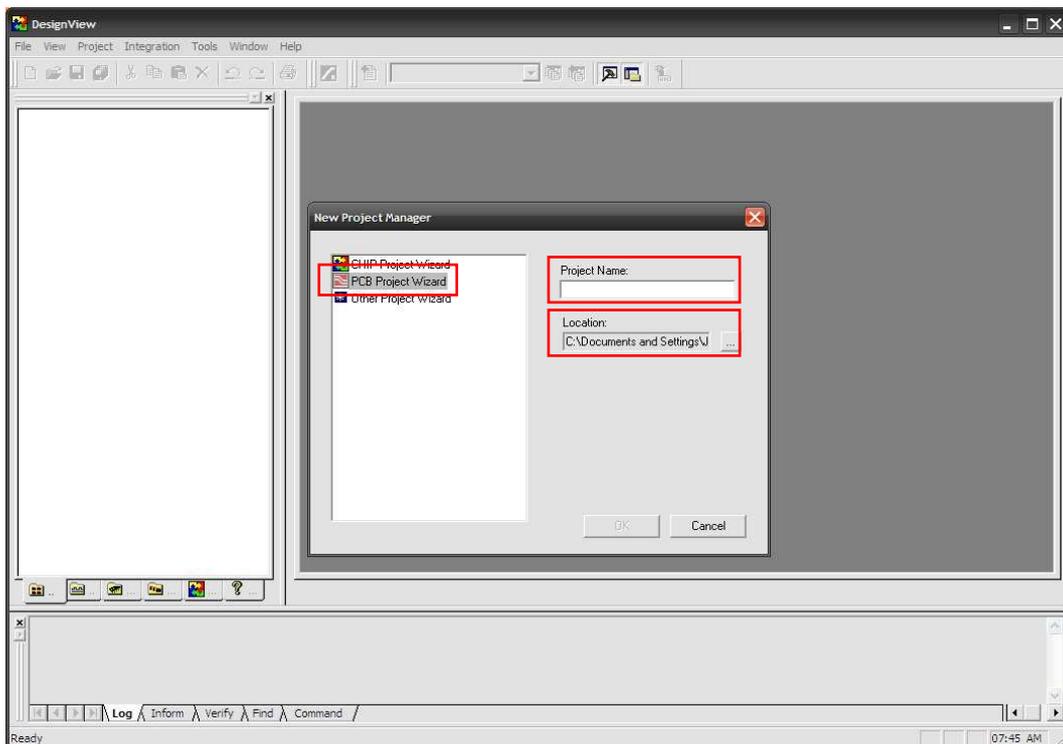


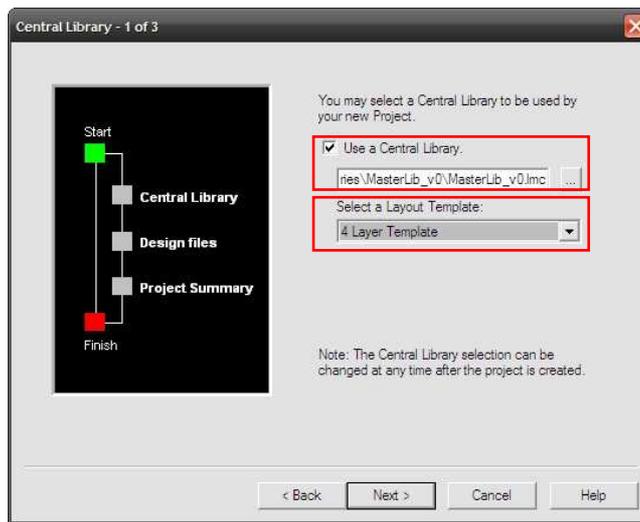
DesignView Schematic Capture Notes

Creating a New Project

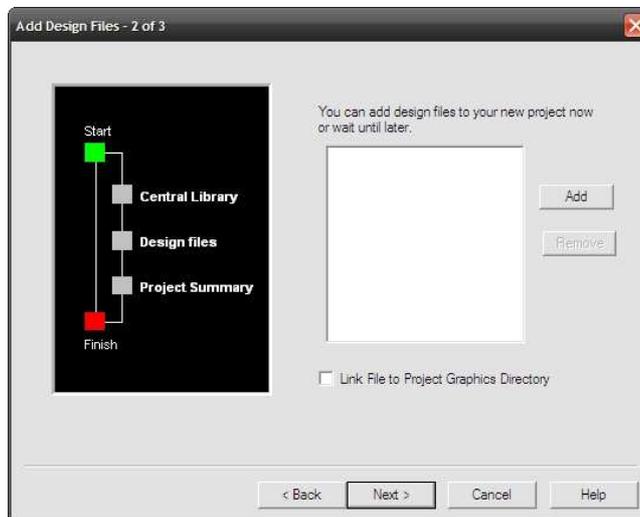
1. Open DesignView and select 'Project > New...' from the menu. In the 'New Project Manager' window, highlight 'PCB Project Wizard' and enter an appropriate top level directory 'Location' for the project files. Enter the desired 'Project Name' (this is automatically appended to the path in the 'Location' box – hence the need to specify a top level directory before setting the name) and click the 'OK' button to launch the project wizard.



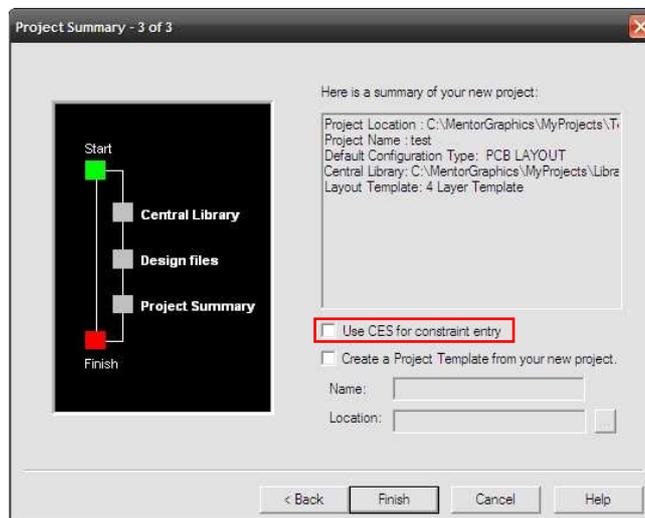
2. On the 'Central Library' dialog, ensure that 'Use a Central Library' is ticked and enter the path to the library file generated by Library Manager. For 'Select a Layout Template' there are only two options: '4 Layer Template' and '8 Layer Template'. At this stage the template selection is not important – it will be defined in earnest once the completed schematic is imported into Expedition PCB for layout. Choose whichever is closest to the planned design and click the 'Next >' button.



3. Ignore the 'Add Design Files' options and just click 'Next >'.

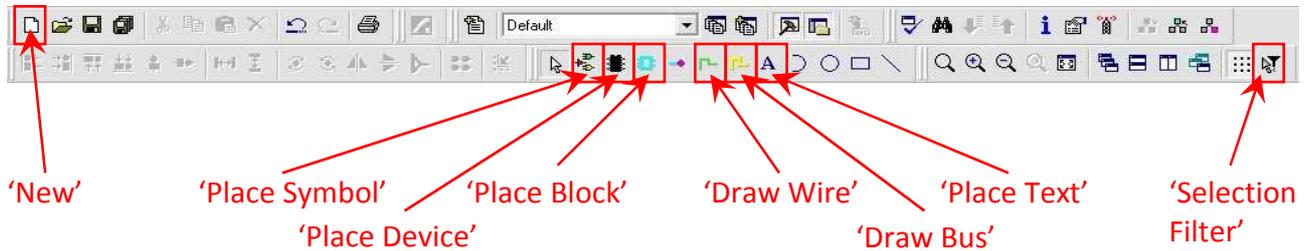


4. In the 'Project Summary' window, ensure that 'Use CES for constraint entry' is unchecked; the Constraints Editor System included with the 2005 release of the Mentor Graphics software package is very buggy and so slow that designs of more than a few hundred nets become entirely impractical. Click the 'Finish' button to complete project initialisation.



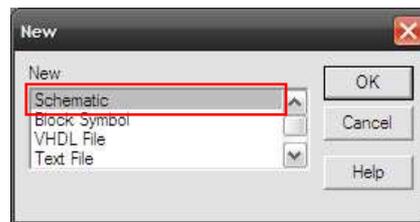
The DesignView Toolbar

The following notes make a number of references to buttons on the DesignView toolbar; these are highlighted in the screenshot below:

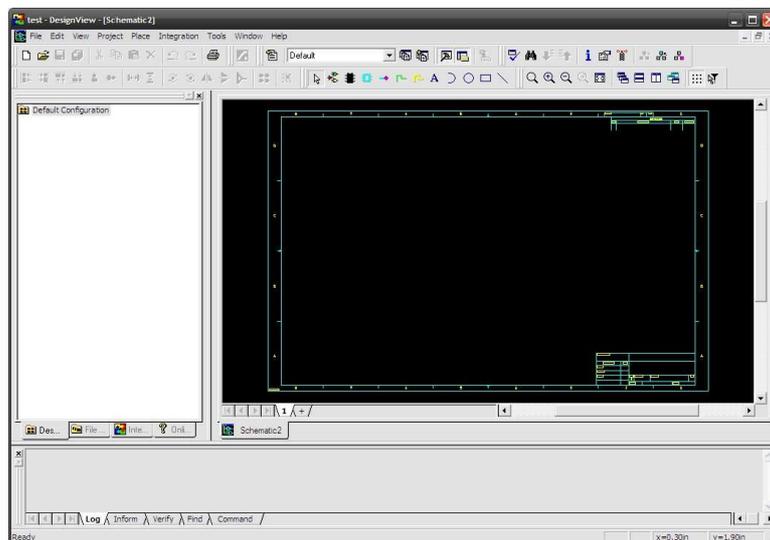


Creating a Schematic

1. Click the 'New' button. In the dialog that opens, highlight the 'Schematic' line and press the 'OK' button.

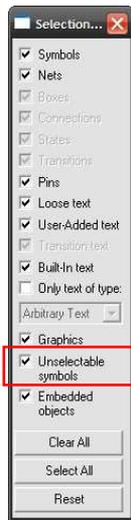


2. DesignView will now show a new blank schematic with the default border frame.

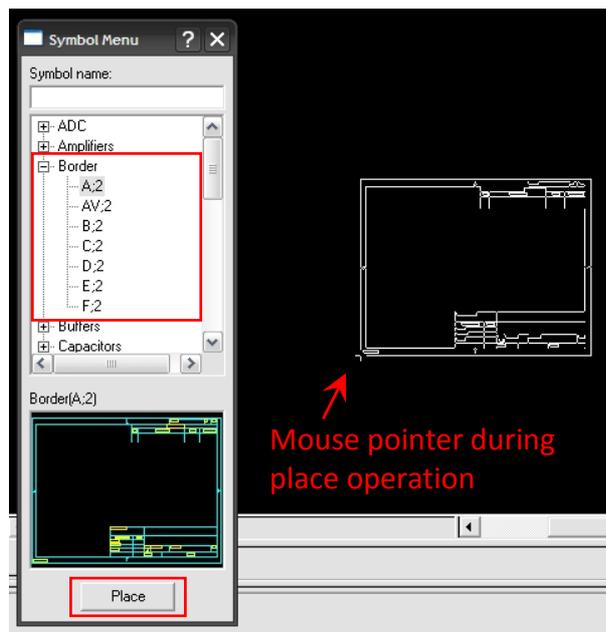


In some cases, it is useful to change the schematic frame (i.e. simple circuits with only a few components should have small frames, in order to minimise empty/wasted space on the schematic page). This may be achieved as follows:

- Click the 'Selection Filter' button
- In the 'Selection Filter' dialog, tick the 'Unselectable symbols' entry



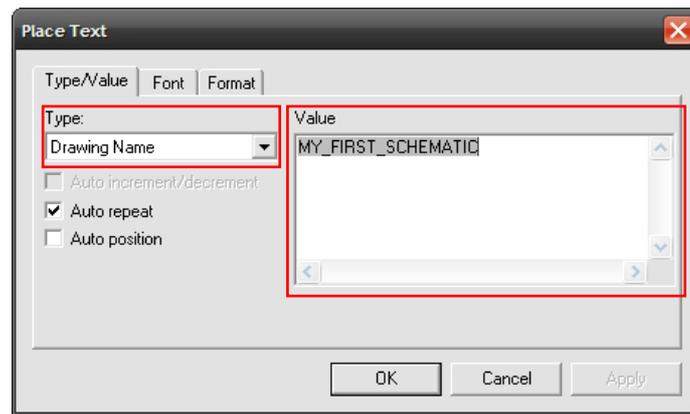
- Highlight the frame in the schematic and press the DELETE key
- Click the 'Place Symbol' button
- In the 'Symbol Menu' dialog, expand the 'Border' entry, highlight the required border type and click the 'Place' button



- The border symbol will be attached to the mouse pointer. Click anywhere on the schematic to place it, then right click and select 'Cancel' to finish
- In the 'Selection Filter' dialog, uncheck the 'Unselectable symbols' entry
- Close the 'Selection Filter' and 'Symbol Menu' dialogs

3. It is good practice to add a 'name' text object to the schematic, to increase readability in multi-page designs:

- Click the 'Selection Filter' button and tick the 'Unselectable symbols' entry
- Click the 'Place Text' button
- In the 'Place Text' dialog, select 'Type' 'Drawing Name' and enter an appropriate name in the 'Value' box

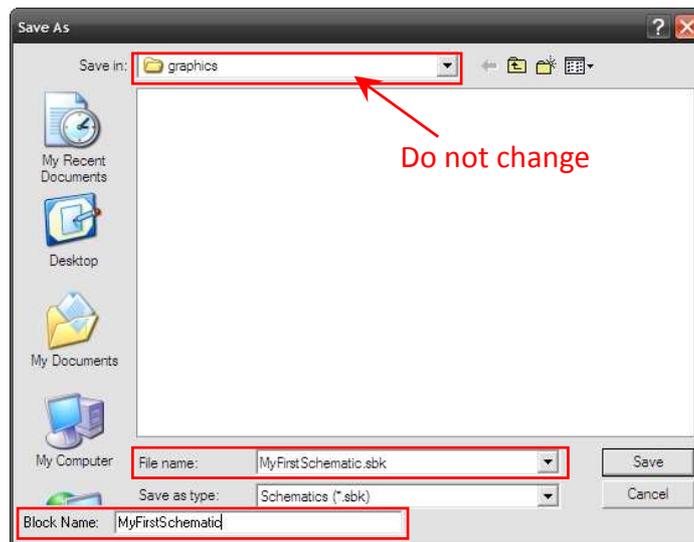


- Click the 'OK' button, then click once on the border symbol (to associate it with the text) and click again on the schematic to place the text

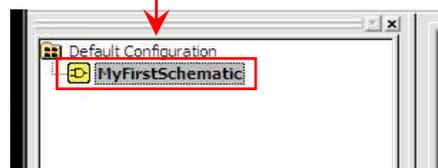
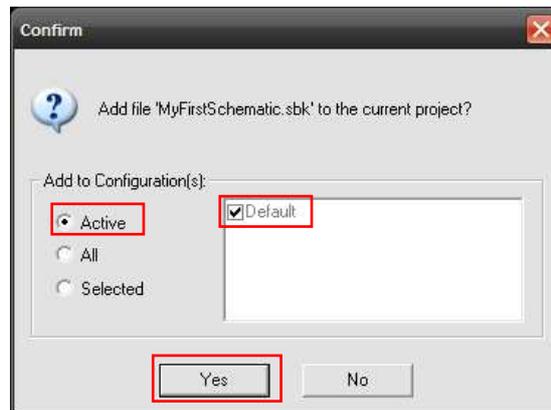


- Right click and select 'Cancel'
- In the 'Selection Filter' dialog, uncheck the 'Unselectable symbols' entry, then close the dialog

4. Save the schematic by selecting 'File > Save As...' from the menu. In the 'Save As' dialog, enter a name for the schematic file and also a 'Block Name'. 'Blocks' will be described in a later section of these notes – however, as a general rule (and in order to reduce the possibility for confusion), the 'Block Name' should be identical to the schematic file name, but without the '.sbk' extension. Always use the default path when saving graphics objects.



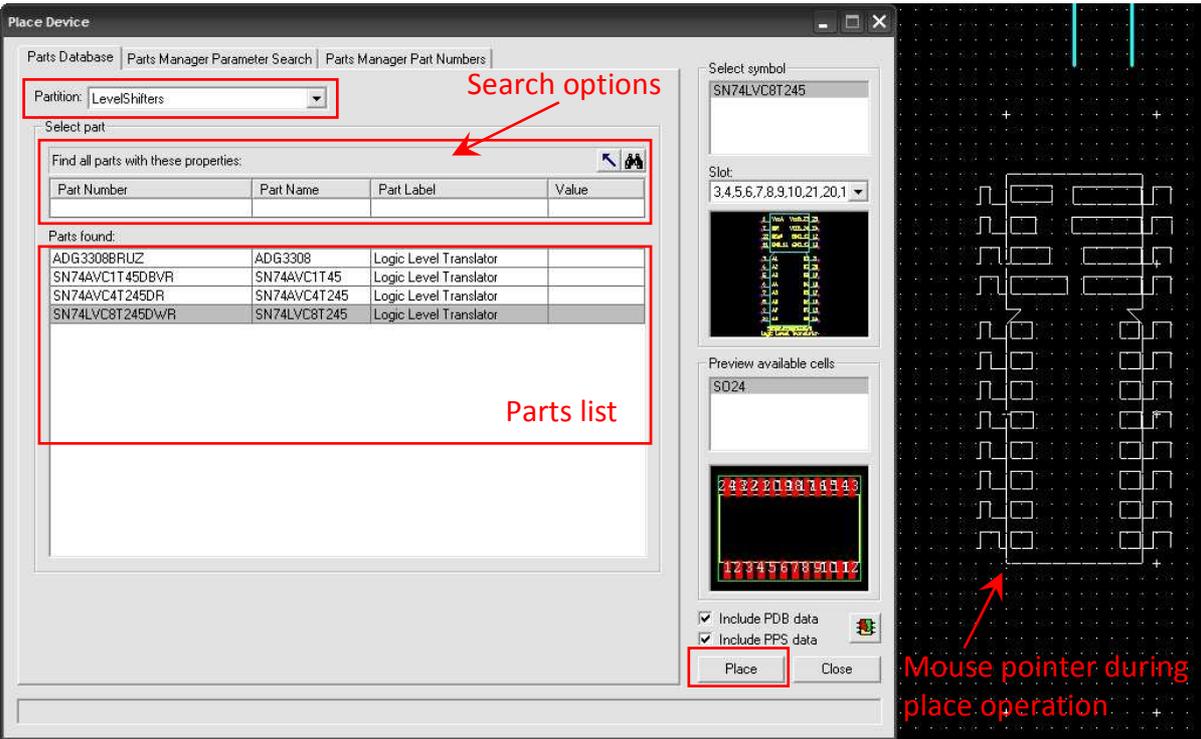
- Once the 'Save' button is clicked, a 'Confirm' dialog will open. Leave the default settings and press the 'Yes' button. The schematic will be added to the project and listed in the 'Design Hierarchy' tab of the Project Manager side panel.



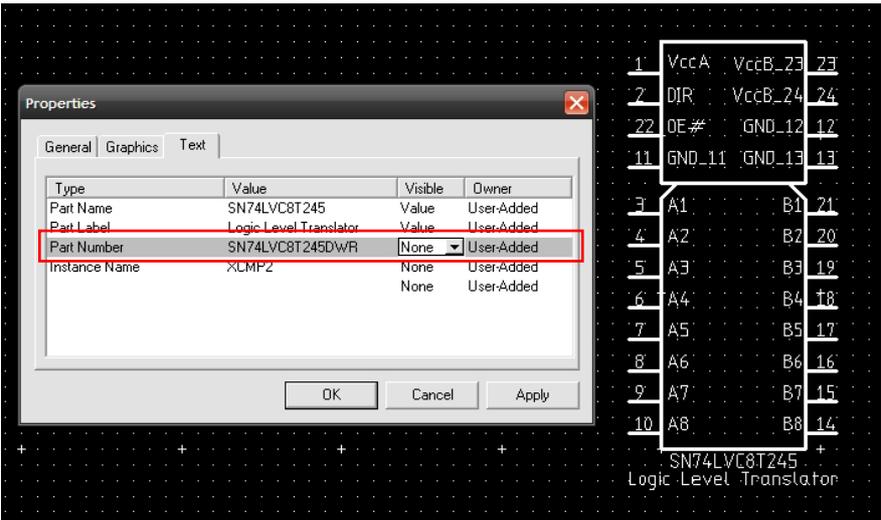
Drawing a Schematic

- The first stage of drawing a schematic is to place the circuit components. Click the 'Place Device' button to open the 'Place Device' window. In the 'Parts Database' tab, select the appropriate partition (i.e. part category) from the Central Library and highlight the required entry in the parts list (the search function is often useful when dealing with very large partitions, e.g. when attempting to locate a resistor of a particular value). When the 'Place' button is pressed, the part symbol will be attached to the mouse pointer – click anywhere on the schematic to drop the symbol (press the R key to rotate it first, if necessary), then right click and select 'Cancel' to leave placement mode. Repeat the process for all required components and close the 'Place Device' window. The orientation of devices that have been

placed on the schematic may be changed by right clicking the symbol and selecting 'Rotate Right' or 'Rotate Left'.

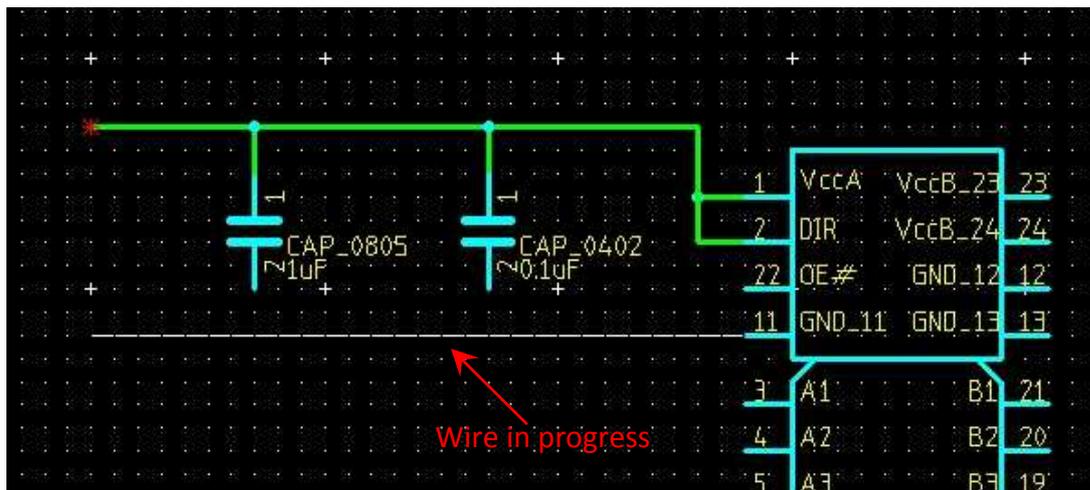


- Each device symbol has associated identification text, some of which is unnecessary at the schematic level. In order to reduce clutter, double click on a symbol to open the 'Properties' window. In the 'Text' tab, change the 'Visible' state for any undesired text to 'None'. Typically, ICs should have visible 'Part Name' and 'Part Label' text, while most passives (resistors, capacitors, etc.) should have 'Part Name' and 'Value' (all components also have a reference designator, but this text is not generated until the schematic is prepped for PCB layout). This technique may also be used to remove unwanted text from symbol pins.

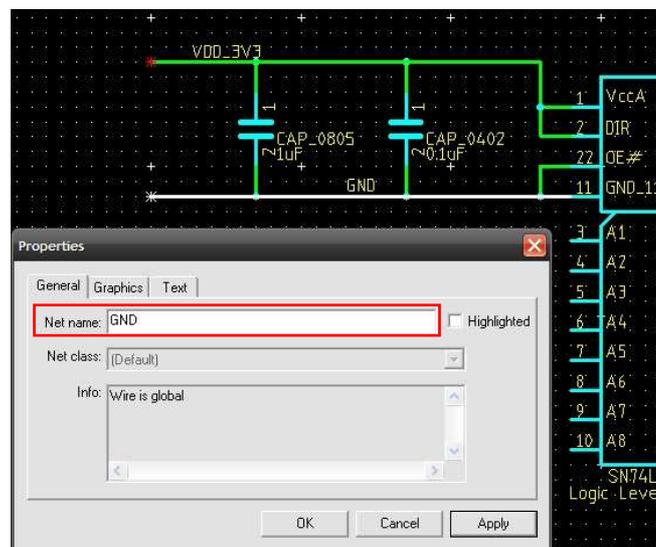


- Once the schematic is populated with symbols, the connections between various components may be defined. Individual signals or voltage levels (referred to as 'nets') are routed between pins using 'wires'. In order to draw a wire, click the 'Draw Wire' button and then click on a pin. Move the mouse pointer to the required location (a second pin, another wire or any free

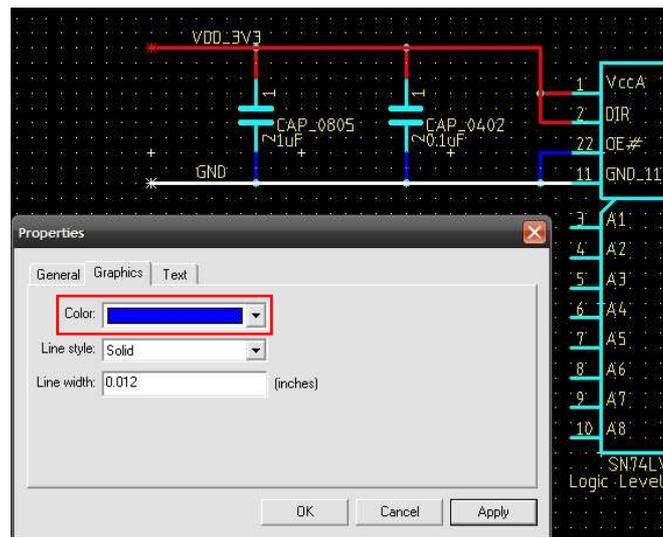
space), and double click to terminate the wire (while moving the mouse pointer, a single click adds a 90° junction). Right click and select 'Cancel' to exit wire placement mode.



- DesignView will automatically assign a net name to each wire. For wires that join two pins on the same schematic page, this is often sufficient. However, if a wire corresponds to a power/ground level or is to be routed to another page of the schematic, manual net assignment is essential. Any nets that require specific constraints at the PCB layout stage must also be named explicitly. This is achieved by double clicking the wire to open the 'Properties' window and entering the net identifier in the 'Net name' box under the 'General' tab.

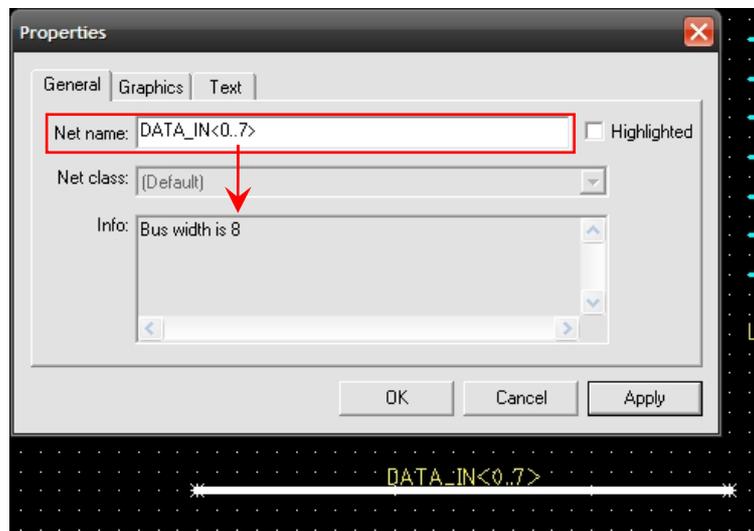


- It is often useful to assign specific colours to wires (e.g. in order to differentiate power/ground and signal nets). The colour selection control can also be found on the wire 'Properties' window, under the 'Graphics' tab.



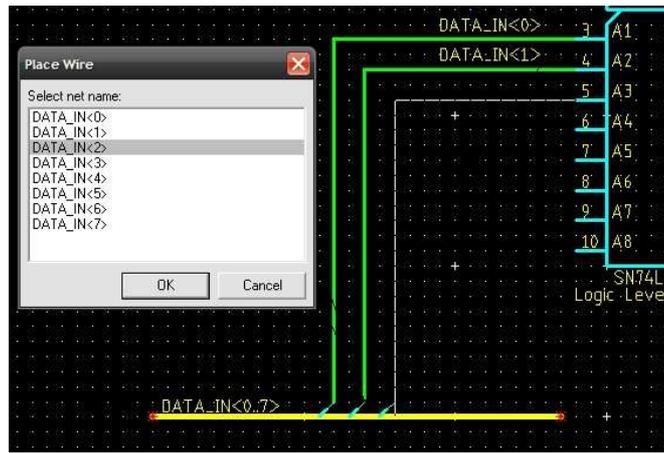
6. When routing certain types of signal between schematic pages (e.g. the various bits of a parallel data stream), it is convenient to group them into a 'bus'. In order to create a bus, click the 'Draw Bus' button, click anywhere on the schematic to start the line and end it with a double click (the controls while drawing are the same as for ordinary wires). The bus cannot be used unless it has been assigned a net name and width; double click the line to open the 'Properties' window and enter a string with the following format in the 'Net name' box under the 'General' tab:

BASE_NAME<FIRST_INDEX..LAST_INDEX>

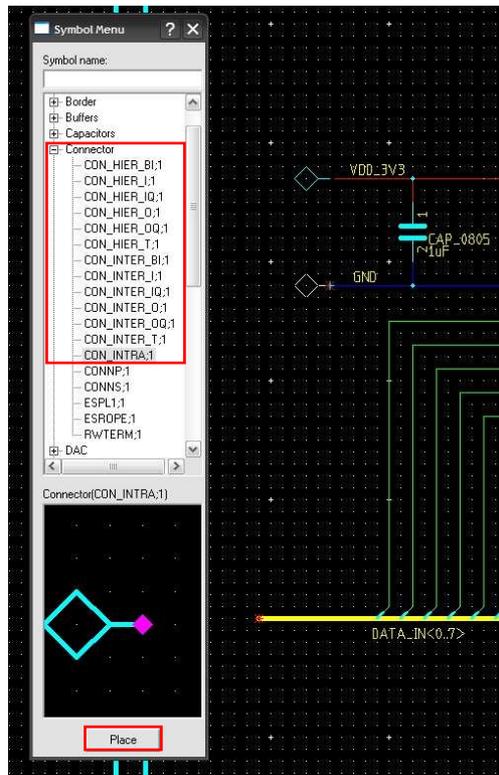


7. In general, buses cannot be attached to the pins on standard device symbols (they may only be attached directly to schematic connectors or 'block' pins – these are described below). Instead it is necessary to fan them out using wires, the procedure for which is as follows:

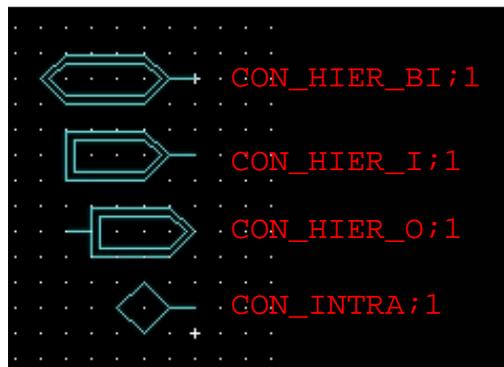
- Click the 'Draw Wire' button
- Click once on the pin of the device that is to be connected (fan out wires *must* start at the component end)
- Draw the wire as usual and terminate it by double clicking the bus
- The 'Place Wire' dialog will open – highlight the net with the required bus index and click the 'OK' button



- Wires and busses that are not attached to symbols at both ends must be terminated with schematic connectors. To add these, click the 'Place Symbol' button to open the 'Symbol Menu'. Expand the 'Connector' entry and highlight a connector of the required type. Press the 'Place' button and click at the end of a wire/bus to drop the symbol (again, the R key may be used to change the orientation of the symbol prior to placement).



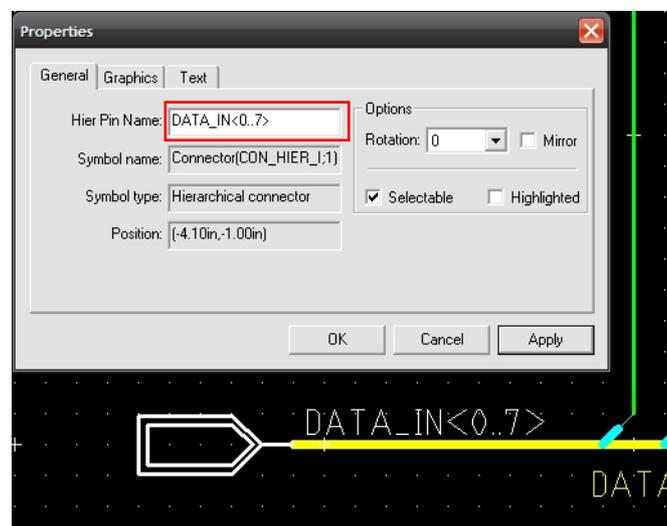
There are a significant number of schematic connectors, but only the following types are required in most standard designs:



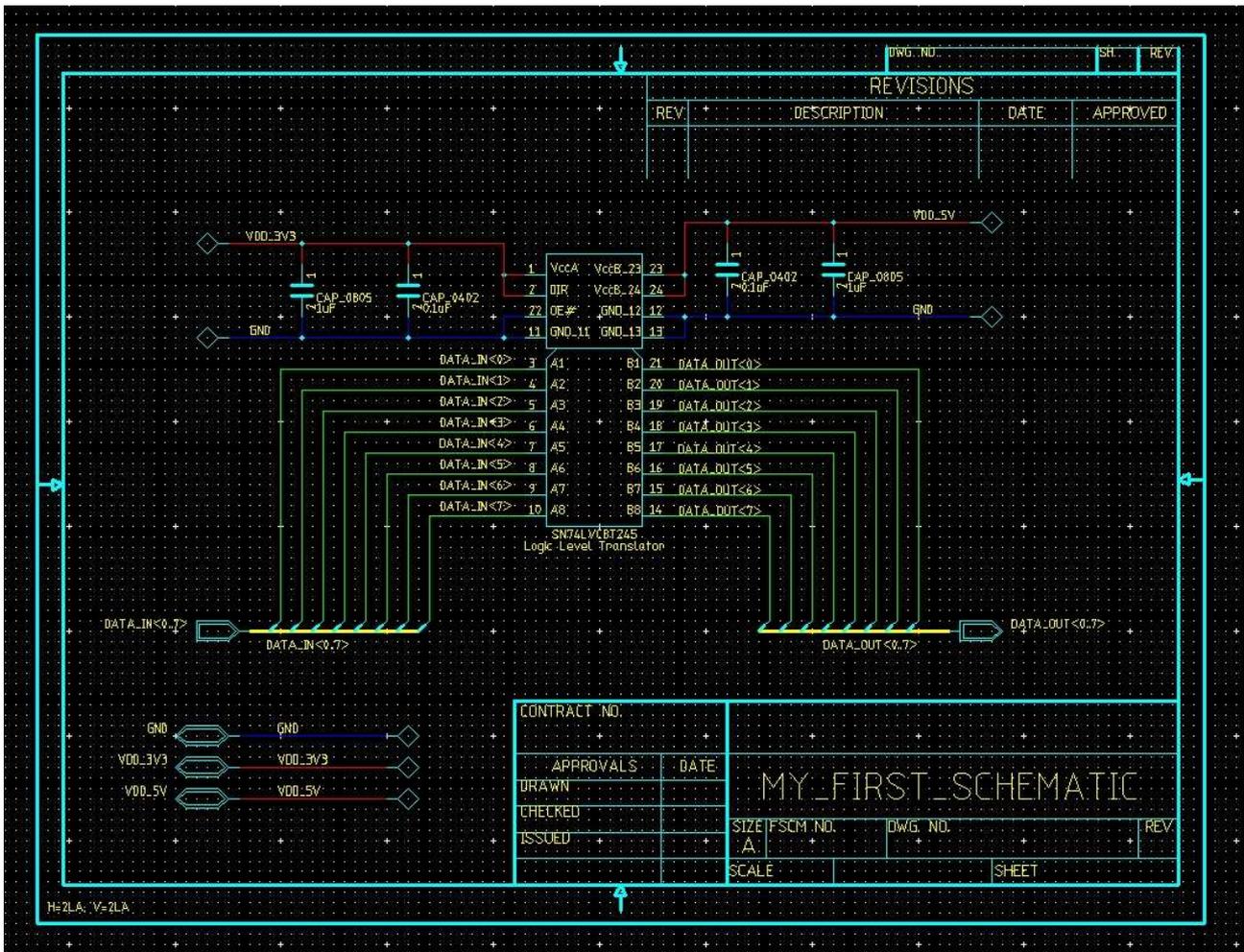
- **CON_HIER_BI;1**: Bidirectional connection between different levels (i.e. different pages) of a schematic
- **CON_HIER_I;1**: Input connection from a different level (page) of the schematic
- **CON_HIER_O;1**: Output connection to a different level (page) of the schematic
- **CON_INTRA;1**: Connection between two wires/busses on a single schematic page

NB: Strictly speaking, the CON_HIER_ family of symbols only connect between different hierarchical levels of a schematic; other types of symbol are used to connect between different pages at the same level. However, since it is typically confusing/bad practice to create 'flat' designs, each level should only consist of a single page anyway – so CON_HIER_* symbols may be treated as general 'inter-page' connectors.*

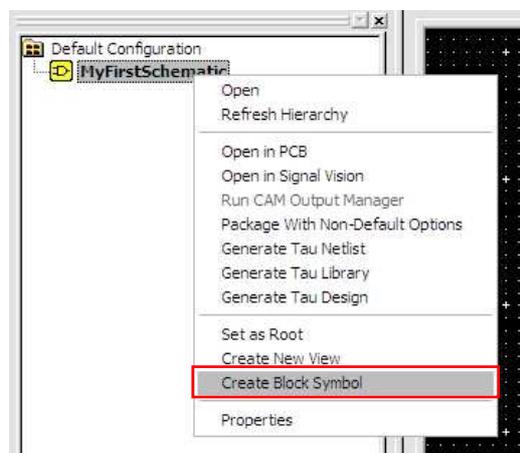
9. All 'inter-page' connectors (CON_HIER_*) must have a name identical to the net of the wire/bus to which they are attached. Double click the symbol to open the 'Properties' window and enter the net name in the 'Hier Pin Name' box under the 'General' tab.



An example (unrealistically minimal) schematic page which demonstrates usage of all the standard connector types is shown in the following screenshot. Note how the 'intra-page' connectors simplify the power/ground wiring layout.



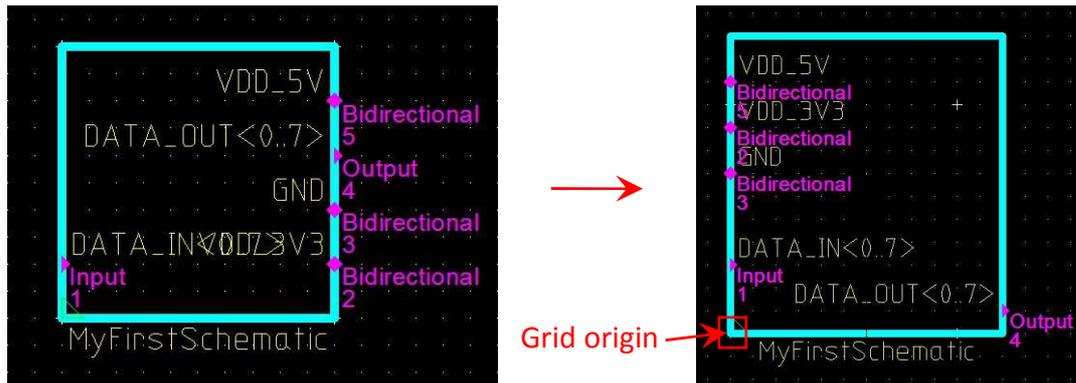
10. All but the simplest designs consist of multiple schematic pages organised in a hierarchy. This allows for an 'object orientated' approach that simplifies architecture management, maximises reusability and increases readability. Any schematic that contains 'inter-page' connectors may be interpreted as a 'block', which can be placed in higher level schematics as though it were an individual component. To create a block symbol representation, right click a schematic in the 'Design Hierarchy' tab of the Project Manager side panel and select 'Create Block Symbol'.



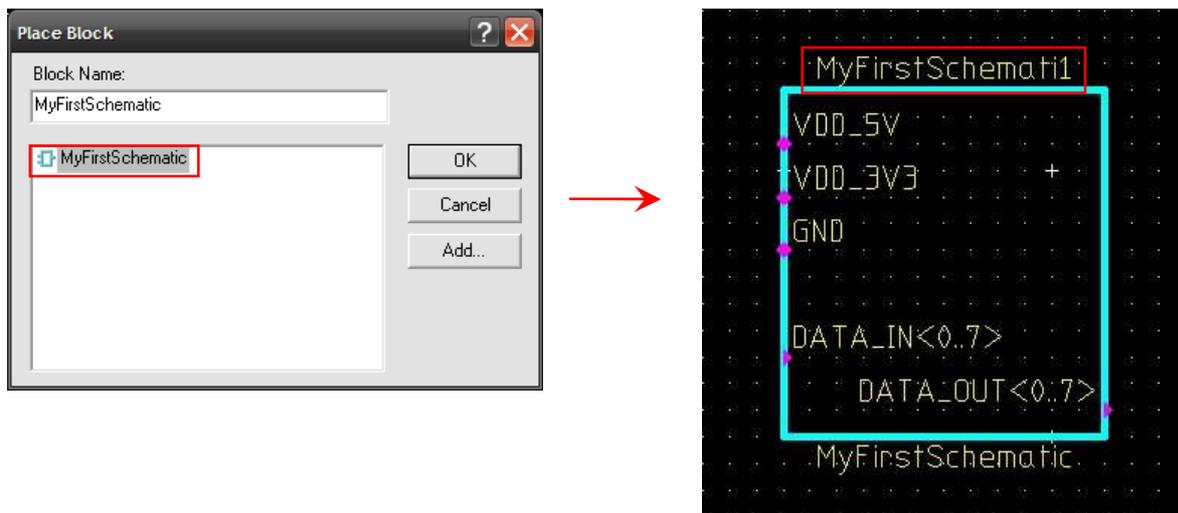
A new editor page will open, containing a simple block symbol with pins matching the 'inter-page' connectors in the underlying schematic. The graphic may be edited in exactly the same way as component symbols in Library Manager, although the pin properties should not be

changed (it is actually possible to modify/add/remove pins at the block level and propagate changes through to the schematic, but it is good practice to always set block interfaces in the schematic itself). There are no specific rules when designing blocks, but in general:

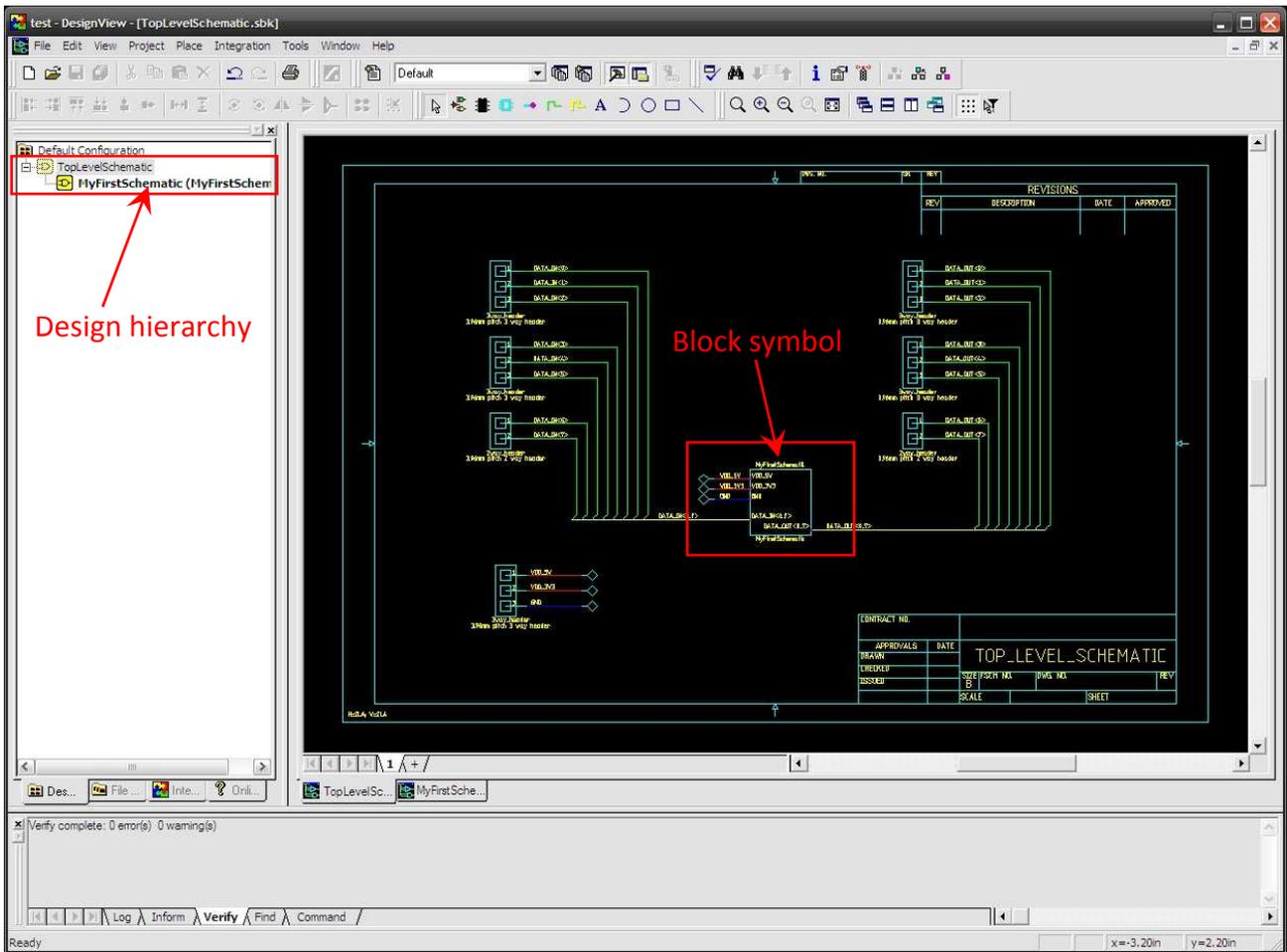
- Place power/ground pins on the upper left hand side of the symbol
- Place input pins on the lower left hand side
- Place output pins on the lower right hand side
- Place the block/schematic name below the symbol
- The grid origin (i.e. the small green triangle) should be at the bottom left hand corner of the symbol



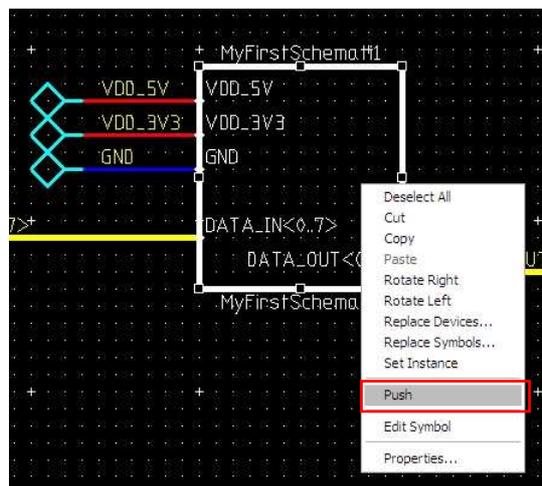
11. To add a block to a higher level schematic, click the 'Place Block' button to open the 'Place Block' dialog. Highlight the name of the required sub-schematic and press the 'OK' button; the block symbol will be attached to the mouse pointer and it may be placed in exactly the same manner as any other symbol. Each instantiated block is automatically given an identifier (a substring of the block name appended with a number), similar to the reference designators associated with physical devices.



As blocks are placed, the 'Design Hierarchy' tab of the Project Manager side panel will automatically change to reflect the structure of the design. A very simple example is given in the following screenshot:



When viewing a page that contains a block, a convenient method for accessing the underlying sub-schematic is to right click the block symbol and select 'Push'.



This concludes the DesignView schematic capture notes. While DesignView has several other important features and configuration options, these all pertain to integration with Expedition PCB for circuit layout operations; they will therefore be covered in the Expedition PCB notes.