Wire EDM

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.

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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.
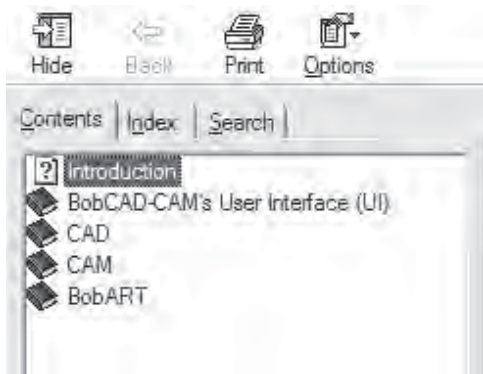
1.7 Training Seminars

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.
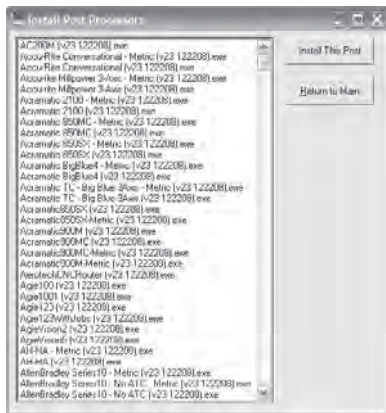
1.11 Technical Support

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.
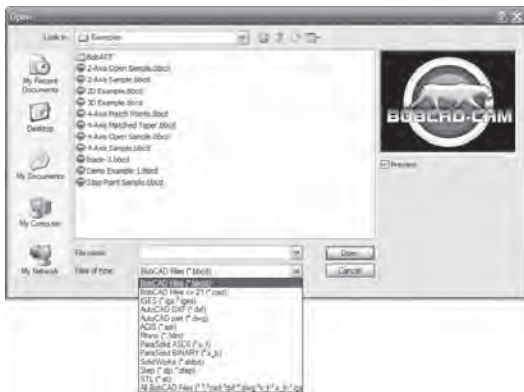
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Chapter 2
Files

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.
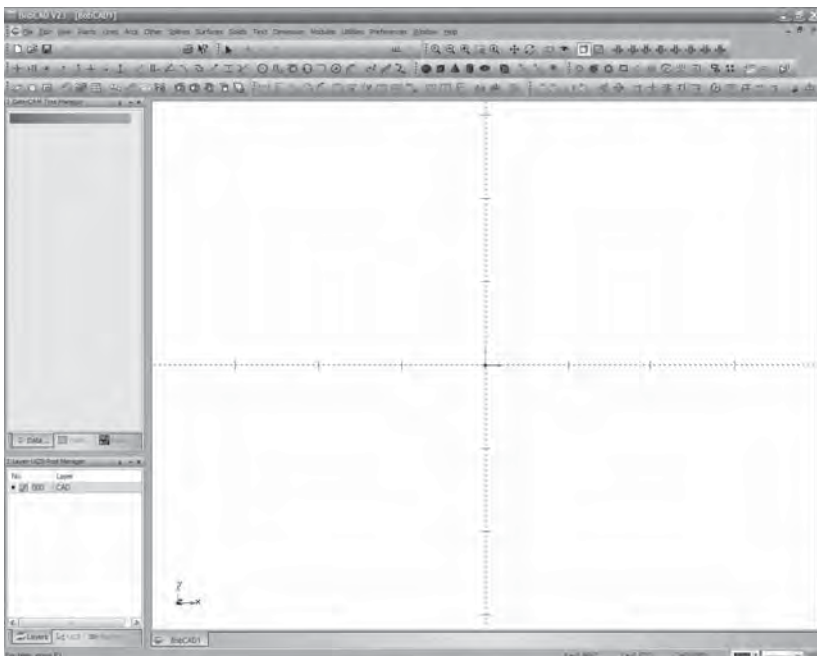
BobCAD-CAM
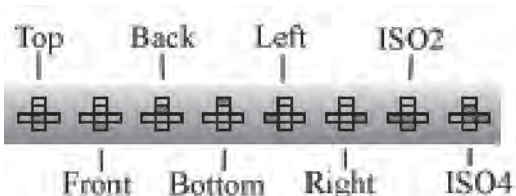# v23

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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
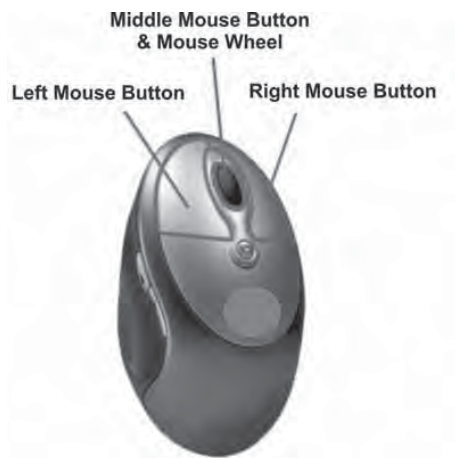
• *Left Mouse Button*

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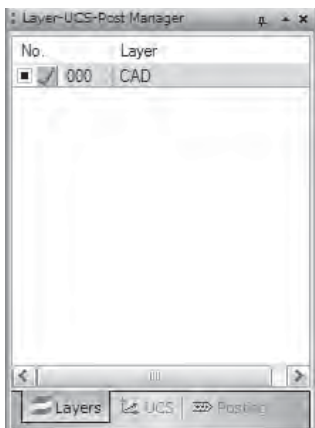
To create a new layer, right-click in the **Layer-UCS-Post Manager** and choose **Add New Layer**. Name the new layer whatever is necessary. BobCAD-CAM Version 23 keeps layers separated in the background by their numbers, not their names, so it is possible to name 2 layers the same if required.

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.

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Chapter 4
2 Axis Wire Features

BobCAD-CAM can draw with wireframes and with solids and surfaces. Generally speaking, wire frames consist of lines, arcs, points, and a few special items like fillets, gears, and ellipses. Solids have 5 different primitive types: cube, sphere, cone, cylinder, and torus. See the **Help** system in the software for a full explanation of each of these.

Described in this chapter is an example of a **2 Axis Inside** feature, from blank drawing to generating the NC program. The steps described apply to all of the 2 Axis features: **2 Axis Inside**, **2 Axis Outside**, and **2 Axis Open**.

4.1 2 Axis Drawing Example

The shape in this example could be used either for a 2 Axis program or for a 4 Axis program with a constant taper all the way around the shape.

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
- Hold the CTRL key down and hit N on the keyboard to start a new drawing.

Step 2: Click on **Other** from the main menu, and then click on **Rectangle**.
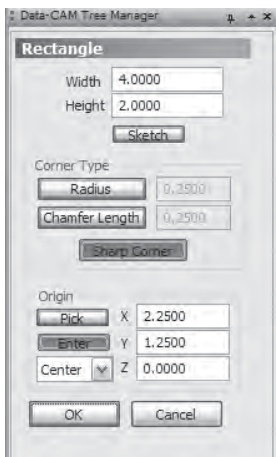
Enter in these values as in the illustration below:

Width: 4.00
Height: 2.00

Be sure **Sketch** is not set. Under **Corner Type**, press **Sharp Corner**. Under **Origin**, click **Enter**, and choose **Center** from the drop-down list if it isn't already set. In the number boxes, set these values:

X: 2.25
Y: 1.25

Click **OK** and the rectangle will appear in the **Workspace**.

Step 3: Click on **Points** in the main menu and then on **End**. Click near all 4 corners of the rectangle.

Step 4: Choose **Arcs** from the main menu and then **Fillet**. In the dialog in the **Data-CAM Tree Manager** on the left, enter in 10 next to **Radius** and click **OK**.

Click on the point on the upper-right corner of the rectangle, then on the point in the upper left. An arc will appear between these two points with a 10 inch radius.

Next, click on the point in the lower right corner, followed by the point in the lower left. Another, identical arc will appear. The drawing should look like the illustration on the next page.
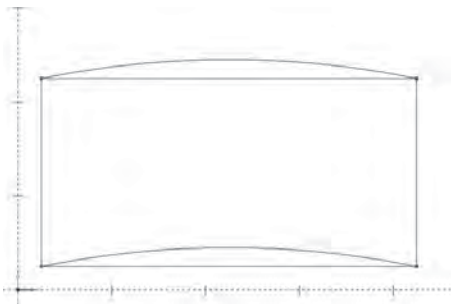
Step 5: Click on **Arcs** again in the main menu and choose **Snap**. Set the radius value in the dialog on the left side of the screen to .5". Set the **Start Angle** to 0, and the **End Angle** to 360. This will ensure that BobCAD-CAM draws a full circle.

Click on the upper arc of the drawing near its center. The circle will be drawn at the 90 degree position of this arc. By default, BobCAD-CAM has snap points at 45° intervals along an arc. Since the mouse was clicked closest to the top point on the arc, the circle will be drawn at 90 degrees on it.
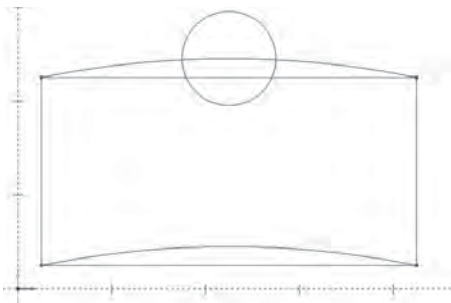
The drawing is nearly finished and only needs some cleanup to remove parts that are not required.

Step 6: Click on the **Utilities** in the main menu and hover the mouse cursor over **Break**. When the fly-out menu appears, choose **Many**.
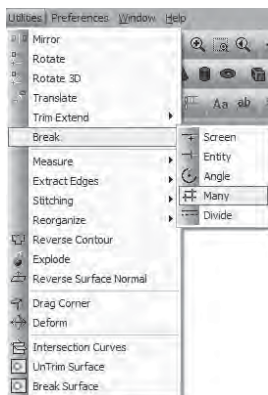
Click the mouse somewhere off the corners of the drawing as below. Press the mouse button and drag the mouse to the opposite corner as in the illustration. Right click anywhere in the **Workspace** and click **OK** from the menu. On release, the **Break Many** function will break every highlighted entity on the drawing where it crosses or intersects any other highlighted entity. This makes it simple to select and delete anything that is not required for the finished drawing.
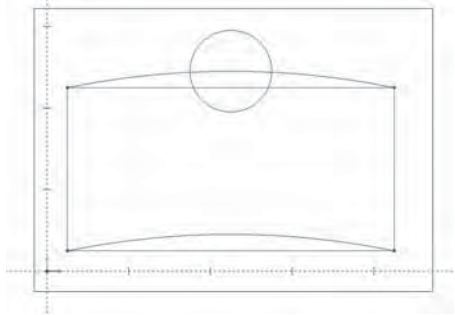
Step 7: Click on the **Selection Mode** icon ▶ on the toolbar. Click on every entity on the drawing that is not required as in the finished drawing illustration on the next page.

Click the trash can icon 🗑 on the toolbar or press the Delete key on the keyboard. The highlighted entities will be deleted from the drawing and the remaining entities should look like the drawing on the next page.
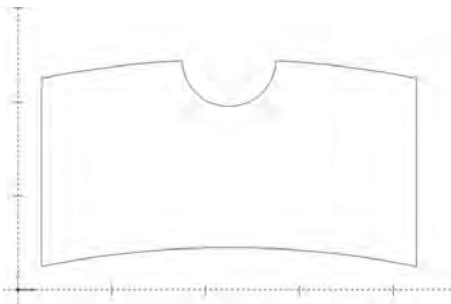
Click on **File** in the main menu and choose **Save As**. In the dialog box, save the file as simple 2 Axis Shape.bbcd as below. Make sure to remember which folder the drawing was saved in; it will be used in a CAM example in the next section.
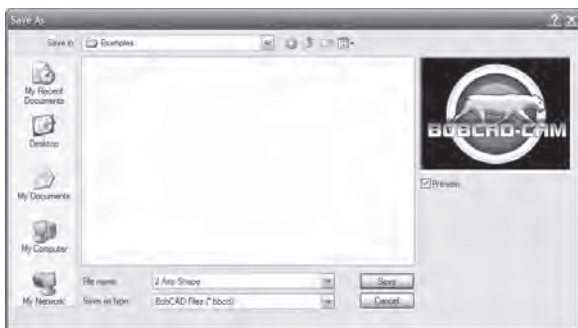
4.2 CAM Terminology

BobCAD-CAM V23's Wire EDM package is equipped with both 2 and 4 axis EDM CAM capability, as well as the ability to choose between several lead-in/out types, coreless 2 axis EDM, and can post code for some of the most difficult machines on the market. Before jumping directly into the examples, however, it will be helpful to define some of BobCAD-CAM's terminology relating to the CAM tree:

- **CAM Part** – This is the top of the CAM tree. At the **CAM Part** level such operations as post processing and toolpath generation work in a global manner, i.e., all the features in the tree will be computed and/ or posted at once. All of the preferred machine settings are also set at this level. Under **CAM Part** are **Milling Tools**, **Turning Tools**, and **EDM Tools**, where each type of machine can be set up.

- **Stock** – The Stock is defined using a contour and a depth. Once defined a transparent solid representing the stock can be displayed. **Material** – Based on the material type selected, the wire diameter, and the stock thickness the system will automatically calculate the cutting conditions.
- **Feature** – A feature consists of the geometry, single or multiple operations, machining parameters and the toolpath that was calculated using all these attributes.
- **Geometry** – The entities that are being used in the feature. Depending on the operation type these can be points, lines, arcs, splines, contours, etc.
- **Associativity** – Each feature contains all the necessary parameters to calculate tool path, including the geometry stored. This association of the geometry within the feature 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. For example, one feature on an EDM part may need several skim passes to maintain accuracy, but another feature on the same part may require only the roughing pass.

  Each of these steps is an operation, and together they make up a **Composite Operation**.

4.3 2 Axis CAM Example

Step 1. Begin by opening the 2 Axis Shape.bbcd file saved in section **4.1**. If this file does not exist, create it first before proceeding with this example so that it will be easier to follow the steps given here.

Step 2. Click on the **CAM** tab of the **Data-CAM Tree Manager**. Right click on **EDM Stock** as shown on the next page.

Hover the mouse cursor over **EDM** in the menu, and choose **2 Axis Inside**. Choosing **Inside** over **Outside** designates this part as a die inside the software, and tailors the options available to this feature to those suitable for a die. This example will include only a single feature in order to keep it short, but it is permitted to have multiple features in the same part.
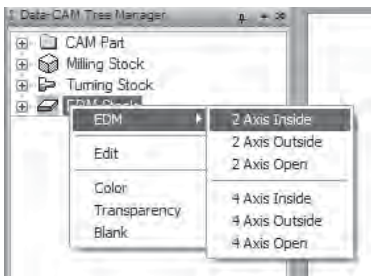
Step 3. To assign the geometry on the screen to this feature, right click on **Boundary** under the new **Feature EDM 2X Inside** feature that will have been inserted into the tree. Choose **Re/Select**.
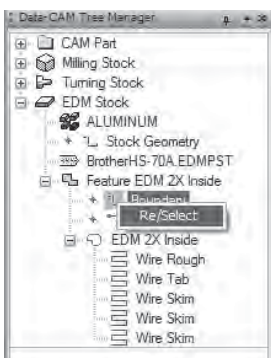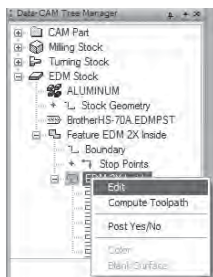

Hold the Shift key on the keyboard and click on the shape. The entire shape should highlight.

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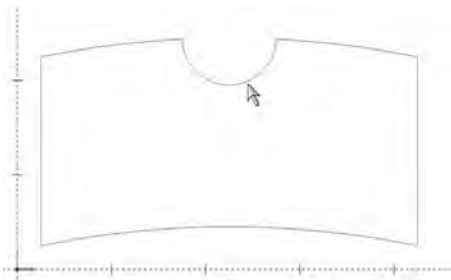
When it is highlighted, right click anywhere in the **Workspace** and click **OK**. The geometry will be associated with the feature in the tree, and the red dot next to **Boundary** under the feature in the tree should disappear.
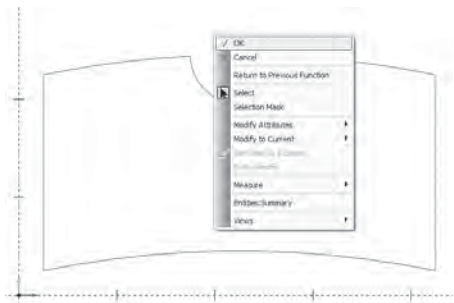
Step 4. Right click on **EDM 2X Inside** and choose **Edit**.

The **2 Axis Contour** dialog should appear as shown below.
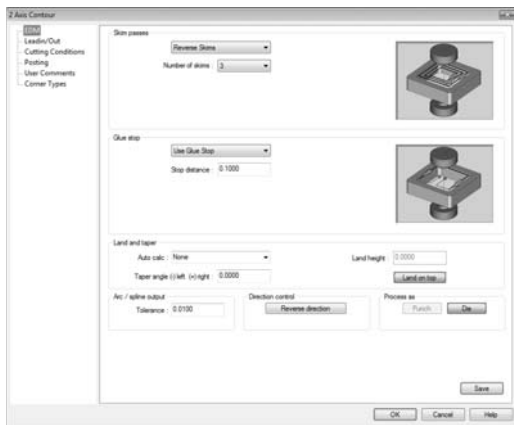
The first category on the left, **EDM**, contains most of the traditional parameters for a wire cut.

Set **Skim passes** to **Reverse Skims**, and set the **Number of Skims** to 2.

*As an aside, for coreless machining, set the **Glue stop** to **Coreless**. **Coreless** is only available on **2 Axis Inside** cuts. The help files contain a more detailed explanation of all of the available options.*
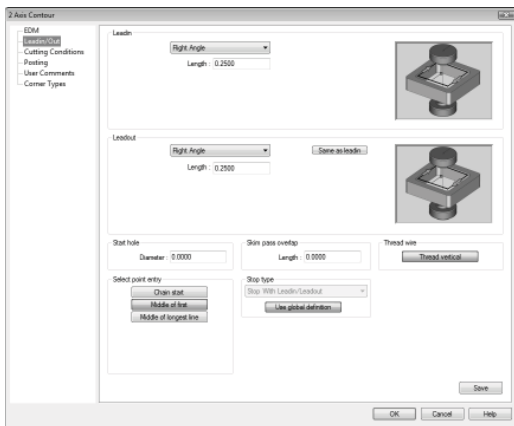
In the **Leadin/Out** category set the **Leadin** to **Perpendicular Blend**. Set the **Radius** to .125 and the **Length** to .25. For **Leadout**, click **Same as leadin** to copy all of the same values from **Leadin**.

Set the **Start hole**'s **Diameter** to .0625, set the **Select point entry** to **Middle of longest line** and leave the rest at default.
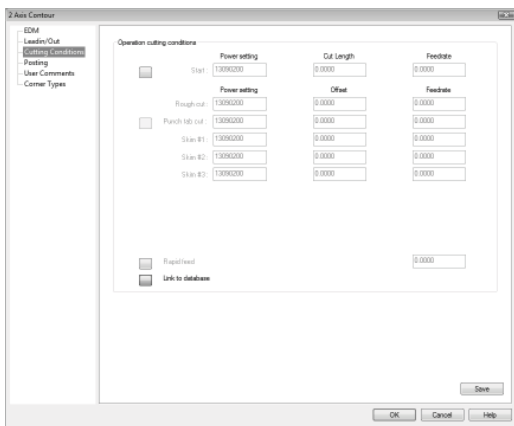
In **Cutting Conditions**, set the **Link to database** button to have BobCAD-CAM pull all of its conditions from the database associated with this machine. In this way, the user is not required to enter in any conditions by hand.
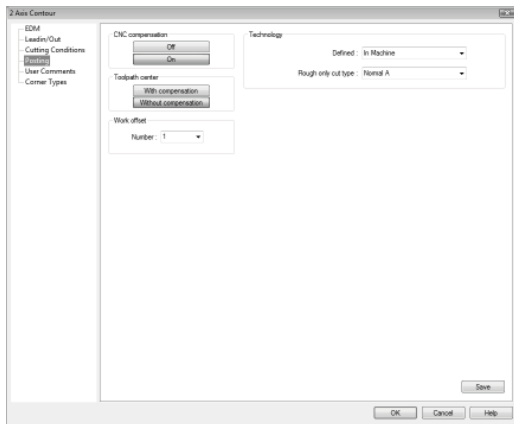
In **Posting**, set the **CNC compensation** to **Off** and the **Toolpath center** to **With compensation**. With it set this way, BobCAD-CAM will not post the G41/42 cutter compensation codes and will offset the wire path directly from the geometry instead.

In **User Comments**, the programmer is permitted to insert any special comments that the operator may need to see in the program. As this example isn't going to be run by an operator, we'll leave them blank.
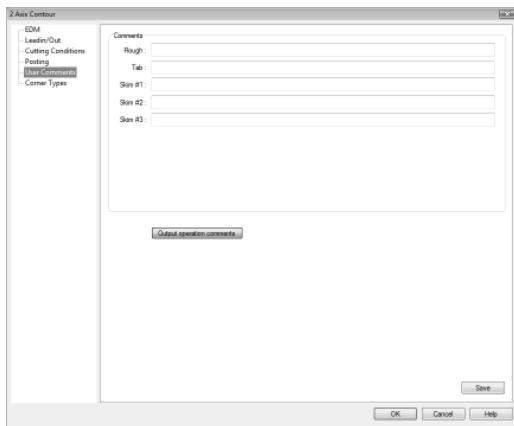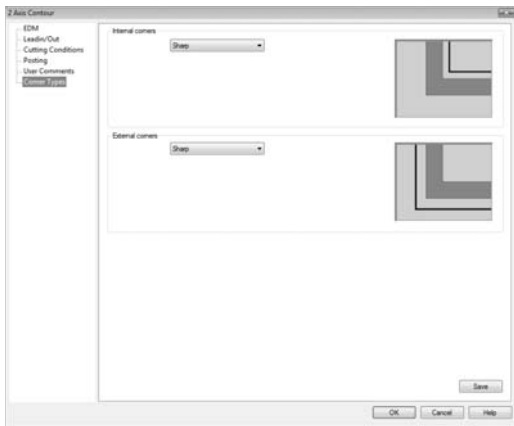
In the last category, **Corner Types**, set the **Internal corners** to **Bisect Line** to relieve the corners of the inside of the die and to allow for a properly square punch. Leave the values at their defaults.

External corners can be left at **Sharp** as there are no external corners in the part.

To finish up, click **OK**.

BobCAD-CAM will automatically compute all of the toolpath for the feature and display it in the **Workspace**. The drawing should now resemble the illustration below.
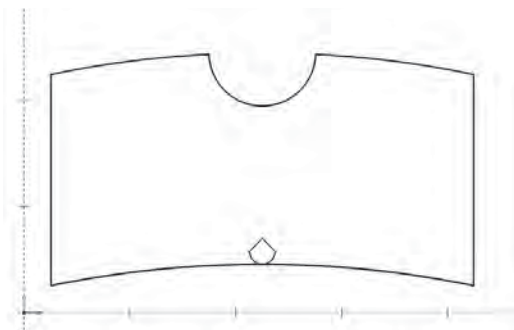
Step 4. The final step is simply to generate the code for the part.

Expand the **CAM Part** item at the top of the tree and right click on **EDM Tools**. Choose **Post** from the menu.

The 2 Axis example part will be complete.

BobCAD-CAM V23

Wire EDM

Chapter 5
4 Axis Wire Features

BobCAD-CAM can machine parts with wildly varying part topologies. There are 2 examples in this chapter, one having a simple constant taper and one with two completely different shapes to illustrate how to cut parts requiring all 4 independent axes.

5.1 4 Axis Taper Example

A simple 4-axis wire shape in BobCAD-CAM can be as simple as two rectangles at differing Z heights, as in this example, or more complex as in the example in **Section 5.2**.

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
- Hold the CTRL key down and hit N on the keyboard to start a new drawing.

Step 2: Click on **Other** from the main menu, and then click on **Rectangle**. Enter in these values as in the illustration below:
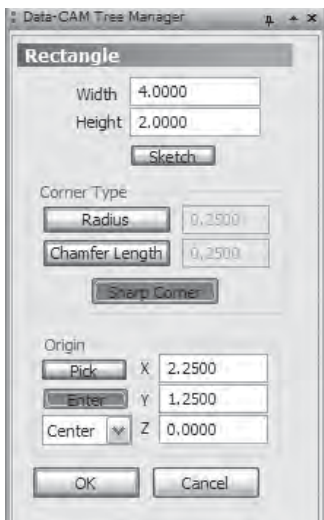
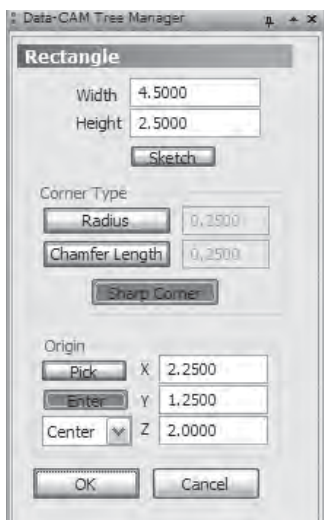**Width**: 4.00
Height: 2.00

Be sure **Sketch** is not set. Under **Corner Type**, press **Sharp Corner**. Under **Origin**, click **Enter**, and choose **Center** from the drop-down list if it isn't already set. In the number boxes, set these values:

X: 2.25
Y: 1.25
Z: 0

Click **OK** and the rectangle will appear in the **Workspace**.

Step 3: Continue in the rectangle function, but change the numbers as below:

Width: 4.5
Height: 2.5

Be sure **Sketch** is not set. Under **Corner Type**, press **Sharp Corner**. Under **Origin**, click **Enter**, and choose **Center** from the drop-down list if it isn't already set. In the number boxes, set these values:

X: 2.25
Y: 1.25
Z: 2

Click **OK** and the second rectangle will also appear in the **Workspace**.

5.2 4 Axis Taper CAM Example

This example is designed to follow directly after the drawing example in section **5.1**.

Step 1. Click on the **CAM** tab in the **Data-CAM Tree Manager**. Right click on **EDM Stock** and choose **Edit**.
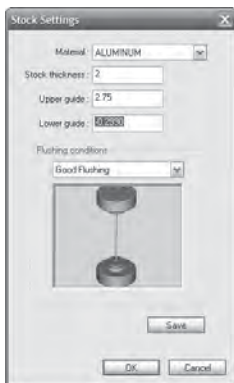
In the **Stock Settings** box that appears, set the **Stock Thickness** to 2" to correspond to the 2" height intended in the drawing, and set the **Upper guide** setting to somewhat above that. In this example, 2.75" is assumed. Click **OK** on this box.

Step 2. Right click on **EDM Stock** again and hover the cusor over **EDM**. From the menu that will fly out, choose **4 Axis Outside**.
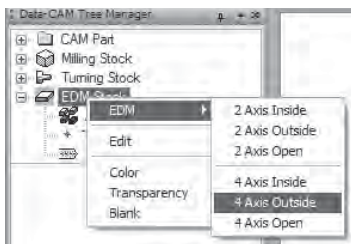

A new **Feature EDM 4 Axis Outside** feature will be inserted in the tree. Under the feature, right click on **Boundary** and choose **Re/Select**.

Hold the SHIFT key on the keyboard and click on the bottom line of the inner rectangle somewhere in between the middle of the line and the left corner as shown on the next page. The entire rectangle should highlight. Do the same thing for the outer shape.
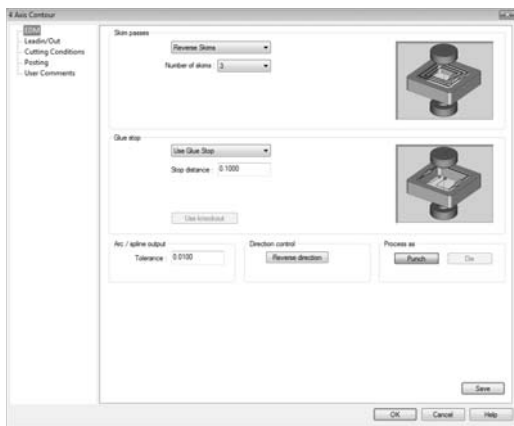
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When both shapes are highlighted, right click anywhere in the **Workspace** and choose **OK** from the menu that will appear.

**Step 3**. Right click on **EDM 4X Outside** and choose **Edit**.
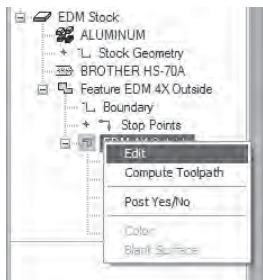
A dialog box with a series of option categories on the left will appear.

In the first category, **EDM**, choose **Reverse Skims** to cause the machine to reverse direction on the skim passes. Choose **1** for **Number of Skims**.

Use a .1 distance for **Glue stop**.

In **Leadin/Out**, set the **Leadin** to **Perpendicular** with a **Length** of .25. For **Leadout**, use an **Arc** with a .125 **Radius**.
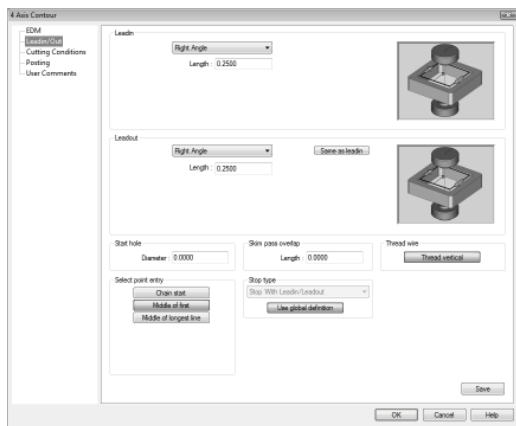

Give a .01 **Length** in **Skim pass overlap**, and set the **Select point entry** to **Middle of first**.
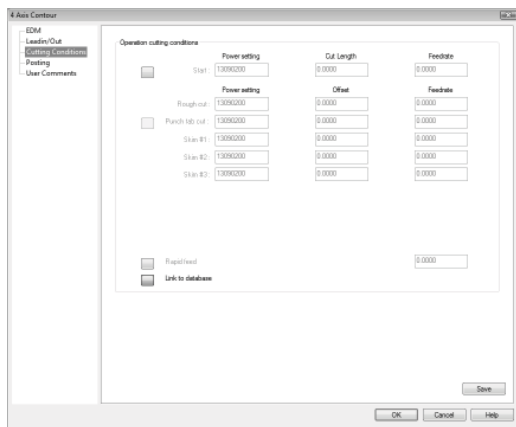
Under the **Cutting Conditions** category, we'd like BobCAD-CAM to pull the conditions directly from the built-in conditions database, so leave all of the parameters at their default values.
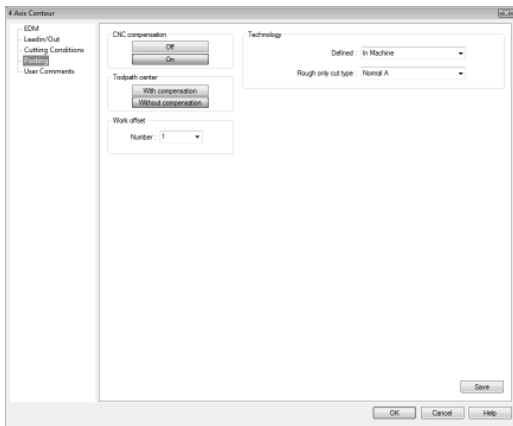
The **Posting** category contains options for compensation, overriding the wire speed and tension, and the flushing parameters. We'll leave these settings at the defaults for this sample.
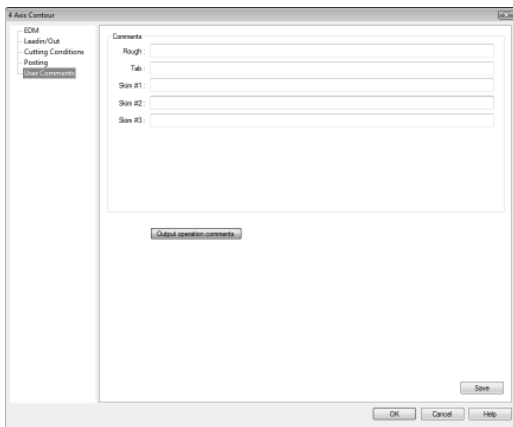

The last category, **User Comments**, allows the programmer to insert any special comments that the operator may need to see in the program. As this example isn't going to be run by an operator, we'll leave them blank.

Click **OK** to finish setting up the operation. BobCAD-CAM will automatically compute the wire path and display it in the **Workspace**.

The computed wire path should look like the illustration below:
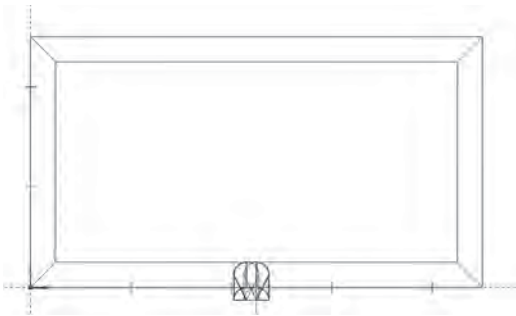
If the view is rotated to show the path in 3D and if the stock settings were correct, the path will appear as below:
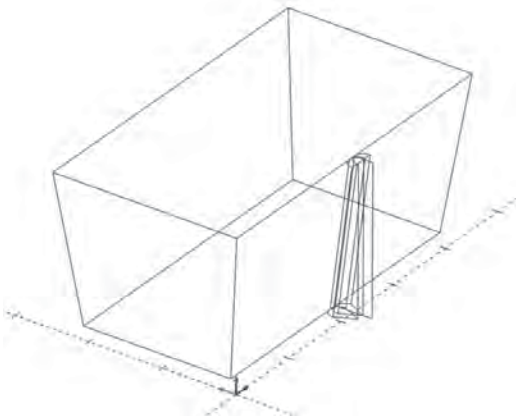
Step 4. The final step is simply to generate the code for the part.

Expand the **CAM Part** item at the top of the tree and right click on **EDM Tools**. Choose **Post** from the menu.

This 4 Axis part is complete.

The drawing given here is fairly simple; the complexity in this particular example doesn't come into play until it is machined. The key is that the entities on the top and bottom shapes are, unlike the previous example, mismatched and will induce a twist in the wire path.

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
- Hold the CTRL key down and hit N on the keyboard to start a new drawing.

Step 2: Click on **Other** from the main menu, and then click on **Rectangle**. Enter in these values as in the illustration below:
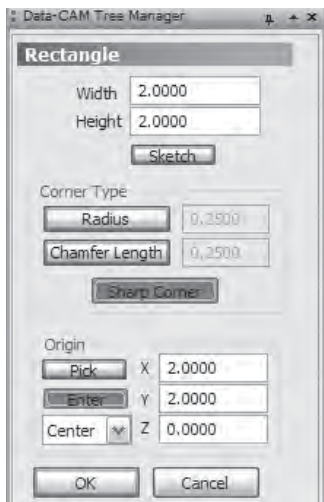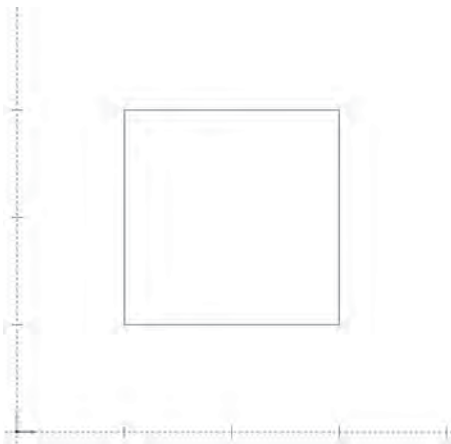

Width: 2.0
Height: 2.0

Be sure **Sketch** is not set. Under **Corner Type**, press **Sharp Corner**. Under **Origin**, click **Enter**, and choose **Center** from the drop-down list if it isn't already set. In the number boxes, set these values:

X: 2.0
Y: 2.0
Z: 0

Click **OK** and the rectangle will appear in the **Workspace** as on the next page.

Step 3: Click on **Arcs** in the main menu and then on **Coordinates**. As shown on the next page, enter these values into the dialog in the **Data-CAM Tree Manager**:
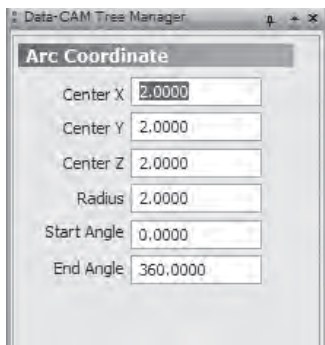
X: 2
Y: 2
Z: 2
Radius: 2
Start Angle: 0
End Angle: 360

Step 4: Choose **Arcs** again from the main menu, then **Snap**. In the dialog on the left side of the screen again, set the **Radius** to .5 and leave the **Start Angle** on 0 and the **End Angle** on 360 then click **OK**. This is not strictly necessary, but when the breaking function is used on them it will be easier to select the sections that don't belong in the final drawing.

Click on the left side of the large circle, and again on the right. The smaller circles will be drawn as the illustration on the next page.
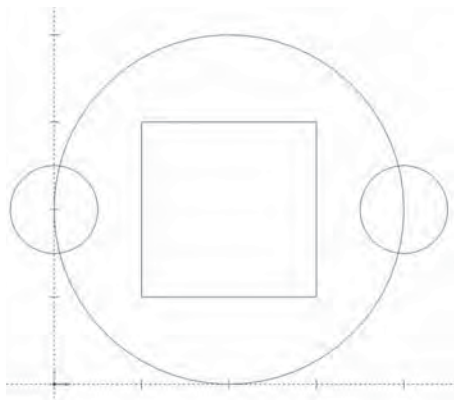
Step 5: Click on **Utilities** in the main menu and hover the mouse cursor over **Break**. When the fly-out menu appears, choose **Many**.

Click the mouse somewhere off the corners of the drawing as shown below. Press the mouse button and drag the mouse to the opposite corner as in the illustration below. Right-click anywhere in the **Workspace** and click **OK** in the menu and BobCAD-CAM will break every entity on the drawing where it crosses or intersects any other entity.
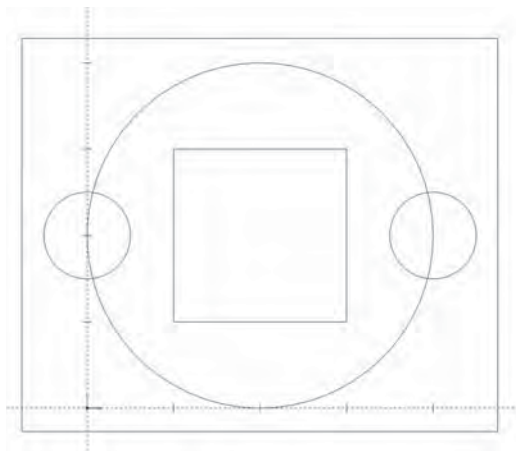
This makes it simple to select and delete anything that is not required for the finished drawing.

Step 6: Click on the **Selection Mode** icon ▶ on the toolbar. Click on every entity on the drawing that is not required for the shape.

Click the trash can icon 🗑 on the toolbar or press the Delete key on the keyboard. The highlighted entities will be deleted from the drawing and the remaining entities should look like the illustration below.
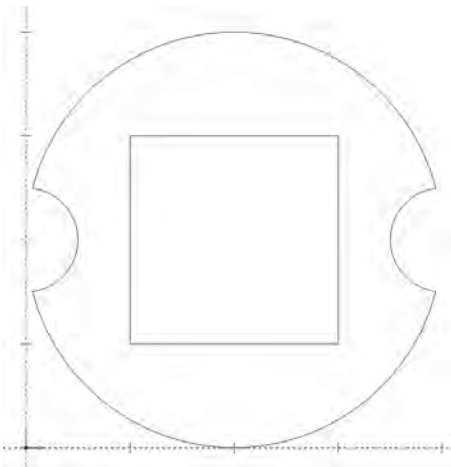
5.4 4 Axis Independent CAM Example

This example and the one following illustrate first a generic 4 Axis independent program and then the use of Match Points to correct for mismatched geometry. Note that most 4 Axis geometry will not require the use of match points if the top and bottom shapes match in terms of number of entities. Since the drawing in Chapter 4.3 was deliberately mismatched, these examples will illustrate how to get a good part from bad geometry.

If the user wishes to repeat this first example with the cleaner drawing produced in **Chapter 4.2**, the toolpath will not require match points to produce a good part.

Step 1. Follow the drawing example in section 5.3 first before proceeding with this example so that it will be easier to follow the steps given here.

Step 2. In order for the NC code to be properly posted and the correct cutting conditions to be applied to the program, the stock will need to be set up prior to producing the toolpath. The stock used in the EDM portion of the CAM tree will apply to the entire Wire EDM part.

Right click on **EDM Stock** and choose **Edit**.
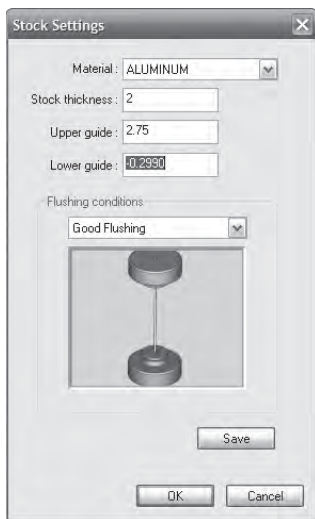
In the box that appears, change the **Stock thickness** to 2" and adjust the upper guide for the wire slightly above that to 2.75". Normally in real usage these numbers will need to be accurate, but for the purposes of this example it will serve that the **Upper guide** simply be above the top of the stock. All stock definitions in the EDM portion of the CAM tree begin with the bottom at Z 0.

Click **OK** at the bottom of the box.

Step 3. Right click on **EDM Stock** again and hover the cursor over **EDM**. From the fly-out menu, click on **4 Axis Inside**. A new **Feature EDM 4X Inside** will be added to the tree.
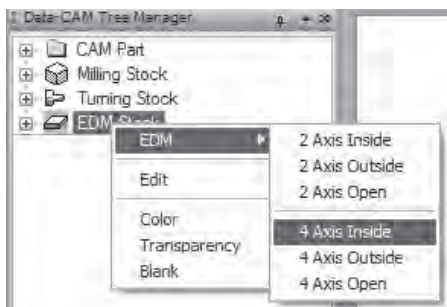
Step 4. Right click on **Boundary** in the feature and choose **Re/Select**.
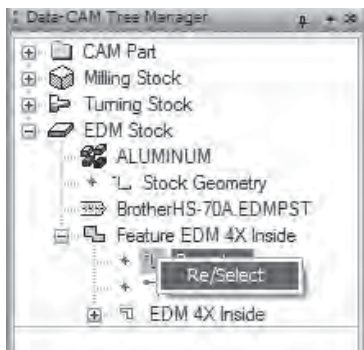
In the **Workspace**, hold the Shift key down on the keyboard and click on the left side of the square near the top, then do the same in the position shown for the larger shape.
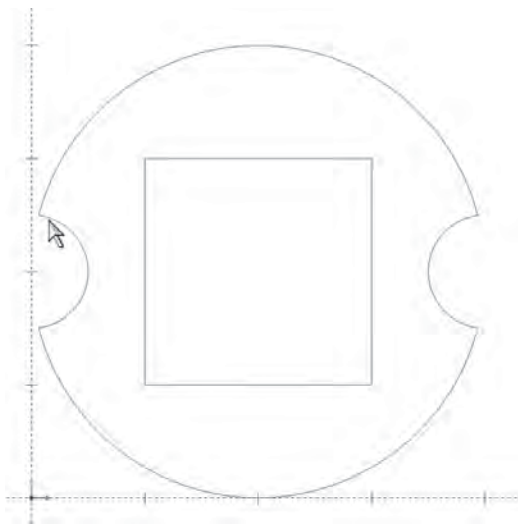
Right click anywhere in the workspace and choose **OK** and BobCAD-CAM will assign the selected geometry to the feature in the tree.
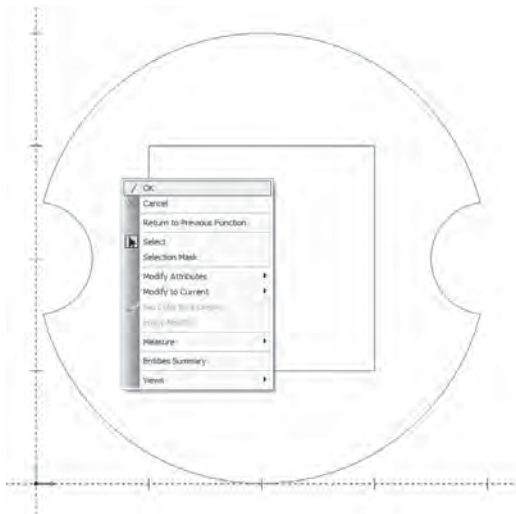
Step 5. Right click on **EDM 4X Inside** and choose **Edit**. Fill in each dialog as shown below.
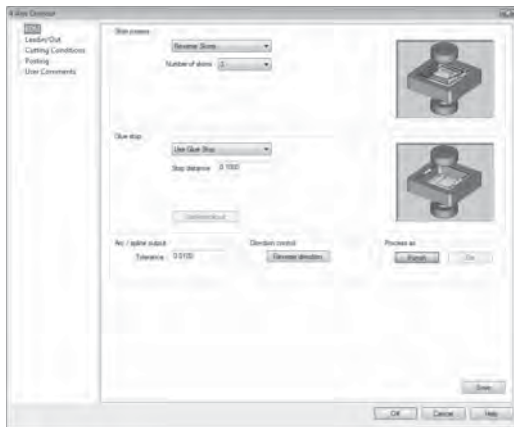
In the first category to the left, **EDM**, set **Skim passes** to **Reverse Skims** and set the **Number of skims** to 3.

Set the **Glue stop** parameter to 0.1" in the **Stop distance** field.
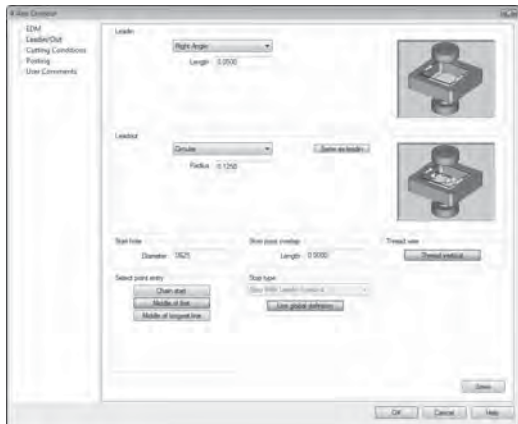
For **Leadin/Out**, set the **Leadin** to **Perpendicular** and set the **Length** to 0.05".

Set the **Leadout** to **Arc** and set its **Radius** to 0.125".

Set **Diameter** in the **Start hole** section to 0.0625", and set the **Select point entry** to **Middle of first** and leave the rest at default.

In **Cutting Conditions**, leave **Link to database** set in order to take advantage of BobCAD-CAM's included cutting conditions files.
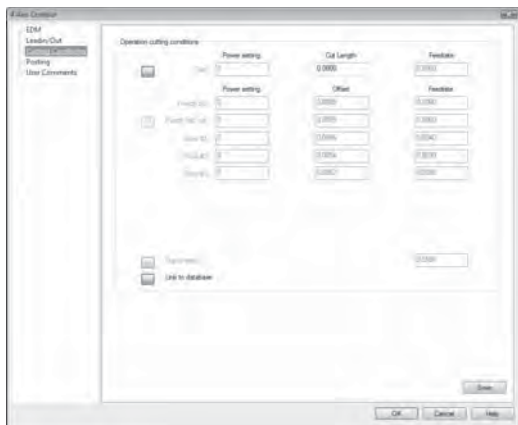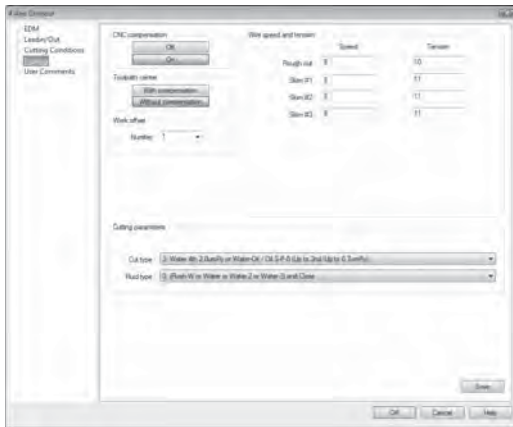
In **Posting**, also leave all of the settings at the defaults. BobCAD-CAM defaults to not offsetting the wire path, but instead uses the (typically) G41/42 codes on the machine to have the controller take care of the wire offset automatically.
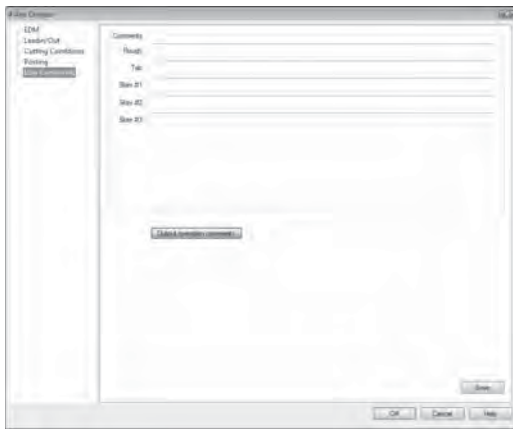
If any optional comments are desired for the individual operations in the feature, enter them here. These comments are for the machine operator to read, and have no actual effect on the NC program produced except for their presence in the code.

Click **OK** on the box. BobCAD-CAM will automatically compute the wire path and display it in the **Workspace**.
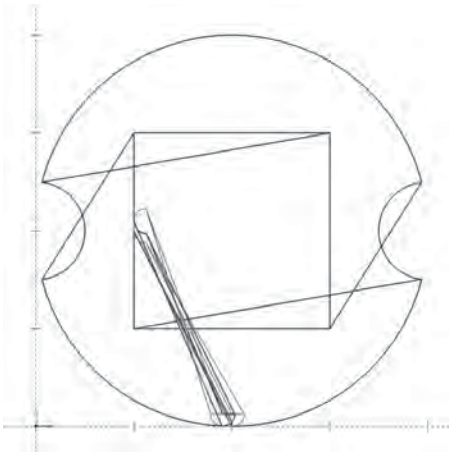
As mentioned at the beginning of the example, this wire path is wrong! Keep going; in section **6.4** steps will be taken to correct it.

As seen in this example, BobCAD-CAM does not automatically correct geometry for mismatched 4 axis shapes. The reason for this is that if it were to be automatically corrected, it would be impossible to induce a twist into the shape on purpose.

5.5 4 Axis Match Point Example

This example will illustrate how to remove an unintentional twist in the wire path.

The reason for the mismatch and thus the reason for the twist is that the arc on the left side is not infact a single arc, it is split into two parts. This leaves 4 entities on the bottom shape and 5 on the top. There are two ways of fixing this; either correct the arc on the top shape, or use a match point at the desired pivot point on the other shape. This exercise needs to illustrate the second method.

Step 1. Since it will be easier to see now rather than later, this is a good time to add the single match point that will be required for this part. Click on **Points** in the main menu and choose **End**.
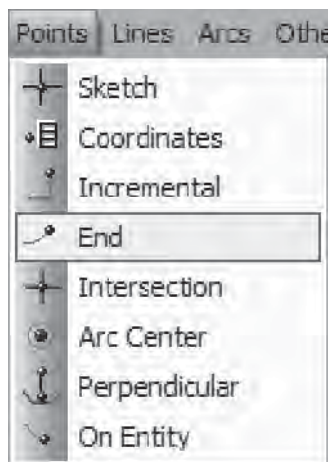
Click near the upper left corner of the square on the shape and a point will appear there.
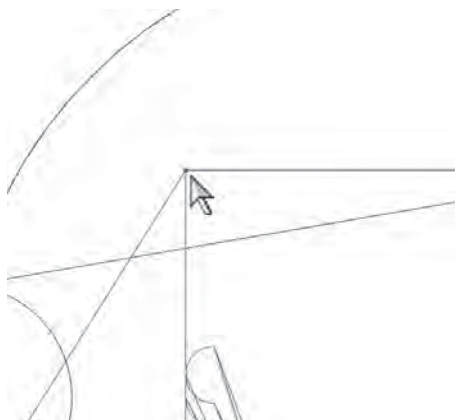
Step 2. Using match points requires that contour entities be created to use as the geometry for the boundary of the feature. To create contour entities, click on **Other** in the main menu, followed by **Contour**.
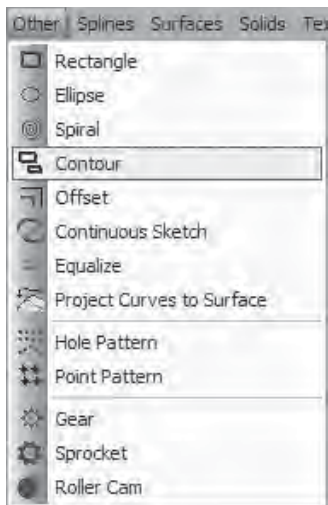
Start by holding the Shift key down on the keyboard and click near the top of the left side of the square just as was done in **Step 1**. Also hold Shift and click near the top of the inward arc on the larger shape.
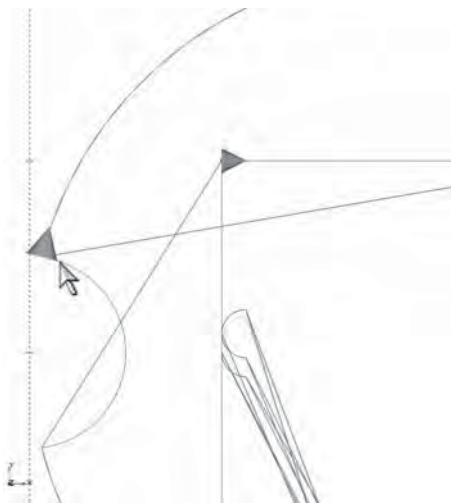
Right click in the **Workspace** and choose **OK**. The contours will appear as large "conic" arrow on the wireframe geometry as shown on the previous page.

Step 3. Re-assign the new contour entities and the match point to the **Boundary** in the feature.

Right click on **Boundary** under **Feature EDM 4X Inside** and choose **Re/ Select**. Click on the geometry in the **Workspace** as shown here.
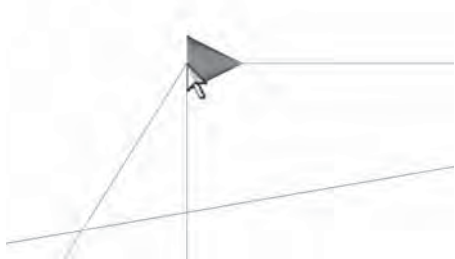
Click anywhere on the square first to get it highlighted. Next, click on the larger shape to add it to the selection in the same way. When both shapes are highlighted, click on the match point in the upper left corner of the square. It may be difficult to see, so zooming in may help with the selection.

Once the point has been selected, right-click in the **Workspace** and click **OK**.

Step 4. All that remains is to tell BobCAD-CAM to regenerate the toolpath using the new boundary geometry. Right-click on **EDM 4X Inside** and choose **Compute Toolpath** from the menu.
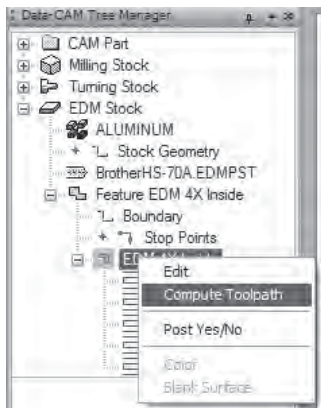
If the selection went well, the result should look like what is shown below:

The match point stops the guide where the point is and allows the other guide a moment to catch up, thus removing any unintentional twist from the part.
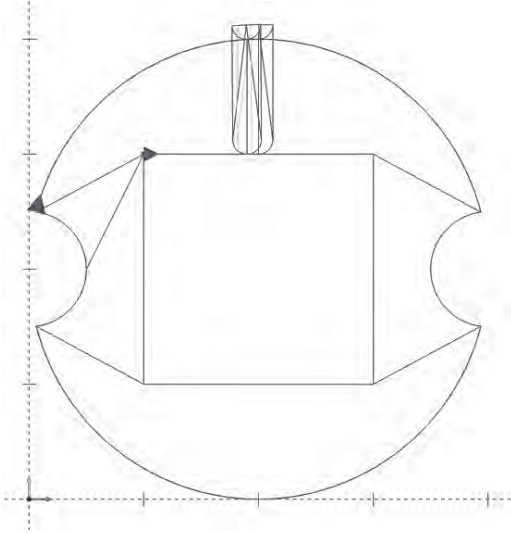
Step 5. To generate the **4 Axis NC code**, expand the **CAM Part** item at the top of the tree and right click on **EDM Tools**. Choose **Post** from the menu.

The 4 Axis example part will be complete.

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Wire EDM

Chapter 6
The Cutting Conditions

BobCAD-CAM's Wire EDM package comes with complete, per-machine cutting conditions databases. Each database is stored as a file on your hard drive under the BobCAD-CAM installation folder, under EdmTechnology and then EDMCuttingConditions. The files can be edited using a database application or peprhaps more easily through the software itself.

6.1 Editing Databases Inside BobCAD-CAM

To look at the database associated with your machine through the software itself, expand **CAM Part** at the top of the CAM tree by clicking on the + button to its left first.

Right click on **EDM Tools** and choose **Current Machine**.
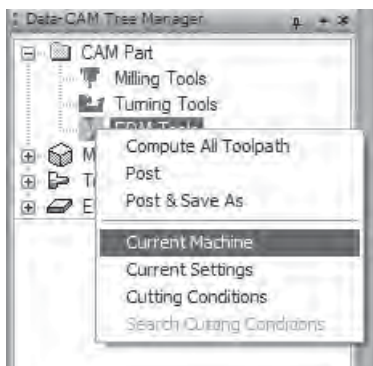
From the drop-down lists, choose the make and model of your machine and then click **OK**.
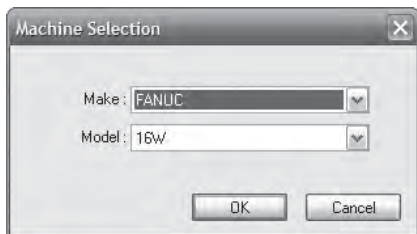
Right click **EDM Tools** once more and choose **Cutting Conditions**.
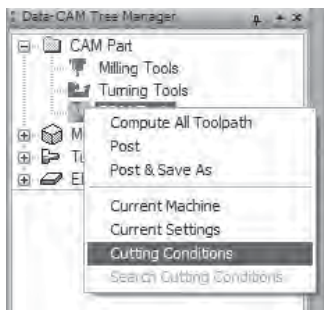
BobCAD-CAM will display the **Edit Cutting Conditions Database** dialog box. At the top of the box will be listed the current **Material**, **Wire Diameter**, and **Stock Thickness**.
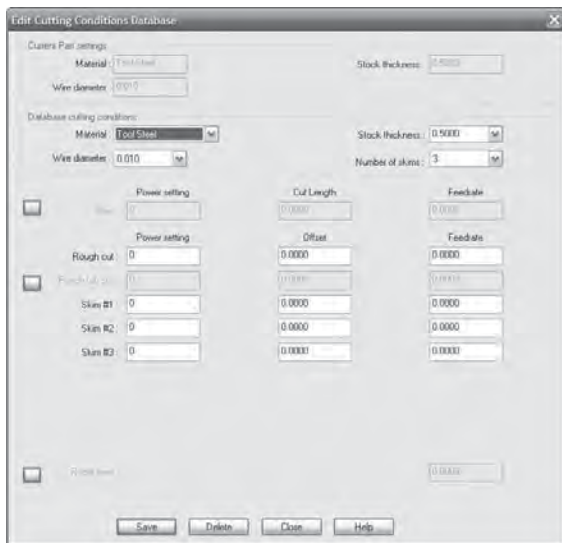
Below that, it is possible to change the **Power Setting**, **Cut Length** and **Feedrate** field for the start of the cut as well as for the rough and skim passes. Choose the **Material** from the list as well as the **Stock Thickness**, **Wire Diameter**, and the **Number of Skims** to edit. If any of the fields below need to be changed, change them and click **Save**. BobCAD-CAM will update the database file for this machine. When finished, click **Close**.

BobCAD-CAM stores the cutting condition in Microsoft Access formatted database files. For the most part, they carry the .EDMCC extension with the exception of a few specialized database files for use with Sodick machines that are labeled with .SODCC instead.

To edit one of these using an external database editor, first make a copy of the database file and rename the file extension to .mdb to match the default Access extension.
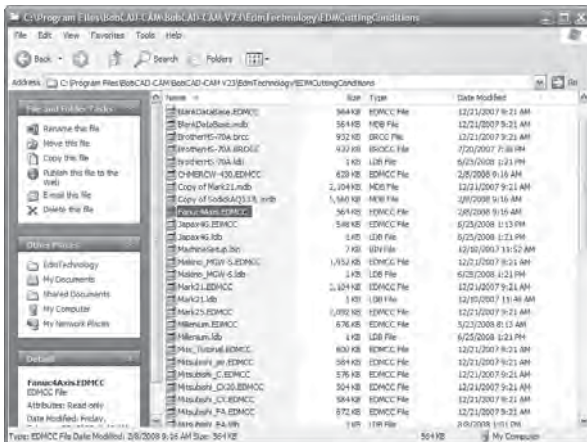
Open the database file with the database software of choice, and choose either the "open" or "connect" option provided in it. Make sure the database type is set to Microsoft Access, and the copied file should open easily.

The tables inside it are labeled and the user will find it simple to query the database as if it had been created for any other purpose. When finished editing the fields, save the database and rename it back to its original extension (either .EDMCC or .SODCC), being careful not to overwrite the original file.

6.3 Changing the Database File

To try out the newly edited database, expand **CAM Part** at the top of the CAM tree by clicking on the + button first.

Right click on **EDM Tools** and choose **Current Machine**. Set the make and model to the machine that the conditions database is for.

Right click **EDM Tools** once more and choose **Current Settings**.
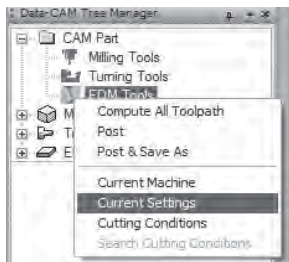
Choose the **Machine Setup** category on the left, then click on **CTC Database** (**C**utting **T**echnology **C**onditions **Database**) in the right pane.

Choose the newly edited file, then click **OK** in the dialog.
The new database file will become associated with this machine until it it is changed by a user at a later date.

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Wire EDM
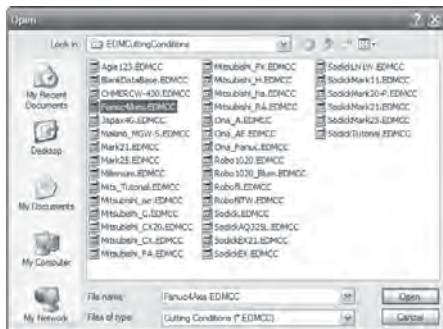
Chapter 7
Getting Code to
the Machine

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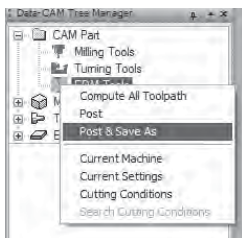
• **Save As** from the **Layer-UCS-Post Manager** – After the program has been posted, right-click in the **Layer-UCS-Post Manager** where the code is displayed and choose **Save As**. The **Save As** dialog will appear.

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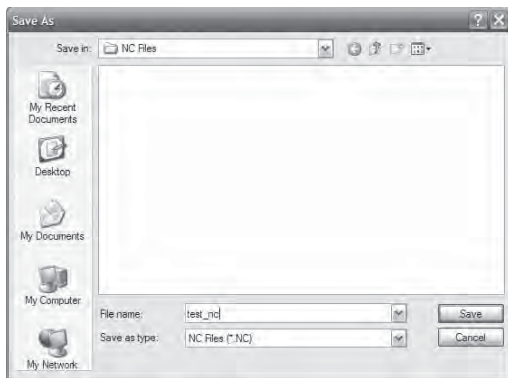
then burn the CD from there instead of attempting to save the file directly to the device.

7.2 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.
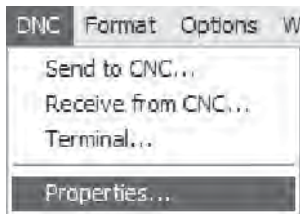
The system will display a series of 5 dialog boxes:

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.
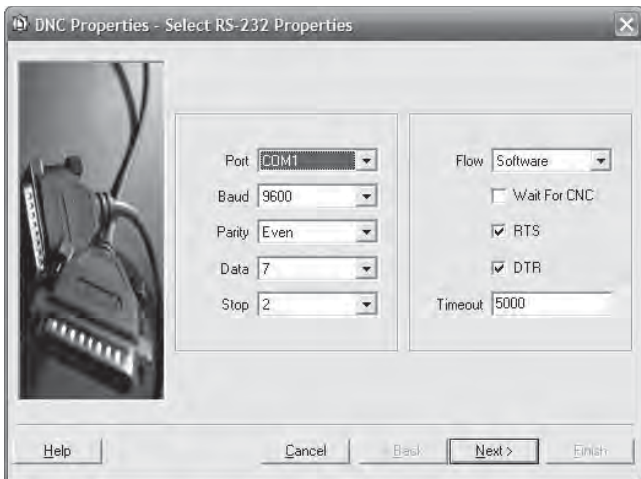
2. **DNC String and Constant Options** – Use this dialog box to set up any special leader or trailer characters the control may require.
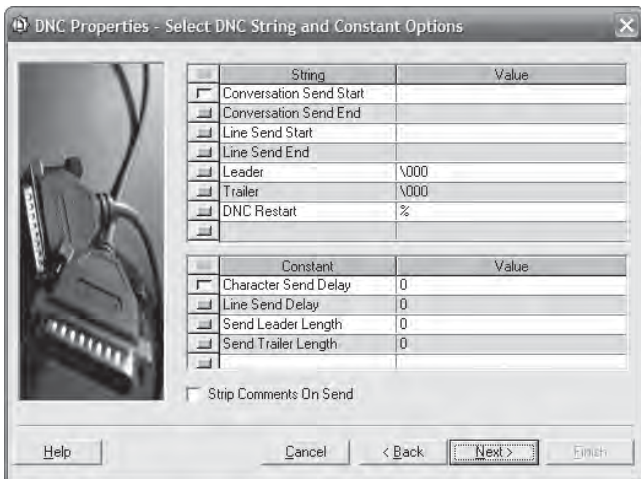
3. **Send Translation Options** – Occasionally a rare control will need a character normally output to be changed into something else entirely in order to read the transferred program. Normally no changes will be required; check the controller's documentation to be sure.

DNC Properties - Select Send Translation Options

	Decimal	Hexadecimal	Character	Name	Action	Value
	000	00		NUL	None	
	001	01	☺	SOH	None	
	002	02	☻	STX	None	
	003	03	♥	ETX	None	
	004	04	♦	EOT	None	
	005	05	♣	ENQ	None	
	006	06	♠	ACK	None	
	007	07	•	BEL	None	
	008	08	◘	BS	None	
	009	09		TAB	None	
	010	0A		LF	None	
	011	0B	♂	VT	None	
	012	0C	♀	FF	None	
	013	0D		CR	None	
	014	0E	♫	SO	None	
	015	0F	☼	SI	None	
	016	10	►	DLE	None	

Help   Cancel   < Back   Next >   Finish

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.

DNC Properties - Select Receive Translation Options

	Decimal	Hexadecimal	Character	Name	Action	Value
	000	00		NUL	Remove	
	001	01	☺	SOH	None	
	002	02	☻	STX	None	
	003	03	♥	ETX	None	
	004	04	♦	EOT	None	
	005	05	♣	ENQ	None	
	006	06	♠	ACK	None	
	007	07	•	BEL	None	
	008	08	◘	BS	None	
	009	09		TAB	None	
	010	0A		LF	None	
	011	0B	♂	VT	None	
	012	0C	♀	FF	None	
	013	0D		CR	None	
	014	0E	♫	SO	None	
	015	0F	☼	SI	None	
	016	10	►	DLE	None	

Help   Cancel   < Back   Next >   Finish

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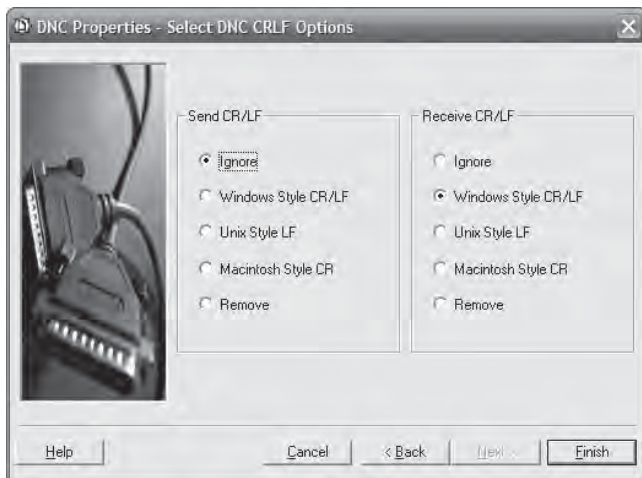
Click **Finish** when the settings are correct. The system will remember them for the next time a program is transferred.

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.

```
IBM PC or PC/XT    25 Pin Serial Cable       MACHINE End
          Pin No.                        Pin No.
F.G.      (1)        × ──────────────── (1)       F.G.
TXD       (2) ─────────┐    ┌────────── (2)       TXD
RXD       (3) ─────────┴────┴────────── (3)       RXD
RTS       (4)                         ┌ (4)       RTS
CTS       (5)                         └ (5)       CTS
DSR       (6)                         ┌ (6)       DSR
S.G.      (7) ──────────────────────── (7)       S.G.
DCD       (8)                         ├ (8)       DCD
DTR       (20)                        └ (20)      DTR
(Female)                                          (Male)
```

```
IBM PC or PC/XT    25 Pin Serial Cable       MACHINE End
          Pin No.                        Pin No.
F.G.      (1)        × ──────────────── (1)       F.G.
TXD       (2) ─────────┐    ┌────────── (2)       TXD
RXD       (3) ─────────┴────┴────────── (3)       RXD
RTS       (4) ─────────┐    ┌────────── (4)       RTS
CTS       (5) ─────────┴────┴────────── (5)       CTS
DSR       (6)                         ┌ (6)       DSR
S.G.      (7) ──────────────────────── (7)       S.G.
DCD       (8)                         ├ (8)       DCD
DTR       (20)                        └ (20)      DTR
(Female)                                          (Male)
```
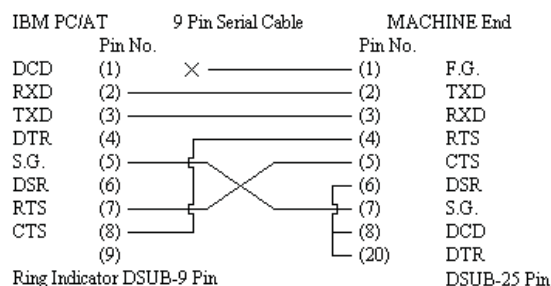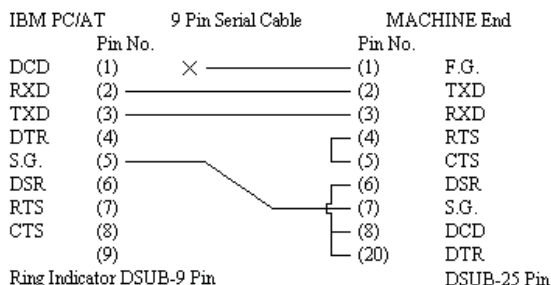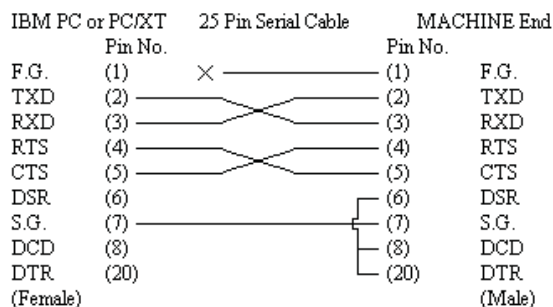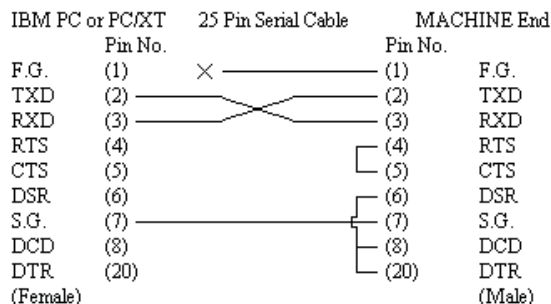
```
IBM PC/AT        9 Pin Serial Cable        MACHINE End
          Pin No.                        Pin No.
DCD       (1)        × ──────────────── (1)       F.G.
RXD       (2) ──────────────────────── (2)       TXD
TXD       (3) ──────────────────────── (3)       RXD
DTR       (4)                         ┌ (4)       RTS
S.G.      (5) ─────────┐              └ (5)       CTS
DSR       (6)          │              ┌ (6)       DSR
RTS       (7)          └────────────── (7)       S.G.
CTS       (8)                         ├ (8)       DCD
          (9)                         └ (20)      DTR
Ring Indicator DSUB-9 Pin                        DSUB-25 Pin
```

```
IBM PC/AT        9 Pin Serial Cable        MACHINE End
          Pin No.                        Pin No.
DCD       (1)        × ──────────────── (1)       F.G.
RXD       (2) ──────────────────────── (2)       TXD
TXD       (3) ──────────────────────── (3)       RXD
DTR       (4) ─────────┐    ┌────────── (4)       RTS
S.G.      (5) ──────┐  └────┼────────── (5)       CTS
DSR       (6)       │       └────────┐ (6)       DSR
RTS       (7) ──────┴────────────────┤ (7)       S.G.
CTS       (8) ───────────────────────┘ (8)       DCD
          (9)                         └ (20)      DTR
Ring Indicator DSUB-9 Pin                        DSUB-25 Pin
```

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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.
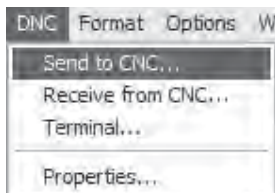
7.3 Transferring by RS-232 (Serial) Link

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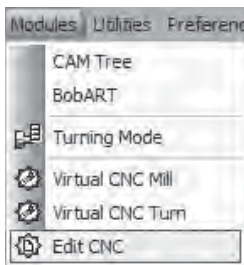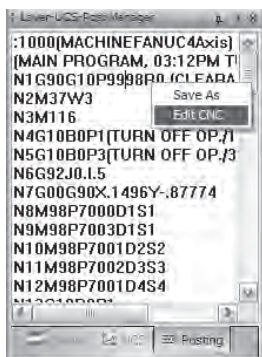
**Step 2**: Choose **Send to CNC…** from the menu

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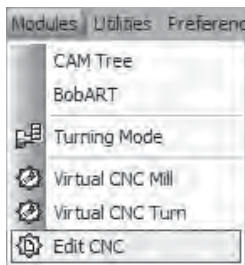
Step 3: The CNC Editor will automatically begin the transfer. The large box near the top of the dialog will display the code as it is received. The dialog will disappear when the transfer is complete.