1.C.1. Changing the Temperature of the Simulation

In the last simulation we found the diode voltage and current at the default temperature of 25 °C. Suppose we want to simulate the circuit at a different temperature? This can easily be done by selecting the temperature option in the simulation profile. We will continue with the circuit of the previous simulation:

We now wish to edit the simulation profile. Since we are using a previously created profile, we need to open the profile rather than create a new profile. Select PSpice and then Edit Simulation Profile from the Capture menus to edit the existing profile:

The Bias Point simulation is already selected from the previous example. Here we need to specify the temperature of the simulation. By default, all simulations are run at 25 °C. To specify a temperature other than the default, select the Temperature (Sweep) option:
The **Temperature (Sweep)** allows us to specify a single temperature or a list of temperatures. If you specify a list of temperatures, the simulation will be run several times, once for each temperature you place in the list. We only need to run the simulation once, so fill in the dialog box as shown:

![Temperature (Sweep) enabled.](image1)

![Simulation will run at 100 °C.](image2)

We have specified the simulation to run at 100 °C. The simulation of the previous simulation did not specify a temperature and the simulation ran at the default temperature of 25 °C. Notice in the screen capture above that there is a checkmark in the box next to the **Temperature (Sweep)** option. The temperature sweep will not run if this box is not checked. The option above specifies that the **Bias Point** simulation will run at 100 °C. Click the **OK** button to return to OrCAD Capture.

Select **PSpice** and then **Run** to run the simulation and then display the diode current and voltage on the schematic: