

Protel98 - Kurs

Part I: Schematics



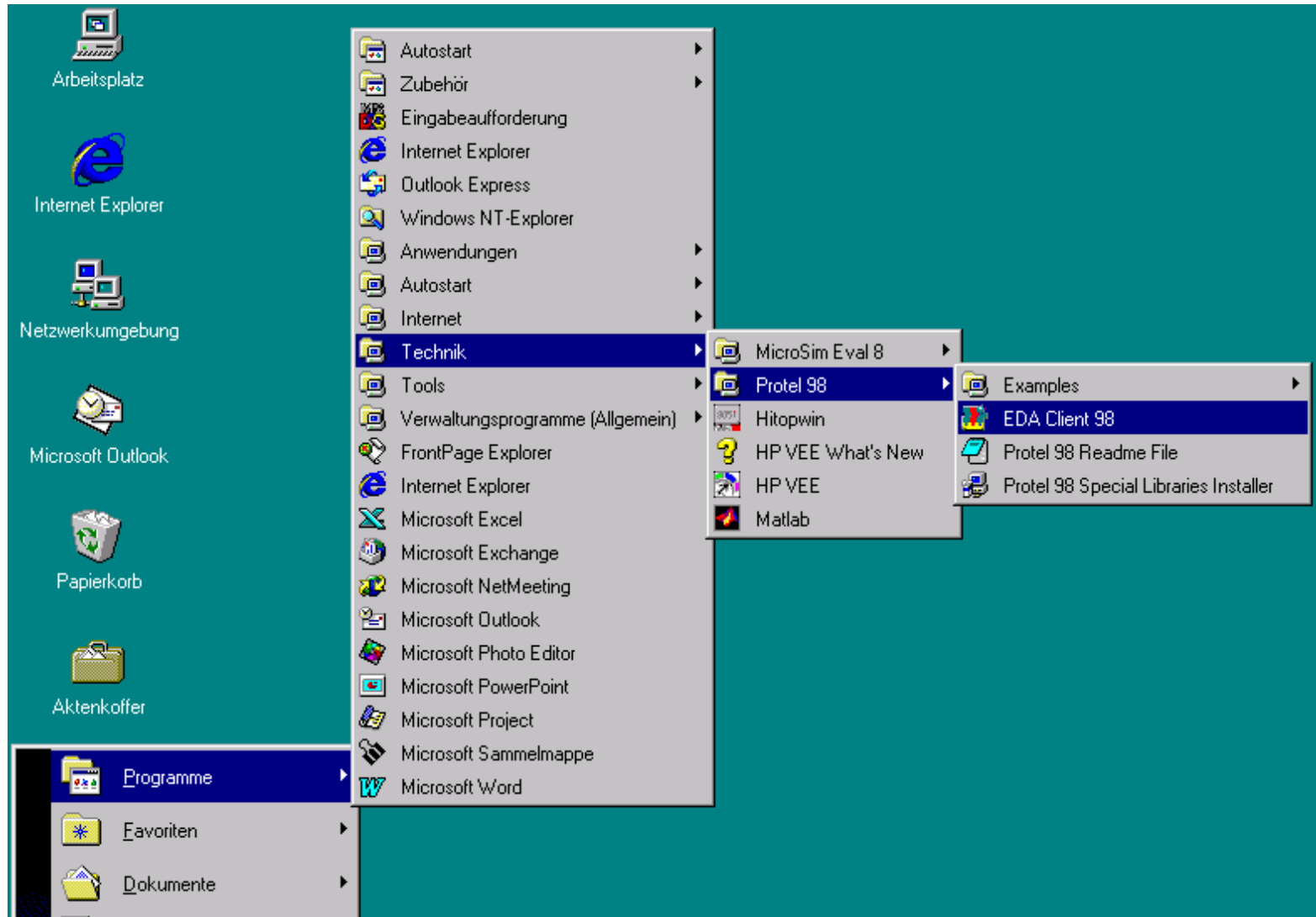
Thomas Kast
F8 Fb.Mechatronik
Email: t.kast@gmx.net

Stand: Oktober 1999

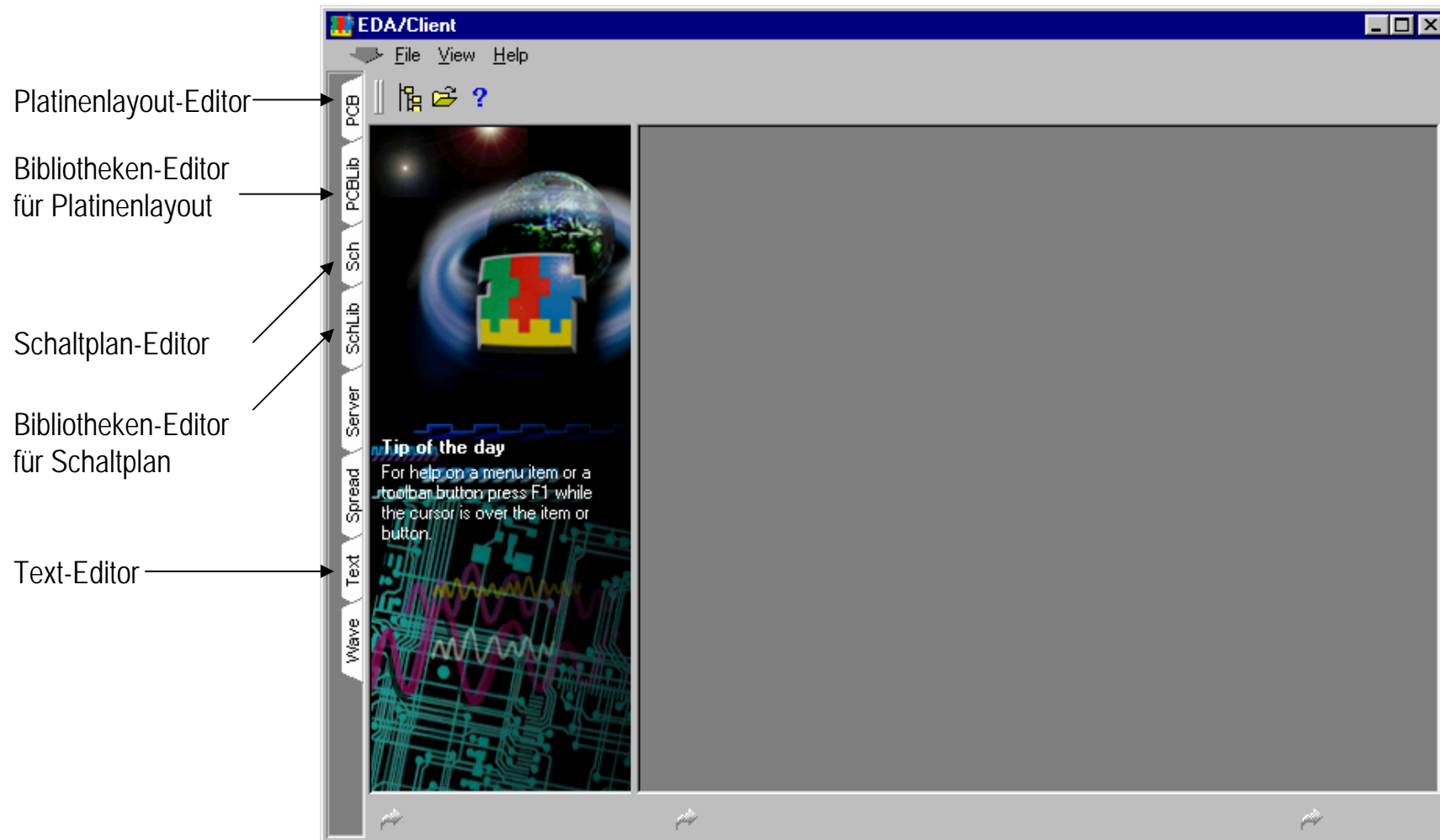
Ziel

- ⌘ Erstellung eines Schaltplans in Schematics zur Weiterverarbeitung in PCB (Platinenlayout)

Programmverzeichnis

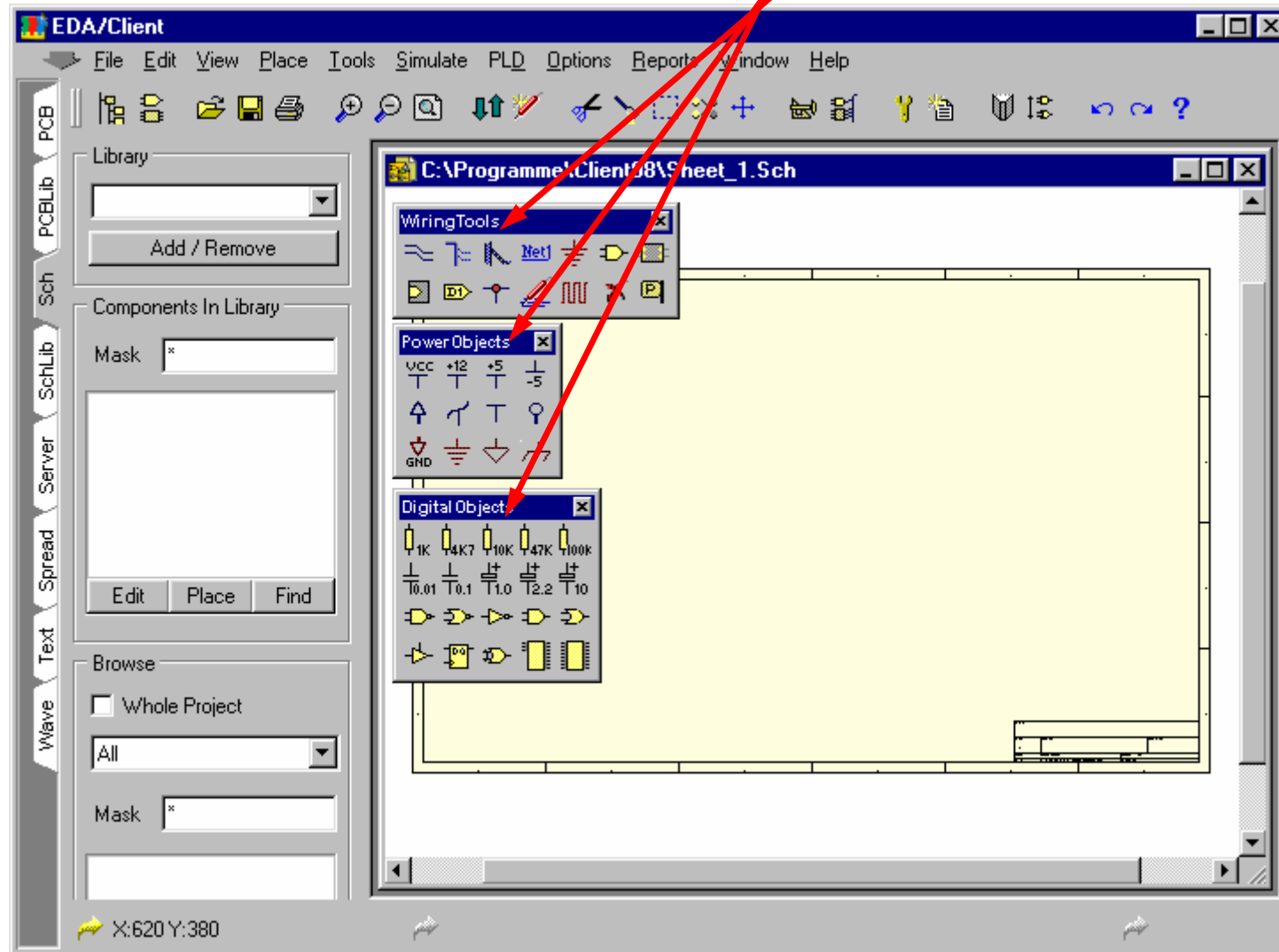


Die Oberfläche

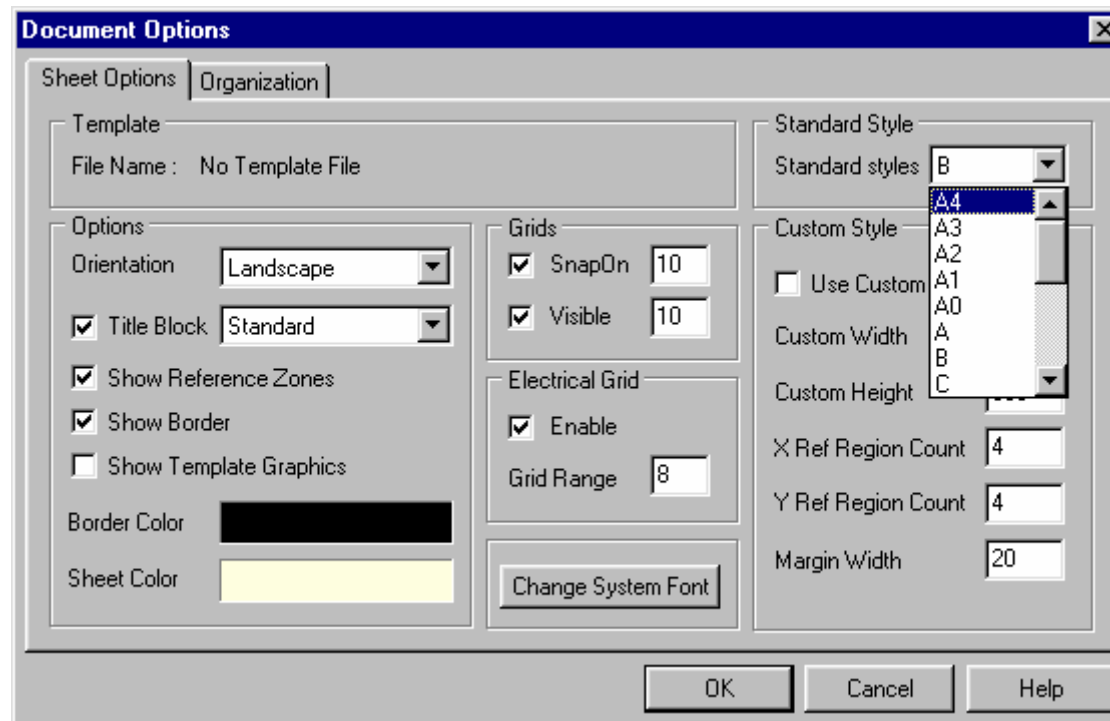


Der Schaltplan-Editor

Zeichenwerkzeug



Arbeitsfläche einrichten (Bild- und Rastergröße)



Nützliche Hotkeys

- **Ansicht**
 - Zoomen mit Bild↑ und Bild↓ (mit Mauszeiger an gewünschte Stelle)
 - Neuzeichnen mit Ende
- **Bauteil und Beschriftungen**
 - 90° drehen mit Leertaste
 - Verschieben - mit linker Maustaste anklicken und gedrückt halten
 - Bearbeiten - Doppelklick mit linker Maustaste
- Zum **Deaktivieren** einer Funktion (Bauteil hängt an Mauszeiger, etc.) rechte Maustaste drücken
- **Liste aller Hotkeys** mit F1 aufrufen, während Wire-Funktion aktiv ist

Protel98 - Kurs

Part I: Schematics

Bauteile finden
und plazieren

Library einbinden

Bauteil finden

The screenshot shows the Protel 98 SE interface with the 'Find Schematic Component' dialog box open. The dialog box has the following settings:

- Find Component:
 - By Library Reference: *RES*
 - By Description
- Search:
 - Scope: Specified Path
 - Sub directories
 - Find All Instances
 - Path: C:\programme\client98
 - File: *.lib
- Found Libraries:
 - C:\programme\client98\SCH98\Library\DEVICE.LIB
 - C:\programme\client98\SCH98\Library\SYMBOLS.LIB
 - C:\programme\client98\SCH98\Library\DosOld\DEVICE.LI
 - C:\programme\client98\SCH98\Library\HP_EESof\HP-EESOF
 - C:\programme\client98\SCH98\Library\Linear\LINEAR.LIB
 - C:\programme\client98\SCH98\Library\Spice\SPICE.LIB
 - C:\programme\client98\SCH98\Library\Spice\SPICE.LIB
 - C:\programme\client98\SCH98\Library\Western Digital\WD_WF
- Components:
 - RES1
 - RES2
 - RES3
 - RES4
 - RESPACK1
 - RESPACK2
 - RESPACK3
 - RESPACK4

The 'Components In Library' list in the background shows:

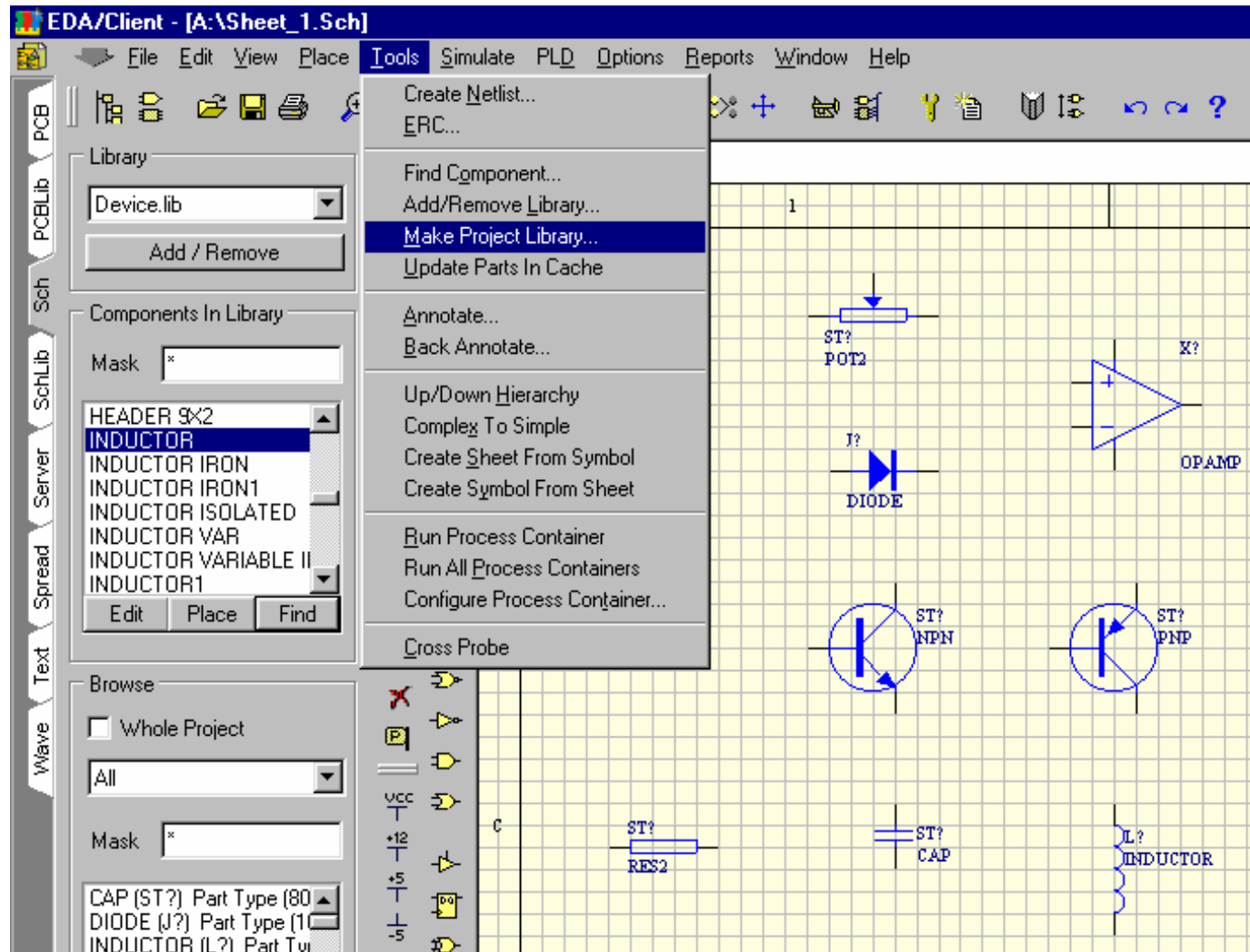
- RELAY-DPDT
- RELAY-DPST
- RELAY-SPDT
- RELAY-SPST
- RES1
- RES2
- RES3
- RES4

The 'Browse' section shows:

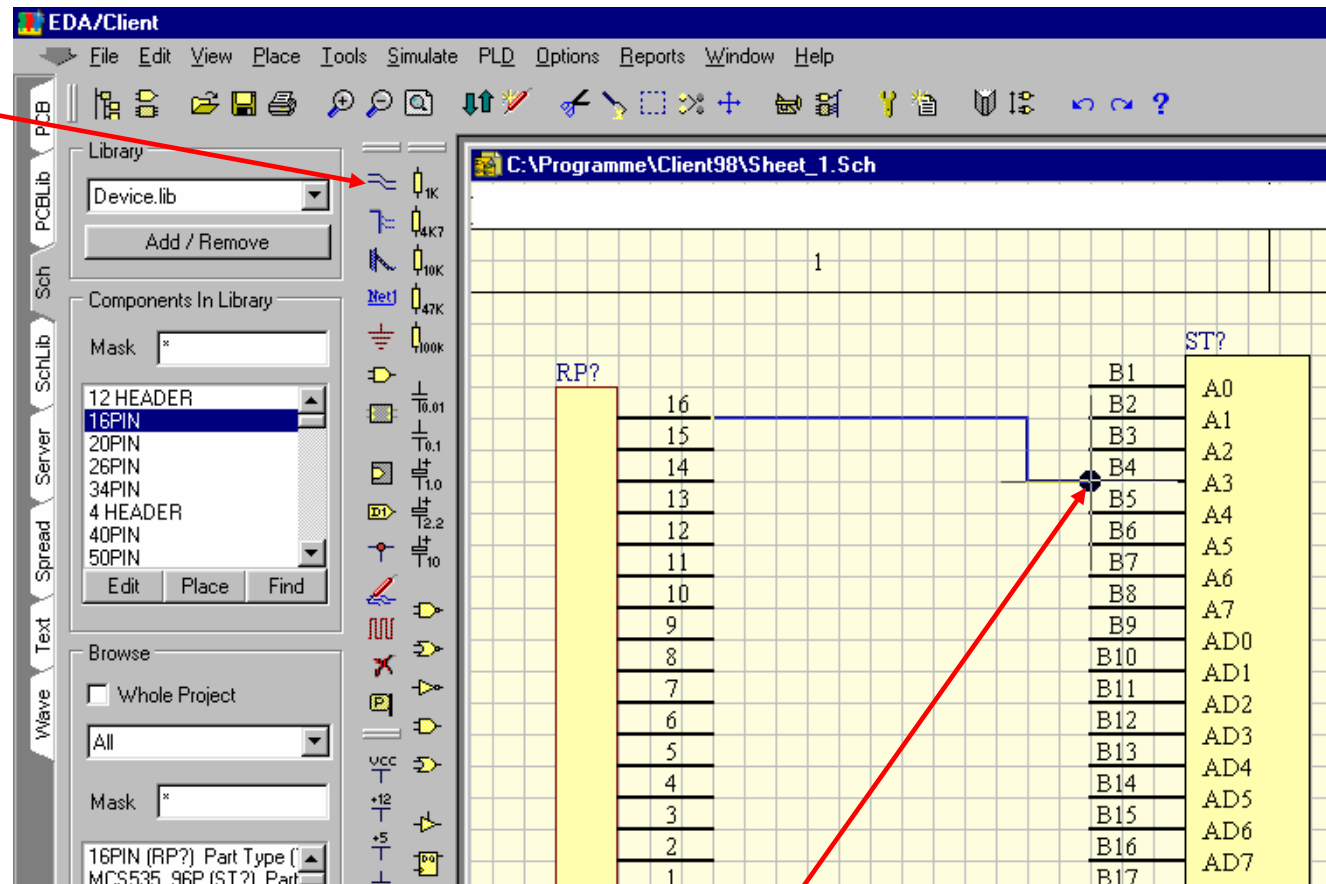
- Whole Project
- All
- Mask: *
- A? Part-Type: RES2 Part I
- A?-1 Passive (330,270) P
- A?-2 Passive (380,270) P
- Part Description (339,106)
- Part Description (339,116)
- Part Description (339,126)
- Part Description (339,136)
- Part Description (339,146)
- Partial Information
- Text Jump Edit
- Update list

In the bottom right corner, a schematic symbol for a resistor is shown with the label 'A?' above it and 'RES2' below it.

Erstellen der Projektlibrary



„Verdrahten“
- einzelne
Leitungen



Punkt erscheint, wenn
Pinanschluß erreicht ist

Protel98 - Kurs

Part I: Schematics

Beschriftung
einzelner Bauteile

Bezeichnung

Wert

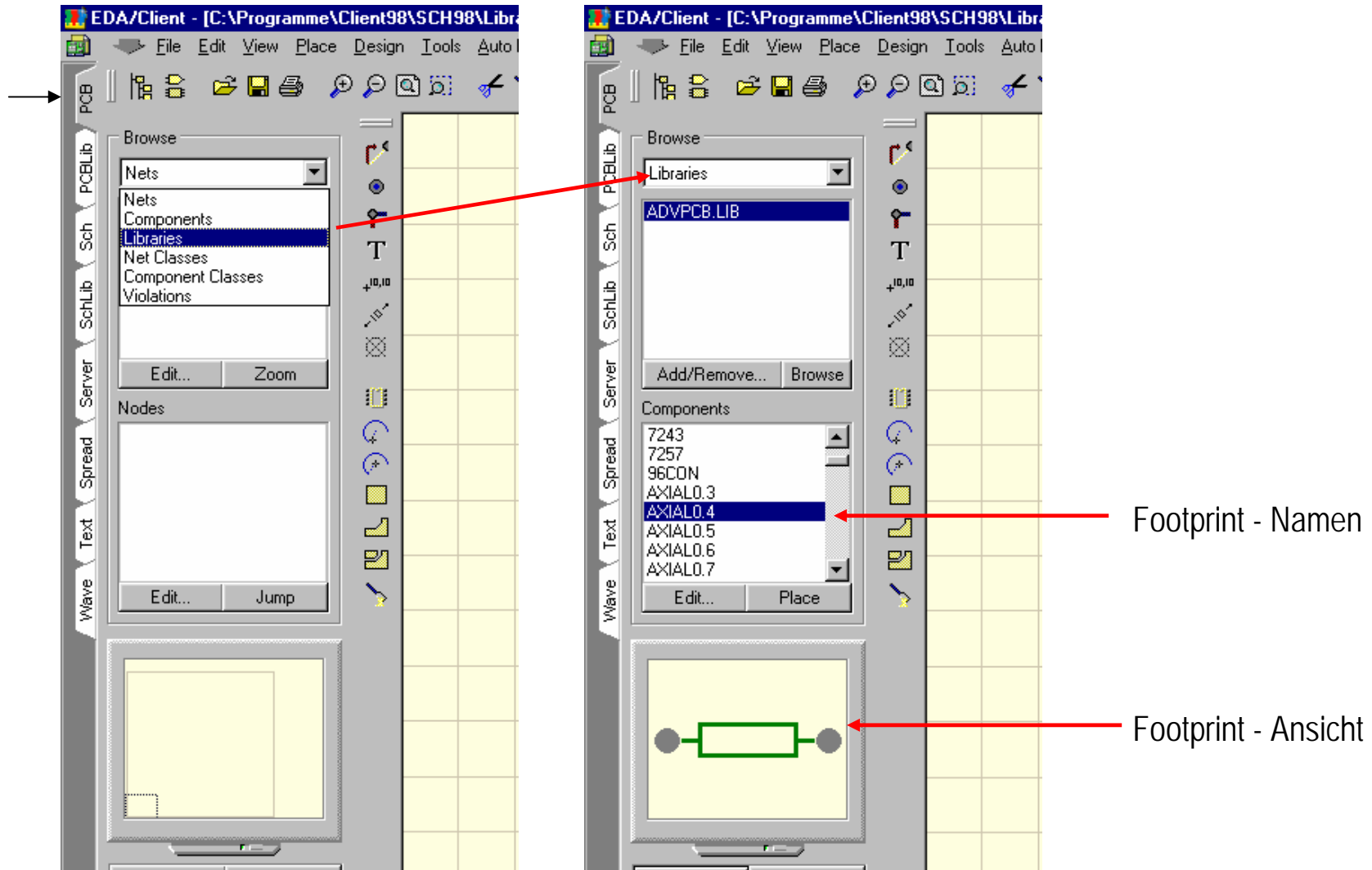
The screenshot shows the Protel 98 SE Schematic Editor interface. The main window displays a schematic diagram with two resistors labeled 'RES2' and a capacitor labeled 'CAP'. The capacitor is connected between a node and ground. A 'Part' dialog box is open, showing the configuration for the selected capacitor. The dialog box has three tabs: 'Attributes', 'Graphical Attrs', and 'Part Fields 1-8'. The 'Attributes' tab is active, showing the following fields:

Field	Value
Lib Ref	CAP
Footprint	
Designator	A?
Part Type	CAP
Sheet Path	*
Part	1
Selection	<input type="checkbox"/>
Hidden Pins	<input type="checkbox"/>
Hidden Fields	<input type="checkbox"/>
Field Names	<input type="checkbox"/>

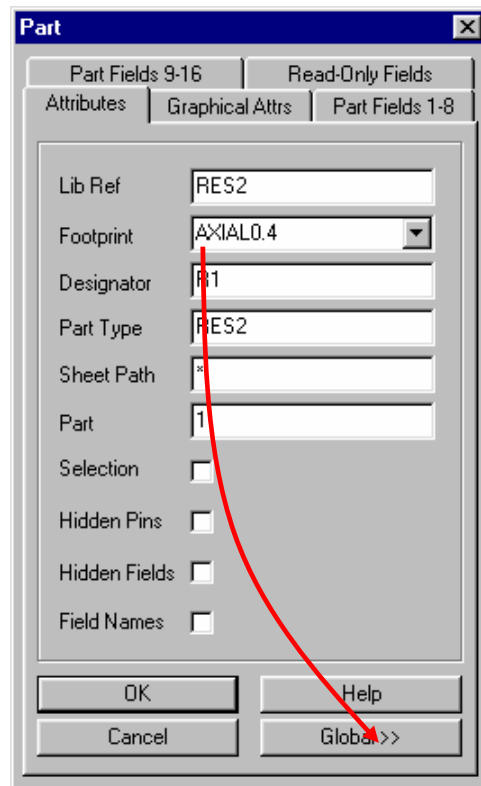
Below the dialog box, a list of components is visible in the 'Browse' area, including:

- (340,560)(390,560) Wire
- (360,560)(360,540) Wire
- 100nF (C3) Part Type (5E
- A? Part-Type:CAP Part (3
- A? Part-Type:RES2 Part (
- A? Part-Type:RES2 Part (
- A?-1 Passive (290,560) P
- A?-1 Passive (360,540) P

Auswählen des geeigneten Footprints

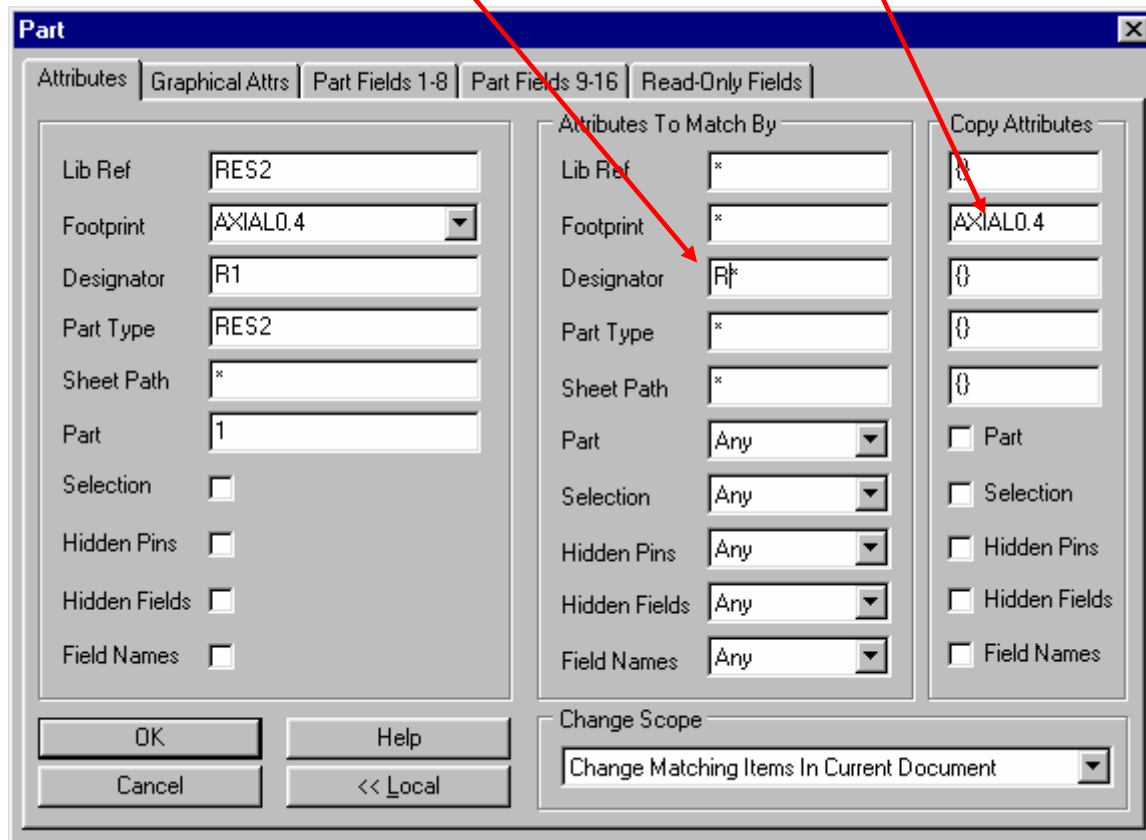


Verknüpfen von Bauteilen und Footprints



Zu verändernde
Bauteile

zu kopierendes
Merkmal



Protel98 - Kurs

Part I: Schematics

Erstellen der Netzliste
mit Footprints
(Basis für Platinenlayout)

The screenshot illustrates the workflow in Protel 98 SE for generating a netlist from a schematic. The main window, titled 'EDA/Client - [C:\Programme\Client98\Sheet_1.Sch]', shows a schematic with components R1 and RES2. The 'Tools' menu is open, and 'Create Netlist...' is selected, with a red arrow pointing to the netlist window.

The netlist window, titled 'EDA/Client - [C:\Programme\Client98\Sheet_1.NET]', displays the following netlist content:

```
[  
C1  
RAD0.3  
100nF  
  
]  
C2  
RAD0.3  
1uF  
  
]  
C3  
RAD0.3  
100nF  
  
]  
R1  
AXIAL0.4  
RES2
```

The 'Error Report' window, titled 'EDA/Client - [C:\Programme\Client98\Sheet_1.ERC]', shows an empty report with the text 'Error Report For' and 'End Report'.

The component list window, titled 'EDA/Client - [C:\Programme\Client98\Sheet_1.Sch]', shows the following components:

- 40,560)(390,560) Wire
- 60,560)(360,540) Wire
- 10nF (C3) Part Type (5F
- ? Part-Type:CAP Part (3
- ? Part-Type:RES2 Part (
- ? Part-Type:RES2 Part (
- ?-1 Passive (290,560) P
- ?-1 Passive (360,540) P

Standard-Bauteile

☒ Widerstand	RES2	Device.lib
☒ Potentiometer	POT2	"
☒ Diode	DIODE	"
☒ Kondensator	Cap	"
☒ Spule	INDUCTOR	"
☒ OP	OPAMP	"
☒ NPN-Transistor	NPN	"
☒ PNP-Transistor	PNP	"
☒ 96-polige Steckerleiste	MCS535_96P	Euro_535.lib

Standard-Footprints

☒ 1 mil = 0.0254 mm

☒ 1 mm = 39.37 mil

☒ Widerstand -> AXIAL0.4

☒ Diode -> DIODE0.4

☒ Kondensator -> RAD0.*

☒ IC's -> DIP8, DIP14....

☒ Steckerleiste -> Euro535

Standard-Bohrlochgrößen

- ☒ Widerstände, Dioden
- ☒ DIP8, DIP16, etc
- ☒ 96-polige Steckerleiste

0.889 mm = 35 mil