DRAW ELECTRONIC SCHEMATICS USING LIBREOFFICE

Draw is a powerful vector graphics drawing software. It is part of the free office suites LibreOffice and Apache OpenOffice.

There are a lot of free EDA software solutions but none of them allows users to 'customize' the schematics as they want (for example part colors - outline, background, including high resolution images in the schematic etc.). This tutorial will cover some aspects of drawing schematics in Draw. Similarly, schematics can be drawn in any other graphics software (for example Inkscape, even GIMP and Scribus).
**Before starting**

1. Set up a grid and keep it consistent for all parts. Go to **Tools - Options - (LibreOffice) Draw - Grid**. I used the default 1 cm resolution and 10 spaces subdivision.

2. When you finish drawing a part group together all objects that form it (drag to select, right click and choose Group). In this way you will be able to move the part as a whole and yet you will not lose the ability to edit the object later.

3. When you move a group (which has become a part) you may accidentally alter its size. To prevent this, right click the group, select **Position and Size** and from **Protect** category check **Size**. You will still be able to move and rotate the object, but you can't resize it.

4. If you haven't used Draw before:
   - override the grid (finer) with **Alt+Arrow** keys
   - when rotating snap at every 15 degrees by holding **Shift** while dragging with mouse
   - draw squares and circles by holding down **Shift** while dragging with mouse
Connections

It might be strange to start with this section, but there are a few things to consider when drawing objects in Draw. There are two ways you can make connections:

- draw lines between objects. That's simple but it can cause troubles when moving parts around as the lines don't follow the part.
- draw connectors between objects. Connectors will remain attached when moving an object that they are attached to but as you will see further, not always can be used because of the way Draw assigns connection points.

These tools are available in the **Drawing** toolbar (activate from **View - Toolbars - Drawing**).

From left to right:

- **Polygon** tool - can be used to draw *lines*. Click and hold to draw the first line, then release the left mouse button and change direction. Click wherever you want to change direction again. Double click to end.
- **Connector** tool - the line changes direction by itself. You can adjust it if you want.
To be different than the line tool, a connector should start and end only on connection points. Click and drag to draw. Supports different arrow styles at endpoints.

- **Line tool** - draws a single line. Click and drag to draw. Supports different arrow styles at endpoints.

**Connection points**

Draw puts all objects (except lines) in a rectangle and offers 4 connection points (left, right, top, bottom). But there are parts that have more than 4 pins. Even parts with less than 4 pins can lead to problems. But there are some tricks.

From left to right:

- The four connection points as they are placed on every object
- A part that at first sight cannot be used with the connector tool but
- A rectangle was drawn around the part in such a manner that its connection points will overlay the transistor pins and at last
- The rectangle contour line and background were set to none. The transistor object and rectangle were grouped together.
While this solution may work for transistor symbols when it comes to ICs there's nothing you can do. Once you've grouped the objects that form a part, Draw treats the group as a rectangle. If you don't group you can't move simultaneously without selecting each and every object with Shift key.

Wire intersection
A simple circle filled with black (or wire color if it is other) of 1 mm diameter will be used for that. But the problem is with grid snapping because it snaps with edges and not center. To overcome this some lines should also be drawn with the circle.

Wire intersection

Parts
Here are some examples of objects that form an electronic symbol:
I will not cover the drawing procedure for each part. Here are some guidelines:

- Draw similar components with the same height/size (by similar I mean all two-pin parts, all kind of transistors etc. - see the image below).
- I suggest you shouldn't include part name and value in the symbol group. Instead place text at last after you have completed drawing the schematic.
- Don't forget to group and protect size.
- If you like to draw colorful schematics don't use all available colors. Instead use a consistent color theme (all resistors are the same color, all transistors the same etc.).
- When drawing ICs keep a consistent spacing between pins.
- You can also import images that will be used as components.

Some common parts. Note that all have the same height (25 mm).
Schematic

- Use the connector tool wherever possible.
- Use a consistent font for labels (name, value etc.)
- Save your parts in a file. Put as many as you can on a single page. To start a new schematic make a copy of your template files and adds a new page (slide). Move back to the first slide and Shift-select parts you need to use then do a simple Copy in Slide 1 - Paste in Slide 2. Position parts and make new copies as necessary. Connect the parts. Create a textbox with two lines (name and value) and copy then modify it for every part.

The schematics on this blog are drawn using this method. Here's another example:

LM317 variable power supply. The font used for labels is Liberation Sans Bold 10.

Source: http://onetransistor.blogspot.in/2014/10/draw-electronic-schematics-using.html