Walkthrough

TI Precision Labs – PSpice® for TI

Start page

- Helpful resources:
  - PSpice for TI E2E
  - PSpice training series: Getting started in PSpice for TI
 - PSpice Users Manual
  - Recent Projects

TI Support & Resources

PSpice for TI FAQ

PSpice for TI E2E™ Design Support Forum

Series: Getting started in PSpice for TI

Quick Start Video

PSPICE® FOR TI

AN INTRODUCTION

PSpice® for TI

Cadence Support & Resources

PSpice User Manual

PSpice Training Course

Upgrade to Full Version

PSpice Resources

Recent Projects

current sources - high-side v-i with bjt

basic circuits - difference amplifier.opj

basic circuits - difference amplifier.opj

basic circuits - difference amplifier.opj
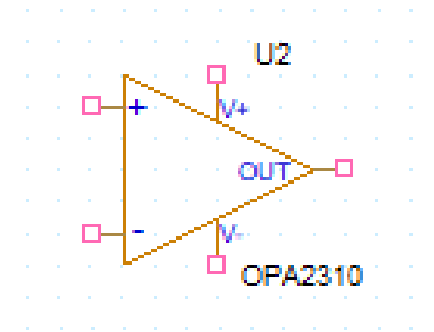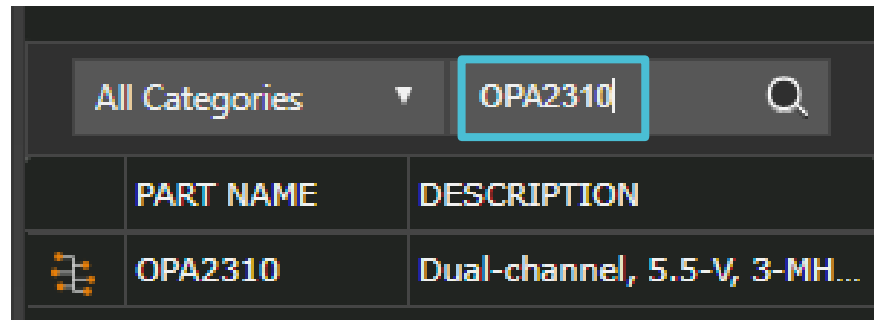
TEXAS INSTRUMENTS

Create new project

- Create a blank project:
  - Navigate to File > New > Project

- Name your project
- Assign it a location
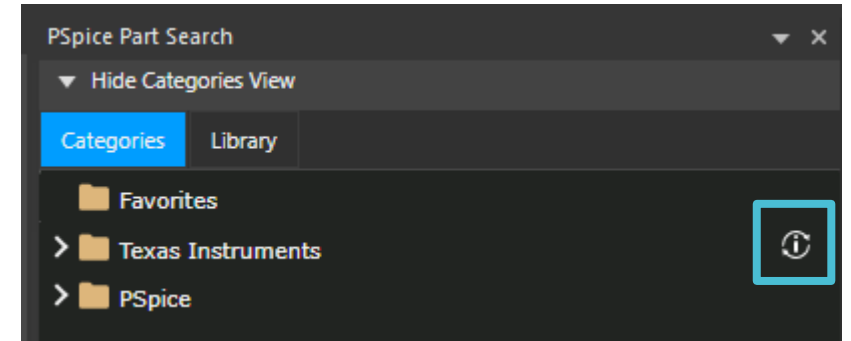- Click "Create a Blank Project"
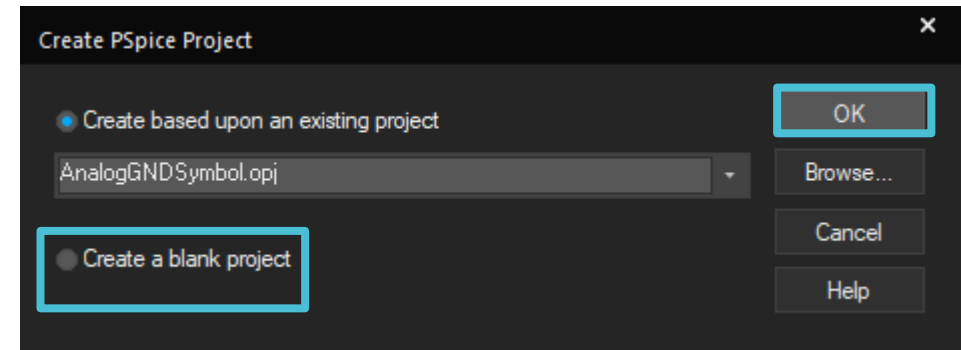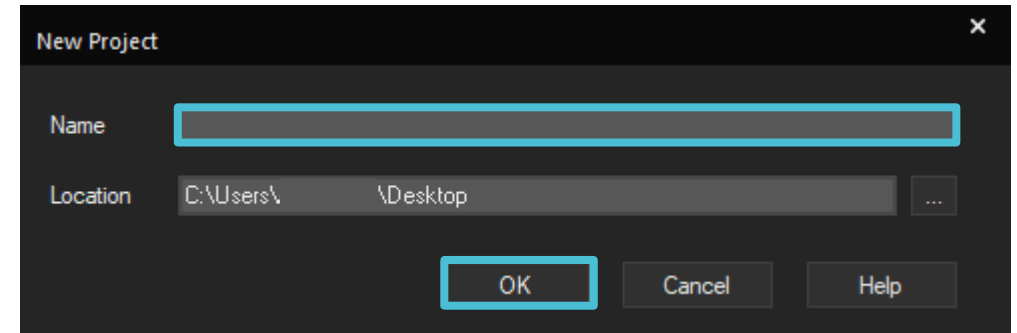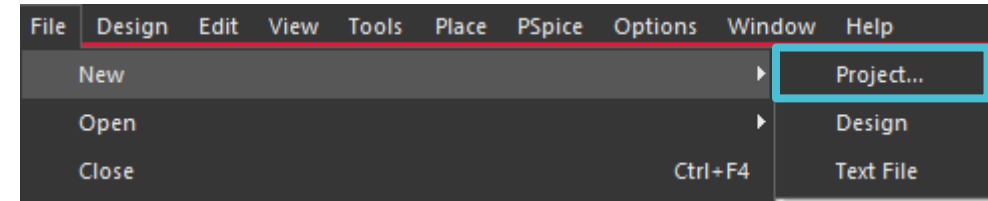
TEXAS INSTRUMENTS

PSpice part search

- PSpice Part Search contains TI components and models native to PSpice
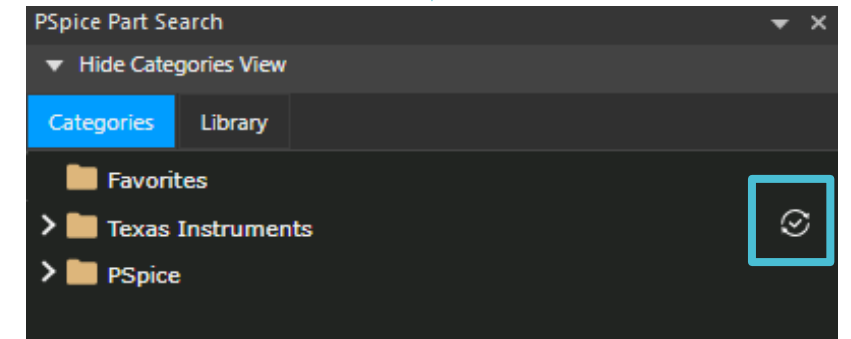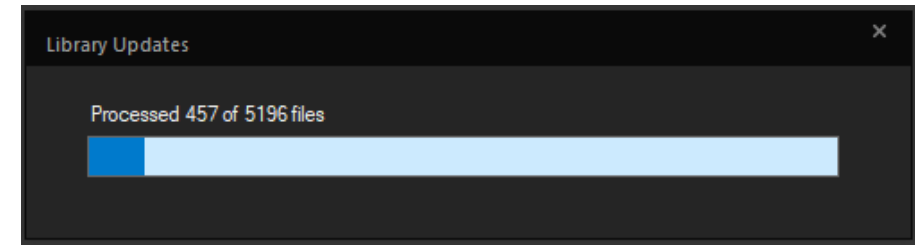- Type in a model number and place onto schematic or search by category

TEXAS INSTRUMENTS

PSpice part search

- Refresh symbol next to "Texas Instruments" folder will show if library is up-to-date

- Restart PFTI to update library

- You can use wildcard to search parts with similar names
  - Asterisk (*): match zero or more characters at the end of a root word
  - Percent (%): match exactly one character to search all variants of a part

Restart PFTI

TEXAS INSTRUMENTS

Project manager tab

- Design Resources:
  - "Project Name".dsn
    - Schematics
      - Pages

- PSpice Resources:
  - Simulation Profiles

Toolbar

- Top / far right of page is the toolbar

- Capture Toolbar
  - File:
  - Edit:
  - View:

- PSpice Toolbar:

- Draw Electrical Toolbar:

- Draw Graphical Toolbar:

TEXAS INSTRUMENTS
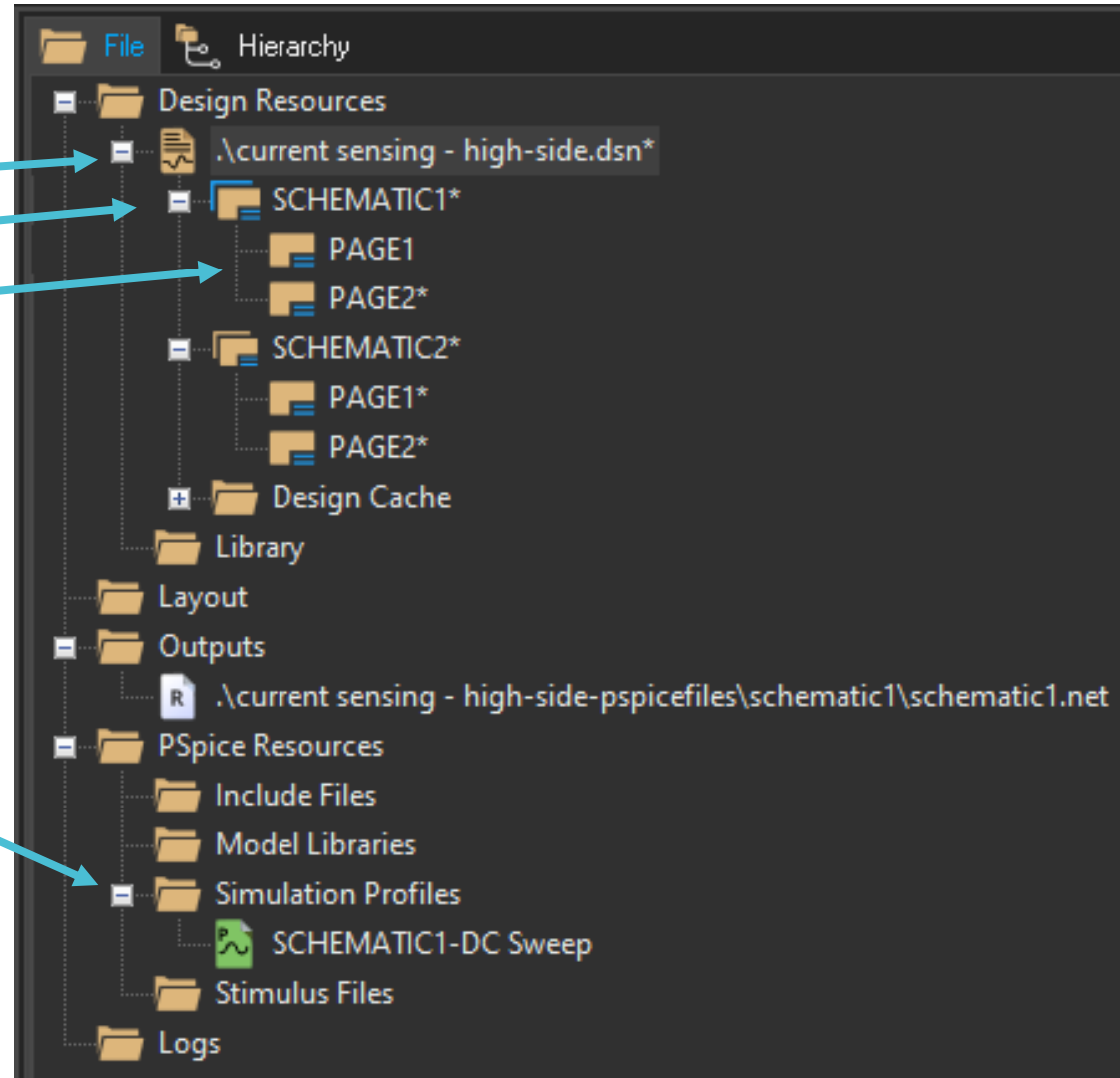
Thanks for your time!

TEXAS INSTRUMENTS

To find more **PSpice® for TI** technical resources and search products, visit **ti.com/tool/PSPICE-FOR-TI**.

TEXAS INSTRUMENTS