## Introduction

During a design of a PCB it can sometimes be necessary to rotate groups of routed components to better utilise the board area. There is a specific procedure for this to work because the basic rotation function uses the symbol origin as the rotation point. This is fine when working on one component but if you have multiple symbols plus their associated routing you will need to define a user pick for the rotation point.

Group Rotate in PCB Editor.

To rotate a group of components, clines and vias in PCB Editor you need to use the Move command.
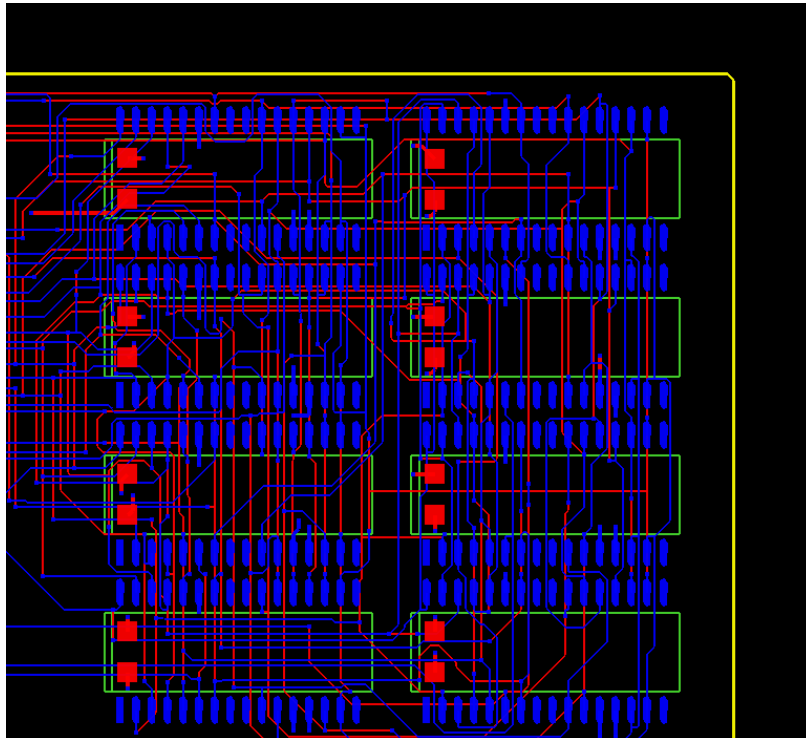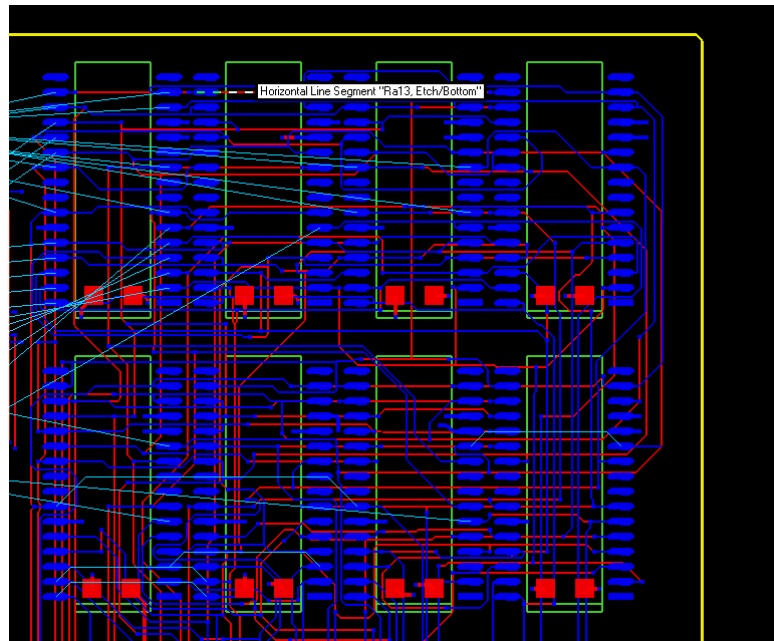

Go to Edit > Move, ensure that the find filter has all the items selected that you wish to move, then move the mouse into the working area and right click and select Temp Group. Then using a left click draw a window around the group of items you wish to rotate. Use a left click to finish the window. All the items will highlight. You can if you wish use the CTRL + left click to de-select items that you may have selected by mistake, and then right click and select Complete.

The command window is now prompting for the user to pick an origin. Go to the options pane and change the **origin point** from **symbol origin** to **user pick** (<u>**this is important for this process to work correctly**</u>). Then go to the working area and left click to select your origin.
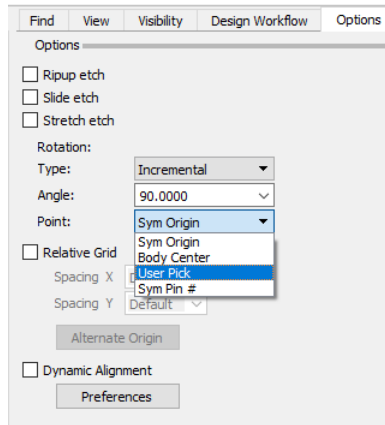
Your group of items are now attached to the mouse cursor. Select right click then choose rotate, the group of components will rotate around your origin point. Left click to confirm your rotation, then left click again to place the group into its new position.