


## Electrical Engineering - General Template

 [In this topic](#)

The Electrical Engineering - General Template includes switch and relay, transmission path, semiconductor, and tube shapes for use in creating general electrical diagrams, such as schematic, one-line, and wiring 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 - General 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 an electrical diagram:

1. Use shapes from the Title Blocks stencil to create a title block or add other reference information, such as a legend.

To open the Title Blocks stencil, choose File > Stencils > Annotation > General - Title Blocks.

2. Add and connect the electrical components. To configure a shape, right-click the shape and choose commands from the shortcut menu.
3. 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.
4. To add text to a shape, select the shape, then type. Many of the electrical shapes have control handles that you can drag to move the shape's text block.

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

See also:

[About creating and revising connected drawings](#)

### Configuring electrical shapes

When you drop some of the shapes from the Semiconductors And Electron Tubes stencil on the drawing page, Visio Technical prompts you for information about the shape's characteristics. For example, when you drop the MOSFET shape, a Custom Properties dialog box prompts you to define shape characteristics such as whether or not the envelope is visible. 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 MOSFET, from the shortcut menu.

Many other electrical shapes can be configured using commands on the shortcut menu. For example, you can right-click the SPST shape to open or close the circuit.

## 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, 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) > OK.

If you want to associate additional data with your electrical 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 electrical shapes, you can run the Property Reporting Wizard to generate inventory or numerical reports based on the data. For example, you could count the number of test points in 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 and connections or nodes and component values. The Netlist Generator can automatically create the netlist from your drawing, 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 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 diagram. 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 electrical 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](#)

