

OpenRoads

Creating Subsurface Utility Models

Existing and Proposed Utilities

Presenter: Robert Garrett, PE

Bentley Systems, Incorporated
685 Stockton Drive
Exton, PA 19341
www.bentley.com

Workshop: *Evaluating and Creating Subsurface Utilities*

Before you Begin: 3

Chapter 1: Review Interface and Base Data 4

 Exercise: Open and review Base Files..... 4

 Exercise: Review Subsurface Utility Engineering Tools 5

Chapter 2: Model Existing Utilities 7

 Exercise: Model Multiple Utilities using a Utility Filter 7

Chapter 3: Model utility line from an External Data Source 11

 Exercise: Configure Model Builder Settings 11

 Exercise: Creating the Utility Model From Model Builder..... 16

Chapter 4: Model a Proposed utility line with Trench 19

 Exercise: Model a Proposed utility line 19

 Exercise: Redesign the Profile 22

Appendix: Configuration Variables used in this course..... 27

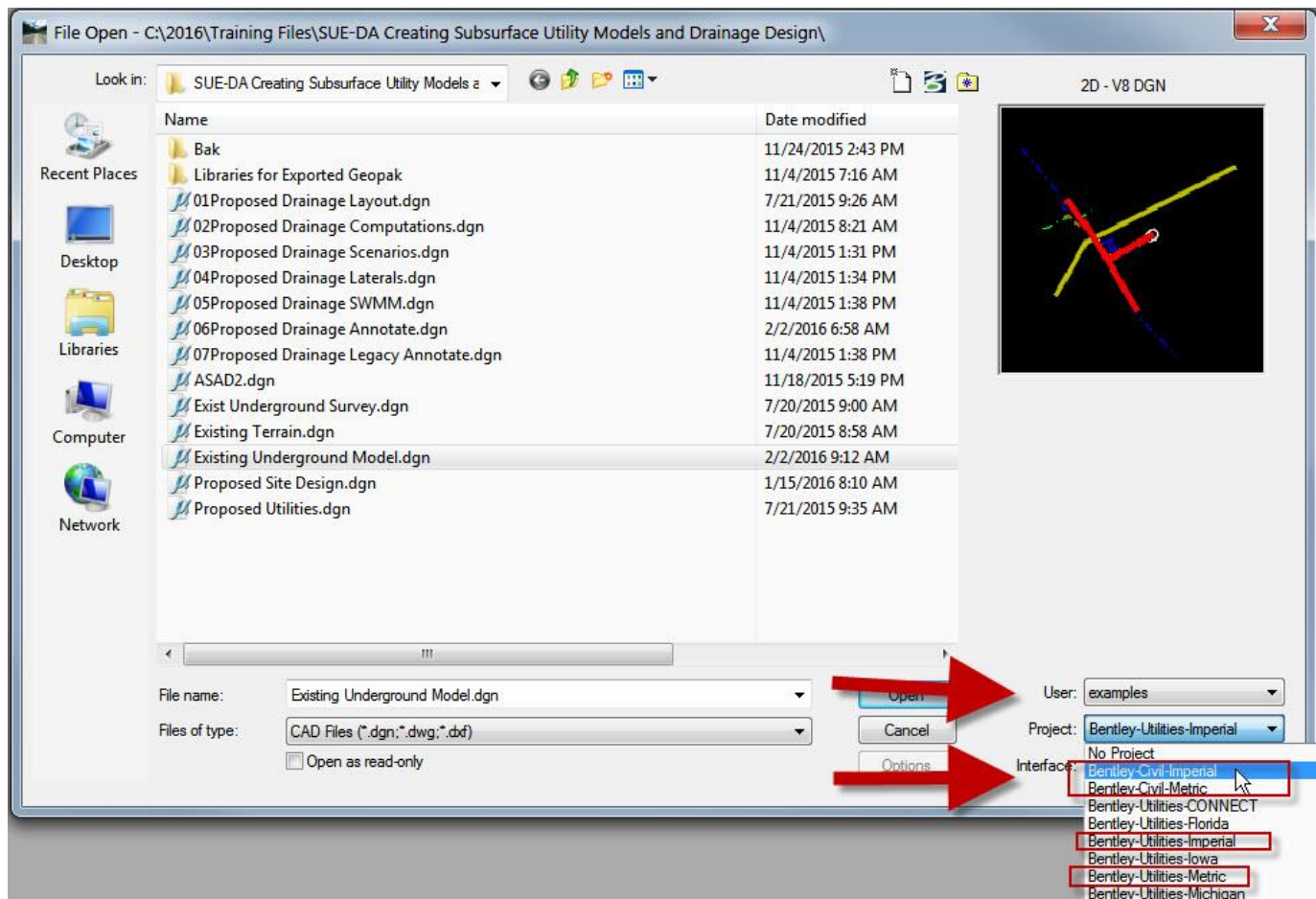
BEFORE YOU BEGIN:

This training session will utilize the sample workspaces provided in the OpenRoads products to facilitate loading of necessary libraries used in the exercises. For this exercise you may use any of the following sample workspaces: Bentley Civil Imperial, Bentley Utilities Imperial.

There are two follow-up course to this one which use the same dataset and cover drainage design.

Subsurface Utility Models (Drainage Analysis) covers the layout and design of drainage network on this project.

Subsurface Utility Models (SWMM Drainage Analysis, Conflict Detection, Annotation) covers some advanced drainage topics, conflict detection and annotation workflows.



Course Overview:

In this hands-on laboratory you will learn to use the OpenRoads Subsurface Utility Engineering tools to:

- Create a model of existing utilities from an external data source.
- Create a model of existing utilities from surveyed data using utility filters.
- Create a model of proposed utilities and trench.

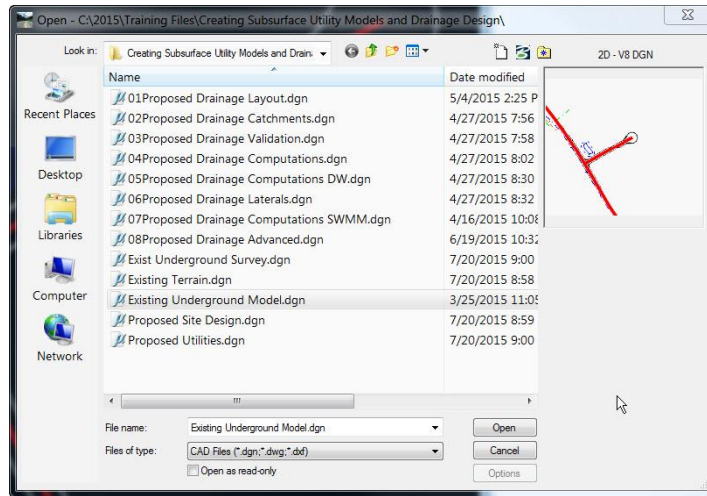
All exercises in this manual are product independent. You may use any OpenRoads product in the GEOPAK, InRoads or MXROAD family.

CHAPTER 1: REVIEW INTERFACE AND BASE DATA

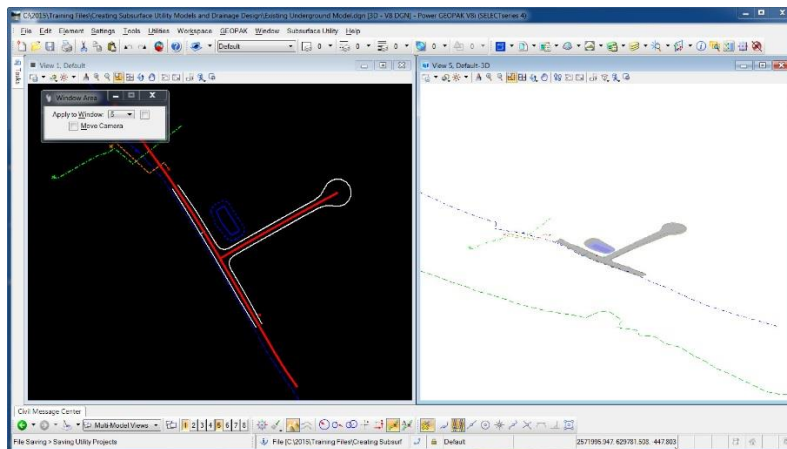
Lesson Objective: In this lesson, we will review the location and function of the Subsurface Utility Engineering tools and review the data which will be used for a start point of our utilities model. This reference data consists of an existing ground surface, and surfaces designed in a road or site workflow.

EXERCISE: OPEN AND REVIEW BASE FILES

1. Instructor will guide you which product to use and the exact path of the data.



2. Open Existing Underground Model.dgn

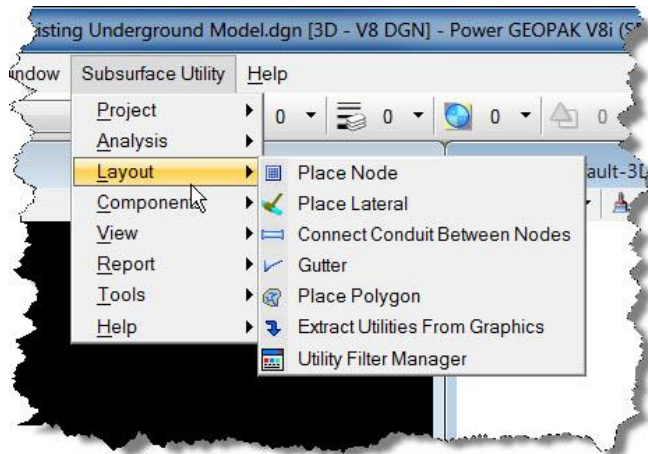


3. DGN files contains:

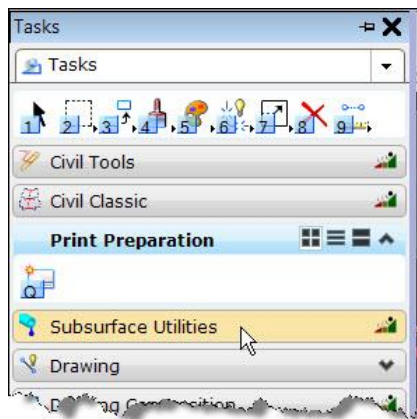
- a. In 2D model:
 - i. Referenced geometry for a road design project
 - ii. Referenced survey information for existing utilities.
- b. In 3D model
 - i. Referenced existing ground terrain.
 - ii. Referenced road design and pond.
 - iii. Referenced survey information for existing utilities

EXERCISE: REVIEW SUBSURFACE UTILITY ENGINEERING TOOLS

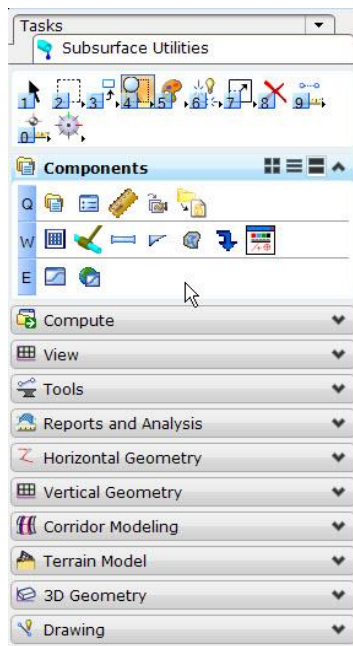
1. The Subsurface Utility tools are located in the top menu.



2. They are also located in Task Navigation, which is where we will reference them in remainder of this document.



3. If you have used Bentley Subsurface Utility Engineering in previous versions then these locations will look familiar to you. There are simply more tools available now to add drainage design functionality.



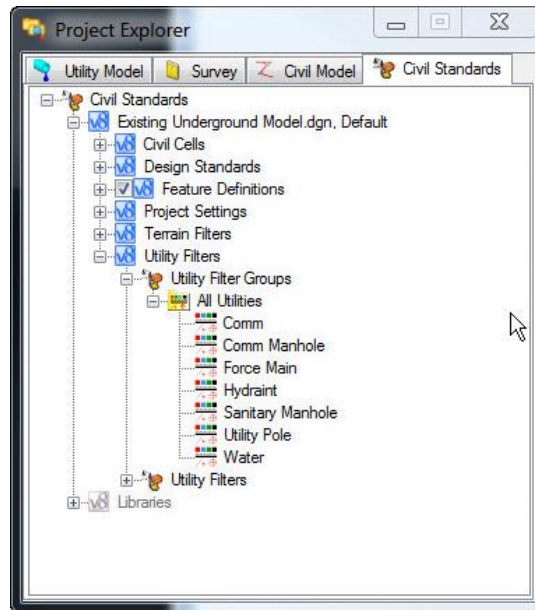
4. Within task navigation you will find the following tools:
 - a. Components – these are tools related to layout and configuration of utility models and drainage models
 - b. Compute – these tools are related to configuring and calculating hydraulic runs.
 - c. View – These tools are related to configuring and viewing various results of the modeling process. For example, opening different views (profile and cross-section), configuring tools tips and custom attributes, queries and other similar.
 - d. Tools – these are various miscellaneous tools such as conflict detection.
 - e. The remaining task panels are duplicates of various OpenRoads tools presented in the utility task panel for convenience of workflow.

CHAPTER 2: MODEL EXISTING UTILITIES

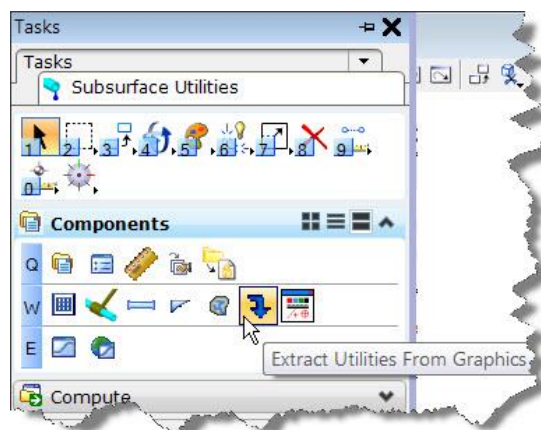
Lesson Objective: In this lesson, we will create a model of multiple existing features in one step using graphic filters. The source features are contained in a reference file and are mix of survey points and survey lines. Using this method, the Subsurface Utility modeler can model many utility lines in only a few steps.

EXERCISE: MODEL MULTIPLE UTILITIES USING A UTILITY FILTER

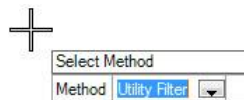
1. Continue in same file **Existing Underground Model.dgn**.
2. Open Project Explorer > Civil Standards tab > Utility Filters branch. You will note that there are several different filters configured. These could have been created in a dgnlib file, but in this case, we made them directly in the DGN where the utilities will be modeled.



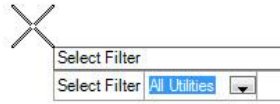
3. Start the Extract Utilities From Graphics command.



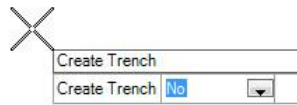
4. Set method to Utility Filter



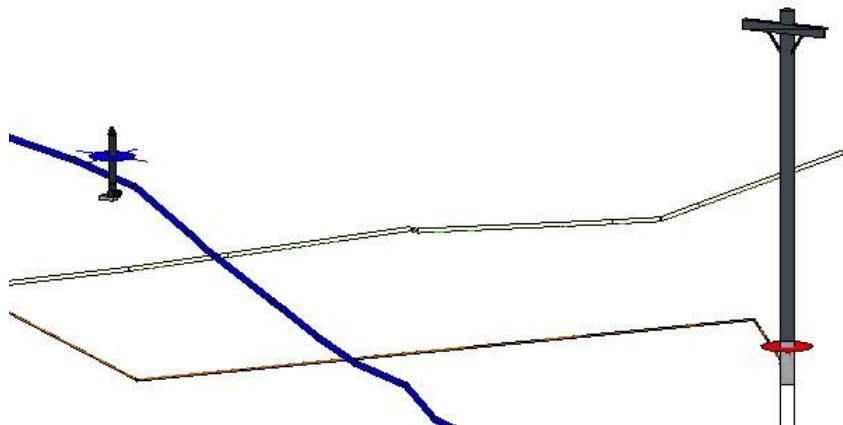
- Then choose the filter group named “All Utilities” in the dialog. In this command you can use a single filter such as the filter for water lines or you can use a filter group which provides the ability to search for many filters at once. The filter group named “All Utilities” contains all the filters to extract water lines, fire hydrants, sewer lines, power poles and more.



- There is no prompt to pick a terrain model because all the survey data is in 3D and our filters are configured to use the element elevations.
- Create trench = No.



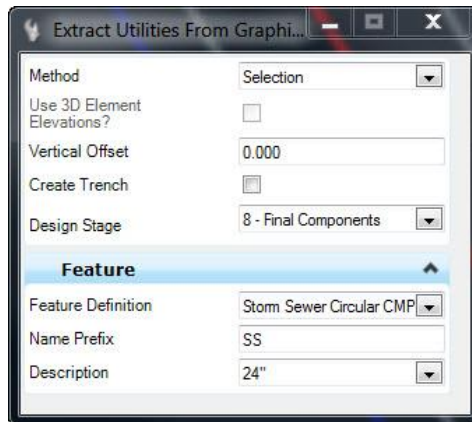
- Many various utility lines in the DGN should have been found and modeled.



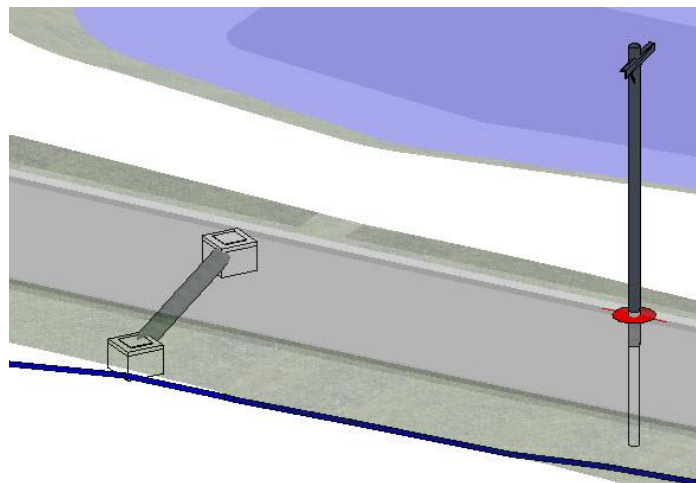
- The instructor will walk you through some review of the model and the filters.
- Notice in the center of the project there is a surveyed existing pipe which was not modeled. This is because our filter group did not have a filter for these pipes. From the survey notes, we know this is a 24” corrugated metal pipe. We can create a model using the same command but use the Selection method.

11. Start the Extract Utility From Graphics Command.

- a. Method = Selection
- b. Use 3D Element Elevations = Yes
- c. Vertical Offset = zero
- d. Create Trench = No
- e. Feature Definition = Storm Sewer Circular CMP
- f. Description = 24"

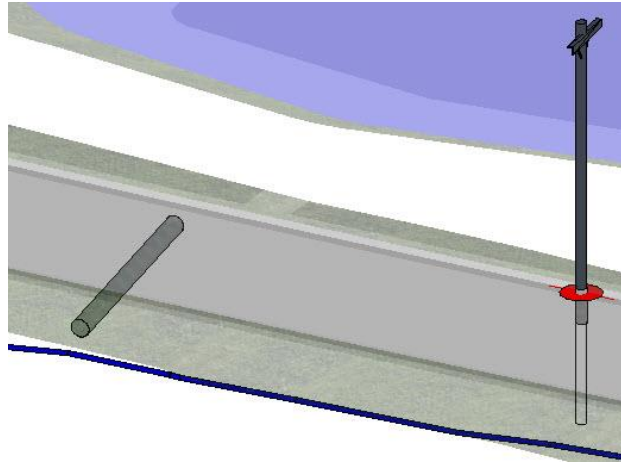


12. Select the surveyed pipe and enter prompts as above.

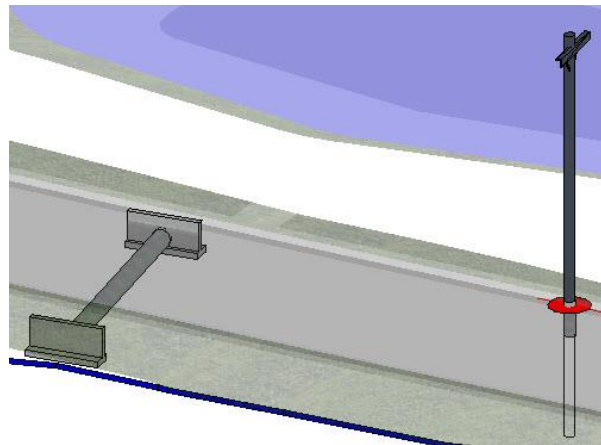


13. You will see that the pipe is modeled.

14. You will also see generic nodes modeled at the ends. These nodes exist because every conduit must have nodes defined at the ends. When the feature definition is not defined then a hard coded generic like you see is used. Since our active level is Default, we can easily turn off these generic nodes.



15. Or we can assign a headwall feature definition to the generic nodes.



WHAT WE HAVE LEARNED:

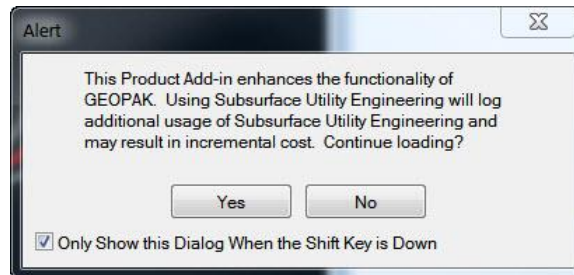
- Using utility filters, we can model many different types and sizes of utilities at once.
- Utility filters can be created in advance in libraries, or on the fly as needed in the active dgn file.
- Extract Utility From Graphics command can use filters individually or filter groups to create the utilities.
- Using utility filters, you can create a model of a drainage network from 3D surveyed nodes and pipes.

CHAPTER 3: MODEL UTILITY LINE FROM AN EXTERNAL DATA SOURCE

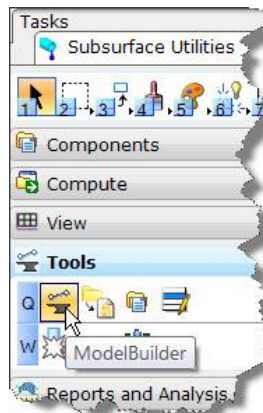
Lesson Objective: In this lesson, we will create a model of an existing gas line using a data supplied to us from the gas line owner.

EXERCISE: CONFIGURE MODEL BUILDER SETTINGS

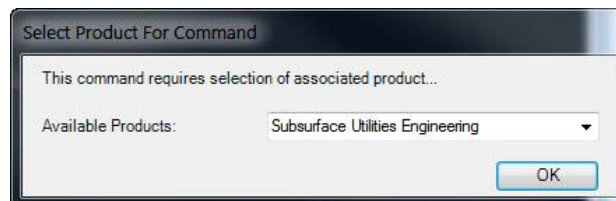
1. Continue in same file, **Existing Underground Model.dgn**
2. We will need to activate a Subsurface Utility Engineering license for this exercise because we will be importing external data into the SUE data fields. **Tools > Product Addins > Subsurface Utility Design and Analysis > Activate Subsurface Utility Engineering**. Answer yes to the warning dialog.



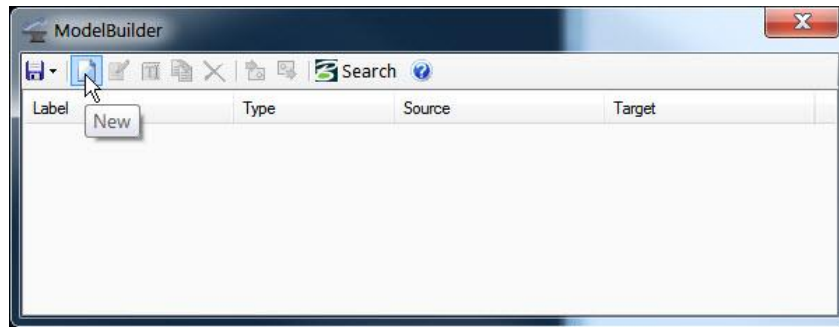
3. Start the Model Builder tool from **Tasks > Subsurface Utilities > Tools**



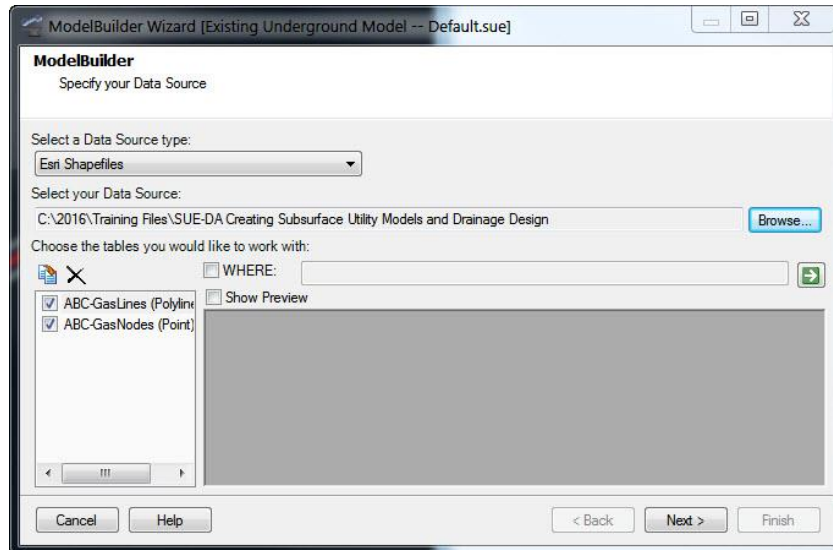
4. Choose Subsurface Utility Engineering from the drop down list. There are two embedded data schemas in the DGN file. One is for SUE information and the other is for hydraulic information. We will be writing SUE data.



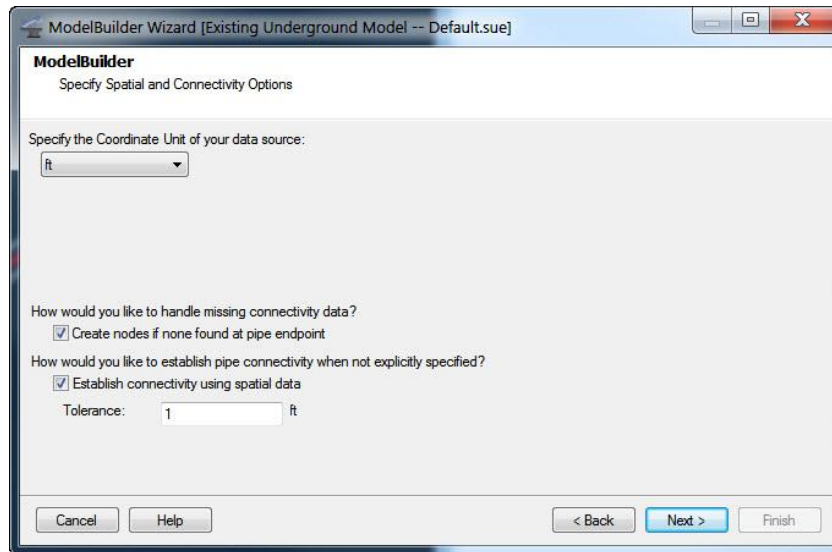
5. The Model Builder list of settings will likely be empty, but if not then just ignore any entries left over from previous use.



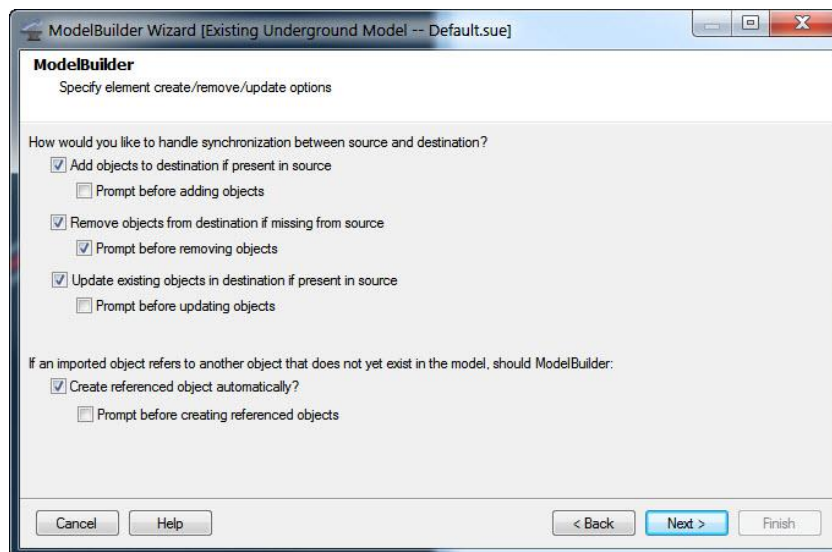
6. Model Builder is used to communicate data into and out of external database sources. This is done by mapping tables and fields in the external source with the utility types and attributes in the utility model. We will define the mapping for a SHP file which we will use in this exercise.
7. Click the New button to start the setup. And the setup wizard will open. Set data source to Shapefiles and select the files named ABC-GasLines and ABC-GasNodes. These are files supplied to us by the utility owner.



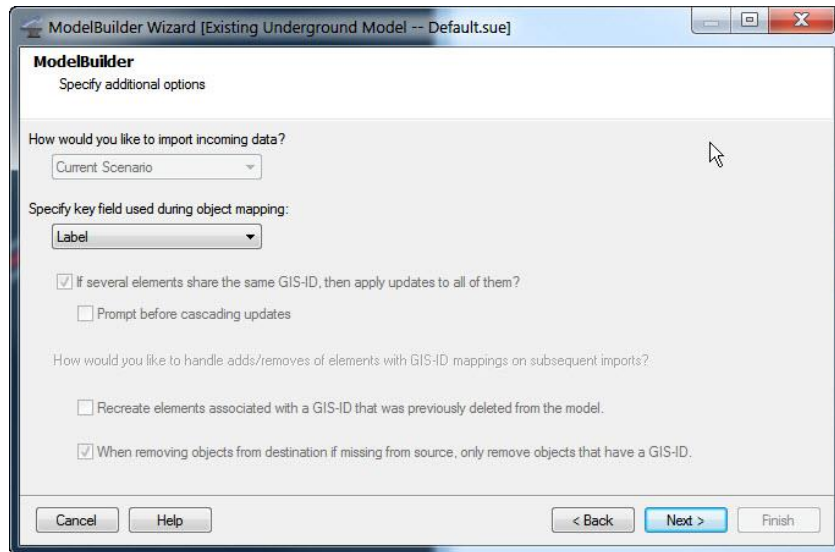
8. Click Next and on the next page set the units of the source data, which is feet. Also set the check boxes to create end nodes as needed and establish connectivity.



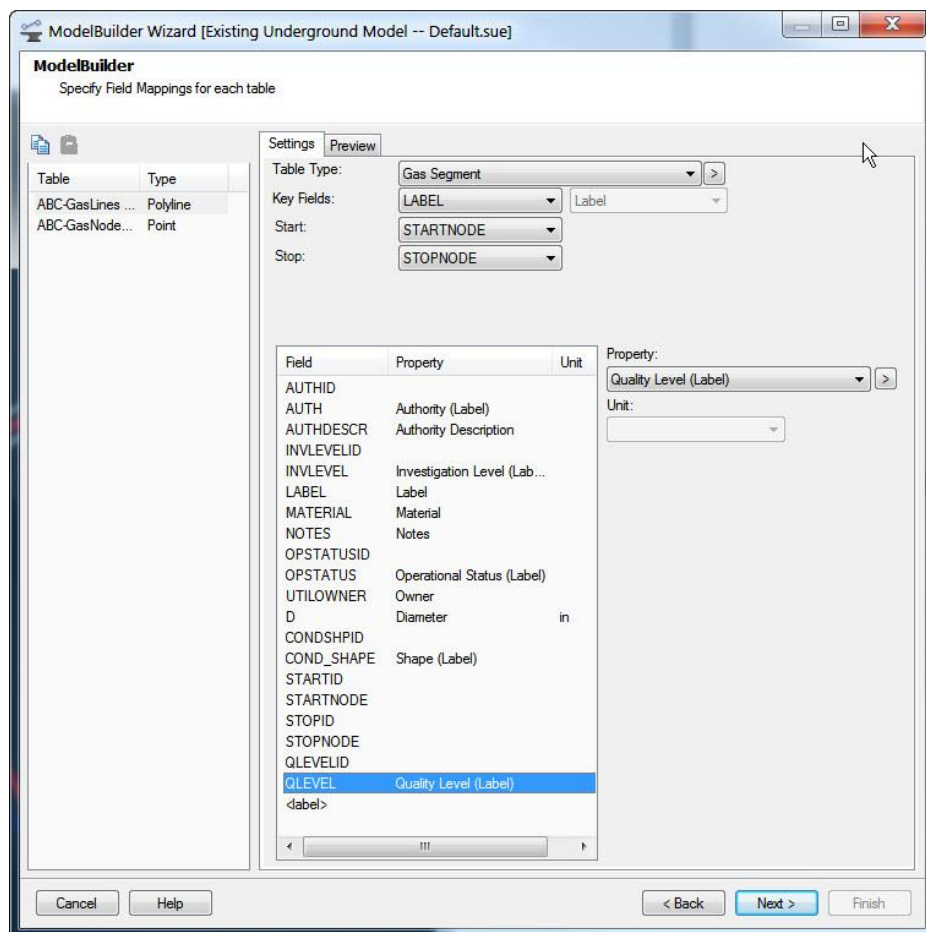
9. Click Next. On the next page define the settings as shown below. These settings define what should occur when synchronizing data between the utility model and the external data.



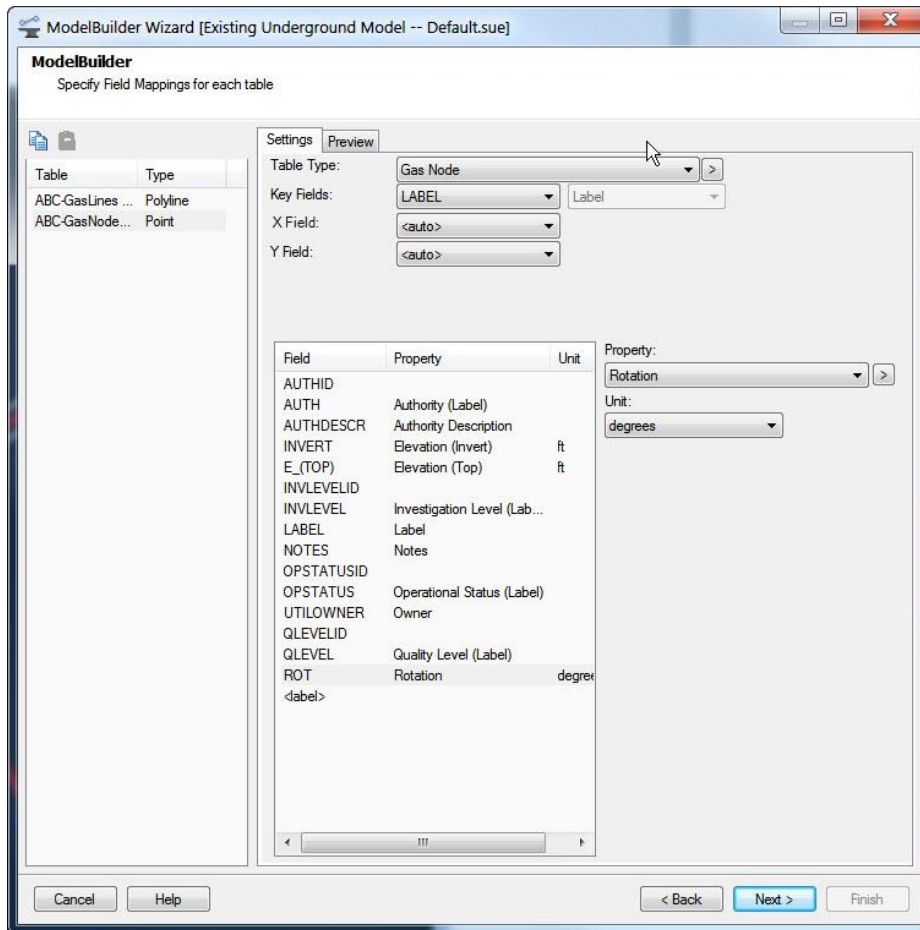
10. Click Next. Set the key field to use Label. The key field is the sometimes referred to as Primary Key and is the unique field in the source data.



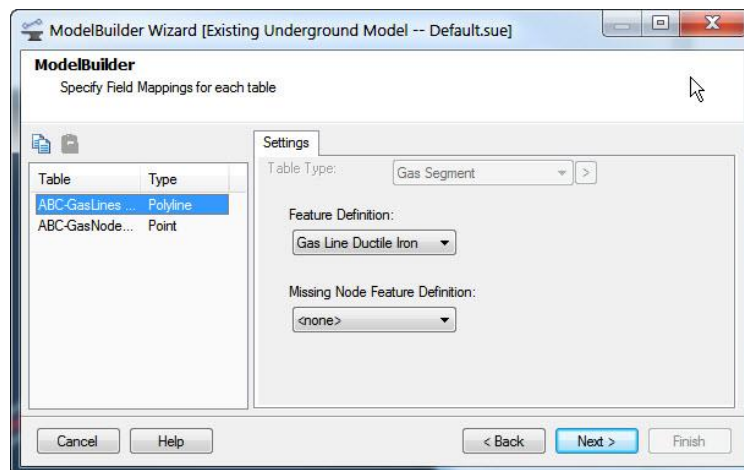
11. Click Next. On the next page is where the table and field mapping occurs. The source tables are listed in the column left. Select the table ABC-GasLines and define the attribute mapping as shown below.



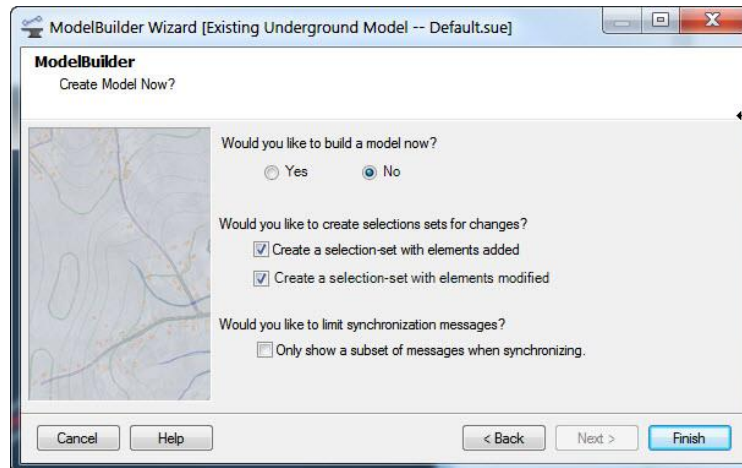
12. Then select ABC-GasNodes and define the mapping as shown here.



13. Click Next. On this page we assign a feature definition to each table. Assign feature definition Gas Line Ductile Iron for the lines and Gas Valve to the nodes.



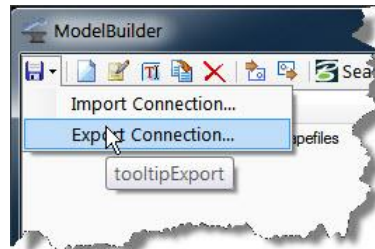
14. Click Next. On the last page of the wizard, define as below.



15. Click Finish.

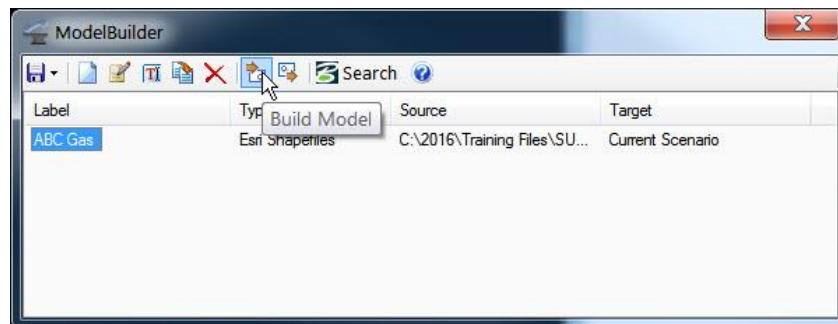
16. The settings are now shown in the Model Builder list and will have the name of DGN file. I renamed it to ABC Gas as shown at start of next exercise.

17. The settings we just configured can be saved for reuse or sharing with others. Select the ABC Gas entry and in the Model Builder menu click on Export Connection



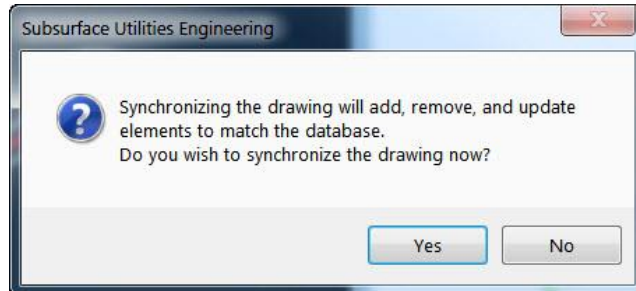
EXERCISE: CREATING THE UTILITY MODEL FROM MODEL BUILDER

1. Continue in same file, **Existing Underground Model.dgn**
2. Open Model Builder tool, if you closed it from previous exercise.

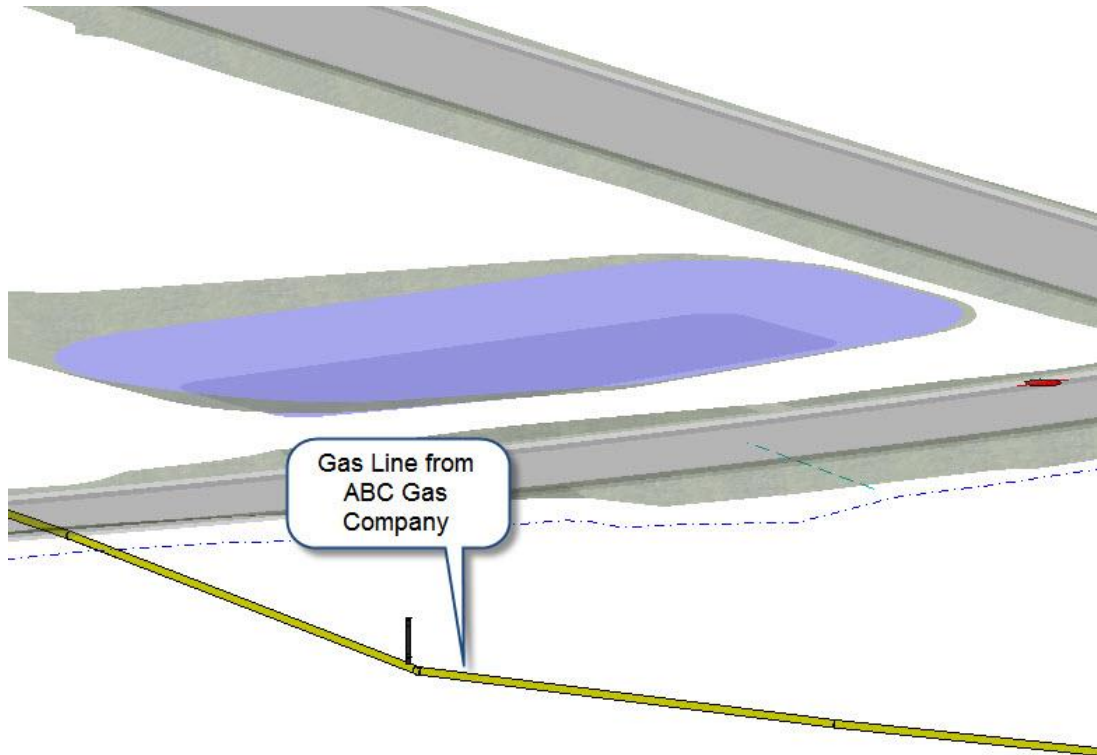


3. As seen in image above, Select the ABC Gas settings we made in previous exercise.
4. In the Model Builder menu click on Build Model.

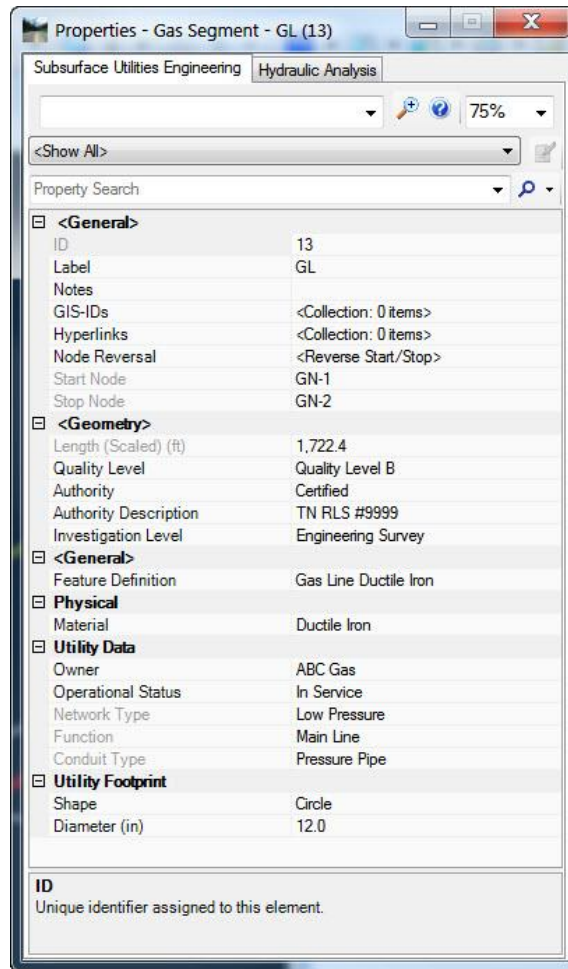
5. There will be a message box which you can review and close. The messages here are telling us that the start node and stop node for the gas line were not found and were created during import.
6. Close the Model Builder dialog. You will get a message to synchronize the DGN graphics and model with the data that was just imported. Answer Yes.



7. You will now have a gas line shown in the DGN file which crosses our roadway.



8. Select the gas line, and on the context toolbox open the utility property panel, which we will discuss in class.

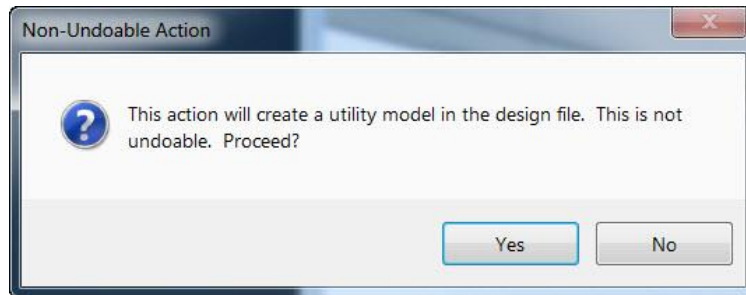


CHAPTER 4: MODEL A PROPOSED UTILITY LINE WITH TRENCH

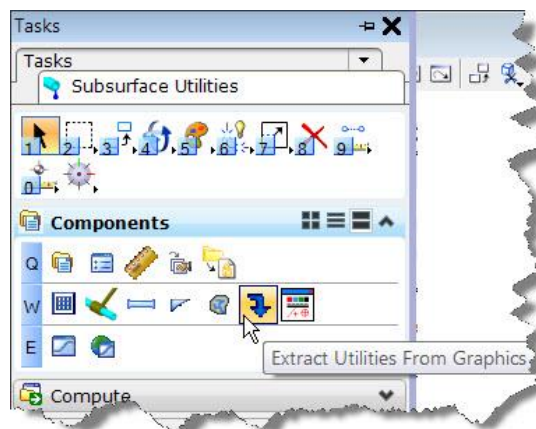
Lesson Objective: In this lesson, we will create a model of a proposed water line using a geometric layout and design profile.

EXERCISE: MODEL A PROPOSED UTILITY LINE

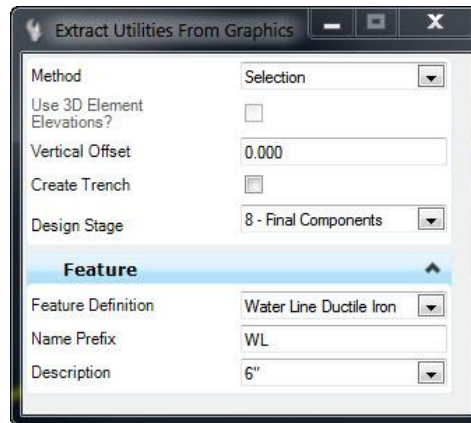
1. Open **...Proposed Utilities.dgn**. On left and right sides of the residential road are alignments for new proposed water and sewer lines. Select the blue geometry and you will recognize that this alignment is created with OpenRoads Geometry tools.
2. Start the Extract Utilities From Graphics command. You will get a prompt to create a utility model.



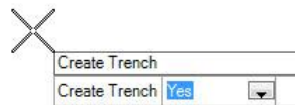
3. Answer Yes to this prompt and then start the Extract Command again. What is happening is the creation of a utility database embedded in the DGN file.



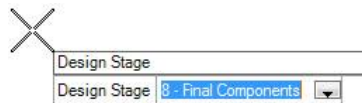
4. This is same command we used in previous exercise.



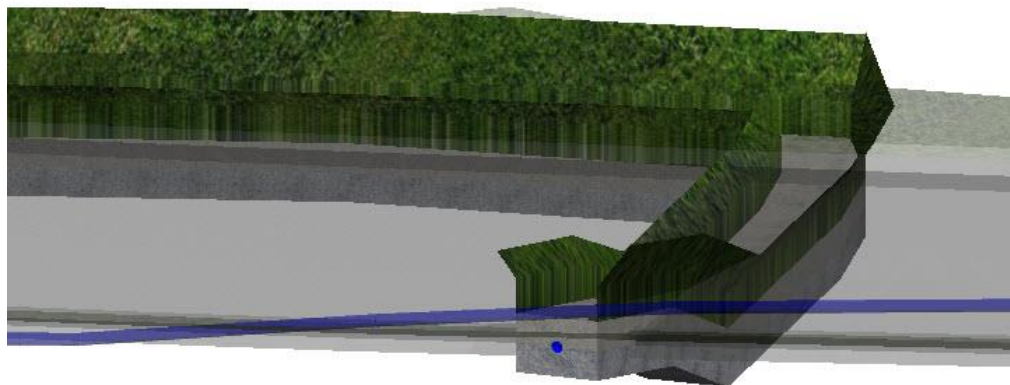
5. The next steps will be identical to what you did in chapter 2 except:
- Set Feature Definition to Water Line Ductile Iron.
 - Set Size/Description to 6"
 - Method = Selection
 - Pick the blue alignments. Reset to stop picking.
 - Pick existing ground terrain model for elevation reference.
 - Vertical Offset to -4.00ft (-1.2m)
6. Set Create Trench to Yes



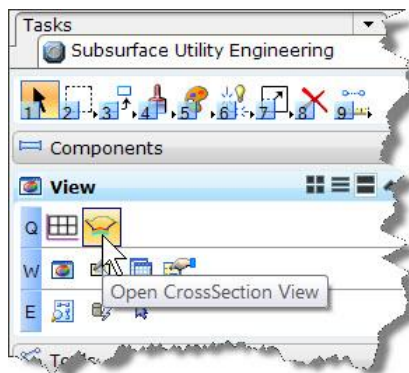
7. Set Design Stage to Final Components



8. Design stages work identically as in OpenRoads corridors. Subsurface Utility Engineering makes a corridor in order to model the trench.

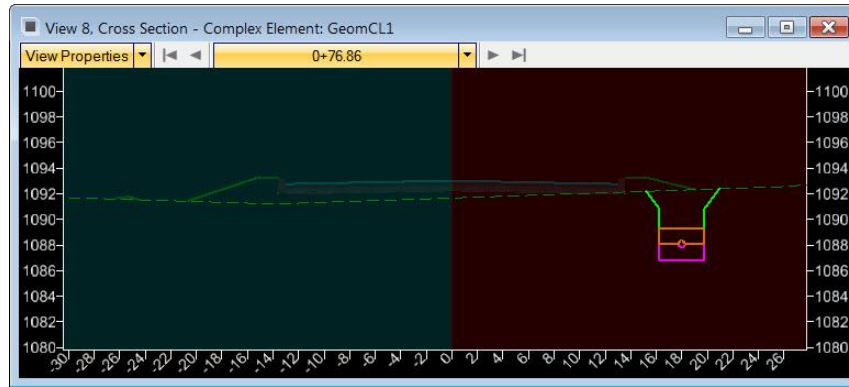


9. We can further check our design by reviewing cross-sections.
10. Click Open Cross Section View in tasks.



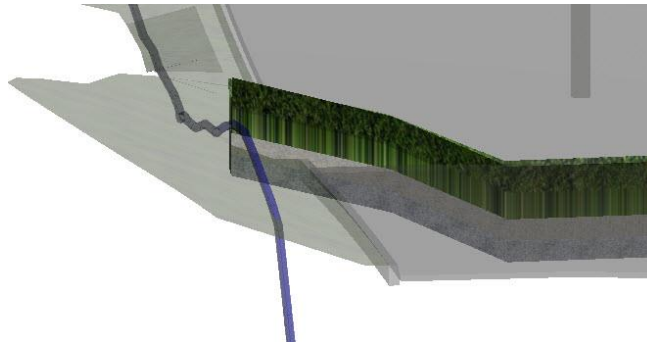
11. Follow the prompts:
 - a. Locate Corridor or Alignment – pick the red side road centerline
 - b. Left and Right offset – enter 30ft or dynamically locate a width
 - c. Start Station – about 0+50
 - d. Interval – 10 ft
 - e. Then open the cross-section in view 8

12. At the beginning, you will note the water line and trench on the left

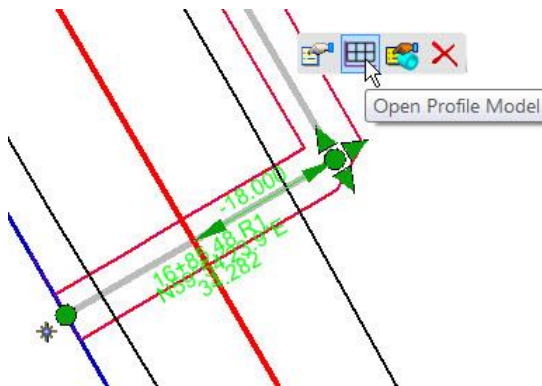


EXERCISE: REDESIGN THE PROFILE

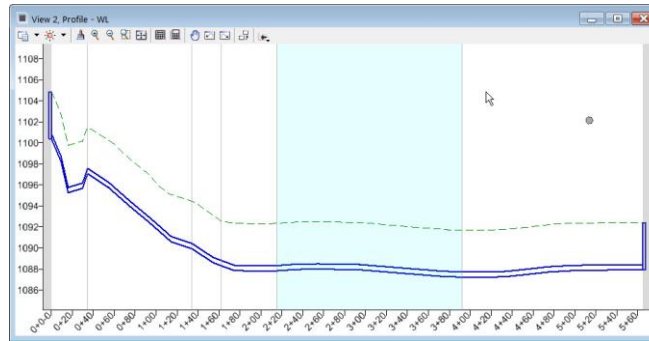
1. If you look closely at the beginning of the proposed waterline, you will note that the assumed depth is insufficient where the water line crosses the road.



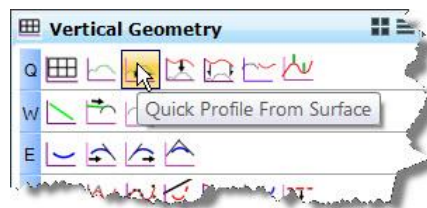
2. We need to redesign a portion of the profile to make sure we are underneath the future cut of the road design. Select the waterline and select Open Profile on the context toolbox.



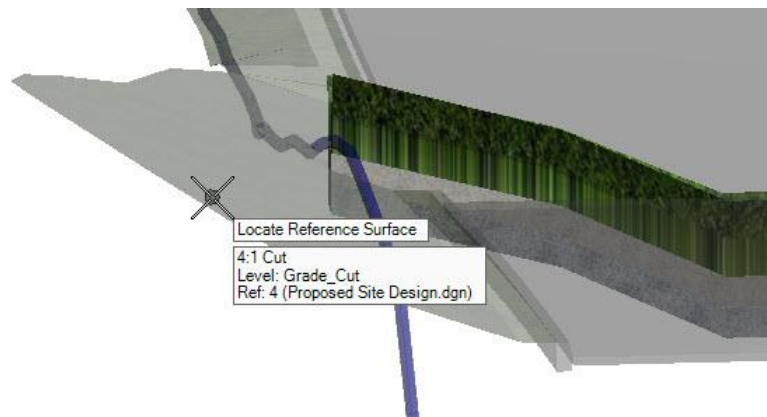
3. Open a view for the profile display



4. We want to display the future road cut on this profile so we can make sure that we are sufficiently underneath it. Start the Quick Profile Command



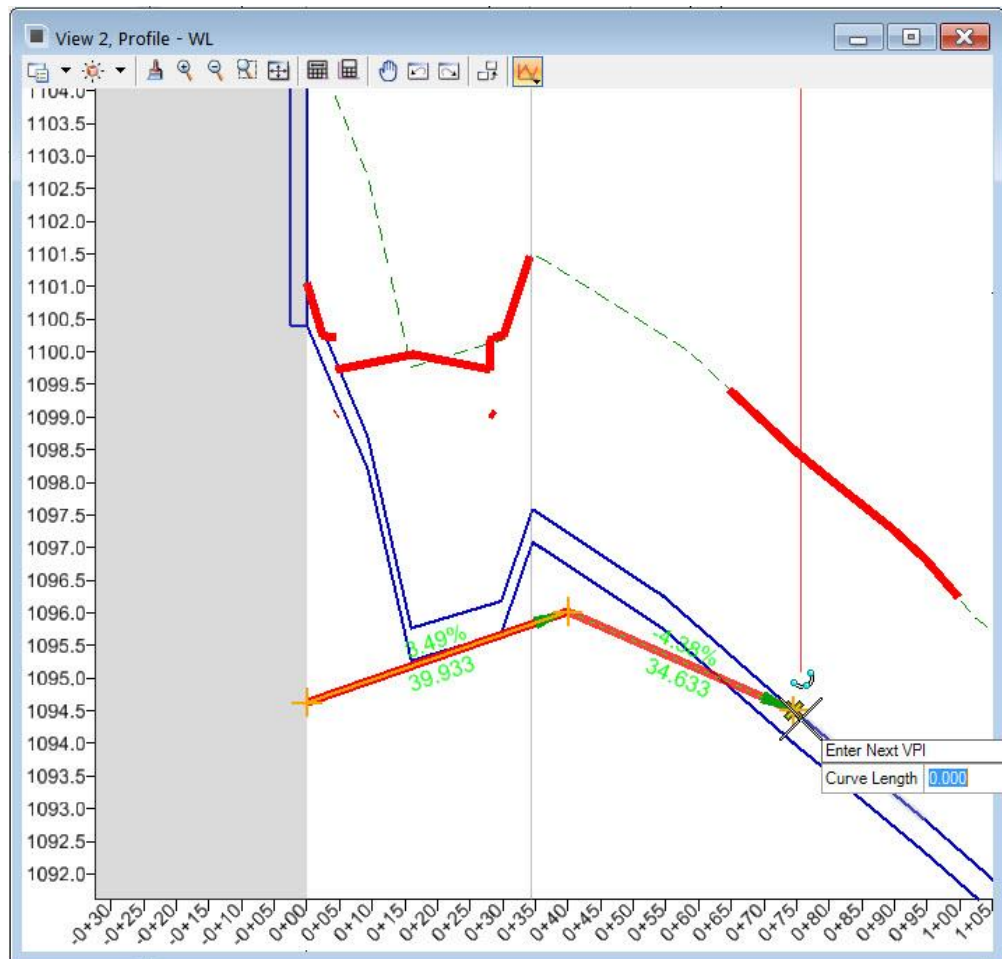
5. Then pick the proposed road surfaces in the 3D view. You can pick all the surfaces if you wish. At minimum you will be interested to see the pavement and the cut slope



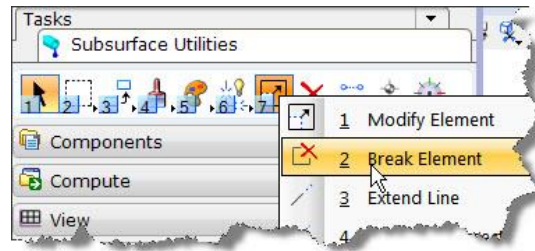
6. This command will show the selected surfaces in your profile. (Note: Red heavy weight added for emphasis)



7. Next, we will use the Profile By PI command to make a new profile for the portion under the road. Notice the snap where we want to join back to the original profile.



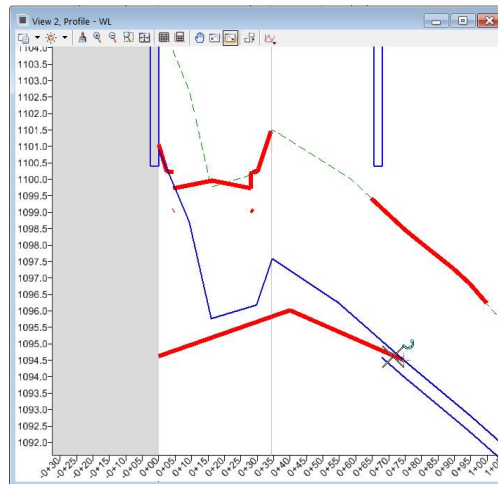
8. Now, start the Break command.



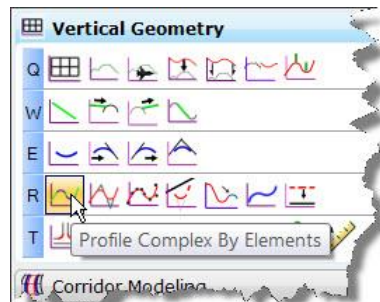
9. Use the Break By Two Points option



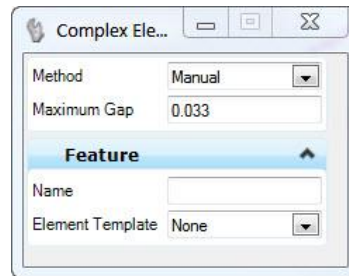
10. And partial delete the portion of original water line profile where we designed new profile.



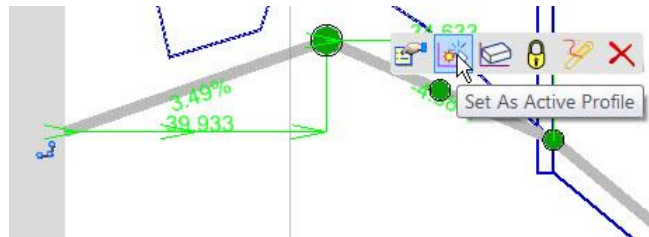
11. Then start the Profile Complex By Elements command



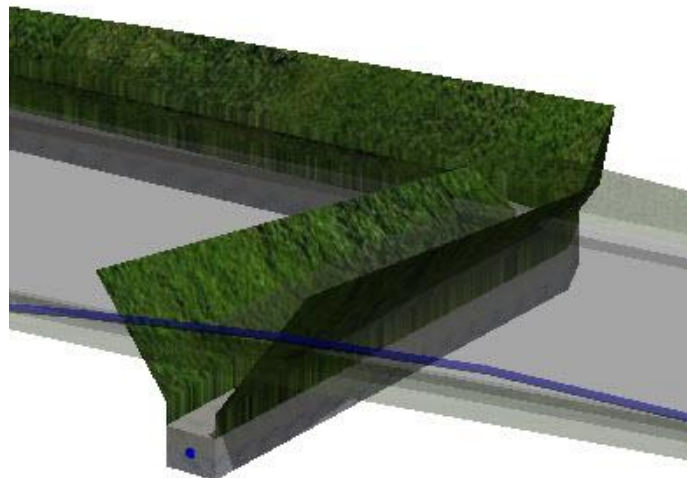
12. You can use Manual or Automatic method to join the new design segment to the original remnant.



13. Finally, select the joined profile and set it as active.



14. Our waterline design is thus updated



WHAT WE HAVE LEARNED:

- The Extract Utility From Graphics command can be used for proposed utilities as well as existing.
- The profile can be redesigned to reflect a new design

APPENDIX: CONFIGURATION VARIABLES USED IN THIS COURSE

In these lessons the workspaces which we used contain the following configuration variables which are relevant to the Subsurface Utilities toolset:

`MS_CELL < $(_USTN_PROJECTDATA)/cell/`

Points to the directory location containing the 2D and 3D cell libraries used by the subsurface utility tools

`MS_CELLLIST < $(_USTN_PROJECTDATA)/cell/*.cel`

Points to the cell libraries themselves.

`CIVIL_CONTENTMANAGEMENTDGNLIBLIST > (_USTN_PROJECTDATA)/dgnlib/SampleDrainageFeatureDefs.dgnlib`

Points to the utility feature definition library

`MS_DGNLIBLIST > $(_USTN_PROJECTDATA)/dgnlib/SampleDrainageFeatureDefs.dgnlib`

Points to the location of element templates used by the feature definitions. Since the element templates are in the same file as the feature definitions then the filename is same.

`CIVIL_CIVILTMDGNLIBLIST > $(_USTN_PROJECTDATA)/dgnlib/SampleDrainageFeatureDefs.dgnlib`

Points to the library containing the utility filters.

`SUDA_SEED_FILE = $(_USTN_PROJECTDATA)/dgnlib/SampleDrainageFeatureDefs.dgnlib`

`SUDA_SEED_MODEL = Default`

Points to the library, and DGN model, which contains hydraulic seed information. Information such as inlet and grate dimensions, pipe roughness factors, catchment runoff factors and more are contained here. Whenever a new utility project is created, the hydraulic seed information is copied to the new dgn file.