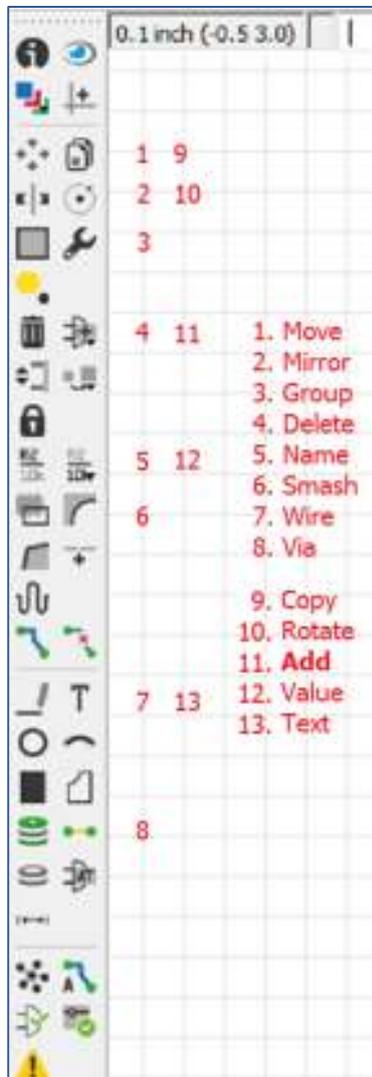


Notes on the Cadsoft Eagle Symbols

We will start with the column on the extreme left and then do the one next to it.

In the picture below I have numbered the symbols that we will need for the Instructable. Beside the picture is an explanation of each symbol. For easy explanation we will call the component side "top" and the wiring side "bottom":



- 1 Move:** Enables movement of components, wires and groups. Right-click to rotate the component/wire/group 90°.
- 2 Mirror:** Converts the component/wire/group into a mirror image. Enables moving items from the top to the bottom of the PCB.
- 3 Group:** Click and drag diagonally across a group of components to create a group. All wires within the area selected will also be part of the group.
- 4 Delete:** Used to delete component/wire/group.
- 5 Name:** Used for naming the components e.g. R1, R2, C1, C2, Q1, Q2, IC1, etc.
- 6 Smash:** Separates the Name (#5) and the Value (#12) from the component so that they can be moved and/or rotated (#1) before being relocated for easy reading.
- 7 Wire:** For drawing the wires. Use the colour menu (that appears below the main menu) for selection of blue (for the bottom wires) and red (for the top wires i.e. Jumpers).
- 8 Via:** Through holes for connecting wires from the bottom wiring to jumpers on top
- 9 Copy:** Makes a duplicate of any component/wire/group
- 10 Rotate:** Same function as "right-click Move" (#1 above)
- 11 Add:** Helps to add components from the library to the PCB
- 12 Value:** Used to enter the value of the component e.g. 10K, 100K, 0.1uF, 10uF, 2N3904, LM386, etc.
- 13 Text:** Add text to the PCB. Use the same colour menu as in #7 to change to red.

Notes

Activating a Symbol: Activate the required symbol by clicking on it. Then click on the item on which you want the action taken e.g. if you want to delete a component, click on the Delete symbol and then on the item you want deleted. If it is a component, click on the "+" mark on the component to avoid e.g. moving or deleting the wrong component or wire. You can, of course, always **Undo** if you make a mistake.

Taking action on a Group: To take action on a group, right click anywhere outside the group. A small window will appear with the action name e.g. Move, Delete, Rotate, etc. depending on the selected function. Click on the window to complete the action.

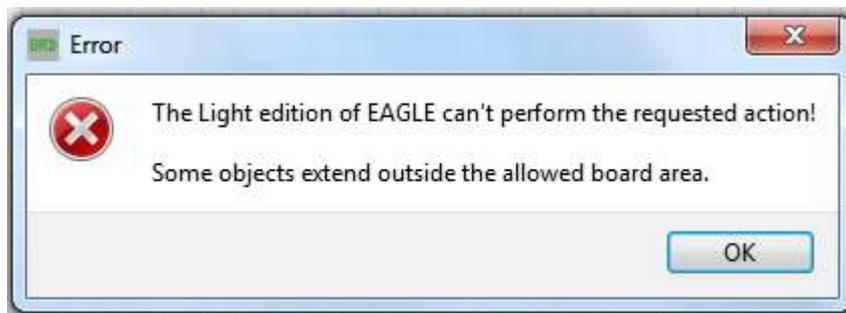
Sub Menus: When you click on some of the symbols, a sub menu will appear on the top-left below the main menu. I will describe below some of the items we will be using in some of these menus.

Notes on drawing Wires: When you click on the Wire symbol, the sub menu will have (from left to right):

- **Wire colour drop down menu:** For the single sided PCBs we are making, we only need **blue** for the wires at the bottom, but we will also use **red** to draw the jumper wires on the top. If you are drawing e.g. a blue track and you want to add in a jumper, you can change from blue to red by selecting it from the colour menu.
- **Wire angle:** The default is right-angle, but you can select the second symbol to draw a diagonal wire. We won't bother with the other selections and the next menu because we are only drawing right-angle and diagonal lines.
- **Wire Width** (or track thickness): Since we are doing our wiring with single strand copper wire, we just need to draw lines. The thickness is of no importance. Leave it at 0.016".
- **Terminating a track:** After you have finished connecting up all the components that need to be connected to a particular track, terminate it by double-clicking at the last component contact point.

If a message pops up in the midst of laying a track, click "OK" if you are sure you are connecting to the correct point and continue. This usually happens when you are adding another component to a track that already has some components connected to it. You are requested to confirm that the connection is correct.

If you move a component or draw a track outside the 100mm x 80mm (4" x 3.2") limit for the free or Light Edition, you will get the error message below. Click OK and move to an area closer to the main PCB design.



Notes on writing Text: If you are writing any text on your PCB design, it will be just to describe something on your design. It will not appear on the PCB since we are not etching.

The same colour selection menu as for Wire drawing appears when you click on the Text symbol. Choose **red**. Blue will give you a mirror image of your text. You can also change the size of the font by selecting from the **Size** menu.

All these changes can be made to the text only after you have typed your text in the **Text** window and clicked on **OK**.

Correcting Errors

If you start a wire by accident, you can cancel it by hitting the **Esc** key.

If you have completed a connection and you find an error, besides the Delete function, there are several levels of **Undo** to get to a point before the error.

If you have opened a menu e.g. the Text window or the Library (when adding components) and you want to get rid of it quickly, hit the **Esc** key twice.