

Netlist / Wirelist Export Import

The procedure to import EDWinXP Schematic Netlist is as described below

Before importing the files, copy and paste the *.wrs and *.wrl files in the JOB directory also the component names in the wrs file should contain the name of the part in the EDWinXP library.

- ? Select **Project -> Netlist/ Wirelist Export & Import**
- ? Choose **File -> Import**
- ? Select **EDWinXP Schematic netlist** for the "Import parameter"-> **Format**
- ? Browse and select the required project file whose schematic is to be generated. It's optional to choose the 'Part Dictionary name'. Click on **Import**

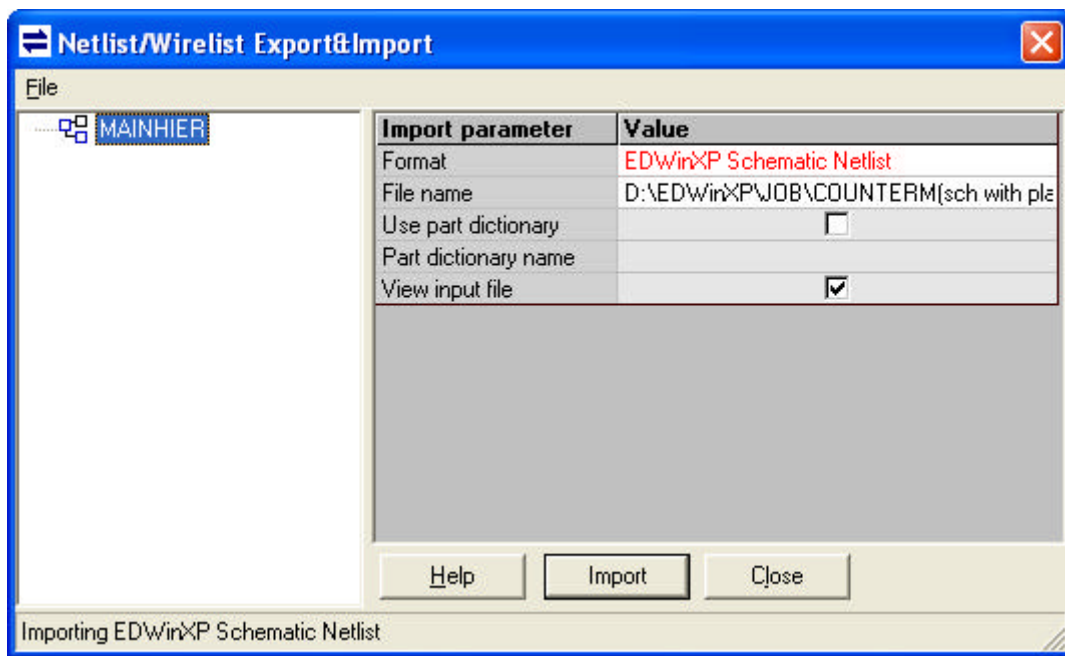


Fig 1

- ? After importing the wrs file, the *wrs file* as well as the *import log* file will be generated. These are given in the below two images (Fig 2 and fig 3)

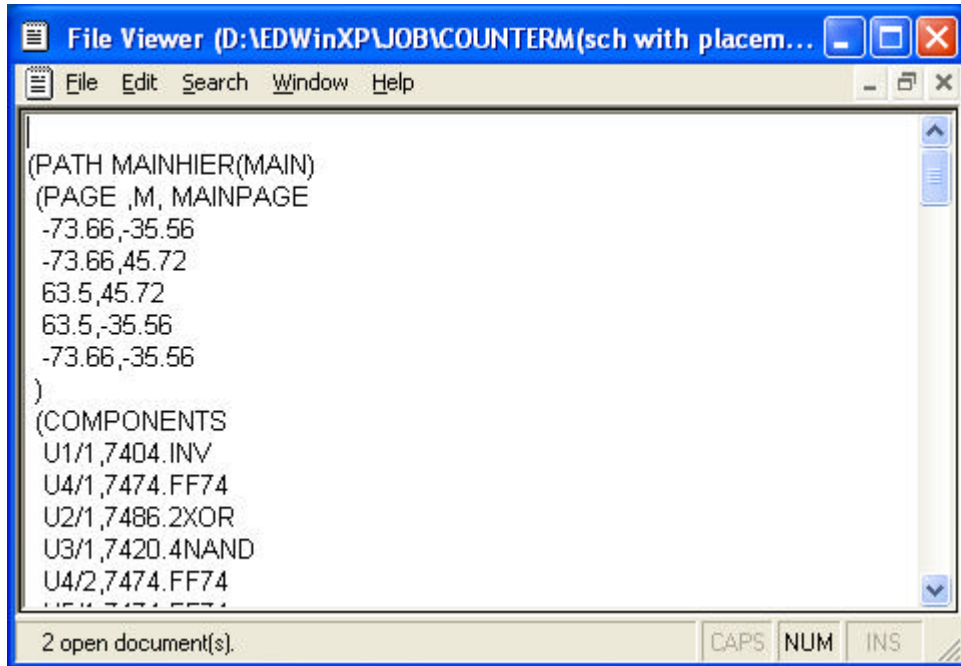


Fig 2

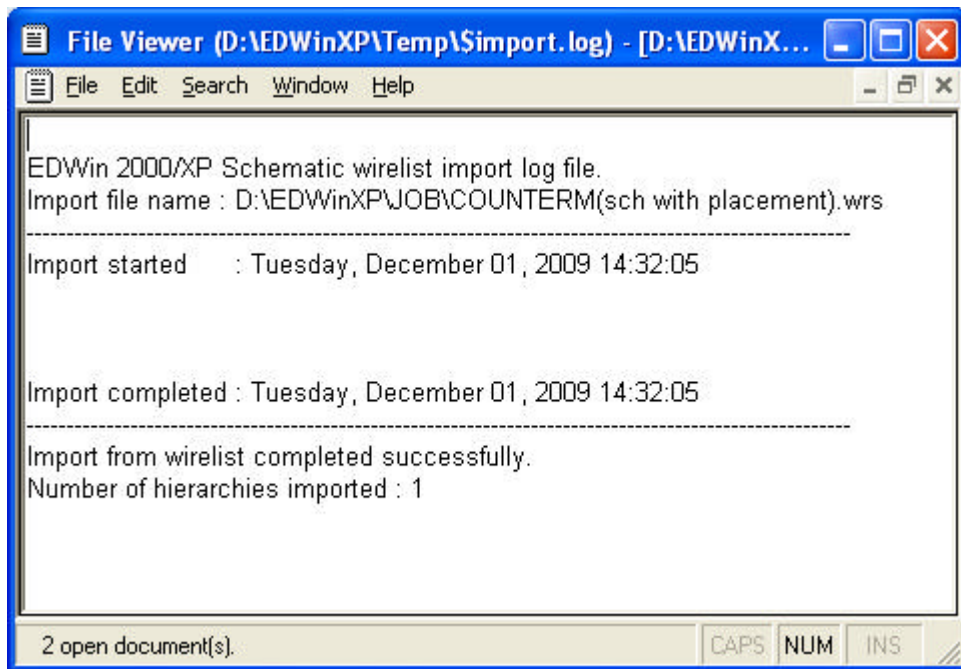


Fig 3

- ? After the import is completed, invoke the **Schematic Editor**. You can view the components are generated and will be stacked at the middle of the page. This is as given below (fig 4)

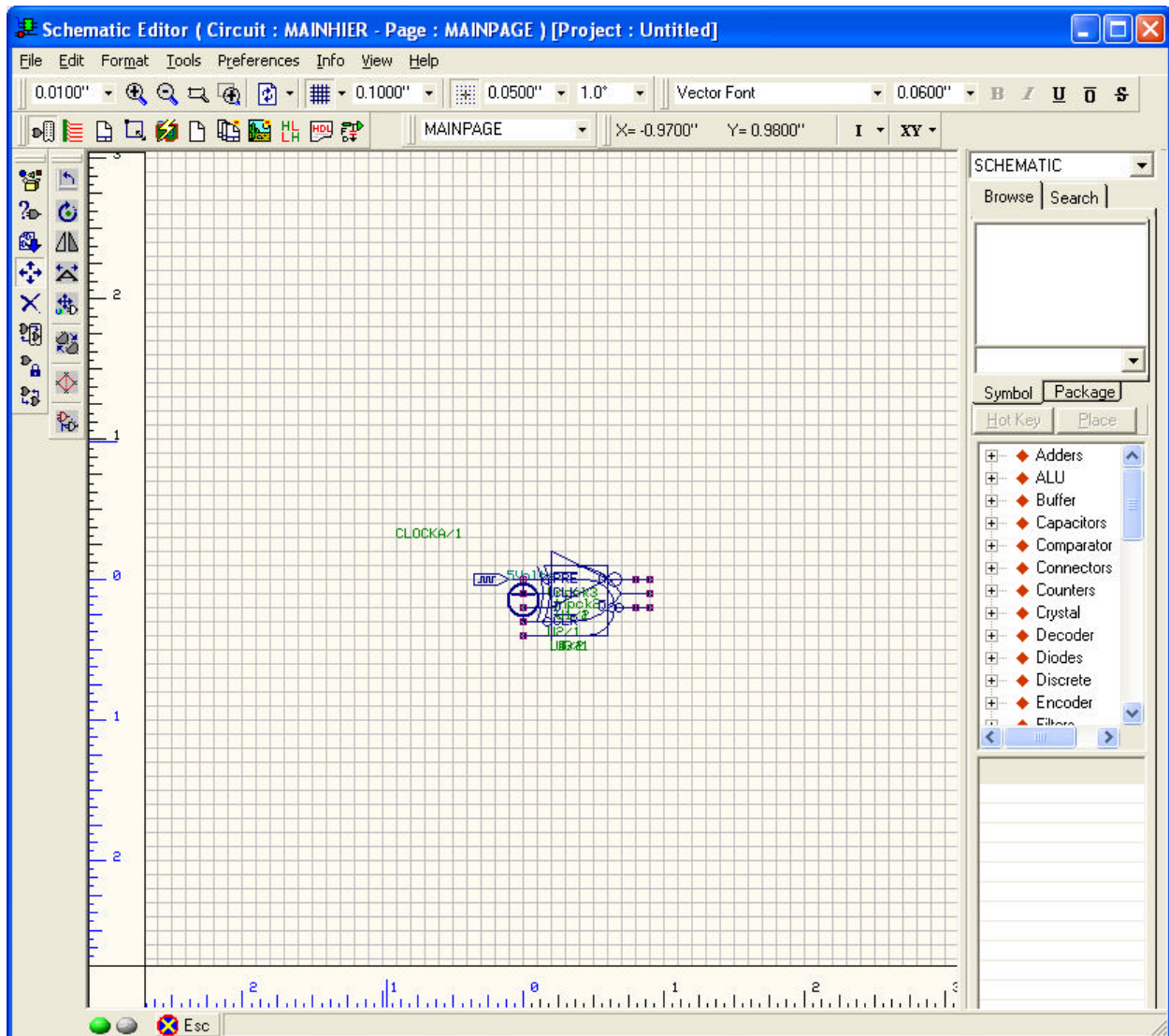


Fig 4

- ? In order to place the components in the Schematic Editor, you can autoplacement the components using the Autoplace option. This function can be accessed from Tools -> Autoplace. Before autoplacement, the **Placement parameters and Design rules** has to be set first which can be accessed from **Autoplace parameters -> Display parameters**
- ? For that invoke **display parameters** and select '**Placement Rules**' and set the parameters as given in the image below (fig 5)

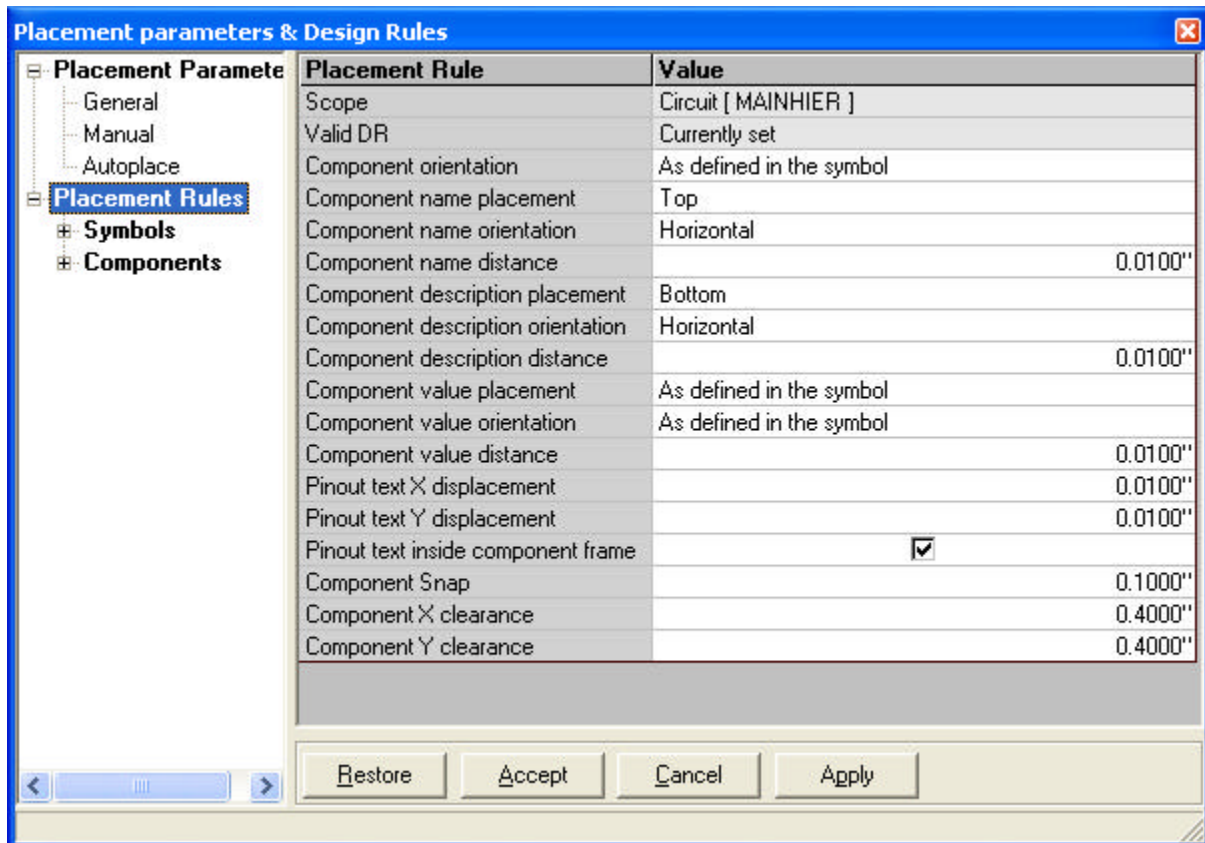


Fig 5

- ? After giving all the necessary input to the **Value**, click on **Apply** button so that all the parameters get applied.
- ? Then select '**Placement Parameters**' -> **Autoplace** and then set the values as given in the below image (fig 6). We are following the *Topological placement of components*. Hence set the *Sequence* as given
 - ? Pre-place at start - *only 'input' side components*
 - ? Autoplace remaining components - *attempting to follow signal flow*

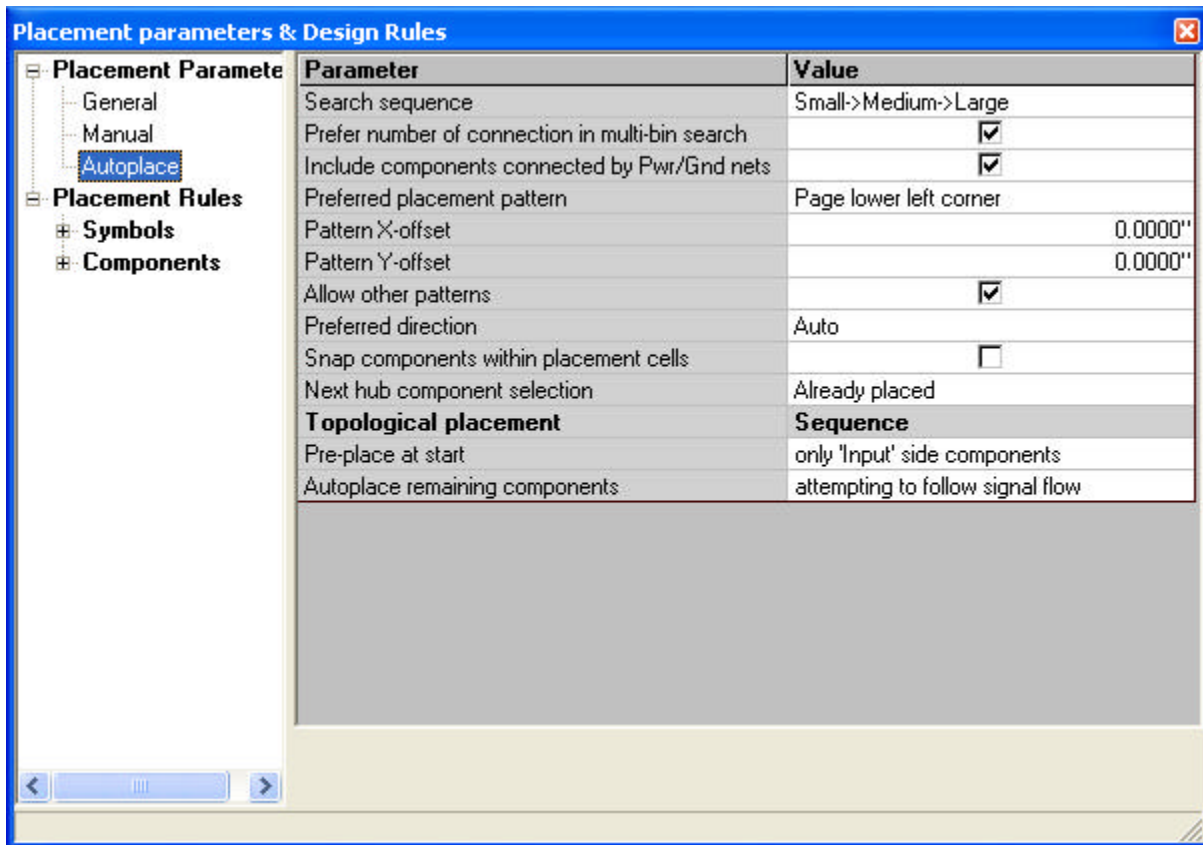


Fig 6

- ? Then select **Autoplace all components** (fifth function tool)-> **Autoplace topologically**

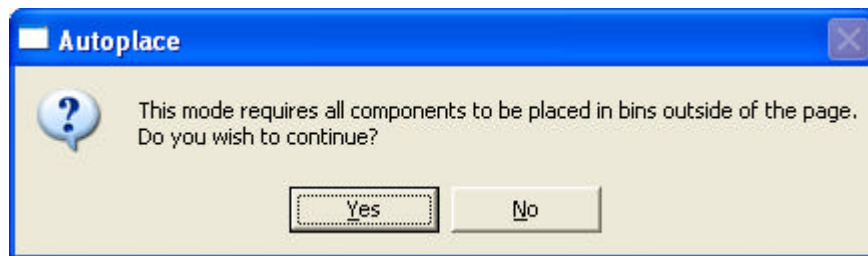


Fig 7

- ? Click on **Yes** to proceed with the autoplacing. The autoplaced components will be distributed in the schematic as shown below (fig 8).
- ? For autorouting of the components, select Tools-> Connections -> Connect Components (first function **tool**) -> **Autoconnect all wires (F6) (sixth option tool)** (fig 9).

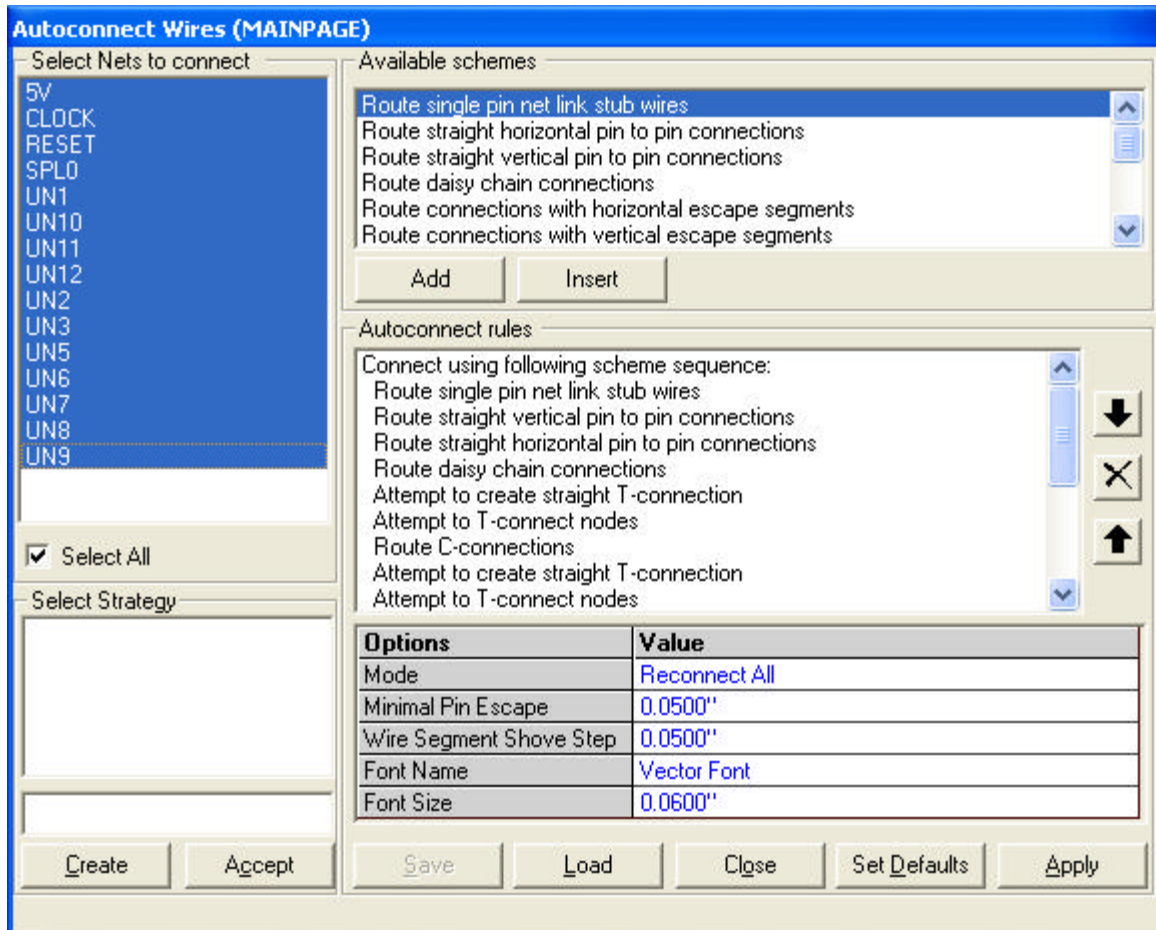


Fig 9

- 2 Choose Select All check box to select all the nets. Then click Apply -> Close (Fig 9).
- 2 The circuit will be autorouted as in the given image below (fig 10).

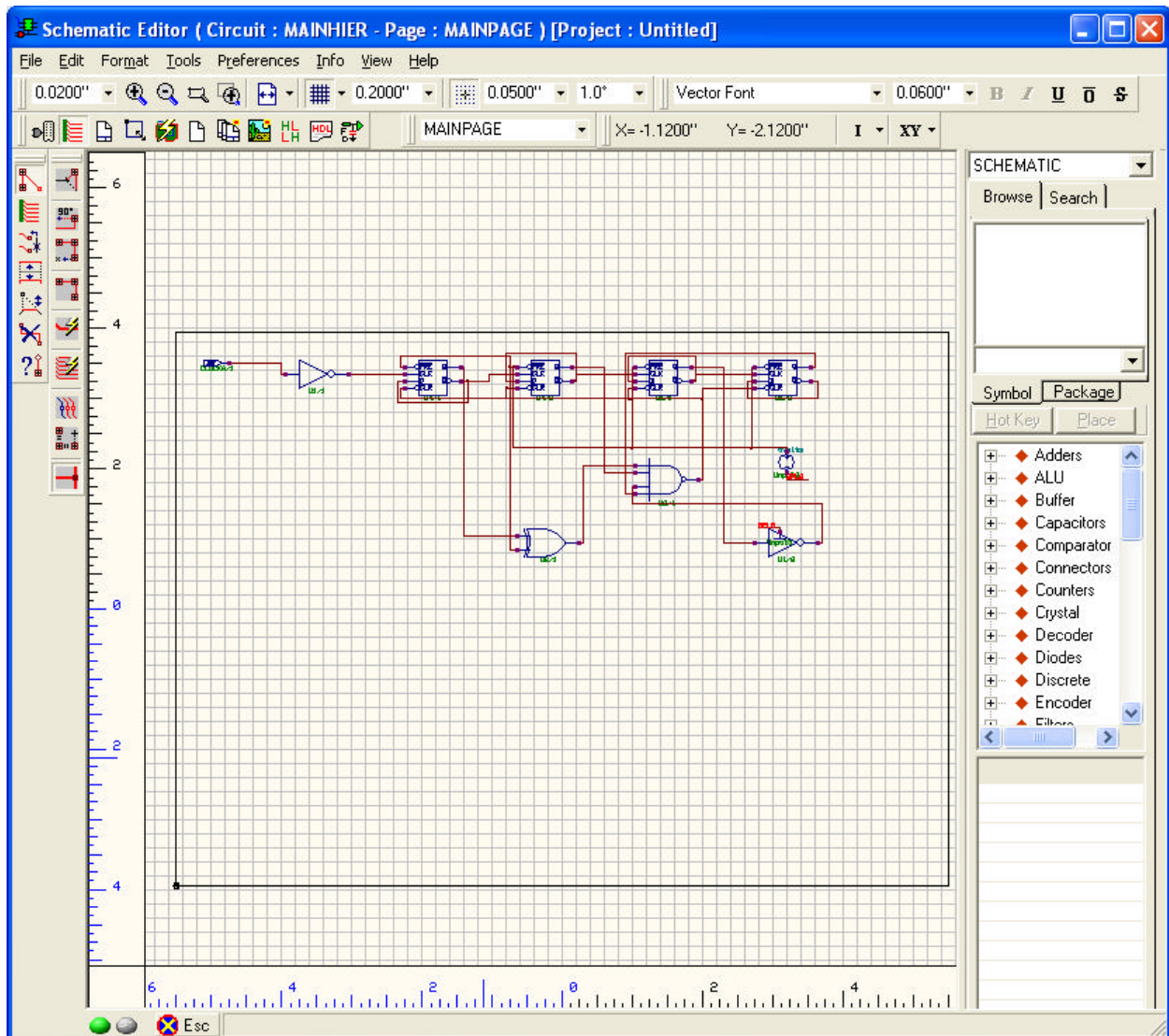


Fig 10

The same principle is followed to import a Layout Wirelist file. Here the procedure is carried out as below

- ? Select **Project -> Netlist/ Wirelist Export & Import**
- ? Choose **File -> Import**
- ? Select **EDWinXP PCB Wirelist** for the "Import parameter"-> **Format**