

LTSpice Work Documentation

Table of Contents

Resources	1
LTSpice Keyboard Shortcuts.....	1
LTSpice Models.....	2

Resources

<https://www.analog.com/en/resources/design-tools-and-calculators/ltspice-simulator.html>

<https://ez.analog.com/design-tools-and-calculators/ltspice/a/faqs-docs/c/getting-started-with-ltspice>

<https://www.ti.com/product/TL431#order-quality>

LTSpice Keyboard Shortcuts

[Analog Devices Keyboard Shortcuts Diagram.pdf](#)

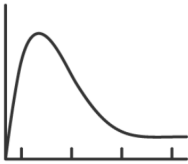
[LTSpice Wiki Keyboard Shortcuts](#)

S : Spice Directive

LTSpice Models

Simulation models

Design tools & simulation ^

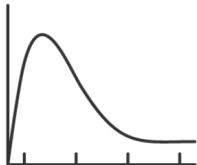


SIMULATION MODEL

[SPICE Model of TL431](#)

SLOJ174.ZIP (0 KB) - Spice Model

Download



SIMULATION MODEL

[TL431 Family Unencrypted PSpice Transient and AC Model \(Rev. B\)](#)

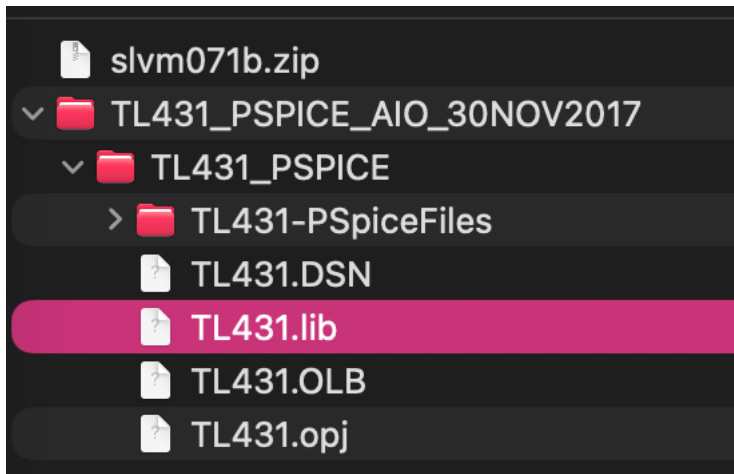
SLVM071B.ZIP (130 KB) - PSpice Model

Download

Utilize TL431 Family unencrpyed PSpice Transient and AC Model

Include Spice Directive

```
.include TL431.lib
```



- ☐ How to import to TL431.lib into ltspice file
- ☐ Do I need to generate a symbol to actually use this library?
- ☐ How to generate a symbol?
 - ☐ How to map the symbol to the TL431.lib file
 - ☐ How to map the pins and wires for the symbol?
 - ☐ Should the symbol look like other parts?
 - ☐ Is there an alternative to generating a symbol such as using opamp2 instead or a 3pin symbol and mapping it with TL431.lib file