

Mill

Getting Started Manual

BobCAD-CAM



v23

Revision 3
August 2009

NOTE: The examples given in this manual are to be used for teaching the basic principles of the BobCAD-CAM software ONLY, and are written with only that goal in mind. Under no circumstances are they to be used for actual part production. Many of the settings and methods have been simplified and shortened to fit into the space allowed and to shorten the time spent waiting for operations to complete. BobCAD-CAM, Inc. claims no liability or responsibility of any kind relating to the use or misuse of the example parts herein.

Copyright © 2006, 2007, 2008, 2009 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. Address inquiries to Documentation Department, BobCAD-CAM, Inc., 28200 US HWY 19 N., Suite E, Clearwater, Florida 33716

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

Chapter 1. Introduction	6
1.1 System Requirements	7
1.2 Installation	8
1.3 Activating BobCAD-CAM	8
1.4 Levels of BobCAD-CAM	10
1.5 Accessing BobCAD-CAM	10
1.6 Using The BobCAD-CAM Version 23 Help system	11
1.7 Training Seminars	11
1.8 Web-Based Training	12
1.9 On-Site Training	12
1.10 Locating and Installing Post Processor Configurations	12
Installing Post Processors from the CD	13
1.11 Technical Support	14
Chapter 2. Files	16
2.1 Opening and Importing	16
2.2 Saving and Exporting	16
2.3 Supported File Types	17
Chapter 3. Using the Workspace	20
3.1 View Presets	20
3.2 Mouse Control	21
3.3 Multiple File Support in the Workspace	23
3.4 Copying Things from One Drawing to Another	23
3.5 Setting Part and System View Defaults	23
3.6 Layer Control	24
3.7 Hiding and Showing Entities	25
3.8 Work Planes	25
Chapter 4. Drawing	28
4.1 Wire Frame Drawing Example	28
4.2 Solid / Surface Example	32
Chapter 5. Generating Tool Paths and Code	42
5.1 The CAM Tree	42
5.2 How BobCAD-CAM Chooses Tooling	43
5.3 2D Pocket Example	44
5.4 Z-Level Rough Example	51
5.5 Drilling and Tapping Example	55
5.6 Posting a Program	59

Chapter 6. Verification	62
6.1 Starting the simulation.....	62
6.2 Controlling the simulation.....	62
6.3 Example	64
Chapter 7. Getting Code to the Machine	68
7.1 Saving Posted Programs	68
7.2 A Word on File Formats and Extensions	69
7.3 Establishing Communication.....	70
Communication Settings	70
Communication Hardware	73
7.4 Transferring Files via RS-232 (Serial) Link	75
Send a Program to the CNC	75
Receive a Program from the CNC.....	76

BobCAD-CAM V23

Chapter 1 Introduction

Mill

Chapter 1. Introduction

Thank you for your purchase of BobCAD-CAM Version 23. We at BobCAD-CAM, Inc. hope that it becomes an indispensable tool in your shop toolbox. Every function of the software is thoroughly tested in shops across the world to ensure the most accurate and reliable results.

There is no way to list all of the features of BobCAD-CAM in a single page in a manual. With that in mind, here are some highlights:

- support for dozens of industry-standard file formats to help ensure compatibility with your customers' designs
- post processor support for programming hundreds of machines
- built-in software toolpath verification from Predator Virtual CNC with the option of upgrading to more powerful versions for those who require more control over their toolpath and machine visualization
- built-in program editing and DNC using the Predator CNC Editor with the option of upgrading to higher levels for editing and communicating with even more machine types
- new toolpath routines in addition to our already robust, proven technology to help use your CNC machines to their fullest extent
- improvements to our existing machining technologies to make them even better than ever
- feature-driven program creation that gives control of the program to the user

This guide is set up to help the new user of BobCAD-CAM Version 23 become acclimated to the software quickly and with minimum effort. For ease of use, it is broken into 7 parts:

- Chapter 1. Introduction
- Chapter 2. Files
- Chapter 3. Using the Workspace
- Chapter 4. Drawing
- Chapter 5. 2D and 3D Toolpath Generation
- Chapter 6. Verification
- Chapter 7. Getting Code to the Machine

We at BobCAD-CAM, Inc. have put a lot of effort into improving the user experience of the software. We hope you will find it a welcome and essential addition to your shop toolbox. In all, we believe BobCAD-CAM V23 to be the best BobCAD-CAM product to date. We are certain that you will find the same to be true.

1.1 System Requirements

The following are the recommended system requirements for BobCAD-CAM Version 23.

Minimum System Requirements:

- 1.0 GHz Processor
- 1 GB Ram (1024 MB)
- 2 GB Available Space on Hard Disk
- Windows 2000, XP, or Vista
- 128 MB Graphics Adapter that supports OpenGL 1.1

If the system has a shared graphics chipset and not an added card, the computer should be equipped with at least 1.0 GB of RAM.

Recommended System Requirements for installation:

- 2.0 GHz Processor or higher
- 2 GB RAM (2048 MB)
- 2 GB Available Space on Hard Disk
- Windows 2000, XP, or Vista
- 512 MB Video Graphics card or higher that supports OpenGL 2.0.

The BobCAD-CAM Version 23 system is a solids and surface modeling system. If using a computer that has less RAM than the minimum requirements listed above, there may be delays in executing functions that relate to rendering models, generating toolpath and G-Code programs. Understand that these delays are not caused by the BobCAD-CAM software. By upgrading the computer to the recommended requirements above, those functions will execute more effectively.

It is highly recommended to use a minimum screen resolution of at least 1024x768 pixels. Any smaller may make it difficult to navigate the menus and toolbars on the screen.

Note for Microsoft Vista Users:

All current versions of the BobCAD-CAM software will work on Microsoft Windows Vista 32 and 64 bit systems. It is highly recommended for performance reasons to use Vista Home Premium or greater.

While BobCAD-CAM will install and run on both 32 and 64 bit systems, it will not take advantage of the extra processing power on 64 bit computers, but will operate

in the same way on both types. Before upgrading any existing Microsoft Windows 98/2000/XP machine to Microsoft Windows Vista it is highly recommend to consult a Computer Repair technician.

To ensure proper operation when using BobCAD-CAM on a Vista computer under a normal user account, instructions for modifying the user permissions for running the BobCAD software programs can be found at <http://www.bobcadsupport.com/> under *Vista & Re-key Information*.

1.2 Installation

BobCAD-CAM Version 23 is an official BobCAD-CAM software product, developed by BobCAD-CAM for automating the manufacturing process.

To install BobCAD-CAM V23, follow these steps:

1. Insert the BobCAD-CAM Version 23 CD into the CD drive of the computer.
2. The software will automatically install. Go through the installation wizard until it has completed the loading and installation process. The installer will display a prompt when the software is fully installed.
3. Start BobCAD-CAM. The software will begin its 5 calendar days in a fully-functional mode until it is registered.

When installation process has completed, the CD is no longer needed in the CD drive and can be removed, and the software will be ready for activation.

1.3 Activating BobCAD-CAM

Password Method: The standard way to activate BobCAD-CAM V23 is password licensing. Once the software has been installed, follow these steps to activate it using the password licensing method:

1. Start BobCAD-CAM V23, and click on **Help** in the main menu. From the menu, choose **Activate License**.
2. Find the **License ID** number on the original invoice included in the box with the software and use it to help fill out the *Version 23 Software Password/Manual Registration Form* also included in the box. The License ID number and the three ID numbers that will appear in the dialog are required.

3. Fax or email the form in to BobCAD-CAM, Inc. at the fax number or email on the form. BobCAD-CAM will fax or email back the 4 required numbers on the bottom portion of the form.
4. Open the **Activate License** dialog again through the **Help** menu and then **Activate License**. Fill in the fields with the numbers that have been returned to you. Click **OK** in the box, then close and reopen BobCAD-CAM V23. The software will be licensed and ready to go.

Hardlock Method: Optional hardlock key licensing is also available for an extra fee from BobCAD-CAM, Inc. To activate the software using this method, follow these steps:

1. Close BobCAD-CAM if it is open and insert the hardware key into an available USB port on the computer.
2. Start BobCAD-CAM V23. The software will automatically read the hardware key and activate the modules that it is licensed for.

To verify which modules the seat has been licensed for, click on **Help** in the main menu. From the menu, choose **License**. In the dialog, verify that the appropriate boxes are checked for this seat of the software.

IMPORTANT!

If your seat of BobCAD-CAM is licensed with the optional hardware key, do NOT lose the key! It is the license to your software! The keys cannot be replaced without purchasing another license. It is suggested that the owner of a hardware-key-protected seat of BobCAD-CAM Version 23 purchase insurance to protect against the loss or theft of the dongle. Keeping track of it is impossible to overstress.

If the user wishes to switch from the passcode licensing system to use a hardware key instead, the software must first be de-authorized with the software system.

See **FAQ #36: How do I move V23 to another computer?** under *V23 Frequently Asked Questions* at the BobCAD-CAM website (www.bobcad.com) and click on the **Support** button at the top of the page to see instructions for de-authorization.

BobCAD-CAM V23 will run in a fully functional mode for 5 calendar days after it is first launched, then will revert to a restricted mode if the software is not yet registered.

This restricted mode has the following:

- Machine communication (DNC, etc.) will not run
- It will post process only the first 20 lines of NC code
- .bbcd files created in restricted demo mode cannot be loaded into BobCAD-CAM in Licensed mode. Files that are created in Licensed mode can be viewed in this demo mode. If those files are then saved in this mode however, they will revert to restricted demo mode files and cannot be re-opened in a licensed copy, so it is a good practice to always use the dongle anytime BobCAD-CAM is in use if it is licensed with one.

Other than these, an installation of BobCAD-CAM Version 23 in restricted mode will be functionally identical to one in licensed mode. After the 5 day period, the software *must* be licensed to be functional.

1.4 Levels of BobCAD-CAM

BobCAD-CAM Version 23's milling functionality comes in 3 different levels to help the user get what is needed at a price that is affordable. Each level includes everything in the previous level and adds what is listed here.

- BobCAD-CAM Version 23 Mill 2X: 2 Axis Laser/Waterjet/Plasma – Profiling routines for 2 Axis cutting machines.
- BobCAD-CAM Version 23 3D Standard: 2 1/2 Axis Programming – Pocketing, Profiling, 2D Engraving, Drilling, 3 Axis Slice programming routines.
- BobCAD-CAM Version 23 3D Pro: All of what comes in Standard, plus upgraded Z-Level Rough and Finish toolpaths, and the Equidistant Offset 3D toolpath for the best part finishes.

1.5 Accessing BobCAD-CAM

There are two ways to access the BobCAD-CAM Version 23 system.

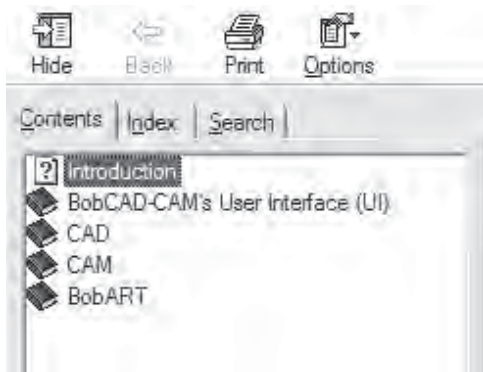
- Directly from the desktop. After the installation is complete an icon will be loaded onto your desktop. Double click on this icon to launch BobCAD-CAM Version 23.
- Directly from the Start menu. Click the Start button from the taskbar, then choose Programs and then BobCAD-CAM Version 23 to launch the software.

1.6 Using The BobCAD-CAM Version 23 Help system

BobCAD-CAM Version 23 includes an extensive help system to anticipate questions and to help with user needs. To access the Help system, simply click on **Help** in the main menu or press the F1 key on your keyboard.

BobCAD-CAM's Help system works similar to help systems in other Windows-based programs. It contains 3 tabs:

- **Contents** – lists the help topics according to which menu they appear in.
- **Index** – the Index lists all of the topics alphabetically.
- **Search** – if it is known which idea is needed but unsure where it may be in the software or the exact name for it, search terms may be typed in and the Help system will search all topics to attempt to find the appropriate topic.



1.7 Training Seminars

BobCAD-CAM customers can attend special 3-day training seminars in areas all over the United States. These training seminars are pre-scheduled in all major cities.

To find out more about a scheduled 3-day training seminar, visit the official BobCAD-CAM website at www.bobcad.com or contact the training department directly at **877-262-2231** or **727-442-3554**. Course certification is provided for the completion of a 3-day class and after having met all class requirements.



1.8 Web-Based Training

BobCAD-CAM customers also have the option of purchasing individual training by the hour on the topic of their choice in a conveniently scheduled Web-based environment. Instructors will provide guidance through whatever topics are necessary on a one-on-one basis and will answer every question.

To schedule a Web-based training session, call **877-262-2231** or **727-442-3554**, or the Technical Support Department directly at **727-489-0003** and a representative will help schedule the training.

1.9 On-Site Training

BobCAD-CAM also offers On-Site Training for customers who are unable to attend scheduled 3-day training classes or would prefer to get trained at their own manufacturing facility. If this is preferable to a 3-day seminar or Web-based training, BobCAD-CAM will train CNC operators and machinists right there in the shop. At the conclusion of the training, professional certification will be provided for those that attend the entire class. For advanced scheduling, information and costs please contact BobCAD-CAM directly at: **877-262-2231** or **727-442-3554**.

1.10 Locating and Installing Post Processor Configurations

The BobCAD-CAM Version 23 system includes a variety of post processors and the ability to customize existing post processor configuration files.

AVAILABLE POST PROCESSORS

3. 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 application. From there, the available post processors are sorted alphabetically by make and model. Locate the necessary post processor and download 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 and the post will install and be ready for immediate use.
4. BobCAD-CAM V23 ships with a second CD containing all of the post processors available. See the steps outlined below for installing from CD.

POST MODIFICATION REQUESTS: BobCAD-CAM Version 23's post processors should not need any changes. However, in the off chance a change is necessary, you may request post processor modifications at no additional charge by contacting technical support at **727-489-0003**.

Post processor requests are serviced on a first-come, first-served basis. Some

modifications can take more time to implement than others, but all requests will be completed as fast as possible. Please allow up to 2 weeks for delivery on any new post processors and modifications; this time will vary depending on request volume and configuration complexity.

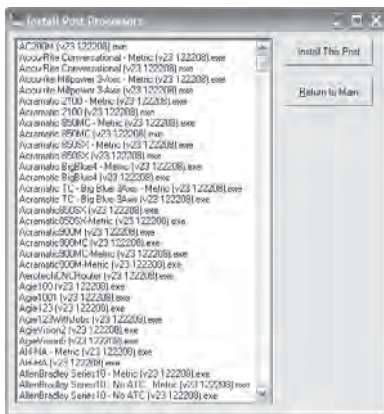
Installing Post Processors from the CD

Step 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**.



Step 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 the that post will install. Repeat this step for each post processor needed. Simply close the application when finished.



1.11 Technical Support

BobCAD-CAM offers customer technical services & software support by phone, fax and by email for all **Technical Support** members.

BobCAD-CAM technical support representatives are standing by to assist with the software if needed. BobCAD-CAM offers Technical Support for all BobCAD customers so that special phone support is available when necessary. Technical Support membership offers a wide range of support benefits and is recommended.



Technical Support is available for calls during the following hours:

8 AM - 6PM Monday- Friday Eastern Standard Time

10AM - 2PM Saturdays Eastern Standard Time

Technical support will be closed on all major holidays recognized in the United States.

The support PHONE line is: 727-489-0003

The support FAX line is: 727-734-8239 - ATTN: Technical Support

The support EMAIL is: support@bobcad.com

IMPORTANT: If not yet signed up for a Technical Support annual membership, all customers may do so by calling **877-262-2231** or the technical support phone number above.

BobCAD-CAM V23



Chapter 2 Files

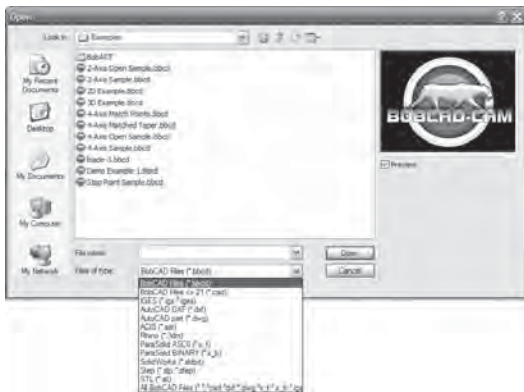
Mill

Chapter 2. Files

All file Import/Export/Open/Save operations are done in the **File** menu at the top of the screen.

2.1 Opening and Importing

Opening and importing files is the same thing in BobCAD-CAM. Both interpret the data in a file and convert it directly for immediate use. Once a file has been opened, all geometry in it is the same as if it had been drawn in BobCAD from the beginning. To open or import a file, click on **File** in the main menu and then on **Open**. Find the file on the disk or on your hard drive, click on it in the box, and then click **Open**.



2.2 Saving and Exporting

Saving and exporting files is also the same thing in BobCAD-CAM. Which ever word is used, it still means to take data from BobCAD-CAM, convert it to some format, and write it to a file.

To save a file, simply click on **File** in the main menu and then on **Save**. If the file has not been saved before, the **Save As** dialog box will appear and you will be able to type in a name for the file and choose the type you wish to save it as.

If at any time you need to save a file to a different format or under a different name, choose **Save As** instead of **Save** and you can get back to this box.

It is important to stay within the Windows™ file naming guidelines when naming your files. Each name can be up to 255 characters long including the

names of all of the folders above it, and must not contain any of the following special characters:

? [] / \ = + < > ; : " , | *

2.3 Supported File Types

- **.bbcd** – Native BobCAD-CAM™ file. Supports CAD and CAM data. Can be both opened and saved.
- **.bbas** - Native BobArt Surface file. Can be opened and saved.
- **.cad** – Older native BobCAD-CAM file types. Can be opened only.
- **.igs / .iges** – International Graphics Exchange Standard. Used for transferring CAD data between systems. Can be opened and saved.
- **.dxf** –AutoDesk™ Document eXchange Format. Can be opened and saved.
- **.dwg** – AutoDesk™ DraWinG file. Can be opened and saved.
- **.sat** – ACIS™ solid file. Can be opened and saved.
- **.3dm** – Rhinoceros™ 3D file. Can be opened only.
- **.x_t** – ASCII ParaSolid file. Can be opened only.
- **.x_b** – Binary ParaSolid file. Can be opened only.
- **.sldprt** – SolidWorks™ part file. BobCAD-CAM will read up to rev. 2008 of these. Can be opened only.
- **.stp / .step** – SStandard for the Exchange of Product model data. Can be opened and saved.
- **.stl** – A STereoLithography file. This is a mesh/faceted data format. BobCAD-CAM Version 23 can generate 3D toolpath directly from .stl meshes. Can be opened and saved.

BobCAD-CAM V23

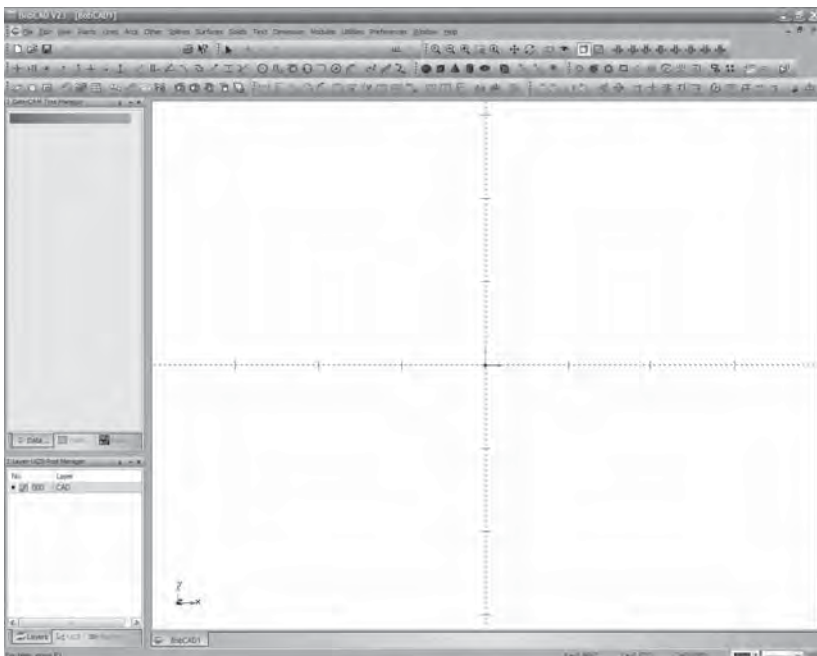


Chapter 3 Using the Workspace

Mill

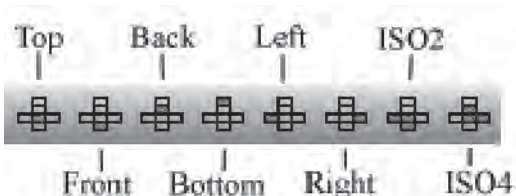
Chapter 3. Using the Workspace

BobCAD-CAM Version 23's **Workspace** has several parts that the user will need to be familiar with.



3.1 View Presets

BobCAD-CAM Version 23's **Workspace** has several presets to help view parts in whatever way necessary. Here are the 8 most often used views, labeled:

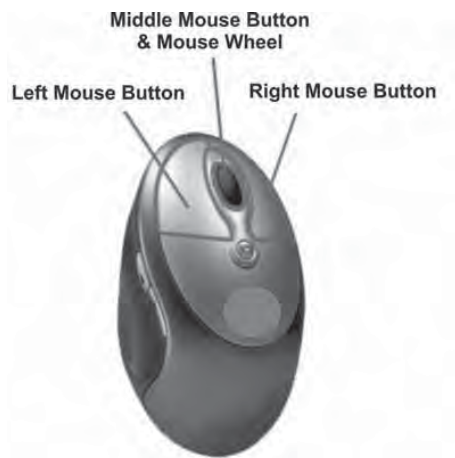


Individual settings for the view, including quality, colors, etc., are all listed under the **Preferences** menu in **Settings Part** and **Settings Default**.

3.2 Mouse Control

BobCAD-CAM Version 23 offers mouse functions that can be used in conjunction with the Shift & Control keys for:

- Panning
- Dynamic Zoom
- Zooming in a Window
- Dynamic Rotation
- Accessing the Utility Drop-Down Menu
- Geometry Selection & Window Picking



- *Left Mouse Button*

The Left mouse button is used primarily for selection. The left mouse button also enables other tasks when used in conjunction with other keys on the keyboard.

To select an entity, face, or solid, rest the cursor over the entity or face that is to be selected, and click the left mouse button to select the entity.

- *SHIFT + Left Mouse Button*

When in selection mode, the left mouse button and shift key on the keyboard combined can be used to chain select geometry in the workspace.

- *CTRL + Left Mouse Button*

The control key with the left mouse button is used for Zoom Window mode.

To zoom in on an area: Hold down both the *CTRL* key and the left mouse button and then drag the cursor over an area of the workspace to zoom in on that area.

- *Middle Mouse Button and Wheel*

If the mouse has a wheel it can be used as a middle or second mouse button for accessing functionality.

Zooming In: scrolling the wheel toward the front of the mouse will cause the software to zoom in toward the center of the screen.

Zooming Out: scrolling the wheel toward the back of the mouse will cause the software to zoom out away from the center of the screen.

- *CTRL + Wheel or Middle Mouse Button*

The control key and middle mouse button combined can be used to adjust the vertical and horizontal position of the drawing within the workspace.

Panning: Hold down the *CTRL* key on the keyboard and the wheel or middle mouse button to freely pan and move the drawing around the workspace.

- *Right Mouse Button*

The right mouse button is used to access the pop-up menu, indicate that selections are completed, cancel a selection, use verify functions, change views and access the **Selection Mask** settings. Clicking the right mouse button and choosing Selection Mask from the drop down menu can access the selection mask options. See below for more on this box.

- *CTRL + Right Mouse Button*

Dynamic Rotation: When holding down the control key on the keyboard and the right mouse button simultaneously, the user is placed in dynamic rotation mode. The user can then move the mouse to rotate the part in the workspace freely.

3.3 Multiple File Support in the Workspace

BobCAD-CAM allows the user to have multiple files open at the same time. If more than one file is open, each one will show up on its own tab at the bottom of the **Workspace**. To switch between them, click on the tab for each one at the bottom.

In the **Window** menu, the user has more methods of controlling how parts are displayed. There are 6 parts:

- **New Window** – this option will create a new tab at the bottom of the Workspace. The new tab is a copy of the existing active drawing.
- **Cascade** – if the user has several drawings open at the same time, the windows displaying the geometry will be reordered to tile on top of each other. The tabs at the bottom will still switch between them.
- **Tile Horizontal** – this will organize multiple drawings in the view stretched out horizontally. This means that the individual drawings will be stacked on top of one another.
- **Tile Vertical** – this will organize multiple drawings in the view stretched out vertically. This means that the individual drawings will be stacked next to one another.
- **Arrange Icons** – if all of the active drawings are minimized within the Workspace and scattered around, this will arrange them in order on the bottom of the view.
- **Active window list** – this section will list all of the drawings that are currently loaded in the Workspace. The active drawing will have a check next to its name.

3.4 Copying Things from One Drawing to Another

For most users there will come a time when parts of one drawing will need to be copied into another. To do this:

1. Highlight the parts of the first drawing to be copied.
2. Click on **Edit** in the main menu and choose **Copy**, or press *CTRL+C* to copy the items to the Windows clipboard.
3. Click on the tab of the drawing it has to be copied to at the bottom of the **Workspace**.
4. Click on **Edit** in the main menu and choose **Paste**, or press *CTRL+V*. The copied parts will appear in the second drawing in the same place they were originally.

3.5 Setting Part and System View Defaults

Part viewing preferences are set for the currently active drawing by clicking on **Preferences** and then **Settings Part**.

All settings for viewing the current part are set here. These include background color, new entity color, the highlight color, the quality of the part rendering, etc. Also set here are the current units the part and all operations are done in, whether in inch (SAE), metric, or some other type of unit.

Settings Default has the same box, but the settings there apply to new drawings only. There is also a **Directories** category not present in **Settings Part** on the left where the user can set the default folders to retrieve data from and save to.

The **Workspace** has a few default shortcut keys of its own for part display:

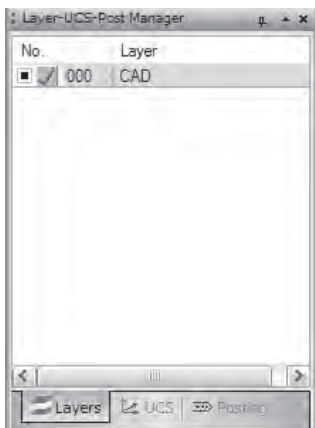
W – toggles wire frame view on and off

S – toggles shaded view on & off

T – toggles transparency for every solid or surface visible

3.6 Layer Control

All layer handling in the software is done through the **Layers** tab of the **Layer-UCS-Post Manager** box.



To create a new layer, right-click in the **Layer-UCS-Post Manager** and choose **Add New Layer**. Name the new layer whatever is necessary. BobCAD-CAM Version 23 keeps layers separated in the background by their numbers, not their names, so it is possible to name 2 layers the same if required.

To switch to another active layer, click on the **Inactive Layer** icon for the layer needed. It will change to a check indicating that it's active. Afterward, all new entities will be drawn on that layer.


To change the layer some existing entities are on, make the layer active first as above. Then, highlight the entities to be moved to the new layer. Right-click in the **Workspace** and choose **Modify to Current Layer**. BobCAD-CAM will then place all of the selected entities on the active layer.

3.7 Hiding and Showing Entities

To hide and show all entities on a layer, click on the **Layer Visible** button. When there is a square in the box, the entities on that layer are visible and when there is not, they are hidden.


To toggle between showing and hiding all of the entities on a single layer, simply toggle the **Layer Visible** button as in the description above. However, to show or hide only a few entities, use the **Blank** and **Unblank** icons:

To hide only a few entities:

1. Click on the **Blank** icon  on the toolbar
2. Left-click on all of the entities that need to be hidden to select them
3. Right-click and choose **OK** in the pop-up menu.

All of the selected entities will be made invisible.

To show some of those entities again:

1. Choose the **Unblank** icon  on the toolbar. All of the currently blanked entities will show and BobCAD-CAM will temporarily hide the currently visible parts of the drawing
2. Click on those entities that need to show again to get them highlighted. Right-click and choose **OK** in the pop-up menu. Those entities that were highlighted will be unblanked and the drawing will be returned to normal.

3.8 Work Planes

BobCAD-CAM Version 23 allows the user to define the working planes for parts. There are three pre-defined planes and the user can create as many others as necessary. When a plane is set active, like layers, all new entities are drawn on that plane. The pre-defined planes are the **Top (X/Y)**, **Front (X/Z)**, and **Side (Y/Z)** planes.

There are several ways to create a new plane. See the Help system for a detailed explanation of all of the methods available.

What follows is an example of using the 3 Points method:

1. Click on the **Point** menu, and then on **Coordinates**. The dialog in the **DATA-CAM Tree Manager** will display fields for the coordinates of the points. 3 points are required.

2. Enter in the values in the fields. Click **OK** after each one.

X:0, Y:0, Z:0

X:1, Y:0, Z:0

X:0, Y:sin(45) Z:cos(45)

Typing in the trigonometric functions will cause BobCAD-CAM to evaluate those functions and change the number in the box to the result of the calculation. It is possible to type in “.7071” in both the **Y** and **Z** fields, but in reality BobCAD-CAM can support many more decimal places than it may be set to display. Using trigonometric functions such as sin(), cos(), and tan() is much more accurate as BobCAD-CAM will evaluate them as far out as it can, i.e. to 15 decimal places.

3. Right-click on the **UCS** tab of the **Layer-UCS-Post Manager**, and choose **Add New UCS**.
4. Choose the 3 Point button in the Data-CAM Tree Manager.
5. Click the points in the order they were drawn in to create the new work plane. This new plane will be rotated 45 degrees off of the **Front** plane around the X-axis, because the **Y** and **Z** numbers are 45 degrees from (0, 0, 0). It is important to click them in this order because the first point highlighted becomes the origin position (0, 0, 0) of the new work plane, the second becomes the direction of the new +X axis, and the third becomes the new +Y.

BobCAD-CAM



V23

Chapter 4
Drawing

Mill


Chapter 4. Drawing

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.

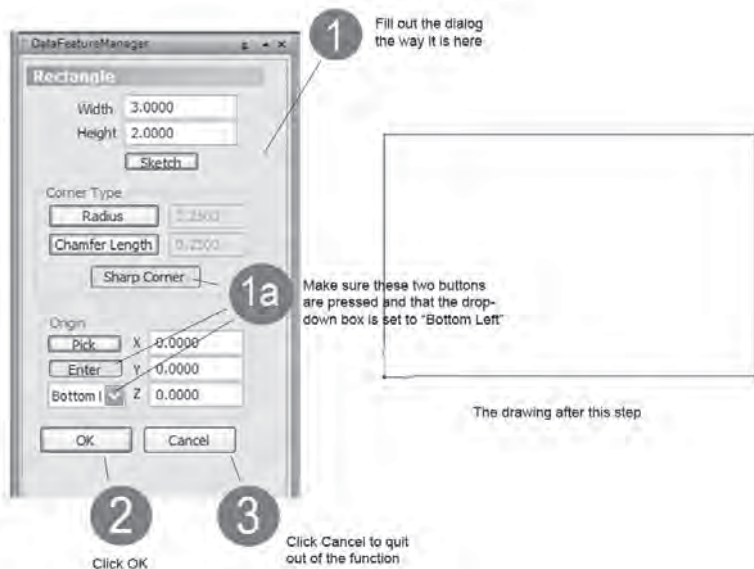
All of the functions used in this chapter are available in all levels of BobCAD-CAM.

4.1 Wire Frame Drawing Example

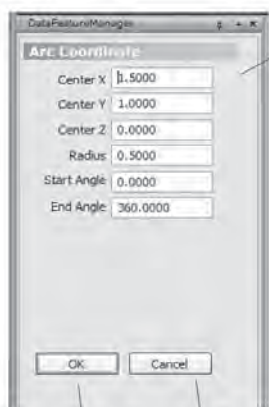
Step 1: Begin with an empty drawing by using any one of these 3 methods:

- Click on **File** in the main menu and then **New OR**
- Click on the **New File** icon in the toolbar  OR
- Type CTRL+N to start a new drawing.

Step 2: Click on **Other** from the main menu, and then click on **Rectangle**.

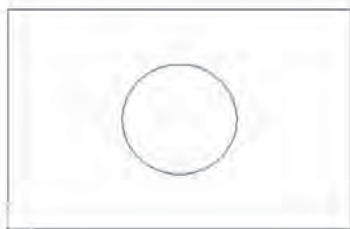


Step 3: Click on **Arcs** from the main menu, then **Coordinates**.



1

Fill out the dialog as it appears here



The drawing should appear like this

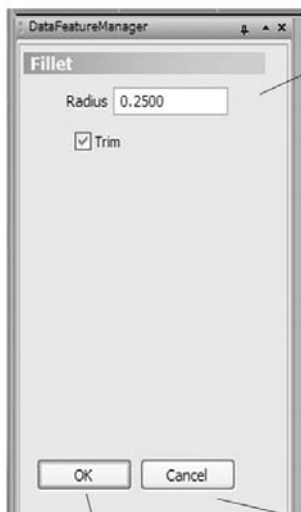
2

Click OK

3

Click Cancel to exit the function

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

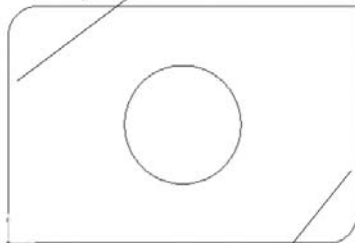


1

Fill out the dialog as shown

3

Click the top line first, then the left line



4

Click on the bottom line, then on the right

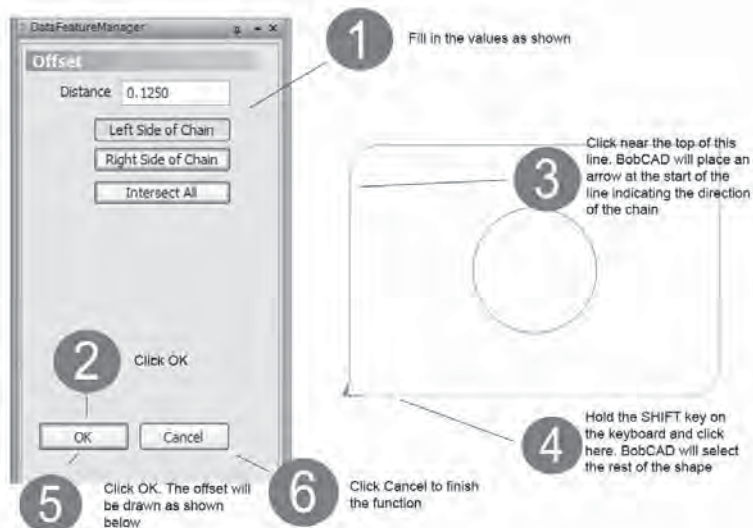
2

Click OK

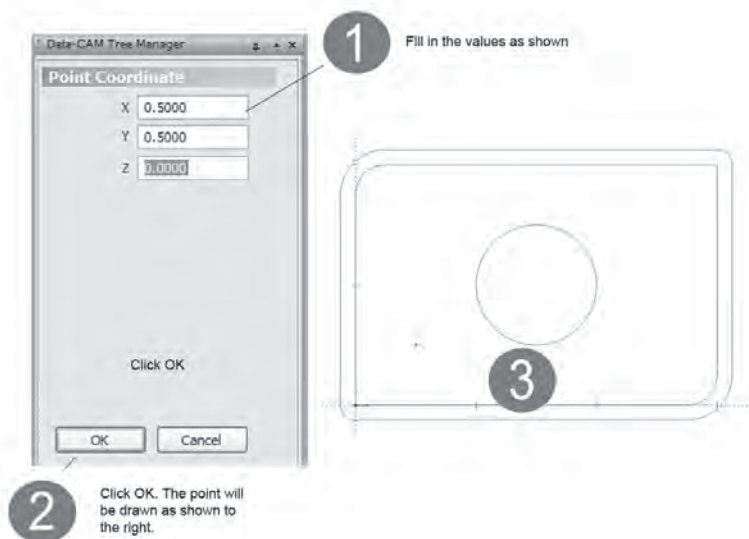
5

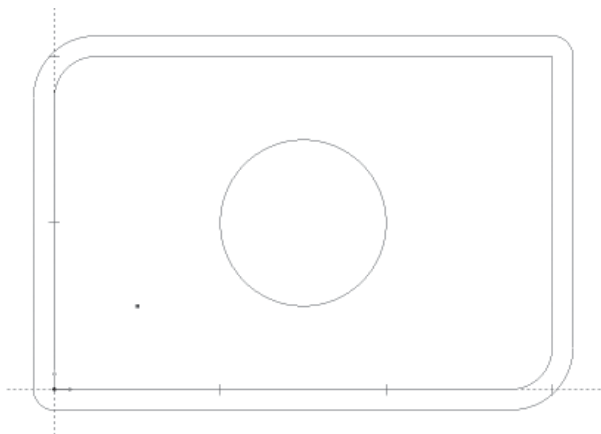
Click Cancel to end the function


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



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

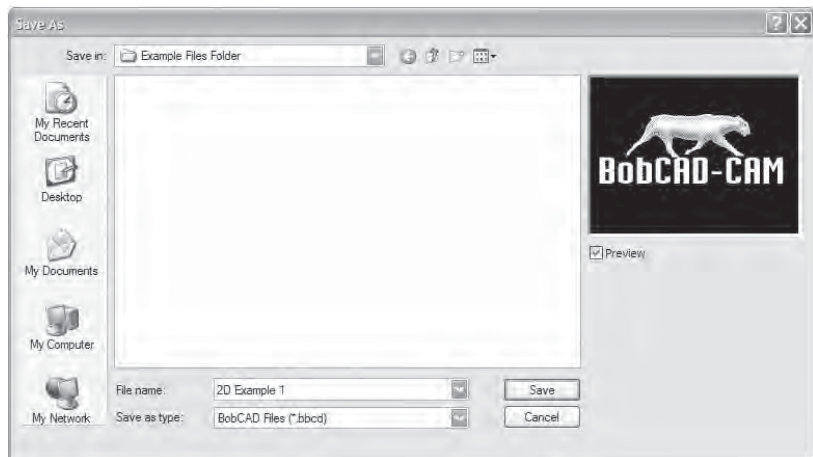




In the main toolbar at the top, click on this icon: . This will center the drawing in the view.

Step 7: Click on File in the main menu and then on Save. Type in “2D Example” in the File Name box and then click on the Save button. Keep in mind what folder the file was saved in; it will be used again in later examples.


***Note:** Make sure you remember what folder you save the file in. At the top of the box, in the **Save In** drop-down box, you can choose nearly any folder on the computer.*



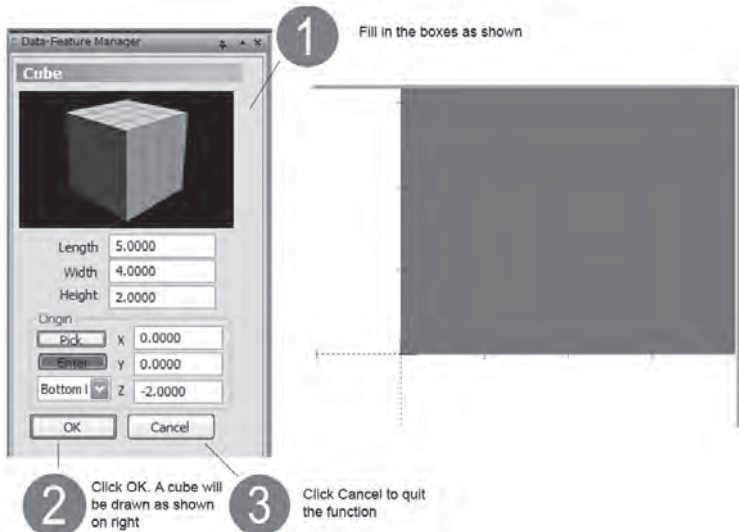
4.2 Solid / Surface Example

This example is long but illustrates some of the most useful functions when drawing with solids and surfaces.

Step 1: Begin with an empty drawing by using any one of these 3 methods:

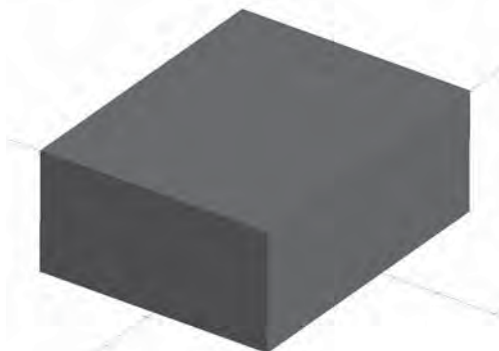
- Click on **File** in the main menu and then **New OR**
- Click on the **New File** icon in the toolbar  OR
- Type CTRL+N to start a new drawing.

Step 2: Click on the **Solids** menu, and then on **Cube**.



For better viewing, click the **ISO1** view button on the toolbar:

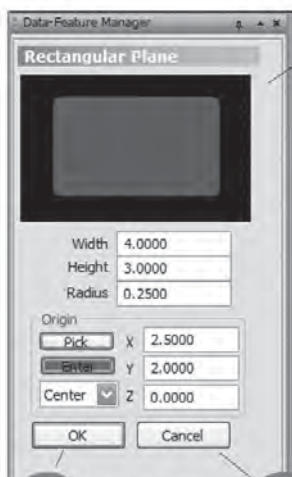




.. and if the view is not centered as shown above, click on the **Zoom Fit All** button:



Step 3: Click on the Surfaces menu, and then on **Rectangular Plane**.



1

Fill in the values as shown



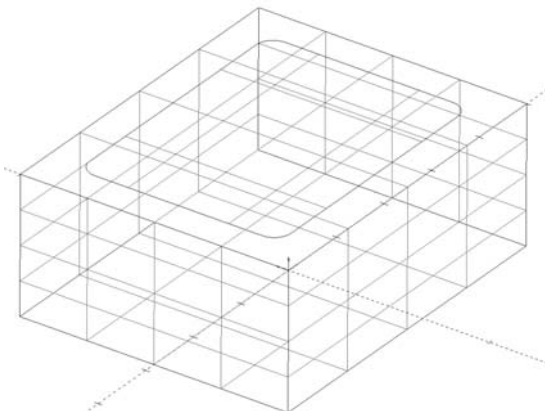
2

Click OK. The plane will appear as shown at left

3

Click Cancel to quit the function

The plane will likely be difficult to see when in a shaded view mode because both it and the top surface of the cube occupy the same 3D space. The plane will be better visible in a wire frame view. To change the view mode, turn off shading by pressing “s” on the keyboard, and turn wire frame on by pressing “w”. These will change the view of the part to look like below:



Step 4: Click on the **Surfaces** menu again, and then on **Extrude Surface**.

1 Fill in the values as shown

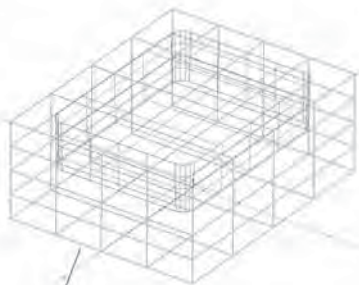
2 Click OK to accept these settings

3 Click on the wire frame of the plane as shown here. It may be difficult to tell which wire frame is which, so be careful to make the selection look like this

4 Click OK again to apply the extrusion

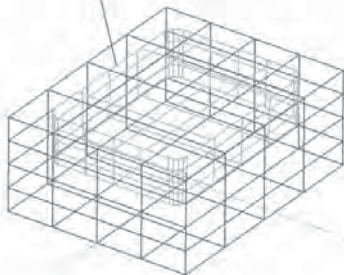
5 Click Cancel to quit the function

Step 5: Click on the **Solids** menu and choose **Subtract**.

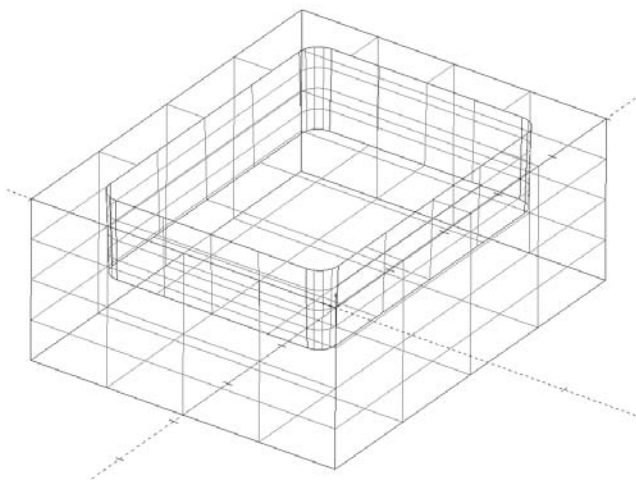


1 Click on the outer cube. This is the area that will be kept

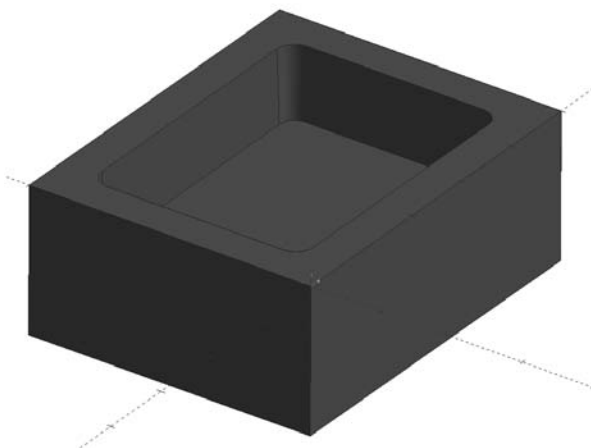
2 Click on the inner extrusion. This is the area that will be removed from the final solid



3 Right-click in the drawing and choose "OK" from the menu that pops up. The final result will look like the picture below



Since the parts that are difficult to see are now complete, switch back to a shaded view by pressing "s" on the keyboard again, and turn wire frame back off by pressing "w" once more. The part should now look like the illustration on the following page:



Step 6: The main portion of the part is complete. To begin drawing the boss in the center, click on **Other** in the main menu and choose **Rectangle**.

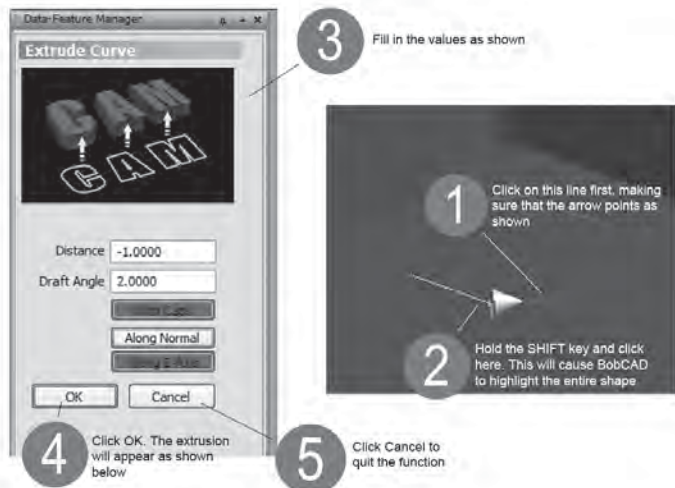
1 Fill in the values as shown

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

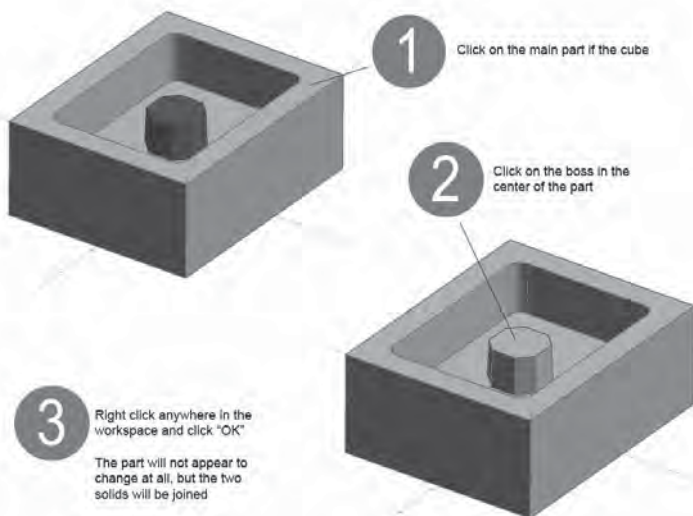
3 Click Cancel to quit the function

This image has been edited to increase the contrast for illustration purposes.

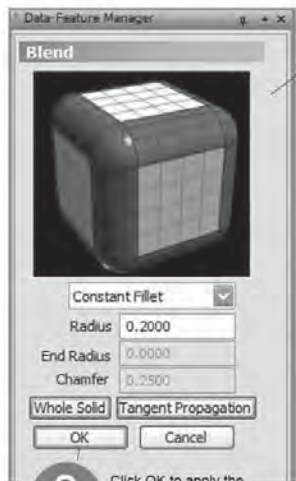
Step 7: To produce the 3D solid of the boss, click on **Surfaces** in the main menu again and choose **Extrude Curve**.



Step 8: The next step is to join the new solid to the cube base. Click on Solids and then on Add.

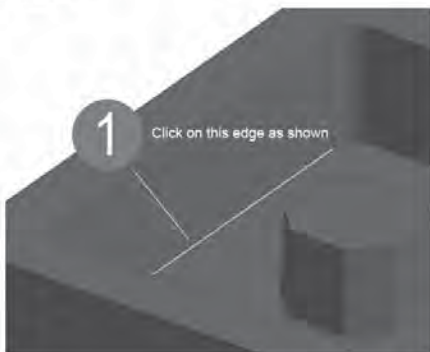


Step 9: All that remains is to add the fillets around the top of the boss, around the inner rim, and on one edge in the interior of the part. Click on **Solids** in the main menu and then on **Fillet**.



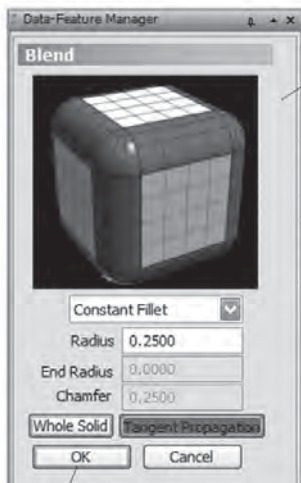
2

Fill in these values



3

Click OK to apply the fillet as shown below. Do not click on "Cancel"!



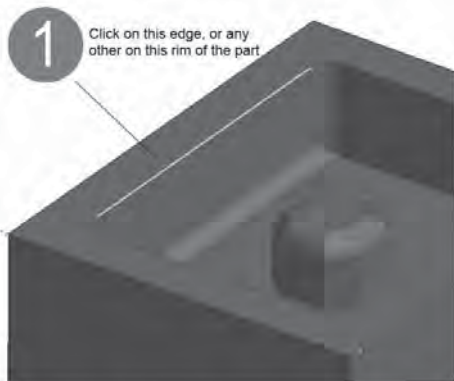
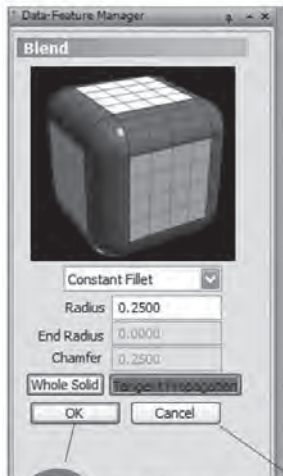
2

Change the values in the boxes to those shown here



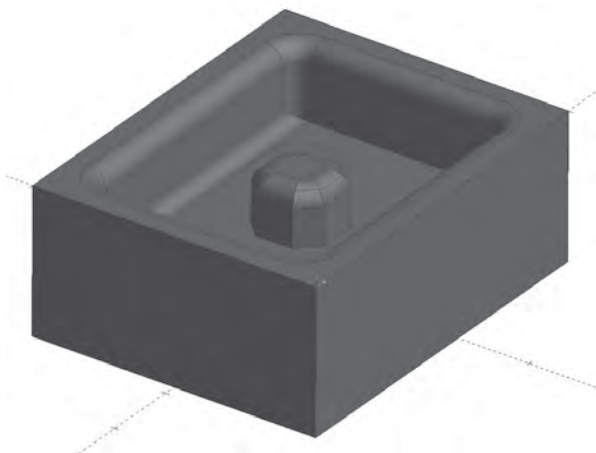
3

Click OK. This will fillet ALL of the edges of this surface, blending it with the surfaces surrounding it as shown below. Again, do not click on "Cancel"!



2 Click OK to fillet the entire edge. Leave all of the settings the same

3 Click Cancel to quit the function



Step 10: The part itself is finished. All that remains is to save it for later use.

Click on **File** in the main menu and then on **Save As**. Type in "3D Example" in the **File Name** box and click on **Save**. Keep in mind what folder the file was saved in; it will be used again in later examples.



***Note:** Make sure you remember what folder you save the file in. At the top of the box, in the **Save In** drop-down box, you can choose nearly any folder on the computer. Keeping all of your files neatly saved in folders with descriptive names avoids messes later on, as you will be able to easily find files saved long ago without having to remember which specific folder a particular file may be in.*

BobCAD-CAM V23



Chapter 5 2D and 3D Toolpath Generation

Mill

5.1 The CAM Tree

What is all this “Feature” stuff? A Feature is an individual element of a part. It is also a collection of all of the operations that fully complete one single task on a part. For example, a pocketing feature includes the geometry that defines the pocket in the drawing, the roughing tool path, the finishing tool path, the tools it uses, and all of the settings related to them.

There are a few other terms the user should be familiar with when using BobCAD-CAM. The most important of these are:

- **Cam Part** – This is the top of the **Cam Tree**. Stock is defined as well as access to posting and verification. At the **CAM Part** level, operations such as posting and verification work in a global manner.
- **Stock** – The Stock is defined using a contour and a depth. Once defined, a transparent solid representing the stock can be displayed. This definition will also be used to create the stock that simulation will use. If the Stock is not defined, the simulator will automatically calculate a stock based on the toolpath being simulated.
- **Material** – Based on the material selected, the system will automatically calculate the cutting feed and spindle speed based on the tool and operation. These values can be found with the tool in the operation. Both the spindle and cutting feeds have a percentage override that is also located with the tool. For example, if the cutting feed is 10 IPM and the **Cutting Feed %** is 150, the output to the machine will be 15 IPM.
- **Feature** – A feature consists of the geometry, single or multiple operations, machining parameters and the toolpath that was calculated using all these attributes.
- **Geometry** – The entities that are being used in the feature. Depending on the operation type these can be points, lines, arcs, splines, contours, point patterns, solids, surfaces, or STL data.
- **Associativity of a Feature** – Each feature contains all the necessary parameters to calculate tool path. This allows the user to change any of these parameters and compute the tool path again without reselecting or redefining all the parameters each time the feature is changed.
- **Operation** – An operation is one element, or task to complete a feature.

Not every feature has all of the same steps. A tapping feature may consist of these operations, for example:

1. center drill the pilot hole
2. spot drill the rest of the hole
3. chamfer the edge of the hole
4. tap the hole

Each of these steps is an operation, and together they make up the composite operation under a feature.

- **Composite Operations** – The group of operations in a feature that define the machining parameters to cut the geometry assigned to the feature.

BobCAD-CAM can handle nearly all aspects of machining operations automatically. All of its speeds and feeds are set conservatively to avoid tool breakage. See the Help in the software for details on how to change these default settings.

All CAM trees in BobCAD-CAM start with stock. To change the stock material, its size, etc., right-click on it and select **Edit**. After the stock is set up, right-click on it and choose a **Drill**, **Mill 2 Axis**, or a **Mill 3 Axis** feature. Right-click on **Geometry** and choose **Re/Select** and then highlight the points or shapes that make up the geometry of the feature.

5.2 How BobCAD-CAM Chooses Tooling

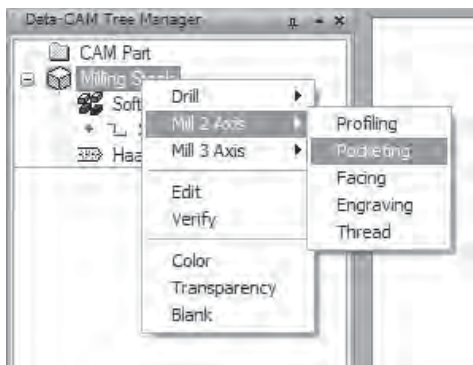
BobCAD-CAM will automatically choose the tooling based on the operation, tool diameter, and corner radius specified in the operation's edit box. For milling operations this is straightforward, simply type in the diameter and corner radius needed and the system will take care of the rest.

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 the 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.

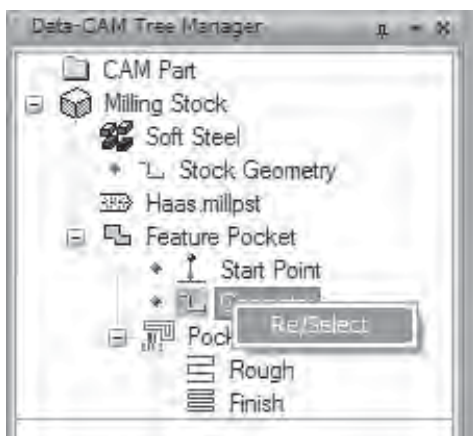
5.3 2D Pocket Example

This example will build on the “2D Example.bbcd” file created in **Chapter 4.1**, and the functions used in this example are present in all 3 levels of BobCAD-CAM. The purpose of this example is to illustrate the basic steps necessary to create any part in the software: 1. Create the feature in the tree, 2. Select the geometry to be used for the feature, 3. Edit the settings for the feature, and 4. Compute the toolpath. All parts in BobCAD-CAM will follow the same pattern of steps.

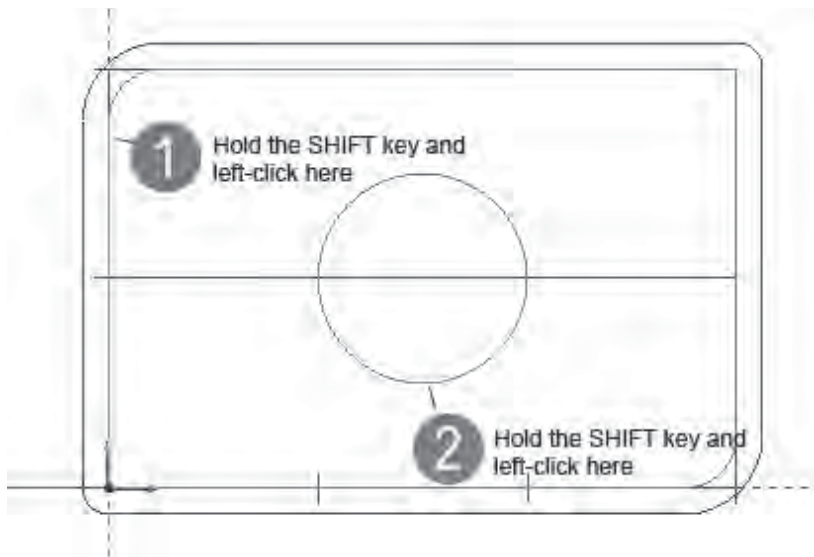
Step 1: Click on the **Cam Tree** tab in the **Data-CAM Tree Manager**. Right-click on **Milling Stock**, choose **Mill 2 Axis**, and click on **Pocketing**.



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

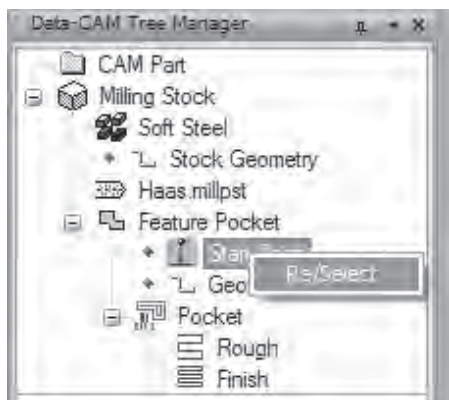


Click on the drawing as shown here:

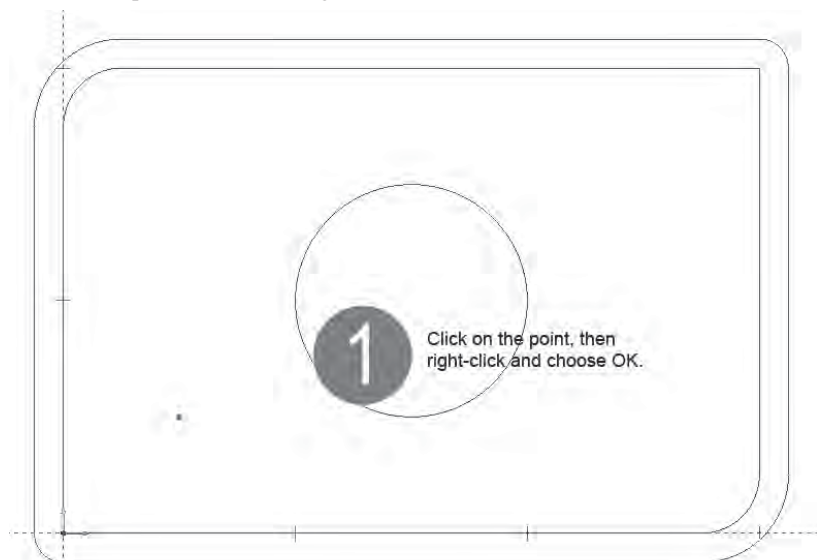


When the two shapes used here have been selected, right-click in the **Workspace** and choose **OK**. Once **OK** has been clicked, BobCAD exits the selection mode and changes the tree to show that the geometry for this feature has been set.

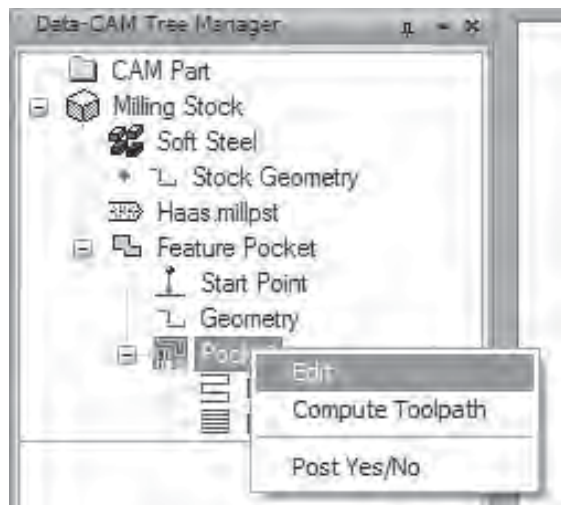
If the starting plunge position of the pocket does not matter, then the start point does not need to be set as BobCAD-CAM will automatically choose a plunge position. If it does matter, however, choose it now by right-clicking on **Start Point** and choosing **Re/Select**.



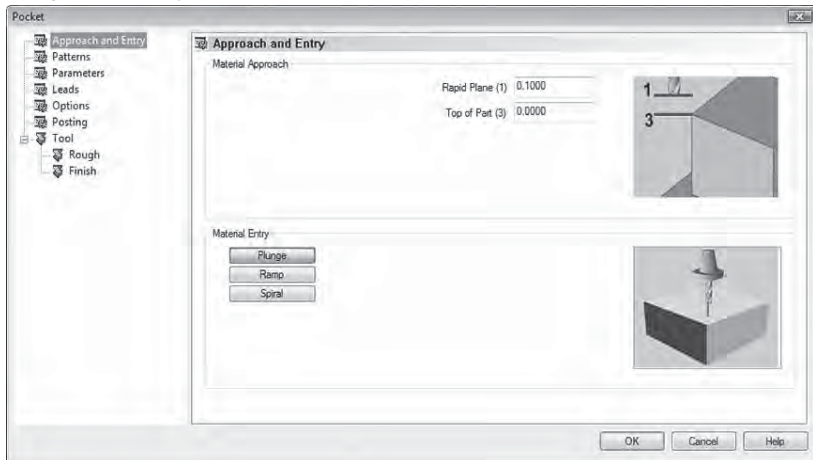
Click on the point on the drawing as shown below:



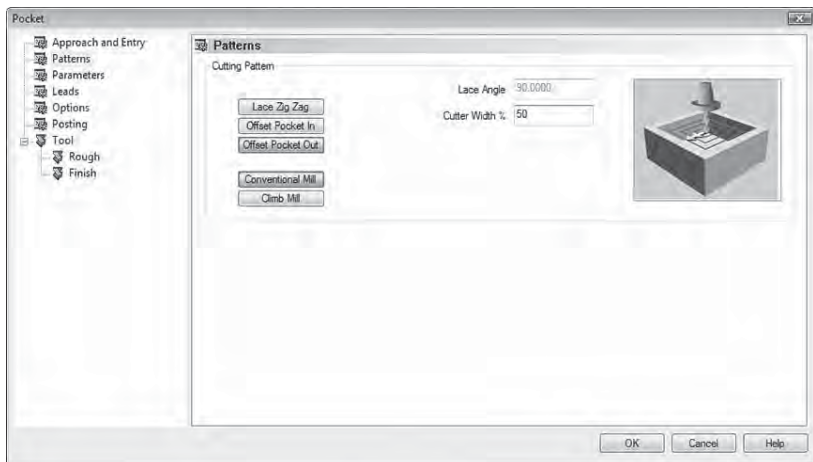
Step 3: Click on **Pocket** under the **Feature Pocket** item in the tree and choose **Edit**.



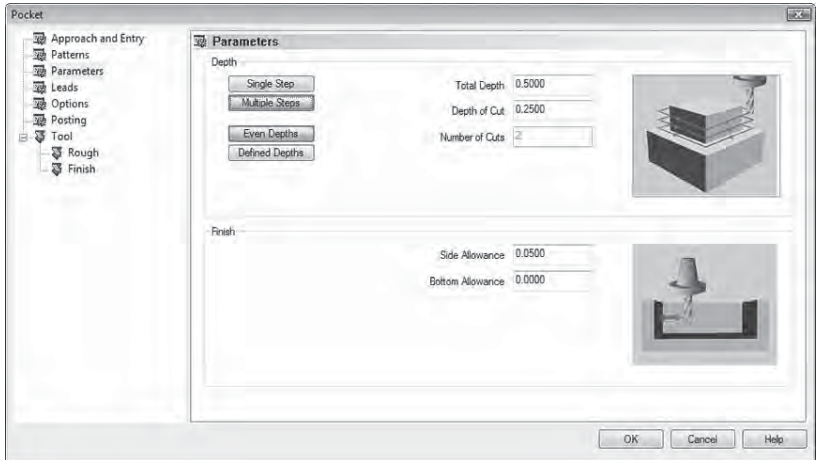
Click on **Approach and Entry** at the left of the box and fill out the rest of the information to appear like below. This example does not call for a ramped entry, though **Ramp** may be chosen if desired.



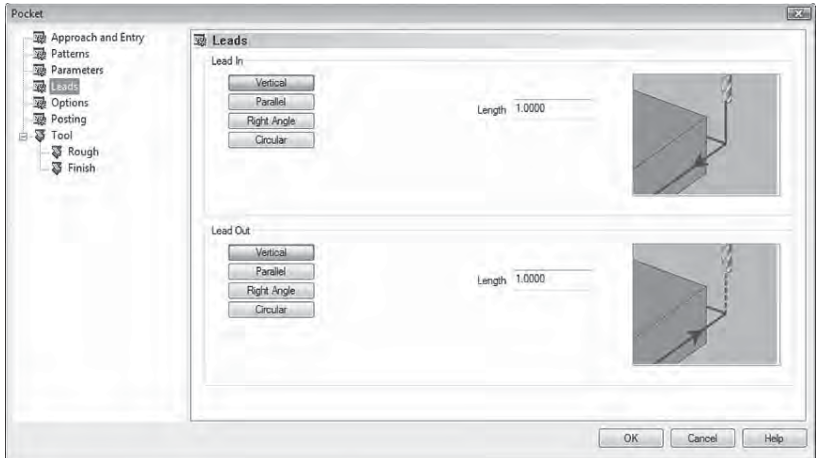
Click on **Patterns** and do the same thing. **Lace Zig Zag** pockets cut from side to side, removing material quickly but leaving scallops on the edges. **Offset Pocket In** and **Offset Pocket Out** do not leave these scallops, but may generate more code.



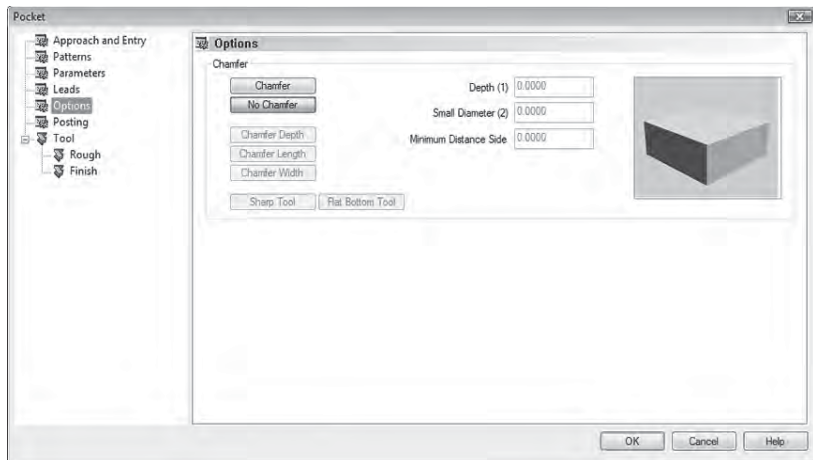
Click on **Parameters** on the left and fill out the dialog. Since in this example the pocket is going to be .5 deep and later on a .5 tool will be specified, choose **Equal Steps** and set the **Depth of Cut** to .25. In this example, there will be a finish tool used so the side allowance is set to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



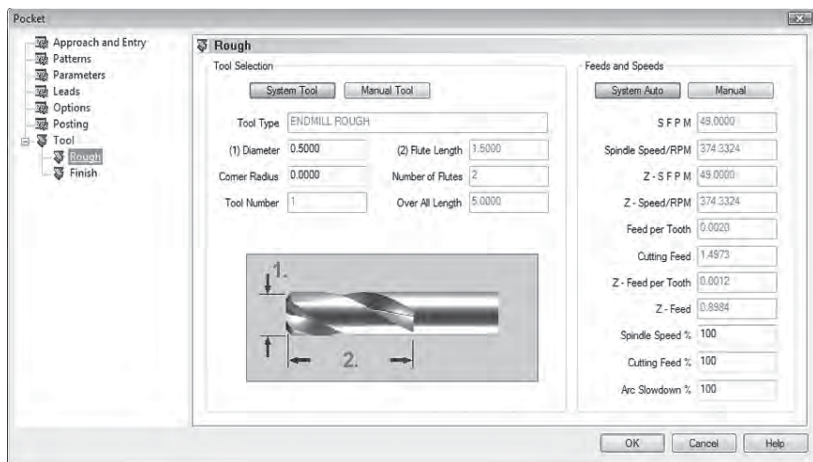
Click on **Leads** to the left. In the pocketing features, these parameters control the approach into the finish pass.



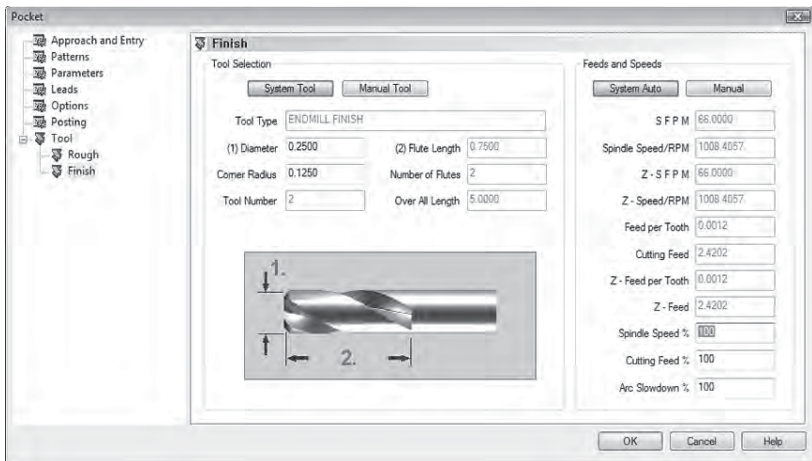
Click on **Options**. It will not be required to break the top edges of the pocket for this example, so leave the settings at the default **No Chamfer**.



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

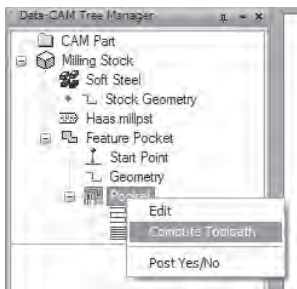


Click on **Finish**. This finish will use a .25 ball mill.

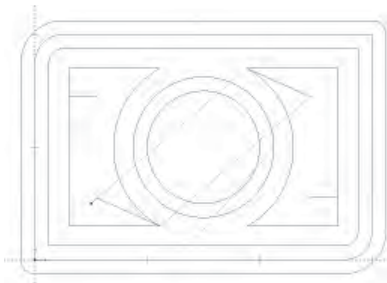


Click **OK** at the bottom right. All of the settings for the pocket operation will be complete.

Step 4: Right-click on **Pocket** one last time and choose **Compute Toolpath**.



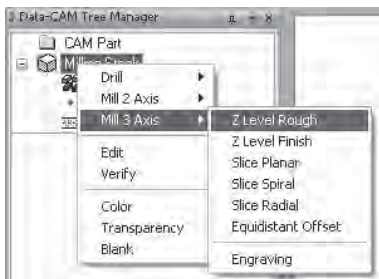
When that has completed, the finished pocket toolpath should look something like this:



5.4 Z-Level Rough Example

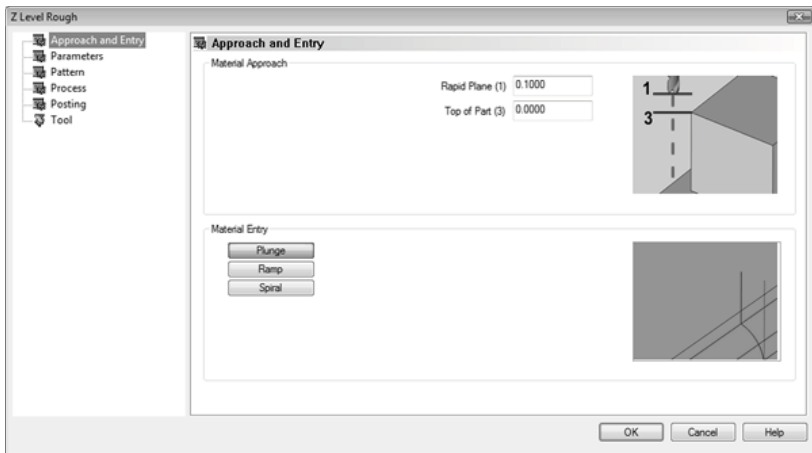
This example uses the “3D Example.bbcd” produced in Chapter 4. The same steps can be used for any 3D file.

Step 1: Right-click on **Milling Stock**, then choose **Z-Level Rough** from the **Mill 3 Axis** menu.

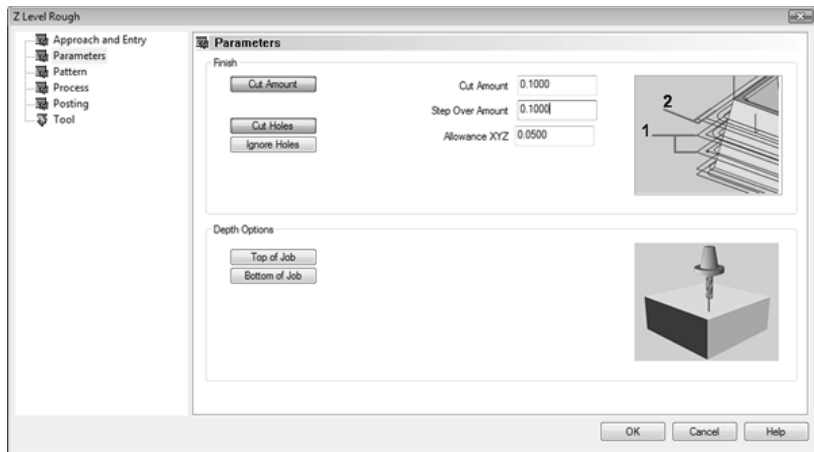


Step 2: Right-click on the **Z-Level Rough** item under **Feature Z-Level Rough** and click on Edit.

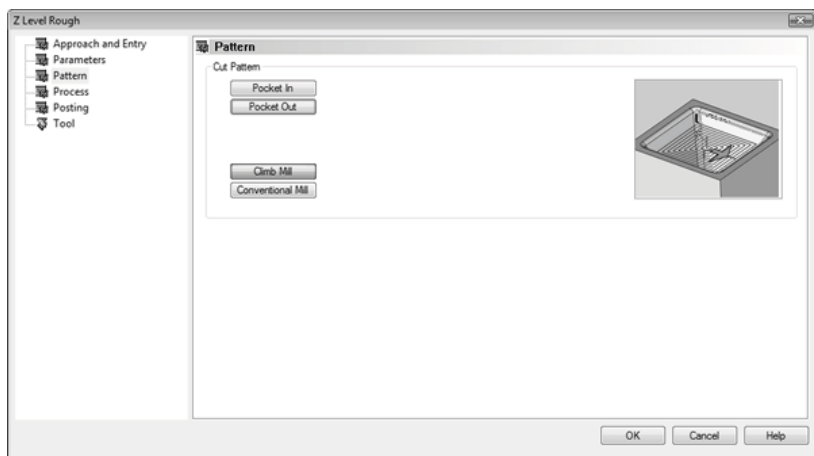
Click on **Approach and Entry** on the left side. The box should already be filled out as shown below, but double-check it to make sure.



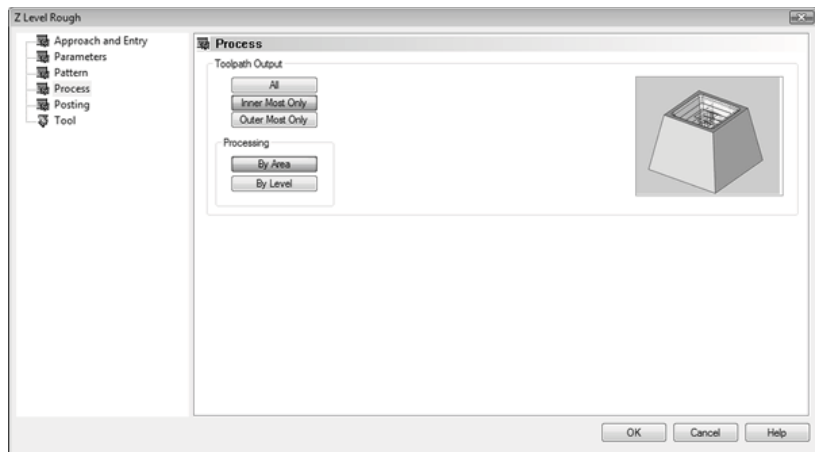
Click on **Parameters**. Change the settings to match what's shown below.



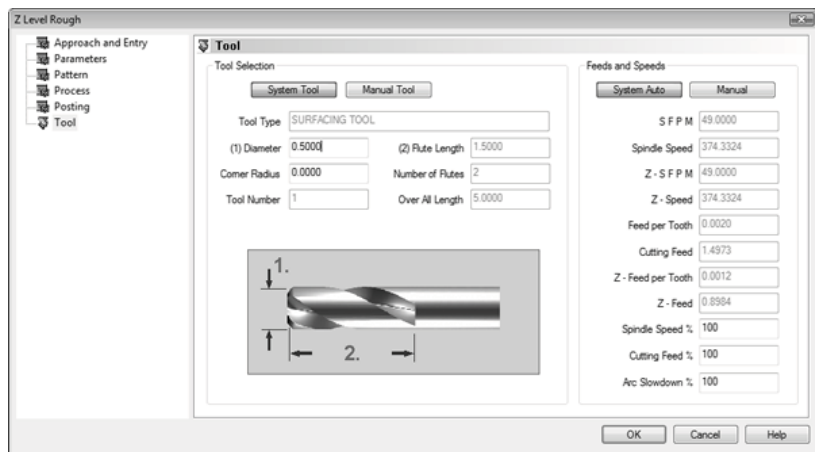
The next item, **Pattern**, needs the Offset Direction changed. Select the Pocket Out option.



The next item on the left, **Process**, has quite a few options to be set. Be sure that **Inner Most Only** is set as that will ensure that the toolpath only cuts the inside pocket.

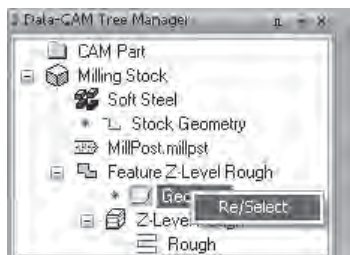


Next, click on **Tool** on the left. Set the tool to a .5 diameter with no corner radius.

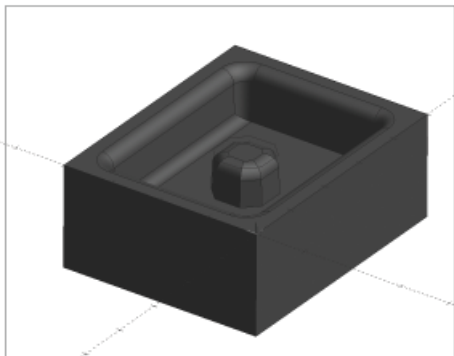


Click **OK** on the lower right to finish editing the operation's parameters.

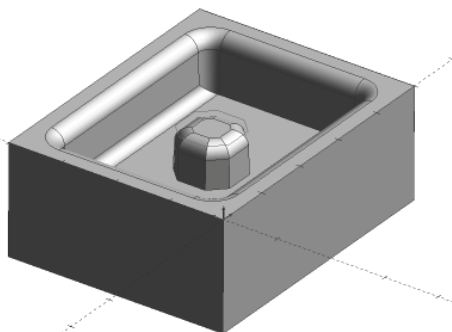
Step 3: Right-click on **Geometry** and choose **Re/Select**.



Click a little away from the geometry and drag the mouse so that a box completely surrounds the part.

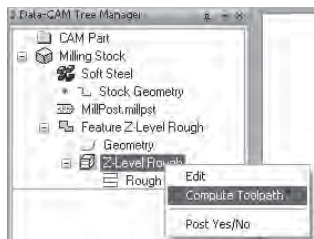


The selected geometry should change color as shown below.

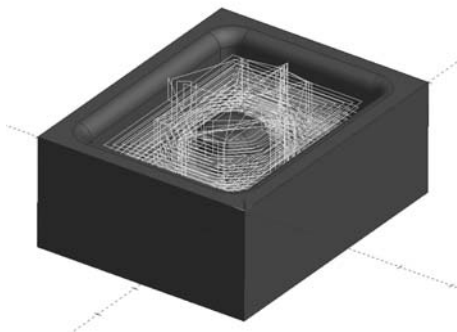


Right-click on the **Workspace** and choose **OK**. The part will change back to its original color.

Step 4: Right-click on **Z-Level Rough** and choose **Compute Toolpath**.



When BobCAD-CAM is finished calculating all of the passes, the toolpath will show as below.



Step 5: Save the file by clicking on the Save icon  in the toolbar.

5.5 Drilling and Tapping Example

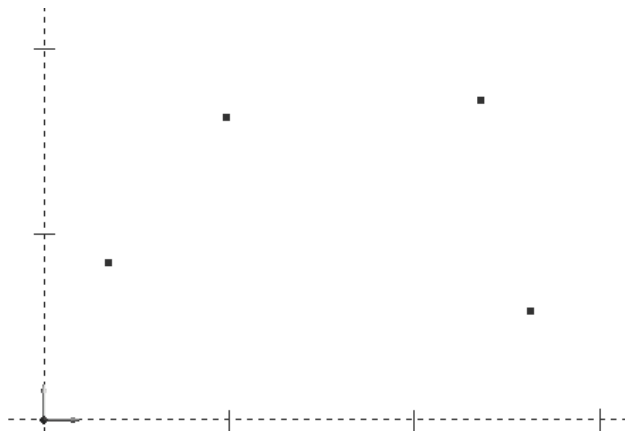
The functions used in this example are available in all levels of BobCAD-CAM.

Step 1: Begin with an empty drawing by using any one of these 3 methods:
Click on **File** in the main menu and then **New** OR

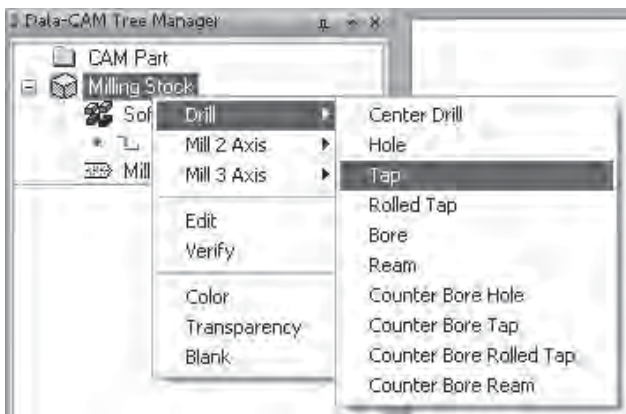
Click on the **New File** icon in the toolbar  OR
Type CTRL+N to start a new drawing.

Step 2: Click on **Points** in the main menu, then on **Sketch**.

Click a few points on the screen. For this example, it's not critical where they are.

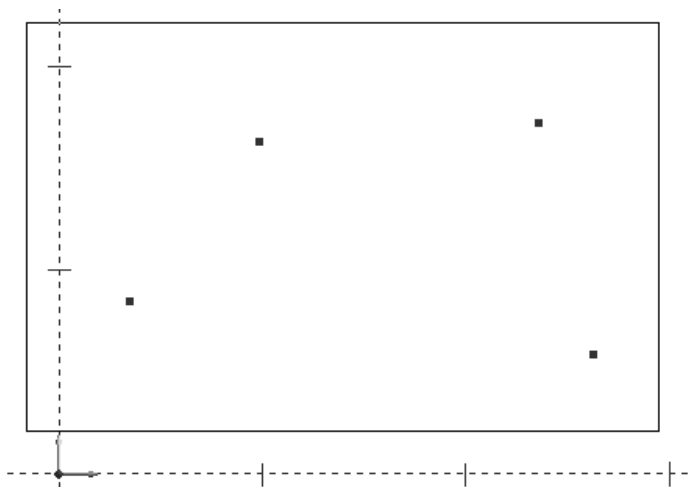


Step 3: Click on the **CAM Tree** tab at the bottom of the **Data-CAM Tree Manager**. Right-click on **Milling Stock** and choose **Tap** from the **Drill** sub menu.



Step 4: Right-click on **Geometry** under the new **Feature Drill Tap** item. Click on **Re/Select**.

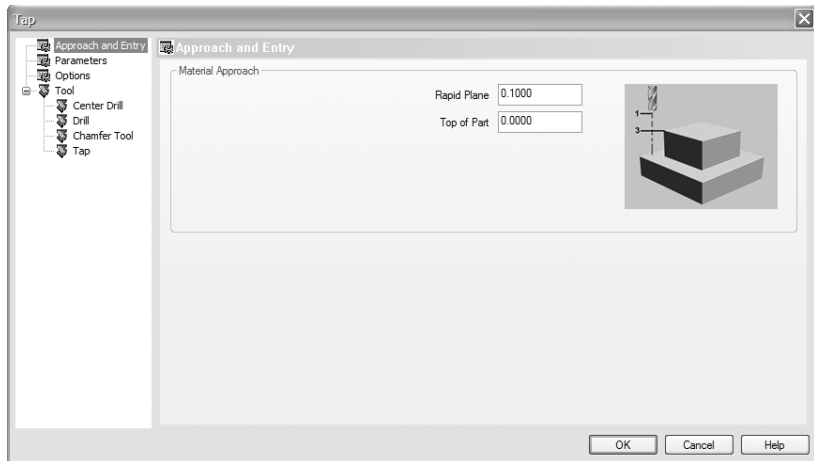
Click slightly away from these points and drag the mouse so that a box draws all the way around the points.



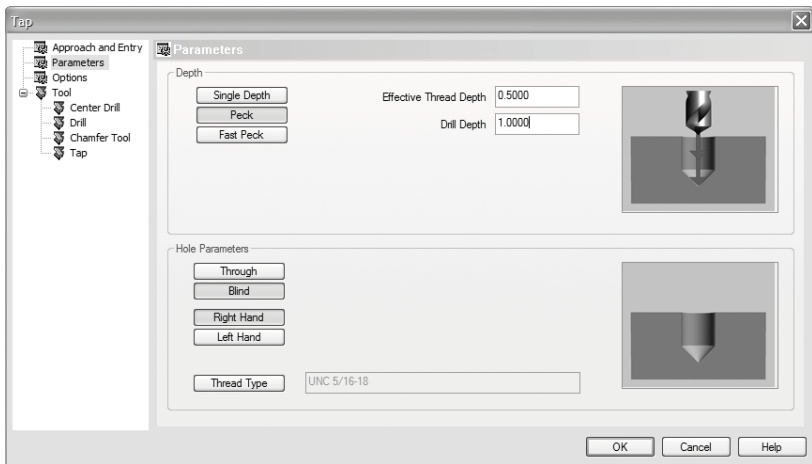
Right-click in the **Workspace** and click **OK**.

Step 5: Right-click on **Feature Drill Tap** and choose **Edit**.

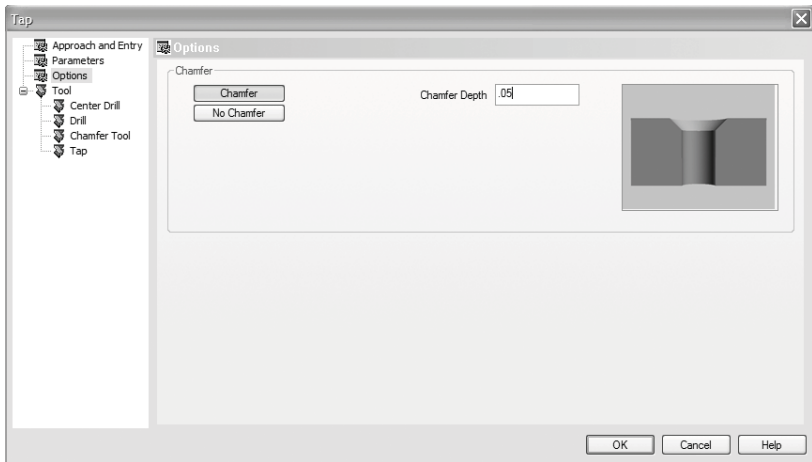
Click on **Approach and Entry** at the left. The default settings should be the same as shown below, but double-check them to be sure.



Click on **Parameters** to the left. These settings are the bulk of the data needed for the cycle. Don't forget to click on **Thread Type** at the bottom and choose a thread.



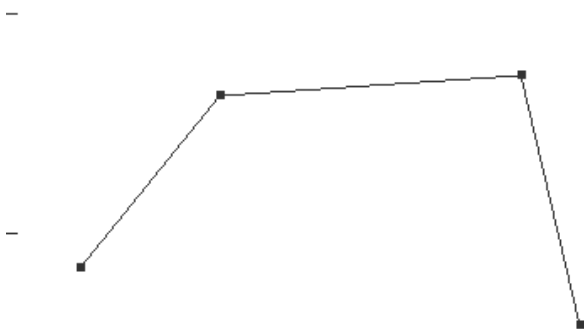
Next, click on **Options** to the left. Match the **Chamfer Depth** setting to .05 as shown below.



There is no need to change anything in the tools. BobCAD-CAM automatically knows which tools to use based on the thread chosen in the **Parameters** page. Click **OK** in the lower right corner.

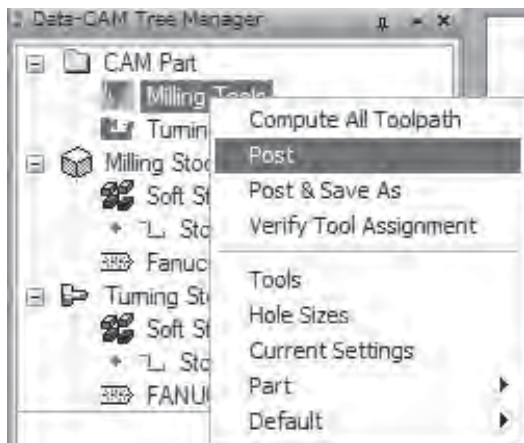
Step 6: Right-click on **Feature Drill Tap** one last time and choose **Compute**.

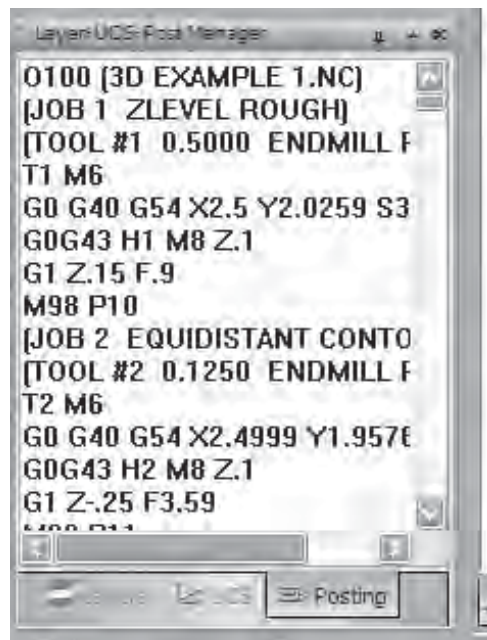
The results in the workspace should resemble the illustration below.



5.6 Posting a Program

After the toolpath for a part has been computed, to generate the code for the program, click on the + symbol next to **CAM Part**, right-click on **Milling Tools** and choose **Post**. The code will appear in the third tab of the **Layer-UCS-Post Manager** window.





BobCAD-CAM V23

Chapter 6 Verification

Mill

Chapter 6. Verification

BobCAD-CAM's verification engine uses a custom Predator™ Virtual CNC® OCX control for highly accurate part cutting simulation.

The new verification system simulates machine cutting based on the computed tool path, the chosen tools, and on the rectangular size of the milling stock. In this way, it is possible to simulate part cutting for any machine that BobCAD-CAM can generate code for, and it can do it before the code is even produced.

6.1 Starting the simulation

Before simulating any cutting features, the tool paths must first be calculated using either **Compute** under each feature or by selecting **Compute All Toolpath** under the right-click menu for **CAM Part**.

To simulate a machine running a program, right-click on **Milling Stock** under **CAM Part** in the **Cam Tree** tab of the **Data-CAM Tree Manager**. Choose **Verify** from the list. The simulation window will immediately appear with a block the size of the part in it.

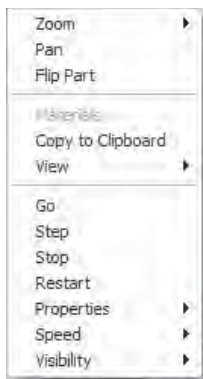
6.2 Controlling the simulation

There are quite a few ways of controlling the simulation. The 6 main controls at the bottom of the window are:

- **Start** – Click this to start the simulation.
- **Step** – Click this to step through the simulation one machine move at a time.
- **Stop** – Click this to stop the simulation. This will not start it over from the beginning; rather, it operates as a pause.
- **Restart** – Click this to start the simulation over from the beginning.
- **Speed Slider** – This controls the speed of the simulation. To the left is slower, to the right is faster.
- **End** – Quit the simulation.

The simulation can also be controlled with the mouse:

- Left-Click + Drag – Rotate the view.
- Wheel Up – Zoom Out
- Wheel Down – Zoom In
- Right-Click – Show the verify pop-up menu



This pop-up menu carries a host of options:

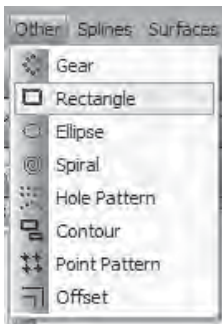
- **Zoom** – This sub menu contains all of the functions for zooming in and out.
- **Pan** – Click this and then left-click + drag in the verify window to pan.
- **Materials** – displays properties for the simulated material, such as color, etc.
- **Copy to Clipboard** – This function copies a screenshot of the current view in the verify window to the clipboard. It can then be pasted into any image editing program for saving, etc.
- **View** – All of the pre-defined viewing options are listed in this sub menu, i.e., Top, Bottom, Isometric views, etc.
- **Go** – Runs the simulation.
- **Step** – Single-steps each machine move.
- **Stop** – Halts the simulation.
- **Restart** – Resets the simulation back to the beginning.
- **Speed** – This sub menu contains all of the options related to the speed of the simulation.
- **Visibility** – Contains settings for the translucency of the image.

6.3 Example

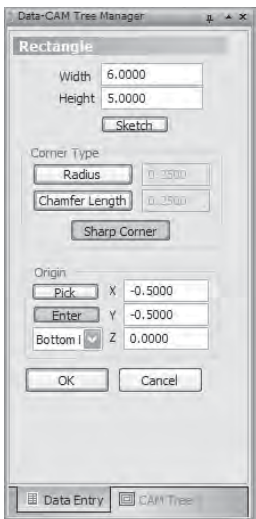
Open the file “3D Example.bbcd” if it was created in the examples in previous chapters.

Step 1: The first thing that will need to be done is to define a stock boundary around the part.

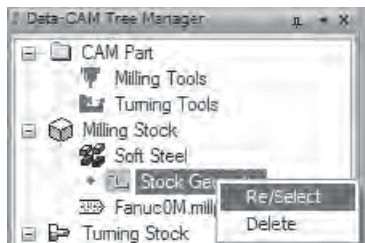
Click on **Other** in the main menu and choose **Rectangle**.



Set the **Width** and **Height** to 6 and 5 as shown below. In the **Origin** section, click on **Enter** and set the X and Y both to -.5 and the Z to 0. In the lower-left of that section, set the drop-down box to **Bottom Left** and click **OK**.



Right-click **Stock Geometry** and choose **Re/Select**.



Hold down the shift key and click anywhere on the rectangle.

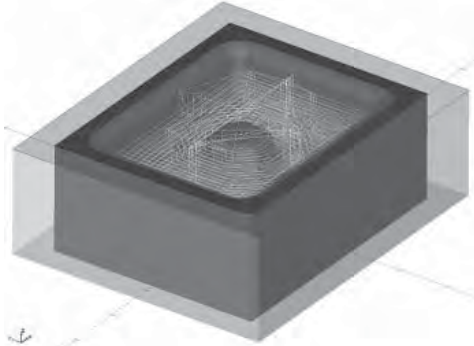


Right-click in the **Workspace** and click **OK**. The stock will appear as a translucent cube.

Right-click on **Milling Stock** and click on **Edit**. In the dialog box that will appear, set **Thickness** to 2 and click **OK**.



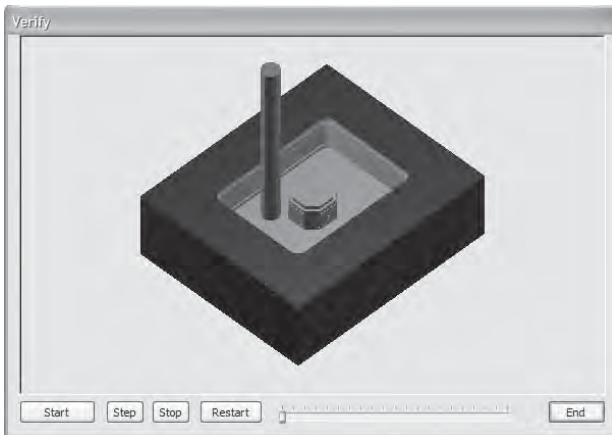
The stock should appear as shown below.



Step 2: Now that the stock is defined, right-click on **Milling Stock** again and choose **Verify**.

The **Verify** window will appear. Click on **Start** to begin the simulation.

The simulation will run. If desired, change the views around while the simulation is running. Note that the tools displayed will match the tools used in the program.



When the simulation is complete, it should look like the illustration above.

BobCAD-CAM V23

Chapter 7 Getting Code to the Machine

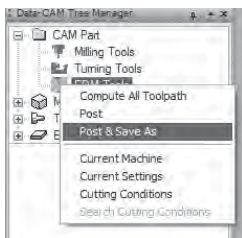
Mill

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.

7.1 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.

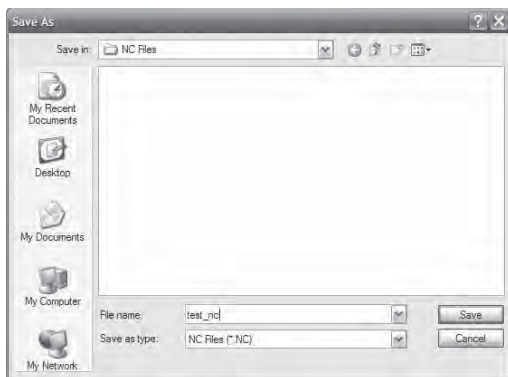


- **Save As from the Layer-UCS-Post Manager** – After the program has been posted, right-click in the **Layer-UCS-Post Manager** where the code is displayed and choose **Save As**. The **Save As** dialog will appear.



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

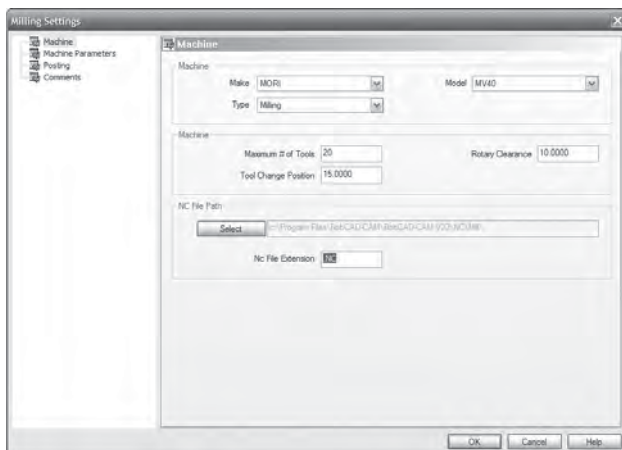


then burn the CD from there instead of attempting to save the file directly to the device.

7.2 A Word on File Formats and 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.

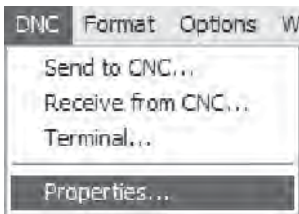
NOTE: ASCII text is the actual file format supported by probably 99% of all CNC machines ever manufactured, regardless of what file extension is required. If there is any doubt whether the machine accepts this format, post a program and try it. It is only in rare exceptions that this will not be the 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.

7.3 Establishing Communication

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.

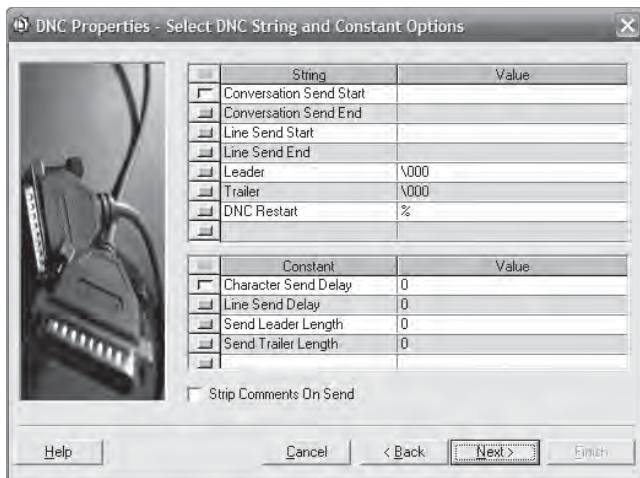


The system will display a series of 5 dialog boxes:

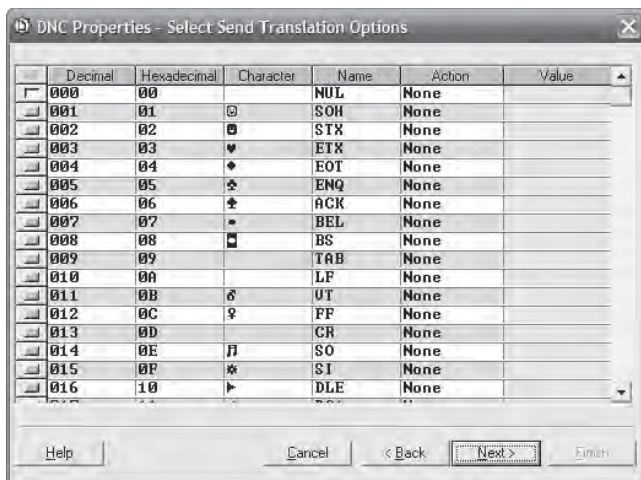
1. **Select RS-232 Properties** – Set the COM port options from here. Most controllers will require that these options be set to match the settings on the machine, but a few rare controls will actually require mismatched settings. See the controller's documentation to ensure the settings are correct.



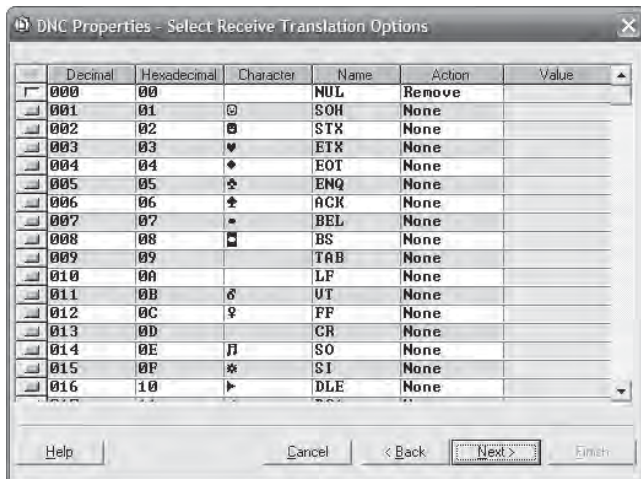
2. **DNC String and Constant Options** – Use this dialog box to set up any special leader or trailer characters the control may require.



3. 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.



4. 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.



5. **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.

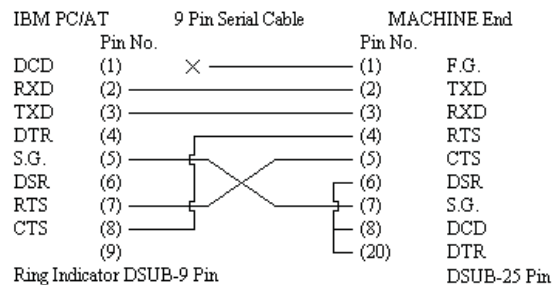
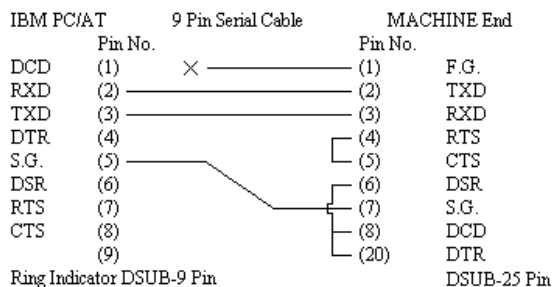
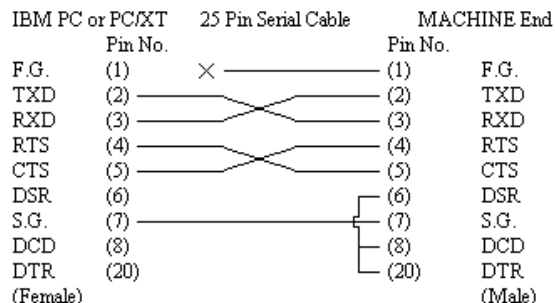
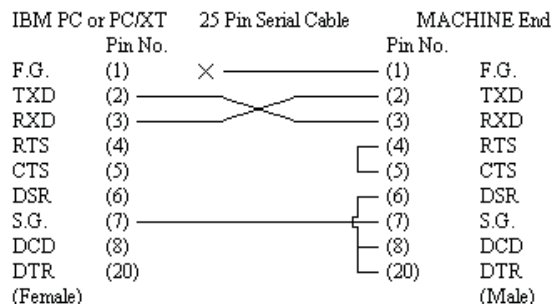


Click **Finish** when the settings are correct. The system will remember them for the next time a program is transferred.

Communication Hardware

To send a file over an RS-232 (or RS-422) link, a properly wired cable is absolutely required. See the next page for the most common wiring diagrams.

These are the common pinouts for nearly all CNC controls.



Cables typically need to 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.

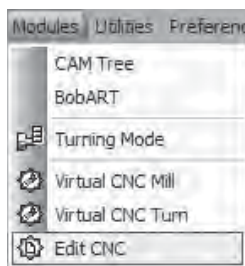
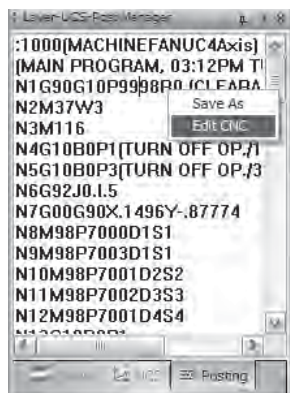
If a good cable is difficult to find or to build, or if the machine is more than 50' from the computer, it is now possible to purchase Predator Grizzly™ cables directly from BobCAD-CAM, Inc. Grizzly™ cables are viable to lengths of up to 400' in a variety of environments.

7.4 Transferring Files via RS-232 (Serial) Link

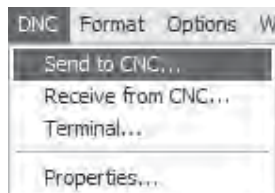
The Predator CNC Editor DNC can transfer a program to and from the CNC controller.

Send a Program to the CNC

Step 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.



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

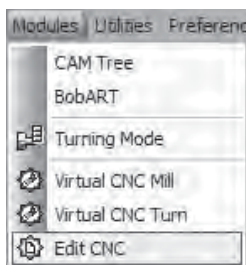


Step 3: The CNC Editor will automatically begin the transfer. The progress bar near the bottom of the dialog will count off the percentage of the file transferred. The progress bar will display 100% when it is complete.

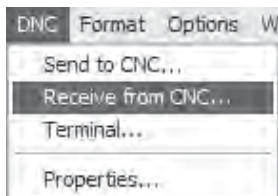


Receive a Program from the CNC

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



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



Step 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.



