Some Notes on Using Protel Software for Schematic Capture and PCB Layout

The design of an electronic circuit is a mixture of engineering science, art, and common-sense. One must choose a circuit configuration, perform supporting calculations, and select actual electronic devices taking into consideration their characteristics, price and availability. The circuit may be prototyped, and/or circuit simulation may be performed. As in all design work, the design process is iterative – you need to make initial selections based on your calculations, and then see if they "work".

Virtually all electronic circuits that are produced in volume are fabricated on printed circuit boards [PCBs, also called printed wire boards, PWBs]. With the introduction of relatively inexpensive ($2k-$10k), high-quality PCB layout software for personal computers, it is now also common for prototype and one-off circuits to be implemented as PCBs.

When designing an electronic circuit, the selection of components and the physical layout of the circuit can, in some cases, be as important as the design of the circuit itself. An electronic circuit must be fabricated from real electronic devices. Real devices have non-zero physical size and fixed terminal positions and sizes. Real devices also have non-ideal electronic characteristics. Electromagnetic interference can be a significant problem, particularly for low-noise analogue circuits. Signals take a finite time to propagate along the electrical tracks which make the circuit connections. This can cause timing problems, particularly if the circuit board is large. At high frequencies (10s MHz-GHz), the conductive tracks behave as transmission lines, with signal reflections occuring where circuit impedence changes abruptly.

Although these effects usually become important for circuits which must operate with very low noise, or at high frequency (above 20MHz, say), it is important to appreciate of some of the practical problems encountered in circuit layout.

1 An Overview of Circuit Design and PDB Layout

1.1 Circuit Schematic Drafting

A schematic of the circuit is usually sketched first by hand, in order to define the concept and topology of the circuit. As the design is refined, perhaps with the aid of calculations and circuit simulation together with information on the price and availability of components, the circuit is often drawn on a personal computer or workstation using a *schematic editor*. This approach has many advantages:

- the circuit can be drawn in a standardised, clear and neat way which promotes understanding;
- changes to the component parts or connections can easily be made;
- some checking of the design can be done automatically – for example, a connection that consists only of output pins is probably wrong!
- the circuit can be exported to a circuit simulation program such as PSpice;
- a netlist that defines the circuit parts and interconnections can be created for export to a PCB-layout program;
- a bill of materials can be created, from which parts can be ordered.

One important advantage of this approach is that a netlist file can automatically be created. A netlist is simply a list of all of the parts and connections in the circuit. Each "net" is one electrically continuous connection: for example, all pins that are connected to +5V form one net which is usually designated VCC. When laying out the circuit, the netlist can be imported into a PCB-layout program. The PCB-

layout program uses the connectivity information in the netlist, which helps to prevent connection errors during manual or automatic track routing. The bill of materials (BOM) is a convenient list from which the components may be ordered.

1.2 An Overview of PCB Design and Fabrication

Once the electronic circuit is drawn by hand, or using a schematic editor, the board layout can begin. *Board layout* begins by deciding where to *place the components* on the board. Placement is usually the most important and difficult step in the layout. Next, *place the tracks* to complete the circuit. Track routing can be done in a number of different ways. The easiest, and often the best for simple circuits, is to use pencil and 5mm squared paper. Note that many through-hole components have their terminals arranged on a standard 0.1" grid. A PCB layout program can be used for manual track placement (*"pad-to-pad routing"*), or an *autorouter* can be used if this facility is provided by the PCB layout software. A *netlist* produced by circuit drafting software is often a great help in reducing connection errors when moving from the schematic to a finalised board layout.

Once the layout is complete, the board can be fabricated. In commercial production, this is a complicated and involved process that involves a large number (100+) of separate photographic and electrochemical processes, together with cutting, drilling, plating, tinning, printing, and testing.

1.2.1 "Hand-made" PCBs

With a little[1] practice and the correct, <u>fresh</u> materials, simple printed circuit boards can be made by hand. With careful registration of the layers, *double-sided* boards can be made. It is not practical to make *plated-through holes* (PTH), so *vias* ("vertical" connections between layers) must be made by hand-soldering wire links or pins ("matrix pins") through a board. If multi-layer boards (commercial boards may have up to up to 12 conductor layers) with plated-through holes are needed, they will need to be fabricated by a commercial PCB manufacturer.

A simplified production process could begin with laser printing a *positive image* of the tracks onto transparent film, and then photographically transferring the image onto *reversal film* to make a *negative* image. This negative is contact-printed onto commercial photo-sensitive coper-clad board that has been pre-coated with a *photoresist* material. To maintain the best image quality, contact printing is done with the film emulsion pressed onto the photoresist using glass plates. The photoresist is exposed to UV light for 15-40 seconds, depending on the light intensity. The positive image of the tracks hardens during photographic developing, and the remainder of the resist can be washed off, exposing the copper layer. All copper which is not protected by the resist layer is then *chemically etched* away, leaving the copper tracks. The photoresist is then stripped, and the board is trimmed and drilled by hand. Before soldering, the copper should be well cleaned (scrubbing with Ajax works well), and the boards should be sprayed with PCB lacquer. The lacquer will prevent the copper from tarnishing, and can be soldered-through.

All that remains is to insert the components ("stuff" the board), to solder them into the circuit, and to test the completed circuit. Then you can revise the design, re-do the board layout, make the second prototype board, etc...

1.3 Circuit and PCB Software

We have licences for a schematic capture program *Advanced Schematic* and a companion PCB layout program *Advanced PCB*. Both programs were written by an Australian company, Protel[2] Pty. Ltd. Altium produces some of the world's most advanced PC-based electronic design automation (EDA) tools. Although our licences are for obsolete versions of the programs, they are adequate for simple design tasks. Full documentation available as on-line help from within the programs, and some brief suggestions for circuit drafting are appended to these notes. Your design should be based around components that are commonly available from electronics hobby stores:

---

[1] A practice little is OK; a **lot** is much better…
[2] Now Altium Ltd.

- David Reid Electronics,   http://www.altronics.com.au/
- Jaycar Electronics,   http://www.jaycar.com.au/
- Dick Smith Electronics,   http://www.dse.com.au/

or component distributors such as:

- Farnell   http://www.farnell.com/
- RS Components   http://www.rs-components.com.au/

The three hobbyist suppliers listed are all in York Street, Sydney – take the train[3] to Town Hall Station. Component price and availability should be considered when choosing between alternatives.

2   Editing Circuit Diagrams with *Advanced Schematic*

2.1   Starting the Program

Start the program from All Programs > Eng > CAD > Protel, and make a new schematic (.sch) file in your file space.

2.2   Part Placement

A part can only be placed if the library that contains the part is loaded. Each parts library is equivalent to a manufacturer's data book. Use Library > Add/Remove (shortcut is L A) to load a library. A useful library to load is device.lib; it contains common devices such as resistors, capacitors, connectors, etc. The libraries d_something.lib are for compatibility with the DOS version of Schematic and should not be used.

Place a part by Place > Part (shortcut is P P). Then type in the component identifier. Note that some parts are identified by manufacturer as well as generic part name: the identifier 7400 for a TTL NAND gate is incorrect, but DM7400 is satisfactory. DM is one of the prefixes used by National Semiconductor. The book by Horowitz & Hill contains a big list of manufacturers' prefixes.

Alternatively, a part can be placed from within the Component Browser. Open the Component Browser by Options > Component Browser (shortcut is O B). Choose the appropriate library and find the part by browsing.

Component types are conventionally designated by letters. The Australian standard designators are :

- **A**   Assembly, such as an IC, amplifier, box of tricks, etc.
- **C**   Capacitor
- **D**   Digital device - typically an IC
- **F**   Protective device (e.g. fuse)
- **L**   Inductor
- **R**   Resistor
- **S**   Switch
- **V**   Semiconductor (transistor, diode, etc.)
- **X**   Connector or terminal

For example, R26 would be a resistor. See *SAA HB3-1986* for more information. Quite often, U is used to designate an IC, and D is used to designate a diode. Some parts, such as the simple TTL gates, have multiple devices within a single part. The individual devices are designated as U2:A, U2:B, etc. That is, U2:A is device A in part U2. U2:B is the second device (gate) in the part U1..

Change the part designator generated by Protel from (for example) U? to U1. The next U part to be placed will then automatically be designated U2

When placing a part, you can "edit at placement" (when the part outline is shown, indicating that the part can be dragged) by pressing the TAB key. Otherwise, edit a placed part by double clicking it. During placement, SPACEBAR will rorate a part, X or Y will flip it.

---

[3] But make sure that you give it back.

Don't forget to place connectors to allow you to get power and signals into and out of your board!

2.3   Footprints

The "footprint" is the physical size, outline, pin location and size, etc. of a part. Some parts have multiple footprints that are defined in Protel's PCB libraries. For example, a DM74LS00 NAND gate can be supplied in a through-hole DIP14 package, or in a surface-mount SOIC14 package. Other parts, such as resistors, capacitors and switches, have no standard footprint, and so do not have a footprint defined in the libraries. If the schematic is going to be used as the basis for a PCB design, you will need to enter an appropriate footprint into the part's footprint field at part placement. Some useful footprints defined in the *Advanced PCB* footprint library are shown in the appendix to these notes.

2.3.1   Predefined Footprints

Note Protel's names of the standard footprints, which are defined in `PFW.LIB`. Open this file from within *Advanced PCB* by using Library > Components to see all the pre-defined footprints. If a footprint that you need is not defined, you will have to create it using Library > Components.

Note that Protel's footprints have been known to be wrong. If you are going to build the board, it is *imperitive* that every footprint be checked against the component manufacturers' specifications.

2.3.2   User Defined Footprints

If a part (e.g. an unusual IC) or component package (e.g. a switch) that you need to use is not defined in Protel's libraries, you will have to define it yourself using the Library > Run Library Editor command from within *Advanced Schematic*.

It is good practice to make pin 1 on each component *square* for ease of identification; the component "footprints" (or "patterns") provided by Protel follow this convention. Note that inch measurements are commonplace in PCB work. Most older components have pins which are on a standard 0.100" (100 mil) grid. One thousandth of an inch is invariably called a 'mil' in PCB work. The *size of a pin* is not standardised.

2.4   Place Wires

Once two or more parts are placed on the schematic, place wires to connect up the pins of parts to complete the circuit. Shortcut is P W. A junction (red dot) indicates that crossing wires are joined. A wire that starts from the end of another wire will be joined to it electrically. Make sure that wires that should not be electrically joined do not have a junction at their crossing point.

2.5   Net Labels, and Power Pins

Sometimes a circuit diagram can become messy or confusing if *every* connection is drawn. Instead of drawing some of the wires in their entirety, a short wire can be drawn from each pin on the net, and a *net label* added to each short wire. Although there is no wire drawn between pins, the pins are electrically connected via the net label. Use Place > Net label (shortcut is P N) to place a net label.

Parts in the Protel schematic libraries may have hidden power pins that are automatically connected to the nets VCC (+5 Volts) and GND (+0 Volt digital ground). This convention applies particularly to TTL chips[4], where the power connections are seldom shown on the schematic. Protel allows you to view the hidden pins, and follows the convention that if you un-hide a hidden pin you have to manually make a connection to it.

2.6   Wiring Tool Bar

Most of the tools needed for placing parts, wires, net labels, etc. are available on the wiring toolbar. (shortcut is O W). Use "edit at placement" to get variants.

---

[4] 74xxxx family parts have power supply at the "top right" pin, and ground at "bottom left".

2.7   Check the Circuit

Protel has a tool called the electrical rules check (ERC) that checks the connections in your circuit against some logical errors, such as a net that only contains output pins. Run it by File > Reports > Electrical Rules Check (shortcut is F R E). Eliminate all of the errors. You can also highlight the connections on any particular net by Edit > Select > Net (shortcut is E S N). To unselect, do by Edit > Unselect > All (shortcut is E E A).

2.8   Netlist and Bill of Materials

Can be generated from File > Reports. The netlist is required for importation into a PCB-layout program.

3   PCB Layout with *Advanced PCB*

If you plan on making a PCB "by hand", design your board using so-called "100 mil" (or "12-12") through-hole technology:

- grid for pin placement   100 mils   (2.5 mm);
- routing grid   25 mils   (0.6 mm);
- *minimum* track width   12 mils   (0.3 mm);
- *minimum* track to track clearance   12 mils   (0.3 mm);
- *minimum* pad size   65 mils   (1.6 mm);
- *minimum* via size   50 mils   (1.3 mm);
- only *one* track between pads.

Remember, 1 mil is 0.001 inch, or 0.0254 mm.

3.1   Starting the Program

Start the program from All Programs > Eng > CAD > Protel. The first thing that you need to do is to get all of the component footprints on the board, and in the best relative positions.

3.2   Netlist

Netlist Load will load the netlist specified, and import the footprints (patterns) associated designated for each part. Note that a component footprint will be found only if the footprint library containing it is loaded at the time the netlist is loaded. After the load, there should be no missing patterns, components or pins.

All components will be placed on top of each other.   Select the components, then centre them on your board area. Use Auto > Placement Tools > Expand Horizontal and Expand Vertical to spread out the components.

Then use Netlist > Optimise > All to re-order the nets in an optimal (shortest path) way. Next, move the components around to get the best layout. This is the most critical step of the layout. Don't bother with Auto Place, as it runs very slowly and cannot outperform manual placement. Some guidelines that you should follow in placing parts are:

- Don't forget that you have to mount the board mechanically somehow. M3 machine screws are often used, so place four pads of diameter 6.0 mm and 3.5 mm hole size in the corners of the board.
- place components so that the track routes are short and as straight as possible;
- place components as close together as possible;
- but leave sufficient room between components to get the tracks in;
- all ICs should face the same way, say pin 1 at top left. Line up components in rows;
- place connectors, adjustment pots, switches and diagnostic LEDs at or near the board edges.

Repeat the Netlist > Optimise and component moving steps until you think that the placement is satisfactory.

3.3   Board Dimensions

Have a guess at the board size - be generous with the size when you begin, as a board that is too small will make track routing very difficult. Place a closed rectangle of purple "keep-out layer" track to define the board limits. The keep-out layer is required for the Auto-router to run.

3.4   Track Routing

Set up the routing clearances, using Netlist > Clearances. For 12-12 technology, set all clearances to 5 mil. Then, set up the auto router by Auto > Setup Auto Route. Choose, for 12-12 technology

- routing grid: 25 mil;
- track width: 12 mil;
- via size: 50 mil;
- via hole: 28 mil;
- Set hugging before 100% (completion), spreading after 100%;
- Select *only* the memory and line-probe router passes at this stage.

Use Auto > Move to Grid to "juggle" the components a little bit so that their pins line up with the routing grid. This step is important, and can have a large effect on the completion percentage. Make sure that the grid that you move to is the same dimension as the grid selected for routing, usually 20 or 25 mil.

Then *save the board before any track routing* is done.

Then run the auto router, by Auto > Auto Route > All.

Then, remove the memory and line-probe passes, and add (say) one maze and one smoothing pass. Run the auto router again. Continue until the board is routed satisfactorily, or you decide to give up and re-do the component placement and board size. You might want to save the board with a slightly different name after each routing pass.

It is important to note that "auto-routing" is very definitely an iterative manual design process. The quality of the final result depends entirely on the component placement, the track width, pad and via sizes chosen, the clearances chosen, the router passes selected, and the number and order of the passes. You will have to do a number of layouts to get a reasonable design.

It will always be necessary to "clean up" the routed board by hand after the auto router has finished.

References

—. Airborne Electronics: http://www.airborn.com.au/ - Information on circuit design and PCB layout.

—. BEC Manufacturing: http://www.becman.com/ - Information on PCB layout.

—. Precision Circuits: http://www.precisionaustralia.com.au/cid/hm_cid.html – a very good set of customer information documents for PCB design and manufacturing.

—. SAA HB3-1986: Electrical and Electronic Drawing Practice for Students. Standards Association of Australia, Sydney, 1986.

Coombs, C.F. (ed.) *Printed Circuits Handbook.* 3ed., McGraw-Hill, 1988.

Ginsberg, G.L. *Printed Circuits Design*. McGraw-Hill, New York, 1991. 621.381531/6

Hall, D.V. *Microprocessors and digital systems*. 2ed., McGraw-Hill, Singapore, 1983. 621.38195/44

Horowitz, P. and Hill, W. *The Art of Electronics*. 2 ed., CUP, Cambridge, 1989.

Jung, W.G. *IC OP-Amp Cookbook*. 3ed., Howard Sams, Indianapolis, 1986.

Lancaster, D. *TTL Cookbook*. Howard Sams, Indianapolis, 1974.

Lancaster, D. *CMOS Cookbook*. Howard Sams, Indianapolis, 1988.

Noble, P.J.W. *Printed Circuit Board Assembly*. Chapman & Hall, London, 1989.

Phillips, G. *Newnes Electronics Toolkit*. Newnes, Oxford, 1993.

Williams, T. *Circuit Designer's Companion*. Butterworth, Kent, 1991. 621.3815/118

Appendix: Footprints of Common Parts

Follows overpage.

Z1
AXIAL0.3

Z2
AXIAL0.4

Z3
AXIAL0.5

Z4
AXIAL0.6

Z5
AXIAL0.7

Z6
AXIAL0.8
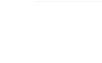
Z7
AXIAL0.9

Z8
AXIAL1.0

Z
DIODE0.4

Z9
DIODE0.7
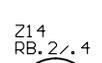
Z10
POLAR0.6

Z11
POLAR0.8
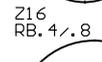
Z12
POLAR1.0

Z13
POLAR1.2

Z14
RB.2/.4

Z15
RB.3/.6

Z16
RB.4/.8

Z17
RB.5/1.0

Z18
TO-3
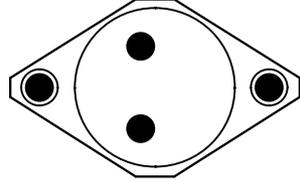
Z19
TO-5

Z20
TO-18

Z21
TO-39

Z24
TO-66
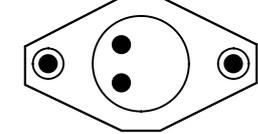
Z25
TO-72

Z26
TO-92A

Z27
TO-92B

Z22
TO-46

Z23
TO-52

Z28
TO-126

Z29
TO-220

Z30
FUSE

Z34
PENTA

Z31
HEPTA

Z32
MULTI11

Z33
MULTI15

Z35
VR1

Z36
VR2

Z37
VR3

Z38
VR4

Z39
VR5

Z40
XTAL1

Z41
BNC
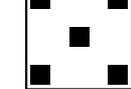
Z42
FLY4

Z43
POWER4

Z44
POWER6

Z45
SPADE

Z46
CAN8

Z47
CAN10

Z48
CAN12

Z73
DIP48

Z74
DIP52
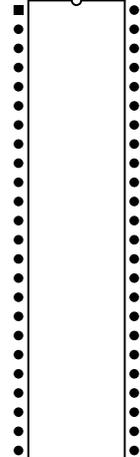
Z49
SIP2

Z50
SIP3

Z51
SIP4

Z52
SIP5

Z53
SIP6

Z54
SIP7

Z55
SIP8
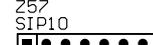
Z56
9

Z57
SIP10

Z58
SIP12

Z59
SIP16

Z60
SIP20

Z61
DIP4

Z62
DIP6

Z63
DIP8

Z64
DIP14

Z65
DIP16

Z66
DIP18

Z67
DIP20
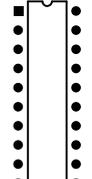
Z68
DIP22

Z69
DIP24

Z70
DIP28
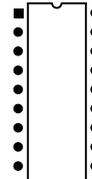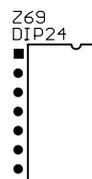
Z71
DIP32
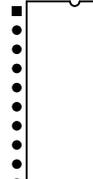
Z72
DIP40

Z76
DIP64
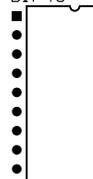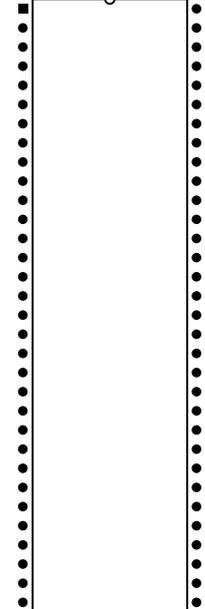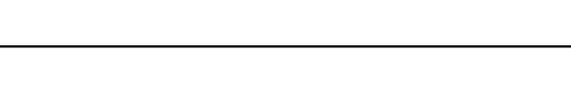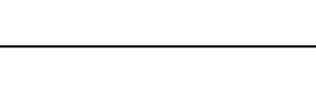
Z1
DB9/F
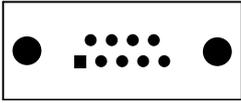
Z2
DB9/M

Z3
DB9RA/F

Z4
DB9RA/F
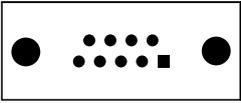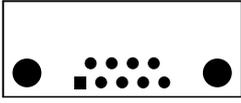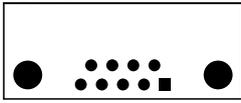
Z5
DB15/F
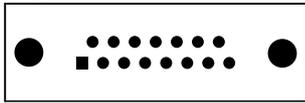
A1
DB15/M
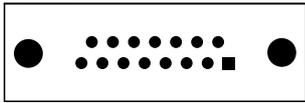
A2
DB15RA/F
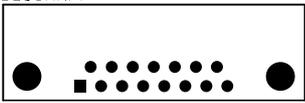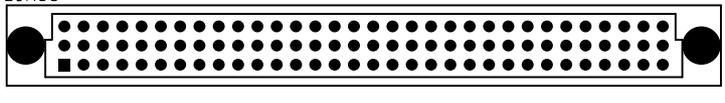
A3
DB15RA/M
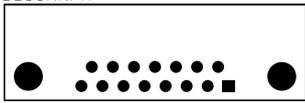
A4
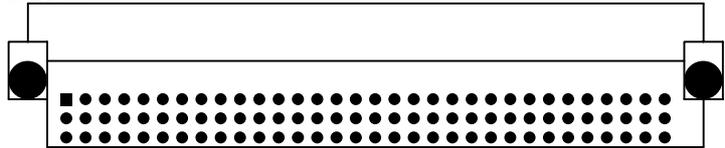DB25/F
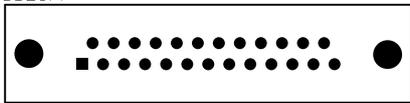
A5
DB25/M
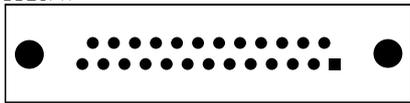
A6
DB25RA/F
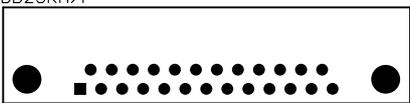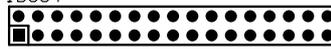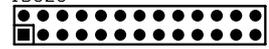
A7
DB25RA/M
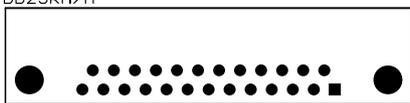
A8
DB37/F
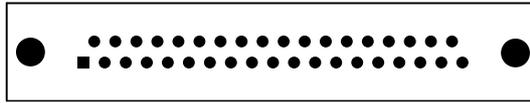
A9
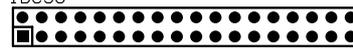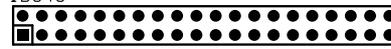DB37/M
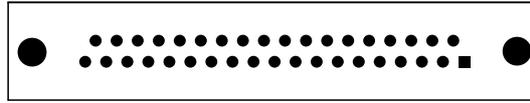
A10
DB37RA/F
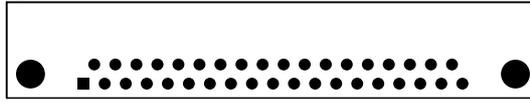
A11
DB37RA/M
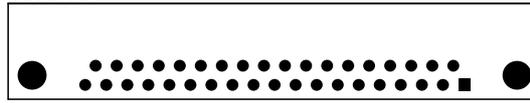
A12
DIN96

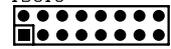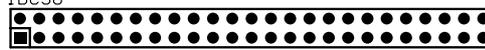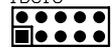A13
DIN96RA

A19
IDC34

A20
IDC26

A18
IDC36

A21
IDC20

A17
IDC40

A22
IDC16

A16
IDC40P

A15
IDC50

A23
IDC10

A14
IDC50P

A1
QUIL64

A8
PGA84x12

A14
PGA124x13

A2
PGA52x9

A9
PGA84x13

A15
PGA128x13

A3
PGA64x10

A16
PGA132x13

A4
PGA68x10

A10
PGA88x13

A17
PGA168x17

A5
PGA68x11

A11
PGA100x10

A6
PGA84x10

A12
PGA114x13

A18
PGA179x18

A7
PGA84x11

A13
PGA120x13

A19
PGA208x17