

Placing a Border on a Schematic rev 5/3/18

Standard practice on professional schematics and block diagrams is to have a border around the drawing and a title block. The title block identifies the company, the author of the drawing, date drawn, possibly revision history, etc. Companies typically have their preferred or required title block format and contents.

The generic library in Pyxis has a border “component” that can be added to schematics. The boarder can be placed before drawing the circuit or added afterward. For small circuits adding it after drawing the circuit is ok although for larger circuits it may be helpful to place the boarder and draw the circuit inside to make sure it fits.

The title block comes as part of the border and most of the fields will be automatically filled. It is also possible to select items that are properties within the title block and edit them. There is a border component and a large_border component in generic_lib. Larger circuits can also be spread across multiple sheets if needed.

For a new sheet:

Start pyxis, create a project, project library, and cell as needed. Create a new schematic. From the Instance icon on the left tool bar open up the file browser to select a component. Navigate to generic_lib and the top item is border. Select and place that in the schematic drawing window.

For an existing schematic:

Start pyxis and open the schematic. Zoom the circuit out so that it is small (only a couple inches long on the screen). Then select Library from the palette and continue as detailed above. Once the border is visible it can be moved such that the circuit is centered in the border.

The border is a component created out of comment text and with properties attached (i.e. date, author, title, etc.) Thus the select filter needs Instances on to select it.