

## Laboratory 4. Editing circuit

### Aim of the laboratory

Edit electronic parts that are not available in libraries.

### Necessary equipment

-Work stations with OrCAD installed

The libraries in Capture CIS cover most of the needs in the matter of defined symbols. The user can also add new compatible libraries (files with .OLB extension) or create their own libraries with new defined parts.

The user can create his own library, independent from the project or attached to the project by accessing File->New->Library.

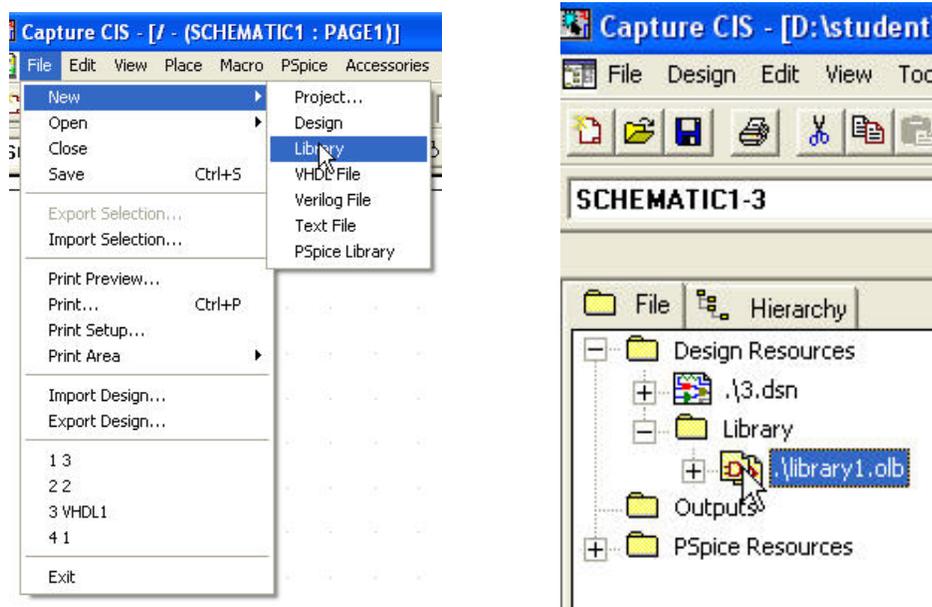


Fig1

For adding a new part or symbol in this library select the library and then from the Design menu click on New Part or New Symbol.

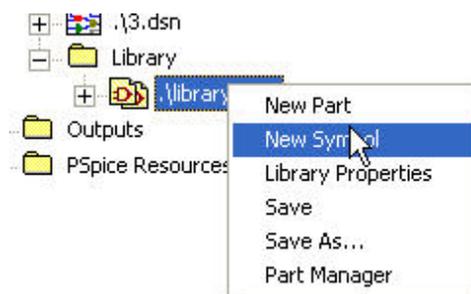


Fig2

After pressing New Part, a window with the proprieties of the part will appear: Name, Part Reference Prefix and other additional specifications:

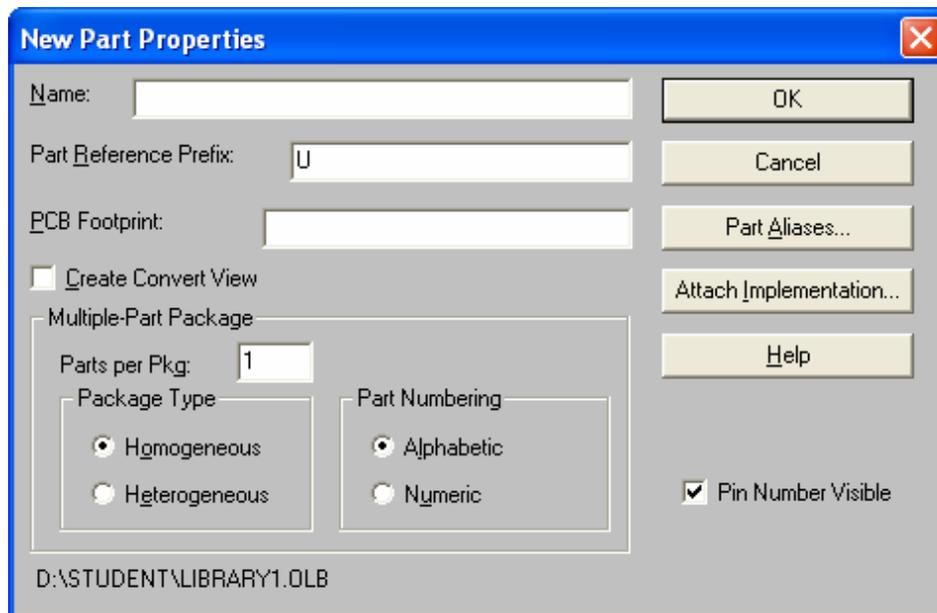
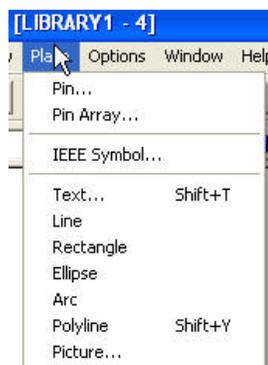


Fig3

The edit window will close after pressing OK.

To define a part we first need to build its body (draw in an outline or inside an outline) and a number of pins (depending on the part's electric properties). The body and the pins of the part can be drawn by using the command Place or the tool palette in the right of the edit window.



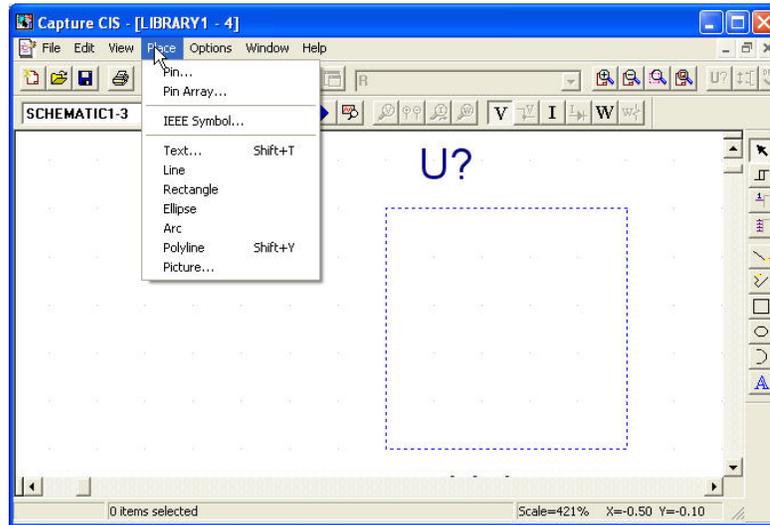


Fig4

If Place->Pin is selected the edit window in figure 5 appears. The properties of a pin are Name, Number, Shape and Type.

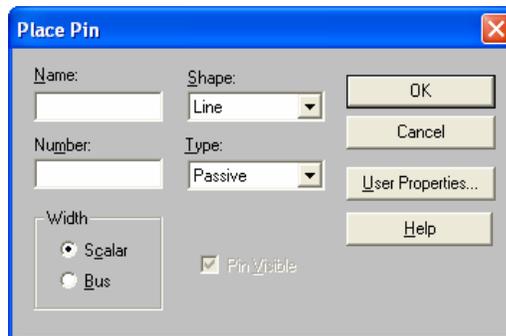


Fig5

**Example**

Edit the symbol of TL494

First step in editing this part is to know its shape, number of pins, their type. Those details can be extracted from the parts PDF.

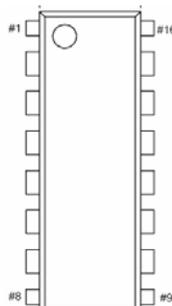


Figure 6

We can observe a capsule containing 16 pins, every pin with its own specific role.

That is why giving a name to every pin is very important. Below we can observe the internal schematic of the capsule and at this moment, the only thing that interests us is the name of every pin.

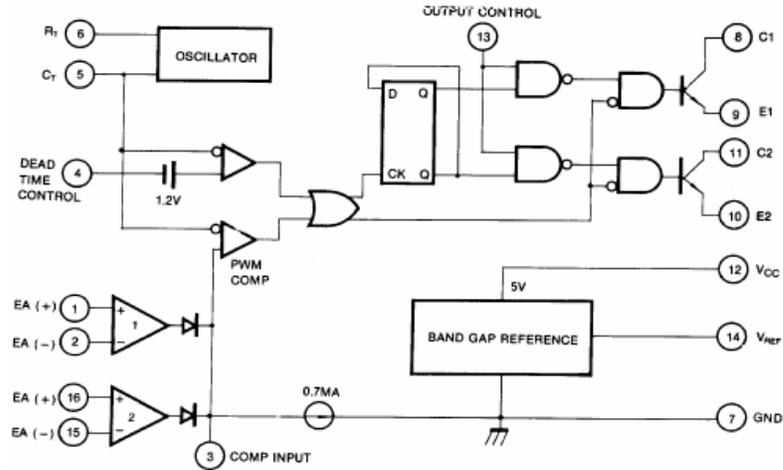


Figure 7

Follow the steps described in figures 1, 2 and 3. Like in figure 3 give the Name of the part. The window from figure 4 will open and we can see the outline of the part, adjust to the wanted size and draw the solid outline.

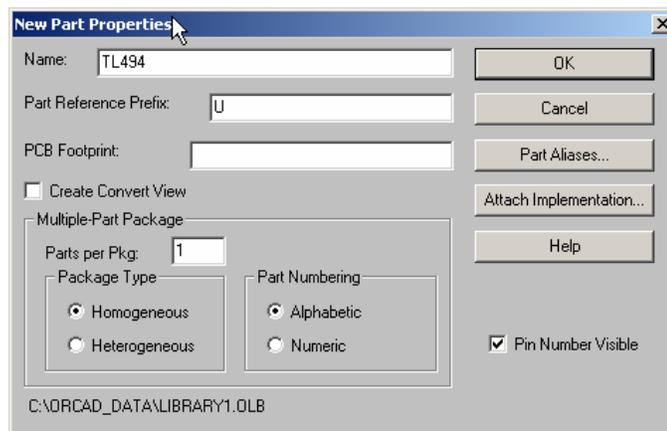


Figure 8

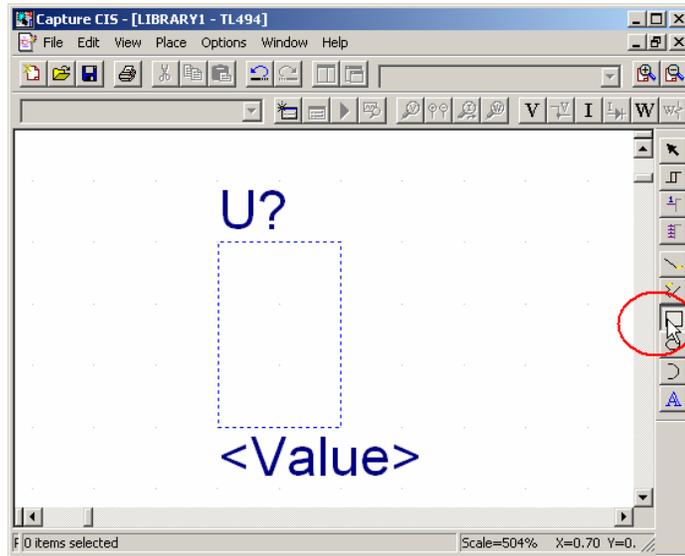


Figure 9

Next step: positioning the pins (figure4 and 5). Give every pin a name and a number. At the end of this step, the part must look like this:

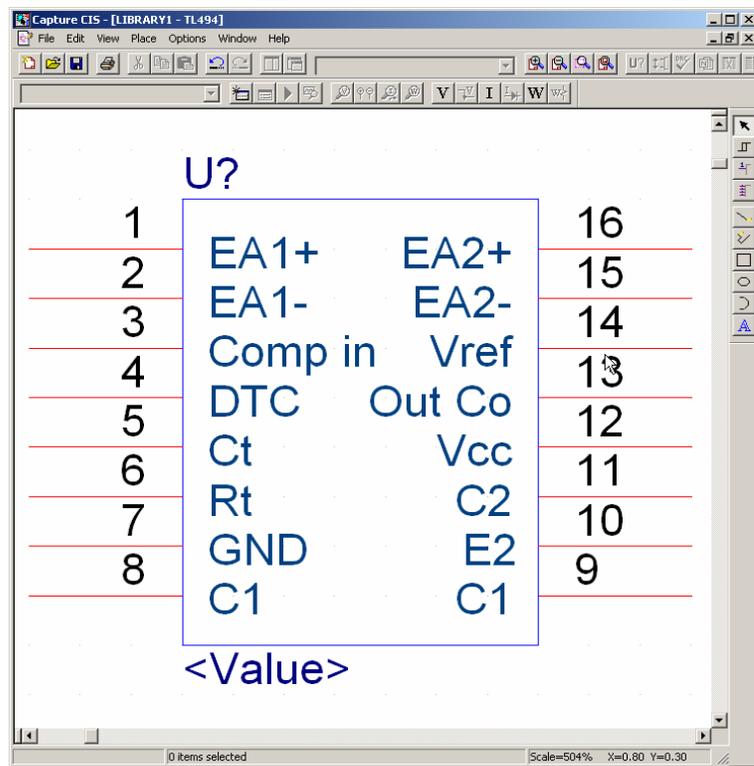


Figure 10

If a certain pin's number or name does not suit us we can change that by double clicking the component and the window bellow will open:

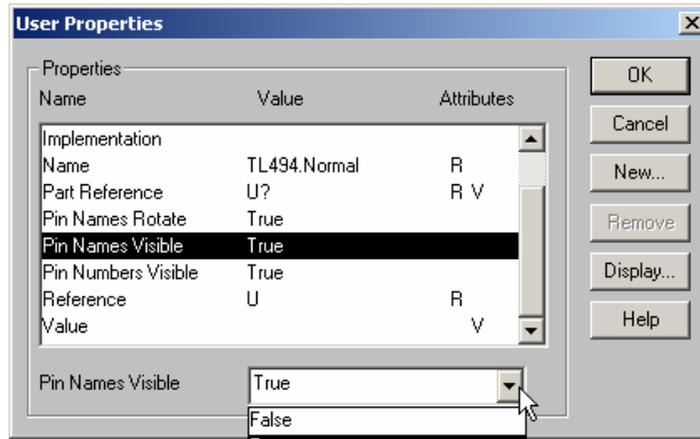


Figure 11

**Homework:**

1. Edit the graphical symbol for the part below:

