

KiCad Hierarchy Tutorial x01

This is an early draft of a tutorial, introducing how to use KiCad to draw hierarchical schematics. This tutorial is still in the writing phase. Your comments will be appreciated.

Jonathan Hill, August 2, 2010
jmhill at hartford dot edu

This tutorial uses an example to introduce how to use the hierarchy capabilities with KiCad.

- This tutorial corresponds to KiCad version 2010-05-05
- I am using 32 bit Windows Vista
- I am right handed and use a right handed mouse
- You should know basic operations involving your operating system, such as selecting files and navigating folders or directories using point and click techniques
- You should have an idea how to use KiCad how to draw a simple one page schematic
- This tutorial also includes topics related to simple TTL circuits

The example uses hierarchy to make a ripple carry adder for a pair of three bit binary values.

Create a New Project

First, create a new project. In the KiCad project window use the mouse to select the following

File => New

I am using a memory stick that has been mounted as the E: drive on my laptop computer. With regards to disk accesses, KiCad is fairly light and it works well having a project stored on a memory stick. At one time KiCad required the name of the project folder to be the same as the project file that it contains. Regardless, we will stick with tradition and name the project file and project folder with the same base name.

In the 'Create new Project' pop-up window, navigate and click the 'New Folder' as necessary. I created a folder named 'KiCad', which contains a folder named 'add3'. Navigate inside the add3 folder and in the 'File name field', enter the name 'add3.pro' and click 'Save'. The following path should now be visible in the title bar:

E:\KiCad\add3\add3.pro

Create the Root Schematic

In the KiCad project manager window, left click the EESchema icon to start the schematic capture tool. To cause the pop-up hint to appear for an icon, float the cursor over a button.



'EESchema' icon

The schematic capture tool will open and the 'Info' pop-up window will produce the following report:

'File <E:\KiCad\add3\add3.sch> not found'

Left click 'OK'. Note that KiCad has already named the new schematic add3.sch. This schematic is also referred to as the 'root' schematic as it will represent the top of the hierarchy. Next, it is a good idea to set the page size. Click the Page Settings icon



'Page Settings' icon

In the 'Page Settings' popup window, the 'Size A' is a good choice for smallish schematics like that for this project. In the 'Title' field enter 'Adder circuit' and in the 'Company' field enter your name. Click to select the 'Export to other sheets' buttons and then click OK.

Place a Hierarchical Sheet Symbol Before Drawing the Underlying Schematic

Left click to select the following in the schematic capture window:

File => Save Whole Schematic Project

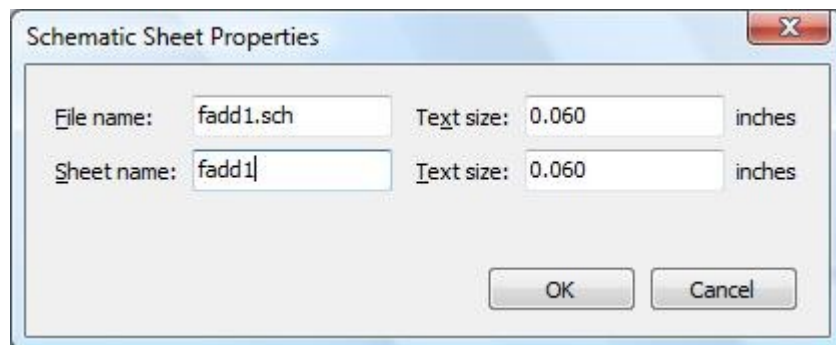
Next, place a hierarchy sheet. Either click the 'Place hierarchical sheet' icon or click to make the following selection:

Place => Hierarchical sheet



'Place hierarchical sheet' icon

In the schematic capture window area, just click left, that is press and depress the left mouse button to start placing the hierarchical sheet. Moving the mouse will scale the size of the box that represents the sheet. Move the mouse so the box is roughly one inch wide by three quarters of an inch tall and left click, so the 'Schematic Sheet Properties' pop-up window appears. Enter 'fadd1.sch' as the 'File name' and 'fadd1' as the 'Sheet name' and then click 'OK' then press the escape key on your keyboard to return to the normal pointing mode.



Schematic Sheet Properties pop-up window

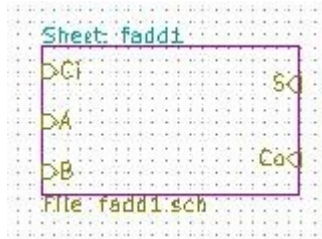
Next, zoom toward the new sheet symbol. To zoom in and out you have some choices:



Zoom In (+) and Zoom Out (-) icons

- Click the Zoom in and Zoom out icons
- Select View=>Zoom In, or View => Zoom Out
- Pressing the F1 and F2 function keys zoom in and out, respectively
- If you have a wheel mouse, the middle wheel will cause the view to zoom in and out

Here we will add pins to the sheet symbol to produce the sheet symbol shown below and then draw the corresponding schematic. In a later example we will use a different technique that is based on an existing schematic.



Sheet symbol for fadd1

Either click left to select the following, or click left on the 'Place hierarchical pin' icon, move the cursor into the sheet symbol, and click left so the 'Pin Sheet Properties' window pops up.

Place => Add Hierarchical Pin to Sheet

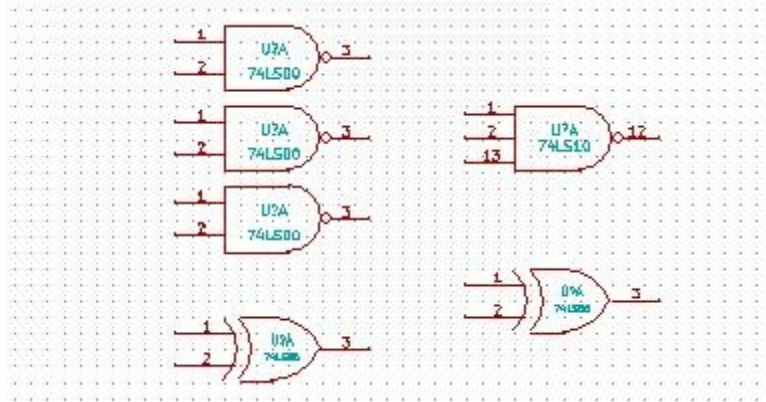


'Place a hierarchical pin' icon

Enter 'Ci' into the 'Text:' field and click the 'Input' button, then click 'OK', then position the new pin to the upper left of the sheet symbol. Likewise, insert input pins for 'A' and 'B' as well as output pins for 'S' and 'Co' as shown above.

Insert Components

Press the escape key to return to the normal point and select mode. To open the new corresponding schematic for fadd1, double click in the fadd1 sheet symbol. With the schematic open, adjust the page size as you did before. We will place components, insert wires, and complete the schematic for fadd1. In the following you will insert gates in a manner like the the following figure.



Full adder components

To insert these components, start by clicking the 'Place a component' icon or make the corresponding selection, move the cursor into the schematic area and click left.

Place => Component



'Place a component' icon

In the 'component selection' window, click 'Select by Browser'. In the 'Library Browser' left pane, click to select '74xx' and in the middle pane, move the slider to find and click '74LS00'. The outline of a NAND gate will appear in the right pane. Click the 'Insert component in schematic' icon.



'Insert component' icon

In the schematic, left click the mouse to place the component. At this point the component designators have not yet been defined and the gates have similar pin numbers. We will address this soon.

Move Components

Move the cursor into the schematic area and click left to place the component. Open the 'component selection' window as before, but this time in the 'History list:' field, click to select '74LS00' and in the schematic place a second and a third 74LS00 gate. Next, place a 74LS10 gate and two 74LS86 gates. Thus far, the schematic should look similar to the figure above.

To move one or more components about, there are several options

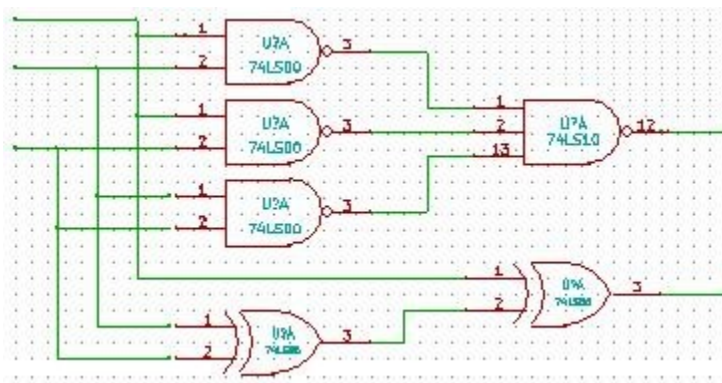
- Point at a line in a component shape and press the 'M' key to *grasp* the component, move the mouse, then left click to *place* the component

- Point at a line in a component shape and then right click, and in the pop-up menu, select the corresponding move command to grasp the component, then left click to place component
- To move one or more components, you can *grasp* the components in a *block*. Imagine a box about the components. Move the cursor to one corner of the box and press the left mouse button, next drag the mouse to the opposite box corner, and then release. Now the selected component move with the cursor. Left click to *place* the components.

Grasping components in a block in that fashion makes provides many options to perform what are called block commands. To start, once the components are grasped, right click to perform a block command. We will investigate component dragging later.

Placing Wires

In the following you will place wires so the schematic will look like the following:



Gates and wires

To place a wire, it is necessary to enter the 'Wiring Mode'.

- With the mouse, select Place => Wire, next position the cursor in the schematic and left click to start placing a wire.
- With the mouse, click the 'Place Wire' icon and then left click to start placing a wire



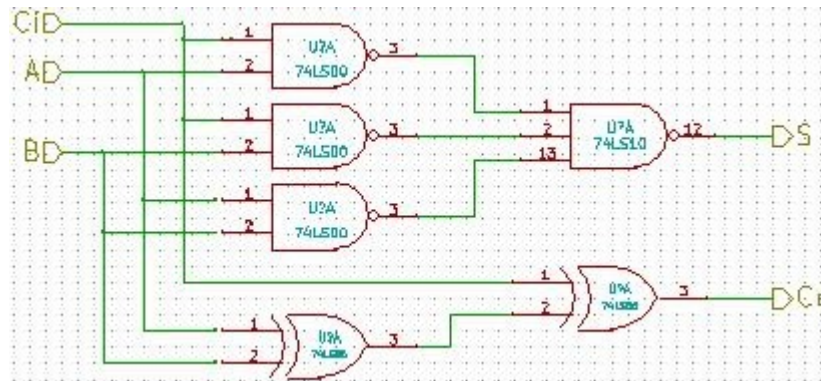
'Place Wire' icon

- Point the cursor at a point in the schematic and press the 'W' key. Unlike the others, this method also starts the actual placement of a wire.

The wire mode inserts horizontal or vertical wire segments. With a wire started, left clicking in space places a wire bend, but left clicking another wire or a component terminal ends the wire. To end a wire in space, either double click, or right click and in the pop-up menu select, 'End Wire'. Right clicking and selecting 'Cancel' prevents the new wire from being inserted in the schematic.

Hierarchical Label

With hierarchical labels the schematic will look like the following:



Components, wires, and labels

The hierarchical pins in a sheet symbol correspond to the hierarchical labels in a schematic. Either click the icon or make the following selection:

Place => Hierarchical Label



'Place Hierarchical Label' icon

Move the cursor into the schematic area and left click. In the pop-up window enter the following:

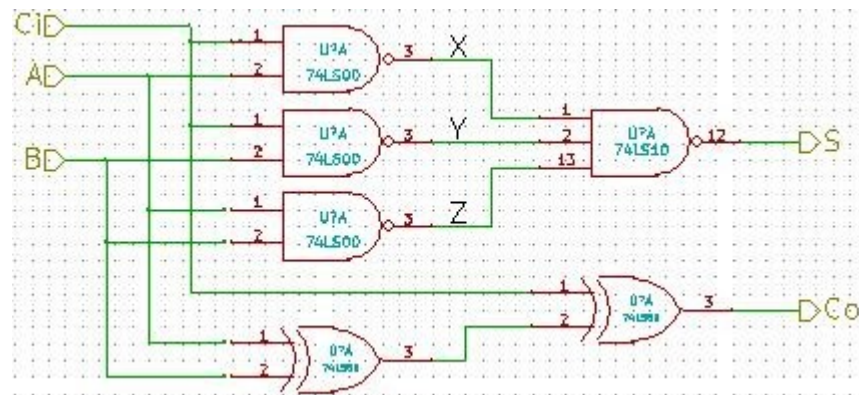
- In the Text field, enter 'Ci'
- In the Size field change the value to 0.120
- In the Style field, click 'Normal'
- In the Orientation field, click 'Right'
- In the Shape field, click 'Input'

Click OK, position the cursor at the upper left wire end and then left click to place the label. Repeat the above steps to place labels for 'A' and 'B'. The 'S' and 'Co' labels are outputs, make the following adjustment to the above steps

- In the Orientation field, click 'Left'
- In the Shape field, click 'Output'

Local Labels

Finally, we can insert labels into a schematic. This makes it easier during a simulation to identify the signals present in the schematic. The schematic will look like the following:



Components and all labels

Click the icon or make the following selection

Place => Label



'Place net name' icon

Left click in the schematic area and in the pop-up window and enter:

- In the Text field, enter 'X'
- In the Size field, enter 0.120
- In the Orientation field, click 'Right'
- In the Style field click 'Normal'

Click OK

Move the cursor to point at upper middle wire as in the figure above and left click to place the label. Repeat the process above to place the Y and Z labels. Press the escape key to exit this tool and return to the normal mode, and then:

File => Save Entire Schematic Project

Hidden Pins, Power, and Global Networks

Some components, such as the logic gates the this schematic have hidden pins for power and ground. To identify these pins, do either of the following:

- Click the 'Show Hidden Pins' icon



Show Hidden Pins icon

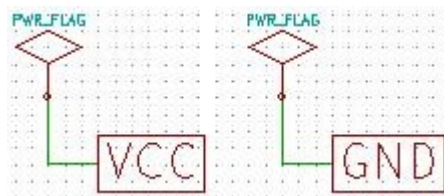
- With the mouse, select Preferences => Options and in the pop-up window click the 'Show hidden pins' button then click OK

Zoom in closer to examine the hidden pins, using the techniques described earlier. Networks in a schematic having the same name are normally connected together. These particular hidden pins are attached to a global network for all the pages in your project, however KiCad is not aware of the significance of these particular global networks. Make a note that the power and ground networks are named VCC and GND. To hide the pins use the steps above to no longer display the hidden pins.

The rules checker needs to be informed of the significance of such networks. There are at least two approaches:

- Use the library editor to create or modify a connector, to indicate that certain pins are of type power in, and connect the ground and power network, each to such a pin. Such a pin indicates that the networks will be driven externally.
- Use flags to indicate that the networks be driven with a power supply. In the following, we use the flags approach.

Move back to the root schematic by moving the cursor into the schematic, right click and select 'Leave Sheet'. The following steps outline how to insert two PWR_FLAG components so that you can attach two bent wires, and two global labels to like the following:



Power Flags

Insert the PWR_FLAG components and wires as described above. To place the global labels, either click the icon or with the mouse select

Place => Global label



'Place Global Label' icon

Next, in the schematic area, left click and in the pop-up window enter:

- In the Text field enter VCC
- In the Size field enter a size, 0.100 or 0.120 is good
- In the Orientation field click 'Left'
- In the Style field click 'Normal'
- In the Shape field click 'Passive'

Press the escape key to exit this tool and return to the normal mode, and then:

File => Save Entire Schematic Project

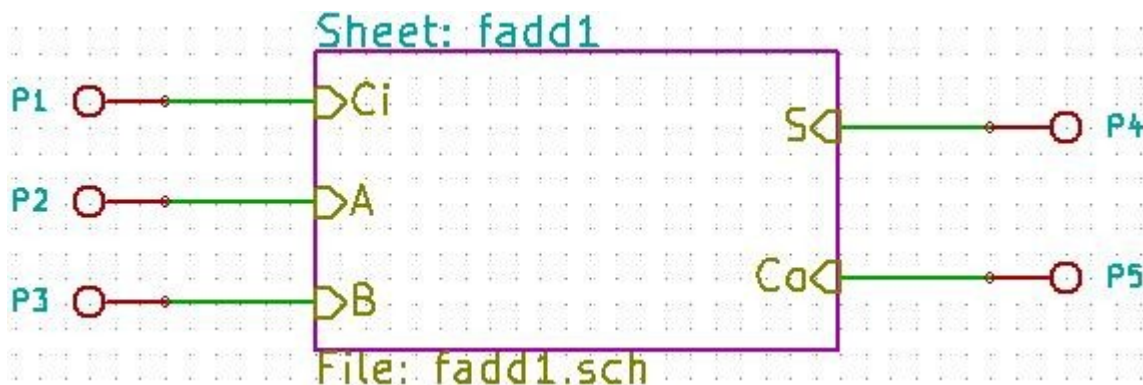
Component designators are an issue in a hierarchical design as each component must have its own unique designator. Component designators may be set manually or may be set by the KiCad tool.

Input-Output Pins

The KiCad rules checker needs to have the inputs driven in a schematic, and a place for the output to go. The schematic for the full-adder uses hierarchical labels that connect to pins in the higher level schematic. The top-level schematic should not use such hierarchical labels as this is the top-level. There are at least two approaches.

- Insert symbols such as signal generators and power supplies. This approach is more common for Spice type simulations.
- Insert connector symbols that allow an external signal generator to provide input

I inserted CONN_1 components from the 'conn' library so that the top level schematic looks like the following. Note that if the rules checker complains that input signal pins are not being driven, then use the symbol editor to open the 'conn' library and edit the CONN_1 symbol. The pin direction should be 'Passive'. It may be enough to simply save the pin and update the library, to satisfy the rules checker.



Top level schematic with a single adder sheet

Producing a Component Symbol

No connect symbol