

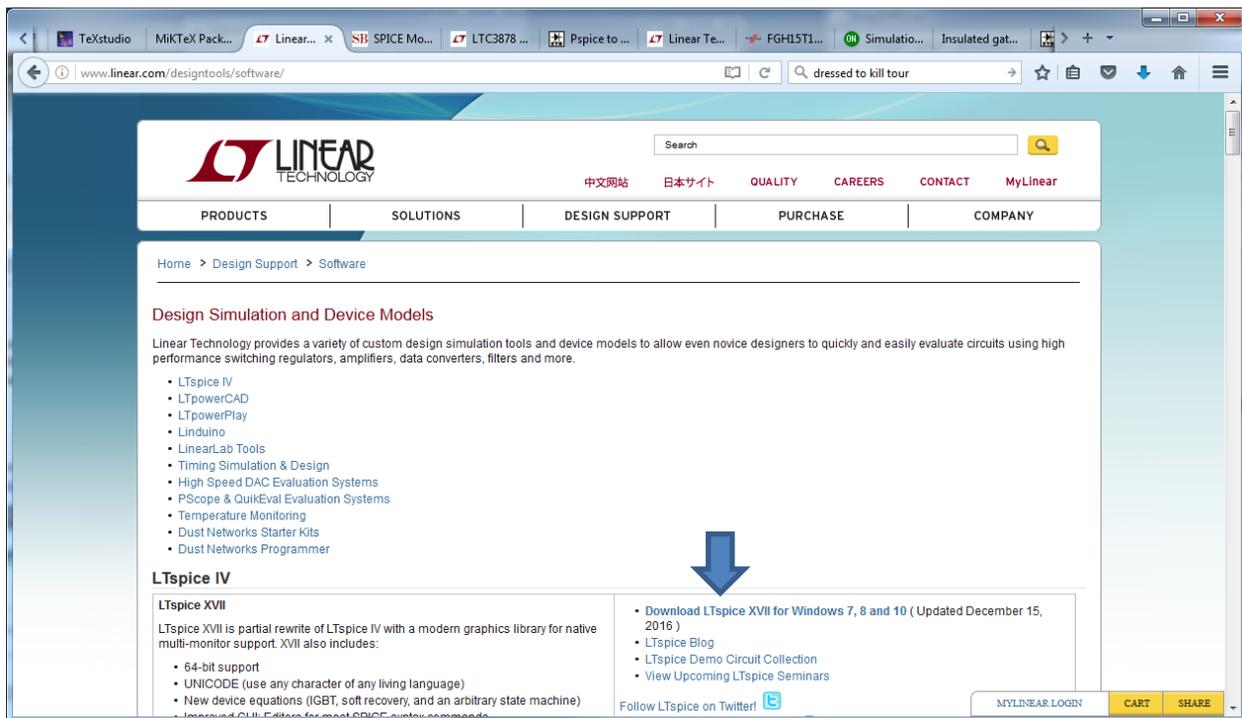
Installation of LTspiceXVII

Introduction

We will use LTspiceXVII in the software lab for the Accelerator Power Electronics course. We have slightly modified the standard installation in order to include some parts that are not in the standard distribution. These instructions detail how to both install the standard distribution and how to modify it to include our additions.

Installation

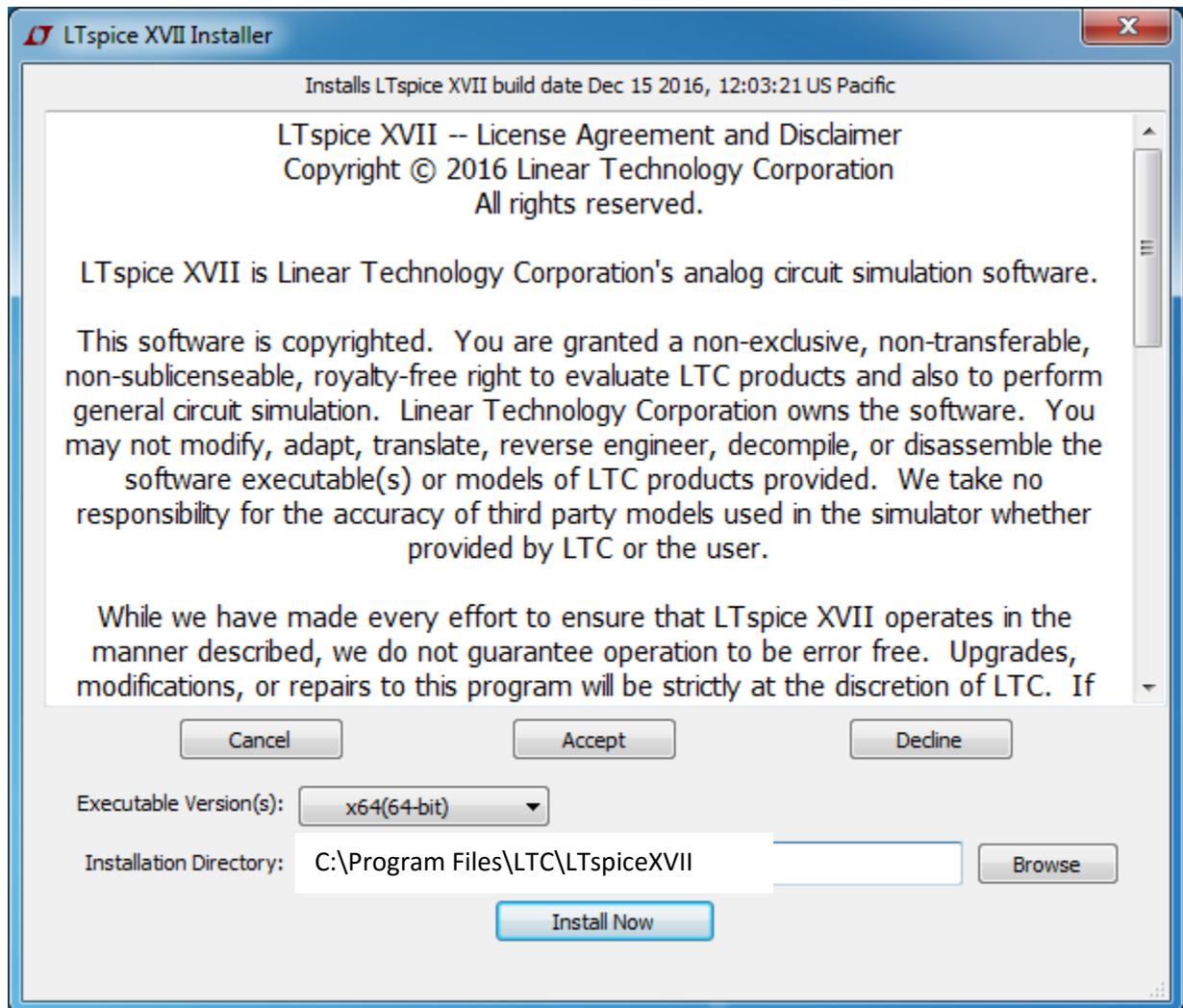
The software is maintained by Linear Technology, Inc., and the software can be freely downloaded from Linear Technology at its website <http://www.linear.com/designtools/software/>



Select *Download LTspice XVII for Windows 7, 8 and 10*. (You do not need to register for a MyLinear account. You can refuse this offer and just download the software.) You will be prompted to save the executable file LTspiceXVII.exe from <http://ltspice.linear-tech.com>. After the file is downloaded, select it to run the installation program, agreeing to the standard Microsoft query about installing a file downloaded from the internet. (You may need administrative privileges to install the program.)

Note that the instructors have provided the LTspiceXVII executable file. There is no need to download it.

The Installation Directory



Maintain the default installation directory C:\Program Files\LTC\LTspiceXVII, accept the license agreement, and then Install Now.

There should now be a directory C:\Program Files\LTC\LTspiceXVII that includes several files, including LTspiceXVIIx64.exe. The desktop shortcut created by LTspiceXVII runs this executable for a 64-bit installation. The directory also contains two subdirectories, "Examples" and "Lib".

The Working Directory

LTspice also creates a copy of the above two subdirectories in the user space `C:\Users\username\My Documents\LTspiceXVII`. This latter location is where LTspice looks for the files it will use in building the schematics and simulations.

The Component Directories

We would like to have a directory in which we store our custom device files. We create a directory structure

`C:\files\LTspiceFiles`

This structure stores our custom devices, those not in the LTspice distribution. We create two subdirectories, one for symbols that we will create and one for library parts that we will use in LTspice.

We create subdirectories

`C:\files\LTspiceFiles\sym`

`C:\files\LTspiceFiles\lib`

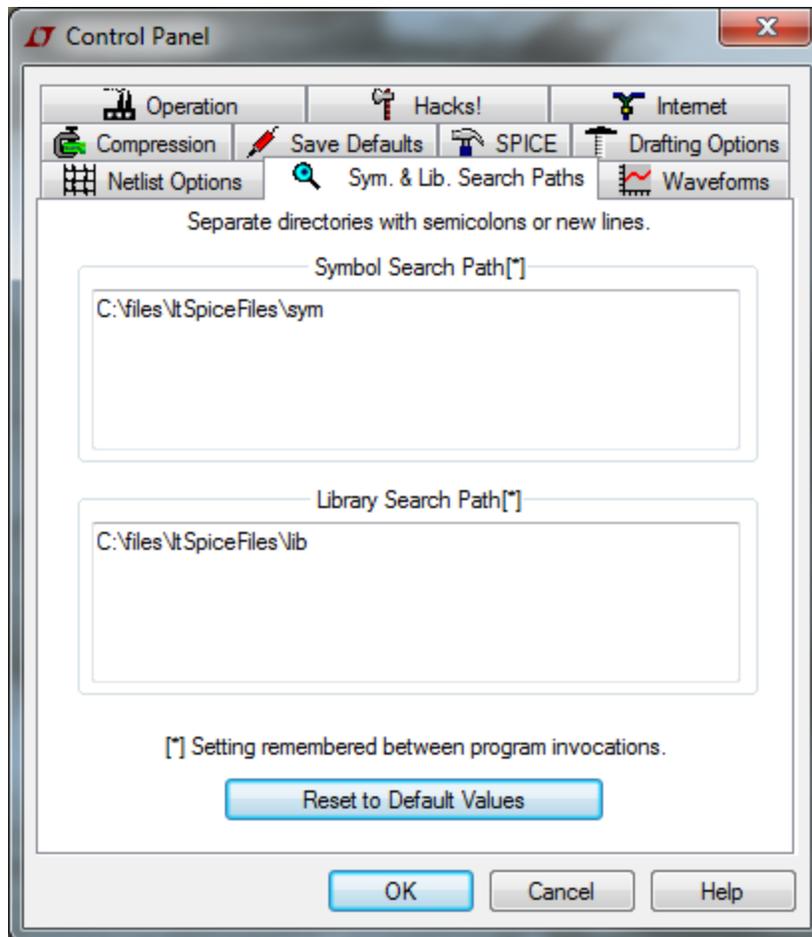
We are able to use these directories after navigating to LTspice->Tools->Control Panel->Sym. & Lib. Search Paths and include them in the LTspice search path, entering the directory names and then clicking OK. After closing LTspiceXVII these paths are stored as the variables `SymbolSearchPath` and `LibrarySearchPath`, respectively, in `LTspiceXVII.ini`.

Unless there are problems, usually there is no need to refer to the setup information.

Copy the file `irgp4063dpbf.spi` into the `C:/Files/LTspiceFiles/Lib` folder

Copy the file `irgp4063dpbf.asy` into the `C:/Files/LTspiceFiles/Sym` folder

We will use the `irgp4063dpbf` IGBT in a Buck Regulator circuit that we will build as part of the LTspiceXVII Laboratory Exercise.



APPENDIX

The following is listed for tutorial purposes. The information below is not a part of the Laboratory exercise. The instructors have built and integrated the IGBT subcircuits into the parts lists that you will use in the Laboratory.

Adding Models to Existing Model Files

The basic element of a SPICE circuit is a model. SPICE uses equations that define the current and voltage characteristics for each model. These characteristics depend on the semiconductor material and fabrication process. The model equations are based on standard device solid state physics device models. An example of a model is a bipolar transistor. To individualize each type of transistor, we then create a transistor model, initializing the value of each model parameter to that transistor characteristic.

A sample model for a 2N2222 transistor is

```
.model 2N2222 NPN(IS=1E-14 VAF=100
+ BF=200 IKF=0.3 XTB=1.5 BR=3
+ CJC=8E-12 CJE=25E-12 TR=100E-9 TF=400E-12
+ ITF=1 VTF=2 XTF=3 RB=10 RC=.3 RE=.2 Vceo=30 Icrating=800m mfg=Philips)
```

In this example, *.model* is the keyword telling SPICE a model definition follows, *2N2222* is the identifier of this model, *NPN* says that this element is an NPN transistor, and most of the equalities in the parenthesis are valid SPICE parameters for an NPN transistor model. The last three parameters, *Vceo*, *Icrating*, and *mfg* appear to be LTspice-specific informational parameters. LTspice extracts them and displays them in the menu that allows us to select the particular transistor for the circuit. The + signs at the beginning of the lines tell SPICE that these lines are continuations of a previous command.

The standard LTspice installation includes many files of standard analog models, such as diodes, bipolar transistors, junction FETs and MOSFETs. These files are all, by default, located in the library

C:\LTC\LTspiceXVII\lib\cmp (for component) and
C:\Users\username\My Documents\LTspiceXVII\lib\cmp.

All of these files are ASCII files with a series of models similar to the one displayed above for the 2N2222 bipolar transistor. To add additional devices to these files, we use an ASCII text editor and copy a SPICE model of the new device, typically obtained from a manufacturer's website, to an existing file. If we also want to add information to the model so that LTspice can display the information during the component selection, we add the appropriate equalities to the entry, using the existing entries as a model to determine the names of the valid informational parameters.

In addition to the model files containing semiconductors, there are also model files for passive devices, ferrite beads, capacitors, diodes, and resistors. These files are stored in a binary format, but we edit them from within LTspice.

Importing and Using IGBT Models

IGBTs are not defined by a standard model in SPICE. SPICE files can appear in different ways. The basic SPICE construct is an *element*. Examples of standard SPICE elements are resistors, capacitors, inductors,

voltage sources, current sources, diodes, bipolar junction transistors, MOSFETs, MESFETs, etc. With each element is associated a model that gives its voltage-current behavior as a function of various parameters based on device physics. If LTspice does not include an element from a particular manufacturer, one can add an element in the appropriate LTspice file, such as standard.dio, standard.mos, etc., by adding a line with the appropriate parameters for that particular element.

For more complicated structures, non-elemental structures, such as an IGBT, there are several options to adding the device to LTspice. The options depend on the information given by the manufacturer and the features that LTspice supports. In some cases, the manufacturer builds up the IGBT by modeling the IGBT as a collection of bipolar junction transistors, MOSFETs, resistors, capacitors, and inductors. The manufacturer then combines these elements into a subcircuit and uses the standard SPICE .SUBCKT construct. In other cases, the manufacturer uses a “non-standard” model, one that is not in the base version of SPICE. If LTspice does not support this model, then it cannot be used in LTspice. However, LTspice does support some of these models. If the manufacturer supplies such a compatible non-standard model, one adds the model differently than one adds a subcircuit.

IGBT Symbol

In either case, we need to assign a symbol to the IGBT so that we can place it in the LTspice schematic. LTspice has made two generic IGBT devices, an N channel, NIGBT.asy, and a P channel device, PIGBT.asy. Both are ASCII files located in

C:\Users\username\Documents\LTspiceXVII\lib\sym\Misc

The text of NIGBT.asy is:

Version 4

SymbolType CELL

LINE Normal 16 80 32 88

LINE Normal 48 96 28 92

LINE Normal 48 96 32 84

LINE Normal 28 92 32 84

LINE Normal 16 48 40 56

LINE Normal 16 8 16 24

LINE Normal 16 40 16 56

LINE Normal 16 72 16 88

LINE Normal 0 80 8 80

LINE Normal 8 16 8 80

LINE Normal 48 0 32 8

LINE Normal 16 16 36 12

LINE Normal 16 16 32 4

LINE Normal 36 12 32 4

WINDOW 0 56 32 Left 0

WINDOW 3 56 72 Left 0

SYMATTR Value NIGBT

SYMATTR Prefix Z

SYMATTR Description N-Channel IGBT symbol for a model you supply.

PIN 48 0 NONE 0

PINATTR PinName C

PINATTR SpiceOrder 1

PIN 0 80 NONE 0

PINATTR PinName G

PINATTR SpiceOrder 2

PIN 48 96 NONE 0

PINATTR PinName E

PINATTR SpiceOrder 3

Most of the information in the file relates to the appearance of the symbol in the LTspice graphical user interface. We do not need to modify these lines when we create our custom IGBT symbols. The LINE lines draw the graphic. The WINDOW lines set the location of text associated with the symbol. The PIN and PINATTR lines tell which parts of the symbol should be connected to the appropriate traces in the schematic, their function in the object and the order in which the elements are given in the SPICE model or subcircuit. For example a typical SPICE file for an IGBT identifies pin 1 as the collector, pin 2 as the gate, and pin 3 as the emitter.

We will have to customize the SYMATTR lines, those which describe the symbol attributes.

For each device we will copy this NIGBT.asy file into our private asy directory,

C:\LTC\LTspiceXVII\lib\sym\Misc

copy NIGBT.asy and PIGBT.asy to

C:\files\ltSpiceFiles\sym

rename the file to identify the particular device we will use, and modify the SYMATTR lines of the file, as appropriate.

Subcircuits

ON/Fairchild FGA180N33ATD

Fairchild supplies a .lib file, fga180n33atd.lib, for this IGBT (ON Semiconductor has acquired Fairchild).

This file, as are all SPICE input files, is an ASCII text file. Although in LTspice, the file extension is arbitrary, Fairchild has chosen to use the .lib extension for its SPICE models and subcircuits. Fairchild has built a subcircuit for this device using resistors, capacitors, inductors, NMOS transistors, a PNP transistor, and diodes. The subcircuit details the connections between the terminals of the elements and the file supplies the relevant parameters for each of these elements. We save this file in our personal lib folder.

We want to create an icon for this device to use in LTspice. (LTspice has a method to create a rectangular box icon, but we prefer to create an icon that looks like the standard IGBT symbol.) We start with the LTspice symbol NIGBT.asy, copy it and rename it NIGBTFGA180N33ATD.asy. In this assembly file we need to define attributes that will associate the file with the vendor subcircuit model. (A

discussion of attributes is in the LTspiceHelp,
LTspice XVII->Schematic Capture->Creating New Symbols->Adding Attributes)

The symbol file needs to have several attributes defined:

- *SYMATTR Prefix X*
The *X* tells LTspice that this symbol is a sub-circuit
- *SYMATTR Value name*
where *name* is replaced by the desired name on the schematic, for example fga180n33atd
- *SYMATTR Description descript*
where *descript* is a user supplied phrase. (I am not sure how this value is used.)
- *SYMATTR ModelFile filename*
where *filename* is the name of the ASCII file containing the definition of the sub-circuit, for example fpga180n33atd. Note that the filename needs to be in the private lib directory that we have created.
- *SYMATTR SpiceModel subcircuitname*
where *subcircuitname* is the name in the sub-circuit definition in *filename*, for example if the definition line in *filename* is *.SUBCKT FGA180N33ATD C G E*, then set *subcircuitname* to *FGA180N33ATD*.

SYMATTR Prefix X

SYMATTR Value FGA180N33ATD

SYMATTR Description N-Channel OnSemi/Fairchild FGA180N33ATD

SYMATTR ModelFile fga180n33atd.lib

SYMATTR SpiceModel FGA180N33ATD

Then we save the newly created *.asy* file into our personal sym folder.

This procedure also works for the International Rectifier (acquired by Infineon) subcircuit models, such as irgp4063dpbf.spi. Note that IR uses an *.spi* extension for its SPICE files.

Model

Some vendors, notably IXYS, do not use a subcircuit; rather they use a phenomenological model that is not a part of the standard SPICE installation. PSpice, a product of Cadence/Orcad was the first variation of SPICE to support this model. Since this is not a standard SPICE model, IXYS refers to their IGBT models as P-Spice models. In the LTspice XVII release, LTspice now also supports this IGBT model. We implement this IGBT model differently than we do a subcircuit. The reference for this inclusion is in

LTspice XVII->LTspice->Circuit Elements->Z. MESFET and IGBT Transistors

LTspice XVII->LTspice->F.A.Q.->Third-party Models

In order to use an IGBT which supplies this model, we need to create the symbol, slightly modify the supplied model file, and add a SPICE command to the schematic that will use this device.

IXYS IXGX35N120C

Symbol Modification

Our symbol file must include three SYMATTR lines

SYMATTR Prefix Z

SYMATTR Value IXGX35N120C

SYMATTR Description N-Channel IXYS IXGX35N120C

The symbol attribute Z informs LTspice that this symbol references a model (X would reference a sub-circuit). The ModelFile and SpiceModel attributes used in sub-circuits are not used for models.

Model File Modification

The first non-comment line in the vendor-supplied model file is

```
.MODEL IXGX35N120C NIGBT
```

Immediately preceding that line, add a line informing LTspice that this is a model file for the particular IGBT, with its pins ordered as collector, gate, and emitter, as described in

LTspice XVII->LTspice->Circuit Elements->Z. MESFET and IGBT Transistors

Add the line

```
Z001 C G E IXGX35N120C
```

immediately before the vendor-supplied .MODEL line.

Schematic Modification

LTspice does not use the SYMATTR ModelFile line in the symbol file to know that the SPICE library file should be used for the schematic in which the symbol is inserted. Therefore one must add an explicit SPICE .include directive in the schematic in order for LTspice to know to import the corresponding model file.

In LTspice

```
Edit->SPICE Directive->.include IXGX35N120C.lib
```

then place the directive on the board and Escape.