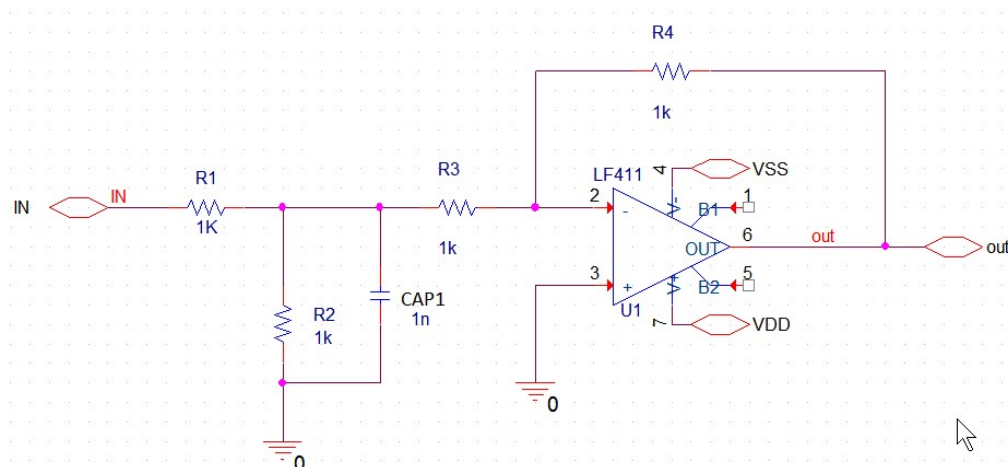## Lab 13-1   Converting a Buffer Circuit to a Subcircuit

Objective: To convert a schematic to a subcircuit.

In this lab, you convert a schematic to a subcircuit.

Creating Buffer2 Project

1. Create a new project based on *Buffer.opj*. Name it *Buffer2.opj*.

2. Delete the VSTIM voltage source.

3. Remove the entire piece of circuitry that includes V2 and V3.

4. Delete the load resistor and capacitor at the output of the LF411.

5. Delete the off-page connectors on VDD and VSS.

6. Add a hierarchical port **portboth-r** left of R1.

7. Add the hierarchical port **portboth-l** to the output node of the LF411 and on the V+ and V- pins of the LF411.

8. **Double**-click each hierarchical port and label it as shown here.

9. Save the design.

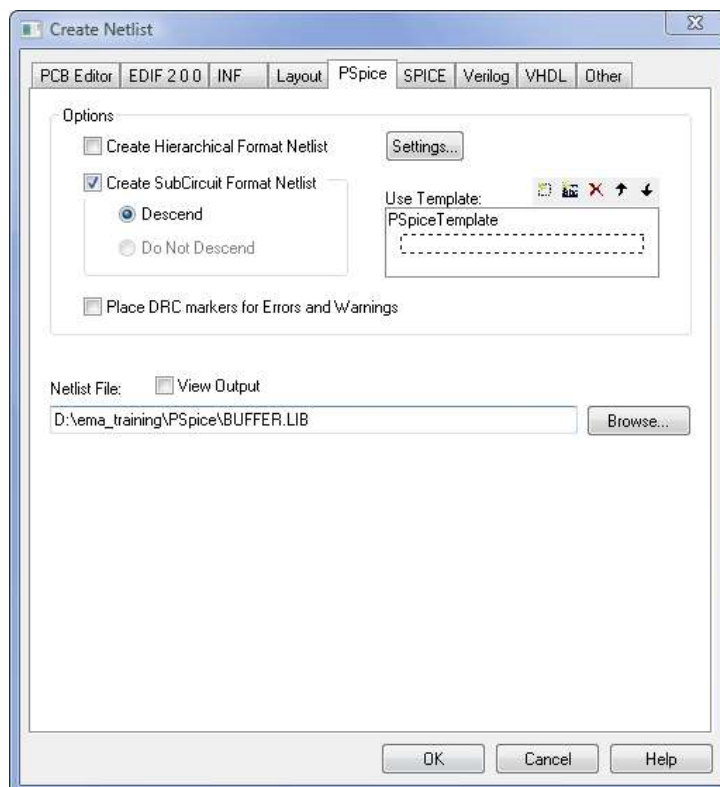**Renaming Schematic Folder to Create Subcircuit**

1. Click the **Project Manager** button, [icon] , then select the **DSN** file.

2. Change the name of the schematic folder to *buffer*.

   Note: The name of the subcircuit is created from the schematic name. If you do not change the name of the schematic folder, the resulting subcircuit is called *schematic1*.

Creating a Netlist

1. Choose **Tools – Create Netlist**.

2. Click the **PSpice** tab.

3. Select the **Create Subcircuit Format Netlist** checkbox.

4. Change the path of the library file to your training directory so that you can find it later. Click **OK**.

5. To view the resulting subcircuit, expand the **Outputs** folder in the OrCAD Capture Project Manager. **Double**-click the library file.

Note: Your listing's pin order and node names can be different from the following example.

Subcircuit Listing

```
* source BUFFER2
.SUBCKT Buffer IN OUT VDD VSS
X_U1          0 N00506 VDD VSS OUT LF411
R_R1          IN N00354  1k
R_R2          0 N00354  1k
R_R3          N00506 N00354  1k
R_R4          OUT N00506  1k
C_CAP1        0 N00354  1n
.ENDS Buffer2
```

Lab Summary

In this lab, you

◆ Converted a schematic to a subcircuit

End of Lab

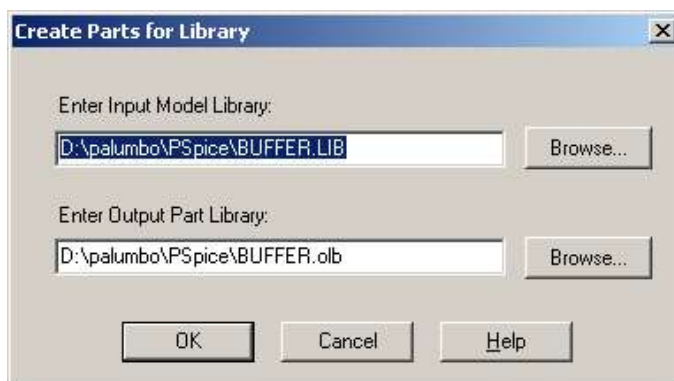## Lab 13-2   Creating a Part with a Wizard

Objective: **To use the Symbol Creation Wizard and create a part for a subcircuit.**

In this lab, you

- Use the Symbol Creation Wizard

- Create a part for the subcircuit created in the previous lab

Creating a Library Part

1. To run the Model Editor, choose **Start – All Programs – Cadence Release 17.2-2016 – Product Utilities – PSpice Utilities – Model Editor**.

2. Select **Capture** and click **Done**.

3. Choose **File – Open.** Navigate to the *buffer.lib* file and click **Open**.

    This file is saved in a folder located in the *<class_dir>/PSpice* directory.

    The Model Editor opens.

4. Choose **File – Export to Part Library** and browse, again, to find your *buffer.lib* file and set the Output Part Library, if necessary.
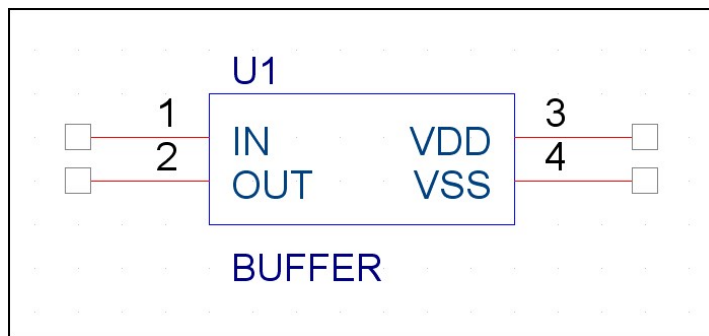

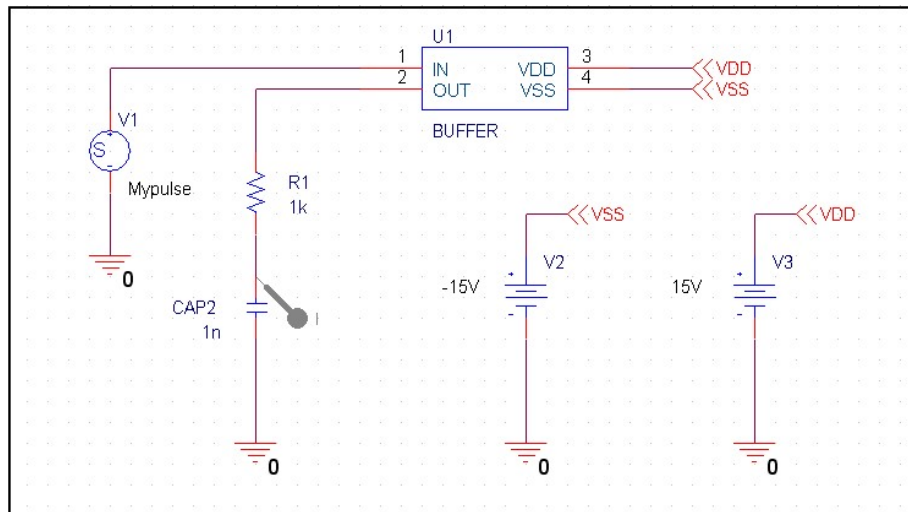5. Click **OK** to create the part. Close the PSpice® Model Editor.

Viewing the Part

1. Return to OrCAD Capture.

2. Create a new project called **Buffer3.opj** based on *empty_all_libs.opj*.

3. Choose **Place – Part**.

4. Click the **Add Library** button. Configure **Buffer.olb**.

   Note: Make sure you use the file from the training directory. There is a *buffer.olb* file in the regular PSpice library folder, but that is not the one you want to use.

5. In the list of parts, **double**-click **Buffer**.
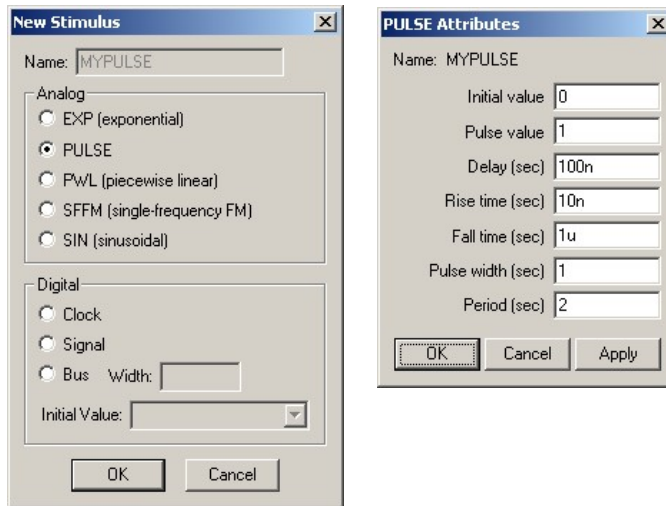
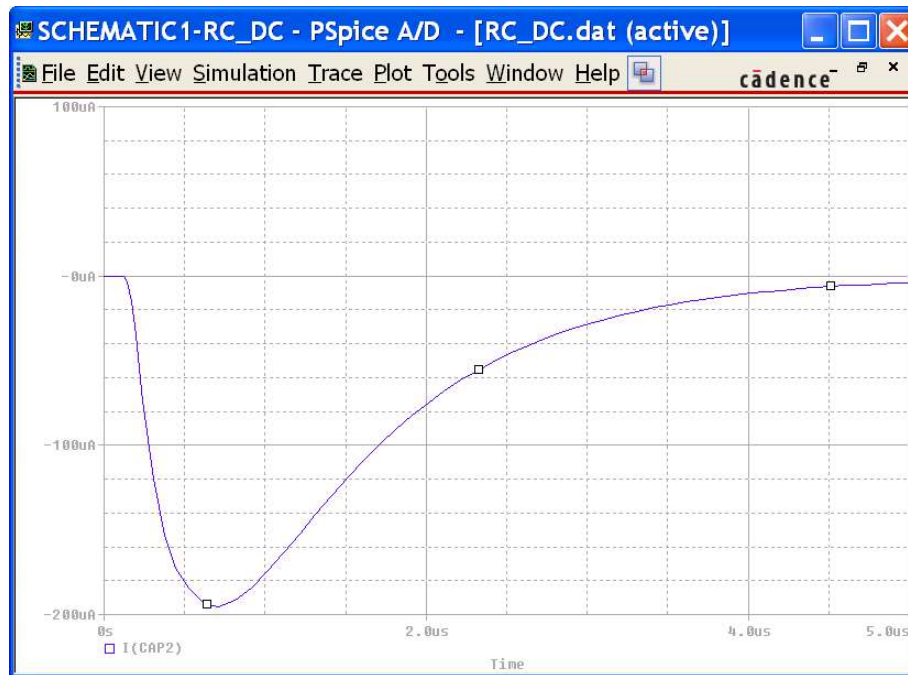6. Place an occurrence of the part.

Using the New Symbol

1. Using the *Buffer3* project, draw this circuit.

Setting Up the Stimulus

1. **Right**-click **V1** and select **Edit PSpice Stimulus**. Set up the MYPULSE stimulus as shown.

2. From the Stimulus Editor, choose **File – Save**. Click **Yes** to update schematic.

3. Choose **File – Exit** from the Stimulus Editor.

4. Configure a simulation profile for a transient analysis that runs for 5uS.

5. In the simulation profile, under the **Configuration Files** tab, select **Library** in the **Category** section. Add *Buffer.lib* to the design.

6. Click **OK** to close the simulation profile dialog box.

7. Run the analysis. Examine the result in the Probe window. It should look like the results in the transient chapter, which are reproduced here.

Lab Summary

In this lab, you

- Used the Symbol Creation Wizard

- Created a part for the subcircuit created in the previous lab

End of Lab

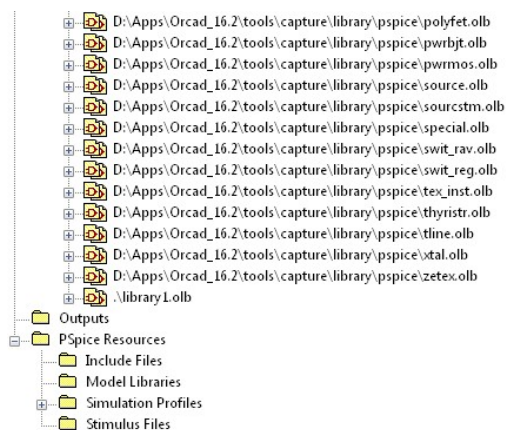## Lab 13-3   Creating a Part from the Beginning

Objective: **To create a new five-pin part for an operational amplifier and configure libraries.**
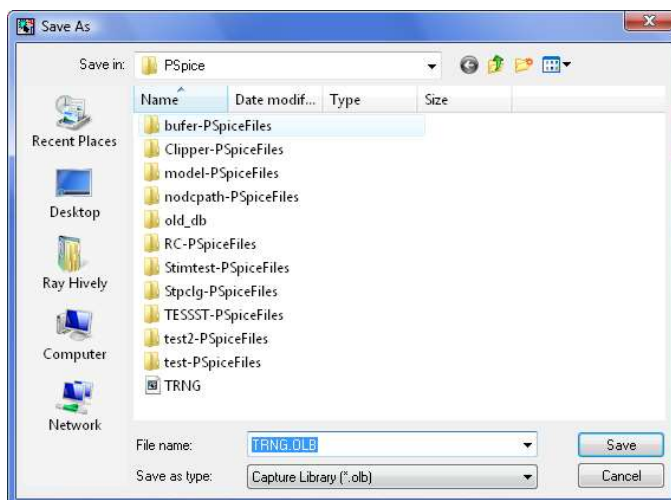
Creating a New Library File

To complete the steps that follow, continue using the *Buffer3* project you created in Lab 12-2.

1. From the Project Manager window, choose **File – New – Library** to create a new library file.
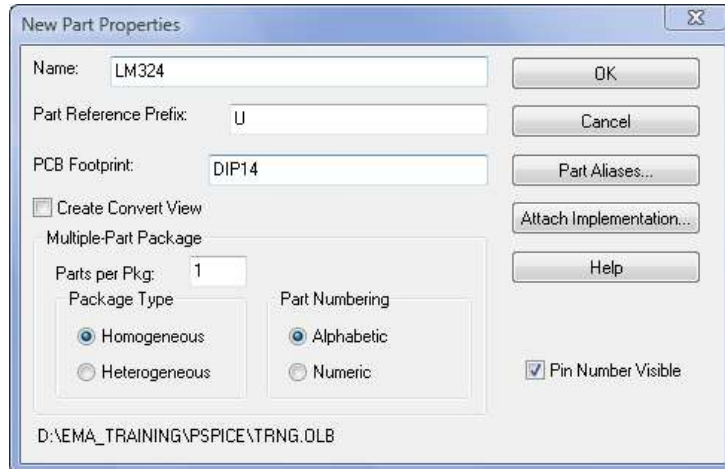
   The new library file, *library1.olb,* is listed in the Project Manager window as shown here.
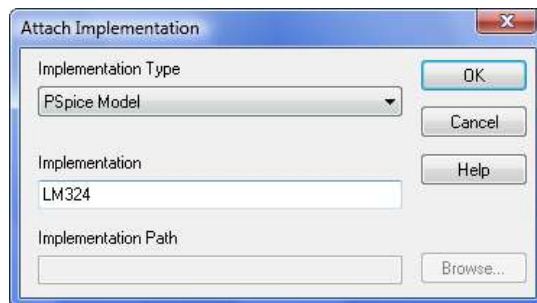

2. Select *library1.olb*. **Right**-click and select **Save As**.

3. Name the library *trng.olb*. Save it in the training directory.

4. Select *trng.olb.* **Right**-click and select **New Part**.

5. Name the new part **LM324**. Configure the properties as shown here.

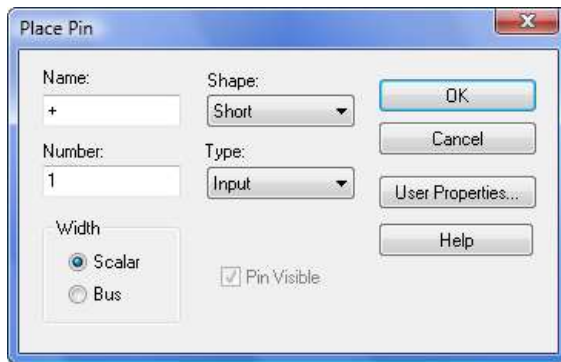6. Click the **Attach Implementation** button. Configure the dialog box as shown here.

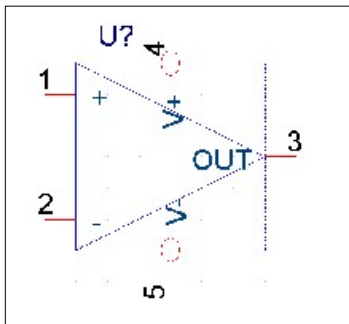7. Click **OK** to accept the settings. Click **OK,** again, to open the Part Editor.

Drawing the Graphics

1. Choose **Place – Line,** *or* click the icon to draw the body of the part. The width and length of the part is six grid spaces.

2. Click the **Place Pin** icon.

   The Place Pin dialog box opens.

3. Define pin number 1 as an input pin, as shown in the following.

4. Place five pins around the body as shown in the following.

- Define pin number 2 as an input pin, named **-** (minus character).
- Define pin number 3 as an output pin named **OUT**.
- Define pin number 4 as a **power** pin named **V+**.
- Define pin number 5 as a **power** pin named **V-**.

Note: If necessary, you can edit a pin definition: **Double**-click the pin and modify the values in the Pin Properties dialog box.

5. **Double**-click pin **V+** and set its **shape** to **Zero Length**.

6. Select the **Pin Visible** checkbox.

7. Repeat steps 4 and 5 for the V- pin.

8. Turn off the snap-to-grid feature.

9. Draw additional lines to connect the power pins to the body of the part.

Editing the Properties

1. Choose **Options – Part Properties** to open the User Properties dialog box.

2. Click the **New** button to open the New Property dialog box.

3. Enter **PSpiceTemplate** in the Name field.

   Note: There is no space between *PSpice* and *Template*.

4. Enter **X^@REFDES %+ %- %V+ %V- %OUT @MODEL** in the Value field.

   Note: There are spaces before and after each pin reference.

5. Click **OK** to close the New Property dialog box.

6. Click **OK** to close the User Properties dialog box.

7. Save the library file.

Using the Part in a Design

1. Either open *buffer.opj* and replace the LF411 with your **LM324,** *or* redraw the circuit with the new symbol.

2. Run the simulation (transient run to 5uS).

3. Examine the results in the Probe window.

End of Lab