Welcome
AltiumLive 2018
University Day

Instructor
John Watson CID
Senior PCB Engineer
Legrand North America
Exploiting Routing Technologies

Review the various options within Altium Designer for both manual
Exploiting Routing Technologies

Agenda

• Introduction
• Preliminary Routing Necessities
• Golden Rules of Routing
• Routing Process Flow
• Routing Options
• Special Routing Tools
• Manual Route Techniques
• Why not just run the Autoroute
• Interactive Route Techniques
• Special Routing Considerations for High Speed and High Power Designs
• Conclusion
Golden Rules of Routing
from Printed Circuit Handbook 7th edition By Happy Holden pg. 463)

1. GRID, GRID, GRID
2. Each signal layer should have routes in the same general direction (E-W or N-S)
3. Do not route Traces across Splits in planes this cuts off signal return paths
4. Do not route high speed traces near the edges of the PCB due to EMI Control
5. Trace width should be no more than 60% of the Pad size due to heat dissipation issues
6. Don’t route signals through a BGA that have nothing to do with that BGA
7. Keep input and output signal separated so they cannot interfere with each other
8. Spread out Routing whenever possible to minimize crosstalk
9. Keep all routing within its own power or ground region for return purposes
10. With trace length tuning, be sure there is three to four time the trace width between loops to the keep the signal from jumping across segments.
Routing Process Flow

1. Route Power connections in a fanout to vias
2. Route I/O connections
3. Route critical nets
4. Protect or “lock” critical until/unless changes are necessary
5. Analog signal (and parts) next and isolate from digital
6. Any location specific signals
7. General routing
Exploiting Routing Technologies

Preliminary Routing Requirements
Command: Setup System Preferences > PCB Editor > Interactive Routing

Personal Settings
- Routing Conflict Resolution
- Interactive Routing Options
- Routing Gloss Effort
- Dragging
- Interactive Routing Width Sources
- Favorites
Preliminary Routing Requirements

Command: Design > Layer Stack Manager

Four factors for all stack-up considerations:
1. The number of layers
2. The number and types of planes (power and/or ground) used
3. The ordering or sequence of the layers.
4. The spacing between the layers.
Preliminary Routing Requirements

Command: Design>Layer Stack Manager

- PCB Routing Preferences
- Layer Stack-up
- Design Rules
- Net Classes

5- Objectives you should try to achieve

1. A signal layer should always be adjacent to a plane.
2. Signal layers should be tightly coupled (close) to their adjacent planes.
3. Power and Ground planes should be closely coupled together.
Preliminary Routing Requirements (Con)

Command: Design>Layer Stack Manager

4. High-speed signals should be routed on buried layers located between planes. In this way the planes can act as shields and contain the radiation from the high-speed traces.

5. Multiple ground planes are very advantageous, since they will lower the ground (reference plane) impedance of the board and reduce the common-mode radiation.
Preliminary Routing Requirements

Command: **Design>Design Rules**

**PCB Routing Preferences**
- Layer Stack-up
- Design Rules
- Net Classes

**STATS**
- 10- Major Categories
- 62- Minor Categories

**ENDLESS Variations of Rules**
Preliminary Routing Requirements

Command: **Design>Design Rules**

- **PCB Routing Preferences**
- **Layer Stack-up**
- **Design Rules**
- **Net Classes**

**STATS**

10- Major Categories

- **ENDLESS Variations of Rules**

- Will be used to guide the routing
**Preliminary Routing Requirements**

Command: Design > Classes

- PCB Routing Preferences
- Design Rules
- Layer Stack-up
- Net Classes

Used to Organize any size PCB Design. By various Categories

Keep in mind that the DRC checks can be setup based on any net class
Routing Options
Criteria: When Routing showing Properties
Command: Place>Track

6 Categories
Net Information
Properties
Interactive Routing Options
Rules
Visualization
Help

• When starting the Route and hitting the Tab key will give access to the Properties Window.
Routing Options

Criteria: When Routing showing Properties
Command: Place>Track

- Net Information
- Properties
- Interactive Routing Options
- Rules
- Visualization
- Help

Specific Details of a net
Net Name: Sleep_Amp/GPIO
Net Class: N/A
Length: 1531.951mil
Routing Options
Criteria: When Routing showing Properties
Command: Place>Track

Key Strokes
Key ‘4’ - Via Hole Size
Key ‘3’ - Width
Shift + R - Routing Mode
Shift + Space - Corner Style
Shift +CTRL + G - Routing Gloss Effort
Shift + D - Routing Selection Choices
Key ‘5’ - Follow Mouse Trail
Shift + C - Pin Swapping
CTRL + W - Display Clearance Boundaries
Shift + G - Show Length Gauge
Routing Commands Net Information

Criteria: When Routing showing Properties
Command: Place>Track

Key Strokes
- Key ‘4’ - Via Hole Size
- Key ‘3’ - Width
- Shift + R - Routing Mode
- Shift + Space - Corner Style
- Shift + CTRL + G - Routing Gloss Effort
- Shift + D - Routing Selection Choices
- Key ‘5’ - Follow Mouse Trail
- Shift + C - Pin Swapping
- CTRL + W - Display Clearance Boundaries
- Shift + G - Show Length Gauge
Routing Commands Net Information

Criteria: When Routing showing Properties
Command: Place>Track

Specific Rules Applied to that net
Routing Commands Net Information

Criteria: When Routing showing Properties
Command: Place>Track

Key Strokes

Key ‘4’ - Via Hole Size
Key ‘3’ - Width
Shift + R - Routing Mode
Shift + Space - Corner Style
Shift + CTRL + G - Routing Gloss Effort
Shift + D - Routing Selection Choices
Key ‘5’ - Follow Mouse Trail
Shift + C - Pin Swapping
CTRL + W - Display Clearance Boundaries
Shift + G - Show Length Gauge
Routing Commands Net Information

Criteria: *When Routing showing Properties*

Command: **Place>Track**
Special Routing Tools
Criteria: With route(s) selected.
Command: Route>Gloss Selected

Gloss
A PCB tool used to clean up routing. By optimizing routing
Special Routing Tools

Criteria: In the PCB
Command: Route>Fan Out

- Gloss
- Fanout
- Slicing
- Swapping
Special Routing Tools
Criteria: In the PCB
Command: Edit>Slice Tracks

Slicing Routes
A PCB tool used to easily slice connections in Routes

- Gloss
- Fanout
- Slicing
- Swapping
Exploiting Routing Technologies

Special Routing Tools
Criteria: In PCB
Command: Part Actions » Configure Pin Swapping or Component Actions

Swapping Categories
Three categories of swapping:
- Pin Swapping
- Differential Pair Swapping
- Sub-part Swapping
Must first be configured

Configuring Swapping
Based on Groups
Pins can be grouped by various methods
Manual Routing Techniques
Criteria: In the PCB
Command: Place>Track

- **Manual Routing**

<table>
<thead>
<tr>
<th>Key Strokes</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Key ‘4’ - Via Hole Size</td>
<td></td>
</tr>
<tr>
<td>Key ‘3’ - Width</td>
<td></td>
</tr>
<tr>
<td>Shift + R - Routing Mode</td>
<td></td>
</tr>
<tr>
<td>Shift + Space - Corner Style</td>
<td></td>
</tr>
<tr>
<td>Shift + CTRL + G - Routing Gloss Effort</td>
<td></td>
</tr>
<tr>
<td>Shift + D - Routing Selection Choices</td>
<td></td>
</tr>
<tr>
<td>Key ‘5’ - Follow Mouse Trail</td>
<td></td>
</tr>
<tr>
<td>Shift + C - Pin Swapping</td>
<td></td>
</tr>
<tr>
<td>CTRL + W - Display Clearance Boundaries</td>
<td></td>
</tr>
<tr>
<td>Shift + G - Show Length Gauge</td>
<td></td>
</tr>
</tbody>
</table>
Why not just run Autoroute?
Interactive Routing Techniques

- Create the net classes
- Shelve existing polygons
- Configure the design rules
- Configure the net topology
- Fanout the design
- Route the power and ground
- Enable the required routing layers

- Setting Up Interactive Routing
  - Multi-Routing
  - Differential Pair Routing
  - Differential Pair Length Tuning
  - Length Tuning
Interactive Routing Techniques

- Multi Routing
- Differential Pair Routing
- Length Tuning
- Differential Pair Length Tuning
AltiumLive 2018
Questions?