LTSpice model for LM393

forum.allaboutcircuits.com/threads/ltspice-model-for-lm393.131516

Thread Starter

spinnaker

Joined Oct 29, 2009

7,835

Jan 22, 2017
#1

LM393LM393

E

eetech00

Joined Jun 8, 2013

1,865

Jan 22, 2017
#2

> spinnaker said:
> Anyone know of a LTSpice model for the LM393? As I am sure you are aware the LM393,
> requires an external pullup. A native LTSpice model I grabbed at random does not seem to
> require a pullup.

Thread Starter

spinnaker

Joined Oct 29, 2009

7,835

Jan 22, 2017
    #3

> eetech00 said:
> You can download a model file at TI.com

Already found that. I could only find models for PSpice on the TI site. Is it compatible with LTspice? LTSpice has two files a symbol and a model. The link on the TI page only has one file.

http://www.ti.com/product/LM393A/toolssoftware?dcmp=dsproject&hqs=sw&#desKits

E

eetech00

Joined Jun 8, 2013

1,865

Jan 22, 2017
    #4

> spinnaker said:
>
> Already found that. I could only find models for PSpice on the TI site. Is it compatible with LTspice? LTSpice has two files a symbol and a model. The link on the TI page only has one file.
>
> http://www.ti.com/product/LM393A/toolssoftware?dcmp=dsproject&hqs=sw&#desKits

Yes....LTspice is compatible with pspice models.

Use the LTspice opamp2 symbol.

Attachments

LM393Test.zip
1.3 KB Views: 389

Thread Starter

spinnaker

Joined Oct 29, 2009

7,835

> eetech00 said:
>
> Yes....LTspice is compatible with pspice models.
>
> Use the LTspice opamp2 symbol.

> spinnaker said:
> Anyone know of a LTSpice model for the LM393? As I am sure you are aware the LM393, requires an external pullup. A native LTSpice model I grabbed at random does not seem to require a pullup.

The excellent LTspice user group at:

http://groups.yahoo.com/group/LTspice/