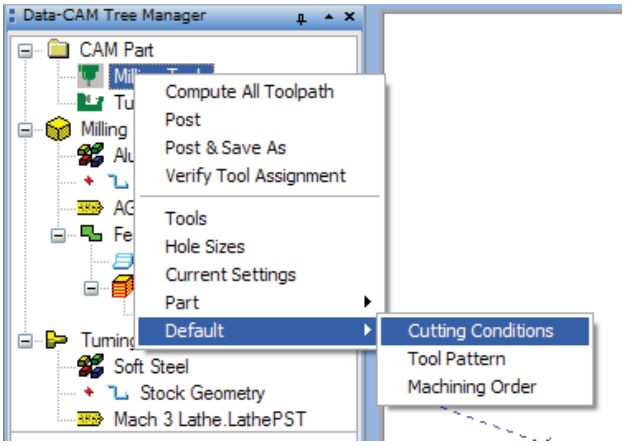
FAQ #3: How do I set my cutting speeds and feeds?

Disclaimer: The default settings given in BobCAD-CAM's tooling tables are extremely conservative so to avoid any tooling mishaps that may occur if speeds are set too high by default. However, it is up to the user to increase these speeds to a more respectable real-world machining level. BobCAD-CAM, Inc. assumes no responsibility, material or otherwise, for any damages resulting from the use or misuse of the information contained herein.

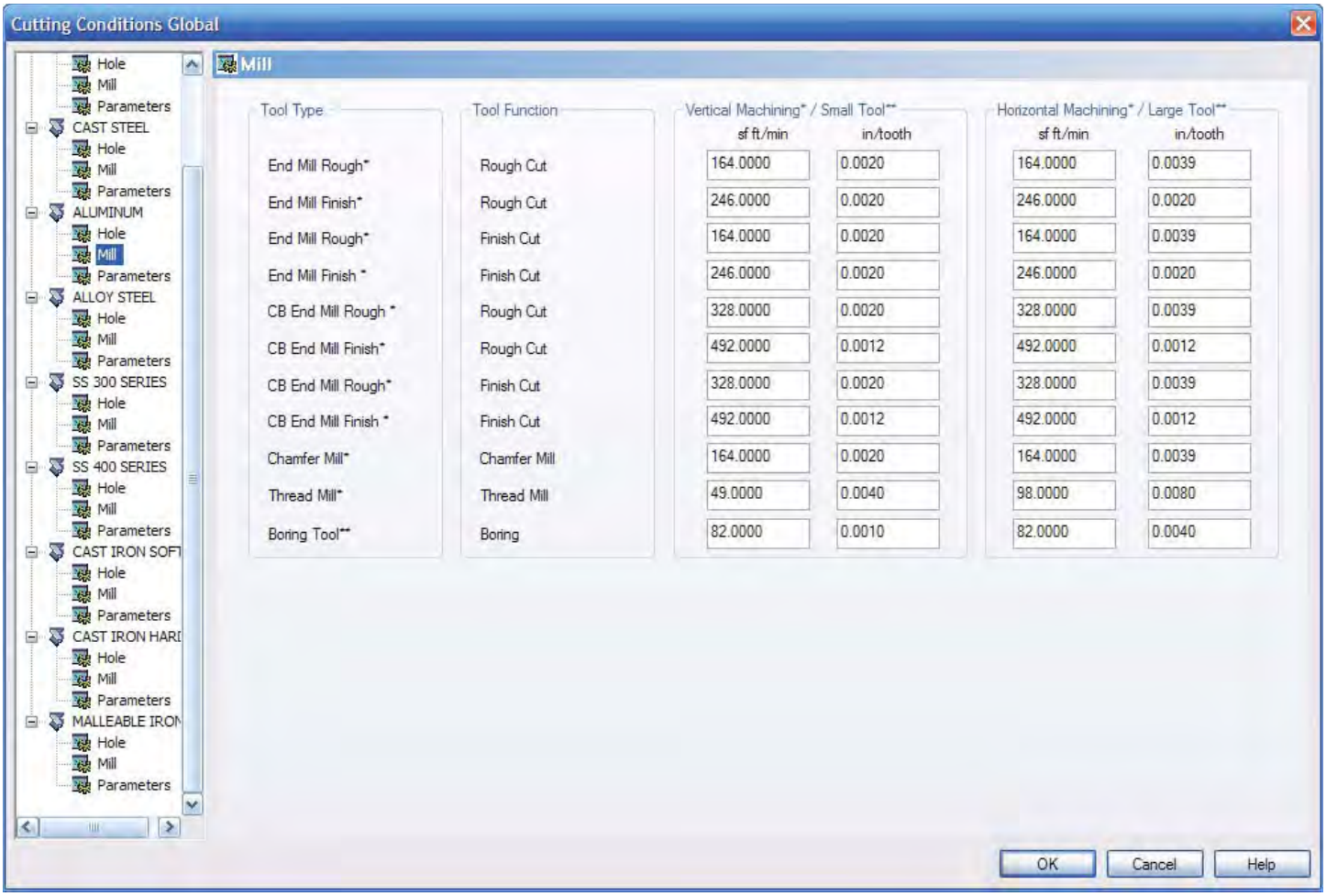
Step 1: Begin with an empty drawing by using any one of these 3 methods:

- 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.



Accessing the default cutting conditions tables inside BobCAD-CAM

Click on the + icon next to **CAM Part**, then right-click on **Milling Tools** (or **Turning Tools**, as appropriate). From the menu that will appear, go to **Default** and pick **Cutting Conditions** as shown. A dialog box will appear as below. The reason for choosing **Default** and not **Part** is so that we don't have to go back and repeat this procedure for each part. We want for the tooling feed settings to be kept for future use in the system, so we have to modify them globally rather than only for the currently open part. This is also why we're not going to draw anything for this part - we want the settings to apply for all future drawings.

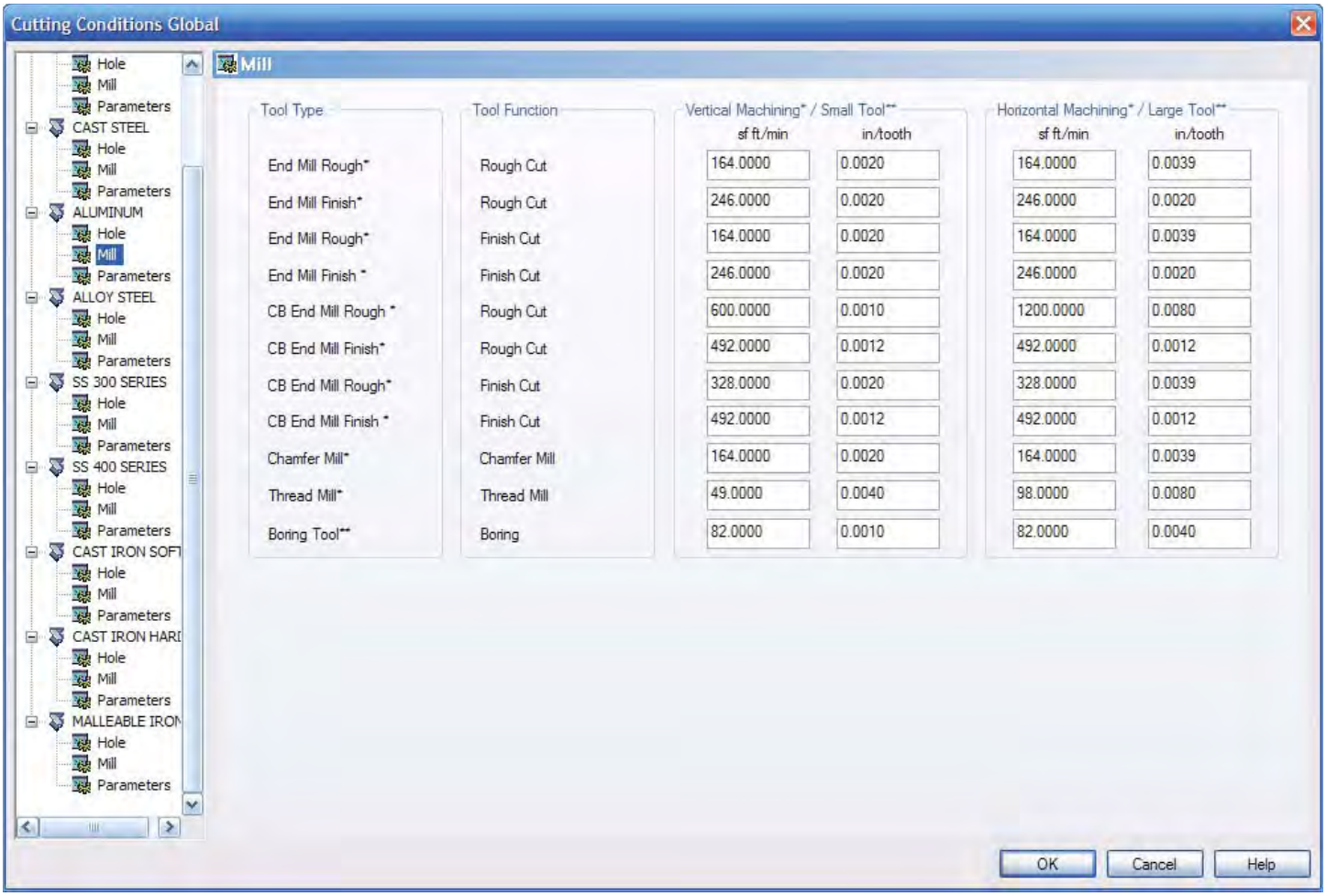


The default cutting conditions table for milling aluminum inside BobCAD-CAM

Step 2: Fill in the table with your tool maker's recommended settings.

What's shown here is the default table setup for milling in aluminum. Aluminum is a material given in every tooling manufacturer's feeds/speeds tables and is one of the most common materials to f nd in a machine shop so it's a good example to use here.

A good guide is to use the charts provided by the tooling maker. BobCAD's cutoff point between large and small tools is .150" in diameter, so tailor the numbers you use around that.



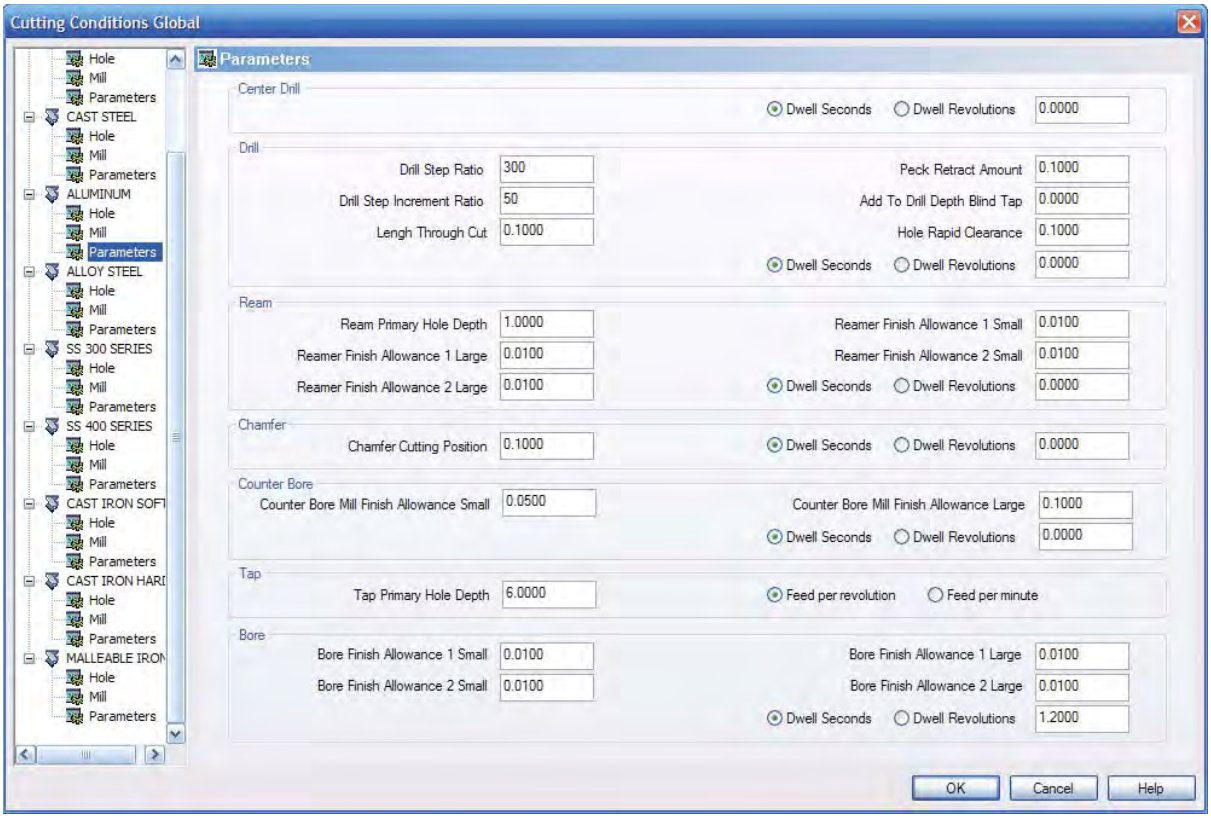
The default cutting conditions table for milling aluminum, changed to ref ect the parameters issued by a tool manufacturer

This illustration shows example low and high range settings for a Carbide End Mill Rough used in rough cuts in aluminum. The small tool setting is at the bottom of this tool manufacturer's recommendations for a 1/4" tool; the large tool setting is at the top range for a 1" tool.

Compare with the previous illustration. Each of the values should be f lled in similarly. It's only required to f ll in the settings that will actually be used for each operation and material on the left, but it's still recommended to f ll out the entire list.

When f nished, remember to click **OK** at the bottom of the box so that the settings will be saved. After entering in all this data, it would be a shame to click the wrong button and not save it!

When the table has been set up and the dialog no longer shows, close the f le by selecting **Close** under the **File** menu.



The default parameters table for aluminum in BobCAD-CAM

Also pay close attention to the Parameters page for each material. Some of the parameters will need some brief explanations:

Center Drill Frame:

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

Drill Frame:

- **Drill Step Ratio**: This parameter governs when the system will automatically apply a peck drilling cycle. By default the value is set to 300 percent. If the depth of the hole is three times greater than the diameter of the drill the system will automatically output a G73 or G83 depending on which option is selected.
- **Drill Step Increment Ratio**: For peck drill this determines the percentage of the diameter of the drill to use as the peck depth.
- **Length Through Cut**: This value is used to def ne the amount, or distance that the tool will travel through the material after it has reached the bottom of the stock in ‘through hole’ machining. This distance does not include the tool point. This value only has effect in hole making jobs, that have been set to ‘through hole’.
- **Drill Step Return**: For Peck Drill this determines the distance that the drill will retract in the hole to start the next peck rather then retracting to the clearance plane.
- **Add To Drill Depth Blind Tap**: This value is used to add an extra constant amount to all drill depths when doing blind hole tapping. The need for this value usually arises when a machine does not have rigid tapping, or when some older machines, in the process of reversing the direction make extra revolutions before stopping. This value acts as a safety so as not to break taps.

- **Stop Before Hole**: This value def nes the distance above the work piece at which the tool movement will switch from rapid feed rate to cutting feed rate. This value is used for all hole making functions (drilling, tapping etc).
- **Dwell Seconds**: When this option is selected the value in the input box will ref ect a specif c measure of time. This measure of time will be applied to the Drill operations that are output by the CAM System.
- **Dwell Revolutions**: When this option is selected the value in the input box will ref ect revolutions the tool must complete during the dwell.

Ream Frame:

- **Ream Primary Hole Depth**: This value is used to automatically calculate the drilling depth in reaming jobs with blind holes, when the entered drilling depth is less than or equal to the reaming depth. This value is used in place of the 'ineffective length" f eld in the assigned reamer tool if this tool parameter is set to 0.0. The addition drilling amount will be equal to the (reamer diameter * entered value).
- **Reamer Finish Allowance 1 Large**: The system will allow the user to use one or two f nishing steps between the drill and reamer operations of the reamer job. These are normally performed with end mills. This f eld is the value used for the f rst step, of a two step f nishing process if the reamer O.D. is less than or equal to 0.118. The endmill that will automatically be selected will have a diameter of (Reamer diam - Reamer f nish allowance one large).
- **Reamer Finish Allowance 2 Large**: The system will allow the user to use one or two f nishing steps between the drill and reamer operations of the reamer job. These are normally performed with end mills. This f eld is the value used the second step of a two step f nishing process if the reamer O.D. is less than or equal to 0.118. The endmill that will automatically be selected will have a diameter of (Reamer diam - Reamer f nish allowance two large).
- **Reamer Finish Allowance 1 Small**: The system will allow the user to use one or two f nishing steps between the drill and reamer operations of the reamer job. These are normally performed with end mills. This f eld is the value used for the one step, or f rst step of a two step f nishing process if the reamer O.D. is less than or equal to 0.118. The endmill that will automatically be selected will have a diameter of (Reamer diam – Reamer f nish allowance one small).
- **Reamer Finish Allowance 2 Small**: The system will allow the user to use one or two f nishing steps between the drill and reamer operations of the reamer job. These are normally performed with end mills. This f eld is the value used for the second step of a two step f nishing process if the reamer O.D. is less than or equal to 0.118. The endmill that will automatically be selected will have a diameter of (Reamer diam - Reamer f nish allowance two small).
- **Dwell Seconds**: When this option is selected the value in the input box will ref ect a specif c measure of time. This measure of time will be applied to the Reaming Operations that are output by the CAM System.
- **Dwell Revolutions**: When this option is selected the value in the input box will ref ect revolutions the tool must complete during the dwell.

Chamfer Frame:

- **Chamfer Cutting Position**: This value def nes the default tool position for chamfer milling operations. The distance is used to determine which part of the angular cutting edge of the chamfer milling tool will be cutting the chamfer. The distance from the bottom of the chamfer, to the bottom of the tool is the value that is needed.
- **Dwell Seconds**: When this option is selected the value in the input box will ref ect a specif c measure of time. This measure of time will be applied to the Chamfering Operations that are output by the CAM System.
- **Dwell Revolutions**: When this option is selected the value in the input box will ref ect revolutions the tool must complete during the dwell.

Counter Bore Frame:

- **Counter Bore Mill Finish Allowance Small**: This field is used to help the system select an endmill that will be used to counter bore milling operation. If the counter bore diameter is less than or equal to 0.5, then the selected endmill will be (less than or equal to, counter bore diameter - counter bore mill finish allowance small).
- **Counter Bore Mill Finish Allowance Large**: This field is used to help the system select an endmill that will be used to counter bore milling operation. If the counter bore diameter is greater than 0.5, then the selected endmill will be (less than or equal to, counter bore diameter – counter bore mill finish allowance large).
- **Dwell Seconds**: When this option is selected the value in the input box will reflect a specific measure of time. This measure of time will be applied to the Counter Bore Operations that are output by the CAM System.
- **Dwell Revolutions**: When this option is selected the value in the input box will reflect revolutions the tool must complete during the dwell.

Tap Frame:

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

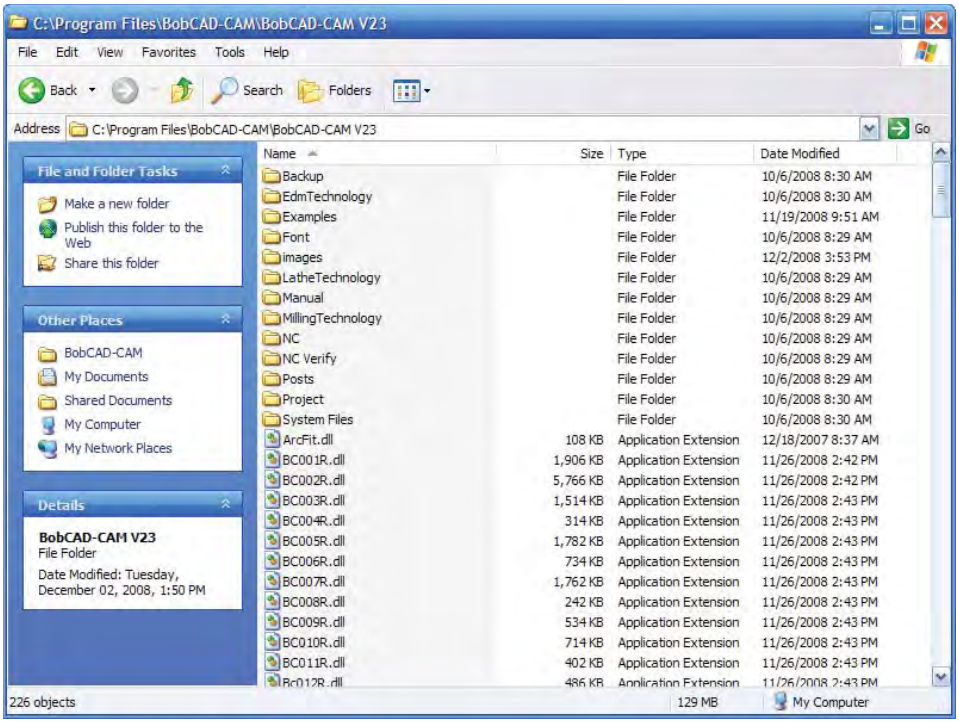
Bore Frame:

- **Bore Finish Allowance 1 Small**: The system will allow the user to use one or two finishing steps between the drilling and boring operations of the boring job. These are normally performed with end mills. This field is the value used for the one step, or first step of a two step finishing process if the bore O.D. is less than or equal to 0.118. The endmill that will automatically be selected will have a diameter of (Bore diam -Bore finish allowance one small).
- **Bore Finish Allowance 2 Small**: The system will allow the user to use one or two finishing steps between the drilling and boring operations of the boring job. These are normally performed with end mills. This field is the value used for the second step of a two step finishing process if the bore O.D. is less than or equal to 0.118. The endmill that will automatically be selected will have a diameter of (Bore diam - Bore finish allowance two small).
- **Bore Finish Allowance 1 Large**: The system will allow the user to use one or two finishing steps between the drilling, and reaming operations of the Boring job. These are normally performed with end mills. This field is the value used for the first step, of a two step finishing process if the bore O.D. is greater than 0.118. The endmill that will automatically be selected will have a diameter of (Bore diam - Bore finish allowance one large).
- **Bore Finish Allowance 2 Large**: The system will allow the user to use one or two finishing steps between the drilling, and boring operations of the boring job. These are normally performed with end mills. This field is the value used the second step of a two step finishing process if the bore O.D. is greater than 0.118. The endmill that will automatically be selected will have a diameter of (Bore diam - Bore finish allowance two large).
- **Dwell Seconds**: When this option is selected the value in the input box will reflect a specific measure of time. This measure of time will be applied to the Bore Operations that are output by the CAM System.
- **Dwell Revolutions**: When this option is selected the value in the input box will reflect revolutions the tool must complete during the dwell.

Step 3: Saving the cutting conditions f le

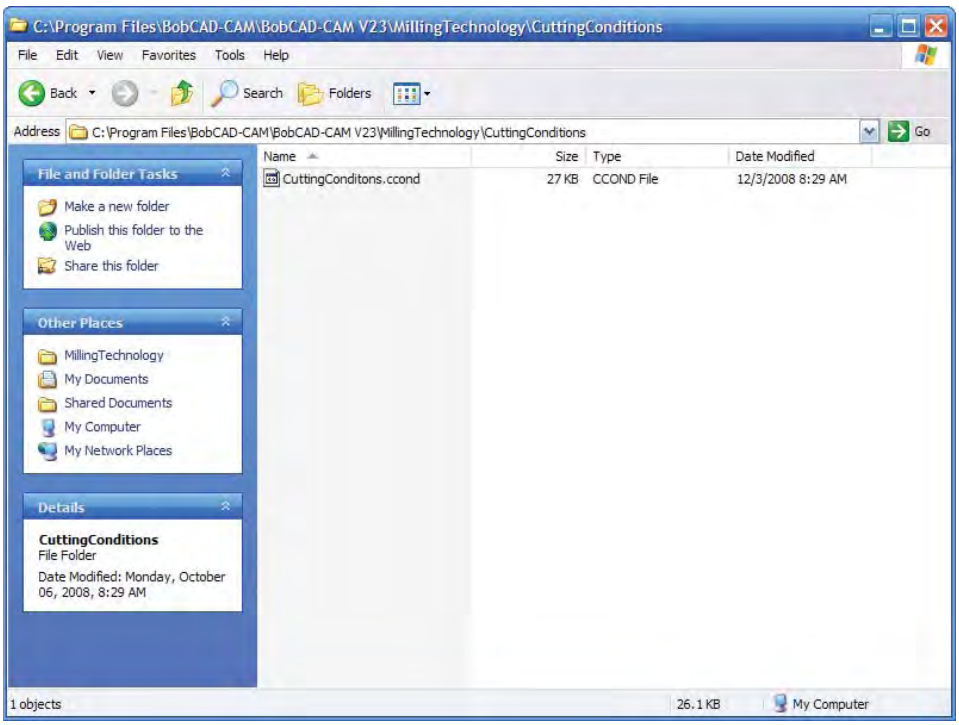
It's highly recommended to back up the cutting conditions f le in case of computer trouble or to transfer to another computer for use with extra seats of the software.

To start, locate the folder that BobCAD-CAM was installed into. Normally, this is **C:\Program Files\BobCAD-CAM\BobCAD-CAM V23**. The software may have been installed into a different folder or onto another drive on the computer entrely, but this is the default location assumed by the installer program.



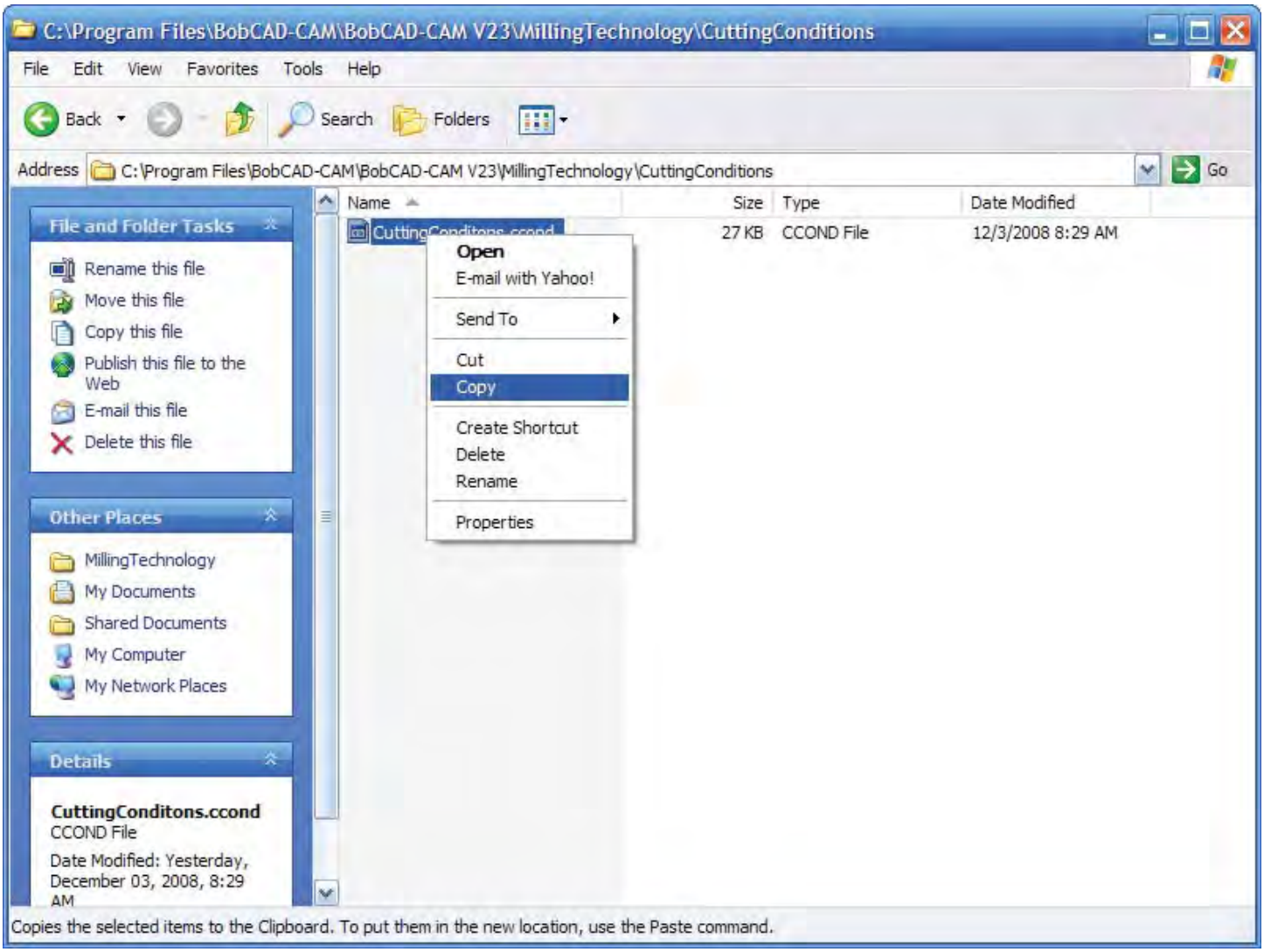
The default location of BobCAD-CAM V23

Once the software has been located on the hard drive, double-click on **MillingTechnology**, and onto **CuttingConditions**. The f le to look for is called **CuttingConditions.ccond**.



The default location of the CuttingConditions.ccond f le

Once the f le has been located, right click on it and choose **Copy**.



Copying the CuttingConditions.ccond f le to the Windows® clipboard for later pasting

From there, the f le can be pasted into another location on the hard drive, pasted into a folder on a network, or perhaps burned onto CD for safekeeping.

To replace the f le again should it become damaged or to install it into another seat of BobCAD-CAM, simply copy the f le back into the **CuttingConditions** folder under **MillingTechnology** in the main BobCAD-CAM installation folder on the destination computer while BobCAD-CAM is not running.