

# CA Design

## Allegro PCB Layout (Editor)

3 days

### Day 1

A – Over-view of Librarian  
Sample part 1 – CAD01

B – Starting a new board  
“Start Board” discussion  
“New Board” exercise  
Board outline  
IDF Import/Export  
Route Keep-in  
Place Keep-in  
Mounting Holes

C – Loading a new net list  
Discuss File > Import > Logic

D – Placement Bin  
Place > Quick-place

E – Schematic cross probe

F – Set-up User Preferences

G – Mechanical placement

H – Rooms

I – Placement Exercise

J – Copy and Assign

K - Mirror

L – Height areas

M – Placement Review

N – Export Placement

## Day 2

A – Review

Parts, Logic Import, Placement, Quick-Place, IDF Import/export

B – Discuss Layer Stack-up and “Rules”

C – Set-up Layer Stack-up

D – Constraint Manager

Set-up Physical Rules

Set-up Spacing Rules

Set-up Diff Pairs

E – Assign Diff Pairs

F – Preliminary Route exercises

Break-out vias

Critical Routes

Access the Autorouter

G – Ground Plane

H – Split Power Plane

Text-Lx-Neg layers

I – Set-up shape parameters

J – Fill areas

K – Copy vias

L – DRCs

M – Status file

N – Highlight

O – Shadow Mode

P – Edit Padstack

Q – Swap Pins/Gates

R – Alternate Parts

S – Re-number Ref Des and Back-annotate

## Day 3

A – Review

B – Off-board Text

C – Set-up Layer Gerbers

D – Set-up Fab Drawing

E – Import/Export Subdrawing

E – Set-up Assembly Drawing

G – Set-up Panel Drawing

H – Run Post Processor

I – Required Fab/Assembly Files

J – Valor ODB++ files

K - Questions