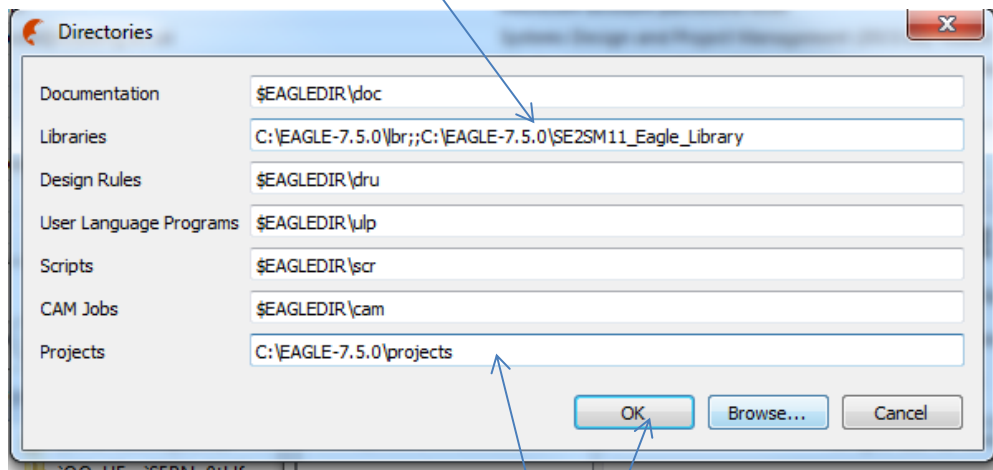


## Using Eagle to Create an Electronic Board.

1. Navigate to Z:\Extra Documents\Cyb Lab Experiments\SE2SM11\Electronic Design Tools
2. Execute eagle-win64-7.5.0.exe Eagle installation file
  - a. Allow the programme to make changes.
  - b. In the WinZip box, click on Setup
  - c. Follow the process
  - d. Accept the C: drive as the destination directory
  - e. Select the “Run as Freeware” licensing option
3. Copy the SE2SM11\_Eagle\_Library directory (as a directory not as individual files) to the C:\EAGLE-7.5.0 directory (where you installed Eagle 7.5).
4. Run Eagle 7.5.0
  - a. Do NOT create any directories at this point if requested – reply No
  - b. From the Control Panel “Options” drop down box select Directories
  - c. In order to make Eagle use the SE2SM11 Library its location must be added to the Libraries list of directories as below:



- d. Add the location where you wish to keep your project files in the Projects box. The default directory (C:\EAGLE-7.5.0\projects) may be used but you must ensure that your files are backed up as the C: drive does get purged (by re-imaging) from time to time, and all user data will be lost. Click on OK when finished.
  - e. Open the Libraries section of the Control Panel by clicking on the small arrow to the left of the word Libraries, right-click the SE2SM11\_Eagle\_Library and select Use All from the drop down menu.
  - f. Eagle 7.5 is now setup ready for use.
5. To start a new project:
    - a. From the Control Panel screen select File – New – Project. It is a good idea to keep all your files for a particular project in one folder, you will create a .sch file for the schematic and from that generate a .brd file for the board layout.
  6. To start a new schematic:

- a. From the Control Panel screen select File – New – Schematic. Note: you will decide where you save the schematic (?sch) when you close the editing session and use the “save-as” option, remember to save it into your designated Projects folder.
- b. Eagle is fairly straight forward once you know what the buttons do, see Figure 1 for the most commonly used, there are more but the labelled ones are sufficient for you to create a schematic and use the schematic to generate the layout of a board:
- c. The basic idea is to ADD components from a library and then NET (wire) them together. From the schematic you generate a board. Use the Create Circuit Error List button Figure 1 to determine if your circuit is satisfactory. Note – Eagle will allow you to build a circuit board layout from an “incomplete – with errors” schematic.
- d. You may decide if you wish to use surface mount technology (SMT) components or through hole components, remembering of course that the processor is a through hole component and 0603 SMT components are difficult to handle as they are so small (1206 or 1210) are more reasonable..
- e. Once you are happy with your schematic use the “create layout from schematic button” as shown in Figure 2
- f. Eagle will not layout the components for you, you need to drag the components from outside to inside the active box (effectively the board you are laying out). The components will be linked by “air wires”, the nets you created in the schematic, between the connected points; you must replace the air wires with tracks. This can be done (lazily and possibly not most effectively) using the Auto Route button from the board toolbox as shown in Figure 3
- g. We are able to manufacture “double sided” boards, that is boards with tracks on the bottom (normal) layer and the top layer of the board, but are unable to “through hole plate” the boards – so there will be no automatic connection between the top layer and the bottom layer of a board. It may be impossible to solder through hole components to pads/tracks on the top layer of a board via their soldered pad on the bottom layer of the board. Surface mount components must have their pads on the top layer of the board but “vias” may be used to route signals from the top layer to the bottom layer, you will then insert wires to create the through hole connection.
- h. Use the Create Board Error List button (Figure 3) to check the layout of your components and tracks. Any air wires left unconverted to tracks should be intentional, i.e. you are going to replace them with physical wires when you populate your board.
- i. Once you are happy the board is ready to print email your .brd file to [s.gould@reading.ac.uk](mailto:s.gould@reading.ac.uk), he will check it and arrange for a time when you can manufacture the board.
- j. Once the board is manufactured it will be returned to you, with components, for drilling and populating. Do not forget to order your components as soon as you have finalised your design.

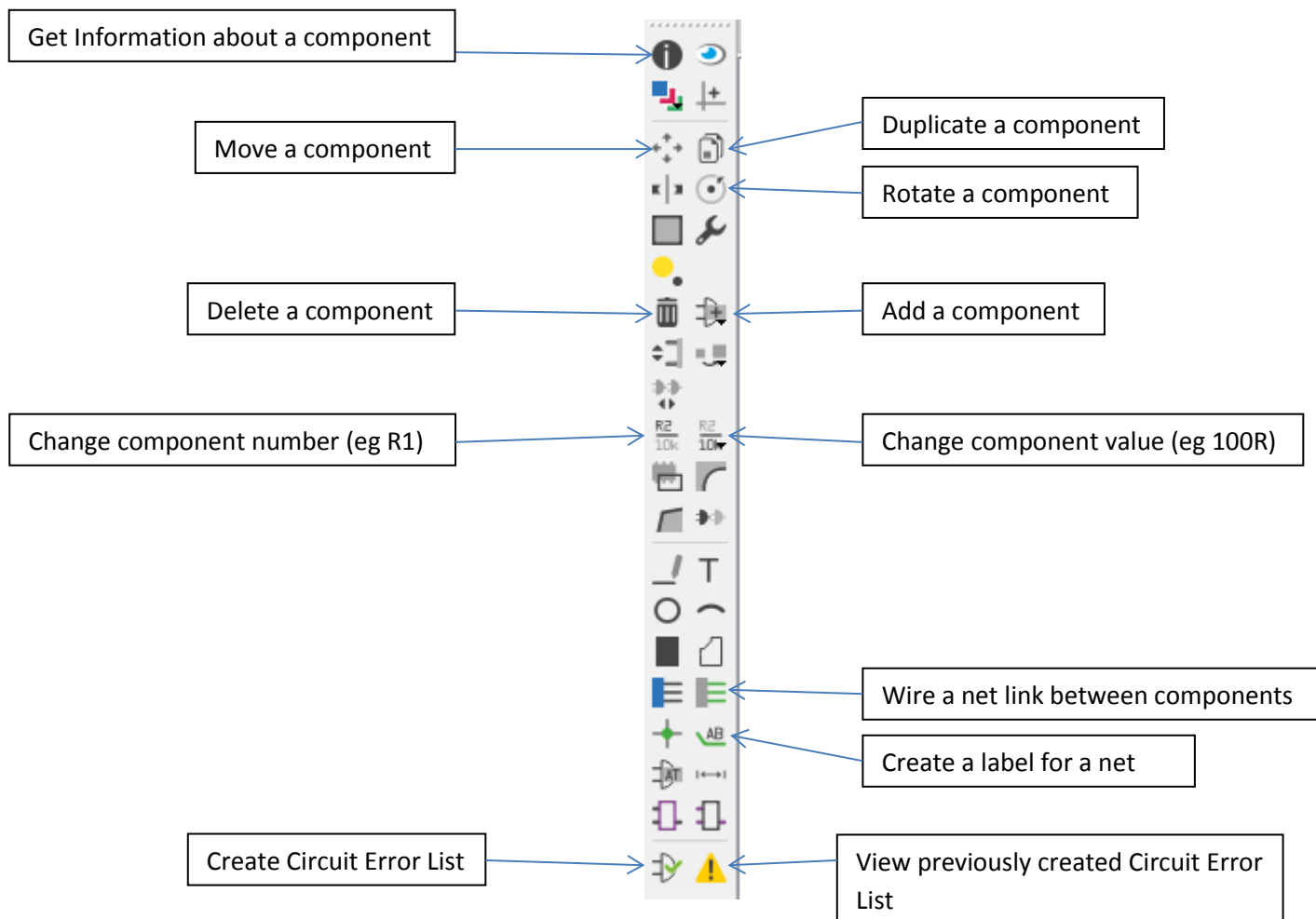


Figure 1: Eagle Toolbox Buttons

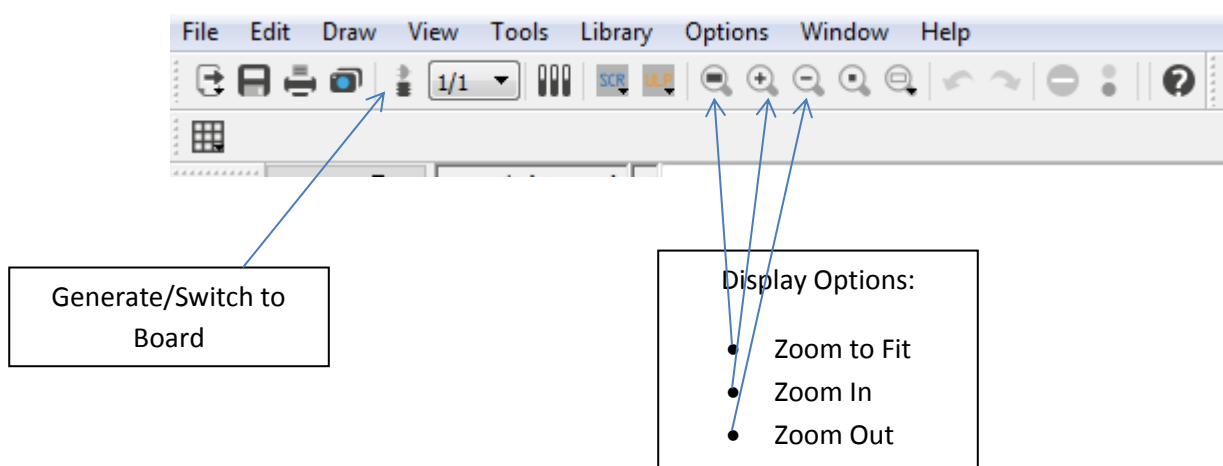


Figure 2: Eagle Headline Commands

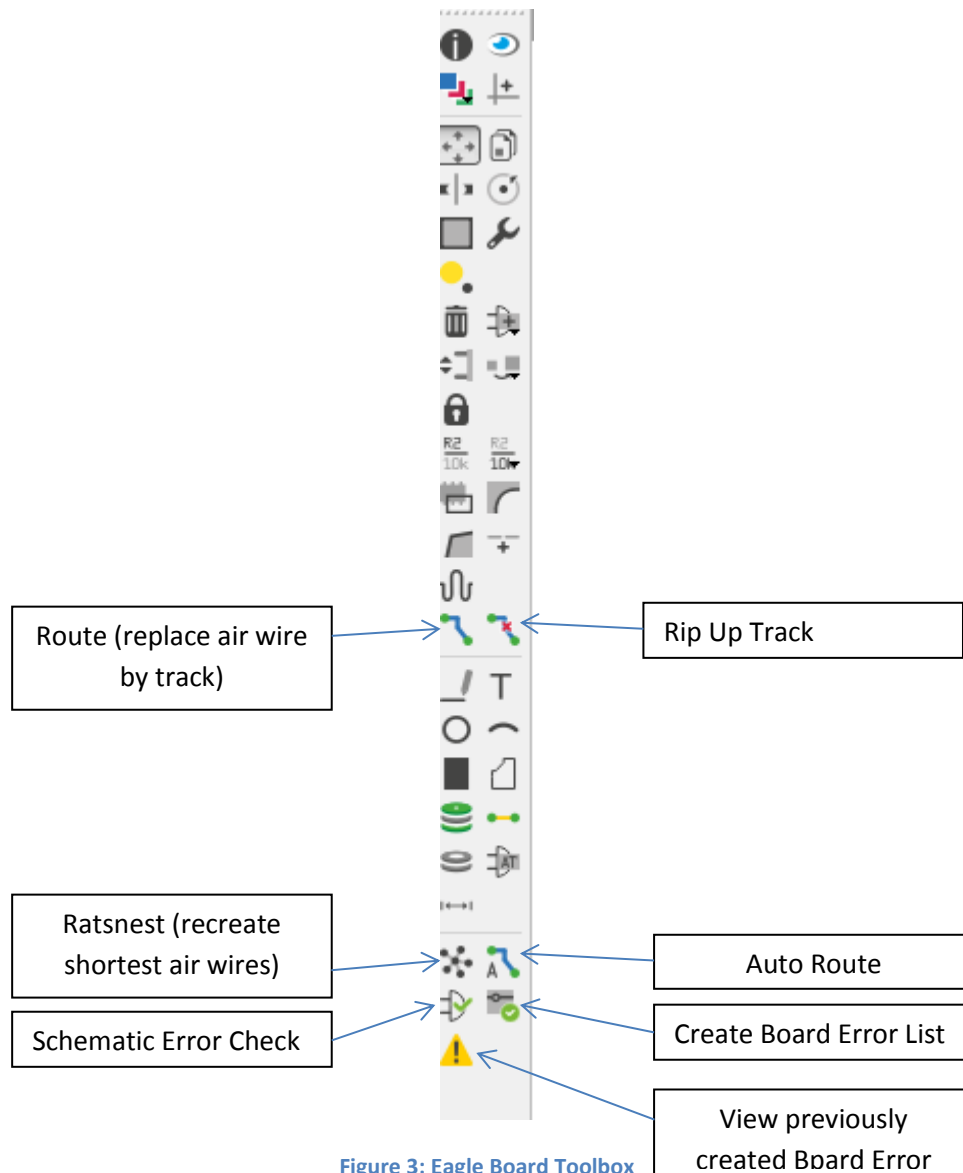


Figure 3: Eagle Board Toolbox