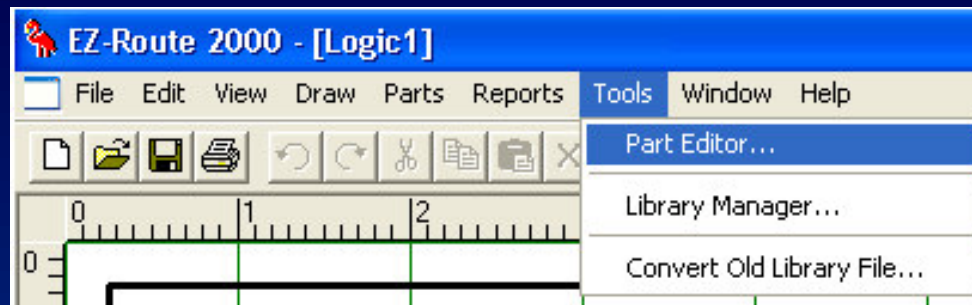


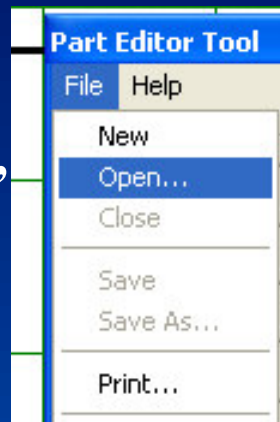
EZRoute2000

Adding new symbol values to existing libraries
Or what to do when you need a 11K and only see 10K

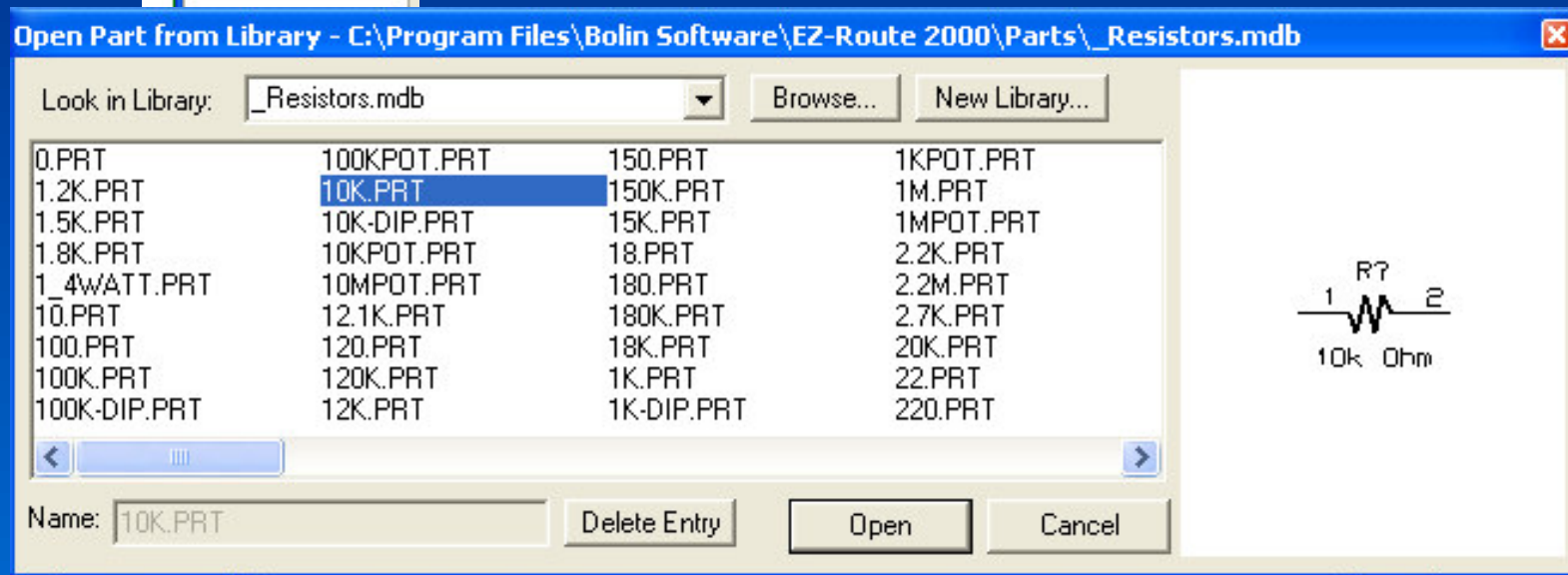
In schematic, open Tools and select Part Editor



In FILE,
Select
OPEN



Select the Resistor Library and select a value. I chose the 10K Press OPEN



**This is the 10K
Part Properties
listing.**

**MODIFY
the description**

Value 1 and

Corp#

Part Editor Tool - [10K.PRT]

File Help

Part Properties

Name:	10K	(set when saved)
Description:	10k Ohm Resistor	
Max Pin #:	2	Corp #: 001-10k-002
Footprint:	DISC-R	Cost: 0.05
Symbols:	R2	Pkg Type: Discrete
Power Pins:		Type:
Value1:	10k Ohm	Family: RESISTOR
Value2:		
Value3:		
Value4:		
Value5:		
Link To:		Browse...

Notice the changes making it 11K. This works for other components as well.

I just made a set of capacitors with nf values using the same procedure.

Part Editor Tool - [10K.PRT]

File Help

Part Properties

Name:	10K	(set when saved)	
Description:	11k Ohm Resistor		
Max Pin #:	2	Corp #:	001-11k-002
Footprint:	DISC-R	Cost:	0.05
Symbols:	R2	Pkg Type:	Discrete
Power Pins:		Type:	
Value1:	11k Ohm	Family:	RESISTOR
Value2:			
Value3:			
Value4:			
Value5:			
Link To:			Browse...

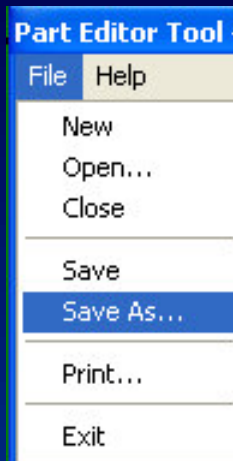
Add the correct cost, and be able to get a cost for your circuit.

Remember, you can't change the part name until you make all the changes, then press save. When the box reappears, look for the part name to be 11k

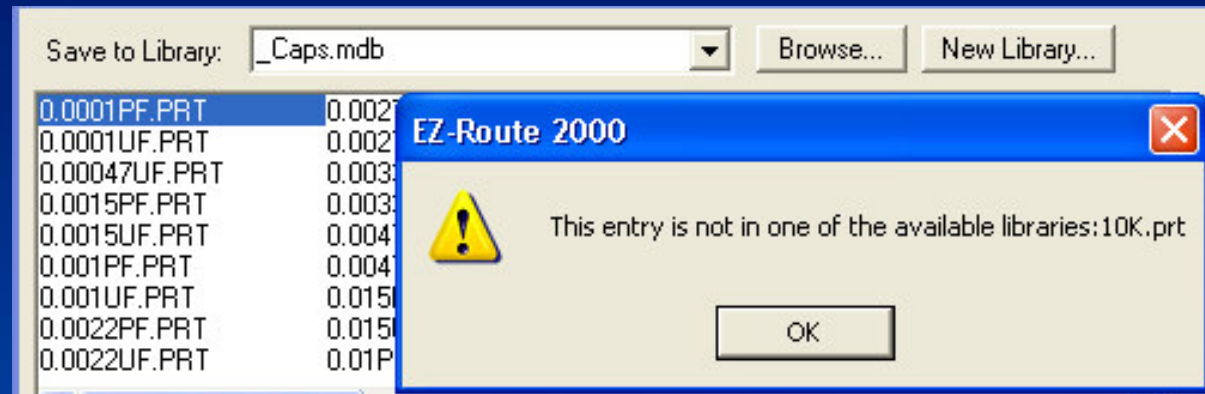
Part Properties

Name:	11K
Description:	11k Ohm Resistor
Max Pin #:	2

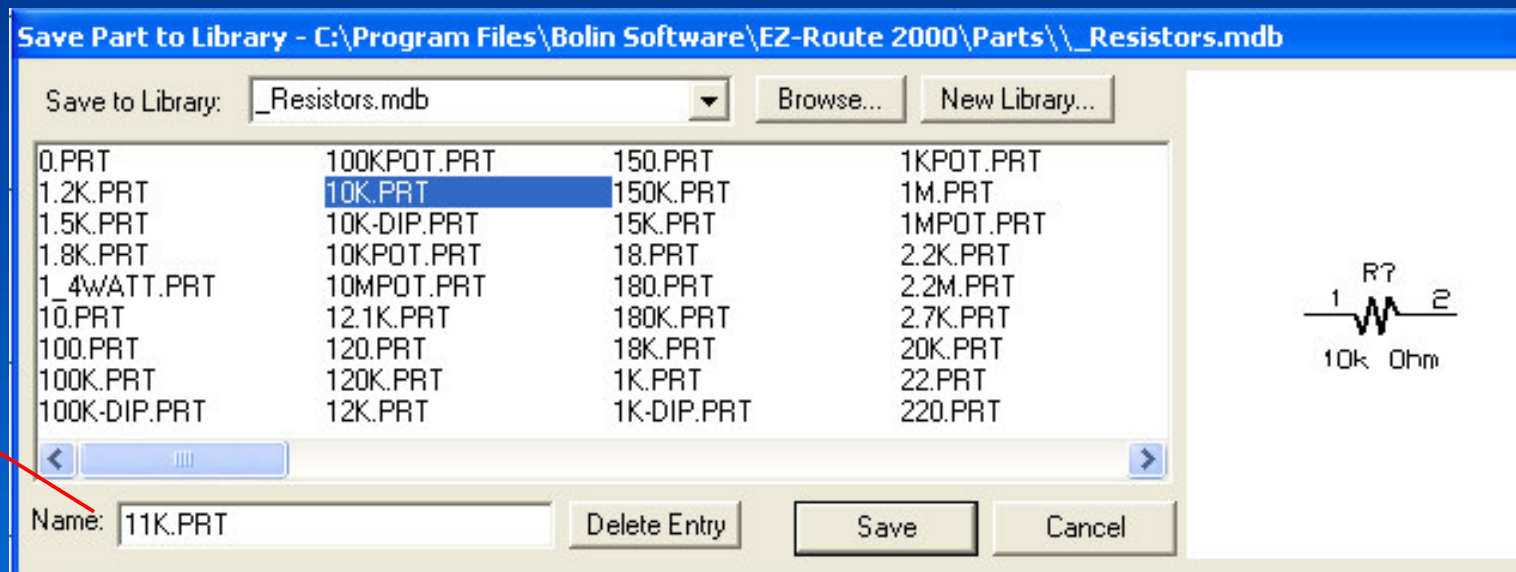
In FILE,
select Save
As



These two error
boxes will
appear, just click
OK



Select the
Resistor
library and
edit to make
the name
11K

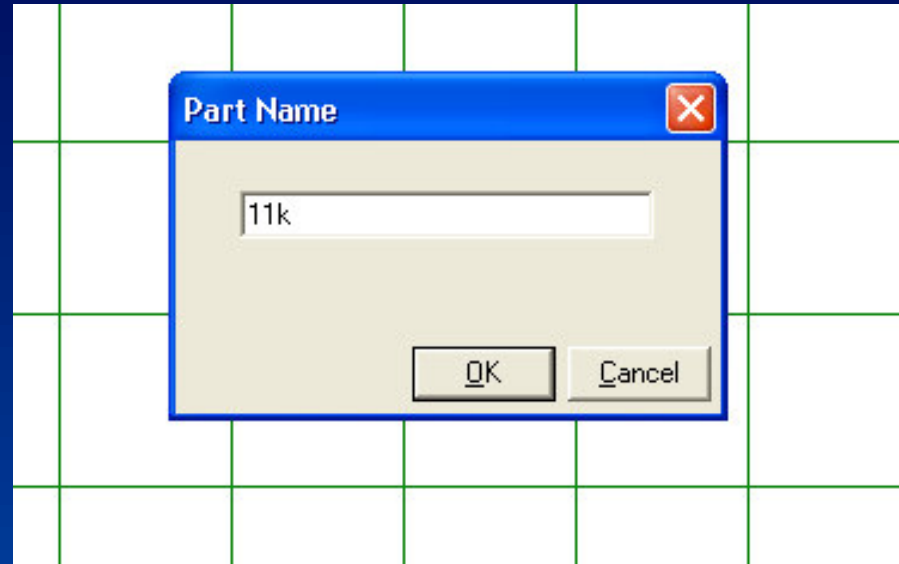


then Save

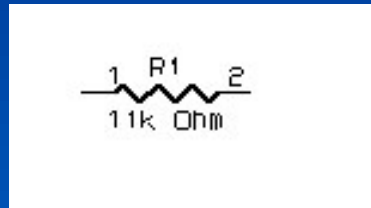
After saving, make sure you **CLOSE** the library. If not, you may need to reload the library.



To test enter the schematic editor, press J and enter 11k then OK



Here's 11K



**Here are some of the nf capacitor values I made.
Using the same procedure.**

