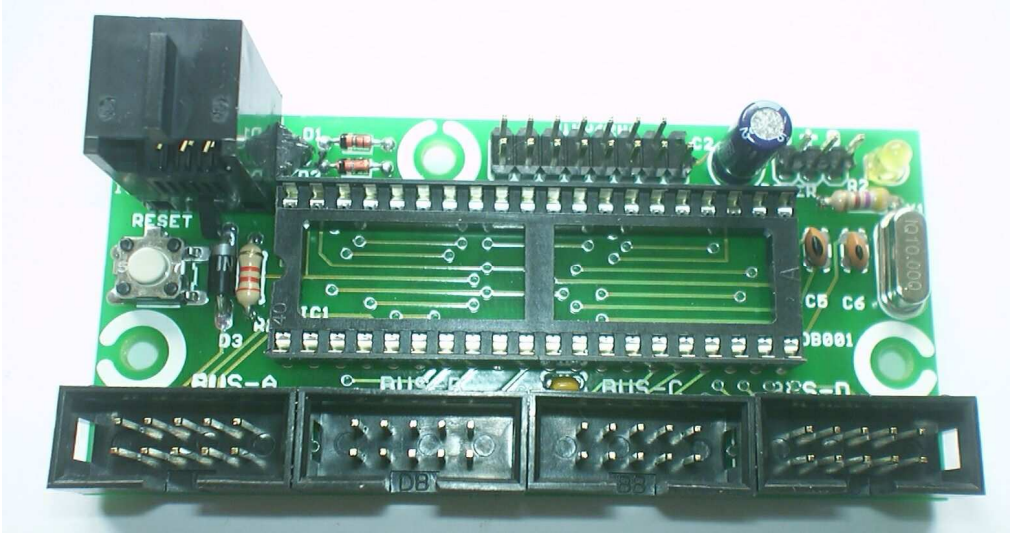
Dwarf Boards

DB001 : 40-pin controller board

(c) Van Ooijen Technische Informatica

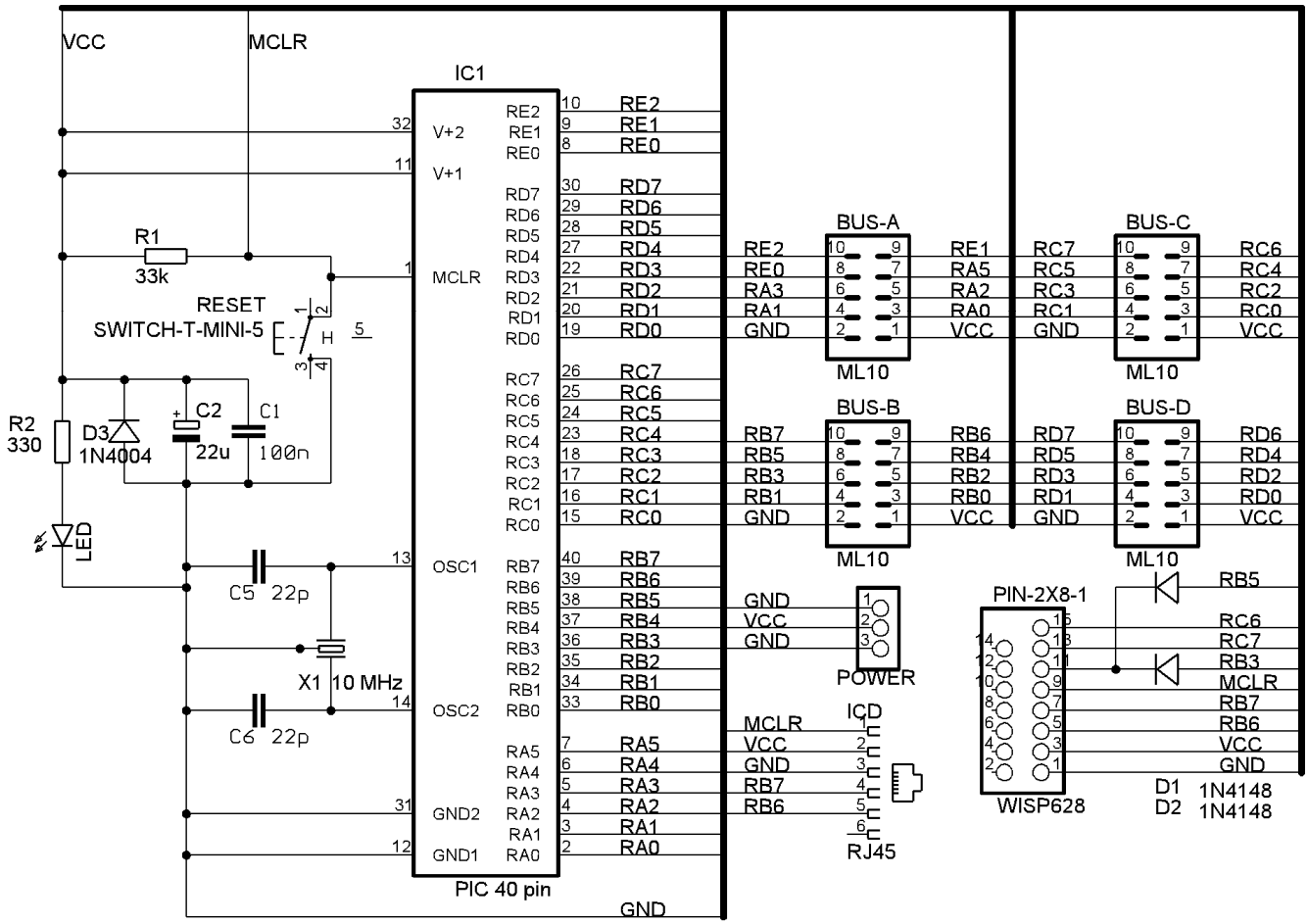
PICmicro, In-Circuit Serial Programming and ICSP are registerd trademarks of Microchip Technology Inc.



Introduction

This document describes the Dwarf Board DB001. This board is a minimal controller board, suited for educational use, experiments or prototyping. The board uses an 40-pin PIC microcontroller, like a 18F452 (16 bit core) or a 16F877A (14 bit core). The board contains Wisp628 and Microchip ICD connectors for In-Circuit Serial Programming. It also has a small pushbutton reset switch and a yellow power LED. The board does not contain any peripherals. The I/O pins of the microcontroller are available on four Dwarf Bus (ML10) connectors for connecting Dwarf Board peripheral boards.

Circuit



The circuit consists of the following sections:

- reset, power decoupling and power LED (left)
- the PICmicro and its crystal (middle)
- the four dwarf bus connectors (right)
- the power and ICSP connectors (lower right)

The reset (R1, switch) and power decoupling (C1, C2) are standard. Diode D3 offers some protection against an power that is accidentally connected the wrong way round. R2 and the LED give visual indication that power is available.

The crystal X1 and the capacitors C5 and C6 provide the clock for the the 40 pin PICmicro IC1. A three-legged ceramic resonator can be used instead of the crystal, in that case C5 and C6 must be omitted. The 10 MHz crystal shown allows an 18F PIC to run at its full speed of 40 MHz (using the x 4 PLL). For a 16F PIC a 20 MHz crystal is required to run the PIC at its full speed.

Note that the capacitors are numbered C1, C2, C5, C6. The designations C3 and C4 are not used.

The four Dwarf Busses are available on the ML10 connectors BUS-A .. BUS-D. Note that BUS-A contains the PIC pins RA0..RA5 and RE0 .. RE2. Pin RA4 is not connected.

The connectors Wisp628 (7/8 pin header) and ICD (RJ11 6/6) can be used for in-circuit programming (ICSP) of the PICmicro using a Wisp628 programmer or Microchip programmer/debugger (for instance the ICD2). The diodes D1 and D2 serve to pull the LVP pin of IC1 low at the appropriate moments. When the ICD connector is used it might be necessary to pull the LVP pin low via de appropriate Dwarf Bus connector.

Dwarf Bus pinout

The tables below show the correspondence between Dwarf Bus pins and PICmicro controller pins.

Dwarf Bus BUS_A	PICmicro port
pin_0	RA0
pin_1	RA1
pin_2	RA2
pin_3	RA3
pin_4	RA5
pin_5	RE0
pin_6	RE1
pin_7	RE2

Dwarf Bus BUS_B	PICmicro port
pin_0	RB0
pin_1	RB1
pin_2	RB2
pin_3	RB3
pin_4	RB4
pin_5	RB5
pin_6	RB6
pin_7	RB7

Dwarf Bus BUS_C	PICmicro port
pin_0	RC0
pin_1	RC1
pin_2	RC2
pin_3	RC3
pin_4	RC4
pin_5	RC5
pin_6	RC6
pin_7	RC7

Dwarf Bus BUS_D	PICmicro port
pin_0	RD0
pin_1	RD1
pin_2	RD2
pin_3	RD3
pin_4	RD4
pin_5	RD5
pin_6	RD6
pin_7	RD7

Jal support

The table below shows the Jal board files for the DB016.

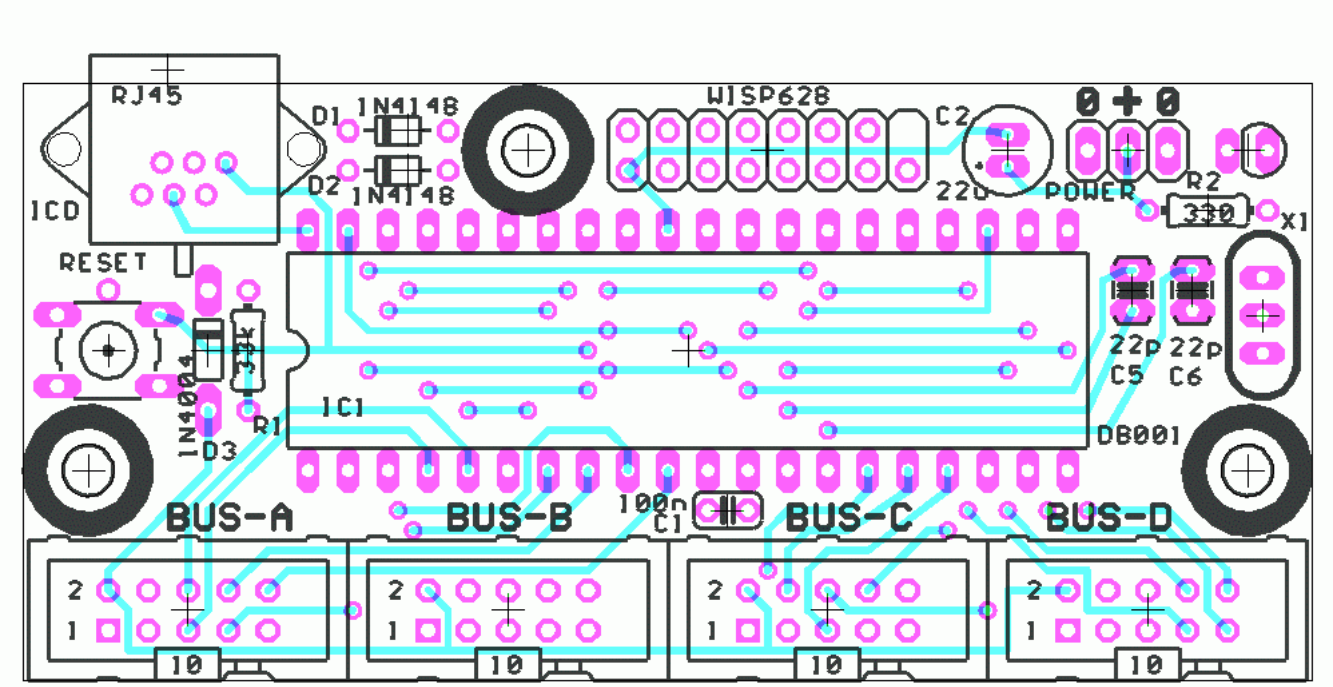
include file	for
DB001_18F452_10	DB016 with an 18F452 and a 10 MHz crystal, running at 10 MHz
DB001_18F452_20	DB016 with an 18F452 and a 20 MHz crystal, running at 20 MHz
DB001_18F452_PLL_40	DB016 with an 18F452 and a 10 MHz crystal, using the PLL to run at 40 MHz (this is the recommended configuration)

Other software

Microchip provides a C compiler for the 18F chips. This compiler is expensive, but a demo version of this C18 compiler can be downloaded from Microchips website. See links at the end of this document.

Component Placement

The figure below shows the placement of the components on the PCB.

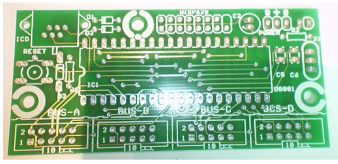
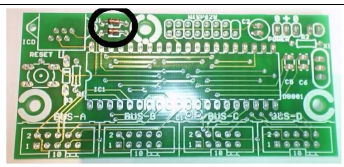
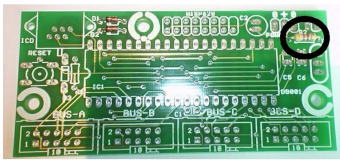
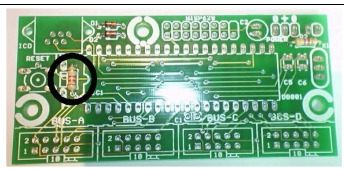
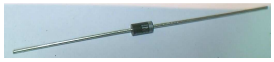
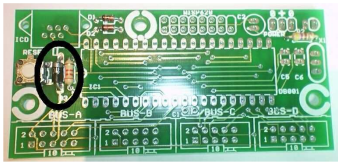
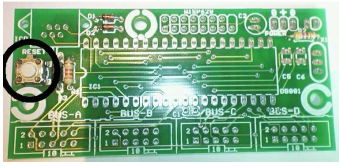
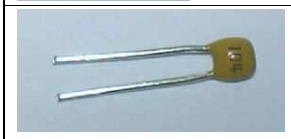
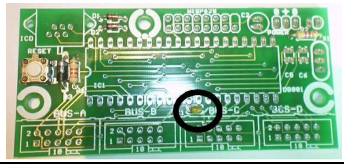
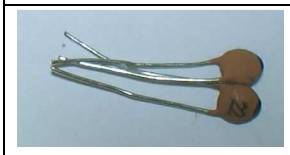
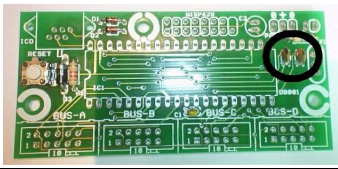


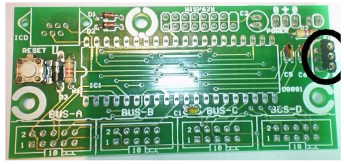
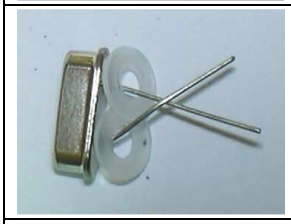
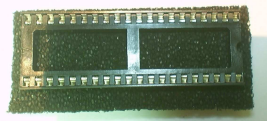
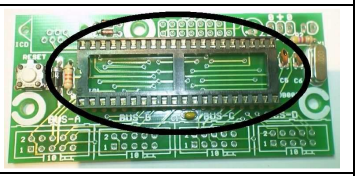
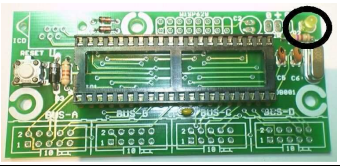
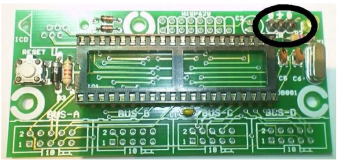
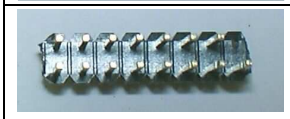
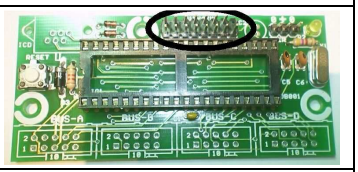
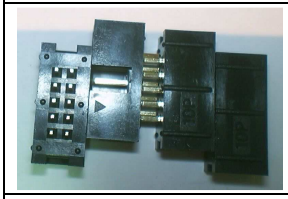
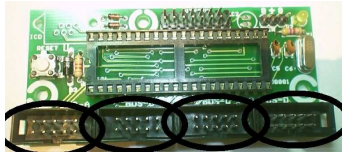
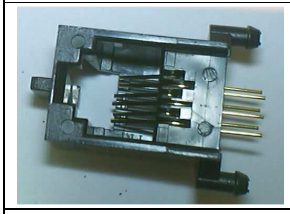
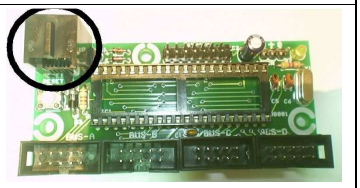
Assembly

The kit contains the PCB and all components as shown in the circuit diagram, except (depending on the kit) the PICmicro controller.

Assembling the kit is not difficult, no fine-pitched or SMD components are used. Just follow the instructions below carefully. Some soldering experience is recommended.

Orient the PCB as shown on the pictures and in the previous paragraph. The silkscreen (component side) text should be readable.

	Orient the PCB as shown on the pictures and in the previous paragraph. The silkscreen (component side) text should be readable.	
	Place and solder the two 1N4148 diodes D1 and D2. The dark band must point to the left.	
	Place an solder the 470 Ω resistors (yellow-purple-brown) R2. (Note: the silkscreen states a value of 330 Ω.)	
	Place an solder the 33 kΩ resistor R1 (orange orange orange).	
	Place an solder the 1N4004 diode D3.	
	Place and solder the pushbutton switch. Some force might be needed to push its legs into the PCB holes.	
	Place and solder the 100nF capacitors (marked 104) C1.	
	Place and solder the two 22pF capacitors (marked 22) C5 and C6.	

	If you want to be able to experiment with different crystals place and solder the 3-cup strip at the indicated position (the position for the crystal). Trim the crystal wires to 5 mm, put the crystal aside, and insert it in the strip when the board assembly is finished. Skip the next step.	
	Place and solder the 10 MHz crystal X1. Note that the two plastic isolators must be put between the capacitor and the board, to prevent shorts between the component-side traces and the crystal case.	
	Place and solder the IC socket. The notch must be at the left side.	
	Place and solder the yellow LED. The flat side of the LED must be at the left.	
	Place and solder the three pin strip. Skip this step if you prefer to solder the 5 Volt power wires directly to the PCB .	
	Place and solder the 2 x 8 pin strip. Note that the missing pin must be at the top right side.	
	Place and solder the four 10-pin connectors	
	Place and solder the 22 uF capacitor C2, directly to the left of the 3-pin header. Sorry, no foto's.	
	Place and solder the RJ connector. Forcing the plastic legs into their holes is difficult. You might need to enlarge the holes a bit.	
	Insert your PIC (not in the kit) in the IC socket. Note: the notches must be at the left and / or lower left.	

Links

- Microchip C18 compiler : **http://www.microchip.com/1010/pline/tools/picmicro/code/mplab18/index.htm**
- The PIClist is a mailing list for PIC users. Heavy traffic, but good information. http://www.piclist.com
- The Jallist is a mailing list for Jal users. http://www.voti.nl/jal/index.html

Errata

The PCB version 1.04 and probably earlier versions too (see solder side) has the ICD2 jack connector connected backwards. Solutions: either solder the jack connector inverted (but this is not very usefull because the opening of the connector will be inwards), or use a cable with the two connectors at the same side (the standard ICD2 cable has connectors at opposite sides of the cable).

Change notes

the latest version of this document can be downloaded from http://www.voti.nl/dwarf

version	date	notes
1.4	2008-11-05	RJ 8/8 changed to RJ 6/6
1.3	2005-11-18	erroneous text 'picture to the left shows...' removed
1.2	2004-07-07	assembly instructions corrected (D4, C2)
1.1	2004-06-28	PCB 1.04 erratum added
1.0	2004-01-13	first version