Electrical Engineering - Systems Template

The Electrical Engineering - Systems Template includes shapes for use in creating electrical or electronic schematic 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 - Systems 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 or electronic schematics diagram:

1. Use shapes from the Title Blocks stencil to add a title block or other reference information.

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

2. Add system component shapes, and use the shortcut menu where possible to configure a shape.

3. Connect the components by gluing the endpoints of 1-D shapes to connection points on 2-D shapes. The endpoints turn red, indicating that they're glued to the connector points.

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

5. Use shapes from the Annotations stencil to add other labels or annotations to the drawing.

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

See also:
About creating and revising connected drawings

Configuring systems shapes

When you drop some of the electrical and electronic shapes on the drawing page, Visio Technical prompts you for information about the shape's characteristics. For example, when you drop the Inductor shape, a Custom Properties dialog box prompts you to set the core type, adjustability type, and number of taps. 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 Set Inductor Properties, from the shortcut menu.

Many other electrical and electronic shapes can be configured using commands on the shortcut menu. For example, you can right-click the Changer shape to set the symbol type: General, Frequency Changer, or Pulse Inverter.

Working with shape properties

A custom property is a field in which you can store information. For example, the Connecting Stage shape includes properties for the number of inlets, outlets, and grading groups. 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 and electronic systems 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 and electronic systems shapes, you can run the Property Reporting Wizard to generate inventory or numerical reports based on the data. For example, you could generate an inventory of the three-conductor polarized connectors 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 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 Web. For example, when you create a large or complex systems 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