

Instructions for Using Bourns GDT Library Data Files with LTspice

Bourns provide gas discharge tube (GDT) library data files for use with LTspice. There is only one library data file for each family of GDT parts with the ability to change parameters, such as sparkover voltages. These library files model the behavior of GDTs under DC and impulse sparkovers.

Note: Simulations using these files require that the solver in LTspice be set to “Alternate” instead of “Normal.” Instructions for changing this setting are described below.

GDT Library Symbol

A 2-external node GDT Library Symbol must first be created. The user may use or modify an existing symbol or draw a new one. At the Symbol Attribute Editor window, enter “X” into the prefix field to indicate that the symbol represents a subcircuit. At the Pin/Port Property window, assign two Netlist nodes. For your convenience, the Bourns Library Model circuit symbol file “Bourns GDT Model.asy” is available for download.

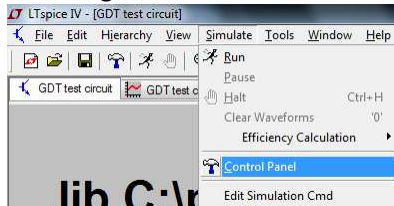
The GDT library file is **not** bi-directional. The simulation will only work when the surge is applied to pin 1 of the symbol (denoted with a “+” on the symbol provided). If the surge is applied to pin 2, the simulation will yield incorrect results.

How to Use Bourns GDT Library Data

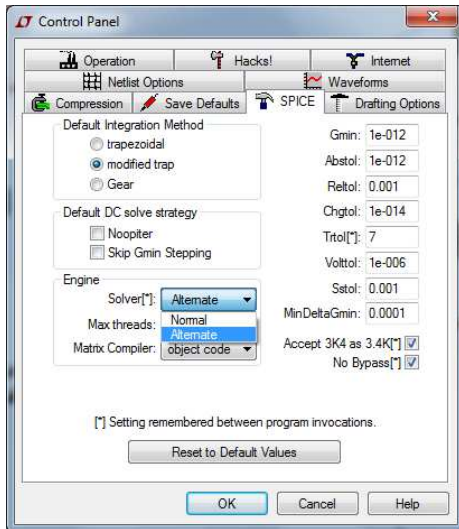
Download Bourns GDT Library data files at www.bourns.com

How to Change Solvers in LTspice

To change the solver from “Normal” to “Alternate,” go to Simulate -> Control Panel.

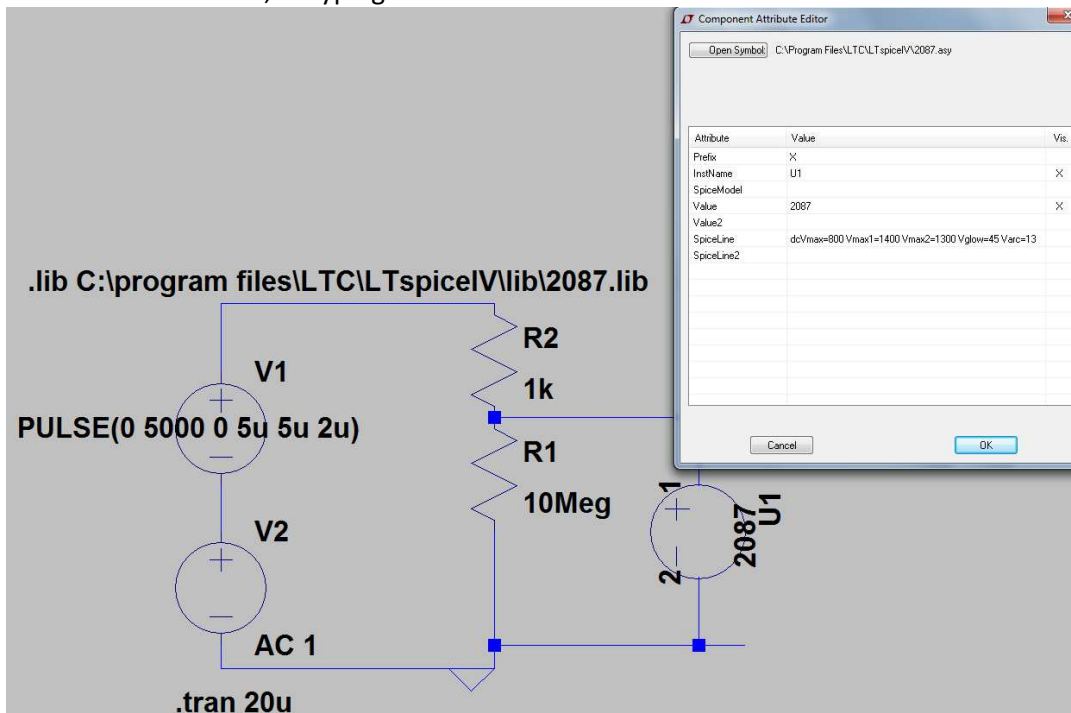


Once the control panel menu pops up, go to the “SPICE” tab. The solver is found in the “Engine” section of SPICE tab. From the solver drop down menu, select “Alternate,” then click “OK.”



Method 1: Spice Directive Syntax

Place the Bourns GDT Model Symbol and connect it to your circuit. Right click the component to bring up the Component Attribute Editor, place the GDT library file name into the Value field. Include the named library file using the .Include SPICE directive `.inc <directory> \ <filename>.lib` <directory> is the path where named file is located. SPICE directives are added by clicking on the “.op” button on the toolbar, or typing “s” in the schematic viewer.

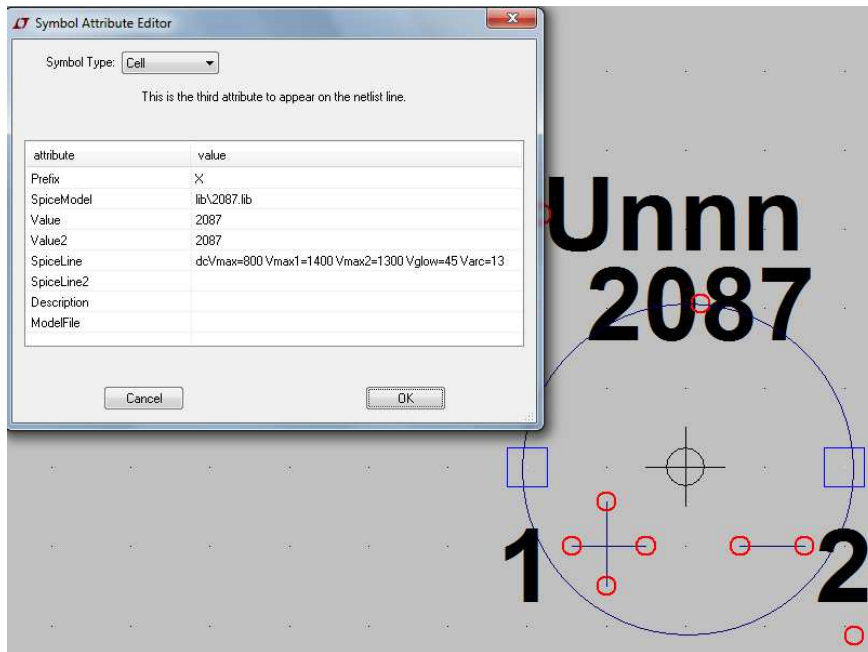


Method 2: Place Bourns GDT Library Data Files into LTspice Library

It may be time-efficient to place a specific part number library data file into the LTspice Library if it is being used regularly. However, doing this will make the symbol unavailable for editing parameters and

will only use the values given at the time it was saved. Copy Bourns library data file to <My-Library> subdirectory (or any other name of subdirectory of your choice) under \LTspiceIV\lib C:\Program Files\LTspiceIV\lib\My-Library For example, C:\Program Files\LTspiceIV\lib\My-Library\2087 (LIB file)

Next, create the circuit symbol as described above.
 At the Symbol Attribute Editor,
 Place “X” into the Prefix field.
 Place “My-Library\<library filename>.lib into the Spice Model field.
 Place “<library filename> into the Value and Value2 fields.
 Place the Bourns GDT Model Symbol connected to the circuit for simulation.



How to Change Parameters for Spice Models

Because the behaviors of GDTs vary from model to model, the option to change parameter values (i.e. sparkover voltages) for different GDTs are provided (i.e. a 90V GDT will behave differently from a 600V GDT). These values are provided in the datasheets. To do so, open the “Symbol Attribute Editor” by either: right-clicking on the symbol from the schematic or by selecting “Edit-> Attributes->Edit Attributes” from the symbol viewer. In the SpiceLine field, you should see various parameters if you downloaded the provided symbol. If you created your own symbol, please add the corresponding parameters in this field. Otherwise, the model defaults to lower sparkover voltages depending on the part number. Follow the instructions below to add these parameters:

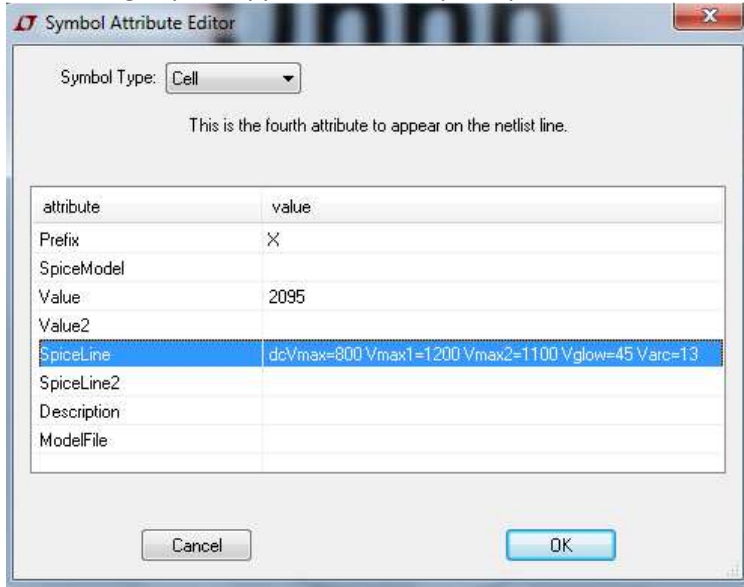
Double click in the SpiceLine field.

Copy and paste “dcVmax=A Vmax1=B Vmax2=C Vglow=D Varc=E”. The parameters are defined below:
 dcVmax and A are the dc sparkover voltage (100V/s) and value, A= 800V for the 2087-80
 Vmax1 and B are the impulse sparkover voltage (1kV/us) and value, B=1400 for the 2087-80
 Vmax2 and C are the impulse sparkover voltage (100V/us) and value, C=1300 for the 2087-80

Vglow and D are the glow voltage and value, D = glow voltage minus arc Voltage (i.e. 60V-15V or 45V for 2087-80)

Varc and E are the arc voltage and value, E = arc voltage – 2(i.e. 15V – 2V or 13V for 2087-80)

Change these values according to the values specified in the datasheet for your part number then click “OK.” Be sure to save the symbol! You may have to delete the symbol from the schematic and add it again for the updated values to be reflected in the simulation. The GDT will now behave according to the new voltages you supplied. An example is provided below for the 2095-300 model.



.LIB -- Include a Library

Syntax: `.lib <filename>`

This directive includes the model and subcircuit definitions of the named file as if that file had been typed into the netlist instead of the `.lib` command. Circuit elements at global scope are ignored.

An absolute path name may be entered for the filename. Otherwise LTspice looks first in the directory `%HOMEPATH%\Documents\LTspiceXVII\lib\cmp` and then `%HOMEPATH%\Documents\LTspiceXVII\lib\sub` and then in the directory that contains the calling netlist.

No file name extension is assumed. You must use `".lib myfile.lib"` not `".lib myfile"` if the file is called `"myfile.lib"`

It is possible to specify a URL of the following form as a file name:

```
.lib http://www.company.com/models/library.mod
```

The file "library.mod" will be http-transferred to the circuit directory and included as a library. For subsequent simulations, in the interest of avoiding downloading the file each time you run the simulation, you can edit the `.lib` statement to

```
.lib library.mod
```

Note that if the URL you specify doesn't exist, most web servers don't return an error, but return a html web page to be displayed in your web browser that explains the error. LTspice can't always read these pages as error conditions so you may get some cryptic error message when the simulation tries to proceed with the included html language error page included in the simulation as valid SPICE syntax.

Encrypted Libraries

LTspice can generate and read a special form of encrypted libraries. This allows one user to prepare a library that another user can use in a simulation without revealing the implementation of the library. A reasonable attempt has been made to make the encrypted library difficult to decode by unauthorized concerns, but it cannot be considered perfectly

secure if for no other reason than it is implemented in software.

To prepare an encrypted library, you need to invoke LTspice from the command line with the command line option "-encrypt". You will need to first backup the library because it will be replaced with the encrypted version. THERE EXISTS NO UTILITY TO CONVERT AN ENCRYPTED LIBRARY BACK TO CLEAR TEXT. Below summarizes the two steps:

1. Make a backup copy of the library. The version you encrypt is deleted.
2. From a command line, type

```
\XVIIx64.exe -encrypt <filename>
```

The file <filename> will be replaced with an encrypted version. The encryption process will take a few minutes.

Once this process is finished, you have an encrypted ASCII file. It's possible to add a copyright notice above the "* Begin:" line, but the first 9 lines of the file must remain unchanged and each line of copyright notice you add must begin with the character '*'.
*'

That is, here an encrypted file written by LTspice:

```
* LTspice Encrypted File
*
* This encrypted file has been supplied by a 3rd
* party vendor that does not wish to publicize
* the technology used to implement this library.
*
* Permission is granted to use this file for
* simulations but not to reverse engineer its
* contents.
*
* Begin:
50 3E 46 0F FA 6E 67 FF B8 4D D9 62 14 32 60 24
36 71 35 0B 66 4F AD 52 B8 F5 9E 22 9F C0 18 8B
FB FE 1D...
```

which you can change to

```
* LTspice Encrypted File
*
```

* This encrypted file has been supplied by a 3rd
* party vendor that does not wish to publicize
* the technology used to implement this library.
*
* Permission is granted to use this file for
* simulations but not to reverse engineer its
* contents.
*
* Copyright © 2005 Acme SPICE Modeling
* For additional information, see
* www.acmespicemodels.com
*
* Begin:
50 3E 46 0F FA 6E 67 FF B8 4D D9 62 14 32 60 24
36 71 35 0B 66 4F AD 52 B8 F5 9E 22 9F C0 18 8B
FB FE 1D...