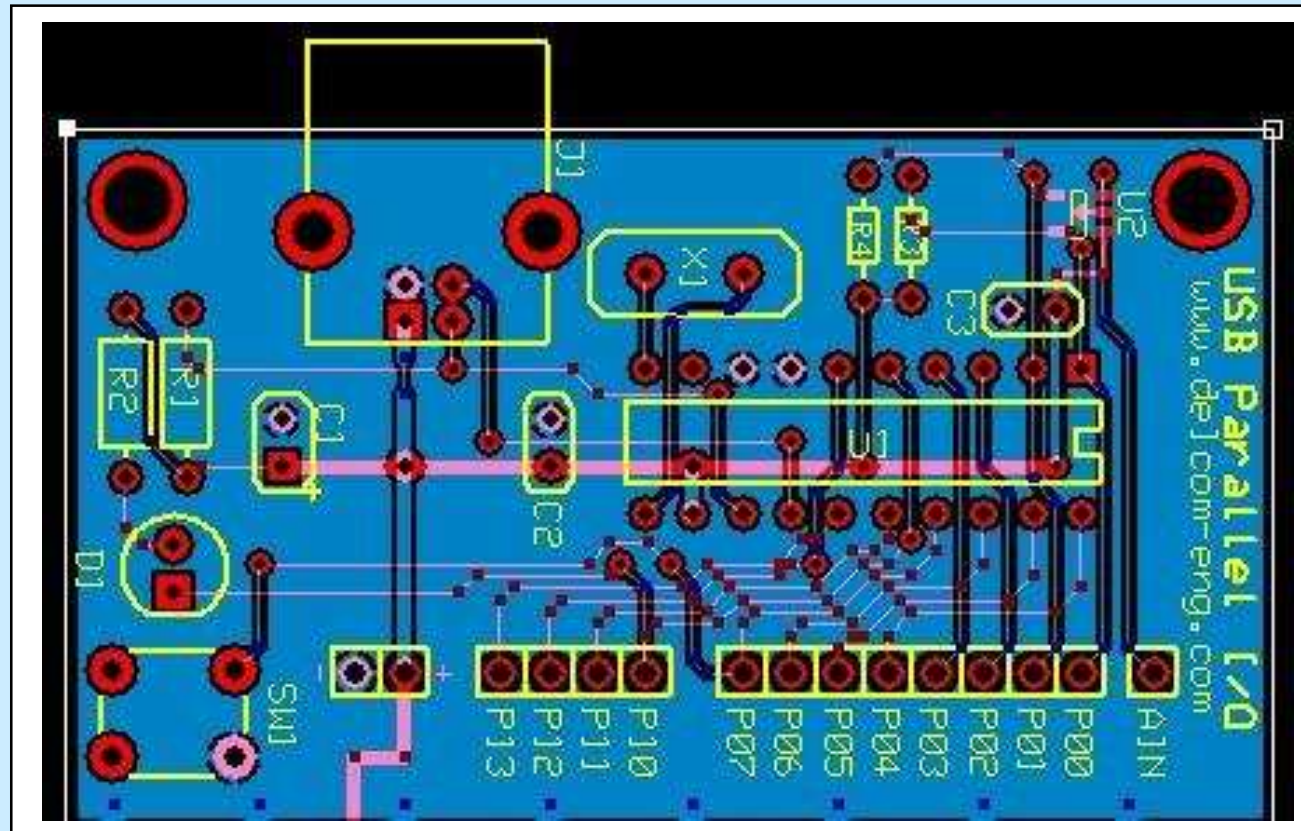


# A Quick Guide to CAD



a look at a few helping hands

# Introduction

Ideally a PCB CAD Program should provide:-

- **Schematic Capture - to include:**
  - A comprehensive Component Library with New-Part Creation & Management
  - Individual Multi-Field Part Specification - Reference, Value, Part No., Footprint
  - Bill-of-Materials Generation
  - Design Rule Checking - Single-Node nets, Unconnected Pins, Shorted Outputs
  - Generation of Master Design Database - Netlist and Component Footprints
- **PCB Layout - to include:**
  - Automatic Extraction of Netlist and Component Footprints from Database
  - Support for Multi-Layer (4 or more) Boards
  - Easy Component Placement - including Rotation and PCB Side
  - Display of Net Connectivity during Placement and Routing - so-called 'Rubberbands'
  - Easy Editing of Track and Segment Widths, Planes and Area-Fills
  - Design Checking - Incomplete Nets, Cu-to-Cu Spacing or Shorts, Duplicate Holes
  - Generation of Board Manufacturing Data - Gerber Files (274-X or 274-D)

# There's CAD...and There's BAD!

...well not 'bad' really - but perhaps 'Maxi' and 'Mini' CAD  
Some 'Mini-CAD' are little more than Drawing Packages.

They Don't:-

- Require the Schematic Capture process - it is Optional
- Provide for comprehensive Part Specification - often just Reference and Value
- Provide a Bill-of-Materials
- Provide Multi-Layer Capability - sometimes only 2
- Perform Design Rule Checking
- Generate a Design Database - limited to a Netlist at best with no Footprint data
- Display Net-Connectivity during Placement and Routeing

They Do:-

- Provide an Easy to Learn and Use method of designing simple PCBs
- Insulate the User from the necessary complexities of 'Maxi-CAD' systems

There are also 'Midi-CAD' packages that fall somewhere in between

# Design Work-Flows

## Maxi-CAD

### Schematic Capture

- Create any new Library Parts
- Organise the new Parts in the Library
- Draw the Schematic
- Enter Part Information for each Component
- Design Rule Check
- Generate the Design Database containing:
  - Component References and Values
  - Component Part Numbers and/or Order Codes
  - Component Inter-Connectivity and PCB Footprints
- Produce Bill of Materials

### Layout

- Create any new Library Footprints
- Organise the new Footprints in the Library
- Import the Design Database containing:
  - Component References and Footprints
  - Netlist of Inter-Component Connectivity
- Create the PCB Outline
- Place Components using any Guidance
- Route Nets using 'rubberbands'
- Design Rule Check
- Generate Board Manufacturing files

## Mini-CAD

### Schematic Capture - Optional

- Create any new Library Parts
- Draw the Schematic
- Enter Part Information for each Component
- Design Rule Check
- Generate the Design Database containing:
  - Component References, Values and Connectivity
- Produce Bill of Materials

### Layout

- Create any new Library Footprints
- Import the Design Database containing:
  - Component References and Connectivity
- Create the PCB Outline
- Manually Add each Component Assigning:
  - Component Reference
  - Component Footprint
- Place Components
- Route Nets Manually
- Design Rule Check
- Generate Board Manufacturing files

(Items shown in grey are CAD system dependant)

# Why Would You Use 'Maxi-CAD' ?

## Well - You Wouldn't:-

- If you were only ever going to design a few fairly simple boards
- You were on a very strict budget - 'Maxi-CAD' can be expensive
- If the thought of managing Libraries gives you a headache!

## Maxi-CAD is rather more complicated to use but it:-

- Forces Rigour into the Design Process - it makes you think carefully
- Provides 'Instant Re-Use' of existing Library Parts and Footprints
- Provides Multi-Layer Capability - useful for Power and/or Ground Planes
- Displays 'Live' Net-Connectivity during Component Placement and Net Routeing:-
  - helps during the Placement Phase as it indicates the ideal Component Orientation
  - helps during Routeing as it shows which Pads are part of the Net
- Performs Design Rule Checking - reducing Silly Mistakes and Design Errors
- Provides Automatic Documentation of the Design including:-
  - a Circuit Diagram - very useful while Placing and Routeing the board
  - Copper Layer Artworks - an aid during final design inspection prior to 'going to press'
  - Bill of Materials - helps with ordering components

# Some Stumbling Blocks

There are a number of CAD features designed to help in the design process:-

- The Drawing Grid
- Schematic Drawing Aids - Labels, Ports & Symbols
- Creating a Custom Schematic Component
- Creating a Custom PCB Footprint
- Managing Libraries

these facilities, once mastered, can lead to easier and better designs

# The Drawing Grid

## Historically, CAD systems used a 0.1" Imperial Basic-Grid

- This is the basis of the very common 0.1" pitch device packages
  - DIL Packages (ICs) 0.3" wide with 0.1" pin pitch
  - SIL Packages (resistors) 0.1" wide with 0.1" pin-pitch
  - IDC Ribbon-Cable connectors with 0.1" pin-grid
- The Basic Grid could be sub-divided to permit routeing between pins
- Many 'modern' component dimensions are still based in the Imperial system
  - SOIC with a pin pitch of 0.05"
  - 1206 Resistor with dimensions of 0.12" x 0.06"

## The Metric Grid

- Board Outlines have long been specified using metric dimensions
  - the Single Euro-Card has dimensions of 160mm x 100mm
- Many ICs now use the metric system as their primary dimension system
  - SSOP (0.50, 0.65mm) QFP (0.50, 0.65, 0.80mm) - some very 'odd' pin-pitch values!
  - 3216 Resistor - the equivalent of the 1206

# Standardising Your Grid System

Using a Standard Grid for all Activities means:-

- You will always be able to finish a Net accurately on a Pad centre-line
- There can be no CAD-caused dimension conversion errors
- You may need to use a fine Grid to create 'foreign' devices
  - even then you may have to 'odds & evens' the Pads
- You may need a temporarily fine Grid to connect to some Pads in Layout

Standardising on an Imperial Grid - my preference

- All simple metric pitch devices are wholly compatible with an Imperial Grid
  - 2-leaded devices such as resistors, capacitors and inductors
- So are many of the more complex metric packages
  - SOIC family (0.05"), some SSOP (0.025"),
- Many metric devices are 'sufficiently' compatible
  - low pin-count MSOP (0.65mm/0.0256"), TO263 Power Device
- Avoid 'awkward' metric-pitch parts where possible
  - they tend to be the fine-pitch devices and you are probably soldering them by hand
  - many devices are available in a more appropriate package style - usually larger but most Mouse builders are not concerned with the ultimate in miniaturisation

# Schematic Drawing Aids

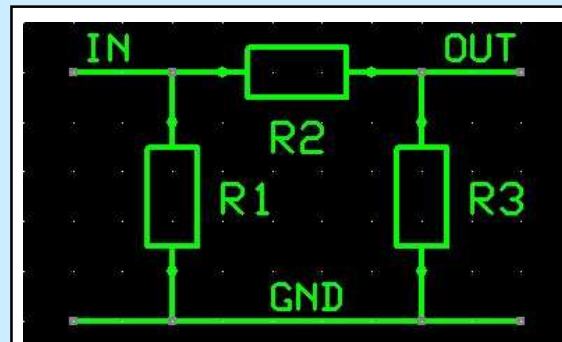
## Net Labels

### Are Used to:

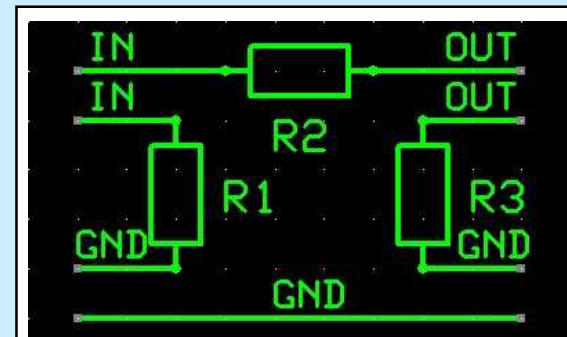
- give a net a specific name - e.g. 'INPUT'  
useful for identifying important nets  
a 'net enquiry' will yield 'GND' not 'N00033'
- link wires separated by distance  
reduces wiring 'clutter' on the Schematic  
multiple labels MUST have exactly the same text
- identify the individual members of a bus  
a bus is just a human visual aid - it means nothing  
to the CAD system  
labels on a bus are simply 'wires separated by distance'  
dissimilar signals may be grouped together and shown  
as a bus to improve schematic readability

### NOTE

- Labels have a 'Hot-Spot' - often bottom left - which is placed against a wire or net to make the association
- Labels are 'local' to the Sheet on which they appear
- Labels should not be used to place simple text on a schematic



Labels simply identifying Wires



Using Labels to connect Wires



Bus  
Labels

# Schematic Drawing Aids

## Sheet Ports

Are Used to:

- permit connections between Sheets  
are 'Global' to the design not 'Local' to the Sheet  
all Ports with the same name are connected  
useful for large, multi-sheet Schematics

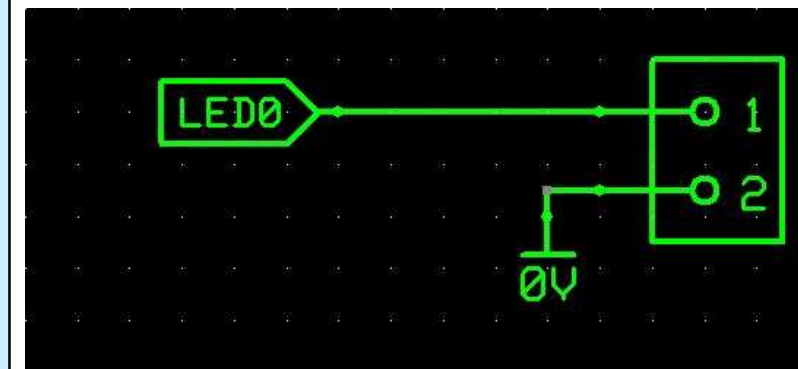
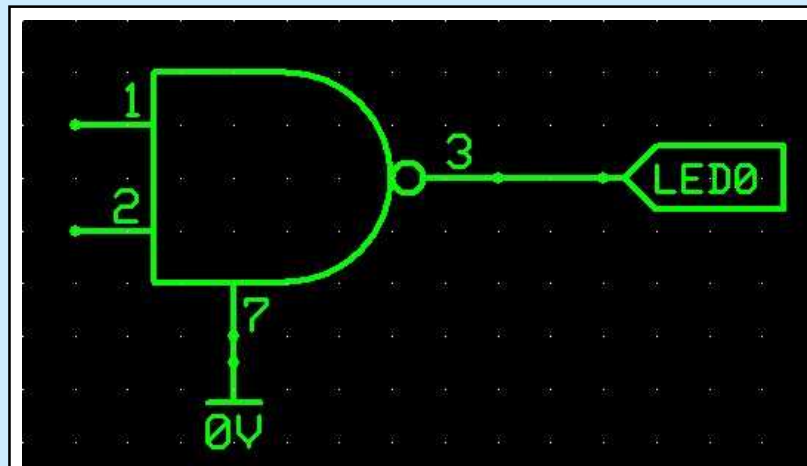
Some CAD systems use Ports to also provide the Label function described earlier. In this case all such 'labels' are Global

## Power Symbols

Are Used to:

- connect all identical power nodes  
are Global to the design  
they reduce wiring clutter on the Schematic  
symbols to be connected MUST have the same name

Multiple 0V nets (analogue or digital) may be kept separate by using different symbols - 0VA or 0VD



Using Sheet Ports and Power Symbols to effect Global connectivity

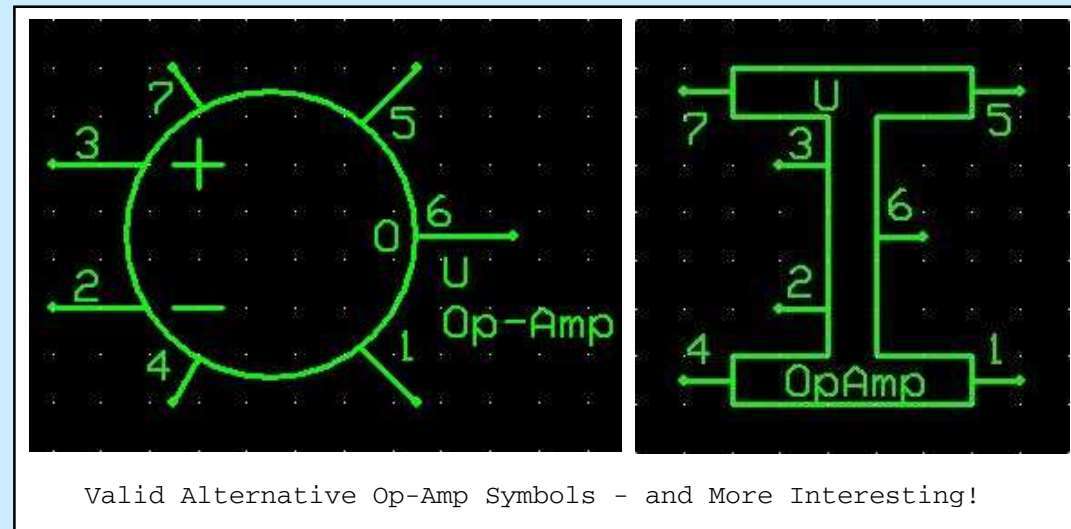
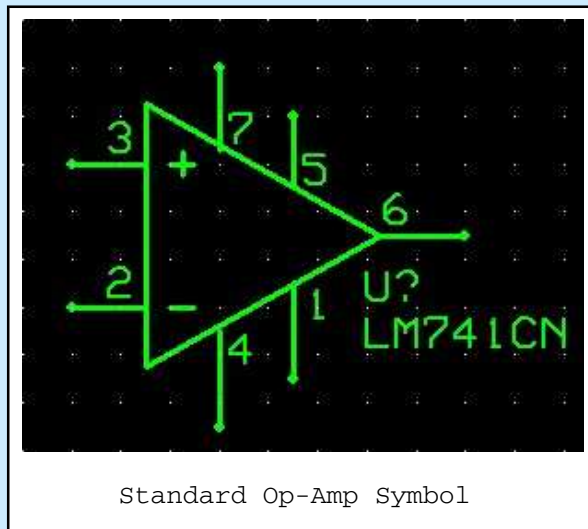
# Creating a Custom Component

The CAD System is only interested in 2 things - Pins & Pads

- Schematic Capture - Pins
- Layout and Tracking - Pads

The system associates each Pad in the Layout with a Pin on the Schematic

The shapes and forms we use to denote the various types of component are simply a convenience for humans



# Creating a Custom Component

## Based On An Existing One

### Option 1 - Generic Component

- 'created' in the Schematic Editor - not the Library
- place the general-purpose part 'Op-Amp'
- edit its 'Part Name' property to LM741CN
- can only be used if both parts have same pin-out

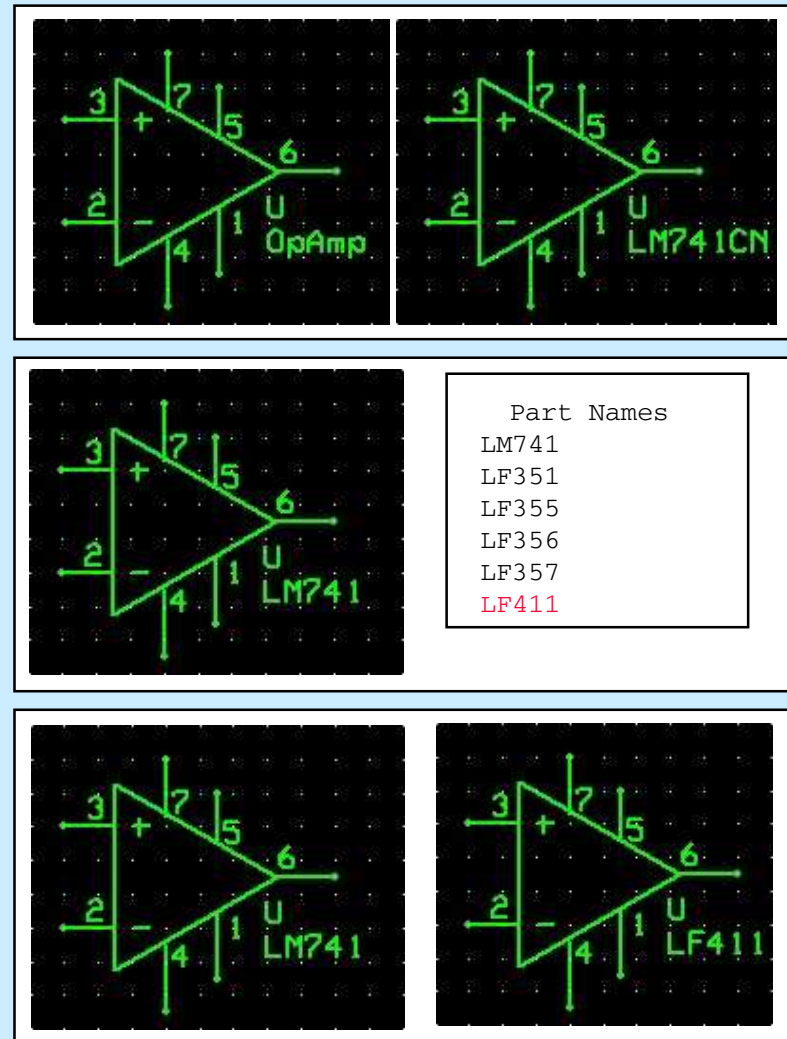
### Option 2 - Multi-Device Component

- in the Library Editor select an identical device
- add the new Part Name to the list
- save the updated Library component
- add the new LF411 to the Schematic

### Option 3 - Single Device Component

- in the Library Editor select an identical device
- edit the Part Name field
- use the Save As function to save the new Part
- add the new LF411 to the Schematic

Note that Library Components do not normally have package suffixes - the 'CN'. This reduces the number of Library Parts and allows package specification - TH or SMD - at design-time



# Creating a Custom Component

## From Scratch

### Draw the Body Outline

- try to ensure that the 'scale' of the new part is appropriate - use a similar part as a guide
- allow enough space on each side for the pins

### Place the Pins

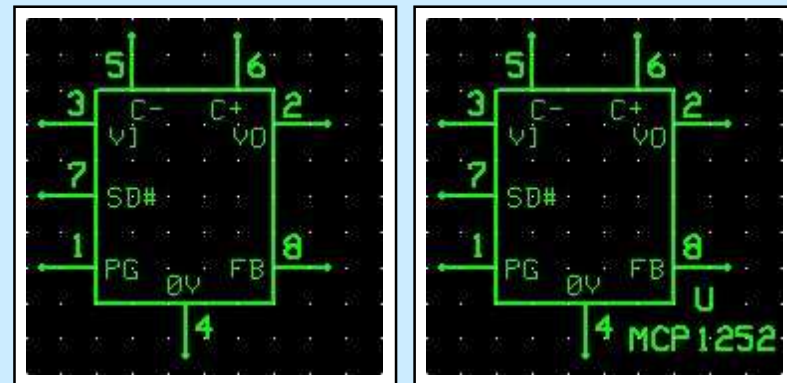
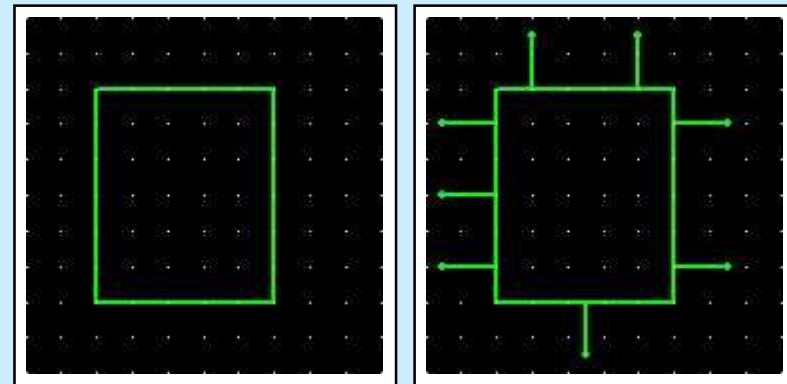
- set pin locations to provide a 'flow' from L to R
- generally place Power pins on the Top edge
- place Ground pins on the Bottom edge
- group pins with related functions - CPU ports etc

### Assign Pin Numbers and Names

- try to use meaningful pin name abbreviations
- indicate 'active low' pins - SD#, #SD, /SD
- set 'Pin Type' if the CAD system allows this (power, input, output, passive, bi-directional)

### Allocate a Reference and Name

- use standard References - U, Q, R, C, L etc
- use a 'generic' name to allow later package spec.



### Note:

Save time by using a similar part as a starting point for a new component by using Save As. The new device can be 'created' by editing the pins - moving/adding/removing/renameing.

# Creating a Custom Component

## Some Further Considerations

### Pin Numbering Consistency

- adopt a consistent scheme for numbering component pins
  - for diodes and polarised capacitors make the +ve (cathode for diode) pin1
  - for transistors/FETs always use the same scheme - e.g: pin1-collector/drain, pin2-emitter/source, pin3-base/gate
- allows the same PCB footprint to be used for multiple devices
  - small axial electrolytics and signal and rectifier diodes (1N4000 series) can share a common PCB footprint
  - bipolar transistors and MOSFETs can share common footprints - TO220, TO126, SOT223, SOT23 etc

### Pin Types

- this parameter is assigned to each component pin
  - typical Types are 'input', 'output', 'bi-dir', 'power' and 'passive' and are used by the DRC to detect 'wiring errors' such as 'output connected to output' or warnings such as 'input connected to power'
  - inappropriate Type assignment can lead to frustrating DRC failures - assigning 'passive' to all pins defeats DRC

### Component Mirroring

- most CAD systems allow component mirroring and rotation
  - this allows a 'left-to-right' oriented component to be placed on the Schematic with a 'right-to-left' flow

# Creating a Custom Footprint

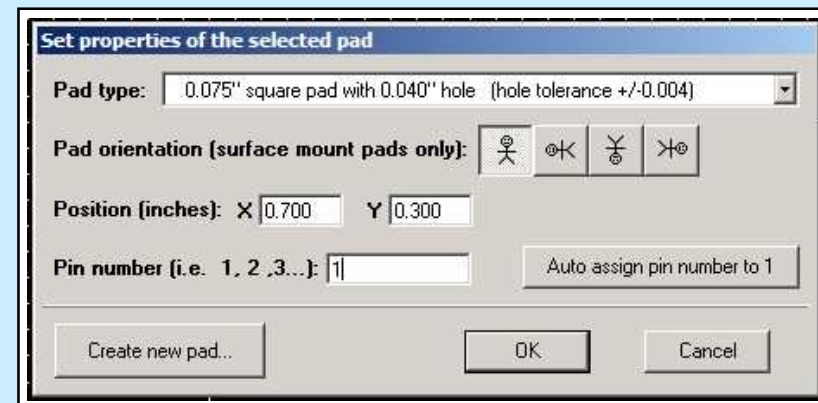
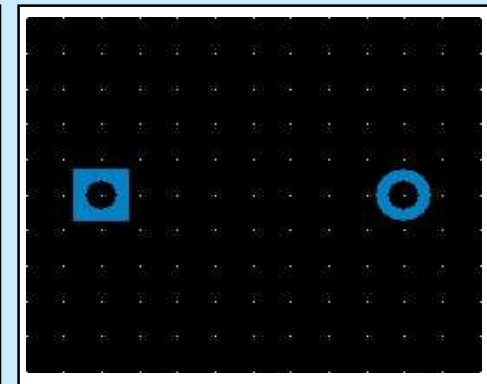
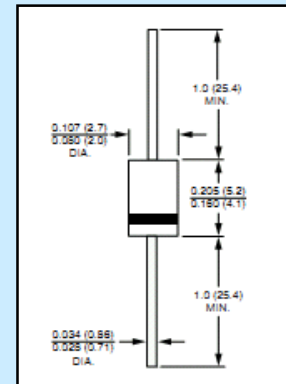
## 1N4000 Series From Scratch

### Establish from the Datasheet:

- maximum lead diameter - 0.035" (0.89mm)
- body dimensions - L: 0.2"(5.2mm) x D: 0.11"(2.7mm)

### Select and Place the Pads

- choose hole diameter larger than lead diameter!  
0.035" lead requires at least 0.039" (1.0mm) hole
- select a square pad for pin1 - round for the rest  
large pads reduce 'lifting' during solder re-work  
small pads allow more routing room between pads  
a good size for this TH component is 0.075" (1.9mm)
- set Drawing Grid to 0.05" and set Snap-to-Grid
- place pads 0.4" (8 grid steps) apart  
this allows a suitable 'body-to-bend' spacing
- assign a Pin Number to each Pad  
use the Pad Properties editor for this task



# Creating a Custom Footprint

## 1N4000 Series From Scratch

### Establish from the Datasheet:

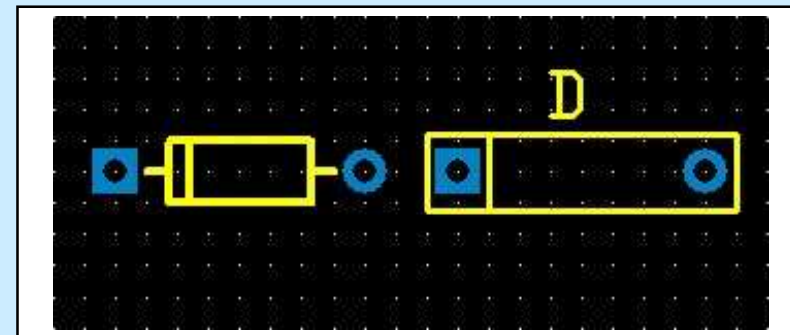
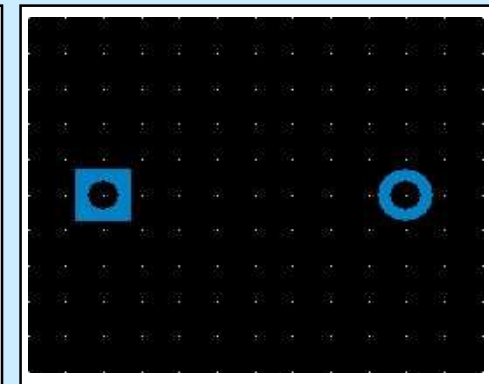
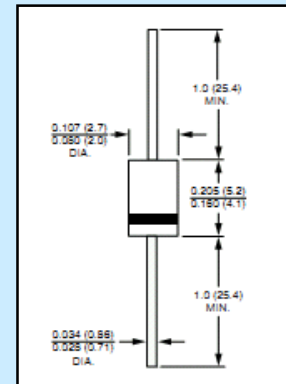
- maximum lead diameter - 0.035" (0.89mm)
- body dimensions - L: 0.2"(5.2mm) x D: 0.11"(2.7mm)

### Select and Place the Pads

- choose hole diameter larger than lead diameter!  
0.035" lead requires at least 0.039" (1.0mm) hole
- select a square pad for pin1 - round for the rest  
large pads reduce 'lifting' during solder re-work  
small pads allow more routing room between pads  
a good size for this TH component is 0.075" (1.9mm)
- set Drawing Grid to 0.05" and set Snap-to-Grid
- place pads 0.4" (8 grid steps) apart  
this allows a suitable 'body-to-bend' spacing
- assign a Pin Number to each Pad  
use the Pad Properties editor for this task

### Add the Silk-Screen Layer Outline

- and a Reference Designator if expected



### Note:

For some components, particularly mechanical ones, accommodating the entire body within the silk-screen outline can be used to prevent 'component clashes'.

# Creating a Custom Footprint

## SOIC8W From SOIC8

These devices are similar except for the body width  
SOIC8 (0.150") - SOIC8W (0.207")

### Establish from the Datasheet:

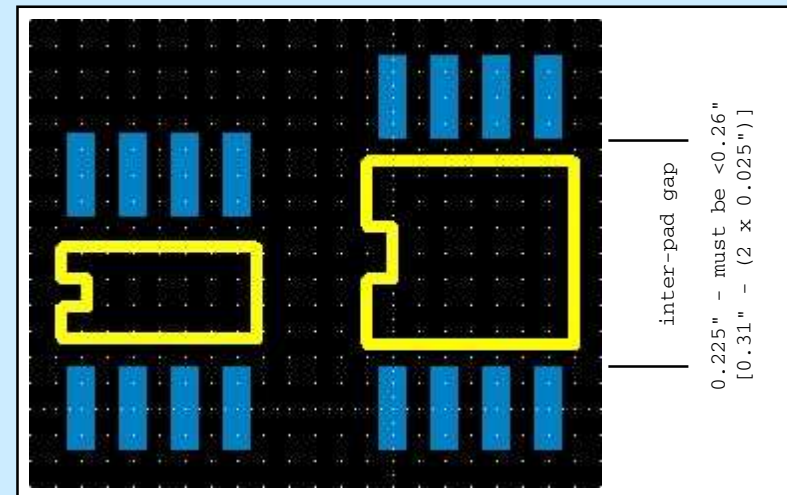
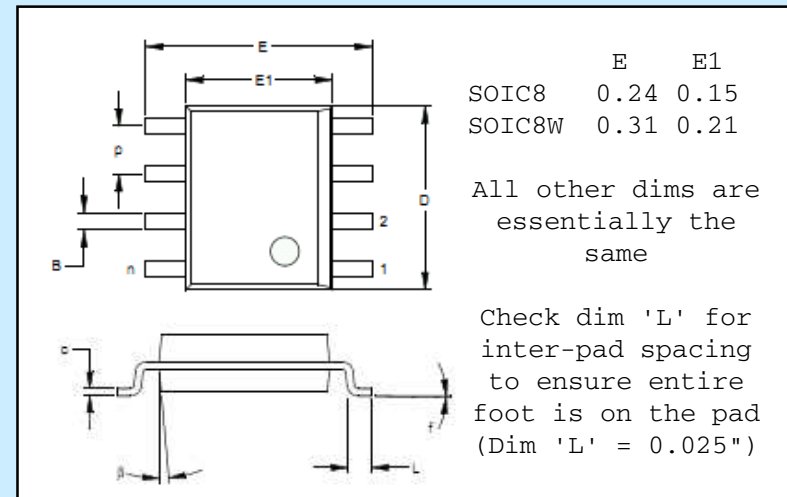
- differences in dimension 'E' and 'E1'
- calculate amount to move pads 5 to 8  
 $0.31" - 0.24" = 0.07"$  - call it 0.075" to stay on 0.025 grid

### Place and Modify Source Footprint

- place Source Footprint onto PCB using Grid  
typically the centre of Pad 1 is the component origin
- set the Drawing Origin to Pad 1 centre  
the XY coordinate display is used to position the pads
- move pads 5 to 8 0.075" (3 grid steps) north  
the pads may need to be 'released' in order to do this
- modify the Silk-Screen outline
- use Save As command to save new Footprint  
choose a meaningful name: 'SOIC8W' or 'SOIC8207'

### Check the new Footprint

- print it 1:1 and place a real part over it



# Creating a Custom Footprint

## 0.65mm MSOP From Scratch

### Establish from the Datasheet:

- lead pitch 0.65mm - converts to 0.0256"
- lead width 0.30mm - converts to 0.012"

### Set Pad Width at 0.014" from:

- pad spacing 0.0256"
- minimum copper-to-copper spacing = 0.01"
- allow a pad position error of up to  $\pm 0.001$ "  
for rounding errors between metric and imperial
- permits a pin-to-pad offset error of  $\pm 0.001$ "  
from lead width of 0.012"

### Place Pads

- set Drawing Grid to 0.001" and set Snap-to-Grid
- place Pad 1
- set Drawing Origin to Pad 1 centre
- place remainder of Pads at X offsets as per Table

determine the pad length and inter-row offset as for the SOIC8 previously examined

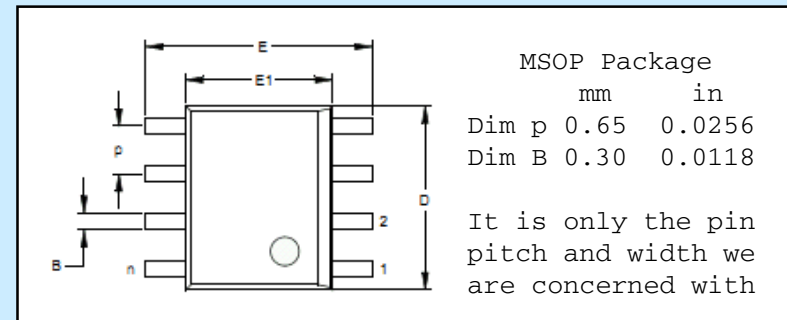
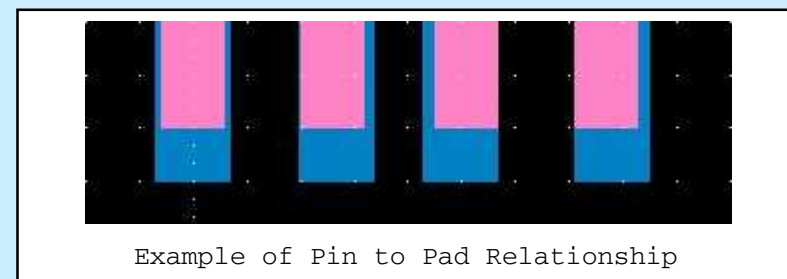


TABLE OF PIN AND PAD CENTRES					
Pin#	Pin Centre	Pad Centre	Pos Error	Pad-Pin Clrnc	Pad-Pad Space
1	00.0	00.0	N/A	N/A	N/A
2	25.6	26.0	+0.4	0.6	12.0
3	51.2	51.0	-0.2	0.8	11.0
4	76.8	77.0	+0.2	0.8	12.0
5	102.4	102.0	-0.4	0.6	11.0
6	128.0	128.0	0.0	1.0	12.0
7	153.6	154.0	+0.4	0.6	12.0
8	179.2	179.0	-0.2	0.8	11.0



# Creating a Custom Footprint

## 2mm Milli-Grid From Scratch

### Establish from the Datasheet:

- lead pitch 2.00mm - converts to 0.0787"
- pin side 0.50mm - 0.71mm (0.028") diagonal

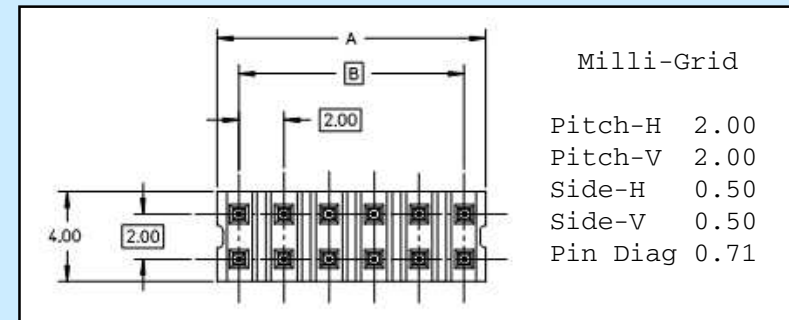
### Footprint Design:

- set pad spacing 0.08" - maintains imperial grid induces an increasing pin-pad error of 0.0013 per pin
- set pad diameter at 0.055" allows a 0.008" trace to be routed between pads with 0.008" clearance on either side
- set hole diameter at 0.035" sets annulus size at 0.01" accommodates 'error-creep' up to  $\pm 4$  pins

### Limitations

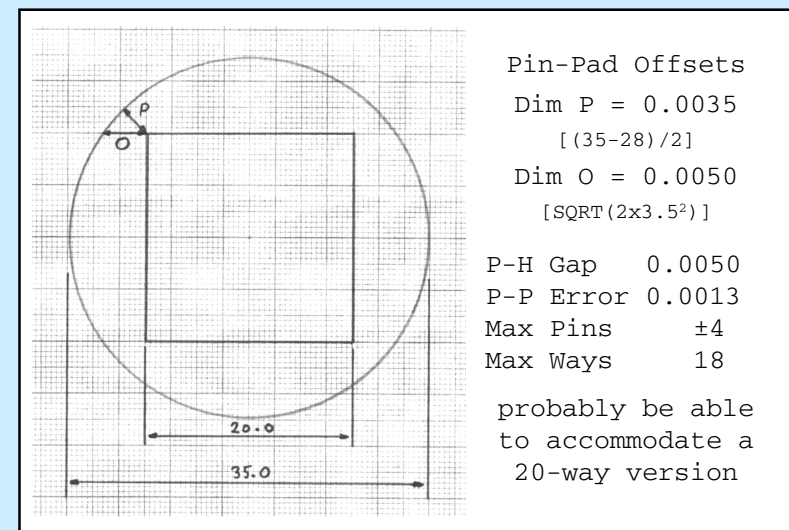
- can only be used up to 20-way (2 x 10) versions

use suitably spaced multiple smaller connectors to fabricate larger pin counts



Milli-Grid

Pitch-H	2.00
Pitch-V	2.00
Side-H	0.50
Side-V	0.50
Pin Diag	0.71



Pin-Pad Offsets

Dim P =	0.0035
	$[(35-28)/2]$
Dim O =	0.0050
	$[\text{SQRT}(2 \times 3.5^2)]$

P-H Gap	0.0050
P-P Error	0.0013
Max Pins	$\pm 4$
Max Ways	18

probably be able to accommodate a 20-way version

# Creating a Custom Footprint

## Some Pad Stack Considerations

### Hole Sizes

- make holes sufficiently larger than the pin/leg to allow for removal during re-work
  - for small diameter pins and legs a hole size 0.01" (0.25mm) larger than the pin is satisfactory
- some pins are rectangular - use the 'diagonal' as the pin dimension

### Pad Dimensions

- allow sufficient area all round the pin/leg for soldering
  - a minimum annulus, typically 0.008", is necessary around the hole to accommodate registration errors
  - insufficient annulus width on a TH pad leads to a 'blobby' joint - ideal pad should be Hole Dia x 1.5
  - on small rectangular SMD pads (SOIC etc) allow extra length (say 0.02") to facilitate 'end soldering' by hand

### Pad to Pad Gaps

- allow sufficient gap between pads to route a single narrow track - if possible
  - this will typically require a pad-pad gap of 0.025" to allow for a 0.008" track with 0.008" track-pad clearance

### SMD Pads

- appear on one side of the board only and PCBs have a 'Component Side'
  - more capable CAD systems can flip components to the other side which automatically takes care of 'pad mirroring' and the silk-screen artwork orientation
  - simpler systems can't 'flip' and require a separate 'mirrored' footprint
- consider placing only 2-pad parts on the reverse side - no need for mirrored footprints

# Creating a Custom Footprint

## A Few Extra Considerations

### Board Manufacturer's Copper Capabilities

- minimum track width - often 0.008" but might be higher
  - minimum copper to copper spacing - often 0.008" but might be larger
  - minimum annulus around PTH - often 0.008" but might be larger
- boards may be cheaper or delivered more quickly if ALL these parameters are exceeded by a sufficient margin

### Board Manufacturer's Drill Rack

- olimex uses 8 standard sizes from 0.7mm to 3.3mm - other sizes are drilled at extra cost
  - design Pad-Stacks to use standard drills
- use any 'Custom' or 'Favourites' pad-library facility offered to make finding and re-using pads easier  
or simply keep a record of any pad-stacks created to match your chosen board manufacturer's specifications

### Finally, if you are proposing to use a diminutive component:

- carefully read and understand the mechanical drawing contained in the datasheet
  - do an 'on-paper' design of the footprint specifying:-
    - pad pitch
    - pad dimension(s)
    - hole size
  - compare the resulting parameters - as above - with the board manufacturer's capabilities
  - abandon part if ANY 'copper' parameters are not met
- it might still be possible to meet the copper parameters by choosing a non-standard hole size

# Managing Libraries

## CAD Programs are shipped with many Library parts

- many - or even most - of which you will never use!
- often created to different standards - inconsistent size, pin naming and 'look'
- but still your favourite component is not included

## Learning to manage Libraries:

- allows the easy selection of known 'tried and tested' entities
  - once a component or footprint has been proven it can be re-used with impunity
- permits a consistency of look-and-feel across components and footprints
  - new entities can be created using existing ones as a starting point - also reduces errors
- reduces the 'pick time'
  - multiple libraries permit grouping of component 'type' - e.g. TTL ICs, analogue ICs, passives
  - listed entities are restricted to only those previously stored
- reduces the time required to realise the project and reduces errors
  - most designs re-use many components and footprints - which have already been verified
  - only need to create the - relatively few - new entities

# Managing Libraries

## Schematic Component Libraries

- Use multiple libraries if your CAD system permits this
  - segregation of component types - no need to wade through 100 TTL devices to get an op-amp
- Use generic or multi-device components if possible
  - reduces the number of distinct Components contained in the library - speeds up your search
  - similar Components created from the same basic library entity look and behave the same

## If the above facilities are not offered

- Use the Custom Library facility
  - import selected components from the standard library and save them in the Custom Library
  - use a 'type prefix' to speed up alphabetical component search - 'OA-LM741' or 'PC-Resistor'
- Use any Favourites facility
  - add your most-used components to the Favourites list - even if you have a Custom library
  - actively manage the Favourites list to prevent it growing too large

## If you discover an error

- Fix It! - and fix it in the Library NOT the job!
  - if you don't fix the problem in the Library you will simply perpetuate the mistake

# Managing Libraries

## PCB Footprint Libraries

- Use multiple libraries if your CAD system permits this
  - allows segregation of footprint types - DIL ICs, SMD parts, TH Passives, Connectors etc
  - each Library can have its own restricted, appropriate and proven Pad Stack list when creating a new part from scratch only suitable Pad Stack options are presented
- Create generic PCB Footprints
  - provided pad numbers are allocated consistently they can be used by multiple devices use a 'derivative suffix' - SOT23A / SOT23B - to indicate a different pad number assignment
  - similar Footprints derived from the same basic library entity look and behave the same

## If the above facilities are not offered

- Use the Custom Library facility
  - import selected components from the standard library and save them in the Custom Library
  - use a 'type prefix' to speed up alphabetical component search - 'IC-DIL8P300' or 'TH-AX050204'
- Use any Favourites facility
  - add your most-used components to the Favourites list - even if you have a Custom library
  - actively manage the Favourites list to prevent it growing too large

If you discover an error ... ?

# Managing Libraries

## Linking Schematic Components to PCB Footprints

- Maxi-CAD will usually provide a way of doing this automatically
  - manual entry into the Component's Part Field during Schematic Capture defines the Footprint
  - a post-process that 'stuffs' the appropriate information into the database from a list
- Mini-CAD probably doesn't provide this feature
  - use a manual table-based method using either a spread-sheet or document format file

Component Footprint Database

Component	Value	Rating	Pri TH F/P	Sec TH F/P	SMD F/Print
Capacitor Elec	10uF	16V	CR01020DIA		
Capacitor Tant	47uF	10V	CR02030DIA		SMCATED
Resistor Standard	All	0.25W	RA04050015		SM1206
Resistor Standard	All	0.50W	RA05060020		SM2010
Inductor B82477 Series	All	All			SMPWRL12MM
IN4000 Series	All	---	DA04050015		
GF1 Series	All	---			SMSMA
T1 LED	All	---	DR01020DIA		
BC182	---	---	TO92A		
TIP120	---	---	TO220H	TO220V	
LM741	---	---	8DIP300		SM08SO-150
LM2594 Series	All	---	8DIP300		SM08SO-150
LM2595/LM2596 Series	All	---	TO220H-05	TO220V-05	TO263-05
PIC16F876-04	---	---	28DIP300		SM28SO-300

# In Summary

- Standardising the Drawing Grid gives Design Consistency
  - reduces tracking issues due to dissimilar basic grid types
- Drawing a Schematic Diagram:-
  - reduces design errors - missing / inappropriate wiring connections are easily spotted
  - acts as visual reference during component placement and routing
- Creating Custom Component and Footprint Libraries:-
  - reduces design errors - pick from and use only verified entities
  - removes 'unwanted' parts from the selection process - improves speed
- Managing Libraries:-
  - is not difficult - it is simply a matter of being organised
  - pays for itself many times over in time/money saved on future projects

these facilities, once mastered, will lead to easier and better designs

# Acknowledgement

## ExpressPCB

The screen-shots used in this presentation were taken from ExpressPCB  
A US-based provider offering an integrated CAD & Board Manufacture service

Their System includes:-

Free easy-to-use CAD software comprising:

- Schematic Capture
  - multi-sheet schematic diagrams
  - creation of custom components and symbols and library maintenance
  - internal-format netlist for use by PCB Layout program
- PCB Layout
  - up to 4 layers (2 track + 2 plane) with component-side silk-screen
  - creation of custom footprints and library maintenance
  - import netlist to allow highlighting of net nodes during tracking

PCB Manufacture and Shipping

- Cost the board within the CAD program based on your design choices
- Order the board(s) from within the Layout program or upload the design files

For more information visit: <http://www.expresspcb.com>

# A Quick Guide to CAD

*That's all Folks*