

## Instructions for Using Bourns Inductor Library Data Files with LTspice

Bourns offers an assortment of inductor models in an LTspice database for the user's application. The SMD Power Inductor LTspice models help to replicate an inductor's characteristics more accurately. The library models are a collection of subcircuit data precisely characterizing an inductor's properties well over its SRF (Self Resonant Frequency) under no-load conditions.

Note: The .lib library model file, will only run on AC Analysis. This type of model, derived from 2-port S-parameter measurements, will not function for transient or DC analysis. For DC analysis, please use the *inductor equivalent circuit*.

### Inductor Library Symbol

A 2-external node Inductor 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 notes.

For your convenience, the Bourns Library Model circuit symbol file "Bourns Inductor Model.asy" is available for download.

### How to Use Bourns Inductor Library Data

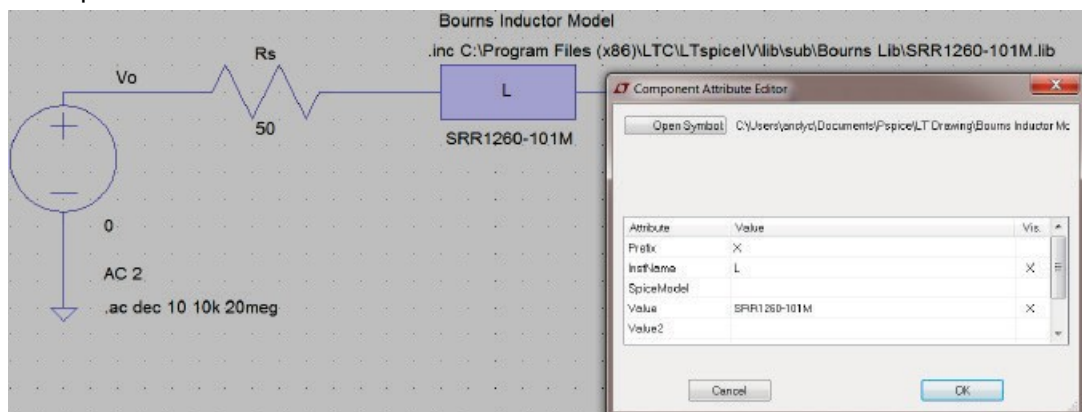
Download Bourns Inductor Library data files at [www.bourns.com](http://www.bourns.com).

#### Method 1: .Include Syntax

Place the Bourns Inductor Model Symbol and connect it to your circuit. At the Component Attribute Editor, place the inductor library file name into the Value field.

Include the named library file using the .Include directive `.inc < directory > \ <filename>.lib`  
< directory > is the path where named file is located.

Example of P/N SRR1260-101M:



#### Method 2: Place Bourns Inductor 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.

Copy Bourns library data file to <My-Library> subdirectory (or any other name of subdirectory of your choice) under \LTspiceIV\lib

C:\Program Files (x86)\LTC\LTspiceIV\lib\My-Library

For example, C:\Program Files (x86)\LTC\LTspiceIV\lib\My-Library\SRR1260-101M (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 Inductor Model Symbol connected to the circuit for simulation.

Example of P/N SRR1260-101M:

