



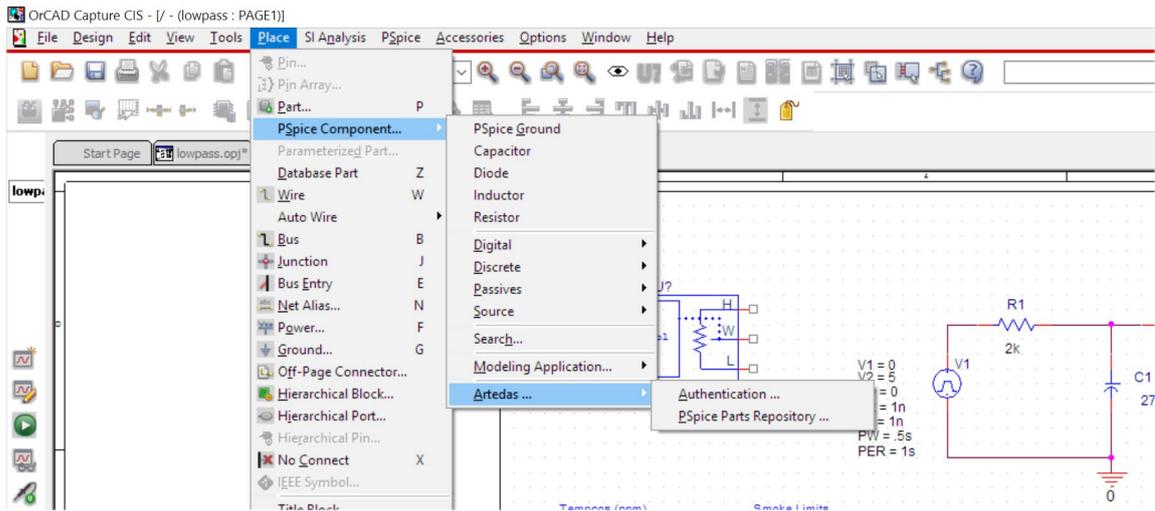
Artedas PSpice Parts Repository Help

This is a collection of silicon vendors PSpice models database.

Each of the PSpice models have their equivalent Capture parts and the added models are either new (not available in the PSpice base packages) or updated version of the existing models.

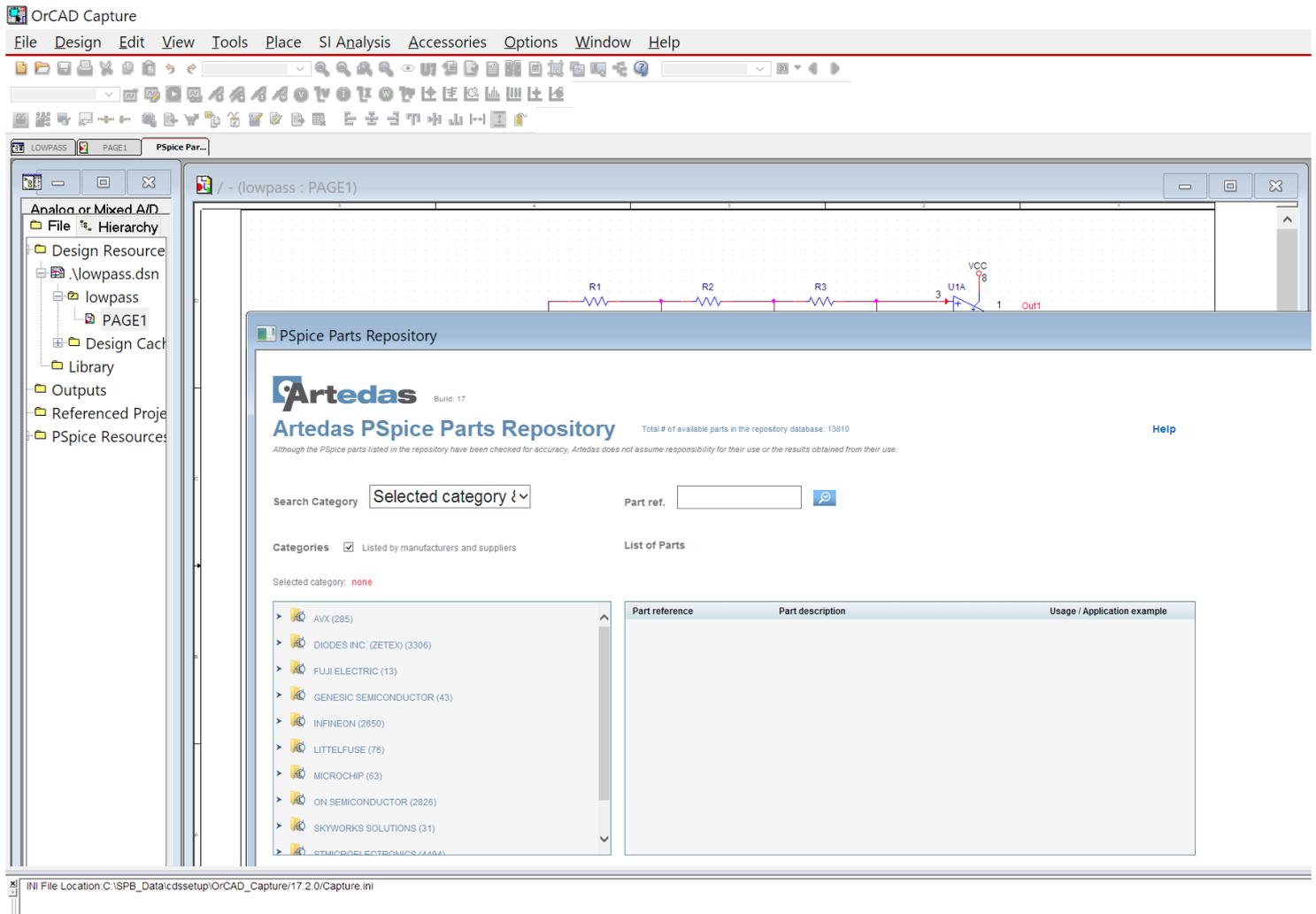
How it works

A homebrewed TCL code needs to be added to Capture. To be able to access the model's database from our server, the user must log in the first time. The login session stays active as long the user's Capture license is under maintenance.

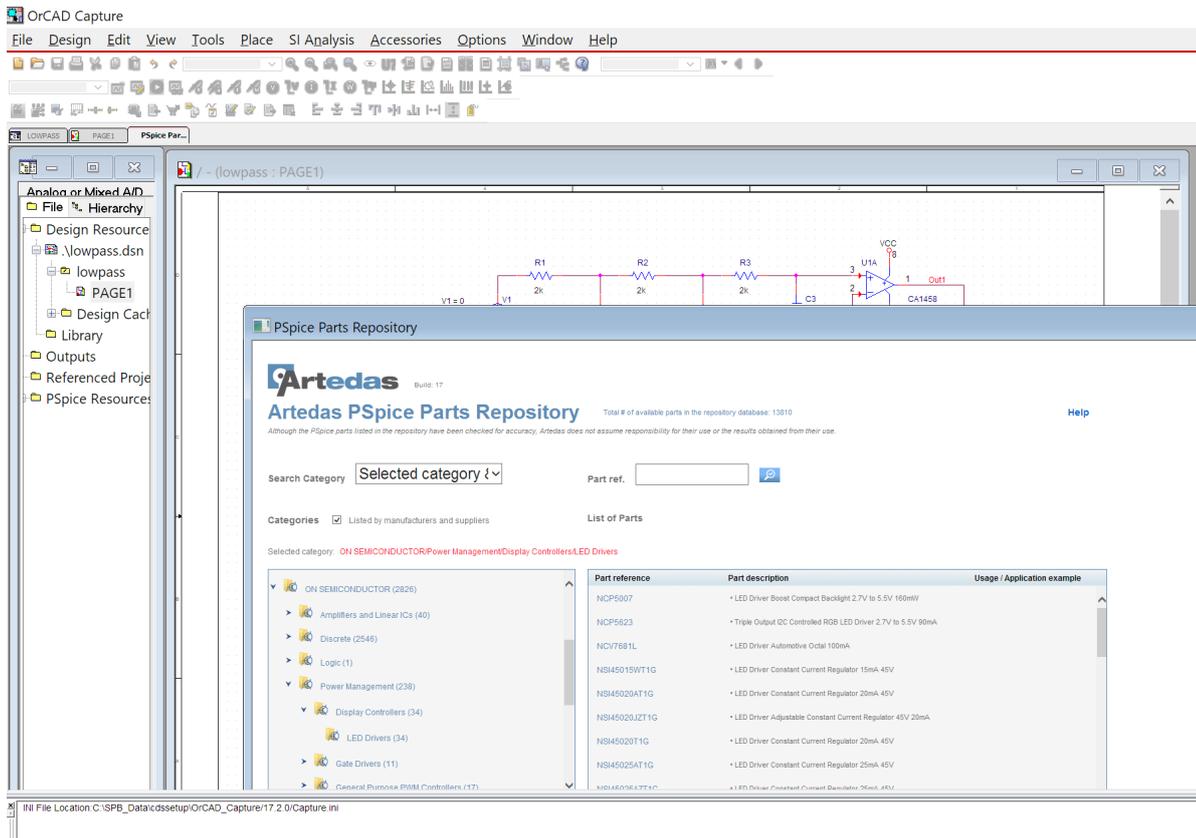


PSpice Parts Repository tab

Select PSpice Parts Repository tab from the Place/PSpice Component/Artedas/PSpice Parts Repository menu.



Selecting a part



If a list of manufacturer/supplier is needed, select the Manufacturer checkbox, the manufacturers will then be listed in the category table.

Click on the arrow to open the subdivisions until a list of available models are displayed in the righthand table. Then, select a part from the “list of Part”.

Note: In the “Usage” column, a PDF icon is displayed when a PDF document is available, providing you more detailed information on how to use the PSpice model.

An image of the selected part is attached to your pointer. You can click the right mouse button to display a pop-up menu with commands that you can use to change the properties of the part before you place it. You can mirror the part horizontally or vertically, rotate the part, or edit the part’s properties.

Move the pointer to the location on your schematic page where you want to place the part, then click the left mouse button.

This action places an instance of the part on your schematic page.

Search features

The Search is not case sensitive. Use the wild card “*” for the search parameter name e.g. ZDT675*, ZD*, etc.

How the process works when placing a part on a sheet?

- 1) The selected Capture symbol (*.OLB) and its associated PSpice model library are copied in the following subdirectory: “Cadence root directory”\Tools\ArtedasLibs.
- 2) During the very first part placement, an ARTEDAS.LIB file is created inside the above directory.
- 3) The ARTEDAS.LIB file is declared in the PSPICE.INI file.
- 4) The library of the model is declared in the ARTEDAS.LIB file; the part model is therefore available for all your subsequent simulation projects.
- 5) The symbol is automatically attached to the cursor for placement on the active sheet.

Note: If you no longer have access to Artedas PSpice Parts Repository database, for your future projects, use the Place/Part tool to declare the previously used symbol libraries (*.OLB) available in the “Cadence root directory”\Tools\ArtedasLibs directory.