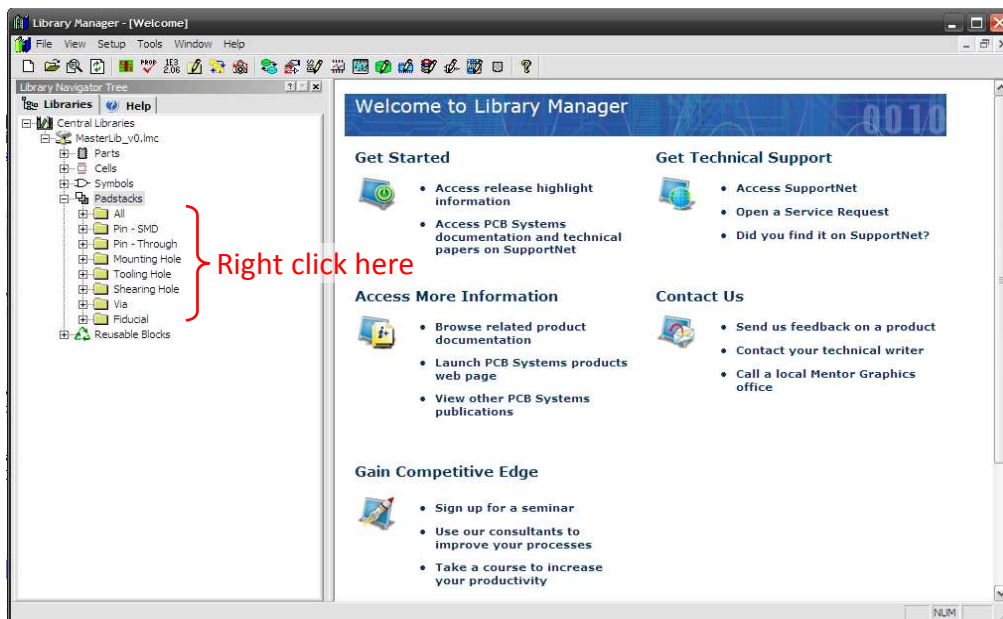# Library Manager Notes

## *Creating a Padstack*

1. Open Central Library:

   → Expand 'Padstacks'
   → Right click entry of required type (SMD, Through, Via, etc.)
   → Select 'New Padstack...'
   → Enter an appropriate name and click 'OK'

2. The Padstack Editor will appear. It is usually necessary to create holes and pads before defining the padstack, but the editor will not allow access to these tabs without first selecting pads for 'Top mount' and 'Bottom mount' (and potentially a hole depending upon the padstack type). Consequently, set the type to 'SMD', select anything for these two surface mount pads and go to the 'Holes' tab (skip to step 4 if creating an SMD padstack)
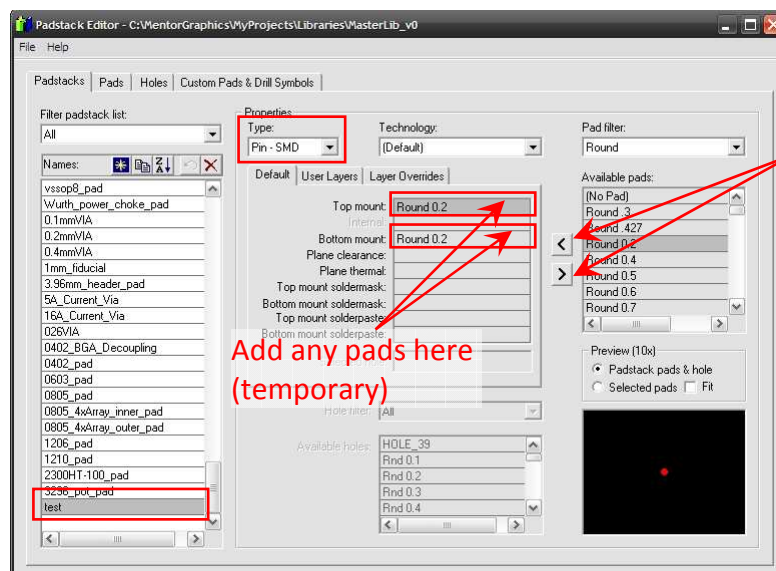
3. On the 'Holes' tab, first make sure that 'Filter units' is set to 'Metric'. Click the 'New Hole' button and a new entry will appear in the list, the name of which will automatically change to reflect the input configuration settings. Leave 'Unit', 'Type' and 'Plated' options at their defaults. It is typically only necessary to specify the hole shape (round, square or slot) and dimensions (tolerances may be ignored). When the hole is complete, go to the 'Pads' tab.
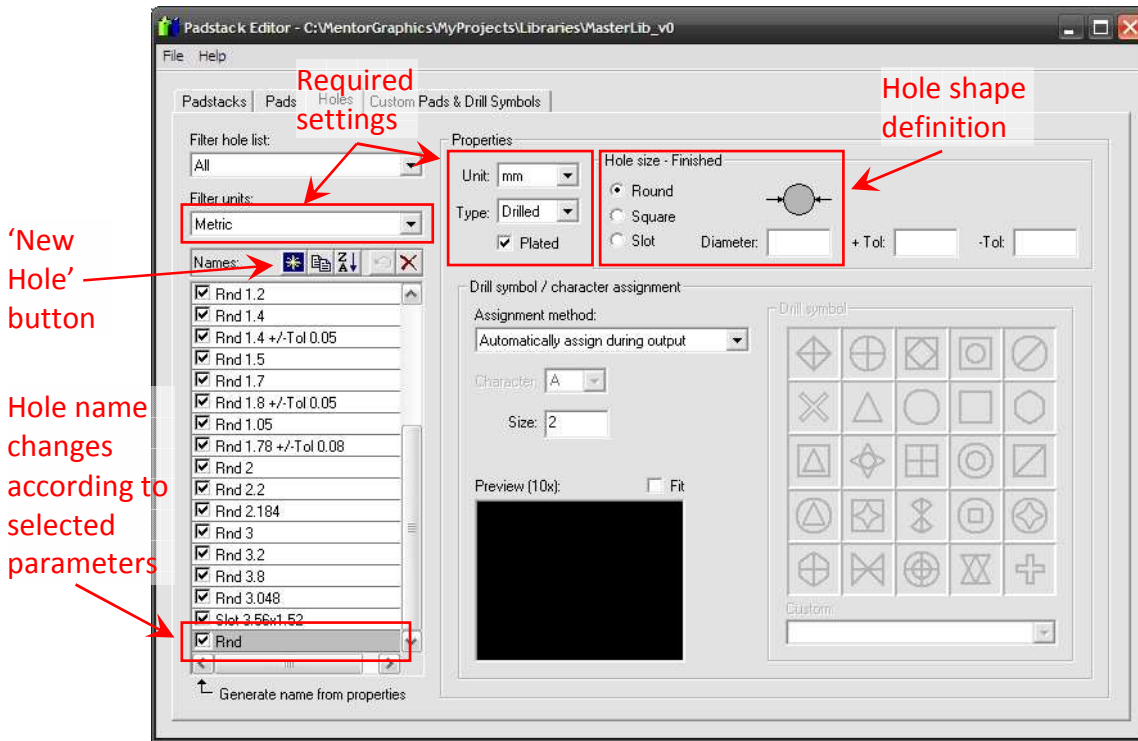
4. On the 'Pads' tab, make sure that 'Filter units' is set to 'Metric'. Click the 'New Pad' button and a new entry will appear in the list, the name of which will again automatically change to reflect the input configuration settings. Leave 'Units' at the default and select a pad type, then enter the pad dimensions. SMD pads are typically square or rectangular (unless dealing with BGAs); through pads are typically round, and require an associated thermal pad (4 Web Round Thermal) and plane clearance pad (Round Donut) – these must be created separately. The 'Pads' tab is also used to define soldermask and solderpaste areas. Once all required pads have been generated, return to the 'Padstacks' tab.
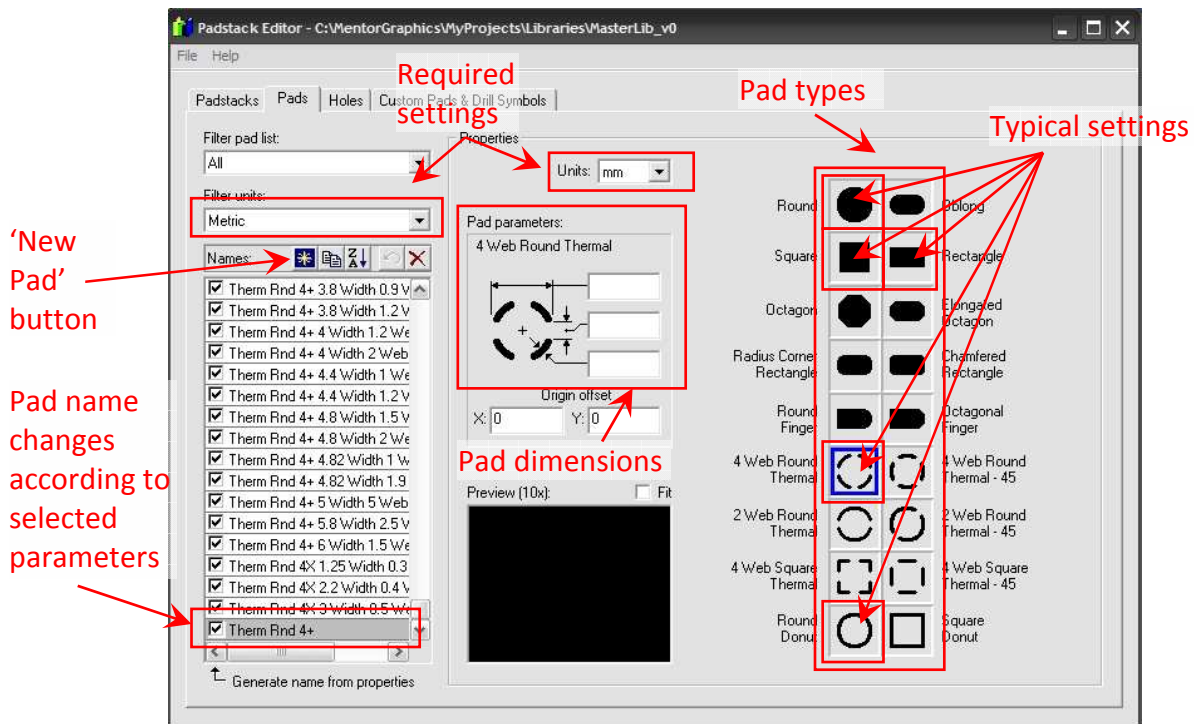
5. On the 'Padstacks' tab, first ensure that the new padstack (with the name defined in step 1) is still selected. Enter the correct type (typically only require 'Pin – SMD', 'Pin – Through', 'Via' and 'Mounting Hole') and then populate the pad list (and choose a hole, if necessary).
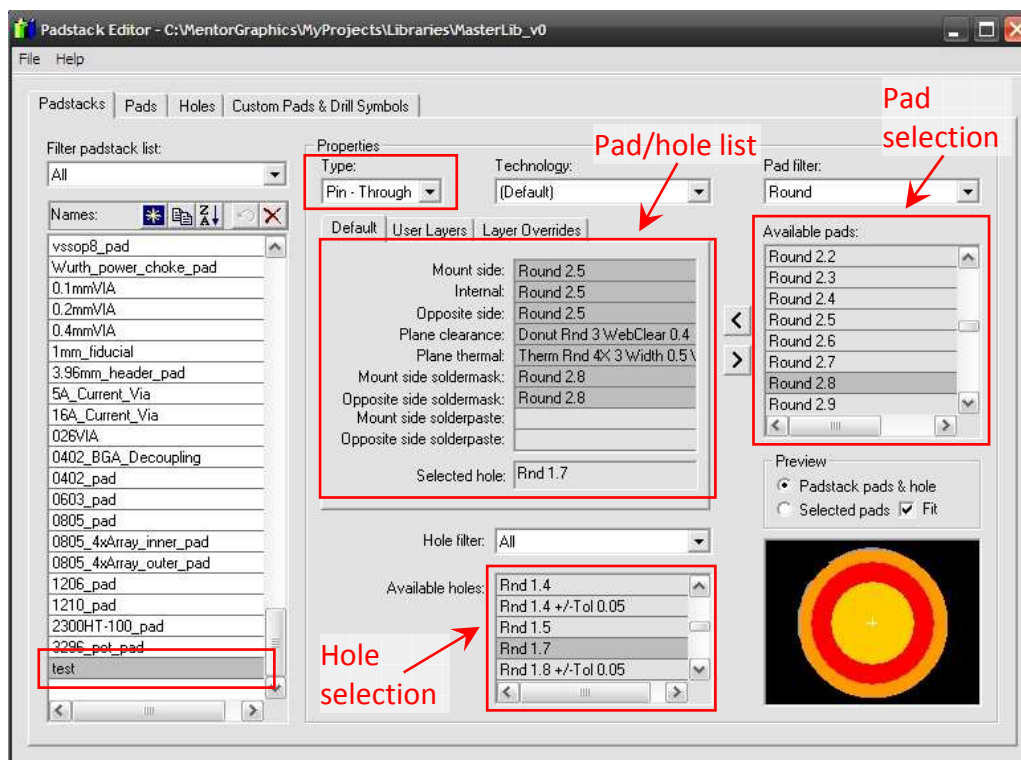
The pad/hole list elements required for each padstack type are as follows:

Mounting Hole:	Plane clearance
	Selected hole

Pin – SMD:	Top mount
	Bottom mount
	Top mount soldermask
	Bottom mount soldermask
	Top mount solderpaste
	Bottom mount solderpaste

Pin – Through:	Mount side
	Internal
	Opposite side
	Plane clearance
	Plane thermal
	Mount side soldermask
	Opposite side soldermask
	Selected hole

Via:	Mount side
	Internal
	Opposite side
	Mount side soldermask (optional – enables probing of via)
	Opposite side soldermask (optional – enables probing of via)

Pad and hole dimensions should be determined from a combination of:

- The relevant component datasheet
- The 'design rules' documents: Through_Hole_Padstack_Guidelines.pdf and SMT_Footprint_Design_Guidelines.pdf
- The 'IPC LP Viewer V2010' utility

However, some general principles are outlined below:

- Top mount = Bottom mount = Top mount solderpaste = Bottom mount solderpaste
- Top mount soldermask = Bottom mount soldermask
- Mount side = Internal = Opposite side
- Plane clearance inner/outer diameter = Plane thermal inner/outer diameter
- Top mount soldermask > Top mount
- Mount side soldermask > Mount side
- Plane clearance inner diameter < Mount side diameter
- Plane clearance outer diameter > Mount side soldermask diameter

For example:

Pin – SMD:

Top mount

Top mount
soldermask

Pin – Through:

Mount side   Plane thermal   Selected hole

Mount side soldermask   Plane clearance
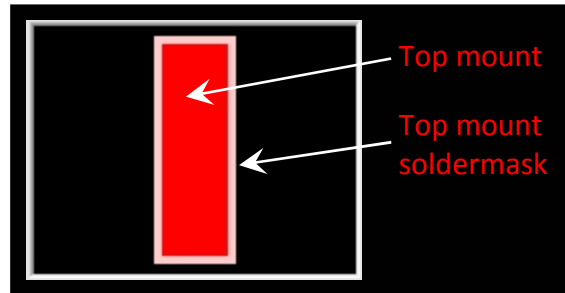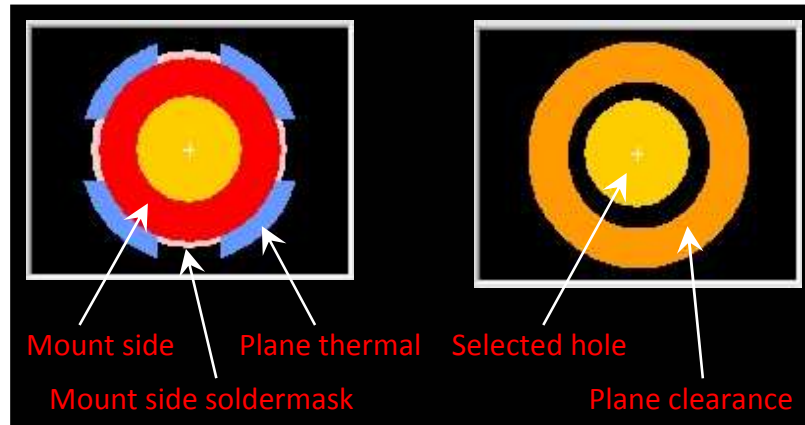
6. Once the padstack is complete, select 'File > Save' from the menu and close the Padstack Editor. The padstack may now be used in a cell design.

Creating a Cell

1. Open Central Library:

   → Expand 'Cells'
   → Right click entry of required type (SMT, Through, Connectors, BGA, etc.)
   → Select 'New Cell...'
   → Enter an appropriate name and click 'OK'

2. The 'Create Package Cell' dialog will open. Verify that the cell name is correct and enter the required pin count. Assign a 'Package Group' to the cell (e.g. IC – BGA, IC – DIP, IC – SIP, IC – SOIC, Connector, Discrete – Axial, Discrete – Chip, etc.) and set the 'Mount type' (either 'Surface' or 'Through') if necessary (note that certain package group selections disable this choice). Click 'Next >>' to launch the Cell Editor.
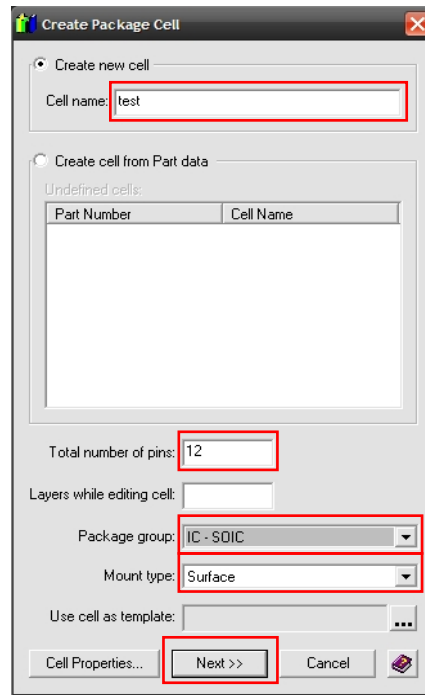
3. When the Cell Editor opens, it will automatically show the 'Place Pins' dialog. Select a padstack for each pin using the drop down menus in the 'Padstack Name' column (multiple pins can be set in a single operation by highlighting them all and pressing the SHIFT key while accessing the drop down menu). Pin rotation (in degrees) may also be specified.
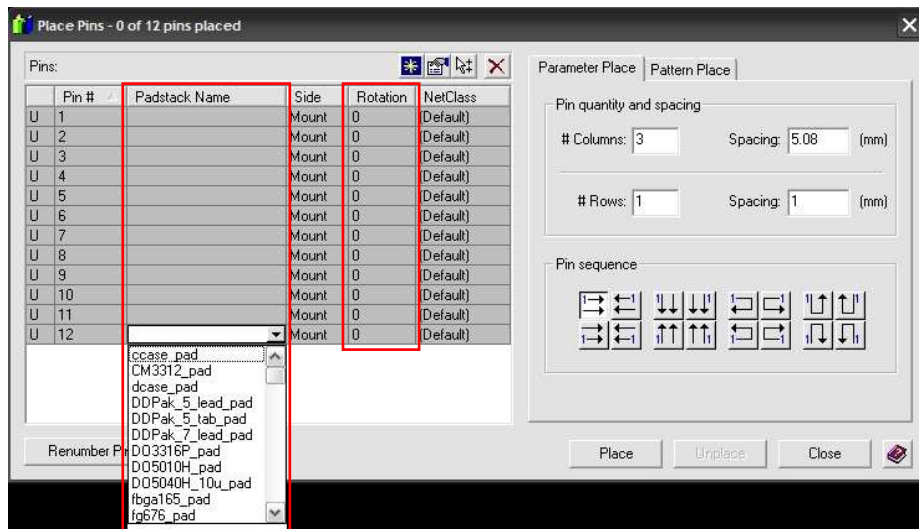
4. Once the pins themselves have been configured they can be added to the cell footprint. For standard components it is convenient to utilise the 'Pattern Place' feature. Click on the 'Pattern Place' tab and specify the 'Pattern Type'; a diagram of the corresponding device package will be displayed, into which the various package dimensions should be entered. Ensure that the 'Include outline' boxes are ticked and click 'Place' to automatically generate the footprint.

Ensure these are ticked

Package dimensions (layout specified by 'Pattern Type')
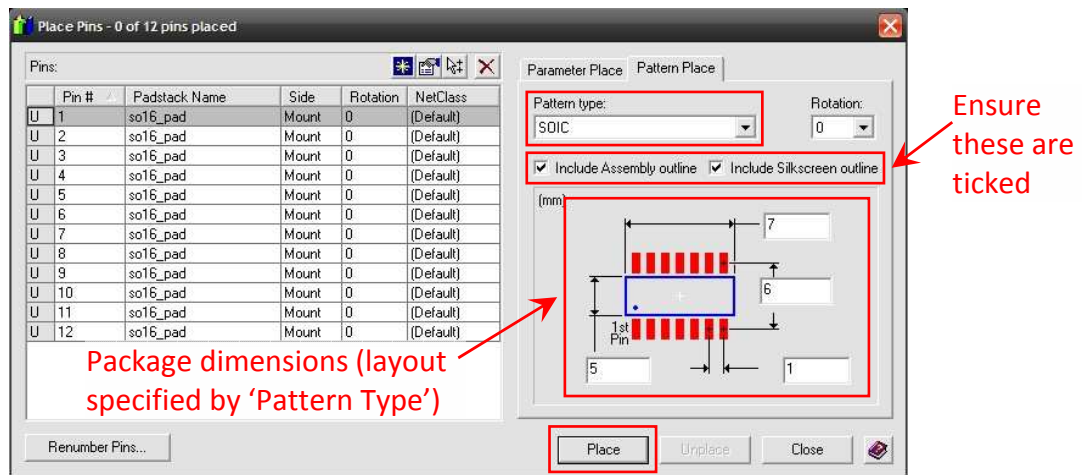
For non-standard packages, pins may be assigned in custom layouts from the 'Parameter Place' tab. Highlight the pins that should be added to the cell, enter the layout parameters (number of columns and rows, pin spacing and the order in which the pins should be indexed) and click 'Place'. The pins will then be attached to the mouse cursor, and should be placed by clicking at the required location in the main Cell Editor window.

Layout dimensions

Specifies the order in which the pin numbers will be sequenced

Only highlighted pins are placed
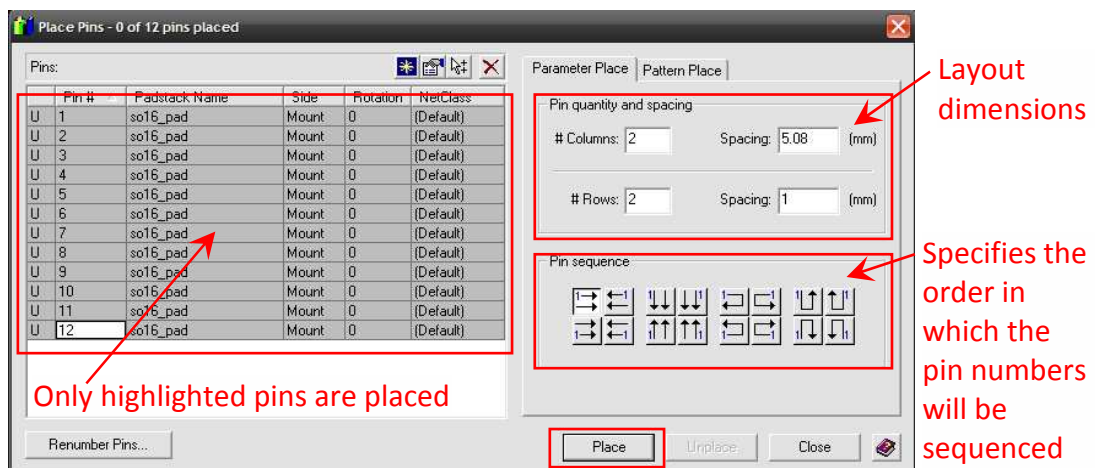
All cell dimensions should be determined from a combination of:

- The relevant component datasheet
- The 'IPC LP Viewer V2010' utility

5. After running a 'Pattern Place', focus will return to the main Cell Editor window and the newly created part will be displayed. In addition to pins, the footprint will consist of package outlines and identification text. If necessary, pins may be reconfigured by clicking the 'Place Pins' button to reopen the 'Place Pins' dialog.
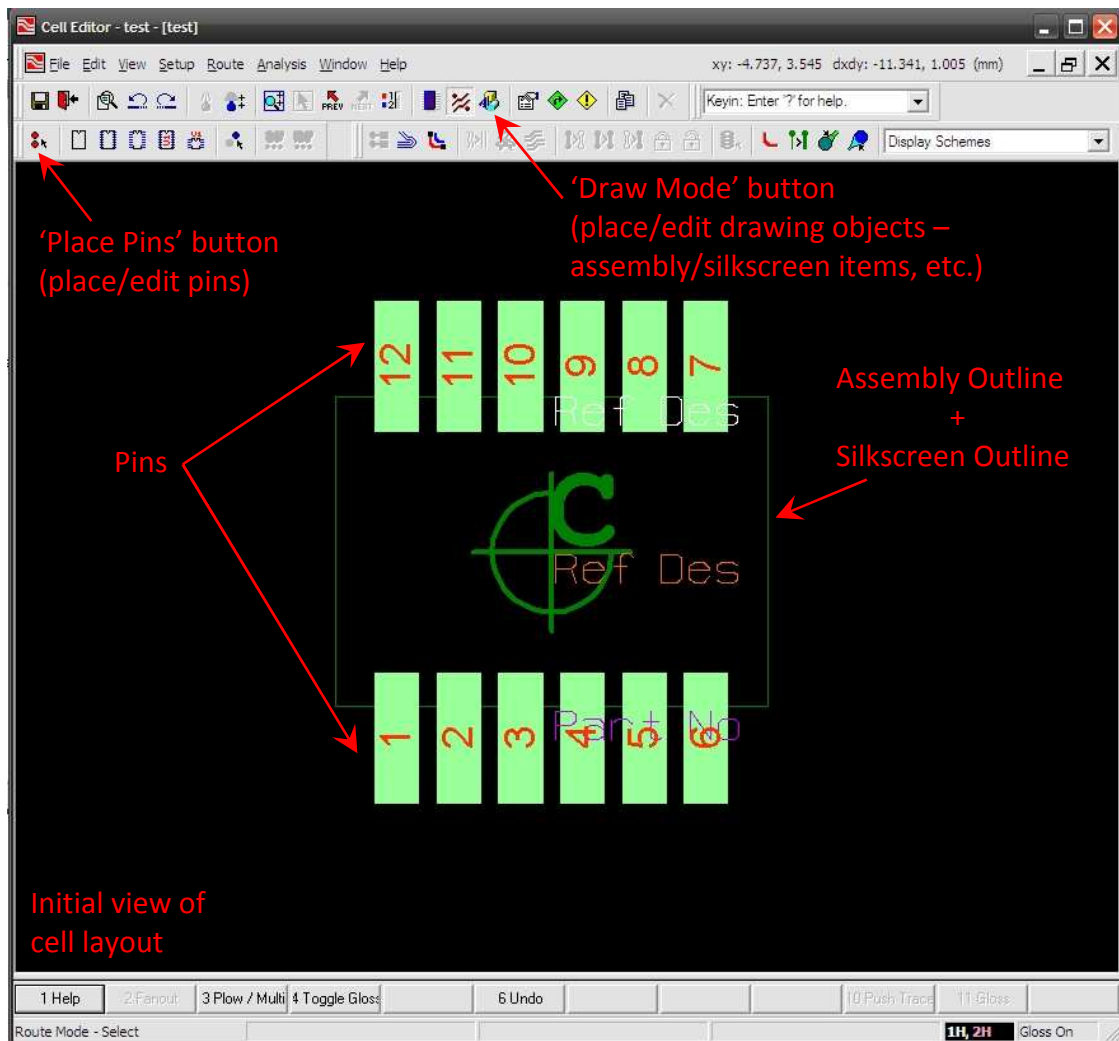
In order to complete the cell, it is necessary to draw a placement outline. Click the 'Draw Mode' button and a 'Properties' window will appear. Select 'Type' 'Placement Outline', choose an appropriate shape (typically a rectangle) from the 'Drawing Toolbar' and then draw a rough outline around the package cell. The precise coordinates of the placement outline vertexes should be entered via the 'Properties' window (the relevant entry boxes will appear when the outline is selected). All placement outlines should be derived from:

- The relevant component datasheet
- The 'IPC LP Viewer V2010' utility

The final step is to set the properties of the text identifiers (again via the 'Properties' window) and move them to standard locations.
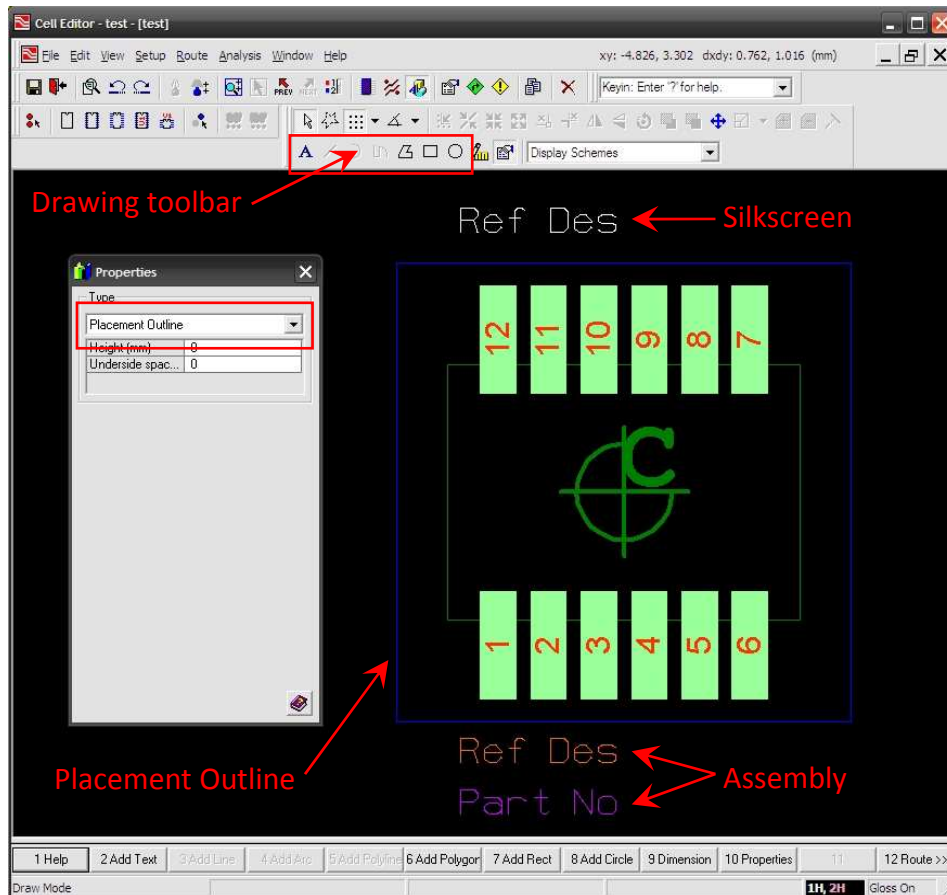
General notes on drawing outlines and text:

- Assembly items are only of significance to the PCB manufacturer. The designer typically will not look at them again once the cell is complete
- Silkscreen items are printed on the surface of the final PCB (no other drawing objects are visible on physical boards)
- Assembly and silkscreen outlines should be identical
- Silkscreen outlines represent the physical outline of a package, excluding the pins (this usually corresponds to the outline of the plastic body of a chip device)
- The placement outline defines the entire PCB area occupied by the package. No other component may overlap this region when placed on the same side of a board. (Placement outlines should include the areas occupied by any auxiliary parts required by a component, e.g. heatsinks)
- Placement outlines are always larger than silkscreen outlines
- It is usual to place the silkscreen reference designator text above the cell, and the assembly reference designator and part number below (as in the above screenshot)
- All text should have a 'Pen Width' of 0.25 mm and a height of 2 mm. However, it is acceptable (or even normal) to reduce the height to 1 mm for tightly packed designs (this simplifies PCB layout, but makes it difficult to read reference designators on the manufactured board)

6. Once the footprint is complete, select 'File > Save' from the menu and close the Cell Editor. The cell may now be used in a part design.

Creating a Symbol

1. Open Central Library:

   → Expand 'Symbols'
   → Right click entry of required type (Capacitors, Resistors, ADC, DAC, etc. – symbols are arranged in finer grained categories than padstacks or cells, as they are often specific to an individual part)
   → Select 'New Symbol...'
   → Enter an appropriate name and click 'OK'

2. The Symbol Editor will open, with a blank main window. Click the 'Pin' button and then click anywhere on the grid to place the required number of pins. Right click and select 'Cancel' when finished.
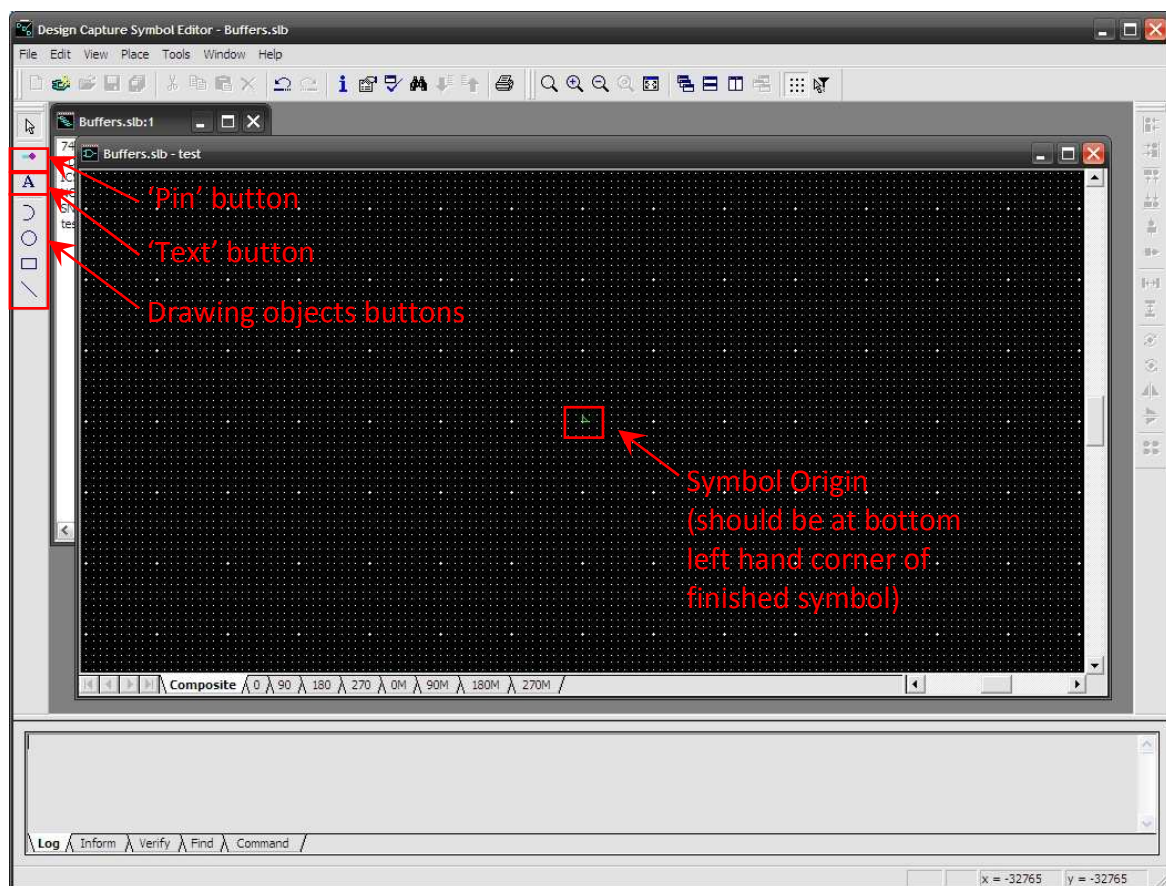

3. Double click on a pin to open the 'Properties' window. On the 'General' tab, enter a value for the 'Pin name' and select an appropriate 'Pin type' (these should be taken from the component datasheet).
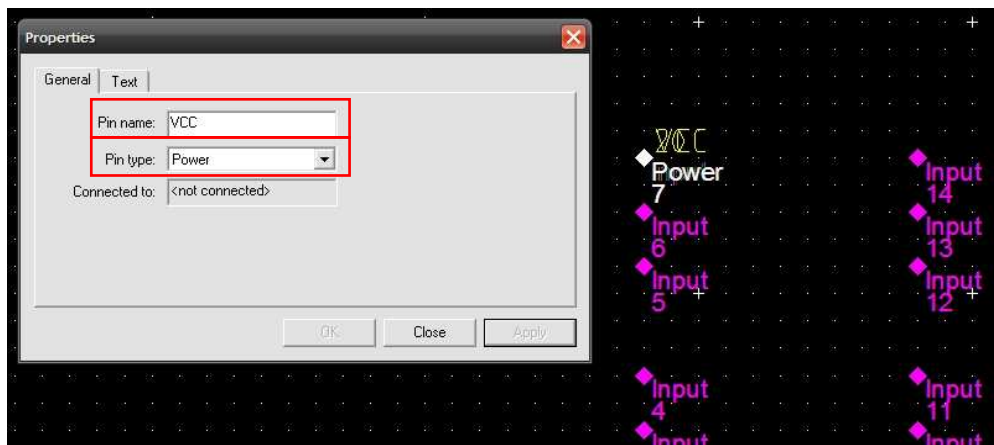
Then go to the 'Text' tab, and click on the blank line below 'Pin Name'. Create a new entry of 'Type' 'Pin Number' and input the pin number value (again from the datasheet). For some parts (e.g. resistors and capacitors), the pin name and number are the same and it is therefore unnecessary to display both. In this case, the state of the 'Visible' column for the 'Pin Number' line should be set to 'None'. Once the pin has been defined, click on the 'OK' button and repeat the process for all other pins on the symbol.
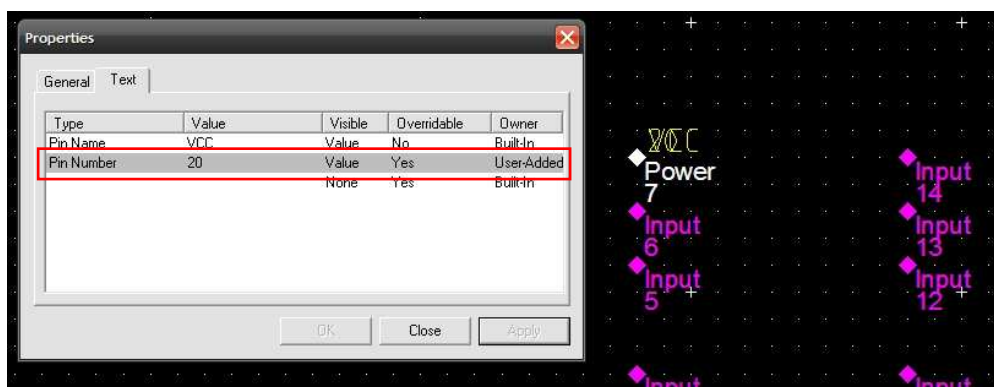

4. Using the drawing objects buttons, create a 'body' for the symbol. This can take the form of a simple rectangle, or it may consist of complex shapes (e.g. standard symbols for amplifiers, resistors, capacitors, transistors, etc.). Each pin should be connected to the 'body' by a straight line no less than two grid squares long. It is usual to place the text for pin names inside the 'body', with pin numbers on the outside. Text can be aligned by selecting multiple items and pressing one of the arrow keys while holding the CTRL key.
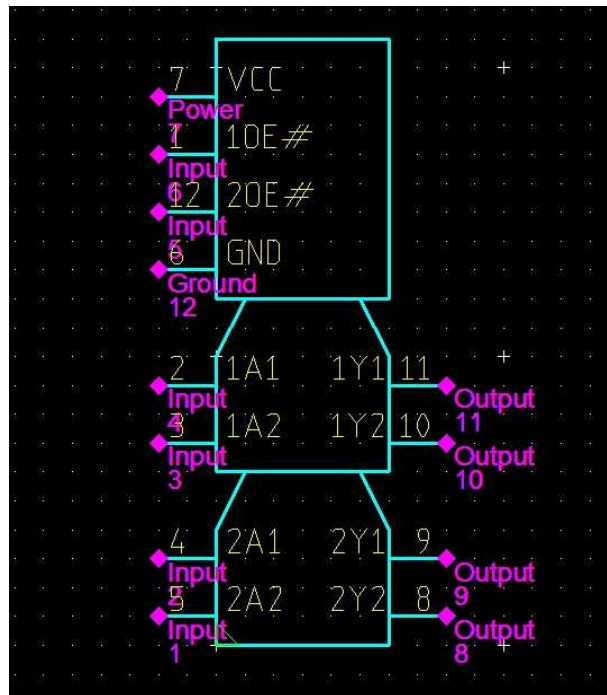
5. The symbol is now functionally complete. The following step is optional, and simply enables the default positions of identification text to be modified (this is usually unnecessary).

   Click the 'Text' button to open the 'Place Text' dialog. Select 'Type' 'Reference Designator', ensure that 'Auto position' is ticked and click the 'OK' button. Repeat this for text of 'Type' 'Part Name' and 'Part Label' (all symbols must display these three text items; certain components, such as capacitors and resistors, should also add text of 'Type' 'Value'). Move the text to appropriate locations relative to the symbol 'body' (it is normal to place the reference designator above the symbol, and the other parameters below).
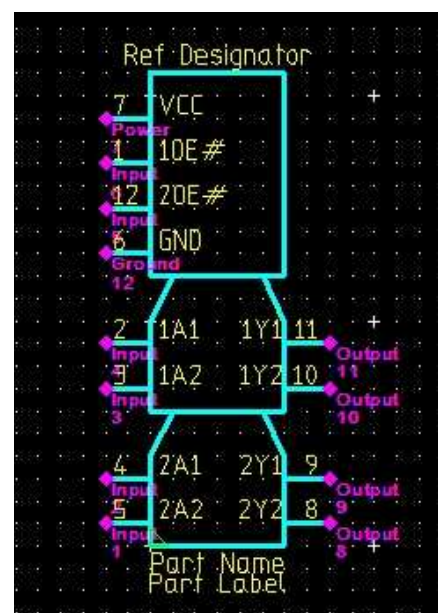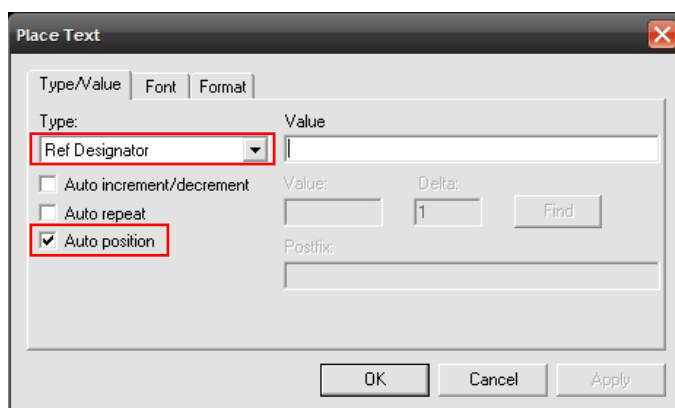

6. Once the symbol is complete, select 'File > Save' from the menu and close the Symbol Editor. The symbol may now be used in a part design.

Creating a Part

1. Open Central Library:

    → Expand 'Parts'
    → Right click entry of required type (Capacitors, Resistors, ADC, DAC, etc. – the categories are essentially identical to those in the symbols branch)
    → Select 'New Part...'
    → Enter the part name and click 'OK'

2. The Part Editor will open, with an incomplete entry for the new part in the 'Parts listing'. For all part types, it is necessary to enter the following information:
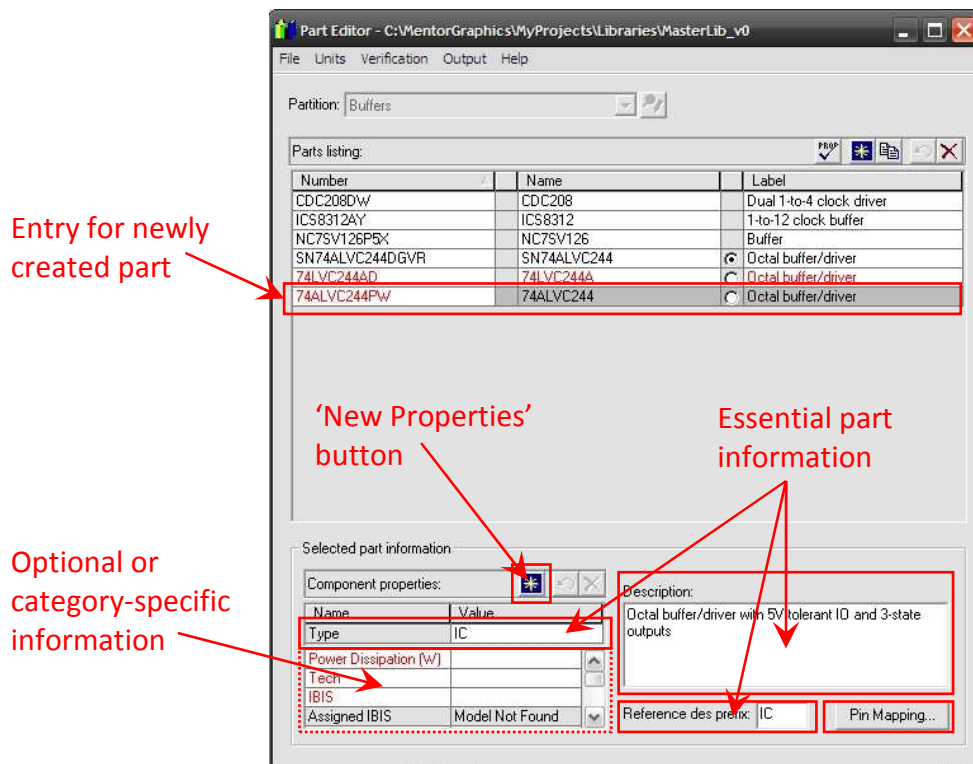
-   **Number**: The full part number taken from the datasheet. This includes any suffixes that relate to the specific physical package type (i.e. many components are made in a variety of package designs – SO, SSOP, TSSOP, etc. – and each has a particular identifier code)
-   **Name**: The name of the component family. This should be the 'Number' *without* the package-type suffix
-   **Label**: A brief (less than 30 characters, if possible) description of the part
-   **Type**: The part category – IC, Resistor, Capacitor, Connector, etc.
-   **Description**: A detailed description of the part
-   **Reference des prefix**: The standard reference designator prefix for the specified part category – i.e. 'R' for resistors, 'C' for capacitors, etc.

Additional information may be entered by clicking the 'New Properties' button and completing the new line that appears in the 'optional information' box below the category type indicator. The following part types require entries in this section:

- Resistors: Power Dissipation (W)
 Tolerance
 Value (Ohm)

- Capacitors: Rating
 Tolerance
 Value (F)

Once all parameters have been recorded, click the 'Pin Mapping' button.

3. The 'Pin Mapping' window will open. This is where the design files for the part are specified. Click the 'Import Symbol' button to open the 'Import' dialog.
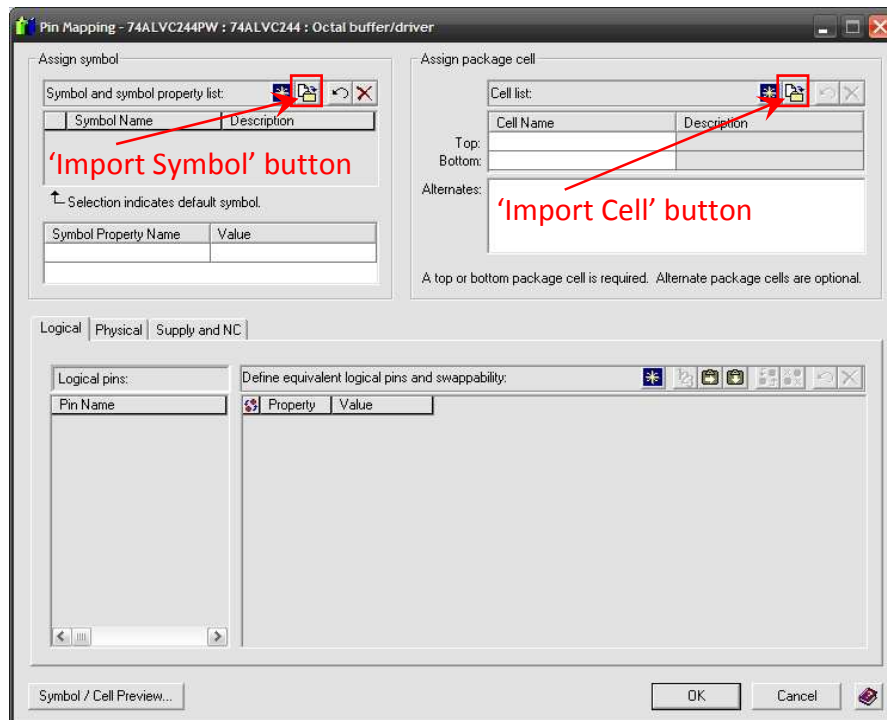

4. In the 'Symbols' tab of the 'Import' dialog, highlight the required symbol. Ensure that 'Create new gate information', 'Include pin properties' and 'Include pin number mapping' are ticked. The 'Number of slots in component' setting indicates the number of instances of the selected symbol that 'fit' inside the physical hardware – for example, a particular chip that contains multiple op-amps may be represented by several instances of a single op-amp symbol. In practice, though, mapping more than one symbol to a cell is confusing and troublesome. It is therefore recommended that the 'Number of slots in component' is always set to one. Click the 'OK' button when finished.

Must be ticked

Typically '1'
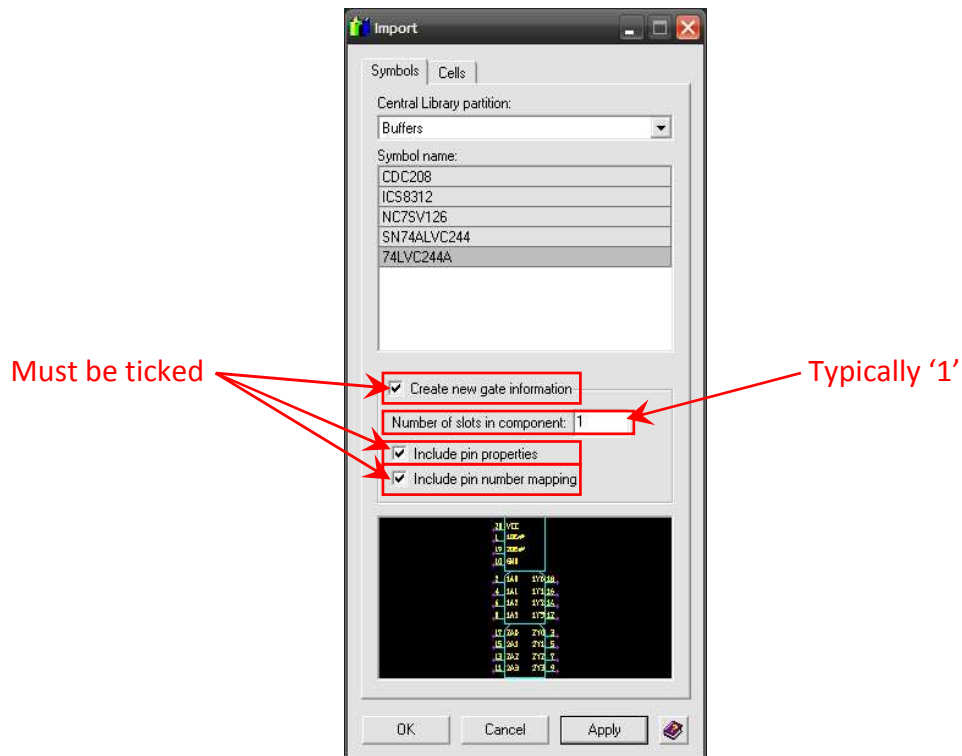
5. Upon returning to the 'Pin Mapping' window, click the 'Import Cell' button. In the 'Cells' tab of the 'Import' dialog, highlight the required cell (the choice of cells is limited to those which have the same number of pins as the selected symbol), click 'Import cell as' 'Top' and press the 'OK' button. (Note that this 'Top' cell will actually be used on both sides of any PCB, unless a different 'Bottom' cell has been explicitly defined)
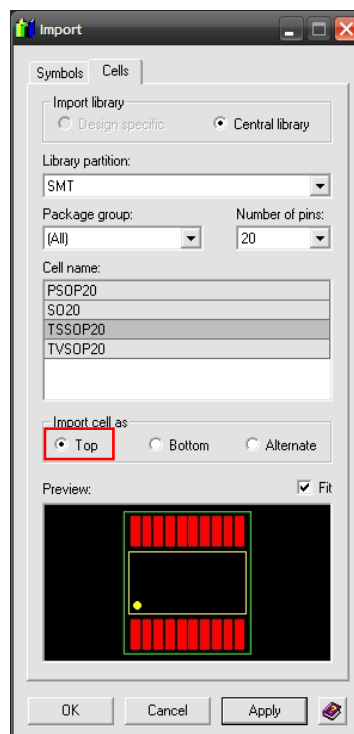
6. In most cases, all pins on the imported cell are automatically mapped to the selected symbol (provided that the pin properties were entered during the symbol design phase).
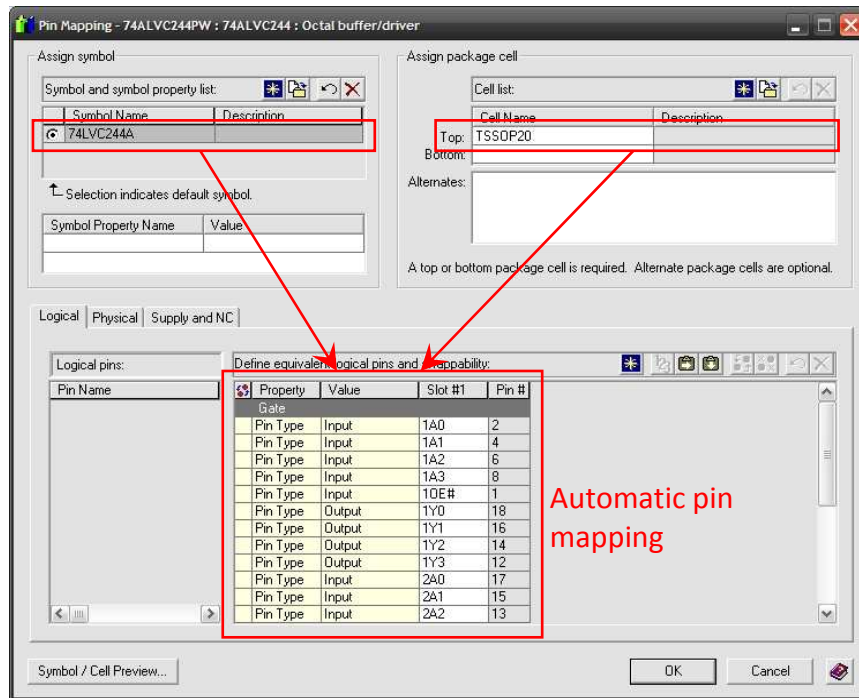
However, NC ('not connected') pins must be assigned manually (since symbols should never include pins of this type). This is achieved by clicking the 'Supply and NC' tab on the 'Pin Mapping' window, and entering any NC pin numbers in the 'No connect' row (each number must go in a separate column – the table will dynamically expand).
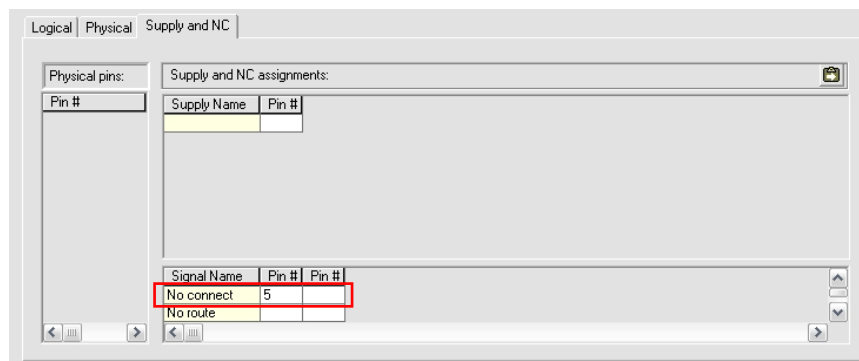
Once all mapping is complete, click the 'OK' button to return to the Part Editor.

7. Select 'File > Save' from the menu and close the Part Editor. The part may now be used in schematic capture and PCB layout.