

## Electrical Engineering - Circuits And Logic Template

 [In this topic](#)

The Electrical Engineering - Circuits And Logic Template includes integrated circuit component, analog and digital logic, transmission path, and terminal and connector shapes for use in creating circuit diagrams.

For information about how a particular shape behaves, right-click the shape, then choose Shape Help from the shortcut menu.

### Setting up your drawing page

By default, the Electrical Engineering - Circuits And Logic Template opens with an unscaled drawing page in portrait (tall) orientation.

#### To change the page settings and drawing scale:

1. Choose File > Page Setup.
2. On the Page Size tab and Drawing Scale tab, choose the settings you want for the drawing size, the printed page size, and the drawing scale, then click OK.

To change the measurement units, click the Page Properties tab and choose the unit you want to use from the Measurement Units list, then click OK.

See also:

[Rotating and resizing pages](#)

[Setting page orientation and scale](#)

#### To create a circuit diagram:

1. Add and connect the circuit components. Use the shortcut menu where possible to configure a shape.
2. Use the Label shape to add a label to each component. To replace the existing text, select the Label shape, then type the new text.

**TIP** You can find other label shapes on the General - Annotations stencil. To open the stencil, choose File > Stencils > Annotation > General - Annotations.

3. To add text to a circuit component, select the shape, then type. Many of the circuit-diagramming shapes have control handles that you can drag to move the shape's text block.

See also:

[About creating and revising connected drawings](#)

### Configuring circuit component shapes

When you drop some of the shapes from the Analog And Digital Logic stencil on the drawing page, Visio Technical prompts you for information about the shape's characteristics. For example, when you drop the Amplifier shape, a dialog box prompts you to set the amplifier type and the number of inputs and outputs. If you want to change the data in the property fields after you've dropped the shape on the page, right-click the shape, then choose the appropriate command, such as Configure Amplifier, from the shortcut menu.

Many other circuit-diagramming shapes can be configured using commands on the shortcut menu. For example, the 4X Building Block-Top shape does not prompt you for information when you drop it on the page, but you can right-click it to set it to a Top, Middle, Base, or Complete building block, and set the number of inputs and outputs and the number of top pins and base pins.

### Working with shape properties

A custom property is a field in which you can store information. You can enter data into a shape's existing fields by selecting the shape, and then choosing Shape > Custom Properties.

Each shape has a label property that you can edit by right-clicking the shape, then choosing Shape > Custom Properties. To display the label text on the shape, select the shape, then press F2 to activate the text block. Move the insertion point to the position for the label's text, then choose Insert > Field > Custom Properties (for Category) > Label (for Field), then click OK.

If you want to associate additional data with your circuit-diagramming shapes, you can run the Custom Properties Editor to add properties.

#### **To run the Custom Properties Editor:**

- Choose Tools > Macro > Custom Properties Editor.

See also:

[Adding, editing, and deleting custom-property fields](#)

#### **Generating reports from properties**

If you've associated custom-property data with your circuit-diagramming shapes, you can run the Property Reporting Wizard to generate inventory or numerical reports based on the data. For example, you could generate a parts list for the diagram.

#### **To run the Property Reporting Wizard:**

- Choose Tools > Property Report.

See also:

[Creating reports from custom data](#)

#### **Generating a netlist for circuit analysis**

You can generate a netlist to help you analyze your electrical circuit diagram using the Netlist Generator tool. The netlist text file includes a list of components used in the diagram and connections or nodes and component values. The Netlist Generator can automatically create the netlist from your diagram, provided that all components are 2-D and the circuit is drawn with all connections glued correctly.

#### **To run the Netlist Generator:**

- Choose Tools > Macro > Electrical and Electronic > Netlist Generator.

OR

- Choose Tools > Netlist Generator (an Electrical And Electronic Template must be open).

See also:

[Using the Netlist Generator](#)

#### **Linking shapes to other drawing pages, other files, or World Wide Web locations**

You can add navigational links to any shape in your diagram, so that users of the diagram can right-click the shape to jump to separate drawing pages, separate files, or documents on an intranet or the World Wide Web. For example, when you create a large or complex schematic diagram, you may want to divide it into smaller, more manageable files, so that each file contains a different section of the drawing. You can then link shapes in one section of the diagram to the Visio file that contains another section of the diagram.

#### **To add links to shapes:**

- Choose Insert > Hyperlink.

See also:

[About using hyperlinks](#)

#### **Placing Visio drawings on the World Wide Web**

You can easily convert a Visio drawing to a format Web browsers can read. Then you can distribute the

drawing on an intranet or the Web.

See also:

[Exporting shapes and drawings in .jpg or .gif format](#)

[Saving drawings as HTML pages](#)

[Configuring circuit component shapes](#)  
[Generating a netlist for circuit analysis](#)  
[Generating reports from properties](#)  
[Linking shapes to other drawing pages, other files, or World Wide Web locations](#)  
[Placing Visio drawings on the World Wide Web](#)  
[Setting up your drawing page](#)  
[Working with shape properties](#)

