1 Printed circuit boards (PCBs)

You are provided with two prototyping PCBs on which to construct your circuits as follows:

1. **PCB1** - Contains an area customised for the construction of 4 line following circuits, together with connectors for you to plug in the IR emitter/detector assemblies. There is also some area for the construction of other circuits. **You are strongly advised to design your line following circuits (see Figure 1) and to construct them on this area of PCB1**. You should aim to achieve this by the end of week 2 at the latest. This will facilitate a major part of the software and mechanical tasks, namely getting the AGV to navigate the playing area.

2. **PCB2** - Mainly prototyping area for the construction of the remainder of your circuits.

   Both PCBs incorporate customised areas for the PCF8574 chip and its addressing. This chip is used to interface between your circuits and the microcontroller. In addition, they have DB9 plugs (9 pin D-shaped plugs/sockets) which are used to physically stack the boards and to connect together common signals across the boards (ground, 5 V, 12 V, ± 15 V and the SDA/SCL data and clock lines)

1.1 Customised vs non-customised PCBs

Customised PCBs have copper tracks laid out so that when the various circuit components are soldered onto the PCB, they are automatically connected together as required by the circuit diagram. As such, customised PCBs require minimal construction work, and are therefore very reliable.

Non-customised PCBs (prototyping PCBs) are useful when attempting to design and validate a circuit. Typically they consist of rows of holes connected by copper tracks (in your case 4 holes) together with 0 V and 5 V tracks. Whilst this type of layout is highly flexible and customisable, and is therefore good for prototyping, you will generally need to make additional connections using plastic-coated copper wire. Hence the price paid for this flexibility is increased construction effort and potentially decreased reliability.

1.2 PCB design and construction

Both PCBs are double sided, with tinned copper tracks on both sides, and with holes drilled so that you can insert and then solder components. Solder resist has been applied between the copper tracks to help prevent accidentally creating solder bridges between pairs of tracks. Some of the PCBs' tracks, as mentioned above, must be connected so that they are common to both boards, and for this to work such tracks require a means of being connected to each other despite existing on opposite sides of the board. Vias are used to achieve this. They are simply holes which have been soldered through so that the tracks on both sides of the board are then electrically connected by the via.

1.3 Exercise and questions

By performing a few simple exercises with your PCBs, and answering the related questions you will gain the understanding needed to successfully design the layout for your circuits, and hence build them. A good working knowledge of your PCBs will also facilitate any circuit debugging you may need to carry out in the latter stages of the project.

- **Exercise 1**

  Identify the 0V, 5V, SDA and SCL pins of the D-type connecters on PCB1

PCB Interconnections (D Plug)		
1	SCL	6 SDA
2	5v	7 +15v
3	12v	8 -15v
4	12v	9 Gnd
5	Gnd	

Using a digital multimeter check for electrical continuity of the respective pins ie that the pins have virtually no resistance between them on opposite sides of the board. Find the vias that enable this continuity for each of these signals, and label them on a photocopy of the track layout of PCB1.

- **Exercise 2**

  Stack PCB1 and PCB2 together my connecting male and female D-type connectors together. Now ensure that there is continuity between the 0V, 5V, SDA and SCL tracks on both boards.

- **Exercise 3**

  On PCB1 there is an area of the board devoted to the 4 line following circuits. By looking at the circuit diagram for the line following circuits (Figure 1 and also the proposed circuit layout (Figure 2) identify and colour in on a photocopy of the track layout of PCB1 the 0V and 5V tracks on the underside of PCB1. Get hold of the data sheet on the Hex Schmitt inverter chip (look in the electrical pages of the IDP website), and by observing the tracks on the underside of PCB1, identify and write down the input and output pin numbers of the Hex Schmitt inverter chip corresponding to each of the 4 line sensor circuits.

- **Exercise 4**

  This exercise concerns the PCF8574P chip, used to interface between your circuits and the microcontroller. First of all, obtain the data sheet for that chip, so that you know what each of the 16 pins is for.

  1. Which are the 0V, 5V, SDA and SCL pins ? Perform continuity checks between the tracks connected to these pins on the underside of PCB1 and the respective pins of the D-type connectors.
  2. To the left of the PCF8574P chip you will see a 3 by 3 grid of pins plus three jumpers which are used to connect pairs of those pins together. By observing the underside of PCB1, find out which pins of the PCF8574P chip are connected to the central column of the 3 by 3 grid of pins. The outside columns of the 3 by 3 grid of pins are connected to 0V and 5V tracks. By performing continuity checks or by obervation find out which is which. Hence explain how the jumpers are used to set the address of the PCF8574P chip, and draw on your photocopy of the track layout of PCB1 the jumper positions to set an address of 101.
  3. By tracing the paths of the tracks on the underside of PCB1 connected to the output pins of the Hex Schmitt inverter chip to the input pins of the PCF8574P chip, write down the bit numbers corresponding to each of the 4 line following sensors.

You should include your answers to these exercises, as well as your annotated photocopy of PCB1 in your First Report.

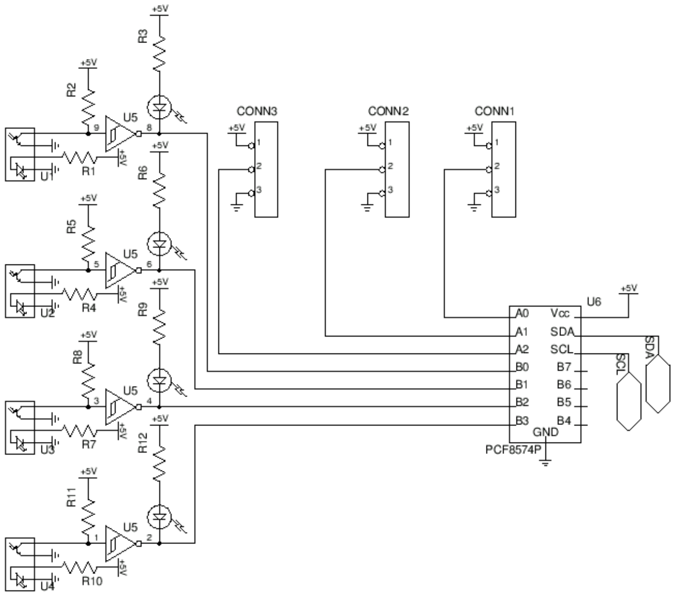

Figure 1: Line Sensor PCB Circuit

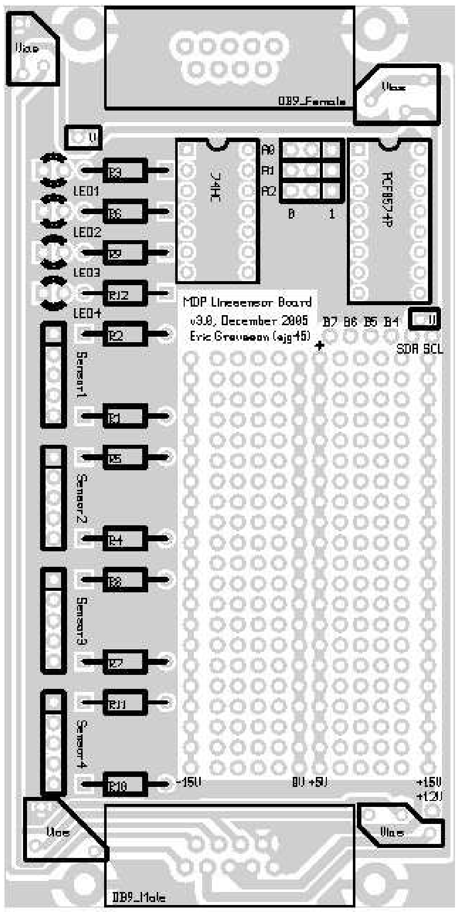

Figure 2: Line Sensor PCB Layout