Page 1

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 circuit board 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 and 2 show toolbars with short descriptions of each
functions by which it is possible to perform all operations in the module
Schematic .

Page 2

Fig. 1. EAGLE *Schematic* toolbar

Fig. 2. The horizontal toolbar of the EAGLE *Schematic* program

An important operation when designing is searching in program libraries
appropriate electronic components identical to those to be used
in a real system. This is possible thanks to the extensive package libraries
EAGLE files with the extension * *.lbr* . These files constitute the data archive
regarding the appearance and type of element housings that can be used when drawing
schematics and creating printed circuits. All libraries with contents
can be viewed in the control panel. When drawing a schematic, only access to
libraries that have been activated (*Library* → *Use*). Fig. 3 shows the selection window and
browsing libraries.

Page 3
Fig. 3. Element library window
To insert the selected item, press the *ADD* button
on
vertical toolbar, then select the library containing this item and
confirm with *OK* - fig. 3. At this stage you can select the element including its housing and external dimensions and leg spacing.

Page 4
Fig. 3. Dialog box for adding and selecting elements
With the help of the mouse it is possible to place any element. Item rotation is obtained with the right mouse button. You can also do this operation with *Rotate* button or make a mirror image with the *Mirror* button .
After selecting and arranging the elements in the diagram, you can proceed to making electrical connections in the diagram by pressing the *NET* icon
.
Once all the elements have been connected together in the diagram (Fig. 4), you can check correct connections with the ERC button. In case of any errors you should correct them. Then you can proceed to the next design stage.

Page 5
Fig. 4. Schematic diagram of the system

4. Arrangement of components on the circuit board in the *Board* module

After connecting the elements correctly, transfer the information in the form of a list connections of the drawn diagram to the *Board* module . This happens after pressing the icon *BOARD* on the horizontal toolbar of the *Schematic* module window .
After switching to the *Board* module , a window will appear containing the maximum allowable
tile area and components with connections - fig. 5.

Page 6
Fig. 5. Window of the *Board* module
Specify the shape and dimensions of the tile, and arrange the elements on it. In front of starting the laying of paths, the technical parameters of the designed tile should be determined, and
so path widths, distance between them, hole diameters, and more. It is possible thanks to the *Drc* function on the vertical toolbar of the *Board* module - fig. 6.

Page 7
Fig. 6. Setting technical parameters of the printed circuit
After setting all the parameters needed to make the tile should
proceed to guide the paths between the feet of the elements. This can be done manually or automatically.
Are drawn in red track top layer - ***top layer*** , while the color
blue ***bottom layer*** path - ***Bottom layer*** . The circuit board is visible from the side

upper, i.e. from the side of the elements.
The *Route* and *Ripup* commands are used to manually route paths .
Automatic track guidance is started by pressing the *Auto* icon on
vertical bar will manage. If the automatic connections made do not meet expectations
designer connections can be improved manually.
After performing these operations, a picture of the finished circuit with elements and is obtained
connections between them - fig. 7.

Page 8
Fig. 7. View of the printed circuit with components and paths
The tile design prepared in this way is saved in a file with the * *.brd* extension . Most
PCB manufacturers accept files in this format as sufficient
production documentation.
Otherwise, it is necessary to prepare production documentation in
the universal Gerber format. Gerber documentation is generated using a module
CAM-Processor launched from the *File* menu .

5. Making a printed circuit

Based on the finished design, the printed circuit can be made in the Department
Electric Drive Automation on the LPKF PROTOMAT C60 milling machine. It is
possible for the needs of engineering projects and diploma theses and requires
prior agreement.
The printed circuit is made of laminate covered completely
a layer of copper is removed with the help of special cutters, the unnecessary copper layer is removed.
The machine also drills holes. Both are made on the milling machine
tile sides. The guides connecting the upper and lower layers must be made by hand through
both sides soldering of wires inserted into drilled holes for grommets.
Leading paths to the ends of the elements must be placed on the bottom layer
tiles - i.e. on the opposite side to the elements.
An example of the electronic system of an engineering project made on a board
printed prepared on the PROTOMAT C60 milling machine are shown in fig. 8 and fig. 9. Layout
was designed as one-sided.

Page 9
Fig. 8. Top view of the circuit - elements side
Fig. 8. Bottom view of the circuit - soldering side

6. Preparation of a new EAGLE program library

The EAGLE program library contains many standard items
e. However, there is often a need to add new items
library. This point presents an example of how to prepare a model
ATMEGA16 microcontroller, which will be added to the element library.
To prepare your own element you need to have knowledge about the library structure in the program
EAGLE. In the library, each element is defined by three components.
The first component is "Device", which is the whole element and a pointer to connect to

---

Page 10

the other two ingredients. The second component is 'Package', which is the element in the form
a printed circuit containing information such as: spacing and width of pins
(ie, the ends of the element), the size of the housing, etc. The third component is the "Symbol" in which
an element's ideological symbol is defined with indications specified by the manufacturer in catalog card.
First, create a new library in "Control Panel": File-> New-
> Library and save it under any new name, e.g. PG.lbr. Library name
should best characterize its content. Then an empty library opens, to
which you need to add the three components of the model to the appropriate element To do this:
1. Edit the library by clicking on
"Device", enter name
new part (e.g. ATMEGAPG) and confirm the addition of the item.
2. Repeat the first step for "Package" and "Symbol" by pressing accordingly
and .
After generating the files, save the whole file and prepare the catalog part data
(in this case the ATMEGA16 microprocessor). Then design the symbol
preferably according to the manufacturer's mark to avoid possible mistakes with
connecting and designing (fig. 9).
Fig. 9. ATMEGA16 processor symbol (Source *: Atmel Datasheets*)
To prepare an element's ideological symbol:
1. Draw the symbol outline using the "Line" button
. Appears
tools at the top of the program:
You will need the "94 Symbol" layer to draw the element outline.
The symbol drawn should be large enough to fit all the pins. one must
keep the resolution according to the set grid.
2. Choose to add "Place pin" pins
and place them around the contour
element as shown in fig. 10.

Page 11

Fig. 10. Preparation of the ATMEGA16 ideological symbol
3. Change the name of the pins in turn using the "Rename" tool
.
4. Name the item using the button
and layers "95 Names" (Fig. 11)
Fig. 11. ATMEGA16 ready symbol
When finished, save the file.
The next step is to design the microprocessor in the form of a printed circuit.
First open the file via "Package"
. Then change the units to millimeters -
mm by selecting the grid button
(the grid in EAGLE is set to inches by default -
inch). Convenient and accurate design of soldering pads (so-called "pads") is possible

thanks to the grid resolution setting according to the pin spacing given in the card
sheet. A grid spacing should be set for the designed ATMEGA16 element
to 2.54 mm (fig. 12).

Fig. 12. Fragment of the ATMEGA16 data sheet with dimensioned drawings
technical element (Source *: Atmel Datasheets*)

After setting the grid:

1. Select soldering pads

- a green octagon indicates a soldering point
a threaded element, while a rectangle is a soldering field of the terminal for
surface mount SMD. Shape and dimensions of the soldering point and
Set the hole diameter for through-hole ends according to the data
catalog item. In the example presented, it is best to choose a pad
long, or "long":

2. In accordance with the dimensions from the catalog, define pad shapes and arrange in
right place. To rotate the pad, click the right mouse button.
3. Draw the shape of the element by selecting a layer from the tools
(for convenience, you can change the grid spacing).
4. Place the element name on the layer
5. Check the pads' marking function
"Show" (information appears on
bottom screen bar). If the pads are not named correctly then this should be
improve with "Rename"
.

The library element should look as shown in fig. 13.

Fig. 13. Prepared ATMEGA16 library element for a printed circuit

In the further stages of preparing the library element you should:

1. Go to the 'Device' component
, add the designed "Add" symbol
(button placed on the vertical bar on the left) and place the symbol
ideological ATMEGA16 in the center of the screen.
2. Add a designed element by clicking the "New" button
in the right
bottom corner of the screen and select the appropriate item.
3. Connect the pads with the symbol's markings by clicking the "Connect" button
. A window will open in which the corresponding symbol pin can be connected
ideological with the correct model pad for the printed circuit (Fig. 14).

Fig. 14. Connecting pins of the element's ideological symbol with the element's model for the circuit
printed

4. After connecting the pins with the pads, provide a description of the prepared element
library in the Description pane (html format is supported in the description).

The final result of the work is shown in Fig. 15.

Page 14

Page 14

Fig. 15. Library element designed in the EAGLE program

7. Preparation of production documentation

The printed circuit production documentation are:

• drill files in Excellon format,

• layer files in Gerber format.

To generate these files, launch CAM-processor in the Eagle program (Fig. 16).
The CAM-processor window will open (fig. 17).

Page 15

Page 15

Fig. 16. Command to start CAM-processor

Fig. 17. The CAM-processor program window

Page 16

Page 16

In the CAM-processor menu, select File → Open → Job. A list of formats will appear
files for generation (Fig. 18).

Fig. 18. List of production documentation file formats

Be the first to choose the Excellon format. The CAM-processor window will have
the appearance shown in figure 19.

Fig. 19. CAM-processor window for generating Excellon drilling files

Only the "Drills" and "Holes" layers are active for the Excellon format. Belongs
start the CAM-Processor work by pressing [Process Job]. In the project's working directory
two text files appear with the extensions .dri (tool list) and .drd (coordinates
drilling holes).

Then select the Gerber format in the CAM-processor window. CAM-processor window
will have the appearance shown in Figure 20.

Page 17

Page 17

Fig. 20. CAM-processor window for generating Gerber files

All layers used in the project will be active for the Gerber format. Belongs
start CAM-Processor operation by pressing [Process Job]. In the project's working directory
resulting text files with the extensions: .whl (aperture list), .cmp (paths
element pages) .sol (soldering page paths), plc (description layer), .stc (mask
soldering element side) and .sts (solder side soldering mask).

The set of files generated by the CAM-processor is the production documentation
circuit board.

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.
2. Open the sample diagram COUNTER_AH indicated by the teacher or
PLYTKA_45 and design the printed circuit.
3. Enter the new library item indicated by the teacher.
4. Draw the diagram and printed circuit of the DC impulse converter
lowering voltage " *Buck converter* " with parameters and using elements
indicated by the teacher. Prepare only the power electronics part

inverters without control system For control signal, power source and
Loads provide for suitable solder pads or connectors.
5. Prepare production documentation in Excellon and Gerber formats. Get to know and interpret the contents of generated files.
6. Verify the correctness of production documentation in free browsers
Excellon and Gerber files, e.g. CAMtastic! ACCEL Technologies or ViewMate Pentalogix.

---

8. Literature

1. EAGLE http://www.cadsoft.de
2. Wieczorek H .: EAGLE, first steps, BTC Publishing House, Warsaw 2007.
3. Rymarski Z .: Materials science and construction of electronic devices. Ed. World Cup, Gliwice 2000.
4. Mika M .: Printed circuits. WkiŁ, Warsaw 1983.
5. Kisiel R., Bajera A .: Basics of constructing electronic devices. Publishing House PW, Warsaw 1999.

Attachment

Basic principles of designing printed circuits

Standard copper path 0.25 mm wide and 0.035 µm thick
resistance 0.018 Ω / cm. The path width should be chosen according to the expected current carrying capacity. It is advisable to use as wide paths as possible to minimize connection resistances.
The distance between adjacent tracks and pads is selected in
voltage dependence. The minimum distances are:
• 0.25 mm for voltages 10..30 V,
• 0.4 mm at voltages up to 50 V,
• 0.5 mm at voltages up to 150 V,
• 0.75 mm at voltages up to 300 V,
• 1.5 mm at voltages up to 500 V.
When designing printed circuits, the goal is to minimize parasitic effects
path inductance. Typical print path inductance is 10 nH / cm. That's why it should be design the circuit so that the paths are the shortest.
The surface area of the electrical circuit loop should be as small as possible
loops are susceptible to interference voltages. Fig. 21 shows an incorrect circuit loop.
Fig. 21. Loop with high inductance
A correct example of connections is shown in Fig. 22. Close track routing
output and return means that the mutual inductance of the circuit is much lower than in the case of Fig. 21.

Fig. 22. Loop with low inductance
Typical wave impedance of a narrow conductive path on one and two-sided wafer
printed is approximately 110 Ω ... 135 Ω. Each path bend changes this
impedance. This can lead to signal reflection. In addition, any sharp bend angle
paths and every sharp edge causes a strong electric field strength. Bad and good
solutions are shown in Fig. 23.
unfavorable

Perfect
Compromise
Fig. 23. Poor and good bending of paths
branching
tracks
change
impedance,
What,
at
signals
high frequency may cause signal reflections. Bad and good solutions
shown in fig. 24.
Wrong
(branched path)
Well
Fig. 24. Poor and good path management
If there are analog and digital circuit elements in the designed circuit
then the masses of both circuits can be connected at one point only. It is advisable to
the common mass of analog signals was distributed in the form of a star. This makes it
possible
Correct measurement of voltages that refer to the same point of the circuit - Fig. 25.

Page 20
Fig. 25. Joint mass connection in complex projects