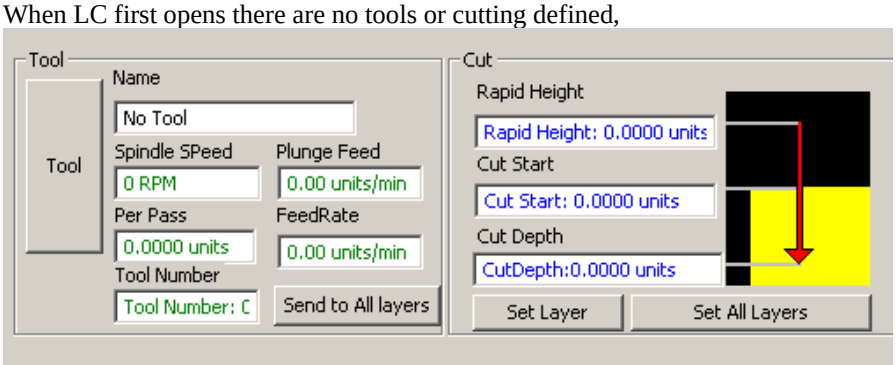
TUTORIAL #6 – MISCLEANOUS

1.0 PLUNGE CLEARANCE

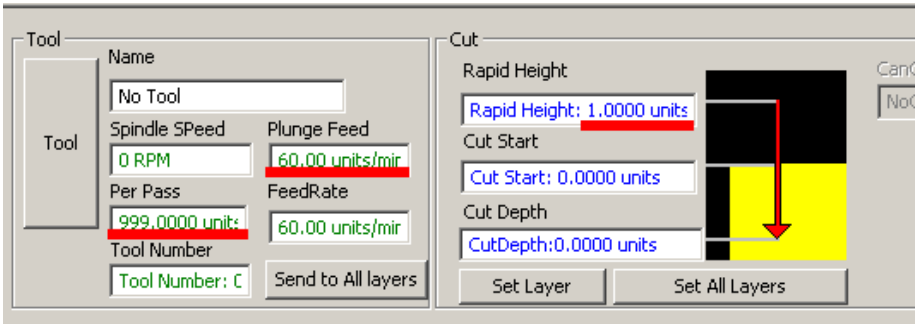
The following write up is about the Plunge Clearance setting. It's provided to make the user aware of how the setting relates to the Cut parameters and posted code.

When LC first opens there are no tools or cutting defined,



T6- FIGURE 1

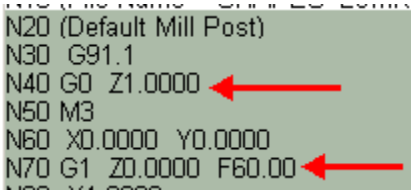
When a DXF is imported, LC provides a default tool and Z rapid plane to each of the created layers / chains in order to show rapid moves from one to another chain.



T6- FIGURE 2

Notice that the default plunge feed is 60 and per pass is very large and a rapid of 1 unit is applied.

If you never did anything to the file and posted it to MACH those settings become part of the generated code.

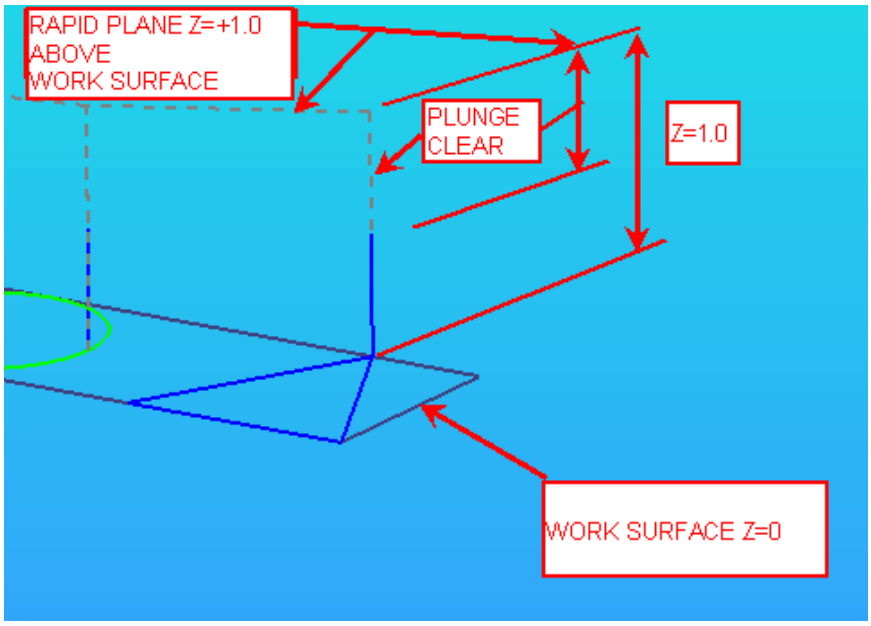


T6- FIGURE 3

The code generated is just a rapid (G00) , moving at the max velocity setting you defined for the motors in motor tuning, from the origin to the clearance plane of Z=1. In this case, since the origin is also at the same location of the first chain to be cut , the tool then does a (G01) interpolated move at a designated feed rate (F60) into the material.

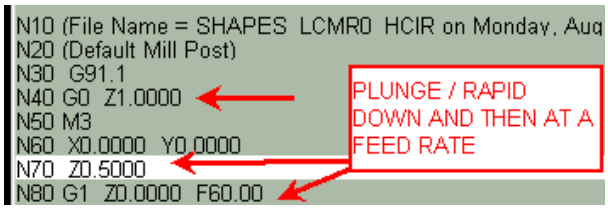
One additional point is that in the setup>posting options, the value for the plunge clearance was set to a zero value. So the Z moves as if it were cutting material all the way down to the piece.

Now if you were to change the Plunge Clearance setting to say a value of ½ the default Z clearance plane value as depicted below, and import the DXF, different code would be generated.



T6- FIGURE 4

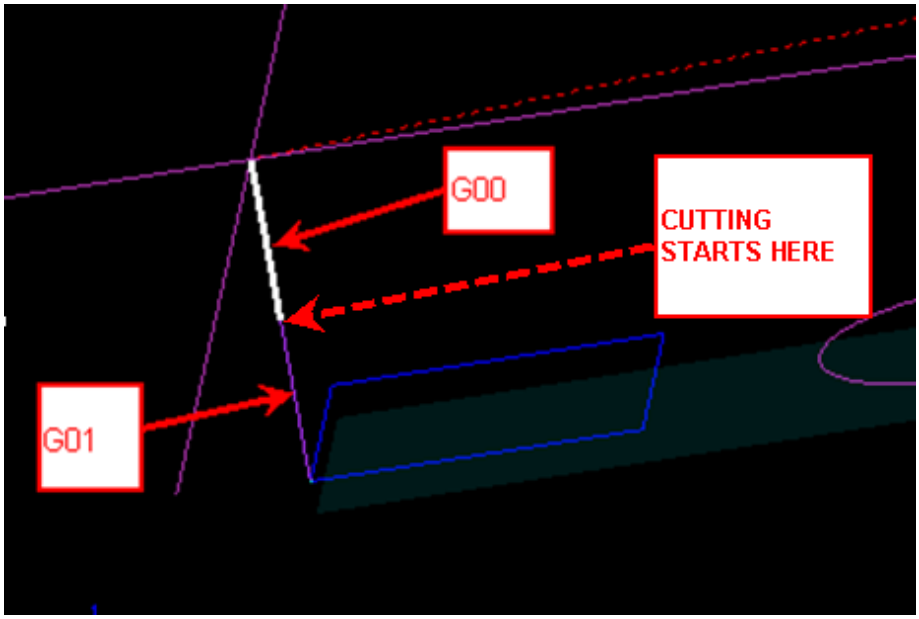
Here is the code after posting to Mach.



T6- FIGURE 5

The code generated is just a rapid (G00) , moving at the max velocity setting you defined for the motors in motor tuning, from the origin to the clearance plane of Z=1. In this case, since the origin is also at the same location of the first chain to be cut , the tool then does a rapid (G00) down to Z=+5 and then a (G01) interpolated move / cutting at a designated feed rate (F60) into the material.

The G01 & G00 moves are shown in Mach Mill below.



T6- FIGURE 6

Your rapid velocity is usually much faster than when cutting. As an example, if the rapid is 100 inches / min and the cutting feed velocity is only 1 inch / min, it would take a lot of time for the Z axis moves and especially if they were long moves. The cutting starts / is done with a G01 move.

Plunge Clearance - The distance ABOVE the Cut starting point where Mach can safely rapid down to without hitting the workpiece. IF set to ZERO it turns OFF the function and mach feeds from the SafeZ height (Rapid Plane) to the cut level instead of rapids.

2.0 JOINING ENTITIES and CHAINS

LC is not a CAD program. LC is not a CAD program. LC is not a CAD program.

LC is not meant to be used as CAD program to move defined machining steps and fix poorly drawn or created contours which are converted into chains. The user must realize that once a DXF is imported they are dealing with further definition of machining and not primitive/ line / curve etc drawn items. You can destroy the accuracy of your contours and the logic of project planning in manipulating machining contours. Yes you can create all kinds of "stuff" by tricking LC. The novice user may not have the machining expertise to realize what was done and how to restore intent.

The majority of problems encountered are just lack of a good drawing representing the machining steps. Most "changes" can be done faster and more easily in your CAD program. If you move a chain, then you have moved a machining step, the rapid moves will be changed, you may move the chain and not place it correctly, cutting directions may change, etc.

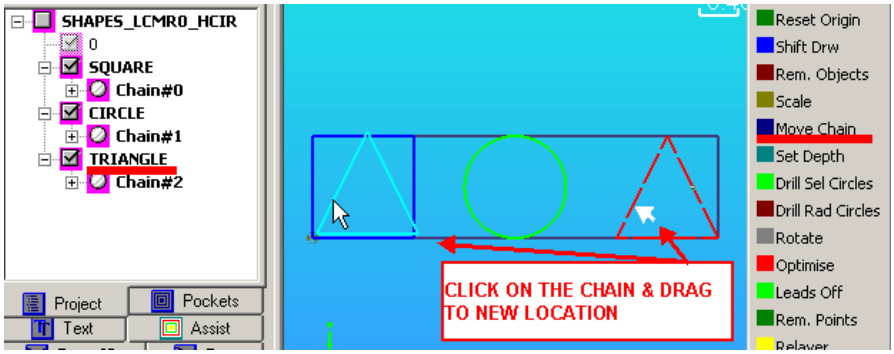
LC's importer is good, but, certainly not perfect. There are tolerances to consider and were discussed elsewhere. Some high end programs will force you to go into their provided Cad module to fix some simple or complex at times.

The Shapes_LCMO_HCIR will be used in the following sections. (it's assumed you have read the manual and gone through the tutorials)

2.1 JOINING CHAINS

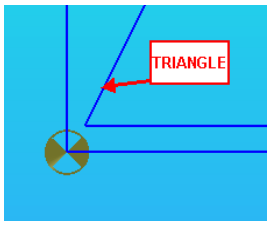
You don't join chains. You actually move a chain and optimize the project, which in turn optimizes "everything".

Import, select the Mill Module, clean the DXF. Then move the triangle into the square.
Select the triangle, use the Move Chain tool, and mouse move / drag the triangle close to the origin.



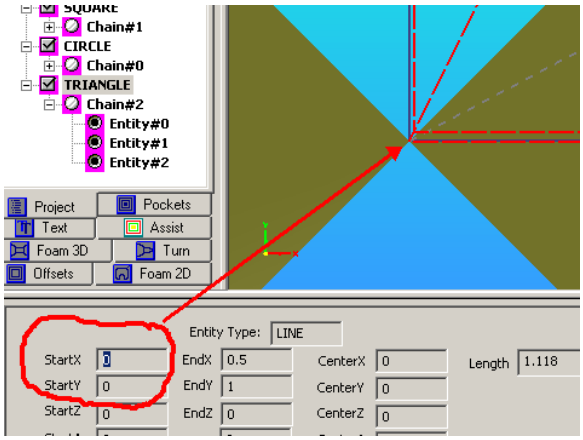
T6- FIGURE 7

Use the Zoom Box tool or mouse to zoom in on the origin. Click the Optimise tool.



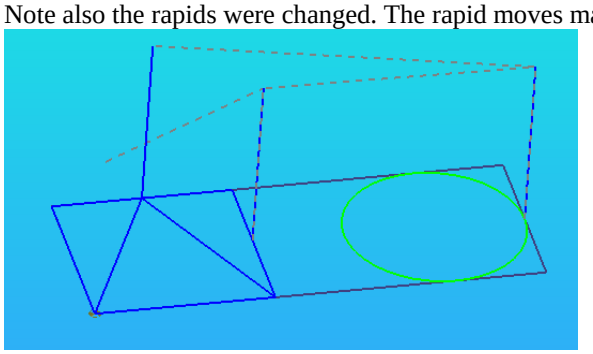
T6- FIGURE 8

The chain can be mouse moved and dragged again. It took two attempts to get it to the origin.
You can use information from the Entities menu to know the location of an entity as shown below.



T6- FIGURE 9

Note also the rapids were changed. The rapid moves may no longer be satisfactory.



T6- FIGURE 10

The move tool can be used to advantage for clamp placement, moving text, etc. where the placement is not critical. LC is not a CAD program as stated numerous times.

Create the machining operations in CAD and not LC!

2.2 JOINING ENTITIES

There may be occasions where you need to join two entities because of a tolerance issue on import.

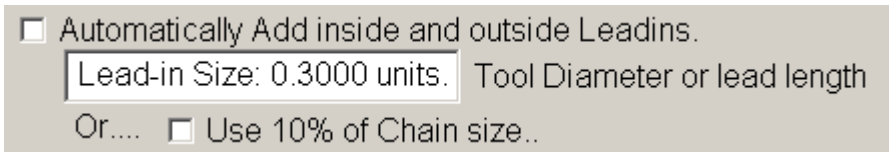
TUTORIAL #7 – ADVANCED MACHING MOVES

PREFACE

This tutorial provides for advanced usage of LC. The tutorial continues to build on prior tutorials and goes into more detail on how to use LC for advanced machining operations. Items covered are lead-in & out's, offsetting, adjusting leads and rapid locations, cutting direction and more. Most of the other tutorials only dealt with accepting what LC provided, now changing will be covered. The user should be totally comfortable with using the miscellaneous tools and menus, screen manipulation and basic operations before doing this tutorial.

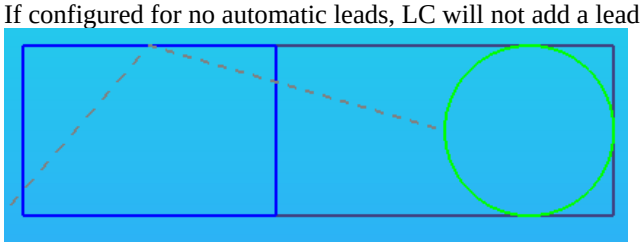
1.0 AUTOMATIC - LEAD IN & LEAD OUT

There are a number of options in LC for producing lead's (lead-in & lead-out).
LC can be configured to automatically add the leads on import by checking the box and defining the lead in size or using the 10% default value of chain size.



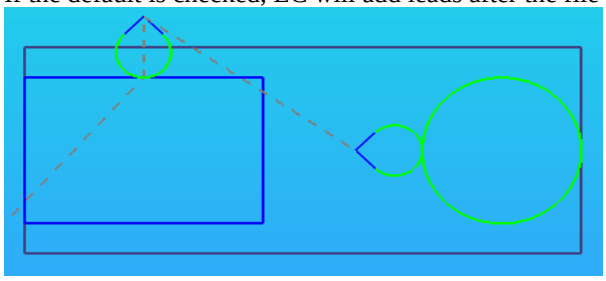
T7 – FIGURE 2

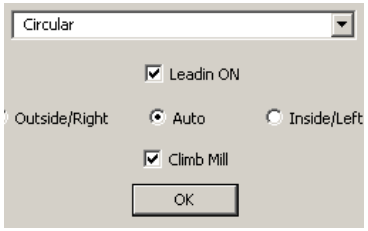
If configured for no automatic leads, LC will not add a lead in.



T7 – FIGURE 3

If the default is checked, LC will add leads after the file is cleaned with the following default settings.

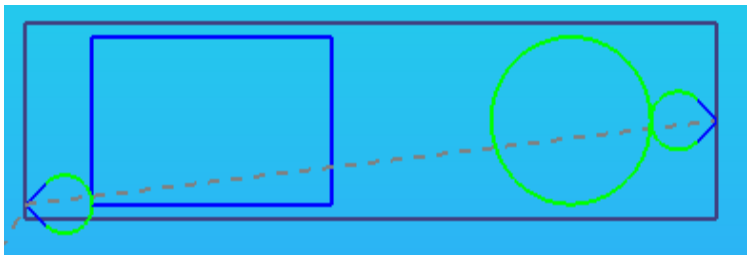




T7 – FIGURE 4

There is no way to change the default setting and the lead location is based on how the

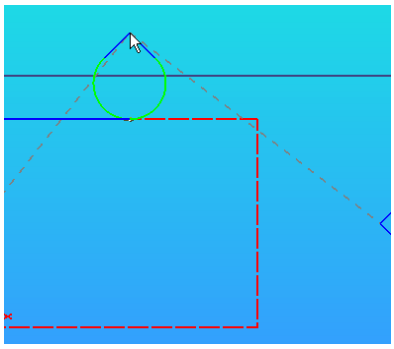
file was automatically optimized. As is, the automatic setting could be used to advantage if all or the majority of the machining operations required the default lead type.



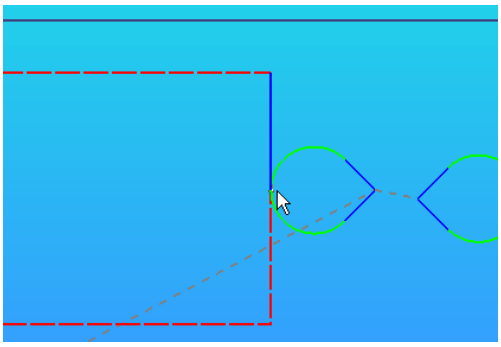
T7 – FIGURE 5

The user has the option to change the location and the type of lead provided.

Changing the location of the lead is done the same the as changing the rapid location.
Click once to select the lead and then click again but hold down the mouse button and drag the lead-in along the contour to it's new location, clicking again at the contour to fix the location. The lead will seem to "snap" at the mid point of the entity.

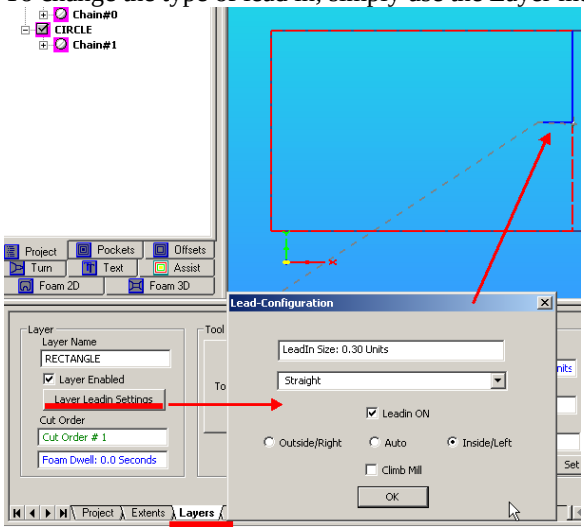


T7 – FIGURE 6



T7 – FIGURE 7

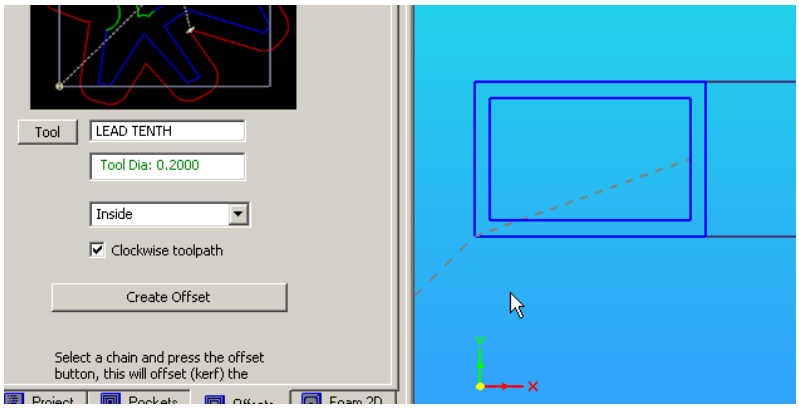
To change the type of lead in, simply use the Layer menu, and select one of the provided options.



T7 – FIGURE 8

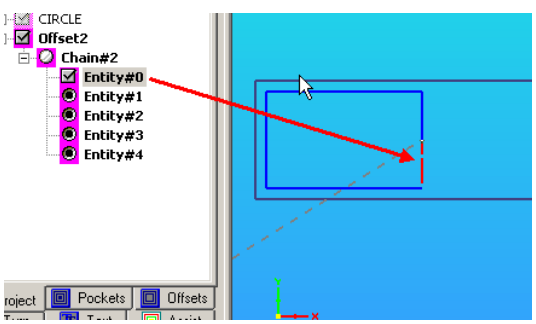
2.0 RAPIDS AND OFFSETS

Create an offset, and in this case inside and clockwise toolpath is chosen

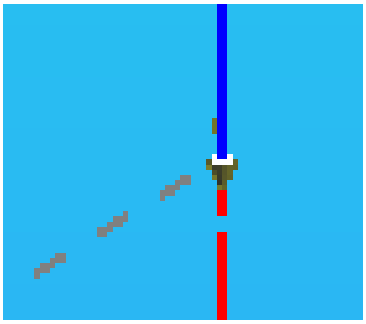


T7 – FIGURE 9

You can turn the original contour off or delete (the layer was turned off for below). Notice that LC breaks one of the entities into 2 entities when a offset is created. Also note the cutting order of the entities. LC leaves the original entity broken two. If you Zoom way in you will see the arrow which indicates direction.

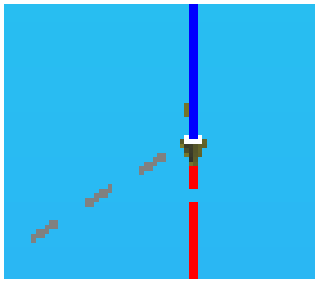


T7 – FIGURE 10

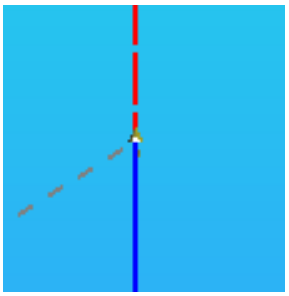


(ZOOMED IN VIEW)

You have a choice between CW (figure 9) and CCW (figure 10) pathing. Note that clicking on the appropriate entity / first in the chain to be cut and zooming in the pointer shows the cut direction.



T7 – FIGURE 11

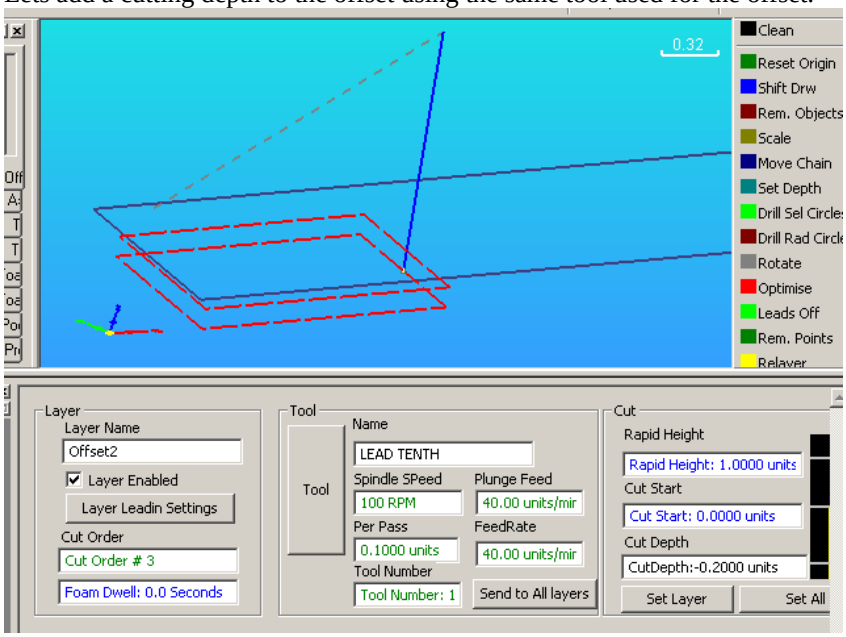


T7 – FIGURE 12

LC optimized the rapids on import, and seems to always default the rapid move to the most positive entity (note that in this file there was a circle so the finished Z move should end as close to the next machining operation as possible. Also note the it's in the middle of the entity. You do not want the Z to penetrate at a corner as the offset would be cut wrong. *G41 and G42 are based the controller looking ahead to the next move to define if it will be a CW or CCW move. Cutter radius compensation is being use and as such, by rules of cutting comp, implementation of compensation should occur after the Z move*.

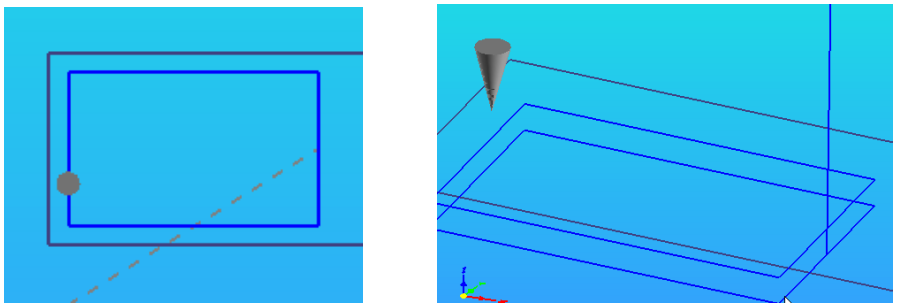
The reader should consult the many books on CNC programming to understand radius compensation relative to leadins, proper Ccode sequencing etc.

Lets add a cutting depth to the offset using the same tool used for the offset.



T7 – FIGURE 13

You can simulate the cutting of the offset in plan or isometric viewing (Alt + S).
The dot and the cone are the tool. There is no setting which will slow the simulation down.

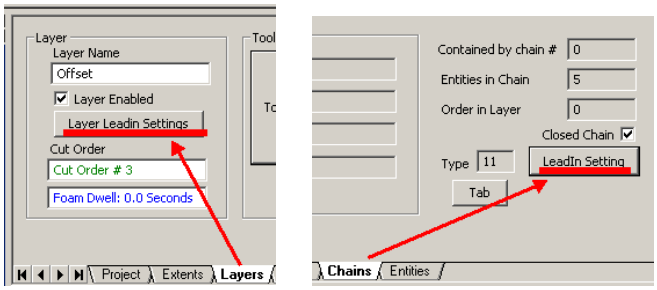


T7 – FIGURE 14

Optionally you can post the file to MACH for a better simulation and walk thru each of the steps relating the code to what is displayed. I have made a few comments in red. Here is the posted code:

N10 (File Name = LEAD on Thursday, August 13, 2009)
N20 (Default Mill Post)
N30 G91.1
N40 M5 M9
N50 M6 T1(TOOL Change LEAD TENTH)
N60 G43 H1
N70 G0 Z1.0000 RAPID RAPID MOVE FROM ORIGIN TO CLEAR PLANE / Z RAPID PLANE
N80 M3 S100
N90 X2.4000 Y1.5000 MOVE FROM ORIGIN TO WHERE THE TOOL WILL FEED INTO THE MATERIAL.
N100 G1 Z-0.1000 F40.00 THE Z AXIS MOVE INTO THE MATERAIL AT A FEED RATE FOR THE FIRST CONTOUR
N110 Y1.1000
N120 X1.1000
N130 Y1.9000
N140 X2.4000 Y1.9000
N150 X2.4000 Y1.5000
N160 G0 Z1.0000 RAPID MOVE BACK TO THE CLEAR PLANE , YOU COULD ADD A DWELL IF SO DESIRED TO ALLOW FOR ADDITIONAL COOLANT TO FLUSH THE TOOL OR MATERIAL IF SO DESIRED
N170 G1 Z-0.2000 THE Z AXIS MOVE INTO THE MATERAIL AT A FEED RATE FOR THE SECOND CONTOUR PASS
N180 Y1.1000
N190 X1.1000
N200 Y1.9000
N210 X2.4000 Y1.9000
N220 X2.4000 Y1.5000
N230 G0 Z1.0000 THE Z RETRACTS TO THE CLEAR PLANE, NOTE: SINCE ONLY THE RECTANGLE OFFSET WAS DIPSLAYED / ENABLED AT POSTING LC CONSIDERS THE MACHINING OPERATION COMPLETE.
YOU WOULD NEED TO ADDITOINAL CODING TO RETURN TO THE ORIGIN IF SO DESIRED.
N240 M5
N250 M30

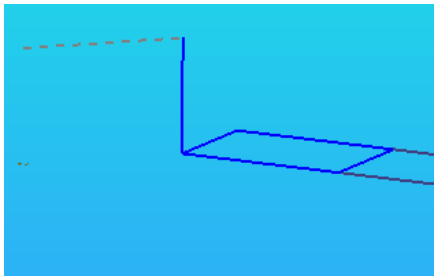
Leads can also be added globally for the layer and chain using the following menus.



T7 – FIGURE 16

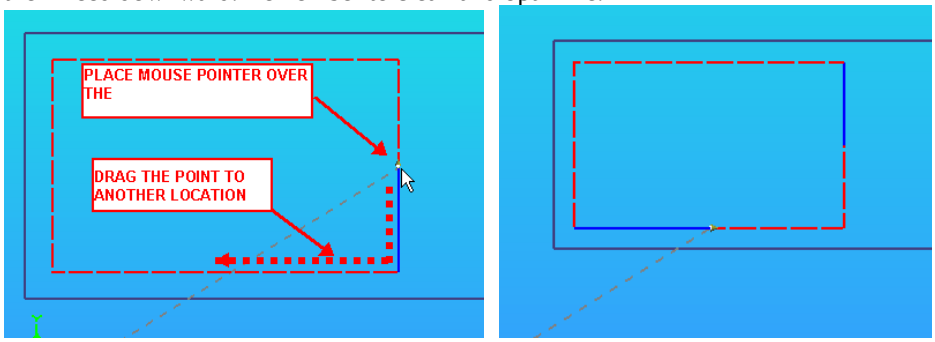
2.1 CHANGING THE RAPID TO A DIFFERENT LOCATION

The rapid to a point on a contour can be changed.
If the rapid cutting point is changed you may not be able to ooops your way back to the beginning.
It is suggested you save your work as a .LCAM file often and clean after any changes.
Below is an example of moving the cutting start point to say the corner which should be avoided!
Rapids should be changed for a reason and with understanding of how it may affect the machining operation.



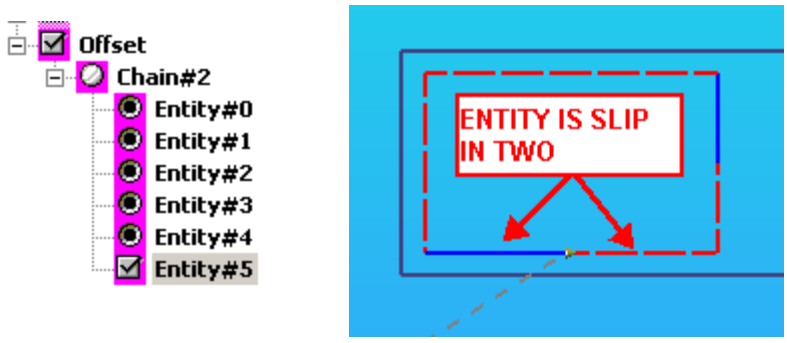
T7 – FIGURE 15

To change the rapid you just place the mouse over the cut direction indicator, click on it and continue holding down the mouse button while you drag it along the coutour. Click on the location you wish to have the Z feed downward. Remember to clean and optimize.



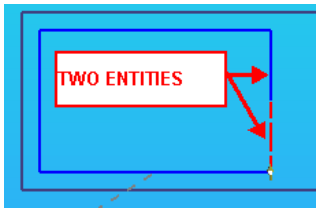
T7 – FIGURE 16

If your happy with the new location, clean and optimize. Notice that LC created another Entity / split the entity into two parts.



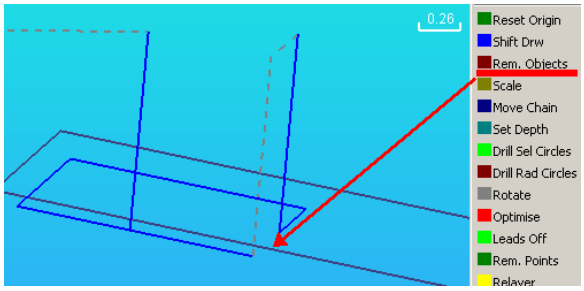
T7 – FIGURE 17

Note that LC will leave the original entity in two parts.



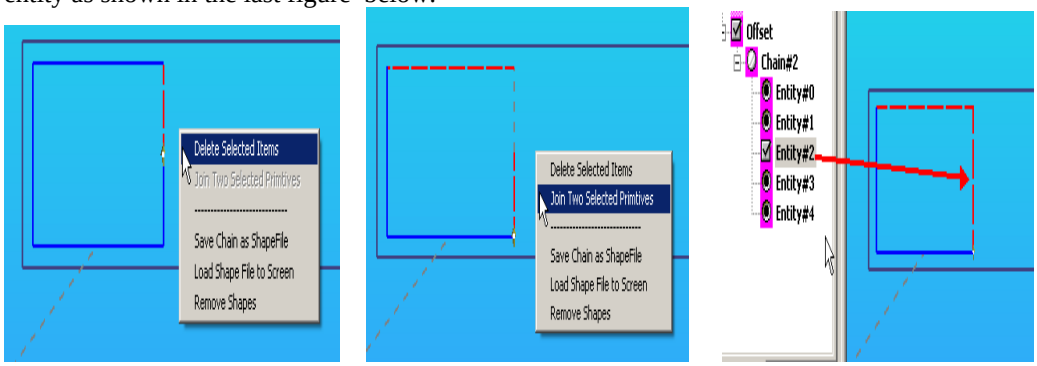
T7 – FIGURE 18

The remove objects tool will delete the highlighted entity and LC provides a rapid to maintain the pathing.



T7 – FIGURE 19

To modify the chain, delete one of the entities, and then use the join entity command if you want only one entity as shown in the last figure below.

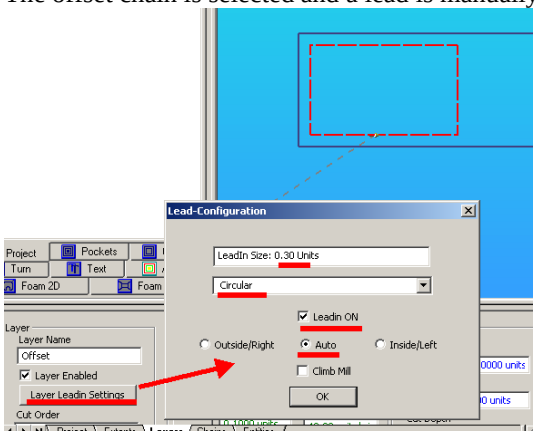


T7 – FIGURE 20

Note, although there are many options which seem to be available when using the tools, you are manipulating machining paths. LC is not a CAD program. You should always check or simulate posted code. Randomly modifying the pathing for no particular reason is senseless. KISS!

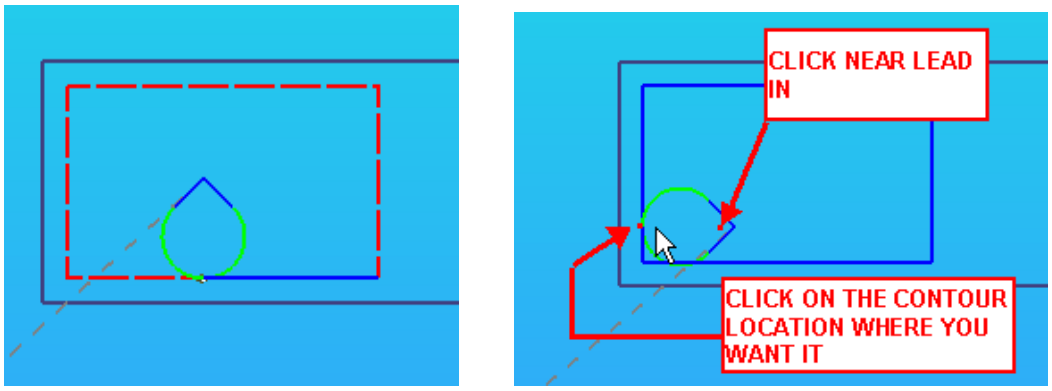
2.2 ADDING LEADS

Lets quickly add a lead and change its location since its very similar to changing the rapid location.
The offset chain is selected and a lead is manually added.



T7 – FIGURE 21

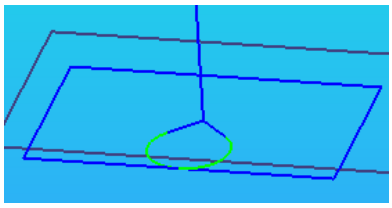
Changing the location of the lead is done the same the as changing the rapid location.
Click once to select the lead and then click again but hold down the mouse button and drag the lead-in along the contour to it's new location, clicking again at the contour to fix the location. The lead will seem to "snap" at the mid point of the entity.



T7 – FIGURE 22

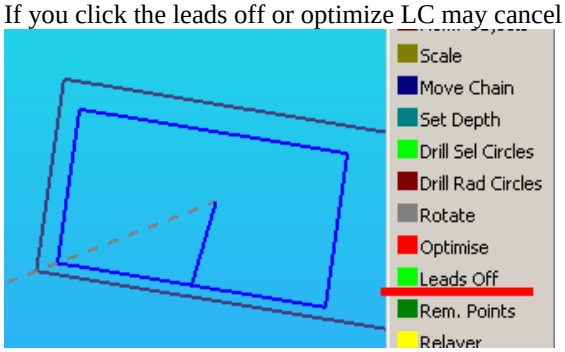
2.3 LEAD'S GENERAL

LC only allows one lead in / lead out per machining operation. Leads are only created at the layer and chain level.



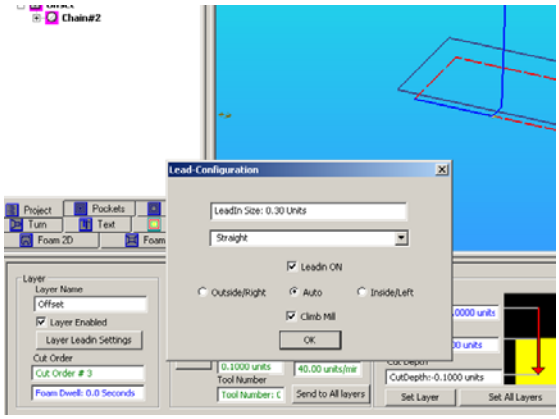
T7 – FIGURE 23

If you click the leads off or optimize LC may cancel any lead you created.



T7 – FIGURE 24

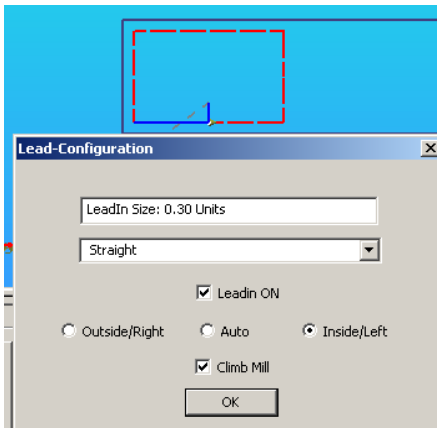
You can add the leads back or select a different type if you wish via the layers menu.



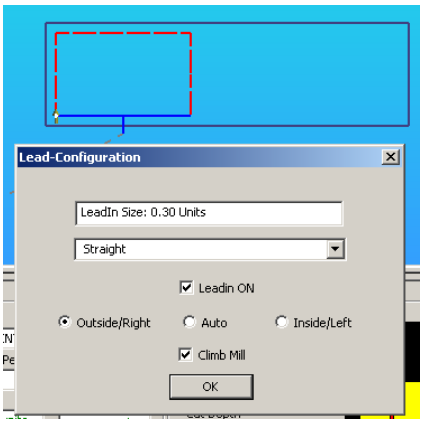
T7 – FIGURE 25

2.4 EXAMPLE OF LEADS

The minimum lead in size should be equal to 2X the cutter diameter.

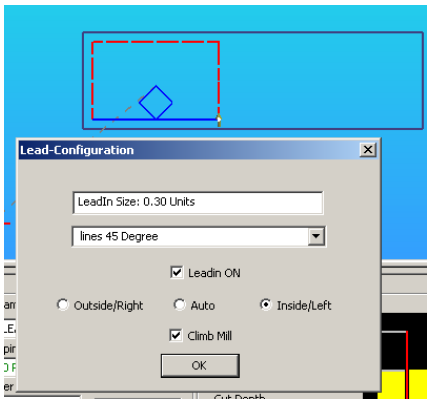


Straight , inside left, climb mill

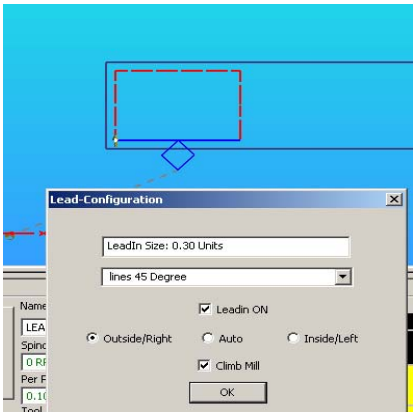


Straight , inside right, climb mill

T7 – FIGURE 26

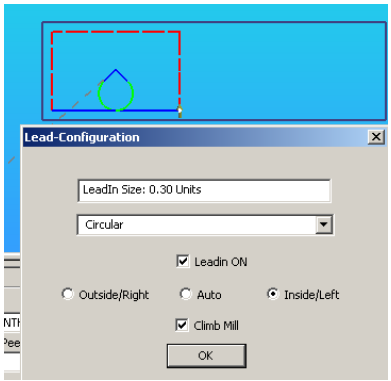


Lines 45 deg , inside left, climb mill

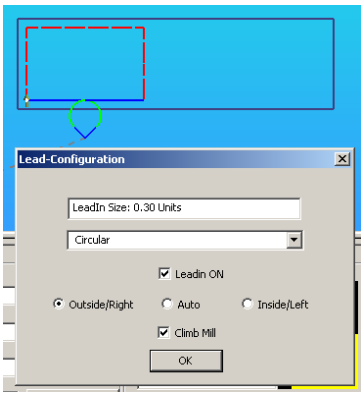


Lines 45 deg , inside right, climb mill

T7 – FIGURE 27



Circular , inside left, climb mill



Circular, inside right, climb mill

T7 – FIGURE 28

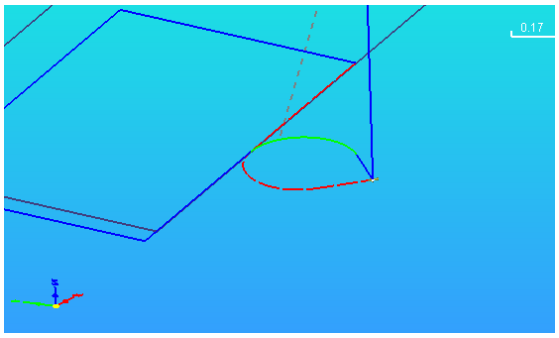
2.5 CLIMB & CONVENTIONAL MILLING

The difference between climb and conventional milling is how the cutter is turning / cutting relative to the material feed direction. Climb milling (down milling) is best for finishing cuts and pulls the cutter into the work. Conventional milling (up milling) will push in the opposite direction. Chip formation and tool loading is different between the two. You should avoid cutting with the full diameter of the cutter (60% may be a good value to use) and use light cuts for a finish cuts. There are book s written about how to use end mills, do facing or slotting cuts, and elaboration on the subject is out of this manuals scope.

2.6 CUT DIRECTION

The direction of material feed (axis movement) will be determined based on which type of milling is selected. The climb milling check box acts a switch. Note that you should and can verify the cut direction by relating to the what is selected to the machining order (entity listing). You can't change the machining order of entities in the chain. Select a different type of milling to change the direction / cutting order.

Changing the cut direction is buggy in LC as described above. You will find that the cutting direction will change, but, you need to go through a sequence to get it to change. There should be a know default setting, and any time the "switch" is checked, the cutting direction is reversed compared to the prior direction. This can be very annoying,



T7 – FIGURE 29

LC BUG / KNOWN PROBLEMS LIST

FOAM 2D - provides for cutting of foam (not covered in this manual)
FOAM 3D - provides for cutting of foam (not covered in this manual)
IMAGE FILE – import an image file and generate Gcode (not covered in this manual)
KEYBOARD
S – allows for simulation of pathing post creation of the paths
NOTE: To stop the simulation hit the S key again (simulation does not work in the lathe module)

New - invokes a window for creating a new drawing border

Highlighting of the all the text "Plunge Clearance 0.0000" and then backspacing will cause LC to shutdown.

To change the Plunge Clearance highlight only the numerical value and type in a new value as shown below.

Highlighting of the all the text "Lead-in Size 0.0000" and then backspacing will cause LC to shutdown.

Remove Redundant Entities – (NOT FUNCTIONAL) Redundant entities are removed if the option is checked in the Setup > Loading Option menu.

Minimize Geometry – (NOT KNOWN)

Fillet Corners – (NOT FUNCTIONAL)

Remove Circles – don't use this command / buggy

The X & Y distance moves are not functional for a chain. Use the Mouse Move.

Click the **Set All Layers** to apply the cutting to all highlighted / enabled layers of the project.
The Set All Layers button can be used, but, the results may not be as expected. LC will randomly select some layers and not others. It is suggested, that for each chain, that a tool and cutting parameters be defined and the set layer button used.

You have choice of no cycle or a G81 drill cycle. The G83 and G73 drill cycles do not work / will not post. You need to manually change the G81 code line.

Outside/Right - defines how the compensation is applied

Left -
RIGHT -

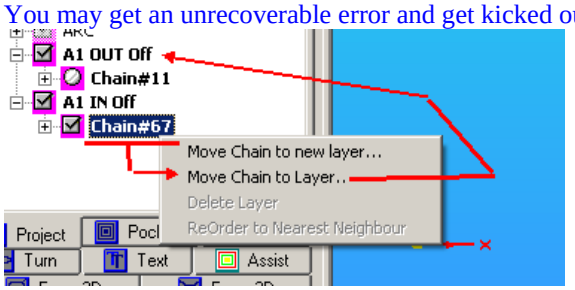
OverHang: - not required and not functional (overhang, as I understand it, would be the amount of un used Cutter flute or can also be the length of cutter extending beyond a holder.

You will get a G81 drill cycle since the **G83 and G73 canned cycles are not functional.**

Note that it is not recommended to create a sub-directory under the Shapes directory and copy shape files into it for use.

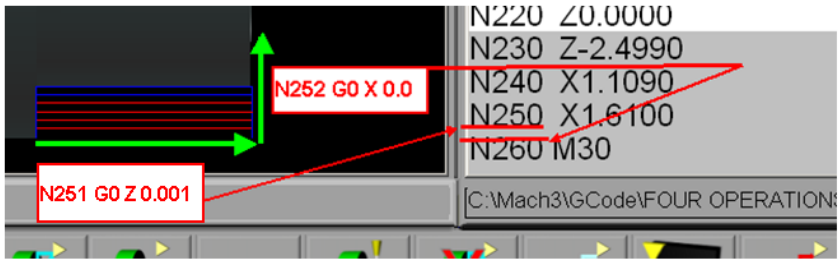
The New command will invoke a flyout which will allow you to define a boarder and LC creates a chain for it. The border can be the size the plate to be engraved.
NOTE: The NEW command can be rather "flacky" at times. It's very easy to delete the chain and start over should there be a display problem.

And then move the chain for the smaller triangle to the A1 OUT Off layer.
Save before you do this!
You may get an unrecoverable error and get kicked out LC doing it this way!



LATHE STUFF

NOTE: *LC will automatically add a move (rapid) from one object to the next or between operations. The moves will diagonally cut through your piece" sometimes. This is not the case as shown here. Simple editing of the Gcode may be required (ie; change the single line so the X move is first and the add the Z move next on a separate line). Some users consider this a bug, but, you can use the "bug" since the rapid "stands out" so to speak. In more advanced programs, you have the option to specify where you want to move and you need to define the point (so use / think of the bug as a more advanced feature asking you where you want to move to!)*



T5 – FIGURE 31

Any changes to the Gcode in MACH3 Turn can be viewed in LC by clicking the MACH3 Load button. LC will "backplot" the code and you can then view it in LC. How good the back plotting will work depends a lot on the extents of the manual changes and how important the information was. Let's just say that it is "buggy".

LC will create pathing based on the tool and cutting parameters.
There is a problem though. LC only allows for defining that the tool is left or right cutting.
There is no center cutting option.

Create the paths by clicking the create profiles button. The rough and finish paths are shown below. Note: I used a clearance of 0.0001" because on occasion in LC when using a zero clearance it doesn't generate a complete finish pass. Don't think that a tenth thou error will cause any problems from a practical point of view!

There is no way to change the default setting and the lead location is based on how the file was automatically optimized.

There is no setting which will slow the simulation down.

The rapid to a point on a contour can be changed.
If the rapid cutting point is changed you may not be able to ooops your way back to the beginning.

Changing the cut direction is buggy in LC as described above. You will find that the cutting direction will change, but, you need to go through a sequence to get it to change. There should be a know default setting, and any time the "switch" is checked, the cutting direction is reversed compared to the prior direction. This can be very annoying,