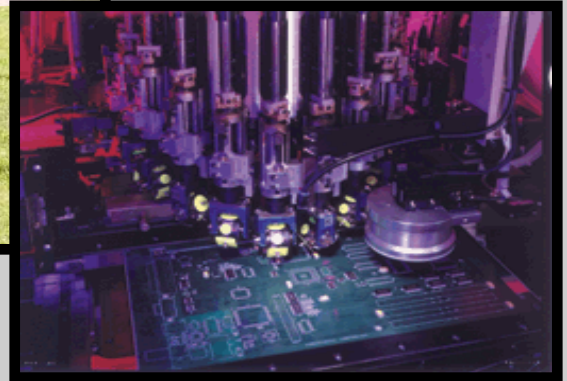
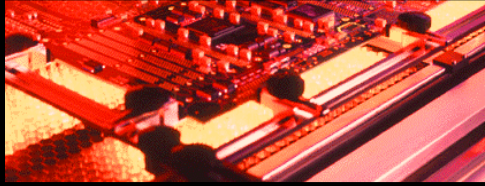


# Benefits of DFX



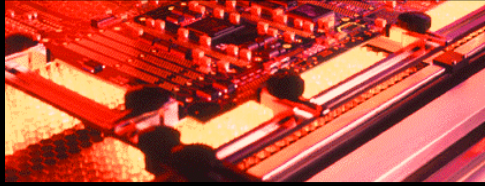
714.979.2228 - Tel  
714.545.1888 - Fax



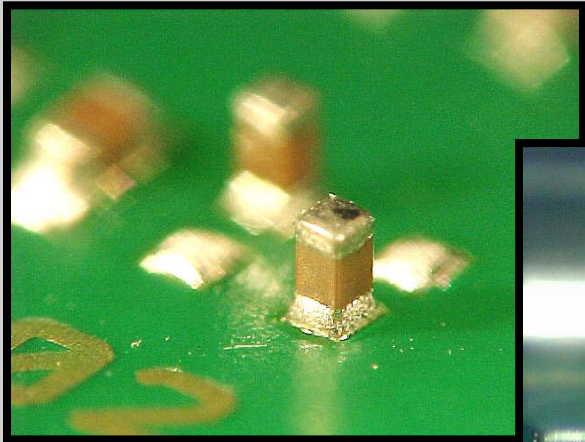
# Benefits of DFX

**Typical Manufacturing defects that are design-related:**

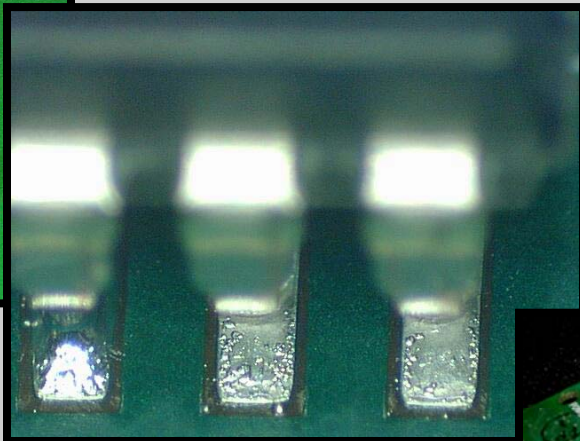
- Tombstoning
- Via in pads
- Component misalignment
- Insufficient solder, poor solder quality
- Components (under the same AVL) mismatch
- Component-to-layout footprint mismatch



# Benefits of DFX



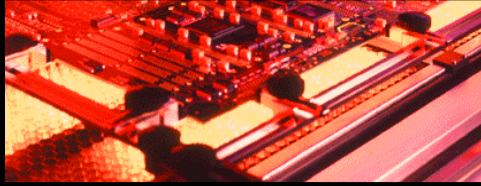
Tombstoning



Via in Pads—Solder Splash



Component Misalignment



# Benefits of DFX

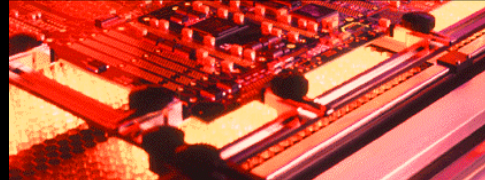
## What is DFX?

DFX stands for **D**esign **F**or **eX**cellence. It is a pre-assembly analysis that enables one to find and minimize design related issues that compromise manufacturing quality; decrease product cost and increase product reliability.

DFX should be a part of the NPI process.

## EMI will check and review the following areas:

1. BOM integrity – double references; quantity that does not match
2. Material pipeline analysis (optional)
3. DFF
  - a) IPC netlist is compared with the Gerber files
  - b) bare board potential fab-related issues
    - b1) gross errors – double drills
    - b2) quality & process related issues
4. DFM (manufacturability, machine interface)
5. DFA (assembly) – Component Package overlay/Pad geometry/multiple AVL overlay
6. DFT (Design for Test)



# Benefits of DFX

## Benefits of DFX

- Minimize design induced manufacturing issues
- Minimize PCB re-spin frequency (typical PCB re-spin cost around \$2K - \$10K)
- Reduce overall cost
- Increase product reliability
- Increase Time To Market
- Can alert QC/inspection station to look for potential areas as found in the DFX report

## Entry Criteria

- BOM with AVL
- Gerber
- Layout CAD files (not just Gerber files)

## Duration of DFX

- Typical duration is 2–4 working days – depending on the complexity and format that is available to EMI.

## Who should use DFX:

1. NPI is the target profile. The design is not finalized yet and can still take input for changes.