MasterCAMM Contour Parameters

Toolpath Type

1. Make sure correct operation is highlighted. (contour)
### Tool

1. Select "Filter" button.
   - Select tool button that matches the cutter to be used.
   - Select check mark.
2. Click Select Library Tool Button.
   - Select correct cutting tool.
3. Change Feed rate, Plunge rate and Retract rate to 60 in/Min (yellow window)
4. Include what operation is about to take place in the comment section.
   - be sure to include such things as operation (drill, pocket or contour), tool diameter, depth of cut and number of passes.
Cut Parameters

1. **Compensation type**: Computer
2. **Compensation direction**: Left or Right based on direction of cut (clockwise or counterclockwise displayed earlier when selecting contour)
3. **Contour type**: 2D
**Depth Cuts** (over ¼" depth cut)

1. Select *Depth Cuts* in left window (parameters list)
2. Turn on *Depth cuts* by selecting or checking box.
3. Input *Max rough step of .250"*
4. *Depth cut order*: Determine if cut is to be done by contour or by depth.
5. If by contour, *Keep tool down* will decrease cycle time.
**Lead In/Out**

1. Uncheck box (turning it off)

Note that it is now disabled.
Linking Parameters

1. Select *Absolute* under: *Retract*, *Feed plane*, *Top of stock* and *Depth*.
2. Input your depth of cut (total) making sure it is a negative(-) value in the yellow window.
Home/Ref Points

1. Change Home position of X,Y and Z to zeros (0). (yellow window)

Click the Green check mark in the lower right corner to generate the program.