# ECE383: Microcomputers – Lab 1
# PCB Artist Introduction

*Goals: The goals of this lab are to introduce students to PCB Artist software to create schematic layouts and generate printed circuit board designs. An additional goal is the creation of a PIC24HJ128GP502 component in PCB Artist.*

## 1. Introduction

This lab introduces students to PCB Artist software. PCB is an acronym that stands for **P**rinted **C**ircuit **B**oard. Students will learn how to design electronic circuits using PCB Artist's schematic tool and then generate a layout of the design using the PCB tool. The final objective will be to design and implement a custom PIC24HJ128GP502 component in PCB Artist.

As always, read through the entire lab and scan the supplied files before starting work. The reporting requirements have you verify your work by uploading screen-captures, photos of hand-work, and answering questions on Blackboard. In all cases, make it easy for the TA to verify your work by showing all work (when necessary), highlighting details in the screenshot to help the TA find them, and make all hand-writing clear and legible.

## 2. TASK 1: Introduction to PCB Artist

For this task we will use video tutorials from Advanced Circuits to introduce PCB Artist and its associated schematic capture capabilities. View the seven step-by-step video tutorials located at http://www.4pcb.com/pcb-software-tips-tools.html. These videos reflect the content of the *PCB Artist Introduction and Tutorial* that can be accessed from the *Help->Tutorials->Design Tutorial* menu within PCB Artist. Complete chapters 1 and 2 of the tutorial. You will be required to show the TA the completed schematic upon entering the lab.

Read the Printed Circuit Board basics tutorial at https://learn.sparkfun.com/tutorials/pcb-basics and answer the following questions on Blackboard.

**Deliverable 1:** Often a PCB is made of four layers of different materials. Draw and label these four layers and describe each of them in your own words. Upload a photo of your drawing (with your name and CWID clearly shown).

**Deliverable 2:** Define the following terms in your own words: a) Trace, b) Pad, c) Panel, d) Plated through hole, e) Silkscreen, f) Soldermask, g) Surface mount, h) Via, and i) Plane.

## 3. TASK 2: 555 Timer Circuit Schematic with PCB Artist

Use PCB Artist to create the digital clock circuit given in Figure 1 below. This circuit includes the following components: a 10KΩ resistor ($R_A$) a 100KΩ resistor ($R_B$), a 1µF capacitor and a 555 timer chip. One possible PCB Artist layout of the circuit is shown in Figure 2. The LM555CJA integrated circuit may be found in the natsemi.cml component library. The capacitors and resistors may be found in the discrete.cml component library. Use the "C" component for the capacitors and the "$R_A$" component for the resistors.

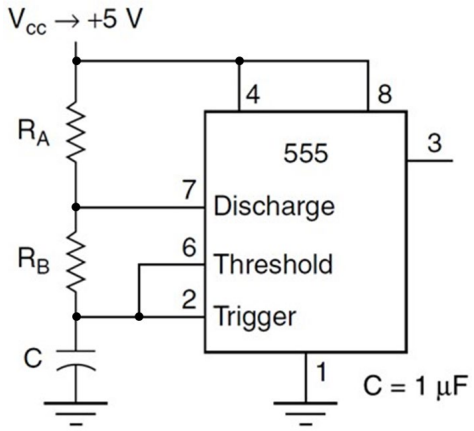


**Figure 1. 555 Timer-Based Digital Clock Circuit**

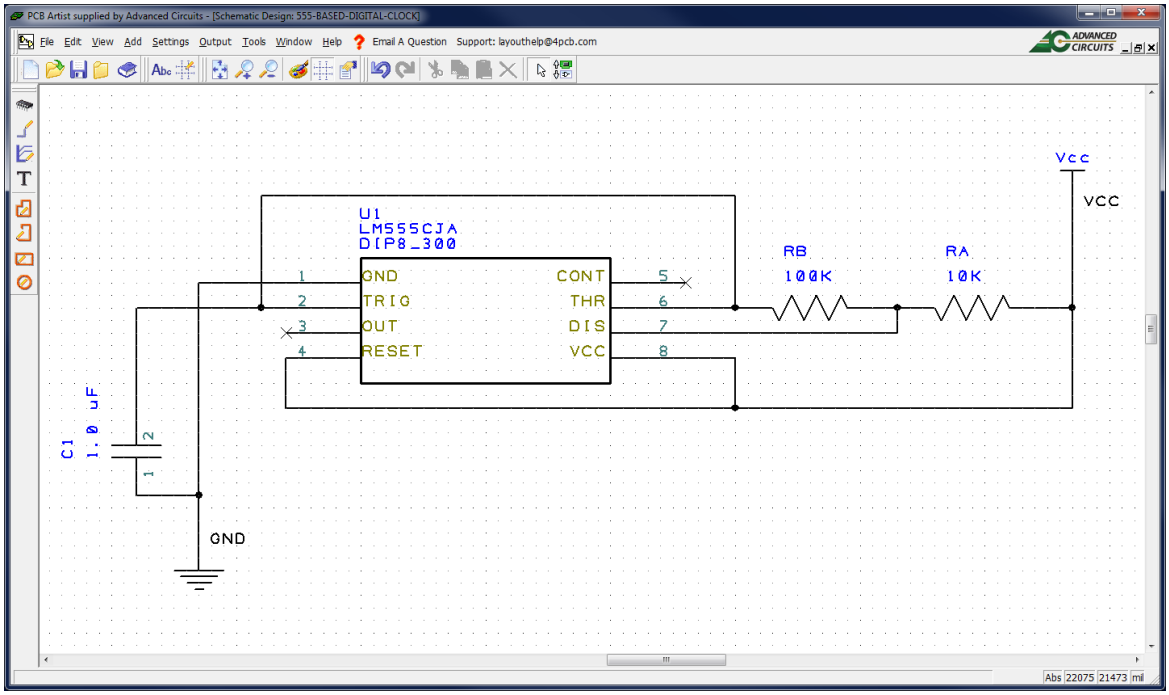


**Figure 2. PCB Artist Layout of 555 Timer-Based Digital Clock Circuit**

**Deliverable 3:** Upload a screenshot of your electrical schematic from PCB Artist for Task 2. Make sure that the names and CWIDs of both group members can be seen (in a text editor window) but do not cover your schematic.

## 4. TASK 3: Creation of a Printed Circuit Board Layout with PCB Artist

For this task we will create a printed circuit board layout from the digital clock circuit schematic created as a part of **Task 2**. Chapter 4 of the *PCB Artist Introduction and Tutorial* describes the basic process of converting the circuit schematic into a printed circuit board layout. Read from the beginning of chapter 4 to the section titled "**Unrouting the design**". Use the described process using the **Schematic To PCB** wizard within PCB Artist. You can accept all the default choices provided by the wizard until the "Place and Route" dialog box is shown. Select "Arrange Outside the Board" for component placement and select "Next". This will result in a PCB that will be similar to the diagram shown in Figure 3 below.

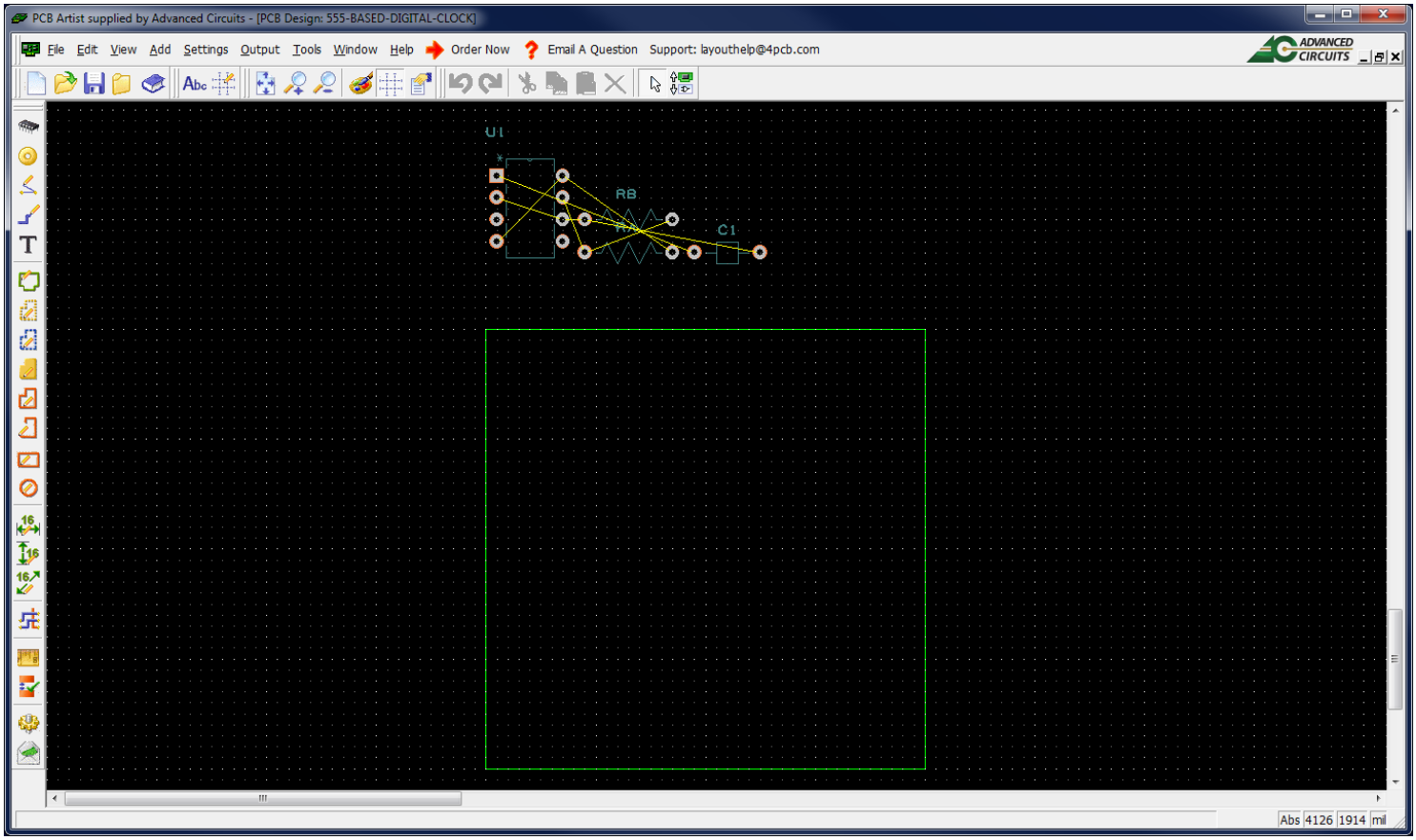


**Figure 3. PCB Wizard Generated Design with Components Unplaced and Nets Unrouted**

Select *Tools->Auto Place Components->All Components* to have PCB Artist auto-place all the circuit components onto the PCB. Select *Tools->Auto Route Nets->All Nets* to have PCB Artist auto-route all the circuit networks (i.e. connections between components) on the PCB. This will result in a printed circuit board similar to the one shown in Figure 4 below.

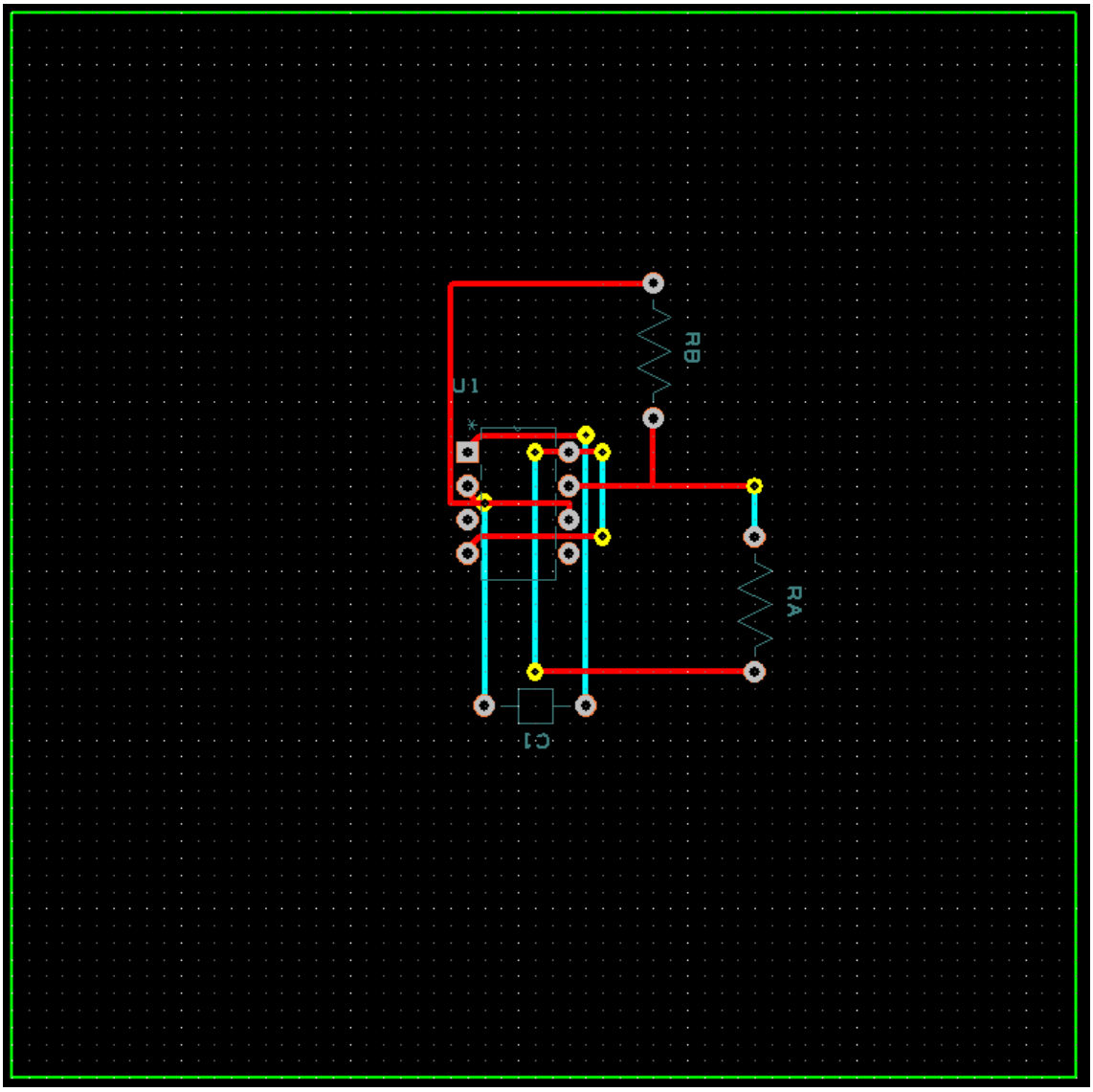


**Figure 4. Printed Circuit Board with all Components Placed and all Nets Routed**

**Deliverable 4:** Upload a screenshot of your printed circuit board layout from PCB Artist for Task 3. Make sure that your name and CWID are visible (in a text editor window) but do not cover your layout details.

## 5. TASK 4: PIC24HJ128GP502 Schematic Symbol, PCB Footprint, and Component in PCB Artist

For this task you will create a component for the PIC24HJ128GP502 microcontroller in PCB Artist. Begin by reading and completing the Library Creation Tutorial within PCB Artist (*Help->Tutorials->Part Creation Tutorial*). After completing the tutorial, perform the following steps:

- Within the Library Manager, create a new schematic symbol library called ece383.ssl. Use the Symbol Wizard tool to create a schematic symbol called PIC24HJ128GP502. Example screen captures for this wizard are shown below.

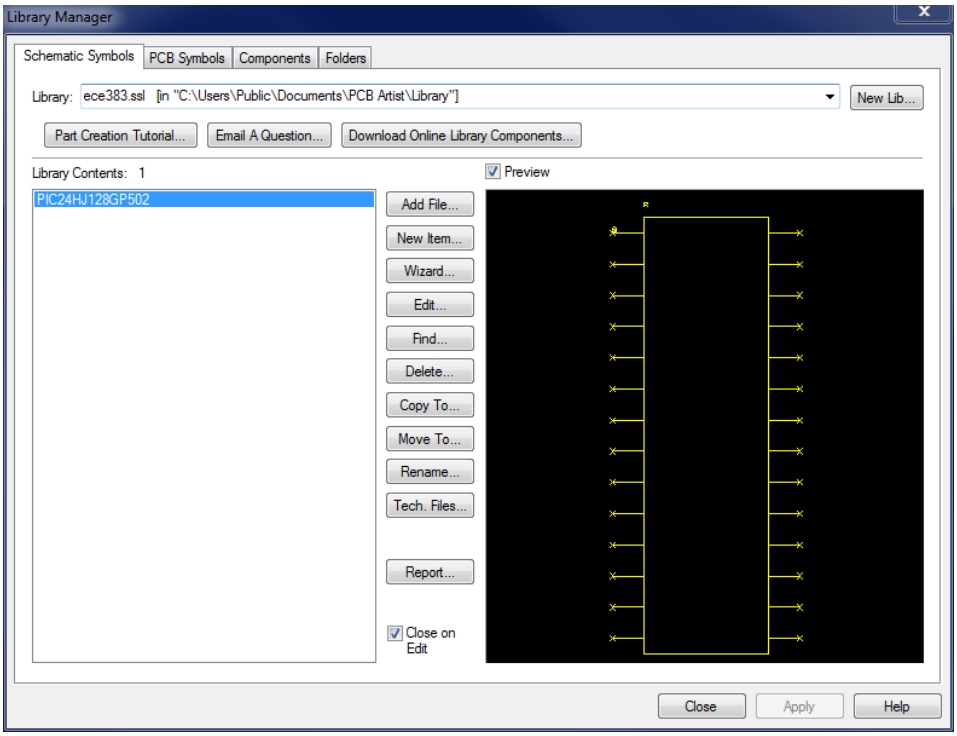


**Figure 5. Click New Lib … button to make ece383 folder**

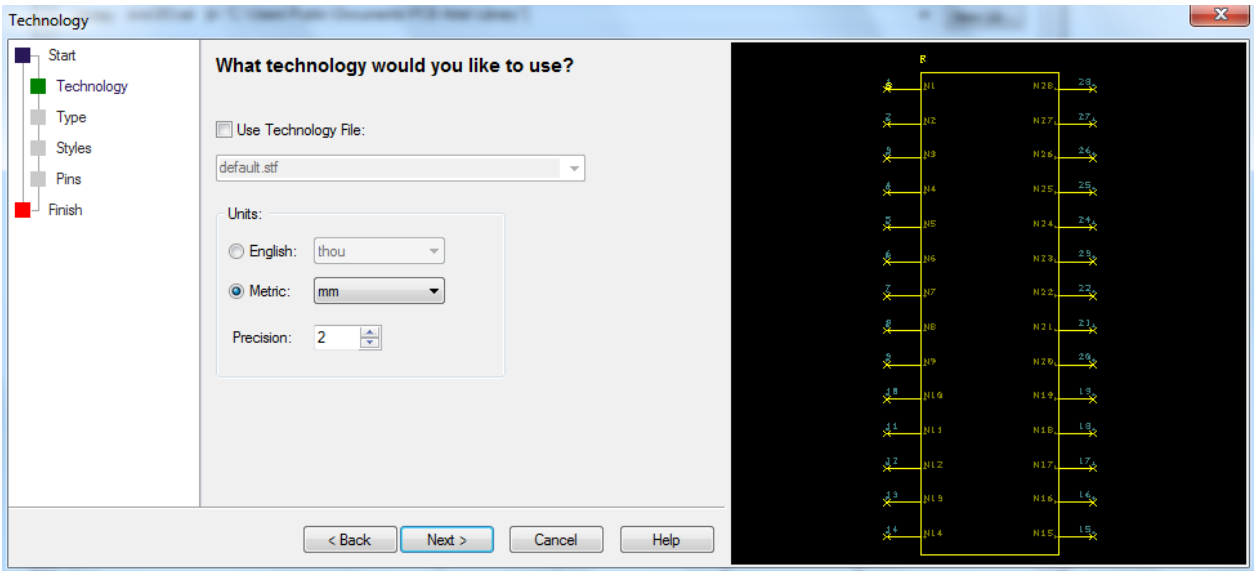


**Figure 6. Technology and Unit Selection**

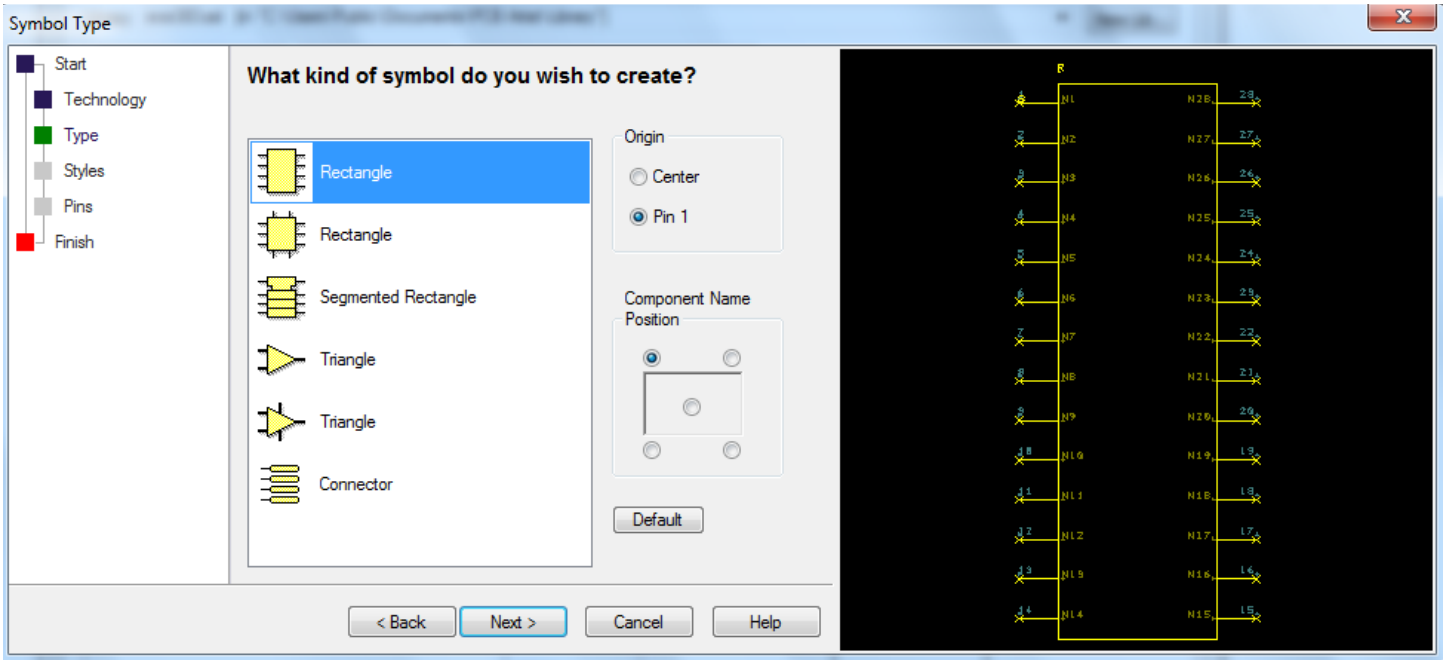


**Figure 7. Symbol Type, Origin, and Component Name Position**

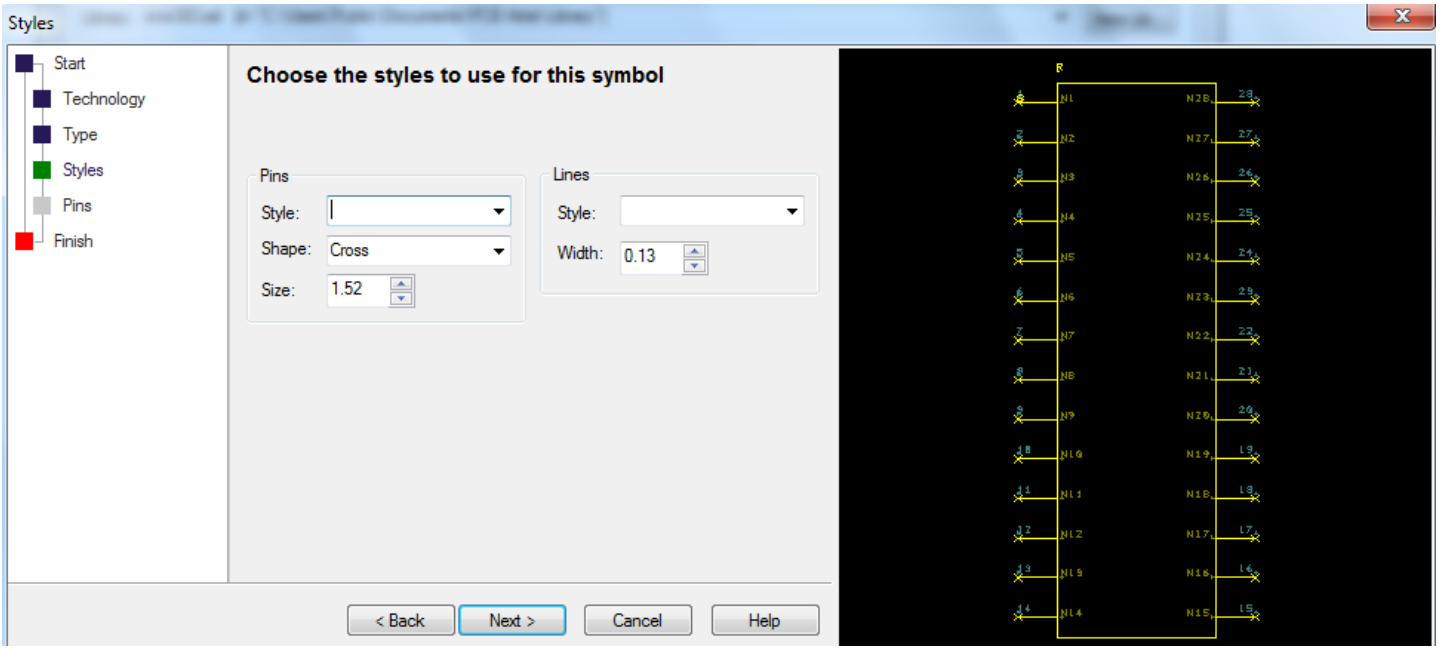


**Figure 8. Symbol Pin and Line Styles**

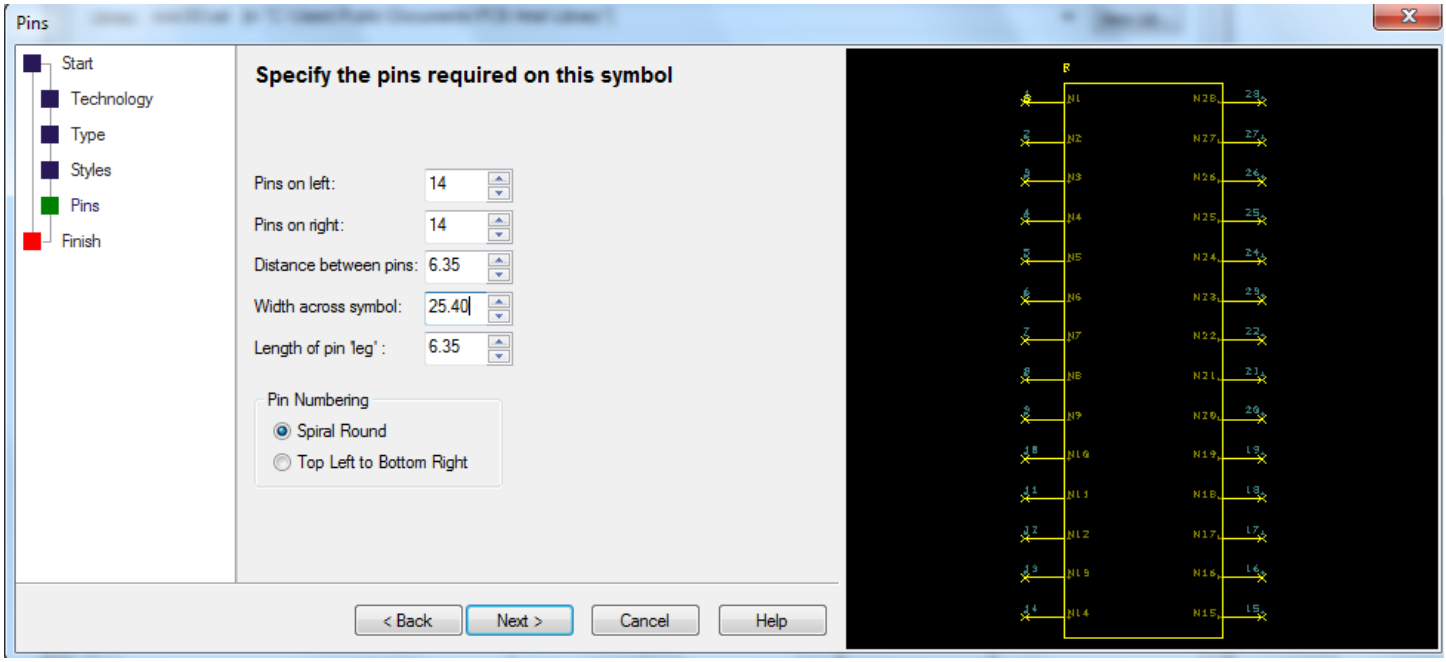


**Figure 9. Pin Layout**

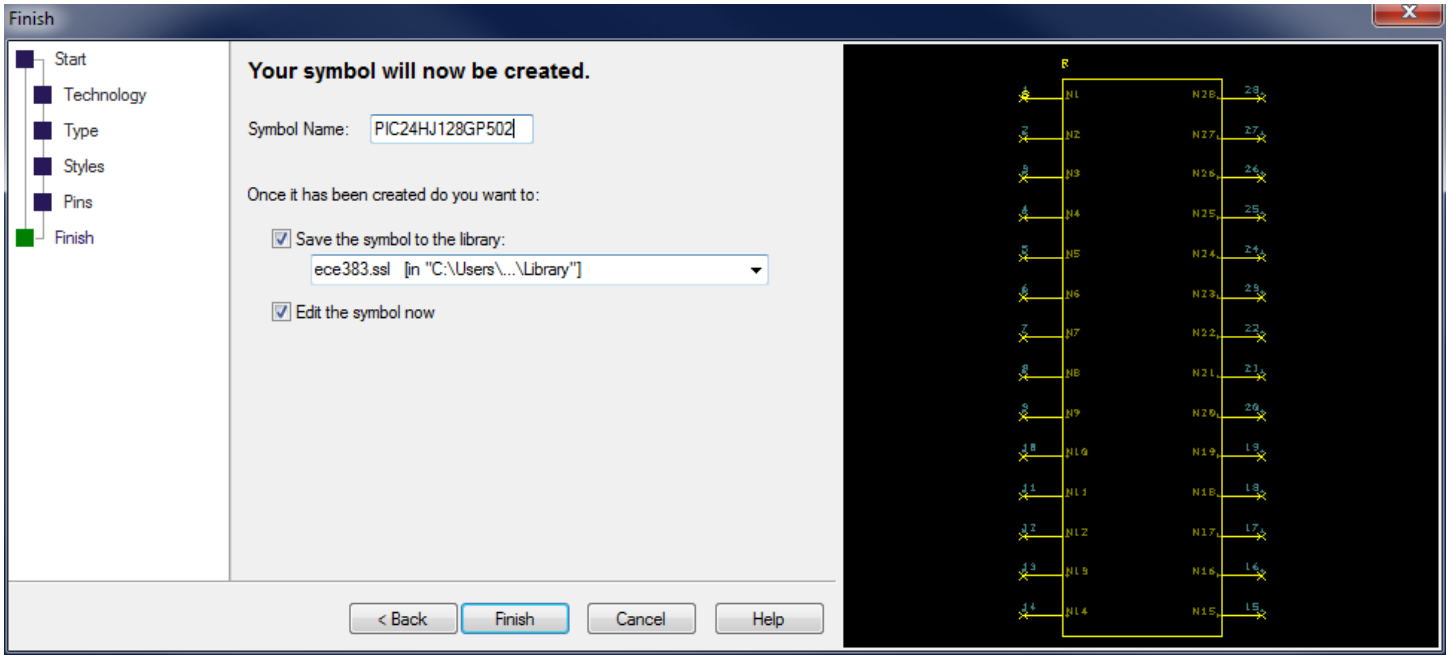


**Figure 10. Symbol Naming and Final Creation Step**

- Within the Library Manager, create a new component library called ece383.cml. Use the Component Wizard tool to create a component called PIC24HJ128GP502. Example screen captures for this wizard are shown below.

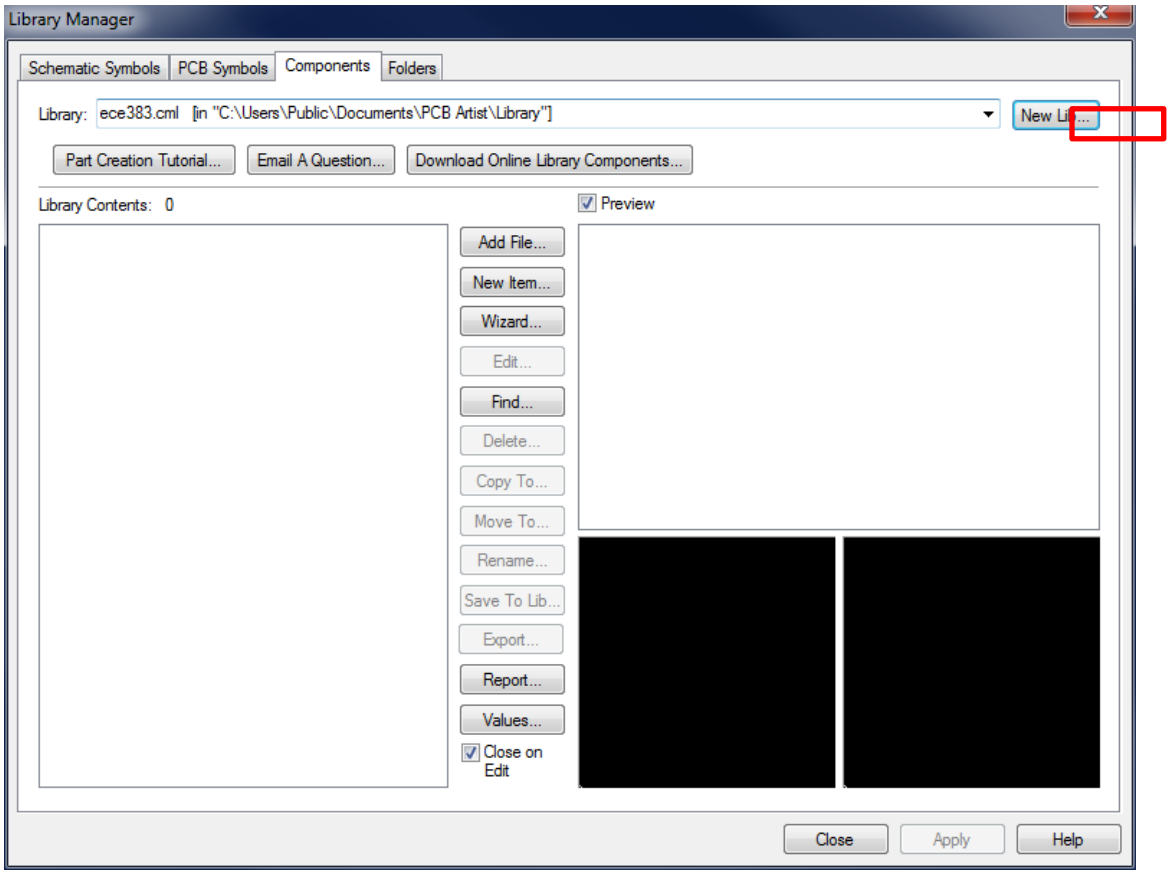


**Figure 11. Click New Lib… to make ece383.cml file in components tab**

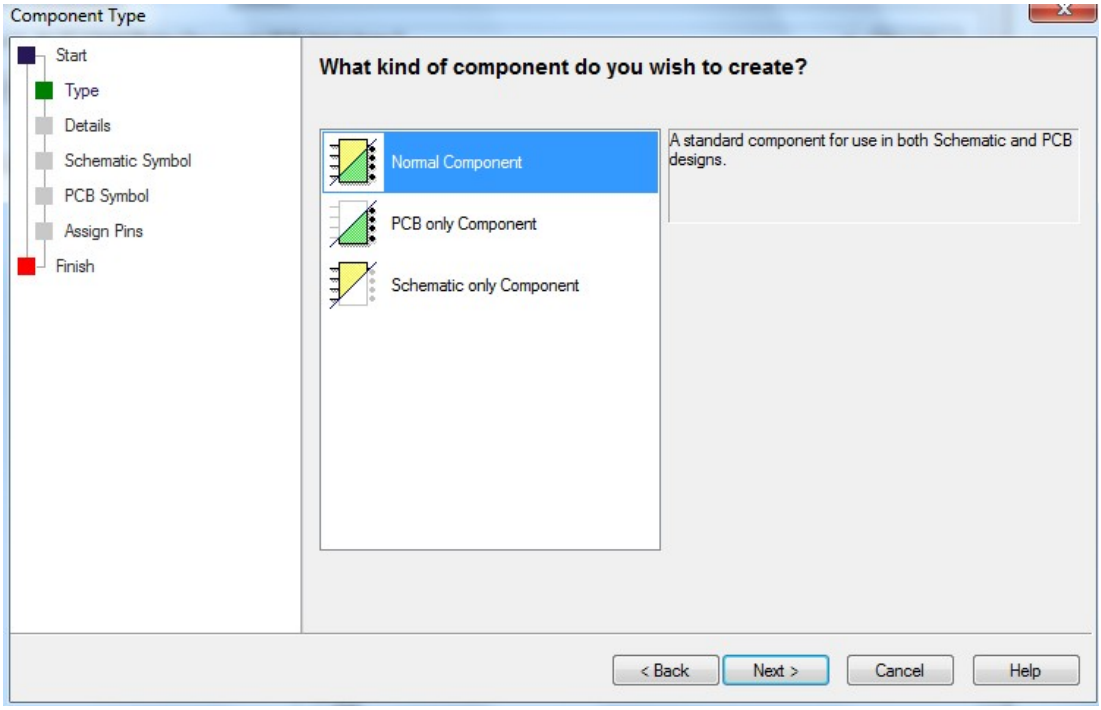


**Figure 12. Create a Normal Component Type**

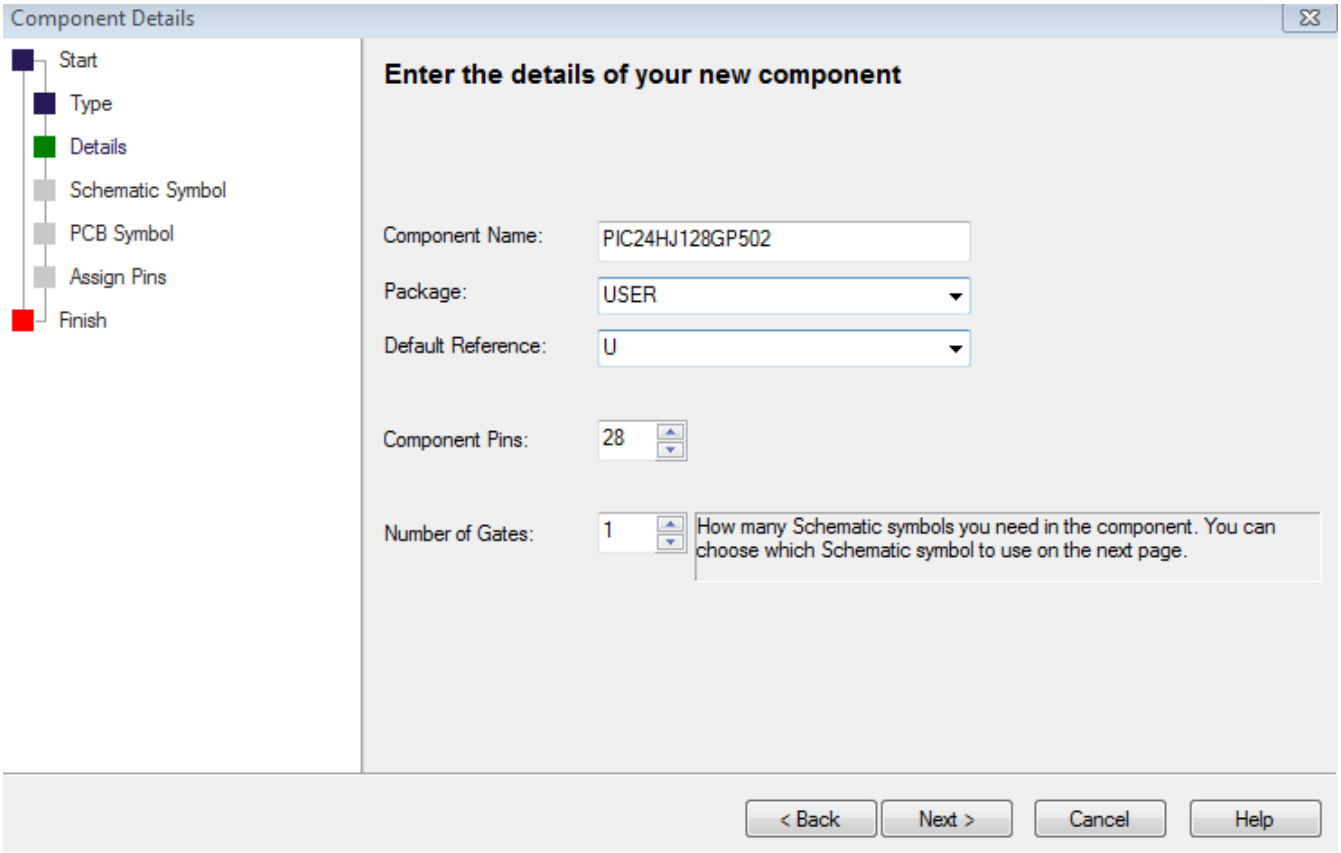


**Figure 13. Component Details**

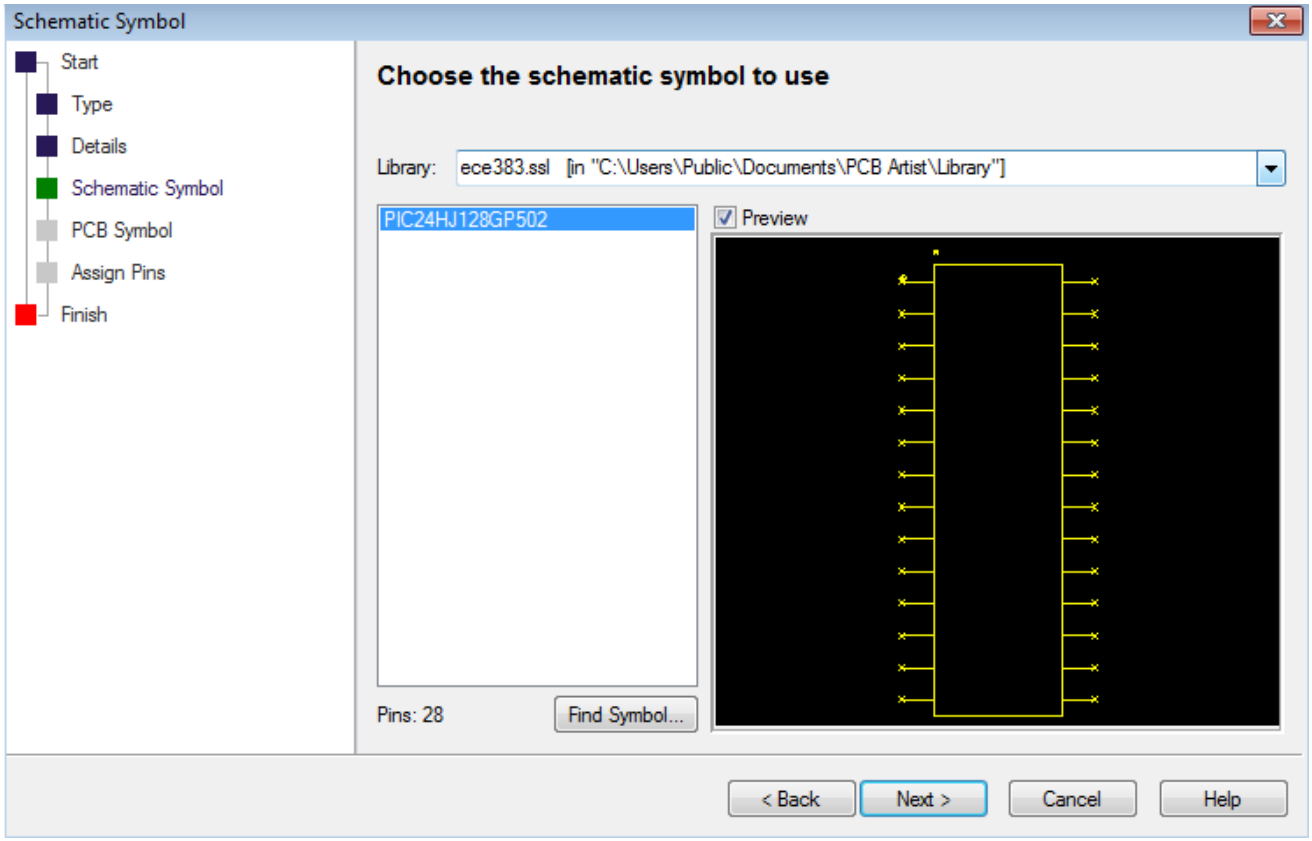


**Figure 14. Schematic Symbol to Use**

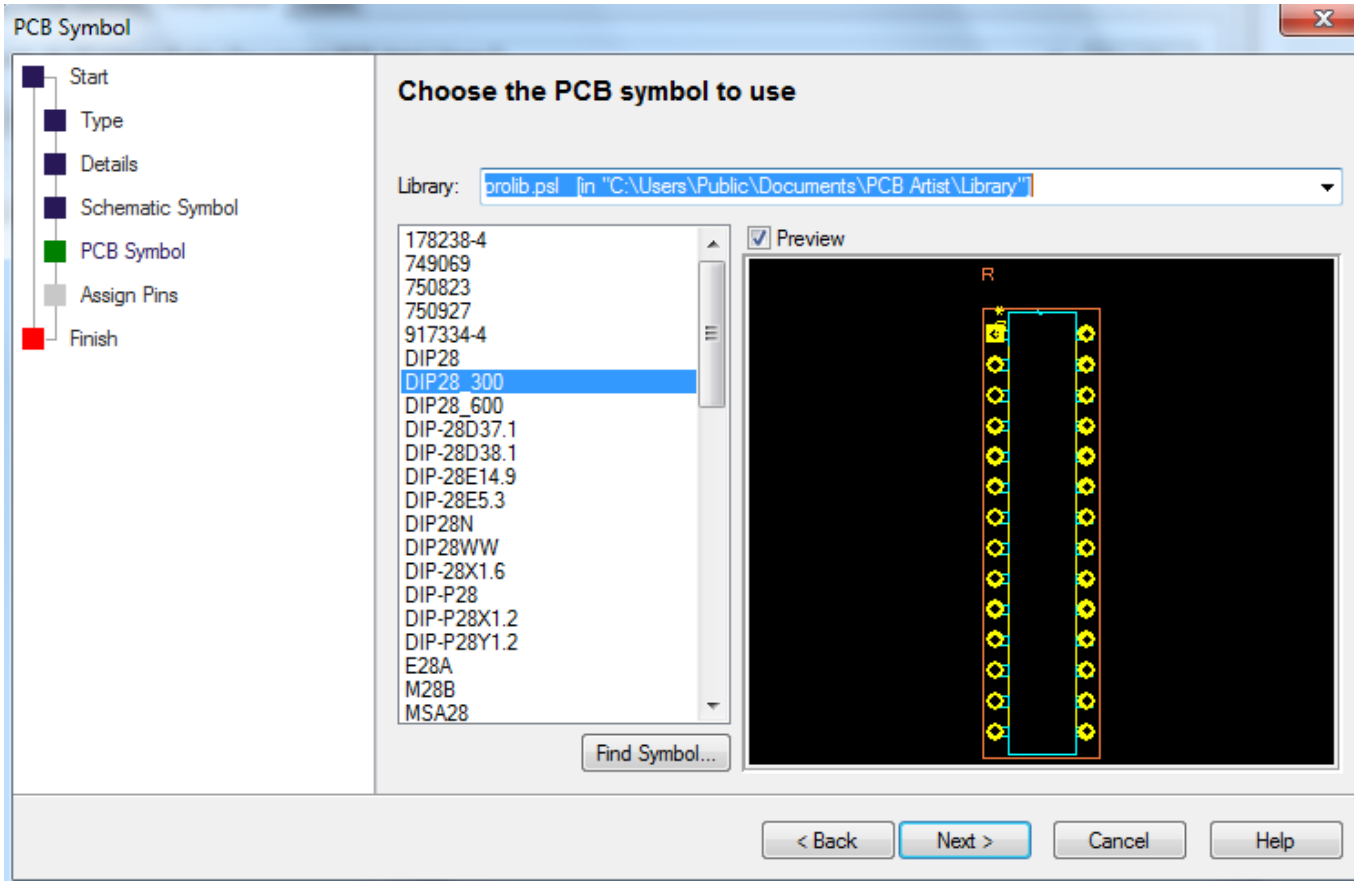


**Figure 15. PCB Symbol to Use (DIP28_300 from the prolib.psl Library)**

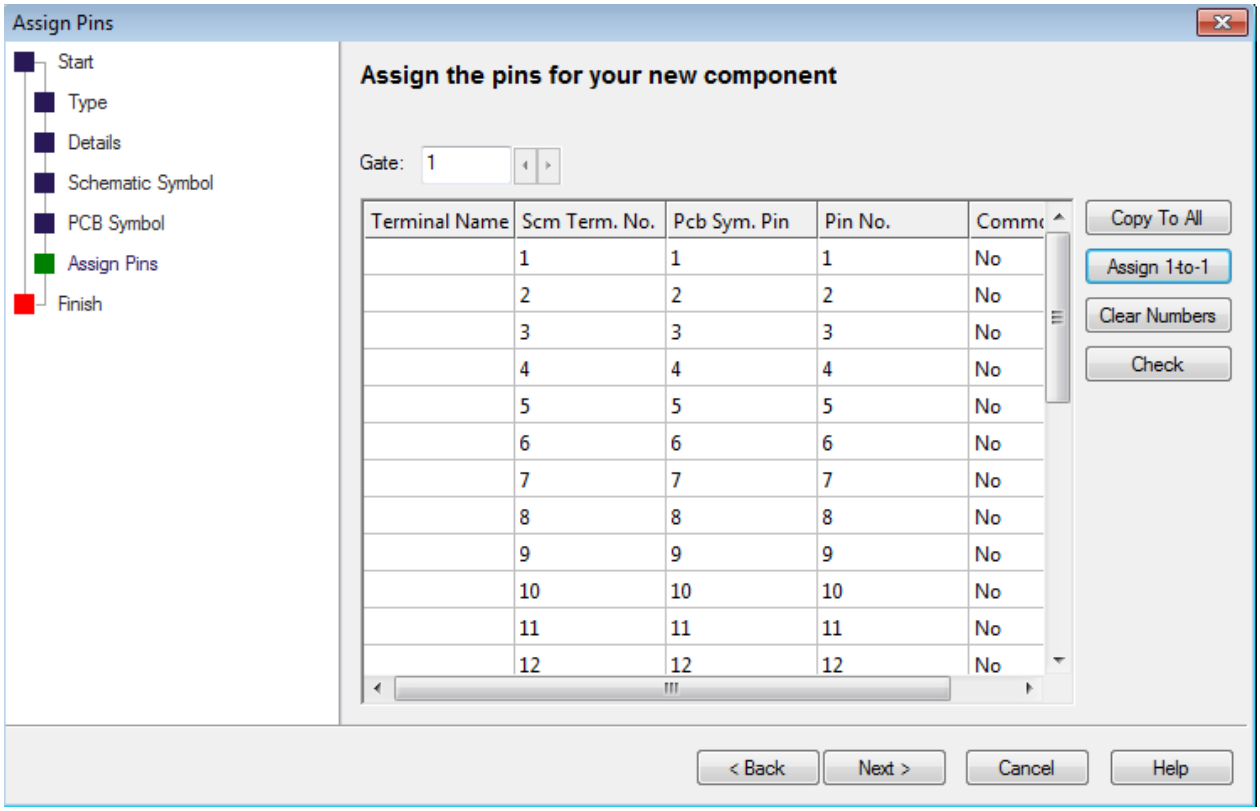


**Figure 16. Assign Pins (Use the Assign 1-to-1 Function)**

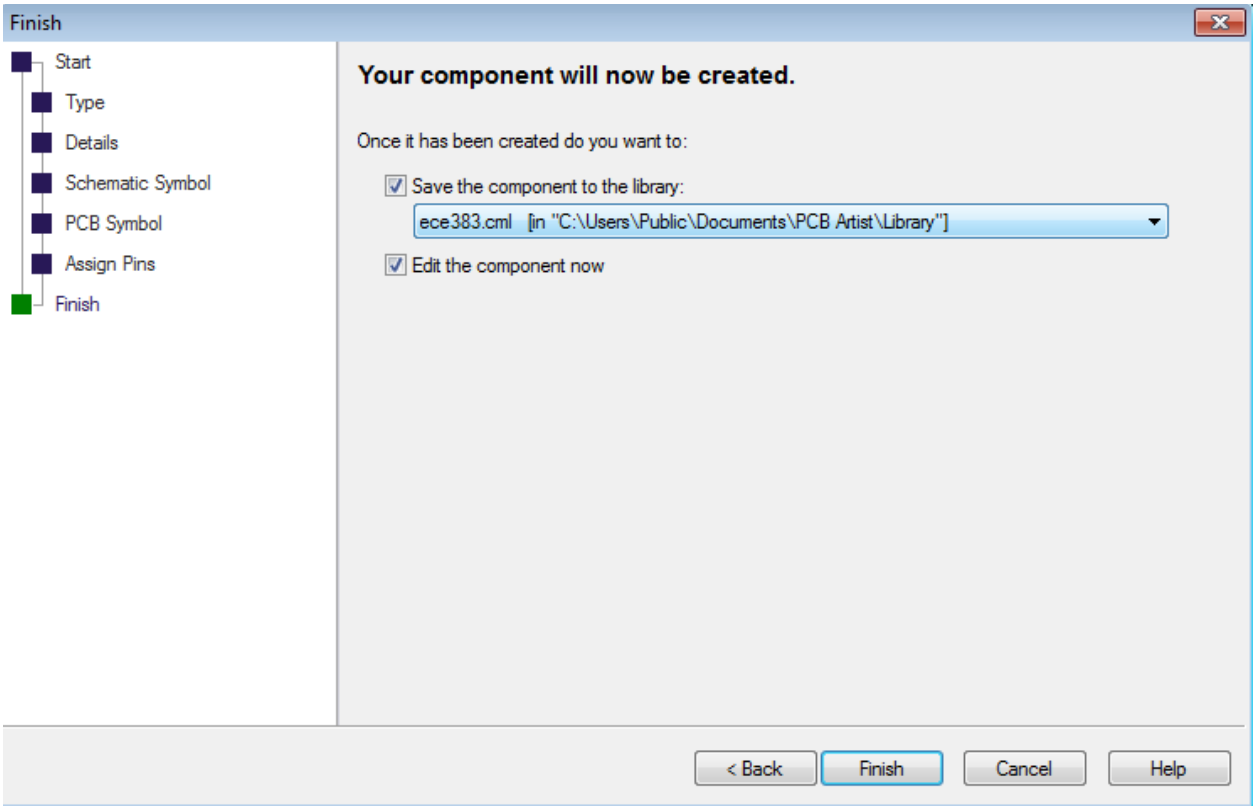


**Figure 17. Save the Component in the Component Library**

- Edit the component and provide terminal names for each pin as shown in the figure below. The terminal names will closely match those given in the PIC24HJ128GP502 diagram shown on page 4 of the PIC24 datasheet on the class website. Save the edited component in the component library.
- Close all the windows. Click either F8 or Add Component button. Find the “ece383.cml" library. Select the PIC24HJ128GP502 and click add. The placed component should resemble that shown in Figure 19.

| Gate Name | Sch Symbol Name | Sch Symbol Terminal Name | Sch Terminal Number | Pcb Symbol Pad Number | Component Pin Name/Number | Net (Class) Name |
|---|---|---|---|---|---|---|
| a | PIC24HJ128GP5( | MCLR# | 1 | 1 | 1 | |
| | | RA0 | 2 | 2 | 2 | |
| | | RA1 | 3 | 3 | 3 | |
| | | RB0 | 4 | 4 | 4 | |
| | | RB1 | 5 | 5 | 5 | |
| | | RB2 | 6 | 6 | 6 | |
| | | RB3 | 7 | 7 | 7 | |
| | | VSS | 8 | 8 | 8 | |
| | | RA2 | 9 | 9 | 9 | |
| | | RA3 | 10 | 10 | 10 | |
| | | RB4 | 11 | 11 | 11 | |
| | | RA4 | 12 | 12 | 12 | |
| | | VDD | 13 | 13 | 13 | |
| | | RB5 | 14 | 14 | 14 | |
| | | RB6 | 15 | 15 | 15 | |
| | | RB7 | 16 | 16 | 16 | |
| | | RB8 | 17 | 17 | 17 | |
| | | RB9 | 18 | 18 | 18 | |
| | | VSS | 19 | 19 | 19 | |
| | | VCAP | 20 | 20 | 20 | |
| | | RB10 | 21 | 21 | 21 | |
| | | RB11 | 22 | 22 | 22 | |
| | | RB12 | 23 | 23 | 23 | |
| | | RB13 | 24 | 24 | 24 | |
| | | RB14 | 25 | 25 | 25 | |
| | | RB15 | 26 | 26 | 26 | |
| | | AVSS | 27 | 27 | 27 | |
| | | AVDD | 28 | 28 | 28 | |

**Figure 18. Component Terminal/Pin Naming**

**Deliverable 5:** Add the created component to a schematic design and take a screenshot of the component placement within the schematic (an example is given below). Upload a screenshot of this Task 4 schematic to Blackboard. Make sure that your name and CWID are visible (in a text editor window) but do not cover your layout details.

**Deliverable 6:** While you used PCB Artist for the electrical schematic and printed circuit board layout, search for and provide the names of 2 other commercially available CAD programs that can also do this.