I. Defining a Complex Device

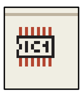
This tutorial is a systematic guide in defining a library element for a pin-leaded TTL chip.

Creating a New Library

Click on the *File/New/Library* menu in the EAGLE Control Panel. The Library Editor window appears, containing a new library, *untitled.lbr*.

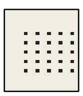
Drawing the Pin-Leaded Package

The component, ADC0804, is manufactured in a pin-leaded package. This is a DIL-20 housing with pin spacing of 0.1inch and package width of 0.250 inch.



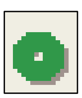
Click on the *Edit a package* icon in the action toolbar, and enter the name to the package in the *New* box of the *Edit* menu, which is the *DIL-20* in our example. Answer *Yes* to the question: *Create new package 'DIL-20'?* The Package Editor window now opens.

Set the Grid



Use the GRID command to set the grid size for the pad placement. 0.05 inch (i.e. 50 mil). Use the F6 key to show or hide the grid.

Place Pads



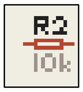
Use the PAD command to place the solder pads in accordance with data sheet specifications. The origin (+) should be somewhere near the center of the package.

Shape and drill diameter specifications can be changed with the parameter toolbar when the PAD command is active. If the parameter tool bar is not present, it needs to be selected by clicking *Options/User Interface* and checking Parameter toolbar under Controls.

The parameter toolbar when the PAD command is active

The pad diameter is usually defined with a value of *auto* with a default diameter of 55 mil. Properties of pads already placed can be changed by clicking *Edit/Change/Shape* or *Edit/Change/drill*.

Pad Name



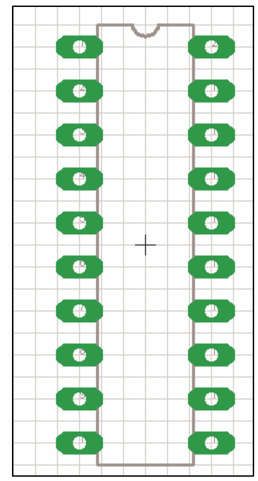
Label pads using the NAME command. EAGLE automatically assigns pad names, *P$1*, *P$2*, *P$3*, etc. as placement proceeds.

Draw the Silkscreen Symbol

Commands WIRE, CIRCLE, and ARC are used to draw the silkscreen symbol. These commands are associated with board layer 21, *tPlace*. This layer contains what will be printed on the board. Line thickness for the silk screen is usually 0.01 inch.

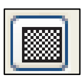
<u>Important: Do not cover any areas that are to be soldered.</u>
Any objects, such as resistor lead wires, which go over pads, are drawn in layer 51, *tDocu*.

On the right is the silkscreen DIL-20. The grid has been changed back to default (0.05 inch).



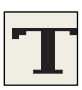
DIL-20 Silkscreen

Areas Forbidden to Components

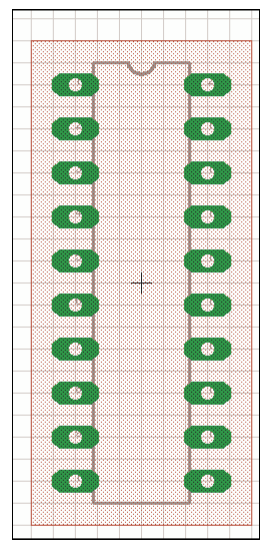


Using the RECT command, a keepout can be constructed over the whole component. This action creates a restricted area, which will send up a DRC warning if any other board components are too close or overlapping. The restricted area display can be turned off by clicking the DISPLAY command icon and unselecting layer 39, *tKeepout*.

Labeling



The TEXT command allows you to place the texts >NAME and >VALUE in those places where in the board the actual name and/or the actual value are to appear. Grid size was changed to 0.01 inch for desired text placement. The SMASH and MOVE commands can be used later to change the position of text relative to the package symbol on the board. The screen shot of the Description Window in the Package Editor (above) displays text were the user will enter the DIL-20 IC component NAME.



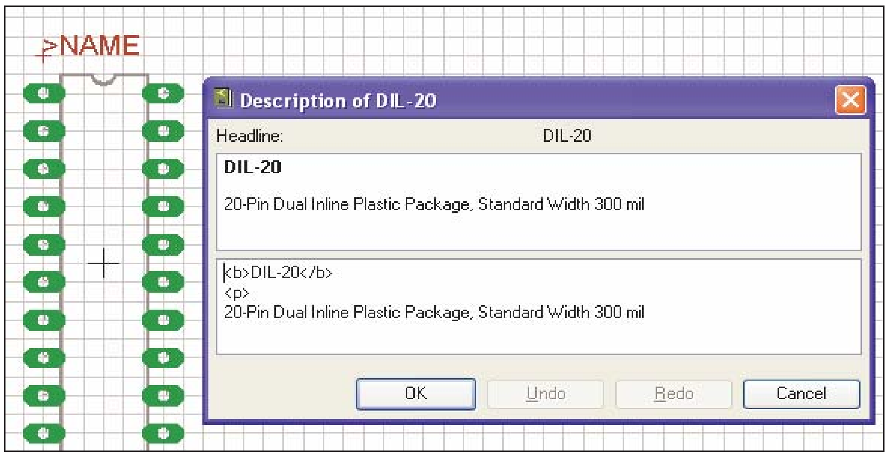
Restricted Area

Description

Clicking on *Description* (located beneath the Package Editor screen) opens a window divided into two sections. Rich Text syntax is entered in the lower part with the resulting description displayed in the upper section. The EAGLE software help system has more information regarding this syntax format under *Rich Text*.

The descriptive text entered for the DIL-20 is:

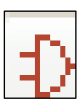
```
<b>DIL-20</b>
<p>
20-pin Dual Inline Plastic Package, Standard Width 300 mil
```



Package Editor Description Window

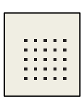
Creating the Schematic Symbol

Define a New Symbol



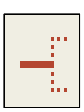
Select the symbol editing mode, and enter the symbol name *DIL-20* in the *New* field.

Set the Grid



The grid of the symbol editing field should be set to 0.1 inch. The pins of the symbol **must** be placed on this grid.

Place the Pins



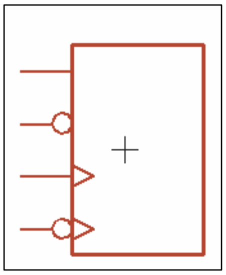
Select the PIN command. Properties of the pins can be set in the parameter toolbar, before placing them with the left mouse button. Pin properties (including groups of pins using the GROUP command) can be changed at a later stage with the CHANGE command as mentioned earlier.

PIN command – Parameter toolbar

Orientation

Pin orientation can be set by either using the four left-hand icons in the parameter toolbar or by rotating with the right mouse button.

Function



Pin function parameters

The function parameter is set with the next four icons on the parameter toolbar. This specifies whether the symbol is to be shown with an inversion circle (Dot), with a clock symbol (CLK), with Both (DotClk) or simply as a stroke (None). The diagram on the right illustrates the four representations on one package.

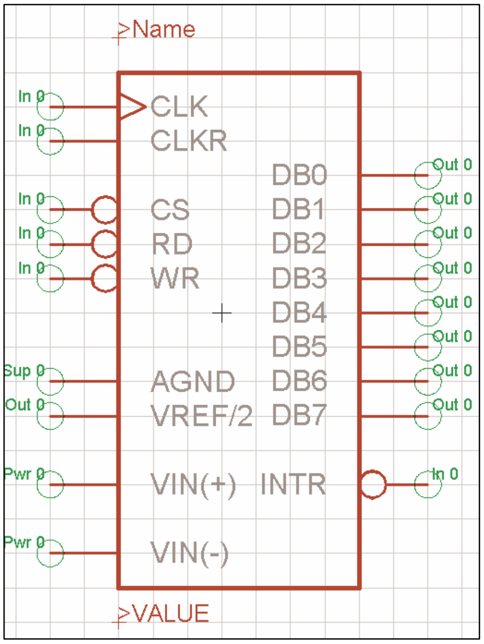
Length

The next four icons after the functions on the parameter toolbar are used to set pin length (0, 0.1 inch, 0.2 inch, and 0.3 inch). The 0 setting is used if no pin line is to be visible.

Visible

The last four icons on the parameter toolbar specify whether the pins are to be labeled with pin name, pad names, both or neither. The symbol being created for the ADC0804 library will display with pin and pad names. Schematic pin numbers, which are not shown in the picture to the right, will be displayed later when pads and pins are connected.

Direction



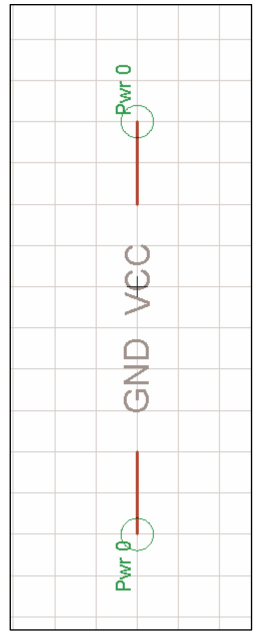
ADC0804 symbol

The direction parameter (drop down box on the parameter toolbar specifies the logical direction of signal flow:

NC	Not connected
In	Input
Out	Output
I/O	Input/Output
OC	Open Collector or Open Drain
Hiz	High impedance output
Pas	Passive (resistors, etc.)
Pwr	Power pin (power supply input)
Sup	Power supply output for ground and power supply symbols

Power Supply Symbol

A separate symbol was created for the two power pins VCC and GND using the same procedure described on the previous page. Both pins were give PWR as their direction with both pin and pad labels chosen with the VISIBLE command.



Power Supply symbol

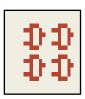
SwapLevel

The swaplevel is a number between 0 and 255. the number 0 means that the pin cannot be exchanged for another pin in the same gate. Any number bigger that 0 means that pins can be exchanged for other pins that have the same swaplevel and are defined within the same symbol. The pins can be swapped in the schematic or in the board with the PINSWAP command.

The two pins of a resistor can have the same swaplevel since they are interchangeable. The pins of the ADC 0804 symbol illustrated at the bottom of the previous page all have a swaplevel of 0 since none of them can be interchanged.

Associating the Package and Symbol to Form a Device Set

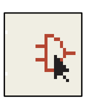
The final step in the process of defining an EAGLE library element is the forming of the Device set. The Device set associates the symbol with the package to form a real component.



Click on the *Edit a device* icon. Enter a name for the device on the *new* line. The device being designed in this tutorial is the ADC0804. After naming the device, the Device Editor window opens.

Note: If the device is to be used in more than one technology, a * is used as a place holder at a suitable location in the device name to represent the different technologies. More information is available on this topic beginning on Page 184 of the EAGLE 4.1 Manual.

Select Symbols



Use the ADD icon to fetch the symbol(s) that belong to the device. A widow opens in which all the symbols available in the current library are displayed.

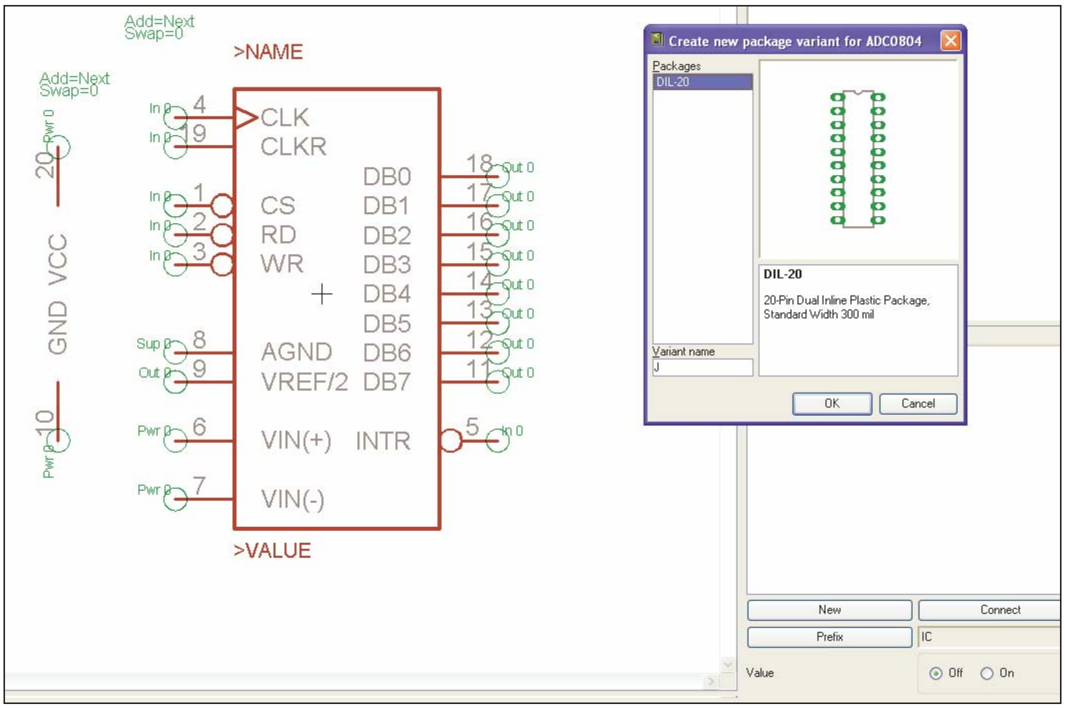
Devices consisting of several schematic symbols which can be placed independently of one another in the circuit (in EAGLE these are known as *gates*), then each gate is to be individually brought into the schematic with the ADD command.

Set an addlevel of *Next* and swaplevel of *0* in the parameter toolbar, and then place the gate near the origin. More information is available on this topic on page 185 and 195 of the EAGLE 4.1 Manual.

Pins the device being constructed in this tutorial cannot be swapped so the swaplevel remains at 0.

Choosing the Package Variants

In the Device Editor window, click on the *New* button at the lower right. A window opens and displays the package defined in this library. The variant *DIL-20* is selected and given the version name J.



Device Editor displaying "Create new package variant" window

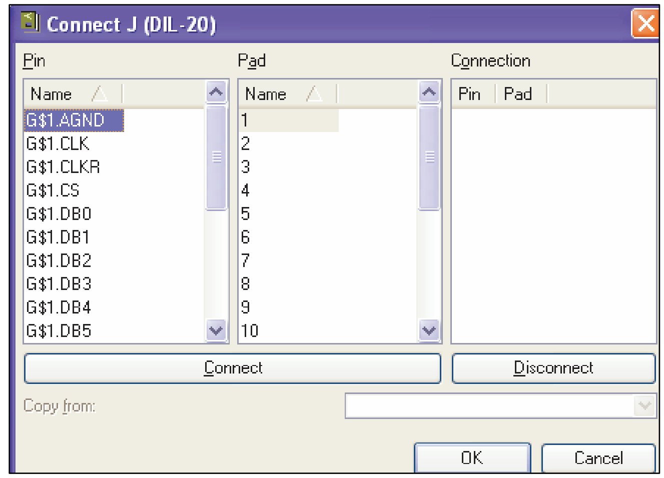
Clicking on a package variant entry with the right mouse button will open a context menu. This allows variants to be deleted, renamed or newly created, technologies to be defined, or the CONNECT command to be called.

Once selected, the variant entry is marked by a yellow symbol with an exclamation mark. This means that the assignment of pins and pads has not yet been (fully) carried out.

Connect – Connections Between Pins and Pads

This is the most important step in the library definition. CONNECT assigns each pin to a pad. The way in which nets in the schematic diagram are converted into signal lines in the layout is defined here. Each net at the pin creates a signal line at a pad. Check the connects in the library with care. Errors that may pass unnoticed here can make the layout useless.

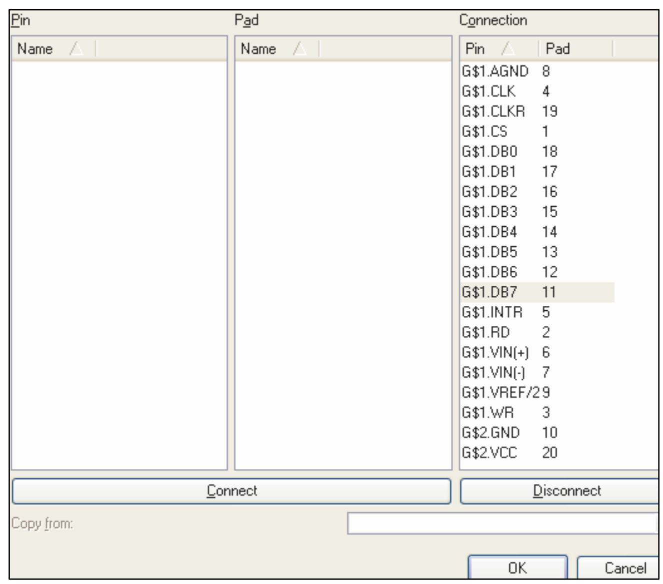
Select the *J* version from the package list and click on the Connect button. The connect window, displayed on the right, opens.



CONNECT dialog

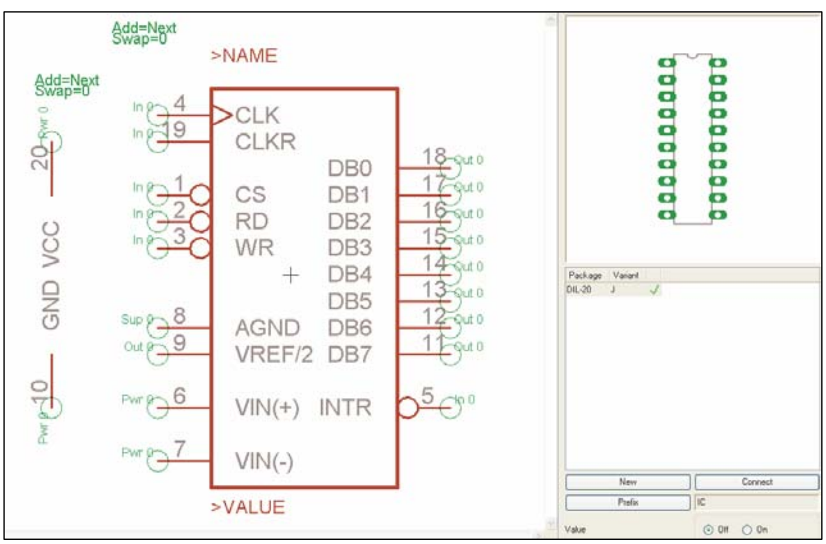
The list of pins is on the left, and the pads are in the center. Click on a pin-entry, and select the associated pad. Both entries are now marked. You join them with the *Connect* button. This pair now appears on the right, in the *Connection* column. Join each pin to its pad in accordance with the data sheet. Finish the definition by clicking on OK. Shown on the right is the CONNECT dialog box with all the pins and pads connected in the DIL-20 package.

One pin has to be connected to exactly one pad! Pins with direction NC (not connected) must be connected to a pad, as well!



Pin and corresponding pad connections

The Device Editor window, pictured below, now shows a green tick to the right of the package variant indicating that the connection is complete. This is only true when every pin is connected to a pad.



Device Editor window displaying completed package variant

Defining Technologies

If you are creating a device that is to be used in more than on technology, refer to the previous heading entitled Associating the Package and Symbol to Form a Device Set.

Specifying the Prefix

The prefix of the device name is defined simply by clicking on the *Prefix* button. *IC* is to be entered in this example.

Value

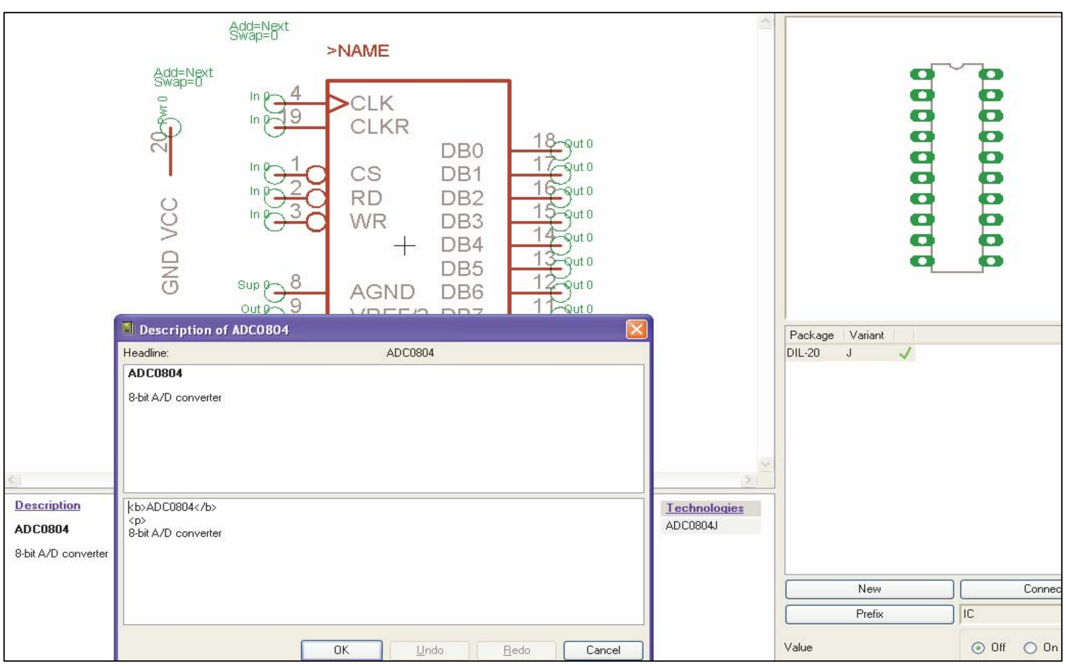
The setting of *value* determines whether the VALUE command can be used to alter the value of the device in the schematic diagram and in the layout. The default setting is *off*, so that alteration is not permitted. Since that appears to be appropriate here too, value is left *off*.

Description

The description box is located beneath the schematic symbols in the Device Editor window. Clicking on *Description* allows a description of the device to be entered. Typical terms should be used so that a keyword search can be performed if needed for the device.

The following Rich Text format results are show in the screen shot below under the Description heading:

```
<b>ADC0804</b>
<p>
8-bit A/D converter
```



Device Editor symbol description box

II. Library and Part Management

Copying of Library Elements

Within a Library

If you want to use a **symbol** or **package**, which already exists in a related manner for a device definition, you can copy it within the library with the commands GROUP, CUT, and Paste. Afterwards it can be modified as requested**.**

The following sections explain how to use an existing DIL 20-pin package/symbol located in the EAGLE linear library, *linear.lbr*, for the previously defined ADC0804.

Open Library

Use the menu *File/Open/Library* in the Control Panel to open the library *linear.lbr* or select the entry *Open* from its context menu of the tree view's expanded *Libraries* branch.

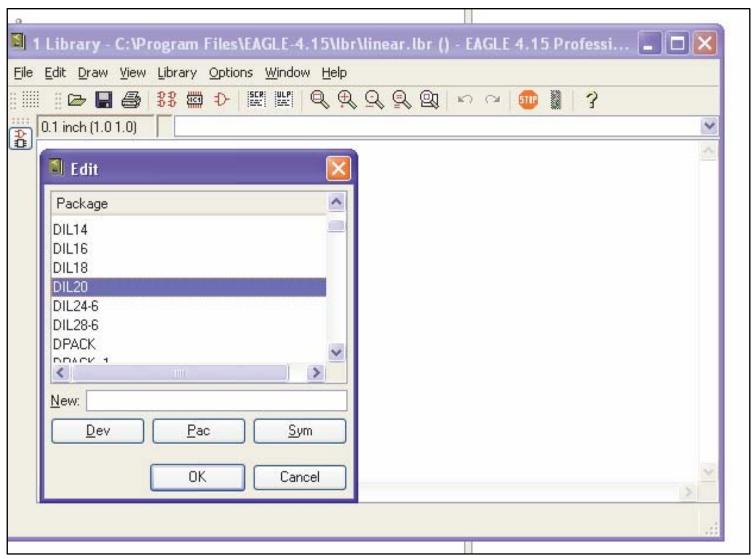
Edit Existing Element

Open the *Edit* window with *Library/Package* and select the package *DIL20*. after clicking *OK* it sis shown in the package editor window.

Use DISPLAY to show all layers.

Draw a frame around all objects to be copied with GROUP.

Now click the CUT icon followed by a left mouse click into the group.

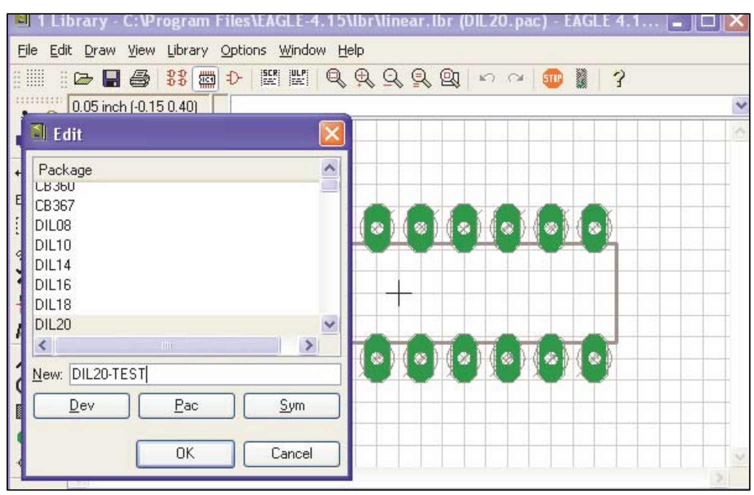


Device Editor *Library/Package/Edit* window

Define New Element

Click the *Edit-a-package* icon in the action toolbar. Enter the name *DIL20-TEST* in the *New* field of the Edit window and confirm with *OK*.

Click the PASTE icon followed by a click at the drawing's reference point. The package will be placed.



Naming the new package in the *Edit* field

Now the package, DIL20-TEST, can be modified as requested. The procedures just described can be applied to symbols as well. Devices, however, can not be copied within a library.

From One Library into Another

Devices

If there is a proper device or Device set that you want to use in your current library you can copy it in two different ways.

In the Control Panel:
Move (with Drag & Drop) the requested Device set from the Control Panel's tree view into the opened Library Editor window. The complete device set with symbol(s) will be copied and newly defined.
As an alternative you could use the entry *Copy to Library* in the context menu of the device entry.

With the COPY command:
Type, for example,

```
COPY ADC0804@ADC0804.lbr
```

or with the whole path

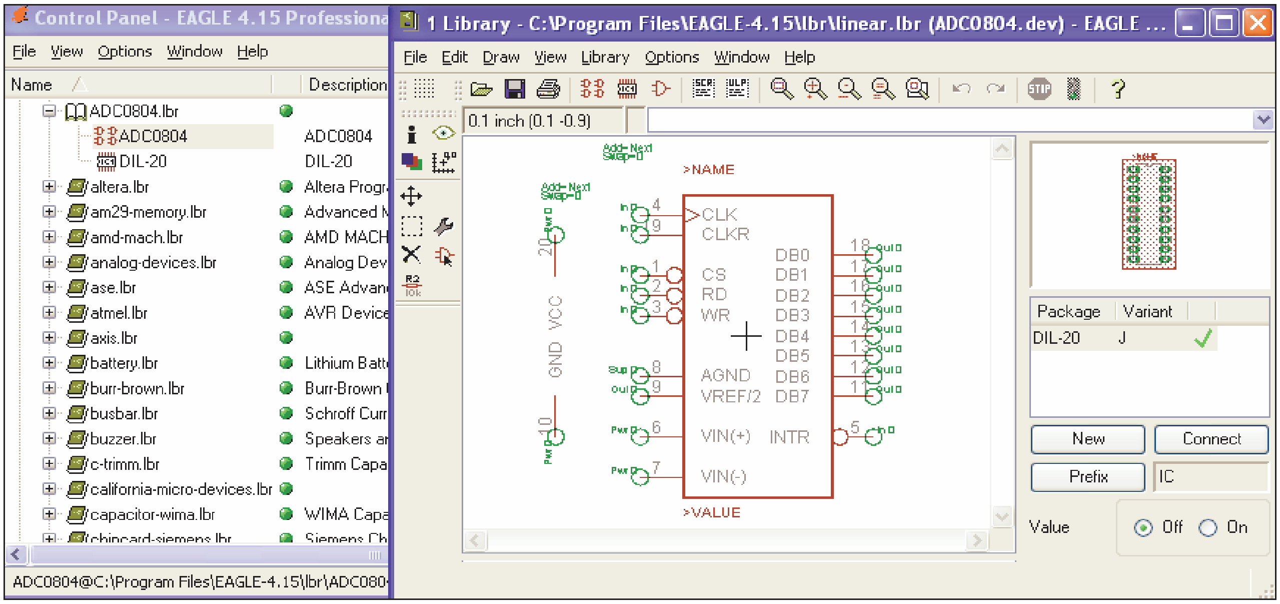
```
COPY 'ADC0804@c:\program files\eagle-
4.15\lbr\ADC0804.lbr'
```

In the command line, the Device ADC0804 from library ADC0804.lbr is copied into the current library.

If the Device should be stored in the current library under a new name simply enter it, like here:

```
COPY ADC0804@ADC0804.lbr ADCNEW
```

The screen shot at the top of the next page displays the Control Panel library tree along with the Library Editor window of the linear.lbr. The ADC0804 device was copied into the Device Editor using the Drag & Drop method described above.



ADC0804 device set copied into the Library Editor window of linear.lbr

Symbols

As symbols are not shown in the tree view of the Control Panel it is not possible to copy them from there into the Library Editor.
Symbols are copied from one library into another the same way as one would do it within the same library. Therefore we use the commands GROUP, CUT, and PASTE (see previous page).

The only difference to copying a symbol with a library is that you have to open the other library (*File/Open*) after CUT and click into the group. Then you define a new symbol that may have the same or another name as before.

In short:

- OPEN the *source* library and EDIT the Symbol
- DISPLAY all layer
- Select all elements with GROUP
- Use CUT and click into the group to set a reference point
- OPEN your *target* library with *File/Open*
- EDIT a new Symbol
- Use PASTE to place it
- Save library

Packages

The procedure to copy packages is nearly the same as to copy devices.

Either move (with Drag & Drop) the requested package for the Control Panel's tree view into the opened Library Editor window. The complete package will be copied and newly defined in the current library. As an alternative you could use the entry Copy to Library in the context menu of the package entry.

Or use the COPY command. Type, for example,

```
COPY DIL-20@linear.lbr
```

in the command line, the package DIL-20 from library linear.lbr is copied into the currently opened library. If the library is not in the current working directory you have to enter the whole path, as for example, in:

```
COPY 'DIL-20@c:\program files\eagle-4.15\lbr\linear.lbr'
```

If the package should be stored in the current library under a new name simply enter it directly in the command line:

```
COPY 'DIL-20@c:\program files\eagle-
4.15\lbr\linear.lbr DIL-20NEW'
```

The package is stored with the new name DIL-20NEW now.

If you want to copy a package that already exist with the same name in the target library the package will be simply replaced.

If the package is already used in a device and the position or the name of one or more pads changes, EAGLE prompts a message in which mode the pads are to be replaced. This procedure can also be cancelled. The package remains unchanged then.

If the enumeration and position of the pads are unchanged but the order is, EAGLE will ask you for the appropriate update mode. Depending on your selection the pin/pad connections of ht device may change (see CONNECT command).

Composition of Your own Libraries

The previously mentioned methods to copy library elements make it very easy to compose your own libraries with selected contents.

Provided the Control Panel and ht Library Editor window are arranged in a manner that both are visible on the screen at the same time, it is very easy to make user-defined libraries while browsing through the library contents in the Control Panel. Simply use Drag & Drop or the context menu *Copy to Library* of the current device or package.

Removing and Renaming Library Elements

Devices, symbols, and packages can be removed from a library with the REMOVE command. Defining a new library element can’t be cancelled by UNDO.

Example:
You would like to remove the package named DIL-20.
Open the menu Library/Remove….A dialog window opens where you can enter the name of the element to be deleted.

This can be done also at the command line:

```
REMOVE DIL-20
```

Packages and symbols can be removed only if they are not used in one of the library's devices. In this case the message *Package is in use!* or *Symbol is in use!* appears. Remove the corresponding device first or delete the particular package or symbol in the device (set).

Use the RENAME command to change a name of an element in a library. Switch to the Package editing mode so that the element that should be renamed is shown first and open the menu *Library/Rename*. A dialog window opens where you can enter the new name of the element.

This can also be done at the command line:

```
RENAME DIL-20 DIL20
```

The package DIL-20 gets the new name DIL20.

The device, symbol, or package name may also be given with its extension(.dev, .sym, .pac), for example:

```
REMOVE DIL-20.PAC
```

In this case it is not necessary to switch to the related editing mode before.

Update Packages in Libraries

As already mentioned in the section *Copying of Library Elements* it is possible to copy packages from one library into another one. An already existing package is replaced in that case.

Each library contains packages which are needed fro device definitions. In many libraries identical types of packages can be found. To keep them uniform over all libraries it is possible to replace all packages of a library with those of another library with the help of the UPDATE command. An existing package with the proper name will be replaced be the current definition.

If you have, for example, special requirements for packages you could define them in a custom-built package or SMD (Surface Mount Device) library. The UPDATE command could transfer them to other libraries.

Therefore open the library to be updated and select *Library/Update*….
Now select the library in which you want to take the packages from.

Having finished the update EAGLE reports in the status bar:
Update: finished – library modified!
If there was nothing to replace: *Update: finished – nothing to do.*

It is also possible to use the command line for this procedure.

If you want to update your library with packages from, for example, dil.lbr, type:

```
UPDATE dil.lbr
```

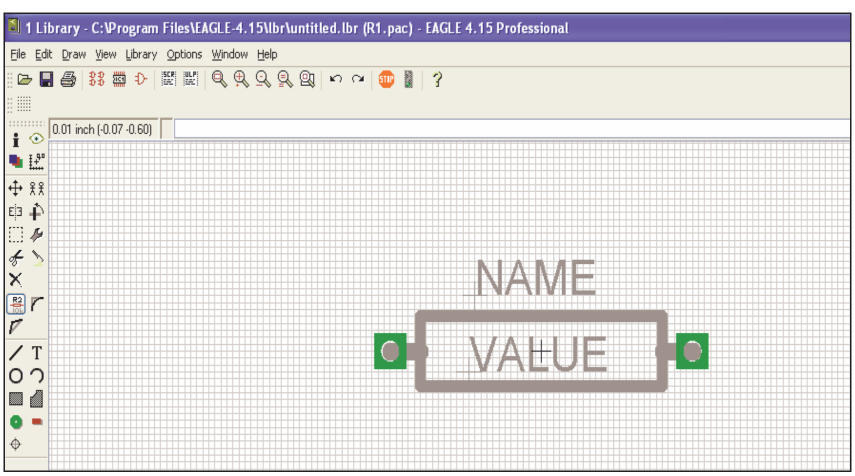
To transfer packages from different libraries, type in one after another:

```
UPDATE dil.lbr smd.lbr special.lbr
```

To update a single package, type in the package name:

```
UPDATE DIL-20@dil
```

The extension .lbr is not necessary. You may also use the whole library path as previously described.



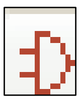
Note

The CHANGE command can be changed at a later stage to alter object properties including but not limited to pad shape or text height.

The GROUP command can be used to change the properties of several objects at one go. Click the CHANGE command, select the parameter and the value, and click on the drawing surface with the right mouse button.

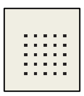
Resistor Symbol

Define a New Symbol



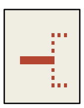
Select the symbol-editing mode, and enter the symbol name *R* in the *New* field.

Set the Grid



The grid of the symbol editing field should be set to 0.1 inch. The pins of the symbol **must** be placed on this grid.

Place the Pins



Select the PIN command. Properties of the pins can be set in the parameter toolbar, before placing them with the left mouse button. Pin properties (including groups of pins using the GROUP command) can be changed at a later stage with the CHANGE command as mentioned earlier.

PIN command – Parameter toolbar