PSpice Schematic Creation

ABSTRACT

The purpose of this FAQ is to walk through setting up a schematic for the purposes of PSpice analysis. In order to generate a bode plot/stability plot for a given schematic, navigate to the FAQ link listed in the E2E post.

Step 1: To begin, set up an empty project in PSpice. This can be done by selecting "File" in the top left corner along with "New" and "Project". Name your project and choose its file location. A pop-up window to "Create PSpice Project" will populate the screen. Select "Create a blank project" if you wish to create a new schematic.

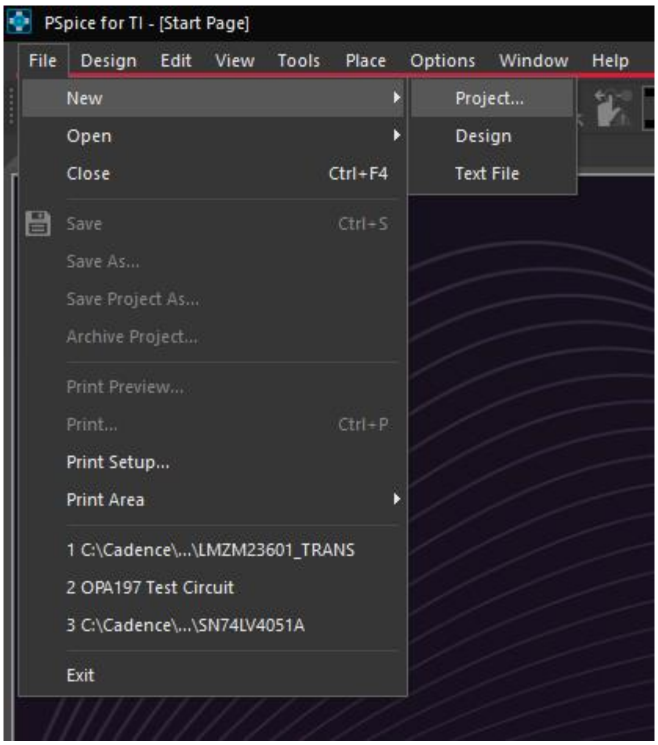

Figure 1: Navigation window options from “File” to “New Project.”

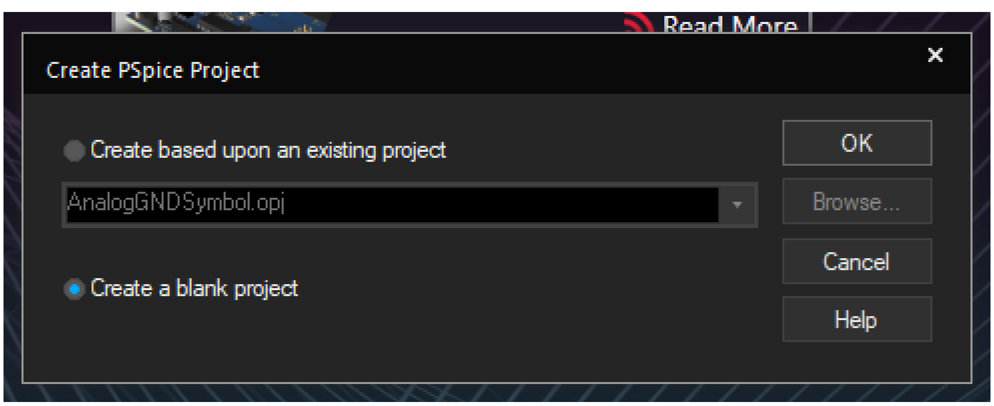

Figure 2: PSpice project creation pop-up.

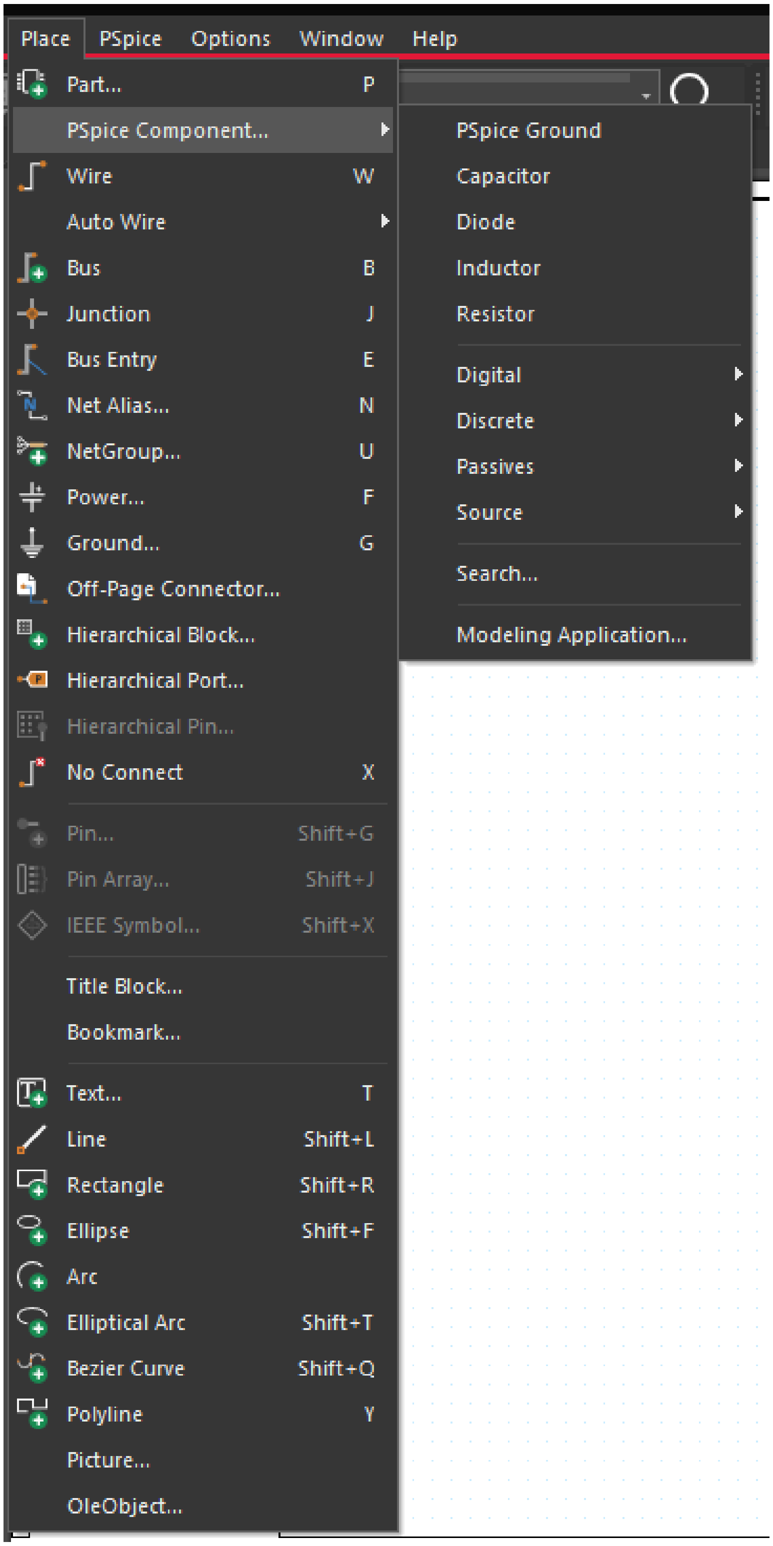

Figure 3: Navigation window options from “Place” to “PSpice Components”

Step 2: Placing components onto your schematic is straightforward. Simply utilize the navigator in the upper left hand corner to find the option for "Place". You may utilize any of the PSpice components and double click on their values to change them.

Available keyboard shortcuts are listed in this tab. The most notable among them are the ones for ground "G" and wire "W". Additional options for the type of ground will pop up. You may select a component without moving it, and input "R" into your keyboard in order to rotate the selected component by 90 degrees. To finish placing your component down, input the ESC key on your keyboard to finish.

Step 3: In order to utilize any Texas Instruments components, use the search function underneath the “Place” tab. A window on the right-hand side will reveal folders. Select the one listed as "Texas Instruments" and use the search bar below. You can search by part number, description, or overall specifications, and PSpice's search function will pull up a list of compatible parts.

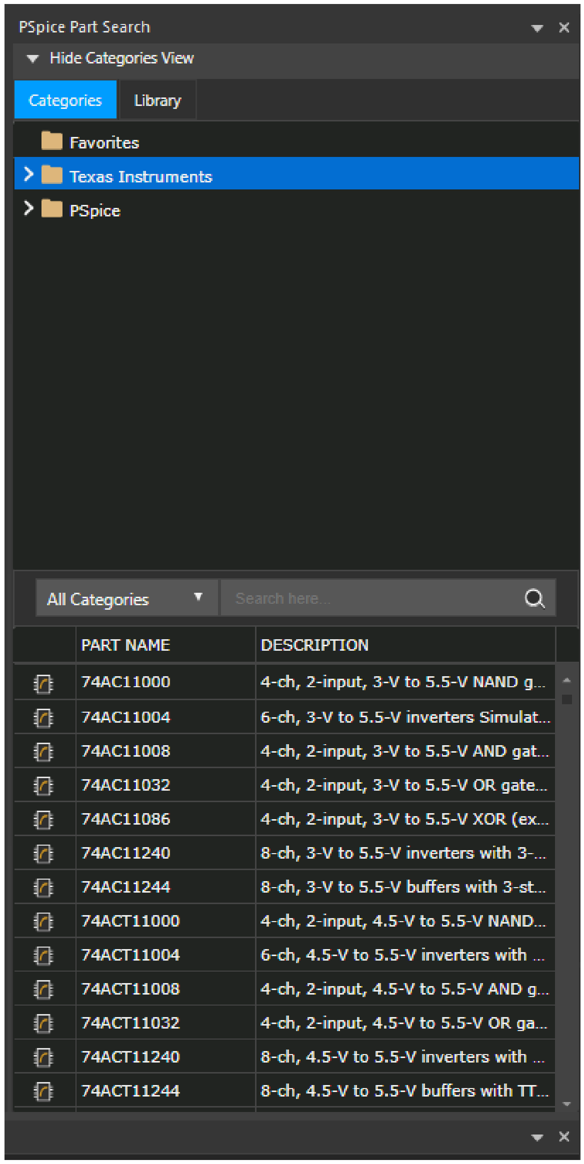

Figure 4: An example of available Texas Instruments specific parts