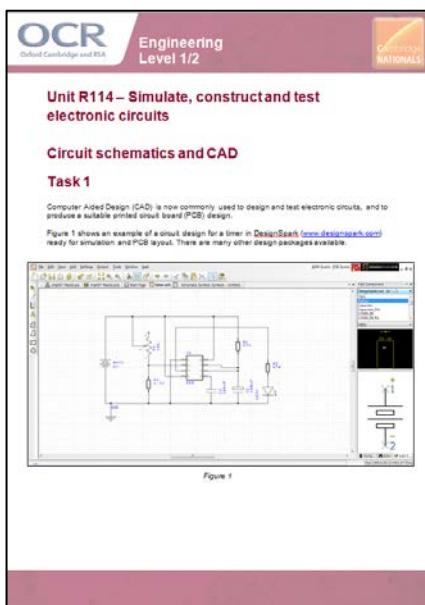


Unit R114 – Simulate, construct and test electronic circuits

Circuit schematics and CAD

Instructions and answers for teachers

These instructions should accompany the OCR resource ‘Circuit schematics and CAD’ activity which supports Cambridge Nationals in Engineering Level 1/2 Unit R114 – Simulate, construct and test electronic circuits.



OCR Engineering Level 1/2 Cambridge NATIONALS

Unit R114 – Simulate, construct and test electronic circuits

Circuit schematics and CAD

Task 1

Computer Aided Design (CAD) is now commonly used to design and test electronic circuits, and to produce a suitable printed circuit board (PCB) design.

Figure 1 shows an example of a circuit design for a timer in DesignSpark www.designspark.com ready for simulation and PCB layout. There are many other design packages available.

Figure 1

Associated Files:
Circuit schematics and CAD

Expected Duration:
Task 1 - approx 1 hour

This resource comprises of one task.



This activity offers an opportunity for maths skills development.

Task 1

This activity will give learners an opportunity to investigate the common key features of computer aided design (CAD) software used for circuit design, simulation and printed circuit board (PCB) layout design. It is anticipated that learners will undertake the activity, as an introduction, prior to using real CAD software with teacher guidance.

Learners may be able to use the internet to research a range of CAD software, or even be given the opportunity to download CAD software to explore practically. The example in the activity was produced using DesignSpark (www.designspark.com). The teacher might already have preferred CAD software available for learners to explore.

The following features are common to most circuit design, simulation and PCB design layout software, however, the teacher may wish to add or remove features for learners to investigate as required. The teacher might also wish to provide guidance to learners in order for them to complete the activity.

Ask the learners to record their research.

Schematic Capture/ Schematic Design	Schematic design is a term used for the process of entering (drawing) a circuit diagram into CAD software. Sometimes it is termed schematic capture. This is the part of the software where the circuit diagram is drawn visually.
Component Library	Software will contain a library of electrical components (eg resistors, capacitors, semiconductors etc) which are used in schematic circuit design. The components will usually have: a circuit symbol; simulation parameters; PCB layout part. New components can often be created or imported.
Connection or Interconnection	Connections (or wires) are used to connect components to produce a circuit diagram.
Grid	A grid is used to ensure the accurate placement of circuit symbols and interconnecting wires. Components and wires snap to set grid points. The grid spacing can sometimes be changed.
Design Checking (or design rule checking)	Most software packages will perform a check to make sure that components are placed correctly and that there are no broken or missing connections. Missing or unconnected wires might be shown a different colour.
SPICE (Simulation Program with Integrated Circuit Emphasis) simulation	SPICE is the name given to the circuit simulation part of schematic design software. It is a program that is used to check and predict circuit behaviour. Sometimes it is built into schematic design software, or is an external software program.
Netlist or Node List	The Netlist is a file that describes the components and connections for the circuit. It is often produced automatically by the schematic design part of the software and the file that SPICE uses to simulate the circuit.
Virtual Instruments	Virtual instruments are the equivalent of real test equipment used within the simulation part of the software. These will include d.c. and a.c. power sources, meters to read voltage and current, signal generators and oscilloscopes. They are virtual (not real) instruments that are used in the software when simulating and testing circuit operation.

<p>Auto component placement and Auto router (for PCB design)</p>	<p>The PCB design part of the software will take a schematic diagram and convert this into a PCB design layout ready for making. Many software packages will automatically place components on the PCB design layout, and automatically route tracks between these. This may or may not be done efficiently. The user often has the option of moving components around if required, or manually placing tracks.</p>
<p>Export Files (eg Gerber, DXF, IDF, csv, txt)</p>	<p>Data can usually be exported from design and simulation software such as the schematic diagram for printing, results of simulations for analysis and PCB layouts for manufacture. Files might also be sent to computer-controlled machines that can automatically produce a PCB.</p>
<p>Bill of Material (BOM)</p>	<p>Many software packages have the capability to produce a list of components required to manufacture the circuit (a Bill of Material) and this might also included manufacturer's part codes and order numbers. Some programmes even have the capability to automatically place an order for components with a supplier.</p>

To give us feedback on, or ideas about the OCR resources you have used, email resourcesfeedback@ocr.org.uk

OCR Resources: the small print

OCR's resources are provided to support the teaching of OCR specifications, but in no way constitute an endorsed teaching method that is required by the Board, and the decision to use them lies with the individual teacher. Whilst every effort is made to ensure the accuracy of the content, OCR cannot be held responsible for any errors or omissions within these resources.

© OCR 2014 - This resource may be freely copied and distributed, as long as the OCR logo and this message remain intact and OCR is acknowledged as the originator of this work.

OCR acknowledges the use of the following content:

Maths and English icons: AirOne/Shutterstock.com