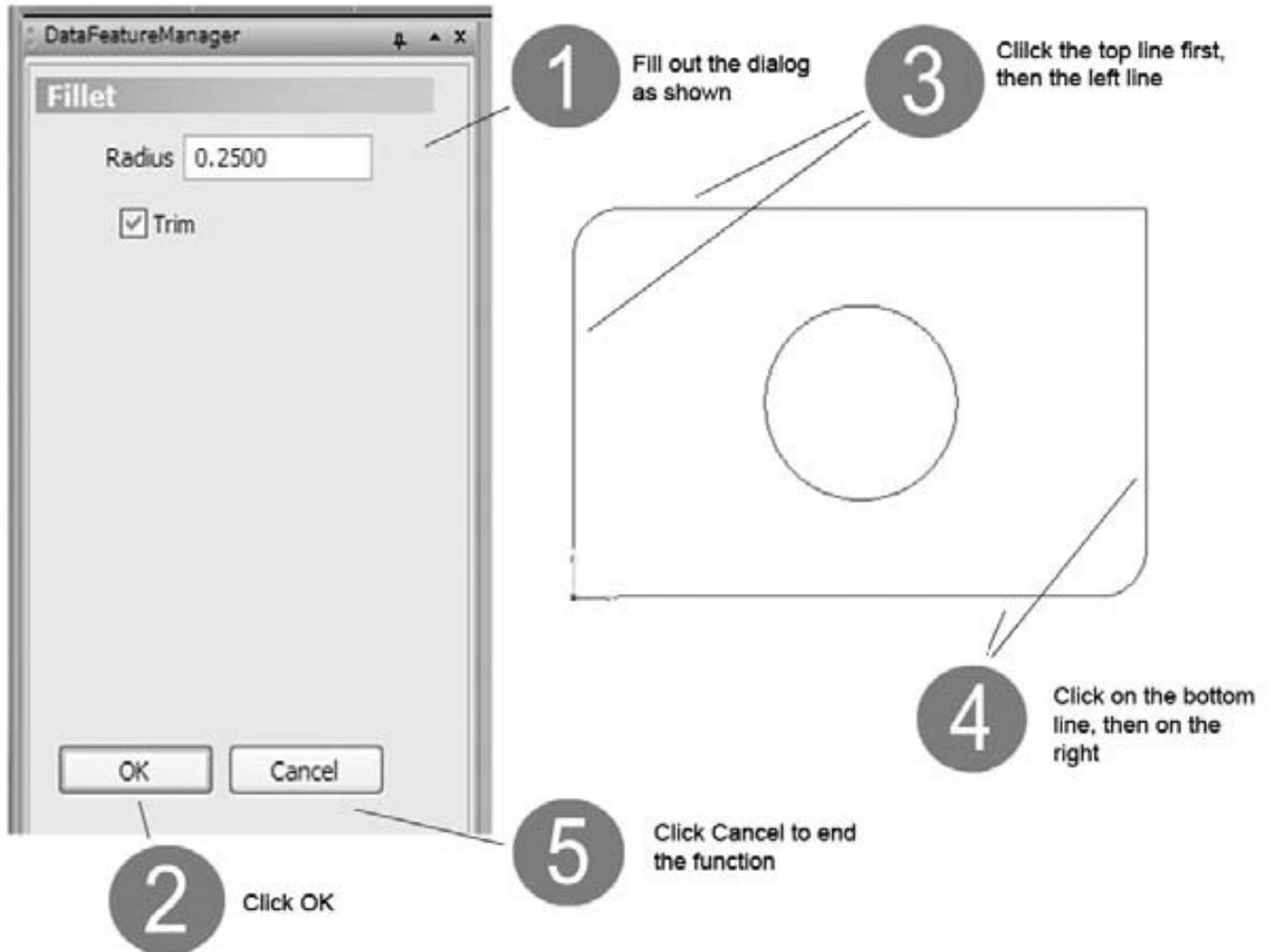
2

Click OK

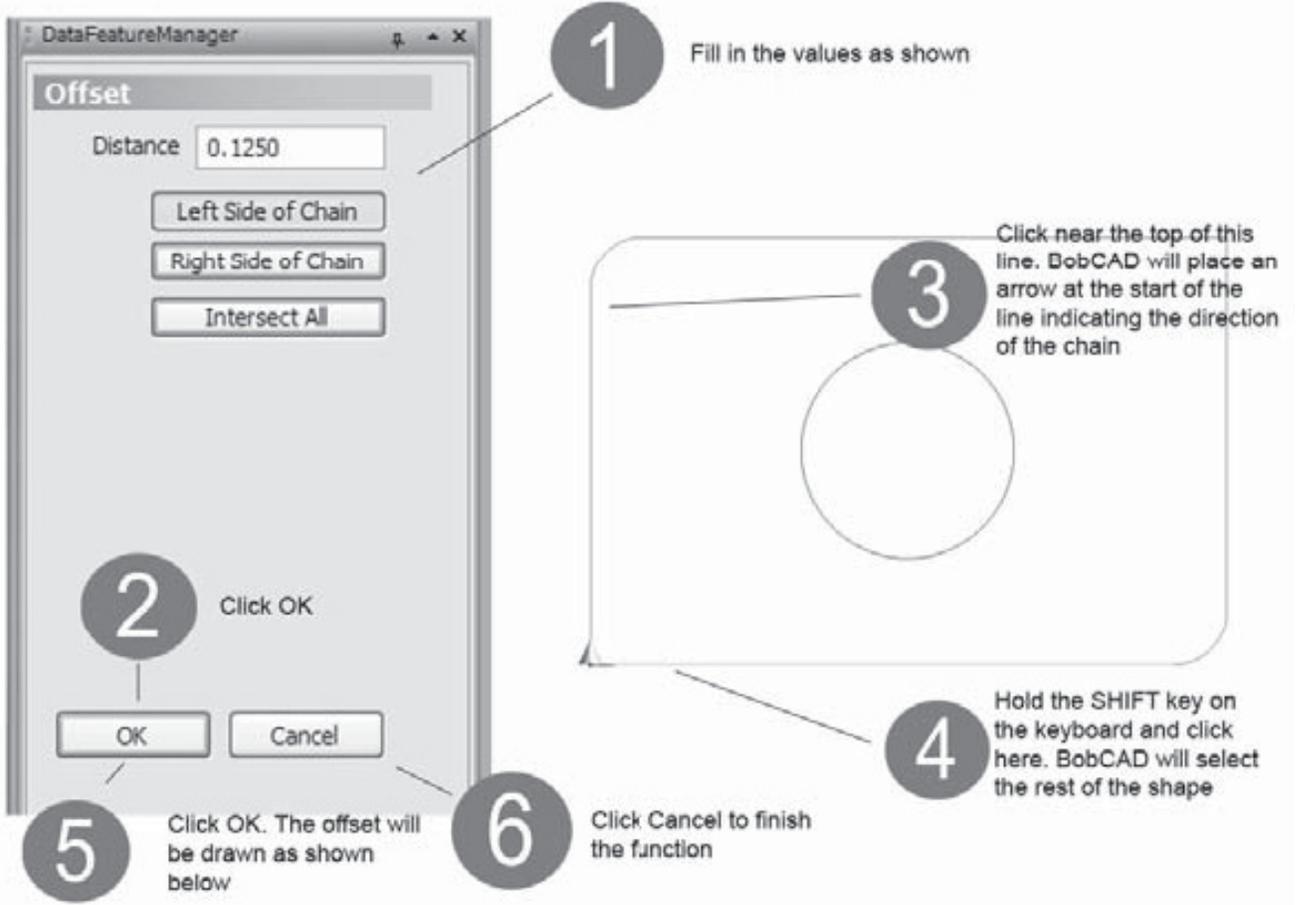
3

Click Cancel to exit the function

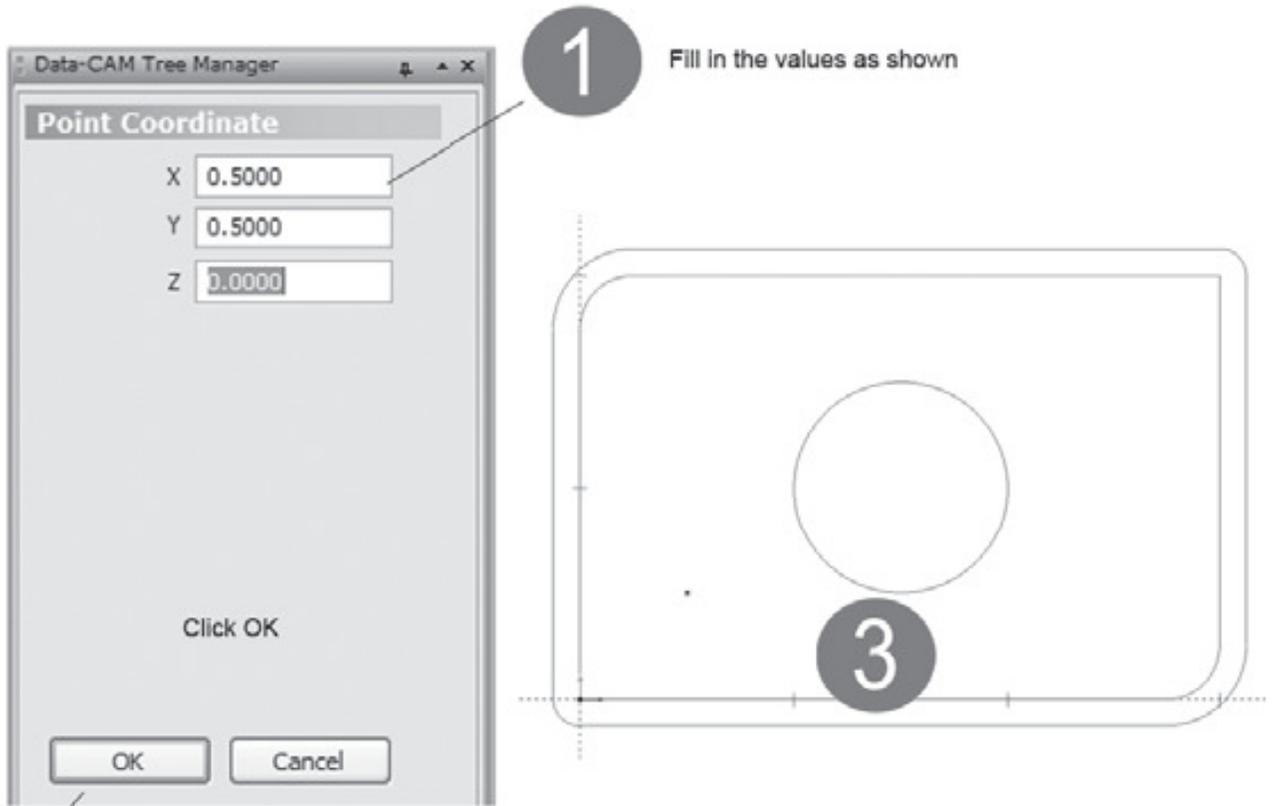
4. Click on **Arcs** from the main menu one more time, then on **Fillet**.



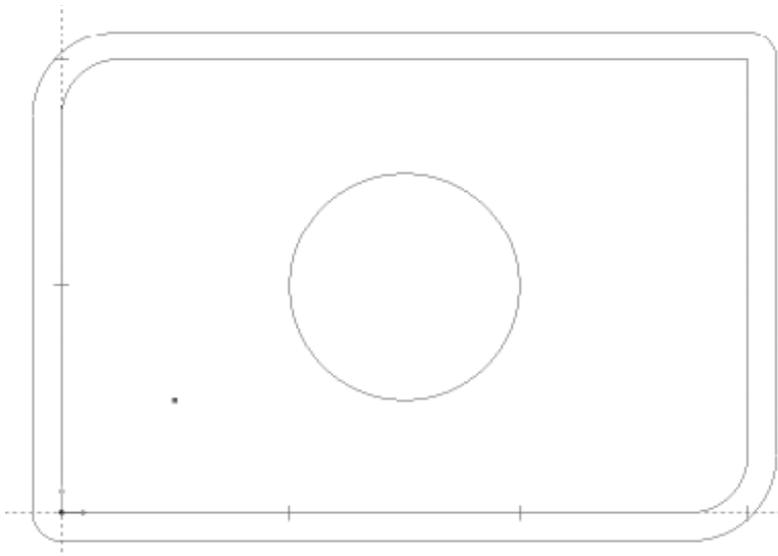
5. Click on **Other** in the main menu and then on **Offset**.



6. Click on **Points** in the main menu and choose **Coordinates**.



2 Click OK. The point will be drawn as shown to the right.

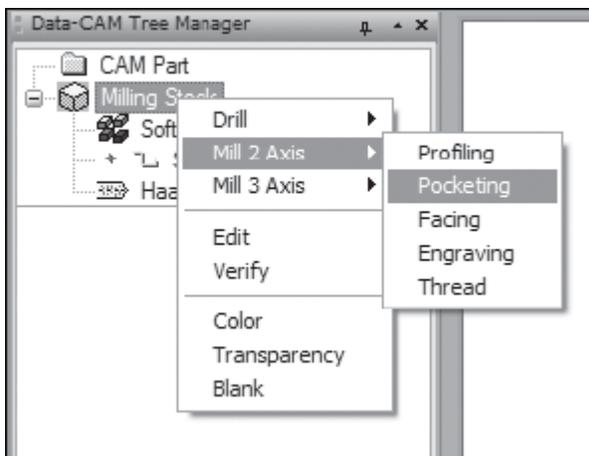


In the main toolbar at the top, click on this icon: . This will center the drawing in the view.

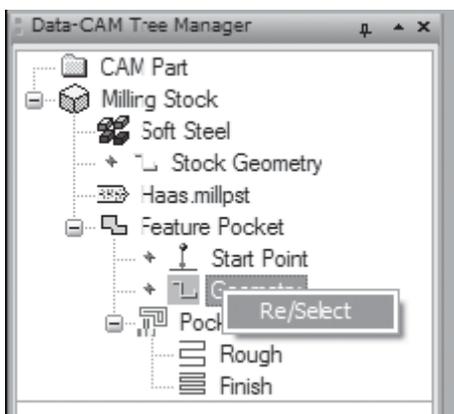
NOTE: BobCAD-CAM will automatically choose the tooling based on the operation, tool diameter, and corner radius specified in the operation's edit box. For milling operations this is straightforward, simply type in the diameter and corner radius needed and the system will take care of the rest. Drilling and tapping operations are more complex. BobCAD will automatically choose the correct center drill, spot drill, ream, countersink, and tap for the specified thread. It will call for tools that are ideal, which will not necessarily be present in the tool library. It is important to note that it remains up to the user to actually place the correct tool in the collet or tool changer, and if a different tool is available other than the ideal tool called for, this is the time to perform the substitution.

Machining The Part

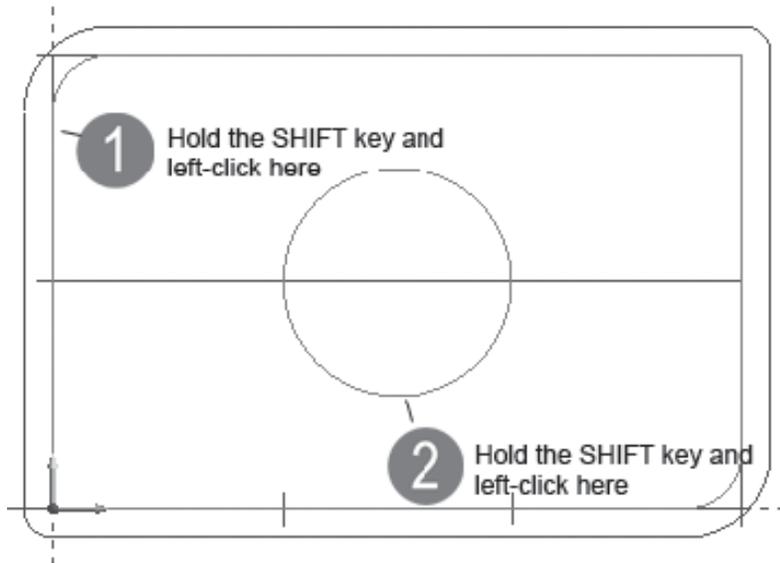
1. Click on the **Cam Tree** tab in the **Data-CAM Tree Manager**. Right-click on **Milling Stock**, choose **Mill 2 Axis**, and click on **Pocketing**.



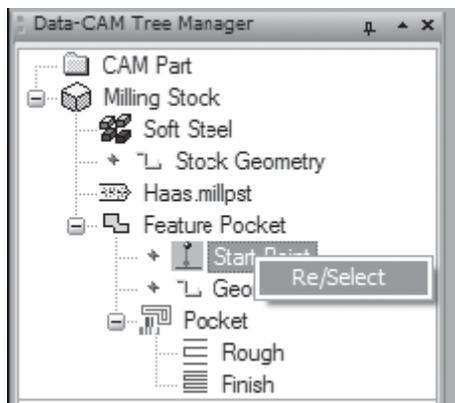
2. Select the shapes used for the pocket by right-clicking on **Geometry** and choosing **Re/Select**.



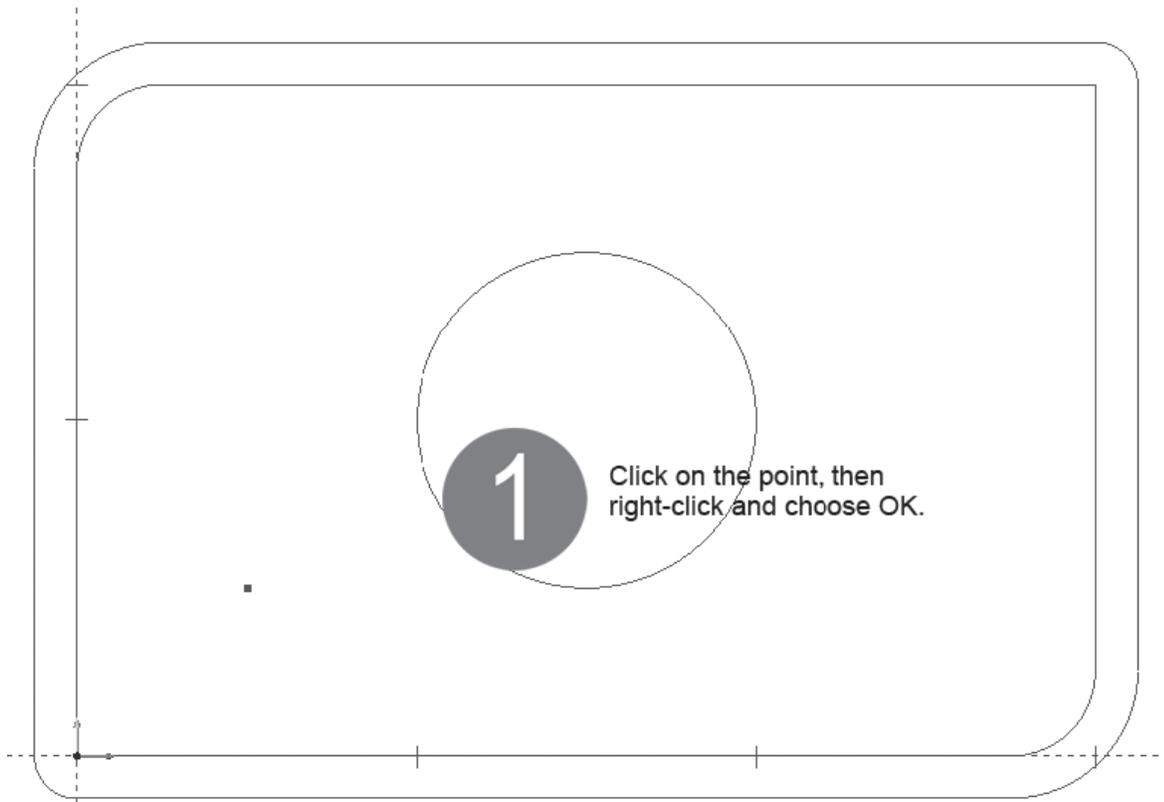
Click on the drawing as shown here:



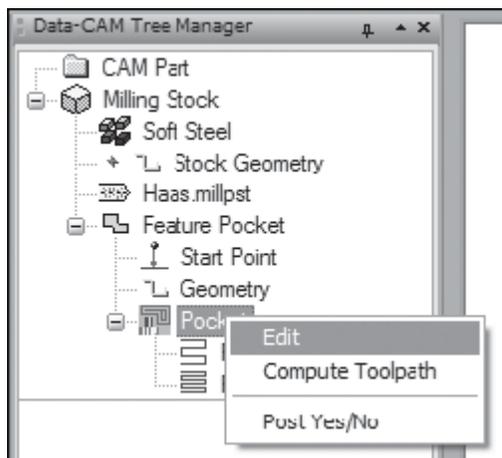
When the two shapes used here have been selected, right-click in the workspace and choose OK. Once OK has been clicked, BobCAD exits the selection mode and changes the tree to show that the geometry for this feature has been set. If the starting plunge position of the pocket does not matter, then the start point does not need to be set as BobCAD-CAM will automatically choose a plunge position. If it does matter, however, choose it now by right-clicking on Start Point and choosing Re/Select.



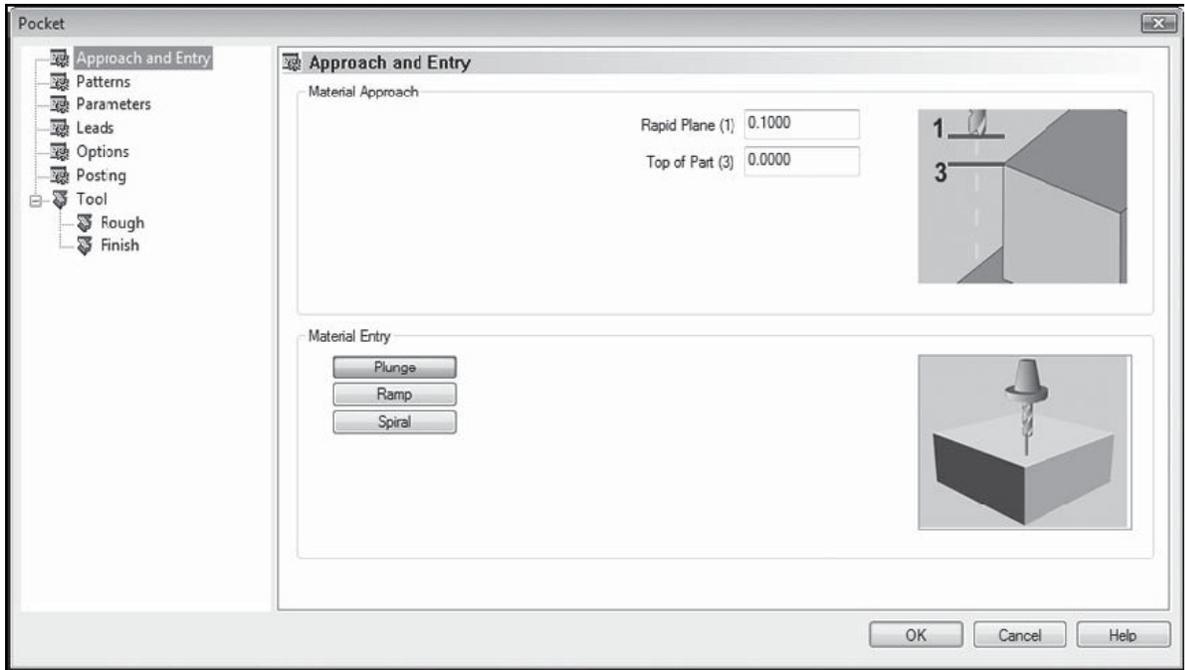
Click on the point on the drawing as shown below:



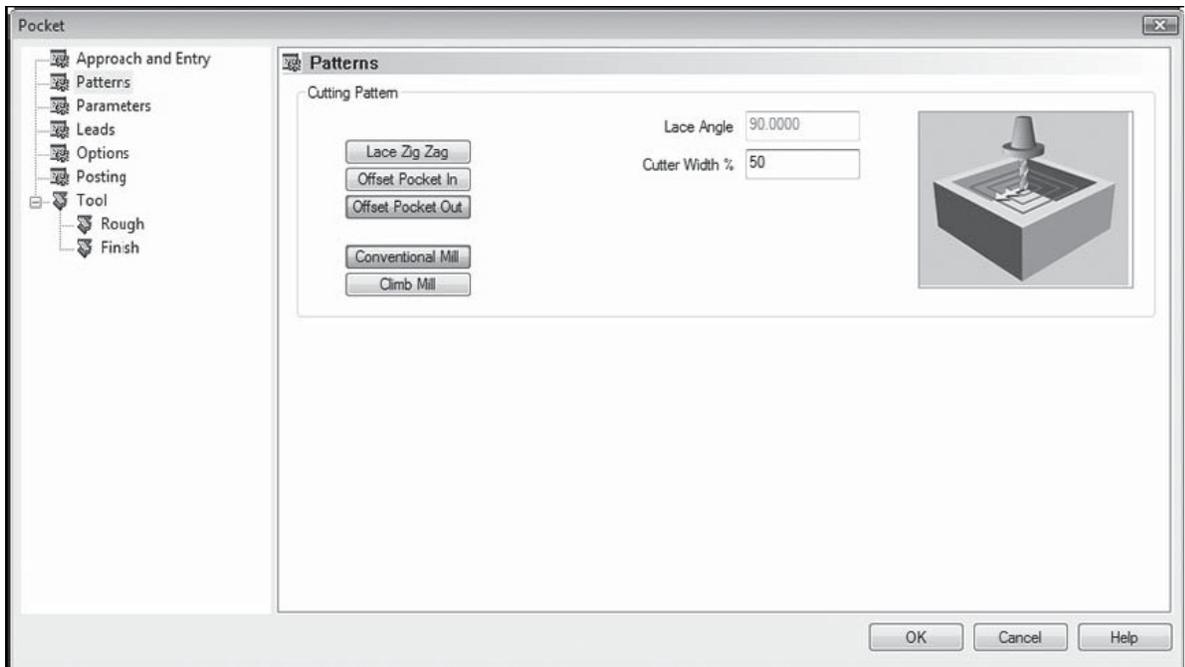
3. Click on **Pocket** under the **Feature Pocket** item in the tree and choose **Edit**.



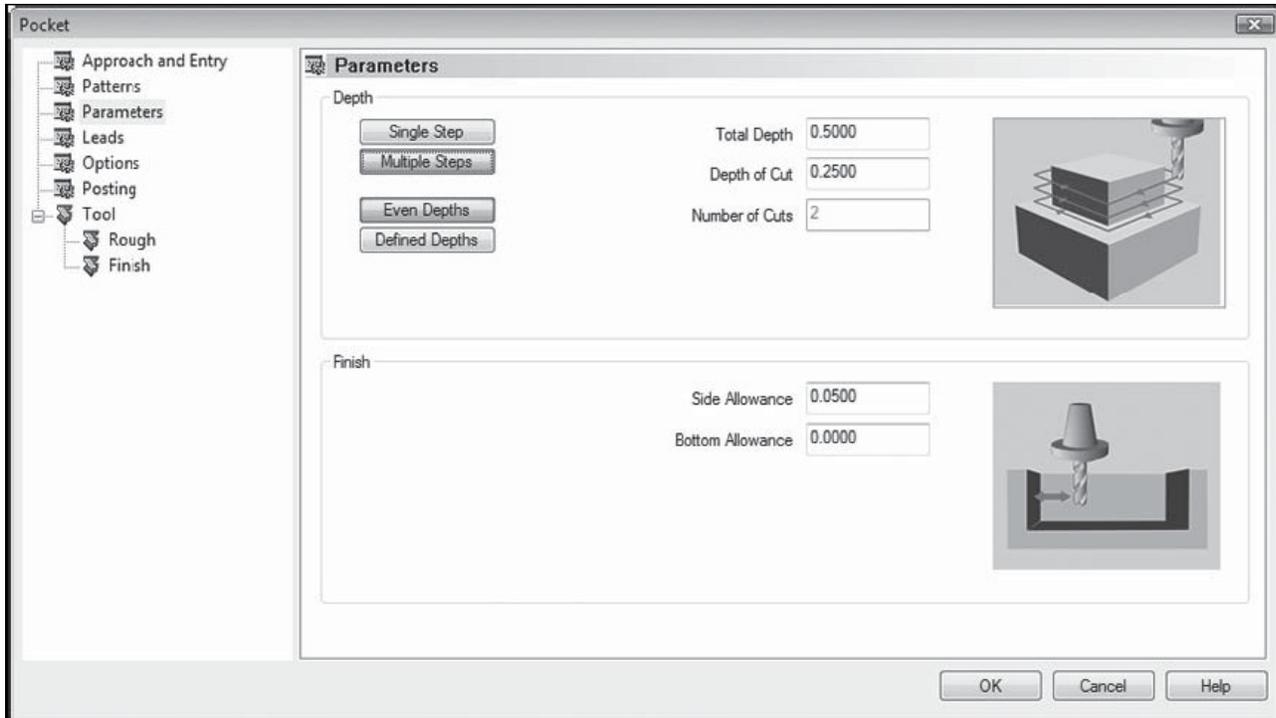
Click on **Approach and Entry** at the left of the box and fill out the rest of the information to appear like below. This example does not call for a ramped entry, though **Ramp** may be chosen if desired.



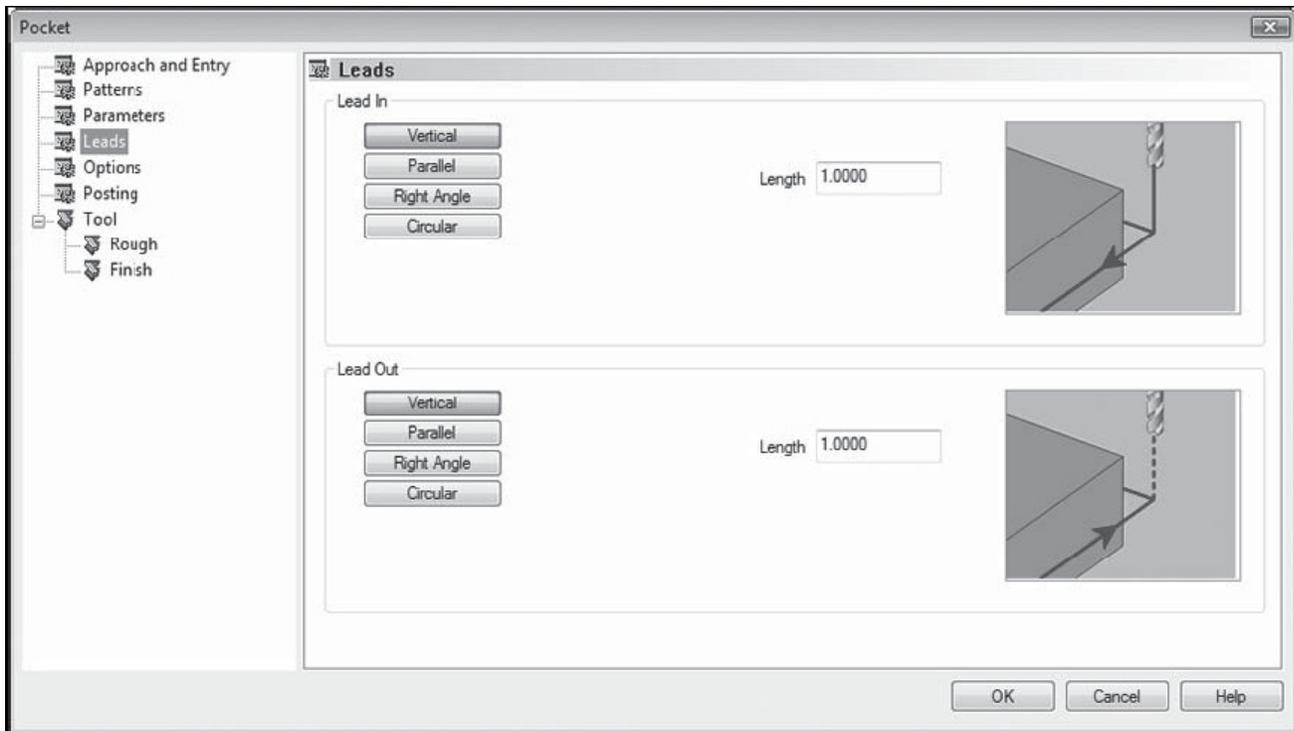
Click on **Patterns** and do the same thing. **Lace Zig Zag** pockets cut from side to side, removing material quickly but leaving scallops on the edges. **Offset Pocket In** and **Offset Pocket Out** do not leave these scallops, but may generate more code.



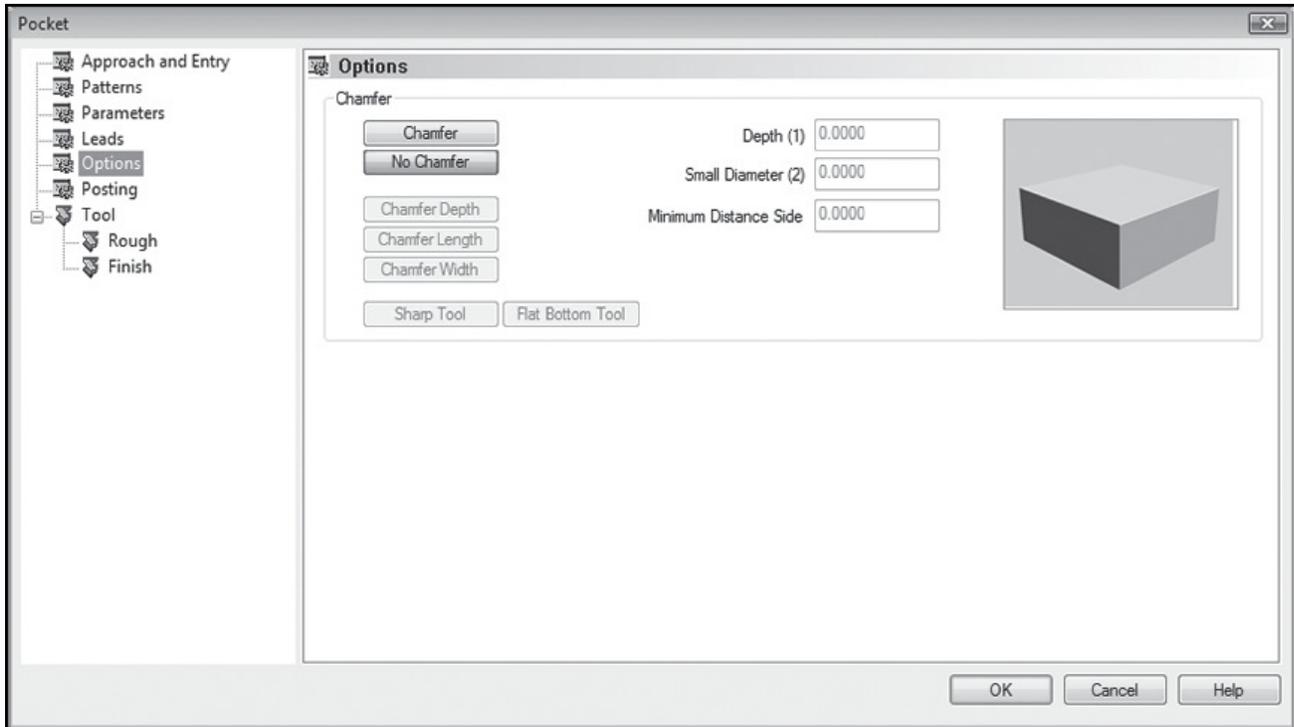
- Click on Parameters on the left and fill out the dialog. Since in this example the pocket is going to be .5 deep and later on a .5 tool will be specified, choose Equal Steps and set the Depth of Cut to .25. In this example, there will be a finish tool used so the side allowance is set to .05 in order to leave this much for the finish pass.



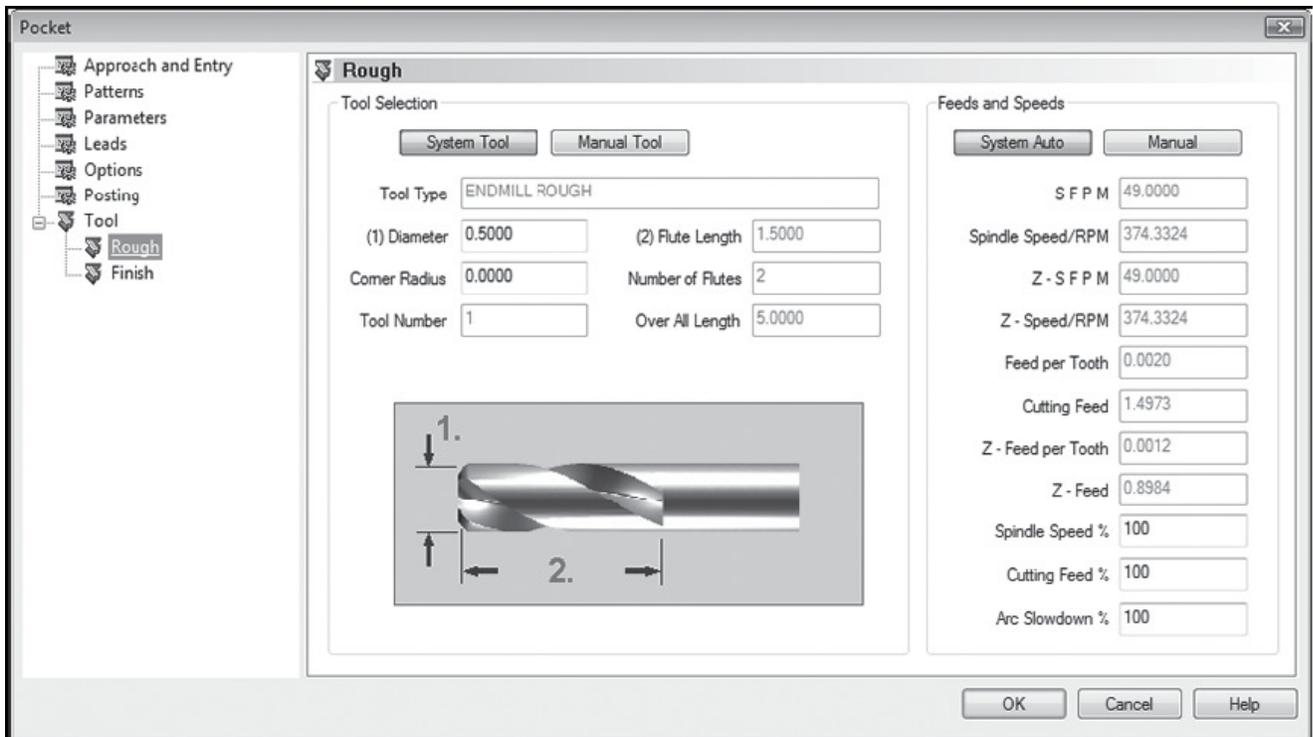
5. Click on **Leads** to the left. In the pocketing features, these parameters control the approach into the finish pass.



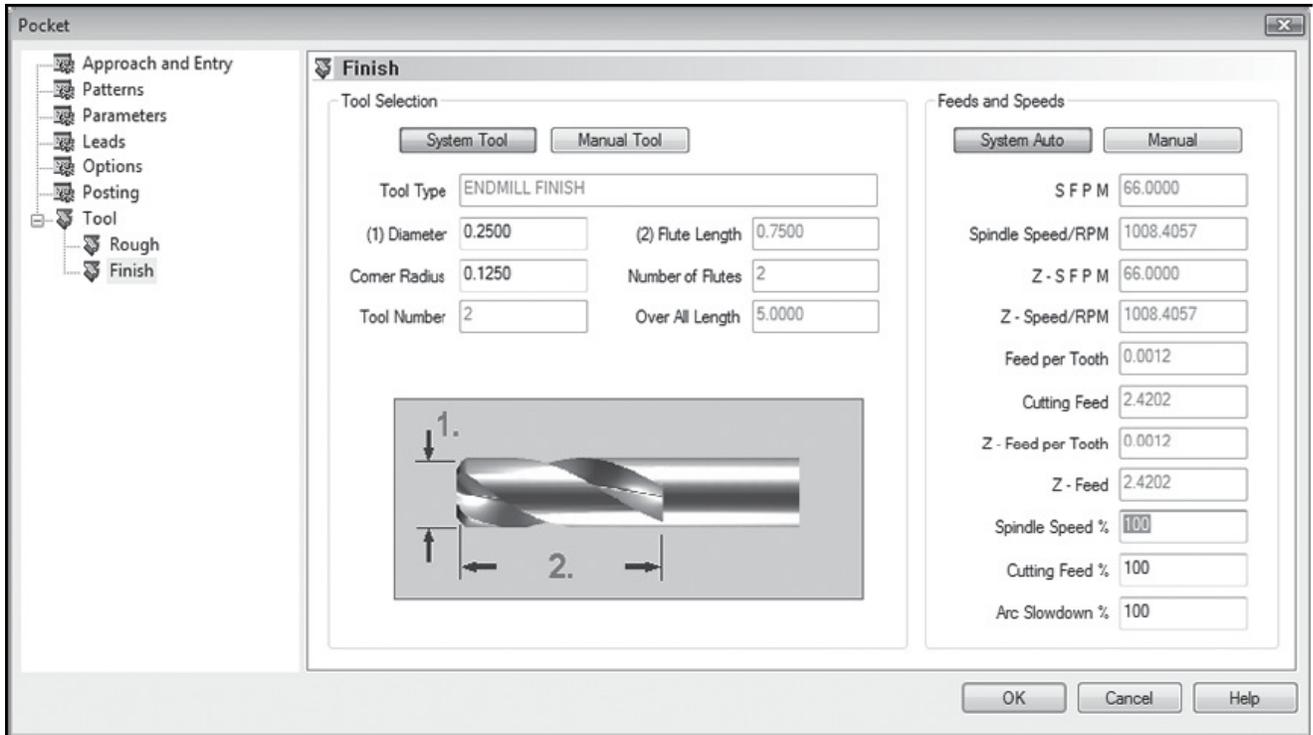
6. Click on **Options**. It will not be required to break the top edges of the pocket for this example, so leave the settings at the default **No Chamfer**.



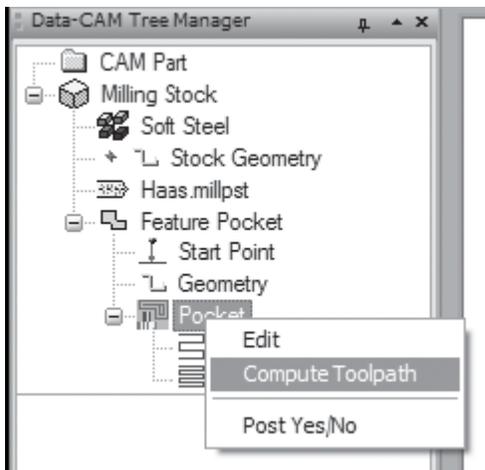
7. Click on **Rough** under **Tool** to set the tool used for the roughing passes. This example it set to a simple .5 diameter fl at milling tool.



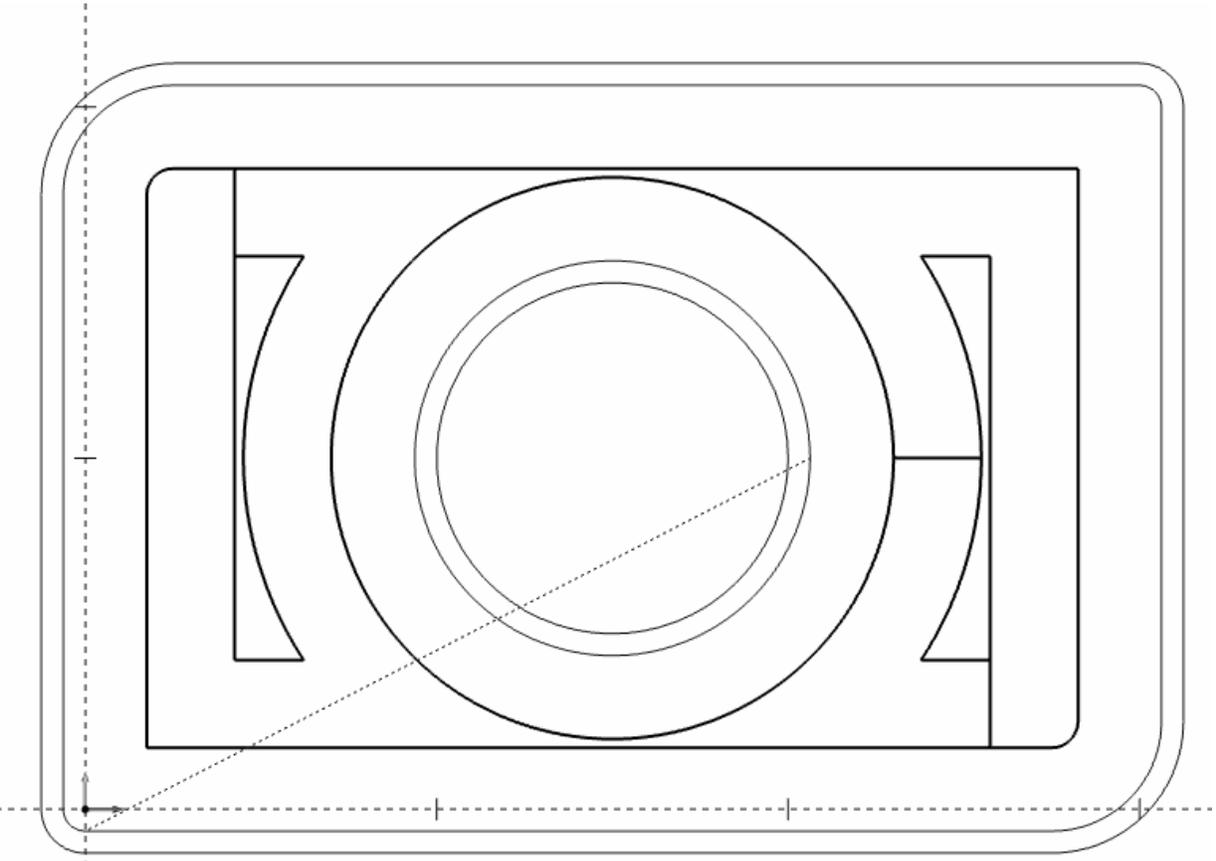
8. Click on **Finish**. This finish will use a .25 ball mill.



- Click OK at the bottom right. All of the settings for the pocket operation will be complete. Now Right-click on **Pocket** one last time and choose **Compute Toolpath**.



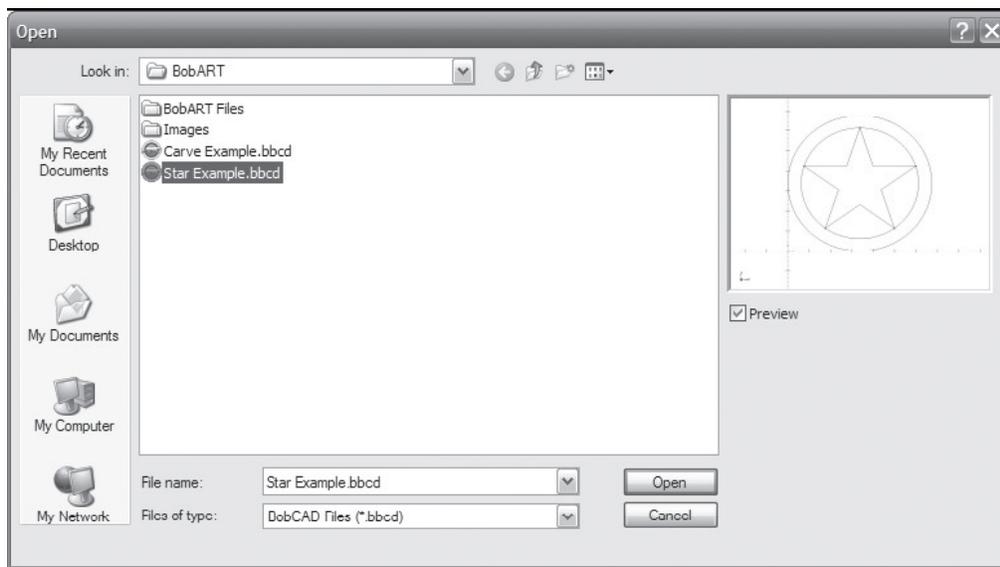
When that has completed, the finished pocket toolpath should look something like this:



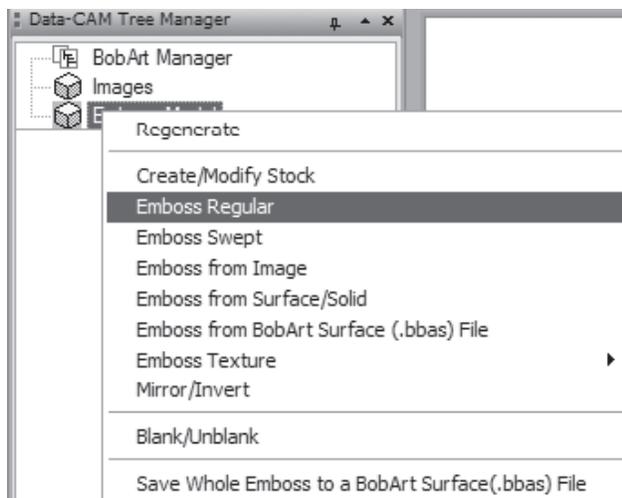
3D BobART Example

BobART is an available add-on module that can be purchased and added to the Version 23 CAD-CAM software. In addition to vectorizing bitmap images, BobART can also produce 3D embossed surfaces that can be machined like any other surface in the software. There are many ways of generating embossed surfaces in the software. This example will show you one method that the software can be used.

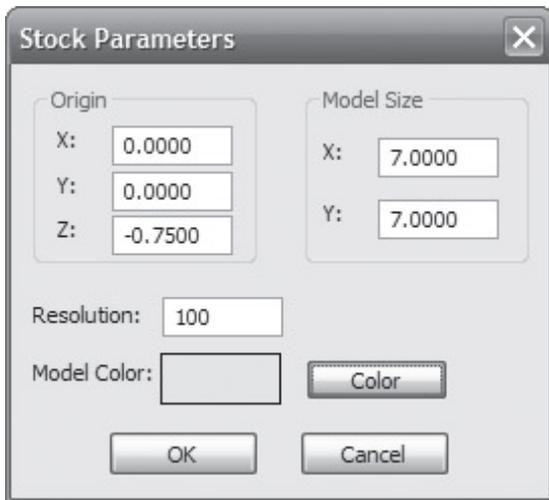
1. Open the “Star Example.bbcd” file in the BobCAD-CAM installation folder. Click on File in the main menu and then Open. Navigate the dialog box to the folder that BobCAD-CAM was installed into. The file will be directly under the Examples/BobART folder there. When you have located the file, click on it to highlight it and then click the Open button in the dialog. The file will open and the drawing will be displayed in the workspace.



2. At the bottom of the Data-CAM Tree Manager, click on the BobART tab to display the BobART tree. Right click on Emboss Model and choose Emboss Regular from the menu that will appear.



The new emboss feature will be inserted into the tree. Since no stock was yet defined, BobCAD-CAM will display the Stock Parameters dialog.



Input the following values:

Origin frame:

X: 0

Y: 0

Z: -.75

Model Size frame:

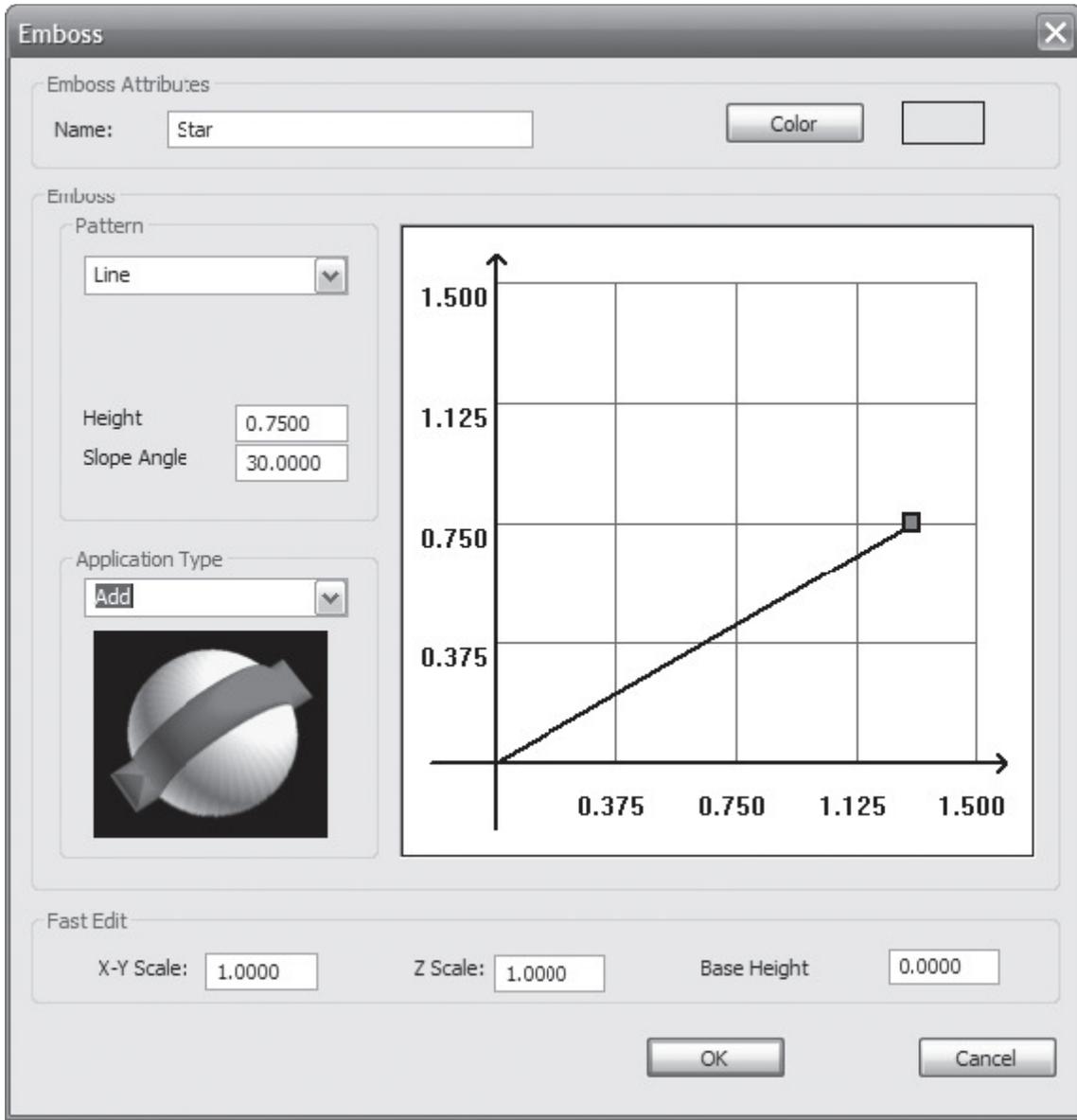
X: 7.0

Y: 7.0

3. Change the Resolution field to 100, and click OK. BobCAD-CAM will draw the defined stock in the workspace. The next dialog that will appear is the Emboss dialog. Change the Name field to something meaningful so that this operation can be easily located in the tree by the user. BobCAD-CAM permits the names of the features in the tree to be changed in order to make them easier to differentiate. Often, a complex set of emboss operations in the BobART tree will contain numerous features with the same name. The larger and more complex the operations get, the more difficult it can be to tell them apart.

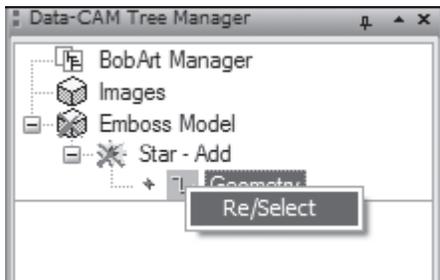
In the dialog, set these values:

- Pattern: Line
- Height: 0.75
- Slope Angle: 30.0
- Application Type: Add



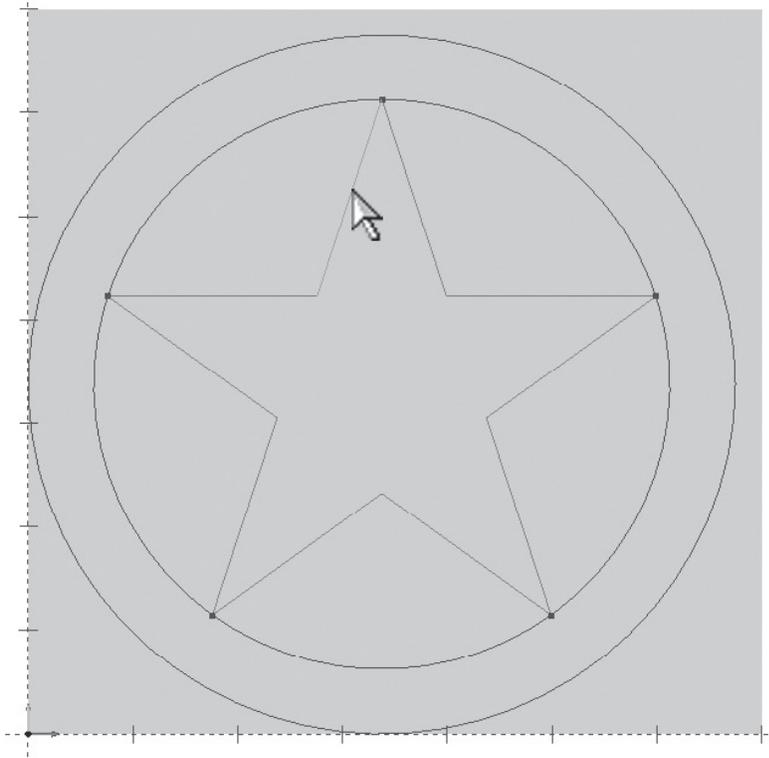
Click OK in the dialog and the box will disappear.

4. Right click on Geometry under the new feature named Star - Add. Choose Re/Select from the pop-up.



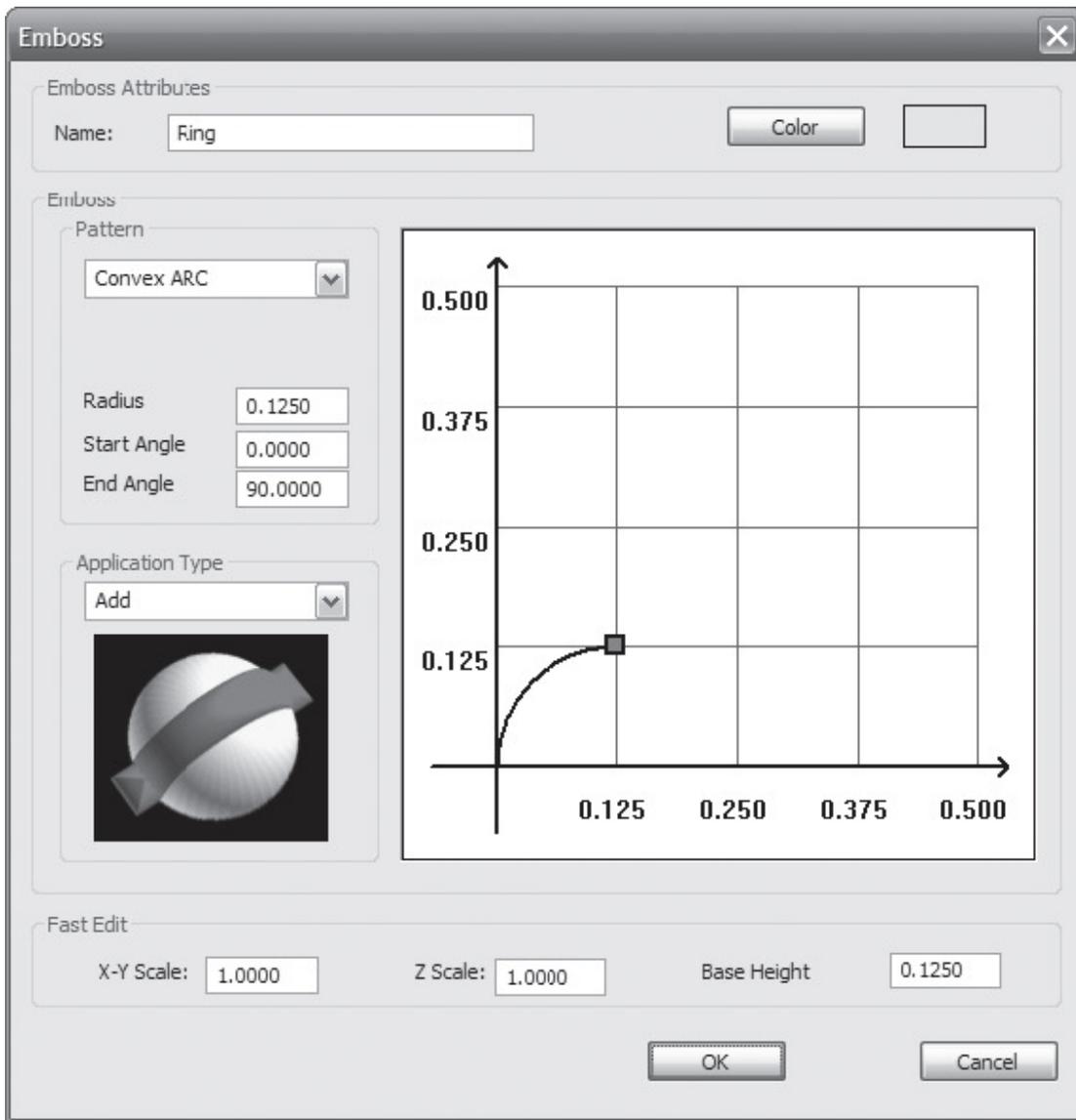
The software will be placed in Selection Mode. While holding the Shift key down on the keyboard, click anywhere on the star shape only in the drawing.

When it has highlighted all the way around, right click anywhere in the workspace and choose OK from the menu that will appear. You can also hit the space bar on your keyboard to indicate OK.



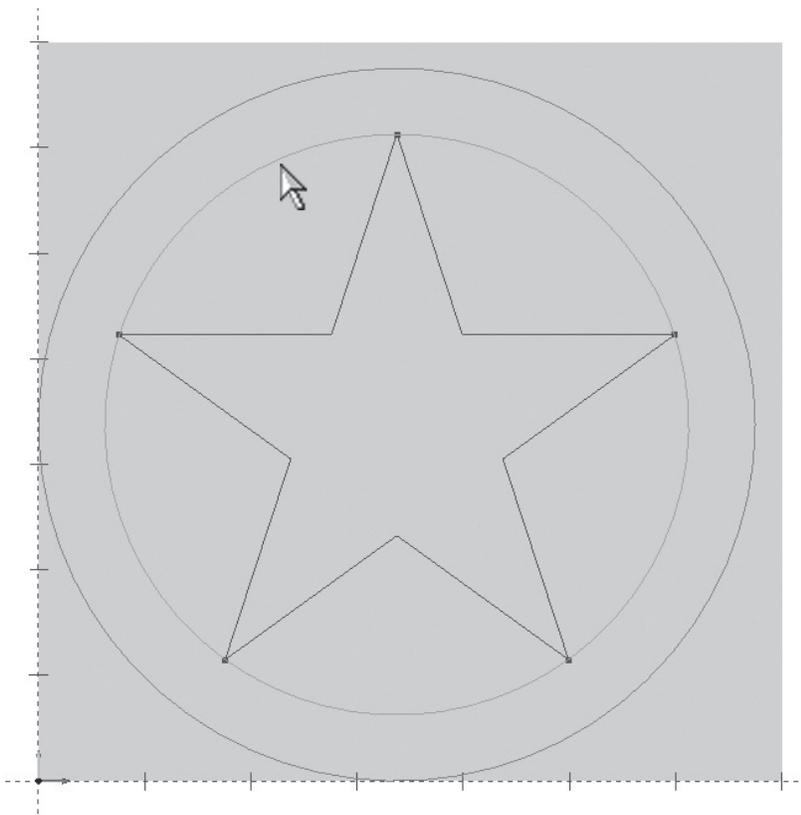
- Right click on Emboss Model and choose Emboss Regular from the menu that will appear again. Since the stock has already been defined, the box that will display first will be the Emboss dialog itself. Change these values in the dialog:

Name: Ring
Pattern: Convex Arc
Radius: 0.125
Base Height: 0.125
Application Type: Add



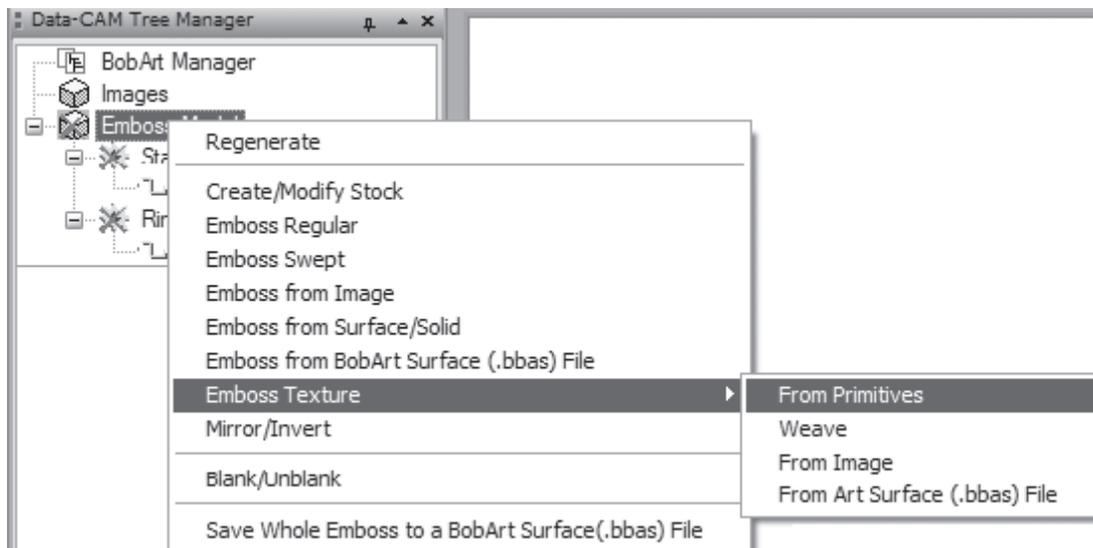
Click OK in the dialog and it will disappear. Right click on Geometry under the new feature named Ring - Add. Choose Re/Select from the pop-up.

- The software will be placed in Selection Mode. While holding the Shift key down on the keyboard, click anywhere on the two circles in the drawing.



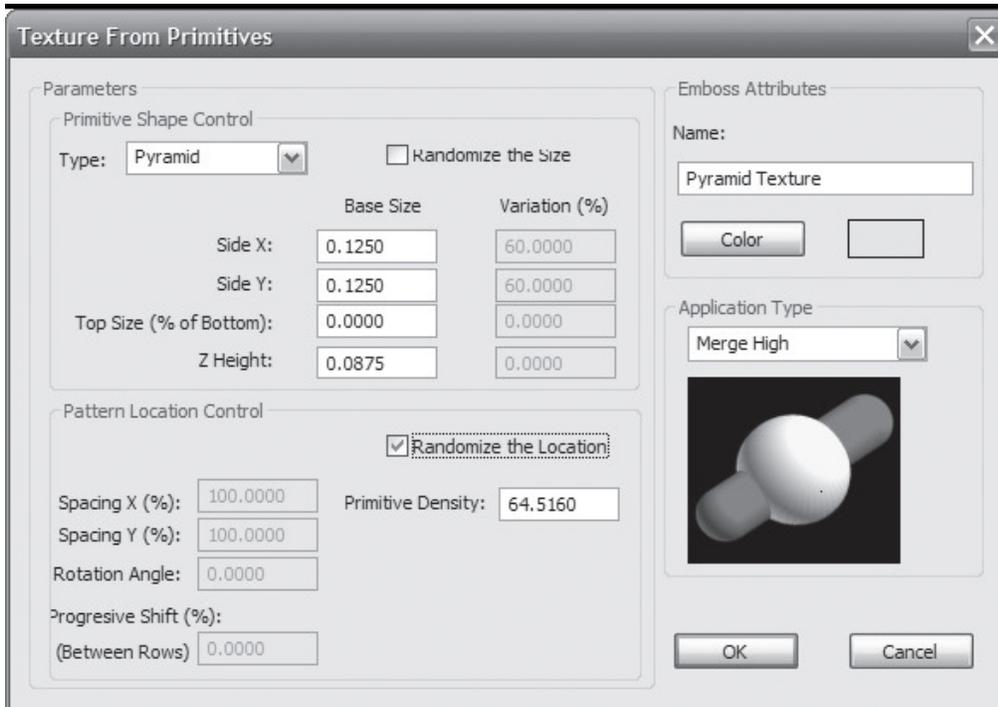
When both of them are highlighted, right click anywhere in the workspace and choose OK from the menu that will appear.

- Right click Emboss Model and hover the cursor over Emboss Texture in the menu. Choose From Primitives from the fly-out.



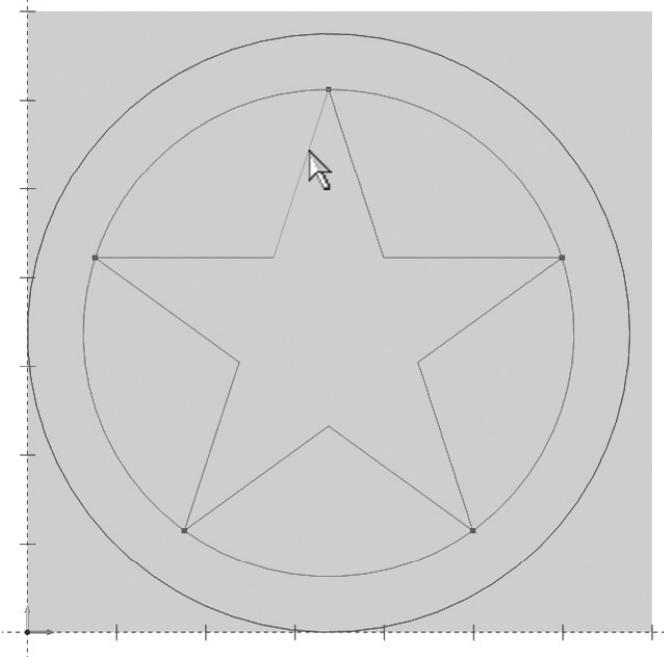
In the dialog that will be displayed, set these values:

Name: Pyramid Texture
Type: Pyramid
Side X: 0.125
Side Y: 0.125
Top Size(% of Bottom): 0
Z Height: 0.0875
Application Type: Merge High



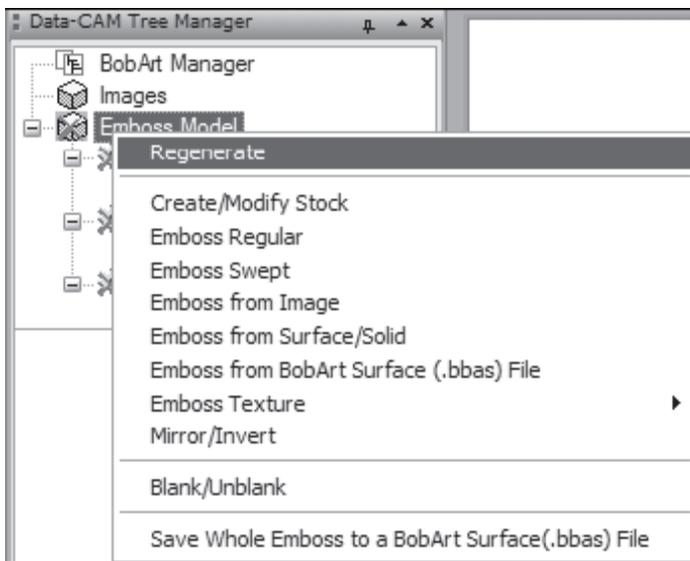
Click on the checkbox labeled Randomize the Location to have BobART scatter the primitives in the texture, then click OK. Right click on Geometry under the new feature named Pyramid Texture and choose Re/Select from the pop-up.

While holding the Shift key down on the keyboard, click anywhere on the the inside circle only, and click anywhere on the star shape in the middle.



When both are highlighted all the way around, right click in the workspace one last time and choose OK to set the selection.

The emboss itself is actually complete. To see the results, right click on Emboss Model in the tree and pick Regenerate.

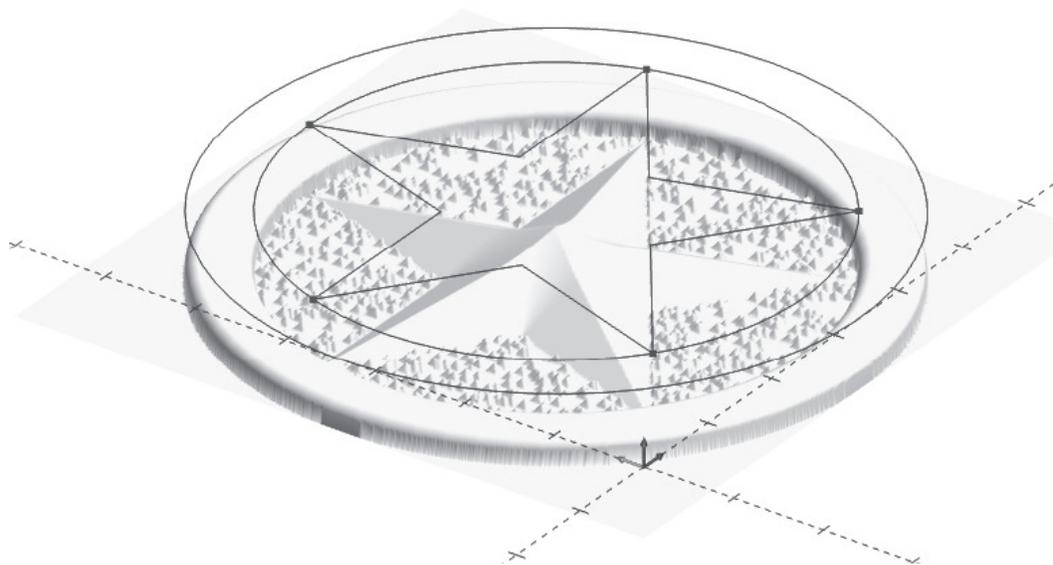


BobCAD-CAM will take a few moments to apply all of the features that were added and will display the results when its finished.

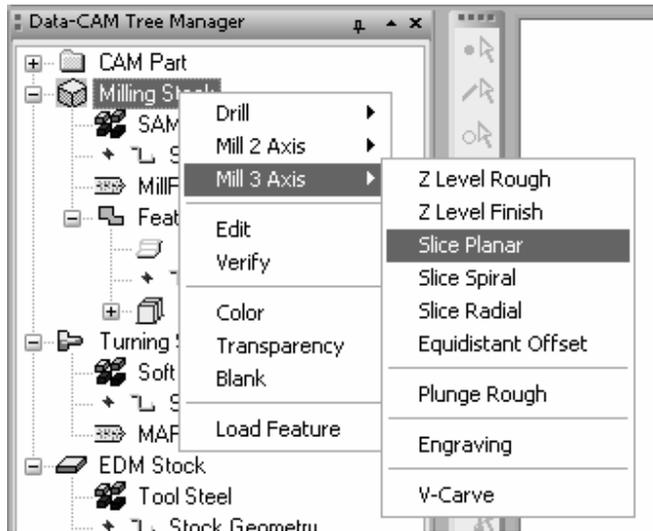
The generated emboss should appear like the illustration here.



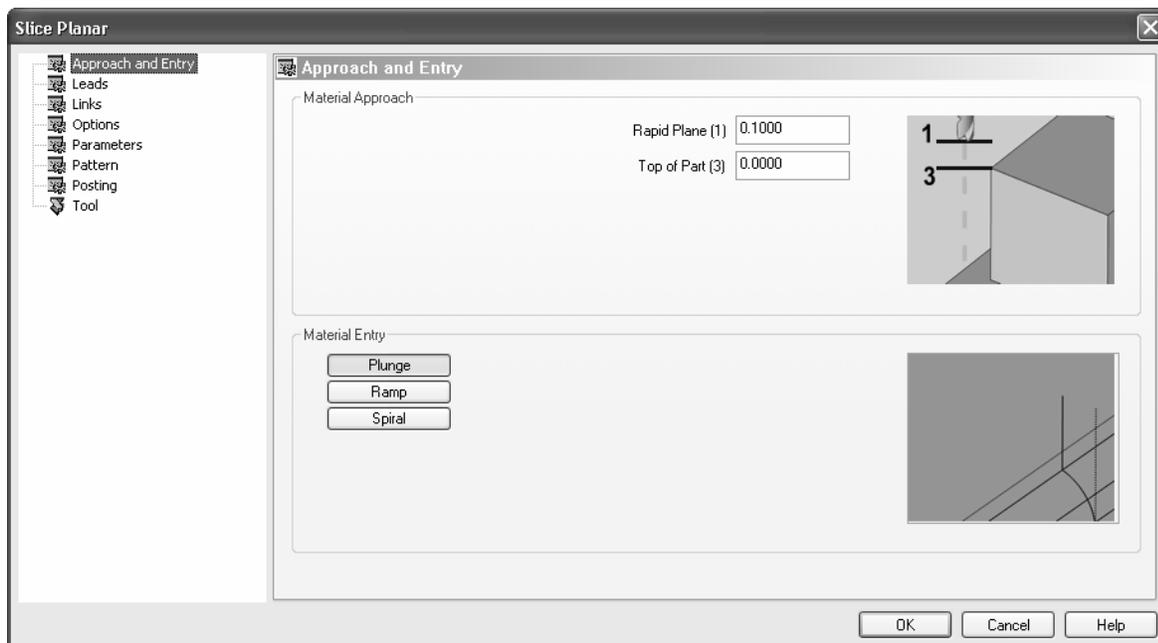
To help illustrate the 3D nature of the emboss, rotate the drawing some in the view.



8. Now we are going to create some 3 Axis Planar toolpath to machine the example part. To do this place your cursor on the CAM Tree Tab of the Data-CAM Tree Manager and click on it to access the CAM Tree.
9. Right click on Milling Stock in the Tree, choose Mill 3 Axis and then click on Slice Planar. This will load the feature into the tree.

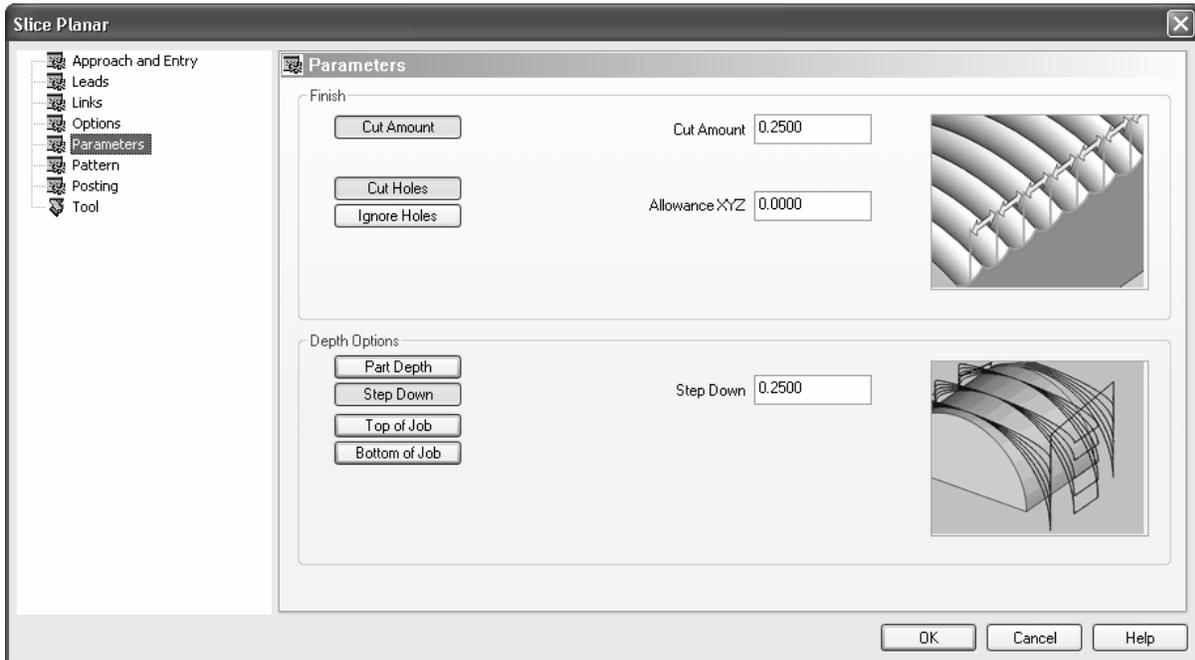


10. Now place your cursor on Geometry just under Feature Slice planar and right click your mouse, then click Re/Select. Then place your cursor anywhere on the embossed model you created and left-click your mouse on it to select it. Then hit the space bar on your keyboard to indicate OK for the selection.
11. Now right-click your mouse on Slice Planar in the Tree just under Boundary and then left-click on Edit. This will bring up the Slice Planar dialog box where you can edit all of your parameters. The first page will be the Approach and Entry Page.



The Rapid Plane is ok at .1 and you can leave the top of part at 0. Click the Plunge entry option. Now click the Leads tab to change to the Leads page.

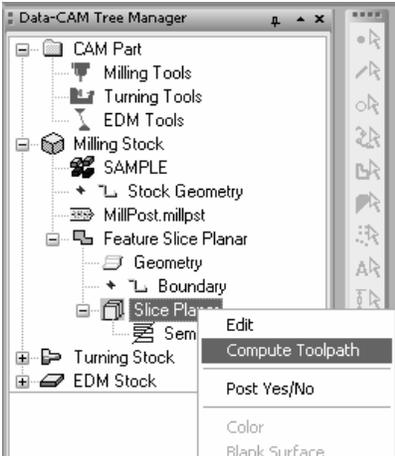
12. Click Plunge for Lead In and Vertical for Lead Out. Now click the Links tab. Select Horizontal/Vertical as the link type and then click the Options Tab. On the Options page click Tool Tip under Point and Extents under Cutting Extents and then click the Parameters page.



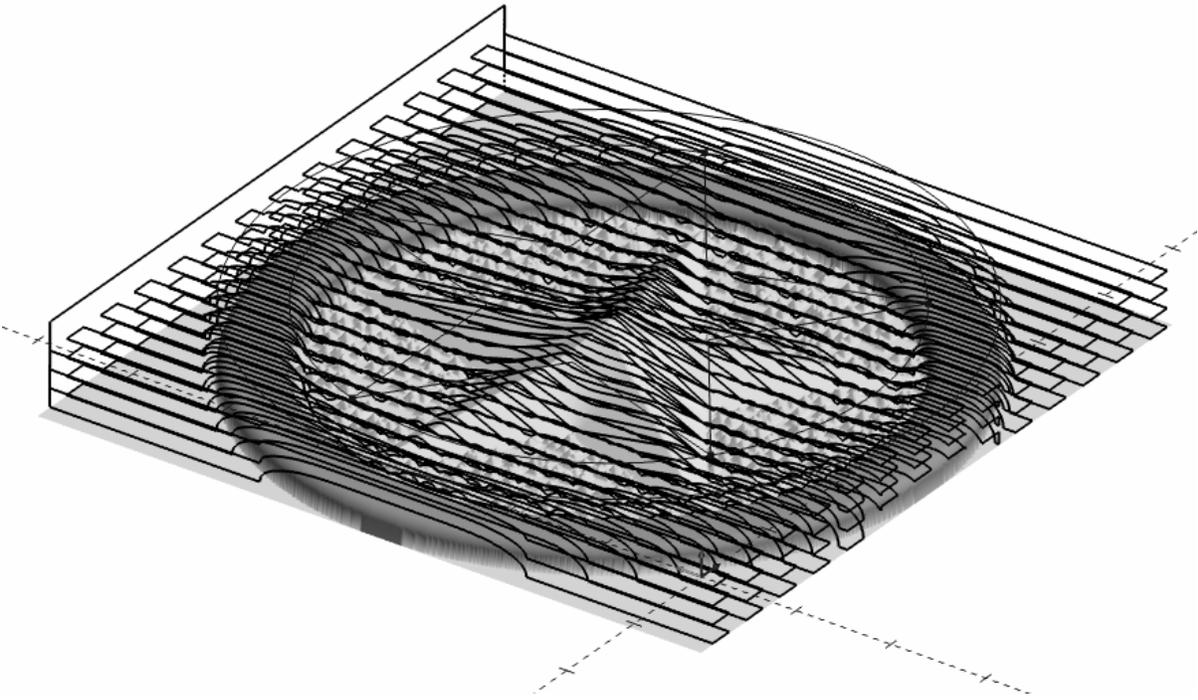
Enter the following variables:

- Cut amount: .25
- Allowance XYZ: 0
- Click the Cut Holes option
- Click the Step Down button and enter .25 for the Step Down amount.

13. Click the Pattern Tab. Click Lace Zig Zag and the Angle should be 90 degrees. Then choose Climb Mill as the cutting action. Now click the Posting Tab and then click on the Arc Fit option. The tolerance option can be left at .0005. The software will use arc segments where it can to reduce the size of the NC program. Now click the Tool Tab. Go ahead and enter .25 for the tool diameter and .125 for the corner radius. Click the OK button at the bottom and then right-click Slice Planar in the tree and choose Compute Toolpath.



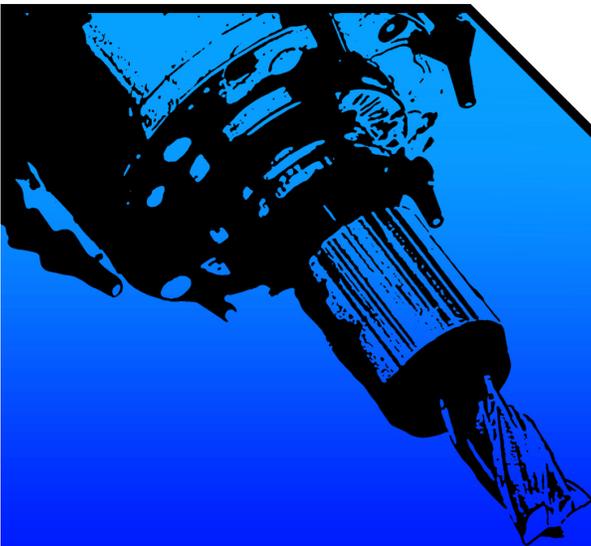
The result:



Chapter 3

Version 23 Lathe

N 6 9 6 X 4 . 4 2 3 8 Y 2 . 6 0 6 6
N 6 9 7 X 4 . 4 1 9 8 Y 2 . 6 0 5 3
N 6 9 8 X 4 . 4 0 6 8 Y 2 . 6 0 1 2
N 6 9 9 X 4 . 4 0 3 2 Y 2 . 6 0 0 1
N 7 0 0 X 4 . 3 8 9 2 Y 2 . 5 9 5 7
N 7 0 1 X 4 . 3 8 6 3 Y 2 . 5 9 4 7
N 7 0 2 X 4 . 3 7 5 3 Y 2 . 5 9 1 3
N 7 0 3 X 4 . 3 7 3 2 Y 2 . 5 9 0 6
N 7 0 4 X 4 . 3 6 5 3 Y 2 . 5 8 8 1
N 7 0 5 X 4 . 3 6 4 Y 2 . 5 8 7 7
N 7 0 6 X 4 . 3 6 0 1 Y 2 . 5 8 6 5
N 7 0 7 X 4 . 3 5 9 2 Y 2 . 5 8 6 2
N 7 0 8 X 4 . 3 5 6 8 Y 2 . 5 8 5 4
N 7 0 9 X 4 . 3 3 4 8 Y 2 . 5 7 8 5
N 7 1 0 X 3 . 1 5 6 5 Y 2 . 2 0 7
N 7 1 1 X 3 . 1 4 3 2 Y 2 . 2 0 2 8
N 7 1 2 X 3 . 1 4 1 9 Y 2 . 2 0 2 4
N 7 1 3 X 3 . 1 4 0 5 Y 2 . 2 0 1 9
N 7 1 4 X 3 . 1 3 9 5 Y 2 . 2 0 1 6
N 7 1 5 X 3 . 1 3 8 6 Y 2 . 1 9 9 8
N 7 1 6 X 3 . 1 3 7 6 Y 2 . 1 9 9 2
N 7 1 7 X 3 . 1 3 6 7 Y 2 . 1 9 7 1
N 7 1 8 X 3 . 1 3 5 8 Y 2 . 1 9 6 4
N 7 1 9 X 3 . 1 3 4 9 Y 2 . 1 9 5 6
N 7 2 0 X 3 . 1 1 1 4 Y 2 . 1 9 2 8



Chapter 3. Version 23 Lathe

This chapter is dedicated to teaching fundamental aspects of how to setup your software for Lathe. The Lathe module can be purchased as an add-on module if you do not already have it.

3.1 Software Setup

BobCAD-CAM's turning features provide the user with a number of options to aid machining with a lathe. There are several aspects that the user should be familiar with prior to machining parts.

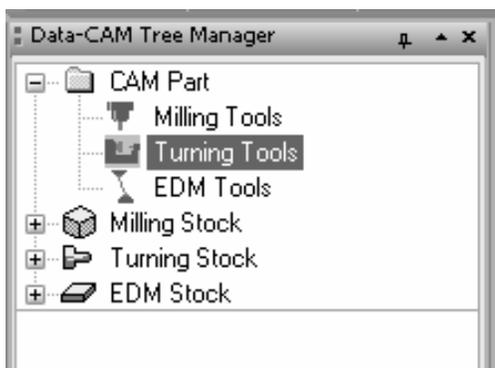
- **Turning Tools:** Turning Tools contains all of the settings relating to the tool library and the general machine settings, including post processing. The user should take the time to review this before machining parts.
- **Turning Stock:** Turning Stock allows the user to change the material to be machined and also serves as the placeholder for all turning features that have been inserted into the tree.

BobCAD-CAM can support several different machining strategies based on the needs of the part. Let's go ahead and look at these.

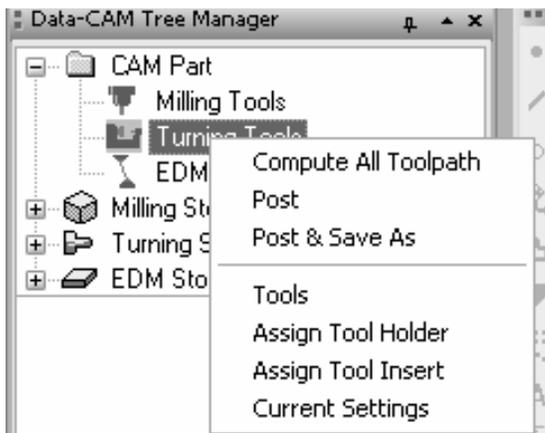
Turning Tools

Turning Tools at the CAM Part level of the tree defines the machine settings and cutting conditions for turning in BobCAD-CAM. It is simpler to describe the items available in the order they appear.

To gain access to these settings, first click the + icon next to CAM Part at the top of the tree to expand it:



Right click on Turning Tools to display the Context menu.



Let's take a quick look at what is in this menu:

Compute All Toolpath: This item in the menu will cause every feature that is already in the tree to recompute their toolpaths in the order that they are listed. Depending on the number, type, and complexity of those features, the re-compute could take some time.

Post: After all toolpath in the tree is computed, click this item to post process a completed NC program. Only those features in the tree under **Turning Stock** that are not marked not to post will be included.

Post & Save As: This item operates the same as Post above, but also automatically opens a Save As dialog to prompt the user where to save the finished NC code.

Tools: BobCAD-CAM permits the user to edit nearly all parameters of the lathe tools in the default tool library to match what is actually present in the user's shop.

Assign Tool Holders: BobCAD-CAM allows the user to change the shape of the tool holder to match the same holder in the shop.

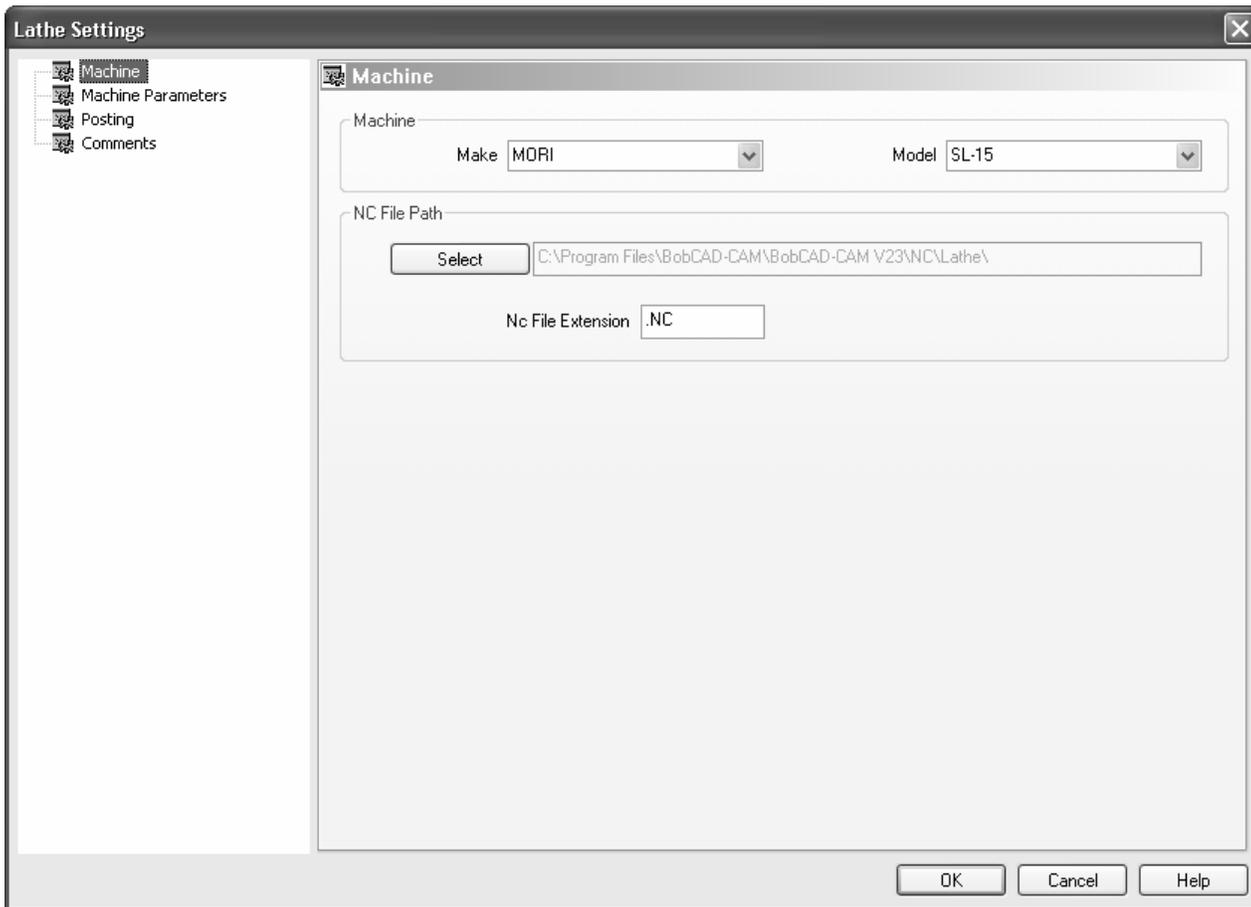
Assign Tool Inserts: BobCAD-CAM allows the user to change the shape of the tool inserts to match the same inserts in the shop.

Current Settings: The settings for the machine can be changed from this menu item.

Let's take a closer look at the Current Settings feature.

The **Lathe Settings** dialog, accessed by clicking on **Current Settings** from the **Turning Tools** pop-up menu in the CAM tree, contains all of the settings related to the current machine in use.

The Machine Page



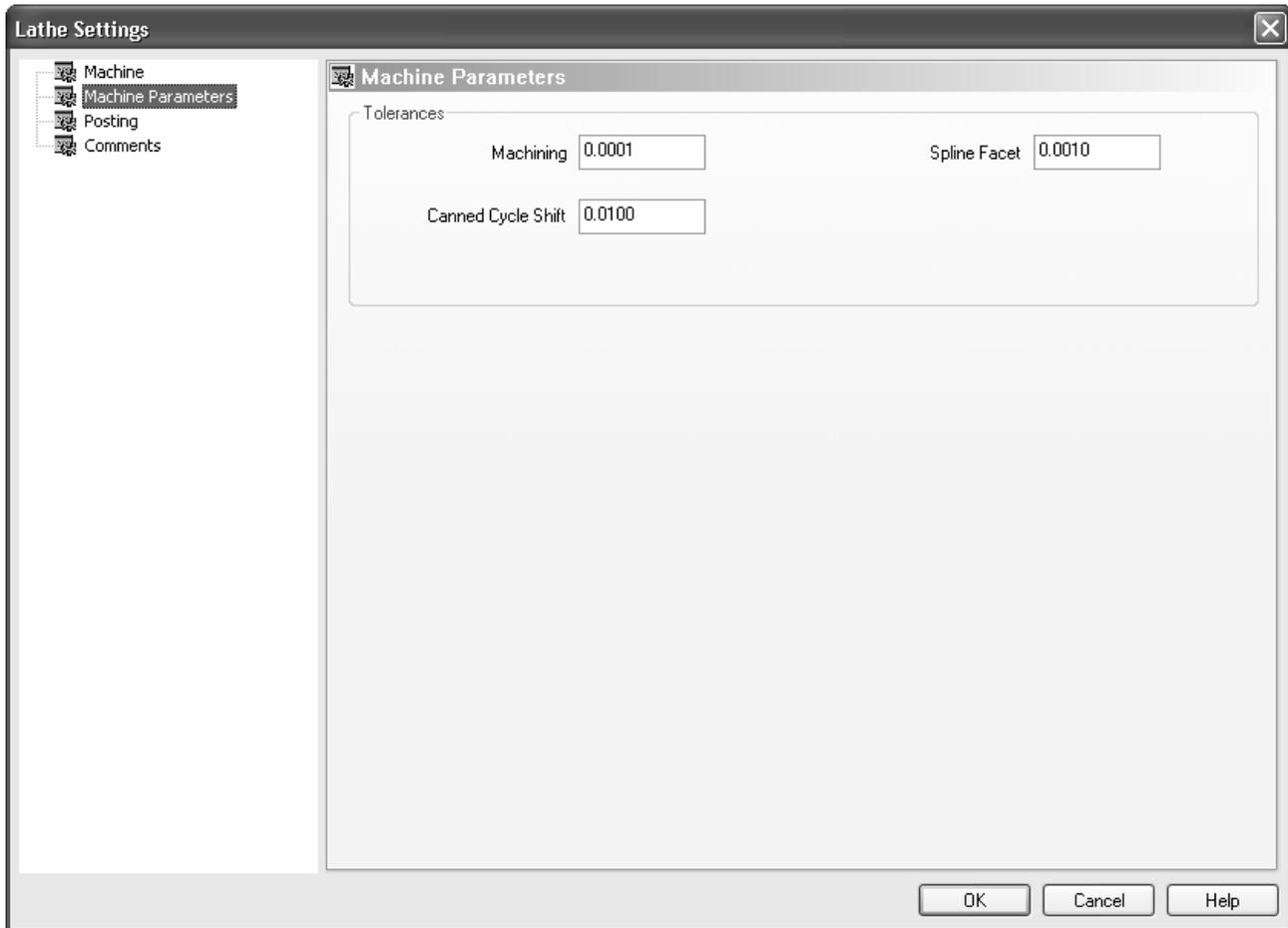
Machine

- **Make:** Designates the make of the desired machine to narrow the actual machine choices down.
- **Model:** Sets the model of the machine in use and sets the post processor to use in the Posting page below.

NC File Path

- **Select:** Sets the folder that BobCAD-CAM will save the finished NC program to.
- **Nc File Extension:** Sets the file extension that BobCAD-CAM will automatically add to the finished NC program file.

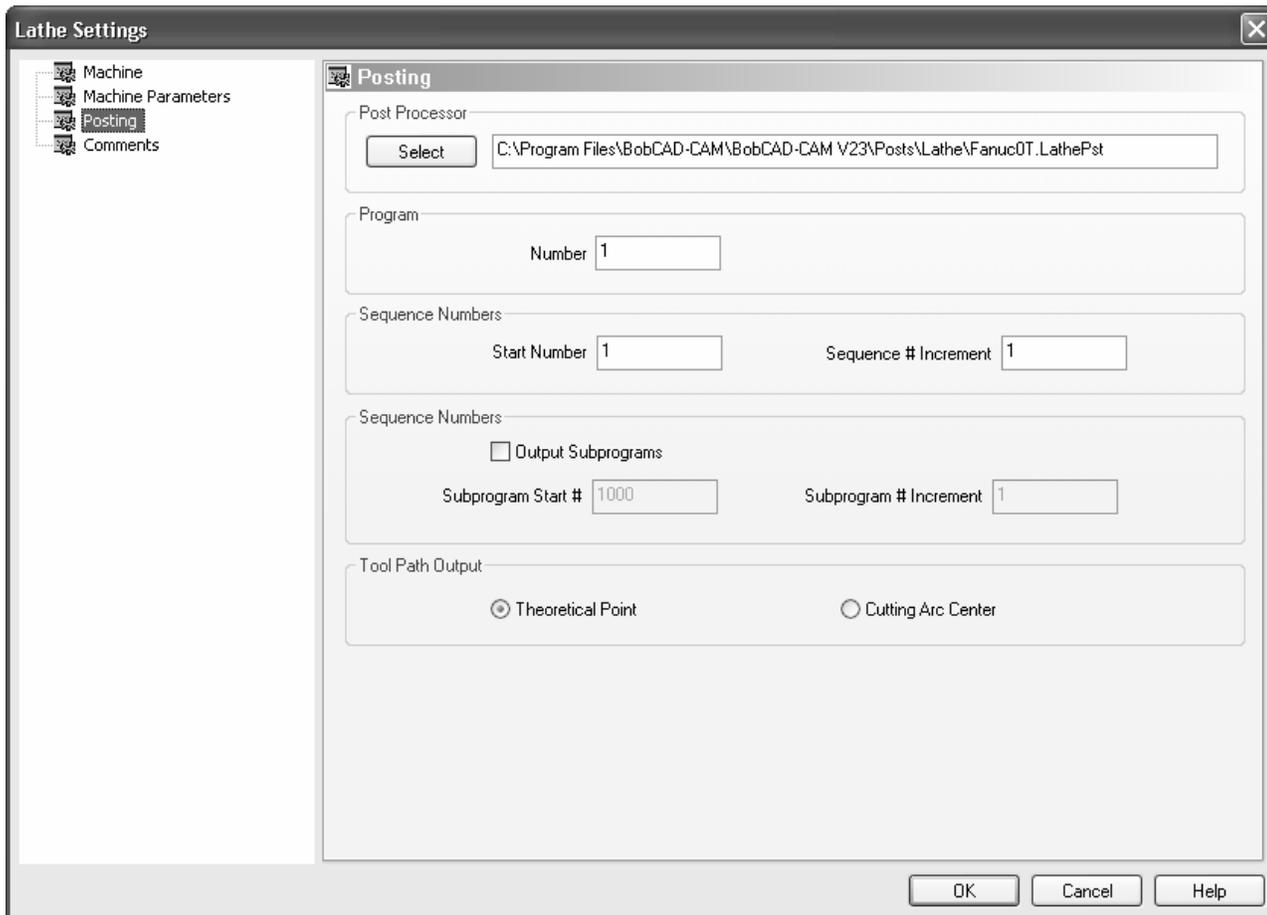
The Machine Parameters Page



Tolerances

- **Machining:** This sets the default tolerance to achieve in the NC program and toolpath. A larger number will result in less NC code but will also cause a poorer finish on the part. A smaller setting will improve the part, but may cause BobCAD-CAM to generate much more NC code.
- **Spline Facet:** When BobCAD-CAM breaks up spline entities into lines and arcs for machining, this setting governs how far away the resulting entities may be from the mathematically correct spline. A larger number results in less code but a much less accurate part, and a smaller number will produce a much better part but may generate more code than desired.

The Posting Page



Post Processor

- **Select:** Click this button and choose a new .LathePst post processor file to override the default post processor used for the current machine.

Program

- **Number:** Change this number to assign a different NC program number in the posted NC program.

Sequence Numbers

- **Start Number:** The number in this field designates the starting line number for events in the NC program if the machine is configured to output them.
- **Sequence # Increment:** This sets the number to be added to each subsequent line number for the next line.

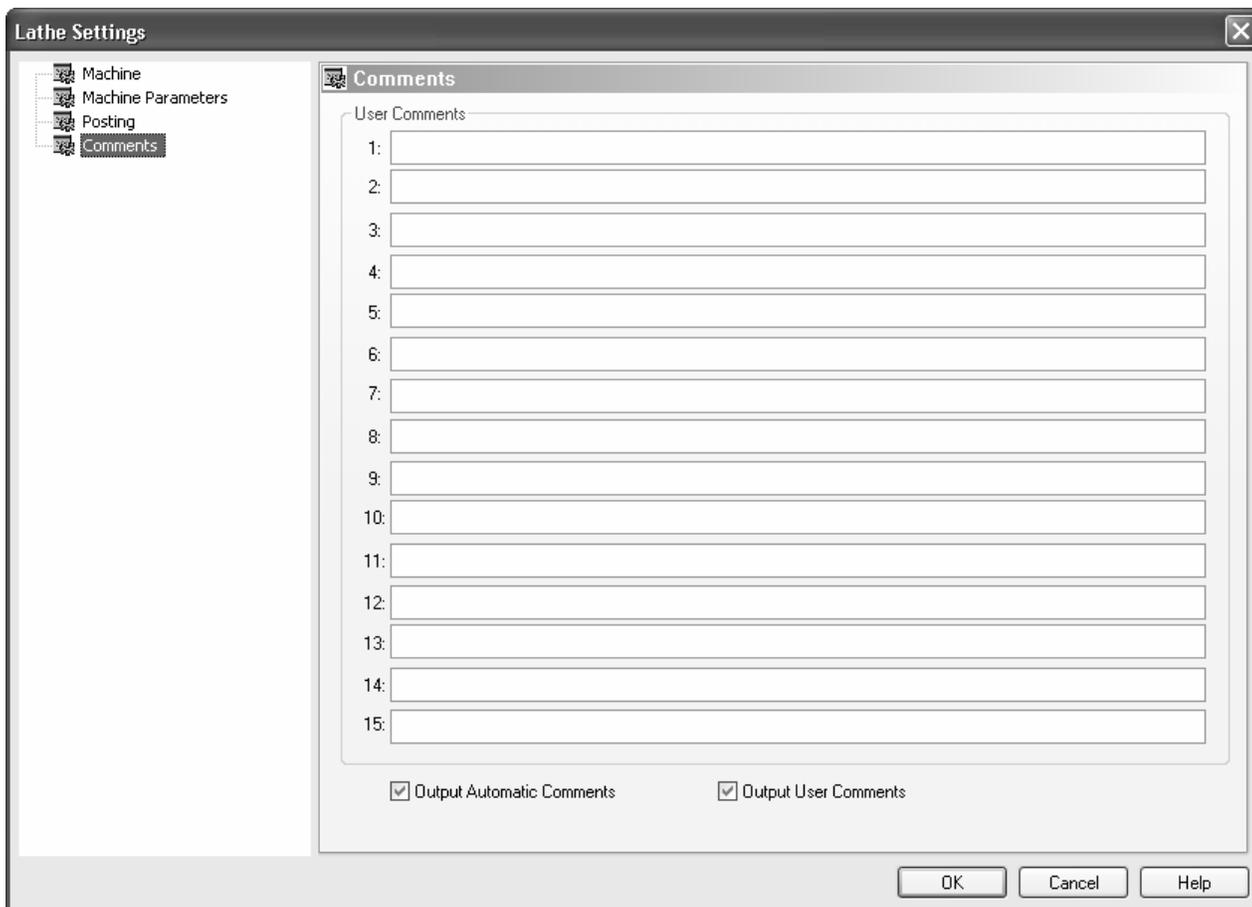
Sequence Numbers

- **Output Subprograms:** A check in this box will allow BobCAD-CAM to generate repetitive subprograms in the posted output if the post processor is set up to do so to save on machine memory.
- **Subprogram Start #:** This sets the first subprogram number to be used in the finished output.
- **Subprogram # Increment:** If more than one subprogram is to be generated, this number sets the number to be added to the previous subprogram for the next.

Tool Path Output

- **Theoretical Point:** This setting dictates to BobCAD-CAM if generating toolpath from the theoretical point of the tool (the intersection of two tangent lines extending from the extents of the cutting area of the tool) is desired.
- **Cutting Arc Center:** This setting sets if BobCAD-CAM should generate the toolpath from the arc center of the tool (inside the extents of the cutting portion of the tool by the size of the tool nose radius).

The Comments Page



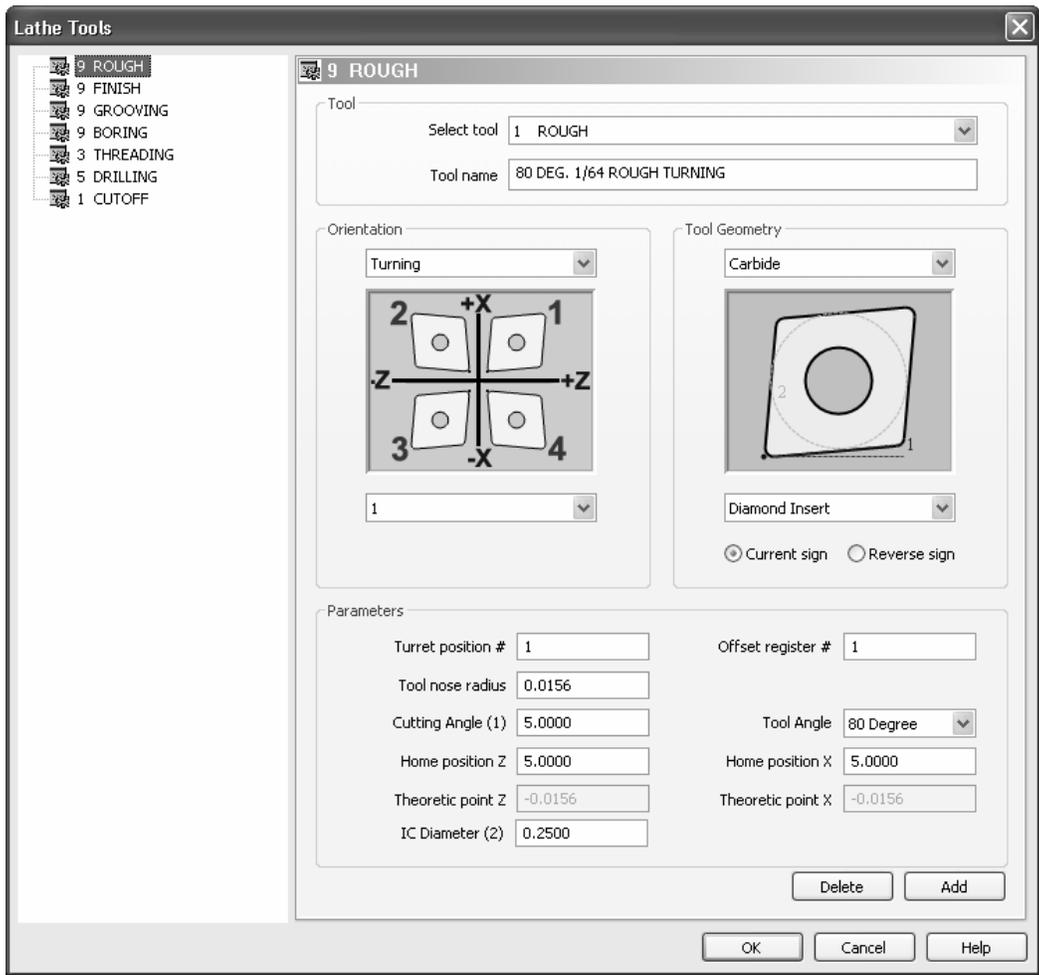
User Comments

- **#1-#15:** This field allows the user to add up to additional comments to be displayed in the program at various points. Note that the post processor configuration file must be set up to specify where each comment should appear in the posted output.
- **Output Automatic Comments:** If this is checked, the system will output the normal feature comments into the program.
- **Output User Comments:** If this button is set, the comments entered into this page will be included in the posted program.

We also want to take a closer look at the Tools feature.

Tools

The Lathe Tools dialog details the settings for every tool set up in the system. Listed here is the full complement of parameters for setting the tools up; the actual available parameters in each page change slightly for each tool type.



Tool

- **Select tool:** The identifier of the current tool being edited in the lathe tool database.
- **Tool name:** The name of the current tool selected.

Orientation

- **Turning:** This parameter identifies the preferred axis of tool motion, in this case, along the Z axis.
- **Facing:** This parameter identifies the preferred axis of tool motion, in this case, along the X axis.
- **Direction (1-4 or 5-8):** Identifies the direction of the tool nose for this tool. See the diagram above it for illustration of each tool tip direction.

Tool Geometry

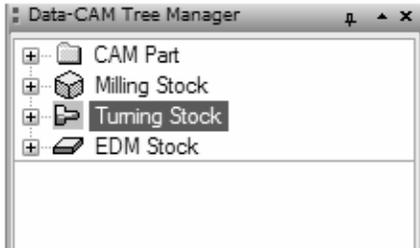
- **Tool Material:** Currently an unused parameter, the system in future versions will make use of this in the materials libraries.
- **Diamond:** Indicates the insert used in this tool is a diamond shape.
- **Triangle:** Indicates the insert used in this tool is a triangular shape.
- **Round:** Indicates the insert used in this tool is a round shape.
- **Square:** Indicates the insert used in this tool is a square insert shape.
- **Current sign:** This tool uses the standard +X cutting direction.
- **Reverse sign:** This tool reverses the standard X direction to be negative.

Parameters

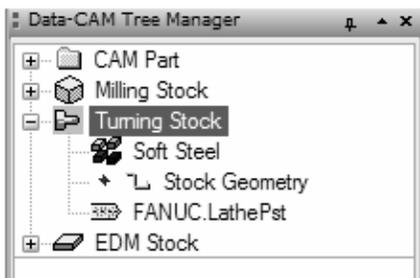
- **Turret position #:** This parameter instructs the software that the tool is loaded into this turret position number on the machine.
- **Offset register #:** Indicates the offset register number to call out in posted NC programs using this tool.
- **Tool nose radius:** Sets the tool nose radius used on the insert for this tool.
- **Cutting Angle:** This parameter indicates the angle of the tip of the tool about the tool nose radius.
- **Tool Angle:** This parameter indicates the angle of the tool in relation to the long axis of the cut.
- **Tool Width:** For certain tools, the width of the tool must be taken into account in the toolpath. Set that here.
- **Drill Diameter:** Available only in the Drilling page, this sets the diameter for the drill.
- **Home position Z:** This parameter indicates the position in Z that BobCAD-CAM will bring the tool when a move to home is required.
- **Home position X:** This parameter indicates the position in Z that BobCAD-CAM will bring the tool when a move to home is required.
- **Theoretic point Z:** This is the distance to the theoretical tip of the cutter from the center of the nose radius in the Z axis.
- **Theoretic point X:** This is the distance to the theoretical tip of the cutter from the center of the nose radius in the X axis.
- **Delete:** Clicking this button will delete the tool currently selected from the library.
- **Add:** If the current tool has been modified, clicking Add will create a new tool in the database with these parameters set.
- **OK:** Click this button when finished editing the library to have BobCAD-CAM accept the changed settings and close the dialog.
- **Cancel:** Click this button to discard any settings that have been changed and close the dialog.
- **Help:** This displays the Help Files.

Turning Stock

Once again, Turning Stock allows the user to change the material to be machined and also serves as the placeholder for all turning features that have been inserted into the tree. Let's take a closer look at this.

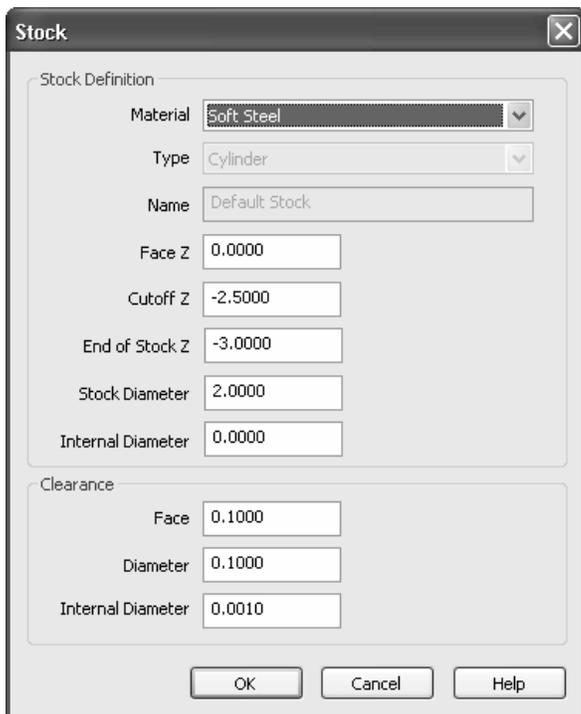


Click on the small + symbol next to Turning Stock to expand it and display these items:



Let's take a quick look at the tree under Turning Stock.

Stock Material: Right-click on this item and choose Edit to display the Material Selection dialog and quick-change the current stock material.



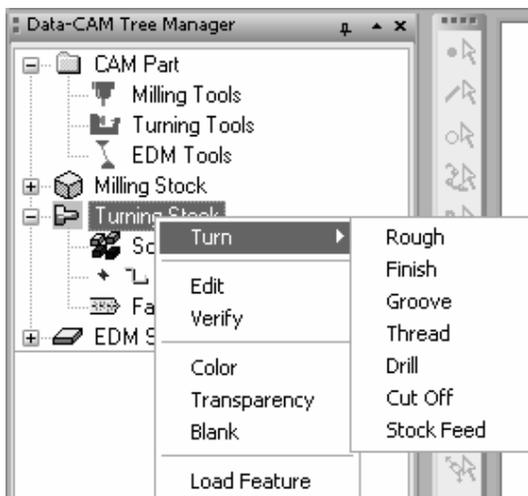
Stock Geometry: By default, BobCAD-CAM will assume a cylindrical stock of a 2" diameter. BobCAD-CAM automatically assumes a 2" diameter cylindrical stock geometry when toolpath is generated. However, occasionally the user will require a different shape for the stock.

To define a new stock shape:

1. Draw the 2D shape of the turning stock to be used.
2. Right click on Stock Geometry under Turning Stock and choose Re/Select.
3. While holding the Shift key on the keyboard, click on the wireframe shape that represents the outer boundary of the stock.
4. Right click anywhere in the workspace and choose OK from the menu that will appear.
5. The stock will be automatically drawn.

Current Post Processor: This item displays the name of the post processor that is currently loaded for milling parts. Right-click on it and choose Edit to display the Set Post Processor dialog, which allows the user to quick-change the current post processor without editing the general machine setup.

Turning Stock also has a right-click context menu. Right click on it to display the different options below:



The following is a list of the options found in this menu:

Turn: This option contains all of the features supported by BobCAD-CAM that can be inserted into the tree and machined. BobCAD-CAM V23 provides the user with numerous toolpath strategies for turning bar stock on a lathe. See the links in the list below for detailed descriptions of each cycle.

- **Rough:** This feature is used to remove material from the stock while leaving an allowance for finishing operations.
- **Finish:** This feature is used to finish the profile after the Rough operation. It also operates like a Pattern Repeat (G73 on Fanuc®-style machines) cycle when multiple passes are used.
- **Groove:** This option is used to create a slot or groove on the face of a part or the diameter of a part, dictated by the orientation of the selected geometry.

- **Thread:** This feature is used to execute a threading operation on the lathe based on user inputs and selections.
- **Drill:** This feature is used to drill a hole in the face of a part.
- **Cut Off:** The Cut Off feature is used to cut a finished part from the bar stock.
- **Stock Feed:** This feature utilizes the bar feeder, on lathes equipped with the option, to advance the stock based on user inputs.

Edit: This option allows the user to modify the parameters of the stock.

Verify: When this option is selected the verification window will be opened for the user to simulate and verify the calculated toolpath.

Color: This function displays the Color dialog to allow the user to modify the color of the stock displayed in the workspace.

Transparency: This function displays the Stock Transparency dialog to allow the user to change the opacity of the defined stock displayed in the Workspace.

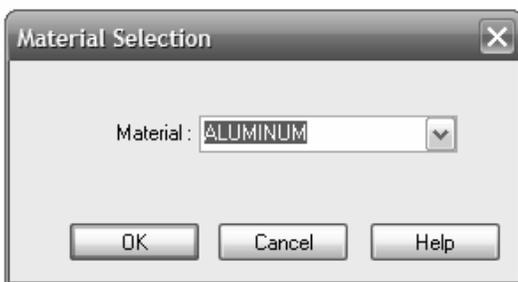
Blank: This option allows the user to hide or show the defined stock in the workspace.

Material Selection

The Material Selection dialog allows the user to quickly change the material used for the stock without the need to go through the Stock editing dialog. To display this dialog, right click on the material name in the tree:



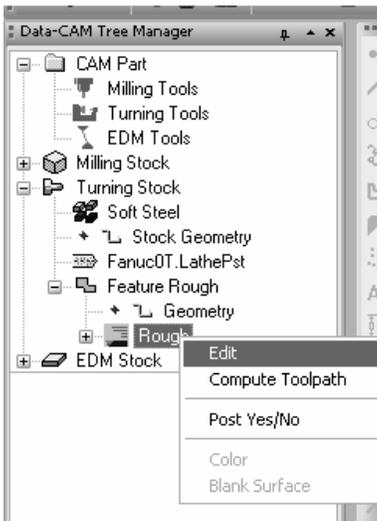
The dialog will appear on the screen:



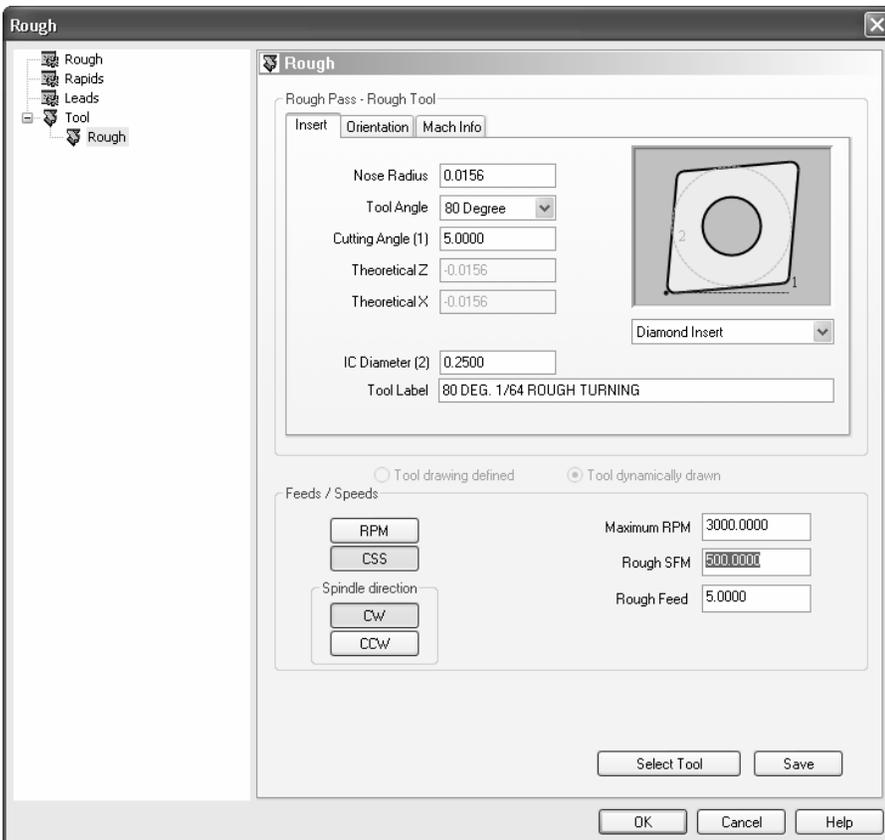
Simply select the stock you want to use from the pull-down menu and click the OK button.

SPECIAL NOTE: The material type actually has no bearing on the program in terms of speeds and feed rates. The reason for this is that there are no available cutting conditions for Lathe at this time. You will manually modify the Speeds and Feeds through the feature Edit dialog in the tree.

In other words you load the cutting feature into the tree, then edit the feature by right-clicking on it in the tree and choosing Edit.



When the Edit dialog appears, you can go to the tool page and edit the speeds and feed rates.



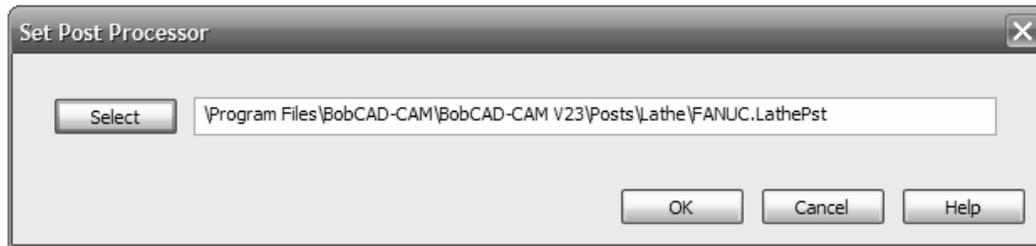
Selecting a Lathe Post Processor

The Set Post Processor dialog is a convenience dialog to allow the user to change the current post processor without using the items in the Lathe Settings dialog box.

To access the dialog, right click on the name of there current post processor in the tree and choose Edit:



When the dialog appears, click on the Select button and an Open dialog will display. Choose the desired post processor file from the list and click OK. The name of the new post will show in the dialog.



Click **OK** to have BobCAD-CAM accept the new post processor setting, or click **Cancel** to close the box without applying the change.

Post processors are also available on the Support Page of www.bobcad.com. You can also contact **BobCAD-CAM Technical Support** for posting requests at 727-489-0003.

Drawing a Lathe Part

The example in this portion of the manual is intended for use with the software in **Diameter** mode.

To make sure that it is in this mode:

1. Click on **Preferences** in the main menu
2. Click on Settings Part, or if all of the user's parts will need to be programmed in **Diameter** mode, click on **Settings Default** instead.
3. Click on **Units** in the tree to the left side of the box.
4. Click on the buttons labeled **Lathe Coordinates (Z, X, Y)** and Lathe **Diameter Mode** both.
5. Click **OK**.

The software will now display all of its X coordinates in **Diameter** mode and will ask for function parameters in the same way.

Turning Profile

This is a simple exercise designed to illustrate some of the functions in the line, point, and arc menus, as well as using the utilities menu to modify entities that are already present in the drawing. This will consist of only a few real steps: the setup, where guide geometry will be drawn; drawing the profile using the guide geometry; then breaking apart the guide geometry and cleaning up the drawing.

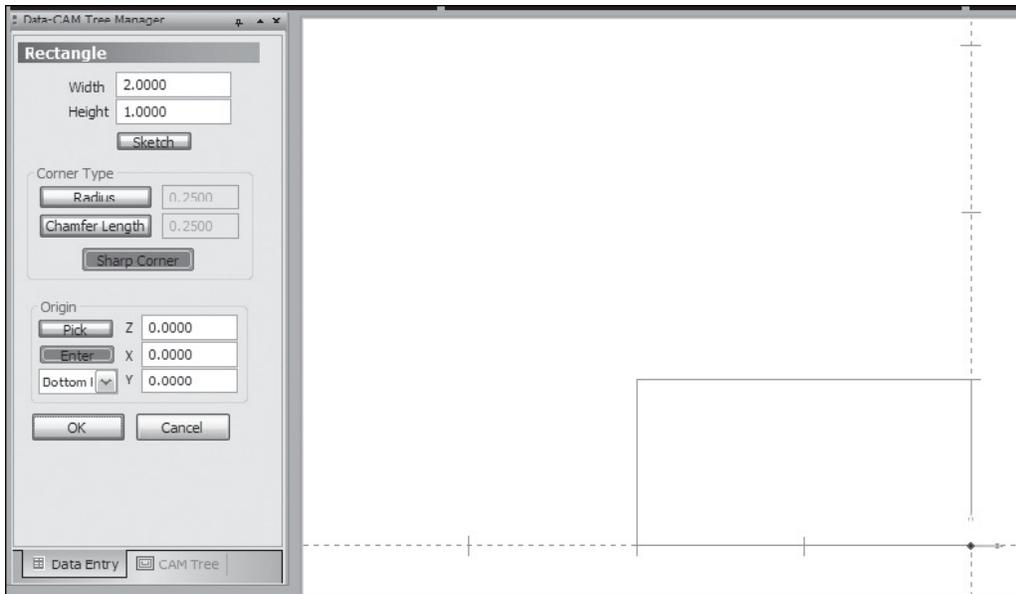
1. Begin with an empty drawing by using any one of these 3 methods:
 - a. Click on **File** in the main menu and then **New**  OR
 - b. Click on the **New File** icon in the toolbar OR
 - c. Type CTRL+N to start a new drawing.

If the **Lathe** icon  on the toolbar is not depressed, click it now to turn **Lathe** mode on.

2. In the main menu, click on **Other** and then on **Rectangle**. In the **Data Entry** tab of the **Data-CAM Tree Manager**, enter in these values:
 - a. Width: 2
 - b. Height: 1

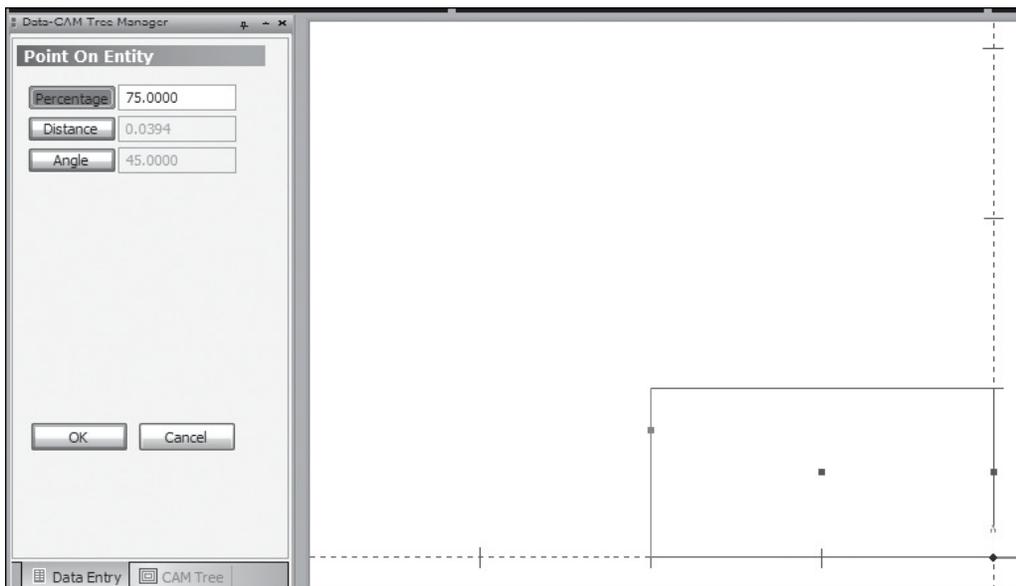
NOTE: The built-in “complex” functions in the **Other** menu, such as **Rectangle**, do not change all of their operations when the software is in diameter or radius mode. The height of the rectangle is always measured from the bottom line of the shape to the top and will not be doubled, as it is a size and not an X-axis location. However, the **Origin X** in all of these functions will follow the software's mode, as it is an X location rather than a size.

Leave the **Sharp Corner** button set, and in the **Origin** section, set the Enter button and choose **Bottom Right** from the drop-down box. Enter 0 for each of the Z, X, and Y axes. Click OK and a rectangle will be drawn.

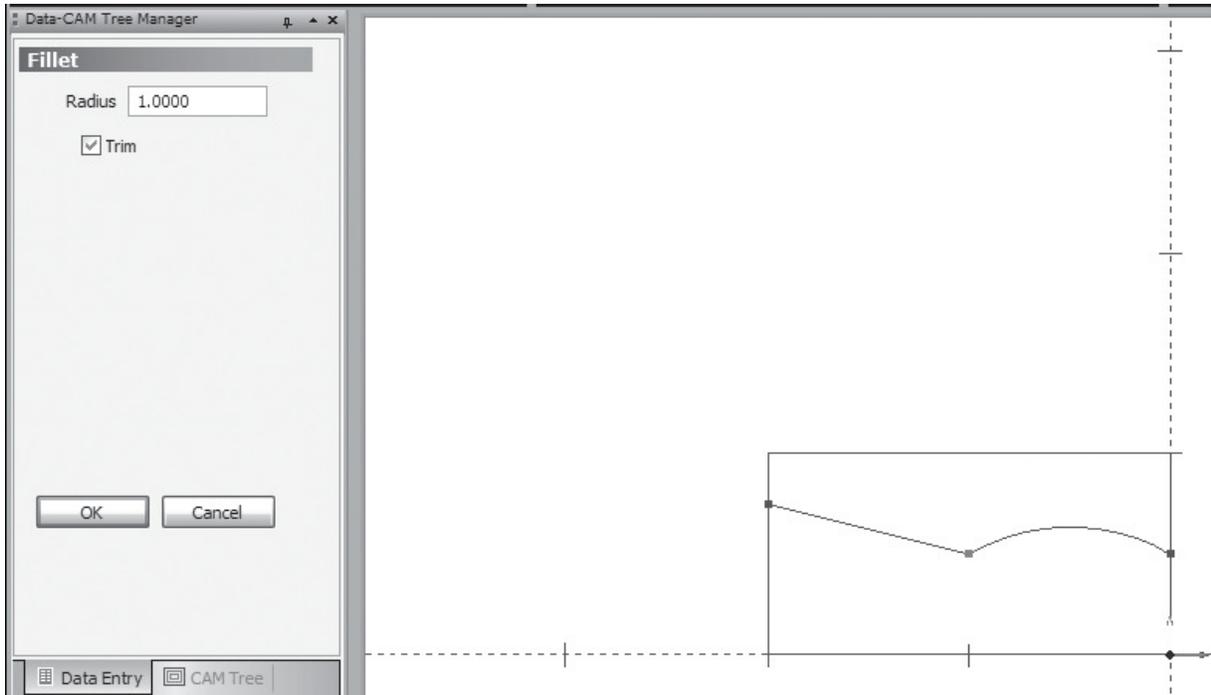


Click on Point in the main menu. Choose Coordinates. Enter -1 for the Z axis, 1 for the X axis, and 0 in Y. Click OK and a point will appear in the middle of the rectangle. Click on Point again in the menu, and choose On Entity. Set the Percentage button and enter in 50. This will put a point in the middle of any wire frame entity clicked on. Click on the right-hand line of the rectangle.

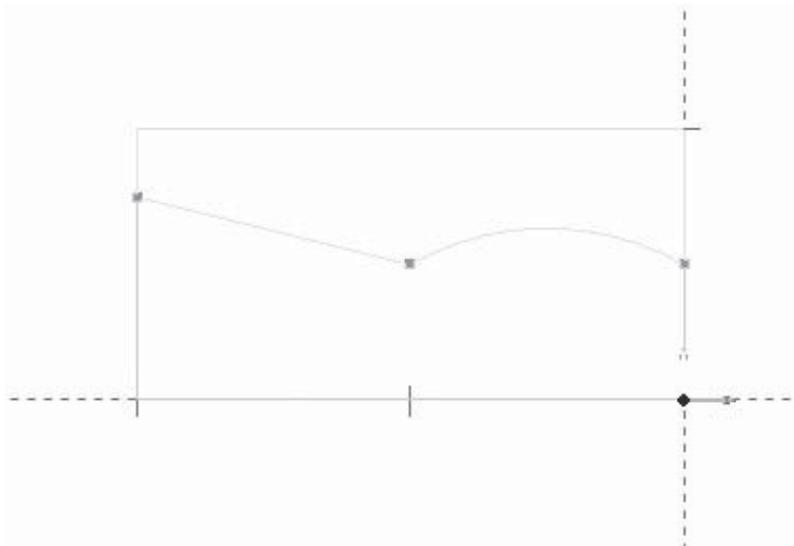
Back in the Data Entry tab, enter in **75 in the Percentage box** and click OK. Click near the bottom of the left line of the rectangle and the point will appear 3/4 of the way to the top of the line. Click Cancel.



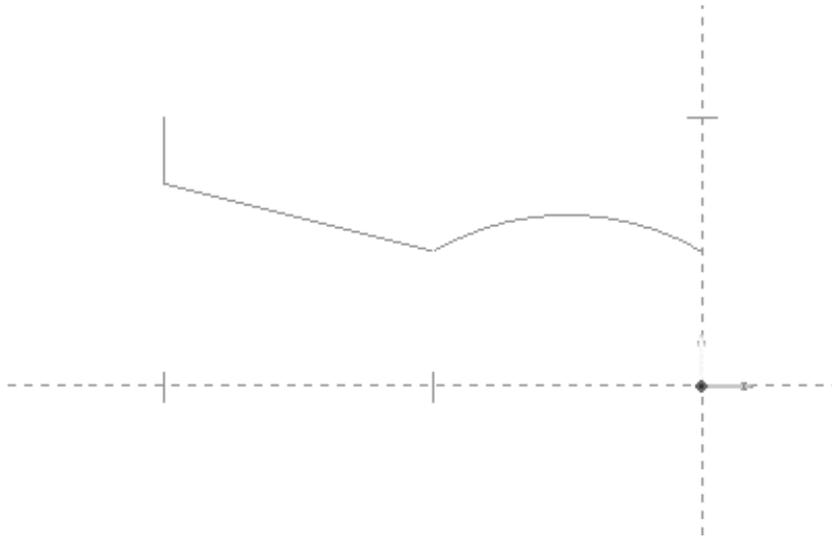
- Click on **Line** in the main menu and then on **Join**. Click on the left two points and a line will appear between them. Click on **Arc** in the menu and then on **Fillet**. In the **Data Entry** tab, enter in 1 in the **Radius** box, and then click on the right most point first, followed by the center point. Click **Cancel** to quit the arc function.



- Click on **Utilities** in the main menu and then choose **Break**, and **Many**. Drag a box over the whole drawing, then right click and choose **OK**. Every line and arc in the drawing will be broken at every point where another entity touches it. If the software is not already in selection mode, turn it on now by clicking on the black arrow icon on the toolbar. Click on every entity that doesn't belong to the final profile then press the Delete key on the keyboard to remove them from the drawing.



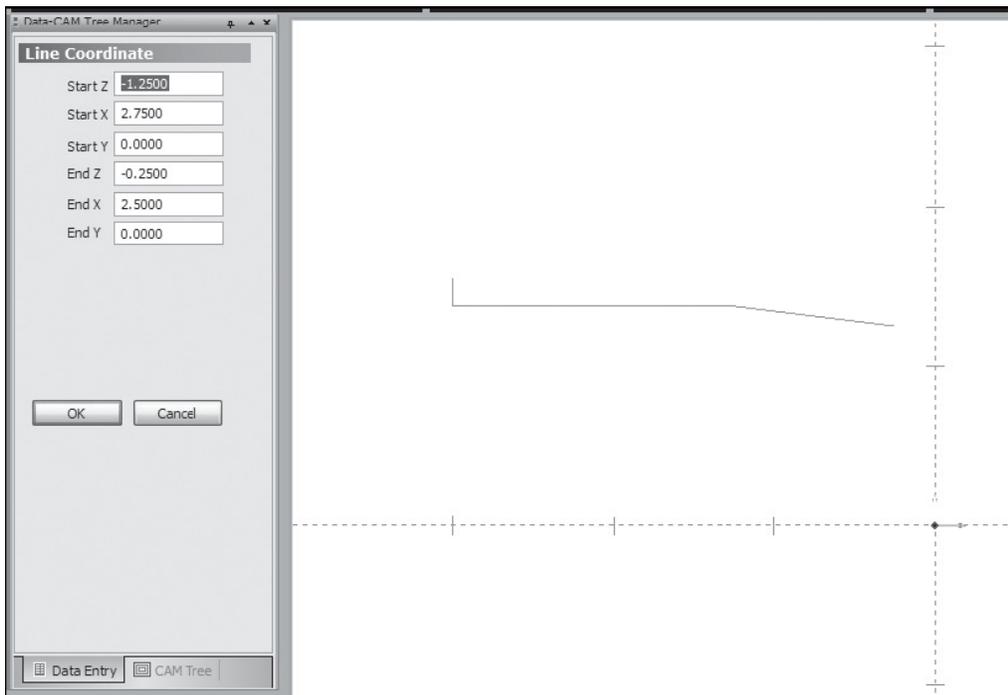
The turning profile is now complete.



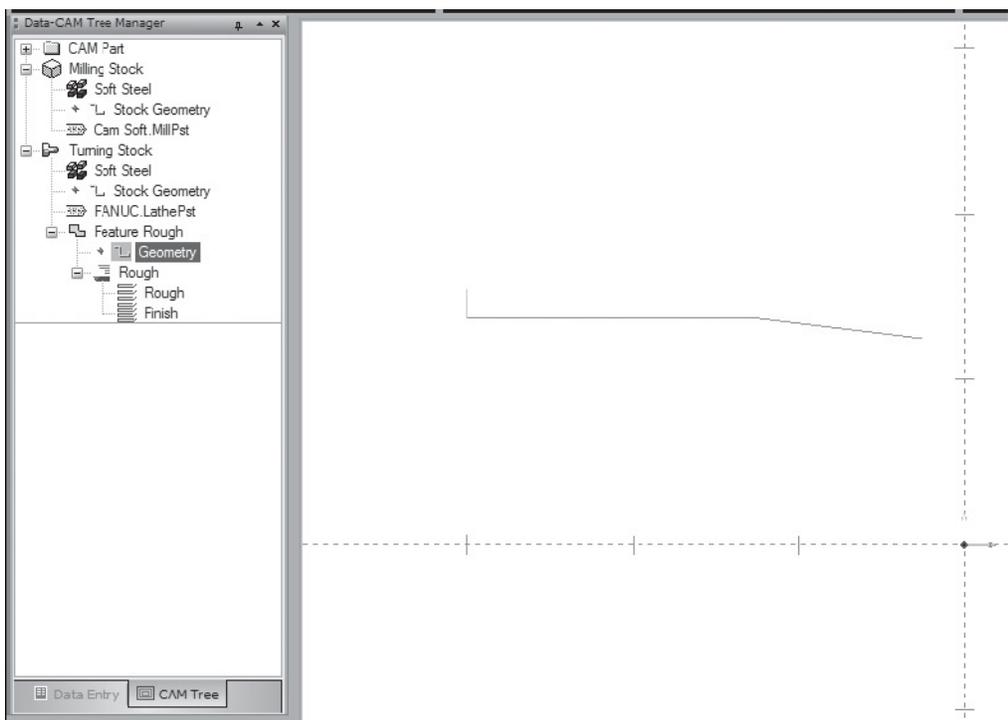
Outside Rough & Finish Example

1. Click on Line and then Coordinates. In the Data Entry tab of the Data-CAM Tree Manager, enter in these values:
 - a. Start Z: -3
 - b. Start X: 2.75
 - c. Start Y: 0
 - d. End Z: -3
 - e. End X: 3.1
 - f. End Y: 0
2. Click OK. The first line will be drawn and the data entry fields will stay available. Enter in this next set of values:
 - a. Start Z: -3
 - b. Start X: 2.75
 - c. Start Y: 0
 - d. End Z: -1.25
 - e. End X: 2.75
 - f. End Y: 0
3. Click OK again. There is one more line to be drawn, so enter these values once more:
 - a. Start Z: -1.25
 - b. Start X: 2.75
 - c. Start Y: 0
 - d. End Z: -.25
 - e. End X: 2.5
 - f. End Y: 0

- Click OK one more time and the last line will appear. Click Cancel in the Data Entry area to quit the function.



- Click on the **CAM Tree** tab of the **Data-CAM Tree Manager**. Right click on **Turning Stock**, choose **Turn**, and then **Rough** from the menu. A new **Feature Rough** feature will be added to the tree.

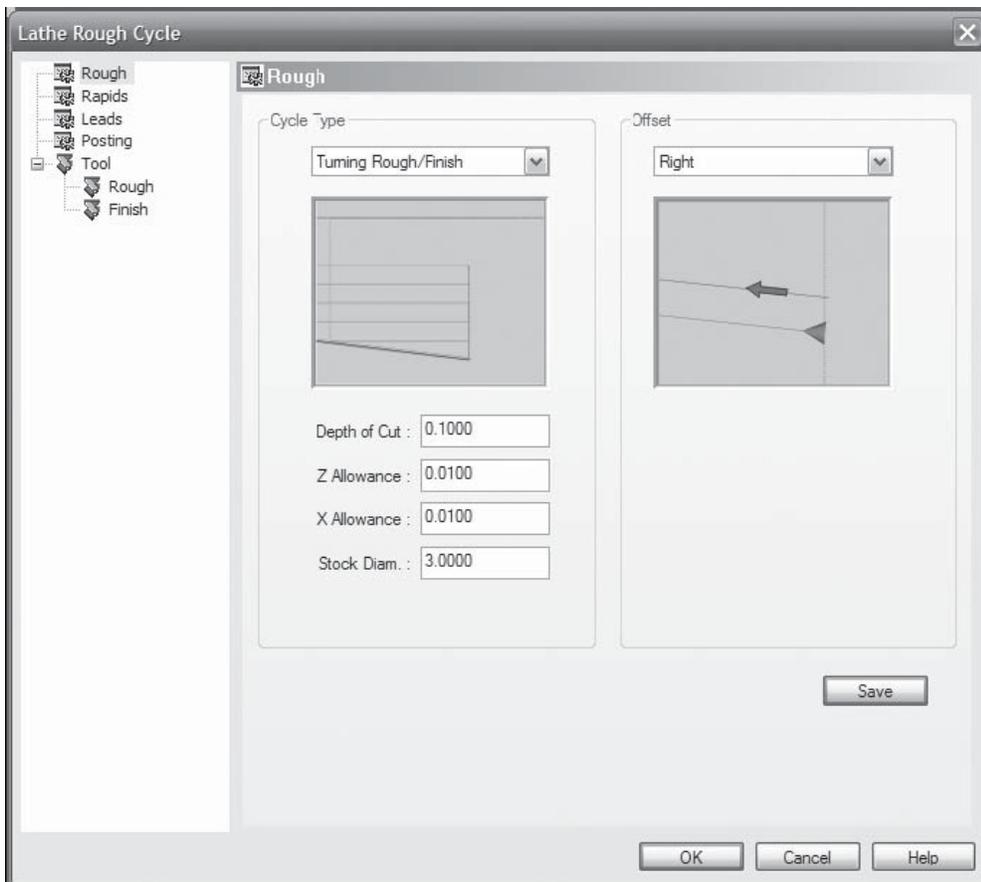


Assign the lines just drawn as the geometry for the feature by right clicking on **Geometry** and choosing **Re/Select**. Hover the mouse cursor over the upper-most end of the left vertical line so that it highlights. Hold down the SHIFT key on the keyboard and left click. All three lines will have become selected. If not, try it again until they have. Right click in the workspace and choose **OK** from the pop-up menu. The geometry is set in the feature now.

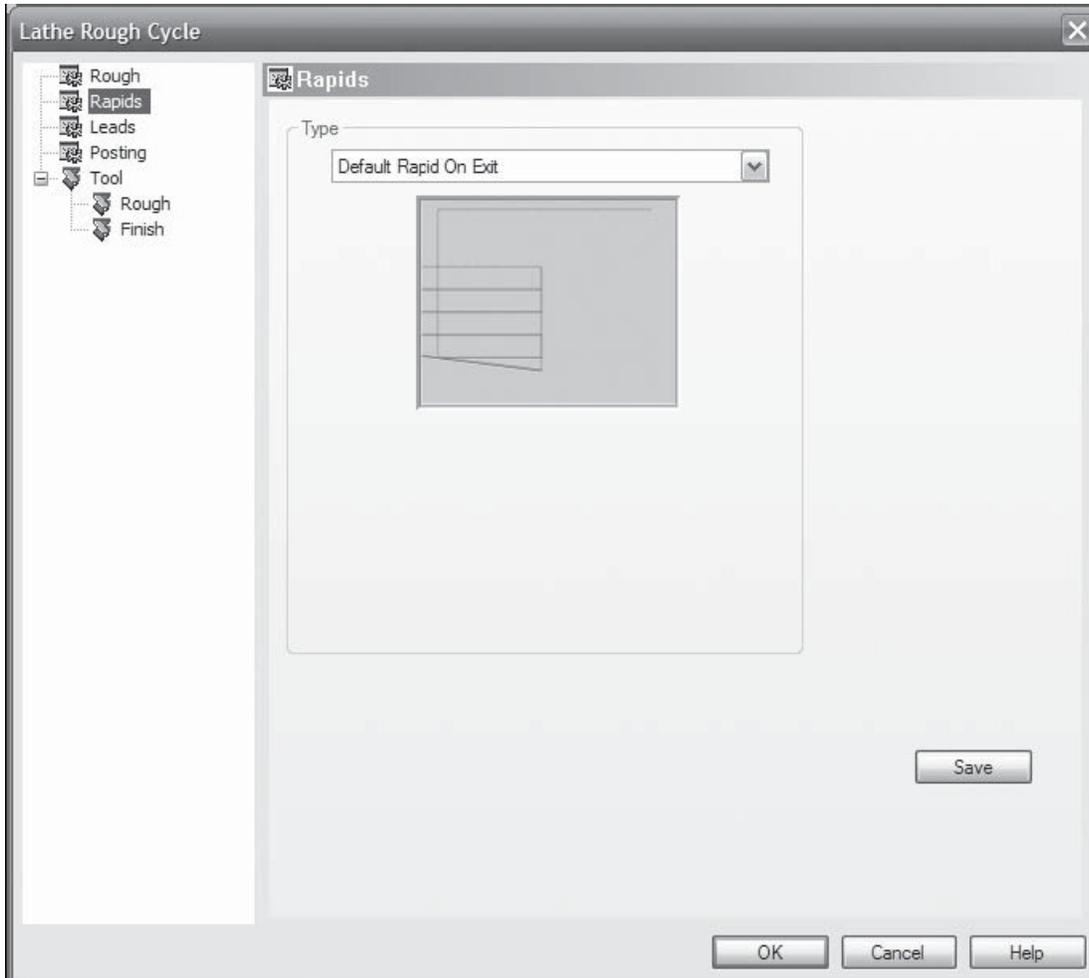
Note: Turning operations require the geometry to be selected in a front-to back direction. By hovering the cursor over the top end of the left line and holding the shift key down before clicking on it, the software will highlight the chain of entities from the beginning of the chain up to the cursor. In this case, that means it will be front-to-back.

6. Right-click on the **Rough** item in the new feature. Choose **Edit**. The first item in the tree to the left of the box that will pop-up is called **Rough**. In the right-hand pane, set the **Cycle Type** to **Turn Rough/Finish**. This will ensure that the tool motion will be horizontal along the turning axis of the stock. Below that section, enter in these values:
 - a. **Depth of Cut:** .1
 - b. **Z Allowance:** .01
 - c. **X Allowance:** .01
 - d. **Stock Diam.:** 3.

To the right under **Offset**, set the drop-down box to **Right**. This will set the tool motion to the right of the drawn path in the direction of cut, in this case, toward the outside of the stock.

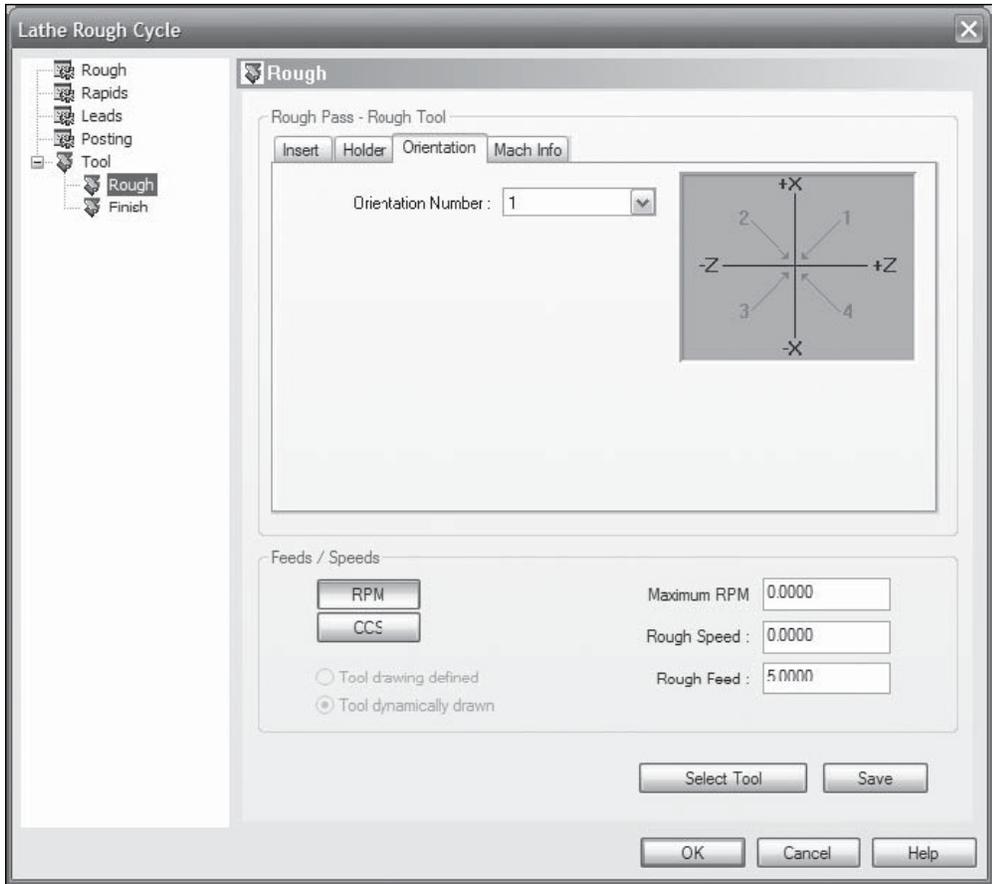


The next item in the tree on the left is **Rapids**. Click on it, then set the dropdown box in the right pane to **Default Rapid on Exit**. For most operations the default is **Rapid on Exit to Tool Home X - Z**. Since this is an OD cut, a rapid in X away from the part first is required, so this is correct.

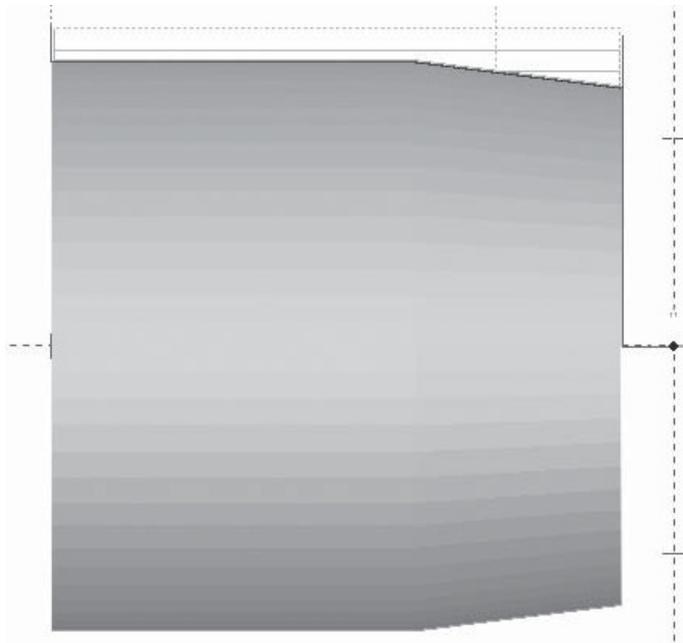


Click on **Leads** in the tree. The default lead-in and lead-out on all of BobCAD-CAM's toolpath is for no extra lead. This is acceptable for most cuts, and is good here. The next item in the tree, **Posting**, will be dealt with in section 5.8. For this example it can be safely skipped. Under **Tool** in the tree, there should be two items, **Rough** and **Finish**. Click on **Rough**. The default tool is fine, but the orientation of the tool insert will need to be checked to make sure it reflects a turning tool. Click on **Orientation** and choose **1** from the drop-down box if it isn't chosen already. Number 1 is the default for turning operations in BobCAD-CAM and it is the correct tool tip direction for this operation.

Do the same for the Finish tool.



The setup for the OD Turning operation is complete, so click **OK**. Right-click on the **Rough** item in the feature and click **Compute** to see the toolpath it generates.

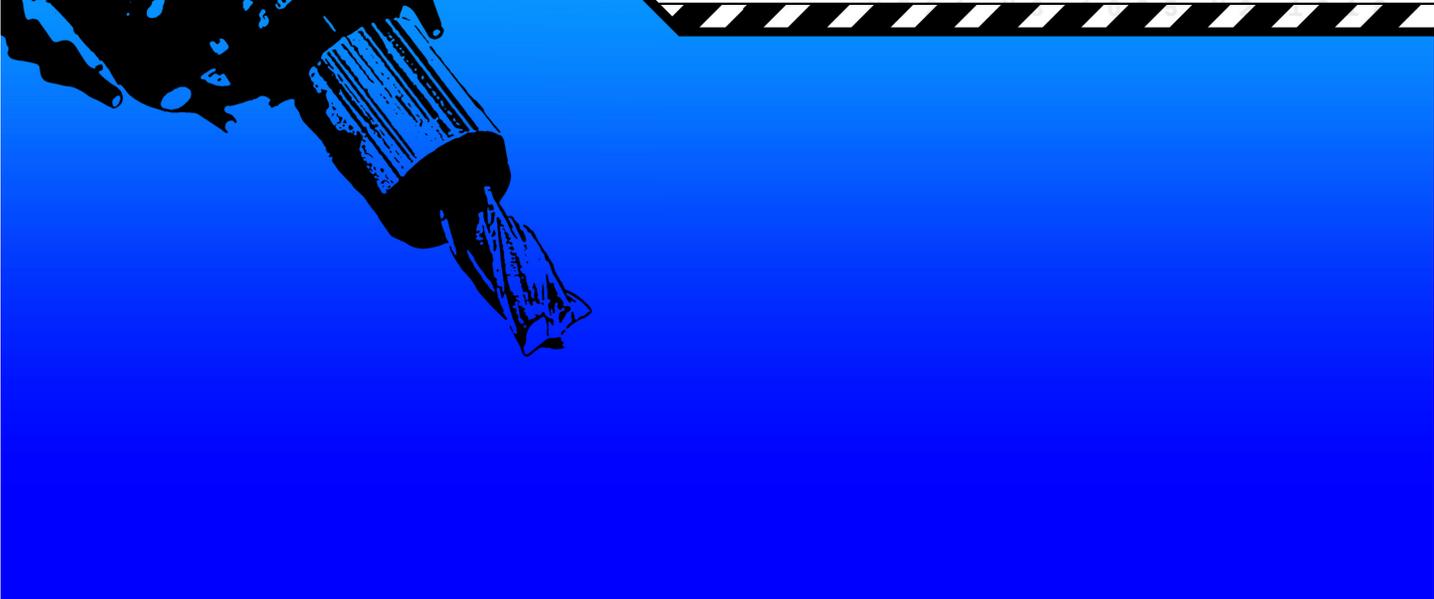
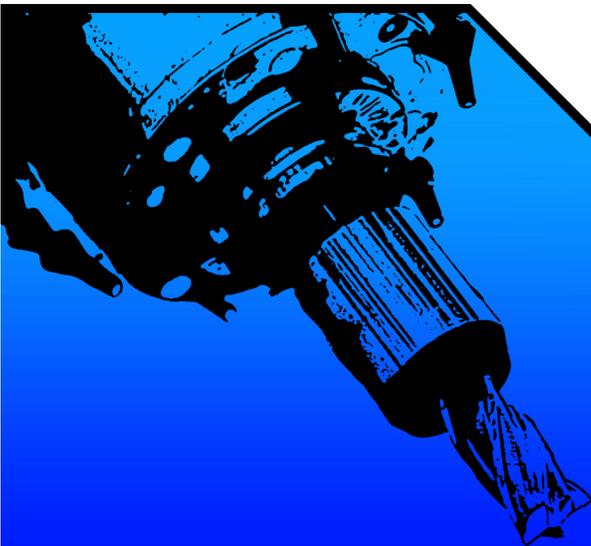


This concludes the lesson.

Chapter 4

Posting & Communications

N 6 9 6 X 4 . 4 2 3 8 Y 2 . 6 0 6 6
N 6 9 7 X 4 . 4 1 9 8 Y 2 . 6 0 5 3
N 6 9 8 X 4 . 4 0 6 8 Y 2 . 6 0 1 2
N 6 9 9 X 4 . 4 0 3 2 Y 2 . 6 0 0 1
N 7 0 0 X 4 . 3 8 9 2 Y 2 . 5 9 5 7
N 7 0 1 X 4 . 3 8 6 3 Y 2 . 5 9 4 7
N 7 0 2 X 4 . 3 7 5 3 Y 2 . 5 9 1 3
N 7 0 3 X 4 . 3 7 3 2 Y 2 . 5 9 0 6
N 7 0 4 X 4 . 3 6 5 3 Y 2 . 5 8 8 1
N 7 0 5 X 4 . 3 6 4 Y 2 . 5 8 7 7
N 7 0 6 X 4 . 3 6 0 1 Y 2 . 5 8 6 5
N 7 0 7 X 4 . 3 5 9 2 Y 2 . 5 8 6 2
N 7 0 8 X 4 . 3 5 6 8 Y 2 . 5 8 5 4
N 7 0 9 X 4 . 3 3 4 8 Y 2 . 5 7 8 5
N 7 1 0 X 3 . 1 5 6 5 Y 2 . 2 0 7
N 7 1 1 X 3 . 1 4 3 2 Y 2 . 2 0 2 8
N 7 1 2 X 3 . 1 4 1 9 Y 2 . 2 0 2 4
N 7 1 3 X 3 . 1 4 0 5 Y 2 . 2 0 1 9
N 7 1 4 X 3 . 1 3 9 5 Y 2 . 2 0 1 6
N 7 1 5 X 3 . 1 3 8 6 Y 2 . 1 9 9 8
N 7 1 6 X 3 . 1 3 7 6 Y 2 . 1 9 9 2
N 7 1 7 X 3 . 1 3 6 7 Y 2 . 1 9 7 1
N 7 1 8 X 3 . 1 3 5 8 Y 2 . 1 9 6 4
N 7 1 9 X 3 . 1 3 4 9 Y 2 . 1 9 5 6
N 7 2 0 X 3 . 1 1 1 4 Y 2 . 1 9 2 8



Chapter 4. Posting & Communications

This chapter is dedicated to providing you with valuable information that will help you understand posting and the communications aspect of the BobCAD-CAM software.

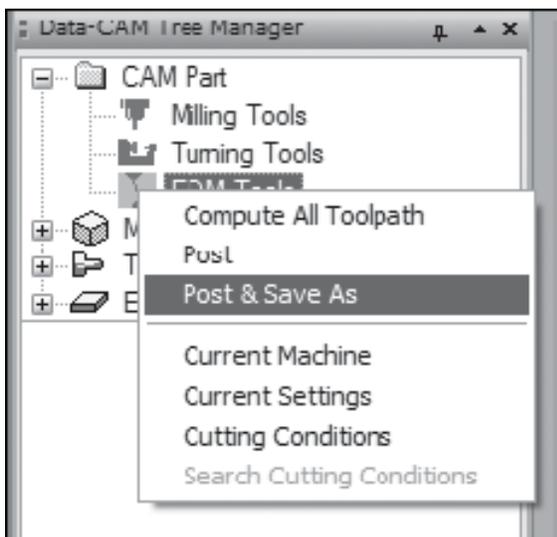
4.1 Posting & Communications

BobCAD-CAM comes equipped with a custom version of Predator™ CNC Editor, widely recognized as the best DNC package available anywhere. When transferring programs to the machine through RS-232 (serial port) links or Ethernet, there simply isn't a better solution.

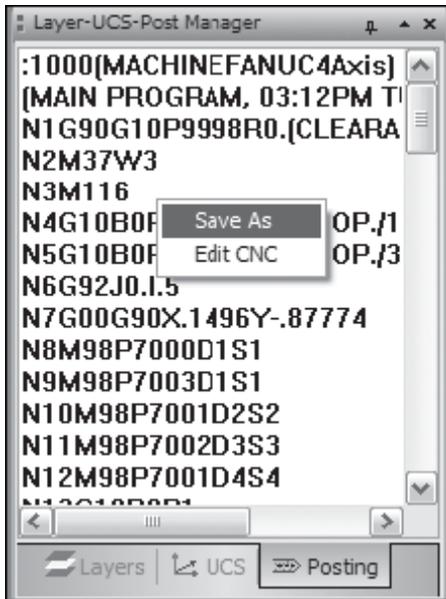
Saving Posted Programs

BobCAD-CAM uses 2 methods to save files to disk.

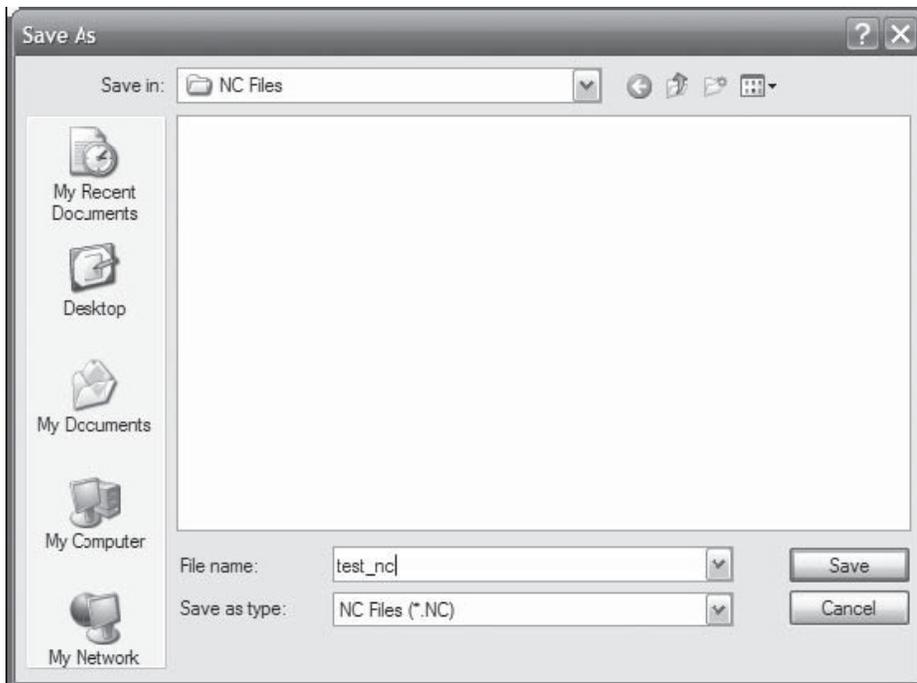
- **Post & Save As:** Right-click on the appropriate machine type under CAM Part (Milling Tools, Turning Tools, or EDM Tools) and choose **Post & Save As**. The software will prompt the user with a **Save As** dialog.



- **Save As** from the **Layer-UCS-Post Manager**: After the program has been posted, right-click in the **Layer-UCS-Post Manager** where the code is displayed and choose **Save As**. The **Save As** dialog will appear.



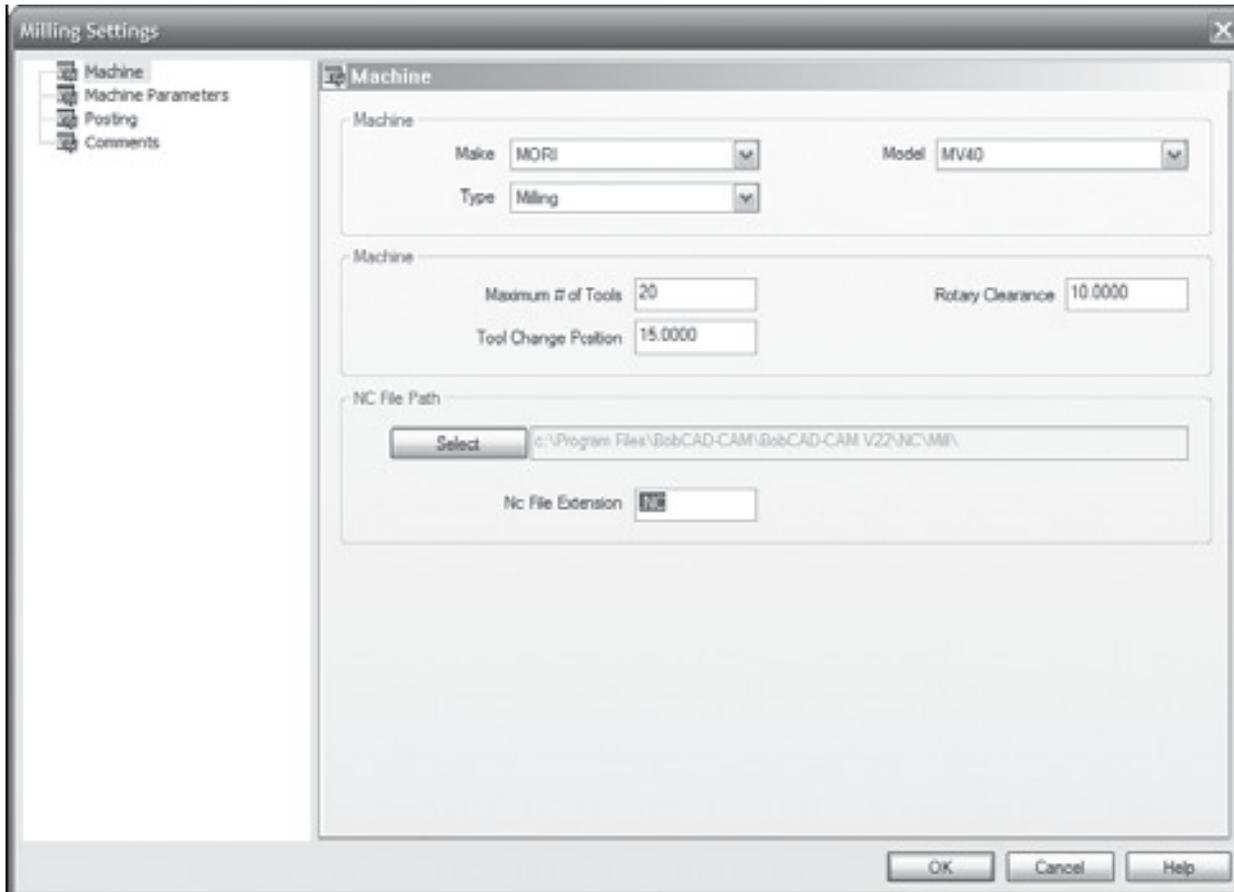
When the **Save As** dialog appears, name the file, choose the appropriate folder to save to, and press **Save**. The file can be saved directly to a floppy if desired.



If it needs to go to the machine on a CD-ROM, it will be easier to save it to another location and then burn the CD from there instead of attempting to save the file directly to the device.

File Formats & Extensions

BobCAD-CAM saves all NC files as ASCII text files. These files may have any extension the user wishes. To change the default NC extension BobCAD uses (when saving for any particular machine), right-click on **CAM Part** and choose **Current Settings**. The **Milling Settings** dialog box will appear. Click on **Machine** in the list to the left.



In the center of the box, there is a field named **NC File Extension**. Change the extension listed to the one recognized by the machine and click **OK**. Make sure to type the dot (".") before the extension letters.

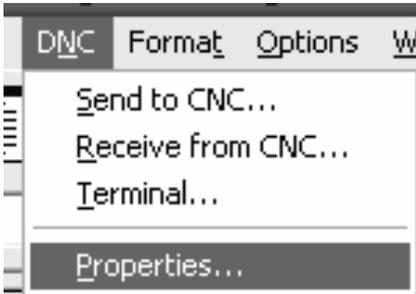
NOTE: ASCII text is the actual file format supported by probably 99% of all CNC machines ever manufactured, regardless of what file extension is required. If there is any doubt whether the machine accepts this format, post a program and try it. It is only in rare exceptions that this will not be the correct file format. If your machine takes another format by default, check with your machine OEM to see if the controller can accept ASCII text.

Establishing Communications

The Predator™ CNC Editor DNC included with BobCAD-CAM can communicate with a very wide variety of machines. It has quite a few parameters that can be set on a per-machine basis. Check your controller's manual for the settings preferred by your machine, as the settings must match between both the controller and Predator™.

Communication Settings

To set the Predator CNC Editor DNC communication settings, click on **DNC** and then **Properties** in the CNC Editor's main menu.

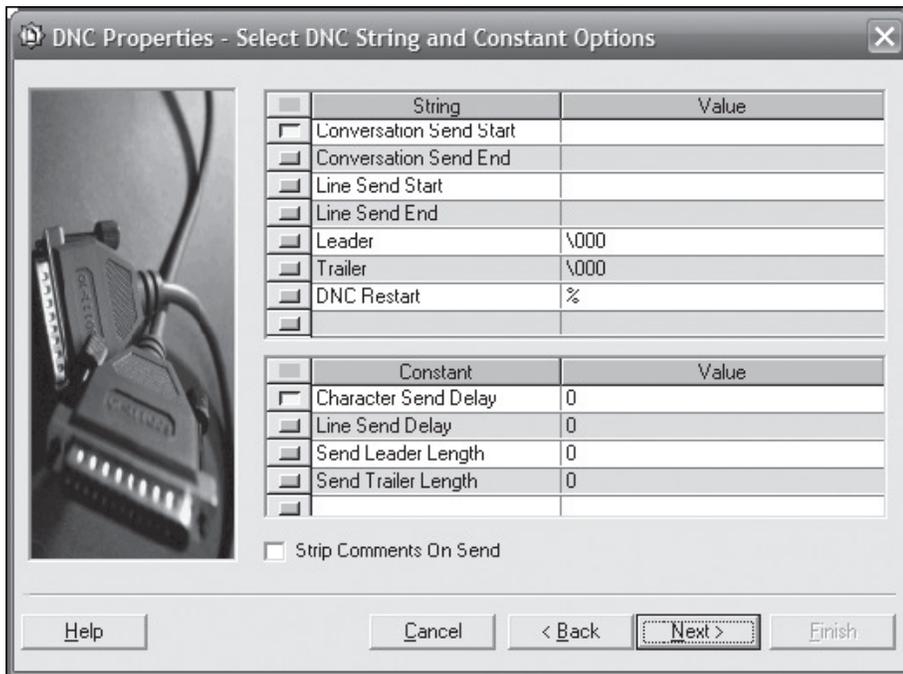


The system will display a series of 5 dialog boxes:

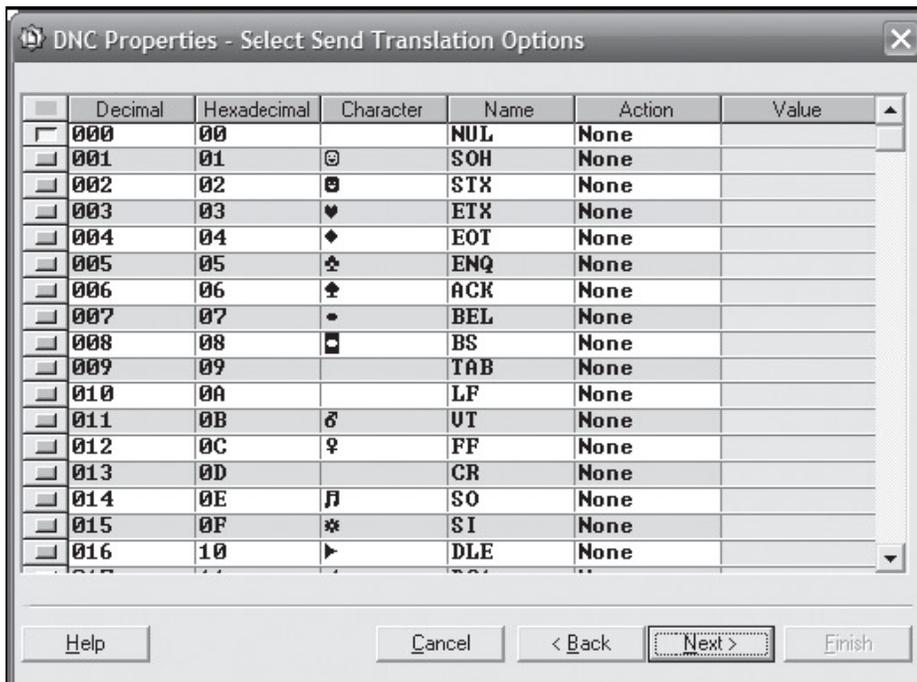
Select RS-232 Properties: Set the COM port options from here. Most controllers will require that these options be set to match the settings on the machine, but a few rare controls will actually require mismatched settings. See the controller's documentation to ensure the settings are correct.



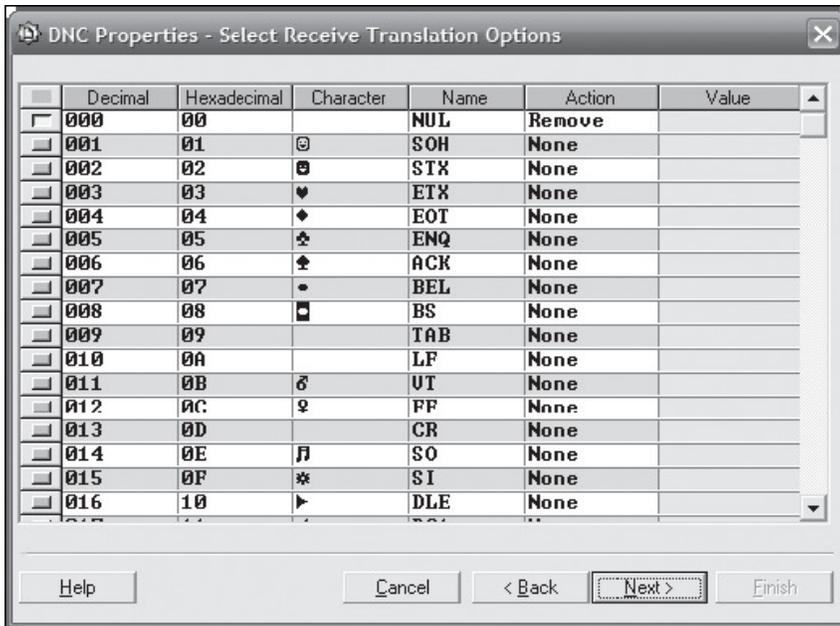
DNC String and Constant Options: Use this dialog box to set up any special leader or trailer characters the control may require.



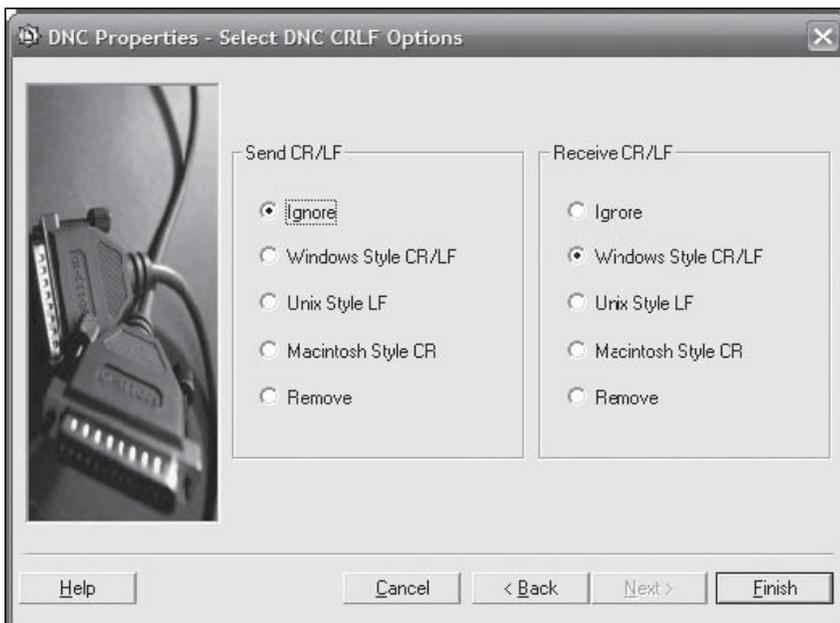
Send Translation Options: Occasionally a rare control will need a character normally output to be changed into something else entirely in order to read the transferred program. Normally no changes will be required; check the controller’s documentation to be sure.



Receive Translation Options: Some controllers will send undesired characters back to the computer. This dialog box can be used to either remove those characters or change them into something else.



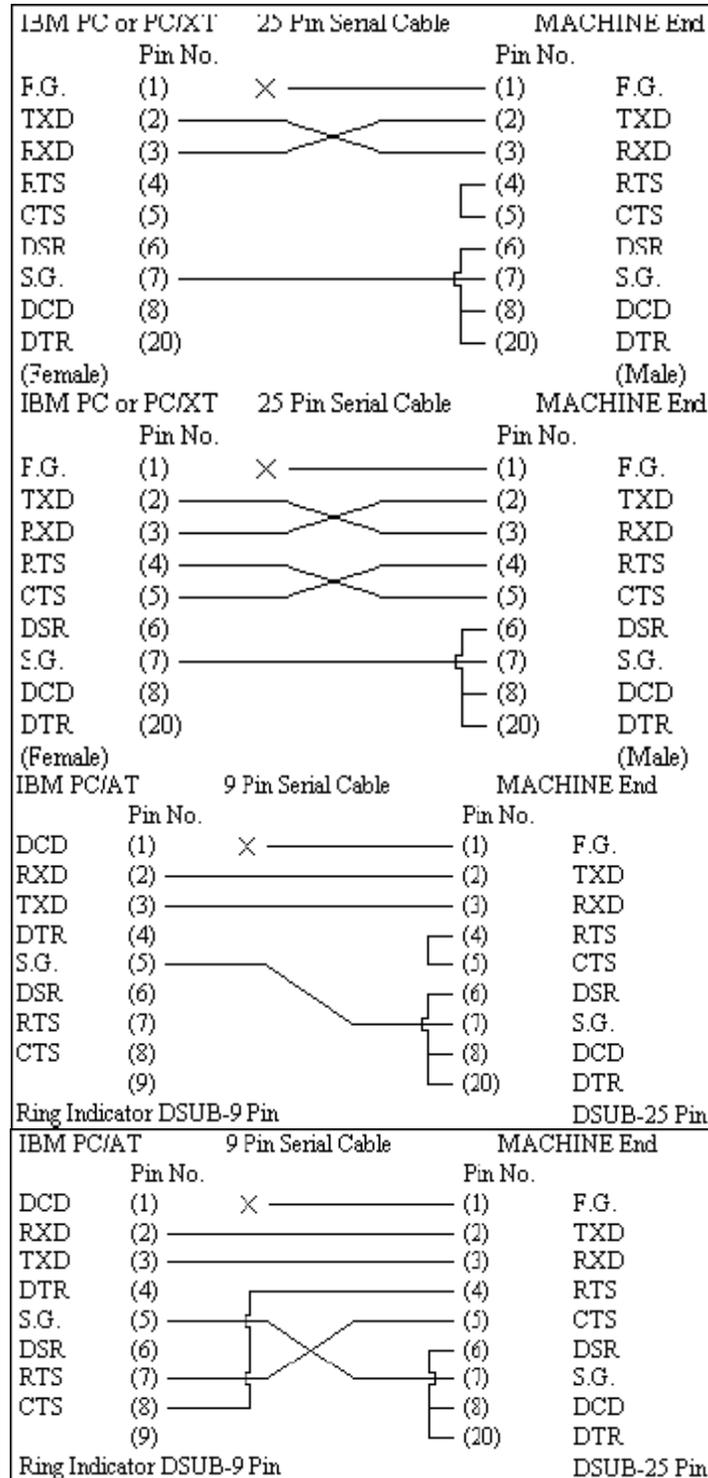
DNC CRLF Options: Some controllers require unusual Carriage Return / Line Feed (CRLF) character combinations. The options for both send and receive are listed in this dialog.



Click **Finish** when the settings are correct. The system will remember them for the next time a program is transferred.

Communication Hardware

To send a file over an RS-232 (or RS-422) link, a properly wired cable is absolutely required. See the following diagram for the most common wiring specifications. These are the common pinouts for nearly all CNC controls.



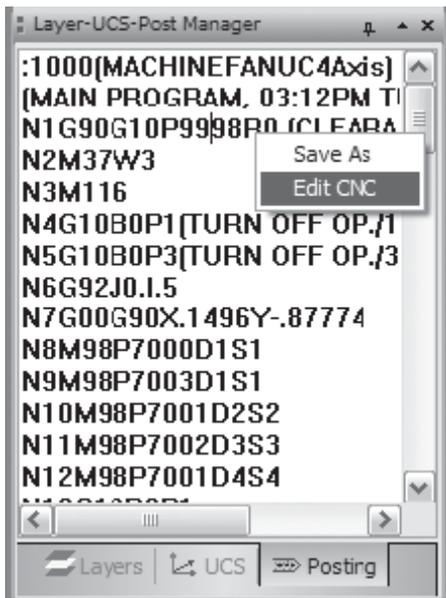
Cables typically need be shorter than 50' and will need to be shielded to help protect against electromagnetic interference. Take care not to coil a long cable or run any cable around any fluorescent lighting or near any electrical transformers or power supplies as these will significantly reduce the strength of the signal through the cable, possibly causing communication to fail entirely. If a good cable is difficult to find or to build, or if the machine is more than 50' from the computer, it is now possible to purchase Predator Grizzly™ cables directly from BobCAD-CAM, Inc. Grizzly™ cables are viable to lengths of up to 400' in a variety of environments.

Transferring Files via RS 232 (Serial) Link

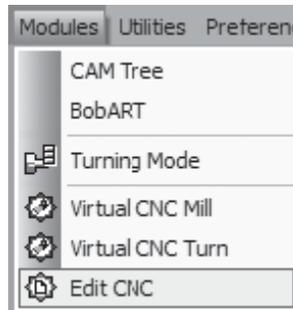
The Predator CNC Editor DNC can transfer a program to and from the CNC controller.

Here is the procedure:

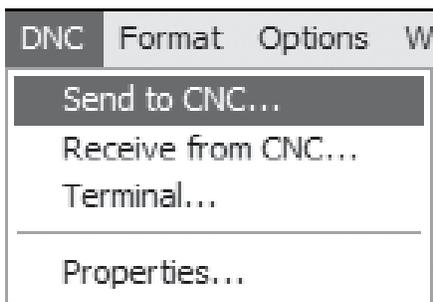
1. Open the CNC Editor by either right-click in the **Layer-UCS-Post Manager** and choose **Edit CNC**, or click on **Modules** in the main menu and choose **Edit CNC** from there.



or



2. Choose **Send to CNC...** from the menu.



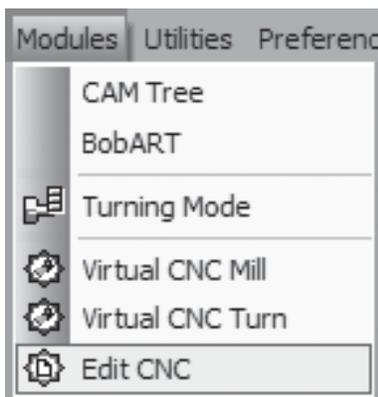
3. The CNC Editor will automatically begin the transfer. The progress bar near the bottom of the dialog will count off the percentage of the file transferred. The progress bar will display 100% when it is complete.



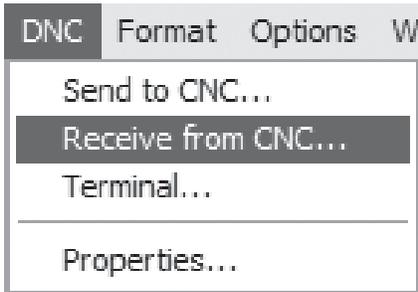
Receiving a Program from the CNC Control

Once again, the Predator CNC Editor DNC can transfer a program to and from the CNC controller. Here is how it is done:

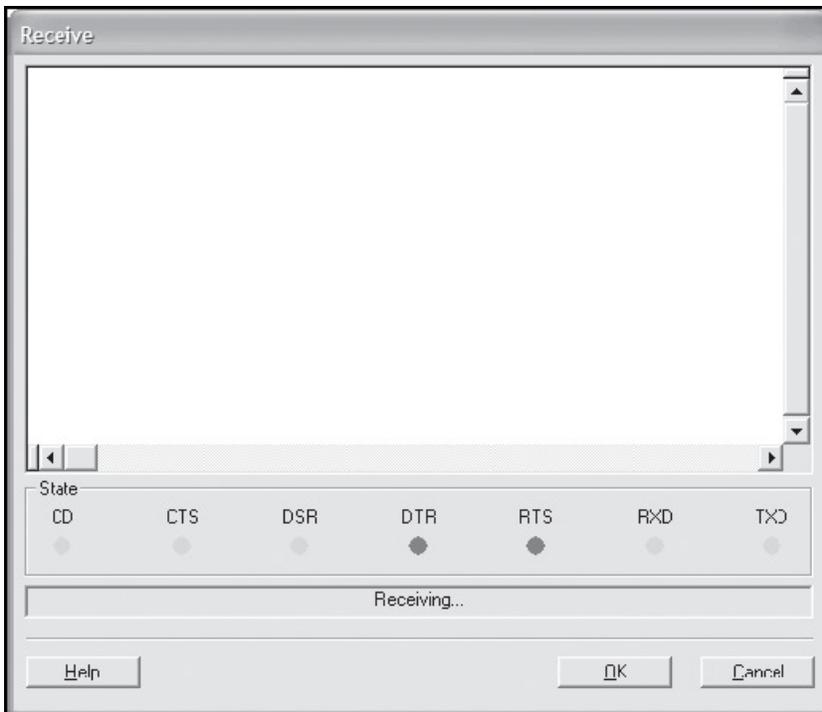
1. To open the CNC Editor, click on **Modules** in the main menu and choose **Edit CNC** from there:



2. Choose **Receive from CNC...** from the **DNC** menu in the CNC Editor.



3. The CNC Editor will automatically begin the transfer. The large box near the top of the dialog will display the code as it is received. The dialog will disappear when the transfer is complete.



That concludes this guide.