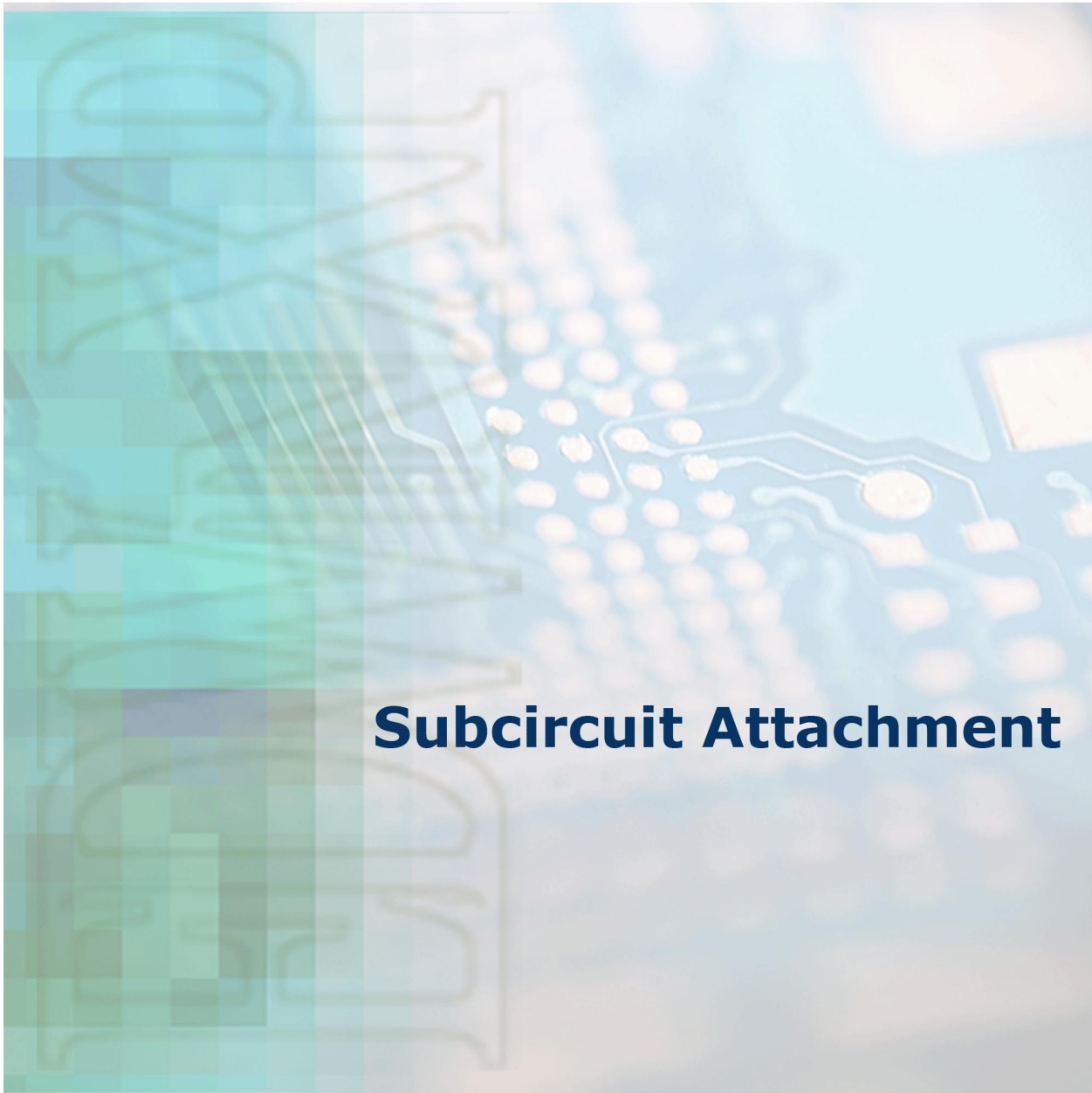


Electronic Design for Windows
EDWIN XP



Subcircuit Attachment

VISIONICS

© Norlinvest Ltd, BVI. Visionics is a trade name of Norlinvest Ltd. All Rights Reserved.

No part of the Subcircuit Attachment document can be reproduced in any form or by any means without the prior written permission of Visionics. Subcircuit Attachment document is subjected to change without notice. Visionics will make changes in a manner that will not affect dependent systems.

Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Visionics. Visionics, EDWinXP, Docone, EDComX, SimWinXP and Mixed Mode Simulator and their respective logos are trademarks or registered trademarks of Visionics. Unauthorized duplication of this work may also be prohibited by local statute.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Visionics. The information contained herein is the proprietary and confidential information of Visionics or its licensors, and is supplied subject to, and may be used only by Visionics's customer in accordance with, a written agreement between Visionics and its customer. Except as may be explicitly set forth in such agreement, Visionics does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Visionics does not warrant that use of such information will not infringe any third party rights, nor does Visionics assume any liability for damages or costs of any kind that may result from use of such information.

Contents

PROCEDURE FOR ATTACHING THE SUBCIRCUIT FILES	4
EDSPICE SYMBOL EDITOR.....	5
ASSIGNING A SUBCIRCUIT TO THE SYMBOL	7

VISIONICS

Procedure for attaching the subcircuit files

In order to simulate a schematic component using EDSpice Simulator, its symbol must be associated with one of the available EDSpice circuit elements or with the subcircuit from EDSpice Subcircuits Library. This application facilitates the association of symbols in the symbol library to the available EDSpice circuit elements or library subcircuits and defining the component parameters. Also EDSpice Simulation references may be added to parts or the already assigned ones may be changed using this module.

Create required component using the Library editor in EDWinXP

Here the assignment of a subcircuit to a regulator is illustrated.

Symbol name: REG

Part name: REG

Package: TO220/3

The screenshot displays the 'Library Editor' window for editing a part named 'REG'. The interface is divided into several sections:

- Part Details:** A table with the following entries:

Name	REG
Prefix	U
Description	REGULATOR
Manufacturer	Generic
Technology	Generic
Type	Analog
External Index Code	
Part Source Library	E:\EDWINXP\LIB\A.Part\REG
- Package Details:** A table with the following entries:

Package	TO220/3
Package Type	PMD
Package JEDEC Name	TO-220
Package Source Library	E:\EDWINXP\LIB\PMD.PACKAGE
- Simulation Parameters:** A section with expandable options.
- Thermal Parameters:** A section with expandable options.
- Constituent Groups:** A table for group configuration:

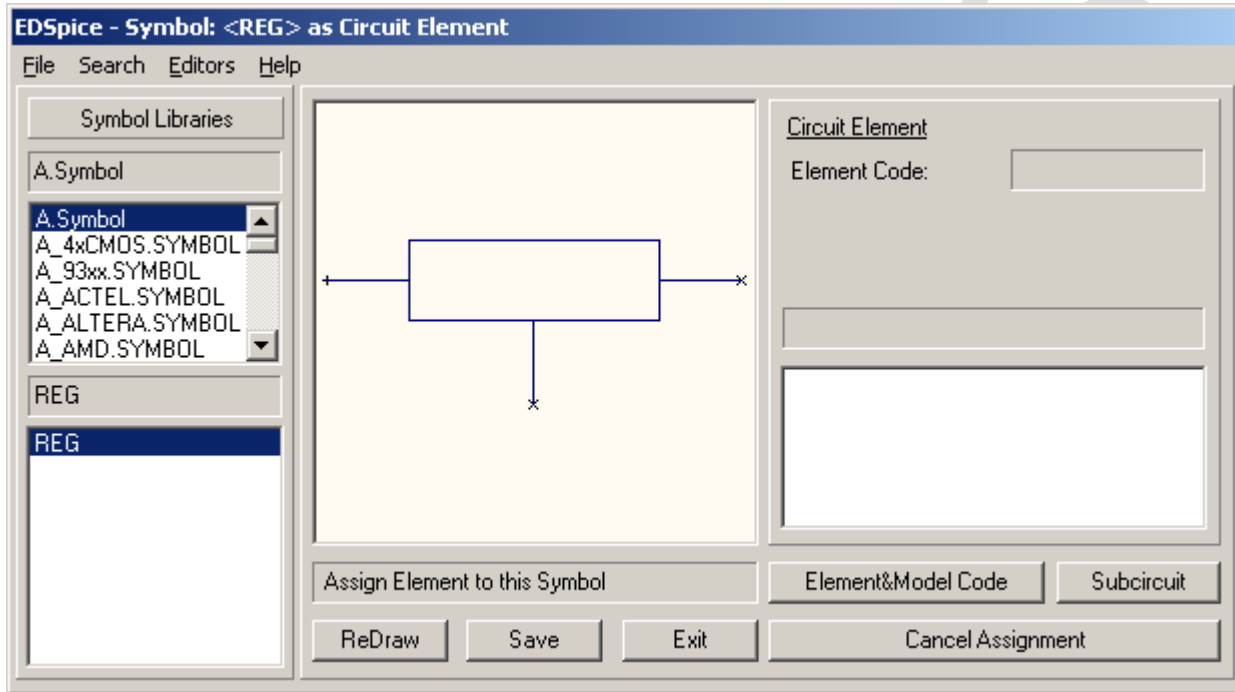
[1] Group : 1 (REG) No UnAssigned	
Group Name	1
Symbol	REG
Un-Assigned Entries	0
MM Simulation Function	0
EDSpice Element Code	X
EDSpice Model Code / SubCircuit	LM7805C
EDSpice Variant / Library	Library: NATIONAL
Symbol Library	E:\EDWINXP\LIB\A.Symbol
- Right Panel:** Shows a schematic diagram of a component labeled 'COMPNAME' with pins 'PA' and 'PDESC'. A dropdown menu shows '(1) Group : 1 (REG)'. Below the diagram, the text 'COMPNAME' and 'COMPDESC' are visible.

At the bottom of the window, there are buttons for 'Part Edit', 'Symbol', 'Package', 'Padstack', and 'Board Cabinet', along with a status bar showing 'Esc'.

Evoking EDSpice Symbol Editor

This module may be invoked from Project Explorer in the following ways.

- Right click Library and select EDSpice **Symbol Editor** from the list.
- Select EDSpice **Symbol Editor** from the Task list or from the Task toolbar.



The next step is to assign a subcircuit for the component. Click on "**Subcircuit**". A window as shown below will popup.

The model as well as subcircuit library structures support the family approach of grouping the models. There are two purposes to group them as *family libraries*.

1. The user may find the models/subcircuits easier.
2. It may happen that model parameters/subcircuits have the same identifier.

For example BC807 from two manufacturers, Motorola and NS. In such cases they may be stored with the same identifier in different family library file and still be unique. This family approach of grouping the models and subcircuits are implemented using family prefixes.

Family prefix, if added to the Simulation Reference, makes the search by EDSpice (with the context as Simulation Reference), faster. In the case of models, the family prefix is the model parameters library file name without the extension. The Family Prefix in the case of subcircuits is the subdirectory under EDS_SBK where the subcircuit is stored.

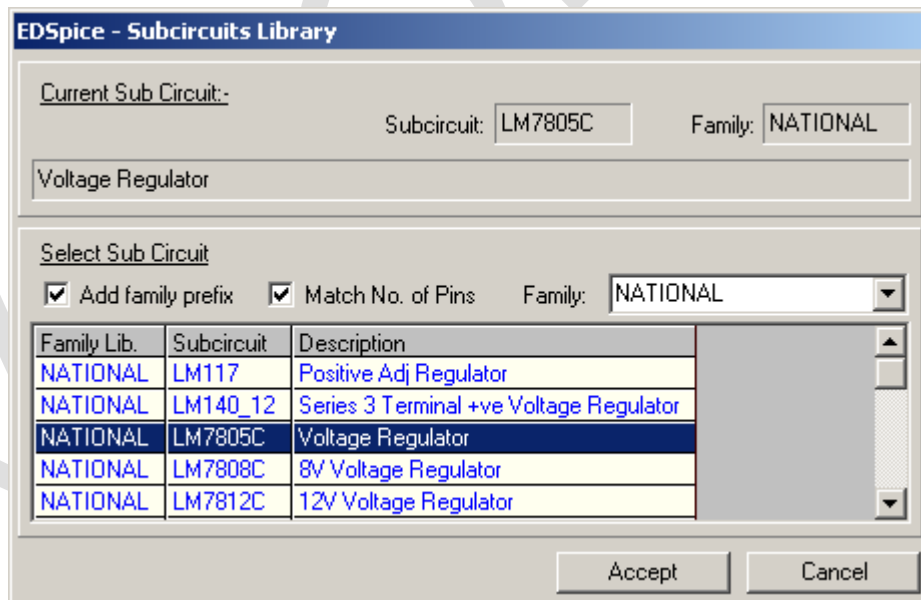
For subcircuits:

Consider the subcircuit TL031_10 adopted from TI. Let this subcircuit be stored in the directory /EDS_SBK\TI_LIN. Here the search string is:-

@TL031_10 - without Family prefix

@TI_LIN.TL031_10 - with Family prefix

When the program searches for subcircuits then in the first case it would go through all subdirectories under EDS_SBK until the subcircuit is found or not. In case when prefix is specified (which is optional) then the search as above is proceeded by attempt to access subcircuit TL031_10 directly in /EDS_SBK\ TI_LIN. If subcircuit is found then OK, otherwise the search continues. Generally it works in similar manner as with model parameters.



Assigning a Subcircuit to the Symbol

The symbol can be assigned with a subcircuit by the following steps:

1. Select the appropriate library subcircuit from the list. To get the list of subcircuits in the library press. If the check box "Add family prefix" is checked, the prefix gets added to the symbol. By checking the list may be filtered by displaying only the subcircuit with the exact number of pin the symbol has
2. Assign element nodes to the appropriate entries of the symbol.

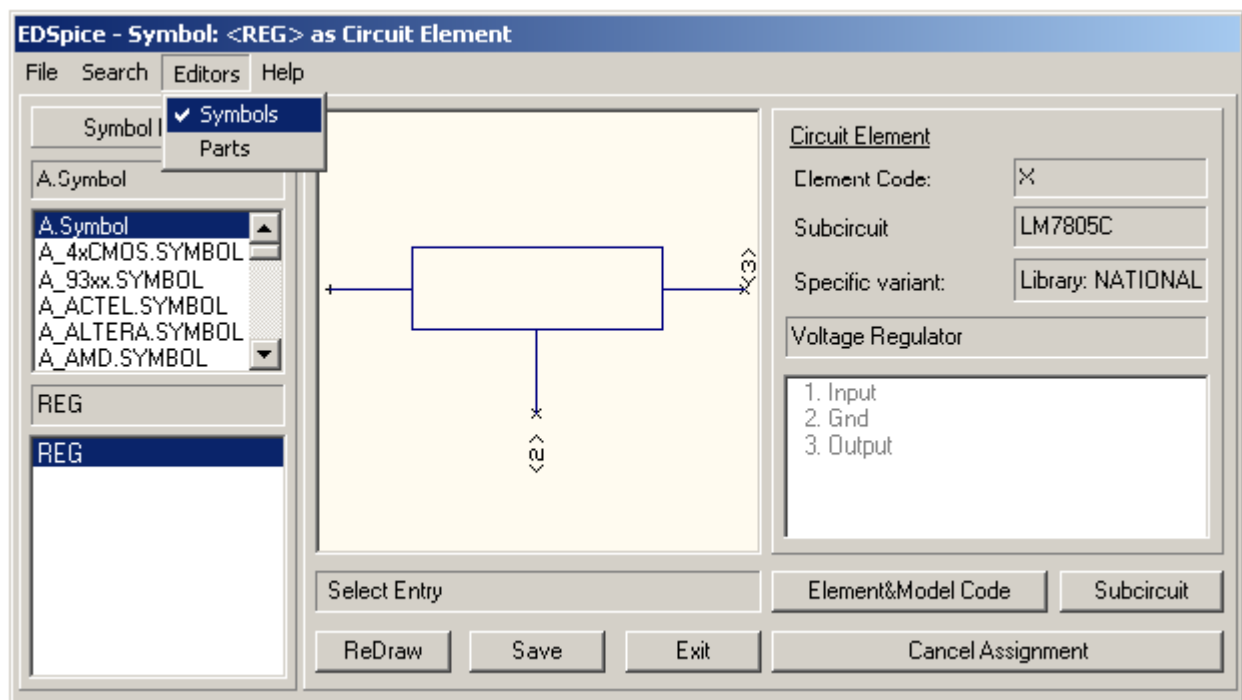
When a subcircuit has been selected, the list of nodes, as expected by EDSpice is displayed.

The sequence of operations is as follows:

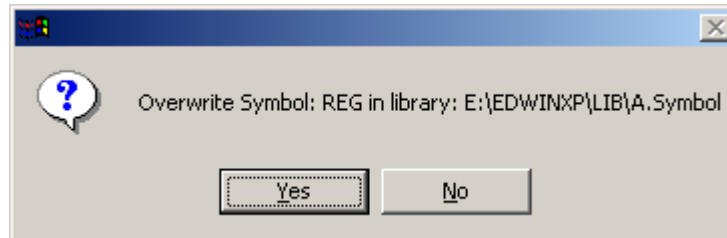
- position the cursor on the entry and click the mouse. The operation is confirmed if the selected entry is marked by a rectangle.
- select the appropriate node for this entry from the nodes list. The operation is confirmed if the entry is marked with the number of the node selected from the list.

The same operation must be repeated for all entries in the symbol.

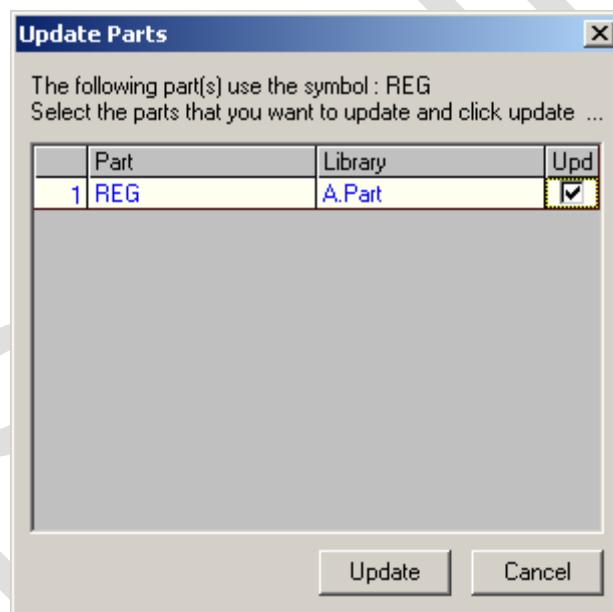
Thus the symbol will be assigned with the subcircuit.



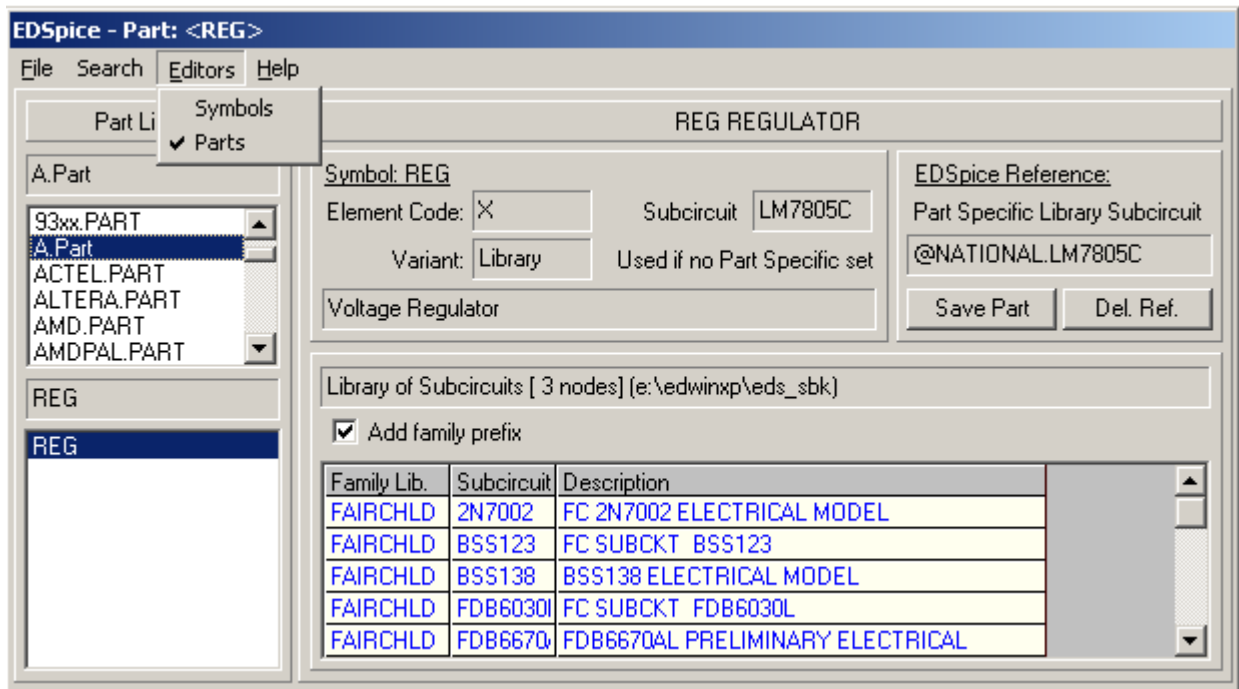
Save the symbol using "Save". Select YES. The symbol REG is saved in the "A.Symbol" library.



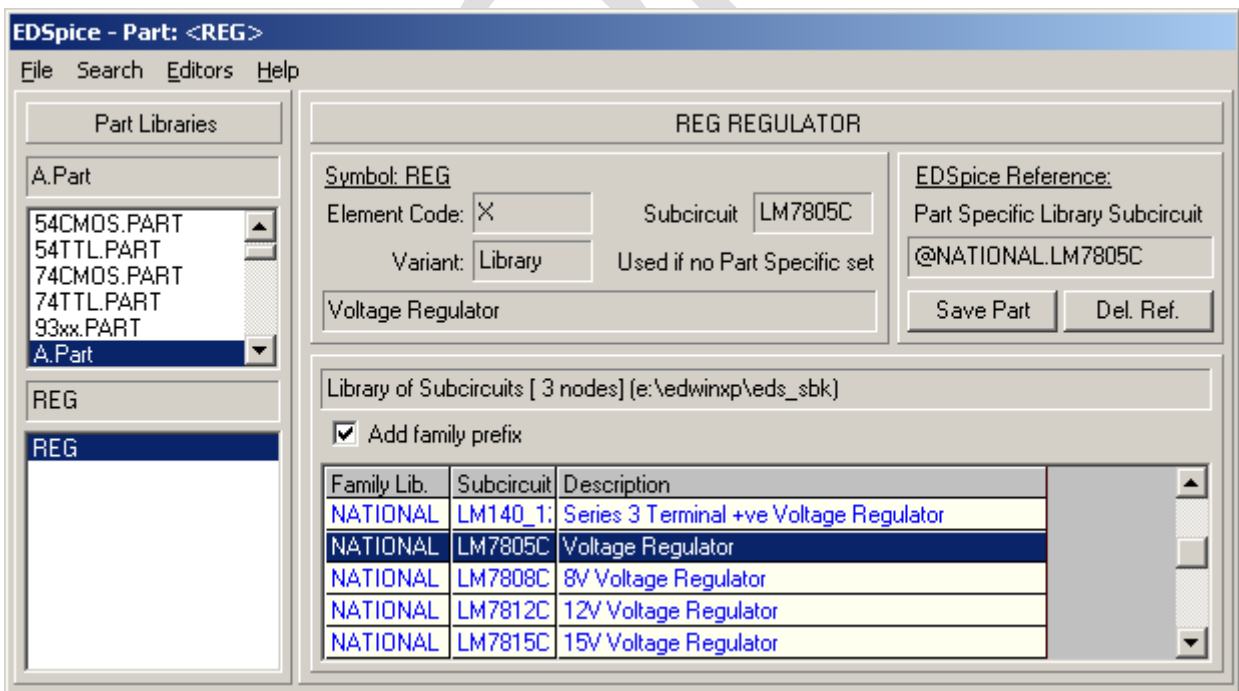
Click on "Update" so that the part get updated. The part REG is saved in the A.PART.



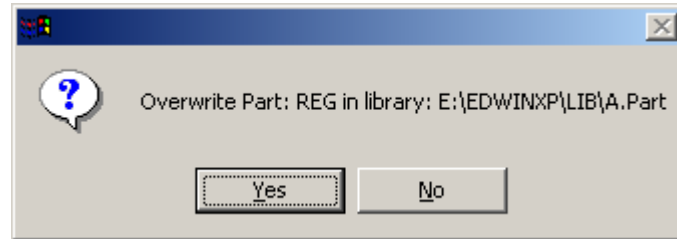
Select **Editor ->Parts**. Thus the Part library gets loaded to the EDSpice Symbol Editor out of which the required part is selected.



Search the Part in the Part library.

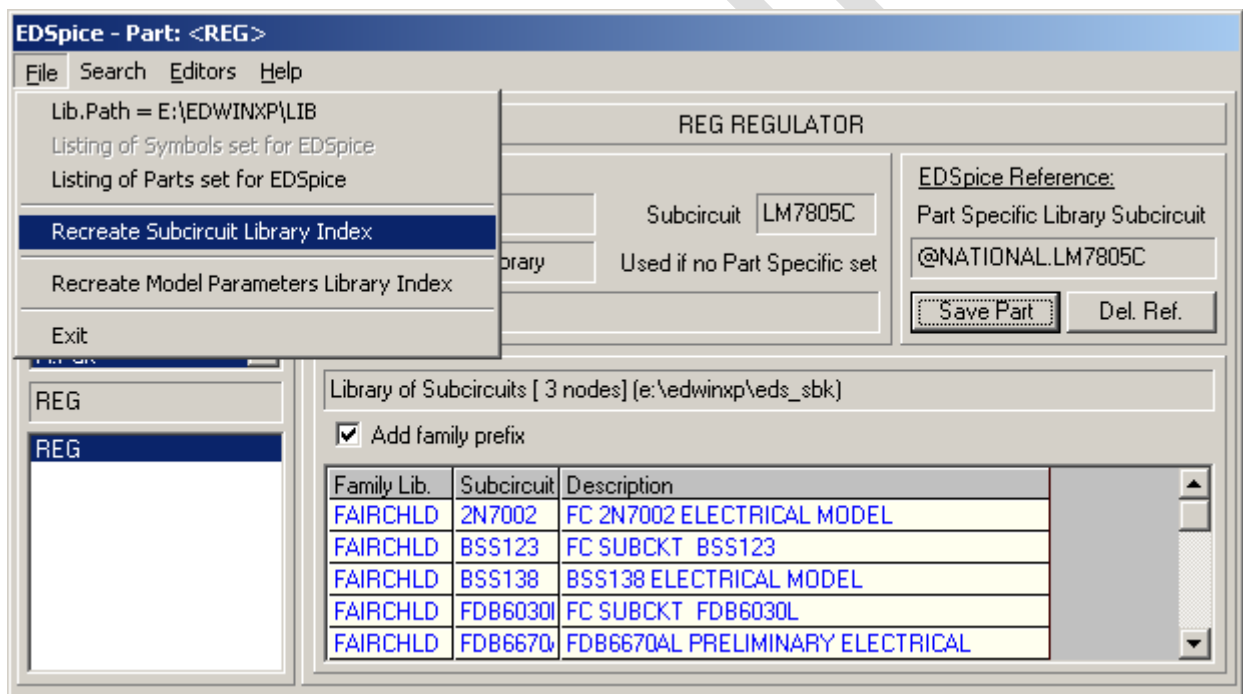


Save the Part by clicking on the "Save Part".



Recreate the Subcircuit index for the updated component to get attached with the parameter.

File -> "Recreate Subcircuit"



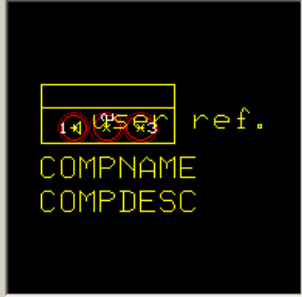
From the **Library Editor -> Simulation Parameters -> EDSpice Simulation Model/ Subckt** indicates the subcircuit attached to the symbol and EDSpice Simulation Ref refers to the attached subcircuit to the Part.

Library Editor (Editing Part : E:\EDWINXP\LIB\A.Part\REG)

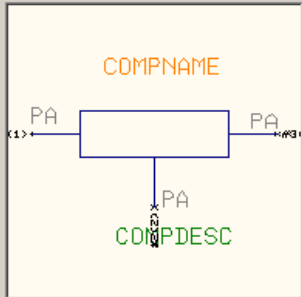
File Edit View Help

Part Details	
Name	REG
Prefix	U
Description	REGULATOR
Manufacturer	Generic
Technology	Generic
Type	Analog
External Index Code	
Part Source Library	E:\EDWINXP\LIB\A.Part
Package Details	
Package	T0220/3
Package Type	PMD
Package JEDEC Name	T0-220
Package Source Library	E:\EDWINXP\LIB\PMD.PACKAGE
Simulation Parameters	
MM Simulation Function	0
EDSpice Simulation Model/Sub Ckt.	X\LM7805C\Library: NATIONAL
EDSpice Simulation Ref	@NATIONAL.LM7805C
Thermal Parameters	

Package : T0220/3



(1) Group : 1 (REG)



Constituent Groups	
(1) Group : 1 (REG) No UnAssigned	
Group Name	1
Symbol	REG
Un-Assigned Entries	0
MM Simulation Function	0
EDSpice Element Code	X
EDSpice Model Code / SubCircuit	LM7805C
EDSpice Variant / Library	Library: NATIONAL
Symbol Library	E:\EDWINXP\LIB\A.Symbol

Part Edit Symbol Package Padstack Board Cabinet

Esc