





---

# Altium Designer Guide

Tutorial v1.0

---

Mateusz Kłosiński  
Identification Number 268081  
`268081@student.pwr.edu.pl`

Faculty of Electronics, Photonics and Microsystems  
Wrocław University of Science and Technology  
Academic Year 20XX-20XX

# Chapter 1

## Introduction

In the rapidly evolving world of electronics, the demand for efficient, reliable, and innovative printed circuit board (PCB) designs is at an all-time high. Engineers and designers are constantly seeking tools that not only keep up with technological advancements but also simplify the complexities involved in PCB design. Altium Designer stands out as a premier solution, seamlessly integrating various aspects of PCB engineering into one cohesive platform.

Altium Designer is more than just a design tool; it's a comprehensive suite that offers an array of features tailored to streamline the PCB design process from conception to production. This powerful software is renowned for its user-friendly interface, robust design capabilities, and extensive library of components, making it an indispensable tool for both novice and experienced engineers alike.

One of the key strengths of Altium Designer is its ability to unify different facets of PCB design. It provides a single environment where schematic capture, PCB layout, and component management are seamlessly integrated. This unification eliminates the need for multiple software packages, reducing the risk of errors and significantly speeding up the design process. Moreover, Altium Designer's advanced simulation and verification tools ensure that designs are not only accurate but also optimized for performance and manufacturability.

Additionally, Altium Designer's collaborative features facilitate teamwork and project management, essential in today's globalized engineering landscape. With real-time collaboration capabilities, design teams can work simultaneously on the same project, ensuring that all members are on the same page and that the design evolves cohesively. This aspect is particularly beneficial for large-scale projects where multiple engineers must contribute their expertise without the bottlenecks of traditional, sequential workflows.

The software's adaptability and scalability are further enhanced by its extensive support for various industry standards and protocols. Whether working on simple consumer electronics or complex aerospace systems, Altium Designer provides the tools necessary to meet stringent industry requirements. The inclusion of advanced routing tools, 3D visualization, and comprehensive design rule checks (DRC) ensures that every design can meet the highest quality standards.

In summary, Altium Designer is not just a tool but a comprehensive solution that empowers engineers to push the boundaries of what's possible in PCB design. Its robust feature set, integrated environment, and collaborative capabilities make it an essential asset for any engineer looking to excel in the field of PCB engineering. As technology continues to advance, tools like Altium Designer will be crucial in shaping the future of electronic design, ensuring that innovation is both achievable and sustainable.

# Chapter 2

## Getting Started

### 2.1 Installation

To begin using Altium Designer, follow these steps to install the software:

1. Download the Altium Designer installer from the official website.
2. Run the installer and follow the on-screen instructions.
3. Choose the desired installation options and specify the installation directory.
4. Complete the installation and launch Altium Designer.

### 2.2 Licensing

After installation, you need to activate your license:

1. Open Altium Designer and go to **Help -> License Management**.
2. Click **Add License** and enter your license details.
3. Follow the prompts to complete the activation process.

### 2.3 User Interface Overview

The Altium Designer interface consists of several key areas:

- **Main Menu:** Located at the top, it provides access to all software functions.
- **Workspace:** The central area where you create and edit your designs.
- **Panels:** Located on the sides, they provide additional tools and information.
- **Toolbar:** Contains shortcuts to frequently used commands.
- **Status Bar:** Displays status information and coordinates.

## Chapter 3

# Project Creation and Management

### 3.1 Creating a New Project

To create a new project in Altium Designer:

1. Go to **File** → **New** → **Project**.
2. Choose the type of project (e.g., PCB Project).
3. Specify the project name and location.
4. Click **Create**.

### 3.2 Managing Project Files

Project files include schematics, PCB layouts, and libraries. To manage these files:

1. Right-click on the project in the **Projects** panel.
2. Select **Add New to Project** to add new files.
3. Use **Add Existing to Project** to add existing files.
4. Organize files by creating folders within the project.

# Chapter 4

## Template

Creating a schematic template is a highly convenient way to start work in Altium Designer. It allows you to set up a standardized layout that can be easily modified to meet your specific preferences and requirements.

### 4.1 Creating a New Schematic Template

To begin creating a template, follow these steps:

1. Right-click on your project in the **Projects** panel and select **Add New to Project → Schematic**.
2. Press **P** to open the **Place** tab in the schematic editor.
3. At the bottom of the **Place** tab, locate the **Drawing Tools** section.
4. Use the **Line** tool to draw a small table at the bottom of your schematic for future data entry.

### 4.2 Adding a Company Logo

To include your company logo in the template:

1. Press **P → Drawing Tools → Import Images**.
2. Select the image file of your company logo that you wish to import.

### 4.3 Specifying Text Data

To add and format text data in the table:

1. Press **P → Text** to add text fields.
2. Use the **TAB** key to switch between text blocks and edit their properties.
3. Define the necessary data for version control and other sheet-specific information.

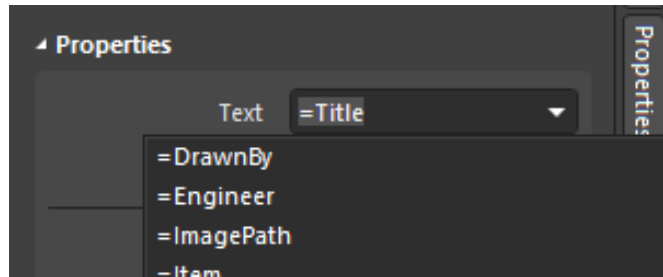


Figure 4.1: Example of the schematic template table with placeholder text.

## 4.4 Setting Up Template Fields

To configure global properties for text fields:

1. Expand the **Properties** panel to view all available options.
2. Define global properties that will automatically populate the text fields in the template.

After setting up the text fields, return to the **Global Sheet Properties** to enter your data strings. Using this template will automatically fill the table with the specified information.

## 4.5 Exporting the Template

To export the completed template:

1. Go to **File** → **Save As**.
2. Choose the **Defaults** tab in the save dialog.
3. Save the file with the **.SchDot** extension to designate it as a template.

## 4.6 Selecting and Using the Template

To use your new template:

1. Go to **Global Properties** → **General** tab.
2. In the **Templates** section, select your newly created template from the list.

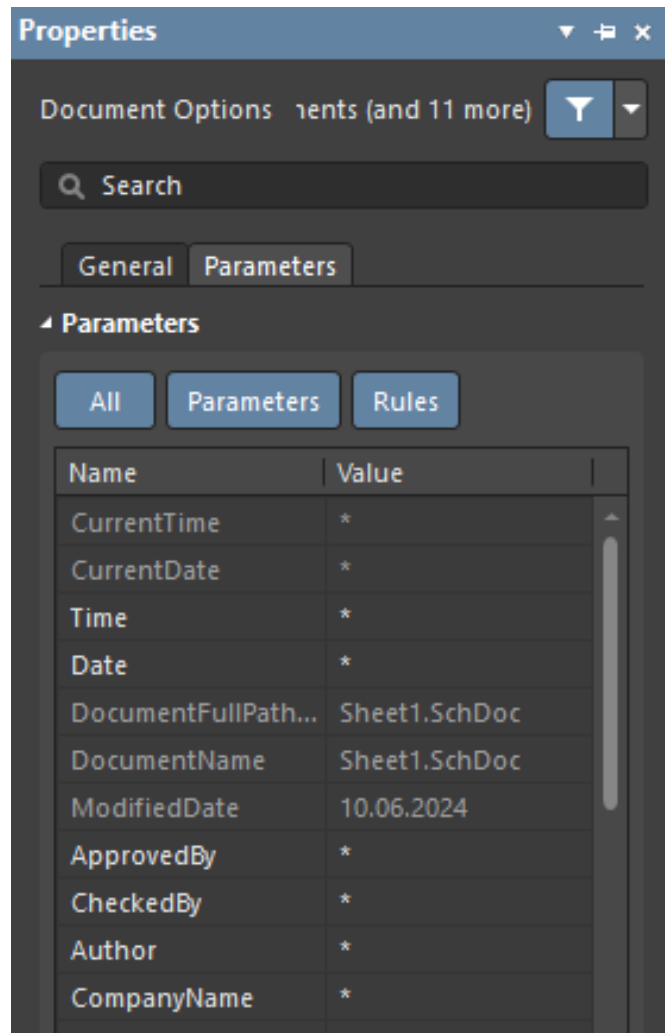


Figure 4.2: Saving the schematic template file.


	<i>Author:</i> Dynamit <i>Title:</i> Title
	<i>Sheet number:</i> 1 / 1 <i>Modified:</i> 10.06.2024
<i>Description:</i> <i>Reviewers:</i>	

Figure 4.3: Completed schematic template ready for use.



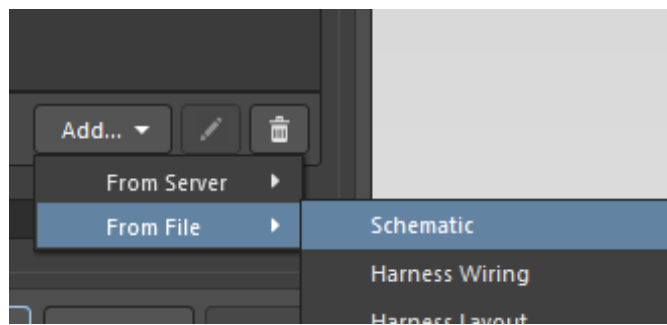


Figure 4.4: Selecting the template from global properties.

# Chapter 5

## Schematics

### 5.1 Working with Schematic and PCB

For efficient workflow, it is beneficial to have both the schematic and PCB views open simultaneously. You can use the Cross-Probe tool to quickly locate components on the PCB layout:

- Right-click on the schematic and select **Crossprobe**.
- Hold **Ctrl** and left-click on an object to highlight its location on the PCB layout.

To highlight common nets between schematic and PCB:

- Press **Alt + Left Mouse Button**.

### 5.2 Parameter Set Directive

The Parameter Set Directive is used to define specific parameters for nets or groups of nets using the Blanket tool. This is particularly useful for setting global properties.

### 5.3 Handling Floating Net Labels and Net Names

By default, floating net labels may not be highlighted as errors. It is advisable to configure project options to flag these labels as errors.

To negate a net name, use the backslash (\) character. Floating power objects can be handled similarly.

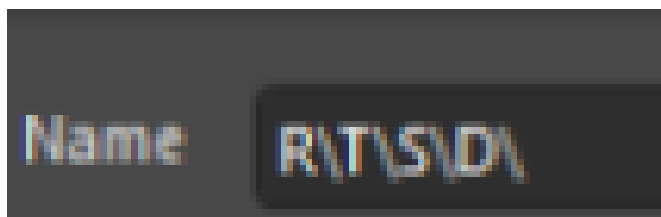


Figure 5.1: Example of net negation.

## 5.4 Using the Connection Matrix

The Connection Matrix tool allows you to quickly modify error settings in project options. This can streamline the process of managing schematic errors.

## 5.5 Creating and Using Buses

Buses are used to connect multiple signals together. To create a bus:

- Right-click → **Place** → **Bus Entry** to create entries for each pin.
- Place the bus on the schematic.
- Attach a port to the bus by right-clicking → **Place** → **Port**.

In project options, ensure that **Generate Net Classes for Buses** is marked to properly define net classes for buses.

## 5.6 Net Classes

Net classes are essential for managing larger projects. Several methods are available to create net classes:

### 5.6.1 Blanket and Parameter Set

- Go to **Place** → **Directives** → **Blanket** to create a blanket directive.
- Attach the **Parameter Set** to specific net names in the schematic.
- Add a net class with the same name as the **Parameter Set**.
- After importing changes to the PCB, the net class will be available.

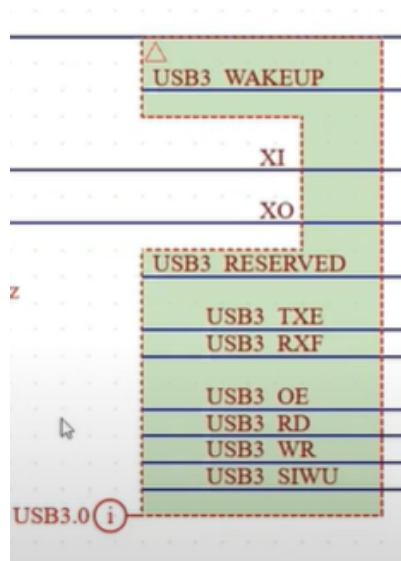


Figure 5.2: Example of a blanket parameter set.

### 5.6.2 By Creating Buses

If **Generate Net Classes for Buses** is checked in project options, net classes should be created automatically. If this is not functioning as expected, consider troubleshooting the settings.

### 5.6.3 Adding Net Name to Signal Harness

This method successfully creates net classes.

### 5.6.4 Using the PCB Editor

- Right-click on a track → **Net Actions** → **Add Selected to NetClass**.
- Note that net classes might be removed from the board when updating schematics. To resolve this:
  - Go to **Project Options** → **Remove Net Class Member and Net Classes**.

## 5.7 Creating Ports

To create ports:

- Press **P** → **Ports**.
- Alternatively, select nets → **Ctrl + C** → **Edit** → **Smart Paste** → **Net Label** → **Port Wire** and **Net Label**.

## 5.8 Harnesses

Harnesses are used for multi-type input traces to connect various schematics, useful for major data lines or magistals:

- To place a harness, go to **Place** → **Harness** → **Connector**.
- For harness entries, go to **Place** → **Harness** → **Entry**.
- Ensure that the harness name and harness port are specified with the same names for proper functionality.
- It is also possible to add a bus to the harness.

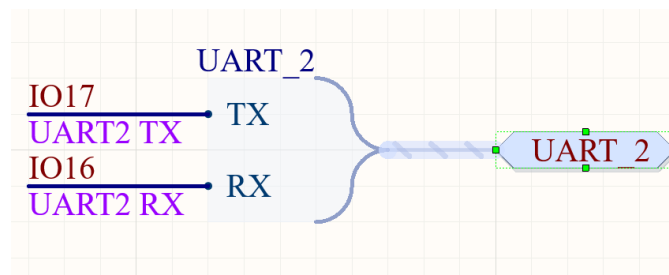


Figure 5.3: Example of a harness setup.

## 5.9 Hierarchical Schematics

To manage hierarchical schematics:

- Go to **Tools** → **Project Options** → **Schematics** → **Graphical Editing** → **Always Drag On**.

### 5.9.1 Down to Up

- Use **Design** → **Create Sheet Symbol from Sheet**.
- Go to **Project** → **Project Options** → **Multi-Channel** → **Change the Replication Suffix**.

### 5.9.2 Top to Down

- Go to **Place** → **Sheet Symbol**.
- Place **Sheet Entries**.
- Use **Design** → **Create Sheet from Sheet Symbol**.

### 5.9.3 Updating Ports

To synchronize ports:

- Right-click on the sheet → **Sheet Symbols** → **Synchronize Sheet Inputs and Outputs**.

To copy ports from a sheet symbol to a schematic:

- Select the ports → **Copy**.
- Use **Smart Paste** → **Smart Paste Wires and Net Labels**.

## 5.10 Stri-Hierarchical Design

Power ports are local, whereas masses must be connected to ports. When placing sheet symbols close to each other, dragging them will automatically connect them.

Adding the word **Repeat** to a sheet symbol will create multiple copies of the sheet. This feature is particularly useful for managing rooms in PCB design. Use **Rooms** → **Copy Rooms** to apply the same layout to all channels.

Channels can be viewed in the lower-left corner of the screen.

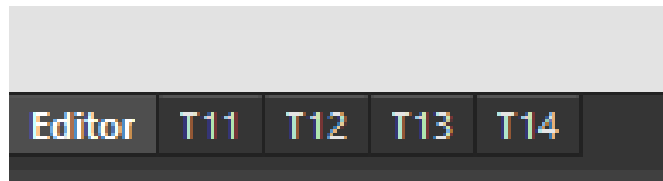


Figure 5.4: View of all channels.

Copying sheet symbols is allowed, but be mindful to use short names for input.

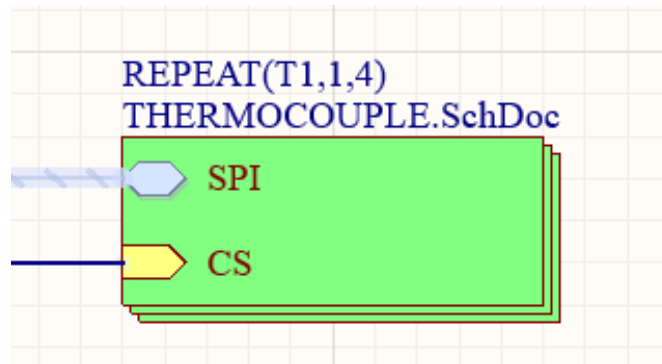


Figure 5.5: Using the repeat function for sheet symbols.

## 5.11 Snippets

Snippets are reusable blocks of schematic or PCB design. To manage snippets:

- Go to **Panels** → **Design Reuse** to view all snippets.
- To create a snippet, right-click on selected objects → **Snippets** → **Create Snippet**.

## 5.12 Colors

You can set different colors for nets:

- Go to **View** → **Set New Colors**.
- Press **F5** to activate or deactivate colors.

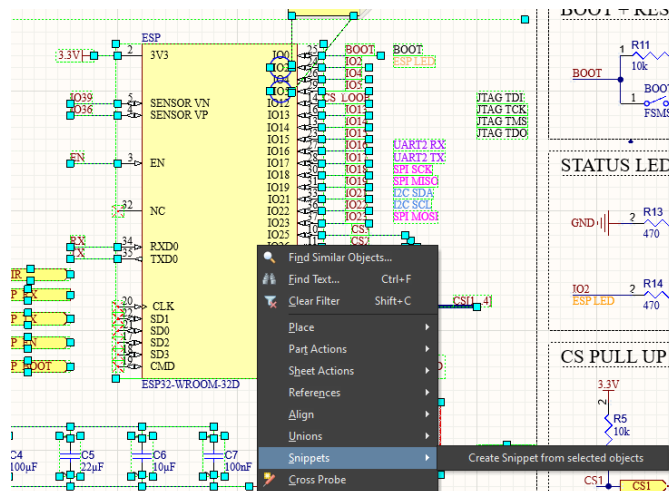


Figure 5.6: Example of snippets management.

## 5.13 Crossovers

To display crossovers:

- Go to **Tools** → **General** → **Display Crossovers**.

## 5.14 Differential Pairs

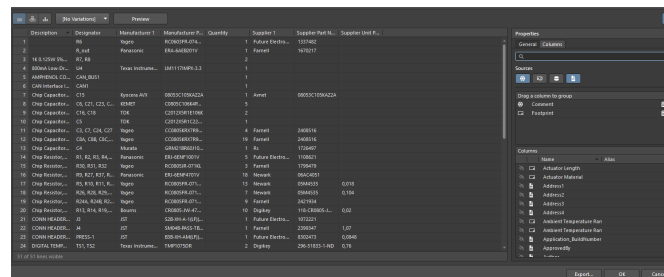
To set designators for differential pairs:

- Go to **Project** → **Project Options** → **Options** → **Diff Pairs**.
- Modify the suffix as required.

## 5.15 BOM Files

To generate a Bill of Materials (BOM):

- Go to **Reports** → **BOM**.
- Choose a template and configure columns for sorting.



Description	Designator	Manufacturer P.	Quantity	Supplier 1	Supplier Part No.	Supplier Unit P.
1	10K	RC0402FR-07	1	Palace Electronics	10K002	
2	10K	RC0402FR-07	1	Palace Electronics	10K002	
3	10K	RC0402FR-07	1	Palace Electronics	10K002	
4	10K	RC0402FR-07	1	Palace Electronics	10K002	
5	10K	RC0402FR-07	1	Palace Electronics	10K002	
6	10K	RC0402FR-07	1	Palace Electronics	10K002	
7	10K	RC0402FR-07	1	Palace Electronics	10K002	
8	10K	RC0402FR-07	1	Palace Electronics	10K002	
9	10K	RC0402FR-07	1	Palace Electronics	10K002	
10	10K	RC0402FR-07	1	Palace Electronics	10K002	
11	10K	RC0402FR-07	1	Palace Electronics	10K002	
12	10K	RC0402FR-07	1	Palace Electronics	10K002	
13	10K	RC0402FR-07	1	Palace Electronics	10K002	
14	10K	RC0402FR-07	1	Palace Electronics	10K002	
15	10K	RC0402FR-07	1	Palace Electronics	10K002	
16	10K	RC0402FR-07	1	Palace Electronics	10K002	
17	10K	RC0402FR-07	1	Palace Electronics	10K002	
18	10K	RC0402FR-07	1	Palace Electronics	10K002	
19	10K	RC0402FR-07	1	Palace Electronics	10K002	
20	10K	RC0402FR-07	1	Palace Electronics	10K002	
21	10K	RC0402FR-07	1	Palace Electronics	10K002	
22	10K	RC0402FR-07	1	Palace Electronics	10K002	
23	10K	RC0402FR-07	1	Palace Electronics	10K002	
24	10K	RC0402FR-07	1	Palace Electronics	10K002	
25	10K	RC0402FR-07	1	Palace Electronics	10K002	
26	10K	RC0402FR-07	1	Palace Electronics	10K002	
27	10K	RC0402FR-07	1	Palace Electronics	10K002	
28	10K	RC0402FR-07	1	Palace Electronics	10K002	
29	10K	RC0402FR-07	1	Palace Electronics	10K002	
30	10K	RC0402FR-07	1	Palace Electronics	10K002	

Figure 5.7: Example of a BOM file.

```
tools-> parametr manager  
  
jak spawdzic podstawki od elemntow i je zmienic automatycznie?  
tools->footprint manager  
skrót -> jc skakanie do komponentu
```

Figure 5.8: Example of additional tips.

To quickly jump to a component, use the **J + C** shortcut. For smart paste operations, use **Ctrl + Shift + V**.

## Chapter 6

# PCB Layout

### 6.1 Creating PCB Layout

To create a PCB layout:

1. Right-click on the project and select **Add New to Project** → **PCB**.
2. Use the **Place** menu to add components and tracks.
3. Define board shape and dimensions.

### 6.2 Editing PCB Layout

- Use the **Properties** panel to adjust component placement.
- Use the **Design Rules** to set constraints for routing and manufacturing.

#### 6.2.1 Design Rules

#### 6.2.2 Layout

#### 6.2.3 Design Rules



## Chapter 7

# Design Verification

### 7.1 Running DRC Checks

To run Design Rule Checks (DRC):

1. Go to **Tools -> Design Rule Check**.
2. Review and resolve any errors or warnings.

### 7.2 Simulating Designs

To simulate designs:

1. Use the **Simulate** menu to set up simulation parameters.
2. Run simulations and analyze results in the **Simulations** panel.

## Chapter 8

# Generating Outputs

### 8.1 Generating Gerber Files

To generate Gerber files for manufacturing:

1. Go to **File -> Fabrication Outputs -> Gerber Files**.
2. Configure layer settings and generate files.

### 8.2 Creating BOMs

To create a Bill of Materials (BOM):

1. Go to **Reports -> Bill of Materials**.
2. Customize the BOM template and export.

## Chapter 9

# Advanced Features

### 9.1 Using Snippets

Snippets allow you to reuse design elements:

1. Select components and right-click to create snippets.
2. Access snippets in **Panels -> Design Reuse**.

### 9.2 Collaborative Design

Collaborate in real-time with your team:

1. Enable **Altium 365** for cloud collaboration.
2. Share projects and collaborate using **Altium 365** features.

# Chapter 10

## Shortcuts

### 10.1 Common Shortcuts

- **Q**: Toggle between imperial and metric units.
- **Z + B**: Zoom to board.
- **Z + A**: Zoom to all objects.
- **Shift + S**: Toggle single layer mode.
- **Shift + Ctrl + Scroll Up/Down**: Change layer.
- **Numpad +/-**: Change layer.
- **Ctrl + V**: Paste.
- **Ctrl + X**: Cut.
- **F5**: Refresh view.
- **Shift + F**: Find elements.
- **Alt + F4**: Close application.

### 10.2 3D View Shortcuts

- **Ctrl + F**: Flip board.
- **P + V**: Place via.
- **Shift + S**: Toggle single layer mode.
- **Ctrl + L**: Toggle layer visibility.
- **Ctrl + D**: Duplicate object.
- **Ctrl + E**: Edit object.

## 10.3 Schematic Shortcuts

- **P + P**: Place part.
- **P + W**: Place wire.
- **P + G**: Place ground.
- **Shift + C**: Compile project.
- **Ctrl + R**: Rotate component.
- **Ctrl + M**: Measure distance.

## 10.4 PCB Layout Shortcuts

- **P + T**: Place track.
- **P + V**: Place via.
- **Shift + S**: Toggle single layer mode.
- **Ctrl + L**: Toggle layer visibility.
- **Ctrl + D**: Duplicate object.
- **Ctrl + E**: Edit object.

Appendix A

Appendix