State of Florida
Department of Transportation

FDOT Traffic Plans - Lighting

CE-11-0118

User Training Manual

March 2, 2015

ENGINEERING / CADD SYSTEMS OFFICE
TALLAHASSEE, FLORIDA
http://www.dot.state.fl.us/ecso/

FDOT Traffic Plans - Lighting
CE-11-0118

Description

This course was developed to teach students fundamental use of GEOPAK, MicroStation, and the FDOT CADD standard resources and applications for Traffic Plans and Lighting design based on a typical Traffic Plans design workflow to produce and present Traffic Plans and Lighting design required for Electronic/Digital Delivery.

This course includes but is not limited to:

- General topics on Lighting Plans and File Creation
- Exploring the FDOT desktop folder
- Create File/Project tool for creating base files for a project
- FDOT Traffic Plans Menu bar
- Levels and Level Filters
- Text Styles
- Annotation Scale
- Models
- Creating a Lighting Key Sheet
- Creating a Summary of Pay Items
- Lighting Tools
- Quantities and Reports
- Sheet Navigator/Sheet Labeling Tools
- Plotting Tools

Objectives

At the conclusion of this course the student will use GEOPAK, MicroStation and the FDOT CADD standard resources and applications for Traffic Plans and Lighting design and work in and understand a typical Traffic Plans design workflow to produce and present Traffic Plans design required for Electronic/Digital Delivery.

Audience

Students involved in preparation of FDOT Lighting Plans at the Beginner, Intermediate and Advanced levels. The complete Traffic Plans course includes the FDOT Signalization course and the FDOT Signing and Pavement Marking course, but students can choose from the three courses as per their job.

Prerequisites

The following courses and some manual drafting or related CADD product experience is recommended:

- FDOT MicroStation Essentials - Part I (CE-11-0114)
- FDOT MicroStation Essentials - Part II (CE-11-0115)
- FDOT Basic GEOPAK Road (CE-11-0099)

Duration: 8 Hours

Professional Credit Hours: 6 PDHs

Table of Contents

1 LIGHTING PLANS

OBJECTIVE

This chapter is divided into three sections:

- CADD Standards & File Creation

    o General discussion on Lighting Plans and File Creation.

    o Exploring the FDOTSS3 folder on the desktop.

    o Using the Create File/Project tool to create the base files for a project.

- Traffic Plans Menu & Workflow

- Levels, Rules, & Models

    o Levels
    o Rules & Filters
    o Civil Features
    o Text Styles
    o Models
    o Annotation Scale

INTRODUCTION

This chapter reviews the FDOTSS3 working environment including how to properly create design files that meet Florida Department of Transportation (Department) CADD standards. New concepts and new tools are introduced that make producing plans much more efficient.

CADD STANDARDS & FILE CREATION

TRAFFIC PLANS AND FILE CREATION

Traffic Plan projects are normally a component of a Roadway plans package. Therefore, the project directory structure usually exists prior to beginning work on a Traffic Plans component. If this is not the case, the same directory structure and file standards that apply to Roadway apply to any other lead component. The Department's *CADD Manual* defines the naming convention used to create the different types of design files required in a project.

The Department uses TIMS Document Management software to manage project files and to create the local directory structure. The Department's Electronic Delivery (EDelivery) software is used to document all project files and track project information. However, beginning July 2015, most of the EDelivery software functionality will be replaced with Digital Delivery. GEOPAK's Project Manager creates a set of binary files to keep track of all the files used by GEOPAK and their purpose. The information tracked by GEOPAK is different from that maintained in Electronic Delivery and TIMS.

EXPLORING THE DESKTOP FDOTSS3 FOLDER

When the Department's CADD software was installed a folder named FDOTSS3 was placed on the desktop. This folder contains shortcuts to applications used when working on the Department's projects. Some of the shortcuts are for starting MicroStation and others are for working with Electronic Delivery.

The figure below shows the contents of the FDOTSS3 folder.
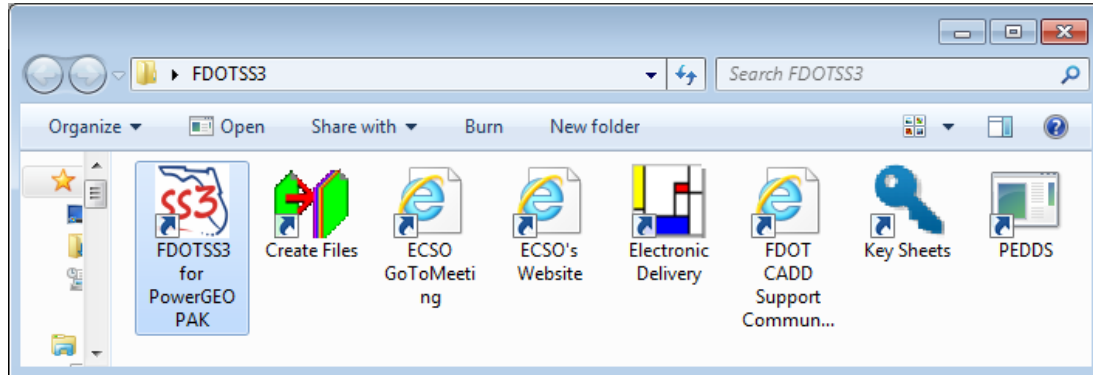
It is recommended to use these shortcuts to start MicroStation. Using these shortcuts will start the workspace environment properly which can alleviate, among other things, issues with using the wrong CADD standards.

The Department now delivers only one (1) shortcut for starting MicroStation:

FDOTSS3 – Uses the V8 file format and FDOTSS3 CADD standards.

Note   FDOTSS3 does not include a metric workspace. Resource files that are required to view and print existing metric files are supplied for this purpose only.

Using this shortcut to start MicroStation, as mentioned previously, opens MicroStation Manager and sets the workspace but does not set the Project. The Project is set by selecting the drop down arrow and selecting the correct project configuration file or *.pcf* file. The workspace remembers the last project worked in and will use the *.pcf* file the next time MicroStation is started with that workspace. This is important to watch as most designers probably work on more than one project at a time. It is recommended to always use the *.pcf* file. This supplements other tools used to produce electronic/digital delivery submittals. Discussion provided later in the course.

©2015 FDOT
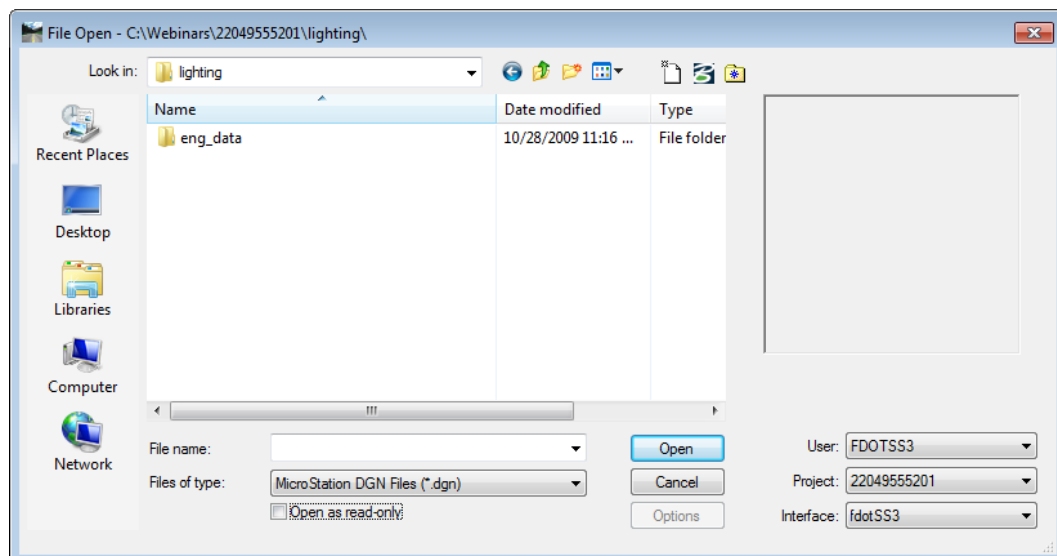
USING CREATE FILE/PROJECT - BASE FILES

The **Create File/Project** application is used to create **MicroStation** design files, Comp book Excel files and other files in accordance with the Department's **CADD** standard file naming conventions. **Create File/Project** uses an ASCII text file, called a Control File (*.ctl) to perform these task(s). This application can also create projects, although Electronic Delivery is the recommended method for creating new projects. **Create File/Project** can be accessed from both inside and outside of **MicroStation**. The advantage to using this tool to create files is that the file will always be named according to the Department's **CADD** standards.

The Figure below shows the **Create File/Project** tool.
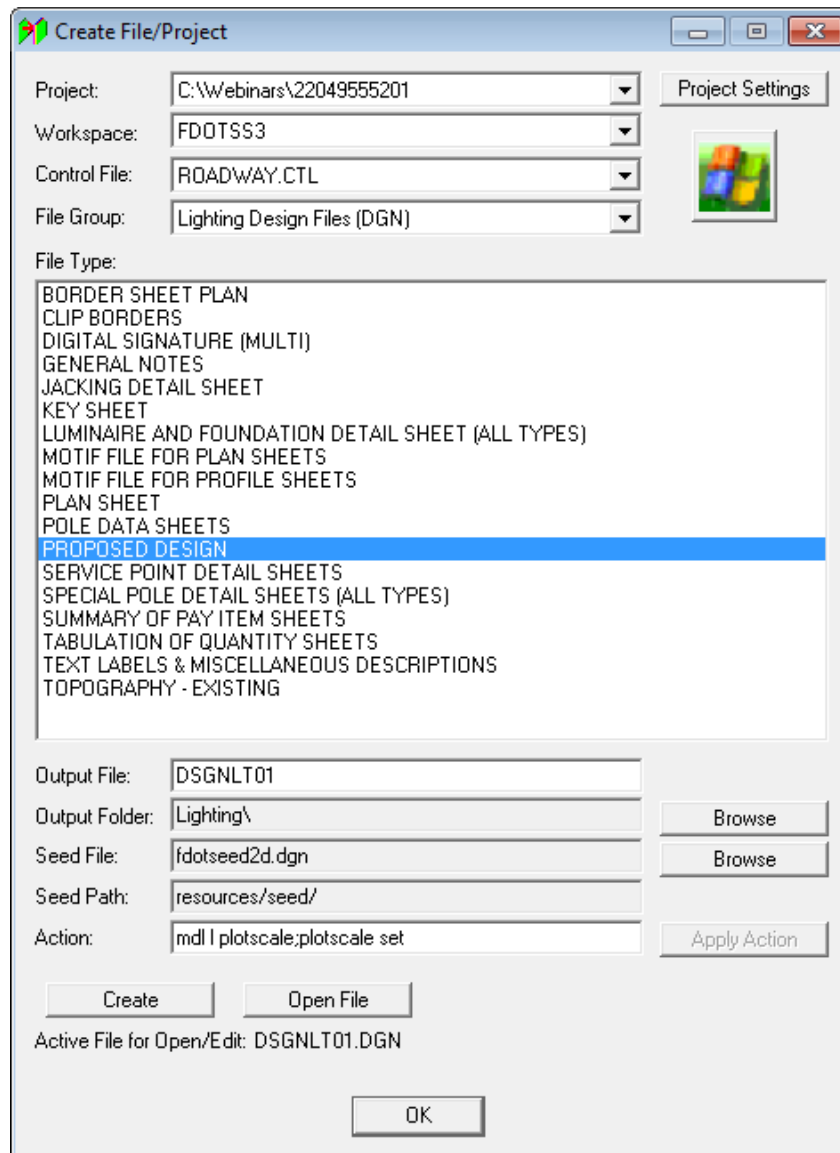
> *Note*   The designer has the option to open the file after it is created or create all the design files and open them later.

Exercise 1.1 *Creating the Base Files for a Project*

➢ *Creating Lighting Project Files*

1. Open the FDOTSS3 folder on the desktop.

2. Double click on the **Create Files** icon . This will open the Create File/Project tool.

Note   When training in the Department's offices, the user must click the Project Navigator button to allow the Create File/Project tool to work locally.

3. On the top right hand side of the Create File/Project tool, select the **Project Settings** button. This will open Project Settings.
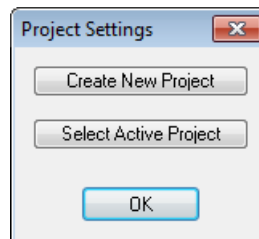
4. **Select Active Project**. This will open the Select Active Project dialog.
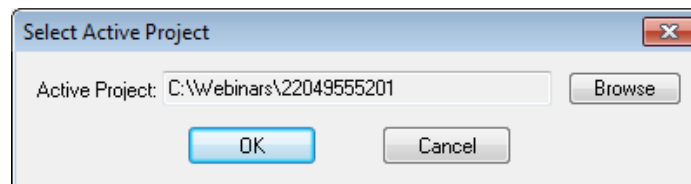
5. Browse to the *Projects* folder and select **22049555201**. Stop at the root folder.
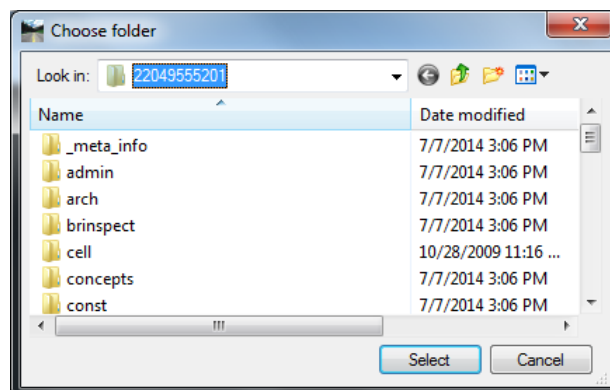
6. Click on the **Select** button. This selects the project and dismisses the Choose Folder dialog and goes back to the Select Active Project dialog.

7. Click **OK** on the Select Active Project dialog.

8. Click **OK** on the Project Settings dialog. This sets the active project.

> ➢ *Load the Appropriate Control File.*

Remember the control file sets, for lack of a better term, the discipline folder to work in. Lighting files are part of the Roadway control file. It is important to know how to do this in case it is required to use a different control file in the future or if the standards ever change.

1. Click the down arrow on the *Control File* dropdown box.

Note   Depending on how the software is installed, the FDOTSS3\RESOURCES\Control_Files folder could be on the server or the local hard drive.
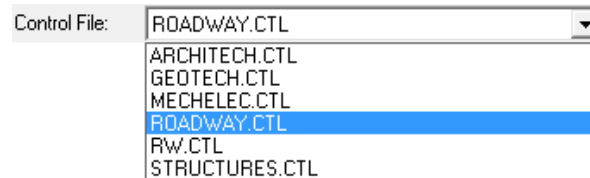
2. Select **Roadway.ctl**. This loads the control file and the Create/Edit application now has enough information to begin creating design files in the specified Output folder.

3. In the *File Group* category, use the drop down arrow to select **Lighting Design Files (DGN)**. This sets the file types to Lighting files and sets the output folder to Lighting.
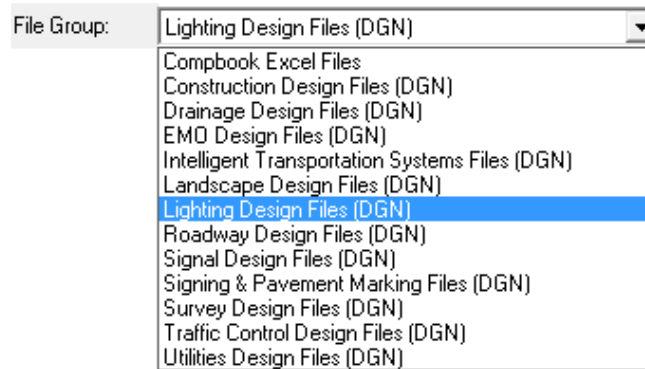
4. In the *File Type* category, select the file **Proposed Design**. This selection populates the *Output File and Seed File* with the correct information.

Note   This version of Create File/Project allows the designer to browse to a different Output Folder if needed. Also, the Workspace field is set to FDOTSS3 by default.
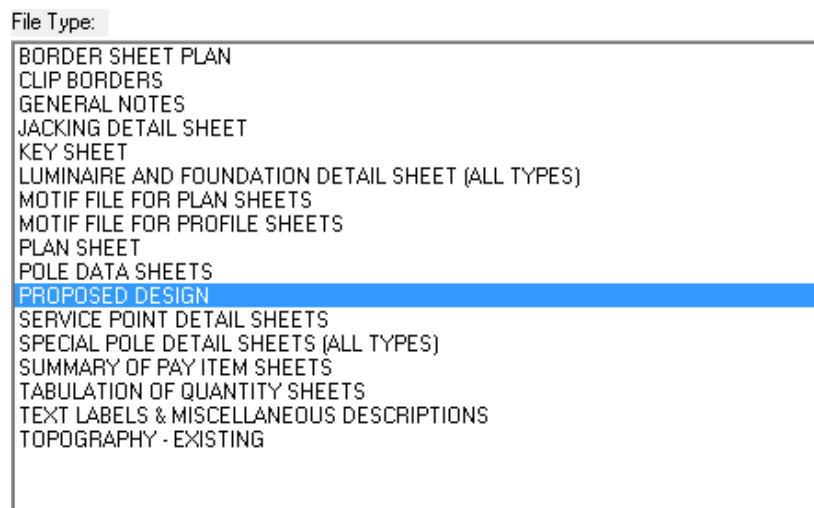
5. Click the **Create** button at the bottom of the dialog.

6. Click the **OK** button to acknowledge the file creation.
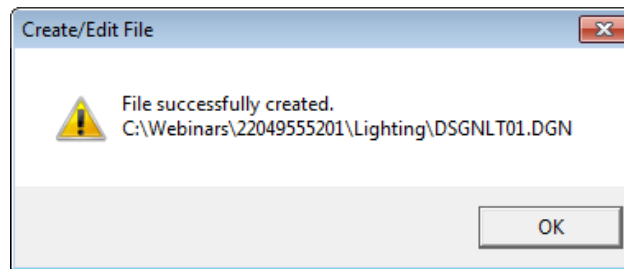
Note Clicking the Open File button next to the Create button at the bottom of the dialog can be used to open the file with this tool, which also registers and checks the file out in Project Navigator when the Create Edit Application is in TIMS mode. Also, notice below the Create and Open File buttons that the file name is shown. If the file Dsgnlt01 already exists in this project, the application will increment the file name to Dsgnlt02 and will not overwrite the file.
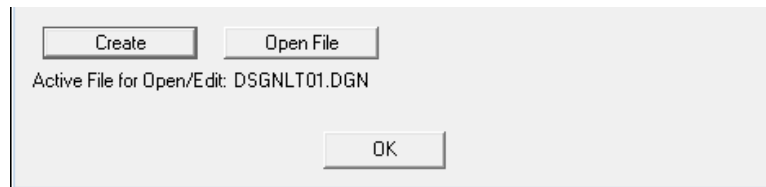
7. Go on to the (*Optional*) Exercise or click **OK** to close the Create File/Project tool.

Exercise 1.2 *(Optional) Create Additional Files*

➢ ***Use same File Group to create additional design files***

1. Using the Create File/Project tool, select the **Tabulation of Quantity Sheets** file.

2. Click the **Create** button.

3. Click **OK** to acknowledge the file creation.

4. Repeat steps 1 thru 3 to create the **Border Sheet Plan** file.

5. Repeat steps 1 thru 3 to create the **Summary of Pay Item Sheets** file.

6. Click **OK** on the Create File/Project tool. This will close the tool.

Exercise 1.3 *Opening MicroStation Using the FDOTSS3 Desktop Shortcuts*

In this exercise, use the FDOTSS3 folder located on the desktop similar to the previous exercise. This folder contains several shortcut icons. It is important to review and understand what each icon is used for.

1. From the FDOTSS3 folder [icon] double click on the icon labeled FDOTSS3. This opens the MicroStation Manager dialog and sets the *Workspace* environment.
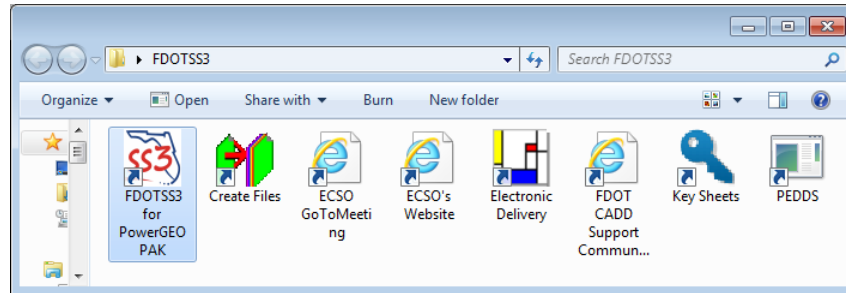
2. In the MicroStation Manager, select the project **22049555201**. This opens the project in the correct root directory.
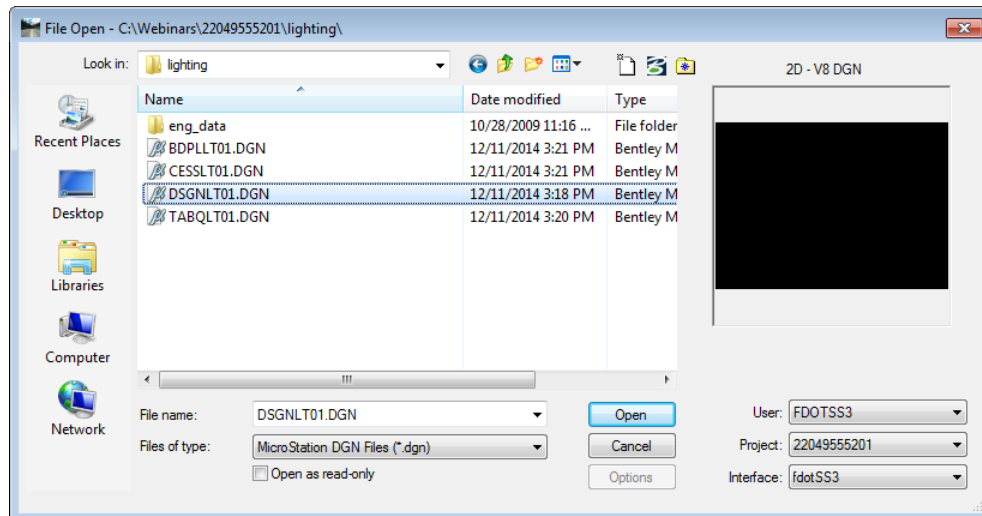
3. Navigate to the *Lighting* folder and select the **Dsgnlt01.dgn** file.

4. Click **Open** or double click on the file to open it

5. Once in MicroStation notice that the FDOT Menu appears. Take a moment to become familiar with this workspace configuration.
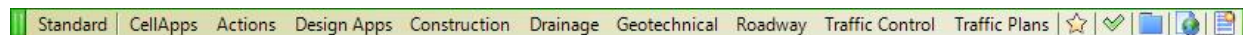
TRAFFIC PLANS MENU & WORKFLOW

ACTIVATING THE TRAFFIC PLANS MENU

As seen in the last exercise, FDOT Menu loads when MicroStation is loaded through the Department delivered FDOT Workspaces. FDOT Menu, by default, opens in the Standard menu which provides an option to configure additional discipline menus as needed.
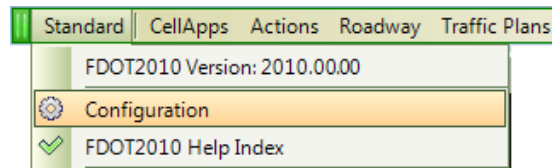
The figure below shows the possible FDOT Menu configurations.
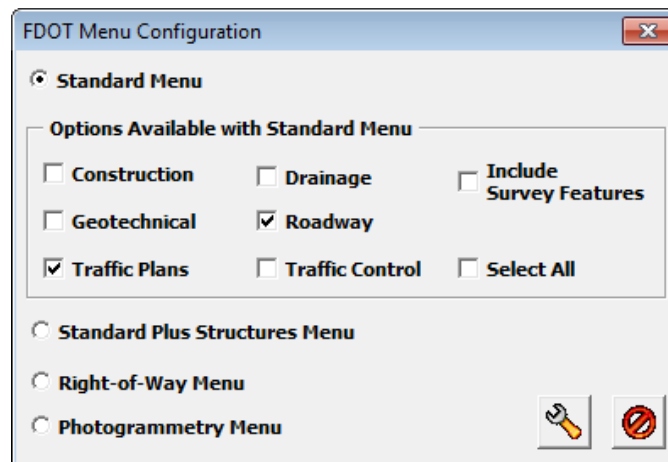
In the top portion of the FDOT Menu **Configuration** dialog provides the option to load one or all of the Roadway menu options. For example, Traffic Plans can be loaded along with Roadway and Typical Sections or it can be loaded by itself.

The Bottom portion of the dialog provides options to load the Structures, Right of Way, or Photogrammetry menus. These menus cannot be loaded at the same time as the Roadway menus. Once the options have been selected click the Update *(wrench icon)* button to load the additional menus or Exit *(slashed circle icon)* to close this dialog without making any changes.

Exercise 1.4 *Activating the Traffic Plans Menu*

1. Continuing in *Dsgnlt01.dgn* select the FDOT Menu option **Standard** from the far left.

2. Select **FDOT Menu Configuration** to open the FDOT Menu Configuration dialog.
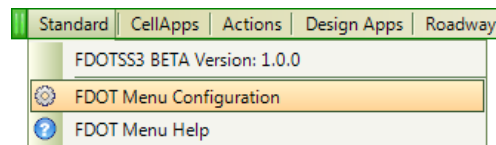
3. Click the option to load the **Traffic Plans** menu. (It's a good idea to activate the Roadway menu as well.)
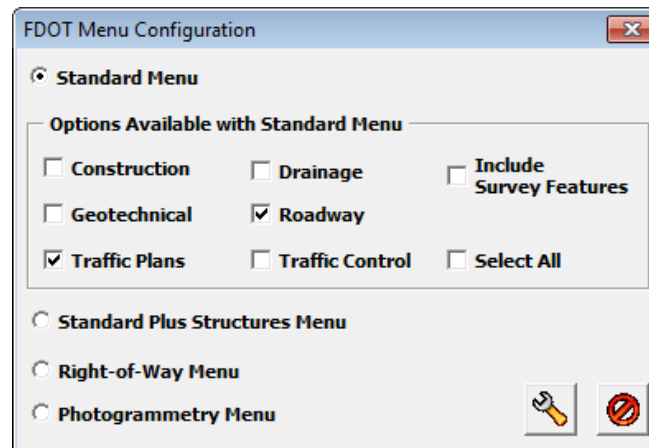
4. Click the **Update** *(wrench icon)* button.

5. Select and hold the **green box** at the front of the FDOT Menu to *drag* to a convenient location.

6. Right-click on the **green box** at the front of the FDOT Menu to access the *Context Menu*.

7. From the *Context Menu*, select **Save Settings** to save the FDOT Menu Bars location.

8. When the standard FDOT Menu is reloaded the Traffic Plans menu is added.

9. Take a moment to become familiar with this workspace configuration and the FDOT Menu.
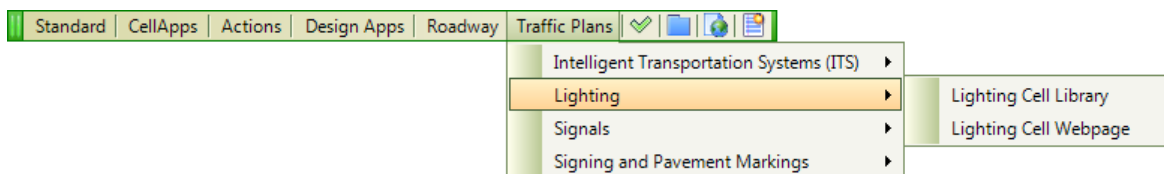
TASK NAVIGATION WORKFLOW - PLANS PRODUCTION > TRAFFIC PLANS

During this intermediary transition to the Task Navigation Menu system, the Department has introduced the basis of a *Plans Production Workflow* for each discipline. This is a location on the Task Navigation Menu System where there is an accumulation of the most common discipline specific tools organized in a common discipline specific, step by step workflow.

Tools located on the FDOT Menu discipline specific submenus may, also, be located in the Task Navigation Menu System.

The change made to the FDOT Menu is an intermediary transition to the Task Navigation Menu system. The Task Navigation Menu System will subsequently replace the FDOT Menu in future releases of the FDOT Workspace/software.
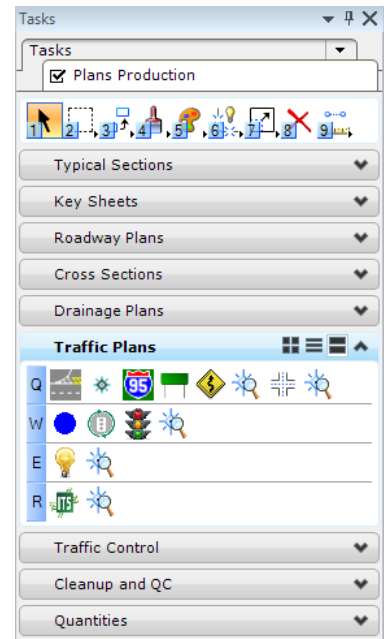
LEVELS, TEXT AND MODELS

LEVELS

The Department has created six Standard Level libraries: Common, Survey, Right of Way, Roadway, Photogrammetry and Structures. There are approximately 1400 levels. The appropriate level library is loaded when a MicroStation file is created or opened using the FDOT Workspaces.

The level name is divided into three components: Level Name, State & View

- The format of the Level / Layer Name is: ***object_sv*** *(max. 18 characters)*

  Where: (***object*** = element type)_ (***s*** = state)(***v*** = view)

(S)tate Designations	(V)iew Designations
p (proposed)	**p** (plan)
d (drafting element)	**r** (profile)
e (existing)	**x** (cross section)
	m (model)

  An example of a Plan View level is: **PoleLight_ep**.

    1. Level Name – **PoleLight**
    2. State - **e**
    3. View - **p**

Note   Some levels do not show a State or View in their name. These levels are set by default to be a proposed plan view element. An example is the level (PullBox) this level is created for proposed Pull Boxes in the plan view.

The levels symbology or color, weight and style are set to ByLevel. What this means is that each level already has a color, weight and style associated to it. Designers cannot create additional levels.

The Department has also created a set of Standard Rules that group the Standard Levels together for the purpose of Quality Control (QC) checking of the Department's Standard design files ensuring compliance with the Department's CADD Standards. The Department delivers MicroStation Filters to reflect each of these Standard Rules to assist in users searching through the level menu. These filters are discussed later in this section. The Standard Level Filters do not turn levels on or off, but they reduce the number of levels visible in the dialog box.

Activating a Standard Level Filter is very easy and can be done from several locations. Level filters can be loaded from the MicroStation Attributes Tool Palette, Level Manager, Level Display dialog, etc. There is a drop down menu as seen in the figure below where level filters can be loaded. This option is only available if "*Active Level Filter*" is selected.
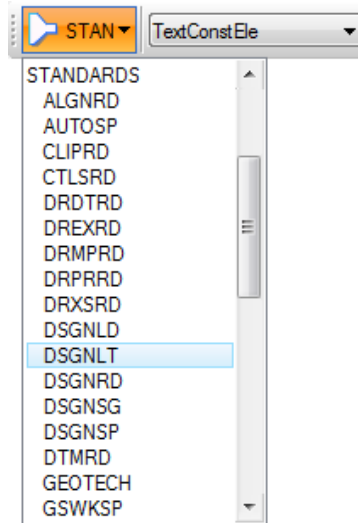
Note   By default the Standard Level Filter is automatically set to specific file names upon opening of the design file.

CIVIL FEATURES

Civil Features define the symbology of elements much like using Place Influence with D&C Manager. The difference is that only the Civil tools use the *Feature Definitions*. Each design Feature has only one definition. Within this definition, symbology is set up for each design View. The tools use the correct symbology for the elements drawn based on the type of model or design View (Plan, Cross Section, Profile, or 3D). The Department has set up the Feature tables using Native Style. Native Style looks in the *.ddb* file for the symbology settings, including the GEOPAK attribute tag and any Adhocs defined in the *.ddb* file.

Feature Definitions are used to control symbology, annotation, and various other properties that are applied to the geometric elements.

The feature definitions are built by using your existing feature table from GEOPAK (DDB), InRoads (XIN), or MX (PSS). These existing feature tables are utilized by way of a link to the file, plus the addition of more settings to enhance capabilities.

The Feature Definitions are used to:

- Define what the geometric elements actually are. What is being modeled such as curb, centerline, edge of pavement, etcetera.
- Control symbology in various views, including capability to define differing symbology in plan, profile, and 3D spaces
- Define terrain modeling attributes (spot, break line, void, etcetera)
- Define surface display characteristics

TEXT STYLES

A *Text Style* is comprised of a group of text attributes, such as font, width, height, spacing and so on. Text styles take away the burden of having to set all of the individual text parameters as required in previous versions of MicroStation. The Department delivers predefined text styles, which should be used whenever possible. The text styles delivered by the Department are created at a scale of 1 to 1. This is important to remember when placing text using Annotation Scale. Annotation Scale is discussed in more detail later in this section.

TEXT STYLES DIALOG

The Text Styles dialog can be accessed by selecting **Element > Text Styles** from the MicroStation menu bar.
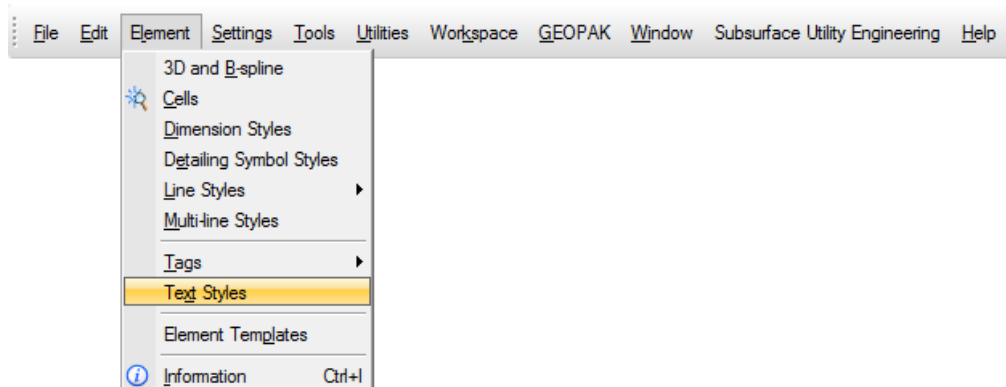
The Text Styles dialog shows all of the text styles that exist in the design file, and all of the parameter settings for each style. When a new file is created or opened an existing file based on the **fdot_levels_v8.dgnlib** all of the appropriate text styles are loaded. No changes should be made to the delivered text styles.
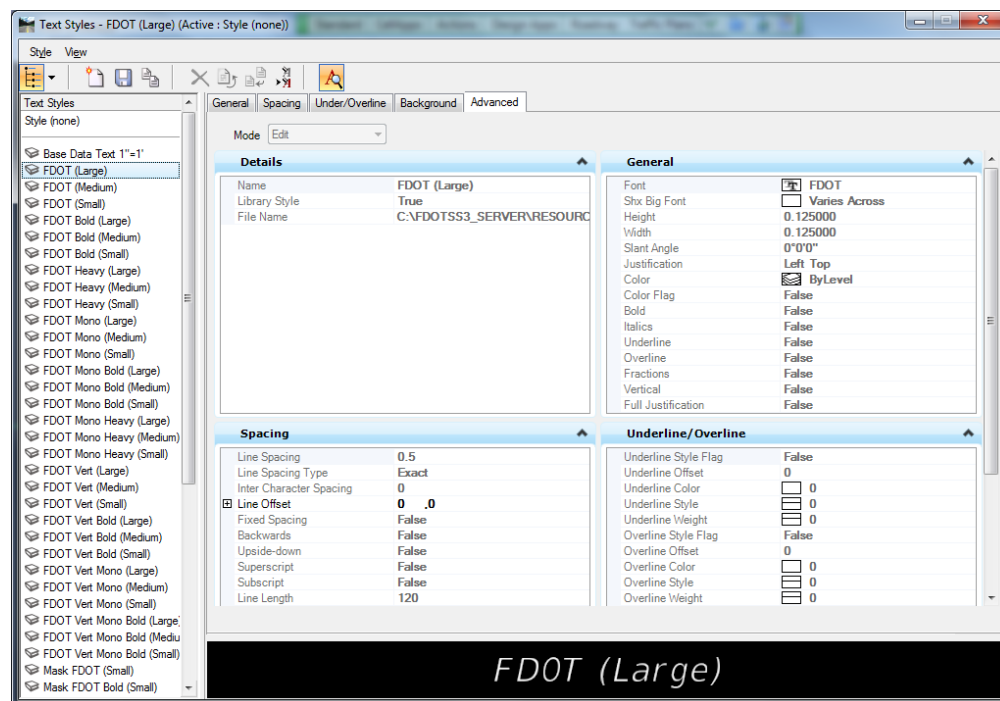
TRUE TYPE FONTS

The Department has integrated the use of True Type Fonts and restructured the Text Styles. True Type Fonts have been implemented for the following:

- Cell Libraries

- DDB (Geopak Database)

- VBAs

- MDLs

- Spreadsheets

True Type Fonts will ignore MicroStation weights. The best method to show thicker text is to use the Bold font. The True Type Fonts delivered by the Department will be installed by the installation and are located in the FDOTSS3\RESOURCES\Fonts\ folder on the server and in the Windows\Fonts folder on the client.

Note   Old fonts will be delivered, but the menus and tools will not be configured to use them.

STYLE ORGANIZATION

Fonts at Common Sizes	40 Scale	50 Scale
Large = .125 x Scale	5.0	6.25
Medium = .1 x Scale	4.0	5.0
Small = .07 x Scale	2.8	3.5

ANNOTATION SCALE

Annotation Scale associates all of the text placed in a model to a specific scale so if the scale of that model is changed the text dynamically changes with it. The Annotation Scale lock must be check on before placing the first piece of text and continue to use Annotation Scale on all of the remaining text for this to function properly.

Using Annotation Scale also removes the burden of having to calculate what text height should be used at a particular scale when placing text. Annotation Scale is model specific.

It is recommended that this be set at the time the model is created and that the Annotation Scale lock is selected. This can also be set in the **Model Properties** as seen in the figure below.

Important items to remember regarding Annotation Scale

- Model Specific

- Annotation Lock must be on from the start

- Can be synced up to the plot scale using FDOT Menu

- Can be set in the model properties

- Don't switch between Annotation Scale on and off
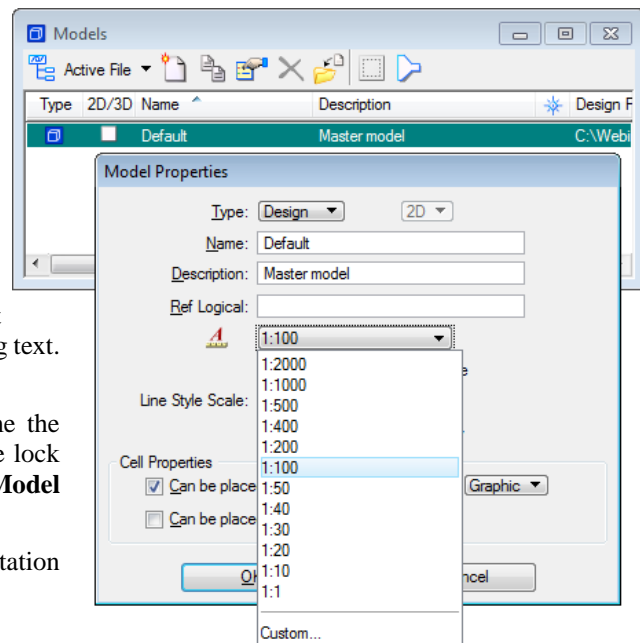
MODELS

Models are independent sets of design data within the same file. Every MicroStation design file has at least one model named Default. Models are used to consolidate files that contain data that is used collectively. An example is the cross section file, this one file (*Rdxsrd*.dgn*) has four models in it.

1. Pattrd – Pattern lines for cross sections
2. Rdxsrd – Cross Sections
3. Xsshrd – Cross Section Shapes
4. Rdxsrd_shg – Cross Section Sheets

This one file with four models replaces four MicroStation design files.

Another example of how to use a model is with the alignment file or (*Algnrd*.dgn*). This file is for displaying the Baseline of Construction. Typically, there could be multiple alignment files to cover all of the different scales. Using the model concept only one MicroStation (*Algnrd*.dgn*) file with several models in it (i.e. a model for each scale) has to be created.

When a new model is created there are several options to define in the model properties, these properties are unique to the model. The figure below shows the Create Model dialog box.
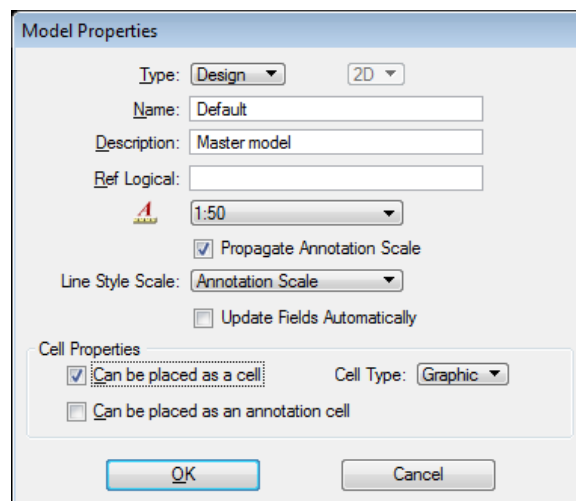
There are two types of models, Design and Sheet. The Design model is the actual geometry or line work and the Sheet model is the sheet file or border. Models can be either 2D or 3D and a 3D model can be referenced to a 2D design file.

> *Note*   If multiple models are used in the DGN files, it is important that their names and uses are documented in the project Journals. This will give downstream users an idea as to what is in the file and whether it is important or not. The same goes for CADD managers who are reviewing QC reports, if temp models are created to work in and the model fails compliancy it may be covered if documented in the project Journal, as long as this Model IS NOT referenced to any other design file.

Exercise 1.5 *Using the Tools*

➢ *Reference Files, MODELS and Plot Scale (Part 1)*

This exercise will cover attaching reference files that have multiple models in them. This reinforces what was discussed earlier regarding models inside of a MicroStation file. There is more than one way to load the References dialog. Use the method most familiar.

1. Continuing with *Dsgnlt01.dgn* open the References dialog box. This dialog is loaded by clicking on the **References** icon on the Primary Tool Bar.
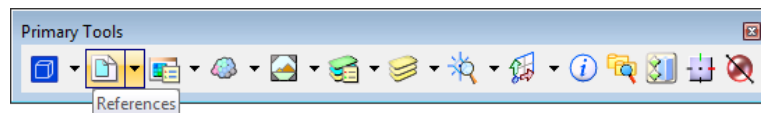
2. In the References dialog, select **Tools > Attach.** This opens the Attach Reference dialog.

3. Navigate to the *Roadway* folder.

4. Attach the reference file **Dsgnrd01.dgn** in the *roadway* folder.

5. Click on **Open**. This opens the Reference Attachment Settings dialog box.

6. Select the **default** *Model* from the drop down list.
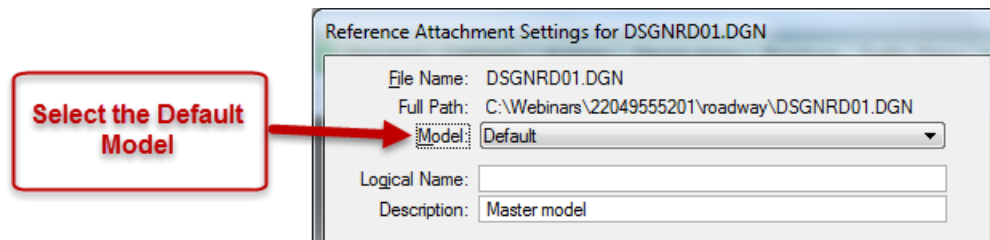
7. Select **Live Nesting with a Nesting Depth of 1** for the *Nested Attachments* setting. This will display the proposed roadway features as well as the *Centerline of Construction without Stationing*.
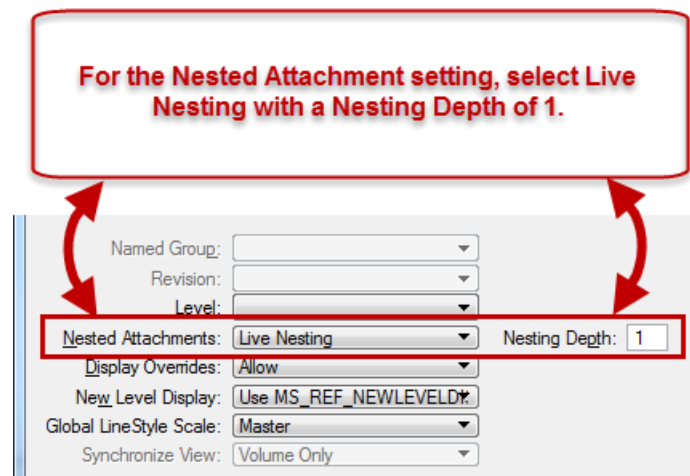
Note   The project alignments have been drawn in the Roadway Design File on the CLConst_dp level, but more importantly it was drawn utilizing the Centerline (CL) Feature Definition. Also, notice there are no Station Tics. Station Tics are drawn in the Alignment file (algnrd*.dgn) at different scales in separate models.

8. Click the **OK** button. This attaches the reference file.

9. In MicroStation do a **Fit View**. This fits the reference file to the active view window.

10. Take a moment and zoom in close to the baseline, notice that there are no tick marks or stations.

11. In the References dialog, select **Tools > Attach.** This opens the Attach Reference dialog.

12. Navigate to the *Roadway* folder and select the reference file **Algnrd01.dgn** in the *roadway* folder.

13. Click **Open**. This opens the Reference Attachment Settings dialog. This is the alignment file that contains the Stationing Tics.
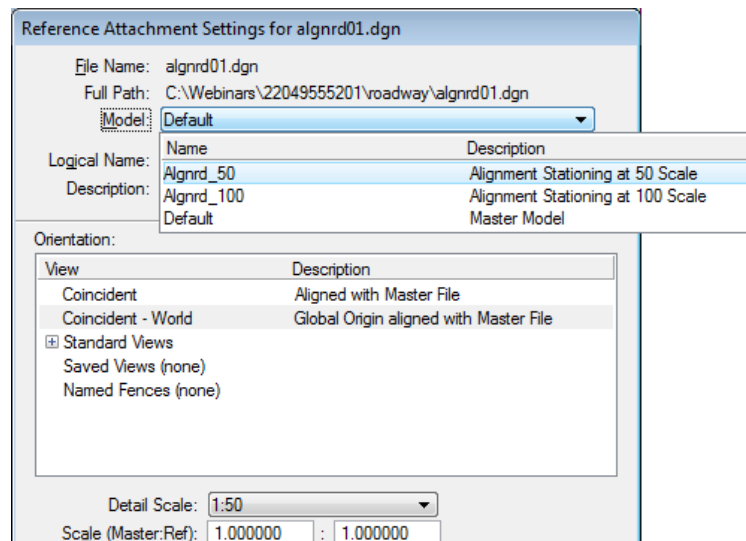
Note   The attachment method should be set to Interactive. Notice the Model name defaults to the Default model. Every MicroStation file has at least one model named Default, except Rdxsrd*.dgn where the default model has been renamed to Rdxsrd. This reference file has additional models to accommodate multiple scales.

15. In the Reference Attachment Settings dialog set the *Model* to **Algnrd_50**.

16. Click the **OK** button. This attaches the reference file.

17. In MicroStation do a **Fit View**. This fits the reference file to the active view window.

Note   The reason for doing it this way is that on Signing and Marking Plans sometimes it's more advantageous to only display the stations and ticks and not the baseline to avoid confusion if close to another pavement marking line or to allow them to be displayed at different scales. This set up makes it easier to turn off the display of the reference file (Algnrd, model BL 100 and/or the referenced Alignment Model in the Dsgnrd file).

18. Zoom in close to the baseline again; notice now that there are tick marks and stations.
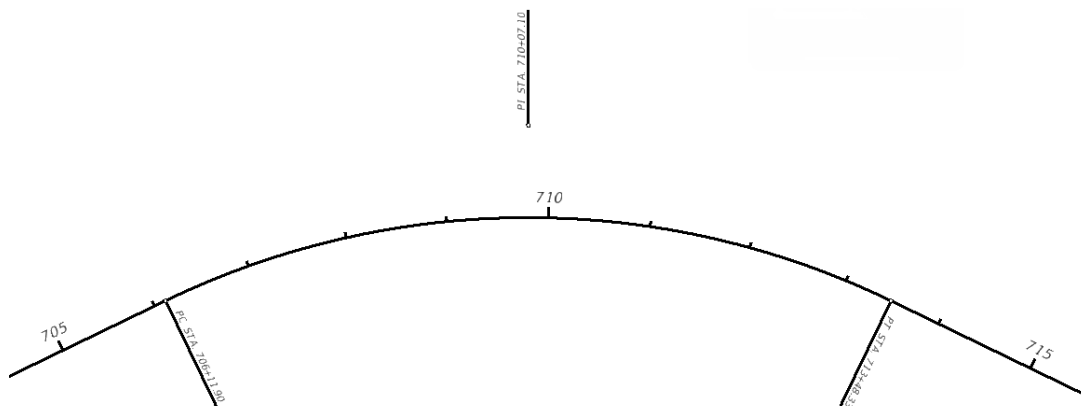
Note   It may be necessary to attach the Topord*.dgn, Utexrd*.dgn and Drexrd*.dgn files depending on the type of project being worked. If there are conflicts with any of the existing features than that reference file should be displayed. Refer to the Plans Preparation Manual Volume II - Chapter 23 for further guidance.

➢ *Using the Tools – Models (Part 2)*

This exercise will cover setting the Annotation Scale for the default model and create a new model in this exercise. This model is a place where to experiment with some tools without adding junk to the default model.

1. Continuing in *Dsgnlt01.dgn* open the Model Properties dialog. This can be opened from the *Primary Tool*.
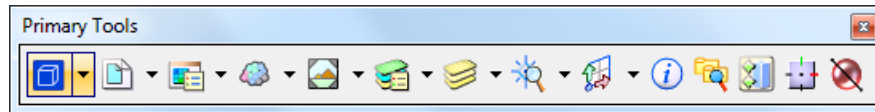
2. In the Model dialog, highlight the **Default** *Model*.

3. Right-click on the **Default** *Model* and select **Properties**. This opens the Model Properties dialog.
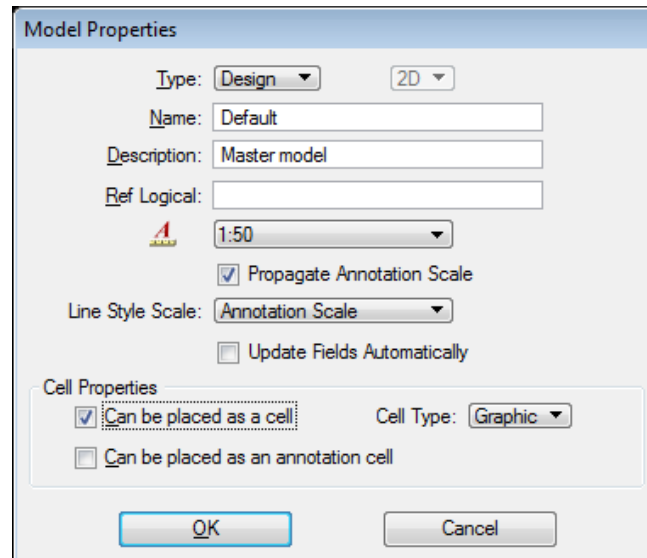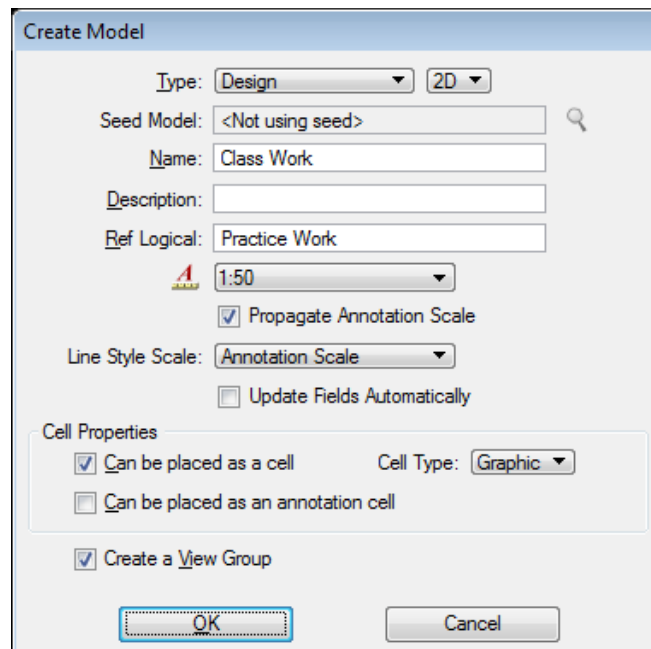
4. For *Annotation Scale* set this to **1"=50'**.

5. Click **OK**. This closes the Model Properties dialog.
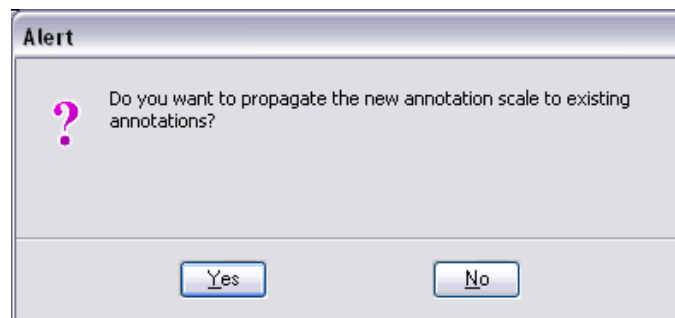
6. Click **Yes** on the Alert dialog (if one appears.). Up to this point nothing is in the design file to change.

7. In the Models dialog, select **Create a new model** ![icon]. This opens the Create Model dialog.

8. Set the *Type* to **Design 2D**. These are the default settings.

9. In the *Name*, enter **Class Work**. This is the *Model* name.

10. *Description* can be left **blank.**

11. For *Ref Logical* enter **Practice Work**. This field, if populated, will fill in the logical name in the reference palette if this file is attached. This is very helpful.

12. For *Annotation Scale* set this to **1"=50'**.

13. Click **OK**. This creates the new model and makes it the active model.

14. Set the *Plot Scale* to **50**.

15. Set the *Units* to **English**.

16. Notice now that the new *Model* shows up in the Model dialog. To switch between models, double click on the model name. The *active model name* will be next to the **View 1** name.
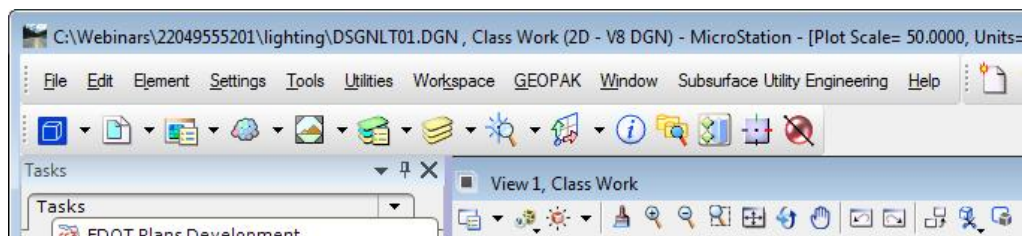
Note   In MicroStation, turn the Annotation Scale Lock on.

➢ *Using the Tools – Levels and Filters (Part 3)*

This exercise will cover investigating Levels and Level Filters delivered by FDOT. The elements placed in this exercise are not part of the final design; they are for experimenting and practicing only.

1. Continuing with *Dsgnlt01.dgn* in the *Class Work Model*, select the **STANDARDS:DSGNLT** option from MicroStation Attributes tool palette. This sets the active level filter.

Note   As a refresher, a level filter isolates the levels seen in the level dialog so it is easier to navigate. The level filters are grouped by discipline and are now set by the filename.

2. In the *Attributes tool palette*, select the drop down arrow next to the *Level Names*.
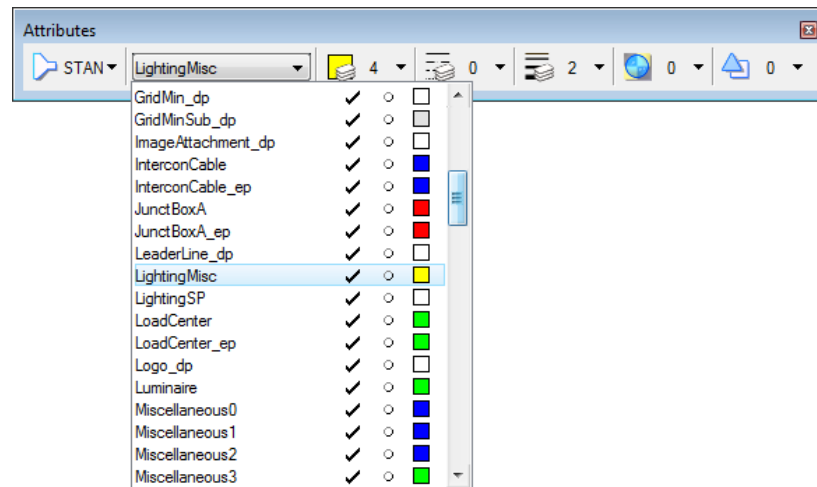
3. Scroll down and select the *Level* **LightingMisc**. This sets the active level to *LightingMisc* and sets the color, Weight and Style because all levels are set up *ByLevel*.

4. In the *Attributes tool palette*, select the drop down arrow next to the *Level Name*.

5. Scroll all the way to the top and select **Filter: STANDARDS:DSGNLT.** This opens the Recall Filter dialog.
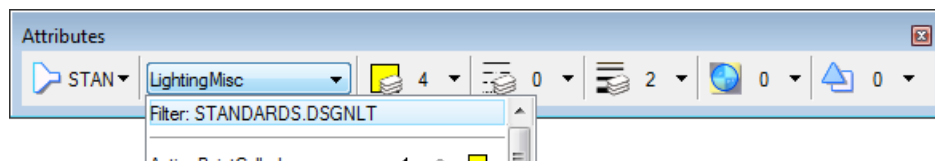
Note   This is another way to load level filters.

6. Scroll down and select **Text (fdot_v8_levels)**.

7. Click **OK**. This loads *Text Levels* filter.

8. Take a moment to review the levels in this filter. Notice that all levels visible are associated to text.

9. Now that the *Filter* is loaded, navigate the level drop down menu and select the level that is appropriate to the item being placed. It is important to understand how useful the level filters are, they will save time when going from Line work to Text and so on.
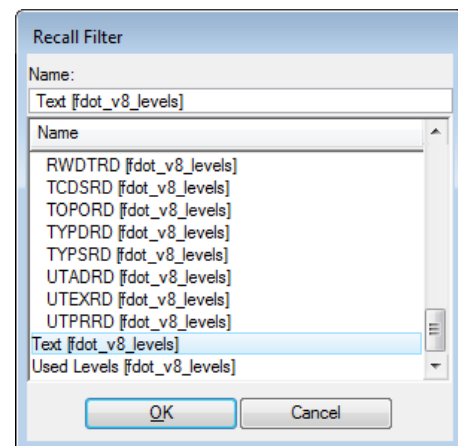
➢ *Annotation Scale and Text Styles (Part 4)*

This is a brief introduction to placing text using the Department delivered Text Styles. This is discussed in more detail later in this course.

1. Continuing with *Dsgnlt01.dgn* in the *Class Work Model*, set the *active level* to **TextLabel.**
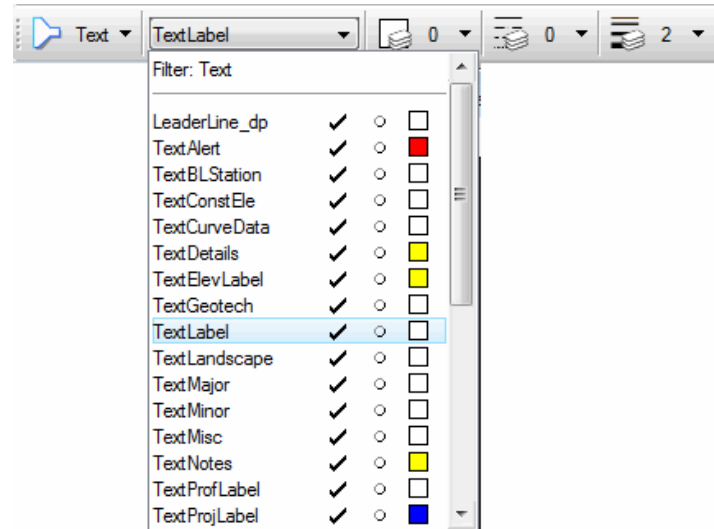
8. From the MicroStation Task Navigator: Main Classic Task, select the **Place Text** tool. This opens the Place Text dialog.
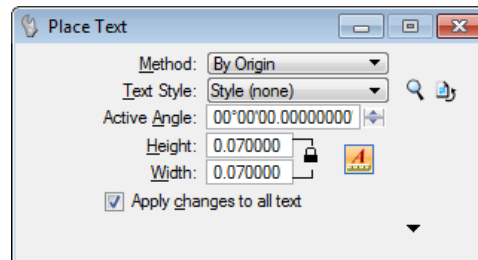
9. Notice that the *Annotation Scale* option is **active**. This was part of our Model Properties set in a previous exercise. Remember with *Annotation Scale* set there is no need to calculate what text height and width to use; this tool automates the calculation.

10. From the *Text Styles* drop down menu, select the **FDOT (Medium)** style.
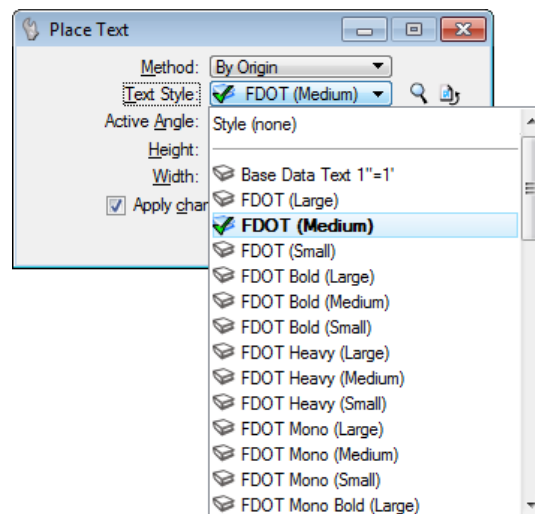
11. In the Text Editor dialog, enter a sample text string and place it in the design file.
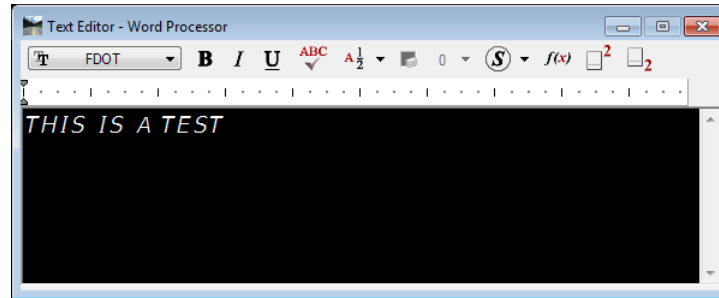
12. From the MicroStation Task Navigator: Main Classic Task, select the **Display Text Attributes** tool

.

13. Select the text just placed and notice the text attributes, the *Height* and *Width* are set to the correct size based on the Annotation Scale. No manual calculation required.
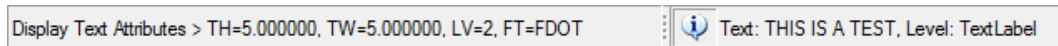
14. Take a moment to become familiar with the Place Text dialog and the other delivered Text Styles.

15. Change the *Model* back to **Default** and select **File > Save Settings**.

©2015 FDOT FDOT Traffic Plans Course **- Lighting**

2   CREATING A KEY SHEET

OBJECTIVE

The objective of this chapter is to teach the student how to create a Lighting Key Sheet that meets the Department's CADD standards.

INTRODUCTION

Once the student has completed this chapter they will be able to create a Key Sheet and all of its components that follow the Department's CADD standards. The Department's CADD standards and the Plans Preparation manual will be adhered to for the creation of this sheet.

GENERAL INFORMATION

The key sheet is the first sheet in the set of construction plans. The information shown on the Lighting plans key sheet varies depending if the Lighting plans are a component of the Roadway plans or the lead component. For example, if the Lighting plans are a component of the Roadway plans, the designer does not need a location map or length of project box because this information is on the lead key sheet. This also applies to the Signalization and Signing & Pavement Markings plans key sheets. Refer to Chapter 3 Volume II of the Plans Preparation Manual for more information.

During the creation of a Key Sheet, the designer is required to take the actions listed below.

Produce the graphical portion of the sheet with these elements:

- Place the standard border cell for a key sheet.
- Place the project location map (only on a lead Key Sheet).
- Place the Florida map cell for a key sheet (only on a lead Key Sheet).
- Place the Section, Township and Range lines (only on a lead Key Sheet).
- Begin Milepost (correct to three decimal places).
- Place the North arrow cell for a key sheet (only on a lead Key Sheet).
- Identify all Railroad Crossings (only on a lead Key Sheet).
- Fill in component Plans (Only on a lead Key Sheet).
- Index of Sheets
- Revision Box

Fill in the project data, including the following:

- Financial Project ID, Number
- Federal Funds (if applicable)
- County Section Number, County Name and State Road Number
- Fiscal Year

PROJECT LOCATION MAP

Florida county maps are available for download from Department's Surveying & Mapping webpage. http://www.dot.state.fl.us/surveyingandmapping/countymap.shtm

These maps are available in DGN or PDF file format. Download the DGN file into the appropriate project directory. This file is used by the Key Sheet Clipping tool to attach, scale and clip the map to the key sheet.

The Project Location Map is placed in the center of the sheet and sized so as not to interfere with other elements on the Key Sheet. Its purpose is to provide enough information so that the project location is easily understood. Township, Range, and County lines and numbers are shown to make the location clear.

USING FDOT MENU TO CREATE A TRAFFIC DESIGN KEY SHEET

The FDOT Menu includes tools to aid in the creation of key sheets with location maps or without. The figure below shows all of the possible scenarios for creating a traffic plans key sheet.
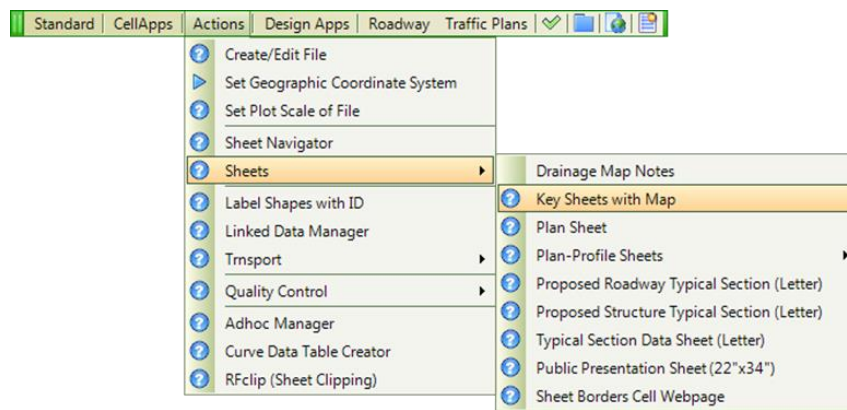
KEY SHEET WITHOUT MAP

In the figure on the above, there is the option to create a Key Sheet with a map. If there is a need to create a Key Sheet without a map, a Key Sheet design file will have to be created first using the Create File/Project tool or some other acceptable method to create the design file using the correct seed file and design file name. In the Key Sheet design file, right click anywhere to access a contextual menu and select *Place Lighting Keysheet* or *Place Lighting Keysheet*, *Federally Funded*.

For Example: If the Lighting plans are a component of the Roadway plans is an example of when to create a Key Sheet without a map.
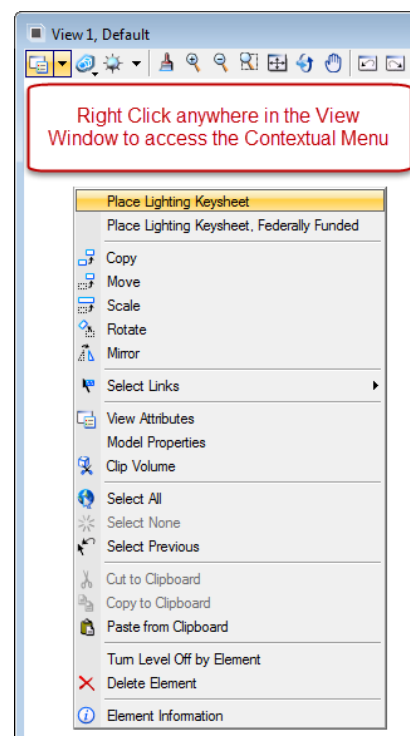
KEY SHEET WITH A MAP

If the project is the lead component, a Key Sheet with a map will need to be created. This option from FDOT Menu opens the *Key Map Sheet Clip* tool. This tool does not require a Key Sheet design file already created; this tool creates it.

Note   This tool by default is set to create the Roadway Key Sheet even if the tool is started in the Lighting folder. It is important to set the Key Map Type first.

- **Scale** - 1"=2miles (10,560), 1"=4miles (21120) or 1"=8miles (42240)

- **Key Map Type** – The list of disciplines for creating key sheets. As the component is selected, the DGN File Name will dynamically change accordingly.

- **Use Federal Funds** – If selected this places the *(FEDERAL funds)* text on the Key Sheet.

  DGN File Name – Displays the working directory path and the design file name of the Key Sheet being created. This dynamically changes based on the Key Map Type setting. The **New** button creates a new file for the sheet cell to be placed in. The file name will default to the next available file number in the standard naming convention for the key map type chosen.

- **Co. Map Name** – This is the county map file downloaded from FDOT's Surveying and Mapping web site. Browse to the file in either in the Roadway discipline folder or the data folder in the project directory. The web site to download the files is:

  www.dot.state.fl.us/surveyingandmapping/countymap.shtm .

- **Clip Area** – This portion of the dialog is for setting the clip limits of the map. Either key in the coordinates, or dynamically select them with the Define Clip Area button.

- **Define Clip Area** - This opens the selected county map so the area to be clipped can be graphically selected. A square will be attached to the cursor. The size of the square is defined by the Scale or Size. Place the square around the area to be clipped and enter a data point. The *Clip Key Map* button will become active.

- **Clip Key Map** – This will clip the map, place the sheet cell, open the file, set the plot scale according to the scale selected, and then launch Sheet Navigator.

Note   If a Key Sheet is created in the roadway directory by error, make sure to delete it to avoid problems with electronic delivery.

WORKFLOWS: PLANS PRODUCTION > KEY SHEET
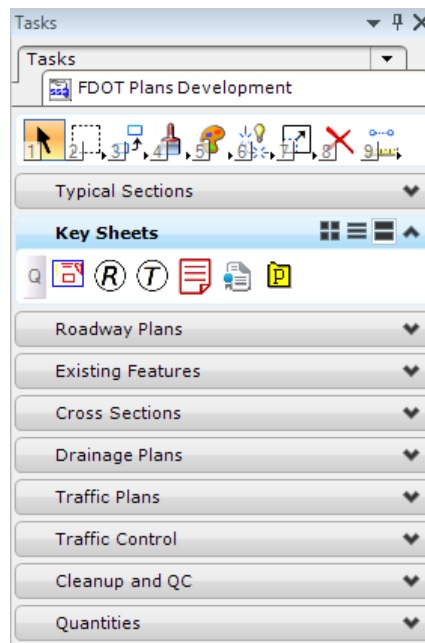
During this intermediary transition to the Task Navigation Menu system, the Department has introduced the basis of a Department Plans Development Workflow for common tasks. This is a location on the Task Navigation Menu System where there is an accumulation of the most common task specific tools organized in a common task specific, step by step workflow.

The change made to the FDOT Menu is an intermediary transition to the Task Navigation Menu system. The Task Navigation Menu System will subsequently replace the FDOT Menu in future releases of the FDOT Workspace/software.

Tools from left to right: Create Key Sheet, Place Range Map Label, Place Township Map Label, Place Key Sheet Revision Note, Place Key Sheet Engineer of Record Information and Place Key Sheet Strung Project Note.

INDEX OF PLANS

The *Index of Plans* is used to describe what sheets and corresponding sheet numbers are in the set of plans. The *Index of Plans* is placed through the Linked Data Manager (LDM) from the FDOT Menu > Actions > Linked Data Manager. This tool will still utilize the *Index of Plans* text file, however; LDM will create data links that will keep the index data on the Key Sheet up-to-date.
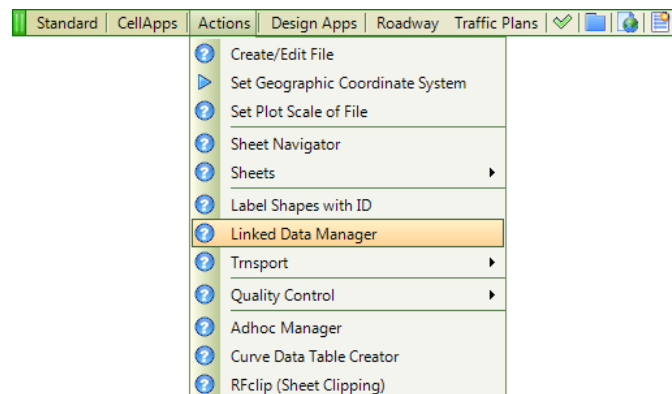
The figure below shows the Lighting Index text file opened in the text editor.

The text height and width of the index text, when placed on the Key Sheet, is generally set to 369.6 which is the default text size for a Key Sheet that is at a scale of 1" = 1 Mile. Though the **Key Map Clipping** tool sets the text height and width values you may desire to check the program for accuracy or modify these values based on the scale of your Key Sheet.

For Example, if your Key Sheet is a scale of 1" = 2 Miles you would multiply 369.6 x 2 to get 739.2. Another way to look at this is to take the text size of 369.6 and divide it by 5280 you would get 369.6 / 5280 = 0.07. This, 0.07, is the desired text height for a 1 to 1 scale. Take 0.07 x 10560 = 739.2 where 10560 is 2 miles.

Once the *Index of Plans* has been placed with LDM the *ksspindex.txt* file placed in the data folder in the project directory can be edited. The changes will be updated in the DGN automatically when the Key Sheet DGN is reopened in MicroStation. The text size placed by LDM is set up to read the Drawing Scale of the active design file and will place correctly as long as the drawing scale is set correctly in the active file.

COMPONENTS OF CONTRACT PLAN SET

The *Components of Contract Plan Set* is a list of all disciplines that are a component to the lead project. The order of the component plans is:

1. Roadway
2. Signing and Pavement Marking
3. Signalization
4. Intelligent Transportation Systems (ITS)
5. Lighting
6. Landscape
7. Architectural
8. Structures

Note   If your project includes Signing and Pavement Markings, Signalization or other component sheets as part of the Roadway Plans and those sheets are numbered consecutively within the Roadway Plans then these are not to be shown as components of the contract plans set.

The *Component Set* is placed through the Linked Data Manager (LDM) from the FDOT Menu > Actions > Linked Data Manager. This tool will still utilize the *Components Set* text file, however; LDM will create data links that will keep the index data on the Key Sheet up-to-date.

Once the *Component Set* has been placed with LDM the *krdcpnts.txt* file placed in the data folder in the project directory can be edited. The changes will be updated in the DGN automatically when the Key Sheet DGN is reopened in MicroStation. The text size placed by LDM is set up to read the Drawing Scale of the active design file and will place correctly as long as the drawing scale is set correctly in the active file.
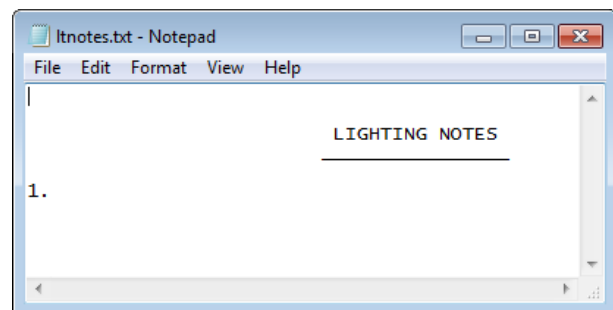
This file is set up for the lead key sheet in the set of plans. If a Lighting Key Sheet without a map is created, this file will not be used because there are no components; hence, there is no *COMPONENTS OF CONTRACT PLANS SET* option on the key sheet.

STRUNG PROJECTS NOTE

Projects that are independently prepared, but are let in the same construction contract shall have the additional Financial Project IDs noted on the right side of the key sheet below the Plans Prepared By block. This cell can be placed from **FDOT Menu**. This tool reads the plot scale for text size.

The *Strung Projects Note* is placed through the LDM from the FDOT Menu > Actions > Linked Data Manager. This tool will still utilize the *Strung Projects Note* text file, however; LDM will create data links that will keep the index data on the Key Sheet up-to-date.

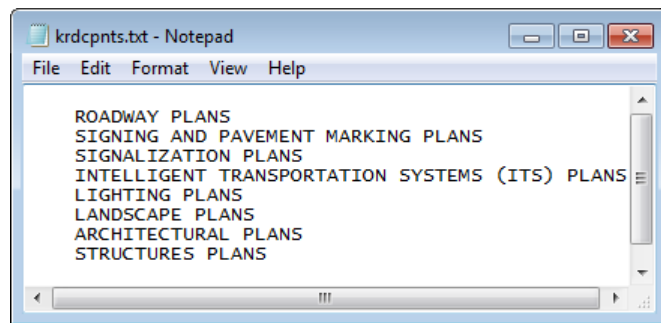Once the *Strung Projects Note* has been placed with LDM the *knletwcontract.txt* file placed in the data folder in the project directory can be edited. The changes will be updated in the DGN automatically when the Key Sheet DGN is reopened in MicroStation. The text size placed by LDM is set up to read the Drawing Scale of the active design file and will place correctly as long as the drawing scale is set correctly in the active file.

NOTE: THIS PROJECT TO BE LET TO CONTRACT WITH FINANCIAL PROJECT ID 000001-1-52-04

Note   The FPID numbers in this note are part of a default note.

TOWNSHIP AND RANGE MAP LABEL

The *Township and Range* are used to better describe the area of the project. On the Key Sheets Workflow there are tools to help place these labels.
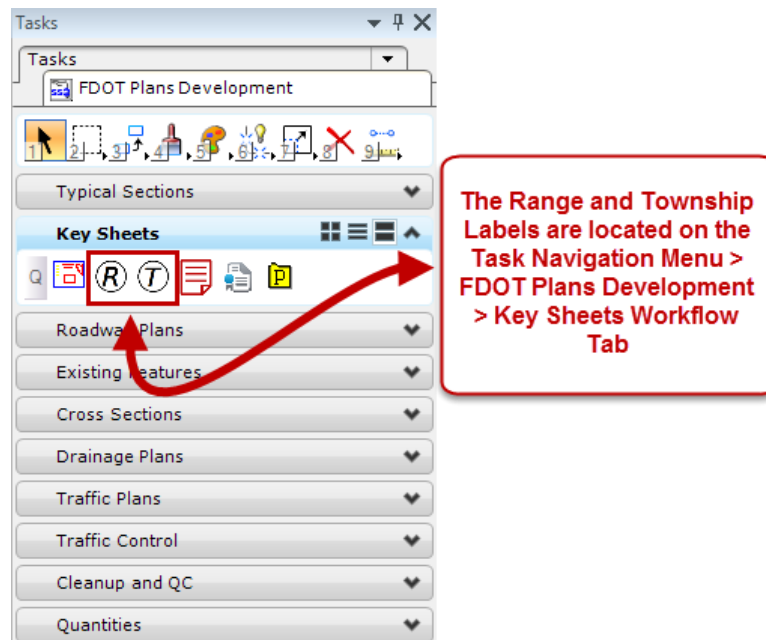
The Township and Range labels are cells with data fields in them, again do not drop the cell to edit the text. The cells are placed based on the plot scale. The figure below shows the Township and Range cells placed from FDOT Menu.
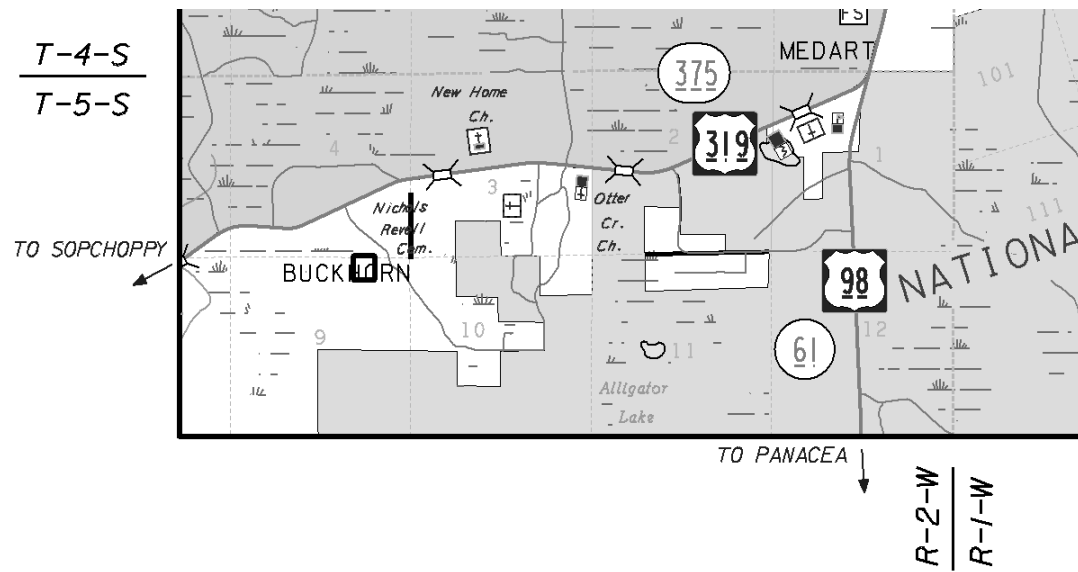
FLORIDA MAP

The Florida map is a cell that can be placed from FDOT Menu select CellApps > FDOT Cell Libraries > *Roadway.cel* library. The Florida Map cell is to be located in the upper right hand corner of the key sheet. This map is only needed on the key sheets without a location map.

> *Note*   Refer to the Plans Preparation Manual Volume II - Chapter 23.2 for more detail.

The insertion point is the upper right hand corner of the border and the cell scale is based on the plot scale. A leader line and text label pointing at the general location of the project are also required.
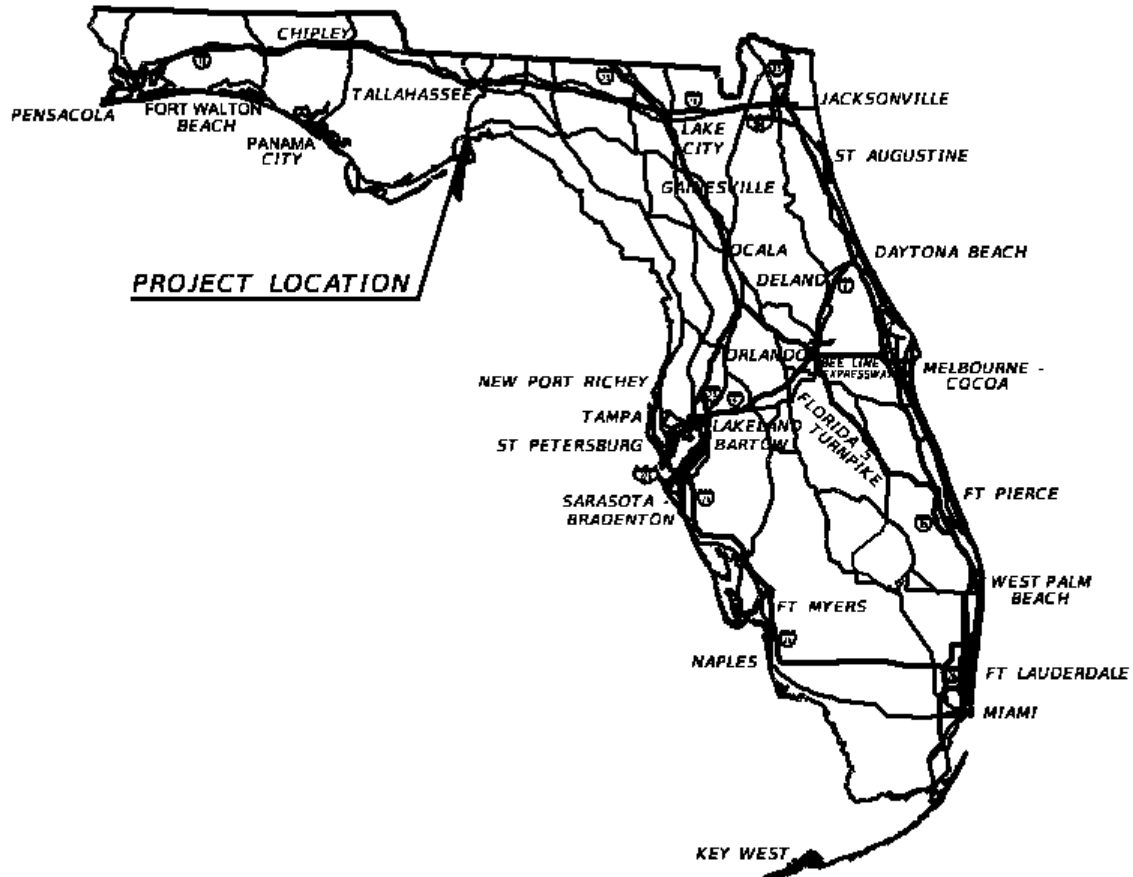
SHEET NAVIGATOR

Sheet Navigator is a tool developed for the Department to aid in the proper identification of plan sheets in a construction set of plans. Sheet Navigator will tag each sheet with information pertinent to the electronic/digital delivery process. Sheet Navigator can also number and renumber sheets in a set of plans.
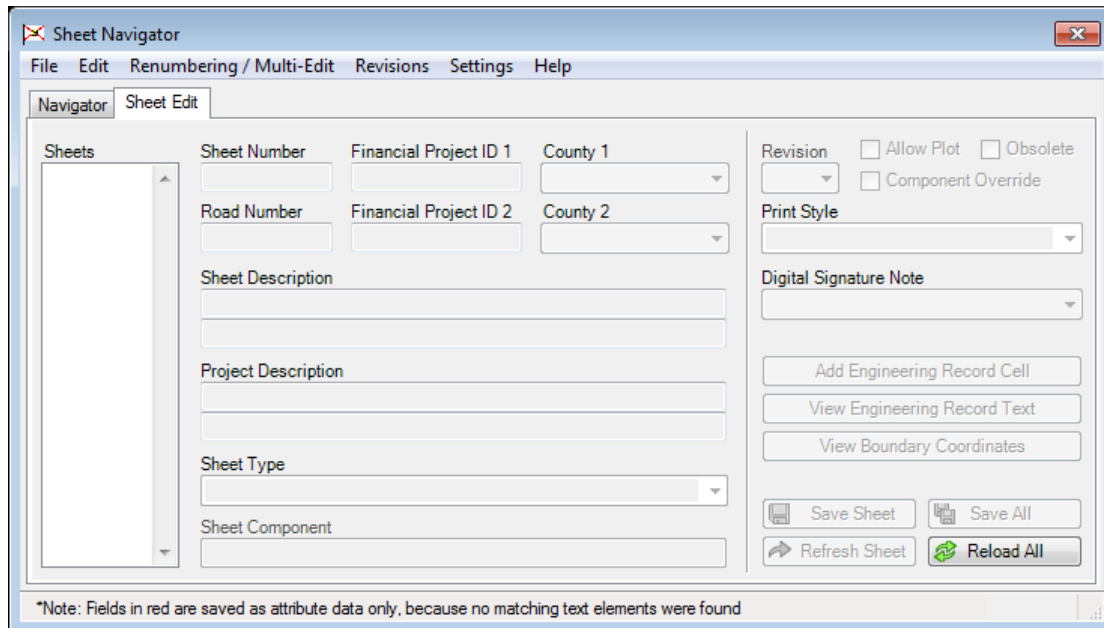
It is important to remember not to put sheet borders in design files that are not going to be part of the plan set. Sheet Navigator along with the Electronic Delivery Indexer (EDI) will look for all files that have sheet borders in them and will tag them as sheets. If the designer absolutely has to put a sheet border in a file that is not part of the plan set, then check the **Obsolete** option in this dialog so that indexer will not select the file as a sheet file. This tool will be covered in some detail later in this course guide when clipping plan sheets.

Exercise 2.1 *Creating a Key Sheet with a Location Map*

> ### *Creating a Key Sheet With a Location Map (Part 1)*

1. Open *Dsgnlt01.dgn* in the *Lighting* folder.

2. From FDOT Menu, select **Actions > Sheets > Key Sheets with Map**. This opens the Key Map Sheet Clip dialog.
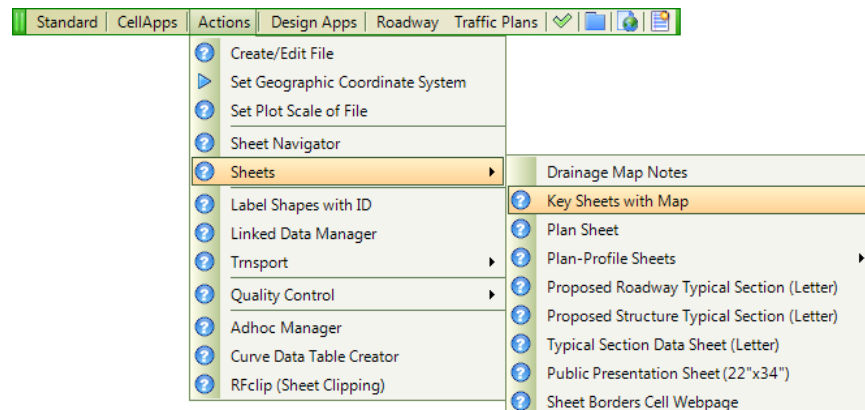
3. Set the top portion of the dialog as follows:

- Scale – **1"=2miles**

- Key Map Type – **Lighting**

- Check to Use Federal Funding
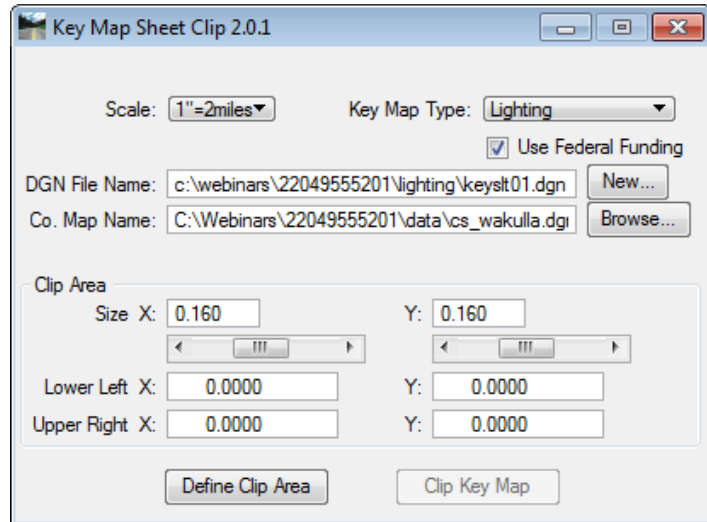
4. Click the **New** button next to the *DGN File Name*. This will open the Key Map Sheet File dialog. This dialog allows the designer to enter a different file name and select a different folder.
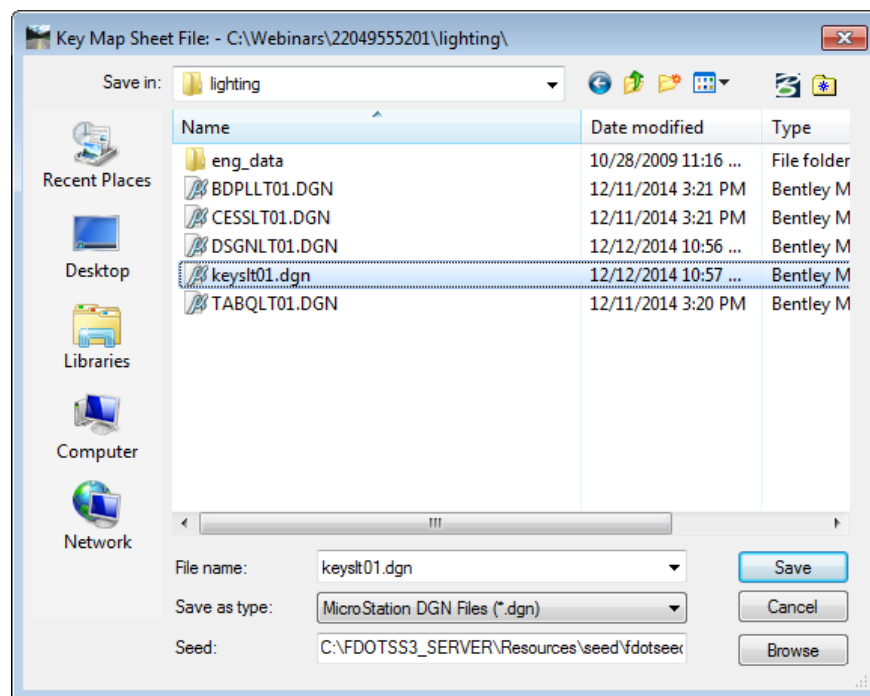
5. Click the **Save** button. This will create the **Key Sheet File**; however, it is an empty file.

6. Click the **Browse** button next to *Co. Map Name*. This is where the designer can select the full county map MicroStation file downloaded from the Survey and Mapping web site.

7. Navigate to the **data** folder in the project. The location that the county map is located in will vary from location to location. The data folder is not necessarily the folder the file will be found in at each individual's office.

8. Select the **cs_wakulla.dgn** file and click **Open**. This will populate the *Co Map Name* field in the Key Map Sheet Clip dialog.
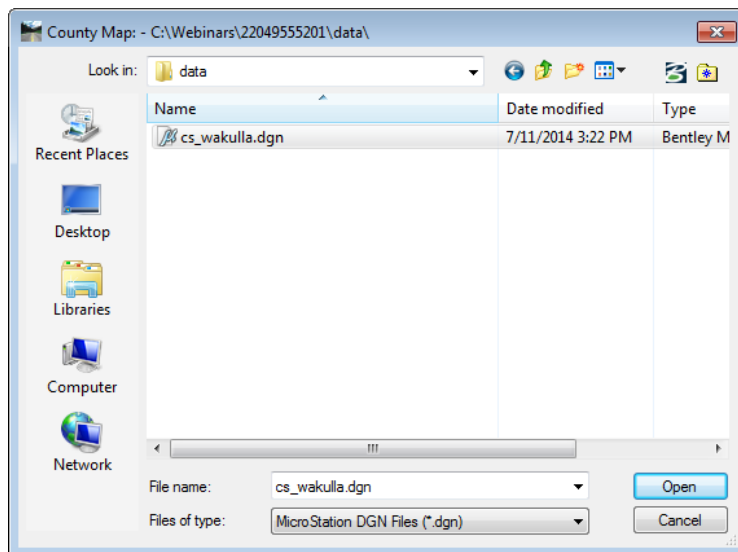
9. The figure below shows how the dialog should look up to this point. Notice all of the fields are populated.
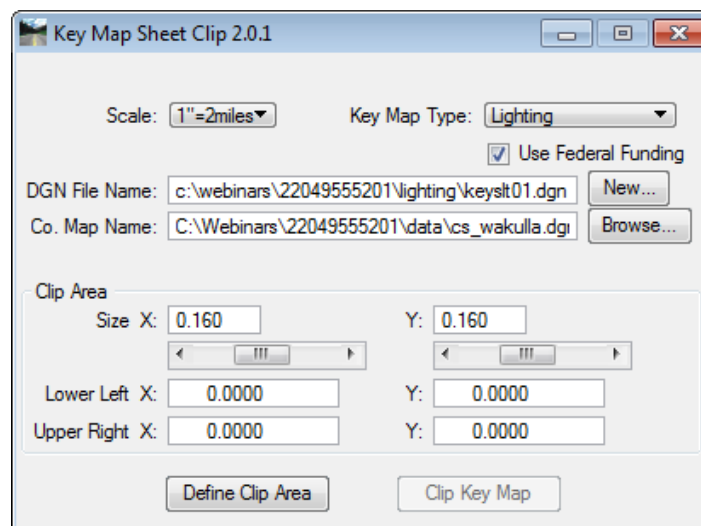
10. Click the **Define Clip Area** button. This opens the county map design file to allow the *Defined Clip Area* to be positioned around the area to be clipped.

11. In MicroStation, **pan** to an area where the clip border will have maximum coverage. This area is not important for this exercise; it is a general location.

Note   Use any of the MicroStation zoom or pan tools to move around the file.

12. Issue a **Data Point** to select the clip area.

Note   To redefine the clip area, click the Define Clip Area button to select a different clip area.

13. Click the **Clip Key Map** button. This will process for a moment and open the key sheet. Sheet Navigator will also open requiring user input.

➢ *Sheet Navigator (Part 2)*

1. Continuing in *keyslt01.dgn*, the *plot scale* should be set automatically to **10560.** This is equal to a **1"=2mile** *scale*.

Note   It is very important to set the plot scale before completing Sheet Navigator; the Digital Signature Note is dependent on the scale.

2. In Sheet Navigator fill in the *Sheet Number* with **L-1**.

3. For the *Financial Project ID 1*, click inside the **blank** filed. This will populate the field with the correct *Financial Project ID*.

4. For the *County* select the drop down arrow and navigate to **Wakulla** County.

5. For the *Road Number* type in **61**. Do not include **SR** in the field; it is part of the sheet file.

6. For the *Digital Signature Note* select the drop down then select **Standard**.

7. Click the **Save Sheet** button. This will tag the file with all of the pertinent information for creating the EDI and populates the fields in the key sheet.

8. In Sheet Navigator, select the **Navigator** tab.

9. Select **File > New**. This opens the Build Index dialog.
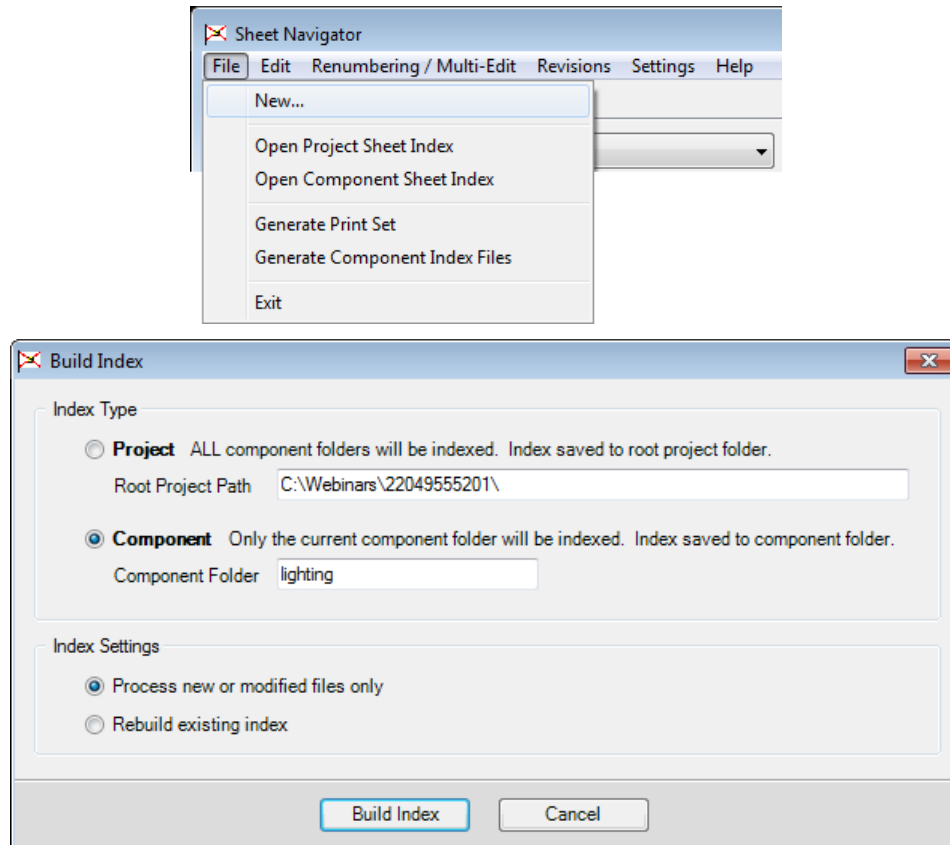
10. Select the **Component** option.

11. Click the **Build Index** button. This creates *sheetinfo.xml* in the *lighting* folder. All other discipline folders are ignored.

12. MicroStation will close and the CADmanage (CM.Batch) process will run.
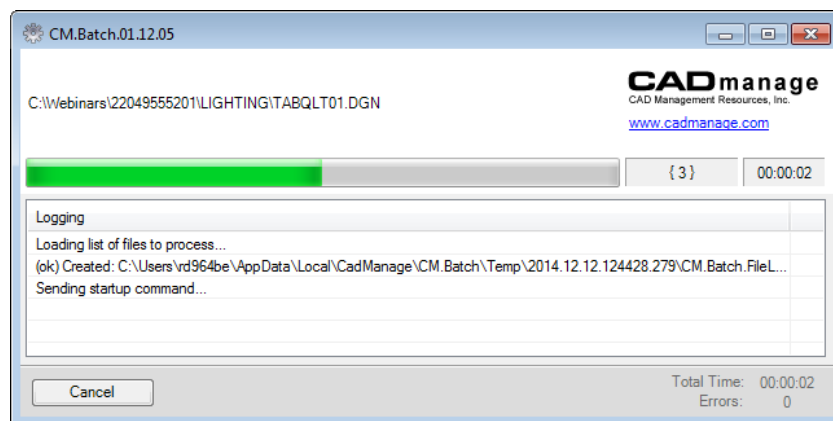
13. When the CM.Batch is complete, MicroStation will be open and Sheet Navigator will display the new **Component Index** on the *Navigator* tab.
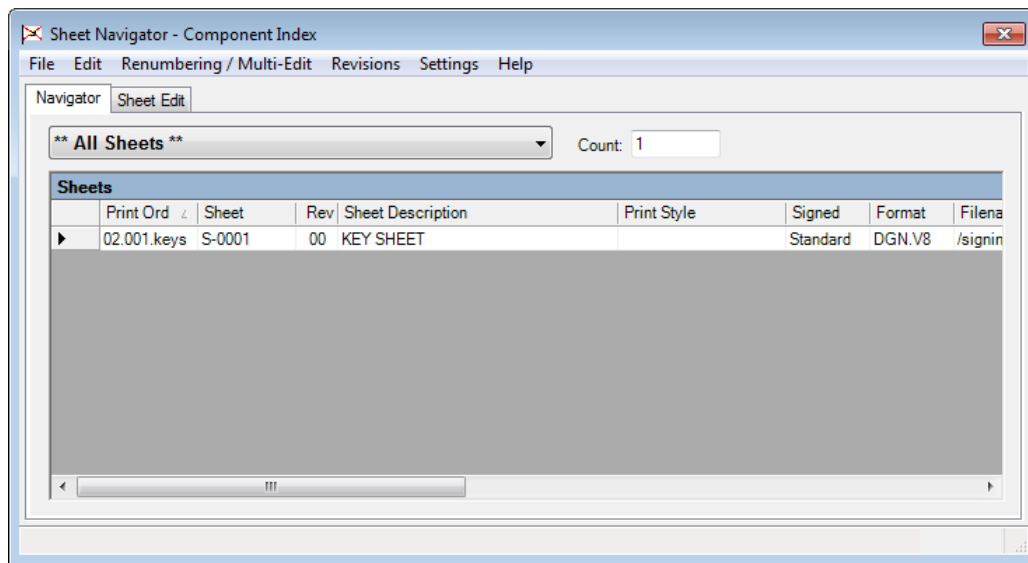
14. Select **File > Generate Component Index Files**. This opens the Generate Component Index Files dialog.
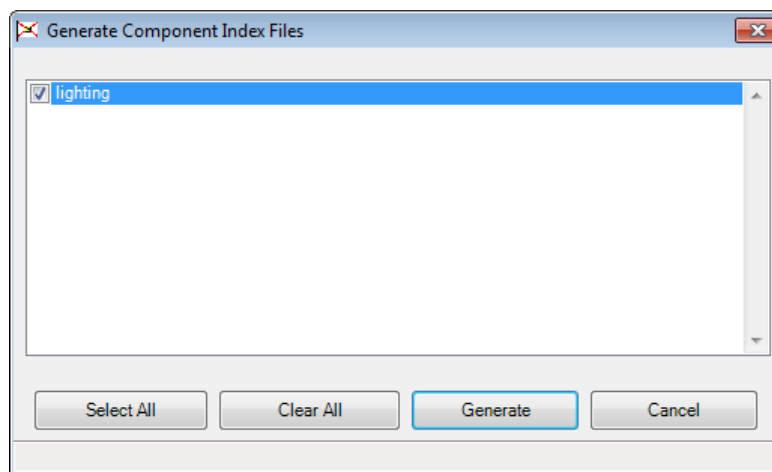
15. Toggle On the check box for lighting.

16. Click the **Generate** button. Sheet Navigator will create 1 component index file.

17. Click the **Ok** button.

18. Click the **X** in the upper right hand corner to close Sheet Navigator.

19. Take a moment to review the key sheet.
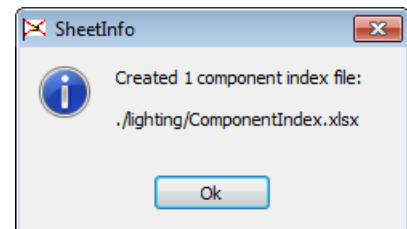
> ➢ *Create LDM Links and Add Index of Sheets (Part 3)*

1. Continuing in *Keyslt01.dgn*, zoom in around the text "INDEX OF LIGHTING PLANS". This is on the left hand side of the sheet.

2. From the FDOT Menu, select **Actions > Linked Data Manager**.
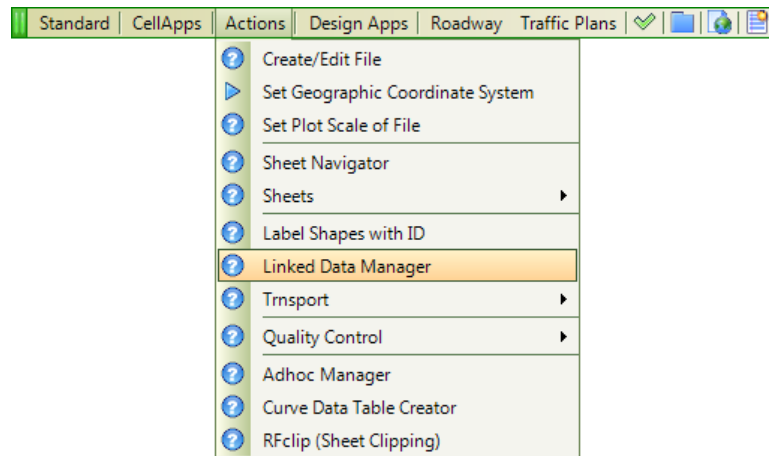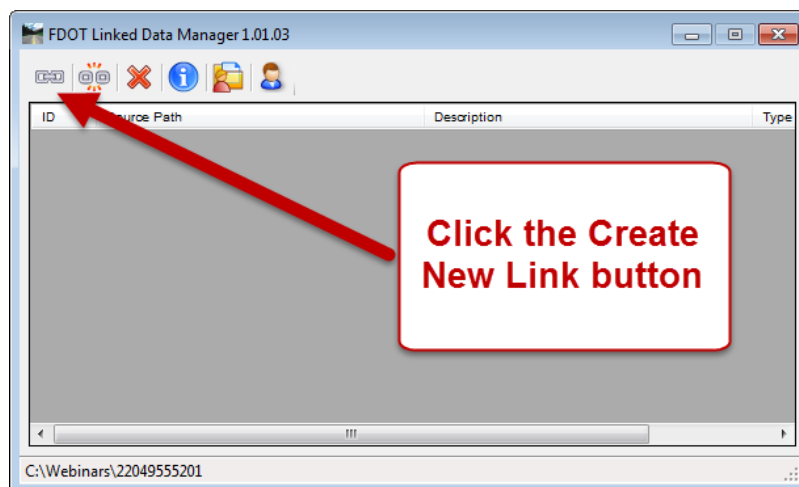
3. Click the **Create New Link** button.

4. Click the **Select Existing File** button.

5. Select the **ComponentIndex.xlsx** and click the **Open** button.

6. For the *Excel Settings*, set the *Worksheet* to **Sheet 1** dialog as shown below.
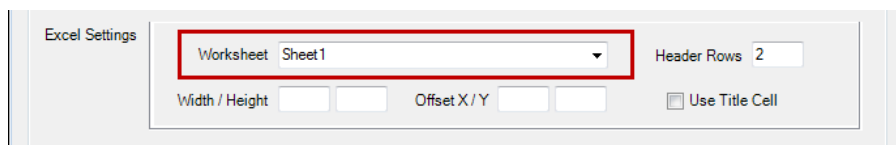
7. Set the *Graphic Settings* as shown below.
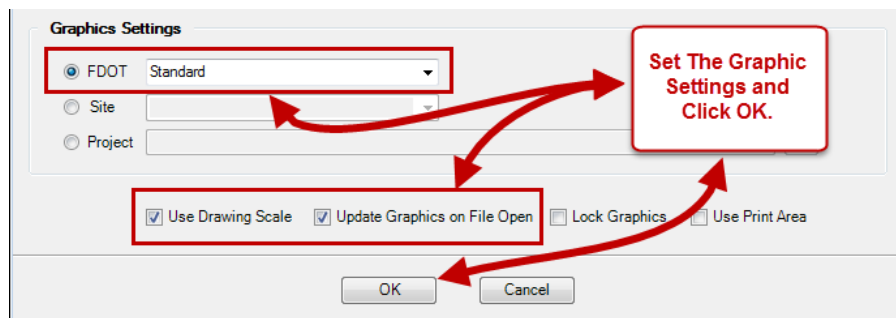
8. Click the **OK** button.

9. Snap to the origin of the *Text Label Index*. This will line up the text properly.

10. Issue a **Data Point** to place the text.

11. Close the LDM.

12. Take a moment to review the Key Sheet.

> ➢ *Creating a Key Sheet with a Location Map (Part 4)*

1. Continuing in *Keyslt01.dgn*, zoom in around location map in the center of the sheet.

2. From Task Navigator, select **FDOT Plans Development > Key Sheets > Range Map Label**. This attaches the *Range* cell at the active plot scale.
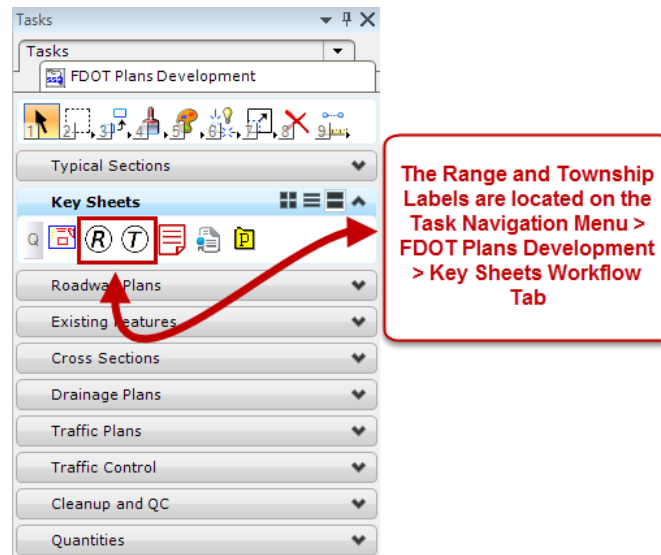
3. Locate where to place the label and issue a **Data Point** to place the *Range* cell. This cell contains data fields so it is not necessary to drop it, use the edit data filed tool to edit the label.

4. From Task Navigator, select **FDOT Plans Development > Key Sheets > Township Map Label**.

5. Locate where to place the label and issue a **Data Point** to place the *Township* label.

6. The figure below shows what the map with labels looks like. These labels come with the default text built into them and they will have to be changed based on the project.
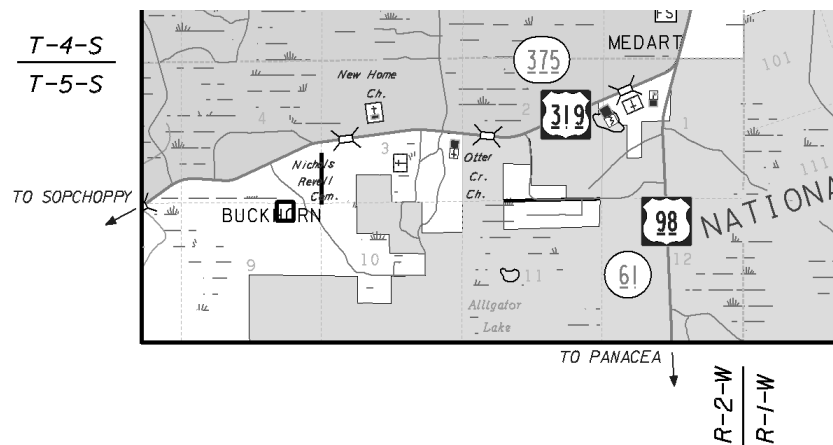
➢ *Placing Project Location Label (Part 5)*

In this exercise the student will draw a leader line with arrow pointing at the general location of the project on the Florida Map then the note "LOCATION OF PROJECT" will be added. This exercise requires the student to use some concepts covered earlier in the course.

1. Continuing in *Keyslt01.dgn*, zoom to the Florida Map area of the key sheet.
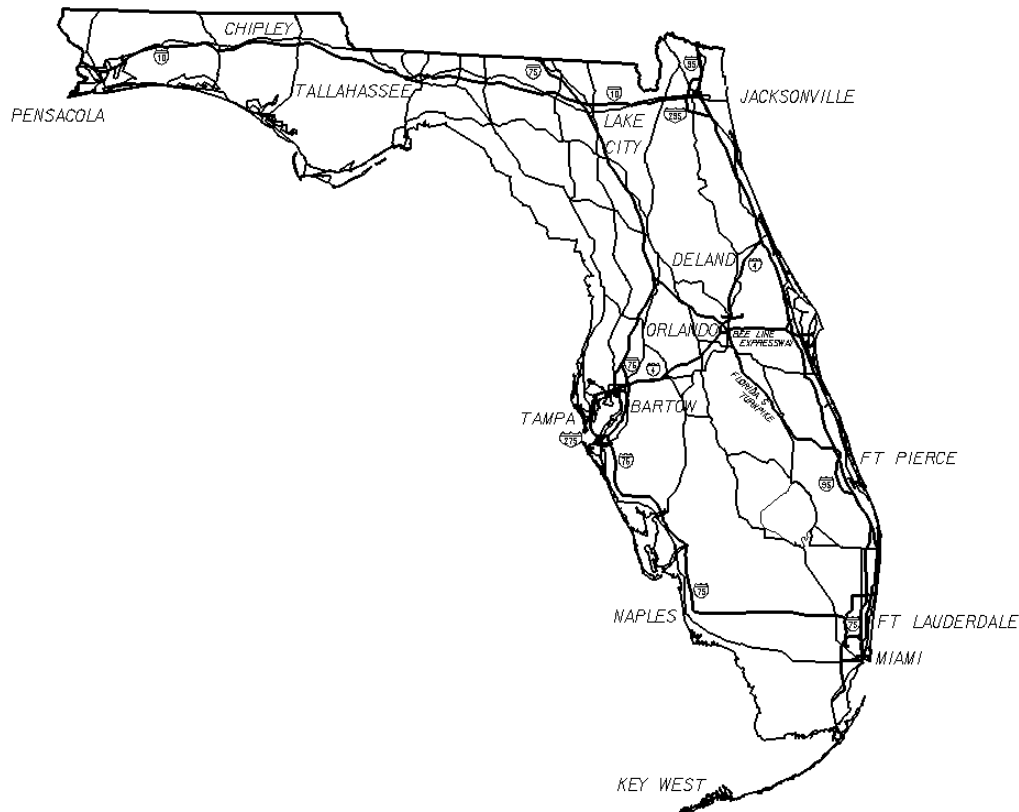
2. Set the MicroStation *Level* to **LeaderLine_dp**. Use the drafting filter to make this easier.

3. Place a line starting from a point near the label **Tallahassee** on the map. The leader line will be a two-piece line or smart line.

4. Draw the second part of the leader line horizontal. The length needs to be long enough to hold the text **"LOCATION OF PROJECT'**. The length can be adjusted after the text is placed.

5. From Task Navigator, select **FDOT Plans Development > Roadway Plans > Place Terminator Arrows** tool.

6. Set the *Terminator* to **ArrTermPrEnd** located in the *roadway.cel* library. This terminator is a personal preference, use whichever arrow meets the projects needs.
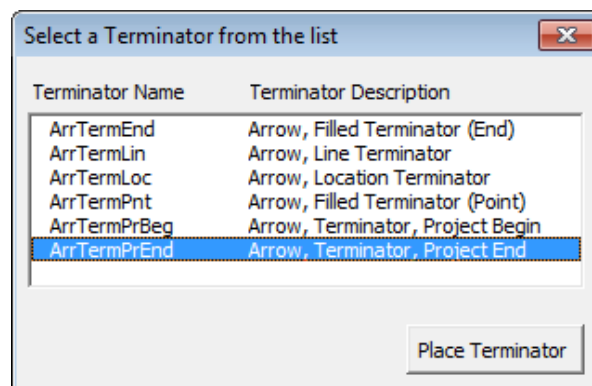
7. Select the *leader line* and accept the line to place the arrow. Select near the end of the line.

8. Set the MicroStation *Level* to **TextLabel**.

9. From the Task Navigator, select the **Main Classic** > **Place Text**.

10. Set the *Text Style* to **General Text Label**.

11. Check on the *Height* and *Width* and set them to **739.20**. (**0.07 x 10560**).

12. In the Text Editor, key-in **LOCATION OF PROJECT**.

13. Place the text above the leader line.

14. Adjust the length of the leader line if necessary.

15. The figure below shows the label and leader line pointing at the project location.
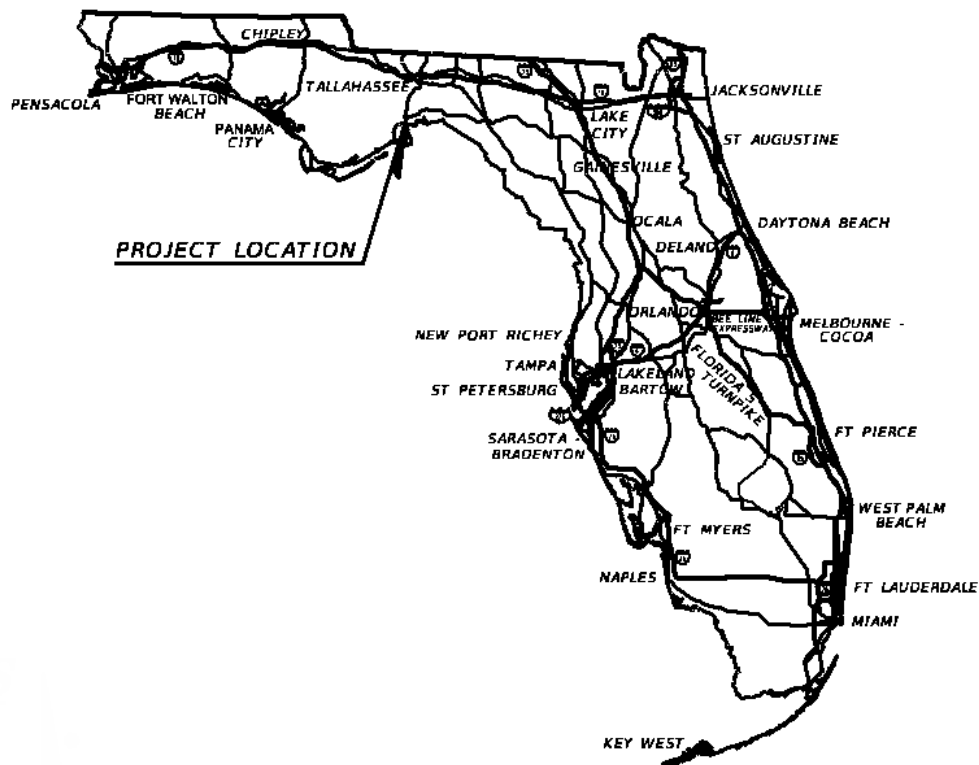
➢ *Place County Number Next to County Name (Part 6)*

In this exercise, the student will use the *Fill in Single Enter_Data Field* tool to place the county number next to the county name. The county number is made up of five numbers, the first two numbers are the county and the next three numbers represent the section of the road being worked on. This county number can be found on the straight line diagrams.

1. Continuing in *Keyslt01.dgn*, zoom to the top center of the key sheet next to the text "WAKULLA COUNTY".

STATE OF FLORIDA
DEPARTMENT OF TRANSPORTATION

CONTRACT PLANS

FINANCIAL PROJECT ID 220495-5-52-01
(FEDERAL FUNDS)
WAKULLA COUNTY (   )
STATE ROAD NO. 61

2. From the Task Navigator, select **Main Classic** > **Fill in Single Enter_Data Field** tool.

3. Issue a **Data Point** in between the parenthesis. A box will appear inside of the parenthesis; this indicates you got the data field.

WAKULLA COUNTY (   )

4. In the Text dialog type in the *County Number* **59010**.

5. Issue a **Data Point** in the view. This will fill in the data field with the county number.

6. Right mouse click to **reset/cancel** the command.

WAKULLA COUNTY (59010)

3 SUMMURY OF PAY ITEMS

OBJECTIVE

The objective of this chapter is to teach the designer how to create the Summary of Pay Items sheet.

INTRODUCTION

The summary of pay items sheet is generated from data exported from TRNS*PORT PES. In TRNS*PORT, there are two different outputs produced for pay item summaries, the Project Summary of Pay Items and the Proposal Summary of Pay Items. Use the appropriate report, based on the project's phase.

For early phase reviews (up to Phase III, or until the proposal has been created), the designer must use the Project Summary of Pay Items Report (the proposal report is not available during this phase of a project). If a designer anticipates the simultaneous release of multiple projects, he/she should print each project's Summary of Pay Items for review. These reports are printed on standard 8.5" by 11" paper. A Project's Summary of Pay Items sheet does not have to be in CADD sheet format for phase review submittals.

For later phase reviews (Phase III or after the proposal has been created), the designer uses the Proposal Summary of Pay Items Report. After the designer submits the report from the designer interface menu, the output is sent to the CADD FTP site, ftp.dot.state.fl.us/outgoing/ces/, normally within 5-10 minutes. The output is transferred to a MicroStation graphics design file and placed on a standard formatted plan sheet via a program available in the Department's CADD software. The quantities listed in the plans must be kept current with the quantities in TRNS*PORT. Any revisions to the quantities in TRNS*PORT must be transferred and updated in the graphics design file. The TRNS*PORT quantities are used to prepare the bid documents therefore the quantities listed in the plans must match.

When the Lighting Plans are a component of the Roadway plans, the Summary of Pay Items Sheet is part of the Roadway plan set and is not included as part of the Lighting Plans.

The CADD version of the Summary of Pay Items Sheet is created by a program available on the FDOT Menu Bar > Actions Submenu > Trns*Port Quantities. This tool transfers the PES Output file, imports it into a design file, and places it on a plan sheet.

GENERAL INFORMATION

The Trns*Port Quantities tool gives the designer the option to place a sheet border into the design file or to only import the PES text file. Do not override the border and settings when updating quantities. The figure below shows how to access the tool.
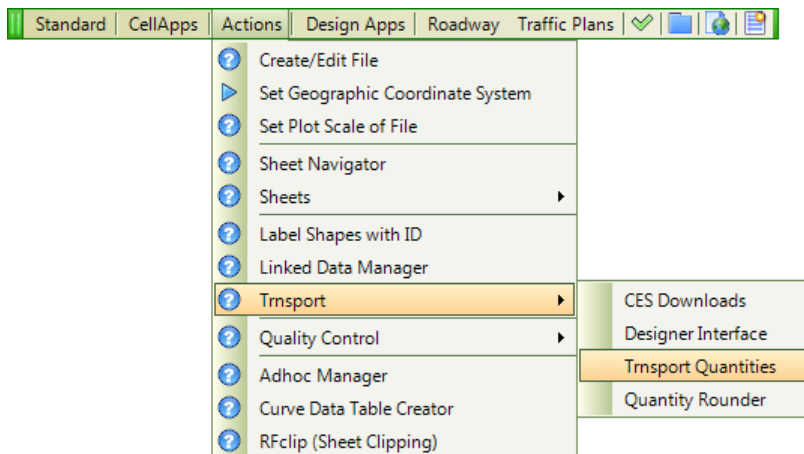
The figure below shows the Trns*Port Quantities tool loaded from FDOT Menu. The dialog is broken into two parts; the top portion of the dialog defines the sheet information and search paths. The bottom part of the dialog defines the label for the title block on the sheet border.
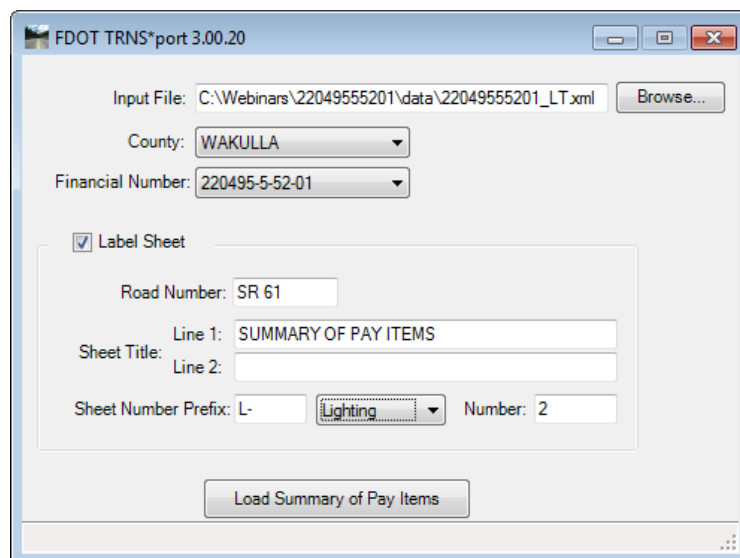
- **Input File Name** - The name of the PES data input file, for example: *p723423.dat.* It doesn't matter where the PES data file is located as long as it is accessible by the file list box. This file may also be an XML file, for example: *22049555201.XML.* Click Browse to navigate to the location of the file in your project directory.

- **Label Sheet -** If these fields are filled in, the program places the *sheet number,* the *title,* and the *project number* in the appropriate fields on the sheet cell when it is placed.

  o *Label Sheet Toggle Box* – will disable the Label Sheet portion of the FDOT TRNS*PORT tool. The labeling then can be completed by the Sheet Navigator application.

Exercise 3.1 *Transferring PES Data to Plan Sheet*

> ➢ **Transferring PES Data to Plan Sheet – FDOT TRNS*port Setup (Part 1)**

In this exercise, the instructor will cover the steps that would normally take place after the designer submits the report from the designer interface menu. These steps will take the designer through accessing the output data and setup for the use of the automated process called FDOT TRNS*port.

1. The designer would access the *Designer Interface* from the FDOT Menu option: **Actions > Trnsport > Designer Interface.**
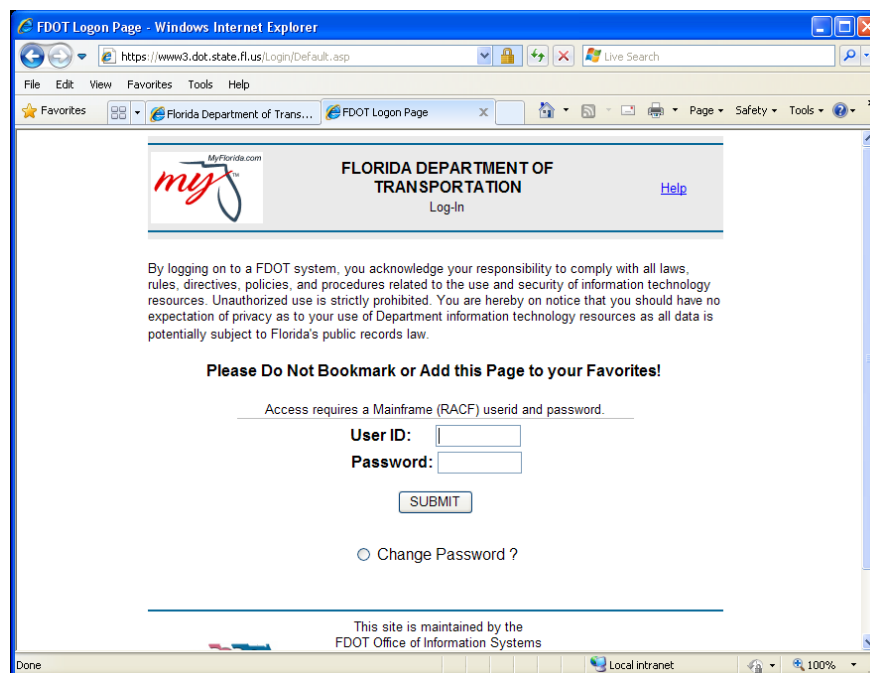
2. The designer would *Login* by typing their *User ID* and *Password* and clicking the **Submit** button.

3. The Trnsport WebGate page displays.
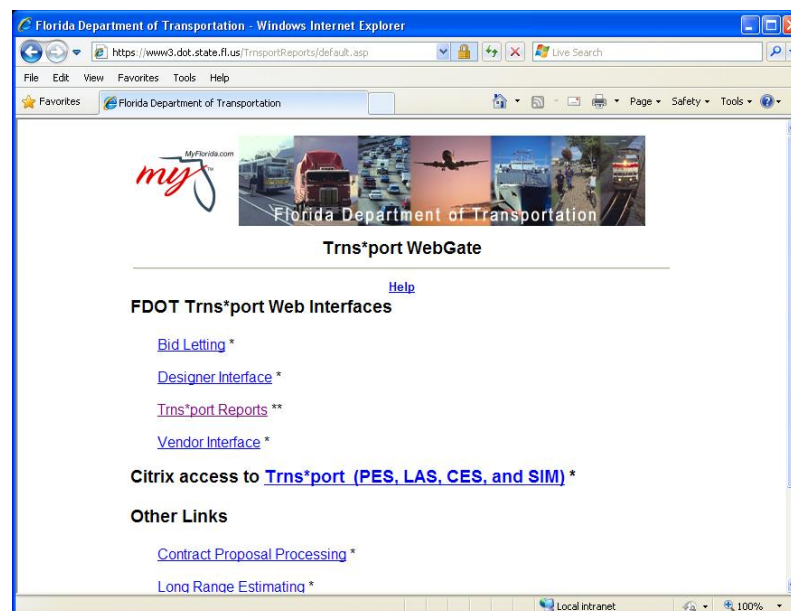
4. Click on the **Trns*port Reports** link to take the designer to the Trns*port Report Menu.
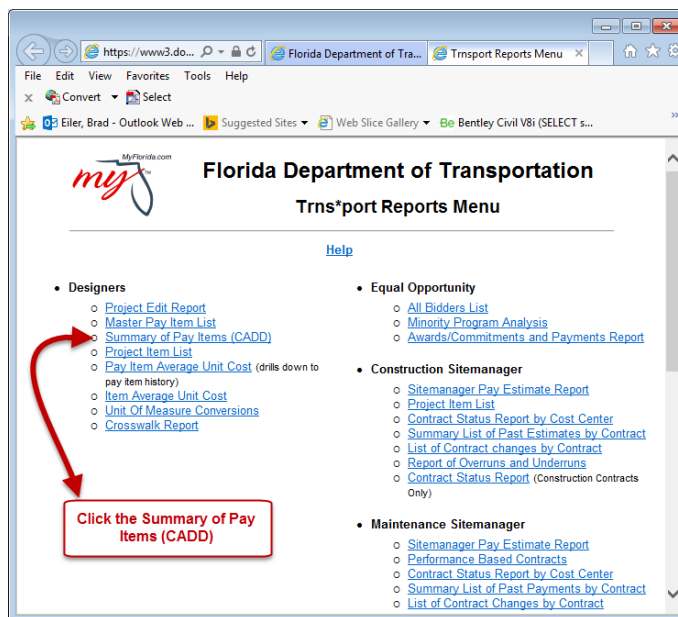
5. Click the **Summary of Pay Items (CADD)** link to take the designer to the Summary of Pay Items (CADD XML) page.

6. The designer will select the radio button for either **Proposal** or **Project** from the *Select Option Type* section. For this project, the **Project** radio was selected.

7. The designer will type in the project **Financial (Project ID)** number in the *Enter Project ID* section field. The *Project ID* is **22049555201**.

8. The designer will select the radio button for either **XML File** or **Report** from the *Select Output Type* section. For this project, the **XML File** radio button was selected.

9. The designer will then click the **Submit** button to submit the request for the output data.

> ➢ *Transferring PES Data to Plan Sheet – Import Trns*port XML Data (Part 2)*

In this exercise, the designer will create a new design file and import the Trns*port XML data at one time. The Department delivers a tool to automate this process called FDOT TRNS*port. This tool can be started from any open MicroStation file.

1. From the MicroStation Menu, select **File > Open** and select the **Cesslt01.dgn** file. This file was created earlier using the Create Edit application.

2. From FDOT Menu, select **Actions > Trnsport > Trnsport Quantities**. This opens the FDOT TRNS*port tool.
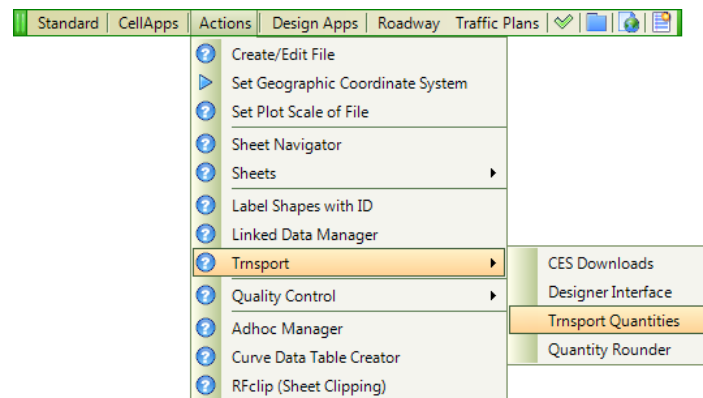
3. For the *Input File*, select **Browse**.
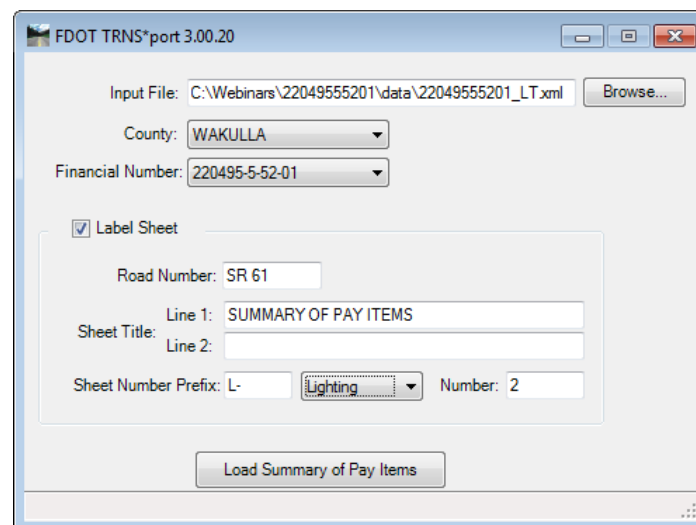
4. Browse to the *data* folder in the project and select the **22049555201_LT.xml** file.

5. Click **Open**. This fills in the *Input File name, County Name* and the *Financial Number* in the FDOT TRNS*port dialog.

6. Toggle **On** the *Label Sheet* check box.

7. Fill in the *Road Number* with **61**.

8. For the *Sheet Title*, leave the **default** text.

9. Set the *Discipline* to **Lighting**. This will set the *Sheet Number Prefix* to **L-**.

10. Set the *Number* to **2**. This will number the sheet **S-2**.

11. Click **Load Summary of Pay Items**. This will load the CES XML file and will place a border sheet.

12. Close the FDOT TRNS*port dialog.

Exercise 3.2 *Update Sheet with Sheet Navigator*

  ➢ *Add Digital Signature Note and Update Labels*

   1. Continuing in *Cesslt01.dgn,* from the FDOT Menu, select **Actions > Sheet Navigator** to launch Sheet Navigator.

> *Note*   Notice the fields that are populated. If the field for the Financial Project ID 1 is blank, click inside the blank field. This will populate the Project ID with the correct number. (This field should already be filled out from the FDOT TRNS*port tool.)

   2. Set the *Digital Signature Note* to **Standard**.

   3. Click the **Save Sheet** button. This updates the sheet border.

  ➢ *Updating Component Index and LDM Synchronization (Part 1)*

   1. In Sheet Navigator, select the **Navigator** tab.

   2. Select **File > New**. This opens the Build Index dialog.

   3. Select the **Component** option.

   4. Select the **Process new or modified files only** option.

   5. Click the **Build Index** button. This creates sheetinfo.xml in the Lighting folder. All other discipline folders are ignored.

6. MicroStation will close and the CM.Batch process will run. When the CM.Batch is complete, MicroStation will be open and Sheet Navigator will display the new **Component Index** on the *Navigator* tab.
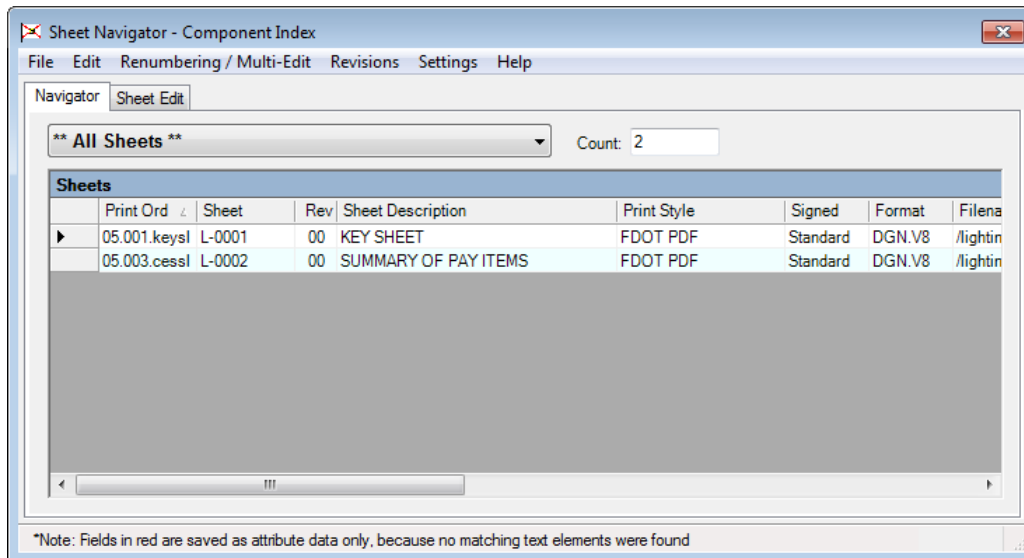
7. Select **File > Generate Component Index Files**. This opens the Generate Component Index Files dialog.
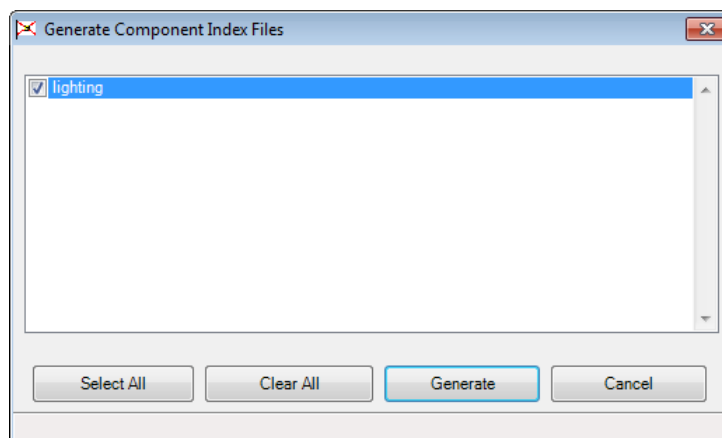
8. Toggle **On** the check box for *lighting*.

9. Click the **Generate** button. Sheet Navigator will create 1 component index file.
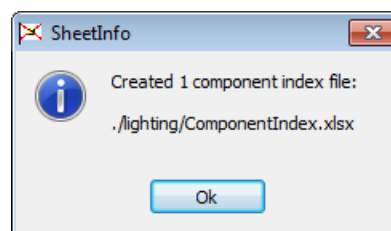
10. Click the **Ok** button.

11. Click the **X** in the upper right hand corner to close Sheet Navigator.

➤ *Updating Component Index and LDM Synchronization – Review (Part 2)*

1. Open *Keyslt01.dgn*, zoom in around the text "INDEX OF LIGHTING PLANS". This is on the left hand side of the sheet.

1. Notice the "INDEX OF LIGHTING PLANS" has been updated by the **LDM** link setup in Chapter 2.

2. Take a moment to review the Key Sheet.

4 LIGHTING TOOLS

OBJECTIVE

The objective of this chapter is to teach the designer how to use D&C Manager, the FDOT Menu and other GEOPAK tools to create proposed lighting features.

INTRODUCTION

The Lighting Tools section introduces several applications that help the designer create Lighting plans.

As explained in Chapter 1 of this course material, the designer will create a new file, *dsgnlt01.dgn*, in which the Lighting proposed design will be drawn. In addition, the designer must reference the proposed Roadway design file, *dsgnrd01.dgn*, existing Topographic file, *topord01.dgn*, existing Utilities, *utexrd01.dgn*, existing Drainage, *drexrd01.dgn*, existing and proposed Right-of-Way files, and any additional files containing existing features along the project.

After the above is done, the designer is ready to clip the sheets. To aid in this task, the designer has two options available, the FDOT Menu Clipping program, commonly known as *rfclip*, and the GEOPAK's Sheet Clipping program.

The next step is to draw/place the proposed Lighting features in accordance with Department's CADD Standards. The FDOT Menu provides tools with the active settings (Level and Symbology) used to create the Lighting features. There are additional tools available to help in the efficient placement of Light poles and Pull Boxes: D&C Manager and the GEOPAK Draw Cell Group by Feature.

Refer to Chapter 25 in the ***Plans Preparation Manual*, Volume II**, for more detail on developing the Lighting plans. Refer to the Design Standards indexes for design criteria.

The Labeling of the plan sheet items has been made easier with the use of GEOPAK's Plan Labeler, which allows the designer to create styles for the placement of repetitive labels.

To assure that the designer is adhering to the ***Traffic Plans CADD Standards***, the FDOT Menu provides an easy way to check and fix symbology to match the CADD standards.

Finally, the designer must compute the pay items used and populate the Tabulation of Quantities Sheet. GEOPAK's D&C Manager can count and report every "each" and "linear" item. D&C Manager generates a report that can be finalized in Microsoft Excel and then imported into the MicroStation Tabulation of Quantities design file using LDM (Linked Data Manager).

CLIPPING LIGHTING PLAN SHEETS

Clipping Lighting Plan Sheets is very similar to clipping Roadway or Lighting Sheets. The designer will use GEOPAK to layout and clip the sheets. On simple projects it may be possible to double stack the plans one below the other. A scale that presents a clear representation of the project should be used. However, the scale should not be smaller than 1" = 100'.

If a Motif file is used, this should be set up prior to starting the clipping process. A Motif file is basically a template MicroStation file with all of the reference files attached that the designer wants attached to each plan sheet. There is an option to use the active file when clipping.

SHEET LAYOUT AND CLIPPING WITH GEOPAK

In this section, the student will learn how to layout clip sheet borders and then clip sheets using GEOPAK. The Department delivers a number of standard sheet clip standards to make the clipping process much easier. As with any process, it is important to communicate with the Department's Project Manager to make sure that all district specific requirements are addressed i.e. Match Lines or no Match Lines, Grey Scaling existing features and so on.

Sheet clipping is, simply, a method to get the information that is in the design file onto a plan sheet. There are several ways to accomplish this with GEOPAK being the most automated. There are advantages to using GEOPAK to clip sheets such as:

- **Consistency -** all of the sheets will look the same with the same reference files and level structure of those reference files will be consistent.

- **Re-clipping -** if the alignment were to change and the sheets needed to be re-clipped it is very easy to do this with GEOPAK.

TO START THE SHEET LAYOUT TOOL

From the MicroStation menu, select **GEOPAK > Road > Plans Preparation > Plan/Profile Sheet Composition** or from the Road tools palette click the **Plan/Profile Sheet Composition** icon.
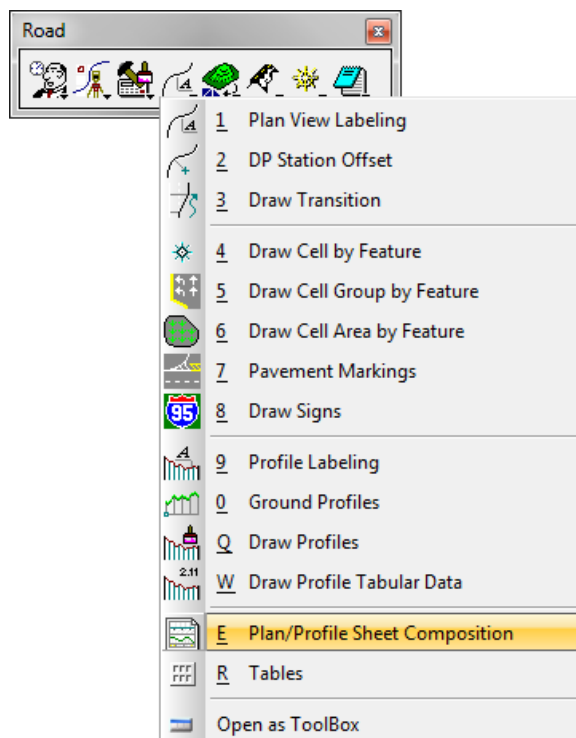
Once activated the Plan Sheet Layout dialog opens as shown below.
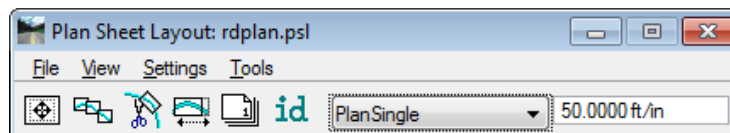
The Plan Sheet Layout tool uses an attached library that controls what sheet types are available. The Plan Sheet Library or .*psl* file is located in the FDOTSS3\geopak\dat_files server folder.
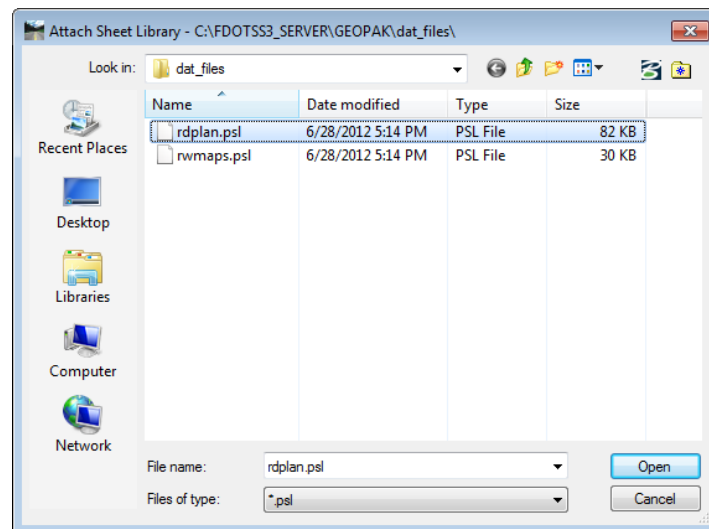
The Department delivers a group of these files, ready to use, that cover the most common of the sheet configurations used. By default, the *rdplan.psl* library is attached. This is shown in the header of the dialog to see which library is currently attached.

SHEET LIBRARY

The sheet library is accessed from the Plan Sheet Layout menu File > Sheet Library. Selecting Edit opens the Sheet Layout: Sheet Library dialog as shown below.
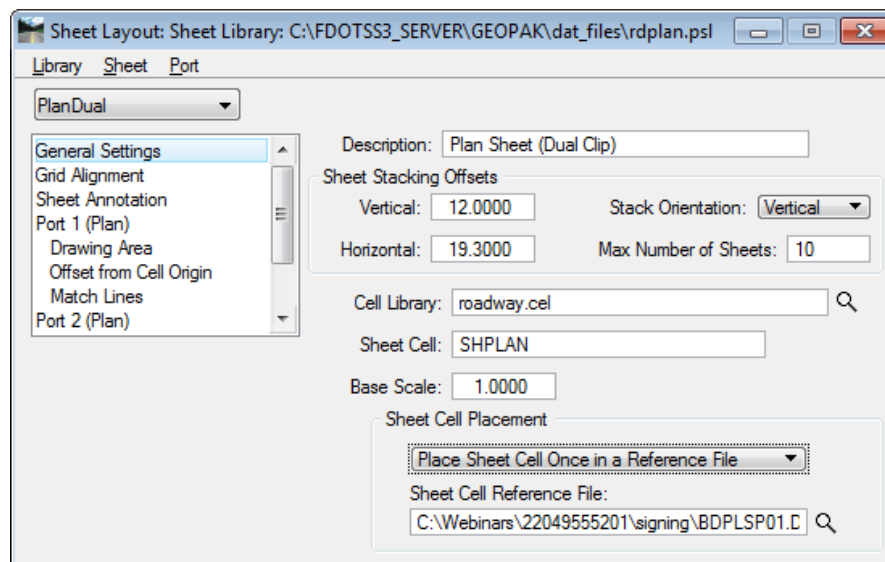
The most important parameters in Sheet Layout: Sheet Library dialog are the following:

- **Cell Library** – Make sure the correct cell library is attached.

- **Sheet Cell** – This is the name of the cell representing the border.

- **Sheet Cell Placement** – Select how the Sheet Cell, Border, is to be placed. Either Place the sheet cell in each file or place it once in a border reference file.

<u>SHEET LAYOUT SETTINGS</u>

The Sheet Layout Settings dialog can be accessed from the settings pull down on the Plan Sheet Layout dialog. This dialog instructs the sheet clipping to cut sheets a certain way. For example: is the clipping to go from Left to Right, is there a lot of elevation relief on the project, if so than set the stair stepping ON and how is the sheet view attributes set up, use the current design file or a Motif file.
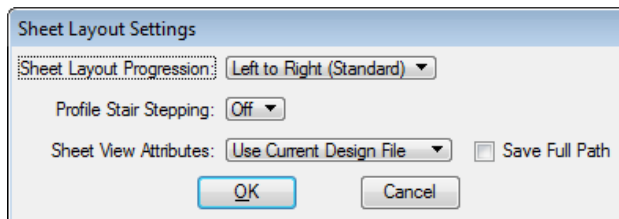
A Motif file is nothing more than a MicroStation design file with reference files attached and level symbology set up the way the plan sheets are to look when clipped. The Department has established a standard design file name *MTPLLT01.dgn*. Once the Motif file is created, attach the appropriate reference files and turn on or off the levels to make the file look the way the plan sheets are to look for the project. This is, also, where the designer can set up any level symbology that is required.

<u>SHEET COMPOSITION</u>

Sheet Composition controls how the limits of the sheets are handled. The options are:

- By Begin Station/Overlap
- By Station Range: Inside Out
- By Station Range: Outside In
- By Station Range: Radial
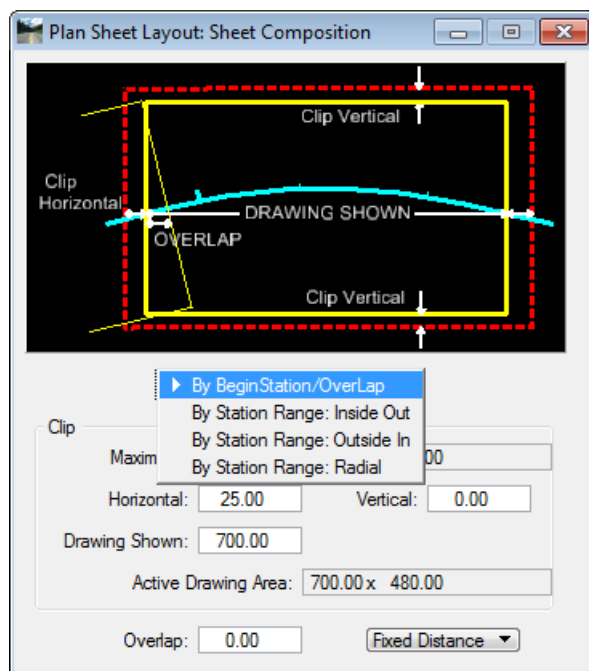
The *Station Range* is a factor of the Clipping Scale. Look in the **PPM, Volume 2, chapter 10**, for the values to be used in the *Station Range*. These values are filled in based on the scale entered, but may need to be adjusted to fix areas in sharp curves where portions of the design files are missed.

These four options are covered in great detail in the **Basic GEOPAK for Roadway Designers** training guide. In this training course, the class will use the option *By Station Range Radial*.

By Station Range Radial – With this option the **Horizontal** distance is used to trim the Station Range. The Maximum Drawing Area, Horizontal and Station Range are inter-related. The resultant clipping shapes are not rectangular; instead, the Vertical edges are oriented perpendicular to the alignment, which results in rectangular shapes along tangent sections and more of a trapezoidal shape along curved sections. Adjusting the Station Range will alleviate the pie shaped wedges along curves.
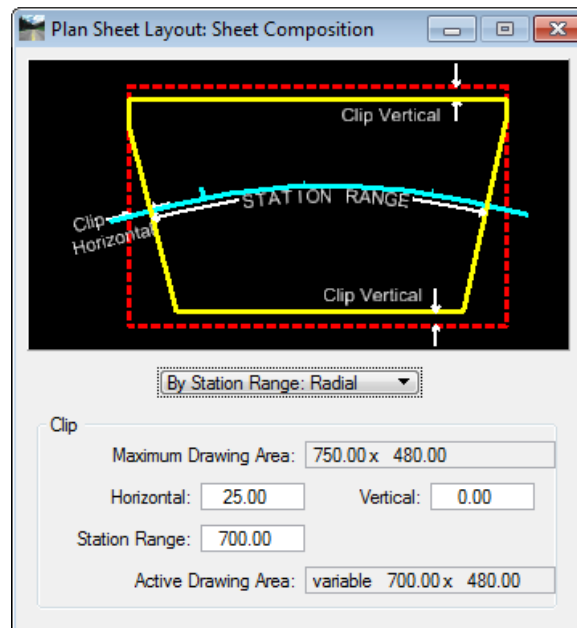
The critical value in this option is the *Station Range*. This should be set based on the standards delivered in the PPM. For example, for 50 scale sheets the Station Range should be set to 700. This is the maximum value and may be decreased to handle alignments with unique conditions like sharp curves or intersections.

Horizontal is used for the clipping limits at the ends of the sheet. If you adjust the *Horizontal* distance the *Station Range* will dynamically change by the Horizontal distance times 2.

Vertical sets an additional clip limit from the defined sheet width. In the figure above, the red dashed rectangle represents the *Maximum Drawing Area*. Entering a value other than zero will reduce the clip limit by this amount.

The sheet layout process is a multiple step process because of the items mentioned earlier, curves and intersections. There are three ways to approach this:

1. Layout all clip sheets from begin project to end project then adjust the sheets at the intersections and around curves to clean up the pie shaped wedges.

2. Layout single sheets at all of the intersections then run the remaining clip sheets up to the intersections and after the intersections. It is good practice to try to center intersections on the plan sheets if possible.

3. Get the clipped sheets from the Roadway group and rename them to signing and pavement marking sheets. As long as the scales are the same this option works fine.

LAYOUT SETTINGS

The Sheet Layout dialog is used to define the Ports and alignments and station range to run the layout along.
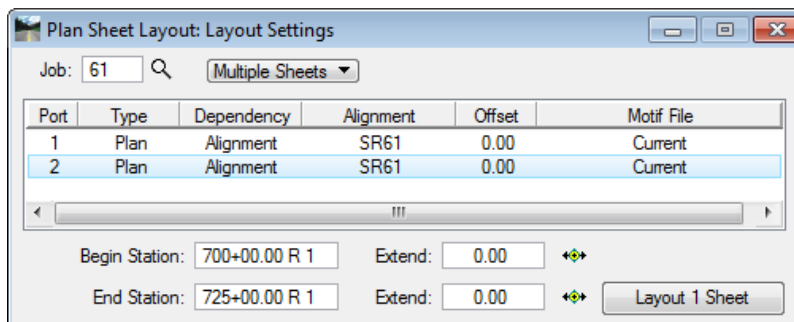
- **Job** – This is the *gpk*. If Project Manager is used, this will be filled in automatically. If Project Manager is not used the job will have to be selected using the magnifying glass icon.

- **Multiple Sheets** – Used for clipping more than one sheet.

- **Single Sheet** – Used for clipping only one sheet.

- **Port** – In a Plan/Profile scenario Port 1 is the Plan view and Port 2 is the Profile. For Plans that are Double Stacked with two plan views, both Port 1 and 2 are Plan.

- **Type** – This is either the Plan or Profile area. In a scenario like Signing and Pavement Markings there will only be a Plan port.

- **Dependency** – This instructs the Port to either be dependent on an Alignment or another port.

- **Alignment** – This is a GEOPAK chain that the sheets are clipped along.

- **Offset** – This allows the designer to set the clip borders at an offset from the Alignment.

- **Motif File** – This shows whether a motif file is used and if so what the name is.

Double Clicking on one of the Plan ports in the *Layout Settings* tool opens the Plan Sheet Layout: Plan Port Data dialog as seen below. This is where modifications are made to set up the Layout Settings dialog.
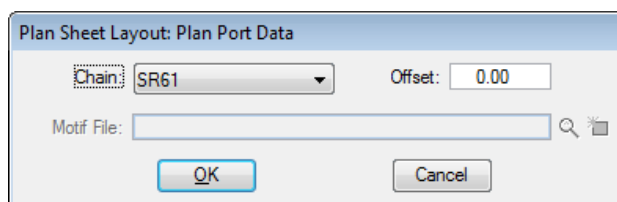
- **Begin and End Station** – This defines where the first sheet starts and the last sheet ends. Either key-in the values or use the Select icons to dynamically select the stations. It is highly recommended to select an even station or snap to an even tick mark for the begin station so that the sheets will all fall on even stations.

- **Extend** – To start the first sheet at a defined distance prior to the Begin Station. This is good for leaving space for General Notes or additional design before the Begin Station. The same applies for the End Station.

- **Layout Sheets** – This button will show the number of sheets needed based on the Begin and End Stations. Clicking this button will draw the clip borders into the active design file.

SHEET NUMBER MANAGER

Sheet Number Manager provides a method for applying sheet and sequence numbers to the clip borders. The main window contains a list of the sheet borders contained in the file. In the case of dual clip borders for a single sheet, i.e. Plan/Profile sheets, the borders are grouped together and only listed once in this window. Select a line in the window, click the up and down arrow buttons to move the Sheet in the sequence order. The next two buttons allow for the manual editing of sequence and sheet numbers. The *ID* button is used to graphically select a sheet to modify. Select the *Highlight Clipping Shape* check box or *Window Center Clipping Shape* if applicable.

Keep in mind that this is the MicroStation design file name for the sheet not the actual sheet number that is placed in the title block. Knowing this, you should always run Sheet Number Manager and add a Prefix of '0' to the sheets so when the sheets are clipped the file names will be, as an example: *Planlt01.dgn* not *Planlt1.dgn*. This will make organizing and managing the files in explorer much easier.
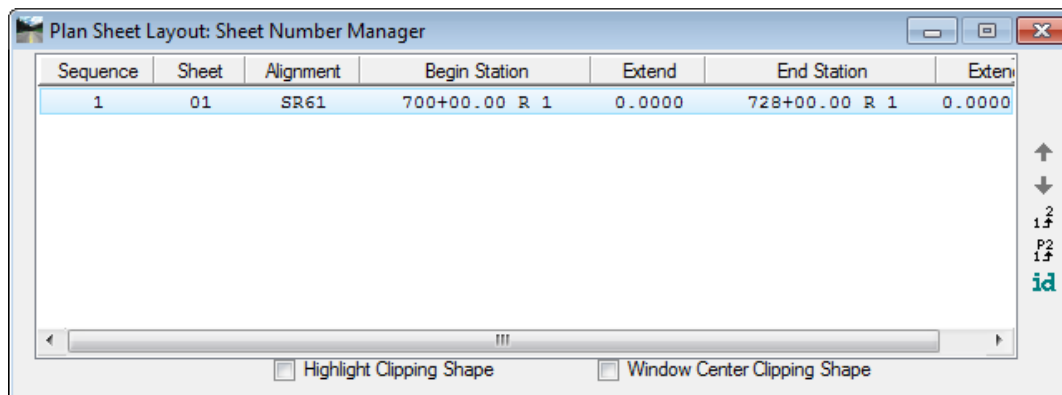
MODIFY SHEETS

This allows the clip sheet borders to be adjusted prior to actually clipping the sheets. The Sheet Modify command opens Plan Sheet Layout: Modify, which operates in two modes, *Slide Sheets or Modify Drawing Area*, represented by the two buttons in the upper left corner. The following figure shows the *Slide Sheets* mode.

Slide Sheets is used to move the clip borders along the alignment. Type a specific station in the box and select the *Left Station, Center Station,* or *Right Station* to apply the new station. *Left Station, Center Station, or Right Station* refers to the portion of the clip border that is located at that station.

Another option is to change the *Sheet Chord Offset* of the border to allow the movement of the sheet parallel to the alignment in either direction for a given distance. *Additional Rotation* can also be applied. This angle is measured from the left station and is in reference to the alignment.

Perform any of these three actions dynamically with the *Dynamic* buttons to the right of the respective function.
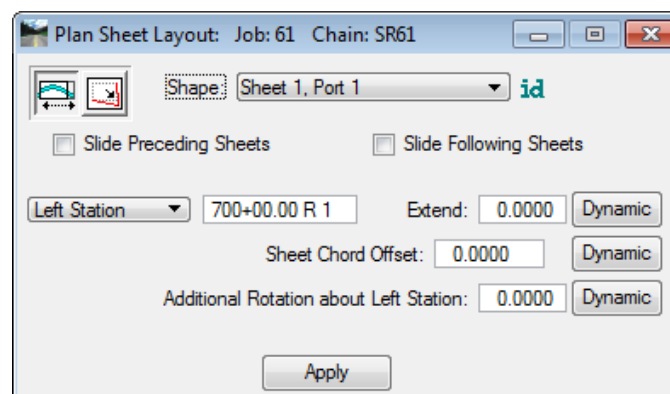
Modify Drawing Area changes the composition of the borders. Type the appropriate dimensions in the *Drawing Shown, Horizontal,* and *Vertical* boxes, and then select *Left Station, Center Station,* or *Right Station* from the *Hold To* list.
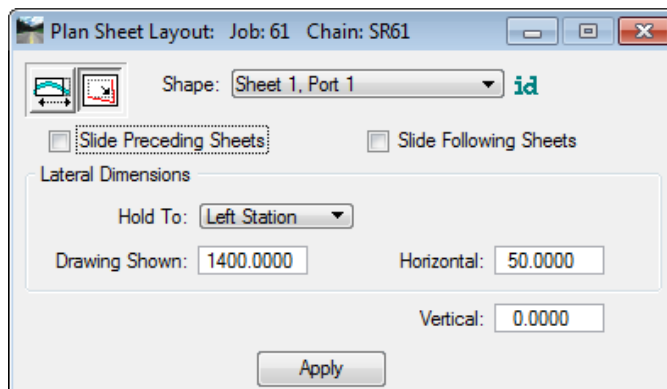
In either mode, select which sheet is being modified from the *Shape* list, or click the *Identify* button to do so graphically. Select the *Slide Preceding Sheets* check box and/or the *Slide Following Sheets* check box to indicate if the sheets preceding and following the modified sheet are to slide to compensate for the changes made to the current sheet border. To make any changes effective, click the *Apply* button.

CLIP SHEETS

This actually clips the plan sheets based on all of the previous settings.
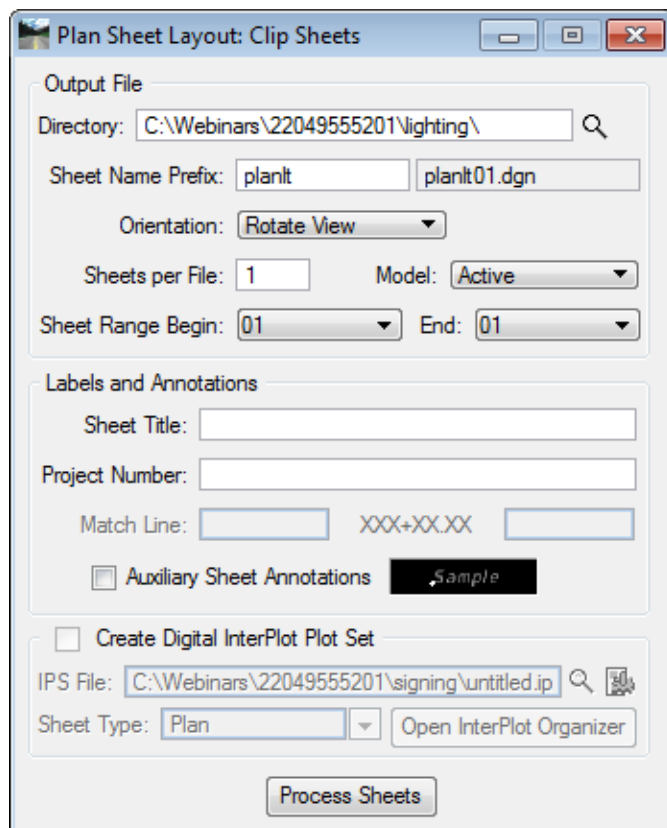
- **Output File** - allows the designer to specify where sheet files are created and file name.

- **Directory** – Select the path where the new sheet files are to be placed.

- **Sheet Name Prefix** – Type the name of the sheet file. Type only the first part of the file name; the software supplies a numerical suffix that corresponds with the sheet number. For example, if PLANSP is entered as the prefix, as the sheets are clipped the files are named PLANSP01.DGN, PLANSP02.DGN, PLANSP03.DGN, etc.

- **Rotate Reference** or **Rotate View** – Rotate Reference is used when stacking more than one sheet per file and Rotate View is used when only one sheet per file is used.

- **Sheets per File** – Type the number of sheets to be created in each design file.

- **Sheet Range Begin** and **End -** Select the range of sheets to be created from the clip borders.

- **Labels and Annotations** allows the designer to enter a **Sheet Title** and **Project Number**, as well as **Match Line** text when applicable. This information cannot be entered if the annotation information is not set up in the Sheet Library. Typically, this section is not used to label the sheets.

Note   If this section is grayed out that is because the Sheet Annotation settings in the attached library are clear as seen in the figure below.
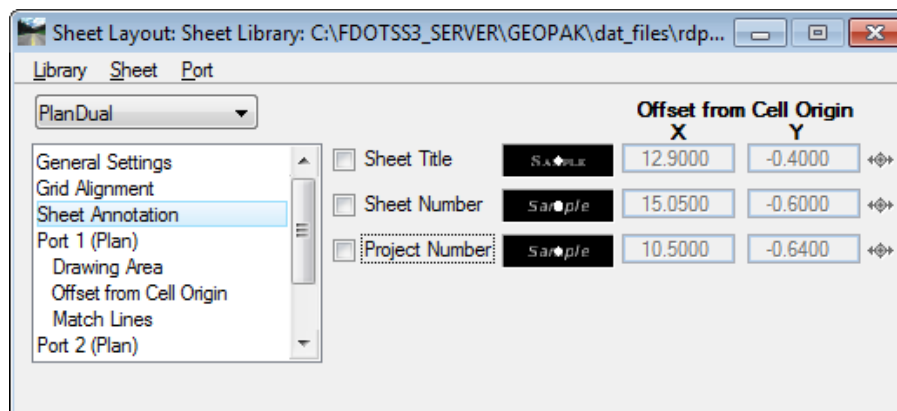
- **Process Sheets** – This button when pressed starts the sheet clipping process.

<u>SHEET LIBRARY ITEMS</u>

There are some settings in the sheet library that may need to be adjusted to make the plan sheets look correct. For example, if after the sheets are clipped and the offset for the left edge of the border to the clip limit is too small this can be adjusted in the library and the sheets re-cut.

The figure below shows the Offset from Cell Origin as 0.200. This is the default as delivered from FDOT. When the sheets are clipped, the space from the left edge of the border to where the clip sheet starts is too small.
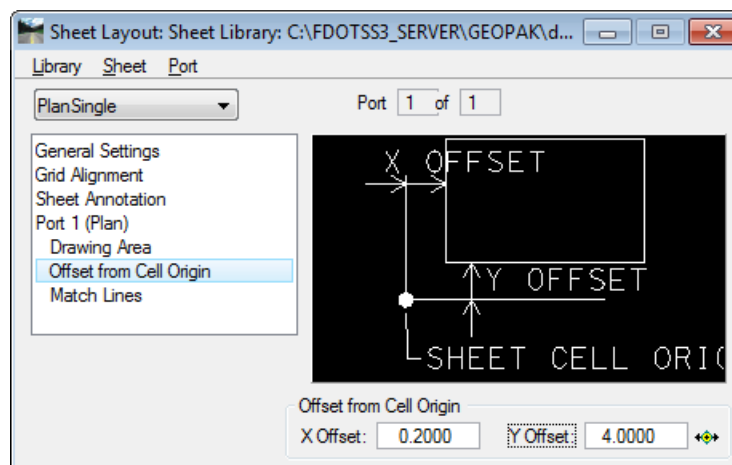
The image below shows the panel with the Offset from cell origin set at 0.200. Changing this value to 0.800 will shift the clip area over to a much better looking condition.
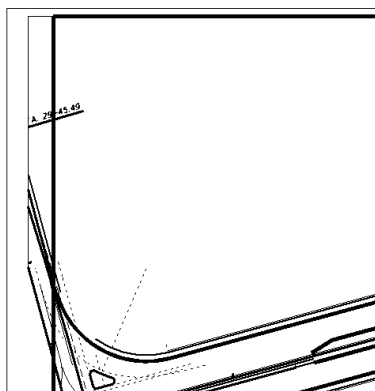
The figure below shows the results after changing the Offset from Cell Origin.
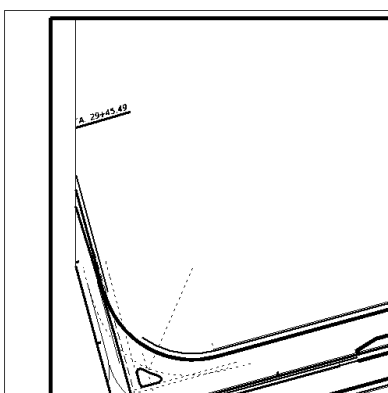
Exercise 4.1 *Clip Sheets*

In this exercise, the student will create the Clip Sheet file and Plan Sheet Border.

➢ *Set up Sheet Clip Design File*

1. From the FDOT Menu, select **Actions > Create/Edit File** to load the Create File/Project application.

2. Using what you have learned in this course guide create the *Clip Borders* file. Create the file in the *lighting* folder using the *Lighting Files Group*. Refer to chapter 1 in this training guide for assistance if needed.

➢ *Open Border Sheet Design File*

3. Open the *Border Sheet*, *Bdpllt01.dgn* created in Chapter 1.

4. Set the *Plot Scale* to **1.00**. No need to place a sheet border at this time, the sheet clipping process will place the border sheet cell.

5. Open the new file *Cliplt01.dgn*. Plot Scale in this file is not critical because it is only for laying out clip borders, no text or line work goes in this file; however, you may set it.

6. Reference in the **Algnrd01.dgn, Dsgnsp01.dgn, Dsgnsg01.dgn, Dsgnrd01.dgn (Default model)** with *Live Nesting* set to **depth of 1,** from the *roadway* folder. This is so you can see the project limits as the sheet are being laid out.

7. Zoom to a **Fit View**.

> ➢ *Start Plan Sheet Layout Tool*

1. Continuing in *Cliplt01.dgn*, select **Plan/Profile Sheet Composition** from the *GEOPAK Road tools palette* or from the **GEOPAK > Road > Plans Preparation > Plan/Profile Sheet Composition** pull down.

2. Select the **SR61.prj** file located in the *roadway* folder. This opens Plan Sheet Layout Tool.

3. In Plan Sheet Layout, select the S*heet Type:* **PlanSingle**. This is the drop down menu on the right hand side of the dialog.

4. Set the *Scale* to **50.00**. This is next to the sheet type.
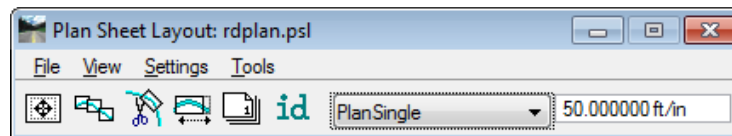
12. In *Plan Sheet Layout*, select **File > Sheet Library > Edit**. This opens Sheet Library.

13. In the *Sheet Cell Placement* portion of the dialog, select **Place Sheet Cell Once in a Reference File**.

14. Use the *magnifying glass* icon to browse to the border sheet created earlier.
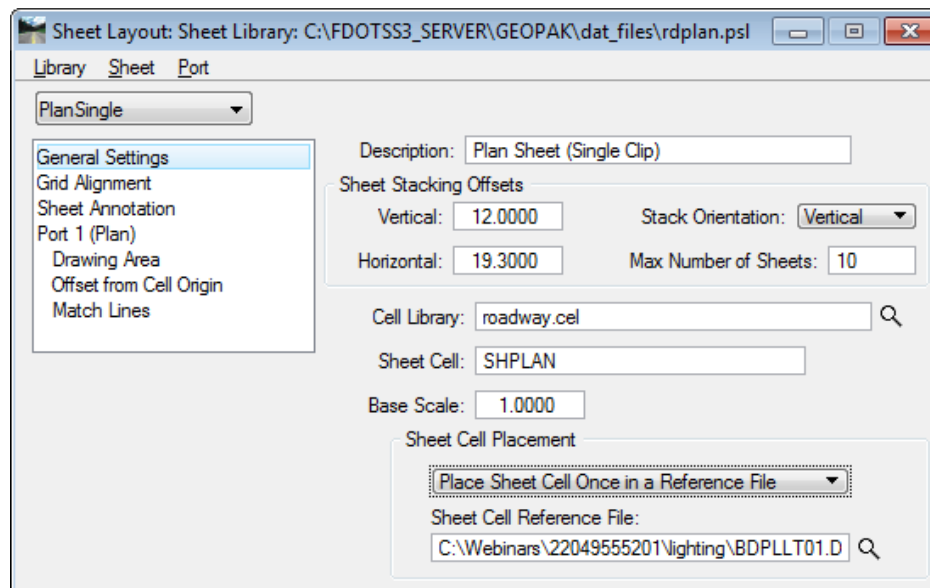
15. Close *Sheet Library* by clicking on the **X** in the upper right hand corner.

16. Click **Yes** to save changes made to the library.

17. In Plan Sheet Layout, select **Settings > Sheet Layout**. This opens Sheet Layout Settings.
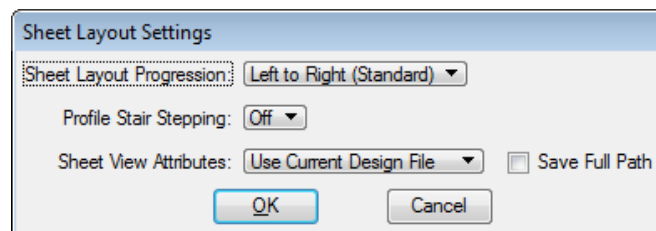
18. Set *Sheet Layout Progression* to **Left to Right (Standard)**.

19. Set *Profile Stair Stepping* to **Off**.

20. Set *Sheet View Attributes* to **Use Current Design File** and click **OK**.

➢ *Sheet Composition*

1. Open Sheet Composition [icon]. This can also be loaded from the **Tools** menu.
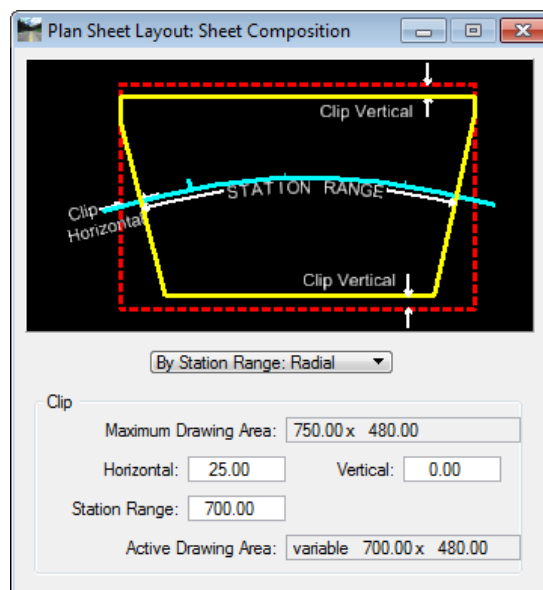
2. Set the *method* to **By Station Range: Radial**.

3. Set the *Station Range* to **700.00**. This will automatically set the Horizontal clipping to **25.00**. The maximum length allowed for a 50 scale sheet as per **PPM Vol. II Chapter 10**.

4. Close Sheet Composition.

➢ *Layout Sheets*

1. Click the **Layout Sheets** icon [icon] or select from the **Tools** menu.
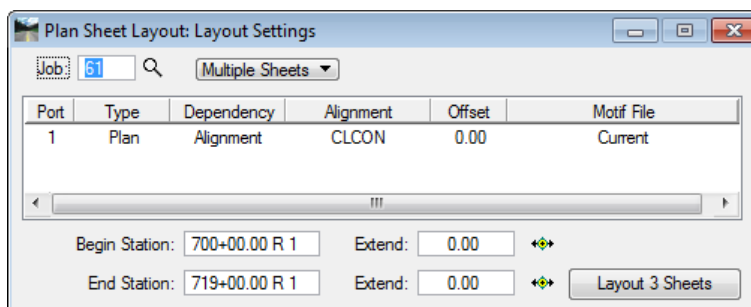
2. The *Job number* should be set to **61**. If not using Project Manager, browse and select the *gpk* file.

3. Set the *method* to **Multiple Sheets**. The options are **Single** or **Multiple**.

4. Double-click on **Port 1**. This opens Plan Port Data.
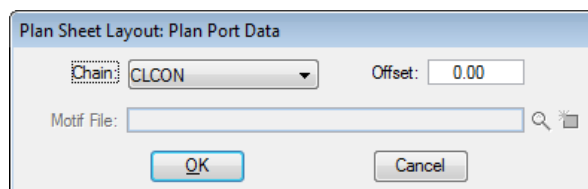
5. Set the *Chain* to **CLCON**.

6. Set the *Offset* to **0.00**.

7. Click **OK**.

8. Set the *Begin Station* to **700+00**.

9. Set the *End Station* to **719+00**.

10. Notice the sheet count is calculated in the Layout button.

11. Click the **Layout 3 Sheets** button. This will draw the clip shapes into the design file.

12. Close Layout Settings.

➢ *Sheet Number Manager*

1. Click the **Sheet Number Manager** icon. This opens Sheet Number Manager.
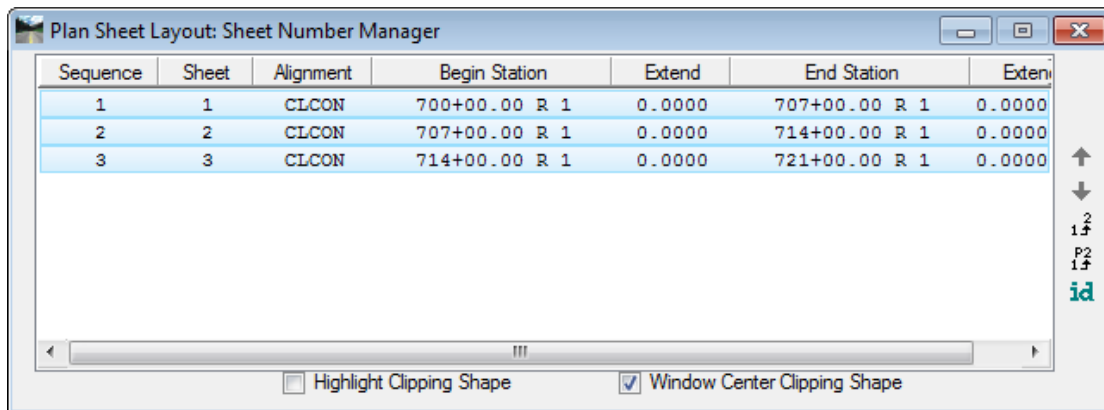
2. Select the sheet from the list in the Plan Sheet Layout: Sheet Number Manager.

3. On the right hand side of the dialog, click the **Edit Sheet Number** button. This opens Edit Sheet Number.
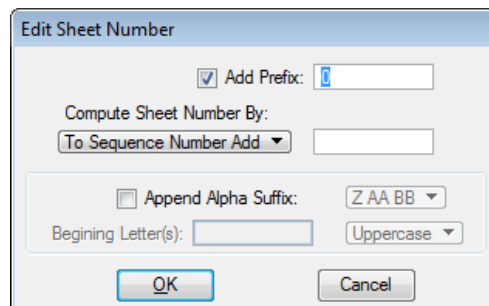
4. Toggle **On** Add Prefix.

5. Enter a **0** for the *Prefix*.

6. Click **OK**. This will change the *Sheet Number* to **01-03**.

7. At the bottom of *Sheet Number Manager*, toggle **On** *Window Center Clipping Shape*.

8. Select **Sheet 01**. This will zoom and center on Sheet 1.

9. Take a moment to familiarize yourself with this function.

10. Close Sheet Number Manager.

11. Click **Yes** to *Save Sheet Number Changes*.

> ➢ *Clip Sheets*

1. Click the **Clip Sheets** button ![icon]. This opens Clip Sheets.

2. For the *Directory*, use the *magnifying glass* to browse to the project **lighting** folder.

3. For the *Sheet Number Prefix* enter **Planlt**. The full file name will appear to the right of the field.

4. For *Orientation*, select **Rotate View**. The options are *Rotate View* or *Rotate Reference*. This is a personal preference.

5. Set the *Sheets per File* to **1**.

6. Set the *Model* to **Active**.

7. *Sheet Range Begin* set to **01**.

8. *End* set to **03**.

9. The sheet range settings allow the user to select a range of sheets to clip. This is useful as a check to make sure all of the settings are the way you want them before clipping a mass number of sheets.

10. Labels and Annotations, this will be grayed out if the options are turned off in the Sheet Library. If they are active, it is not necessary to fill them in as the Sheet Navigator is used to populate the title block.

11. The next figure shows the *Sheet Library* and the *Sheet Annotation* options toggled **Off**.
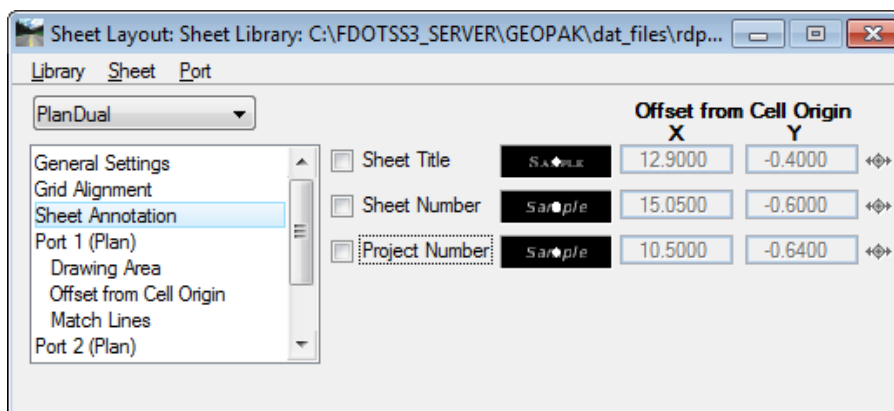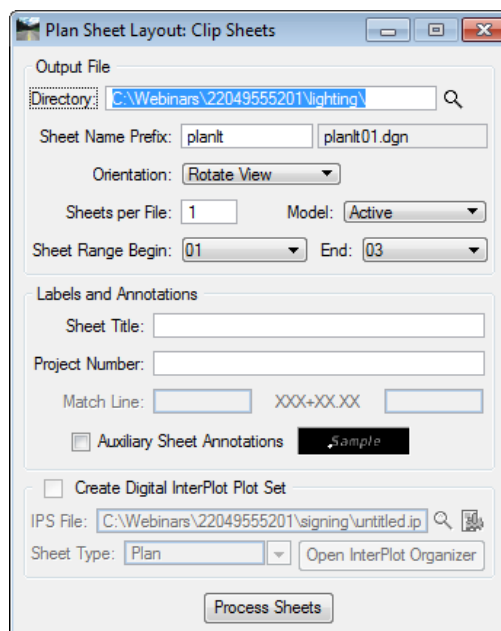
12. Click **Process Sheets**. This will start the sheet clipping process.

13. Close Clip Sheets.

14. Open **Planlt01.dgn** in the *lighting* folder.

15. Take a moment to review the sheet and reference files and level settings. If the settings are not correct, go into the **Motif** file or the **Cliplt** file to fix them. Then go back into the **Cliplt01.dgn** file and re-clip the sheets.

16. Close Plan Sheet Layout.

17. Click **Yes** to save settings to Project Manager.

➢ *Place North Arrow*

➢ *Now that there is a clip border it is easy to know where to place the North Arrow so that it falls within the plan sheet clip limits. It is very important that the designer set the view to the setting Unrotated before placing the North Arrow.*

1. Open *Dsgnlt01.dgn* in the *lighting* folder.

2. Zoom to *station* **707+00** on the **SR 61** *alignment*.

3. Attach the reference file **Cliplt01.dgn** from the *lighting* folder.

4. From the MicroStation view commands, select **Rotate View** and set to **Unrotated**.
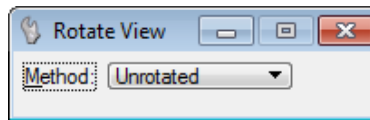
5. From Task Navigator, select **FDOT Plans Development > Roadway Plans > Place North Arrow**.
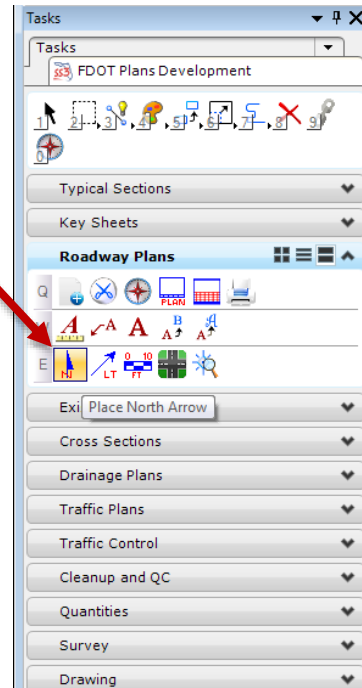
6. Select the **Place North Arrow** icon at the far left of the dialog.
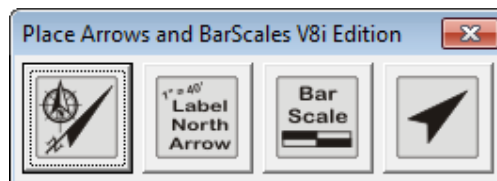
7. Place the **North Arrow** near the upper-right corner of the clip limits.

8. Right -click to **Cancel** the command.

➢ *Label North Arrow with the Scale*
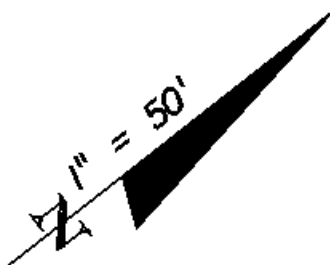
1. Rotate the view by **2 points**, selecting the bottom of the North Arrow line and then the point.

2. From the Place Arrows and Barscales toolbox**,** select the **Label North Arrow** icon.

3. Select the **North Arrow** just placed.

4. Issue a **Data Point** to place the label.

5. Repeat steps 5-12 for the remaining 2 clip borders to place the **North Arrows** and **Scale Labels**.

➢ *Fill In Title Block using Sheet Navigator*

1. Open **Bdpllt01.dgn** in the *lighting* folder.

2. From FDOT Menu**,** select **Actions > Sheet Navigator (Label Sheets)**.

3. Leave the *Sheet Number* field **blank**, this will be covered later using the Auto Numbering process.

4. For the *Financial Project 1***,** select the blank field, this will automatically populate.

5. Set the *County* to **Wakulla**.

6. For the *Road Number* enter **SR 61**.

7. Set the *Digital Signature Note* to **Standard**.

8. Click **Save Sheet**. This tags the sheet.

9. Close Sheet Navigator.

Note   The Sheet Description will be populated in the Plan Sheet.

EXPLORING D&C MANAGER

The Design and Computation Manager (D&C) Manager uses a proprietary database that is provided by the Department. When the Department's CADD software is installed, the latest D&C Manager database (DDB) is placed either on the server or on the local hard drive. For new projects, it is recommended to copy the latest DDB file from the FDOT installation folder into the project *symb* folder and rename it to the eleven digit FIN number. For the *22049555201* project used as the example in this manual, the *FDOTSS3.ddb* is renamed to *22049555201.ddb*. This allows the designer to modify the database for specific parameters, and protect it from being overwritten by any future maintenance updates. The latest FDOT DDB file is located in the \FDOTSS3\geopak\databases\ folder.

The naming format is *fdot####.ddb,* where **####** is the year or version of the Department's CADD software.

When D&C Manager is opened during a design session it should not be closed, minimize the dialog. This tool loads slowly because of the number of items in the database.

It is highly recommended that every user that works on projects becomes very familiar with D&C Manager. D&C Manager should be the standard tool used by everyone whether they are drawing simple lines or designing major interchanges. Many other applications and processes rely on the features that D&C Manager places on elements, if these features are not found then drawing cross sections, for example, would be nearly impossible.

This database has been set up specifically by the Department to create elements with the correct level symbology according to the Department's CADD Standards. The **.ddb** file is set up with discipline folders called categories. Inside of each category are items.
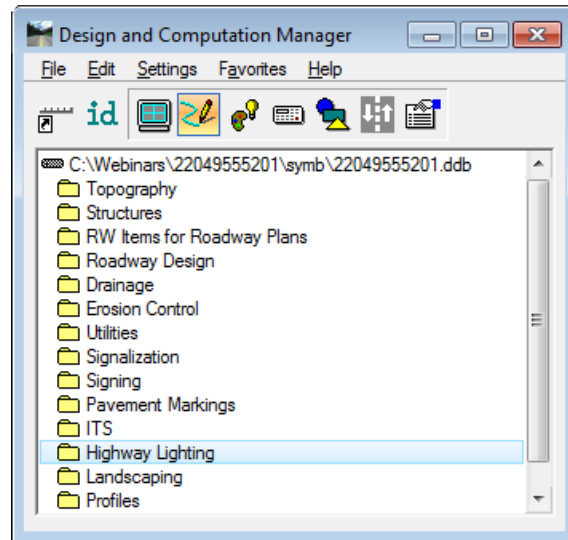
- **Categories -** The basic component of the hierarchical tree is the Category, which is represented by a folder icon. The FDOTSS3.ddb database categories are divided by discipline. The figure above shows the Categories with a description relative to their discipline. Inside these Categories, you have either subcategories or items. Items are represented by one of three icons as described below.

- **Items -** The other database component is the item. An item could be a drafting item, a compute item or a default item. Items contain specific functions related to defined element symbology or quantity calculations. Items are represented by one of three icons:

- **Default Icon -**These items are used to set drafting standards for MicroStation commands or to draw COGO elements without annotation. In many cases, this is a 3PC routine.

- **Drafting Standards Icon** - These items are used to set drafting standards for MicroStation commands or to draw COGO elements with annotation.

- **Calculator Icon** - These items are used to set drafting standards for MicroStation commands or to draw COGO elements. The graphics can be tagged with a pay item attribute for additional stratification of features when running computations. Computation parameters are defined for these items.

D&C MANAGER MENU BUTTONS

D&C Manager has a toolbar to quickly access different modes. The function of each button is summarized below. The tools used to create pavement marking plans will be covered in more detail later in this chapter.
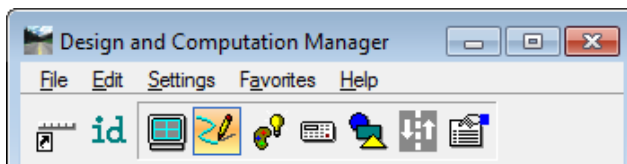
 - **Switch To Toolbox Mode**. D&C Manager is set up to work in two different modes, as a dialog box or a toolbox. This button activates the toolbox mode. This toolbox can be resized and docked. To change the display back to the whole dialog box, click the **Switch to Dialog Mode** button. The Place Influence check box is at the left end of the toolbox.

 - **Identify Item**. The **Identify Item** button is used to set the D&C Manager item to match a selected MicroStation element previously drawn by the D&C Manager or elements drawn with Graphical COGO and the SMD file. If the item you ID does not match an item in the active database, a message appears in the status bar saying: **No matching database item**. This is a very useful tool for new users learning the hierarchy of the database.

 - **Display**. The **Display** button filters the display of MicroStation elements in the design file so only the selected features are displayed, highlighted or hidden. This button expands D&C Manager to show a collection bin. This collection bin is for controlling the display of multiple items at one time. To add items to the collection bin, double-click the item. This tool also opens a second tool box with four buttons as seen in the figure below.

From left to right the buttons are Normal Display, Highlight Selection, Hide Selection and Display Only Selection. You will cover all of these in the exercises.

 - **Design**. The **Design** button is used to plot COGO and MicroStation elements into MicroStation with the defined symbology by use of Draw Plan and Profile dialog or in conjunction with MicroStation commands when **Place Influence** is selected.
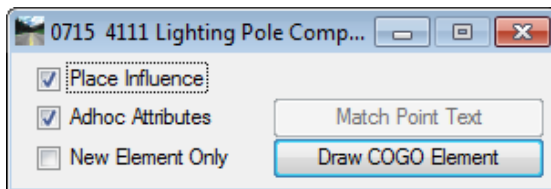
 - **Set**. The Set button is used to set the symbology of previously drawn MicroStation graphic elements in accordance with the parameters of a selected item in the database. This is the tool you use if an element needs to be fixed to meet CADD standards.
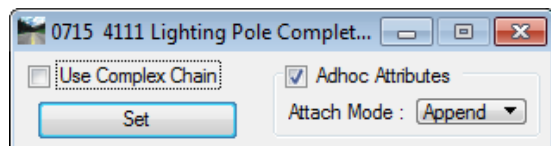
 - **Compute**. The Compute button is used for tabulating quantities of items that have been placed as a Pay Item by use of the Design or Set mode or have been drawn in MicroStation and they match the search criteria of the DDB file. This tool also expands D&C Manager to show the collection bin; this allows you to process multiple pay items at one time. A second toolbox containing the computation results is also opened. Computing is covered later in this course.
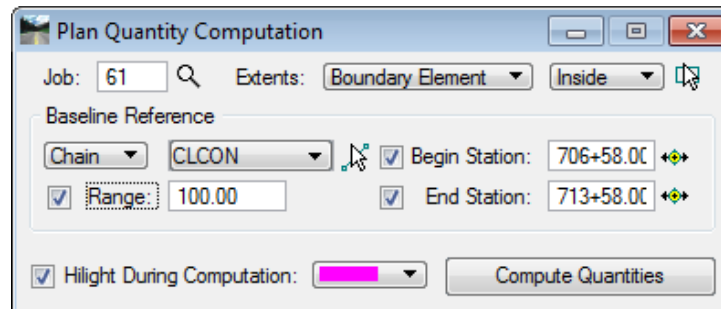
 - **Shapes**. The Shapes button uses plan view MicroStation graphics that defines an enclosed area to create a filled shape for computing area quantities. This tool will not be used in the creation of Pavement Marking Plans. However, this tool could be used to calculate Island Nose paint.

 - **Pavement Marking**. This mode provides additional options for placing pavement striping and markings. This tool adds four additional tools to D&C Manager as shown in the figure below.
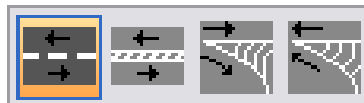
The four tools from left to right are: Striping, Separation, Chevron Diverge, & Chevron Merge

 - **Preference**. This expands the toolbar to include four additional buttons to configure D&C Manager. These tools are also accessible from the **Edit** menu.

DESIGN SETTINGS

Some settings need to be addressed before using D&C Manager to draw or compute items. These are the Design Settings loaded from the Settings > Design menu in D&C Manager.
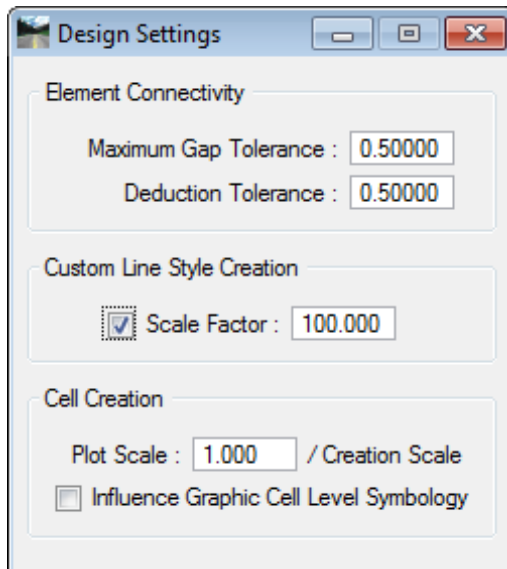
- **Maximum GAP Tolerance** – If the distance between two specified elements in a MicroStation file is smaller than the **Max Gap Tolerance**, the software assumes the two elements intersect and act accordingly. If the distance is larger than the **Max Gap Tolerance**, GEOPAK assumes the two elements do not connect.

- **Deduction Tolerance** –Utilized in the Compute mode. For example, if the pay item is specified for a curb line, and for each manhole (drawn in as a cell), there is a deduction of six feet. The origin of the cell does not have to be on the curb line, but must be within the deduction tolerance in order for the cell to be recognized and the deduction to be made.

- **Custom Line Style Creation** – The Custom Line Style *Scale Factor* utilized during the Draw Plan & Profile dialog. Note this scale is only for custom line styles. Text and other labeling are controlled by the Label Scale on the Draw Plan & Profile dialog.

- **Cell Creation** – **Plot Scale** is utilized as a ratio with the Creation Scale within the setup of D&C Manager. If the Cell Creation Scale is 10 and the designer wants the cell twice as large as a cell placed with D&C Manager, utilize 20 as the Drawing Scale.

  If a drawing scale is entered, all subsequent cells that are placed from the D&C Manager are scaled accordingly if the item is set to use creation scale. When a cell is placed with place influence on, GEOPAK utilizes the drawing scale setting to compute an X & Y scale for use in the MicroStation Place Cell dialog. Why do this? Otherwise, the designer would have to type the X & Y scale in the Place Cell dialog whenever another item is placed from D&C Manager. This also allows use of a metric DDB with English cells by using a different creation scale.

- **Influence Graphic Cell Level Symbology** – When active, GEOPAK utilizes the symbology within the D&C Manager, ignoring the element symbology defined in the cell.

DESIGN MODE

Design Mode is the default mode when the D&C Manager is opened. Design Mode is used for the following functions:

- Set Drafting Standards by use of the **Place Influence** command for the placement of MicroStation elements using MicroStation commands.

- Print COGO elements into the design file according to the drafting standards set by the item's defined parameters.

- Place Adhoc Attributes on an element.

Whether an item is placed in the file by generic MicroStation commands or printed from the COGO database, the items can be placed as pay items for future tabulation. The following sections will detail the procedure to set drafting standards for MicroStation commands and for the printing of COGO elements.

By selecting *Place Influence*, the designer can use MicroStation commands to place elements utilizing the element attributes established for the currently selected item in the GEOPAK D&C Manager database. With some items, a GEOPAK attribute is placed with the element for calculating quantities.
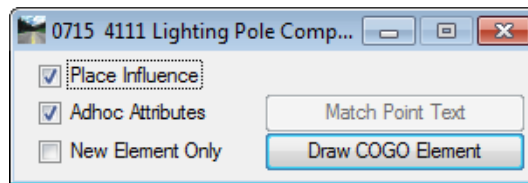
When *Place Influence* is selected, the MicroStation level symbology is set. Any MicroStation command to draw a line, copy a line or place a cell is set to this symbology.
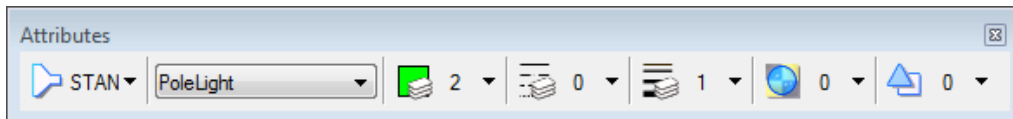
When finished placing the elements for a selected item it is important to remember to turn off **Place Influence**.

It cannot be stressed enough how important it is that all users working on projects become familiar with D&C Manager and use this tool for everything they do. D&C Manager should be the first tool opened when MicroStation is started and the last tool closed when the work day is done.

ADHOC ATTRIBUTES
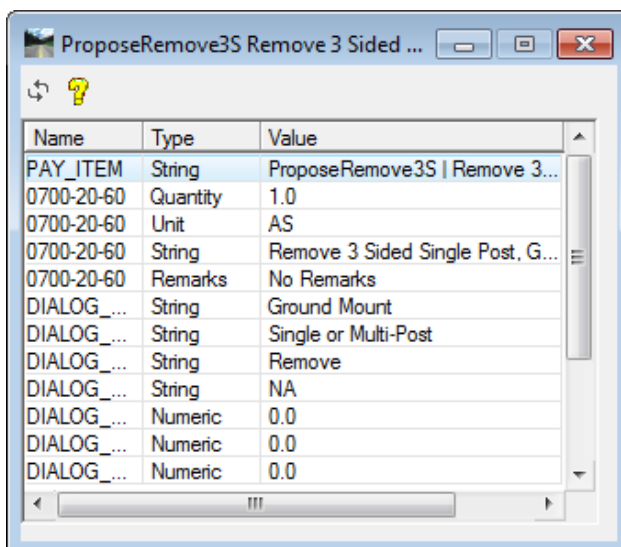
An Adhoc Attribute is additional information about a particular element. Look at it as another property for the MicroStation element. Example, a line has this fundamental data associated to it:

- Level

- Color

- Weight

- Style

By placing an Adhoc on the line, the designer is giving that line additional information. That information could be a chain name, cross slope, profile name, thickness, etc. It is almost limitless as to what can be associated to an element with Adhocs. These Adhocs can be used by other down-stream applications to generate quantities, draw cross sections or many other tasks.
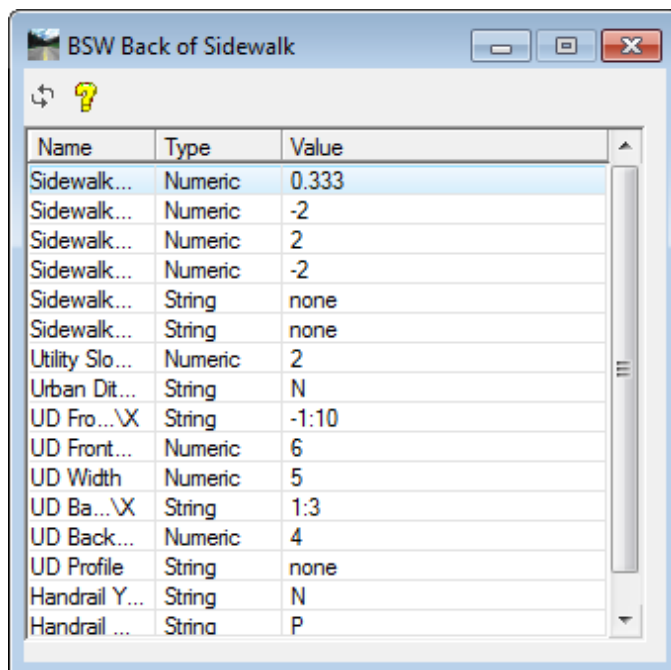
Some Adhocs are placed in the background and the designer has no interaction with them. For example, when a proposed & remove sign post is placed with the Draw Sign Program, a 3pc file is utilized to access the D&C Manager (.ddb) to place Adhocs on the post symbol for computing quantities.

The figure below shows the Adhocs that are associated to an item in the **D&C Manager**. These Adhocs are part of the item and are set by default. Nothing indicated this information was set.

Other Adhocs are interactive meaning the designer must fill in the parameters. An example is when placing the back of sidewalk. There are several pieces of additional information that can added to the sidewalk line. This information is then used to draw the sidewalk in the cross sections.

Adhoc Attributes are comprised of three types of information that must be defined:

1. **Name** - The *Name* is an identifying term used when GEOPAK is searching for a specific Adhoc Attribute.

2. **Type** - The *Type* identifies the nature of the information, and can be set to various options: Numeric, String, Unit, Quantity, and Remarks.

3. **Value** - The *Value* is the actual information to be used by GEOPAK, and is determined by the Type. For example, if the Type is set to Numeric then the Value must be a number.

DRAWING LIGHT POLES

Generating the proposed lighting design is the responsibility of the engineer. It is also the engineer's responsibility to make sure an acceptable program is used to generate the proposed design. Currently the Department uses AGI 32 when the design is done in-house. It is possible that the proposed design is done by the lighting manufacturer or Power Company; however, they will have to generate an acceptable report that is to be submitted to the Department for review. This report can be used by the in-house designers to create the lighting design files i.e. light pole spacing.

There are a couple of ways to approach drawing the Light poles into MicroStation, one is to import the elements from the AGI 32 lighting program into MicroStation and then place the light pole cell on top of the AGI 32 element. Another way is to reference the AGI 32 file into the lighting design file then place the light pole cell on top of the AGI 32 element. A third option is to take the report generated from AGI 32 or another acceptable lighting program which details the spacing of the light poles then use D&C Manager in conjunction with other GEOPAK tools to draw the light poles at the specified spacing.

DRAWING PULL BOXES

Pull Boxes are also an important part of the lighting plans. In this course the designer will learn how to place the pull boxes in two different scenarios; one is at driveways or side streets where the conduit may change from underground to under pavement. The other is at the light poles, each proposed light pole has a pull box however these pull boxes are not always drawn in the plans due to plan clarity but they are quantified. The designer will learn how to apply an Adhoc attribute on the light poles to include a pull box for automated quantities.

DRAW CELL BY FEATURE

Another option the designer has to draw the light poles is the *Draw Cell by Feature* (best suited for placing one cell at a time) or *Draw Cell Group by Feature* tools in GEOPAK (designed for multi cell placement).

> *Note* When using the Draw Cell Group by Feature and/or Draw Cell by Feature tools, always ensure to load the correct Geopak Database (*.ddb) as the tool will 'reset' the Database to the *.ddb specified in the resource file every time the tool is closed and reopened. This Geopak Database (*.ddb) may not be the database intended for use with your active project.

The *Draw Cell by Feature* tool can be loaded from the MicroStation menu option Applications > Road > Plans Preparation > Draw Cell by Feature.

This tool can also be loaded from the *Road* tools palette.

The *Draw Cell by Feature* tool is very user friendly and requires little explanation.
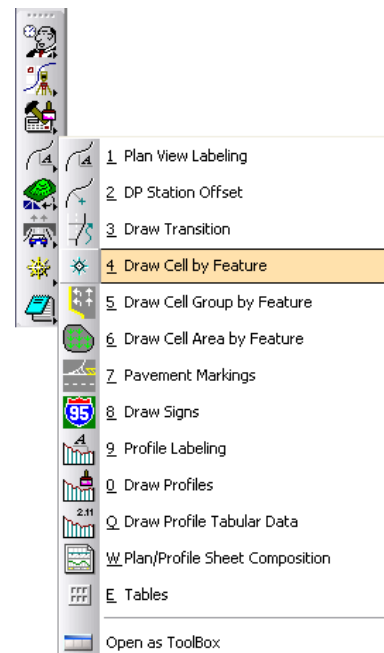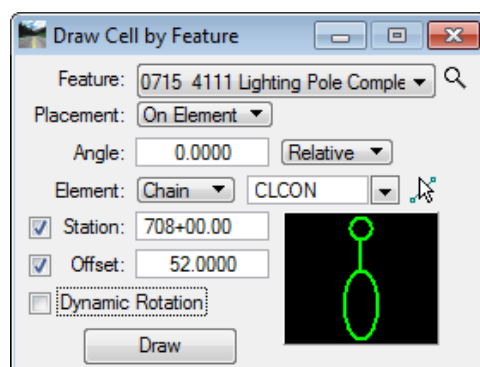
DRAW CELL GROUP BY FEATURE

Another option the designer has to draw the light poles is the Draw Cell Group by Feature tool in GEOPAK. This tool works well when placing multiple cells at one time.

The Draw Cell Group by Feature tool can be loaded from the Applications pull down Applications > Plans Preparation > Draw Cell Group by Feature.

This tool can also be loaded from the **Road Tools** palette.

FEATURE GROUP TAB

The *Feature Group* defines the D&C Manager item to use which in turn selects the cell to be placed and the Offset from the selected element to place the item.

Based on the items that are checked the designer can place up to three different cells with this tool. The Magnifying glass next to each item allows the designer to browse to D&C Manager and select the appropriate item which will attach the cell to be drawn. The Offset is the distance off of the selected DGN Element or Chain when drawing the cell.

Road Tools palette menu:
- 1 Plan View Labeling
- 2 DP Station Offset
- 3 Draw Transition
- 4 Draw Cell by Feature
- 5 Draw Cell Group by Feature
- 6 Draw Cell Area by Feature
- 7 Pavement Markings
- 8 Draw Signs
- 9 Profile Labeling
- 0 Draw Profiles
- Q Draw Profile Tabular Data
- W Plan/Profile Sheet Composition
- E Tables
- Open as ToolBox
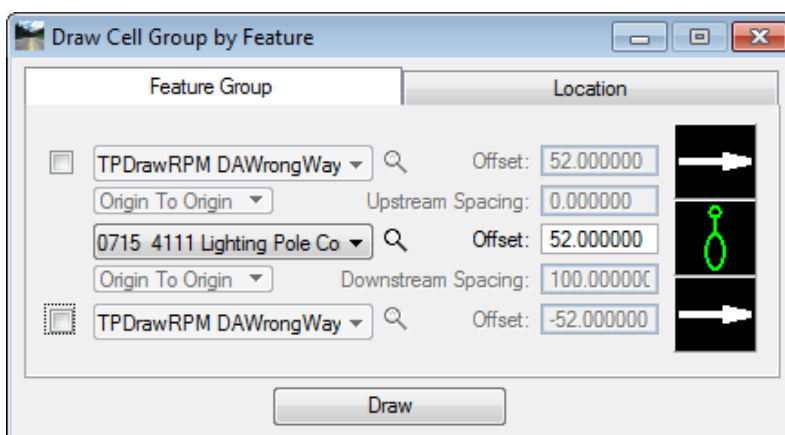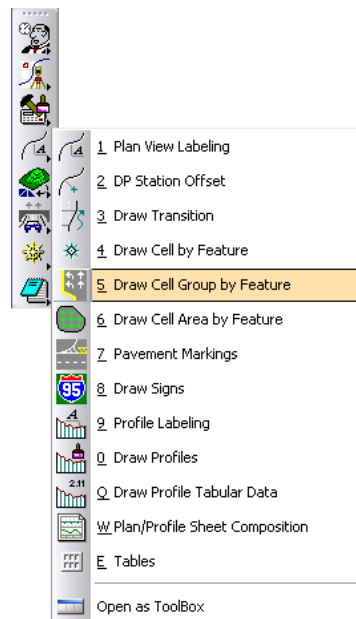
LOCATION TAB

The *Location* defines whether the designer uses a DGN element or Chain, Begin and End stations and Spacing.
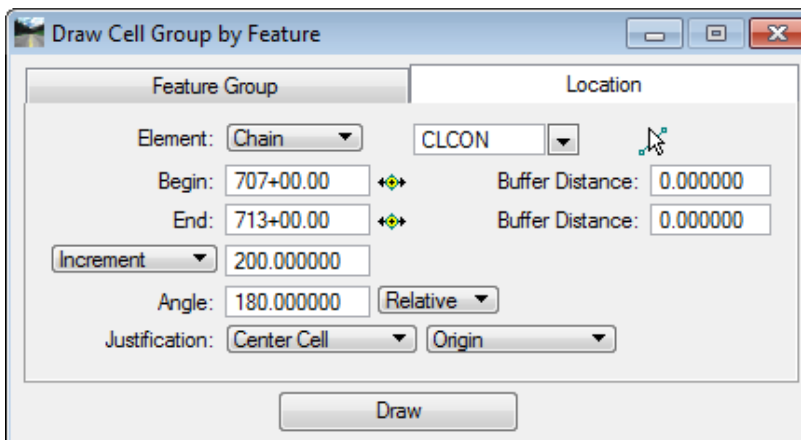
- **Element** – This is either a GEOPAK Chain or DGN element. This functions just like the Draw Striping tool. The Begin and End fields will be populated with the appropriate data based on the element selected. They can be modified using the buttons or by keying in the value.

- **Buffer Distance** – This is for setting where to start the first cell and end the last cell.

- **Spacing** – There are several options available to set the spacing between the cells, they are:

- **Increment** – This is a set value between cells.

- **Even** – Evenly spaces the cells at user specified spacing, the buffer distances are ignored.

- **Max Spacing** – The location of the beginning and ending cell are determined, than a sufficient number of cells are placed in between, so that the distance between them is no more than the specified Max Spacing.

- **Once** – Only one set or cell is placed. The ending buffer distance is ghosted.

- **Each Vertex** – The origin of the cell is placed at each vertex of the selected element. The Begin and End and buffer distances are ghosted.

- **End Points** – The origin of the cell is placed at each end point of the selected element. The Begin and End and buffer distances are ghosted.

- **Angle** – If set to *Absolute* the angle is based on 0 degrees as horizontal. If set to *Relative* the cell is placed relative to the element selected.

- **Justification** – The justification can be based on the center cell (only option if only one row is placed), the upstream or downstream cell. Only those toggled on in the **Feature Group** are available for setting Justification.

REVIEW LIGHT POLE ITEM IN D&C MANAGER

Before placing the *Light Poles* it is important to understand how the Light Pole items are set up. The DDB file that the Department delivers will have Adhocs assigned to the Light Pole item to account for a Pull Box. This allows Pull Boxes to be automatically quantified without having to draw them. The next figure shows the Light Pole item in D&C Manager.
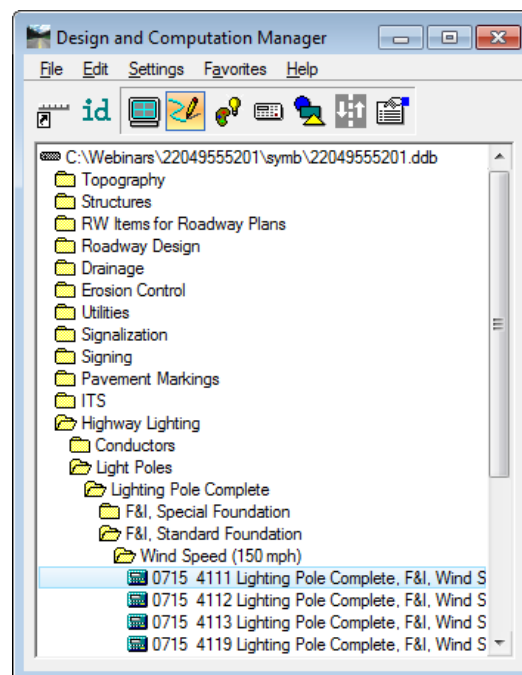
Reviewing the *Light Pole* item the designer will notice:

- The *Adhoc* option is checked **On**.

- The *Compute Parameters* have been set to compute by **Adhoc.**

Reviewing the *Adhoc Attributes* the designer will notice that there are two Adhocs assigned to the Light Pole.

- Pull Box.

- Light Pole.

The Light Pole has to be assigned as an Adhoc because of how the *Compute Parameters* are set. Notice in the figure above that the Unit for the Light Pole and Pull Box are set to EA. Even though the quantities are being computed using Adhocs, it will still break the items out as EA.

The next figure shows the *Compute Parameters*, notice that the *Compute Method* is set to **Adhoc Attribute**.

Exercise 4.2 *(Optional Exercise) Placing Light Poles Using Draw Cell by Feature*

In this exercise the designer will place Light Poles at a specified spacing and offset supplied from a referenced AGI 32 Lighting DXF file.

1. Continuing in *Dsgnlt01.dgn*, reference in the **AGI32Lighting.dxf** file. The light poles locations are along the **SR61 Corridor** offset from the *CLCON Chain*.

2. Read DWG/DXF Units dialog and click the **OK** button. (Leave the dialog set to **Survey Feet**.)

3. Select **Model** from the drop down on the Reference Attachment Settings dialog. The selection is *Model* or *Layout*.
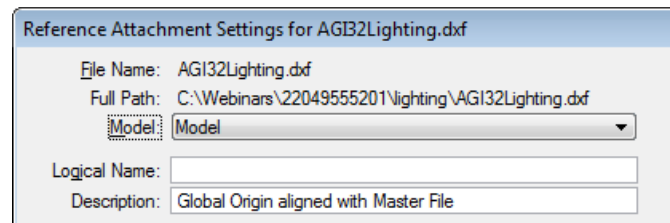
4. Click the **OK** button on the Reference Attachment Settings dialog. This attaches the **AGI32Lighting.dxf** file.

5. Close the Reference dialog.

6. Zoom into *station range* **716+00 to 719+00** and **Rotate the View** accordingly, if necessary.

7. From the Road Tools palette, click on the **Draw Cell by Feature** tool.

8. Select the **0715 4111 Lighting Pole Complete, F&I, Wind Speed 150, Pole Height 40'** item from the DDB by clicking the *Feature magnifying glass* and navigating to: **Highway Lighting > Light Poles > Light Pole Complete > F&I, Standard Foundation > Wind Speed (150 mph)** category.

9. Set the *Placement* to **On Element**.

10. Set the *Element* to **Chain CLCON**.

11. Set the *Station* to **716+00** and toggle on the lock or use MicroStation tools to place the light pole cell on the AGI32 marker.

12. Set the *Offset* to **52** and toggle on the lock or use MicroStation tools to place the light pole cell on the AGI32 marker.

13. Click the **Draw** button on the Draw Cell by Feature dialog. With the *Station* and *Offset* lock toggles **On** the light pole cell origin is locked in the correct location.

14. Repeat **steps 11-14** for the remaining 3 light poles to place within the Station range specified in step 6 with following information.

> *Hint*   Try to use MicroStation tools to place the light pole cell on the AGI32 markers.

Angle	Station	Offset	Side
180	717+00	52	Right
0	718+00	52	Left
180	719+00	52	Right

15. Take a moment to review the Adhocs being placed on the poles.
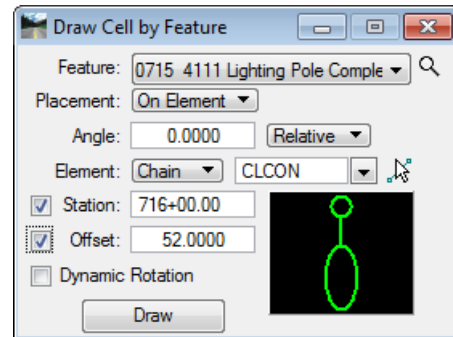
Exercise 4.3 *Placing Light Poles Using Draw Cell Group by Feature*

➢ **Place Light Poles at Specified Spacing and Offset (Part 1)**

In this exercise the designer will place Light Poles at a specified spacing and offset. On a real project this spacing and offset information would come from the AGI 32 Lighting design software or another Department acceptable design method.

✓ *Load DDB and Select Light Pole Item*

1. Continuing in *Dsgnlt01.dgn*, zoom to near **station 702+00**.

2. From the Road tools palette, click on the **Draw Cell Group by Feature** tool.
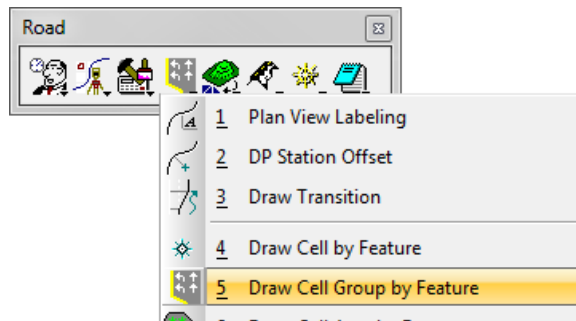
3. On the *Feature Group* tab click on the *magnifying glass* icon to open D&C Manager.
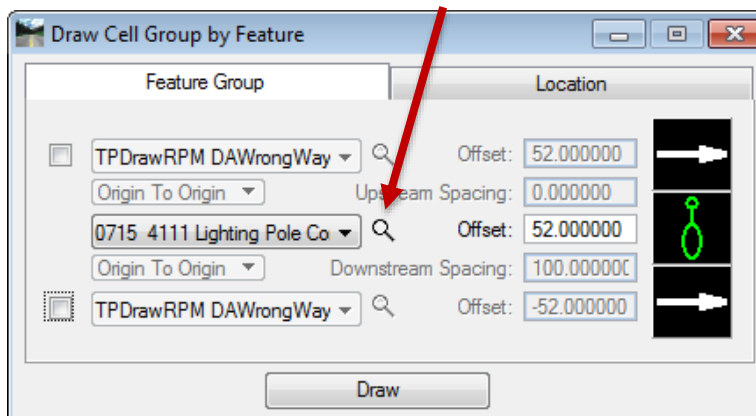
4. Verify the project specific DDB (**22049555201.ddb**) is loaded. If not, select **File > Open** from the Design & Computation Payitem dialog.

Note   If the correct DDB file is loaded, skip the following steps and move on to step 7.

5. Navigate to the *symb* folder and select **22049555201.ddb**. This is the project DDB file; it is a copy of the current *FDOTSS3.ddb* file renamed to the eleven digit fin number.

6. Click **OK**. This loads the selected DDB file.

7. Navigate to the **Highway Lighting > Light Poles > Light Pole Complete > F&I, Standard Foundation > Wind Speed (150 mph)** category.

8. Select the **0715 4111 Lighting Pole Complete, F&I, Wind Speed 150, Pole Height 40'** item.
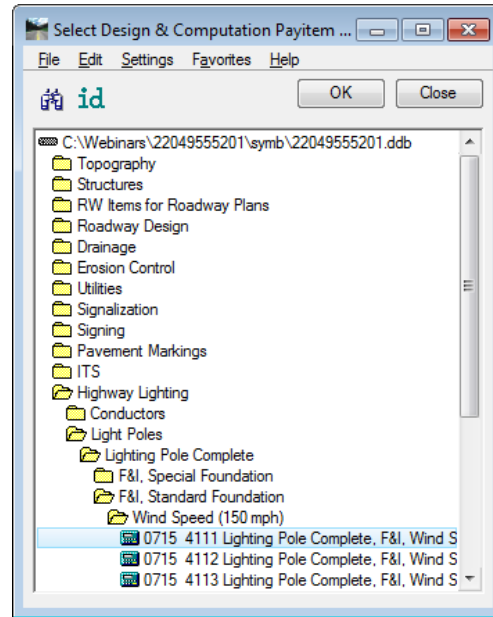
9. Click the **OK** button or double-click on the item. This will close Select Design & Computation Payitem and push these items over to the *Feature Group* tab.

10. On the Draw Cell Group by Feature, from the drop down arrow next to the *magnifying glass* select the item **0715 4111 Lighting Pole Complete, F&I, Wind Speed 150, Pole Height 40'**. This sets the item in the tool.
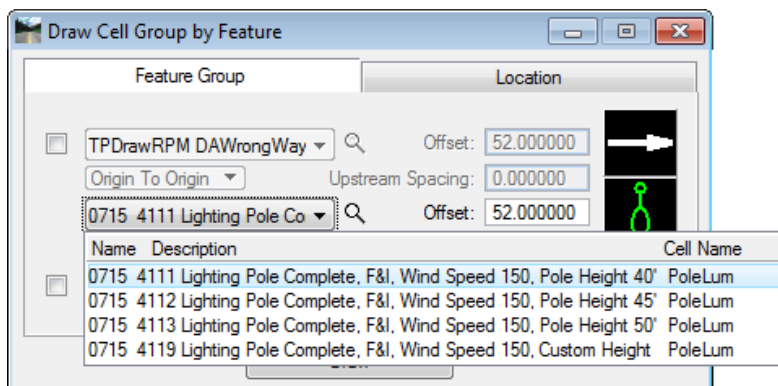
11. Set the *Offset* to **52.0.** This will be the offset from the **CLCON Chain** to the center of the light pole origin. Typically, this information would come from the lighting design software.

12. Select the **Location** tab.

13. Set the *Element* to **Chain**.

14. From the *chain* drop down menu select **CLCON**.

15. Set the *Begin station* to **702+00**.

16. Set the *End station* to **706+00**.

17. *Begin* and *End Buffer Distance* should be **0.00**.

18. From the drop down menu, set to *Increment* and the *Value* to **200.00**. Typically, this value would come from the lighting design software.

19. The *Angle* should be **0.00** and set to **Relative**. *Angle* **0.00** is for the *left side* of the road, **180.00** would be for the *right side* of the road.

20. Set the *Justification* to **Center Cell** and **Origin**.
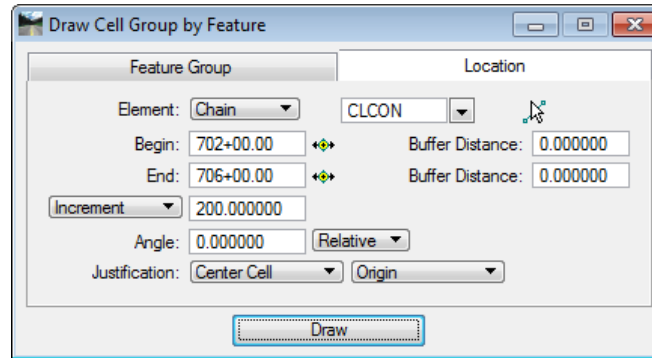
21. Click the **Draw** button. Next the designer will place the cursor on the left side of the road.

22. This will open the Adhoc Editor dialog. This allows the designer to edit the default values if needed. Items that are in Red are not modifiable, they are locked.

23. Move the pointer to the *left side* of the road and issue a **Data Point**. Light Poles should display.

➢ ***Draw the Light Poles on Right Side of Road (Part 2)***

In this part of the exercise the designer will draw the light poles on the right side of the road.

1. Continuing in the *Location* tab of the Draw Cell Group by Feature, set the *Begin station* to **701+00**.

2. Set the *Begin Buffer Distance* and *End Buffer Distance* to **0.00**.

3. Set the *End station* to **705+00**.
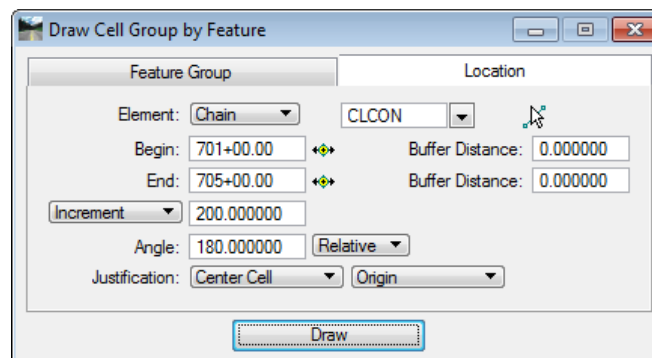
4. Set the *Angle* to **180.00**.

5. Click the **Draw** button. Next the designer will place the cursor on the right side of the road.

6. Move the pointer to the *right side* of road and issue a **Data Point**. Light Poles should display staggered.

Note   The designer could also stagger the placement of the poles by adding 100' to the Begin and End station Buffer Distances.

➢ *Draw Light Poles on Left & Right Sides of Road (Part 3)*

In this part of the exercise the designer will draw the light poles on the left and right sides of the roadway practically at the same time. This exercise is to show the capability of the Draw Cell Group by Feature tool. In some cases it may not be practical to use this technique when another method may be more efficient. This technique requires the designer to create a false compute item in the DDB specifically for cell placement of the light poles and also requires a little cleanup after light pole cell placement. However, this technique allows for more control of cell placement, especially with stationing around curves.

✓ *Load Project Specific DDB and Create 'False' Computation Item*

1. Continuing in *Dsgnlt01.dgn***,** zoom to *station range* **707+00 – 714+00**.

2. Open the D&C Manager**.**

3. Verify the project specific DDB (**22049555201.ddb**) is loaded. If not, select **File > Open** from the Design & Computation Payitem dialog.

Note	If the correct DDB file is loaded, skip the following steps and move on to step 6.

4. Navigate to the *symb* folder and select **22049555201.ddb**. This is the project DDB file; it is a copy of the current *FDOTSS3.ddb* file renamed to the eleven digit fin number.

5. Click **OK**. This loads the selected DDB file.

6. Navigate to the **Highway Lighting** category.

7. Right click on the *Highway Lighting* category and select **New Category**.

8. In the Name field type **Misc**. Click **OK**.
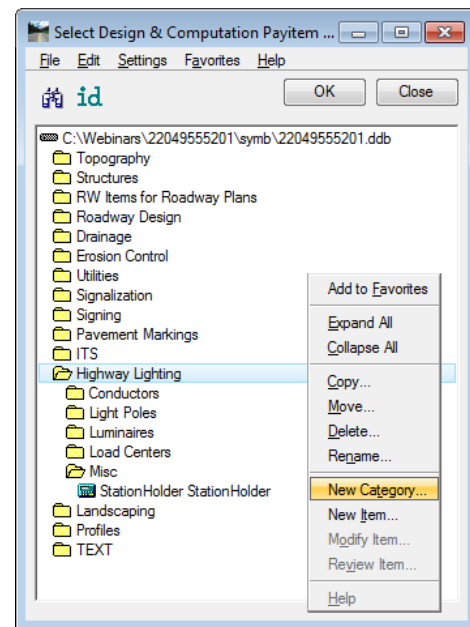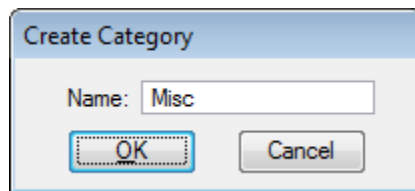
9. Right click on the new **Misc** *category* and select **New Item**.

10. Type **StationHolder** in the *Item ID* field.

11. Copy the **Item ID** text to the *Description* field.

12. From the drop down list next to the *Item ID,* select **Compute Parameters**.

Note	Very important! In order for the Draw Cell Group by Feature tool to recognize and place this item, it has to be set to Compute Parameters.

13. Toggle **On** *Attribute* and type **STAHOLD** in the field.

14. Type **ActivePoint** in the *Cell Name*.

Note	This cell is a tiny point set as a construction element and could be left in the file without being printed if the designer uses a Department delivered Pen Table.
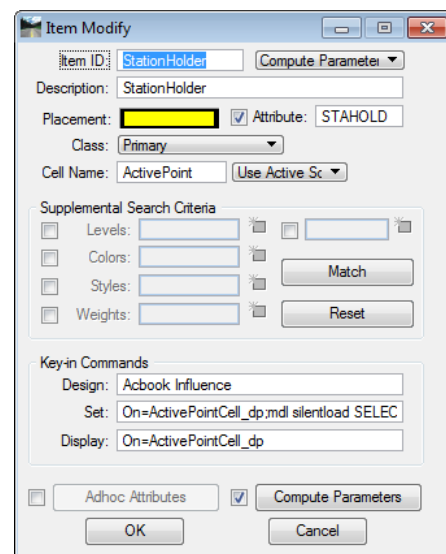
15. In the *Key-in Commands* fields type in the following:

    Design: **Acbook Influence**

    Set: **On=ActivePointCell_dp;mdl silentload SELECTBY dialog**

    Display: **On=ActivePointCell_dp**

16. Toggle **On** *Compute Parameters* and click the **Compute Parameters** button.

17. Set the *Computation Method* to **Standard Unit**.
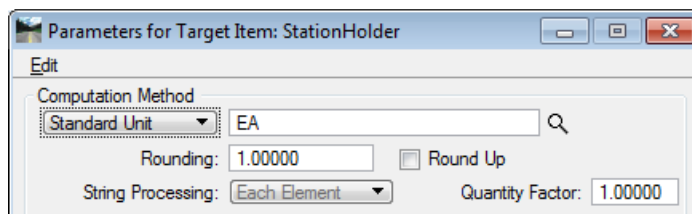
18. Click the *Computation Method magnifying glass* and select **EA** from the *English Unit* table.
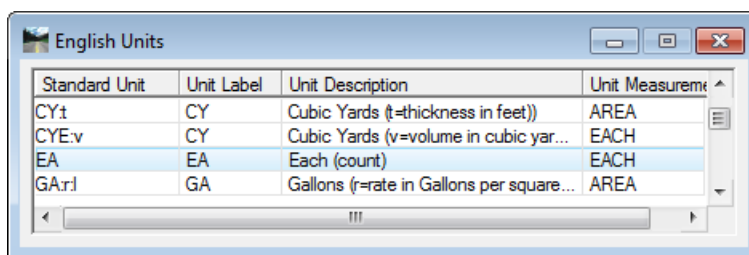
19. Click the red **X** in the corner of the English Units dialog to close the dialog.

20. Click the **OK** button on the Parameters for Target Item dialog.

21. Click the **OK** button on the Item Modify dialog.

22. Navigate to **File > Save** on the D&C Manager dialog to save changes to the *Project Specific DDB*.

23. Close the D&C Manager dialog.

    ✓ *Load Project Specific DDB and Select 'False' Computation Item & Lighting Pole Item*

24. On the *Feature Group* tab click on the *magnifying glass* icon to open D&C Manager.
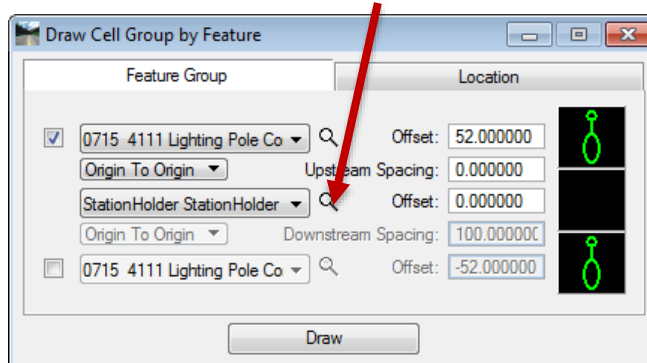
25. Navigate to the **Highway Lighting > Misc** category.

26. Select the **StationHolder** item.

27. Click the **OK** button on the Select Design & Computation Payitem dialog.

28. Set the *Offset* to **0.00**.

29. Toggle **On** *Item* above the *Station Holder* on the *Feature Group* tab and click on the *magnifying glass* icon to open D&C Manager.

30. Navigate to the **Highway Lighting > Light Poles > Light Pole Complete > F&I, Standard Foundation > Wind Speed (150 mph)** category.

31. Select the **0715 4111 Lighting Pole Complete, F&I, Wind Speed 150, Pole Height 40'** item.
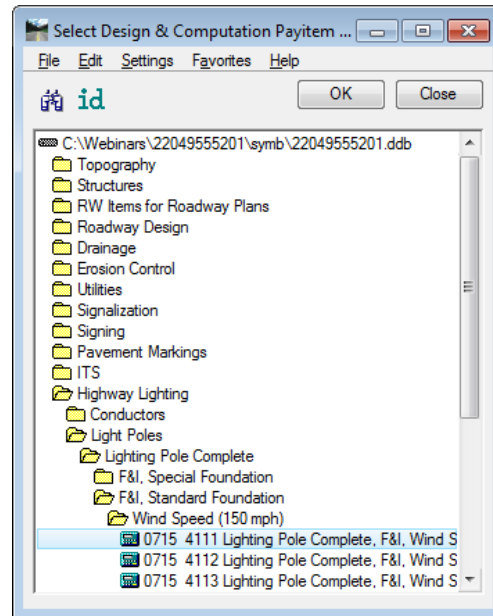
32. Click the **OK** button or double-click on the item. This will close Select Design & Computation Payitem and push these items over to the *Feature Group* tab.

33. On the Draw Cell Group by Feature, from the drop down arrow, select the item **0715 4111 Lighting Pole Complete, F&I, Wind Speed 150, Pole Height 40'**. This sets the item in the tool.

34. Set the *Offset* to **52.0.** This will be the offset from the **CLCON Chain** to the center of the light pole origin. Typically, this information would come from the lighting design software.

35. Select the **Location** tab.

36. Set the *Begin Station* to **707+00**.

37. Set the *Begin Buffer Distance* and *End Buffer Distance* to **0.00**.

38. Set the *End Station* to **714+00**.
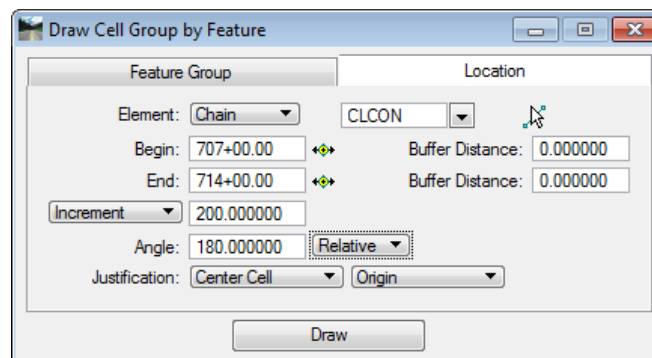
39. From the drop down menu, set to **Increment** and the *Value* to **200.00**. Typically, this value would come from the lighting design software.

40. The *Angle* should be **180.00** and set to **Relative**. *Angle* **0.00** is for the *left side* of the road, **180.00** would be for the *right side* of the road.

41. Set the *Justification* to **Center Cell** and **Origin**.

42. Click the **Draw** button. Next the designer will place the cursor on the right side of the road.

43. Move the pointer to the *right side* of the road and issue a **Data Point**. Light Poles display staggered.

44. Technically, the designer could copy the PoleLum cell, rotate the copy 180° and save the cell to a project specific cell library. Then the designer could set up another DDB entry in their project specific DDB to call out the new cell and the Left and Right side light pole cell placement could be done at the same time. Due to time constraints in the classroom setting, this detail was not performed in the exercise.

However, to place the light poles on the left side of the roadway using this technique, there are a number of small adjustments that can be made on the Draw Cell Group by Feature dialog (on the *Feature Group* tab and/or the *Location* tab) at this point in the process. The following are a few of the most notable:

o On the *Feature Group* tab: toggle **On** the *DDB* item below the *StationHolder*, enter **52** in *Offset*, and enter **100** in *Upstream Spacing*. Switch to the *Location* Tab: change the *End station Buffer Distance* to **-100** *(enables placement of the light pole at station 714+00)* and change the *Angle* to **0.00**. Click the **Draw** button and move the cursor to the *left side* of the roadway and issue a **Data Point**.
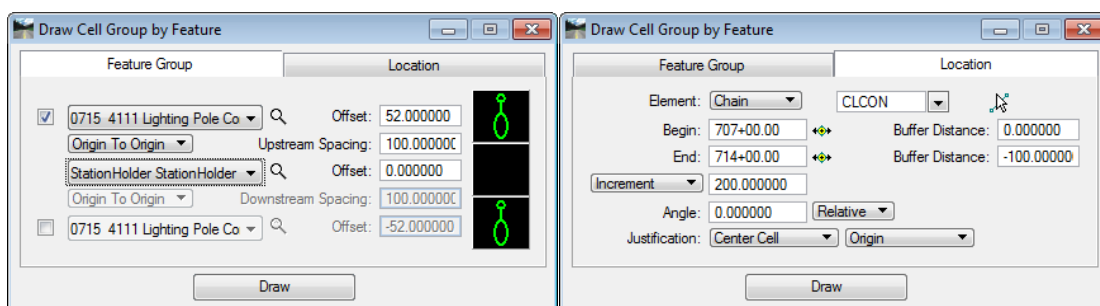
o On the *Feature Group* tab: toggle **On** the *DDB* item above the *StationHolder* and enter **52** in *Offset*. Switch to the *Location* Tab: change the *Begin Station Buffer Distance* to **100** *(enables placement of the light pole at station 714+00)* and change *Angle* to **0.00**. Click the **Draw** button and move the cursor to the *left side* of the roadway and issue a **Data Point**.
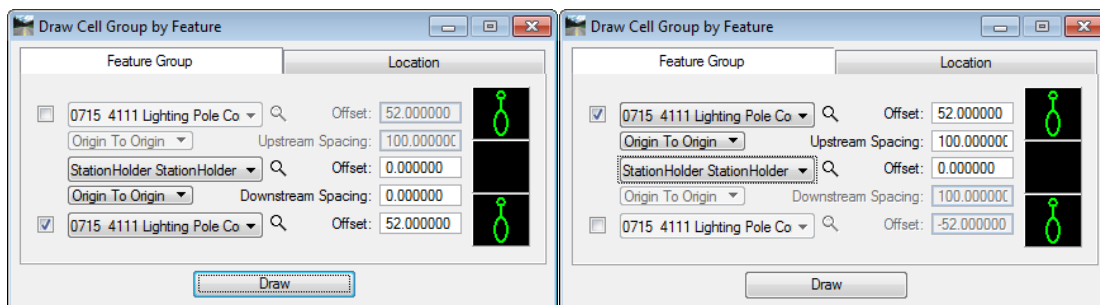
o On the *Feature Group* tab: toggle **On** the *DDB* item below the *StationHolder* and enter **52** in the *Offset*. Switch to the *Location* tab and change *Angle* to **0.00**. Click the **Draw** button and move the cursor to the *left side* of the roadway and issue a **Data Point**.
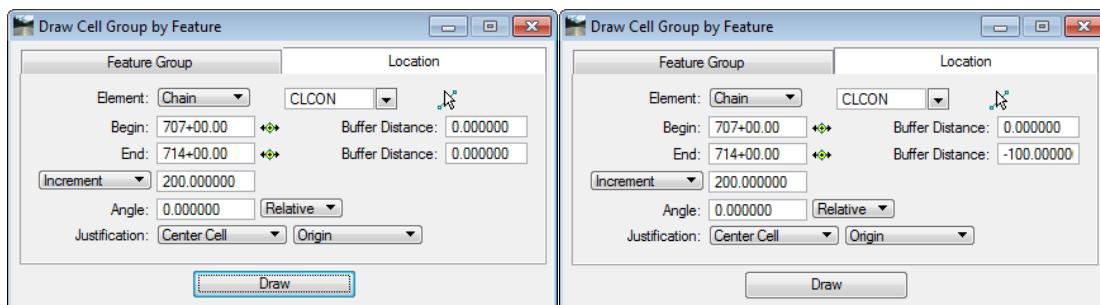
45. Use one of the above methods to place light poles on the *left side* of the roadway around the curve.

46. After clicking the **Draw** button, just mouse over the curve area *without* issuing a Data Point. This allows for a preview of the light pole placement before actually placing the light pole cells.

47. Issue a **Data Point** on the *left side* of the roadway after testing the methods referred to in step above.

48. Clean up the '*Station Holders'* along the curve on chain **CLCON**.

49. Close Draw Cell Group by Feature.

REVIEW ADHOC ATTRIBUTES

GEOPAK's Adhoc Attribute Manager is used to view items placed with D&C Manager to see if any Adhocs have been tagged to elements and if there are Adhocs this tool will allow the designer to edit the values if necessary.

The Adhoc Attribute Manager tool can be loaded from the Road tools palette.
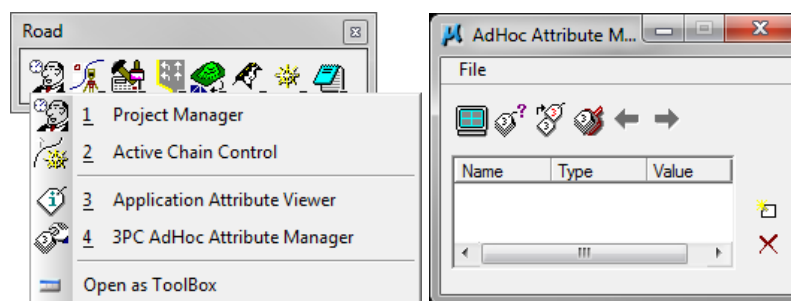
Clicking on the question mark icon and then selecting one of the Light Poles will display any Adhocs tagged to the selected element as shown next.
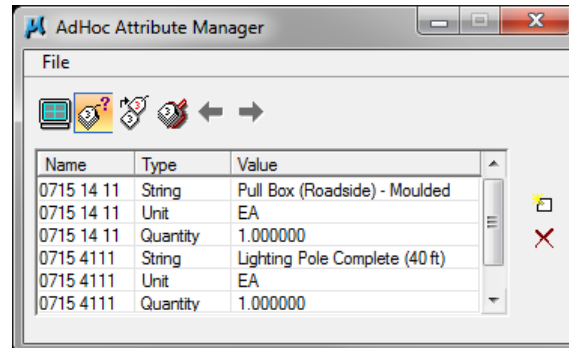
Users can be assured that any items that display in this dialog will quantify properly.

Exercise 4.4 *Modify Pull Box Item in D&C Manager*

➢ **Edit Pull Box Item To Set Scale (Part 1)**

In this exercise the designer will edit the pull box item to set the scale option to use active scale. If the Pull Box item is already set to **use Active Scale**, skip this exercise.

1. Open D&C Manager if closed. Navigate to and double-click on the **Highway Lighting** category.

2. Double-click on the **Pull Boxes** category.

3. Double-click on the **Furnish & Install** category.

4. Right-click on the item **0715 14 11 Pull Box (Roadside) – Moulded**.

5. From the popup menu, select **Modify** item. This opens Item Modify.
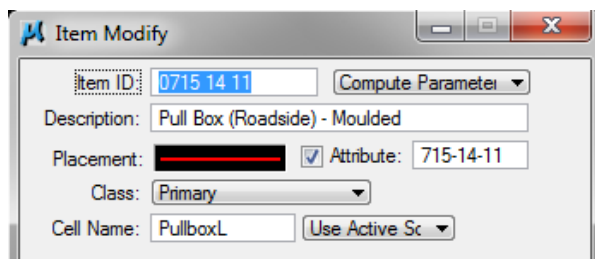
6. From Item Modify right side, set *Scale* to **Use Active Scale** from the drop down menu.

7. Click **OK**.

8. In D&C Manager, click **File > Save** to permanently save the changes.

➢ **Placing Pull Boxes (Part 2)**

In this exercise the designer will use the *Draw Cell by Feature* tool in conjunction with D&C Manager to place Pull Boxes at a driveway.

1. Continuing in *Dsgnlt01.dgn*, zoom to the *right side* of the roadway at *station* **704+20**.

2. **Rotate the view** as necessary.

3. From Road tools palette, open Draw Cell by Feature.
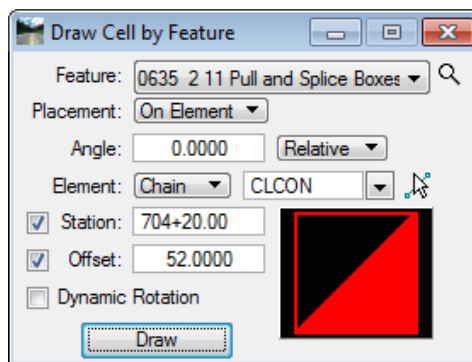
4. Click the *Feature magnifying glass* icon. This opens Select Design & Computation Payitem. Make sure the correct *DDB* file is loaded.

5. Navigate to and double-click on the **Signalization** category.

6. Double-click on the **Pull and Junction Boxes** category.

7. Select the **Pull and Splice Boxes > Pull and Splice Boxes > Item 0635 2 11 Pull and Splice Boxes, F&I, 13" x 24"**.

8. Click **OK** or double-click on the item. This will close Select Design & Computation Payitem and push the pull box items into the Draw Cell by Feature dialog.

9. On Draw Cell Group by Feature, click on the *Feature* drop down list and select **Item 0635 2 11 Pull and Splice Boxes, F&I, 13" x 24"**.

10. Set the *Placement* to **On Element**. The other option is *Point*.

11. Set the *Angle* to **0.00** and **Relative**.

Note   Relative will rotate the cell dynamically as the user slides along the Chain or MicroStation element.

12. Set *Element* to **Chain**.

13. From the drop down menu, select **CLCON**.

14. Type in **704+20** in *Station* and toggle **On** the *Station* lock. The designer can leave this un-checked to place the pull box dynamically.

15. For *Offset* check it **On** and enter **52.00**. The designer can leave this un-checked to place the pull box dynamically.

16. Leave *Dynamic Rotation* **Un-Checked**.

17. Click **Draw**.

18. Move the pointer to the *right side* of the road near the drive way and issue a **Data Point**. This will place the pull box in the design file.

19. Place a second pull box on the *opposite side* of the drive way using the same *Offset* at *Station* **704+68**. The figure below shows the two pull boxes next to the drive way.
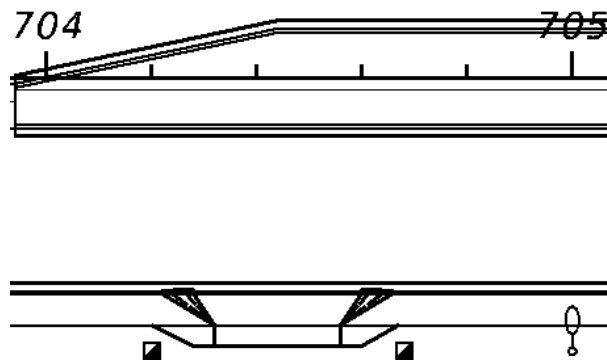
20. Finish placing pull boxes on *both sides* of any *side road or driveway* on *both sides* of the roadway on the **SR61** chain.

21. Close Draw Cell by Feature.

DESIGN SETTINGS

The custom line styles used to draw the Conduit lines in the next exercise can be adjusted so that the line style displays graphically pleasing. At times it may be necessary to adjust the line style scale to get the symbols built into the line style to display. To adjust the line style scale in D&C Manager the designer would go to Settings > Design this opens the Design Settings dialog.
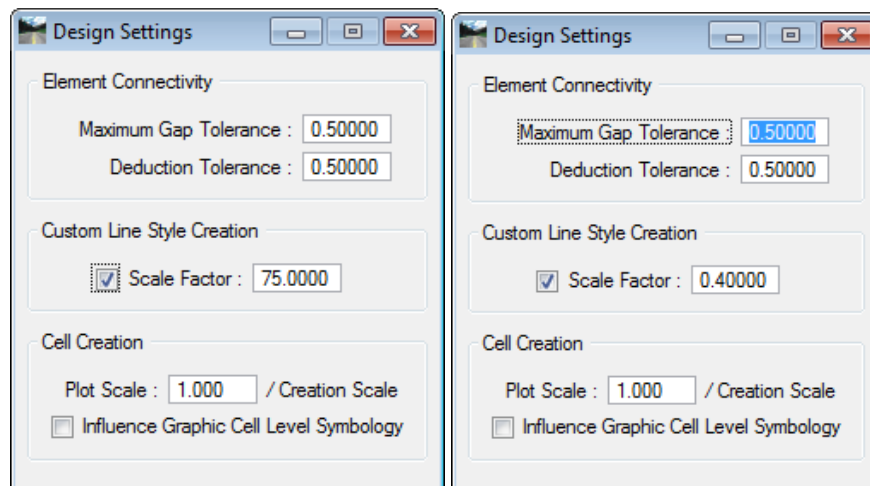
Plot Scale   *Annotation Scale*

In the center of the Design Settings dialog is the item for *Custom Line Style Creation*. This setting controls how a line style is displayed when drawn with D&C Manager. The designer should check on the *Scale Factor* and set the scale. In most cases the Plot Scale will work but in some it may be necessary to adjust this value as with the *Conduit* on a *100* scale plan sheet, a scale of *75.0* gives a better representation of the *Conduit* line style. Changing this value will not affect the quantities; however, it is very important that the designer communicate any changes to all design team members for consistency.

Exercise 4.5 *Draw Conduit under Ground between Light Poles*

In this exercise the designer will use D&C Manager to draw the conduit between the light poles and pull boxes.

1. Continuing in *Dsgnlt01.dgn*, from Task Navigator select **Civil Tools > Horizontal Geometry > Line Between Points** .

2. From the Line dialog, click the **Feature Definition** drop down menu.

3. Select **Traffic Plans > Signalization > Conduit > Conduit, Furnish & Install, Open Trench or Contractor's Options** for conduit placed from *Light Pole* to *Light Pole*.

    However, when crossing the *Roadway, Side Roads* or *Driveways*, use **Conduit, Furnish & Install, Underground or Underpavement - Directional Bore**.
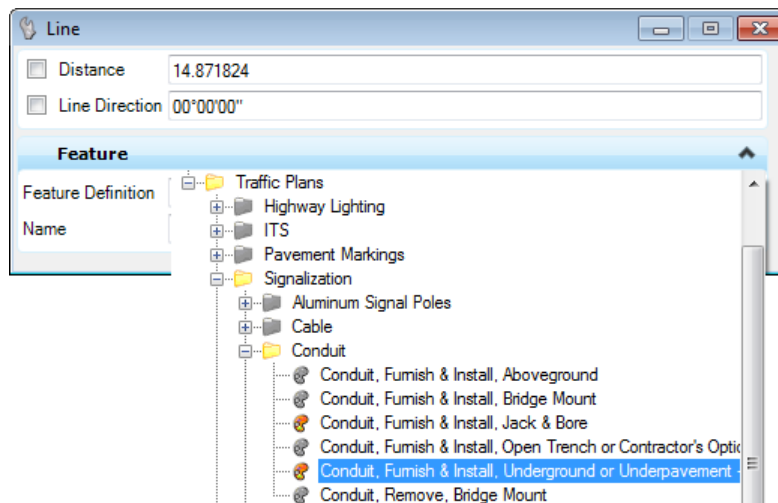
Note   This is equivalent to opening the D&C Manager and selecting the Signalization category > Conduit Furnish & Install > item 0630 2 13 Conduit, Furnish & Install, Underground or Underpavement - Directional Bore with Place Influence toggled on.

4. Zoom to *station* **701+00** the right side of the road.

5. Reference in the **dsgnsg01.dgn** file from the *signals* folder.

6. Draw a line from the **edge of the Pull Box** adjacent to *Pole 2* in the referenced *dsgnsg01.dgn* file to the **center of first Light Pole.**

7. Draw a line from one **Light Pole** to the next. Set the MicroStation *snap* to **Origin** to make this task easier.

8. Repeat this process until the last **Light Pole** is on the *right side* of the road. The figure below displays the *Conduit* passing through the *Light Poles* and the *Pull Boxes*.
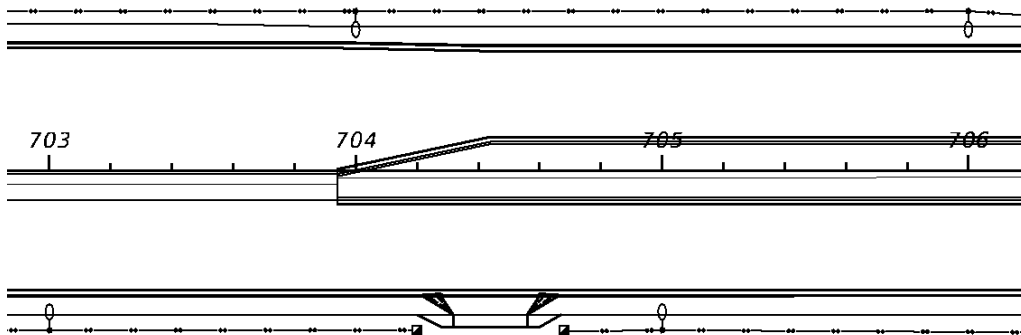
9. Reset to **Cancel** the *Place Line* command.

PLAN LABELING

GEOPAK delivers a tool, Plan View Labeler, to aid in the placement of plan labels. These labels can be simple callouts or complex blocks of text with stations and offsets built into them. It is possible to create custom labels for those pieces of text that are placed repeatedly into a style file similar to a library. The Department delivers style files with custom labels already created.

Labels created with labeler can be synced with D&C Manager for level symbology. This resolves issues with inconsistency between users. Labels can also be created at a scale of 1 to 1 so that they will work on any scale sheet.

Labels can be created with leader lines and arrow heads as part of the custom label. The arrow heads can be terminators delivered by the Department or the standard arrows built into the labeler. Again this makes for very consistent plan labeling if all users use the same labels.

Plan View Labeler can be launched from the Road tools palette.
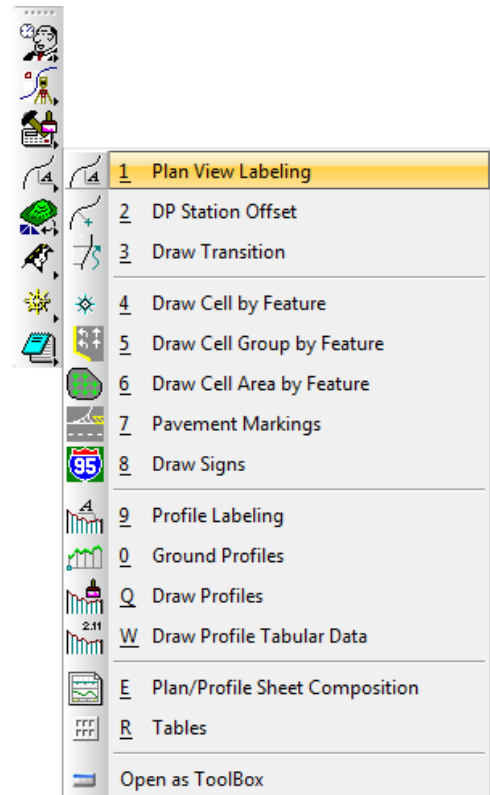
OR from the MicroStation menu Applications > GEOPAK ROAD > Plans Preparation > Plan View Labeling.

The general work flow of the Labeler is:

- Open Labeler.

- Open a Label Style File or *.lsf.*

- Select a Custom Label.

- Create a Custom Label.
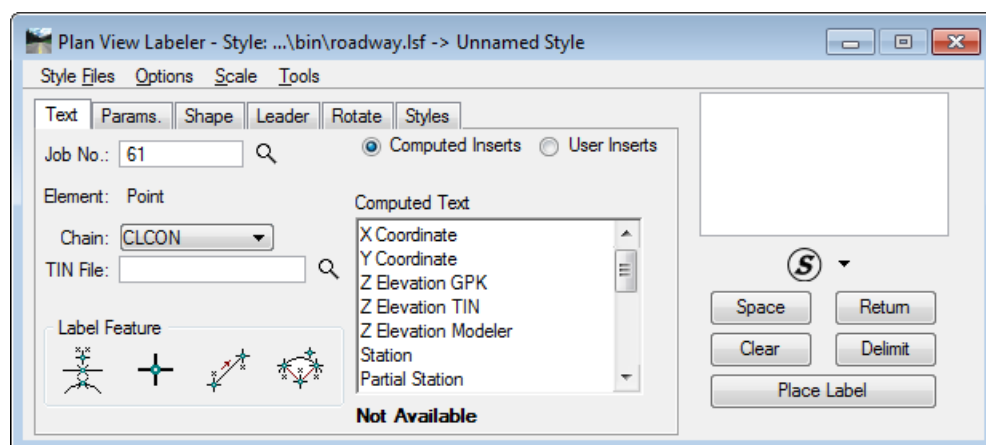
The Label Feature buttons on the Text tab are discussed next.
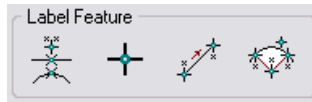
Select GEOPAK or MS Element – Prompts the designer to select the GEOPAK or MicroStation element. From this selection the designer can retrieve the geometry and place that information in MicroStation.

Data Point Location – Prompts the designer to snap to or issue a data point in the design file. Usually used when placing station and offsets.

2 GPK Point Line - Works with visualized points from COGO. When the points are selected, a display line is drawn between the points and a circle is placed at the midpoint. The bearing of that line is displayed and can be placed in MicroStation.

3 GPK Point Arc - Works with visualized points from COGO representing the P.C. and P.T. of the curve. When the points are selected, a display curve is drawn and the designer is prompted to enter the chord direction point, which is identified dynamically. Now any element of the curve's geometry can be displayed or placed in MicroStation.

Exercise 4.6 *Create a Custom Text Label*

➢ ***Create Custom Text Label for Light Poles (Part 1)***

In this exercise the designer will create a custom label for the Light Poles as shown in the next figure.

✓ *Open Project lsf File*

1. Continuing in *Dsgnlt01.dgn* in the *lighting* folder.

2. Open Project Manager.

3. Zoom to *station* **701+00**.

4. Rotate the view **by 2 point**; use the *station tick marks* **701+00** and **702+00** as the 2 points.

5. Open Plan View Labeler. Either from the Road tools palette or from the GEOPAK menu.
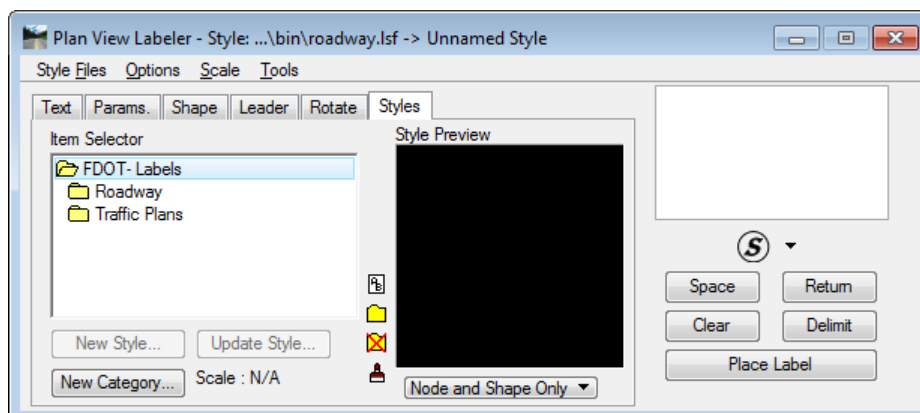
✓ *Styles Tab*

6. Select the **Styles** tab.

7. From the *Style Files* pull down menu, select **Open**.

8. Navigate to the Project *symb* folder and select **Roadway.lsf**.
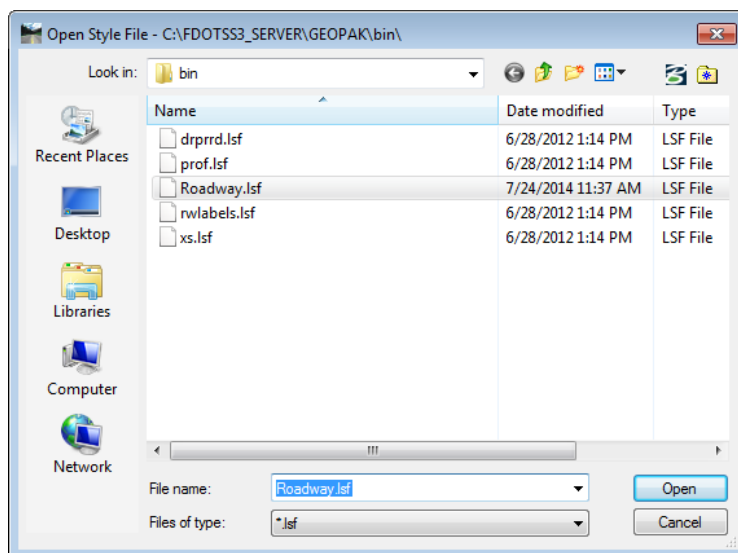
9. Click **Open**. This loads the *.lsf* file in the Project folder.

10. In the *Styles* tab, double-click on the **Traffic Plans** category. This expands the category and shows the standard labels delivered by FDOT.

✓ *Params Tab*

11. Continuing in Plan View Labeler, select the *Scale* pull down menu and select **Change Scale**. This opens Scale Style.

12. Set the *New Scale* to **1.00**. This allows the designer to create a custom label that can be used at any scale.

13. Click **OK**.

14. Open the *Params* tab. This is where the designer will set the *text size* and *symbology*.

15. Double click on the **Sample window** to access the Set Feature dialog.

16. Click on the **Paddle Lock** icon to lock it.

17. Set the text *Height* to **0.07**. The *Width* automatically matches the height.

18. Set the *Justification* to **Left Center**. Click on the nodes to adjust this.

19. Set the *Font t*o **FDOT Bold**.

20. Set the *Level* to **TextMisc**.

21. Set the *Color* and *Weight* to **ByLevel**.

22. Set the *text Line Spacing* to **0.07**.

23. Click **OK** on the Set Feature dialog.
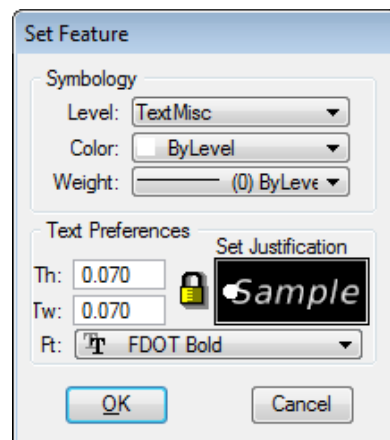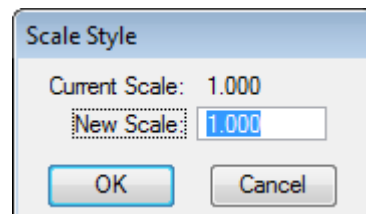
✓ *Shape Tab*

24. Open the **Shape** tab.

25. Select the first option: **No Shape**.

✓ *Leader Tab*

26. Open the **Leader** tab.

27. Set the *Leader type* to **One Point Leader**.

28. Set the *Terminator* to **No Terminator**.

29. Set the *Level* to **LeaderLine_dp**.

30. Set the *Color* and *Weight* to **ByLevel**.
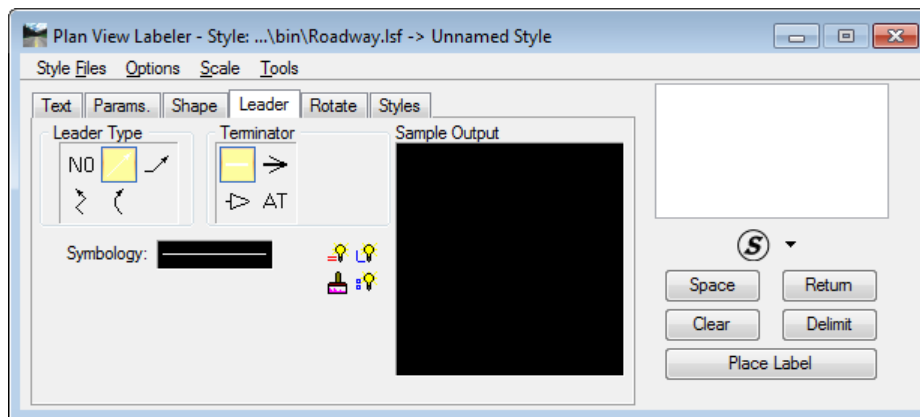
✓ *Text Tab*

31. Open the **Rotate** tab.

32. Set the *Current Angle* to **0.00**. This is not critical at this point as the designer can change this as the labels are being placed. Setting this now will make the preview look better.

✓ *Text Tab*

33. Open the **Text** tab.

34. Make sure the *Job No. id* set to **61**. Use of Project Manager will insure this.

35. Set the *Chain* drop down menu to **CLCON**.

36. In the *Key-In window* of the *Text* tab, enter **STA**.

37. Click the **Space** button. This is located under the Key-In window.
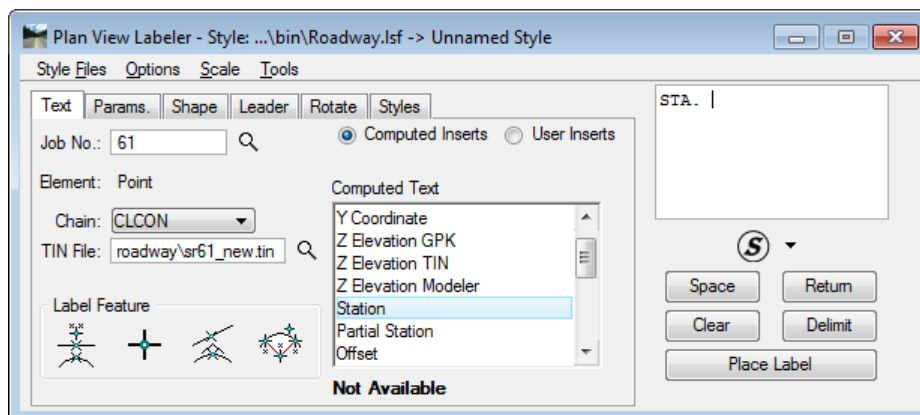
38. In *Label Feature* select **Data Point Location**. This is the second icon.

39. Set the *snap* option to **Origin** and snap to the first light pole on the left side of the road.

40. In *Computed Text* select **Station**. Do Not double-click on *Station*.

41. Set the *Station rounding* to **0**.

   ✓ *Complete the Label*

42. Double-click on **Station**. This pushes the station text over to the Key-In window.
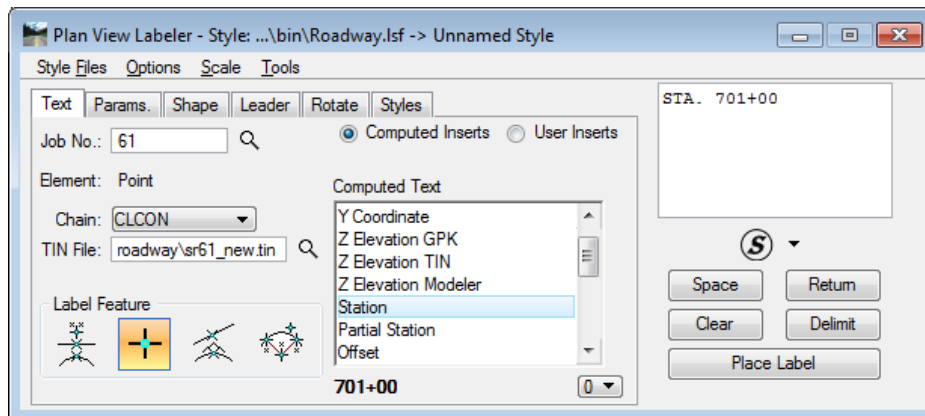
43. Click the **Delimit** button. This is under the Key-In window. This will automatically draw the leader line between to two lines of text in the label.

> **Hint**   Very important that the user use the Space button when building a label, not the space bar on the key board.

44. In the *Key-In* window, enter **POLE**.

45. Click the **Space** button and enter **NO**.

46. Click the **Space** button and enter **X.** This is a place holder to be edited at the time label is placed.

47. Click the **Space** button and enter **CKT**.

48. Click the **Space** button and enter X-X. This is a place holder to be edited at the time label is placed.
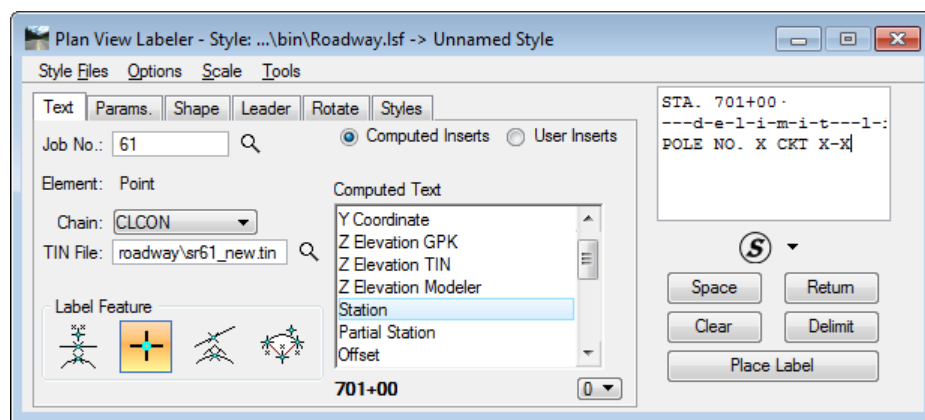
✓ *Place the Label*

49. Zoom in very close to the *center of the light pole*. This is necessary because the label being placed is at a scale of 1: 1. This is a temporary label.

50. Click the **Place Label** button.

51. Move the cursor *near the light pole* and issue a **Data Point** to place it.

52. Move the cursor *left and right along the label*, notice how the hinge point changes as the cursor gets close to the ends of the label.

53. Issue a **Data Point** to connect the leader line to the light pole and complete the label placement.

✓ *Save New Custom Label to the .lsf File.*

54. Open the **Styles** tab.

55. Select the **Traffic Plans** category.

56. Click **New Style**. This opens Create Style. The designer must place the temporary label before this option is available.

57. Enter the **Style Name Light Pole**.

58. Leave the *Style Scale* to **1.0**.

59. Click **OK**. This adds the new style to the *Item Selector* list.

60. Click the **Styles Files** pull down menu and select **Save**.

61. In MicroStation delete the label just placed; this was only temporary.
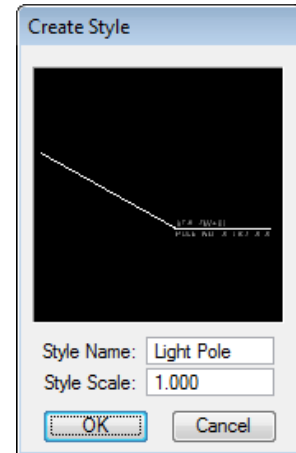
➢ **Place Custom Text Label (Part 2)**

In this exercise the designer will place the new custom label on the light poles.

✓ *Place Labels*

1. Click the **Clear** button. This is located under the Key In window.

2. In the *Item Selector*, select the new style **Light Pole**.

3. Double-click on the *style* or click the **Blue** check mark icon.

4. On the *Scale* pull down menu, select **Change Scale**.

5. Change the *Scale* to **50.00**.

6. Click **OK**.

7. Zoom out so the label will fit in the view.

8. In the *Key In window*, edit the *Pole Number* to be **POLE NO. 1**.

9. Edit the *Circuit* to be **CKT. A-1**.

10. The *Station* is still set from when the label was originally placed.

11. Click **Place Label**.

12. Move the cursor to the *right and below the light pole* and issue a **Data Point**.

13. Move the cursor to the *left of the label* and issue a **Data Point**. This completes the label placement.

✓ *Label Remaining Poles*

1. Pan to the second light pole on the right side of the road.

2. Open the **Text** tab.

3. Click the **Data Point Location** button.

4. Snap to the **Origin** of the second light pole. Notice the station dynamically changes.

5. In the *Key In window*, edit the *Pole Number* to be **POLE NO. 2**.

6. Edit the *Circuit* to be **CKT. A-2**.

7. Click **Place Label**.

8. Move the cursor to the *left and below the light pole* and issue a **Data Point**.

9. Move the cursor to the *right of the label* and issue a **Data Point**.

$$STA.\ 702+00$$
$$POLE\ NO.\ 2\ CKT\ A-2$$

10. Repeat this process on the remaining light poles.

11. Close Plan View Labeler.

CREATING POLE DATA AND LEGEND SHEET

The *Pole Data* and *Legend Sheet* contain details and notes pertaining to pole placement and construction. This sheet provides a listing of each pole by pole number. The following information is also provided for each pole:

- Circuit Number

- Roadway Station and Offset

- Arm Length

- Luminaire Wattage

- Mounting Height

- Pay Item Number

FDOT Menu provides a tool for the creation and placement of this sheet. The process to develop this sheet is:

1. Create the *Pldtlt01.dgn* file. This is the standard MicroStation file for the Pole Data and Legend Sheet.

2. From FDOT Menu, use LDM to *place* the Pole Data and Legend Summary Box, *edit* excel spread sheet data and *update* the excel data into the Pole Data and Legend Sheet.

Exercise 4.7 *Placing the Pole Data and Legend Sheet*

In this exercise the designer will create the Pole Data and Legend Sheet design file and place the standard sheet and data into the file.

➢ ***Create the MicroStation Design File***

1. From the FDOT Menu, select **Actions > Create/Edit File**.
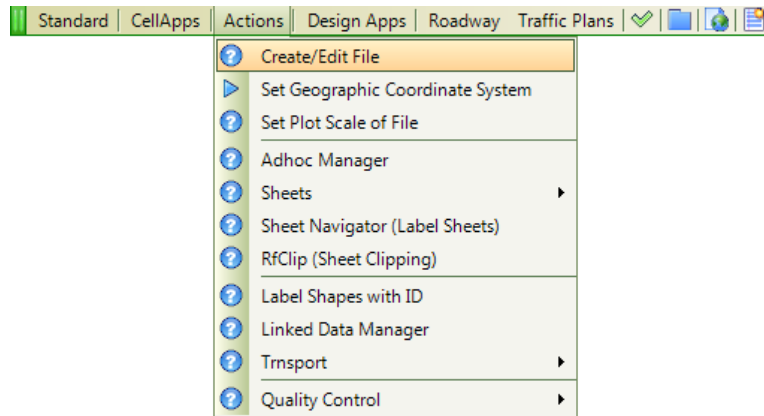
2. From *File Type*, select **Pole Data Sheets**. This populates the *Output File* name.

3. Click **Create**.

4. Click **OK** to acknowledge the file creation.

5. Click **Open DGN**.

6. Click **OK** on the Create File/Project dialog.

7. Click **OK** on the Set/Update Plot Scale dialog. The default scale of 50 is fine for this sheet.

➢ ***Place Pole Data Sheet***

1. From FDOT Menu, select **Actions > Sheets > Plan Sheet**.
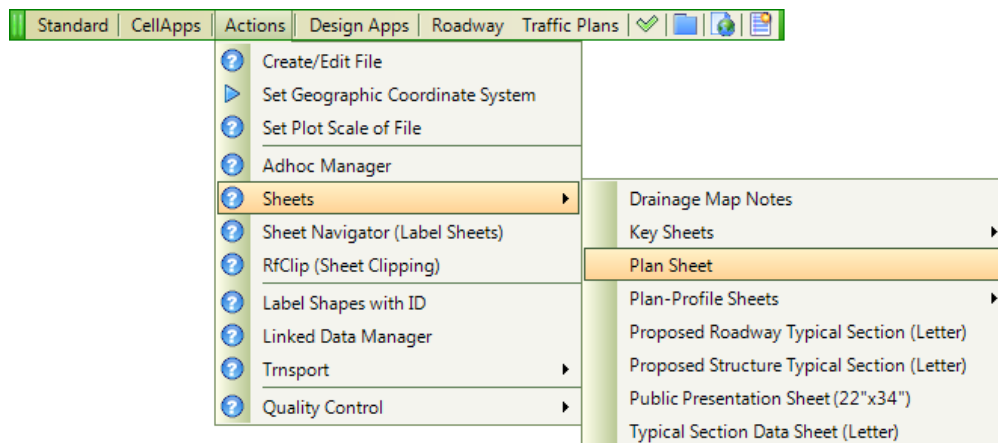
2. Issue a **Data Point** in the drawing to place the sheet.

3. Perform a **Fit View**.

4. From FDOT Menu, select **Actions > Linked Data Manager**.

5. Click the **Create New Link** button.
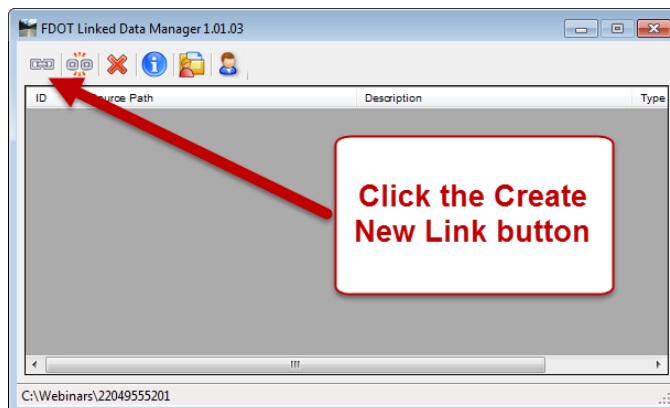
6. Click **Create New from Template** on the Link Information dialog.
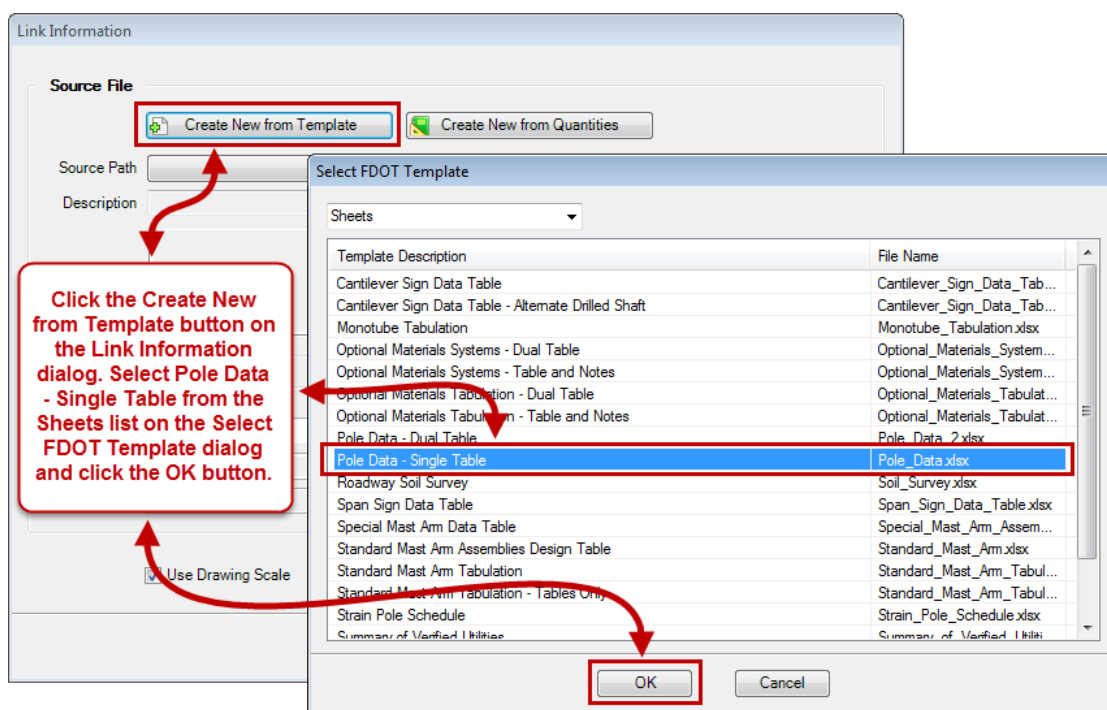
7. Select **Pole Data – Single Table** from the list of templates on the Select FDOT Template dialog.

8. Click the **OK** button to continue.

9. Save the **Pole_Data.xlsx** file to the *lighting* folder.
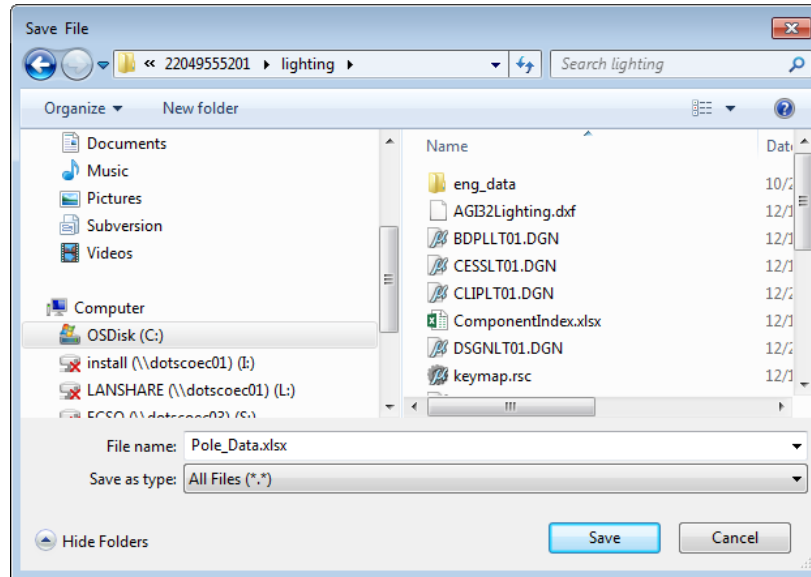
10. Set the Link Information dialog as shown below:
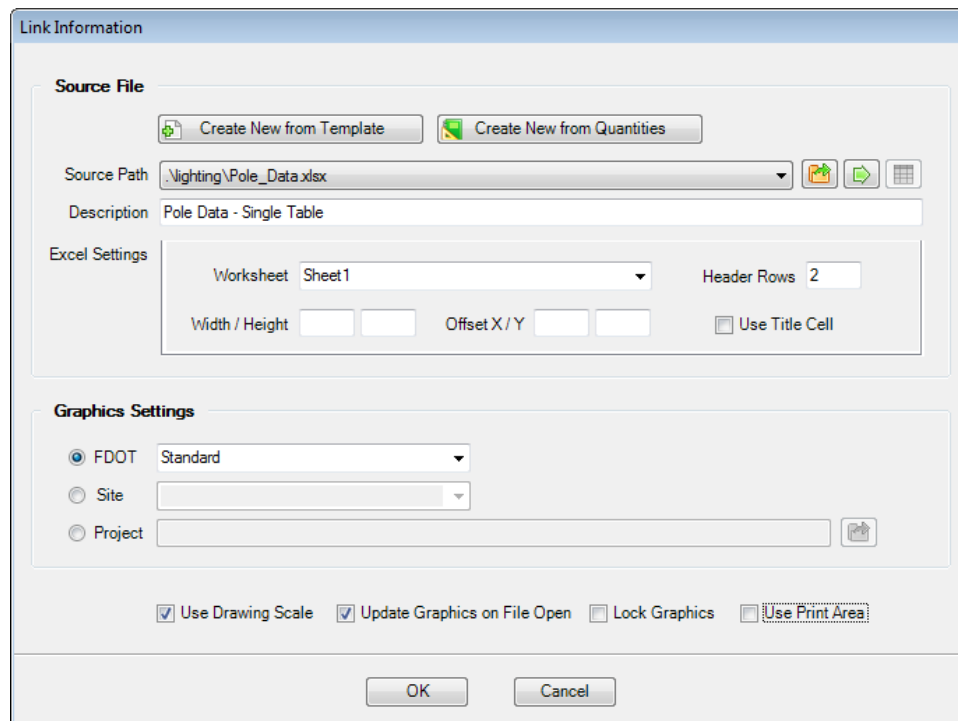
11. Click the **Green Arrow** button next to the *Source Path* drop down list to open the **Pole Data Spread Sheet**.

12. Populate the excel spread sheet with the following data and save the excel spread sheet.

Pole No. (s):	**1 thru 18**
Circuit:	**A-1 thru A-18**
Station:	**701+00.00** thru **719+00.00** *(Station 715+00.00 has no Light Pole.)*
Dist.or Arm:	**10'**
Luminaire Wattage:	**250**
Mounting Height:	**40'**
Pole Setback:	**15'**
PayItem:	**0715 4111**

13. Click the **Ok** to place the *Pole Data Summary Box* on the Plan Sheet placed earlier.

14. Close the Excel file.

15. From FDOT Menu select **Cellapps > FDOT Cell Libraries** and select the **Roadway.cel** *Library* from the Cell Libraries dialog.

16. Scroll to find the **SHPoleDataLegend** cell.

17. Snap to the *upper left corner* of the sheet cell to place the legend.

> *Hint*   If the data needs to be edited and replaced, use LDM to update the link.

Note   Notice in the MicroStation file under the Legend that there are several data fields that should be filled using the MicroStation Fill in Single-Enter Data Field tool.

➢ ***Fill in Title Block***

1. From FDOT Menu, open Sheet Navigator.

2. Populate Sheet Navigator as shown below.

3. Click the **Save Sheet** button. This fills the title block and adds the digital signature note for electronic delivery.

4. Close Sheet Navigator.

5 QUANTITIES AND REPORTS

OBJECTIVE

In this chapter the following topics will be covered:

- Defining Adhocs for Quantities

- Import Project Properties from TRNS*PORT

- Generate Quantities and Export to Quantity Manager

- Generate CSV File in Quantity Manager

- Export Quantities for TRNS*PORT

- Place Quantities on Tabulation of Quantities Sheet using Linked Data Manager (LDM)

INTRODUCTION

This section covers applications used to produce quantities and then place the quantities on a plan sheet or computation book. Being able to produce quantities based on your design elements should be your ultimate goal. Producing automated quantities shows the individuals knowledge of the software and best design practices. Team work is crucial to achieving this goal, if everyone in the design squad is not following the same procedures, as it applies to CADD standards, producing automated quantities will be much more difficult.

GENERATE QUANTITIES

Quantities are generated from the elements designed and drawn in MicroStation using Design and Computation (D&C) Manager. All of the elements drawn in the previous exercises are now able to be automatically quantified because D&C Manager was used to draw the elements.

D&C Manager has the ability to generate individual sheet quantities or the entire projects quantities at one time. D&C Manager can export to a CSV file that can be imported into the Tabulation of Quantities file or it can export to a data base that can then be opened with Quantity Manager.

Quantity Manager is a standalone program that can organize and manipulate the data created by D&C Manager and quantities generated manually then create reports which can be used to produce an electronic comp book. Quantity Manager can also export the quantities to TRNS*PORT, which takes away the need to have a designer to manually enter all of the project pay items. This task alone can save hours over the lifespan of a project, not to mention removing the possibility of human error. Also, Quantity Manager can export to a CSV file which can then be used to create the Tabulation of Quantities sheet.

Lighting Plans quantities are considered sheet quantities, which means there is a Tabulation of Quantities sheet in the plan set that breaks down the quantities per sheet.

> *Note*   When calculating quantities with D&C Manager, Locate must be OFF in all attached Reference files. Otherwise, GEOPAK will crash.

QUANTITIES TO TRNS*PORT WORKFLOW

1. Generate quantities using **D&C Manager** then export to a database file.

2. Export the project properties *(.xml)* file from TRNS*PORT. This can be done any time after the notice to proceed has been executed. At this time the designer establishes the funding rule in **Quantity Manager**, this is really the item category. Example 0400 is the category for Signal plans. This funding rule associates the items in **Quantity Manager** to a category in TRNS*PORT.

3. Open the project in **Quantity Manager**, this is the database file, usually Access database *(mdb)* format.

4. Import the project properties into **Quantity Manager**. This is the *(.xml)* file from TRNS*PORT.

5. Apply the funding rule to the appropriate items.

6. Export the Quantities from **Quantity Manager** to a format TRNS*PORT can read. This is also an *(.xml)* file. Name it using the *11 digit FIN number.xml.*

7. Upload the quantities to TRNS*PORT. This alleviates the need for a designer to manually enter pay items and quantities.

8. Export the quantities from **Quantity Manager** into a *(.csv)* file to be used on the Tabulation of Quantities Sheet.

D&C MANAGER QUANTITIES
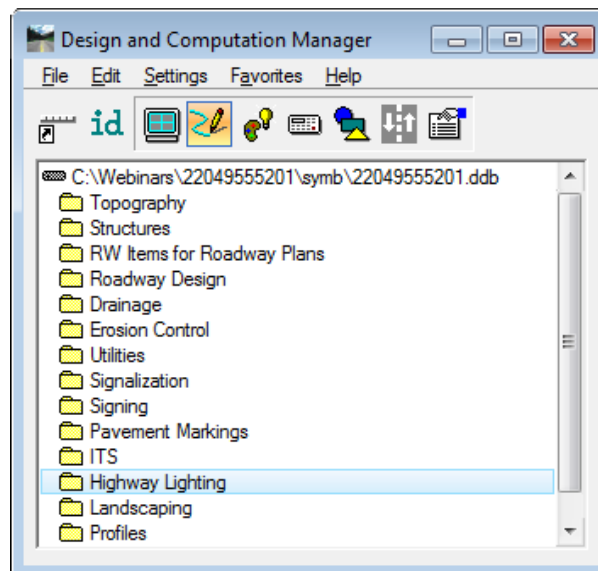
D&C Manager expands to add a list box at the bottom of the dialog, when the Compute option is selected. This is the collection bin, where the designer adds the items from **D&C Manager** to be quantified. Double click on the item to load the items into the collection bin. To clear the collection bin, right click in the collection bin area and select clear collection. Once all of the items are added to the collection bin they can be saved for future use. To save a collection, right click in the collection bin area and select **Save Collection**. Save the file to the project.

The figure below displays the collection bin populated with items ready for quantity calculation. The items will not be in numerical order when loaded into the collection bin; they are displayed in the order that they are added.
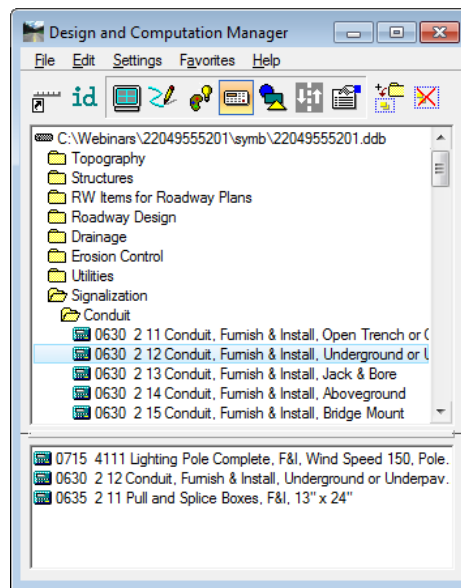
> ➢ *To Open and Organize the Collection:*

1. In D&C Manager, select **Favorites > Organize Favorites**. This opens My Favorites.

2. In My Favorites, click **Open**.
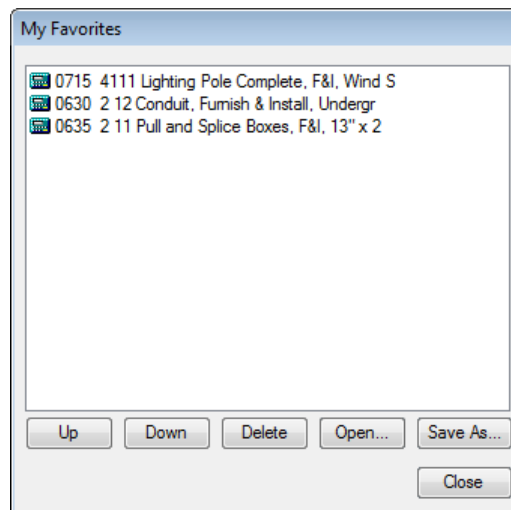
3. Browse to the folder where the designer saves the collection and select the file. The collection file will have a *(.col)* extension.

4. Use the **Up** and **Down** buttons to organize the items.

5. Click **Save As** to save the changes.

Note   The items can also be organized in the output from D&C Manager or in Quantity Manager prior to going onto sheets.

REVIEW ITEMS PRIOR TO COMPUTING QUANTITIES

Prior to computing any quantities it is important to review the items that are about to be computed. D&C

Manager's *Display* tool will aid in this task. To activate the *Display* tool, click on the [icon] icon.

- [icon] **Normal Display** – the items of interest change back to their original element symbology.

- [icon] **Highlight Selection** – the desired elements are displayed in the current MicroStation highlight color, while any other elements remain unchanged.

- [icon] **Hide Selection** – the desired items are not displayed in the view, however all other elements are displayed. This is the reverse of the **Display Only Selection** tool.

- [icon] **Display Only Selection** – only the items in the collection box appear, while the rest are not displayed in the view.

Using the *Highlight Selection* the designer would zoom in close to the project elements and make sure that whatever is placed in the collection bin for computing quantities highlights.

> *Note*   Always set the option back to Normal Display before closing the tool.

COMPUTING QUANTITIES

Plan Quantity Computation opens when the Compute is selected.
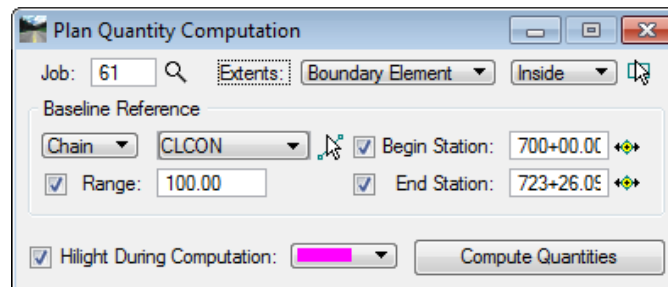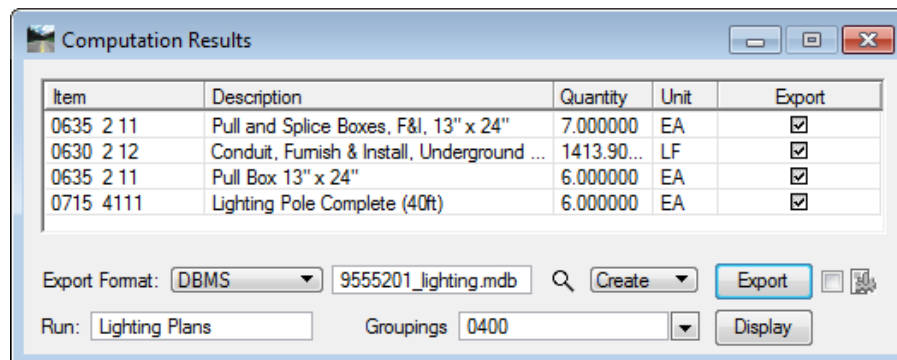
- **Job** is the GEOPAK *(.gpk)* file, if the designer is using Project Manager than this will be set.

- **Extents** are a list of options for limiting the area and elements included in the quantity calculations. The next figure shows the options available.
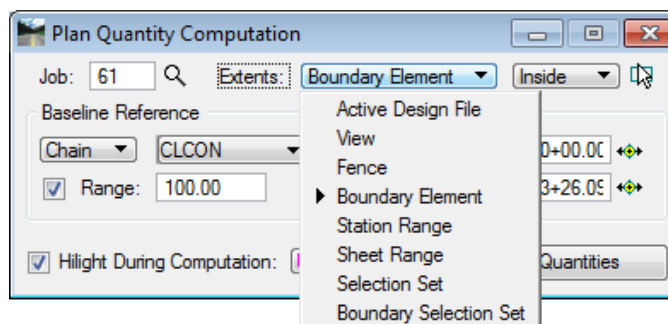
- o   *Active Design File* - All elements in the Active Design File are candidates for computation.
- o   *View* - Only the selected items that are displayed in MicroStation view one are computed. If the view includes area outside of the *Range,* the *Range* will override.

- o *Fence* - A MicroStation fence must be placed, and all specified graphical features which satisfy both the fence and the *Range* will be tabulated.
- o *Boundary Element* - A previously drawn closed shape is used to determine quantities. When selected, the designer is prompted to select the closed shape. This mode is sensitive to the MicroStation Inside, Overlap and Clip modes.
- o *Station Range* - A Baseline Reference (chain or dgn) must be defined for this option. The range fields default to the extent of the chain. The designer can key in a station or by clicking Begin or End Station icons graphically define the station range. Perpendicular projections to the station(s) define the extent of the computations. This option is not ideal for sheet quantities.
- o *Sheet Range* - Sheets placed with the Plan Profile Sheet Composition tool can be utilized. Select the option which displays the sheet range. This option will work in our scenario.
- o *Selection Set* - A MicroStation selection set of the candidate compute elements must be created prior to computing.
- o *Boundary Selection Set* - A MicroStation selection set of the Boundary (not the actual candidate compute elements) must be created prior to computing. This option will work in our scenario. When this option is used it uses the element ID number in Quantity Manager to organize the data for each individual sheet.

- **Baseline Reference** allows the designer to define a Chain or DGN element for offsets and station values for reports. In addition the *Range*, if set, is measured from the selected Chain or DGN element to search for candidate items. Setting the *Baseline Reference* to *None* limits the type of output that can be generated as no station / offset values can be computed.

- When using **Sheet Range** or **Boundary Elements** it is not necessary to toggle on Begin and End Station limits.

- **Highlight During Computation** when toggled on, all MicroStation elements computed is highlighted in the selected highlight color.

- **Compute Quantities** starts the computation process and when completed opens the Computation Results dialog.

Item	Description	Quantity	Unit	Export
0635 2 11	Pull and Splice Boxes, F&I, 13" x 24"	7.000000	EA	☑
0630 2 12	Conduit, Furnish & Install, Underground ...	1413.90...	LF	☑
0635 2 11	Pull Box 13" x 24"	6.000000	EA	☑
0715 4111	Lighting Pole Complete (40ft)	6.000000	EA	☑

Export Format: DBMS ▼ 9555201_lighting.mdb 🔍 Create ▼ Export ☐ 📇

Run: Lighting Plans   Groupings 0400 ▼ Display

Note   It is imperative that the designer review these results before going any further. If there are any errors or omissions now is the time to fix or add them.

- o *Export Format* - sets the type of output the designer wants to generate from the reported quantities. There are several formats to choose from:

  Comp Book – A more detailed report that lists not only quantity summaries, but also geometric properties such as plan view coordinates and station/offsets for located elements. File is in ASCII format.

  Item Report – A Quantities Summary listing pay items, descriptions, units and total quantities for located elements. File is in ASCII format.

  Item Table – Contains the same information as the Item Report, but formatted in tabular form.

  DBMS – Very detailed information including calculated and rounded quantities, geometric properties, pay item numbers, descriptions, station / offset values, etc. The format is the selected database (i.e., Microsoft Access, Oracle, SQL Server, and dbase is set in the Compute Settings tool). This option is required when taking the quantities into Quantity Manager. This is the option used in this training guide.

  CSV By Item –Summary listing pay items, descriptions, units and total quantities for located   elements. Format is CSV (coma separated values).

  CSV By Element – A more detailed report that lists not only quantity summaries, but also geometric properties such as plan view coordinates and station/offsets for located elements. Format is CSV (coma separated values).

  Table – User defined table column properties. A column for designating the appropriate symbol is also included, as well as Adhoc attribute data. This option is used to generate legends such as for landscape plans.

- o Next to the *Export Format* is a field for entering the file name of the output file; using the hourglass allows the designer to browse to a specific folder.

- o When exporting the quantities there are two options to choose from: *Create* or *Append*. This will place the quantities into a new file or append them to a previously created file.

- o *Run* and *Phase* are only used for the DBMS export format to Quantity Manager. The **Run** is a user key in and any logical description, like Lighting Plans or Preliminary, can be used. This description will be passed to Quantity Manager. There is several default Phases included in the drop down menu, however, the designer is only concerned with Design Estimate. This is the recommended Phase when exporting to Quantity Manager, as this is the only phase that can be passed to TRNS*PORT.

- o *Export* commences the output file process and creates the quantity file. In this training the designer will be exporting to DBMS which will create an MDB file that can then be opened with Quantity Manager.

Exercise 5.1 *Create and Save a Collection of Favorites*

In this part of the exercise the student will create a collection of items used to generate quantities and save the collection as a favorite to be used with future processes.

1. If closed, open **Dsgnlt01.dgn** in the *Lighting* folder.

2. Open D&C Manager.

3. Zoom in near *Sta* **535+00**.

4. Click on the **Compute** icon in D&C Manager. This is the calculator icon.

5. If there are items in the *collection bin* of D&C Manager, right-click in the collection area and select **Clear Collection**.

6. Use the **ID** tool in D&C Manager to identify and add lighting items to the collection. Once all of the items are identified Save the collection.

7. Click the **ID** icon in D&C Manager.

8. Select one of the **Light Poles** and accept it to move to the item **0715 4111 Lighting Pole Complete, F&I, Wind Speed 150, Pole Height 40'**.

9. Right-click on the item and select **Add to Collection**. This places item in collection bin.

10. Click the **ID** icon, select **Conduit** line and accept.

11. Right-click on the item and select **Add to Collection**.

12. Repeat this process to add the **Pull Boxes** at the driveway.

13. Right-click in the *Collection Bin* and select **Save Collection As**. This opens Save Collection to a File. The location will default to the working directory.
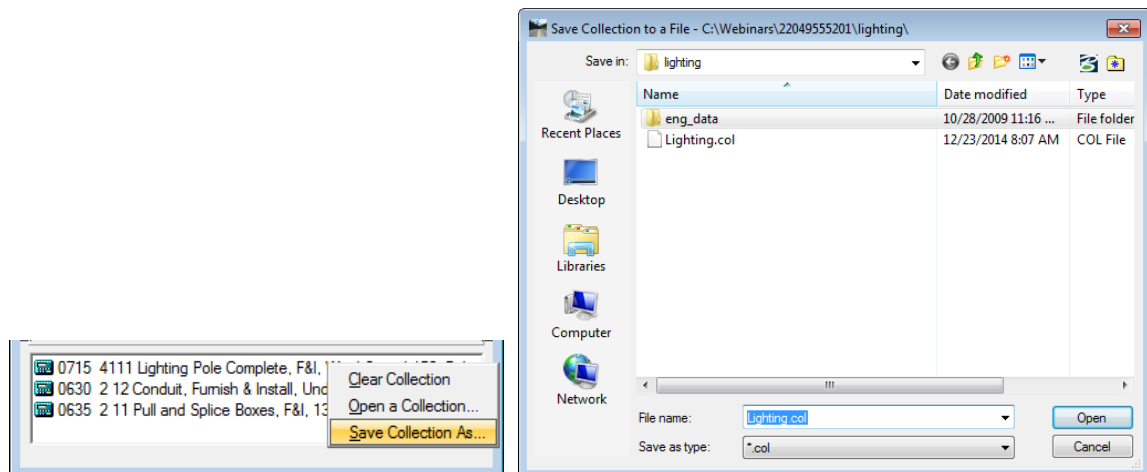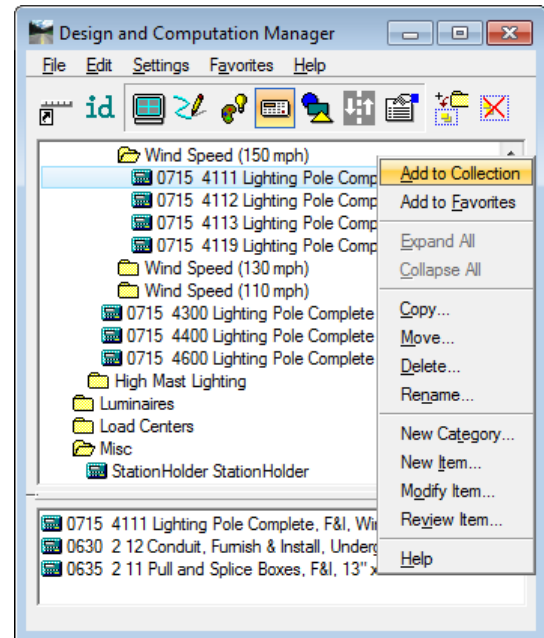
14. Enter a *file name* for the collection, example: **Lighting**. The extension *(.col)* is automatically added.
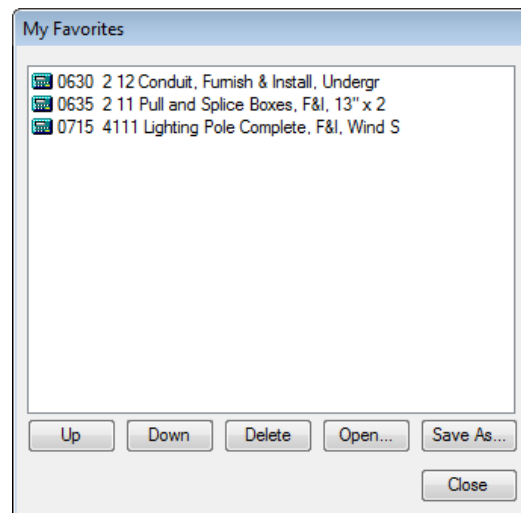
15. Click **OK**.

Exercise 5.2 *Organize the Collection*

Organizing the collection is only to make it easier on the designer when working with a group of items as they will now be in numerical order. Organizing the collection will have no impact on creating the Tabulation of Quantities Sheet; this is organized in Quantity Manager.

1. In D&C Manager, select **Favorites > Organize Favorites**. This opens My Favorites.
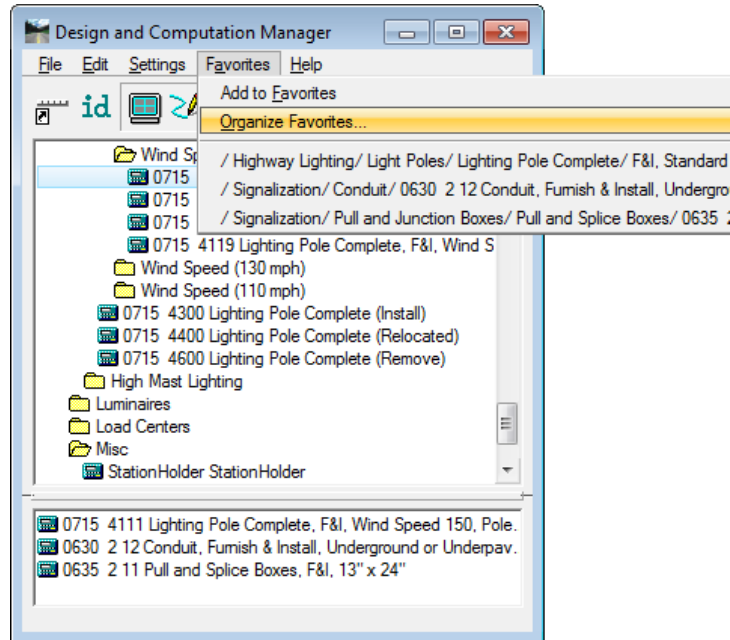
2. In My Favorites click **Open**.

3. Navigate to the *lighting* folder and select **Lighting.col**. This loads the D&C Manager items into My Favorites.

4. Using the **Up** and **Down** buttons, put the items in numerical order from *lowest to highest.*

5. Click **Save As**.

6. Select the **Lighting** file and click **OK** to overwrite.

7. Click **OK** on the Alert dialog warning you that the file already exists.

8. Close My Favorites.

9. Right-click in the *Collection Bin* and select **Open a Collection**.

10. Select the **Lighting** collection in the *Lighting* folder.

11. Click **OK** to open the file. This will load the items in the collection in numerical order.

Exercise 5.3 *Review the Quantity Items Using Display Tool*

1. Continuing in *Dsgnlt01.dgn***,** click the **Display** tool on D&C Manager.


2. On the Display tool, click the **Highlight** icon. This will highlight all of the items in the collection.

3. Take a moment to review the design file and experiment with the other display options.

4. Set the *display* option to **Normal**.

5. Close the Display.

Exercise 5.4 *Generate Quantities and Export to Quantity Manager*

➢ ***Compute quantities***

1. Continuing in *Dsgnlt01.dgn***,** zoom out to see the first four clip borders.

2. On D&C Manager, click the **Compute** icon. This opens Plan Quantity Computation.

3. The *Job* number is **61**. This will be set using Project Manager.

4. Set the *Extents* to **Boundary Selection**.

5. Set *Method* to **Clip**.

6. Set the *Baseline Reference* to **Chain**.

7. Select the *Chain* to **CLCON**.

8. Toggle **On** *Range*.

9. Set the *Range Value* to **300.00**. This is distance from the chain for items to compute.

10. Toggle **On** *Highlight During Computation*.

11. In MicroStation, use **Power Selector** to select the 3 clip borders.
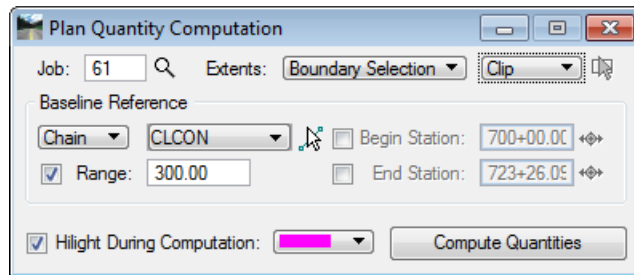
12. Click **Compute Quantities**. This opens Computation Results.
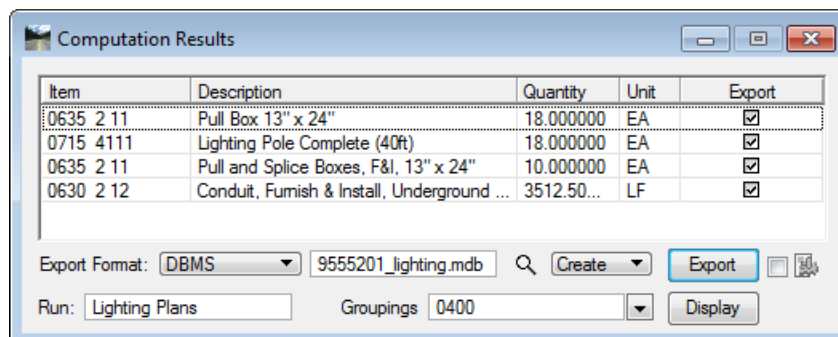
➢ **Export Quantities to DBMS for Quantity Manager**

1. Set the *Export Format* to **DBMS**. This is the required format for Quantity Manager.
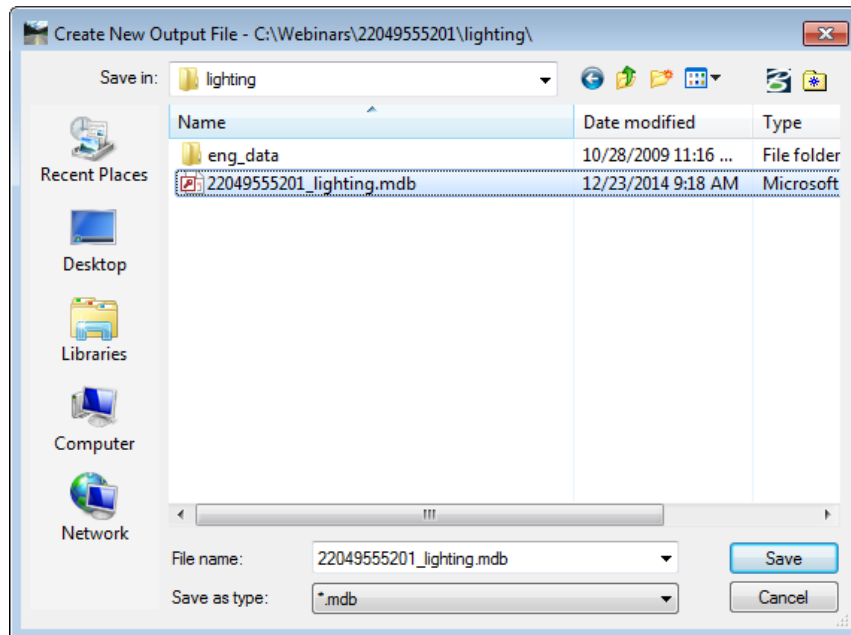
2. Using the *magnifying glass* icon, browse to the **lighting** folder.

3. Enter a *file name*; the Department recommends using the **fin number**. The *(.mdb)* file extension is automatically added.

4. Click **OK**.

5. Set to **Create**. The options are *Create* or *Append*.

6. For *Run* enter **Lighting Plans**.

7. For *Groupings* select or type in **0400**. This option must be selected.
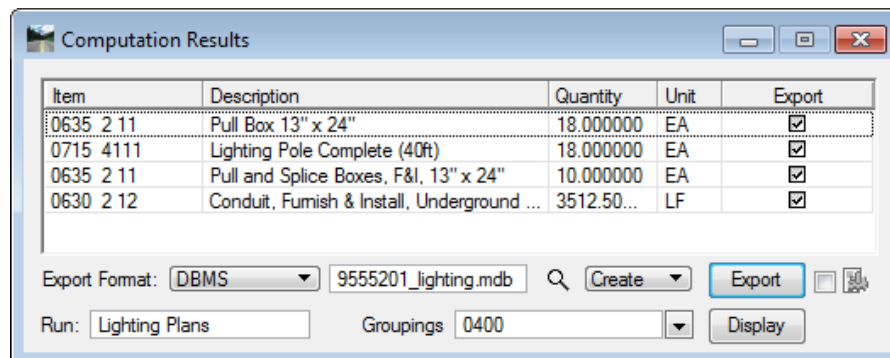
8. Click **Export**. This creates the *(.mdb)* file.

9. Close Computation Results. Open Windows Explorer and confirm the file was created.

10. From FDOT Menu, select **Standards > Explore Current Working Directory**.

11. The *(.mdb)* file displays in the **lighting** folder.

12. Close Windows Explorer.

QUANTITY MANAGER OVERVIEW

As mentioned earlier in the training guide, Quantity Manager is a stand-alone program that organizes and manages the quantities generated from D&C Manager. Quantity Manager can also be used to enter non graphic quantities such as Mobilization. Once the information is loaded into Quantity Manager, the designer can then export the data from Quantity Manager into a spread sheet that can then be placed in CAD on the Tabulation of Quantities sheet using Linked Data Manager. Quantity Manager also creates the Computation Book and can export the quantities to an xml file to be uploaded into TRNS*PORT.

Quantity Manager can be loaded from the Road Tools Palette or from the Applications pull down menu.
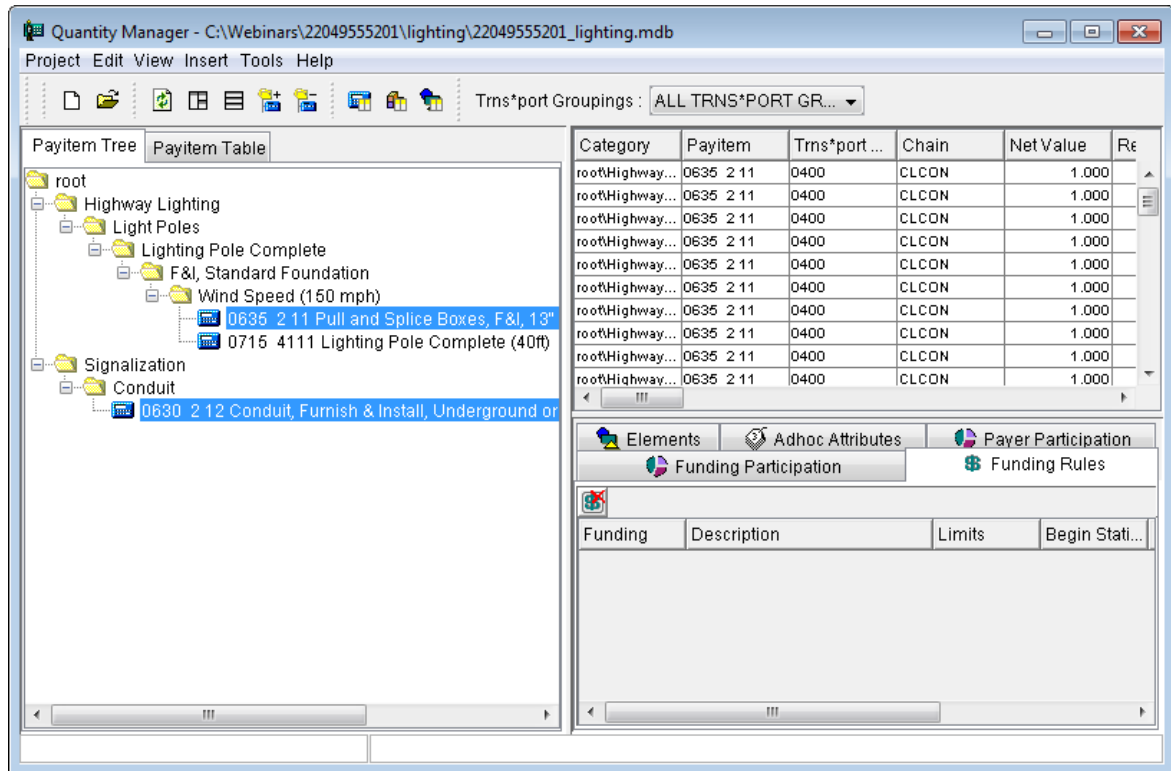
QUANTITY MANAGER WORKFLOW

1. Open Quantity Manager.
2. Open the Project created by D&C Manager. This is the *(.mdb)* file created in D&C Manager.
3. Import the Project Properties *(.xml)* file. This is created in TRNS*PORT then exported to an *(.xml)* file.
4. Export quantities to *(.xml)* file to be loaded into TRNS*PORT.
5. Create *(.csv)* file, using delivered styles from FDOT, to be used to create Tabulation of Quantities sheet.

Note   The Department offers a full training course on Quantity Manager that most users have already attended. In this training guide you will only use some basic functions of Quantity Manager.

Exercise 5.5 *Import Project Properties from TRNS*PORT*

➢ *Open Quantity Manager and Import Project Properties*

1. Continuing in *Dsgnlt01.dgn*. Quantity Manger can be opened in any design file.

2. From the Road tools palette, select Quantity Manager. The icon is located under the Project Manager icon in the same menu as D&C Manager.
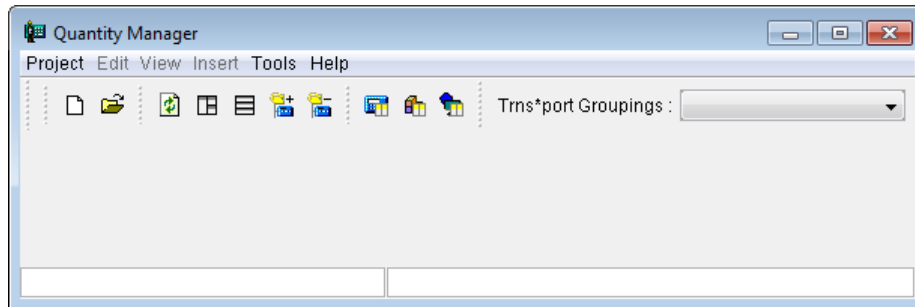
3. In Quantity Manager, select **Project > Open**. This opens Connect to Database.

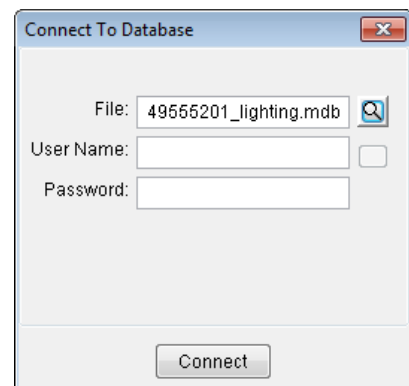4. Set the *Database* to **MS Access 2000**.

5. For the *File*, click the *magnifying glass* icon.
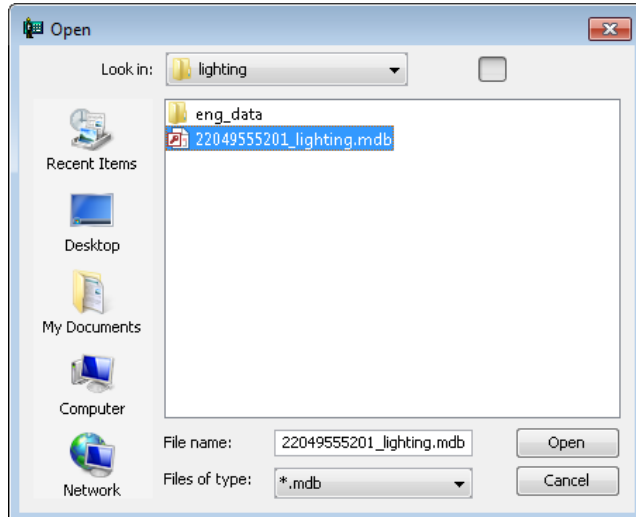
6. Browse to the *lighting* folder and select the project *(.mdb)* file. In this exercise select **22049555201_lighting.mdb**. This is the file created in D&C Manager.

Note   If your discipline is part of the Roadway set of plans you will be required to make sure that all of your MicroStation elements are drawn with D&C Manger and check the quantities but not create a database. The Roadway user will typically create one project database of all quantities. In this example the Lighting plans are stand alone.

7. Click **Open**. This displays Connect To Database.

8. Click **Connect**. This loads the project database into Quantity Manager.

> ### *Load Project Properties from TRNS*PORT*

1. Continuing in Quantity Manager, select **Project > Properties**. This opens Project Properties.

2. In **Project Properties**, select **Project > Import aecXML**.

3. Browse to the project *data* folder and select **22049555201_PI.xml** (PI for Project Information). This is the file exported from TRNS*PORT.

> *Note*   The 22049555201 Project Information XML file is the aecXML Export from the Designer Interface. This file is not the same as the 22049555201_LT.xml file. The _LT file is a Summary of Payitem Report exported from the Designer Interface whereas the _PI file contains information such as TRNS*PORT Groupings.

4. Click **Open**. This loads the project properties. As seen in the figure below notice that the *General Information* is now filled in.

5. Select the **0400 Lighting Grouping** from the *Trns*port Groupings* list.

6. For the *Location Information*, select the drop down menu then select the chain **CLCONST**. This will populate the *Begin* and *End Station*.

7. For the *Begin Mile Post* enter **0.000.**

8. For *End Mile Post* enter **.441** .

9. For Midpoint Coordinates set Latitude to **59 58 28**.

10. Set *Longitude* to **30 00 32**.

11. Click **Import**. This loads the Project Properties into Quantity Manager and marries them with the project database *(.mdb)* file.

Note   Unless something in TRNS*PORT changes that affects the project properties you will not have to reload the project properties xml file for this database.

Exercise 5.6 *Export Quantities for TRNS*PORT*

1. In Quantity Manager in the left hand pane, select the **Pay item Table** tab.

2. Select the first item then using the *Shift* key on the keyboard select the *last item in the table*.

3. Select **Project > Export > Export**. This opens the Export dialog.

4. Set the *Export Style* to **aecXML + Funding** from the drop down menu.

5. Toggle **On** *Rename TRNS*PORT Groupings As* and enter **0400**. By default, it is set to *DesignEstimate*.
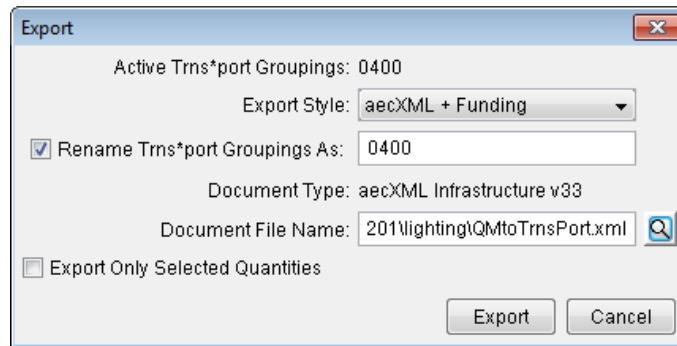
6. For *Document File Name* click the *magnifying glass* icon and browse to the **lighting** folder.

7. Enter a new *file name* for the TRNS*PORT *(.xml).* For this example use **QMtoTrnsPort**. This name is not important, but must be remember for the upload process.

Note   Do not overwrite the _PI xml file. This is the project properties (.xml) file. You may need this to import into Quantity Manager if errors are found.

8. Click **Open.** This loads the file name into the Export dialog.

9. Click **Export.**

10. Click **OK** on the *Information* box to confirm the export process.
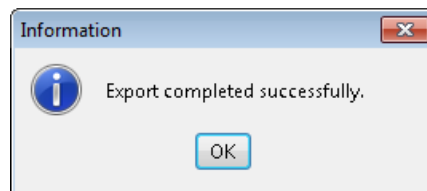
11. Close the Export dialog.

This data is now ready to be imported back into TRNS*PORT using the Designer Interface. The Designer Interface can be opened from the State Specifications and Estimates Office web page at: http://www.dot.state.fl.us/estimates/BOE/BOEonline.shtm.

Exercise 5.7 *Generate CSV File in Quantity Manager*

In this exercise the student will create a csv output file from Quantity Manager. This *(.csv)* file will then be converted to *SBTBLT.csv*, which tools on the FDOT Menu can open and import into the Tabulation of Quantities Sheet.

1. In Quantity Manager, select **Tools > Reports > Create**. This opens Create Report.
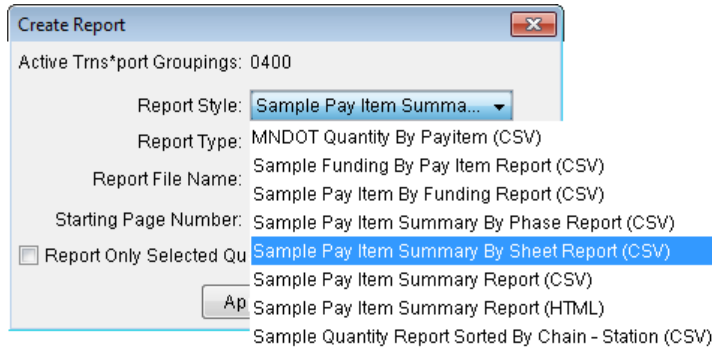
2. From *Report Style* select select **Sample Pay Item Summary By Sheet Report (CSV)** from the drop down menu.

3. For *Report File Name* click *magnifying glass* icon and browse to the **lighting** folder.

4. Enter a *file name*, for this example use **SBTBLT** the *(.csv)* extension is automatic.

5. Click **Open**.

6. On the Create Report, click **Create.** This opens the *(.csv)* file in Excel. Do not close Excel.
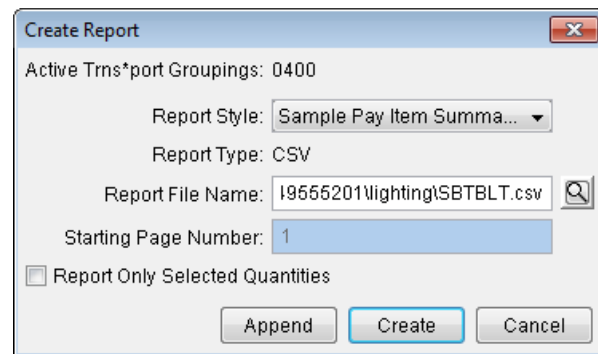
7. Click **Cancel** on the Create Report.

8. Close Quantity Manager. In Excel, select the **Data Tab > Sort & Filter** and **Sort A to Z** icon.

9. *Sort the data* by the **Name** column.

10. Click **Save**. *Do not close Excel*.

Exercise 5.8 *Create Tabulation of Quantities Sheet*

Next, you will create the **Tabulation of Quantities Sheet** to import the Excel data created in Quantity Manager.

1. If MicroStation was closed, reload MicroStation using the FDOTSS3 shortcut from the desktop folder.

2. Open the **Tabqsp01.dgn** file was created in an earlier exercise, from the *lighting* folder and skip steps **3-7**.

3. If the *Tabqlt01dgn* file was NOT created, open the **Dsgnlt01.dgn** in the *lighting* folder.

4. From FDOT Menu, select **Actions > Create/Edit File**.

5. Using Create File/Project, select the active project and create the **Tabulation of Quantity Sheet**. This file goes in the *lighting* folder.

6. Open the **Tabqlt01.dgn** file.

7. Accept the *Plot Scale* of **50**.

8. From FDOT Menu, select **Actions > Sheets**.

9. Place the **Plan Sheet** in the *Tabqlt01.dgn* file.

Exercise 5.9 *Import Excel Data to Tabulation of Quantities Sheet Using LDM*

Next, the designer will import the Excel data created in Quantity Manager using LDM.

> ### Create LDM Links and Add Index of Sheets

1. From the FDOT Menu select **Actions > Linked Data Manager**.
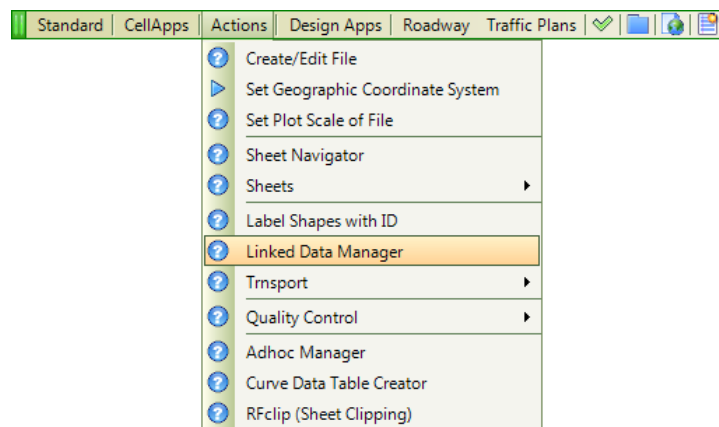
2. Click the **Create New Link** button.
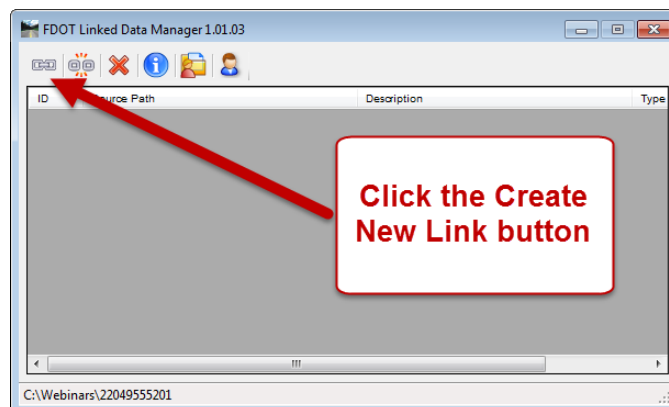
3. Click the **Create New from Template** button.
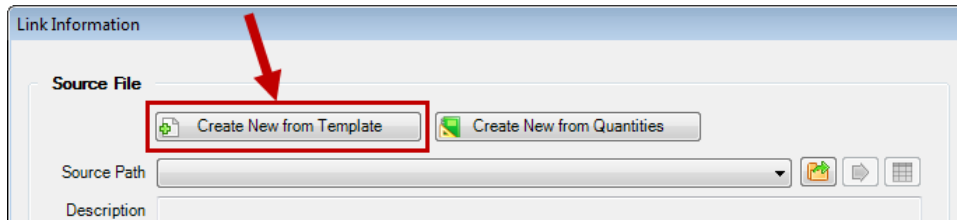
4. Select the **Sheets** from the drop down *Selection List*.
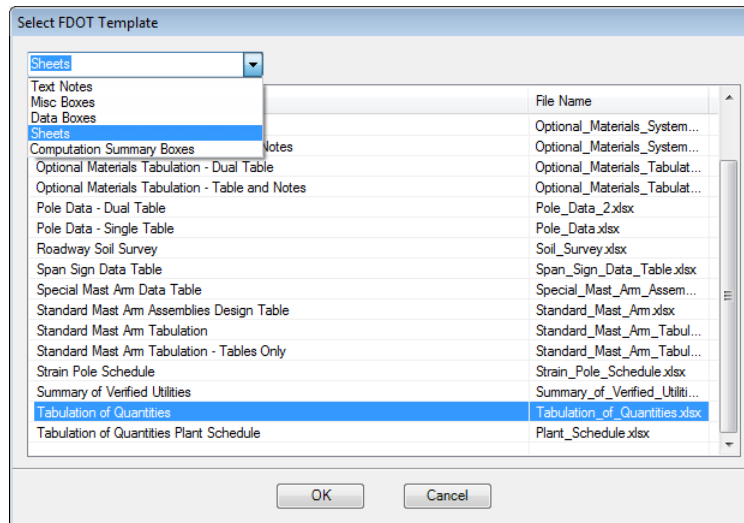
5. Select the **Tabulation of Quantities** *template*.

6. Click the **OK** button.

7. On the Save File dialog, navigate to the *lighting* folder and click the **Save** button to save the **Tabulation_of_Quantities.xlsx** file to that location.

8. Enter **Lighting Tabulation of Quantities** in the *Description*.

9. For the *Excel Settings*, set the *Worksheet* to **Sheet 1** as shown below.
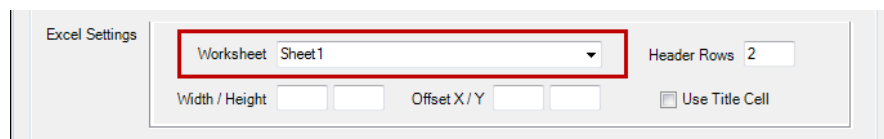
10. Set the **Graphic Settings** as shown below.
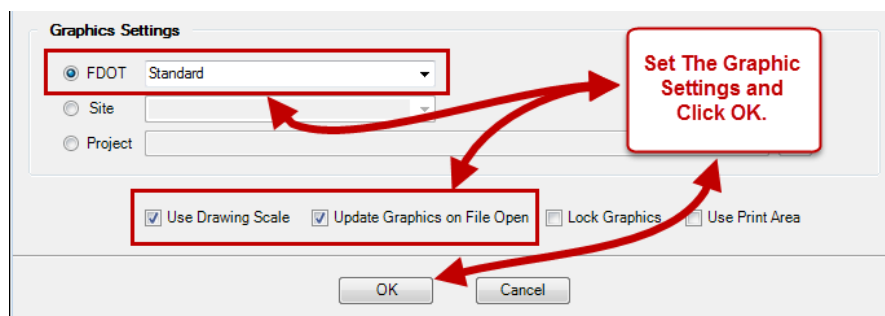
11. Click the **OK** button.

12. Snap to the upper left hand corner of the blue sheet border.

13. Issue a **Data Point** to place the *Tabulation of Quantities* tabulation table.

> ➢ *Update LDM Links and Add Data to the Tabulation Sheet*

1. Open both the **Tabulation_of_Quantities.xslx** file and the **SBTBLT.csv** file.

2. In the *SBTBLT.csv* file, highlight the first pay item starting at **Column A, Row 2** to **Column C down to the last pay item** highlighting all of the cells in the grid.

Note   Do not highlight the column headers in row 1.

3. Copy and Paste the **highlighted cells** into the *Tabulation_of_Quantities.xslx* file using *Paste Special, No Borders*.
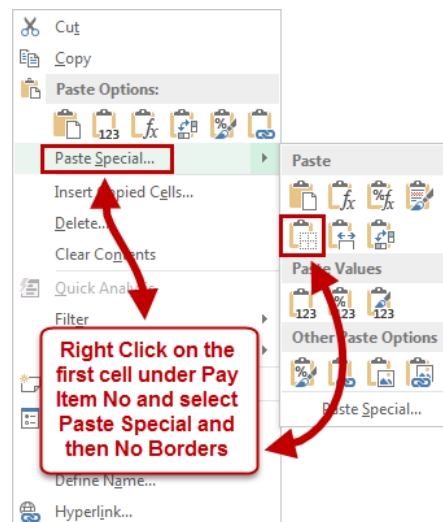
4. In the *Tabulation_of_Quantities.xslx*, enter the **sheet names** in the *title cells* under *Sheet Numbers* as shown in the image below.

S - 1		S - 2		S - 3	
PLAN	FINAL	PLAN	FINAL	PLAN	FINAL
8		10		10	
6		8		4	
1260.5		1306.3		945.7	

5. Copy and paste the individual **Sheet** *data columns* into the corresponding *Sheet Name, Plan Column*.

6. Copy and Paste the **Total** *data* in the *Totals This Sheet, Plan* column.

7. Adjust the **column widths** in the *Tabulation_of_Quantities.xslx* file, if necessary and save the file.

8. Right click on the link created in the LDM and select **Update Now**.
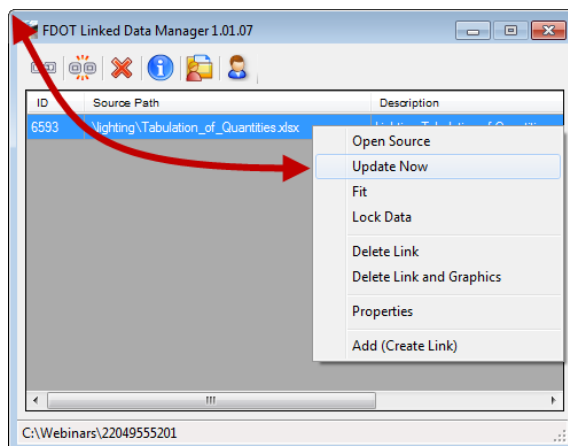
9. **Adjust** the *columns in the file* so the Tabulation Table fits on the *Plan Sheet*, **Save** the Excel *spread sheet* when changes are complete and **Update** the link to *apply the changes*.

10. Close the LDM and the Excel files.

➢ *Add Sheet to Sheet Navigator*

1. Take a moment to review the *Tabulation of Quantities Sheet.*

2. Run Sheet Navigator to fill in the **Title Block**, **Sheet Number**, **Financial Number**, **County**, **Road Number** and add the **Digital Signature Note**.

Title Block:	– **TABULATION OF QUANTITIES** *(Should be filled out already.)*
Sheet Number:	– **L-4**
Financial Number:	– **220495-5-52-01** *(Tab or double click this field to fill automatically.)*
County:	– **Wakulla**
Road Number:	– **SR 61**
Digital Signature Note:	– **Standard**

3. Close Sheet Navigator.

6 SHEET NAVIGATOR

OBJECTIVE
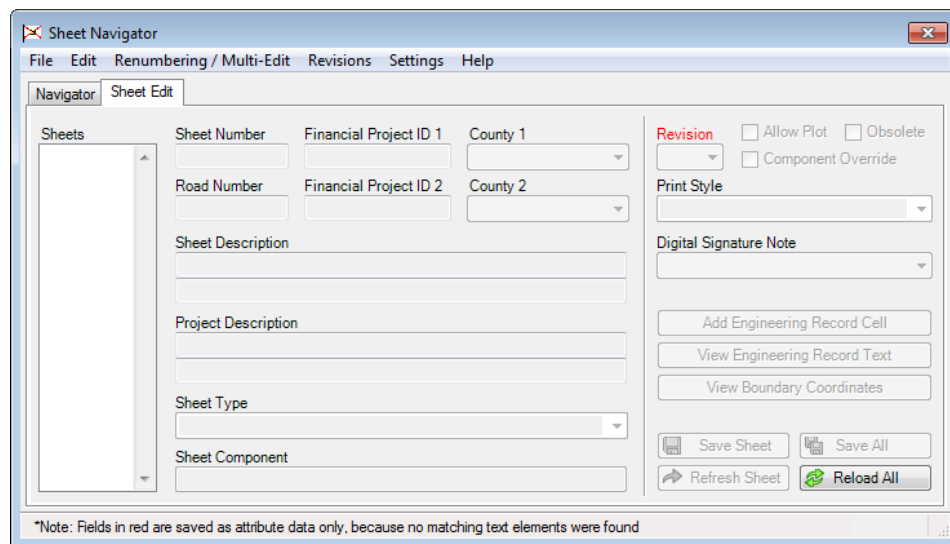
In this chapter the following topics will be covered:

- Overview of Sheet Navigator
- Sheet Navigator Work Flow
- Auto Numbering and Renumbering
- Edit Multiple Sheets

INTRODUCTION

This section covers the application Sheet Navigator, which is used to label and tag sheets for digital delivery and/or electronic delivery. Its functions include entering of sheet title block information, auto numbering of sheets, renumbering sheets and performing Multiple Edits on sheets. Using this tool will greatly reduce the time needed to number sheets in a set of plans or even renumber sheets after sheets have been added or removed from the set, which is a common occurrence over the life span of a project.

SHEET NAVIGATOR OVERVIEW

Sheet Navigator is one of the single most important tools in preparing a project for Electronic Delivery and/or Digital Delivery. Sheet Navigator has the ability to Auto Number sheets, Number single or multiple sheets and Re-Number sheets. Sheet Navigator can also tag a sheet as Obsolete so that Electronic Delivery Indexer (EDI) does not select the sheet and index it. This allows the designer to create exhibits or temporary sheets in the project without the worry of Electronic Delivery including them in the plans package. Sheet Navigator can also perform multiple sheet edits at one time such as adding the Road Number or the Digital Signature Note to multiple sheets.

Sheet Navigator creates an index named Sheetinfo.xml located in either the root project folder or in the component folder depending on how the designer created the index. This index is crucial to Sheet Navigator functioning properly when working in the Auto Numbering, Renumbering or Multi Sheet Edit mode.

Sheet Navigator is used to:

- Tag sheets for Electronic Delivery
- Number Sheets
- Label Sheet Title Blocks
- Auto Number sheets
- Renumber Sheets
- Apply Revision text

SHEET NAVIGATOR WORKFLOW

1. Create Sheet Files Using GEOPAK or FDOT Menu.

   Note   The Border can be in the active file or a reference file.

2. Open Sheet File in MicroStation.

3. Start Sheet Navigator from FDOT Menu: Actions > Label Sheets (Sheet Navigator).

4. Populate Fields in Sheet Navigator.

5. Add Digital Signature Note.

6. Save Data to Sheet File.

SHEET EDIT TAB

The *Sheet Edit* tab is used for populating the standard title block information.

- **Sheets** – This shows the number of sheets in the active file and also the sheet number if they have been numbered. Sheets show as Undefined if they have not been numbered.

- **Sheet Number** – This is the actual sheet number example S-3.

- **Financial Project ID 1** – Eleven digit FIN number. This is automatically populated by clicking in the field. A project configuration file must be selected when opening MicroStation for this to work.

- **County 1** – If the project spans two counties this is for the first county. From the drop down menu, select the county.

- **Road Number** – This is the State Road number of the project alignment.

- **Financial Project ID 2** – For stung projects, this is for the second project FIN number.

- **County 2** – This is for the second county name if your project spans two counties.

- **Sheet Description** – Example: Tabulation of Quantities.

- **Project Description** – This is used on Structural sheet borders and provides a brief description of the project.

- **Sheet Type** – This is filled in automatically and is triggered by text in the sheet cell. 01-Plan Sheet is the default sheet type if none of the standard text strings are found. Refer to FDOT Menu for more information on this tool.

- **Revision** – Shows what revision if any the selected sheet has applied to it.

- **Allow Plot** – Controls whether the sheet will be converted to a PostScript image when using FDOT's EDelivery software.

- **Obsolete** – Toggled On will make the selected sheets obsolete or inactive which means the EDelivery software ignore these files.

- **Component Override** – Allows the designer to assign a sheet to a different discipline other than the default discipline. The default discipline is the discipline in which the MicroStation file resides.

- **Sheet Component** – Describes what is the plan set component of the selected sheet based on the sheet number. Example: S-4 is component Signing and Pavement Marking where 4 is the Primary component.

- **Digital Signature Note** – From the drop down menu, select the type of Electronic Delivery Signature to place in the file. This cell uses the plot scale of the file.

- **Add Engineering Record Cell** – Places the Engineer of Record cell in the file.

- **View Engineering Record Text** – reads the data in the Engineer of Record cell and displays it.

- **View Boundary Coordinates** – Displays the coordinates of the sheet boundary.

- **Reload All** – Closes and re-opens Sheet Navigator. If changes have been made to Sheet Navigator the designer will be warned to save the changes or cancel.

- **Refresh Sheet** – Refreshes Sheet Navigator and MicroStation. If changes have been made to Sheet Navigator the designer will be warned to save the changes or cancel.

- **Save Sheet** – Saves changes and updates the MicroStation file.

- **Save All** – Saves changes to all sheets and updates the MicroStation files.

NAVIGATOR TAB

The Navigator tab is used for the Auto Numbering, Renumbering and Editing of multiple sheets at one time.
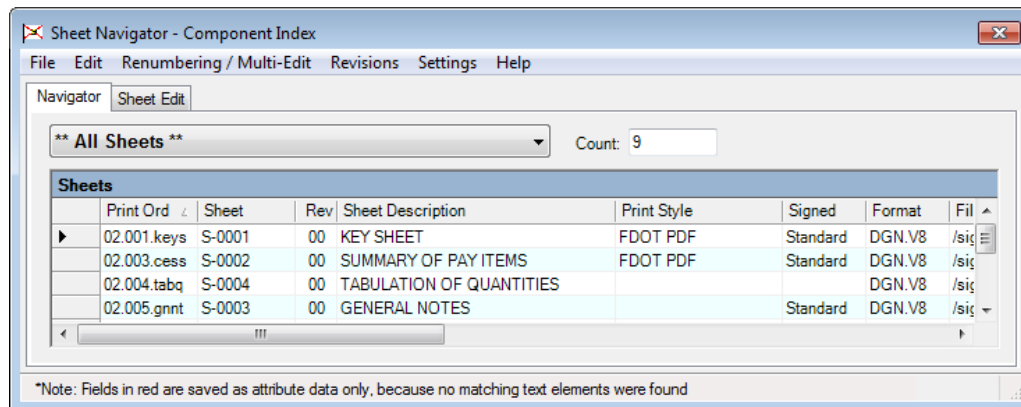
AUTO NUMBERING AND RENUMBERING

This is a multi-step process where the designer will need to first auto number the sheets then renumber those sheets because the auto numbering process uses arbitrary numbers based on the file name. Example: Keyslt01.dgn is already numbered L-1, when Auto Numbering is started, Planlt01 is auto numbered L-1. This process can save the designer a lot of time on large projects and it takes the tedious operation of having to open every sheet file and run Sheet Navigator manually. If a project has to be renumbered because of adding or removing sheets, Sheet Navigator automates the entire process.

Auto Numbering or Renumbering Workflow:

1. Create a Sheet Navigator *Index* file named **sheetinfo.xml**. This can either be a *Project Index* or *Component Index*.
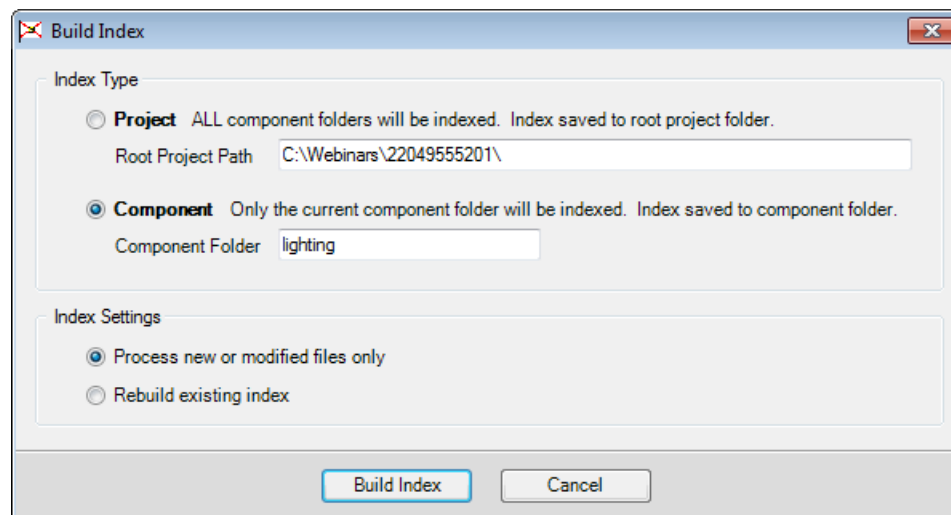
o Selecting *Project* will create an index file that covers all disciplines in the project; this is the option to use if the designer is the Prime and need to number all of the sheets on the project.

o Selecting *Component* is used when the designer is only concerned with a specific discipline. The discipline name will default to the working directory that Sheet Navigator started in.

Note Very important that the Sheetinfo.xml is always current. If there is any doubt as to how up to date this file is always create a new Sheetinfo.xml. It only takes a short amount of time to create this file and it hurts nothing to override the existing file.

2. Open the **sheetinfo.xml** file.

3. Start the *Numbering* process by selecting the process to run: **Auto Number, Renumber** or **Edit Sheets**.
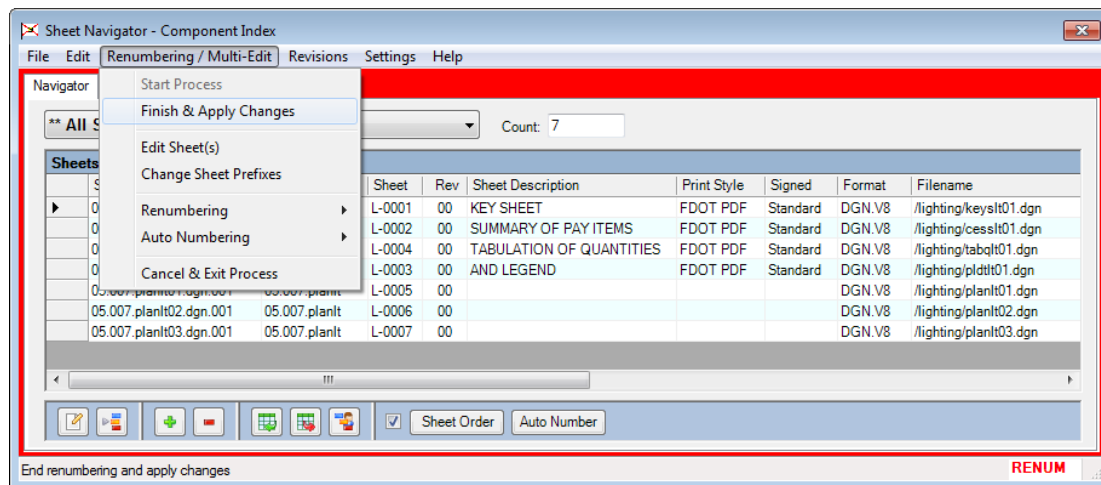
4. **Finish** and **Apply** changes.

5. **Cancel & Exit Process** allows the designer to stop the process without making any changes.

Exercise 6.1 *Auto Numbering and Renumbering Sheets*

➢   **Auto Numbering of Sheets**

1. Open *dsgnlt01.dgn*. This exercise can be completed from any design file.

2. From FDOT Menu, select **Utils > Label Sheets (Sheet Navigator)**. This opens Sheet Navigator.
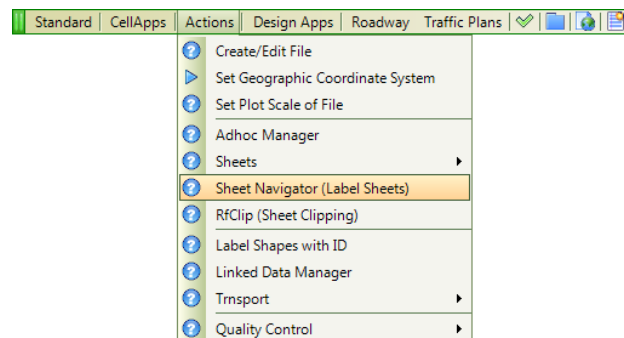
3. In Sheet Navigator, select the **Navigator** tab.

4. In the *Navigator* tab, select **File > New**. This opens Build Index.
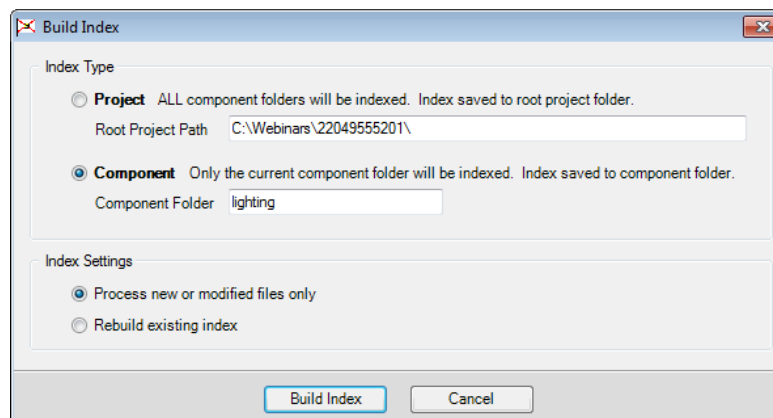
5. Select the **Component** option.

6. Click the **Build Index** button. This creates **sheetinfo.xml** in the *lighting* folder. All other discipline folders are ignored.

7. MicroStation will close and the **CADmanage (CM.Batch)** process will run.
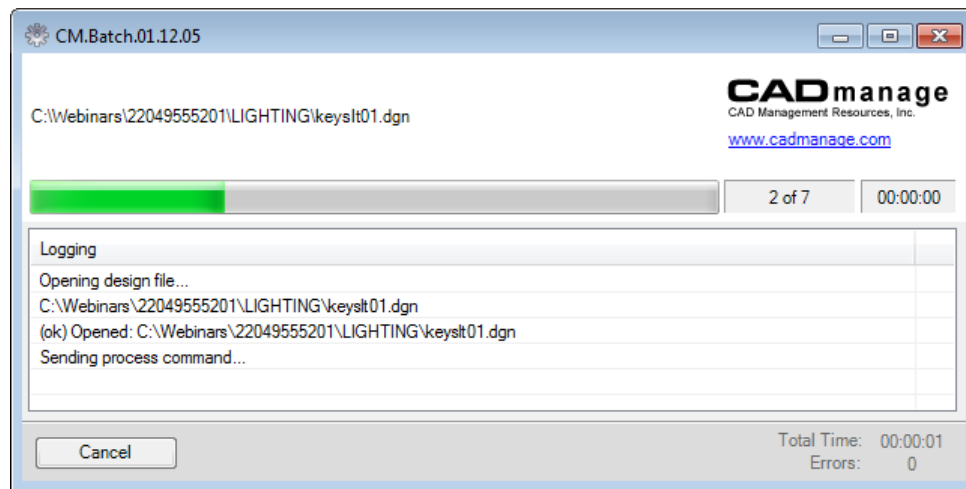
> ➢ *Renumbering of Sheets*

After the index process is complete, the Navigator tab will be populated with a list of sheets found in the lighting folder. Up to this point, there has been some sheets already filled out using Sheet Navigator.

However, there are a number of sheets that have missing information and are not numbered. Sheet Navigator can be used to fill out missing information, tag the sheet with the standard Digital Signature Note and number each individual sheet in a batch process. The following steps will cover the basics of this process.

1. Select **Renumbering/Multi-Edit > Start process**

2. Click **OK** on the Warning box.
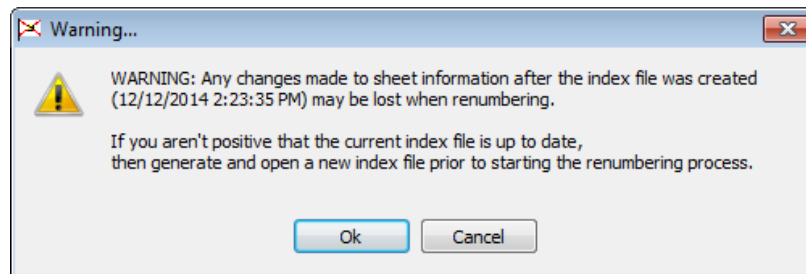
3. Several options have been added to the Sheet Navigator dialog.

4. Select **Renumbering/Multi-Edit > Auto Numbering > Auto Numbering Mode**. You could also use the check box at the bottom of the dialog to activate this option.

5. The figure below shows all of the sheets, some are numbered and others are tagged as undefined which means the undefined sheets have not been numbered. When auto numbering is run, *planlt01* might be numbered **L-0001**.
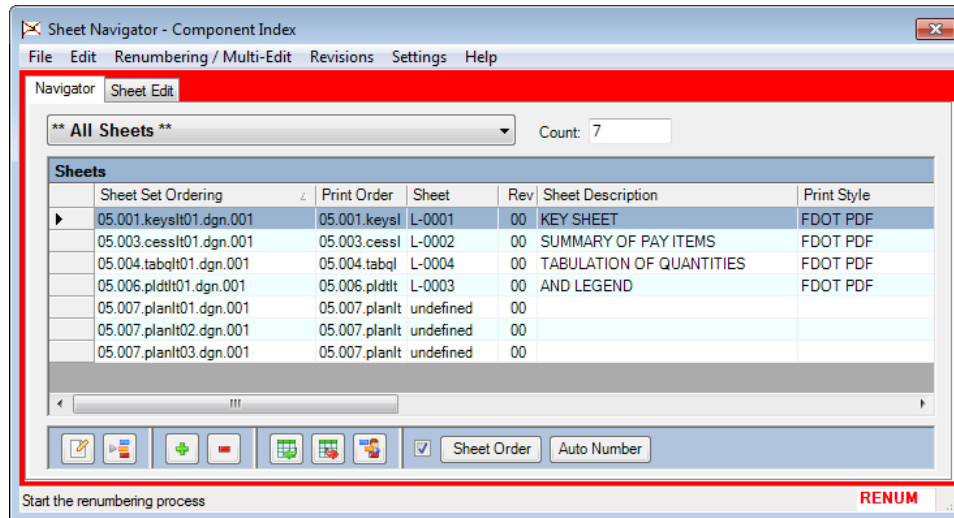
6. Click **Auto Number**.

7. Click **OK** on the Auto Number Sheets dialog. This is information about the logic used to number sheets.
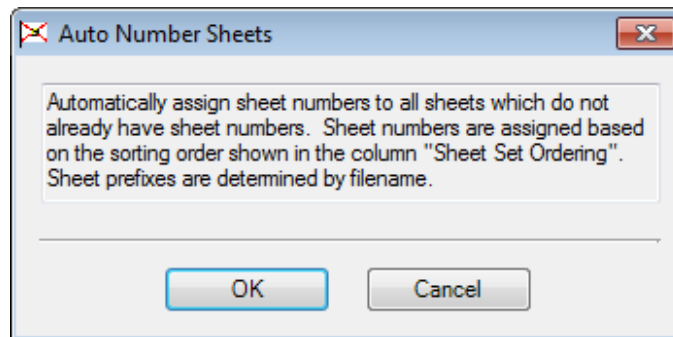
8. Notice in the figure below that the undefined sheets now have numbers, however, they may not be correct. If the sequence numbers are incorrect, you will need to renumber these sheets manually.
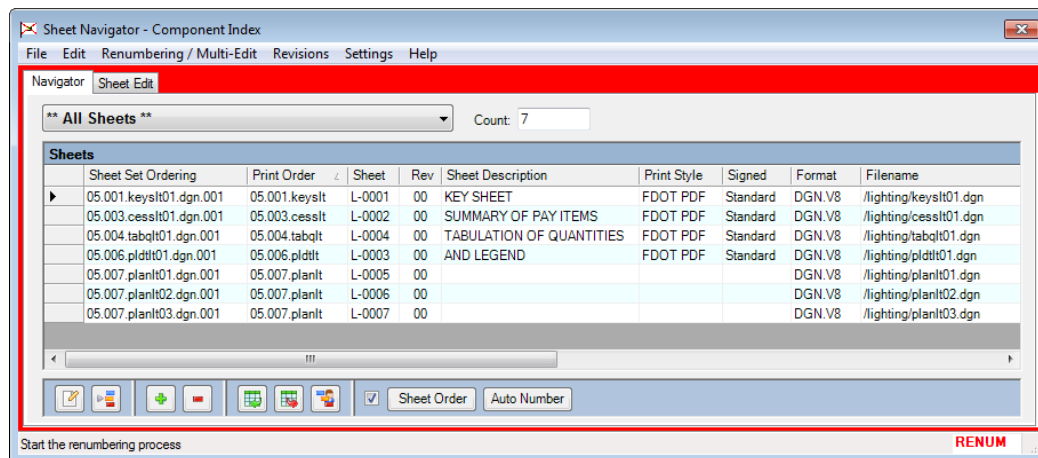
9. If the sheets are numbered correctly, skip to step 19.

10. If the sheets are sequenced incorrectly, click in the **grey column** before the *Sheet Set Ordering* column and drag across the sheets sequenced incorrectly.

11. Click the **Sequence Sheets** button at the bottom of the Sheet Navigator dialog. This will make *Planlt01* sheet number **L-0005** and increment the other selected sheets accordingly.
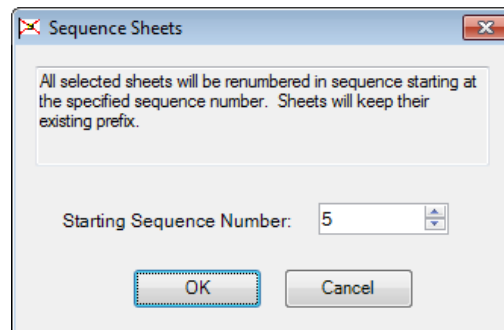
12. Set *Starting Sequence Number* to **5**. This changes *Planlt01* from **L-0001** to **L-0005** and increments the other selected sheets by the same number.

13. Click **OK.** Notice the Sheet numbers have now been changed. Next, you need to save the changes and update the design files.
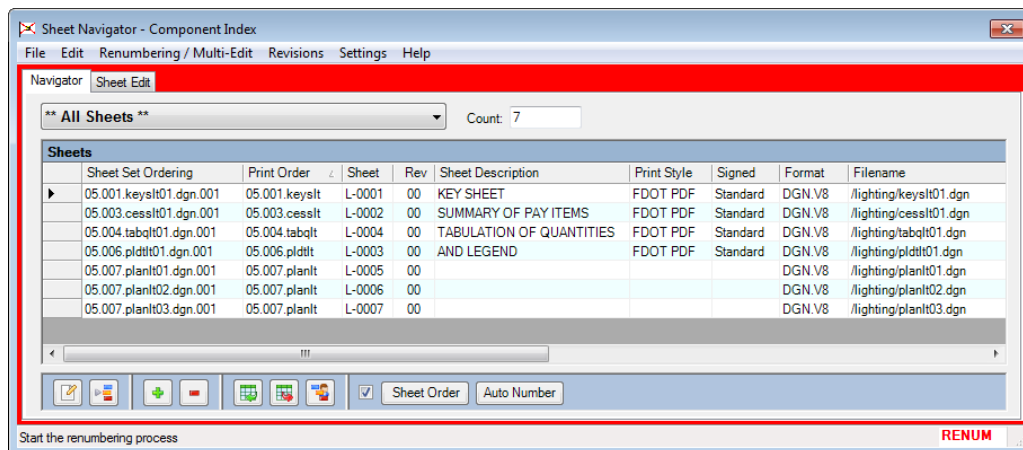
Note   It may be necessary to modify the sheet prefixes to get the correct numbering sequence.

14. Click **Renumbering/Multi-Edit > Finish and Apply Changes**.
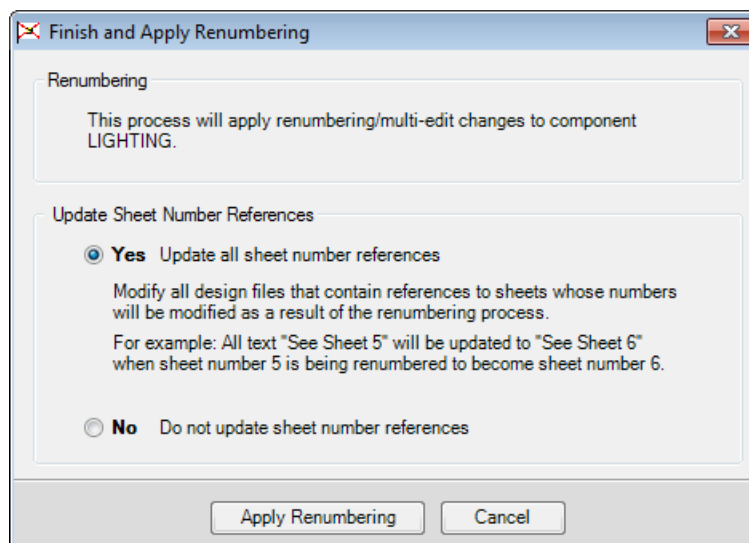
15. Toggle **On** *Yes* to *Update* all sheet number references.

16. Click **Apply Renumbering** on the Finish and Apply Renumbering dialog to apply changes. This will open the MicroStation files and add the sheet numbers to the sheets.

17. Notice in the figure below that the sheets are now numbered sequentially. This was accomplished without having to open any of the sheet files and was completely automated. This is a huge time saver on large projects.

Note   Sheet Navigator can also be used to open files in the index by double clicking on the file.
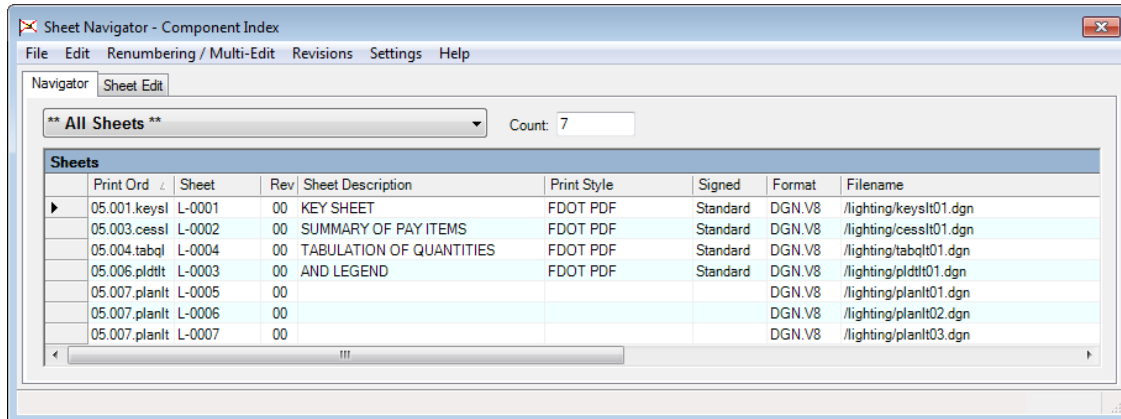
18. Double click on *sheet* **L-0005**, this will open *planlt01.dgn* and set Sheet Navigator to the *Sheet Edit* tab. Notice that the design file has been edited to have a sheet number.

19. In Sheet Navigator open the **Navigator** tab. Notice that several sheets do not have a *Road Number, Standard Digital Signature Note, Print Style* or a *Sheet Description*. In the next exercise you will use this tool to add the missing information to multiple sheets.

Exercise 6.2 *Use Multi-Edit to add Road Number*

1. Continuing in the *Dsgnlt01.dgn*. This exercise can be completed from any design file.

2. Select **Renumbering/Multi-Edit > Start process**.

3. Click **OK** on the Warning dialog.

4. Click the *sheet* **L-0005** and drag down across *sheet* **L-0007**. This selects the 3 sheets with the missing *Road Number*.

   Hint   You must click and drag over the box on the far left side next to the Sheet number where the black arrow is shown. You can also use the Ctrl and Shift keys to select files.

5. Select **Renumbering/Multi-Edit > Edit Sheet(s)**. This opens Renumbering – Edit Sheets dialog.
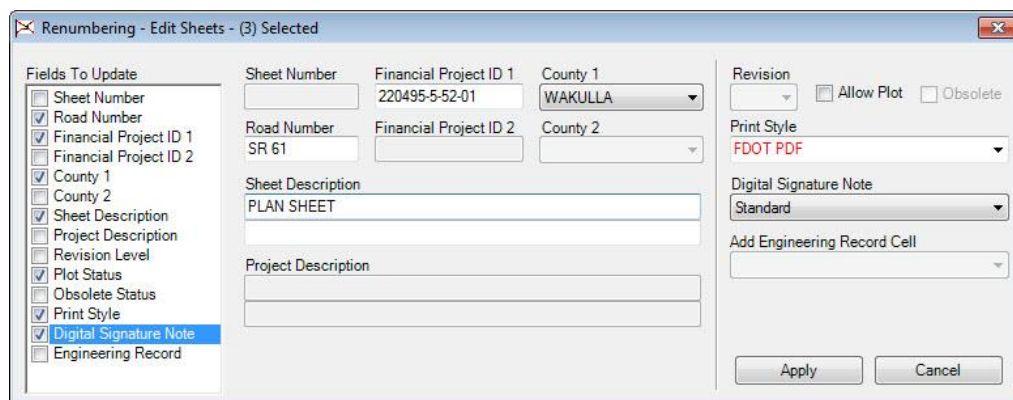
6. Toggle **On** *Road Number*. This activates the *Road Number* field in the dialog.

7. Enter **SR 61** in the *Road Number* field.

8. Toggle **On** *Financial Project ID 1*. This will fill in automatically.

9. Toggle **On** *County 1*.

10. Select **Wakulla** from the *County* list.

11. Toggle **On** *Print Style*.

12. Select **FDOT PDF** from the *Print Style* drop down list.

13. In the *Sheet Description* field enter **PLAN SHEET**.

14. Toggle **On** *Digital Signature Note*. This activates the *Digital Signature Note* drop down list in the dialog.

15. Select **Standard** from the *Digital Signature Note* list.

16. Click **Apply**.

17. Click **OK** on the Warning box.
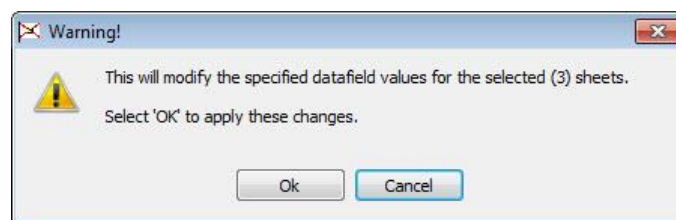
18. Click **Renumbering/Multi-Edit > Finish and Apply Changes**.

19. Toggle **On** *Yes* to *Update* all sheet number references.

20. Click **Apply Renumbering** on the Finish and Apply Renumbering dialog to apply changes. This will open the MicroStation files and add the sheet numbers to the sheets.

21. Click **OK** on the SheetInfo dialog. This will start the editing process. When finished Sheet Navigator will reload and show the multi-sheet edit changes. Use the proceeding steps to make any additional sheet edits such as *Digital Notes* or *Sheet Descriptions*.
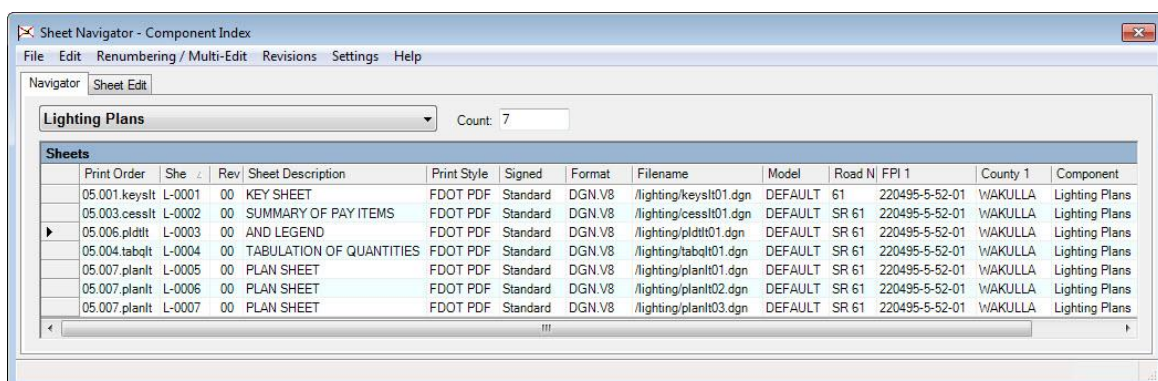
Note Double Click on any sheet selected and review the change.

22. In Sheet Navigator, select the **Navigator** tab.

23. Select **File > New**. This opens the Build Index dialog.
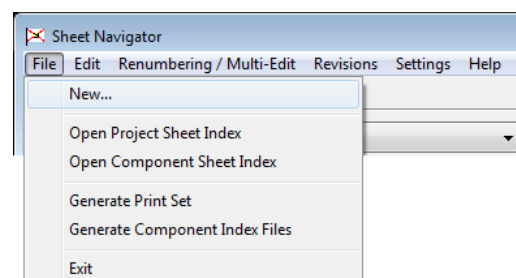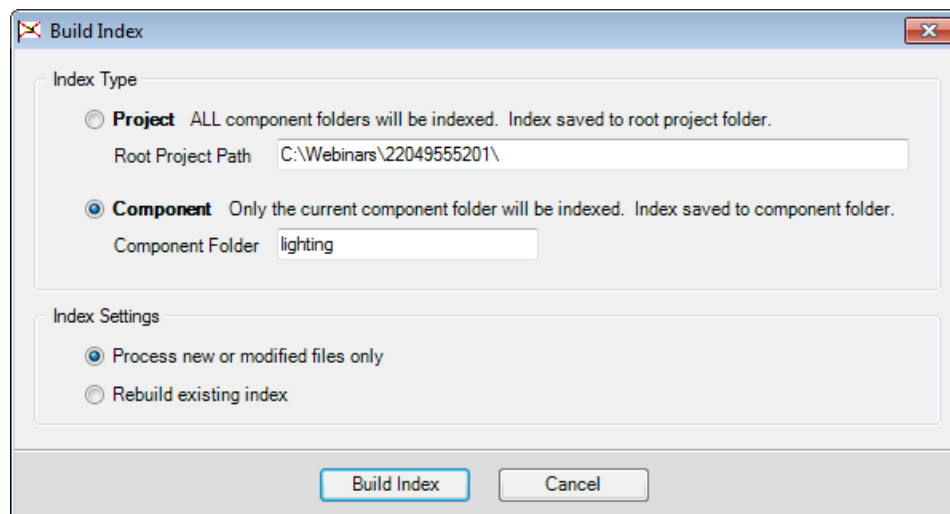
24. Select the **Component** option.

25. Click the **Build Index** button. This creates *sheetinfo.xml* in the *lighting* folder. All other discipline folders are ignored.

26. MicroStation will close and the **CADmanage (CM.Batch)** process will run.

27. When the **CADmanage (CM.Batch)** is complete, MicroStation will be open and Sheet Navigator will display the new *Component Index* on the *Navigator* tab.

28. Select **File > Generate Component Index Files**. This opens the Generate Component Index Files dialog.
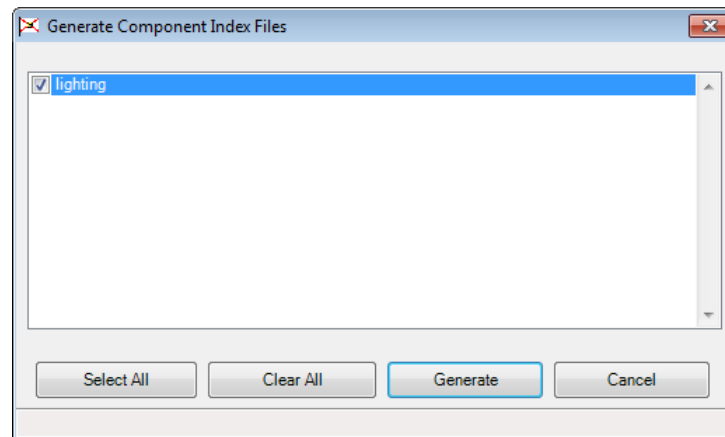
29. Toggle **On** the check box for **lighting**.

30. Click the **Generate** button. Sheet Navigator will create 1 component index file.
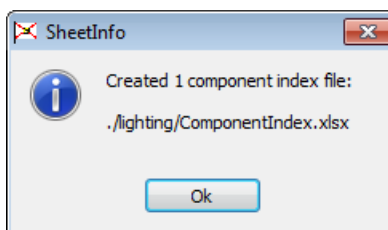
31. Click the **Ok** button.

32. Click the **X** in the upper right hand corner to close Sheet Navigator.

Exercise 6.3 Create LDM Links and add Index of Sheets

1. Open the *Keyslt01.dgn*, zoom in around the text "**INDEX OF LIGHTING PLANS**". This is on the left hand side of the sheet. Notice the *Key Sheet Index* has already been updated. This was accomplished by setting up the initial data link in Chapter 2 using LDM and then using Sheet Navigators Generate Component Index Files tool to update the *ComponentIndex.xlsx* file.

    However, if the link is broken or not setup, please see the following steps to setup the link.

Note   If the data is wrong or out of order, open the ComponentIndex.xlsx file and make changes. In LDM (Linked Data Manager) right click on the link and select Update Now.

2. From FDOT Menu select **Actions > Linked Data Manager**.
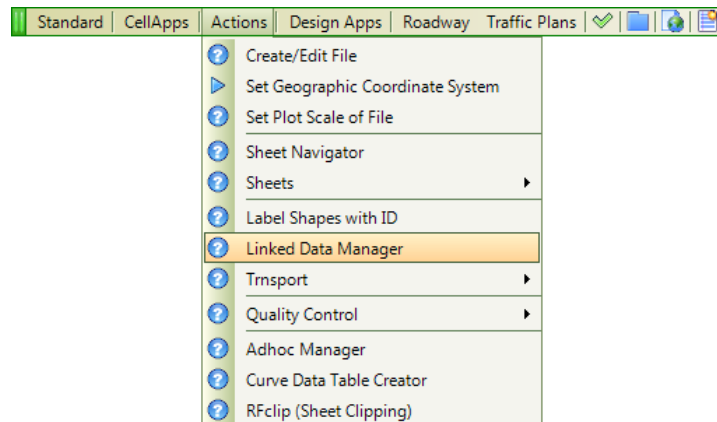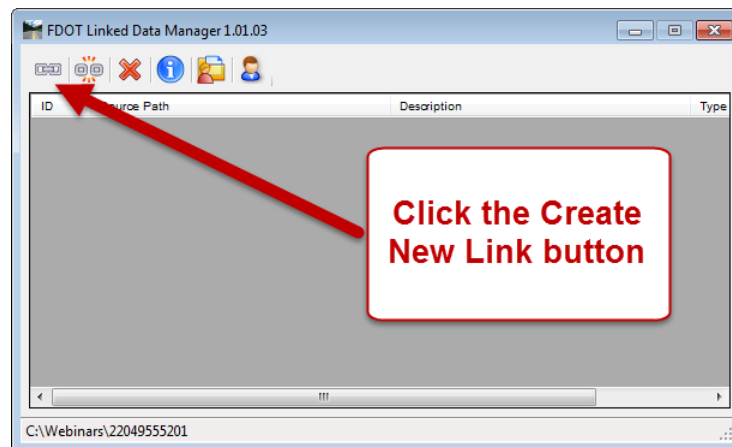
3. Click the **Create New Link** button.

4. Click the **Select Existing File** button.

5. Select the **ComponentIndex.xlsx** and click the **Open** button.

6. For the *Excel Settings*, set the *Worksheet* to **Sheet 1** dialog as shown below.
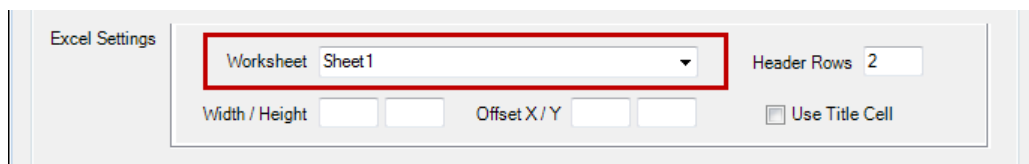
7. Set the *Graphic Settings* as shown below.
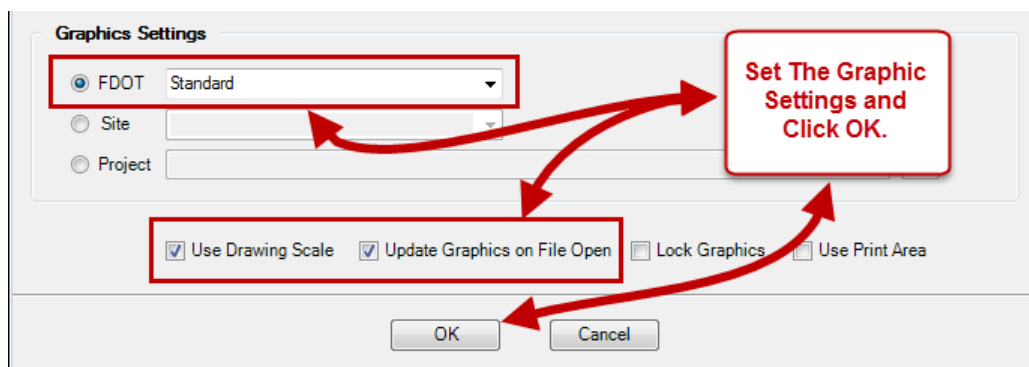
8. Click the **OK** button.

9. Snap to the *origin* of the text label **Index**. This will line up the text properly.

10. Issue a **Data Point** to place the text.

11. Close the LDM.

12. Take a moment to review the *Key Sheet*.

7 PRINTING TOOLS

OBJECTIVE

In this chapter the following topics will be covered:

- Printing from the MicroStation Print Dialog

- Print to PDF

- Create Print Set File

- Printing from the MicroStation Print Organizer

INTRODUCTION

This section covers the basics of printing from within MicroStation. The Department delivers a number of plot drivers that can be used to plot hard copy paper plots or electronic post script or pdf files. The plot drivers control how MicroStation displays the graphics on paper or in an electronic file.

Designers can produce plots several different ways through MicroStation. Each district office may have its own set of printing procedures and plot drivers, contact the CADD manager if there are any questions regarding printing. The intention of this chapter is to explain two printing options, printing through the MicroStation Print Dialog and printing through the MicroStation Print Organizer.

> *Note*   Print Organizer replaces Batch Print. When the designer needs to produce plots from sets of design files, models and/or Project Explorer links, use Print Organizer. Print Organizer creates print sets, which are used to identify, save, recall, and print a particular set of files and/or models.

The Department delivers a document in the FDOTSS3\Resources\plot\ directory that better explains all of the delivered plot drivers. Review this document before printing, as there is important information that may be helpful.

The MicroStation print engine required many changes for the V8i platform to ensure compatibility with this edition's reworked graphics display subsystem. In addition, there are a number of other upgrades to printing functionality.

We will discuss what is new with printing in MicroStation V8i and will see how to edit the new printer drivers and pen tables later in this chapter.

PRINT DEFINITIONS/SETS

A print set is a collection of print definitions and folders. Print definitions are created when the designer adds files, models, or Project Explorer links to Print Organizer. Each print definition consists of a reference to a DGN file, like a sheet model in a DGN file with properties, such as size and scale.

> *Note*   Project Explorer links are note covered in this course guide. For more information on Project Explorer links, please see the MicroStation help file.

The steps required to create print definitions are:

1. Open the Print Organizer dialog.

2. Add the DGN files, models, or Project Explorer links.

3. Specify the print definition creation options.

Print Organizer creates print set (*.pset)* files as a container for print definitions, folders that group print definitions, and global set information pertaining to the print set.

If the same set of files are to be printed on a regular basis, the designer can save the set of files as a print set (*.pset*) file. A print set is used to identify, save, recall, and print a particular set of files and/or models.

The designer can open multiple instances of the same print set file, but only the first instance has write access. Subsequent instances of the print set will open in read-only mode. Read-only mode allows the designer to preview and print the files, but if the designer wants to make changes they will need to save the print set with a different name.

Print Organizer allows the designer to define hierarchical organized print sets. To define this type of print set, the designer will first create folders and sub-folders below the root folder. Print definitions, which are created when files are added, are then organized in the folders and sub-folders.

NAME EXPRESSIONS

The print definition name expression is used to determine the print definition names. The print definition name expression can also be used to rename print definitions.

An expression is composed of a series of components, where a component may be a symbol, a number, a string, or an operator. The symbols available for use in expressions are published by various symbol providers, and are grouped by symbol sets; such as, *System.String, System.Path, PrintSet,* and *PrintDefinition*. To generate an HTML report showing all available symbol sets and the symbols available in each, select Utilities > Report Symbols from the Named Expressions dialog.

An example output file name expression is shown in the following table. The result of the output file name expression is 001–plans.pdf, assuming the print set name is "*plans.pset*" and the printer driver configuration file is *pdf.pltcfg*.

Expression	Result
System.String.Format ("{0:D3}", PrintDefinition.SetPrintDefNumber) & "-" & System.Path.GetFileNameWithoutExtension (PrintDefinition.SourceFile)	001–planlt01.pdf

PRINT STYLES

Print styles provide an easy way of defining a default print definition properties for both Print Organizer and the Print dialog.

The following can be done using print styles:

- Store print definition properties for easy reuse.

- Maintain drawing standards by managing display parameters, using a particular scale, or pen table.

- Set properties for the entire print set; such as, printer name or printer driver configuration file.

- Apply print definition properties when creating print definitions or changing printers; such as, paper size, pen table, plot driver, fence definitions *(plot border)*.

Print styles are useful if the designer frequently uses the same print definition properties every time they print. For example, if the designer prints at a particular size with a specific pen table, they can define those print definition properties in a print style and apply them when the designer creates print definitions or when they change printers. This is accomplished by identifying a print style as a default print style or by assigning a print style to a printer driver configuration file.

The designer can also store groups of commonly used print definition properties in a print style and then reference them on-demand. This method of referencing print styles is helpful when the designer uses a group of print definition properties for more than one project.

The following should be considered and/or set when creating and applying Print Styles:

- If print area should be fenced.

    o A fence will be created based on the Print Definition created on the Fence tab.
    o Rasterized should be unchecked unless the style is intended to show Rasterized features like transparencies.

- If workspaces are left blank, the current workspace in use will be used.

- Create Print Definitions from Models should be set to "All Design Models."

- Define as many search attributes as possible.

    Note   These options will reduce the search time by filtering out elements.

- Define the display attributes to be used when creating prints.

- Select the Printer Configuration file to be used with the print style.

WHERE ARE PRINT STYLES STORED?

Print Styles are stored in DGN Library (DGNLIB) files. In the FDOT Workspace the default styles are stored on the server in *FDOT_PrintStyles.dgnlib*. However, the following are the recommended files and locations for additional Print Styles:

- Site Specific Print Styles can be stored in a file named *Custom_PrintStyles.dgnlib* in the FDOTSS3\RESOURCES\Dgnlibs directory on the server.

- User Specific Print Styles can be stored in a file named *USER_PrintStyles.dgnlib* in the local \FDOTSS3\Workspace\prefs directory.

- Project specific Print Styles can be stored in the *symb* directory.

PRINT DIALOG

The Print dialog provides the necessary tools for printing at any stage during a project. The Print dialog is used to produce plots of the active design file and its references. The designer can perform all printing operations from the Print dialog, which is opened by choosing File > Print for the MicroStation menu.

The following can be done from the Print dialog:

- Select a printer driver and edit its configuration file.

- Create a pen table to resymbolize the print.

- Specify monochrome, grayscale, or true color output.

- Create print definition files.

The Print dialog lets the designer work with two types of printers: the Windows system and a Bentley driver. The two (2) types are:

- **Windows driver** — the designer has access to the functionality of the Windows printer driver to print directly to the system printer device, create an Enhanced Windows metafile, or save to print later.

- **Bentley driver** — the designer can save the print to disk to send to a printer later.

USING PRINT STYLES IN PRINT DIALOG

There are many benefits to using Print Styles in the single sheet Print dialog.

- No need to place a fence.

- No need to select a plot driver.

- No need to select a pen tables.

- No need to adjust display settings.

Note   Once a Style has been applied you can override settings if needed.

APPLYING A PRINT STYLE

The steps to apply a Print Style in the Print Dialog are:

1. To apply a *Print Style*, select **Settings > Apply Print Style** from the Print dialog.

2. You will be prompted to select a *Print Style* from a drop down list of available styles.

PRINT ORGANIZER

Print Organizer is a batch utility for printing and reprinting sets of design files, models and/or Project Explorer links that are stored in a print set *(.pset)* file.

The following can be done using Print Organizer:

- Print, preview, or publish multi-sheet print sets.

- Drag and drop files from Windows Explorer, drag and drop models from the Models dialog, or drag and drop links from Project Explorer to compose print sets.

- Create hierarchical organized print sets.

- Set print preferences per item.

- Edit the properties of individual or groups of print definitions.

- Define and reuse named collections of print definition properties.

- Create PDF output with hierarchical bookmarks.

- Integrate with Project Explorer, to view, print, compose, and navigate print sets.

Print Organizer lets the designer work the same printer driver configuration files as the Print dialog. This allows for printing to be sent either directly to the default system printer, or to a file saved to print later.

In Print Organizer, same as the Print dialog, lets the designer work with two types of printers: the Windows system and a Bentley driver. The two (2) types are:

- **Windows driver —** the designer has access to the functionality of the Windows printer driver to print directly to the system printer device, create an Enhanced Windows metafile, or save to print later.

- **Bentley driver —** the designer can save the print to disk to send to a printer later.

Print Organizer stores project settings in a print set file with a .pset extension. Each print definition consists of a reference to a DGN file, like a sheet model in a DGN file, along with properties, such as size and scale.

The individual files and models within a print set file are referred to as print definitions and can be hierarchically grouped in folders and sub-folders.

Print definitions are created when the designer adds files and/or models to Print Organizer. Assigning the proper values to print definition properties is an essential part of creating a print set; therefore, Print Organizer provides several methods to assign print definition properties and each method is used in a different context. These methods are:

- **Print Styles —** Allows the designer to define and reuse named collections of print definition properties that are stored in a DGN library. The designer can apply a print style when creating print definitions. To create a print style, select Print Organizer's Tools > Define Print Styles menu item.

- **Print Organizer's in-place editing feature —** Allows the designer to edit individual properties for one or more print definitions by double-clicking the property in Print Organizer's right pane to change its value. To edit a property for a group of print definitions, select the print definitions, and then double-click the property to change its value.

- **Single or multi-edit properties dialogs —** Allows the designer to edit a group of print definition properties for one or more print definitions. To edit a single print definition, double-click it to open the Properties dialog or right-click it and select Properties from the pop-up menu. To edit a group of print definition properties, select the print definitions from the Print Organizer dialog and select Edit > Properties.

- **Print Definition Creation Options dialog —** Allows the designer to specify print definition properties when creating print definitions. To open this dialog, select Manually Specified Options on the Create Print Definitions dialog.

Print Organizer supports named expressions for print definition and output file names. Named expressions are created using the Define Named Expression dialog and are stored in the configured DGN libraries. Once a named expression is defined, it is accessible in Print Organizer.
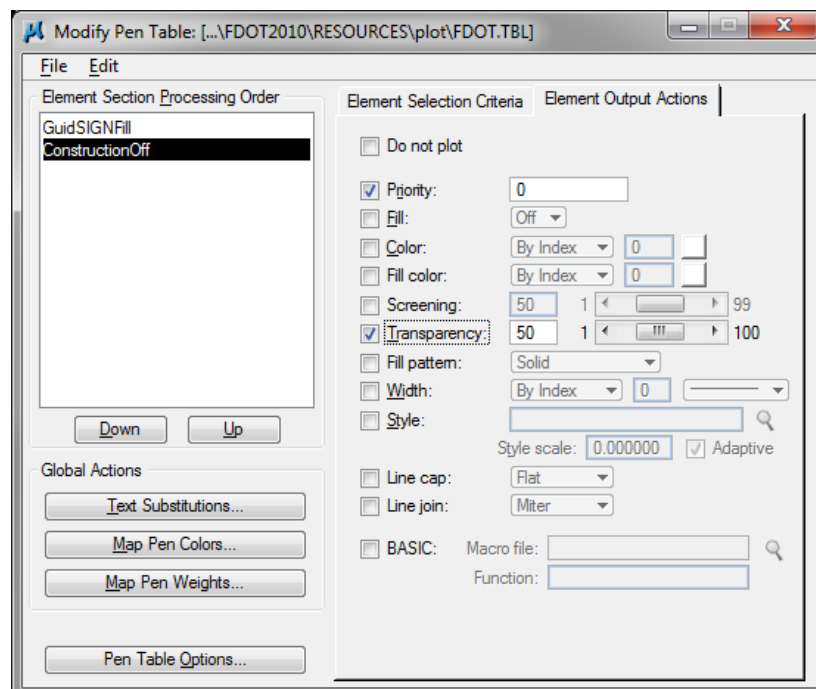
MODIFY PEN TABLE ENHANCEMENTS

ELEMENT SELECTION CRITERIA
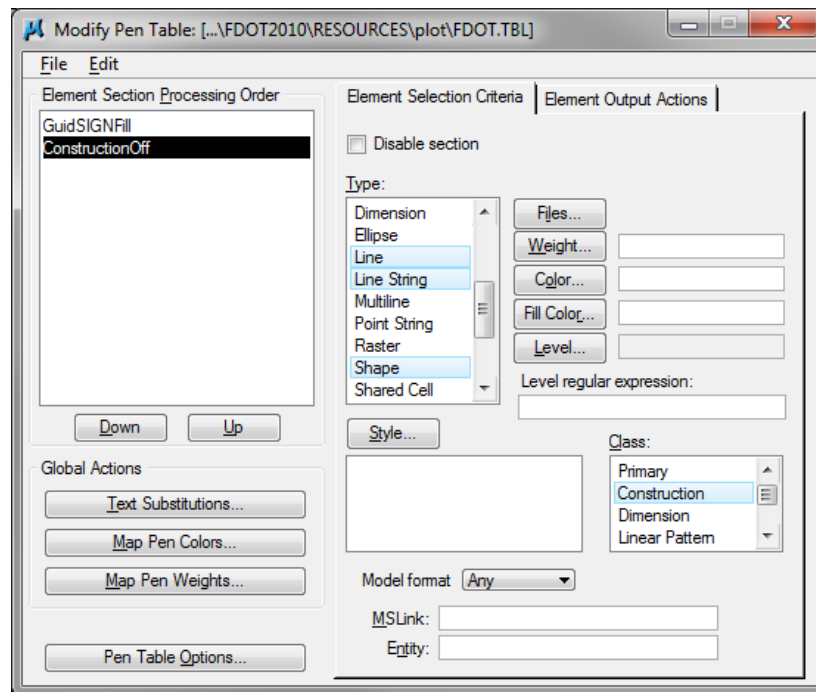
Wildcards can be used to select levels by entering the expression in the "Level regular expression" field and Tags have been added to the list of element types available.

ELEMENT OUTPUT ACTIONS

TRANSPARENCY

Pen table transparency is supported for any printer driver capable of printing in rasterized mode. If pen table transparency is used when printing in non-rasterized mode, the transparency values do not have any effect.

PRIORITY

Pen table priority does not have any effect for printing 3D files. Priority is implemented for printing 3D files through Z depth, using hardware acceleration when possible, so the pen table is no longer required to make multiple passes through the element list. This permits some actions that were not possible with the 2004 Edition, such as assigning different priorities to individual components of a shared cell.

PEN TABLE OPTIONS
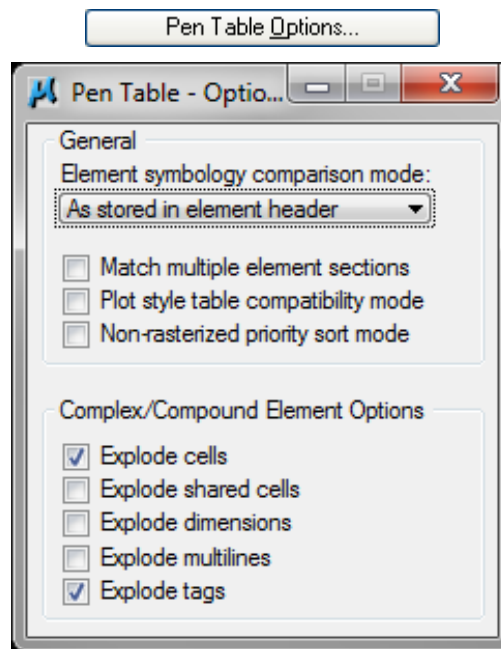
The designer can set up a pen table to apply more than one section to an element. If Match multiple element sections is on in the Pen Table Options dialog box, the sections will be selected and applied in descending order, starting at the top of the list and working downwards.

If this setting is off (the default), once a section has been selected by its input criteria and its output actions applied, pen table processing on that element stops. In the V8i Edition the Element symbology comparison mode setting in the Pen Table Options dialog box affects only the manner in which the input criteria treats level symbology. There are two modes from which to choose:

- **As stored in element header** — the element header symbology is matched against the input criteria.

- **As displayed in view** — the view symbology is matched against the input criteria.

MODIFY THE FDOT PEN TABLE

The following are the steps that will modify the standard FDOT Pen Table *(FDOT.tbl)* so that all of the elements in a referenced file will be printed as if they were all color 20.

1. Have the **Print Definition File** already open before proceeding.

2. To edit the *Pen Table* select **PenTable > Edit** from the menu. This will open the Modify Pen Table.

3. Make a copy of the standard *FDOT Pen Table* by selecting **File > Save As** from the menu and give it a logical name.

4. Select the *last item* in the *Processing Order* field and then select **Edit > Insert New Section Below**.

5. The Insert Section dialog will display prompting you for a *Name*. Enter a **logical name**.

6. Click **OK** to continue.

Note   Make sure the new item is selected in the Processing Order field and then select all of the items in the Type field. The easiest way to do this is to click on any one of the items and the hit Ctrl + A on the keyboard. If you only wanted to gray scale a certain type of element like a shape then you would only select the appropriate items in the Type field.

7. Click the **Files** button to define the references to be processed.

8. The Identify Files dialog will display. In the bottom field enter the *logical name* of the reference to apply the *resymbolization* to or part of the name and wildcards for files with multiple sheets.

Note   When specifying a logical name for a reference in the pen table, you can enter the complete logical name, or you can enter an expression pattern consisting of one or more wild cards. The following table identifies the wild cards that you can use for matching logical names.

Wild Card	Used to
(.)	Match any character. This serves as a single character wild card.
(*)	Match zero or more occurrences of the preceding character. This is used in combination with any character or with the single character wild card (.). Automatically includes the root word.
(+)	Match one or more occurrences of the preceding character. This is used in combination with any character or with the single character wild card (.). Automatically excludes the root word.

9. Click **Add**.

10. Add any other *logical names* to process and then click **OK** to return to the Modify Pen Table dialog.

11. From the Modify Pen Table dialog, click on the **Element Output Actions** tab.

12. Select the *Color* checkbox and set the color to **20** as shown below.

13. Select **File > Save** from the menu to save the changes.

14. Close the Modify Pen Table dialog.

15. Select **File > Save Print Definition File** and save changes to the open *Print Definition File*.

16. Select **File > Reload Printer Driver Configuration**.

PRINTING IN MICROSTATION FROM THE PRINT DIALOG

The Print dialog can be opened from the tool bar in MicroStation or from the menu option File > Print or by using the keyboard shortcut *Ctrl P*. This will open the Print dialog as shown below. Depending on the office/site standards, the Print dialog opens with a standard plot driver. This is a Workspace Configuration setting and should be set by the CADD manager. In this course guide, the class will be using the *fdotprinter.pltcfg* for printing to a device. *Fdotprinter.pltcfg,* as described in the help document, is a copy of the Bentley *printer.pltcfg* driver with weights and styles set to the Department's CADD standards.

> *Note*   Many of the old file formats (PLT, INI, SPC) were changed back in MicroStation XM. Some of the old formats can be used in V8i, but any files created using the new formats in V8i will not be backwards compatible.
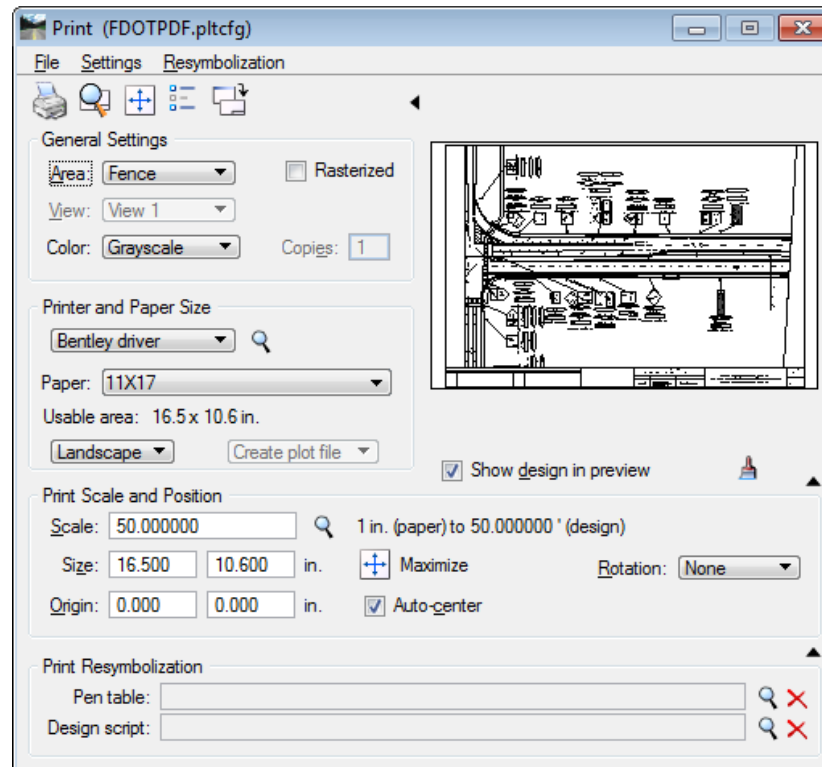
All options for adjusting printing settings are contained in the menu bar at the top of this dialog and the icon bar directly below it. The selected printer driver configuration file appears in the title bar of the dialog.

FILE MENU

The *File* menu contains options for setting up the plots, previewing the plot, selecting and editing a plot driver/configuration, and reloading the driver after editing.

The format of the printer drivers has changed to XML and they now have the extension PLTCFG. MicroStation V8i will continue to support many of the old PLT files but Bentley encourages users to convert their existing drivers because they will drop support of the PLT format at some point in the future. The Department has converted all of the drivers normally provided and include only the PLTCFG version in the FDOTSS3 Software Suite.

SETTINGS MENU

The *Settings* menu contains options for setting the *Units* (inches, feet, mm, cm, dm, m), *Update from View, Print Attributes* (like level symbology and whether or not to plot the print border along with several other attributes), *Set Fence from Fit All, Set Fence from Fit Master, Apply Print Style, Raster Options, 3D Printing* (ghosted), *Hidden Line Removal* and *Preferences*.

A new Preferences dialog was added enhancing the capabilities in a more user friendly layout.

RESYMBOLIZATION MENU

Pen Tables allow the designer to resymbolize their design. In other words, the designer can produce printed output that looks different from the on-screen view of the design. Thus, from a single design file, by using different pen tables, can produce printed output having different element symbology to that of the original design. Pen tables are created using the Print dialog's pen table editor. They are supported in both Print Organizer and the Print dialog.

The *Resymbolization* menu is for *Creating, Attaching* and *Editing Pen Tables*. Pen Tables can be used to do several things during the plot process. The Department delivers one Pen Table that fills in the fields at the bottom of the plots with the file name, date and user name. This file can be modified to perform other functions but understand that any modifications made will affect all users if the Pen Table is located on the server.

PRINT ICONS

 - Submits the Print

 - Opens a Print Preview Window
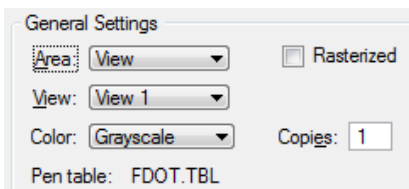
 - Maximizes the Print to Fit the Printable Area

 - Opens the Print Attributes Dialog

 - Updates the Print Preview to Match the Design File

GENERAL SETTINGS

The *General Settings* area is where to select the plot area and which view to plot from. It also shows which if any pen table is loaded, how many copies to plot and the color status.


Rasterized option was added in MicroStation XM and is available in V8i. This option should only be used for rendering or transparencies because it generates larger plot files and is a considerably slower process. When this option is selected MicroStation will convert both raster and vector data into a single raster image and send it to the printer.

Note   The Rasterized toggle option is not for printing files.

PRINTER AND PAPER SIZE

The **Printer and Paper Size** section is for selecting whether to use a Windows printer or a Bentley driver. Clicking on the magnifying glass allows the designer to browse and load a plot driver. Clicking the Windows icon lets the designer configure the windows printer. The designer can also select the paper size and whether or not to plot as Landscape or Portrait. The Full Sheet toggle, when on will maximize the print size to the selected paper size as seen in the figure below.
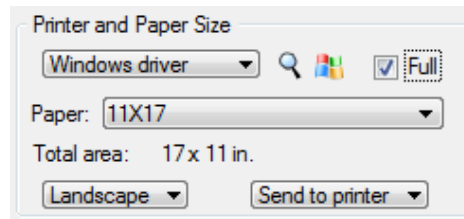
The figure below shows the same plot area with Full Sheet cleared.
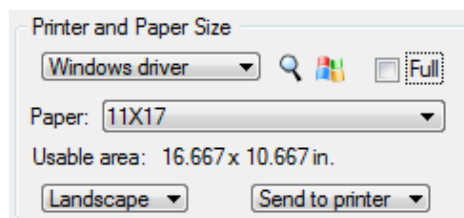
PRINT SCALE AND POSITION

Print Scale and Position is where to fill in an exact plot scale. Typically, how this works is the designer places a fence around the plot border of the design file; this will calculate a scale and display it in this portion of the dialog as seen below. Next, round this odd number up to the actual scale. In the example shown below the actual print scale is 100.00.
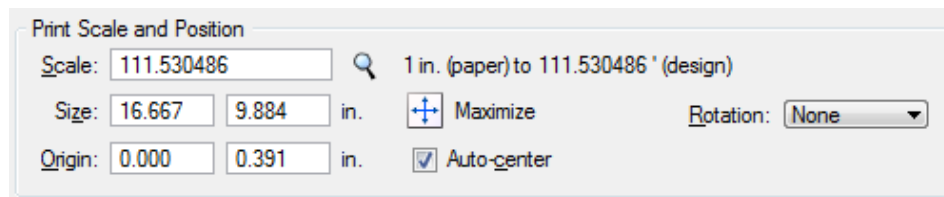
> **Note** This is one of the settings that can be saved into the definition file so when loaded for each plot the scale will always be set to the correct scale.

The designer has the ability to adjust the X Origin and Y Origin print position. These values change the position on the print on the paper. The right hand side of the Print dialog shows the preview.

Exercise 7.1 *Printing with the Print Dialog*

➢ *Printing to a PDF File with the Print Dialog (Part 1)*

1. Open *Planlt01.dgn* in the *lighting* folder.

2. Place a **fence** on the *sheets plot border*. This is the red shape around the outside of the border.

3. Click on the **Print** icon on the MicroStation Standard tool palette or select **File > Print**. This opens the Print dialog and sets a default plot driver. If a default plot driver is not set, the Select Printer Driver Configuration Files dialog will display. If so, skip to step 5.
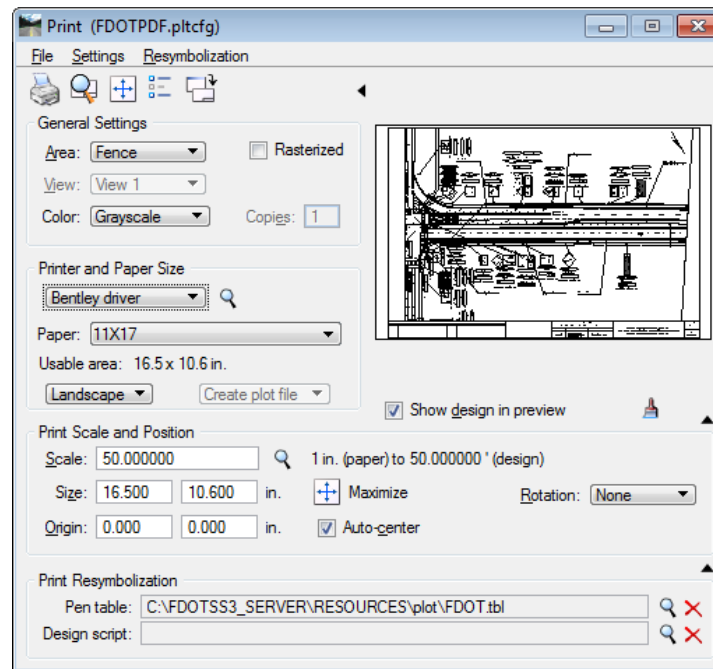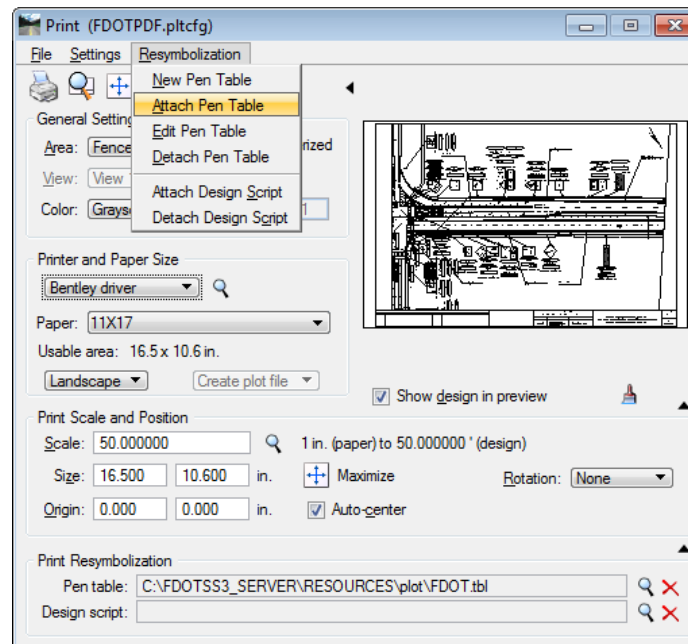

4. In the Print dialog, select **File > Select Bentley Driver**. This opens Select Printer Driver Configuration Files dialog.
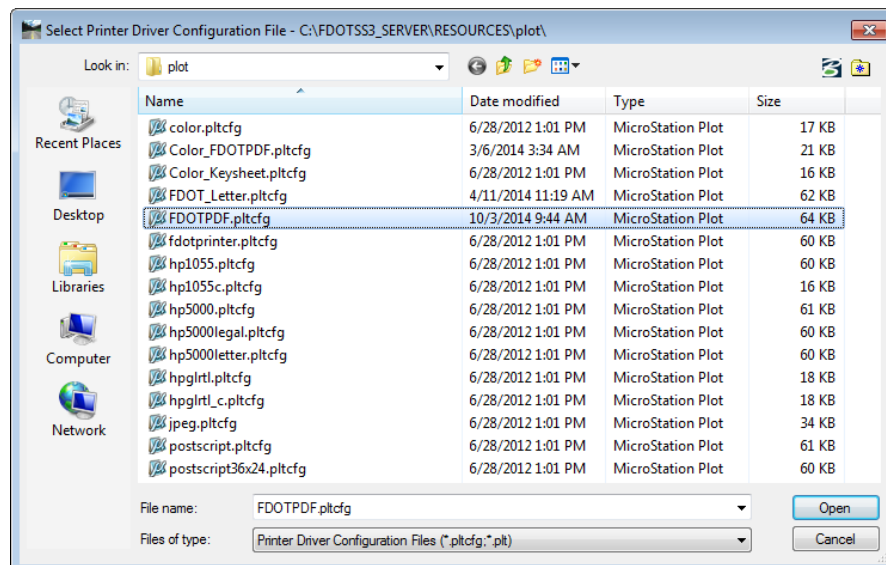
5. Browse to *FDOTSS3\RESOURCES\plot* and select **fdotpdf.pltcfg**.

6. Click **Open**. This loads the *fdotpdf plot driver* and changes the settings in the Print dialog.
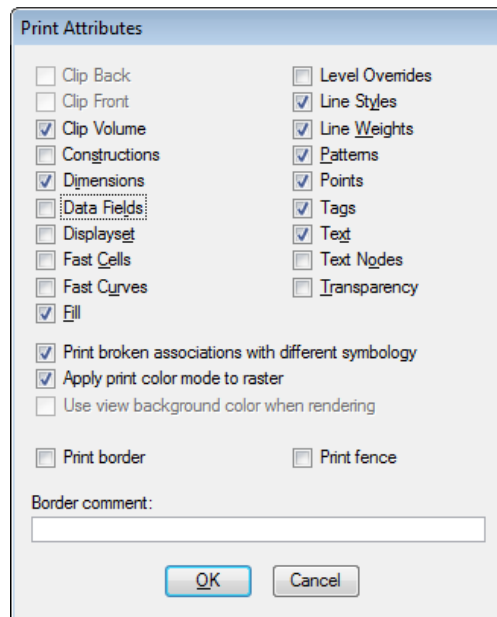
7. Select **Resymbolization > Attach Pen Table** on the Print dialog.

8. Set the *Paper Size* to **11 x 17**. This is the default setting in the plot driver.

9. Set the *Print Scale* to **50.00**.

10. Click the **Print Attributes** icon or select **Settings > Print Attributes**. This opens Print Attributes.
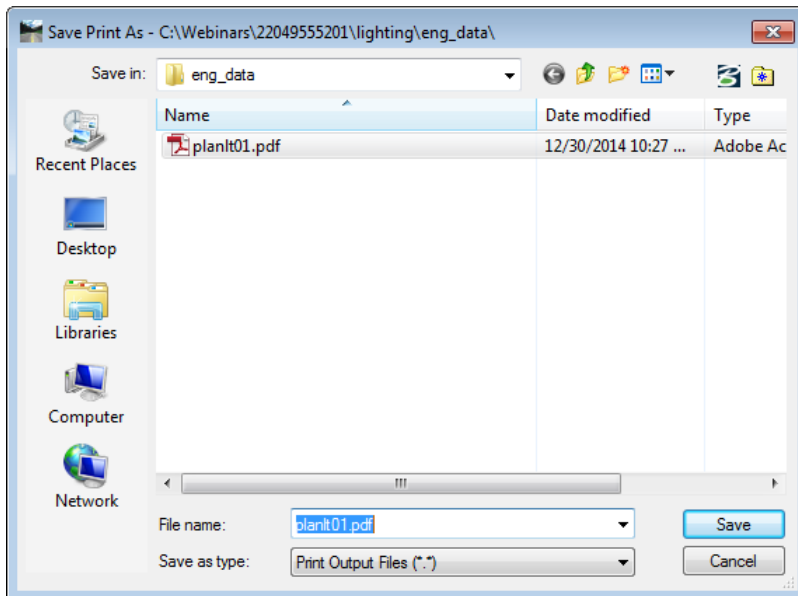

11. Toggle **Off** *Constructions* and *Data Fields*. This turns off the red plot border.

Note   It is also recommended to toggle off the Print Border and Fence Boundary options.

12. Click **OK**.

13. Click the **Print** icon ![print icon]. This opens Save Print As. The *default location* for the plot files is the **eng_data** folder under the discipline folder as shown below.


14. Click **Save**. This accepts the default file name and location and creates the *(.pdf)* file.

15. From FDOT Menu, select **Standard > Explore Current Working Directory**.

16. Browse to the **eng_data** folder.

17. Double click on **Planlt01.pdf**. This will open the plot file for review.

18. Close the *(.pdf)* file.

➢ *Using a Print Style with the Print Dialog (Part 2)*

Using Print Styles can drastically reduce the number of steps needed to complete this process because all settings can be predefined in the Print Style. For example, with a properly defined Print Style you would simply have to:

1. Open *Planlt01.dgn* in the *lighting* folder.

2. Click on the **Print** icon on the MicroStation Standard tool palette or select **File > Print**. This opens the Print dialog and sets a default plot driver.

3. Select **Settings > Apply Print Style** from the menu.

4. Choose the appropriate *Print Style* from the list and click **OK**.
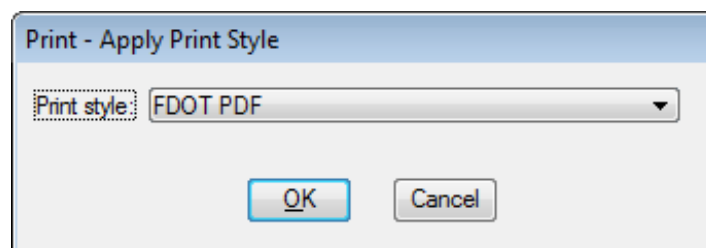
5. Click the **Print** icon ![print icon] and view the *(.pdf)* file as before.

➤ *Save the Print Settings with the Print Dialog (Part 3)*

Now that you have all of the print settings set the way you want them, it is a good idea to save these settings for future use and also for other users to use so that you get consistent output.

1. In the Print dialog, select **File > Save Print Definition File**. This opens Save Print Definition File dialog.
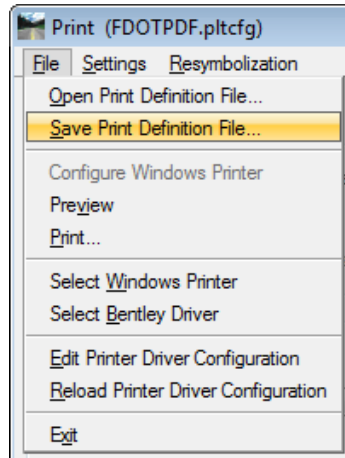
2. In Save Print Definition File, enter **LIGHTING-PDF**. The file name will default to the MicroStation file name.
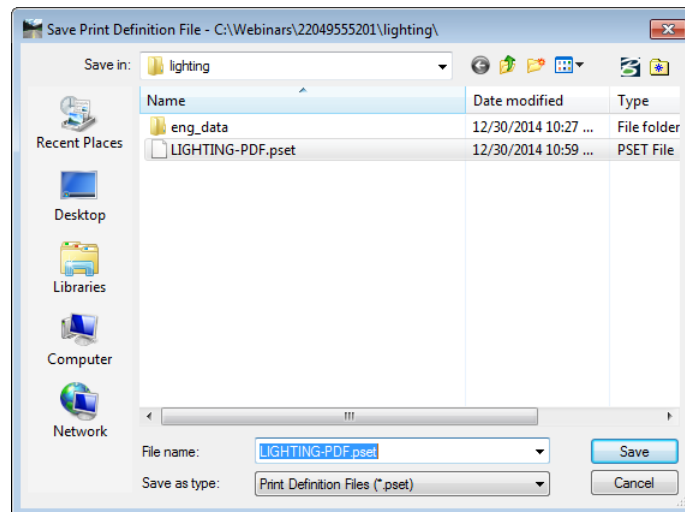
3. Click **Save**. This saves the settings to an *(*.pset)* file that can be used in subsequent plot jobs.

4. In the Print dialog, select a different *plot driver* from **File > Select Bentley Driver**. Select **fdotprinter.pltcfg** for this example. Notice how all of the plot settings have changed.

5. In the Print dialog open the *Definition* file you saved**, File > Open Definition File**.

6. In the *lighting* folder, select **LIGHTING-PDF.pset**.

7. Click **Open**. This will set all of the *plot preferences* the way you saved them for the *(.pdf)* plot.

8. Close the Print dialog.

PRINTING IN MICROSTATION FROM THE PRINT ORGANIZER

PRINT ORGANIZER INTERFACE
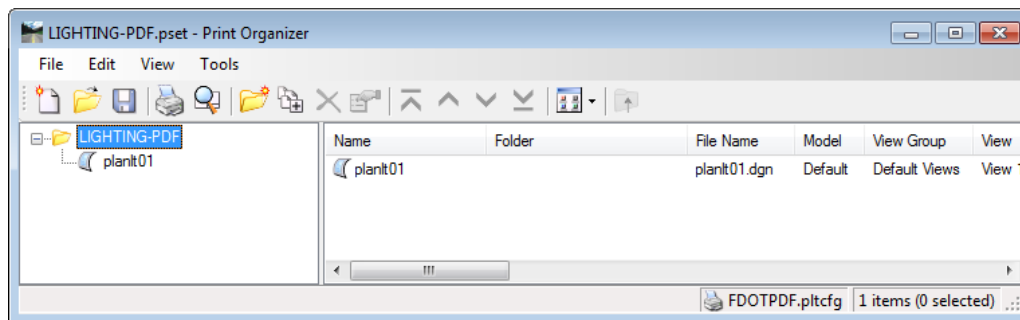
Before executing a print job from Print Organizer, open a file not included in the print job. Monitor the Message Center. If any problems occur with the print job there will be a notification in the Message Center which might assist the designer in correcting the problem.

With Print Organizer, the designer can import old JOB *(*.job)* files. However, when the old JOB file is opened with Print Organizer, a conversion will take place. The old Batch Specification file must be available. A warning dialog will appear indicating the plot border level (PlotBorder_dp) is not in the reference files. After the conversion, the designer can organize the Print Set as desired and save the configuration as a Print Set file *(*.pset)*.

A *Print Set* file *(*.pset)* is easily created and saved with the following steps:

1. Open a design file (preferably one (1)) not included in the print job.

2. Select **File > Print Organizer** from the MicroStation menu.

3. In Print Organizer, select **File > Save As.**

4. Define the *name* and *location* to store the *(.pset)* file before clicking the **Save** button.

Once the Print Set file has been created, the designer can begin to organize the Print Set by adding folder and/or Files. To add folders to the Print Set, use one (1) of the following methods:

- Select **File > Add Folder to Set** from menu.

- Click **Add Folder to Set** icon.

- Right-click parent folder and select **Add Folder to Set.**

To add filers to the Print Set, use one (1) of the following methods:

- Select **File > Add Files to Se**t from menu.

- Click **Add Files to Set** icon.

- Right-click parent folder and select **Add Folder to Set.**

- Drag and Drop from Windows Explorer.

When adding folders and/or files to the *Print Set*, the designer is given the opportunity to apply a Print Style or to manually define options (Manually Specified Options).
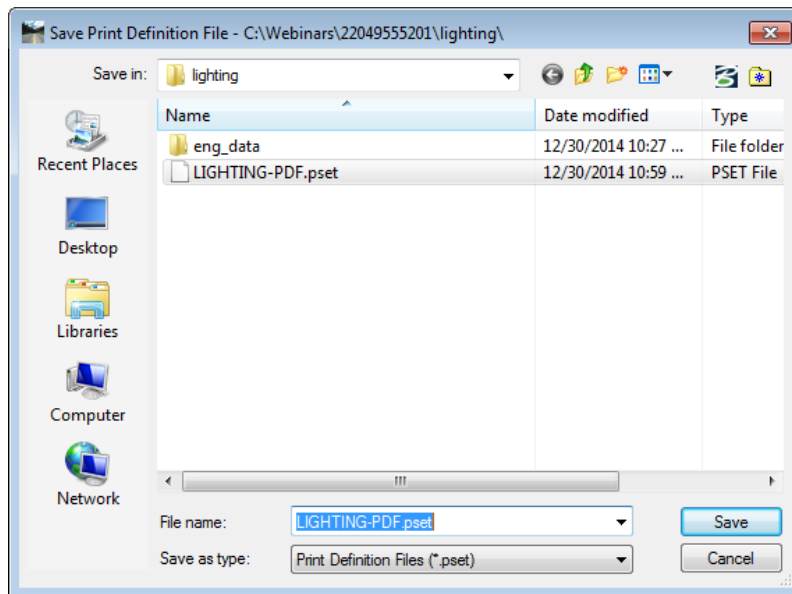
The Print Organizer interface can be arranged to easily view the information the designer is most interested in. From the Print Organizer Menu Bar, navigate to **View > Show/Hide Columns** or Right-click on the column header in the Print Definition pane to turn on or off the column headers. The designer can also drag columns to position the columns in the order that best suits them.

FILE MENU

The *File* menu contains options for creating new Print Set Files, editing existing Print Set Files, setting the Default Print Definition Name (Named Expression), setting the Output File Names (Named Expression), adding folders to the Print Set, adding files to the Print Set, previewing the plot, selecting and editing a plot driver/configuration, and reloading the driver after editing.
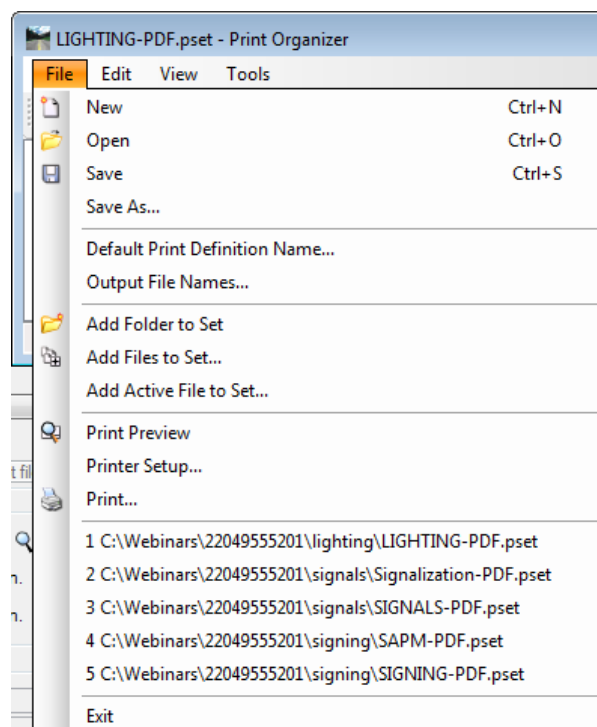
EDIT MENU

The *Edit* menu contains options for Renaming Print Definitions, Deleting Print Definitions, organization of the Print Definitions, selecting Print Definitions, managing the paths of selected or all Print Definitions, Setting Preferences and Properties of the selected Print Definition(s).
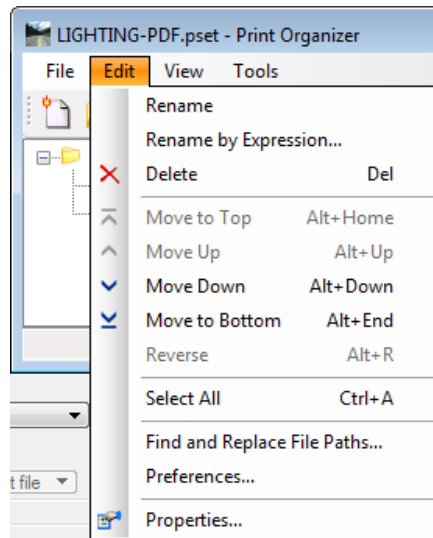
VIEW MENU

The *View* menu contains options for Toggling on/off the Toolbar, Toggling on/off the Status Bar, Show/Hide Columns in the Print Definition pane, whether to view Details or List in the Print Definition pane and View all Print Definitions or Show Print Definitions in Groups.
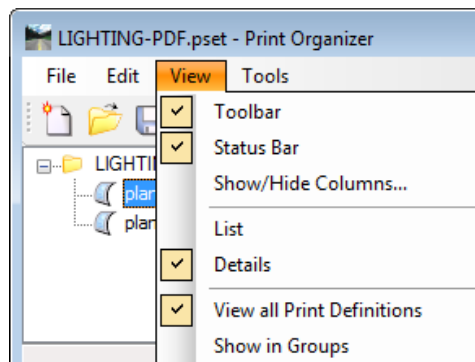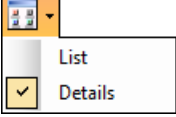
TOOLS MENU

The *Tools* menu contains options for loading the *Define Print Styles* tool or loading the *Apply Print Styles* tool.
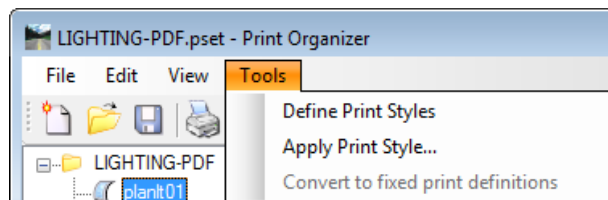
PRINT ORGANIZER ICONS

 - New Print Set file.

 - Opens an Existing Print Set file

 - Saves a Print Set file

 - Prints to the selected Printer or File

 - Opens the print preview window

 - Add Folders to Print Set

 - Add Files to Print Set

 - Deletes Print Definition(s)

 - Opens the Print Definitions Properties dialog

 - Moves selected Print Definition to the Top of the list in the Print Definition pane

 - Moves selected Print Definition up one position in the Print Definition pane

 - Moves selected Print Definition down one position in the Print Definition pane

 - Moves selected Print Definition to the Bottom of the list in the Print Definition pane

 - Toggles between Details and list view for the Print Definition

PRINT DEFINITION PROPERTIES

MAIN TAB

When a *Print Definition* is selected, navigate to Edit > Properties from the Print Organizer menu or click the Properties icon on the Print Organizers toolbar to access the Print Definition Properties. The Main tab is much like setting up the General Settings (Area and Resymbolization) Printer and Paper Size (Paper) and Print Scale and Size (Layout) sections on the Print dialog.
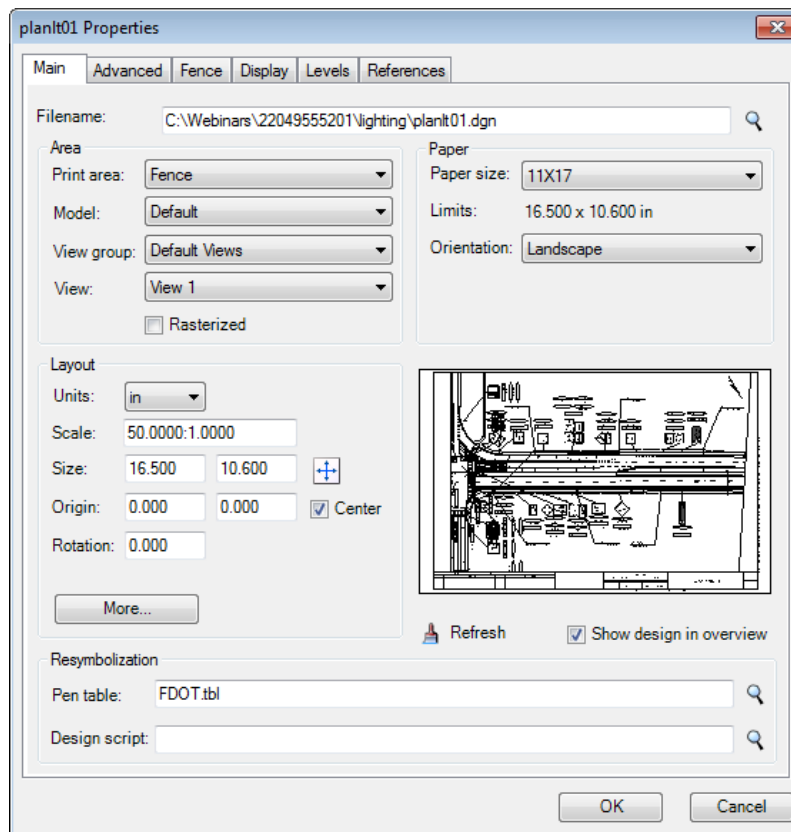
- **Area** – section is where to select the plot area and which view to plot from.

Note   Rasterized option was added in MicroStation XM and is available in V8i. This option should only be used for rendering or transparencies because it generates larger plot files and is a considerably slower process. When this option is selected MicroStation will convert both raster and vector data into a single raster image and send it to the printer. The Rasterized toggle option is not for printing files. PDF files created with the Rasterized option turned on will not have searchable text.

- **Paper** – section is for selecting the paper size and whether or not to plot as Landscape or Portrait.

- **Layout** – section is where to fill in an exact plot scale, size and adjust the X Origin and Y Origin print position. How this worked is the designer placed a fence around the plot border which would calculate a scale and display it in this portion of the dialog. This odd number would have to be rounded up to the actual scale. This is now done by applying a Print Style to the Print Definition.

Note   The designer has the ability to adjust the X Origin and Y Origin print position. These values change the position on the print on the paper.

The right hand side of the Print dialog shows the preview.

- **Resymbolization** – shows the path and which pen table is loaded. The Pen Table is applied by the Print Style.

<u>ADVANCED TAB</u>

When a Print Definition is selected, navigate to *Edit > Properties* from the Print Organizer menu or click the *Properties* icon on the Print Organizers toolbar to access the *Print Definition Properties*. The *Advanced* tab is used to specify workspace, color options and raster options (color/quality), update from design file and update print definition name for the selected print definition.
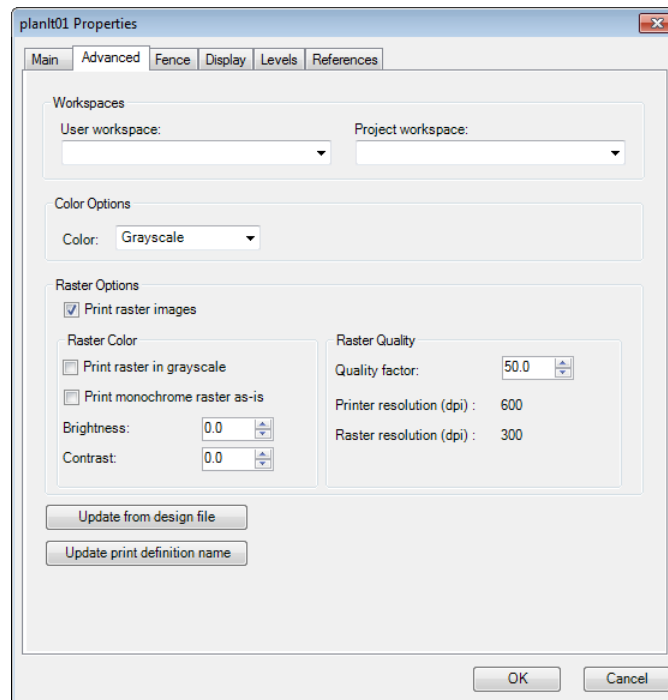
- **Workspaces** – section is where to specify a User Workspace and/or a Project Workspace.
    - *User Workspace* – determines which resource files to use based on that workspaces configuration in the user configuration file *(.ucf)*. Such resources may include fonts and line styles.
    - *Project Workspace* – essentially the project name. This could also determine which resources to use if resources such as fonts and line styles are configured in the project configuration file *(.pcf)*.

Note   Leaving these the workspace options blank allows Print Organizer to use the defaults set up by the workspace configuration.

- **Color Options** – section is for selecting Monochrome, Grayscale, or True Color from and list. The available options depend on the selected printer.

Note   When the Plot to 3D check box is selected, the color is set to True Color and cannot be changed.

- **Raster Options** – section is where to enable or disable printing all raster types such as Raster Manager Attachments and imported DGN Rasters. If the check box is toggled off the Raster Color and Raster Quality are disabled (ghosted).

Note   For more information on the Raster Option settings, please see the MicroStation help file.

- **Update from design file** – Updates the print definition with current data from the design file and its reference files.

- **Update from design file** – Updates the print definition name using the expression name specified on the Default Print Definition Name Expression dialog.

FENCE TAB

When a Print Definition is selected, navigate to Edit > Properties from the Print Organizer menu or click the *Properties* icon on the Print Organizers toolbar to access the *Print Definition Properties*. The *Fence* tab is used to specify fence creation methods for the selected print definition.
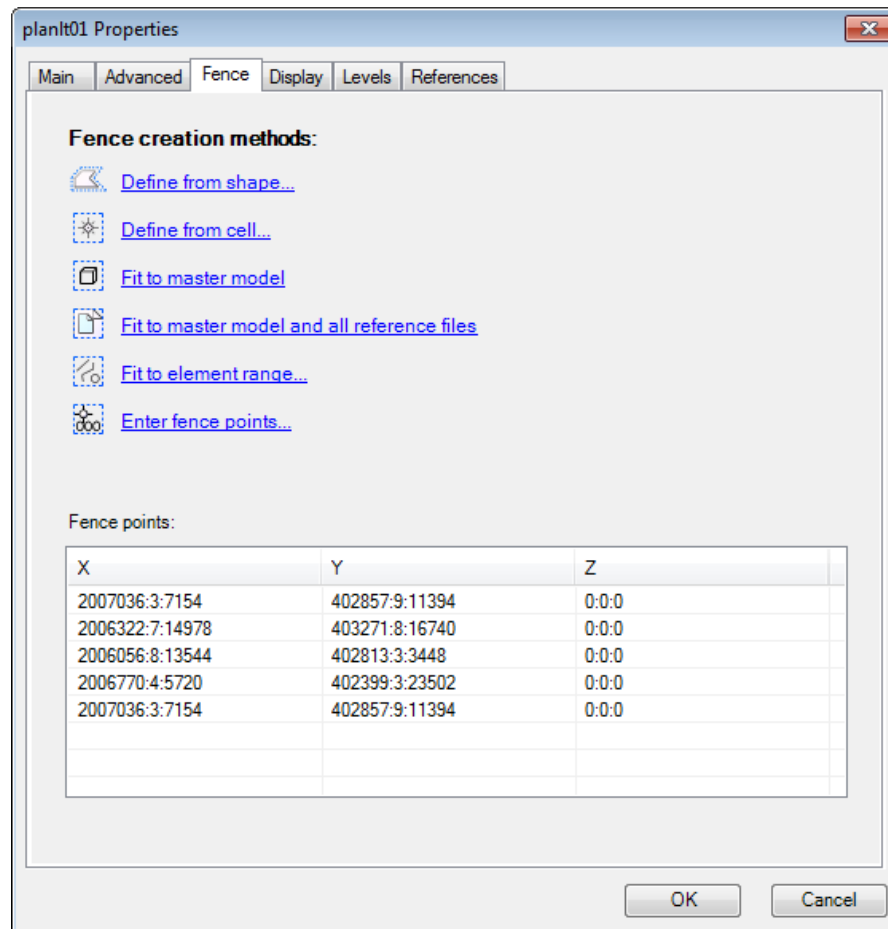
- **Define from Shape –** section is used to specify the plot area by describing shapes whose actual coordinates define the print area.

- **Define from cell –** section is used to specify the plot area by describing a cell whose actual coordinates define the print area.

- **Fit to master model –** section is automatically calculated to include every element in the master model. The fence points are displayed in the Fence points list box.

- **Fit to master model and all reference files –** is automatically calculated to include every element in the master model and all references. The fence points are displayed in the Fence points list box.

- **Fit to element range –** is used to specify the plot area by describing a fence whose coordinates are based on the range of specified elements. The fence points are displayed in the Fence points list box.

- **Enter fence points –** is used to specify the fence points that define a print area.

DISPLAY TAB
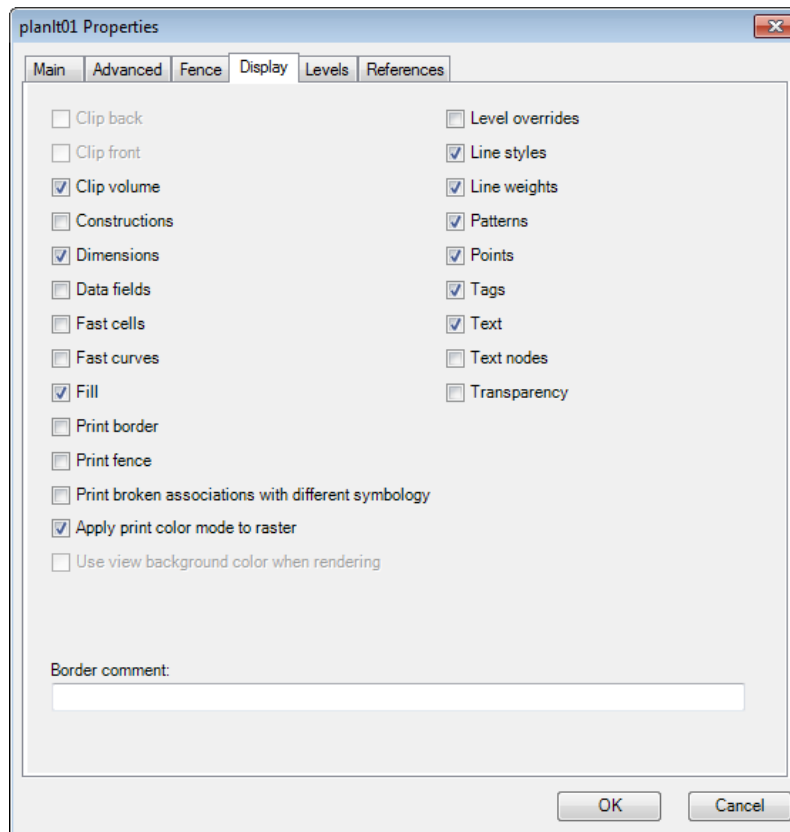
When a Print Definition is selected, navigate to *Edit > Properties* from the Print Organizer menu or click the *Properties* icon on the Print Organizers toolbar to access the *Print Definition Properties*. The *Display* tab is used to specify various display attributes for the selected print definition.

LEVEL TAB

When a Print Definition is selected, navigate to *Edit > Properties* from the Print Organizer menu or click the *Properties* icon on the Print Organizers toolbar to access the *Print Definition Properties*. The *Levels* tab is used to specify levels to control for the selected print definition.

REFERENCE TAB

When a Print Definition is selected, navigate to *Edit > Properties* from the Print Organizer menu or click the *Properties* icon on the Print Organizers toolbar to access the *Print Definition Properties*. The *Display* tab is used to specify reference files to control for the selected print definition.

PRINT PREVIEW

A *Print Preview* can be generated of a file, selection of files, or entire folder of files by either selecting *File > Print* preview from the Print Organizer Menu Bar, clicking the Print Preview Icon or Right-clicking the file name and select Print Preview.
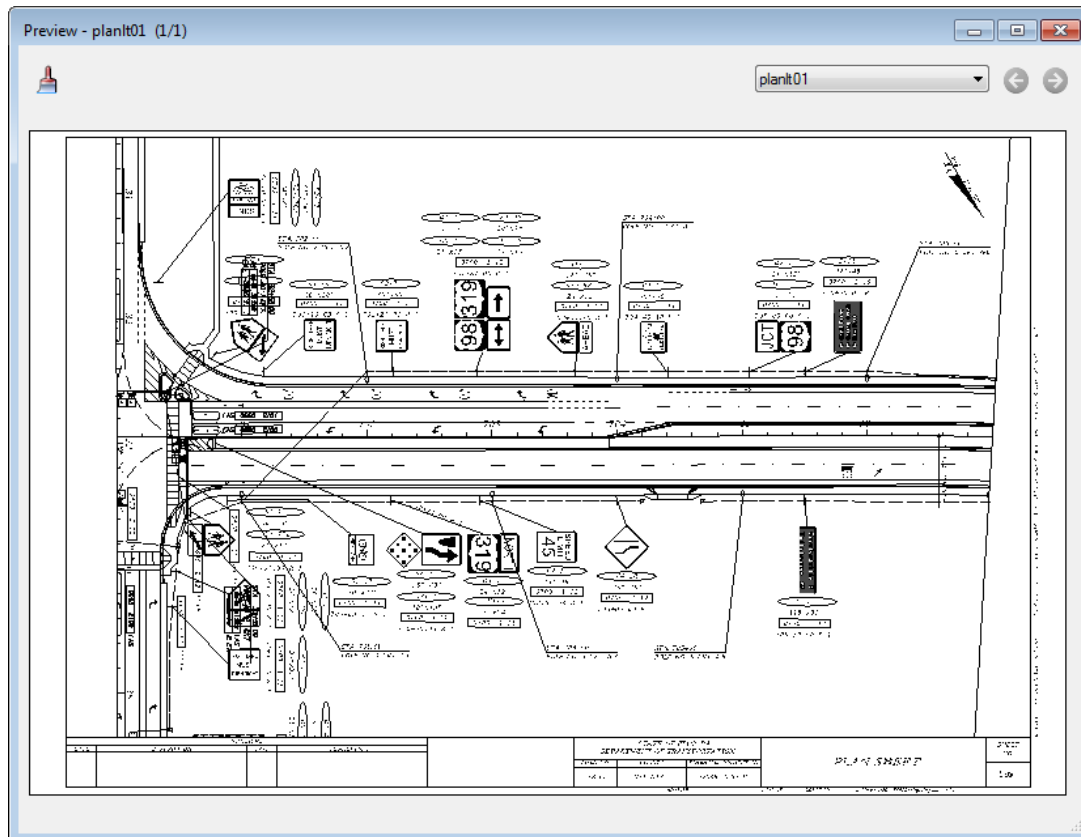
Exercise 7.2 *Printing with Print Organizer*

➢ ***Open and Modify an Existing Print Set (Part 1)***

In the previous exercise the class saved a Print Set *(*.pset)* file based on the setup of the Print Dialog. As previously mentioned in this chapter, Print Set files and Print Styles can be utilized by the Print Dialog and Print Organizer. In this exercise the student will open and make specific changes to the *LIGHTING-PDF.pset* file and save those changes to the existing Print Set file.

✓ *Open an Existing Print Set*

1. Open the *Dsgnlt01.dgn* file.

2. From the MicroStation menu, select **File > Print Organizer.** This opens the *Print Organizer* dialog.
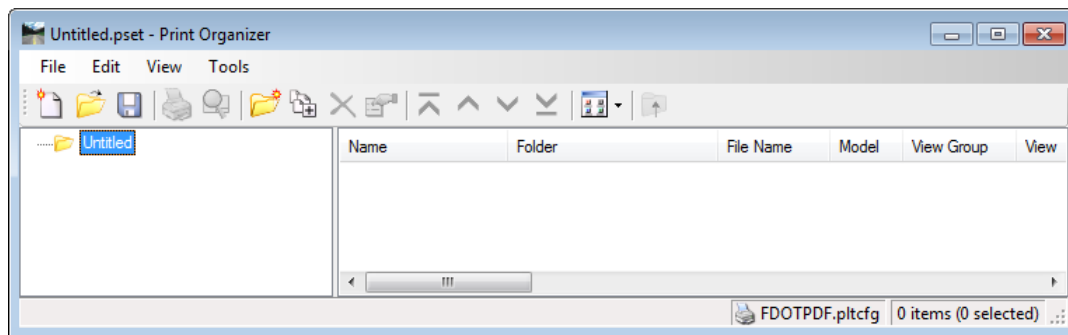
3. In the Print Organizer dialog, select **File > Open**. This opens the Open Print Set File dialog.

Note   The Open Print Set file dialog should open to the c:\e\projects\22049555201\lighting folder.

4. Select the **LIGHTING-PDF.pset** file created in the previous exercise. The *Print Set* file information will now populate the Print Organizer dialog.
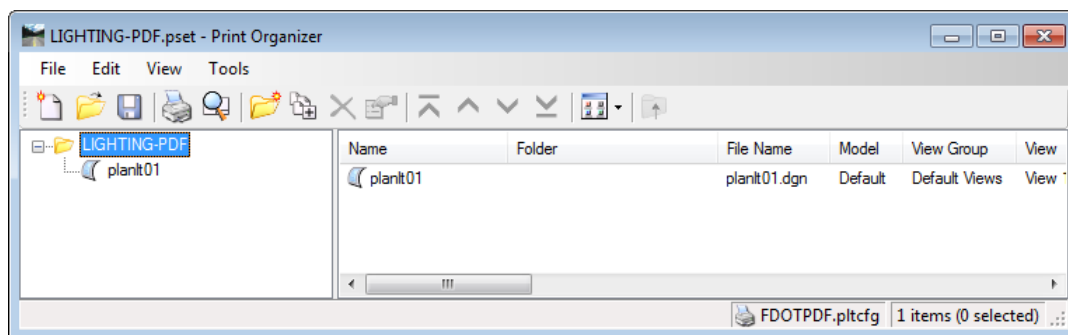
✓ *Modify an Existing Print Set – Output File name Expression*

First, the class will modify the *Output File Name Expression*. In the previous exercise the Output File was named *Planlt01.pdf*. Currently the Output File would be *001-LIGHTING-PDF.pdf*. The following steps will make Print Organizer match the previous exercises Output File.

5. From the Print Organizer menu, select **File > Output File Names**. This will open the Output File Name Expression dialog.

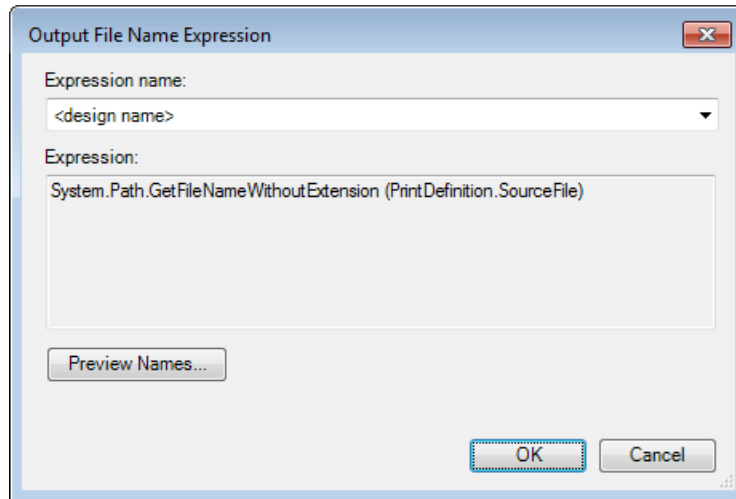6. Click the **Preview Names** button. This will open the Output File Name dialog. Notice the output file name for the default *(001-LIGHTING-PDF.pdf)*.

7. With the Output File Name dialog still open click the down arrow for the *Expression Name* list to view all the available output file name expressions.

Note   For custom Output File Name Expressions select Custom from the list.

8. The *default* is set to **<print counter>-<print set name>**.

9. From the *list* select **<design name>**.

10. Notice the *Output File Name* has changed to the correct output **Planlt01.dgn**.
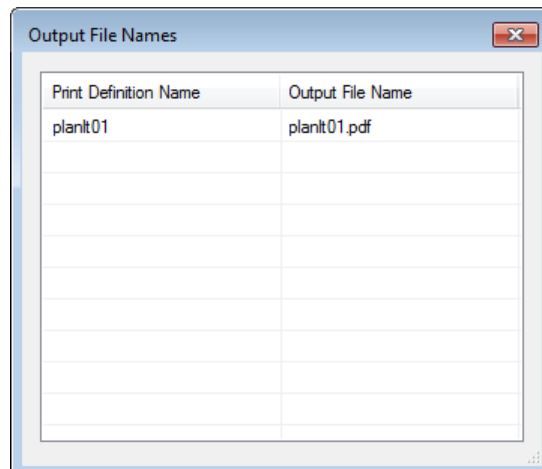
11. Close the Output File Name dialog by clicking the red **X** in the upper right hand corner of the dialog.

12. Click the **OK** button on the Output File Name Expression dialog to set the selected *Output File Expression* to the **Print Set**.

Note   All files added to this Print Set will now have the same output file name until changed by the designer.

✓ *Modify an Existing Print Set – Add Files to Set*

Now, the class will add the *Planlt02.dgn* file to the Print Set.

13. From the FDOT Menu, select **Standard > Explore Current Working Directory.** This will open Windows Explorer to the *Lighting* folder of the current project *22049555201*.

14. Select **Planlt02.dgn** in the Windows Explorer window.

15. Drag and drop **Planlt02.dgn** on the *LIGHTING-PDF Print Set* category in Print Organizer. This opens the Create Print Definitions dialog.
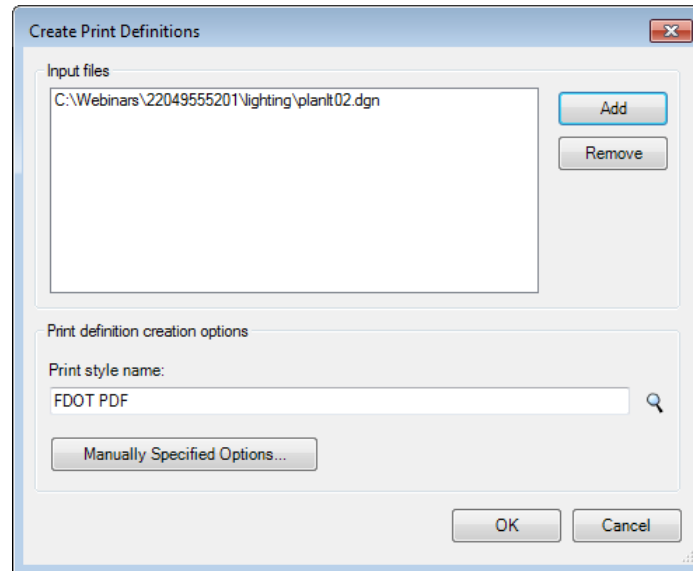
16. Click the *magnifying glass* icon next to the *Print style name* and select FDOT PDF Print Style from the list of Print Styles.

17. Click the **OK** button on the Apply Print Style dialog.
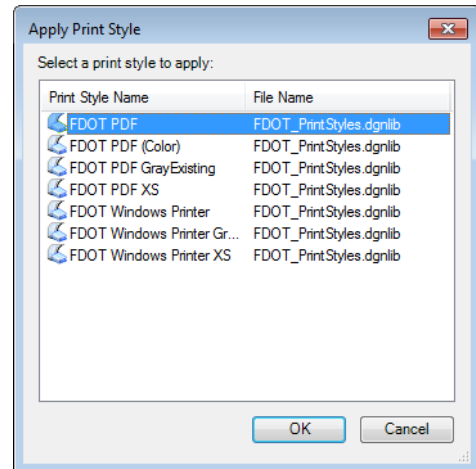
18. Click the **OK** button on the Create Print Definitions dialog. A Print Organizer Status box displays as Print Organizer runs through its process.
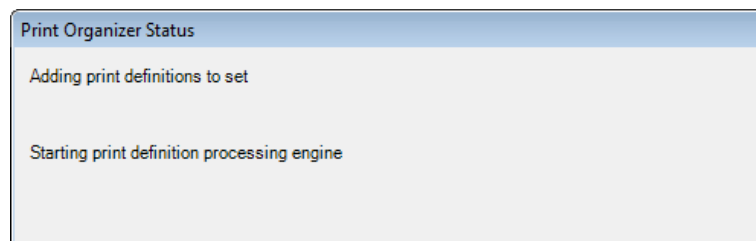
Note   By setting the Print Style from the Department delivered FDOT_PrintStyles.dgnlib (FDOT PDF), the Print Set Pen Table may revert to the standard pen table for that Print Style.

✓ *Modify an Existing Print Set – Setting Pen Table for Multiple Print Definitions*

The class will apply the previously created Pen Table to all the Print Definitions listed for the existing Print Set.

19. In the Print Definitions pane, highlight the first **Print Definition**, hold the *Shift* key on the keyboard down and click the last **Print Definition**. This will highlight all *Print Definitions* listed.
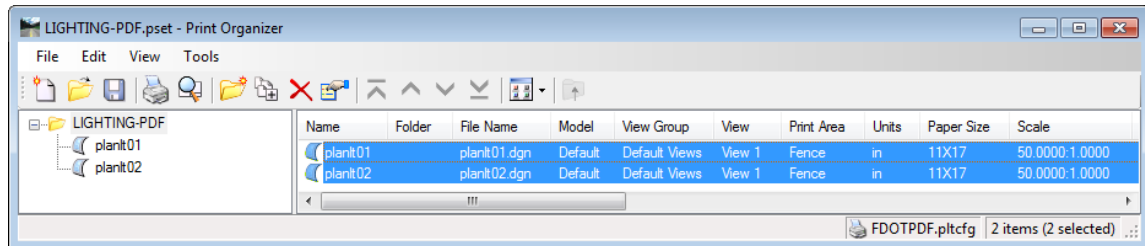

20. Right-click on the highlighted list of **Print Definitions**.
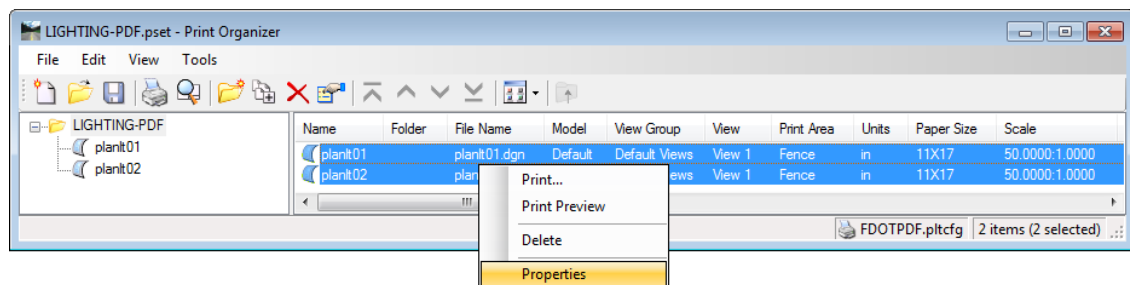
21. Select **Properties** from the contextual menu. This opens the Modify Properties dialog.

22. On the *Main* Tab, in the *Resymbolization* section of the dialog, click the *Pen Table magnifying glass*. This opens a *Windows Explorer* window to navigate to the location where the **FDOT.tbl** file is stored.

23. Navigate to the \\FDOTSS3\RESOURCES\plot folder and select the **FDOT.tbl** Pen Table

24. Click the **Open** button.
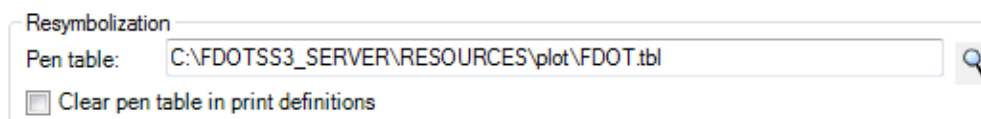
25. Click the **OK** button on the Modify Properties dialog to accept the select Pen Table for the select *Print Definitions*.

26. From the Print Organizer menu, select **File > Save** to save the changes made to the *LIGHTING-PDF.pset Print Set* file.

✓ *Modify an Existing Print Set – Preview Sheets in The Print Set*

The class will now preview the sheets in the Print Set.

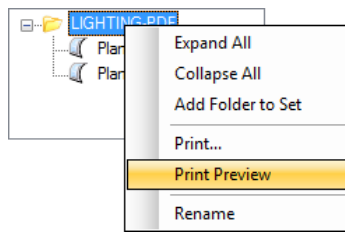27. In the *Print Set* category pane, right-click on the **LIGHTING-PDF** category (folder).

28. Select **Print Preview** from the contextual menu. This will open the Preview dialog.

29. Click the **Preview Navigation** buttons or drop down list to review the *Plan Sheets* in the Print Set.

30. Close the Preview dialog by clicking the red **X** in the upper right hand corner of the dialog.

➢ **Create New Print Set File & Print Multiple Sheets to PDF with Print Organizer (Part 2)**

In the previous exercise the class modified an existing Print Set *(*.pset)* file which was based on the setup of the Print Dialog. In this exercise the student will create a new Print Set file and print multiple sheets to PDF format through Print Organizer.

1. Continuing in the *Dsgnlt01.dgn* file. Open Print Organizer from the MicroStation menu, select **File > Print Organizer**.

2. Once Print Organizer is open, select **File > New** or click the **New** icon from Print Organizer.

3. Select **File > Save** or click the **Save** icon from Print Organizer. This opens the Save Print Set File dialog.

4. In the *File Name* field enter **LIGHTING_Plans-PDF.pset** *(.pset will be filled in by the program)*.

5. Click the **Save** button to save the **LIGHTING_Plans-PDF.pset** file to the *lighting* folder.

✓ *Creating A New Print Set – Add Folders to Set*

Now, the class will add Folders to the Print Set. This will produce a multi-sub category Print Set to allow the designer to control different Print Definitions for the multiple subcategories. Typically, this will be done for a project Print Set adding folders by discipline and sub category (folders) under the discipline categories for different intersections or side streets.

6. From the Print Organizer, right-click on the **LIGHTING_Plans-PDF** category.

7. Select **Add Folders to Set** from the contextual menu. This will pop in a new folder under the *LIGHTING_Plans-PDF Print Set* category.

8. Rename the *folder* **Keysheet**.

9. Repeat steps 6-8 for the following folders.

> Plan Sheets
> Summary of Quantities
> Pole Data
> Tabulation of Quantities
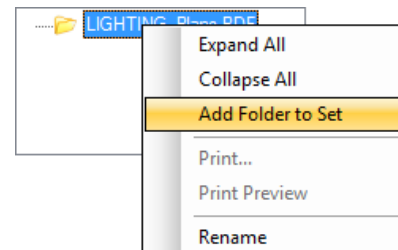
Note   If the folders are out of order (personal preference) use the arrow icons to move the folders into the desired position.

10. From the Print Organizer toolbar, click the **Save** icon.

✓ *Creating A New Print Set – Add Files to Set: Drag & Drop*

Now, the class will add Files to the Print Set.

11. From the FDOT Menu, select **Standard > Explore Current Working Directory**. This will open Windows Explorer to the *Lighting* folder of the current project *22049555201*.

12. Select the **Keyslt01.dgn** in the Windows Explorer window.

13. Drag and drop **Keyslt01.dgn** on the *LIGHTING_Plans-PDF Print Set > Keysheet* category in Print Organizer. This opens the Create Print Definitions dialog.

14. Click the *magnifying glass* icon next to the *Print Style Name* field to select a **Print Style**.

15. Select the **FDOT PDF Print Style** from the list of *Print Styles*.

16. Click the **OK** button on the Create Print Definitions dialog. A Print Organizer Status box displays as Print Organizer runs through its process.

17. Click the **OK** button on all Print Organizer 'Warning' box. (If any appear.)

✓ *Creating A New Print Set – Add Files to Set: Menu Selection*

18. In the Print Organizer, highlight the **Summary of Quantities** folder in the *Print Set* category pane.

19. From the Print Organizer toolbar, select the **Add Files to Set** icon or navigate to **File > Add Files to Set** from the Print Organizer menu. This opens the Create Print Definitions dialog.

20. Click the **Add** button to open the Select Files dialog.

21. Navigate to the *Lighting* folder and select the **Cesslt01.dgn** file.

22. Click the **Done** button.

23. Click the *magnifying glass* icon next to the *Print style name* field to select a **Print Style**.

24. Select the **FDOT PDF Print Style** from the list of *Print Styles*.

25. Click the **OK** button on the Create Print Definitions dialog. A Print Organizer Status box displays as Print Organizer runs through its process.

26. Repeat steps 1-8 for the remaining folders placing the appropriate sheet file in the associated folder.

✓ *Modify an Existing Print Set – Setting Pen Table for Multiple Print Definitions*

The class will apply a Pen Table to all the Print Definitions listed for the existing Print Set.

27. Highlight the ***LIGHTING_Plans-PDF*** folder.

28. Select **View > View all Print Definitions**.

29. In the *Print Definitions* pane, highlight the **Keyssp01.dgn** file.

30. Hold the *Shift* key on the keyboard down and click the **Gswksp01.dgn** file.

31. Right-click on the highlighted list of ***Print Definitions***.

32. Select **Properties** from the contextual menu. This opens the Modify Properties dialog.
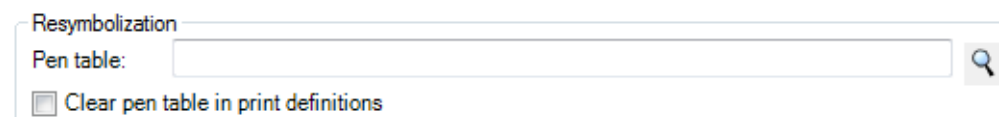
33. On the *Main* tab, in the *Resymbolization* section of the dialog, click the *magnifying glass* icon next to the *Pen Table* field. This opens a Windows Explorer window to navigate to the location where the **FDOT.tbl** file is stored.

34. Navigate to the **\\FDOTSS3\RESOURCES\plot** folder.

35. Select the **FDOT.tbl** *Pen Table*.

36. Click the **Open** button.

```
Resymbolization
Pen table:    C:\FDOTSS3_SERVER\RESOURCES\plot\FDOT.tbl          🔍
☐ Clear pen table in print definitions
```
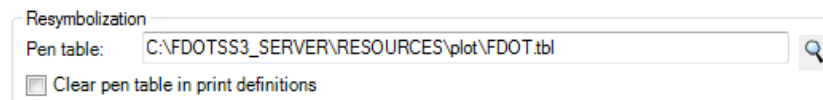
37. Click the **OK** button on the Modify Properties dialog to accept the select Pen Table for the select **Print Definitions**.

38. From the Print Organizer menu, select **File > Save** to save the changes made to the **LIGHTING_Plans-PDF.pset** *Print Set* file.
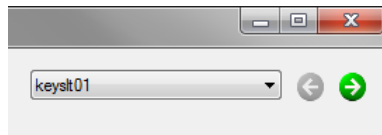
✓ *Modify an Existing Print Set – Preview Sheets in The Print Set*

The class will now preview the sheets in the Print Set.

39. In the *Print Set* category pane, right-click on the **LIGHTING_Plans-PDF** category (folder).

40. Select **Print Preview** from the contextual menu. This will open the Preview dialog.

41. Use the **Preview Navigation** buttons or drop down list to review the **Plan Sheets** in the *Print Set*.

```
[  keyslt01  ▼ ]  ⬅  ➡
```

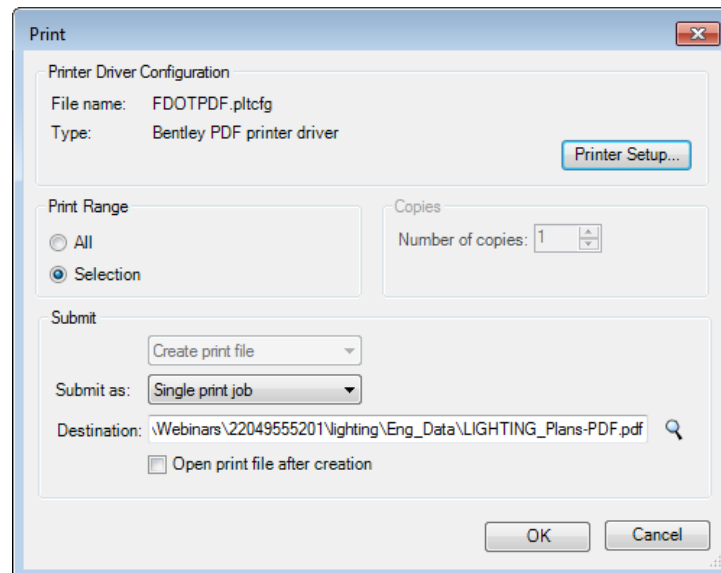42. Close the Preview dialog by clicking the red **X** in the upper right hand corner of the dialog.

✓ *Modify an Existing Print Set – Preview Sheets in The Print Set*

The class will now plot all sheets in the Print Set to PDF through Print Organizer.

43. In the *Print Set* category pane, right-click on the **LIGHTING_Plans-PDF** category (folder).

44. Select **Print** from the contextual menu or click the **Print** icon 🖨 for the Print Organizer toolbar. This will open the Print dialog.

```
Print                                                    [  X  ]
┌─ Printer Driver Configuration ──────────────────────────┐
│  File name:    FDOTPDF.pltcfg                            │
│  Type:         Bentley PDF printer driver                │
│                                    [ Printer Setup... ]  │
└──────────────────────────────────────────────────────────┘
┌─ Print Range ──────────────┐  ┌─ Copies ─────────────────┐
│  ○ All                     │  │  Number of copies: 1 ⇅   │
│  ● Selection               │  │                          │
└────────────────────────────┘  └──────────────────────────┘
┌─ Submit ─────────────────────────────────────────────────┐
│              [ Create print file         ▼ ]              │
│                                                           │
│  Submit as:  [ Single print job          ▼ ]              │
│  Destination: \Webinars\22049555201\lighting\Eng_Data\LIGHTING_Plans-PDF.pdf  🔍 │
│               ☐ Open print file after creation           │
└───────────────────────────────────────────────────────────┘
                                    [  OK  ]  [ Cancel ]
```

45. Click the **OK** button at the bottom of the Print dialog.

46. From the FDOT Menu, select **Standard > Explore Current Working Directory**. This will open Windows Explorer to the *Lighting* folder of the current project *22049555201*.

47. Double click the **eng_data** folder and open and review the **LIGHTING_Plans-PDF.pdf** file.