PCB design with Mentor Graphics PADS

- a few tips / guidelines:
  - start from what someone has already done ("good artists borrow; great artists steal" -M. Andrew, 2018)
  - number all your 10k Ohm resistors R1..N, all your 50 Ohm resistors RT1..M (ditto capacitors, etc)
  - get to know your keyboard equivalents:
    - schematic ("PADS logic"):
      - F2 - start wire
      - ctrl-space - add "ground"
      - shift-space - add "power"
      - alt-space - add "off-page"
    - layout ("PADS layout"):
      - F2 - start routing a trace
      - ctrl-e - move
      - ctrl-r - rotate
      - l1 - switch to layer 1 and draw it in front of the others (similar for l2, l3, l4...)
      - F6 - select whole net
      - home - zoom to whole board
      - alt-B - mirror the view
      - check your gerbers in an external program
  - our example project is a LVDS-to-TTL converter board
PCB design with Mentor Graphics PADS

- add a voltage regulator
- add necessary/recommended discrete components (resistors, capacitors, etc)

3.3V to 2.5V Regulator

```
V_IN > 3V
IN
LT1963A-2.5
SHDN
SENSE
GND

2.5V
1.5A

+ 10µF*

+ 10µF*

SHDN
SENSE
GND

* TANTALUM, CERAMIC OR ALUMINUM ELECTROLYTIC

TOP VIEW

OUT 1
SENSE/ADJ 2
GND 3
NC 4
GND 6
GND 7
SHDN 5

S8 PACKAGE
8-LEAD PLASTIC SO

*PIN 2 = SENSE FOR LT1963A-1.5/LT1963A-1.8/
LT1963A-2.5/LT1963A-3.3
= ADJ FOR LT1963A
T_JMAX = 150°C, θ_JA = 70°C/W
```
PCB design with Mentor Graphics PADS

- add a LVDS receiver
- add a power supply decoupling capacitor

SN65LVDS2 and SN65LVDT2
DBV Package
(TOP VIEW)

110-\( \Omega \) Resistor for LVDT Only

M. Andrew

Coffee and Technic
- add a power connector
- add a LEMO connector
- add some test points if you like
- connect everything together
- only after you've verified the design makes sense in schematic, push it to layout
PCB design with Mentor Graphics PADS

- initial view in layout is something like this...
PCB design with Mentor Graphics PADS

- do rough part placement/rotation to make everything close to where it needs to be
- this may involve flipping some components to the other side of the board; depends what your mechanical constraints are
PCB design with Mentor Graphics PADS

- connect the dots
PCB design with Mentor Graphics PADS

- add copper planes for "power" and "ground"
PCB design with Mentor Graphics PADS

- "pour" the copper planes
"pour" the copper planes
PCB design with Mentor Graphics PADS

- add vias for powers/grounds that are not yet connected
PCB design with Mentor Graphics PADS

- move/rotate silkscreen so it will still be visible after fabrication and assembly
- it helps to view just top_copper+top_silkscreen
PCB design with Mentor Graphics PADS

- then switch to view just bottom_copper+bottom_silkscreen
- alt-B mirrors the view, which is useful when viewing the bottom layer with text not mirrored
PCB design with Mentor Graphics PADS

- check design rules (DRC/clearance), fix any problems
PCB design with Mentor Graphics PADS

- check layout versus schematic (LVS/connectivity), fix any problems
PCB design with Mentor Graphics PADS

- repeat DRC and LVS until everything is resolved
- generate gerber files
PCB design with Mentor Graphics PADS

- check gerbers with an external program
PCB design with Mentor Graphics PADS

- check gerbers with an external program
- if they look good, send them off to fabrication house (or bring upstairs to PCB mill)
PCB design with Mentor Graphics PADS

- if you need to define a new part (or add a package to an existing part):
  - start by making the PCB/layout decal if you don't already have it
  - then enter pin definitions
  - then assign the pins to gates as appropriate
  - then make the gates if necessary
PCB design with Mentor Graphics PADS

- from part editor in PADS layout, make the decal
- use the wizard if you like
- save the decal to your library (optional)
PCB design with Mentor Graphics PADS

- from part editor in PADS logic, assign the decal you just made
- enter pin definitions
- assign the pins to gates as appropriate
PCB design with Mentor Graphics PADS

- make the gate(s)
- save the part to your library (optional)
- add the part to your schematic

**ADCMP580/ADCMP581/ADCMP582**

**FUNCTIONAL BLOCK DIAGRAM**

- VCC IN5
- VCC IN16
- VCC OUT9
- VCC OUT12
- IN+
- IN-
- OUT
- OUT
- COMPARE
- VTERM
- VTERM+
- VTERM-
- HYS
- VEE
- GND
- COMP1
PCB design with Mentor Graphics PADS

- wire it up
- ECO-to-PCB
PCB design with Mentor Graphics PADS

- next time: quick-and-dirty PCBs with the PCB mill...