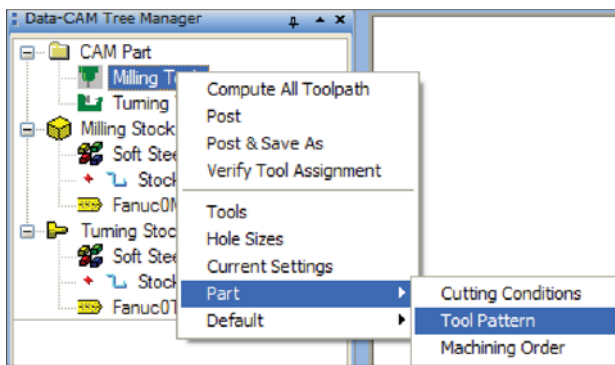


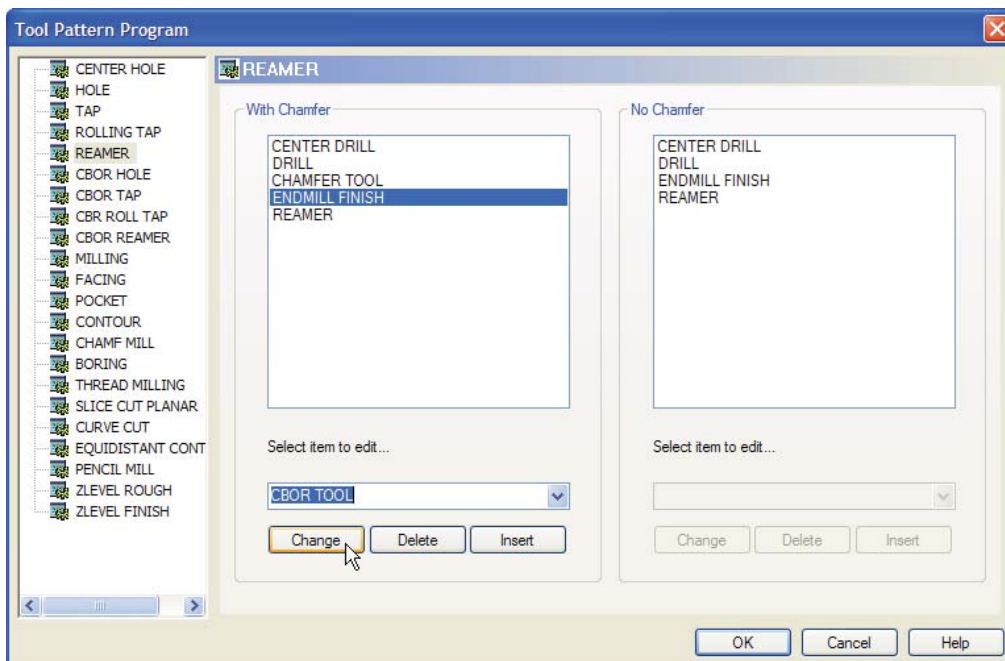
FAQ #13: How to I change the pattern of operations in a drilling cycle?

Not all drilling operations are the same, especially when one considers what the end result of a holing feature should look like. For this reason, BobCAD-CAM V22 offers a way to change the pattern of operations executed on any feature. For example, it's common practice to countersink (chamfer) a tapped hole before running the tap into it, but it is also common not to do so in certain circumstances. This FAQ will illustrate change a finish mill operation from a reamed hole feature into a more common counterbore. Note that the same steps given here also apply to turning features; simply apply the same technique to Turning Tools instead of Milling Tools.

Step 1. Click on the + symbol next to **CAM Part** to open it up. Right-click on **Milling Tools** and hover the mouse cursor over **Part**. If the change is meant to be permanent, choose **Default** instead. Click on **Tool Pattern**.



Step 2. From the dialog that will appear, choose **REAMER** from the list to the left (5th item from the top). Click on **ENDMILL FINISH** in the section labeled **With Chamfer**. From the drop-down list under this box, choose **CBORE TOOL** and then click on **Change**. The **ENDMILL FINISH** will be switched out for a counterboring operation. Click **OK** to have BobCAD accept the change.



It is also possible to remove items from the list completely and to insert new items.

To remove an item from the list, simply highlight the item and choose the **Delete** button below. Click **OK** and it will be removed from any matching features currently using it in the tree.

To insert an item, highlight where it should be inserted in the list (before / after which items), choose the desired operation and click **Insert**. Again, click **OK** to have BobCAD accept the change.