



SPICE Netlist Import

VISIONICS

© Norlinvest Ltd, BVI. Visionics is a trade name of Norlinvest Ltd. All Rights Reserved. No part of the SPICE Netlist Import document can be reproduced in any form or by any means without the prior written permission of Visionics. SPICE Netlist Import 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

SPICE NETLIST IMPORT	4
INVOKING SPICE NETLIST IMPORT	4

VISIONICS

SPICE NETLIST IMPORT

This utility allows to import both circuit (.cir) and subcircuit (.sbc) files to the system. The Schematic diagram is generated by converting the imported file data to the project. All the Instance parameters and Model parameters are set during conversion. The circuit is also set for various analyses. All the parameters are set for various types of analysis, which is specified in the circuit file. This routine deletes the currently loaded project upon start. Hence it is to be ensured that the database is saved before invoking this application.

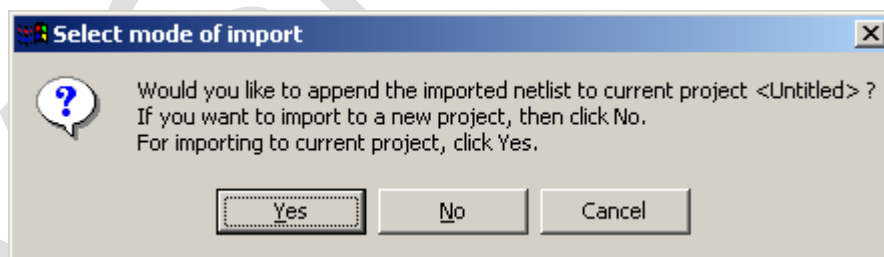
Invoking Spice Netlist Import

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

- Right click Project and select Import Spice Netlist from the list.
- Select Import Spice Netlist from the tasklist or from the task toolbar.

Note: By default, system does not display the task toolbar. It may be enabled from View menu in the Project Explorer.

The system prompts for saving the project. A window pops out to select the mode for importing the files. The imported files may be created as a new project or as new hierarchies in an existing project.



Click **No**

The Spice Netlist Import window appears. Spice Netlist Import collects all the parts information in the library at the time of form load. It keeps a table of these information's for extraction of parts while importing. Select File | Open. A dialog box pops up from where the required .cir or .sbc file can be selected. The project is named after importing the file.

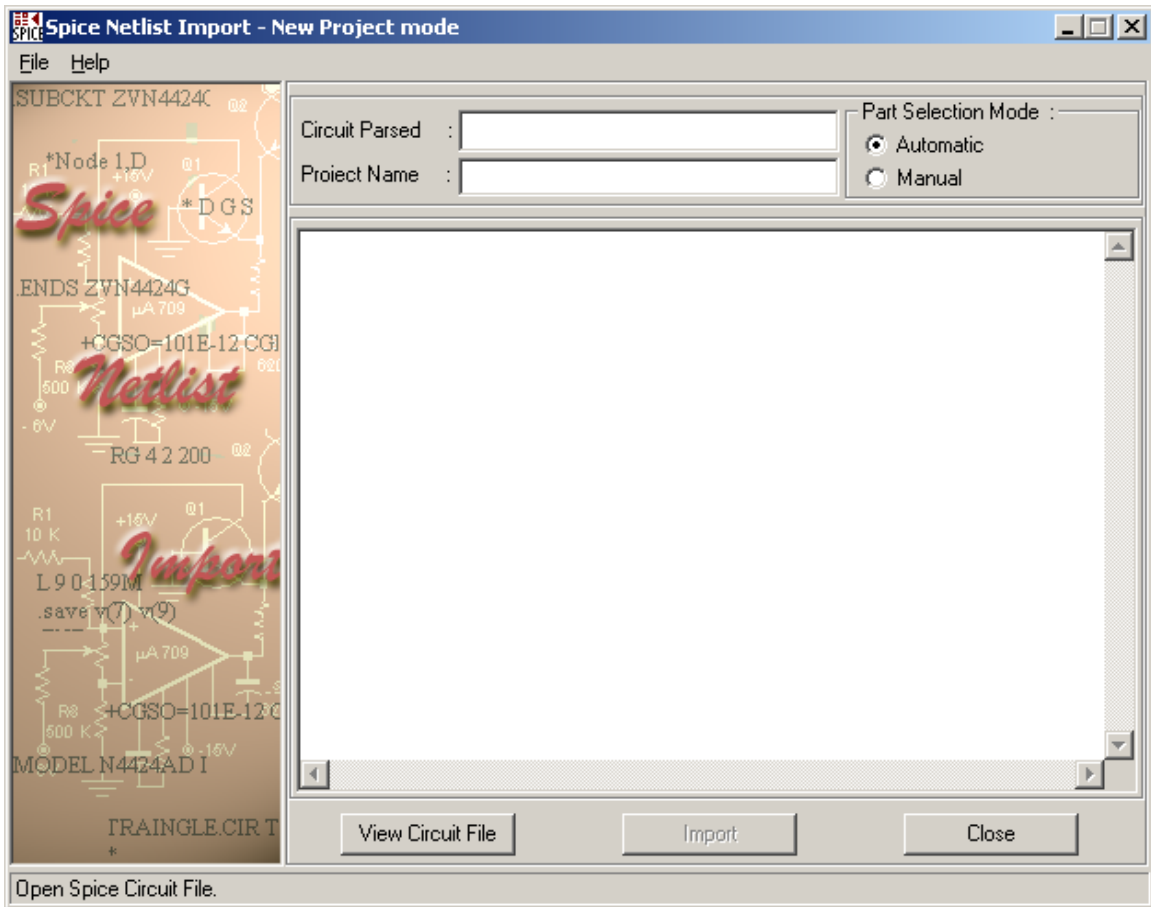
While importing, there are two modes for part selection.

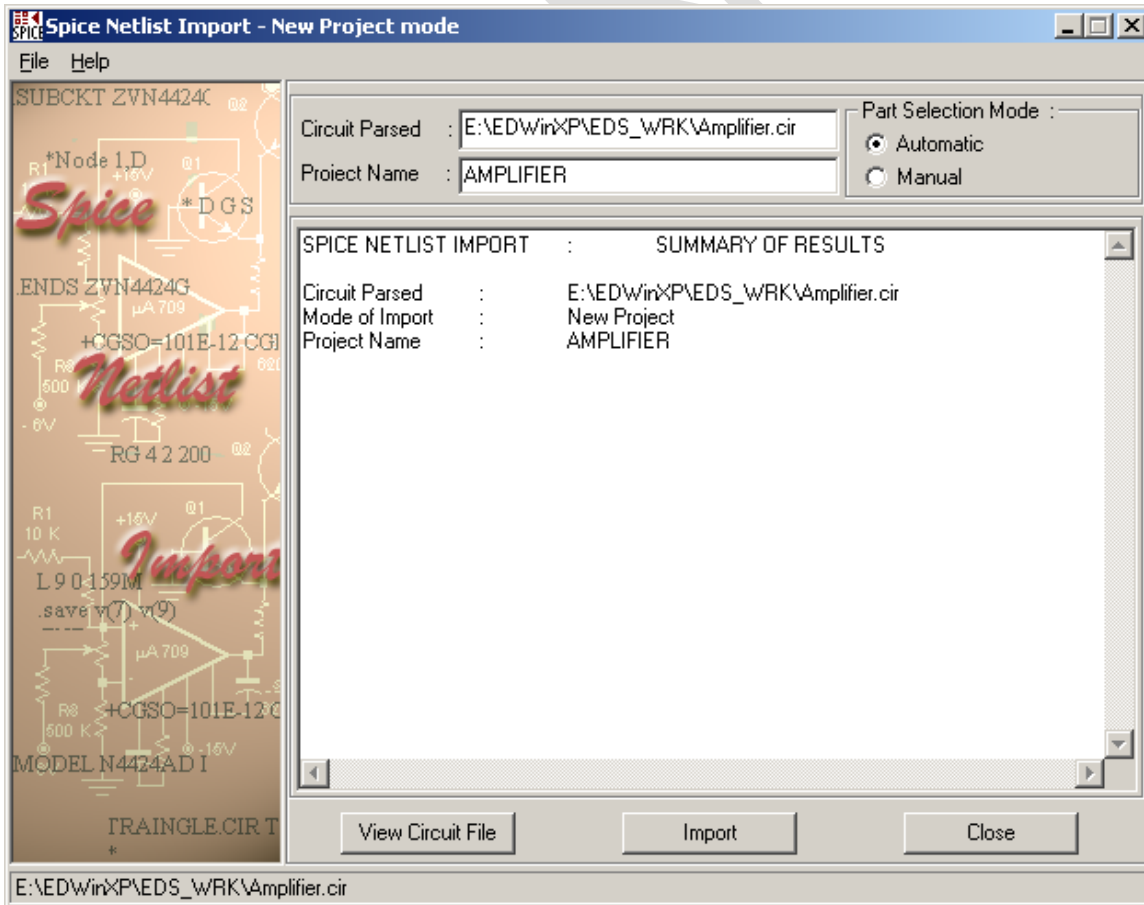
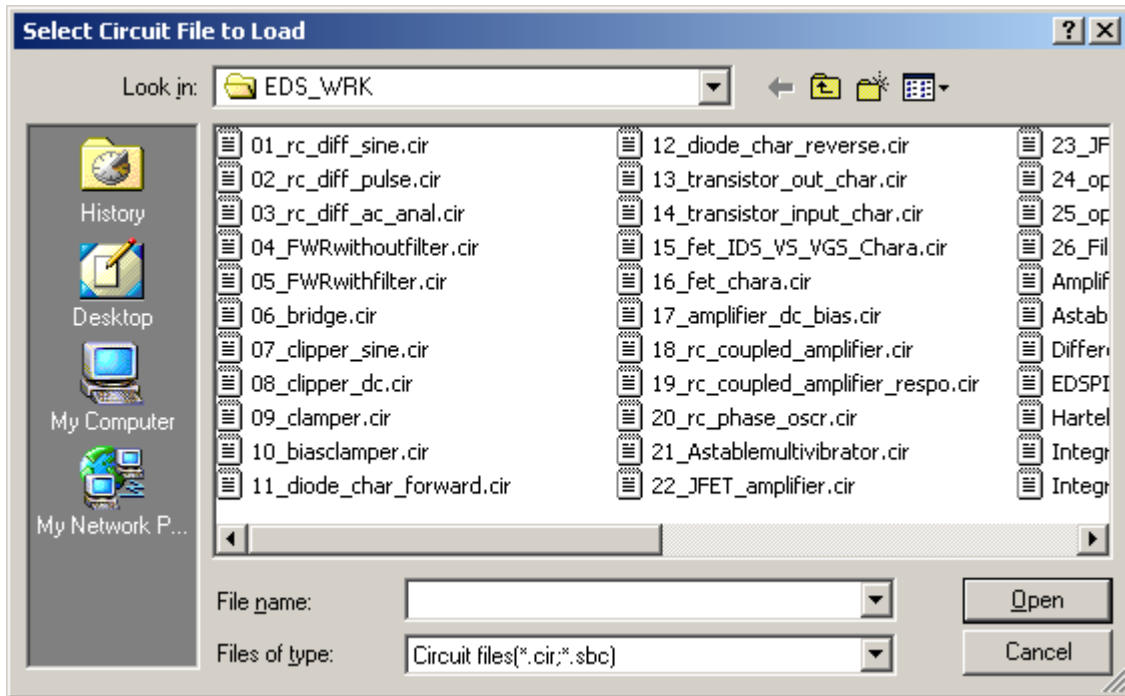
Automatic: In this mode the system searches and selects the first component that matches the criteria in the circuit file definition and loads it to the project.

Manual: In this mode system lists all the parts satisfying the conditions in a table, from which user can select one. Similarly for all the definition of various components in the circuit file, several components are listed. The selected components are loaded to the project.

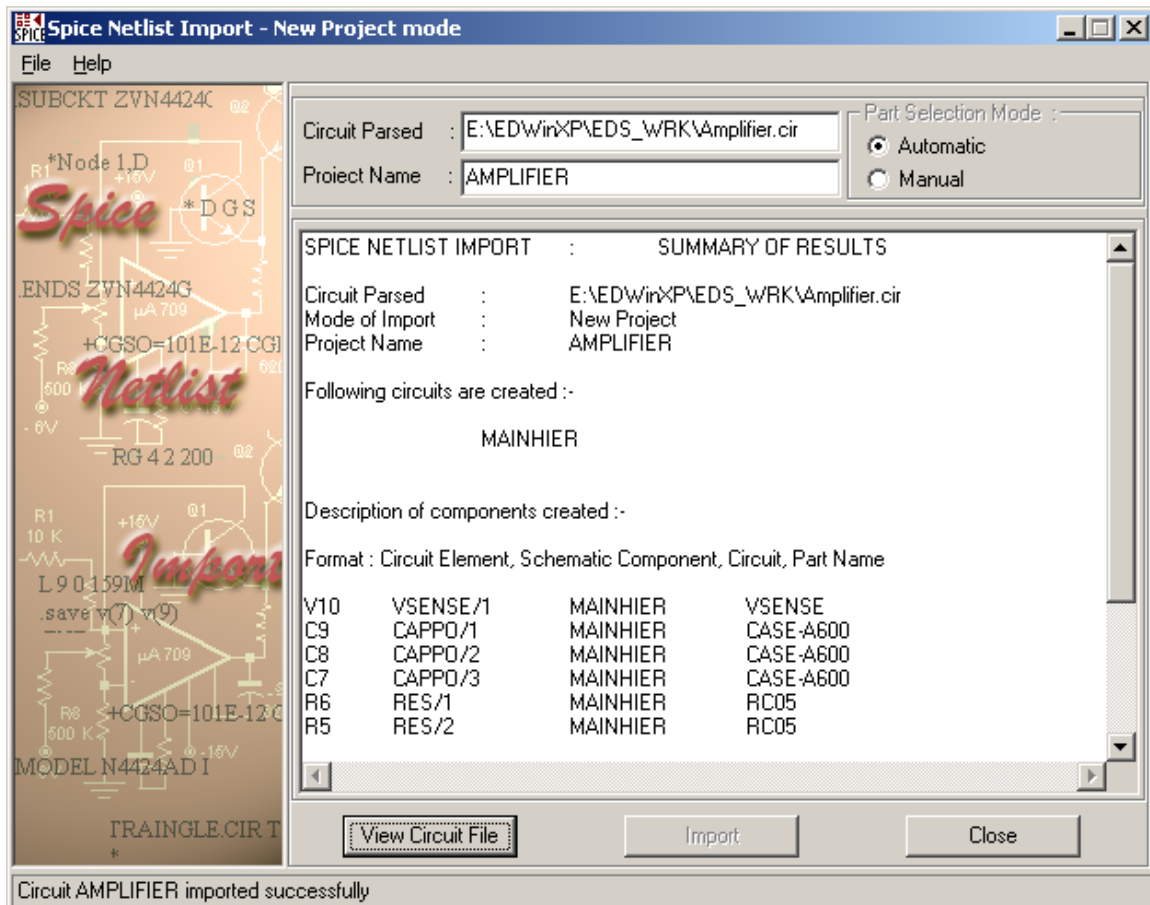
If none of the parts in the library satisfies the search criteria, then system asks the user whether to create a dummy part to keep the netlist correctly. Or user can ignore this part creation and the related nets remains partially connected. Dummy parts created can be recognized by their formatted names.

In case of subcircuits system creates a part and assigns the corresponding hierarchy to this part automatically. Parts like this also appears with formatted names.

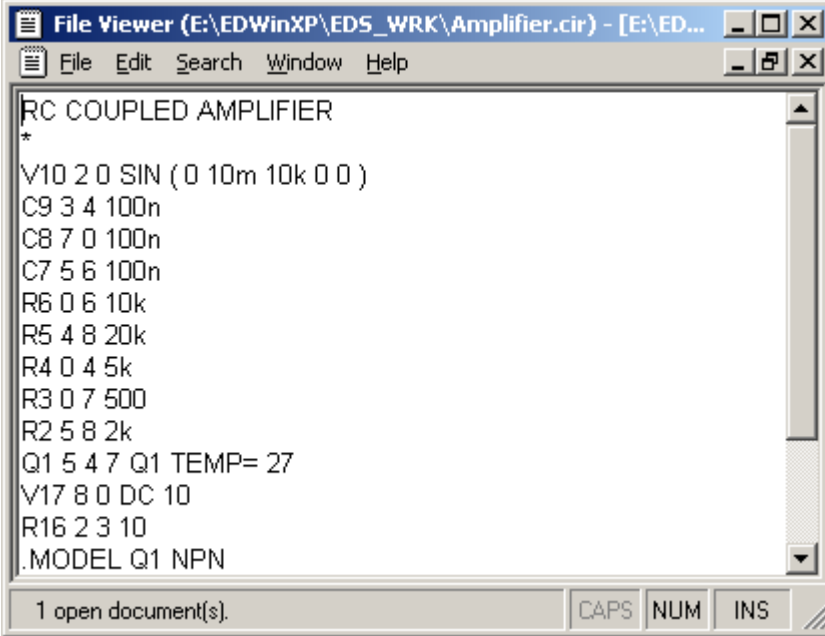




After loading all the components, click the **Import** button to import these components and netlist information to the project. The project is build within a short time.



Click the **View File** button to view circuit file in a file viewer.



The screenshot shows a 'File Viewer' window with the following text:

```
RC COUPLED AMPLIFIER
*
V10 2 0 SIN (0 10m 10k 0 0)
C9 3 4 100n
C8 7 0 100n
C7 5 6 100n
R6 0 6 10k
R5 4 8 20k
R4 0 4 5k
R3 0 7 500
R2 5 8 2k
Q1 5 4 7 Q1 TEMP= 27
V17 8 0 DC 10
R16 2 3 10
.MODEL Q1 NPN
```

The window title is 'File Viewer (E:\EDWinXP\EDS_WRK\Amplifier.cir) - [E:\ED...'. The status bar at the bottom indicates '1 open document(s)' and has buttons for 'CAPS', 'NUM', and 'INS'.

Click the **Close** button to exit Spice Netlist Import window.

Invoke the Schematic Editor, to view the net connection using ratsnets and start routing manually or using Autorouter.

All information's like node modifiers, instance/model parameters, transient initial conditions, Initial node voltage guesses, set up for all types of analysis etc are read from the file and are created in the newly created project. All instance/model parameters having a spice name are successfully imported.

.SAVE, .PRINT, .PLOT commands are ignored.

