Knowledge Test 01 /2020 – Answer Sheet

Lector: Alexander A. Istomin

1. The FT245RL Microcircuit in the Cryptuino01, when connected to the PC, creates a communication port (**COMx** under Windows, **/dev/ttyACMx** under Linux).

   a) Write down at least three values (max 5) you can send to this communication port, that, if read, will reset the ATTINY.

   b) Write down also two values that will put the ATTINY in a working state again.

The FT245RL has a 8-bit parallel IO Bus = 256 Values.

You need to pull the RESET pin LOW (line on top of Reset= signal inverted)

Also, page 3 of the ATTiny datasheet:

$\overline{\text{RESET}}$

Reset input. A low level on this pin for longer than the minimum pulse length will generate a reset, even if the clock is not running and provided that the reset pin has not been disabled. The minimum pulse length is given in Table 22-3 on page 201. Shorter pulses are not guaranteed to generate a reset. The Reset Input is an alternate function for PA2 and dW.

The reset pin can also be used as a (weak) I/O pin.

List of all numbers sent to the FT245RL that reset the ATTiny:

0 1 2 4 5 6 16 17 18 20 21 22 32 33 34 35 36 37 38 39 48 49 50 51 52 53 54 55
64 65 66 68 69 70 80 81 82 84 85 86 96 97 98 99 100 101 102 103 112 113 114
115 116 117 118 119 128 129 130 132 133 134 144 145 146 148 149 150 160 161
162 164 165 166 176 177 178 180 181 182 192 193 194 196 197 198 208 209 210
212 213 214 224 225 226 228 229 230 240 241 242 244 245 246

List of all numbers sent to the FT245RL that reenable the ATTiny from reset:

3 7 8 9 10 11 12 13 14 15 19 23 24 25 26 27 28 29 30 31 40 41 42 43 44 45 46
47 56 57 58 59 60 61 62 63 67 71 72 73 74 75 76 77 78 79 83 87 88 89 90 91 92
93 94 95 104 105 106 107 108 109 110 111 120 121 122 123 124 125 126 127 131
135 136 137 138 139 140 141 142 143 147 151 152 153 154 155 156 157 158 159
163 167 168 169 170 171 172 173 174 175 179 183 184 185 186 187 188 189 190
191 195 199 200 201 202 203 204 205 206 207 211 215 216 217 218 219 220 221
222 223 227 231 232 233 234 235 236 237 238 239 243 247 248 249 250 251 252
253 254 255

The following C program does it automatically:

```
#include <stdio.h>
#include <stdbool.h>

int main()
{
  int i;
  printf("List of all numbers sent to the FT245RL that reset the ATTiny:\r\n");
  for (i = 0; i < 256; ++i)
  {
    bool b0 = (i & 0x01);
    bool b1 = ((i >> 1) & 0x01);
    bool b2 = ((i >> 2) & 0x01);
    bool b3 = ((i >> 3) & 0x01);
    bool b4 = ((i >> 4) & 0x01);
    bool b5 = ((i >> 5) & 0x01);
    bool b6 = ((i >> 6) & 0x01);
    bool b7 = ((i >> 7) & 0x01);
    bool c = !(b0 & b1);
    bool d = !(b7 | (!b5));
    bool e = !(c | d);
    bool res = !((!e) & (!b3));
    if (!res)
    {
      printf("%d ", i);
    }
  }
  printf("\r\n\r\nList of all numbers sent to the FT245RL that reenable the ATTiny from reset:\r\
n");
  for (i = 0; i < 256; ++i)
  {
    bool b0 = (i & 0x01);
    bool b1 = ((i >> 1) & 0x01);
    bool b2 = ((i >> 2) & 0x01);
    bool b3 = ((i >> 3) & 0x01);
    bool b4 = ((i >> 4) & 0x01);
    bool b5 = ((i >> 5) & 0x01);
    bool b6 = ((i >> 6) & 0x01);
    bool b7 = ((i >> 7) & 0x01);
    bool c = !(b0 & b1);
    bool d = !(b7 | (!b5));
    bool e = !(c | d);
    bool res = !((!e) & (!b3));
    if (res)
    {
      printf("%d ", i);
    }
  }
  return 0;
}
```

Q2: What are the metric dimensions of a 0201(Imperial) Capacitor?

0201(Imperial) = 0.6mm x 0.3mm (0603 metric)

Q3: What is the standard pitch (there are rare exceptions) for SOIC Chip Pins

1.27mm or 50mil

Q4: Is the electric schema dependent on a footprint of one or several components?

No, there can be many footprints for a symbol (Integrated circuit sold in different packages) and many symbols for a footprint (millions of different chips with a QFP-44 footprint)

Q5: What is the function of these passive components?

Decoupling capacitors

Q6: Can a VIA of 0.6mm/0.3mm exist in a NPTH of 0.4mm?

NO!!!!! A VIA of 0.6/0.3mm has a hole of 0.3mm, a NPTH (non plated through hole) of 0.4 mm is a bigger hole.
Can you make a smaller hole inside a bigger hole ? That's Metaphysics, not PCB Design!

Q7: What is this symbol ? 

It is a symbol for an Open **Solder Jumper** with 2 contacts

Q8: What is Edge.Cuts used for ?

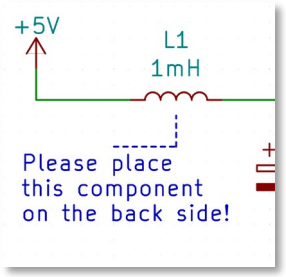
This Layer is used to define the PCB Board Outline (also the cutouts)

Q9: Assume the PCB is in the XY Plane. How does the Z-height of a component influence its footprint?

No influence whatsoever – mechanical stability is not a necessary requirement for all components

Q10: How (and where) do I indicate in the Schema Editor (`EESchema`) that a component is placed on the backside of the PCB?

For all who answered: you flip it to the other side with the red/green button – you're wrong! That is a function of the PCB EDITOR, not the Schema Editor!!
Short Answer : You can't …. or you write a text field like this ☺:



Q11: BONUS 1: What is a Rats Nest?

The rat's nest is maze of lines between the pads of the components showing you the electrical connections as defined by the schematic who are not yet connected by traces or zones.

Q12: BONUS 2: What is `Eco2.User` used for?

This Layer is a custom User Layer. It has no predefined purpose and can be used for any graphical information at the discretion of the designer.

Q13: BONUS 3: What does the inscription $\overline{\text{IORQ}}$ on a component or a signal line mean (the line on top)?

The line on top of IORQ means that this pin or signal line is active, when it is pulled LOW (to GND)!
Line on top = signal inverted