# ArcGIS® 9.3

## ArcGIS Schematics Designer Tutorial I: Working With the Standard Builder

Copyright © 1999-2007 ESRI

All rights reserved

**Copyright information**

The information contained in this document is the exclusive property of ESRI and its licensor(s). This work is protected under United States copyright law and other international copyright treaties and conventions. No part of this work may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying and recording, or by any information storage or retrieval system, except as expressly permitted in writing by ESRI. All requests should be sent to Attention: Contracts and Legal Services Manager, ESRI, 380 New York Street, Redlands, CA 92373-8100, USA.

The information contained in this document is subject to change without notice.

**U.S. Government Restricted/Limited Rights**

Any software, documentation, and/or data delivered hereunder is subject to the terms of the License Agreement. In no event shall the U.S. Government acquire greater than RESTRICTED/LIMITED RIGHTS. At a minimum, use, duplication, or disclosure by the U.S. Government is subject to restrictions as set forth in FAR §52.227-14 Alternatives I, II, and III (JUN 1987); FAR §52.227-19 (JUN 1987) and/or FAR §12.211/12.212 (Commercial Technical Data/Computer Software); and DFARS §252.227-7015 (NOV 1995) (Technical Data) and/or DFARS §227.7202 (Computer Software), as applicable. Contractor/Manufacturer is ESRI, 380 New York Street, Redlands, CA 92373-8100, USA.

**ESRI Trademarks**

@esri.com, 3D Analyst, ACORN, ADF, AML, ArcAtlas, ArcCAD, ArcCatalog, ArcCOGO, ArcData, ArcDoc, ArcEdit, ArcEditor, ArcEurope, ArcExplorer, ArcExpress, ArcGIS, ArcGlobe, ArcGrid, ArcIMS, ARC/INFO, ArcInfo, ArcInfo Librarian, ArcInfo—Professional GIS, ArcInfo—The World's GIS, ArcLessons, ArcLocation, ArcLogistics, ArcMap, ArcNetwork, *ArcNews*, ArcObjects, ArcOpen, ArcPad, ArcPlot, ArcPress, ArcQuest, ArcReader, ArcScan, ArcScene, ArcSchool, ArcScripts, ArcSDE, ArcSdl, ArcSketch, ArcStorm, ArcSurvey, ArcTIN, ArcToolbox, ArcTools, ArcUSA, *ArcUser*, ArcView, ArcVoyager, *ArcWatch*, ArcWeb, ArcWorld, ArcXML, Atlas GIS, AtlasWare, Avenue, Business Analyst Online, BusinessMAP, Community, CommunityInfo, Data Automation Kit, Database Integrator, DBI Kit, EDN, ESRI, ESRI—Team GIS, ESRI—The GIS Company, ESRI—The GIS People, ESRI—The GIS Software Leader, FormEdit, GeoCollector, Geographic Design System, ESRI BIS, Geography Matters, Geography Network, GIS by ESRI, GID Data ReViewer, GIS Day, GIS for Everyone, GISData Server, JTX, MapBeans, MapCafé, MapData, MapObjects, Maplex, MapStudio, ModelBuilder, MOLE, NetEngine, PC ARC/INFO, PC ARCPLOT, PC ARCSHELL, PC DATA CONVERSION, PC STARTER KIT, PC TABLES, PC ARCEDIT, PC NETWORK, PC OVERLAY, PLTS, Rent-a-Tech, RouteMAP, SDE, Site Reporter, SML, Sourcebook•America, Spatial Database Engine, StreetEditor, StreetMap, Tapestry, the ARC/INFO logo, the ArcAtlas logo, the ArcCAD logo, the ArcCAD WorkBench logo, the ArcCOGO logo, the ArcData logo, the ArcData Online logo, the ArcEdit logo, the ArcEurope logo, the ArcExplorer logo, the ArcExpress logo, the ArcGIS logo, the ArcGIS Explorer logo, the ArcGrid logo, the ArcIMS logo, the ArcInfo logo, the ArcLogistics Route logo, the ArcNetwork logo, the ArcPad logo, the ArcPlot logo, the ArcPress for ArcView logo, the ArcPress logo, the ArcScan logo, the ArcScene logo, the ArcSDE CAD Client logo, the ArcSDE logo, the ArcStorm logo, the ArcTIN logo, the ArcTools logo, the ArcUSA logo, the ArcView 3D Analyst logo, the ArcView Data Publisher logo, the ArcView GIS logo, the ArcView Image Analysis logo, the ArcView Internet Map Server logo, the ArcView logo, the ArcView Network Analyst logo, the ArcView Spatial Analyst logo, the ArcView StreetMap 2000 logo, the ArcView StreetMap logo, the ArcView Tracking Analyst logo, the ArcWorld logo, the Atlas GIS logo, the Avenue logo, the BusinessMAP logo, the Community logo, the Data Automation Kit logo, the Digital Chart of the World logo, the ESRI Data logo, the ESRI globe logo, the ESRI Press logo, The Geographic Advantage, the Geography Network logo, the MapCafé logo, the GIS Day logo, the MapObjects Internet Map Server logo, the MapObjects logo, the MOLE logo, the NetEngine logo, the PC ARC/INFO logo, the Production Line Tool Set logo, the RouteMAP IMS logo, the RouteMAP logo, the SDE logo, The Geographic Advantage, The World's Leading Desktop GIS, *Water Writes*, www.esri.com, www.esribis.com, www.geographynetwork.com, www.gis.com, www.gisday.com, and Your Personal Geographic Information System are trademarks, registered trademarks, or service marks of ESRI in the United States, the European Community, or certain other jurisdictions.

Other companies and products mentioned herein may be trademarks or registered trademarks of their respective trademark owners.

# Contents


| | |
|---|---|
| Introducing the ArcGIS Schematics Designer Tutorial I | 1 |
| Exercise 1: Creating a schematic dataset and defining the main parameters | 2 |
| Exercise 2: Generating schematic diagrams on the fly | 4 |
| Exercise 3: Customizing the schematic diagrams display | 10 |
| Exercise 4: Specifying schematic rules | 20 |

# Introducing the ArcGIS Schematics Designer Tutorial I

Welcome to the ArcGIS Schematics Designer Tutorial I.
This first tutorial has been designed to help you understand the ArcGIS® Schematics Standard Builder and some of the predefined Schematics rules. It details a simple way to quickly generate schematic diagrams from any feature selection set or trace result highlighted in a map document. The only requirement is that your feature classes must be organized into a geometric network.

## Introducing the sample database's content

### A summary of the Tutorial I database

When the ArcGIS Schematics tutorials were installed on your computer, the C:\arcgis\ArcTutor\Schematics\Schematics folder should have been created. This directory contains the ElecDemo geodatabase on which the first tutorial is based.

This sample database contains an electrical network and a land base.

- The data describing the electrical network is contained in the ElectricNetwork feature dataset. This feature dataset comprises the ElectricNetwork_Net geometric network, which is composed of the following feature classes: the CapacitorBank, ElectricNetwork_Net_Junctions, Feeder, OpenPoint, ProtectionDeviceBank, RegulatorBank, ServiceLocation, Substation and TransformerBank. These feature classes contain all the geometric network feature junctions. The PrimaryLine and SecondaryLine feature classes concern the geometric network feature edges.
- The LandBase feature dataset contains the County polygon features.

### Specifications for the tutorial schematic project

The purpose of this tutorial is to create a diagram type to automatically generate schematic diagrams from any feature set highlighted after a trace or selection operation in a map containing this geometric network. The symbology used to represent the schematic elements in the generated diagrams will be similar to that used in the map.

You will also learn how to configure Schematics rules that will be executed during your diagram generation to automatically reduce some schematic elements contained in your diagram or complete your diagram by adding new elements such as containers around some specific elements.

**Note:** In this tutorial, you will use specific functionality and commands (Diagram Type Assistant, Import From Feature Layers, and so on.) that allow you to quickly configure your schematic project in a minimum amount of time. To understand in detail how this functionality operates, it is recommended that you continue through the ArcGIS Schematics Designer Tutorial II where the less automated Designer functions are used.

## Introducing the tutorial main steps

In this tutorial, the schematic project conception is organized into four exercises:

- The purpose of the first exercise is to create the schematic dataset that will contain all the schematic data related to the schematic project you will build and to specify some default parameters for the related schematic project.
- During the second exercise, you will create the schematic diagram type that will be used to generate schematic diagrams on the fly from any set of features highlighted in a .mxd document containing the ElectricNetwork_Net geometric network data. This diagram type will work with the Standard Builder.
- The purpose of the third exercise is to customize the schematic diagram display.
- The last exercise concerns the predefined Schematics rules that can be automatically executed during the generation of diagrams that work with the Standard Builder. You will learn how to specify some rules parameters.

# Exercise 1: Creating a schematic dataset and defining the main parameters

In this exercise, you will learn to create a new schematic dataset, edit the related schematic project parameters within Schematics Designer, and specify some predefined default values for main parameters.

## Creating a schematic dataset

1. Start ArcCatalog™.
2. Click the Tools menu and click Extensions.
3. Check the Schematics check box.
4. Click Close
5. Select C:\arcgis\ArcTutor\Schematics\Designer in the Catalog tree.
6. Create a new file geodatabase in the Catalog tree and type a new name for this geodatabase ("TutorialI_SchematicDatabase", for example).

**Note:** You can also create a new personal geodatabase or use an existing ArcSDE® geodatabase.

7. Right-click this geodatabase, point to New, then click Schematic Dataset. A new schematic dataset appears in the Catalog tree.
8. Type a new name for this schematic dataset ("TutorialI", for example).

**Note:** An ArcEditor™ or ArcInfo® license is required for schematic dataset creation.

## Editing the related schematic project within Designer

1. Right-click the new schematic dataset in the Catalog tree and click Edit Project.
   The Schematics Designer application starts, loads the associated schematic project parameters in memory, and opens directly in design mode. As no diagram type already exists in this new schematic dataset, the Diagram Type Assistant automatically opens.
2. Cancel the Diagram Type Assistant for the moment.

**Note:** "Exercise 1: Creating a schematic dataset and editing the schematic project" of the ArcGIS Schematics Designer Tutorial II provides you with more details about creating and editing schematic datasets and projects.

## Setting schematic project default parameters

The Default Node Symbol parameter, available from the General tab is used by default to represent schematic nodes when their own Symbol parameter (defined for their related node element type) is not set.

To define this default symbol, follow the steps below:

1. Click the Default Node Symbol field.
2. Click the Edit Symbol button to launch the NgUSymbolEditor utility, the Schematics CGM Symbol Editor and Vector Drawing tool.
3. Use the NgUSymbolEditor drawing tools to build your CGM symbol as desired and type a name for the current symbol (Click File and click Save As).
4. Close NgUSymbolEditor (Click File and click Exit).

The Default Node Symbol field is automatically populated by the name of the symbol you have just created.

Parameters defining text effects that will be used by default to display the labels when these effects are not defined during the label definition, are available from the Schematic Project Effects tab. They are assembled into the Text Effects subset of parameters.

5. Click the Effects tab.
6. Click the Text Color field that will define the default color for the labels and click the color box that is displayed to launch the Table Color Editor.
7. Choose or define a color and click OK.
8. Click the Text Font field that will define the default text font for the labels.
9. Click one of the default fonts already defined from the drop-down list that appers, or click the ellipsis button displayed on the right to open the Font Editor. The Font Editor lists all fonts installed on your computer and allows you to choose the font you want.

Parameters defining the selection style are assembled on the Highlighting tab.

Three selection styles are available:

- With the Handle style, a selected schematic element is singled out by the display of its bounding box.
- With the Highlight style, a selected element is singled out by specific color, width, style, and fill.
- With the Handle And Highlight style, the Handle and Highlight selection styles are combined.

When the Highlight or Handle And Highlight style is used, some line, fill, and text effects must be set to complete the selection style definition and the appearance of links, nodes, and labels when they are selected in a schematic view.

10. Click the Selection Style drop-down arrow and choose Handle And Highlight.
11. To specify the color for the selected lines, click the Line Color parameter, click the color box to launch the Table Color Editor, then choose or define the desired color that will be used to highlight lines when they are selected.
12. To define the selected lines' width, click the Line Width parameter and type the desired value.
13. For defining the effects for the selected fills, click Solid from the Fill Style drop-down list and define the Fill Color parameter. These fill attributes will be used to display the symbol fill of the nodes that are selected.
14. To define the effects for the selected labels, click the color box that is displayed when clicking the Text Color field. Choose or define the color you want from the Table Color Editor that opens.
15. Click OK.
16. Click Save.
    By default, the schematic project definition coherence is checked before Schematics saves the project. This causes the Please verify dialog box to open before the saving is done. This dialog box sums up some points that may have been forgotten during your schematic project building such as creating an element type without associating it with a diagram type, creating a link element type without specifying the OriginNode and ExtremityNode attributes, or creating an element type without specifying its identifier. Note that the Please verify dialog box lists warnings with a critical-level mark (red, orange, or green traffic light) without preventing the saving. In fact, each warning is only informative and your project definition can be completely operational even if red warnings display on the dialog box.
17. Uncheck the Do not perform this verification when saving anymore check box and click Yes.
18. Close Schematics Designer.

**Note:** The schematic project parameters are more detailed in exercise 2 of the ArcGIS Schematics Designer Tutorial II. For more information about the parameters available from the Schematic Project General, Effects, and Highlighting tabs, please read "Exercise 2: Setting the schematic project's default parameters".

# Exercise 2: Generating schematic diagrams on the fly

The purpose of this exercise is to create a new schematic diagram type that works with the Standard Builder to generate schematic diagrams on the fly from any selected feature set or trace result in ArcMap™. Creating such a diagram type is easier than creating a diagram type that works with the Custom Query Based Builder because there are specific commands in Designer that allow you to define it with minimum setup work and configuration. The standard way to create such a diagram type is organized into the following two steps:

1. Create an ArcMap document containing the geometric network data you want to use to generate schematic diagrams, then symbolize this data.
2. Create the schematic diagram type that will implement the schematic diagrams, and use the Import From Feature Layers command to quickly create the schematic element types that will implement the schematic diagrams' content based on the ArcMap document's Table Of Content (TOC).

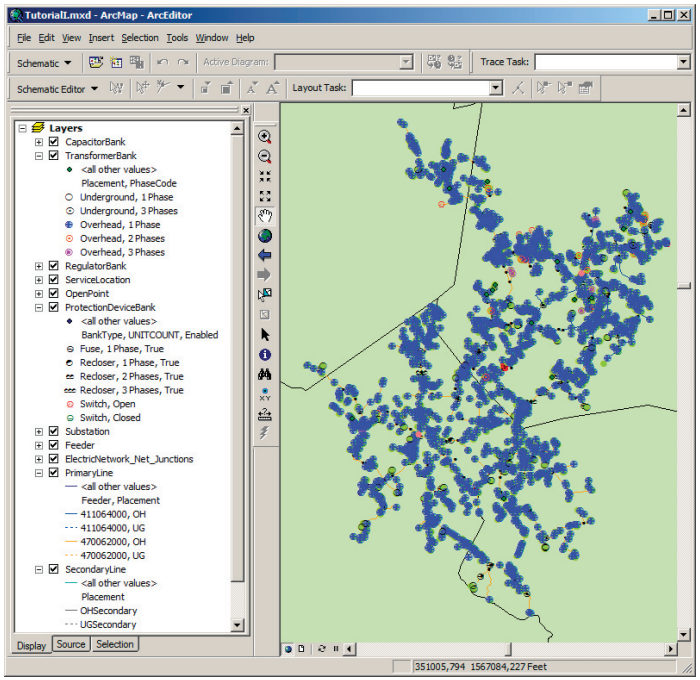
## Starting from an ArcMap document

When working with Schematics, you often have a map, map template, or layer file that contains the feature layers that comprise the geometric network with which you want to work and generate schematic diagrams. The symbology used to represent the feature layers in your ArcMap project is often already defined, categories have been created, and you are familiar with the representation. ArcGIS Schematics offers the possibility of using these layer definitions to quickly create schematic element types. This point will be detailed in the next section. The purpose of this exercise is to create an ArcMap document based on the ElectricNetwork_Net geometric network stored in the ElecDemo database.

1. Start ArcMap with a new empty map.
2. Click Add Data and browse to the ElecDemo database that should have been installed in C:\arcgis\ArcTutor\Schematics\Schematics.
3. Expand the ElectricNetwork feature dataset, select ElectricNetwork_Net, then click Add.
4. Click Add Data again, browse to the ElecDemo database, select LandBase, then click Add.
5. Click Selection and click Set Selectable Layers.
6. Click Select All and uncheck the HartGIS.County check box.
7. Close the Set Selectable Layers dialog box.
8. Configure the feature layers as you want. You can define categories for some of them if you want.
9. Save your .mxd file.

Below is the configuration of the TutorialI.mxd file that should have been installed in C:\arcgis\ArcTutor\Schematics\Schematics:

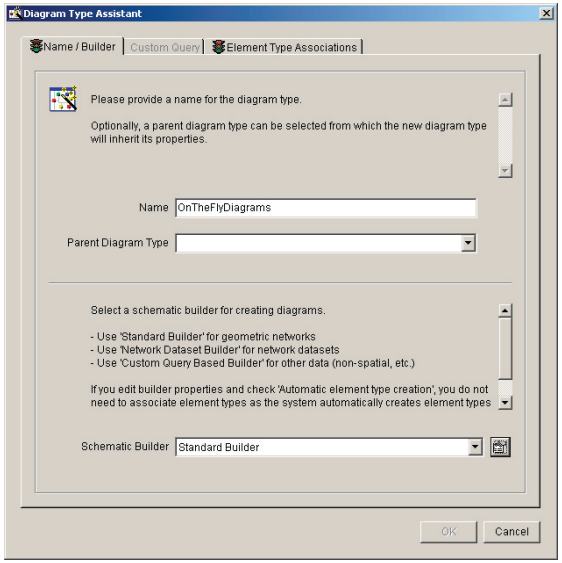
- A particular symbol has been specified to represent the CapacitorBank, ElectricNetwork_Net_Junctions, Feeder, OpenPoint, RegulatorBank, ServiceLocation, and Substation feature junctions.
- TransformerBank junctions are categorized according to their Placement and PhaseCode couple values.
- ProtectionDeviceBank junctions are categorized according to their BankType, Unitcount, and Enabled triplet values.
- SecondaryLine feature edges are categorized according to their Placement subtype values.
- PrimaryLine feature edges are categorized according to their Feature and Placement couple values.



## Creating a schematic diagram type to generate schematic diagrams from a map document

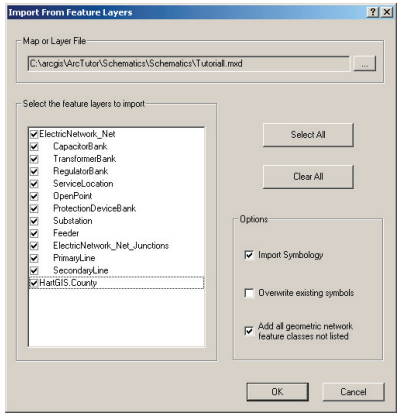
Create a schematic diagram type to generate schematic diagrams on the fly after a selection or a trace operation from any selected feature set in the .mxd document you have created.
In this exercise, you will learn how to use the Diagram Type Assistant to quickly create and specify such a diagram type.

1. Start ArcCatalog and activate the Schematics extension.
2. Browse to the geodatabase that contains the schematic dataset you created in exercise 1.
3. Right-click this schematic dataset and click Edit Project.
   Designer starts and your schematic project opens.
   Since no schematic diagram type already exists in this schematic dataset, the Diagram Type Assistant automatically opens.
4. Type a name for the schematic diagram type in the Name text box on the Name/Builder tab. For example, type “OnTheFlyDiagrams”.
5. Make sure that Standard Builder is selected in the Schematic Builder drop-down list.
   The Name/Builder tab content should now be similar to the following:

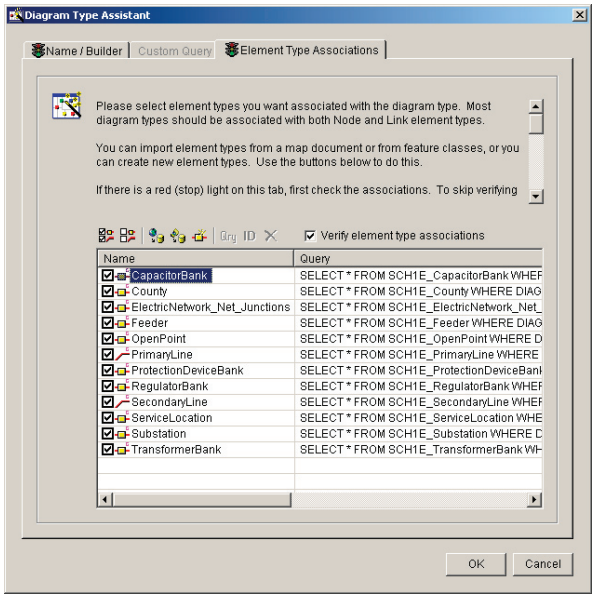



Next, define the schematic element types that will implement the schematic elements contained in the schematic diagrams related to this diagram type. As you want to use this diagram type to generate schematic diagrams from selected feature sets in the .mxd document you created in the previous section, the best way to complete this diagram type definition is to use the Import From Feature Layers command. This command creates schematic element types related to the feature layers displayed in a .mxd document. The rest of this tutorial is based on the provided TutorialI.mxd file. Even if your own .mxd file and the TutorialI are similar, use the TutorialI.mxd file so the screen shots in the rest of this document are consistent with the steps you will be following.

6. Click the Element Type Associations tab.
7. Click the Import From Feature Layers button.
   The Import From Feature Layers dialog box opens.
8. At the right of the Map or Layer File field, click the ellipsis button, browse to the TutorialI.mxd file installed in C:\arcgis\ArcTutor\Schematics\Schematics\, select it, then click Add.
9. All feature layers contained in this .mxd document should be displayed in the Select the feature layers to import area.
   The 11 feature layers that comprise the ElectricNetwork_Net geometric network should be out of line to the right of the first ElectricNetwork_Net entry. The HartGIS.County feature layer that is not included in the geometric network appears in the list at the same level of the geometric network entry. The check boxes corresponding to the ElectricNetwork_Net feature layers should all be checked by default. Keep them checked.
10. Check the HartGIS.County check box.

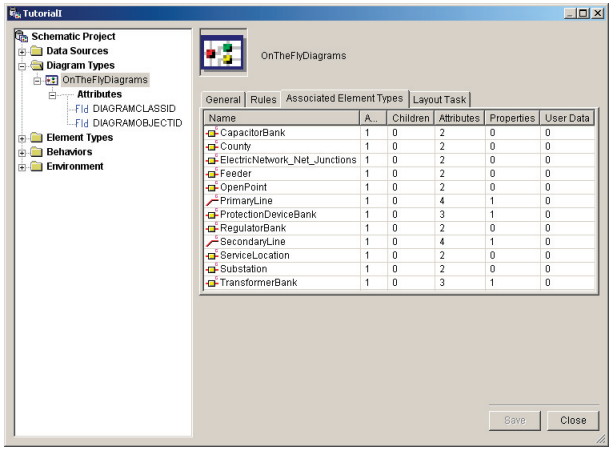


11. Click OK.
    The Import From Feature Layers dialog box closes. The Element Type Associations tab appears as follows:

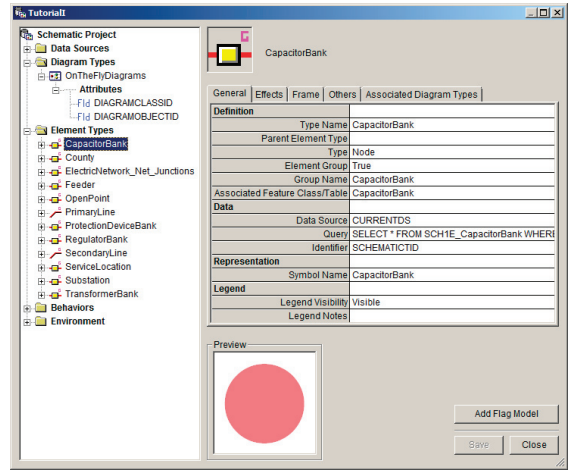



12. Click OK to close the Diagram Type Assistant dialog box.

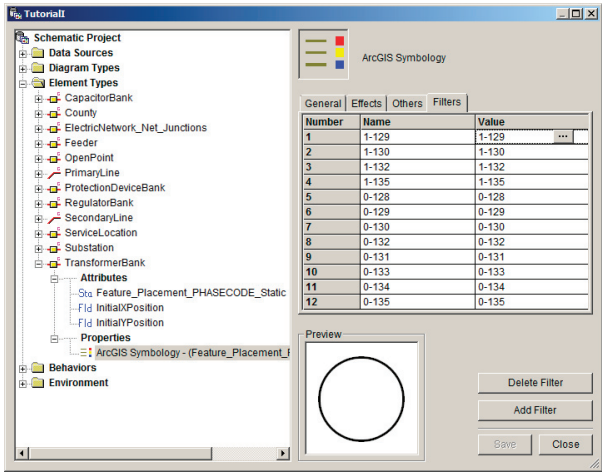
In this example, after the complete import, the OnTheFlyDiagrams new diagram type appears under the Diagram Types entry in the Designer tree.



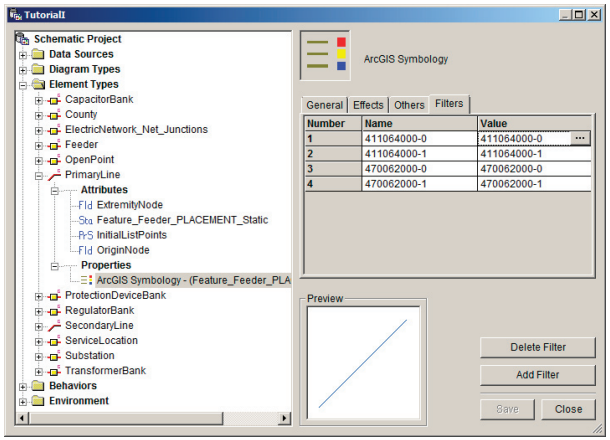

The Associated Element Types tab related to this diagram type contains all the element types that have been created during the import process and that correspond to the feature layers you checked.
If you expand the Element Types entry, you will find that an element type related to the CapacitorBank, ElectricNetwork_Net_Junctions, Feeder, OpenPoint, RegulatorBank, ServiceLocation, Substation, TransformerBank, ProtectionDeviceBank, PrimaryLine, and SecondaryLine geometric network feature layers and to the HartGIS County feature layer have been created.




A symbol has been created to display each schematic node in the same way as in the ArcMap document. When checking the ProtectionDeviceBank and TransformerBank node element types, you can note that an attribute and a property have been created to display each related schematic node according to database field values in the same way as their associated junctions are categorized in ArcMap.




When expanding the PrimaryLine and SecondaryLine element type entries, you can also see that an attribute and a property have been created to display the related schematic links according to database field values as they are in the .mxd file.

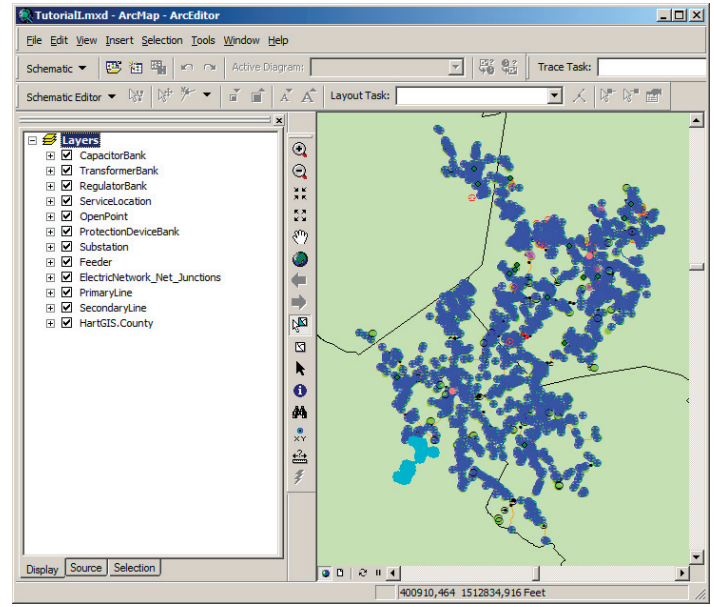



13. Click Save.

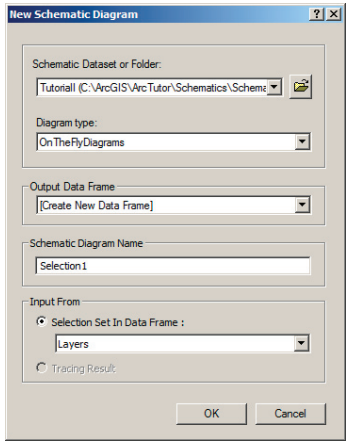
All schematic components required to generate a schematic diagram from a selected feature set or trace operation highlighted in the TutorialI.mxd document are now available. You can test how this new diagram type works.

14. Close Designer.

15. Start ArcMap and activate the Schematics extension.
16. Activate the Schematic, Schematic Editor, Schematic Network Analyst and Utility Network Analyst toolbars.
17. Load the TutorialI.mxd file.
18. Make sure the HartGIS.County layer is not selectable.
19. Click the Select Features tool and create a selection set of features in your map.

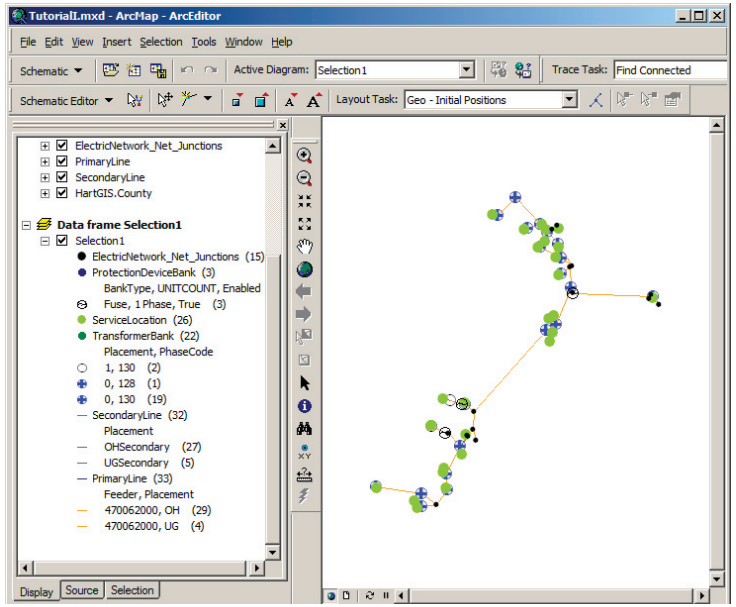



20. Click the Generate New Schematic Diagram command. The New Schematic Diagram dialog box opens.
21. Click the browse button next to the Schematic Dataset or Folder drop-down list to select the TutorialI schematic dataset in the TutorialI_SchematicDatabase geodatabase.
22. Click the Diagram type drop-down arrow, click OnTheFlyDiagrams.
23. In the Schematic Diagram Name text box, type a name for the schematic diagram that will be generated (Selection1, for example).
24. By default, the Selection Set In Data Frame option is checked in the Input From area.



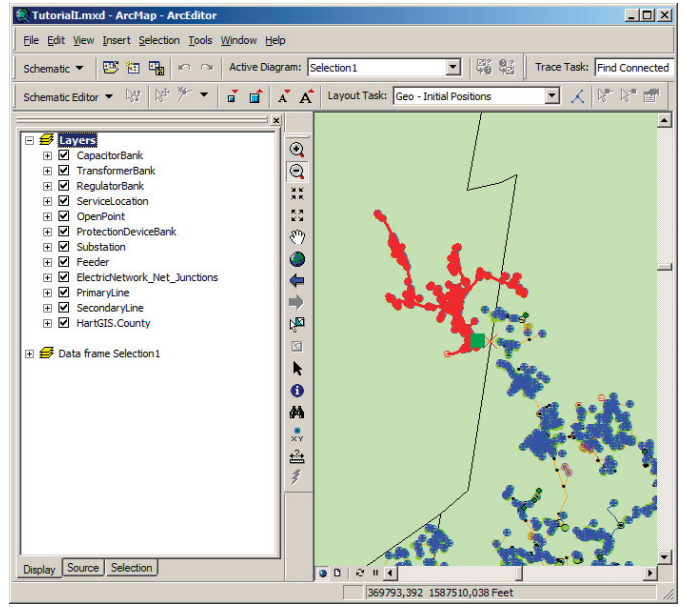

25. Click OK.

The new schematic diagram is generated. At the end of the generation process, a new schematic diagram containing the schematic elements associated with the selected set of features is displayed. The symbology used to represent the schematic elements should be similar to the symbology used to represent the associated features in the map.




26. Reactivate your map data frame and use the Utility Network Analyst toolbar to create a trace result.

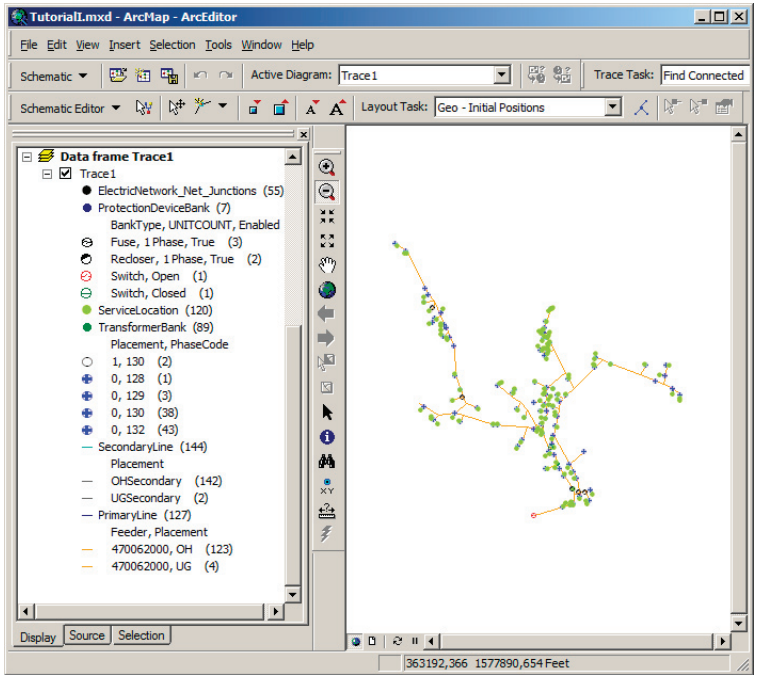
Below is a sample trace result obtained with the Find Connected trace task:




27. Click the Generate New Schematic Diagram command.
28. On the New Schematic Diagram dialog box that opens, make sure that the TutorialI schematic dataset is currently selected in the Schematic Dataset or Folder drop-down list .
29. Click the Diagram type drop-down arrow, click OnTheFlyDiagrams.
30. Type a name for the schematic diagram that will be generated (“Trace1”, for example) in the Schematic Diagram Name text box .
31. By default, the Tracing Result option is checked in the Input From area.
32. Click OK.

At the end of the generation process, a new schematic diagram containing the schematic elements associated with the set of features currently highlighted in the map is displayed.

33. Click Analysis on the Utility Network Analyst toolbar and click Clear Results. Repeat this operation to clear the flags and barriers you have specified.



# Exercise 3: Customizing the schematic diagrams display

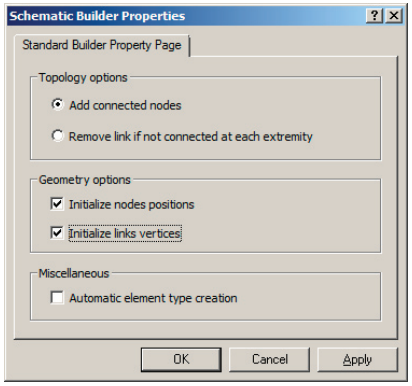
The purpose of this exercise is to learn how the generated schematic diagrams display can be customized.

- In the first section, you will learn how vertices that are displayed along the edges in the map can also be reported and displayed on the associated schematic link.
- In the second section, you will work with schematic attributes and properties to customize your schematic elements display by:

  - Adding and displaying labels on Feeder nodes,
  - Changing the ServiceLocation nodes' default symbol,
  - Creating a new script attribute to regroup values related to TransformerBank schematic nodes and using this attribute to display them.

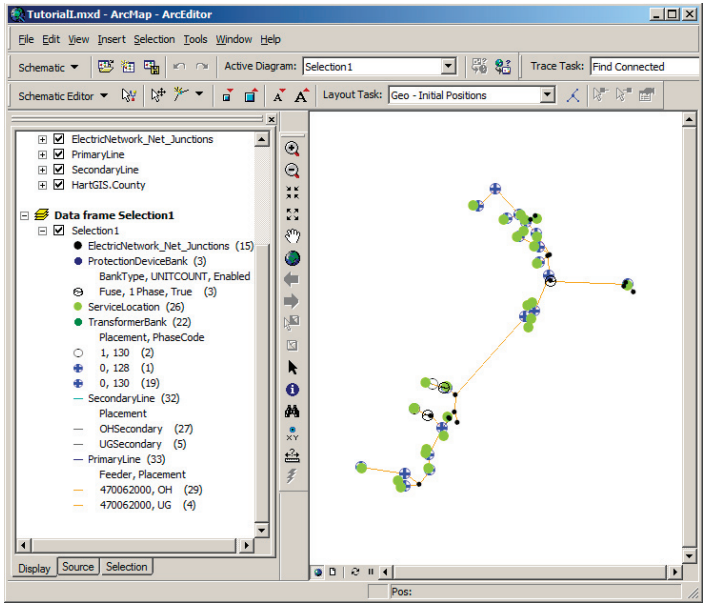
## Displaying vertices along links

1. Close ArcMap and edit your schematic project with Designer.
2. Click the OnTheFlyDiagrams entry under the Diagram Types main entry in the Designer tree and click the Schematic Builder field on the General tab. Click the ellipsis button that is displayed on the right. The Standard Builder Property Page tab is displayed.
3. Check the Initialize links vertices check box.



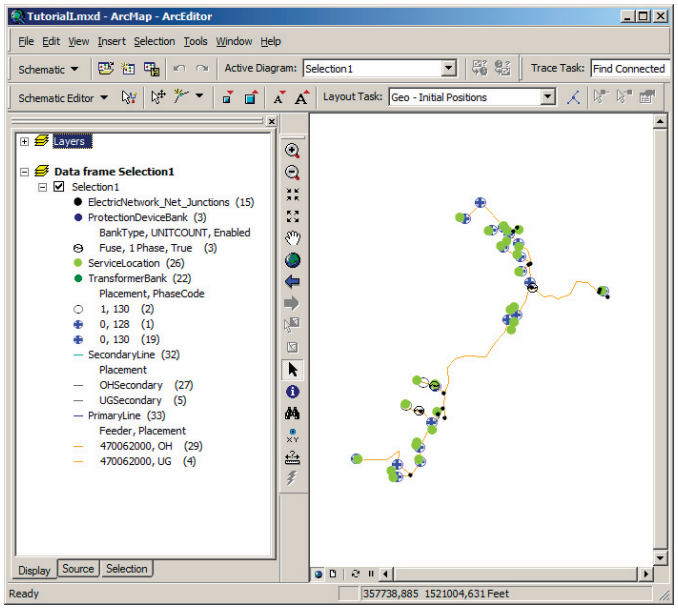

4. Click OK.
5. Click Save and close Designer.
6. Start ArcMap and open the TutorialI.mxd file. Select a set of features in the map and use the Generate Schematic Diagram command to generate the new OnTheFlyDiagrams schematic diagram from the current selection set.

A new schematic diagram opens. Schematic links contained in this diagram are now displayed with their vertices.




Note that for all schematic diagrams you have previously generated, if you reopen them, they display without vertices along links. To display vertices along these diagrams' links, you need to update the schematic diagram content.

7. Click Open Schematic Diagrams, browse to your schematic dataset, then choose one of the diagrams you generated previously.
8. Click the Schematic menu and click Update Diagram. The Update Diagram dialog box opens.
9. Make sure the Initial features option is selected and click OK. After the diagram updates, vertices are displayed along the links.




10. Close ArcMap without saving the .mxd file.

## Creating properties and attributes

Attributes and properties can be created to document and symbolize the content of schematic diagrams that work with the Standard Builder.

In this section, you will display the circuit description of the feeders at the top of each Feeder node and animate the ServiceLocation nodes according to the value of the OnWhat database field related to the associated feature junction.

1. Edit your schematic project with Designer.

### Labeling the Feeder nodes

Before creating the textual property that will be used to display the labels at the top of the Feeder nodes, you must create the attribute that will return the Feeder CIRCUITDESC field values.

Schematic elements contained in diagrams generated by the Standard Builder are managed in differently from those managed by the Custom Query Based Builder. During each schematic diagram generation, the Standard Builder proceeds in two steps.

- First, it analyzes the highlighted features in the ArcMap document, decodes the topology related to these features from the geometric network, builds all associated schematic nodes and links, and pulls these elements into specific schematic tables it has created in the schematic dataset.
- Second, by querying the set of elements stored in the schematic tables, the generated diagram opens.

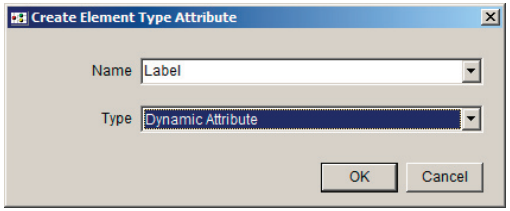
For the element types managed by the Standard Builder, two specific types of attributes can be created:

- The dynamic attribute retrieves field values directly from the feature class or table with which the schematic element type is associated.
- The static attribute imports field values from the feature class or table with which the schematic element type is associated into the schematic dataset during the diagram generation process.

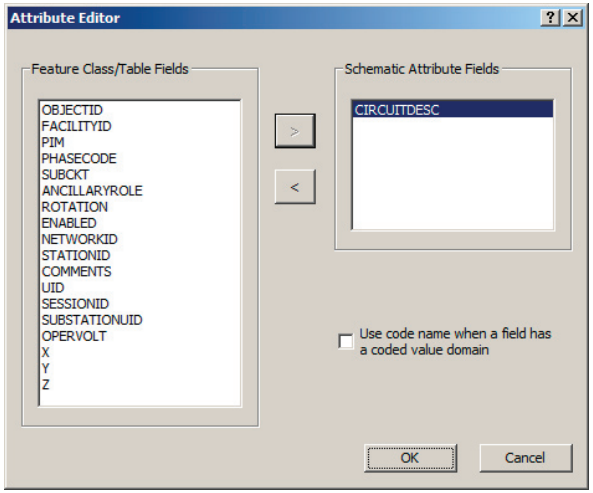
Note that if you create a static attribute and use it to document or animate your schematic elements, you will have to update any previously generated diagrams so the attribute values can be pulled into the schematic database and the elements displayed according to these values. With a dynamic attribute, the impact on the schematic element representation is automatic.

In this example, you are going to create a dynamic attribute related to the CIRCUITDESC field stored in the Feeder feature class:

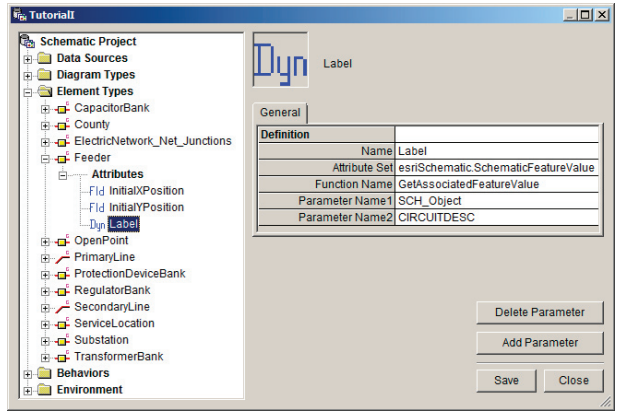
2. Right-click the Feeder element type entry that corresponds to the Feeder feature layer created during the import process.
3. Click Create Attribute. The Create Element Type Attribute dialog box opens.
4. Type a name for the attribute (“Label”, for example).
5. Click the Type drop-down arrow and click Dynamic Attribute.




6. Click OK.
   The Attribute Editor dialog box opens.
7. Click CIRCUITDESC in the Feature Class/Table Fields list and click the right arrow to validate this field.

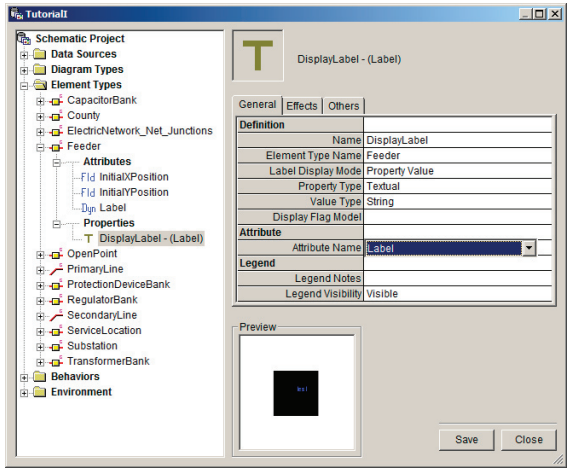



8. Click OK.
   The newly created attribute is displayed as follows:



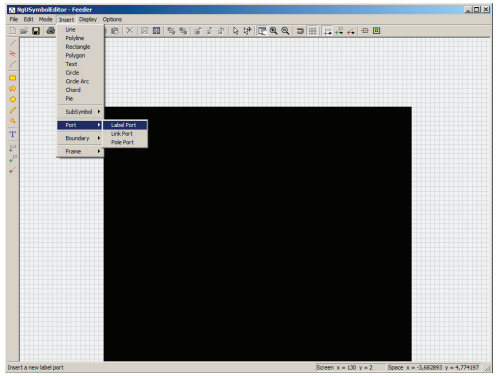

Next, you have to create the textual property that will be used to display this label.

9. Right-click the Feeder element type entry in the Designer tree and click Create Property. The Create Property dialog box opens.
10. Type a name for the property (“DisplayLabel”, for example) and keep the Textual option and the default String Value Type.
11. Click OK.
12. Click the attribute you have just created in the Attribute Name drop-down list on the newly created property’s General tab.

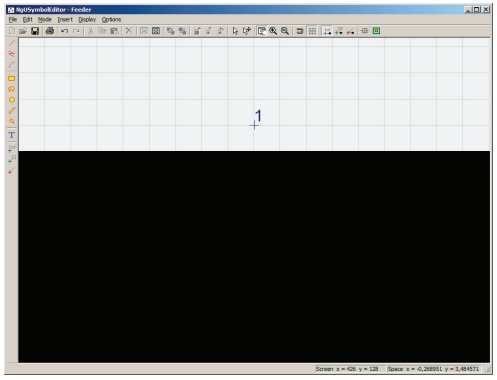


You must now specify how the labels will be displayed on each feeder node's symbol. Labels on nodes are usually placed according to the reference point of the symbol used to represent these nodes. This reference point is the center of the symbol. In this example, you will learn how to create label ports that will be used to attach your labels.

13. Click the Feeder element type entry in the Designer tree, click the Symbol Name field on the General tab, then, click the Edit Symbol button that opens the vector drawing tool (NgUSymbolEditor).
14. Click Fit All from the Edit menu and use the Zoom Out and Zoom In tool to fit on top of the symbol.
15. Click Insert, point to Port, and click Label Port.



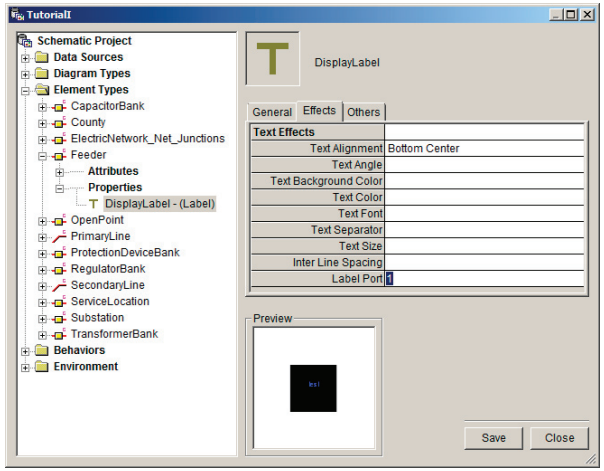
16. Click the top of the symbol to place a label port.
17. If needed, zoom to this location to check the label port position and reposition it if you want.



18. Click Save and close NgUSymbolEditor.

You can specify that you want your label to be attached to these label ports.

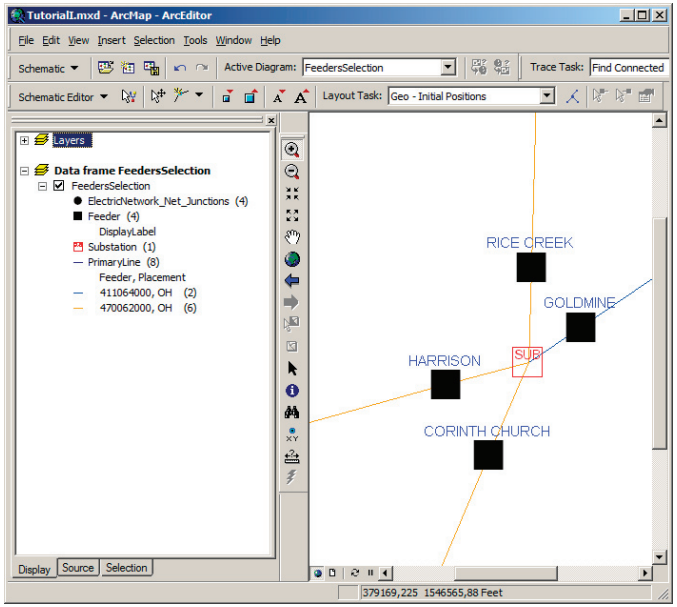
19. Click the DisplayLabel textual property you previously created and click the Effects tab.
20. Choose Bottom Center for the Text Alignment parameter; this will cause the label to be automatically centered at the top of the label port (in other words, the port will be bottom centered regarding to the label to which it will be attached).
21. Type "1" for the Label Port parameter.




22. Click Save, click Close, then exit Designer.
23. Start ArcMap and load the TutorialI.mxd file.
24. Activate your map data, right-click the Feeder feature layer, then point to Selection and click Select All.
25. Click Selection and click Zoom To Selected Features.
26. Use the Select Features tool to redefine a selection set containing the already-selected Feeder junctions and some of their neighbor features.
27. Use the Generate Schematic Diagram command to generate the new OnTheFlyDiagrams schematic diagram from the current selection set.
    A new schematic diagram opens.

28. If needed, use the Increase/Decrease Symbol Sizes and Increase/ Decrease Text Sizes buttons to adapt your elements' sizes.

After having zoomed in on the Feeder schematic elements in this newly generated diagram, your result will be similar to the following:



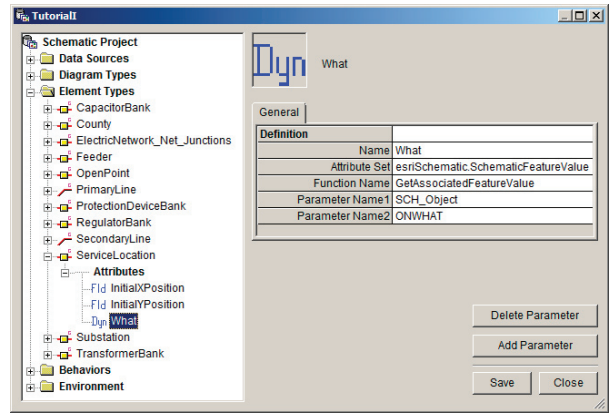

29. Close ArcMap without saving the diagram layout or the .mxd file.

## Symbolizing the ServiceLocation schematic nodes

In this example, you are going to create a discrete property that will be used to categorize the ServiceLocation nodes according to the OnWhat database field values. There are seven possible values for this field: Building, Cabinet, Other, Pole_Cons, Pole_Prim, Pole_Sec, and Temp. You will focus on the Building service location only; that is, you will represent the ServiceLocation schematic nodes with a particular symbol only when these elements are associated with a ServiceLocation feature junction for which the OnWhat field value is Building. For the other ServiceLocation nodes, the symbol used to represent them in the schematic diagrams will be similar to that used in the map to represent the associated ServiceLocation feature junction.
You will create a dynamic attribute related to the OnWhat field stored in the ServiceLocation feature class:

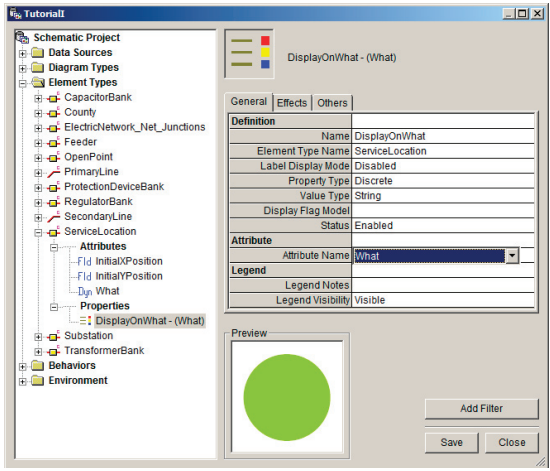
1. Edit your TutorialI schematic project within Designer.
2. Right-click the ServiceLocation element type entry in the Designer tree and point to Create Attribute. The Create Element Type Attribute dialog box opens.
3. Type a name for the attribute ("What", for example), click Dynamic Attribute from the Type drop-down list, and click OK.
4. In the Attribute Editor dialog box that opens, double-click OnWhat from the Feature Class/Table Fields list and click OK.

The newly created attribute display as following:



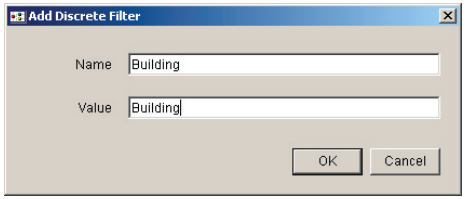
Create the discrete property that will be used to categorize the ServiceLocation nodes according to these attribute values:

5. Right-click the ServiceLocation element type entry in the Designer tree and click Create Property. The Create Property dialog box opens.
6. Type a name for the property (“DisplayOnWhat”, for example), check the Discrete option, and keep the default String Value Type.
7. Click OK.
8. Click the attribute you have just created in the Attribute Name drop-down list on the newly created property’s General tab.



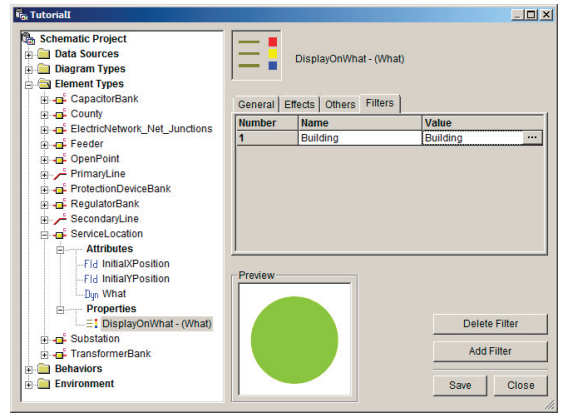

Next, you need to create the discrete filter that will be used to filter out the ServiceLocation whose related OnWhat field value is Building.

9. Click Add Filter. The Add Discrete Filter dialog box opens.
10. Type a name for the filter in the Name text area (“Building” in this example).
11. Type “Building” in the Value text box, and click OK.



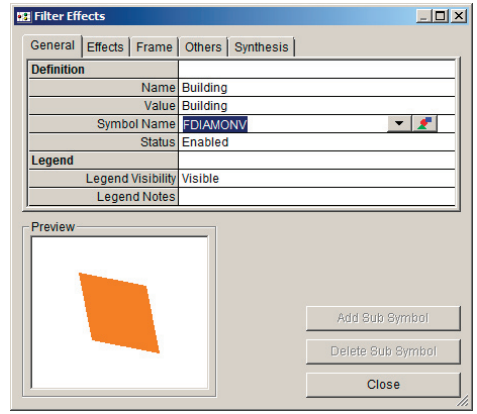

The new Filters tab is displayed among the discrete property’s tabs with a single item related to the Building filter.

12. Select this item and click the ellipsis button that appears at the end.

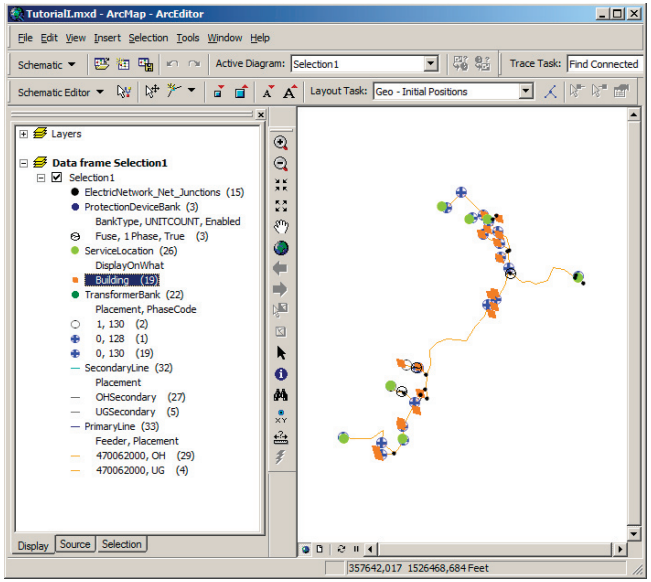



The Filter Effects dialog box opens. Specify the parameters you want to customize for the symbol used to represent the schematic nodes that will verify this filter.
In this example, the default symbol has been changed (to the FDIAMONV CGM symbol), the symbol color has been modified, and a specific angle has been set.

**Note:** When the CGM symbol used to represent the element type uses a specific font character, modifying the parameter of the symbol consists of modifying the Text parameters available from the Text Effects section on the Effects tab.



13. Close the Filter Effects dialog box.
14. Click Save, click Close, then exit Designer.
15. Start ArcMap and load the TutorialI.mxd file.
16. Click Open Schematic Diagrams and open one of the diagrams you have previously generated.
    If this diagram contains ServiceLocation schematic nodes, the symbol representing these nodes should now be different for the Building ServiceLocation nodes.



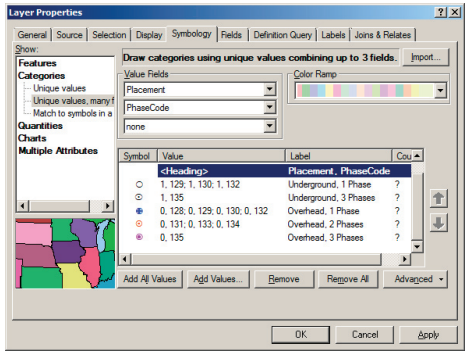

**Note:** If the new symbol used to display the Building ServiceLocation nodes is too small/big compared to the others' node elements, you can specify the Scaling parameter using Designer. This parameter is on the Others tab of the Filter Effects dialog box that opens by clicking the ellipsis button that is displayed at the end of the Building filter. For example, if you use the FDIAMONV CGM symbol, specify 6 for the Scaling parameter so all the nodes in your diagram have a very near scale.

**Note:** If instead of creating a dynamic attribute related to the OnWhat database field you had created a static attribute, you would have to update your generated diagrams so the new property effects would be reflected in these diagrams.

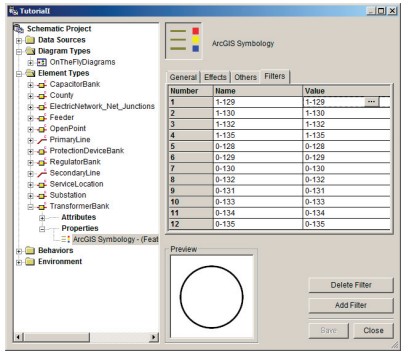
17. Close ArcMap without saving your diagram and .mxd file.

## Creating a script attribute to regroup property discrete filters

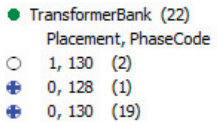
In this example, at the beginning of exercise 2, the TransformerBank features symbology had been customized to be represented according to the Placement and Phasecode values. Groups of values had been defined as follows:




Because Schematics discrete filters cannot be associated with several values, when the associated element type was created in the schematic dataset using the Import Feature Layers function and the discrete properties was automatically created to be coherent with the features symbology, these regroupments were lost. Each value constituting a group has been expanded into several filters, each being represented by the same symbology.

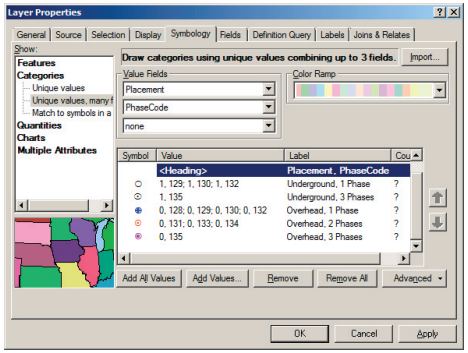


When generating a diagram based on several TransformerBank features related to a shared category in the map, the associated schematic elements might be dispatched in several schematic filters, and several entries with the same symbology might be displayed in the diagram legend.



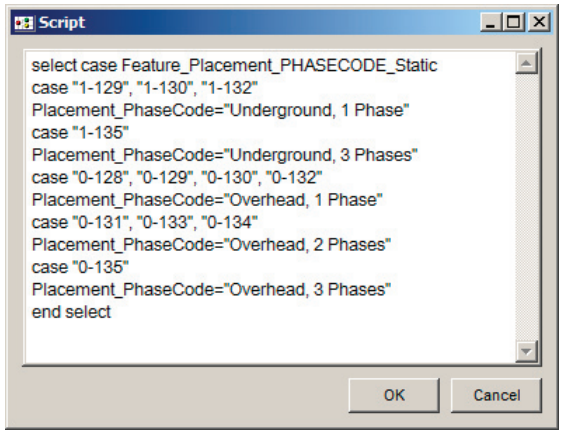

In this section, you will learn how these filters can be regrouped by creating a new script attribute and associating this attribute with the existing discrete property.

1. Edit your TutorialI schematic project within Designer.
2. Right-click the TransformerBank element type entry in the Designer tree and point to Create Attribute. The Create Element Type Attribute dialog box opens.
3. Type a name for the attribute (“Placement_PhaseCode”, for example), click Script Attribute from the Type drop-down list, then click OK.
4. Click the ellipsis button that is displayed at the end of the Script field on the General tab related to the newly created attribute.
5. The Script dialog box opens. Type the VB script code that will convert and regroup the desired values. In this case, the regroupment will operate on the Feature_Placement_PHASECODE_Static attribute values which already contain these initial values. In the map, the categories are defined as follows:

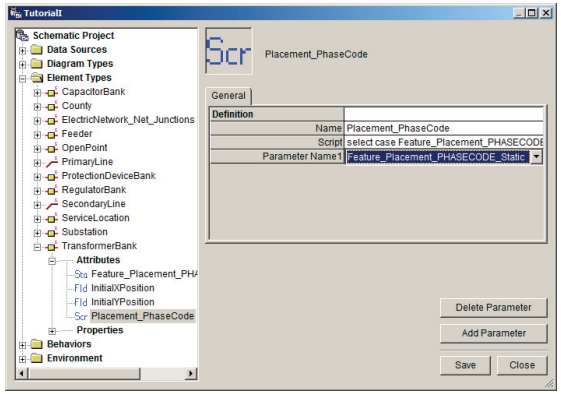



The VB script will be used to operate the exact same regroupments.

6. Type the following VB script code on the Script dialogbox:

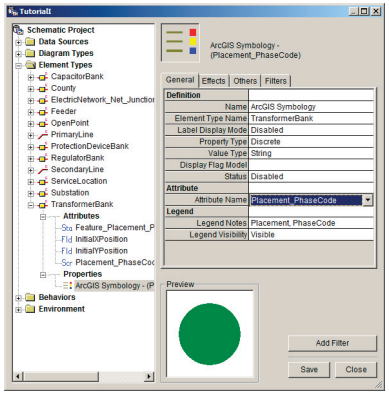



7. Click OK.
8. As the Feature_Placement_PHASECODE_Static attribute is used as a parameter in the script, click Add Parameter, click the Parameter Name drop down arrow, then select this attribute.

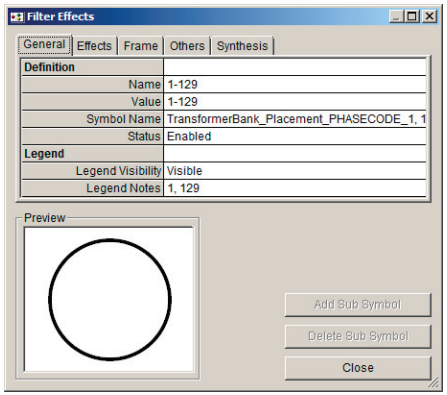



The attribute script definition is complete. You must now change the ArcGIS Symbology discrete property currently asssociated with the Feature_Placement_PHASECODE_Static attribute so it works with this new Placement_PhaseCode attribute script.

9. Start by changing the property-attribute association. Click the ArcGIS Symbology property that displays under the TransfomerBank entry in the Designer tree, and choose Placement_PhaseCode from the Attribute Name drop-down list.

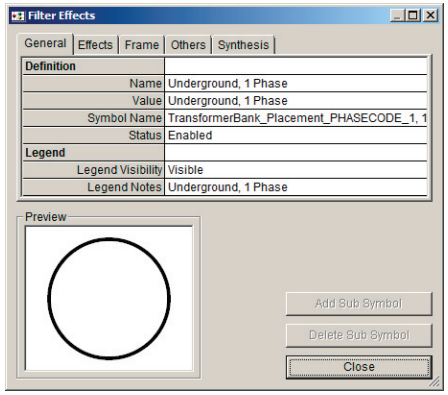



10. Click the Filters tab, click the first discrete filter parameter line related to the 1-129 value, then click the ellipsis button that is displayed to edit the parameters.
The Filter Effects dialog box opens.




According to the script code specified for the new Placement_PhaseCode attribute, the 1-129 filter value now corresponds to the Underground, 1 Phase value:

11. Change the Name, Value, and Legend Notes parameter values into Underground, 1 Phase as shown in the following screen shot.

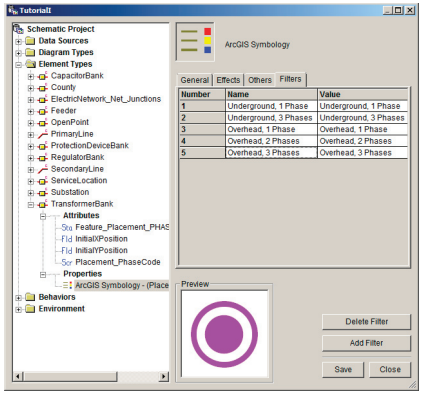



12. Click Close.

Since schematic elements related to the previous 1-130 and 1-132 discrete filters will be managed by the Underground, 1 Phase filter, these filters can be removed:

13. Select the discrete filter parameter related to the 1-130 value, and click Delete Filter. Repeat this operation for the next 1-132 filter.
14. Repeat steps 10 to 12 to change the 1-135 discrete filter into Underground, 3 Phases.
15. Repeat the same steps for the 0-128 discrete filter that must become Overhead, 1 Phase.
16. Remove the 0-129, 0-130, and 0-132 discrete filters that will be managed by the Overhead, 1 Phase filter.
17. Repeat steps 10 to 11 again to change the 0-131 discrete filter into Overhead, 2 Phases.
18. Because they will be managed by this Overhead, 2 Phases filter, remove the 0-133 and 0-134 discrete filters.
19. Edit the last 0-135 discrete filter parameters and change its Name, Value, and Legend Notes parameter values into Overhead, 3 Phases. Close the Filter Effects dialog box.

The ArcGIS Symbology discrete property Filters' tab appears as follows:




20. Click Save, click Close, then exit Designer.
21. Start ArcMap and load the TutorialI.mxd file.
22. Click Open Schematic Diagrams and open one of the diagrams you have previously generated. If this diagram contains TransformerBank schematic nodes, these nodes should be now grouped into the new discrete filters.

Figures 1 and 2 clearly show the impact of these changes in the legend. Figure 1 shows a diagram sample while the original ArcGIS Symbology discrete property was activated. Figure 2 shows the same diagram after that property has been redefined and associated with the new Placement-PhaseCode script attribute. The four legend entries with the same blue symbol that appeared in the TOC before the changes (figure 1) are now all regrouped under a unique legend entry (figure 2).

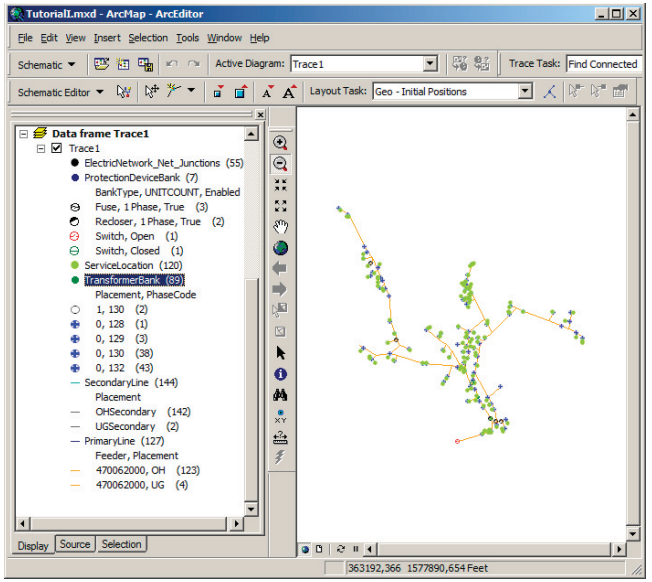



Figure 1: Before ArcGIS Symbology redefinition

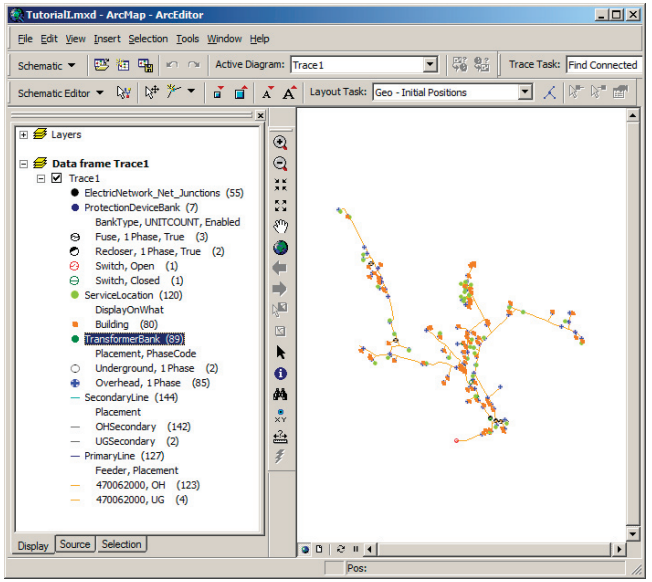



Figure 2: After ArcGIS Symbology redefinition

# Exercise 4: Specifying schematic rules

In this exercise, you will learn about schematic rules. Predefined schematic rules can be used to simplify diagrams (reduction rules) or create new elements, such as containers that are related to the elements in these diagrams (relationship rules).
Predefined rules provided with ArcGIS Schematics 9.3 only work with diagrams managed by the Standard or Network Analyst builder. Rules specified for these diagram types are executed during schematic diagram generations.

In the following sections, you will learn about four rules: Node Reduction By Priority, Relationship, Spatial Query, and Expand Link rules.

- The Node Reduction By Priority rule allows you to generate schematic diagrams after all nodes or some particular nodes related to a given schematic element type have been removed and after the links that connected these removed nodes have been reconnected on a given target node so the topology is preserved.
- The Relationship rule allows you to create schematic elements and relations regarding the information stored in a relationship class from the set of features currently selected in the map.
- The Spatial Query rule allows you to add new schematic elements from one or more schematic element classes based on where their associated features are located in relation to the features related to another schematic element class.
- The Expand Links rule allows you to generate schematic diagrams after all links or some particular links related to a given schematic element type have been expanded into several links based on specific values. These values may be brought by the original links themselves or by any other attributes brought by other elements contained in the diagram.

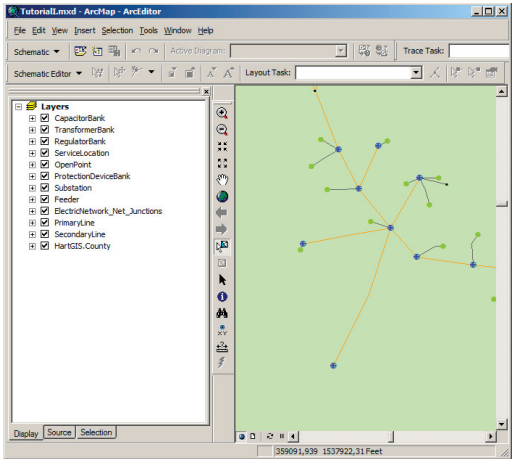
Several rules can be specified for your diagrams. They are chained in their entry sequence order during each diagram generation process.

- In this exercise, you will first specify parameters for a Spatial Query rule that will be used to detect and automatically complete the generated schematic diagrams by the secondary network elements that are very close to each selected TransformerBank feature.
- Then you will specify relationship rules that will be used to automatically add the counties related to the TransformerBank and ServiceLocation schematic nodes contained in a diagram and display them as containers around the TransformerBank and ServiceLocation in this diagram.
- Next, you will specify parameters for a Node Reduction By Priority rule that will be used to simplify your generated diagrams by reducing the ElectricNetwork_Net_Junctions elements. Only the junctions with one connection (one link) will be kept.
- Finally, you will create an Expand Links rule to expand the Primary Lines schematic links according to their PhaseCode field values. The Separate Overlapping Links layout automatic triggering will also be specified to allow users to clearly visualize that rule effects at the diagram loading.

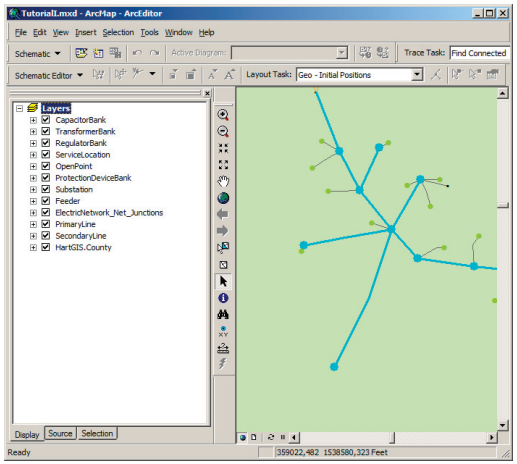
## Specifying a Spatial Query rule

At first, generate a sample diagram that will mainly contain TransformerBank features connecting SecondaryLine edges without the associated SecondaryLine schematic links to be included in the generated diagram. After specifying a Spatial Query rule to automatically add SecondaryLine links, you will analyze the impact of the rule execution on that sample diagram.

1. Start ArcMap and load the TutorialI.mxd file.
2. Zoom in on a part of the map that contains several TransformerBank junctions with incident SecondaryLine edges.

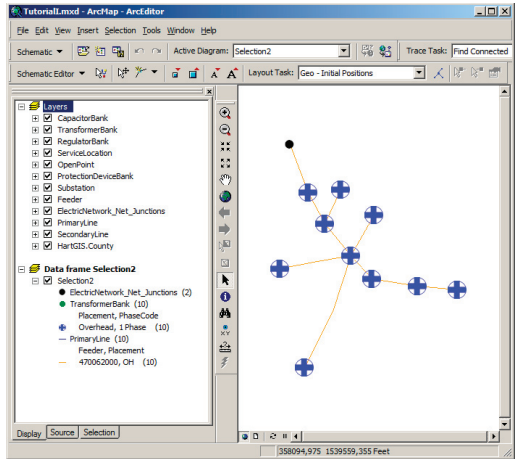


3. Click Selection and click Set Selectable Layers. The Set Selectable Layers dialog box opens.
4. Uncheck SecondaryLine and ServiceLocation.
5. Click the Select Features tool and select some PrimaryLine and TransformerBank features in the map.




6. Click the Generate New Schematic Diagram command. The New Schematic Diagram dialog box opens.
7. Select the TutorialI schematic dataset in the TutorialI_SchematicDatabase geodatabase.
8. Click the Diagram Type drop-down arrow and click OnTheFlyDiagrams.
9. Type a name for the schematic diagram that will be generated ("Selection2", for example) in the Schematic Diagram Name field, and click OK.

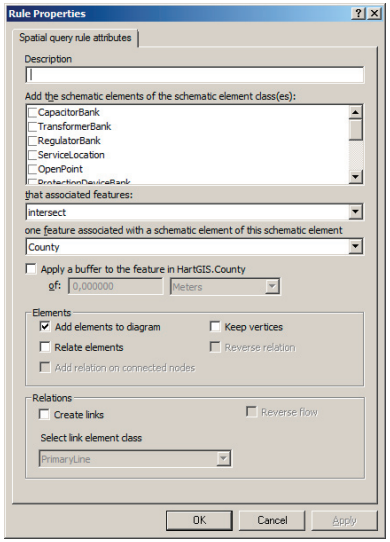
The following screen shot shows the sample diagram obtained.




10. Close ArcMap without saving the .mxd file.

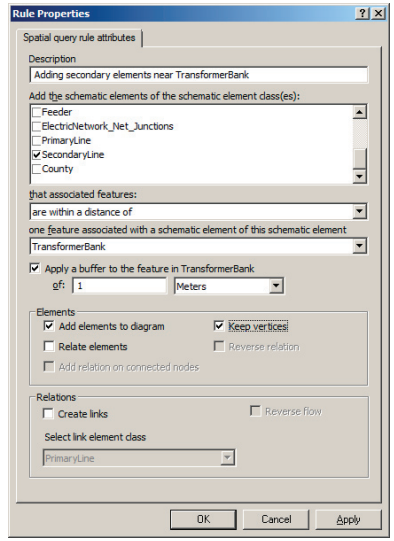
You will now edit the schematic project and create a Spatial Query rule that will be used to automatically add the secondary network elements when these elements are not initially in a diagram.

11. Edit your TutorialI schematic project within Designer.
12. Click the diagram type entry in the Designer tree (OnTheFlyDiagrams in this example).
13. Click the Rules tab from the right part of the Designer Editor window.
14. Click Add Rule at the bottom right corner of this tab.
    A new empty item appears in the first line on the tab.
15. Click the Name field and choose Spatial Query Rule from the drop-down list.
16. Click the Description field and click the Ellipsis button that is displayed at the end of the rule item.
    The Spatial query rule attributes tab opens.




In this example, the Spatial Query rule will be used to detect all secondary network elements within a distance of 1 meter from any TransformerBank and automatically add these elements into the generated diagrams. When link elements are added, their origin and extremity nodes are also automatically added by the rules, you only need to ask for adding the SecondaryLine links.

17. Type a name for this rule in the Description text box (for example, “Adding secondary elements near TransformerBank”).
18. Check the SecondaryLine element class check box in the Add the schematic elements of the schematic element classes list.
19. Choose ‘are within a distance of’ from the that associated features drop-down list.
20. Choose TransformerBank from the ‘one feature associated with a schematic element of this schematic element’ drop-down list.
21. Make sure the Apply a buffer to the feature in TransformerBank of check box is checked.
22. Type “1” in the text box and choose Meters from the drop-down list.
23. Since you want the rules to simply add the detected elements to the generated diagram, make sure that the Add elements to diagram check box is checked.
24. Check the Keep vertices check box to preserve the vertices along links.

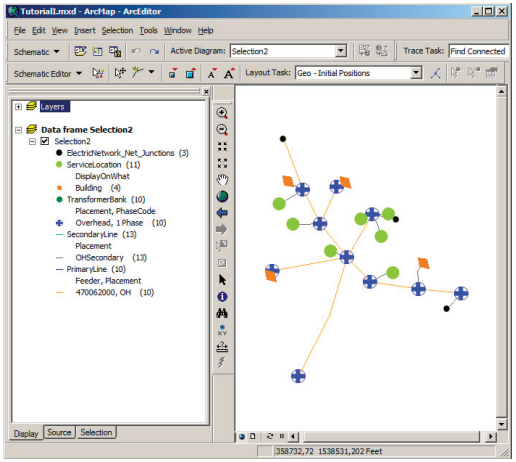



25. Click OK.
26. Save the schematic project definition, close Schematics Designer then exit.

You are ready to test the impact of this first rule in the schematic diagram generated in step 9.

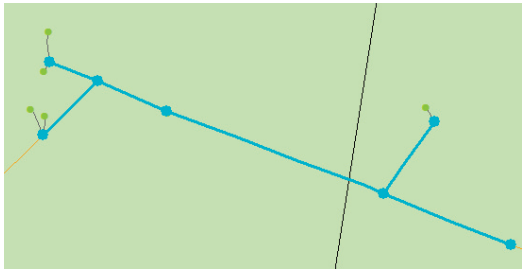
27. Start ArcMap and load the TutorialI.mxd file.
28. Open the diagram.
29. Click Schematic and point to Update Diagram.
30. Make sure the Initial features option is selected on the Update Diagram dialog box that opens and click OK.

Following is the result of the Update diagram command on the sample diagram.

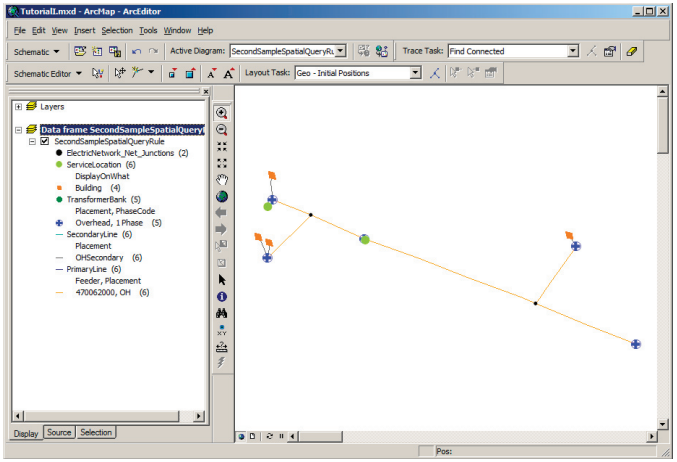



If you compare this updated diagram with the same diagram before the update (see page 21), you can see that (1) SecondaryLine links that are very close to TransformerBank nodes initially contained in the diagram are now also contained in the diagram, (2) ServiceLocations and Electric_NetworkNet_Junctions nodes that often constitute the SecondaryLine links' extremities have been also automatically added in the updated diagram.

Here is another sample of a generated diagram from some TransformerBank features selected in the map.



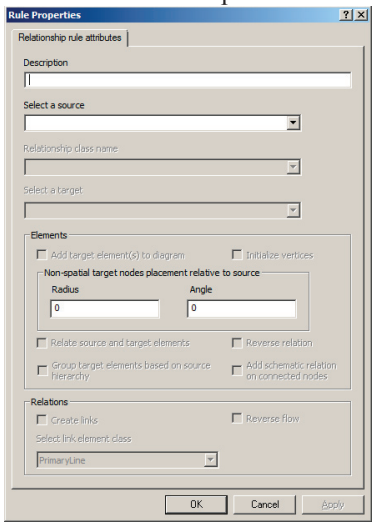
The above screen shot shows the set of features selected in the map from which the diagram generation starts. The following screen shot shows the resulting schematic diagram:



## Specifying a Relationship rule

You will now chain this Spatial Query rule execution with a second set of rules, Relationship rules, which will be used to add and display county elements as containers around the TransformerBank and ServiceLocation elements in a schematic diagram.

1. Close ArcMap and edit your TutorialI schematic project within Designer.
2. Click the diagram type entry in the Designer tree (OnTheFlyDiagrams in this example).
3. Click the Rules tab from the right part of the Designer Editor window.
4. Click Add Rule at the bottom right corner of this tab.
   A new empty item appears in the second line of the tab.
5. Click the Name field and choose Relationship Rule from the drop-down list.
6. Click the Description field and click the ellipsis button that displays at the end of the rule item.
   The Relationship rule attributes tab opens.




7. Type a description for this rule in the Description text box (for example, “Adding containers related to TransformerBank”)
8. Click the Select a source drop-down arrow and click TransformerBank..

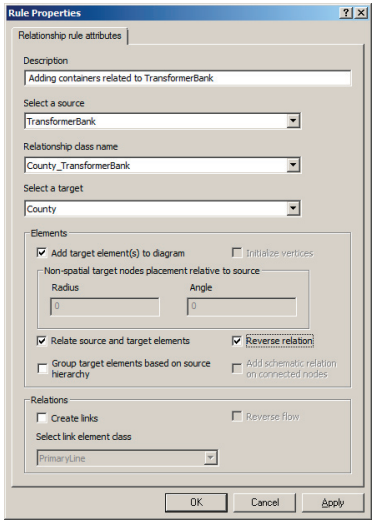
As in the geodatabase, there is a relationship class related to the TransformerBank feature class. All the controls on the tab are automatically available.

9. Make sure that County_TransformerBank is selected in the Relationship class name drop-down list.
10. Make sure that County is specified as the target element class in the Select a target drop-down list.
11. Check Add target element(s) to diagram so the counties related to the TransformerBank elements as specified in the County_TransformerBank relationship class are automatically added in the generated diagram.

Next, you have to specify how the relation must be represented in the generated diagram. Checking the Relate source and target elements check-box causes the schematic source elements to be represented as containers around the schematic target elements to which they are related. In this example, since it is the County targets you want to be represented as containers around the TransformerBank sources, you will have to reverse the relation.

12. Check the Relate source and target elements check box.
13. Check the Reverse relation check box.

The Relationship rule attributes tab now appears as follows:

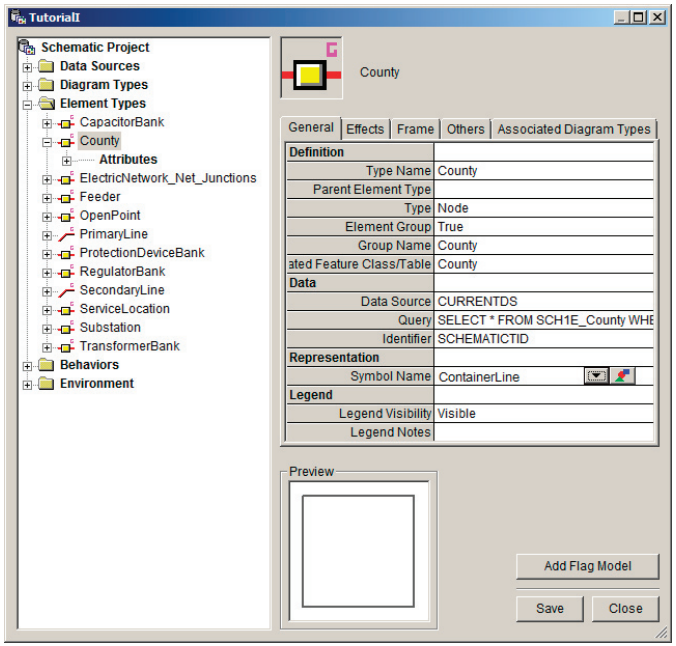


14. Click OK to close the tab.
15. Click Save.

The symbol used to represent the container elements must be appropriate. Generally, using a squared polyline is the best way to graphically represent them. In this example, you need to create and specify the symbol that will be used to represent the County as containers.

16. Expand the Element Types entry in the Designer tree and choose County.
17. Click the Symbol Name field on the related General tab and click the Select Symbol button that is displayed.
18. The Symbol Browser opens. Select the ContainerLine symbol and click OK.

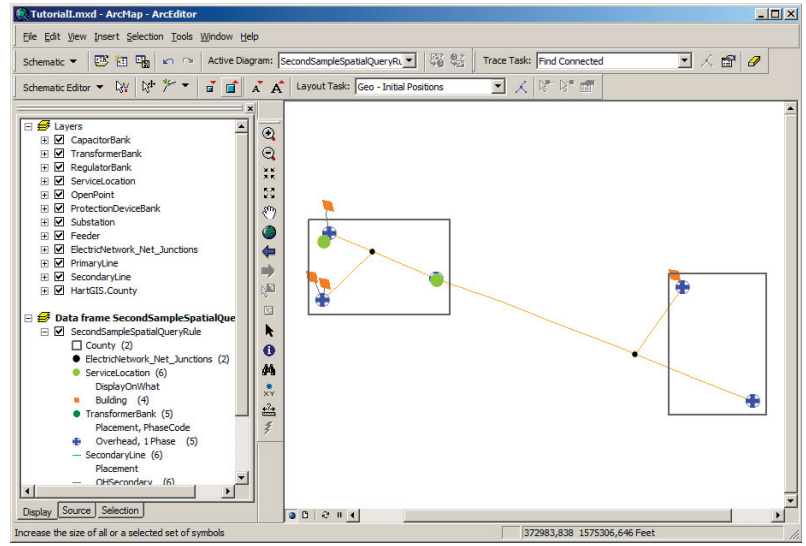
The following screen shot shows the County General tab content obtained after this configuration.




19. Save the schematic project definition and close Schematics Designer.

20. Start ArcMap and load the TutorialI.mxd file.
21. Update any diagram you have previously generated.
    It is displayed with one or several containers representing the county related to the TransformerBank elements contained in the diagram.

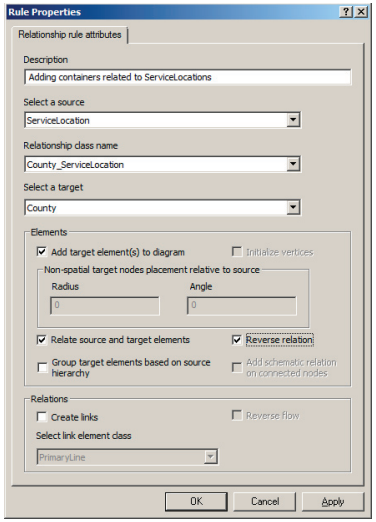
The following screen shot shows the update of the diagram obtained from the selected TransformerBank features that were used to exemplify the previous rule (see pages 23). Since the TransformerBank elements contained in this diagram were related to two counties, two containers are now displayed around these TransformerBank elements in the updated diagram.




In the ElecDemo geodatabase, there is a second relationship class, County_ServiceLocation, that specifies the county related to each ServiceLocation feature.

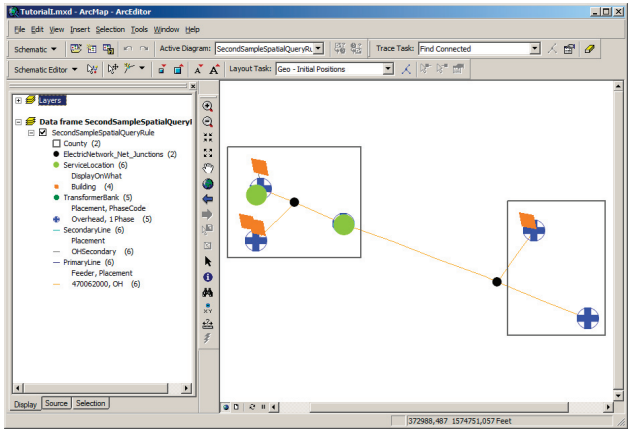
22. Repeat steps 1 to 13 to create a new Relationship rule that will relate each ServiceLocation schematic element to its county in the generated schematic diagrams.

The following screen shot shows the Relationship rule attributes tab content after these steps:




23. Click OK to close the tab.
24. Click Save.
25. Test the impact of this new definition in your generated diagrams by updating them in ArcMap.

Here is the result obtained in the previous sample diagram:

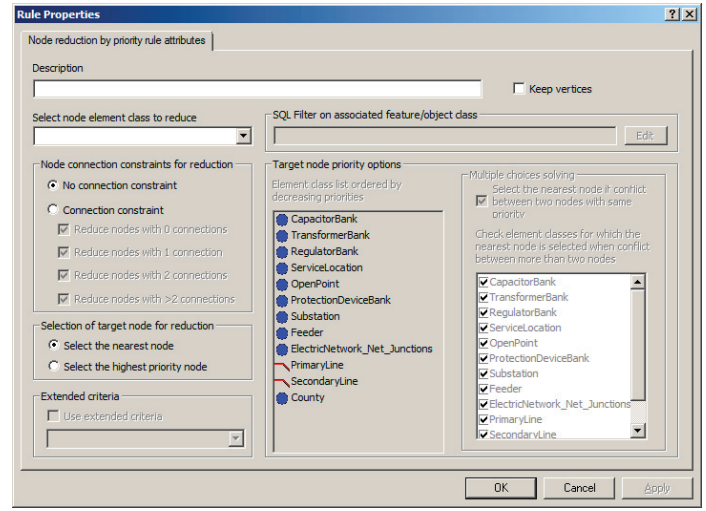



Since the ServiceLocation elements are now related to the County, the containers that represent the counties automatically enlarge to incorporate the ServiceLocation elements.

## Specifying a Node Reduction By Priority rule

In this section, you will specify a Node Reduction By Priority rule that will cause the reduction of ElectricNetwork_Net_Junctions elements in your generated schematic diagrams.
Only the ElectricNetwork_Net_Junctions nodes that have one connected link will remain; that is, the ElectricNetwork_Net_Junctions node with a degree of 1. The target node used to reconnect the incident links of the reduced nodes will be determined according to a list of element types in order of priority: Substation, Feeder, TransformerBank, ServiceLocation, ProtectionDeviceBank, and CapacitorBank.

1. Edit your schematic project within Designer.
2. Click the diagram type entry in the Designer tree and click the Rules tab from the right part of the Designer Editor window.
3. Click Add Rule at the bottom right corner of this tab.
   A new, empty item appears in the fourth line of the tab.
4. Click the Name field and choose Node Reduction By Priority from the drop-down list.
5. Click the Description field and click the ellipsis button that is displayed at the end of the rule item.
   The Node reduction by priority rule attributes tab opens.




6. Type a name for this rule on the Description text box (for example, “Reducing NetworkNet Junctions”).

7. Check the Keep vertices check box so the vertices displayed along the links are kept after nodes reduction and so a vertex is also displayed at the reduced node location.
8. Click the Select node element class to reduce drop-down arrow and click ElectricNetwork_Net_Junctions.
9. In the Node connection constraints for reduction area, check Connection constraint and uncheck the Reduce nodes with 1 connection check box so such nodes in the diagram are not reduced during the rule execution.

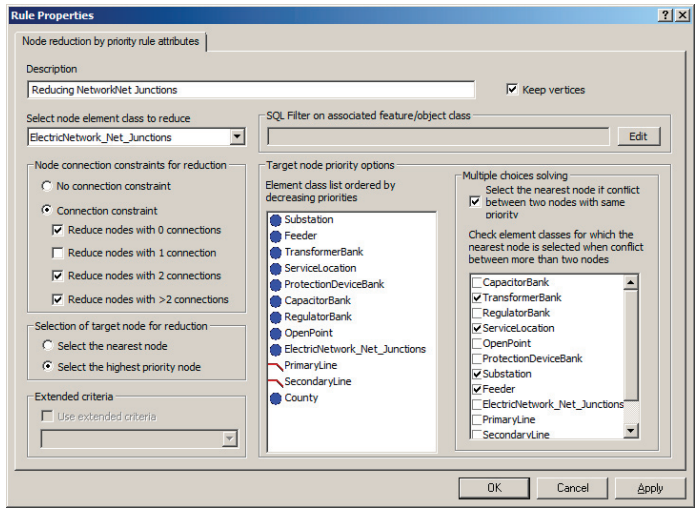
Next you will define how the topology of the network will be preserved after the specified node reducing.To accomplish this, you will specify the target nodes that will be used to reconnect the incident links related to the reduced nodes. By default, the Select the nearest node option is checked; that is, the nearest node connecting to the reduced node will become the target node. In this example, you will use the Select the highest priority node option so the target node depends on the element type of the nodes that connect the reduced ElectricNetwork_Net_Junctions node.

10. Check the Select the highest priority node check box.

With this option, for each node connected to the reduced node, the reduction rule will analyze the options specified in the Target node priority options section before detecting the right target node. The target element classes will be ordered from the highest to the lowest priority as follows: Substation, Feeder, TransformerBank, ServiceLocation, ProtectionDeviceBank, and CapacitorBank.

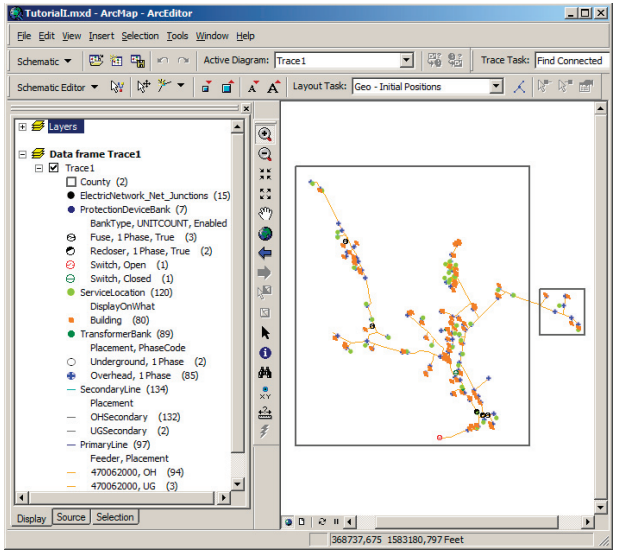
- Following this order, the Target node will be detected as follows: If among all the nodes that connect the reduced node, only one node belongs to the element type with the highest priority (that is, Substation), this node becomes the target node.
- If among all the nodes that connect the reduced node, no node belongs to the element type with the highest priority, the rule will check for a connected node belonging to the element type defined with the second-level priority (that is, Feeder), and so on.
- If among all the nodes that connect the reduced node, several nodes belong to the element type with the highest priority, the rule will detect the target node according to the options specified in the Multiple choices solving subsection. In this example, you will keep the Select the nearest node if conflict between two nodes with same priority option checked so the node nearest the reduced node systematically becomes the target node when the rule detects several possible target nodes.

11. In the Target node priority options area, click the Substation entry in the Element class list ordered by decreasing priorities list and drag the entry to the top of the list.
12. Repeat the operation so the Feeder element class is the second highest priority class, and so on, for the TransformerBank, ServiceLocation, ProtectionDeviceBank, and CapacitorBank element classes.
13. Examine the Multiple choices solving area. This area groups parameters used to decide whether the reduction will be done when there are several nodes of a shared element class candidate to become the target. Make sure that the Select the nearest node if conflict between two nodes with same priority check box is checked.
14. Uncheck all element classes in the element classes list except the Substation, Feeder, TransformerBank, and ServiceLocation entries. This means that if an ElectricNetwork_Net_Junctions node is selected to reduce and it connects several Substation element candidates to be the target node, the rule chooses the nearest Substation. The rule will also work in the same way to resolve conflicts for the Feeder, TransformerBank and ServiceLocation element classes.
    By unchecking the other element classes in the list, you ensure that ElectricNetwork_Net_Junctions nodes won't be reduced if any conflicts occur when determining target nodes for elements of the other classes. The tab content should is now displayed as follows:



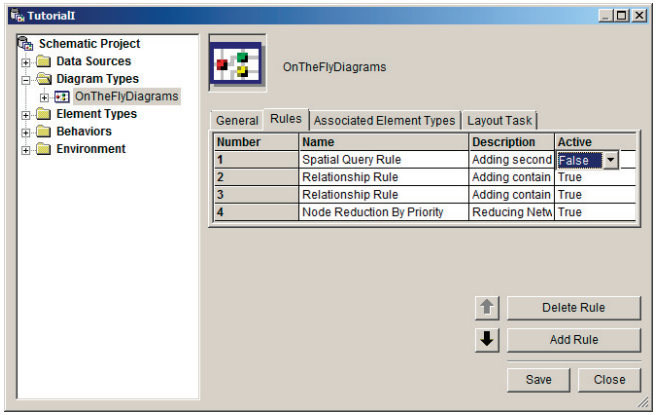
15. Click OK.
16. Save the schematic project definition.

You can test the impact of this new rule on one of your generated diagrams by updating. Here is the result obtained on the Selection1 sample diagram (see page 16).

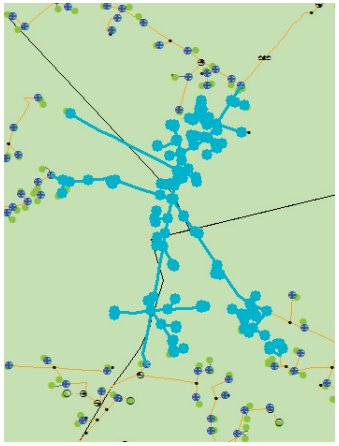



Among the 55 ElectricNetwork_Net_Junction elements in the diagram, only 15 of them are kept after the reduction rule execution—those that correspond to nodes with only one link connection.

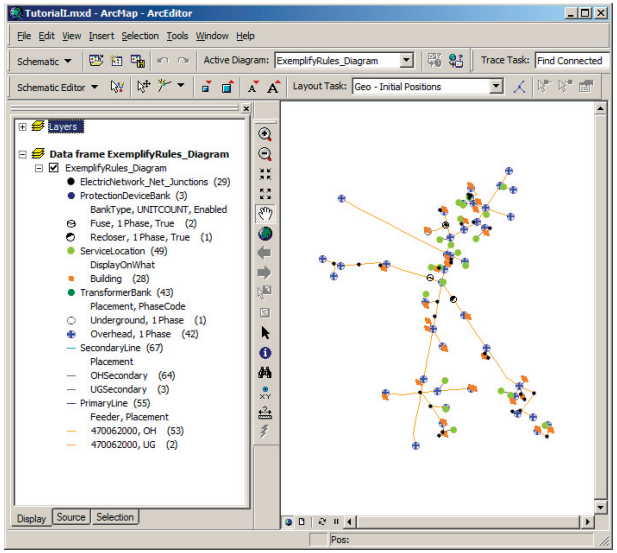
If you want to deactivate any rule defined for a schematic diagram type, you only need to click its Rules tab and click False in the Active field related to the desired rule.



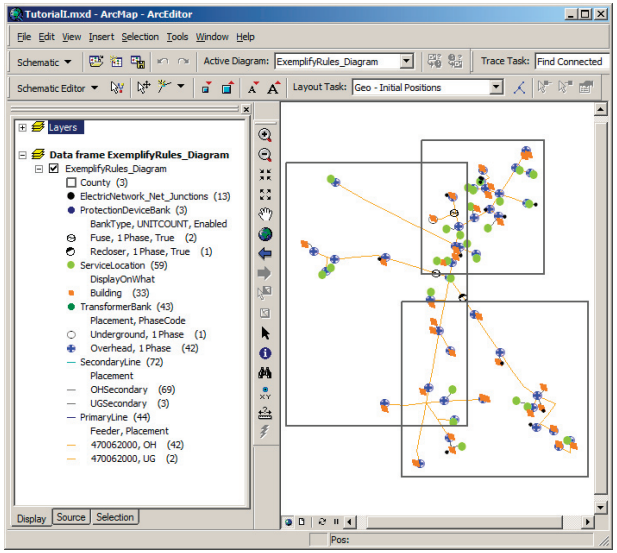

In this last example, you can analyze the diagrams obtained from a set of selected features located near the boundaries of three counties in the map.



The following diagram is obtained while all rules are deactivated during the generation. It displays only the schematic elements related to each selected feature in the map.




After having activated  all rules on the Rules tab and saved the schematic project definition, the resulting updated diagram appears as follows:

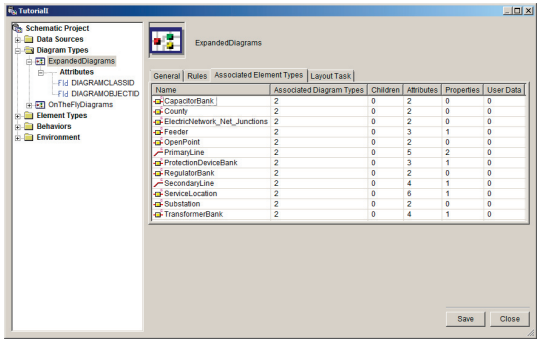



**Note:** In this example of configuration, the generation and update phases can take several minutes when the diagram is based on many TransformerBank features. As the Spatial Query rule execution detects the SecondaryLine neighbors of each TransformerBank by working directly on the feature class table, more time will be required to execute this rule depending on the number of TransformerBank features.

## Generating new diagrams with expanded links

In this section, you will create a new diagram type associated with the same element types as the OnTheFlyDiagrams diagram type created in exercise 2. This diagram type will be used to generate diagrams in which the PrimaryLine edges will be automatically expanded according to the PhaseCode value stored in the associated feature class table.

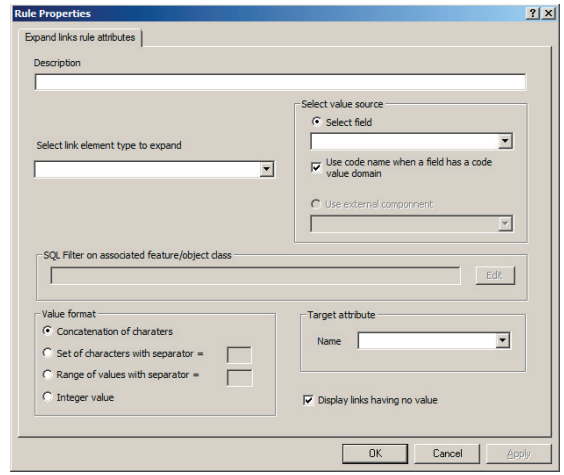
1. Close ArcMap.
2. Edit your TutorialI schematic project within Designer.
3. Right-click the main Diagram Types entry in the Designer tree and point to Create. The Create Diagram Type opens.
4. Type a name for your new diagram type (ExpandedDiagrams, for example) and click OK.
   The new diagram type entry is displayed in the Designer tree.
5. On the General tab, verify that the specified schematic builder is Standard Builder. Click this field and click the ellipsis button that is displayed. The Schematic Builder Properties dialog box opens. Check Initialize link vertices and click OK.
6. Click the Associated Element Types tab related to the newly created diagram type.
7. Right-click the background's tab and click Edit Associations.
8. The Element Type Associations dialog box opens. Click the Select All button at the bottom left corner so all the existing element types in the project are associated with this second diagram type and click OK.
   The Associated Element Types tab content related to the new diagram type now appears as follows:




9. Click Save.

You must now specify the rule that will be used to automatically expand the PrimaryLine links contained in diagrams implemented by this new diagram type in several links according to the PhaseCode field values stored in the associated PrimaryLines feature class.

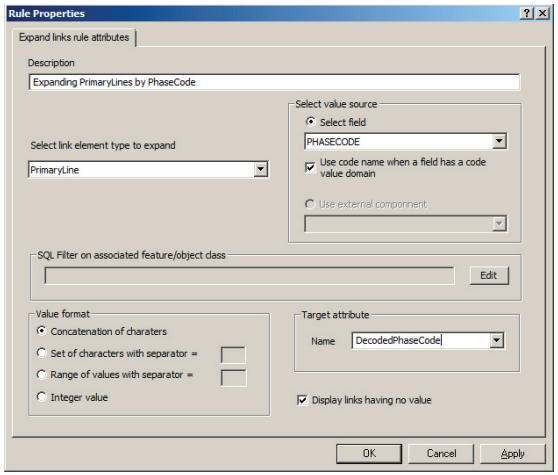
10. Click the Rules tab from the right part of the Designer Editor window.
11. Click Add Rule at the bottom right corner of this tab.
    A new, empty item appears in the fourth line of the tab.
12. Click the Name field and choose Expand Links from the drop-down list.
13. Click the Description field and click the ellipsis button at the end of the rule item.
    The Expand links rule attributes tab opens.



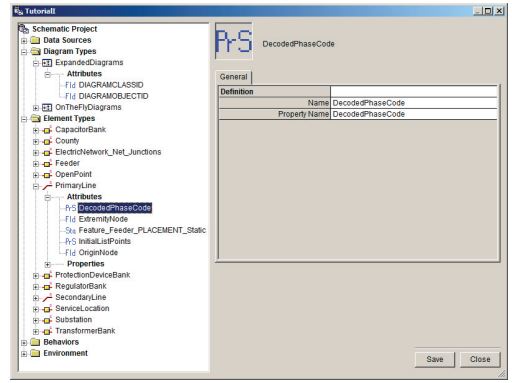

14. Type a name for this rule in the Description text box (for example, "Expanding PrimaryLines by PhaseCode").
15. Click the Select link element type to expand drop-down arrow and click PrimaryLine.
16. You must now specify the values that will determine how the original link will be expanded in several links. In this example, the values will come from the PHASECODE field. Select PHASECODE in the Select field drop-down list in the Select value source area.

The PHASECODE field value is encoded, so keep the Use code name when a field has a code value domain option checked.

17. Next, you have to specify the type of values that will be used to expand the links in the Value format area. The encoded values related to the PHASECODE field are A, B, C, AB, AC, ABC, and so forth. They correspond to a concatenation of characters. Therefore, keep the default Concatenation of characters option checked in the Value format area.
18. The Target attribute area has to do with how decoded values of expanded links are stored in the schematic database. Type a name for the new PropertySet attribute you want to be created for that storage ("DecodedPhaseCode", for example).
    The tab content should now appears as follows:

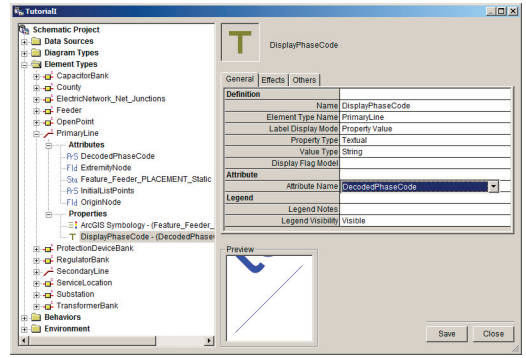



19. Click OK.
20. Expand the PrimaryLine element type entry in the Designer tree. The DecodedPhaseCode attribute has been automatically created. The value related to each link that will be expanded by that rule will be stored in this attribute during the rule execution.




These values can be used to label each expanded link. This can be easily done through a textual property.

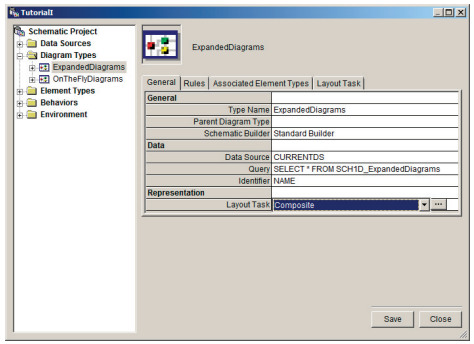
21. Right-click PrimaryLine in the Designer tree and click Create Property. The Create Property dialog box opens.
22. Type a name for the property ("DisplayPhaseCode", for example). Keep the default Textual option, the String value type checked. Click OK.
23. Click DecodedPhaseCode from the Attribute Name drop-down list on the General tab.



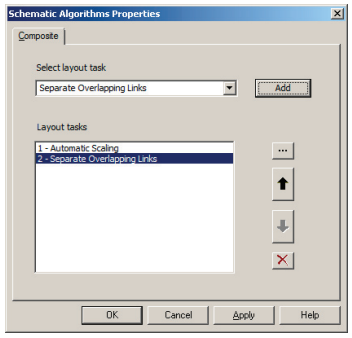

If you try to generate a diagram based on this new diagram type now, you will generate a diagram for which all PrimaryLine links whose PHASECODE values are ABC, AB, BC, and so on, will be automatically expanded but will overlap. For those links to be automatically separated, you can specify the Separate Overlapping Links algorithm to be activated at the diagram loading. Moreover, to keep the size of your symbols

appropriate in the generated diagram, it's better to combine this Separate Overlapping Links algorithm execution with the Automatic Scaling algorithm using a Composite algorithm. Follow these steps to specify such an algorithm execution:

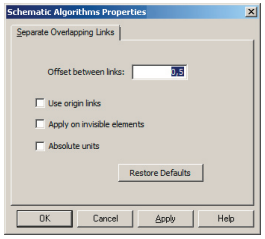
24. Select the entry related to the diagram type in the Designer tree (ExpandedDiagrams in this example).
25. Click the Layout Task drop-down arrow on the related General tab, and choose Composite.




26. Click the ellipsis button to specify the Composite algorithm parameters. The Schematic Algorithm Properties dialog box opens.
27. Click Automatic Scaling in the Select layout task drop-down list and click Add.
28. Click Separate Overlapping Links from the same list and click Add.



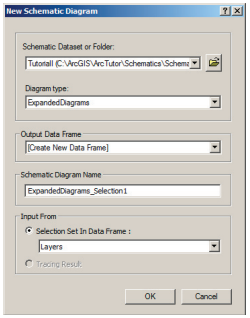

29. Click the ellipsis button while the second algorithm task is selected in the Layout tasks list to verify the properties set by default for the Separate Overlapping Links algorithm. The related Schematic Algorithms Properties dialog box appears as follows:



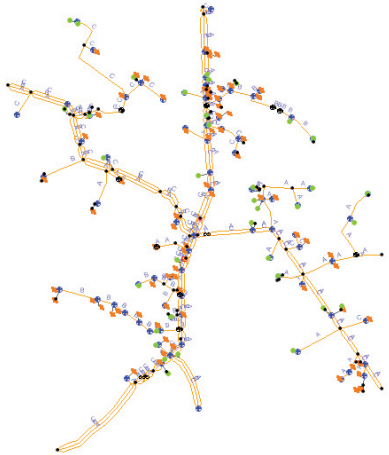

30. Click OK to close the dialog box.
31. Click Save, close Designer, then exit.

Now you can generate a diagram based on this second diagram type definition.

32. Start ArcMap and load your .mxd file.
33. Select an area in your map that contains PrimaryLine edges with PHASECODE field values equal to ABC, AB, BC, or AC.
34. Click Generate New Schematic Diagram. The New Schematic Diagram dialog box opens.
35. Click the browse button next to the Schematic Dataset or Folder drop-down list to choose the TutorialI schematic dataset in the TutorialI_SchematicDatabase geodatabase.
36. Click ExpandedDiagrams for the second diagram type you have just created from the Diagram type drop-down list.
37. Type a name for the schematic diagram that will be generated ("ExpandedDiagrams_Selection1", for example) in the Schematic Diagram Name text box.
38. By default, the Selection Set In Data Frame option is checked in the Input From area.



39. Click OK.

The new schematic diagram is generated. All the links related to a PHASECODE field values equal to ABC, AB, AC, or BC have been expanded, and the Separate Overlapping Links algorithm activated at the diagram loading has separated them.

Following is a sample diagram generated from some PrimaryLine features selected in the map.