Laboratory 2.
Introduction to LTspice (simulator of electronic schematics and diagrams)

Aim of this laboratory

Learn how to:
- Create, open and delete a file designed in LTspice
- Edit the electronic circuit's schematic using LTspice libraries

Necessary equipment

- Work stations with LTspice installed

Theoretical Approach

Building a project

Launch LTspice from Windows Programs:
1) New schematic
2) Save your project on C:\Temp\My_Folder
3) Using the STspice libraries implement the electronic scheme from Fig.1

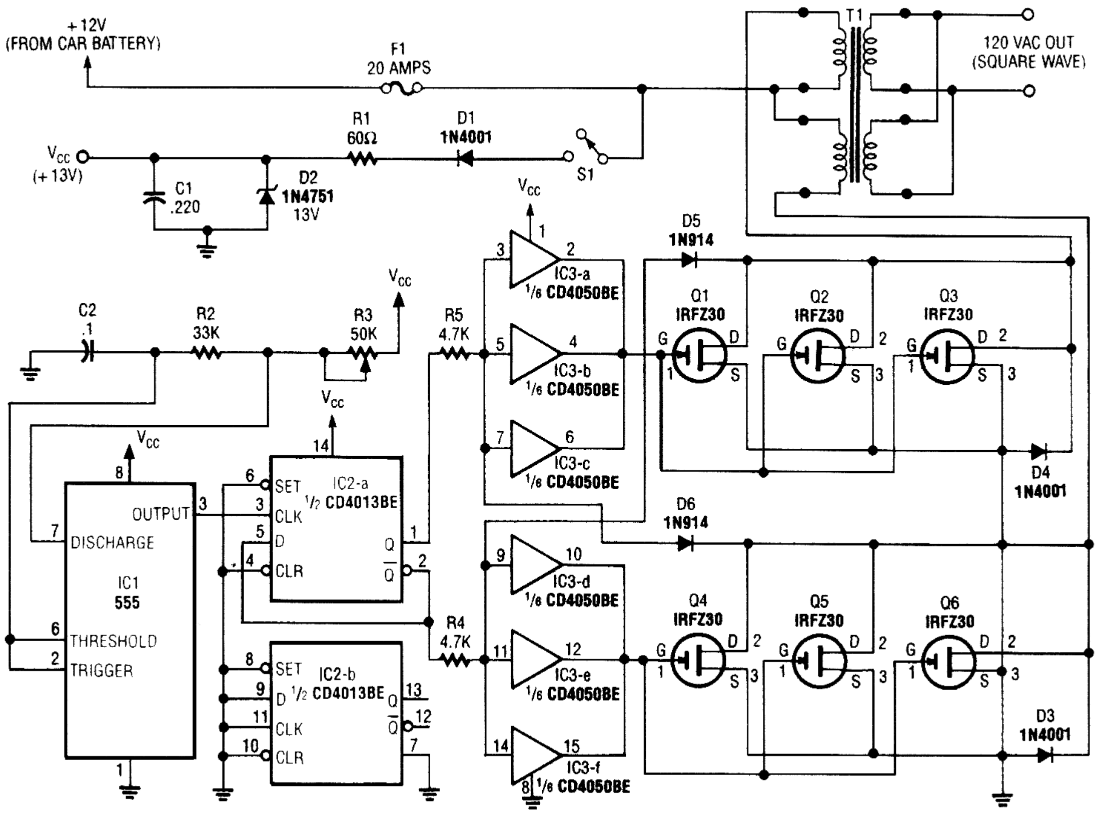

Figure 1. Electronic scheme

4) For the components that you cannot find in the LTspice libraries create your own components, like this:

- New schematic
- Save your project (recommended to be your component name)
- From "Hierarchy" menu, choose "Create a New Symbol"
- Draw and place text on your component with options from the Draw menu.
- Save your component in the same directory with the project and give the same name.

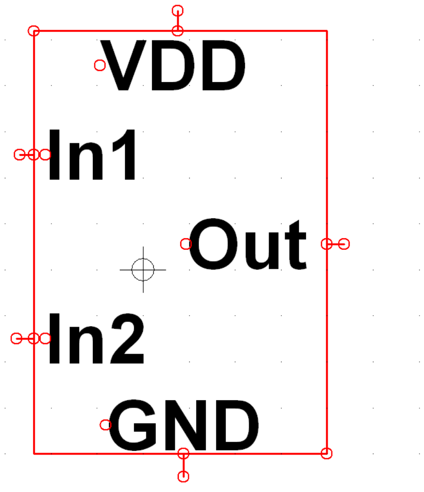

Figure 2. Using the "Create a New Symbol" option

5) You can import your components in the same maner like the other LTspice components.

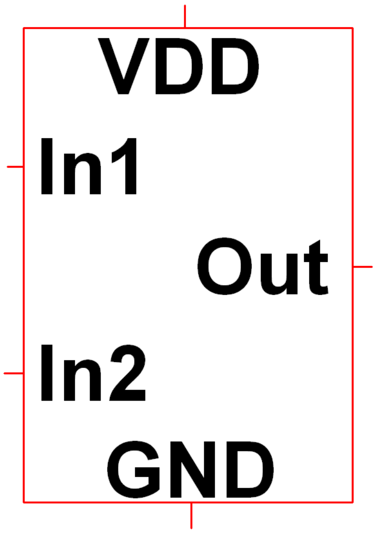

Figure 3. The created component, imported in the project