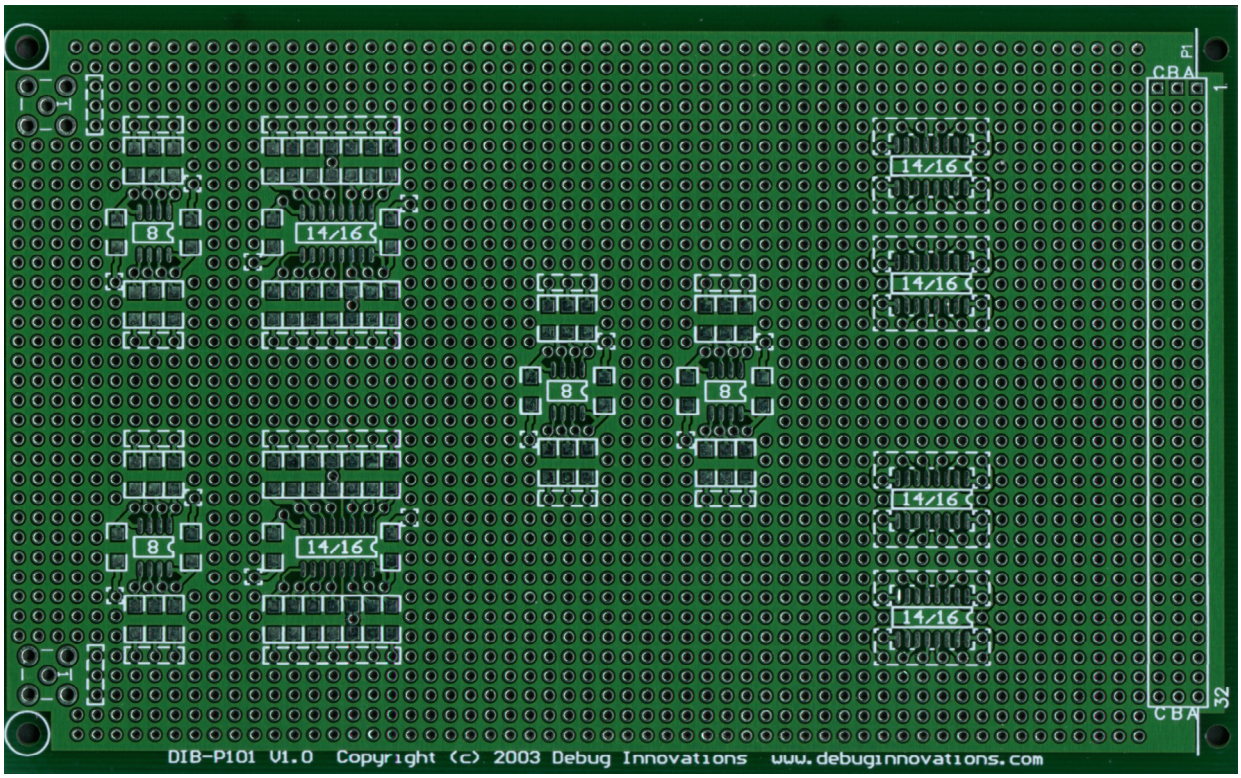
DIB-P101

High Speed Surface Mount Prototyping Board

User Guide

V1.0



www.debuginnovations.com

Introduction

Thank you for purchasing a prototyping solution from Debug Innovations. The DIB-P101 is designed to take the hassle out of prototyping high speed circuits with a host of innovative features:

Ideal for

- High speed logic prototyping
- Mixed analogue / digital device prototyping
- Mixed RF / digital prototyping
- Surface mount device evaluation

Large 0.1" matrix with power/ground planes for

- DIL devices
- Connectors
- Discretes
- Adaptor sockets for non-SOIC footprint devices e.g. FPGAs

Careful layout of matrix allows space for

- SMD to SMD or SMD to through hole connections
- 0.1" pitch edge connectors
- Connectors for stacking or sideways expansion
- 2 x SMA/SMB/SMC connector patches for RF connections
- Buffer or termination components near to the connectors

Single Eurocard format with 96-way DIN41612 connector patch

- Industry standard size for compatibility with a wide range of racks and PCB products
- Standard Eurocard mounting holes and 96 way connector
- High quality fibre glass laminate PCB construction
- Hot air levelled tin plated pads throughout for ease of soldering
- Legend on all patches shows pre-wired pins
- Solder mask on all component positions reduces the risk of shorts

Unique multi-patch architecture

- 4 x SOIC-14/16 minimal footprint patches designed primarily for use with digital logic and similar devices. All pins are tracked to holes on the 0.1" matrix. Each patch has a pre-wired decoupling capacitor space on the reverse side of the board across pins 16 to 8 and 14 to 7.

- 2 x SOIC-14/16 patches with pre-wired spaces for discrete devices on every pin designed for analogue and special function ICs requiring support resistors, capacitors etc. All pins are tracked to holes that can be directly wired or used as test points then connected through a 1206 patch to holes on the 0.1" matrix. Each patch has a pre-wired decoupling capacitor space on the reverse side of the board across pins 16 to 8 and 14 to 7.

- 4 x SOIC-8 patches with pre-wired spaces for discrete devices on every pin designed for op-amps and other small ICs requiring support resistors, capacitors etc. All pins are tracked to holes that can be directly wired or used as test points then connected through a 1206 patch to holes on the 0.1" matrix. Each patch has a pre-wired decoupling capacitor space on the reverse side of the board across pins 8 to 4 and 7 to 4 and a pre-wired feedback resistor/capacitor space across pins 6 to 2 for use with standard pinout op-amps.

- Total pinout flexibility – all device pins are open circuit unless the discrete patches are used

- Discrete patches use 1206 footprint for manageable hand soldering and wide component choice

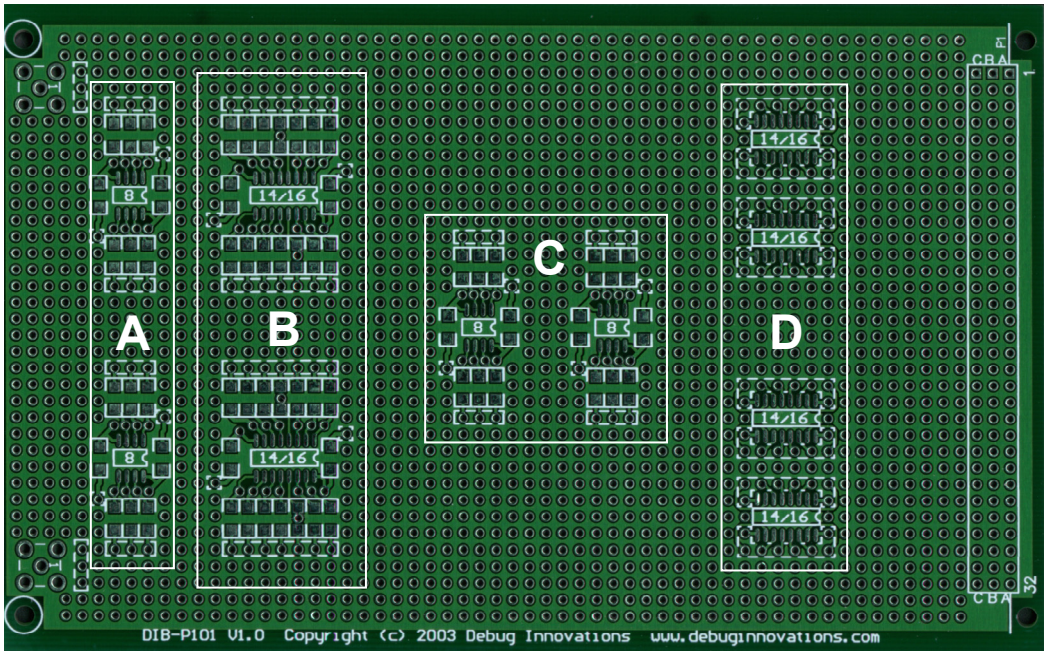
This unique design allows prototyping of circuits incorporating both surface mount and through hole components. Special patches incorporating pre-wired discrete component positions simplify the design of analogue and high speed digital circuits avoiding the need for special sockets and reducing the amount of wiring required. Accessible power planes and pre-tracked decoupling capacitor patches give excellent electrical performance making this board suitable for precision analogue and RF prototyping.

In this user guide you will find details of our unique patch architecture and other features. As usual, good planning can save a lot of time. Please familiarise yourself with the construction methods described in this guide before starting to build your project. If you don't understand anything in this guide or you have a suggestion for a design change or new product, please email support@debuginnovations.com.

DIB-P101 Overview

Top Edge

Interface Connector Area



96 way DIN 41612

Bottom Edge

The photo above shows the component side of a DIB-P101 prototype board. The main features are clearly marked. They are:

Backplane Connector

On the right hand side is a standard 96-way DIN 41612 connector patch complete with standard fixing holes. This patch will accommodate 32, 64 or 96-way connectors, male or female, straight or right angle as required. If not required this part of the board can be used for components (there are no special connections, all pads are free).

Interface Connector Area

On the left hand side is an area that can be used for I/O connectors. There are 2 special patches which can be fitted with SMA, SMB or SMC connectors for coaxial cables. There is enough space to accommodate right angle 0.1" pitch IDC connectors in between the RF types for I/O purposes in a typical rack mount environment.

Board Stacking

Enough space has been left along the top and bottom edges to fit more 96-way DIN 41612 connectors to enable board stacking with other DIB-P101 boards or custom designed motherboards. When not used in a rack environment, the top and bottom edges can be used as extra connector or component space.

SOIC Patches

Areas A-D contain the surface mount SOIC patches. Areas A and C can accommodate 4 x 8-pin devices with a passive component on every pin. Area A is positioned near to the RF connectors for a short path to buffers, opamps, PLLs, A/D converters and other high speed or noise sensitive circuits. Area B can accommodate 2 x 14-pin or 16-pin devices with a passive component on every pin. Area D can accommodate 4 x 14-pin or 16-pin devices but no SMD passive components. All patches have decoupling capacitor patches on the solder side of the board. The layout chosen mimics common requirements of a rack-mount mixed signal design – a few digital devices in area D, the gap to the connector for through-hole logic chips and high speed analogue chips in areas A and B. The spaces above and below area C are large enough to accommodate sockets for small PLDs / FPGAs.

Power Planes

The whole of the remaining area of the board is covered with 0.1" pitch through-holes for general components. In the space between the holes, there is a complete power plane. The component side and solder side power planes are completely separate. Typically you would use one as Vcc and the other as GND.

To connect a component hole to a power plane, use a small screwdriver or knife to scrape away the solder resist coating to expose the tin plated copper below. It is straightforward to bridge the gap between the pad and the plane with a small amount of solder. After working this way for a short time, it becomes second nature to use the power planes in this way and is much quicker (and electrically better) than using wires.

One word of warning – if you use a voltage regulator or power transistors in TO220 or similar packages, be careful not to let the metal tab get too close to the power plane. General handling causes them to quickly scrape away the solder resist and short to the power plane, causing potentially disastrous results.

Board Markings

Holes that have a special use, for example those that are connected to a patch location, are clearly marked with a square around them on the component side legend. Holes that have no markings are unconnected and free for general use. Of course, you can use any special holes as general purpose holes if you know they are not being used.

Solder Side

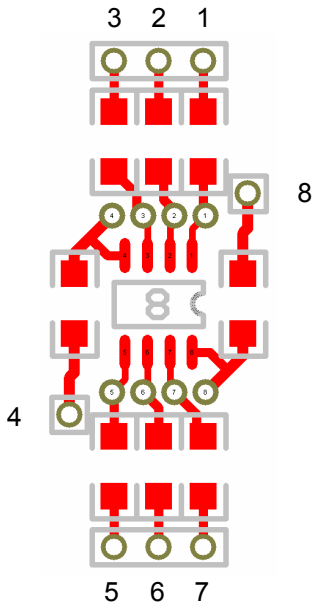
On the solder side of the board are a number of 1206 sized SMD patches. Each SMD patch on the component side has 2 pre-connected decoupling capacitor patches on the solder side. For the 8-pin patches, capacitor spaces are connected between pins 8 and 4, and 7 and 4 to suit the most common power pinouts. In addition a component patch is wired between pins 6 and 2 for a feedback resistor / capacitor in op-amp circuits. For the 14/16-pin patches, capacitor spaces are connected between pins 14 and 7 or 16 and 8 (fit the one that matches the size of the device fitted).

Board Size

The DIB-P101 board is "Single Eurocard" sized and the 96-way connector is placed in the industry standard position for compatibility with 3U 19" racks and many other board products.

In the following sections, each of the patch types is described in more detail.

8-pin Universal SOIC + Passives Patches



8-pin patch

8-pin Device Patch

This type of patch is for 8-pin SOIC devices and associated passives. There are 4 of this type of patch on the board in areas A and C in the preceding photograph. Typical uses are single and dual op-amps, PLLs, power supply ICs and other small analogue chips.

Passive Patches

All device pins are connected to the numbered vias shown in the diagram and also to a 1206 size passive component position, the other side of which is connected to a hole on the 0.1" grid. The track arrangement is such that the outer ring of numbers follows a circular pattern similar to standard IC pin numbering. All holes on the 0.1" grid that are pre-tracked are clearly marked with a rectangle on the top surface of the board though they may be used freely if there is no component fitted. Any unmarked holes can be considered 'free' for general use.

If a passive component position is not required on a pin, you can wire directly to the inner via or use a zero ohm resistor to bridge the unused component position. Holes in the vias are deliberately larger than normal so as to allow for fitting of connector pins or coaxial cables to aid testing.

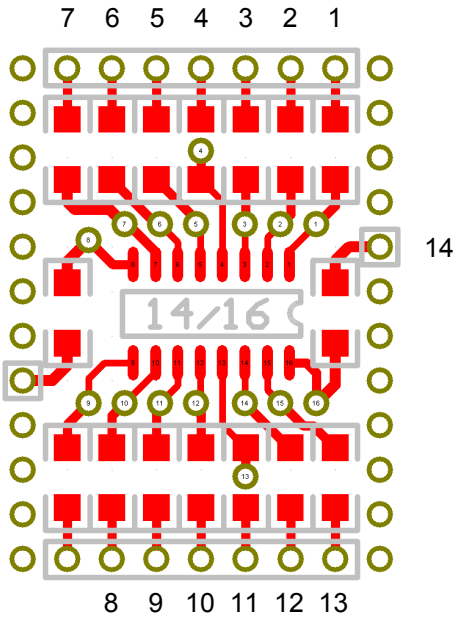
Power Pins

The tracks on the corner pins are wider than normal for minimum resistance / inductance. Inductors or resistors can be fitted in the power lines using the passive component position for noise suppression. On the solder side of the board there are 2 patches for decoupling capacitors – one for a single op-amp pinout connected between pins 7 and 4, and the other for a corner power pinout connected between pins 8 and 4.

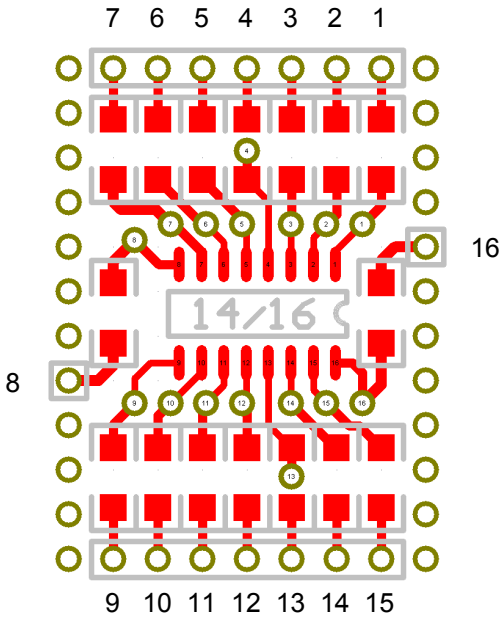
Feedback Passive Patch

On the solder side there is also a 1206 size passive component position connected between pins 6 and 2 for mounting feedback resistors or capacitors when using a standard pinout single op-amp device.

14/16-pin Universal SOIC + Passives Patches



14-pin device pin numbers



16-pin device pin numbers

14/16-pin Device Patch

This type of patch is for 14-pin or 16-pin SOIC devices and associated passives. There are 2 of this type of patch on the board in area B in the preceding photograph. Typical uses are quad op-amps, comparators, power supply ICs and other medium sized analogue chips.

Device Alignment

The 14/16-pin universal patches can be fitted with 14-pin or 16-pin SOIC package devices. When fitting a 14 pin device, the component should be aligned to the right of the patch as viewed above i.e. pin 1 should be in the same place as a 16-pin device's pin 1 and pins 8 and 9 of a 16-pin device would not be used.

Passive Patches

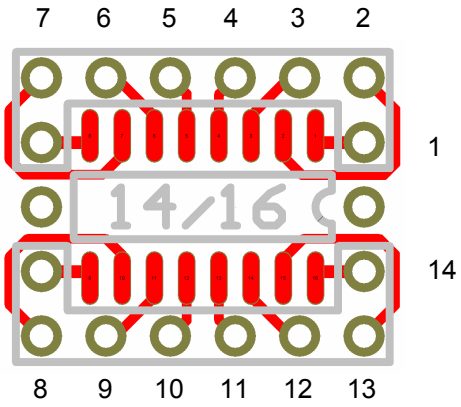
All device pins are connected to the numbered vias shown in the diagram and also to a 1206 size passive component position, the other side of which is connected to a hole on the 0.1" grid. The track arrangement is such that the outer ring of numbers follows a circular pattern similar to standard IC pin numbering. All holes on the 0.1" grid that are pre-tracked are clearly marked with a rectangle on the top surface of the board though they may be used freely if there is no component fitted. Any unmarked holes can be considered 'free' for general use.

If a passive component position is not required on a pin, you can wire directly to the inner via or use a zero ohm resistor to bridge the unused component position. Holes in the vias are deliberately larger than normal so as to allow for fitting of connector pins or coaxial cables to aid testing.

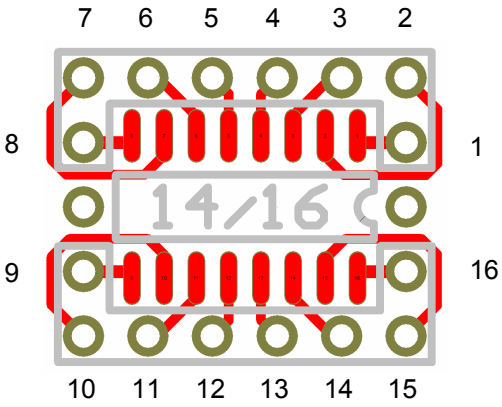
Power Pins

The tracks on the corner pins of 14 or 16 pin devices are wider than normal for minimum resistance / inductance. Inductors or resistors can be fitted in the power lines using the passive component position for noise suppression. On the solder side of the board there are 2 patches for decoupling capacitors – one for a 14-pin device connected between pins 14 and 7, and the other for a 16-pin device connected between pins 16 and 8.

14/16-pin Universal SOIC Patches



14-pin device pin numbers



16-pin device pin numbers

14/16-pin Device Patch

This type of patch is for 14-pin or 16-pin SOIC devices and has no provision for associated passives. There are 4 of this type of patch on the board in area D in the preceding photograph. Typical uses are digital circuits and bus interface chips.

Device Alignment

The 14/16-pin universal patches can be fitted with 14-pin or 16-pin SOIC package devices. When fitting a 14 pin device, the component should be aligned to the right of the patch as viewed above i.e. pin 1 should be in the same place as a 16-pin device's pin 1 and pins 8 and 9 of a 16-pin device would not be used.

Connections

All device pins are connected to a hole on the 0.1" grid. The track arrangement is such that the ring of numbers follows a circular pattern similar to standard IC pin numbering. All holes on the 0.1" grid that are pre-tracked are clearly marked with a rectangle on the top surface of the board though they may be used freely if there is no component fitted. Any unmarked holes can be considered 'free' for general use.

Power Pins

On the solder side of the board there are 2 patches for decoupling capacitors – one for a 14-pin device connected between pins 14 and 7, and the other for a 16-pin device connected between pins 16 and 8.