



**GDAŃSK UNIVERSITY
OF TECHNOLOGY**

FACULTY OF ELECTRICAL AND CONTROL ENGINEERING
Introductory CDIO Project, PG_00049763

Exercise name:
Printed circuit design

Instructions for the laboratory exercise

Patryk Strankowski, Jarosław Guziński

1. Aim of the exercise

The purpose of the exercise is to prepare a printed circuit board (PCB) design using EAGLE.

2. Description of the EAGLE program

Currently, there are many programs for designing printed circuit boards varied options and price. The most popular of them are: Protel, OrCad, Tango and EAGLE. In hobby applications, the EAGLE program dominates, for which it is available there is a free version (so-called freeware) .

The free EAGLE version can be used for non-commercial projects. Dimensions of the designed circuit are limited to a 100 mm × 80 mm plate. There can be one tile or double layer. In addition, the program is fully functional without time limits.

The package includes three basic modules:

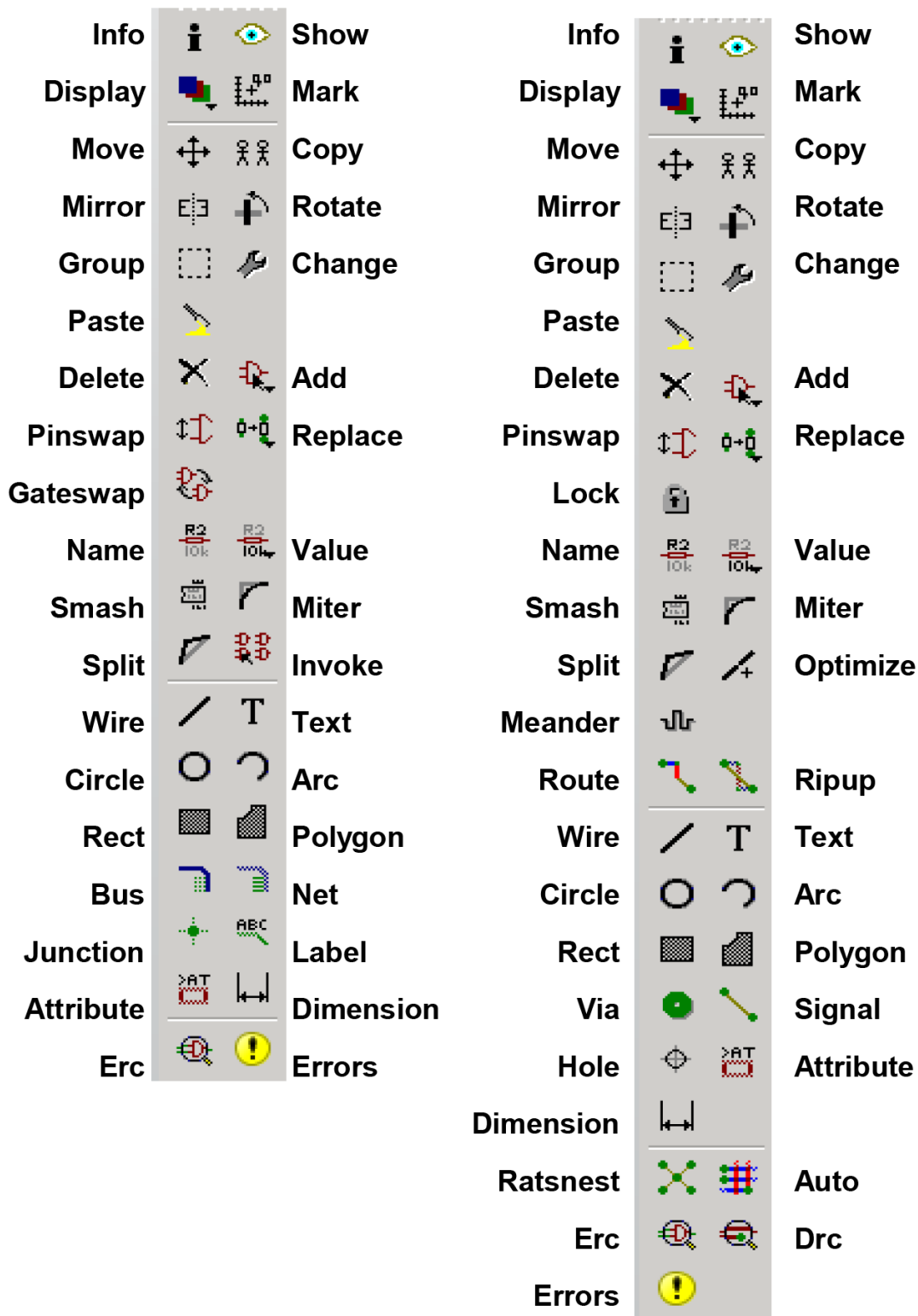
- *Control Panel* - a control panel for managing and facilitating file management work with large projects,
- *Schematic* - a module for creating and editing diagrams,
- *Board* - a module for creating and editing printed circuit boards.

Further descriptions of the EAGLE program operation refer to the system design example microprocessor control.

3. Creating the schematic diagram in the *Schematic* module

The circuit board design process can be divided into three basic stages. The first is to create a schematic diagram of the designed tile. Another the step is to transfer information, i.e. the list of connections, the drawn diagram to the module *Board* . The final step is to determine the size of the tile, arrangement of layout elements on circuit board and manual or automatic connection between them.

Fig. 1 show toolbars with short descriptions of each functions by which it is possible to perform all operations in the module Schematic and Layout (board) .



➤ *Command toolbar of the Schematic Editor (left) and the Layout Editor (right)*

Fig. 1. EAGLE Schematic toolbar and Layout (board) toolbar

8. Exercise program

1. Open the **RECORDER** project example containing the schematic diagram and ready circuit board design. Check the operation of the EAGLE program.

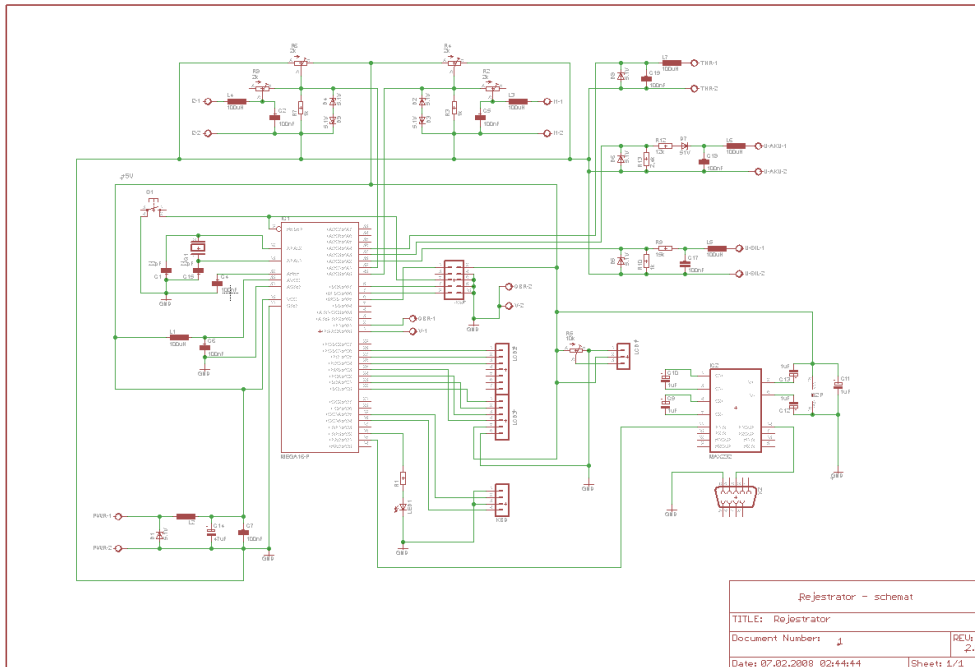


Fig. 8.1. Project example – Recorder file – *schematic diagram*

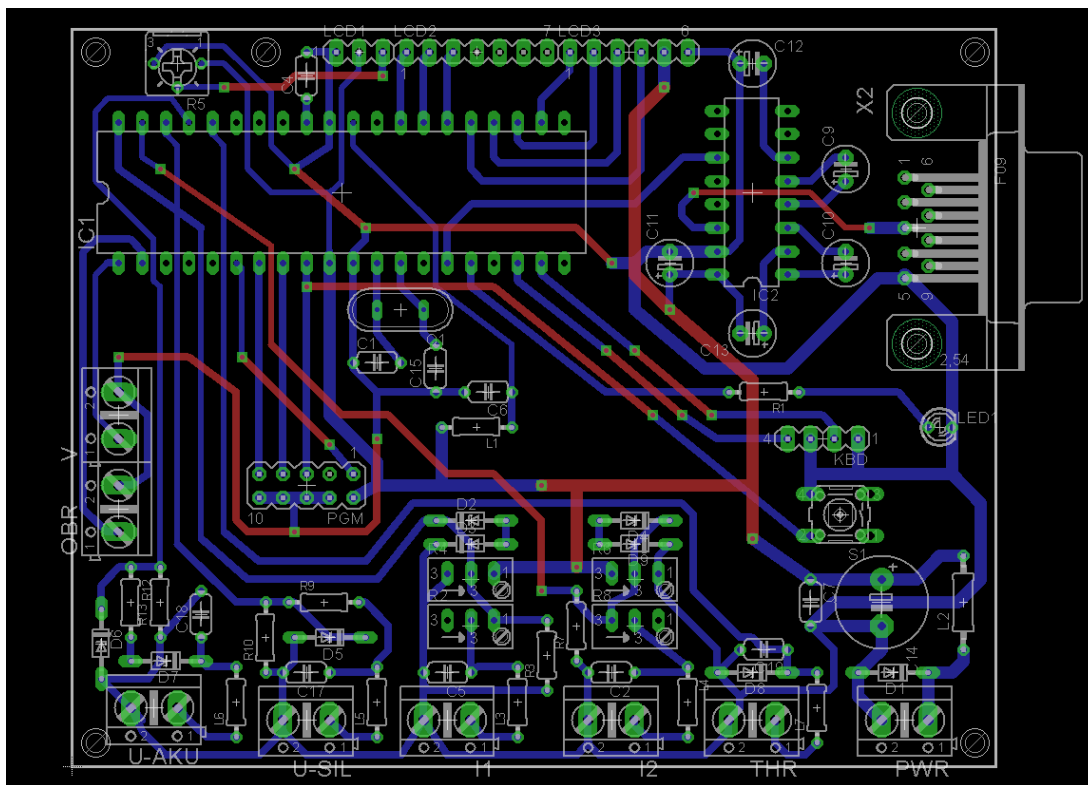
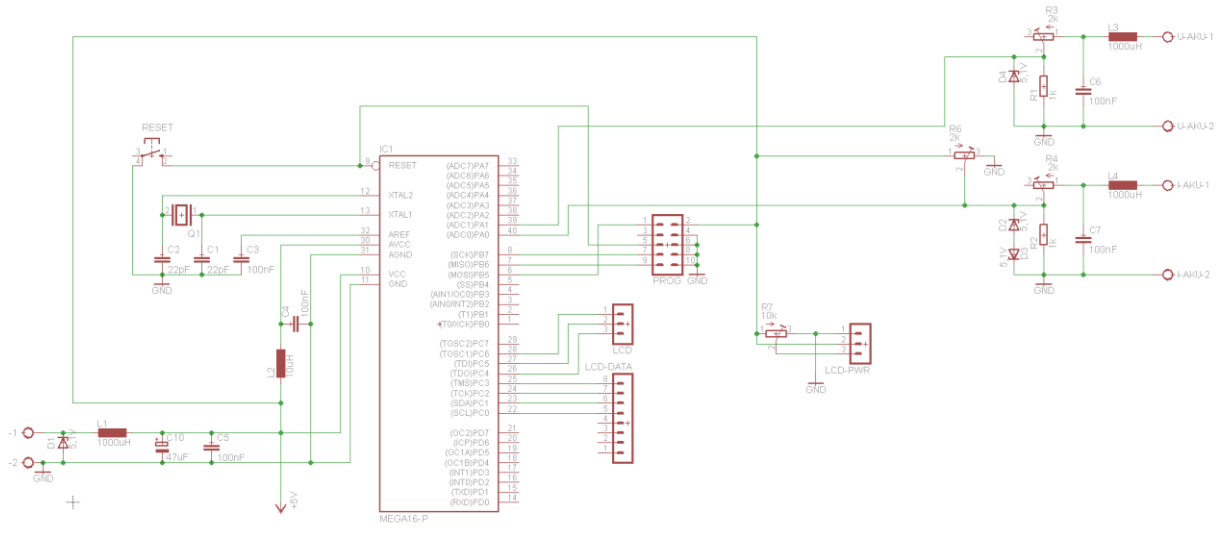
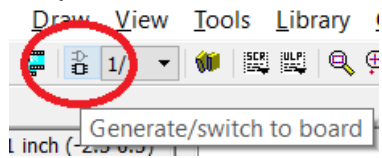


Fig. 8.1. Project example – Recorder file – *ready circuit board design*

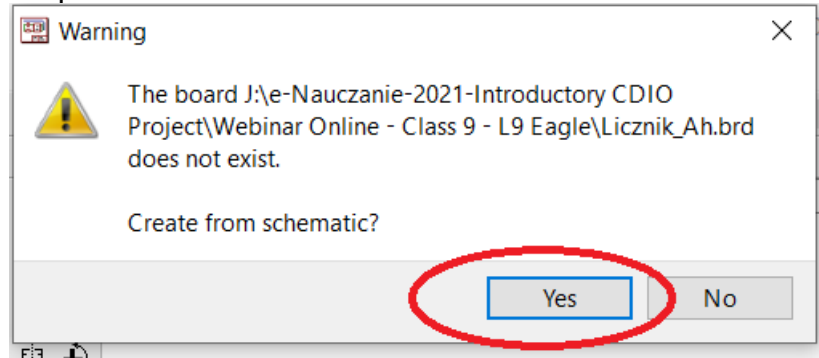
2. Open the sample diagram COUNTER_AH and design the printed circuit.



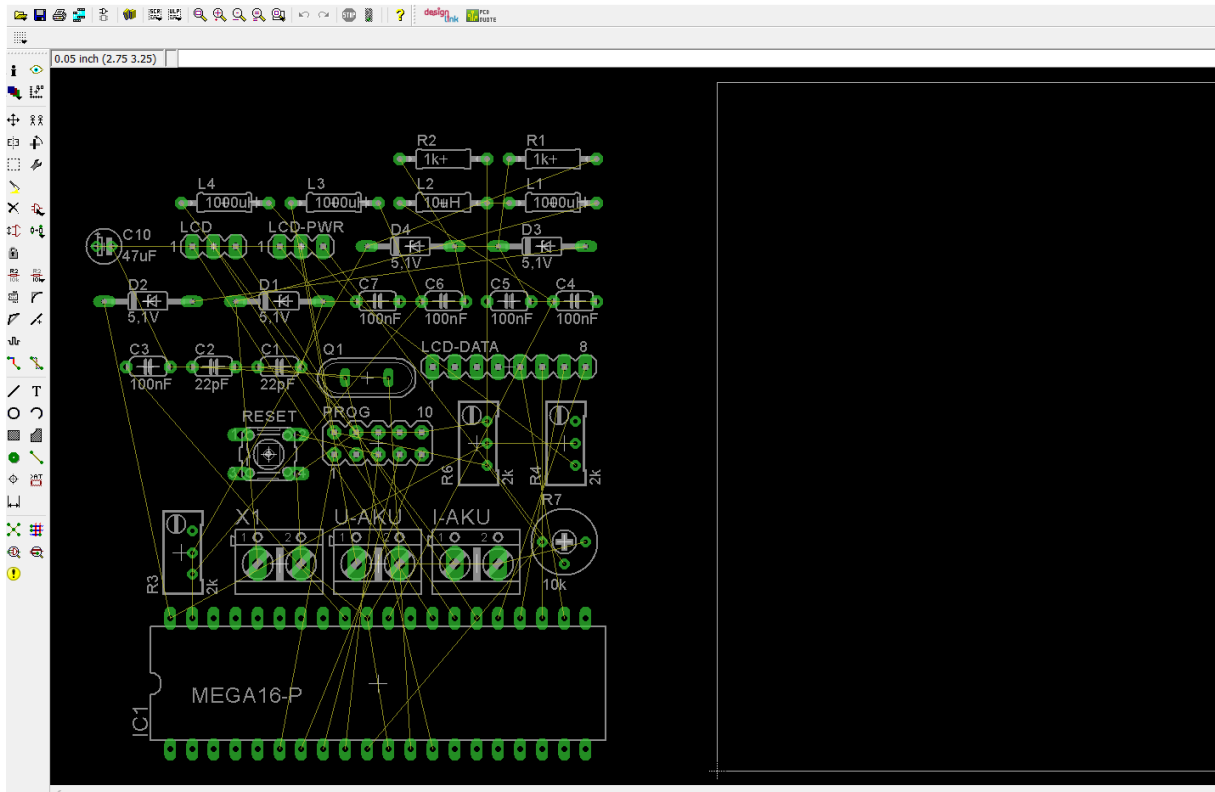
Step 1



Step 2

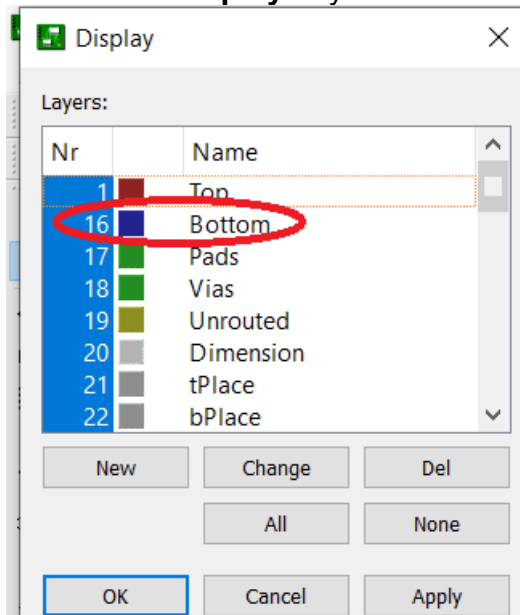


Step 3



Step 4

Use as much connections as possible using the bottom layer (blue colour).
Use button **Display Layers** from the toolbar to check layers.



Use the EAGLE ver. 6 – Tutorial, pdf file from e-nauczanie website.

- Chapter 15. Designing a PC Board.
- Creating a Board from a Schematic – page 48

1) Generating a Board File

2) As a first step, draw the board contour. It is made up of wires in the layer 20, *Dimension*. **Program suggest the dimensions. Check the layer 20**

3) Component Placement

Etc...