



FAQ #46: How do I set up BobCAD-CAM for a mill?

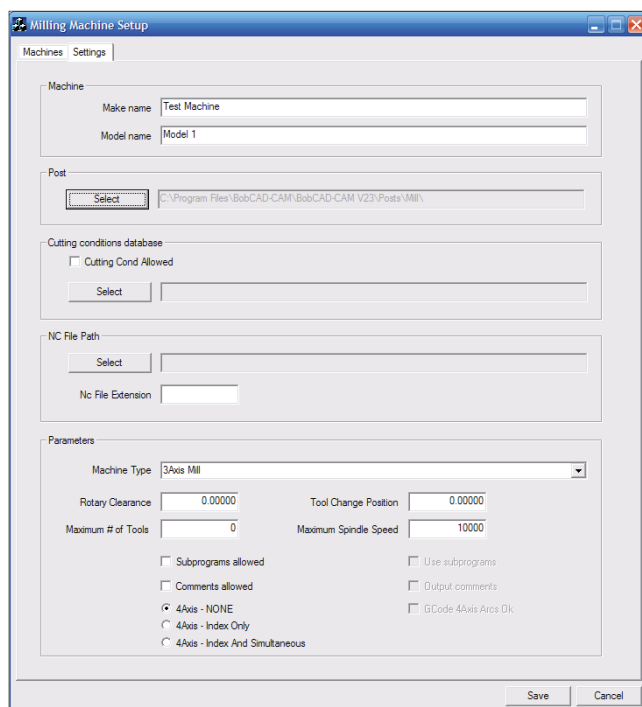
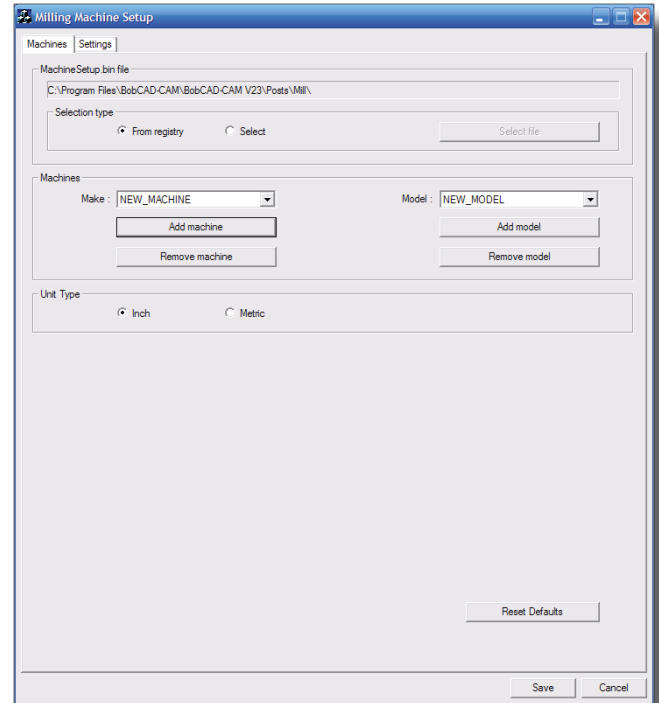
Q: The milling machine I use isn't listed when I try to pick a machine. How do I get it in there?

A: If the mill isn't listed, it will need to be set up. It's not difficult to do. A milling machine setup in BobCAD-CAM V23 consists of a setup that describes the machine's capabilities, the post processor configuration file, and the cutting conditions database. Most milling machines will use either the default conditions database or a modified version of it. Follow these steps:

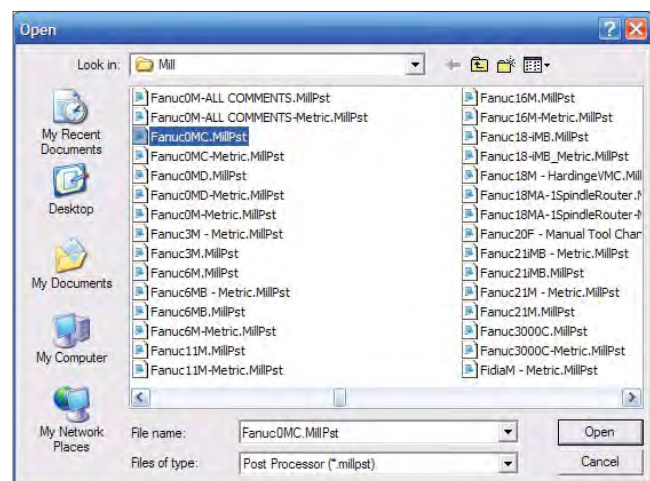
Step 1: Close BobCAD if it's open. Double click the MillingMachineSetup.exe program normally located in the BobCAD-CAM installation folder, usually C:\Program Files\BobCAD-CAM\BobCAD-CAM V23\, and the program will start.

Check for the make and model of the machine. If the make exists but the model doesn't, click on **Add Model**. If the whole machine make doesn't exist, click on **Add Machine**. In this example, both a new make and a new model will be added.

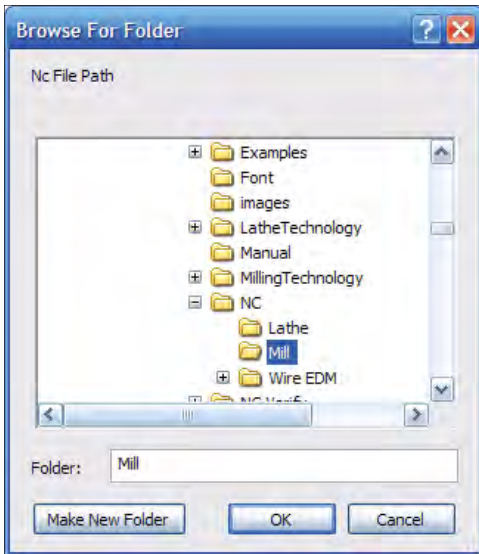
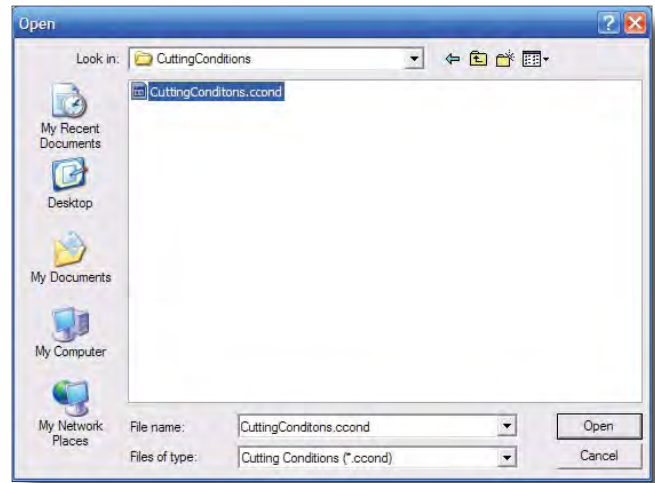
Step 2: Click on the **Settings** tab and type in the new name for the make or model. In this example, both are to be added.



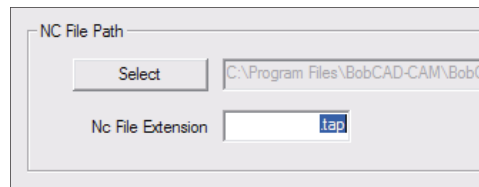
Click on the **Select** button in the **Post** section and choose the post processor to be used for this machine, then click **OK** on the **Open** dialog that will have appeared. Post processor configuration is covered elsewhere.



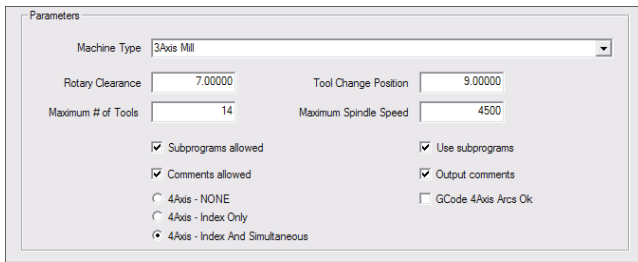
Generally a cutting conditions database is necessary, so check **Cutting Cond Allowed** and then the **Select** button. Navigate to the folder containing the desired .ccond file. BobCAD-CAM will usually go straight to the cutting conditions database folder for mill, which will contain the default database. Choose that unless there is another database to be used. Click **OK** in the dialog.



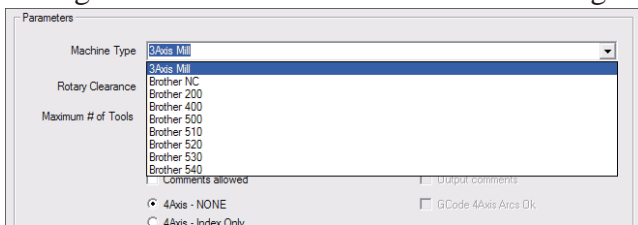
Click the **Select** button under **NC File Path** and choose a default folder to save post processed nc code to. BobCAD-CAM will use this folder by default when this machine is loaded. Type in the file extension to be used with this machine for the NC files. Don't forget to include the leading dot (".") before the extension.



In the **Parameters** section, choose the options that best fit the machine:



- **Machine Type** - The milling machine type can be chosen from this drop-down list. The most often used setting is 3Axis Mill. Choose this even if using a 4th axis.



- **Rotary Clearance** - This setting dictates the clearance retract height in Z when indexing the 4th axis.
- **Tool Change Position** - This sets the Z height of tool changes.
- **Maximum # Tools** - If the machine has a maximum number of tools allowed in a single program, enter that value here.
- **Maximum Spindle Speed** - BobCAD-CAM will not exceed the maximum spindle RPM set in this box.

- **Subprograms Allowed** - If the machine can use subprograms to help reduce the amount of repetitive code, check this box to enable BobCAD-CAM to produce them.
- **Use Subprograms** - If it is desired to use subprograms by default, check this box.
- **Comments Allowed** - If the machine cannot accept comments in a program, uncheck this.
- **Output Comments** - If it is desired to view comments in the NC code on the control and it can accept them, check this box and the comments will not be stripped from the NC file.
- **4axis - NONE** - Set this to disable all 4th axis output.
- **4axis - Index Only** - If the machine can index the 4th axis but not machine with it, set this option.
- **4axis - Index And Simultaneous** - If the machine has a fully functional 4th axis and it is desired to use it, set this option to nable BobCAD-CAM's 4th capabilities.
- **GCode 4Axis Arcs OK** - Not all machines, even with fully functional 4th axes, can accept arc moves using it. To ensure that they are not produced by BobCAD-CAM check this box.

Step 3: Click on the **Machines** tab at the top of the dialog again. Check over the settings, including the **Unit Type** section, and make sure that the machine make and model names are correct. Click **Save** at the bottom of the box. When BobCAD-CAM V23 is next started, this machine will be available to choose from the list and will automatically be set as the default machine.

*Note that many of these settings can be changed from within BobCAD-CAM itself, under **CAM Part, Milling Tools, and Current Settings**. If they are changed, they will override what is set here and will be saved to the machine configuration. The next time MillingMachineSetup.exe is run it will reflect the changes made from within the software.*

