



# User Manual

## for the Caltrans

# Pavement Management System

# PaveM

---

*Caltrans Implementation • March 2012*



AgileAssets Inc.  
Pavement Management System

Caltrans User Manual, March 2012.

©2012 AgileAssets Inc.

All rights reserved. All text and figures included in this publication are the exclusive property of AgileAssets Inc. for the sole use of its clients and employees. This material may not be copied, reproduced, or used in any way without the express written permission of AgileAssets Inc.

Information in this document is subject to change without notice and does not represent a commitment on the part of AgileAssets Inc.

This document may contain registered trademarks, trademarks, and service marks that are owned by their respective companies or organizations. AgileAssets Inc. disclaims any responsibility for specifying which marks are owned by which companies or organizations.

If you have any comments or suggestions regarding this document, please send them to:

AgileAssets Inc.  
3144 Bee Caves Road, Austin TX 78746  
Phone (512) 327-4200 • Fax (512) 328-7246  
[www.agileassets.com](http://www.agileassets.com)

## **Table of Contents**

<b>1. Welcome to AgileAssets Pavement Analyst®.....</b>	<b>1</b>
1.1. System Overview .....	1
1.2. Basic Operation .....	2
1.2.1. Step 1: Filling Data Tables and Preparing Calculated Results.....	2
1.2.2. Step 2: Creating the Performance and Network Master Files.....	3
1.2.3. Step 3, Part 1: Performance Analysis.....	3
1.2.4. Step 3, Part 2: Network Analysis .....	3
<b>2. Getting Started.....</b>	<b>4</b>
2.1. Conventions Used in this Manual .....	4
2.2. Browser Information .....	4
2.3. Log On Procedure.....	4
2.4. Description of the Dashboard .....	6
2.4.1. Left Gutter Icons .....	6
2.4.2. Quick Links.....	7
2.4.3. Location Reference.....	10
2.4.4. Color Schema.....	10
2.4.5. Messages .....	10
2.4.6. Design Mode.....	11
2.5. Forgotten Password Procedure.....	11
2.6. Change Password .....	12
2.7. Common Commands .....	12
2.7.1. Change Graph Properties .....	12
2.7.2. Check Spelling.....	17
2.7.3. Color Settings .....	19
2.7.4. Delete.....	19
2.7.5. Export Data .....	19
2.7.6. Filter.....	20
2.7.7. Filter By This Value .....	24
2.7.8. Find.....	24
2.7.9. Insert.....	26
2.7.10. Insert Like .....	26
2.7.11. Massive Update.....	26
2.7.12. Maximize Data Pane.....	27
2.7.13. Selection Commands .....	27
2.7.14. Set Bookmark.....	27

2.7.15.	Show on Bing Map.....	27
2.7.16.	Show on Google Map .....	28
2.7.17.	Show on Map.....	29
2.7.18.	Show Report .....	29
2.7.19.	Show Videolog .....	29
2.7.20.	Sort.....	29
2.7.21.	Update Target Table .....	30
2.7.22.	Work with Attachments.....	31
2.8.	Floating Map Window .....	33
2.8.1.	Layers Pane.....	34
2.8.2.	Map Pane .....	35
2.8.3.	Layer Properties Dialog Box.....	37
2.8.4.	Layer Selection Dialog Box.....	41
2.8.5.	Map Selection Dialog Box.....	42
2.9.	Overview of User-controlled Tables, Windows, and Calculations .....	42
2.9.1.	Data View of Location-referenced Data.....	43
2.9.2.	Graph View of Location-referenced Data.....	44
2.9.3.	List Data Windows .....	46
2.9.4.	Table-specific Windows .....	46
2.9.5.	"As-of Date" Windows .....	46
2.9.6.	Record Selection.....	46
2.10.	Software Use via the Keyboard.....	47
2.10.1.	Left Gutter Navigation.....	47
2.10.2.	Menu Navigation .....	48
2.10.3.	Navigation Within a Window .....	48
2.10.4.	Tab Order.....	50
<b>3.</b>	<b>Roadway Module .....</b>	<b>51</b>
3.1.	Setup PMS Columns .....	51
3.2.	Calculated Expressions .....	53
3.2.1.	Description of the Available Parameters .....	53
3.2.2.	Expressions Pane .....	54
3.2.3.	SQL Pane.....	54
3.2.4.	Columns Pane .....	54
3.3.	Material Code ID.....	54
3.4.	Work Code.....	55
3.4.1.	Description of the Work Codes Window .....	56
3.4.2.	How to Define a Work Code .....	56

3.5.	Budget Categories.....	57
3.6.	Inventory .....	57
3.6.1.	Setup Status .....	57
3.6.2.	Setup Class Code .....	58
3.6.3.	Inventory Window .....	59
3.7.	Standard Sections.....	59
3.7.1.	Standard Sections Window Description.....	59
3.7.2.	How to Create a Standard Section.....	60
3.8.	Current Pavement Management Section Data.....	60
3.9.	Construction History .....	61
3.9.1.	Description of the Construction History Window .....	62
3.9.2.	How to Insert a New Record .....	63
3.9.3.	How to Add a New Layer .....	63
3.9.4.	How to Insert a Standard Section .....	64
3.10.	Pavement Structure .....	64
3.11.	Pavement (Profile/Cross-section) .....	64
3.11.1.	Route Pane .....	64
3.11.2.	Cross-Section Pane.....	65
3.11.3.	Pavement and Layer Details Pane.....	65
3.11.4.	Section Profile Graph .....	65
3.11.5.	Cross-section Graph .....	65
3.11.6.	How to Display a Section Profile Graph.....	66
3.11.7.	How to Display a Cross-section Graph .....	66
3.12.	Structure Defaults .....	66
3.13.	Conditions Menu .....	66
3.14.	Traffic.....	67
3.15.	Performance Master.....	67
3.16.	Performance Models.....	68
3.16.1.	Decision Tree Pane.....	69
3.16.2.	Attributes Pane.....	70
3.16.3.	Models Pane .....	71
3.17.	Models .....	71
3.17.1.	Types of Models Supported .....	71
3.17.2.	Description of the Right-click Shortcut Menu Commands .....	72
3.18.	Default Model Structure .....	73

3.18.1.	Description of the Window .....	73
3.18.2.	How to Add a Node .....	74
3.18.3.	How to Set Decision Parameters.....	74
3.18.4.	How to Delete a Node.....	75
3.19.	Setup Condition State.....	75
3.20.	Section Performance .....	76
3.20.1.	Description of the Sections Pane.....	76
3.20.2.	Description of the Graph Pane.....	77
3.21.	Upper Level Decision Tree .....	77
3.22.	Treatments .....	78
3.22.1.	Description of the Treatments Window .....	78
3.22.2.	How to Edit a Treatment .....	82
3.22.3.	How to Configure Improvements .....	83
3.23.	Decision Trees .....	84
3.23.1.	Decision Trees Window Description.....	84
3.23.2.	How to Create a New Decision Tree Set.....	86
3.23.3.	How to Create a New Lower-level Decision Tree .....	86
3.23.4.	How to Modify a Lower-level Decision Tree.....	88
3.23.5.	How to Assign a Lower-level Decision Tree.....	88
3.23.6.	How to Disassociate a Lower-level Decision Tree .....	88
3.23.7.	How to Delete a Lower-level Decision Tree.....	89
3.24.	MWP Project Status .....	89
3.25.	Setup RSL Categories .....	89
3.26.	Work Plan Type .....	90
3.26.1.	Work Plan Type Window Description.....	90
3.26.2.	How to Create a Work Plan Type.....	90
3.27.	Network Master .....	90
3.28.	Section Current Needs .....	91
3.29.	Optimization Analysis .....	92
3.29.1.	Optimization Analysis Window Description.....	92
3.29.2.	How 'Number of Nodes' Works .....	98
3.29.3.	How 'Project Cost Threshold' Works .....	98
3.29.4.	How to Define and Run an Optimization Analysis .....	100
3.29.5.	How to Include Projects from a Work Plan.....	102
3.30.	Detailed Optimization Results .....	102

3.31.	Setup Constraint Partitions.....	103
3.31.1.	Description of the Window .....	103
3.31.2.	How to Create Constraint Partitions.....	104
3.32.	Work Plan Data .....	104
3.32.1.	How to Edit a Work Plan.....	105
3.32.2.	Example of Editing a Work Plan.....	106
3.32.3.	How to Copy Line Items from a Scenario Analysis .....	106
3.33.	Strategy Configuration.....	107
3.34.	Section Strategies .....	107
3.34.1.	Road Sections Pane .....	107
3.34.2.	Strategies Assigned to Road Section Pane.....	107
3.34.3.	Strategy Details Pane.....	108
3.35.	Road View.....	108
3.35.1.	Description of the Panes in the Road View Window.....	108
3.35.2.	Description of the Right-click Shortcut Menu Commands .....	109
3.36.	GIS Data Explorer .....	110
<b>4.</b>	<b>System Module .....</b>	<b>111</b>
4.1.	Access Level Settings.....	111
4.1.1.	Description of the Access Level Settings Window .....	112
4.1.2.	How to Create an Access Profile.....	113
4.2.	Actions Rights .....	113
4.2.1.	Description of the Actions Rights Window.....	114
4.2.2.	How to Set Right-to-See Values .....	115
4.3.	Columns .....	115
4.3.1.	Description of the Columns Window .....	116
4.3.2.	Split Rules .....	119
4.3.3.	Precautions in Setting Access Levels.....	119
4.3.4.	Setting Right-to-See and Right-to-Edit Values.....	120
4.4.	Menus .....	121
4.4.1.	Modules Pane .....	121
4.4.2.	Module Menus Pane.....	121
4.4.3.	How to Change a Menu Name .....	121
4.4.4.	How to Delete a Window .....	122
4.5.	Setup Popup Window Titles .....	122
4.6.	Setup Window Titles .....	123

4.7.	System Parameters.....	124
4.8.	Tables.....	128
4.8.1.	Description of the Tables Window .....	128
4.8.2.	Description of the Import Config Window .....	133
4.8.3.	How to Create a Table with Location Data.....	142
4.8.4.	How to Create a List Table .....	142
4.8.5.	How to Create an Inventory Table .....	143
4.8.6.	How to Create a Generic Table .....	147
4.8.7.	How to Add a New Column to a Table .....	148
4.8.8.	How to Add an Existing Column to a Table.....	148
4.8.9.	How to Create an Index for the Table.....	149
4.8.10.	How to Make a Data or List Window .....	149
4.8.11.	How to Make a Graph Window .....	150
4.8.12.	How to Make a Popup Window .....	150
4.8.13.	How to Make an "As-of Date" Window.....	151
4.8.14.	How to Modify Column Definitions .....	152
4.8.15.	How to Change the Order of Columns.....	152
4.8.16.	How to Update When Source Data Changes.....	153
4.9.	Text Resources.....	153
4.10.	User Names and Access .....	153
4.10.1.	User Names and Access Window Description .....	154
4.10.2.	How to Add a New User Without Copying .....	156
4.10.3.	How to Add a New User by Copying.....	157
4.10.4.	How to Reset a Password .....	158
4.11.	Version Information .....	158
4.12.	Views .....	158
4.12.1.	Description of the Views Window .....	158
4.12.2.	How to Create a View .....	160
4.12.3.	How to Make a View Window .....	161
4.13.	Copy Window .....	162
4.14.	Administrative Units .....	162
4.14.1.	Description of the Admin. Units Window .....	163
4.14.2.	How to Add a New Admin. Unit .....	163
4.15.	Asset Type.....	164
4.15.1.	Description of the Asset Type Window .....	164
4.15.2.	How to Create an Asset Type.....	165
4.15.3.	How to Create PM Triggers.....	166
4.16.	Setup File System .....	167

4.17.	Module .....	167
4.18.	Units.....	168
4.19.	Setup LRM.....	168
4.19.1.	Description of the Setup LRM Window .....	169
4.19.2.	Alternate LRM Windows.....	170
4.19.3.	How to Change Reference Systems.....	170
4.19.4.	Example of Changing the Reference System.....	171
4.20.	Concurrent Location.....	172
4.21.	Manage GIS Data Sources.....	173
4.21.1.	Description of the Manage GIS Data Sources Columns .....	173
4.21.2.	Description of the Right-click Shortcut Menu.....	174
4.22.	Manage GIS Layers.....	175
4.23.	GIS Images .....	175
4.24.	GIS Coordinate Reference System.....	175
4.25.	Network Lanes .....	176
4.26.	Routes List (Setup Network Lines).....	177
4.27.	Network Transactions .....	177
4.27.1.	Network Transactions Window Description .....	178
4.27.2.	How to Run a Network Transaction .....	181
4.28.	Setup Dynamic Aggregation .....	181
4.28.1.	Main Pane .....	182
4.28.2.	Details Pane .....	182
4.28.3.	Aggregation Rules .....	183
4.28.4.	Description of the Right-click Shortcut Menu Commands .....	183
4.29.	Dynamic Segmentation .....	184
4.29.1.	Route Selection Pane .....	185
4.29.2.	Segment Boundaries Pane .....	185
4.29.3.	Columns to Graph Pane.....	186
4.29.4.	Graph Pane .....	186
4.30.	Setup Update from Geometry .....	186
4.31.	System Archive .....	186
4.31.1.	Description of the Window .....	187
4.31.2.	How to Perform an Initial Archive.....	188
4.31.3.	How to Perform Subsequent Archives .....	188
4.31.4.	How to Retrieve Archived Data .....	189

4.31.5. How to Create a New Archive Set.....	189
4.32. Groovy Scripts.....	190
4.32.1. Description of the Groovy Scripts Window .....	190
4.32.2. Description of the Groovy Script Dialog Box.....	191
4.32.3. Parameters for Each Script Type.....	191
4.33. System Job Executables .....	194
4.34. System Job Schedules .....	194
4.34.1. Jobs Tab .....	195
4.34.2. Log Tab.....	196
4.34.3. CRON Expression Editor .....	196
4.35. Import.....	199
4.35.1. Description of the Import Window .....	200
4.35.2. Date-type Column Formats .....	204
4.35.3. How to Set Up an Import Process .....	206
4.35.4. How to Import Data with the Import Window .....	209
4.36. Import Runner.....	210
4.36.1. Description of the Import Window .....	210
4.36.2. How to Import Data with the Import Runner Window .....	212
4.37. System Logs.....	213
4.38. Notifications .....	214
4.38.1. System Alerts/News Pane.....	214
4.38.2. Target Users Pane .....	215
4.39. Notification Types.....	215
4.39.1. Description of the Columns.....	215
4.39.2. Description of the Right-click Shortcut Menu Commands .....	216
4.40. Set Substitutes for Notifications.....	216
4.41. Threads.....	217
<b>5. Reports.....</b>	<b>218</b>
5.1. Reports Management Window .....	218
5.1.1. Reports Pane.....	219
5.1.2. Report Details Pane .....	222
5.1.3. Share Report Window .....	223
5.1.4. Schedule Email Window.....	223
5.2. How to Insert a New Report .....	226

5.3.	How to Display a Report .....	227
5.4.	How to Create a Share Group .....	227
5.4.1.	Create a Share Group by Selecting Security Profiles and Administrative Units.....	228
5.4.2.	How to Create a Share Group by Directly Inserting Users .....	229
5.5.	How to Share a Report .....	229
5.6.	How to Stop Sharing a Report .....	230
5.7.	How to Make a Report Public .....	230
5.8.	How to Make a Report Private.....	231
5.9.	Dashboard Reports .....	231
5.9.1.	How to Add a Report to the Dashboard.....	231
5.9.2.	How to Refresh a Dashboard Report .....	232
5.9.3.	How to Remove a Report from the Dashboard .....	232
5.10.	Tabular Reports Configuration Dialog Box.....	232
5.10.1.	List of Columns Pane .....	234
5.10.2.	Order By Pane .....	235
5.11.	Graph Reports Configuration Dialog Box .....	236
5.11.1.	Graph Reports Setup Tab.....	236
5.11.2.	Graph Tab .....	243
5.11.3.	Data Tab .....	243
5.12.	GIS Reports Configuration Dialog Box.....	244
5.12.1.	GIS Reports Setup Tab .....	244
5.12.2.	Report Tab (GIS Reports).....	245
5.13.	JasperReports.....	245
5.13.1.	How to Add a Report Without Sub-reports.....	246
5.13.2.	How to Add a Report With Sub-reports.....	246
<b>6.</b>	<b>Information for Administrators.....</b>	<b>248</b>
6.1.	Details of User-controlled Tables and Windows .....	248
6.1.1.	How to Tell if a Table is User-modifiable.....	248
6.1.2.	How to Delete a Table.....	249
6.1.3.	How to Tell if a Window is User-modifiable .....	249
6.1.4.	How to Delete a Window .....	249
6.2.	Adjustment of Window Layouts .....	249
6.2.1.	How to Activate Design Mode.....	250
6.2.2.	User Control Properties Dialog Box .....	250

6.2.3.	How to Change the Size of a Pane.....	260
6.2.4.	How to Change the Size of a Column.....	261
6.2.5.	How to Add a Column to a Pane.....	261
6.2.6.	How to Delete a Column from a Pane .....	261
6.2.7.	How to Hide Columns .....	262
6.2.8.	How to Change a Pane's Format.....	262
6.3.	Record History Column.....	263
6.4.	Update Source and Where Columns.....	263

## 1. Welcome to AgileAssets Pavement Analyst®

AgileAssets Pavement Analyst® is designed to assist decision-makers in the process of managing a network of pavements. It stores, retrieves, and processes pavement-related condition and inventory data, allowing you to analyze the current condition, future performance, and expected monetary needs of the pavement network. The flexible pavement management environment makes AgileAssets Pavement Analyst a powerful pavement management tool.

### 1.1. System Overview

AgileAssets Pavement Analyst is a fully functional pavement management system designed to help you, the decision-maker, effectively manage a pavement network. This section presents an overview of the main features of the system and the layout of the software. The system consists of three types of pavement management functions:

- Set-up functions — The functions contained in the set-up menu establishes parameters for the way the software calculates or uses raw data contained in the database. These parameters are defined by your agency to control the way data is combined, calculated, and used for the analysis functions contained in the Analysis module.
- Database access — The Database menu consists of a fully functional database management system that contains all the pavement management data collected by your agency. This data is related to pavement inventory, pavement condition and performance, pavement structures and maintenance, rehabilitation/reconstruction activities, traffic and other data configured for your agency.
- Analysis functions — The Analysis menu contains the functions used to analyze pavement performance as well as estimate current and future pavement work needs. The analysis portion of the software uses the parameters specified in the Set-up portion and the data contained in the Database portion to make these estimates.

The system uses three basic types of data for pavement management:

- The first data type, Inventory, Class, and Management, defines the basic attributes of the pavement network. This data defines the extent of the network, its location referencing, and other attributes of each section in the network. This allows the software to use information such as functional classifications, traffic levels, environmental zones, and other information that may be useful for pavement management analyses.
- The second data type, Construction History, defines the pavement structures that exist on each pavement section. Using this information, the software can analyze performance and assign appropriate maintenance and rehabilitation treatments for planning and network analysis.
- The last data type, Condition, defines the pavement condition of your network. The software uses pavement condition for performance modeling, reporting, decision-making, and prioritization of candidate projects based upon the decision-making parameters.

The system combines these three data types into master files for analysis within the system:

- The first master file — the Network Master File — contains the most current information from all three data types for every section in the network. This file is the starting point for future condition prediction and network planning analyses within the system. You may also produce reports with current information from this file.

- The second master file — the Performance Master File — contains a historical record of pavement network information. The performance master file is primarily used for performance analysis of pavements. Using historical information, the software is able to estimate deterioration equations for the pavements in the network, which allows you to predict future pavement condition for planning analysis. In addition, the performance master file can be used to produce reports showing historical trends in condition, traffic, or other information as needed.

## 1.2. Basic Operation

Using AgileAssets Pavement Analyst involves three major steps:

1. Filling raw data tables with data collected from the pavement network. These tables are filled by direct data entry, using import routine(s), or a combination of these two. The system contains utilities for converting this data into calculated results that can be analyzed by the system. These calculated results are displayed in the distress and performance indices windows, pavement structure tables and graphs, and the traffic window. The system includes windows (such as the Pavement Structure window) and functions to view this data in convenient formats.
2. Processing raw data and storing results in the performance and network master files and other derived data tables. This step uses the performance set-up parameters combined with raw data and the database calculated results from the first step to create two master files: one for performance analysis and the other for network analysis.
3. Analyze the data contained in the performance and network master files to produce reports and document the analyses.

### 1.2.1. Step 1: Filling Data Tables and Preparing Calculated Results

Once raw data is recorded in the database, you convert it (according to database set-up parameters) to create calculated results that can be used later in reporting as well as performance and network analysis. The Database menu typically provides at least the following calculated results:

- Distress Survey — Pavement performance per distress survey interval according to the distress index score. A window is available that combines all major pavement types as well as roughness (if kept separately from the standard distress survey data).
- Pavement Structure — Pavement construction history along a pavement management section and over time to the last reconstruction. The results are shown in tabular form (pavement structure table) and in graphical form (pavement structure graph).

The following configuration steps are taken for calculated tables:

- Define in which Oracle table each calculated item will reside. As a prerequisite for performing network analysis, define the Network Master table contents, and as a prerequisite for performing performance analysis, define the Performance Master table contents.
- Define for each calculated item the calculation/aggregation formula and the order in which calculations are to take place.
- Define the method by which section limits are produced.
- Define the data review window for each table and where it'll be located in the system.
- Define whether the data in each table can be graphed down the road and will be available in the Road View window.

### **1.2.2. Step 2: Creating the Performance and Network Master Files**

Once you have filled the database with the latest available raw data and calculated the distress survey, you are now ready to begin making the performance and network master files. These files are composed of fields that are calculated using performance set-up parameters, raw data, and data aggregation techniques as described in the previous step and in the technical manual. The first step in making the performance and network master files is to specify or adjust the parameters in the construction set-up menus.

The set-up parameters for performance analysis are:

- Road structure categories define the general categories of pavement structures in the network according to the maintenance and rehabilitation actions.
- Work codes define the work for a construction contract and assign a road structure category to the work.

### **1.2.3. Step 3, Part 1: Performance Analysis**

The first step in performing analyses is to create the latest year's performance master file and re-calculate the performance master file as needed. Once the performance master file is filled, performance models can be reviewed and adjusted from the historical data supplemented with the latest year's information.

Models may be developed using two methods:

- The first method, building manual pavement performance models, allows you to define expected pavement performance curves using your engineering judgment and experience.
- The second method is to create pavement performance models in the performance analysis module based on historical data contained in the performance master file. Once models are developed for all possible groupings, each model can be reviewed individually and adjusted as necessary.

Models may be assigned by groups of sections or by individual sections.

After performance models have been created in the performance analysis module, you can view the predicted pavement management performance section assuming that no treatments were performed on the section. This estimation of a section's performance is available from the section performance menu selection under performance analysis. From the performance window, you can graphically display individual pavement section performance.

### **1.2.4. Step 3, Part 2: Network Analysis**

After you have completed developing the performance master and developing/reviewing pavement performance models, you are ready to perform network analysis. Network analysis is controlled through the network set-up menu where your agency establishes decision trees and available treatments. It also controlled via the Utilities menu in the System module, where your agency establishes priority formulae and sets parameters that affect the roadway simulation due to future treatment. The network analysis module allows you to run various scenarios to investigate the impact of different budget levels on your network condition or to estimate budgets necessary to attain a given condition level.

## 2. Getting Started

This section shows you how to log on and provides general information about the features of the application.

### 2.1. Conventions Used in this Manual

To assist you in differentiating types of information or actions required of you, this tutorial uses the following typographical conventions:

- > The greater-than sign separates menu levels (for example, File > Open).
- CAPITALS Upper case text indicates the names of keys on the keyboard (for example, SHIFT or CTRL or ALT+F4).
- + When used between keyboard keys, the plus sign indicates keys that must be pressed simultaneously to perform a command (for example, ALT+F4).
- Bold** Commands are shown in a heavy bold font (for example, **Retrieve**).
- Tahoma The Tahoma font is used for text that appears in a window and statements that are to be typed in exactly as shown.
- Title Caps Title Capitalization often indicates a specific data element type (for example, Task, Service Request, Work Function).

### 2.2. Browser Information

AgileAssets software is web-based. It may be accessed from any browser (Internet Explorer, Mozilla Firefox, Google Chrome, etc.). However, some minor differences in the appearance and behavior may be noted depending on the browser you use. For example, when the focus is on a field in Internet Explorer, the text in the field is highlighted — but in Firefox and Chrome a dotted rectangle appears around the text.

This manual assumes that the Internet Explorer browser is utilized. The descriptions of appearance and behavior of the application in this manual will therefore conform to the conventions of this browser.

### 2.3. Log On Procedure

To launch and log on to the application, you need the following information:

- The URL (web address) where the system resides;
- Your user ID;
- Your user password;
- The administrative unit that you want to use for this session; and
- The security profile you want to use for this session.

Once you have the information needed to log on, follow these steps to log on to the system:

1. Launch your Internet browser.
2. In the address window of the browser window, type the URL for where the system resides and press Enter. (Note: If this is the first time you are accessing the system, you should save this URL as a "favorite" to facilitate return visits.) After the Enter key is pressed, the browser window will show the log on window for the system. An example of this window is shown below.



User ID

User Password  [Forgot your password?](#)

**Login**

AgileAssets Management System [PaveM] Version 6.8 Build 13512

3. Type your user ID and password in the appropriate fields. Passwords are case-sensitive. (If you have previously logged on to the system, the system will remember your user ID and so you will only need to enter your password.)

**NOTE**

If you forget your password, click the [Forgot Your Password?](#) link. See Section 2.5 on page 11 for more information.

4. Click **Login**. The system checks the information you entered.

If the User ID and/or password are not recognized, the system will display a message telling you of this. Repeat step 3 to continue. Note that you are only allowed five attempts to log on. After the last unsuccessful attempt, you will be locked out of the system. You must contact the System Administrator to be allowed to proceed.

If the User ID and password are recognized and you are assigned to only one administrative unit and one security profile, the system will automatically use these assignments and will proceed to log you on to the application.

If the User ID and password are recognized and you are assigned to more than one administrative unit and/or more than one security profile, the system displays a window to select your administrative unit and security profile. An example of this window is shown below.

Department

Security Profile

**Submit**

AgileAssets Management System [PaveM] Version 6.8 Build 13512

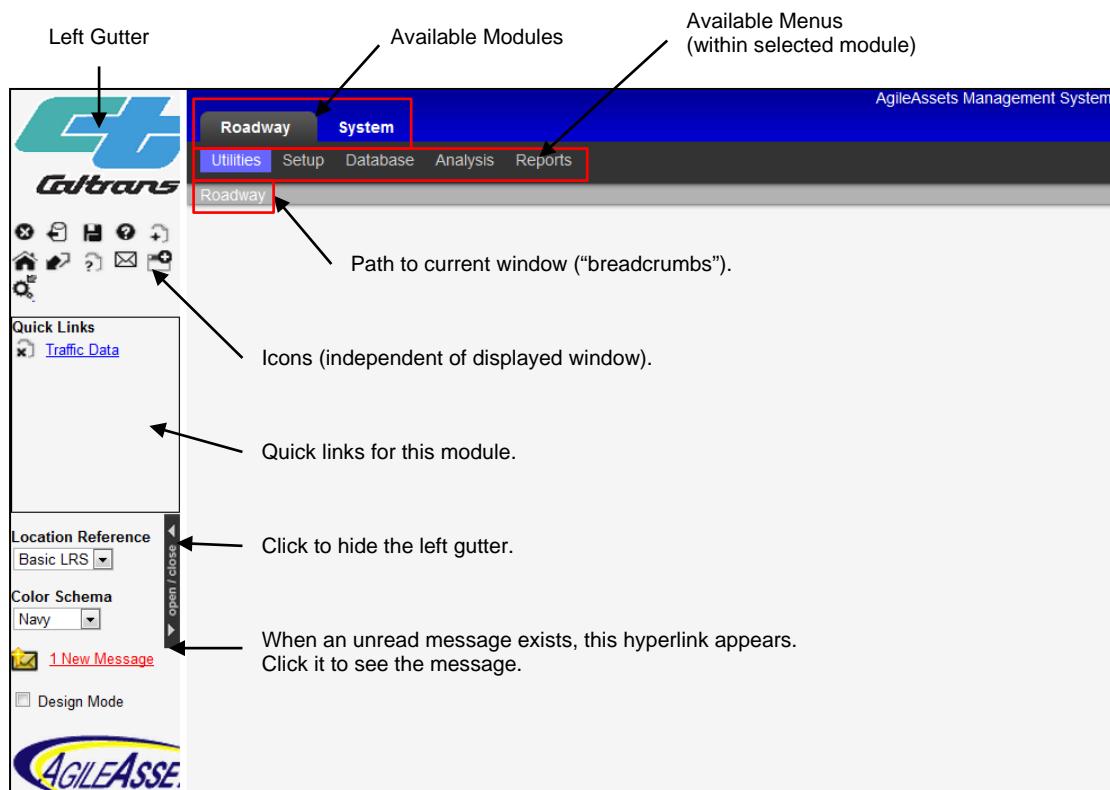
The system completes the fields with the administrative unit and security profile you used the last time you logged on. If the displayed selections are correct, proceed to the

next step. If you need to change a selection, click the down arrow to display the list of choices, locate the one you wish to use, and then click that choice.

Note: The selection of an administrative unit is important. At various points in the system, the phrase "use current" appears. The current unit is the unit you selected when you logged on. Furthermore, some windows only show data for one administrative unit and so you may not see the data you want unless you select the correct administrative unit. Also note that you may select a different administrative unit once logged on by using the  icon that appears in the left gutter.

5. Click **Submit**. You are now fully logged onto the system. The system displays the dashboard of the module you last used in the PaveM system. A typical dashboard is shown in the example on the following page.

Note, however, that your security profile determines what is displayed. For example, the dashboard shown on the next page shows all modules and windows. This could be what a person with the System Administrator security profile would see. On the other hand, a person with a Pavement Engineer security profile may only see the Roadway module – and that module would not have any setup windows.



## 2.4. Description of the Dashboard

A typical dashboard is shown above. The various elements that appear in the dashboard are described in the following sections.

### 2.4.1. Left Gutter Icons

The left gutter, which may be viewed or hidden by clicking the black bar at the right edge, provides the icons shown in the table below.

- |   |  |
|---|--|
|  | Exit – This icon closes the application. |
|---|--|

-  Retrieve – This icon retrieves the latest data for the displayed window from the database. Note: This will overwrite any new, unsaved data (meaning that the unsaved data will be lost).
-  Save – This icon stores the new data in the window in the database.
-  Help – This icon displays Help information for the displayed window.
-  Add Quick Link – This icon adds a link to the displayed window in the left gutter as well as the Quick Links group selected in the Manage Quick Links window. (Quick Links allow you to more easily access a window. See Section 2.4.2 on page 7 for more information on Quick Links.)
-  Remove Quick Link – This icon appears beside each entry in the Quick Links list, and is used to remove an entry from the list as well as the Manage Quick Links window.
-  Home – This icon displays the Home window for the selected module. (The Home window is typically a module's dashboard.)
-  Administrative Unit Selection – This icon displays the window for selecting an administrative unit and a security profile. This allows you to select a different administrative unit and/or security profile without logging off and then logging back on.
-  Manage Window Links – This icon displays a new window in which you may configure Quick Link groups (which in turn determine what Quick Links are shown in the left gutter) as well as text that is displayed when you hold the cursor over a Quick Link hyperlink. See Section 2.4.2 on page 7 for more information.
-  Send Email – This icon allows you to send an email that contains a link to the displayed window. When you click this icon, the system displays a dialog box with the link. You may then use the dialog box to compose your message and who will receive the email.
-  Create New Session – This icon allows you to start a new, independent session without ending the current session. After clicking the icon, the system displays a new tab in your browser that shows the dashboard of the current module. You may then alternate between the tabs to see and modify different windows in the same application.
-  Reset User Window Settings – This icon resets any customized window settings for the current window to their default settings.

#### **2.4.2. Quick Links**

##### **NOTE**

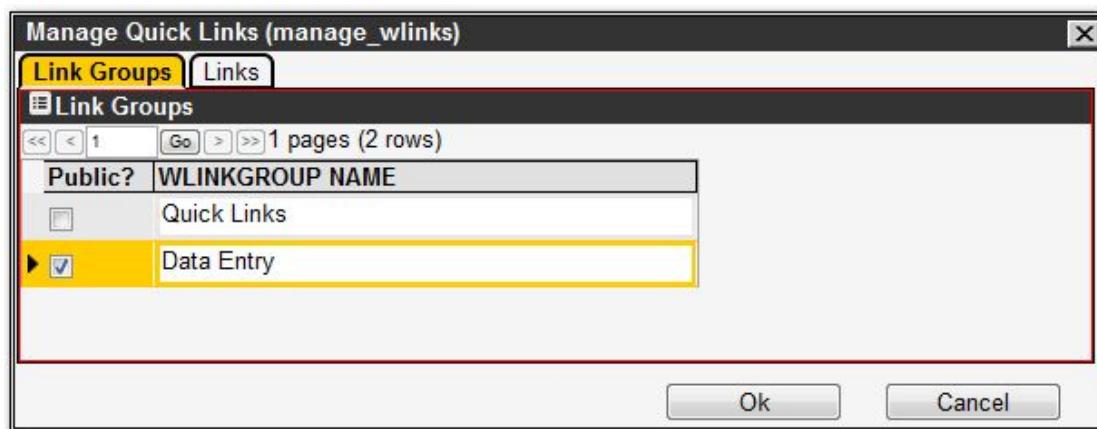
When you create a Quick Link, it is added to the Quick Link Group selected in the Manage Quick Links window. Therefore, if you are using multiple Quick Link Groups, first select the group in which you want the link to appear and then create the link to the window. (If you have not yet created links for the selected group, you will not know what group is selected except by displaying the Manage Quick Links window.)

A Quick Link is a navigational shortcut to quickly display a particular window within a module. Each module has its own set of Quick Links, and the Quick Links for one module are not displayed when a different module is selected. They greatly speed the display of a window that you frequently use.

For example, say you often use the Construction History window. To navigate to this window you would need to select the Roadway module, then click Database, then hold the cursor over Construction, and finally click Construction History to display the window. If you save this window as a Quick Link, the Quick Links list in the left gutter will show the name of this window as a hyperlink after selecting the Roadway module. You may then click the hyperlink, which displays the window directly without the intermediate steps.

To create a Quick Link for a window, click the  icon. The name of the window will then appear in the Quick Links list on the left side of your browser window. If you want to remove the window from the list of Quick Links, click the  icon that appears beside the name of the window.

In addition to managing the list of Quick Links by simply adding and deleting links, you may also use the  icon to display the Manage Quick Links window. An example of this window is shown on the following page.



This window allows you to configure Quick Link groups as well as what is displayed when you hold the cursor over a Quick Link hyperlink in the left gutter. By selecting a particular Quick Link group, you select what Quick Links are shown in the left gutter. You also may determine whether the Quick Links are for you alone or are shared amongst all members of your agency.

**NOTE**

The default Quick Link group is called Quick Links. Each user has a similarly-named group. This group is intended for private use only and should not be made public.

The Manage Quick Links window has two tabs: Link Groups and Links. The Link Groups tab shows the different groups into which the Quick Links are divided. A check mark in the Public? column indicates that the Quick Link group is shared amongst all users. The group that is selected is the one that appears in the left gutter.

For the link group selected in the Link Groups tab, the Links tab shows the windows of the links in the Link Groups tab. The text in the Comments column is what is displayed when you hold the cursor over the Quick Link.

### **How to Create a Quick Link Group**

To create a Quick Link group, follow these steps:

1. Display the Manage Quick Links window by clicking the  icon. This window shows all groups of Quick Links that are yours as well as those shared amongst all users in your agency.
2. In the Link Groups tab, right-click and then click **Insert**. A new row is added to the table.
3. In the WLINKGROUP NAME column in the new row, type the title that you would like to appear above the Quick Links in the left gutter.
4. Optionally, if you would like this group to be available to all members of your agency, click the check box in the Is Public? column. The application places a check mark in the check box to denote that it is selected.
5. Click the **OK** button to save the new group and close the Manage Quick Links window.

### **How to Add a Quick Link to a Quick Link Group**

Once the Quick Link group to which a window's Quick Link will be added exists, follow these steps to create a link to the window within the Quick Link group:

1. Display the Manage Quick Links window by clicking the  icon.
2. In the Link Groups tab, click the desired Quick Link group to which the Quick Link will be added. The system highlights the group to show that it is selected.
3. Click the **OK** button to save your selection and close the Manage Quick Links window.
4. Display the window that you would like to have a Quick Link as part of the selected Quick Link group.
5. Click the  icon to create the Quick Link. The name of the window appears in the Quick Link list in the left pane.
6. Repeat step 5 for any additional windows you would like to be part of this Quick Link group.
7. When all Quick Links are set, re-open the Manage Quick Links window by clicking the  icon.
8. Click the Links tab to view the Quick Links that are part of the group.
9. For each Quick Link in the Links tab, enter in the Comments column the text you would like to be displayed when a user holds the cursor over the link.
10. Optionally, you may determine in what order the Quick Links are displayed by editing the Order column. The default order is the order in which the Quick Links

were created. If you would like a different order, edit the numbers in the column to reflect the desired display order (top to bottom).

11. Click the **OK** button to save the new group and close the Manage Quick Links window.

#### **How to Select What Quick Links Are Displayed**

If the Manage Quick Links window shows only one Quick Link group, then all Quick Links are always shown in the left gutter.

However, if more than one group is shown, you may select what Quick Links are displayed as follows:

1. Display the Manage Quick Links window by clicking the  icon. This window shows all groups of Quick Links that are yours as well as those shared amongst all users in your agency.
2. Click the row showing the Quick Link group you wish to use. (If you want to see what Quick Links are actually in a group, select the group and then click the Links tab.)
3. Click the **OK** button. The Manage Quick Links window closes and the application displays the Quick Links of the selected group in the left gutter.

#### **2.4.3. Location Reference**

The Location Reference field contains a drop-down list of all location referencing systems utilized by the database. The various location referencing systems are configured in the Setup Location Reference Methods (LRMs) window, which is described on page 168.

You may change from one referencing system to another by selecting a different LRM from the list that is displayed by clicking the down arrow in this field. Once a new LRM is selected, the next time you open a window with a table that contains location data, the locations will be identified by the newly selected LRM. See page 171 for an example of changing the referencing system.

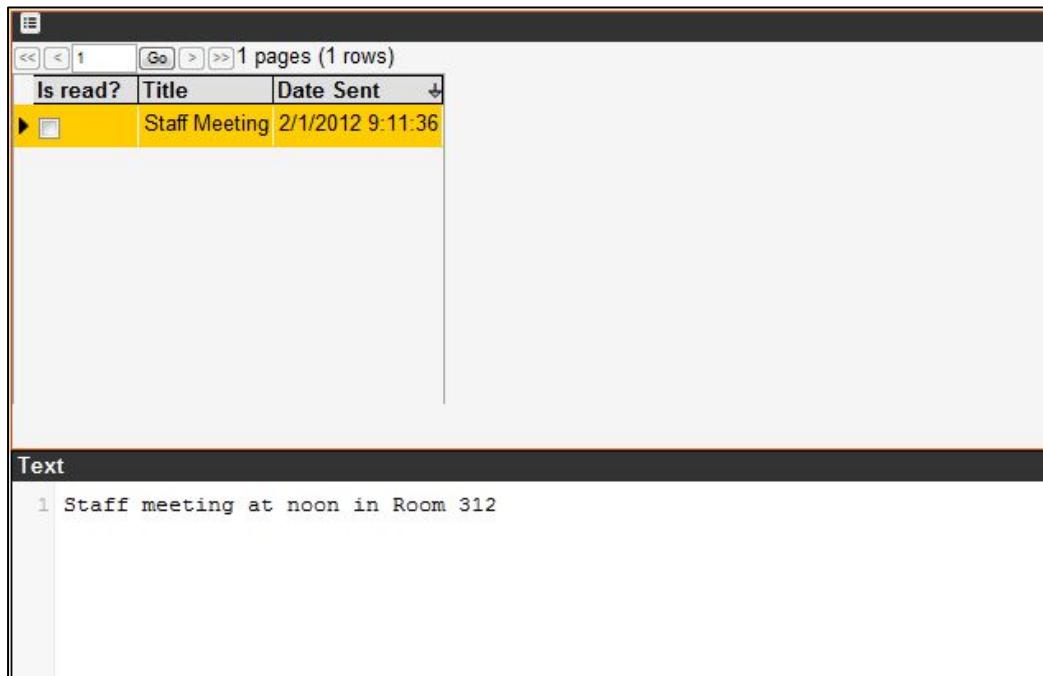
#### **2.4.4. Color Schema**

The left gutter provides a Color Schema field with a drop-down list of colors. The setting of this window determines the background color of the window as well as the color used for highlighting selected records. For example, in the screen shot shown on page 6, the Color Schema field is set to Navy. This causes the upper border and hyperlinks to be in blue and, while not shown, selected records would be in gold.

You may select a color from the list to change the appearance of the window and what color indicates a selected record.

#### **2.4.5. Messages**

While each module provides its own Messages window, the contents of the window are the same for all modules. When new messages are received, the phrase New Message is displayed in the left gutter. You may then click the phrase to display the Messages window, with the most recent message being the first entry in the table. The lower portion of the window shows the text of the message selected in the upper pane. A typical Messages window is shown below.



After reading a message, you may denote that the message is read by placing a check mark in the Is Read? column (by clicking the check box in this column). When you close or refresh the Messages window (after saving any changes), the system deletes all messages marked as read.

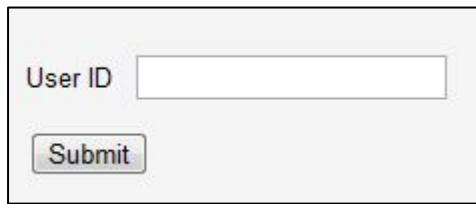
#### **2.4.6. Design Mode**

The Design Mode check box is only available for those users who are designated as an "administrator" (that is, has the Is Admin? check box selected) in the User Names and Access window.

When you select this check box, the system enters Design Mode. In this mode you can configure the shape and size of windows and panes. See page 249 for further information on customizing the appearance of the application.

### **2.5. Forgotten Password Procedure**

The log-in window provides a [Forgot Your Password?](#) hyperlink. If you forget your password, click this hyperlink to have a new password emailed to you. When you click the hyperlink, the system displays a new browser tab (or window, depending on your browser) with a field for entering your User ID. An example of this new tab is shown below.



A simple user input form consisting of a rectangular box. Inside the box, the label 'User ID' is positioned above a text input field. Below the input field is a 'Submit' button.

Enter your User ID in the field and then click the **Submit** button. The system will then generate a new password and send it to the email address configured in your user profile that is stored in the User Names and Access window.

Once you receive the email with the new password, use it in the log-in window. Provided you enter the password correctly, the system will log you in and then display the window for changing your password. See the following section for assistance on changing your password.

Once you have successfully changed your password, the system will display the window for selecting an administrative unit and a security profile (or will simply log you in if you are only assigned one administrative unit and one security profile).

## 2.6. Change Password

A password is needed to access the application and, if configured, to execute certain commands (as set in the Actions Rights window). Periodically, in the interest of maintaining good system security, you should change the password. Additionally, if you forgot your password and the system emailed you a new one, you must create a new password immediately after logging in with the new password.

To change your password, follow these steps:

1. Display the Change Password window. An example of this window is shown below.



The screenshot shows a software interface titled "Roadway > Utilities > Change Password". It features three text input fields: "Current Password" (with a red border), "Enter new password", and "Confirm new password". A "Change Password" button is located at the bottom right.

2. Type your current password in the Current Password field.
3. Press the tab key and then type your new password in the Enter New Password field.
4. Press the tab key and then type your new password in the Confirm New Password field.
5. Click **Change Password**. The system checks that the passwords you typed in the bottom two fields match and, if so, displays a message saying your password is changed. You may then use the new password the next time you sign on. (If the passwords do not match, the system alerts you to this condition and you will need to re-enter the new password in both fields to proceed.)

## 2.7. Common Commands

Many commands are used throughout applications developed by AgileAssets and perform the same actions regardless of the window that is displayed. These are termed common commands. These commands generally appear on shortcut menus that are displayed when you point to a window or pane and right-click (that is, click the button on the right side of the mouse).

The following sections describe these commands.

### 2.7.1. Change Graph Properties

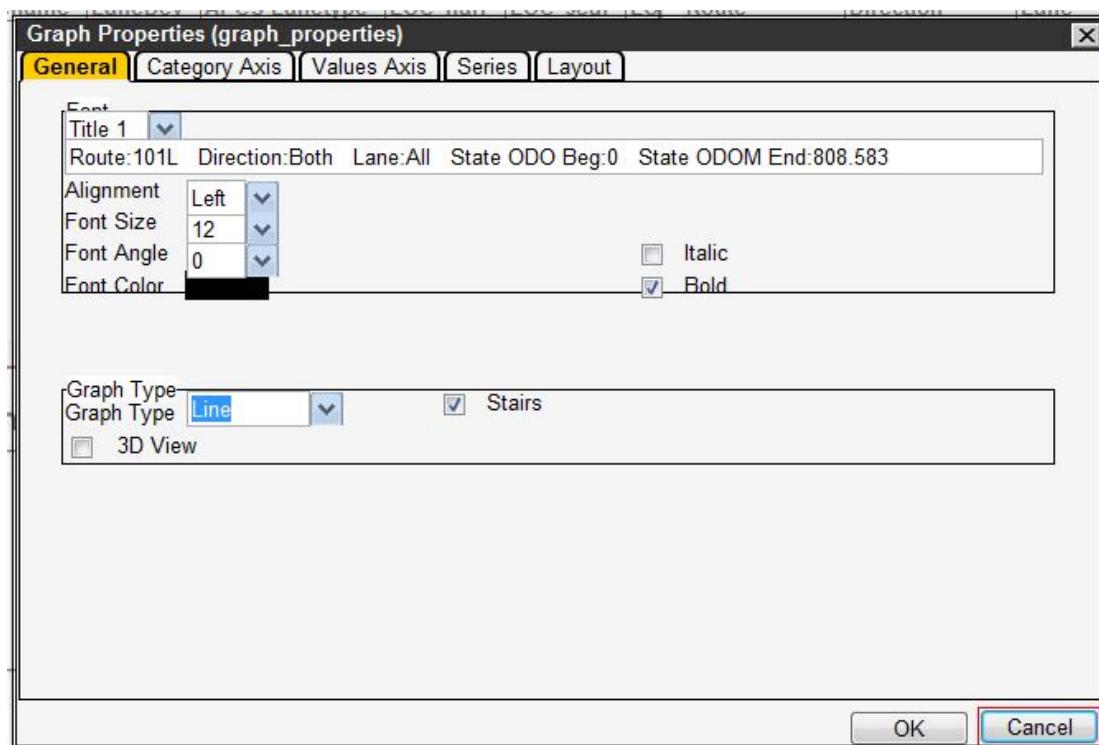
When graphs (and graph reports) are displayed, how the graph displays may be altered by using the **Change Graph Properties** command. This command displays a dialog box, an example of which is shown on the next page. The dialog box contains the following tabs:

- General – This tab configures the type of graph (line, bar, scatter, or area) and the appearance of the graph title(s).

- Category Axis — This tab configures the X axis, including the label for the axis and how the axis is divided.
- Values Axis — This tab configures the Y axis, including the label for the axis and how the axis is divided.
- Series — This tab configures the legend area of the graph.
- Layout — This tab configures where the graph appears in the graph pane and the size of the graph relative to the overall size of the pane.

### General Tab

The General tab of the Graph Properties dialog box configures the title(s) of the graph and what type of graph is displayed.



The upper portion of the tab configures the title(s). You may display one or two titles for a graph, with the title selected from the drop-down list at the top of the tab (Title 1 is the uppermost title). For the title selected from the drop-down list, the remaining fields in the upper part of the tab display:

- The title wording. For Title 1, the system automatically generates the title working. Note that you may change the wording to something else, but it will not be permanent; the next time the graph is displayed, the original, automatically generated title will return (and your changes will be lost).
- The alignment of the title (left, center, or right). The boundaries of the title are the width of the graph, not the width of the pane.
- The font size, whether the font is bold, italic, both, or neither.
- The orientation of the title (Font Angle field). The title may be rotated from the default horizontal position (Font Angle = 0). Positive values rotate the title counterclockwise around the left end of the title; negative values rotate the title clockwise around the right end of the title. Note that +90° is not the same as -90° — both result in a vertical title, but +90° causes the title to read up, while -90° causes the title to read down.

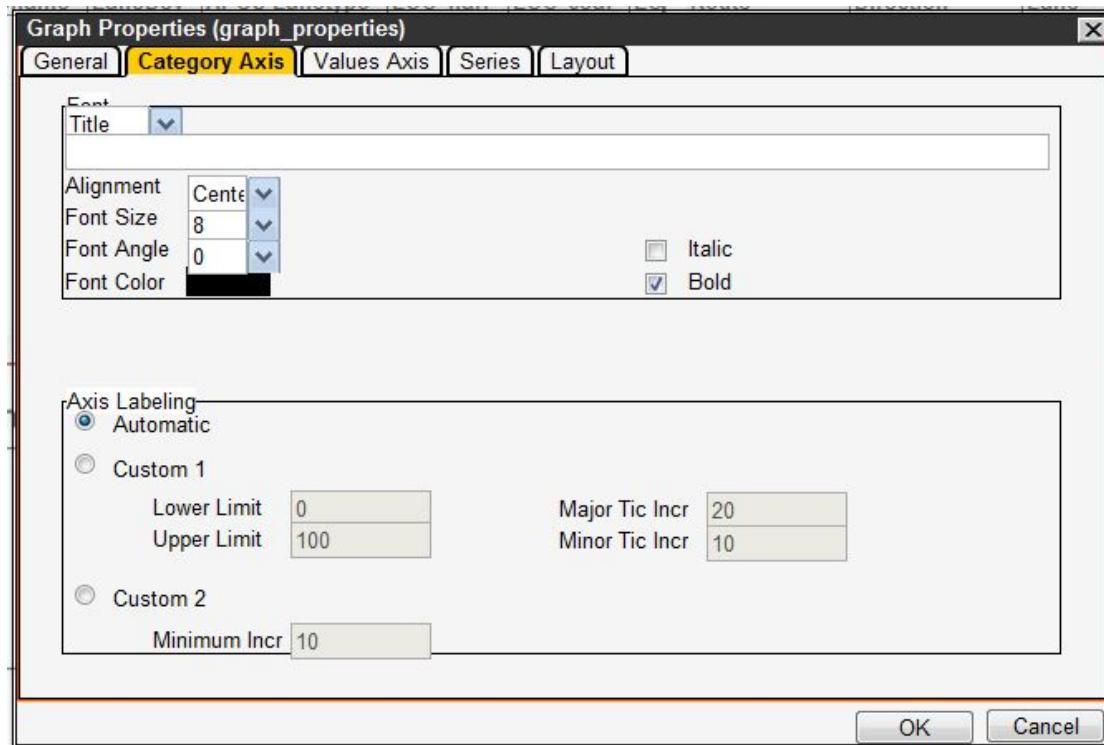
- The color of the title. If you want to change the color, double-click the displayed color. The system will then display the Color Settings dialog box, and you may select a different color.

The lower portion of the tab configures the type of graph. The type of graph is selected from the drop-down list in the Graph Type field. You may select one of four types of graphs: line, bar, scatter, or area. Depending on what type is selected, a second drop-down field may appear to further determine the type of graph as described below:

- If you select Line, then a Stairs check box is shown. When this check box is selected, the curve will show discrete "steps" between data points. When it is clear, the curve will flow more smoothly from data point to data point.
- If you select Bar, then a second field appears to the right that allows you to choose either Stack or Side. In a Stack bar graph, each X-axis value has a single, layered bar (where each layer is a different series). In a Side bar graph, each X-axis value has multiple, side-by-side bars (where each bar is a different series).
- If you select Area, then a second field appears to the right that allows you to either Stack or Percentage. In a Stack area graph, each series' value is added to the previous series' value to show a cumulative graph using actual data. In a Percentage area graph, each series' value is the percentage of total value.
- If you select Scatter, no additional fields are displayed.

For all types of graphs, you may elect to give the graph a 3-D effect by checking the 3-D View check box. When selected, this check box causes a shadow to be added to the curve and axes to give the illusion of depth.

### Category Axis Tab



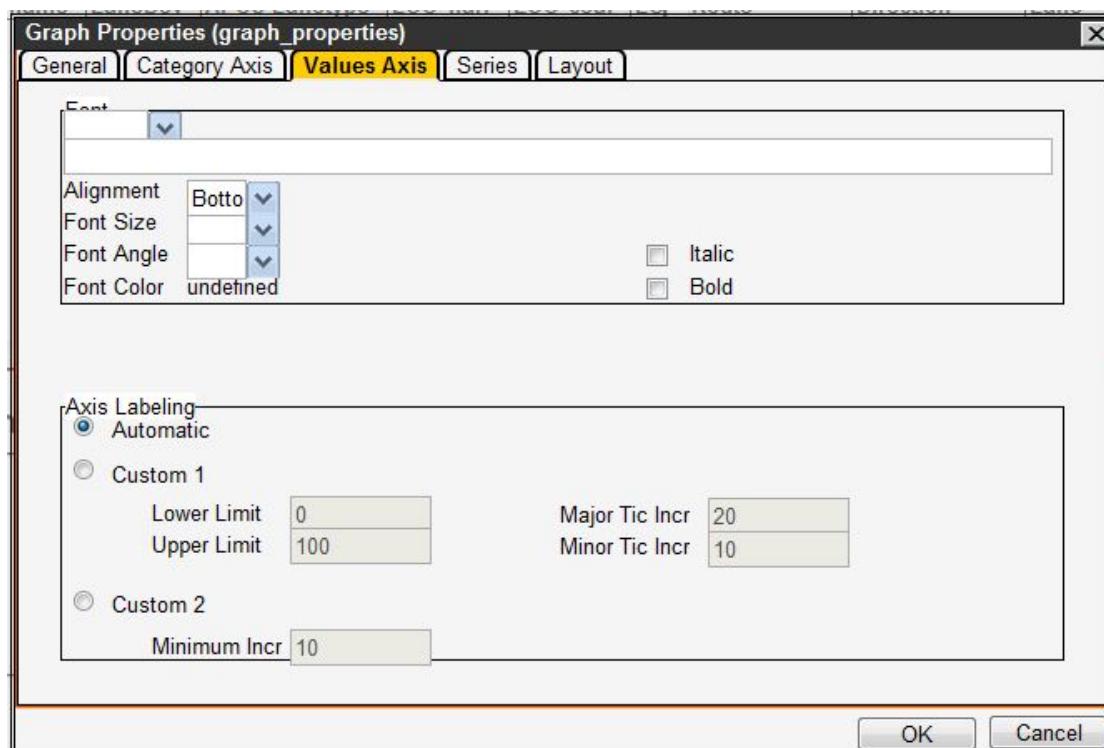
The Category Axis tab configures the X axis of the graph. The X axis has two labels: the title of the X axis and the values shown along the X axis. You select which one is being configured by selecting the desired label from the drop-down list at the top of the tab. In the list, Title

refers to the title of the X axis and Label refers to the values shown along the axis. The fields for configuring the appearance of the labels are the same as found on the General tab.

The lower portion of the tab is only displayed if the X axis shows numeric values. (This means that for graph reports, which always show non-numeric values, the lower portion of the tab is not displayed.) This section of the tab provides three choices for configuring the displayed values along the X axis:

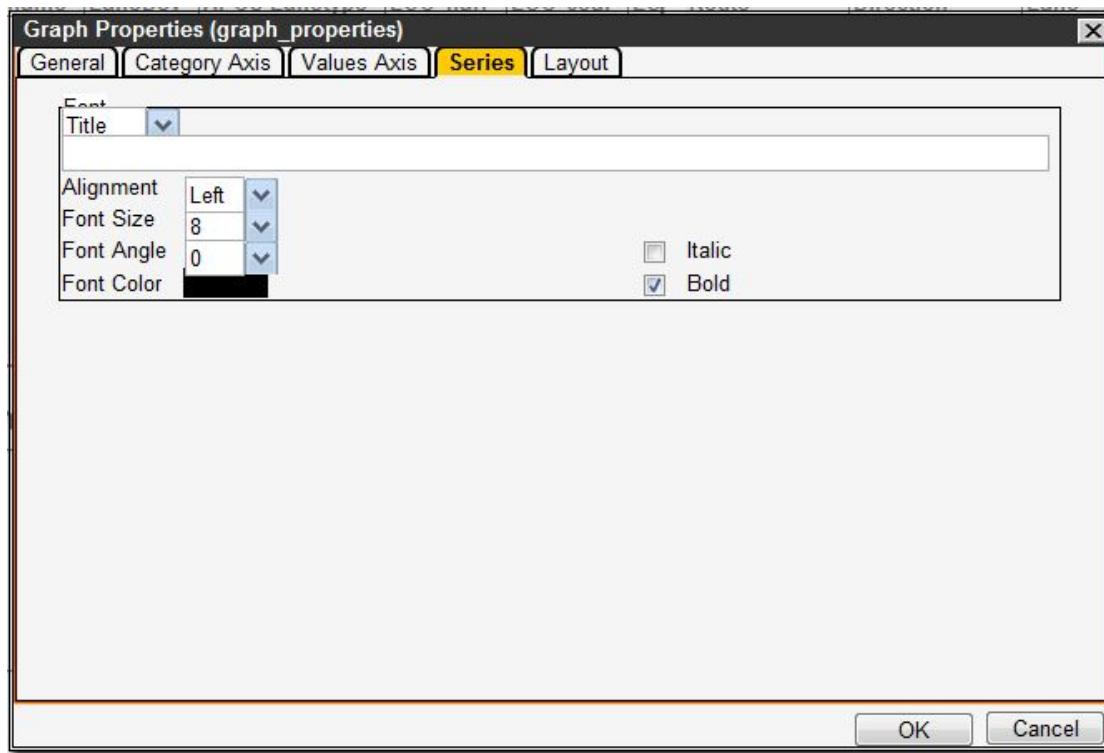
- Automatic — When this radio button is selected, the system determines the starting and ending values for the X axis as well as the major divisions of the X axis between these values. Note that minor, unlabeled tick marks are not displayed with this selection.
- Custom 1 — You determine what the starting (Lower Limit) and ending (Upper Limit) values are for the axis. You also set the intervals at which the major and minor tick marks will be shown. (Note that only major tick marks are labeled.)
- Custom 2 — You specify the minimum interval between the major tick marks, and the system uses this information to divide the axis as it deems appropriate for the data range to be shown by the axis.

### Values Axis Tab



The Values Axis tab configures the Y axis of the graph. The fields of this tab are the same as for the Category Axis tab.

### Series Tab

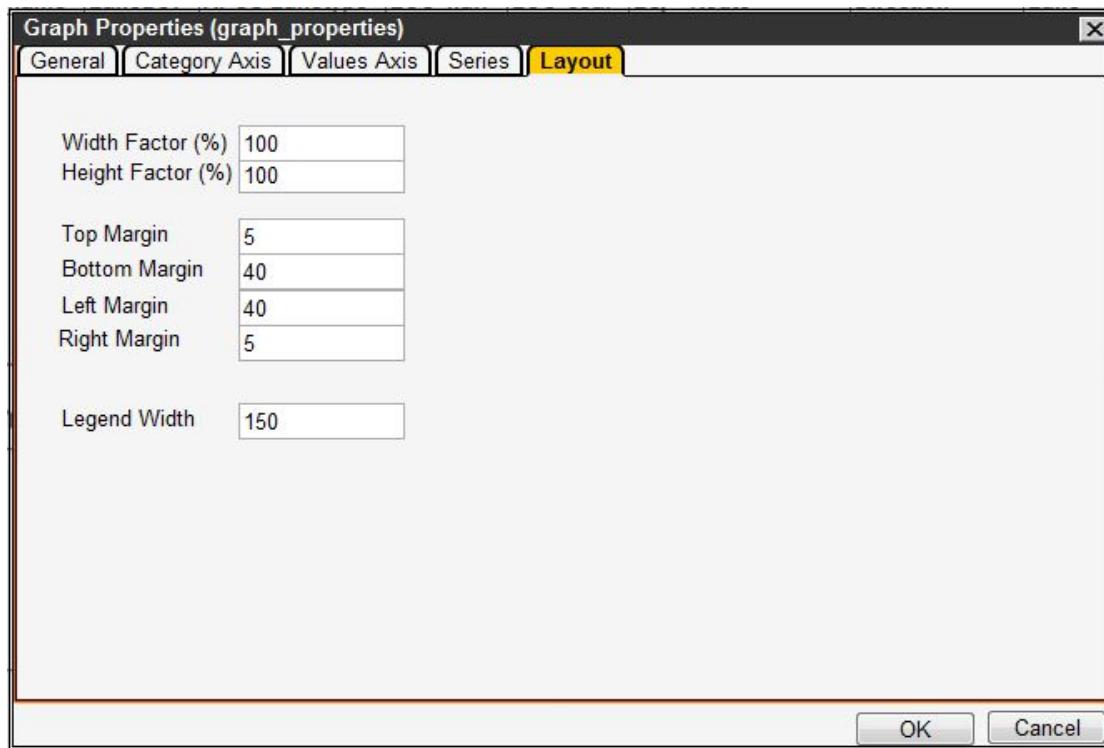


The Series tab configures the appearance of the "legend" area of the graph (that is, the part of the graph on the side that indicates what colors represent what data). The tab provides a drop-down list to select what you wish to configure: the title of the legend area or the text in the legend area. The fields on this tab are the same as found on the General tab.

**NOTE**

The Color field refers to the color of the text, not the color of the data points in the graph.  
(The color of the data points cannot be edited.)

### Layout Tab



The Layout tab configures where the graph appears in the pane. It provides the following fields:

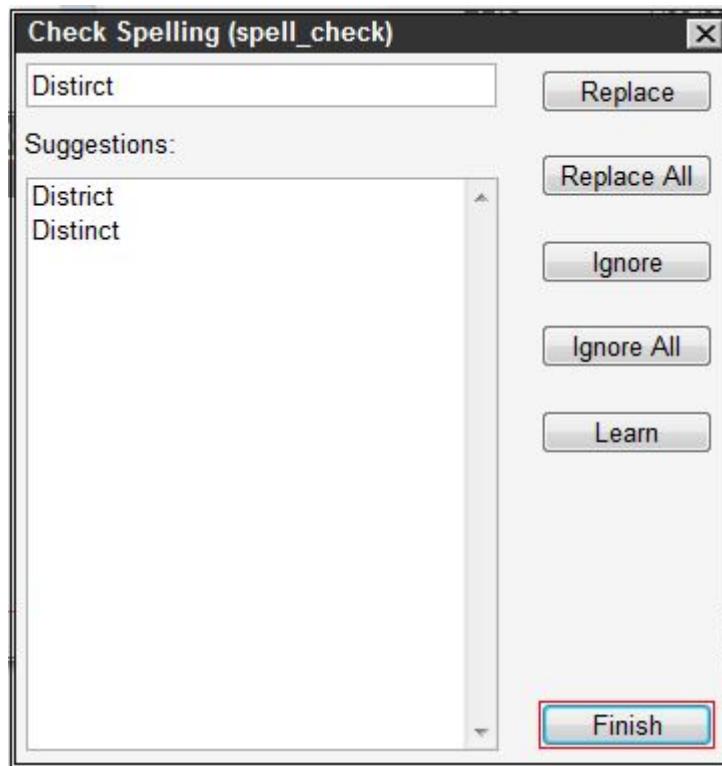
- Width and Height Factors — These two fields determine the size of the graph relative to the size of the pane, and are expressed as a percentage. A value of 100% is the maximum (that is, the graph extends the full distance between the margins set in the following four fields).
- Margins — These four fields determine the distance between the actual axes of the graph and the labels and titles around the axes. The values are actual distance from the edges of the pane, not a percentage. For example, if you increase the value of the left margin, the space between the label for the Y axis and the axis itself will increase (with the label remaining stationary and the Y axis moving to the right). Generally, the intent of the margin fields is to allow you to configure sufficient space for the values shown along the axes.
- Legend Width — This field determines the width of the legend area of the graph. Note that selecting a different value will cause the graph area of the graph to shrink or expand to fill the space configured for the entire graph. The text in the legend area will wrap around if the width is too small.

#### 2.7.2. Check Spelling

The **Check Spelling** command is available for text fields. You find this command by first double-clicking the text field to display the text string in a pop-up window. You then right-click the window to display a shortcut menu that contains the **Check Spelling** command.

To initiate spell-checking, select the command from the shortcut menu. The system then begins checking the text string for words not in the spell-check dictionary. If no misspellings are found, the system displays a message indicating that the check is complete.

On the other hand, if a misspelling is found, the system displays the Spell Check dialog box with the misspelled word shown at the top of the box. An example of this dialog box where the word "district" is misspelled is shown below.



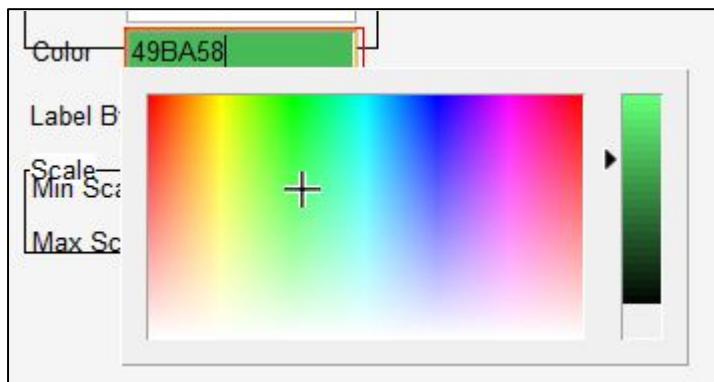
The dialog box also displays suggestions for how the word should be spelled. You may then respond by:

- Selecting one of the suggestions and then clicking **Replace** to replace the misspelled word with the suggestion you highlighted (or **Replace All** if you wish to correct all instances in the entire text string).
- Highlighting the flagged word and typing how it should be spelled and then clicking **Replace** or **Replace All**.
- Clicking **Ignore** to cause the system to continue spell checking without changing the flagged word (or **Ignore All** to prevent the system from flagging other instances of the word in the entire text string).
- Clicking **Learn** to add the word to the spell-check dictionary.

After clicking one of the buttons, the system takes the requested action and then proceeds with spell-checking. When the entire text string is checked, it displays a message saying the check is complete.

You may abort the spell-check process before it completes by clicking **Finish**.

### 2.7.3. Color Settings



In some windows of the system, information is indicated by a color as are labels. You may change this assignment by double-clicking the color in the table. The system will then display the Color Picker window with available colors. Slide the right-pointing triangle up the slide on the right to set the saturation and then click the desired color in the color palette. Crosshairs indicates the selected color. The selected color and color number is also shown in the underlying field from which you launched the Color Picker.

### 2.7.4. Delete

The **Delete** command removes the highlighted row or selected node from the table or tree view.

**Note:** It is strongly recommended that you save before deleting anything. This action prevents changed information from being lost should the system not allow you to perform the deletion.

For example, say you were working in the Labor Classification Codes window and made several changes to the information shown in the Details table. You then decided to delete a code. However, this code is already in use. Therefore, the system will not allow you to delete the code. When the deletion is disallowed, the system restores the table to the condition it was in at the last save – and thus all your changes will be lost. If you had saved before attempting to delete, the system would restore the table with your changes intact.

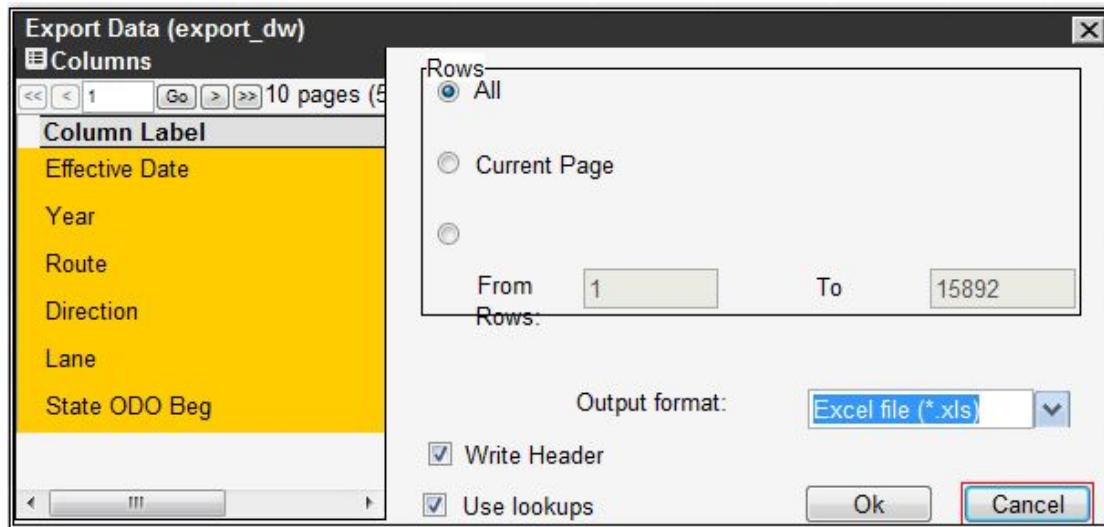
### 2.7.5. Export Data

The **Export Data** command is available in windows or panes that show tabular data. It allows you to select what table columns and records you wish to export and then save the data to a file outside of the system. The data may be saved as a comma-separated text (.csv) file; an HTML file; a Microsoft Excel (.xls) file; a DBF file; a Google Earth KML file; or a Shape file. (If the Shape type is selected, the application generates a zip archive with .shp, .shx, .dbf and .prj files. This type is only available for data windows with location information and for systems that have an attached “route” map.)

When you select the command, the system responds by displaying a dialog box so you may select the columns you wish to export. An example of this dialog box is shown on the following page.

#### NOTE

If the window or pane is filtered from which you select the **Export Data** command, the columns shown in the dialog box will also be filtered.



You may also select what records to export – either all records, just those currently displayed, or certain pages of records. The Use Lookups check box allows you to choose whether the lookup label is exported (selected) or the ID is exported (not selected).

After making your selections, click the **OK** button and the system will transfer control to your computer, which asks you whether you wish to display the exported data in a default application or save the file. Make your desired selection to complete the export process.

#### 2.7.6. Filter

##### NOTE

Changing the LRS system may cause filter criteria to be cleared if the criteria include location-referencing columns.

Graph View ("graph down the road") windows are not affected by filters set in Data View or other windows. For example, filtering the route to one route in a Data View window will not restrict the routes shown in the Graph View window (even though the Route (ID) column appears in both views). You must set a separate filter to restrict the records displayed in a Graph View window.

Whether a column may be filtered is set in the Data Window tab of the User Control Properties window (which may only be displayed when in Design Mode). In this tab, the upper pane provides a Filterable? check box that (when selected) determines whether all columns in the table may be filtered. **If filtering is not currently enabled for a window, do not turn it on without first consulting with AgileAssets software development. For example, the Filterable? checkbox should not be selected when the data source is a 'DW\_ procedure.**

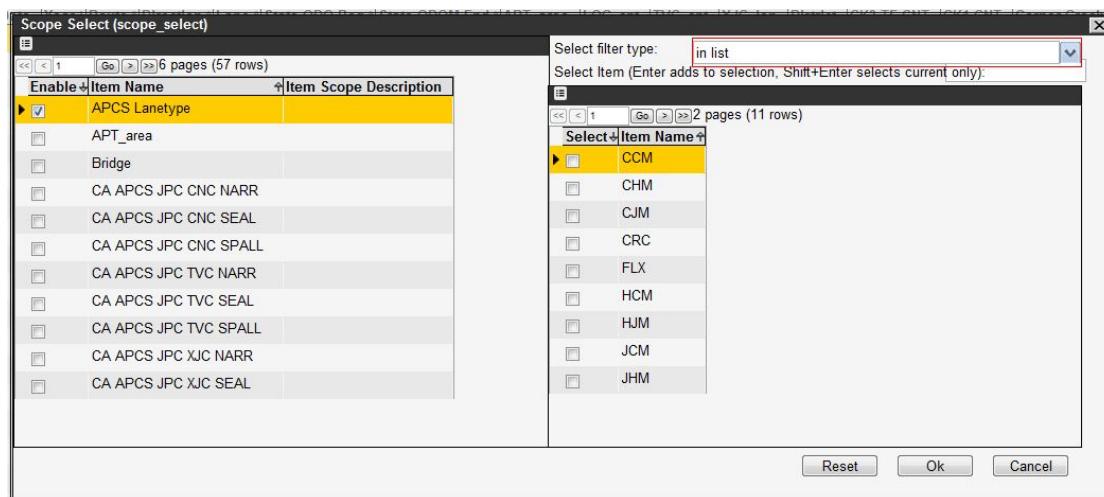
In addition to filters that you set, data records may be automatically filtered to only those records that have a value in an "owner" column that equals the administrative unit you selected when you logged on. This occurs when the Access-regulated? check box in the Data Window tab of the User Control Properties window is selected.

The **Filter** command allows you to restrict the data that appears in a column wherever that column appears in the module. In other words, filtering a column in one window affects all windows and panes in the module that also contain the filtered column. The filter remains in effect until removed or changed (logging off or changing modules will not remove the filter).

In addition, a filter is only for a particular user. That is, each user may configure his or her own filter for a column.

When data is filtered, a funnel on a yellow background ( will appear in the title bar of the window or pane. (Double-clicking the icon in the title bar will also display the Filter window.)

A typical Filter window is shown below.



### **Filter Operators**

In the right pane of the Filter dialog box is a drop-down list that changes depending on what type of column is selected in the left pane. The following table summarizes the operators that are available for each type of column and what results from using the operator. For List data, this table also describes the field for directly entering filter criteria as well as the **Select Like** right-click command.

Note: This table is the general list of available operators, but not all implementations include all of these. Contact AgileAssets if your particular implementation does not include an operator you need and ask them to amend the WHERE clause of the SETUP\_FILTER table.

Data Type	Operator	Description
Date	=	All records where the date in the column matches the entered date exactly are displayed.
	>=	All records where the date in the column matches or is later than the entered date are displayed.
	<=	All records where the date in the column matches or is earlier than the entered date are displayed.
	>	All records where the date in the column is later than the entered date are displayed.
	<	All records where the date in the column is earlier than the entered date are displayed.
	<>	All records where the date in the column is not the same as the entered date are displayed.

Data Type	Operator	Description
	Between	All records where the date in the column is within the time period set by the two entered dates (the earlier date on the left, the later on the right) are displayed. Note: The displayed dates include the earlier date, but exclude the later date. Therefore, the later date should be set to one day later than the latest date you want displayed.
	Today	All records where the date in the column matches your computer's current date are displayed.
	Yesterday	All records where the date in the column is one day earlier than your computer's current date are displayed.
	This Month	All records where the date in the column has the same month as the month in your computer's current date are displayed.
	Previous Month	All records where the date in the column is one month earlier than the month in your computer's current date are displayed.
	This Week	All records where the date in the column falls on a day that is within the same week as the day in your computer's current date are displayed. (A week is defined as being from Sunday to Saturday.)
	Previous Week	All records where the date in the column falls on a day that is within the previous week as the day in your computer's current date are displayed.
	This Year	All records where the year of the date in the column matches the year of your computer's current date are displayed.
	This Fiscal Year	All records where the date falls within the current fiscal year. (The fiscal year is defined directly in the Oracle database, not in the AgileAssets application.)
	Previous Year	All records where the year of the date in the column is one year earlier than the year of your computer's current date are displayed.
	Previous Fiscal Year	All records where the date falls within the previous fiscal year.
	Current Pay Period	All records where the date falls within the current pay period.
	Previous Pay Period	All records where the date falls in the previous pay period.
List	Is Null	All records where the column is null ('blank') are displayed.
	Is Not Null	All records where the column is not null ('blank') are displayed.
	In List	All records where the column has the selected data element(s) are displayed.
	Not In List	All records where the column does not have the selected data element(s) are displayed.
	Is Null	All records where the column is null ('blank') are displayed.
	Is Not Null	All records where the column is not null ('blank') are displayed.

Data Type	Operator	Description
	Direct Entry Field	<p>After selecting either In List or Not in List, the system displays a list of values along with a field for the direct entry of the item to be selected. Type the string and as you type the system will attempt to identify what you are typing by displaying suggestions. You may continue typing or use the arrow keys to highlight a choice from the displayed suggestions.</p> <p>Once the desired string is shown in the field, press the Enter key to select the item in addition to items already selected, if any. Alternately, you may press Shift+Enter to select only the item (de-selecting all other selections, if any). The system places a check mark in the check box to denote that the item is selected.</p>
	Select Like	<p>Once a list of values is displayed by selecting either In List or Not in List, this command is available on a shortcut menu displayed by right-clicking the list. After selecting the command, the application displays a dialog box. In the dialog box, type a string and then click the <b>Select</b> button. The application will then select all values in the list that have the string anywhere in the value.</p>
Number	=	All records where the number in the column matches the entered number exactly are displayed.
	>=	All records where the number in the column matches or is larger than the entered number are displayed.
	<=	All records where the number in the column matches or is smaller than the entered number are displayed.
	>	All records where the number in the column is larger than the entered number are displayed.
	<	All records where the number in the column is smaller than the entered number are displayed.
	<>	All records where the number in the column does not match the entered number are displayed.
	Between	All records where the number in the column is within the range set by the two entered numbers (the smaller on the left, the larger on the right), inclusive (that is, the number is between or matches the entered numbers).
	Is Null	All records where the column is null ('blank') are displayed.
	Is Not Null	All records where the column is not null ('blank') are displayed.
Text	=	All records where the column has the text string exactly are displayed.
	<	All records where the column has text that comes before the entered text (that is, earlier in the alphabet).
	>	All records where the column has text that comes after the entered text (that is, later in the alphabet).
	LIKE	This is similar to the equal sign, but is used for wildcard searches. Permitted variables for wildcard searches are the percentage sign (%), which stands for one or more characters, and the underscore (_), which stands for a single character, with the variable appearing in the text string at the location where various characters could occur.
	Is Null	All records where the column is null ('blank') are displayed.
	Is Not Null	All records where the column is not null ('blank') are displayed.

## How to Set a Filter

To set a filter, follow these steps:

1. Display a window that contains the column you wish to filter.
2. Right-click the selected record in the window (or pane of the window) that contains the column to be filtered and then click **Filter** from the shortcut menu. The application responds by displaying the Filter window, which contains all columns in the window (or pane).
3. In the left pane of the Filter window, locate the column to be filtered and click it to select it.
4. In the right pane of the Filter window, click the down arrow to display the different filter types and then click the desired filter type in the list. The application responds by displaying the data values that are available for filtering. It also places a check mark in the Enabled column of the left pane to denote that the column is filtered.
5. In the right pane, select the desired data values by clicking the check box for each value.

Note: For List data types, you also have the option of directly entering the filter criteria in the field above the list values or utilizing the right-click **Select Like** command.

6. Once all values are selected, repeat steps 3 through 5 for any other columns that you would like to filter.
7. Once all filters are set, click the **OK** button. The application activates the filter and adjusts the records in the window (or pane) accordingly. The application also places a funnel on a yellow background () in the title bar to denote that the data is filtered.

## How to Remove a Filter

To remove a filter, display the Filter window from a window (or pane) that contains the filtered column. In the left pane of the Filter window, click the record for the column to select it. Then click the Enable check box to remove the check mark and disable the filter. Finally, click the **OK** button to close the Filter window.

Note: To remove all filters, click the **Reset** button. This clears all check boxes in the Enable column. You may then click **OK** to close the Filter window.

### **2.7.7. Filter By This Value**

The **Filter By This Value** command allows you to select a value in a table and then filter the display to show only those records with the selected value. When you select this command, the application displays the Filter window. However, unlike the window that is displayed by the **Filter** command, this left side of the window only shows the column that contains the selected value (rather than all columns that may be filtered). Additionally, the filter criteria on the right side are automatically set to be equal to the selected value.

If the filter window shows the proper value and filter expression, you may click the **OK** button to activate the filter. If desired, you may instead edit the filter value and/or expression before clicking the **OK** button.

### **2.7.8. Find**

You use the **Find** command to locate a particular node in a tree view or a particular entry in a column of a table (the system does not search the entire table). It differs slightly depending on whether you select it in a tree view or a table view.

When you right-click a node in a tree view and then click the **Find** command, the system displays the following dialog box:



When you right-click a column in a table and then click the **Find** command, the system displays the following dialog box:



These two dialog boxes operate in the same way. (Since the drop-down list in the table version of the Find dialog box has only one entry, it may be ignored.)

To search for a particular word or phrase in the hierarchy or table column, type what you're looking for in the field and then click **Find**. The system will search for all instances that include (either wholly or in part) the string of characters that you type, without regard to case. When searching in a table, you may better target the search by utilizing the check boxes:

- By selecting the Match Case check box, the system will only find those instances where the case matches.
- By selecting the Exact Match check box, the system will only find those matches that wholly include the entered string.
- By selecting the Use Wildcards check box, you may use "wild cards" in place of text in the search string. Use a percentage sign (%) for multiple characters and an underscore (\_) for a single character. Also, for these searches, use the LIKE operator.

The system searches from the node or row to which you pointed to initiate the **Find** command to the end of the hierarchy or table. It will then "wrap around" to the beginning of the hierarchy or table and continue searching. Once a match is found, the system moves the viewable portion of the hierarchy or table and highlights the node or row with the sought string. The Find dialog box will remain displayed after locating an instance of the sought-for word or phrase to allow you to search for the next occurrence by re-clicking the **Find** button.

As an example of using the **Find** command, say you are looking for road section AL01Z. You would initiate the command by right-clicking in the Section Name column and then clicking **Find**. In the dialog box, you would have the following options for searching:

- You may simply type al01 and click **Find**. The system would then find all road sections that contained al01 — that is, AL01A, AL01B, AL01C, and so on. (It would also find lower case and mixed case versions of these.)
- You may type al01z, select the Exact Match check box, and then click **Find**. The system would then find only those road sections designated al01z whether in lower case, upper case, or mixed case (that is, al01z, AL01Z, AL01z, aL01Z, etc.).

- You may type AL01, select the Match Case check box, and then click **Find**. The system would then find only those road sections that contained AL01 in upper case (that is, AL01A, AL01B, AL01C, etc.).
- You may type AL01Z, select both the Exact Match and the Match Case check boxes, and then click **Find**. The system would then find only the road section designated AL01Z.

Note: In all of these searches, the system will only find instances that are in the Section Name column. Any instances that occur in other columns will not be found.

### 2.7.9. Insert

The **Insert** command adds a blank record to a table. In the new record, fields that must have data for the record to be saved ("required fields") are marked by an asterisk (\*) to the left of the field name.

#### NOTE

For windows that show a tree view and a table view, the command **Add Branch** is used instead of **Insert**.

### 2.7.10. Insert Like

The **Insert Like** command adds a new record to the table and copies the data from the selected record to the new record. By default, the data from all columns is copied. However, the data from a column may be prevented from being copied by clearing the Copyable check box for the column's record that is found in the User Control Properties dialog box > Data Window tab > Columns pane.

### 2.7.11. Massive Update

The **Massive Update** command allows you to modify the values of one or more columns of all selected records at one time (provided the columns are marked as being capable of being updated by this command in the User Control Properties dialog box).

When you select this command the application displays a dialog box. The left side of this dialog box shows all the columns in the table that may be updated. (A column is designated as capable of being updated by selecting the Massive Updatable? check box that is found in the Data Window tab of the User Control Properties dialog box).

The right side of the dialog box provides a field to enter (or select) the value to which the column selected on the left side will be set. (After entering the value in the field, the application places a check mark in the Modified? check box on the left side to indicate that the column's value will be changed.)

#### NOTE

For location-referenced data tables, type NULL in the field to set the selected column's value to NULL. For list data tables, select the Set to Null check box.

### ***Calculations Involving Numeric Columns***

For numeric columns ("R" view type), you can set all selected records either to some constant or use a Groovy script. The script may include the column ID of any numeric column in the table, not just the column that is being updated. Examples of update expressions are shown below:

IRI\_IND\*1.5+IRI\_OLD [column updated using the values in two columns]

Math.log(COL\_1) [uses Math Functions from Java.Math Library]

#### NOTE

When evaluating formulas, the original data from the data window (before opening the dialog box) is used. For example, if the formula for COL\_1 is COL\_2 + 1 and formula for COL\_2 is COL\_1 \* 3, the original values for COL\_1 and COL\_2 are used to evaluate both formulas.

#### **2.7.12. Maximize Data Pane**

A data pane may be increased in size by double-clicking the title bar for the data pane. After double-clicking, the data pane will fill the window (or, if the data pane is in a tab with other data panes, the width of the tab).

To restore the data pane to its original size, double-click the title bar.

#### **2.7.13. Selection Commands**

At many points in the system, tree views are provided. To select particular items in the tree view, the following commands are available when you right-click a node in a tree view:

- **Select This / Deselect This** – These commands work as a "toggle" to alternately select and de-select the node to which you pointed.
- **Select Branches / Deselect Branches** – These commands work as a "toggle" to alternately select and de-select all sub-nodes that are beneath the node to which you pointed. (Note that the "parent" node to which you pointed is not selected; you will need to use the **Select This** command separately to select it.)
- **Select All / Deselect All** – These commands work as a "toggle" to alternately select and de-select all nodes in the tree view regardless of whether they are visible.
- **Select Visible Items / Deselect Visible Items** – These commands work as a "toggle" to alternately select and de-select only the nodes that are visible in the tree view.

#### **2.7.14. Set Bookmark**

This command allows you to set what record is highlighted when the window opens for your current session (all bookmarks are cleared when you log off). The criterion by which a record is "bookmarked" is the data in one or more columns (as set in the User Control Properties dialog box). Each module may have a different bookmark.

For example, you could set a bookmark for a particular road section. Then, when you open any other window that shows road sections, the system will automatically display and highlight the first record that includes (or intersects) the bookmarked road section.

#### NOTE

The system only allows you to set a bookmark for one road section; selecting this command for a new road section de-selects the formerly bookmarked road section.

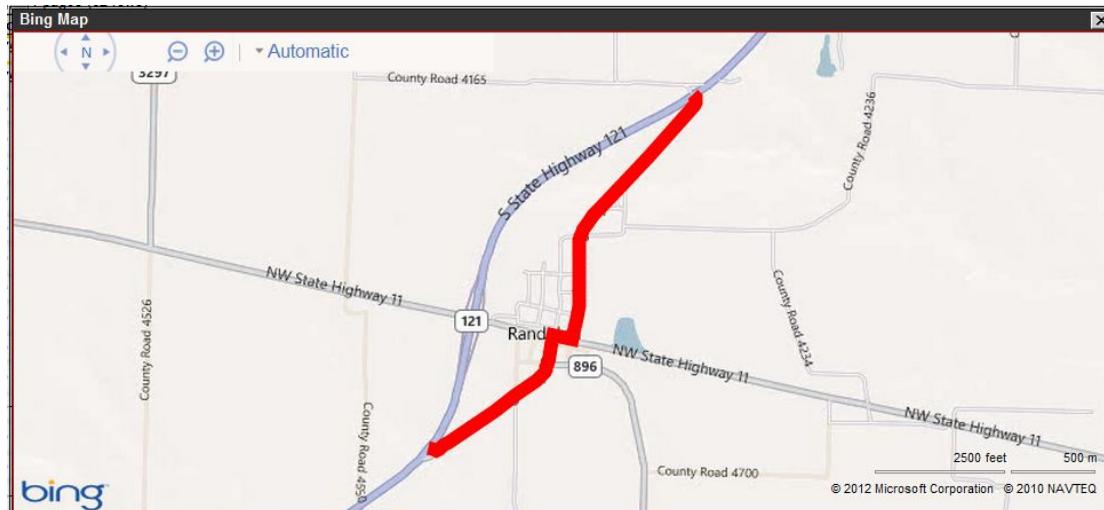
To set a bookmark, right-click the record showing the desired road section and then select this command from the shortcut menu that is displayed.

#### **2.7.15. Show on Bing Map**

The **Show on Bing Map** command is available for all windows that display a table that contains location data and that have a Bing map attached. When you right-click a record and

then select this command from the shortcut menu, the application displays a new window with the location shown on a Bing map. If you select a different record, the window with the Bing map remains displayed and the location changes to that of the newly selected record.

An example of a Bing map is shown below.

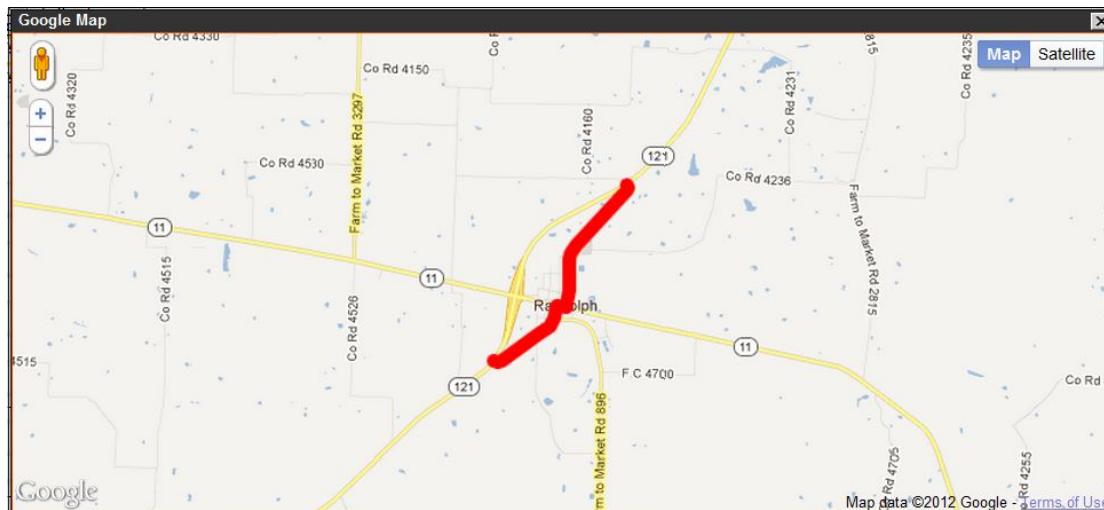


### **2.7.16. Show on Google Map**

The **Show on Google Map** command is available for all windows that display a table that contains location data and that have a Google map attached. When you right-click a record and then select this command from the shortcut menu, the application displays a new window with the location shown on a Google map. If you select a different record, the window with the Google map remains displayed and the location changes to that of the newly selected record.

Once the Google map is shown, if you right-click the map two additional commands are available to change the view: **Show Traffic** (which colors the map to show traffic volumes) and **Show Street View** (which shows a picture of the street and allows you to navigate along the route by clicking [if such data has been captured for the location]). You may also drag the "Person" icon to a place on a road to see the street view at that location (if such a street view exists) or click the "Terrain" button to display a quasi-topographical view of the map.

An example of a Google map is shown below.



### **2.7.17. Show on Map**

The **Show on Map** command is available for all windows that display location data from a table. When you right-click a record and then select this command from the shortcut menu, the application displays a new window with a map. The location in the selected record is shown in red on the map. Icons are available to manipulate the display of the map, although the Select Features icon () is disabled. See the Floating Map window on page 33 for more information.

#### **NOTE**

If you select a new record in the data table, the map window remains displayed and the application changes the highlighted section of the map to show the location in the newly selected record.

### **2.7.18. Show Report**

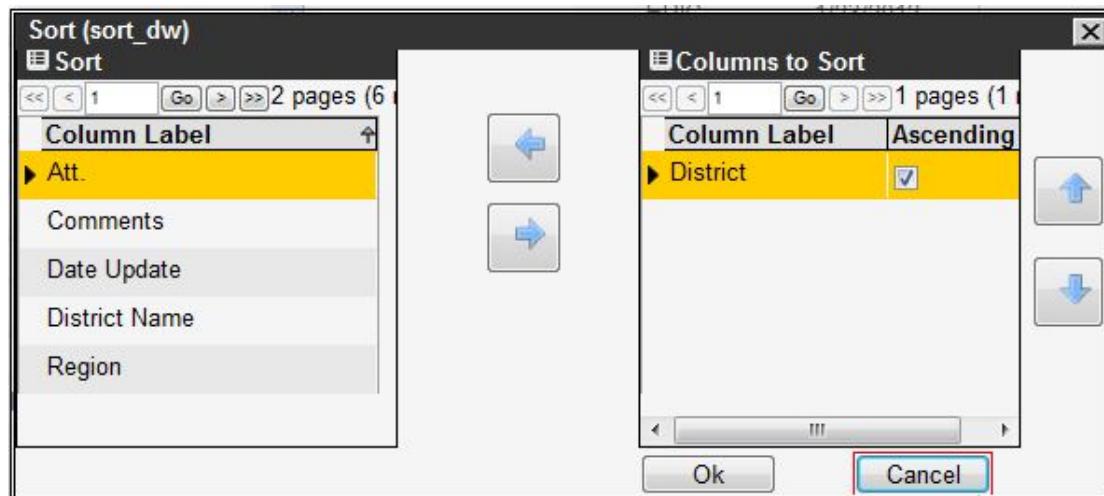
Some windows are configured to allow you to generate a report with data from the window or pane. The report is first created in the Reports module and then this report is configured to display from a particular window or pane via a database table (or, in some implementations, the Configure Window Reports window in the System module). Once configured, the **Show Report** command becomes available in the window or pane so you may display the report.

### **2.7.19. Show Videolog**

The **Show Videolog** command is available in any window that contains location data. When you execute this command, a new window opens that shows the video log. The upper part of the new window shows the route and mile point for the image shown in the bottom part of the window. A drop-down list sets what is shown in the bottom pane: center of road section, right side, left side, and "pavement" (straight down).

Controls are also provided in the bottom pane to step through the video log. In order from left to right, the icons perform the following operations: fast reverse, reverse, pause, forward, and fast forward.

### **2.7.20. Sort**



The **Sort** command allows you to sort a table using multiple criteria. (While you could use the **Sort** command to sort a table by a single criterion, it is easier to just double-click the column head of the column by which the table will be sorted. This sorts the data in the column in ascending order. If you double-click again on the column head, the data is sorted in descending order. A small arrow appears in the column head to denote that the table is sorted

by the data in that column, with the direction of the arrow indicating whether the sort is ascending or descending.)

When you select the **Sort** command, the system displays the Sort dialog box. The left side of the dialog box shows the columns available for sorting. The right side shows the columns currently being used for sorting. (If this is the first time you have selected this command since opening the window (or the underlying table is not already sorted), the right side will be blank. Otherwise, it will show the last sort specified.) The arrows between the columns select a column for use in sorting (rightward pointing arrow) or remove a column from being used for sorting (leftward pointing arrow).

To perform a sort for the first time, follow these steps:

1. In the left pane, click the primary column to be used for sorting.
2. Click the right-pointing arrow to move the column to the right pane.
3. The system assumes that the sort will be in ascending order. If you wish to sort in descending order, click the check box in the Ascending column. This clears the check box.
4. Repeat steps 1 - 3 for the secondary and subsequent sorts.
5. Once the columns for sorting are selected, click **OK**. The system will then sort the table by the criteria that you specified. The system will also remember how the table is sorted and will sort the table in the same way the next time you open the window.

If you wish to modify the criteria by which a table is sorted, re-display the Sort dialog box. The dialog box will then show the last sort specified. You may then modify this sort as follows:

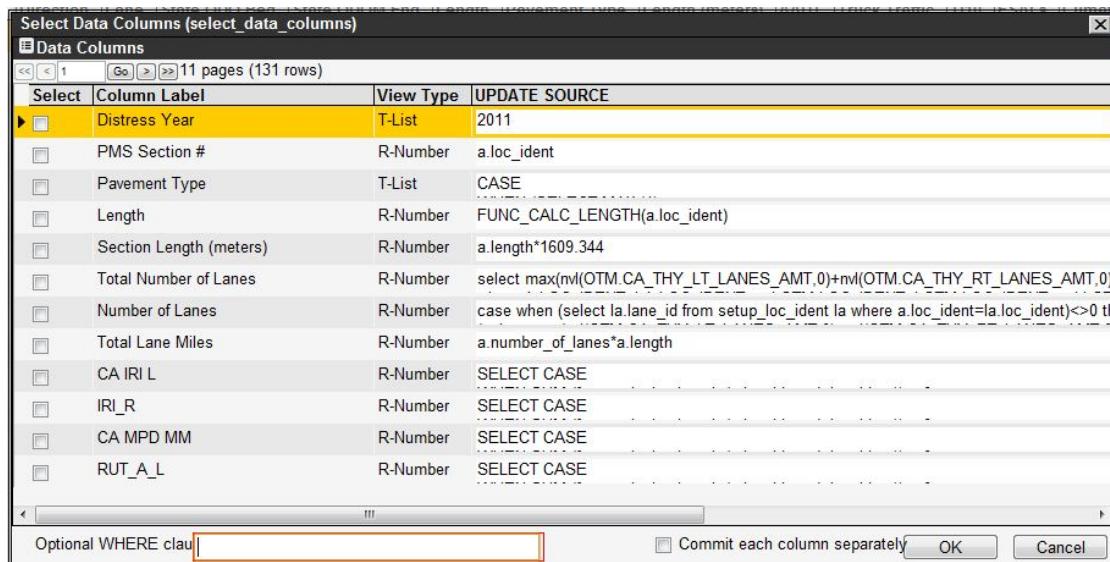
- To change the order of the sort (that is, what is the primary criterion, what is the secondary criterion, etc.), use the large arrows on the side of the dialog box. Click the row showing the sort criterion and then click the appropriate arrow to move the row up or down in the table. (The system sorts by the first, topmost row, then the second row, and so on.)
- If you wish to change the order of the data in a sort column, check (for ascending) or clear (for descending) the check box in the Ascending column for the row showing the column.
- If you wish to remove a sort criterion from the table, select the row showing the criterion in the right pane and then click the left-pointing arrow in the middle of the dialog box.
- If you wish to add a sort criterion to the table, click the column in the left pane and then click the right-pointing arrow to move it to the right pane. Note that this always adds a row to the end of the table regardless of what row you select. Therefore, after completing the row as directed above, you may need to use the large up arrow on the side of the dialog box to move the row to the proper position in the table.

### **2.7.21. Update Target Table**

Whenever a section data table in a data entry window contains columns with calculated values (and/or are derived from data in other tables), then the **Update Target Table** command is available. This command re-calculates the values (and/or retrieves the values from the source table) for existing records and non-location reference data columns. You may select which columns are updated.

After clicking **Update Target Table**, the system displays a dialog box from which you select the column(s) to update. An example of this dialog box is shown on the following page. You select the columns by clicking the check box in the Select column for the desired column. Note

that if you wish to select more than one column, hold down the CTRL key when clicking the check boxes of additional columns. (You may also select a "block" of adjacent rows by clicking the check box of the first row, holding down the SHIFT key, and then clicking the last row of the block.) Also, if you wish to select all columns, right-click and then click the **Select All** command. (Conversely, you may use the **Deselect All** command to de-select all columns.)



If desired, you may also optionally include a Where clause to further restrict what columns and/or data is updated.

Once you have selected the desired columns to be updated and entered a Where clause (if desired), are selected, click **OK**. The system then updates each selected calculated column, using the current data from which it is calculated.

#### NOTE

Only data for the currently selected data set is updated. In other words, any filters that may have been set determine what data is updated.

The section limits of the section data table are not affected by this action. No rows (sections) are added or deleted when the **Update Target Table** command is executed; only the attribute values for each section are affected.

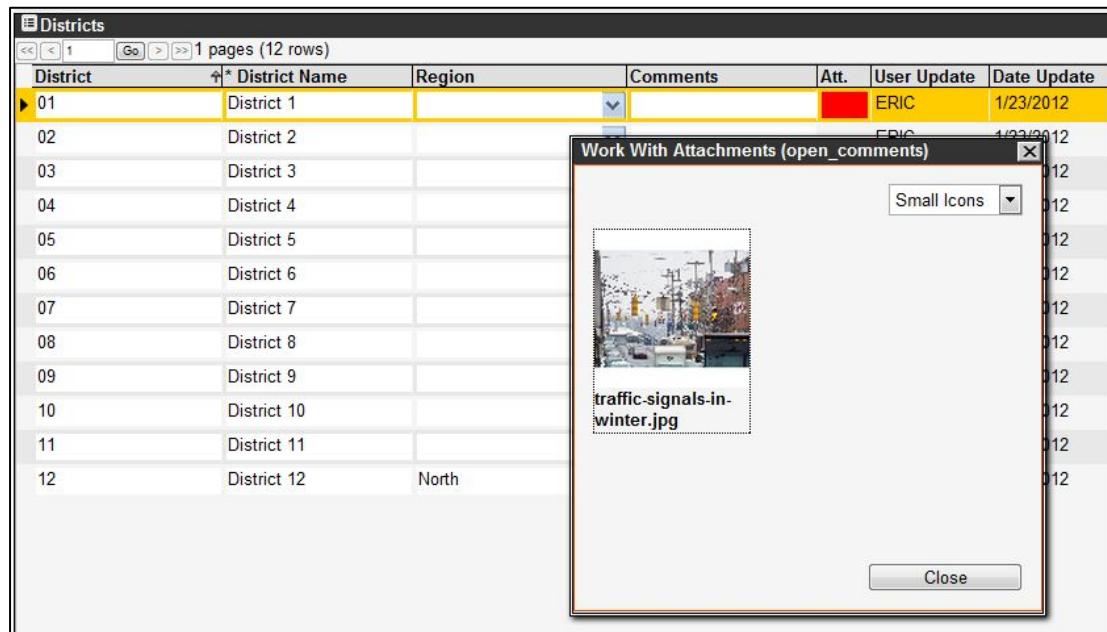
#### **2.7.22. Work with Attachments**

In some windows of the system you may want to include additional information with a record. This information may be a graphic file or a text file, and is termed an "attachment." The presence of one or more attachments is indicated in the Attach. (or Att.) column. If the background color for the cell in this column for the record is red, an attachment is present. You may double-click the red square to view thumbnail images of the attachments (or right-click the red square and then click **Work with Attachments** from the shortcut menu that is displayed). Once thumbnail images are displayed, you may double-click an image to display the attachment.

The example at the top of the next page shows a window with the Work with Attachments dialog box. Note that the selected record has a red cell in the Att. column and the thumbnail size is set to Small Icons.

The Work with Attachments window is dynamic, which means that when you change records in the underlying window, then the thumbnails in the Work with Attachments window also changes to be those of the underlying window's selected record.

To add or modify attachments, right-click the record and then click **Work with Attachments** from the shortcut menu that is displayed. This displays a dialog box that shows the file(s) that contain the attachment(s). To view an attachment, you must download the file to your computer and then open it with the appropriate application.



When you right-click a file in the dialog box (or just the dialog box itself if no files are shown), the following commands are displayed along with the common commands:

- **Insert Local File** – This command adds a single file on your computer as an attachment. (The **Insert File(s) from Server** command described below allows you to add multiple files at one time, although those files reside on your network server rather than your local computer.)

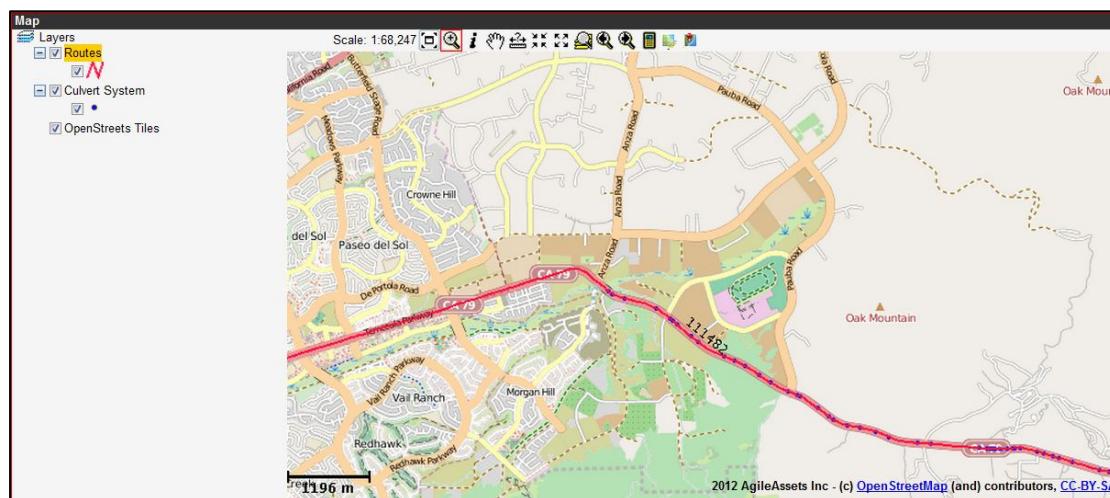
After selecting the command, the system displays the Select File dialog box. You enter the file name of the attachment in the field by clicking **Choose File** to display a second dialog box to locate the file name on your computer. You then double-click the desired file name, which closes the second dialog box and shows the file name in the Select File dialog box. Finally, click **Upload** to attach the file to the record (and show it in the Work with Attachments dialog box).

- **Update Local File** – For the selected attachment file that was obtained from your local computer, this command retrieves the most current version of the file. Other than operating on an existing attachment rather than creating a new attachment, it behaves in the same way as the **Insert File** command.
- **Insert File(s) from Server** – This command adds one or more files from the network server. When you select this command, the system displays a list of files and folders. (The files and folders are those in the root directory specified by the Attachment Root parameter that is found in the System Parameters window.) You may filter this list by using the File Filter field at the bottom left of the dialog box. Once you have located the files you wish to attach, click the check box for each file to select the files. Once all files are selected, click the **OK** button to upload the files to the AgileAssets system.

- **Update File from Server** — For the selected attachment file that was obtained from the server, this command retrieves the most current version of the file. (Although multiple files may be added at one time, only one file at a time may be updated.)
- **Insert File Names/Paths** — This command works the same as the **Insert Files from Server** command except the file is not stored on your local machine. Instead, the system uses the specified path to locate and display the attachment.
- **Update File Name/Path** — This command allows you to update the file name and path for the selected attachment.
- **Delete File** — This command removes the selected attachment file from the record.
- **Move Up** — This command moves the selected file image up one position in the list.
- **Move Down** — This command moves the selected file image down one position in the list.
- **Download File** — This command starts a process to place the selected attachment file on your computer. Once on your computer, you may use the appropriate application to view the file. (This can also be accomplished by double-clicking the file thumbnail image.)

When the Work with Attachments dialog box shows the desired attachments, click the  icon in the upper right corner to close the dialog box. If desired, you may change how the attachments are displayed by selecting a display type from the drop-down list.

## 2.8. Floating Map Window



The Floating Map window is a window that may be launched from various windows throughout the system. It appears on top of the window from which it is launched (other than the Data from Map window, which is essentially the Floating Map). It will remain on top until you close it, which allows you to switch between the underlying window and the Floating Map window to, for example, select different routes or inventory items.

When the Floating Map window opens, the underlying window determines what GIS layers are initially shown in the window. These layers are the default layers for the window and cannot be altered or removed. You may, if desired, add additional GIS layers (from your library of GIS layers) to the default layers by utilizing the **Open Map** or **Add Layer** command found on the right-click shortcut menu. The system will remember what additional layers you add and will display them the next time the Floating Map window launches. (Every user thus has his or her set of maps that are customized for particular data windows.)

The Floating Map window contains two panes: the left pane (Layers) shows the various GIS layers that comprise the map, which is shown in the right pane. These two panes are described in more detail in the following sections.

### **2.8.1. Layers Pane**

The Layers pane on the left side of the Floating Map window shows the layers associated with the selected data source in the Manage GIS Data Sources window (see page 173). The order of the layers in this pane is the order of the layers in the Map pane (that is, the topmost layer listed is the top layer in the map). You may change the order of the layers by clicking a layer to select it, hold down the mouse key, drag the layer to the desired position, and finally release the mouse key.

When you right-click this pane, a shortcut menu is displayed. This menu contains the common commands along with the following special commands:

- **Edit Layer Properties** – This command displays the Layer Properties dialog box, which you use to modify how the selected layer is shown on the map. (Note: Instead of using this command you may also double-click the layer name or the symbol used for the layer to display the Layer Properties dialog box.) See page 37 for more information on the Layer Properties dialog box.
- **Save as Layer** – After modifying a layer, this command allows you to save the modifications. It displays the Save Layer dialog box, where you may name the layer and indicate whether it is a public layer (that is, a layer that is available to all authorized users). A saved layer can then be re-used by you (or anyone if it is public) in any Floating Map window by using the **Add Layer** command.
- **Add Layer** – This command displays the Layer Selection dialog box. This dialog box contains those layers that you previously saved as well as those marked as public. See page 41 for more information on the Layer Selection dialog box.
- **Remove Layer** – This command deletes the layer from the map and the list of layers. (Note: If you only wish to remove the layer from the map, but keep the layer available in the list of layers, clear the check box beside the name of the layer.)
- **Copy Layer** – This command copies the existing layer to the clipboard.
- **Paste Layer** – This command copies the layer on the clipboard to the Floating Map window.
- **Show Labels** – This command places labels on the map to identify the elements of the selected layer. The formatting of the labels is accomplished via the Layer Properties dialog box. See page 37 for more information on the Layer Properties dialog box.
- **Hide Labels** – This command removes labels from the map.
- **Show Attributes...** – This command displays the inventory items of the active layer.
- **Set Layer Active** – This command is only available for those layers that are associated with an inventory table. When available, the command makes the selected layer active. This means that the Select Features icon () may be used to extract and display data from the active layer.
- **Zoom to Layer** – This command changes the magnification of the map so the entire extent of the layer is displayed. (The maximum extent of the layer is defined in the Full Extent column of the Manage GIS Data Sources window. See page 173 for more information on the Manage GIS Data Sources window.)

- **Save Map** — This command allows you to save the currently displayed map. When you select this command, the system displays the Map Selection dialog box. See page 42 for more information on this dialog box.

You enter a name for the map in the Name field and then click the **OK** button. The system then saves the new map under the entered name within the My Maps folder. (Note: If you would prefer that the map be a "public" map [that is, available to all users], after saving the map navigate to the Manage GIS Layers window and select the Is Public check box for the map you just saved. See page 175 for further information on the Manage GIS Layers window.)

- **Save Map as PDF** — This command saves the map in a PDF file and (provided your computer is equipped with the Adobe application) displays the PDF file.
- **Open Map** — This command displays the Map Selection dialog box. Select the desired map and then click the **OK** button. The system then closes the dialog box and displays the selected map in the Floating Map window. See page 42 for more information on the Map Selection dialog box.
- **New Map** — This command removes all layers from the map other than those generated by the underlying window.

### 2.8.2. Map Pane

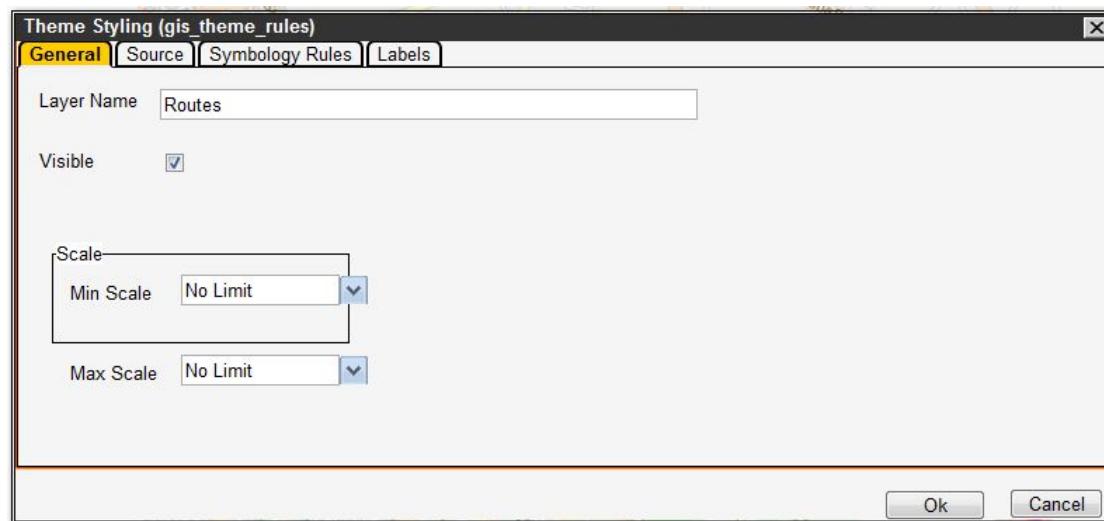
The Map pane shows an area of your transportation network with the layers selected in the Layers pane displayed. Icons are available to manipulate the map image:

-  **Close Layers Pane** — This icon closes the Layers pane to provide more room for the display of the map.
-  **Open Layers Pane** — This icon opens the Layers pane to show the list of layers.
-  **Enlarge Area** — This icon allows you to enlarge an area of the map. After selecting this icon, draw a rectangle around the area you wish to enlarge by placing the cursor at one corner of the rectangle, holding down the mouse key, and dragging the cursor to the diagonally opposite corner. A rectangle appears as you drag the cursor. When the rectangle encompasses the desired area, release the mouse key. When you release the mouse key, the system enlarges the selected area to fit in the pane.
-  **Select Features** — This icon displays information from the data table associated with the active layer. After selecting this icon, draw a rectangle around the area of interest (as described above). When you release the mouse key, the system displays a new window with inventory records. The records in the table will be only those records with data elements that are within, or intersecting, the drawn rectangle.
-  **Select Features (Polygon)** — This icon works like the Select Features icon described above. The difference is that instead of defining a rectangle by dragging, you define a series of points that describe a polygon that encloses the area of interest. You click to set each "corner" of the polygon and double-click to set the last point of the polygon. After double-clicking, the new window with inventory information is displayed.
-  **Identify** — When this icon is selected and you click an element on the map, the application displays the label of the element you clicked.

- ⑦ **Pan** – This icon allows you to move the map so a different area of the entire map is displayed. To use this icon, click the icon and then click and hold down the mouse key while dragging the map as desired. When you release the mouse key, the system will fill in the "blank areas" resulting from dragging the map.
- ⑧ **Calculate Distance** – This icon allows you to measure the distance between two points. After selecting the icon, click the starting point for measurement. Then drag the cursor to the ending point for measurement (as you drag, a red line will be displayed to indicate what is being measured) and click again. The application will then display the distance between the two points in a dialog box. The units for the measurement may be selected in the dialog box.
- ⑨ **Zoom In** – This icon allows you to zoom in on an area of the map. The application zooms in a fixed amount each time you click the icon. You may also use the scroll button on the mouse.
- ⑩ **Zoom Out** – This icon allows you to zoom out from an area of the map. The application zooms out a fixed amount each time you click the icon. You may also use the scroll button on the mouse.
- ⑪ **Full Extent** – This icon restores the map to its full extent.
- ⑫ **Previous Extent** – This icon displays the previous zoom level. Note that the arrow in the magnifying glass changes in size depending on whether the previous zoom level is an enlargement or a reduction. The arrow is large if it is an enlargement (zoom in); the arrow is small if it is a reduction (zoom out).
- ⑬ **Next Extent** – After displaying a previous extent, this icon displays the former zoom level. Note that the arrow in the magnifying glass changes in size depending on whether the previous zoom level is an enlargement or a reduction. The arrow is large if it is an enlargement (zoom in); the arrow is small if it is a reduction (zoom out).
- ⑭ **Track Location by GPS** – When your version of the system includes the Online GPS Functionality (OGF) feature and the GpsGate Client application is running, this icon is available. When you select it, and provided a GPS signal is present (which is indicated by a red X appearing on the icon), the system will show your location on the map and will update this location as you move. Your location is indicated by a green plus sign (+).

Note: GpsGate Client is freeware that is available from Franson Technology AB. If you do not have it on your machine, the system will ask if you wish to install it when you click the icon. See your System Administrator for more information.
- ⑮ **Set Default Location** – This icon causes the system to remember the currently displayed location. You may then return to this location by clicking the icon that is to the right of this icon.
- ⑯ **Go to Default Location** – This icon causes the system to re-draw the map so the default location is displayed.

### 2.8.3. Layer Properties Dialog Box



You display the Layer Properties dialog box by right-clicking a layer and then clicking **Edit Layer Properties** in the shortcut menu that is displayed. This dialog box allows you to edit the various styling attributes for line, polygon, and point types of GIS data.

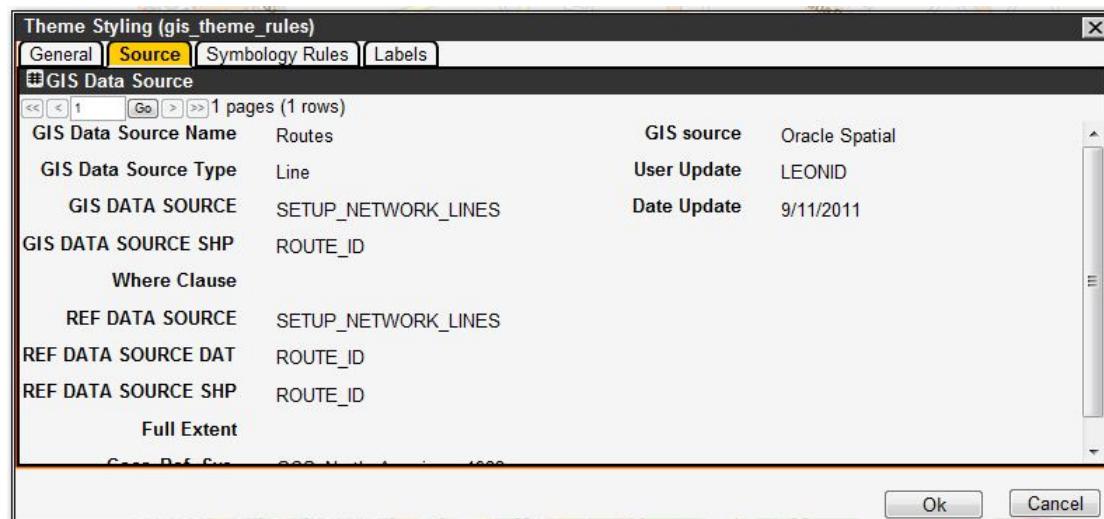
The dialog box contains four tabs: General, Source, Symbology Rules, and Labels. These tabs are described in more detail in the following sections.

#### ***The General Tab***

The General tab provides the following fields to set the general properties of the layer:

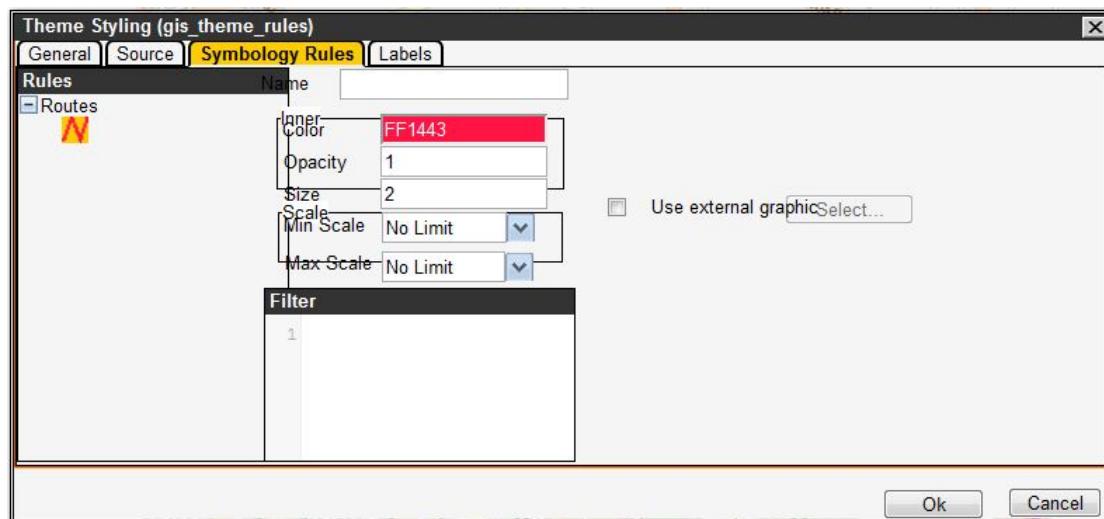
- **Layer Name** – This field shows the name of the layer as it appears in the left pane of the Floating Map window.
- **Visible** – When this check box is selected, the layer will be displayed (within the limits set by the minimum and maximum scale values, if any). (Note: This is the same check box that appears next to the layer name and layer symbol in the list of layers in the left pane, and so you can also turn a layer on or off directly in the Layers pane.)
- **Min Scale** – This field sets the lower scale limit. When the map scale falls below this limit, the layer will not be displayed. If the No Limit item is selected from the drop-down list, the layer is always displayed.
- **Max Scale** – This field sets the upper scale limit. When the map scale exceeds this limit, the layer will not be displayed. If the No Limit item is selected from the drop-down list, the layer is always displayed.

### The Source Tab



The Source tab is read-only. It shows information about the data source that is associated with the layer. This information is taken from the Manage GIS Data Sources window. If you need to modify any of the displayed information, navigate to that window.

### The Symbology Rules Tab



The Symbology Rules tab is composed of two panes. The Rules pane on the left shows the symbols used to represent layer data on the map. For the symbol selected in the left pane, the right pane provides fields to format the symbol.

When you right-click the Rules pane, a shortcut menu is displayed with commands for configuring how the layer is divided into "sub-layers" based on the values of a column. (For example, you could use a GIS layer that contains routes and then have a different sub-layer for each route type [interstate, primary, and secondary]. These sub-layers may be turned on or off and formatted just like layers.) These commands are described below:

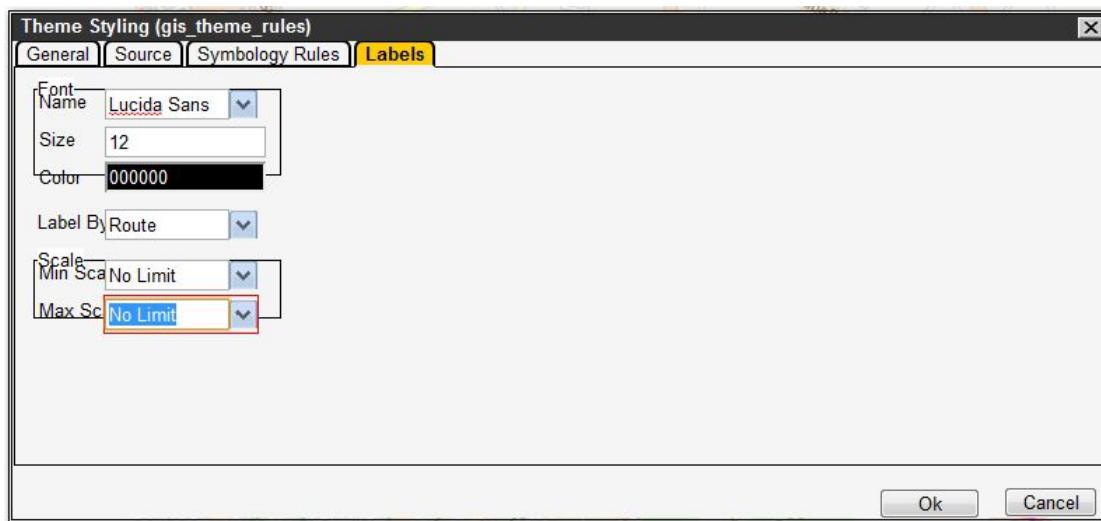
- **Set Rules Automatically** – With this command you select a column that will be used to divide the GIS data into groups and then the system automatically creates sub-layers for each distinct value in the column. (For numeric columns, the system determines the range of the values in the column and then creates three, roughly equal groups – so only three sub-layers are created for these types of columns.) Once the sub-layers are created, you may edit them as you would any other layer.

- **Reset to Default** – This command restores the layer to its default settings. Any modifications to the layer (such as sub-layers) are lost.
- **Add Rule** – This command allows you to manually specify how the values of a column are to be grouped to form sub-layers.
- **Delete Rule** – This command deletes the selected sub-layer.

The fields in the right pane vary depending on the type of GIS data (point, polygon, or line) represented by the symbol selected in the Rules pane. The following table describes the fields that are available and notes the type of GIS data that causes the field to be displayed.

Field Name	Type of GIS Data	Description
Name	All	This is the label that appears next to the GIS data icon.
Border Size, Color	Polygon	These two fields define the color and size (in pixels) of the border for polygon types of GIS data. You may specify a color by directly entering a color code or by selecting a color from the color palette that is displayed by clicking the field.
Inner Size	Line, Point	For point types of GIS data, this field defines the size (in pixels) of the icon. For line types of GIS data, it defines the width (in pixels) of the line. Note: The size does not vary with the scale of the map. Note: This field is hidden when the Use External Graphic check box is selected.
Inner Color	All	This field defines the color of the line, polygon, or point. You may specify a color by directly entering a color code or by selecting a color from the color palette that is displayed by clicking the field. Note: This field is hidden when the Use External Graphic check box is selected.
Scale Min, Max	All	When the scale of the map is within the thresholds set by these two fields, the settings of the Inner, Border, and Shape fields are in effect.
Filter	All	The Filter field allows you to configure a filter to determine what GIS data is displayed on the map. When you right-click this field, a shortcut menu appears with an Edit command, which displays the Filter dialog box (see page 20).
Shape	Point	This field contains a drop-down list of images that may be used to denote a point in a point type of GIS data representation. Note: This field is hidden when the Use External Graphic check box is selected.
Use External Graphic check box	Line, Point	This check box allows you to use an external image file for the icon that denotes points and lines. The available external image files are stored as attachments in the GIS Images window (see page 175). When you click the check box, the system displays the available image files for the GIS data type (either line or point). You then click the desired image to select it and finally click the Select button to place the image in the Layer Styling dialog box. Note: Once the check box is selected, you may use the <b>Select</b> button in the dialog box to select a different image file. Tip: Since the size of icon does not vary with the scale of the map, you may want to create multiple image files of an icon, with each file containing a different size of the icon. You would then use the Scale fields to display different sizes of the icon (that is, different image files) depending on the scale of the map.

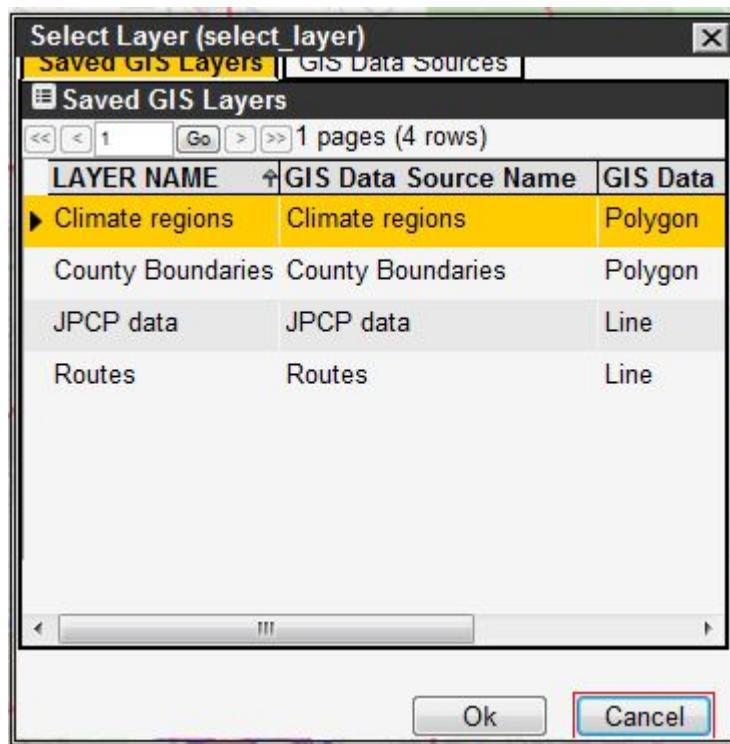
### The Labels Tab



The Labels tab provides fields so you may configure the appearance of the labels shown with the GIS data icons as well as the information displayed in the label. (The labels are displayed by using the **Show Labels** command that is found in the shortcut menu that is displayed by right-clicking in the Layers pane of the Floating Map window.) These fields are available for all types of GIS data and are described below:

- Font Name — This field shows the font used for lettering.
- Font Size — This field shows the size of the font used for lettering. The size does not vary with the scale of the map.
- Font Color — This field shows the color of the font. You may enter a color code directly or you may select a color from the color palette that is displayed by clicking the field.
- Label by — This field contains a drop-down list of the columns in the table from which GIS data is taken. By selecting a column you configure what information is displayed in the label.
- Scale Min, Max — These fields are thresholds that determine at what map scales the label will be displayed.

#### 2.8.4. Layer Selection Dialog Box



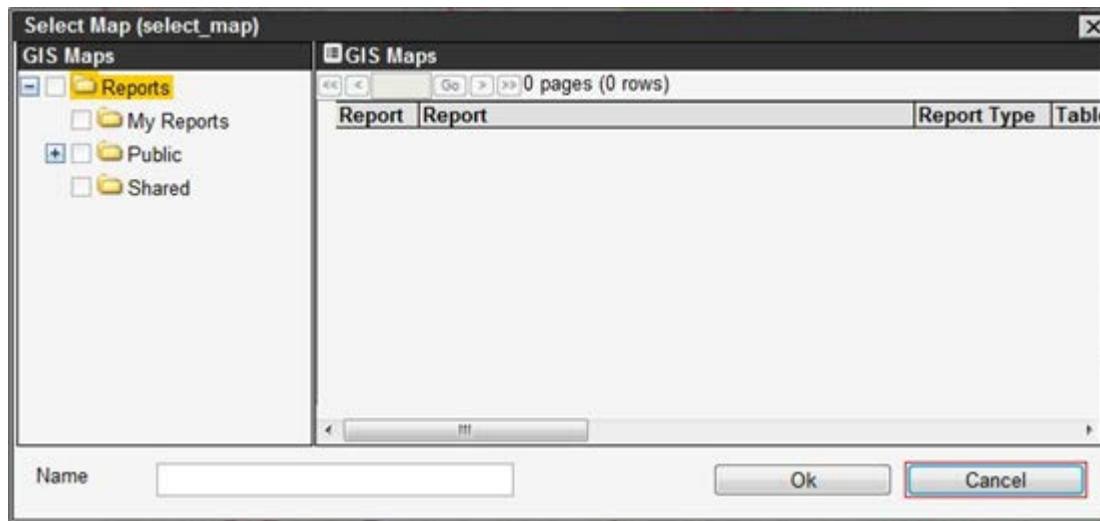
The Layer Selection dialog box appears when you execute the **Add Layer** command that is found on the shortcut menu that is displayed by right-clicking the Layers pane. This dialog box essentially contains the "library" of layers that may be added to the map.

This dialog box contains two tabs: Saved GIS Layers and GIS Data Sources. These tabs are described below:

- The Saved GIS Layers tab shows those layers that you previously saved from this or any other window in the system. It also shows those layers saved by other users if the user marked the layer as a "public" layer when the user saved the layer.
- The GIS Data Sources tab shows the data sources that are saved in the Manage GIS Data Sources window (see page 173). When you select a data source, the system applies the default styling to the source to create a layer and then displays the layer in the Floating Map window.

After you find and select the desired layer, click the **OK** button to close the dialog box and show the layer in the Floating Map window.

### 2.8.5. Map Selection Dialog Box



A "map" may be thought of as a collection of GIS layers. In addition to saving and selecting individual layers via the Layer Selection dialog box that was described in the previous section, you may also save and select maps (that is, multiple layers) via the Map Selection dialog box.

The Map Selection dialog box appears after you execute the **Save Map** or **Open Map** commands. This dialog box contains two panes and mimics the functionality found in the Reports Management window.

The GIS Maps pane on the left shows how maps may be categorized: maps solely for your use in the My Maps folder; maps that are available to all authorized users in the Public folder; and maps that were specifically shared amongst certain users in the Shared folder.

The GIS Maps pane on the right lists all available maps without regard to category and provides more detailed information for maps. When a map is selected in the left pane, the corresponding record is also selected in the right pane.

The Name field shown at the bottom of the dialog box is used when saving a map.

## 2.9. Overview of User-controlled Tables, Windows, and Calculations

This topic presents an overview of the tables, windows, and calculations that may be controlled by users.

Speaking very broadly, the system supports the following types of tables:

- Strictly location-referenced data tables.

In this type of table, the data is identified strictly by location (route with "from" and "to" point boundaries). An example of location-referenced data is road surface distress data that is collected by route and start/end mile point.

This type of table and the windows associated with it are created via the Tables window, which is found in the Utilities menu of the System module. See page 128 for more information on the Tables window.

The data in location-referenced data tables is displayed in two types of windows: data view windows (see Section 2.9.1 on page 43) and graph view windows (see Section 2.9.2 on page 44). The data view window shows the data in a tabular format, while the graph view window shows the data along a section of road.

- Inventory data tables.

In this type of table, the data is primarily identified by its asset ID number, but may also have location associated with it. An example of inventory data without location data is a data table for radios where each radio has its own identification number. An example of inventory data with location data is a data table for intersections in your network where each intersection is identified by intersection name -- but also includes a secondary reference by location (that is, the route and mile point where the intersection is located).

This type of table is first created in the Asset Type window by specifying the name of the inventory table and then the windows associated with it are created via the Tables window. The Asset Type window is described in more detail on page 164, while the Tables window is described on page 128.

Inventory data tables are displayed via a data view window specifically for inventory that is supported by status and class code windows. (If the implementation also includes a maintenance management system, two additional windows for periodic maintenance are also associated with the inventory data table.)

- List data tables.

In this type of table, the data does not have any location associated with it; each record is identified by an internal ID number. The information in list data tables is typically used in drop-down lists throughout the system, and enhances the information provided by a record. See Section 2.9.3 on page 46 for more information.

This type of table and the windows associated with it are also created via the Tables window; see page 128 for more information.

- Generic data tables.

These are tables that are defined by the user in the Tables window. They are used for a wide variety of purposes specific to a particular implementation.

- Derived data tables.

In this type of table, all data values are calculated from other data sources. For example, all records and information in the Pavement Structure window comes from the data shown in the Construction History window.

Records are inserted into this type of window in one of the following three ways (with a particular method selected during system implementation): by system job; by dynamic segmentation; or by finest partition.

The order of operations for these types of tables is that first records are inserted and then the data values are calculated by executing the **Update Target Table** right-click command.

The windows used for derived data tables are those used for location-referenced data tables, which are described above.

### **2.9.1. Data View of Location-referenced Data**

The Data View shows data from a location-referenced data table in a tabular format. You may choose to display the data in "grid view" or in "record view."

In grid view, all records are shown as a table (subject to any filtering that may be active). In record view, a single record is shown and you may navigate from record to record using the navigation arrows below the title bar. (Record view is often used when a large number of columns exist in the table and you would like to see all of them at one time in the window.)

Shown below is a grid view of a data table. Following that example is the same data in a record view.

Grid View Example:

Effective Date	Year	Route	Direction	Lane	State ODO Beg	State ODOM End	APT area	LOC_cnt	TVC_cnt	XJC_len	District	CK3 TF CNT	CK1 CNT	Corner Cracks	CA APCS JF
6/9/2011	2011	113L	Desc.	1	22.125	22.136	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.139	22.19	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.194	22.247	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.25	22.303	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.306	22.357	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.36	22.414	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.416	22.468	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.47	22.522	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.524	22.575	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.578	22.629	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.632	22.684	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.686	22.737	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.74	22.791	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.795	22.845	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.848	22.866	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.87	22.919	0	0	0	0		0	0	0	0
6/9/2011	2011	113L	Desc.	1	22.921	22.972	0	0	0	0		0	0	0	0

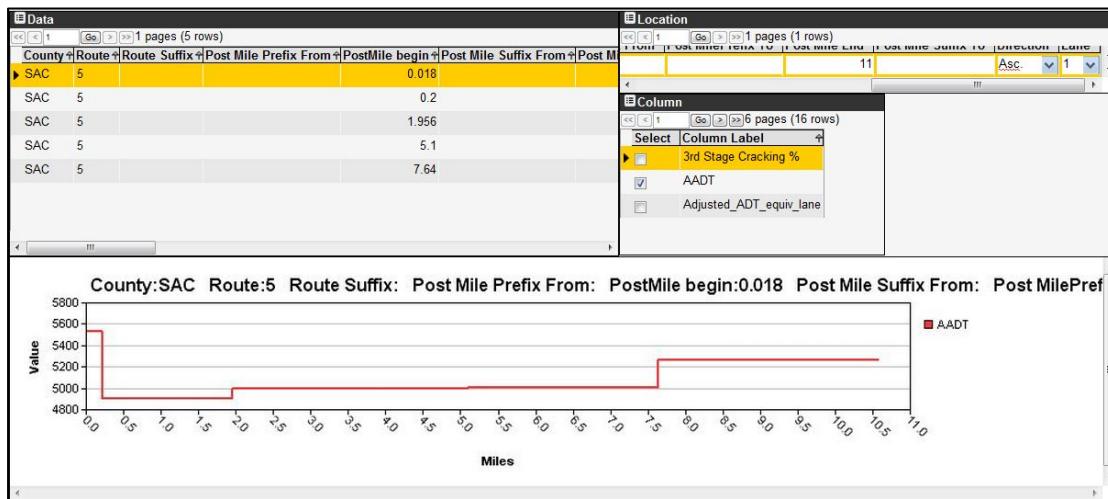
Record View Example:

Effective Date	Year	Route	Direction	Lane	State ODO Beg	State ODOM End	APT_area	LOC_cnt	TVC_cnt	XJC_len	District	CK3 TF CNT	CK1 CNT	Corner Cracks	CA APCS JF
6/9/2011	2011	113L	Desc.	1	22.125	22.136	0	0	0	0		0	0	0	0
					JPC LOC Seal		0				Cattle	<input type="checkbox"/>			
					JPC LOC SPALL		0				Climate Region				
					CA APCS JPC TVC NARF		0				Constr	<input type="checkbox"/>			
					CA APCS JPC TVC SEAL		0				DSBegin		17700		
					CA APCS JPC TVC SPAL		0				DSEnd		17725		
					CA APCS JPC XJC NARF		0				Filename	16903800			
					CA APCS JPC XJC SEAL		0				LaneDev	<input type="checkbox"/>			
					CA APCS JPC XJC SPAL		0				APCS Lanetype	JPC			
					CA APCS LJC NARR		0				Length Rated (m)		24.5		
					CA APCS LJC SEAL		0				Lowspeed	<input type="checkbox"/>			
					CA APCS LJC SPALL		0				Pavewidth		0		
					CA APCS TJC NARR		0				Rinfo	NA			
					CA APCS TJC SEAL		6				RRX	<input type="checkbox"/>			
					CA APCS TJC SPALL		0				Shld_Type	FLX			
					Comments		0				Shld_Wid	N			
					Att.		0				Direction	SB			
					User Update		0				Elevation		12.9		
					Date Update		0				Latitude	38.5349229			

When grid view is shown, the icon will change the grid view to a record view. The icon will then change to this icon , which will change the record view back to a grid view.

### 2.9.2. Graph View of Location-referenced Data

The Graph View is also known as the graph-down-the-road view. It shows data from a section data table (such as traffic) along a particular road section. An example of a Graph View is shown in the example below (this is the Network Master Graph window).



The following table summarizes what records are graphed depending on the selections you make for the Direction and Lane columns. For example, setting the Direction column to Increasing and the Lane column to All will result in all records where Direction = All or Increasing AND Lane = All or 1 being graphed.

Selection for Direction Column	Selection for Lane Column	What Records Are Displayed (Direction Value, Lane Value)
All	All	(All, All), (Increasing, All), (Increasing, 1)
Increasing	All	(All, All), (Increasing, All), (Increasing, 1)
Decreasing	All	(All, All), (Decreasing, All), (Decreasing, 1)
Increasing	{Lane Number}	(All, All), (Increasing, All), (Increasing, {Lane Number})
Decreasing	{Lane Number}	(All, All), (Decreasing, All), (Decreasing, {Lane Number})

### Description of the Graph View Window

The window used for the Graph View contains the following elements:

- Data pane — This pane is located on the upper left side of the window. It lists distress or traffic data for data collection sections limits.
- Route pane — This pane is located in the upper right corner of the window. It shows the available routes in a drop-down list. You use this pane to select the route, lane, direction, and start and end mile points for which data will be displayed. If you right-click this pane, a shortcut menu appears with the following commands:
  - Show Data and Graph — This command displays the graph in the lower pane as well as data in the Data pane.
  - Reset Mile Points to Entire Route — This command sets the Start and End mile points to be the entire route.
- Measure/Index pane — This pane is located in the middle of the window, between the Data and Year panes. It selects the data to be displayed on the graph. The variables available in this pane are configured in the Setup Data Tables window.
- Year pane — This pane is located in the middle right side of the window. It selects the years for data displayed on the graph.
- Graph pane — This pane is located at the bottom of the window. It displays a curve for each combination of index and year selected in the Measure/Index and Year panes for the route selected in the Route pane. (For example, if you selected two distress indices and three years, the Graph pane would show six curves.)

If would like to change how the graph displays, right-click and then click **Change Graph Properties**.

### How to Display a Graph

To display a graph:

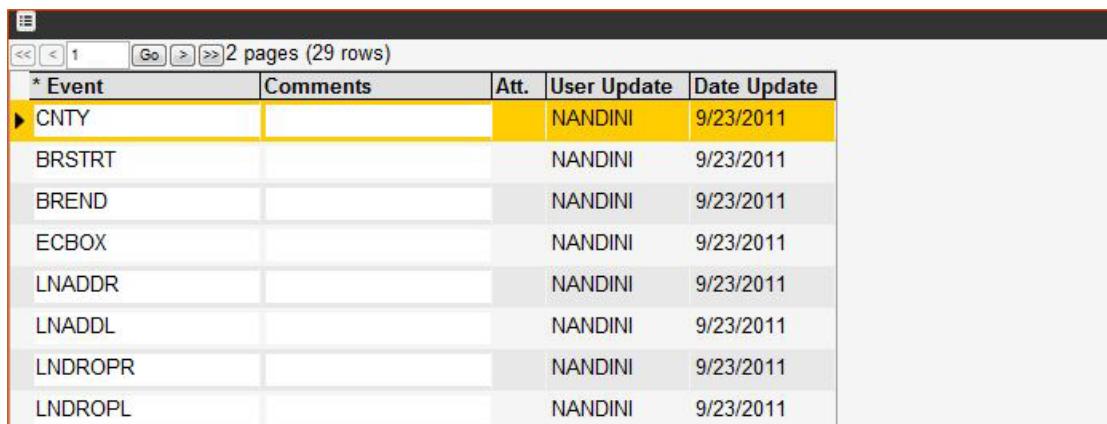
1. In the Route pane, enter the route of interest by typing the route in the Route column (or select the route from the drop-down list that is displayed by clicking the down arrow). If you want to see only part of the route, adjust the mile points and/or lanes and/or direction to show the desired part of the route.

Note: Since data will only be displayed for routes that belong to the administrative unit you selected when you logged on, only routes assigned to that administrative unit are shown in the drop-down list.

2. In the Measure/Index pane, select the index or indices you wish to graph by clicking the check box beside each index.
3. In the Year pane, select the year or years you wish to graph by clicking the check box beside each index.
4. In the Route pane, right-click and then click **Show Data and Graph**. The system then draws curves for each combination of index and year selected in the Measure/Index and Year panes for the selected route. It also displays data in the Data pane.

#### **2.9.3. List Data Windows**

The data from list data tables is generally displayed in a simple window that shows the values that appear in a particular drop-down list. These are generally found in the Setup menus of the various modules. An example of a list data window is shown below.



* Event	Comments	Att.	User Update	Date Update
CNTY		NANDINI	9/23/2011	
BRSTRRT		NANDINI	9/23/2011	
BREND		NANDINI	9/23/2011	
ECBOX		NANDINI	9/23/2011	
LNADDR		NANDINI	9/23/2011	
LNADDL		NANDINI	9/23/2011	
LNDROPR		NANDINI	9/23/2011	
LNDROPL		NANDINI	9/23/2011	

As with location-referenced data tables, you may display the data in grid view or record view.

#### **2.9.4. Table-specific Windows**

Data from any table may be displayed in a simple user-defined window showing all rows and columns for that table. These windows may be found anywhere throughout the system. As with location-referenced data tables and list data tables, you may choose to display the data in grid view or record view.

#### **2.9.5. "As-of Date" Windows**

An "as-of" date window shows data as it was at any particular past time. The user enters a date and time at the top of the window and then the data as of that date appears in the window. This window is read-only because past-data is fixed. Maps showing the data with the same as-of date are also available within this window. Any data can have a corresponding "as of" date window.

The creation of this type of window as well as the ability to retain temporal data (from which this window derives its data) is accomplished in the Tables window (see page 128).

#### **2.9.6. Record Selection**

You may select multiple records in data and list tables. These records may be selected as a block by clicking the first record of the block, holding down the SHIFT key, and then clicking the last record in the block. You may select non-adjacent records by holding down the CTRL



key and clicking each record that you want to select. The selected records are highlighted in color to denote that they are selected.

Within the selected records, one particular record is "active." This active record is indicated by a right-pointing triangle in the leftmost column of the displayed table (see the previous page for an example). (Note: While it is possible to have an active record that is not part of a set of selected records, the moment you right-click the active record to execute a right-click command, it becomes both active and selected.)

This distinction between selected records and the active record becomes important when executing commands. While you may have multiple records selected, a command that only operates on a single record (such as **Delete**) will only affect the active record -- not all selected records. Conversely, a command that operates on multiple records (such as **Massive Update**), affects all selected records (including the active record), rather than just solely the active record.

## 2.10. Software Use via the Keyboard

You may navigate through AgileAssets software solely via the keyboard (that is, without using the mouse or other pointing device). This section describes how to use keyboard keys to navigate within the software.

In this discussion, we talk of where the "focus" is. This means what section or object in the window is active. Focus is indicated in a variety of ways. It may be a dotted line around the name of a module or a drop-down field; it may be a red square around a pane within a window; or it may be where a field or object is highlighted. You should pay careful attention to what is shown on the screen as you press keys to navigate to the desired object.

Note: You may also read the message line in the lower left corner of the browser window to help you discover the object on which the focus is placed.

The navigation pathway is circular. If you keep pressing the Tab key (or Shift-Tab keys), you will eventually return to where you started. For example, if the focus is on the left gutter, tabbing moves the focus through the gutter, then to the browser, then to the modules and menus of the AgileAssets application itself, and finally back to the left gutter.

### 2.10.1. Left Gutter Navigation

The left gutter contains icons for performing common tasks (such as saving or opening Help), Quick Links to open windows you commonly use, and various drop-down list fields and check boxes.

To navigate to the left gutter, press the Tab key until the focus shifts to the icons. The focus is indicated by an icon becoming highlighted in the color of the selected color scheme. When the focus is on the desired icon, press the Enter key to select the icon.

After tabbing through the icons, the focus then moves to the Quick Links (if any). As with the icons, once the focus is on the desired text portion of the Quick Link, press the Enter key to select the Quick Link.

#### NOTE

If the focus is on the icon next to the text portion of the Quick Link, pressing Enter will delete the Quick Link rather than open the window associated with the Quick Link.

After the Quick Links section of the left gutter, further tabbing will move the focus to the drop-down list fields. Focus for one of these fields is indicated by the field becoming

highlighted. Once the focus is on the desired field, press the down arrow key to step through the drop-down list for the field. (You may also use the up arrow key to step in the opposite direction in the list. You cannot "wrap-around" from the beginning of the list to the end or vice versa -- so you may need to use both keys to find the desired list value.)

For check boxes, press the Tab key until the check box is a finely dotted line. Then press the space bar to place a check mark in the check box (or clear the check box). Finally, press the Enter key to activate the new state of the check box.

If you wish to close the left gutter (or, conversely, open it), slowly press the Tab key while keeping an eye on the message line in the lower left corner of the browser window. When the message line shows javascript:OnCollapse(), press the Enter key to open or close the left gutter.

#### **2.10.2. Menu Navigation**

You move the focus between the different tabs of the application (which correspond to the different modules of the application) by pressing the Tab key to move left to right. (You may also move right to left by pressing the Shift+Tab keys.) The focus is indicated by a heavy dotted line around a tab.

Once the focus is on the desired tab, press the Enter key to select the module. The system displays the dashboard for the new module and places the focus on the Utilities menu within the module. The focus is indicated by a square in the same color as the color scheme.

To navigate between the different menus in the module, press the right arrow key. This moves the focus to the right. (Once the focus is past the Utilities menu, you may use the left arrow key to move the focus to the left. The focus will not "wrap-around" from the Utilities menu to the Reports menu or vice versa.)

Once the focus is on the desired menu, press the down arrow key to move to the first menu item within the selected menu. You may then use the up and down arrow keys to place the focus on the desired menu item. (For those menu items that lead to submenu items, use the right arrow key to move to the submenu and then use the up and down arrow keys to move within the submenu. You may return to the previous menu by pressing the left arrow key.) Once the focus is on the desired menu item, press the Enter key to open the window associated with the menu item.

#### **NOTE**

Once the menu items under a main menu are displayed, you may close these items and return to navigating within the main menu by pressing the Shift+Tab keys. (If the focus is currently in a submenu, you may need to press Shift+Tab multiple times.)

#### **2.10.3. Navigation Within a Window**

Once a window is open, the focus within the window is indicated by a red square (regardless of the color scheme chosen).

For those panes that show tabs, press the Tab key (or Shift-Tab keys) to move the focus until the red square is around the entire pane showing the tabs. Then use the right and left arrow keys to step between the different tabs within the pane.

When the desired pane is displayed and you have placed the focus on it (by pressing the Tab key), press Ctrl+Y to activate the pane. You may then manipulate the records and data in the pane using the following techniques:

- To switch between Form View (a single record) and Grid View (multiple records), press Ctrl+G.

- To move from record to record, press the up or down arrow keys. The record on which the focus rests is highlighted in the color of the selected color scheme.
- If the pane shows multiple pages of records, press the Page Down key to move to the next page of records. To return to a previous page, use the Page Up key.
- Once the desired record is highlighted, press the Tab key to move the focus from column to column within the record. (The focus only moves between editable fields.) You may move the focus in the opposite direction by pressing the Shift+Tab keys.
- For fields that show values, when the focus is on the field you may press the Delete key to remove the current data and then you can type the new data. Alternately, you may use the right and left arrow keys to step through the characters of the data, press the Backspace key to remove one or more characters, and then type the new characters.
- For drop-down list fields, when the focus is on the field begin typing the characters that will appear in the field. The system will display all items from the list that match what you have typed as you type. You may continue typing the remaining characters or you may use the down arrow key to step through the displayed list items until the desired item is highlighted. Once the desired value is shown in the field, press the Tab key to select the value and move the focus to the next editable field.
- For check boxes, when the focus is on the field press the space bar to change the state of the check box. After pressing the space bar, press the Tab key to keep the new check box state.
- To display the shortcut menu for the pane, press Ctrl+M. You may then use the up and down arrow keys to highlight the desired command in the shortcut menu. Once the desired command is highlighted, press the Enter key to execute the command. (You may close the shortcut menu without selecting a command by pressing the ESC key.)
- To close a dialog box or other pop-up window, press Ctrl+S.

The above techniques are for panes that show tabular data. For panes that show tree views, the techniques are similar:

- The up and down arrow keys move the focus between nodes at the same level.
- The left arrow key moves the focus up one level in the hierarchy. Conversely, the right arrow key moves the focus down one level in the hierarchy.
- To expand a branch of a hierarchy, press the Enter key. To collapse the branch, press the Shift+Enter keys.
- To select a node, press the space bar. If you press the space bar again, the node is de-selected.

When you are finished working in a pane, press CTRL+U to de-activate the pane and return to navigating from pane to pane with the Tab key. You may also return to menu selection by pressing CTRL+, (Control+Comma), which places the focus on the Utilities menu.

To save data, press Ctrl+>.

#### **2.10.4. Tab Order**

The principal means of moving the focus via the keyboard is by pressing the Tab key (or pressing the Shift+Tab keys to move the focus in the opposite direction). The tab order determines the order in which the focus moves from object to object. The following points may be helpful in understanding the tab order:

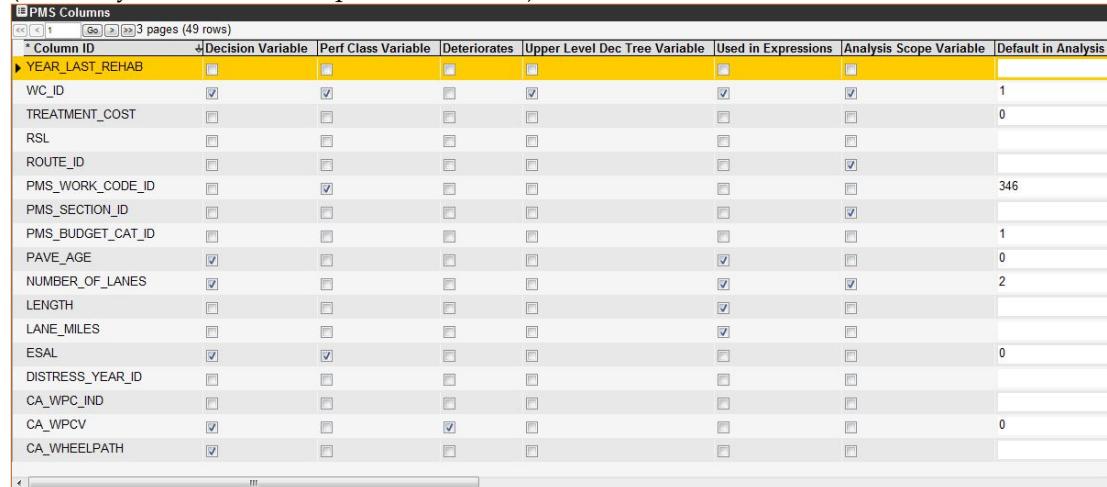
- The tab order for browser objects, left gutter objects, and panes is preset and fixed.
- The tab order for main menu items follows the order in which they are seen on the screen (as per the Order column in the Menu window).
- The tab order for columns within a pane follow the order as set in the Order column, which is found in the bottom pane of the Data Window tab of the Change Control Properties dialog box.
- When setting values for tab order, the order runs from the lowest non-zero integer value to the highest. A value of zero forces the focus to go to this column last. A value of negative one (-1) forces the focus to skip the column.

## 3. Roadway Module

The Roadway module provides the features and data structures needed to model and manage your pavement network.

### 3.1. Setup PMS Columns

(Roadway > Utilities > Setup PMS Columns)



Column ID	Decision Variable	Perf Class Variable	Deteriorates	Upper Level Dec Tree Variable	Used in Expressions	Analysis Scope Variable	Default in Analysis
YEAR_LAST_REHAB	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
WC_ID	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1
TREATMENT_COST	<input type="checkbox"/>	0					
RSL	<input type="checkbox"/>						
ROUTE_ID	<input type="checkbox"/>	<input checked="" type="checkbox"/>					
PMS_WORK_CODE_ID	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	346
PMS_SECTION_ID	<input type="checkbox"/>	<input checked="" type="checkbox"/>					
PMS_BUDGET_CAT_ID	<input type="checkbox"/>	1					
PAVE_AGE	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0
NUMBER_OF_LANES	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	2
LENGTH	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
LANE_MILES	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
ESAL	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0
DISTRESS_YEAR_ID	<input type="checkbox"/>						
CA_WPC_IND	<input type="checkbox"/>						
CA_WPCV	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0
CA_WHEELPATH	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	

Several lists of variables are used in various aspects of network analysis and performance analysis. You use the Setup Analysis Columns window to create and maintain variables for those lists.

Each row in the Setup Analysis Columns window is a numeric or list column as initially defined in the Columns window (see page 115).

#### NOTE

If you insert a new row, you must enter a Column ID and this Column ID must exist in the Columns window. Furthermore, its column type must either be numeric or list. If these conditions are not met, an error message will be displayed.

The Column ID of a column in this window should also be a Column ID in the Network Master table (NMF). If not, then the settings in this window for the column have no effect on network analysis. See page 90 for more information on the Network Master table.

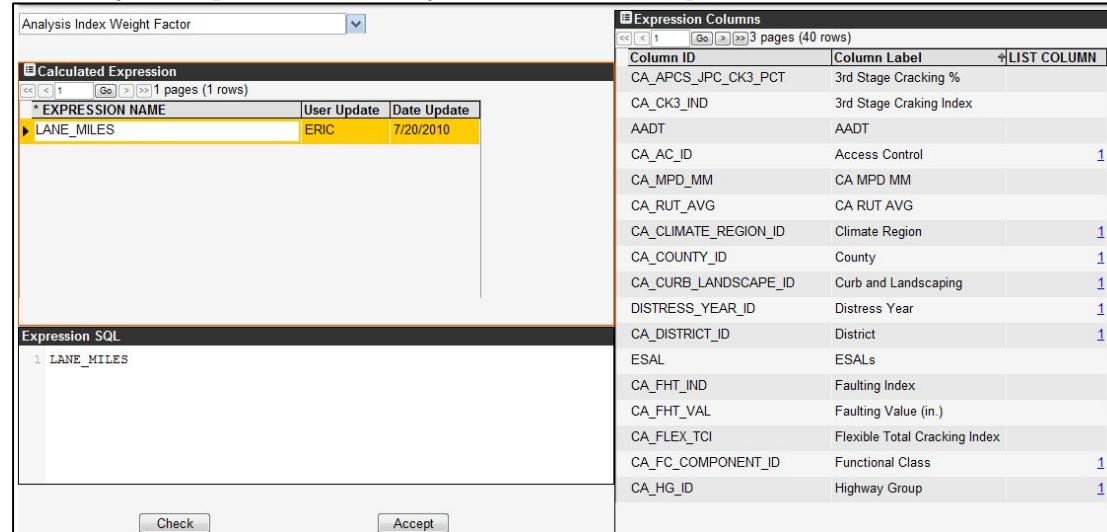
Once you define a PMS column in this window, then its assignment to the various "analysis" lists is determined by the value in each of the following fields:

- Decision Variable — This variable can influence rehabilitation need during network analysis by its use in decision trees. Specifically, when selected, the column will appear in the decision variable list shown in the dialog box displayed by selecting **Edit Decision Var Limits** command in the Edit/View tab of the Decision Tree window.
- Perf Class Variable — This variable can influence deterioration during network analysis by its use in performance models. Specifically, when selected, the column will appear in the decision variable list shown in the dialog box displayed by selecting **Edit Decision Var Limits** command in the Default Model Structure window.

- Deteriorates — This variable causes deterioration to be applied to the column's values during network analysis. It could be used as the Index value in network analysis and section current needs. Specifically, when selected, the column will appear:
  - In the Condition Attributes drop-down list in the Improvements pane of the Treatments window.
  - In the PMS Analysis Index drop-down list in the Network Analysis window.
  - As columns in the Road Sections pane of the Section Current Needs window.
  - As rows in the Attributes pane of the Performance Model window.
- Upper Level Dec Tree Variable — This variable can influence rehabilitation need during network analysis by its use in decision trees. Specifically, when selected, the column will appear in the Upper Level (left) pane in the Selection tab of the Decision Tree window.
- Used in Expressions — This variable can influence several factors during network analysis by its use in calculated expressions. Specifically, when selected, the column will appear in the right pane in the Calculated Expressions window (and so can be used in calculated expressions in that window).
- Analysis Scope Variable — This variable can influence the scope of road network analyzed during network analysis. Specifically, when selected, the column will appear in the Edit Scope dialog box that is displayed by selecting the **Edit Scope** command in the Network Analysis window.
- Default in Analysis — This column contains SQL that is used to supply a value for the column when it has no value in either the Network Master file or the Performance Master file.
- Allow Change After Treatment — This check box indicates whether the column's values change after a treatment where the values are not based on a performance model (that is, a non-PI-based attribute). When selected, the column will appear in the drop-down list in the Changing Attributes column of the Other Improvements pane in the Treatments window.
- Change with Year SQL — This field shows the SQL that determines how the value of the non-PI-based attribute changes each year in the future for scenario analyses.
- Update Order — For non-PI-based attributes, this field contains a number that indicates the order in which the attributes will be updated when multiple attributes either have "Change with Year SQL" in this window or are shown in the Other Improvements pane in the Treatments window.
- Is Constr. Column — This check box indicates that the column may be used as a constraint in the Work Plan Optimization window.
- Is Add Constr. Column — This check box indicates that the column may be used to subdivide constraints in the Setup Constraint Subdivision window.
- Ch with Time Groovy Script — This field is equivalent to the Change with Year SQL field except that Groovy scripts are utilized rather than SQL.
- Groovy Script for Calculated Columns — This column shows the Groovy script that calculates the value of the column. It is used in analysis in all future years. (The type of Groovy script is Calc Columns in Analysis.)

### 3.2. Calculated Expressions

(Roadway > Setup > Network Analysis... > Calculated Expressions)



The screenshot shows the 'Calculated Expression' window. On the left, there's a list pane titled 'Calculated Expression' with a single item: 'EXPRESSION NAME: LANE\_MILES, User Update: ERIC, Date Update: 7/20/2010'. Below it is an 'Expression SQL' pane containing the text '1 LANE\_MILES'. On the right, there's a large table titled 'Expression Columns' with 20 rows, each containing a column ID, column label, and a 'LIST COLUMN' indicator. The table includes columns like CA\_APACS\_JPC\_CK3\_PCT, 3rd Stage Cracking %, CA\_CK3\_IND, 3rd Stage Craking Index, AADT, AADT, CA\_AC\_ID, Access Control, CA\_MPDM\_MM, CA MPD MM, CA\_RUT\_AVG, CA RUT AVG, CA\_CLIMATE\_REGION\_ID, Climate Region, CA\_COUNTY\_ID, County, CA\_CURB\_LANDSCAPE\_ID, Curb and Landscaping, DISTRESS\_YEAR\_ID, Distress Year, CA\_DISTRICT\_ID, District, ESAL, ESALs, CA\_FHT\_IND, Faulting Index, CA\_FHT\_VAL, Faulting Value (in.), CA\_FLEX\_TCI, Flexible Total Cracking Index, CA\_FC\_COMPONENT\_ID, Functional Class, and CA\_HG\_ID, Highway Group.

The purpose of the Calculated Expressions window is to supply user-defined information for parameters used in analysis. This information is either the definition of a drop-down list for use in other windows or a formula for direct use in analysis. Each module has its own Calculated Expressions window, and the expressions only apply to analysis in that module.

The parameters are shown in the drop-down list in the top right corner of the window. By selecting from this drop-down list, you can then define the expressions for the parameter. (Multiple expressions are allowed for all parameters other than Analysis Index Weight Factor and Analysis Priority. These two parameters are only allowed one expression.)

#### NOTE

Some parameters require SQL expressions and some require Java expressions. The application will know which is required for a selected parameter and will configure the window accordingly. SQL expressions appear in the lower left pane; Java expressions appear in the Java Expression Text column of the left middle pane.

In addition to the parameters drop-down list, the Calculated Expressions window contains three panes: the Expressions pane on the left, the SQL pane in the bottom left, and the Columns pane on the right.

#### 3.2.1. Description of the Available Parameters

The available parameters are:

- Analysis Index Weight Factor – This parameter affects the benefits resulting from a scenario analysis regardless of the objective of the analysis. An SQL expression is required for this parameter. This parameter acts as a default when a Groovy script is not selected in the Analysis Weight Groovy Script column in the Optimization Analysis window.
- Analysis Priority – This parameter is a single formula that is the priority to rehabilitate. It is applied in network analysis when the prioritization method is chosen. The higher the number the sooner that the road section will be rehabilitated. An SQL expression is required for this parameter. This parameter acts as a default when a Groovy script is not selected in the Priority Groovy Script column in the Optimization Analysis window.

- Budget Group – When this parameter is selected, every record in the Budget Categories window (see page 57) is entered in the Expressions pane. Then, for each record in the Expressions pane, the ID of the budget category record in the Budget Categories window is entered in the SQL pane.
- Treatment Cost – This parameter is composed of the set of values of all treatment costing methods. An SQL expression is required for each treatment cost method and is entered in the SQL pane. For example, if treatment cost is by square feet, then the formula would look like this: "UNIT\_COST \* NVL(SEC\_WIDTH,24)\*(LENGTH) \* 5280 /900", where unit cost, length, and width are columns in the Network Master file. Treatment costs appear in a drop-down list in the Cost column in the Treatments pane of the Treatments window.

### **3.2.2. Expressions Pane**

In the Expression pane, you configure the value of the parameter selected in the drop-down list in the upper left corner of the Calculated Expressions window. When the selected parameter may have multiple, discrete values that appear in a drop-down list, then multiple records in this pane are configured; when the parameter's values result from a formula, then only one record is configured.

### **3.2.3. SQL Pane**

The SQL pane in the bottom left corner of the window contains the calculated Oracle SQL expression pertinent to the currently selected parameter and record in the Expression pane. It is this expression that is used in analysis to provide the appropriate calculation. The Oracle SQL expression must be limited to columns that exist in the Network Master table and must be valid. (To check that the SQL expression is valid, click the **Check** button.) The Columns pane lists the columns that may be included in the SQL expression.

Once an SQL expression is completed, activate the new SQL expression in the network analysis process by:

1. Clicking the **Check** button to be sure that the expression is valid.
2. Clicking the **Accept** button.
3. Clicking the  icon to save the new expression.

#### **NOTE**

An SQL expression must be accepted before it can be saved. The expression will neither be accepted nor saved if it is not valid or it is saved without first being accepted.

### **3.2.4. Columns Pane**

The Columns pane on the right side contains the list of columns that may be used in expressions. If a column is a list, then a hyperlink is provided in the List Column column to display the values in the list.

A column of the Network Master file is marked for use in expressions by selecting the Used in Expressions check box in the PMS Columns in Analysis window (see page 51).

## **3.3. Material Code ID**

(Roadway > Setup > Construction Setup > Material Code ID)

Material Code ID						
Material Code Name		Layer Category	Color	Comments	User Update	Date Update
Milling			1	0- Code for a Milling Lay	ERIC	1/19/2012
AB			0	AGGREGATE BASE	LYN	1/25/2012
AS			0	ASPHALT TREATED SL	LYN	1/25/2012
ATB			0	ASPHALT TREATED B/	LYN	1/25/2012
ATPB			0	ASPHALT TREATED PE	LYN	1/25/2012
BASE			0	BASE	LYN	1/25/2012
BWC			1	BONDED WEARING CC	LYN	1/25/2012
CFRB			0		LYN	1/25/2012
CRACKSEAL			1		LYN	1/25/2012
CTB			0	CONCRETE TREATED I	LYN	1/25/2012
CTPB			0		LYN	1/25/2012
CTS			0		LYN	1/25/2012

You use the Material Codes ID window to define and maintain the list of materials used by your agency in road construction. The columns in this window provide the following information:

- Material Code – This column shows the user-defined identification code for each material used in the construction and maintenance of roads in your network.
- Color – This column shows the color assigned to the material. The assigned color may be changed by double-clicking the displayed color. This causes a new window to open with a color palette. Click the desired color and then click **OK** to assign the color to the material.
- Comments – This column allows you to enter additional information. Typically, this is a description of the material or what the ID code means.

### 3.4. Work Code

(Roadway > Setup > Construction Setup > Work Code)

Work codes				
Work Code Name	Pavement Type	Work Type	Comments	
Thin Overlay	Flexible	Rehabilitation		
Seal Coat	Flexible	Preventive Maintenance		
Slab replacement	JPC	Rehabilitation		
Crack, Seat and Overlay JPC		Reconstruction		
Full Depth Reclamation	Flexible	Reconstruction		
Seal Cracks	Flexible	Preventive Maintenance		
Grind with Slab Replace	JPC	Rehabilitation		
Grind	JPC	Rehabilitation		
PCC lane replacement	JPC	Rehabilitation		
Cold In-Place	Flexible	Rehabilitation		
Unknown		Routine Maintenance		
Medium Overlay	Flexible	Rehabilitation		
Thick Overlay or Recons	Flexible	Reconstruction		
Very Thin Overlay	Flexible	Preventive Maintenance		

You use the Work Code window to define work codes, which, typically, represent general categories of work accomplished in a contract. A work code is assigned to each contract in the database.

You may define as many work codes as necessary to fully describe the type of work performed on a pavement network. You may also add, delete, or edit existing work code definitions.

Each work activity is also assigned one of three categories depending on the scope of the work to be performed. The three categories are:

- Rehabilitation activities consist of treatments designed to restore or improve the structural capacity of a pavement management section. These activities can range from simple, thin overlays to base repair to removal of poorly performing materials and thick overlays.
- Construction activities represent the pavement management section functioning in its first performance period (before rehabilitation). Pavements are considered to be reconstructed when at least a substantial amount (if not all) of the select material associated with the previous performance period has been removed and replaced with new, different, or rejuvenated material.
- Maintenance activities are considered to be thin pavement surface treatments designed to seal cracks in the pavement surface or improve skid resistance. They generally do not add structural capacity to a pavement; however, they may assist in prolonging a section's performance period.

#### NOTE

Road structure calculated values are only developed from data when the Work Types are identified as "rehabilitation" or "construction."

#### **3.4.1. Description of the Work Codes Window**

The columns in the Work Codes window provide the following information:

- Work Code Name — This column shows the user-defined identification code for each work activity.
- Pavement Type — This column indicates the surface type with which the work activity is concerned.
- Work Type — This column shows the category assigned to the work activity (R = rehabilitation; C = construction; and M = maintenance). Models are only developed from data when the Work Type is identified as "rehabilitation" or "construction."
- Comments — This column allows you to enter additional information. Typically, this is a description of the material or what the ID code means.

#### **3.4.2. How to Define a Work Code**

To define a work code:

1. Display the Work Code.
2. In the Work Code window, right-click and then click **Insert**. A new row is added to the table.
3. In the Work Code Name column, highlight the default name and then type the name of the work code.
4. In the Pavement Type column, click the down arrow and then click the surface type.

5. In the Work Type column, click the down arrow and then click the work type category (rehabilitation, construction, or maintenance) for this work code.
6. Click the  icon to save the new work code.

### 3.5. Budget Categories

(Roadway > Setup > Network Analysis... > Budget Category)

Budget Category	
Budget Group	* Budget Category
► STIP	STIP
HM1	HM1
HA-22	HA-22

The Budget Categories window allows you to create and maintain budget categories that are used in work plans and work orders. Currently, the entries in the Budget Category column appear in drop-down lists in the Issue Service Requests, Issue Work Orders, Budget Plans, Scenario Analysis, Treatments, and Master Plan windows, where you select a budget category to assign to planned work.

#### NOTE

A budget category is assigned to each activity in the Setup Activities window. When a service request or work order is issued for a particular activity, if a budget category is assigned to the activity, then the budget category column will default to the assigned budget category. Similarly, line items for work plans can be grouped by budget category using the activity assignment for the line items.

### 3.6. Inventory

The fundamental data element of the maintenance management system is an asset type. These are the different assets (signs, road sections, buildings, traffic signals, etc.) that are "owned" by your administrative unit.

Inventory data can be collected for each asset type. Inventory data is divided into classes and assigned a status. For example, a road section (an inventory item) may be an Interstate road (the class) and may be active (the status).

#### 3.6.1. Setup Status

(e.g. Roadway > Database > Culverts Data > Culvert System Status)

Roadway > Database > Culverts Data > Culvert Status			
1 pages (2 rows)			
* Asset Status Name	* Asset Status Type	User Update	Date Update
► Active	Active	EUGENE	1/3/2012
Inactive	Inactive	EUGENE	1/3/2012

The Setup Status window allows you to create and maintain the various status values needed for an asset type. They are used as a criterion for displaying items from the asset type's inventory as well as for performing certain actions.

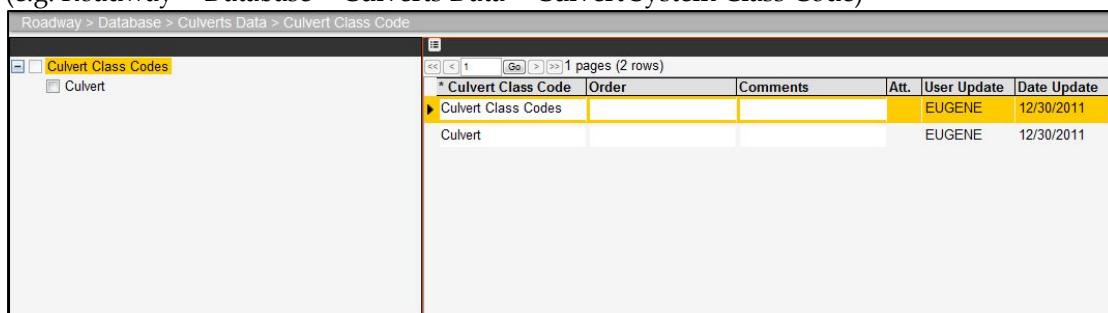
The Asset Status Name column shows the name of the status, which is what is displayed in various windows throughout the module.

Each status value must be assigned a status value type. (A type may be assigned to more than one status value.) Four types are available:

- New – This type is for a status value that indicates that an inventory item was recently added to the system.
- Active – This type is for a status value that indicates that an inventory item is currently in use.
- Inactive – This type is for a status value that indicates that an inventory item is no longer in use.
- Purged – This type is for a status value that indicates that an inventory item is no longer part of the system.

### 3.6.2. Setup Class Code

(e.g. Roadway > Database > Culverts Data > Culvert System Class Code)



The screenshot shows a software interface titled 'Roadway > Database > Culverts Data > Culvert Class Code'. On the left, there is a tree view pane showing a single node 'Culvert Class Codes' which is expanded to show a child node 'Culvert'. On the right, there is a table view pane with the following data:

* Culvert Class Code	Order	Comments	Att.	User Update	Date Update
Culvert Class Codes			EUGENE		12/30/2011
Culvert			EUGENE		12/30/2011

Classification codes (abbreviated as "class codes" or "classes") divide an asset type into categories. For example, road sections (an asset type) may be divided into Interstate, Primary, Secondary, and Gravel classes. Each asset type has its own set of classification codes, although some asset types have only one class that is identical with the asset type.

The Setup Class Code window allows you to configure and maintain classification codes for an asset type. It also allows you to specify how those codes relate to each other by developing a hierarchy of classification codes. The classification codes configured in this window appear in various windows throughout the module, and are used to select what inventory items are displayed.

The Setup Class Code window contains two panes: on the left, a tree view that shows the hierarchical arrangement of the codes; and on the right a table view that provides additional information about each code. The tree view is expanded by clicking a plus sign (+), or contracted by clicking a minus sign (-).

When you select a code in the tree view pane, the row corresponding to that code is highlighted and displayed in the table on the right. The converse is also true – selecting a row also selects a code in the tree view.

When you right-click a record in the table view pane, a shortcut menu is displayed. This menu contains the following command in addition to the common commands:

- **Add Branch** – This command adds a new node to the tree view that is subordinate to the node named in the row to which you pointed. A corresponding record is also added in the table view.

### 3.6.3. Inventory Window

Each asset type has its own Inventory window. The layout of the Inventory window is the same for each asset type. It contains the following:

- Inventory tab – This tab lists all inventory items and provides basic information about the item.
- Attribute tabs – Optionally, tabs may be available that provide additional information (such as defect survey findings) about the inventory item selected in the Inventory tab.

## 3.7. Standard Sections

(Roadway > Setup > Construction Setup > Standard Sections)

Standard Sections					
1 pages (1 rows)					
* Standard Section Name   Comments   Att.   User Update   Date Update					
198-ASP					
Milling	1 0- Code for a Mi ERIC	1/19/2012	TODD	2/2/2012	
* Material Code Name   Color   Layer Category   Comments   User Update   Date Update					
AB	0 AGGREGATE F LYN	1/25/2012			
AS	0 ASPHALT TRE/ LYN	1/25/2012			
ATB	0 ASPHALT TRE/ LYN	1/25/2012			
ATPB	0 ASPHALT TRE/ LYN	1/25/2012			
BASE	0 BASE LYN	1/25/2012			
BWC	1 BONDED WEA LYN	1/25/2012			
* Layer   Material Code   * Thickness (ft.)   Comments   Att.   User Update   Date Update					
1 AB		0.4		TODD	2/2/2012

In the Standard Sections window, you can view, edit, and modify data that represents standard-constructed road sections by defining the type of work and the road layers involved in each type of standard construction. Standard sections are used to facilitate data entry in the Layers pane of the Contracts window.

When actual contract work is constructed to "standard" specifications (as represented in the Standard Sections window), then when you right-click in the Layers pane of the Contracts window, a standard section can be copied into the Layers pane as the actual contract construction event.

### 3.7.1. Standard Sections Window Description

The Standard Sections window contains the following panes:

- Standard Sections – This pane shows the available standard sections.
- Material Codes – This pane shows the materials that are configured in the system and which are used for the layers of the standard section. The intent of this pane is to help you select the appropriate material from the drop-down list that is available in the Material Codes column of the Layers pane.
- Layers – For the standard section selected in the upper pane, this pane shows the layers that comprise the standard section. Note that milling is always assigned the code of 99 and the thickness is expressed positively (that is, not as a negative number even though material is removed). Even though it is expressed as a positive number, the system will subtract the amount from the total road thickness.

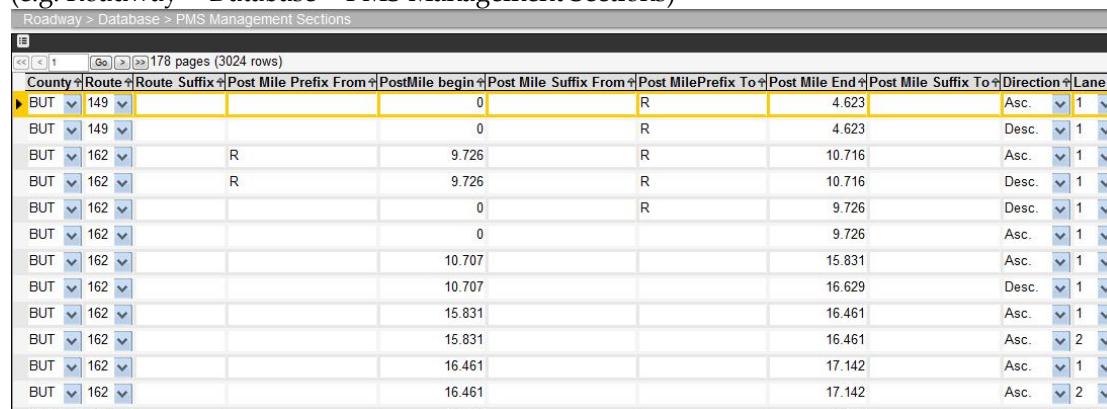
### **3.7.2. How to Create a Standard Section**

To create a standard section:

1. Display the Standard Sections window.
2. In the Standard Sections pane, right-click and then click **Insert**. A new row is added to the table.
3. In the Standard Section Name column, highlight the default name and then type the name of the standard section. Since this name is what will appear in other windows when you are selecting standard sections, this name should be specific and informative.
4. In the Layers pane, right-click and then click **Insert**. A new row is added to the pane.
5. Click in the Layer column and then type 1 to indicate that this is the first layer of the standard section. (The first layer is defined as the topmost layer.)
6. Click the down arrow in the Material Code column and then click the type of material for this layer in the drop-down list.
7. Click in the Thickness column and then enter the thickness of this layer.
8. Repeat steps 4 through 7 for each additional layer in the standard section.  
Note: Milling is always assigned as layer 99 and given a positive thickness.
9. When all layers are entered for the standard section, click the  icon to save the new standard section.

## **3.8. Current Pavement Management Section Data**

(e.g. Roadway > Database > PMS Management Sections)



Roadway > Database > PMS Management Sections								
County + Route + Route Suffix + Post Mile Prefix From + Post Mile begin + Post Mile Suffix From + Post Mile Prefix To + Post Mile End + Post Mile Suffix To + Direction + Lane +								
BUT	149			0	R	4.623	Asc.	1
BUT	149			0	R	4.623	Desc.	1
BUT	162		R	9.726	R	10.716	Asc.	1
BUT	162		R	9.726	R	10.716	Desc.	1
BUT	162			0	R	9.726	Desc.	1
BUT	162			0	R	9.726	Asc.	1
BUT	162			10.707		15.831	Asc.	1
BUT	162			10.707		16.629	Desc.	1
BUT	162			15.831		16.461	Asc.	1
BUT	162			15.831		16.461	Asc.	2
BUT	162			16.461		17.142	Asc.	1
BUT	162			16.461		17.142	Asc.	2

The Management Sections data window displays current Pavement Management Sections data in a table format (a graph window is also provided to display the information graphically). You use the data window to define and maintain the current list of pavement management sections and boundaries. The system uses these pavement management sections for analyses performed via Analysis menu items.

The limits for the current pavement management sections are used for model development and network analysis. Therefore, section breaks should be made where logical limits for pavement projects are expected. AgileAssets recommends that section breaks be made where:

- A discontinuity based on construction history (date, structure, materials) occurs.
- Pavement condition changes.

For each pavement management section, only one classification may be set for each classification variable assigned to the section. Therefore, if a classification variable changes, the pavement management section should be broken at that point. An exception may be made for a classification such as sub-grade where a localized change in the sub-grade does not significantly affect pavement performance, pavement project selection criteria, etc.

The current Pavement Management Sections data form shown below is used to define each pavement management section. The minimum definition for the section includes:

- Route ID.
- Extension.
- Lane Direction.
- Lane Identification.
- Point from the designated route beginning point that defines the beginning of the pavement management section (From Point).
- Point from the designated route beginning point that defines the ending of the pavement management section (To Point).

Other important information that should be shown includes section length, typical width of the pavement management section, any additional paved areas (such as cul-de-sacs or continuous left turn lanes), and the pavement type.

Classification data is also associated with each pavement management section. AgileAssets established this classification data based on your agency's requirements. Each pavement management section can have only one classification value for each defined variable. The Route Description, From, and To physical description are optional, and are only shown in this window to allow you to orient yourself concerning the pavement management section location in the field.

### 3.9. Construction History

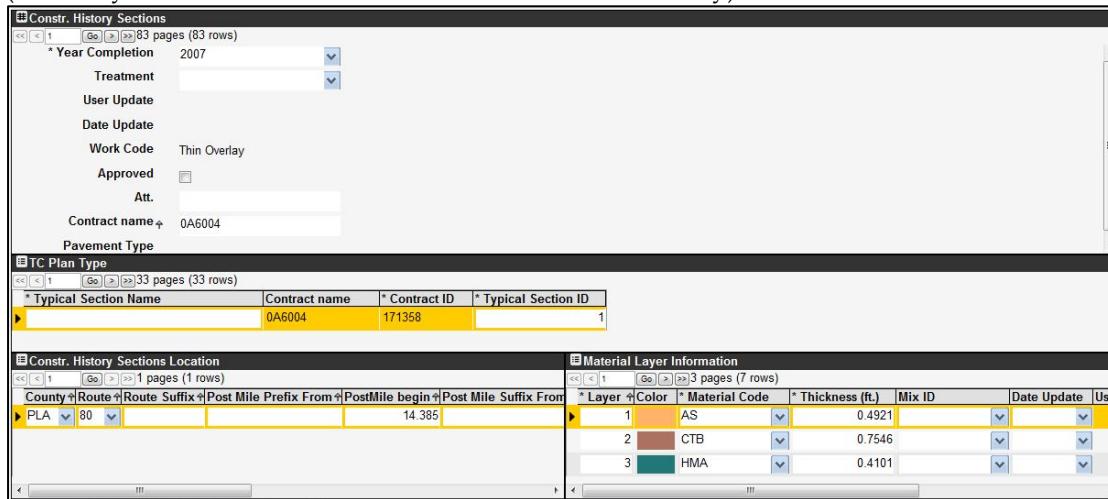
In the Construction History window, you can view, edit, and modify data related to work performed on pavements. Data is recorded in this window by the limits of the work as it was performed on the road. The Construction History window contains the following information by work:

- Construction History Section ID, which is a unique record number that may be automatically sequenced by the system depending upon your version.
- Route ID.
- Physical limits of the work.
- Year the work is completed.
- The type of work performed on the road referenced by the work code category and the materials placed or removed during the work.

As with other database windows, the limits for a project do not have to coincide with the limits of any pavement management sections. This is because the software will automatically apply pavement structure information to pavement management sections when the master files are calculated. The window allows you to view pavement projects exactly where they occur. The system then uses an algorithm to build the entire road structure for all mileage based on the construction history information.

### 3.9.1. Description of the Construction History Window

(Roadway > Database > Construction > Construction History)



The Contracts window contains the following panes: Construction History Sections, Material Codes, and Layers. These panes are described in more detail in the following sections.

#### ***Construction History Sections Pane***

The Construction History Sections pane contains information about a constructed section of road such as route, project limits, construction date, and a physical description of the project. Three important pieces of information are recorded for each project: road location description, work code, and completion year.

#### ***Material Codes Pane***

The Material Codes pane allows you to define several important sets of properties for each material type. You then assign these material codes to layers for each contract. You use this window to define and maintain the list of materials used by your agency. Information that can be recorded for each layer includes a user-defined material code, a user-defined material description, and a layer type.

#### **NOTE**

Note: The layer category assigned to each material is very important because it controls how road structure categories are assigned to a section. Any material assigned a layer category of either asphalt or concrete is used to estimate the pavement thickness. The pavement thickness is used to calculate the road structure category as discussed in the Work Code topic (see page 55). AgileAssets staff members establish an initial set of materials for each agency based on input during system configuration meetings. You may add, modify, and remove layers from this window if you have sufficient authority.

This pane also allows you to modify the color settings that represent each material. To do this, double-click the color circle. This displays the Color Settings window.

#### ***Layers Pane***

The Layers pane shows the layers placed or constructed during the contract defined in the upper window. You use this window to add, modify, or delete individual layer information, which includes layer number, material code, and thickness.

Layers are ordered with the smallest layer number on top. Milling is indicated by a layer number of 99 and should contain a positive value for milling thickness. In structure calculations, this milling thickness will be subtracted.

When you right-click this pane, the system displays a shortcut menu with the common commands along with the following special command:

- **Copy Layers from Standard Sections** – This command replaces the information shown in the pane with a "standard section," which is defined in the Standard Sections window (see page 59). After selecting the command, the system displays a dialog box so may select the desired standard section.

### **3.9.2. How to Insert a New Record**

To add a new construction history section record, follow these steps:

1. Display the Construction History window.
2. In the Construction History Sections pane, right-click and then click **Insert**. A new row is added to the table.
3. Enter the road location information by selecting route, direction, and lane from the drop-down lists and by entering begin and end mile points in the appropriate columns.
4. Tab to the other columns and type the information if known. You must type data in the following columns: Year Completion and Work Code. If either of these columns is left blank, then the record cannot be saved.
5. Click in the Constr. History Section column of the new record and type a unique alphanumeric identifier.
6. In the Layers pane, define the layers placed or removed as part of the contract. This may be accomplished by inserting a standard section or by right-clicking and then clicking **Insert** to add records to the pane. In each new record added via the **Insert** command:
  - a. In the Layer column, type the code for the layer. (Milling is always defined as Layer 99.)
  - b. In the Material Code column, select the material for the layer from the drop-down list accessed by clicking the down arrow.
  - c. In the Thickness column, type the thickness of the layer.
7. Click  to save the new record.

### **3.9.3. How to Add a New Layer**

To add a new layer to an existing contract:

1. Display the Construction History window (Roadway > Database > Construction > Construction History).
2. In the Construction History Sections pane, click the row showing the road section to which layer information will be added. The system will highlight it to show that it is selected.
3. In the Layers pane, right-click and then click **Insert**. A new row is added to the table.
4. In the new record in the Layers pane, click in the Layer column and then type the ID code for the layer. The value entered for a layer ID may be any two-digit number (other than 99) that designates the vertical position of the layer, with the smallest

number indicating the top of the layer. For milling, set the Layer column to 99 and enter a positive value for milling thickness.

5. In the Material Code column, click the down arrow and then click the material for the layer.
6. In the Thickness column, type the thickness of the layer .
7. Repeat steps 3 through 6 for any additional layers.
8. When all layers are entered, click the  icon to save the new information.

#### **3.9.4. How to Insert a Standard Section**

To facilitate data entry, different types of "standard sections" are defined in the Standard Sections window. A standard section is then inserted in the Layers pane as follows:

1. In the Construction History Section pane, click the row showing the road section for which layer information will be added.
2. In the Layers pane, right-click and then click **Copy Layers from Standard Sections**. The system displays a dialog box listing all standard sections defined in the Standard Sections window.
3. In the dialog box, click the row showing the desired standard section and then click **OK**. The system closes the dialog box and displays the layers of the selected standard section in the Layers pane.
4. Click the  icon to save the new information.

### **3.10. Pavement Structure**

The Pavement Structure feature shows changes in the construction history along a route or portion of a route. Although the window for viewing contracts is useful for reviewing individual pavement paving projects, it is not a convenient way to review the entire construction history of the roadway across multiple contracts. AgileAssets developed the Pavement Structure feature to provide "one look" access to the entire history of construction activity.

You may view Pavement Structure as data in a table format or graphically along a route.

#### **3.11. Pavement (Profile/Cross-section)**

The Pavement Profile/Cross-section window shows changes in the construction history along a route or portion of a route. Although the window for viewing Contracts is useful for reviewing individual pavement paving projects, it is not a convenient way to review the entire construction history of the roadway across multiple contracts. AgileAssets developed the Pavement Profile/Cross-section window to provide "one look" access to the entire history of construction activity.

The bottom of this window will display a "Section Profile (Longitudinal)" graph after clicking the  icon. When you click the "Section Profile (Longitudinal)" graph, a "Cross-section" graph will be displayed on the right for the road section on which you clicked. These graphs show a representation of the pavement structure within the limits set in the Route pane, which is shown in the upper left of the Pavement Profile/Cross-section window.

##### **3.11.1. Route Pane**

The Route pane in the upper left corner of the Pavement Profile/Cross-section window shows the route, and section of the route, to be graphed. The Show Layers Since column determines

what construction data is displayed in the graph by allowing you to select all layers regardless of when they were constructed; only those layers since the last reconstruction; or only those layers since a particular year.

**NOTE**

For those applications that have an Approved column in the Construction Contracts and Data window(s), you may select the Approved check box in this pane to display only those layers that are approved.

This pane regulates what is graphed in the Section Profile Graph, which can only display a single road lane. When All is selected for either lane or direction, then the system uses this convention to select the lane to graph: if Direction is All, then use the Inc (N/E) direction; if Lane is All, then use the Right lane.

### **3.11.2. Cross-Section Pane**

The read-only Cross-section pane in the top middle of the Pavement Profile/Cross-section window displays all section limits for data displayed in the graph. These section limit breaks are a combination of all construction history between the two graph points.

**NOTE**

When you click a mile point in the Section Profile graph, the system will display the record for the mile point where you clicked in the Cross-section pane.

### **3.11.3. Pavement and Layer Details Pane**

This read-only pane in the upper right corner of the Pavement Profile/Cross-section window shows the construction history (including all contracts and layers) at a particular point on the road. This point is set in either of the following ways:

- When you click a mile point in the Section Profile graph, the Contract and Layer History pane will show all layers and contracts associated with that point. In addition, the Cross-section graph on the right and the Section Breaks pane in the top middle will be set to that road point.
- When you adjust the Section Breaks pane to another section, this identifies a section of road for which all road points have identical construction history. The Contract and Layer History pane then shows all layers and contracts associated with any of those points.

### **3.11.4. Section Profile Graph**

The Section Profile graph shows detailed construction information for the roadway specified in the Route pane from the year given in the Show Layers Since column to the present.

You may change the vertical and horizontal scales for the graph to better display layer information. You accomplish this by right-clicking the graph and then clicking the **Change Graph Properties** command on the shortcut menu.

### **3.11.5. Cross-section Graph**

When you click a mile point in the Section Profile graph, the system displays the cross-section profile at that mile point in the Cross-section graph on the right side of the window. (When the graph is first displayed in the Section Profile pane, the Cross-section graph will not be shown; click a mile point to see the graph.)

You may change the vertical and horizontal scales for the graph to better display layer information. You accomplish this by right-clicking the graph and then clicking the **Change Graph Properties** command on the shortcut menu.

### **3.11.6. How to Display a Section Profile Graph**

To display a Section Profile graph:

1. Display the Pavement Profile/Cross-section window.
2. In the Route pane, check the route displayed. If this is the correct route, go to the next step. Otherwise, click the down arrow and then click the desired route from the list.
3. If the route is subdivided by direction, click the down arrow in the Direction column and then click the desired direction.
4. If the route is subdivided by lane, click the down arrow in the Lane column and then click the desired lane.
5. Click in the Start Milepoint column and then type the starting mile point of the route to be graphed.
6. Click in the End Milepoint column and then type the ending mile point of the route to be graphed.
7. Click the down arrow in the Show Layers Since column to display the drop-down list and then click the starting year for the construction history information that is to be graphed.
8. Click the  icon to display the graph for the selected roadway and time period.

### **3.11.7. How to Display a Cross-section Graph**

To display a Cross-section graph:

1. Display the Pavement Profile/Cross-section window.
2. Display the Section Profile graph for the roadway for which you wish to see the cross-section (see the previous section).
3. Locate the mile point in the Section Profile graph for which you wish to see the cross-section and then click the graph at this point. The system will then display the cross-section at this mile point on the right side of the window.

## **3.12. Structure Defaults**

The Structure Defaults window is used to view non-detailed pavement structure data. AgileAssets recommends that the information from the last structural improvement be displayed in this window when detailed construction information is not available for that section of pavement. For those sections where adequate detail is available, it should be entered in the Construction History window.

Note: Whenever data is in both the Construction History window and the Structure Defaults window, the system will use data from the Construction History window for filling the master files.

## **3.13. Conditions Menu**

The Conditions menu provides access to the different types of distress data that your agency collects as well as any condition index. This data typically includes flexible distress, rigid

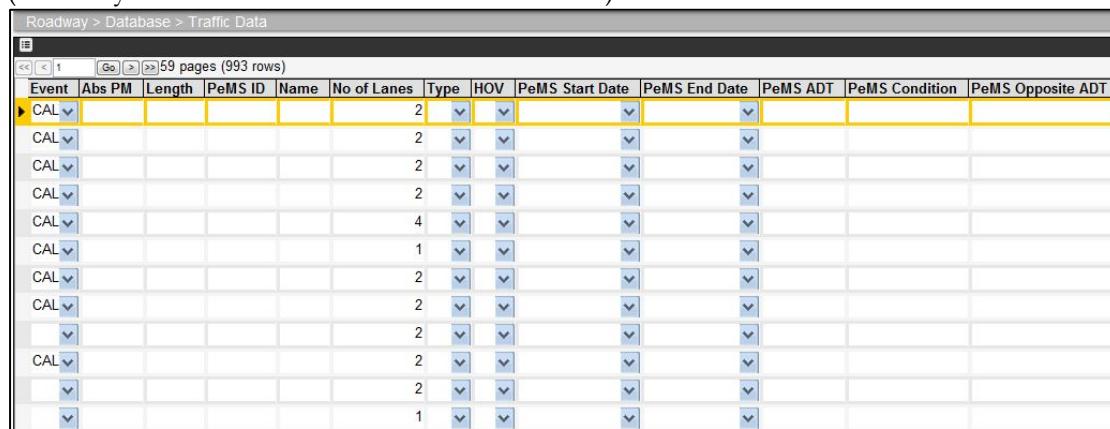
distress, skid, and ride. Two, paired windows are provided for each type of condition data: a data window and a graph window. See page 42 for more information on data and graph windows.

**NOTE**

The displayed condition indices (that is, the calculated values) are read-only and must be calculated using the **Update Target Table** command (which is found on shortcut menus that are displayed by right-clicking in certain windows).

### 3.14. Traffic

(Roadway > Database > Traffic Data > Traffic Data)



Event	Abs PM	Length	PeMS ID	Name	No of Lanes	Type	HOV	PeMS Start Date	PeMS End Date	PeMS ADT	PeMS Condition	PeMS Opposite ADT
CAL					2	▼	▼		▼	▼		
CAL					2	▼	▼		▼	▼		
CAL					2	▼	▼		▼	▼		
CAL					2	▼	▼		▼	▼		
CAL					4	▼	▼		▼	▼		
CAL					1	▼	▼		▼	▼		
CAL					2	▼	▼		▼	▼		
CAL					2	▼	▼		▼	▼		
	▼				2	▼	▼		▼	▼		
CAL					2	▼	▼		▼	▼		
	▼				2	▼	▼		▼	▼		
	▼				1	▼	▼		▼	▼		

The Traffic section of the database captures information about traffic levels and predicted growth along a route. Traffic data is assigned to a route independently of the pavement management section in much the same way that inventory data is assigned to a route. Data shown in the Traffic windows generally includes the Annual Average Daily Traffic (AADT), Equivalent Single Axle Load (ESAL), and the percent growth.

Two, paired windows are provided for traffic information: a data window and a graph window. See page 42 for more information.

**NOTE**

The displayed values are read-only and must be calculated using the **Update Target Table** command (which is found on shortcut menus that are displayed by right-clicking in certain windows).

### 3.15. Performance Master

(Roadway > Analysis > Performance Analysis... > Performance Master > Performance Master Data)

Performance Master Data												
Lane	Year	AADT	Left Paved Shoulder Width	Right Paved Shoulder Width (ft)	Speed Limit	Number of Lanes	Comments	Att.	User Update	Date Update	Mix Design ID	
	2010	1,470	0	2	65	2					Unknown	
2010	30,000	0	0	35	5						Unknown	
2010	35,499	0	0	45	4						Superpave 3/8" PG64-28 50	
2010	34,500	0	0	35	4						Superpave 3/8" PG64-28 50	
2010	8,900	0	0	45	4						Unknown	
2010	7,300	0	0	35	2						Unknown	
2010	13,100	0	0	35	4						Unknown	
2010	1,816	0	4	35	2						Unknown	
2010	1,600	0	2	65	2						Unknown	
2010	1,400	0	4	65	2						Unknown	
2010	1,400	0	2	45	3						Unknown	
2010	1,400	0	2	65	3						Unknown	
2010	1,382	0	2	65	2						Unknown	
2010	1,300	0	2	65	2						Unknown	
2010	357	0	1	55	2						Unknown	
2010	300	0	1	55	2						Unknown	
2010	300	0	0	55	2						Unknown	

Performance analysis uses pavement management history information to provide a feedback mechanism to improve the predictive capabilities of pavement performance models. The primary repository for data used in performance analysis is the Performance Master file (PMF).

The PMF contains a record for each management section that contains all available condition data. It is the foundation upon which pavement modeling (and therefore estimated performance in network analysis) is based. You build and maintain models in the Performance Models window. They are based on historical pavement condition and other data stored in the PMF. The Section Performance window shows the past and expected performance of individual pavement management sections based upon the performance model group.

The PMF is a vital system component that allows you to make good, consistent pavement management predictive models from performance index and other performance classification data. The PMF is a compilation of calculated results from the data contained in the Database module. Specific calculation rules for each column in the Performance Master table are coded into the update target SQL values for each column. Typically, each column is also commented with additional information about the data rules for the table.

### 3.16. Performance Models

(Roadway > Analysis > Performance Analysis... > Performance Models)

Performance Model Tree												
Roadway > Analysis > Performance Analysis... > Performance Models												
<b>Performance Model Tree</b>												
Work Code: Full Depth Recla												
<b>Attributes</b>												
Column Label												
3rd Stage Cracking %	0	100	<input checked="" type="checkbox"/>									
Faulting Value (in.)	100	0	<input type="checkbox"/>									
Flexible Total Cracking	100	0	<input type="checkbox"/>									
IRI AVG	40	500	<input type="checkbox"/>									
CA MPD MM	0	4.15	<input type="checkbox"/>									
WPCV	0	100	<input type="checkbox"/>									
<b>Models and Expressions</b>												
PMS MODEL NAME												
CA_IRL_AVG-PCC, Unk Power	MODEL TYPE	MODEL EXP	Comments	User Update	Date Update							
CA_IRL_AVG-PCC, Unk Power		88.480781759779 + -0.2C 625	CA_IRL_AVG									
CA_IRL_AVG-PCC, Unk Power		88.480781759779 + -4.0E 628	CA_IRL_AVG									
CA_IRL_AVG-PCC, Unk Power		88.480781759779 + -2.6E 627	CA_IRL_AVG									
lin life 27.5	Linear	100 + -1.8182 * T	SYSTEM	5/12/2006								
lin life 26.4	Linear	100 + -1.8939 * T	SYSTEM	5/12/2006								
lin life 24.2	Linear	100 + -2.0661 * T	SYSTEM	5/12/2006								
lin life 22	Linear	100 + -2.2727 * T	SYSTEM	5/12/2006								

The Performance Models window shows the decision tree for determining what deterioration model is to be used for a certain type of road for a particular condition attribute. (A condition

attribute is essentially a measurement index such as rutting or cracking, with the model predicting how this measurement index changes over time for a particular type of road.) The models configured in this window are then used in network analysis.

The structure of the tree itself, and the criteria that cause certain child nodes to be selected and not others, are set in the Default Model Structure window (see page 73). The structure and branching criteria cannot be modified in the Performance Models window.

You use the Performance Models window to assign a model to a node of the decision tree for each condition attribute. Almost always, models are assigned only to the rightmost nodes. However, when all child nodes for a parent node would utilize the same model, you may assign a model to the parent node as a shortcut.

The Performance Models window has three panes: the Decision Tree pane, the (condition) Attributes pane, and the Models pane.

### **3.16.1. Decision Tree Pane**

For the condition attribute selected in the Attributes pane, the Decision Tree pane shows what models are assigned to the different nodes of the decision tree. The box representing each node shows the assigned model — unless no model is assigned, in which case the decision variable used in selecting the node is displayed. (If you hover over a node, the system will display the decision variable and value that causes that node to be selected.)

When you right-click a node in the decision tree, the following special commands are available:

- **Set Selected Model for this Node** – For the condition attribute selected in the Attributes pane, this command assigns the model selected in the Models pane to the decision tree node that you right-clicked. If a model is already assigned to this node, the new selection replaces the previous selection. If a model is already assigned to this node, but is protected, and the system displays an error message when you select this command because you cannot assign a model to a protected node.
- **Set Model for Children** – When all child nodes of a parent would utilize the same model for the selected condition attribute, you may right-click the child nodes' parent node and use this command as a shortcut to assigning the model to each child node. Note that the parent node must have a model already assigned to it before using this command. (If no model is configured for the parent node, this command is not available.)
- **Protect/Release Model** – Provided you have sufficient authority, you may "protect" the assignment of a model to a node. This means that the assignment cannot be changed until the protection is released (by you or someone with sufficient authority). Once a model assignment is protected, an asterisk (\*) appears before the start of the model name. If you do not have sufficient authority, or if a model is not yet assigned to the node, this command is not available.
- **Disassign Model** – When a node has a model assigned to it, this command disassociates the model from the node. (Note: If you want to replace an existing model, you may use the **Set Selected Model for this Node** command.)

### 3.16.2. Attributes Pane

Roadway > Analysis > Performance Analysis... > Performance Models

Column Label	MODEL START	MODEL FINISH	Used in RSL	CONDITION STATE	Script for rem life threshold	Used for ben. calc.	INDEX YEAR COLUMN	Benefit Threshold Group
3rd Stage Cracking %	0	100	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	DISTRESS_YEAR_ID	
Faulting Value (in.)	100	0	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	DISTRESS_YEAR_ID	
Flexible Total Cracking	100	0	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	DISTRESS_YEAR_ID	
IRI AVG	40	500	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	DISTRESS_YEAR_ID	
CA MPD MM	0	4.15	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	DISTRESS_YEAR_ID	
WPCV	0	100	<input type="checkbox"/>		<input checked="" type="checkbox"/>	<input type="checkbox"/>	DISTRESS_YEAR_ID	

The Attributes pane shows all available condition attributes configured for your system. (A condition attribute is essentially a measurement index such as rutting or cracking.) They are defined in the Deteriorates column of the PMS Columns in Analysis window.

What you select in this pane determines what is displayed in the nodes of the Decision Tree pane. In other words, the models displayed in the decision tree are specific to a certain condition attribute. When you select a different attribute, the names of the models displayed in the nodes will also change. (Note: Just the displayed text that changes; the structure and branching criteria are fixed and do not change if the condition attribute changes.)

The pane provides the following columns:

- Column Label — This column shows the name of the attribute.
- Model Start — This column shows the upper boundary value for the condition attribute. The default value is 100, but it may be set to a different value.
- Model Finish — This column shows the lower boundary value for the condition attribute. The default value is 0, but it may be set to a different value.
- Used in RSL — When this check box is selected, the attribute is used to determine remaining service life (RSL).
- Condition State — This column contains an expression that changes a raw performance index score into an integer value in the range of 1 to N, where N is the number of condition states. The integer value is then assigned a label in the Setup Condition State window to provide a word or phrase to describe the integer value — for example, 1 could be Very Good, 2 could be Good, and so on to 5 being Poor. These labels appear in the Detailed Scenario Analysis window as well as reports.
- Script for Rem Life Threshold — This column provides a drop-down list that contains the various ways that the RSL threshold may be determined. This list contains all scripts configured in the Groovy Scripts window that are of the PI Threshold for RSL or Benefit type.
- Used for Ben. Calc. — When this check box is selected, the attribute is used to determine benefit.
- Index Year Column — This is the name of the network master column that contains the year that the given index value is calculated from when filling the network master file.
- Benefit Threshold Groovy Script — This column provides a drop-down list that contains the various ways that the benefit threshold may be determined. This threshold is the lower boundary below which benefit is not calculated. This list contains all scripts configured in the Groovy Scripts window that are of the PI Threshold for RSL or Benefit type.

### 3.16.3. Models Pane

Models and Expressions					
PMS MODEL NAME	Model Type	MODEL EXP	Comments	User Update	Date Update
CA_IRI_AVG-PCC, Unk Power		88.480781759779 + -0.20625	CA_IRI_AVG		
CA_IRI_AVG-PCC, Unk Power		88.480781759779 + -4.0E 628	CA_IRI_AVG		
CA_IRI_AVG-PCC, Unk Power		88.480781759779 + -2.6E 627	CA_IRI_AVG		
lin life 27.5	Linear	100 + -1.8182 * T	SYSTEM	5/12/2006	
lin life 26.4	Linear	100 + -1.8939 * T	SYSTEM	5/12/2006	
lin life 24.2	Linear	100 + -2.0661 * T	SYSTEM	5/12/2006	
lin life 22	Linear	100 + -2.2727 * T	SYSTEM	5/12/2006	
lin life 20.2	Linear	100 + -2.4752 * T	SYSTEM	5/12/2006	

The Models pane shows all available performance models. These are created and maintained in the Setup Models window (see page 71).

When you right-click a record (or group of records) in this pane, a shortcut menu is displayed with the following special command:

- **Show Selected Models** – For the model(s) selected in the table, this command displays a new window that shows the selected models on a graph of time versus condition. (Select multiple rows by using either CTRL+click or SHIFT+click.)

#### NOTE

The permitted upper value along the Y axis is always 100. If the function of the curve is to exceed 100, then the system will display the curve as a flat line at 100 until the function of the curve returns to a value less than 100. Similarly, the X axis always displays 50 years – and if the curve reaches 0 before 50 years, the system will display the curve as a flat line along the X axis from the X-intercept to 50 years.

## 3.17. Models

(Roadway > Setup > Performance Setup... > Performance Models List)

Models and Expressions		
* PMS MODEL NAME	* Model Type	MODEL EXP
a-high;beta-high,ro-high	Sigmoidal	100 + -80 * EXP( -1 * (20 / T ^ (3)))
a-high;beta-high,ro-low	Sigmoidal	100 + -80 * EXP( -1 * (10 / T ^ (3)))
a-high;beta-high,ro-med	Sigmoidal	100 + -80 * EXP( -1 * (15 / T ^ (3)))
a-high;beta-low,ro-high	Sigmoidal	100 + -80 * EXP( -1 * (20 / T ^ (1)))
a-high;beta-low,ro-low	Sigmoidal	100 + -80 * EXP( -1 * (10 / T ^ (1)))
a-high;beta-low,ro-med	Sigmoidal	100 + -80 * EXP( -1 * (15 / T ^ (1)))
a-high;beta-med,ro-high	Sigmoidal	100 + -80 * EXP( -1 * (20 / T ^ (2)))
a-high;beta-med,ro-low	Sigmoidal	100 + -80 * EXP( -1 * (10 / T ^ (2)))
a-high;beta-med,ro-med	Sigmoidal	100 + -80 * EXP( -1 * (15 / T ^ (2)))
a-low;beta-high,ro-high	Sigmoidal	100 + -90 * EXP( -1 * (20 / T ^ (3)))
a-low;beta-high,ro-low	Sigmoidal	100 + -90 * EXP( -1 * (10 / T ^ (3)))
a-low;beta-high,ro-med	Sigmoidal	100 + -90 * EXP( -1 * (15 / T ^ (3)))

In the Models window, you manually define deterioration models. You may then view these deterioration models.

### 3.17.1. Types of Models Supported

The system currently supports the following types of models:

- Exponential – In this type of model, the pavement condition remains nearly constant for a number of years, and then rapidly deteriorates.
- Hyperbolic – In this type of model, the pavement condition deteriorates steadily with the greatest rate of deterioration occurring in the early years.

- Inverse Exponential — This type of model is essentially the same as the Exponential model type.
- Linear — In this type of model, the rate of deterioration of the pavement condition is the same from year to year.
- Piecewise Linear — In this type of model, you specify up to six coordinates and the system draws a straight line between each coordinate. This type allows you to specify points in time where the rate of deterioration "jumps" to a new value, where it again remains constant until the next discontinuity.
- Power — This type of model is a special form of the exponential type. Like an exponential-type graph, the pavement condition remains nearly constant for a large number of years, and then rapidly deteriorates. However, the rate of deterioration is based directly on time rather than a logarithm.
- Sigmoidal — In this type of model, the pavement condition remains nearly constant for a number of years, then rapidly deteriorates, and then remains nearly constant again.

You assign a model type to each user-defined model by using the drop-down list in the Model Type column. You then set the model's parameters by using the **Edit Model Parameters** command (which is found on the right-click shortcut menu). These parameters dictate the model's rate of deterioration from a starting constant value.

In addition to user-defined models, models may also be generated via regression analysis in the Performance Models window (see page 68 for a description of this window). You can tell these types of models from user-defined models by looking at the User Update column. This column shows System for models defined via regression analysis. Also, for these models, you are not allowed to edit the parameters.

Models defined and viewed here are assigned to road sections in the Performance Models window and the Section Performance window (see page 76).

### **3.17.2. Description of the Right-click Shortcut Menu Commands**

When you right-click a record in the table, the system displays a shortcut menu with the common commands along with the following special commands:

- **Show Selected Models** — For the model(s) selected in the table, this command displays a new window that shows the selected models on a graph of time versus condition. (Select multiple rows by using either CTRL+click or SHIFT+click.)

#### **NOTE**

The permitted upper value along the Y-axis is always 100. If the function of the curve is to exceed 100, then the system will display the curve as a flat line at 100 until the function of the curve returns to a value less than 100. Similarly, the X-axis always displays 50 years — and if the curve reaches 0 before 50 years, the system will display the curve as a flat line along the X-axis from the X-intercept to 50 years.

- **Edit Model Parameters** — This command is only available for user-defined models (that is, those that do not show System in the User Update column). When available, it allows you to view and adjust the parameters that determine the model's rate of deterioration for the model that you right-clicked. After activating this command, the system displays a dialog box that you use to adjust the parameters. Once the parameters are adjusted as desired, click **OK** to save the adjusted model parameters.

(You may abort the change process, and return to the previous parameter settings, by clicking **Cancel**.)

#### NOTE

All model equations start with a constant value from which the rate of deterioration is subtracted. This constant value should not exceed 100. If a greater value is entered, the system will use 100 instead.

### 3.18. Default Model Structure

The Default Model Structure window shows the decision tree that determines what deterioration model to use for a particular type of road. "Type of road" means not only the type of road and how the road section is constructed but also the condition attributes associated with that road section.

Each node of the tree represents the point at which a decision must be made. The tree then branches depending on the result of the decision. You select what decision variable is to be used in making the decision as well as the values for that variable that cause the different branches from the node to be selected. The tree is read from left to right, with the leftmost node representing the most general and the rightmost nodes the most specific.

The decision tree configured here is used in the Performance Models window, where a deterioration model is assigned to the rightmost nodes for each condition attribute.

#### 3.18.1. Description of the Window



The Default Model Structure window shows a decision tree. Decision trees consist of parent nodes, child nodes, and the branches between them. The nodes are shown as boxes and the branches are the lines connecting the boxes. A parent node requires a user-selected decision variable and user-defined values to distinguish, and branch to, the parent's child nodes. The decision variables that may be used are selected in the Perf Class Variable column in the PMS Columns in Analysis window.

Each box in the decision tree shows the decision variable that is configured for that node. (If no decision variable is configured, the phrase No Model is shown.)

To view the values for the decision variable, right-click the node and then click **Edit Decision Var Limits**. Alternately, hover over one of the child nodes until the system displays a text box, which will show the value that causes that child node to be selected.

When you right-click a node, the system displays a shortcut menu with the common commands along with the following special command:

- **Edit Decision Var Limits** — This command is only available for parent nodes (that is, nodes that have child nodes). After selecting this command, the system displays a dialog box in which you select the decision variable and values that will be used to determine which tree branch (performance model) will be followed.

### **3.18.2. How to Add a Node**

#### **NOTE**

You may only add children to a node if unused decision variables are available.

To add a branch to an existing node in the Default Model Structure window, right-click the node to which the branch will be added and then click **Add Branch**.

If the node did not have any existing child nodes, the system creates two new child nodes. Each child node will be given the default description of No Model.

If the node already had child nodes, then the system will create only one new child node. This child node will be given the default description of No Model.

After the node(s) are created, right-click the parent node and select the **Edit Decision Var Limits** command to select the decision variable and set the values that determine when the new child node(s) will be activated. See the following section for more information.

### **3.18.3. How to Set Decision Parameters**

Only parent nodes have parameters. These parameters determine which path through the decision tree is followed depending on the conditions encountered in the performance model. In other words, these parameters distinguish one child node from another.

To set the parameters for a parent node:

1. Right-click the parent node that will have its parameters set and then click **Edit Decision Var Limits**.

The system displays the dialog box for setting the node's parameters. The Variables pane on the left side of the dialog box shows the decision variables that may be used for this node. Note that in any branch of the decision tree, a decision variable may only be used once. If no decision variables are shown in the left pane, this means all decision variables have been used in this branch. To add decision variables, navigate to the PMS Columns in Analysis window and select the desired variables in the Perf Class Variable column.

The Limits pane on the right side of the dialog box shows the value assigned to each child node for the variable selected in the left pane. (Child nodes are identified by a number, with node number 1 at the top of the window.)

Variables may be configured to have either numeric or character values. If the variable is configured to have numeric values, then the right pane shows a row for each child node. Each row shows the upper threshold at which the next child node will be activated. The upper threshold of the first child node then becomes the lower threshold of the next child node, and the process repeats for any additional child nodes.

If the variable is configured to have character values, then the right pane shows all character values that may be assigned to the variable. You assign a node to each character value, with that node being activated when the variable equals the assigned

character value. Note that not all values need to be assigned; use Node 0 for any character values you do not wish to assign.

In other words, with numeric values you assign threshold values to the child nodes, while with character values you assign child nodes to the values.

2. In the left pane of the dialog box, click the decision variable that you want to use.
3. If the selected variable is configured to have numeric values, in the first row of the right pane, click in the High Limit column of the first row. Type the value for the threshold when the next child node will be activated. If necessary, repeat this action for any additional rows in the table (other than the last row, which is not assigned an upper threshold).
4. If the selected variable is configured to have character values, for the first character value listed in the right pane, click the down arrow to display the list of nodes and then click the node to which this value will be assigned. (If the value is not to be assigned, select Node 0.) Repeat this action for each character value in the right pane.
5. When all values are assigned, click **OK** to close the dialog box and save the new values.

#### **3.18.4. How to Delete a Node**

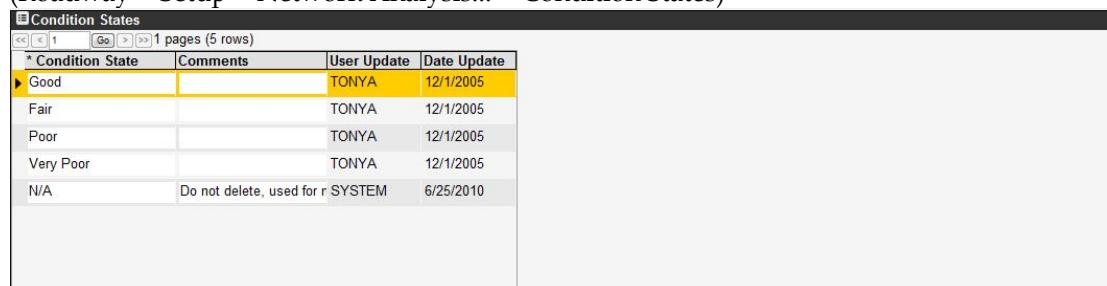
##### **NOTE**

Only child nodes may be deleted. In addition, if the deletion of a child node leaves the parent node with only one child node, then the other child node will be deleted as well.

To delete a node (and the branch associated with it), right-click the node and then click **Delete**. The system will then delete the node and branch.

### **3.19. Setup Condition State**

(Roadway > Setup > Network Analysis... > Condition States)



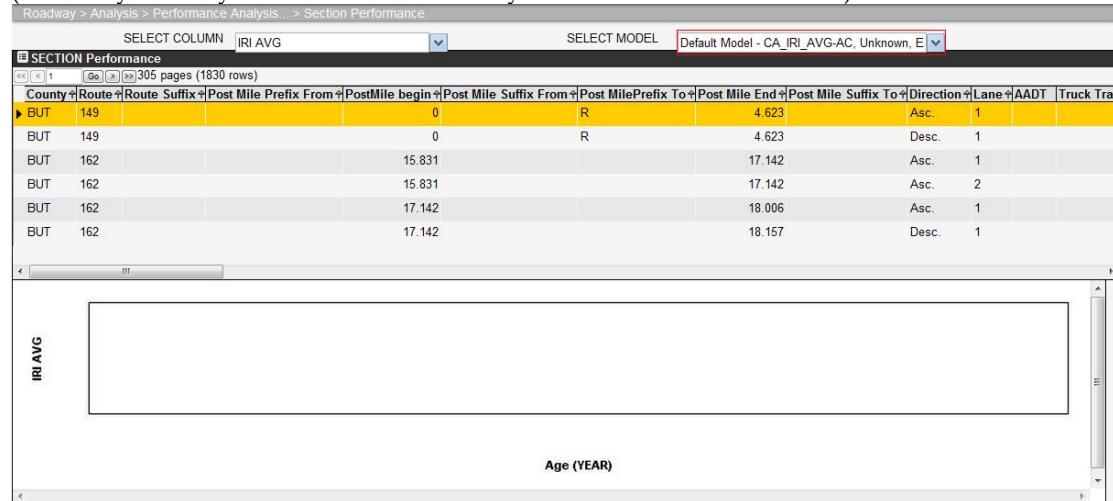
* Condition State	Comments	User Update	Date Update
Good		TONYA	12/1/2005
Fair		TONYA	12/1/2005
Poor		TONYA	12/1/2005
Very Poor		TONYA	12/1/2005
N/A	Do not delete, used for SYSTEM		6/25/2010

The Attributes pane of the Performance Models window contains a column called Condition State. This column contains an expression that changes a raw performance index score into an integer value in the range of 1 to N, where N is the number of condition states. The integer value is then assigned a label to provide a word or phrase to describe the integer value -- for example, 1 could be Very Good, 2 could be Good, and so on to 5 being Poor. These labels appear in the Detailed Scenario Results window as well as reports.

This window, the Setup Condition State window, is where the labels for the integer values are created and maintained. The records in this window are generally in order from 1 to N (from the top to the bottom) as indicated by the value in the Native ID column.

## 3.20. Section Performance

(Roadway > Analysis > Performance Analysis... > Section Performance)



The screenshot shows the 'SECTION Performance' window. At the top, there are two dropdown menus: 'SELECT COLUMN' set to 'IRI AVG' and 'SELECT MODEL' set to 'Default Model - CA\_IRI\_AVG-AC, Unknown, E'. Below these are two tables. The first table, titled 'SECTION Performance', has columns for County, Route, Route Suffix, Post Mile Prefix From, Post Mile begin, Post Mile Suffix From, Post Mile Prefix To, Post Mile End, Post Mile Suffix To, Direction, Lane, AADT, and Truck Traf. It contains several rows for 'BUT' sections. The second table, titled 'IRI AVG', has a single column 'Age (YEAR)' with a value of 4. Below the tables is a large empty rectangular area labeled 'IRI AVG' on its left side.

You use the Section Performance window to view the performance model assigned to each road section as well as set section-specific performance models.

The Section Performance window contains two panes: the Sections (top) pane and the Graph (bottom) pane. It also contains two fields with drop-down lists:

- Select Column – This field shows the condition attribute that will be graphed and what data will be used for regression.
- Select Model – For the condition attribute selected in the Select Column field and for the road section selected in the Section pane, this field shows the model that generated the curves in the Graph pane.

The drop-down list in this field contains either one or two entries. When the road section uses the Default Model Structure performance model, then only that model appears in the drop-down list. When the road section uses a model assigned (in this window) specifically to this road section, then two models appear in this drop-down list: the section-specific model (the name for which starts with Section Model) and the model that would be assigned if the Default Model Structure was used (the name for which starts with Default Model).

### NOTE

If a road section uses a section-specific model, then this model will be the model selected in the Select Model field and shown in the graph when this window first opens.

### 3.20.1. Description of the Sections Pane

The Sections pane in the upper part of the window lists all available road sections. When you select a section, its performance model is shown in the Graph pane (within the constraints applied by the selections in the Select Column and Select Model fields).

When you select a road section in the table, the system re-draws the graph to show the model associated with this road section and selected in the Select Model field. (The available models in the drop-down list for this field will also change to show what may be selected for the road section and condition attribute selected in the Select Column field.)

### 3.20.2. Description of the Graph Pane

The Graph pane shows the data and two deterioration curves for the currently selected road section, condition attribute, and performance model. The two deterioration curves both use the selected model. One curve is produced by applying the model from the year last rehabilitated. The other curve is produced by applying the model from the latest year's condition value.

#### NOTE

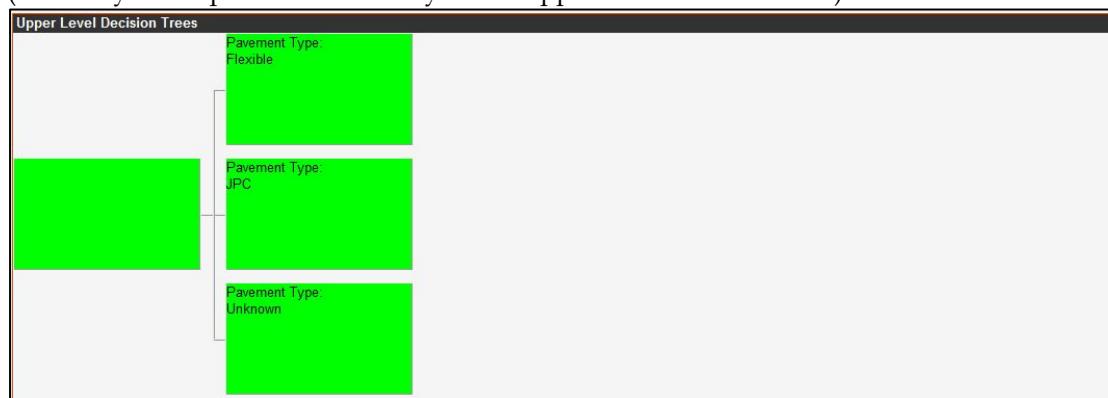
When you hold your mouse cursor over a point or line within the graph, the application displays a popup window showing the Index and Year values at that point.

When you right-click on a node in the hierarchy, the following special commands are available:

- **Revert to Default Model** – When you select this command, the section-specific model currently in use for the selected section is removed and replaced with the section's Default Model Structure model. This command is enabled when a section-specific model is active.

### 3.21. Upper Level Decision Tree

(Roadway > Setup > Network Analysis... > Upper Level Decision Tree)



The Upper Level Decision Tree window configures all but the lowest level of the treatment decision tree shown in the left pane of the Selection tab of the Decision Trees window (which is described on page 84).

This window defines the upper portion of the decision tree for treatments where, typically, the type of road surface (for example, asphalt or concrete) and road functional class (for example, interstate, primary, or secondary) are defined and related to each other. (The lower portion of the decision tree for treatments, where specific road conditions such as roughness or traffic count determine a treatment, are configured in the Decision Trees window.)

The text in each box in the decision tree is an abbreviated description. To view the full text (including a description of the level's setting), hold your mouse cursor over the box until the system displays the full text in a popup window.

When you right-click a node in the hierarchy, the system displays a shortcut menu with the following special commands:

- **Edit Decision Var Limits** – This command is only available for nodes that have children. After selecting this command, the system displays a new window in which you select the variables and values that determine which tree branch will be selected.

The set of decision tree variables that may be selected are configured in the Upper Level Decision Tree column in the PMS Columns in Analysis window.

- **Add Branch** – This command adds a node or nodes as children beneath the node you right-clicked. After selecting this command the system displays a dialog box so you may enter the number of nodes needed.
- **Edit Node Name** – This command displays a dialog box so you may edit the name of the node.
- **Set Item Height** – This command displays a dialog box that shows the current vertical dimension of the node boxes (in pixels). You may modify this value to change the vertical size of all node boxes.

## 3.22. Treatments

You define treatments for long-term projects in this window, which allows them to be available for optimization analysis. Treatments are used in all analyses with decision trees to provide an estimate of the treatment type and costs of recommended future projects as well as the condition consequences of these projects. The system provides you with the ability to add, remove, and maintain as many treatments as desired.

### 3.22.1. Description of the Treatments Window

(Roadway > Setup > Network Analysis... > Treatments)

The screenshot shows the 'Treatments' window with three main panes:

- Treatments pane:** A grid of treatment records. One record, 'Seal Cracks', is highlighted. The columns include Treatment, Treatment Name, Unit Cost, Selection Priority, Exclusion Priority, Exclusion Years, Date Update, User Update, and Comments.
- Improvements pane:** A grid of improvement records. One record, 'CA\_FLX\_TCV', is highlighted. The columns include Condition Attributes, Date Update, User Update, and Comments.
- Other Improvements pane:** A grid of other improvement records. One record, 'Work Code Seal Cracks', is highlighted. The columns include Changing Attributes, Condition Improvement Script, Other, Date Update, and User Update.

The Treatments window provides the following panes: Treatments, Improvements, and Other Improvements. The difference between the Improvements pane and the Other Improvements pane is that the Improvement pane is for attributes that vary according to a performance model (that is, performance index values such as IRI or roughness) and the Other Improvements pane is for attributes that do not vary according to a performance model (such as traffic or pavement age).

For example, a treatment could be to apply a 2" overlay. The resulting improvements from this treatment could then be to improve the IRI value, reset the pavement age to zero, and reduce bridge clearances by 2". The Improvements pane would show how the IRI value changes and the Other Improvements pane would change the pavement age and bridge clearances.

#### Treatments Pane

The Treatments pane lists all treatments used for long-term projects and the costs associated with each. It includes the following columns:

- Treatment Name

This column provides the name of the treatment.

- Unit Cost

This column shows the unit costs for the treatment. The costs are calculated using the method selected in the Cost column.

- Selection Priority

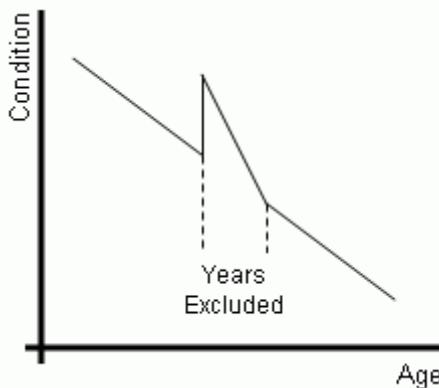
This column shows the Treatment Priority number, which determines which treatment from the decision trees should be used in optimization analysis. More than one decision tree may be defined for each bridge element. For a given performance class and condition attribute, multiple decision trees may also be defined. During the decision-making process, a recommended rehabilitation from each of these decision trees is calculated. The rehabilitation with the highest treatment priority number becomes the selected strategy. **Treatment priority must be a unique number.** The user should assign treatment priority based on local knowledge and your agency's policy. This number is not used for INCBEN analysis.

- Exclusion Priority

This column sets the value for the treatment exclusion priority, which sets the priority of this treatment in regard to other treatments. When the selected treatment is applied, then all other treatments with equal or lower treatment exclusion priority values will not be applied for the number of years given in the Year column.

- Exclusion Year

This column shows exclusion years. For those treatments with a future deterioration type of either Improvement Deteriorates in N Years or Increases Index RSL by N Years (as set in the Future Detr Type column in the Improvements pane), the value in this column indicates how long the improvement in condition should last until the condition returns to that experienced by the pavement section (according to its underlying performance model). The graph below shows how these years are applied to a maintenance treatment:



Exclusion years define the minimum number of years after a treatment is performed before another treatment with equal or higher treatment exclusion priority value can be assigned to a pavement management section. (The treatment exclusion priority value is configured in the Treatment PRCD column.)

- Cost

This column specifies the method by which the unit cost is calculated. The method is selected from a drop-down list that contains the available unit-cost calculations.

- Budget Category

This column shows the budget category assigned to the treatment. The budget category is typically configured to be one of three values: (1) Reconstruction; (2) Rehabilitation; or (3) Resurfacing. The value for the budget group is set in the Calculated Expressions window.

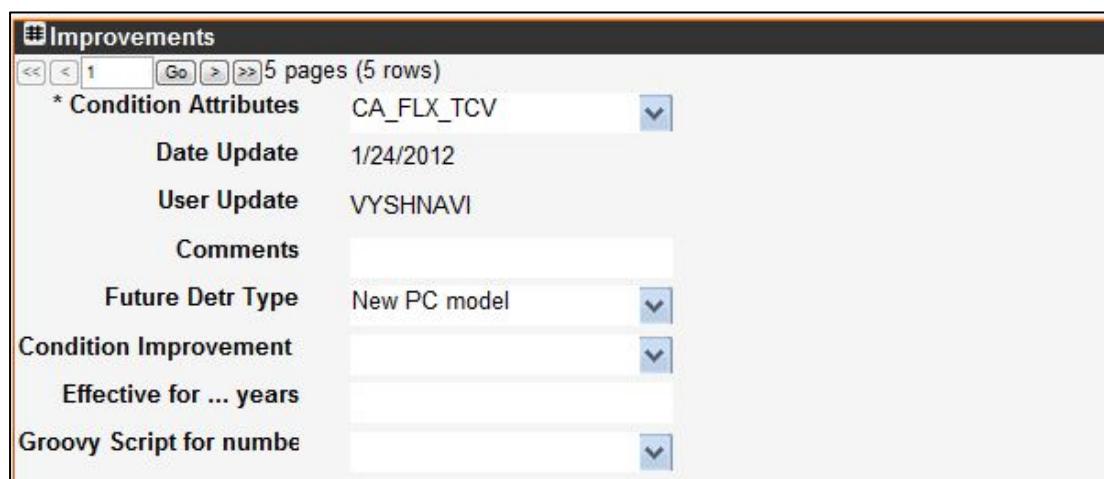
During optimization analyses that are constrained by budget, the system assigns a treatment to a bridge element and attempts to subtract the cost of the treatment from the budget category assigned in this column. If the budget category has sufficient funds for the treatment, the treatment is assigned to the bridge element and the budget category's value is decremented by the treatment's cost. On the other hand, if the budget category has insufficient funds to cover the cost of the treatment, then the bridge element is skipped and is not considered for improvement in the current analysis year.

For analyses that are not constrained by budget, the system ignores this column.

- Work Code

This column specifies the work code for the treatment. The work code is selected from a drop-down list. Note that if a work code is not assigned to a treatment, any line item in a work plan that shows the treatment cannot be used to generate a construction history record.

### **Improvements Pane**



The screenshot shows the 'Improvements' pane with the following data:

Condition Attributes	CA_FLX_TCV
Date Update	1/24/2012
User Update	VYSHNAVI
Comments	New PC model
Future Detr Type	New PC model
Condition Improvement	
Effective for ... years	
Groovy Script for number	

After the treatment selected in the Treatments pane is applied, the condition attributes (also called performance indices) listed in this pane (the Improvements pane) will change according to the rules configured in this pane. (The condition attributes not listed in this pane are unaffected by the treatment and will not change.)

The Improvement pane contains the following column:

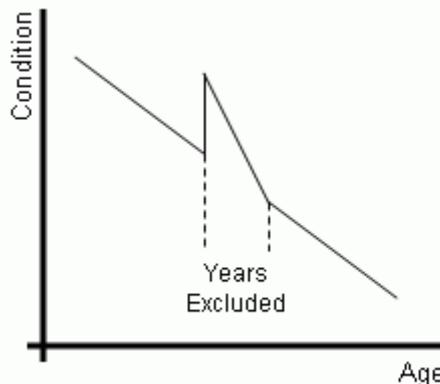
- Condition Attributes – This column provides a drop-down list that contains all condition attributes that change according to a performance model. These condition attributes are columns in the Network Master file and have the Deteriorates check box selected in the BMS Columns in Long Term Analysis window.
- Future Detr Type – This column contains a drop-down list with the following deterioration methods:
  - Increases Index RSL by N Years – For the condition attribute, after treatment and improvement, the deterioration is set based upon the model selected using

the Default Model Structure window's decision tree, but the model's coefficients will be automatically adjusted to extend the life of the road section by N years (based upon the RSL threshold as defined in the Performance Models window).

This life extension is over and above the life extension due to the improvement itself. (The value of N is either set directly in the Effective for Years column or results from the Groovy script specified in the Groovy Script for Number of Years column.)

- Improvement Deteriorates in N Years — For the condition attribute, after treatment and improvement, the deterioration is set to return to the "before treatment" model after N years after which the "before treatment" model is used. (The value of N is either set directly in the Effective for Years column or results from the Groovy script specified in the Groovy Script for Number of Years column.)

Note: For those treatments with a future deterioration type of either Improvement Deteriorates in N Years or Increases Index RSL by N Years (as set in the Future Detr Type column in the Improvements pane), the value in this column indicates how long the improvement in condition should last until the condition returns to that experienced by the pavement section (according to its underlying performance model). The graph below shows how these years are applied to a maintenance treatment:

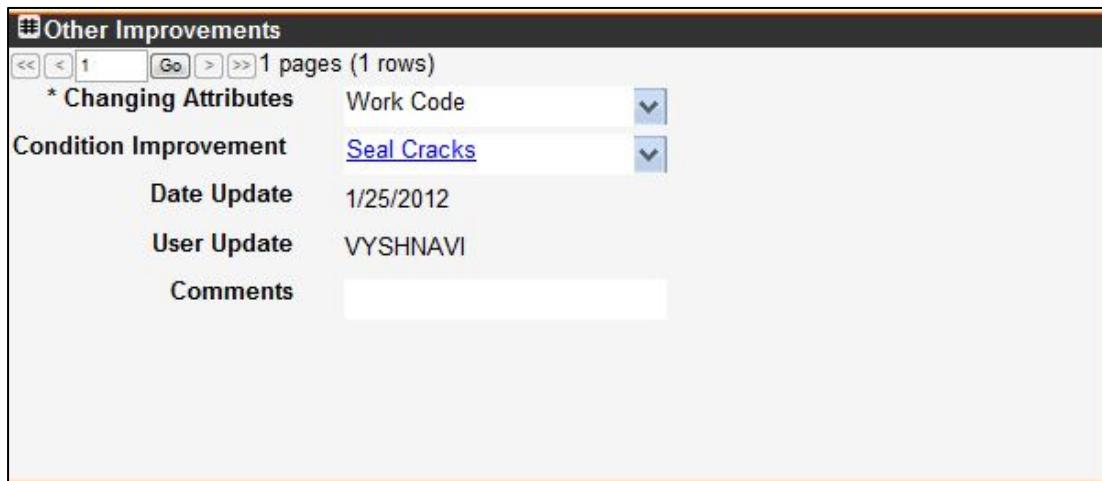


- New PC Model — A new deterioration model for the condition attribute that deteriorates is applied based upon the model selected using the Default Model Structure window's decision tree.
- Section Model If Exists, Otherwise New PC Model — If there is a model for the section, then, after treatment, it will be used; otherwise, the model selected using the Default Model Structure window's decision tree will be used. Note: Section models are always used during the first years of analysis prior to treatment. The same section model will continue to be used after treatment if you select this option.
- Condition Imp — This column is provided for those implementations that use the Scenario Analysis window (that is, those implementations that use prioritization analysis). In these implementations, this column shows the expression for determining the "condition improvement" estimate. The expression is selected from a drop-down list, the contents of which are configured by the After Treatment Improvement parameter that is found in the Calculated Expressions window (see page 53 for more information).
- Effective for ...Years — This column specifies how the N value for the two future deterioration types Increases Index RSL by N Years and Improvement Deteriorates in N

Years is determined. Provided the Groovy Script for Number of Years column is null, the system uses the value entered in this column directly for the N value. (If the Groovy Script for Number of Years column is not null, the N value from the Groovy script overrides any value entered in this column.)

- Groovy Script for Number of Years — This column specifies how the N value for the two future deterioration types Increases Index RSL by N Years and Improvement Deteriorates in N Years is determined. When a script is selected in this column, the system uses it to calculate the N value (and ignores the Effective for ... Years column).

### ***Other Improvements***



The screenshot shows the 'Other Improvements' pane with a single record displayed. The record details are as follows:

* Changing Attributes	Work Code
Condition Improvement	Seal Cracks
Date Update	1/25/2012
User Update	VYSHNAVI
Comments	[Empty]

After the treatment selected in the Treatments pane is applied, the non-PI-based attributes listed in this pane (the Other Improvements pane) will change according to the rules configured in this pane. When multiple records appear in this pane, the order in which the attributes are updated is set by the value in the Update Order column of the PMS Columns in Analysis window.

The Other Improvement pane contains the following columns:

- Changing Attributes — This column provides a drop-down list that contains all non-PI-based attributes (that is, attributes that don't change according to a performance model, such as pavement age or traffic). These attributes are columns in the Network Master file and have the Restart with Treatment check box selected in the BMS Columns in Long Term Analysis window.
- Condition Improvement Script Other — This column shows the name of the Groovy script that determines the "condition improvement" estimate. The script is selected from a drop-down list, the contents of which are configured in the Groovy scripts window under the Condition Improvement type of Groovy script.

#### **NOTE**

Rather than editing a script in the Groovy scripts window, you may instead modify a script by clicking the name, which then displays the Edit Groovy Script dialog box.

### **3.22.2. How to Edit a Treatment**

To add or edit a treatment:

1. Display the Treatments window.

2. In the left pane, click the treatment to be edited. To add a new treatment, right-click and select **Insert**.
3. Edit or enter a unique treatment code (user defined).
4. Edit or enter the name of the treatment.
5. Edit or enter the unit cost.
6. Edit or enter a unique number for the treatment priority.
7. Enter or edit the number of years that a treatment should not be applied to the section or how long the effect of maintenance should last (for example, four years). Recall that if condition warrants, a treatment with lower Exclusion Priority can be applied to the section.

To configure the improvements resulting from the treatment, if any, see the following section.

### **3.22.3. How to Configure Improvements**

If the default condition consequences for a treatment resulting from optimization analysis are not what you desire, follow these steps to override the default values:

1. Display the Treatments window.
2. In the Treatments pane, click the treatment to be edited.
3. In the Improvements pane, right-click and then click **Insert**. The system adds a new record to the pane.
4. In the Condition Attributes column in the new row, click the down arrow and then click the desired condition attribute.
5. If your implementation shows the Condition Improvement column, click the down arrow and then click the desired condition improvement expression.
6. In the Future Detr Type column, click the down arrow and then click the deterioration method.
7. If you selected the Increase Index RSL by N Years or Improvement Deteriorates in N Years method in step 6, select the way in which N is calculated in the Effective for ...Years column.
8. Repeat steps 3 through 7 for any additional condition attributes that are affected by the treatment.
9. If non-PI-based attributes will also change as a result of the treatment, right-click in the Other Improvements pane and then click **Insert**. The system adds a new record to the pane.
10. In the Changing Attributes column in the new row, click the down arrow and then click the desired attribute.
11. In the Condition Improvement Script Other column, click the down arrow and then click the name of the Groovy script that will determine condition improvement.
12. Repeat steps 9 through 11 for any additional non-PI-based attributes that are affected by the treatment. (Note: If you configure multiple records in the Other Improvements pane, be sure the attributes are updated in the proper order by checking the values in the Update Order column in the PMS Columns in Analysis window.)
13. Click  to save the new information.

### 3.23. Decision Trees

The decision tree for pavement treatment defines the rules that indicate need for pavement treatment. The tree contains both the need for treatment and the type of treatment required.

In AgileAssets software, a full decision tree is the combination of an upper-level decision tree (built in the Upper Level Decision Tree window; see page 77) and a lower-level decision tree. Lower-level decision trees are created and attached as an extension of the upper-level decision tree in this window, the Decision Trees window (Roadway > Setup > Network Analysis... > Decision Trees).

Lower-level decision trees are grouped in sets. The software supports an unlimited number of decision tree sets, which gives you great flexibility in creating various decision trees for use in optimization analysis.

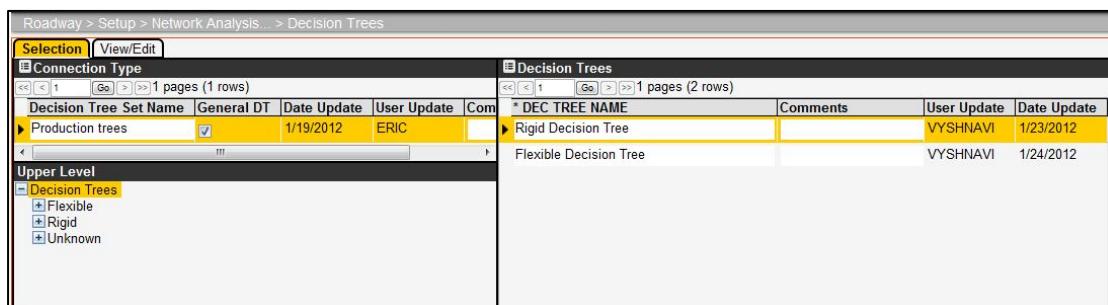
Each decision tree set is categorized as either a General decision tree set or an INCBEN decision tree set. The differences between General and INCBEN decision tree sets are described below:

- General Decision Tree Sets – For a General decision tree set, several lower-level decision trees may be assigned to the upper-level portion of the decision tree. The lower-level decision trees may have only one treatment assigned per limb. During analysis, then, a road section can have a treatment recommended for each lower-level decision tree that is bundled together. The final single recommended treatment is the one with the highest treatment priority (see the Treatments window on page 78 for more information).
- INCBEN Decision Tree Sets – For an INCBEN decision tree set, only one lower-level decision tree may be assigned to the upper-level portion of the decision tree. The lower-level decision trees may have up to three treatments assigned per limb. During analysis, then, a road section can have several recommended treatments. The final single recommended treatment is determined internal to the INCBEN algorithm, which uses a sliding scale where:
  - The treatment with the highest benefit/cost ratio is chosen when budgets are lean; and
  - The treatment with the highest benefit is chosen when budgets are ample.

#### 3.23.1. Decision Trees Window Description

The Decision Trees window (Roadway > Setup > Network Analysis... > Decision Trees) contains two tabs: the Selection tab and the View/Edit tab. The following sections describe these tabs in more detail.

##### Selection Tab



DEC TREE NAME	Comments	User Update	Date Update
Rigid Decision Tree		VYSHNAVI	1/23/2012
Flexible Decision Tree		VYSHNAVI	1/24/2012

The Selection tab contains the following panes: Decision Tree Sets, Upper Level, and Decision Trees. These panes are described in more detail below:

- Decision Tree Sets Pane

The Decision Tree Sets pane shows the available decision tree sets. The Standard Decision Tree? check box indicates whether the set is a General or an INCBEN decision tree set, with a check mark indicating General.

- Decision Trees Pane

The table shown in the Decision Trees pane lists all decision trees created for all decision tree sets.

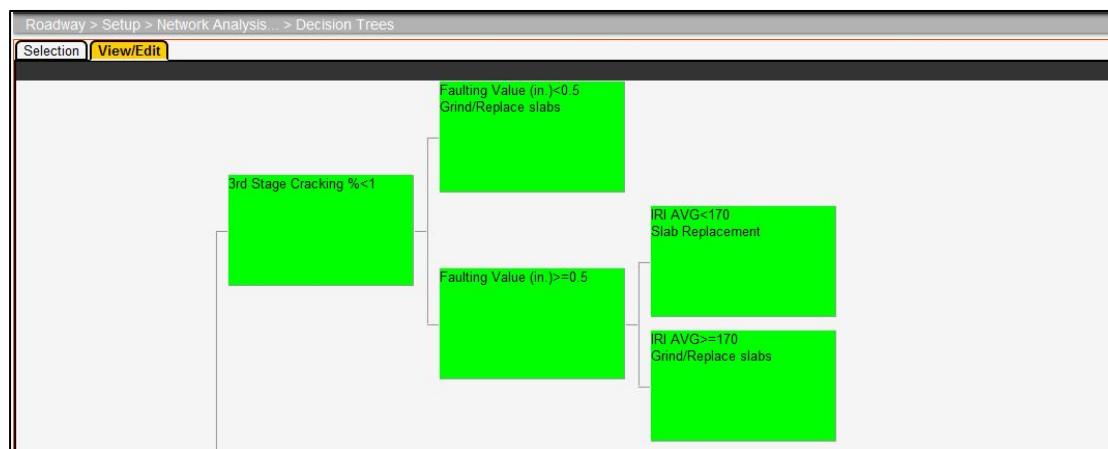
- Upper Level Pane

For the decision tree set selected in the Decision Tree Sets pane, the Upper Level pane shows the upper-level decision tree (as configured in the Upper Level Decision Tree window) and what lower-level decision tree(s) are assigned to each node of the upper-level decision tree. (For General decision trees, multiple lower-level decision trees may be assigned to each node of the upper-level decision tree; for INCBEN decision trees, only one lower-level decision tree may be assigned to each node of the upper-level decision tree.)

When you right-click the lowest node in the upper-level decision tree shown in the pane, the following special commands are available:

- **Assign Selected Tree** — **Only execute this command from the lowest level of the upper-level decision tree.** For the lower-level decision tree selected in the Decision Trees pane, this command attaches the lower-level decision tree to the node of the upper-level decision tree that you right-clicked. See page 88 for further information on assigning a lower level decision tree.
- **Remove Tree** — **Only execute this command from the lowest level of the upper-level decision tree.** Once a lower-level decision tree is attached to a node of the upper-level decision tree, this command becomes available to disassociate the lower-level decision tree from the upper-level decision tree node.

#### View/Edit Tab



Note: For INCBEN decision trees, treatments must be ordered by cost with the:

- Lowest cost treatment in the treatment (top) position.
- Next lowest cost treatment in the first alternative treatment (middle) position.
- Next (second) lowest cost treatment in the second alternative treatment (bottom) position.

For the lower-level decision tree selected in the Decision Trees pane of the Selection tab, the View/Edit tab shows the nodes of the lower-level decision tree. You use the View/Edit tab to

create and maintain lower-level decision trees by editing their structure, decision thresholds, and treatment outcomes. For parent nodes, the decision variable is shown. For child nodes, the treatment is shown (or treatments, for INCBEN decision trees). The decision tree flows from left to right, and the nodes are numbered from the top to the bottom.

When you right-click a node in the decision tree, the following special commands are displayed:

- **Edit Decision Var Limits** – This command is only available for nodes that have children. After selecting this command, the system displays a new window in which you select the index and thresholds that will be used to determine which treatment alternative (tree branch) will be selected. The set of decision tree variables that may be selected are configured in the Decision Variables column in the PMS Columns in Analysis window.
- **Edit Treatment** – This command is only available for nodes that do not have children. After selecting this command, the system displays a new window in which you select the treatment to be applied.
- **Edit First Treatment Alternative** – This command is only available for INCBEN decision trees and nodes that do not have children. After selecting this command, the system displays a new window in which you select the treatment to be applied if the indicated treatment cannot be applied.
- **Edit Second Treatment Alternative** – This command is only available for INCBEN decision trees and for nodes that do not have children. After selecting this command, the system displays a new window in which you select the treatment to be applied if neither the indicated treatment nor the first alternative can be applied.

### **3.23.2. How to Create a New Decision Tree Set**

As discussed in the introduction, decision trees are grouped in sets. These are shown in the Decision Tree Sets pane. To create a new set, follow these steps:

1. Display the Decision Trees window.
2. In the Decision Tree Sets pane, right-click and then click **Insert**. A new record is added to the pane.
3. In the Decision Tree Set Name column of the new record, enter a name for the set. (The name you enter here will appear in the drop-down list in the Optimization Analysis window.)
4. If this set is for General decision trees, click the check box in the Standard Decision Tree? column. Otherwise, for INCBEN decision trees, leave the check box clear.
5. Click the  icon to save the new information.

### **3.23.3. How to Create a New Lower-level Decision Tree**

To create a new lower-level decision tree, you may find it useful to first draw the decision tree on a separate piece of paper. On the separate piece of paper, you should diagram the different levels needed in the hierarchy and what decision variables and values are required to select the branches of the decision tree. The diagram should also indicate what treatments result at the termination points of each branch in the decision tree.

Once you have your decision tree diagrammed, follow these steps to enter it into the application:

1. Display the Decision Trees window.

2. In the Decision Trees pane of the Selection tab, right-click and then click **Insert**. A new record is added to the table. (The decision tree set selected in the Decision Tree Sets pane does not affect the creation of a lower-level decision tree.)
3. Type the tree name in the Dec Tree Name column.
4. Click the View/Edit tab. The application shows a single node that is the start of the decision tree.
5. Right-click the node and then click **Add Branch**. The system displays a dialog box so you may enter the number of nodes required.
6. Using the hierarchy you created on the separate piece of paper, enter in the dialog box the number of nodes needed below the node you right-clicked and then click the **OK** button. The system adds the entered number of nodes to the decision tree.
7. Repeat steps 5 and 6 to create the hierarchy you diagrammed on the separate piece of paper.
8. Once the decision tree is created, the next step is to add the decision variables and values that govern what branches are selected. Right-click a parent node and then click **Edit Decision Var Limits**. The system displays a dialog box so you may select the desired decision variable. (For INCBEN analysis, this should be the least expensive candidate treatment.)
9. In the left pane of the dialog box, click the desired decision variable. The system responds by configuring the right pane for entry of the values that cause each child node to be selected.
10. Configure the right pane of the dialog box to appropriately select each child node.
11. Click **OK** to close the dialog box.
12. Repeat steps 8 through 11 for the remaining parent nodes in the decision tree.
13. Now that the decision variables and values are set, the final task is to assign treatments to the termination points of each branch in the decision tree. For General decision trees, you will assign one treatment; for INCBEN decision trees, you will assign up to three candidate treatments.

To assign the first treatment, right-click the first termination node and then click **Edit Treatment**. The system responds by displaying a dialog box so you may select the desired treatment.

14. In the dialog box, click the down arrow to display the list of treatments and then click the desired treatment.
15. Click **OK** to close the dialog box. The selected treatment is shown in the termination node (alternate treatments are shown by hovering over the node).
16. For an INCBEN decision tree only, right-click the same termination node and then click **Edit 1st Treatment Alternative**. Repeat steps 14 and 15 to select the first alternate treatment.
17. For an INCBEN decision tree only, right-click the same termination node and then click **Edit 2nd Treatment Alternative**. Repeat steps 14 and 15 to select the second alternate treatment.
18. Repeat steps 13 through 17 for the remaining termination nodes of the decision tree.
19. Click the Selection tab.
20. Click the  icon to save the new information.

### **3.23.4. How to Modify a Lower-level Decision Tree**

To modify a lower-level decision tree, follow the advice given below:

- To change the name of a decision tree in the table of the Selection tab, highlight the existing name and then type the new name.
- To change the treatment assigned to a termination node, right-click the node and then click the appropriate command for which treatment you wish to change.
- To change the values that determine which branch is followed, display the View/Edit tab. Then right-click the parent node and click **Edit Decision Var Limits**. In the dialog box that displays, re-configure the right pane with new values to determine what branches are followed depending on the value of the decision variable.
- If you want to change the decision variable that determines which branch is followed, you can only use a decision variable that has not been used at a different level of the decision tree. This may mean that you will have to alter the decision variables for more than one node. To change the decision variable, use the **Edit Decision Var Limits** command to select the decision variable and the values that determine what branch is followed.
- If you want to remove a node from the decision tree, only child nodes may be deleted. Furthermore, if a parent only has two child node and you delete one of them, the system will automatically delete the other as well. To delete a node, right-click the child node and then click **Delete**.

### **3.23.5. How to Assign a Lower-level Decision Tree**

To join a lower-level decision tree to an upper-level decision tree node:

1. Display the Decision Trees window.
2. In the Decision Tree Sets pane, select the appropriate decision tree set by locating the record for the desired set and then clicking the record to select it. The system responds by highlighting the record to show that it is selected and also refreshes the Upper Level pane to show the lower-level decision tree assignments for the selected decision tree set.
3. In the table in the Decision Trees pane of the Selection tab, click the record showing the lower-level decision tree to be assigned. The system highlights the record to show that it is selected.
4. In the Upper Level pane on the lower left, find the child node of the upper-level decision tree to which you want to assign the selected lower-level decision tree.
5. Right-click the child node found in the previous step and then click **Assign Selected Tree**. The system refreshes the hierarchy shown in the Upper Level pane to show the lower-level decision tree attached to the upper-level decision tree.
6. Click the  icon to save the new information.

### **3.23.6. How to Disassociate a Lower-level Decision Tree**

To disassociate a lower-level decision tree from an upper-level decision tree node, follow these steps:

1. Display the Decision Trees window.
2. In the Upper Level pane of the Selection tab, find the node that corresponds to the lower-level decision tree that will be disassociated from the upper-level decision tree.
3. Right-click the node found in the previous step and then click **Remove Tree**.

4. Click the  icon to save the new information.

### **3.23.7. How to Delete a Lower-level Decision Tree**

To delete a lower-level decision tree, follow these steps:

1. Display the Decision Trees window.
2. In the table in the Decision Trees pane of the Selection tab, right-click the lower-level decision tree to be deleted and then click **Delete**. The application checks whether the decision tree is assigned. If it is not assigned, the tree is deleted.

If the decision tree is assigned, the application displays a message that says, "This sub-tree is still assigned, do you still want to delete?" Click **Yes** if you want to delete it; otherwise, click **No**.

3. Click the  icon to save the new information.

## **3.24. MWP Project Status**

(Roadway > Setup > Network Analysis... > MWP Project Status)

Master Work Plan Project Status List				
<< <   Go   > >> 1 pages (6 rows)				
MWP Project Status	Comments	Att.	User Update	Date Updated
Programmed		EUGENE	1/6/2012	
PS&E		KSTRAUSS	8/20/2010	
Awarded		KSTRAUSS	8/20/2010	
Scenario Results		EUGENE	1/6/2012	
Scenario Recommended		ERIC	8/31/2010	
Completed		EUGENE	1/13/2012	

You use the MWP Project Status window to define the various project status levels for work plans, which are maintained in the Work Plan Data window.

The project status levels defined here are used in the following windows:

- In the Work Plan Data window, where it appears in a drop-down list and is assigned to every record in the work plan.
- In the Scenario (Network) Analysis and Optimization Analysis windows. When a work plan is included in an analysis, then what is included from the work plan can be filtered by project status using the right-click **Edit MWP Scope** command.

## **3.25. Setup RSL Categories**

(Roadway > Setup > Network Analysis... > RSL Category)

Remaining Service Life Category				
<< <   Go   > >> 1 pages (6 rows)				
* RSL Category	Comments	Att.	User Update	Date Update
RSL < 2		LREDD	11/28/2007	
RSL 2-5		LREDD	11/28/2007	
RSL 5-10		LREDD	11/28/2007	
RSL 10-15		LREDD	11/28/2007	
RSL 15-20		LREDD	11/28/2007	
RSL > 20		LREDD	11/28/2007	

The Setup RSL Categories allows you to create groups of remaining service life (RSL) values. These groups are then used in summary reports that provide information on the RSL for certain routes in your transportation network.

## 3.26. Work Plan Type

(Roadway > Setup > Network Analysis... > Work Plan Type)

Work Plan Types					
* Work Plan Type	Comments	Att.	User Update	Date Update	Level ID
HQ Strategic Recommendations – Rea		EUGENE	1/11/2012		2
Approved Program (STIP)		EUGENE	1/11/2012		2
HQ Strategic Recommendations – Scr		EUGENE	1/11/2012		2
District Strategic Recommendations –		ERIC	9/23/2010		4
District Scratch Pad 1		EUGENE	1/11/2012		4
District Operations Workplan		ERIC	9/23/2010		4
HQ Strategic Recommendations- Origii		KSTRAUSS	12/30/2010		2
District Scratch Pad 2		EUGENE	1/11/2012		4
District Scratch Pad 3		EUGENE	1/11/2012		4
Approved Program (HM1)		EUGENE	1/11/2012		2
Approved Program (HA-22)		EUGENE	1/11/2012		2

The system allows you to develop multiple work plans. (A work plan is essentially a list of planned projects.) The different work plans are created in this window, while the data for each work plan is entered in the Work Plan Data window (see page 104).

### 3.26.1. Work Plan Type Window Description

The Work Plan Type window shows a simple table with each type a record in the table. The Work Plan Type column shows the name of the work plan type, which is what will appear in the drop-down lists found in the Work Plan Data, Maintenance Analysis, and Optimization Analysis windows.

### 3.26.2. How to Create a Work Plan Type

To create a new type of work plan, follow these steps:

1. Display the Work Plan Type window.
2. Right-click and then click **Insert**. A new row is added to the table.
3. In the Work Plan Type column of the new row, highlight the default name and then type the name of the new work plan type. Since this name will appear in drop-down lists, make it as specific and informative as possible.
4. Click the  icon to save the new work plan type.

## 3.27. Network Master

The Network Master table (NMF) is the starting point for all network analyses. These analyses depend on the columns included in the NMF. At a minimum, you must have the PMS\_SECTION\_ID and PAVE\_AGE columns (with UPDATE\_SOURCE) in the NMF.

The table contains the most current information for all pavement sections. It represents the current structure, traffic, classification, and condition for the network as a whole. Since the network master file provides input to analysis, it must contain information for the full network. In those cases where data is not available, default data is applied.

The NMF is the input data stream for network analysis. It must be re-calculated at least once per year for the current data set to be used in network analysis. It should also be re-calculated each time any of the condition indicator settings change.

The NMF normally uses Current PM Sections as the master for pavement section definitions. Other data (including pavement management inventory, traffic, condition in the form of performance index, and construction history) are all assigned to the NMF when it is filled.

Data for the NMF is aggregated according to rules developed for your particular version of the system, and is documented in your Configuration Document.

Network Master data is shown in two windows, one providing a data view (Roadway > Analysis > Network Analysis... > Network Master > Network Master Data) and the other a graphical view (Roadway > Analysis > Network Analysis... > Network Master > Network Master Graph). Shown below is the Network Master Data window; for the Network Master Graph window, see page 44.

Distress Year	County	Route	Route Suffix	Post Mile Prefix From	Post Mile begin	Post Mile Suffix From	Post MilePrefix To	Post Mile End	Post Mile Suffix To	Direction	Lane
2011	NEV	80		R	6.475 R		R	6.729 R		Asc.	2
2011	NEV	80		R	6.729 R		R	6.841 R		Asc.	2
2011	NEV	80		R	6.195 R		R	6.307 R		Asc.	2
2011	NEV	80		R	5.479 R		R	5.648 R		Asc.	2
2011	NEV	80		R	5.76 R		R	5.872 R		Asc.	2
2011	NEV	80		R	5.648 R		R	5.76 R		Asc.	2
2011	NEV	80		R	6.841 R		R	6.978 R		Asc.	2
2011	NEV	80		R	6.002 R		R	6.195 R		Asc.	2
2011	NEV	80		R	5.261 R		R	5.479 R		Asc.	2
2011	NEV	80		R	5.872 R		R	6.002 R		Asc.	2
2011	NEV	80		R	6.307 R		R	6.475 R		Asc.	2
2011	NEV	80		R	5.153 R		R	5.226 R		Asc.	2
2011	NEV	80		R	5.261 R		R	5.479 R		Asc.	3
2011	NEV	80		R	7.842 R		R	9.037 R		Asc.	2
2011	NEV	80		R	6.475 R		R	6.729 R		Asc.	1
2011	PLA	80		R	64.178 L		R	64.259 L		Desc.	2
2011	NEV	80		R	9.065 R		R	11.395 R		Asc.	2

### 3.28. Section Current Needs

(Roadway > Analysis > Network Analysis... > Section Current Needs)

Roadway > Analysis > Network Analysis... > Section Current Needs											
Road Sections											
<a href="#">Road Sections</a>											
County	Route	Route Suffix	Post Mile Prefix From	PostMile begin	Post Mile Suffix From	Post MilePrefix To	Post Mile End	Post Mile Suffix To	Direction	Lane	
BUT	149		R	1.182	R		3.169		Asc.	1	
BUT	149		R	1.196	R		3.224		Desc.	1	
BUT	149		R	3.201	R		3.633		Asc.	1	
BUT	149		R	3.261	R		3.679		Desc.	1	
BUT	149		R	3.647	R		4.623		Asc.	1	
BUT	149		R	3.696	R		4.623		Desc.	1	
BUT	149			0	R		1.179		Asc.	1	
Treatment Needs											
Decision Tree Set	Treatment (Alt1)	Treatment Cost (Alt1)	Treatment (Alt2)	Treatment Cost (Alt2)	Treatment (Alt3)	Treatment Cost (Alt3)	Treatment (Alt3)	Treatment Cost (Alt3)			
Production trees	HMA-Thin->0.1<=0.2	\$2,073,600.00									

The purpose of the Section Current Needs window is to show the general decision tree treatment recommendation for every section in the Network Master file. This is exactly the same set of treatments you would get if you ran a one-year prioritization method network analysis with unlimited budget.

This window contains the following panes:

- Road Sections pane — This pane is located at the top of the window and contains every available road section in the Network Master file.
- Needed Treatments pane — The pane located at the bottom of the window contains current rehabilitation recommendations. The left pane shows the recommendations resulting from general decision trees. The pane on the right shows the recommendations from INCBEN decision trees. If no treatment is recommended for the current road section, then nothing appears in a pane.

### NOTE

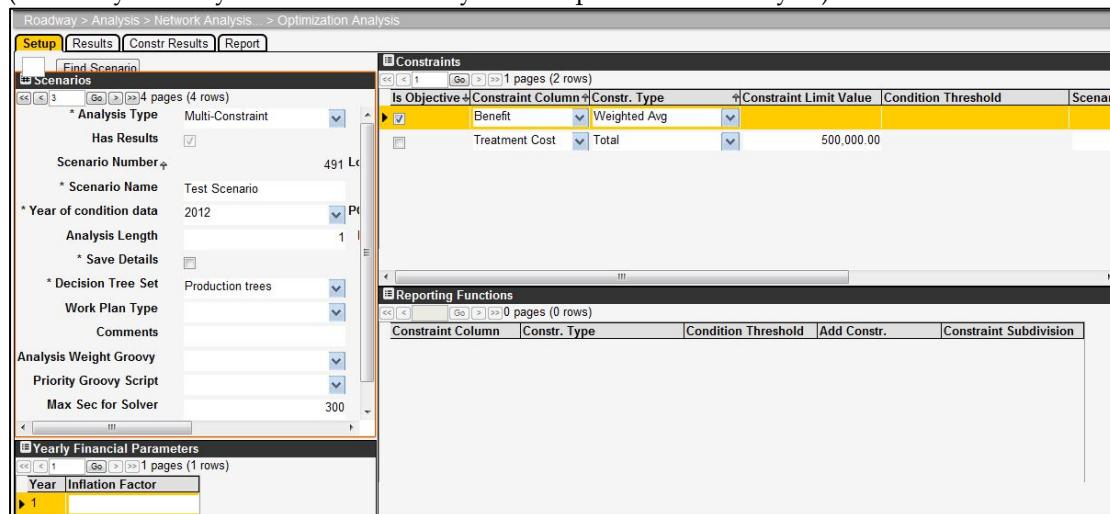
The recommended treatments in the Needed Treatments pane and the road sections in the Road Sections pane are always current. If you change your decision tree or Network Master file, these changes will be reflected in this window immediately and automatically.

## 3.29. Optimization Analysis

The purpose of optimization analysis is to create an optimal work plan using a single objective and multiple constraints (and constraint subdivisions, if so configured) across one or more years. If you are running a multiple-year optimization, each section needs to be assigned a strategy in the Section Strategies window before running the optimization routine. (See page 107 for more information on the Section Strategies window.)

### 3.29.1. Optimization Analysis Window Description

(Roadway > Analysis > Network Analysis... > Optimization Analysis)



The Optimization Analysis window is where all optimizations are performed. In this window you can define and run multiple optimizations, each having different settings, budgets, analysis periods, and so forth. The results from each optimization are stored separately and can be reviewed in this and other windows throughout the system.

The window provides the following tabs that allow you to switch between different components of project optimization:

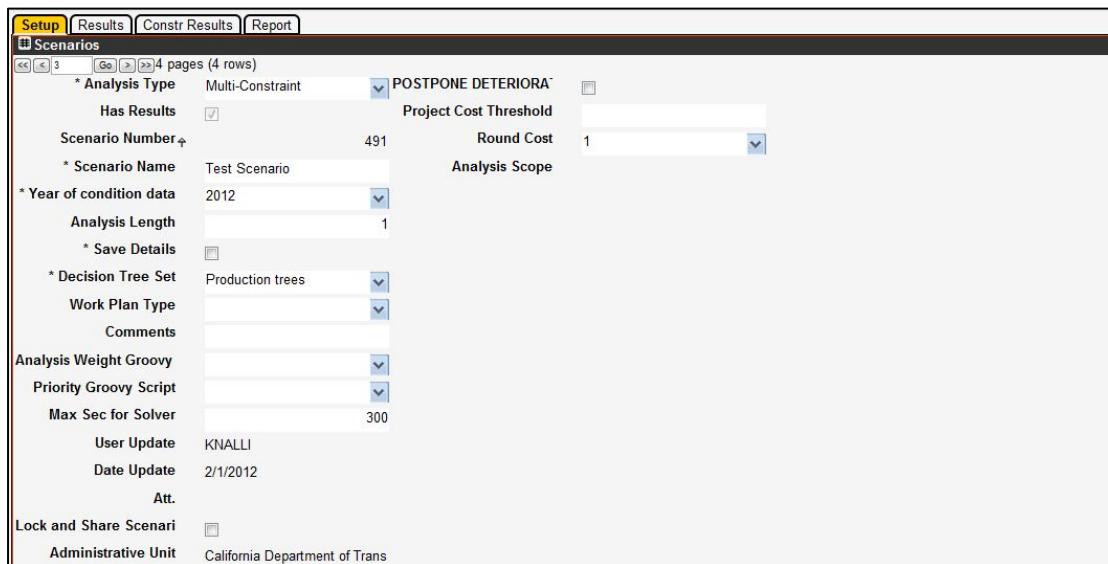
- **Setup** – This tab is where all parameters for an optimization are established.
- **Results** – This tab shows the optimal recommended work plan.
- **Constr. Results** – This tab shows the predicted values of each constraint compared to the input constraint value. This allows you to identify the constraints that have controlled the analysis results.
- **Report** – This tab shows the constraints selected in the Reporting Functions pane of the Setup tab and the value of each constraint after optimization.

These tabs are described in more detail on the following pages.

## Setup Tab

The Setup tab is where all parameters for a work plan optimization are established. When the Optimization Analysis window first opens, the Setup tab is always the tab that is displayed. This tab contains the Scenarios pane on the upper left, a Yearly Fin. Parameters pane on the lower left, a Constraints pane on the upper right, and a Reporting Functions pane on the lower right. These panes are described in the sections below.

### Scenarios Pane



The Scenarios pane displays the following configuration details:

Setting	Value
Analysis Type	Multi-Constraint
Has Results	<input checked="" type="checkbox"/>
Scenario Number	491
Scenario Name	Test Scenario
Year of condition data	2012
Analysis Length	1
Save Details	<input type="checkbox"/>
Decision Tree Set	Production trees
Work Plan Type	<input type="button" value="▼"/>
Comments	<input type="text"/>
Analysis Weight Groovy	<input type="button" value="▼"/>
Priority Groovy Script	<input type="button" value="▼"/>
Max Sec for Solver	300
User Update	KNALLI
Date Update	2/1/2012
Att.	<input type="text"/>
Lock and Share Scenario	<input type="checkbox"/>
Administrative Unit	California Department of Trans

The Scenarios pane is where you select project optimizations to review. It may also be used to create new optimization definitions or delete optimizations that you no longer need.

This pane contains the following fields:

- Analysis Type — This field determines what optimization routine will be performed.  
Note: If you select the Ranking analysis type, the system will rank projects until the constraint is reached — and then add one more project into the ranking list.
- Has Results — This check box indicates if the optimization has already been run. When a check mark is not displayed, then the results information shown in the other tabs is irrelevant.
- Scenario Number — This is a sequential number automatically assigned to identify the scenario. It cannot be edited. (To find a particular scenario, enter the scenario number in the field above the Scenarios pane and then click the **Find Scenario** button.)
- Scenario Name — This is a short description of the scenario. This description should be informative to other users.
- Year of Condition Data — The year you enter in this field is the year when condition data was collected, which sets the first year of optimization as this year plus one. (In most cases, the Network Master File contains only a single year's data and the value in this field sets the year of that data. For those cases where the Network Master File contains data from multiple years, the year in this field configures what records will be used for optimization.)
- Analysis Length — This is the number of years in the optimization period.
- Save Details — This check box determines whether the system saves the details of the optimization. When it is checked, the details from the analysis are saved and may be

viewed in the Detailed Optimization Results window (see page 102). When it is not checked, the details are not saved, although the overall results of the analysis will still be available in the Results tab.

- Decision Tree Set – This field determines what decision tree set will be used in the analysis. See page 84 for more information on decision trees.
- Work Plan Type – This field provides a drop-down list of all available work plans (which are defined in the Setup Work Plan Type window). When you select a work plan from the drop-down list, then the projects and costs from this work plan are first included in the optimization and then remaining rehabilitation recommendations are taken from decision tree results.

If you do not select a work plan, then the optimization is run without consideration of a pre-defined work plan; it gets its rehabilitation recommendations entirely from decision tree results. (To remove a work plan from the Work Plan Type field so no plan is selected, highlight the displayed work plan and then press the Delete key.)

- Comments – Optionally, you may include information in this field to describe the optimization.
- Analysis Weight Groovy Script – This field shows the name of the Groovy script that determines the weighting used to calculate Benefit. If a Groovy script is not utilized and this field is null, the value is taken from the Calculated Expression window where the drop-down list at the top of the window is set to Analysis Index Weight Factor.
- Priority Groovy Script – This field shows the name of the Groovy script that determines how analysis factors are prioritized against one another when the analysis type is Prioritization. If a Groovy script is not utilized and this field is null, the value is taken from the Calculated Expression window where the drop-down list at the top of the window is set to Analysis Priority.
- Max Sec for Solver – This field sets the maximum amount of time (in seconds) that the optimization analysis will be allowed to run before settling on a solution. (In some optimization analyses, many optimal solutions are available and the routine will "hunt" between each solution without settling on one and ending. Since these solutions are only marginally different than one another, forcing the routine to finish by setting the maximum amount of time that it can "hunt" is necessary and appropriate.)
- Lock and Share Scenario – If this check box is selected, then no modifications may be made to the settings for the scenario. The scenario is in effect a template for creating a new scenario. To create a new scenario from the template, right-click in the Scenarios pane and then click **Copy Scenario**. The system then creates a copy of the template and you may modify it as needed.
- Administrative Unit – In some implementations analyses are restricted by administrative unit. For these implementations, this field is included in the Setup tab and shows what unit "owns" the displayed analysis. For example, if the field shows Headquarters, then only users who select the Headquarters administrative unit when logging on may see and run the analysis.
- Postpone Deterioration – By default, the system performs a one-year deterioration of the network before beginning the analysis of treatment options. When this check box is selected, the system does not perform this initial one-year deterioration.
- Project Cost Threshold – When performing a priority analysis with a budget constraint, this field determines the amount of a project's cost that must be within budget to be included. The value may range from 0.0 to 1.0 (corresponding to 0% to 100% of the budget), with 0.5 (50%) being the default value. A setting of 0.0 will

always result in the budget constraint being reached or exceeded; a setting of 1.0 will result in the budget constraint never being exceeded. See page 98 for more information on how the setting of this field affects what projects are included in a ranking.

- **Round Cost** — This field causes the system to round the cost of projects to the selected rounding value before submitting the project to the solver engine. The greater the rounding, the faster the solver will converge on a solution.
- **Analysis Scope** — This read-only field shows the data elements that are included in the optimization. The scope is set with the **Edit Scope** command.

When you right-click this pane, a shortcut menu is displayed. This menu contains the common commands along with the following special commands:

- **Run Scenario** — This command executes the displayed optimization.
- **Edit Scope** — This command allows you to set the analysis scope for the optimization. Analysis scope allows you to limit the data elements included in the optimization.

When you execute this command, the application displays a new window. This window shows the variables that may be utilized to limit the scope of the analysis. You use this window in the same way as the Filter window (see page 20). (The variables are designated (and can be changed) in the Setup Analysis Columns window; see page 51 for more information.)

After setting the analysis scope, the selected elements are shown in the Analysis Scope field.

- **Copy Scenario** — This command allows you to copy the current scenario, which may then be pasted into another scenario to create a new one.
- **Download MPS File** — This command allows you to store on your local computer a copy of the MPS-formatted file that is the input to the solver.

### Yearly Financial Parameters Pane

Setup	Results	Constr Results	Report
<b>Yearly Financial Parameters</b>			
1 pages (1 rows)			
Year	Inflation Factor		
► 1			

The Yearly Financial Parameters pane allows you to enter values for discount rate and inflation for each year in the analysis period. These values must be entered as decimals (for example, 3% is entered as 0.03 — not 3).

The inflation rate is the rate of increase of a price index (the percentage rate of change in price level over time). The inflation rate affects the budgeted dollar amount by the following formula: Value in Next Year = Value in Current Year x (1 + Entered Inflation Value).

The discount rate is the value of money over time. A positive value decreases the value of money over time. The discount rate affects the budgeted dollar amount by the following formula: Actual Dollar Value = Budgeted Dollar Value - (Discount Rate x Budgeted Dollar Value).

## Constraints Pane

Setup   Results   Constr Results   Report						
Constraints						
<< < > >> 1 pages (2 rows)						
Is Objective	Constraint Column	Constr. Type	Constraint Limit Value	Condition Threshold	Scenario Yea	Add Constr.
<input checked="" type="checkbox"/>	Benefit	Weighted Avg				
	Treatment Cost	Total	500,000.00			

### NOTE

The constraints shown in the drop-down list in the Constraint Column column are those columns with a check mark in the Is Constraint check box in the Setup Analysis Columns window (see page 51 for more information on this window).

Each optimization requires one objective and, typically, multiple constraints. The Constraints pane allows you to configure the objective and constraints for the work plan optimization.

When you right-click this pane, the following special commands are available along with the common commands:

- **Propagate Years** – This command inserts a record for each of the years in the analysis period that is the same as the record you right-clicked (other than year).
- **Set to Included MWP Budget** – For each of the records selected in the Constraints pane, this command identifies the MWP projects that match the year and constraint partition of the record (and are within the analysis and MWP scope). It then calculates a total cost for these identified projects and inserts the cost in the Constraint Limit Value field of the record.

If the record in the Constraints pane does not include a year, then the command returns an error.

For example, say the constraint is on treatment costs for the year 2012 with an "Interstate" functional class partition. The system finds all of the interstate sections included in the analysis scope and then finds all MWP projects within the year 2012 that intersect those sections. It then places the total project costs for those projects in the constraint record.

- **Activate Constraint Subdivisions** – This command inserts records for each of the constraint subdivisions into which the selected constraint is subdivided as configured in the Setup Constraint Subdivisions window.

## Reporting Functions Pane

Setup   Results   Constr Results   Report				
Reporting Functions				
<< < > >> 1 pages (1 rows)				
Constraint Column	Constr. Type	Condition Threshold	Add Constr.	Constraint Subdivision
IRI/Traffic Weighted	Percent Above Threshold	0.4		

This pane configures what constraints will be shown in the Report tab. It has no bearing on the optimization routine itself. Essentially, this pane allows you to see the value of a constraint

after optimization without using the constraint to affect the outcome of the optimization routine.

When you right-click this pane, the following special commands are available along with the common command:

- Activate Constraint Subdivisions** — This command inserts records for each of the constraint subdivisions into which the selected constraint is subdivided as configured in the Setup Constraint Subdivisions window.

### Results Tab

Analysis > Network Analysis... > Optimization Analysis										
Setup	Results	Constr Results	Report	Work Plan Results						
<< < 1 > >> 70 pages (1105 rows)										
Plan Year	Budget Group	Treatment	Estimated Cost	Route	Direction	Lane	Milepost From	Segment From	Milepost To	Segment
6 Reconstruction	4R- New Construction or Reconstruction		\$14,105,000.00	I015	Both	All	17.196	001330	26.773	001330
7 Restoration	RC - 3R Crack/Seal/Plant Mix Overlay		\$4,596,178.00	I015	Desc.	All	30.870	001330	36.000	001330
2 Preservation - PM	RC - Surface Coat		\$12,419.00	I015	Asc.	All	36.000	001330	36.200	001330
3 Preservation - PM	Sealcoat (Chip Seal)		\$1,195,000.00	I015	Both	All	36.200	001330	46.700	001330
2 Reconstruction	4R- New Construction or Reconstruction		\$5,265,000.00	I015	Both	All	46.903	001330	47.460	001330
5 Preservation - PM	Sealcoat (Chip Seal)		\$1,625,000.00	I015	Both	All	46.903	001330	66.541	001330
2 Preservation - PM	Sealcoat (Chip Seal)		\$275,000.00	I015	Both	All	72.600	001330	76.000	001330
3 Preservation - PM	Sealcoat (Chip Seal)		\$680,000.00	I015	Both	All	85.638	001330	92.700	001330
2 Preservation - PM	Sealcoat (Chip Seal)		\$718,000.00	I015	Both	All	92.210	001330	99.490	001330
6 Preservation - PM	RC - Surface Coat		\$93,814.00	I015	Desc.	All	95.872	001330	97.115	001330
3 Preservation - PM	RC - Surface Coat		\$524,709.00	I015	Asc.	All	119.500	001330	127.548	001330
3 Preservation - PM	RC - Surface Coat		\$554,178.00	I015	Desc.	All	119.500	001330	128.000	001330
1 Preservation - 1R	Thin Plant Mix Overlay (less than 0.15')		\$2,642,000.00	I015	Asc.	All	127.548	001330	135.148	001330
4 Preservation - PM	RC - Surface Coat		\$30,943.00	I015	Asc.	All	127.548	001330	128.000	001330
4 Preservation - PM	RC - Surface Coat		\$489,333.00	I015	Asc.	All	128.000	001330	135.148	001330
1 Preservation - 1R	Thin Plant Mix Overlay (less than 0.15')		\$4,647,000.00	I015	Both	All	135.148	001330	143.367	001330

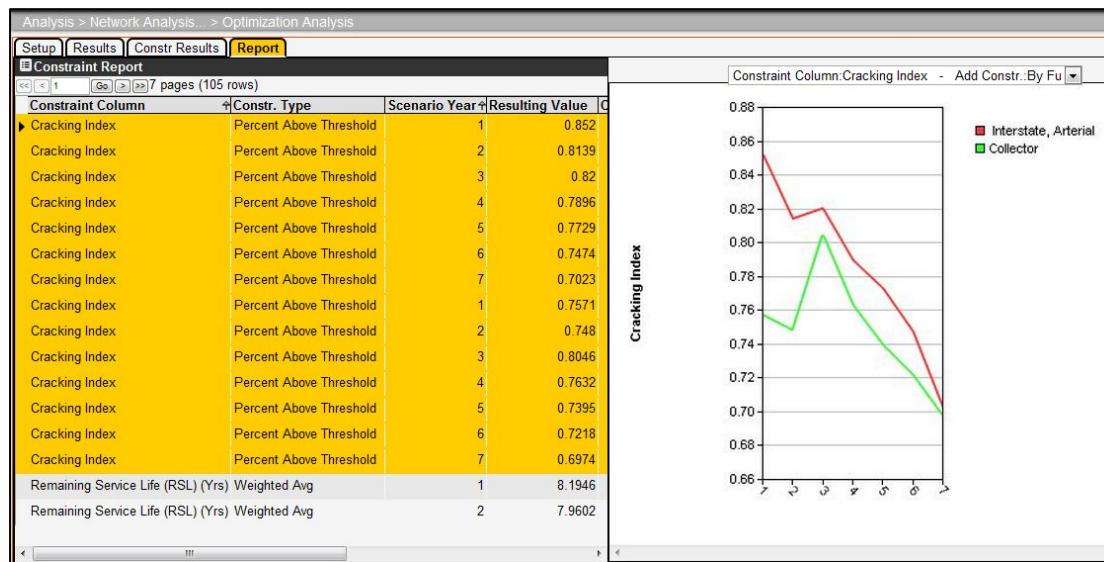
The Results tab provides a list of pavement sections produced by the optimization routine, based upon the input criteria configured in the Setup tab. These pavement sections constitute the recommended work plan.

### Constraint Results Tab

Analysis > Network Analysis... > Optimization Analysis						
Setup	Results	Constr Results	Report	Constraint Results		
<< < 1 > >> 1 pages (14 rows)						
Scenario Year	Constr. Type	Constraint Column	Constraint Limit Value	Condition Threshold	Resulting Value	Add Constr.
1 Total	Treatment Cost		199,818,000.00		199817940.8357	
2 Total	Treatment Cost		118,972,000.00		118971877.7148	
3 Total	Treatment Cost		147,983,000.00		147982802.4651	
4 Total	Treatment Cost		120,928,000.00		120927804.126	
5 Total	Treatment Cost		131,454,000.00		131453631.5992	
6 Total	Treatment Cost		121,728,000.00		121727983.674	
7 Total	Treatment Cost		126,813,833.00		126813788	
1 Weighted Avg	Benefit				662.0926	
2 Weighted Avg	Benefit				635.5488	
3 Weighted Avg	Benefit				570.8543	
4 Weighted Avg	Benefit				595.9851	
5 Weighted Avg	Benefit				566.2256	
6 Weighted Avg	Benefit				540.8676	
7 Weighted Avg	Benefit				502.3105	

The Constraint Results Tab shows the values of the constraints that affect the optimization routine at the end of the routine.

## Report Tab



After the optimization routine finishes, the left pane of this tab shows the constraints you selected in the Reporting Functions pane of the Setup tab and the values of the constraints.

The right pane shows a graph of the values of a particular constraint (which is selected from the drop-down list field shown at the top of the pane). The Y-axis of the graph shows constraint values and the X-axis shows the number of years in the analysis. The graph is a line graph for all constraint types except Total, which is a bar graph. You may modify the appearance of the graph by using the right-click **Change Graph Properties** command.

### 3.29.2. How 'Number of Nodes' Works

The Number of Nodes in Solution field sets the maximum number of decision nodes passed into the Integer Solver. This field affects the analysis as follows:

1. The system runs a linear programming solution (LPS) iteratively. It uses this type of solution because it is fast.
2. For road sections in the LPS result whose “portion to include [treat]” into the optimal result is close to 0 or 1, the system rejects or includes these sections entirely in the solution.
3. The system repeats steps 1 and 2 on the reduced set of sections until there are only N nodes (sections) left in the undetermined state (that is neither accepted [treated] or rejected [not treated]).
4. For these remaining nodes (sections), run the integer program (where the 0/1 solutions from LPS are hard-wired as inputs into the integer program).

#### NOTE

The objective of linear programming is to pick the portion of each road section to treat that will optimize the objective function, while the objective of integer programming is to pick/reject each section to treat in its entirety.

### 3.29.3. How 'Project Cost Threshold' Works

The Project Cost Threshold field determines the amount of the project cost (expressed as a decimal percentage) that must be within the budget constraint to be included in the priority

ranking. To better understand how this works, say an agency has the following five projects with the assigned priorities and costs:

Project Name	Project Cost	Project Priority
Project 1	\$2,000	190
Project 2	\$3,000	90
Project 3	\$1,000	150
Project 4	\$500	20
Project 5	\$1,800	70

The agency wants to rank these projects by priority with a budget constraint of \$4,400. The first part of the analysis is to simply rank the projects by priority. This results in the projects being sorted into the order shown in the following table.

Project Name	Project Cost	Project Priority
Project 1	\$2,000	190
Project 3	\$1,000	150
Project 2	\$3,000	90
Project 5	\$1,800	70
Project 4	\$500	20

If the budget was unlimited, the analysis could stop here and all the projects would be in the ranking. However, since the budget is constrained to \$4,400, the analysis engine must consider whether a project can be included in the ranking based upon the cost of the project. The criterion for whether a project can be included in the ranking is the setting of the Project Cost Threshold field, which determines the percentage of the project cost that must be within the budget constraint to be included in the ranking.

In essence, the analysis engine takes each project in priority order, adds the cost of the project to the running total, and compares the new running total to the budget constraint. If the new running total exceeds the budget constraint, the analysis engine multiplies the project's cost by the setting of the Project Cost Threshold field, adds the resulting value to the running total cost of all projects, and evaluates whether the new running total exceeds the budget constraint. If it does not exceed the budget constraint, analysis continues; if it equals or exceeds the budget constraint, analysis stops.

The following table shows three settings for the Project Cost Threshold field and notes which projects will be included in the project because of the value of this field.

Value of the Project Cost Threshold (PCT) Field	Projects Included in the Ranking
0.0 (This value results in the budget always being exceeded unless the list of projects is exhausted.)	Project 1 is included ( $\$2,000 \leq \$4,400$ ). Project 3 is included ( $\$1,000 \leq \$2,400$ [remaining budget after including Project 1]). Project 2 is included (while $\$3,000 > \$1,400$ , the $[PCT\ value * \$3,000] \leq \$1,400$ ). Ranking stops because running total ( $\$6,000$ ) exceeds budget constraint ( $\$4,400$ ).
0.5 (This is the default value.)	Project 1 is included ( $\$2,000 \leq \$4,400$ ). Project 3 is included ( $\$1,000 \leq \$2,400$ ). Project 2 is not included (the $[PCT\ value * \$3,000] > \$1,400$ ). Project 5 is included (the $[PCT\ value * \$1,800] \leq \$900$ ). Ranking stops because running total ( $\$4,800$ ) exceeds budget constraint ( $\$4,400$ ).
1.0 (This value results in the budget never being exceeded.)	Project 1 is included ( $\$2,000 \leq \$4,400$ ). Project 3 is included ( $\$1,000 \leq \$2,400$ ). Project 2 is not included (the $[PCT\ value * \$3,000] > \$1,400$ ). Project 5 is not included (the $[PCT\ value * \$1,800] > \$900$ ). Project 4 is included ( $\$500 \leq \$900$ ). Ranking stops because list of projects is exhausted; budget constraint not reached.

### **3.29.4. How to Define and Run an Optimization Analysis**

Follow these steps to perform a work plan optimization:

1. Open the Optimization Analysis window (Roadway > Analysis > Network Analysis... > Optimization Analysis).
2. Right-click in the left pane and then click **Insert**. The application creates a new application.
3. Highlight the text in the Scenario Name field and then type the name for this optimization.
4. Click in the Year of Condition Data field and then enter the year (in the format YYYY) of the most recent condition data that will be used in the optimization. Note that the year given in this field determines the starting year of the results of the optimization, which is the entered year plus one.
5. Click in the Analysis Length field and then type the number of years covered by this analysis.
6. If you wish to save the details of the optimization for further study, click the Save Details check box. (The details may be viewed in the Detailed Optimization Results window.)
7. For the Decision Tree Set field, click the down-arrow and then click the decision tree set you wish to use in the optimization.
8. If you wish to limit the data that is used in the analysis, right-click in the left pane and then click **Edit Scope**. This command displays a data selection window that you may use to select what data is used in the analysis.
9. If you wish to use a discount rate (that is, the value of money over time) and/or inflation in the analysis, complete the records in the Yearly Fin. Params pane.

10. You are now ready to enter the records for the objective and constraints of the optimization. The first record in the right pane should be for the objective of the optimization. To enter the record for the objective of the optimization, follow these steps:
  - a. In the right pane, right-click and then click **Insert**. A new record is added to the pane.
  - b. Click the **Is Objective** check box.
  - c. Click the down arrow in the **Constraint Column** column and then click the desired objective.
  - d. Depending on the objective selected, only a certain value may be permitted for the **Constr. Type** column and so the application automatically sets this column. If this does not occur, click the down arrow in the **Constr. Type** column and then click the appropriate constraint type.
11. While only one objective is allowed for an optimization, one or more constraints may be configured. A constraint record may apply to all years in the optimization period or multiple constraint records may be created for each year in the optimization period. Finally, for those constraints that have subdivisions as configured in the **Setup Constraint Subdivisions** window, you may create records for each constraint subdivision. The following steps provide a general process for entering subdivided constraint records:
  - a. In the right pane, right-click and then click **Insert**. A new record is added to the pane.
  - b. Click the down arrow in the **Constraint Column** column and then click the desired constraint.
  - c. Click the down arrow in the **Constr. Type** column and then click the desired constraint type.
  - d. In the **Constraint Limit** column, enter the value for the constraint. Note: Percentages are entered as a decimal value between 0 and 1 (for example, 5% is entered as 0.05 not 5).
  - e. If the constraint type is **Percentage Above Threshold**, enter the threshold value in the **Condition Threshold** column.
  - f. If the constraint will apply to all years in the optimization period, leave the **Scenario Year** field blank. Otherwise, right-click the constraint record and then click **Propagate Years** in the shortcut menu. The application enters a constraint record for each year in the optimization period (which was set in step 5) as a copy of the record you right-clicked. It also enters the year in the **Scenario Year** field. You will now need to edit each newly inserted record to reflect the appropriate constraint limit.
  - g. If the constraint will apply to all subdivisions of the constraint (if any), then you are finished with this constraint and may proceed to enter additional constraints as needed.
12. On the other hand, if the constraint limit will vary for the different constraint subdivisions, right-click the constraint record and then click **Activate Constraint Subdivisions** in the shortcut menu. The application will then insert a record for each child node shown in the **Setup Constraint Subdivisions** window with the name of the node shown in the **Node Name** column. You will now need to edit each newly inserted

record to reflect the appropriate constraint limit. After editing each record, you may proceed to enter additional constraints as needed.

13. Once all constraints are configured, you are ready to run the optimization. Right-click the left pane and then click **Run Scenario**. The application performs the optimization and then displays the results in the Results tab. The Constr. Results tab will also show the actual constraint values at the end of optimization.

#### NOTE

You may improve performance of an optimization analysis by experimenting with the values entered in the Number of Nodes in Solution and/or Max Sec for Solver fields. In general, increasing the number of nodes improves the quality of the optimization, but at the cost of increasing the amount of time that the analysis runs. Similarly, increasing the amount of time that the Solver runs will improve the quality of the optimization, but at the cost of lengthier run times.

### **3.29.5. How to Include Projects from a Work Plan**

The goal for work plan optimization is the development of an optimal work program. The system includes a window for maintaining that work program. Maintenance of the work program definition is covered more fully in the discussion of the Work Plan Data window (see page 104).

By default, optimization does not include or consider projects from a work plan because the objective of the optimization is to select the most appropriate projects given the input criteria. However, when running an optimization, there will be times where the input criteria should or must include projects that are already in the planning stages. The **Edit MWP Scope** command (which is found by right-clicking in the Setup tab) is used to include projects from a work plan.

When you select the **Edit MWP Scope** command, the Project Selection window opens. This window shows work plan attributes of status, plan year, and treatment. Work plan sections that meet the criteria that you define in this window will be placed into the optimization's recommended work plan. This data selection controls which projects are automatically included in the optimal work program by year, treatment, and/or status. In multi-year analyses, after pavement section(s) are assigned according to their status, they later become eligible for normal scenario work plan development.

## **3.30. Detailed Optimization Results**

(Roadway > Analysis > Network Analysis... > Detailed Optimization Results)

Analysis > Network Analysis... > Detailed Optimization Results											
Select Scenario LN_506 Test Decomposition											
Road Sections											
Record #	Route	Direction	Lane	Milepost From	Segment From	Milepost To	Segment To	Length	Number of Lanes	PMS Section #	IN MWP
19 I090	Asc.	All		0.000	001660	5.500	001660	5.5	2	17703161	<input type="checkbox"/>
20 I090	Asc.	All		5.500	001660	10.920	001660	5.42	2	17703162	<input type="checkbox"/>
21 I090	Asc.	All		10.920	001660	11.151	001660	0.231	2	17703163	<input type="checkbox"/>
22 I090	Asc.	All		11.151	001660	14.781	001660	3.63	2	17703164	<input type="checkbox"/>
Named Details											
Scenario Year	Benefit	Exclusion Yr	Pvmnt Age	Remaini	Budget Group	Prior	Treatment	Condition Attributes	Treatment Cost	Pavement Type	Performance Model Type
1	772.98		4	9.3727				Cracking Index		Composite	ITD - Flexible/Composite - 1R Prese

For all work plan optimizations that have the Save Details check box in the Setup tab of the Optimization Analysis window checked, the Detailed Optimization Results window provides

an in-depth look at the data resulting from the optimization. You select the desired optimization from the drop-down list in the Select Scenario field at the top of the window. After choosing the optimization, the system displays the information from the optimization in the window.

The upper pane (Road Sections) shows the Network Master road sections (with finer partitions if the Master Work Plan is used in the optimization). For the road section selected in the upper pane, the bottom pane (Named Details) shows each year's results for the road section.

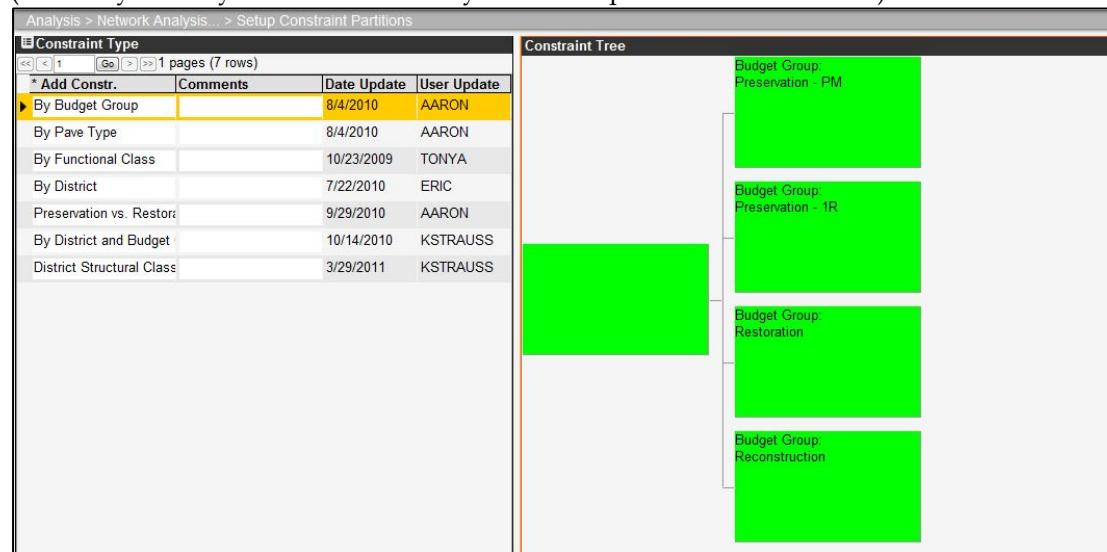
### 3.31. Setup Constraint Partitions

Constraints are used in the Optimization Analysis window. If desired, these constraints may be subdivided to allow a more finely tuned optimization.

For example, you could select CCI as an index for constraining treatment costs. This index would apply across all types of roads in the transportation network. However, if you would prefer to have different CCI values for different types of roads, you could use this window to subdivide the CCI constraint by, say, Interstates, Primary, and Secondary. These could then be used to create additional constraint records in the Optimization Analysis window.

#### 3.31.1. Description of the Window

(Roadway > Analysis > Network Analysis... > Setup Constraint Partitions)



The screenshot shows the 'Setup Constraint Partitions' window. On the left is a grid table titled 'Constraint Type' with columns: \* Add Constr., Comments, Date Update, and User Update. The table contains seven rows with various constraint types and their details. On the right is a 'Constraint Tree' diagram showing a hierarchical structure of budget groups:

- Budget Group: Preservation - PM
  - Budget Group: Preservation - 1R
- Budget Group: Restoration
- Budget Group: Reconstruction

The Setup Constraint Partitions window contains two panes. The left pane shows the constraints that have partitions. For the constraint selected in the left pane, the right pane shows the hierarchical arrangement of the constraint's partitions.

#### NOTE

When you add additional records to the Constraints tab of the Optimization Analysis window, records are only created for child nodes; parent nodes are ignored.

When you right-click a node in the hierarchy, the system displays a shortcut menu with the following special command:

- **Edit Decision Var Limits** – This command is only available for parent nodes. It allows you to configure the thresholds for the selection of each of the child nodes.

- **Edit Node Name** – This command allows you to modify the text shown in the node box.

### **3.31.2. How to Create Constraint Partitions**

To partition a constraint, follow these steps:

1. Display the Setup Constraint Partitions window.
2. In the left pane, right-click and then click **Insert**. A new record is added to the pane.
3. In the new record, highlight the default text and then type the column name of the constraint that will be subdivided.
4. In the right pane, right-click the parent node and then click **Add Branch**. The application displays a dialog box so you may enter the number of nodes needed.
5. In the dialog box, enter the number of nodes needed.
6. Repeat steps 4 and 5 for any additional partitions that are needed.
7. Right-click the parent node and then click **Edit Decision Var Limits**. The application displays a dialog box so you may select the criteria by which each child node is activated.
8. In the left pane of the dialog box, select the variable that will be used to determine branching. (The available variables are those columns in the Setup Analysis Columns window that have a check mark in the **Is Add Constr. Column** check box.)
9. In the right pane of the dialog box, enter the criteria that will cause the different child nodes to activate.
10. Click **OK** to close the dialog box. The application labels each of the child nodes with the variable and value that causes the node to activate.
11. If desired, you may further partition one or more of the child nodes by repeating steps 4 through 10.

12. Once the partition hierarchy is as desired, click to save the new partitions.

### **3.32. Work Plan Data**

(Roadway > Analysis > Network Analysis... > Master Work Plan)

Master Work Plan										
* MWP Project Status	* Plan Year	* Budget Group	* Treatment	* Estimated Cost	* Route	* Direction	* Lane	* Milepost From	* Segment	
Awarded	2011	Restoration	Bonded Overlay (Concrete)	\$0.00	I090	Both	All	0.000	001660	
Awarded	2011	Preservation - 1R	Thin Plant Mix Overlay (less than 0.1 in.)	\$5,062,000.00	I090	Both	All	44.500	001660	
Awarded	2011	Restoration	3R- Mill and Inlay, Overlay	\$1,552,000.00	SH057	Both	All	18.978	001620	
Awarded	2011	Restoration	3R- Mill and Inlay, Overlay	\$1,736,000.00	SH057	Both	All	27.150	001620	
Scenario Recommended	2011	Preservation - PM	RC - Surface Coat	\$14,429.00	SH200	Both	All	55.416	001610	
Scenario Recommended	2011	Preservation - PM	RC - Surface Coat	\$8,042.00	US095	Asc.	All	436.000	001540	
Scenario Recommended	2011	Preservation - PM	RC - Surface Coat	\$111,767.00	US095	Both	All	481.840	001540	
Scenario Recommended	2012	Preservation - PM	RC - Surface Coat	\$310,774.00	I090	Asc.	All	63.000	001660	
Scenario Recommended	2012	Preservation - PM	RC - Surface Coat	\$24,216.00	SH004	Both	All	5.720	001780	
Scenario Recommended	2012	Preservation - PM	RC - Surface Coat	\$57,374.00	SH005	Both	All	3.476	001820	
Scenario Recommended	2012	Preservation - PM	RC - Surface Coat	\$8,072.00	SH005	Both	All	4.420	001820	
Scenario Recommended	2012	Preservation - PM	RC - Surface Coat	\$45,514.00	SH058	Both	All	2.210	001810	
Scenario Recommended	2012	Preservation - PM	RC - Surface Coat	\$34,151.00	SH200	Both	All	29.740	001610	
Development	2012	Preservation - PM	Sealcoat (Chip Seal)	\$206,830.32	US002	Both	All	7.400	001590	
Development	2012	Preservation - PM	Sealcoat (Chip Seal)	\$26,846.85	US002	Both	All	16.030	001590	

**NOTE**

The data displayed in this window is not affected by filtering. To facilitate locating a particular record, you may sort the records (either by double-clicking a column head or using the right-click **Sort** command) or use the right-click **Find** command.

All transportation agencies keep a list of planned projects, which is updated regularly for when the project will be performed, where it will be performed, the intended treatment, and estimated project cost. This list is termed a work plan, and it constitutes the final list of sections proposed for repair based upon network analysis results and, as necessary, user intervention.

The system allows you to develop multiple work plans, with the different work plans being defined in the Work Plan Type window and the data for each work plan being entered in this window. The data is then used in scenario analysis and reports. (See page 90 for more information on the Work Plan Type window.)

When you right-click the table, a shortcut menu is displayed. This menu contains the common commands along with the following special commands:

- **Recalculate Cost** – This command utilizes the treatment cost formula configured in the Calculated Expressions window. This formula uses the input parameters for treatment costs found in the Master Work Plan window. The result of this formula is then displayed in the Project Cost column.
- **Copy Analysis Results** – This command displays a dialog box from which you select the scenario analysis and then the line items you want to copy to the work plan data window. See page 106 for more information on using this command.
- **Create Pavement Construction Record** – This command creates a construction history section record from the currently selected master plan record provided a work code is designated for the treatment assigned to the master plan record. The construction history section record is then available for edit and review in the Construction History window.

Note: Once a line item in a work plan is used to create a construction history record, this command is no longer available for that line item.

This following information is copied from the master plan record to the construction history section record:

- The work code from the treatment assigned to that master plan record.
- The layer information from the standard section assigned to the master plan record. If no standard section is assigned to the master plan record, then the standard section assigned to the treatment associated with the master plan record will be used. If a standard section is not associated with either the master plan record or the treatment associated with the master plan record, a construction history section record will still be created, but it will have no layers.
- The cost, location, and construction year from the master plan record.

### **3.32.1. How to Edit a Work Plan**

To edit a work plan:

1. Open the Work Plan Data window.
2. Click the arrow in the Select WP Type field above the table and then click the desired work plan from the displayed list.

3. To add a row to the plan, right-click in the Master Work Plan pane and then click **Insert**. The application adds a new row to the table, and you may enter the necessary information in each column.

To modify information in a row, highlight the existing information and then type the new information.

To remove a row from the plan, point to the row to be deleted, right-click, and then click **Delete** as appropriate to edit the work program.

4. When you have made all desired changes, click  to save the new information.

### **3.32.2. Example of Editing a Work Plan**

Assume that a rural highway will have an aggregate source open nearby in the year 2010. The output from network analysis may recommend a thin overlay on pavement sections along this route for 2008. However, the local engineer knows that the structural capacity for this road will not be adequate for the expected, undocumented traffic (in this case, "undocumented" means not included in system data).

The engineer therefore decides to alter the timing and treatment for the affected pavement section(s) along this route by changing the year to 2009, the treatment to a heavier type, and altering project price(s) accordingly. The engineer next assigns a project status code to the project that will be excluded from network analysis (for example, "excepted"). All these changes are made in a particular work plan that was selected from the drop-down list at the top of the window.

The engineer can continue to address work plan changes for each section as appropriate. Once changes are made to the work plan, any network analysis used to create scenario work plans must be re-run to account for those projects established and "fixed" by the engineer. This last network analysis run is essential for distributing budgets effectively while taking into consideration project specific requirements.

### **3.32.3. How to Copy Line Items from a Scenario Analysis**

The system supports multiple scenario analyses, and you may copy line items out of one of these scenario analyses into a work plan by following these steps:

1. Open the Work Plan Data window.
2. In the Select WP Type field at the top of the window, click the down arrow and then click the work plan into which you will copy line items. The system displays the line items for the selected work plan after you make your selection.
3. Right-click the table and then click **Copy Analysis Results**. The system displays a dialog box for selecting the scenario analysis out of which you will copy line items.
4. In the Select Scenario field at the top of the dialog box, click the down arrow and then click the scenario analysis out of which you will copy line items. After selecting the scenario analysis, the system displays the line items from the analysis scenario.
5. Select the line items you wish to copy. Use SHIFT+click to select multiple, adjacent line items or CTRL+click to select multiple, non-adjacent line items.
6. Once the records are selected, right-click and then click **Copy Selected Items**. The system copies the selected line items to the Work Plan Data window. Note that the dialog box remains open so you may select additional line items if necessary.
7. When all desired line items are copied, click the close window icon in the upper right corner to close the dialog box.

8. In the Work Plan Data window, click  to save the new line items in the selected work plan.

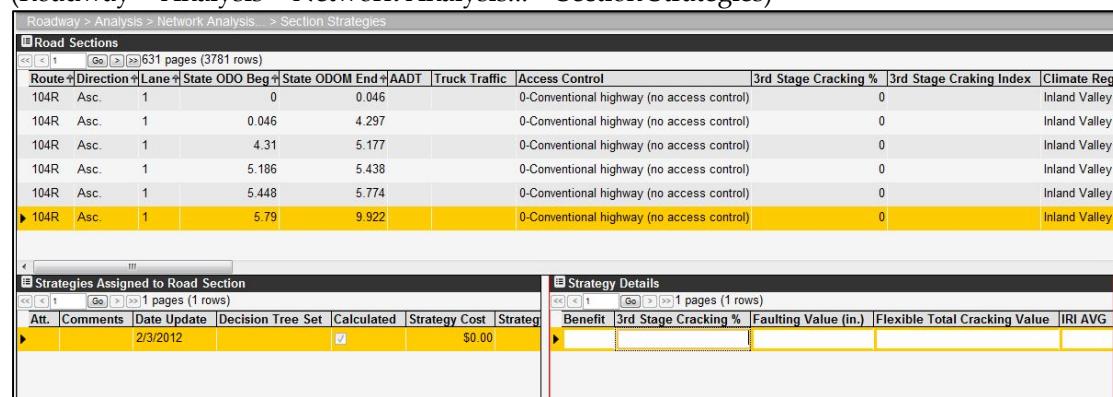
### 3.33. Strategy Configuration

The Strategy Configuration window allows you to create and maintain the various strategies that are assigned to road sections in the Section Strategies window (see the following section). A strategy is a set of pavement treatments specified for one or more years on a pavement section.

The window contains two panes. The left pane (Strategies) lists the various strategies that have been configured. For the strategy selected in the Strategies pane, the right pane (Strategy Details) shows the years covered by the strategy. For each year in the pane on the right, a check mark in the Budgeted column indicates whether money may be spent in the indicated year. (The right pane may be populated a single year at a time by using the **Insert** command or all years [using the value in the Strategy Duration field in the left pane] at once by using the **Insert All** command.)

### 3.34. Section Strategies

(Roadway > Analysis > Network Analysis... > Section Strategies)



Roadway > Analysis > Network Analysis... > Section Strategies								
Road Sections								
631 pages (3781 rows)								
Route	Direction	Lane	State ODO Beg	State ODOM End	AADT	Truck Traffic	Access Control	3rd Stage Craking %
104R	Asc.	1	0	0.046		0-Conventional highway (no access control)	0	Inland Valley
104R	Asc.	1	0.046	4.297		0-Conventional highway (no access control)	0	Inland Valley
104R	Asc.	1	4.31	5.177		0-Conventional highway (no access control)	0	Inland Valley
104R	Asc.	1	5.186	5.438		0-Conventional highway (no access control)	0	Inland Valley
104R	Asc.	1	5.448	5.774		0-Conventional highway (no access control)	0	Inland Valley
▶ 104R	Asc.	1	5.79	9.922		0-Conventional highway (no access control)	0	Inland Valley
Strategies Assigned to Road Section								
1 pages (1 rows)								
Att.	Comments	Date Update	Decision Tree Set	Calculated	Strategy Cost	Strateg	Benefit	3rd Stage Cracking %
		2/3/2012			\$0.00			
Strategy Details								
1 pages (1 rows)								
Benefit	3rd Stage Cracking %	Faulting Value (in.)	Flexible Total Cracking Value	IRI AVG				

The Section Strategies window allows you to develop, edit, and assign rehabilitation strategies to particular road sections. (A strategy is a set of pavement treatments specified for one or more years on a pavement section and are configured in the Strategy Configuration window.)

This window contains the following panes: Road Sections, Strategies Assigned to Road Section, and Strategy Details. These panes are described in more detail in the following sections.

#### 3.34.1. Road Sections Pane

The Road Sections pane lists all road sections in the Network Master file. This set may be filtered as needed. When you select a section in the top pane, the lower panes are filled with information related to the selected section.

#### 3.34.2. Strategies Assigned to Road Section Pane

For the road section selected in the Road Sections pane, the Strategies Assigned to Road Section pane lists the strategies currently assigned to the road section. You may insert and delete these strategies as desired.

When you right-click this pane, the following special commands are available along with the common commands:

- **Build Strategy Data** – This command populates the Strategy Details pane with the computed treatments for the section based on the strategy funding parameters. **This will overwrite any manual changes that you made in the Strategy Details pane.**
- **Calculate Strategy Data** – This command uses the treatments (if any) shown in the Strategy Details pane to calculate costs, conditions, and other data.
- **Build Strategies of This Type for All Sections** – This command functions the same as the **Build Strategy Data** command, but affects all road sections in the upper pane, not just the selected road section. Note: Those strategies assigned to a road section that have the Protected check box selected will not be affected by this command.

### **3.34.3. Strategy Details Pane**

For the strategy selected in the Strategies pane, this pane provides the details of the strategy including what treatment in what year is recommended and the costs associated with that treatment.

## **3.35. Road View**

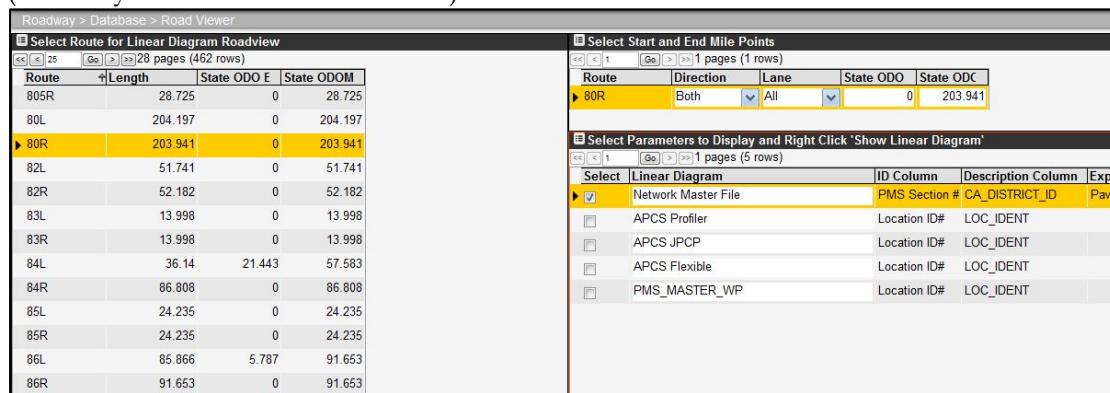
### **NOTE**

The Road View window always uses the base linear referencing system regardless of what is selected in the Location Reference field on the left side of the browser window.

You use the Road View window (Roadway > Database > Road Viewer) to graph several types of data at once. In the window, you select which columns are to be graphed. You then use the **Show Linear Diagram** command (found by right-clicking in the Linear Diagram pane) to display a new window (the Linear Diagram window) that shows graphs of the selected columns.

### **3.35.1. Description of the Panes in the Road View Window**

(Roadway > Database > Road Viewer)



The Road View window contains the following panes:

- The Route Selection pane – This pane contains a list of all routes in the road network. You highlight the route, which will be graphed in the Linear Diagram window.
- The Boundary Selection pane – This pane shows the route selected in the Route Selection pane and its end points. You may graph a portion of the route by adjusting the route's boundary points.

- The Linear Diagram pane – This pane identifies the data that will be graphed in the Linear Diagram window. It shows one row for each table with data that can be graphed.

In this pane, you:

- Select whether the road segments for a table will be graphed. (Only if a check mark appears in the check box in the Select column will road segments from this table be graphed.)
- Select by what column's values the road segments for a table will be colored on the graph.
- Select whether the road segments for a table will be displayed on the graph in one line or expanded to a line for each column value.
- Further constrain the portion of the route to be graphed.
- Invoke the Linear Diagram window to view the graph.

### **3.35.2. Description of the Right-click Shortcut Menu Commands**

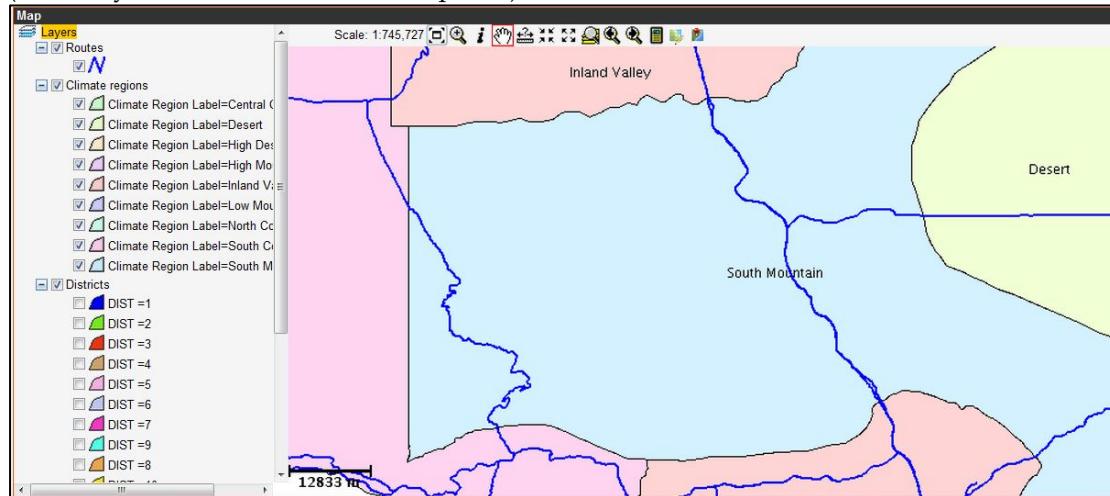
When you right-click a record in this pane, a shortcut menu is displayed. This menu contains the following special commands:

- **Edit Scope** – This command displays a new window, from which you set further constraints on the portion of the route to be graphed. These constraints are based upon the data pertinent to the route. This window has the same capabilities as the Filter window available from reports.
- **Expand By** – This command displays a new window, from which you select the column whose values will expand the line graph, with each value being on a different line. Select the No Column entry in the drop-down list when you don't want the column expanded.
- **Color By** – This command displays a new window, from which you select the column whose values will color the line graph, with each value having a different color. Select the No Column entry in the drop-down list when you don't want to color by any column's values.
- **Show Linear Diagram** – This command displays a new window, which shows the road segments for the selected tables in a linear diagram graph. Across the top of the graph is a scale showing each mile to graph for the route. Down the left side is a list of the tables whose road segments are to be graphed. In the body of the graph are a series of parallel lines, one for each table (or, if the Expand By capability is used, then one for each selected column's value). Each line is colored by the column value as selected using the **Color By** command.

Additionally, when you right-click a road segment in the body of the graph in the Linear Diagram window, a shortcut menu is displayed that contains the **Show Information** command. This command displays a second, new window that shows the data pertinent to the road segment upon which you right-clicked.

### 3.36. GIS Data Explorer

(Roadway > Database > GIS Data Explorer)



#### NOTE

The first time that you display the Data from Map window it will be blank (that is, no themes will be shown in the left pane and no map in the right pane). You must add themes to the left pane via the **Add Layers** command that is found on the shortcut menu that is displayed by right-clicking the left pane. After adding a theme, the system will display the theme in the right pane. The system will remember the theme(s) you select and will display the theme(s) the next time you display the Data from Map window.

The Data from Map window is essentially the Floating Map window; see page 33 for more information.

## 4. System Module

The System module allows you to create and maintain the codes needed for the asset management system as well as the administrative functions of the software. A System Administrator should use the features of this module.

### 4.1. Access Level Settings

An "access profile" is the access permissions ("access levels") granted for menus in all modules. It determines what a user may see and access. While the access profile is for all modules as a whole, the permissions may vary from one module to another. For example, one type of access profile could provide complete read/write access to all windows found in a particular module while disallowing (actually, not showing) access to the other modules.

The Access Level Settings window allows you to establish different access profiles by setting access permissions for each menu. The permissions range from 0 to 7 (most restrictive to least restrictive, respectively). An access profile is then assigned to a user in the User Names and Access window. See page 113 for information on creating an access profile.

In all, three windows maintain access level settings, each of which contributes to what data may be viewed and maintained by an access profile. The windows are:

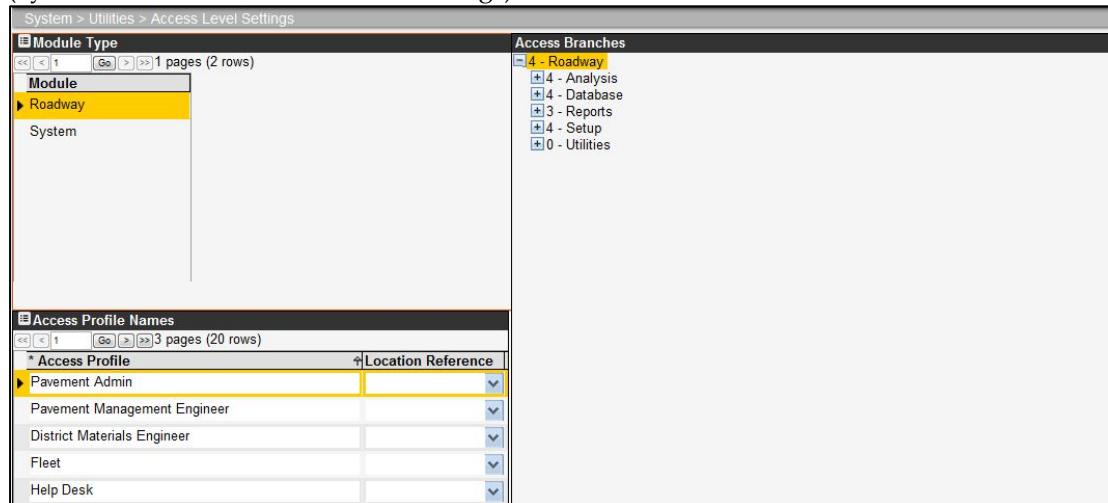
- Access Level Settings – This is the window currently being discussed, where level settings for window access are set. This window access level is also used to determine user access to columns and right-click commands.
- Actions Rights – This is where level settings for accessing right-click commands are set (in other words, can the user view and use commands found on shortcut menus that are displayed by right-clicking a window or pane).
- Columns – This is where level settings for a user's ability to view data in specific columns and/or edit the data are set.

Within a window, the user's access to right-click commands and columns depends on how his or her access level settings relate to the window access level setting in the user's access profile:

- When the window access level setting is greater than or equal to the access level setting for the right-click command, then the user can activate the command in the window. Otherwise, the command is disabled for that user.
- When the window access level setting is greater than or equal to the access level setting in the Right to See column, then the user can see the column in the window. Otherwise, the user cannot see the column.
- When the window access level setting is greater than the access level setting in the Right to Edit column, then (assuming that the user has edit capability in the window in general) the user can edit the column. Otherwise, the user cannot edit the column.

#### **4.1.1. Description of the Access Level Settings Window**

(System > Utilities > Access Level Settings)



The Access Level Settings window contains three panes: the Module pane (upper left) for selecting a module; the menu structure as displayed in the Module Menu tree view pane on the right; and the Access Profile pane (lower left) that shows the different access profiles configured.

For the module selected in the Module pane and the access profile selected in the Access profile pane, the Module Menu hierarchy shows the access permission assigned to each menu item. (The access permission appears as a number to the left of the name of the menu item.)

##### **Module Pane**

This pane lists all modules found in the system. This is read-only, and only the common commands are provided on the shortcut menu found by right-clicking.

##### **Access Profile Pane**

This pane shows the names of all access profiles and the number of users that are assigned each profile. The names shown in this pane are what appear in the drop-down list found in the User Names and Access window. To create an access profile, see page 113.

The Location Reference column determines whether a security profile allows access to one or all location reference systems (LRS). If the column is left blank, then the profile allows access to all LRS and the Location Reference drop-down field will be shown in the left gutter. Otherwise, the security profile will only allow access to the selected LRS and the Location Reference drop-down field will be not shown in the left gutter (because the user only has access to that LRS and cannot change it).

##### **Module Menu Pane**

This is the right pane in the Access Level Settings window. For the currently selected module and access profile, it displays all of the menu items of the module in a tree view along with the access permission assigned to each item. (The access permission is shown to the left of the name of the menu item.)

When you right-click a menu item in the tree view, a shortcut menu is displayed with the following commands:

- **Set This to the Branch** – This command assigns the access level setting for the menu item to which you pointed to all sub-items.

- **Set Access Level** – This command displays a new dialog box in which you may select the access level for the item to which you pointed. The following access levels are available:
  - (0) Invisible – The user cannot see the menu item.
  - (1) Disabled – The user can see the menu name, but cannot open the window.
  - (2) Read Only – The user can see the window content, but cannot modify it.
  - (3) Read/Write – The user can read and write data.
  - (4) Accept/Confirm – The user can read and write data and has approval capability.
  - (5) - (7) Super Users – The user has all the rights to edit data and grant approvals plus the additional rights defined in the Actions Rights and Columns windows.

#### **4.1.2. How to Create an Access Profile**

To create an access profile, follow these steps:

1. Display the Access Level Settings window.
2. In the Access Profile pane, right-click and then click **Insert**. The system adds a new record to the pane.
3. In the row showing the new record, highlight the default name (New Role) and then type the name of the access profile.
4. In the Module pane, click the first module to select it.
5. In the Module Menu pane, right-click the first menu item and then click **Set Access Level**. The system displays a dialog box that lists all access permissions and what each defines.
6. In the dialog box, click the down arrow to display the list of access permissions and then click the desired access permission.
7. Click **OK**. The system closes the dialog box.
8. If the menu item has sub-items and these are to be assigned the same access permission as the parent menu item, right-click the parent menu item and then click **Set This to Branch**. Otherwise, skip this step.
9. Repeat steps 5 through 8 for the remaining menu items.
10. In the Module pane, select the next module and repeat steps 5 through 8 for the menu items in this module.
11. Repeat step 10 for the remaining modules.
12. When access permissions are assigned to all menu items in all modules, click .

## **4.2. Actions Rights**

### **NOTE**

Only personnel authorized as a System Administrator should be allowed read/write access to this window.

This window is used to re-name and set access levels for user action properties (that is, the commands in the shortcut menus found by right-clicking a pane or window [these are termed "right-click commands"]). It is also used to require that a user enter his or her password to execute certain commands.

Note that three windows maintain access level settings, each of which contributes to what data may be viewed and maintained by an access profile. These are:

- Access Level Settings – This window is where access profiles are created and maintained.
- Actions Rights – This is the window described in this topic, which is where you specify a user's ability to view and execute right-click commands.
- Columns – This is where the user's ability to view and/or edit particular columns in a window is specified.

Within a window, the user's access to right-click commands and columns depends on how his or her access level settings relate to the window access level setting in the user's access profile:

- When the window access level setting is greater than or equal to the access level setting for the right-click command, then the user can activate the command in the window. Otherwise, the command is disabled for that user.
- When the window access level setting is greater than or equal to the access level setting in the Right to See column, then the user can see the column in the window. Otherwise, the user cannot see the column.
- When the window access level setting is greater than the access level setting in the Right to Edit column, then (assuming that the user has edit capability in the window in general) the user can edit the column. Otherwise, the user cannot edit the column.

#### 4.2.1. Description of the Actions Rights Window

(System > Utilities > Actions Rights)

Action Rights						
MENU ITEM ID	MENU ITEM LABEL	Right To See	Password Required?	Confirm Required?	Comments	
>	-	5 - System Administrator	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
sec_level_0	0 - Invisible	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
sec_level_1	1 - Disabled	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
sec_level_2	2 - Read Only	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
sec_level_3	3 - Editable	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
sec_level_4	4 - Approval	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
sec_level_5	5 - Super5	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
sec_level_6	6 - Super6	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
sec_level_7	7 - Super7	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
sec_level_8	8 - Super8	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
sec_level_9	9 - Super9	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
accept	Accept as Current	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
model_accept	Accept model for Analysis	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		
buy_new_equipment	Acquire This Equipment	3 - Read / Write	<input checked="" type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/>		

The Actions Rights window contains the following columns:

- Menu Item Label – This column describes the right-click command and is the text seen by the user. You can edit the label for any right-click command, but be aware that this may cause the appearance of the system and documentation to differ.
- Right to See – This column shows the minimum access level setting in an access profile (for the window) that will allow the user to execute the right-click command throughout the system (also see Access Level Settings on page 111 for a description of rights).

- Password Required – When this check box is selected, the user must enter his or her password to execute the command.

**NOTE**

Each row in this window represents a different right-click command. Several right-click commands cannot be restricted and are always available to the user. These are the common commands that are in most shortcut menus (such as **Find**).

When completing this window, avoid changing any action label to something that does not describe the action.

#### **4.2.2. How to Set Right-to-See Values**

The user's ability to see a right-click command depends upon both:

- The user's access level setting for the window containing the command; and
- The Right to See access level setting in this window.

The table below shows which ranges of these two settings yield what level of access.

<b>The Access Level Setting for the Window</b>	<b>Right-click Command's "Right to See" Setting</b>	<b>The Effect on Access</b>
0 – 1	0 – 7	Can't even access the window. Consequently, cannot use the command.
2	0 – 2	Can use the command, but cannot edit data in the window. This may cause difficulties when the command requires subsequent editing in the window.
2	3 – 7	Cannot use the command.
3	0 – 3	Can use the command.
3	4 – 7	Cannot use the command.
6	0 – 6	Can use the command.
6	7	Cannot use the command.

#### **4.3. Columns**

**NOTE**

Only personnel authorized as a System Administrator should be allowed read/write access to this window.

This window is used to set descriptions, rights, and formatting for all fields in the asset management system.

In all, three windows maintain access level settings, each of which contributes to what data may be viewed and maintained by an access profile. The windows are:

- Access Level Settings – This window is where access profiles are created and maintained.

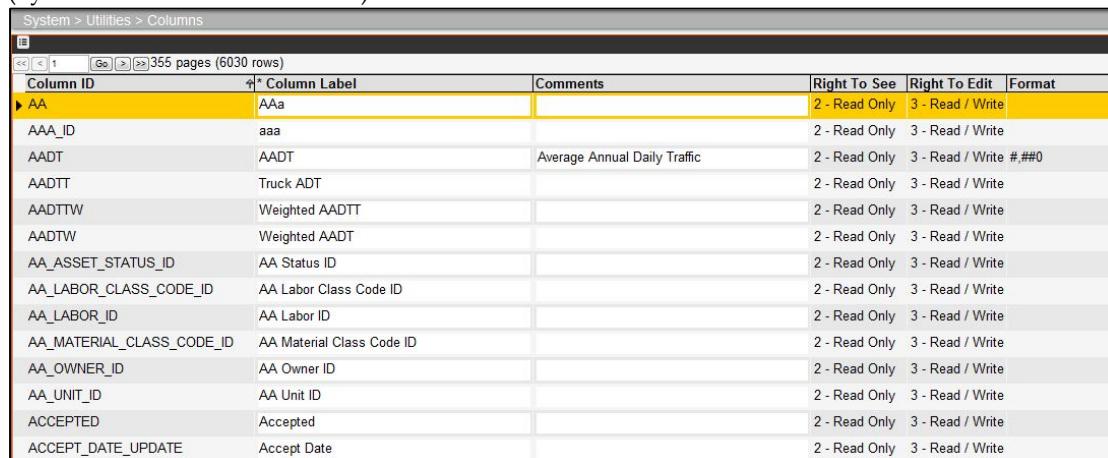
- Actions Rights – This is where level settings for accessing and executing commands found on shortcut menus are set (in other words, can the user view and use shortcut commands found by right-clicking a window or pane).
- Columns – This window, which is the subject of this topic, defines the access levels required to both view and edit specific fields within the system. If a user does not enter a window with a high-enough access level to view a column, the column will be invisible to that user. (This is appropriate for fields such as salary or social security number.) If a user enters a column with sufficiently high-enough access level to see a field, but not enough to edit the field, it will be visible but disabled so the user cannot change the value.

Within a window, the user's access to right-click commands and columns depends on how his or her access level settings relate to the window access level setting in the user's access profile:

- When the window access level setting is greater than or equal to the access level setting for the right-click command, then the user can activate the command in the window. Otherwise, the command is disabled for that user.
- When the window access level setting is greater than or equal to the access level setting in the Right to See column, then the user can see the column in the window. Otherwise, the user cannot see the column.
- When the window access level setting is greater than the access level setting in the Right to Edit column, then (assuming that the user has edit capability in the window in general) the user can edit the column. Otherwise, the user cannot edit the column.

#### 4.3.1. Description of the Columns Window

(System > Utilities > Columns)



Column ID	Column Label	Comments	Right To See	Right To Edit	Format
AA	AAa		2 - Read Only	3 - Read / Write	
AAA_ID	aaa		2 - Read Only	3 - Read / Write	
AADT	AADT	Average Annual Daily Traffic	2 - Read Only	3 - Read / Write #,##0	
AADTT	Truck ADT		2 - Read Only	3 - Read / Write	
AADTTW	Weighted AADTT		2 - Read Only	3 - Read / Write	
AADTW	Weighted AADT		2 - Read Only	3 - Read / Write	
AA_ASSET_STATUS_ID	AA Status ID		2 - Read Only	3 - Read / Write	
AA_LABOR_CLASS_CODE_ID	AA Labor Class Code ID		2 - Read Only	3 - Read / Write	
AA_LABOR_ID	AA Labor ID		2 - Read Only	3 - Read / Write	
AA_MATERIAL_CLASS_CODE_ID	AA Material Class Code ID		2 - Read Only	3 - Read / Write	
AA_OWNER_ID	AA Owner ID		2 - Read Only	3 - Read / Write	
AA_UNIT_ID	AA Unit ID		2 - Read Only	3 - Read / Write	
ACCEPTED	Accepted		2 - Read Only	3 - Read / Write	
ACCEPT_DATE_UPDATE	Accept Date		2 - Read Only	3 - Read / Write	

#### NOTE

The ITEM\_COL\_NAME and DESCR\_COL\_NAME columns must contain real names. (These columns are in the table from which the Columns window takes its data but not the window itself. As such, these columns may only be edited directly in the database.)

The Columns window contains the following columns:

- Column Label – This is the name of the column as seen by a user. (The Column ID column shows the name of the column within the system.)

- Right to See – This is the minimum access level setting in a user's profile (for the window) to allow the user to see the column within that window (also see Access Level Settings on page 111).
- Right to Edit – This is the minimum access level setting in a user's profile (for the window) to allow the user to edit the column within that window (also see Access Level Settings on page 111).
- Use for Weighted Average – A check mark indicates that the column may be used to create weighted-average aggregations in tabular reports.
- Format – This column configures how the data in the column is formatted. Dates and times are formatted automatically as set in the System Parameters window. Monetary values are formatted by putting [Currency] in the column, with the currency format being that which is configured in the System Parameters window. Finally, numbers may be formatted using Java formatting techniques; see this website for more information: [http://www.unicode.org/reports/tr35/#Number\\_Format\\_Patterns](http://www.unicode.org/reports/tr35/#Number_Format_Patterns).

Note: You cannot edit the Format column directly. Instead, right-click the record for the column and then click **Edit**. The application displays a dialog box. Click **Next** to view the second page of the dialog box and, provided the View Type is R-Number, the field for setting the format is at the bottom of the dialog box.

- View Type – This column only affects the display of data when used and filtered in standard reports. This allows a System Administrator to create new columns for reporting and choose how the columns will be configured for the user in filtering and displaying the data in reports. The permitted variables for View Type are as follows:
  - B – Color.
  - C – Check box (a variable that has two values displayed as a check box).
  - D – Date.
  - G – Geometric data.
  - H – Big List (a filterable list that displays in a popup window).
  - R – Number (real).
  - S – String (character).
  - T – If the Table\_Name and Parent\_Col\_Name columns are completed, this indicates a tree-type list. If the columns are not completed, this indicates a simple list.

When the View Type variable is T (tree), the following additional variables may be used:

- DB Tables and Views – This is the name of the lookup table that contains the list of legal values (codes) and a description of each.
- ID Column – This is the name of the column in the lookup table that is the code. This is the actual data value stored in the database. (This field does not display until a table is selected in the DB Tables and Views field.)
- Label Name – This is the name of the column in the lookup table that is the literal description of the code. This is what the user will see on the screen when reporting or filtering data based on this column. (This field does not display until a table is selected in the DB Tables and Views field.)
- Where Clause – This is the Oracle "where" statement to apply to the Oracle lookup table in order to get only the set of values that apply to this column ID.

(This field does not display until a table is selected in the DB Tables and Views field.)

- PK Tables — If you pick a view in the DB Tables and Views field, this field supplies the underlying table needed for the foreign key that is required for future automatic processing.
- Rule for Finest Partition Data Split — When the Master Work Plan is included in network analysis, the system performs a finest-partition operation between it and the Network Master file. The type of finest partition operation is selected from the drop-down list in this column.

Note that none of these columns can be edited directly in the window itself. Instead, you use the **Insert** and **Edit** commands, which have special functionality for column attribute manipulation:

- **Insert** — This command inserts a new column into the system. Once a column is defined in this window, other windows (that build or adjust data table structures) can assign it to one or more data tables.

When you select this command, the system displays a dialog box. You enter the column ID, descriptive label, comments, and other column attributes. See the **Edit** command description below for details on column attributes.

Note: The text you enter for comments does not just reside in the AgileAssets database; it is also entered into the Oracle system table that stores comments.

Note: The following rules must be observed when naming new columns:

- The column name must begin with a letter.
- The column name cannot be longer than 30 characters.
- The column name must be made up of alphanumeric characters or the following special characters: \$, \_, and #. If the column name uses any other characters, it must be enclosed in double quotation marks.
- The column name cannot be an Oracle "reserved word." (See the following site for reserved words:  
[http://download-uk.oracle.com/docs/cd/A87862\\_01/NT817CLI/appdev.817/a42525/apb.htm](http://download-uk.oracle.com/docs/cd/A87862_01/NT817CLI/appdev.817/a42525/apb.htm).)

- **Edit** — This command provides edit capability for the attributes of an existing column. When you select this command, a new window appears that is identical to the window displayed with the **Insert** command — except that the column ID is already present and cannot be changed. Three column attributes are editable here: column label, right to see, and right to edit. To edit the remaining column attributes, click the **Next** button. The information available and editable in the subsequent pane varies depending upon the column type selected as shown at the top of the pane:

- When the T-List column type is selected, then you select the table from which the data for the T-List column will originate. Then you select the column in that table that uniquely identifies the data. Next, you select the column in that table that will be displayed when the T-List is used as a drop-down list. Finally, you set the "where clause" if you want only a portion of the data selected to be shown for the T-List.
- When the String column type is selected, then the size of the column (how many characters) must be provided.
- When the Color, Date, or Checkbox column type is selected, then the rest of the pane is blank.

- When the R-Number column type is selected, then the following information must be provided:
  - The Numeric Type. Select the type by clicking the appropriate radio button: Integer, Decimal, or Real. If Decimal is selected, then additionally provide the size (number of digits on both sides of the decimal point).and scale (number of digits to the right of the decimal point).
  - The minimum value and maximum value used for data validation.
  - Whether the values in the column may be used for weighted-average aggregations.
  - The Split Rule if your agency has implemented either the AgileAssets Pavement Management System or Data Manager product. When one of these products is implemented, you need to specify the Split Rule because that rule is needed in the Finest Partition operation; it determines how the numeric value will be adjusted when road section limits are more finely partitioned. See the following section for the available Split Rules.

#### NOTE

When editing, in the first popup window that is displayed, you may change any of the fields. However, for the second popup window that is displayed by clicking **Next**, do not change any fields without first consulting AgileAssets.

In addition to the **Edit** and **Insert** commands, a **Delete** command is also available. However, this should not be used without first consulting with AgileAssets. Unpredictable operation may occur if you delete a column that is in use.

For a description of "rights" (that is, access permissions), see the Access Level Settings window on page 111.

#### **4.3.2. Split Rules**

The following are the available Split Rules:

- Split along and across – Any value that varies as road section area varies normally would be split along and across. An example would be cost to maintain a road section.
- Split along – Any value that varies as road section length varies, but is not affected by road section width changing, normally would be split along. An example would be roadway length.
- Split across – Any value that varies as road section width varies, but is not affected by road section length changing, normally would be split across. An example would be pavement width pertinent to this record.
- Don't split – Any value that varies as road section width varies, but is not affected by road section length changing, normally would be split across. An example would be average elevation of this road section.

#### **4.3.3. Precautions in Setting Access Levels**

When setting access levels, you should **avoid**:

- Setting the "Right To See" value higher than the "Right to Edit" value.
- Changing any column label to something that does not describe the column.

Note: If you are trying to change a column label, but cannot find it in this Columns window, call an AgileAssets technical representative. You should also call

AgileAssets if you are unsuccessful in changing a column label to its proper description (throughout the system).

- Changing values in any of the following columns without first consulting an AgileAssets technical representative:
  - Column ID.
  - View Type.
  - Table Name.
  - Item Column Name.
  - Column Name Description.
  - Where Clause.
  - Show Details.

Note: These columns usually have their right to edit set to "5 – Super" or higher. This provides needed security to these sensitive variables.

#### **4.3.4. Setting Right-to-See and Right-to-Edit Values**

##### **NOTE**

Although not required, it is normal practice to set a column's "right to see" less than its "right to edit."

The settings for a user's "right to see" and/or "right to edit" columns depends upon both his or her profile's access level setting for the window containing the column and the "right to see" / "right to edit" access level setting in this window (Columns). The table below shows which ranges of these three settings yield what level of access:

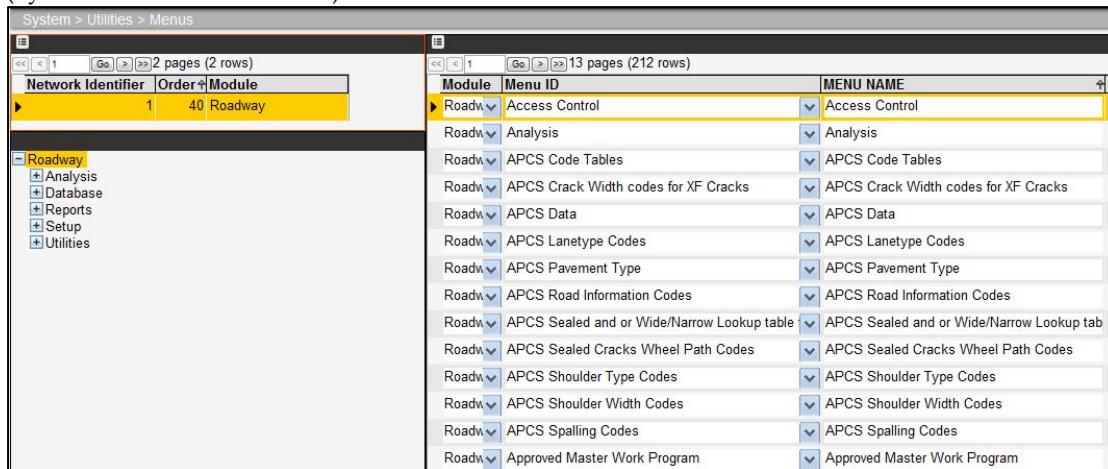
<b>The Profile's Access Level Setting for the Window</b>	<b>Column's "Right to See" Setting</b>	<b>Column's "Right to Edit" Setting</b>	<b>The Effect on Access</b>
0 – 1	0 – 7	0 – 7	Can't even access the window. Consequently, cannot see or edit the column.
2	0 – 2	0 – 7	Can see the column, but cannot edit the column (since you cannot edit anything in the window).
2	3 – 7	0 – 7	Cannot see (nor edit) the column.
3	0 – 3	0 – 3	Can see and edit the column.
3	0 – 3	4 – 7	Can see the column, but cannot edit the column.
3	4 – 7	0 – 7	Cannot see (nor edit) the column.
6	0 – 6	0 – 6	Can see and edit the column.
6	0 – 6	7	Can see the column, but cannot edit the column.
6	7	0 – 7	Cannot see (nor edit) the column.

## 4.4. Menus

### NOTE

Although you can change the labels for the main menu commands, be aware that this could cause the appearance of the system to differ from the documentation provided. For this reason, changing the labels for menu commands is not recommended.

### (System > Utilities > Menus)



The screenshot shows the 'System > Utilities > Menus' window. The left pane, titled 'Modules', displays a table with columns 'Network Identifier', 'Order', and 'Module'. A single row is selected for 'Roadway'. The right pane, titled 'Module Menus', displays a table with columns 'Module', 'Menu ID', and 'MENU NAME'. The 'Roadway' module is selected, and its menu items are listed: Access Control, Analysis, APSCS Code Tables, APSCS Crack Width codes for XF Cracks, APSCS Data, APSCS Lanetype Codes, APSCS Pavement Type, APSCS Road Information Codes, APSCS Sealed and or Wide/Narrow Lookup tab, APSCS Sealed Cracks Wheel Path Codes, APSCS Shoulder Type Codes, APSCS Shoulder Width Codes, APSCS Spalling Codes, and Approved Master Work Program. The 'Analysis' menu item is highlighted in yellow.

The Menus window is used to maintain labels for each of the menu commands in the main menu structure of the asset management application. It displays a table and tree view of the menu commands for each of the modules, and is composed of two panes: Modules and Module Menus. These panes are described in the following sections.

#### 4.4.1. Modules Pane

The upper part of the left Modules pane shows the available modules, with the selected module highlighted.

The lower part of the Modules pane shows the hierarchical arrangement of the menus and menu items found in the module selected in the upper part of the pane. When you click a node in this hierarchy the corresponding entry in the table in the right pane is also selected.

#### 4.4.2. Module Menus Pane

The right Module Menus pane shows a table view of all menus and menu items in the module selected in the upper left pane. The record in this table that is selected corresponds to the node in the lower left pane that is also selected (provided the selection is made from the lower left pane; if you select a record in the table first, the corresponding node in the hierarchy is not also selected). You may change the labels and menu ordering for the menu items; see the following section for information on editing labels.

#### 4.4.3. How to Change a Menu Name

While it is generally not recommended to change the name of a menu, in rare instances it may be necessary. This is accomplished as follows:

1. Display the Menus window.
2. In the Modules pane, select the module for which a label is to be changed by clicking the record showing the name of the module. The module name is highlighted and the menu hierarchy for the module is shown in the lower part of the left pane.

3. In the hierarchy in the lower left pane, locate the label to be changed and click it. (If necessary, click a + sign to expand the hierarchy.) The system highlights the node you clicked in the hierarchy and also the corresponding row in the table in the Module Menus pane on the right.
4. In the Menu Label column of the Module Menus pane, modify the name as desired.
5. Click . The new label is now in effect and will be seen the next time the module and menu is accessed. (Note: If you changed the name of a menu in the System module, you must first select another module and then re-select the System module to see the new name.)

#### 4.4.4. How to Delete a Window

To delete a window, you delete the menu item associated with the window. This is accomplished as follows:

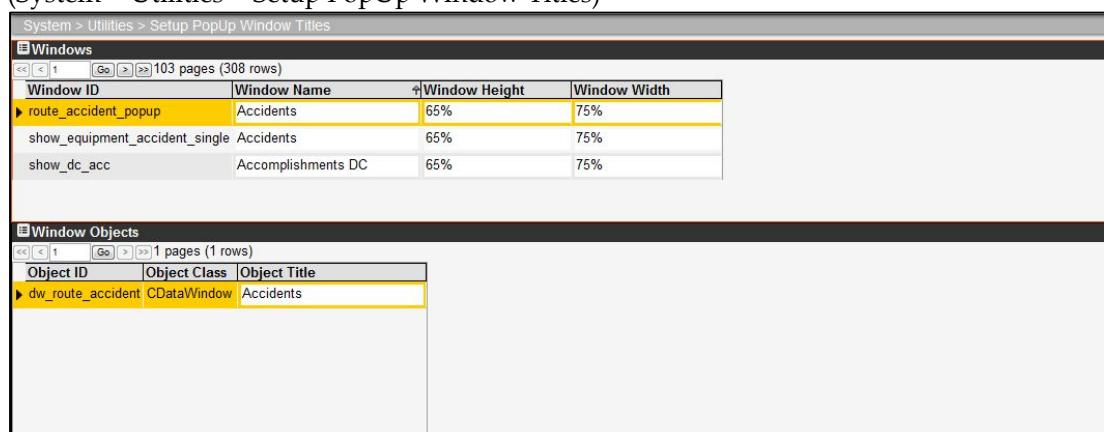
1. Display the Menus window.
2. In the upper left pane, select the module in which the menu item for the window appears. Once the module is selected, the menu hierarchy in the lower left pane changes to reflect the hierarchy of the selected module.
3. In the bottom left pane, locate the menu item for the window.
4. Right-click the menu item and then click **Delete**.
5. Click the  icon.

### 4.5. Setup Popup Window Titles

#### NOTE

Although you can change the labels for popup windows, be aware that this could cause the appearance of the system to differ from the documentation provided. For this reason, changing the labels for popup windows is not recommended.

(System > Utilities > Setup PopUp Window Titles)



The screenshot shows the 'Setup PopUp Window Titles' page with two main sections:

**Windows** table:

Window ID	Window Name	Window Height	Window Width
route_accident_popup	Accidents	65%	75%
show_equipment_accident_single	Accidents	65%	75%
show_dc_acc	Accomplishments DC	65%	75%

**Window Objects** table:

Object ID	Object Class	Object Title
dw_route_accident	CDataWindow	Accidents

The Setup Popup Window Titles window is used to maintain labels for popup windows that appear throughout the application (and the objects that appear in them). It contains the following panes:

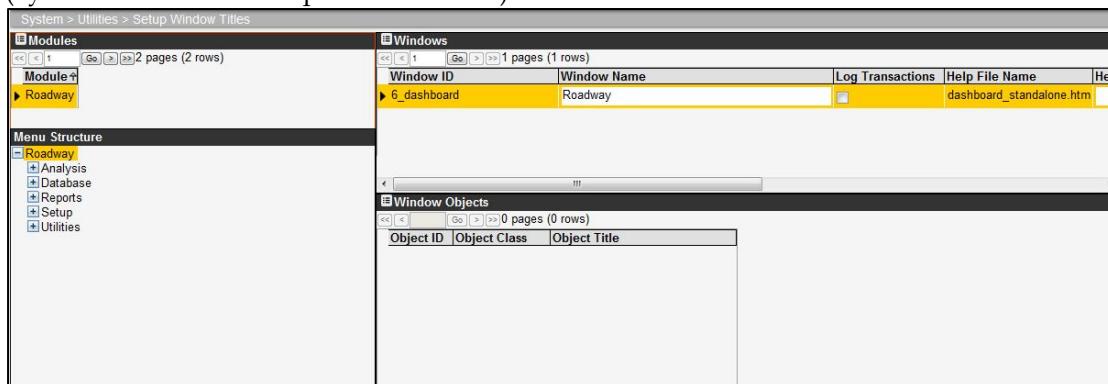
- Windows – The upper pane shows the available popup windows. You may edit the title of a popup window by highlighting the existing text in the Window Name column and typing the new title.
- Window Objects – The lower pane shows all components of the popup window selected in the upper pane. You may edit the label applied to a component by highlighting the existing text in the Object Title column and typing the new label.

## 4.6. Setup Window Titles

### NOTE

Although you can change the labels for windows and panes, be aware that this could cause the appearance of the system to differ from the documentation provided. For this reason, changing the labels for windows and panes is not recommended.

(System > Utilities > Setup Window Titles)



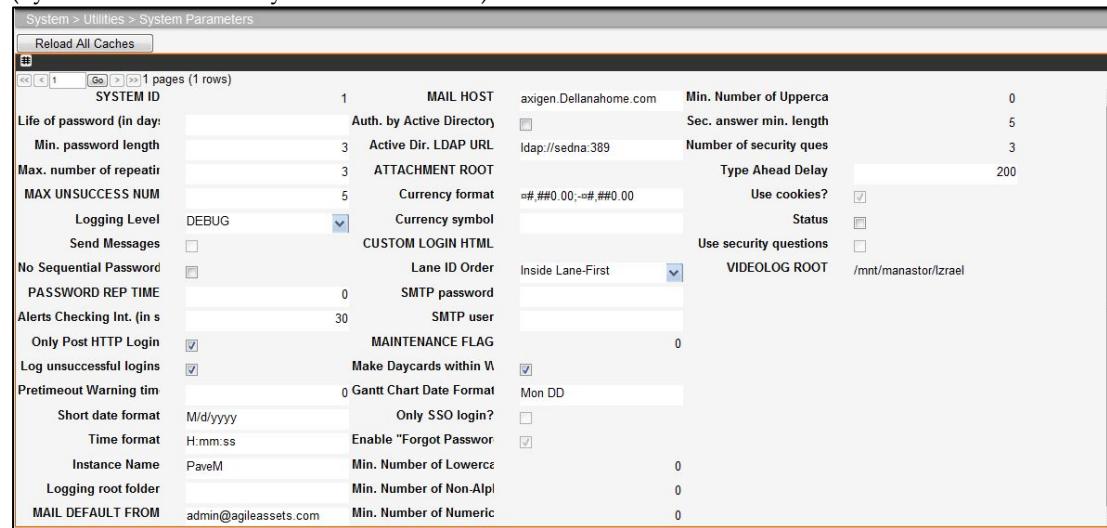
The Setup Window Titles window is used to maintain labels for the windows and panes throughout the application. It contains the following panes:

- Modules (and module's menu hierarchy) – The left pane shows the available modules, with the selected module highlighted. Below the table that shows the various modules of the application is the hierarchy of menu items for the selected module.
- Windows – The upper right pane shows the windows associated with the menu item selected in the hierarchy in the lower left pane. You may edit the title of a window by highlighting the existing text in the Window Name column and typing the new title.
- Window Objects – The lower right pane shows all components of the window selected in the upper Windows pane. You may edit the label applied to a component by highlighting the existing text in the Object Title column and typing the new label.

This window may also be used to "re-map" the association between a window and the Help topic that describes that window so different information displays when the Help icon is clicked. This different information may be in a file in the same folder as the other HTML Help files or, more typically, a web page. You re-map the association by displaying the desired window in the hierarchy on the left and then, in the Windows pane, changing the file name in the Help File Name column to the file or web page ('<http://<URL>>').

## 4.7. System Parameters

(System > Utilities > System Parameters)



The screenshot shows the 'System > Utilities > System Parameters' window. It displays a grid of system parameters with their current values. Some fields are dropdown menus or checkboxes. The parameters include:

Parameter	Value	Parameter	Value
SYSTEM ID	1	MAIL HOST	axigen.Dellanahome.com
Life of password (in day)		Auth. by Active Directory	<input type="checkbox"/>
Min. password length	3	Active Dir. LDAP URL	ldap://sedna:389
Max. number of repeatit	3	ATTACHMENT ROOT	
MAX UNSUCCESS NUM	5	Currency format	=#,##0.00,-#,##0.00
Logging Level	DEBUG	Currency symbol	
Send Messages	<input checked="" type="checkbox"/>	CUSTOM LOGIN HTML	
No Sequential Password	<input type="checkbox"/>	Lane ID Order	Inside Lane-First
PASSWORD REP TIME	0	SMTP password	
Alerts Checking Int. (in s)	30	SMTP user	
Only Post HTTP Login	<input checked="" type="checkbox"/>	MAINTENANCE FLAG	0
Log unsuccessful logins	<input checked="" type="checkbox"/>	Make Daycards within W	<input checked="" type="checkbox"/>
Prettimeout Warning tim	0	Gantt Chart Date Format	Mon DD
Short date format	M/d/yyyy	Only SSO login?	<input type="checkbox"/>
Time format	H:mm:ss	Enable "Forgot Passwor	<input checked="" type="checkbox"/>
Instance Name	PaveM	Min. Number of Lowerc	0
Logging root folder		Min. Number of Non-Alpi	0
MAIL DEFAULT FROM	admin@agileassets.com	Min. Number of Numeric	0

The System Parameters window controls many miscellaneous system settings. A knowledgeable System Administrator should maintain these settings as many of them affect the security of the system and critical data items.

The **Reload All Caches** button causes all AgileAssets cache-retained data to be refreshed from the current data in the database. AgileAssets software uses cache for quick access to a copy of much AgileAssets-specific GUI configuration information. At times, portions of this cache data get out-of-date relative to the original, but recently modified, data contained in the database — and this button may be used to bring the cache up-to-date.

### NOTE

The **Reload All Caches** button is made available to authorized users so that cache data updating can occur at any time. Alternatively (or additionally) the functionality of this button can be automatically applied nightly via a System Job.

The following table lists all system parameters and provides a description of each. Note that this is a general list and not all parameters are found in every AgileAssets application.

Parameter Title	Parameter Description
Active Dir. LDAP URL	This field contains the server address where the Active Directory is stored.
Alerts Checking Int. (in sec)	The value in this field determines how often (in seconds) the system checks for new alerts. Note: If this parameter is set to zero, then not only will the system not generate alerts, it will also not generate any types of messages.
Attachment Root	This is the root folder to which a user has access to obtain a source file to import or attach an external file to a record.

Parameter Title	Parameter Description															
Auth. by Active Directory?	<p>When this check box is selected, the password stored in the Active Directory is used for authentication rather than the password stored in the AgileAssets application.</p> <p>Note: A third field is also added to the log-on window for entry of the domain. The system administrator will need to provide this information to a user to allow her or him to log on.</p>															
Currency Format	This field sets how monetary values are formatted in column displays. To apply the format to a column, in the Columns window put [Currency] in the Format column for the column that is a monetary value.															
Custom Login HTML	This field customizes the appearance of the log-in screen.															
Date Format	<p>This field sets how dates are displayed. These functions use the same 'format' strings as the java.text.SimpleDateFormat class as shown in the following table:</p> <table border="1"><thead><tr><th>Value</th><th>Format (Full)</th><th>Format (Short)</th></tr></thead><tbody><tr><td>Year</td><td>yyyy (4 digits)</td><td>yy (2 digits) y (2 or 4 digits)</td></tr><tr><td>Month</td><td>MMM (name or abbr.) NNN (abbr.)</td><td>MM (2 digits) M (1 or 2 digits)</td></tr><tr><td>Day of Month</td><td>dd (2 digits)</td><td>d (1 or 2 digits)</td></tr><tr><td>Day of Week</td><td>EE (name)</td><td>E (abbr.)</td></tr></tbody></table> <p>For example, a typical coding for an American-style date (say, 10/22/2008) would be MM/dd/yyyy and for a sortable date (say, 2008-10-22) it would be yyyy-MM-dd.</p>	Value	Format (Full)	Format (Short)	Year	yyyy (4 digits)	yy (2 digits) y (2 or 4 digits)	Month	MMM (name or abbr.) NNN (abbr.)	MM (2 digits) M (1 or 2 digits)	Day of Month	dd (2 digits)	d (1 or 2 digits)	Day of Week	EE (name)	E (abbr.)
Value	Format (Full)	Format (Short)														
Year	yyyy (4 digits)	yy (2 digits) y (2 or 4 digits)														
Month	MMM (name or abbr.) NNN (abbr.)	MM (2 digits) M (1 or 2 digits)														
Day of Month	dd (2 digits)	d (1 or 2 digits)														
Day of Week	EE (name)	E (abbr.)														
"Forget Password?" Link Enabled	If this field is checked, then the Forgotten Password feature is enabled and the "Forget Password?" prompt appears in the log-on screen. See page 11 for more information on this feature.															
Gantt Chart Date Format	This specifies the format for the dates shown at the top of the Schedule and Road View windows. It follows the same conventions as Date Format given above.															
Instance Name	This is the literal description of the environment that is displayed at log-in and in the top right corner of the main screen.															
Lane ID Order	This parameter determines how divided multi-lane roadways are numbered. The selection "Inside Lane First" indicates that the number 1 lane is nearest the median; the selection "Outside Lane First" indicates that the number 1 lane is furthest from the median.															
Life of Password (in days)	The number of days that a password can be active. Once expired, on the next user logon, the user will be required to change his or her password.															
Log Unsuccessful Logins	When this check box is selected, the system tracks and stores information on unsuccessful attempts to log in. This information may be viewed by displaying the standard report for unsuccessful log-in attempts.															

Parameter Title	Parameter Description
Logging Level	<p>You may configure logs to be kept each day to monitor system activity. These logs are stored in the location specified by the Logging Root Folder parameter and are accessed via the System Logs window (see page 213). The following types of logging may be configured by selecting the appropriate type from the drop-down list:</p> <ul style="list-style-type: none"><li>Error — This type logs only that an error occurred.</li><li>Debug — This type logs that an error occurred as well as where (in the Java code) that the error occurred.</li><li>Performance — This type is the same as Debug but with additional timing statistics. It shows User ID, Window ID, action, and time to execute.</li><li>None — Logs are not generated.</li></ul> <p><b>Note: Setting this parameter to a value other than None also writes log-in and log-out activity by user to the SYSTEM_LOG table.</b></p>
Logging Root Folder	All logging files are stored in a folder called Logs. This field specifies where that folder is stored relative to the application root directory. (If the field is blank, the logs are stored in the application root directory under the Logs folder.)
Mail Default From	This field configures the email address that will be the "From" e-mail address for all mail sent by the system.
Mail Host	This field configures the server name through which email is routed.
Mail Password	This is the password used to access the e-mail server, if needed.
Mail Username	This is the username used to access the e-mail server, if needed.
Make Day Cards within WO	When this check box is selected, users may only create or edit Day Cards that are within the start and end dates (inclusive) of the work order.
Max. Idle Session	When the user does not interact with the system for the length of time (in minutes) given in this field, the system will automatically log off the user.
Max. Number of Repeating Characters	The maximum number of characters that may repeat consecutively in a password. For example, if this parameter was set to three, a password like AAAA01 would not be allowed (although AAA001 and AA01AA would be allowed).
Max. Unsuccess Num.	The value in this field specifies the number of attempts a user may make to log on before he or she is locked out of the system. When the value in this field is exceeded, the user will need to contact the System Administrator to unlock his or her account.
Min. Number of Lowercase Characters	The value in this field indicates the minimum number of lowercase characters that must be in a password.
Min. Number of Non-alphanumeric Characters	The value in this field indicates the minimum number of non-alphanumeric characters that must be in a password.
Min. Number of Numeric Characters	The value in this field indicates the minimum number of numeric characters that must be in a password.
Min. Number of Uppercase Characters	The value in this field indicates the minimum number of uppercase characters that must be in a password.
Min. Password Length	The minimum number of characters for a password and user name.
No Sequential Passwords	If this check box is selected, sequential passwords are not allowed. For example, if a user's current password is USER01, with this check box selected the system would not allow the user to change the password to USER02.

Parameter Title	Parameter Description																								
Number of Security Questions	When the Forgotten Password feature is enabled, this field sets the number of security questions that the user must configure.																								
Only Post HTTP Login	When this check box is selected, the User ID is included as part of the application's URL and so the User ID field is not displayed at login.																								
Password Rep Time	The value in this field is the amount of time (in days) in which a password cannot be repeated. (If the value is zero, there is no constraint on the re-use of passwords.)																								
Pretimeout Warning Time	This is the time (in minutes) prior to session timeout when a user gets a warning message that his or her session is about to expire. (The warning appears in a pop-up window.) If this parameter is set to zero, then a warning message is not generated.																								
Prevent Sequential Passwords	When this check box is selected, a user cannot create a new password that sequentially follows his or her existing password. For example, if her password is USER01, selecting this check box would prevent her from creating a new password called USER02.																								
Right to Left	When this check box is selected, text in name fields will be positioned so as to read from right to left. When it is clear, text will read from left to right.																								
Security Answer Minimum Length	This is the minimum number of characters required for the answer(s) to the security question(s) used for the Forgot Password feature.																								
Send Messages	When this check box is selected, material transaction messages are allowed to be sent.																								
Session Timeout	When no activity occurs for the time (in minutes) specified by this parameter, the system will automatically log-out the user. If this parameter is set to zero or null, a user's session will never time-out.																								
Table Import Root	This field is for importing and shows the root of the network share.																								
Time Format	This field sets how time is displayed. These functions use the same 'format' strings as the java.text.SimpleDateFormat class as shown in the following table: <table border="1"><thead><tr><th>Value</th><th>Format (Full)</th><th>Format (Short)</th></tr></thead><tbody><tr><td>Hour (0 - 11)</td><td>KK (2 digits)</td><td>K (1 or 2 digits)</td></tr><tr><td>Hour (12 -24)</td><td>hh (2 digits)</td><td>h (1 or 2 digits)</td></tr><tr><td>Hour (0 - 23)</td><td>HH (2 digits)</td><td>H (1 or 2 digits)</td></tr><tr><td>Hour (1 - 24)</td><td>kk (2 digits)</td><td>k (1 or 2 digits)</td></tr><tr><td>Minute</td><td>mm (2 digits)</td><td>m (1 or 2 digits)</td></tr><tr><td>Second</td><td>ss (2 digits)</td><td>s (1 or 2 digits)</td></tr><tr><td>AM/PM</td><td>a</td><td></td></tr></tbody></table>	Value	Format (Full)	Format (Short)	Hour (0 - 11)	KK (2 digits)	K (1 or 2 digits)	Hour (12 -24)	hh (2 digits)	h (1 or 2 digits)	Hour (0 - 23)	HH (2 digits)	H (1 or 2 digits)	Hour (1 - 24)	kk (2 digits)	k (1 or 2 digits)	Minute	mm (2 digits)	m (1 or 2 digits)	Second	ss (2 digits)	s (1 or 2 digits)	AM/PM	a	
Value	Format (Full)	Format (Short)																							
Hour (0 - 11)	KK (2 digits)	K (1 or 2 digits)																							
Hour (12 -24)	hh (2 digits)	h (1 or 2 digits)																							
Hour (0 - 23)	HH (2 digits)	H (1 or 2 digits)																							
Hour (1 - 24)	kk (2 digits)	k (1 or 2 digits)																							
Minute	mm (2 digits)	m (1 or 2 digits)																							
Second	ss (2 digits)	s (1 or 2 digits)																							
AM/PM	a																								
Type Ahead Delay	This is the time threshold (in milliseconds) for the type-ahead feature to disengage.																								
Use Cookies?	When this check box is selected, a User_ID cookie is saved on the browser.																								
Use Master Code Cost	This parameter is usually hidden, but may be accessed directly through the database. If this parameter is set to 1, then all material unit costs come from the MATERIAL_MASTER table. Generally, this parameter is set to 1 for agencies that use a third-party material management system having statewide unit costs. (When this parameter is set to 0, material unit costs are calculated internally by stock bin within each warehouse as total cost of the stock bin divided by the total quantity in the stock bin.)																								

Parameter Title	Parameter Description
Use Security Questions	This check box is displayed in some applications. When selected, the Forgotten Password feature is enabled. See page 11 for more information on this feature.

#### NOTE

Setting a parameter's value to zero or leaving the field blank means that no constraint is applied. For example, setting Life of Password to 0 (zero) means that a user's password never expires.

## 4.8. Tables

The Tables window allows you to create most any table as well as a window and menu item to display the information in the table.

Note: The Tables window is part of the feature set that allows users to control the creation and maintenance of certain tables, windows, and panes. See the Overview of User-controlled Tables, Windows, and Calculations section on page 42 for further information on the different types of tables.

### 4.8.1. Description of the Tables Window

(System > Utilities > Tables)

In DB?	Table Name	Table Label	Split Type	Table Type	XY Coor. Ref. Sys.	Comments
<input checked="" type="checkbox"/>	AAA			General		
<input type="checkbox"/>	ACC_ACTIVITY_OBJ_MATERIAL			General		
<input type="checkbox"/>	CARCASS_CLASS_CODE			Class Code		
<input type="checkbox"/>	CARCASS_INVENTORY		Don't split data on split location	Inventory		
<input type="checkbox"/>	CA_CULVERTS_PREFIX_XWALK	Culverts Prefix Xwalk		General		Culverts Prefix Xwa
<input type="checkbox"/>	CA_TIGER_ROUTES_MODIFIED_SHP			General		
<input type="checkbox"/>	CA_WEB_DOCS	Caltrans Web Documents (Reports and Maps)		General		
<input type="checkbox"/>	EQUIPMENT_CLASS_CODE			Class Code		
<input type="checkbox"/>	EQUIPMENT_INVENTORY			Inventory		
<input type="checkbox"/>	LABOR_CLASS_CODE			Class Code		
<input type="checkbox"/>	LABOR_INVENTORY	Labor Inventory		Inventory		
<input type="checkbox"/>	MODULE_REPORT_TABLE			General		Controls in which
<input type="checkbox"/>	MS_TREATMENT			General		This table has the
<input type="checkbox"/>	NETWORK_LANES_COUNT			Location		Table that lists the
<input type="checkbox"/>	NETWORK_MASTER	Network Master Data	Split data on split location	Location		Analysis table util
<input type="checkbox"/>	NETWORK_SECTIONS	Network master pavement sections	Split data on split location	Location		

The Tables window contains the following tabs:

- Tables – This tab lists all tables created via this window as well as all inventory and class code tables created from the Asset Type window.
- Columns – For the table selected in the Tables tab, this tab lists all columns in the table.
- Indexes – For the table selected in the Tables tab, this tab lists the indexes and primary keys for the table.

- Java Triggers – For the table selected in the Tables tab, this tab lists the Java triggers and the AfterChange type of Groovy scripts that affect the table. (The Groovy scripts are essentially given a Java "wrapper" so they will run in an Oracle environment.)

### **Tables Tab**

The Tables tab lists each table created using the Tables window. It also shows any inventory and class-codes tables created via the Asset Type window. The Table Name column shows the name of the table as it exists in the database; it is set by the user during the Insert process.

The Keep Change History? column allows you to introduce temporality into the database. When this check box is selected for a table, the application maintains previous versions of the data in the table. You may then create an "as-of date" window to display the data at a certain time in the past. (See page 151 for information on creating an "as-of date" window.)

#### **NOTE**

If you introduce temporality into a table, all related tables must also have temporality. If the table has LOC\_IDENT and/or COMMENTS\_ID, the system will automatically create mirror tables for these. You must manually set up temporality for any SETUP\_\* table having a column in the main table as well as the SETUP\_NETWORK\_LINES table (and any other alternate location-referencing tables). Finally, if the main table has GEOM in it, then the map also has temporality.

When you right-click the tab, a shortcut menu is displayed. This menu contains the following special commands that apply to the table selected in the Tables tab:

- **Insert** – This command provides the ability to include an already existing table in the Tables window or create a new table. When this command is selected, a dialog box appears. This dialog box provides two radio buttons for selecting whether the table is new or already exists in the database; a text field to enter the name of the table; and a drop-down list to select the type of table. The following are the available types of tables that may be selected:

- Location – This type corresponds to tables with location information.
- General – This type corresponds to all tables not covered by the other two.
- Setup – This type corresponds to list tables.

Note: Inventory tables are omitted from this list because they are created in the Asset Type window; see page 164.

See page 142 for more detailed instructions on creating a data table with location information; see page 142 for more detailed instructions on creating a table that will provide list data.

- **Delete** – This command deletes all appropriate records from "metadata" tables (SETUP\_TABLES, SETUP\_TABLE\_COLUMNS, etc.). After deletion, you will also need to delete the menu item for the table in the Menus window.

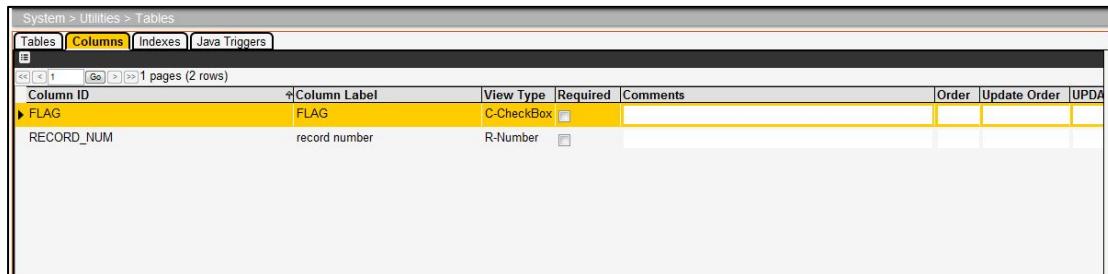
Note: This command is not available for inventory and class code tables; to delete these types of tables, delete the name of the table from the Table Name column of the Asset Type window.

- **Make Window** – This command creates a window for the table. See page 149 for further instructions on using this command.

- **Apply Changes** – This command modifies the table structure and/or indexes in the database. After you make changes in the Columns tab or Indexes tab, select this command to implement the changes in the database.

Note: Any additions or modifications to the Preparer's Comments column are sent to the Oracle system table that stores comments when you execute this command.
- **Import Table Data** – This command opens the Import Config window, which is used for configuring the routine that will import data into the selected table. See page 133 for more information on the Import Config window.

### Columns Tab



Column ID	Column Label	View Type	Required	Comments	Order	Update Order	UPDA
COLUMN_ID	FLAG	C-CheckBox	<input checked="" type="checkbox"/>				
RECORD_NUM	record number	R-Number	<input type="checkbox"/>				

The Columns tab shows the columns in the table selected in the Tables tab. When the **Apply Changes** right-click command in the Tables tab is executed, it updates the table structure using the column definitions from this tab.

The columns in this tab provide the following information:

- Column ID – This is the internal name that identifies the column.
- Column Label – This is the name of the column as seen in the application.
- View Type – This column identifies the type of column. This value controls how the column is configured in the Oracle table as well as how it is displayed in data windows. Valid values are: B - Color; C - Check Box; D - Date; G - Geometry; H - Big List; R - Number; S - String; and T - List.
- Required – When the check box in this column is checked, data must appear in the column. The system will also prefix the column name with an asterisk (\*) to denote that it is required.
- Audit – When the check box in this column is checked, a change history for the data in the associated column is kept and is available by clicking the Audit Link column's value in the appropriate data entry window. Note: Auditing only works on tables that have a single-column unique index.
- Update Order – This field configures the order in which calculated columns are calculated. This order is important whenever one calculated column is based upon the results of a second; in this case, the first column's Update Order must be larger than the second's. This field is ignored whenever the Update Source field is blank.
- Update Source – This field, when filled, indicates that this row in the Columns pane is a calculated column and is the SELECT SQL statement that performs that calculation. See page **Error! Bookmark not defined.** for more information on this column.
- Where Clause – This field, when filled, is the WHERE clause portion of the SELECT SQL provided in the Update Source field. This field is ignored whenever the Update Source field is blank. See page **Error! Bookmark not defined.** for more information on this column.

- Has Message — A check appears in this check box when the display of a system message is configured that depends on data in the column. See the **Setup Messages** command below for more information.

When you right-click a row, a shortcut menu is displayed. This menu contains the following commands in addition to the common commands:

- **Add New Column** — This is the first of two ways in which records can be inserted into this pane. This command defines a new column name in the system and applies it to the current table.

When you select this command, the system displays the first of a series of dialog boxes. Enter the column's ID (by convention this should be in upper case and end in `_ID`), column label (name), right to see, right to edit, View Type (see description above), split rule (see page 119), and other attributes of the column.

Note: If the column's ID already exists, an error message will appear and this process will be canceled. If it does not already exist, then a record will be inserted into this pane. The column will be added to the table selected in the Tables pane and will also be added to the available columns shown in the Columns window.

Note: Any information you include in the Comments text field in the dialog box is sent to the Oracle system table that stores comments.

- **Add Existing Column** — This is the second of two ways in which records can be inserted into this pane. This command accesses an existing column name and applies it to the current table.

When you select this command, the system displays a new window. Enter the column's ID in this window (you need to know the exact ID; access the Columns window to obtain this). If it does not already exist, an error message will appear and this process will be canceled. If it does exist, then a new record will be inserted into this pane and the column will be added to the table selected in the Tables pane.

- **Setup Message** — This command displays the Setup Message window. This window allows you to configure a message that is triggered by data in the column when the system job Send Column Specific Message is run. The Setup Message window contains three panes:

- The upper Messages pane configures the text of the message and the conditions that cause the message to be generated. It also indicates what security role the user must possess to be able to view the message.
- The lower left Modules pane indicates what modules will show the message (in the module's Messages window). A New Message alert is shown in the left gutter of the module to inform the user that a message is available.
- The lower right Access Regulated Columns pane lists the columns in the table that are access-regulated. When an access-regulated column is selected, then the data that is intended for the column becomes a final criterion for whether the message will display. The user will only see the message if the value for the access-regulated column entered at log-on matches the value in the record that is otherwise causing the message to be displayed.

## Indexes Tab

The screenshot shows the 'Indexes' tab in a software application. The top pane displays a table of indexes for the selected table. The middle-left pane lists all available columns for the table. The middle-right pane lists the columns currently defined in the selected index. Two arrows between the middle panes allow for moving columns between them.

TABLE INDEX NAME	Primary Key?	Is Unique?	Comments	Att.	User Update	Date Update
CA_CULVERTS_PREFIX_XWALK_UI	<input type="checkbox"/>	<input checked="" type="checkbox"/>		EUGENE		1/10/2012

Column ID	Column Label	View Type	Required
CA_CUL_DEF	Definition	S-String	<input type="checkbox"/>
CA_CUL_PREFIX	Prefix	S-String	<input type="checkbox"/>
COMMENT_ID	Att.	R-Number	<input type="checkbox"/>
COMMENT_STR	Comments	S-String	<input type="checkbox"/>
DATE_UPDATE	Date Update	D-Date	<input type="checkbox"/>
USER_UPDATE	User Update	S-String	<input type="checkbox"/>

Column ID	Column Label	View Type	Order
CA_CUL_CODE	Code	S-String	1
CA_CUL_SIDE	Side	S-String	2

The Indexes pane in the upper part of the tab lists the indexes for the table. The Available Columns and Index Columns panes in the lower part of the window are used to see and/or adjust the column components of the currently selected index.

The Indexes tab shows the indexes pertinent to the table identified in the Tables tab. When the **Apply Changes** right-click command in the Tables tab is executed, it updates the indexes definitions from this tab.

The Indexes tab contains three panes:

- Indexes pane – This pane lists all indexes for the table selected in the Tables tab.
- Available Columns pane – This pane lists all columns for the table selected in the Tables tab and is used to select columns to be put into the current index.
- Index Columns pane – This pane lists all columns in the index currently selected in the Indexes pane.

The index tab also contains two buttons with arrows to add and remove columns from the index. The upper arrow key adds the column selected in the Available Columns pane to the Indexes pane. The lower arrow key removes the column selected in the Indexes pane and places it in the Available Columns pane.

The columns in this tab provide the following information:

- The Primary Key? column in the Indexes pane sets the index to be the primary key. Only one index (per table) can be the primary key.
- The Unique? column in the Indexes pane sets the index to be unique.
- The Order column in the Index Columns pane sets the order of the columns in the index.

## Java Triggers

System > Utilities > Tables		
Tables	Columns	Indexes
Java Triggers		
<<	1	>> 1 pages (1 rows)
Class Name	Order	Dynamic Create Parameters
com.agileassetsinc.trigger.Groovy	12	

The Java Triggers tab lists those Java scripts that trigger changes in the table selected in the Tables tab. It also lists AfterChange type Groovy scripts that affect the table.

Since Groovy scripts cannot run in an Oracle environment as such, each is given a Java "wrapper" that enables the Groovy script to run. The name of this wrapper is the same for all AfterChange Groovy scripts: com.agileassetsinc.trigger.GroovyScriptTrigger. The particular AfterChange Groovy script is given in the Dynamic Create Parameters column, which shows the ID number of the Groovy script.

When you right-click a record that shows com.agileassetsinc.trigger.GroovyScriptTrigger in the Class Name column, the following command becomes available:

- **Edit Groovy Script** – This command displays the Edit Groovy Script dialog box. You use this dialog box to create the Groovy script. For information on the command buttons found in this dialog box, see page 191. After you create and test the Groovy script, you click the **OK** button to close the dialog box. The system then inserts the ID number of the new Groovy script in the Dynamic Create Parameters column (regardless of what was entered in the Name field of the Edit Groovy Script dialog box). The system also adds the new Groovy script to the Groovy Scripts window under the AfterChange type of scripts.

### NOTE

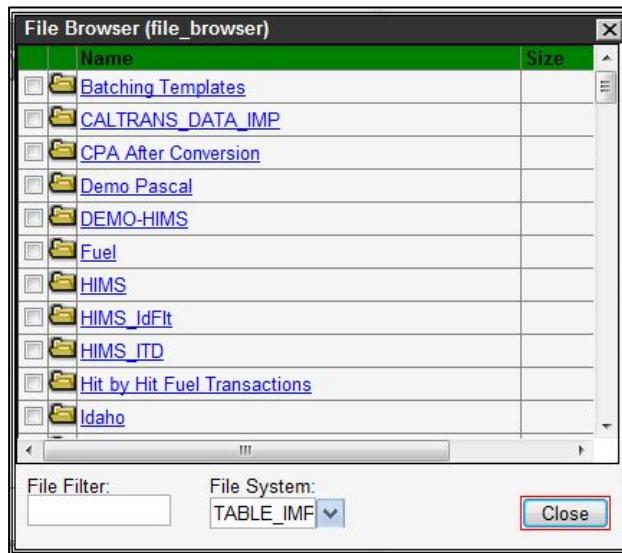
If you wish to use an existing AfterChange Groovy script rather than create a new one, do not use this command. Instead insert a new record in the Java Triggers tab, enter com.agileassetsinc.trigger.GroovyScriptTrigger in the Class Name column, and enter the ID number of the existing Groovy script in the Dynamic Create Parameters column.

### **4.8.2. Description of the Import Config Window**

The Import Config window is displayed by executing the **Import Table Data** command that is found in the shortcut menu that is displayed by right-clicking a table record in the Tables tab of the Tables window. The Import Config window contains three tabs: Imports, Import Detail, and Import Log. These tabs are described in more detail in the following sections.

Regardless of the tab that is selected, three command buttons are available. These are described below:

- **Edit Files** – This command button displays a dialog box that shows the available source files for imports. An example of this dialog box is shown below.

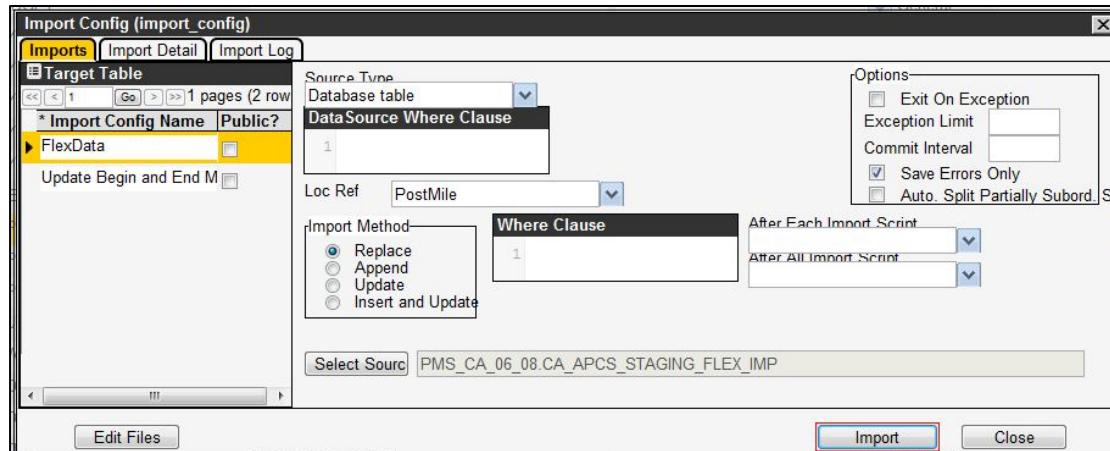


You may select a particular file (by clicking the check box for the file) and then use the right-click **Upload File** command to move the file from the shared (that is, user accessible) drives to the drive for importing. (To assist you in locating a particular file, you may enter a filter for the file in the File Filter field at the bottom of the dialog box. When you press the Enter key on the keyboard, then only those files matching the filter criteria will be displayed.) If desired, you may move files from the drive for importing to the shared drive by using the **Download Selected** command.

Note: The system supports multiple root folders. If multiple root folders are configured in the Setup File System window (which is found in the Setup menu of the System module), then the File System field will be shown at the bottom of the File Browser dialog box. This field contains a drop-down list of all available root folders. When you select a folder from the list, the system will refresh the list of files to show those files in that folder.

- **Import** – This command button imports the data from the source table into the target table. The result of the import is shown in the Import Log tab.
- **Close** – This command saves any changes you may have made and then closes the Import Config window.

### Imports Tab



**NOTE**

The **Insert Like** command creates a new import configuration and copies fields from **both** the Imports and Import Detail tabs.

The Imports tab of the Import Data window lists all import routines that have been configured to import data into the table selected in the Tables window. The left pane lists the import routines.

In the left pane, the **Is Public?** check box determines whether the routine will be available in the Import Runner window. When this check box is selected, the routine appears in the Import Runner window and may be run from that window rather than the Tables window.

The right pane provides the fields for configuring the general aspects of the import routine selected in the left pane. This pane also provides fields in the Options box to halt the import if desired. When halted, the system rolls back all changes since the last commit to the database. The fields in the Options box are described below:

- When the **Exit on Exception** check box is selected, then the system will stop the import process when the number of exceptions entered in the **Exception Limit** field is reached (or the first exception if this field is left blank).
- Regardless of the setting of the **Exit on Exception** check box, the **Commit Interval** field determines when records are committed to the database based on the number of records processed. For example, if this field is set to 1000, then after the 1,000th record is processed the system will commit the last 1,000 records to the database.

Note: Once records are committed to the database, an exception will not cause those records to be restored to the condition they were in before the import process began.

- If the **Save Errors Only** check box is selected, then the system will save the text describing exceptions to a log file.

The fields that are displayed in the right pane are dependent on the selected Source Type. The following table lists the different types of sources from which data may be imported and the fields that become available when you select a particular source type. Following the table is a description of the different fields that may appear in the right pane.

Source Type	Select Source	Data Source Where Clause	Import Method	Where Clause	After Each Import Script	After All Import Scripts	Has Header
Access file	x	x	x		x	x	
Comma separated	x		x		x	x	x
Database table	x	x	x		x	x	
Dbf file	x		x		x	x	
Excel file	x		x		x	x	x
Fixed length	x		x		x	x	x
SQL server table	x	x	x	x	x	x	
Shape file	x		x		x	x	

The fields are described below:

- **Select Source** — This button allows you to select the source that contains the data to be imported. When you click the button, the application displays a new window with the sources that are appropriate to the selected source type. (For example, selecting Access file as the source type would cause all Microsoft Access database files [\*.mdb] to be displayed when you click the **Select Source** button.)

Note: The location where the application looks for files is set at the time of configuration. When the project manager (typically) enters a root folder in the File System window for imports, then you will see some or all folders and files under that root folder when you select this command provided your user ID has an administrative unit set in the User Names window.

- If the administrative unit column is blank, then you will see all folders and files under that root folder.
- If the administrative unit column is filled, then you will only see a file folder and its contents under that root folder that was especially made for your administrative unit. Ask your system administrator for that subfolder's name. (Note to system administrator: The file folder name must be the internal ID of the administrative unit.)
- Data Source Where Clause — If you would like to restrict the data imported from the data source, you may enter an SQL statement in this field to identify the records you want to import.
- Import Method — This field provides the following radio buttons to select what will happen to the records in the target table:
  - The Replace method truncates the target table and then begins inserting new records from the data source.
  - The Append method does not truncate the target table; new records are simply inserted at the end of the existing target table.
  - In the Update method, the import routine looks for an existing record in the target table that matches a record in the data source and then updates the target table's record with data from the data source's record. (If a record is not found in the target table, the record in the data source is ignored.)
  - The Insert & Update method is the same as the Update method except if a record is not found in the target table, then the record is added to the target table (rather than being ignored).

Note: When you select either the Update or Insert & Update method, an additional field is added to the pane called Update by Index. This is a drop-down list that contains the different means by which a record may be identified.

- Where Clause — When you select the Insert import method, you may enter an SQL statement in this field to identify what records are deleted from the target table.
- After Each Import Script — This field contains a drop-down list of After Each Import Groovy scripts. If desired, you may select one to run after each record is imported.
- After All Import Script — This field contains a drop-down list of After All Import Groovy scripts. If desired, you may select one to run at the very end of the import routine.
- Has Header — When this check box is selected, the first record of the data source is a header record.

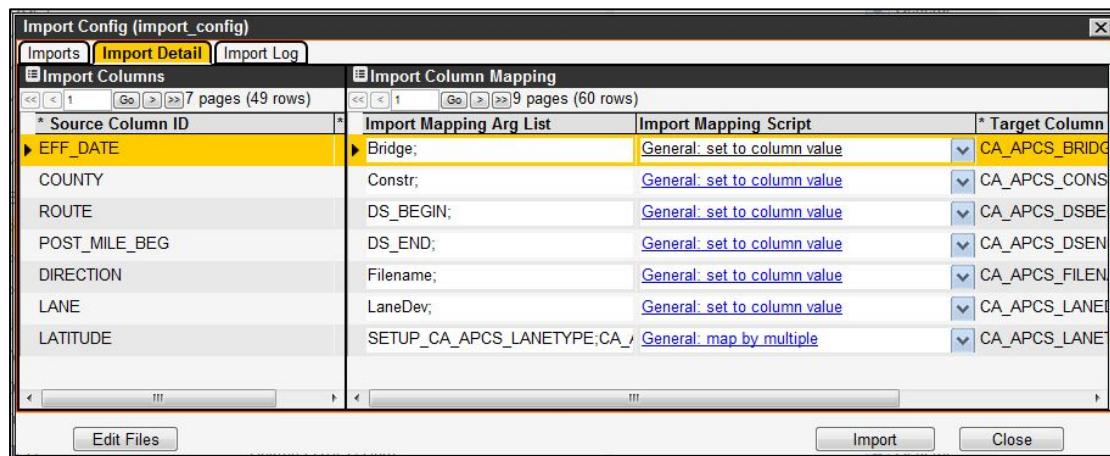
In addition to the fields that vary depending on the source type, two check boxes are always available:

- Exit on Exception — When this check box is selected, the first detected error causes processing to stop and data is restored to its condition before the import routine began. The system does not generate a log file, but the Log tab shows the fatal error message.
- Save Errors Only — When this check box is selected, then only records with errors are saved in the log file.

After the details of the import are set in the Import Detail tab, the **Import** button at the bottom of the Import Data window will execute the import selected in the Target Table pane of the Imports tab.

Instead of running the import immediately, you may instead click the **Close** button, which saves the changes to the import definition so that the import may be run at a later time from the Tables window or the Import Runner window (if the Is Public? check box is selected for the routine).

### **Import Detail Tab**



For the import routine selected in the Imports tab, the Import Detail tab provides the mapping from the columns of the data source to the columns of the target table. This tab contains two panes: Import Columns and Import Column Mapping.

#### Description of the Import Columns Pane

The left pane shows the columns in the data source. The application will automatically populate this pane with the column names from the data source except for the Fixed length source type and provided the Has Header check box is selected for the Comma separated and Excel file source types. (If the check box is not selected, the application still creates records for each column, but these are not labeled. You may subsequently edit the records to include the appropriate column name.)

For the Fixed length source type, you will need to use the **Insert** command to add records to the pane. You then enter in each record the name of the column and its size.

When you right-click this pane, a shortcut menu is displayed. This menu contains the common commands along with a command called **Refresh Source Columns**. This command updates the list of columns shown in the pane.

### Description of the Import Column Mapping Pane

The right pane shows the mapping from one or more columns in the data source to a column in the target table. Each record in this pane is for one column in the target table as indicated by the Target Column column. (You may update the list of target table columns by right-clicking the pane and then selecting the **Refresh Target Columns** command.)

The Import Mapping Arg List column contains the input parameters for the ImportMapping Groovy script selected in the Import Mapping Script column that transfers and, if desired, adjusts the data from the source to a target column.

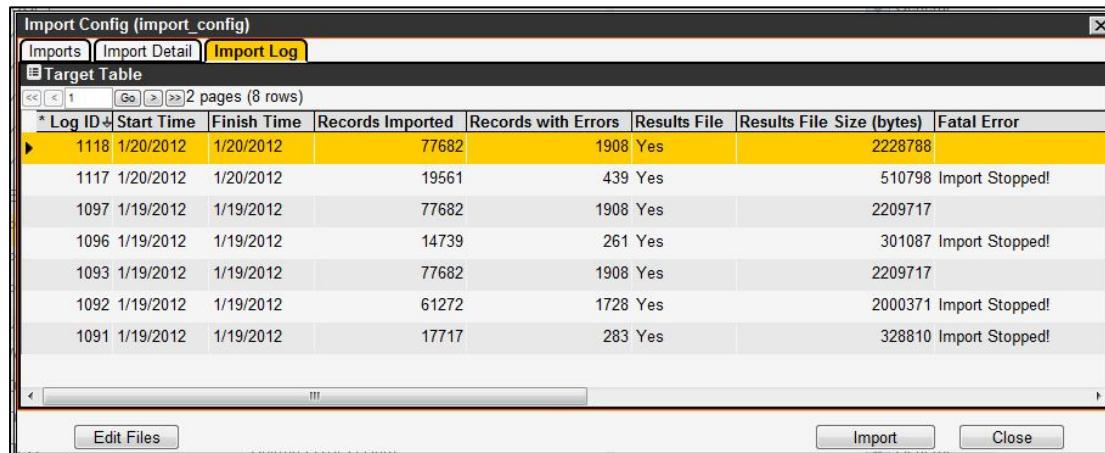
To enter (or modify) input parameters, right-click the selected record and then click **Edit Arguments**. The application responds by displaying a dialog box. The left side of this dialog box contains the list of columns shown in the Import Columns pane of the underlying window. The right side of the dialog box shows the input parameters. Two arrows are provided between the panes to move a column from one pane to the other. Additionally, an **Add Argument** button is provided so you may insert a blank record in the argument list on the right. The arguments in the right pane are executed from the top to the bottom (first to last record) as indicated by the value in the Order column. If desired, you may change the values in this column to change the order of argument execution.

When the dialog box shows the desired argument, click the **OK** button to close the dialog box. (The **Cancel** button discards any changes you made and closes the dialog box.)

#### NOTE

Rather than selecting an ImportMapping Groovy script, you may create a new script by utilizing the **Create/Edit Groovy Script** command. This command is found by right-clicking the selected record in the right pane. After selecting the command, the application displays a dialog box for creating or modifying the Groovy script. More information on this dialog box may be found on page 191.

### **Import Log Tab**



*	Log ID	Start Time	Finish Time	Records Imported	Records with Errors	Results File	Results File Size (bytes)	Fatal Error
▶	1118	1/20/2012	1/20/2012	77682	1908	Yes	2228788	
	1117	1/20/2012	1/20/2012	19561	439	Yes	510798	Import Stopped!
	1097	1/19/2012	1/19/2012	77682	1908	Yes	2209717	
	1096	1/19/2012	1/19/2012	14739	261	Yes	301087	Import Stopped!
	1093	1/19/2012	1/19/2012	77682	1908	Yes	2209717	
	1092	1/19/2012	1/19/2012	61272	1728	Yes	2000371	Import Stopped!
	1091	1/19/2012	1/19/2012	17717	283	Yes	328810	Import Stopped!

Each time the import routine selected in the Imports tab is run, a record is inserted into the Import Log tab. The record shows when the import was run, the number of records imported, the number of errors (if any), and whether a results ("log") file is available. (To view the actual data transferred by the import routine and any errors that were detected, right-click the record for the import routine and then click **Download File**. See below for more information on this command.)

**NOTE**

A system job is available to remove all records that are over one month old except for the record of the last run of the import routine.

When you right-click the tab, a shortcut menu is displayed. This menu contains the following special commands that apply to the table selected in the Tables tab:

- **Download File** — This command is only available when Yes is shown in the Results File column. When available, it sends the data transferred by the import routine and any detected errors to an external CSV file. You may then use an application such as Microsoft Excel to view the transferred data.

**NOTE**

If the Save Errors Only check box is selected on the Imports tab, then the downloaded log file will only show those records with errors.

If the Exit on Exception check box is selected on the Imports tab, then a log file is not available. Instead, the record on the Import Log tab will show the error that was detected and stopped the import routine.

### ***ImportMapping Groovy Scripts***

The following table lists the ImportMapping Groovy scripts that are most commonly available for selection in the Import Mapping Script column of the Import Column Mapping pane. (There may be other ImportMapping Groovy scripts that are project-specific; see the project manager for more details.)

Name of Groovy Script	What It Does	Arguments
Concatenate	Concatenates any number of strings including constants.	Each piece (source column / constant) to concatenate in order.
concatenate 2 strings	Concatenate two strings.	The two strings (source column / constant) in order.
convert to integer	Converts value to an integer.	Source column / constant.
map by column comment	Works for target "list" type columns. Matches source column with the COMMENT_STR column in the SETUP table of the "list" column.  On exception sends NULL to target.	Source column / constant.
map by column name	Works for target "list" type columns. Matches source column with the *_NAME column in the SETUP table of the "list" column.  On exception sends NULL to target.	Source column / constant.
map by column name required	Does exactly the same as "map by column name" except that exception processing is different.  On exception Issues error and does not import the row.	Source column / constant.
map by xwlk	Works for target "list" type columns. Is a "double table" lookup intended to map many source values into a few target "ID" values.  1. Matches source column with the SOURCE_NAME column	Source column / constant.

	in the SETUP_*_XWLK table of the “list” column, then 2. Matches the SOURCE_NAME column in the SETUP_*_XWLK table with the *_NAME column in the SETUP table of the “list” column.  On exception sends NULL to target.	
set to column value	Sends source value directly to target.	Source column / constant.
to date	Converts the source value to a “date” type target column.	Source column / constant.
today	Sends SYSDATE to a “date” type target column.	
Y_N_column to C-type column	Y_N_column to C-type column.	Source column / constant.

### ***How to Import Into Loc\_Ident***

When a table to be imported contains the column LOC\_IDENT, then the Import Column Mapping pane of the Import Details tab automatically includes all “location” columns, those noted below are for the PostMile Location Reference Method:

- LOC\_IDENT
- CA\_COUNTY\_FROM (County)
- CA\_COUNTY\_TO (County)
- CA\_PM\_PREFIX\_FROM (Post Mile Prefix From)
- CA\_PM\_PREFIX\_TO (Post MilePrefix To)
- CA\_PM\_SUFFIX\_FROM (Post Mile Suffix From)
- CA\_PM\_SUFFIX\_TO (Post Mile Suffix To)
- CA\_POSTMILE\_BEG (PostMile begin)
- CA\_POSTMILE\_END (Post Mile End)
- CA\_ROUTE\_FROM (Route)
- CA\_ROUTE\_TO (Route)
- CA\_RTE\_SUFFIX\_FROM (Route Suffix)
- CA\_RTE\_SUFFIX\_TO (Route Suffix)
- LANE\_DIR (Direction)
- LANE\_ID (Lane)
- PERPEN\_OFFSET (Offset)

To identify “location” there should be a mapping to each of these columns from the source data. For example, if the source data had the normal system route name and mile points, L/R for direction, no lanes and no perpendicular offset, then the mapping would be as shown in the table below (Note: This example mapping is from Caltrans APCS data.):

<b>Import Mapping Arg List / Source</b>	<b>ImportMapping Groovy Script</b>	<b>Target</b>	<b>Notes</b>
1;	Max loc_ident	LOC_IDENT	Always use this ImportMapping Groovy script to set LOC_IDENT.
SETUP_CA_COUNTY;CA_COUNTY_ID;County;CA_COUNTY_ABBREV;	General: map by multiple	CA_COUNTY_FROM (County)	
SETUP_CA_COUNTY;CA_COUNTY_ID;County;CA_COUNTY_ABBREV;	General: map by multiple	CA_COUNTY_TO (County)	
PM_PREFIX;	set to column value	CA_PM_PREFIX_FROM (Post Mile Prefix From)	
PM_PREFIX;	parse from Post Miles	CA_PM_PREFIX_TO (Post MilePrefix To)	
PM_SUFFIX;	set to column value	CA_PM_SUFFIX_FROM (Post Mile Suffix From)	
PM_SUFFIX;	set to column value	CA_PM_SUFFIX_TO (Post Mile Suffix To)	
POST_MILE_BEG;	set to column value	CA_POSTMILE_BEGIN (PostMile begin)	
POST_MILE_END;	set to column value	CA_POSTMILE_END (Post Mile End)	
Route;	Parse Ca Route and Rte Suffix	CA_ROUTE_FROM (Route)	
Route;	Parse Ca Route and Rte Suffix	CA_ROUTE_TO (Route)	
Route;	Parse Ca Route and Rte Suffix	CA_RTE_SUFFIX_FROM (Route Suffix)	
ROUTE;	Parse Ca Route and Rte Suffix	CA_RTE_SUFFIX_TO (Route Suffix)	
DIRECTION;"NB";1;"SB";2;"EB";1;2;	General: case statement	LANE_DIR (Direction)	
Lane;	General: set to column value	LANE_ID (Lane)	
0;		PERPEN_OFFSET (Offset)	

#### **4.8.3. How to Create a Table with Location Data**

Follow these steps to create a new table that contains location data with the Tables window:

1. Open the Tables window.
2. In the Tables tab, right-click and then click **Insert**. The application displays a dialog box.
3. For the purposes of this example, assume that the table does not yet exist in the database. Therefore, in the Insert dialog box click the radio button beside **New**.
4. Click the arrow for the drop-down list and then click **Location**.
5. Enter the internal name of the new table in the text field in the lower part of the dialog box.
6. Click **OK**. The application closes the dialog box and adds a new record to the table. (The table is not actually created in the database at this point.)
7. In the new record, enter the name of the new table in the Table Label column and, optionally, any additional information in the Comments column.
8. Click the Columns tab. This tab shows the default columns for any table (comments, user ID, and user update date).
9. Right-click the table in the Columns tab and then select the appropriate command: either **Add New Column** if the column does not exist in the database or **Add Existing Column** if the column does exist.
10. Repeat step 9 for each column.
11. If desired, click the Index tab and create an index for the table; see page 149 for more information.
12. Click the Tables tab.
13. Right-click the record for the new table and then click **Apply Changes**. The application now creates the table in the database with the columns specified in the Columns tab.
14. Click the  icon.

The table is now created. Next you will need to create the window for the display of data from the table. Three general types of windows may be created: one that displays data in a tabular format (a data window; see page 149); one that displays data graphically along a route, provided location information is in the database table (a graph window; see page 150); and a popup window that is launched via hyperlinks in a column (a popup window; see page 150).

#### **4.8.4. How to Create a List Table**

Follow these steps to create a new table that contains data that will be used in a drop-down list:

1. Open the Tables window.
2. In the Tables tab, right-click and then click **Insert**. The application displays a dialog box.
3. For the purposes of this example, assume that the table does not yet exist in the database. Therefore, in the Insert dialog box click the radio button beside **New**.
4. Click the arrow for the drop-down list and then click **Setup**.

5. Enter the internal name of the new table in the text field in the lower part of the dialog box.
6. Click **OK**. The application closes the dialog box and adds a new record to the table. (The table is not actually created in the database at this point.)
7. In the new record, enter the displayed name of the new table in the Table Label column and, optionally, any additional information in the Comments column.
8. Click the Columns tab. This tab shows the default columns for any table (comments, user ID, and user update date).
9. Right-click the table in the Columns tab and then select the appropriate command: either **Add New Column** if the column does not exist in the database or **Add Existing Column** if the column does exist.
10. Repeat step 9 for each column.
11. If desired, click the Index tab and create an index for the table; see page 149 for more information.
12. Click the Tables tab.
13. Right-click the record for the new table and then click **Apply Changes**. The application now creates the table in the database with the columns specified in the Columns tab.
14. Click the  icon.

The table is now created. Next you will need to create the window for the display of data from the table. See page 149 for more information.

#### **4.8.5. How to Create an Inventory Table**

A table that contains inventory information for a particular asset type is handled differently than other types of tables. An inventory table is created in the Asset Types window. (The application also automatically creates a class codes table to categorize the items in inventory when the inventory table is created.)

##### **NOTE**

Making an inventory table (and its associated class codes table) via the Asset Types window is the preferred and recommended method. However, as an exception, you may create inventory tables (that is tables with names ending in \_INVENTORY) and class code tables (that is, tables with names ending in \_CLASS\_CODES) in the Tables window via the **Insert** command. In this case, once you have inserted the tables, begin the steps shown below at step 5.

Once this is accomplished, the names of the inventory and class codes tables will appear in the Tables window. You may then create windows to display the data from these tables along with supporting status and PM windows if desired.

The following steps describe how to create an inventory and other supporting windows:

1. If you have not already done so, open the Asset Types window and enter the name of the inventory table for the asset type as described on page 165.
2. Open the Tables window.

3. In the Tables tab, locate the record for the class codes table for the asset type. Right-click this record and then click **Apply Changes**. The class codes table now exists in the database.
4. In the Tables tab, locate the record for the inventory table for the asset type. Right-click this record and then click **Apply Changes**. The inventory table now exists in the database.
5. If your application utilizes LRS transactions, you must create an inventory details table that is associated with the inventory table. (For applications that do not utilize LRS transactions, it is optional whether you create an inventory details table.) To create an inventory details table, perform the following steps:
  - a. In the Tables tab, right-click and then click **Insert**. The application displays a dialog box.
  - b. Click the radio button beside **New**.
  - c. Click the arrow for the drop-down list and then click **Inventory Details**.
  - d. In the lower part of the dialog box, enter the internal name of the new table in the text field. This name must end in **\_INVEN\_DET**.
  - e. Click **OK**. The application closes the dialog box and adds a new record to the table.
  - f. In the new record, enter the name of the new table in the **Table Label** column and, optionally, any additional information in the **Comments** column.
  - g. Click the **Columns** tab. This tab shows the default columns for the table.
  - h. Right-click the table in the **Columns** tab and then select the appropriate command: either **Add New Column** if the column does not exist in the database or **Add Existing Column** if the column does exist.
  - i. Repeat the previous step for each column.
  - j. If desired, click the **Index** tab and create an index for the table.
  - k. Click the **Tables** tab.
  - l. Right-click the record for the new table and then click **Apply Changes**. The application now creates the table in the database with the columns specified in the **Columns** tab.
  - m. Click the  icon.
6. In the list of tables, find the record for the inventory table. Right-click this record and then click **Make Window**. The application displays a new window. The default selection in the dialog box is to create an inventory window.
7. Since the inventory table option is already selected, just click **Next**. The application displays a second new window.
8. In the second new window, check that the window title is what you want. If necessary, modify the title as desired. (The **Editable?** check box is already selected as is appropriate.)
9. In the lower part of the dialog box, expand the menu hierarchy to locate the parent menu item under which a new menu item for all windows (inventory, class codes, status, etc.) associated with the asset type will be placed.

10. Right-click the parent menu item and then click **Add Branch**. The application displays a new dialog box so you may enter the name of the menu item.
11. In the dialog box, in the New Menu ID field enter the internal ID for the menu item under which the menu items to open all the windows associated with the asset type will appear. (This ID should be lower case and not contain spaces.) Also, in the New Menu Name field, enter the name of the menu item that will be displayed in the menu hierarchy.
12. Click **OK** to close the dialog box. A new node is added subordinate to the parent menu item you right-clicked in step 10. Note: The application automatically selects this node, so you will need to de-select it to continue.
13. Click the square beside the name of the new menu item to remove the check mark from it.
14. Right-click the new menu item created in step 10 and then click **Add Branch**. The application displays a new dialog box so you may enter the name of the menu item that will display the inventory window for the asset type.
15. In the dialog box, in the New Menu ID field enter the internal ID for the menu item that will open the inventory window. (This ID should be lower case and not contain spaces.) Also, in the New Menu Name field, enter the name of the menu item that will be displayed in the menu hierarchy.
16. Click **OK** to close the dialog box. A new node is added subordinate to the menu item you right-clicked in step 12. This node is automatically selected, which is appropriate this time.
17. Click **OK**. The window and menu item for the display of the asset type's inventory is now created. If you created an inventory details table that is associated with the inventory table, the system automatically splits the inventory window into two panes (the upper pane shows inventory information and the lower pane shows the details). Otherwise, the window is not split.
18. Now create a status window to categorize the items in inventory. To start, right-click the record for the inventory table and then click **Make Window**. The application displays a new window.
19. The Main radio button is already selected as required, so click the down arrow to display the list and then click Status Window.
20. Click the **Next** button. The application displays a second new window.
21. In the second new window, check that the window title is what you want. If necessary, modify the title as desired. (The Editable? check box is already selected as is appropriate.)
22. In the lower part of the dialog box, expand the menu hierarchy to locate the menu item created in step 10 under which all windows associated with the asset type will be placed.
23. Repeat steps 12 through 15. The Status window and menu item are now created.
24. If you would like to perform PM activities on items in inventory, you may create a PM by Inventory window and menu item as described in the following steps:
  - a. Right-click the record for the inventory table and then click **Make Window**. The application displays a new window.
  - b. The Main radio button is already selected as required, so click the down arrow to display the list and then click PM by Inventory.

- c. Click the **Next** button. The application displays a second new window.
  - d. In the second new window, check that the window title is what you want. If necessary, modify the title as desired. (The Editable? check box is already selected as is appropriate.)
  - e. In the lower part of the dialog box, expand the menu hierarchy to locate the menu item created in step 10 under which all windows associated with the asset type will be placed.
  - f. Repeat steps 12 through 15. The PM by Inventory window and menu item are now created.
25. In various windows of the application, you may select a portion of a map and display information for items of a selected asset type that fall within the selected area. If you would like to have this capability for this particular asset type, perform the following steps:
- a. Right-click the record for the inventory table and then click **Make Window**. The application displays a new window.
  - b. Click the Popup radio button. The Popup list field is activated.
  - c. Click the down arrow to display the list and then click **Inventory Popup**.
  - d. Click the **Next** button. The application displays a second new window.
  - e. In the second new window, check that the displayed information is correct. Modify or add any information that is needed.
  - f. Click **OK** to complete the process.
26. You now need to create the window and menu item to display the information in the class codes table that is associated with this asset type's inventory. Locate the record for the class codes table, right-click this record, and then click **Make Window**. The application displays a new window. The default selection in the dialog box is to create a class codes window.
27. Since the class codes table option is already selected, just click **Next**. The application displays a second new window.
28. In the second new window, check that the window title is what you want. If necessary, modify the title as desired. (The Editable? check box is already selected as is appropriate.)
29. In the lower part of the dialog box, expand the menu hierarchy to locate the menu item created in step 10 under which all windows associated with the asset type will be placed.
30. Right-click the menu item and then click **Add Branch**. The application displays a new dialog box so you may enter the name of the menu item that will display the class codes window.
31. In the dialog box, in the New Menu ID field enter the internal ID for the menu item that will open the class codes window. (This ID should be lower case and not contain spaces.) Also, in the New Menu Name field, enter the name of the menu item that will be displayed in the menu hierarchy.
32. Click **OK** to close the dialog box. A new node is added subordinate to the menu item you right-clicked. This node is automatically selected, which is appropriate.

33. Click **OK**. The window and menu item for the display of the asset type's class codes is now created.
34. If you would like to perform PM activities on items in inventory by utilizing class codes, you may create a PM by Class Codes window and menu item as described in the following steps:
  - a. Right-click the record for the class codes table and then click **Make Window**. The application displays a new window.
  - b. The Main radio button is already selected as required, so click the down arrow to display the list and then click **PM by Class Code**.
  - c. Click the **Next** button. The application displays a second new window.
  - d. Repeat steps 26 through 31. The **PM by Class Code** window and menu item are now created.

The process for creating inventory and associated windows and menu items is now complete.

#### **4.8.6. How to Create a Generic Table**

A generic table is one that is not a table with location data, nor is it a table that contains a list used for setup, nor does it contain inventory data. To create this type of table, follow these steps:

1. Open the Tables window.
2. In the Tables tab, right-click and then click **Insert**. The application displays a dialog box.
3. For the purposes of this example, assume that the table does not yet exist in the database. Therefore, in the Insert dialog box click the radio button beside **New**.
4. Click the arrow for the drop-down list and then click **General**.
5. Enter the internal name of the new table in the text field in the lower part of the dialog box.
6. Click **OK**. The application closes the dialog box and adds a new record to the table. (The table is not actually created in the database at this point.)
7. In the new record, enter the displayed name of the new table in the **Table Label** column and, optionally, any additional information in the **Comments** column.
8. Click the **Columns** tab. This tab shows the default columns for any table (comments, user ID, and user update date).
9. Right-click the table in the **Columns** tab and then select the appropriate command: either **Add New Column** if the column does not exist in the database or **Add Existing Column** if the column does exist.
10. Repeat step 9 for each column.
11. If desired, click the **Index** tab and create an index for the table; see page 149 for more information.
12. Click the **Tables** tab.
13. Right-click the record for the new table and then click **Apply Changes**. The application now creates the table in the database with the columns specified in the **Columns** tab.
14. Click the  icon.

The table is now created. Next you will need to create the window for the display of data from the table. [topic How to Make a DataWindow](#) See page 149 for more information.

#### **4.8.7. How to Add a New Column to a Table**

During the creation of a new table, or the modification of an existing table, you may add a new column as follows:

1. Open the Tables window.
2. In the Tables tab, click the record for the table into which the column will be added to select it.
3. Click the Columns tab.
4. In the Columns tab, right-click and then click **Add New Column**. The system displays the first of two dialog boxes to create the new column.
5. Complete the fields of the first dialog box as appropriate. Remember that the convention for the column ID is that it is in upper case and ends in \_ID.
6. Click **Next**. The system displays the second dialog box.
7. Complete the fields of the second dialog box to configure the column as desired.
8. Click **OK**. The system checks whether the ID exists. If it does exist, an error message will appear. If it does not exist, then a new record will be inserted into this pane.
9. Click the Tables tab.
10. In the Tables tab, right-click the selected table and then click **Apply Changes**. The system adds the column to the table.

11. Click the  icon.

#### **4.8.8. How to Add an Existing Column to a Table**

During the creation of a new table, or the modification of an existing table, you may add an existing column as follows:

1. Open the Tables window.
2. In the Tables tab, click the record for the table into which the column will be added to select it.
3. Click the Columns tab.
4. In the Columns tab, right-click and then click **Add Existing Column**. The system displays a dialog box to enter the ID or label of the column.
5. In the dialog box, type the column's ID or label. As you type the system will attempt to match what you type with existing column IDs and labels. These potential matches are shown below the field. You may continue typing or click the potential match that you want.
6. Click **OK**.
7. Click the Tables tab.
8. In the Tables tab, right-click the selected table and then click **Apply Changes**. The system adds the column to the table.

9. Click the  icon.

#### **4.8.9. How to Create an Index for the Table**

Follow these steps to create an index for a table created with the Tables window:

1. Open the Tables window.
2. In the Tables tab, select the desired table by clicking the record for the table.
3. Click the Indexes tab.
4. In the Indexes pane, right-click and then click **Insert**. The application displays a dialog box.
5. In the dialog box, enter the name of the new index and then click **OK**. The application closes the dialog box and adds a new record to the pane.
6. In the Available Columns pane, select the column to be used for indexing by clicking the record showing the column.
7. Click the right-pointing arrow between the two lower panes to move the selected column to the Index Columns pane.
8. Repeat steps 6 and 7 for any additional columns to be used for indexing.
9. When all desired columns are shown in the Index Columns pane, complete the Order column to indicate the relationship of the index columns to each other.
10. Click the Tables tab.
11. Right-click the record selected in step 2 and then click **Apply Changes**. The new index columns are now in force.

#### **4.8.10. How to Make a Data or List Window**

To create the window that will display the data in a table, follow these steps:

1. Display the Tables window.
2. Locate the table for which you will create a window. Right-click the record showing the table and then click **Make Window**. The application displays a new window. The default selection in the dialog box is to create a data window, which is the goal of this example.
3. Since the data table option is already selected, just click **Next**. The application displays a second new window.
4. In the second new window, check that the window title is what you want. If necessary, modify the title as desired.
5. If the data in the window may be modified, click the **Editable?** check box to select it.
6. In the lower part of the dialog box, expand the menu hierarchy to locate the parent menu item under which the menu item to display the window will be placed.
7. Right-click the parent menu item and then click **Add Branch**. The application displays a new dialog box so you may enter the name of the menu item.
8. In the dialog box, in the **New Menu ID** field enter the internal ID for the menu item that will open the window. (This ID should be lower case and not contain spaces.) Also, in the **New Menu Name** field, enter the name of the menu item that will open the window.
9. Click **OK** to close the dialog box. A new node is added subordinate to the parent menu item.
10. Right-click the new node and then click **Select This**. The system places a check mark in the square beside the menu name to denote that it is selected.

11. Click **OK** to make the window.

#### **4.8.11. How to Make a Graph Window**

Rather than create a window that displays data in a tabular format, you may also display data as a graph along a route (provided the underlying table in the database contains location information). This is accomplished as follows:

1. Display the Tables window.
2. Locate the table for which you will create a window. Right-click the record showing the table and then **Make Window**. The application displays a new window. The default selection in the new window is to create a data window, not a graph window.
3. For the first option (Main), click the down arrow to display the list of types of windows and then click the **Graph Along the Route Window** entry.
4. Click **Next**. The application displays a second new window.
5. In the second dialog box, check that the window title is what you want. If necessary, modify the title as desired.
6. If the data in the window may be modified, click the **Editable?** check box to select it.
7. In the lower part of the dialog box, expand the menu hierarchy to locate the parent menu item under which the menu item to display the window will be placed.
8. Right-click the parent menu item and then click **Add Branch**. The application displays a new dialog box so you may enter the name of the menu item.
9. In the dialog box, in the **New Menu ID** field enter the internal ID for the menu item that will open the window. (This ID should be lower case and not contain spaces.) Also, in the **New Menu Name** field, enter the name of the menu item that will open the window.
10. Click **OK** to close the dialog box. A new node is added subordinate to the parent menu item.
11. Right-click the new node and then click **Select This**. The system places a check mark in the square beside the menu name to denote that it is selected.
12. Click **OK** to make the window.

#### **4.8.12. How to Make a Popup Window**

You may elect to display data from a table in a window that "pops-up" when you click a hyperlink in a column or retrieve inventory information via a map. This is accomplished as follows:

1. Display the Tables window.
2. Locate the table for which you will create a popup window. Right-click the record showing the table and then **Make Window**. The application displays a new window. The default selection in the dialog box is to create a data window, not a popup window.
3. Click the radio button for the second option (Popup).
4. Click **Next**. The application displays a second window.
5. In the second window, check that the window title is what you want. If necessary, modify the title as desired. Also, write down the popup window ID as this will be needed when configuring the hyperlink or map theme.
6. If the data in the window should not be modified, clear the **Editable?** check box.

7. Enter the retrieval argument and, if necessary, the Where Clause that determines what data appears in the popup window.
8. Click **OK** to make the window.

Once the popup window is available, you need to configure the column that contains the hyperlink that will display the popup window. This is accomplished as follows:

1. Invoke Design Mode by clicking the Design Mode check box in the left gutter.
2. Display the window that contains the column that will have the hyperlink.
3. Right-click the pane containing the column and then click **Change Control Properties**. The application displays the User Control Properties dialog box.
4. Click the Data Window tab.
5. In the lower pane of the tab, locate the column that will contain the hyperlink.
6. In the record showing the column, click the **Is Link** check box.
7. In the Target Window ID column, enter the popup window ID that you wrote down in the previous set of instructions.
8. In the Retrieval Args column, enter the argument for the data to be retrieved and passed to the popup window.
9. Click **OK** to close the dialog box.
10. Click the  icon. The application asks if you wish to save the new layout.
11. Click **OK** to save the new layout.
12. Exit Design Mode by clicking the Design Mode check box in the left gutter.

#### **4.8.13. How to Make an "As-of Date" Window**

##### **NOTE**

An "as-of date" window displays data from a table at a specific time in the past. You must select the Keep Change History? column for the table in the Tables tab or no data will be displayed in the "as-of date" window that you create with this procedure.

To create the window that will display the data in a table, follow these steps:

1. Display the Tables window.
2. Locate the table for which you will create a window. Right-click the record showing the table and then click **Make Window**. The application displays a new window.
3. For the first option (Main), click the down arrow to display the list of types of windows and then click the "As-of Date" Window entry.
4. Click **Next**. The application displays a second new window.
5. In the second new window, check that the window title is what you want. If necessary, modify the title as desired.
6. If the data in the window may be modified, click the **Editable?** check box to select it.
7. In the lower part of the dialog box, expand the menu hierarchy to locate the parent menu item under which the menu item to display the window will be placed.
8. Right-click the parent menu item and then click **Add Branch**. The application displays a new dialog box so you may enter the name of the menu item.

9. In the dialog box, in the New Menu ID field enter the internal ID for the menu item that will open the window. (This ID should be lower case and not contain spaces.) Also, in the New Menu Name field, enter the name of the menu item that will open the window.
10. Click **OK** to close the dialog box. A new node is added subordinate to the parent menu item.
11. Right-click the new node and then click **Select This**. The system places a check mark in the square beside the menu name to denote that it is selected.
12. Click **OK** to make the window.

#### **4.8.14. How to Modify Column Definitions**

To modify the SQL statement or any other information in a column, follow these steps:

1. Open the Tables window.
2. In the Tables pane, click the record for the table for which the columns need to be modified.
3. Click the Columns tab.
4. In the Columns tab, make the desired modifications to the calculation SQL.
5. Click .
6. Open the window that shows the data from the table.
7. Right-click and then click **Update Target Table**. The system then re-calculates any calculated columns using the new column definitions.

#### **4.8.15. How to Change the Order of Columns**

##### **NOTE**

If a table is created from a DW\_ procedure, the order of the columns cannot be modified. Also, the order of the constituent columns of the LOC\_IDENT column (see page 258) is set in the LRM window.

The columns in the data window for a data table appear in the order specified in the Order column of the Data Window tab of the User Control Properties dialog box. Number 1 signifies the leftmost column and then increments to the right. To change the order in which the columns appear in a window, follow these steps:

1. Invoke Design Mode by clicking the Design Mode check box in the left gutter.
2. Display the Tables window.
3. Right-click the record for the table that contains the columns to be re-ordered and then click **Change Control Properties**. The application displays the User Control Properties dialog box.
4. Click the Data Window tab.
5. In the lower pane of the tab, modify the values in the Order column to indicate the desired order.
6. Click **OK** to close the dialog box.
7. Click the  icon. The application asks if you wish to save the new layout.
8. Click **OK** to save the new layout.

9. Exit Design Mode by clicking the Design Mode check box in the left gutter.

#### 4.8.16. How to Update When Source Data Changes

If the source data changes (for example, if survey data for a section data table is re-imported), the re-calculation of the values based on this data (for example, the indices based on the survey data) is not performed automatically. Instead, you must perform the re-calculation manually as described below:

1. Navigate to the window that shows the data from the data section table.
2. Right-click and then click **Update Target Table**. The system displays a dialog box so you may select what calculated columns you wish to update.
3. Click the check box for each calculated column you wish to re-calculate and then click **OK**. The system re-calculates the values of the specified columns.

### 4.9. Text Resources

#### NOTE

Consult with an AgileAssets staff member before changing any text messages.

The Text Resources window lists all text messages that may be generated by the application. Variables in a message are preceded by a percent sign (%). You may alter a message to better suit the situation in which it appears, but remember that a message may appear in more than one situation. Therefore, avoid making the message so specific that it is unlikely to work in multiple situations. An example of the Text Resources window is shown below.

(System > Utilities > Text Resources)

System > Utilities > Text Resources					
		73 pages (1227 rows)			
TEXT RESOURCE ID	TEXT RESOURCE	Comments	Att.	User Update	Date Update
953	Transaction for %1 cannot be completed because conversion from %2 units is no				
1217	%1 is duplicate equipment name				
1214	%1 is invalid equipment name				
1215	%1 is invalid stock ID				
1216	Not enough material in stock				
1256	"%1" between %2 and %3				
1167	"Budget constraint of %1 is less than Master Work Program cost of %2"				
249	%1 .....				
1036	%1 - cannot unambiguously translate into basic location				
459	%1 - Invalid 'from' location		LEONID	12/18/2006	
460	%1 - Invalid 'to' location		LEONID	12/18/2006	
818	%1 - is invalid file or directory				
462	%1 - Offset 'From' is greater than offset 'To'				

### 4.10. User Names and Access

This window allows System Administrators (or others with the proper security access) to:

- Add or remove users.
- Assign the administrative unit(s) that a user may use when logging on.
- Assign one or more security profiles to a user.
- Determine whether the user may have design privileges.
- Activate or disable the user's account.

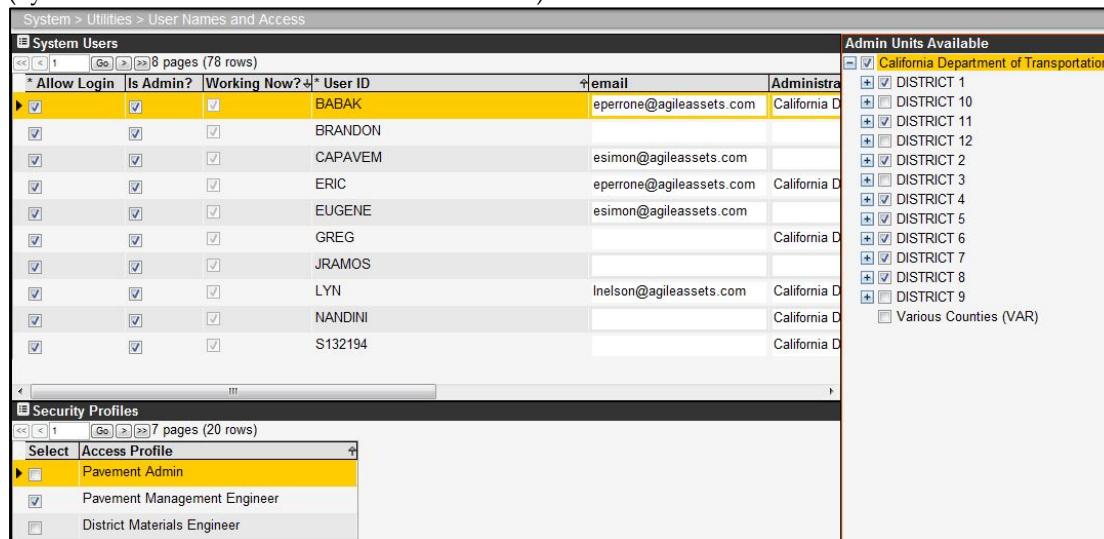
**NOTE**

**Do not delete the user called SYSTEM.** This user must be present to allow scheduled system jobs to run. (If this user is not present, you may add it to the window as any other user is added except that log-in does not necessarily need to be allowed.)

If your implementation utilizes the Forgotten Password feature, it is vital that an accurate email address be included in each user's record. Without an email, this feature will not work.

#### **4.10.1. User Names and Access Window Description**

(System > Utilities > User Names and Access)



The screenshot shows the 'User Names and Access' window with three main panes. The left pane displays 'System Users' with columns for Allow Login, Is Admin?, Working Now?, User ID, email, and Administra. The middle pane shows 'Security Profiles' with options like Select, Access Profile, Pavement Admin, Pavement Management Engineer, and District Materials Engineer. The right pane lists 'Admin Units Available' including California Department of Transportation, DISTRICT 1 through DISTRICT 9, and Various Counties (VAR).

The User Names and Access window contains three panes: the list of System Users on the left, the available Security Profiles below, and a tree view of the Available Admin. Units on the right. Before users can be configured in this window, the following codes and settings must be defined:

- Administrative units (System module).
- Access profiles in the Access Level Settings window.

##### ***System Users Pane***

**NOTE**

**Do not delete the user called SYSTEM.** This user must be present to allow scheduled system jobs to run. (If this user is not present, you may add it to the window as any other user is added except that log-in does not necessarily need to be allowed.)

If your implementation utilizes the Forgotten Password feature, it is vital that an accurate email address be included in each user's record. Without an email, this feature will not work.

In the Users pane, users may be added or deleted, and some of their attributes may be edited. Columns that may be edited are: access profile, the user's administrative unit (although this column has no bearing on what the user may select when logging on), his or her email address, and whether the user is allowed to log on (that is, whether the user name is "active").

When a new user is inserted, the user's password is also entered in upper case as an additional piece of information that is not viewable in the pane.

A check mark in the Allow Login check box indicates that the user is allowed to log on. If a user enters an incorrect password the number of consecutive times set in the System Parameters window, the system disables that user's access and clears the Allow Login check box. The System Administrator can restore the user's access by clicking the check box and saving the change. The check box may also be used to suspend a user's access privileges.

A check mark in the Is Admin? check box indicates that the user has design privileges. This means that the user is allowed to adjust a window's properties (column widths, sizes and position of panes, labels, titles, etc.). Note that the application must be placed in Design Mode for the user with design privileges to modify the application. The user with design privileges places the application in design mode by clicking the Design Mode check box and then opening the window to be modified. The application will remain in design mode until the check box is cleared or the user logs off. See page 249 for more information on adjusting window layouts.

A check mark in the Make Change Password? check box indicates that the user must change his or her password when they next attempt to log on. After the user changes his or her password, this check box is automatically cleared.

A check mark in the Working Now? column indicates that the user is currently logged on to the system.

When you right-click this pane, a shortcut menu is displayed with the following commands in addition to the common commands:

- **Insert Like** – This command provides an insert and copy capability. It creates a new user by copying all of the information from the currently highlighted user into a new record with the user name and password still to be completed. (This is different from the **Insert** command because that command inserts a new, blank row instead of copying information into the new row.) See page 156 for an example of using this command.
- **Reset Password** – This command allows a System Administrator to set a new password for a user. Note that this command does not require the entry of the old password. This allows the System Administrator to assign a new password to a user who has forgotten his or her password. See page 158 for instructions on resetting a password.

#### ***Admin. Units Available Pane***

When a user logs on, he or she must select an administrative unit that serves as his or her "home" unit. The administrative units that are available for the user to select are set in this pane.

In the Admin. Units Available pane, you can view and/or change the administrative units to which the currently selected user has access for logging on. An empty square next to the administrative unit name indicates that it is not selected. On the other hand, a check mark in the square indicates that the administrative unit is selected. Typically, the initial configuration would allow each user access to his or her own administrative unit and the units that he or she manages.

#### **NOTE**

The administrative unit selected in the Users pane is only informational and must also be selected in this pane to actually be an available administrative unit.

## Security Profiles Pane

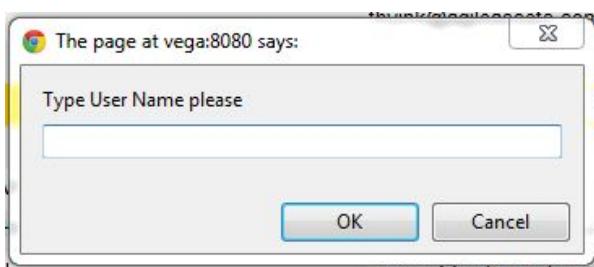
The Security Profiles pane shows all available security profiles as configured in the Access Level Settings window (see page 111). A check mark in the check box indicates that the security profile is assigned to the user selected in the System Users pane. (More than one profile may be selected.)

The security profiles selected in this pane appear in a drop-down list that is displayed during log-on (in the same window where he or she selects an administrative unit).

### 4.10.2. How to Add a New User Without Copying

For a new user that is unlike an existing user, you should use the **Insert** command. The following steps show how to use this command:

1. In the left pane, right-click, and then click **Insert**. The system displays the window shown below.



2. Enter the new user's name and then click the **OK** button. A message window will notify you that the default user password is set to the user's name and that the user should change the password when he or she first logs on. (The user changes his or her password as directed by the system, which automatically displays a dialog box to change the password.)

Note: The default password is the user name in upper case (passwords are case-sensitive).

3. Click **OK** to close the message window. The system closes the window and adds a record to the User pane.
4. Enter the person's email address in the Email column.
5. In the Available Admin Units pane, select the administrative unit(s) to which the new user is assigned and which he or she may select when logging on. To accomplish this, right-click the desired administrative unit and then click **Select This**.

Note: A **Select Branches** command is also available so you may assign the user all the sub-administrative units beneath an administrative unit header. A **Select All** command is also available if you wish to assign the user to all administrative units. (If you want to change a user's assignment, "de-select" versions of all these commands are also available.)

Note: This step is very important. If you do not assign an administrative unit to the new user, he or she cannot use the system because the user cannot select an administrative unit when logging on. Since it is possible to save the new user and close the window without assigning an administrative unit, and since the system will not warn you of this condition, double-check that these assignments are made before closing the window.

Note: Selecting an administrative unit in the Administrative Unit column of the User pane is not the same as selecting an administrative unit in the Available Admin Units

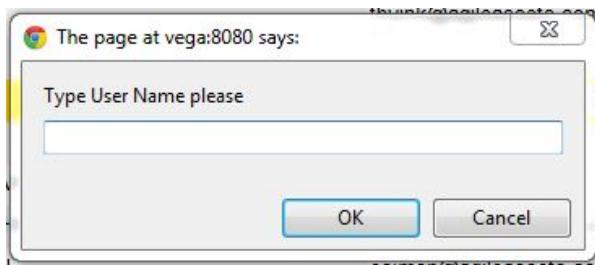
pane. The column in the User pane is for informational purposes only and is not used by the system.

6. In the Security Profiles pane, select each security profile to be assigned to the user by clicking the check box in the Select column.
7. Once all information is assigned and recorded, click . The new user may now access the system.

#### **4.10.3. How to Add a New User by Copying**

If a new user is very similar to an existing user, the quickest way to add the user is by using the **Insert Like** command. This command creates a new record and sets all parameters, except user name and password, for the new user to that of the selected user. To create a new user with this command:

1. In the left pane, right-click an existing user whose settings will serve as the basis for the new user and then click **Insert Like**. A new window will be displayed. The system displays the window shown below.



2. Enter the new user's name and then click the **OK** button. A message window will notify you that the default user password is set to the user's name and that the user should change the password when he or she first logs on. (The user changes his or her password as directed by the system, which automatically displays a dialog box to change the password.)

Note: The default password is the user name in upper case (passwords are case-sensitive).

3. Click **OK** to close the message window. The system closes the window and adds a record to the User pane.
4. Enter the person's email address in the Email column.
5. In the Admin. Units Available pane, check the selected administrative unit(s) to which the new user is assigned and which he or she may select when logging on. Edit as necessary.

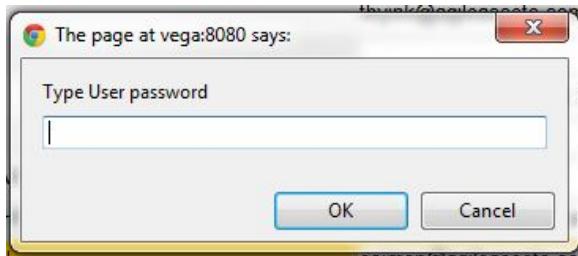
Note: Selecting an administrative unit in the Administrative Unit column of the User pane is not the same as selecting an administrative unit in the Admin. Units Available pane. The column in the User pane is for informational purposes only and is not used by the system.

6. In the Security Profiles pane, check the selected profile(s). Edit as necessary.
7. Once all information is assigned and recorded, click . The new user may now access the system.

#### **4.10.4. How to Reset a Password**

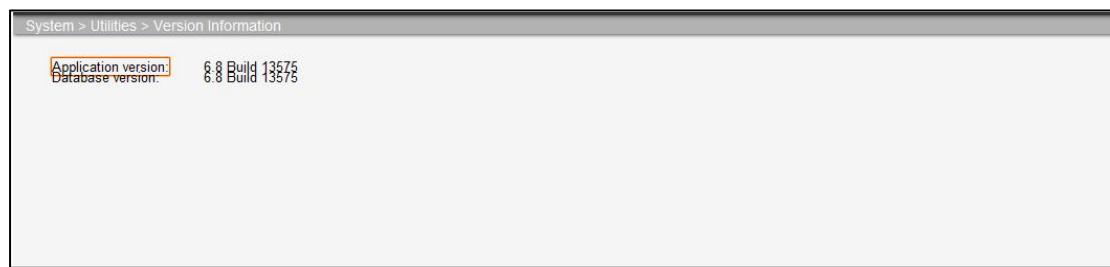
To reset a password, follow these steps:

1. In the System Users pane, right-click the user's name and then click **Reset Password**.  
The system displays the following dialog box.



2. Type the new password in the field. Remember that passwords are case-sensitive.
3. Click the **OK** button.
4. Click  . The new password is now in effect. When the user next tries to log in, he or she will need to use the new password.

#### **4.11. Version Information**



The Version Information window shows you the version of the system that you are currently using.

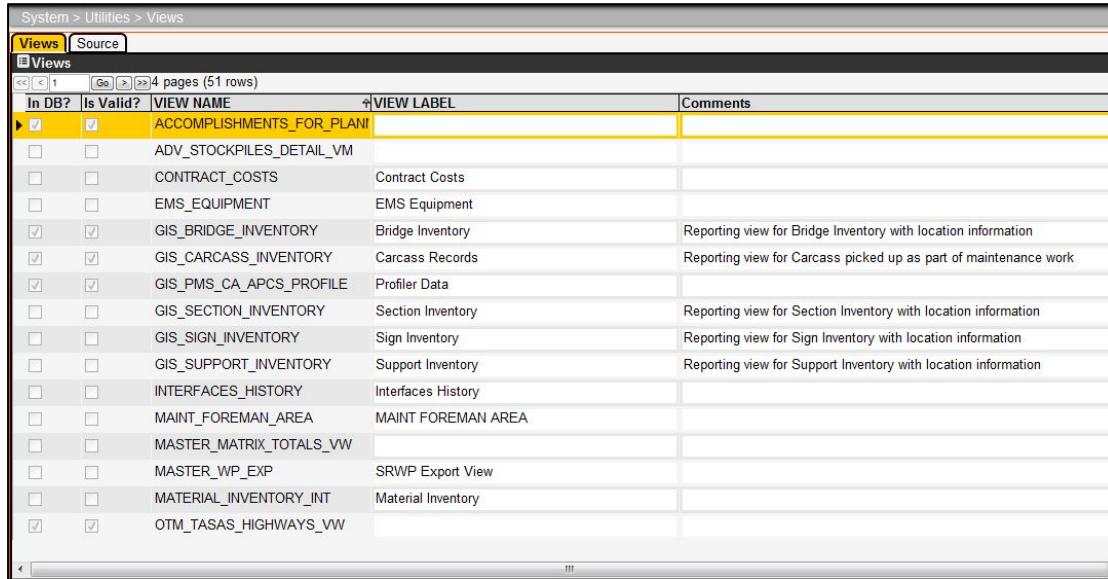
#### **4.12. Views**

The Views window (System > Utilities > Views) allows you to create and modify Oracle views, which can then be used within the system. From the Views window you may also create a window from which to review the contents of an Oracle view.

##### **4.12.1. Description of the Views Window**

The Views window contains two tabs: Views and Source. These tabs are described in more detail in the following sections.

## The Views Tab



In DB?	Is Valid?	VIEW NAME	VIEW LABEL	Comments
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	ACCOMPLISHMENTS_FOR_PLAN		
<input type="checkbox"/>	<input type="checkbox"/>	ADV_STOCKPILES_DETAIL_VW		
<input type="checkbox"/>	<input type="checkbox"/>	CONTRACT_COSTS	Contract Costs	
<input type="checkbox"/>	<input type="checkbox"/>	EMS_EQUIPMENT	EMS Equipment	
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	GIS_BRIDGE_INVENTORY	Bridge Inventory	Reporting view for Bridge Inventory with location information
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	GIS_CARCASS_INVENTORY	Carcass Records	Reporting view for Carcass picked up as part of maintenance work
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	GIS_PMS_CA_APCS_PROFILE	Profiler Data	
<input type="checkbox"/>	<input type="checkbox"/>	GIS_SECTION_INVENTORY	Section Inventory	Reporting view for Section Inventory with location information
<input type="checkbox"/>	<input type="checkbox"/>	GIS_SIGN_INVENTORY	Sign Inventory	Reporting view for Sign Inventory with location information
<input type="checkbox"/>	<input type="checkbox"/>	GIS_SUPPORT_INVENTORY	Support Inventory	Reporting view for Support Inventory with location information
<input type="checkbox"/>	<input type="checkbox"/>	INTERFACES_HISTORY	Interfaces History	
<input type="checkbox"/>	<input type="checkbox"/>	MAINT_FOREMAN_AREA	MAINT FOREMAN AREA	
<input type="checkbox"/>	<input type="checkbox"/>	MASTER_MATRIX_TOTALS_VW		
<input type="checkbox"/>	<input type="checkbox"/>	MASTER_WP_EXP	SRWP Export View	
<input type="checkbox"/>	<input type="checkbox"/>	MATERIAL_INVENTORY_INT	Material Inventory	
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	OTM_TASAS_HIGHWAYS_VW		

The Views tab lists all views created in the application. A check mark in the In DB? column indicates that the view is part of the underlying database (which means the **Apply Changes** command has been executed at least once). A check mark in the Is Valid? column indicates that the SQL is valid. These check boxes cannot be updated manually and, instead, are automatically updated after the **Apply Changes** command is executed.

This tab also provides a right-click shortcut menu. This menu contains the common commands along with the **Make Window** command. The **Make Window** command creates a window that displays the view's data; see page 161 for further information.

The shortcut menu also includes an **Insert** command. After selecting this command, the system displays a dialog box into which you type the view name. If the view exists, then its SELECT portion is copied into the Source tab. If the view does not exist, then you enter the syntax for the view's SELECT statement into the Source tab.

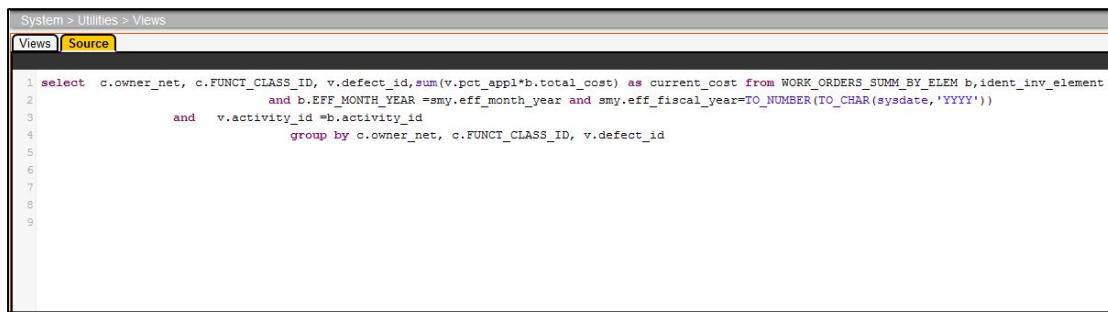
### NOTE

All columns in the SELECT statement must have column names/aliases because the system automatically builds the view-definition portion of the view statement from the SELECT statement in the Source tab. If any column names/aliases are missing, then an error will occur when you select the **Apply Changes** command.

The menu item that allows you to display the window for the view must be created (in the Menus window of the System module) before selecting the **Make Window** command.

Any additions or modifications to the Preparer's Comments column are sent to the Oracle system table that stores comments when you execute the **Apply Changes** command.

### The Source Tab



```
System > Utilities > Views
Views | Source

1 select c.owner_net, c.FUNCT_CLASS_ID, v.defect_id,sum(v.pct_appl*b.total_cost) as current_cost from WORK_ORDERS_SUMM_BY_ELEM b,ident_inv_element c
2           and b.EFF_MONTH_YEAR = smy.eff_month_year and smy.eff_fiscal_year=TO_NUMBER(TO_CHAR(sysdate,'YYYY'))
3           and v.activity_id = b.activity_id
4           group by c.owner_net, c.FUNCT_CLASS_ID, v.defect_id
5
6
7
8
9
```

For the view selected in the Views tab, this tab provides the SQL that generates the view.

This tab also provides a right-click shortcut menu with one command: **Apply Changes**. After modifying the SQL, this command applies the changes to the database.

#### NOTE

All columns in the SELECT statement must have column names/aliases because the system automatically builds the view-definition portion of the view statement from the SELECT statement in the Source tab. Furthermore, for existing views, the aliases must match the original view column names. If any column names/aliases are missing, then an error will occur when you select the **Apply Changes** command.

#### **4.12.2. How to Create a View**

To create a view, follow these steps:

1. Display the Views window.
2. In the Views tab, right-click and then click **Insert**. The application displays a dialog box so you may enter the internal name of the view.
3. Type the name of the view in the dialog box. The convention for names is all upper case, no spaces.
4. Click **OK**. The application closes the dialog box and inserts a new record in the Views tab. Also, if the view exists, then its SELECT portion is copied into the Source tab. (If the view does not exist, you will need to enter the syntax for the view's SELECT statement into the Source tab.)
5. In the new record, optionally enter a display name for the view and any comments.
6. Click the Source tab.
7. In the Source tab, if necessary enter the SQL that will produce the view. (If the view exists, this data may already be present.)

Note: All columns in the SELECT statement must have column names/aliases because the system automatically builds the view-definition portion of the view statement from the SELECT statement in the Source tab. If any column names/aliases are missing, then an error will occur when you select the **Apply Changes** command.

8. Right-click the tab and then click **Apply Changes**. The application checks the SQL and, if valid, adds the view to the database. The view is now created.

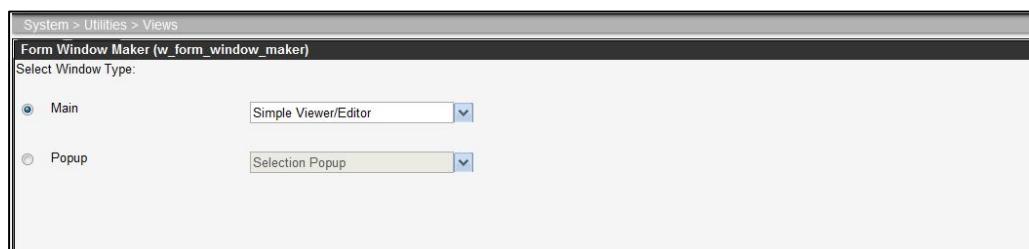
#### **4.12.3. How to Make a View Window**

The first task in creating a window to display data from a view is to create the menu item that a user will select to display the window. This is accomplished as follows:

1. Display the Menus window.
2. In the upper left pane, find the record for the module in which the menu item will appear and then click the record to select it. After selecting the module, the application displays the menu hierarchy for the module in the lower left pane.
3. In the lower left pane, if necessary expand the hierarchy to display the parent menu item under which the new menu item will be placed.
4. Right-click the parent menu item and then click **Add Branch** in the shortcut menu that is displayed. The application displays a dialog box so you may enter the name of the new menu item.
5. In the dialog box, type the name of the new menu item.
6. Click **OK**. The application closes the dialog box and adds a new node under the parent menu item as well as a new record in the table in the right pane.
7. Click the  icon to save the new menu item.

The menu item that will display the window now exists. You may now create the window that will display the data in a view as described in the following steps:

1. Display the Views window.
2. Locate the view for which you will create a window. Right-click the record showing the view and then **Make Window**. The application displays a dialog box similar to that shown below. The default selection in the dialog box is to create a data window, which is the goal of this example.



System > Utilities > Views  
Form Window Maker (w\_form\_window\_maker)  
Select Window Type:  
 Main Simple Viewer/Editor  
 Popup Selection Popup

3. Since the data table option is already selected, just click the **Next** button. The application displays a second dialog box.



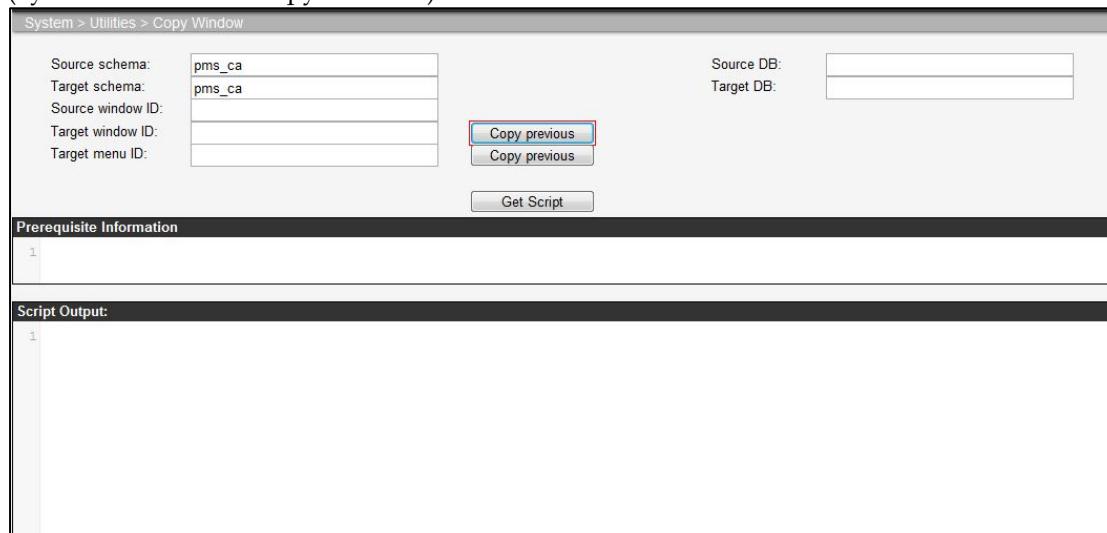
System > Utilities > Views  
Form Window Maker (w\_form\_window\_maker)  
Window Id: accomplishments\_for\_planning\_data Window Title: ACCOMPLISHMENTS\_FOR\_PLANNING  
 Editable?  
Select Menu  
+ Roadway  
+ System

4. In the second dialog box, check that the window title is what you want. If necessary, modify the title as desired.
5. If the data in the window may be modified, click the **Editable?** check box to select it.

6. In the lower part of the dialog box, expand the menu hierarchy to locate the menu item that a user will select to display the window. Click the check box beside the name of the menu item to select it.
7. Click **OK** to make the window.

### 4.13. Copy Window

(System > Utilities > Copy Window)



The screenshot shows the 'Copy Window' dialog box. It has three tabs: 'System > Utilities > Copy Window', 'Prerequisite Information', and 'Script Output'. The 'System > Utilities > Copy Window' tab is active. In this tab, there are several input fields: 'Source schema' (pms\_ca), 'Target schema' (pms\_ca), 'Source window ID', 'Target window ID', 'Target menu ID', 'Source DB', and 'Target DB'. Below these fields are two buttons: 'Copy previous' (which is highlighted with a red border) and 'Copy previous'. At the bottom of this tab is a 'Get Script' button. The 'Prerequisite Information' and 'Script Output' tabs are also visible but inactive.

The Copy Window window (System > Utilities > Copy Window) allows you to generate the scripts necessary to copy a window from one schema to another (or from one menu to a new menu). The system creates an Oracle SQL script that creates (or re-creates) a specific window according to the defined parameters.

To copy a window, follow these steps:

1. Open the Copy Window window.
2. Define the necessary parameters (Source Schema, Target Schema, Source Window ID, Target Window ID, and Target Menu ID).
3. If necessary, define the Source DB and Target DB (by default, the system will generate a script using the same Database Sources).
4. Click the **Get Script** button. The system generates the script.

Once the script is generated, you can copy the script and execute it in a different instance using a database editing tool.

**NOTE**

You should review the generated script before executing it. This operation only should be executed by qualified personnel.

### 4.14. Administrative Units

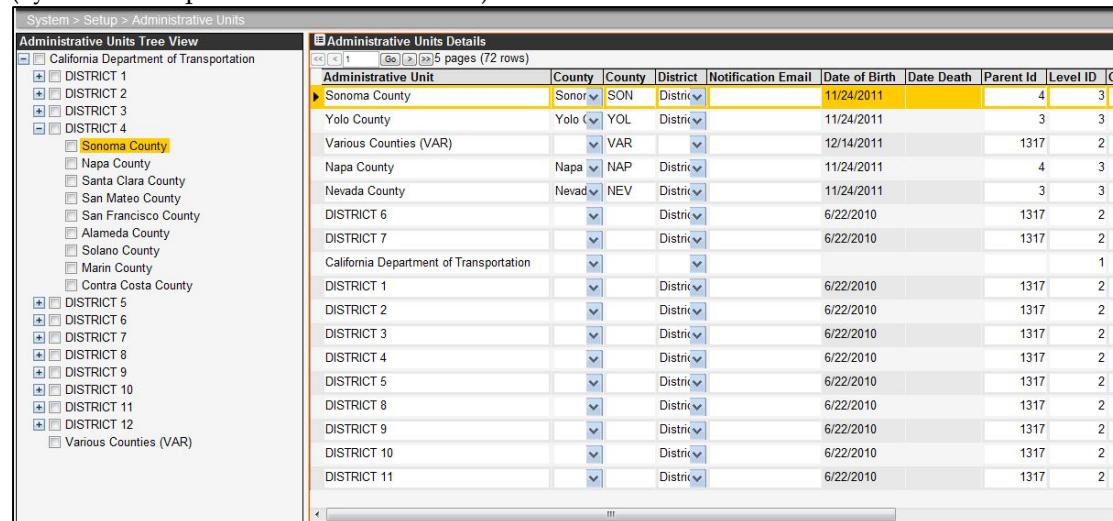
The system allows an agency to view and define their administrative structure using a simple tree view of the organization. The Setup Administrative Units window allows the agency to define the relationship between central office, district office, maintenance yards, and so forth. The hierarchy also controls certain user functions.

**NOTE**

If you wish to share labor resources with another administrative unit, use the Setup Resource Sharing window. This window provides commands to give permission to certain administrative units so their employees, equipment, and materials may be used on work orders created by a different administrative unit.

#### **4.14.1. Description of the Admin. Units Window**

(System > Setup > Administrative Units)



The screenshot shows the 'Administrative Units' window with two panes. The left pane is a 'Administrative Units Tree View' showing a hierarchical structure of administrative units, including districts and counties. The right pane is a 'Administrative Units Details' table with columns for Administrative Unit, County, District, Notification Email, Date of Birth, Date Death, Parent Id, Level Id, and Order. The table lists various administrative units like Sonoma County, Yolo County, and Nevada County, along with their respective details.

The Administrative Units window is comprised of two panes that give different views of the same information.

The tree view (on the left side) shows the hierarchy in graphical form.

The table view (on the right) gives the information in tabular form and also includes the "birth" (and "death," if applicable) dates of each administrative unit.

**NOTE**

The Email Address column contains the address of the person to whom notices of low stock are sent. This is separate from the email address used for the forgotten password feature (which may or may not be the same).

When you right-click either pane, the following special command is available:

- **Add Branch** – This command adds a new, subordinate node to the tree view and a corresponding record in the table view. See the following section for the procedure on how to add an administrative unit.

#### **4.14.2. How to Add a New Admin. Unit**

To add a new administrative unit to the hierarchy, follow these steps:

1. In either view, right-click the administrative unit under which the new unit will be added and then click **Add Branch**. The system expands the tree view and adds the new node as subordinate to the administrative unit to which you pointed. A new record is also added to the table.
2. In the new record of the table, double-click New in the Administrative Unit column. The word New is highlighted.

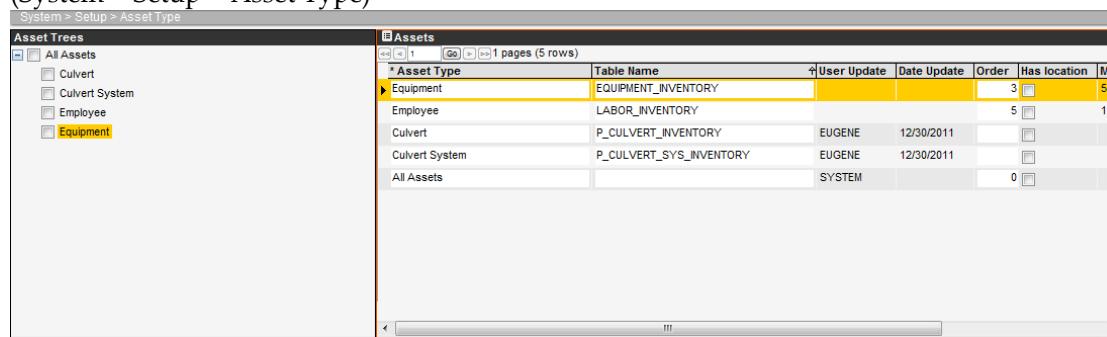
3. Type the name of the new administrative unit.
4. Click  to save the new administrative unit.

## 4.15. Asset Type

The Setup Asset Types window allows you to create and maintain a hierarchy of all asset types utilized by your agency. It also allows you to create the name of the inventory and class codes tables that will be associated with the asset type (if any) and what event(s) will trigger periodic maintenance (PM) for the asset type. (Each asset type has its own PM triggers, and these may differ from asset type to asset type.)

### 4.15.1. Description of the Asset Type Window

(System > Setup > Asset Type)



Assets						
* Asset Type	Table Name	User Update	Date Update	Order	Has location	Mo
Equipment	EQUIPMENT_INVENTORY			3	<input checked="" type="checkbox"/>	5
Employee	LABOR_INVENTORY			5	<input checked="" type="checkbox"/>	1
Culvert	P_CULVERT_INVENTORY	EUGENE	12/30/2011		<input type="checkbox"/>	
Culvert System	P_CULVERT_SYS_INVENTORY	EUGENE	12/30/2011		<input type="checkbox"/>	
All Assets	SYSTEM			0	<input type="checkbox"/>	

The Asset Type window contains two panes: on the left, a tree view that shows the hierarchical arrangement of the asset types; on the right, a table view that provides additional information about each asset type. The tree view is expanded by clicking a plus sign (+), or contracted by clicking a minus sign (-).

When you select an asset type in the tree view pane, the record corresponding to that node is highlighted and displayed in the table on the right. The converse is also true — selecting a record in the table also selects a node in the tree view.

The Location Applicable check box determines whether the inventory table requires a location to be specified for each item in inventory. If the check box contains a check mark, a location is required.

When you right-click a node in the tree view pane, a shortcut menu is displayed. This menu contains the following command in addition to the common commands:

- **Add Branch** — This command adds a new, subordinate node to the tree view and a corresponding record in the table view. See the following section for the procedure on how to add an asset type.

When you right-click a record in the table view pane, a shortcut menu is displayed. This menu contains the following commands in addition to the common commands:

- **Add Branch** — This command adds a new, subordinate node to the tree view and a corresponding record in the table view. See the following section for the procedure on how to add an asset type.
- **Edit PM Trigger** — This command allows you to select what events will be used to trigger periodic maintenance (PM) activities for this asset type. The PM triggers entered here appear in the drop-down list in the PM by Inventory and PM by Class Codes windows. See page 166 for more information on what triggers are available and how they are associated with the asset type.

**NOTE**

Do not make the MODULE\_ID column editable because changing the module in this window will not refresh the MODULE\_OWNER\_SHARE\_ELEM view (it must be refreshed via a system job). Although the module that is used to manage an asset type is an attribute of the asset type, if you want to change the module under which an asset type is managed, this should be done in the Setup Modules window.

#### **4.15.2. How to Create an Asset Type**

You must perform two tasks to create an asset type. You must first create the asset type in the Asset Type window and then include this asset type in a particular module in the Module window.

To accomplish the first task (creating an asset type), follow these steps:

1. Open the Setup > Asset Type window in the System module.
2. In the Asset Hierarchy pane on the left, expand the hierarchy to find the parent node under which the asset type will appear.
3. Right-click the parent node and then click **Add Branch**. The application adds a new child node called New to both the hierarchy and the table in the right pane.
4. In the table in the right pane, highlight the word New and then type the name of the new asset type.
5. If particular items of this asset type will be maintained in an inventory window, click in the Table Name column and type the name of the inventory table. Note: The table name cannot exceed 23 characters and must end in \_INVENTORY (in other words, the unique part of the name must be between 1 and 13 characters in length).
6. Click the Location Applicable check box to require that each item in inventory have a location.
7. Click  . The new asset type is now created.

Note: If you entered an inventory table name, the application creates records in the Tables window for the inventory and class codes tables. (The class codes table has the same prefix as the inventory table name and ends with \_CLASS\_CODE.) These tables do not yet exist in the database; they will only exist in the database once the **Apply Changes** command is executed for each table in the Tables window. You may also use the Tables window to create the windows to display the data in the tables along with Status, PM by Inventory, and PM by Class Code windows if desired. See page 128 for more information on the Tables window.

The asset type (and any associated inventory and class codes tables to support the asset type) now exists in the database. The next task is to select the module in which the asset type will appear. This is accomplished as follows:

1. Navigate to the Setup > Module window in the System module.
2. In the Modules pane on the left, click the module in which the asset type will appear. The application highlights the record to show that it is selected.
3. In the Selected Asset Types pane on the right, right-click and then click **Edit Selection**. The application displays the Asset Selection dialog box.
4. In the dialog box, locate the asset type that you just created in the Asset Type window and click the square beside the name of the asset type. The application places a check mark in the square to denote that it is selected.

5. Click **OK**. The application closes the dialog box and displays the selected asset type in the Selected Asset Types pane.
6. Click . The asset type is now associated with a module.

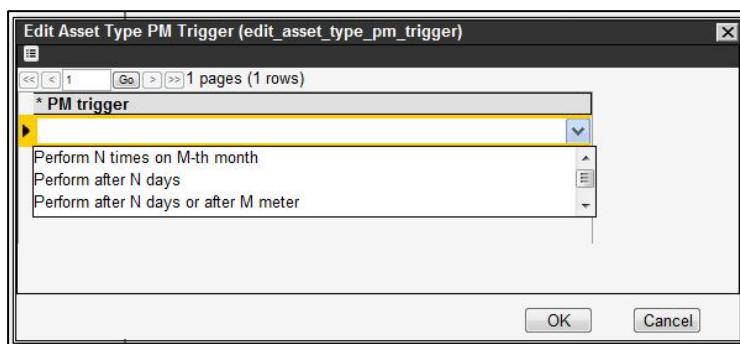
#### **4.15.3. How to Create PM Triggers**

The following criteria may be associated with an asset type to trigger a PM activity:

- Perform N times on the M-th month — The activity is performed a user-defined number of times in a user-specified month.
- Perform after N days — The activity is performed after a user-defined number of days from the start date of the last PM activity.
- Perform after N days or M meter — The activity is performed either after a user-defined number of days from the start date of the last PM activity or when the difference between the meter reading when the PM activity was last performed and the current meter reading reaches a user-defined value (whichever occurs first).
- Perform after N Meter — The activity is performed when the difference between the meter reading when the PM activity was last performed and the current meter reading reaches a user-defined value.
- Perform after defect N score is below M or after K days — This activity is performed either after a user-defined number of days have elapsed since the start date of the last occurrence of the activity or the defect score is below a user-defined threshold.
- Perform on defined month — The activity is performed once in the user-specified month.

To associate one or more PM triggers with an asset type, follow these steps:

1. Display the Asset Types window.
2. In the right pane, locate the record for the asset type with which you will associate the PM triggers.
3. Right-click the record and then click **Edit PM Triggers** from the shortcut menu that is displayed. The application displays a dialog box so you may enter the desired triggers.
4. In the dialog box, right-click and then click **Insert** from the shortcut menu that is displayed. The application inserts a record in the dialog box.
5. In the new record, click the down-arrow to display the list of PM triggers and then click the desired PM trigger as shown below.



6. Repeat steps 4 and 5 for any additional PM triggers that will be associated with this asset type.

7. When all desired PM triggers are shown in the dialog box, close the dialog box by clicking the **OK** button.
8. Click the  icon to save the new information. The selected PM triggers will now be available in the trigger drop-down lists in the PM by Inventory and PM by Class Codes windows for this particular asset type.

## 4.16. Setup File System

(System > Setup > Setup File System)

System > Setup > Setup File System					
File System					
* File System	File System Root	Comments	Att.	User Update	Date Update
► TABLE_IMPORT_ROOT	/mnt/import_mnt		LEN		6/24/2011
LOCAL_IMPORT_ROOT	c:\BufTmp				

The Import feature of the Tables window allows multiple root folders for the selection of data files to be imported. These root folders are configured in this window, the Setup File System window. This window shows the name of the root folder and its description.

Once multiple folders are entered in this window, then the File Browser dialog box launched from the Import Config dialog box shows a field called File System. (See page 133 for more information on the Import Config dialog box.) This field contains a drop-down list, with the contents of the list being the names of the root folders configured in this window.

## 4.17. Module

The Module window (System > Tools > Module) displays the modules included in the system and the asset types assigned to each module. It shows the name of the module as displayed on your computer, the order in which the modules appear as tabs at the top of the window, and the type of assets with which the module is concerned.

System > Tools > Module			
Modules			
Selected Asset Types			
Module	Network Identifier	Order	
► Roadway		1	40
System		1	90
Asset Type			
Culvert			
Culvert System			

To associate an asset type with a module, select the desired module in the left Modules pane. Then, in the right Select Asset Types pane, right-click and then click **Edit Selection** in the shortcut menu. This command displays a dialog box that shows all available asset types and their arrangement. Select an asset type by clicking the square next to the name of the asset type. The application places a check mark in the square to denote that it is selected. Alternately, you may right-click and use one of the selection commands to select multiple asset types. When all asset types are selected, click **OK** to close the dialog box and display the selected asset types in the Setup Module window.

## 4.18. Units

(System > Setup > Units)

System > Setup > Units					Unit Conversions	
Measurement Unit	External Unit ID	Comments	User Update	Date Upd	Measurement Unit	Conversion Factor
Acre	ACRE	acre			Site	
Bag	BAG	bag			Skein	
Bale	BALE	bale			Sleeve	
Ball	BALL	ball			Spool	
Band	BAND	band			Square Foot	43560 17844000001
Bi-Annual	BIAN	bian			Square Inch	
Bi-Monthly	BIMO	bimo			Square Yard	4840.019827
Bi-weekly	BIWK	biwk			Tank	
Block	BLCK	blk			Test	
Bundle	BNDL	bndl			Thousand	
Book	BOOK	book			Thousand Foot	
Box	BOX	box			Thousand Gallon	
Brick	BRCK	brck			Ton	
Bottle	BTL	btl			Ton Mile	
Bushel	BU	bu			Tray	
Bulk	BULK	bulk			Trip	
Hundred	C	c			Troy Ounce	

The Setup Units window is used to define the units of measure for all quantities used throughout the system. Once the units are defined in this window, they can be assigned to defects, activities, material stock items, etc. In addition, conversion factors may be configured to translate one unit to another.

The Setup Units window provides the following information:

- In the left pane, the Unit Name column lists all units that are defined for the system. You may modify the data in this column or insert new rows for new units. The text that appears in this column is what appears in the unit drop-down lists found throughout the system. **Note: Once a unit is added to the system, it cannot be deleted if it is referenced anywhere in the system.**
- In the left pane, the External Unit ID column provides a cross-reference from the units used in the application to units used in other, external applications with which the Agile application interfaces (such as an accounting system). **Note: Each unit code in this column must be unique.** If duplicate unit codes exist, the internal interfaces will fail when the unit is sent from the external application to the Agile application.
- The right pane lists units to which the selected unit in the left pane may be converted. The conversion factor to convert from the unit in the left pane to the unit in the right pane is shown in the Conv Factor column. For example, if TONS is selected in the left pane and LBS is selected in the right pane, then 2000 would be entered in the Conv Factor column.

## 4.19. Setup LRM

### CAUTION

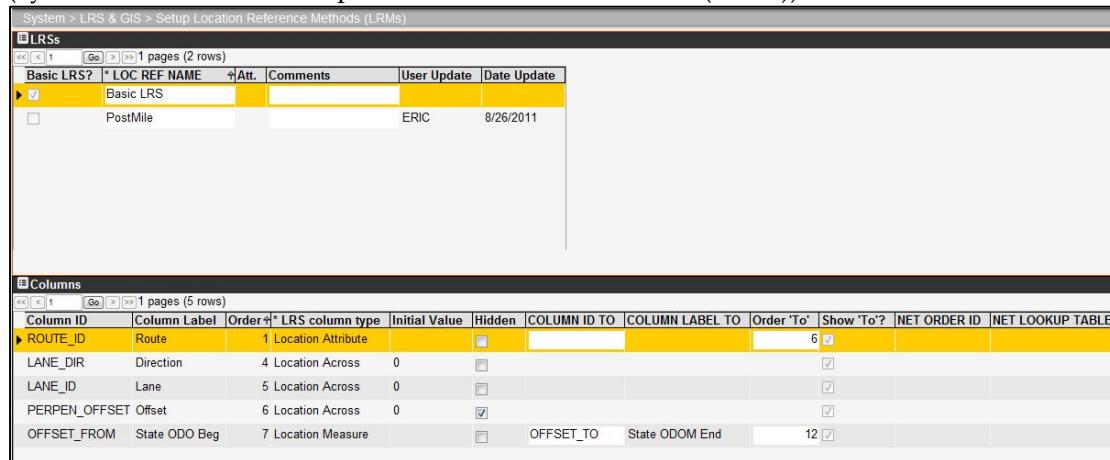
Do not modify the data in this window without first consulting with AgileAssets. Failure to do this may result in the loss of data from your system.

The Setup LRM [Linear Reference Methods] window allows you to define the Base Location reference system as well as any alternate reference systems that are associated with the Base

Location reference system. These reference systems are shown in the Location Reference drop-down list that is below the icons on the left side of the window.

#### 4.19.1. Description of the Setup LRM Window

(System > LRS & GIS > Setup Location Reference Methods (LRMs))



The screenshot shows the 'System > LRS & GIS > Setup Location Reference Methods (LRMs)' window. It has two main panes:

- LRSs Pane:** Shows a list of location reference systems. One row is selected, showing 'Basic LRS?' checked, 'LOC REF NAME' as 'Basic LRS', 'Comments' as 'PostMile', 'User Update' as 'ERIC', and 'Date Update' as '8/26/2011'.
- Columns Pane:** Shows a table of columns for the selected LRS. The columns are: Column ID, Column Label, Order, LRS column type, Initial Value, Hidden, COLUMN ID TO, COLUMN LABEL TO, Order 'To', Show 'To?', NET ORDER ID, and NET LOOKUP TABLE. The data includes:

Column ID	Column Label	Order	LRS column type	Initial Value	Hidden	COLUMN ID TO	COLUMN LABEL TO	Order 'To'	Show 'To?'	NET ORDER ID	NET LOOKUP TABLE
ROUTE_ID	Route	1	Location Attribute					6			
LANE_DIR	Direction	4	Location Across	0							
LANE_ID	Lane	5	Location Across	0							
PERPEN_OFFSET	Offset	6	Location Across	0							
OFFSET_FROM	State ODO Beg	7	Location Measure			OFFSET_TO	State ODOM End			12	

##### **Description of the LRSs Pane**

The upper, LRSs [Linear Reference Systems] pane shows all reference systems defined for your system. A check mark in the Basic LRS? column identifies the reference system that is the Base Location reference system. All other reference systems are alternates of this system.

The displayed name of each reference system is shown in the Loc Ref Name column, and is what is displayed in various parts of the system. It is also what is displayed in the Location Reference field that appears in the left gutter.

When you right-click an alternate reference system in the upper pane, the following special command is displayed in a shortcut menu:

- **Make Window** – This command displays a new window that shows the tree view of the menu items of the system. In the new window, click the parent menu item into which a menu item for the window that will show the cross-reference table between the Basic Reference System (BRS) and the alternate systems that you right-clicked.

After selecting the parent menu item, click **OK** to close the window. Then log off and log back on, and the menu item will be in the designated place, with the name of the menu item being what is entered in the Loc Ref Name column. Open the new window to see the cross-reference table. The BRS is shown on the left and the alternate system is on the right.

##### **Description of the Columns Pane**

##### **CAUTION**

The values set in this pane for the base LRM should never be changed without first consulting AgileAssets.

The Columns pane shows the columns that define the reference system selected in the upper pane. The following columns appear in this pane:

- Column ID – This column shows the internal column ID used in the system.
- Column Label – This column shows the column heading label that appears for the column throughout the system.

- Order — This column indicates the order (left to right) in which the columns are seen in data windows throughout the system.
- LRS Column Type — This column indicates whether the column is the "location measure" column. In defining an alternate reference system, one and only one column must be set as the "location measure"; all other columns must be set as "location attribute".
- Initial Value — If a value appears in this column, then the value will be automatically set to any record that is location-referenced when it is inserted into the system.
- Hidden — When this check box is selected, the column will not be displayed.
- Column ID "To" — This column shows the internal column ID used in the system. The contents of this field are not used if the Show "To" check box is clear.
- Column Label "To" — This column shows the column heading label that appears for this column throughout the system. The contents of this field are not used if the Show "To" check box is clear.
- Show "To"? — If the check box in this column is checked, then this LRM field will be shown twice in any location-referenced window when an alternate LRM is selected. It is shown once for the "from" point and then again for the "to" point. If unchecked, then the LRM field appears once and applies to both "from" and "to" points. Note: This column must be checked for the column designated "location measure."
- Net Order ID — This column indicates the order (left to right) in which the columns are seen in data windows throughout the system. Note: "To" columns are always situated to the right of "from" columns regardless of what is specified here.
- Net Order Table — This column indicates the order (left to right) in which the columns are seen in data windows throughout the system

#### **4.19.2. Alternate LRM Windows**

For each alternate reference system defined in the Setup Reference Location Methods window, the application creates a window that allows you to create and maintain the sectioning of your road network under the alternate reference system.

A row in this window describes a length of road in the alternate LRM and also cross-references that length of road to its base LRM description. This window contains two sets of columns:

- The set of columns that define location in the base LRM. This is the set of columns shown in the bottom pane of the Setup Location Referencing Methods window when the base LRM is selected in its top pane.
- The set of columns that define location in the alternate LRM (pertinent to this window). This is the set of columns shown in the bottom pane of the Setup Location Referencing Methods window when this alternate LRM is selected in the top pane.

#### **NOTE**

The Lane Direction column will only appear in this Alternate LRM window if the alternate LRM is configured with "use location across" in the top pane of the Setup Location Referencing Methods window.

#### **4.19.3. How to Change Reference Systems**

To change the method of identifying locations, navigate to the home window by clicking the  icon. Then click the drop-down arrow in the Location Reference field to display the list of referencing systems. Finally, click the desired LRS in the drop-down list.

After you change the method of identifying locations, the change will be implemented the next time that a window with location data is displayed (or is refreshed). The change may result in additional columns being added to the view of the table in the window. See the example in the following section for further information.

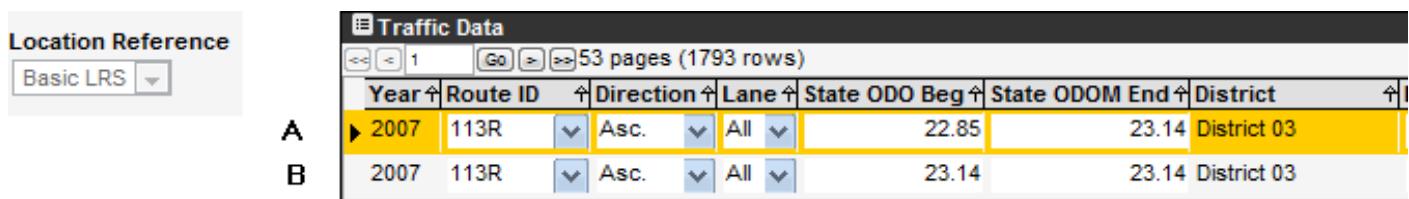
#### **4.19.4. Example of Changing the Reference System**

This example will show two traffic sections. The first reference system will identify locations by route and mile point. Then the LRS will be changed to an alternate referencing system where locations are identified by county post mile.

The traffic sections, in route and mile point reference, are both on route FL4040:

- The first section (marked "A") is between mile point 22.85 and 23.14.
- The second section (marked "B") is between mile point 23.14 and 23.14.

In the system, these sections look like this:

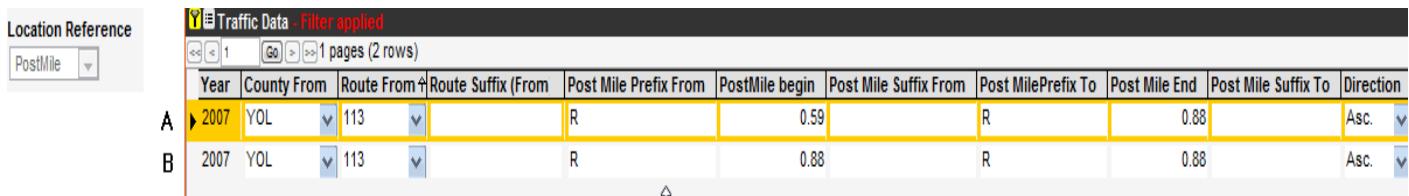


Traffic Data								
	Year	Route ID	Direction	Lane	State ODO Beg	State ODOM End	District	
A	2007	113R	Asc.	All	22.85	23.14	District 03	
B	2007	113R	Asc.	All	23.14	23.14	District 03	

The LRS is now changed to the alternate referencing system. In this system, the traffic sections Route, County, PostMile identification, which are:

- The first section (marked "A") is on Route 113 in YOL county between postmile 0.59 and 0.88.
- The second section (marked "B") starts on Route 113 in YOL county between postmile 0.88 and 0.88.

In the system, these sections look like this:



Traffic Data - Filter applied											
	Year	County From	Route From	Route Suffix (From)	Post Mile Prefix From	PostMile begin	Post Mile Suffix From	Post MilePrefix To	Post Mile End	Post Mile Suffix To	Direction
A	2007	YOL	113		R	0.59		R	0.88		Asc.
B	2007	YOL	113		R	0.88		R	0.88		Asc.

The system deduces the alternate location reference of county and post mile from the basic reference of route and mile point because of the data provided in the alternate reference system window shown on the following page.

Using this information:

- Traffic section data can be entered by route and mile point -- and then the system knows the county post mile; or
- Traffic section data can be entered by county post mile, and then the system translates and stores the information as route and mile point. Then, upon next accessing this information, it can be viewed or edited utilizing either route and mile point or county

post mile.

Y# - Filter applied												
Route ID	Direction	Lane	State ODO Beg	State ODOM End	Route From	County From	Post Mile Prefix From	Post Mile Suffix From	Route Suffix (From)	PostMile begin	Post Mile End	ROUTE SEQUENCE
113R	Asc.	All	19.447	19.894	113	SOL				19.637	20.084	19.637
113R	Asc.	All	21.05	21.799	113	SOL	R		R	21.24	21.989	21.24
113R	Asc.	All	21.799	22.26	113	SOL	R			21.989	22.45	21.989
113R	Asc.	All	22.26	33.402	113	YOL	R			0	11.142	0
113R	Asc.	All	33.402	33.446	113	YOL				11.296	11.34	11.296
113R	Both	All	0	11.688	113	SOL				0	11.688	0
113R	Both	All	11.688	19.447	113	SOL				11.878	19.637	11.878
113R	Both	All	19.894	21.05	113	SOL				20.084	21.24	20.084
113R	Both	All	33.446	43.254	113	YOL				11.34	21.148	11.34
113R	Both	All	43.254	43.454	113	YOL	M			21.148	21.348	21.148
113R	Both	All	43.454	43.64	113	YOL	R			21.348	21.534	21.348
113R	Both	All	43.64	44.195	113	YOL				21.521	22.076	21.521
113R	Both	All	44.195	54.736	113	SUT				0	10.541	0
113R	Both	All	54.736	56.11	113	SUT	R			10.541	11.915	10.541
113R	Both	All	56.11	60.563	113	SUT				11.927	16.38	11.927

## 4.20. Concurrent Location

(System > LRS & GIS > Concurrent Location)

System > LRS & GIS > Concurrent Location								
Dominant Location								
<< <   1   Go   > >> 87 pages (604 rows)								
Route	Direction	Lane	State ODO Beg	State ODOM End	User Update	Date Update	Comments	
101R	Both	All	438.996	441.273				
101R	Both	All	533.313	535.033				
101R	Both	All	535.399	541.803				
101R	Both	All	567.903	570.724				
101R	Both	All	571.762	579.667				
101R	Both	All	589.214	593.379				
101R	Both	All	594.034	605.909				
Subordinate Locations								
<< <   1   Go   > >> 1 pages (1 rows)								
Route	Direction	Lane	State ODO Beg	State ODOM End	Same Direction	User Update	Date Update	Comments
101L	Both	All	438.328	440.605	✓			

The Concurrent Location window allows you to associate alternate locations with a particular, principal location for the Basic LRS referencing system. The upper pane shows the principal location, while the lower pane shows the alternate locations ("aliases") that are associated with the principal location selected in the upper pane.

## 4.21. Manage GIS Data Sources

(System > LRS & GIS > Manage GIS Data Sources)

System > LRS & GIS > Manage GIS Data Sources							
GIS Data Source Name	GIS Data Source Type	GIS DATA SOURCE	GIS DATA SOURCE SHP COL	Where Clause	REF DATA SOURCE	REF DATA SOURCE DAT COL	REF DATA SOURCE
Assembly Districts	Polygon	ASSEM00	OBJECTID				
Bing Hybrid Tiles	Image						
Bing Map Tiles	Image						
Climate regions	Polygon	SETUP_CA_CLIMATE CA_CLIMATE_REGION_ID			SETUP_CA_CLIMATE CA_CLIMATE_REGION_ID		CA_CLIMATE_REG
Coastal Zones	Line	COASTZN	OBJECTID				
Congressional Districts	Polygon	CONGR00	OBJECTID				
County Boundaries	Polygon	CNTY_BND	OBJECTID				
County Boundaries w/o Ba	Polygon	CNTY_BND_WO_BA	OBJECTID				
County Lines	Line	CNTY_LIN	OBJECTID				
Culvert ET Shape	Point	ET-ShapeFile					
Culvert Shape	Line	Culvert_shape					
Culvert System	Point	P_CULVERT_SYS_IN P_CULVERT_SYS_ID			P_CULVERT_SYS_IN P_CULVERT_SYS_ID		
Districts	Polygon	DISTRICT	OBJECTID				
HOV Lanes	Line	HOV	OBJECTID				
JPCP data	Line	PMS_CA_APACS_JPC LOC_IDENT			PMS_CA_APACS_JPC LOC_IDENT		LOC_IDENT
Maintenance Facilities	Point	MAINT_FAC	OBJECTID				
MPO_DOT	Polygon	MPO_DOT	OBJECTID				

The Manage GIS Data Sources window lists all sources of GIS data. You use the window to add and maintain the various sources that display data on a map. The sources configured here are used in the Floating Map window when adding layers to a map. (See page 33 for more information on the Floating Map window.)

### 4.21.1. Description of the Manage GIS Data Sources Columns

The following table describes what data is entered in each column of the Manage GIS Data Sources window depending on the GIS data source type.

Column Name	GIS Source				
	ArcGIS Online, ArcGIS Online Tiles	Open Streets Tiles	Image	Shape	Oracle Spatial
GIS Data Source Name	This field shows the name of the GIS data source as displayed in the Map pane.				
GIS Data Source Type	This field is a drop-down list that indicates the type of GIS layer (point, line, polygon, or image).				
GIS Data Source	Path to desired layer in a semi-colon separated list (no ending semi-colon).	Table name of desired layer.	Name of the Image file.	Name of the Shape file.	Table name of desired layer.
GIS Data Source Shp Col	Leave blank.	Same as Shape.	Leave blank.	ID Column in GIS data source that identifies each feature to which data will be linked. If this column is null, features are given a dummy sequence by their record number.	
Where Clause	Leave blank.	SQL statement that limits the data that is retrieved.	Leave blank.	Leave blank.	SQL statement that limits the data that is retrieved.

Column Name	GIS Source						
	ArcGIS Online, ArcGIS Online Tiles	Open Streets Tiles	Image	Shape	Oracle Spatial		
Ref Data Source	Leave blank.	Same as Shape.	Leave blank.	The table within the AgileAssets schema that is a crosswalk between the GIS_DATA_SOURCE_SHP_COL and REF_DATA_SOURCE_DAT_COL columns.			
Ref Data Source Dat Col	Leave blank.	Same as Shape.	Leave blank.	A column within the REF_DATA_SOURCE table that matches data from within the AgileAssets schema.			
Ref Data Source Shp Col	Leave blank.	Same as Shape.	Leave blank.	The column within the REF_DATA_SOURCE table that matches the value in the GIS_DATA_SOURCE_SHP_COL column.			
Full Extent	This column is used to limit the maximum extent shown for a map in AgileAssets software. It is a pair of xY coordinates, with all four numbers being space-delimited.						
Coor. Ref. Sys.	This column contains a drop-down list that is comprised of the coordinate reference systems defined in the GIS Coor. Ref. Systems window (see page 175). You select the reference system that is to be associated with the layer and data source.						

#### 4.21.2. Description of the Right-click Shortcut Menu

When you right-click a record in the Manage GIS Data Sources window, the system displays a shortcut menu. This menu contains the common commands along with the following special commands:

- **Create New GIS Data Source** – This command begins the process of adding a new GIS data source.
- **Delete GIS Data Source** – This command removes the selected record from the Manage GIS Data Source window.
- **Recreate Window for Showing Attributes** – This command enables the **Show Attributes** command found in right-click shortcut menus in GIS-related windows.
- **Recreate Window for Editing Attributes** – This command enables the pop-up windows to open from the Data from Map window.
- **Preview** – This command displays the Floating Map window (see page 33).

## 4.22. Manage GIS Layers

(System > LRS & GIS > Manage GIS Layers)

System > LRS & GIS > Manage GIS Layers				
LAYER NAME	*GIS Data Source Name	GIS Data Source Type	Public?	User ID
Climate regions	Climate regions	Polygon	<input checked="" type="checkbox"/>	CAPAVEM
County Boundaries	County Boundaries	Polygon	<input checked="" type="checkbox"/>	CAPAVEM
Districts	Districts	Polygon	<input checked="" type="checkbox"/>	CAPAVEM
JPCP data	JPCP data	Line	<input checked="" type="checkbox"/>	LYN
Routes	Routes	Line	<input checked="" type="checkbox"/>	LYN

The Manage GIS Layers window lists all layers either created by you under the User ID you used when signing on or that are "public" layers. You use this window to review and manage your "library" of available layers. You can view how a layer appears (and begin the process of further customizing a layer) by using the **Preview** command, which displays the Floating Map window (see page 33).

## 4.23. GIS Images

(System > LRS & GIS > Gis Images)

System > LRS & GIS > Gis Images					
Gis Image	* GIS Data Source Type	Comments	Att.	User Update	Date Update
Bridge icon	Point	<input type="button" value="▼"/>	TODD	2/6/2012	
Culvert icon	Line	<input type="button" value="▼"/>	TODD	2/6/2012	

You use the GIS Images window to create a library of symbols that may be used in line and point theme styles in GIS maps. The symbols are image files that are associated with a record in this window via the **Work with Attachments** command.

### NOTE

The image files are recommended to be PNG format.

## 4.24. GIS Coordinate Reference System

(System > LRS & GIS > GIS Coor.Ref. Systems)

System > LRS & GIS > GIS Coor.Ref. Systems					
Coor. Ref. Sys.	* WKT	Comments	Att.	User Update	Date Update
GCS_North_American_1983	GEOGCS["GCS_North_American_1983",				
olditm	PROJCS["olditm",GEOC				
IDTM83	PROJCS["IDTM83",GEC				
NAD_1983_Albers	PROJCS["NAD_1983_A		ERIC	10/18/2011	

### NOTE

Only qualified personnel should modify the data shown in this window.

You use the GIS Coordinate Reference System window to create and maintain the various coordinate reference systems that may be selected in the Coor. Ref. Sys. column of the Manage GIS Data Sources window (see page 173).

## 4.25. Network Lanes

(System > LRS & GIS > Linear Network Tools > Geometry > Network Lanes)

System > LRS & GIS > Linear Network Tools > Geometry > Network Lanes							
Route	Direction	Lane	State ODO Beg	State ODOM End	Number of Lanes	Comments	Att.
101L	Desc.	All	0	1.161	3		
101L	Desc.	All	1.161	1.361	2		
101L	Desc.	All	1.361	3.148	3		
101L	Desc.	All	3.148	3.249	4		
101L	Desc.	All	3.249	3.371	5		
101L	Desc.	All	3.371	4.359	4		
101L	Desc.	All	4.359	4.507	5		
101L	Desc.	All	4.507	4.834	4		
101L	Desc.	All	4.834	4.974	5		
101L	Desc.	All	4.974	8.28	4		
101L	Desc.	All	8.28	8.409	3		
101L	Desc.	All	8.409	8.523	4		
101L	Desc.	All	8.523	8.655	3		
101L	Desc.	All	8.655	9.814	4		
101L	Desc.	All	9.814	11.546	5		

The Network Lanes window sets the number of lanes in each direction for your entire road network. Each record in this window indicates the number of lanes for a road section as identified by route, direction, start mile point, and end mile point. When entering data into this window, observe the following:

- For lane direction, **never** set the lane direction to "All."
- For lane ID#, **always** set the lane ID# to "All."

This "number of lanes" information is used for finest partition operations in:

- The Finest Partition window; and
- The System Jobs window, where it is used in the procedure to update the Finest Partition for the Pavement Structure Profile. (It updates the information needed for the Pavement Structure [Profile/Cross Section] window.)

When you right-click in the window, the following special command is displayed in a shortcut menu:

- **Fill Table** – This command causes the "lanes" information for the entire road network to be re-generated from the Pavement Management window's road sections and number of lanes values. Each pavement management record yields either one or two "lanes" records. Two records are generated, one for each direction, whenever the pavement management record's direction is "All." In this case, the number of lanes is also halved.

## 4.26. Routes List (Setup Network Lines)

(System > LRS & GIS > Linear Network Tools > Geometry > Routes List)

The Routes List window shows all of the routes (current and former) in the Caltrans linear network along with the minimum and maximum mile points of each. Those routes that are no longer in use will have the **Is Inactive** check box selected. The measurement points are editable. All other adjustments to the linear network (linear reference system [LRS]), such as add route, delete route, etc., are made by other means specific to your project's implementation, including:

- Use of the linear network supplied within the third-party GIS map that is used in the system;
  - Use of the GIS transactions window (if your agency does not have an interface to an external linear reference management system); or
  - Use of an interface that sends LRS adjustments and then applies them to this system.

## 4.27. Network Transactions

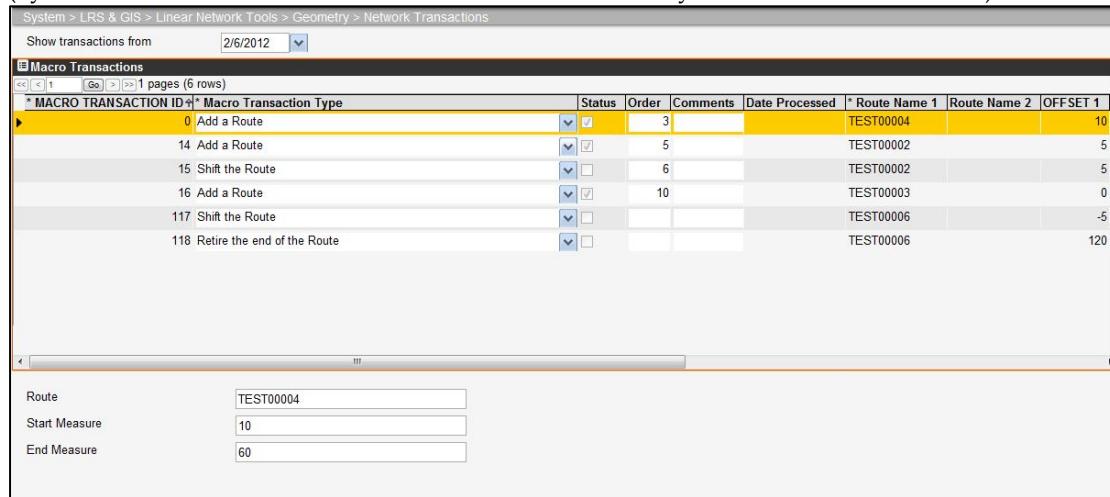
## Caution

You should only use this window if you do not have an interface to the AgileAssets Network Manager or your own agency's linear referencing management system. Contact AgileAssets before making any changes to this window if you are unsure.

The purpose of the Network Transactions window is to perform all adjustments to the base location reference system for a linear network within the AgileAssets system. All route additions, deletions, or mile point adjustments are made through this window by entering transactions and then committing them. ("Committing" a transaction means that the results of the transaction are permanently applied throughout the database and GIS map.)

#### 4.27.1. Network Transactions Window Description

(System > LRS & GIS > Linear Network Tools > Geometry > Network Transactions)



The screenshot shows the Network Transactions window. At the top, there is a toolbar with icons for search, refresh, and other functions. Below the toolbar, a message bar says "System > LRS & GIS > Linear Network Tools > Geometry > Network Transactions". A dropdown menu "Show transactions from" is set to "2/6/2012". The main area is divided into two panes: "Macro Transactions" and "Transaction Input".

**Macro Transactions:** This pane displays a table of transactions. The columns are: Macro Transaction ID, Macro Transaction Type, Status, Order, Comments, Date Processed, Route Name 1, Route Name 2, and Offset 1. The transactions listed are:

Macro Transaction ID	Macro Transaction Type	Status	Order	Comments	Date Processed	Route Name 1	Route Name 2	Offset 1
0	Add a Route	✓	3			TEST00004		10
14	Add a Route	✓	5			TEST00002		5
15	Shift the Route	✓	6			TEST00002		5
16	Add a Route	✓	10			TEST00003		0
117	Shift the Route	✓				TEST00006		-5
118	Retire the end of the Route	✓				TEST00006		120

**Transaction Input:** This pane contains three input fields: "Route" (TEST00004), "Start Measure" (10), and "End Measure" (60).

This window contains the following elements:

- The Show Transactions From field – By entering a date at the top of the window (in the format MM/DD/YYYY) and clicking the  icon, all uncommitted transactions as well as those committed since the entered date will be displayed in the Macro Transactions pane. When no date is entered, only uncommitted transactions are retrieved.
- Macro Transactions pane. This pane is described in the following section.
- Transaction Input pane. This pane is described on the following page.

#### Macro Transactions Pane

When the Network Transactions window is first displayed, the Macro Transactions pane at the top of the window shows all uncommitted transactions. If a date is entered in the Show Transactions From field, then all committed transactions since the entered date are also shown (along with all uncommitted transactions). To see earlier committed transactions, change the date to a date earlier than the desired date of the committed transaction and then click the  icon.

In the Macro Transactions pane, you enter and define new transactions. You may also edit uncommitted transactions. (An uncommitted transaction is one where the status field is not checked and consequently has no processed date.) After creating a new transaction or editing an uncommitted transaction, you may then commit the transaction, which applies the results of the transaction throughout the database and GIS map.

When you right-click a transaction in the table, a shortcut menu is displayed. This menu contains the following special commands along with the common commands:

- **Decompose Macro Transaction** – After selecting this command, the system displays a new window that shows the ordered series of basic transactions that compose the transaction you right-clicked. The transaction's input parameters are applied to these basic transactions as appropriate.

Note: Internally, the system operates using basic transactions. The **Decompose Macro Transaction** command is primarily used as a communications tool to convey how it operates internally. This is important, for example, during import routine development with a third-party system that controls the network. The eight basic

transactions are described in the Interface Design document for the third-party system.

- **Commit Transaction** - This command commits the currently selected transaction, applying it throughout the database and the GIS map. Once a transaction is committed, the system places a check mark in the transaction's Status column and displays the processed date in the transaction's Date Processed column.

### **Transaction Input Pane**

For the transaction and transaction type currently selected in the upper Macro Transactions pane, the Transaction Input pane at the bottom of the window displays the transaction inputs for the transaction type and allows you to enter the parameters for the transaction inputs. The transaction inputs vary depending upon transaction type. The table below identifies the inputs for each transaction type.

	Inputs				
Transaction Type	First Route	Second Route	First Value	Second Value	Third Value
Add second route to start of route.	Main route.	Route to add.	New start measure.		
Add second route to end of route.	Main route.	Second route.			
Add lane inside.	Route name.		Start measure.	End measure.	Lane direction.
Add lane outside.	Route name.		Start measure.	End measure.	Lane direction.
Add a route.	Main route.		Start measure.	End measure.	
Add asphalt to the end of a route.	Main route.		New pavement length.		
Add asphalt to the start of a route.	Main route.		New pavement length.	New start measure.	
Add new length.	Route name.		Start measure added.	End measure added.	
Delete lane inside.	Route name.		Start measure.	End measure.	Lane direction.
Delete lane outside.	Route name.		Start measure.	End measure.	Lane direction.
Delete portion of the route.	Route name.		Start measure deleted.	End measure deleted.	
Merge two routes.	Main route.	Route to add.			
Realign portion of the route.	Route name.		Start measure.	Original end measure.	New end measure.
Realignment with renaming of old roadbed.	Route retained.	New name of original pavement portion.	Start realignment.	End realignment.	Length of new pavement.
Realignment with retirement.	Route to realign.		Start of realignment.	End of realignment.	Length of new pavement.
Rename a route.	Existing route name.	New route name.			

	Inputs				
Transaction Type	First Route	Second Route	First Value	Second Value	Third Value
Rename a portion of a route.	Route name.	New route name.	Start measure.	End measure.	Start measure of new route.
Retire a route.	Route name.				
Retire a portion of a route.	Route name.		Start measure.	End measure.	
Reverse a route's mile points.	Route name.				
Shift a route's mile points.	Route name.		Miles to shift.		
Split data on a route at mile point.	Route name.		Mile point for splitting.		
Split a route and give a new name to the beginning of the route.	Route to split.	New name for beginning mileage.	Mile point for splitting.		
Split a route and give a new name to the end of the route.	Route to split.	New name for ending mileage.	Mile point for splitting.		
Trade dominance.	Subordinate route.		Start measure on subordinate route.	End measure on subordinate route.	

Several transactions not only adjust location reference but also work directly on attribute data records, splitting one record into two. This "split" capability is needed as it is the only way for data that spans a transaction mile point (that is, the attribute data's "from" point and "to" point are on either side of the transaction mile point) can be assigned to each of the two routes that are the result of the transaction. The transactions listed below use the "split" capability:

- Add lane inside.
- Delete lane inside.
- Delete portion of the route.
- Realign portion of the route.
- Realignment with rename of old roadbed (use "split" twice).
- Realignment with retirement (use "split" twice).
- Rename a portion of route (use "split" twice).
- Retire a portion of route (use "split" twice).
- Split data on a route at mile point (use "split" once).
- Split a route and give a new route name to the start (use "split" once).
- Split a route and give a new route name to the end (use "split" once).
- Trade dominance.

#### **4.27.2. How to Run a Network Transaction**

To run a network transaction, follow these steps:

1. Open the Network Transactions window.
2. Right-click the table and then click **Insert**. The system adds a new record to the table.
3. In the Macro Transaction Type column of the new record, click the down arrow to display the list of macro transactions and then click the desired transaction. The system displays the fields for the information needed for the transaction in the lower pane.
4. In the lower, Transaction Input pane, complete the fields with the necessary information.
5. In the upper, Macro Transaction pane, right-click the new record and then click **Commit Transaction**. The system runs the transaction. After running the transaction, the system displays today's date in the Date Processed column as an indication that the transaction ran successfully.

**NOTE**

If you are running multiple transactions that require the result of one transaction for the next transaction, use the Order column to enter the order in which the transactions should be run. The transactions will still need to be manually run one at a time, but this will help you to run them in the proper order.

#### **4.28. Setup Dynamic Aggregation**

You use the Setup Dynamic Aggregation window to establish and save dynamic aggregation "rule sets." These rule sets specify the source and destination tables, which columns to use as merging criteria (and the merging rules), and which columns to extract (and their aggregation rules such as average, min or max, first value observed, value over greatest distance, length-weighted average, etc.).

**NOTE**

Once these rule sets have been defined, they can be accessed and run at any time to re-segment the network or any selected subset of the network using the Dynamic Aggregation window.

To define a dynamic aggregation operation, this information is needed:

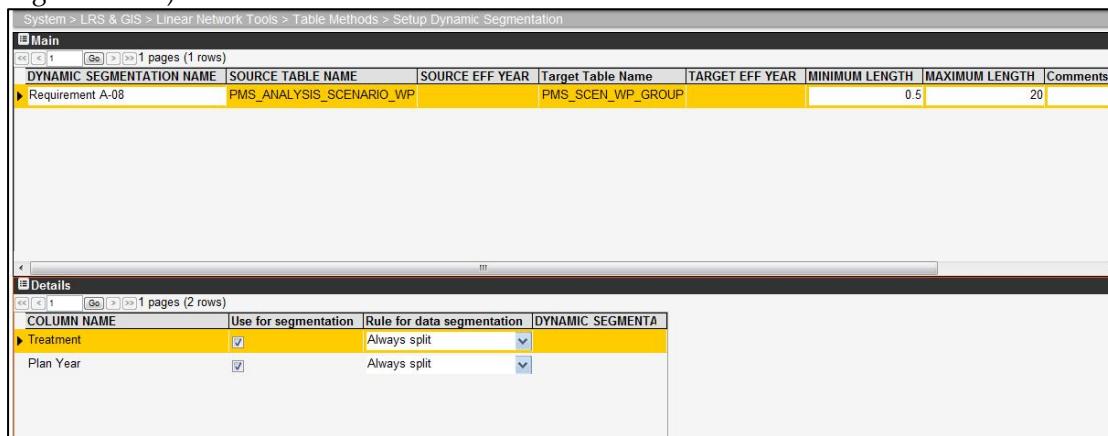
- An input table.
- An output table of "merged" road segments.
- For each column in the table, a set of "merging" rules for road segment boundaries and a set of "aggregation" rules to estimate data values for each "merged" road segment.

**NOTE**

Only columns that are shared by both the input table and the output table are candidates for dynamic aggregation. Only columns in the input table are candidates for use in the "merging" rules.

An example of the Setup Dynamic Aggregation window is shown below. It contains two panes: the [Main](#) pane and the [Details](#) pane. These panes are described in more detail in the following sections.

(System > LRS & GIS > Linear Network Tools > Table Methods > Setup Dynamic Segmentation)



The screenshot shows the 'Setup Dynamic Segmentation' window with two main panes:

- Main Pane:** Displays a table with one row. The columns are: DYNAMIC SEGMENTATION NAME, SOURCE TABLE NAME, SOURCE EFF YEAR, Target Table Name, TARGET EFF YEAR, MINIMUM LENGTH, MAXIMUM LENGTH, and Comments. The row data is: Requirement A-08, PMS\_ANALYSIS\_SCENARIO\_WP, PMS\_SCEN\_WP\_GROUP, 0.5, 20, and an empty comments field.
- Details Pane:** Displays a table with two rows. The columns are: COLUMN NAME, Use for segmentation, Rule for data segmentation, and DYNAMIC SEGMENTA. The rows are: Treatment (checked, Always split) and Plan Year (checked, Always split).

#### 4.28.1. [Main Pane](#)

##### NOTE

To save a record in this pane, the input table and output table must be named and exist in the Oracle database.

The top pane contains a list of the saved rule sets giving the name of the rule set, and the source and target table names. A minimum length is defined if the user wishes to omit segment breaks that would create segments shorter than a specified length.

Columns that are of particular importance are the following:

- Input Table Name — This is a required column containing the name of an existing table in the Oracle database. This table is used as the input to the dynamic aggregation operation.
- Output Table Name — This is a required column containing the name of an existing table in the Oracle database. This table is used as the output of the dynamic aggregation operation.
- Minimum Length/Maximum Length — These two columns determine the minimum and maximum lengths for the road segments that are output from the dynamic aggregation operation. The values entered in these columns are used in the "aggregation" rules. Adjacent input road segments will be merged so that each road segment is no shorter than the minimum length value and no longer than the maximum length value.

If one or both columns are blank, then the road segments are not constrained by that variable.

#### 4.28.2. [Details Pane](#)

The Details pane identifies all column-specific "merging" and "aggregation" rules. These rules (together with the Minimum Length rule from the Main pane) form the entire set of dynamic aggregation rules.

Once you insert a new rule set in the Main pane, the details regarding when segment breaks should be inserted and which data are then aggregated to the resulting segments are configured in the Details pane. The Details pane contains the list of fields from the source table that are used in the aggregation rules and/or are to be aggregated to the resulting sections. You can insert these from a list of all available fields from the source table.

The Use for Aggregation check box defines whether the field is to be used in establishing segment breaks. After selecting the Use for Aggregation check box, you select the aggregation rule in the Rule for Data Aggregation column.

#### **4.28.3. Aggregation Rules**

The following rules are available (note that the description of these rules assume that the output table starts as one segment for each entire route and that the rules then split the entire route into pieces, with each piece being one or more "merged" road segments from the input table):

- Always Split — This rule will insert a break whenever the value changes (but will omit breaks that would cause a segment length shorter than the minimum specified).
- Always Split (Ignore Minimum Lengths) — This rule will insert a break whenever the value changes (but will ignore the specified minimum length).
- Level Partition — This rule will insert a break based on user-defined levels. A break occurs whenever two adjacent values straddle a level.
- Statistical (Ignore Nulls) — This rule will insert a break based upon two levels. An average value is calculated for each contiguous set of sections where route and direction are the same. A break occurs whenever two adjacent values straddle a level. The two levels are calculated as an average multiplied by a user-input parameter called % Deviation (Average [ $1 \pm (\% \text{ Deviation} / 100)$ ]).

If you select the Level Partition or Statistical rule, a dialog box will be displayed to select the argument for the rule. After selecting the argument for the rule, it will be shown in the Dynamic Aggregation Rule Argument column. You may change the argument for the rule by utilizing the **Define Arguments** command.

#### **4.28.4. Description of the Right-click Shortcut Menu Commands**

When you right-click a record in this pane, a shortcut menu is displayed. This menu contains the following special commands:

- **Define Argument(s)** — This command allows you to modify the values shown in the Dynamic Aggregation Argument Rules column. When this command is selected, a dialog box opens that is appropriate for the type of rule selected. (This dialog box also appears when you select a rule in the Rule for Data Aggregation column.) Enter the new value(s) and then click **OK** to update the rule's parameter(s).

Note: The Dynamic Aggregation Argument Rules column will not update until you close and re-open the window or click the  icon.

- **Refresh Column List** — This command updates the list of columns shown in the pane.

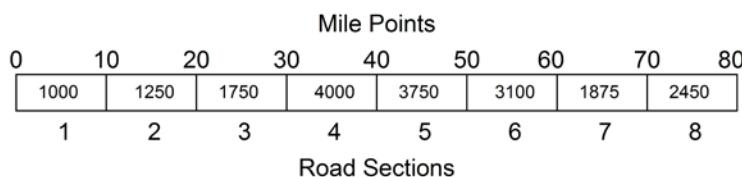
## 4.29. Dynamic Segmentation

(System > LRS & GIS > Linear Network Tools > Table Methods > Dynamic Segmentation)

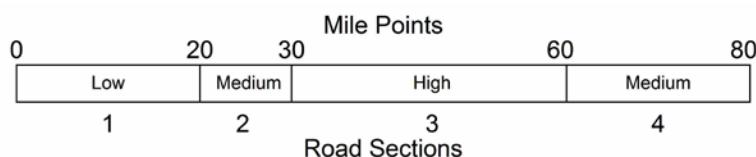
The screenshot shows the 'Dynamic Segmentation' window with three main sections:

- Route Selection:** A table with columns: Route, Direction, Lane, State OF, and State OD. One row is selected: 101L, Both, All, 0, 808.583.
- Segment Boundaries:** An empty table with columns: Route, Direction, Lane, State ODO Beg, and State ODOM End.
- Columns to Graph:** A list of checkboxes for 'Column Label' including: Preservation Benefit (selected), Plan Year, and Estimated Cost.

Dynamic Segmentation aggregates road sections so the resulting sections are homogenous for a particular data parameter according to the rules configured in the Setup Dynamic Segmentation window. For example, say traffic volumes are collected along a route as shown below:



You would like to aggregate these different volumes into three categories: low (less than 1500), medium (1500 to 2500), and high (greater than 2500). Using Dynamic Segmentation, the eight road sections of the route would be aggregated into four sections as shown below:



You use the Dynamic Segmentation window to perform a Dynamic Segmentation operation. Note that you cannot perform a Dynamic Segmentation operation until the operation has been defined in the Setup Dynamic Segmentation window (see the previous section). Once a Dynamic Segmentation rule set has been created in the Setup Dynamic Segmentation window, the aggregated output table may be refreshed at any time using the Dynamic Segmentation window.

The first step in using the Dynamic Segmentation window is to choose one of the Aggregation rule sets from the drop-down list shown in the top left corner of the window. After this, all further activity in this window is applicable only to this selected Dynamic Segmentation operation. This activity includes:

- Viewing the results of the Dynamic Segmentation operation graphically and literally without affecting the output file.
- If needed, making manual adjustments to the results.

- When the results are as desired, sending them to the output file.

**NOTE**

When the Dynamic Segmentation operation is selected from the drop-down list, then (if the temporary table is filled with previously run results of this operation) the Graph pane and the Segment Boundary pane are refreshed with the road segments produced from this previous run.

The Dynamic Segmentation window contains the following panes: Route Selection, Segment Boundaries, Columns to Graph, and the Graph. These panes are described in the following sections. Above the panes, the window also contains a drop-down list of Aggregation Rules and the following buttons that provide functionality for the selected Dynamic Segmentation operation:

- **Show Graph** – This button displays or refreshes the graph shown at the bottom of the window.
- **Do Aggregation** – This button performs the Dynamic Segmentation operation on the set of road segments as constrained by any filtering. The output is placed in a temporary table, which is displayed in the Graph pane and the Segment Boundary pane. If desired, the output may be adjusted in the Segment Boundary pane.

After the Dynamic Segmentation operation is performed, the Graph pane is refreshed. The prerequisites to display a graph in the Graph pane are: (1) select one or more columns in the Columns to Graph pane; and (2) have just previously clicked the **Do Aggregation** button for the currently selected Dynamic Segmentation operation.

- **Move Data** – This button becomes available after selecting the **Do Aggregation** button (which places data in the temporary table). It replaces the contents of the output table with the contents of the temporary table for all road segments as constrained by any filtering that may have been set.

#### **4.29.1. Route Selection Pane**

The Route Selection pane identifies the route and mile point boundaries that will be shown in the Graph pane. You select the route from a drop-down list. After selecting the route, the default mile points are shown. If desired, you may adjust the beginning and/or ending mile points to see a particular section of the route.

#### **4.29.2. Segment Boundaries Pane**

The Segment Boundaries pane lists the "temporary set" of road segments resulting from previously run results of the current Dynamic Segmentation operation (that is, the operation selected in the Aggregation field) for the route selected in the Route Selected pane. This pane is filled by clicking the **Do Aggregation** button.

Note the Show Records column. This column shows the input table's road segments and data that apply to this "output" road segment.

When you right-click a record in this pane, a shortcut menu is displayed. This menu contains the following special commands, which apply to segment boundaries:

- **Cut This Segment into Two** – This command displays a new window, into which you type a mile point that is within the boundaries of the current road segment. When you click **OK**, the current road segment becomes two road segments, split at the entered mile point. The Segment Boundaries pane is then refreshed accordingly.
- **Glue This to the Next Segment** – This command takes the currently highlighted road segment and the next road segment (the adjacent one with higher mile points)

and turns these two road segments into one. The Segment Boundaries pane is then refreshed accordingly.

#### 4.29.3. Columns to Graph Pane

The Columns to Graph pane lists the columns whose data can be graphed in the Graph pane. To graph one or more columns of data, click the appropriate check box.

#### 4.29.4. Graph Pane

For the selected route, the Graph pane graphs each column's data (as selected in the Columns to Graph pane) as well as the road segment boundaries produced by previously clicking the **Do Aggregation** button. A vertical line indicates each road segment boundary. The column data that is graphed is from the input table.

#### NOTE

If the incoming data has a gap in it, only the second section break on the gap is shown in the graph.

### 4.30. Setup Update from Geometry

(System > LRS & GIS > Configure Update from Geometry)

System > LRS & GIS > Configure Update from Geometry			
 << < < 1 (Go) > >> 1 pages (2 rows)			
* UPDATE TARGET COLUMN ID	* Geometry Update Target Table name	* UPDATE SOURCE COLUMN ID	* UPDATE SOURCE TABLE NAME
CA_CLIMATE_REGION_ID	PMS_CA_APACS_JPCP	CA_CLIMATE_REGION_ID	SETUP_CA_CLIMATE_REGION
CA_CLIMATE_REGION_ID	NETWORK_MASTER	CA_CLIMATE_REGION_ID	SETUP_CA_CLIMATE_REGION

When the **Update Target Table** command executes and encounters the Java string com.agileassetsinc.update.UpdateFromGeometrySource for a column (rather than an SQL statement), the system looks to this window to determine how to update the column. Essentially, this window defines how you acquire data from a polygon Shape file and use it as an attribute of a location-based data table.

The Update Target Column ID and Geometry Update Target Table Name columns show the column that will be updated and the table in which the column resides. The Update Source Column ID shows where the data to be used to update the column will come from, with the Update Source Table Name column being the table in which the source column resides.

### 4.31. System Archive

#### NOTE

Only a user of the system may perform the archive function and, further, he or she must be the only user logged in when the archive function is initiated. **Once the archive function is underway, log-in is disabled until the archive function completes.**

Archiving allows a System Administrator to move data from the operational portion of the database to a location that is inaccessible to the users. The data that is archived is generally historical data that is no longer of immediate value. By archiving unneeded data, the size of

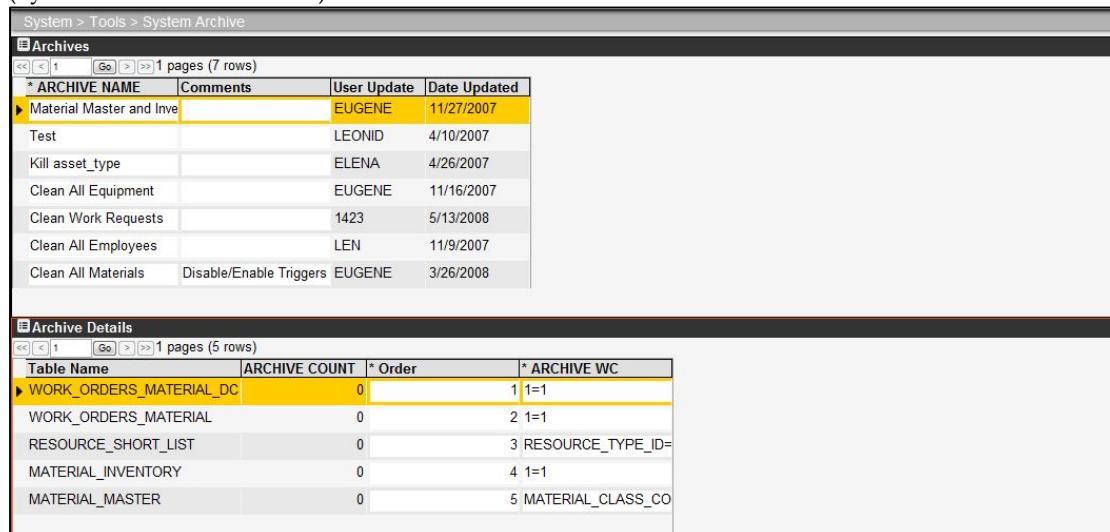
the operational portion of the database is reduced — which in turn reduces the amount of time needed to access data.

The archiving of data is a powerful feature, yet also potentially destructive to the integrity of the data. For this reason, the archiving of data must be approached cautiously and with a great deal of forethought. AgileAssets recommends that you only use existing archive sets to archive data.

If necessary, archived data may be restored to the operational portion of the database so it is again accessible.

#### 4.31.1. Description of the Window

(System > Tools > Archive)



System > Tools > System Archive			
Archives			
<<	<<	1	[Go] [>] [>>]
1 pages (7 rows)			
* ARCHIVE NAME	Comments	User Update	Date Updated
Material Master and Inv		EUGENE	11/27/2007
Test		LEONID	4/10/2007
Kill asset_type		ELENA	4/26/2007
Clean All Equipment		EUGENE	11/16/2007
Clean Work Requests		1423	5/13/2008
Clean All Employees		LEN	11/9/2007
Clean All Materials	Disable/Enable Triggers	EUGENE	3/26/2008

Archive Details			
Table Name ARCHIVE COUNT * Order * ARCHIVE WC			
WORK_ORDERS_MATERIAL_DC	0	1 1=1	
WORK_ORDERS_MATERIAL	0	2 1=1	
RESOURCE_SHORT_LIST	0	3 RESOURCE_TYPE_ID=	
MATERIAL_INVENTORY	0	4 1=1	
MATERIAL_MASTER	0	5 MATERIAL_CLASS_CO	

The window is divided into two portions: the Archives list (top pane) and the Archive Details (bottom pane). For each archive record in the Archives pane, records in the Archive Details pane indicate what data in a single database table will be archived based upon the constraints established in the Archive\_WC column. In other words, the archive record serves as an umbrella for the various actions needed to archive data from a single database table.

#### Archives Pane

The Archives pane shows all defined archive sets in the database. An archive set is composed of the archive actions indicated in the Archive Details pane.

When you right-click an archive set, a shortcut menu is displayed. This menu contains the common commands along with the following special commands:

- **Archive** — This command initiates the archive actions for the highlighted archive set. For archive sets with multiple detail records, the archive actions are performed in the order, from lowest to highest, indicated in the Order column of the Archive Details pane.

Note: If you attempt to use this command to archive data previously archived, the system will warn you that this will destroy the previously archived data. You may then choose to proceed with the archive or abort the process.

- **Restore** — This command moves the data specified in the highlighted archive set to the user-accessible portion of the database.

### Archive Details Pane

The Archive Details pane defines all tables and the portion of data per table that will be archived when the Archive command is activated for the selected archive set in the Archive pane. Each detail record defines the data archived for a single table. To prevent the loss of data, several detail records may be needed for a single archive operation and the order that these tables are archived must also be set appropriately. The order is set in the Order column, and the archive actions are performed starting with the lowest-ranked table.

#### NOTE

If a table is included in an archive, then all tables that have a foreign key to that table must also be included in the archive with a lower value.

The Archive Details pane contains the following columns:

- Archive WC [Where Clause] – This column shows the SQL WHERE statement that controls what data is archived.
- Archive Count - This read-only column indicates the number of records archived for the table associated with the detail record.
- Order - This column indicates the order in which each table is archived. Tables are archived in order of lowest to highest. Remember that tables with foreign keys must have lower numbers than their parent table.
- Archive\_WC - This column contains the Oracle WHERE statement that indicates what portion of the table's data is to be archived. This statement typically defines a time period for which data will be archived. For example, "archive all data more than 3 years old" would be translated into DATE\_WORK < SYSDATE - 365\*3 for the Labor day cards table.

When you right-click this pane, the common commands are available. However, the **Insert** command works slightly different than in other windows. It not only inserts a new record in the Archive Details pane, but also displays a window that contains a drop-down list of all table names so you may select the appropriate table for the new detail record.

#### **4.31.2. How to Perform an Initial Archive**

To perform an initial archive, follow these steps:

1. Display the System Archive window.
2. In the Archives pane, locate the archive set to be used.
3. Right-click the record showing the desired archive set and then click **Archive**. The system will begin archiving the data specified in the Archive Details pane.

#### **4.31.3. How to Perform Subsequent Archives**

After initially archiving data, additional archiving of the same data will destroy the data previously archived. You should, therefore, not perform subsequent archives without careful consideration of the consequences nor without assistance from AgileAssets. If you do decide to proceed with a subsequent archive, follow these steps:

1. Display the System Archive window.
2. In the Archives pane, click the desired archive set to select it.
3. Write down all information shown in the Archive Details pane for the selected archive set. (Alternatively, print the screen to capture this information.)

4. In the Archives pane, right-click and then click **Insert**. A new record is added to the pane.
5. In the Archive Details pane, using the information acquired in step 3, duplicate the details of the initial archive by inserting the appropriate number of rows and entering the data. Also adjust the Archive\_WC column to accurately define the historic data to be archived.
6. Once you have configured the Archive Details pane, write down all information shown in this pane. (Alternatively, print the screen to capture this information.)
7. Label the information gathered in step 3 and in step 6 so it is clear which information is for which archive set (initial or duplicate) and then send the information to AgileAssets for evaluation.

AgileAssets will review the information and then indicate whether you may proceed with the subsequent archive or whether additional refinement of the duplicate archive set is necessary. Once AgileAssets approves the duplicate archive set and explicitly instructs you to proceed:

1. Copy the entire Oracle database to a test schema and retain a copy of the dump file.
2. In the Archive pane of the test schema, right-click the new archive set and then click **Archive**. The system will begin performing the archive actions.
3. Once the archive operation completes, check the system as a user to see that everything works. If everything works as expected, you may now perform the archive in the production schema. If problems are found, consult with AgileAssets for assistance in resolving them.
4. In the Archive pane of the production schema, right-click the new archive set and then click **Archive**. The system will begin performing the archive actions.
5. Once the archive operation completes, check the system as a user to see that everything works. If problems are found, replace the production schema with the contents of the dump file and advise AgileAssets to evaluate the situation.

#### **4.31.4. How to Retrieve Archived Data**

To retrieve archived data, follow these steps:

1. Display the System Archive window.
2. In the Archives pane, locate the archive set to be used.
3. Right-click the record showing the desired archive set and then click **Recoup**. The system will begin moving the data specified in the Archive Details pane to the user-accessible portion of the database.

#### **4.31.5. How to Create a New Archive Set**

##### **CAUTION**

**Due to the potential for inadvertently destroying data, you should not create a new archive set without consulting with AgileAssets. If none of the existing archive sets meet your needs, please contact AgileAssets for further assistance.**

The following example shows the level of detail and complexity required to archive data, which emphasizes why it is imperative that AgileAssets' staff be involved in the creation of new archive sets. In this example, old equipment repair orders are archived, where "old" is defined as all repair orders with repair order IDs less than 1000.

Table Name	Archive Count	Order	Archive WC
WORK_ORDERS_LABOR_DC	0	1	WORK_ORDER_ID IN ( SELECT w.WORK_ORDER_ID FROM WORK_ORDERS w, SETUP_PROJECT p WHERE p.PROJECT_ID = w.PROJECT_ID AND p.PROJECT_ID < 1000 and p MODULE_ID = 5 )
WORK_ORDERS_MATERIAL_DC	0	2	WORK_ORDER_ID IN ( SELECT w.WORK_ORDER_ID FROM WORK_ORDERS w, SETUP_PROJECT p WHERE p.PROJECT_ID = w.PROJECT_ID AND p.PROJECT_ID < 1000 and p MODULE_ID = 5 )
WORK_ORDERS	0	4	WORK_ORDER_ID IN ( SELECT w.WORK_ORDER_ID FROM WORK_ORDERS w, SETUP_PROJECT p WHERE p.PROJECT_ID = w.PROJECT_ID AND p.PROJECT_ID < 1000 and p MODULE_ID = 5 )
EQUIPMENT_REPAIR_DIRECT	0	3	PROJECT_ID < 1000
SETUP_PROJECT	0	5	PROJECT_ID < 1000 and MODULE_ID = 5

## 4.32. Groovy Scripts

### CAUTION

The Groovy Scripting feature provides considerable flexibility to the software. Due to the power of this feature, it should only be used by people trained in Java and its use within this application. Used improperly, this feature can cause serious problems.

The Groovy Scripts window (System > Tools > Groovy Scripts) is where Groovy scripts are developed and maintained. Groovy scripts provide the ability to add features to the application by writing and applying appropriate Java code. For example, this feature is often used to add data-checking capabilities for data entry. The scripts configured in this window are then available in the appropriate drop-down lists found in other parts of the application.

### 4.32.1. Description of the Groovy Scripts Window

The screenshot shows the 'Groovy Scripts' window with two main sections:

- Groovy Script Type:** A table listing Groovy Script Type Name and Description. One row is highlighted in yellow: 'AfterInsert' with the description 'Parameters: DataLayer dl, // instance of DataLayer DataStore dsln, // datastore with inserted row in it'.
- Groovy Script:** A table listing Groovy Script ID, Groovy Script Name, and Script. Several rows are highlighted in yellow, including 'set\_labor\_id', 'SET\_INV\_ITEM\_DATE', 'set\_contract\_name\_to\_id', 'ITD\_Request\_User', and 'FLEET\_Set\_EQ\_Attributes'. The 'Script' column for 'set\_labor\_id' contains the following Groovy code:

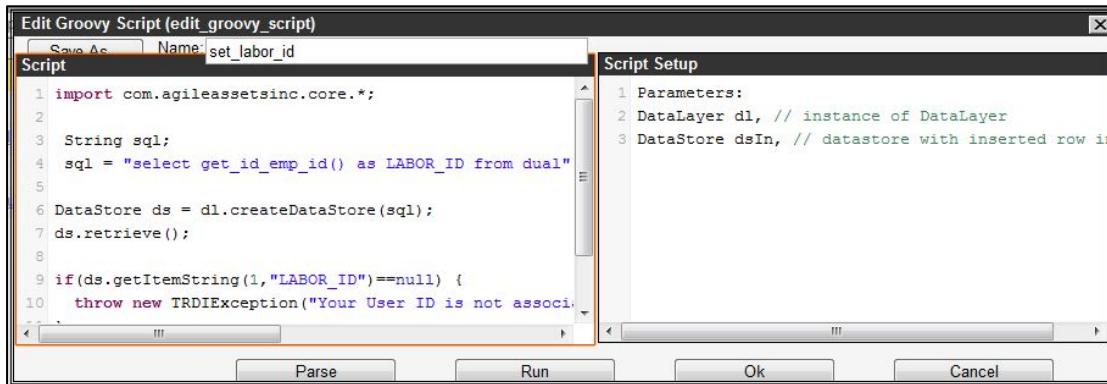
```
import com.agileassetsinc.core.*; String sql; sql = "select get_id_emp_id() as LABOR_ID from dual"; DataStore ds = dl.createDataStore(sql); import com.agileassetsinc.core.*; String sql; sql = "select getdate() as WORK_DATE from dual"; DataStore ds = dl.createDataStore(sql); ds.setDate("WORK_DATE", getdate());
```

The top pane of the Groovy Scripts window shows the different types of Groovy scripts that are available.

For the type selected in the upper pane, the lower pane shows the Groovy scripts that have been created for this type. The lower pane also provides the following command in addition to the common commands:

- **Edit Groovy Script** – This command displays the Groovy Script dialog box, in which you may modify the selected Groovy script. See the following section for more information.

#### **4.32.2. Description of the Groovy Script Dialog Box**



The Groovy Script dialog box is displayed after selecting the **Insert** or **Edit Groovy Script** command from the Groovy Scripts window. This dialog box allows you to create or modify Groovy scripts. (Note: If you use the **Insert** command, the application will first ask you to enter the ID for the new script. Then, after entering the ID, the application will display the Groovy Script dialog box.)

The left pane shows the actual Groovy script. (The internal name of the script is shown in the ID field, while the displayed name for the script is shown in the Name field.) The right pane shows the allowed parameters for this type of script and serves as a reference when creating or modifying scripts.

The following command buttons are also available:

- **Save As** – This button allows you to copy the current Groovy script as a basis for creating a new Groovy script. After clicking the button, the application asks for the ID of the new script. After entering the new ID, you assign a new name to the script and adjust the script as desired.
- **Parse** – This button checks the Groovy script's syntax and displays what is found.
- **Run** – This button runs the Groovy script and shows the output. This button will only work for those types that have no input parameters; otherwise an error will occur (saying the script needs input).
- **OK** – This button saves the Groovy script and closes the Groovy Script dialog box.
- **Cancel** – This button discards any changes made in the Groovy Script dialog box and closes it.

#### **4.32.3. Parameters for Each Script Type**

The following table shows the different types of Groovy scripts. Each Groovy script uses particular parameters (and only these parameters). The following table lists the input parameters for each type of Groovy Script.

Type	Parameters	Description
Activity	DataLayer dl	Instance of DataLayer.
	DataStore dsIn	Datastore with all modified rows in it.
AfterTreatmentChange	DataStore dsIn	Datastore with optimization engine.
	int rowIn	Current record.
	double val_in	Value of column being edited.
	Return	Desired changed value.
AfterChange Notification Trigger		
AfterImport	Varies	May contain various parameters.
AfterInsert	DataLayer dl	Instance of DataLayer.
	DataStore dsIn	Datastore with inserted row in it.
BeforeOpenMenu		
BeforeSave	DataLayer dl	Instance of DataLayer.
	DataStore dsIn	Datastore with all modified rows in it.
CalcColumnsinAnalysis		
ChangeWithTime	DataStore ds_run	Datastore with optimization engine.
	int rowIn	The current row within the optimization engine.
	Double valueIn	The value of the column that is being changed (within rowIn).
	newValueOut	Return value (string or integer or double).
ConditionImprovement	DataStore ds_run	Datastore with optimization engine.
	int rowIn	The current row within the optimization engine.
	Double valueIn	The value of the column that is being changed (within rowIn).
	newValueOut	Return value (string or integer or double).
ImportMapping	Varies	May contain various parameters.
MMS Models	DataStore dsIn	Datastore with current data in it.
	int rowIn	Current row of dsIn.
	currentScore	Current score.
	targetScore	Target score.
	currentBudget	Current budget.

Type	Parameters	Description
	Return double	The future budget.
MMS Util Function	DataStore dsIn	Datastore with current data in it.
	int rowIn	Current row of dsIn.
	currentScore	Current score.
	Return double	The utility value for the given score.
OnItemChange	DataLayer dl	Instance of DataLayer.
	DataStore dsIn	Datastore with current row in it.
	String columnIn	The column name in which the change occurs.
	String newValueIn	The new value as a string.
OnLinkClick	DataLayer dl	Instance of DataLayer.
	DataStore dsIn	Datastore with current row in it.
	Return string	URL of the link.
PIThresholdForRSL	DataLayer dl	Instance of DataLayer.
	DataStore dsIn	Network master or analysis engine DataStore.
	int rowIn	Row in dsIn.
	return double	Threshold value in natural PI scale.
System Job Notification		
Treatment	DataLayer dl	Instance of DataLayer.
	DataStore dsIn	Datastore with all modified rows in it.

### 4.33. System Job Executables

(System > Tools > System Job > Executables)

System Job Executables				
	* System Job Executable	System Job Executable Source	Comments	User Update
1	* 1 Update Alt Refs in Location Table	com.agileassetsinc.job.UpdateSetup	This job resets the alt re	
0	0 FillDefSurvSummJobGen	com.agileassetsinc.job.FillDefSurvSu	Fills the DEFECT_SURV	
0	0 FillInvSummAmountJob	com.agileassetsinc.job.FillInvSummA	This job summarizes inv	
1.	1. Save Quantity Guidelines	com.agileassetsinc.job.SaveQgJob	Saves a copy of Activity	
2.	2. Restore Quantity Guidelines	com.agileassetsinc.job.RestoreQgJob	Restores a previously sa	
	Advance FTE Fiscal Year	com.agileassetsinc.job.AdvanceFteF	fills a new year's (year i	
	Buffer Employee Update	com.agileassetsinc.job.BufferFillEmp	Fill tables from buffer En	
	Build Dashboard Reports	com.agileassetsinc.job.BuildDlReport	This procedure refreshes	
	Built View for QA/QC reports	com.agileassetsinc.custom.wy.Creat	This job rebuilds the view	
	CalculateUncoveredSections	com.agileassetsinc.job.CalculateUnc	Find and fill sections not	
	Check help files	com.agileassetsinc.job.CheckHelpFil	Finds out where there ar	
	Clean abandoned locations	com.agileassetsinc.job.CleanAbando		
	Clean Abandoned Sessions	com.agileassetsinc.job.CleanAbando	Cleans abandoned Orac	
	Clean Grid Report Buffer Table	com.agileassetsinc.job.CleanGridRe		
	Clean Import Config Log	com.agileassetsinc.job.CleanImportC	Clean Import Config Log	
	Clean Loc Idents	com.agileassetsinc.job.CleanSetupL	This job updates the SE	
	Clean tmp folder	com.agileassetsinc.job.CleanTmpFol	Clean tmp folder on the :	

The Executables window contains the list of programs that may be called by system jobs. A program may be either a Java class or can be an Oracle procedure. These programs are stored elsewhere in the database/application, and only the name of the program is stored in this window.

The programs configured here appear in the drop-down list in the System Job Executable column that is found in the System Jobs Schedules window (see the following section).

When you right-click the window, a shortcut menu is displayed. This menu provides the common commands along with the following commands:

- **Insert Executable** — This command adds a record to the window.
- **Delete Executable** — This command removes the selected record.

### 4.34. System Job Schedules

#### NOTE

Scheduled jobs will only run if a user called SYSTEM is in the User Names and Access window.

Only personnel knowledgeable in Oracle and authorized as a database administrator should be allowed read/write access to this window.

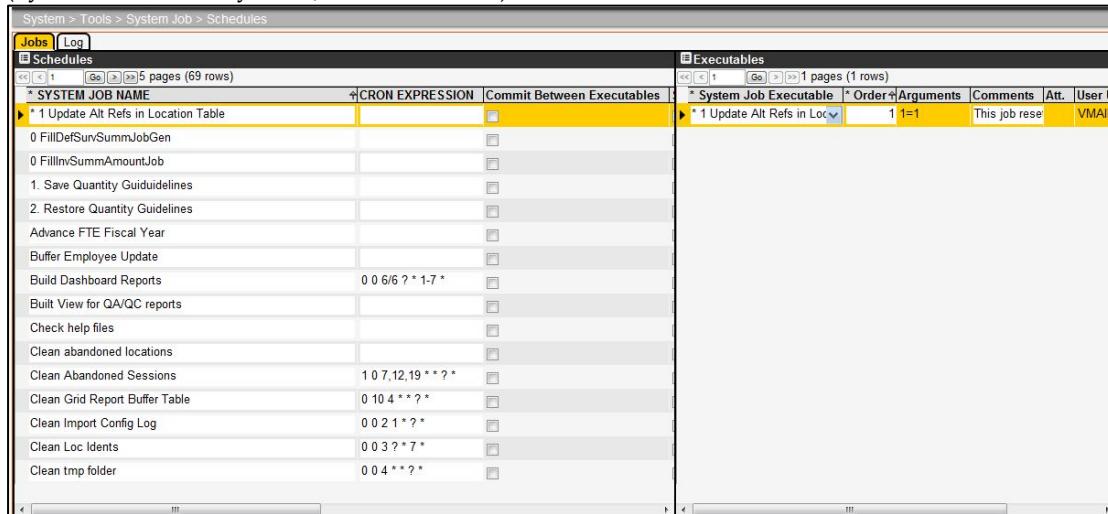
Since the CRON expression will no longer be processed once the day specified in the End Time field is reached, and so the job will not execute, it is not necessary to delete a job. You may instead simply set the End Time field to a day in the past to prevent the job from executing.

This window displays all system jobs available to your agency and is used to invoke system jobs. From this window, system jobs may be either run "on-demand" or scheduled to automatically execute periodically.

There are two tabs in this window: the Jobs tab and the Log tab. These tabs are described in more detail in the following sections.

#### 4.34.1. Jobs Tab

(System > Tools > System Job > Schedules)



SYSTEM JOB NAME	CRON EXPRESSION	Commit Between Executables
* 1 Update Alt Refs in Location Table		<input checked="" type="checkbox"/>
0 FillDefSurvSummJobGen		<input type="checkbox"/>
0 FillInvSummaAmountJob		<input type="checkbox"/>
1. Save Quantity Guidelines		<input type="checkbox"/>
2. Restore Quantity Guidelines		<input type="checkbox"/>
Advance FTE Fiscal Year		<input type="checkbox"/>
Buffer Employee Update		<input type="checkbox"/>
Build Dashboard Reports	0 0 6/6 ? * 1-7 *	<input type="checkbox"/>
Built View for QA/QC reports		<input type="checkbox"/>
Check help files		<input type="checkbox"/>
Clean abandoned locations		<input type="checkbox"/>
Clean Abandoned Sessions	1 0 7,12,19 * * ? *	<input type="checkbox"/>
Clean Grid Report Buffer Table	0 10 4 * * ? *	<input type="checkbox"/>
Clean Import Config Log	0 0 2 1 * ? *	<input type="checkbox"/>
Clean Loc Idents	0 0 3 ? * 7 *	<input type="checkbox"/>
Clean tmp folder	0 0 4 * * ? *	<input type="checkbox"/>

System Job Executable	Order	Arguments	Comments	Att.	User U
* 1 Update Alt Refs in Loc	1	I=1	This job rese		VMAIN

The Jobs tab contains two panes: Schedules and Executables. The Schedules pane shows the various system jobs that are available. The Executables pane shows what program or programs will execute when the system job selected in the Schedules pane runs. These panes are described in more detail in the following sections.

#### ***The Schedules Pane***

The Schedules pane shows all system jobs. These jobs may be run "on demand" or scheduled to automatically execute periodically.

This tab contains the following columns:

- Job Name – This column shows the name of the system job.
- CRON Expression – This column shows an abbreviated description of when the job is scheduled to run. The value in this column is created or modified by using the **Edit CRON Expression** right-click command.
- Scheduled? – If the check box in this column is selected, then the schedule set for this job (by using the right-click **Edit CRON Expression** command) will be followed and the job will run automatically according to that schedule provided a user called SYSTEM is in the User Names and Access window.
- Comments – This column contains a description of what the job does.
- Start Time – This column sets the beginning of the time period in which the system will process the CRON expression. You may enter a date directly or double-click the field to display a calendar.
- End Time – This column sets the end of the time period in which the system will process the CRON expression. You may enter a date directly or double-click the field to display a calendar. Once the day specified in this field is reached, the job will no longer run.

When you right-click a particular system job in the list, in addition to the common commands the following special commands are available:

- **Insert Schedule** – This command inserts a new schedule record in the pane.
- **Delete Schedule** – This command deletes the selected schedule record from the pane.
- **Next Fire Date** – For scheduled jobs, this command will display when the selected job will next run.
- **Run Job** – This command initiates the system job that you right-clicked.

Note: If you right-click the Run Interface system job, the system displays a dialog box with the list of available interfaces in a drop-down list. Click the down arrow to display the list of interfaces, click the desired interface to be run, and finally click **OK**.

- **Edit CRON Expression** – This command allows you to set the periodic schedule by which this job will be automatically run. After selecting the command, the system displays the CRON Expression Editor dialog box. See below for more information on this dialog box.

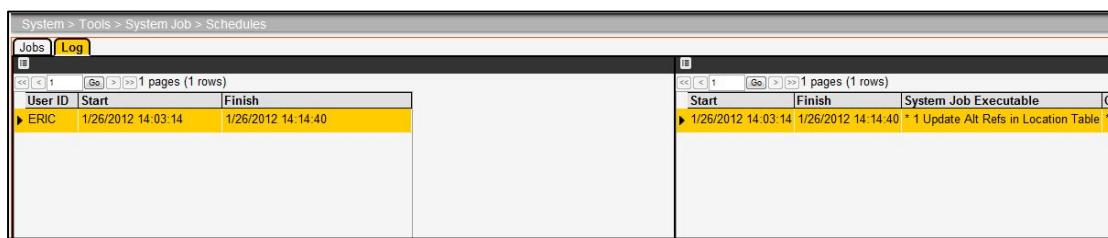
#### **The Executables Pane**

For the system job selected in the Schedules pane, the Executables pane shows one or more programs that will execute when the system job runs. (When multiple programs are listed, they execute in the order set by the Order column, with the next program starting when the previous job finishes. If any program in the series fails, then the entire job halts and all changes are rolled back.)

The programs shown in the drop-down list in the System Job Executable column are those listed in the Executables window. (See page 194 for more information on this window.)

The input values for a program are set by right-clicking the record for the program and then clicking **Define Argument(s)**. The system responds by displaying a dialog box so you may enter the input values. Once you enter these values and close the dialog box, the values will appear in the Arguments column.

#### **4.34.2. Log Tab**



System > Tools > System Job > Schedules		
Jobs		
1 pages (1 rows)		
User ID	Start	Finish
► ERIC 1/26/2012 14:03:14 1/26/2012 14:14:40		
Log		
1 pages (1 rows)		
Start	Finish	System Job Executable
► 1/26/2012 14:03:14 1/26/2012 14:14:40 * 1 Update Alt Refs in Location Table *		

The Log tab shows the run history of the job selected in the Jobs tab. For each run, the start and end dates are shown as well as any Error Text produced when that run did not finish satisfactorily.

#### **4.34.3. CRON Expression Editor**

The CRON Expression Editor is a dialog box that is displayed by executing the right-click **Set Schedule** command. You use this dialog box to set when the selected job will run within the time period specified by the Start Time and End Time fields found in the Jobs pane. The dialog box contains four tabs:

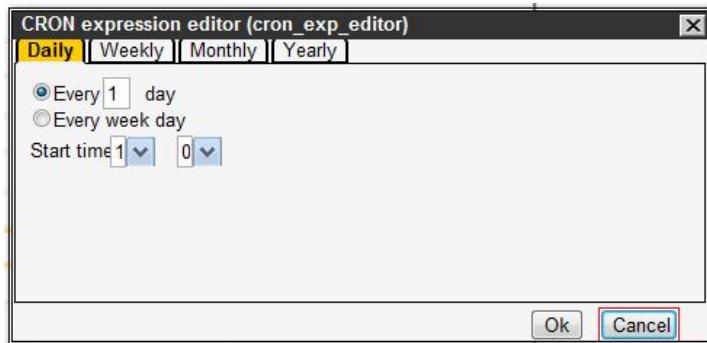
- Daily Tab.
- Weekly Tab.

- Monthly Tab.
- Yearly Tab.

#### NOTE

The user is responsible for using this dialog box wisely. For example, setting an automatic schedule for procedures with input parameters may result in "process" problems. If it is scheduled, the procedure will run automatically using its default parameter value set as identified in the window. This may lead to negative results. Consequently, the user is responsible for making the correct decision whether to schedule jobs that have input parameters.

#### **Daily Tab**



The Daily tab of the CRON Expression Editor dialog box allows you to set a job to run once on certain days. (You cannot specify a job to run multiple times on one day, although you could get around this restriction by inserting multiple instances of the job in the Jobs tab and then set each job to run at different times of a certain day.) The Daily tab contains two radio buttons:

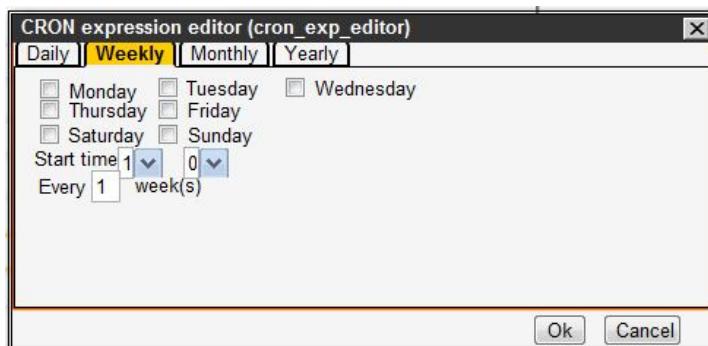
- The upper radio button sets the selected job to run on days counted iteratively rather than certain days of the week or dates. After selecting the radio button, enter a value in the field for how many days until the job runs again.

For example, entering 1 in the field means the job will run every day; entering 2 means the job runs every other (that is, every second) day; and so forth. The time of the day when the job runs is set in the Start Time field at the bottom of the dialog box.

- The lower radio button sets the job to run on every week day (that is, Monday through Friday, inclusive). The time of the day when the job runs is set in the Start Time field at the bottom of the dialog box.

Note: Selecting this radio button has the same effect as selecting the Monday through Friday check boxes and setting the iteration to every week in the Weekly tab.

### Weekly Tab

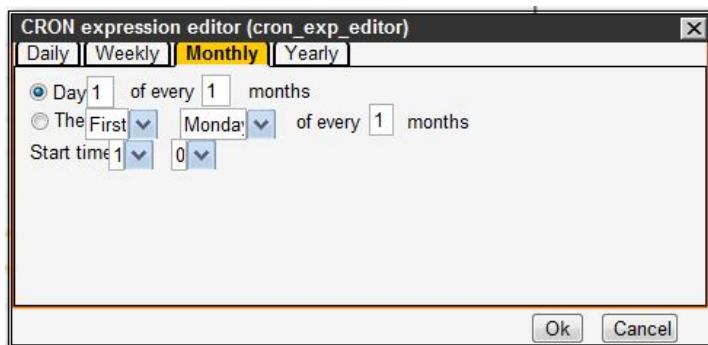


The Weekly tab of the CRON Expression Editor dialog box allows you to set a job to run on certain days of certain weeks. The tab provides seven check boxes, one for each day of the week. You click a check box to have the job run on that day. The time when the job runs is the same for all selected days and is set in the Start Time field.

Once you have specified the days and time when the job will run, you then set whether the job runs each week, every other week, or at some other iteration. You specify the weekly iteration in the Every {number} Weeks field.

For example, entering 1 in this field will cause the job to run on the specified days every week. Similarly, entering a 4 in this field will cause the job to run on the specified days every fourth week (that is, run one week, skip two weeks, and then run again).

### Monthly Tab



The Monthly tab of the CRON Expression Editor dialog box allows you to set a job to run on a certain day (either by day of the month or day of the week) of a certain month. The time of day when the job runs is set in the Start Time field. The tab provides two radio buttons:

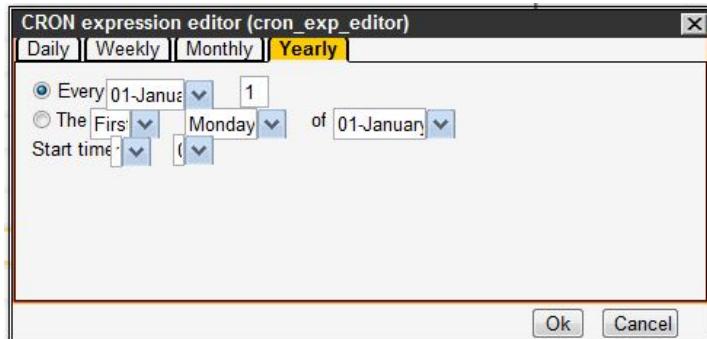
- The upper radio button sets the job to run on a certain day of the month, with the day of the month being entered in the first field of the radio button label. The second field of the label determines what monthly iteration the job will run.

For example, if you configure the label to read, "Day 1 of every 3 months," the job would run on the first day of a month, then skip two months, and then run again on the first day of the fourth month. This would be a typical schedule for quarterly reports.

- The lower radio button sets the job to run on a certain day of the week of certain months. The first two fields of the radio button label set which day the job will run (for example, "first Monday," "fourth Friday," etc.). The third field of the label determines what monthly iteration the job will run.

For example, if you configure the label to read, "The First Monday of every 1 months," the job would run on the first Monday of every month. This would be a typical schedule for, say, a report of monthly work order backlogs.

### Yearly Tab



The Yearly tab of the CRON Expression Editor dialog box allows you to set a job to run on a certain day (either by day of the month or day of the week) of one particular month of the year. The time of day when the job runs is set in the Start Time field. The tab provides two radio buttons:

- The upper radio button sets the job to run on a certain day of a particular month. The first field is a drop-down list containing the months of the year. The second field is where you enter the day of the month.

For example, to run a job once every year on June 30th, you would select 06-June from the drop-down list and then enter 30 in the second field.

- The lower radio button sets the job to run on a certain day of the week of a particular month. The first two fields of the radio button label set which day the job will run (for example, "third Wednesday," "second Tuesday," etc.). The third field of the label sets the month that the job will run.

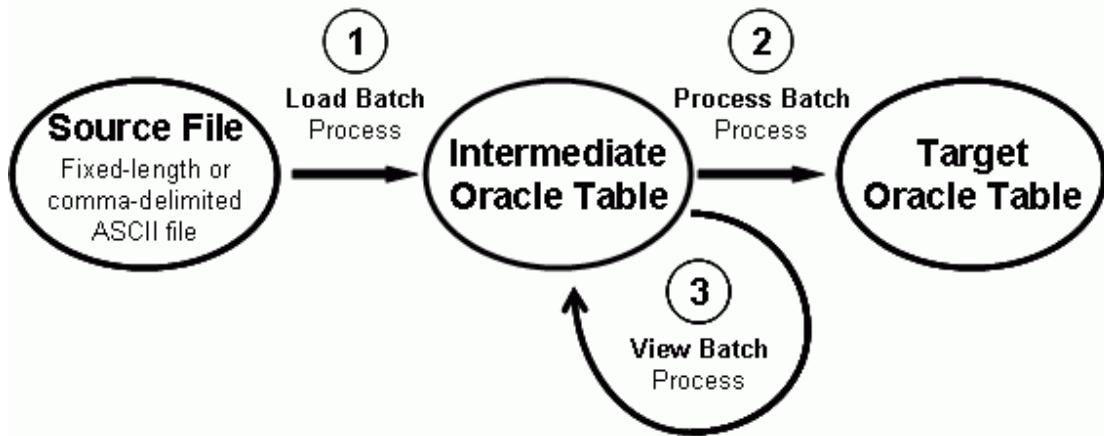
For example, if you configure the label to read, "The Fourth Friday of every 12-December," the job would run on the fourth Friday of December of every year.

## 4.35. Import

The Import feature allows you to:

- Import data from fixed-length ASCII text files, comma-separated (CSV) ASCII text files, Oracle database tables, or Microsoft Excel files into the application.
- Configure import processes.

Importing data is a three-step process involving three files as shown in the figure below. The Load Batch and Process Batch processes move (that is, copy) data from one file to the next. The View Batch process identifies which records have error messages (which means that these records were not moved to the target table).



#### NOTE

Importing normally occurs on an on-going basis. The data that you are importing is called a "batch." A source file is a batch. Also, data is grouped by batch in the intermediate Oracle table. Once data is processed into the target Oracle table, it loses its batch designation.

#### **4.35.1. Description of the Import Window**

The Import window contains three tabs:

- All Imports – This tab displays each import process.
- Import's Setup – This tab displays the record layout of the input file and its Oracle intermediate table structure. This information is set during software configuration, after which it cannot be edited.
- Import's Batches – This tab is where users perform import of data into the system and which shows the history of all previous import attempts.

These tabs are described in more detail in the following sections.

#### **All Imports Tab**

(System > Tools > Import)

* IMPORT NAME	* Source Data Type	* Table Name	DB Procedure	Encoding	Has header row?	Import
APCS Flex import	Comma separated	CA_APACS_RAW_FLEX_IMP	proc_conv_apcs_raw_flex		<input checked="" type="checkbox"/>	
APCS JPC to Staging Import	Comma separated	CA_APACS_RAW_JPC_IMP	proc_conv_apcs_raw_jpc		<input checked="" type="checkbox"/>	
APCS Profiler to Staging Import	Comma separated	CA_APACS_RAW_PROF_IMP	proc_conv_apcs_raw_profiler		<input checked="" type="checkbox"/>	
AsBuiltData	Comma separated	CA_CONV_ASBUILT	proc_conv_asbuilt		<input checked="" type="checkbox"/>	
GPR Import	Comma separated	CA_CONV_GPR	proc_conv_gpr_raw		<input type="checkbox"/>	

The All Imports tab of the Import window lists the imports that are currently available for use or are under development. Usually, a user of the import feature will only utilize this tab to select the import process that he or she wants to employ. One record exists for each type of import. **Information in this tab should only be adjusted by the system administrator who is responsible for setting up new import processes.**

The following columns are shown in this tab:

- Import Name — This column provides a description of the import process.
- Source Data Type — This column specifies the overall format of the input file for the import process. This format may either be fixed length (which is an ASCII text file of fixed length format), comma separated (which is an ASCII text file that is comma delimited — that is, CSV formatted), database table, or Excel file.

Note: The database table must be in Oracle (or other system with ODBC connection) with SELECT connection to the application schema and that the column order must be irrelevant.

- Table Name — This column shows the name of the intermediate Oracle table for this import process. It must be entered in upper case without blank spaces.
- Proc Name — This column shows the name of the Oracle procedure that will be used in the import process. This procedure validates the information in the intermediate Oracle table and then moves the error-free records from the intermediate table into the application.
- Encoding — This column contains the AgileAssets-supplied text that identifies the coding system to use when data in the input file uses a non-standard coding system (for example, Hebrew, Arabic, Chinese dialects, etc.). When the input file uses a standard coding system, this column is left blank.
- Has Header Row? — When the check box in this column is selected, the first row in the input file is a header row.
- Import Location Mapping — This column provides a drop-down list that includes the different methods used to translate a client's linear reference system to AgileAssets' linear reference system. The method selected in this column is the method used during the import process. The different methods are defined and maintained in the Import Location Mapping window.

#### **Import's Setup Tab**

##### **NOTE**

This tab is only to be used by the system administrator responsible for setting up new import processes.

* Order	* COLUMN NAME	* Data Type	DATA PRECISION	DATA SCALE	COLUMN FORMAT	Target Column	Mapping Type (for list columns)	Nullable?
1	STATE	Number		0				
2	EFF_DATE	Date			yyyyMMdd			
3	UNIT_SYSTEM	String		2				
4	District	String		2				
5	County	String		3				
6	Route	String		4				
7	Post_Mile	String		9				
8	Direction	String		2				
9	Lane	String		2				
10	Latitude	Number		13	8			
11	Longitude	Number		13	8			
12	Elevation	Number		6	1			
13	LaneODO	Number		8	1			
14	Filename	String		8				
15	DS_Begin	Number		12	5			
16	DS_End	Number		12	5			

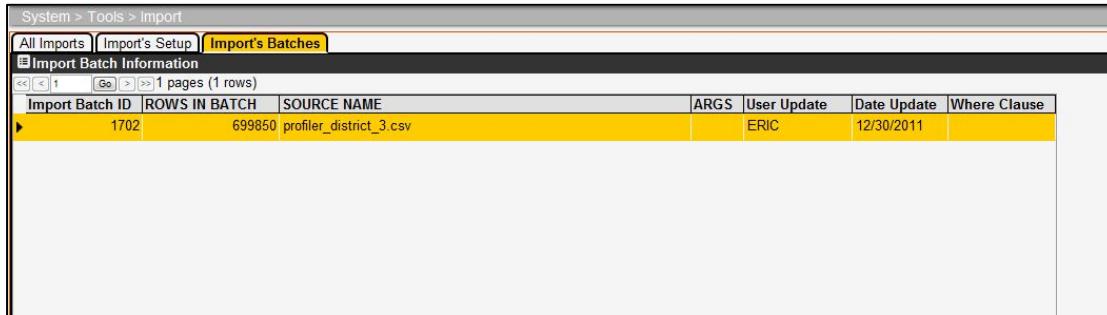
For the import process selected in the All Imports tab, the Import's Setup tab describes the input file's record layout as well as the structure of the intermediate Oracle table. In this tab you can define and modify the record layout and the Oracle column descriptions. One record exists for each column (field). See How to Set Up an Import Process on page 206 for more information.

This tab provides the following columns:

- Order — This column describes the position of the data column within the input file.
- Column Name — This column shows the Oracle intermediate table column name. It must be entered in upper case without blank spaces or hyphens. It also cannot start with a number nor contain Oracle-reserved words.
- Data Type — This column describes the data type (either string, number, or date) for the column for both the input data and the Oracle intermediate table.
- Data Precision — This column shows the length of the column in the Oracle intermediate table.
- Data Scale — This column shows the number of decimal places for numeric data type columns in the Oracle intermediate table.
- Column (Date) Format — This column shows the date format for all date data type columns in the input file. See page 204 for more information on formats.
- Target Column — This column provides a drop-down list from which you may select the existing column in the AgileAssets database that corresponds to the column in the input file.
- Mapping Type — For a given field to be imported, this column identifies the column in the Setup window that contains the set of legal codes used in the import source file. The following are the available mapping types and what each means:
  - By code (internal ID) — This is the internal ID# in the Setup table for the given target column. Select this value only if directed to do so by AgileAssets.
  - By comments — This is the Comments column in the Setup window for the given target column.
  - By name — This is the Name column in the Setup window for the given target column.

- Data Source Length – If the Source Data Type is fixed-length (as identified in the All Imports tab), this column is displayed and shows the length of the data column in the input file.

### ***Import's Batches Tab***



Import Batch Information						
Import Batch ID	ROWS IN BATCH	SOURCE NAME	ARGS	User Update	Date Update	Where Clause
1702	699850	profiler_district_3.csv		ERIC	12/30/2011	

For the import process selected in the All Imports tab, the Import's Batches tab is used to initiate the import data process and also contains a log of all import attempts (batches). (A batch is a set of data that is imported into the system.)

The import process generally occurs as follows:

1. The desired import process is selected in the All Imports tab.
2. In the Import's Batches tab, the batch starts as an input file.
3. The user points to the tab, right-clicks, and clicks **Insert** to copy the batch into the intermediate Oracle table.
4. The user points to the tab, right-clicks, and clicks **Process Batch** to move all valid data records into the system from the intermediate Oracle table, with invalid records remaining in the intermediate Oracle file for review.

#### **Description of the Columns in the Import's Batches Tab**

The Import's Batches tab contains the following columns:

- Rows in Batch – This column identifies how many records remain in the batch (that is, in the intermediate Oracle table). Before the **Process Batch** command is run, this number is the total number of records in the input file. After the **Process Batch** command is run, this number is the number of invalid records that remain (the valid records having been moved into the system).
- Source Name – This column shows the input file name for the batch including its subdirectories.

#### **Description of the Right-click Shortcut Menu Commands**

When you right-click a record in this tab, a shortcut menu is displayed. This menu shows the common commands along with the following special commands:

- **Insert** – This command loads data from the user-selected input file into the batch (the intermediate Oracle table). If this load is successful, a batch record is added to the table.
- **Reimport Batch** – This command performs the same actions as the **Insert** command except that the result that goes into the intermediate Oracle table overwrites the currently selected batch.
- **View Batch** – This command allows you to see the data in the batch as well as the associated error messages for invalid records.

- **Process Batch** – This command moves all error-free records from the intermediate Oracle table to the application's target table. This command processes the batch by:
  1. Validating all records in the batch and attaching error messages to the invalid records.
  2. Moving all valid records into the application.
  3. Removing the valid records from the batch (the intermediate Oracle table).

For further information, see How to Import on page 209.

#### **4.35.2. Date-type Column Formats**

All date-type columns in the Import's Setup tab must have a proper column format. Examples of proper formats are shown below:

- MM/dd/yyyy
- MM-dd-yyyy
- yyyy/MM/dd
- yyyy/MM/dd HH:mm:ss

**Note:** Data column formats are case sensitive.

#### ***Date and Time Patterns***

Date and time formats are specified by date and time pattern strings. Within date and time pattern strings, unquoted letters from 'A' to 'Z' and from 'a' to 'z' are interpreted as pattern letters representing the components of a date or time string. All other characters are not interpreted; they're simply matched against the input string during parsing. See page 206 for the list of pattern letters.

Pattern letters are usually repeated, as their number determines the exact presentation. The following table lists pattern letters for different types of data.

Data Type	Pattern for Formatting	Pattern for Parsing
Text	If the number of pattern letters is four or more, the full form is used; otherwise, a short or abbreviated form is used (if available).	Both forms are accepted — independent of the number of pattern letters.
Number	The number of pattern letters is the minimum number of digits, and shorter numbers are zero-padded to this amount.	The number of pattern letters is ignored unless it's needed to separate two adjacent fields.

Data Type	Pattern for Formatting	Pattern for Parsing
Year	If the number of pattern letters is two, the year is truncated to two digits; otherwise it is interpreted as a number.	If the number of pattern letters is more than two, the year is interpreted literally regardless of the number of digits. So using the pattern "MM/dd/yyyy", "01/11/12" parses to Jan 11, 12 A.D.  For parsing with the abbreviated year pattern ("yy"), the abbreviated year relative to some century must be interpreted. It is done by adjusting dates to be within 80 years before and 20 years after the current time. For example, using a pattern of "MM/dd/yy" and assuming it was created on Jan 1, 1997, the string "01/11/12" would be interpreted as Jan 11, 2012 while the string "05/04/64" would be interpreted as May 4, 1964. During parsing, only strings consisting of exactly two digits, will be parsed into the default century. Any other numeric string, such as a one-digit string, a three-digit (or more) string, or a two-digit string that isn't all digits (for example, "-1"), is interpreted literally. So "01/02/3" or "01/02/003" are parsed, using the same pattern, as Jan 2, 3 AD. Likewise, "01/02/-3" is parsed as Jan 2, 4 BC.
Month	If the number of pattern letters is three or more, the month is interpreted as text; otherwise, it is interpreted as a number.	
General Time Zone	Time zones are interpreted as text if they have names. For time zones representing a GMT offset value, the following syntax is used (the format is locale independent and digits must be taken from the Basic Latin block of the Unicode standard):  <pre> GMTOffsetTimeZone: GMT Sign Hours:Minutes     + or -                   +--&gt; Sign                   +--&gt; Hours:Minutes                           +--&gt; Hours             +--&gt; Minutes                                   +--&gt; 0 to 23                 +--&gt; 00 to 59   +--&gt; one or two digits                     +--&gt; must be two digits     </pre>	For parsing, RFC 822 time zones are also accepted.
RFC822 Time Zone	For formatting, the RFC822 four-digit time zone format is used:  <pre> RFC822TimeZone: Sign Hours:Minutes     + or -                   +--&gt; Sign                   +--&gt; Hours:Minutes                           +--&gt; Hours             +--&gt; Minutes                                   +--&gt; 00 to 23                 +--&gt; 00 to 59   +--&gt; must be two digits                     +--&gt; must be two digits     </pre>	For parsing, general time zones are also accepted.

### Examples

The following examples show how date and time patterns are interpreted in the U.S. locale. The given date and time is 2001-07-04 12:08:56 local time in the U.S. Pacific time zone.

Date and Time Pattern	Result
yyyy.MM.dd G 'at' HH:mm:ss z	2001.07.04 AD at 12:08:56 PDT
EEE, MMM d, "yy	Wed, Jul 4, '01
h:mm a	12:08 PM

hh 'o'clock' a, zzzz	12 o'clock PM, Pacific Daylight Time
K:mm a, z	0:08 PM, PDT
yyyy.MMMM.dd GGG hh:mm aaa	02001.July.04 AD 12:08 PM
EEE, d MMM yyyy HH:mm:ss Z	Wed, 4 Jul 2001 12:08:56 -0700
yyMMddHHmmssZ	010704120856-0700

Note: As shown in the examples above, to avoid interpretation problems text can be quoted using single quotes (').

#### **Pattern Letters**

Letter	Date or Time Component	Presentation	Examples
G	Era designator	Text	AD
y	Year	Year	1996; 96
M	Month in year	Month	July; Jul; 07
w	Week in year	Number	27
W	Week in month	Number	2
D	Day in year	Number	189
d	Day in month	Number	10
F	Day of week in month	Number	2
E	Day in week	Text	Tuesday; Tue
a	Am/pm marker	Text	PM
H	Hour in day (0-23)	Number	0
k	Hour in day (1-24)	Number	24
K	Hour in am/pm (0-11)	Number	0
h	Hour in am/pm (1-12)	Number	12
m	Minute in hour	Number	30
s	Second in minute	Number	55
S	Millisecond	Number	978
z	Time zone	General time zone	Pacific Standard Time; PST; GMT-08:00
Z	Time zone	RFC 822 time zone	-0800

#### **4.35.3. How to Set Up an Import Process**

Three, broad stages are needed to establish an import process:

- Planning the import process (outside of the application).

- Developing the import process (inside the application).
- Troubleshooting.

### ***Plan the Import Process***

The first stage in setting up an import process is to gather the necessary information for the process (usually with the assistance of AgileAssets support staff):

1. Identify the data to be imported.
2. Create a process that converts the data to a fixed-length ASCII file or a comma-delimited (CSV) ASCII file. These two types of files are the only types that are acceptable.
3. Identify all data validity checks and data formatting checks that are needed.
4. Identify the target table(s) in the application into which the data will be imported.
5. Identify the crosswalk data transformations from the data columns of the input file into the columns of the target table.
6. Write the validity-checking procedure.
7. Create one or more test input data files. At least one test file should contain enough errors to verify all data validity checks that were requested to be in the procedure.

### ***Develop the Import Process***

Once the import process is planned and you have the procedure name from AgileAssets, develop the import process:

1. Launch the application, log on, and navigate to the Import window.
2. In the All Imports tab of the Import window, right-click the table and then click **Insert**. The system will add a new row to the table.
3. In the Import Name column of the new row, type the name of the import process.
4. In the Source Data Type column, click the down arrow and then click the type of file.
5. Press the tab key to move the cursor to the Table Name column and type the name of the target table. This name must be in upper case, without blanks.
6. Press the tab key to move to the Proc Name column and type the procedure name provided by AgileAssets.
7. If the input file has a header row, click the check box in the Has Header Row? column.
8. Click the Import's Setup tab.
9. Place the cursor anywhere below the column heads, right-click, and then click **Insert**. A new record is added to the table. Note the value in the Order column. The number shown here corresponds to the input file field (that is, a 1 signifies the first field, a 2 the second field, and so on.) **Do not edit the value in the Order column.**
10. In the Column Name column of the new row, create a column name (in upper case and without blanks) that describes the data in the input file field that corresponds to the value in the Order column.
11. In the Data Type column, click the down arrow and then click the data type in the input file field that corresponds to the value in the Order column.
12. For numeric- and string-type data, tab to the Data Precision column and type the maximum length of the data in the input file field that corresponds to the value in the

Order column. This information is required and is used for sizing the columns in the intermediate Oracle table. For date-type columns, this column may be left blank.

13. For numeric-type data, tab to the Data Scale column and type the value which is the maximum number of decimal places in the input file field that corresponds to the value in the Order column. For other data types, leave this column blank.
14. For date-type data, tab to the Column Format column and type a value which describes the data formatting in the input file field that corresponds to the value in the Order column. See page 204 for more information on the allowed data formats for dates. For other data types, leave this column blank.
15. If the type of import process is for a fixed-length ASCII file, the Data Source Length column will be displayed. Tab to this column and type the length of the data in the fixed-length type input file.

Note: Do not skip characters in the description of a fixed-length type input file. This means that the columns defined in this tab must cover every character in the fixed-length input file record. Consequently, if data exists in the input file that you do not want to import, you should create a dummy column for it in this table and then instruct AgileAssets to write the import procedure to ignore this dummy data.

16. Click  . The import process is now ready for testing and use. See page 212209 for instructions on importing data.

### Troubleshoot

After attempting to import data, the import process may not work as you expected. Consider the following when attempting to resolve the unexpected results:

- If the Import Set-up was incorrect, then the **Insert** command in the Import's Setup tab will fail for almost every input file record. Suggestions for identifying the error source are shown in the table below.
- If the Import Set-up is correct, and you have not yet run the **Process Batch** command, then the data in the intermediate Oracle table will mirror that of the input file.
- If the Import Set-up is correct, and you have run the **Process Batch** command, then data in the intermediate Oracle table will exhibit all data consistency errors.

Potential Source of Error							
Type of Error	Source Data Type	Data Source Length	Column Format	Short Data Precision	Short Data Scale	Column Data Type	Order
All data types <sup>1</sup>	✓	✓					
Numeric column <sup>1</sup>				✓	✓	✓	✓
String column <sup>1</sup>		✓		✓			
Date column <sup>1</sup>		✓	✓			✓	✓

Potential Source of Error					
Type of Error	Oracle Table Name	Oracle Column Name	Oracle Proc Name	Column Data Type	Order

"Table Exists" Oracle error <sup>1</sup>	✓				
"Cannot Create Table" Oracle error <sup>1</sup>	✓	✓			
"Procedure Doesn't Exist" Oracle error <sup>2</sup>			✓		
Other Oracle error <sup>2</sup>				✓	✓

Notes:

1. Error occurs when you select **Insert** in the Import's Batches tab.
2. Error occurs when you select **Process Batch** in the Import's Batches tab.

#### **4.35.4. How to Import Data with the Import Window**

To import data, follow these steps:

1. Open the Import window.
2. On the All Imports tab, click the import template you wish to use.
3. Click the Import's Batches tab.
4. Right-click in the Import's Batches tab and then click **Insert** from the shortcut menu that is displayed. The system displays a dialog box to select the file to be imported.
5. Click the **Browse** button. The system displays a typical file explorer window.
6. In the file explorer window, locate the file to be imported and then double-click the file name to display the file name in the dialog box.
7. Click **Upload**. The system imports the data into the temporary, intermediate table. When the import process is finished, a new record appears in the Import's Batches tab.
8. Right-click the new record and then click **View Batch** from the shortcut menu that is displayed. The system displays a new window that shows the records that were loaded from the source table. If any format errors were detected, they are shown in the Import Err String column. You must correct the errors before continuing.
9. Provided that no errors were detected, close the Batch View window.
10. In the Import's Batches tab, right-click the new record and then click **Process Batch** from the shortcut menu that is displayed. This command executes the data validation and transfer procedure for the import.
11. After the **Process Batch** command has finished, right-click the new record and then click **View Batch** from the shortcut menu that is displayed. The system displays a new window that shows either no records (if all records are error-free) or the processed records that have errors.

If no records are shown, close the window. The import process is now complete.

If records with errors are shown, you may recover in either of the following ways:

- o Correct the errors in the View Batch window, close the window, and then return to step 10 to continue; or
- o Use the contents of the window to identify the records in your original file and correct the errors in the original file. Once the records in the original file are corrected, return to the Agile Pavement Manager system and, in the Import's

Batches tab, right-click the record showing the last import attempt and then click **Reimport Batch**. The system again asks you for the data source and then re-imports the data, displaying a new record in the Import's Batches tab. Return to step 6 to continue.

## 4.36. Import Runner

Some implementations include the Import Runner window (System > Tools > Import Runner). This window allows you to manage your batch files. You can create folders, upload files, and delete files. Once batch files are uploaded, you can then process them.

## NOTE

The import routines are created in the Tables window and appear in this window when the Is Public check box is selected for the routine.

#### **4.36.1. Description of the Import Window**

The Import Runner window contains three tabs:

- Imports – This tab displays each "public" import routine.
  - Import Detail – This tab displays the record layout of the input file and its Oracle intermediate table structure. This information is set during software configuration, after which it cannot be edited.
  - Import Log – This tab is where users perform the import of data into the system and which shows the history of all previous import attempts.

These tabs are described in more detail on the following pages.

## **Imports Tab**

System > Tools > Import Runner	
Imports	Import Detail
Target Table	
<<	1
(Go)	>>
2 pages (20 rows)	
Import Config Name	Target Table Name
Bridges Data Import	PMS_CA_PONTIS_BRIDGES
CAS Contract Items Import	PMS_CA_EWB_CONTRACT_ITEMS
CAS Projects Import I	PMS_CA_EWB_CURR_PROJECTS
CAS Projects Import II	PMS_CA_EWB_CURR_PROJECTS
Culvert Inspection Import	PMS_CA_CULVERT_INSPECTION
Culvert Inventory Import No Loc	P_CULVERT_INVENTORY
End Treatment Assessment Import ( Culvert)	PMS_CA_ENDTR_ASSESSMENT
End Treatment Import No Loc (Culvert)	PMS_CA_END_TREATMENT
FlexData (APCS)	PMS_CA_APACS_FLEX
GIS Precision Import (Culvert)	PMS_CA_GIS_PRECISION
Import from SETUP_CA_CLIMATE_REGIONS	SETUP_CA_CLIMATE_REG_ARCH
JPC Data (APCS)	PMS_CA_APACS_JPCP
No LOC Culverts System Import	P_CULVERT_SYS_INVENTORY
OE Data Import	PMS_CA_OE_DATA
Profile Import ( APCS)	PMS_CA_APACS_PROFILE
PRSM Data Import	PMS_CA_PRSM

The Imports tab of the Import Runner window lists the "public" imports that are currently available for use. It also shows the target table into which the imported data will be placed.

When you right-click a record in this tab, a shortcut menu is displayed. This menu shows the common commands along with the following special command:

- **Edit Config** – This command displays the dialog box for configuring the import routine.

### **Import Detail Tab**

The screenshot shows the 'Import Detail' tab in the PaveM application. It has two main panes:

- Import Columns:** A grid showing columns from a data source. The columns listed are: DISTRICT, COUNTY, FACILITY\_CARRIED, LOCATION, DIR, LAST\_INSP, YEAR\_BUILT, YEAR\_CONSTRUCTED, LENGTH, DECK\_AREA, LATITUDE, LONGITUDE, BRIDGE\_NO, BRIDGE\_NAME, and ROUTE\_N. The first column is labeled 'Source Column ID' and the second is 'Order + Comments'. The 'DISTRICT' column is highlighted in yellow.
- Import Column Mapping:** A grid showing the mapping from the data source columns to target table columns. The columns listed are: BRIDGE\_NAME, YEAR\_CONSTRUCTED, DECK\_AREA, YEAR\_BUILT, DISTRICT, FACILITY\_CARRIED, LENGTH, LOCATION, LOCATION, LOCATION, LOCATION, LOCATION, LOCATION, LOCATION, and LOCATION. The 'Import Mapping Arg List' and 'Import Mapping Script' columns show the mapping logic. The 'Target Column' column lists the target table columns: BRIDGE\_NAME (Bridge Name), YEAR\_CONSTR (Year Built), CA\_DECK\_AREA (Deck Area), CA\_YEAR\_BUILT (Year Built), COMMENT\_ID (Att.), COMMENT\_STR (Comments), DATE\_UPDATE (Date Update), DISTRICT\_ID (District), FACILITY\_CARRIED (007 FACILITY CARRIED), LENGTH (Length), LOC\_DESC (Location Desc), LOC\_IDENT (Location ID#), CA\_ROUTE\_FROM (Route), CA\_COUNTY\_FROM (County), CA\_PM\_PREFIX\_FROM (Post Mile Prefix From), and CA\_PM\_SUFFIX\_FROM (Post Mile Suffix From). The 'Order #' column shows the sequence of mappings.

For the import routine selected in the Imports tab, the Import Detail tab provides the mapping from the columns of the data source to the columns of the target table. This tab contains two panes:

- Import Columns – This pane shows the columns in the data source.
- Import Column Mapping – This pane shows the mapping from one or more columns in the data source to a column in the target table.

The information shown in this tab is read-only. If you need to modify any information shown in this tab, use the **Edit Config** command found in the Imports tab.

### **Import Log Tab**

The screenshot shows the 'Import Log' tab in the PaveM application. It has a single pane displaying a log entry:

Log ID	Start Time	Finish Time	Records Imported	Records with Errors	Results File	Fatal Error	Comments	Att.	User Update	Date Update	Administrative Unit
829	12/20/2011	12/20/2011	11742	1205	Yes			EUGENE		12/20/2011	

For the import process selected in the Imports tab, the Import Log tab shows previous attempts (if any) at importing data.

When you right-click this tab, a shortcut menu is displayed. This menu shows the common commands along with the following special commands:

- **Import New Data** – This command loads data from the server where the file containing the data is located to the target table in the application.
- **Manage Files** – This command allows you to move the desired files to the server in preparation for importing.

- **Download File** — After importing data, this command allows you to download the error file for the import process to your computer.

#### **4.36.2. How to Import Data with the Import Runner Window**

Importing data via the Import Runner window is accomplished in two parts. First the file containing the data to be imported is placed on the server that is used for importing data. Then the data is imported from the file on the server to the target table. These tasks are described in more detail in the following sections.

##### ***Make the Data Files Available for Importing***

The first task is to place the files containing the data to be imported on the server that is used for importing data. This is accomplished as follows:

1. Open the Import Runner window.
2. Click the Import Log tab.
3. Right-click the Import Log tab and then click **Manage Files**. The system displays a dialog box for selecting files. This dialog box shows the root folder structure for your agency (other agencies cannot see this structure).
4. If necessary, click the hyperlink for the folder into which the file containing the data to be imported will be placed. This opens the folder. (If you "go too deep," you can return to a higher level in the directory by clicking the hyperlink beside the up-pointing arrow at the top of the table.)

Note: If the desired folder does not exist, you may create it by right-clicking the dialog box and then clicking **Make Directory** from the shortcut menu. The system displays a new dialog box so you may label the new folder and then displays the new folder in the directory. You may then open the new folder by clicking it.

5. Once the correct folder is open, right-click the dialog box and then click **Upload File**. The system displays a dialog box to locate the desired file.
6. Click the **Browse** button. The system displays a typical file explorer window.
7. In the file explorer window, locate the file to be imported and then double-click the file name to display the file name in the dialog box.
8. Click the **Upload** button. The system places the selected file in the opened folder.
9. You may upload additional files in preparation for importing or close the dialog box by clicking the **Close** button.

##### ***Import the Data into the Target Table***

Once the file containing the data to be imported is placed in your agency's root directory structure, you may then import the data into the target table. This is accomplished by performing the following steps:

1. Open the Import Runner window.
2. On the Imports tab, click the import process you wish to perform.
3. Click the Import Log tab.
4. Right-click the Import Log tab and then click **Import New Data**. The system displays a dialog box that shows your agency's root directory structure.
5. Locate the file that contains the data to be imported. You open a folder by clicking the hyperlinked folder name shown beside the folder icon. If necessary, you can return to

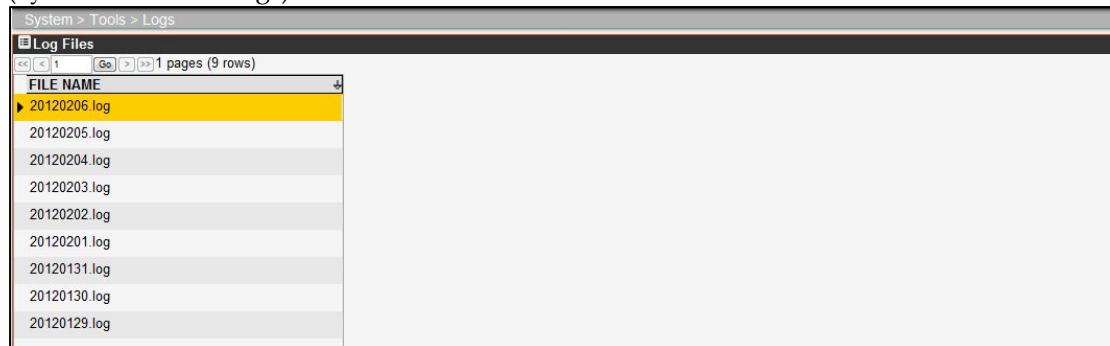
a higher level of the directory structure by clicking the hyperlink beside the up-pointing arrow at the top of the table.

6. Once the desired file is found, click the square shown beside the name of the file. The system places a check mark in the square to denote that it is selected.
7. Click the **OK** button. The system imports the data from the selected file to the target table in the application. The system also shows a window that indicates the progress of the import process.
8. Once the import process is finished, the system displays a summary of the import process. Click **OK** to close the summary. This completes the import process.

Note: If errors occurred during the import process, the system will prompt you to download the error file to your computer. You may then use this file to diagnose the source of the error. Generally, errors must be corrected in the input file. Once the errors are corrected in this file, you will need to copy the file to the server as described in the previous section and then re-import the data as described in this section.

#### 4.37. System Logs

(System > Tools > Logs)



The purpose of the System Logs window is to capture error statements that are transmitted by the system. These logs are then sent to AgileAssets support staff to aid in their debugging efforts.

When you report a problem to AgileAssets you may be instructed to "turn on the log" and then send the results to AgileAssets. This is accomplished by setting the Logging Level parameter in the System Parameters window to a value other than None (see page 124 for more information on the System Parameters window). The application will then log all errors that occur in the system. The logs are then listed in this window, the System Logs window.

You open a particular log by first downloading a log to your computer by using the **Download File** right-click command. Then, once the log file is downloaded, you can open it using a tool such as Notepad or send it to AgileAssets as instructed.

## 4.38. Notifications

(System > Tools > Notifications)

The screenshot shows the Notifications window with two main panes:

- System Alerts / News**: A grid of messages. One message is highlighted with a yellow background and border. The columns are: \*Notification Type, Effective Date, Expiration Date, Title, \*Text, Comments, User Update, Date Update, Alert.
- Target Users List**: A grid of users. One user is highlighted with a yellow background and border. The columns are: User ID, Date Read, Date Sent.

System Alerts / News								
* Notification Type	Effective Date	Expiration Date	Title	* Text	Comments	User Update	Date Update	Alert
General	2/1/2012 9:13:00	2/1/2012 9:15:00	Staff Meeting	Staff meeting at noon in Room	TODD	2/1/2012	<input type="checkbox"/>	
General	1/31/2012 15:32:00	1/31/2012 15:40:00	Time Sheets Due	Time sheets are due at the end of the month	TODD	1/31/2012	<input checked="" type="checkbox"/>	
General	1/31/2012 15:00:00	1/31/2012 15:03:00	System interruption	TAMS will be down for maintenance	TODD	1/31/2012	<input checked="" type="checkbox"/>	
General	4/27/2011 0:00:00	4/27/2011 12:00:00	We are having errors on the system	We are having errors on the system	VMAIN	4/27/2011	<input type="checkbox"/>	
General	4/13/2011 7:00:00	4/13/2011 7:33:00		TAMS is going down for maintenance	VMAIN	4/13/2011	<input type="checkbox"/>	
General	12/17/2010 0:00:00	12/24/2010 0:00:00	Welcome	Welcome to TAMS. This system	VMAIN	12/17/2010	<input type="checkbox"/>	
General	11/30/2010 11:45:00	11/30/2010 12:10:00	System interruption	TAMS will be down for maintenance	VMAIN	11/30/2010	<input type="checkbox"/>	

Target Users List		
User ID	Date Read	Date Sent
HIMANSHU	11/15/2010	11/15/2010 11:20:50

You use the Notifications window to create messages that are delivered at a certain time (unless the Alert check box is selected, in which case the message is sent immediately). (Other messages may be configured to be sent when a certain action occurs; these are configured in the Notification Types window; see page 215.)

The messages created in this window are termed General messages. The messages are sent to the users listed in the Target Users pane or, if that person so designates in the Set Substitutes for Notifications window (see page 216), another recipient.

The following sections describe the two panes found in the Notifications window.

### 4.38.1. System Alerts/News Pane

The System Alerts/News pane is where new messages are created. It also shows you previous messages. This pane contains the following columns:

- Effective Date — This is when the message will be sent. After saving the new message, when this date and time is reached (and allowing for the alert polling interval set in the System Parameters window by the Alerts Checking Int. parameter), the message is sent to all users listed in the Target Users pane.

When the message arrives, the system displays a hyperlinked announcement of a new message in the left gutter. When you click the hyperlink, the system displays the Messages window, from which you can view the message.

Note: If you double-click the field, the system displays a calendar pop-up window with a time field. Set the time before selecting the date because when you click a date the calendar pop-up window closes.

- Expiration Date — This is when the message expires. After this date and time, if a user has not read the message, the system will remove the announcement of the message from the user's left gutter and the message will not be shown in the user's Messages window. As with the Effective Date field, double-clicking the field will display a calendar pop-up window.
- Title — This is the title of the message.
- Text — This is the message.
- Alert — When this check box is selected, the message is sent as soon as a recipient is added to the Target Users pane and saved. The message will appear in a pop-up window on the recipient's computer after the amount of time set by the Alerts Checking Int. parameter (which is found in the System Parameters window). The

message will also be available in the Messages window until it is marked as read. (If the user is not logged on when the alert is sent, the system will display the message when he or she logs on.)

The pop-up window that shows the message contains two command buttons: **Read?** and **Close**. When you click the **Read?** button, the system places a check mark in the Is Read? check box of the pop-up window. You may then click the **Close** button to close the pop-up window. (If you only click the **Close** button, the system will continue to re-display the alert pop-up window until you click the **Read?** button.)

Note: By clicking the **Read?** button, you essentially delete the alert message because read messages are not shown in the Messages window.

Note: You do not need to set an Effective Date or Expiration Date for alerts because the system sends the message immediately and will continue to send it until the recipient marks it as read. However, if you do set these fields, then the system will only send the message within the time period set by the fields.

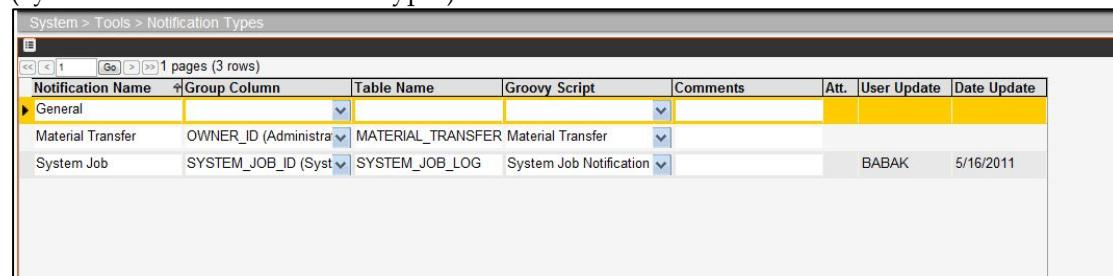
#### **4.38.2. Target Users Pane**

The Target Users pane shows to whom the message selected in the Message pane will be sent. It also indicates whether a recipient has read the message.

You add users to this pane by right-clicking the pane and then clicking the **Insert** command. The system then displays a dialog box with all users that are not yet selected as recipients of the message.

### **4.39. Notification Types**

(System > Tools > Notification Types)



System > Tools > Notification Types							
Notification Name	Group Column	Table Name	Groovy Script	Comments	Att.	User Update	Date Update
General							
Material Transfer	OWNER_ID (Administra	MATERIAL_TRANSFER	Material Transfer				
System Job	SYSTEM_JOB_ID (Syst	SYSTEM_JOB_LOG	System Job Notification		BABAK	5/16/2011	

You use the Notification Types window to configure messages that are sent when a certain action occurs. (You may also configure messages that are sent at a certain time; this type of message [called a General message] is configured in the Notifications window; see page 214.)

The different types of notifications vary depending on the needs of the agency using the AgileAssets system. Two typical types utilized by several agencies are described below:

- Material Transfer – This type generates messages when a material transfer occurs, that is when a request for stock transfer is made and when the stock to be transferred is “fulfilled” (sent from the warehouse that had the stock originally).
- Work Request – This type sends a message to the user(s) who will be doing the work described in the work request when the work request is inserted or the owner assigned is changed.

#### **4.39.1. Description of the Columns**

The following columns are found in this window:

- Notification Name – This column shows the label for the notification type.

Note: The General type is by default always present. However, it has no function in this window and so the fields for this type are left empty. Its use is in the Notifications window.

- Group Column – This column shows the group of users to whom the message will be sent. You may modify the message's recipients by using the **Edit Grouping** command, which is found on the shortcut menu that is displayed by right-clicking the record.
- Table Name – This column shows the name of the table in which a data change will cause the message to be sent (based upon the criteria in the Groovy script).
- Groovy Script – This column contains the name of the Groovy script that determines when the message will be sent and also what the message will contain. The Groovy script is selected from a drop-down list, the contents of which are all Groovy scripts of the AfterChange Notification Trigger type.

#### **4.39.2. Description of the Right-click Shortcut Menu Commands**

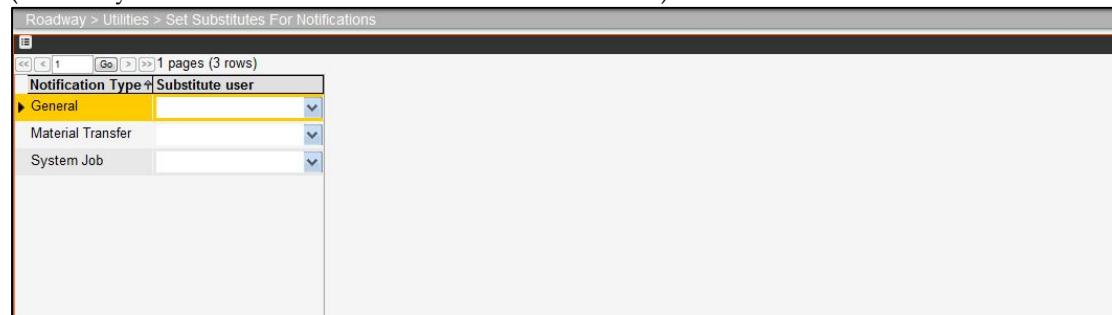
When you right-click a record, the system displays a shortcut menu. This menu contains the common commands along with the following commands:

- **Create/Edit Script** – This command is only available when the Groovy Script field is active. When enabled, this command displays the Groovy Script dialog box; see page 191 for more information on this dialog box.
- **Edit Grouping** – When you select this command, the system displays a dialog box showing the users who are part of the group. You may then edit who belongs to the group and save the new data.

Note: Some versions of the AgileAssets system allow a user to designate an alternate user to receive his or her messages. This is configured in the [Set Substitutes for Notifications](#) window.

### **4.40. Set Substitutes for Notifications**

(Roadway > Utilities > Set Substitutes For Notifications)



The screenshot shows a table with two columns: 'Notification Type' and 'Substitute user'. The 'Notification Type' column lists 'General', 'Material Transfer', and 'System Job'. The 'Substitute user' column contains dropdown menus. The 'General' row has its dropdown open, showing it is currently set to 'Substitute user'. The other two rows have their dropdowns closed.

Notification Type	Substitute user
General	Substitute user
Material Transfer	
System Job	

The Set Substitutes for Notifications provides a means for a user to designate an alternate recipient for his or her messages for each notification type. For example, this feature allows a user to designate another person to handle his or her messages while he or she is on vacation. This window is found in each module (in the Utilities menu), which allows a user to configure different alternate recipients for module.

The Set Substitutes for Notifications window lists all the notification types configured in the Notification Types window (see page 215). For each type, an alternate user is selected from the drop-down list of users. Once this information is set and saved, messages will be sent to both the original recipient as well as the people he or she designated. When the original user no longer wants the alternate person to receive his or her messages, the user sets the Substitute User column to null.

## 4.41. Threads

(System > Tools > Threads)

Thread	Base Class	Creation Date	Status	Start Date	Finish Date	Stoppable	Stopped	User ID	Access Profile	Administrative Unit
▶ 1796	class com.agileassetsinc.main.DataImporter	2/6/2012 14:24:39	Done	2/6/2012 14:24:39	2/6/2012 14:24:41	<input checked="" type="checkbox"/>	<input type="checkbox"/>	ERIC	System Role	California Department
1795	class com.agileassetsinc.main.DataImporter	2/6/2012 14:23:00	Done	2/6/2012 14:23:00	2/6/2012 14:24:08	<input checked="" type="checkbox"/>	<input type="checkbox"/>	ERIC	System Role	California Department
1794	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 14:12:40	Done	2/6/2012 14:12:40	2/6/2012 14:12:42	<input checked="" type="checkbox"/>	<input type="checkbox"/>	KNALLI	System Role	California Department
1793	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 14:09:38	Done	2/6/2012 14:09:38	2/6/2012 14:09:41	<input checked="" type="checkbox"/>	<input type="checkbox"/>	KNALLI	System Role	California Department
1792	class com.agileassetsinc.main.DataImporter	2/6/2012 14:03:58	Done	2/6/2012 14:03:58	2/6/2012 14:04:02	<input checked="" type="checkbox"/>	<input type="checkbox"/>	ERIC	System Role	California Department
1791	class com.agileassetsinc.core.SystemJob	2/6/2012 13:34:53	Done	2/6/2012 13:34:53	2/6/2012 13:34:55	<input type="checkbox"/>	<input type="checkbox"/>	LYN	System Role	DISTRICT 3
1790	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 13:22:56	Done	2/6/2012 13:22:56	2/6/2012 13:23:39	<input checked="" type="checkbox"/>	<input type="checkbox"/>	BRANDON	System Role	California Department
1789	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 12:54:36	Done	2/6/2012 12:54:36	2/6/2012 12:56:07	<input checked="" type="checkbox"/>	<input type="checkbox"/>	BRANDON	System Role	California Department
1788	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 12:47:22	Done	2/6/2012 12:47:22	2/6/2012 12:48:32	<input checked="" type="checkbox"/>	<input type="checkbox"/>	BRANDON	System Role	California Department
1787	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 12:40:57	Done	2/6/2012 12:40:57	2/6/2012 12:42:07	<input checked="" type="checkbox"/>	<input type="checkbox"/>	BRANDON	System Role	California Department
1786	class com.agileassetsinc.core.SystemJob	2/6/2012 12:29:18	Done	2/6/2012 12:29:18	2/6/2012 12:45:02	<input type="checkbox"/>	<input type="checkbox"/>	ERIC	System Role	California Department
1785	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 12:22:44	Done	2/6/2012 12:22:44	2/6/2012 12:28:55	<input checked="" type="checkbox"/>	<input type="checkbox"/>	BRANDON	System Role	California Department
1784	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 12:17:01	Done	2/6/2012 12:17:01	2/6/2012 12:22:35	<input checked="" type="checkbox"/>	<input type="checkbox"/>	BRANDON	System Role	California Department
1783	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 12:07:30	Done	2/6/2012 12:07:30	2/6/2012 12:11:26	<input checked="" type="checkbox"/>	<input type="checkbox"/>	BRANDON	System Role	California Department
1782	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 11:36:35	Done	2/6/2012 11:36:35	2/6/2012 11:37:32	<input checked="" type="checkbox"/>	<input type="checkbox"/>	ERIC	System Role	California Department
1781	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 11:26:42	Done	2/6/2012 11:26:42	2/6/2012 11:28:04	<input checked="" type="checkbox"/>	<input type="checkbox"/>	ERIC	System Role	California Department
1780	class com.agileassetsinc.main.optim.SoverIntegerOptim	2/6/2012 11:19:24	Done	2/6/2012 11:19:24	2/6/2012 11:20:41	<input checked="" type="checkbox"/>	<input type="checkbox"/>	ERIC	System Role	California Department

The Threads window shows all processes that are running (or have run) in the background. The Stoppable column indicates whether a process may be halted. To halt a process, right-click the record showing the process and then click **Stop Thread**. The Stopped column will then show that the process was halted short of completion (which is instead indicated by the Status column).

## 5. Reports

The Reports window provides commands for displaying lists of reports that exist for the selected module. This window provides extensive reporting capabilities. At the broadest level, it supports the following types of reports:

- Tabular reports – This type of ad-hoc report displays data (either detailed or aggregate) in rows and columns.
- Graph reports – This type of ad-hoc report displays data in a graphical format.
- GIS reports – This type of ad-hoc report displays data via a color-coded map.
- GIS Maps – This is a map composed of various layers assembled by a user.
- Jasper reports – This is a “canned” type of report built using open source software and then available for use within the Reports window.

This window organizes access to reports through a user-modifiable “folder” hierarchy that categorizes reports into:

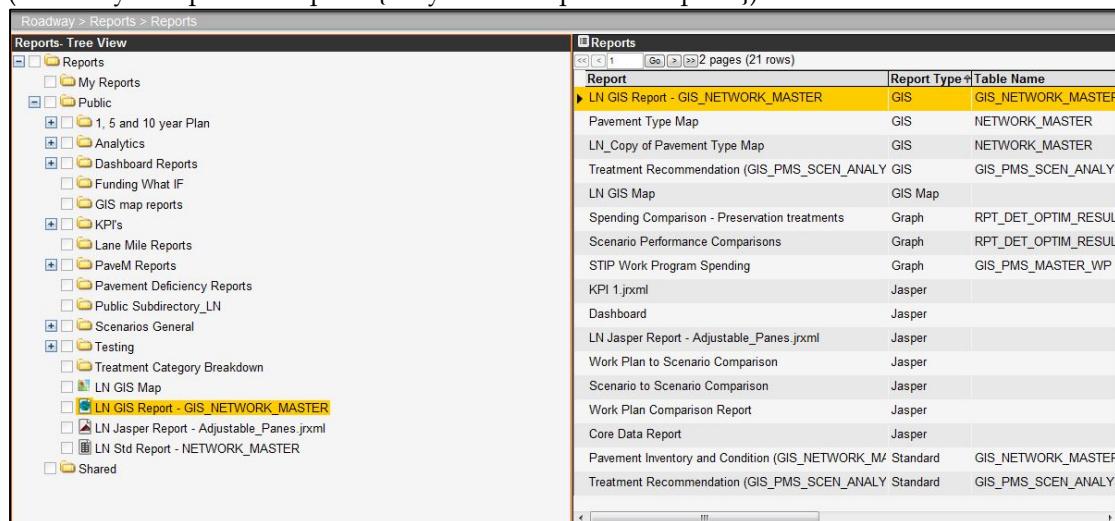
- Private folders (folders containing reports that are available only to the creator of the report).
- Shared folders (folders containing reports that are available to those users that were provided access by the report’s creator).
- Public folders (folders containing reports that are available to all authorized users).

Reports are each created from a "view" of the database. These views are essentially various ways of organizing and relating the parameters and variables of the database, and serve as the basis for reports. A report may be thought of as an excerpt of a particular view – certain columns of data are extracted from the view, sorted or filtered (or otherwise manipulated) as needed, and then displayed on your computer. Any number of report views can be added to the system without software modification.

Note: Each module has its own set of reports and report views.

### 5.1. Reports Management Window

(Roadway > Reports > Reports {& System > Reports > Reports})



The screenshot shows the Reports Management Window. On the left, there is a 'Reports - Tree View' pane showing a hierarchical structure of reports. On the right, there is a 'Reports' list view showing a table of report details.

Report	Report Type	Table Name
LN GIS Report - GIS_NETWORK_MASTER	GIS	GIS_NETWORK_MASTER
Pavement Type Map	GIS	NETWORK_MASTER
LN_Copy of Pavement Type Map	GIS	NETWORK_MASTER
Treatment Recommendation (GIS_PMS_SCEN_ANALY)	GIS	GIS_PMS_SCEN_ANALY
LN GIS Map	GIS Map	
Spending Comparison - Preservation treatments	Graph	RPT_DET_OPTIM_RESULT
Scenario Performance Comparisons	Graph	RPT_DET_OPTIM_RESULT
STIP Work Program Spending	Graph	GIS_PMS_MASTER_WP
KPI 1.jrxml	Jasper	
Dashboard	Jasper	
LN Jasper Report - Adjustable_Panes.jrxml	Jasper	
Work Plan to Scenario Comparison	Jasper	
Scenario to Scenario Comparison	Jasper	
Work Plan Comparison Report	Jasper	
Core Data Report	Jasper	
Pavement Inventory and Condition (GIS_NETWORK_MASTER)	Standard	GIS_NETWORK_MASTER
Treatment Recommendation (GIS_PMS_SCEN_ANALY)	Standard	GIS_PMS_SCEN_ANALY

When you click the Reports menu item, the application displays the Reports Management window. This window contains two panes: the Reports pane and the Report Details pane.

### **5.1.1. Reports Pane**

The Reports pane shows a hierarchical structure that organizes the reports to which you have access. The principal divisions of this structure are: Public Reports, Shared Reports, and My Reports. These divisions are described more fully below:

- Public Reports – This folder contains all reports that are shared amongst all users in your agency and for which you have sufficient security clearance. These reports are read-only. To modify a report for your own use, copy it to your My Reports folder by right-clicking the report and then clicking **Copy Report**. (Alternatively, you could display the configuration dialog box for the report [by using the right-click **Setup>Show Report** command], making the desired modifications, and then clicking the **Create Copy** button that is provided in the configuration dialog box.)

Note: The folder structure within the Public Reports folder as well as the names of the subfolders cannot be modified except by users with a security level of 5 - System Administrator or greater. (The names of public reports cannot be changed regardless of a user's security level.)

- Shared Reports – This folder contains all reports shared specifically with you from another user and for which you have sufficient security clearance. The system inserts a subfolder in this folder for each user that shares a report with you. The shared report is then placed in this subfolder. Additionally, when the other user shares the report, he or she may select whether the report is read-only or may be edited.

Note: The security level of the report overrides the selection of read-only or read-write when sharing a report. For example, say a report is configured as read-only. If you then share this report and select read-write access, the shared report will still be read-only.

Note: The reports shown in the Shared Reports folder are not copies of the original report that a user shared. They are the original reports. This means that any changes you make to the report are reflected in the report shown in the My Reports folder of the user who shared the report (as well as any other users with which he or she shared the report).

Note: You cannot modify the folder structure within the Shared Reports folder nor can you edit the name of a folder or report. Also, even if a report was shared with read/write access, you cannot change the security level assigned to the report (as shown in the Right-to-See field).

- My Reports – This folder contains all reports to which only you have access.

Tip: You may find it useful to create subfolders within this folder, one for each type of report (Tabular, Graph, GIS, and/or Jasper).

Note: Unlike Public and Shared reports, you may change the title of reports and folders in My Reports. To do this, double-click the name of the report and type the new name.

When you right-click a folder or a report name, the system displays a shortcut menu with various commands. These commands are described in the following sections.

### **Right-click Commands for Folders**

When you right-click a folder, a shortcut menu is displayed. This menu shows the common commands along with the following special commands:

- **Insert Tabular Report** – This command is only available when you right-click the My Reports folder or one of its subfolders. It inserts a new Tabular report title within the folder you right-clicked.
- **Insert Graph Report** – This command is only available when you right-click the My Reports folder or one of its subfolders. It inserts a new Graph report title within the folder you right-clicked.
- **Insert GIS Report** – This command is only available when you right-click the My Reports folder or one of its subfolders. It inserts a new GIS report title within the folder you right-clicked.
- **Insert GIS Map** – This command is only available when you right-click the My Reports folder or one of its subfolders. It inserts a new GIS map title within the folder you right-clicked.
- **Insert Jasper Report** – This command is only available when you right-click the My Reports folder or one of its subfolders. It inserts a new JasperReport title within the folder you right-clicked.
- **Insert Directory** – This command is only available for the My Reports folder and its subfolders. (If you have sufficient authorization, it is also available for the Public folder and its subfolders. You cannot add folders to the Shared folder.) When available, this command inserts a new folder inside the folder you right-clicked. You may change the default name of the new folder by double-clicking the name and then typing the desired name.
- **Delete** – This command is only available for folders that are empty (that is, that have no reports in them). When selected, it deletes the folder from the My Reports hierarchy.
- **Delete Selected** – This command will only execute for folders that are empty. When selected, it deletes all folders that have a check mark in the square beside the folder name.

### **Right-click Commands for Individual Tabular, Graph, and GIS Reports and GIS Maps**

When you right-click a report title for a Tabular, Graph, GIS, or GIS Map report (in other words, any report other than a JasperReport), a shortcut menu is displayed. This menu shows the common commands along with the following special commands:

- **Show Report** – This command is only available for Tabular reports. It opens a new window and displays the report using current data.  
  
Note: If a tabular report is configured with an “As Of” date that is different than the current date, the report generated with this command may differ from the same report generated via the Tabular Reports Configuration dialog box.
- **Setup/Show Report** – This command displays the configuration dialog box for the report type of the report you right-clicked. See page 232 for further information on the Tabular configuration dialog box; page 245 for the Graph configuration dialog box; and page 232 for the GIS configuration dialog box. For GIS Maps, see the Floating Map window on page 33. Other than Tabular reports, you must display the configuration dialog box for a report to display the report.

- **Share Report** — This command displays a dialog box so you may select the user or users with which you wish to share the report you right-clicked. See page 229 for more information on using this command.
- **Put On My Dashboard** — This command places the selected report on the dashboard. After selecting this command, a dialog box appears so you may select the quadrant in which the report will appear. See page 231 for more information on dashboard reports.

**NOTE**

The maximum size of a tabular report that can be shown on the dashboard is set by the value in GRID\_REPORT\_IN\_MEM\_SIZE\_THRS in the SYSTEM\_ADM table. This value is the number of cells (rows times columns) that may be displayed.

- **Make Report Public** — This command is available for reports in your My Reports folder that are not currently public. When you select this command the system copies the report you right-clicked to the Public folder. It may then be accessed by anyone in your agency.
- **Make Report Private** — This command is only available for a report that you (that is, your User ID) made public. It removes the copy of the report from the Public folder. (It may be initiated from either the Public folder or your My Reports folder).
- **Delete** — This command deletes the selected report from the My Reports hierarchy. (You cannot delete reports shown in the Shared or Public folders.)
- **Delete Selected** — This command deletes all reports in the My Reports hierarchy that have a check mark in the square beside the report name. (You cannot delete reports shown in the Shared or Public folders.)
- **Copy Report** — This command creates a copy of the report you right-clicked and places it in your My Reports hierarchy.
- **Schedule Report to be Emailed** — This command allows you to send the selected report to other people as an email attachment. When you select this command, the system displays the Schedule Email window; see page 223 for more information.

#### ***Right-click Commands for Individual Jasper Reports***

When you right-click a report name for a JasperReport, the following special commands are added to the shortcut menu that is displayed by right-clicking the name of the JasperReport:

- **Show Report (Excel)** — This command creates the report in a format readable by the Microsoft Excel application and, if this application is available, displays the report in Excel.
- **Show Report (HTML)** — This command opens a new browser window and displays the report.
- **Show Report (PDF)** — This command creates the report in a format readable by the Adobe Reader application and, if this application is available, displays the report.
- **Edit Jasper Subreports** — This command displays a new window that allows you to edit the subreports contained in the JasperReport you right-clicked.
- **Download Report** — This command starts a process to place the highlighted report on your computer. Once on your computer, you may use the appropriate JasperReports development tool (such as iReport) to modify the report template to suit your needs.

- **Update Report** – This command allows you to replace the highlighted report with a new version (in other words, modify the report).

### **5.1.2. Report Details Pane**

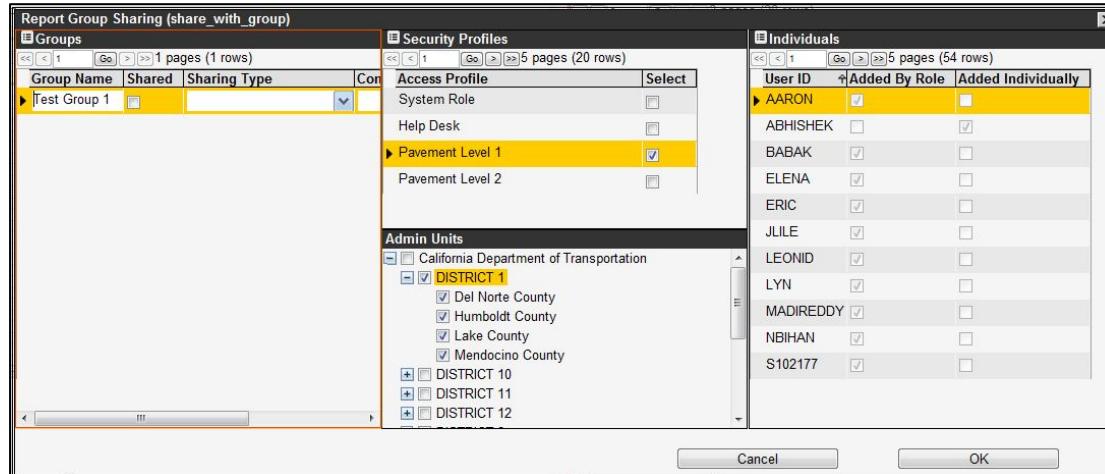
The Report Details pane contains records for all reports to which you have access (regardless of whether the reports are private, public, or shared). When you click a report in the left pane, the record in this pane corresponding to the report is highlighted. The Report Details pane shows the following fields:

- Report – This column shows the name of the report.
- Public – When the report is a public report, a check mark appears in the check box in this column. This check box is not editable.
- Shared – This field indicates the number of users with which the report is shared. The value is a hyperlink that will display a popup window when you click the hyperlink. The popup window lists the user names of all users with which the report is shared.
- Comments – This field provides information that helps you to better understand the report.
- Right to See – This field shows the security level assigned to the report. Only users with this access level (or a greater level) may view or edit the report.

Note: The security level is selected from a drop-down list of security levels. The levels shown in the list are those equal to your security level or lower.

- Date Updated – This field shows when the report was last modified.
- User Update – This field shows the name of the user who last modified the report.
- On Dashboard – A check mark in the check box indicates that the report is shown on your dashboard. This check box cannot be modified by clicking it; instead, to put a report on the dashboard, you use the **Put on My Dashboard** command that is found by right-clicking the report you wish to put on your dashboard. To remove a report from the dashboard, display the dashboard, right-click the report that you want to delete, and then click the **Delete Report** command.
- Report Owner – This field indicates the user that created the report. Only this user can change the Right to See setting and only this user can delete the report.
- Report Type – This field shows the type of report (Tabular, Graph, GIS, GIS Map, or Jasper).
- Table Name – This field shows the name of the table or view from which data for the report is drawn.
- Last Shown – This field shows when the report was last viewed.

### 5.1.3. Share Report Window



#### NOTE

Do not assign a Sharing Type when creating a share group. Assigning a Sharing Type causes the report to be shared.

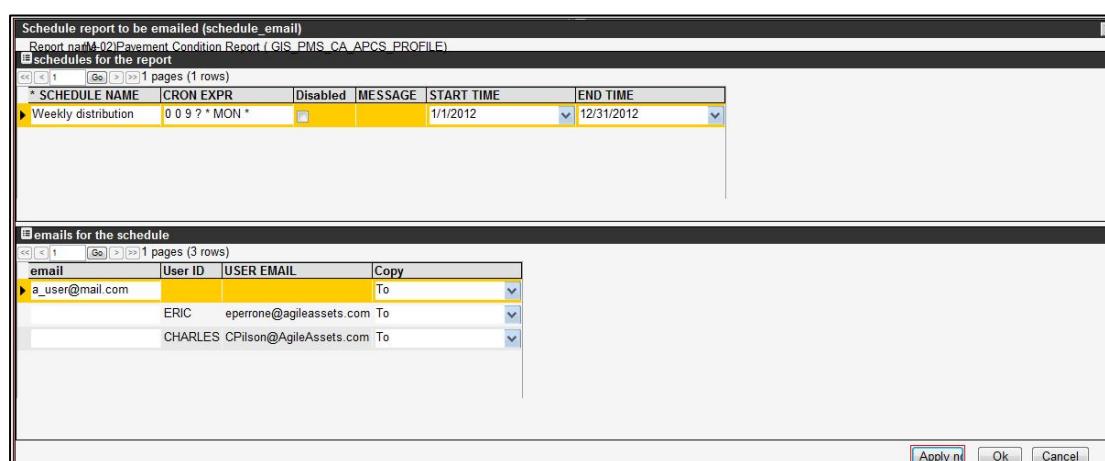
The Share Report window is displayed by right-clicking one of your own reports in the left pane to display a shortcut menu and then clicking the **Share Report** command that appears in the shortcut menu. This window shows all Share Groups that you have created (in the Groups pane) and the users that are in the selected Share Group (in the Individuals pane). The Individuals pane also contains two columns to indicate how a user was added to the pane:

- A check mark in the Added by Role column indicates that the user was added to the pane by the selection of a security profile and administrative unit. (The middle two panes, Security Profiles and Admin Units, show what security profile(s) and admin unit(s) are selected for the selected Share Group.)
- A check mark in the Added Individually column indicates that the user was added to the pane directly via the **Insert** command.

For more information on how a Share Group is created, see page 227.

The means by which a report is shared is the assignment of a Sharing Type to a Share Group. See page 229 for more information.

### 5.1.4. Schedule Email Window



The Schedule Email window is displayed when you right-click a report and then click the **Schedule Report to Be Emailed** command. This window allows you to create a schedule for when the report will be emailed to the desired recipients by inserting a record in the upper pane (and completing the fields of the record using right-click commands) and inserting the recipients in the lower pane.

**NOTE**

The shortcut menu of the upper pane contains a command called **Email Now**. Since this command sends the report immediately rather than at a scheduled time, you do not need to complete the CRON Expression, Start Time, or End Time fields.

The Schedule Email window contains two panes: Schedules for Report and Emails for the Schedule. In addition, the window also contains three command buttons at the bottom of the window. The functions of these buttons are described below:

- **Apply Now** — This command button saves the data in the window and enables the **Email Now** command.
- **OK** — This command button saves the data in the window and closes the window.
- **Cancel** — This command button discards any changes made to the window and closes the window.

#### ***The Schedules for Report Pane***

The Schedules for Report pane shown at the top of the Schedule Email window is where you configure the schedule for when the selected report will be sent.

#### **Description of the Columns in the Schedules for Report Pane**

The Schedules for Report pane contains the following columns:

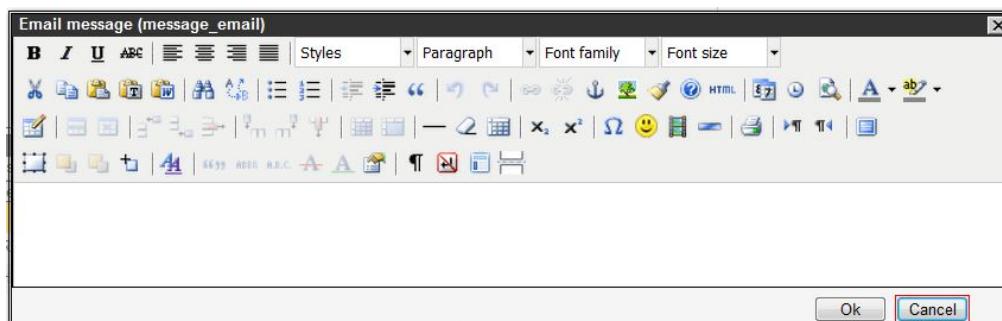
- Schedule Name — This is the name of the schedule.
- CRON Expression — This column contains the expression that determines when the report is sent to the recipients (within the time period specified by the Start Time and End Time fields). The CRON expression is created and entered into the field by right-clicking the record to display the shortcut menu and then clicking the **Set Schedule** command. See the following section for more information on this command.
- Disabled — When this check box is selected, the schedule is disabled and so the system will not distribute the selected report.
- Message — This column shows the optional email message that will accompany the distributed report. The message is created by right-clicking the record to display the shortcut menu and then clicking the **Edit Email Message** command.
- Start Time — This column sets the beginning of the time period in which the CRON expression will be processed to distribute the selected report. You may enter a date directly or double-click the field to display a calendar.
- End Time — This column sets the end of the time period in which the CRON expression will be processed to distribute the selected report. You may enter a date directly or double-click the field to display a calendar. Once the day specified in this field is reached, the report will no longer be sent.
- Report Args — This column shows the expression that determines what data is shown in the distributed report. The expression is created by right-clicking the record to display the shortcut menu and then clicking the **Define Argument(s)** command,

- Format – This column contains a drop-down list of formats for the distributed report.

#### Description of the Commands in the Schedules for Report Pane

When you right-click a record in the Schedules for Report pane, the system displays a shortcut menu. This menu contains the common commands along with the following special commands:

- **Set Schedule** – This command sets when the report will be distributed. It displays the CRON Expression Editor dialog box (see page 196 for more information on this dialog box). After composing the CRON expression and closing the dialog box, the system places an abbreviated version of the expression in the CRON Expression column.
- **Edit Email Message** – This command displays the Message dialog box, an example of which is shown below. You use this dialog box to compose the (optional) email message that will accompany the distributed report. After composing the message and closing the dialog box, the system places the message in the Message column.



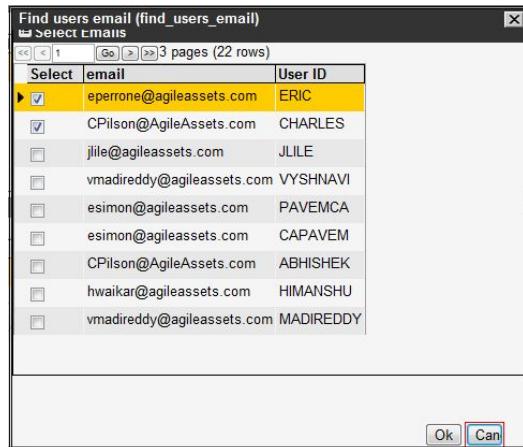
- **Define Arguments** – This command box displays a dialog box so you may enter any data required for the report. After entering the data and closing the dialog box, the system will put the entered data in the Report Args column. (If input data is not required for the report, this command is not available.)
- **Email Now** – This command becomes available after entering the recipients of the report (which may only be done after inserting a record in the Schedules for the Report pane) and then clicking the **Apply Now** command button. Once it is available, it immediately sends the report to the specified recipients. After the command executes, it is a good practice to select the Disable check box in the upper pane to ensure that the report is not sent again by the system.

#### ***The Emails for the Schedule Pane***

The Emails for the Schedule pane shown at the bottom of the Schedule Email window contains the recipients of the report. You may enter a recipient in either of two ways (note that the commands shown below do not become available until a record is inserted in the upper pane):

- Right-click the pane to display the shortcut menu and then click **Insert**. The system inserts a blank record. You may then directly enter the email address to which the report will be sent in the Email column.

- Right-click the pane to display the shortcut menu and then click **Insert Users Email Address**. The system responds by displaying a list of users with email addresses. An example of this list is shown below.



You then select the desired recipient(s) by clicking the check box beside the recipient's name. Once all recipients are selected, close the dialog box and the system inserts a record for each selected recipient.

#### NOTE

In the first method, the recipient's email address appears in the Email column. In the second method, the recipient's email address appears in the User Email column rather than the Email column. This difference is of no consequence.

After entering the email address of the recipients, you may use the drop-down list in the Copy field to set whether the recipient appears on the To line of the email; the CC line of the email ("carbon copy"); or BCC line of the email ("blind carbon copy" — that is, the recipient receives a copy but he or she does not appear on the emails sent to the other recipients).

## 5.2. How to Insert a New Report

The procedure for inserting a new report into your My Reports folder (or one of its subfolders) is the same for the Tabular, Graph, and GIS types of reports. (You cannot insert a new report into the Public or Shared folders. For JasperReports, see page 245 for more information.)

Follow these steps to insert a new Tabular, Graph, and GIS report:

- Display the Reports Management window.
- Right-click the folder in which the new report will appear and then click the **Insert** command appropriate for the type of report you will be creating. For all types other than GIS Maps, the system displays a dialog box so you can select the table or view from which the data for the report will be taken. For GIS Maps, the system simply inserts a new record; go to step 5 to continue.
- In the dialog box, find the record showing the table or view from which the data for the report will be taken and then click this record to select it.
- Click the **OK** button. The system closes the dialog box and displays the new report in the folder that you right-clicked.
- If desired, double-click the name of the report and type a new name for the report.
- Click the  icon to save the new report.

**NOTE**

At this point you have only created the record for the report, not the report itself. The next steps are to right-click the report and then click the **Setup/Show Report** command. The system will then display a configuration dialog box for the particular report type. Once the report is configured, the Report Details pane will display and you may set the security level for the report by selecting an entry from the drop-down list shown in the Right to See field.

### 5.3. How to Display a Report

To display a report, perform the following steps:

1. Display the Reports Management window.
2. In the report hierarchy on the left, locate the report you wish to view and click the report name to select it. The system displays the Report Details pane on the right, which shows information about the report.
3. Select one of the following methods to display the report depending on the type of report (which is shown in the right pane):
  - o If the report is a Tabular report, right-click the report name and then click **Show Report**. The system displays the report in a new window.
  - o If the report is a Graph report, right-click the report name and then click **Setup/Show Report**. The system displays the Graph Reports Configuration dialog box. Click the Graph tab to view the Graph report.
  - o If the report is a GIS report, right-click the report name and then click **Setup/Show Report**. The system displays the GIS Reports Configuration dialog box. Click the Report tab to view the GIS report.
  - o If the report is a GIS map, right-click the report name and then click **Setup/Show Report**. The system displays the Floating Map window.
  - o If the report is a JasperReport, right-click the report name and then click the command to display the report in the desired format (either PDF, Excel, or HTML).

### 5.4. How to Create a Share Group

**NOTE**

Do not assign a Sharing Type when creating a Share Group. Assigning a Sharing Type causes the report to be shared.

A Share Group is a collection of individuals that have a common factor (such as "all field supervisors" or "all employees in Division 1"). It determines which users have access to a shared report. A Share Group is defined in one of the following ways:

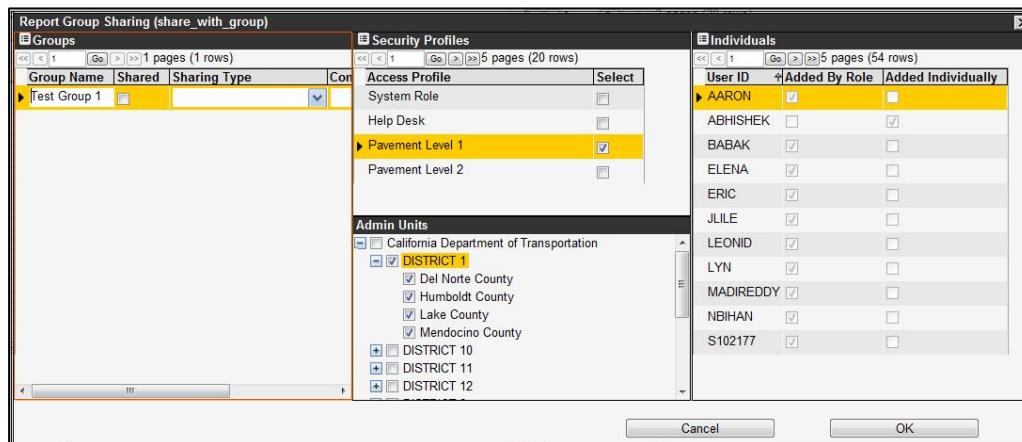
- Selecting one or more security profiles and one or more administrative units. The selection of these two items in turn determines the individuals that will have access to the shared report.
- Directly inserting the User IDs of the individuals with whom the report will be shared.
- A combination of the two previous methods. That is, you first specify the people with whom the report will be shared by selecting one or more security profiles and one or

more administrative units — and then you augment this list by inserting additional users.

#### **5.4.1. Create a Share Group by Selecting Security Profiles and Administrative Units**

To create a Share Group that contains people that were selected by specifying one or more security profiles and one or more administrative units, perform the following steps:

1. Display the Share Report window by right-clicking a report in the My Reports section of the hierarchy in the left pane and then clicking **Share Report**. An example of this window is shown below.



2. In the Groups pane of the Share Report window, right-click and then click **Insert**. The system adds a new record to the pane.
3. In the new record, provide a name for the group by entering it in the Group Name column.
4. **Leave the Sharing Type column blank.**
5. If you would like to include any additional information about the Share Group, enter that information in the Comments column.
6. In the Security Role pane, click the check box for the security role assigned to the people with whom the report will be shared. The system places a check mark in the Select column to denote that the security role is selected.
7. Repeat step 6 for any additional security roles.
8. In the Admin Units pane, click the check box beside the name of the administrative unit that contains the people with whom the report will be shared. If you wish to select multiple administrative units, you may click the check box for each administrative unit or use the selection commands that are found by right-clicking the name of an administrative unit. The system places a check mark in the Select column to denote that the administrative unit is selected. In addition, the system displays all users that are in the selected administrative unit(s) and that are assigned the selected security profile(s) in the Individuals pane.
9. Once the intended people are shown in the Individuals pane, click the **OK** button to save the new Share Group and close the Share Report window.

#### **5.4.2. How to Create a Share Group by Directly Inserting Users**

To create a Share Group that contains people that were selected by directly inserting their User IDs, perform the following steps:

1. Display the Share Report window by right-clicking a report in the My Reports section of the hierarchy in the left pane and then clicking **Share Report**. An example of this window is shown in the previous section.
2. In the Groups pane of the Share Report window, right-click and then click **Insert**. The system adds a new record to the pane.
3. In the new record, provide a name for the group by entering it in the Group Name column.
4. **Leave the Sharing Type column blank.**
5. If you would like to include any additional information about the Share Group, enter that information in the Comments column.
6. In the Individuals pane, right-click and then click **Insert**. The system displays a new window that lists all users.
7. In the new window, find the User ID of the person with whom you wish to share the report and then click the check box beside the User ID. The system places a check mark in the Select column to denote that the user ID is selected.
8. Repeat step 7 for any additional users.

Note: If you insert a User ID by mistake, you may remove it from the Individuals pane by right-clicking the User ID and then clicking **Delete**. (This command is only available for a User ID added by insertion rather than by selecting a security profile and administrative unit.)

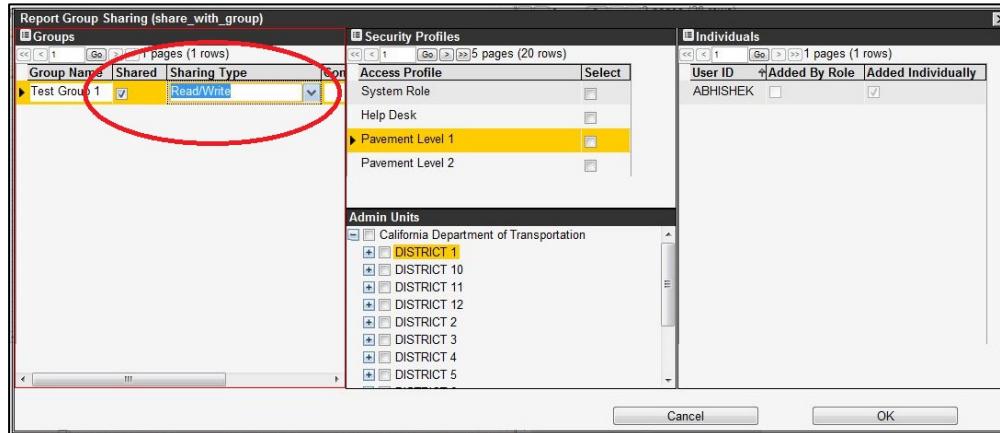
9. Once the intended people are shown in the Individuals pane, click the **OK** button to save the new Share Group and close the Share Report window.

#### **5.5. How to Share a Report**

To share a report with specific users, you must first create a Share Group as described in the previous sections. Once Share Groups are defined, follow these steps to share a report:

1. Display the Reports Management window.
2. In the My Reports area of the report hierarchy, right-click the report you wish to share and then click **Share Report**. The system displays the Share Report window.

3. In the Groups pane, find the Share Group that contains the individuals with whom you wish to share the report. In the record that shows the desired Share Group, set the Sharing Type column to the desired value (either read-only or read/write). The system places a check mark in the Shared check box to denote that the report is shared with this group. An example of the window at this point is shown below.



4. Repeat step 3 for any additional Share Groups.
5. Click the OK button. The shared report will now appear in the Shared folder of each user with whom you shared the report.

## 5.6. How to Stop Sharing a Report

If you no longer wish to share a report with all members of a Share Group, clear the Shared check box in the Share Report window.

If you wish to stop sharing a report with a particular administrative unit and/or security role, clear the check box for the administrative unit and/or security role.

If you wish to stop sharing a report with an individual, and that individual was added manually rather than by using security roles and administrative units, find the person's record in the Individuals pane, right-click the record, and then click **Delete**.

If you wish to stop sharing a report with an individual, and that individual was added by using security roles and administrative units, you cannot stop sharing with that individual (without stopping sharing with all members of his or her security group and administrative unit).

## 5.7. How to Make a Report Public

To make a report public (that is, available to all users in your agency), follow these steps:

1. Display the Reports Management window.
2. In the My Reports area of the report hierarchy, right-click the report you wish to make public and then click **Make Report Public**. The system places a copy of the report you right-clicked in the Public folder.

## 5.8. How to Make a Report Private

To remove a report from the public area, you must be the user that initially made the report public. Provided you are this user, you may make the report private from either the Public folder or your My Reports folder. To make a report private again, follow these steps:

1. Display the Reports Management window.
2. In either the Public or My Reports area of the report hierarchy, right-click the report you wish to make private and then click **Make Report Private**. The system removes the copy of the report shown in the Public folder.

## 5.9. Dashboard Reports

The Dashboard feature allows you to display up to four reports (from the same module) at the same time in the module's Home window. Each report occupies one quarter of the window (regardless of the number of reports you select to be displayed in the Dashboard window). Scroll bars are available to display the portions of the report that fall outside of the pane in which the report is displayed. If desired, you may display a report in the entire window by double-clicking the title bar at the top of the pane. (You restore the report to its original size by again double-clicking the title bar.)

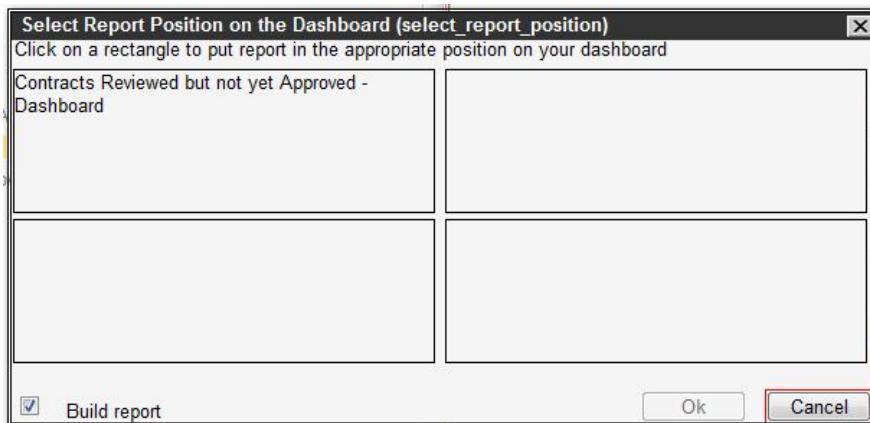
### NOTE

The maximum size of a tabular report that can be shown on the dashboard is set by the value in GRID\_REPORT\_IN\_MEM\_SIZE\_THRS in the SYSTEM\_ADM table. This value is the number of cells (rows times columns) that may be displayed.

### 5.9.1. How to Add a Report to the Dashboard

To add a report to a module's Home window, follow these steps:

1. Select the module by clicking the appropriate tab.
2. Display the Reports Management window by clicking the Reports menu item.
3. In the report hierarchy, find the report you wish to show on the dashboard. Right-click the name of the report and then click **Put On My Dashboard**. The system displays a dialog box so you may select the quadrant in which the report is to appear. An example of this dialog box is shown below.



Note: Any reports already configured to appear on the dashboard will be shown in the quadrant where they appear. You may replace an existing report with a different report by selecting the quadrant showing the report to be replaced.

4. Click the desired quadrant. The system colors the selected quadrant to denote that it is selected.
5. If you want the report to appear immediately on the dashboard, leave the Build Report check box selected.  
Otherwise (for example, if the report will take a considerable amount of time to generate), clear the check box by clicking the check mark. The report will then not appear on the dashboard immediately; instead, it will be generated and placed on the dashboard when the system job "Build Dashboard Reports" is run. Typically, this job is run each evening when more time is available for generating reports.
6. Click **OK**. The system closes the dialog box. The selected report will appear the next time that the Home window is displayed (or after the system job is run).

### **5.9.2. How to Refresh a Dashboard Report**

The data shown in a report on the dashboard is current as of when the system job "Build Dashboard Reports" was last run. If you wish to update the report sooner than the next time this system job is run, follow these steps:

1. Select the appropriate module by clicking the tab for the module.
2. If necessary, display the module's Home window.
3. Right-click the report you wish to update and then click **Refresh Report**. The system will then retrieve the most current information and display it in the report.

### **5.9.3. How to Remove a Report from the Dashboard**

To remove a report from a module's Home window, follow these steps:

1. Select the appropriate module by clicking the tab for the module.
2. If necessary, display the module's Home window.
3. Right-click the report you wish to remove and then click **Delete Report**. The system will then remove the report from the Home window. (Note that the report is only removed from the Home window; the report is still available in the list of reports in the Reports Management window.)

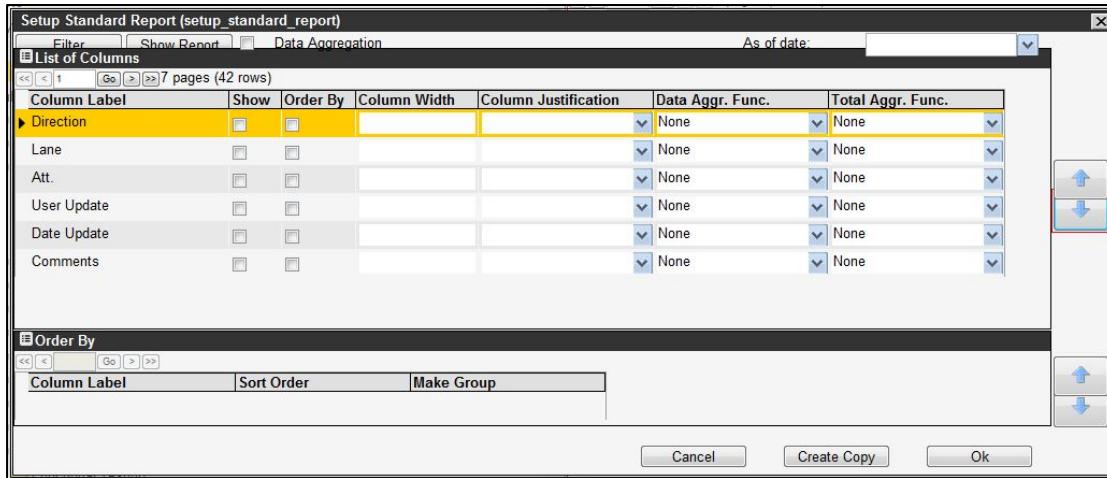
## **5.10. Tabular Reports Configuration Dialog Box**

### **NOTE**

For the Internet Explorer 8.0 browser, to enable scroll bars to appear in the resulting report you must select the Enable Websites to Use the Search Pane check box (found on the Advanced tab of the Internet Options dialog box).

Tabular reports display information in a tabular (row and column) format. Each module has its own set of Tabular reports.

The configuration dialog box for Tabular reports is launched by right-clicking a Tabular-type report and then clicking **Setup/Show Report**. An example of this dialog box is shown below.



This dialog box contains the following:

- **Filter** — This button displays the Filter dialog box, which you may use to restrict the data used in the report. See page 20 for more information on filtering.
- **Show Report** — This button displays the report in a new browser window. The report is displayed in one continuous table that is divided into virtual pages. The size of each virtual page varies between 100 and 900 rows. The default value is 100. To change this value, select the desired number of rows from the drop-down list shown at the top right of the report. (The setting of the drop-down field has no effect on printed output.)

Note: Tabular reports may be downloaded in a variety of formats by selecting the desired format and then clicking **Download** on the browser page showing the report.

For XLS and XLSX formats, the size of the report is limited by the value in SYSTEM\_ADM.GRID\_REPORT\_IN\_MEM\_SIZE\_THRS. If the size of a report (columns times rows) exceeds this value, the system generates the message, "The entire report cannot be downloaded in this format. The limit is [value] (rows \* columns)." To recover, you must either export fewer records or change the limit.

Also, for CSV files, all fields are enclosed in double quotes to handle the presence of any commas in the output data.

- **Data Aggregation** check box — When this check box is selected, all records in the database are summarized into a single record for each non-numeric column that you select to be included in the report. The numeric values in each record that are summarized are aggregated as specified in the Data Aggr. Func. column in the List of Columns pane. (When you select the check box, the aggregation method is automatically set to Total for each column containing numeric data. This helps you to see which columns contain numeric data.)

For example, say you are creating a report showing total costs for 10 administrative units. The database contains 5000 Day Cards. If you do not select the Data Aggregation check box, the report will show 5000 rows — one row for each Day Card. However, if you do select the check box, then the report will show only 10 rows — one row for each administrative unit (with the total costs being the sum of all Day Cards for that administrative unit). If you elected to include subunits in the report (and each administrative unit contained two subunits), then the report would contain 20 rows — one row for each administrative unit/subunit combination.

Note: If you change the aggregation type from Total to some other type, or if you clear and then re-select the Data Aggregation check box, your selection will be lost. (It will be restored to the default type, which is Total.)

- As of Date — This field allows you to use data from a particular time in the past (rather than current data) for the report. Note: This will only work if the tables on which the report is based are configured to support temporality.
- List of Columns pane — This pane lists all columns available in the view that underlies the report. The columns that are actually used in the report are indicated by a check mark in the Show column. The order of the listed columns from top to bottom corresponds to the order of the columns in the report from left to right. See the following section for more information on this pane.
- Order By pane — This pane lists the columns that are used to sort the information in the report and whether the sort is in ascending or descending order. It also indicates if subtotals are to be included in the report. The columns that appear in this pane are those with a check mark in the Order By column in the List of Columns pane. See the section on the following page for more information on this pane.

#### **5.10.1. List of Columns Pane**

The List of Columns pane shows all columns in the view that underlies the selected report. The columns that actually appear in the report are indicated by a check mark in the Show column.

##### **NOTE**

If the list of columns extends beyond what will fit in the pane, use the vertical scroll bar for the pane to view the remaining columns; the vertical scroll bar for the browser window will not do this. (You may need to use the horizontal browser scroll bar to scroll the window so the vertical pane scroll bar is visible.)

To the right of the table are two large arrows. These are used to set the order in which the columns will appear in the report. For the selected record in the table, clicking the upward pointing arrow moves the record upward in the table — which means that the column will appear further to the left in the displayed report. Conversely, clicking the downward pointing arrow moves the selected record downward in the table and further to the right in the displayed report.

##### **NOTE**

Clicking an arrow and holding down the mouse key will cause the record to move continually until you release the mouse key.

The columns in this pane are described below:

- Column Label — This column shows the name of the column. It is what will appear in the report as the title of the column.
- Show — When the check box is selected, the column is included in the report.
- Order By — When the check box is selected, the column is used to sort the information in the report. The column will also be shown in the Order By pane.
- Column Width (px) — This column specifies the width of the column in pixels. If you do not specify a value, the default value will be used. (The default value is the width needed to display the column head on one line. If you make the value smaller than the

default value, the column head will be truncated rather than wrapping to a second line.)

- Data Aggr. Func. – When the Data Aggregation check box is selected, the system summarizes data. Non-numeric data is automatically summarized into single instances for each non-numeric value (or combination). Numeric values are aggregated as specified by what is selected in this column.

Note: By default, the Data Aggr. Func. column is set to Total for columns with numeric values when you select the Data Aggregation check box. However, if you would prefer to use the numeric value to "group by" rather than aggregate (such as for year), then Total may be changed to None.

Note: Some entries in the drop-down list found in this column are column names prefixed with the abbreviation WA. This abbreviation means "weighted average" and indicates that the values of the named column are used to create a weighted-average aggregation.

- Total Aggr. Func. – When set to a value other than None, this column inserts a summary value for all records at the end of the report beneath the column being summarized. (You do not need to select the Data Aggregation check box to aggregate the total for a column.) You choose how the data will be summarized by selecting an aggregation type from the drop-down list. **Note: You may only select a data aggregation type for columns containing numeric data.**
- Aggr. Func. By – For each column that has a check mark in the Make Group column in the lower Order By pane, a column will appear in this pane to allow you to select how the numeric data is to be aggregated. (You cannot aggregate non-numeric data.) When an aggregation type is selected, the report will "break" after the last row of the group and the aggregation value will be inserted beneath the column. This essentially allows you to insert "subtotals" into the report. Note that you do not need to select the Data Aggregation check box to aggregate data for a group.

### **5.10.2. Order By Pane**

The Order By pane shows those columns by which the information in the report will be sorted. The columns shown in this pane are those with check marks in the Order By column in the List of Columns pane.

The order in which the rows are listed in this pane is the order by which the data is sorted. That is, the first (top-most) row is the primary order; the next row down is the secondary sort order; and so on for all rows. You use the large arrows to the right of the table to change the order of the rows. (Note: Clicking an arrow and holding down the mouse key will cause the record to move continually until you release the mouse key.)

The Sort Order column indicates whether the sorting is in ascending or descending order.

When the check box in the Make Group column is selected, instances of the same type are grouped together. This allows you to summarize the data in the report for each unique instance as well as all instances.

For example, if you selected the check box for Administrative Unit, each administrative unit would be grouped together. Then, using the column that is added to the List of Columns pane, you would select which numeric columns will be summarized and how that data would be summarized. The report would then "break" after each administrative unit and show the summary data values for the specified columns. (If you also selected an aggregation type in the Total Aggr. Func. column, a summary value for all administrative units would be shown at the end of the report.)

**NOTE**

The Make Group check boxes must be selected in order from top to bottom. In other words, you cannot select the check box for the second row of the table if the first row is not also selected. For example, if you sorted a report by administrative unit (primary) and section class code (secondary), you cannot summarize data just by section class code. You would also need to summarize by administrative unit (or make the section class code the primary sort).

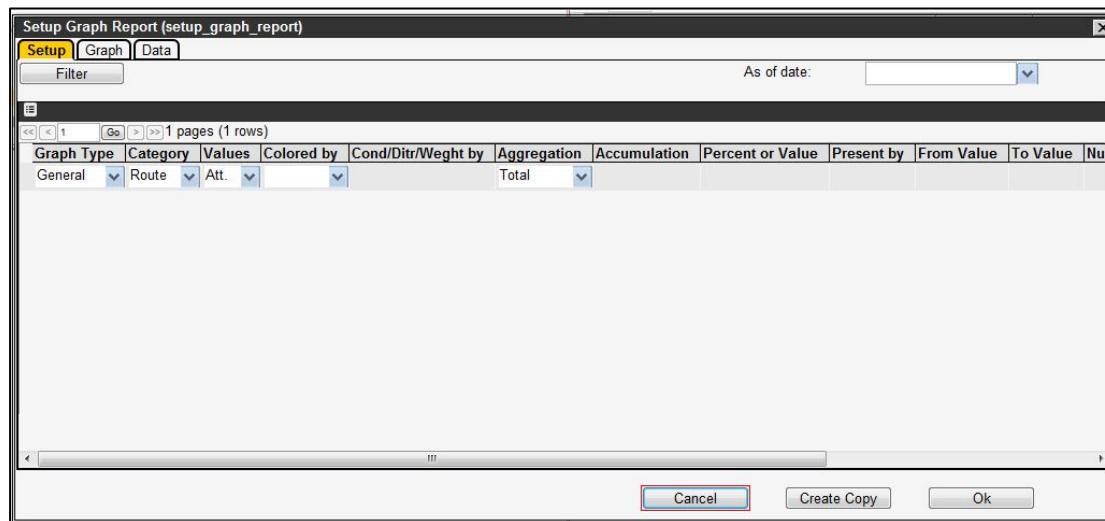
## 5.11. Graph Reports Configuration Dialog Box

Graph reports display information in a graphical representation. The system supports five general types of graphs: cumulative, distribution, conditional, general, and weighted average. Each module has its own set of Graph reports.

The configuration dialog box for Graph reports is launched by right-clicking a Graph-type report and then clicking **Setup/Show Report**. This dialog box contains the following tabs:

- Setup (which allows you to edit the parameters for a report to create a new report). This tab is described below.
- Graph (which shows the actual Graph report). This tab is described on page 243.
- Data (which shows the data from which the graph was created). This tab is described on page 243.

### 5.11.1. Graph Reports Setup Tab



The Setup tab allows you to configure the data that will appear in the graph as well as the type of graph that will be displayed. The tab contains the following:

- Filter — This button displays the Filter dialog box, which allows you to restrict the data used in the graph before it is retrieved from the database. See page 20 for more information on filtering.
- As of Date — This field allows you to use data from a particular time in the past (rather than current data) for the report. Note: This will only work if the tables on which the report is based are configured to support temporality.

- Graph type set-up table — The table shown in the window shows the type of graph to be created as well as columns to select the data to be shown in the graph. One of five types of graphs may be selected:
  - General. See the next section for a description of this type.
  - Conditional. See page 238 for a description of this type.
  - Distribution. See page 239 for a description of this type.
  - Cumulative. See page 241 for a description of this type.
  - Weight Average. See page 242 for a description of this type.

Depending on the type of graph selected, different columns will appear to configure the data for the graph.

#### NOTE

Missing values in the Category field and/or the Colored By field are treated as an extra "N/A" group. However, if values are missing from the Values field or the Cond/Distr/Wgt By field, then the record is not used for the graph and its results.

#### **General Graph Type**

The General type of graph plots any X-axis variable against a user-defined summarization of a Y-axis value. This graph differs from the other graph types in that you can define how the Y axis is summarized. The available aggregate functions include:

- Average of a user-selected variable.
- Minimum of a user-selected variable.
- Maximum of a user-selected variable.
- Summation of a user-selected variable.
- Count of a user-selected variable (number of non-null occurrences of the variable within each category and series combination).

When you select General from the drop-down list in the Graph Type column, the graph set-up table displays the following columns to configure the General graph type:

- Category — This column provides a drop-down list that contains the available categories that will be displayed along the X axis of the graph. The system will partition the data by the category you select in this column.
- Values — This column provides a drop-down list that contains the available values for the Y axis of the graph. The system processes the selected data according to the function selected in the Aggregation column.
- Colored By — This column provides a drop-down list that contains the available parameters to further partition the data selected in the Category column. The data will be partitioned by category as well as by the parameter you select in this column. For example, if the Category value is Section and the Colored By value is Direction, then the graph will show colored bars for each direction for each section. (If you select the value No Series for this column, the data is not further partitioned.)
- Aggregate — This column provides a drop-down list that contains functions. The system will process the data for the Y-axis values within each category and "colored by" combination according to the function you select in this column.

### **Conditional Graph Type**

The Conditional type of graph summarizes values only if the values pass a specified criterion. This type of graph first retrieves all rows from the database that pass the filter. Then, the application groups the data to create "cells" by category (and, optionally, by a secondary, "by color" category). Finally, for each cell, it calculates and reports the percentage of the measurement units that pass the conditional criteria out of all measurements within that cell.

A Conditional graph retrieves all the rows in the database that pass through the filter, but then calculates a percentage that meet the conditional criterion established by the user. A subtle but significant difference exists between a filter and the conditional criterion: a Conditional graph always reports the percentage of measurement units that pass the conditional criteria compared to the total measurement units that pass through any defined filters.

Note that for conditional percentage calculations:

- The numerator of the percentage (how much meets the condition) excludes from this total any record where the "condition by" column value doesn't exist (is null).
- The denominator of the percentage (the entire quantity) includes in this total all records regardless of whether the "condition by" column value exists or not (is null).
- If you want a conditional percentage to exclude missing values in the conditional variable, use the **Filter** button to filter the conditional variable to ">=0". This will remove missing conditional values from the formula (that is, from the denominator, since the numerator always excludes them).

#### **Example**

To illustrate this important point, suppose we were to create two conditional graphs on a network with 1000 miles of pavement and we were interested in knowing the percentage of the network that was in "Fair or Better" condition. The conditional criterion defined for these graphs is the number of roadbed miles in fair or better condition as defined by the user (for example, Ride Quality Index is greater than 2.5).

For this example, suppose also that for the selected year five hundred miles were in fair or better condition (50% of the total network mileage). If we set-up a filter using the Fair or Better criterion, then the graph would only retrieve rows in fair or better condition. Thus the graph would report that 100% of the network is in fair or better condition (remember we retrieved only fair or better roads by defining the filter). This result, although correct, is clearly misleading and not what was intended.

On the other hand, if we let the full network pass through the filter and then summed the mileage actually in fair or better condition (500 miles), the graph would correctly report that only 50% of the network is in fair or better condition. Thus it pays to be careful when defining filters when creating conditional distribution graphs.

#### **Columns to Configure a Conditional Graph**

When you select Conditional from the drop-down list in the Graph Type column, the graph set-up table displays the following columns to configure the Conditional graph type:

- Category – This column provides a drop-down list that contains the available categories that will be displayed along the X axis of the graph. The system will partition the data by the category you select in this column.
- Values – This column provides a drop-down list that contains the available values for the Y axis of the graph. The system processes the selected data according to the function selected in the Aggregate column.
- Colored By – This column provides a drop-down list that contains the available parameters to further partition the data selected in the Category column. The data will

be partitioned by category as well as by the parameter you select in this column. For example, if the Category value is Section and the Colored By value is Direction, then the graph will show colored bars for each direction for each section. (If you select the value No Series for this column, the data is not further partitioned.)

- Condition By — This column provides a drop-down list that contains the available conditional variable columns. Select the column you wish to use for specifying the conditional criterion.
- Condition Sign — This column provides a drop-down list that contains the operators for the condition, which are the following:
  - < The conditional column must be less than the specified value to pass the condition.
  - <= The conditional column must be less than or equal to the specified value to pass the condition.
  - = The conditional column must be equal to the specified value to pass the condition.
  - > The conditional column must be greater than the specified value to pass the condition.
  - >= The conditional column must be greater than or equal to the specified value to pass the condition.

**Note:** Missing values in the conditional column always fail the condition.

- Condition Value — This column shows the value for the condition.

### **Distribution Graph Type**

The Distribution type of graph shows summarized values of one variable plotted against another variable. It is usually displayed as a bar chart. A simple example is the number of roadbed miles within each maintenance district on a highway network.

When you select Distribution from the drop-down list in the Graph Type column, the graph set-up table displays the following columns to configure the Distribution graph type:

- Category — This column provides a drop-down list that contains the available categories that will be displayed along the X axis of the graph. The system will partition the data by the category you select in this column.
- Values — This column provides a drop-down list that contains the available values for the Y axis of the graph. The system processes the selected data according to the function selected in the Aggregate column.
- Colored By — This column provides a drop-down list that contains the available parameters to further partition the data selected in the Category column. The data will be partitioned by category as well as by the parameter you select in this column. For example, if the Category value is Section and the Colored By value is Direction, then the graph will show colored bars for each direction for each section. (If you select the value No Series for this column, the data is not further partitioned.)
- Accumulation — This column determines how the data is accumulated. A drop-down list provides two choices: Frequency or Cumulative. When you select Frequency, the results indicate "how much" (as defined by the indicator in the Present By column) there is for each category and "colored by" combination. When you select Cumulative, the system takes the "frequency" results and then accumulates for each "colored by" level across all categories.

- Present By – In this column you select the type of distribution to be calculated. The choices are:
  - Value – When the Accumulation column is set to Frequency, this selection shows in the Values column the total summation of all values for each category and "colored by" combination. When the Accumulation column is set to Cumulative, these values are then accumulated for each "colored by" level across all categories.
  - % by Series – When the Accumulation column is set to Frequency, this selection shows in the Values column the percentage of its total for each category and "colored by" combination. When the Accumulation column is set to Cumulative, these values are then accumulated for each "colored by" level across all categories.

For example, in the table shown in the following section, this setting would hold the category constant (say, District 1), total the values across all road types (Interstate + Primary + Local = 7), and then show the percentage of each road type to the total of all road types (Interstate = 1/7; Primary = 2/7; and Local = 4/7).

- % by Category Values – You should only select this type when the Accumulation column is set to Frequency and there is a "colored by" variable. When these conditions are met, this type shows in the Values column the conditional percentage per series.

For example, in the table shown in the section below, this setting would hold the road type constant (say, Interstate), total the values across all districts (which equals 5), and then show the percentage for each district to the total for all districts (District 1 = 1/5; District 2 = 2/5; and District 3 = 2/5).

- % of Total – When the Accumulation column is set to Frequency, this selection shows in the Values column the percentage of its total for each category and "colored by" combination. When the Accumulation column is set to Cumulative, these values are then accumulated for each "colored by" level across all categories.

For example, in the table shown in the following section, this setting would calculate a total for all road types and all districts (which is 21). It would then calculate the percentage of each district/road type combination as a total of all combinations (District 1/Interstate = 1/21; District 1/Primary = 2/21; and so forth).

### Example Results

Shown below are two sets of tables. The first shows original data, while the second shows the graph results based upon this data for all combinations of graph attributes.

Original data, showing what values should appear for each category and "colored by" combination, is shown in the table below.

		SERIES (Section Class)		
		Interstate	Primary	Local
CATEGORY	District #1	1	2	4
	District #2	2	1	3
	District #3	2	1	4

The following table shows the graph values per administrative unit (category) and road class ("colored by"). It shows what values would appear for each "colored by," accumulation, and "present by" selection. Note that these combinations result in nine different graphs.

COLORED BY	ACCUMULATION	CATEGORY (Administrative Unit)	PRESENT BY								
			Value per road class (Colored By)			% by Total per road class (Colored By)			% by Category per road class (Colored By)		
			Interstate	Primary	Local	Interstate	Primary	Local	Interstate	Primary	Local
Column Selected	Frequency	District #1	1	2	4	5%	10%	20%	20%	50%	36%
		District #2	2	1	3	10%	5%	15%	40%	25%	25%
		District #3	2	1	4	10%	5%	20%	40%	25%	36%
Column Selected	Cumulative	District #1	1	2	4	5%	10%	20%	Not Applicable		
		District #2	3	3	7	15%	15%	35%	Not Applicable		
		District #3	5	4	11	25%	20%	55%	Not Applicable		
"No Series"	Frequency	District #1	7			35%			Not Applicable		
		District #2	6			30%			Not Applicable		
		District #3	7			35%			Not Applicable		
"No Series"	Cumulative	District #1	7			35%			Not Applicable		
		District #2	13			65%			Not Applicable		
		District #3	20			100%			Not Applicable		

### Cumulative Graph Type

#### NOTE

"Missing" (that is, null) distributed data is excluded from Cumulative graph types.

The Cumulative type of graph is utilized for building frequency or probability density, or cumulative probability density functions, for selected variables. For example, the distribution of alligator cracking is best represented as a histogram graph that displays the distribution of the Alligator Index variable for one or more years.

When you select Cumulative from the drop-down list in the Graph Type column, the graph set-up table displays the following columns to configure the Cumulative graph type:

- Values – This column provides a drop-down list that contains the available values for the Y axis of the graph. The system processes the selected data according to the function selected in the Accumulation column.
- Colored By – This column provides a drop-down list that contains the available parameters to further partition the data selected in the Category column. The data will be partitioned by category as well as by the parameter you select in this column. For example, if the Category value is Section and the Colored By value is Direction, then the graph will show colored bars for each direction for each section.
- Distributed By – This column provides a drop-down list that contains the available parameters for "distributed by" (category) values on the X axis of the graph. When you select a parameter from the list, the application will divide the data into these categories according to the settings for From Value, To Value, and Number of Divisions columns.
- Accumulation – This column determines how the data is accumulated. A drop-down list provides two choices: Frequency or Cumulative. When you select Frequency, the graph is a normal histogram that shows the sum of the measurement column values within each bin. When you select Cumulative, the graph is a cumulative distribution histogram that shows the sum of the measurement column with values less than or equal to the value of each bin.
- Present By – This column determines how the data is presented. A drop-down list provides two choices: Value or %. When you select Value, the real value (that is, the summation) will be shown. When you select %, the percentage of values compared to the total on the Y-axis of the graph will be shown.
- From Value – This column shows the value at the beginning (nearest the Y axis) of the X axis.
- To Value – This column shows the value at the end of the X axis.
- Number of Divisions – This column shows how the range between the From Value and To Value is divided. For example, if the From Value is 0 and the To Value is 4, setting Number of Divisions to 4 causes the X axis to have five equally spaced tick marks – one tick mark for each value (0, 1, 2, 3, 4). On the other hand, if you set Number of Divisions to 2, there will be three equally spaced tick marks (0, 2, 4).

### **Weighted Avg. Graph Type**

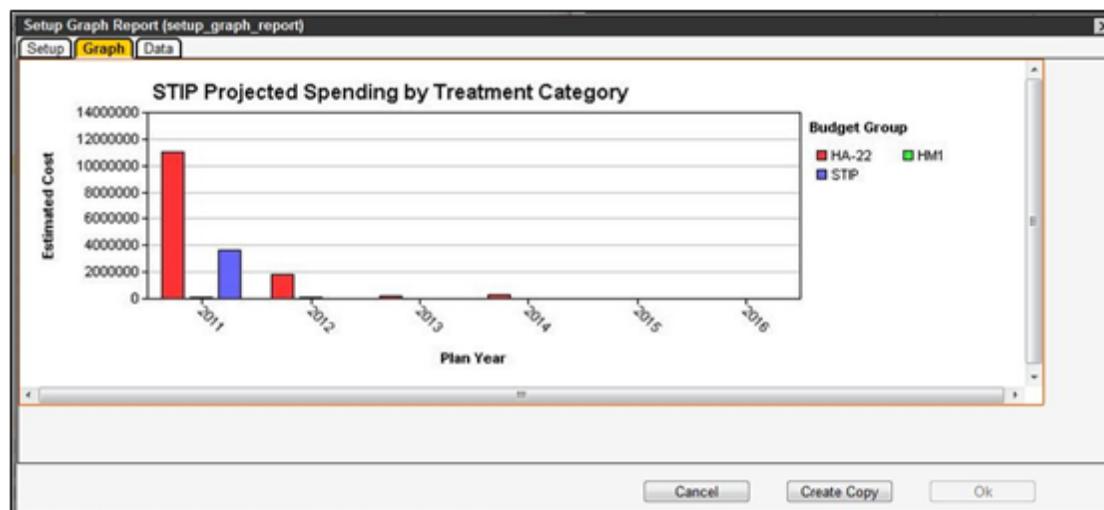
The Weighted Average type of graph plots any X-axis variable against a weighted average of a Y-axis value. This graph is similar to the General type of graph, with two exceptions: (1) the aggregation function is always "average"; and (2) it is a weighted average, not a simple average, and so needs a "weighting" variable.

When you select Weighted Avg. from the drop-down list in the Graph Type column, the graph set-up table displays the following columns to configure the Weighted Average graph type:

- Category – This column provides a drop-down list that contains the available categories that will be displayed along the X axis of the graph. The system will partition the data by the category you select in this column.
- Weighted Average Of – This column shows the column being averaged. A drop-down list is provided for you to select the column being averaged. This list contains the available "value" columns for the Y axis of the graph.

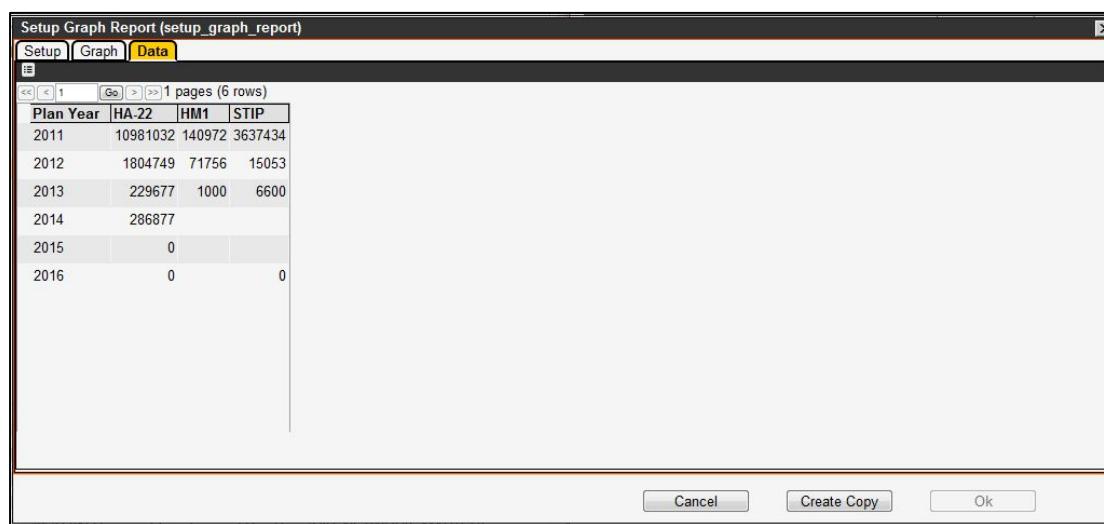
- Colored By – This column provides a drop-down list that contains the available parameters to further partition the data selected in the Category column. The data will be partitioned by category as well as by the parameter you select in this column. For example, if the Category value is Section and the Colored By value is Direction, then the graph will show colored bars for each direction for each section.
- Weighted By – This column provides a drop-down list that contains the available parameters for use in calculating a weighted average. For example, for a length-weighted average of daily traffic, select Length from the drop-down list in this column (and select Daily Traffic from the drop-down list in the Weighted Average Of column).

### 5.11.2. Graph Tab



Once the type of graph and data for the graph are selected, the resulting graph is displayed in this tab. To format the graph, right-click the graph and click **Change Graph Report Properties**. This command displays a new window, which allows you to change the parameters for the graph. See page 12 for more information.

### 5.11.3. Data Tab



Plan Year	HA-22	HM1	STIP
2011	10981032	140972	3637434
2012	1804749	71756	15053
2013	229677	1000	6600
2014	286877	0	0
2015	0	0	0
2016	0	0	0

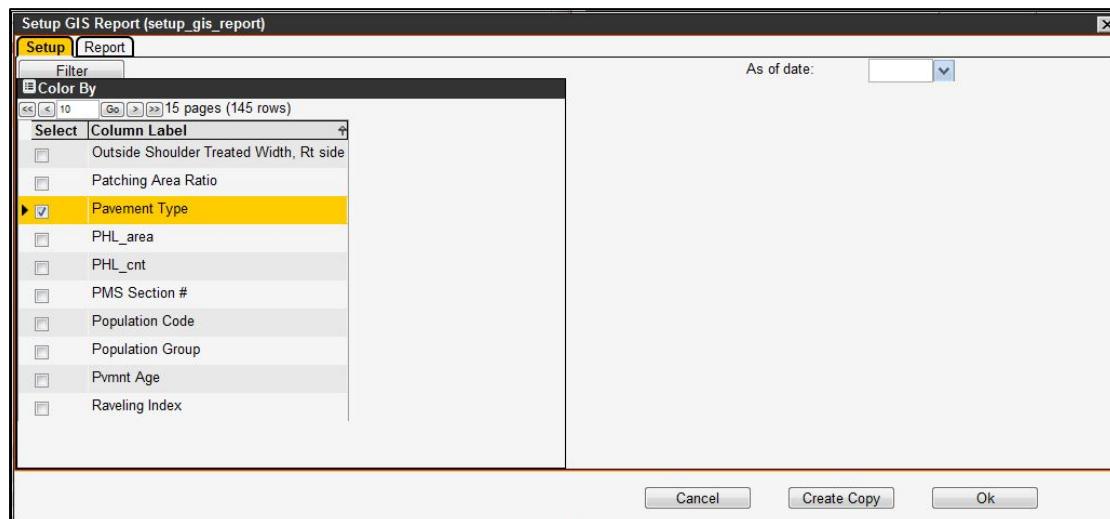
The Data tab displays the data from which the graph was created.

## 5.12. GIS Reports Configuration Dialog Box

GIS reports display information via a color-coded map, with each color denoting a particular data element or data range. Each module has its own set of GIS reports, although some modules (such as the System module) do not contain information that can be displayed in a GIS report and so this type of report is not available.

The configuration dialog box for GIS reports is launched by right-clicking a GIS-type report and then clicking **Setup/Show Report**. This dialog box contains two tabs: Setup and Report.

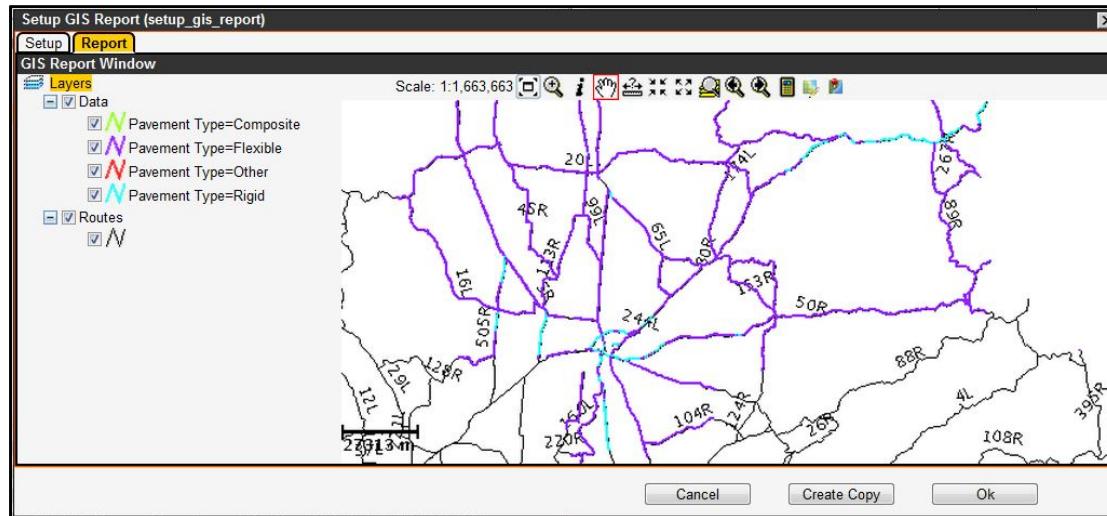
### 5.12.1. GIS Reports Setup Tab



You use the Setup tab to edit or set the parameters for the GIS report. The Setup tab contains the following:

- **Filter** – This button displays the Filter dialog box, which allows you to restrict the data used in the graph before it is retrieved from the database. See page 20 for more information on filtering.
- **As of Date** – This field allows you to use data from a particular time in the past (rather than current data) for the report. Note: This will only work if the tables on which the report is based are configured to support temporality.
- **Color By** – The Color By pane lists all columns that are in the table on which the GIS report is based. You may select one (and only one) of these columns to display in the GIS report by selecting the check box next to the name of the column. The system then assigns colors to the data from this column and overlays the selected theme with the color that represents the data value along each route in the map.

### 5.12.2. Report Tab (GIS Reports)



### **5.13.1. How to Add a Report Without Sub-reports**

This procedure describes how to add a report template to the AgileAssets application so a JasperReport may be generated. For the purposes of this example, it is assumed that the report template has already been created in a JasperReports development tool such as iReport and does not contain sub-reports.

Once the report template is available, follow these steps to add the report template to the AgileAssets application:

1. In the module of the AgileAssets application to which the report will be added, display the Reports Management window.
2. In the reports hierarchy in the left pane, locate the folder in which the report will be stored, right-click this folder, and then click **Insert JasperReport**. The system displays a dialog box so you may select the file that contains the report template.
3. In the dialog box, use the **Browse** button to locate the desired file and place it in the dialog box. Once the file is shown in the dialog box, click the **Upload** button. The system retrieves the specified file and adds the report to the folder you right-clicked.
4. If desired, double-click the default report name and type a more appropriate name for the report.
5. Right-click the new record and then click the Show Report command corresponding to the desired output to display the newly uploaded report. Check that the report is the one you intended to be added.
6. Click .

### **5.13.2. How to Add a Report With Sub-reports**

This procedure describes how to add a report template that contains sub-reports to the AgileAssets application so a JasperReport may be generated. For the purposes of this example, it is assumed that the report template was developed with the JasperReports development tool called iReport; if you use a different development tool, you will need to adjust the procedure accordingly (with the most important adjustment being the need to turn all sub-reports into parameters).

Once the report template is created and functioning properly in iReport, you will need to change the sub-reports into parameters. Note that once you do this, the report will no longer function in iReport. Instead, you will need to upload the report and display it through the AgileAssets application to view the report. To change the sub-reports to parameters, follow these steps in iReport 3.7.2:

1. Create a new parameter for the sub-report. The name of the parameter should be the name of the sub-report.
2. Double-click the parameter name to display the Add/Modify Parameter/(Properties) dialog box. In the dialog box, set the Parameter Class Type/Parameter Class field to the following: `net.sf.jasperreports.engine.JasperReport`. **Note that this value does not appear in the drop-down list.** You must highlight the existing text in the field and then type (or copy) this value into the field.
3. In the report template itself, double-click the text field showing the sub-report to display the properties dialog box for the sub-report.
4. Click the Subreports (Other) tab to select it.
5. Set the Subreport Expression Class field to `net.sf.jasperreports.engine.JasperReport` by selecting it from the drop-down list.

6. Change the value in the Subreport Expression field to a parameter. This is accomplished by clicking the **Expression Editor** button on the right side of the field, which displays the Expressions Editor dialog box.

In the Expression Editor dialog box, display the available parameters. Find the parameter corresponding to the sub-report and then double-click the name of the parameter to show it in the upper pane of the dialog box. Click the **Apply** button to close the dialog box and send the selected parameter to the properties dialog box. The Subreport Expression field should now show the desired parameter for the sub-report.

7. Close the properties dialog box.
8. Repeat these steps for any additional sub-reports.
9. Compile the report.
10. Save the report.

The report template is now in a state that can be uploaded to the AgileAssets application. Follow these steps to add the report template and associated sub-reports to the AgileAssets application:

1. In the module of the AgileAssets application to which the report will be added, display the Reports Management window.
2. In the reports hierarchy in the left pane, locate the folder in which the report will be stored, right-click this folder, and select **Insert Jasper Report**. The system displays a dialog box so you may select the file that contains the report template.
3. In the Select File dialog box, click the **Browse/Choose File** button to display a second dialog box to locate the file name on your computer.
4. Locate the file for the main report (not the sub-reports) and then double-click the desired file name, which closes the second dialog box and shows the file name in the Select File dialog box.
5. In the Select File dialog box, click the **Upload** button. The files for the main report are uploaded to the application server. A new record appears in the Reports tab that shows the main report.
6. In the reports hierarchy in the left pane, locate the folder in which the report will be stored, right-click this report, and select **Edit Jasper Subreports**. The system displays a dialog box that shows the sub-reports associated with the Jasper Report.
7. For the first sub-report listed in the tab, right-click the sub-report and then click **Insert/Update Report** from the shortcut menu that is displayed. The application displays a dialog box to select the file for the highlighted sub-report.
8. Repeat steps 6 and 7 for the remaining sub-reports listed in the Sub-reports tab.
9. Right-click the new record and then click the Show Report command corresponding to the desired output to display the newly uploaded report.
10. Click .

## 6. Information for Administrators

The section describes commands found in nearly all windows of the system as well as how to adjust window layouts.

The responsibility of an administrator of an AgileAssets application is to:

- Keep the application operational in the client's computer environment.
- Implement upgrades as provided by AgileAssets.

The information that follows identifies the qualifications needed to accept that responsibility and provides instructions for performing upgrades.

	For Software	For Database
Administrator Qualifications	Client must provide a person that is qualified to deploy/un-deploy a WAR file.	Client must provide a qualified Oracle DBA to support the client's Oracle database and the Oracle schema for the AgileAssets application.
System Components	The software is composed of a web archive (WAR) file. This file contains JavaScript files and Java classes along with other required configuration files, packaged in a standard format for Web servers.	The system is retained in a single Oracle schema. The software obtains linkage to this Oracle schema through the configuration file web.xml existing in the WAR file.
System Upgrades	A web archive (WAR) file will be provided. To upgrade, un-deploy the current application from the web server and deploy the provided WAR file.	A text file containing a series of SQL statements will be provided. To upgrade, run the file against the Oracle schema.
Troubleshooting	Contact your AgileAssets project manager or Leonid Samokhine ( <a href="mailto:lsamokhine@agileassets.com">lsamokhine@agileassets.com</a> ).	Provide the log file for the SQL run by contacting your AgileAssets project manager, Leonid Samokhine ( <a href="mailto:lsamokhine@agileassets.com">lsamokhine@agileassets.com</a> ), or Len Moser ( <a href="mailto:lmoser@agileassets.com">lmoser@agileassets.com</a> ).

### 6.1. Details of User-controlled Tables and Windows

An important feature set within AgileAssets software is the ability for users to define and control tables, windows, and their components (columns and panes). This section provides in-depth information on this feature set.

#### 6.1.1. How to Tell if a Table is User-modifiable

Any table is user modifiable. If the table is an inventory table or a list column table, then they are already defined and available in their respective "window creating" windows: Set Inventories by Asset Types window and Manage List Columns window, respectively.

For any other table, they are all user modifiable. This is accomplished in the Tables window. If the table to be modified:

- Already exists in the Tables tab of the Tables window, then select it and modify it.
- Does not exist in the Tables tab of the Tables window, then insert it. At that point, all of the table's characteristics will be automatically added to (all of the tabs in) the Tables window. From there, you can modify/add any characteristic of that table.

#### **6.1.2. How to Delete a Table**

##### **WARNING**

Deleting tables can be very dangerous. Consequently, AgileAssets strongly recommends that AgileAssets be advised of your intent to delete a table before you actually do it. Once AgileAssets evaluates the table, AgileAssets will advise you whether it ought to be deleted or not.

If it is determined that deleting a table is permissible, to delete a table you use the **Delete** right-click command in the primary data pane of the appropriate "window making" window.

#### **6.1.3. How to Tell if a Window is User-modifiable**

All windows, or more specifically the panes that are the components of all windows, have user-modifiable characteristics. Refer to the Adjustment of Window Layouts on page 249 for descriptions of these abilities.

#### **6.1.4. How to Delete a Window**

To delete a window, you delete the menu item associated with the window. This is accomplished as follows:

1. Display the Menus.
2. In the upper left pane, select the module in which the menu item for the window appears. Once the module is selected, the menu hierarchy in the lower left pane changes to reflect the hierarchy of the selected module.
3. In the bottom left pane, locate the menu item for the window.
4. Right-click the menu item and then click **Delete**.
5. Click the  icon.

## **6.2. Adjustment of Window Layouts**

The application provides a feature called Design Mode. For authorized users, this feature allows the user to:

- Adjust the height and width of panes.
- Adjust the width of columns.
- Add or remove columns.
- Modify the format applied to the pane via a CSS file.
- Modify pane-specific parameters.

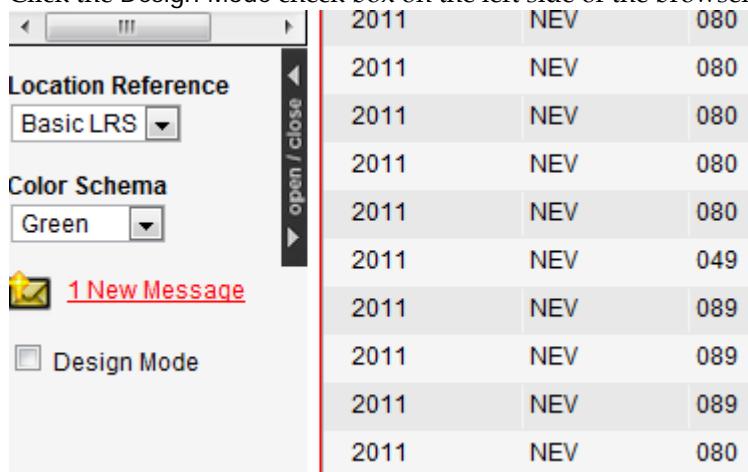
These adjustments may only be made while in Design Mode, and this mode can only be activated by users that are granted design privileges in the User Names and Access window (see page 153).

Once Design Mode is active, the **Change Control Properties** right-click command becomes available. This command displays the User Control Properties dialog box; see the following section for more information.

### **6.2.1. How to Activate Design Mode**

To activate Design Mode, you must be a user with design privileges. This designation is made in the User Names and Access window by selecting the Is Admin? check box for the user's record. Once you have design privileges, follow these steps to invoke Design Mode:

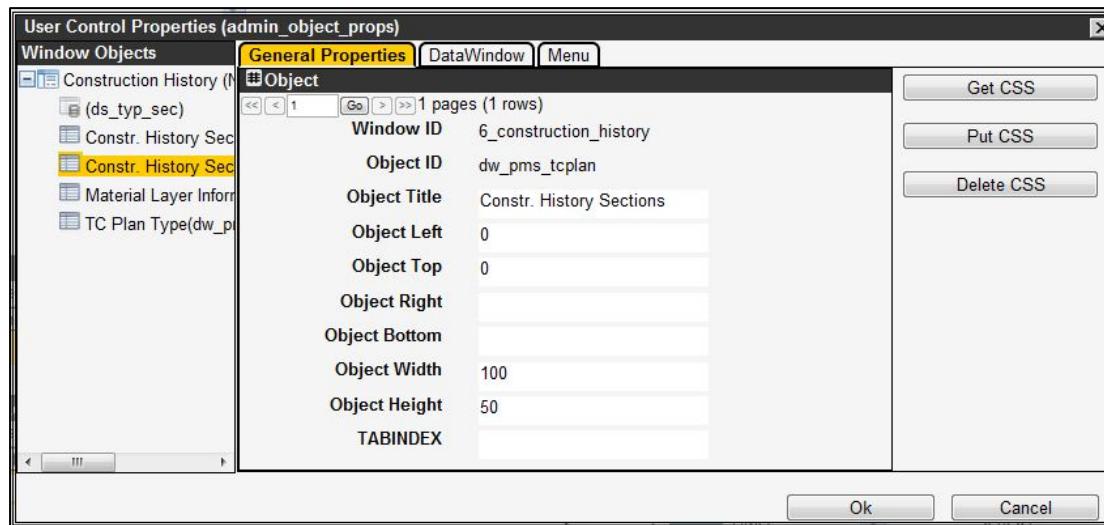
1. Log on to the application.
2. Click the Design Mode check box on the left side of the browser window.



3. Open the window to be adjusted.

Once the window is open, right-click the pane you wish to adjust and then click **Change Control Properties** in the shortcut menu that is displayed. This command displays the User Control Properties dialog box, in which you may adjust the layout of the pane as well as the columns that appear in the pane. Note: This command is not needed to adjust column widths (which are adjusted by clicking and dragging) or column or field labels (which are modified in the Columns window).

### **6.2.2. User Control Properties Dialog Box**



When you select from the shortcut menu that is displayed by right-clicking a pane, the application displays the **Change Control Properties** command the User Control Properties dialog box. (This command only is available when Design Mode is active.)

This dialog box lists all objects in the window along the left side of the dialog box. Depending on what object is selected in this list, different tabs (and fields within the tab) are displayed. The entire set of available tabs are General Properties, Data Window, and Menu. These tabs are described in more detail in the following sections.

The dialog box also provides the following command buttons:

- **OK** – This command applies any changes you made and then closes the dialog box
- **Cancel** – This command discards any changes you made, reverts to the previous (saved) version, and closes the dialog box.

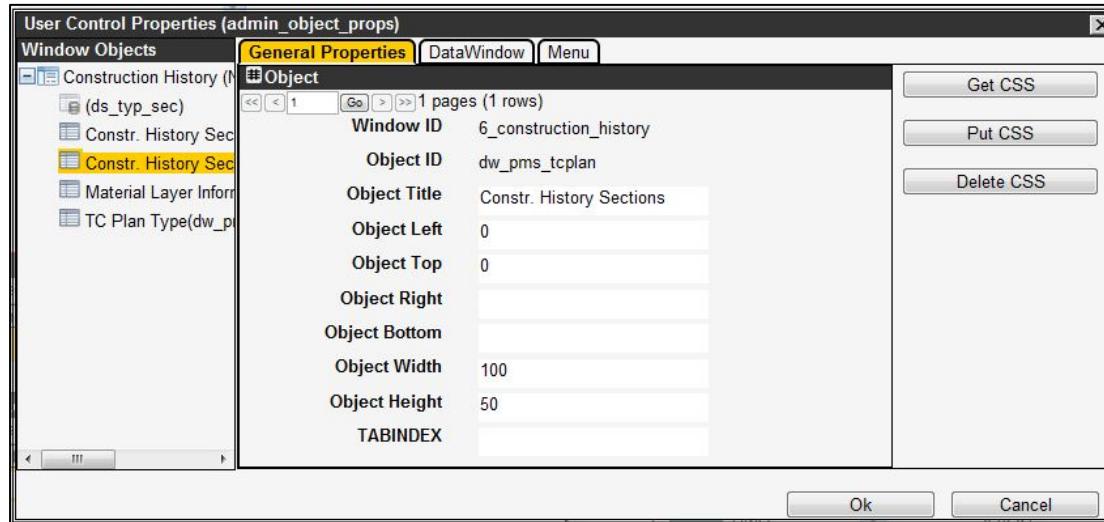
#### NOTE

Even though changes are applied to the window, they are not yet permanently saved until you attempt to close the window. The application then asks if you wish to save the changes – and responding by clicking **OK** permanently saves the changes.

#### **Window Objects Pane**

The Window Objects pane on the left side of the dialog box lists all objects that are found within the window. By default, the object selected is the object on which you right-clicked to launch the **Change Control Properties** command. What you select in this pane determines what tabs and fields are available in the User Control Properties dialog box.

#### **General Properties**



This tab provides the fields that are needed to modify the object selected in the Window Objects pane.

A typical set of fields are as follows:

- Window Title – This read-only field shows the internal name of the window.
- Object ID – This read-only file shows the procedure that created the window.
- Object Title – This field shows the word or phrase that appears in the bar above the pane.

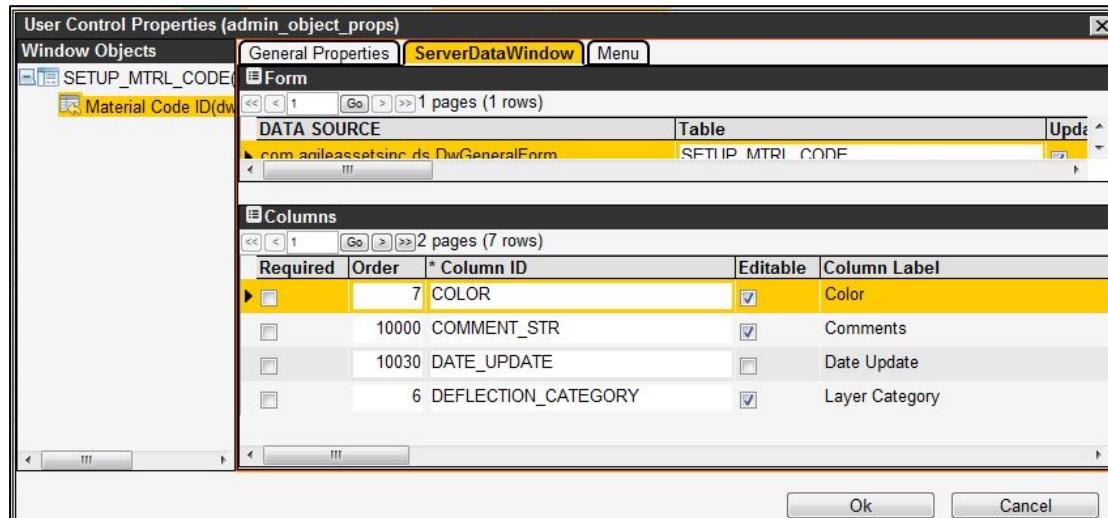
- Object Left, Object Top – These fields set the position of the upper left corner of the pane within the window.
- Object Right, Object Bottom – These fields set the position of the bottom right corner of the pane within the window.
- Object Width – This field shows the width of the pane in pixels.
- Object Height – This field shows the height of the pane in pixels.
- Tab Index – This field sets the order in which the "focus" in the window moves from pane to pane. It is recommended that this be used only by AgileAssets personnel.

Typically, these fields are not edited directly. Instead, the size of the pane is adjusted by dragging the edges of the pane to the desired size.

When the User Control Properties dialog box is launched when record view is active (that is a window that shows only a single record and the columns appear as fields), command buttons are added to the General Properties tab for CSS file manipulation. These command buttons are described below:

- **Get CSS** – This command button sends the default format to a CSS file in a folder selected by you. This allows you to modify this CSS file (using third-party software) supplying the desired format.
- **Put CSS** – This command button allows you to identify the CSS file (that now contains the desired formatting) that the application acquires and uses to format the pane.
- **Delete CSS** – This command button removes the user-created CSS file that is formatting the pane and returns the pane to the default record view formatting.

### **ServerDataWindow Tab**



When a pane is selected in the Window Objects pane, the ServerDataWindow tab is available and sets what columns appear in the pane. The tab provides two panes: Form and Columns.

#### Description of the Form Pane in the ServerDataWindow Tab

The upper, Form pane shows the following information:

- Data Source – This columns shows the source from which the data in the window is taken.

- Update Table — This column shows the name of the table from which the data in the pane is drawn.
- Updateable? — When this check box is selected, data in the pane may be modified.
- Is Grid? — When this check box is selected, the form is a grid. When it is clear, the form is free-form. (A "grid" form type shows multiple records in a table-like display; a "free-form" form type shows a single record at a time.)
- Max Rows — This column shows the maximum number of records that may appear in the pane. Note: If the value is null, the maximum number of records is 10,000. If the value is zero, all records from the table are shown in the pane.
- Sort Str — This column configures how the records in the window are sorted. It is an "SQL order" clause.

Note: If you display the User Control Properties dialog box from the Filter window, setting this column to Order ID will cause the column records in the Filter window to be sorted in the same column order as the underlying window (that is, the window from which the Filter window was launched). Otherwise, the column records will be in alphabetical order.

- Retrieval Args — This column provides the retrieval arguments (column IDs separated by semicolons) pertinent to the table selected in the Tables pane. It is only applicable when the form is used for a linked pop-up window.
- Where Clause — This column is used to filter the records shown in the pane. It is an "SQL where" clause for the selected table in the Tables pane.
- Filterable? — When this check box is selected, the columns in the table may be filtered via the **Filter** command in the right-click shortcut menus.
- Access-regulated? — When this check box is selected, the data in the table is automatically filtered to only those records where an "owner" column is the administrative unit you selected when you logged on.
- Bookmark Column — This column specifies the internal column name by which bookmarks are set.
- Default Column Width — This column sets the default width for the column. If it is left null, then the columns will be as wide as the column headers.
- Can Reorder Columns — When this check box is selected, the user can change the order of the columns without utilizing Design Mode and without affecting the order of the columns for any other users. (To move a column, the user clicks the column head and drags the column to the desired position.) The new order is retained after closing the window.
- BeforeSave Script — This column provides a script that executes after the  icon is clicked, but before the actual save operation (and after basic required-field and data-type checks). These scripts are usually used to verify changed rows prior to committing the data to the database. In technical terms, the data is saved to the database but not committed before this script is run. This allows you to create Groovy scripts that can query across any tables for validation purposes.
- AfterSave Script — This column provides a script that executes after a new row is inserted into the data pane. Typically, these scripts will set some default values in the row based on the user ID, the user's login administrative unit, or current date or time.

- Custom HTML – This column contains HTML code that customizes the appearance of the pane when in free form (single record) view (provided the pane is formatted via a CSS file). This code is activated dynamically.

Note: To display the results of newly added HTML code, display the General Properties tab, click the **Get CSS** button, and then immediately click the **Put CSS** button. Then close the Change Control Properties dialog box, close the window, and re-open the window. The effect of the HTML code should be shown in the pane.

When you right-click a record in this pane, a shortcut menu is displayed. This menu contains the following special commands:

- **Switch to Form** – This command is for AgileAssets staff use only.
- **Create/Edit Script** – This command displays a new window in which you enter a Groovy script or modify an existing script. These scripts are for server-side event handlers (mostly data checking routines) for BeforeSave and AfterSave events. After entering the script and closing the window, the new (or modified) script is available in the drop-down lists found in the BeforeSave Script and AfterSave Script columns.

Note: This command only becomes available when the cursor is in a script field.

- **Add Record History Link** – This command adds a Record History column to the table (and to the Columns pane in the Data Window tab). This column contains a hyperlink that opens a new window that shows changes that were made to the selected record. See page 263 for more information.

Note: This command is not available if the column already exists in the table.

- **Add Maintenance History Link** – This command adds a Maintenance History column to the table (and to the Columns pane in the Data Window tab). This column contains a hyperlink that opens a new window that provides a log of maintenance that was performed on an inventory item.

Note: This command is not available if the column already exists in the table.

#### Description of the Columns Pane in the ServerDataWindow Tab

##### NOTE

If the pane contains a column or columns that provide location information, see page 258 for more information on these types of columns.

The lower, Columns pane shows the columns that appear in the pane that you right-clicked. By default, this pane lists all columns that are in the table shown in the Forms pane. You may add or remove columns from this pane by utilizing the right-click **Select Columns** command. This in turn determines what columns are shown in the pane that you right-clicked to launch the User Control Properties dialog box.

When you right-click a record in this pane, a shortcut menu is displayed. This menu contains the following special command:

- **Select Columns** – This command displays a new window in which appear all columns for the table shown in the Forms pane as defined in the Tables window. Select or de-select any set of column names by using the appropriate right-click selection commands. When you close the window, the set of columns in the Columns pane is adjusted accordingly.

Note: If the column is a list column, then all columns in the list column table are also available for selection and are shown as children in the list of all columns.

Note: Be careful in using the **Select All** command as it selects all columns including all child columns. Instead, the **Select Visible Items** command is generally used because it ignores these child columns.

Note: If you wish to include a column that is not in the table, use the **Insert** command. This command adds a new, blank record into which you may type the name of the column you wish to include.

- **Create/Edit Script** — This command displays a new window in which you enter a Groovy script or modify an existing script. These scripts are for server-side event handlers (mostly data checking routines) for ItemChanged events. After entering the script and closing the window, the new (or modified) script is available in the drop-down list found in the OnItemChange Script column.
- **Renumber Order** — This command inserts values in the Order column for all columns that are not hidden. The values start at 10 and increment by 10 from the first record to the last record. (If values are already present in the Order column, these values are overwritten.)

The columns that are of particular importance are the following:

- **Required** — When this check box is selected, the column must have data (be non-null) for the record to be saved.
- **Order** — This column sets the order of the columns within the pane (from left to right). If the table is created by a DW\_proc, this column is read-only.
- **Column ID** — This column is the database column identification.
- **Column Label** — This column is the description of the column that is used in the column heading of the window pane.

Note: The label may be edited to be more descriptive than the default label. The edited label will only appear in the window from which the Change Control Properties dialog box was accessed. (If you wish to return the label to its default version, clear the edited label from the Column Label column.)

- **Editable** — When this check box is selected, data in the column may be modified (provided your security profile allows you to edit the data). When it is clear, the data is read-only. (Note: All calculated columns should have this check box clear.)
- **Massive Updatable?** — When this check box is selected, the column will be updated when the **Massive Update** command is executed.

Note: If none of the columns have this check box selected, the **Massive Update** command will not appear in the shortcut menu that is displayed by right-clicking the window or pane.

- **Hidden** — When this check box is selected, the column is not displayed. (While removing the column from the pane accomplishes the same thing, hiding the column allows you to easily re-display the column if you need to.)
- **Copyable?** — When this check box is selected, the data in the column will be copied to the new record that is inserted with the **Insert Like** command.
- **Initial Value** — Placing a value in this column is optional. When a value is placed in this column, it is used to fill the column as each record is inserted.
- **Change DDDW** — This column is applicable only when the column contains a drop-down list and is used to filter the entries in the drop-down list. The Change DDDW column shows the "SQL where" clause that filters the drop-down list to just the

desired items. (Note: To limit the number of entries in the drop-down list, use the Max Rows column.)

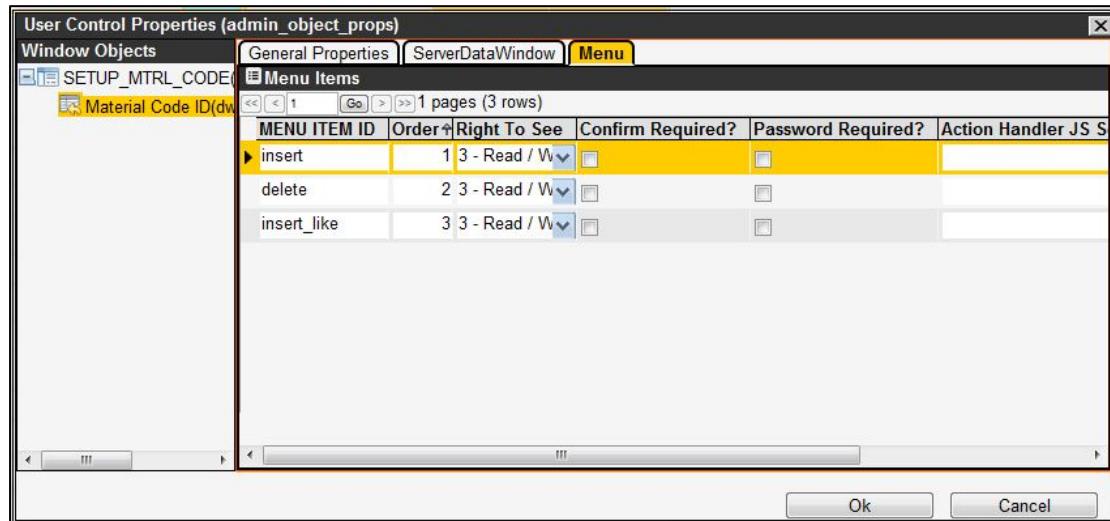
- Dropdown Filterable — This check box determines whether the items in the drop-down list in the column may be filtered. When selected, the items are filtered as the column is filtered. When clear, the list is not filtered (even if the column is filtered).
- Column Width — This column sets the width of the column (in pixels).
- Select Str — This column provides the SELECT statement calculation that defines this column's value. If the table is created by a DW\_ proc, this column is read-only.
- Max Rows — If the column is a drop-down list, this column determines the number of entries in the list. If the value is null, the maximum number of entries is 10,000. If the value is zero, it is the actual number of entries (which may be greater than 10,000).  
Note: If the value in this column is smaller than the number of candidates for inclusion in the list, then the selection of candidates for inclusion is random (not predictable).
- Is Link — When this check box is selected, each value in the column is a hyperlink and displays a popup window or URL when clicked. To define the hyperlink, you use the following columns:
  - Target Window ID — This column contains the WINDOW\_ID of the pop-up window or the URL of the website that will be displayed when the hyperlink is clicked. (If the WINDOW\_ID is not known, use the Setup Window Titles window in the System module, which shows the WINDOW\_IDS for windows.)
  - Retrieval Args — This column provides the retrieval arguments (column IDs separated by semicolons) that are passed to the pop-up window and determine what is displayed in the window. These arguments may also be input parameters for the URL, with <1> indicating the first parameter, <2> indicating the second, and so forth.
- OnItemChange Script — This column provides a Groovy script that may be used to validate data and/or determine the value of other column(s) based on what is entered in a column. See page 258 for more information on this type of Groovy script.

Note: The script is triggered when focus moves off of the column, not upon save. This means that if, for example, a column's value is determined from the values in two other columns and one of these values is null, the script needs to be written so that the third column's value is not generated until values exist in both of the other columns.

Note: Exercise care in creating scripts. This is a powerful feature and utilizing a large number of scripts will degrade performance.

- OnLinkClick Script — This column provides a Groovy script that executes when a user clicks a hyperlink.
- Right to See — This column determines the security level for viewing the column. If this column is set to Null, then the security level for the window is used.
- Right to Edit — This column determines the security level needed to modify data in the column. If this column is set to Null, then the security level for the window is used.
- Master Columns / Master WC — The list in a drop-down field may be configured to have different entries depending on the record selected. The Master Columns column contains the name of the column that is used to constrain what is shown in the drop-down list. The Master WC column supplies the where clause that sets what is displayed in the drop-down list.

## Menu Tab



The Menu tab allows you to configure what commands appear in the shortcut menu (and in what order) that is displayed by right-clicking the pane or window from which you launched the User Control Properties dialog box. This tab provides the following columns:

- Menu Item ID — This column shows the internal identifier of the command. (To discover the actual label for the command that appears in the shortcut menu, look up the menu item ID in the Action Rights window.)
- Order — This column sets the order of the commands in the shortcut menu (1 is the topmost position in the shortcut menu).
- Right to See — This column sets the security level for the command. Only users with a security level equal to or greater than the value configured in this column will be able to see the command.
- Confirm Required — When this check box is selected, then the system will prompt the user to confirm actions (such as delete) before proceeding with the action.
- Password Required — When this check box is selected, the system will prompt the user to enter his or her password before proceeding.
- Action Handler JS Script — You may create a custom command that executes a Java script when it is selected. This column lists the available Java scripts, which are created by AgileAssets staff and stored in the database itself. (The Java scripts cannot be created or modified from within the application.)

Note: When you create a custom command, you must first create a menu item ID for the command in the Action Rights window and also request that AgileAssets create the Java script for the command's action. This is necessary because when you insert a new record in this tab, the system will ask you for the ID of the command. Similarly, in the new record, the Java script must be stored in the database so you can select it from the drop-down list in this column.

- BeforeOpen Script — This column contains a drop-down list of BeforeOpenMenu-type Groovy scripts, which are configured in the Groovy Scripts window. These scripts affect the display of the command in the shortcut menu (not the action of the command itself). For example, you could select the Groovy script that hides the command when a user displays the shortcut menu.

### ***LOC\_IDENT Column Information***

When a pane is to include location information, you include LOC\_IDENT in the list of columns that appear in the User Control Properties dialog box. LOC\_IDENT is an "umbrella" that includes the five base location reference columns used to specify a location: route, lane direction, lane ID, offset from, and offset to. (There may be alternate referencing systems used in some implementations that use different columns to specify location, and what is said here for the five base columns also applies to these alternate columns.)

These constituent columns do not appear in the list of columns in the dialog box; the application inserts them in place of LOC\_IDENT when the pane is displayed. The order of the constituent columns are determined by the order shown in the Setup Location Reference Methods (LRMs) window, which cannot be controlled from within the application but rather through direct editing of the underlying database.

In other words, the five constituent columns always appear in the same order – but where these five columns appear in relation to the other columns of the data form is determined by where LOC\_IDENT appears in the list of columns in the User Control Properties dialog box (and which is determined by the setting of the Order column).

While you cannot determine the order of the constituent columns, you can control whether a particular location-identifying column is shown in the pane. This is accomplished by inserting the location-identifying column that you wish to not display (along with LOC\_IDENT) and then selecting the Hidden check box to prevent the column from being displayed. The following steps spell this out in more detail:

1. Display the User Control Properties dialog box for the desired pane.
2. In the Columns pane of the Data Window tab, right-click and then click **Insert** from the shortcut menu. A new record is added to the pane.
3. In the Column ID column of the new record, type the name of the location-identifying column that you wish to hide. (The column IDs for location-identifying columns are shown in the Setup Location Reference Methods [LRMs] window.)
4. Select the Hidden, Required, and Editable check boxes by clicking each check box. A check mark appears in the check box to denote that it is selected.
5. If the column to be hidden is LANE\_ID and if LOC\_IDENT is set to be required and editable, then enter 0 (zero) in the Initial Value column.
6. Click the  icon to save the form.

### ***OnItemChange Script Information***

The OnItemChange Script column contains a drop-down list of existing OnItemChange Groovy scripts that are available within the database. You may also right-click this column and then click the **Create/Edit Script** command to either edit the currently assigned script or create a new one. Scripts configured here are typically used for validating data as it is entered by a user. These scripts run whenever a user has edited the value in a column and the user moves the cursor off of the field by tabbing or saves data.

Certain variables that are available at run time may be included in the script. A partial list of the variables that are available is shown in the table below.

#### **CAUTION**

**DO NOT CHANGE the values of these variables in the script; this could cause system instability.**

Variable	Description
OWNER_ID	This is the ID value of the user login admin unit.
new_val_in	This variable contains the proposed edit of the column by the user. This is typically the data value you would validate.
row_in	This is the row being edited. It is typically used to set or retrieve values of other data in the row for validation purposes.
col_in	This is the Column ID being edited.
user_id	This is the User ID of the person editing the data.
module_id	This is the current module ID.

NOTE: During evaluation, the new\_val\_in variable is always passed as a string to the script – that is, if the column is a number then the proposed new value is still a string until you cast the value as a number. This should be done when performing calculations. The following are examples of this:

```
parseFloat(new_val_in); — This gives a number with decimals.  
parseInt(new_val_in); — This gives an integer.  
StrToDate(new_val_in); — This converts a date string to a date.
```

### Validation

To validate data, you write a custom set of commands to check if the proposed edit by the user is allowed. If it is not allowed, you issue the following command:

```
EventReturnValue = -1;
```

### Messaging

The function to display a message to the user is:

```
basic_window.ShowAlert(text identifier here);
```

To display actual text, use the following function:

```
basic_window.ShowAlert("Message Goes Here!!!");
```

### Some Useful Non-standard Functions Provided by AgileAssets Software

- GetItemDisplay() – If you need the text label shown on screen for a drop-down list, use the GetItemDisplay function. Note: This function should not be used for the column you are currently validating (use the GetChild().Lookup() function for this purpose). An example of this function is shown below:

```
this.GetItemDisplay(row_in, "ROUTE_ID")
```

This function always returns a string value.

- GetChild( ).Lookup() – This function will return the display text for the column currently being evaluated. For example, if you are validating a change to the ROUTE\_ID column and it is a drop-down list, you could use this function to get the ROUTE\_NAME corresponding to the ID the user chose (which is held in the variable new\_val\_in):

```
this.GetChild("ROUTE_ID").Lookup(new_val_in)
```

This function always returns a string value.

- SetItem() — This function sets the value of a column to the specified value. For example, the following statement sets the LANE\_DIR column to 2 for the current row:  
this.SetItem(row\_in,"LANE\_DIR",2);

This function is not data-type dependent, but you should use proper data types for the column being adjusted.

- GetItem() — This function gets the value of a column. Note that for a drop-down list, this returns the ID value and not the drop-down list text label. For example, the following statement gets the value of the WC\_ID column for the current row:

```
this.GetItem(row_in, "WC_ID");
```

This function always returns the data type of the underlying column.

#### Example: Validate IS\_ACTIVE\_SECTION Field

The following example determines whether a road section is active.

```
/*Validate IS_ACTIVE_SECTION Column
Rules are
1. WC_ID must be 1,2, or 3
2. Sous Code must be 000
Only process if being set to Active (value 1)/*
if(new_val_in == "1") {
/*Get WC_ID Value/
    var wc_id = this.GetItem(row_in, "WC_ID");
/*Get Sous Code*/
    if(this.IsValidColumn("ROUTE_ID"))
    {
        var dir_code = this.GetItemDisplay(row_in,
"ROUTE_ID").substr(13,3);
    }
    else
    {
        var dir_code = this.GetItem(row_in,
"SOUS_ROUTE").substr(1,3);
    }
/*Check Rules and Show message and Reject Edit if necessary*/
    if((wc_id != 2 && wc_id != 3 && wc_id != 1 )|||dir_code != "000" )
    {
        basic_window.ShowAlert("Le segment ne peut être à « Utilisé »
s'il est sur gravier, sur structure ou sur une sous-route non
principale.");
        EventReturnValue = -1;
    }
}
```

#### **6.2.3. How to Change the Size of a Pane**

Once Design Mode is activated, you may adjust the size of a pane by hovering over the edge of the pane that you wish to move until the cursor changes to opposite-pointing arrows. You may then drag the edge to the desired position.

#### **6.2.4. How to Change the Size of a Column**

Regardless of whether Design Mode is activated, you may adjust the size of a column by hovering over the edge of the column that you wish to move until the cursor changes to opposite-pointing arrows. You may then drag the edge to the desired position. The new size will then be retained when you close the window. (Each user may have a different column size.)

#### **6.2.5. How to Add a Column to a Pane**

To add a column to a pane, follow these steps:

1. Ensure that the Design Mode check box is selected and then open the window that contains the pane.
2. Right-click the pane and then click **Change Control Properties** from the shortcut menu. The User Control Properties dialog box is displayed.
3. Click the Data Window tab.
4. In the Columns pane, right-click and then click **Select Columns** from the shortcut menu. The application responds by displaying a dialog box to select the columns that will appear in the form.
5. In the dialog box, select the column that will appear in the pane. You select a column by locating it in the hierarchy, right-clicking the node, and then clicking **Select This**.
6. Click **OK** to close the dialog box and display the selected column in the Columns pane.
7. In the Columns pane, set the Order, Required, and Editable fields as necessary.
8. If desired, complete the remaining fields for the column if needed.
9. When all information is entered, click **OK** to close the User Control Properties dialog box.
10. Click the  icon. The application asks if you wish to save the new layout.
11. Click **OK** to save the new layout.
12. Exit Design Mode by clicking the Design Mode check box in the left gutter.

#### **6.2.6. How to Delete a Column from a Pane**

To delete a column from a pane, follow these steps:

1. Ensure that the Design Mode check box is selected and then open the window that contains the pane.
2. Right-click the pane and then click **Change Control Properties** from the shortcut menu. The User Control Properties dialog box is displayed.
3. Click the Data Window tab.
4. In the Columns pane, locate the column to be removed.
5. Right-click the record showing the column and then click **Delete** from the shortcut menu. The record for the column is removed from the pane.

Note: Instead of deleting the column, you could also hide the column by clicking the Hidden check box.

6. Click **OK** to close the User Control Properties dialog box.
7. Click the  icon. The application asks if you wish to save the new layout.

8. Click **OK** to save the new layout.
9. Exit Design Mode by clicking the Design Mode check box in the left gutter.

### **6.2.7. How to Hide Columns**

The User Control Properties dialog box determines what columns are displayed in a window or pane for all users. If desired, you may further customize the display of columns for just yourself by performing the following steps:

1. Display the User Control Properties dialog box by selecting the Design Mode check box, right-clicking the window, and then clicking **Change Control Properties**.
2. In the left pane of the dialog box, select the pane for which you wish to customize the display of columns.
3. Click the Data Window tab.
4. In the Form pane of the Data Window tab, locate the Columns Hideable check box and then click it to select it. The system places a check mark in the selection box to indicate that it is selected.
5. Click the **OK** button. The system saves the changes and closes the User Control Properties dialog box.
6. Clear the Design Mode check box.
7. Close and then re-open the window. (This causes the system to place the command for showing/hiding columns in the right-click shortcut menu.)
8. In the pane for which you wish to customize the display of columns, right-click and then click **Show/Hide Columns**. The system displays a dialog box. This dialog box contains all columns that may be hidden by individual users. (Columns that are marked as hidden in the User Control Properties dialog box will not appear in this dialog box.)
9. For the column(s) you wish to hide, click the check box beside the name of the column.
10. Click **OK**. The system closes the dialog box and hides the designated columns. The system also displays a hyperlink command in the title bar called **Hidden Columns** to alert you that columns are hidden. You may use this command rather than the right-click **Show/Hide Columns** to further modify the display of columns in the pane.

### **6.2.8. How to Change a Pane's Format**

The format for a pane is controlled by a CSS file. The format for the pane may therefore be changed by applying a new or modified CSS file to the pane as described below:

1. Ensure that the Design Mode check box is selected and then open the window that contains the pane.
2. If necessary, place the window in record view by double-clicking the  icon in the title bar.
3. Right-click the window and then click **Change Control Properties** from the shortcut menu. The User Control Properties dialog box is displayed.
4. In the General Properties tab, click the **Get CSS** button and save the CSS file in a convenient location on your computer.
5. Using third-party software, edit the CSS file to reflect the desired formatting.

6. In the General Properties tab, click the **Put CSS** button and retrieve the edited CSS file from your computer. The application applies the retrieved CSS file and applies it to the window.
7. Click the  icon. The application asks if you wish to save the new layout.
8. Click **OK** to save the new layout.
9. Exit Design Mode by clicking the Design Mode check box in the left gutter.

#### NOTE

The edited CSS file does not cause the original CSS file to be deleted; rather, it "overlays" the original. This allows you to remove the edited CSS file (by utilizing the **Delete CSS** button in the User Control Properties dialog box) and thereby restore the original formatting as set by the original CSS file.

### **6.3. Record History Column**

At the time of implementation, certain columns in specific windows are selected for tracking. For these columns, the system logs when the value in the column changed, what the old value was, what the new value is, and who made the change.

This log is accessed via a hyperlink in the Record History column. When Yes is shown in this column, clicking the word displays a new window that shows all changes made to the value in the tracked column for the selected record. A new record is added to this window each time the value changes, which serves as a history of changes for the tracked column.

### **6.4. Update Source and Where Columns**

The definition of columns includes the option of using SQL statements to update the data in the column. These statements are placed in the Update Source and Where Clause fields. For the data to be successfully and correctly updated, you must complete these fields properly.

The general form for the SQL statement is:

```
SELECT ( update_source ) AS column_name FROM table_name A WHERE  
where_clause
```

Such that:

- The SELECT statement is legal and when run without the where\_clause selects one and only one row for every row in the table\_name.
- table\_name is the appropriate table being updated (as identified in the currently selected record in the top pane of the window you're using)
- column\_name is the appropriate column being updated (as identified in the currently selected record in the Columns pane of the window you're using)
- update\_source is the text in the UPDATE\_SOURCE field. If filled, then this record in "Columns" pane is a calculated column.
- where\_clause is the text in the WHERE\_CLAUSE field. It need not be filled, but, if filled, then the calculation is only applied to those rows in the table that pass the where clause criteria

Note: The best way to learn to write Update Source SQL is to find several existing examples and place them into the SELECT statement configuration shown above.

To test if your update\_source text is correct, place it into the SELECT statement configuration shown above without the where\_clause. It should yield the same number of rows as in the table with no duplication on the primary key or any unique index.

To test if your where\_clause text is correct, place it into the SELECT statement configuration shown above and see that it yields the expected filtered results.