

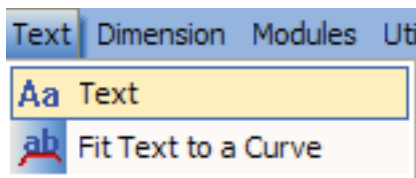
FAQ #18: How do I engrave text in BobCAD-CAM?

By special request, FAQ #18 covers simple text engraving using BobCAD-CAM. Engraving text is even simpler than most operations in the software. In most cases, it's as simple as "place text; cut text!"

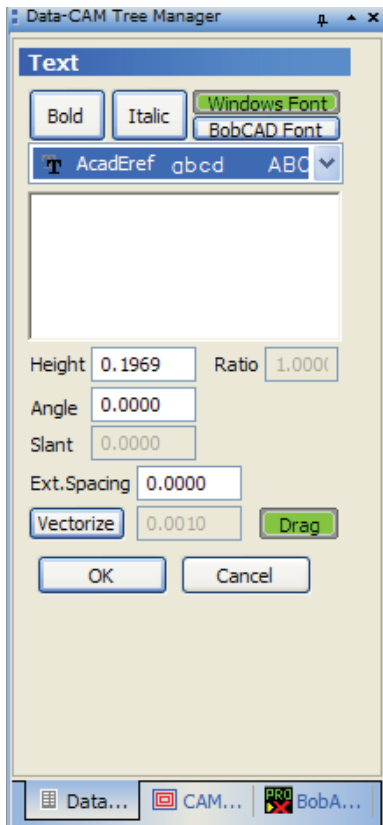
This example will illustrate the basic method for a 2-dimensional engraving.

Step 1: Begin a new drawing by typing Ctrl+N on the keyboard, selecting **File** from the main menu and then **New**, or by using the **New File** icon on the toolbar.

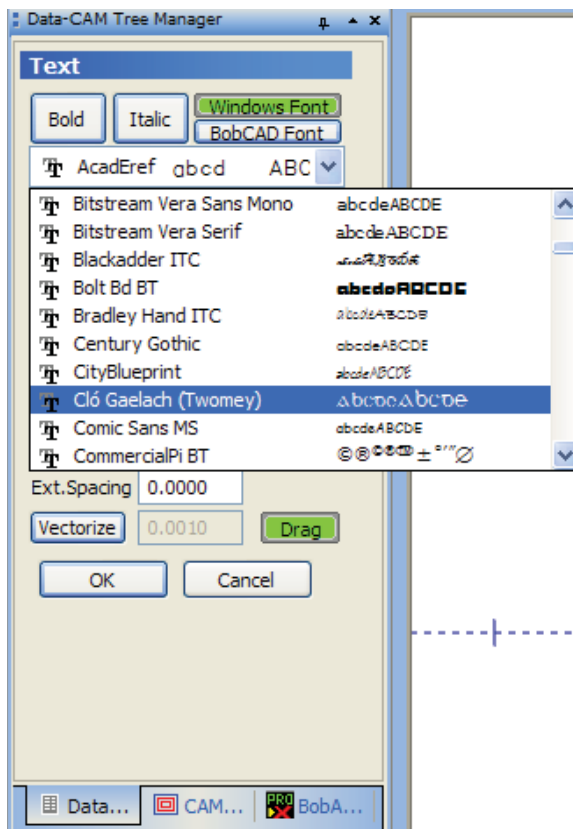
Step 2: From the main menu, select **Text** and then **Text**.



In the box that will appear to the left of the screen, all of the options available for the text entity type can be filled in.

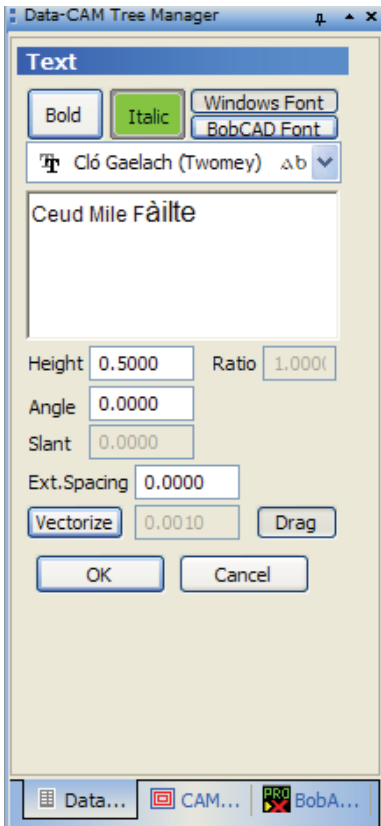


The illustration at left is the default dialog box. To begin, choose an appropriate font from the drop-down box as shown below:

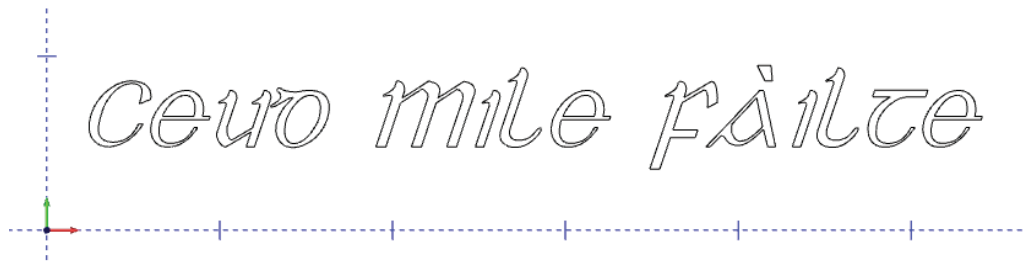


For this example, the Clò Gealach (Twomey) font was chosen. Don't worry too much about the font used or the text placed in the example, as nearly every computer in existence has a different font list. BobCAD-CAM does ship with a few fonts of its own; these are available when the user clicks on the **BobCAD Font** button at the top of the dialog. If **Windows Font** is pressed, BobCAD-CAM will list the fonts available throughout Windows. As an interesting tip, if more fonts are desired, sometimes it helps to install other programs, such as a word processor, in order to increase the fonts available to BobCAD! There are also several websites offering fonts for free download that BobCAD-CAM can readily use as soon as they are installed for use in the operating system, so be free to experiment.

To finish out the box, the example below has **Italic** set, the **Height** changed to .5, the **Drag** button set, and some text typed in. Click **OK** and BobCAD-CAM will allow the user to drag the text around the screen.



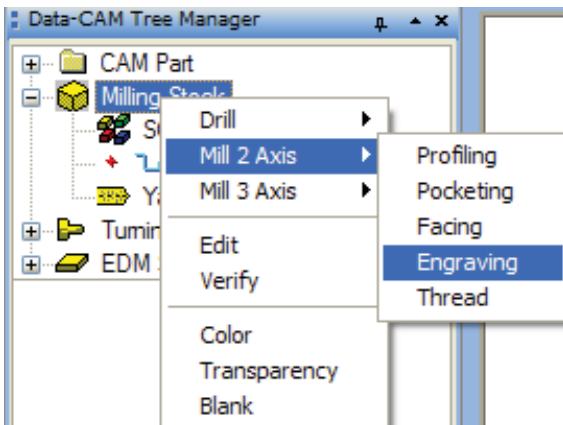
When the text is in the desired position, click the mouse once to set it.



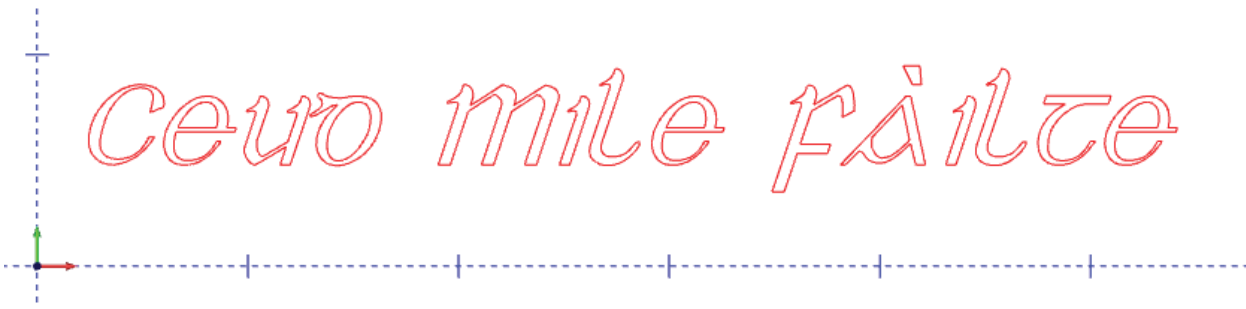
Right click anywhere in the **Workspace** and click **OK**, then click **Cancel** in the **Text** dialog to quit the function.

Step 3: Set up the machining feature in the CAM tree.

Click on the **CAM** tab at the bottom of the **Data-CAM Tree Manager**. Right click on **Milling Stock** and hover the cursor over **Mill 2 Axis**. From the fly-out menu, choose **Engrave** from the list.

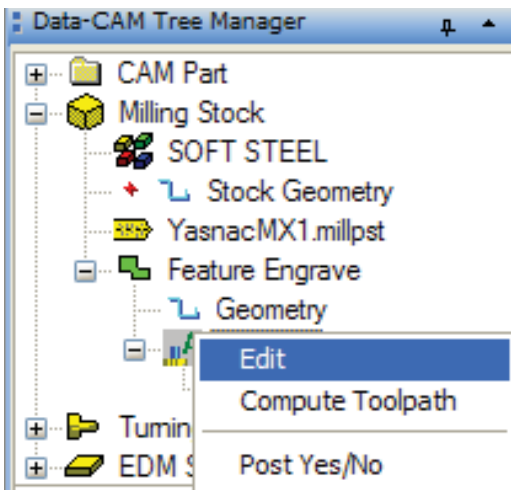


A new **Feature Engrave** feature will be added to the tree. Underneath this in the list, right click on **Geometry** and choose **Re/Select**. Click on the text in the **Workspace**. When it is highlighted, right click and choose **OK**.

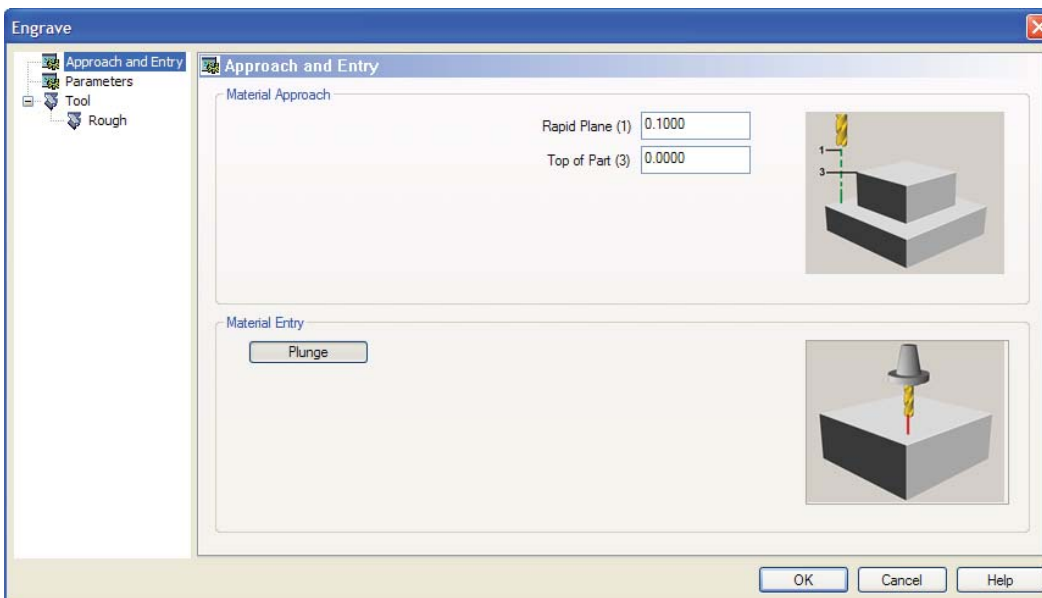


When the text is set, the **Geometry** item under **Feature Engrave** will lose the red dot next to it, indicating that the geometry has been properly set.

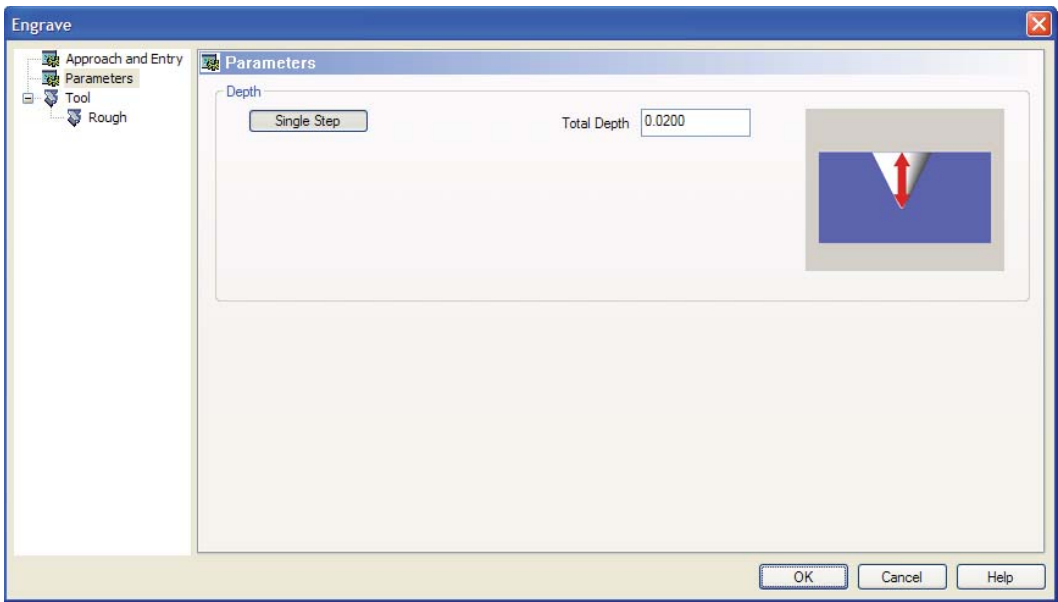
Right click on **Engrave** in the feature and choose **Edit** as shown below:



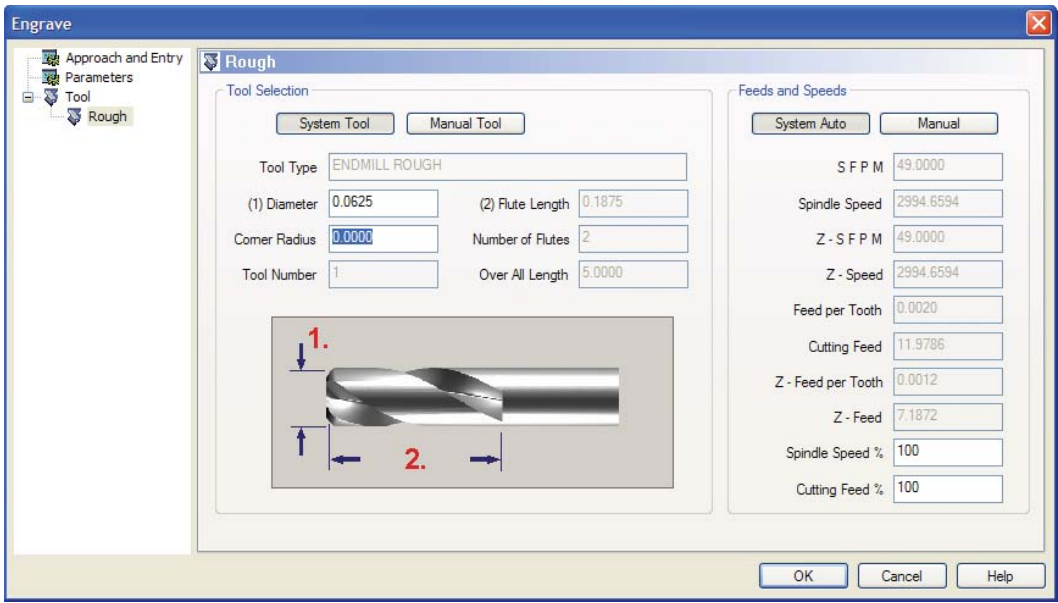
The feature's **Parameters** dialog box will display. The **Engrave** dialog is shorter than most:



The first item, **Approach and Entry**, is actually just fine in the defaults for this example. If not setting the top of the material at Z0 the numbers here will change, but for now they're just fine.



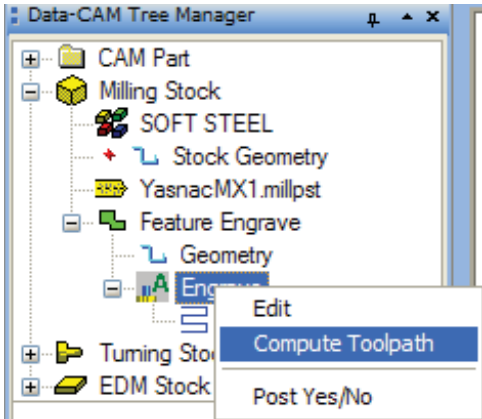
In **Parameters**, set the **Total Depth** to .02 to go that deep into the material.



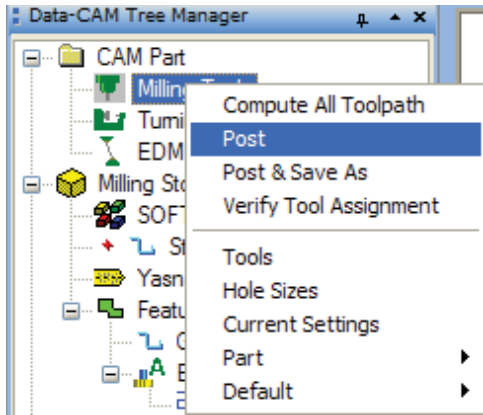
In the **Rough** sub-item of **Tool**, set the **Diameter** to .0625 to use a 1/16" cutter.

That's it for the cutting parameters, so click **OK** to finish.

Step 4: The feature is set up, so all there is left to do is computer the toolpath and post the NC code. Right click on **Engrave** as shown below and click **Compute Toolpath**.



When the toolpath is computed, open **CAM Part** at the top of the tree by clicking on the + icon next to it. Right click on **Milling Tools** and select **Post**.



That's it, the engraving is complete!