Lathe

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.*

Table of Contents

BobCAD-CAM

V23

Lathe

Chapter 1
Introduction

Chapter 1. Introduction

Thank you for your purchase of BobCAD-CAM Version 23. We at BobCAD-CAM, Inc. hope that it becomes an indispensible 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 Accessing BobCAD-CAM

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

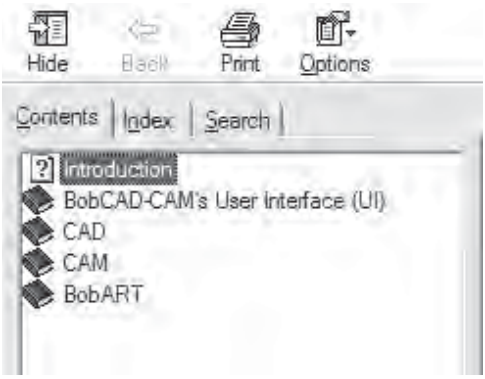
- Directly from the desktop. After the installation is complete an icon will be loaded onto your desktop. Double click on this icon to launch BobCAD-CAM Version 23.
- Directly from the Start menu. Click the Start button from the taskbar, then choose Programs and then BobCAD-CAM Version 23 to launch the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

1.5 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 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

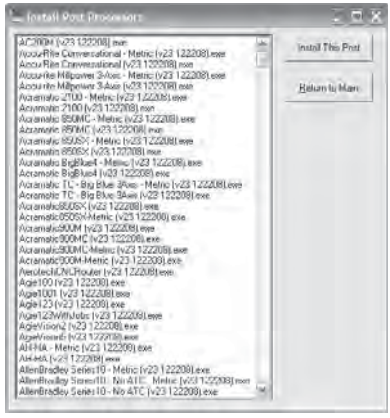
1. BobCAD-CAM, Inc. will post all of the currently available post processors as they become available on its **http://www.bobcad.com/** website. To access these, click on the Support link on the main page, then choose the appropriate Post Processor link for your application. From there, the available 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.
2. 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

Lathe

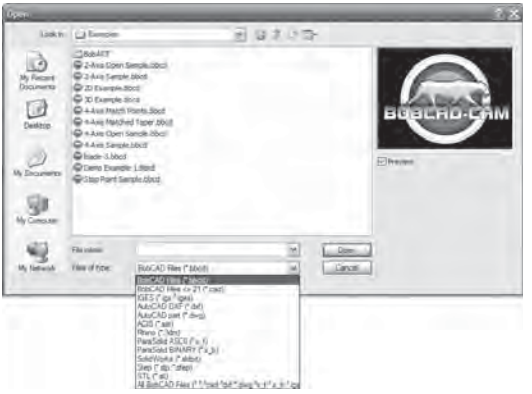
Chapter 2
Files

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. Whichever 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** – STandard 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.

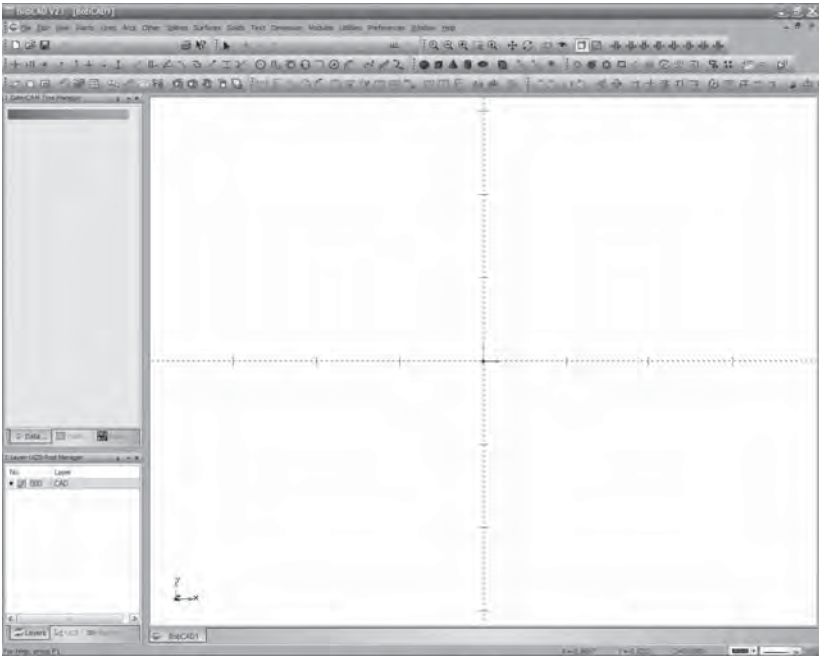
BobCAD-CAM V23

Lathe

Chapter 3
Using the Workspace

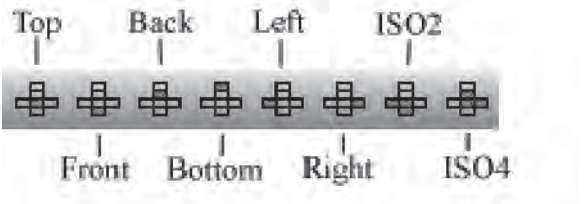
Chapter 3. Using the Workspace

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:

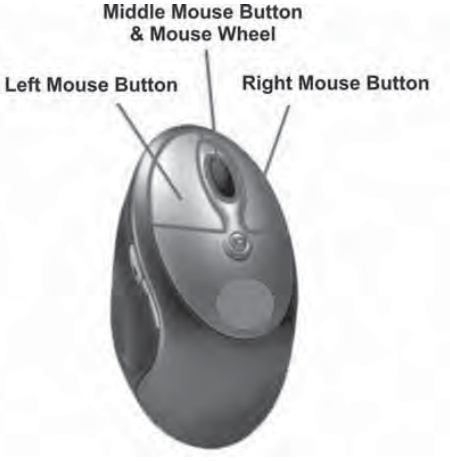


Individual settings for the view, including quality, colors, etc., are all listed under the **Preferences** menu in **Settings Part** and **Settings Default**.

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.

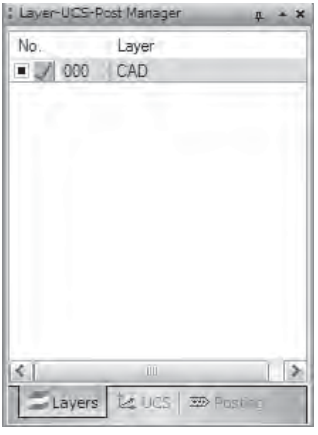
Settings Default has the same box, but the settings there apply to new drawings only. There is also a **Directories** category not present in **Settings Part** on the left where the user can set the default folders to retrieve data from and save to.

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

Lathe

Chapter 4
Drawing for Lathe

Chapter 4. Drawing for Lathe

BobCAD-CAM can draw both with wire frames and with solids and surfaces. Generally speaking, wire frames consist of lines, arc, points, and a few special versions of them, i.e., fillets, gears, and ellipses. Solids have 5 different primitive types, cube, sphere, cone, cylinder, and torus. Nearly any combination is possible, and many are sometimes combined to produce unusually shaped surfaces. See the **Help** system in the software for a full explanation of each of these.

The examples in this portion of the manual are 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.

4.1 Wire Frame Turning Profile Example

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 3 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.

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.

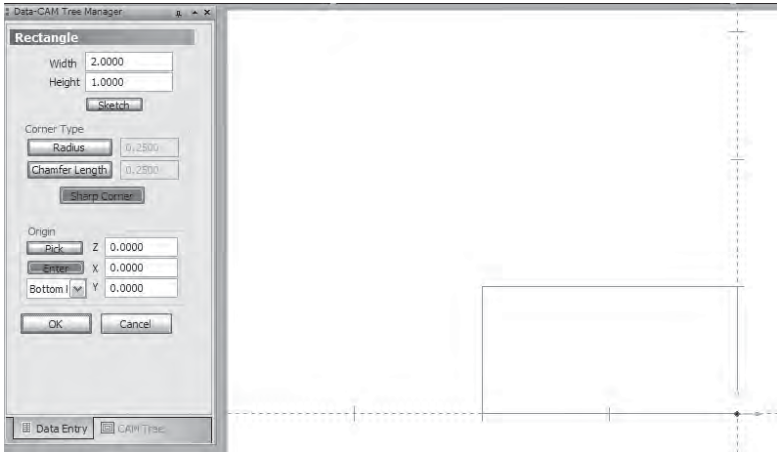
If the **Lathe** icon  on the toolbar is not depressed, click it now to turn **Lathe** mode on.

Step 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:

Width: 2.
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.*

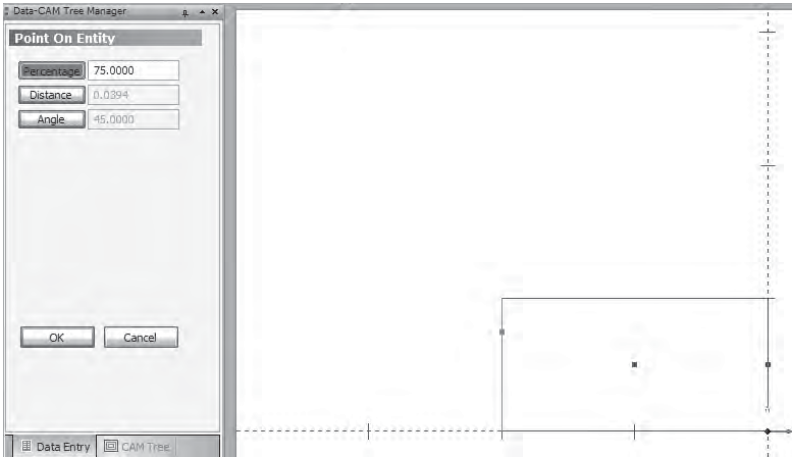
Leave the **Sharp Corner** button set, and in the **Origin** section, set the **Enter** button and choose **Bottom Right** from the drop-down box. Enter 0 for each of the **Z**, **X**, and **Y** axes. Click **OK** and a rectangle will be drawn.



Click on **Point** in the main menu. Choose **Coordinates**. Enter -1 for the **Z** axis, 0.5 for the **X** axis, and 0 in **Y**. Click **OK** and a point will appear in the middle of the rectangle.

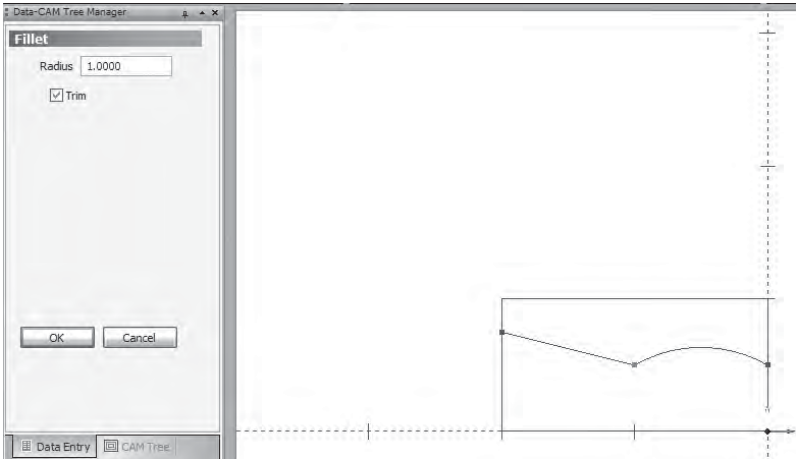
Click on **Point** again in the menu, and choose **On Entity**. Set the **Percentage** button and enter in 50. This will put a point in the middle of any wire frame entity clicked on. Click on the right-hand line of the rectangle.

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**.



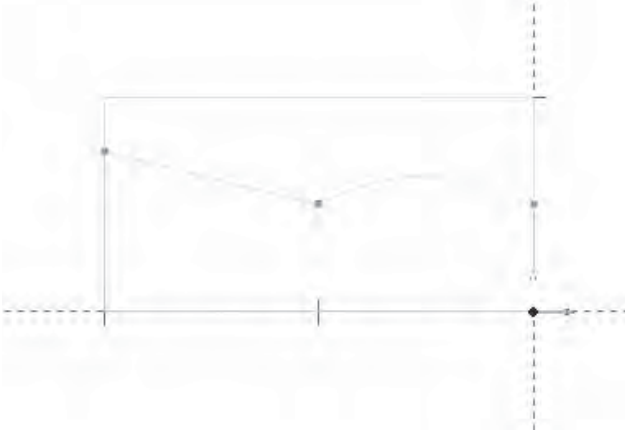
Step 3: Click on **Line** in the main menu and then on **Join**. Click on the left two points and a line will appear between them.

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.

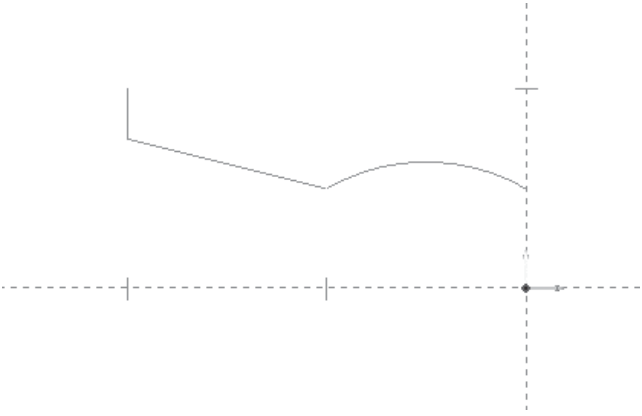


Step 4: Click on **Utilities** in the main menu and then choose **Break**, and **Many**. Drag a box over the whole drawing, then right click and choose **OK**. Every line and arc in the drawing will be broken at every point where another entity touches it.

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.



That's it; the turning profile is complete.



4.2 Wire Frame Facing Profile Example

The only real difference between a turning profile and a facing profile is that a turning profile generally opens upward (or downward, for ID profiles), and a facing profile opens to the right.

This exercise will expand a little on the turning profile example by using a few different drawing functions. This is to help the user get a better feel for the way BobCAD-CAM approaches drawing.

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

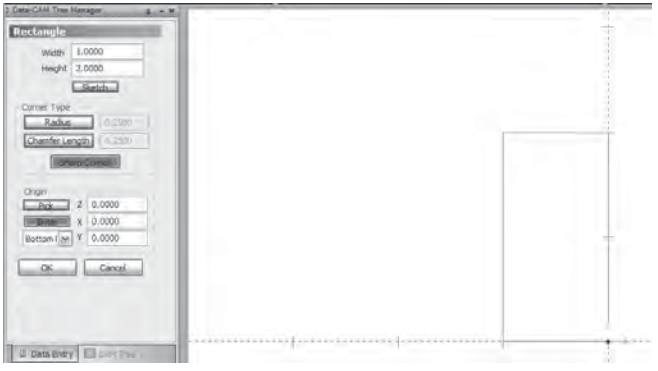
Click on the **New File** icon in the toolbar OR
Type CTRL+N to start a new drawing.

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

Step 2: Click on **Other** and then **Rectangle** from the main menu. Enter in these values in the **Data Entry** tab:

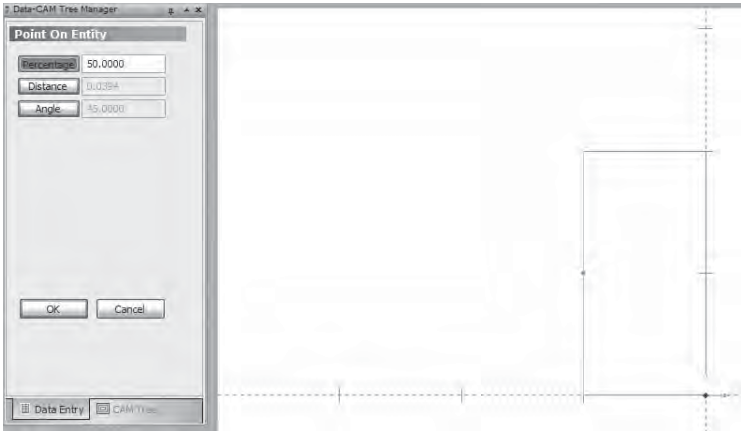
Width: 1.
Height: 2.

In the **Origin** section, make sure the **Enter** button is activated and select **Bottom Right** from the drop-down box. Enter in 0 for all 3 axes then click **OK**. The starting rectangle will appear in the **Workspace**.

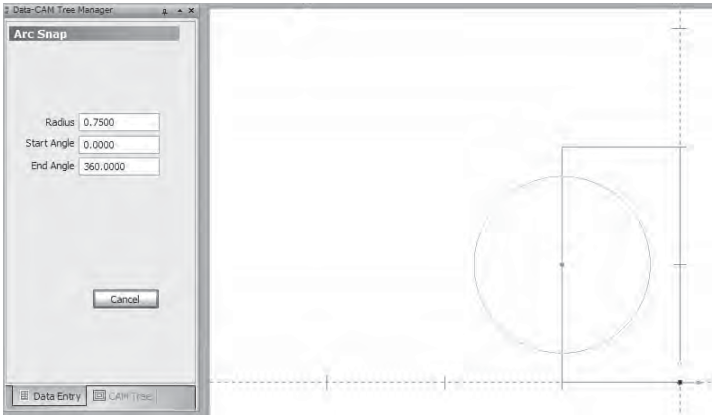


Click on **Point** in the main menu and choose **Coordinates**. Enter 0 for all 3 axes then click **OK**. The point may be difficult to see, but it's there.

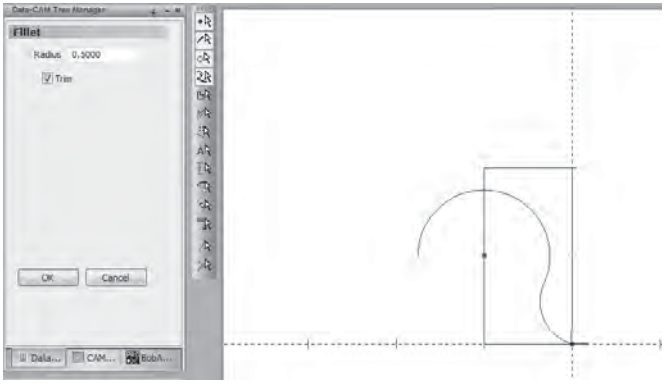
Click on **Point** again from the menu and choose **On Entity**. Make sure the **Percentage** button is activated and enter 50 in the box next to it. Click **OK** and then click on the left side line of the rectangle. Click **Cancel** on the function to quit from it.



Step 3: Click on **Arc** in the main menu, then **Snap**. Enter .75 for the radius of the arc, and enter 180 and 540 in the **Start Angle** and **End Angle**, respectively. Click on the point in the middle of the left line and a full circle will appear centered on it. Click **Cancel** in the **Data Entry** tab.

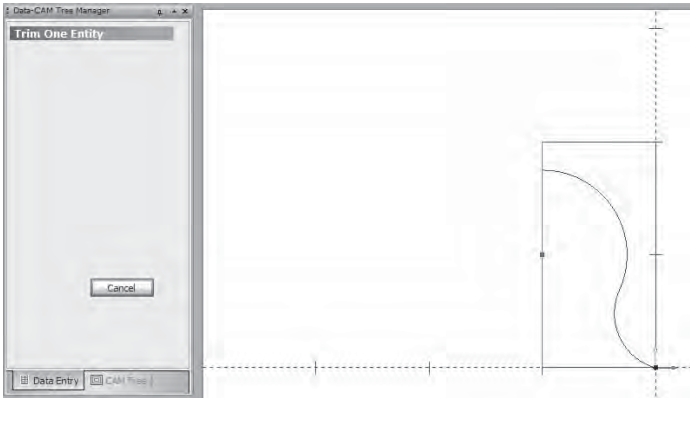


Click on **Arc** again in the main menu and pick **Fillet**. Enter .5 for the radius. Click near the bottom right of the large circle just drawn then again at the 0,0 point. Make sure that the two lines of the rectangle do not highlight before you click on the point. If they do, the fillet will be drawn to them and not to the point.



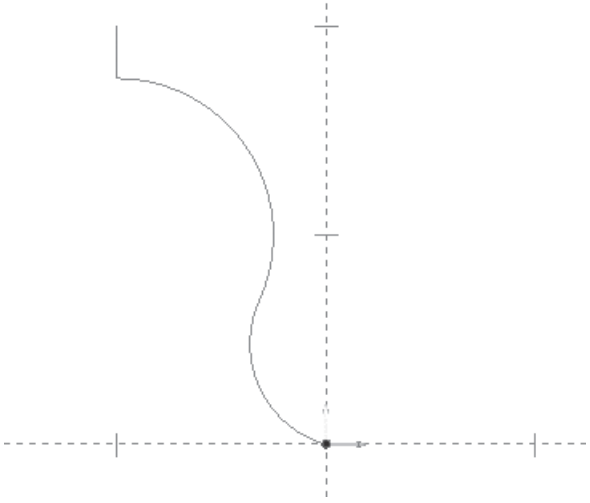
When the fillet is drawn, most of the large circle will disappear. The fillet will shorten a circle to the 0 degree point when it is set to automatically trim, so the larger circle will need to be extended back to the left line of the rectangle.

Click on **Utilities** in the main menu and choose **Trim Extend** and **One Entity.** Click on the top of the "S" shape, inside of the rectangle, that is on the screen first, then near the top of the left line of the rectangle. The arc will be trimmed to the line.



Step 4: The profile is drawn; all that remains is the cleanup. Click on Utilities in the main menu, and then on Break and Many. Drag a box around the drawing and release the mouse button. Right-click anywhere in the workspace and choose OK, then click on Cancel in the Data Entry tab.

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 facing profile is finished!

4.3 Solid Profile Extraction Example

Often parts from engineers and design houses will be sent as solid models from which the lathe programmer must then extract the profiles from in order to machine them. This exercise illustrates how to get those profiles in a usable form.

Steps 1 through 3 draw a solid shape. If a part has been imported from another source these steps can be skipped entirely.

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.

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

Step 2: Click on **Solids** in the main menu and then on **Cylinder**. In the **Data Entry** tab of the **Data-CAM Tree Manager**, enter in these values:

Radius: 2.
Height: 3.

In the **Origin** section, click on **Enter** and leave the **Z**, **X**, and **Y** coordinate all on 0. Click **OK**, then on **Cancel** to quit the function.

The cylinder will be in the wrong orientation, so click on **Utilities** and **Rotate**. Click once on the cylinder in the **Workspace**, and then enter these values in the **Angle Around Axis** section of the **Data-CAM Tree Manager**:

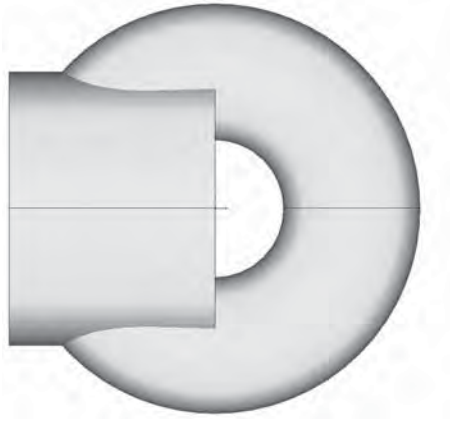
Z: 0.
X: -90.
Y: 0.

Highlight **Enter** in the **Origin** section, and leave the **Z**, **X**, and **Y** coordinates all at 0. Click **OK** once to accept the settings, **OK** again to rotate the part.

Step 3: Click on **Solids** in the main menu again, and then on **Torus**. In the **Data Entry** tab again, type in these values:

Major Radius: 2.
Minor Radius: 1.

In the **Origin** section, make sure that **Enter** is selected and set the **Z**, **X**, and **Y** coordinates to 0. Click **OK** and then on **Cancel** to quit the function.



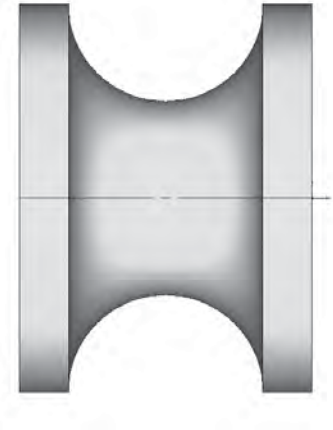
Click on **Utilities** again in the main menu, and choose **Rotate**. Click on the new torus once, then set the settings for it to the same as used for step 2. Click **OK** twice again and the torus will rotate.

Click on **Utilities** once more and choose **Translate**. Click on the torus, then in the **Data Entry** tab click on **Delta** and choose **Enter** under **End**. Set the values as here:

Z: -1.5
X: 0.
Y: 0.

Click **OK** twice again, then **Cancel** to quit the function. The part is nearly complete.

Click on **Solids** in the main menu, then on **Subtract**. Click the cylinder first, then the torus. Right-click in the **Workspace** and choose **OK**. The torus will be removed from the cylinder. Click **Cancel** in the **Data Entry** tab again and the part will be complete.



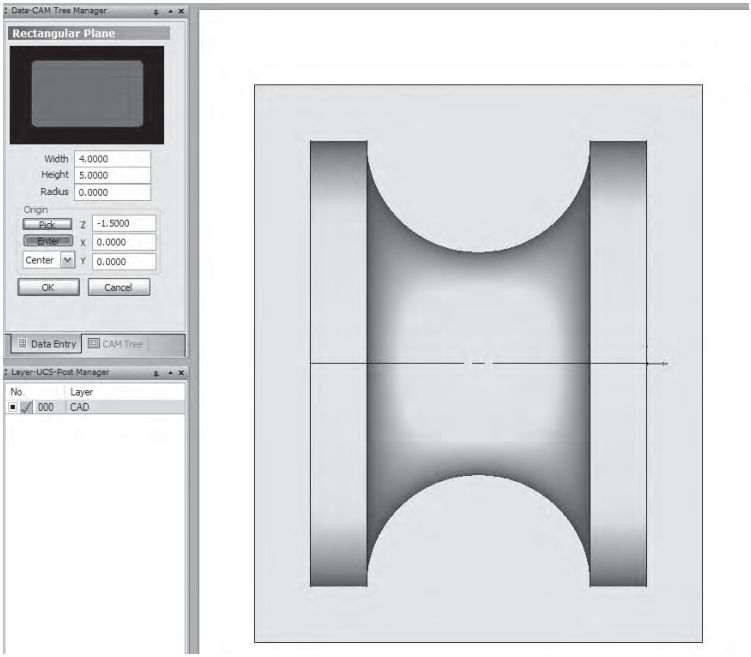
Step 3: In order to get the cross section profile out of the part, it is necessary to first make a cutting plane. Click on **Surfaces** in the main menu and then on **Rectangular Plane**. In the **Data Entry tab**, enter these values:

Width: 4.
Height: 5.

Under the **Origin** section, make sure that **Enter** is selected, the drop down box in on **Center**. Enter in these values in the coordinate fields:

Z: -1.5
X: 0.
Y: 0.

The plane should intersect all of the edges of the model, so if skipping here to use a pre-existing solid adjust the numbers so that the plane completely surrounds the edges of the model as seen in the illustration. Click **OK** to draw the plane then click **Cancel** to exit the function.

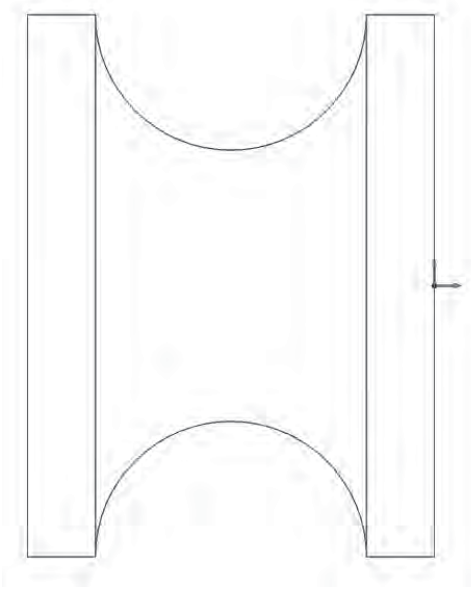


Step 4: Click on **Solids** once more, then on **Subtract**. Highlight the solid of the part first, then on the plane. Click **OK**. The plane will be removed from the solid. As the plane has no thickness, all that will be left is a new edge completely around the part.

Since line and arc geometry are necessary for the cutting functions, the next step is to extract the edges from the model.

Click on **Utilities** from the main menu. Choose **Extract Edges** and from the pop-up menu choose **From Solid**. Click on the solid once, then right-click on the **Workspace** and choose **OK**. All of the edges in the model will be extracted as lines and arcs. Press **Cancel** in the **Data Entry** tab to quit the function.

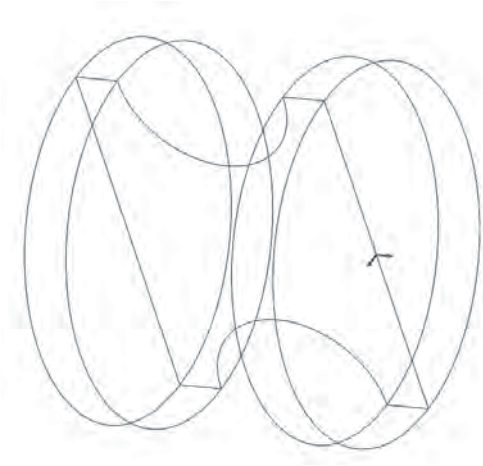
The solid is no longer necessary in the drawing and can be deleted. If not already in selection mode, click the black arrow icon on the toolbar, then click on the solid. It should highlight. Press Delete on the keyboard and the solid will disappear, leaving only the edges that were extracted.



Step 5: The last step is to clean up the edges so that they are ready for use.

Click on **Utilities** in the main menu then choose **Break** and *Many*. Click off of the model to some corner and drag a box over the entire drawing. When the mouse button is released, BobCAD-CAM will automatically break every line and arc where it touches any other. This will make it easy to remove the parts of the drawing that are not necessary. Click **Cancel** in the **Data Entry** tab.

Rotate the drawing some using the dynamic rotate icon on the toolbar. Click on the black arrow icon on the toolbar again to put BobCAD-CAM back into selection mode. Highlight those lines and arcs that do not belong in the drawing and press the delete key on the toolbar to remove them.



When finished, select the Top icon on the toolbar to return to the top view. The profile will now be ready for use.

BobCAD-CAM V23

Lathe

Chapter 5
Turning Toolpaths

Chapter 5. Generating Tool Paths and Code

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, and the tools it uses.

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 such operation 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** – The material being machined. This item has no impact on the actual code that is generated. This is used primarily as a reference for the programmer.
- **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. For Turning opeerations, these can be lines, arcs, points and contour entities.
- **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 Turning feature for a lathe may consist of these operations, for example:

  1. Roughing the profile.
  2. Finishing the profile.

  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 in 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 **Turning Stock** and select **Edit**.

After the stock is set up, right-click on it and choose a **Turn** 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 Introduction to the Examples

The examples in this portion of the manual are intended for use with the software in **Diameter** mode. To make sure that it is in this mode:

1. Click on **Preference** 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.

The exercises are also intended to follow in order, but are designed in a way that the user can approach them each on their own. Each one includes a step for drawing the geometry required for the exercise.

***Note**: The direction of the toolpath is determined by the orientation of the tool. To override the direction of the toolpath it is necessary to create a Contour out of the geometry in use. The direction of the contour will dictate the direction the tool travels.*

Most of the examples also contain important notes about the geometry or tool types each function requires. Keep these in mind when programming real-world parts.

5.3 Facing Example

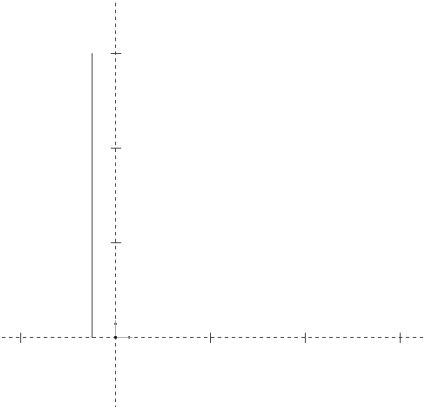
Step 1: Begin with an empty drawing by using any one of these 3 methods:
Click on **File** in the main menu and then **New** OR

Click on the **New File** icon in the toolbar OR
Type CTRL+N to start a new drawing.
If the **Lathe** icon on the toolbar is not depressed, click it now to turn **Lathe** mode on.

Step 2: Click on Line in the main menu and then on **Coordinates**. In the **Data Entry** tab of the **Data-CAM Tree Manager**, enter in these value:

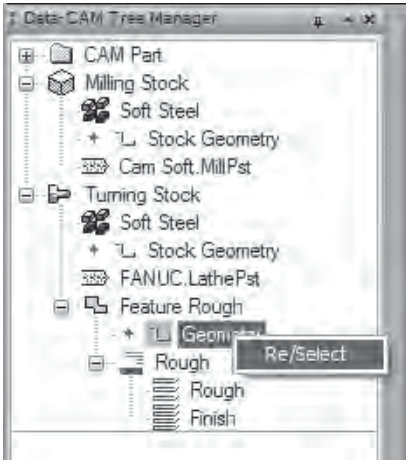
Start Z: -.25
Start X: 4.
Start Y: 0.
End Z: -.25
End X: -0.01
End Y: 0.

Then click **OK** or press Enter and the line will be drawn in the Workspace.



Step 3: 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.

Step 4: Assign the lines just drawn as the geometry for the feature by right-clicking on **Geometry** and choosing **Re/Select**.



Drag a window around the geometry in the Workspace to highlight the entities that were just created.

Right-click in the **Workspace** and choose **OK** from the pop-up menu. The geometry is set in the feature now.

***Note**: The direction of the toolpath is determined by the orientation of the tool. To override the direction of the toolpath it is necessary to create a Contour out of the geometry in use. The direction of the contour will dictate the direction the tool travels.*

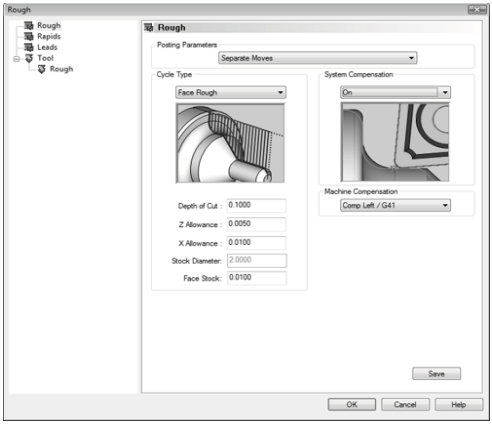
Step 5: Right-click on the **Rough** item in the new feature. Choose **Edit**.

The first item in the tree to the left of the box that will pop-up, called **Rough**. In the right-hand pane, set the **Posting Parameters** to **Separate Moves.** This will cause the system to generate the program using line, arc and rapid moves only, instead of Canned Cycles.

Then set the **Cycle Type** to **Face Rough**. This will ensure that the tool motion will be vertical along the face of the stock. Notice that the Stock Diameter field is grayed out and the Face Stock field is made available.

In the **Cycle Type** fields, set the parameters to these numbers:

Depth of Cut: 0.1
Z Allowance: 0.005
X Allowance: 0.01
Face Stock: 0.01



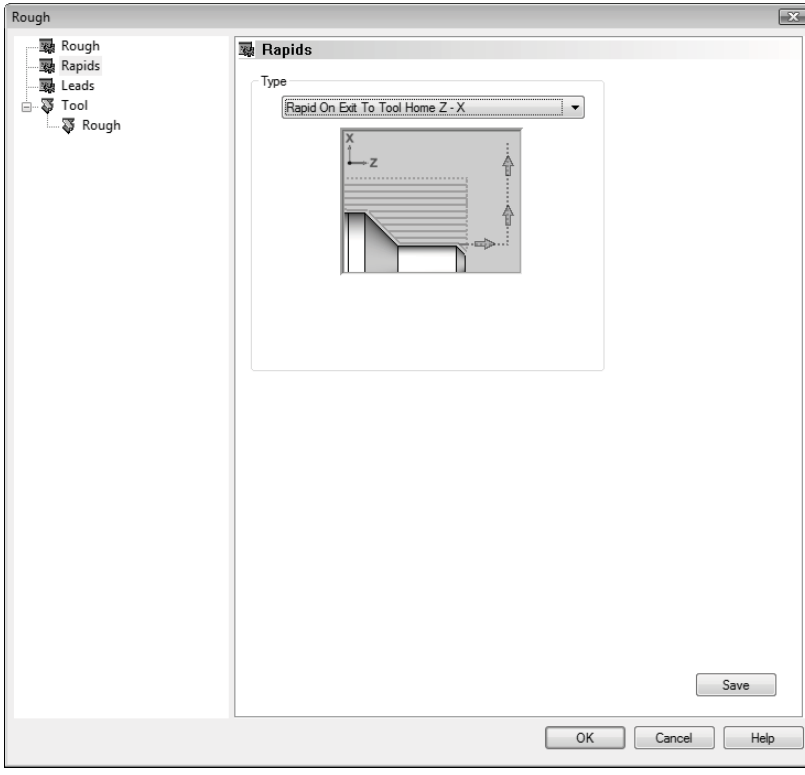
The **Face Stock** field tells the software where to begin the facing operation, while the geometry that we have assigned to the feature tells the facing operation where it is to stop.

To the right under **System Compensation**, set the drop-down box to **On**. This option determines whether or not the toolpath will be compensated for the toolnose radius.

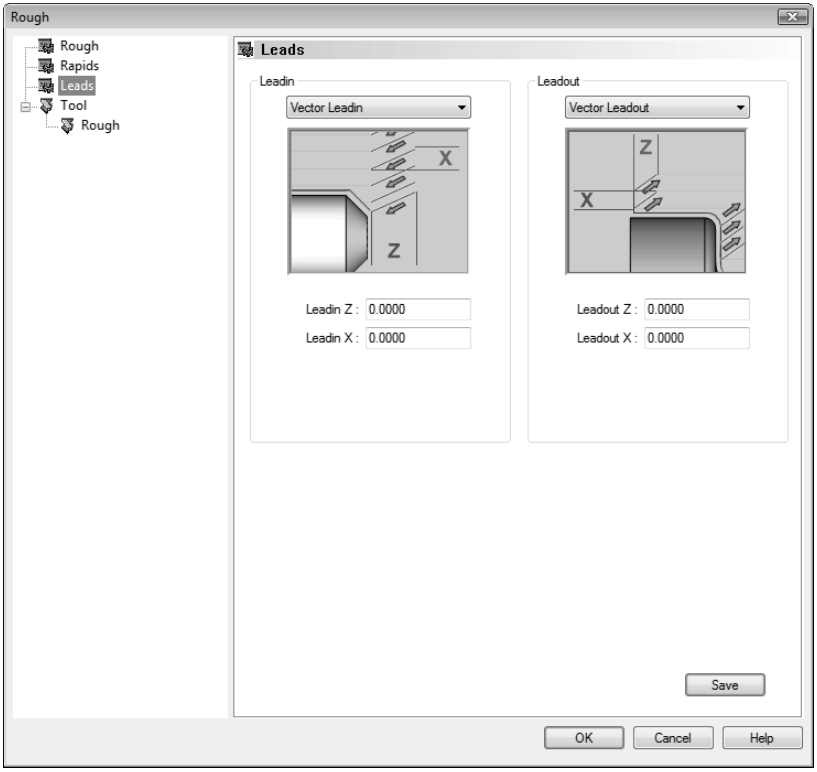
Under **Machine Compensation**, set the drop-down box to **Comp Left / G41**. This will set the tool motion to the left of the drawn path in the direction of cut and will cause the system to output the command for Left Compensation in the posted program.

***Note**: In this example only one line has been drawn. This is because the facing operation uses the minimum and maximum X values to determine where the toolpath starts and stops in X. The Face Stock value defines the Z start point and the maximum X value determines the X start point of the toolpath.*

The next item in the tree on the left is **Rapids**. Click on it, then set the drop-down box in the right pane to **Rapid on Exit To Tool Home Z – X**. This setting will cause the tool to retract in Z first before executing any X moves after the toolpath is generated.

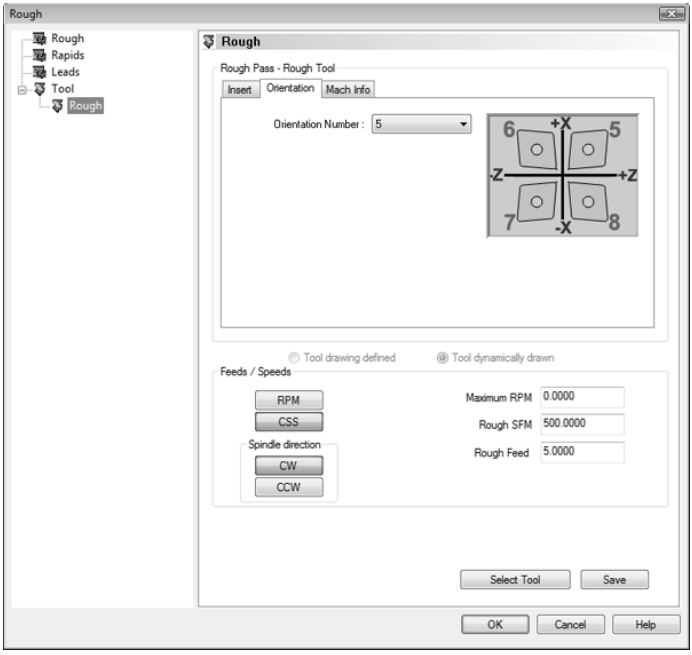


Click on **Leads** in the tree. The default lead-in and lead-out on all of BobCAD-CAM's toolpath is for no extra lead. This is acceptable for most cuts, and is good here.



***Note**: The direction of the leads are tied directly to the orientation of the tool. For tools using orientations #1 and #5, the leads are not adjusted. For orientations #2 and #6, the Leadin Z value is reversed. For orientations #3 and #7 both the Leadin X and Leadin Z are reversed and for orientations #4 and #8 the Leadin X is reversed. This is done so that when switching from one orientation to another, the Leadin and Leadout style reflect the location of the feature being machined.*

Under **Tool** in the tree, there should only be one item, **Rough**. Click on it. The default tool is fine, but the orientation of the tool insert will need to be checked to reflect a facing tool. Click on **Orientation** and choose **5** from the drop-down box if it isn't chosen already. Number 5 is the default for facing operations in BobCAD-CAM and it is the correct tool tip direction for this operation.



The setup for the facing operation is complete, so click **OK**. The system will automatically calculate the toolpath based on the inputs provided. If the feature is edited and any values are changed, the system will re-compute the toolpath automatically.

5.4 Outside Rough / Finish Example

This exercise is designed to follow section 5.3, or it may be done in a new file. If a new file is preferred, follow the instructions in **Step 1** of the facing exercise to start with a clean drawing.

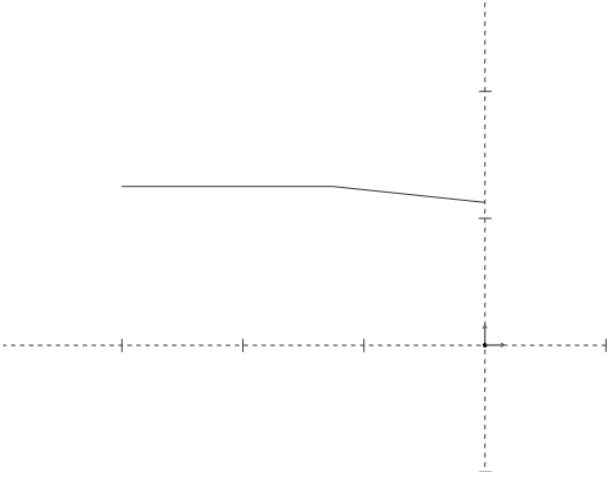
Step 1: Click on **Line** and then **Coordinates**. In the **Data Entry** tab of the **Data-CAM Tree Manager**, enter in these values:

Start Z: -3.
Start X: 2.75
Start Y: 0.
End Z: -1.25
End X: 2.75
End Y: 0.

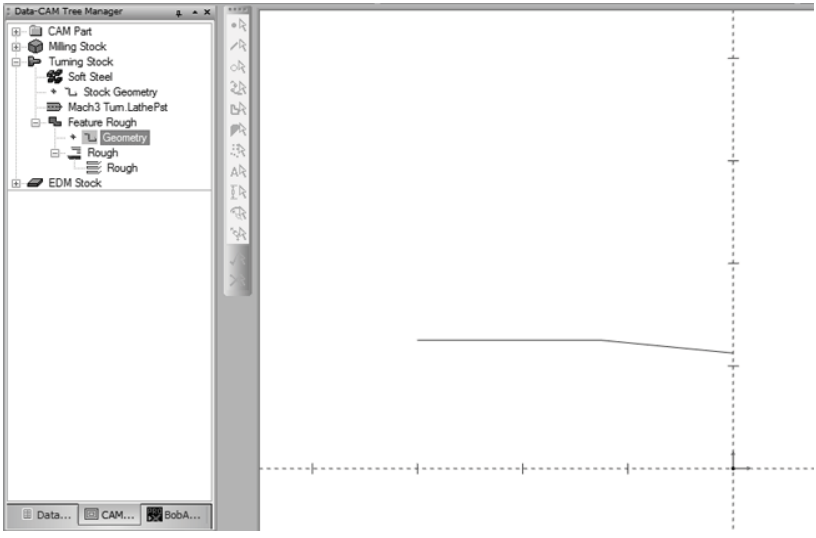
Click **OK**. The first line will be drawn and the data entry fields will stay available. Enter in this next set of values:

Start Z: -1.25
Start X: 2.75
Start Y: 0.
End Z: 0.
End X: 2.5
End Y: 0.

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



Step 2: 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.



Step 3: Assign the lines just drawn as the geometry for the feature by right-clicking on **Geometry** and choosing **Re/Select**.

Drag a box around the geometry to select it all.

Then right-click in the **Workspace** and choose **OK** from the pop-up menu. The geometry is set in the feature now.

***Note**: The direction of the leads are tied directly to the orientation of the tool. For tools using orientations #1 and #5, the leads are not adjusted. For orientations #2 and #6, the Leadin Z value is reversed. For orientations #3 and #7 both the Leadin X and Leadin Z are reversed and for orientations #4 and #8 the Leadin X is reversed. This is done so that when switching from one orientation to another, the Leadin and Leadout style reflect the location of the feature being machined.*

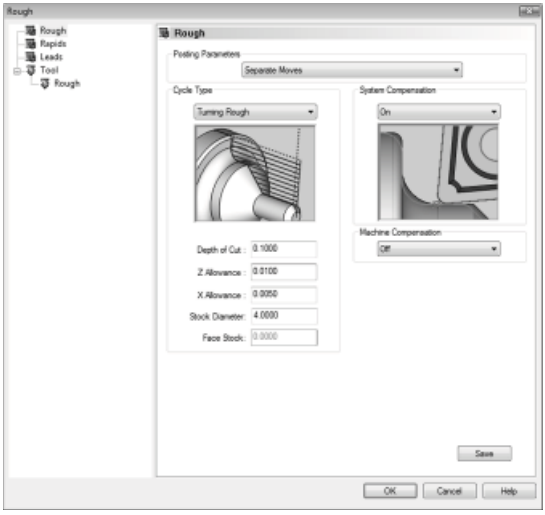
Step 4: Right-click on the **Rough** item in the new feature. Choose **Edit**.

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 **Posting Parameters** to **Separate Moves**. This will cause the system to output line, arc and rapid moves only, instead of a Canned Cycle.

Then set the **Cycle Type** to **Turn Rough**. This will ensure that the tool motion will be horizontal along the turning axis of the stock. Also notice that the **Face Stock** field is grayed out and the **Stock Diameter** field is available for input.

In the Cycle Type section, enter in these values:

Depth of Cut: .1
Z Allowance: .01
X Allowance: .005
Stock Diam.: 4.

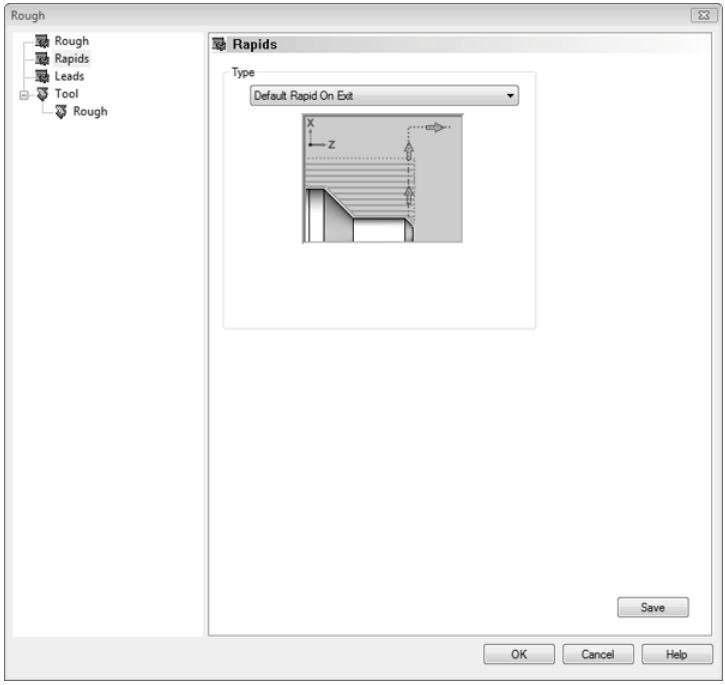


To the right under **System Compensation**, set the drop-down box to **On**. This option will cause the system to generate a toolpath that is compensated for the toolnose radius.

Under **Machine Compensation**, set the drop-down box to **Comp Right / G42**. This will set the tool motion to the right of the drawn path in the direction of cut and will cause the system to output the command for Right Compensation in the posted program.

***Note**: In this example only two lines have been drawn. This is because the turning operation uses the minimum and maximum Z values to determine where the toolpath starts and stops in Z. The Stock Diamter value defines the X start point and the minimum Z value determines the Z start point of the toolpath.*

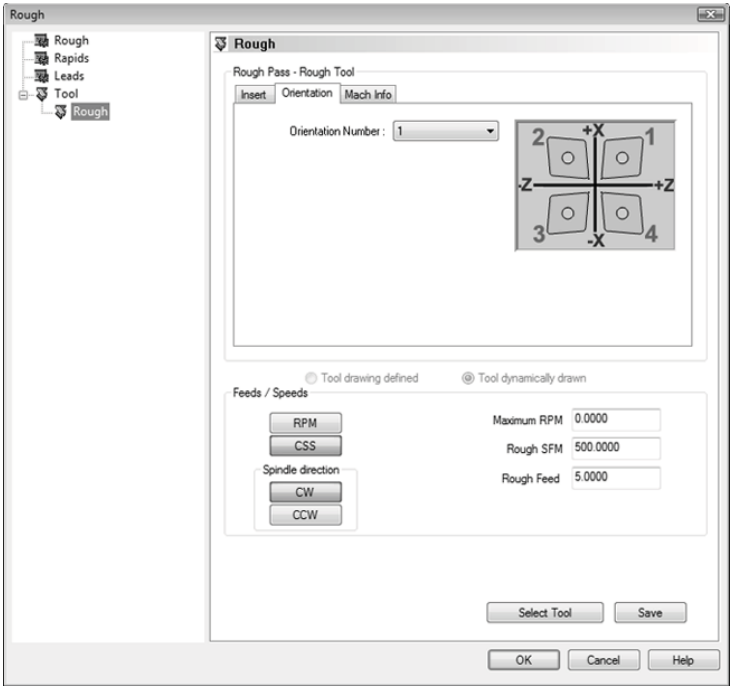
The next item in the tree on the left is **Rapids**. Click on it, then set the drop-down box in the right pane to **Default Rapid on Exit**. For Turning operations the default is **Rapid on Exit to Tool Home X - Z**. Since this is an OD cut, a rapid in X away from the part first is required, so this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

***Note**: The direction of the leads are tied directly to the orientation of the tool. For tools using orientations #1 and #5, the leads are not adjusted. For orientations #2 and #6, the Leadin Z value is reversed. For orientations #3 and #7 both the Leadin X and Leadin Z are reversed and for orientations #4 and #8 the Leadin X is reversed. This is done so that when switching from one orientation to another, the Leadin and Leadout style reflect the location of the feature being machined.*

Under **Tool** in the tree, there should be one item, **Rough**. 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.



The setup for the OD Turning operation is complete, so click **OK**. The system will automatically calculate the toolpath based on the input values. Whenever a value is modified within a turning operation, the system will calculate the toolpath with the new values automatically.

5.5 Drill Example

This exercise is designed to follow section 5.3 or it may be done in a new file. If a new file is preferred, follow the instructions in step 1 of the facing exercise to start with a clean drawing.

Step 1: Click on **Point** and then **Coordinates**. In the **Data Entry** tab of the **Data-CAM Tree Manager**, enter these values:

Z: -3.5
X: 0.
Y: 0.

Click **OK** and then **Cancel** to quit the function. Only one point is required for the drilling operation in Turning mode.

Step 2: Right-click on **Turning Stock** in the **CAM Tree** tab of the **Data-CAM Tree Manager**. Choose **Turn** and then **Drill**. A new **Feature Drill** feature will be added to the tree.

Right-click on **Drill Depth Point** and choose **Re/Select**. Click on the point in the drawing. Right-click in the drawing window and choose **OK**. This point represents the final depth of the drilling operation.

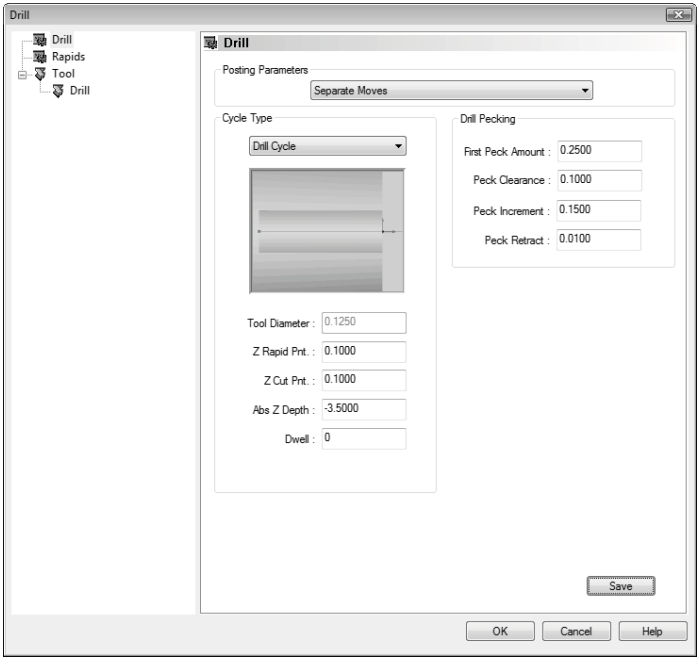
Step 3: Right-click on the **Drill** item in the feature and choose **Edit**. In the **Posting Parameters** field select **Canned Cycles**. This will make the system output the drilling operation using the canned cycle definition in the post processor.

In the Cycle Type fields, set these values:

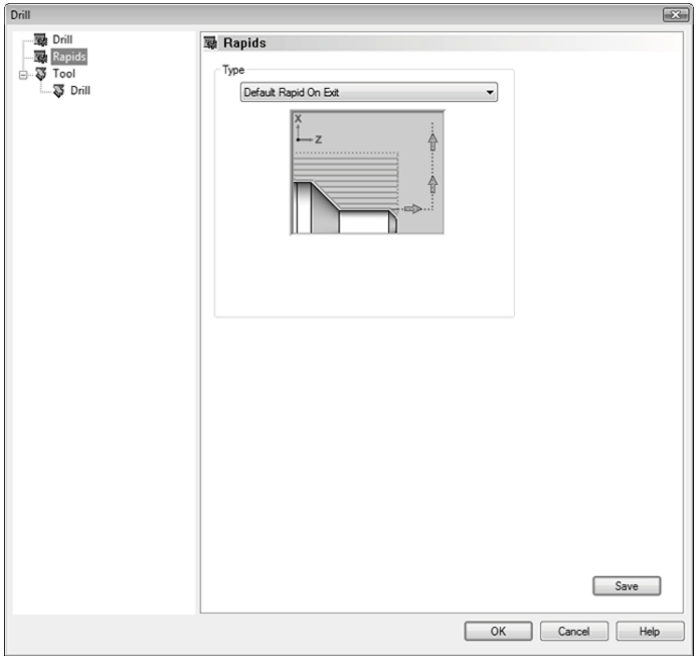
Z Rapid Point: .1
Z Cut Point: .1
Abs. Z Depth: This will already say -3.5, as this is the Z position of the point.
Dwell: 0.

To the right of the **Cycle Type** fields will be the pecking parameters. Since the total depth of the hole is 3.5" and the drill diameter will be .5", it will be a good idea to peck it. Enter in these values in the boxes:

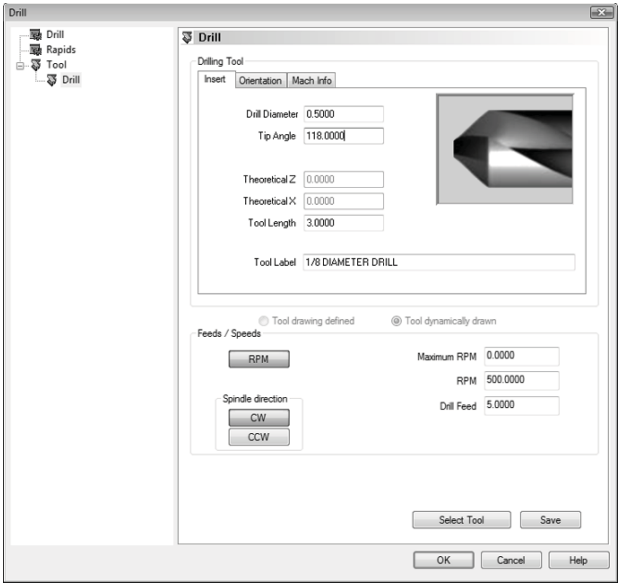
First Peck Amount: .25
Peck Clearance: .1
Peck Increment: .15
Peck Retract: .01



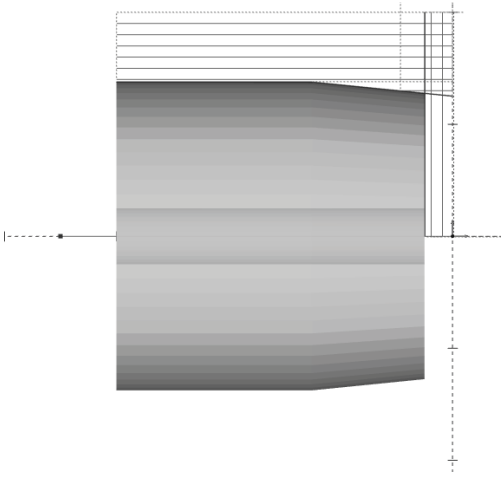
Click on the **Rapids** item in the tree to the left. Set the **Type** to **Rapid on Exit To Tool Home Z - X**. This will ensure that the drill will retract all the way to the home position in the Z axis first.



Under **Tool** in the list to the left, choose **Drill**. In the pane to the right, set the **Drill Diameter** to .5. Now the software will know to drill a .5” inch hole.



The setup for the drilling feature is complete, so click **OK**. The system will automatically calculate the toolpath for the drilling operation and update the preview of the stock in the workspace.



5.6 Inside Rough / Finish Example

This exercise is designed to follow section 5.5 or it may be done in a new file. If a new file is preferred, follow the instructions in step 1 of the facing exercise to start with a clean drawing.

Step 1: Click on **Line** and then **Coordinates**. In the **Data Entry** tab of the **Data-CAM Tree Manager**, enter in these values:

Start Z: -2.
Start X: 1.5
Start Y: 0.
End Z: -.2
End X: 1.5
End Y: 0.

Click **OK**. A line will be drawn and the data entry fields will stay available. Click on **Cancel** in the **Data Entry** dialog. Only one line is needed for this operation.

Step 2: 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.

Left click on the line that was created in the previous step. When it is highlighted, right-click in the Workspace and choose **OK** to finish assigning the geometry.

***Note**: The direction of the leads are tied directly to the orientation of the tool. For tools using orientations #1 and #5, the leads are not adjusted. For orientations #2 and #6, the Leadin Z value is reversed. For orientations #3 and #7 both the Leadin X and Leadin Z are reversed and for orientations #4 and #8 the Leadin X is reversed. This is done so that when switching from one orientation to another, the Leadin and Leadout style reflect the location of the feature being machined.*

Step 3: Right-click on the **Rough** item in the new feature. Choose **Edit**.

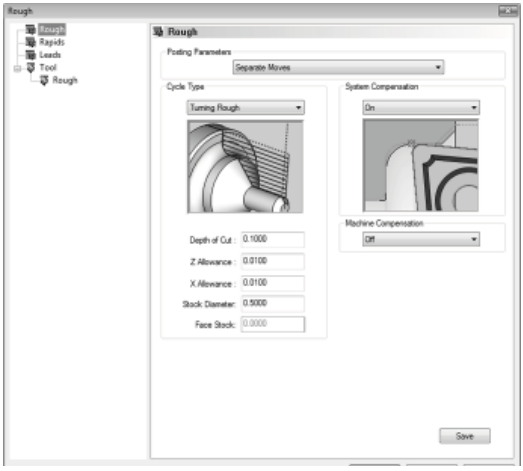
Then click on the first item in the tree to the left of the box that will pop-up, called **Rough**. In the **Posting Parameters** field, select the **Separate Moves** option.

In the **Cycle Type** field, 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.

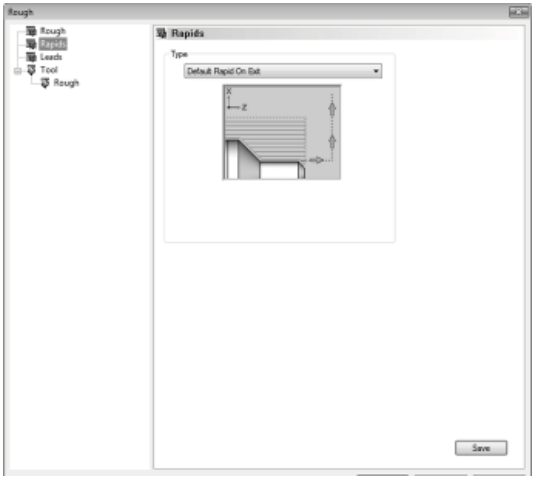
Below that section, enter in these values:

Depth of Cut: .1
Z Allowance: .01
X Allowance: .01
Stock Diam.: 0.5

To the right under **System Compensation**, set the drop-down box to **On**. This will set the tool motion to the left of the drawn path in the direction of cut, in this case, toward the inside of the stock.

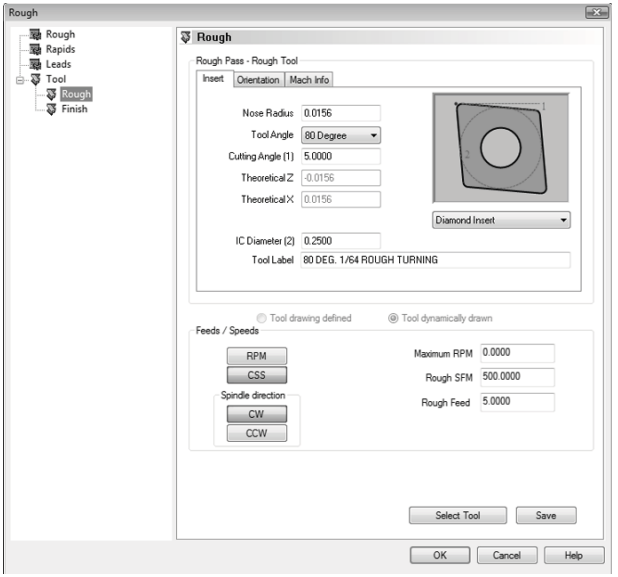


The next item in the tree on the left is **Rapids**. Click on it, then set the drop-down box in the right pane to **Rapid on Exit to Cycle Start X - Z**. For most operations the default is **Rapid on Exit To Tool Home X – Z**, but since this is an ID cut, a rapid in X toward the center first is required. The cycle start point will be lower-right corner of the generated toolpath. If the default had been used, it would rapid to tool home in X first, through the stock. For ID cuts, cycle start is a better place because of its location.



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.

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 an ID turning tool. Click on **Orientation** and choose **4** from the drop-down box if it isn't chosen already. Number 1 is the default for turning operations in BobCAD-CAM Version 23, but that is an incorrect tool tip direction for this operation. Do the same for the **Finish** tool.



The setup for the ID turning operation is complete, so click **OK**. The system will automatically calculate the toolpath and display the toolpath in the **Workspace**.

***Note**: The tool tip orientation is what determines if a turning cut is an ID or OD cut. If the toolpath appears on the wrong side of the geometry after it is computed, it is likely that this setting is the cause.*

5.7 OD Thread Example

This exercise is to show the user the basics of using the threading feature. It will be a simple 1/2-13 UNC OD screw thread. It will use the default 1/8 .003R 60° threading tool.

Step 1: Right-click on **Turning Stock** and choose **Edit**. This will bring up the **Stock Edit** dialog. Change the **Stock Diameter** field to 2.0 and click **OK**.

Step 2: Click on **Line** and then **Coordinates**. In the **Data Entry** tab of the **Data-CAM Tree Manager**, enter in these values:

Start Z: 0.
Start X: 2.
Start Y: 0.
End Z: -1.
End X: 2.
End Y: 0.

This line is being drawn on the O.D. of the stock intentionally. When creating a threading operation it is necessary for the geometry being used to lay on the stock. Whether it is the original stock, or the stock remaining after another operation. If the geometry is not located on the stock the preview will be incorrect.

Step 2: Click on the **CAM Tree** tab of the **Data-CAM Tree Manager**. Right-click on **Turning Stock**, choose **Turn**, and then **Thread** from the menu. A new **Feature Thread** feature will be added to the tree.

Step 3: Right-click on the **Thread** item under the new feature and choose **Re/Select**. Then left click on the line to select it. The direction of the tool will be determined by its' orientation, which will be addressed later in this example.

Right-click on the **Workspace** somewhere and choose **OK**. The geometry will be set in the feature and will no longer have the red dot next to it.

***Note**: Threading operations require the geometry to be only ONE single line.*

Step 4: Right-click on the **Thread** 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 **Thread**. In the right-hand pane, set the **Thread Type** to **Canned Cycle**. This will ensure that the posted NC code will use the post processor's complex thread canned cycle, normally a G76 cycle on most controls.

Set the **Thread Leadin Angle** to **60 Degrees**. This is the most common leadin angle for threading so it is the default in the feature.

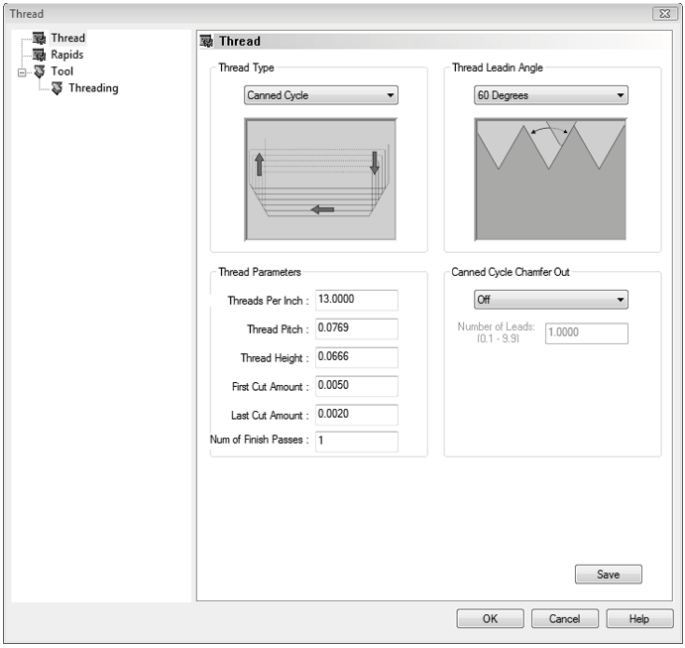
Since this is an OD thread for a screw there will normally be a wall to the left of the thread, so set the **Canned Cycle Chamfer Out** to **Off**. If this were a pipe thread, there would normally be a chamfer on the end, and in that case, it would need to be set to **On**.

Under **Thread Parameters** is most of the data used to cut the thread. Enter in these values:

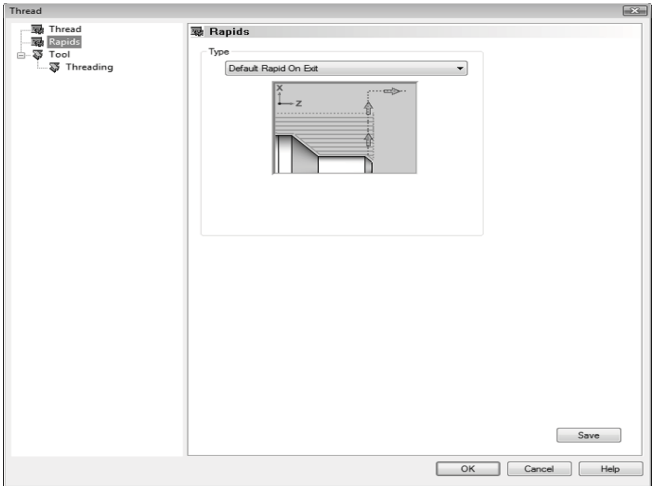
Threads Per Inch: 13
First Cut Amount: .005
Last Cut Amount: .002
Num. of Finish Passes: 1

While the other values in this field are editable, they will remain unchanged in this exercise.

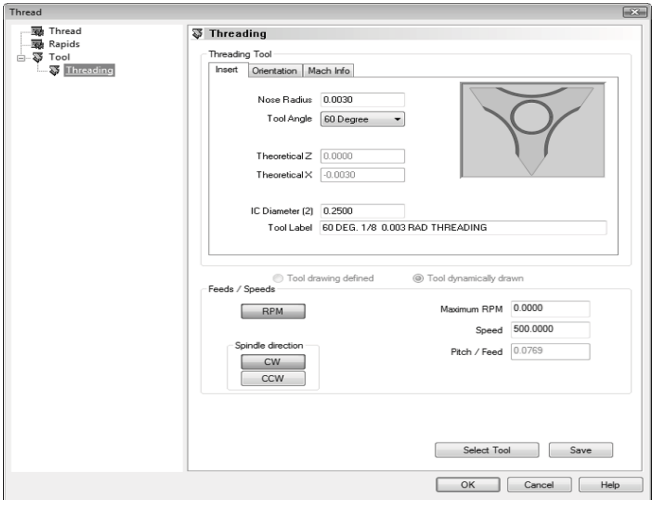
***Note**: The* **Threads Per Inch** *and the* **Thread Pitch** *settings work from one another. When one of them is changed, the other changes to match. For example, if the* **Thread Pitch** *were set to .05, the* **Threads Per Inch** *parameter would automatically change to 20, and vice-versa.*



The next item in the tree on the left is **Rapids**. Click on it, then set the drop-down box in the right pane to **Rapid on Exit to Cycle Start X - Z**. For most operations the default is **Rapid on Exit To Tool Home X – Z**, but since this is an OD thread cut, there's really no reason to have to return it to tool home as the tool change on the machine will perform the same action. The cycle start point will be upper-right corner of the generated toolpath.

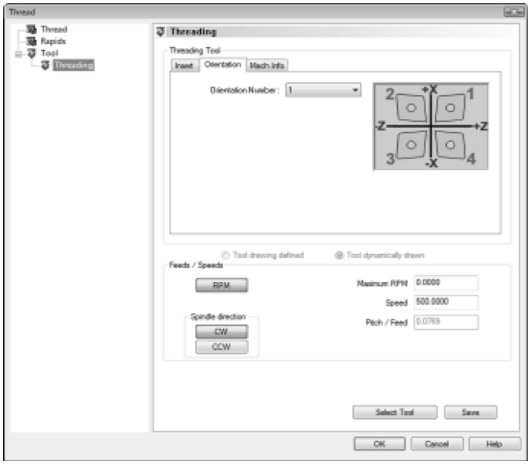


Under **Tool** in the tree there will be one item, **Threading**. Click on it to access the tooling parameters. The default tool is a 1/8” 60 degree threading tool with a .003” radius, and this is the tool to use for this thread.

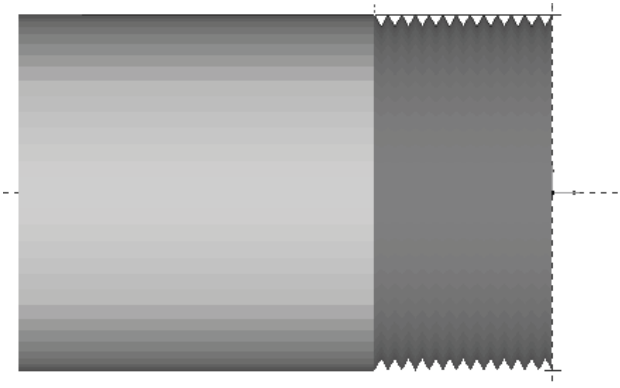


***Note**: The form the thread will take will be dependent on the tool loaded into the machine. If the thread is required to have a square form, as in an ACME thread, for example, the proper tool must be loaded into the machine. BobCAD-CAM calculates threading passes according to the angle on the flank of the thread, not its form.*

Next, click on **Orientation** and choose **1** from the drop-down box if it isn't chosen already. Number 1 is the default for OD threading operations in BobCAD-CAM. To cut an ID thread, simply switch the orientation to an ID direction and the operation will automatically reverse direction.

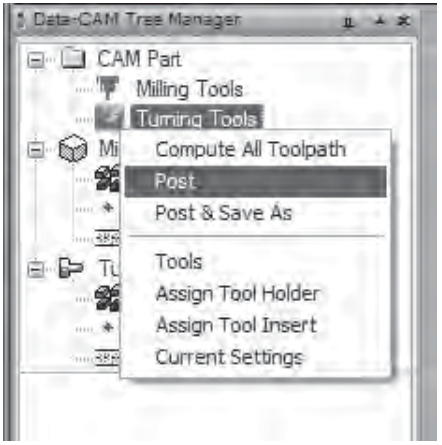


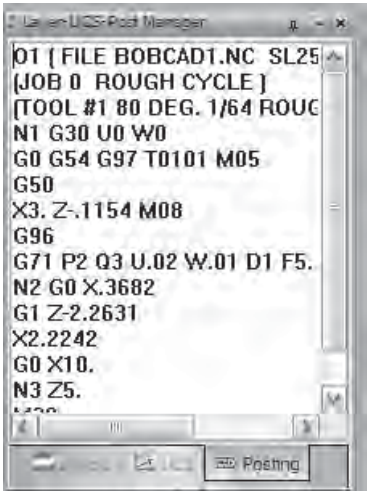
Step 5: The setup for the threading operation is complete, so click **OK**. Right-click on the **Thread** item in the feature and click **Compute**. The thread toolpath produced should look like the illustration below.



5.8 Posting a Program

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





BobCAD-CAM V23

Lathe

Chapter 6
Verification

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 diameter size of the turning 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 **Turning Tools** under **CAM Part**.

To simulate a machine running a program, right-click on **Turning Stock** in the **Cam Tree** tab of the **Data-CAM Tree Manager**. Choose **Verify** from the list. The simulation window will immediately appear with a cylinder the size of the stock 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.

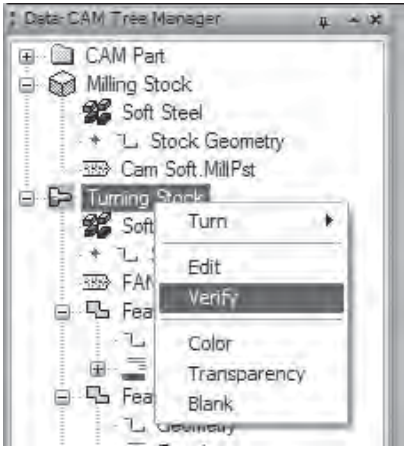
This pop-up menu carries a host of options:

- **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.

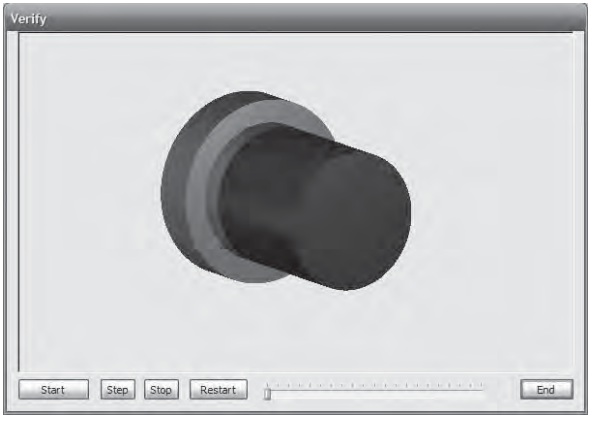
6.3 Example

Open any part file that has a computed CAM tree with toolpath in it, or execute one of the examples in **Chapter 5** prior to attempting this exercise. The simulation needs to have a part generated first before anything can be simulated.

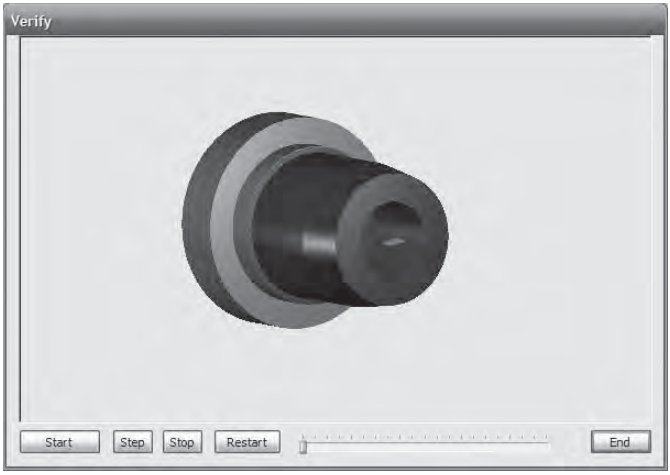
Step 1: Right-click on **Turning Stock** in the CAM tree and select **Verify**.



Step 2: In the simulation box that will appear, rotate the part around some when it cuts to see any inside cutting or drilling that may be in the part.



To run the simulation, simply click **Start**.



BobCAD-CAM V23

Lathe

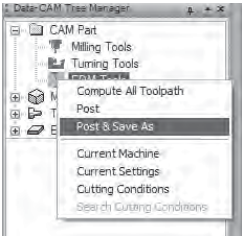
Chapter 7
Getting Code to the Machine

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.

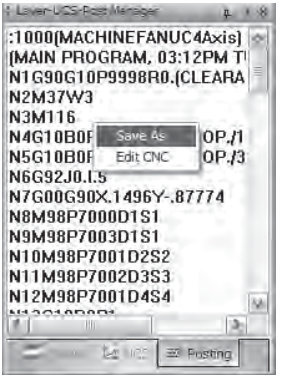
7.1 Saving Posted Programs

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.

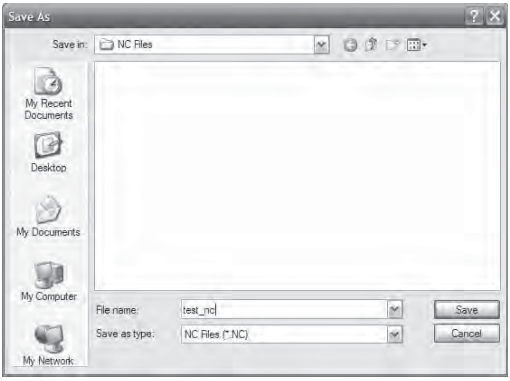


- **Save As** from the **Layer-UCS-Post Manager** – After the program has been posted, right-click in the **Layer-UCS-Post Manager** where the code is displayed and choose **Save As**. The **Save As** dialog will appear.



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

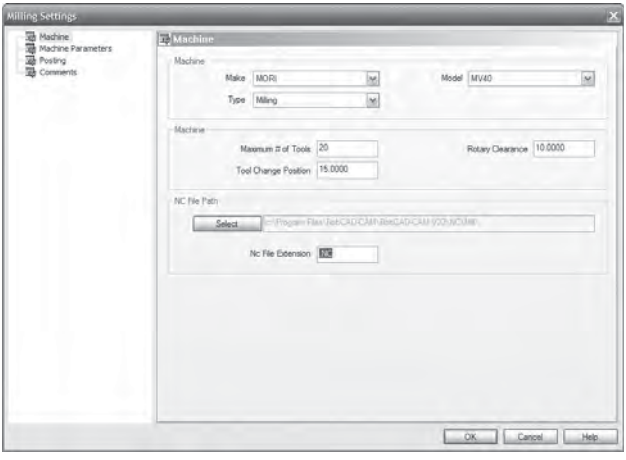


then burn the CD from there instead of attempting to save the file directly to the device.

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.

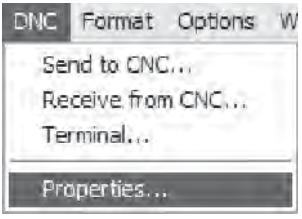
***NOTE**: ASCII text is the actual file format supported by probably 99% of all CNC machines ever manufactured, regardless of what file extension is required. If there is any doubt whether the machine accepts this format, post a program and try it. It is only in rare exceptions that this will not be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

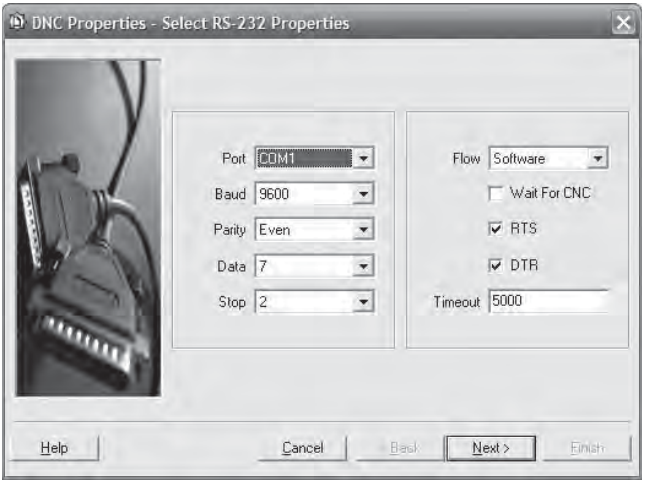
Communication Settings

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

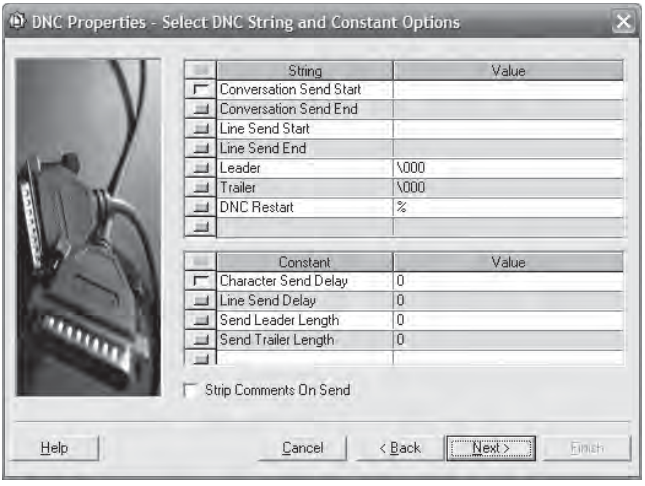


The system will display a series of 5 dialog boxes:

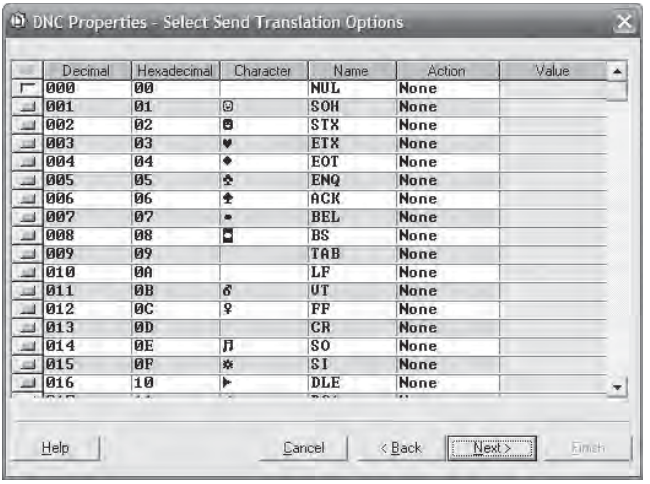
1. **Select RS-232 Properties** – Set the COM port options from here. Most controllers will require that these options be set to match the settings on the machine, but a few rare controls will actually require mismatched settings. See the controller's documentation to ensure the settings are correct.



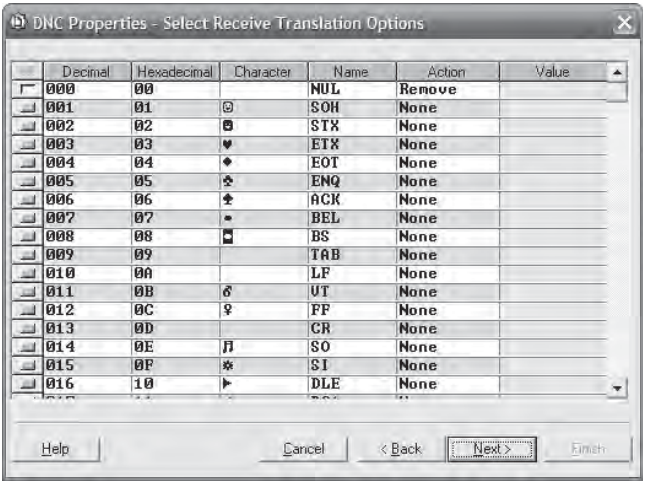
2. **DNC String and Constant Options** – Use this dialog box to set up any special leader or trailer characters the control may require.



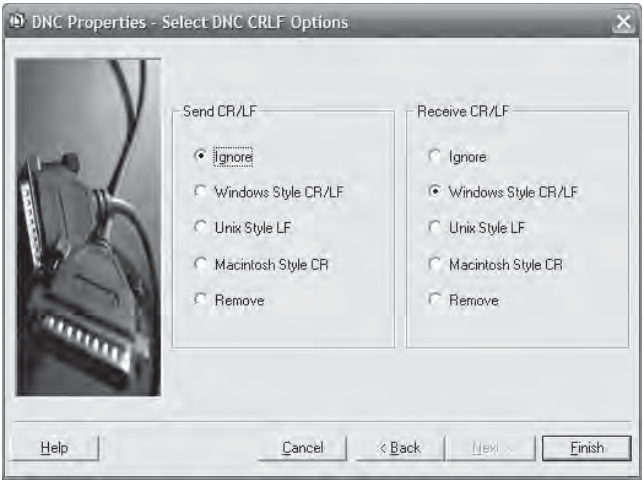
3. **Send Translation Options** – Occasionally a rare control will need a character normally output to be changed into something else entirely in order to read the transferred program. Normally no changes will be required; check the controller's documentation to be sure.



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.

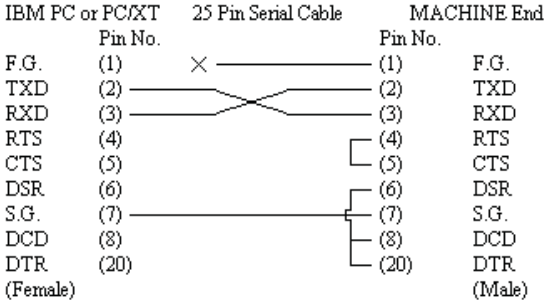


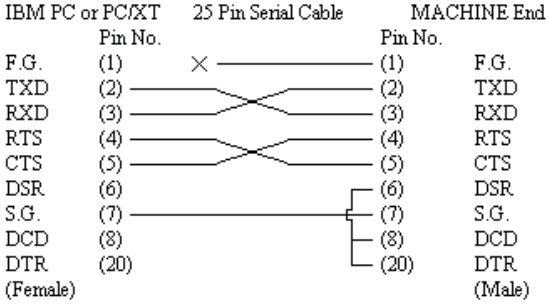
Click **Finish** when the settings are correct. The system will remember them for the next time a program is transferred.

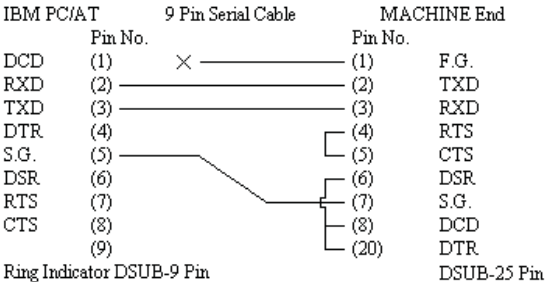
Communication Hardware

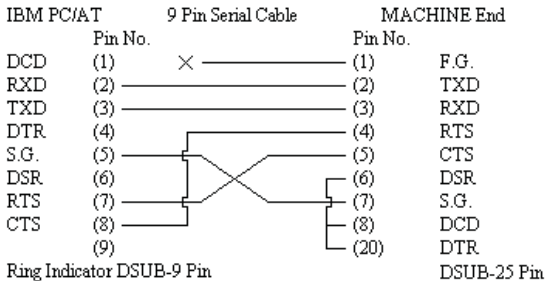
To send a file over an RS-232 (or RS-422) link, a properly wired cable is absolutely required. See the next page for the most common wiring diagrams.

These are the common pinouts for nearly all CNC controls.









Cables typically need 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.

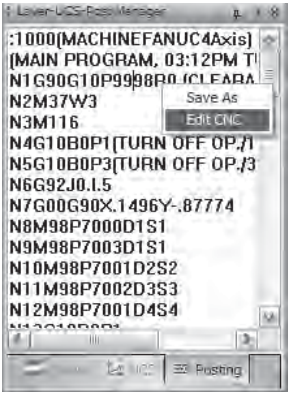
If a good cable is difficult to find or to build, or if the machine is more than 50' from the computer, it is now possible to purchase Predator Grizzly™ cables directly from BobCAD-CAM, Inc. Grizzly™ cables are viable to lengths of up to 400' in a variety of environments.

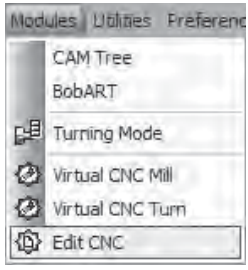
7.4 Transferring Files via RS-232 (Serial) Link

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

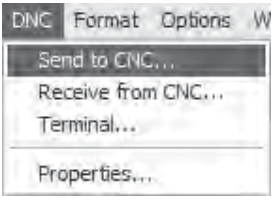
Send a Program to the CNC

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.





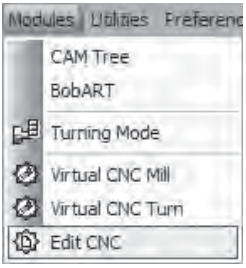
Step 2: Choose **Send to CNC…** from the menu



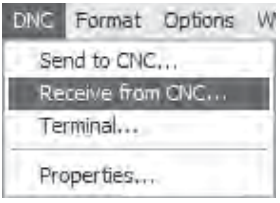
Step 3: The CNC Editor will automatically begin the transfer. The progress bar near the bottom of the dialog will count off the percentage of the file transferred. The progress bar will display 100% when it is complete.

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.