

GSCEM HINTS

Start schematic editor

- Double click on the "Start Xterm" icon on the desktop.
- In the "bash" window that comes up, type gschem (and return) to run the schematic editor.

Save

- File/Save Page to save the currently displayed page. Give the file a .sch file extension.
- Or use the Save button on the toolbar.

Zoom

- Type "z" to zoom in, centred on current cursor position, or "Z" (shift z) to zoom out. The mouse scroll wheel can also be used to zoom in and out.
- Type "ve" (view extents) to zoom to the whole page.

Add Components

- Click on the little gate icon in the toolbar (or menu item Add/Component, or press the "i" key). Select the "Libraries" tab.
- Scroll down to the library you want and click on its little arrow to expand. Then click on the component you want (the symbol is shown in the "Preview" pane). Left click in your main gschem window to place the component.
- Left click again to add more of the same, then press escape (or right click) to end component entry mode.
- Middle mouse button click to rotate a component before placing it (or see below).

Delete Components or Connections

- Select what you want to delete (click on it).
- Press the delete key¹.
- Use undo (control z, upper case U or toolbar icon) to bring it back.

Keyboard shortcuts

- Use the usual ctrl-c and ctrl-v to copy and paste, ctrl-z and ctrl-y to undo and redo.

¹ With the windows version of gschem you need to cancel the "Select Component" window before using keyboard commands such as delete. A quick way to do this is to press the "escape" key a few times.

Move Components

- Select the component (left click on it) and drag the component around using the left mouse button. If you make a mistake, use undo (control z, upper case U or toolbar icon).
- Rotate a component by selecting it (left click) and typing "er" (entity rotate)¹.
- The reference designator (eg R1) can be moved relative to the component or rotated in the same way.
- Move a block of components/wires etc by selecting the block (using the select tool, drag a box around what you want to move). Now you can drag the selected block to where you want.

Printing

- Use the File/print menu. From here, you can print directly or to a postscript file
- You can save to other image formats using the "Write Image" option in the file menu.

Grid

- Components and connections normally snap to the grid. You can change this setting and the grid size under the "Options" menu.
- If grid snap is off, "Snap Off" is displayed on the status bar in the bottom right hand corner, otherwise snap is on.

Connecting up

- Connect up using "Add nets mode", which you can select by clicking on the toolbar icon with the blue lines and red dots.
- You have to connect to the little red ends of the terminals on the components (zoom in until you can see them).
- Right clicking finishes that wire but stays in add nets mode, so you can start another connection.

Name nets (signals)

- Name signals (nets) by selecting the net (left click on it). Then right click and select "Add Attribute". In the box that pops up choose "netname" as the attribute name (usually comes up by default). Type the name you want in the "Value" field. "Visible"

¹ With the current version of gschem under Windows, you need to cancel the "Select Component" window before using keyboard commands such as "er". A quick way to do this is to press the "escape" key a few times.

checked and "Show Value Only" are usually what you want.

- After clicking on OK, move the net name to where you want it (select and drag, as above).
- A net can have more than one name, but it's not recommended.
- Nets with the same name will be connected together in the netlist, even if they are on a different page.

Multiple Pages

- Designs can span multiple pages (.sch files). Each page can be named anything you like (but .sch file extension is recommended).
- Additional pages can be created either within gschem (Page menu) or in separate gschem sessions.

Reference Designators

- All components (except ground symbols) MUST have a unique reference designator (refdes), but they don't have to be consecutive. Components with the same reference designator will be treated as the same component by the netlist generator. (With care, you can use this feature to split a large component into a number of smaller separate symbols.) For PCB design, refdes values can be anything you like, but for spice simulation they have to follow spice conventions (R for resistor etc, see Spice Hints document).
- Several different ways exist to number components (eg R1, R2 etc)
 1. Use "Autonumber text" from the "Attributes" menu in gschem. This has a drop-down list of certain built-in designators (C,Q,R,U and X). You can type in your own as well (eg CONN). Make sure "Search Focus" is set to what you want and "all" or "unnumbered" is set as you want it (often you want "all", to renumber all components, including those which have already been given a number, perhaps incorrectly). This technique won't renumber across multiple pages.
 2. Or, manually number them in gschem. Either double click on the refdes and modify the value, or select the component, right click and choose "Edit...".
 3. Or, on the command line, run "refdes_renum file1.sch file2.sch..." , where file1.sch etc are your schematic pages. (Run refdes_renum without any parameters to get a summary of the options.) You'll probably want to print out your schematic again after running this, since all refdes values are likely to have changed.

Symbol Attributes

- By default, components in the libraries have at least a `refdes` attribute when loaded into gschem. Other attributes may also be present and the user can add more.
- Unless you really know what you are doing, don't delete any attributes from a part freshly loaded from a library.
- To change an existing attribute, double click on it.
- If a symbol has a `"symversion"` attribute, don't change it, or you will get error messages relating to symbol versions.
- To change several attributes of a component and/or to add attributes to it, select the component, right click and click on `"Edit..."`. Click on an existing attribute to change its value. Right click on it for the option to delete it. Select an attribute from the drop down `"Name"` list to add one.
- Attributes have a name (eg `"refdes"`) and a value (eg `"R1"`). You can make both, either or neither of these visible on the schematic page by clicking on the `"Vis?"` (visible), `"N"` (name) and/or `"V"` (value) tick box. Usually you just want the value visible, and only for the `"value"` and `"refdes"` attributes.
- See the Spice Hints for information on required attributes for simulation.
- PCB footprints (`"footprint"` attribute) must correspond exactly to the name of a footprint file.
- There is nothing to stop you having the same attribute name more than once, but this can cause strange errors later on, so check with the `"Edit"` dialogue that this doesn't happen. You can delete an attribute using the menu you get by right clicking on that attribute in the edit dialogue.
- Visible attributes can be dragged, rotated and deleted in the same way as component symbols. Dragging or rotating a component symbol drags or rotates its visible attributes at the same time.

GSchem Options

- Much of the behaviour of gschem can be customised. If you copy the system configuration file to your working directory, your local copy will override the system one:

```
cp /usr/local/share/gEDA/system-gschemrc gschemrc
```

Your copy, `gschemrc` is a text file with many comments explaining the various options. Just edit it to see what you can do and alter it to get what you want. Make sure you delete from around “End of path related keywords” to the end of the file, since that part should not be duplicated.

- Another system file, `system-gafrc`, also in `/usr/share/gEDA/`, controls other aspects of the gEDA suite and can also be copied and customised in a similar way.
- Using the above, your customisations will be applied whenever you invoke `gschem` in that working directory. If you want these customisations to always apply, make yourself a directory called `".gEDA"` (note the dot and the case) in your home directory and put your `gschemrc` and `gafrc` in there. These can still be overridden for a particular project by making a `gschemrc` and/or `gafrc` in your working directory. [NOTE: this will not work in E222.]
- One customisation you may want to make is to tell `gschem` where your symbols are (eg if you make some of your own). Make a file called `"gafrc"` in your working directory containing the following lines (or add them to your existing `gafrc`):

```
(define current-working-directory (getenv "PWD"))
(component-library current-working-directory)
```

then symbols in your working directory will be accessible. Alternatively (or in addition)

add similar `component-library` lines with `current-working-directory` replaced by the path to a directory containing your symbols, eg:

```
(component-library "/pcb/geda/uod-symbols")
```

This way you could keep all your own symbols together in one place.