

PADS Logic

- For PADS9.0.X

苏州泓泰软件科技有限公司

TEL:0512-82285118

FAX:0512-82285119

MAIL:tech@mosteceda.com.cn

课表

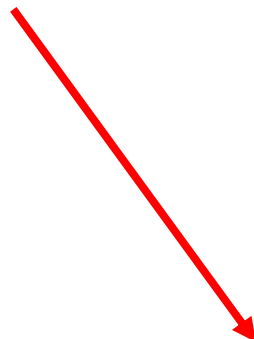
第一课	介绍和安装
第二课	使用介面与环境设置
第三课	编辑绘制
第四课	零件库
第五课	绘制线路图
第六课	PCB设置与规则
第七课	资料输出
第八课	连结器

第一课

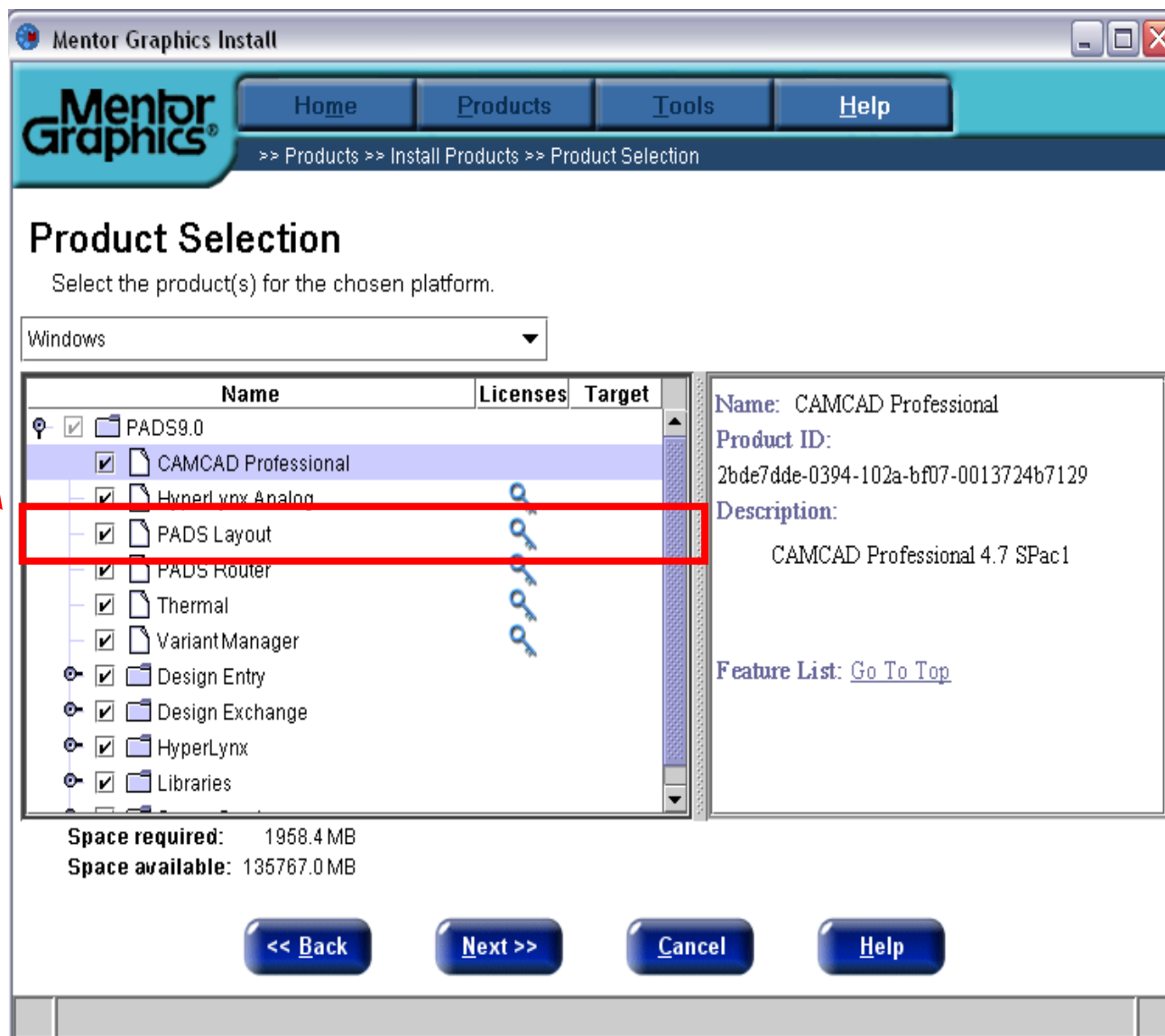
介绍和安装

Logic 安装

- PADS-Logic
- 勾选Layout项目



预设程序安装路径为
**C:\MentorGraphics\9.0.
X PADS**



软、硬件建议需求

建议操作系统

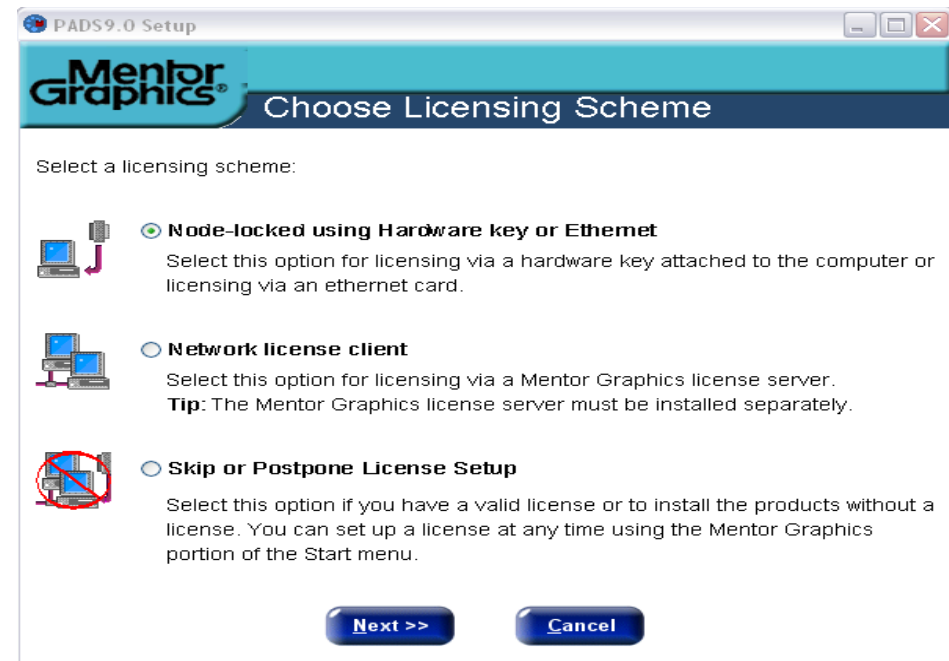
- Windows XP
- Windows VISTA(2007.2以上版本)

建议硬件配备

- CPU至少要Pentium III 500 MHz
- 存贮器至少要256MB (建议512MB以上)
- 三键或滚轮鼠标器 (建议)
- 1024 X 768, 256色显示
- 需要1.0 GB硬盘空间(建议1.5 GB硬盘空间)

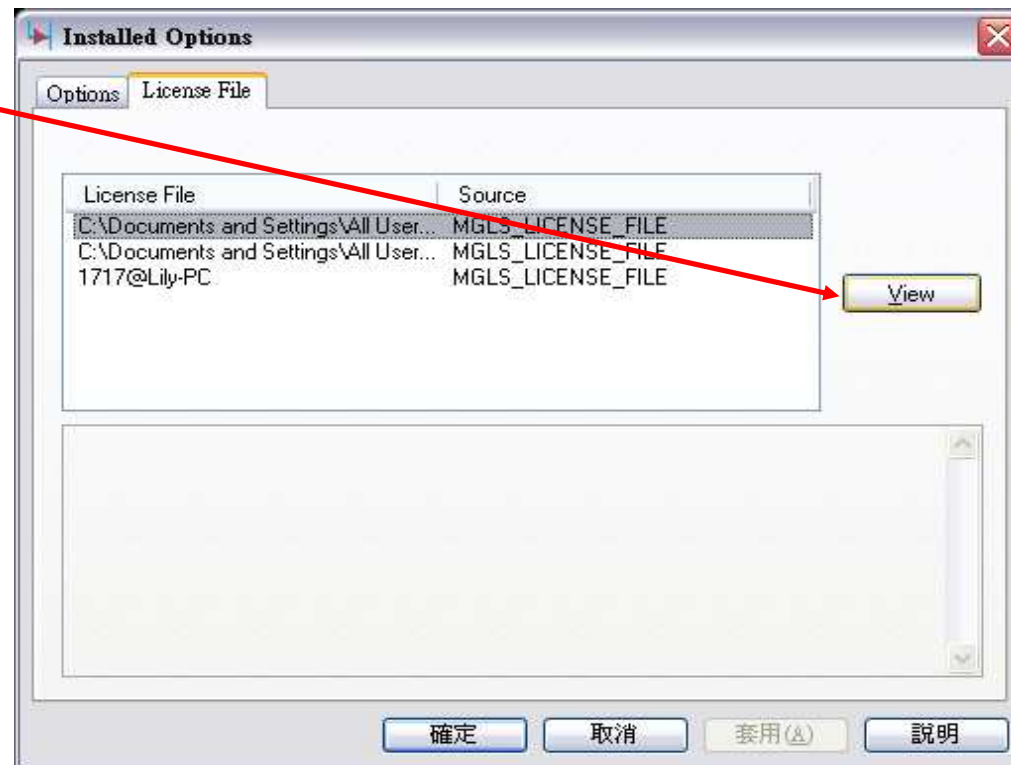
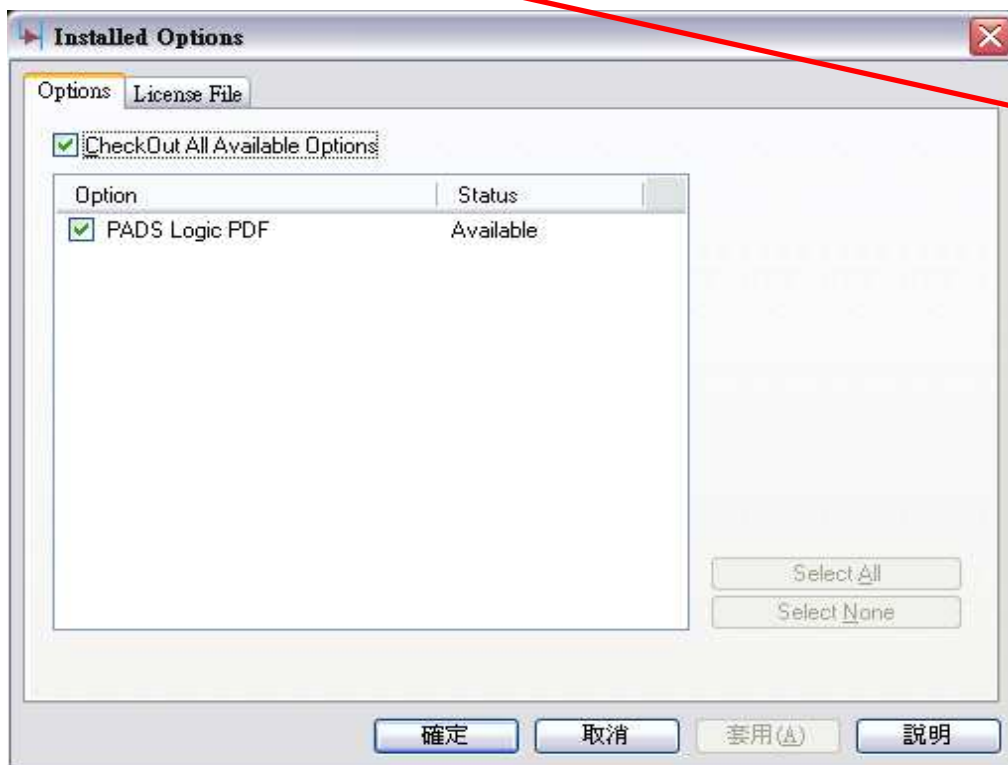
保全方式

- **Logic**需要单独的**License ID**，相关细节可参考**License file**中的内容
- **C:\Documents and Settings\All Users\Application Data\pads\9.0.X\License.dat**
- **Floating License Server** 是属于额外的安装选项
- 可从安装光盘中找到安装程序



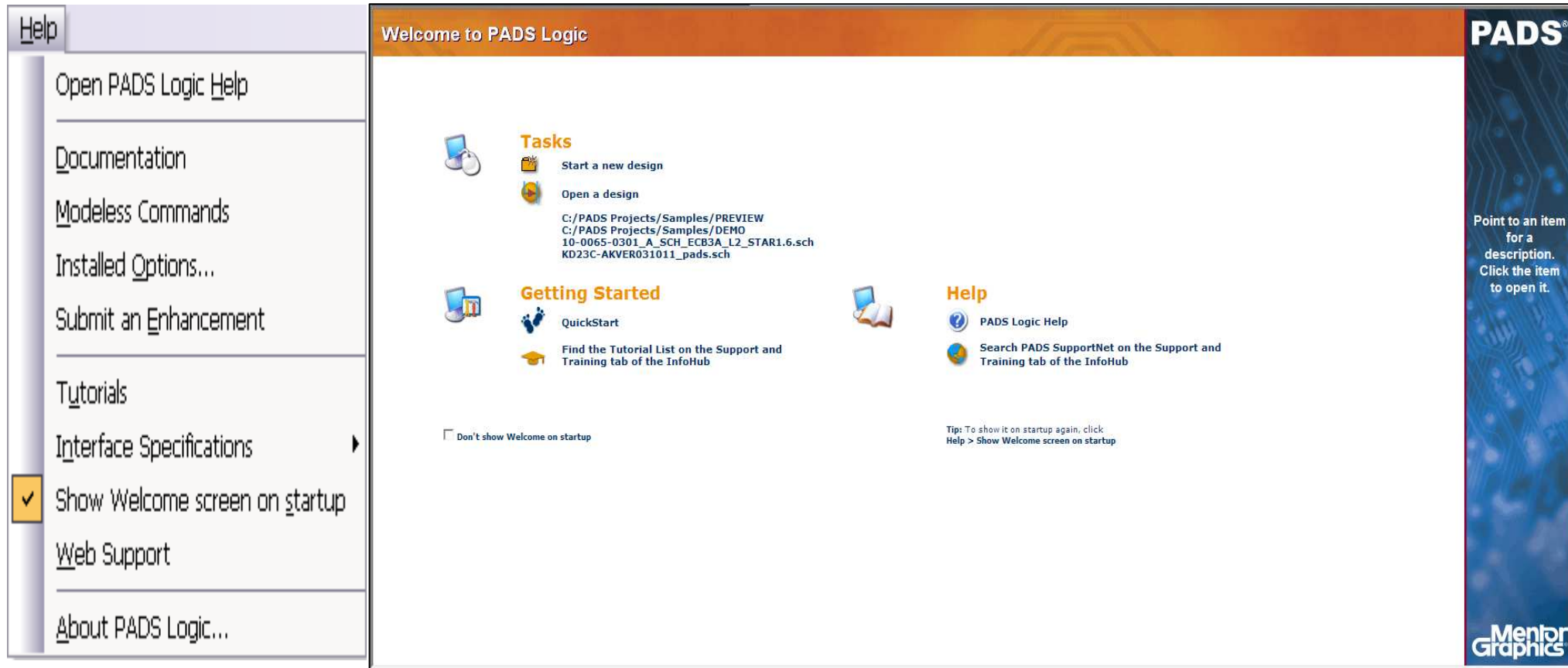
安装功能与 License file

- 功能安装选项：
 - Help > Installed Options...
- 查看 license file 内容：
 - 选择 “License File” 页面
 - 按 “View” 按钮



PADS 相关资源

- 可由Logic中的Help下拉菜单中选取



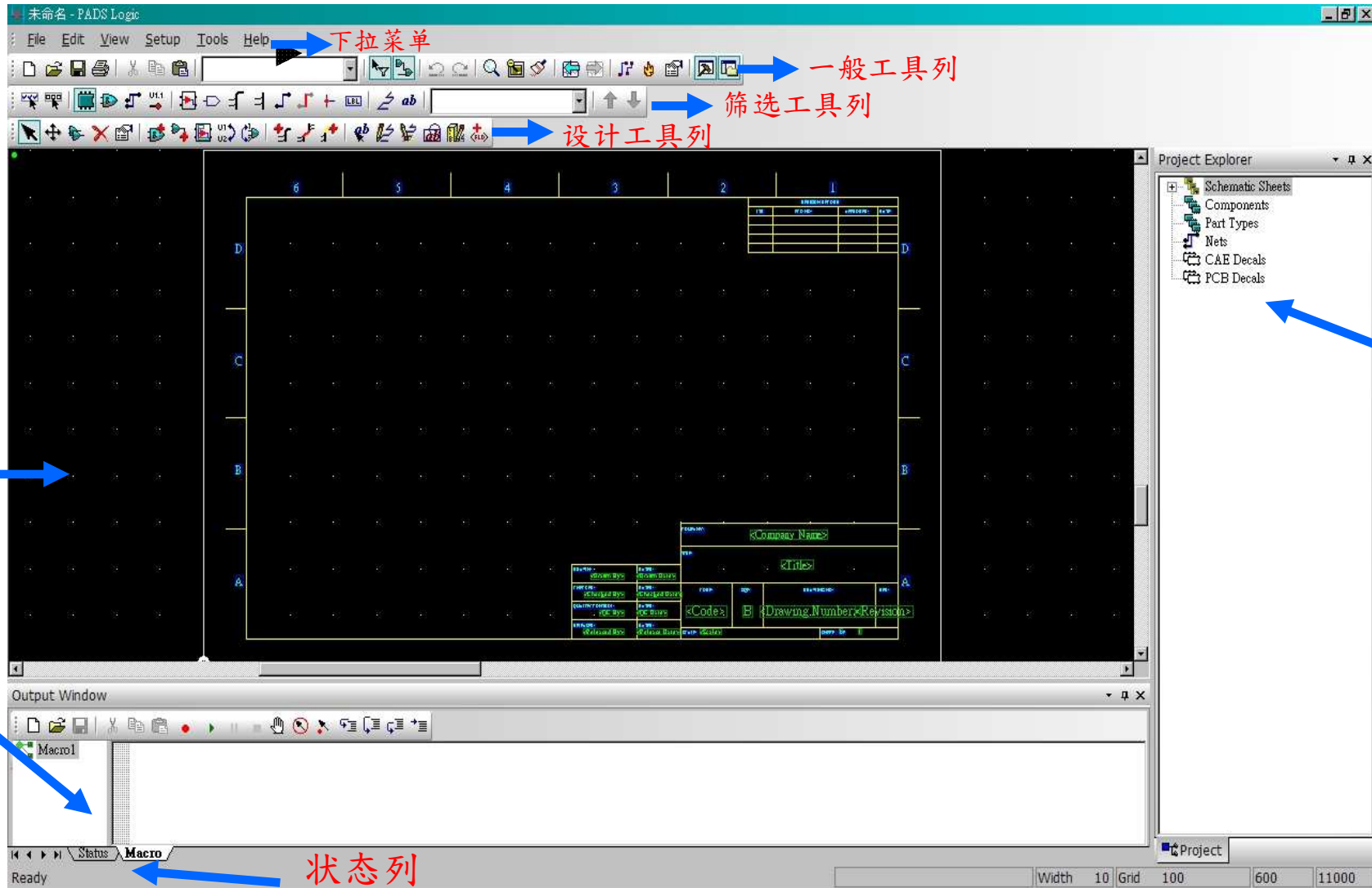
PADS Logic help 联机即时说明

Web Support 连结原厂联机说明 <http://supportnet.mentor.com/> (需于原厂网站上注册)

第二课

使用介面与环境设置

Logic 环境介绍



Logic 环境介绍

◆ Project Explorer 面板

树状结构式的窗口，提供对整张线路图物件资料的存取

◆ 输出窗口 共分成二个页面：

Status 显示目前编辑状态

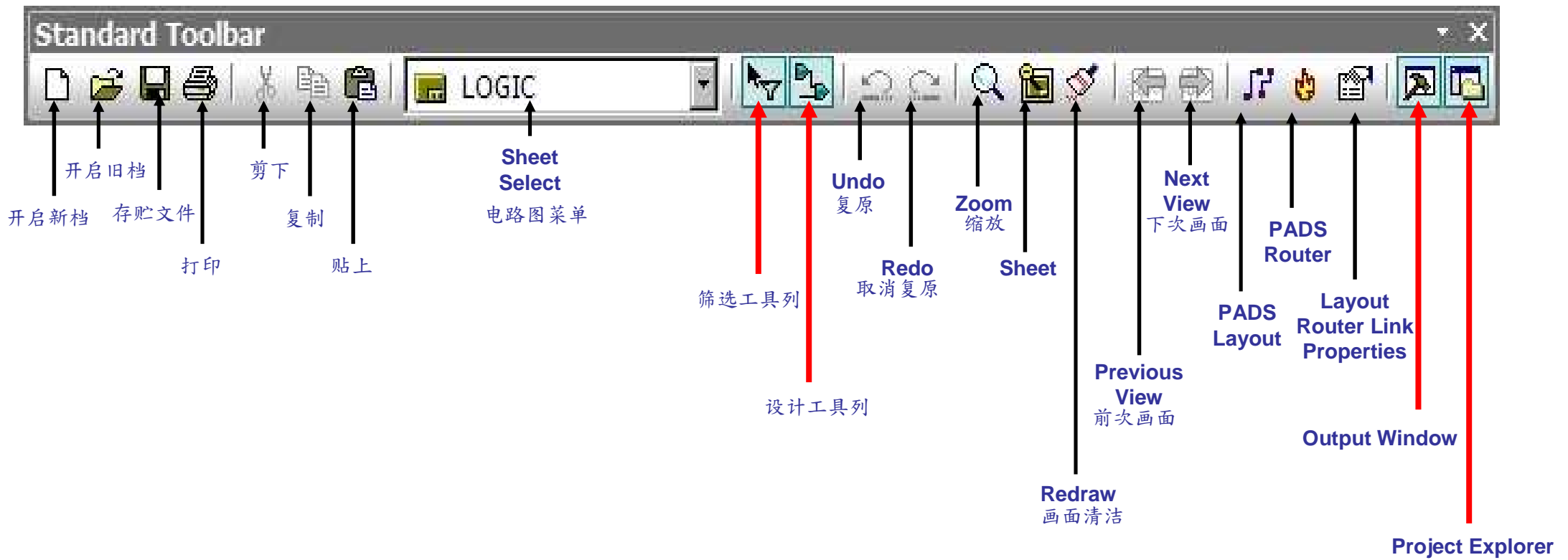
Macro 编辑，录制巨集

◆ 状态列

提供指令执行时相关信息

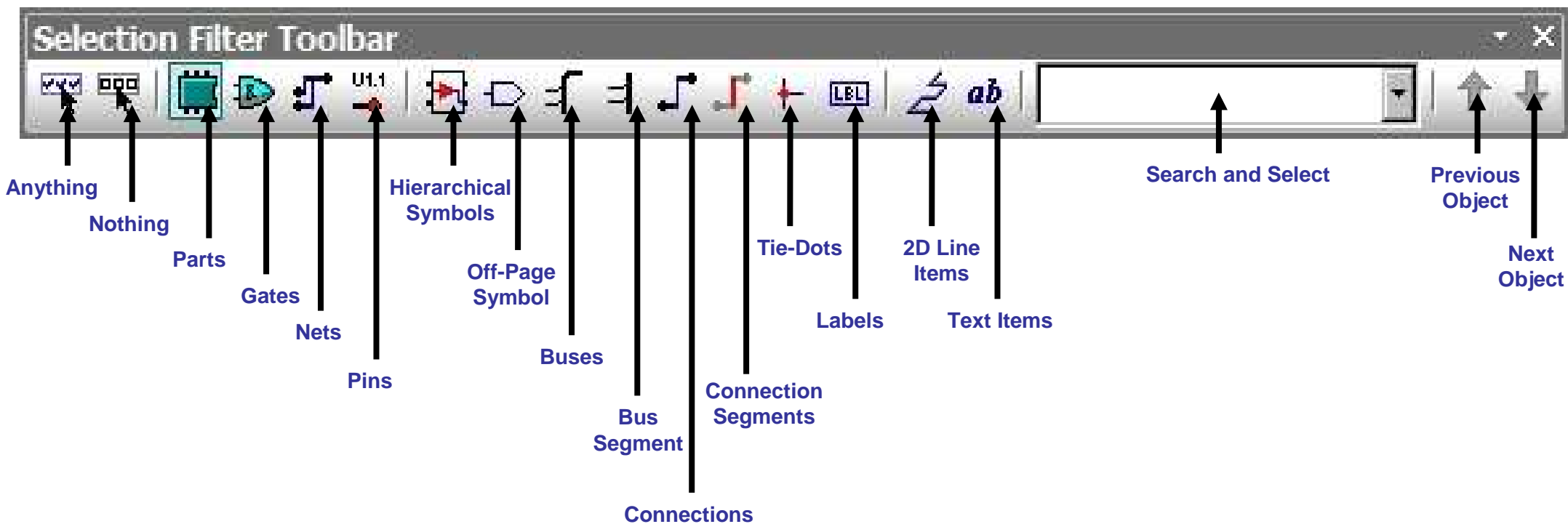
工具列

◆ 一般工具列



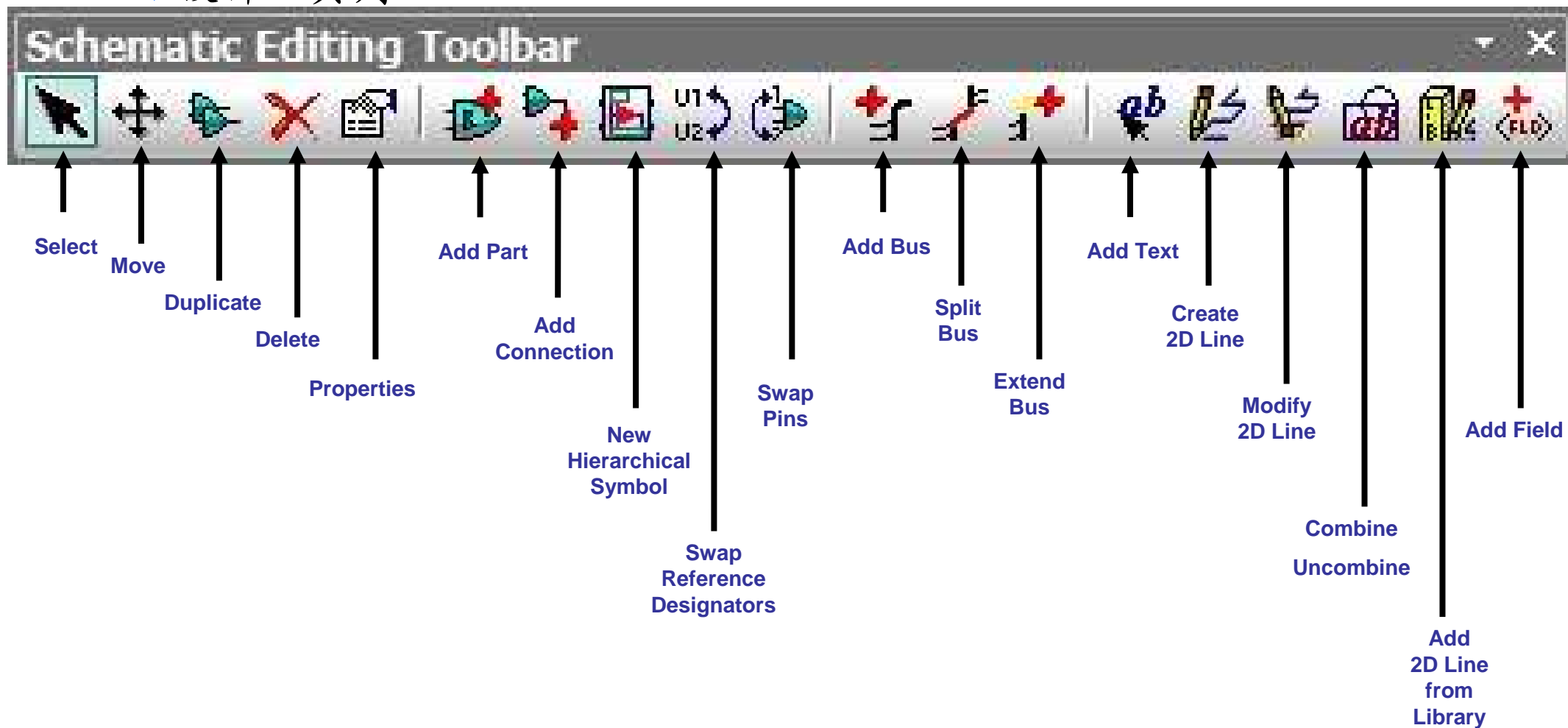
工具列

◆ 筛选工具列



工具列

◆ 设计工具列



鼠标器指令

鼠标器中键 (MMB)

- Click 偏移画面
- Hold and Drag 放大缩小
- Ctrl + Wheel 放大缩小
- Wheel 上下
- Shift + Wheel 左右

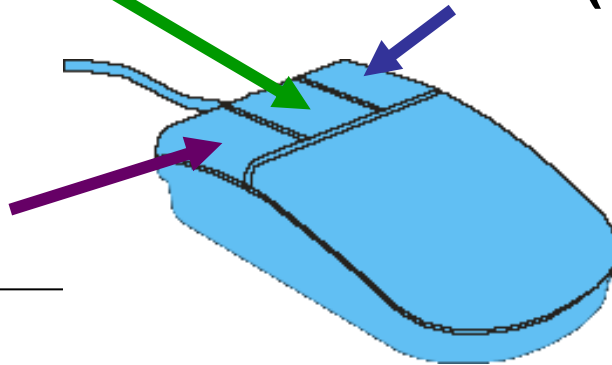
鼠标器右键 (RMB)

- Nothing Selected 右键菜单
- Object Selected 物件右键菜单

Middle Mouse
Button (MMB)

Right Mouse
Button (RMB)

Left Mouse
Button (LMB)

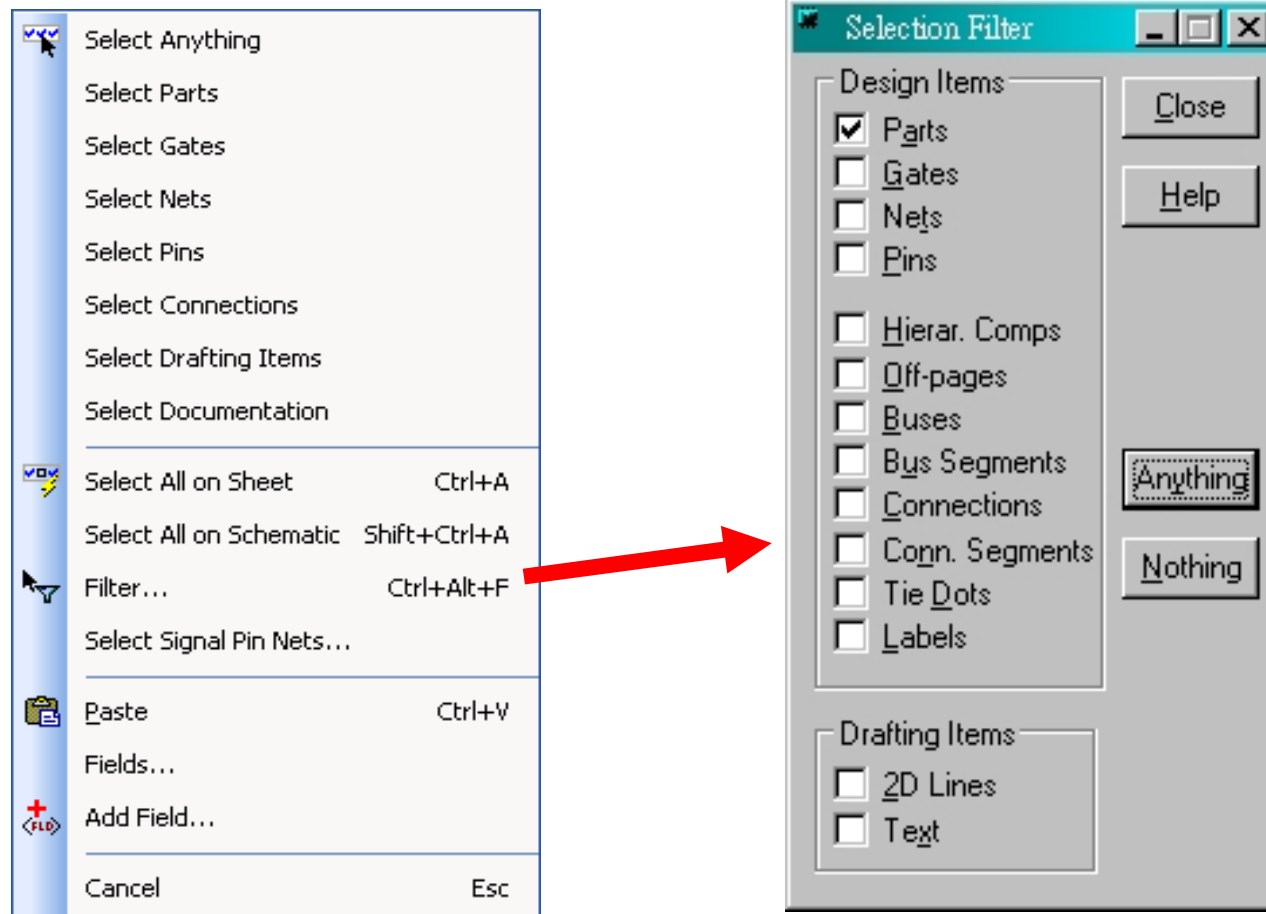


鼠标器左键 (LMB)

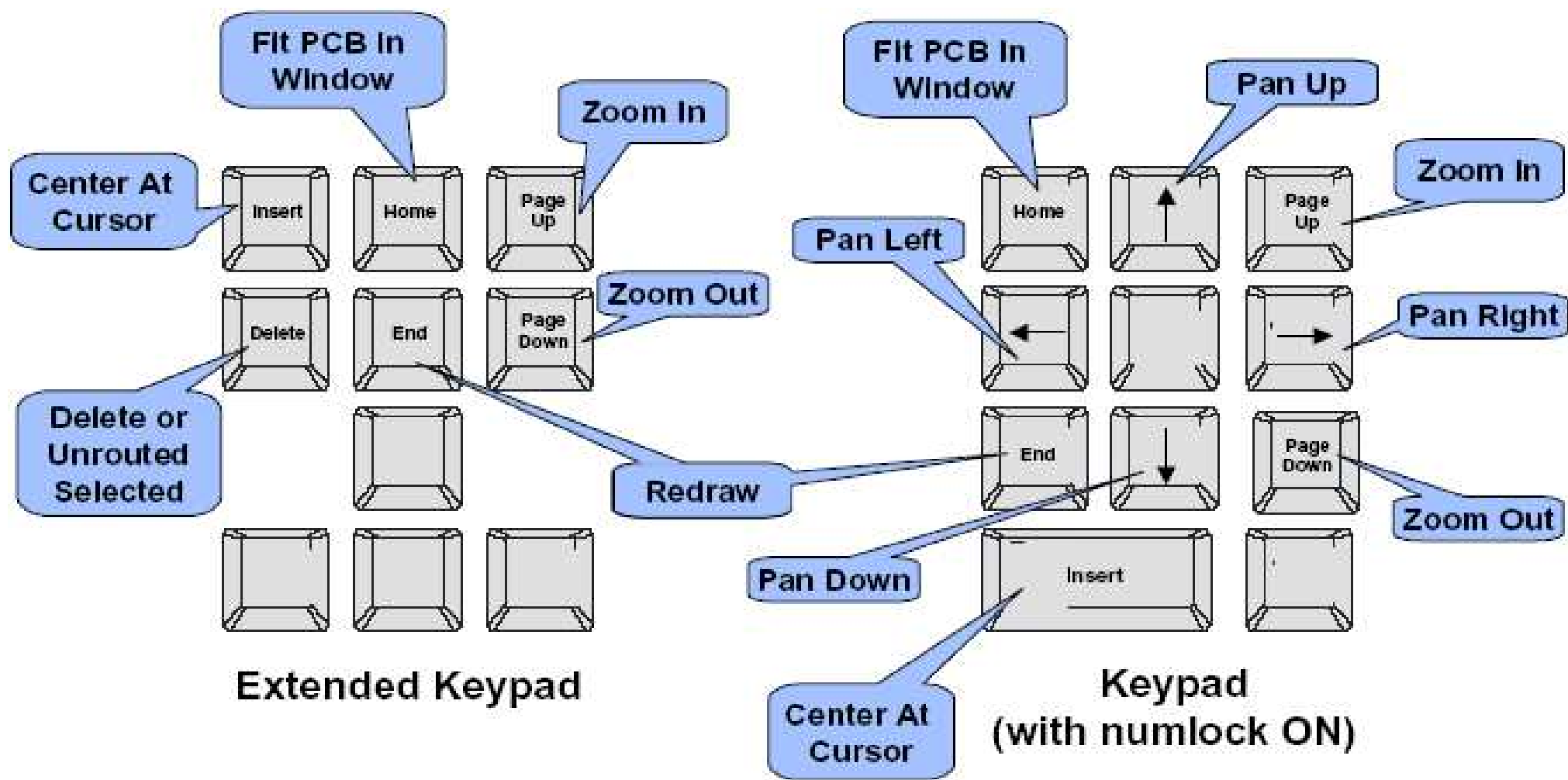
- Click 选取
- Ctrl + Click 加选与取消加选
- Double Click 物件Properties
- Depress LMB in space + Drag a box 框选

一般工具列

◆ 右键筛选工具列(对应筛选工具列)

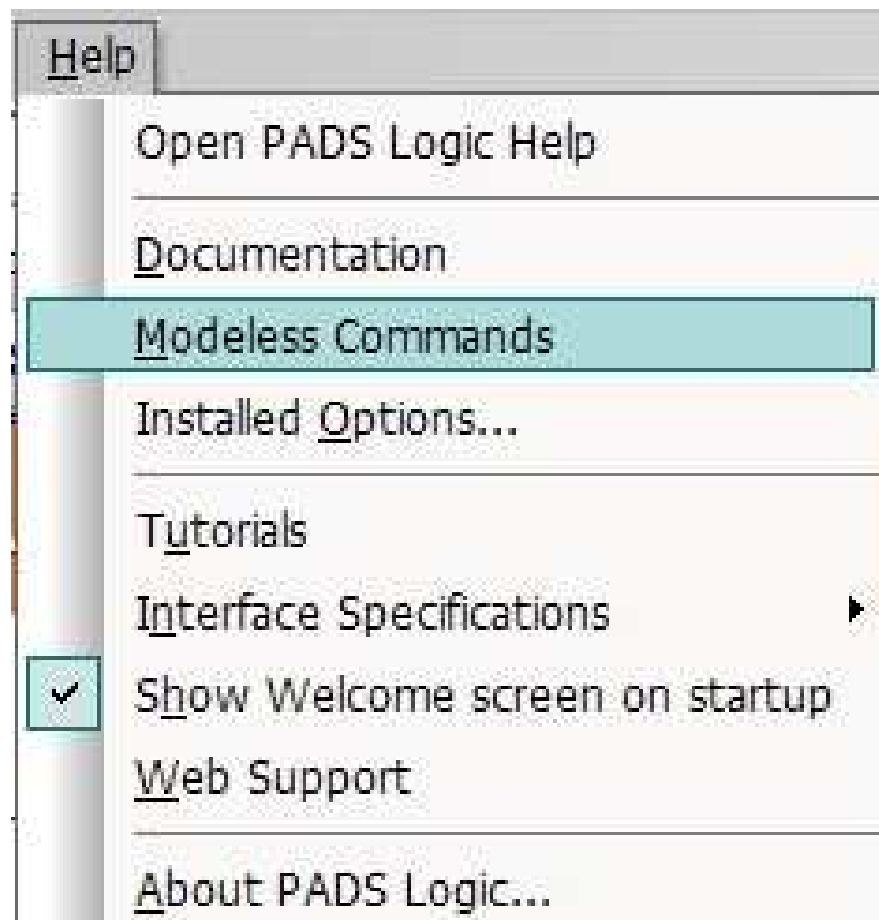


键盘指令



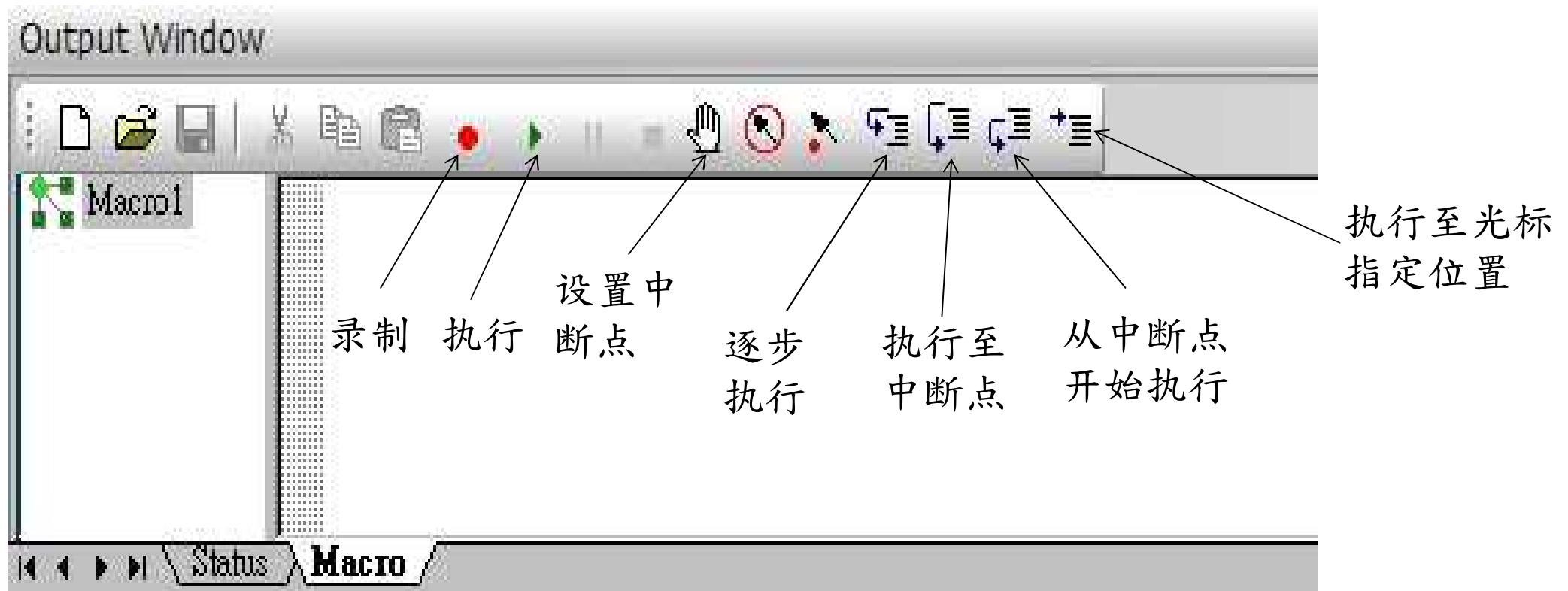
热键指令

- Help > Modeless Commands



巨集录制

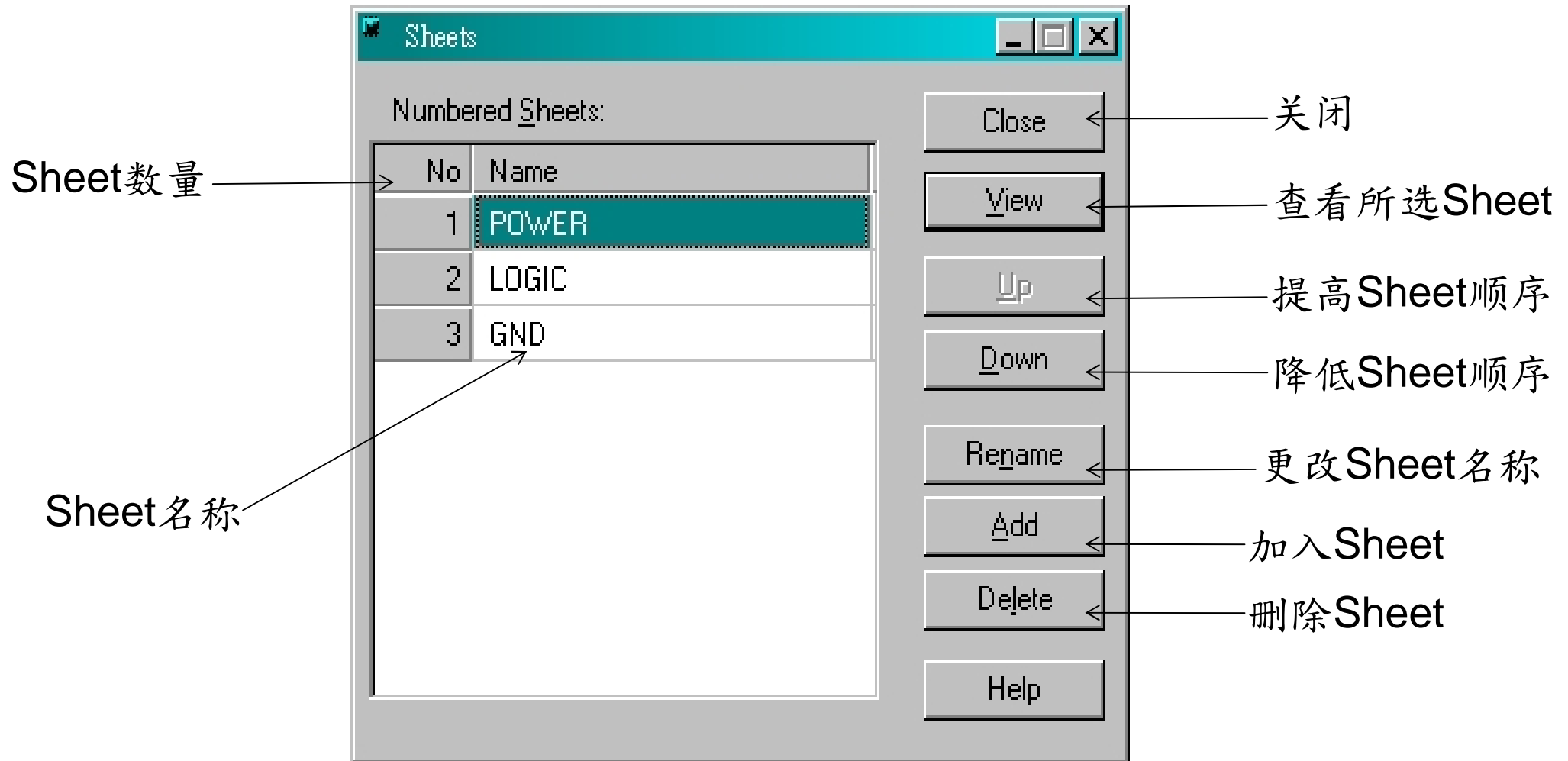
- Output Window > Macro



PS: 先存档后方可使用

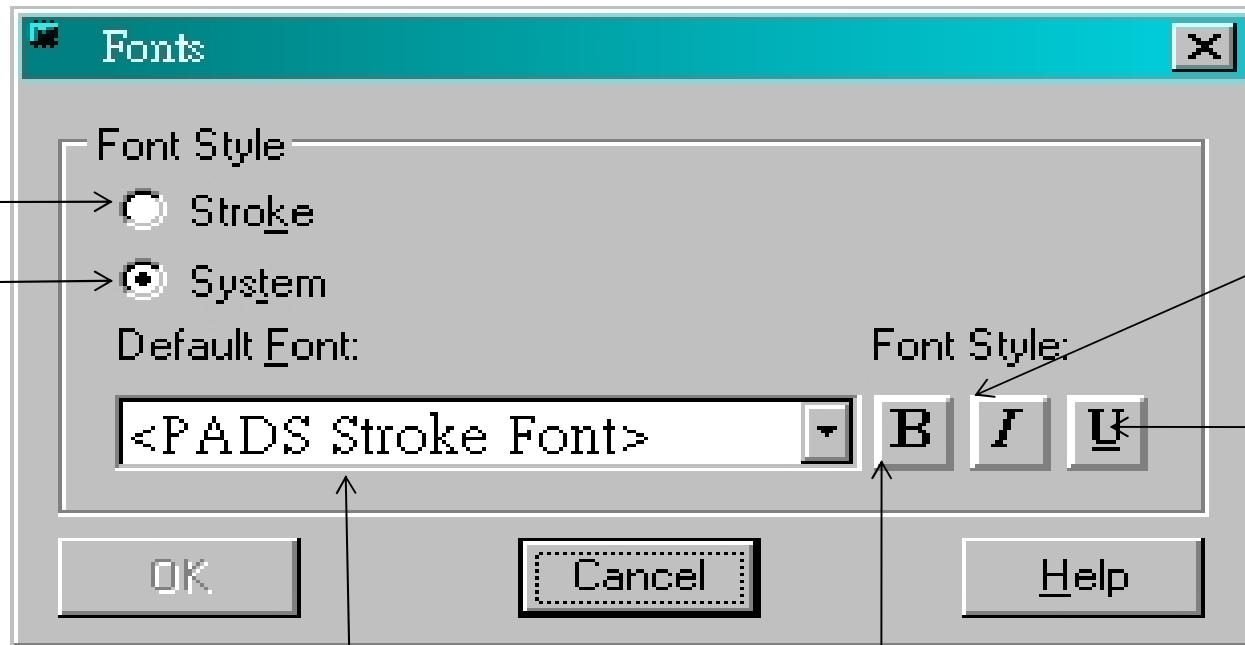
Sheets

- Setup > Sheets



Fonts

- Setup > Fonts



PADS预设字型

系统内字型

斜体

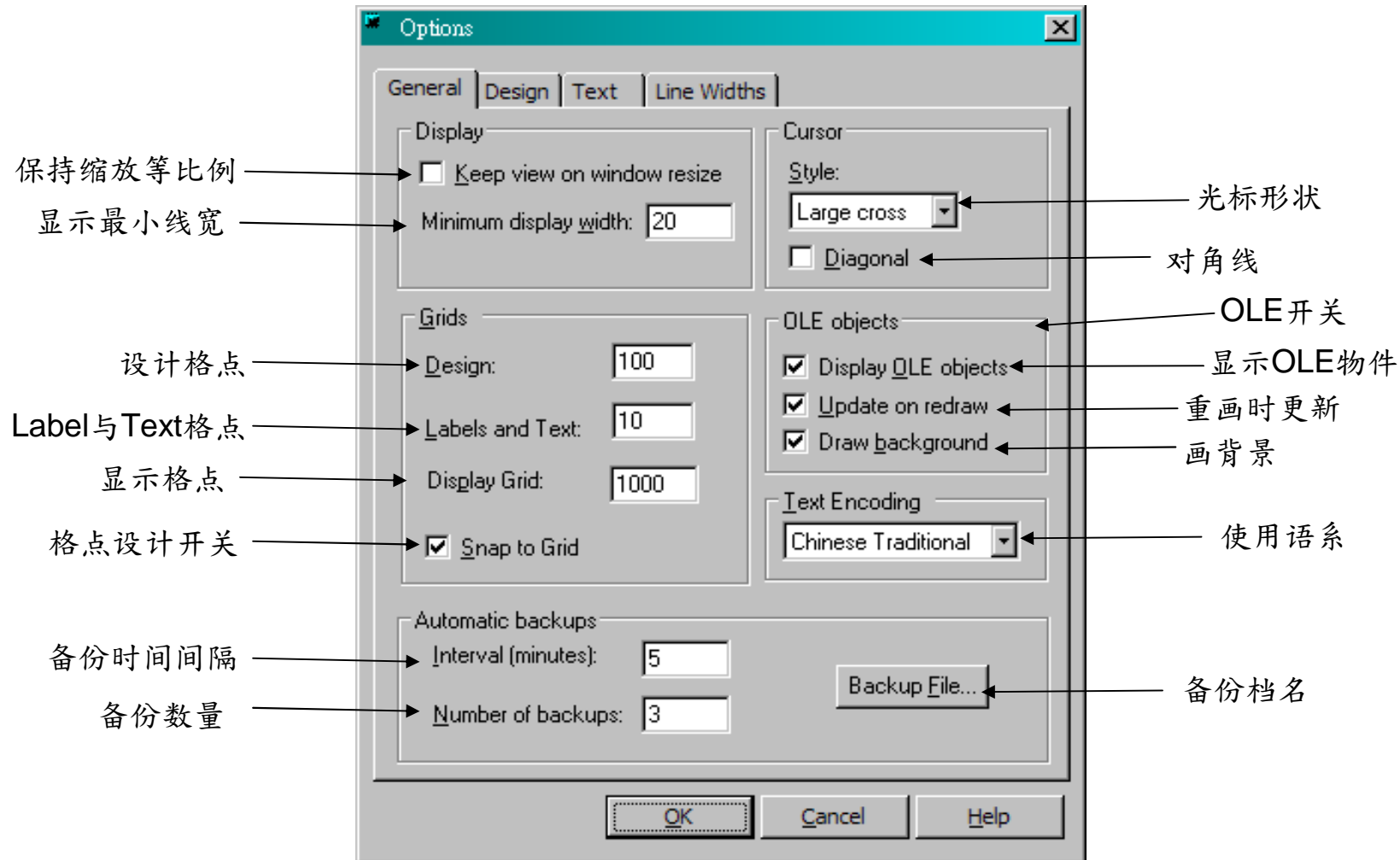
底线

字型列表

粗体

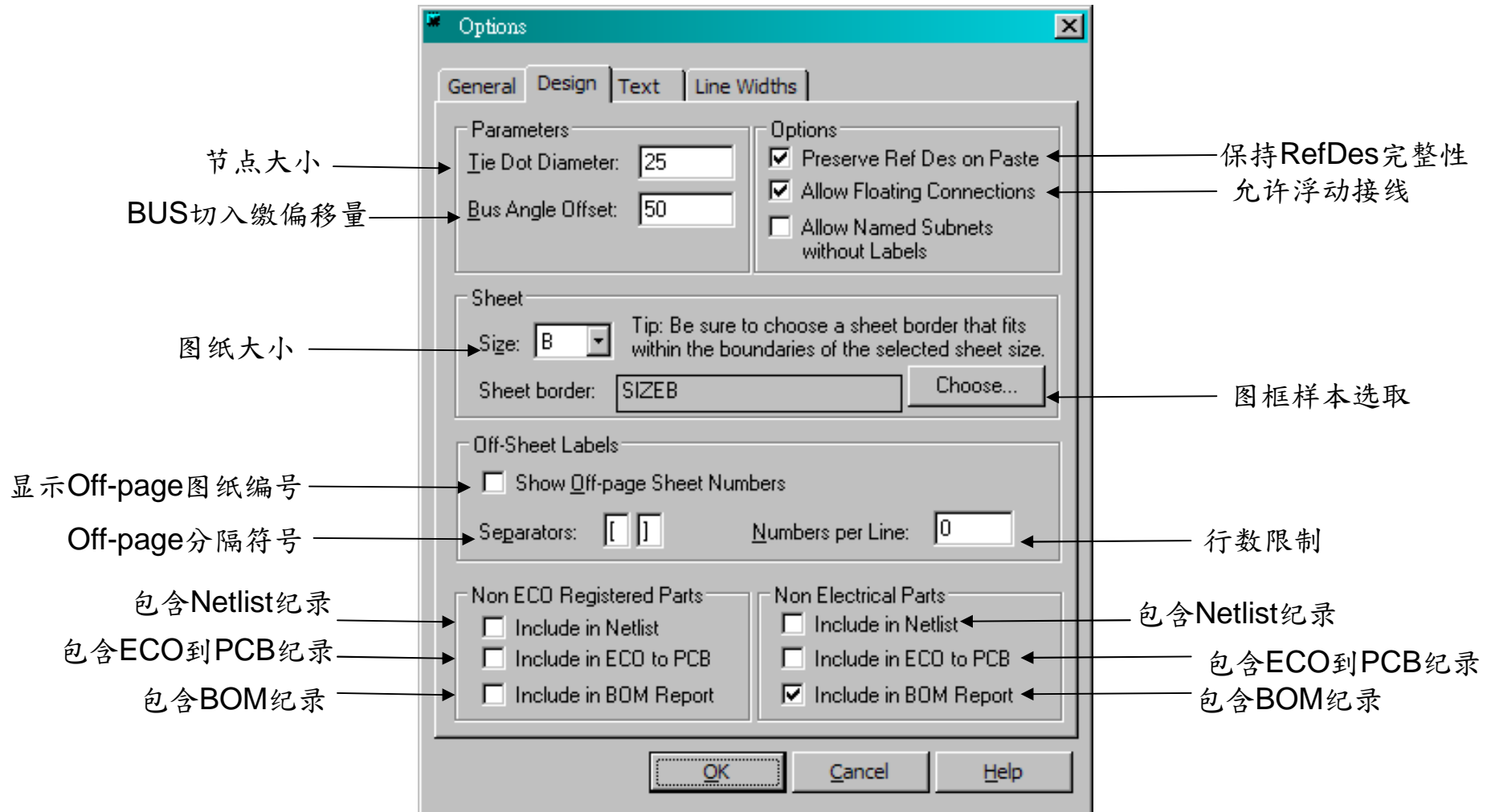
Options

◆ Tools > Options > General



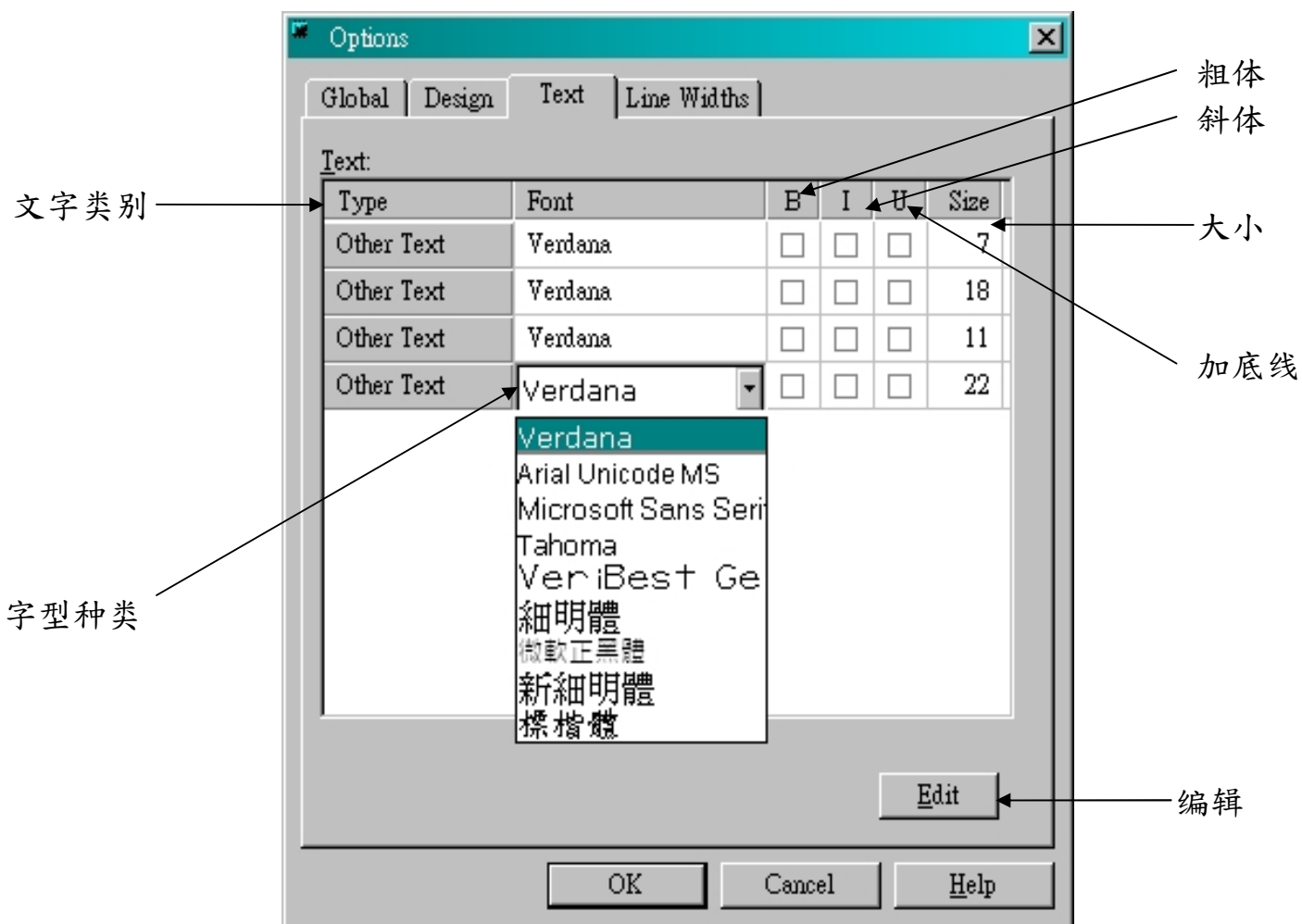
Options

◆ Tools > Options > Design



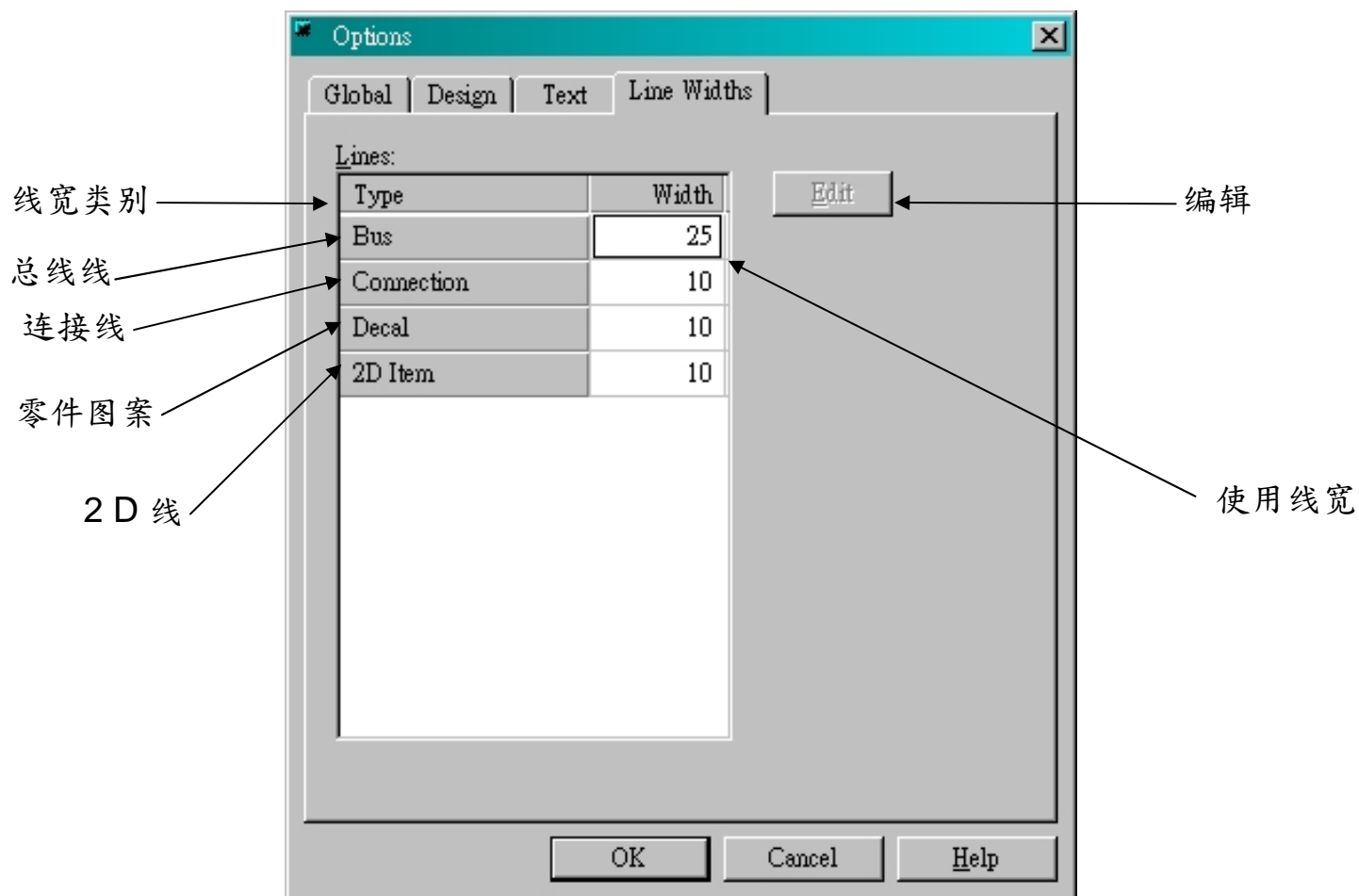
Options

◆ Tools > Options > Text



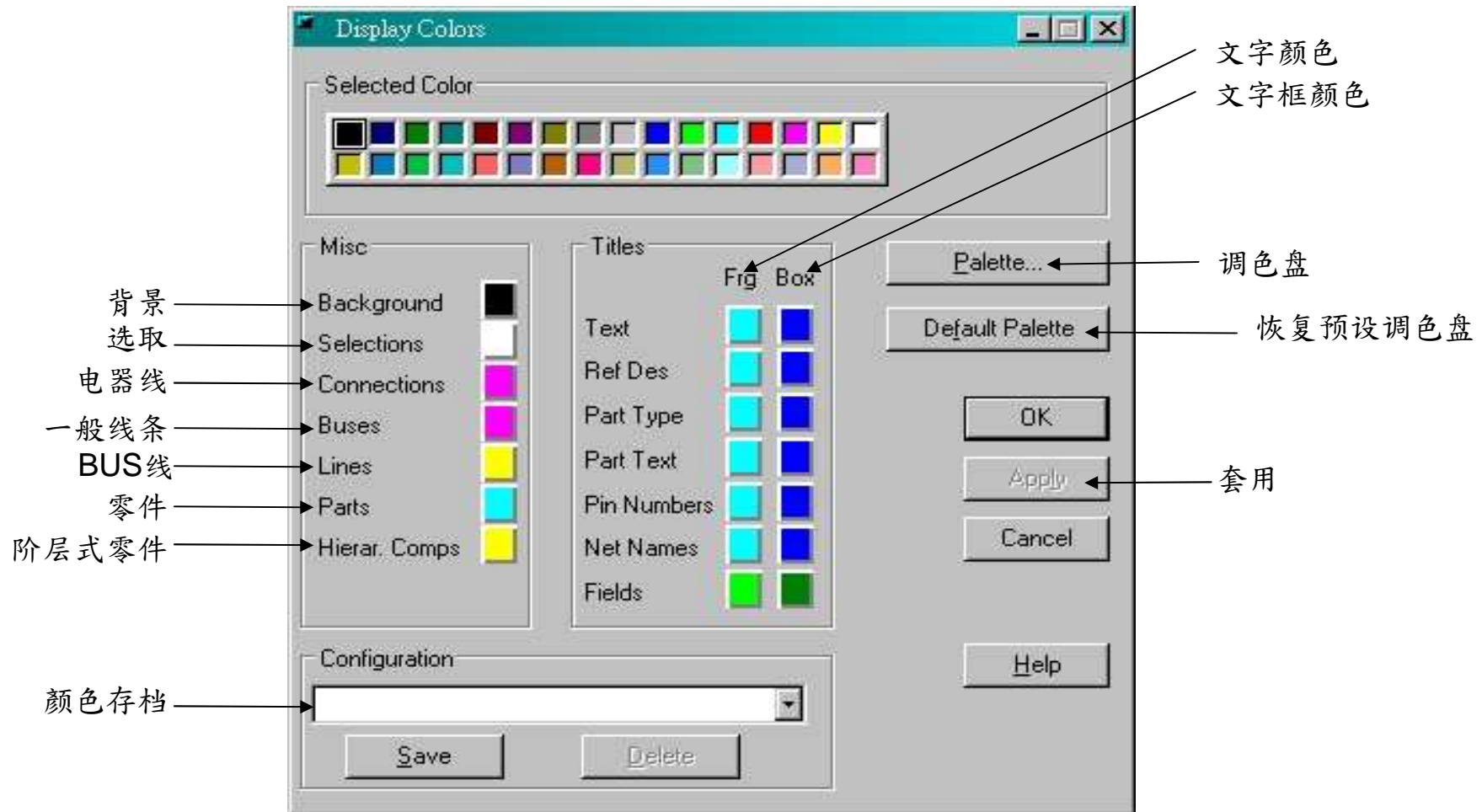
Options

◆ Tools > Options > Line Widths



Display Colors

- Setup > Display Colors



练习一

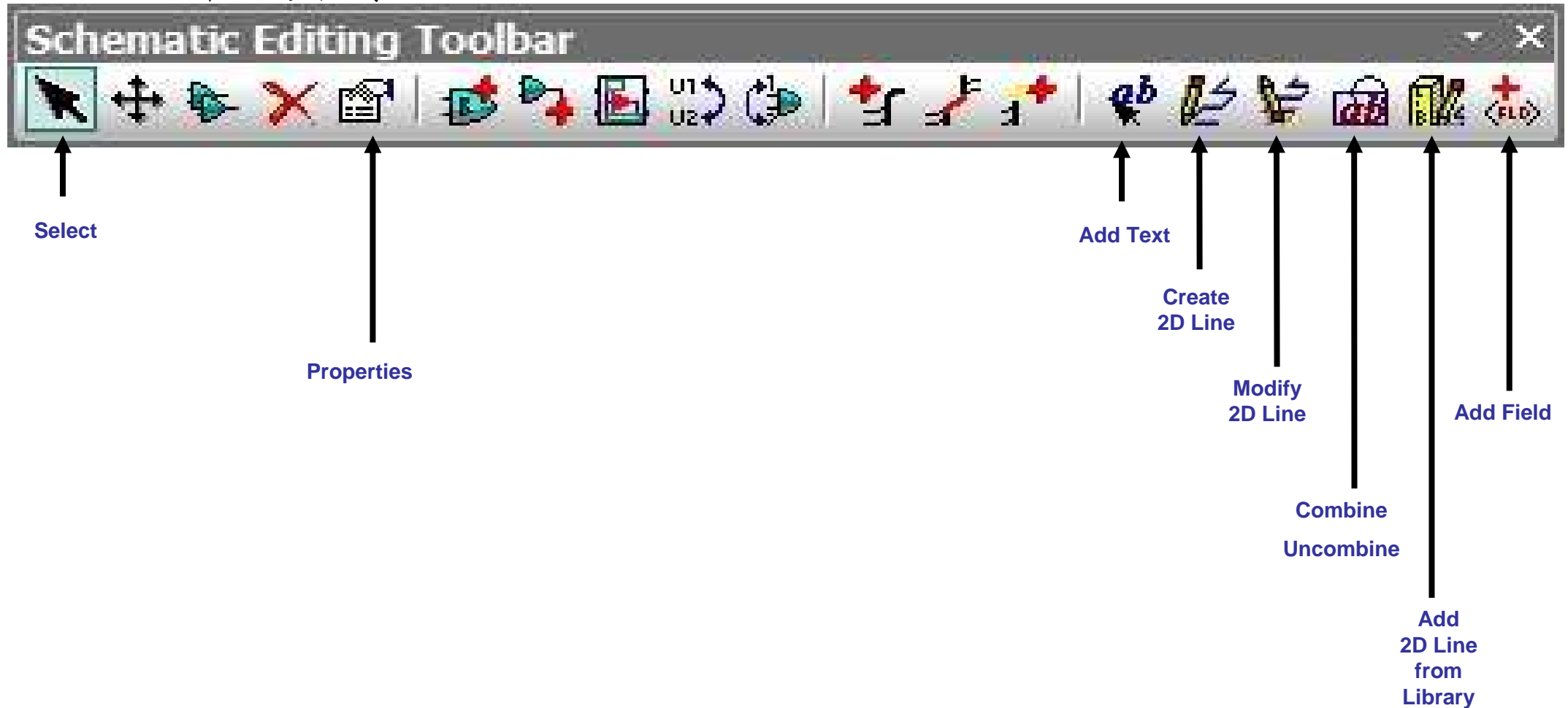
- Project Explorer 面板、Help 面板、输出窗口、状态列、Macro
- 一般工具列、筛选工具列、设计工具列
- 鼠标器指令
- 键盘指令
- Options
- Display Colors
- 增加 Sheets、设置 Fonts

第三课

编辑绘制

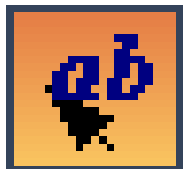
Design Toolbar

◆ 设计工具列



写文字

- Design Toolbar > Add Text



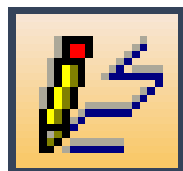
The 'Add Free Text' dialog box contains the following fields and controls:

- Text:** A text input field for the content of the text.
- X:** A text input field for the X-axis coordinate.
- Y:** A text input field for the Y-axis coordinate.
- Font:** A section containing:
 - Rotation:** A dropdown menu set to 0.
 - Line Width:** A text input field set to 10.
 - Size (pts):** A text input field set to 7.
 - Font Style:** Three buttons for Bold (B), Italic (I), and Underline (U).
 - Font:** A dropdown menu showing 'Arial Unicode MS'.
- Justification:** A section with two columns of radio buttons:
 - Horizontal:** Radio buttons for Right, Center, and Left (selected).
 - Vertical:** Radio buttons for Up, Center, and Down (selected).

Labels on the left side of the dialog point to: 文字内容 (Text), X坐标值 (X coordinate), 旋转角度 (Rotation), 文字大小 (Text size), and 选用字型 (Font selection). Labels on the right side point to: Y坐标值 (Y coordinate), 文字线宽 (Text line width), 文字形态 (Text style), and 垂直定位点 (Vertical alignment point).

画线

- Design Toolbar > Create 2D Line



完成线段	Complete	<dbl_click>	
增加转角	Add Corner	<click>	
删除转角	Del Corner	Back	
增加弧线	Add Arc		
	Width...	{W<nn>}	设置线宽
封闭多边形	Polygon	{HP}	
圆形	Circle	{HC}	
矩形	Rectangle	{HR}	
线段	Path	{HH}	
	Orthogonal	{AO}	90度角
	Diagonal	{AD}	45度角
	Any Angle	{AA}	任意角
	Cancel	Esc	

修改线段

- Design Toolbar > Modify 2D Line



圆弧线 → Pull Arc
 增加转角 → Split
 删除转角 → Del Segment Back

填满

Drafting Properties

Width: 10 Filled:

Style
 Solid: Dotted:

Rotation: 0 Mirror by X: Mirror by Y:

OK Cancel Help

线宽

实心线

旋转角度

虚线

水平翻转

垂直翻转

Width... {W<nn>} ← 设置线宽

Solid Style ← 实心线

Dotted Style ← 虚线

Filled

Orthogonal {AO} ← 90度角

Diagonal {AD} ← 45度角

Any Angle {AA} ← 任意角

Cancel Esc

设置线宽

实心线

虚线

90度角

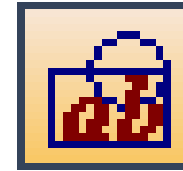
45度角

任意角

PS:填满效果仅适用于多边形与矩形

组合物件

Design Toolbar > Combine/Uncombine



- Combine方法

- 选取后 > 鼠标器右键 > Combine
- 按下 Combine/Uncombine 按钮 > 鼠标器右键 > Combine > 点选所需物件 > 鼠标器右键 > Complete

- Uncombine方法

- 按下 Combine/Uncombine 按钮 > 鼠标器右键 > Uncombine > 点选不需要的物件 > 鼠标器右键 > Complete

- Explode方法

- 选取后 > 鼠标器右键 > Explode
- 按下 Combine/Uncombine 按钮 > 鼠标器右键 > Explode > 点选所需物件 > 鼠标器右键 > Complete

存贮组合物件

- 存贮步骤：
 - 选取组合物件后
 - 鼠标器右键
 - Save to Library
 -



选择零件库

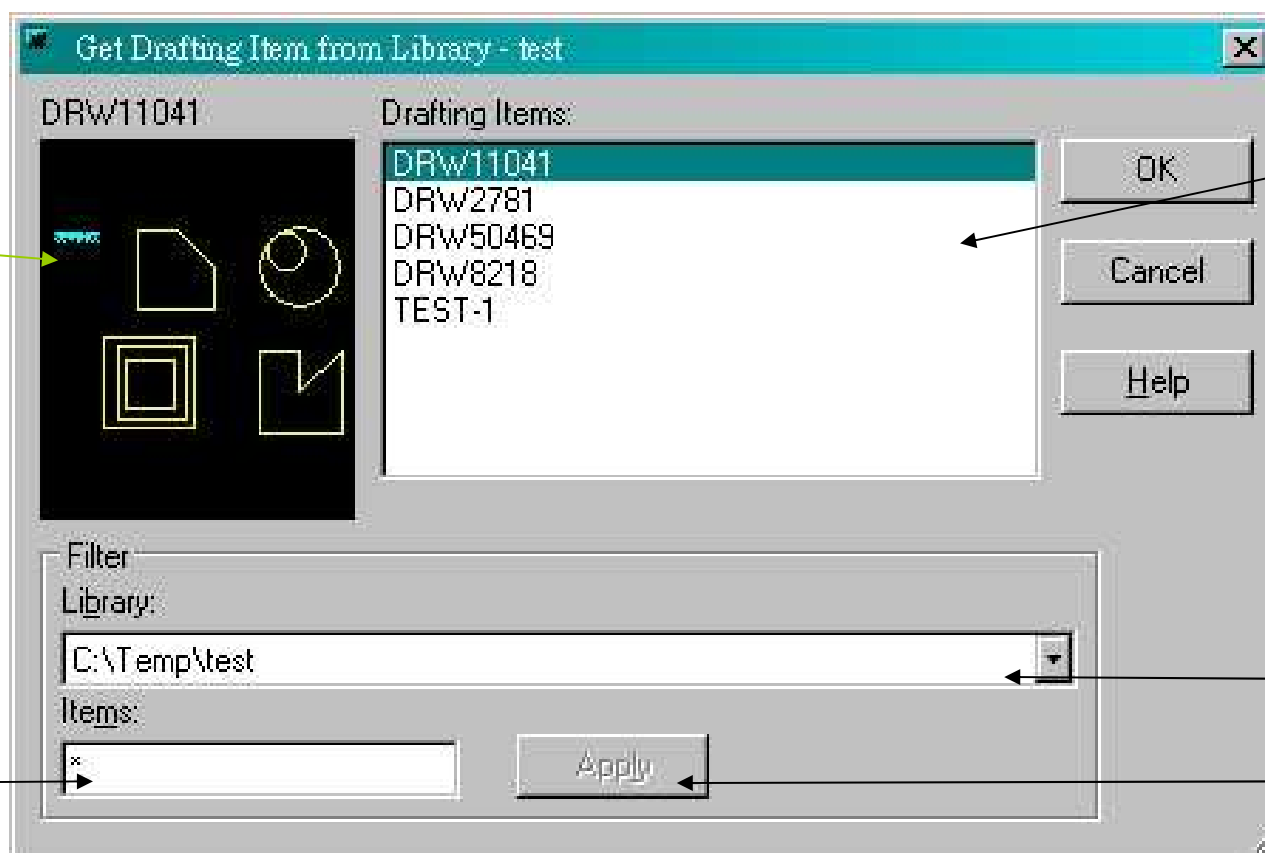
储存名称

取用组合物件

- 取用步骤：



Design Toolbar > Add 2D Line from Library



零件预览

零件列表

选取零件库

筛选控制

套用

Field

- Design Toolbar > Add Field



Add Field

Name:

Value:

Tip: to create a new user field, type the field name in to the Name box.

X: Y:

Font

Rotation: Line Width:

Size (pts): Font Style: **B** *I* U

Font: Arial Unicode MS

Justification

Horizontal: Right Center Left

Vertical: Up Center Down

OK Cancel Help

Field类别

文字内容

X坐标值

Y坐标值

旋转角度

文字线宽

文字大小

文字型态

选用字型

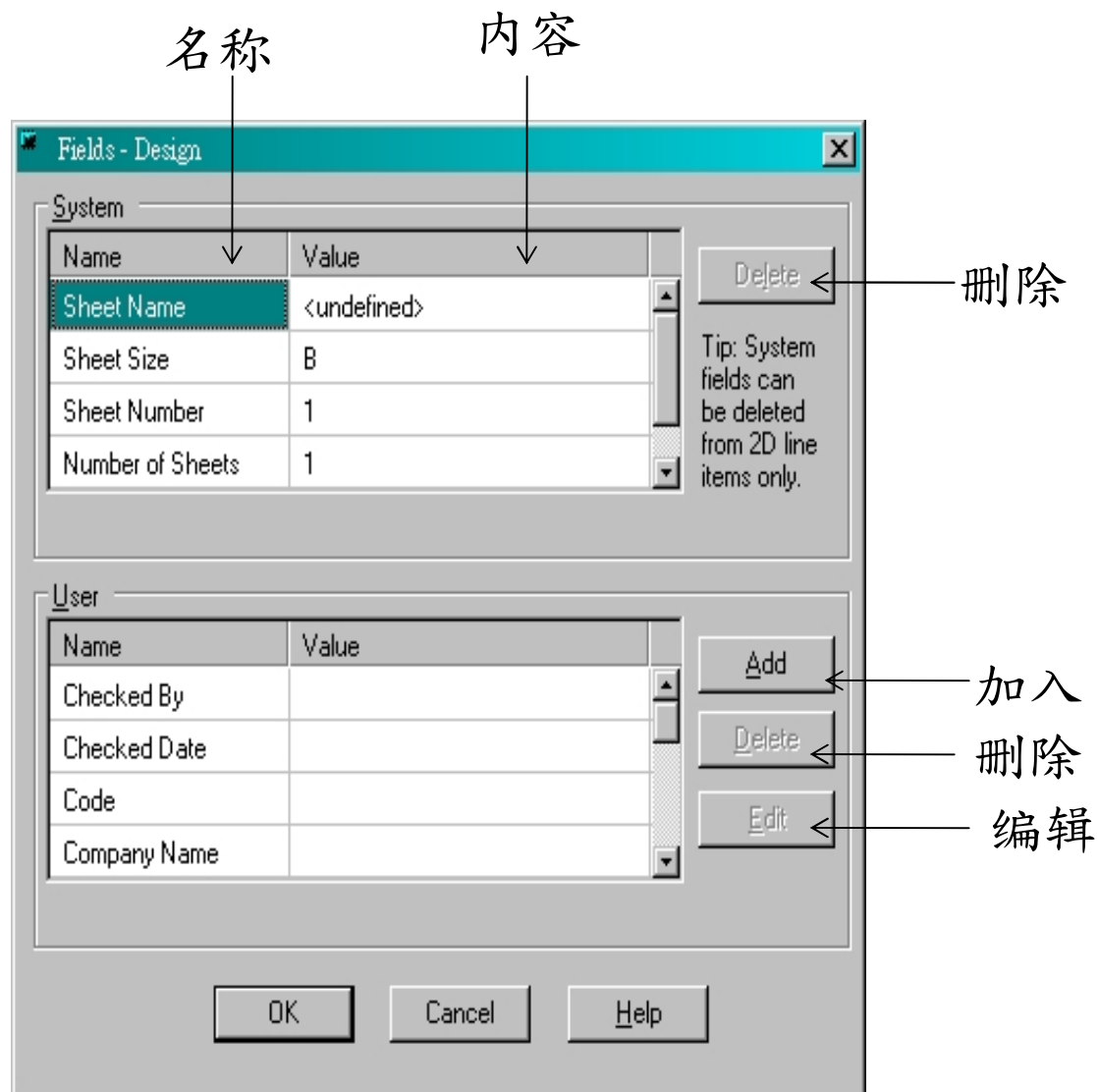
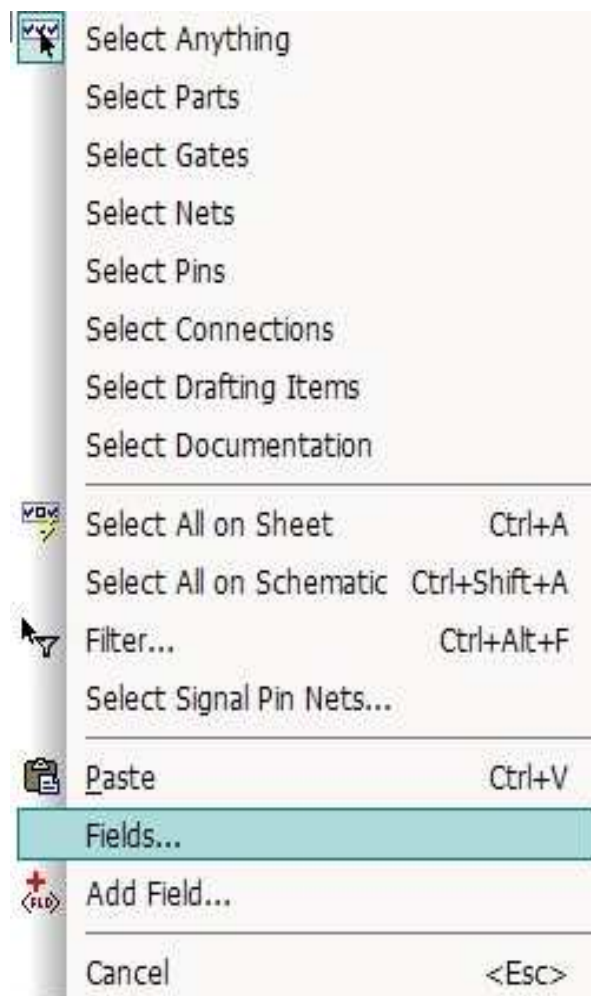
水平定位点

垂直定位点

- Sheet Name
- Sheet Size
- Sheet Number
- Number of Sheets
- Last Saved Date
- CHECKED
- CODE
- COMPANY
- COMPANY ADDRESS
- Checked By
- Checked Date
- Company Name
- DRAWING NO
- DRAWN
- Drawing Number
- Drawn By
- Drawn Date
- QC By
- QC Date
- QUALITY CONTROL
- RELEASED
- REV
- Release Date
- Released By
- Revision
- SCALE
- TITLE

Field

- 鼠标器右键 > Fields...



练习二

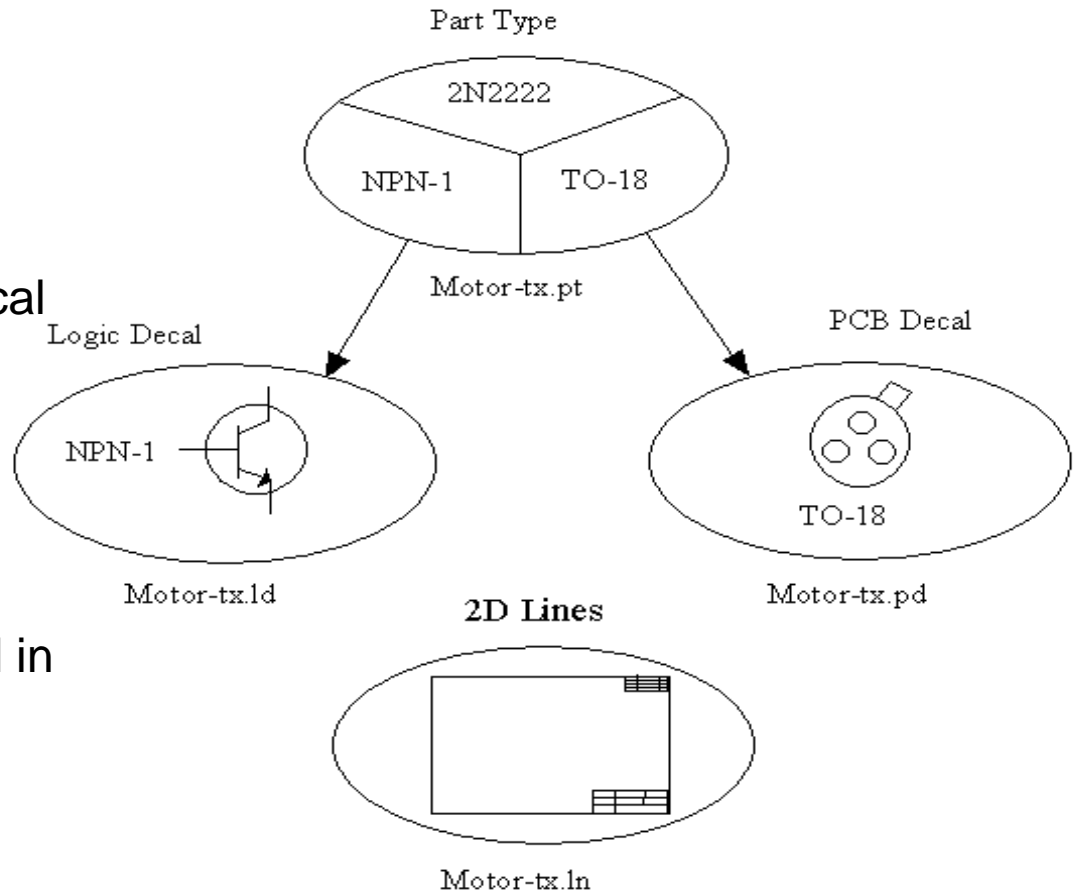
- 设计工具列
- 写文字
- 画线、修改线段
- 组合物件
- Field

第四课

零件库

零件库结构

- **Part Type (.pt07)**
 - Part name
 - Gate and pin information
 - Attributes
 - Points to the PCB Decal and CAE Decal
- **PCB Decal (.pd07)**
 - “Footprint” on PCB board
 - Pad stack information
 - Pin number assignments
- **CAE(symbol) Decal (.ld07)**
 - Schematic graphical symbol displayed in schematic layout editor
 - No pin number assignments
- **2D Lines (.ln07)**
 - Any drafting item (border, arrow markers, charts, board, copper outlines etc.)



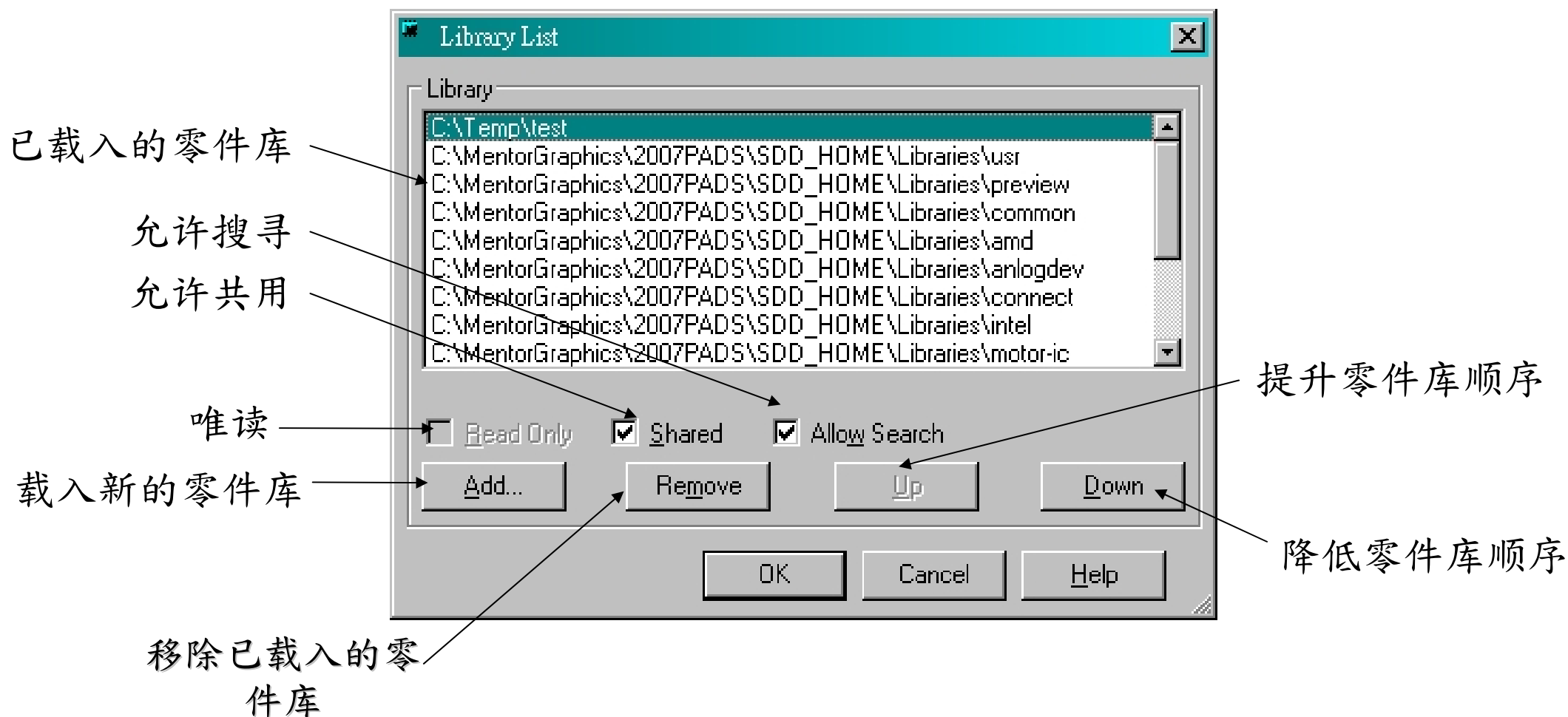
零件库管理器

File > Library

The screenshot shows the 'Library Manager - Browsing All Libraries' window. It features a 'Library' dropdown menu at the top left, currently set to '(All Libraries)'. Below it are three buttons: 'Create New Lib...', 'Manage Lib. List...', and 'Attr Manager...'. A central preview area shows a logic gate symbol (MY-7400) on a black background. To the right of the preview is a 'Filter' section with icons for 'Decals', 'Parts', 'Lines', and 'Logic', and a search input field containing an asterisk (*). Below the preview is a 'Part Types' list with entries like 'test:MY-7400', 'test:MY-RES2', and 'test:NEW_PART_1'. To the right of this list are buttons for 'New...', 'Edit...', 'Delete', and 'Copy'. At the bottom are 'Import...', 'Export...', and 'List to File' buttons, along with 'Close' and 'Help' buttons. Chinese annotations with arrows point to various elements: '浏览零件库' (Browse component library) points to the Library dropdown; '建立新零件库' (Create new component library) points to 'Create New Lib...'; '预览零件' (Preview component) points to the logic gate symbol; '零件列表' (Component list) points to the Part Types list; '导入零件' (Import component) points to 'Import...'; '管理零件属性' (Manage component attributes) points to 'Attr Manager...'; '管理零件库' (Manage component library) points to 'Manage Lib. List...'; '筛选控制' (Filter control) points to the search input field; '套用' (Apply) points to the 'Apply' button; '建立新零件' (Create new component) points to 'New...'; '编辑零件' (Edit component) points to 'Edit...'; '删除零件' (Delete component) points to 'Delete'; '复制零件' (Copy component) points to 'Copy'; '列表零件信息' (List component information) points to 'List to File'; and '导出零件' (Export component) points to 'Export...'.

管理零件库

- File > Library > Manage Lib. List...



建立零件 1/13

1. File > Library > Parts > New
2. File > New
- 3.

一般零件 → Part Type
连接符号 → Connector
线路图符号 → CAE Decal
Pin脚符号 → Pin Decal



建立零件 2/13

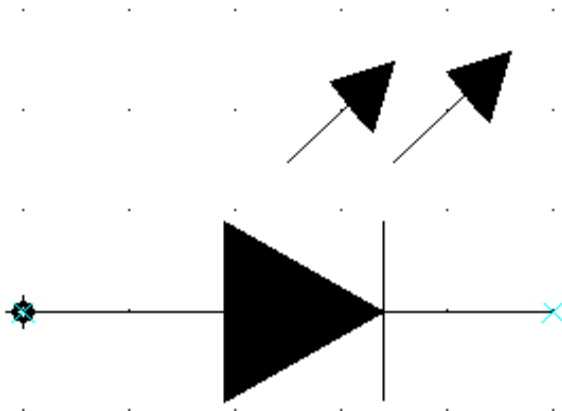
4.



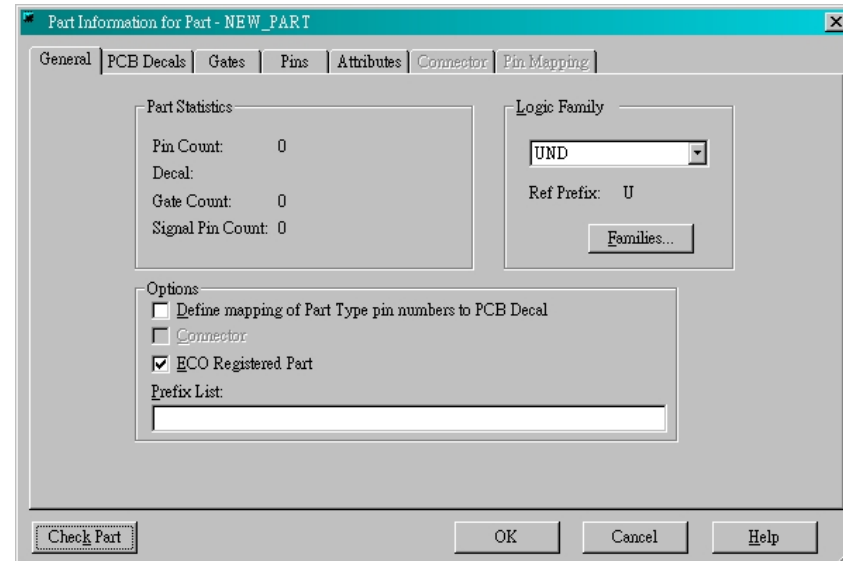
编辑线路图符号外型

编辑零件属性

建立线路图符号(CAE)外型

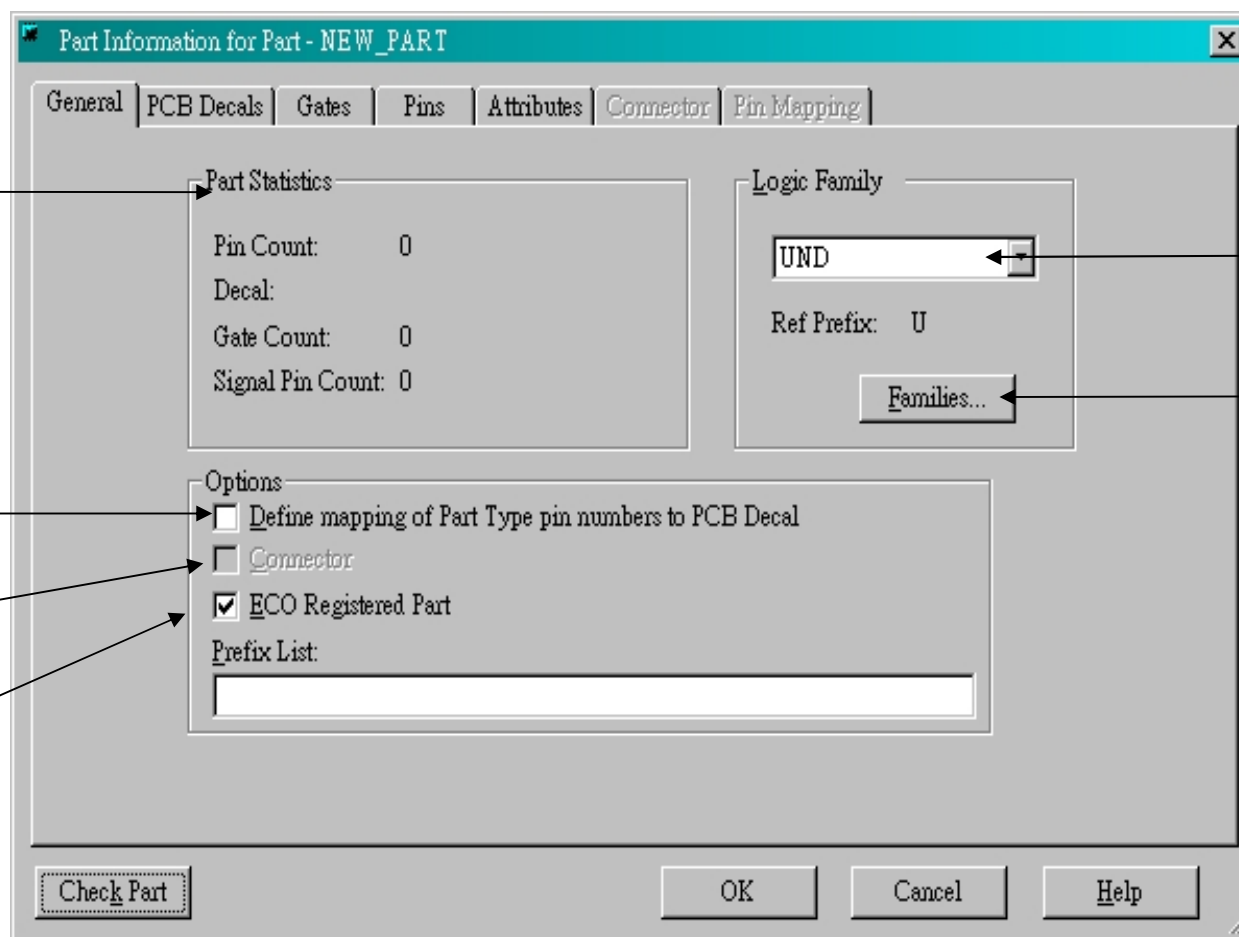


Set number of pins in the part, logic family, PCB decal, CAE decal, signal pins, gate and pin swaps, implicit pins, alphanumeric and connector symbols



建立零件 3/13

5. General 页面



零件Pin脚数量

RefDes选取

自订RefDes

自订Pin脚名称

连接符号

为ECO纪录元件

建立零件 4/13

6. PCB Decals 页面

Part Information for Part - NEW_PART

General PCB Decals Gates Pins Attributes Connector Pin Mapping

LED/RA

Library: (All Libraries)

Filter: * Pin Count: [] Apply

Show only Decals with pin numbers matching Part Type

Unassigned Decals:

- preview:LED/RA
- preview:OSC
- preview:SHIELD
- common:0402
- common:0603
- common:0805
- common:0805MIL
- common:1005
- common:1005MIL

Assigned Decals:

Assign >> << Unassign

Assign New Up Down

Reset

Check Part OK Cancel Help

预览PCB外型

PCB外型包装列表

重设

选择零件库

筛选名称

筛选Pin脚数量

套用

选用

选用新的

取消选用

提高顺序

降低顺序

已选用列表

建立零件 5/13

7. 页面

预览Gate

修改

重设

增加

删除

Part Information for Part - NEW_PART

General | PCB Decals | **Gates** | Pins | Attributes | Connector | Pin Mapping

Reset

Edit Add Delete

Gate	Pins	Swap	CAE Decal 1	CAE Decal 2	CAE Decal 3	CAE Decal 4
A	0	0	NAND2			
B	0	0	NAND2			
C	0	0	NAND2			
D	0	0	NAND2			

Check Part OK Cancel Help

建立零件 6/13

8. Pins 页面

Part Information for Part - NEW_PART

General | PCB Decals | Gates | Pins | Attributes | Connector | Pin Mapping

Tip: Multiple values in one column can be edited at once by using the Edit button.

Pin Group	Number	Name	Type	Swap	Seq.
Gate-A	1		Undefined	0	1
Gate-A	2		Undefined	0	2
Gate-A	3		Undefined	0	3
Gate-B	4		Undefined	0	1
Gate-B	5		Undefined	0	2
Gate-B	6		Undefined	0	3
Gate-C	8		Undefined	0	1
Gate-C	9		Undefined	0	2
Gate-C	10		Undefined	0	3
Gate-D	11		Undefined	0	1
Gate-D	12		Undefined	0	2
Gate-D	13		Undefined	0	3
Signal Pin	7	GND			
Signal Pin	14	VCC			

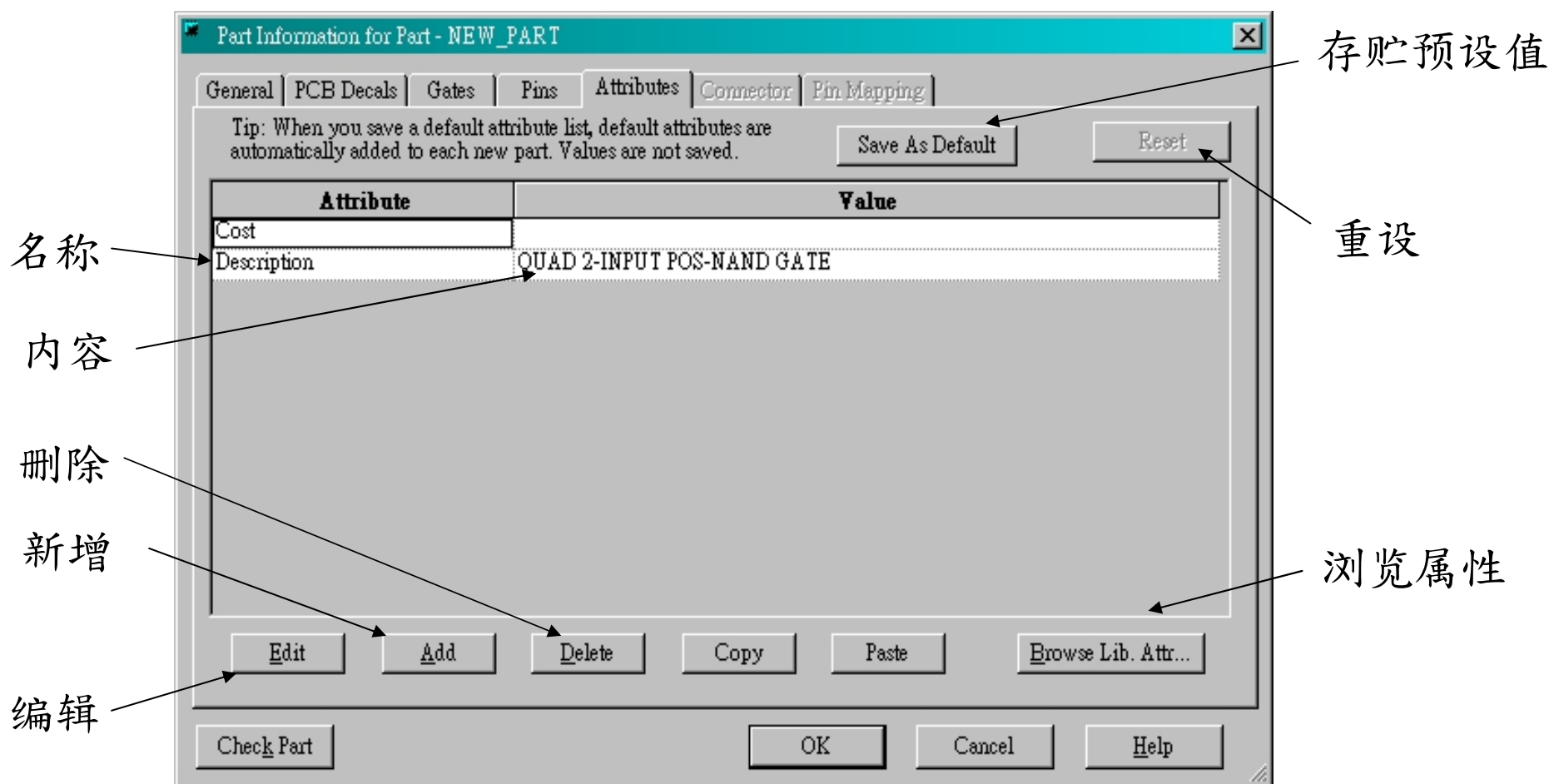
Buttons: Reset, Edit, Add Pin, Add Pins..., Delete Pins, Renumber..., Copy, Paste, Import CSV

Annotations:

- Pin脚编号 (Pin Number)
- 讯号名称 (Signal Name)
- 重设 (Reset)
- 加入 (Edit)
- 大量加入 (Add Pin)
- 移出 (Add Pins...)
- 更换Pin编号 (Delete Pins)
- 可大量编辑 (Renumber...)
- 复制 (Copy)
- 贴上 (Paste)
- 导入CSV档 (Import CSV)

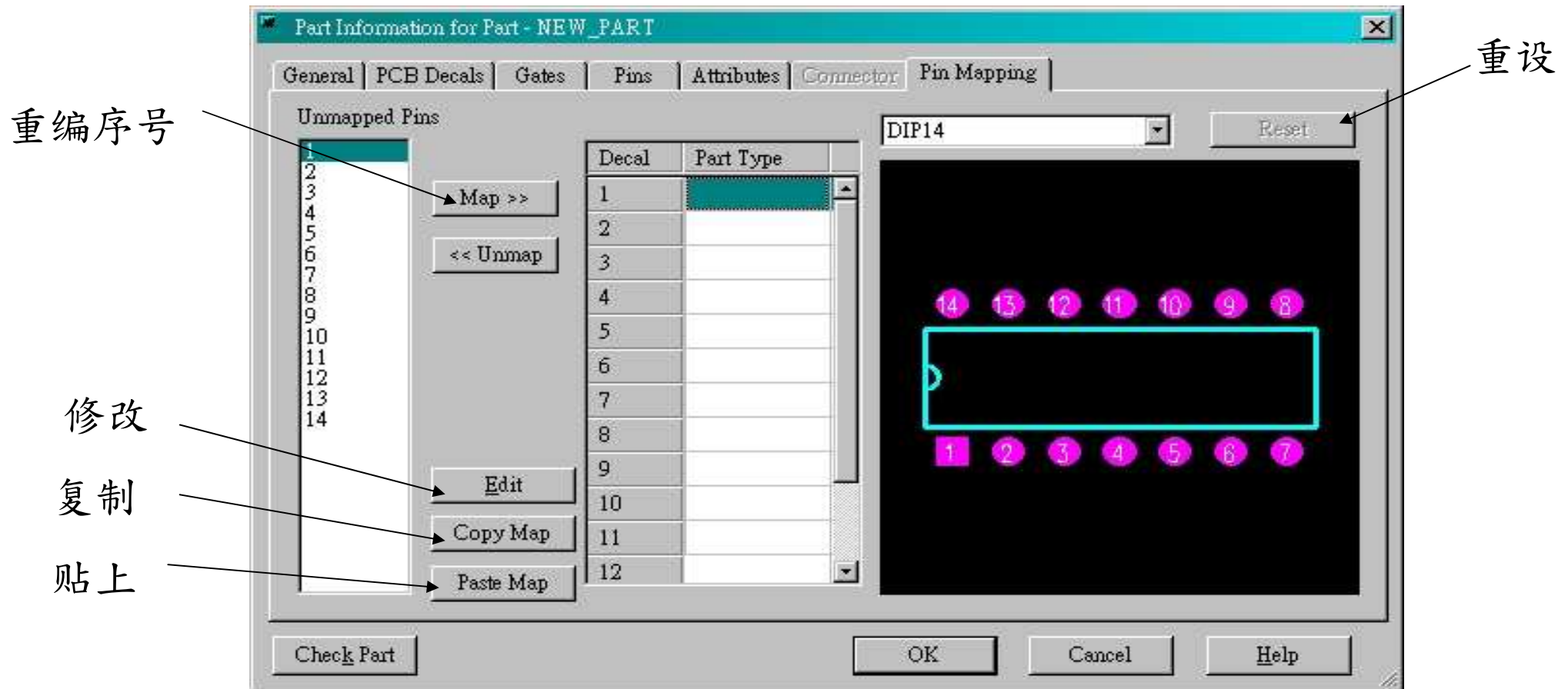
建立零件 7/13

9. Attributes 页面



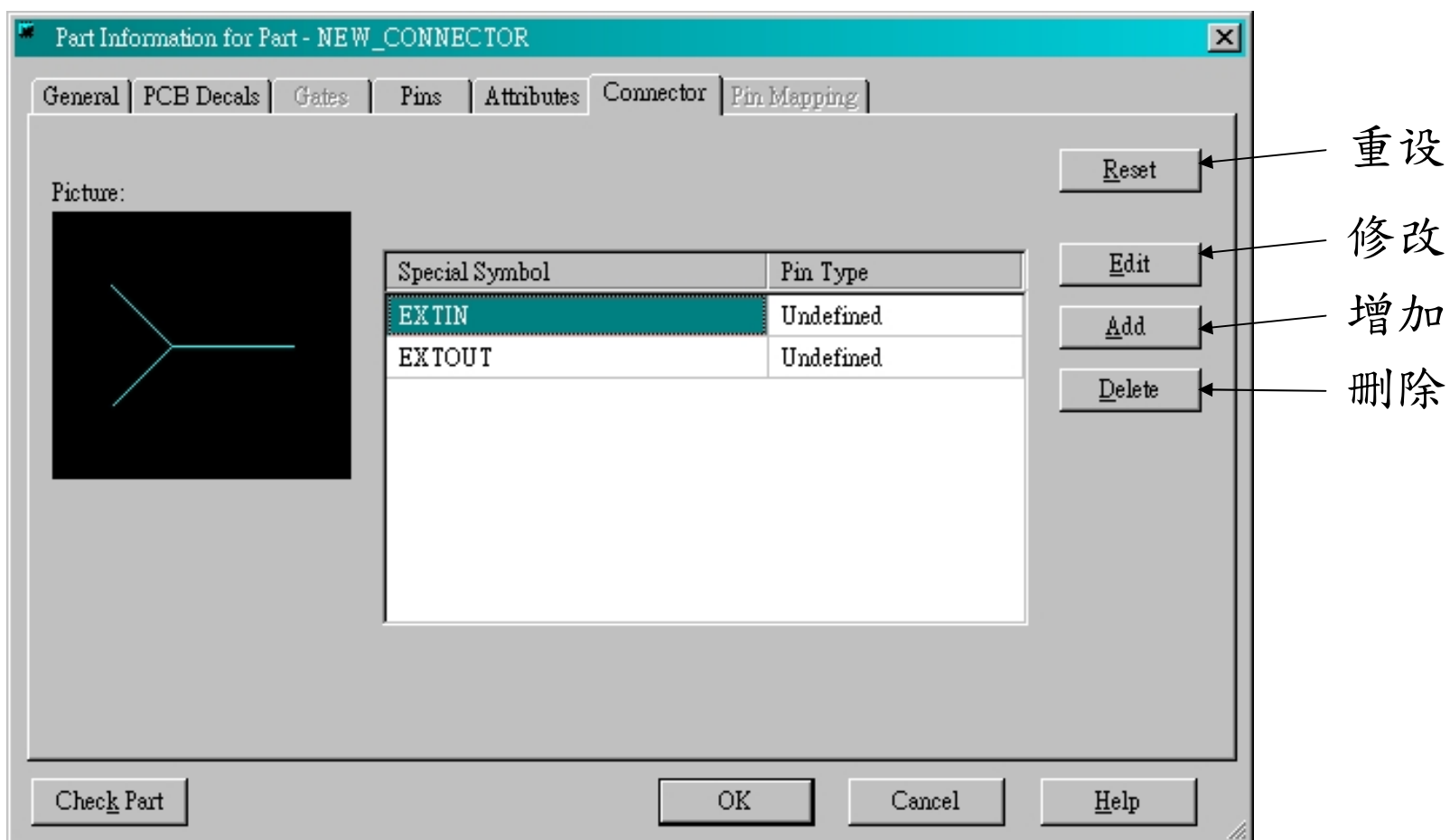
建立零件 8/13

10. Pin Mapping



建立零件 9/13

11. Connector 页面



建立零件 10/13

12.



绘制工具
(非电气特性)

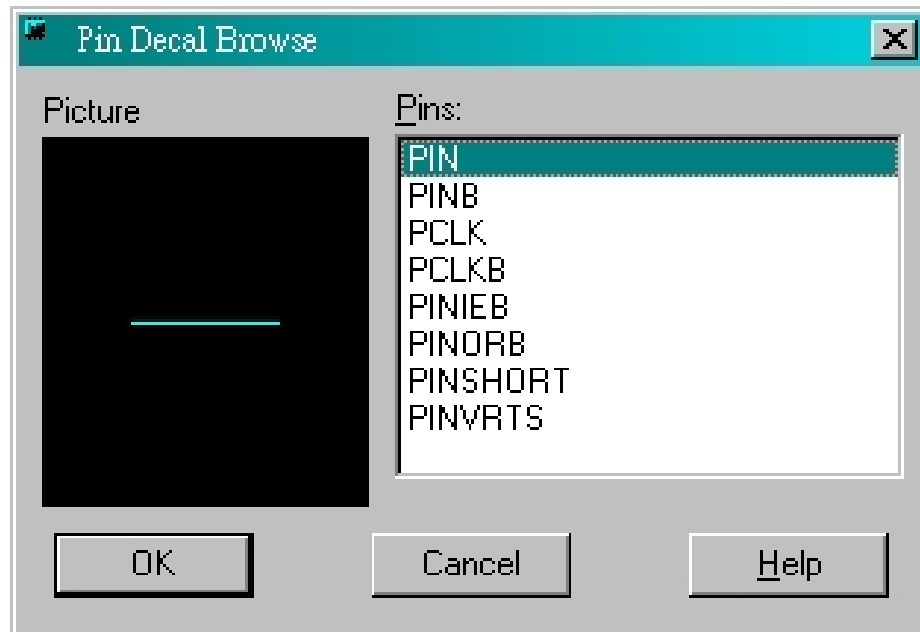
精灵

增加属性
标签

Pin脚工具
(有电气特性)

建立零件 12/13

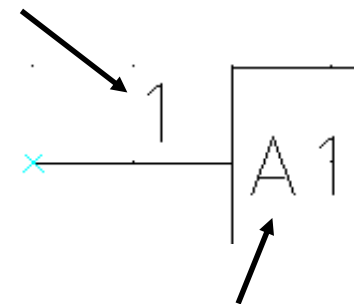
14. Design Toolbar > Add Terminal



#1:TYP=U SWP=0 NETNAME.

Sequence Number Pin Type Pin Swap

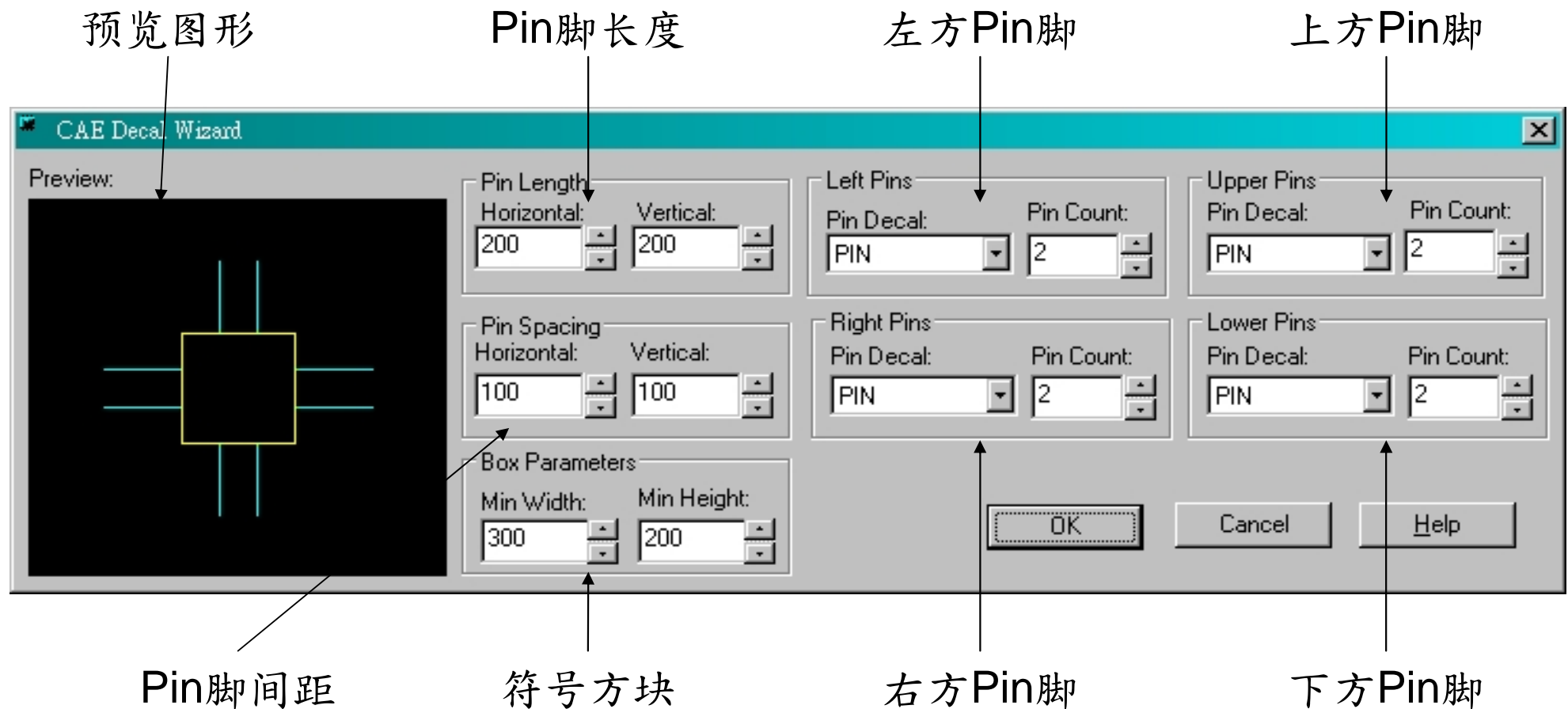
Pin Number



Pin Name

建立零件 13/13

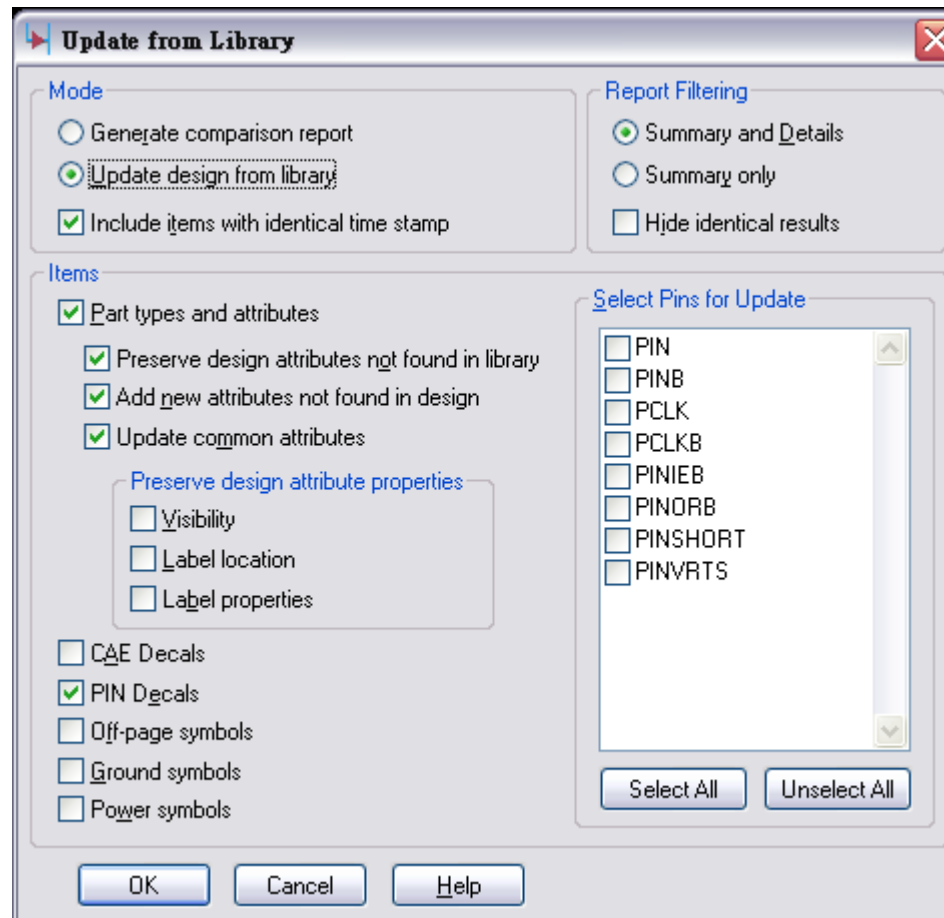
14. Design Toolbar > CAE Decal Wizard



零件库更新

Tools > Update from library

设计者可确认最近一次所使用数据库里符号和组件的定义，并对零件做更新



练习三

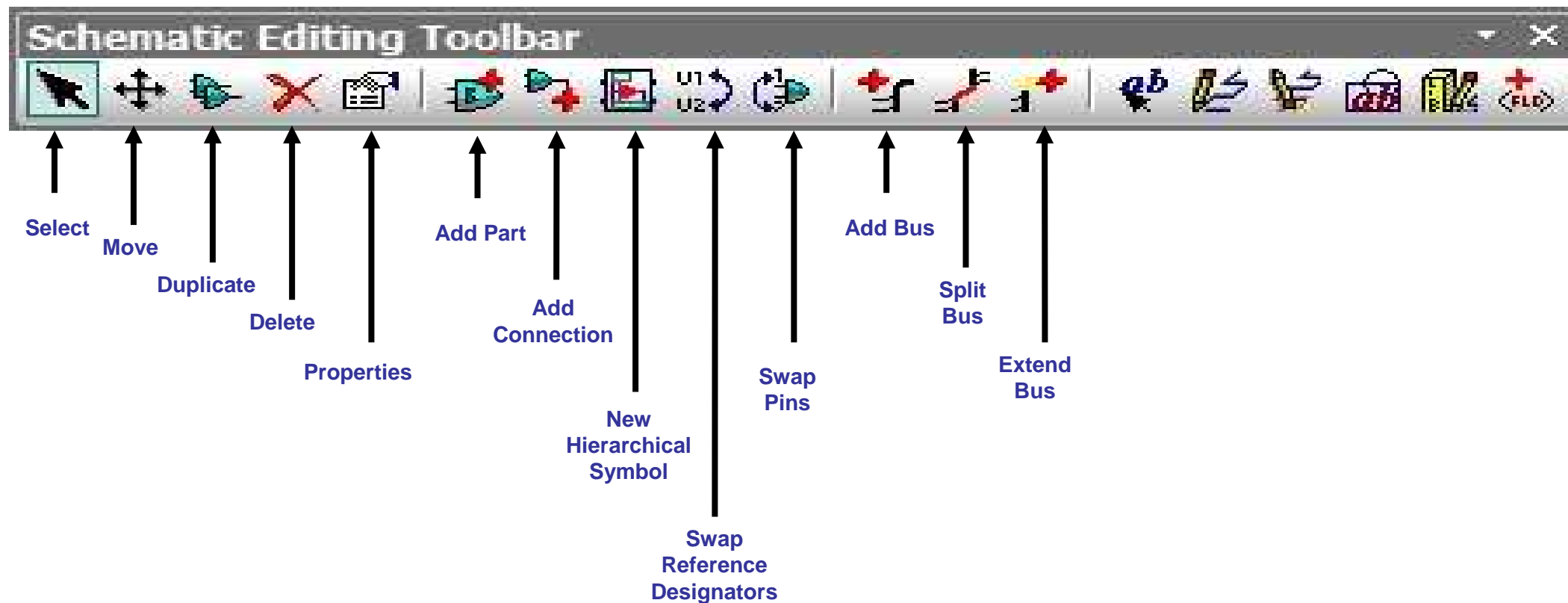
- 建立电阻
- 建立电容
- 建立7400
- CAE Decal Wizard

第五课

绘制线路图

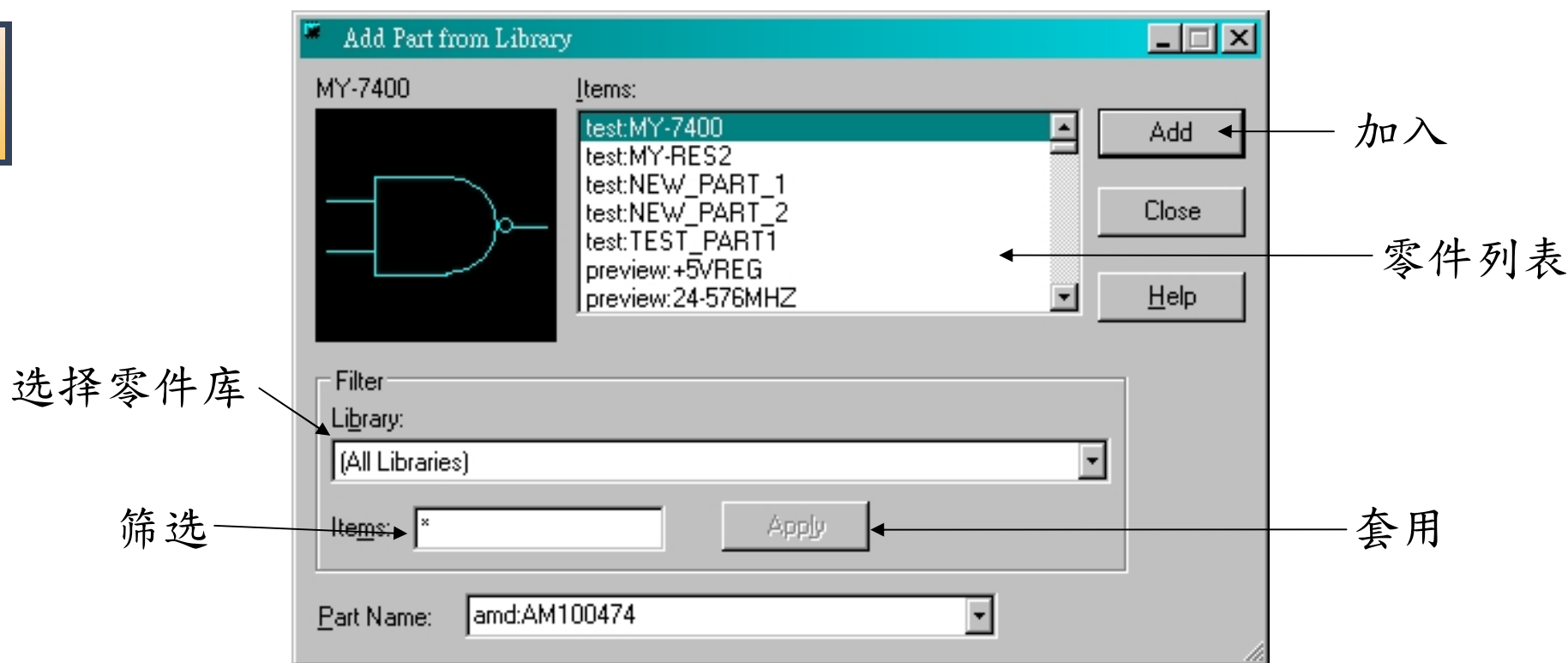
Design Toolbar

◆ 设计工具列



摆放线路图零件符号

- Design Toolbar > Add Part



画电气线

- Design Toolbar > Connection



删除转角	→	Del Corner	Back
选择角度	→	Angle	
选择跨页符号	→	Off-page	Alt+Space
选择接地符号	→	Ground	Ctrl+Space
选择电源符号	→	Power	Shift+Space
暂停	→	End	
显示电源Net的名称	→	Display PG Name	
		Cancel	Esc

	Rotate 90	Ctrl+R
	X Mirror	Ctrl+F
	Y Mirror	Shift+Ctrl+F
	Alternate	Ctrl+Tab
	Cancel	Esc

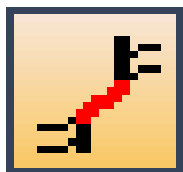
选择其他样式

画Bus线

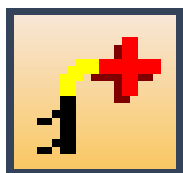
- Design Toolbar > Add Bus



画Bus线



修改Bus线



延伸Bus线

Bus名称

选取部份

所有相同名称部份

混合式

固定式

显示Bus名称

新增

修改

删除

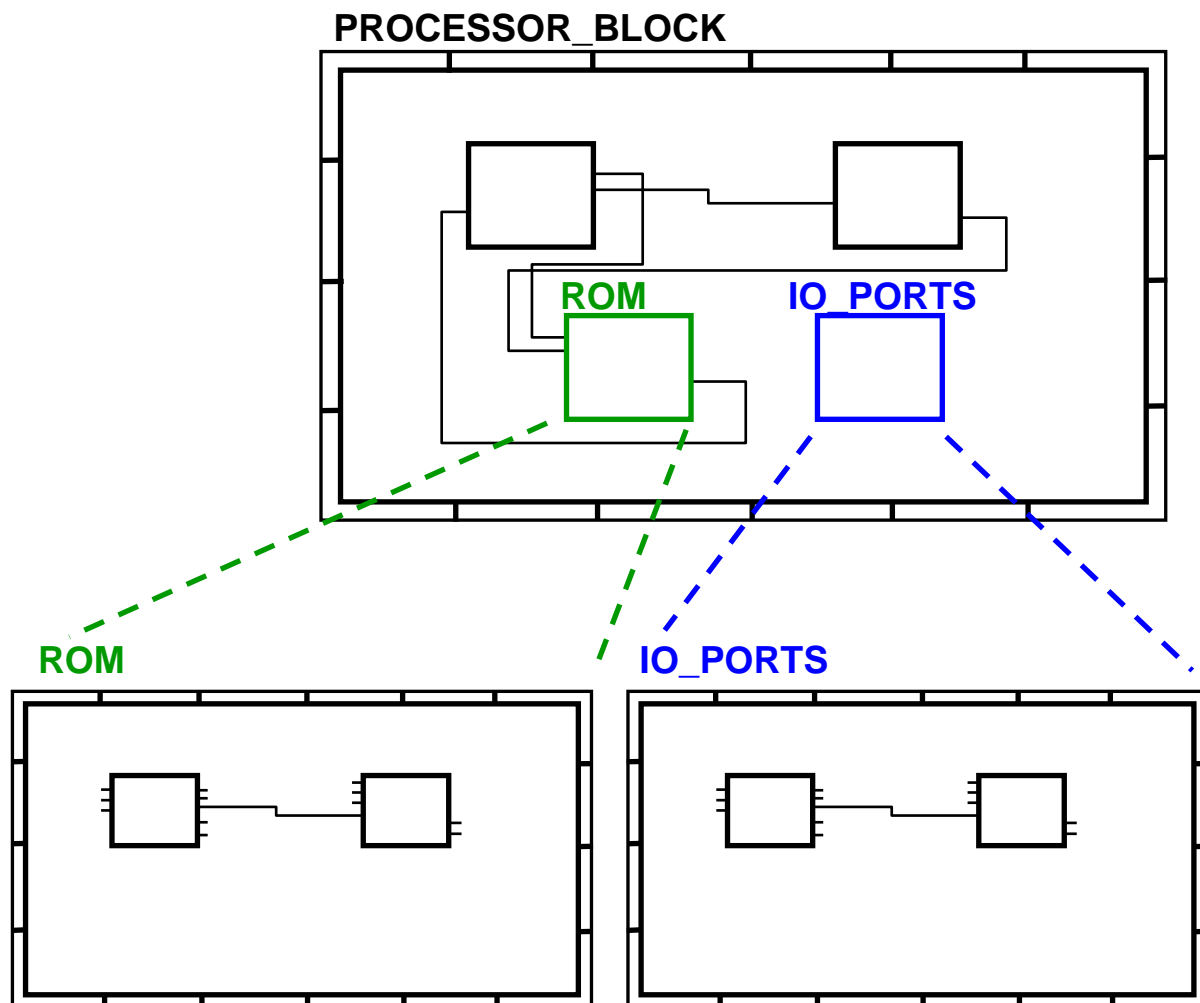
降低顺序

提高顺序

建立阶层式线路图 1/2

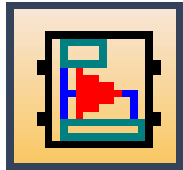
Design Toolbar > New Hierarchical Symbol

- 具有Net与母层有所连接(ROM)
- 具有Net不与母层有所连接(IO_PORTS)



建立阶层式线路图 2/2

- Design Toolbar > New Hierarchical Symbol



预览图形 Pin脚长度 Pin脚间距 输入Pin脚 阶层式线路图选取

The screenshot shows the 'Hierarchical Symbol Wizard' dialog box with the following fields and annotations:

- Preview:** A preview window showing a symbol block with four pins. Annotation: 预览图形.
- Pin Parameters:** Fields for Pin Length (200) and Pin Spacing (150). Annotations: Pin脚长度 and Pin脚间距.
- Input Pins:** Fields for Pin Decal (PIN) and Pin Count (2). Annotation: 输入Pin脚.
- Output Pins:** Fields for Pin Decal (PIN) and Pin Count (2). Annotation: 输出Pin脚.
- Hierarchical Sheet:** Fields for Sheet Number (<New Sheet>) and Sheet Name (empty). Annotation: 阶层式线路图选取.
- Box Parameters:** Fields for Box Width (400) and Min Box Hgt (300). Annotation: 符号方块.
- Buttons:** OK, Cancel, and Help buttons. Annotation: 阶层式线路图名称.

练习四

- Add Part
- Add Connection
- Add Power、Ground、Off-page
- Add Bus
- Add Hierarchical Symbol

第六课

PCB设置与规则

层面堆叠

- Setup > Layer Definition

The screenshot shows the 'Layers Setup' dialog box with the following details:

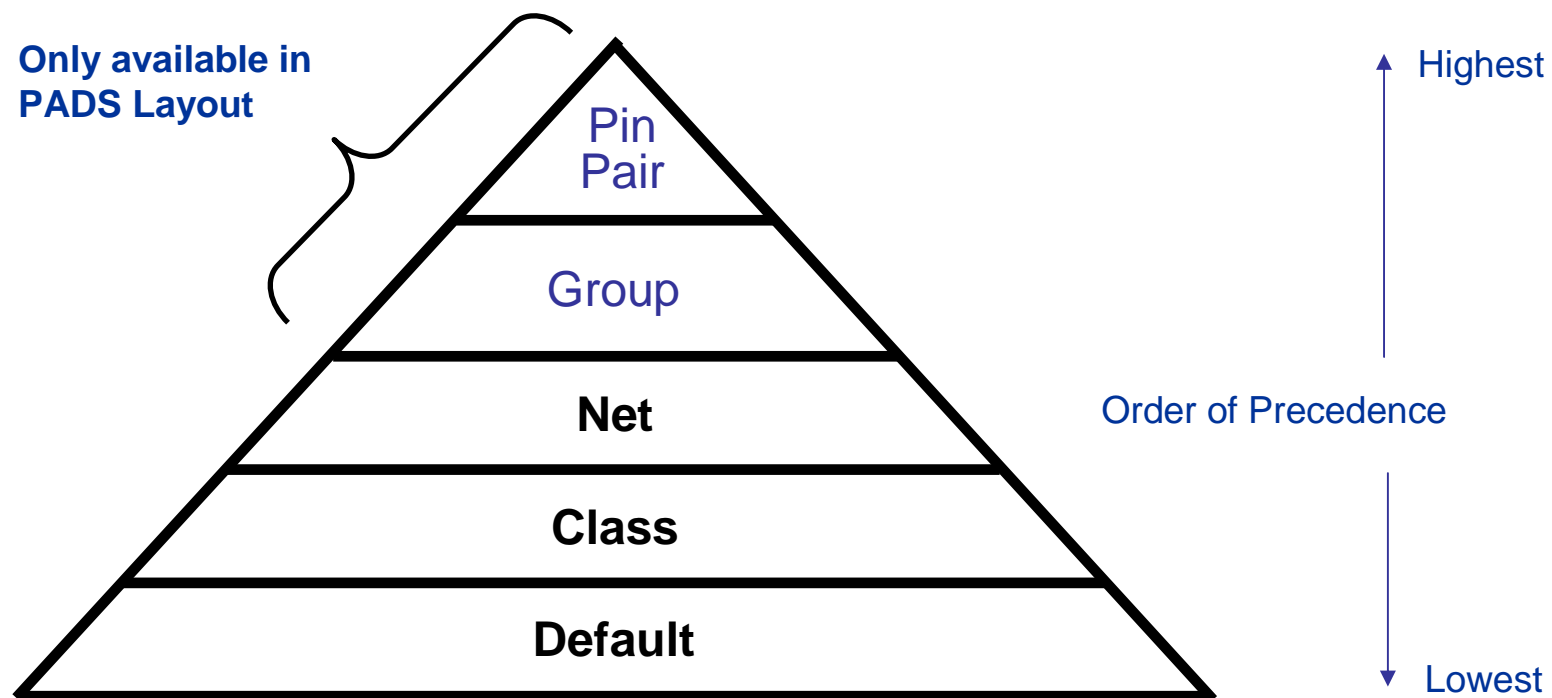
Lev.	Type	Dir.	Name
1	CM	H	Top
2	CM	V	Bottom
3	GN		Layer_3
4	GN		Layer_4
5	GN		Layer_5
6	GN		Layer_6
7	GN		Layer_7
8	GN		Layer_8

Annotations and their corresponding UI elements:

- 层面列表 (Layer List) → Points to the table of layers.
- 层面名称(可中文) (Layer Name (can be Chinese)) → Points to the 'Name' field.
- 层面类别 (Layer Category) → Points to the 'Electrical Layer Type' section.
- 层面种类 (Layer Type) → Points to the 'Plane Type' and 'Routing Direction' sub-sections.
- 设置布线层面数量 (Set the number of wiring layers) → Points to the 'Count' field.
- 设置布线角度 (Set wiring angle) → Points to the 'Routing Direction' options.
- 设置层面堆叠顺序 (Set layer stacking order) → Points to the 'Count' field.
- 设置层面厚度与电气参数 (Set layer thickness and electrical parameters) → Points to the 'Thickness' button.
- 层面最大限制 (Maximum layer limit) → Points to the 'Max Layers...' button.

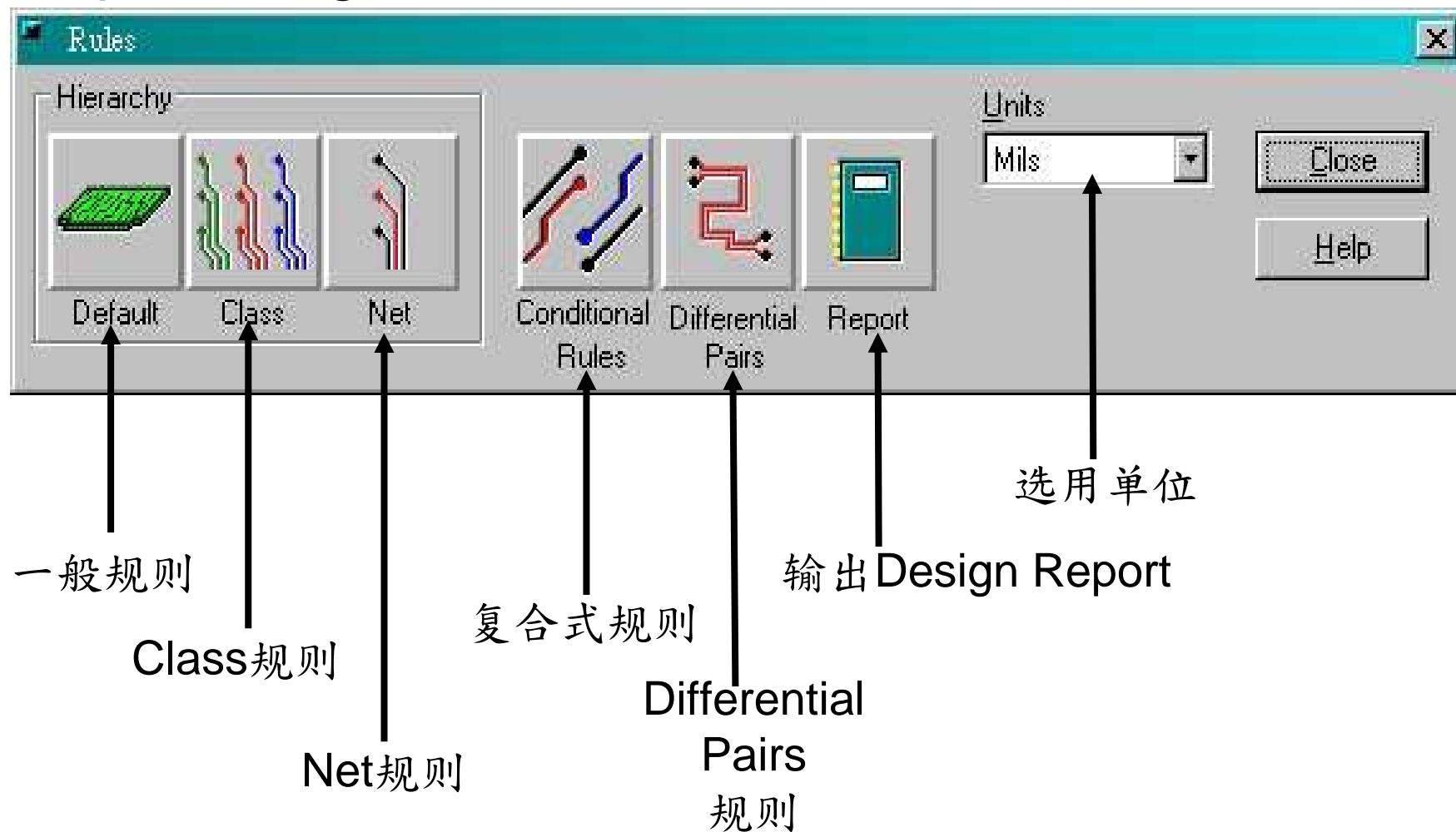
设置规则

- 规则优先顺序



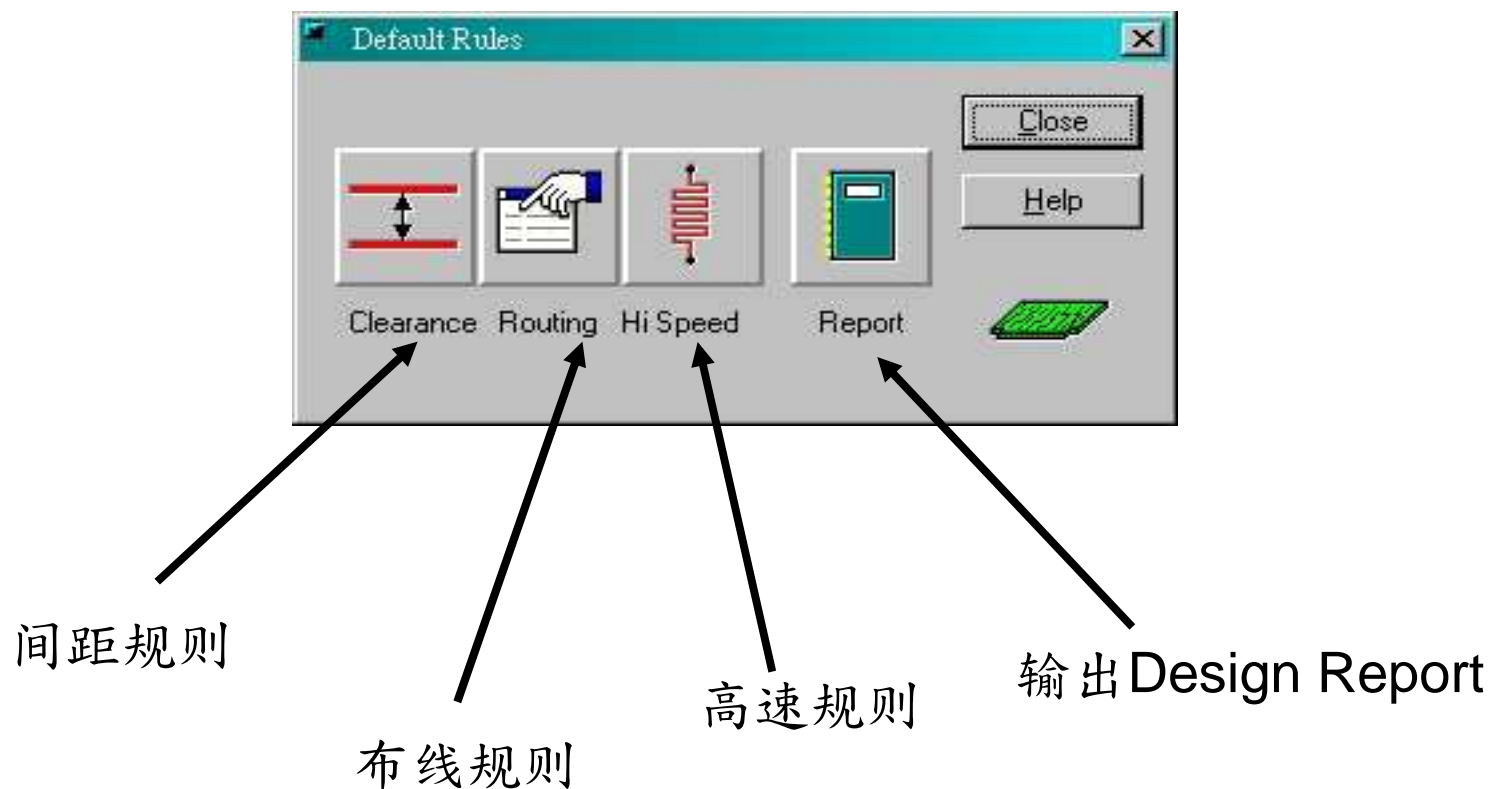
设置规则

- Setup > Design Rules



设置规则

- Setup > Design Rules > Default



Setup > Design Rules > Default > Clearance

Callouts for 'Same Net' table:

- 当同NetName时的安全间距检查 (Safety clearance check when same NetName)

Callouts for 'Trace Width' table:

- 最小线宽 (Minimum trace width)
- 标准线宽 (Standard trace width)
- 最大线宽 (Maximum trace width)

Callouts for 'Clearance' table:

- 钻孔与钻孔安全间距 (Safety clearance between drill holes)
- 元件与元件安全间距 (Safety clearance between components)
- 一般的安全间距检查 (General safety clearance check)

Legend (Red boxes):

- 走线 (Traces)
- 贯孔 (Through holes)
- 焊点 (Solder joints)
- SMT
- 文字 (Text)
- 铜箔 (Copper foil)
- 板框 (Board frame)
- 钻孔 (Drill holes)

Setup > Design Rules > Default > Routing

布线选项

结构类型

目前系统可用层面

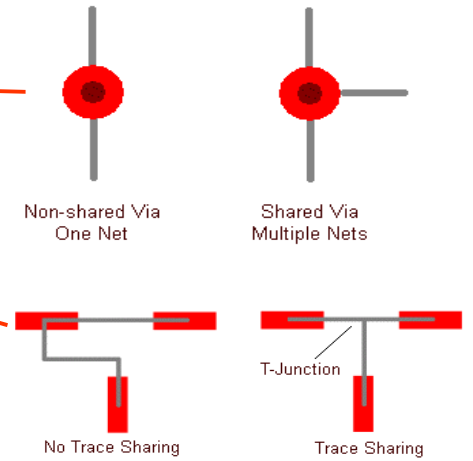
设置可走线的层面

目前系统可用贯孔

设置贯孔类型

设置可用的贯孔

限定可使用的贯孔数量



将目前系统中
Partial Via
设为可用的贯孔

Setup > Design Rules > Default > HiSpeed

The screenshot shows the 'HiSpeed Rules' dialog box with the following parameters and annotations:

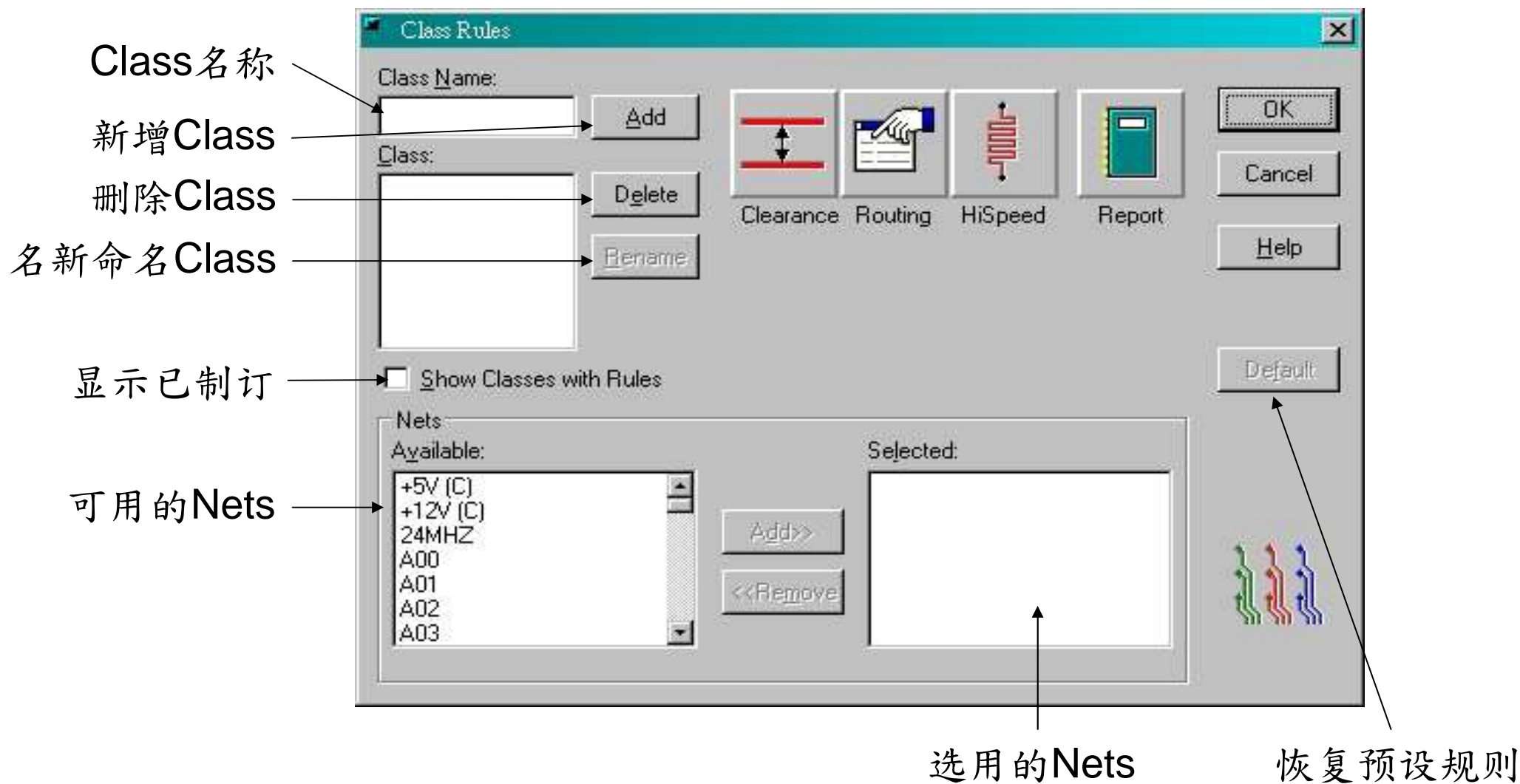
- Parallelism:** Length: 1000, Gap: 200
- Parallelism:** Tandem: 1000, Gap: 200
- Aggressor
- Shielding:** Shield, Gap: 200
- Use Net:** [Dropdown menu]
- Rules Table:**

Rules	Minimum	Maximum
Length:	0	50000
Stub Length:		0
Delay (ns):	0.000	10.000
Capacitance (pF):	0.000	10.000
Impedance (Ohm):	50.000	150.000
- Matching:** Match Lengths, Tolerance: 200

Annotations (in red boxes):

- 最短长度 (Shortest length) - points to the '0' in the Length Minimum field.
- 最大长度 (Maximum length) - points to the '50000' in the Length Maximum field.
- 允许走T线的长度 (Allow T-line length) - points to the '0' in the Stub Length field.
- 以Delay Time最小最大 (Minimum/Maximum Delay Time) - points to the '0.000' and '10.000' in the Delay (ns) row.
- 以电容值最小最大 (Minimum/Maximum Capacitance) - points to the '0.000' and '10.000' in the Capacitance (pF) row.
- 以阻抗值最小最大 (Minimum/Maximum Impedance) - points to the '50.000' and '150.000' in the Impedance (Ohm) row.
- Route时是否走线到等长长度 (用于Group AutoRoute) (Route whether to route to equal length (used for Group AutoRoute)) - points to the 'Match Lengths' checkbox.
- Route时等长长度的Tolerance (Tolerance of equal length when routing) - points to the '200' in the Tolerance field.

Setup > Design Rules > Class



Setup > Design Rules > Conditional Rule

复合式Design Rules
第一组设置

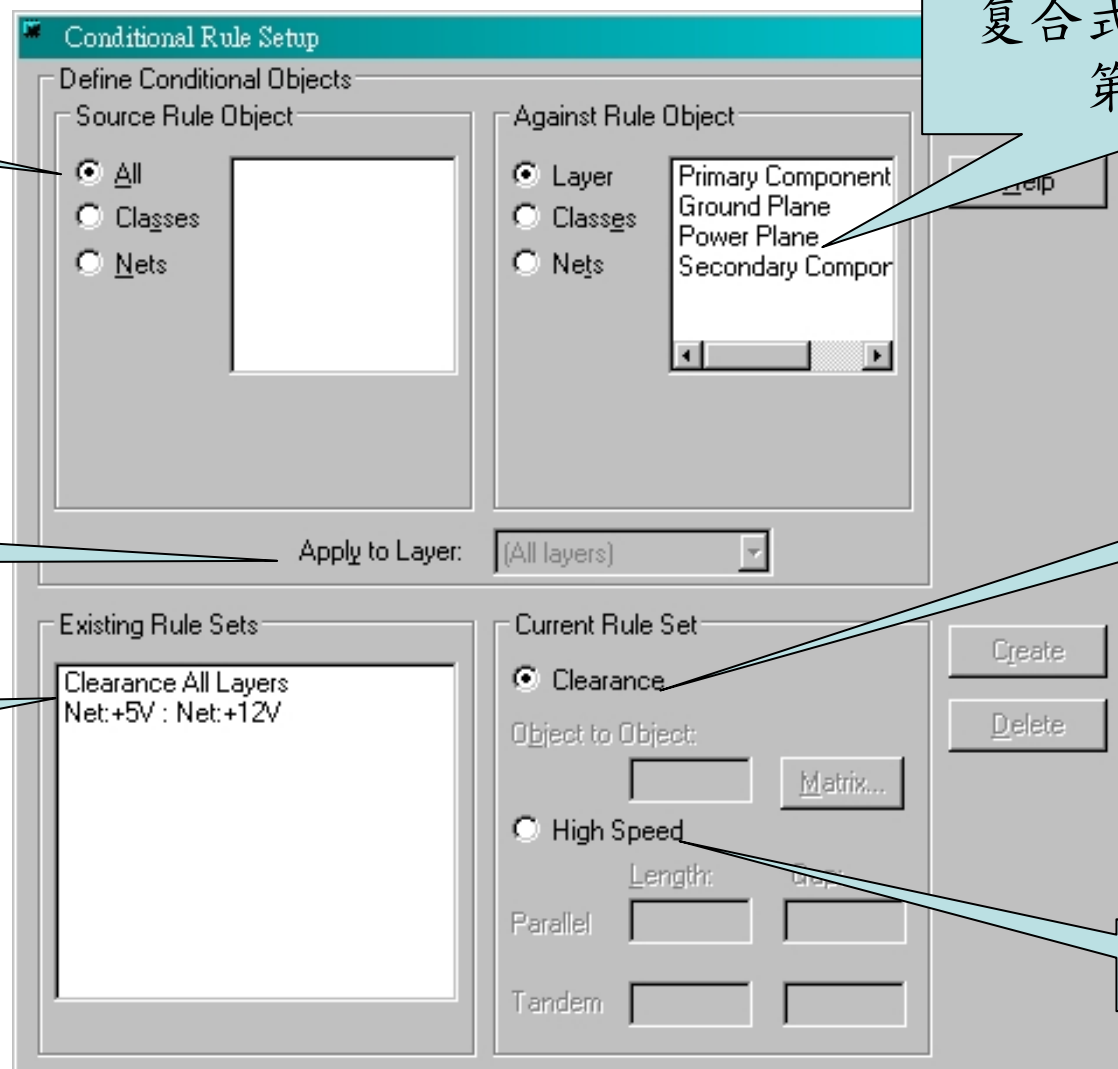
复合式Design Rules
第二组设置

限定层面

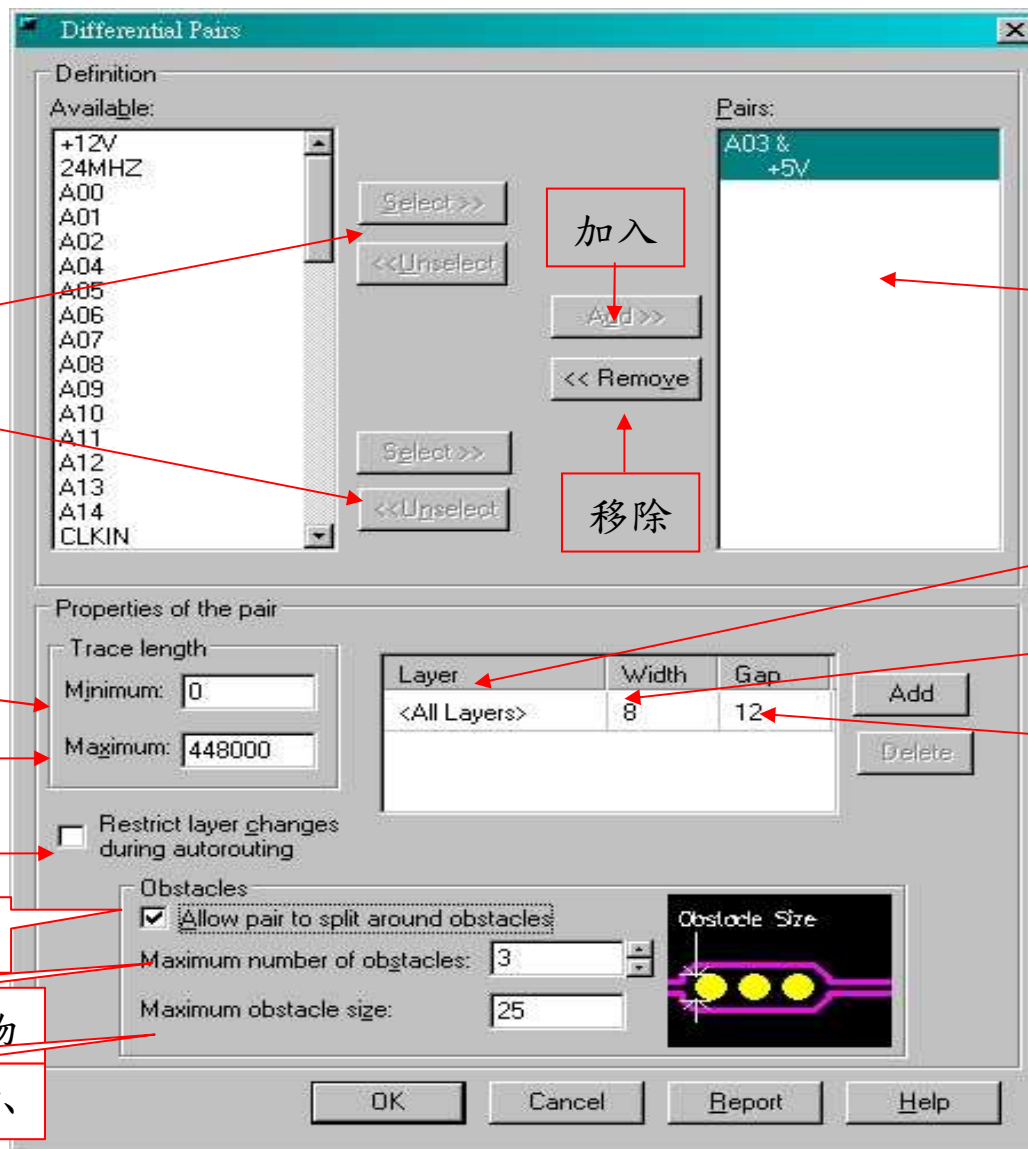
Design Rules 描述

安全距离设置

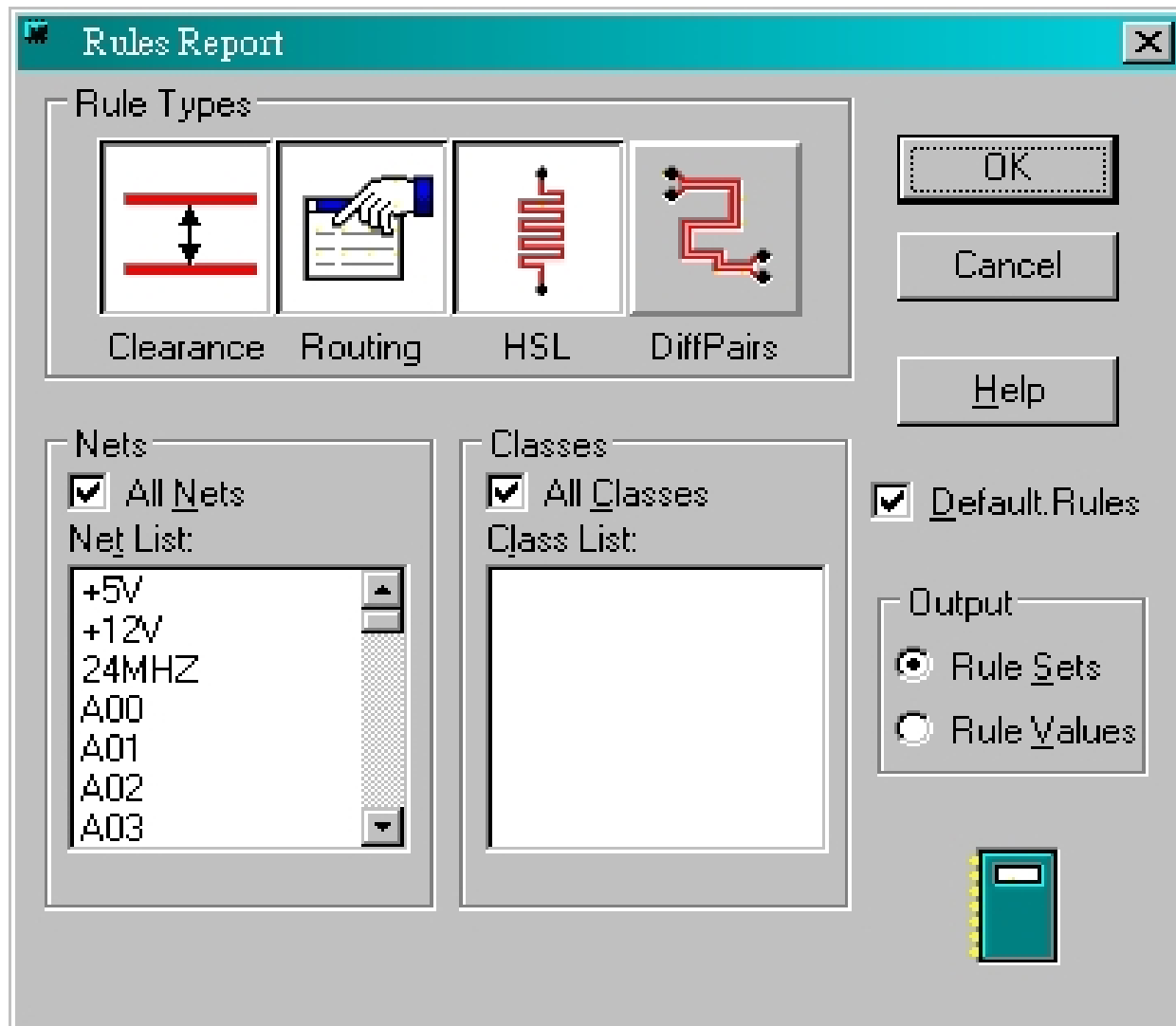
High Speed设置



Setup > Design Rules > Differential Pairs



Setup > Design Rules > Report



练习五

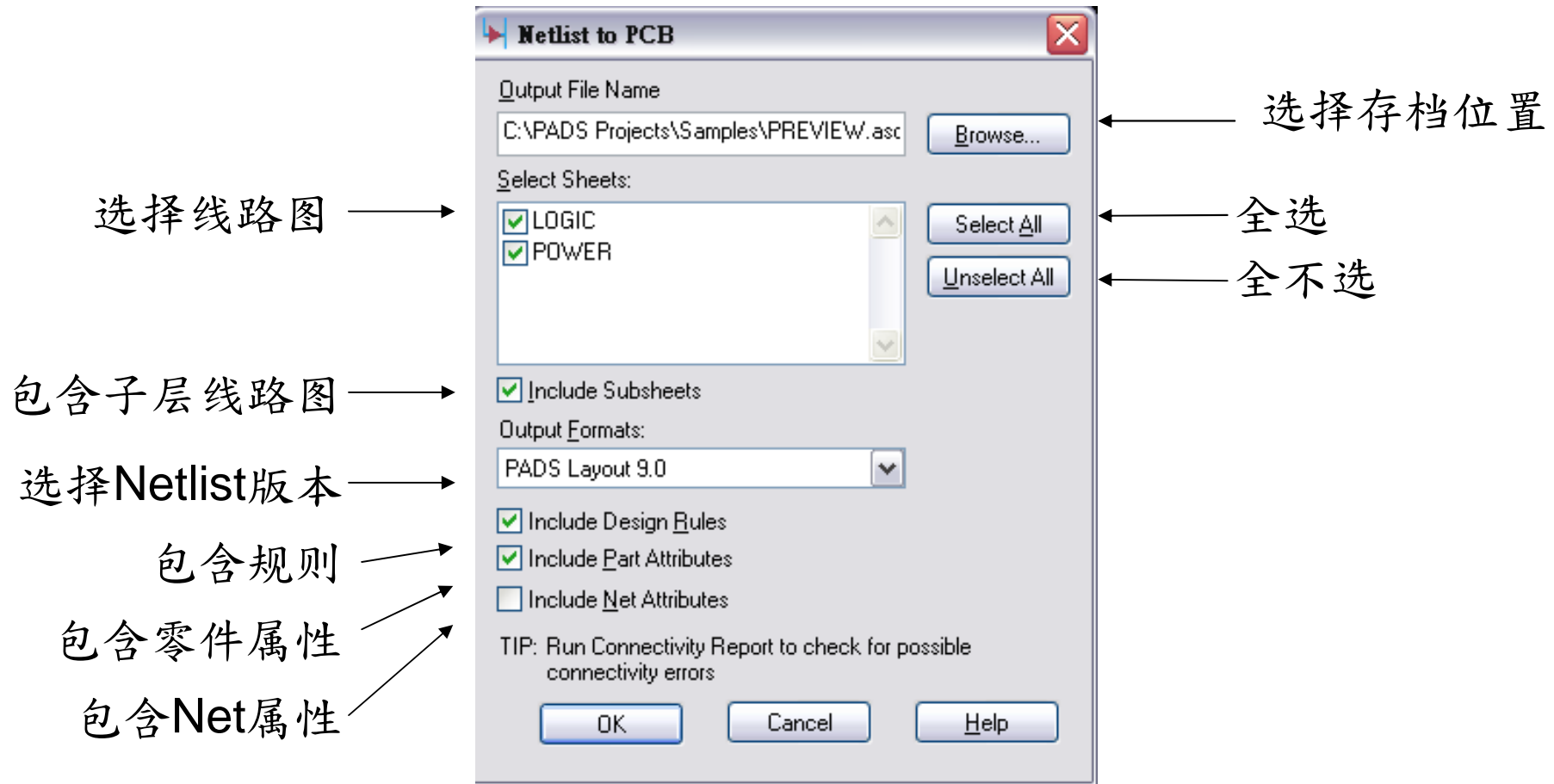
- 层面堆叠
- Clearance Rule
- Routing Rule
- HiSpeed Rule
- Class Rule
- Conditional Rule
- Differential Pairs
- Report

第七课

资料输出

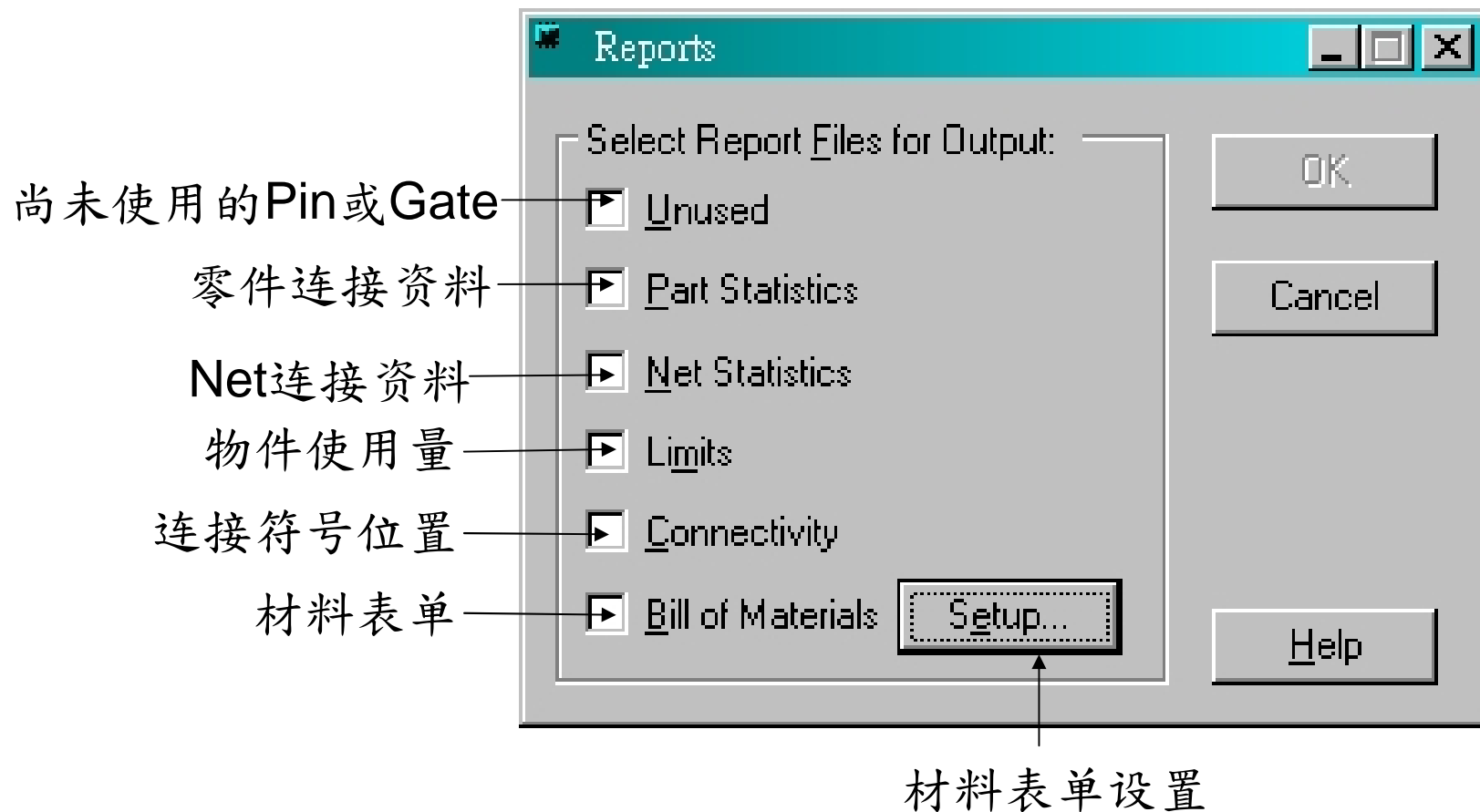
输出连线关系 (Netlist) 文件

- Tools > Layout Netlist



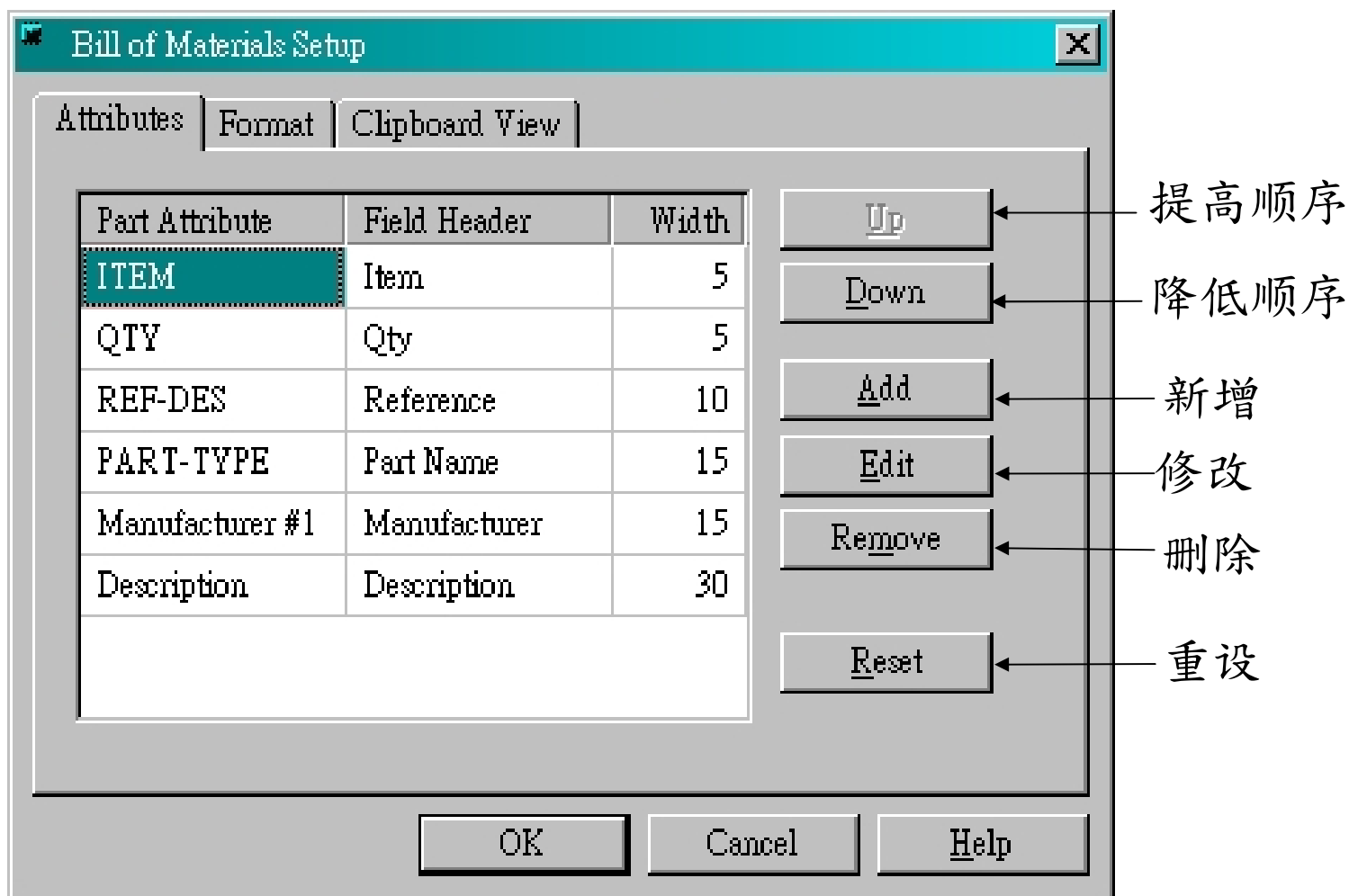
输出报表

- File > Reports



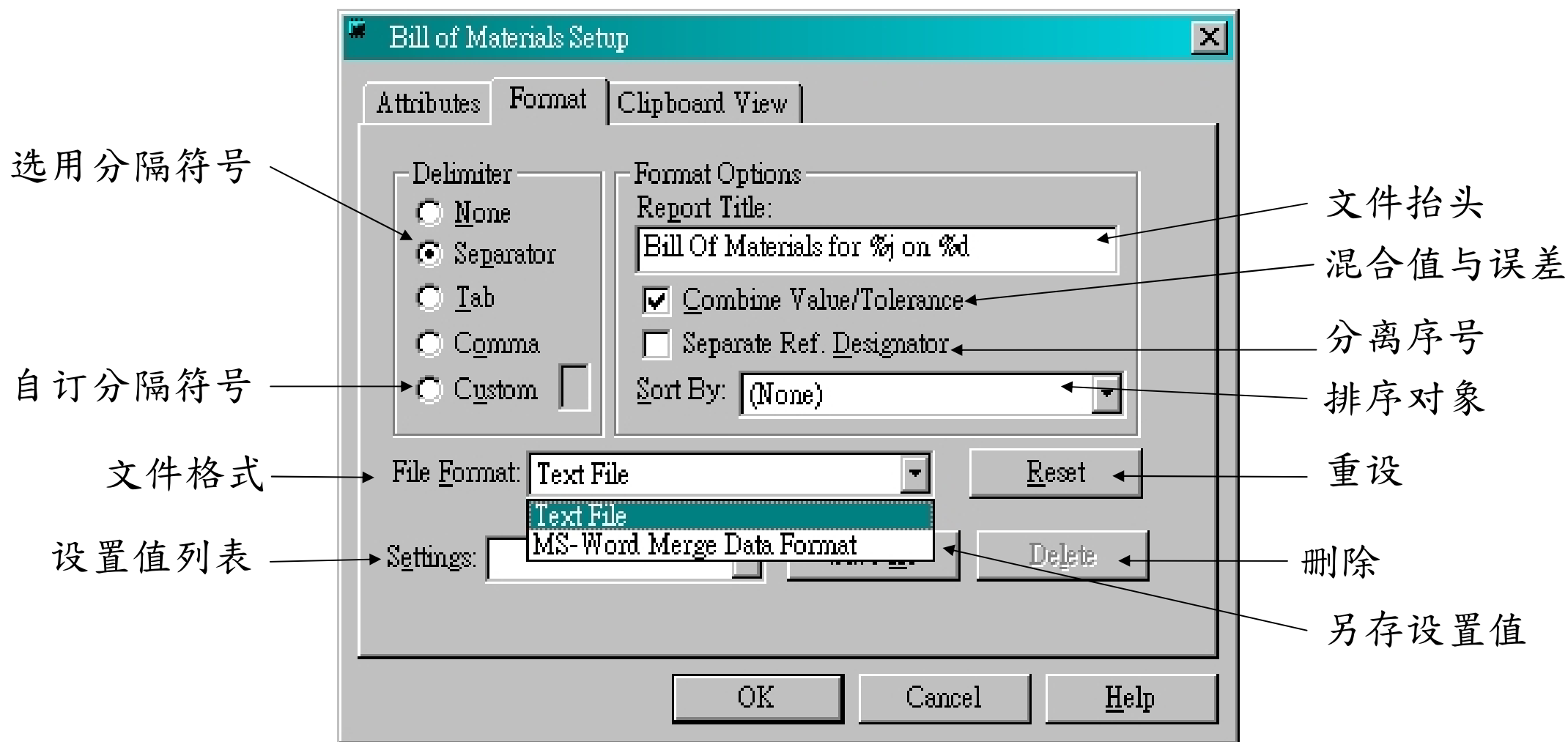
输出报表 1/3

- File > Reports > Setup > Attributes 页面



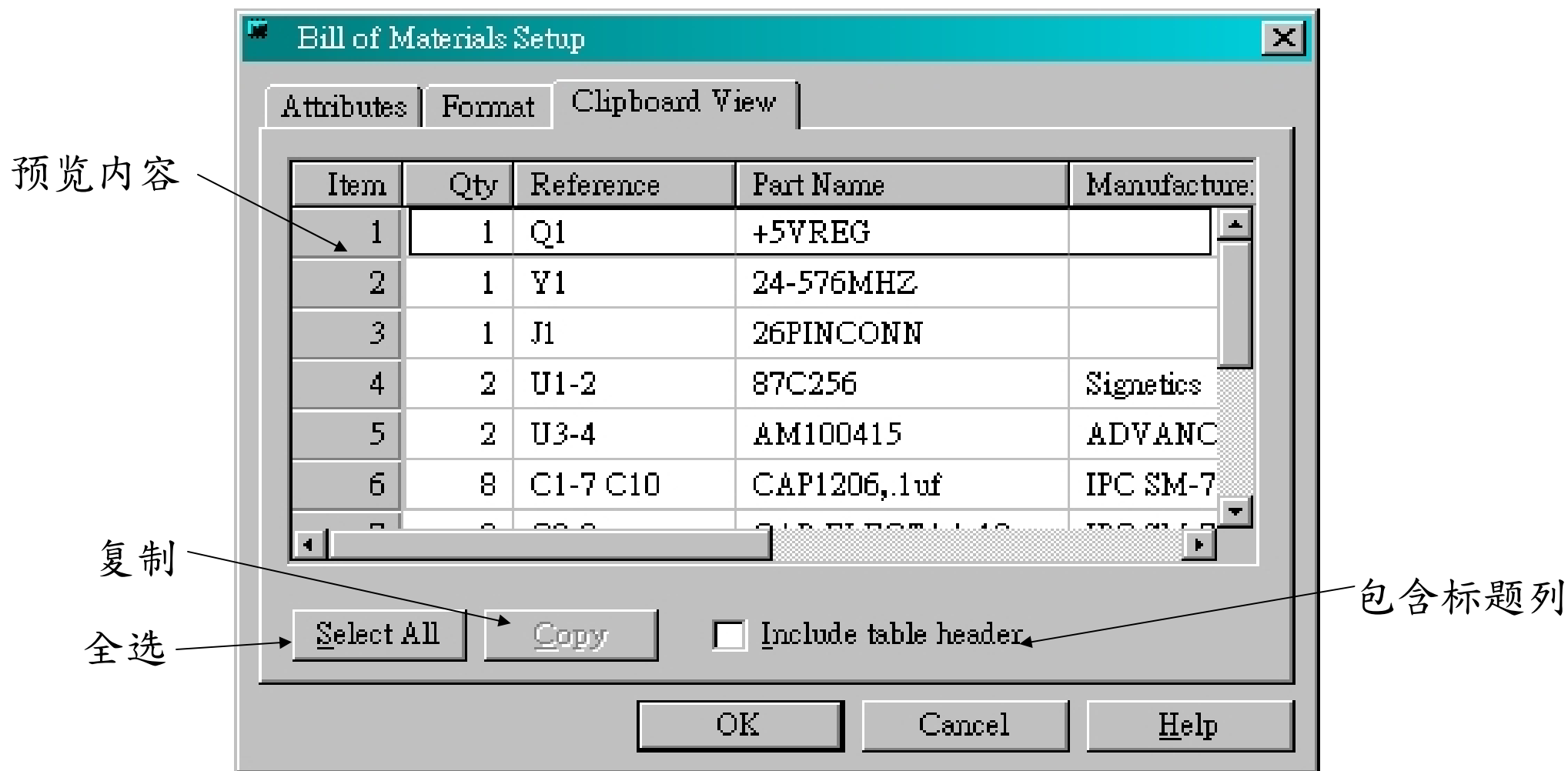
输出报表 2/3

- File > Reports > Setup > Format 页面



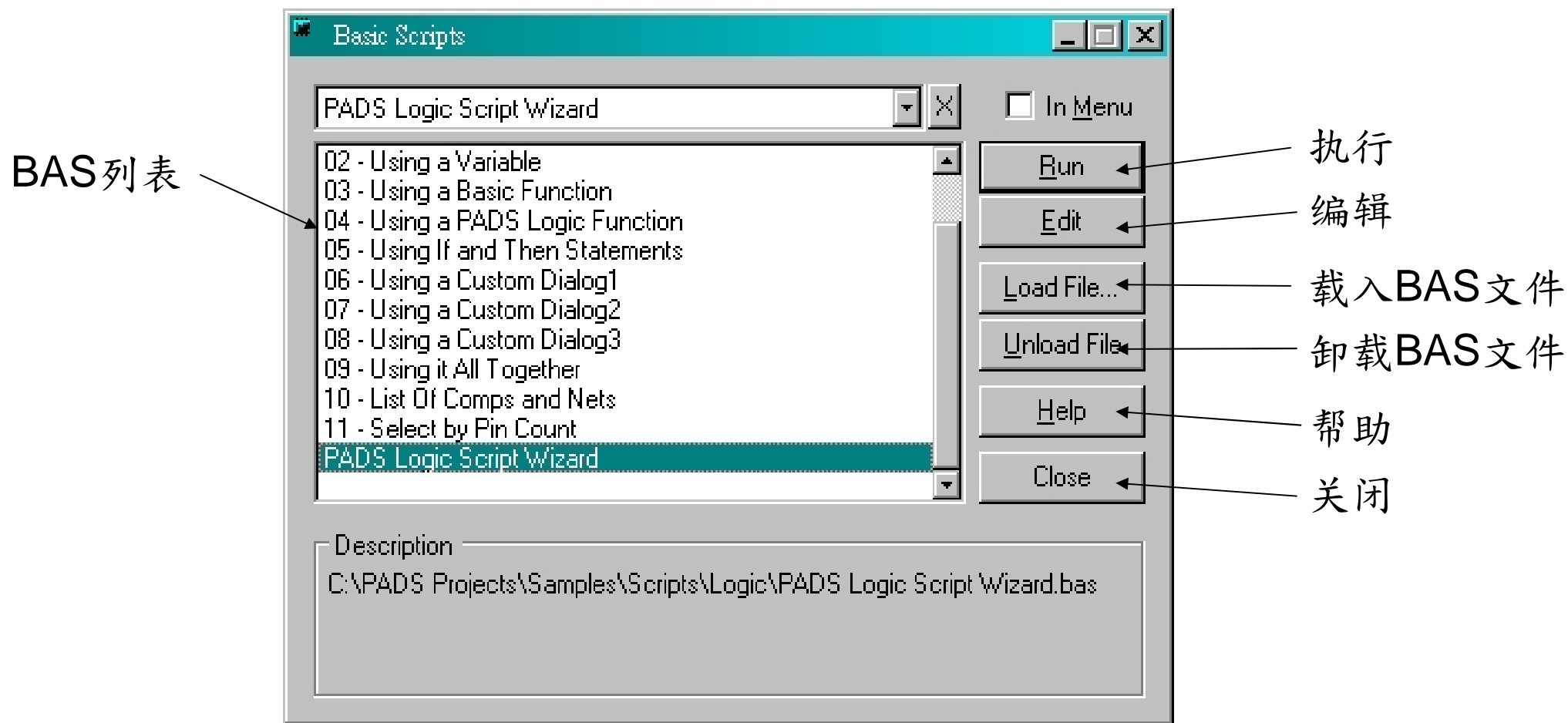
输出报表 3/3

- File>Reports>Setup>Clipboard View 页面



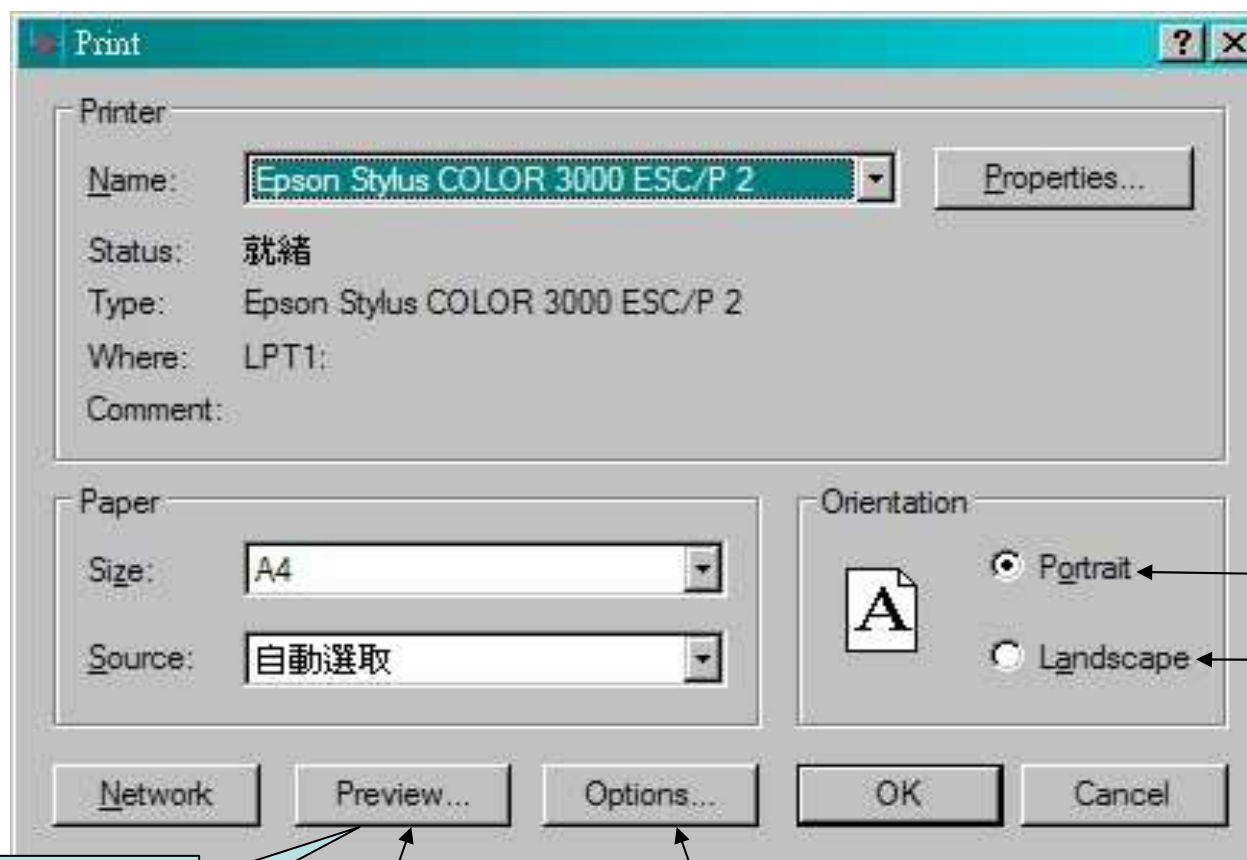
辅助程序与范例

- Tools > Basic Scripts > Basic Scripts...



打印 1/3

- File > Print



同 File > Print Preview

预览打印

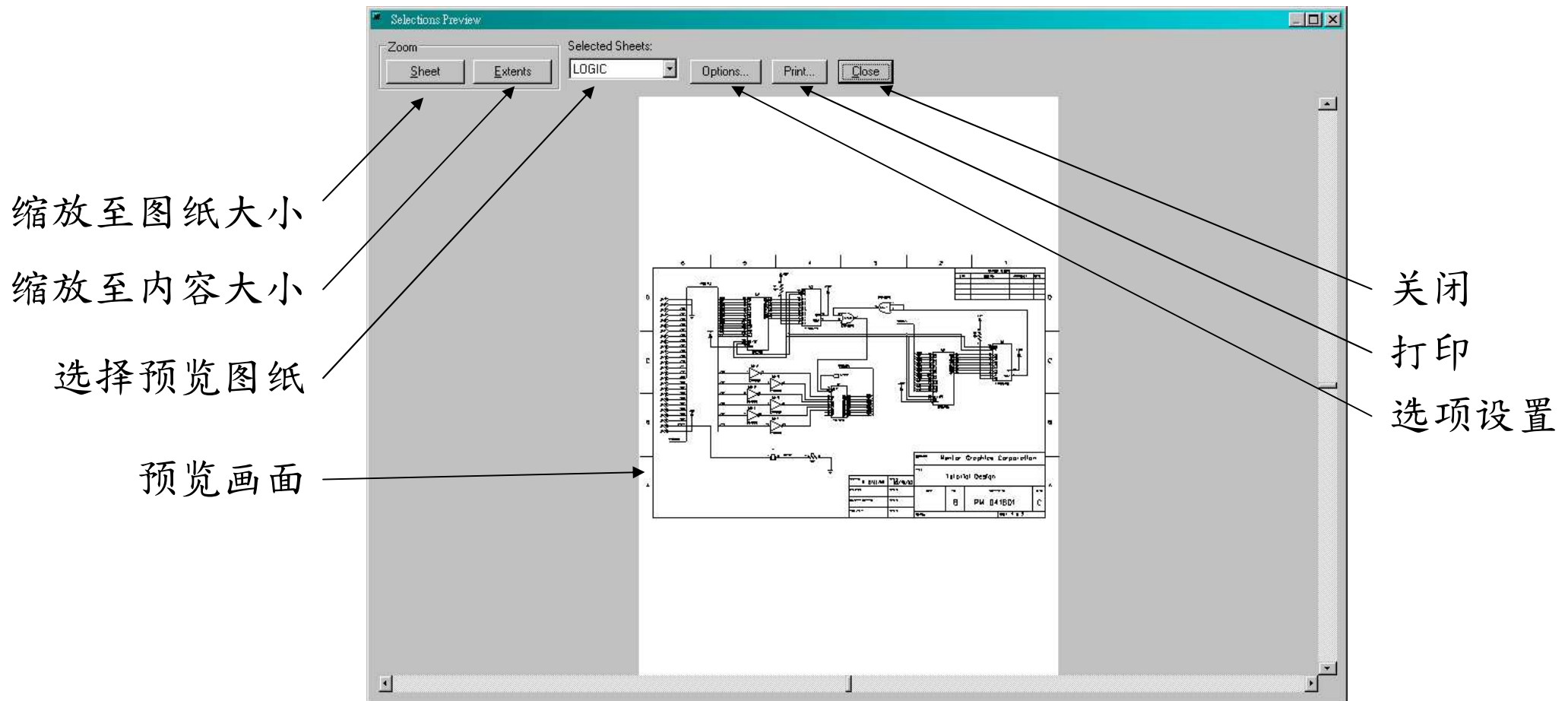
选项设置

直向

横向

打印 2/3

- File > Print > Preview
- File > Print Preview



打印 3/3

- File > Print > Options
- File > Print Preview > Options

选用线路图

加入选取线路图

加入所有线路图

可用线路图

移除选取线路图

移除所有线路图

颜色设置

可用色块

角度

图纸校正方式

X轴偏移量

Y轴偏移量

图纸比例

内容比例

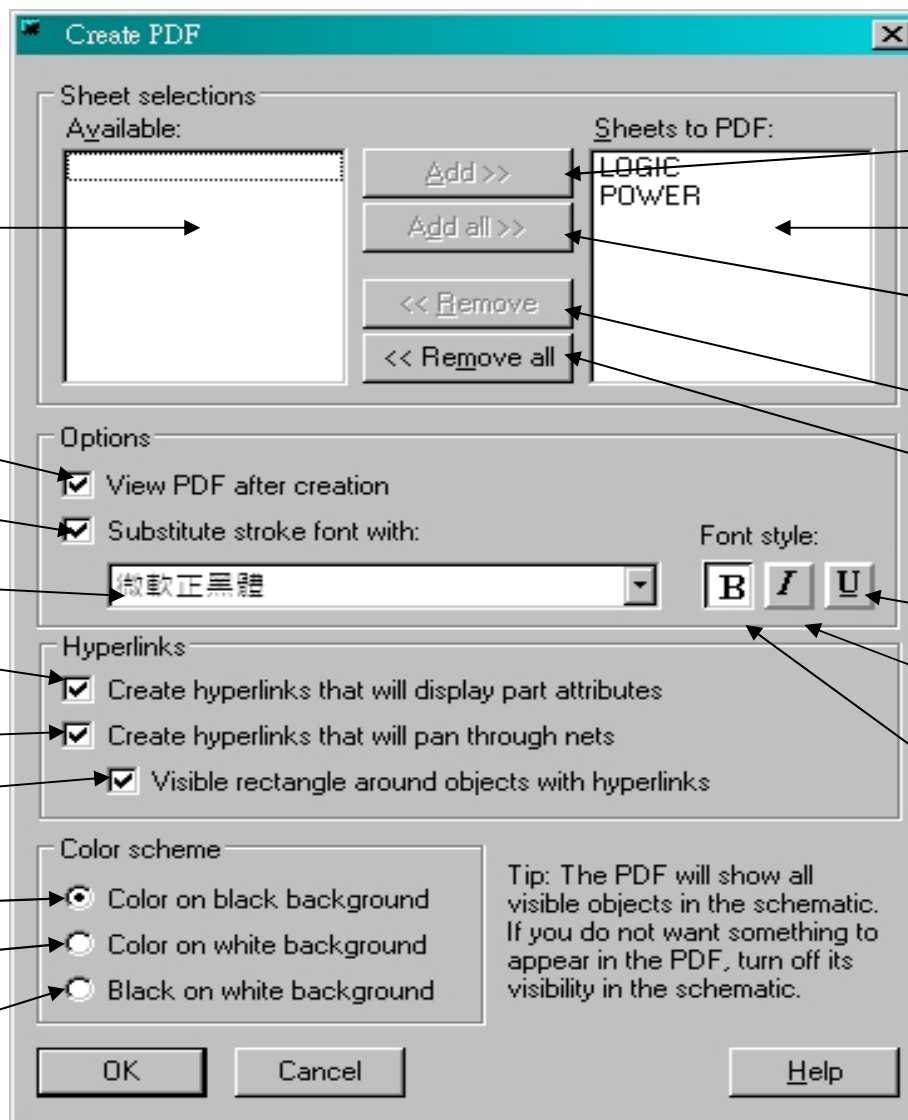
预览大小比例

打印文件名称

打印图纸外框

建立PDF文件

- File > Create PDF



不輸出至PDF的线路图

加入选取线路图

輸出至PDF的线路图

加入所有线路图

移除选取线路图

移除所有线路图

建立后查看PDF

设置字型

选用字型

建立零件属性

建立Net属性

建立物件范围方框

加底线

斜体

粗体

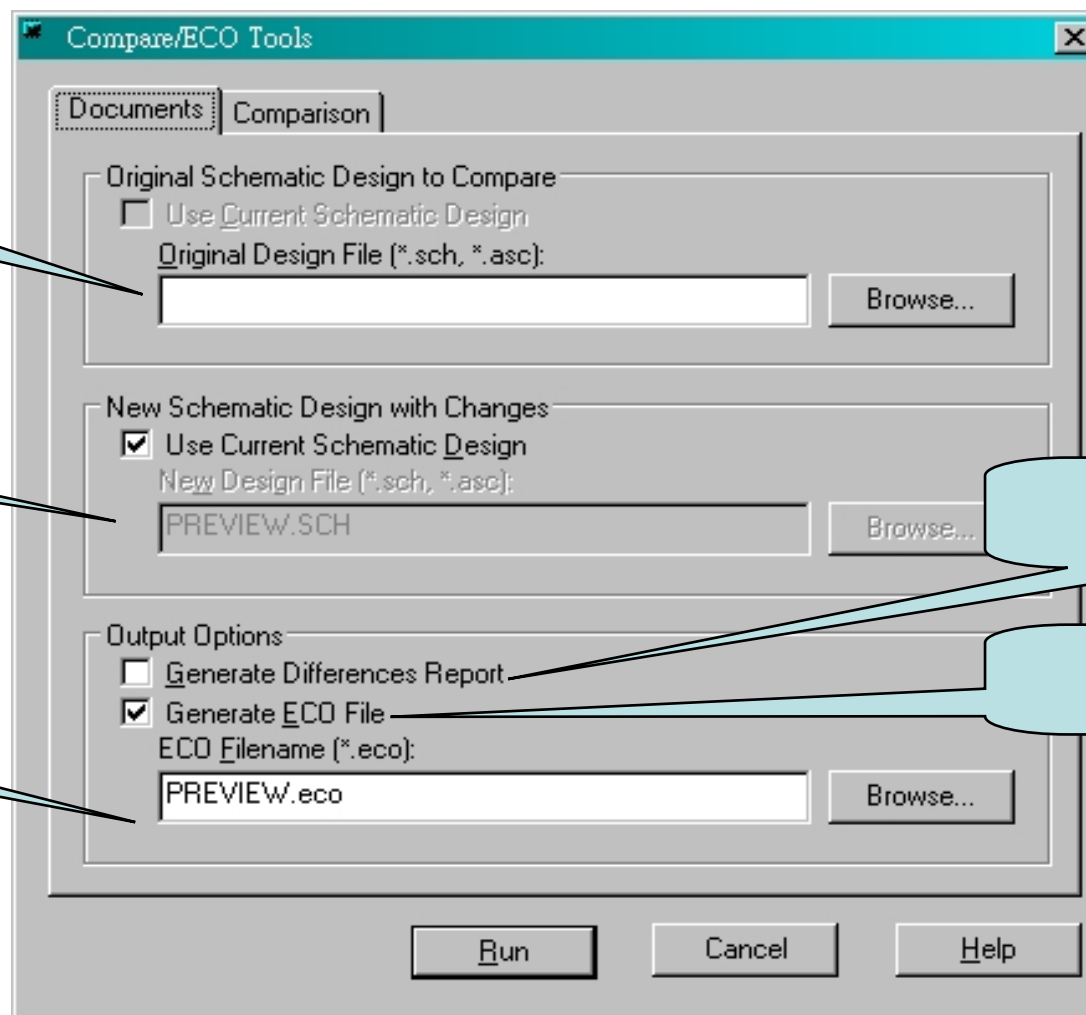
彩色黑底

彩色白底

黑白黑底

ECO比较 1/2

- Tools > Compare/ECO > Documents 页面



旧的版本

新的版本

ECO档名

产生差异报表

产生ECO档

ECO比较 2/2

- Tools > Compare/ECO > Comparison 页面

The screenshot shows the 'Compare/ECO Tools' dialog box with the 'Comparison' tab selected. The dialog is divided into several sections with various options and checkboxes. Chinese annotations with arrows point to specific features:

- 比较Part属性** (Compare Part Attributes): Points to the checked 'Compare Part Attributes' checkbox.
- 比较Net属性** (Compare Net Attributes): Points to the unchecked 'Compare Net Attributes' checkbox.
- 比较Decal属性** (Compare Decal Attributes): Points to the unchecked 'Compare PCB Decal Assignments' checkbox.
- 比较规则** (Compare Design Rules): Points to the checked 'Compare Design Rules' checkbox.
- 比较名称，以重新命名为主** (Compare Names, Rename as Necessary): Points to the selected radio button option.
- 比较名称，以更换物件为主** (Compare Names, Prefer to Add or Delete Parts): Points to the unselected radio button option.
- 比较结构，以重新命名为主** (Compare Connectivity and Topology): Points to the unselected radio button option.
- 忽略空Pin的Net** (Ignore the Unused Pins Net): Points to the checked checkbox.
- 指定忽略的Net** (Specify Net Name): Points to the text input field containing 'NOT_CONNECTED'.

Buttons at the bottom include 'Run', 'Cancel', and 'Help'.

练习六

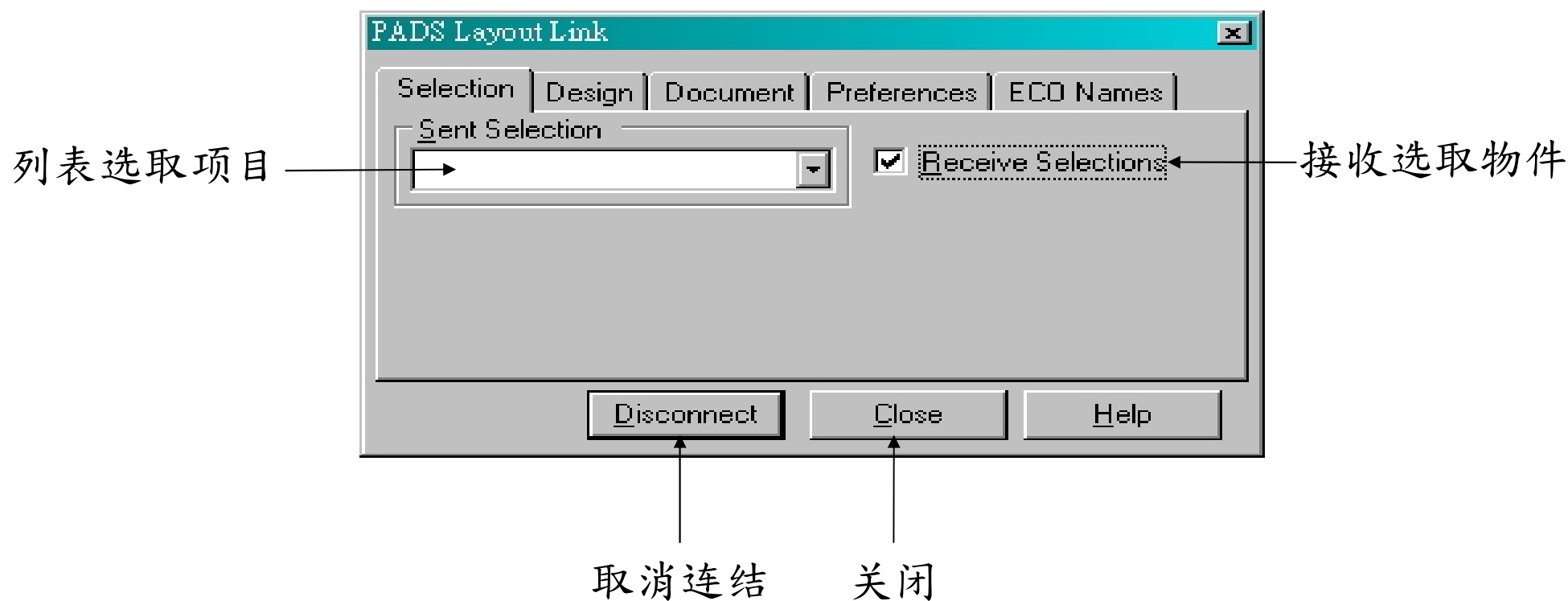
- 输出连线关系 (Netlist) 文件
- 输出报表
- 辅助程序与范例
- 打印
- 建立PDF文件
- ECO比较

第八课

连结器

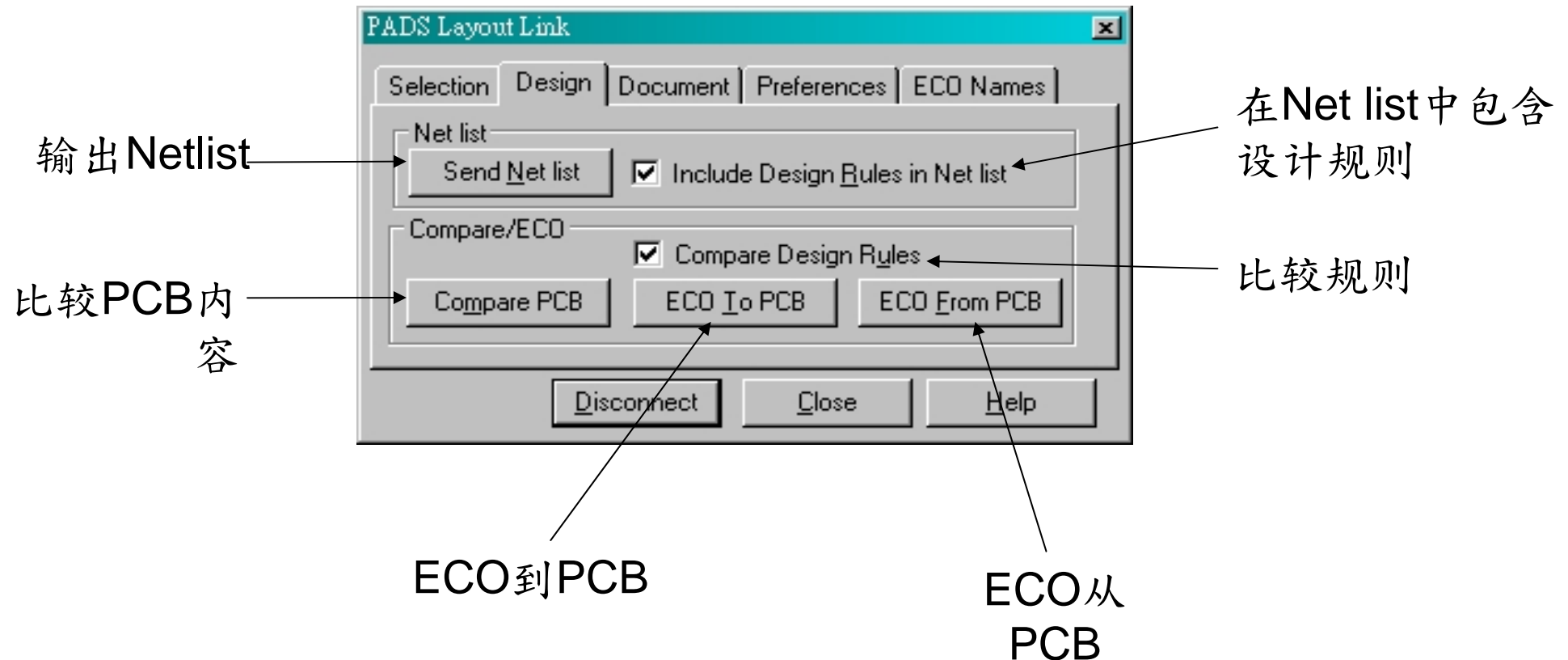
PADS Layout 连结器 1/5

- Tools > PADS Layout > Selection 页面



PADS Layout 连接器 2/5

- Tools > PADS Layout > Design 页面



PADS Layout 连结器 3/5

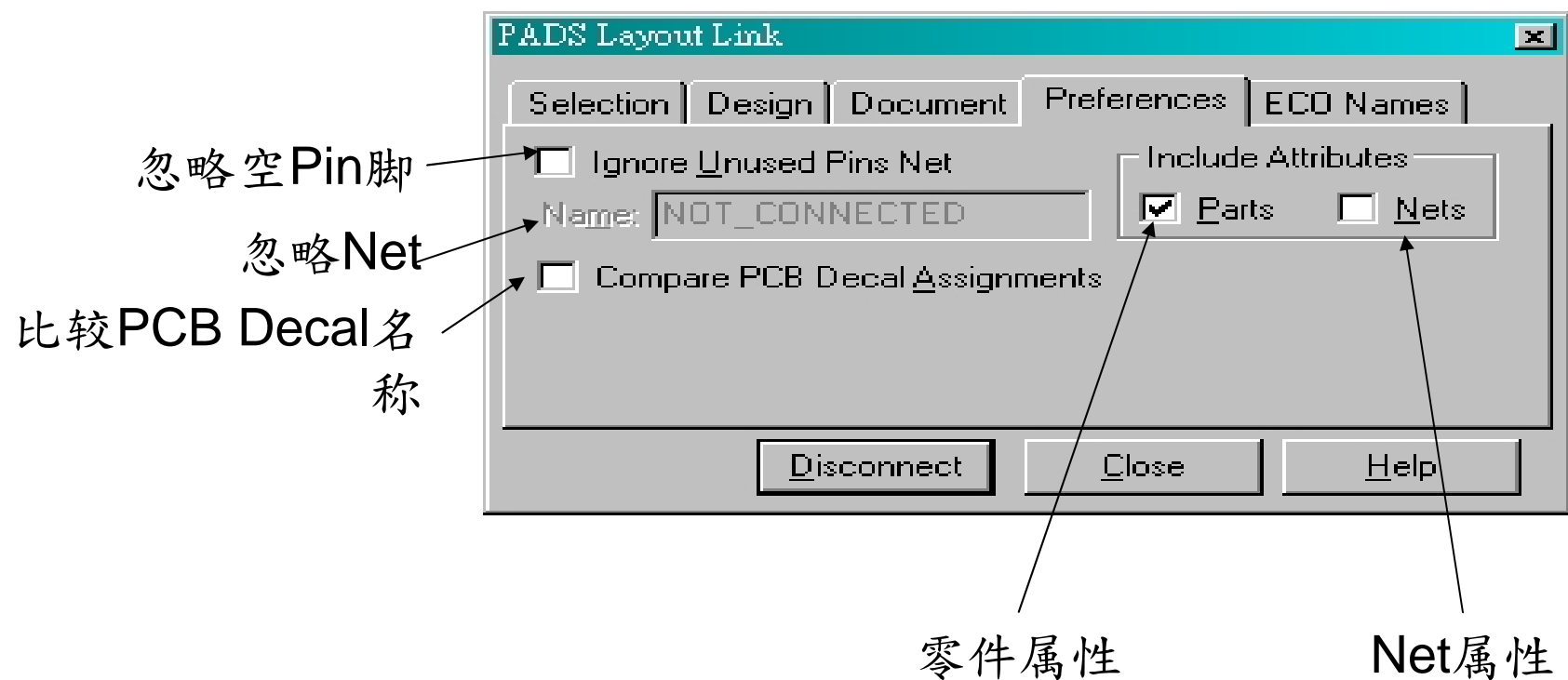
- Tools > PADS Layout > Document 页面



连结Layout文件名称

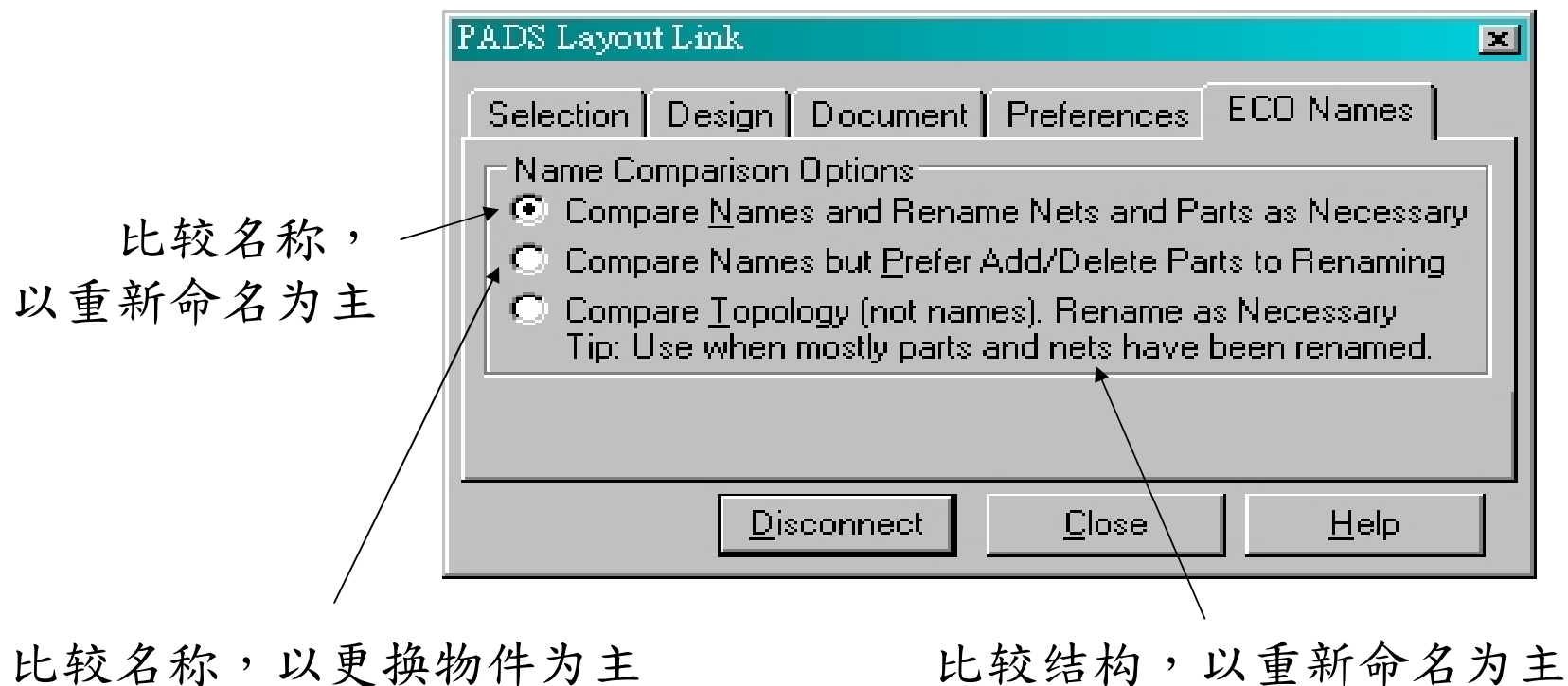
PADS Layout 连接器 4/5

- Tools > PADS Layout > Preferences 页面



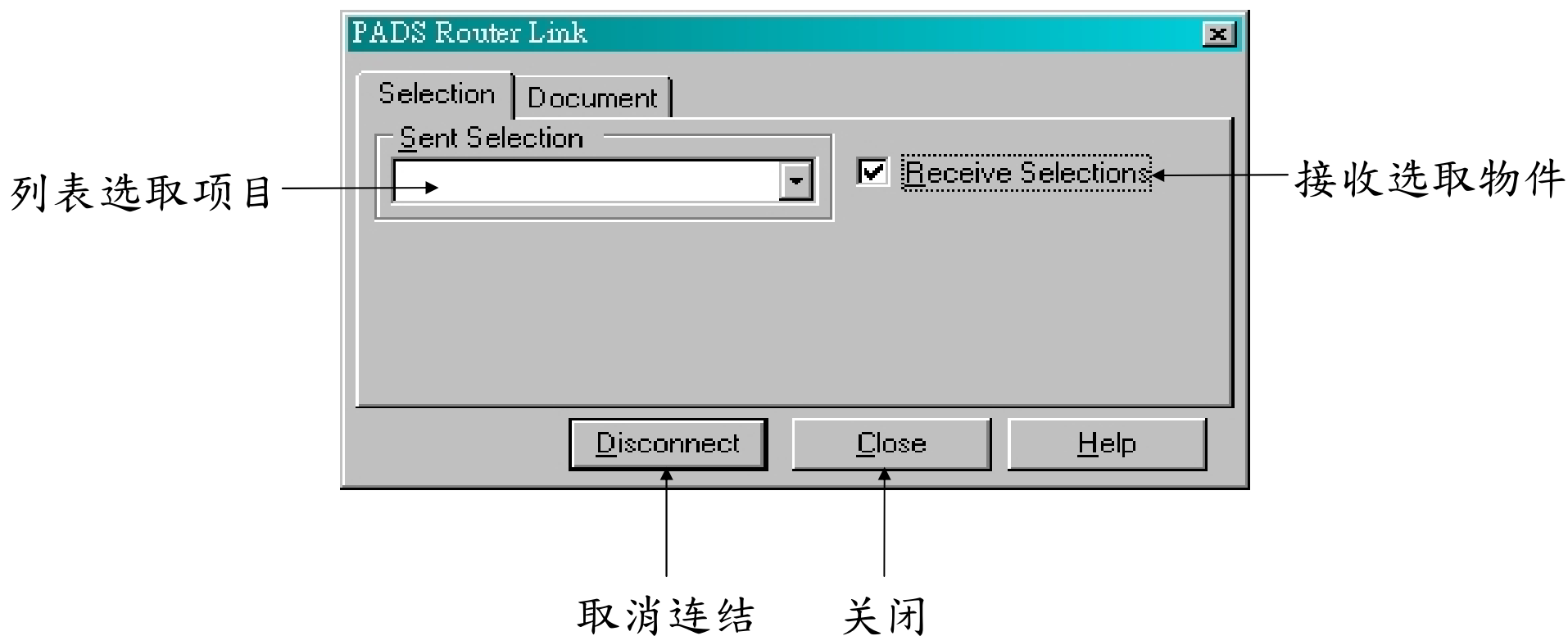
PADS Layout 连接器 5/5

- Tools > PADS Layout > ECO Names 页面



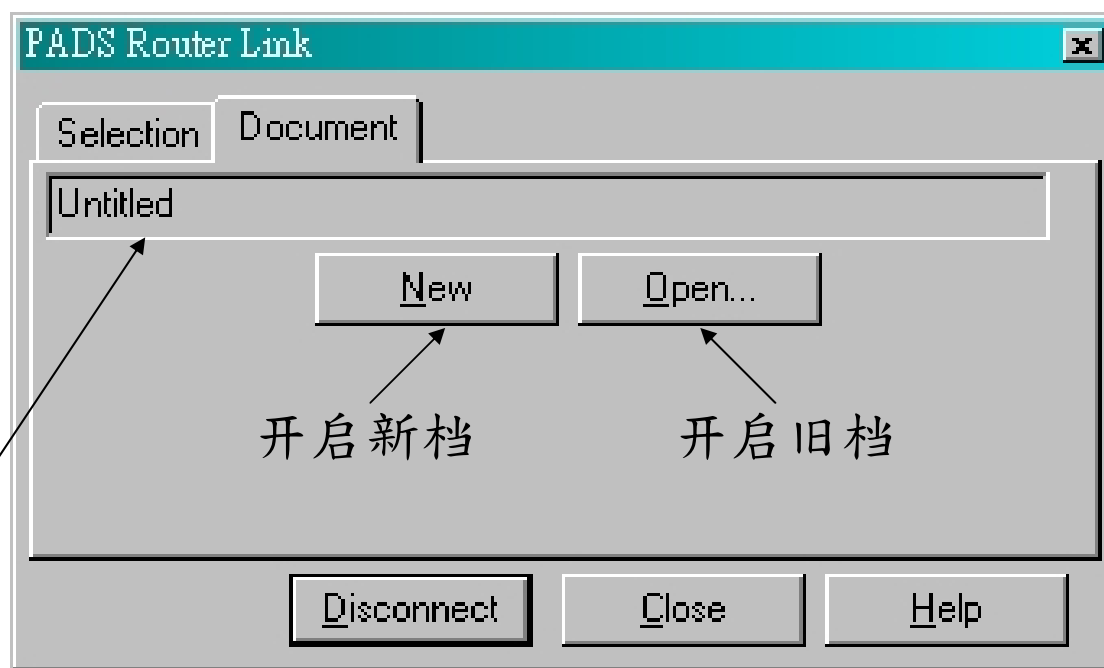
PADS Router 连结器 1/2

- Tools > PADS Router > Selection 页面



PADS Router 连接器 2/2

- Tools > PADS Router > Document 页面



连结Router文件名称

练习七

- PADS Layout 连接器
- PADS Router 连接器

课程结束

苏州泓泰软件科技有限公司

TEL:0512-82285118

FAX:0512-82285119

MAIL:tech@mosteceda.com.cn