CHAPTER 1
Getting Started

Those of you who are familiar with setting up projects and drawing schematics in Capture may want to skip this chapter, as it has been written for those of you who have little or no experience of using Capture. This chapter will describe how to start Capture and how to set up the project type and libraries for PSpice simulation.

At the end of each chapter there are some exercises to do and as you go through the book, each chapter will build upon the exercises from previous chapters.

1.1 STARTING CAPTURE

Circuit diagrams for PSpice simulation are drawn in either Capture or Capture CIS schematic editor. The CIS option, which stands for Component Information System, allows you to select and place components from a component database instead of selecting and placing components from a library. For this book, it does not matter whether the circuits are drawn in Capture or Capture CIS.

If you have the OrCAD software installed, launch Capture or Capture CIS, by clicking on:

Start > Program Files > OrCAD xx.x > Capture

or

Start > Program Files > OrCAD xx.x > Capture CIS
where xx.x is the version number, e.g. 10.5, 11.0, 15.5, 15.7, 16.0, 16.2, 16.3 or 16.5.

At the time of writing this book, the current version is 16.5 and is started by:

Start > Program Files > Cadence > Release 16.5

If you have the Cadence software installed, the tools are installed under the Allegro platform name. In this case, only Capture CIS is available and is branded as Design Entry CIS:

Start > Program Files > Allegro SPB xx.x > Design Entry CIS

### 1.2 CREATING A PSpICE PROJECT

New designs started in Capture will automatically create a project file (.opj) which will reference associated project files such as the schematics, libraries and output report files.

Before the circuit diagram is drawn, the project type and libraries required for the project need to be set up. First of all a new project is created by selecting from the top toolbar:

File > New > Project

In the New Project window (Figure 1.1), you enter the name of the project and then you have a choice of one of four project types:

- **Analog or Mixed A/D** is used for PSpice simulations.
- **PC Board Wizard** is used for schematic to PCB projects.
• **Programmable Logic Wizard** is used for CPLD and FPGA designs.
• **Schematic** is used for schematic and wiring diagrams.

When you select a Project type, the **Tip for New Users** gives a brief explanation of the project type. For PSpice projects, select **Analog or Mixed A/D**. This will activate the PSpice menu on the top toolbar in Capture.

It is recommended that a new directory location (folder) is created for each new project. This can be done by clicking on the **Browse...** button shown in Figure 1.1, which opens up the **Select Directory** window shown in Figure 1.2.

![FIGURE 1.2](image1.png)

Creating a project folder location.

By selecting the **Create Dir...** button, the **Create Directory** window (Figure 1.3) appears, which allows you to name the directory (folder).

![FIGURE 1.3](image2.png)

Creating the project folder.

The created folder, PSpice Exercises in this example, will appear in the **Select Directory** window. However, you must highlight and select the folder by clicking twice with the left mouse button, which will show the ‘open’ yellow icon as shown in Figure 1.4. A further subdirectory or folder can be created by clicking on the **Create Dir...** in the **Select Directory** window button and following the same procedure above.
The project folder location will then appear in the Location box of the New Project window (see Figure 1.1).

**FIGURE 1.4**
The project folder has been selected.

An alternative method of creating the project folder is to type in the folder location directly into the Location box in the New Project window in Figure 1.1 and Capture will automatically create the folder.

**NOTE**
It is a common mistake to create a project folder and not select the folder. Make sure you double click on the created folder name in the **Select Directory** window (Figure 1.4).

The next window to appear is the Create PSpice Project window, which sets up the project for PSpice simulation (Figure 1.5).

The pull-down menu option allows you to select preconfigured Capture-PSpice libraries for the project. The most commonly used option for new projects is **Simple.opj**, which adds the following five default libraries to the project:

- Analog.olb
- Breakout.olb
- Source.olb
- Sourcstm.olb
- Special.olb
These libraries contain the most commonly used parts for PSpice projects and are recommended for new projects.

There is also an option to create updated versions of an existing project, i.e. to create a newer version 2 based upon the original version 1 project. In the Create PSpice Project Window (Figure 1.5), select the function **Create based upon an existing project** and then **Browse** to select an existing project. This will copy the existing project and all its associated files into the new project. This is similar to using the **File > Save As** function.

If the **Create a blank project** option is selected, then no Capture-PSpice libraries are added to the project. The libraries can be added later. This will be demonstrated in one of the exercises at the end of this chapter.

When a new project is created, a **Project Manager** window is created (Figure 1.6) which shows the absolute path to the libraries. Remember that these are Capture symbol libraries which define the graphics for the parts. They are not the PSpice model libraries. The Capture libraries are installed by default and can be found, depending on the OrCAD or Cadence software version you are using, for example, at:

```
<software install path> OrCad > OrCAD_10.5 > tools > capture > library > pspice
```

or

```
<software install path> Cadence > SPB_16.3 > tools > capture > library > pspice
```
Normally the *software install path* is the C: drive.

![Software Install Path](image)

**FIGURE 1.6**
Project Manager showing the Capture parts libraries and their location.

---

**TIP**

If the *Project Manager* window is not displayed, select from the top toolbar, *Window > <project name>.opj* file (Figure 1.7). Here the project name is resistors. Note the project name file extension .opj.

![Project Manager Window](image)

**FIGURE 1.7**
Displaying the Project manager window.

Alternatively, click on the Project manager icon or .
1.3 SYMBOLS AND PARTS

1.3.1 Symbols

Before drawing a schematic diagram, it is useful to know the difference between a part and a symbol. Symbols differ from parts in that they are not placed from the Place Part menu in Capture. You have to select the symbol from the Place menu (Figure 1.8).

The Place menu also shows the corresponding shortcut keys. For example, to place a Power symbol, press F and the Place Power menu appears as shown in Figure 1.9.

Wires connected to symbols take on the name of the symbol. For example, to define a wire to be connected to zero volts, you place a ‘0’ symbol. To define a +5 V connection you can use a VCC_CIRCLE symbol and rename it +5 V. All wires connected to the +5 V symbol will take on a net name of +5 V. A net is a wire connection. There are many different symbols you can use to define the power and grounds connections and you can rename them accordingly.
In the Place Power menu in Figure 1.9, a VCC_CIRCLE symbol has been selected and its name has been changed to +5 V. Any wires (nets) connected to +5 V will take on the net name +5 V.

Other symbols include hierarchical ports and off-page connectors which allow signals to be connected together throughout the design. These will be discussed in Chapter 20.

There are two symbol libraries, source and capsym. Capsym contains all the analog ground and power symbols, while source, which also contains the analog 0V symbol, contains the digital $D_HI$ and $D_LO$ symbols, which are used to set a digital level of 'hi' or 'lo' on a wire or pin of a digital device.

1.3.2 Parts

To place a part, select Place -> Part. Figure 1.10a shows the Place Part menu for version 16.0 and Figure 1.10b shows the Place Part menu for version 16.3.

Although the two menus look different they have the same functionality in that they display the list of libraries available and the parts available in the libraries; and they both provide a part search function. In Figure 1.10a, only the analog library has been highlighted and so only those parts for that library are shown in the Part List.
In Figure 1.10b, all the libraries have been highlighted and so you see the name of the part and which library it comes from. If you place the cursor over any part in the Part List, a tool tip rectangular bar appears showing the absolute path to the library part.
NOTE
Batteries, voltage sources and current sources are found in the source library from the Place Part menu (Place > Part) and are not to be confused with the power symbols (VCC_circle, 0V, etc.) from the capsym library (Place > Power or Place > Ground), which are effectively used to ‘invisibly’ connect wires with the same net name together.

In the Place Power or Place Ground window (Figure 1.11) there is a source library which contains only the digital HI, digital LO and ground 0V symbols.

To recap, symbols are placed from the Place menu and parts are placed from the Place > Part menu. Also note that both Part libraries and Symbol libraries have an .olb extension and are the Capture graphical parts.

1.4 DESIGN TEMPLATES
From version 16.3 onwards, Design Templates have been added, which are complete electronic circuits and topologies including simulation profiles for analog, digital, mixed and switched mode power supplies. You can select any of these templates from the pull-down menu in the Create PSpice Project window when you create a new project (Figure 1.12).

Figure 1.13 shows the Design Template for a Single Switch Forward Converter which includes the schematic and explanatory text.