PADS-PowerPCB 4 Tutorial (with Blazeroute)

PADS-PowerPCB is the ultimate design environment for complex, high-speed printed circuit boards.

PROCEDURE FOR SIMULATION IN SCHEMATICS

- 1. Importing Design Data from PowerLogic or Using ECO mode to draw the circuit,
- 2. Edit component,
- 3. Setting the PCB,
- 4. Routing Connection,
- 5. Creating the Copper Pour,
- 6. Printing.

PowerPCB version 4.0:

STARTING PADS

1. To open a PADS-PowerPCB :

Demo Version :

PrePowerPcb PowerPcbv40

1. Click "PrePowerPcb Demo" icon,

Demo

2. Click "PowerPcbv40 Demo" icon to open DEMO version.

Full Version :

1. Click "PrePowerPcb V40" icon,

2. Click "PowerPcbv40 Demo" icon to open FULL version.



Importing Design Data

To avoid missing the connection line, Design data can be imported from PADS-PowerLogic.



Demo

To open the PADS-PowerLogic,

Double-click on the **powerlogic-4** icon.

Start PowerLogic and draw the schematic file.

Click **Design** icon, and then click the **add part** icon.



Select Part Name in Add Part windows.

Or Click Browse to search New Part Name to select new part.

| Add Part Part Name: OK OK | ■ Browse Help | |
|---|--|------------------------------|
| RES-1/8W | Part-Types: misc:RES-1/2W misc:RES-1/2W misc:RES-1/4W misc:RES-1W misc:RES-2W misc:RES-2W misc:RES-2W misc:RES-0IP147RES misc:RES-0IP168RES | OK Cancel <u>H</u> elp |
| Filter Li <u>b</u> rary: [All Libraries] Ite <u>m</u> s: R* | Apply | |



Click Add Connection icon to connect the components.



2

After connect all the components. Then you can use two methods to send the design data to PowerPCB.

- 1. Using Netlist to PCB method with .asc file.
- 2. Using OLE PowerPCB Connection method. (Need open PADS-powerPCB first !)

1. Using Netlist to PCB method :

Select Tools/Netlist to PCB to send a Netlist to PADS-PowerPCB.



Save the output file name *default.asc*.



Open PADS-PowerPCB, Select **File/Import** to import .asc file.

2. Using OLE PowerPCB Connection method.

Select Tools/OLE PowerPCB Connection.

| Tools | <u>W</u> indow <u>H</u> elp |
|----------------|-----------------------------|
| Part | Editor |
| Part | Attribute <u>M</u> anager |
| Upda | ate Off-page from Library |
| Upda | ate Pins from Library |
| <u>S</u> ave | Off-page to Library |
| <u>N</u> etli: | st to PCB |
| Eorw | ard ECO to PCB |
| Back | ward Annotate from PCB |
| <u>E</u> xpo | rt Rules to PCB |
| Impo | rt Rules from PCB |
| O <u>L</u> E | PowerPCB Connection |
| OLE | BlazeRouter Connection 😽 |
| Visua | al Basic Scripting |
| Macr | OS |

Select **Design** tab from PowerLogic's OLE PowerPCB Connection dialog box.

Click Send Netlist icon to automatically export a netlist to PADS-PowerPCB as a netlist import.

| Selection | Design D | ocument P | referer | nces | |
|-------------------------|-----------|--------------------|---------|----------------------|---|
| Con | npare PCB | <u>R</u> ules To F | РСВ | Send <u>N</u> etlist | Ţ |
| Synchronize <u>P</u> CB | | Rules From PCB | | Synchronize SCH | |

Disperse Components

To arranges all of the selected objects on grid sites without overlapping.

After import or send design to PADS-PowerPCB, all components are overlapping.

Select Tools/Disperse components to disperse the components.

| <u>I</u> ools <u>W</u> indow <u>H</u> elp | |
|---|---|
| Decal Editor | |
| Automatic <u>C</u> luster Placement | |
| Disperse Components | 1 |
| Length Minimization K Ctrl+M | |

4



Before dispersion :



After dispersion :



Using ECO

Use the **ECO** Toolbox to perform (Engineering Change Order) operations, which modify the net list, or to add a new component and connection.



Click ECO icon, save the eco file. Click OK.

| Z Autore ECO file | |
|---|--------------|
| Append to file. | Cancel |
| Filename: | |
| C:\Temp\pwr\Untitled.eco | <u>H</u> elp |
| Browse Write ECO file after glosing ECO toolbox. | |
| Browse Write ECO file after closing ECO toolbox. Note: This will clear undo buffer. | |
| Browse Write ECO file after closing ECO toolbox. Note: This will clear undo buffer. ECO Output Options Attribute Expansion | |
| Browse Write ECD file after closing ECD toolbox. Note: This will clear undo buffer. ECD Output Options Attribute Expansion Expand Part Attributes | |

Click Add Component icon to add component.



Decal Editor

PADS-PowerPCB use components from the parts libraries. Every part in the library has a decal associated with a part type in a parts library.

Use the Decal Editor to create and/or edit these decals.

1. Create new Decal

Select Tools/Decal Editor.

|) C | :\TE | MP\tes | st.pcb - | Power | PCB | |
|--------------|--------------|--------|---------------|-------------|---------------------|--------------|
| <u>F</u> ile | <u>E</u> dit | ⊻iew | <u>S</u> etup | Tools | <u>W</u> indow | <u>H</u> elp |
| G | 2)L | | l) Top | <u>D</u> ec | al Editor | |
| | | | | Auto | matic <u>C</u> lust | ter Place |

Click Drafting icon.

Click Terminal icon to draw terminal.



2. Create new library associate with new part.

Select **File/Library** to open the Library Manager dialog box.



Click **Decals** icon as the library type.

Click New Lib... icon to save a new library in TEMP folder.

3. Save the New Decal.

Select File/Save Decal to save decal in new library.

6

Select Library and type the name of decal.

| 📉 Decal: New - Powe | Save PCB Decal to Library | | × |
|---|---------------------------|----------|--------|
| <u>File</u> <u>E</u> dit <u>V</u> iew <u>S</u> etup | Library: | | |
| <u>O</u> pen Decal | C:\TEMP\test | <u> </u> | OK |
| Save Decal | Name of PCB Decal: | | Cancel |
| Save Decal <u>A</u> s ^N | connect] | | |

Select File/Exit Decal Editor to return to PADS-PowerPCB.

4. Add new Decal.

Click ECO icon and Click Add Component icon.

Select Library and Part Type.

Click Add icon.

| 🕻 Get Part Type f | rom Library - test | _ 🗆 × |
|-----------------------------|--------------------|---------------|
| CONNECT | Part Types: | |
| | test:CONNECT | <u>C</u> lose |
| | | Add |
| 000 | | Help |
| Filter Li <u>b</u> rary: | | |
| C:\TEMP\test Items: | | |
| 1* | Apply | |

Work Area and Grid Settings

1. Status Window :

The Status Window is a floating window with useful summary information, preference controls, and a viewing control.



The Status Window is a floating window that contains: •The net name and pin connects of of a selected part. •A Snap to Grid check box. Snaps the cursor to the current design grid. •The Postage Stamp viewer. Use it to pan and zoom.

2. Setting Origin :

The design positions are relative to the origin.



Select **Setup/Set Origin** then click on the workspace to indicate a new position for the origin.



3. Setting Grids :

Sets the spacing of the design grid which controls the general placement of parts.

Select Setup/Preferences and select the Grids tab to view or set the current display grid setting.

| Setun Tools Window Help | Preferences |
|-------------------------------------|--|
| Layer Definition | Global Design Routing Thermals Auto Dimensioning Teardrops Drafting Grids Split/Mixed Plane Fanout |
| Drill Pairs Jumpers | Design Grid |
| Design <u>R</u> ules | Y: 100 Y: 25 Y: 25 |
| Preferences Ctrl+Alt+G ECO Prefs | Image: Signap to Grid Image: Signap to Grid Image: Signap to Grid Image: Signap to Test Point Grid |
| Set <u>O</u> rigin | Display Grid Hatch Grid |
| Display <u>C</u> olors Ctrl+Alt+C | X: 1000 Copper: 10 Y: 1000 Keepout: 100 |
| | <u>B</u> adial Move Setup |
| | |
| | OK Cancel Apply Help |

| Hodeless C |
|--|
| Command: g500 |
| GRID format: G{null(RM/D} <n> {<n>) Turn POLAR GRID On/Off: GP</n></n> |

Or typing **G500** and pressing Enter to set the Design Grid in one step.

Set Grids to 500, typing G500 and pressing Enter.

Set the Display Grid to 200, typing GD200 and pressing Enter.

4. Setting Units of Measure :

Select **Setup/Preferences** to change the unit of measure to inches, mils (default setting 1000 = 1"), or metric units (1 mm).

The Design Units combo box is on the Global preferences tab.

Now, the current design units set to Mils.



5. Using the Selection Filter :

To focus your selections on specific objects, PowerPCB has a Selection Filter. The Selection Filter allows you to specify which design objects can be selected.

• To access and review the Selection Filter:

Select **Edit/Filter** to open the Selection Filter dialog box. Objects are organized into three categories—*Design Items*, *Drafting Items*, *and Layers*.

• Selection Filter Shortcuts :

Click the right mouse button while no object is selected, a pop-up menu containing a list of **Selection Filter** shortcuts appears.

Select one of these shortcuts updates the Selection Filter to include only the items in the shortcut description.

Select Nets shortcut and note how the selection filter is updated to allow selection of nets only.



6. Creating a Board Outline :

It defines the board boundary.



Click **Board Outline and Cut out icon** to Draw the shapes of board used for PCB design. (Default: Polygon [HP])

| Until | led - Po | werPCE | } | | | | | | | | |
|------------------------|------------------|---------------|-------|----------------|--------------|-------------|--------|-----|-----|---|---|
| <u>File</u> <u>E</u> d | lit <u>V</u> iew | <u>S</u> etup | Tools | <u>W</u> indow | <u>H</u> elp | | | | | | |
| B | | Тор | | - | 0 | 0 | × | 45 | *** | | ~ |
| K | 5 | | | | 0 | € bc | | | 2 | 2 | Q |
| X St. | | × | | E | Board O | utline a | nd Cut | Out | | | |

After draw the board outline to define the board boundary.

Click the right mouse button to open the pop-up menu, then choose *Complete*, or Double-click the left mouse button, to close and complete the polygon.

| Complete | <dbl_click></dbl_click> | | - C. | 5 | 151 | |
|------------|-------------------------|----|------|---|-----|--|
| Add Corner | <click></click> | | | | | |
| Add Arc | | 2 | | | | |
| Width | {W <nn>}</nn> | 27 | | | | |
| Layer | {L <nn>}</nn> | | | | | |
| Auto Miter | | | | | | |
| ✓ Polygon | (HP) | 35 | | | | |
| Circle | {HC} | | | | | |
| Rectangle | {HR} | | | | | |
| Path | {HH} | | | | | |
| Orthogonal | {AO} | | | | | |
| 🗸 Diagonal | {AD} | | | | | |
| Any Angle | {AA} | 54 | | | | |
| Cancel | Esc | | Ŷ | | | |
| | | | | | | |
| | | | | | | |
| | | | | | | |



Then, Click the **Board icon [Ctrl-B]** from the toolbar to fit the board outline to the screen.

7. Setting the Layer of the PCB :

To assign the number of layers, the nets associated with embedded plane layers, layer stackup, and layer thickness.

Select Setup/Layer Definition. The Layers Setup dialog box appears.

| 📉 Layers Setup | × |
|---|-------------------|
| Lev. Type Dir. Name | |
| 1 CM H Top | ОК |
| 3 GN Layer_3 | Canaal |
| 4 GN Layer_4 5 GN Layer_5 | |
| 6 GN Layer_6 7 GN Layer 7 | Help |
| 8 GN Layer_8 | |
| Name: Top Electrical Layer Type Image: Component in the componen | Associations |
| Electrical Layers Count: 2 <u>I</u> ncrease <u>R</u> eassign | <u>T</u> hickness |

8. Setting Layer Colors :

Select Setup/Display Colors. The Display Colors Setup dialog box appears.

| Setup | | | | | | | | | | | | | | | _ | |
|-------|------------|--|---|--------|--------|---|--|--|--|--|---|---|--|--|--|--|
| | | | | | | | | | | | | | Pal | lette. | | |
| 0.0 | 0.0 | C | 0 | œ | c | c | c | 0 | 0 | | | C |)e <u>f</u> au | lt Pa | ette | |
| Apply | fo All Obi | ects | □ | esigr | n Item | ns | | | | La | bels | | [01 | utline | s | |
| 18 | | | Pads | Traces | Vias | Lines | Text | Copper | Errors | Ref. Des | Type | Attribute: | Keepout | Top | Bottom | |
| | | | • | • | • | • | • | • | V | | | • | • | • | • | |
| | | ₽ | | | | | | | | | | | | | | - |
| | | 1 | Π | | П | | Γ | | | Π | | | | | | |
| | | 2 | | | | Π | | | | | | | | | п | |
| | | 2 | | | | | | | | | | | | | | |
| | | 2 | | | П | | Ē | F | | | F | F | Π | | | |
| | | ন | | | | Ī | | Ī | | | | Ē | | | | |
| | | | | | | Cor | nfigu | ration | , <u> </u> | | | | | | | |
| | Board (| Dutline | • [] | | | Г | | | | _ | _ | - | _ | _ | . 18 | - |
| | Connec | ction | Ľ | | | | <u>5</u> | ave | | | | | | Del | ete | |
| | | Setup C C C C Apply To All Obj S Board (Connec | Setup Apply To All Objects s Board Dutline Connection | Setup | Setup | Stup Apply To All Objects Board Outline Connection | Stup Apply To All Objects S Board Outline Connection | Stup Apply To All Objects S Board Outline Connection | Stup Apply To All Objects Board Outline Connection State Connection State Configuration State Configur | Stup Apply To All Objects Board Outline Connection Save | Setup Apply To All Objects S Board Outline Connection Save | Setup Apply To All Objects S Board Outline Connection Save | Setup Apply To All Objects S S S S S S S S S S S S S | Setup Period Apply To All Objects S S S S S S S S S S S S S | Setup Palette. Default Pal Acply To All Objects S Configuration Board Duttine Connection Configuration Configuration Configuration Description Configuration Description Configuration Description De | Setup Palette Palette Palette Default Palette Acply To All Objects Palette Design Items Palette Palette Design Items Palette Palette Palette Palette Palette Palette Palette |

9. Setting Design Rules and Trace width :

To assign and edit rules, or width, spacing and general routing. Select **Setup/Design**, Click **Default** icon and **Clearance** icon to set Trace Width.



10. Setting Pad Stack width :

Example From Get Part, Select SIP-1P.



12



Select the Pads of components. and click right mouse to select Query/Modify to modify Pad Stack.



Click the Pad Stack icon.

| Query/Modify | Pin | _ 🗆 × |
|--------------------------------------|-----------------------------------|--------------|
| Ref. Des.: R1 | | ОК |
| Pin No: 1 | | |
| Pin Name: | | Apply |
| Function: | | Cancel |
| Swap Id: | | |
| Net: | | <u>H</u> elp |
| Coordinates | | |
| X: -1000 |) | |
| Y: 4000 | | |
| Connectio □ □ Leard ☑ Plane | n: rops 🔲 Tes Thermal 🗖 Top | st Point |
| 00 | | |
| Pad Stack | Component | Attributes |

Input the Diameter of Pad. Click OK.

| Query/Modify Pad S | tacks - SIP-1P (Mils) | > |
|--|--|-------------------------------|
| Pad Stack Type © Decal © Via Decal Name: SIP-1P | Pin: Plated: Sh.: Sz.: Layer: All (P) CNN 60 <mounted side<br="">CNN 60 <inner layers=""> CNN 60 <opposite side<br="">CNN 80 Layer_25</opposite></inner></mounted> | e> OK e> Cancel |
| | Add Add Oglete Delete | Help |
| | Parameters Use Global Defaults | <u>List</u> |
| | Pad Style: Pad | |
| | Pad Size Relative to Drill Size | Preview |
| Add <u>Via</u> Delete Vias Name: | | |
| C Through C Pagti | al Drill Sige: 37 F Plated | |
| Start Layer. | Slot Parameters Slotted Length: 0 | Decal Units Mils Metric |

Click All or Selected components.



After increase the diameter to 150.



11. Change the pads shape:



Click left-click mouse on the pad of component.



Click right-click mouse to select [Query/Modify]



Click Pad Stack icon





Select the icon pad shape.

| Diameter: | Inner Diam: | |
|-----------|-------------|--|
| 90 T ÷ | 20 ÷ | |

Input the size of inner diameter and diameter of pad.

Then select other pin number of component.

| Pad Stac <u>k</u> Type | Pin: Plated: |
|--|-------------------|
| De <u>c</u> al Name: | 1(P) K |
| | |
| Apply Pad Stack Changes | × |
| Apply Pad Stack Changes | components |
| Apply Pad Stack Changes Do you want to apply the changes to all with decal type DIP14 | components |
| Apply Pad Stack Changes Do you want to apply the changes to all with decal type DIP14 or just the selected componen | components ts? |
| Apply Pad Stack Changes Do you want to apply the changes to all with decal type DIP14 or just the selected componen Keep Attributes | components |
| Apply Pad Stack Changes Do you want to apply the changes to all with decal type DIP14 or just the selected componen Keep Attributes | components ts? |

Click OK. Then click All if change all component's pads.

Controlling Selections

1. Parts Placement and Moving Components :

Click the **Design toolbox** icon.



S

Click the **Move Component mode** icon.

Select the component and click the left mouse button to move the component.

2. Rotating Components :

Click **Rotate** icon to rotate 90.

Click **Spin** icon to rotate free angle.



Select component and Click the Right mouse or Press Ctrl-F to flip Side.



3. Enable Automatic Component Nudging :

PowerPCB's placement features let you automatically shove or nudge adjacent components whenever components are placed too close to each other or if they overlap.

To enable automatic component nudging:

Select Setup/Preferences. The Preferences dialog box appears.

| <u>S</u> etup | Tools | <u>W</u> indow | Help |
|----------------|------------------|----------------|-------|
| Laye | r Definiti | on | |
| Pad | <u>S</u> tacks | | |
| Drill <u>F</u> | Pairs | | |
| Jump | ers | | |
| Desig | gn <u>R</u> ules | s | |
| Prefe | rences. | Ctrl+ | Alt+G |
| FCO | Profe | | 7 |

Select the **Design** tab.

Select **Automatic** from the **Nudge** area of the dialog box to enable automatic nudging. Choose OK to apply the changes and close the Preferences dialog box.

| Preferenc | es | | | | |
|-----------|--|--------------------------|------------|--------------------------|--------|
| Teardrops | Drafting | Grids | Split/Mixe | ed Plane | Fa |
| Global | Design | Routing | Thermals | Auto Dir | mensio |
| | ve Preference Move By Ori Move By Cu | igin Irsor Location | | e Automatic Prompt | ß |
| Ċ | Move By Lu Move By Mi | irsor Location dpoint | | c | C Off |

Adding Route

1. Using the Manual Route Editor :





Select the Pin.

Before Selection:





After Selection:

After select the correct pin, you can draw a line and path to connect other pin.

During connection:



After connection:



2. Using the Dynamic Route Editor (DRE) :

The dynamic route editor (DRE) is an interactive autorouter that follows the direction of your cursor as you move it, seeking optimal paths and installing corners as the route progresses. [Note: It is not reliable! Save file before execute Auto-Route]

To enable Dynamic route:

In order to enable the On-line Design Rule Checking (DRC), Select **Setup/Preference**, Select **Prevent Error** in **On-Line DRC** on Design Tab. Then click OK.



After setting the DRC,

Click **Design** icon and then click **Auto Route** icon to route the connection.



Select all components in box, then one connection will auto route and change colour.

Or Double-Click the pin to auto route connection.



Select all connection again, then another connection will auto route and change colour.



Creating Poured Copper

1. Creating the Copper Pour Outline :

The pour outline defines the boundaries of the copper pour area. Define the Pour Outline :

Select the **Drafting toolbox** icon from the toolbar.

Click the **Copper Pour** icon from the Drafting toolbox.

Draw the copper pour area.

2. Flooding the Pour Outline :

Click the **Flood** icon.



Select the flooding copper pour outline area. Then Choose Yes.

After flooding the copper pour outline, you can see the area of poured copper.

Before Flood:



After Flood:



Setting the Units for the Dimensions

1. Setting the Units :

Select Setup/Preference,

Choose the Global tab and set design units box.

2. Assigning General Settings, Alignment and Arrows and Text Properties of Auto Dimension :

Select Setup/Preference,

Choose the Auto Dimensioning label tab.

| Preferences | Design Routing Thermals Auto Dimensioning |
|---|---|
| Teardrops Drafting Grids Split/Mixed Plane Fanout Global Design Routing Thermals Auto Dimensioning General Settings | Alignment and Arrow |
| | Arrows Arrow Length: 0.1 → ↓ ↓ ↓ Arrow Size: 0.05 Text Gap: 0.03 ⊥ail Length: 0.05 Line Width: 0.01 |
| Design Routing Thermals Auto Dimensioning Text Image: Constraint of the stress of the s | |

3. Adding the Dimension :

Auto-orient mode automatically establishes the orientation of newly added dimensions.



Click Autodim icon to add dimension

Click the Auto icon or other dimension icon.

After adding dimension:



Printing the PCB

Use the CAM command on the File menu to produce laser printouts.

Select File/CAM, The Define CAM Documents dialog box appears.

| Click Add | icon |
|-----------|------|
|-----------|------|

| Exhibition Lawson | |
|-------------------|--------------------|
| | Fabrication Layer: |

Click Layers icon to define which layers and items should appear on printouts.

| Document <u>N</u> ame: | | | OK | |
|--|--------------|-----------------|--------------|-------|
| I Document <u>T</u> ype: Output <u>F</u> ile: | | | Cance | 1 |
| Custom 🗾 gen00.pho | | | <u>H</u> elp | |
| Fabrication Layer: | | | Dur | |
| <unassigned></unassigned> | | - | Hun | |
| Summary: | | | Set Layer | rs |
| 1 | | | | |
| Customize Document | - Output Dev | ice | | |
| Customize Document | Output Dev | ice | | |
| Customize Document | Output Dev | ice A Pen | Photo | Drill |

Select the Layer in Layer Selections, Click Add icon.

Check item in Items on Primary. E.g. Pads, Lines

| Available: Bottom Layer_3 Layer_4 Layer_5 Layer_6 Layer_7 | Add >> | pp | OK Cancel <u>P</u> review |
|---|------------------|-------------|---------------------------------|
| Other | Items on Primary | | <u>H</u> elp |
| Board Outline | ☐ Pads | 🗖 Ref. Des. | |
| Connections | Traces | Part Type | |
| Component outlines | Lines | Text | |
| Top Mounted | 🗖 Vias 🧏 | Attributes | |
| Bottom Mounted | Copper | 🗖 Outlines | |
| | | Test Points | |
| Selected Color | | | |

Click **Preview** icon to preview the output.

Click **Board** icon to zoom the output of PCB board. Then **close**.



Click Print icon in Output Device box, and Input the Document Name.



Click **Run** to print the output to Printer.

BlazeRouter

In PowerPCB, Select Tools/BlazeRouter

| <u>T</u> ools | <u>₩</u> indow | <u>H</u> elp | | | | | | | |
|---|---|--------------|--|--|--|--|--|--|--|
| <u>D</u> eca | al Editor | | | | | | | | |
| Auto Disp Leng Clys Auto | Automatic <u>C</u> luster Placement Disperse Components Length Minimization Ctrl+M Cluster Manager Auto Nudge | | | | | | | | |
| Vie <u>w</u> SPE Boar C <u>A</u> M | Draw CCTRA dSim 350 | | | | | | | | |
| Blaz | eRouter | | | | | | | | |

Select Launch BlazeRouter Only in Action box Then press Proceed

| BlazeRouter Link | | | _ 🗆 X |
|--|--------------------|---------------------------|-------------------|
| Action © Run in <u>B</u> ackground | Preferences – | C <u>C</u> learance Rules | Proceed Cancel |
| Launch BlazeRouter Only Save for Further Use | C Grjd C Design | O DFI | Help |
| Routing Strategy | | Setup | |
| U:\pcb\Untitled_blz.pcb | | | Save <u>A</u> s |

Click Ignore in BlazeRouter Window

(Hint: Wait about 1 min, it hide by other program, so minimum all program window first)





Select Tools/Options in BlazeRouter Program



Change File Location in General Tab. (otherwise cannot open file for auto route)

| Options | | | | | | | | | | |
|--|--|--|--|--|--|--|--|--|--|--|
| General Display Routing | Strategy Fanout DFT Audit | | | | | | | | | |
| Cursor Pick radius: 10 Cursor style: Vertical Large Cross | Graphics Backup Minimum display width: 12 Interval (minutes): 10 Image: Second state sta | | | | | | | | | |
| File locations: | Design <u>U</u> nits: Mils ▼ | | | | | | | | | |
| File Types: Locations: | | | | | | | | | | |
| File Types: Locations: Designs \\ele-pine\netapps\pwrpcb\BlazeRouter\ Reports U:\pcb Log U:\pcb Strategy \\ele-pine\netapps\pwrpcb\BlazeRouter Macro U:\pcb Color Scheme U:\pcb Backup \\ele-pine\netapps\pwrpcb\BlazeRouter\ VB Scripts \\ele-pine\netapps\pwrpcb\ole\VBscripts | | | | | | | | | | |
| | OK Cancel Apply Help | | | | | | | | | |

Change File Types of Designs and Backup Locations

| File locations: | |
|--|--|
| File Types: | Locations: |
| Designs Reports Log Strategy Macro Color Scheme Backup VB Scripts | U:\pcb U:\pcb \\ele-pine\netapps\pwrpcb\BlazeRouter U:\pcb U:\pcb U:\pcb \\ele-pine\netapps\pwrpcb\ole\VBscripts |

Now, can open file for auto-route

Select Length Minimization to short connection



Click Properties icon to change design properties



In Clearance Tab, click All icon to change the space between all thing.



In Routing Tab, change the value in Trace width



In Layers Tab, check the Route box Top or Bottom only when only one layer design is used.

| D | esign | Propertie | s | | | | | | _ 🗆 X |
|-----|------------|-------------------|-----------------|----------------|-----------------------|------------|----------------|--------------------|--------------|
| | Cle San | earance ne net | Rou Route Op | ting Itions | Via Bia: Test Poir | ing Its | Laye Layers | er Biasing Grid | ОК |
| | | | | | | - Pre | eview —— | | Apply |
| II. | | Layer | Route | Dir | ection | | | | Cancel |
| | 1 | Top | | Horizor | Tai | | | | |
| U | 2 | Bottom | ☑ | Vertica | l | | \rightarrow | • | <u>H</u> elp |
| J. | | | | | | | | | |
| | | | | | Þ | | \uparrow | | |

After properties setup, click Routing icon,

| 🖉 ехе | cise9.pcb* - BlazeRouter | |
|-----------------|--------------------------------------|------------------------|
|]] <u>F</u> ile | idit ⊻iew <u>T</u> ools <u>H</u> elp | |
|] @ | 🖬 🥥 (H) Top 💽 🗹 🛠 🖈 | ※ 2 - 2 - U Q マロ 国际米留校 |
| 4 | | Routing |

And Click Start Autorouting icon to start auto route.



Click Select icon to select component

| B | - 8 | | Í | (H) | Тор |) | | | • |] 🖻 | Ś | C | × | *** | |
|---|-----|---|---|-----|-------|-----|----|-----|---|-------------|---|---|---|-----|--|
| 4 | ii> | Ш | Ч | × | | 141 | ## | 147 | | • + + | * | * | # | ţ÷ţ | |
| | | | | Se | elect | | | | | | | | | | |

Select Component with cursor and click right-click mouse, Select Unroute to delete the route.





After Change the design properties and autorouting again.



Exercise 1 Disperse Components







27 PADS-PowerPCB 4 Tutorial (Electronics Industrial Training Programme 2002, EE, CUHK)

Exercise 3 Add new Decal



Exercise 4 Add new Decal with setting grids



Exercise 5 Selection Filter



28

Exercise 6 Change the pads



Exercise 7 Rotating Components



Exercise 8 Manual Route



Exercise 9 Dynamic Route



Exercise 10 Flooding the Pour Outline

Exercise 11 Adding the Dimension

Exercise 12 Printing the PCB