

Specifying Impedance Controlled Printed Circuit Boards

Alec Groves – BAE Systems Insyte, Isle of Wight

the power to see :

the information
to decide :

the knowledge
to command :

Specifying Impedance Controlled PCBs

INTRODUCTION

WHY do we need control of impedance

Z axis becomes important

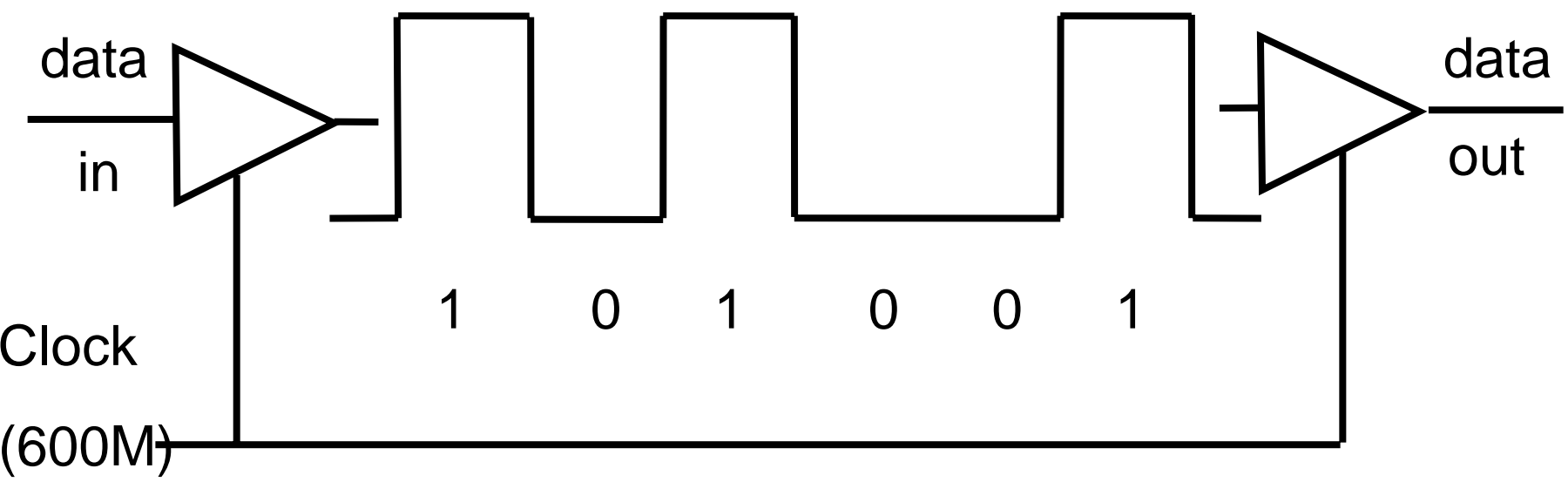
Potential skills gap

Specifying Impedance Controlled PCBs

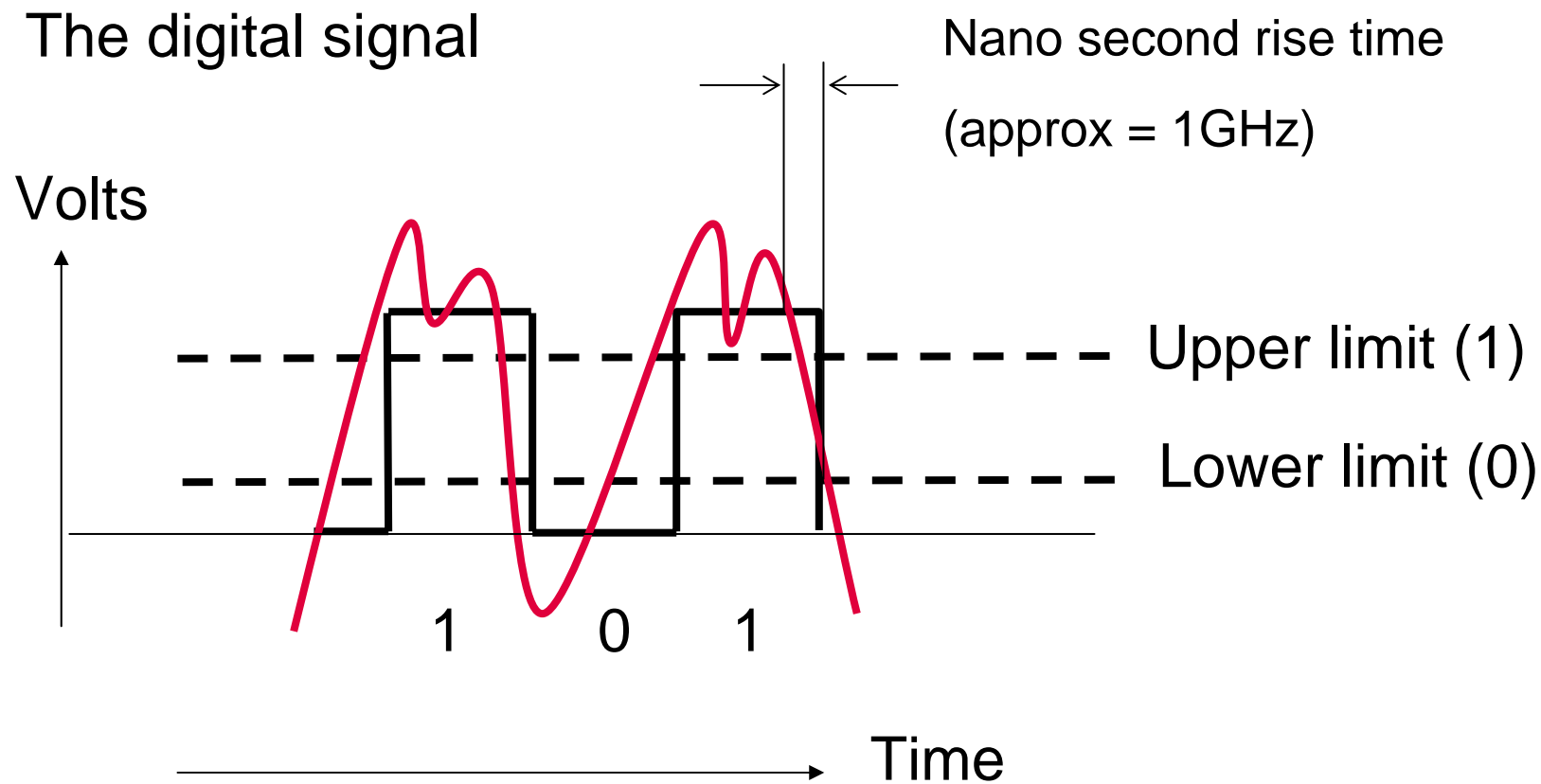
“Learning how to specify printed circuit boards is a continual journey of enlightenment”

Specifying Impedance Controlled PCBs

The digital signal (2G data rate)

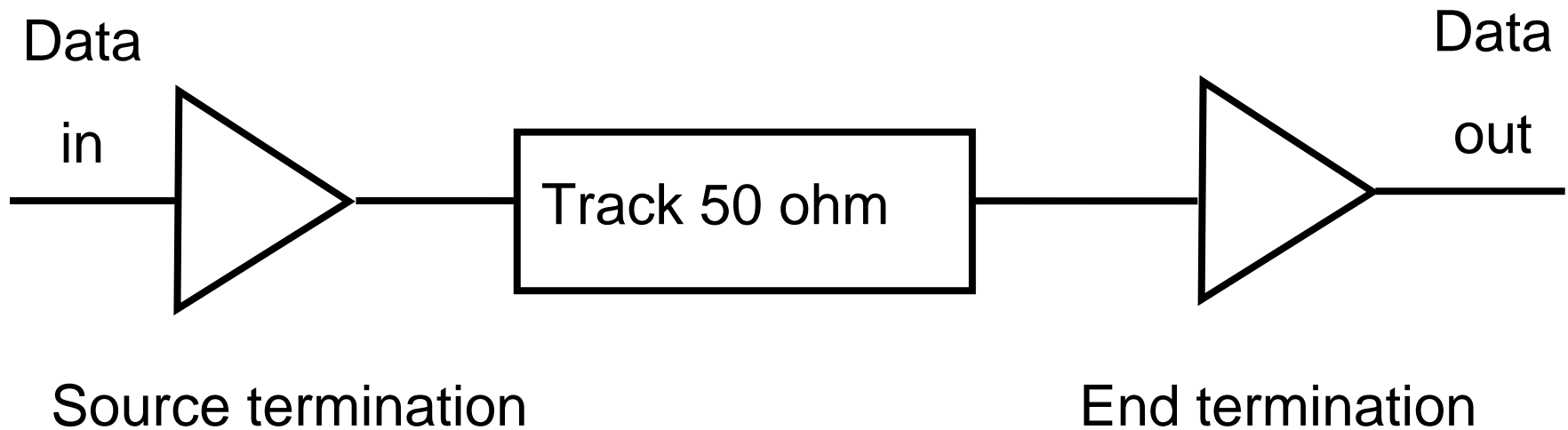


Specifying Impedance Controlled PCBs



Specifying Impedance Controlled PCBs

INTRODUCTION



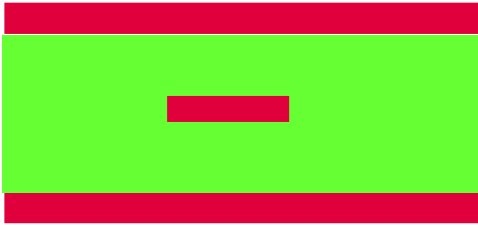
Specifying Impedance Controlled PCBs

INTRODUCTION

If all circuit components are not matched, the signal can be distorted and / or signal reflections can occur

Specifying Impedance Controlled PCBs

Z Axis becomes important



Example controlled impedance track feature

Track width (and thickness)

Z axis spacing

Laminate electrical properties

Specifying Impedance Controlled PCBs

Z axis has become important

Use of power / ground planes

Increasing circuit density

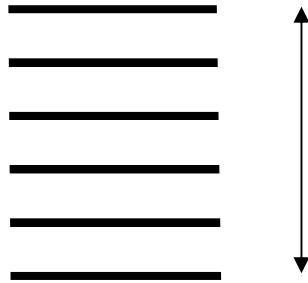
The need to segregate signals

High layer counts

Teams of Design Engineers involved

Specifying Impedance Controlled PCBs

The Design data shall describe all features that effect “fit”, “form” and “function”.



6 artworks 1.6mm thick



6 artworks individual spacing specified

Specifying Impedance Controlled PCBs

Not all 6 layer 1.6mm thick PCBs are made the same.

Fabricators can have different preferred materials

Different presses with the same materials can result in different dielectric thickness

PCB construction must be achievable

CAD tools of little assistance

Specifying Impedance Controlled PCBs

The I/C data advises 50 ohm +/- 10 ohm

The Design Engineer needs to cope with this requirement along with other signal integrity issues

Formulae and software of varying degrees of sophistication are available to help

IPC Guidelines are available (IPC-2141 Design Guide for High – Speed Controlled Impedance Circuit Boards)

Feature sizes, material availability, manufacturing tolerances, laminate electrical performance tolerances – all must be considered.

Specifying Impedance Controlled PCBs

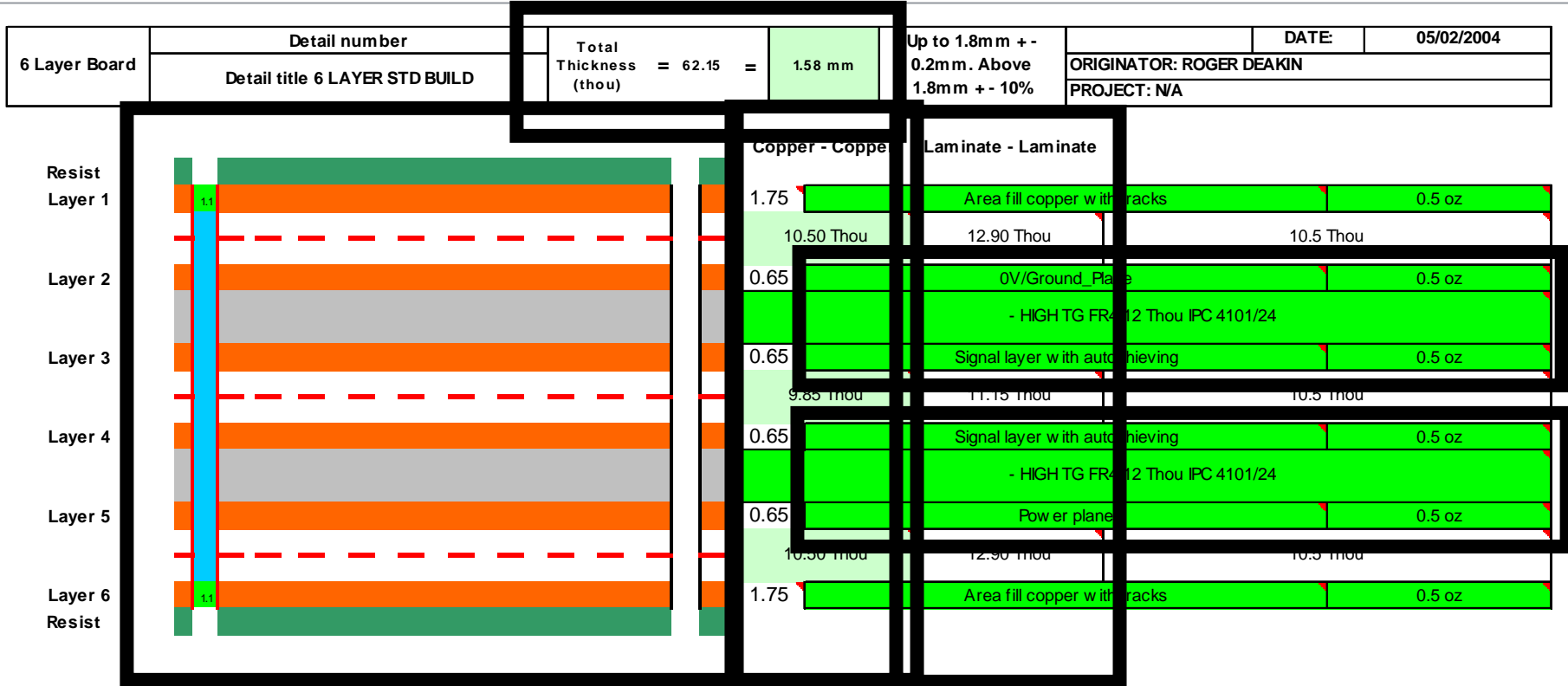
The “Design Authority” is responsible (fit, form and function)

The PEC Fabricator has most of the knowledge.

Impedance control – not an exact science (microwave)

There is a clear need for a design aid – new requirements to be specified.

Specifying Impedance Controlled PCBs



Key for board section (view above):

- Solder Resist
- Laminate
- Un-plugged vias and microvias
- Plated Through-Hole
- Copper
- Sequential Build Point
- Blind Via (filled with resin during bonding)
- Non-Plated Through-Hole
- Pre preg

Specifying Impedance Controlled PCBs

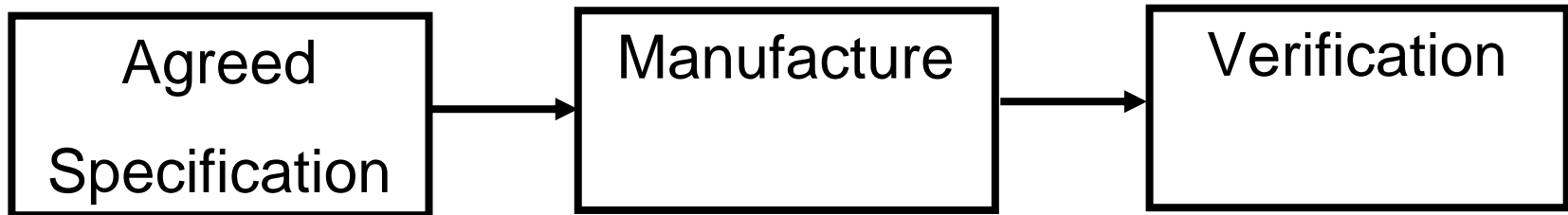
With an eye on via aspect ratio and cost, tool used in the early design stages.

Many “standard” builds agreed with our Fabricators.

Data transferred to Engineering Drawings.

Specifying Impedance Controlled PCBs

The impedance controlled track is a “component” and can’t be tested in circuit.



Specifying Impedance Controlled PCBs

Verification:

IPC-6012 Qualification and Performance Specification for Rigid Printed Boards

IPC-6018 Microwave End Product Board Inspection and Test

Specifying Impedance Controlled PCBs

Thank you for listening

Are there any questions?