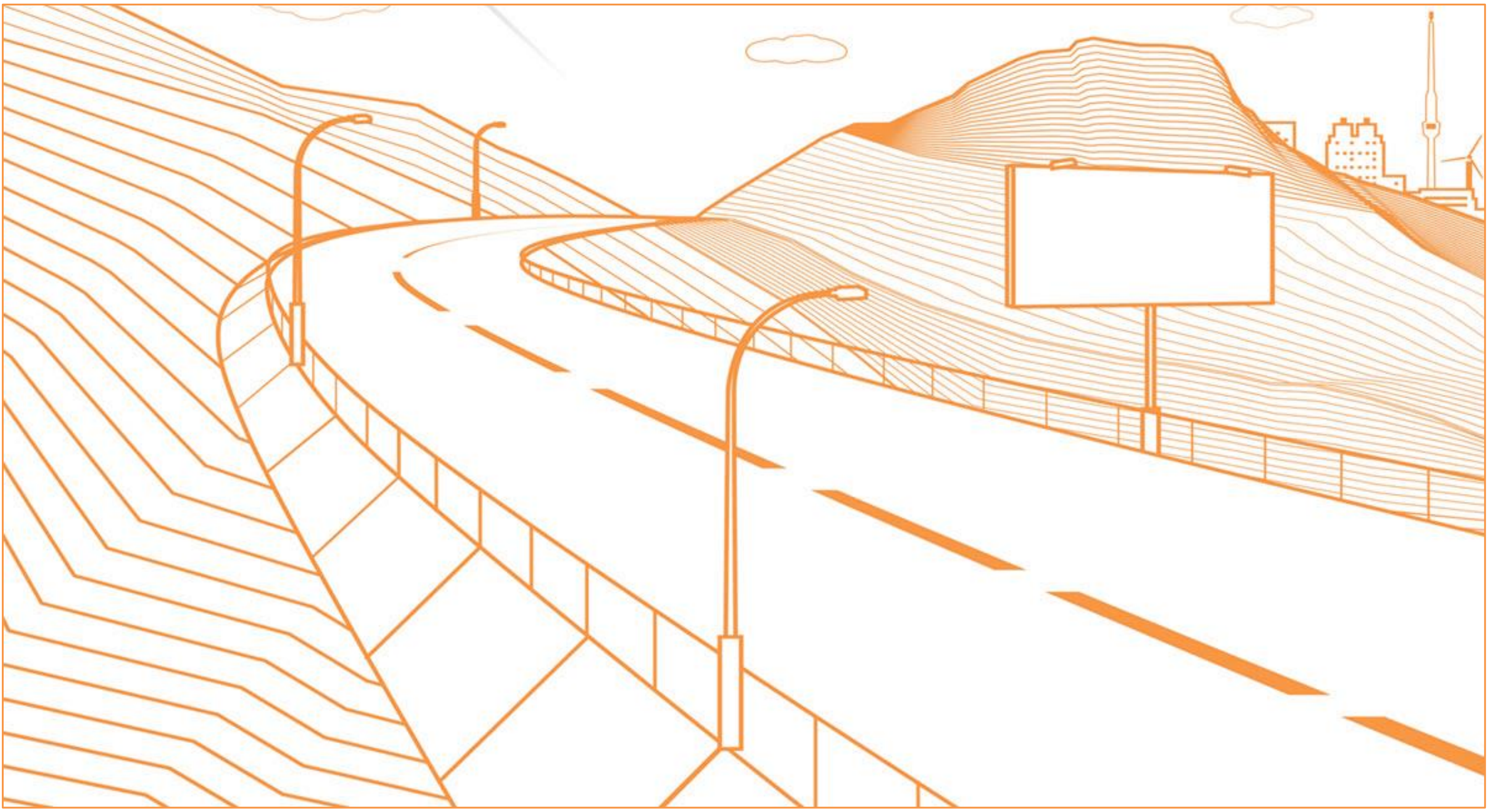
Infratech
www.infratechcivil.com



# AUTODESK CIVIL 3D

## 2018

# TABLE OF CONTENTS

## PRELUDE

**About the authors**

"Infratech Civil Solutions" ("Infratech") is a one-stop consulting, training and drafting solution for AEC (Architecture, Engineering, and Construction).

Infratech leverages the skills of a collaborative team of registered professionals, content developers and information technology experts to offer optimum, customized and technologically advanced solutions.

Infratech core services include:

Project consulting and sub-consulting services.

- eLearning and custom training designed to provide solutions that are tailored and customized to meet the requirements of each situation.
- Post-training support with a package combining technical support, hands-on project guidance, assistance and drafting standard implementation.
- Continued education and certification services to help professionals stand out among peers.
- Assisting qualified organizations with training grant and funding assistance.

**Credits:**

The current material was developed using data and assistance provided by various organizations. Infratech would like to acknowledge: The City of Kamloops, BC for data provided under its open government license, The City of Ottawa, the City of Gatineau, Autodesk Inc., The National CAD Standards, College Ahuntsic, ESRI, Trimble Inc. Leica Inc. and Scyform Montreal.



### Infratech Civil

78 George Street, Suite 204
Ottawa, Ontario Canada K1N 5W1
844.444.6372 (844.HI.INFRA)
training@infratechcivil.com
www.infratechcivil.com

### Disclaimer:

This book explains the process of designing a residential subdivision and other civil engineering infrastructures. Project names, data, locations, and procedures are either the products of the author’s experience or creativity and used in an experimental or in a fictitious manner. Infratech has tried to apply experience accumulated over a decade of professional engineering, surveying, and drafting. Although the author and publisher have made every effort to ensure that the information in this manual follow best practice guidelines, the author and publisher do not assume and hereby disclaim any liability to any party for any loss, damage, or disruption caused by errors or omissions, whether such errors or omissions result from negligence, accident, or any other cause.

This manual is not intended as a substitute for local regulatory requirements. The user should consult and ensure that local guidelines are applied. The information in this book is meant to supplement practical design methodologies. Like any profession requiring a duty of care and public protection, the practice of engineering, architecture, and construction poses some inherent risks. The authors and publisher advise trainees to take full responsibility in the use of knowledge acquired using this manual.

# 1 INTRODUCTION

Hello and welcome to **Infratech**'s **Civil 3D Essentials** course.

AutoCAD is **Autodesk**'s flagship software for computer-aided drafting and design or **CADD** as it's more commonly known. **CADD** is used across the board in professions such as engineering, architecture, construction, and surveying. These professionals use **CADD** to create electronic drawings, plans, calculations, and designs, for printing, visualization and information sharing.

**Civil 3D** is the AutoCAD vertical product, dedicated to the civil engineering industry.

This course, entitled "**AutoCAD Civil 3D Essentials**" is geared towards beginners and intermediate users. The course is purposefully designed to be a hands-on duplication of the real-life experiences of the authors. So, this will allow first-time users to save significant time in the inherent trial and error process of learning a new design software.

More advanced users can also find value in this course, as it will help them reinforce their capabilities and most likely learn more efficient techniques to accomplish some tasks. At the same time, they can compare their design practices and processes to the methods used by the experienced designers that have designed this course.

To supplement this course, **Infratech** also offers advanced and specialized modules such as **Advanced Surveying**, **Advanced Storm and Sanitary Design** and **Advanced Roadway Design**.

## 1.1 Course Objectives

A civil engineering design project is usually a set of moving parts, but in general, Civil 3D is used to accomplish three major tasks:

1. First, **processing and analyzing existing <u>data</u>**. In general, this will involve collecting field data through the process of land

NOTES

NOTES

surveying. This typically involves sending a survey team to the field to assess the lay of the land, using appropriate survey gears and personnel. The collected information is then transferred and analyzed in an office environment, where the existing ground is modeled to the closest replica possible.

2. Second, **the conceptual and detailed design**: At this stage, CADD (Computer Aided Drafting and Design) is initially used to provide a broad stroke of the components that will make up the project. These components must be placed in relation to the immediate environment of the project. For example, is there a previous phase of the project to tie to? What are the project's pre-existing conditions such as the natural drainage? What is the capacity of existing utilities and roads to absorb proposed development volumes? These are just a few questions that must be answered, or at least asked during the conceptual phase. Once those questions are answered, we can then move to the **preliminary design phase**. This phase will aim to ensure that the project can potentially work to the satisfaction of all involved parties including developers, competent authorities and stakeholders. It should be noted that **Autodesk** has other software, such as **Infraworks**, that may be better suited than Civil 3D, for this phase. Then, we have the **detailed design phase**. This is the phase of the project, where using Civil 3D is the most advantageous. The strength of Civil 3D resides in its use to perform detailed calculations, design, drafting, and production of final construction plans.

3. Finally, the design intent must be **illustrated** through the creation of **construction plans and documentation**. In this phase, qualified builders and contractors can submit bids and tenders, based on the plans designed in **Civil 3D**!

NOTES

It’s important to note that Civil 3D is built on top of Map 3D, which was also built on top of AutoCAD. As a result, Civil 3D inherits most of AutoCAD and Map 3D's base commands and functions.

That means if you have Civil 3D, more often than not, you do not need to purchase or install a copy of AutoCAD or Map 3D.

All you need to do is go to the Civil 3D workspace and switch to AutoCAD. This will literally change the interface of Civil 3D to that of AutoCAD. If you need Map 3D functionalities, the same operation will allow you to switch to the Map 3D user interface and have most options available.

This means that knowing AutoCAD will undoubtedly be a benefit in learning Civil 3D. However, this benefit should not be overstated as Civil 3D has concepts and techniques that are often foreign to "Vanilla" AutoCAD users.

It is important to note that Civil 3D is as much a drafting software than a design one. A basic knowledge of civil engineering concepts is essential to get a proper grasp of the software.

In this course, we cover topics such as

1. The Civil 3D user interface, where we see the tools available to us to complete a successful project, including the drawing area, the command line, the ribbon, the application menu and much more.

2. After the interface, the next logical step is to learn how to set up a project through the creation of styles and templates.

3. Then, it will be time to bring in some survey data and learn how to process it, by using points, point groups, and description keys.

4. After that, we will learn about surfaces and how to create, modify and annotate them.

5. We will also learn how to subdivide a land by creating individual parcels.

6. Then we will kick off the roadway design portion of the course, by learning how to create alignments and profiles.

NOTES

7. Then, an **assembly or cross-section**, an alignment and a profile are brought together to create corridors.

8. Next, we will learn how to design **wet utilities** such as sanitary and stormwater sewers.

9. When the detailed design is completed, we need to estimate the project **quantities** by using **material lists**.

10. Finally, we will see how to create **constructions plans and documentation** using Civil 3D.

Okay - we have a lot of ground to cover ahead. So, without any further delay, let's start learning **AUTOCAD Civil 3D Essentials**.

## 1.2 Common terms

Below is a list of definitions of common terms used in this course. Some of these terms are listed as defined in AutoCAD Civil 3D® glossary:

**Alignment:** "A series of 2D coordinates (northings and eastings), connected by lines, curves, or spirals, used to represent features such as the road centerlines, edges of pavement, sidewalks, or rights-of-way."

**Assembly:** "An AutoCAD Civil 3D drawing object (AECCAssembly) that manages a collection of subassembly components, such as travel lanes, curbs, shoulders, and ditches, to form the structural elements of a roadway or other corridor-type structure".

**Breakline**
"A line used to connect the data representing a distinct surface feature, like a ridge line, edge of pavement, toe of a slope, the centerline of a road, or flowline of a ditch or stream. When a breakline is defined, the surface triangulation must first follow the breakline, by placing triangle edges coincident with the breakline segments. This ensures the feature in the model is accurately depicted. Then, the rest of the interpolation is performed based on proximity. Breaklines are typically critical to creating an accurate surface model. It is the interpolation of the data, not just the data itself, that determines the shape of the model. See also non-destructive breakline"

**Catchment Area**
"The area tributary to a lake, stream, or drainage system."

**Contour**
"A line that connects points of the same elevation or value relative to a specified reference datum."

**Corridor**
"Any path, the length, and location of which are typically governed by one or more horizontal and vertical alignments. Examples are roadways, railways, traveled ways, channels, ditches, utility runs, and airport runways."

**Curb return alignment**
"An alignment that connects the edges of two intersecting roadways. The most common curb return geometry is a simple circular fillet. In a typical intersection, curb return alignments are placed in each of the four quadrants, between edges of the pavement."

**Data shortcut**
"An object that can create a data reference between drawings in a project. Data shortcuts are not used with Vault projects."

**Daylight line**
"A line showing the line of zero cut or fill within the job area. For grading objects, it represents the target line produced by grading to a specified surface, distance, or elevation."

**Project Management**
"AutoCAD Civil 3D offers a set of data management options. Master these options; you are able to determine the data structure best suited to your projects. "

**DEM**
"(Digital Elevation Model) An array of elevations taken on a regularly spaced horizontal grid."

**Easting**
"A linear distance eastward from the North-South line which passes through the origin of a grid. Equivalent to the X coordinate in an XYZ coordinate system."

**Field book**
"The permanent detailed record a surveyor makes of all observations made in the field. In AutoCAD Civil 3D, a field book file, (*.fbk), it can be used as a source of survey data."

**Field to Finish**
"Use the Civil 3D topography functions to download, create, analyze, and adjust topographic data. These features complement the basic Civil 3D functionality by simplifying the process of transferring topographic data collected from the field to and from your office."

NOTES

NOTES

**Figure**
"Special linework automatically created in a drawing when you import a field book containing figure data. Also created when using AutoCAD Civil 3D survey figure commands. You can control the layers for figures by using figure prefixes".

**Foresight**
"A point to which an instrument sighting is made to measure or establish its elevation or horizontal position."

**Figure prefix**
"Used to automatically place figures on user-defined layers based on the beginning part of the figure name. For example, the figure prefix EP can be used to place the figures EP, EP1, EP2, and EPL all on the same layer. Using figure prefixes in conjunction with description keys can largely automate the process of constructing a well-organized base plan."

**Frontage**
"The parcel segments that are adjacent to a right-of-way. The frontage is also considered to be the parcel segments located at the front of the parcel."

**Full description**
"The expanded description of a point after description key matching has taken place."

**Grade**
"A method of reporting ground inclination in which the change in elevation is expressed as a percentage of the horizontal distance traveled. For example, if the ground rises one linear unit (meter or foot) over a horizontal distance of five units, the grade is 20%. See also slope."

**Grading**
"The process used to model the finished ground surface."

**Hydraulic Grade Line**
"A line in a pipe network or storm sewer drainage system that represents the elevation head and pressure head of fluid at any point along a system."

**Inverse**
"An inverse calculation on a closed figure calculates the bearings and distances between coordinates and reports the area. Because you use exact numbers (coordinates) to determine corners, no closure error is reported."

**LiDAR**
"Light Detection and Ranging. Typically associated with airborne laser scanning."

NOTES

**Northing**
"A linear distance northward from the East-West line which passes through the origin of a grid. Equivalent to the Y coordinate in an XYZ coordinate system."

**Point Group**
"Collection used to group the points in a drawing into smaller, more manageable units. For example, you can create a point group that contains all of the points in a drawing that meets certain elevation criteria."

**Point of Vertical Intersection:**
"(PVI) In a profile, the point where two tangent lines meet."

**Profile**
"An object that contains elevation data along a horizontal alignment or another line. There are two main types of profiles: surface and layout. Profile data objects can be viewed within a profile view object."

**Profile View**
"An object that manages the graphic display of profile data objects within a drawing. A profile view is essentially a graph with two primary axes: the x-axis represents horizontal distance along the referenced horizontal alignment (or another linear feature). The y-axis represents elevations. Profile view objects can also include grid display components and data bands."

**Prospector Tab**
"The part of Toolspace where you access drawing and project objects. Objects are arranged in a tree or hierarchy with folders and subfolders that you navigate. See also Settings tab."

**Raw Description**
"The original description of a point, before description key matching takes place. Often corresponds to the point description entered in the field by a surveyor."

**Right-Of-Way**
"(ROW) The allowable work area for an alignment. Property lines of the property owners who reside adjacent to the construction site generally specify these limits, which are called right-of-way lines."

**Site**
"A collection of objects that are managed by means of common topology. The objects that participate in the topology are parcels, alignments, and grading. See also topology."

**Slope**
"A method of reporting surface inclination as a ratio that expresses the horizontal distance in which the elevation changes by one linear unit.

NOTES

For example, if the ground rises three units over a horizontal distance of 15 linear units (meters or feet), the slope is 5:1 (5 to 1)."

**Spot Elevation**
"The elevation of a single point in the drawing. Used to define areas that are sparse in contour data when generating a TIN using contour information. Areas that may also need spot elevations are the top of hills, valleys, and bottom of swales."

**Stakeout**
"The process of placing stakes in the ground at control points on a site that is being developed. For example, after you place points in your drawing, or after you design an alignment, you can create stakeout reports that list the coordinates of each stake. Someone else can then use these stakeout reports to place (or adjust) the stakes at the site."

**Station**
"A distance along a horizontal alignment."

**Style** "A logical collection of settings that applies to a class of objects. Styles simplify the process of apply settings by simply referencing a style. Modifying a style affects all the objects referencing that style."

**Subassembly**
"An AutoCAD drawing object (AECCSubassembly) that defines the geometry of a component used in a corridor section. The AutoCAD Civil 3D tool palette and tool catalogs provide a variety of preconfigured subassemblies, such as travel lanes, curbs, shoulders, and ditches."

**Subdivision**
"An unimproved tract of land surveyed and divided into parcels for purposes of sales."

**Superelevation**
"The change in cross slope or 'banking' on a horizontal curve to help counterbalance the centrifugal force of a vehicle traversing the curve. See also cant."

**Surface**
"A network of elevation data (either TIN or Grid). The points of a surface are connected into either triangles or a grid, which are then used to interpolate contours, and to generate profiles and cross-sections. A surface represents the ground condition at a particular time or event."

**Template**
"A collection of default settings and styles used to create a drawing."

**TIN (Triangular Irregular Network)**
"A TIN surface is the most common method of interpolating elevation data. The points are connected into triangles that are used to interpolate for contours, and to generate profiles and cross-sections.

NOTES

The lines that make up the surface triangulation are called TIN lines. See also 3D face.

**Transparent Command**
"A command that you can run while another command is in progress. Transparent commands begin with an apostrophe (')."

**Traverse**
"A method of surveying in which length and directions of lines between points on the Earth are obtained by or from field measurements and used in determining positions of the points."

**Vertical Curve** "A curve used on a profile (most commonly on layout profiles) to provide a gradual change in slope from one tangent to the other. There are three types of vertical curve: parabolic, circular, and asymmetrical."

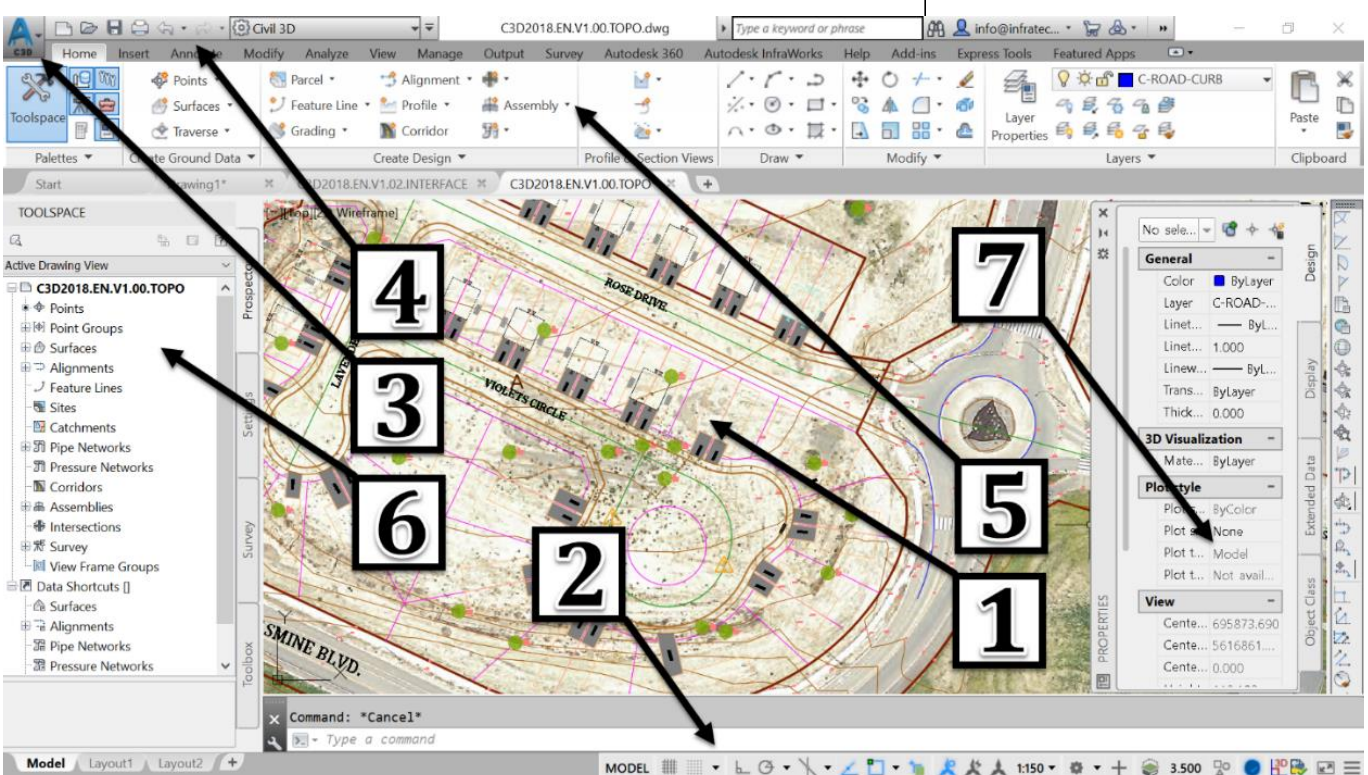
**Watersheds**
"Catchment areas for rainfall that are delineated as the drainage areas producing runoff. Base flow in a stream also usually comes from the same area."

NOTES

# 2 INTERFACE

Let's first introduce the items we will explore in more details in this lesson. The interface is the main working environment where we interactively locate and use the needed Civil 3D tools. Among the main components of the Civil 3D interface we have the:

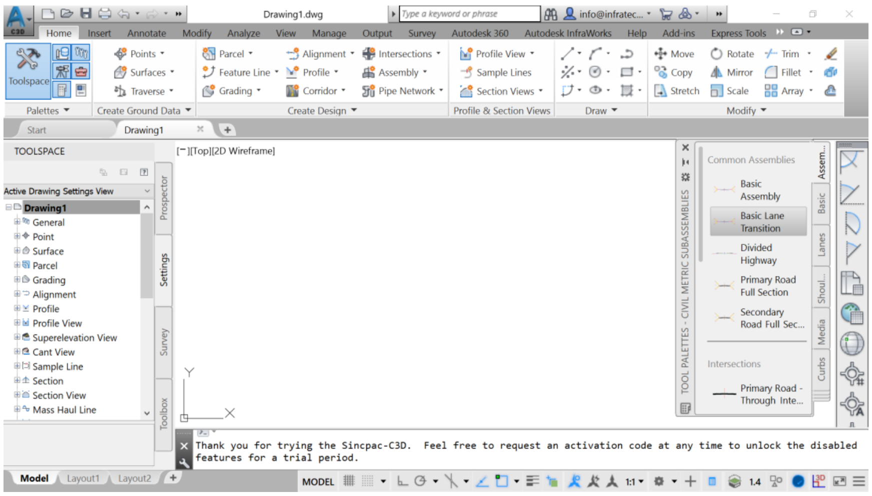
1. **Drawing window**: this is the main drawing and design window. It allows you to manipulate and visualize objects in your current work.
2. The **Command line**: It allows you to launch an AutoCAD command or input drafting options
3. The **Application Menu: It** allows you to navigate and manage files.
4. The **Quick Access Toolbar**: It is used to display most commonly used commands.
5. The **Ribbon**: Contains commands needed for routine tasks. The **Ribbon** is a feature implemented starting with the 2010 version of Civil 3D. It's a practical replacement of the old menu bar. This menu still exists but is hidden by default. The ribbon is accessed by typing **ribbon** at the command line.
6. **Toolspace:** This is the main window for managing data, setting parameters and styles for the drawing.
7. **Properties Window**: Displays specific properties of selected entities



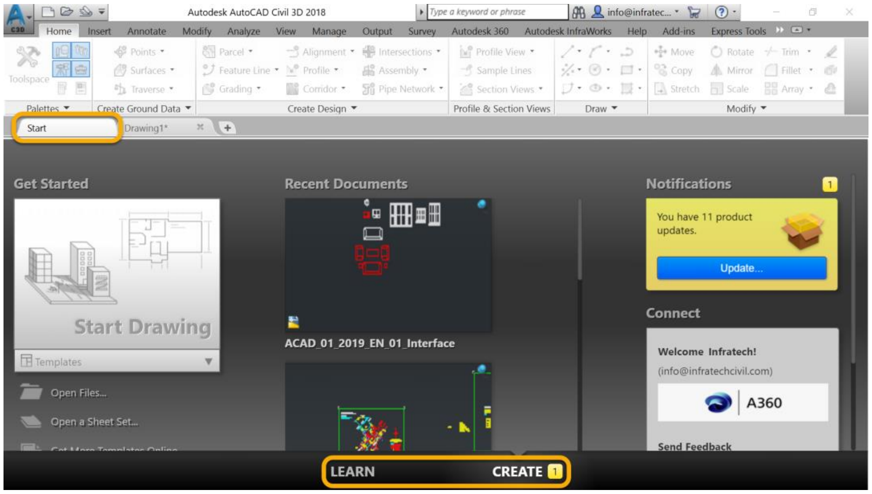
## 2.1 Start Screen

1. Now, we are going to explore a few items accessible from the **Start Screen**. It will be advantageous to know how to access some of the options and Autodesk services before diving deep into **Learning Civil 3D**.

2. Launch Civil 3D by clicking a shortcut, usually from your PC's desktop, program menu or taskbar. Civil 3D opens, and your default template file, **drawing 1**, is displayed.



3. Next, click on **Start**. You will notice two more tabs at the bottom of the screen, **Learn** and **Create**.



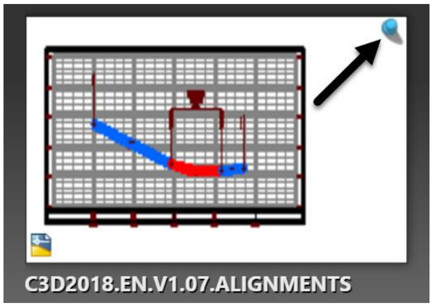

4. Let's focus on the **Create** tab, the default one. On this screen we have three sections: **Get started**, **Recent Documents** and **Notifications**.

NOTES

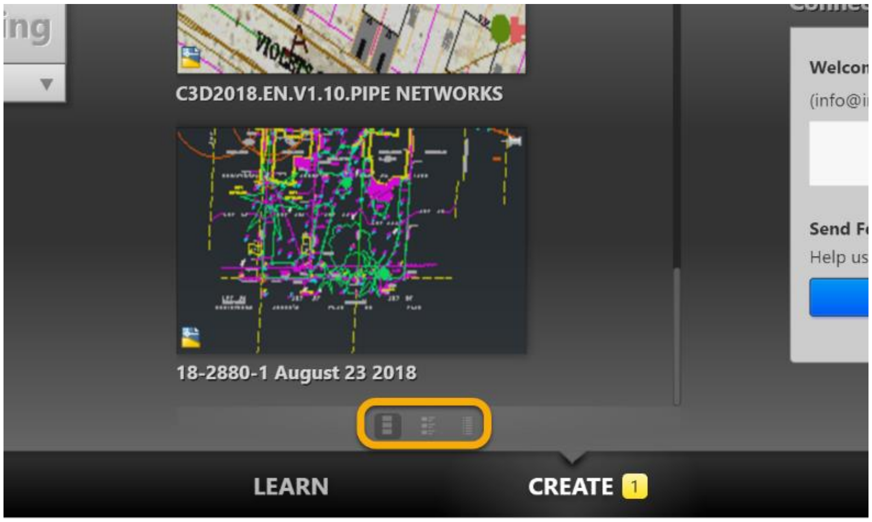
NOTES

5. The **Get Started** section allows us to do things like,
   - Opening drawing files and **sheetsets**.
   - get templates online, such as localized country-specific kits; and
   - explore sample drawings on our desktop.

6. The middle section shows a list of **Recent Documents** we have worked with. These documents are shown in chronological order. So, if we are looking for a recent document that we don't quite remember the folder it was saved in, this is a good place to look. Furthermore, we can pin a document, to permanently display it in this list, by clicking on the **Grey pin** in the top **right corner**. Once we are done with working with that file or don't need to remember it anymore, we can simply unpin it.

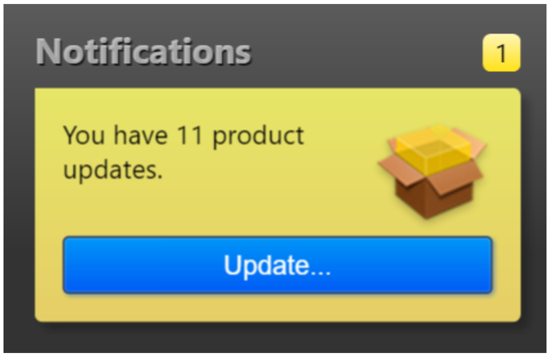



7. At the bottom, we have an option to display the files in different modes: Big and medium icons, with previews, or list view modes.

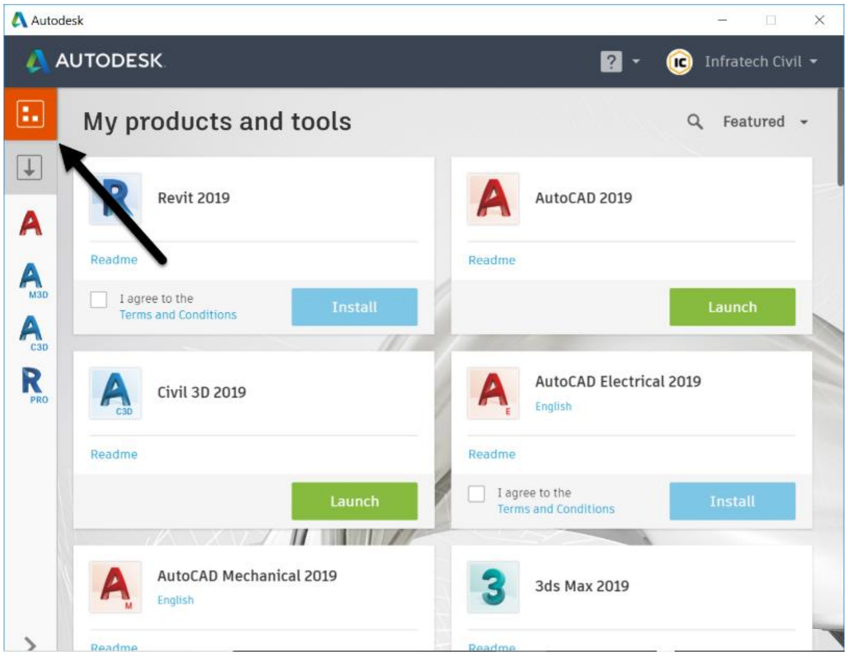


NOTES

8. Now, let's explore the third section, Notifications. In here, we get notifications from **Autodesk** to make sure our subscriptions and services are up to date and running smoothly. If you are logged into your AutoDesk account, you will see the number of items that need your attention. In our case, we have one item, which is a number of products that need to be updated. Let's check it out. Click on **Updates**.



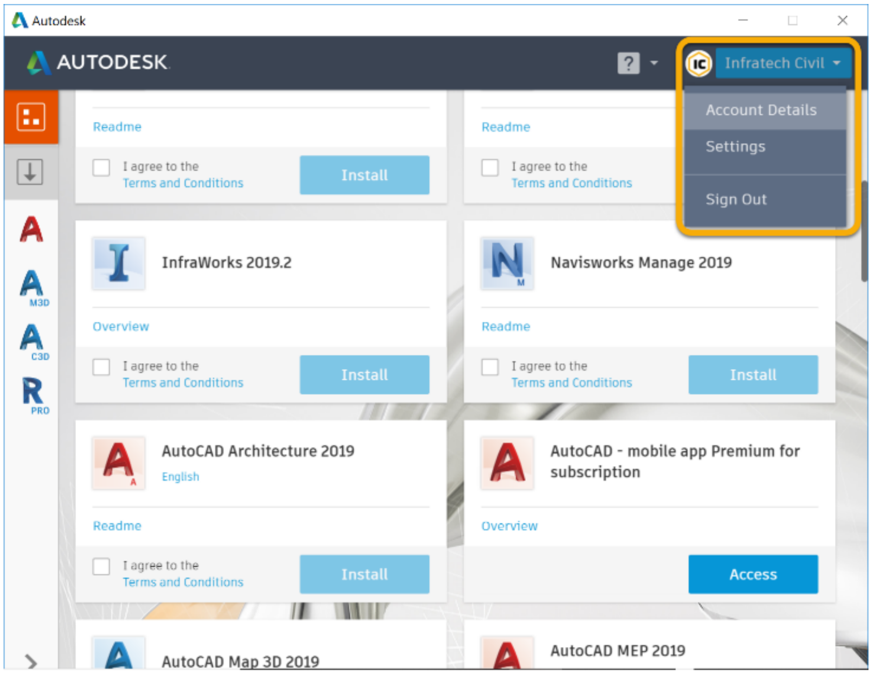

9. A new window is displayed. The icon in the top left corner allows us to see a full list of products. This list will be different depending on your subscription package. But the process is usually the same. To see the list of updates that can be made, Click on **My products and tools.**



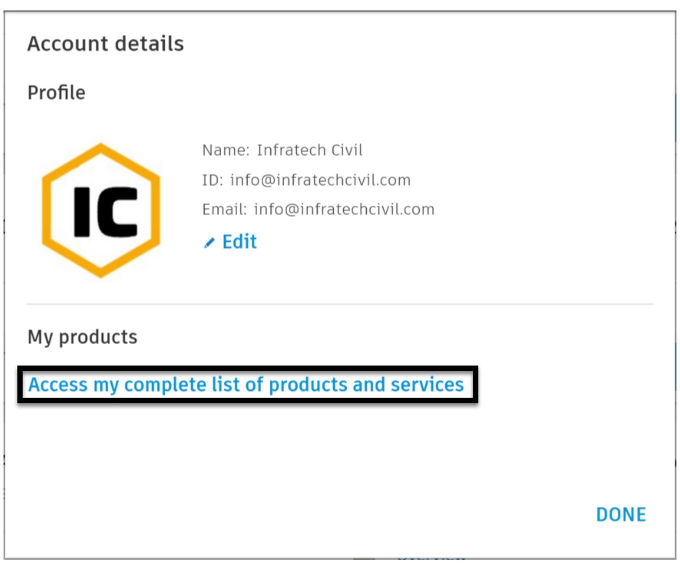

10. What is shown here is a comprehensive list of all products in your subscription package. In this case, we have access to the **AEC** collection (Architecture, Engineering, and Construction), which gives us access to software such as **Civil 3D**, **Revit**, **Recap**, **Map 3D**, and many more. This is a good place to check if you ever need a design software.

11. You may already have access to one in your collection, without knowing it. To install a software, simply agree to the terms and conditions, and you are on your way.

12. One more thing you can do is to access your full **Autodesk Account** from the menu in the top right corner.



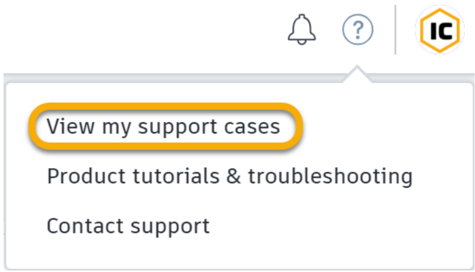

13. Once there, click on the **Account Details**, then on **Access my complete list of products and services**.



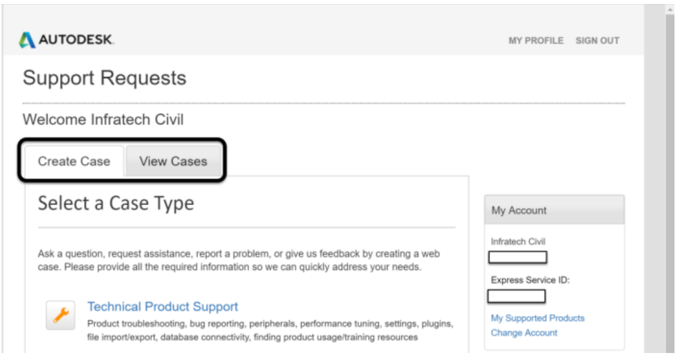

14. Here, you can,
    - manage your profile or company settings, if you have account administrative rights.
    - change contact and billing information; and
    - access cloud services.

NOTES

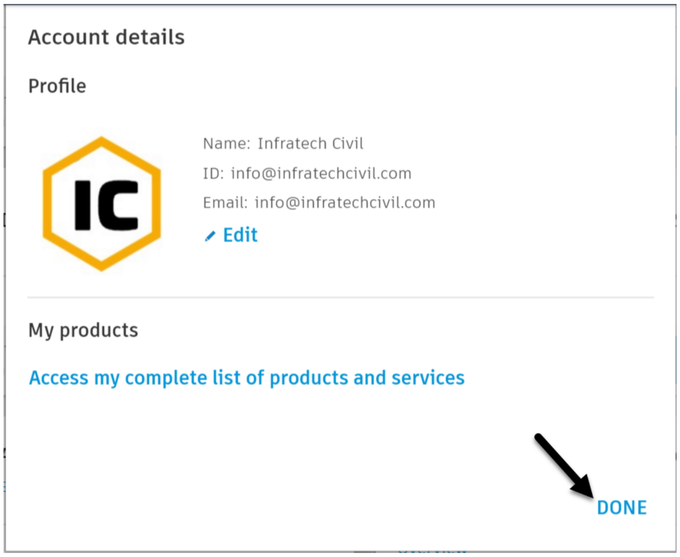
15. But, one of the most useful services is a direct access to **Autodesk Technical Support**. To do that, click on the support icon in the top right corner.




16. From there, you can check on updates on your existing support cases or simply create new ones:




17. Then,
    - Close your internet browser window to return to the **Autodesk Portal**.
    - Click on done to close the **account details** window.




18. Next, close the main portal access window to return to the **Start Screen.**

This concludes the section on the **Start Screen**. Next, we will start exploring in more details, the Civil 3D interface and the working environment.

NOTES

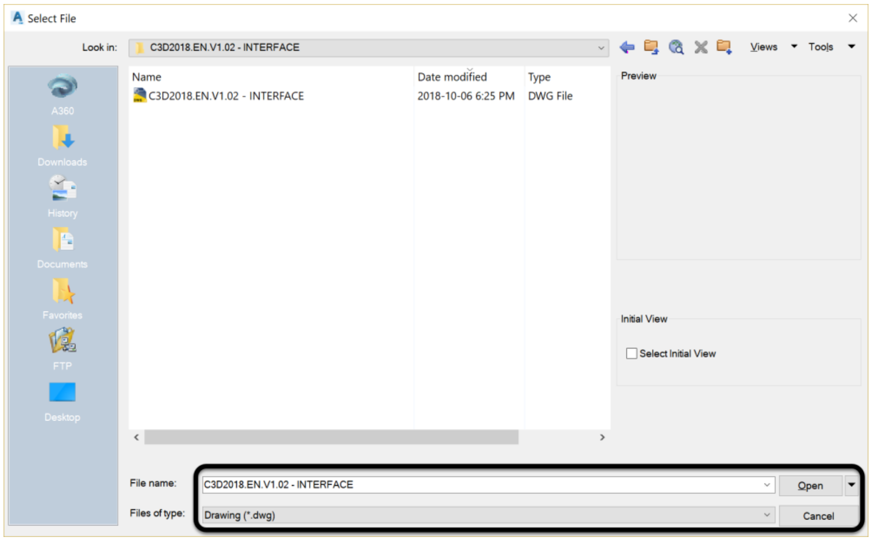
## 2.2 Application Menu

Like any other software, the best way to start learning Civil 3D is to first familiarize ourselves with the user interface. That's the working environment where the tools and needed commands are located.

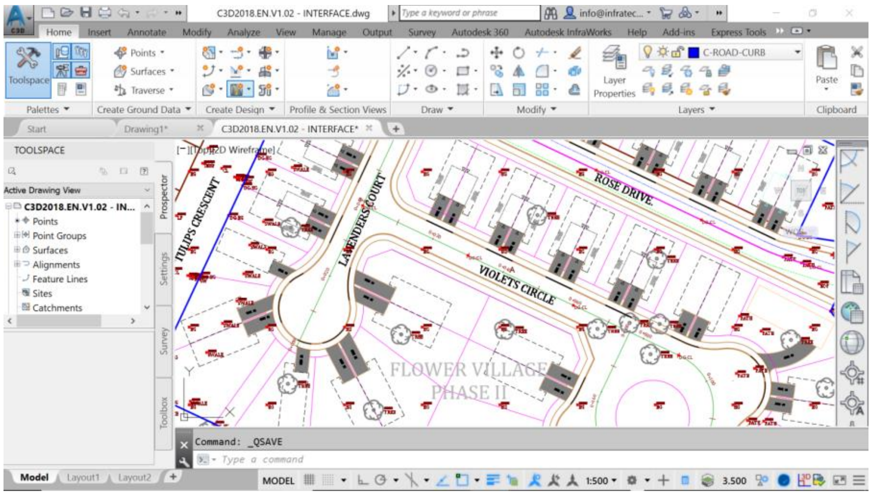
Let's open a file and explore the Civil 3D interface.

1. In the top left corner, click on the **Open** command to browse to the **02.01-Interface dwg** file in **Lesson 02** practice folder.

2. In the new window, make sure you have selected a **dwg** file format, then select the **02-Interface** file and click on **Open**.

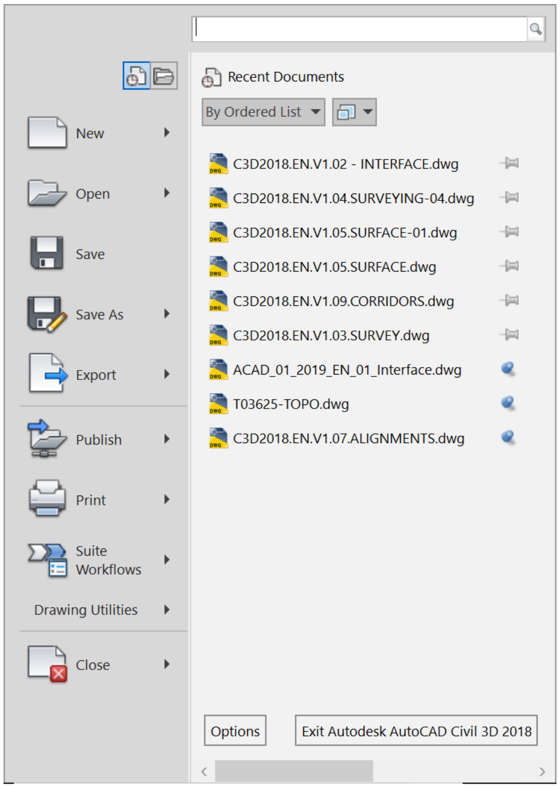



3. The drawing opens. We see the proposed **Flower Village Townhouse Subdivision**, which is the project we will work on throughout this training. We will try to design it **from-field-to-finish**. We will see all the necessary steps from the moment we received field survey data to the creation of the final field stakeout. This process will involve organizing and adjusting survey data, creating road alignments and profiles, site grading, designing utilities, stormwater management, estimating volumes and much more.



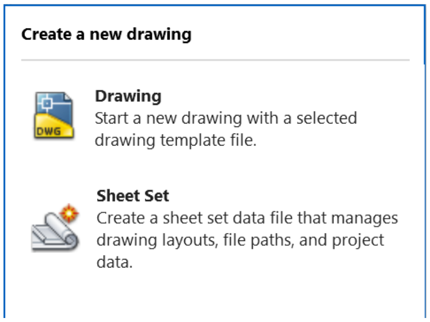

NOTES

4. Now, let's explore some components of the **Civil 3D** interface. First up, the **Application Menu**. It is in the top left corner and is represented by the Autodesk Civil 3D logo . Once you click on it, you'll notice that you have more options. We are going to explore a few of them. Most of the items on the application menu are similar to the AutoCAD Application menu. So, for more details, please refer to the AutoCAD course. We will talk about these commands here, but not in as many details because they are mostly basic AutoCAD commands.



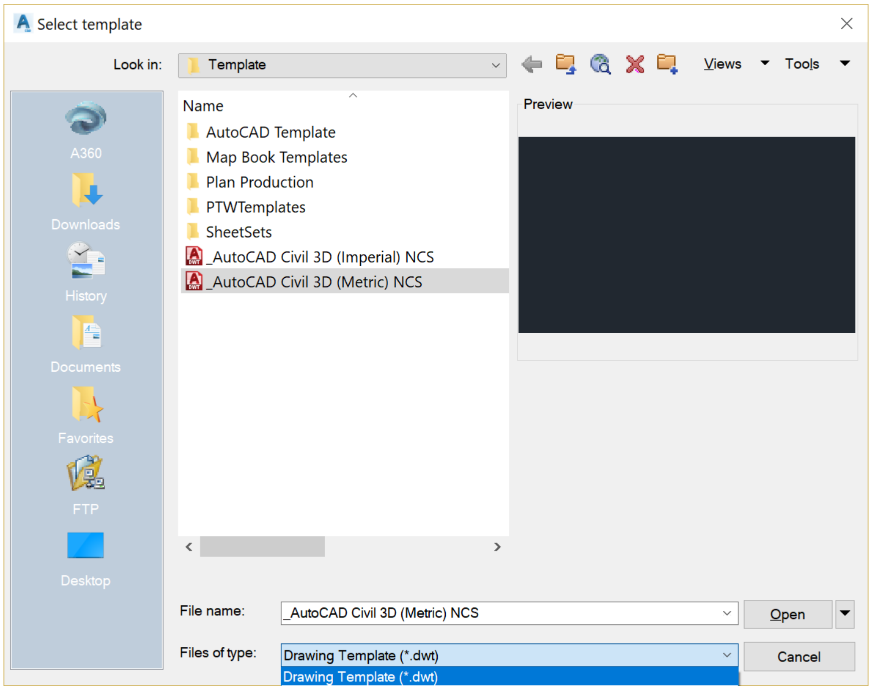

5. We have the **New** command at the top, click the arrow to the right to expand it for more options. From there, we can either create a new **drawing** or **sheetset**. A **sheetset** is a drawing file specifically created to manage sheets for printing. Before going further, let's note that we will be using the terms **Plotting**, **Printing**, and even **Publishing** interchangeably. They are originally meant for different things, but as the demarcation line between them is becoming finer, the more we move to the electronic age of things. But that's a discussion for another day. For now, all three terms mean creating an output file for sharing with others.

6. Click on **Drawing**.




NOTES

7. A new window opens. Here, be careful of what type of file you choose to create. Most of the time we are creating two types of files:
   - A **dwg** file, which is a standard Civil 3D file; or
   - A **dwt** file, which is a Civil 3D template file. We will ignore the third type, the **dws** for now.



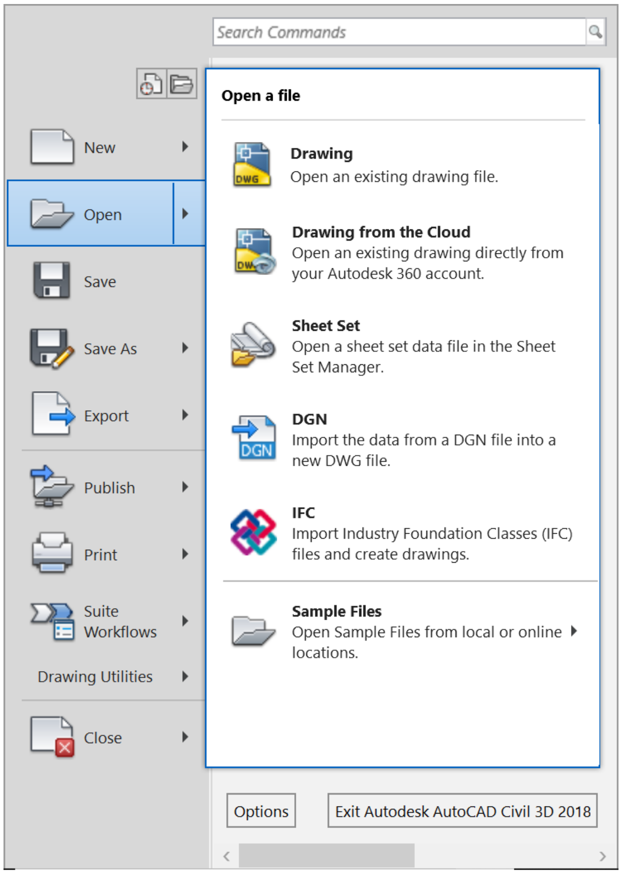

8. The **dwt** is a template file, which means that it contains standards and settings commonly used, such as unit formats and precisions, title blocks and borders, styles and label settings, and much more. It's always recommended to start a new drawing from a template to make it much easier, because you have a lot of parameters already set up in previous jobs, and you don't need to recreate them.

9. On the other end, you can choose to create a new **dwg** file if,
   - you only need to create a quick drawing and you aren't all that interested in the setup.
   - Or, if you have already specified a default template in Civil 3D. In that case, Civil 3D opens your template by default. We will see later how to create a default template.

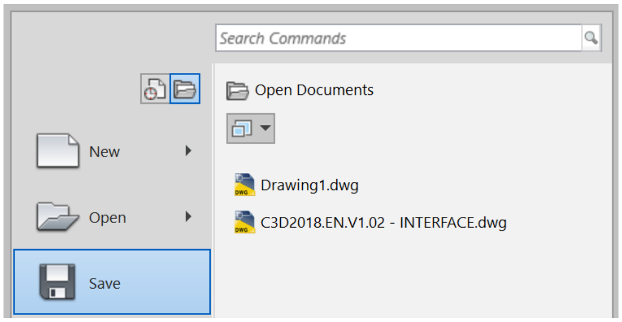
10. For the time being, let's cancel and return to the application menu.

11. The next line in the Application menu list is the Open command. Let's expand it by hovering over it. A scrolling menu shows up with options such as:
   - **Drawing**, to open a pre-existing drawing that has been worked on already, not from a template;
   - **Drawing from cloud,** to open a drawing stored somewhere on the internet;
   - **SheetSet**, to open a printing setup file;
   - Additional formats such as **DGN** and **IFC** files.

NOTES




- Next, on the **Application Menu**, we can save the current drawing in the default **dwg** format.

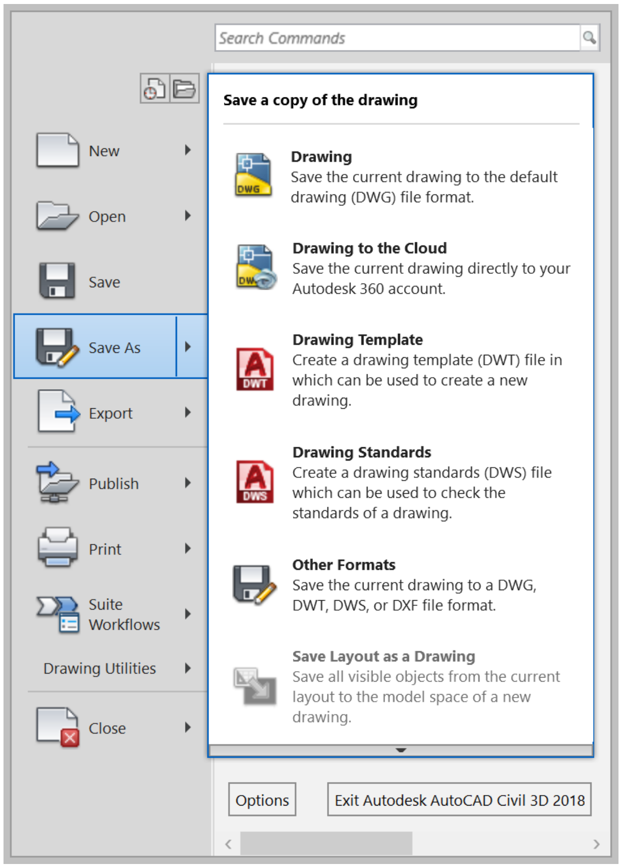



- Or, we can use **Save As** to choose a different file format or name. For instance, we can **save as** under:
- a standard Civil 3D desktop file, or a mobile friendly drawing format;
- a drawing template, the standard **dwt** format we talked about earlier;
- a drawing standard file;
- other formats, such as **dxf**;

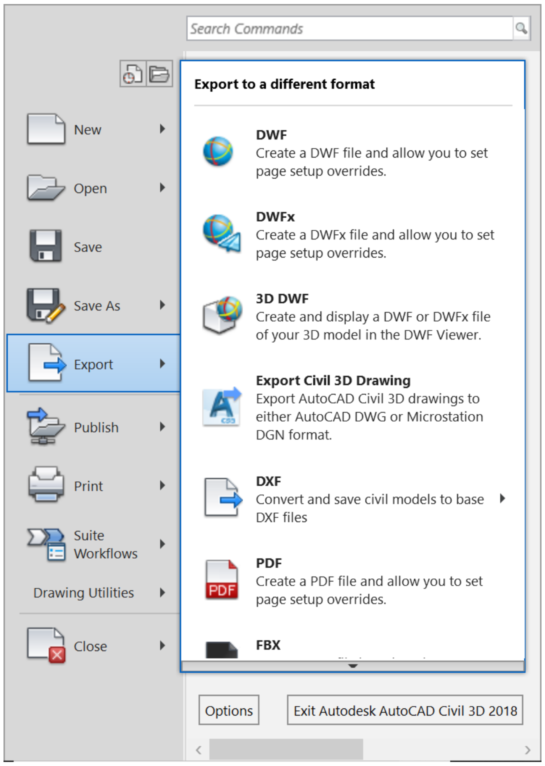
12. We can even save a layout as a completely separate AutoCAD file. How great is that? If for example, we want to create a file for only our

NOTES

grading or drainage sheet layout, we can do that and send it to whoever is requesting it.



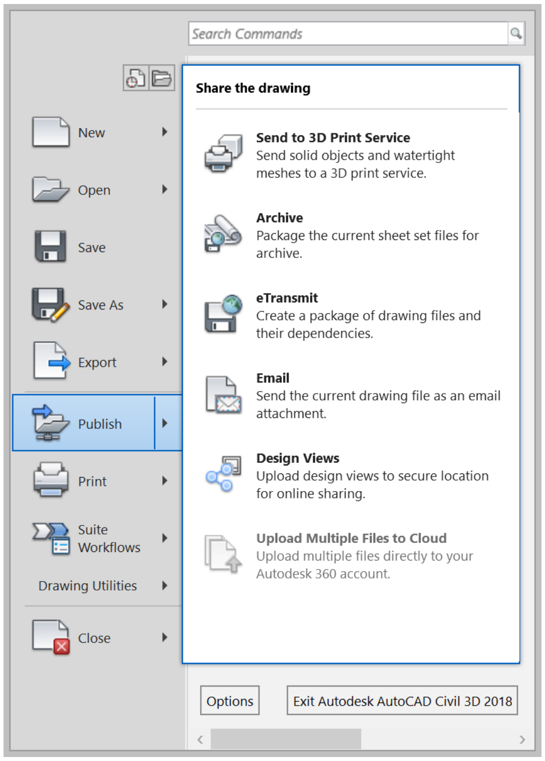

13. We can also export a Civil 3D drawing to different formats such as **pdf** or **dwf**. A **dwf** is essentially the Autodesk **pdf** equivalent. It's also very convenient as you can open a **dwf** file on most modern desktops without installing any specific software.



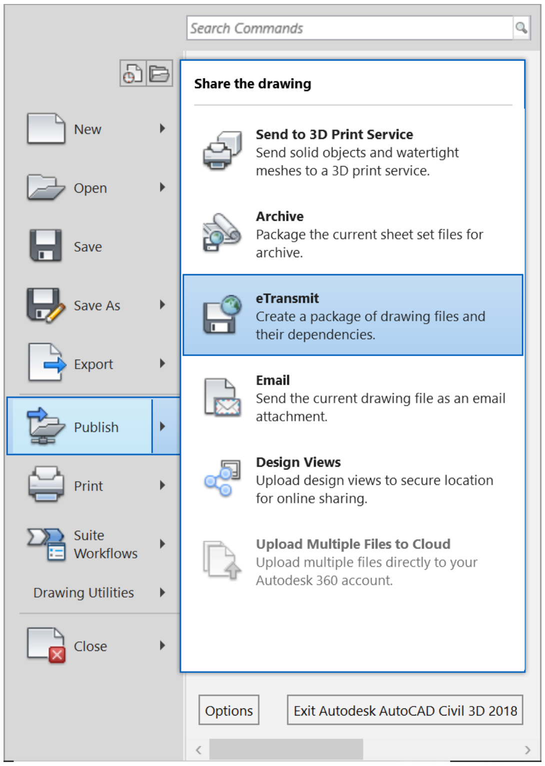

NOTES

NOTES

14. Next, we can **publish**. In Civil 3D, publishing is simply printing for the purpose of sharing electronically. To publish a drawing, we have multiple options.




15. Among the different forms of publishing, one that we need to mention is the **eTransmit** option. This method is highly recommended. It makes sure all files related to the current drawing, such as external reference files, fonts, and plot styles, are embedded in the published package.

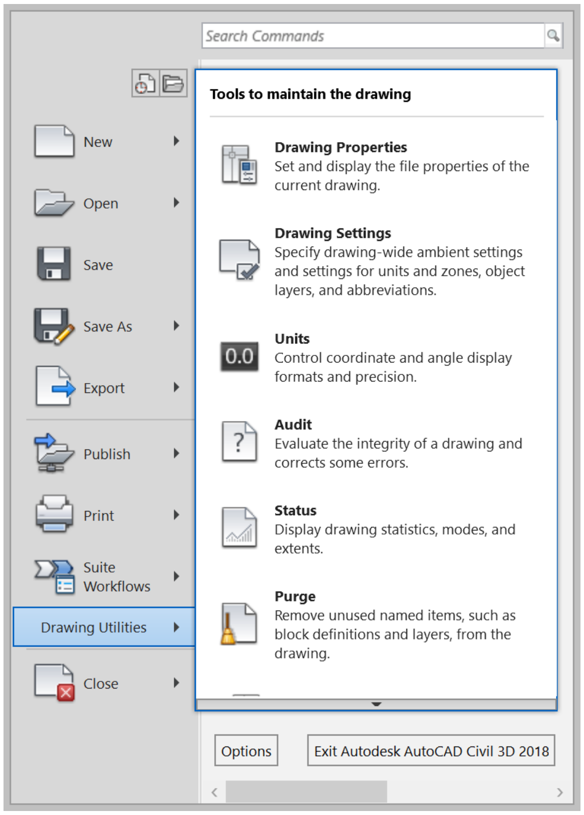


NOTES

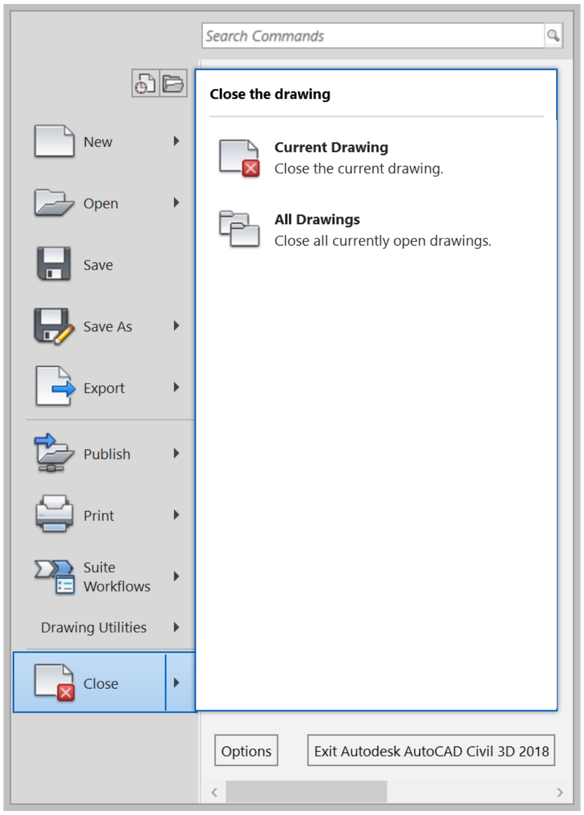
16. In the **Print** section, we can perform various printing related tasks. We will talk more about them in the **Output** chapter.

17. Next, are the **Drawing utilities**, where we can set various properties and parameters for the current drawing. Once again, refer to the **AutoCAD Essentials** course for more details on how to use these settings. For now, just know that you can make operations such as:

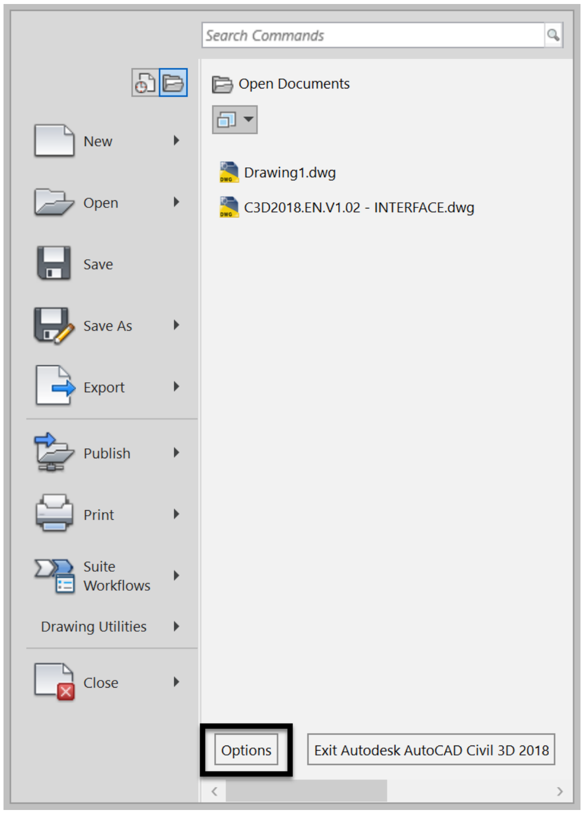
 - consulting the drawings properties and statistics;
 - accessing the Civil 3D settings, which we can also do from the **Toolspace**, which we will talk about in a minute;
 - setting the Drawing Units;
 - auditing and recovering the drawings to fix corrupted files;
 - displaying the **Files Status**, for more statistical information;
 - And, performing a purge, a typical AutoCAD operation to get rid of unused drawing objects.



18. We can also close the current or all opened drawings and exit Civil 3D.



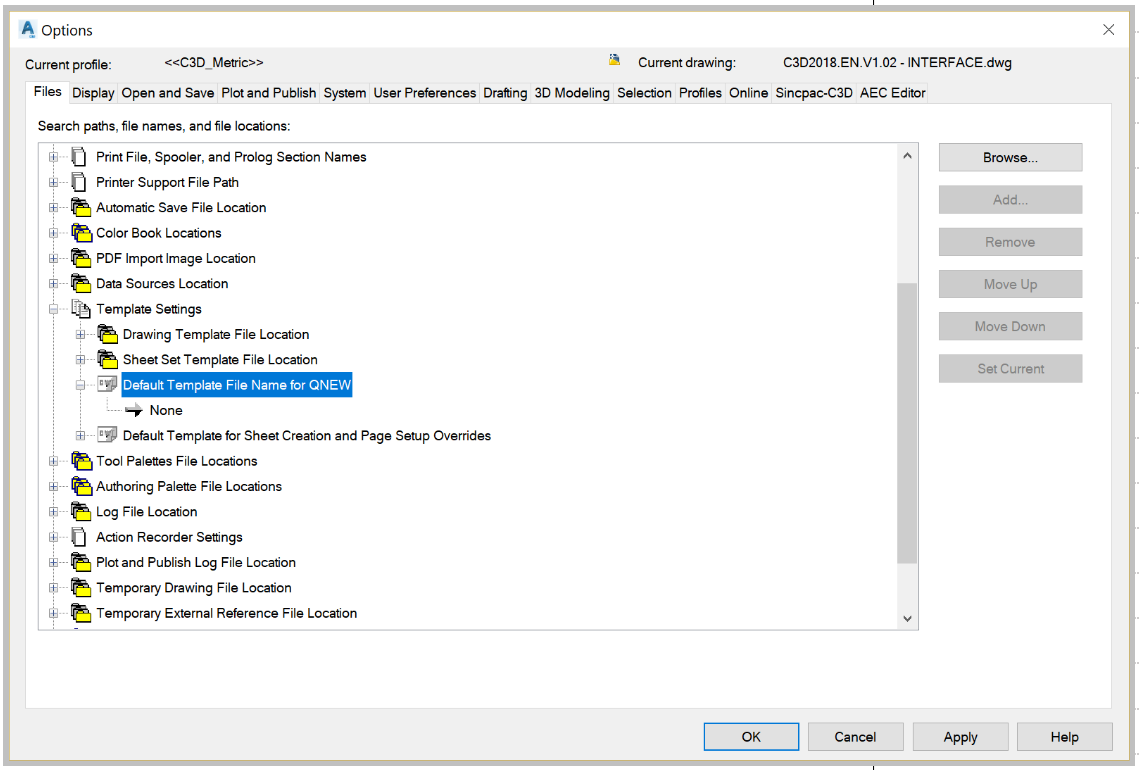

19. One more thing we can do from the **application menu** is to to access the current drawing's global **options**.




NOTES

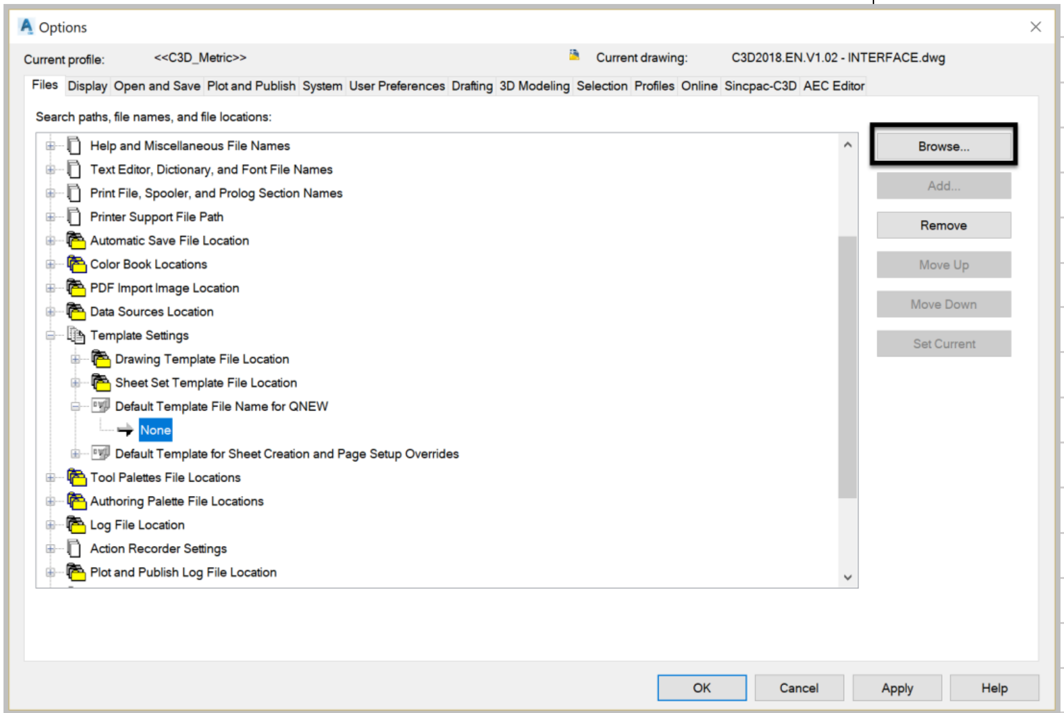
NOTES

20. The **Options** window is where most of the drawing's underground preferences and settings are stored.
21. Among the choices, we have:
22. **The file tab is** where we can find all the information for Civil 3D support file locations, including paths for the program drivers, menus, and other files.

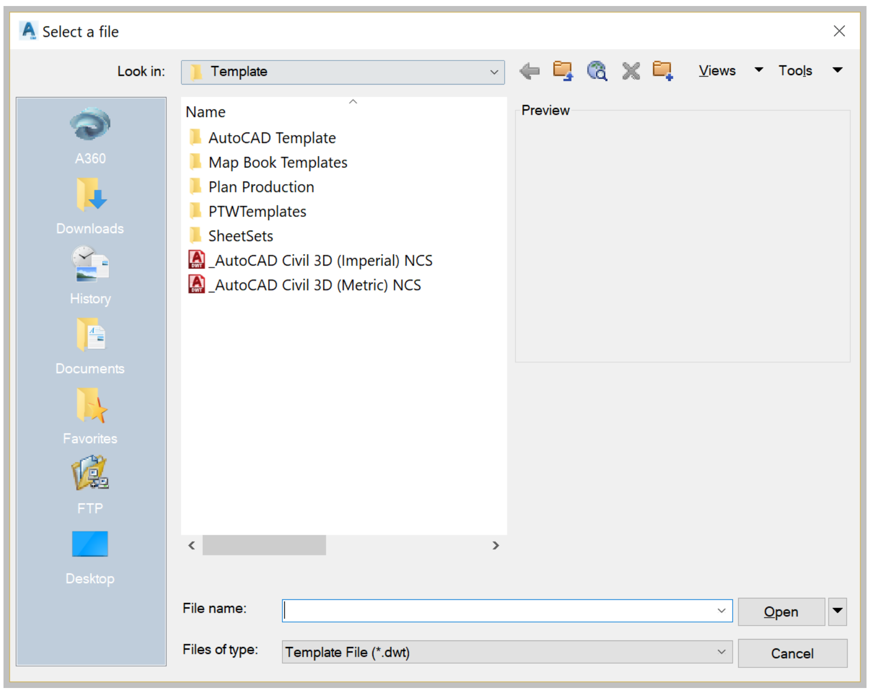



23. On this tab, we also have user-defined settings such as default templates and **temporary saved** files. For instance, we can come in here and specify where to store our **automatic save** files.
24. We can also specify the default template to use when we create a new drawing. We currently haven't specified one. If we haven't set a default template, Civil 3D will prompt us to choose one every time we try to create a new drawing. Let's change that and specify which template to use by default every time we hit the New drawing button or type **Qnew** at the command line.

NOTES

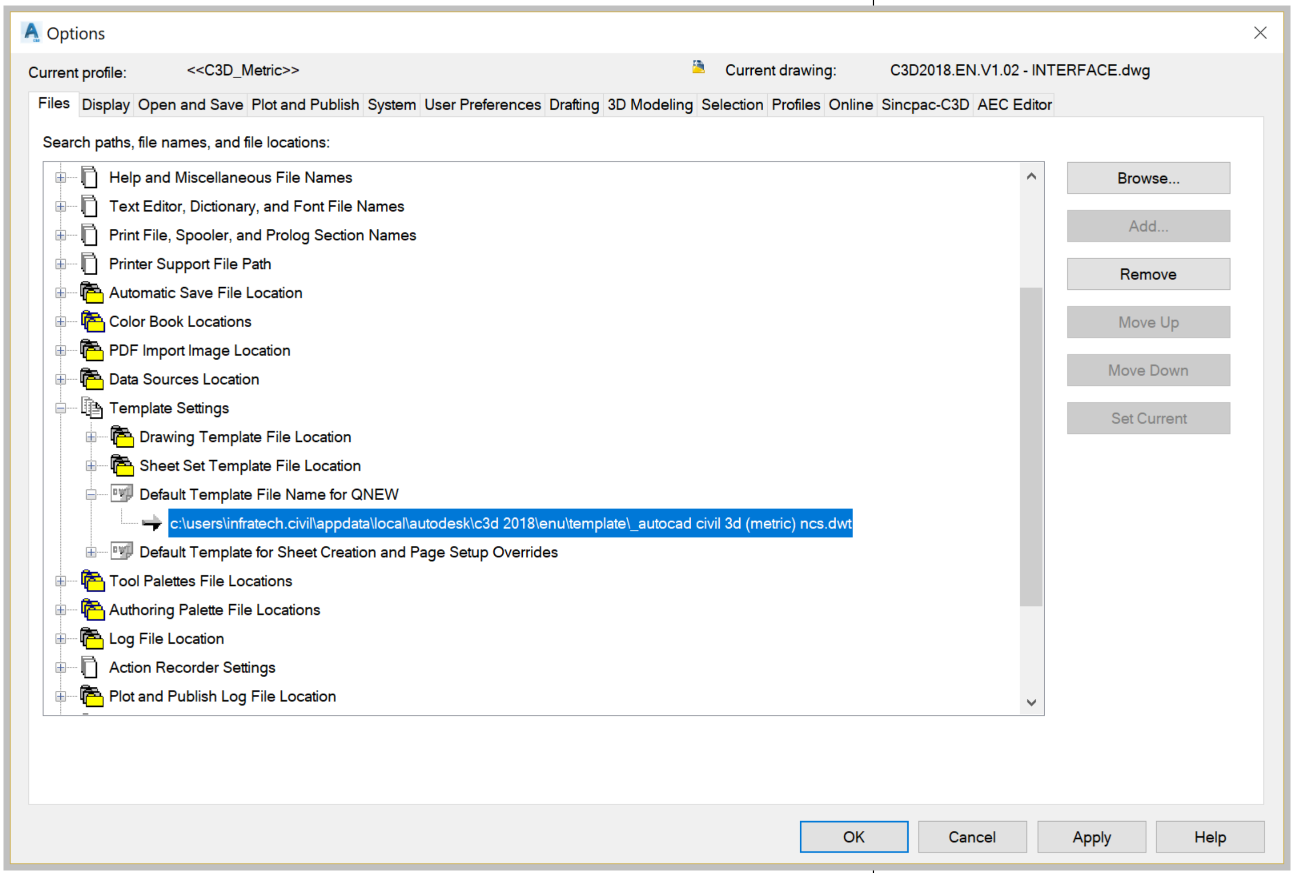



25. Once we hit browse, we are automatically directed to the default folder containing all Civil 3D templates.



NOTES

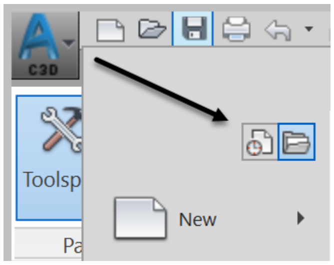
26. Choose a default template file, metric or imperial setup, depending on your geographic location or jurisdictional requirements. If you don't have a template set up for your organization, the default Civil 3D templates are a good place to start. You should, however, improve them. Organizations such as **Infratech Civil** can help you set up your own standards or provide you with improved pre-made ones. Most of our designs will be in metric or imperial units. We then need to choose the default template provided out-of-the-box by Civil 3D, or choose our own, if we have created one.



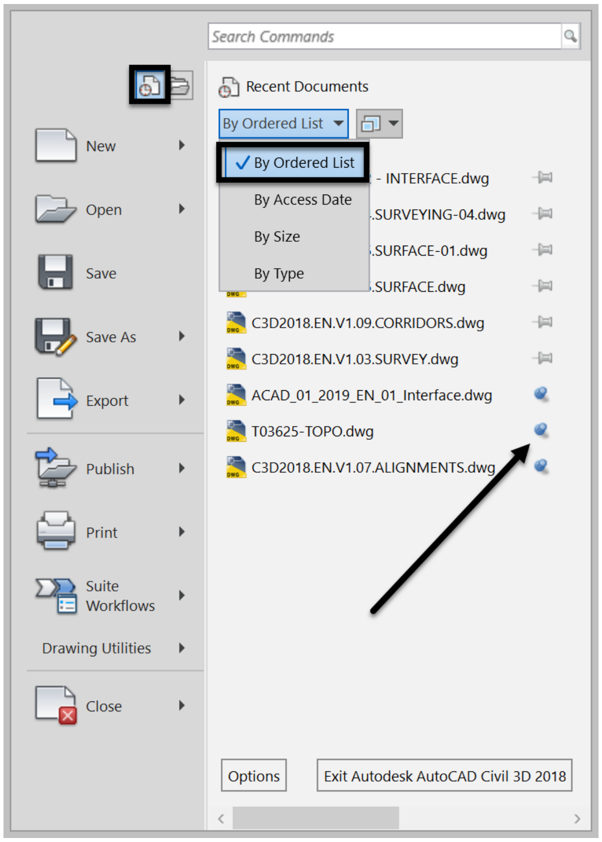

27. A default template is now set and will be used each time we hit the **Qnew** command on the ribbon or enter it at the command line.

28. Don't forget to cancel, when exiting this window, if you do not want to change any settings during this training.

29. Let's go to the **Application Menu**. Don't forget that, that is how we got here. There are also other ways to access the **Options** windows, and the **Application menu** is just one of them. At the top right, we have two icons.

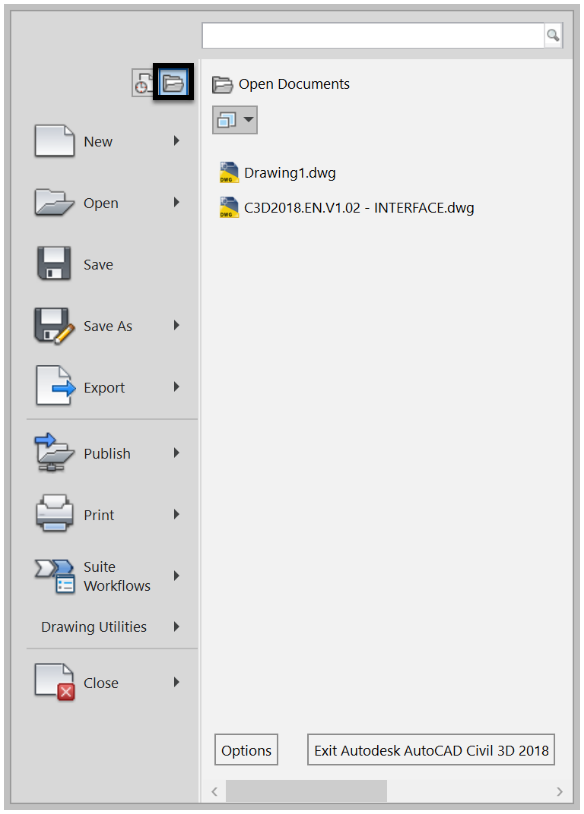



30. The first, the white icon to the left, allows us to display the recently opened files, sorted in chronological order. We can pin files if we want them to be always displayed in this list. Just like we did on the **Start window**. We can also sort files by list, access date, size or type.



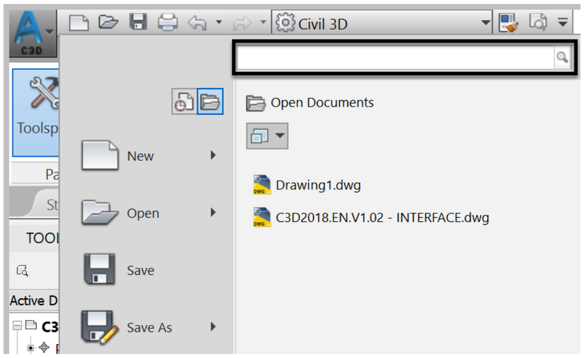

NOTES

31. To the right, we have an option to display only currently opened files. We see on the list the two files that are currently opened.



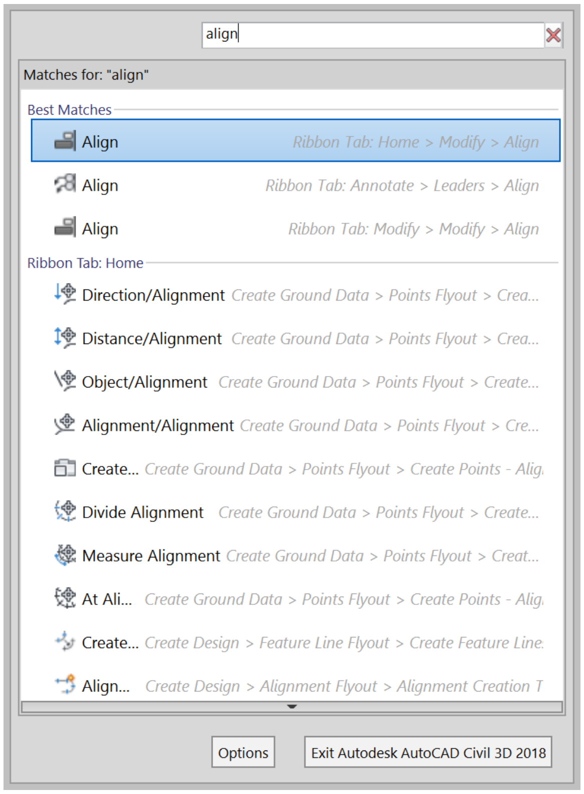

32. Once again, we have an option to change the display mode to list small or large icons. We can also sort files by date or size.

33. Lastly, from the **Application Menu**, at the top right, we have the **search commands** textbox, where we can perform specific searches.

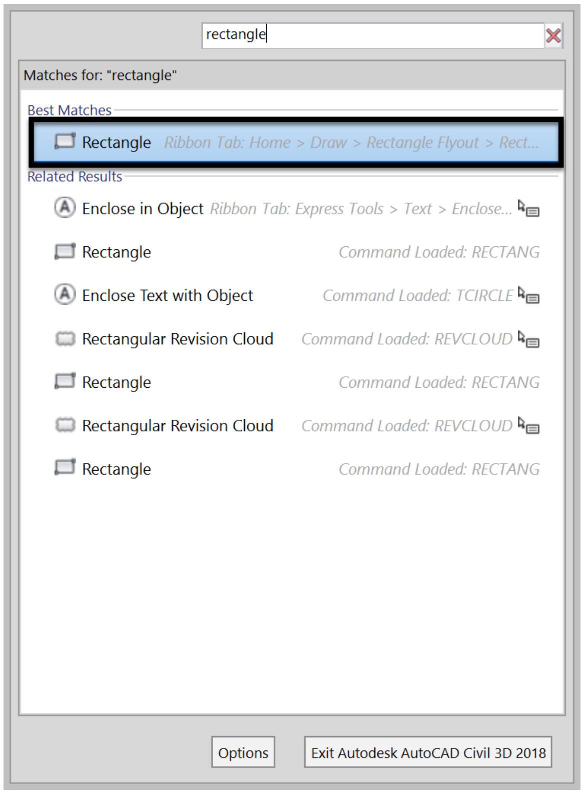



NOTES

34. For example, let's say we want to design an alignment. By just typing align in this box, we have a whole list of suggestions related to the word **Align**. Look at the list of commands below. We can create alignments using all kind of methods.



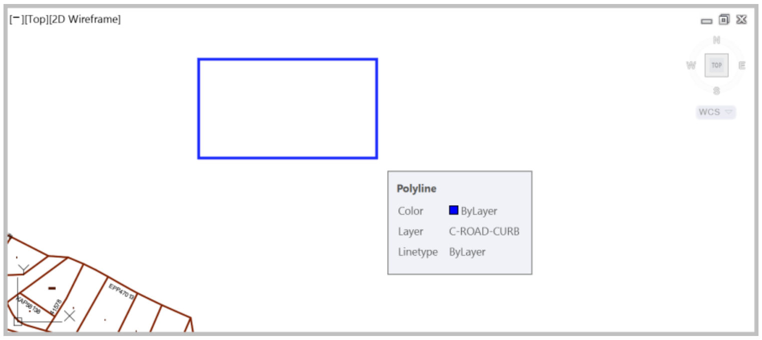

35. Let's try the word **Rectangle**. By simply typing that word, we have a suggestion to run the **Rectangle** command from the ribbon, and many other options.




NOTES

NOTES

36. If we click on it, we can come to the drawing area and simply click the start and end point and draw a rectangle. So, these are actual commands we can run, it is not just an information help menu!




37. The **search commands** box is very useful when we are looking around for commands that we don't know or don't recall.

38. Select the polyline that we have just created and erase it using the delete button from your keyboard.

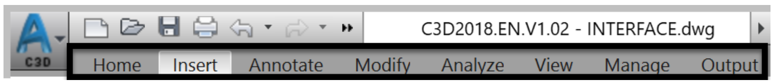
39. That concludes the section on the **Application Menu**. Next up, the **Ribbon**!

## 2.3 Ribbon

The ribbon is a relatively new feature for long-time Civil 3D users. It's a rare sight these days, but you can still encounter some users who prefer to use the good old scrolling **menu bar**.

1. Several items constitute the **ribbon**. First, at the top are the **tabs**. They are used to group commands with similar functions.

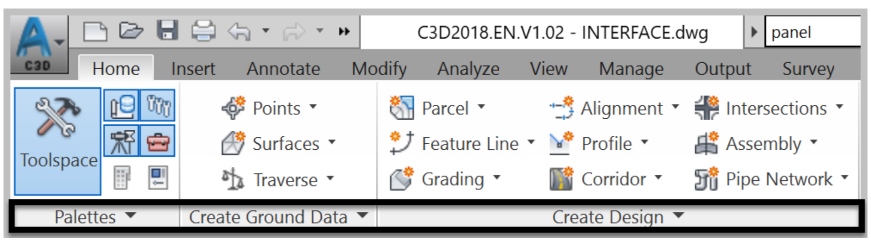
2. For example, the **Insert** tab has commands that help insert new or existing data into the drawing, like blocks, survey points, aerial imagery or other external references.



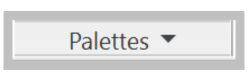

3. On the **View** tab, we can manage the display of the **Drawing Area**. We can use 2D or 3D displays, a different **UCS**, visualize from different angles or browse in 3D mode.

4. Additionally, each tab is grouped further into panels. For example, on the **Home** tab, we have **panels** called:

- **Palettes,** where we can organize the Civil 3D command palettes.
- **Create Ground Data,** for creating points and surfaces.
- **Create Design,** for design related commands.

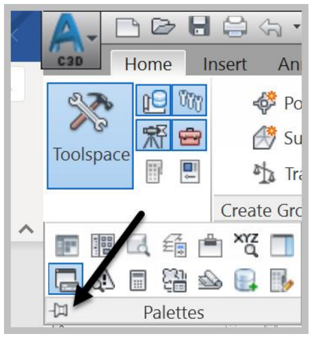



5. A panel can be expanded to show more commands by clicking on the downward arrow to the right of its name.

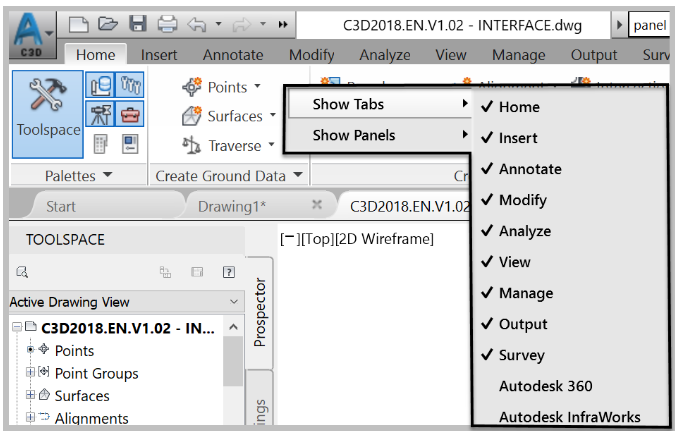


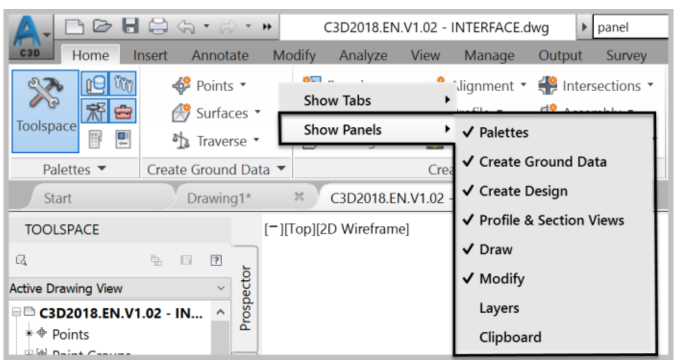

6. We can expand and pin the panel to expose additional hidden commands. Unpinning the panel will collapse it to its original state.

NOTES



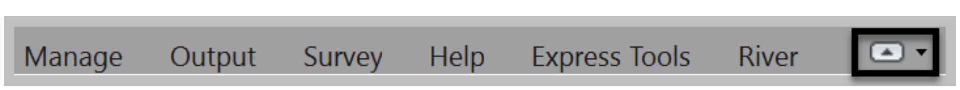

7. The ribbon can be further organized by unloading items. To do that, **right-click** anywhere in the ribbon and uncheck the items to unload or check the items to display.




8. To make the **ribbon** more readable, especially when working at a lower than recommended display resolution, unload unneeded **panels**.

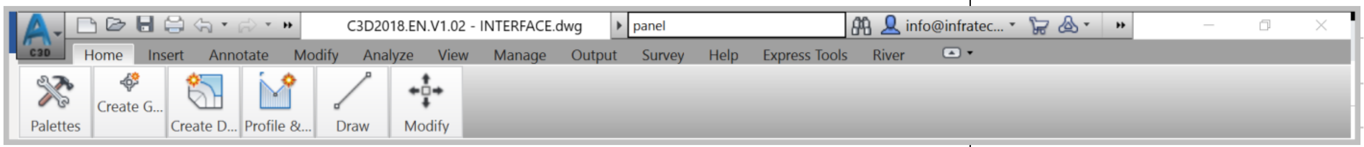
9. An additional way to manage the ribbon is by collapsing it by using the **Cycle Through** button, the upward triangle to the right.



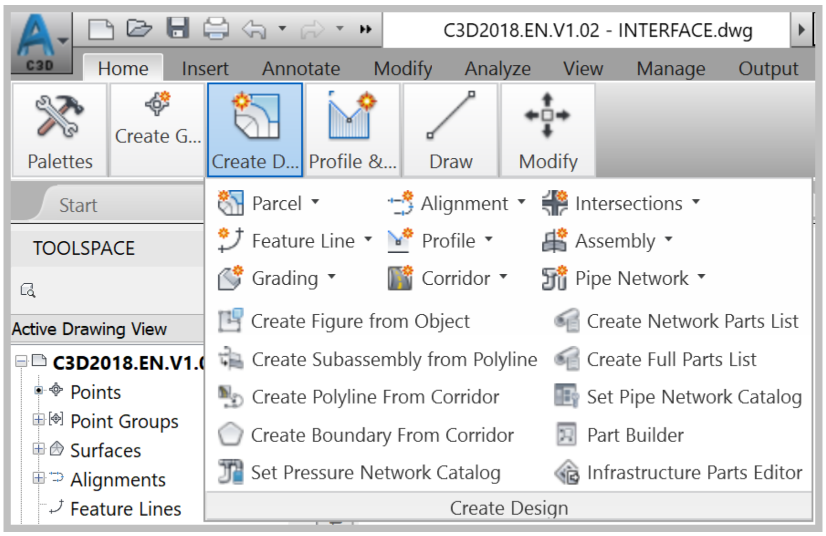

NOTES

NOTES

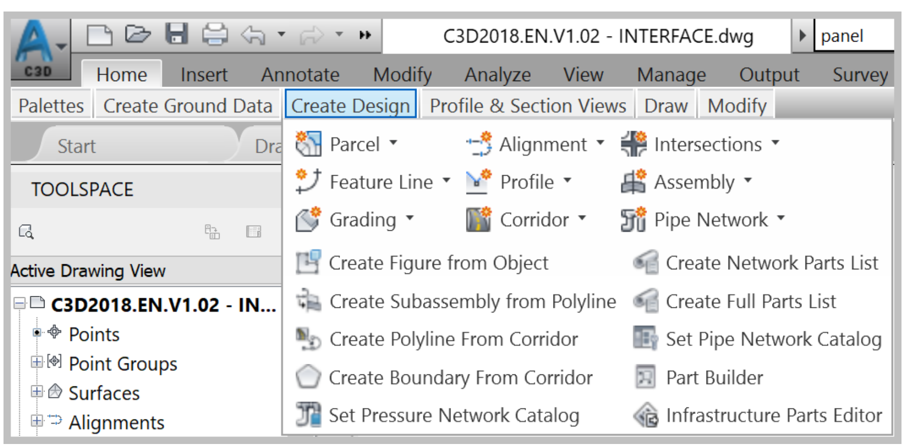
10. A first click will collapse and make the ribbon compact. If you look closely, you will see that we still have access to all commands. They are just more compacted.




11. By clicking on the **Create Design** panel the collapsed commands are expanded.



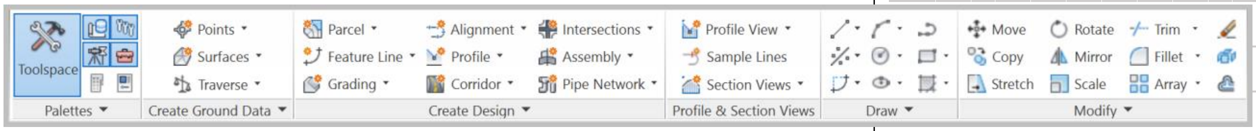

12. Click on the **cycle through** arrow a second time, and the **ribbon** will adjust again.

NOTES

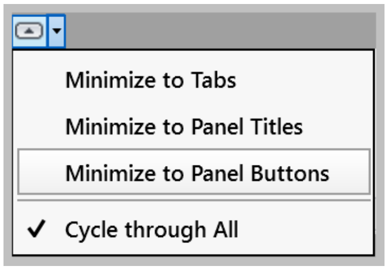
13. A third click will completely collapse the ribbon and minimize it to tabs only.


14. Clicking on any tab will display the full ribbon.




15. One last click on the **Cycle Through** arrow, and we complete the cycle and return to the original state of the ribbon.

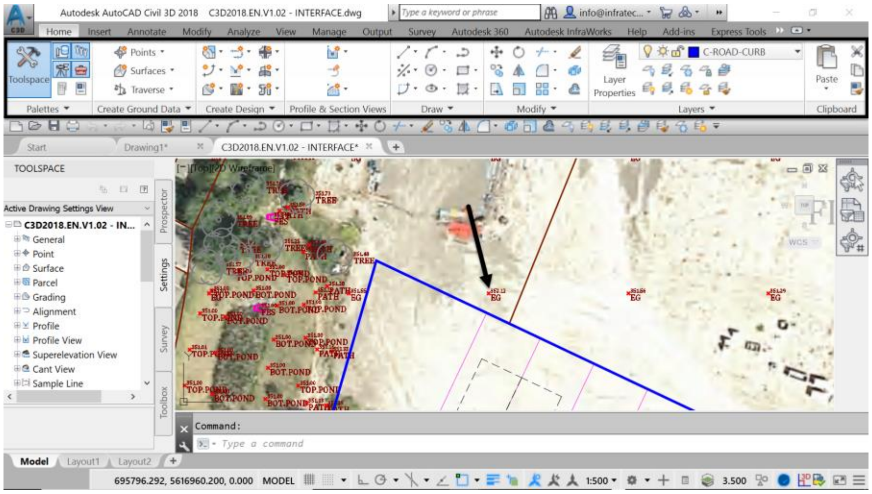
16. Instead of going through the four cycles, we can just switch back and forth between the full display and one of three states of our choice, between **Minimize to Tabs**, **Minimize to Panel Titles** and **Minimize to Panel Buttons**.



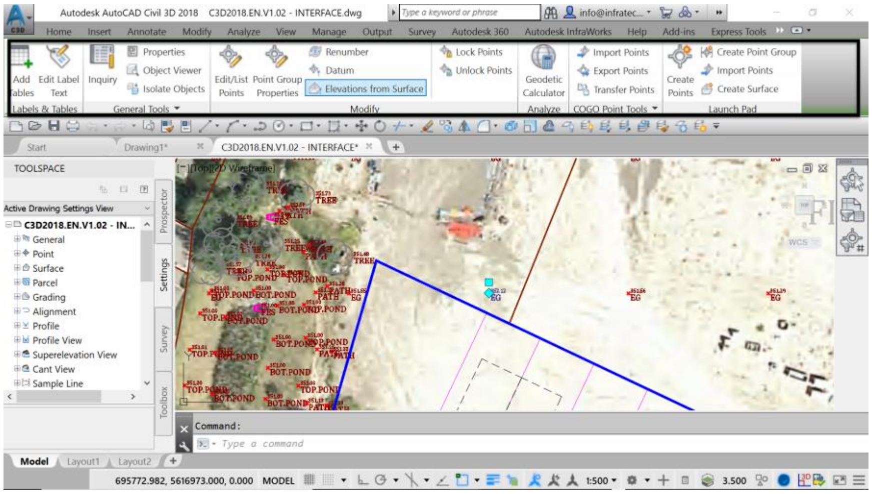

17. Last but not least, one of the most significant benefits of the ribbon is its **contextual responsiveness**. It adapts to present you with a set of commands depending on the currently selected entity. For example, if you are working with **points**, the ribbon automatically changes to present you with the commands related to **points**.

NOTES

18. While working in the same interface drawing, zoom to a survey point, for example, one of the **EG** points to the northwest of the proposed site. Take note of the current state of the **Ribbon** with the **Home** tab activated.



19. Now, click on the point. Notice how the ribbon adjusts and only shows commands related to points operations such as editing points, labels styles, importing and exporting points.

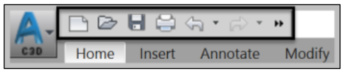


20. That concludes the section on the **Ribbon**, up next, the **Quick Access toolbar**.

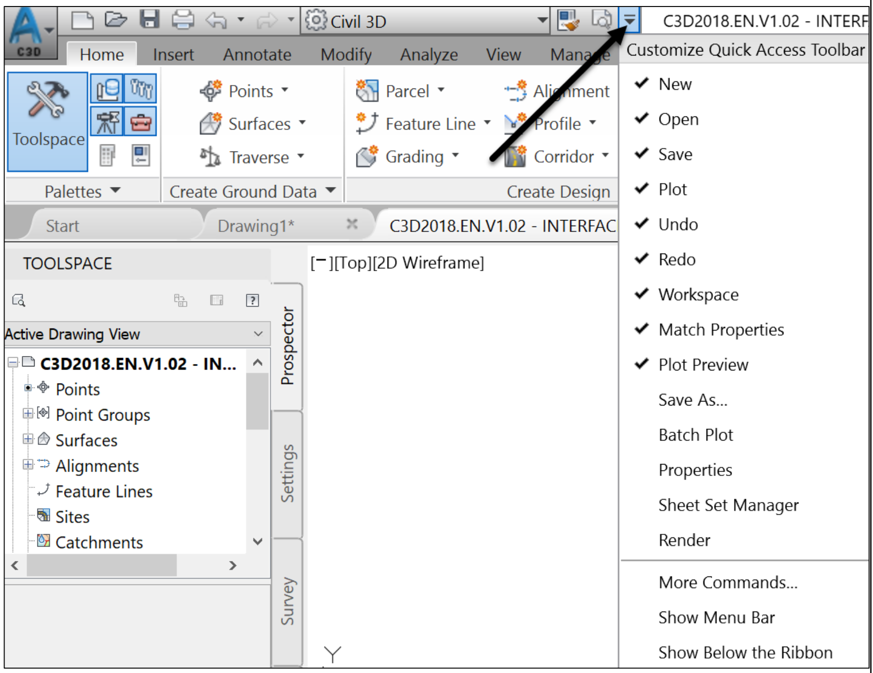
## 2.4 Quick Access Toolbar

Next item on our perusal of the Civil 3D interface is the **Quick Access toolbar**, a typical windows application feature. We have it in most office programs.

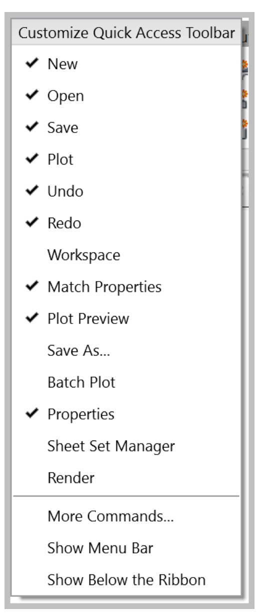
The **Quick Access Toolbar** gives us an option to stack a set of frequently used commands, that we want to have at our disposal at all times. By default, it shows common commands such as **New**, **Open**, **Print**, **Undo,** and **Redo**.




1. If we click on the arrow to the right, we can see a list of additional commands that we can add to the quick access toolbar.




2. Let's add a frequently used command like **Match Properties** and remove one that is not really used that often, like **Work Space Switching**, since we can also access it through the status bar, in the bottom right corner.

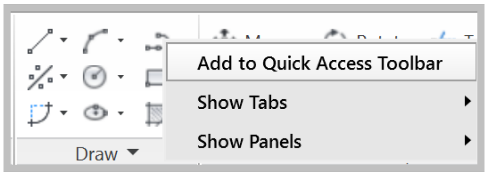



NOTES

3. The new **Quick Access Toolbar** responds to our request and adjusts accordingly.

4. One thing that we like to do is to relocate the **Quick Access Toolbar** below the **ribbon**, to have more real-estate, and display as many commands as we need. We will see in a second why that is important.

5. For the time being, let's keep adding commands to our **Quick Access Toolbar**. To add a command from the **Ribbon**, right-click on it and select **Add to Quick Access Toolbar**.

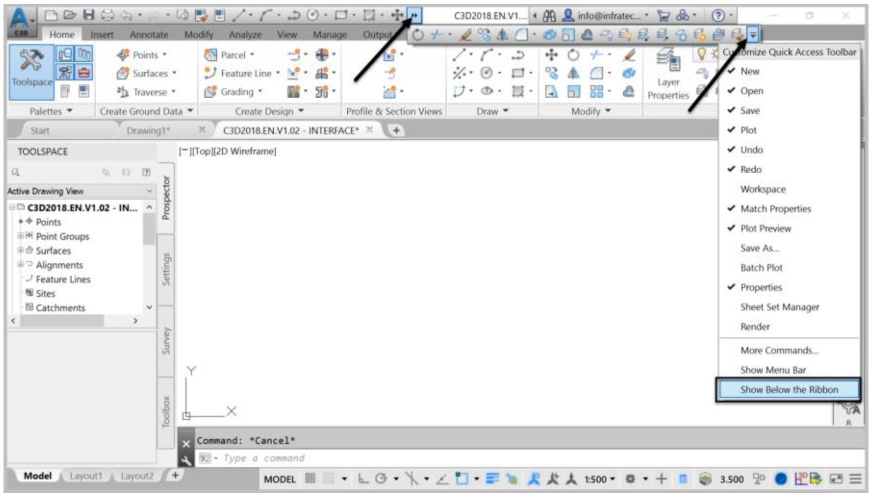



6. Let's add the frequently used ones such as drafting commands (line, polyline, **rectangle**, **circle**, and **hatch**), modification commands such as (**move**, **rotate**, **trim** and **extend**, **copy**, **mirror**, **fillet**, **scale** and **offset**), layer management commands (**turn on** and **off**, **layer isolate** and **un-isolate**, **thaw** and **unthaw**, **make current** and **match layer**). We typically also add a few design commands that we frequently use. It would be a good idea to organize them by groups, to make it easier to locate them.

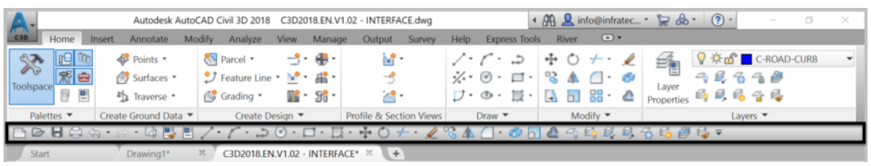

NOTES

NOTES

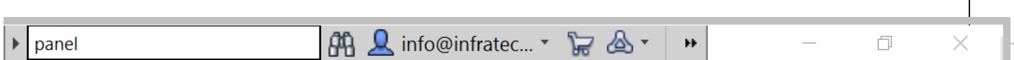
7. After adding all these commands, you will notice that not all of them are displayed. That is because we just don't have the necessary real-estate to show them at the top of the **Ribbon**. To fix that, we will need to go to the **Quick Access Bar** display options and choose to display it below the **Ribbon**.




8. We will then have all the commands displayed and readily available, regardless of the ribbon tab we are working with. This will save us time by not having to switch back and forth between tabs to find our most used commands. In the long run, this adds up to a gain in productivity.



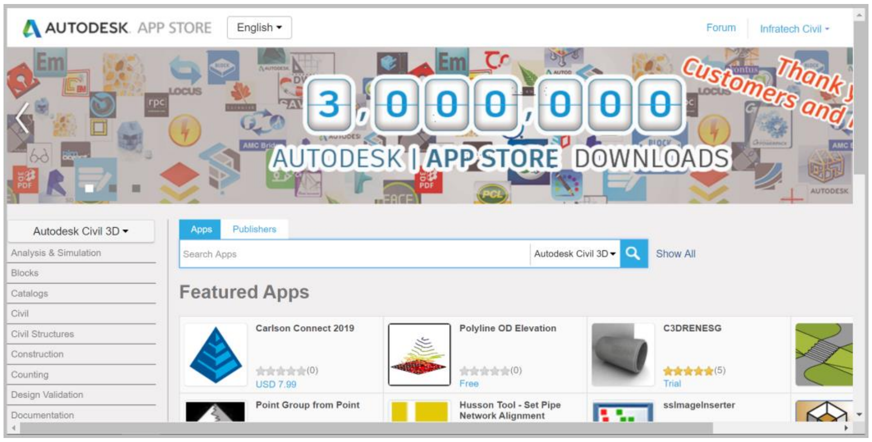
9. At the top, to the right of the quick access toolbar, we have a few tools mostly related to searching and Autodesk account management. The first textbox is for searching online for help. If we type **Line,** for example, it brings up a few suggestions regarding the word line.




10. To its right, we have the **Online Search** command, which takes us to the online help itself, to search on there.

NOTES

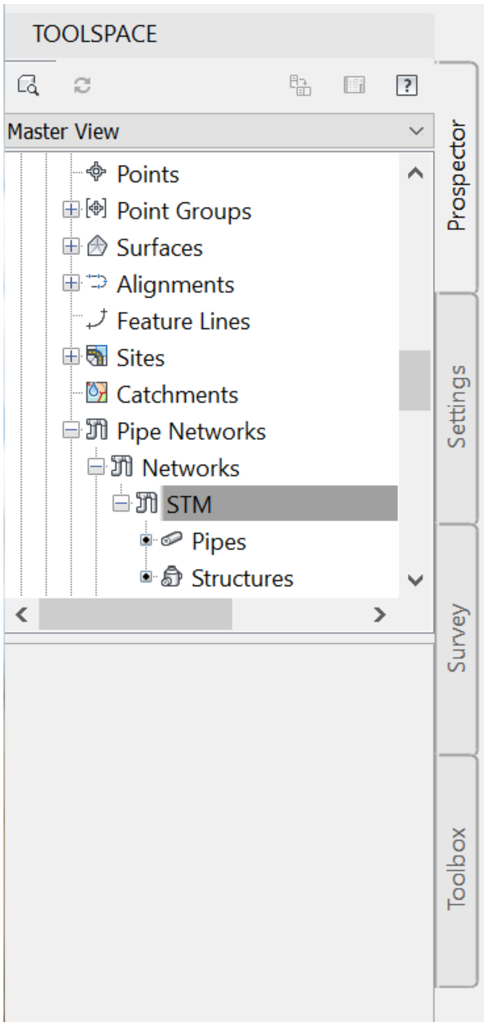
11. Next, you can access the Autodesk app store  to browse for add-on apps, created by third-party developers. These are useful apps you can add to Civil 3D, to perform tasks that are not natively possible with out-of-the-box. AutoCAD Civil 3D is a great application, but no software does everything out-of-the-box. We always need to find some workaround for some specific tasks. Some of these apps are free while most are paid-applications.




12. Let's close this window. Next, we have our **Autodesk Account portal** entry window. There, we can pretty much do all our Autodesk Account management, as we've seen in the **Start Screen** section of this lesson.

## 2.5 The Toolspace

The Toolspace is the main window for managing data, settings, and styles in Civil 3D. It is accessible from the **Ribbon**, on the **Palettes** panel.




The Toolspace is organized in four tabs.

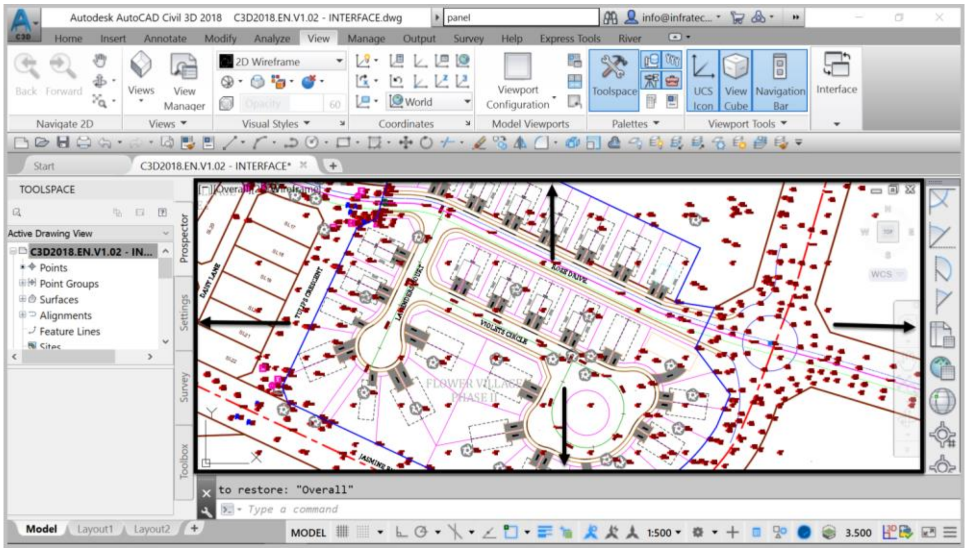
a. The **Prospector** tab, where we will work most of the time, is used to manage data and create design entities. It has several collections or items, where Civil 3D data is organized. Among the collections, we can find Points and Point Groups, Surfaces, Alignments, Sites, Catchments, Pipe Networks and many more.
b. The **Settings** tab is used to manage style and settings.
c. The **Survey** tab is used to import and manage survey data by creating databases including the survey database, the equipment database, the figure prefix database, and linework code sets.
d. The **Toolbox** tab is used to create reports. It can generate output for points, surfaces, parcels, pipes, profiles, and more.

NOTES

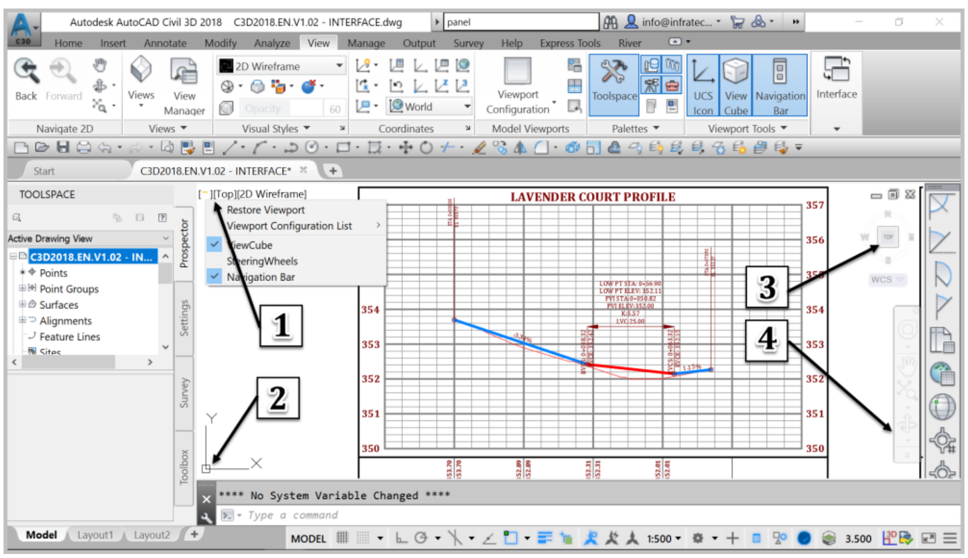
NOTES

## 2.6 Drawing Area

The Civil 3D **drawing area** is the central rectangular area where most of the action takes place. It is the center-stage where we draw, visualize, select and manipulate objects.




In the **drawing area,** there are some elements that need to be noted. For more details on the elements, you can refer to the **AutoCAD Level 1** course. But let's mention them briefly.




1. In the top left corner, **Viewport and Visual Style** controls the view directions and styles.
2. The **UCS** or **User Coordinate System** is the active coordinate system that establishes the XY
3. or work plane and the vertical direction for drawing and modeling.
4. The **View Cube** is where you can see drawing elements from different view directions by rotating the view.
5. The **Navigation Bar** contains additional tools like **Pan**, **Orbit**, and more.

That concludes the section on the Drawing area, up next, the **Command Line**.

## 2.7 Command Line

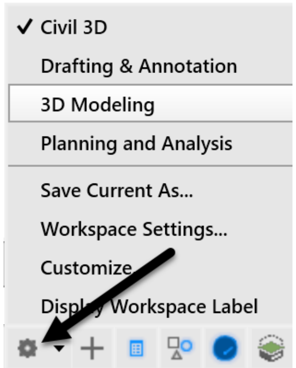
The **Command Line** is the textbox at the bottom of the drawing area. It is the main intake for any command transmitted using the keyboard. It is displayed or turned off by simultaneously pressing **CTRL** and number **9**.


## 2.8 Status Bar

Just below the **Command Line**, we have the **Status Bar**. It has a set of drafting and display tools that provide ease of use and precision drafting.

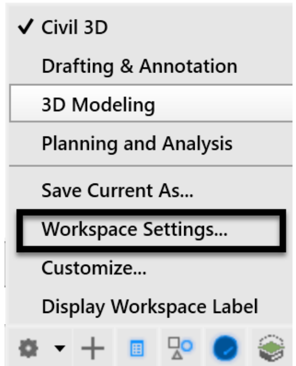

These settings and tools are discussed in more details in the **Basic AutoCAD** course. Nonetheless, let's discuss one of them, the **Workspace Switching.** As previously mentioned, Civil 3D is built on top of Map 3D, the Autodesk's **GIS** (Geographic Information System), which in turn is built on AutoCAD. So, as it turns out, if you have Civil 3D, you can just switch between multiple working environments, like basic AutoCAD 2D drafting, 3D modeling, Map 3D GIS analysis or Civil 3D design mode.

Simply click on the **Workspace Switching** icon and pick the appropriate work environment. You can also make changes to your workspace by clicking on **Customize,** then on **Save Current As,** to rename the workspace.

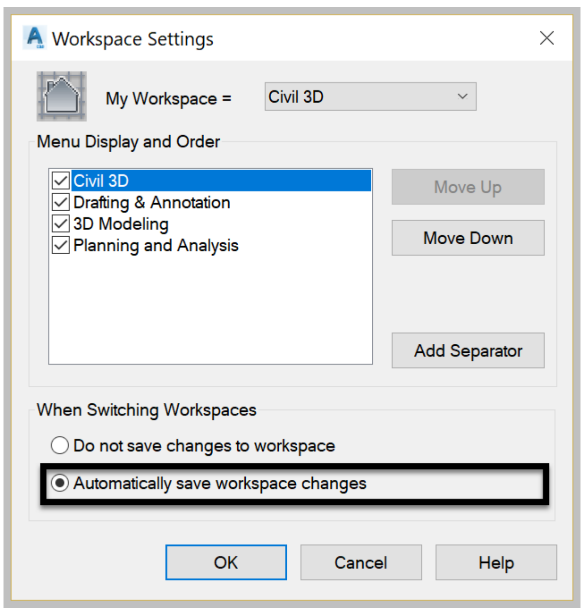



NOTES

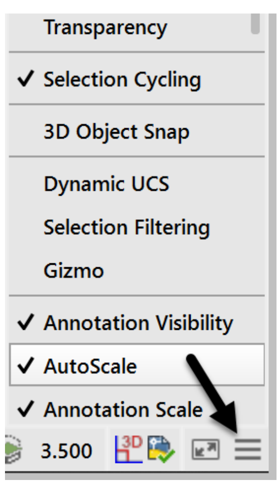
Another option you should be aware of, as mentioned in the **Quick Access Toolbar** section, is saving the changes that you are making to the interface. To have Civil 3D memorize these changes, click on **Workspace Settings**.




From there, select the option to **Automatically save workspace** changes. This action should not be performed if you are not currently working on your usual and personal workstation.




Finally, note that all **Status Bar** display options are available from the button  in the right bottom corner.




NOTES

NOTES

# 3 STYLES AND TEMPLATES

## 3.1 Definition of Styles

Styles are the essential elements that allow us to control the appearance of Civil 3D objects such as points, alignments, profiles, etc., and their label styles, the information that goes with these objects.

The process and the different dialog boxes used for creating these styles are often similar from one type of style to another.

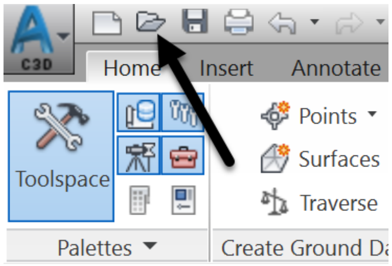
## 3.2 Definition of Templates

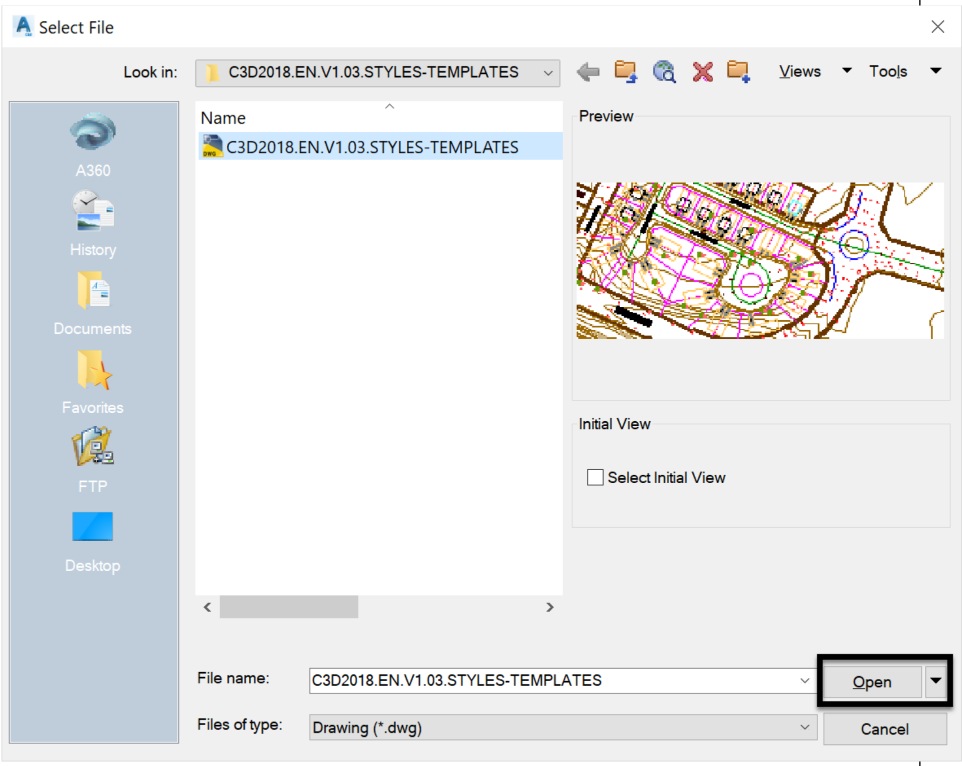
Templates can be defined as AutoCAD Civil 3D files (**dwt** format) that contain the standards for creating and displaying Civil 3D objects. A template file can increase productivity and consistency, by avoiding the need to repeat the same operations from one project to another. Using a template file allows the conservation of layer standards, line types, colours, text sizes, and more, throughout multiple drawings.
Generally, a pre-existing template file should be used each time a user is starting a new project. By doing this, we have a leg up in the design process by reutilizing a set of pre-established standards, instead of recreating them from scratch.

## 3.3 Creating templates

In this exercise, we will use already existing styles, modify them, and create a file template for our future uses:

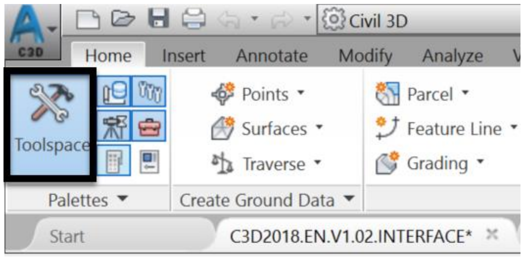
1. Open the **03.01-Styles-Templates dwg** file in the **Lesson 03** practice folder.





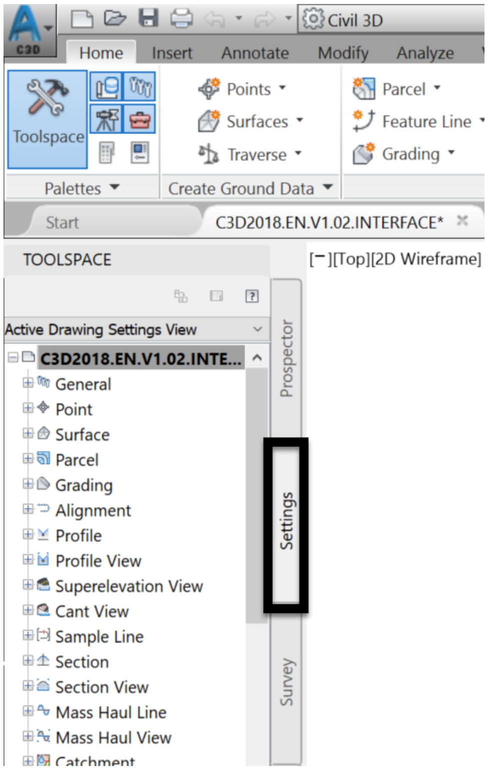

2. Let's start editing the file and establish some standards for object styles, units of measurement, labels, and the like. The creation and maintenance of template files is a never-ending process; It's just a fact of life that we are always striving for a higher standard. Besides, projects and regulatory agency requirements are always evolving; our template needs to adapt accordingly.

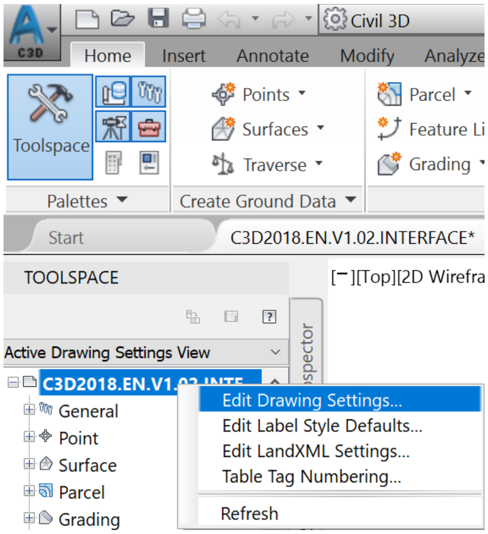
- First, make sure the **Toolspace** is active and visible.




- Now, switch over to the drawing settings tab. Right-click on the file name, then **Edit File Settings**. Most standards (settings and styles) are created here. On this tab, we can create, copy, modify, or delete styles.

NOTES

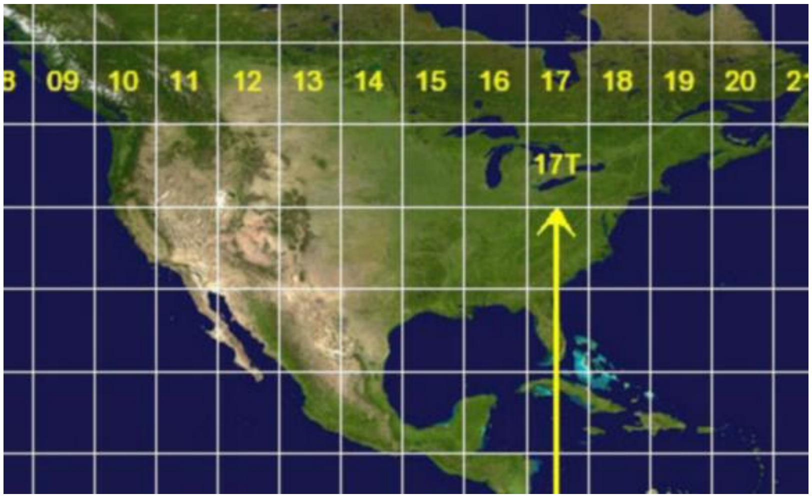


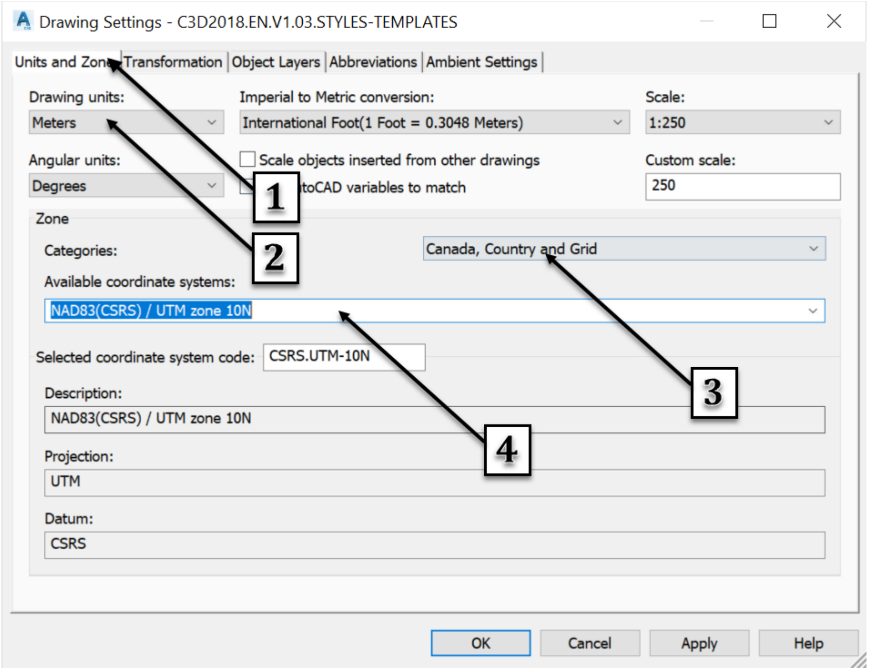

3. Under the **Units and Zone** tab, you can define parameters such as linear and angular units, annotation scale and the coordinate system.

4. Let's set the coordinate system for this project. We will see why this may be helpful on the next tab. This site is in the **Universal Transverse Mercator** (UTM) projection system, **Zone 10**. Therefore, we chose the **CSRS.UTM-10N** system. We recommend that users consult their local Geographic Information System authority, for a recommended coordinate system for each project.

NOTES




5. Next, we need to set-up the drawing to match the appropriate Units and zone.

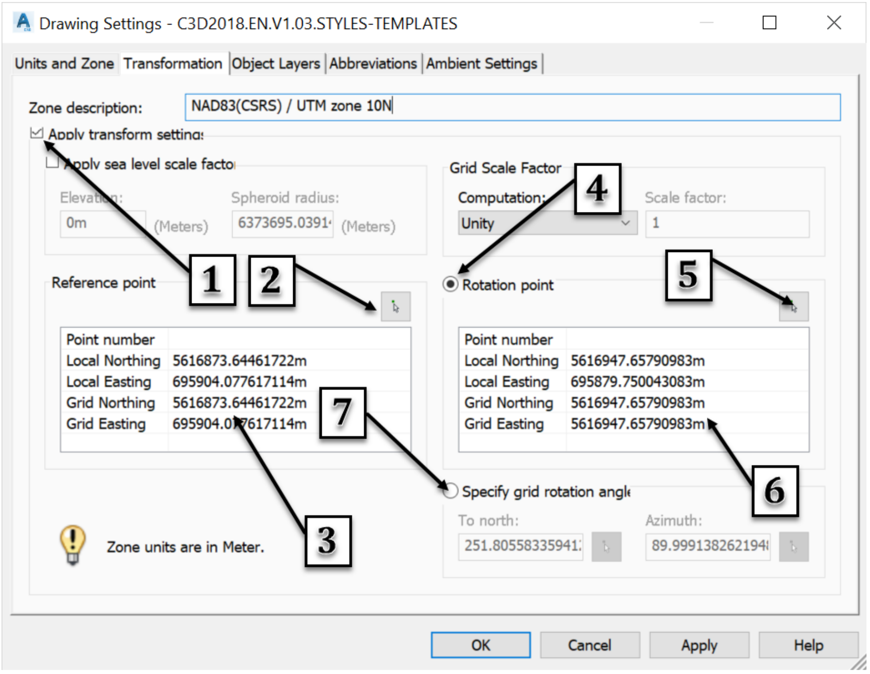



6. The **Transformation** tab gives us an option to transform the coordinate system specified on the previous tab to a grid system. For example, if we have a survey data that was collected using an arbitrary local system, we can transform that data into a known grid system. To do that, we must have at least two known reference points (for **translation** and **rotation**). Before any transformation, a coordinate system must also be set on the **Units and Zone** tab, which we already did by setting the projection to **UTM North America Zone 10**.

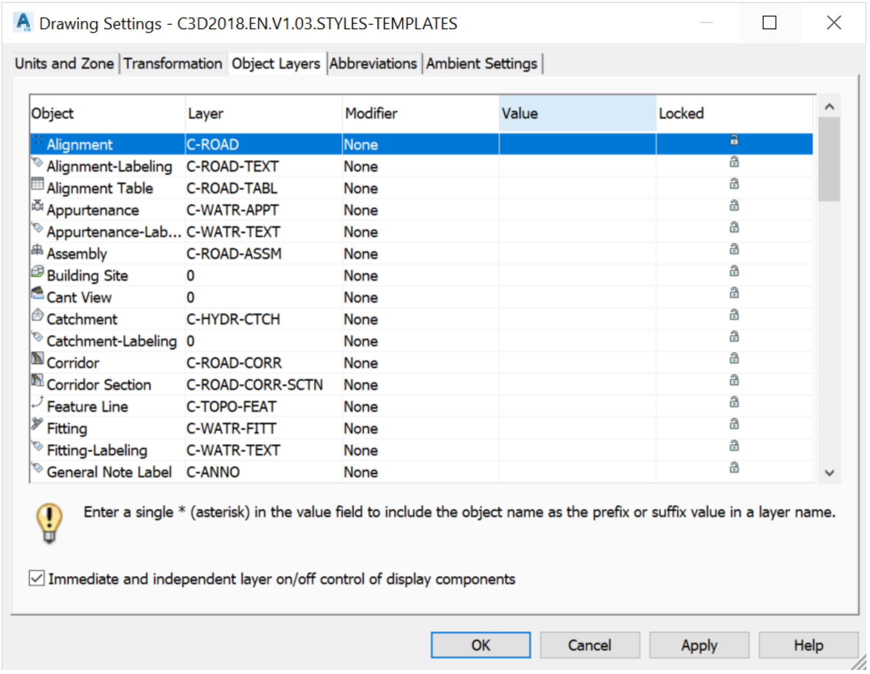
To make a coordinate transformation:

NOTES

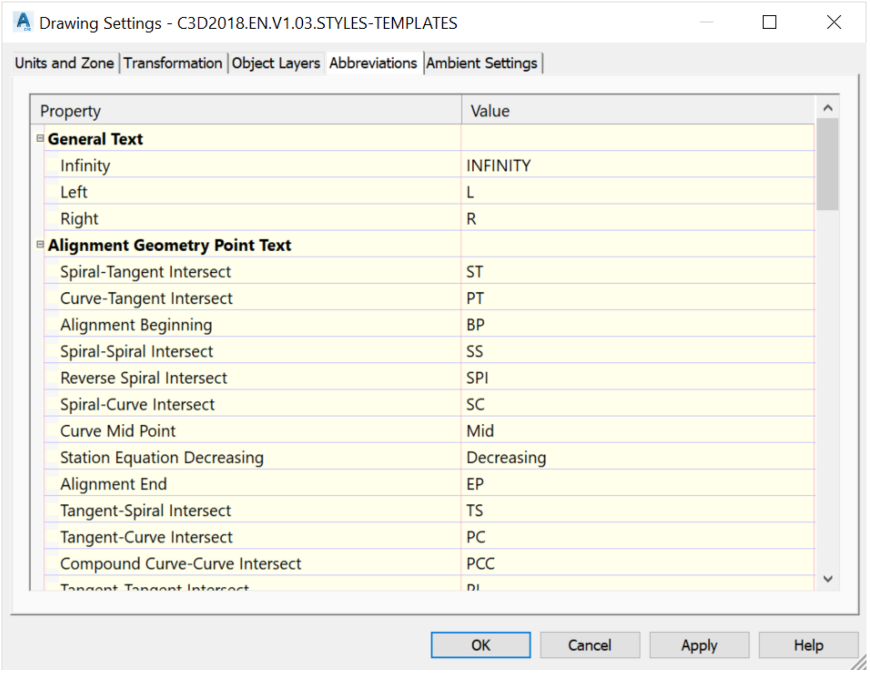
- First, activate **Apply the transform settings** checkbox.
- Then, pick a reference point in the drawing. Usually, this would be a point with known and georeferenced coordinates. In this case, our survey is already georeferenced. So, we are picking a random location, just to explain the process.
- After that, type in the known coordinates of the point you selected, in the drawing in the previous step. This will complete the translation phase of the transformation.
- Next, we need to perform the **rotation** phase. For that, activate the rotation option as shown on the **Transformation** tab.
- Then, pick the second known point in the drawing. Again, click on a random point, to choose the second georeferenced point.
- Finally, type in the known coordinates. Alternatively, you can use a rotation angle, if you know one.



7. On the **Object Layer** tab, we can specify default layers for various objects; when we create an object, a Civil 3D point for instance, it will automatically be placed on the layer specified here.



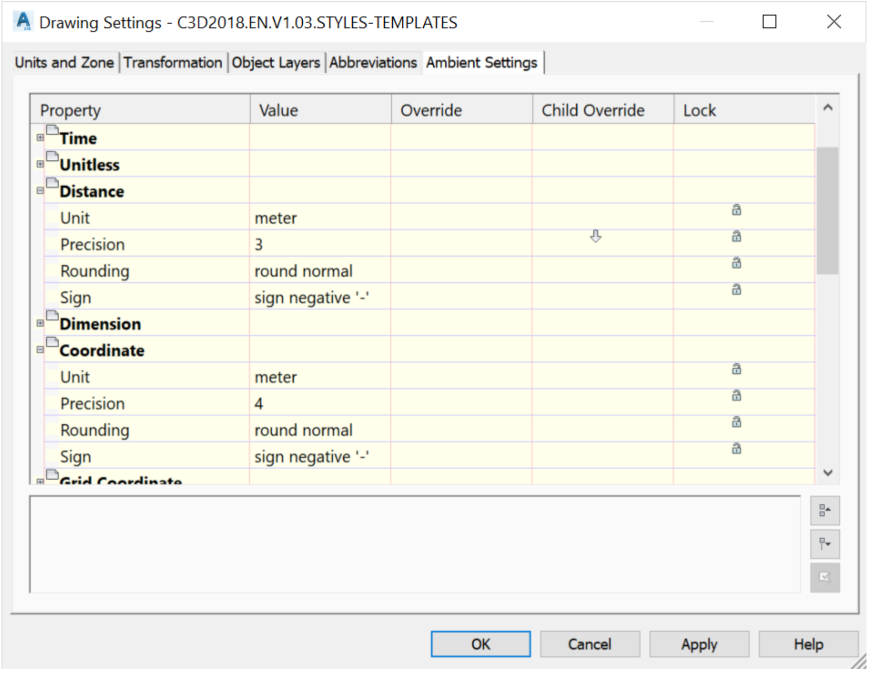

8. The **Abbreviations** tab allows you to manage different default abbreviations when creating labels and reports.




NOTES

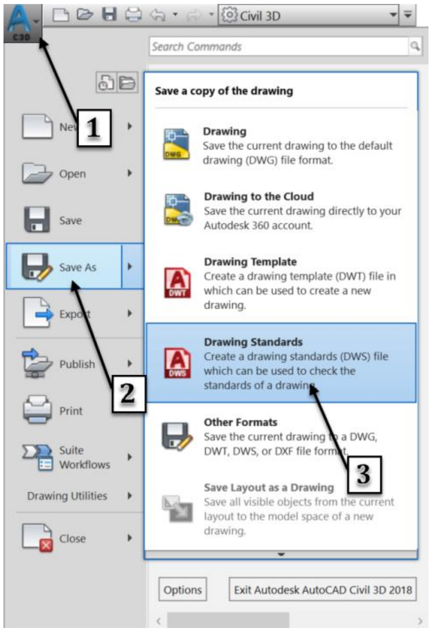
NOTES

9. The **Ambient Settings** tab specifies the current file's units and parameters.

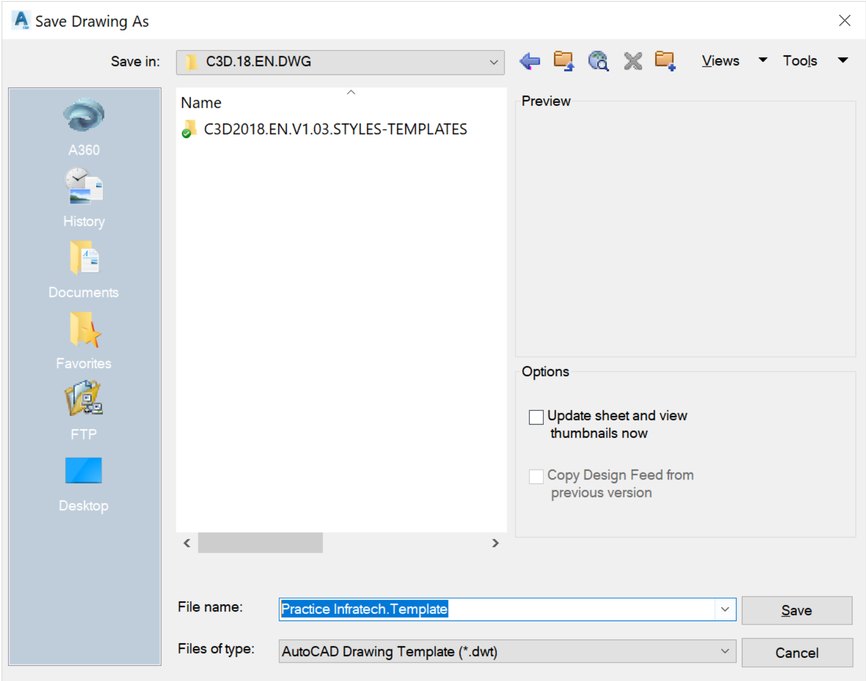



10. You can change the units to meters or feet, square meters or square feet, cubic meters or cubic feet, and so on, by clicking on the "**+**" sign and opening the sections for **Distances**, **Coordinates**, **Elevation**, **Area**, **Volume**, and **Station**.

11. Now that some parameters have been created, save the file in a **dwt** format by proceeding as follows.

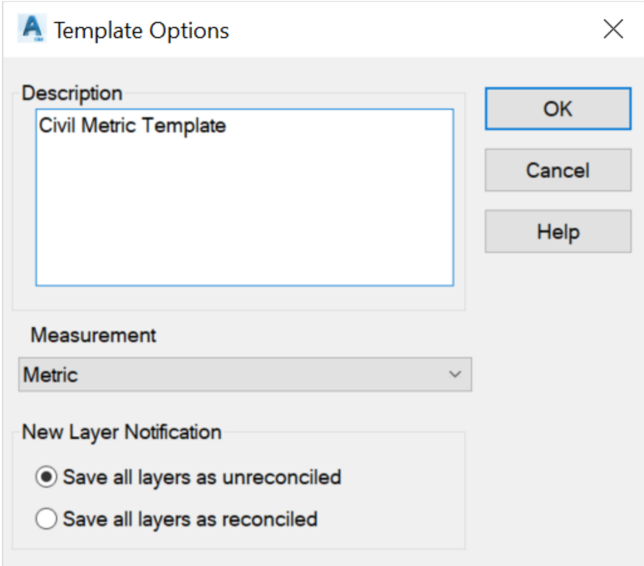


12. Choose the **Lesson 3** working folder, that is the **Templates** tutorial folder.




13. Choose a **DWT** file format and click on **Save.**

14. Then, enter the description of the template. This part is optional, but as we will recommend throughout this course to always try to provide self-explanatory descriptions.

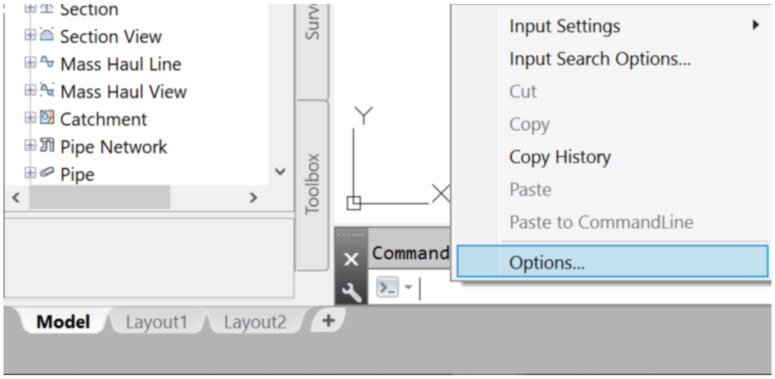



15. Click on **OK** to create the template. Now, every time that we are starting a project, we can open this template file and rename it. As an option, we can also define this file as a template. It would

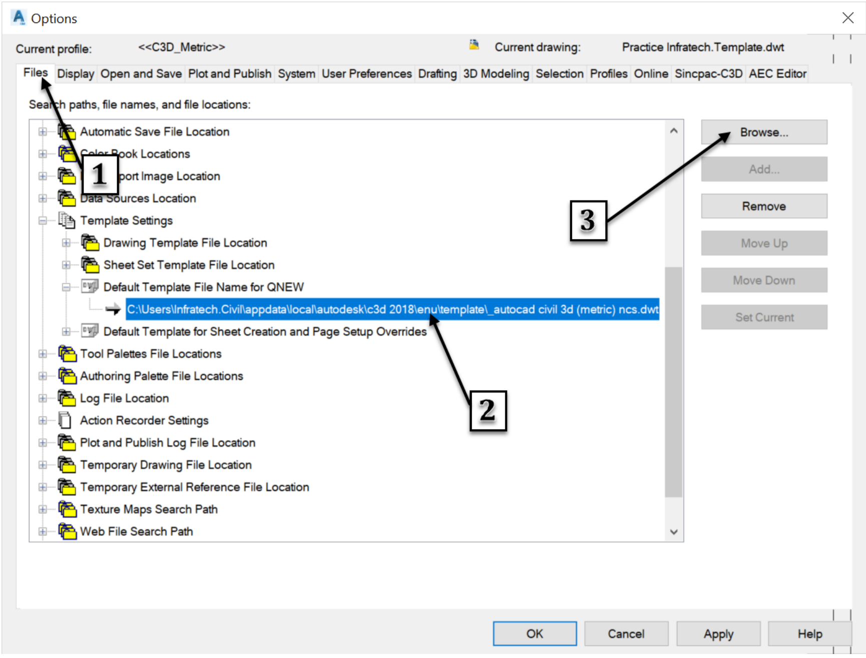
NOTES

be loaded whenever we create a new drawing (using the **QNEW** command). To set the default template, use the following steps:

16. Type **Options** at the command line or right-click in an empty space of the drawing area and choose **Options.**



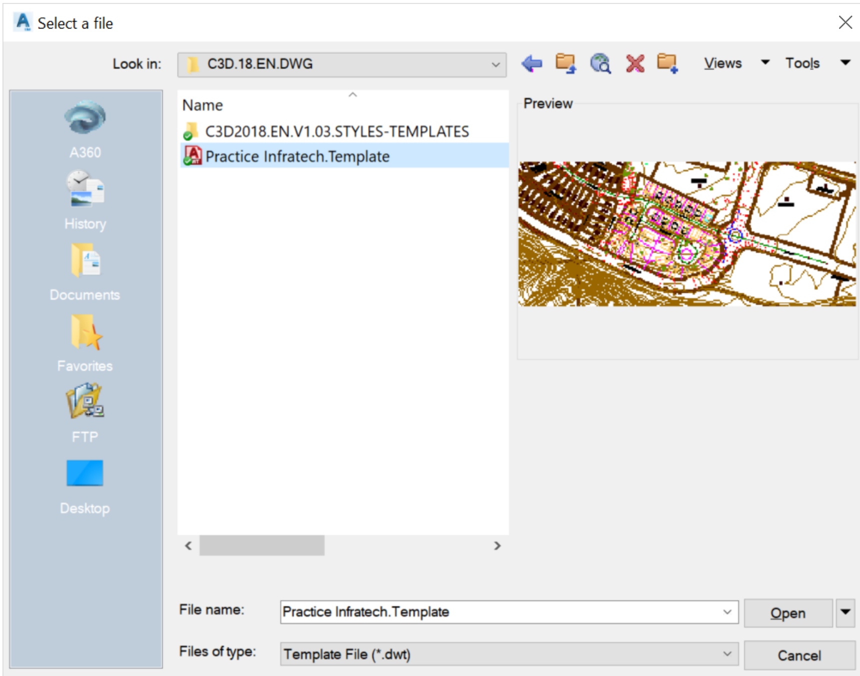

17. Browse to choose a default template file:



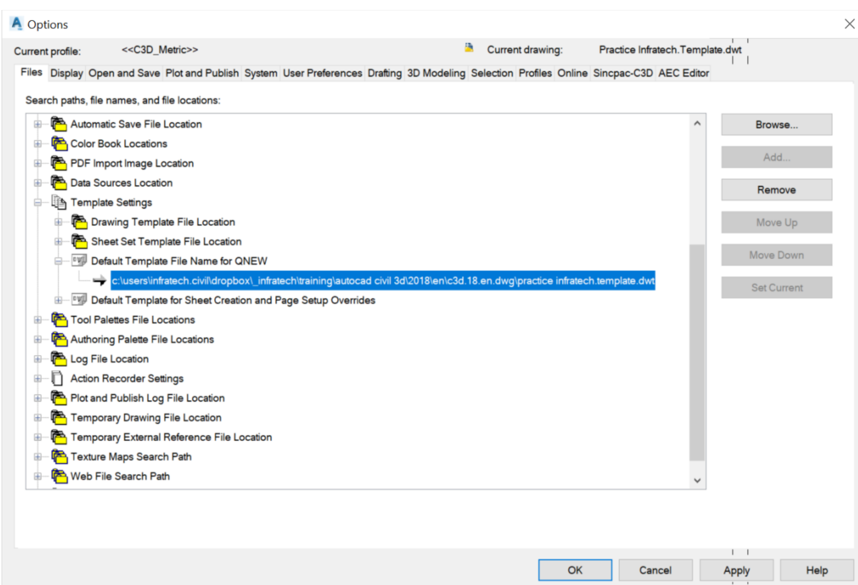

NOTES

NOTES

18. Then, find and select the template file you would like to use as a default and click on **Open**.




19. We have, at this point, defined the default template file to use each time we create a new drawing. For now, click on **Cancel** as we are just practicing and do not wish to change any setting that will affect your production.




20. Now, we know how to create a template file and how to set it as the default for creating new projects.

21. The next step is to maintain and improve the template with new styles, as we are creating them.

NOTES

## 3.4 Creating Styles and Labels

In Civil 3D, most types of objects (points, alignments, profiles, and so on) have a style and a label style. Let's learn how to create them, starting with the example of a Point Style and a Point Label Style. The two names sound similar, but they are two different things that we will learn to differentiate.

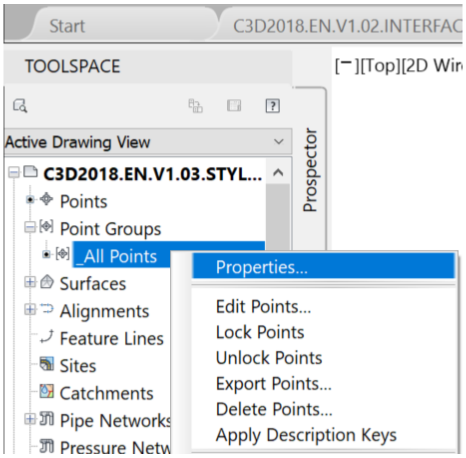
### 3.4.1 **Point Styles**

In general, most manipulations of styles and settings are done from the **Settings** tab of the **Toolspace**. However, to make the software easier, we will use the Autodesk Civil 3D **Prospector** and right-click menu. By establishing this habit, we will create a routine that will make it easier to learn the software.

As stated before, there are several ways to access Civil 3D's commands (right-click menu, ribbon, prospector, and the command line are the usual options).

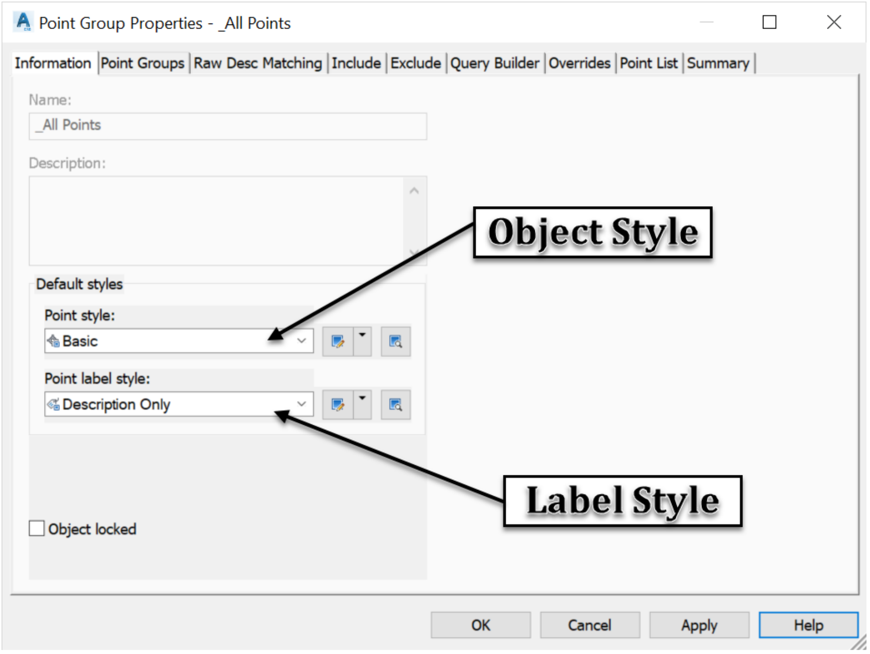
To create and edit point styles, proceed as follows:

1. Activate the **Prospector** tab and navigate to the **Point Groups** section.
2. Then, select **_All Points** and right-click on **Properties**




3. On the **Information** tab of the **Point Group Properties** window, we have two main boxes, each representing a type of styles:
   - The first one is **Object Styles:** it controls the appearance of any point object belonging to this point group.

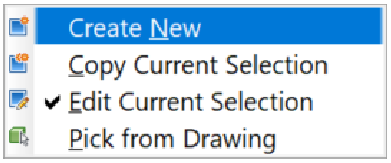
- The second one is **Label Styles:** It dictates the appearance of the text associated with the point, including coordinates, descriptions, point numbers and elevations.




4. In the **Point Group Properties** window, both types of styles can be created, copied, or edited. Let's create a new point style (for example, a circle and an **"X"** sign). To create and edit a **point object style**,

    - from the previous window, create a new style, by clicking on the arrow on the right, in the **Point Style** section. 

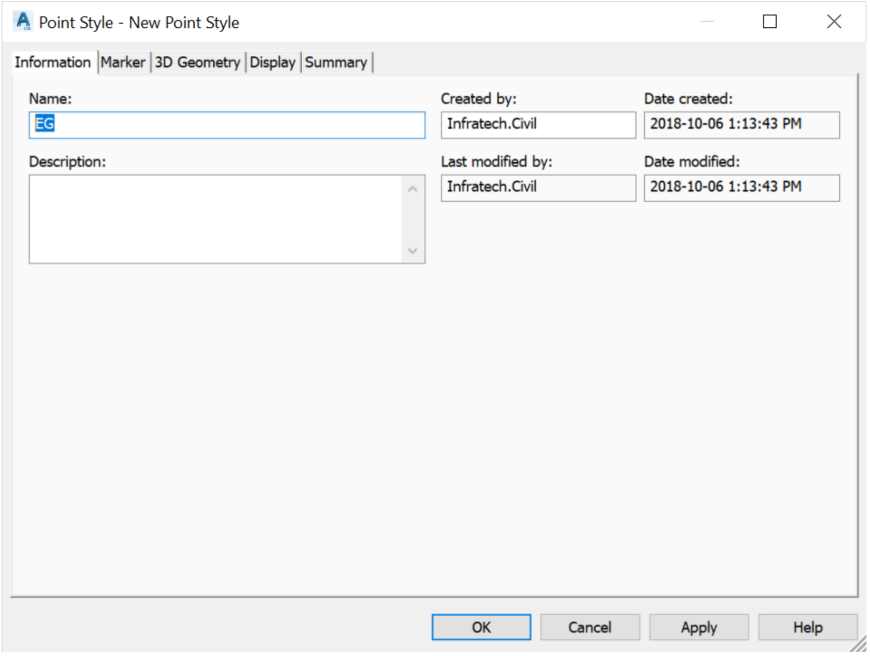
    - next, click on **Create New**



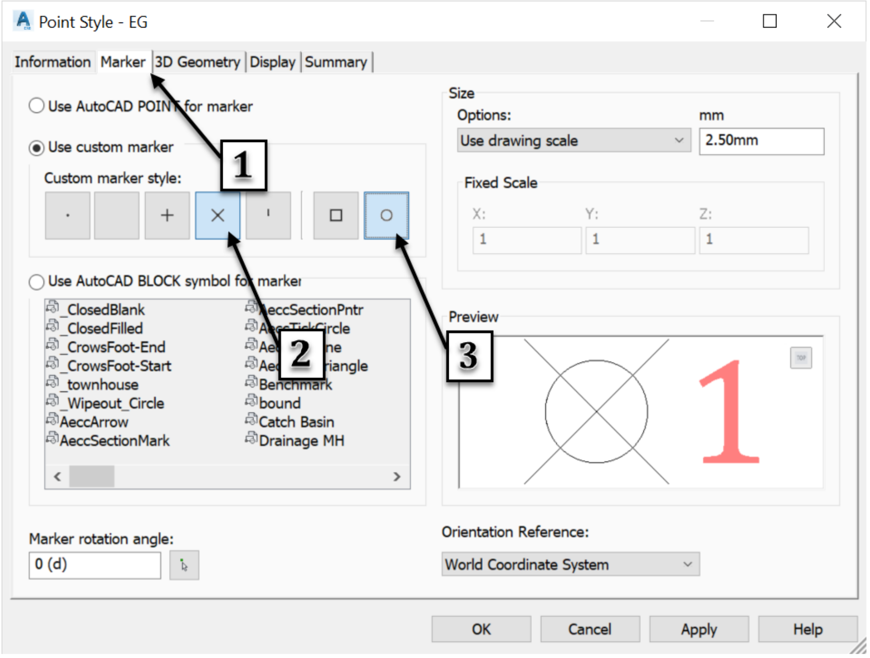

NOTES

NOTES

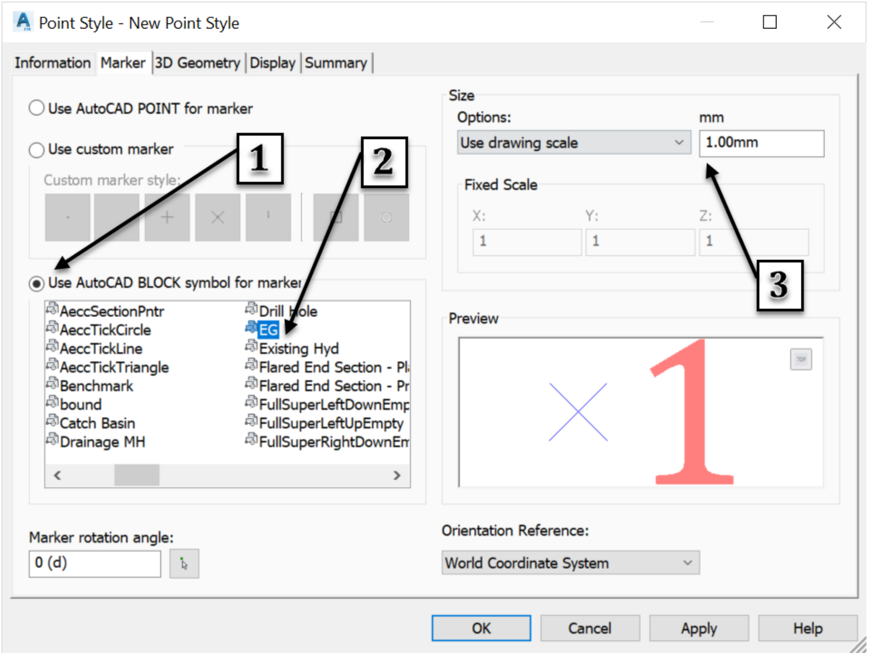
- then, on the **Information** tab, in the **Name** textbox, enter the name of the point style, for example, **EG**, for Existing Ground.




- Now switch to the **Marker** tab. For the point style, you can either choose an **AutoCAD point** for the marker, a custom **marker**, or a pre-existing AutoCAD block to represent the existing ground.



- The option of using pre-existing AutoCAD blocks is particularly useful in a situation where we want to choose an existing block from an external AutoCAD file. Case in point, when styles have been created by a client or a permitting agency like a department of transportation or municipality. In that case, we simply pick a style from the list of symbols.




- After choosing a style, you can go to the display tab and change the colours, line types, scales, and many other options. For now, click on **OK** to close the point styles editor. We will see later how to apply the styles.

NOTES

NOTES

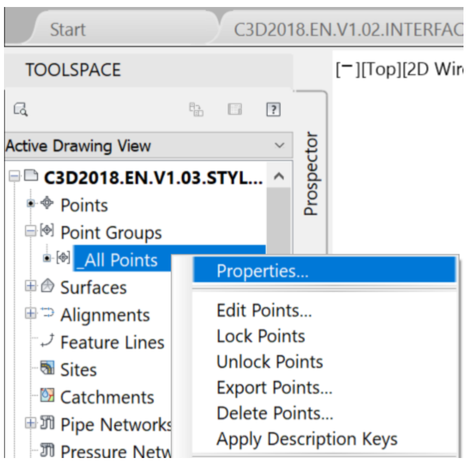
### 3.4.2 Creating Label Styles

Label styles are annotations that allow us to display an object's information. For example, for a survey point, one may be interested in displaying the coordinates, elevation, description, and much more.

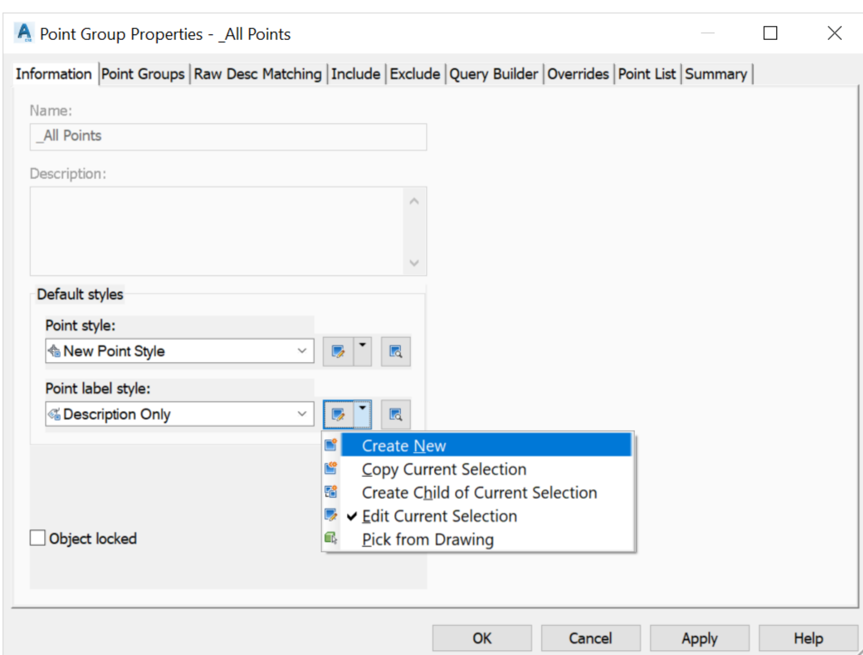
To practice this, let's create a style that will display the elevation, description, and coordinates of a point.

To create a point label style, do the following:

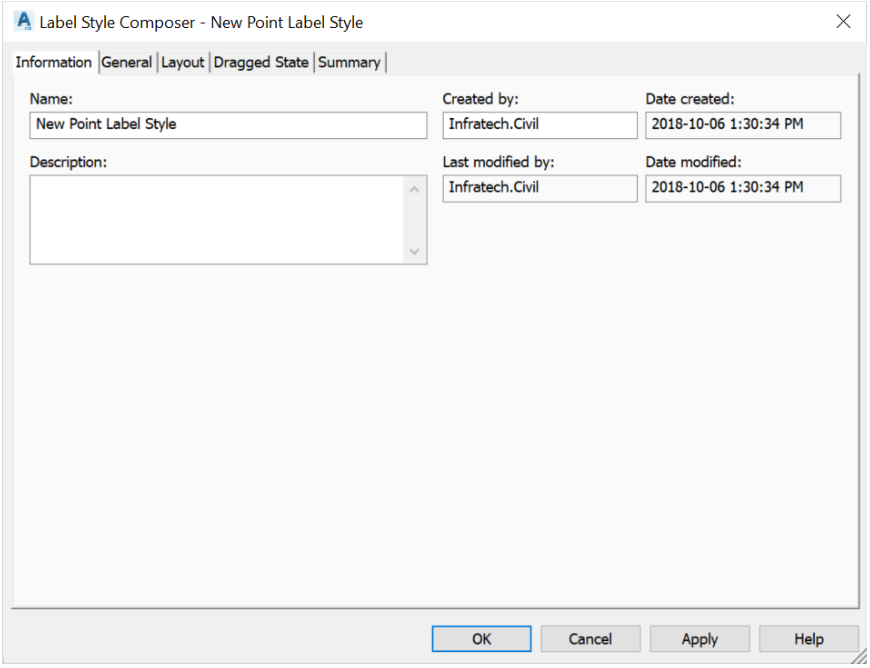
1. Open the **Properties** window of the point group to which you want to apply a point style.




2. In the window, this time in the **Point Label Style** section, create a label style that will display **elevation, description, and coordinates**.



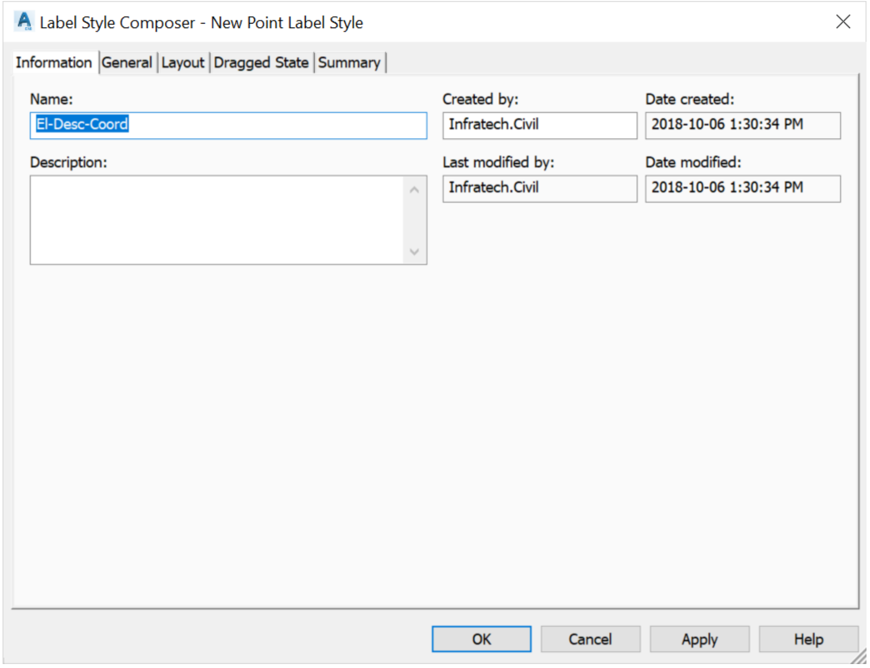
3. The **Label Style Composer** window will appear with the following tabs:
   i. The **Information** tab, which controls general information such as style name, creation date, and authors.
   ii. The **General** tab, for some basic label information.
   iii. The **Layout** tab, which defines the content of the label's text.
   iv. The **Dragged State** tab, which defines the style of the label when dragged from its original location.
   v. The **Summary** tab, which gives a general overview of the different parameters of the Style.



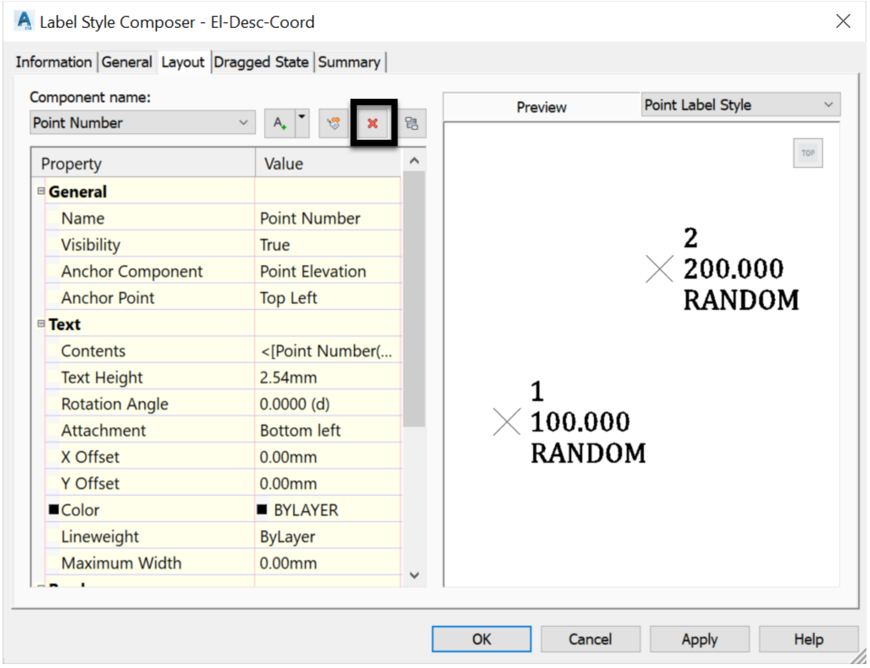

NOTES

NOTES

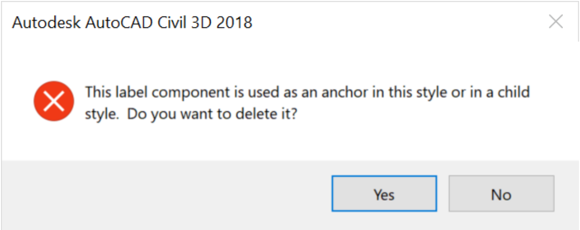
2. On the **Information** tab, name the new style. In general, try to give the style a name that clearly indicates its purpose. For example, we are going to call it **EL-DESC-COORD** to indicate that this label style will show the point's elevation, description, and coordinates.



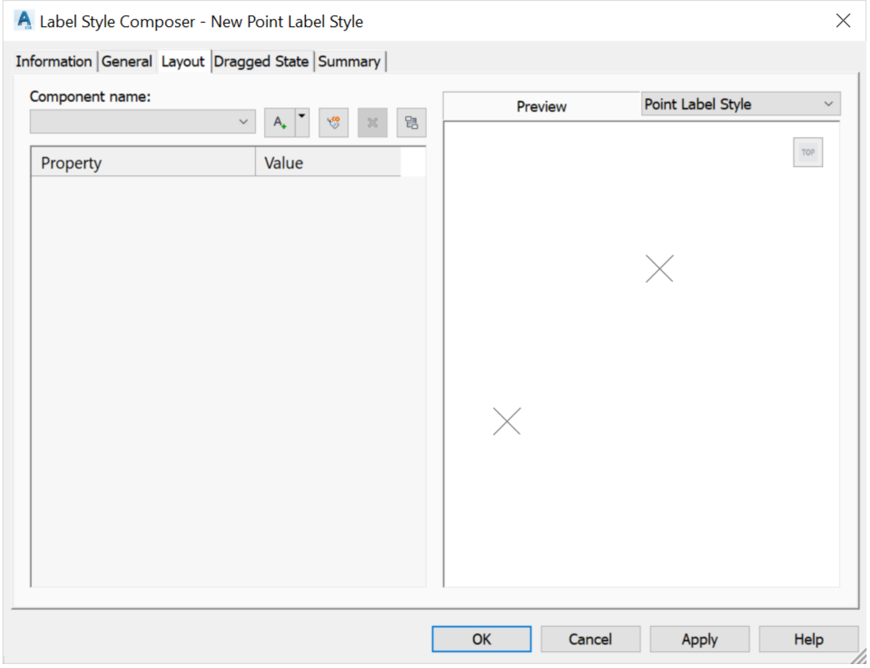

3. On the **Layout** tab, delete the existing data (point number, altitude, and description). We will learn how to recreate them.



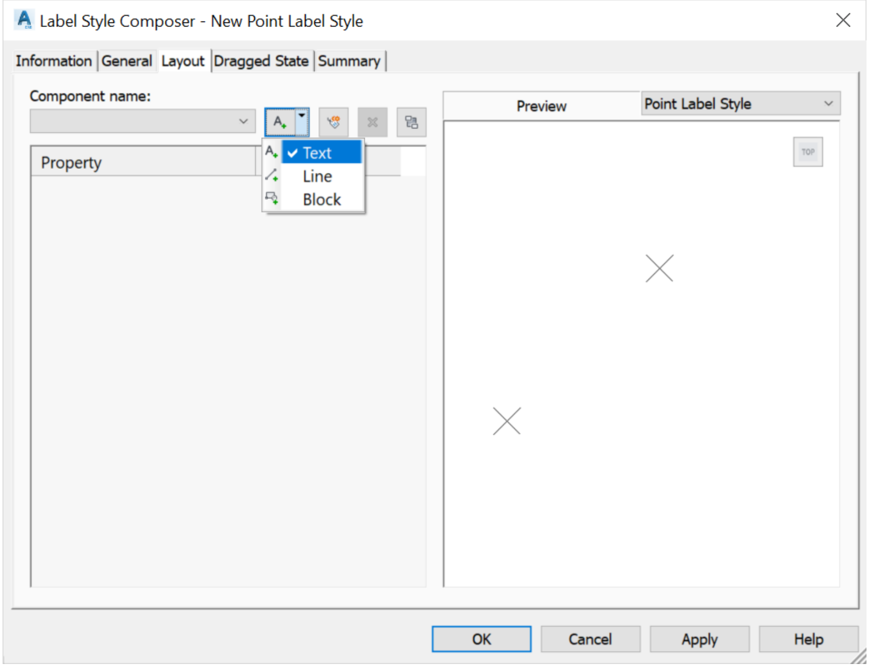
4. If a label component you are trying to delete is used by another label, you will receive a warning. For now, ignore potential warnings and, click **Yes** to delete the label components.




5. Finally, we must have an empty **Layout** window, with no any data:



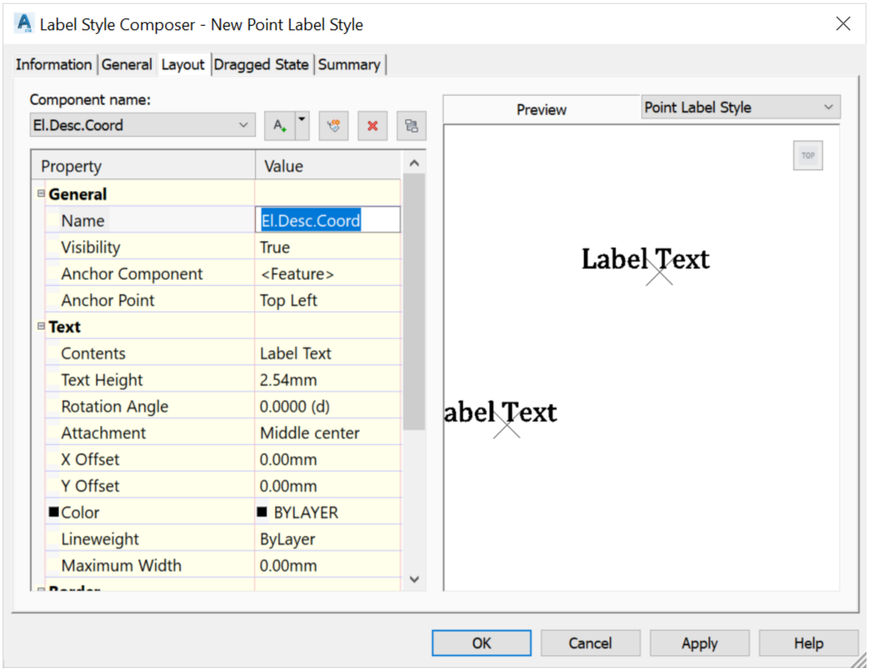

6. Next, we will recreate a style from scratch. To do so, select a text component.



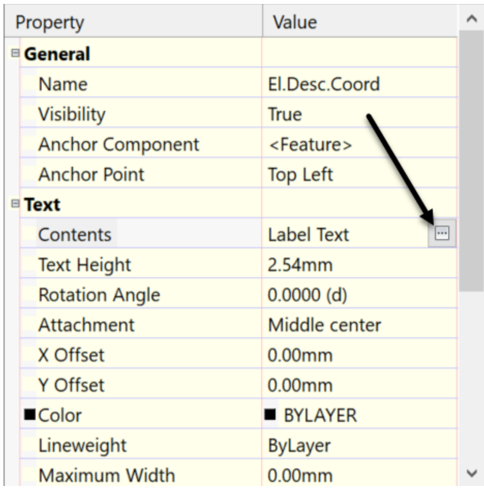

NOTES

NOTES

7. Fill in the necessary data such as the name of the new style, text sizes, colours, and so forth.

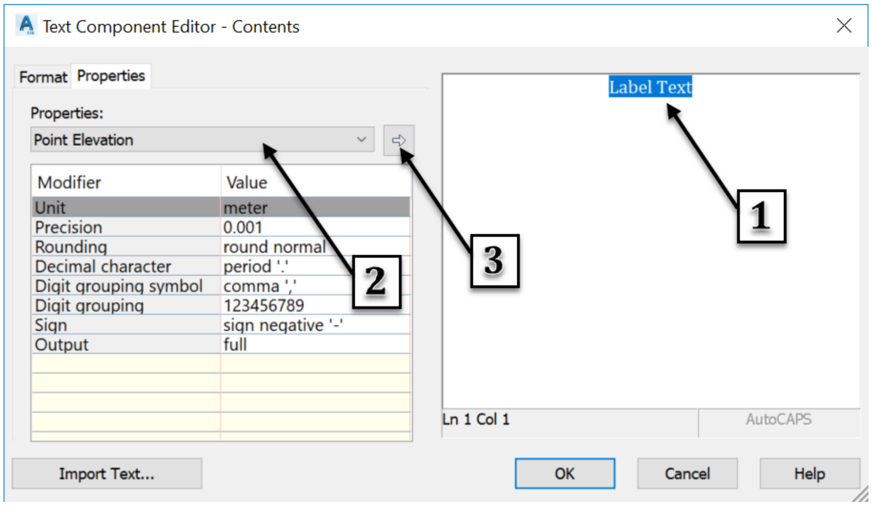



8. To determine the content of the label, meaning the information that the label will convey to the readers, click on the icon with three small dots on the **contents** line.

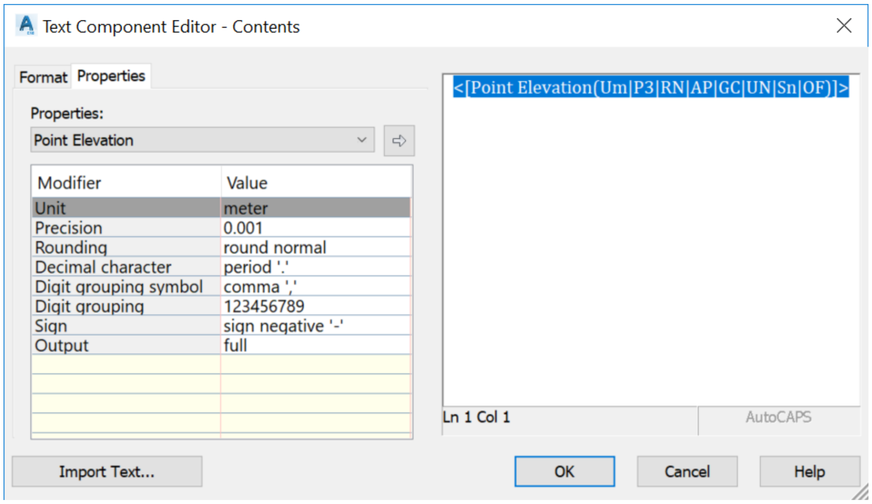


NOTES

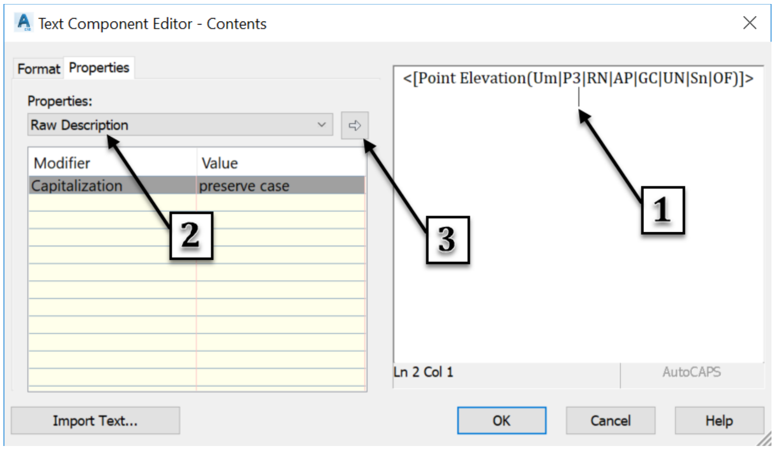
9. Next, we will add the following parameters: Elevation, Description, and Coordinates. Before we define our label, we need to delete the existing text, or select it and simultaneously click on the horizontal arrow to replace it with the parameter in the **Properties** drop-down box.

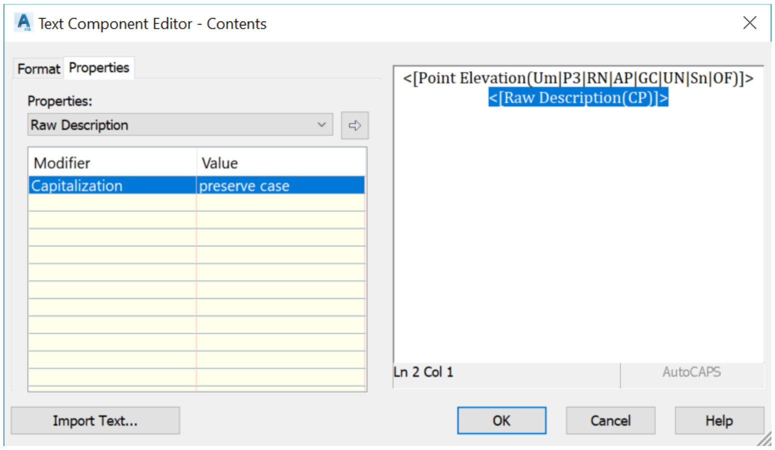



10. We have now defined the elevation component of the label.

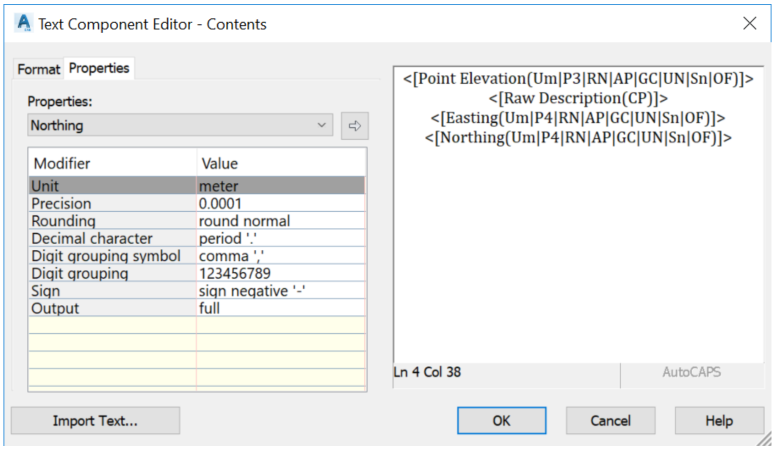


11. Next, let's add the **Point Description** field of the style definition. Go to the next line after the **Point Elevation** component, then select **Raw Description** in the properties drop-down box and click the arrow to the right to add the description component of the label.



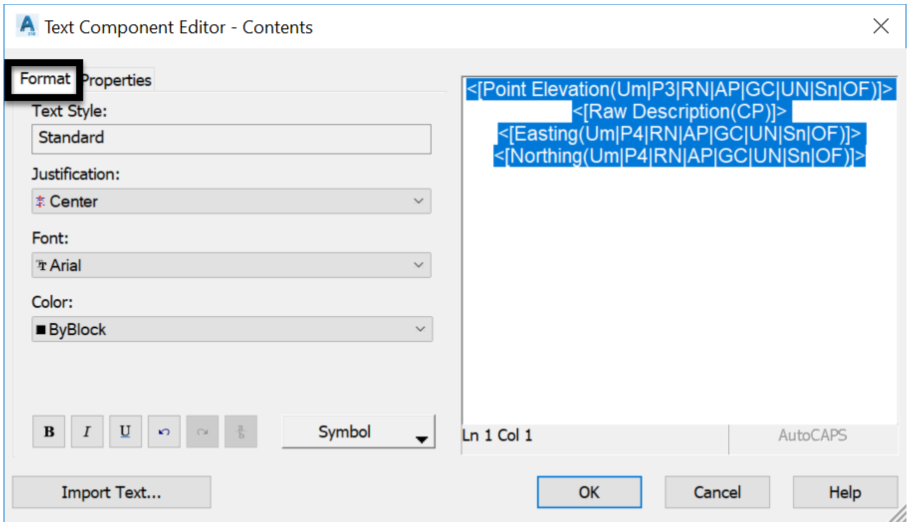




12. The last component to add is the **coordinate** information. Follow the steps we previously used for the **elevation** and **description** to make that happen. First, select **Easting** in the drop-down **Properties** box and click on the arrow to the right. Then, select **Northing** and click the arrow to the right to add the two coordinate elements to the definition of the label.



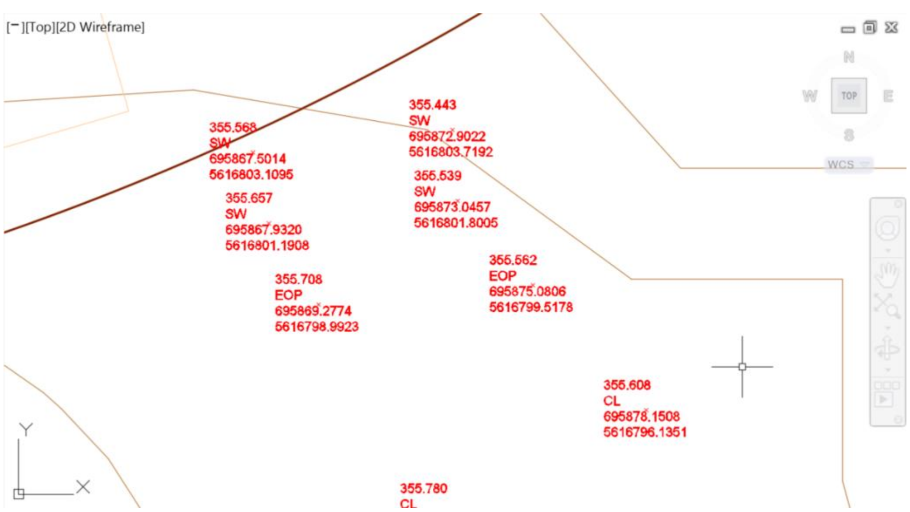

NOTES

13. the **Format** tab. On here, we can change the text style, justification, font, colours, and so forth.




14. Now, click **OK** three times to close all the windows we had to open to get here.

15. Finally, all points in the drawing should appear in accordance with our elevation, description, and coordinates label style.






NOTES

NOTES

## 3.5 Description Keys

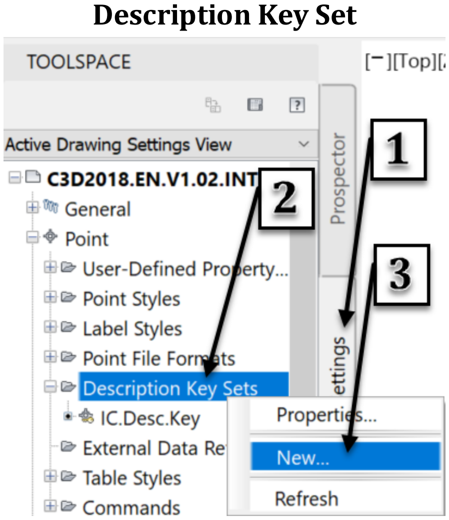
One of the most common land surveying post-processing operations is to assign symbols to survey points to give them a distinctive and self-explanatory appearance. For example, we would like to show specific symbols for trees, fire hydrants, manholes, and the like. Traditionally, this is done in AutoCAD by using a library of blocks. Now, this process can be automated in Civil 3D by using **description keys**. Description keys can be applied during or after the import process. They just need to be created and defined before the points. If not, an update must be made to the point group to apply the **description keys**.

Let's see how to apply styles and labels to survey points, using a **Description Key Set**.

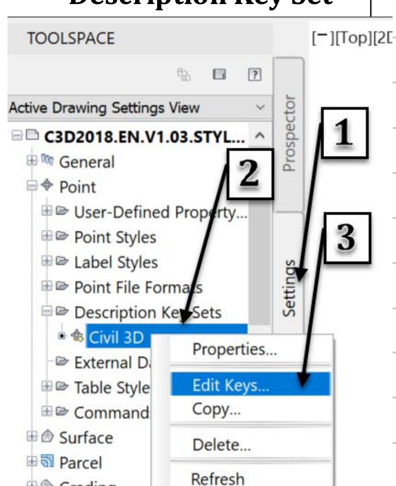
1. First, we need to define the **Description Key Set**. To do so, you can edit the existing description key set **(IC.Desc.Key)**. You can also start with the out-of-the box **Civil 3D** key set, if working from the default template file.

2. Another option is to simply create a new set by right-clicking on **Description Key Sets** on the **Settings** tab, in the **Toolspace**. You can choose either option for this exercise. But, just to be consistent with the steps in this training, let's pick the option to edit the existing **Description Key Set**.

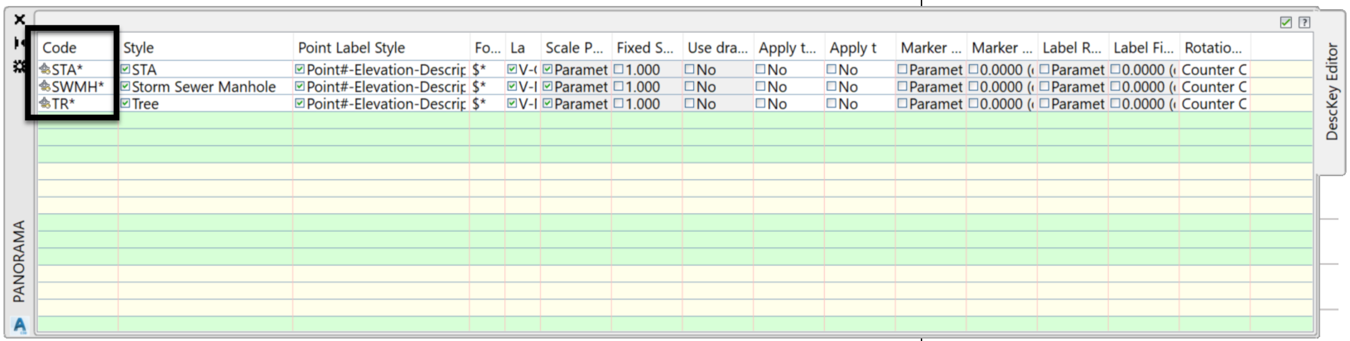
**To Create a New Description Key Set**



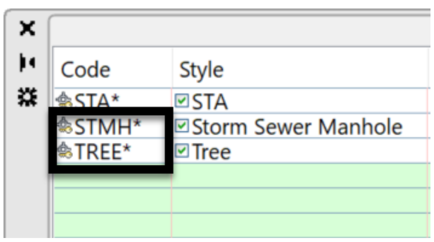

**To Edit an existing Description Key Set**



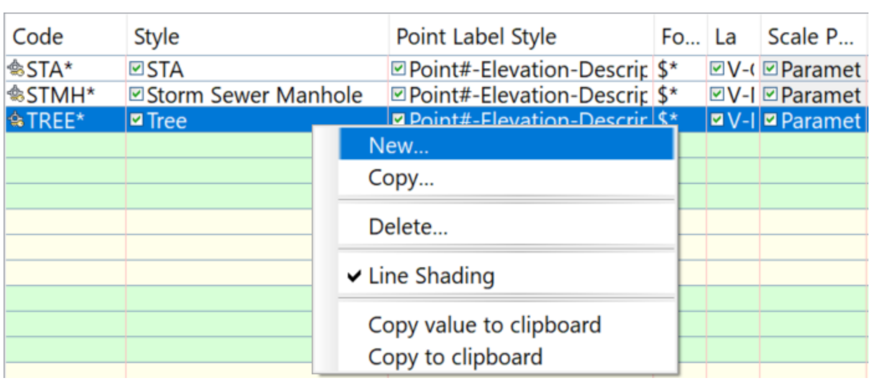
3. In the **Panorama Window**, that we will encounter in Civil 3D whenever we need to edit an object in a tabular view, we have a few pre-defined codes (**STA*** for Station, **SWMH*** for Stormwater Manholes, **TR*** for Trees).




4. Now, we need to modify a couple of the pre-defined codes to match our requirements. In our survey file, Storm Manholes are described with the code **STMH** instead of the code **SWMH** used in the description key set. Also, our surveyor used the code **TR** for the trees and not **TREE**, as described in the key set. So, we need to make a change to the code, to match the description used in our survey file. Click on the two codes to rename them to **STMH** and **TR** respectively.

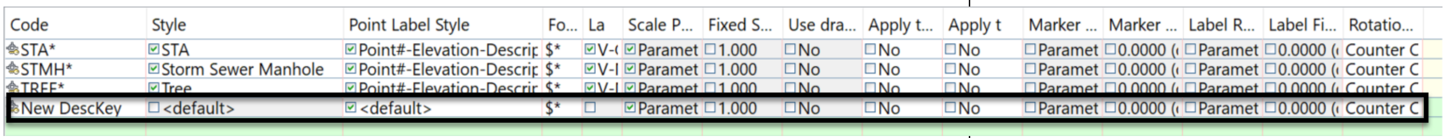



5. Next, we want to create a code for the following items: Sanitary Sewer Manhole (**SSMH**), Fire Hydrants (**FH**), Water Valves (**WV**), Existing Ground (**EG**), and Light Poles (**LP**). Let's start with the **SSMH**. To create a description key, in the editor vista (**Panorama window**), right-click on any line and click **New.**

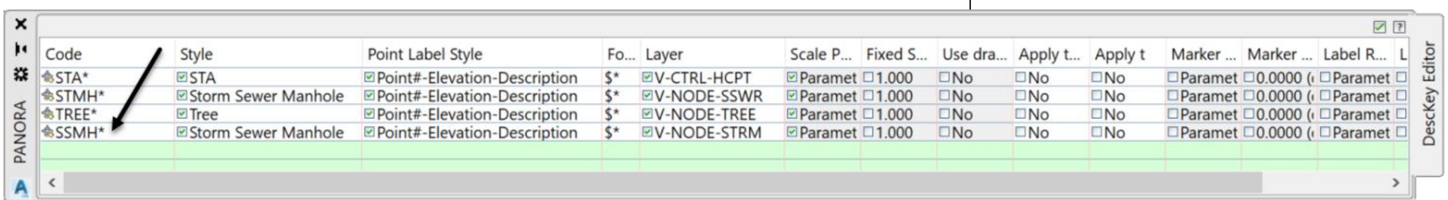



NOTES

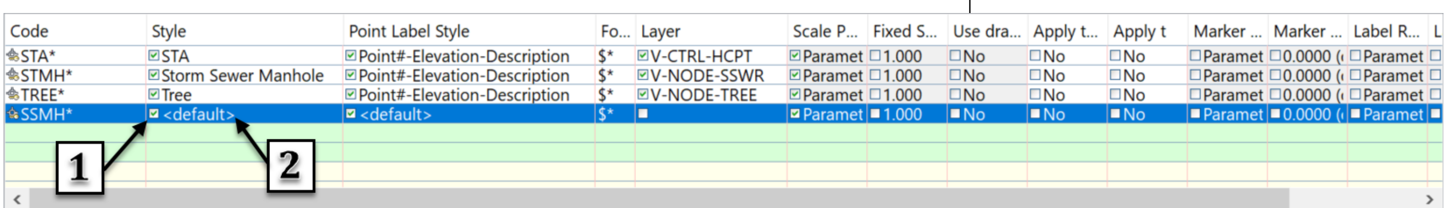
6. A line is added to the **Description Key Set**.

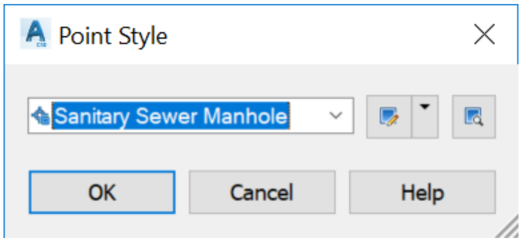



7. Now, edit the new **Description Key** to match the codes in our current and future survey files: rename the new Key to **SSMH***. We should mention that contrary to point groups, and other definitions in Civil 3D, all entries in the **Code** column of the **Description Key Editor** are case sensitive. Therefore, we should be mindful of that.




8. Next, apply the style for **SSMH** to determine the appearance of the points. For all **SSMH** points, select the **Sanitary Sewer Manhole** style.



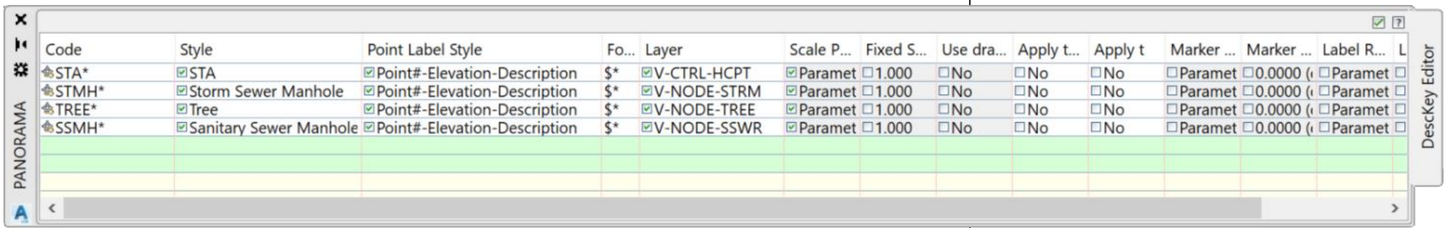




9. Repeat the previous two steps to change the **Points Label Style** to **Point Number-Elevation-Description**. Then, assign the layer to **V-NODE-SSWR**.

NOTES

NOTES

10. The new **Description Key** windows should be all set now and ready to be utilized.



| Code | Style | Point Label Style | Fo... | Layer | Scale P... | Fixed S... | Use dra... | Apply t... | Apply t | Marker ... | Marker ... | Label R... | L |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| STA* | STA | Point#-Elevation-Description | $* | V-CTRL-HCPT | Paramet | 1.000 | No | No | No | Paramet | 0.0000 ( | Paramet | |
| STMH* | Storm Sewer Manhole | Point#-Elevation-Description | $* | V-NODE-STRM | Paramet | 1.000 | No | No | No | Paramet | 0.0000 ( | Paramet | |
| TREE* | Tree | Point#-Elevation-Description | $* | V-NODE-TREE | Paramet | 1.000 | No | No | No | Paramet | 0.0000 ( | Paramet | |
| SSMH* | Sanitary Sewer Manhole | Point#-Elevation-Description | $* | V-NODE-SSWR | Paramet | 1.000 | No | No | No | Paramet | 0.0000 ( | Paramet | |



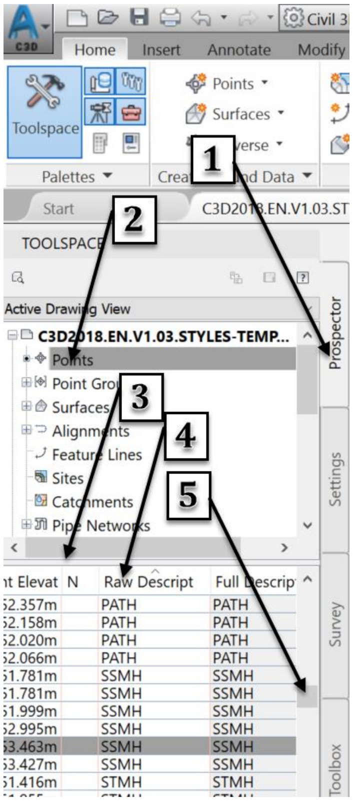
11. However, before going further, we should make a couple of observations:
    i. The asterisk is a wild-card character that allows us to apply a catch-all definition to a code, followed by any other character. For example, **STMH*** will include STMH1, STMH-2, STMH-Ex, and so on. We must also note that, when a Point or Label style is not applied from the **Description Key Set**, the **Point Group** properties will take precedence and control the style and label style.
    ii. Finally, the $* sign in the format column specifies that a point's **raw description** will be used as **full description** in the point label. If we want a different value for raw and full description, this is the place to specify that.

12. Now we know how to create Description **Keys.** We can use these steps to create a full Description **Key Set** for our company. A sample more advanced description key set will be provided for your use, in the **Infratech Template** file included in the resources provided, as part of this training.

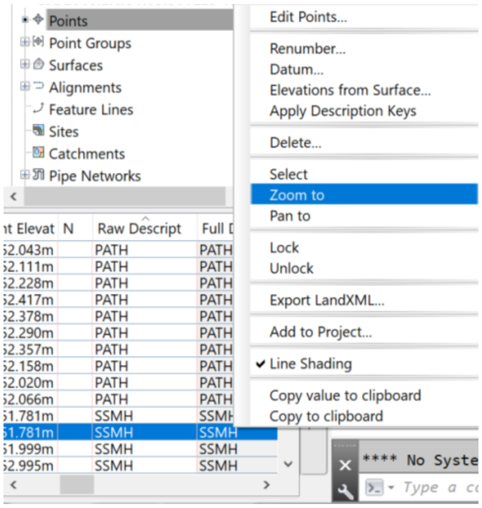
13. We can now close the **Panorama** windows by clicking on the green check mark at the top right to close the **Description Keys Editor**.

14. Let's check what happens to the points for which a description key has been created.

    - Activate the prospector tab in the Toolspace.
    - Click on **Points**,
    - Re-arrange the **Panorama** (table at the bottom) to display the **Raw Description** column,
    - Sort the points by **Raw Description**,
    - Scroll down or up to display the point description **SSMH**.

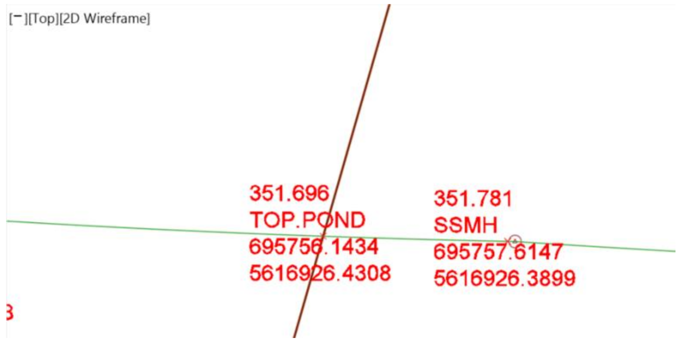



- In the table, select one of the points with **SSMH** as raw description, right-click and **Zoom to**.



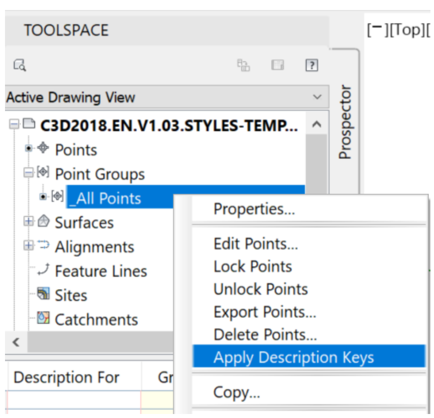

- The point is displayed in the center of the screen. We need to zoom closer.

NOTES

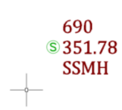


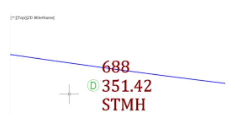

- We can see that the point is shown just like any of the other points. The reason for this is that we have created the **Description Keys** after the points have already been created or imported. Therefore, we need to apply the **Description Key** set to points that have already been created, before the **key set.**

- To apply the **keyset,** go to the point group, in this case, the **_all points**, select it, right-click and click **Apply Description Keys.**




- Now, notice how the Points style and Label style have been changed to the specifications in the KeySet table?



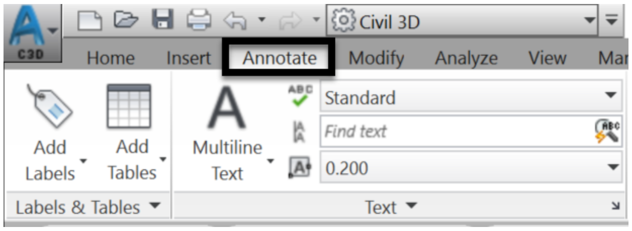




NOTES

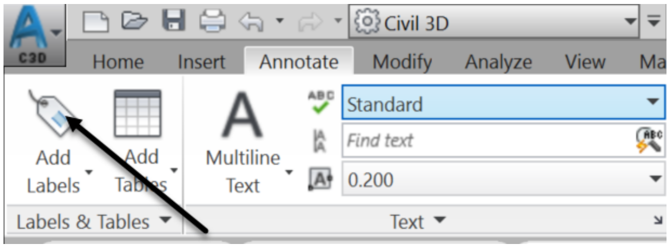
## 3.6 Annotation Dialog Box

Civil 3D has a global dialog box that simplifies annotating objects. This dialog box is a unique window that can be used to annotate most common objects. Options available in the dialog box vary, depending on the selected object. For example, to add annotations to multiple lines segments, you need to:

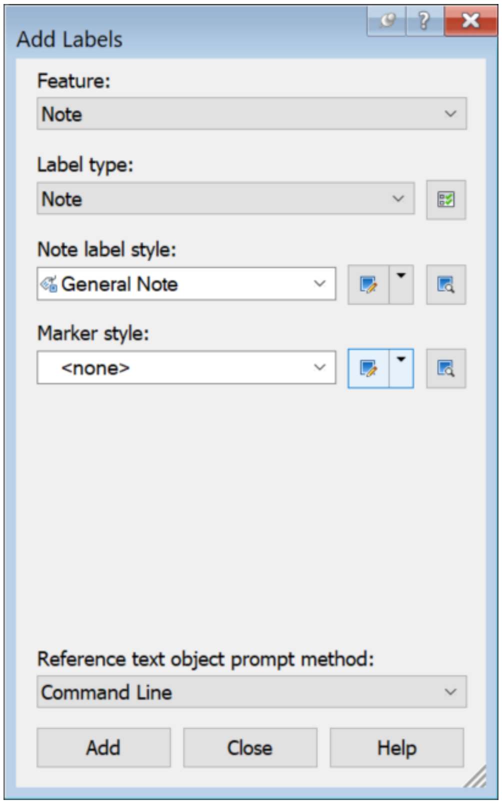
1. Click on the **Annotation** tab.




2. Then Click on the upper part of the icon (not on the **Add Labels** text or arrow at the bottom).



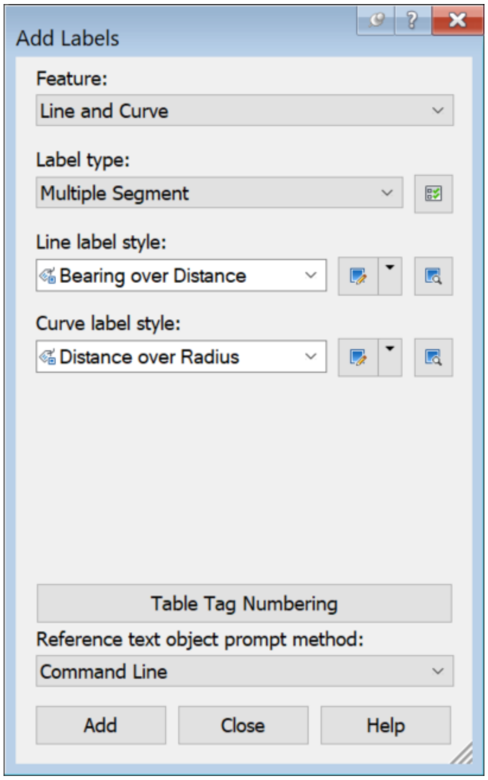

3. In the new window, add labels for most types of Civil 3D objects (lines, surfaces, parcels, pipes, and so forth)




NOTES

Let's try it, by adding label styles to parcel segments (the blue line representing the boundary of the current project).

4. In the **Add Labels** window, choose the different label styles to apply. In this case, multiple segments.



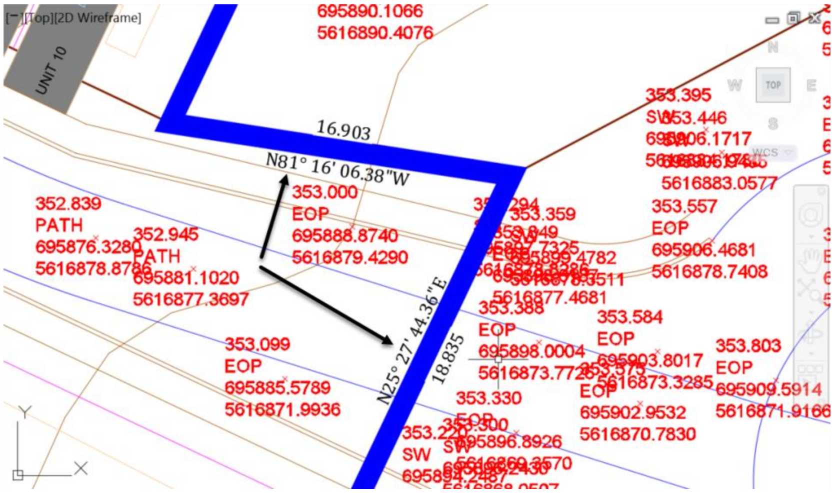

5. Click on **Add.**

6. Then, click anywhere on the blue boundary polyline.


NOTES

NOTES

7. We can now see the bearing, distance and angle labels of the site boundary polyline.



# 4 SURVEYING

## 4.1 Introduction

In general, most civil engineering jobs involve a phase of field data collection and a stakeout of design data for construction. These two operations are usually carried out by a licensed surveyor in the project's jurisdiction. In the first scenario, the design team would like to know the lay of the land before proposing any modification.

Depending on the project's setup, the survey team may be responsible for collecting the data, make necessary adjustments and create a usable digital terrain model. In some cases, the surveyor is solely responsible for collecting the data while the design team will create the model. Whatever the case may be, Civil 3D offers the tools for processing survey data to create a terrain model or surface.

Survey data is usually imported into Civil 3D in either of two ways: importing raw survey files (**JOB**, **FBK**, **TDS**, and others.) or creating custom file formats such as **TXT** or **CSV**.

NOTES

## 4.2 Importing raw files

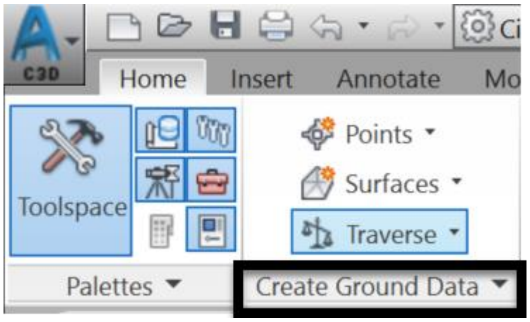
To import a raw file from different survey equipment, we have two main options.

### 4.2.1 Survey Data Collect Link tool

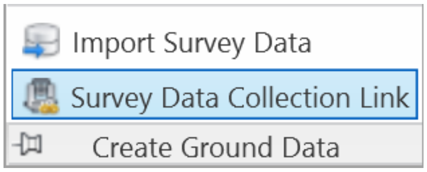
The first option is to convert the raw file into a Civil 3D acceptable format (**CSV**, **TXT** or **FBK)** using the **Survey Data Collection Link**. To access it:

1. Activate the **Home** tab.

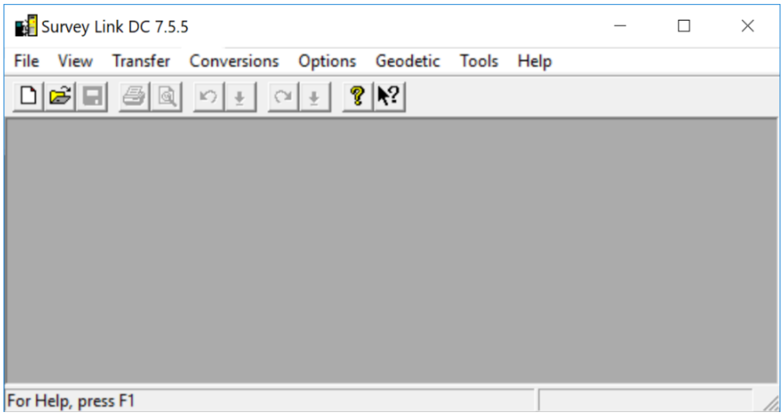
2. Expand **Create Ground data** section.




3. Launch the module **Survey Data Collection Link.**

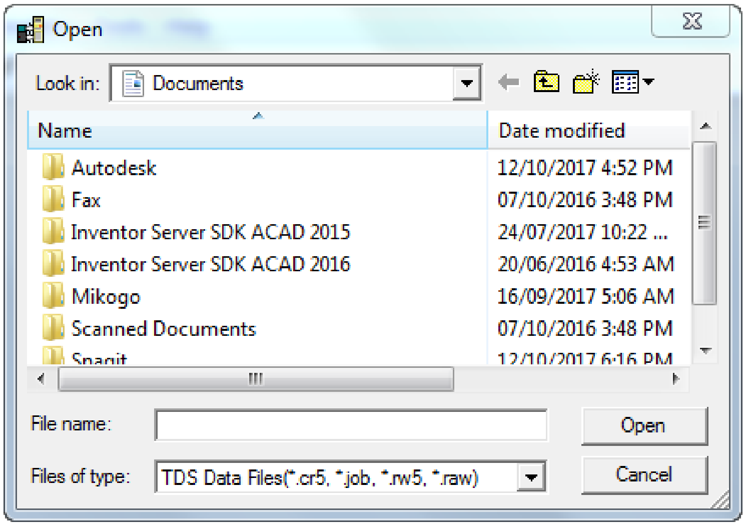



4. The **Survey Link DC 7.7.5** module opens:




NOTES

5. You can then search for the file exported from your survey equipment and convert it to a Civil 3D compatible format (**TXT**, **CSV**, Land XML, and the like.).




### 4.2.2 Third-party software

A second option is to use a software from third-party developers (Carlson Connect, Stringer Connect, and others) or the equipment manufacturers themselves. They provide tools for converting raw files from their instruments to Civil 3D compatible format. As an example of conversion software, we have **TrimbleLink** for Trimble Equipment, **Leica Exchange** for Leica equipment or **Survey Office** for Spectra Precision, and many others. More information can be obtained by contacting the manufacturers or resellers of the equipment.

## 4.3 Importing custom file formats

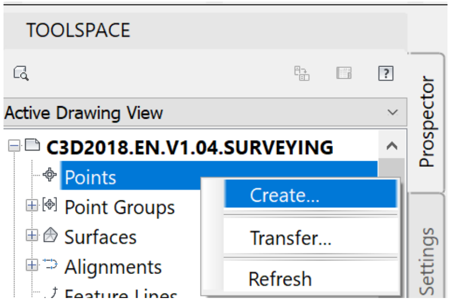
Generally, we can export a survey fieldbook from the field equipment to custom format files like an Excel spreadsheet (**CSV**) or text file (**TXT**). The extracted files are most of the time in the form of columns **P** (Point Number) **N** (Northing coordinates), **E** (Easting coordinates), **Z** (Elevation) and **D** (Description). There are some variations of formats (mostly a combination of these five fields) like **PENZD**, **PENZ, ENZ**, and so forth).

Let's see how to use these types of files in Civil 3D.

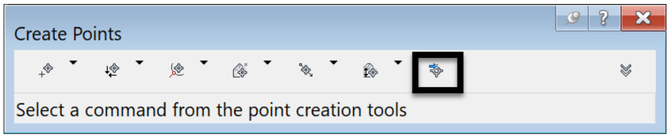
1. Open the **04.01-Survey dwg** file in **Lesson 04** practice folder.

2. Launch the **point creation Toolbar** by right-clicking on **Points** in the **Prospector** tab of the Toolspace.

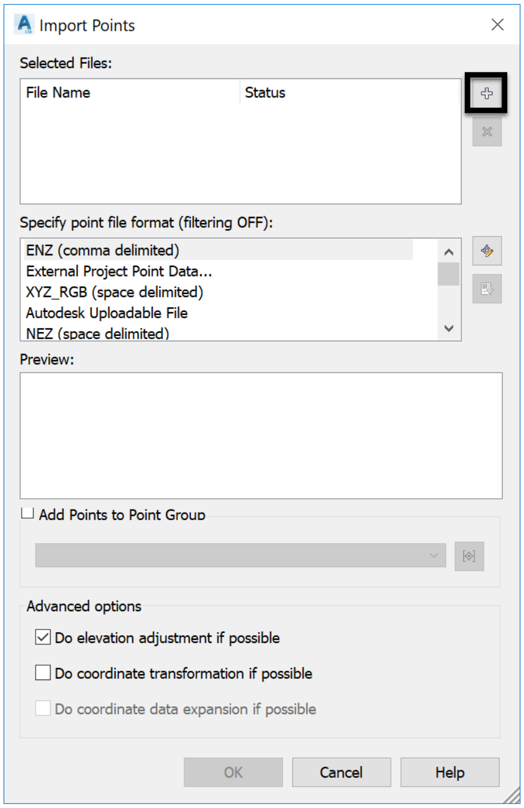
NOTES




3. Run the **Import Points from file** command.

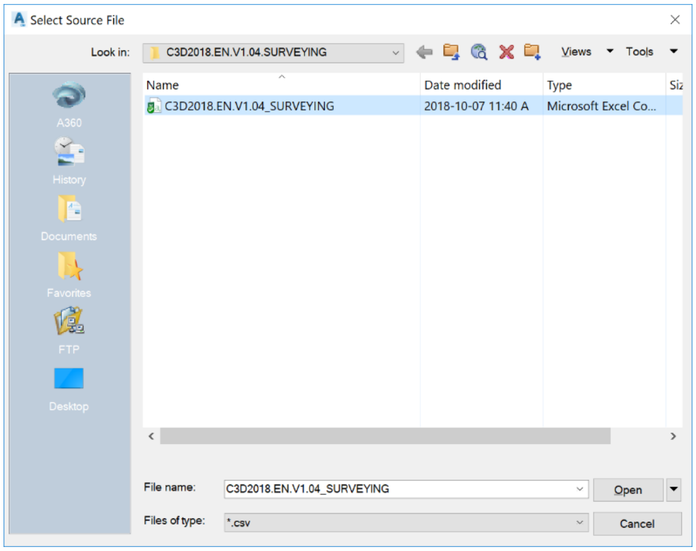



4. Click on **Add Files** to choose the survey file to import.




NOTES

5. Browse to the survey folder to select the **CSV** file. Make sure the right type of file is selected in the **Files of type** drop-down box. In this case, we need to choose a **CSV** file format.




6. Click on Open.

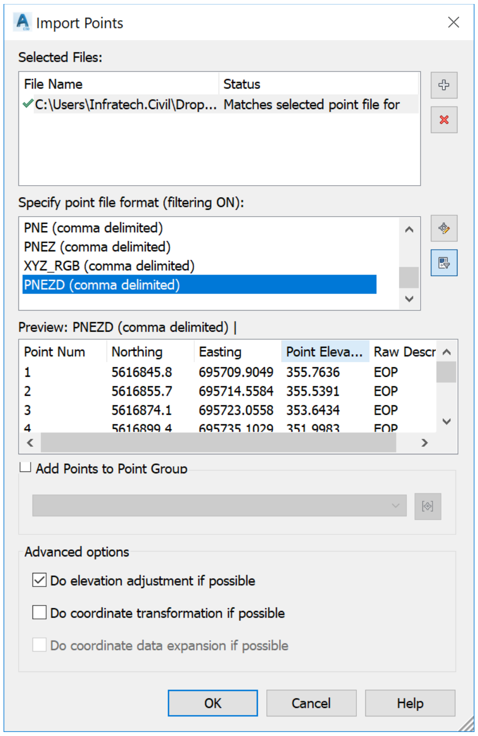
7. Choose the appropriate point file format. To determine the right format, we have a few options:

   I. Ask the person who created the file.

   II. Open the file in a text editor and identify the different columns that compose it.

   III. Proceed by trial and error by choosing different formats until we get the one that fits the base file.

The first option is obviously the recommended one.

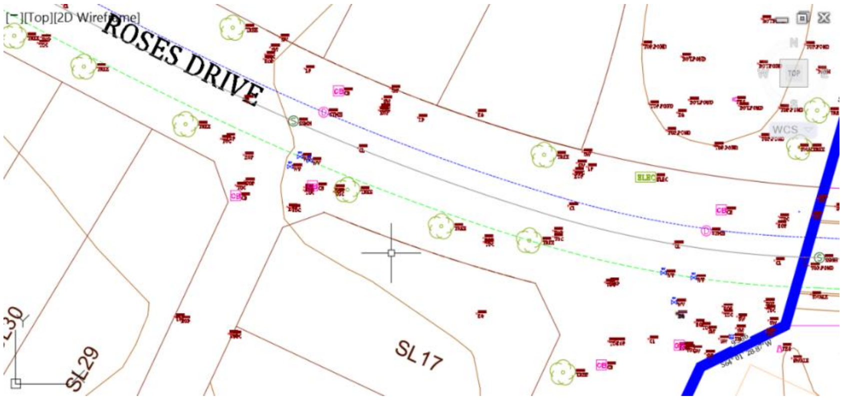



NOTES

NOTES

8. Click OK to import the points.

9. The points now appear in the drawing. Furthermore, the styles and labels defined in the **Description Keys** set are assigned to each point in accordance to their raw description.



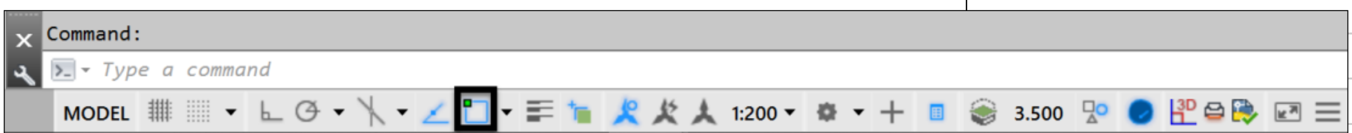
## 4.4 Creating points

We've seen how to import existing points collected from a land survey. However, we often need to create additional points. The reason may be for design purposes, incomplete field survey data that needs to be enhanced in the office, construction stakeout, or other motives.

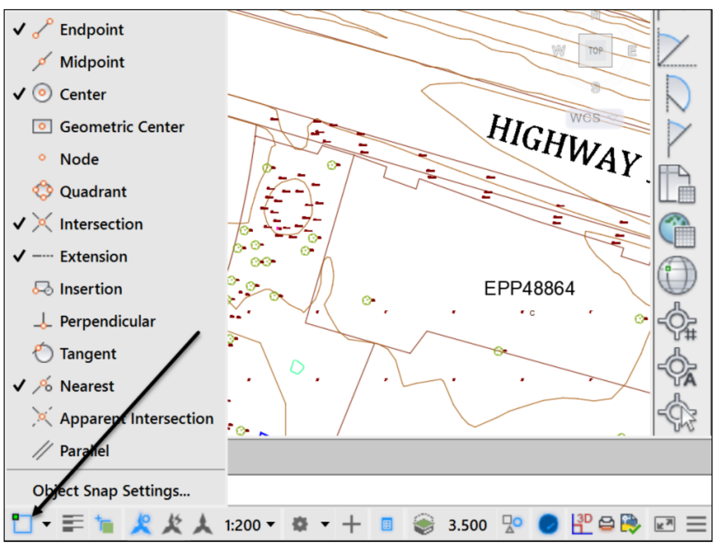
Whatever the case may be, Civil 3D has a tool called the **Point Creation Toolbar** that allows us to create the points needed for different circumstances.

Let's explore a few cases where we need to create points.

1. Continue working with the same **SURVEYING.DWG file.**
2. Make sure **osnap** is activated (It will be blue when it's active and dark when disabled):

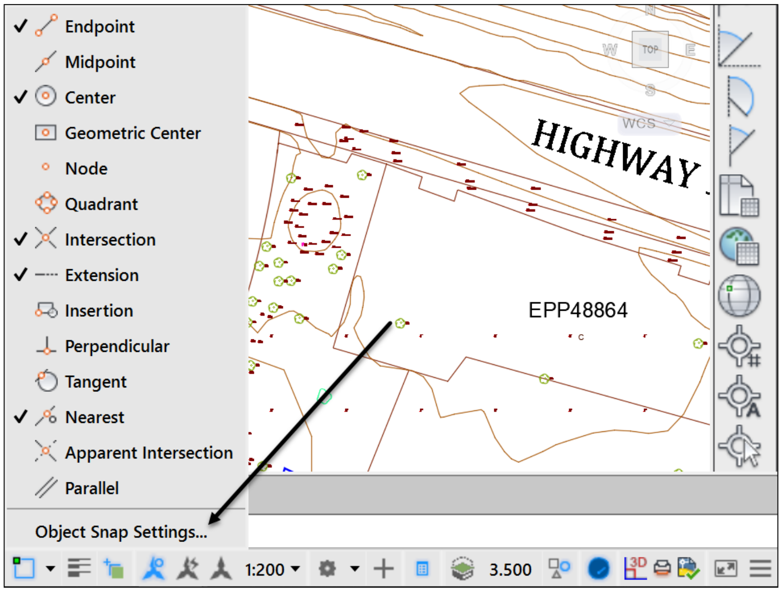



3. Right-click on **Object Snap** to activate the following modes: Endpoint, Center, Intersection.

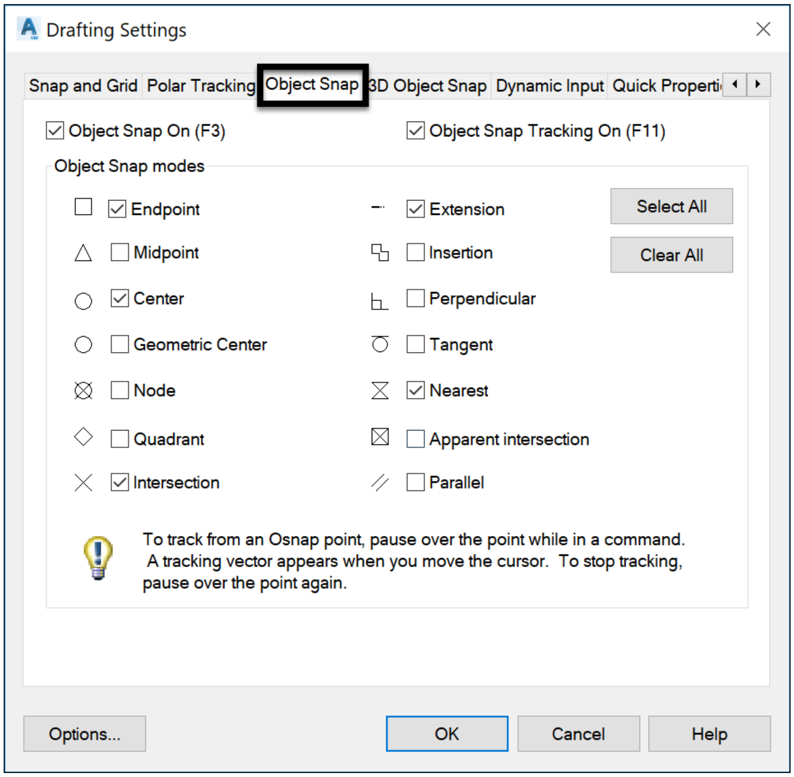



NOTES

4. Alternatively, you can select **Object Snap Settings** and select the Snap modes of your choice.




5. Then, from the **Object Snap** tab, select the modes to activate.




6. Click OK

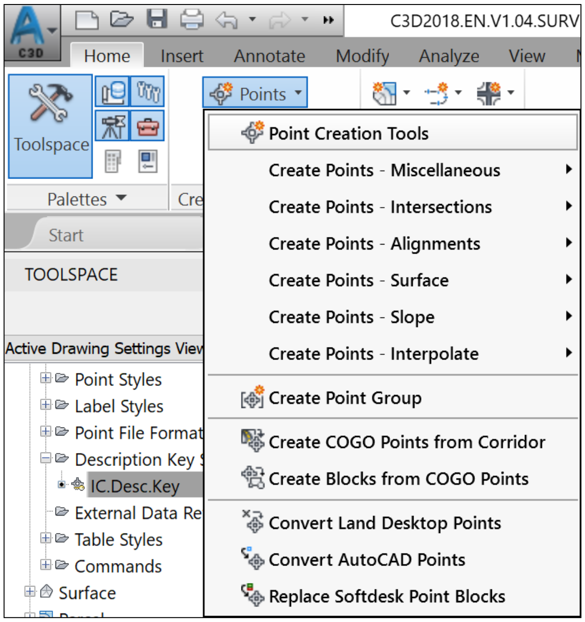


NOTES

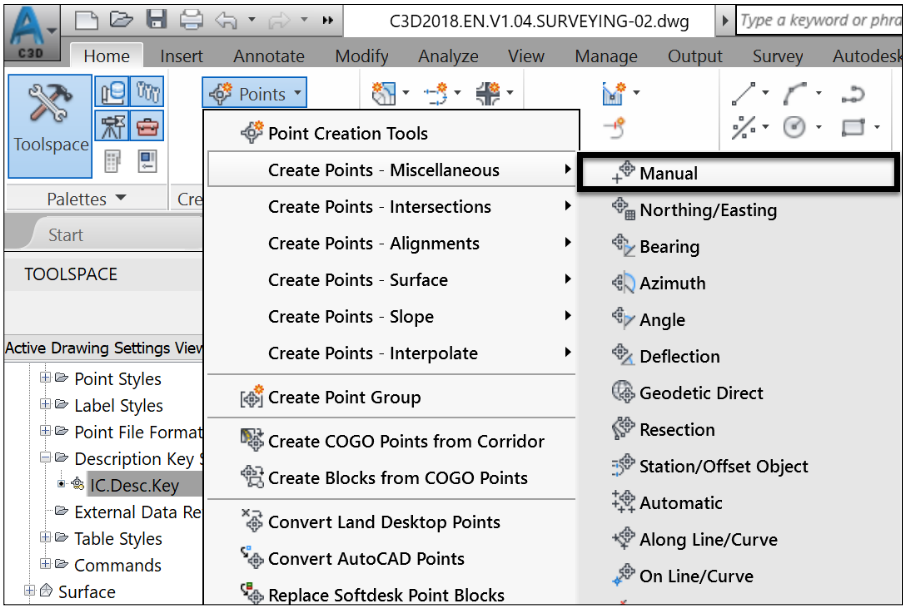
## 4.5 Creating points with data coordinates

Let's assume we have a monument (or survey benchmark) for which we have coordinates available on paper and have just learned about their existence. We can use the paper coordinates to recreate it in the survey file or simply draw it in Civil 3D at the exact location.

1. To create a point manually, launch the **Point Creation Tools.** Let's learn a new way to launch the tools instead of right-clicking from the prospector. From the ribbon, you can launch the **Point Creation Tools**.




2. Or you can directly execute the **Manual** point creation command without loading the toolbar.




NOTES

NOTES

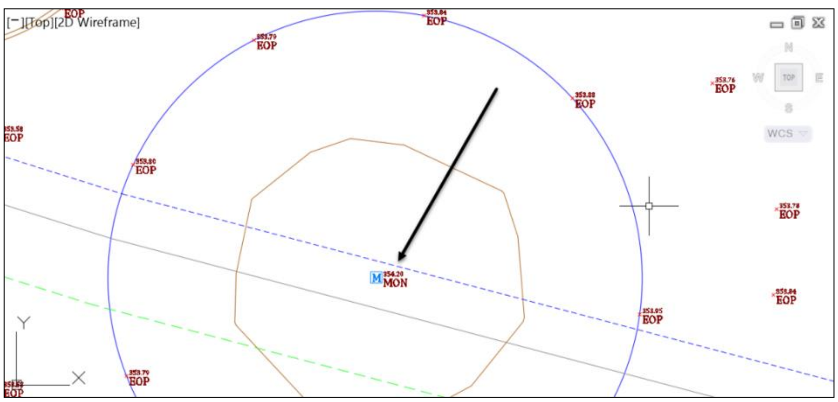
3. Then, specify the location of the point at the command line. Spoiler alert! The monument is going to be in the center of the roundabout. So, you may need to zoom around that area to see it. When asked for the point location, type the following coordinates "**695920.20, 5616867.00**" at the command line, then press **Enter.**

4. Next, enter **MON** (in upper case), for monument, to specify the description.

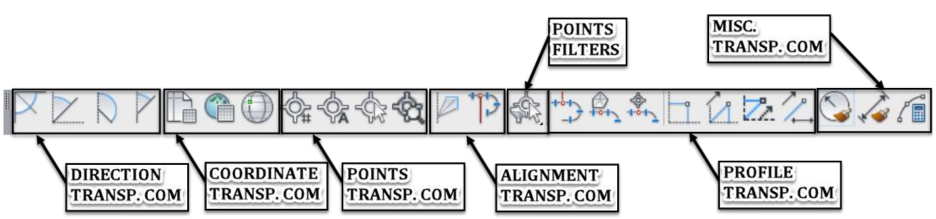
5. Finally, when prompted for elevation, enter **354.20m** or **1,162.10ft**

6. Hit **Escape** to end the command. If we needed to add more points, we would continue and specify a new point. Alternatively, instead of entering the coordinates at the command line, we could simply click to insert to point, had we known the exact location.

7. Zoom in to see the point created.



8. Note that you can also use the transparent commands (the vertical toolbar usually found on the right) to specify the locations of points.



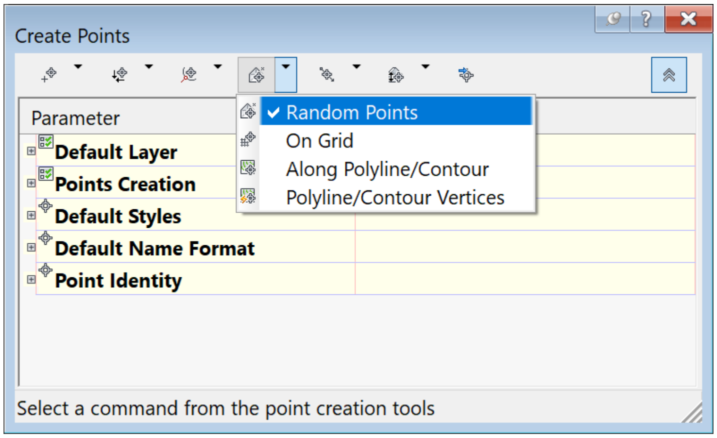

9. Transparent commands are useful, as we can improve precision drafting and design, by entering an exact value at the command line, based on known information, when prompted for a point, a distance, or a radius.

Transparent commands cannot be run on their own. They can only be used within a running command. We will, soon, release a few more lessons including one on using transparent commands. Check our release roadmap for that.

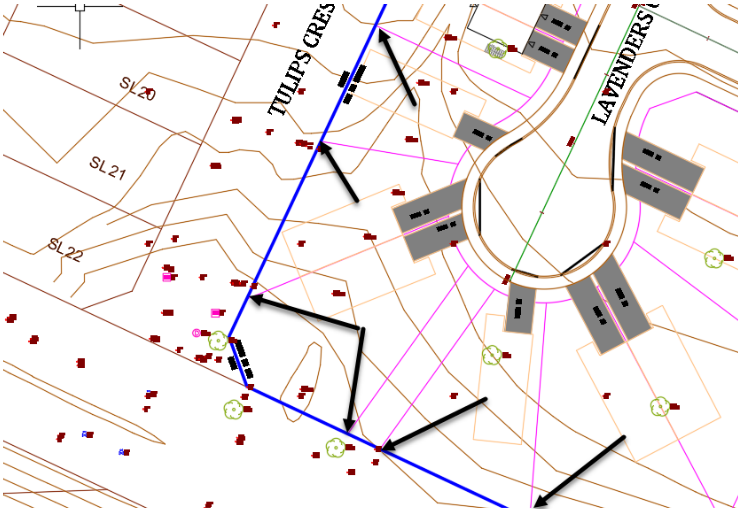
NOTES

## 4.6 Creating points from a surface

1. Each project needs to be integrated into an existing environment. For instance, we need to tie proposed utilities to the existing infrastructure; we need to daylight existing proposed grades to the existing ground on the edges of the site. In this case, there are lots and roads already built in a previous phase. So, we must maintain the elevations along these pre-existing lots and roads. The best way to do it is to create points with elevations matching the existing ground as surveyed. To create points from a surface, use the point created by the surface command:




2. Click on the back of lot corners along **Tulips Crescent** and **Jasmine Blvd**.




3. When prompted for the description of the point, type **DG-EG** at the command line. These are design points with **Existing Ground** (**EG**) elevations.

NOTES

4. Continue to press **Enter** at the command line to accept the description **DG-EG** for the next points. If you make a mistake, you can always return and restart the point creation command.

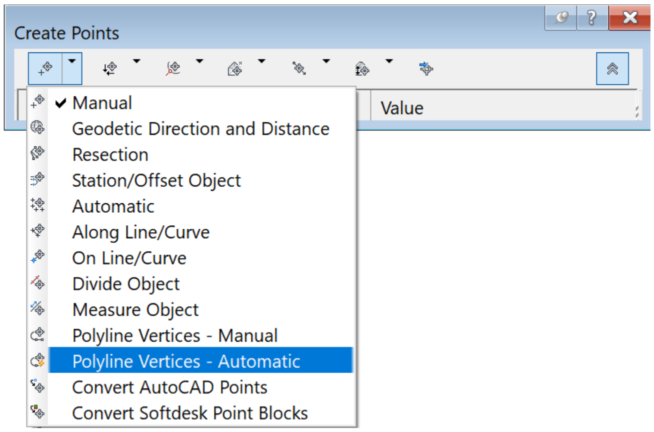
NOTES

NOTES

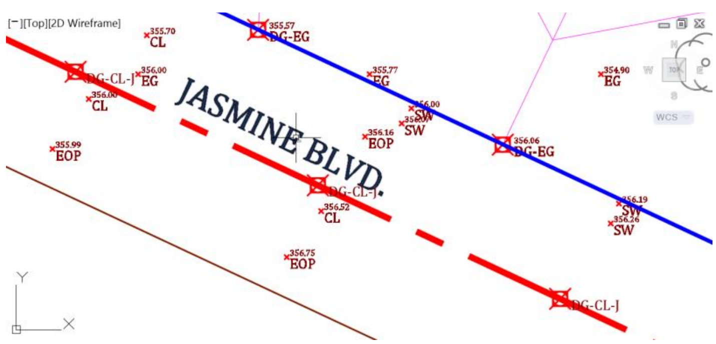
## 4.7 Creating points on a polyline by vertices

You may sometimes need to create points along a polyline, maybe a survey baseline, a road that needs to be staked out, or for other reasons. To do that,

1. Reopen the **Point Creation Toolbar**, if you have closed it. If not, follow along.
2. Run the **Polyline Vertices – Automatic** command on the point creation toolbar.

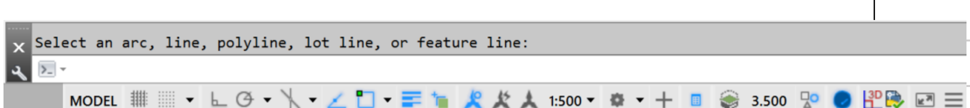



1. Select the Red polyline representing the centerline of **Jasmine Blvd.**




2. For the description Enter **DG-CL-J** (Design centerline for **Jasmine Blvd**).

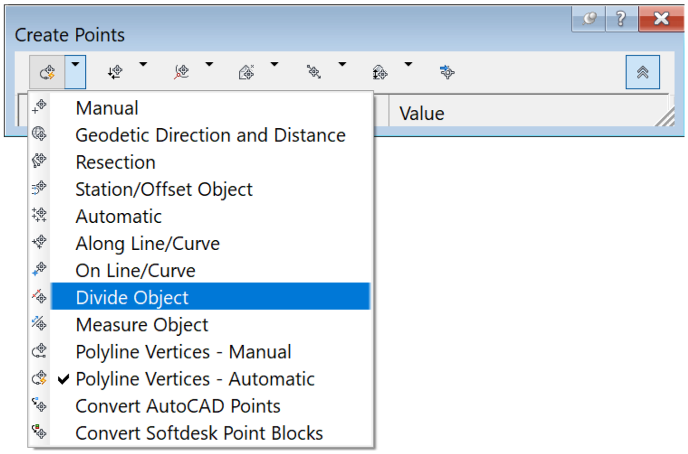
3. Keep entering **Enter** at the command line until it becomes blank:



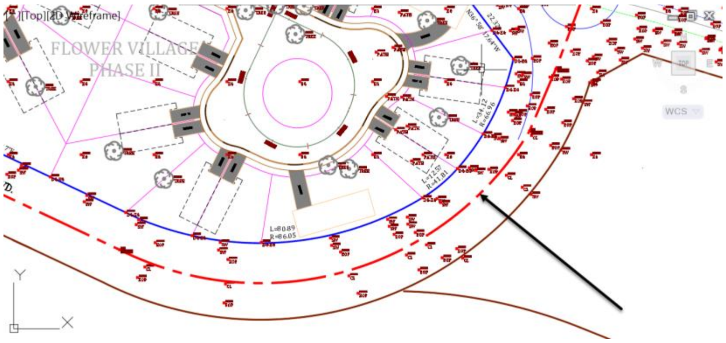

4. The new centerline points (**DG-CL-J**) are created along the polyline (not to be confused with the survey centerline points (**CL**)). For now, don't worry about the zero elevations. We will see how to assign elevation by design or from an existing ground.

## 4.8 Creating a given number of points on a polyline per segment: Divide Object

1. Now let's say we want to have a given number of points on the road. To do that, start the point creation toolbar. Then, run the **Divide Object** command:




2. Select the red line representing the centerline of **Jasmine Blvd**.



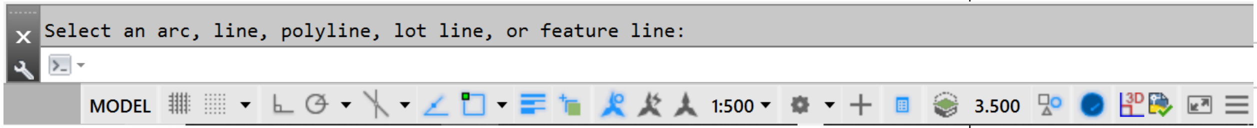

3. Enter 30 at the command line to divide the polyline into 30 segments and create points at the end of each segment;

4. Enter 0 for offsets;

5. Enter DG-CL (design points representing the centerline) for descriptions;

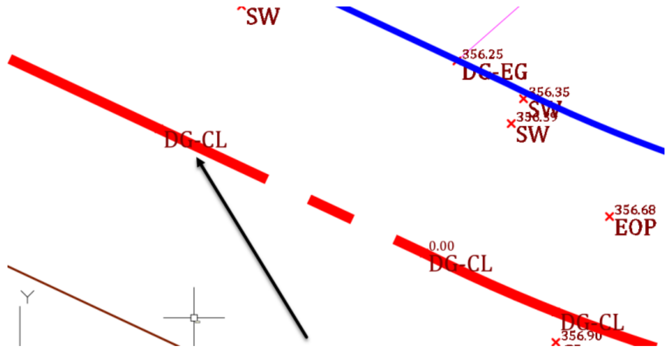
6. When prompted for an elevation press Enter to ignore the request; We will see later how to assign design elevations using different criteria.

7. Continue typing Enter at the command line until it becomes blank:




NOTES

8. The **DG-CL** points are created and appear in the drawing, without point elevation as specified.



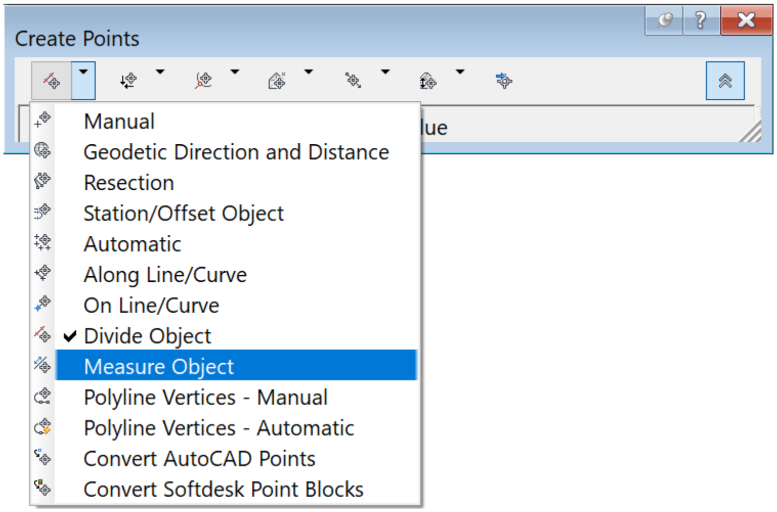

NOTES

NOTES

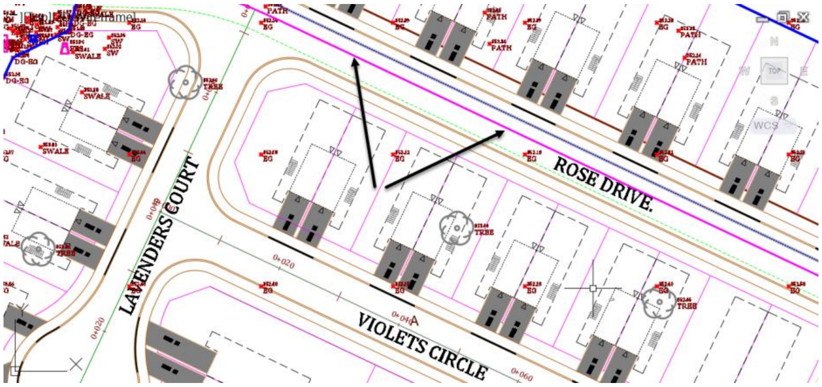
## 4.9 Creating Points on a polyline by intervals.

Instead of creating a set number of points, we can create points at set intervals, for example, a point every **30m** or **100ft**.

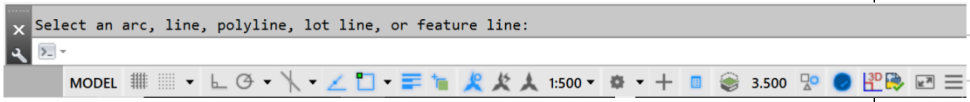
1. Run the **Measure-Object** command to create points.




Click on the centerline of **Rose Drive**, the magenta polyline.




2. For starting station, hit Enter to accept 0.000 (the starting point of the polyline)
3. For end station, press Enter to accept 460.42m or 1,510.56ft (the end of the polyline).
4. Enter at the command line to accept an offset of 0 from the line.
5. For interval type 30m or 100ft at the command line. This indicates that we want to create points every 30m or 100ft.
6. For the Description Enter DG-CL at the command line
7. Press enter to ignore the request for elevation.
8. Keep pressing Enter until the command line is blank.



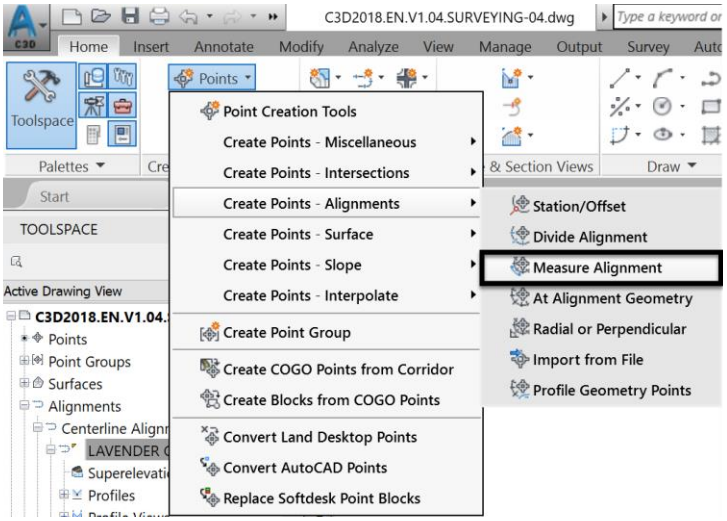

9. We have now created points at given intervals along the centerline. Case in point, this would be useful to designers when they receive a request from a surveyor for points at the major station of a baseline.

10. Save your work.

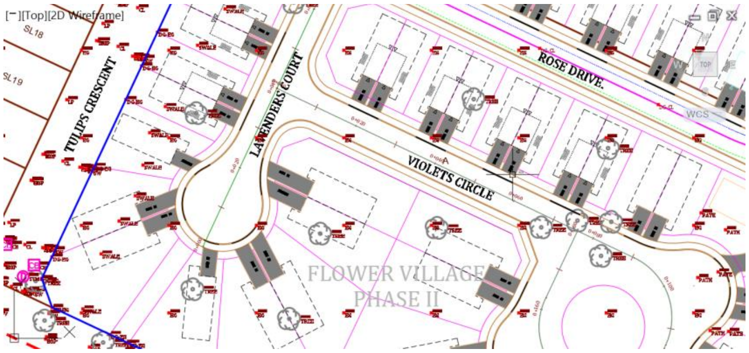
NOTES

## 4.10 Creating points on an alignment

1. The previous exercise can be used for an alignment instead of a polyline. In this case, the command needed would be **Measure Alignment**:




2. Click on the centerline of **Violets Circle**




3. For starting station hit Enter to accept 0+000 (the starting point of the polyline)

4. For end station, press Enter to accept 0+179.74m or 589.70ft (the end of the alignment).

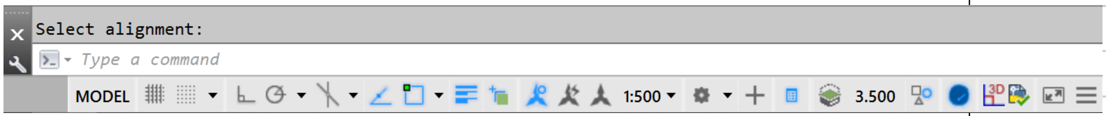
5. Press Enter to accept an offset of 0.00m (0.00ft) from the line.

6. For interval, type 30m or 100ft at the command line. This indicates that we want to create points every 30m or 100ft.

7. For the Description Enter DG-CL.

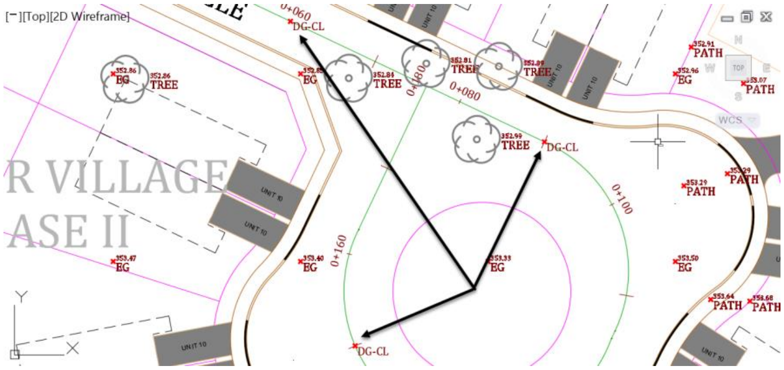
8. Press Enter to ignore the request for elevation.

9. Keep pressing Enter, until the command line is blank.




NOTES

10. New points are now created along **Violet Circle**, at a **30m** or **100-ft** interval.



NOTES

## 4.11 Creating Point Groups

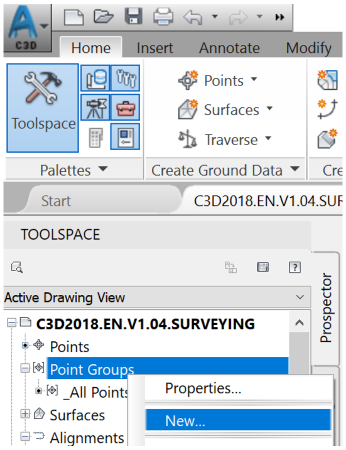
**Point Groups** are used to organize and manage points in a project. They also allow us to assign display styles and labels to points belonging to the same group. This can be done by overriding the property assigned by a description key set, if one is used. For example, we would like to separate the existing ground points from the design points. Maybe existing points don't need to display elevation information, since contours will be more than enough.

On the other hand, design elevations need to be shown with precision at specific locations, since the builder or contractor will use them to set grades. To do this, we simply need to create two point groups. One for the existing points and another for the proposed points. Each group will have its own **Point Style** and **Label Style**.

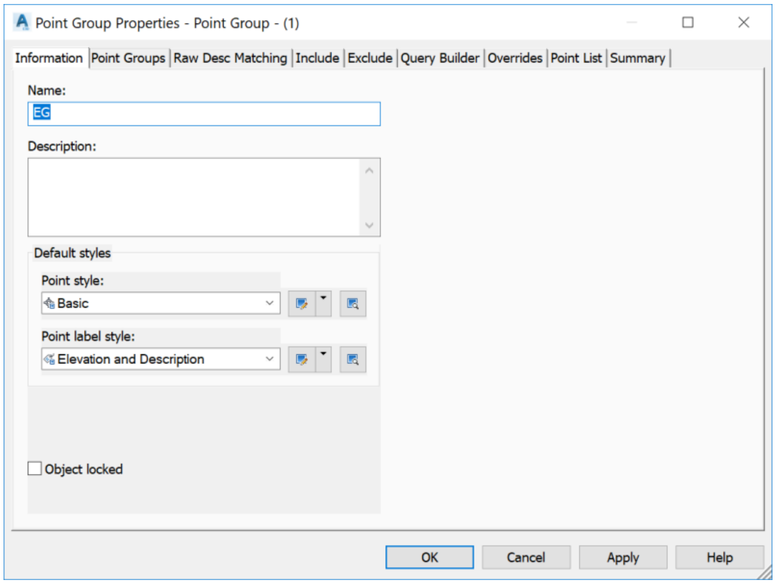
To create a point group:

1. Activate the prospector tab on the Toolspace

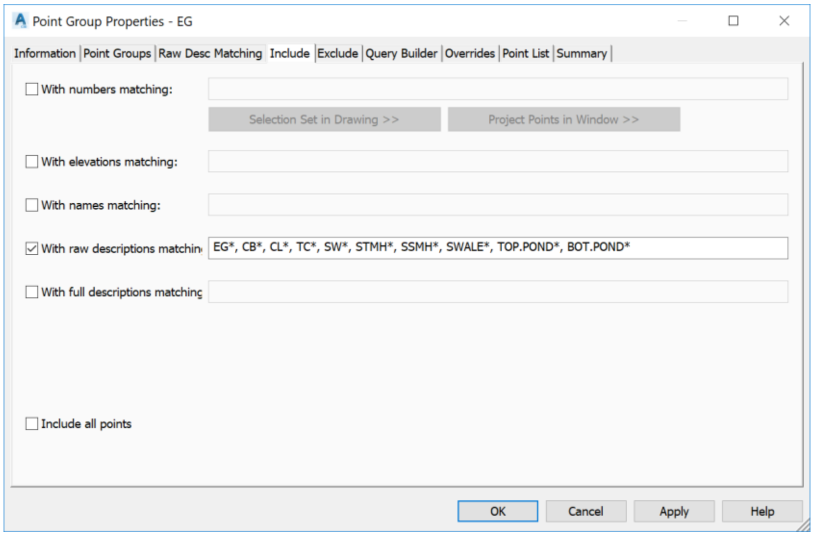
2. Right-click on Point Groups and select New.




- **Information** tab: assign the name **EG** (Existing Ground), style **Basic** and label style **Elevation and Description.**

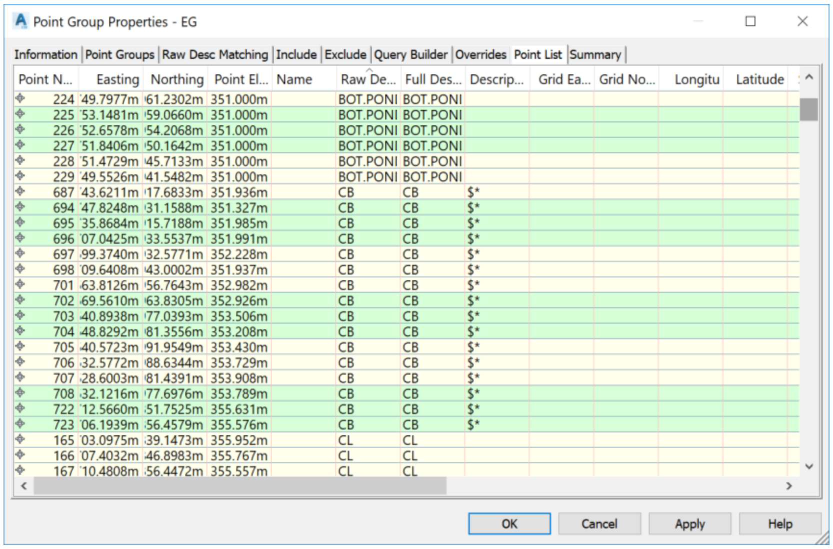


- **Include** tab: At the line **With Raw description matching,** type the following **EG*** (for Existing Ground survey shots), **CB*** (for Catch Basin rim elevations), **TC*** (for Top of Curbs) **SW** (for Sidewalks), **STMH** (for Storm Manholes), **SSMH** (for Sanitary Sewer Manholes), **Swale** (for Top of bank and bottom of swale elevations), **Top.Pond** (for Top of Pond), **Bot.Pond** for (Bottom of Pond).




We include the * wildcard sign to also include points, in situations where additional numbering is needed during the surveying process. For example, when names such as STMH-1, STMH-2, and the like, are used to collect survey manholes with unique identifiers. Using STMH* will make sure all STMHs are included in the point group.

1. Before exiting, we can verify the list of points included in the group, on the **Point List** tab.



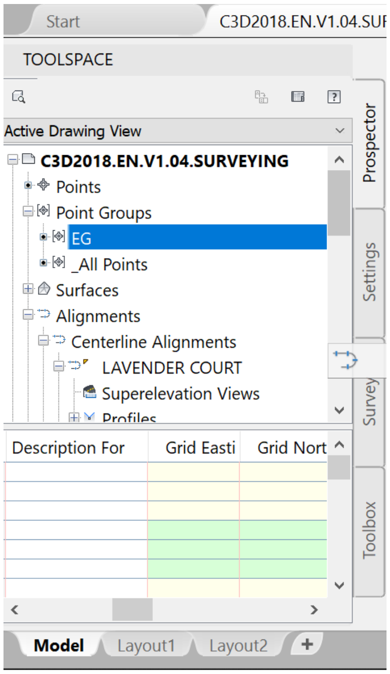

| Point N... | Easting | Northing | Point El... | Name | Raw De... | Full Des... | Descrip... | Grid Ea... | Grid No... | Longitu | Latitude |
|---|---|---|---|---|---|---|---|---|---|---|---|
| 224 | 49.7977m | 61.2302m | 351.000m | | BOT.PONI | BOT.PONI | | | | | |
| 225 | 53.1481m | 59.0660m | 351.000m | | BOT.PONI | BOT.PONI | | | | | |
| 226 | 52.6578m | 54.2068m | 351.000m | | BOT.PONI | BOT.PONI | | | | | |
| 227 | 51.8406m | 50.1642m | 351.000m | | BOT.PONI | BOT.PONI | | | | | |
| 228 | 51.4729m | 45.7133m | 351.000m | | BOT.PONI | BOT.PONI | | | | | |
| 229 | 49.5526m | 41.5482m | 351.000m | | BOT.PONI | BOT.PONI | | | | | |
| 687 | 43.6211m | 17.6833m | 351.936m | | CB | CB | $* | | | | |
| 694 | 47.8248m | 31.1588m | 351.327m | | CB | CB | $* | | | | |
| 695 | 35.8684m | 15.7188m | 351.985m | | CB | CB | $* | | | | |
| 696 | 07.0425m | 33.5537m | 351.991m | | CB | CB | $* | | | | |
| 697 | 99.3740m | 32.5771m | 352.228m | | CB | CB | $* | | | | |
| 698 | 09.6408m | 43.0002m | 351.937m | | CB | CB | $* | | | | |
| 701 | 63.8126m | 56.7643m | 352.982m | | CB | CB | $* | | | | |
| 702 | 69.5610m | 63.8305m | 352.926m | | CB | CB | $* | | | | |
| 703 | 40.8938m | 77.0393m | 353.506m | | CB | CB | $* | | | | |
| 704 | 48.8292m | 81.3556m | 353.208m | | CB | CB | $* | | | | |
| 705 | 40.5723m | 91.9549m | 353.430m | | CB | CB | $* | | | | |
| 706 | 32.5772m | 88.6344m | 353.729m | | CB | CB | $* | | | | |
| 707 | 28.6003m | 81.4391m | 353.908m | | CB | CB | $* | | | | |
| 708 | 32.1216m | 77.6976m | 353.789m | | CB | CB | $* | | | | |
| 722 | 12.5660m | 51.7525m | 355.631m | | CB | CB | $* | | | | |
| 723 | 06.1939m | 56.4579m | 355.576m | | CB | CB | $* | | | | |
| 165 | 03.0975m | 39.1473m | 355.952m | | CL | CL | | | | | |
| 166 | 07.4032m | 46.8983m | 355.767m | | CL | CL | | | | | |
| 167 | 10.4808m | 56.4472m | 355.557m | | CL | CL | | | | | |

NOTES

NOTES

2. Now click Ok to close the Point Groups Properties window

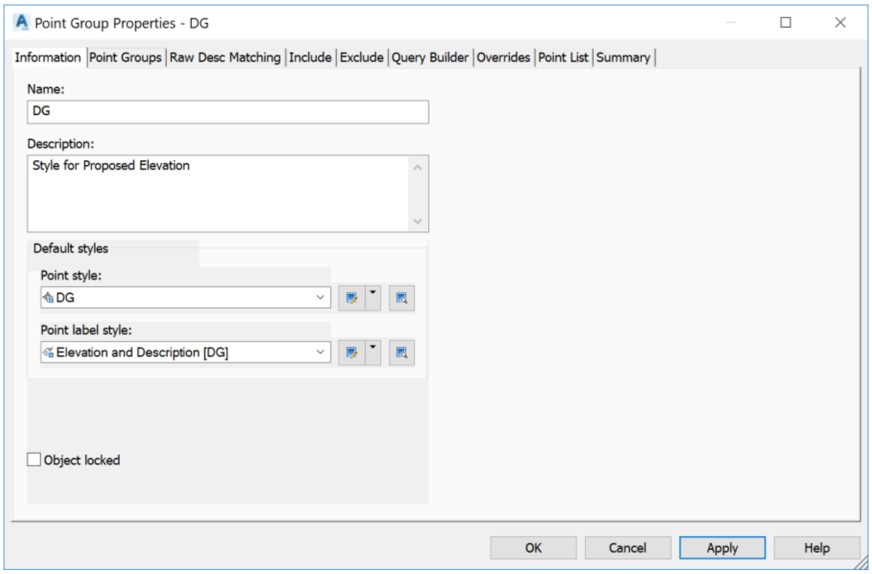
3. The point group EG is now created, and we see that it is added to the point groups collection in the Prospector.




Now let's create a **Point Group** of design points.

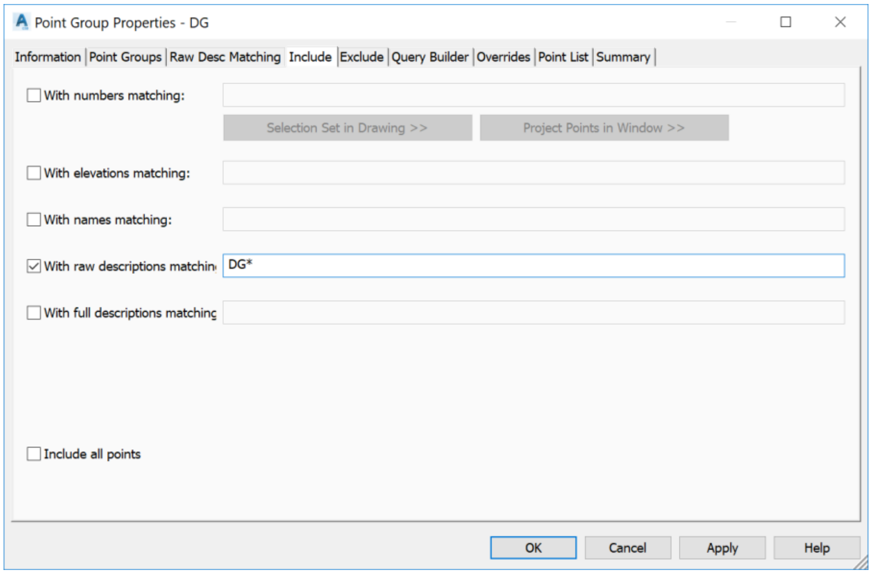
1. Repeat the previous two steps, creating a point group and assigning its definition.

2. On the Information tab: assign the name DG (Design Ground), a Basic style and an Elevation and Description style.

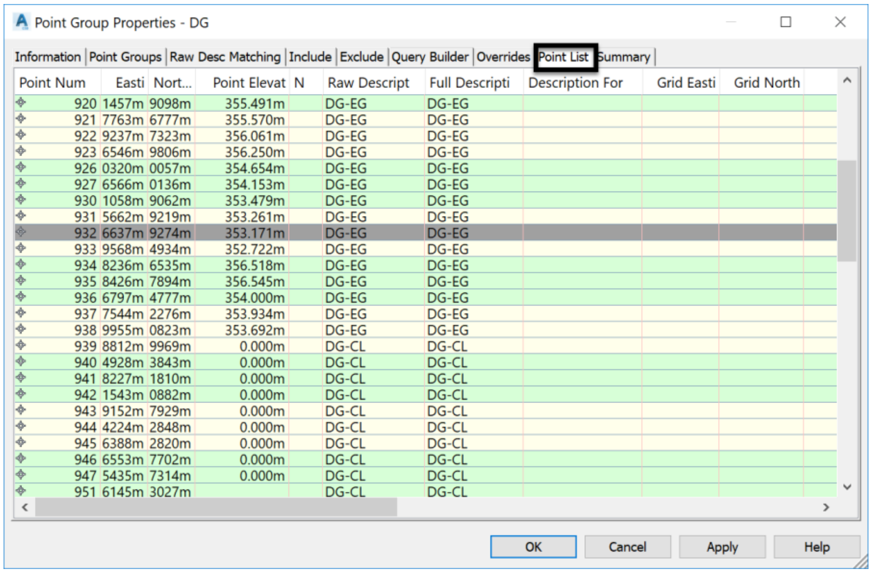


NOTES

3. On the **Include** Tab, set **With Raw definition** to **DG *** (the wildcard * will include all points starting with DG).




4. You can check the list of points included in the **DG** point group from the **Point List** tab. Currently, that includes all **DG-EG** and **DG-CL** points created in previous exercises.




5. Click **OK.**

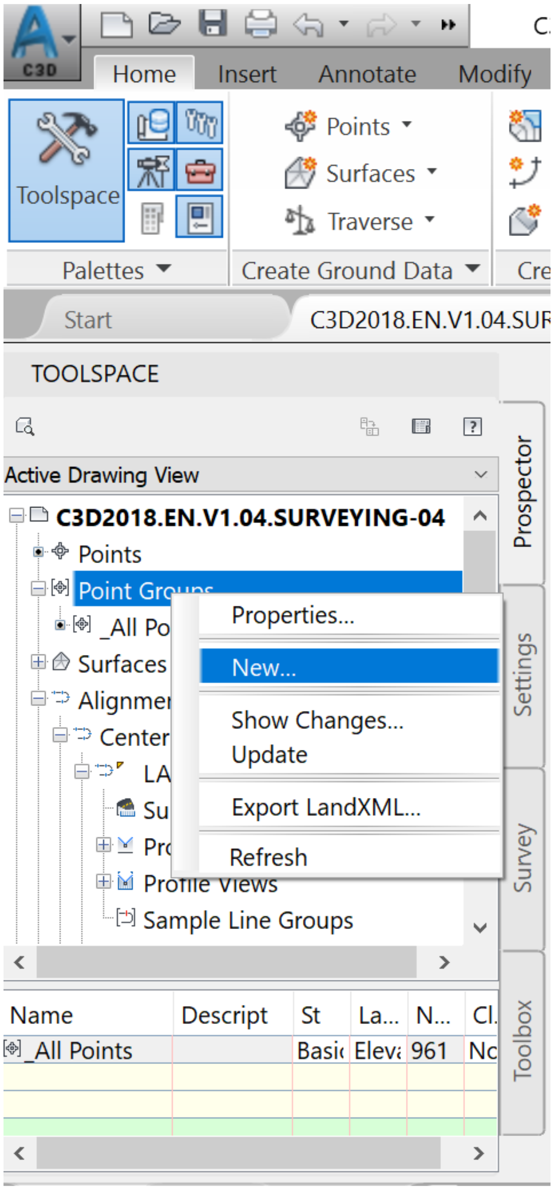
NOTES

## 4.12 Modifying Points

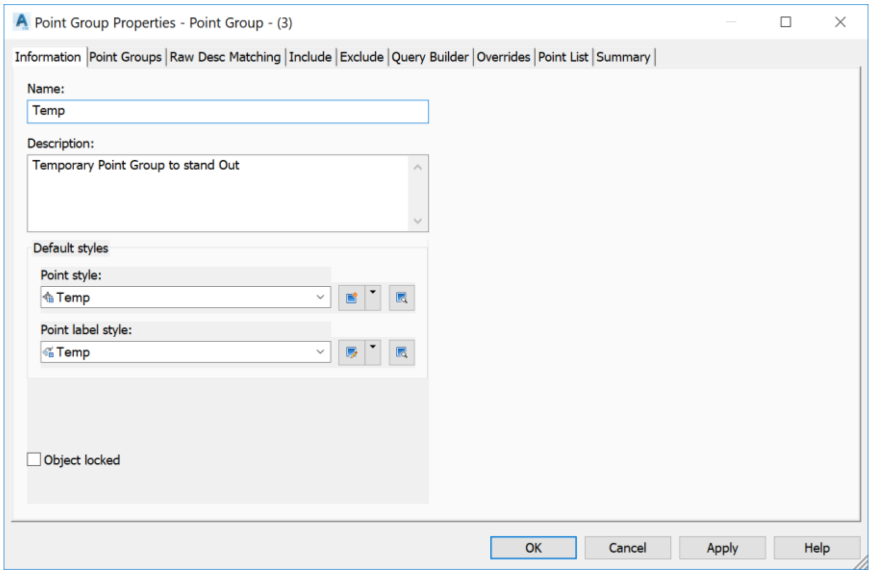
### 4.12.1 **Elevations from Surface.**

Now let's see a practical use of point groups. Remember the Centerline points we have created along **Jasmine Blvd**? Now we can select only those points and make several other operations to them, using the point group feature. This will be a simple example. However, the applications in which we can use **Point Groups** are virtually endless.

1. Create a temporary **Point Group**, just for the **DG-CL-J** previously created, and call it **Temp**.

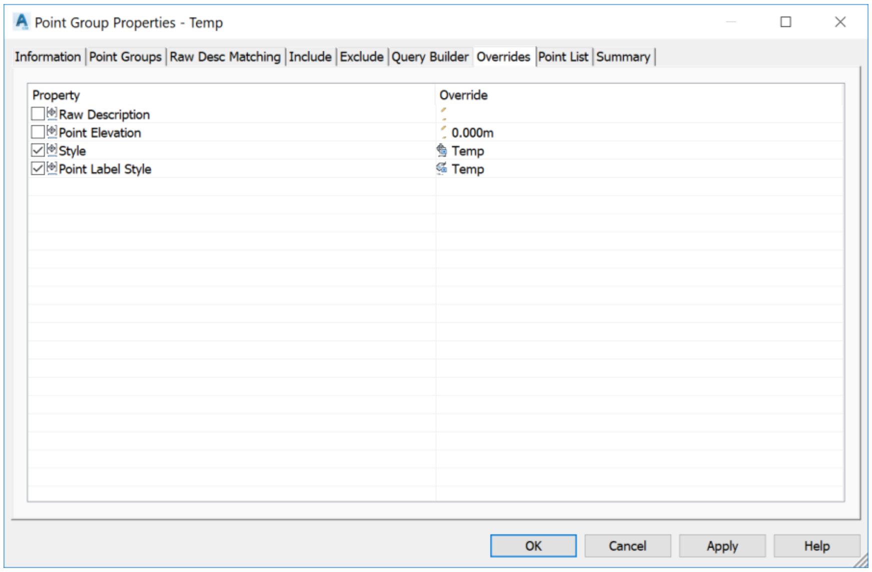


2. Name it **Temp** and assign the **Temp** styles. These styles are used when we want to make points standout for a rapid selection.




3. On the **Include** Tab, include all points with **the raw description matching**: DG-CL-J

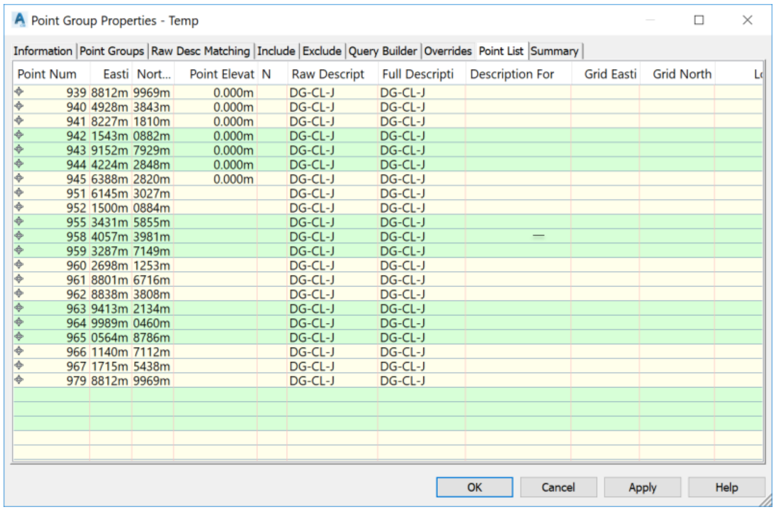
4. On the **Overrides** tab, activate **Style** and **Point Label Style**. We select this to override any style applied by the **Description Key Set** that we've created in the **Style and Templates** chapter.




NOTES

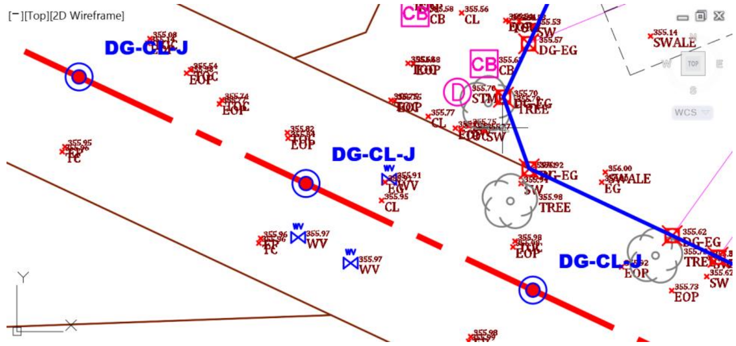
NOTES

5. Now, let's check the list of point picked up by the Group. Looking at the list, this is exactly what we wanted. We have picked up only the **DG-CL-J** points. Nothing else is included. We can now click **Ok** and go back to the **Prospector.**

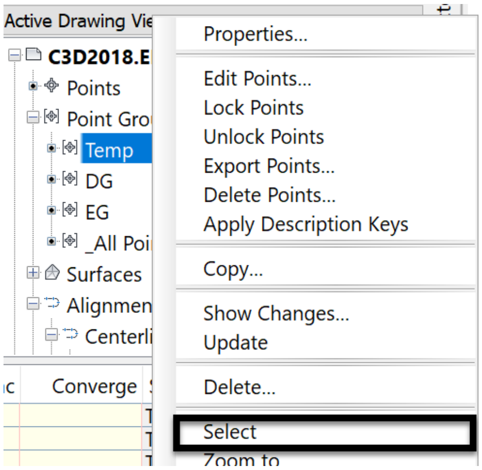



| Point Num | Easti | Nort... | Point Elevat | N | Raw Descript | Full Descripti | Description For | Grid Easti | Grid North | Lo |
|---|---|---|---|---|---|---|---|---|---|---|
| 939 | 8812m | 9969m | 0.000m | | DG-CL-J | DG-CL-J | | | | |
| 940 | 4928m | 3843m | 0.000m | | DG-CL-J | DG-CL-J | | | | |
| 941 | 8227m | 1810m | 0.000m | | DG-CL-J | DG-CL-J | | | | |
| 942 | 1543m | 0882m | 0.000m | | DG-CL-J | DG-CL-J | | | | |
| 943 | 9152m | 7929m | 0.000m | | DG-CL-J | DG-CL-J | | | | |
| 944 | 4224m | 2848m | 0.000m | | DG-CL-J | DG-CL-J | | | | |
| 945 | 6388m | 2820m | 0.000m | | DG-CL-J | DG-CL-J | | | | |
| 951 | 6145m | 3027m | | | DG-CL-J | DG-CL-J | | | | |
| 952 | 1500m | 0884m | | | DG-CL-J | DG-CL-J | | | | |
| 955 | 3431m | 5855m | | | DG-CL-J | DG-CL-J | | | | |
| 958 | 4057m | 3981m | | | DG-CL-J | DG-CL-J | | | | |
| 959 | 3287m | 7149m | | | DG-CL-J | DG-CL-J | | | | |
| 960 | 2698m | 1253m | | | DG-CL-J | DG-CL-J | | | | |
| 961 | 8801m | 6716m | | | DG-CL-J | DG-CL-J | | | | |
| 962 | 8838m | 3808m | | | DG-CL-J | DG-CL-J | | | | |
| 963 | 9413m | 2134m | | | DG-CL-J | DG-CL-J | | | | |
| 964 | 9989m | 0460m | | | DG-CL-J | DG-CL-J | | | | |
| 965 | 0564m | 8786m | | | DG-CL-J | DG-CL-J | | | | |
| 966 | 1140m | 7112m | | | DG-CL-J | DG-CL-J | | | | |
| 967 | 1715m | 5438m | | | DG-CL-J | DG-CL-J | | | | |
| 979 | 8812m | 9969m | | | DG-CL-J | DG-CL-J | | | | |

6. In the drawing, we now see the **DG-CL-J** point standing out due to the applied **Temp** style. We will later see how to create these types of styles and labels.

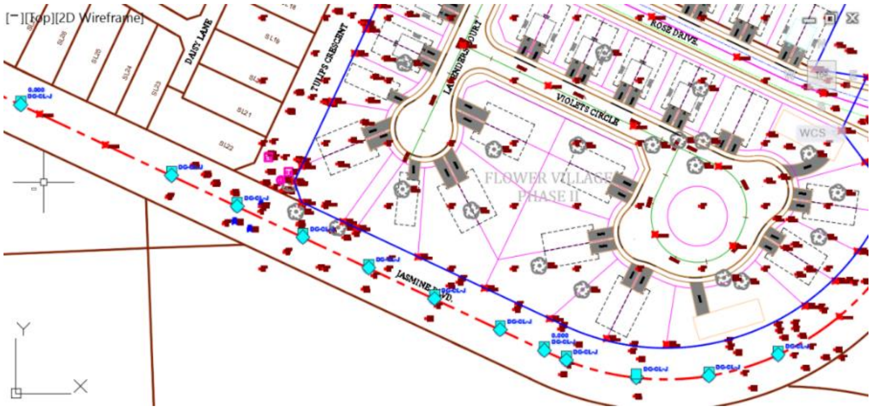



7. Now, **Jasmine Blvd** is already an existing road. So, we may want to assign all points to an existing surface elevation. This is very easy to do with a point group.  Simply go to the **Temp Point Group**, select the points and assign elevation from the **EG surface**.

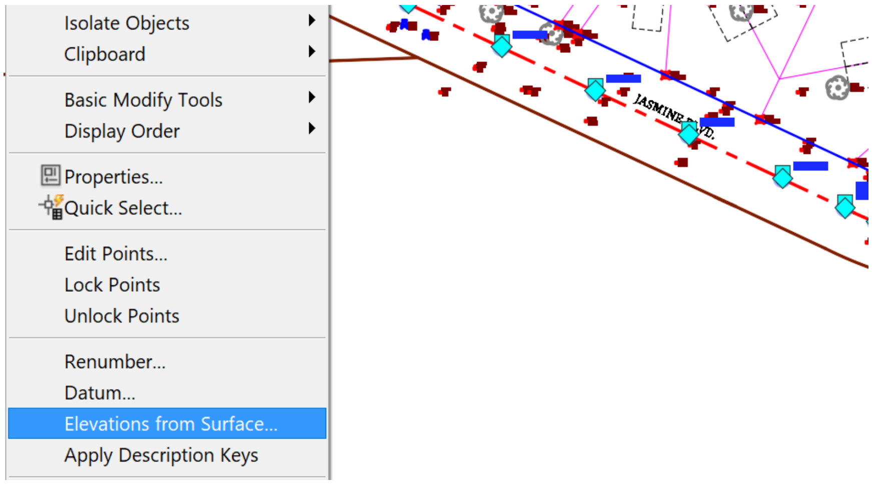


NOTES

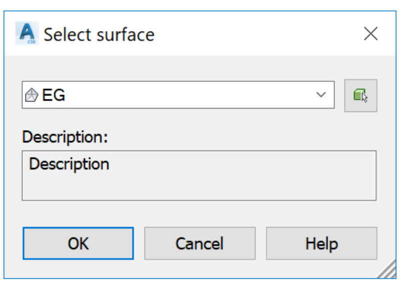
8. All points belonging to the group are selected.




9. Right-click and select **Elevations from Surface**

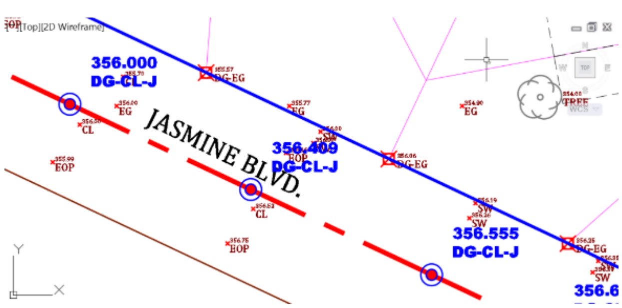



10. Choose **EG** in the **Select Surface** window




11. Click **Ok**

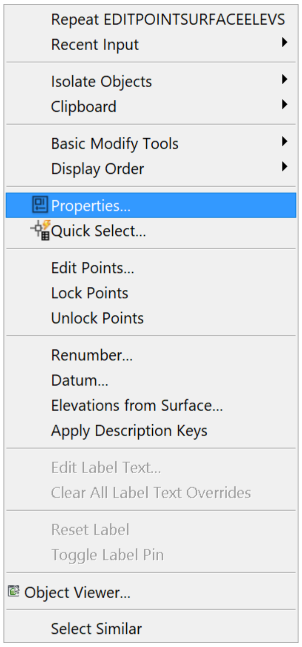
12. All points in the **Temp** point group are now assigned elevations matching the existing ground.



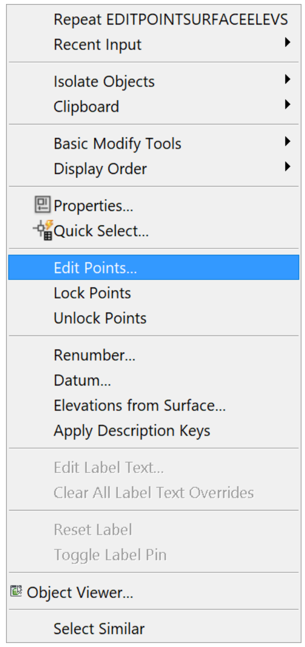
NOTES

In addition to making them stand out and assigning elevations from a surface, we can also select the **Point Groups** and do the following:

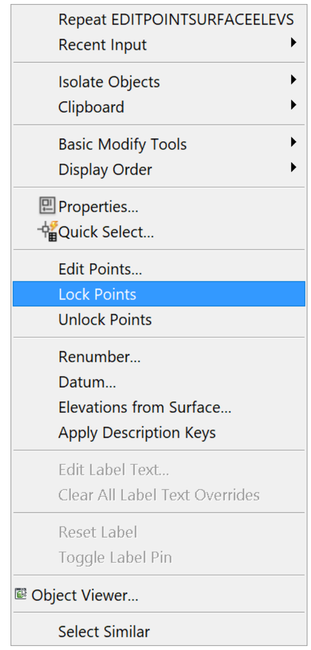
**Change the properties** of points such as: Raw description, styles, layers, colours , scale, etc.



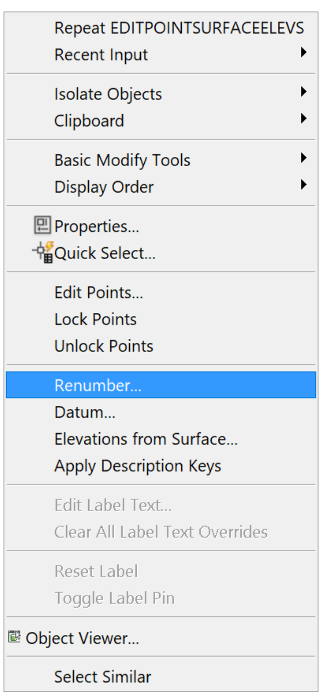

**Edit points** to change properties such as: numbers, coordinates, descriptions, and the like.



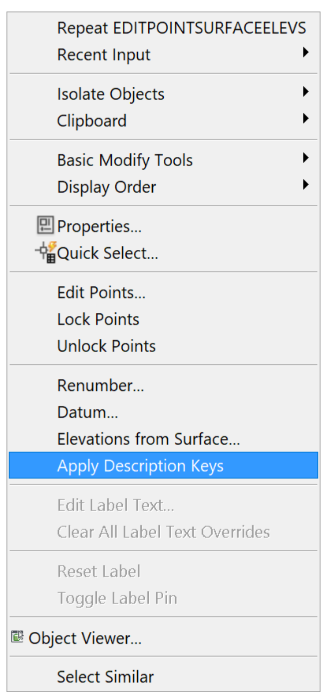

**Lock and unlock points**. This is particularly useful for survey points. We don't want to inadvertently modify them.



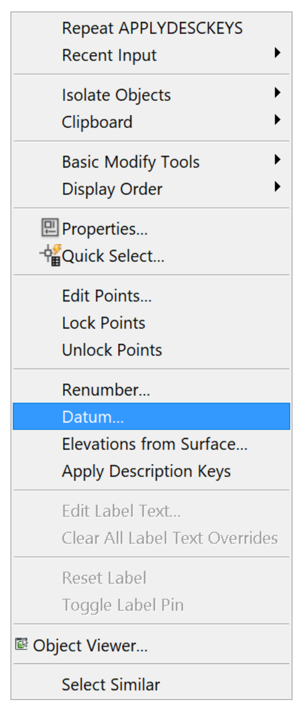

**Renumber Points**: As we add and delete points, point numbers get offset. To keep a tidy point group, we may need to renumber them.




**Apply Description Keys**: As we create points and update the description Key Set, updates to both may be needed.



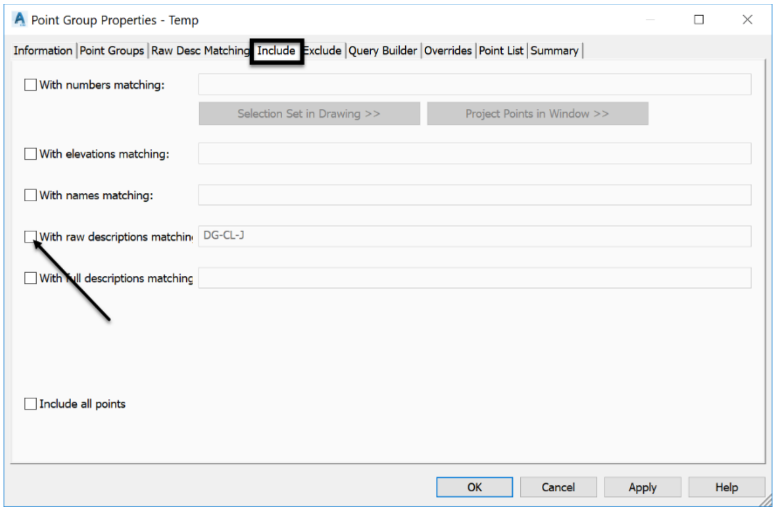

**Change Point Datum**: We need to change a datum for points (raise or lower all points by a given value). Example, correcting a survey.



### 4.12.2 **Disabling a Point Group.**

Since we have achieved the intended goal of the **Temp** point group, you need to disable it. We have several options to do that. Since it's just a temporary point group intended for momentary and specific tasks, we can just empty it by clearing the point group definition. To do that:

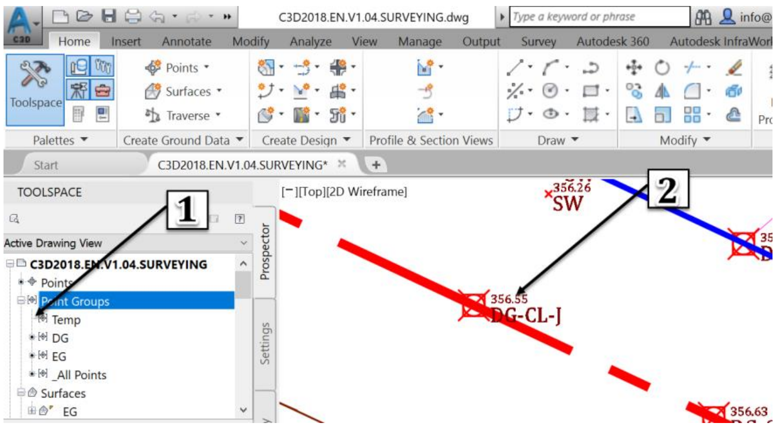
1. Return to the **Point Group Properties**, and clear the definition on the **Include** tab.




The **Temp** point group is now empty. Note that the points have not been deleted. They simply are not part of this group anymore. We can make the following observations:

2. The Temp point group does not show the dot to the left, compared to the other groups (DG and EG). This means that it is empty and does not have any point included.

3. The points in the drawing do not display with the temporary style and label anymore. Note that you may need to regen your drawing to have the styles updated.




NOTES

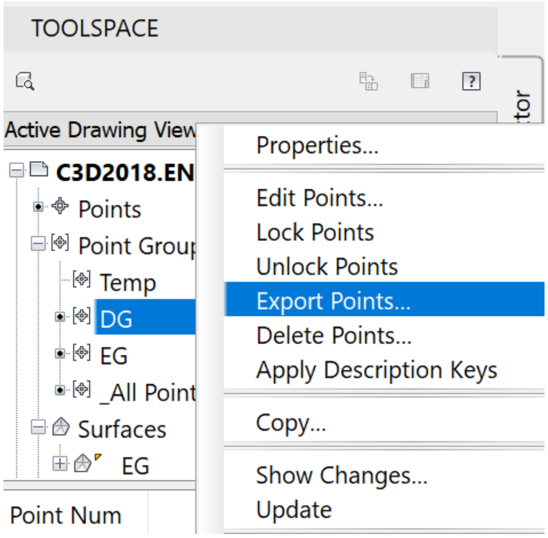
NOTES

## 4.13 Exporting points

When a design is complete, we need to share the information with a construction team. Often time, we will need to stake out the design intent using points. Therefore, we need to be able to export points for the design drawing. The exported points can then be imported to a survey gear or other **BIM** (Building Information Modeling) software.

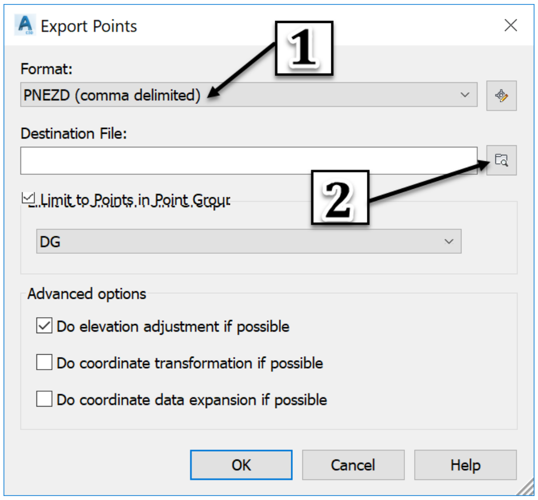
Let's export the few design points created so far:

1. To export a point, from the **Prospector**, in the point group collection, right-click on **DG.** Then click on **Export points**.

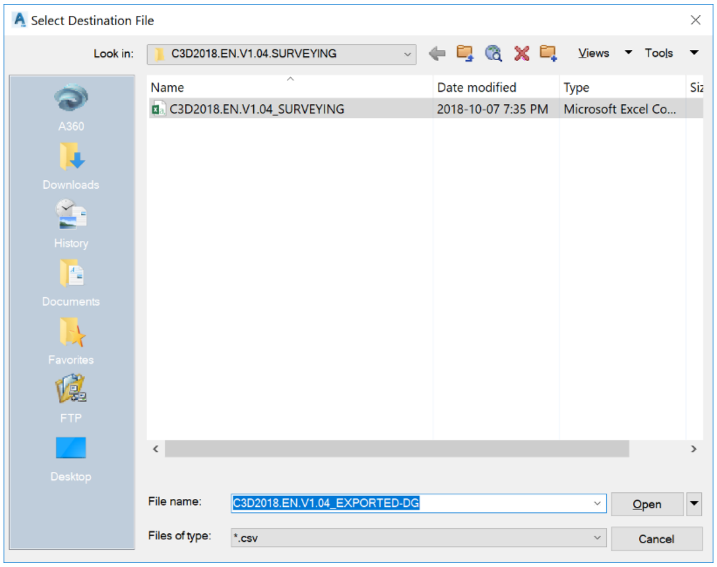


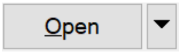

2. Choose a folder on the computer to export the text file.

3. Next, choose the requested file format (PNEZD CSV file is usually a good choice), then click the directory to save the file.



4. After that, select the folder to save the exported file to:




5. Click on 

.

6. Then, click on 


7. The **DG** point group is now exported, and the CSV created can be used for field stakeout or importing in another drawing or survey equipment.

NOTES

NOTES

# 5 SURFACES

Conventional field survey data usually only collect existing ground at specific spots. Typically, these are locations with particular features that may impact the design of the proposed project. These features may include utility infrastructures, manhole rims, inverts, or pipe fittings. LiDAR technology and photogrammetry are facilitating survey data collection. However, they are not always practical or cost-effective. Especially for legal boundary determination, or engineering design.

Therefore, we should create a terrain model (or surface) in the office, to fill in the gaps in the field survey. The creation of a surface has several advantages:

1. First, we can get information, by an approximation, at locations not originally surveyed;
2. Second, we can compare several surfaces to estimate earthworks quantities;
3. And third, they facilitate field stakeout of the design.

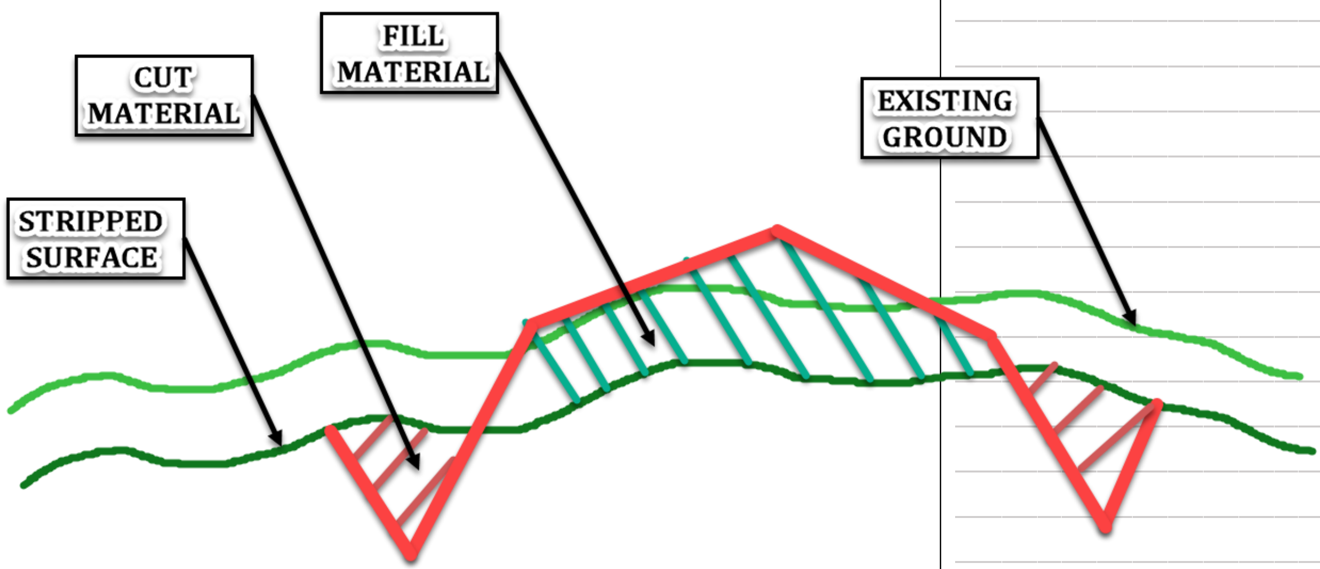
Civil 3D has a set of tools that allows you to use basic drawing objects (points, polylines, polyfaces, **DEM** files, and more) to create these surfaces.

Civil 3D makes it also possible to create different kinds of surfaces, including:

1. An **Existing Ground** surface, to replicate the surface before any development operation.
2. A **Stripping Surface**. It is essentially the natural ground with the layer of topsoil, typically the first **10cm** or **4in** to **30cm** or **12in**, removed. This operation allows construction to be done on clean and stable ground. The topsoil can be reused at a later stage on the current project or a separate one.
3. And lastly, the **Final Surface**: This is the finished or final grade surface of the site.

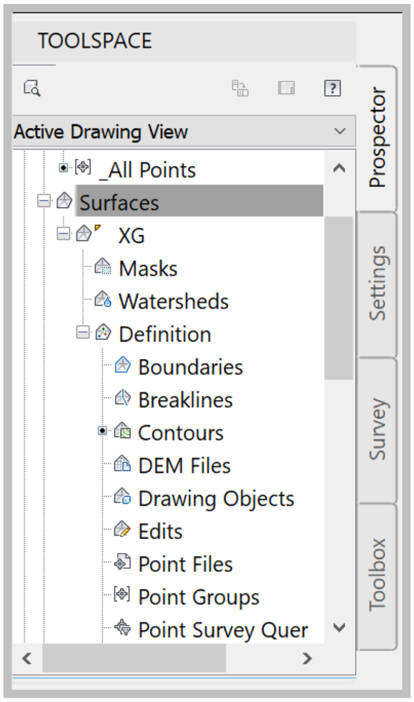
We also have intermediate surfaces. These include subbase and base surfaces. They are not always shown on construction plans, but calculated instead, from typical cross sections.

This figure summarizes the different types of surfaces we will most frequently use.




## 5.1 Creating and defining surfaces

A surface is almost always created and defined from the **Definition** section in the surface tree structure.




NOTES

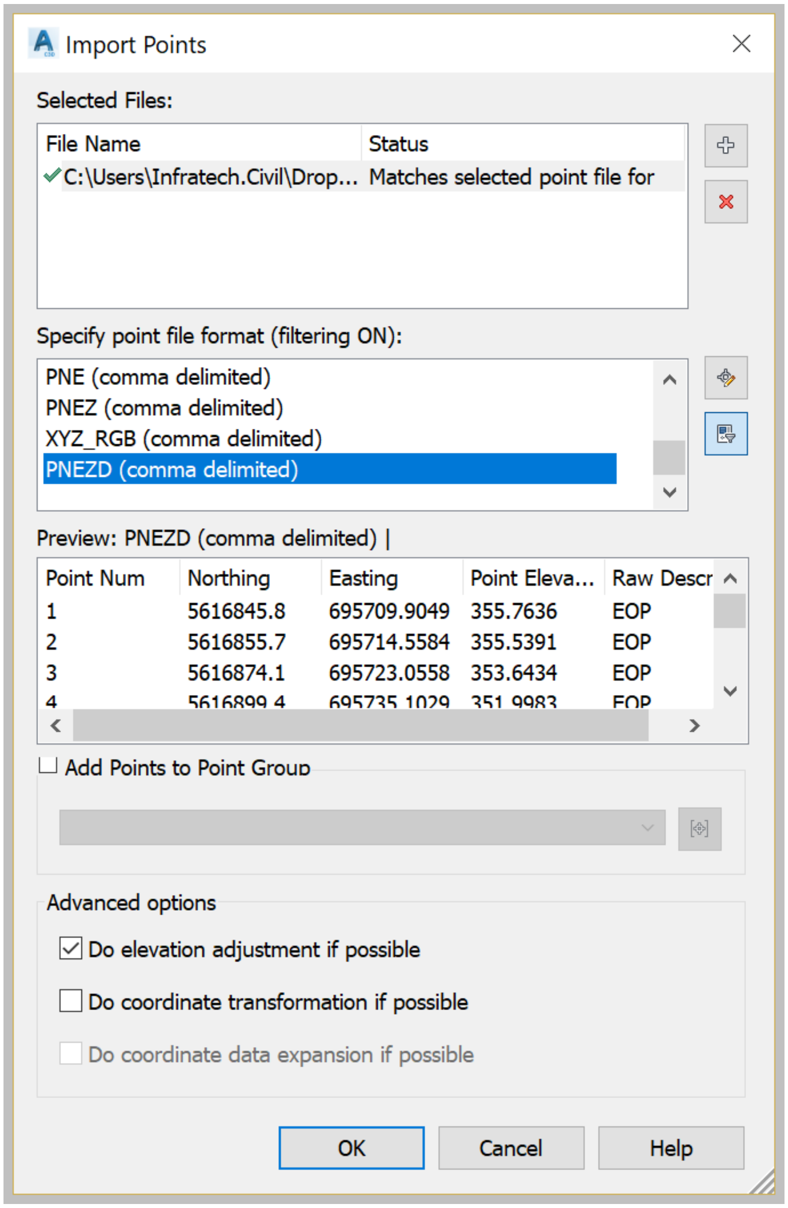
NOTES

Let's practice working with surfaces.

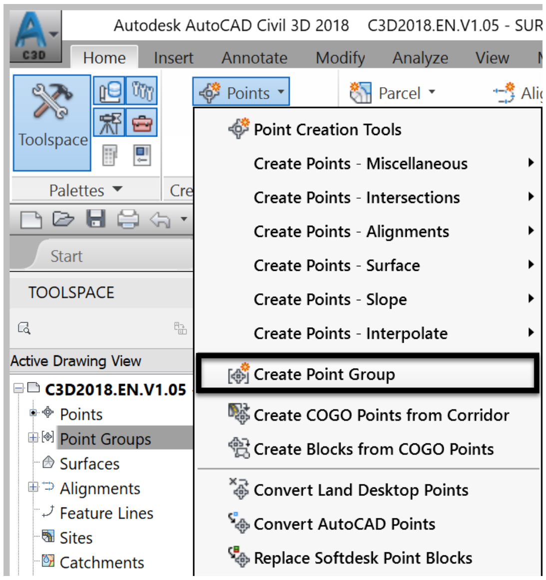
1. First, Open the **05.01-Surface dwg** file in the **Lesson 05** practice folder.

2. Import the **CSV** point data file located in the Survey folder. We have already done it in the Survey lesson but let's review the steps.

   - Click on the **Insert** tab
   - Select **Points from file.** Optionally, you can launch the point creation toolbar and run the import command.
   - From there, import the point file by choosing the **PNEZD** commas delimited format.
   - Lastly, click **OK**

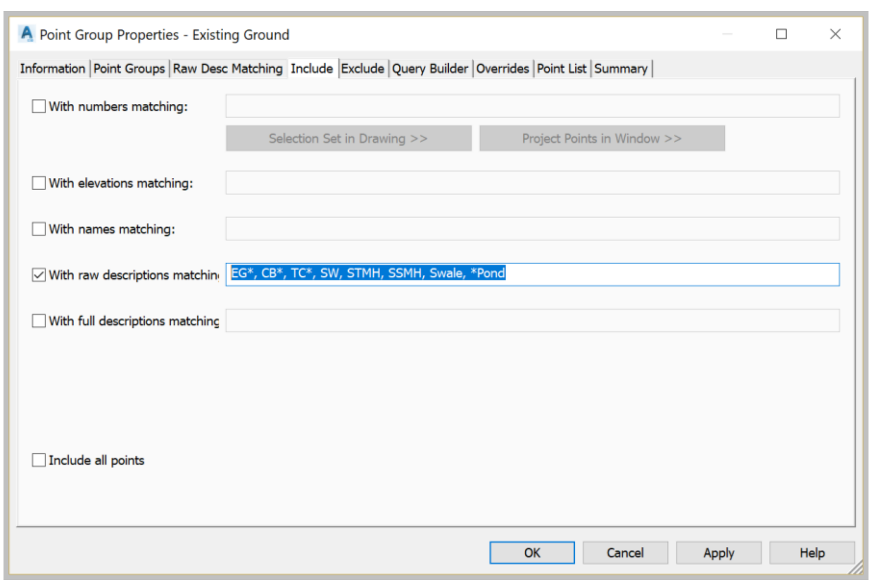



3. Create a point group from the prospector in the **Point groups** section or from the ribbon.




4. Name it **Existing Ground**. This point group will include all points that represent the existing ground elevations. We are not necessarily using all the points in the survey file, because points such as pipe inverts need to be excluded.

5. Next, switch to the **Include** tab. At the **With raw description matching** line, click to check the box. Then, include the raw descriptions in the definition. That would include **EG*** (for Existing Ground survey shots), **CB*** (for Catch Basin rim elevations), **TC*** (for Top of Curbs), **SW** (for Sidewalks), **STMH*** (for Storm Manholes), **SSMH*** (for Sanitary Sewer Manholes), **Swale** (for Top of bank and bottom of swale elevations), and ***Pond** (for Top.Pond and Bot.Pond).

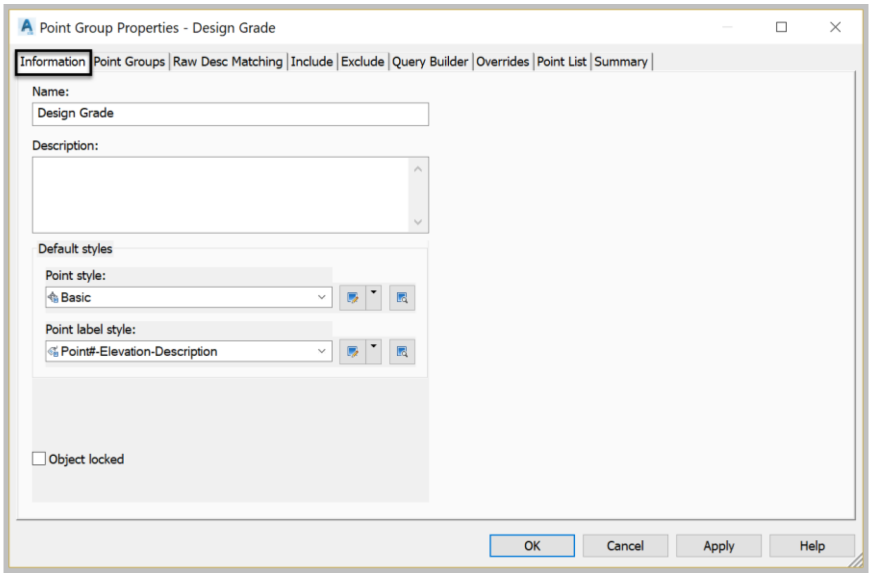



6. Click **OK**

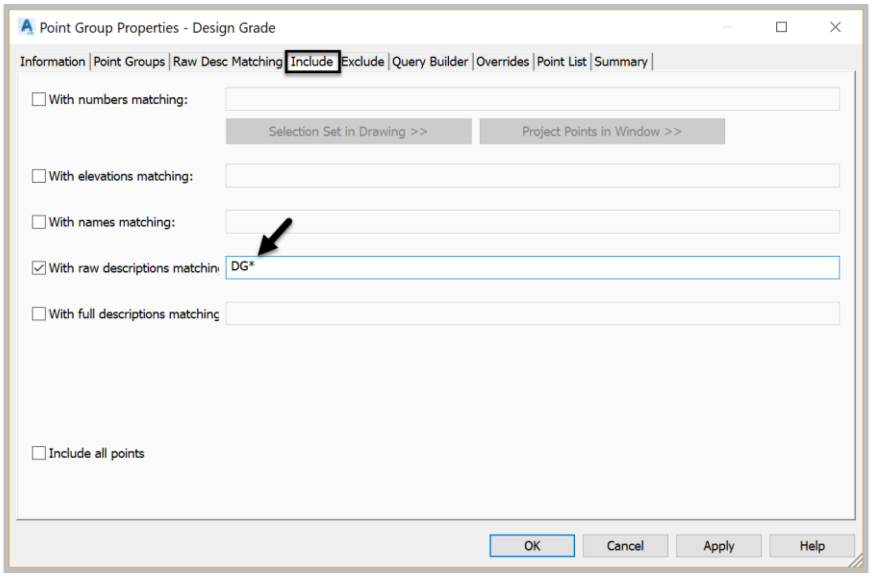
NOTES

NOTES

7. Next, create another point group and name it **Design Grade**.




8. Define the point group from the **Include** tab. Include all design points with the description starting with **DG.** Use the * wildcard to include points such as DG-OG, DG-CL, DG-FFE, and so forth. Once we do that, we will need to make sure all future design points will have a description starting with **DG.**

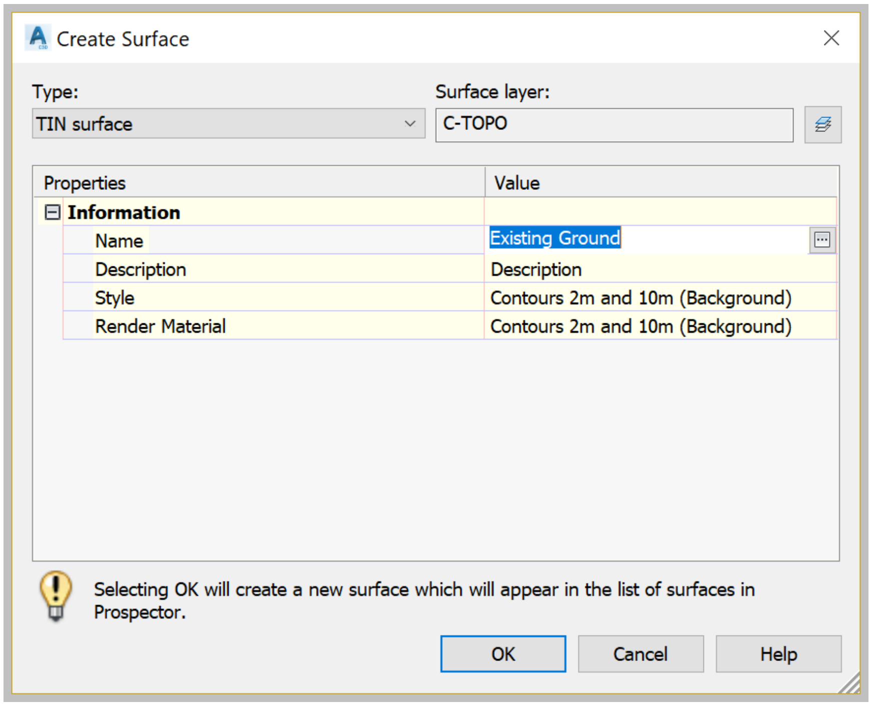


NOTES

## 5.2 Creating an Existing Ground Surface

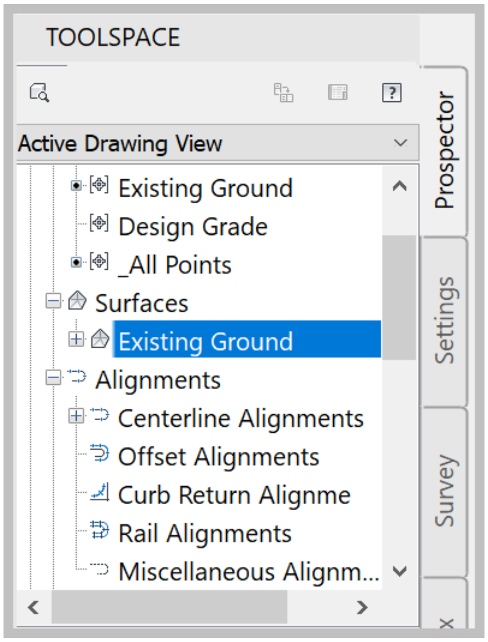
1. Now that we have defined the point groups, we need to create the surfaces. We can do that either by right-clicking on **Surface** in the **Prospector** and selecting **New** or by using the **Create Surface** command from the ribbon.

2. In the new window, name the Surface **Existing Ground** and select the default surface styles.




3. Click on **OK**

4. The **Existing Ground** surface now appears in the Prospector's window

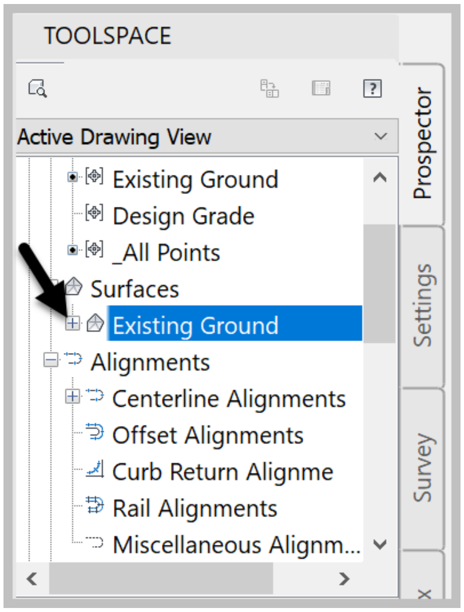


## 5.3 Defining a Surface

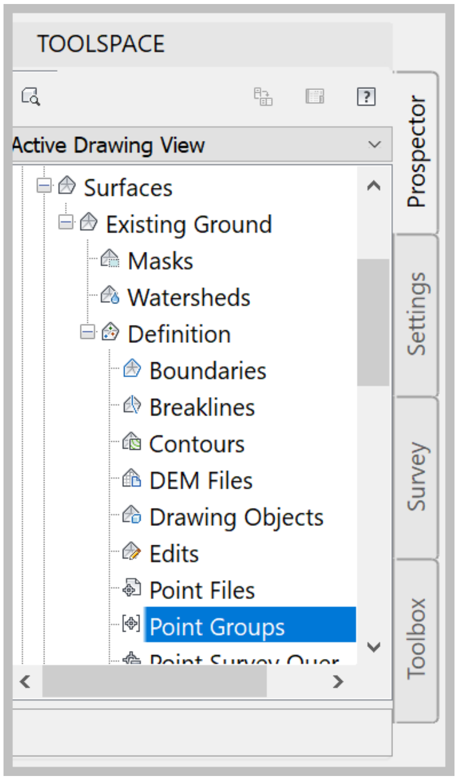
How we define a surface generally depends on the information on hand. Based on the data available, we can define a surface from survey points, contour data, breaklines, or simple AutoCAD entities, including lines, blocks, texts, and the like.

### 5.3.1 Defining a Surface by Point Groups

1. Open the **Surface Definition** section by clicking the **"+"** sign to the left of the surface.

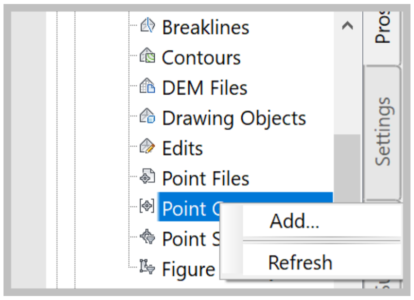



2. Scroll to the **Point Group** section.

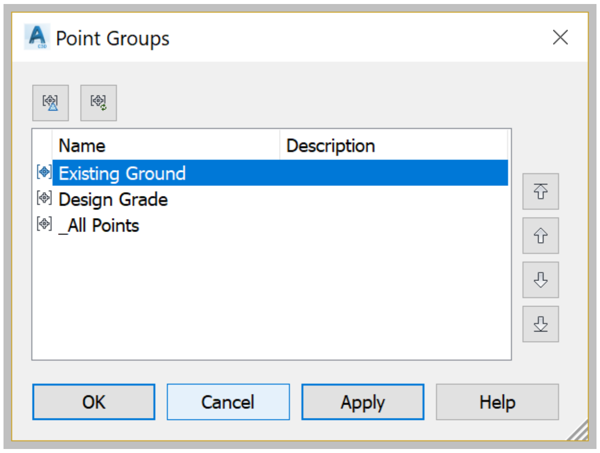



NOTES

3. Right-click and select **Add.**




4. Since we are creating a surface for the existing ground, we are going to select the **Existing Ground** point group that we have created before.

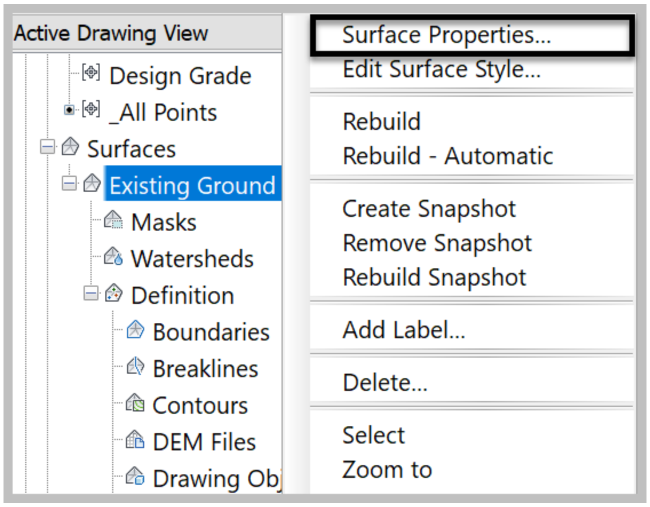



5. Click on **OK.**

6. The surface is now defined, using the existing ground point group.

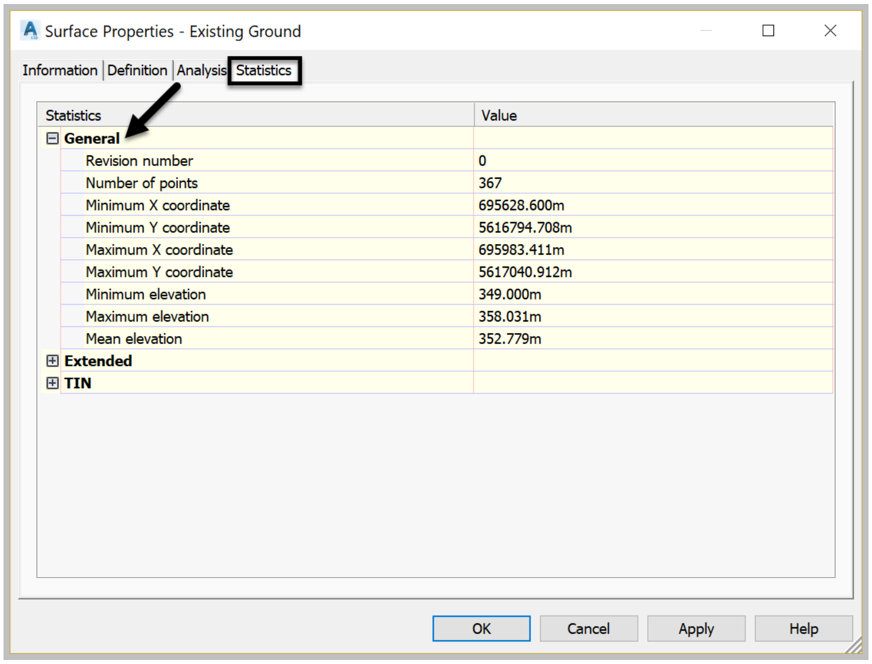
7. We can quickly check a few statistics on the surface. Just to make sure nothing abnormal happened during the process.

8. To do that, select the surface in the prospector, right-click and select **Surface Properties**




NOTES

9. Activate the **Statistics** tab and expand the **General** section.




10. For now, looking at the elevations, we are happy with the preliminary results. The elevations seem to be within the range of the survey data, the coordinates are reasonable, and the number of points is as expected.

11. Click on **OK** to close the statistics windows and return to the drawing.

We have now seen one of the most commonly used methods to define a surface, **Point Groups**.

We will explore more options as we progress through this course.

NOTES

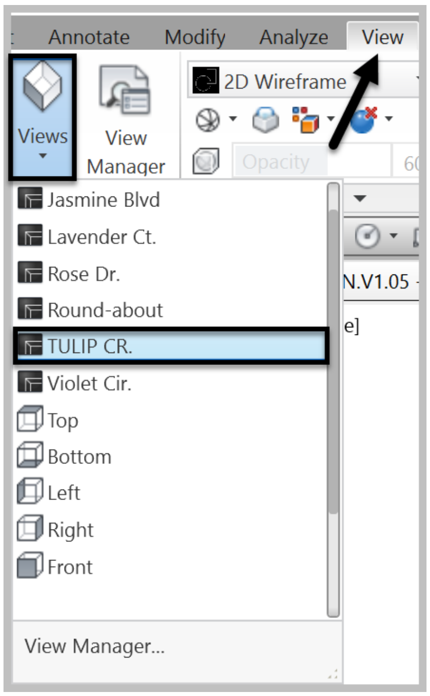
## 5.3.2 Defining a surface from breaklines

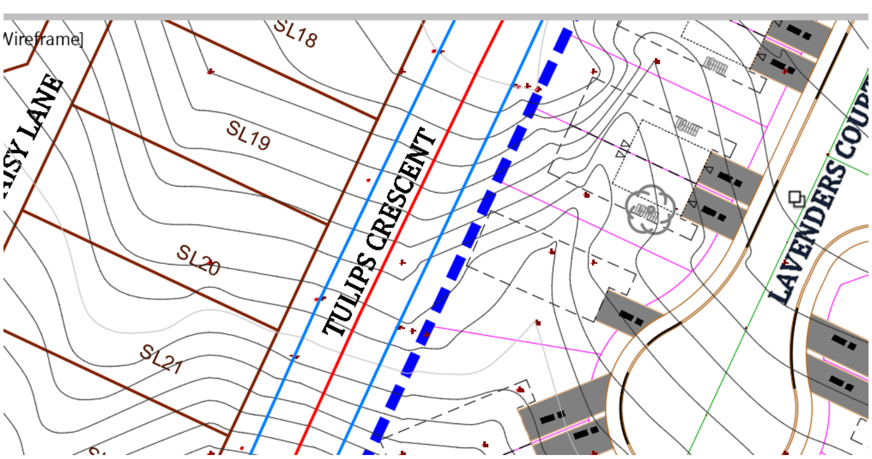
Breaklines are line data that can improve a surface. They provide more information to the surface and influence how it is triangulated. Adding a breakline forces the surface to create triangles connecting the vertices on the breakline. For example, we can create a more accurate surface by adding centerline, edge of pavement, top and back of curb breaklines to a surface instead of using points only.

Let's see an example.

1. Zoom to **Tulips Crescent**. If you don't know the streets yet, we've created named views for each street to help you find them. To zoom to a street:

   ○ On the ribbon, click on the **View** tab, then expand the **Views** drop-down box and select the view you want to zoom to, in this case, **Tulips Crescent**.



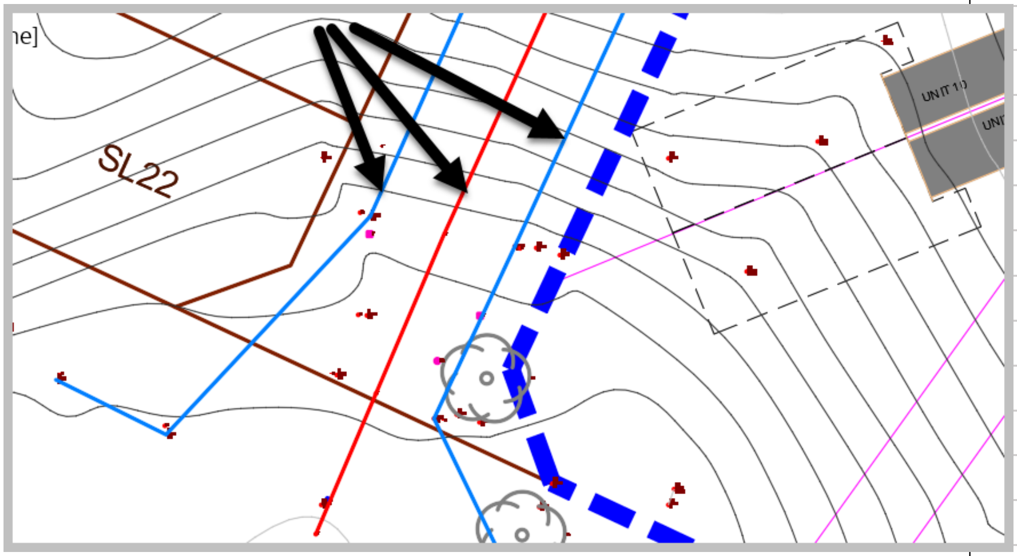




NOTES

NOTES

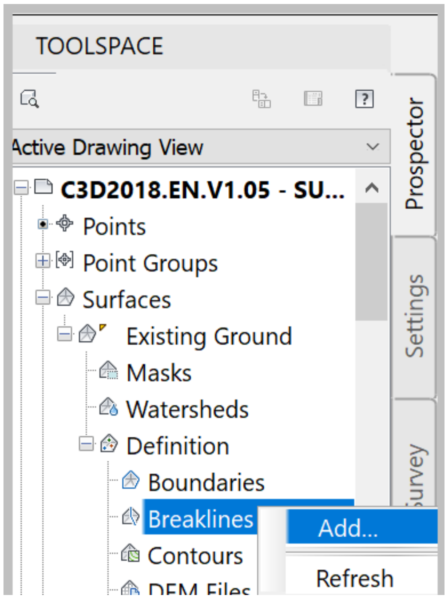
2. Notice the contours of the existing surface, created from point group. Now let's add the **breaklines** and see what difference they make.

3. Select the two polylines representing the **Edge of Pavement** (blue lines) and the **Centerline** (Red middle polyline).



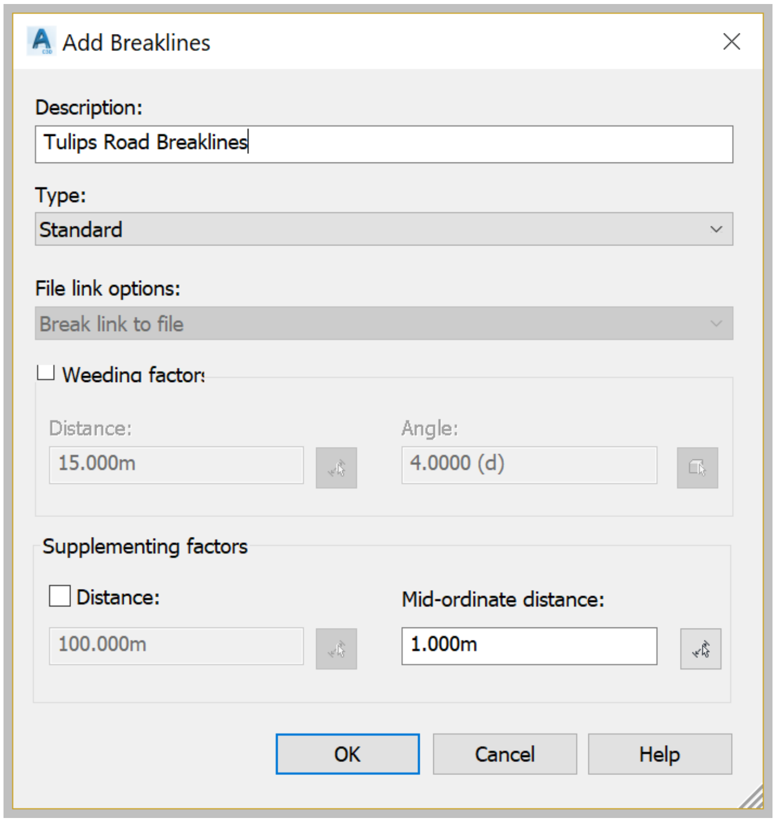

4. Then, expand the definition tree of the **Existing Ground** surface.

5. Right-click on **Breaklines** to select **Add.**



NOTES

6. In the **Description** field, use a text describing the purpose of the breakline.

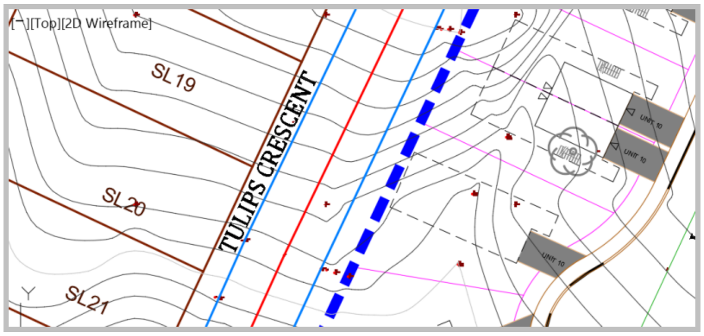



7. Usually, descriptions in Civil 3D are not required. Due to that reason, they are often overlooked. However, it is recommended that you always use explicit descriptions when prompted to. It makes it much easier to manage the surfaces and other entities. Especially, when the number of entities in the drawing becomes significant, and you need to track them.

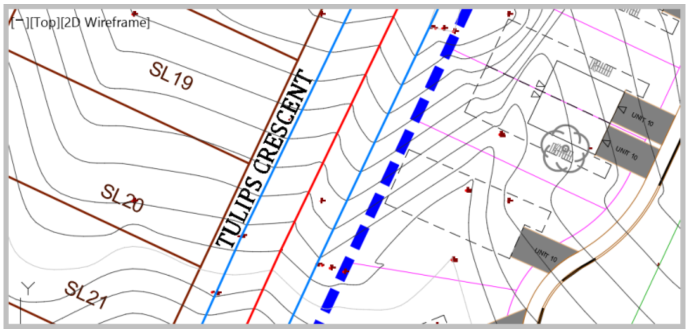
8. Click **OK** to close the **Add Breaklines** window.

9. Notice the surface contours **Before** and **After** adding breaklines. The breaklines make the surface look more realistic with better-looking road contours.

**BEFORE**




**AFTER**



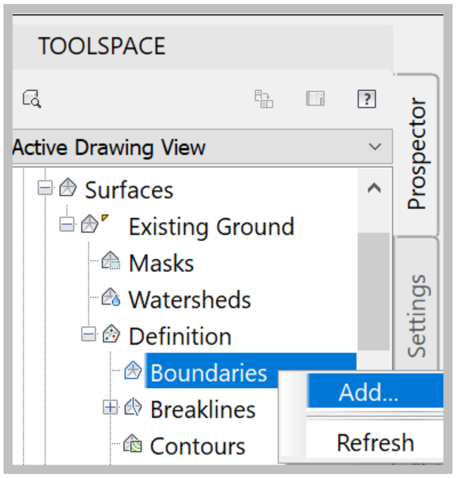
NOTES

### 5.3.3 Defining a surface Boundary

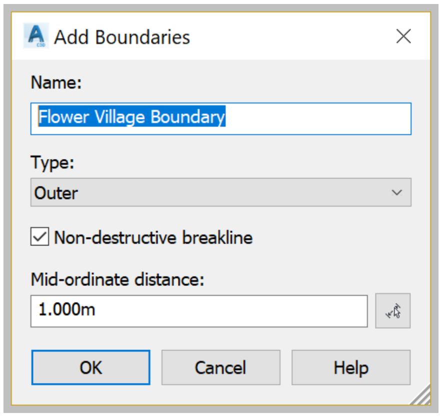
It is often essential to limit a surface within a defined boundary. This allows you to focus on the area of interest. It also helps allocate more computer processing power to the actual required work. Indeed, boundaries help limit Civil 3D's calculations to a smaller surface area. This is especially important as Civil 3D is still a graphics intensive software.

To add a boundary to the surface:

1. Expand the **Existing Ground** Surface definition.
2. Right-click on **Boundaries** in the definition.
3. Select **Add.**

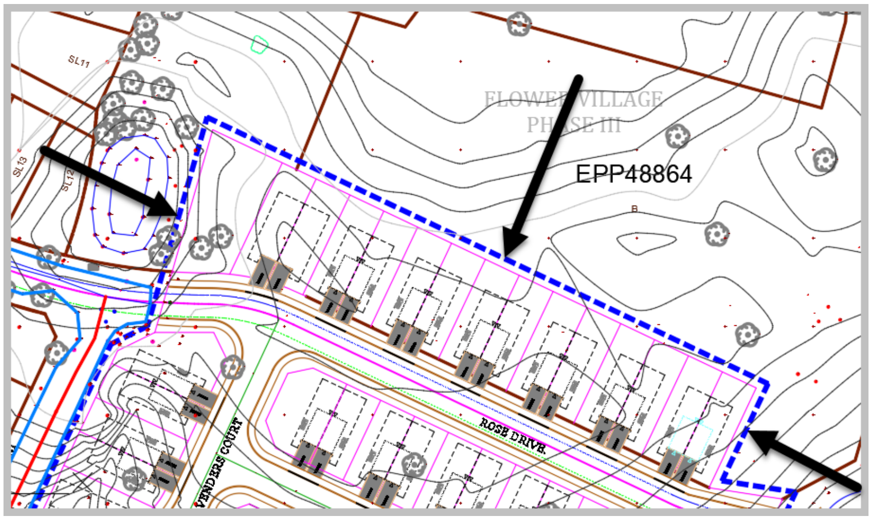



4. Call the boundary **Flower Village Boundary** for example.

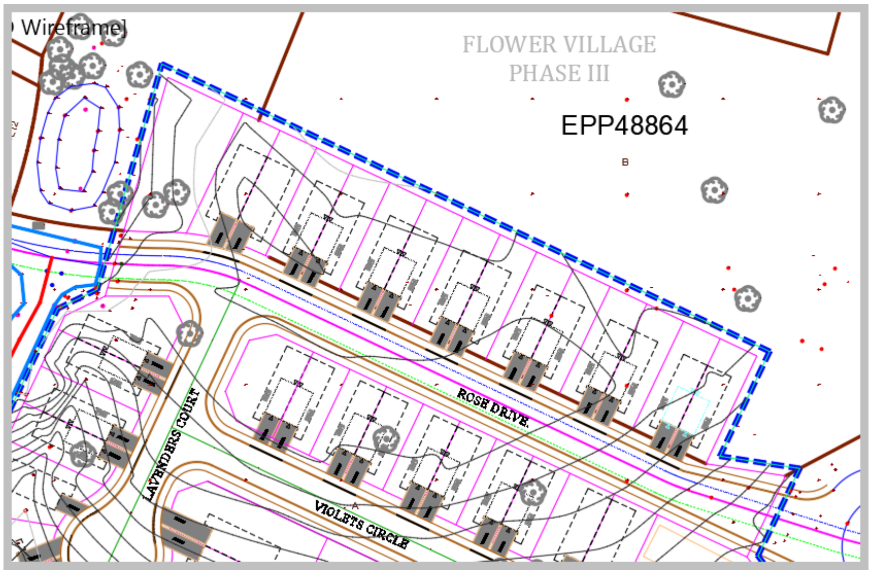



5. Click on **OK.**

6. Click on the dashed **blue line** representing the site boundary.




7. Notice how the surface is now constrained to the specified area.




NOTES

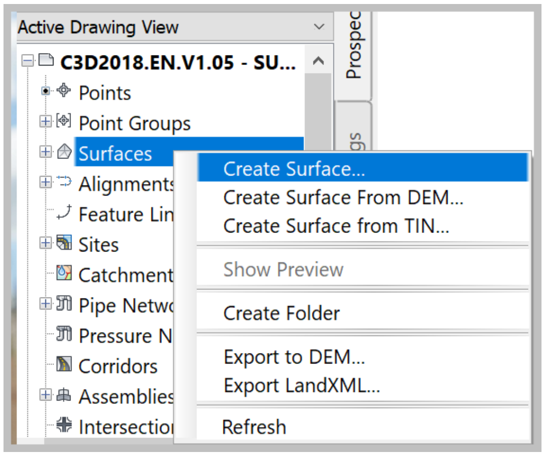
### 5.3.4 Defining a surface by Edits

We can also define a surface by editing existing data and adding them to the new surface definition, as **Edit** operations on the surface. Among those **Edit** operations are **Paste, Raise/Lower, add** or **delete points**, **add** or **delete lines**, and more.

Let's see how to use a couple of these edit operations. We will create an entirely new surface, a stripped surface.

To create a stripped surface:

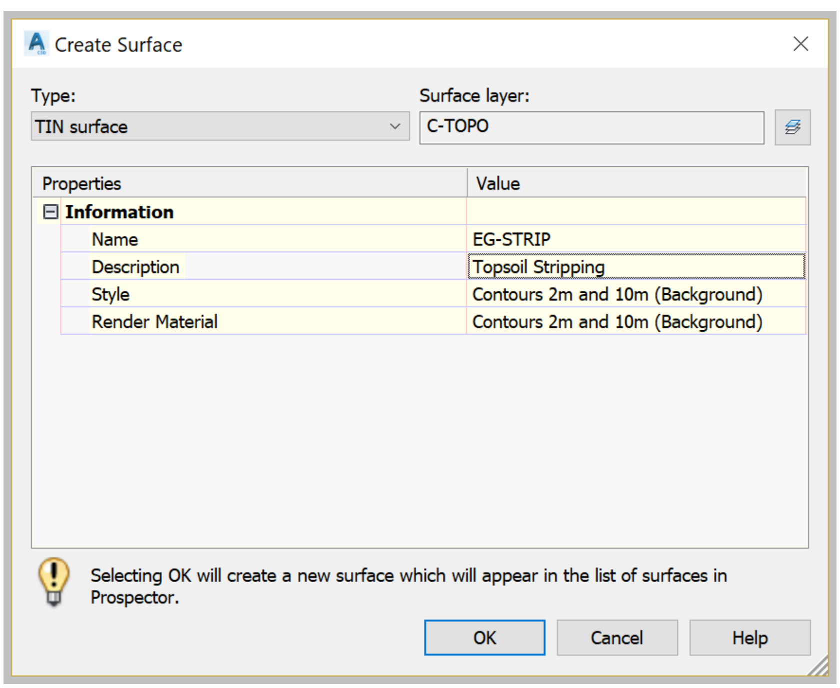
1. From the **Ribbon** or the **Prospector**, create a new surface:




2. In the **Create Surface** window, name the new surface **EG-STRIP.**

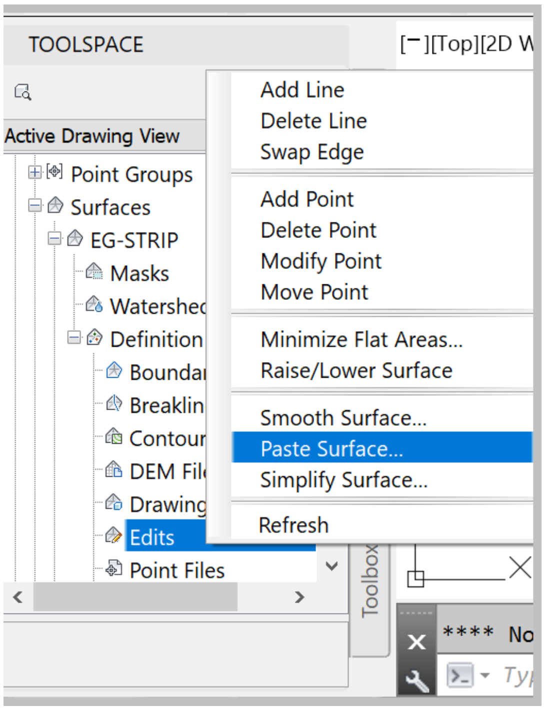
3. Provide an explicit description such as **Topsoil Stripping**

4. Keep the default style and Render Material




NOTES

5. Click on **OK.**
6. All we have done so far is create a name and style for the stripping surface. We still need to define it. To do that, first, we need to **Paste** the **Existing Ground** surface. Remember, the stripping surface is basically the **Existing Surface** that is shaved a given depth to remove the topsoil.
7. Now, expand the **EG-STRIP Definition**
8. Then, right-Click on the **Edits** section
9. And, select **Paste Surface**




10. When prompted, select the **Existing Ground** surface and click **OK.**

11. The EG-Strip surface is now defined. However, at this stage, the existing and stripped surfaces are basically the same. That is because we have only copied and pasted the existing surface into the stripped surface.

12. The next step is to shave the topsoil, say a depth of **15cm or 6in.** To do that, from the surface definition, **Edits** section, right-click and select **Raise/Lower.**

13. For elevation type **-0.15m (-0.5ft)** on the command line. In the Raise/Lower command, a negative value means lowering, and a positive value means raising the surface.

14. The strip surface is now defined and fully functional. We will later see how to use it.

NOTES

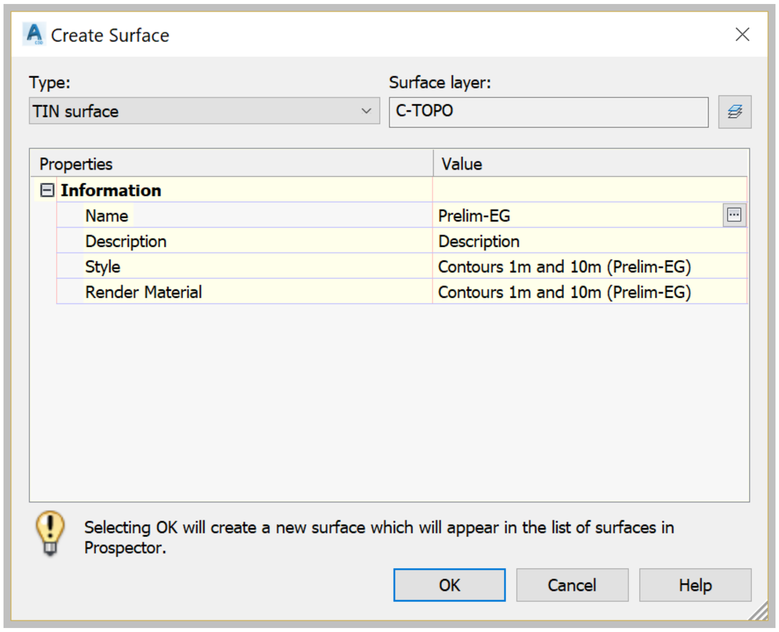
## 5.4 Defining a surface from Contours

Another option to define a surface is by contour data. Contours are available from government repositories, design software packages, photogrammetry, and other sources. If they are not created from a reliable field surveyed source, the data must be utilized with caution. Typically, using such data is a cost-effective way to perform a preliminary design or analysis. But, the final design should always rely on verifiable field-collected data, by a licensed surveyor.

Let's assume that we obtained the contour data before the topographical survey. We are going to create a preliminary existing ground surface, just to learn how to create a surface from contours.

1. Using the previous steps, create a surface and name it **Prelim-EG.** This will represent the existing surface created from preliminary contour data.

2. At the **Create Surface** step, assign the **Name**, **Style** and **Render Material**.



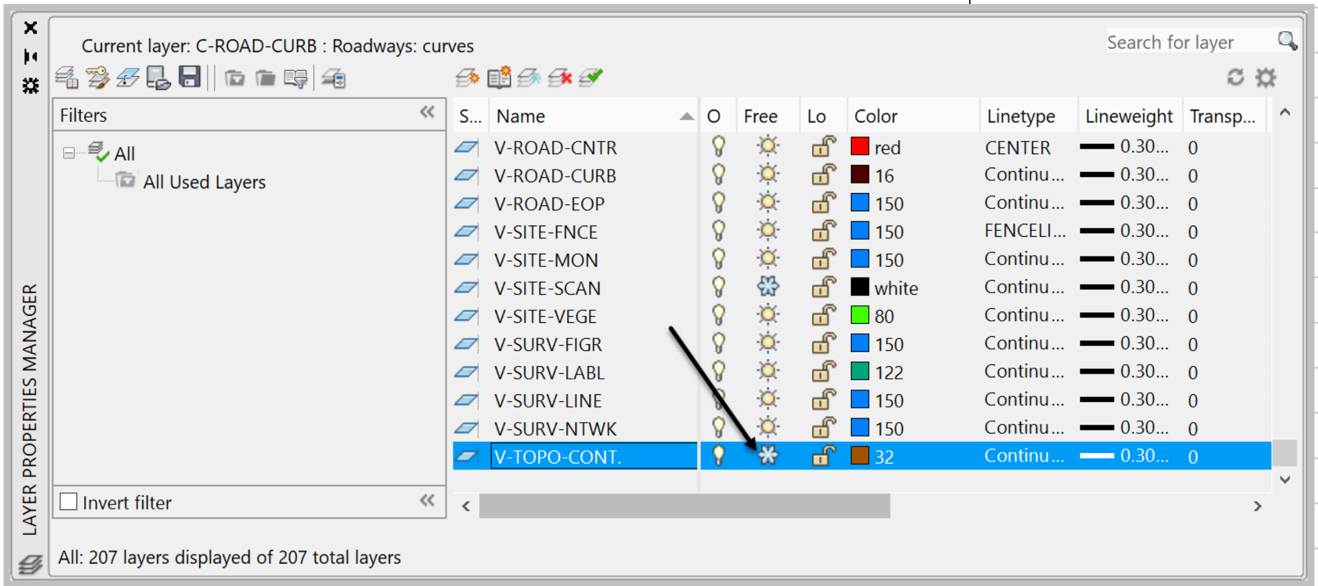

3. Click **OK**.

4. Now let's turn on the contour layer to bring in the contours data.

5. Bring up the layer management window by clicking on the layer manager button on the ribbon or by typing layer on the command line.

NOTES

6. Scroll down to the **V-TOPO-CONT** layer and turn it on.




7. Then, close the **Layer Properties Manager** by clicking the **X** sign , at the top left. The contour information is now displayed.

8. Next, select one of the contours.

9. **Right-Click** anywhere in the **Drawing Area** and click **Select Similar.**

10. All contours are now selected. The **Select Similar** command is a very useful tool to select objects sharing the same properties (type, layers, colours, and so forth)

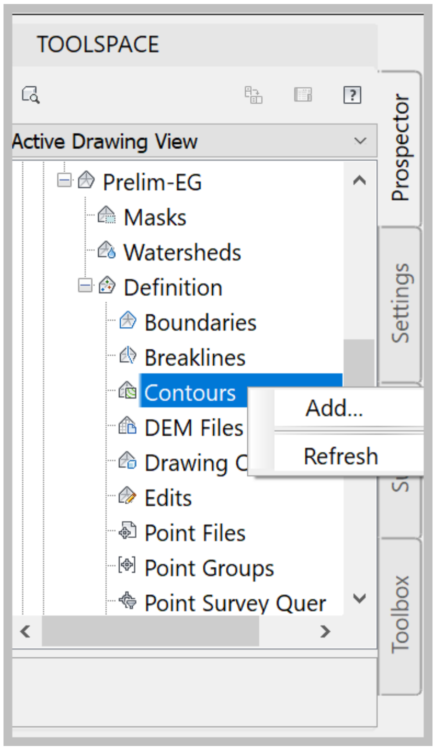
11. Now that we have selected the contours, what we want to do is to use them to define a surface.

12. To do that, go to the **Prospector**, scroll in the window until you find the **Prelim-EG** Surface.

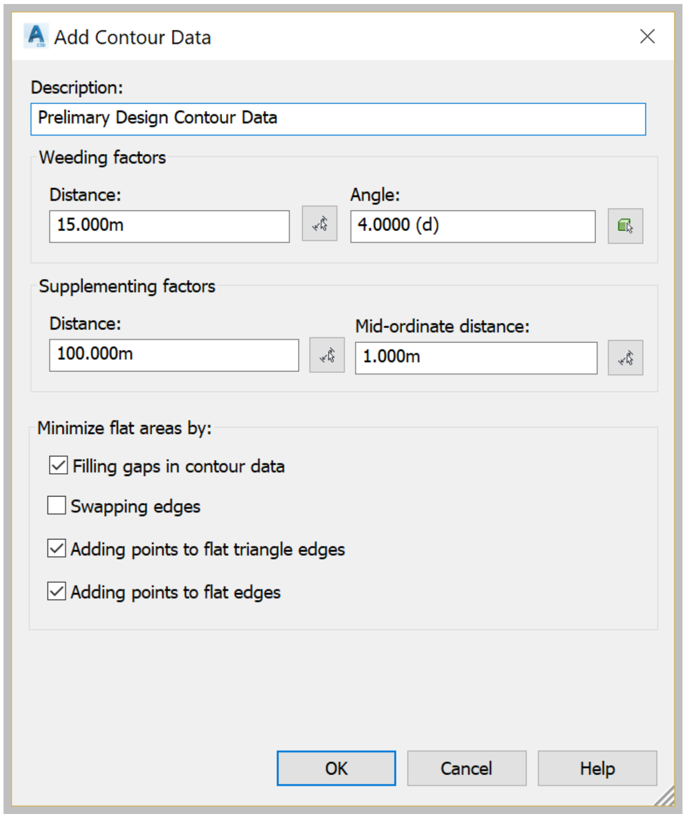
13. Open the surface definition tree and Click on the line that says **Contours.**

NOTES

14. Right-click and select **Add.**



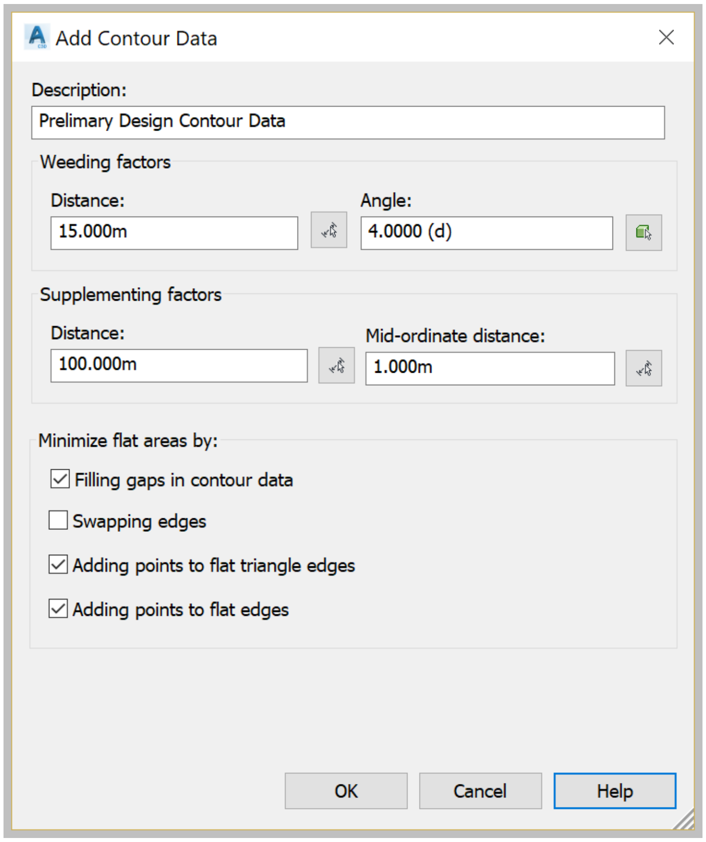

15. Fill out the next window with meaningful information in the Description box.




NOTES

NOTES

16. When we create a surface, it's not always going to be perfect. Maybe we don't have enough data, or the quality is simply not good enough. Thus, inaccuracies exist. Civil 3D represents these inaccuracies with flat areas. We can fix them by adding more data. For the time being, let's accept the default option in the **Minimize Flat Areas By** section and click on **OK.**




17. Next, go back to the **Layer Properties Manager** windows and freeze the **V-TOPO-CONT** layer, since we don't need it anymore.

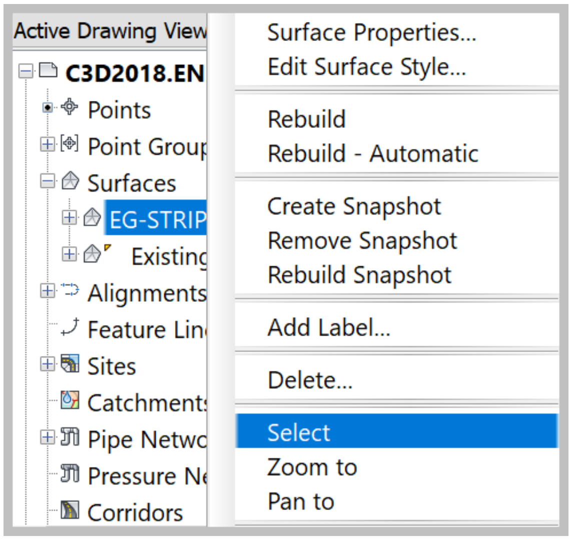
18. The newly created **Prelim-EG** surface appears on the screen and is very close to the surveyed topo contours.

19. Once the surface is created, we don't need to show the contour lines anymore. Therefore, we need to go back to the layer manager and turn off the **V-TOPO-CONT** layer.

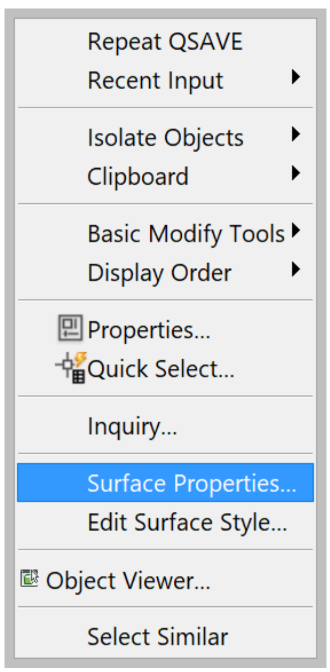
## 5.5 Surface Properties

Once a surface is created, its properties can be viewed or edited. To access the properties of the surface,

1. Select it in the drawing or from the prospector




2. Right-click in an empty area and select **Surface Properties**.



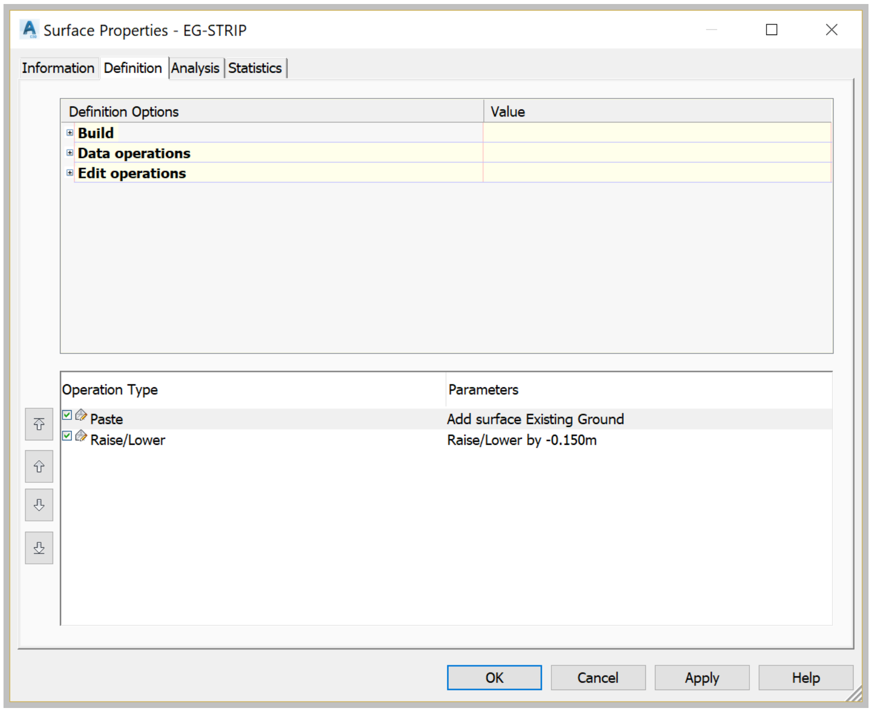

3. The **Surface Properties** window opens and displays four tabs.

   - The **Information** tab is where we set the name, description, and the default styles, meaning **Surface Style** and the **Render Material**. On this tab, we can also lock the surface to prevent any further editing and, show or hide the display of tooltips.
   - The second tab is **Definition**: it allows you to specify definition parameters of the surface. For instance, it is

NOTES

possible to determine a minimum or maximum altitude allowed for the construction of the surface. For example, you can predefine a maximum allowed elevation of **400m** or **1300ft** and a minimum elevation of **300m** or **1000ft.** We know that the elevations for the current site are around **350m** (or **1200ft**). Thus, we immediately eliminate certain types of errors in the survey data. With a minimum and maximum elevation set, we don't have to worry when the surveyor makes a typo and enters **3500m** or **11500**ft instead of **350m** or **1150ft**. We can also set more definition options, such as the maximum triangle lengths, crossing breaklines, and much more.

The **Operation Type** section of this window gives you a complete list of actions performed to define the surface. You can change their order by selecting an operation. For example, adding a **Point Group**, and clicking the **Ascending** and **Descending** arrows on the left. The chronological order is from top to bottom. It is also important to note that in Civil 3D surface definition, the most recent operation, **Raise/Lower** in this case, takes precedence. Let's see what that means.

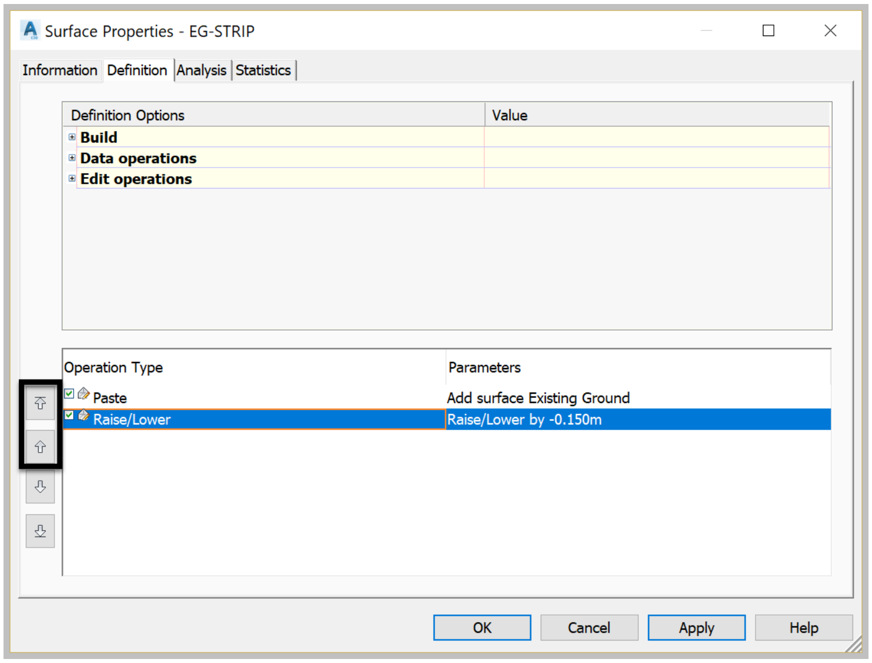



4. Click on the Statistics tab, last to the right, and take note of the current surface information. Let's write down the value of the **Mean Elevation.**

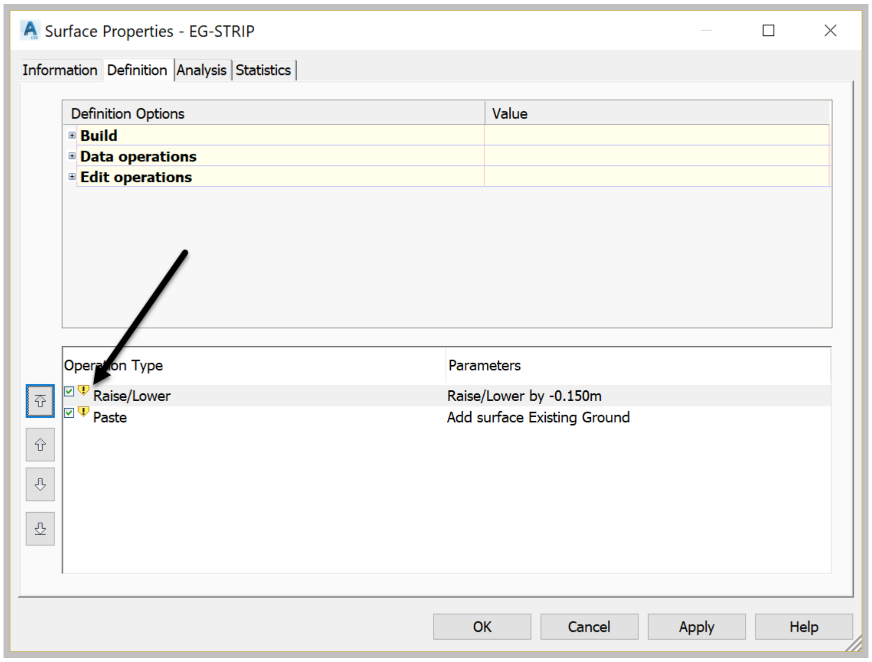
5. Now, go back to the **Definition** tab, second from the left.

6. Select the last operation, **Raise/Lower**, and move it up using any of the **Up** arrows to the left.

NOTES




7. The **raise/lower** operation is now at the top. You get a notification, the two yellow exclamation marks, warning you of changes to the surface. That's exactly what we are trying to do, change the surface.




8. By changing the order of the operations, in practical terms, we have first lowered the existing ground by the depth of the topsoil (**15cm** or **6 inches**). This is because the raise/lower operation is now at the top, with less priority. We then, pasted the existing ground on top of it. Which brings us to square one, the existing ground.

9. Since Civil 3D is warning us that a surface definition has changed, we need to acknowledge that. Click on **OK** to close the

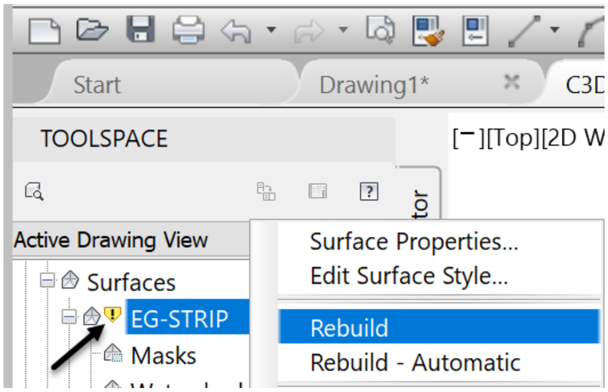
NOTES

NOTES

**Surface Properties** window. We will be back right here once we let Civil 3D know what to do with the changes.

10. Now let's go and check the **Statistics** tab and verify that's indeed what happened.

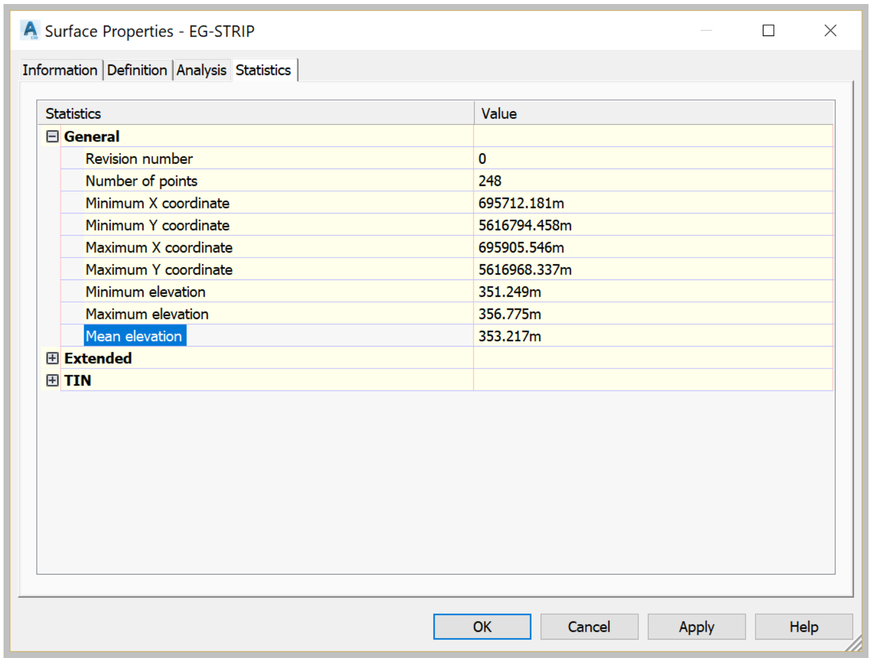
11. In the Prospector, right-click on the surface with the yellow warning sign. We have two options. The first one is to **Rebuild**. With this option, we will rebuild the surface and continue to get warnings every time we make changes to the surface definition. This is good, as it helps us be aware of potential accidental changes to the surface. The drawback is that intentional changes will not automatically propagate through the design. Thus, other entities , such as profiles, manhole elevations, pipe rules, and earthworks volumes are not updated. Despite that these entities depend on the surface. We would need to manually rebuild the surface to make them aware of the new changes.




Our second choice is to **Rebuild – Automatic.** This simply tells Civil 3D to go ahead and automatically update the surface and not bother us with warnings while we are busy doing other things. We may save time with this option. However, there is a potential for unintentional changes to the surface to slip through the cracks. They will not be detected because we have decided not to be warned of changes. Our recommendation is always to update manually. The benefit of avoiding costly mistakes could justify the lost time caused by manual updates. So, select the manual rebuilt mode and let's go back to the **Surface Properties** window and check if, indeed, the new value for the mean elevation is changed. We have predicted that the mean value would be changed to that of the original surface.

NOTES

12. Open the properties windows with a right-click on the surface, then go the **Statistics** tab.

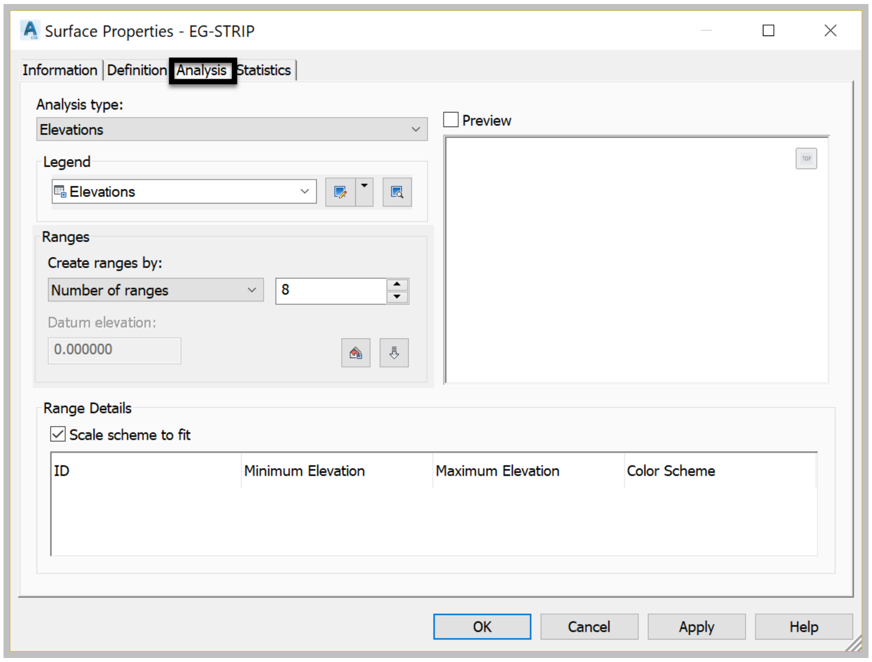



13. As it turns out, the new value from the **Mean Elevation** is now **15cm** or **6in** higher than the stripped surface. That tells us that we are in fact, back to square one (existing ground elevations). In summary, it's important to understand that as far as creating surface is concerned, the last operation always wins. As shown by the stripped surface example, we must first copy the existing ground surface then lower it. Not the other way around.

NOTES

## 5.5.1 Analysis Tab

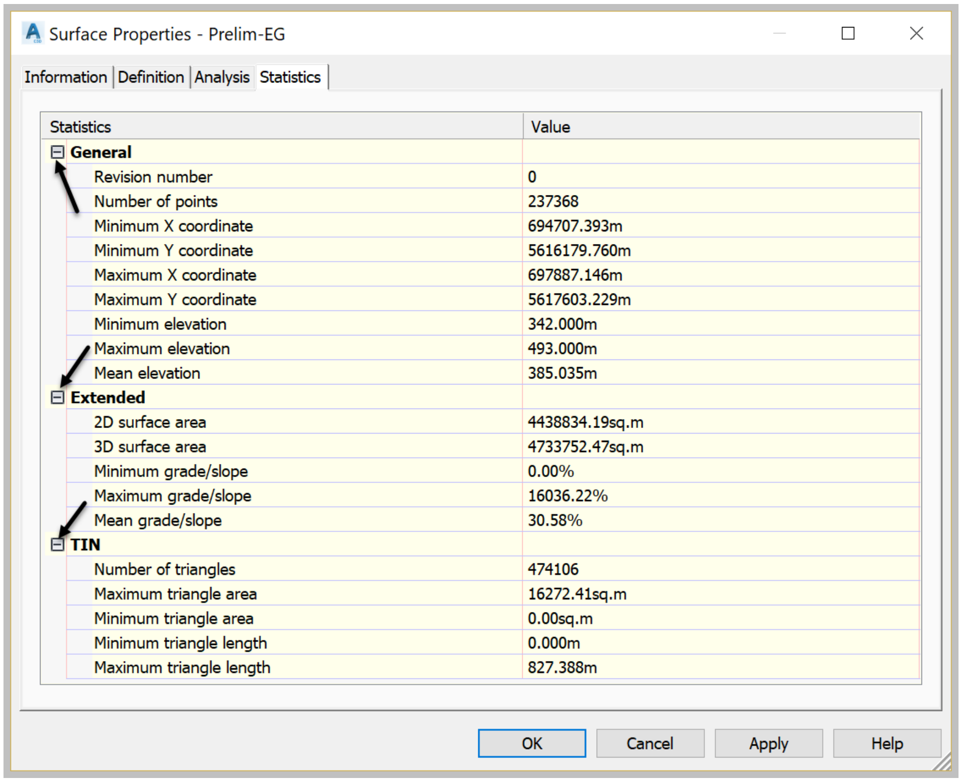
1. Now let's keep exploring the **Surface Properties** window. If you have already closed it, reopen it and activate the **Analysis** tab, the 3rd tab from left to right.




2. This tab allows you to perform a visual analysis on the surface. For example, you can carry out an analysis for **Contours, Directions, Slopes, Elevations, and Watershed**. We will see how to do these analyses in a minute.

### 5.5.2 **Statistics Tab**

Finally, as we have seen before, the **Statistics** tab allows us to have basic statistics on the surface. We can determine things such as the number of points, maximum, mean and minimum elevations, the number of triangles, and areas. It's easy to check this window for a quick reference and detect possible errors in the creation of a surface, especially regarding minimum and maximum elevations. To expand each section, click on the plus sign to the left of the section's title.




| Statistics | Value |
|---|---|
| **General** | |
| Revision number | 0 |
| Number of points | 237368 |
| Minimum X coordinate | 694707.393m |
| Minimum Y coordinate | 5616179.760m |
| Maximum X coordinate | 697887.146m |
| Maximum Y coordinate | 5617603.229m |
| Minimum elevation | 342.000m |
| Maximum elevation | 493.000m |
| Mean elevation | 385.035m |
| **Extended** | |
| 2D surface area | 4438834.19sq.m |
| 3D surface area | 4733752.47sq.m |
| Minimum grade/slope | 0.00% |
| Maximum grade/slope | 16036.22% |
| Mean grade/slope | 30.58% |
| **TIN** | |
| Number of triangles | 474106 |
| Maximum triangle area | 16272.41sq.m |
| Minimum triangle area | 0.00sq.m |
| Minimum triangle length | 0.000m |
| Maximum triangle length | 827.388m |

NOTES

NOTES

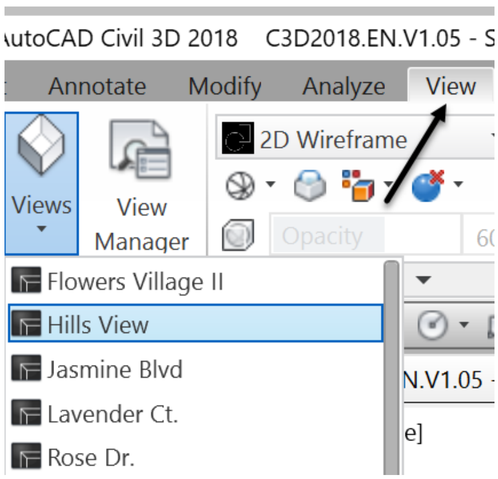
## 5.6 Surface analysis

### 5.6.1 Water Drop Path

The **Water Drop** analysis allows us to analyze runoff and trace the path that a drop of water would take across a surface. This analysis is especially useful for designing surfaces such as parking lots, subdivision streets , and so forth. We can easily identify low spots on a surface, and install necessary stormwater catch basins in these low spots.

To use the **Water Drop** tool,

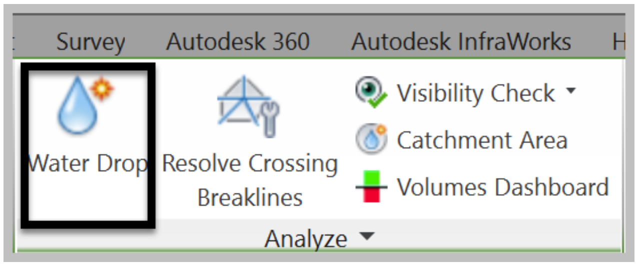
1. Zoom to the **Hills View** named view or manually zoom out using the mouse wheel.



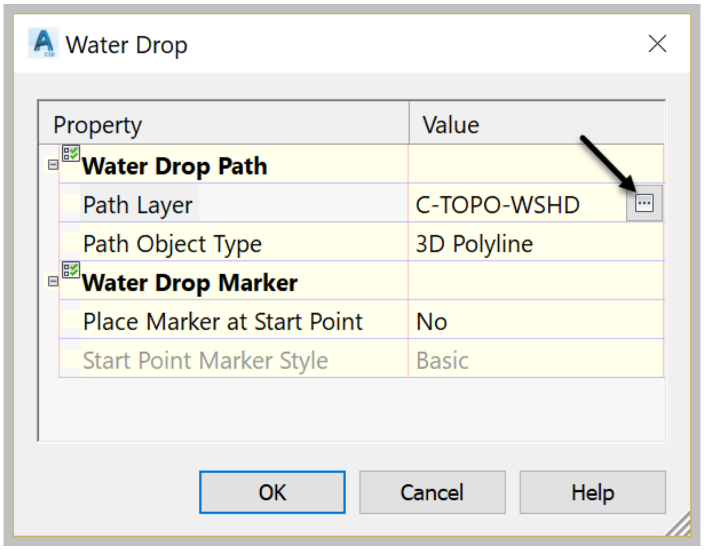

2. Select the **Prelim-EG** surface in the drawing.

NOTES

3. On the ribbon, click the **Water Drop** command for analysis.

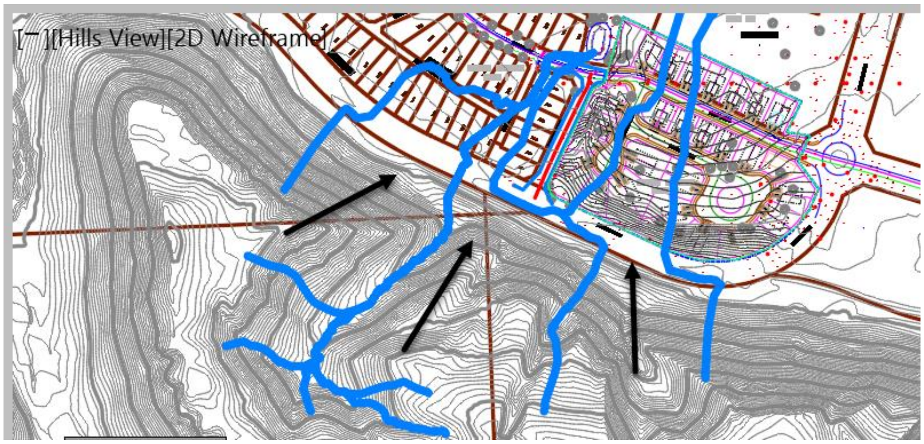



4. In the **Water Drop** window, change the layer to the watershed layer **C-TOPO-WSHD** and choose **3D Polyline** for **Path Object Type.**




5. Click **OK.**

6. In the drawing, click on a few points on the surface and observe the flow direction. We notice that as expected, water is running downhill from the mountainous areas to the lower areas and will ultimately flow down to the river.




7. Zoom in further, and you will notice that most of the runoff from our proposed development is going through existing **Pond 2.**

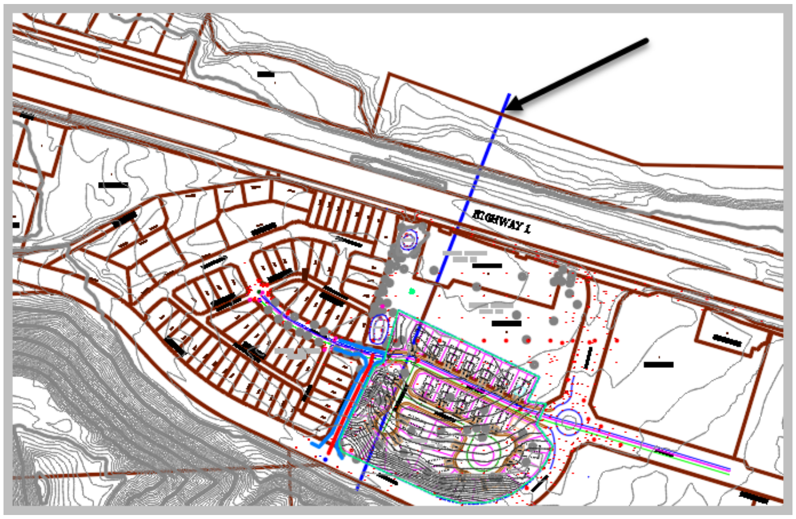
Meanwhile, the flows from **Flower Village Phase 1** are going through **Pond 1**.

8. Now, delete the Water Drop analysis lines (blue lines) by selecting one of them. Then, right-click, **Select Similar** and press delete on the keyboard. Or, you can use the layer isolation method to isolate the **C-TOPO-WSHD** layer and delete them. We will use the water drop method more when we design subdivision streets.

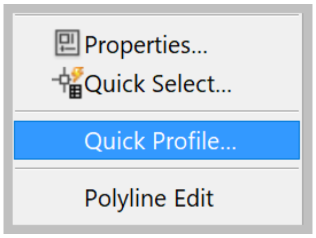
### 5.6.2 Analysis by Quick Profiles

Another method of quickly analyzing a surface is by the **Quick Profile Method**.

1. Now, draw a polyline that crosses our proposed site, from south of **Jasmine Blvd** all the way to the river.

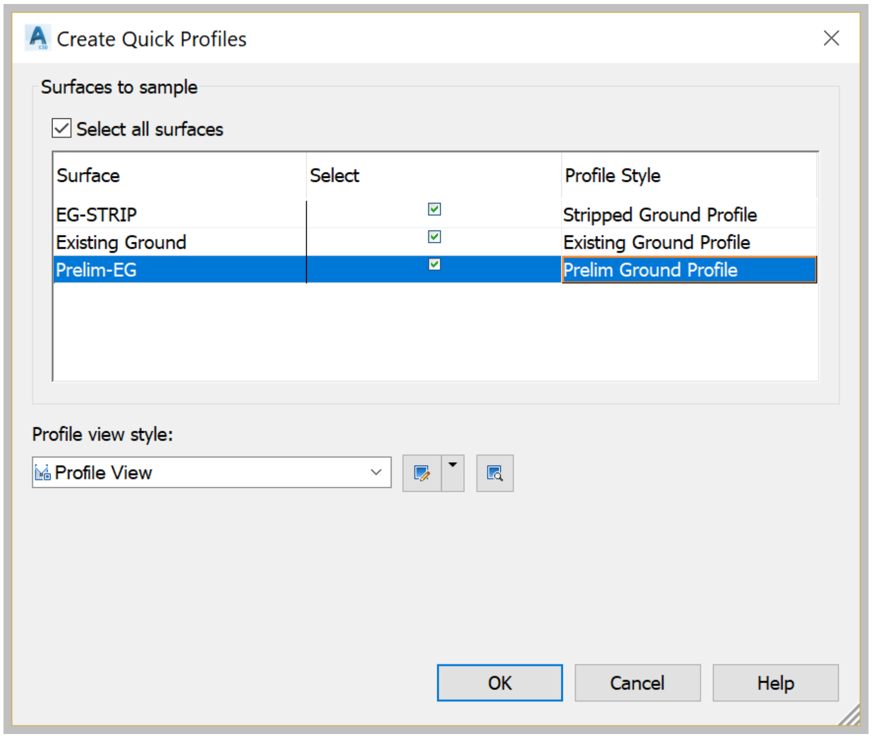



2. Select the polyline, then Right-click and choose **Quick Profile.**

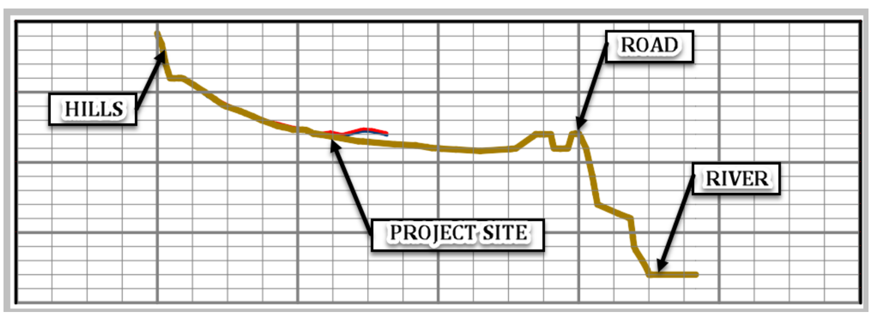



NOTES

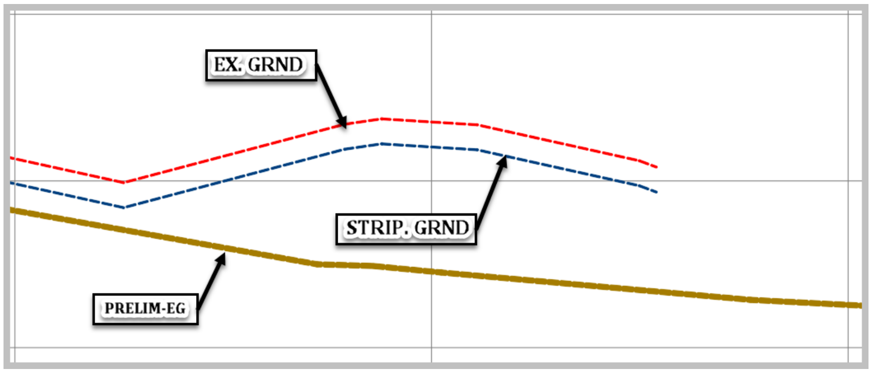
3. In the **Create Quick Profiles** window, pick the adequate styles for the surface **EG-STRIP**, the **Existing Ground** and **Prelim-EG** surface.




4. Next, click **OK** and click on an empty spot on the screen to select the origin of the profile. In the profile view, you should be able to identify the major features of the site, such as the hill, major roads, and the river.




5. If you zoom in closer, you will notice the different surfaces, including the surveyed existing ground, the stripped surface, and the preliminary contours surface.

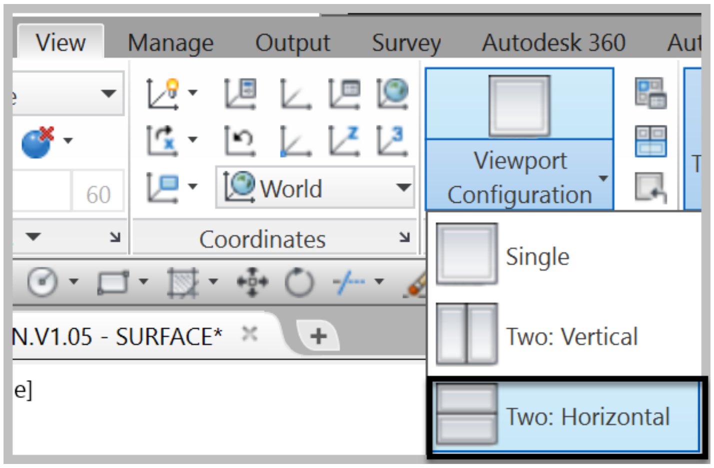



NOTES

6. Furthermore, you can observe the profiles in two horizontal view windows. Then, interactively analyze different areas of the site. First, split the **Drawing Area** into **two horizontal** views. To do that,

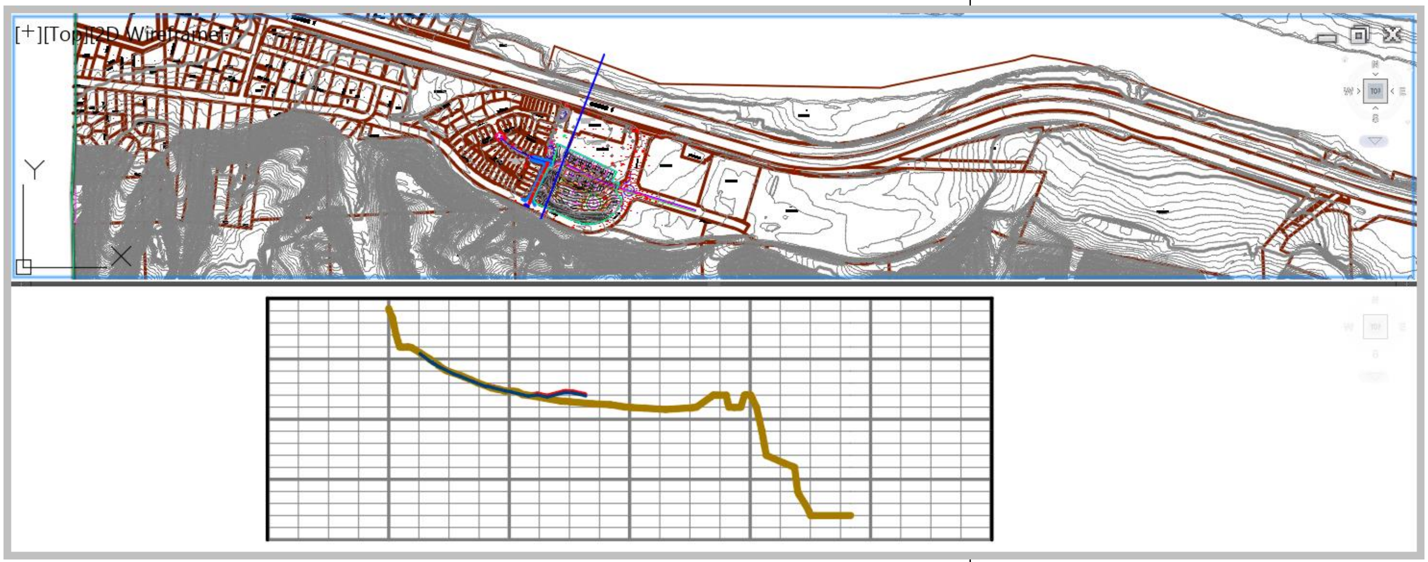
7. Click on the **View** tab.

8. Then in the **View Configuration** drop-down menu, select **Two Horizontal** views.




9. Now in the drawing area, in the top window, zoom in to see the polyline that you drew to cross the site.

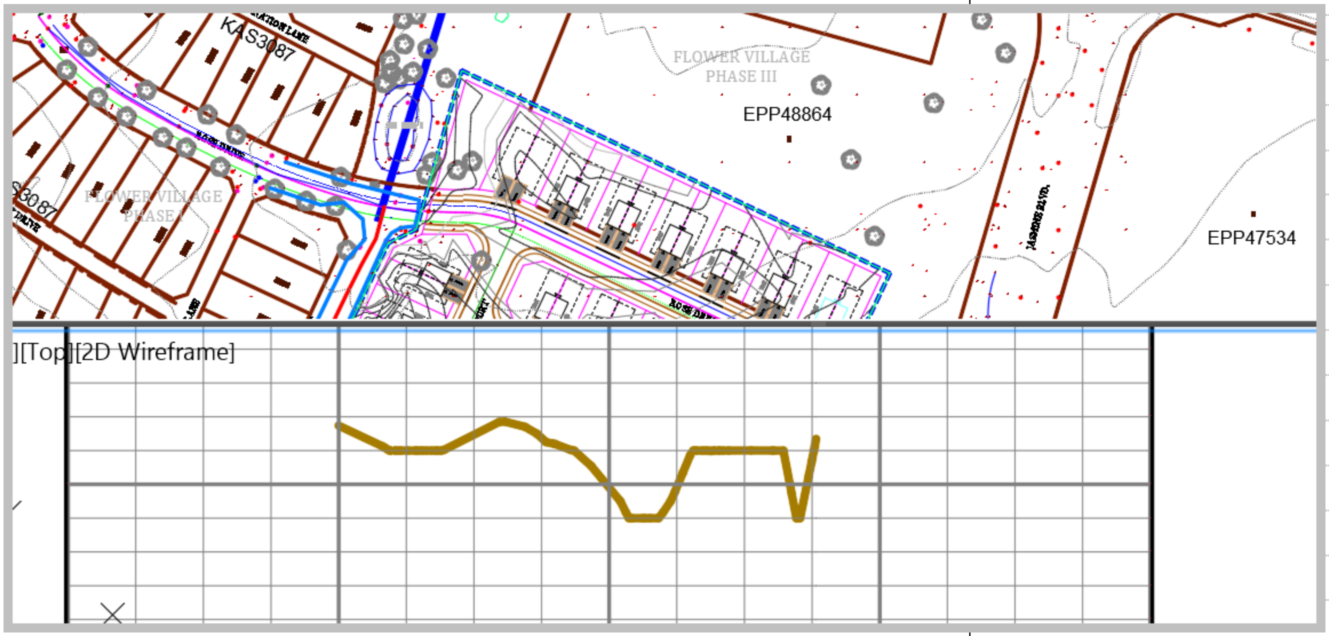
10. In the bottom window, zoom and center the view to fit the profile view just like this.



11. When we start the design process, we are going to make changes to the existing ponds, since we are adding more impervious to the site. So, we probably need bigger ponds to retain all that extra runoff. To help the decision making, we probably need to know what is going on in the area of the ponds.

NOTES

12. Select the polyline and move it to cross the two ponds. Something like this.




13. Notice how the profile automatically adjusts to reflect the area crossed by the polyline.

14. From here, we can observe that **Pond 1** is higher than **Pond 2**, and that the ultimate project Outfall will be in **Pond 2**, right to the south of **Highway 1**.

15. Now, select the quick profile polyline and hit delete. Notice how the profile disappears? That's because this profile is dependent on the polyline. One more thing to mention; if you ever saved the drawing, the profile will also disappear, as it's a temporary analysis tool. That's why it's called a **Quick Profile**. It cannot be saved and used later. To do that, we will need a more powerful profile. We will see later how to create them. For now, make sure the polyline is deleted and that you return to single view. To do that, select **Single** in the **View Configuration** drop-down menu, from the **View** tab.

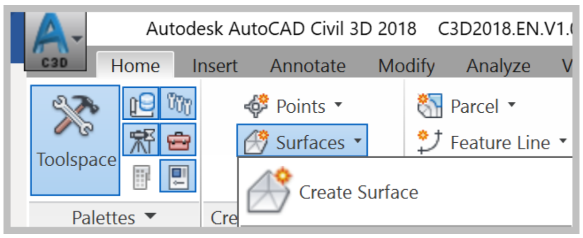
NOTES

## 5.7 Volume Surfaces (Cut and Fill)

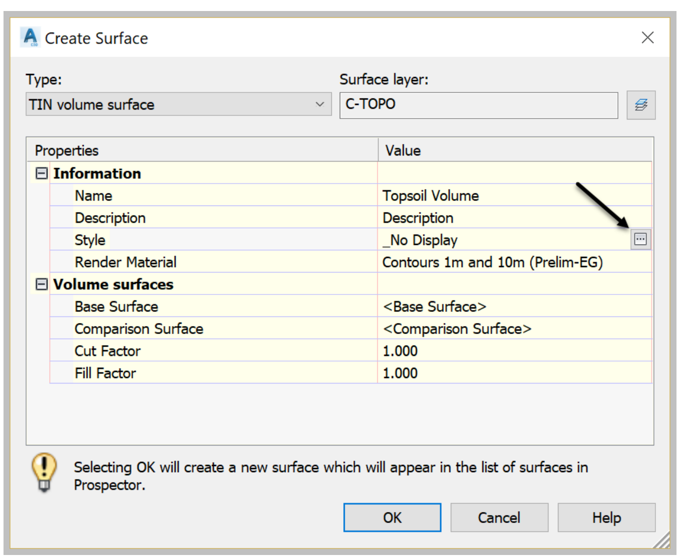
So far, we have learned how to create a **TIN Surface** or a "regular" surface. But, there's a second type of surface, a **Volume Surface**. You guessed it, the main purpose of this type of surface is for estimating volumes or earthworks quantities. Volumes are the amount of material, **Cut** and **Fill,** that you will need to transform a surface from an initial to a final state. For example, removing topsoil, altering a site from **Existing Ground** to **Final Ground**.

Let's see that in practice by estimating the total volume of topsoil that needs to be removed from the site.

1. From the **Home** tab, launch the **Create Surface** command.



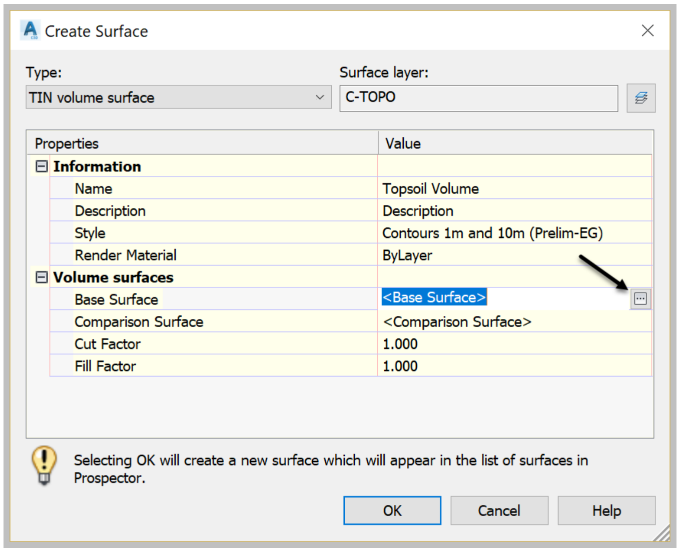

2. In the **Create Surface** window, for **Type**, choose **TIN Volume Surface**, call the surface **Topsoil Volume** and assign a **_No Display** Style. A No-Display style is basically a style that we assign to entities we don't need to show in Civil 3D. In AutoCAD, we typically turn off or freeze the layer of entities we don't need to show. In Civil 3D, we do the same thing by assigning the **No Display** style to specific objects, without needing to manipulate layers back and forth. Since we only need the **Volume Surface** to estimate earthworks information, there is no need to display it. Sometimes, we are required to create a Cut and Fill areas map. In those cases, we need to assign an elevation style to show the needed information. In this current situation, we don't need to. So, we are using a **No Display** style.



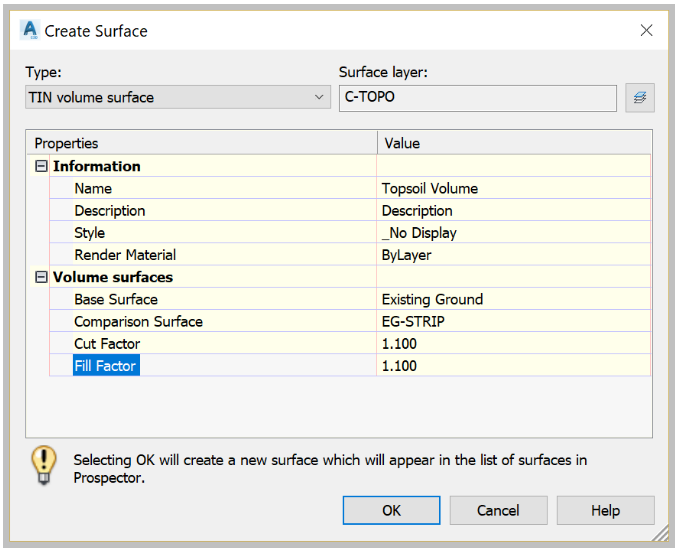

NOTES

NOTES

3. Next, define the **base surface**, the existing ground, by clicking on the three dots to the right




4. After that, define the **comparison surface**, the Stripped surface, by clicking on the three dots on the same line.



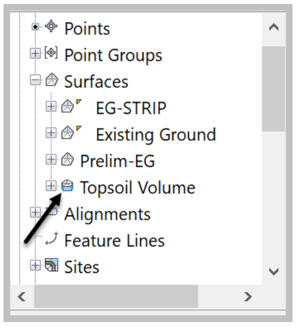

5. For **Cut** and **Fill Factor**, let's use **1.10**. These factors control, depending on the soil type, how much the material swells after removal, or how much it is compacted when put in place. This means that for every cubic meter or feet of cut material we will need to haul **1.1** cubic meter or feet. Obviously, it would be the inverse for the fill material. To fill every cubic unit in place, we will need to haul in 1.1 cubic unit. A basic rule of thumb for this is that you are always transporting more volume than what's in the ground. Either when you remove or when you are putting in place.

6. Now, click on **OK** to close the window.

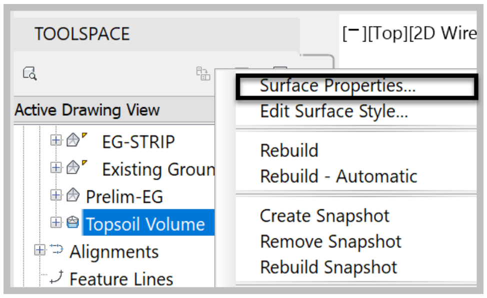
7. Nothing shows up in the **Drawing Area** because we have assigned a **No Display** style to the volume surface.

NOTES

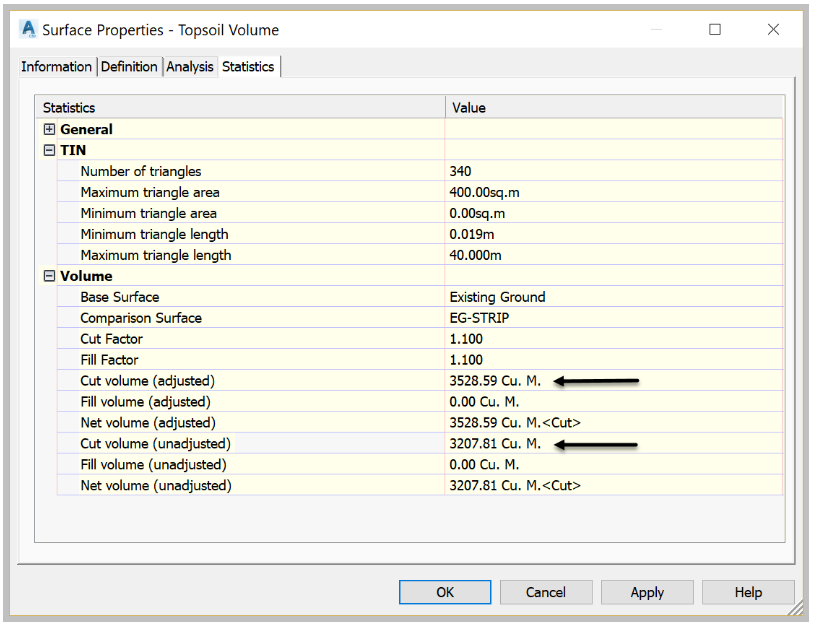
8. However, the **Topsoil Volume** surface still appears in the prospector. It has a different symbol than the other surfaces because it represents a **Volume surface**, while the others are “regular" TIN Surfaces. **TIN** stands for **Triangulated Irregular Network.** It is the most common method of interpolating elevations when we are creating a surface.




9. The last step consists of checking the volumes. This is done from the surface's statistics. In the **Prospector**, select the surface, right-click and go to **Surface Properties**.




10. Then, on the **Statistics** tab, expand the **Volume** tree. Notice the adjusted and unadjusted Cut Volumes. As expected, the adjusted volume is 10% more. And of course, the volume of Fill is zero because we are estimating a topsoil volume, which consists only of cut material.



In summary, this is a quick method for estimating volumes, especially when we have two surfaces we want to compare.

NOTES

NOTES

## 5.8 Surface Styles and Labels

Surface label and styles control the appearance of surfaces and their annotations.

### 5.8.1 **Surface Styles:**

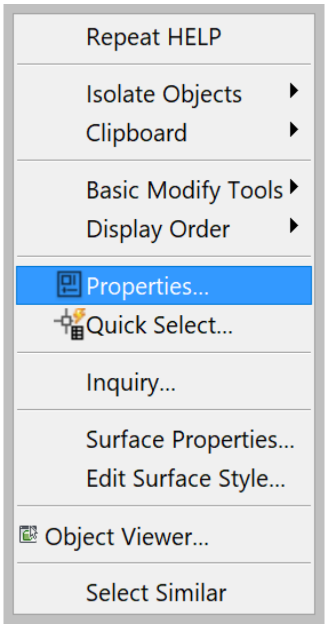
Like most entities in Civil 3D, applying a surface style controls the display of the surface. Using a style, we can manage the display of things such as contours and their intervals, triangles, points, slope arrows, watersheds, etc.

Let's see how to apply styles to the surfaces in the drawing. Say we want to display the high and low areas on the preliminary existing ground surface. To do this, we will first need a Style that displays Elevation. We must then decide what range of elevations we will consider high or low ground.

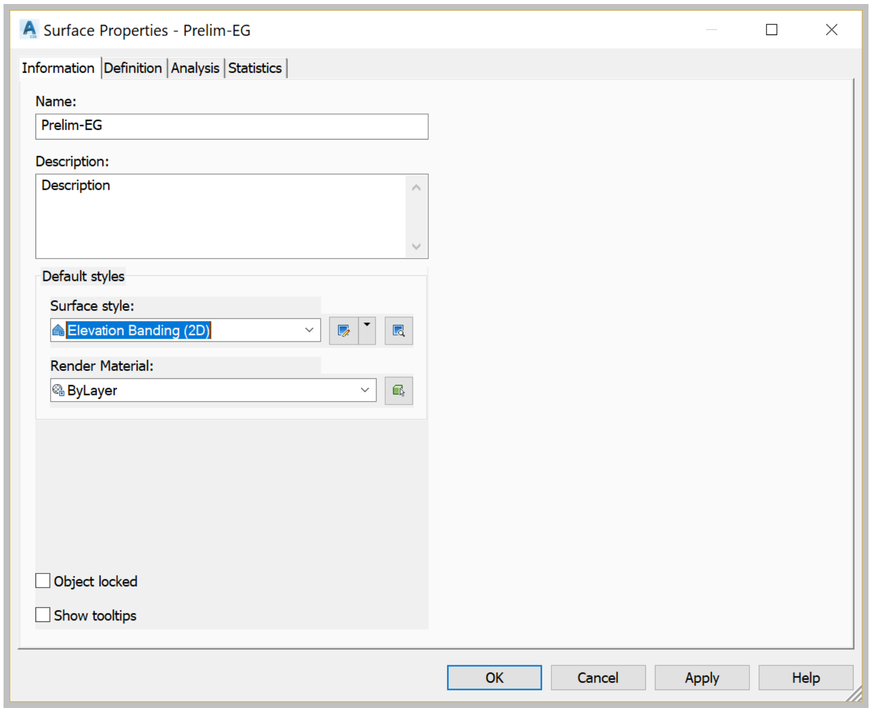
To apply or create a new surface style,

1. Select the working surface, **Prelim-EG** surface, by clicking anywhere on its contours in the drawing, or by selecting it in the Prospector. Right-click then **Select.**

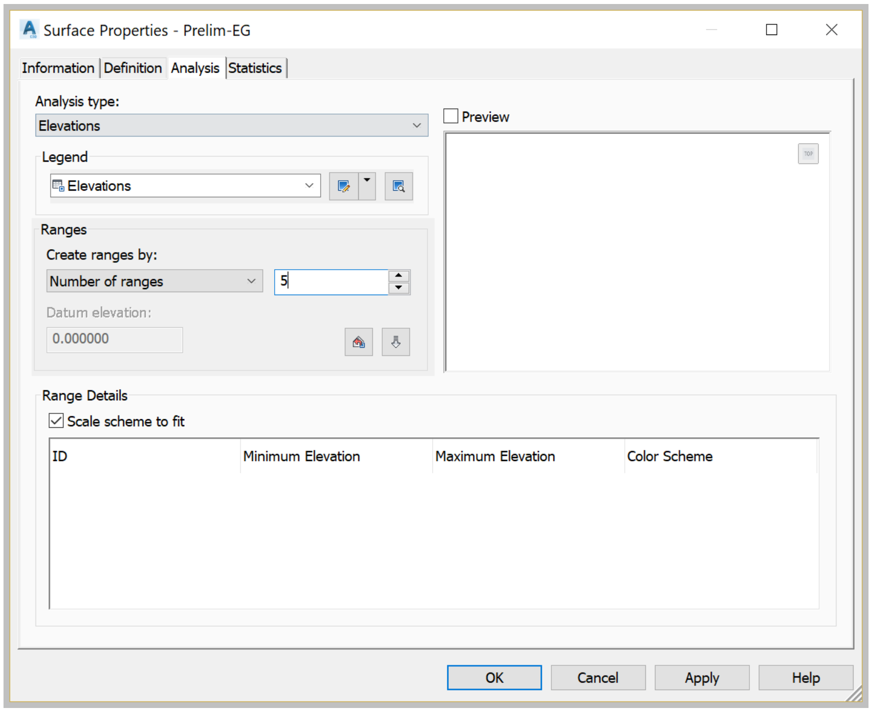
2. Right-click in an empty spot in the drawing area and select **Surface Properties**.




3. In the **Surface Properties** dialog box, on the **Information** tab, for **Surface Style**, select **Elevation Banding (2D)**. This will allow us to display the elevations in hatch (solid or predefined). This will complete the first step in displaying an elevations map.



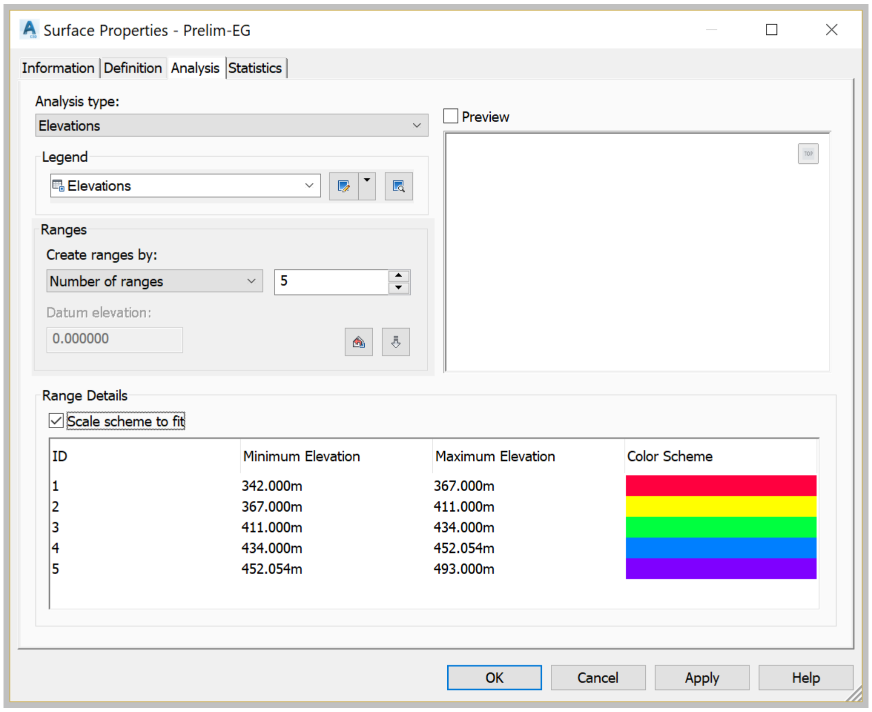

4. Next, we need to specify the range of elevations. For that, click on the **Analysis** tab, and select Elevation for the type of analysis. The Legend should also be elevation. For **Number of ranges**, let's choose **5.** The default number of ranges and schemes are set in the **Analysis** tab of the **Surface Styles** dialog box that we will see later.



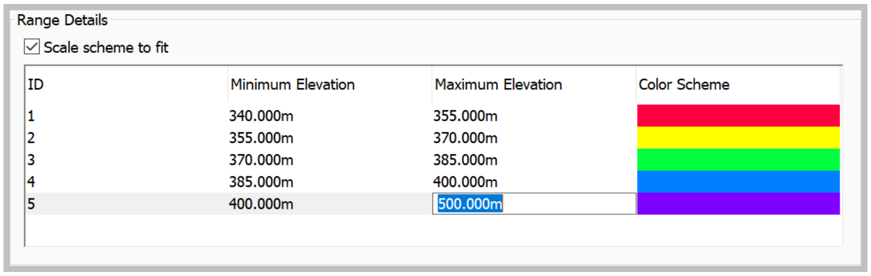

NOTES

NOTES

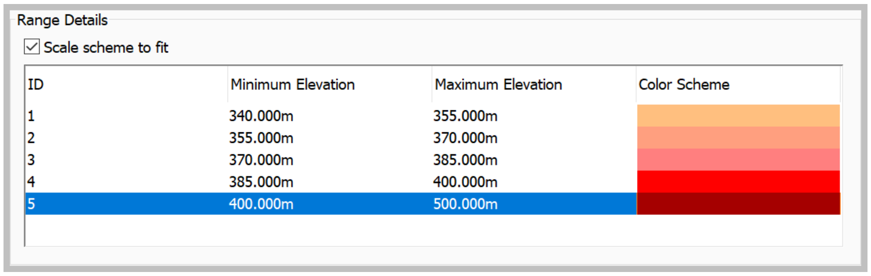
5. Click the downward arrow to display range elevations and schemes.



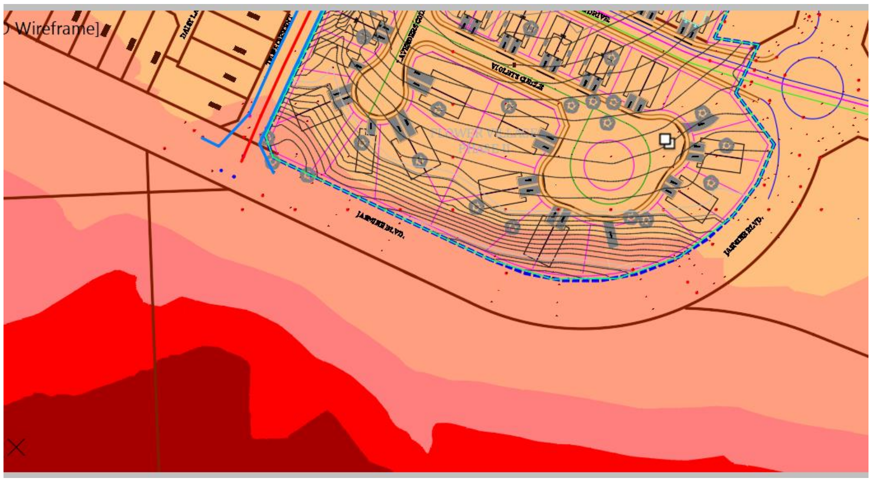

6. Let's change the colour scheme and elevation ranges. We may need to make the last **Minimum Elevation** a little higher, say anything above **400m or 1,300 feet.** We can then go at **15m** or **50ft** intervals for the rest.




7. For the colour, we can change them individually by clicking on them. Let's choose a gradually intense red colour scheme.



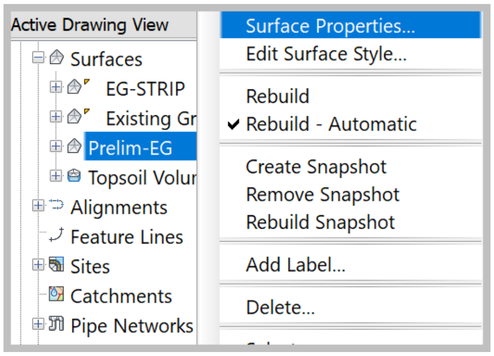

8. Click on **OK** to close the window and observe the changes. The elevations map is displayed in the drawing, and we can differentiate between the high and low ground areas. We can further refine the elevation ranges by increasing or tightening them, to do more analysis.



9. Now change the Style back to the **Prelim-EG** contours style by selecting the surface. Then go to the surface properties and apply the contours style on the information tab.

10. Next, let's see how to **Create** or **Edit Surface Styles**. Assume the project reviewing agency requires that existing contours be displayed at **0.25m** or **1-feet** intervals. To create a style that meets that requirement, we can either modify an existing style or simply create a new one.

11. First, hide the **Prelim-EG**, by applying a **No Display** style, so that we can easily select the surveyed surface. Select the **Prelim-EG** surface, right-click and select **Surface Properties**.




12. Next, apply the **No Display** style and click **OK** to close the window.

13. We also need to repeat the previous steps to hide the **EG-Strip** surface, since we don't need to display it anymore.

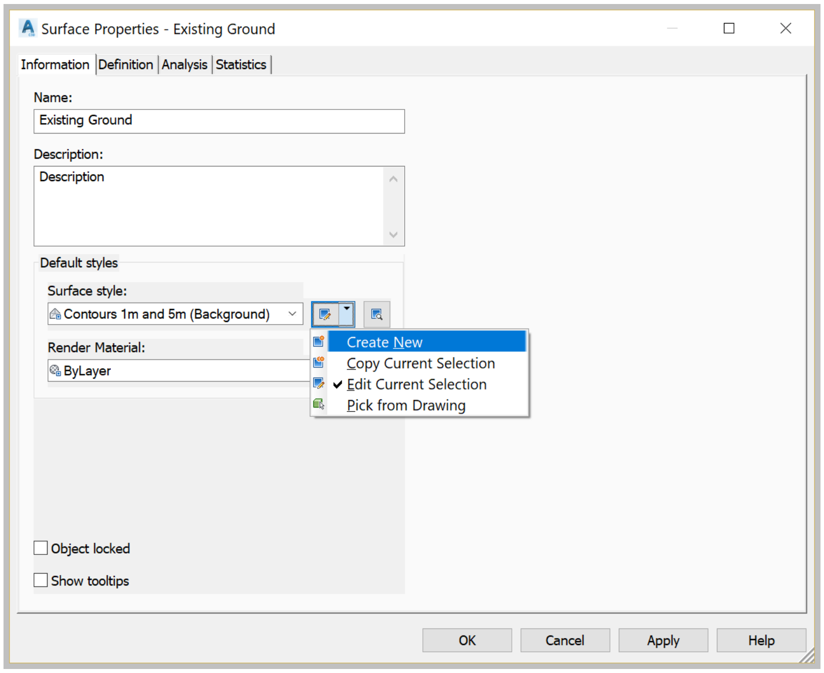
14. Now we need to see how to create a Contours Style for the required **0.25m** or **1-feet** intervals by our reviewing agency. Select the **Existing Ground** surface and open the **Surface Properties** window as we just did for the **Prelim-EG** surface.

15. If we click in the **Surface Style** drop-down box, we will see a list of all styles available in our drawing. We can simply pick one and apply it to the currently selected surface. Unfortunately, we

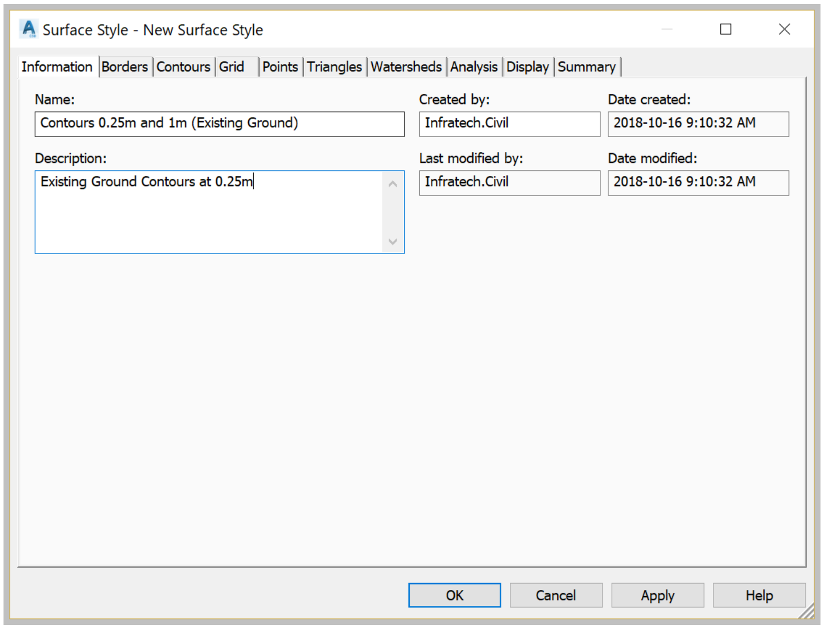
NOTES

don't have the one required by the reviewer, the **0.25m** or **1ft** contour**.** So, let's create it.

16. Click on the downward arrow to the right of **Surface Style** dialog box and click on **Create New**. Note that we could also select **Copy Current Selection** to copy the currently active style and then modify it to meet our requirements. That would be a smart thing to do if we have settings in that style that we want to copy over. This case actually fits our current situation, since we only need to change contours intervals. Nonetheless, let's create a new style so that we can learn how to do it.




17. In the **Surface Style** window, first, start with the **Information** tab. It has a record of the surface identification, including the surface style name, description, dates, and authors of the modifications.



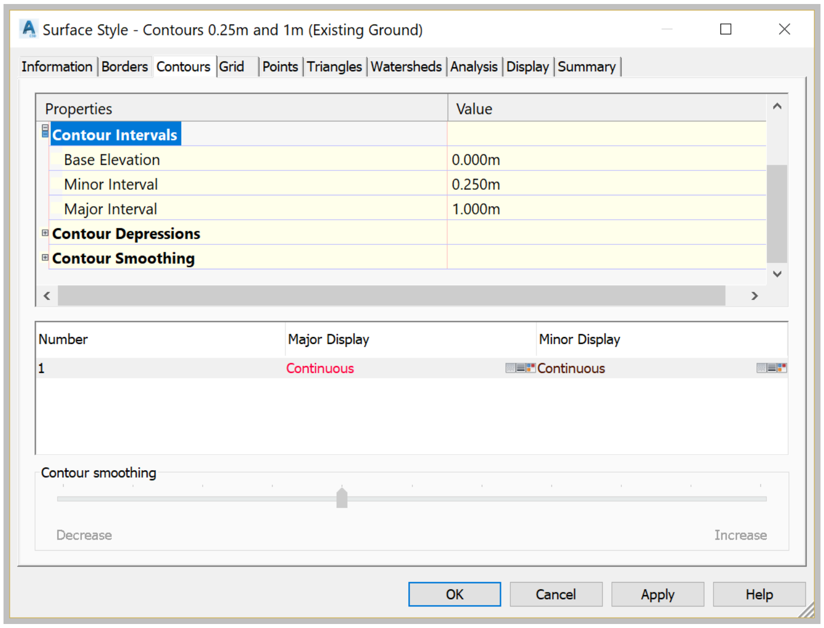

NOTES

NOTES

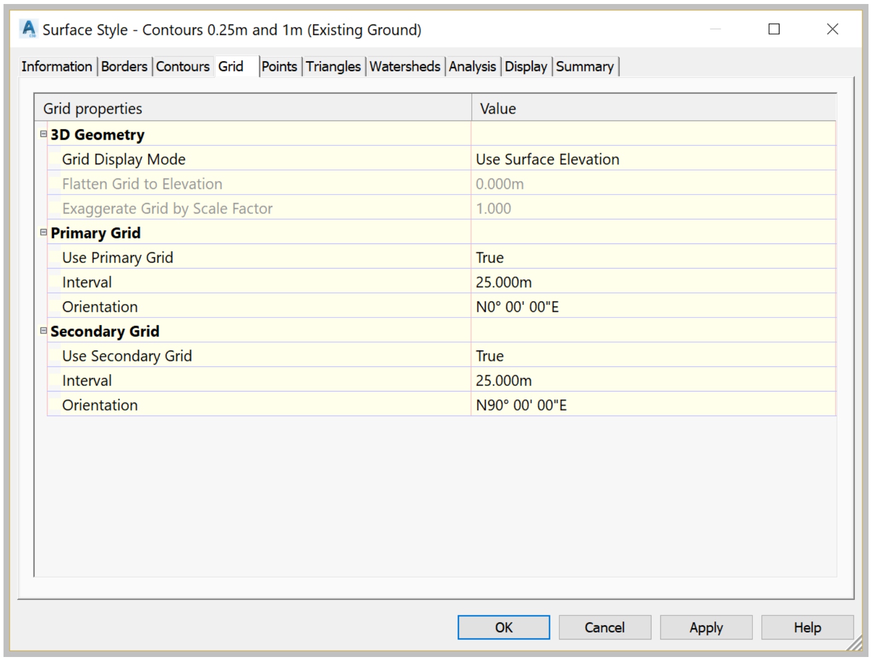
18. That takes care of the surface identification. The most important information we need next is the contours intervals. But before we do that, let's explore what is on the other tabs and what they can help us accomplish later.

19. The next tab is **Borders**. It specifies the settings for the surface border, which display the extents of a surface. It should not be confused with **Boundary** in the surface definition. A boundary is a physical object, most of the time a polyline, that helps define a border.

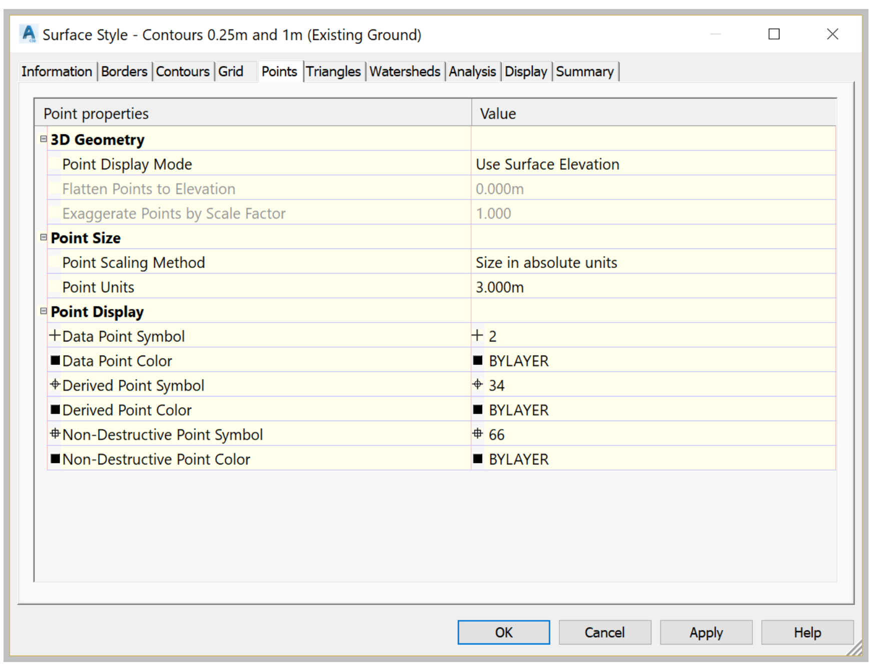
20. Up next is **Contours**, the tab we are looking for to create our style. Most of the time you only need two areas in this tab. The first is the **Contour intervals** and the second one is the **Contour Smoothing**, used to smooth contours and display them as more polished polylines. Let's open the section by clicking on the + sign close to **Contour Intervals**. The section extends, and we see three lines. The base elevation which specifies the relative base elevation for the major contours. That means the contours counting will start from here. It should always be set at a lower elevation. Otherwise, contours will be missed. We then have the **Major** and **Minor** contours. Now, set the **Minor Interval** at **0.25m** or **0.5ft** as required, and the **Major Interval** at **1m** or **3ft**.



21. On the **Grid** tab, we can specify the settings for the grid lines. Should we want to display our surface in grids, say cells of 50m by 50m or 150ft by 150ft.



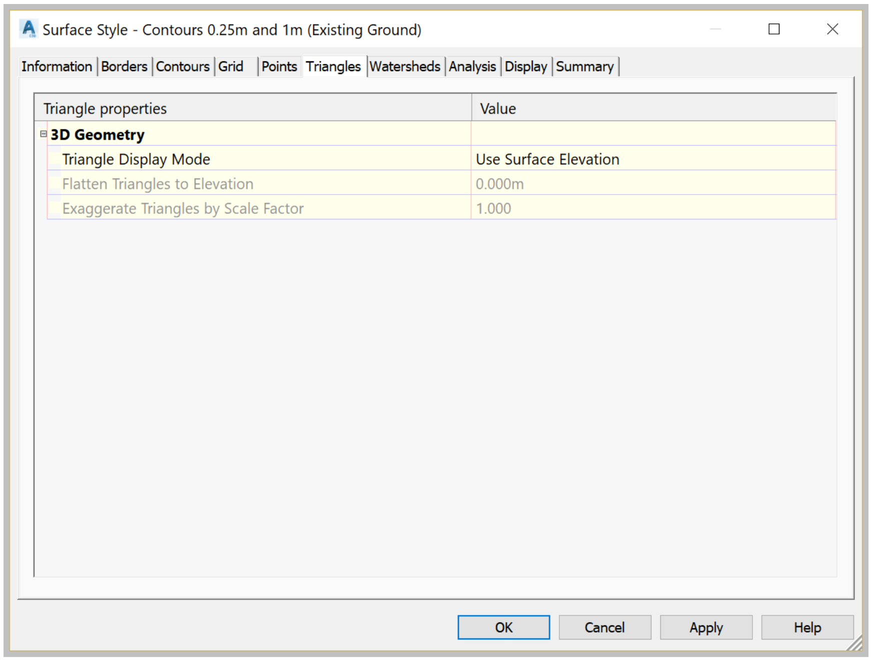

22. On the **Points** tab, we can set the settings for points of the surface entities. These are all the points used for the surface triangulation, cogo points, breakline vertices, etc.



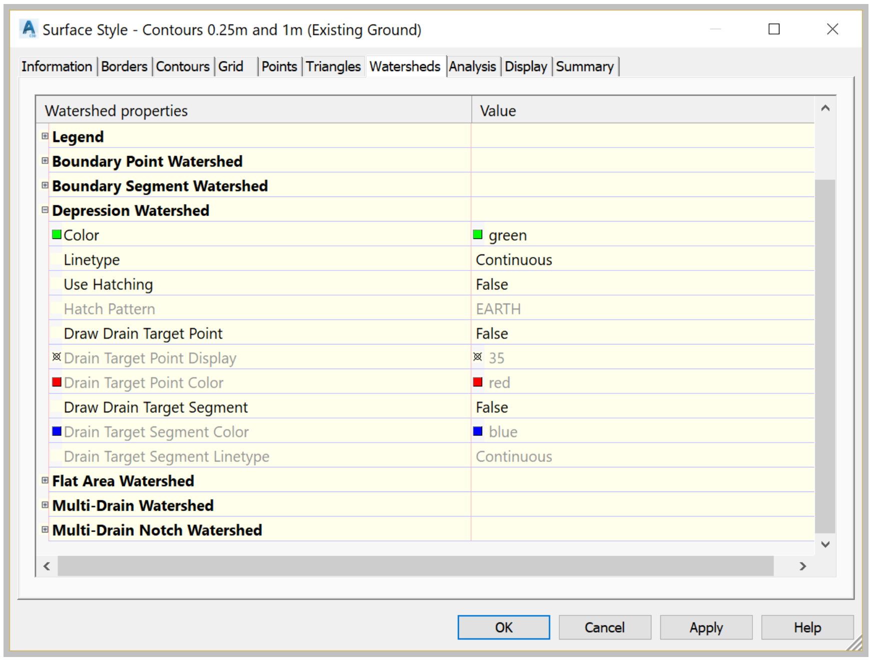

NOTES

NOTES

23. Next is the **Triangles** Tab, which specifies the properties of the triangles that constitute the surface.

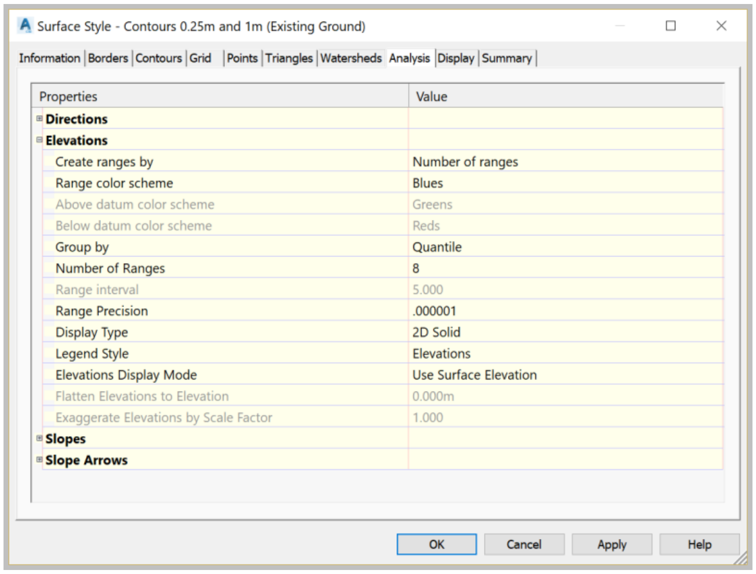



24. We then have the **Watersheds** tab, used to analyze how water flows along and off a surface.

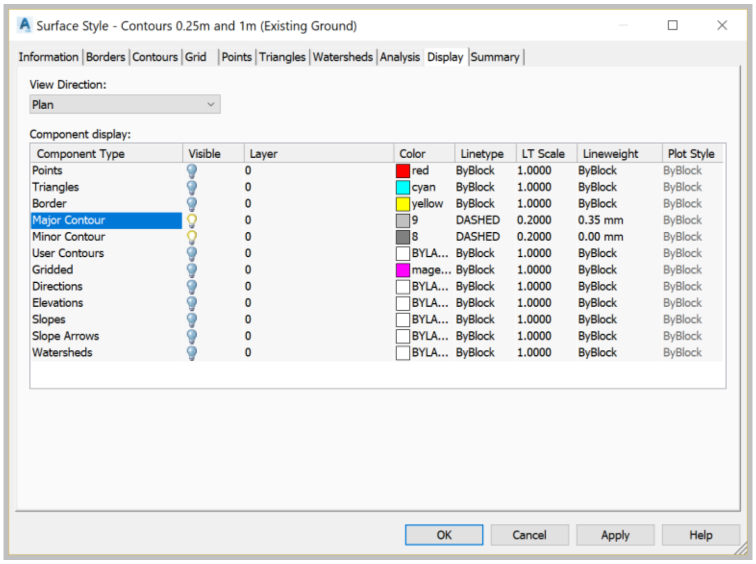


NOTES

25. Then we have the **Analysis** tab, which specifies the display parameters for the surface analyses. We've seen an example of surface analysis when we created the Elevation Map. This is where the colour and range defaults are managed. So, any time you are doing an analysis, this is where you should look for the parameter settings.




26. Then, probably the most important tab is the **Display tab**. This is where we put it all together by deciding what needs to be visible or not, among all the components that make up a surface. We can change the visibility, colour, or layer for the surface components at any time. Now, let us set the display settings for the surface style. Since this is an existing ground style, we want to use colours that will print light in our plot style file, say 9 for major contours and 8 for minor contours. We also want to make them dashed. By convention, we will display existing lines in a dashed pattern and proposed lines in continuous linetype. Let's set it to 0.2-line type scale, and for line lineweight, we will use **0mm** for major contours and **0.35mm** for major contours. This pretty much sets up our style.

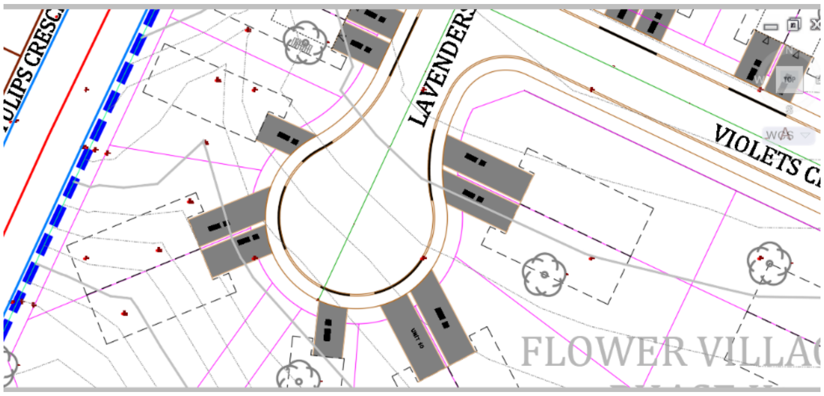


NOTES

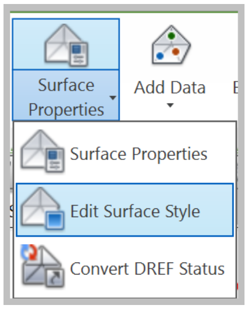
27. The last tab gives us a summary of most things we've talked about in this window. Modifications can be made here as well.

28. Let's click **OK** twice and see how our style does. Remember, creating styles is a tried-and-true process. You need to get something that looks good, prints well and meets requirements.

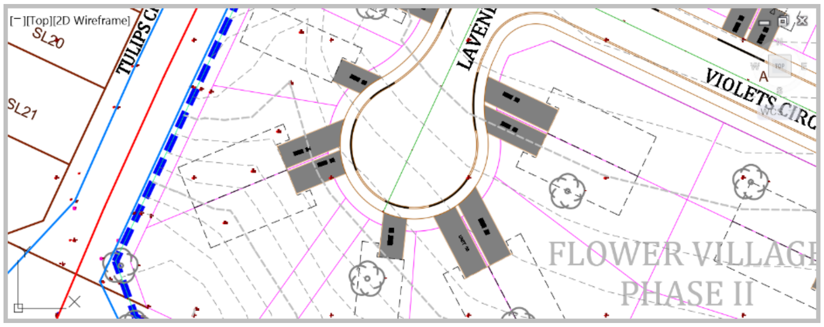
29. Looking in, our style doesn't look too bad for a first trial. Maybe we just need to adjust the linetype scale to show the dashed lines a little better.




30. Let's go back to the surface style display tab and adjust that. Select the surface, and on the ribbon click on **Edit Surface Style**. As you can see, we've been using the right-click options, but you can also access these commands from the ribbon. It all depends on which option is easier for you.




31. Now on the **Display** tab, change the **LT Scale** from **0.2** to **2** and click **OK** to close the **Surface Style** window. Now we see the contours dashed lines appear much better.



32. In summary, this is how you create a surface style. Should it be for contours, points, elevations or slopes, the process is the same. First, go the component tab to set the parameters, and manage the visibility on the **Display** tab. That's pretty much all for Surface Styles. The same process works for alignments, profiles and many other objects, with few differences.

NOTES

NOTES

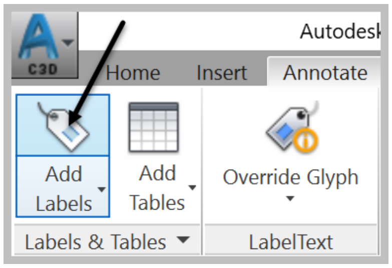
## 5.8.2 Surface Label and Table Styles

As we've seen in the lesson on **Points**, most Civil 3D entities have two types of styles: the object style and the label style. The first controls the "**look**" of the object itself, while the latter controls the **annotation** that goes with the entity. For example, a surface will need spot elevation, contour elevations, and slope values. While an alignment would need stations and offsets, and manholes would need Rim and pipe invert elevations. These are all called Labels, and the way they are displayed constitute the **Label Style**.

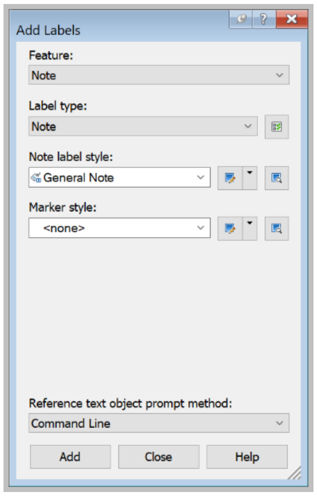
Let's create a couple of labels for the surface: slopes and spot elevations styles. To make it easier to create labels, use the global label creation dialog box.

On the ribbon,

1. Click on the **Annotate** tab, then to the far left, click on **Add Labels**. When you hover over, there are two sections, the Top and Bottom part. You need to be careful on which one you click, as they will lead you to two different paths. You may still be able to find your way and add labels, but the process will be different. We want to click on the top part.

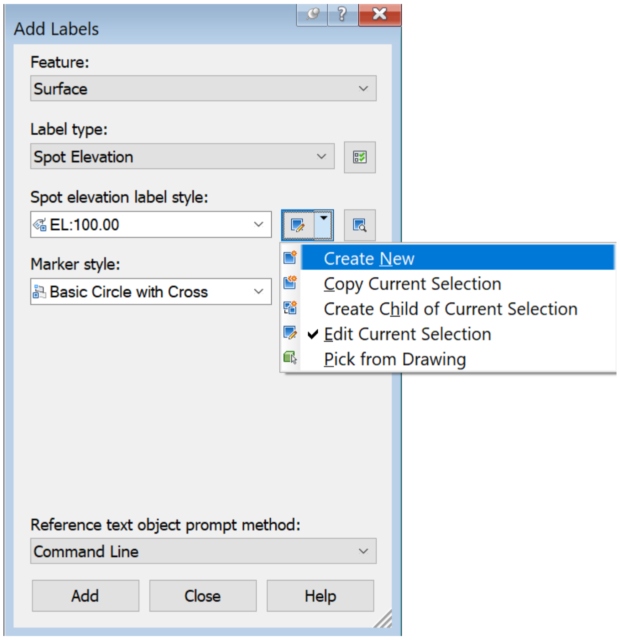



2. That brings up the **Add Label** dialog box. This is a global window for labels, as it is common to most objects. Options available in the dialog box change depending on the selected feature. If we choose to label a surface, the type of labels we will be able to create will be different than those for say alignment or profiles.

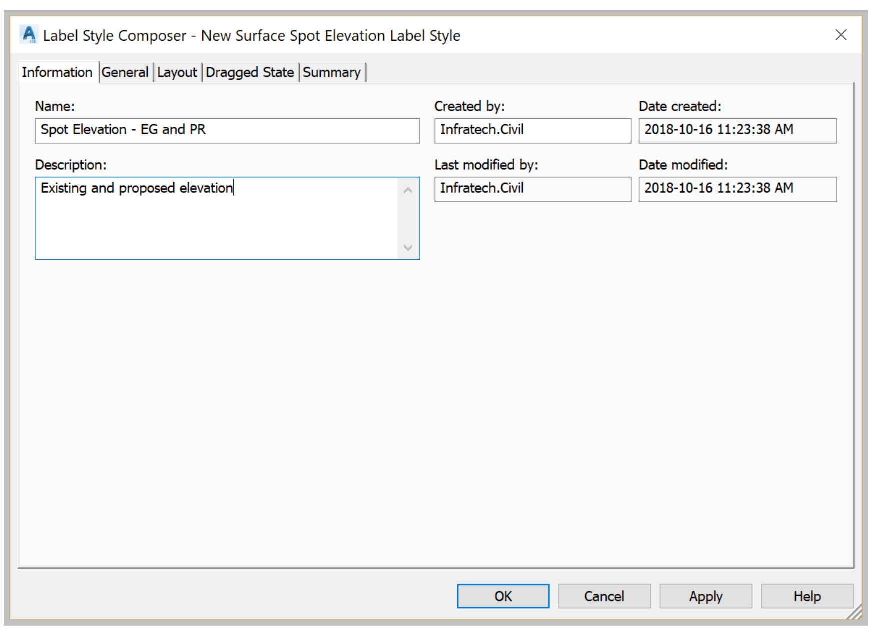


NOTES

3. Change the feature to **Surface**. For label type, choose **Spot Elevation**. We already have a few labels styles that come with our default template like **EL: 100.00**, which display elevations in that format. Now, suppose that we want to create an elevation style that will display information on two surfaces, at tie-in locations like the back of lots. At these locations, we want to display the proposed elevation and the existing ground elevation. We don't currently have a style that can do that. So, we will have to create one.




4. On the **information** tab enter the name of the label, say **Spot Elevation - EG and PR** and the description, **existing and proposed elevation** for example.



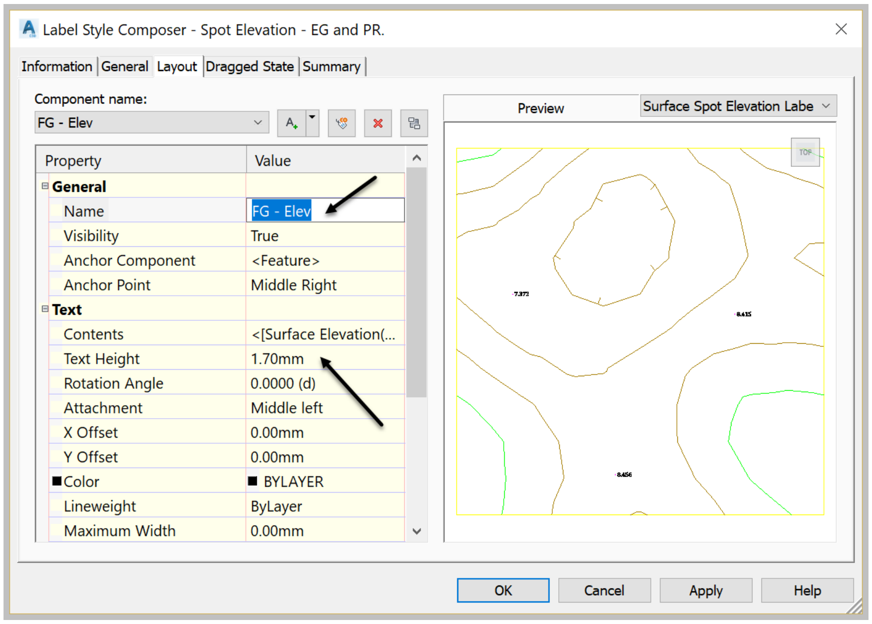

5. On the **General** tab, we can specify things like label visibility, label orientation, layer, and the like.

NOTES

6. The **Layout** tab is where most of the label creation action takes place. This is where we create the label components and control the visibility. The components already created are displayed in the **Component Name** box. We can create new components  such as text, lines, blocks, etc. by clicking on the A+ symbol to the right.

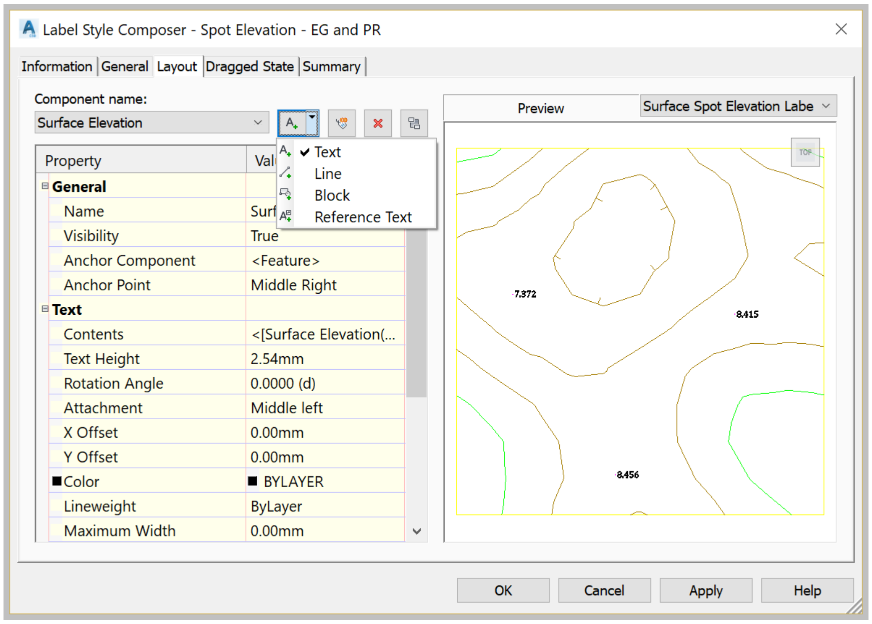
7. First, let's change the name of the default text component from **Surface Elevation** to **FG – Elev.** This will make it clear that this component provides elevations for the **FG – Finished Ground**. We can also change the text size for the label, and other properties.

8. Let's also change **Surface Elevation** component name to **FG – Elev**, to make it clear that this component is for **FG** or Finished Ground. In this window, we can also change elements of the label like text styles and height, rotations, colours , lineweight, border, and more.

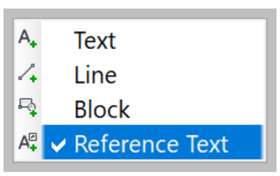


NOTES

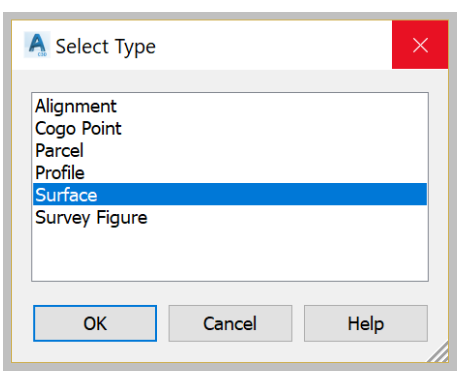
9. For our second surface elevation, we need to add a second text, besides the default, **Surface Elevation** text component that we just edited. The type of text we will need this time is not the first type that says **Text** but the last one, **Reference Text**. The difference is that using **Text** will give us information on the current surface we are attaching the label to, the proposed design surface. While the **Reference Text** will reference another surface, the existing ground surface.




10. So, click on the **reference text**.

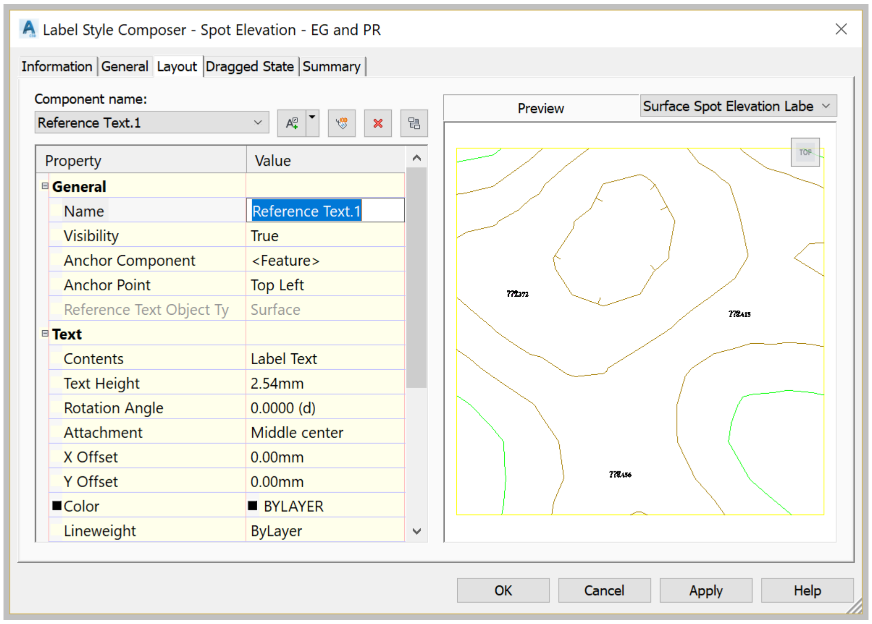



11. Our surface label will be able to grab information from different types of entities such as alignments, profiles, parcels, and the like. This capability has several great applications in creating all types of labels. In the **Select Type** window, choose **Surface.** That's because, besides our proposed surface, we also want the label a reference elevation from the existing ground.

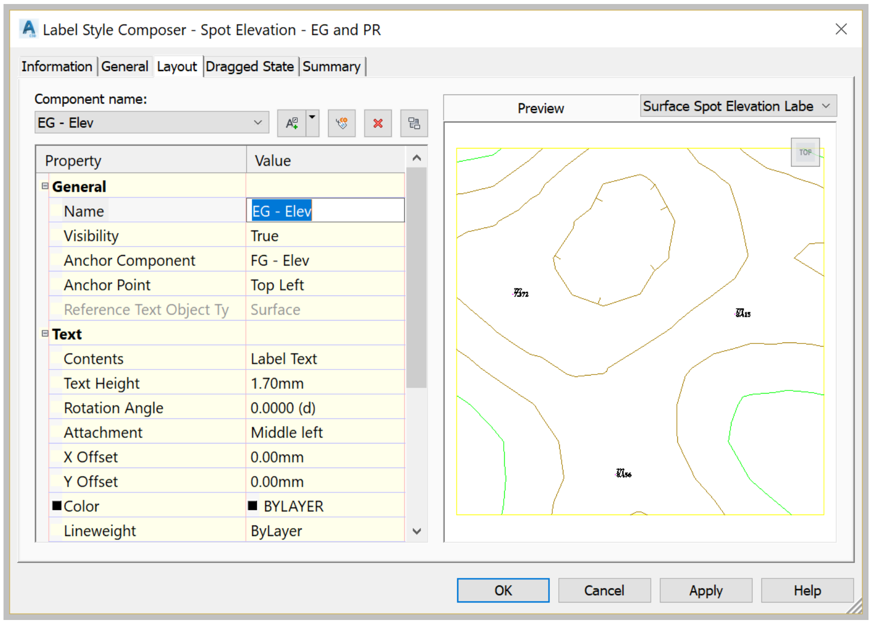



12. Click **OK**.

13. A new text component is created with the name: **Reference Text.1**.

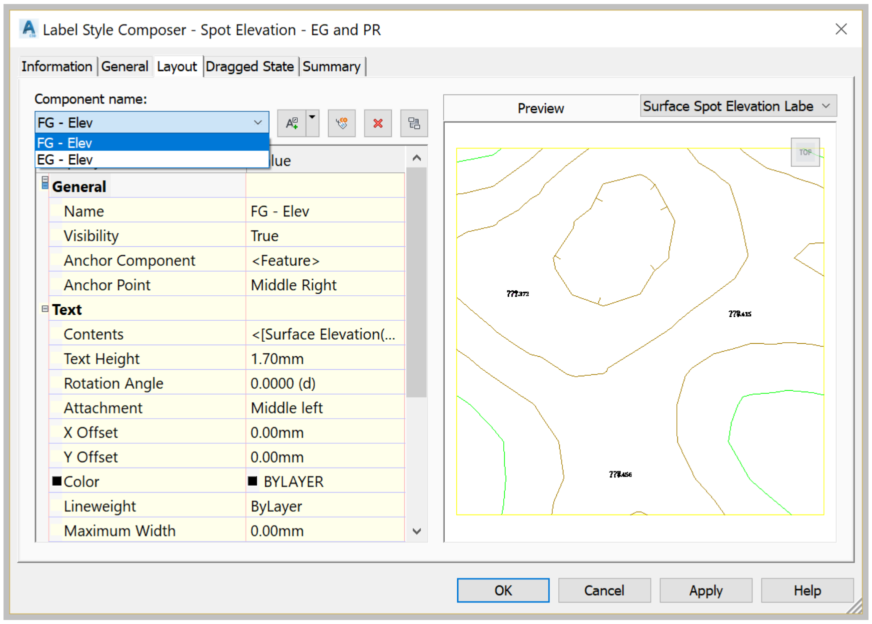



14. Next, change the name of the newly created component from **Reference Text.1** to **EG – Elev** to make its function more explicit. Giving obvious names is crucial. It facilitates managing styles and providing better template readability for future users of the file.




NOTES

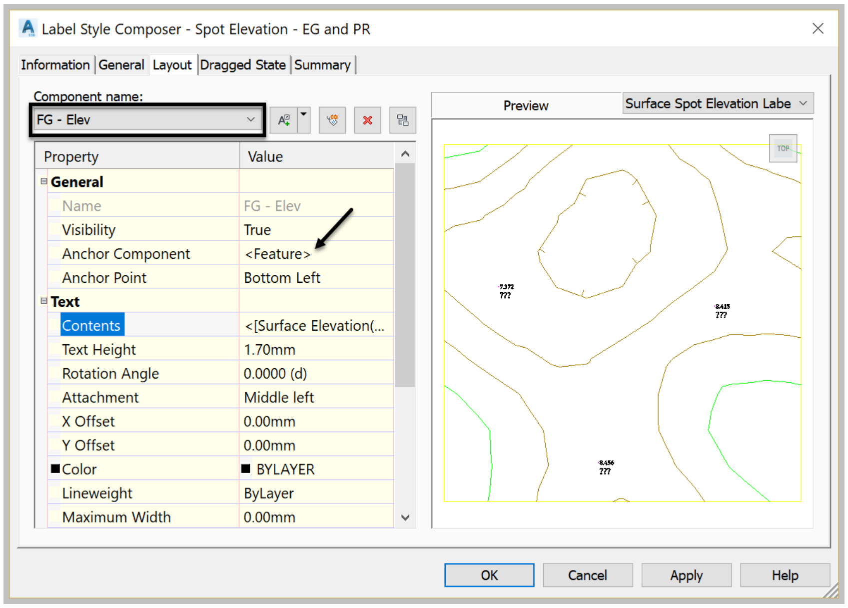
15. When you open the **Component Name** box, you should have two components. If you have more than two, you can always click on the red X to the right  to delete unwanted ones.



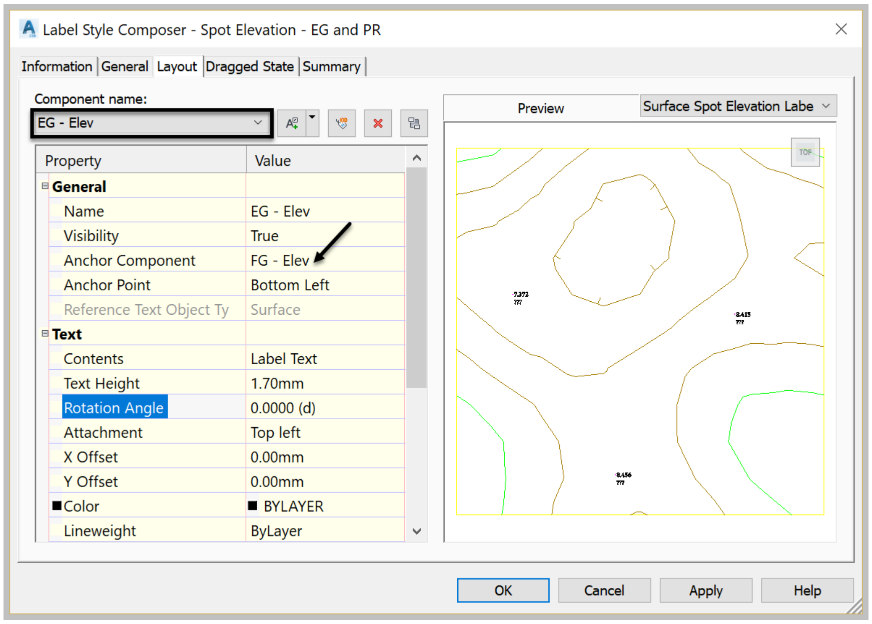

16. One important thing, we need to understand is how to stack the different components we are creating. For good readability, we have different options to anchor the components in relations to each other.

- First is the **Anchor Component**, which means the object to which we want to tie the label. We can choose **Feature** to specify that we want to tie the component to the feature, meaning the point where we are creating the label.
- Once we specify the **Anchor Component**, we also need to determine the **Anchor Point.** It means in relation to the **Anchor Component** where do we want the label to be (left, right or middle)? For this label, let's decide to
  - Anchor **FG – Elev** to the Top Right of the Feature, mean the point where we are putting the label
  - Then we will anchor **EG – Elev** to the bottom of FG – Elev. To do that we can anchor the component **FG – Elev** to **Feature**.

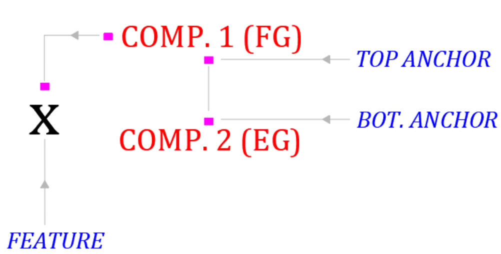
NOTES




17. Next, we need to position the **EG – Elev** component. We can also anchor it to **Feature**. But we don't want it to overlap **FG – Elev**. So, instead, we are going to anchor it to **FG – Elev**.



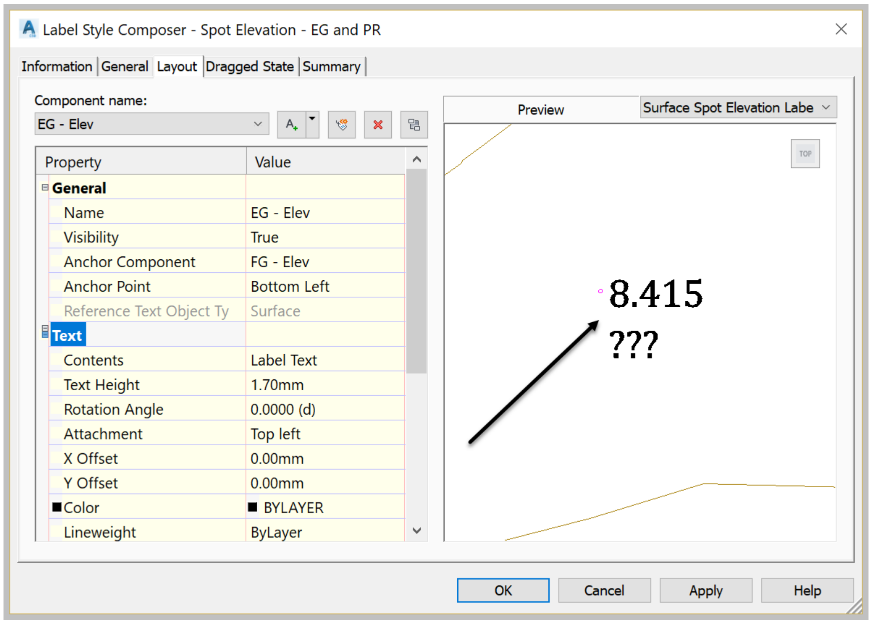

18. To summarize, so far, the proposed label component (**FG**) is anchored to **Feature** or the insertion point of the label, while the existing label component (**EG**) is anchored to the proposed label.




NOTES

NOTES

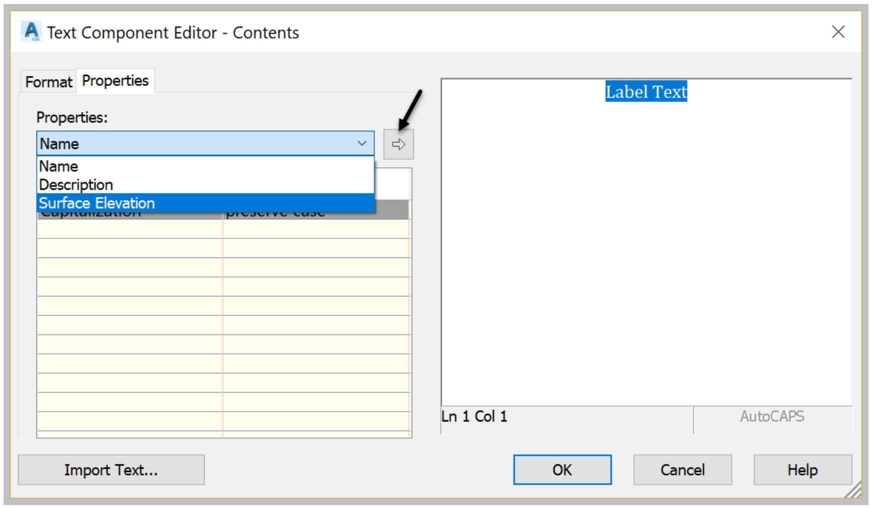
19. Don't forget that you have a preview window that allows you to see how your labels stack while you are still creating them.




20. Notice the question marks in the preview? That is because the **EG – Elev references** an external surface, which we have not specified yet.

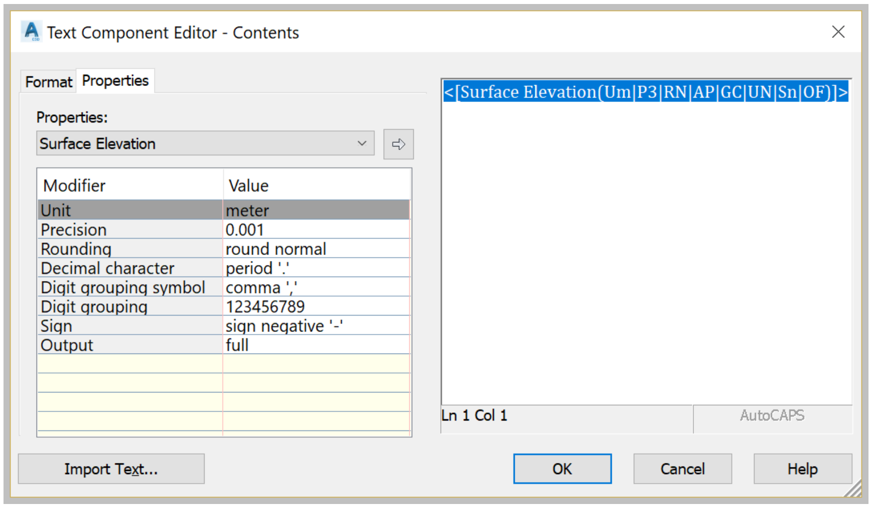
Now that we have positioned the label the way we want let's work on the content, the actual information we want to convey with these labels.

21. Start with the **EG – Elev** component. The proposed label is probably already set. To change the content, on the **Contents** line, when you click on the **Label Text** cell, you will notice an icon with three small dots. Click on it. We are familiar with this window. We have used it before in the template and styles lesson. Now first select **Label Text** in the editor, then the **Properties** drop-down box, select **Surface** and click on the horizontal arrow to apply that label.

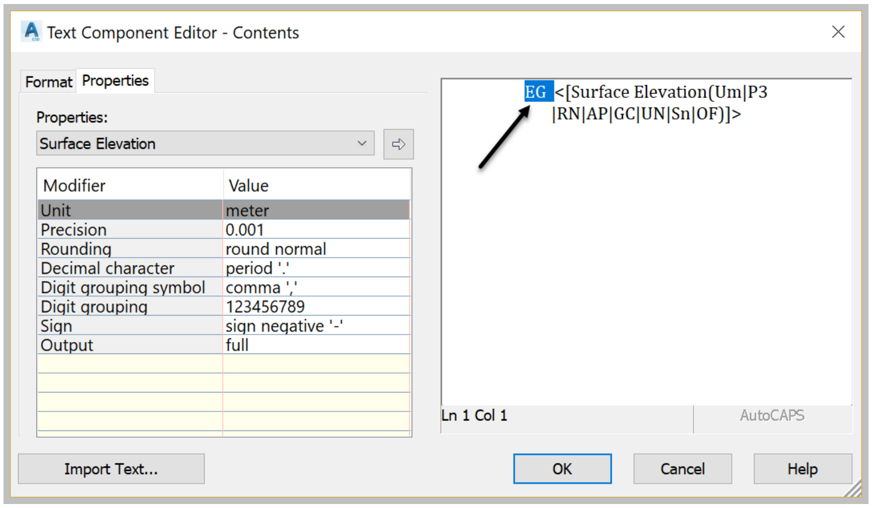


NOTES

22. Once we do that, the reference surface elevation code will be applied. We also have an option to format the label display such as the label units (meter or feet), precision (decimal places), and others.

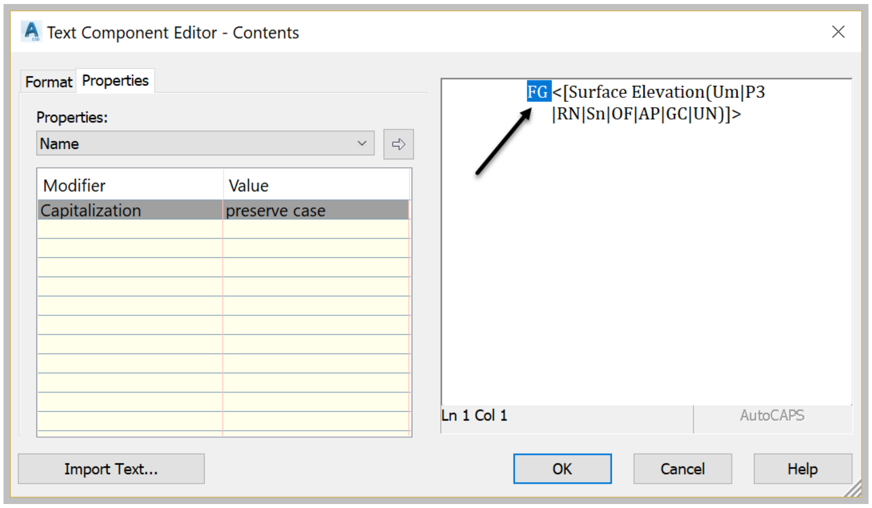



23. One more thing, we will have two labels stacked on top of each other. So, we need to be able to differentiate them. Let's add a text in front of each, so we know what surface elevation they represent. In front of the elevation code, enter the surface name or initials, like EG.



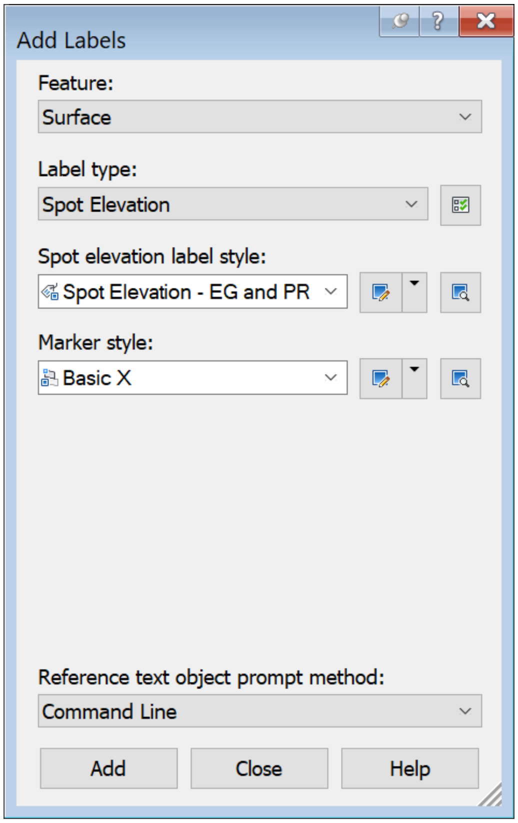

24. Click **OK** to return to the **Label Composer** window. Now select the **FG – Elev** component and do the same thing. First, go to the content line, click the content editor icon , then add the prefix **FG** before the elevation code.

NOTES

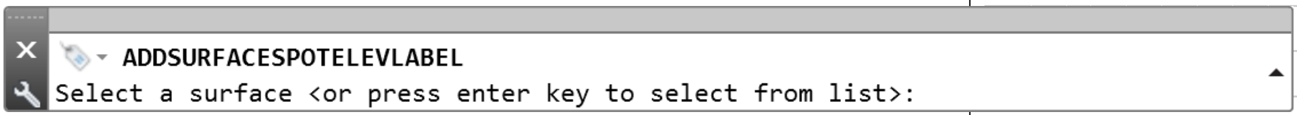



25. Click on **OK** twice to close the **Text Component Editor** and the **Label Composer** window to return to the **Add label** window. If it's closed, you can reopen it from the **Annotate** tab.

26. Make sure our new style **Spot Elevation – FG and EG** is active and select **Basic X** style.



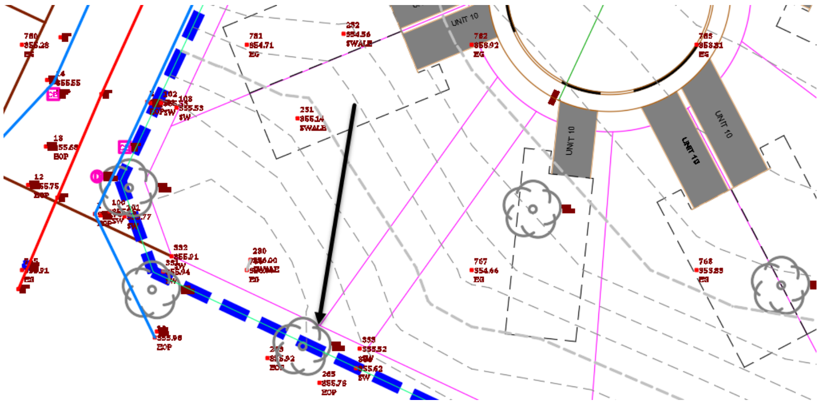

27. Click on **Add**. At the command line, you are prompted to select a surface to annotate. You are presented with two options. You can either click on the surface in the drawing, if you can identify it. Or, you can press **enter** at the command line.



NOTES

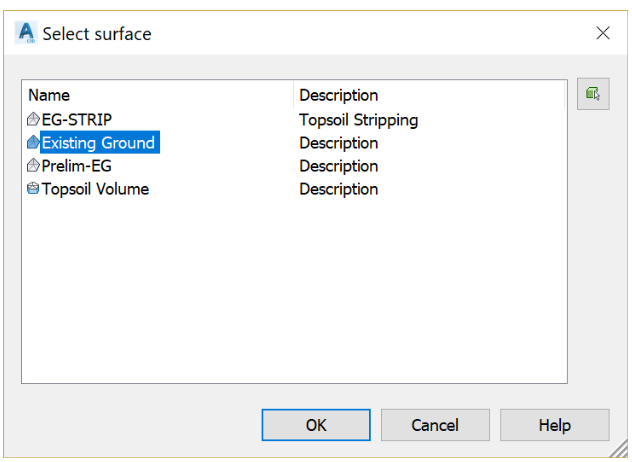
28. Press **enter** at the command line. Ideally, at this step, we would pick the **FG** (Finished Ground) surface. Since we are not that far in the design, we don't have one yet. So, for the time being, let's select the **Prelim-EG** surface, just to simulate what would happen once we get our **FG** surface.

29. You are now prompted for the location of the label. Let's choose the back of the lot for one of the parcels in the **Lavender Court** cul-de-sac , to the west of the utility easement.

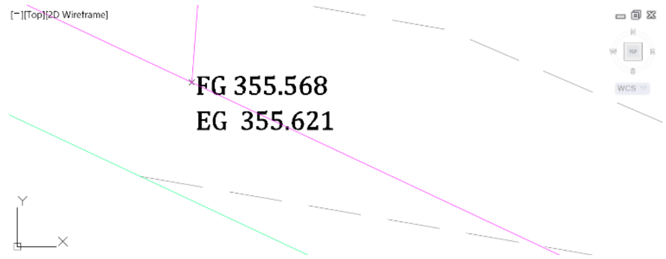


30. Now that we have placed the FG label, we are prompted to select the surface that we are trying to reference for EG label.


31. Press **Enter** at the command line to select **Existing Ground** for reference surface and click **OK**.




32. We are then prompted to select another point to label. Let's just click on one more point and end the command.




33. We now have exactly what we wanted, a label style that displays both **EG** and **FG** elevations.

34. You can press escape at the command line to end this command.

We will see later, how to add other types of labels like slopes and contours after we are done designing the **Finished Ground**.

NOTES

NOTES

# 6 PARCELS

## 6.1 Introduction

The methods we will learn in the current lesson can be applied to residential, commercial or industrial lots. Also, we can be more creative and apply these methods to the right-of-ways, roads, green spaces, and other public easements.

To do this, Civil 3D provides us with a set of commands that we can use to design the lot. We can set design parameters to meet the standards of the permitting authority, for minimum lot dimensions and areas.

In addition, we can also create parcels from existing objects like lines or polylines. These are typically obtained from the property appraiser's office.

Let's see how to create parcels using the two methods. First, we will start by creating parcels using the **Parcel Creation Tools.**

## 6.2 Sites

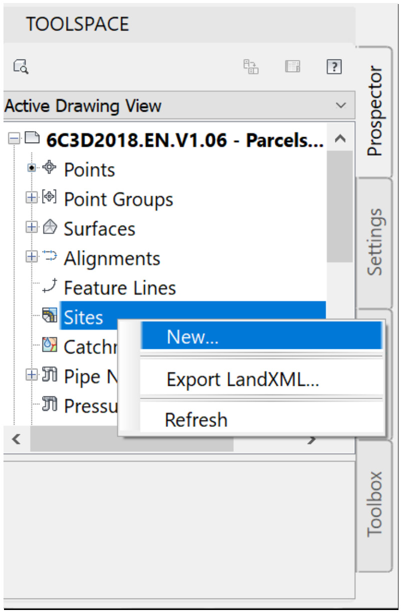
1. Start by opening the **06.01-Parcels** dwg file in **Lesson 06** practice folder.

2. Next, we need to create a **Site**. Before going any further, we need to talk a little bit about **Sites** as there is quite a bit of misconception on **Sites**. Some users, even certain long-term ones, use the term **Site** literally. It is considered a place where we can categorize construction sites, jobs, or phases of a bigger project. While you may be able to use them for that, they are created for an entirely different concept. In Civil 3D, **Sites** are used to manage objects with a common topology. **GIS** professionals are more versed in the notion of topology. In a **Site**, entities such as parcels, alignments, gradings, and feature lines can share a topology. The **Oxford dictionary** defines a topology in mathematics as: "The study of geometrical properties and spatial relations unaffected by the continuous change of shape or size of figures". This is a fancy definition to say that objects in a topology, or a site in this case, interact with each other. Examples of sites include soil maps, watersheds, or subdivisions. Sites can geographically overlay each other.

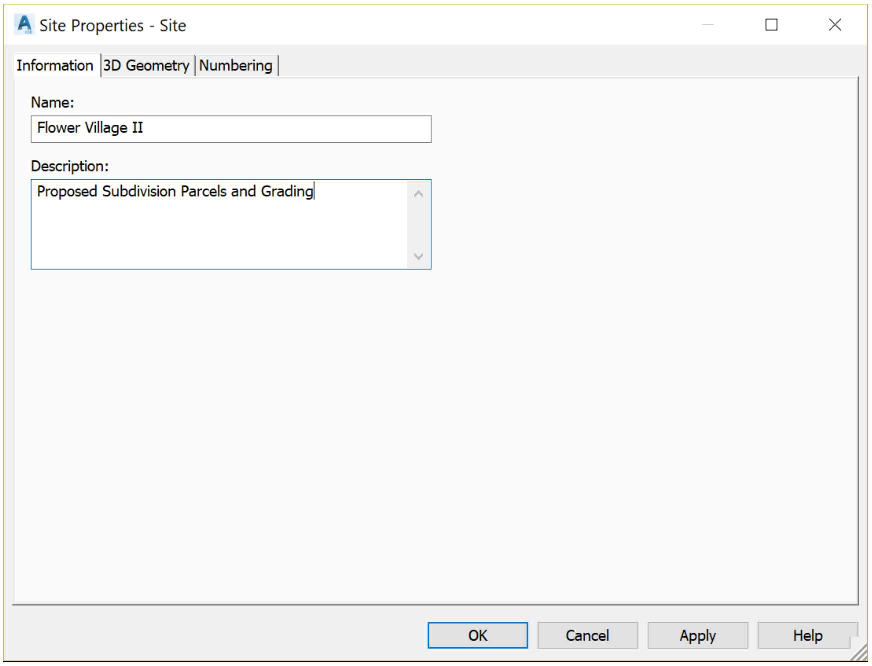
3. Case in point, in a subdivision with roads and lots, we can include parcels and right-of-way alignments in the same **Site.** The right-of-way parcels are then automatically created using the alignments. However, parcels would never be affected by an alignment if they don't belong to the same site, even if

they overlap. The most common case of topology in Civil 3D is grading with feature lines. When you are creating a grading, say for a pond, your top of pond and grading group need to be in the same site. That will allow you to create a bottom of pond from the top feature lines. In summary, feature lines need to share a topology to interact with each other for grading purposes. Thus, they can share an elevation at crossing points. That means they need to be in the same site. Now that this has been cleared up, we can start creating the parcels.

4. Let's create a **Site**. To create one, in the prospector, right-click on **Sites**, then **New**.




5. In the new dialog box, assign a name and a description, for instance, **Proposed Flowers Subdivision** parcels and grading.

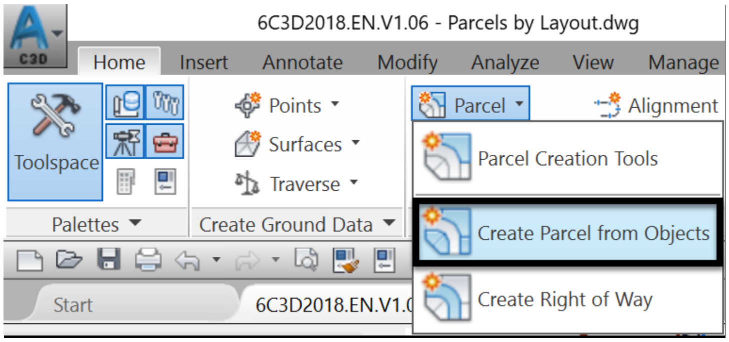



6. Click **OK**

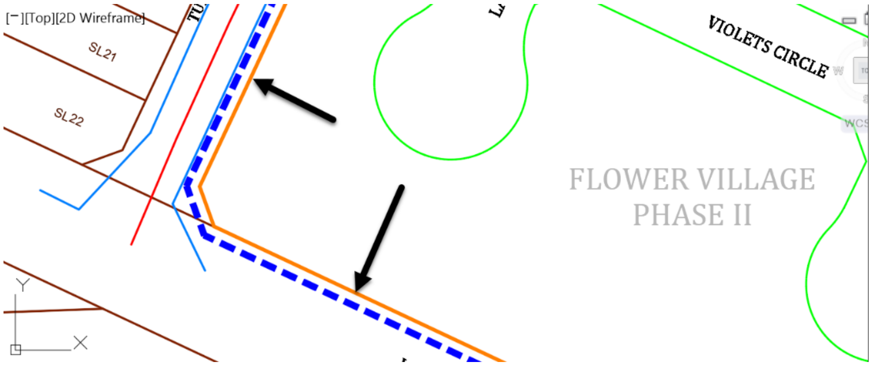
NOTES

## 6.3 Parcel from Object

1. Create a parcel from the main lot (blue boundary line). To do that, from the Ribbon, click on the parcel command and **Create from Object** from the drop-down menu.




2. Click anywhere on the Orange line representing the site boundary.



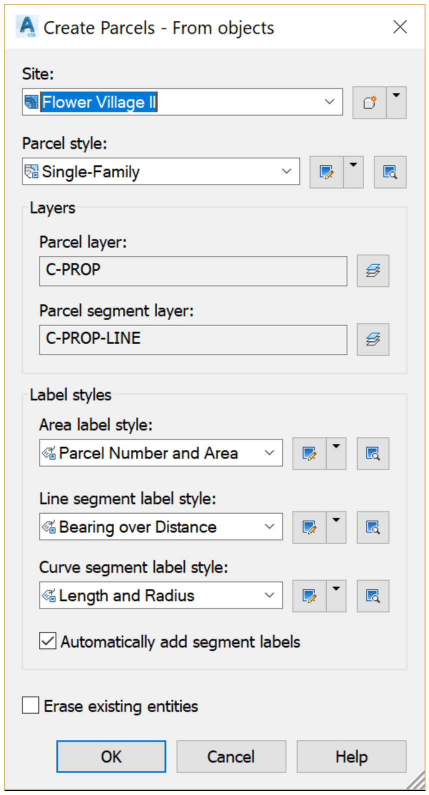

3. Type **Enter** at the Command Line input.

4. In the "**Create Parcels**" window,

    - Choose the options to put the created parcel lines into the **Site**, **Flower Village II.**
    - Apply a single-family style with the townhouse lots we will be creating.
    - Accept default layers.
    - Use **Parcel Number and Area** for the area's labels, **Bearing over a distance** for the line segments, **Length and Radius** of curve segments.

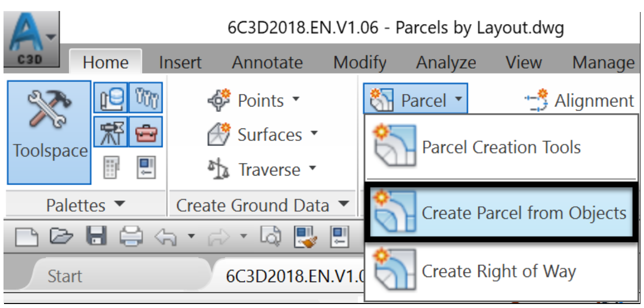
NOTES

- We will also decide to keep the original lines; even though we can erase them. We may need them to create future grading surface boundaries.

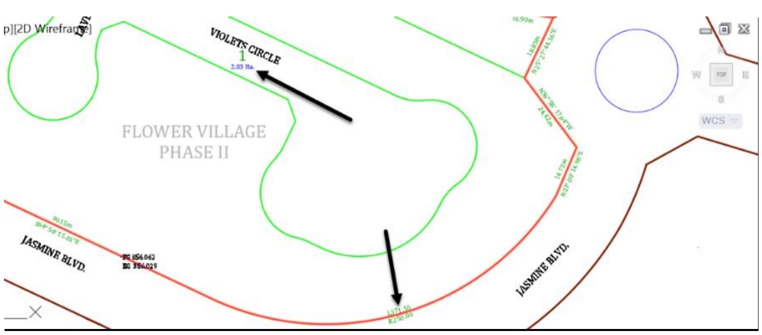



5. Click **Ok.**

6. Now, define the road right of way. Rerun the same command to create parcels from objects.



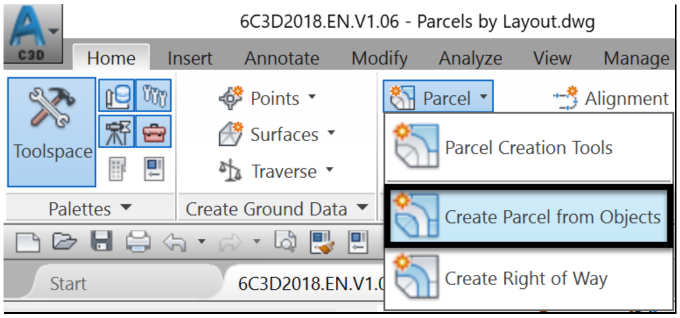

7. You will notice that the overall parcel has been created from the boundary polyline. Segment labels have also been added.



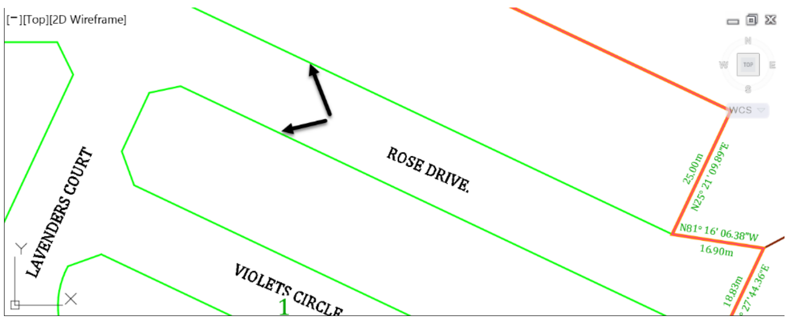

NOTES

NOTES

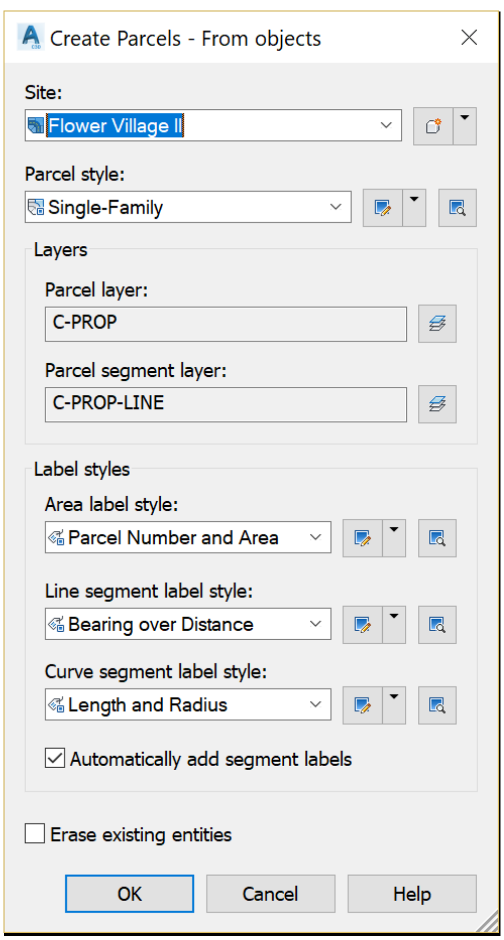
8. Next, let's create a parcel for the road right-of-way. Rerun the **Create Parcels for Objects** command from the ribbon.




9. This time select the two road right-of-way polylines.



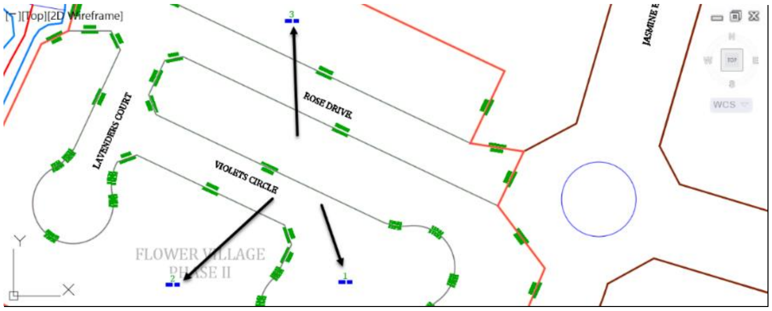

10. In the next box, accept the same default settings previously used for the boundary polyline.



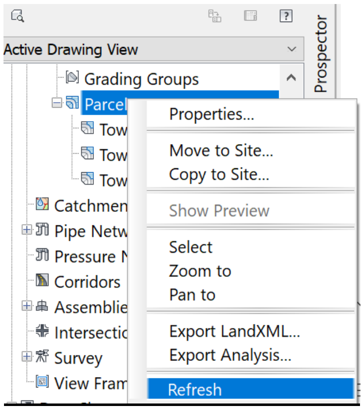
NOTES

11. Click **Ok.**

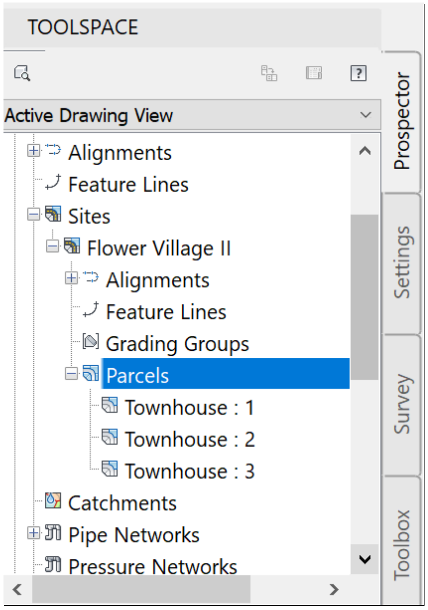
12. We have now created three lots. Two residential parcels and another one representing the road right-of-way.



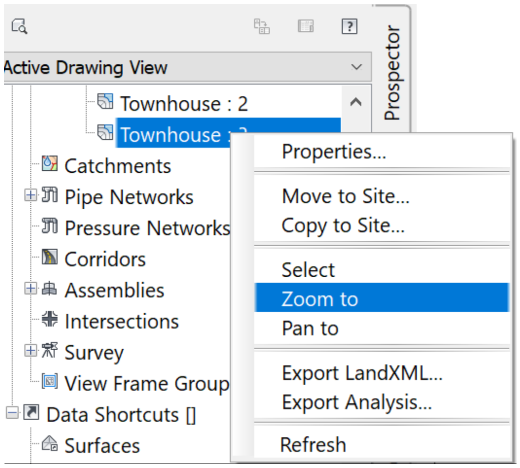

13. While we are focused on creating parcels in the drawing areas, the database is being updated with new information. Let's check it. In the **Prospector**, expand the **Flower Village II** site collection.



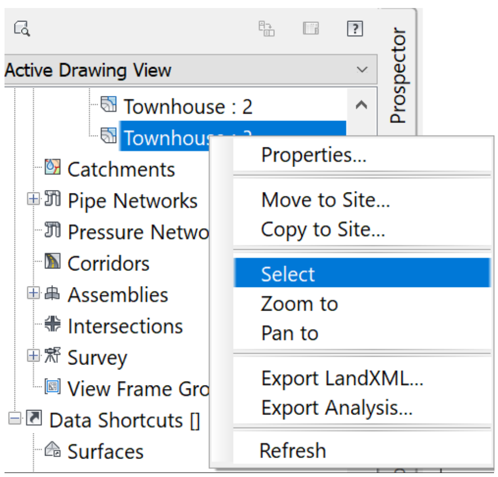

14. After a **right-click** on parcels and **refresh**, the parcels appear in the site.

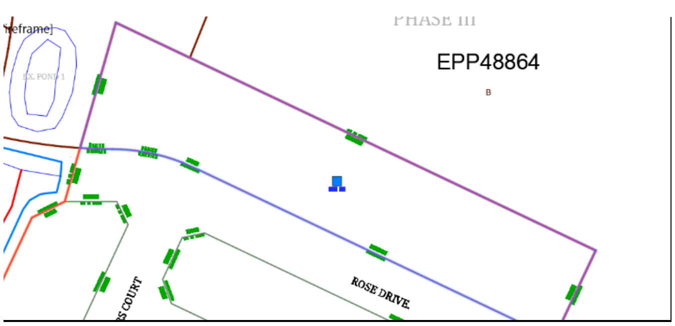


15. From here, we can zoom to a specific parcel by selecting it, right-clicking and select **Zoom to**.




16. We can also select a parcel if we need to change its properties or perform other operations on it.




NOTES

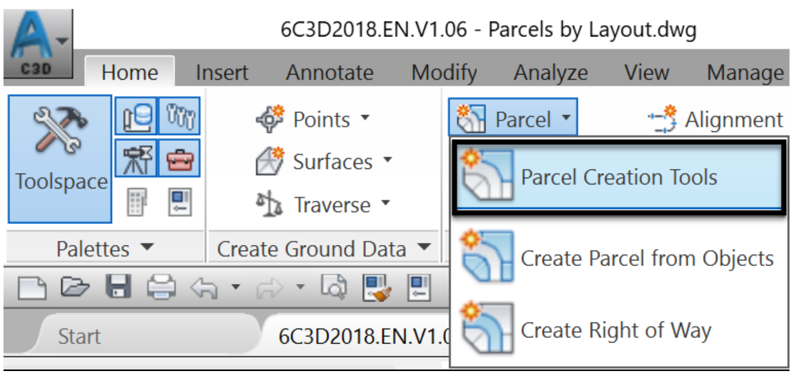
## 6.4 Parcel Creation Tools

1. Had a plat already been done for the subdivision, we could convert individual AutoCAD parcel lines or polylines to Civil 3D parcels. The process would be the same as what we have done for the boundary and right-of-way lines. Simply,

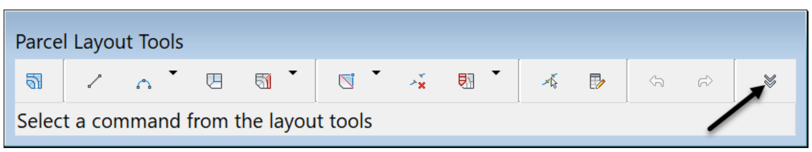
    - run the **Parcel from Object** command,
    - set the parcel creation parameters,
    - then, create parcels by selecting the objects (lines or polylines).

2. But, then again, we are creating a layout for a vacant lot, so we are going to run another parcel creation command, the **Parcel Creation Tool**. Municipalities have design standards for residential developments lots layouts. This includes lot frontage width, lot depths, and minimum areas. These standards must be met for a subdivision plat to be accepted. Civil 3D gives us the needed tools to meet these standards while optimizing the use of our land.

3. Keep working on the same drawing or open the **02-Parcels-by-Layout** dwg file in the same folder. On the Ribbon launch the command to create parcels **by layout**, the **Parcel Creation Tool**.



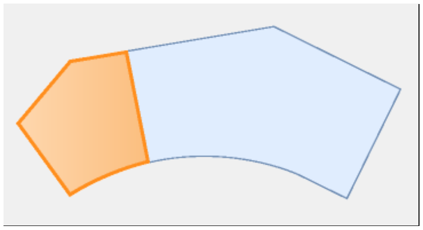

4. Expand the toolbar if it is not already.



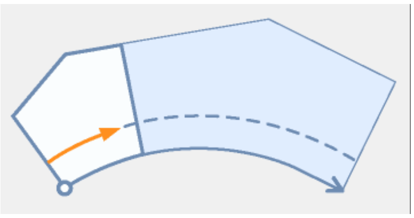

5. In the **Parcel Layout Tools**, **Parcel Sizing** area, enter the following design parameters.

    - A minimum parcel area of **150** square meters or **1,600** square feet depending on your units

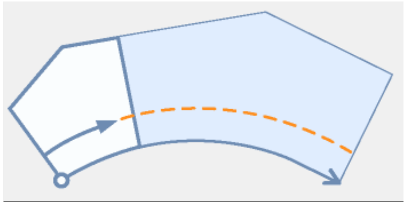
NOTES



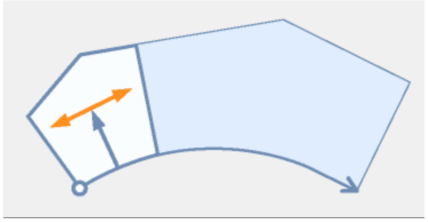
- A minimum frontage of **8.5m** or **30ft**



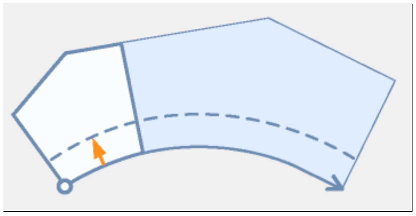
- **Yes,** for minimum frontage at offset.



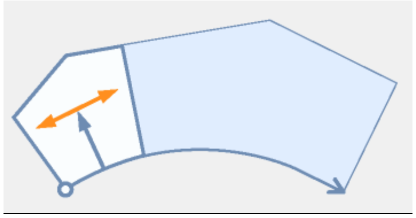
- A minimum width of **8.5m** or **30ft**. This is the minimum width of the lot at any location along the parcel.



- A frontage offset of **5m** or **15ft**

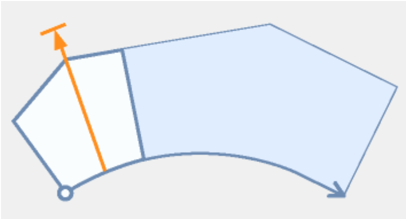


- A minimum depth of **15m** or **20ft**.



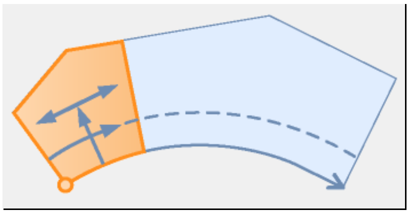
NOTES

- A maximum depth of **40m** of **150ft**.

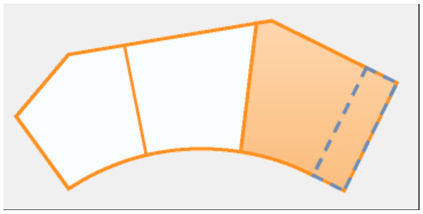


- When multiple solutions are possible, let's go with **Smallest Area** to maximize the land use.

- Let's turn off the **Automatic Mode** so that we can follow individual lots when they are being created.



- And finally, let's redistribute any remaining area that is too small to meet our lot creation criteria into the last created parcel. Typically, that would be a corner lot.

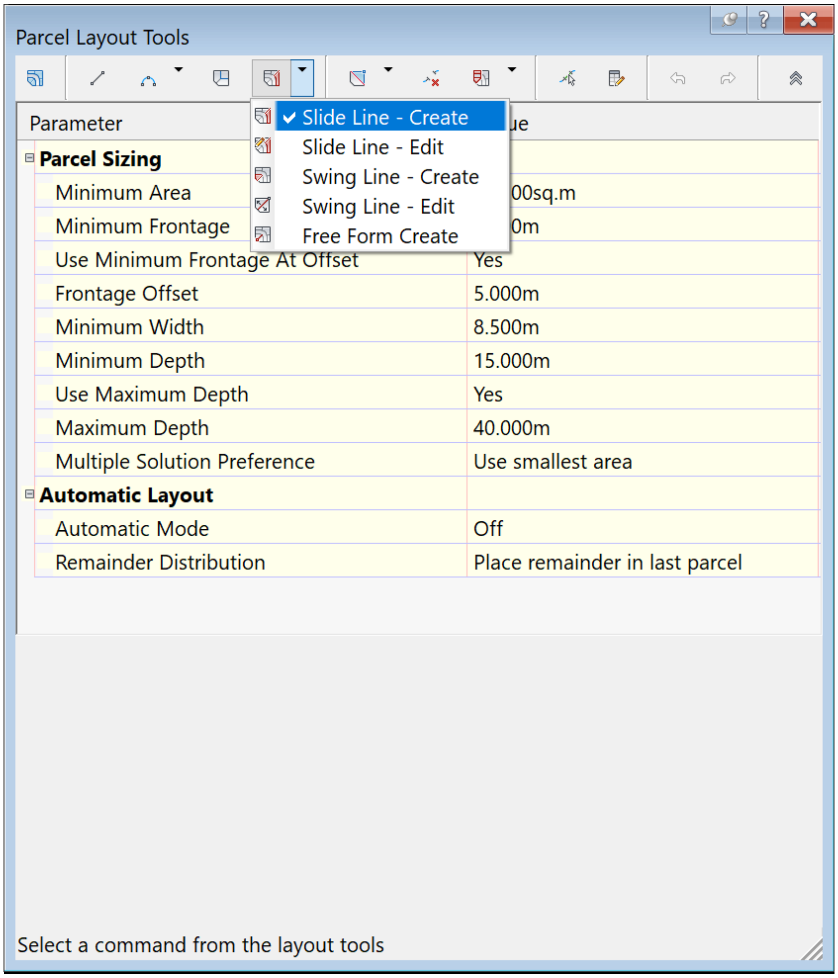


NOTES

NOTES

### 6.4.1 Slide Line – Create

1. We now have our parameters set, and we are ready to start laying out our parcels. We are going to run the **Slide Line – Create** to have lot lines, or **Parcel Segments**, created by sliding along the frontage.




2. In the **Create Parcels – Layout** dialog box:

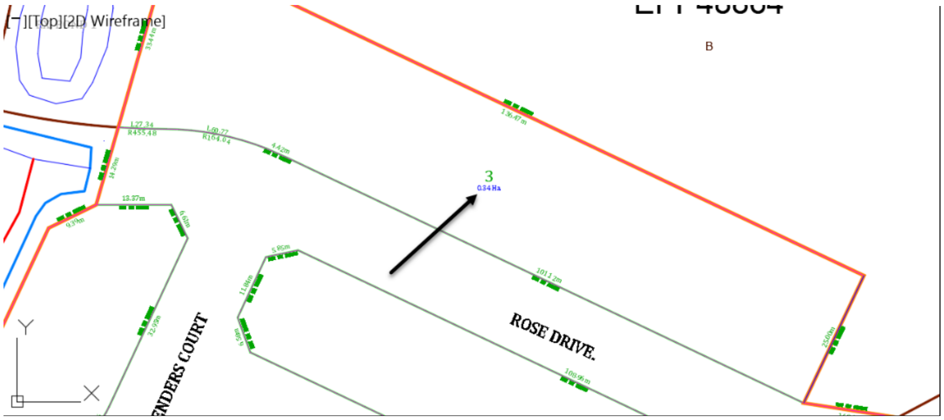
   - Set the Site to **Flower Village II**,
   - With a **Townhouse** Parcel style
   - Leave the **Parcel Layers** to the default settings in our template
   - Assign the **Label Style** for the area, the line segments and curve segments to your preferences, or as required by the reviewing agency.
   - Check the **Automatically add segments labels** checkbox.
   - And finally, click **OK**.

NOTES

3. At the command line, you are prompted to specify the parcel to be subdivided.

```
Select from the layout tools or [eXit]:
CREATEPARCELBYLAYOUT Select parcel to be subdivided or [Pick]:
```

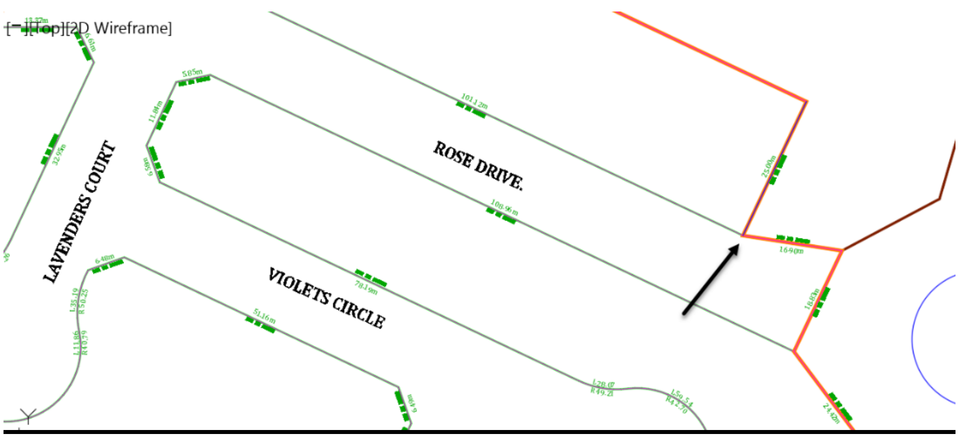
4. Click on **Parcel 3**, the one to the north of the main street, **Rose Drive**.




5. You are then requested to specify the **Start Point of frontage.**

```
Select parcel to be subdivided or [Pick]:
CREATEPARCELBYLAYOUT Select start point on frontage:
```

6. To specify that, we want to click on the southeast corner of the parcel.

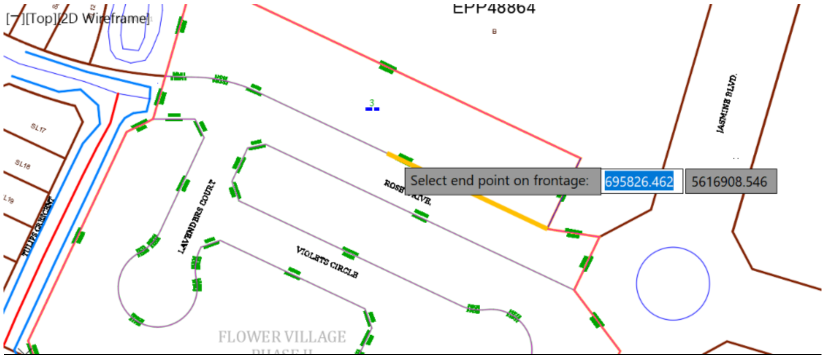



7. Next, we must specify the endpoint of the frontage.

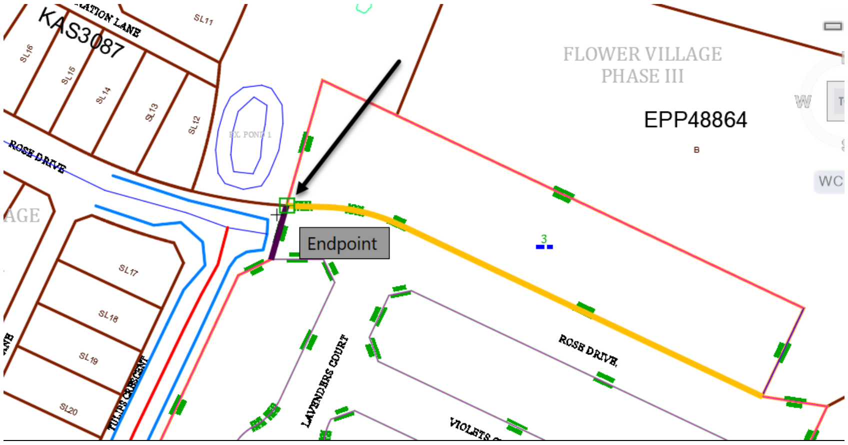
```
Select start point on frontage:
CREATEPARCELBYLAYOUT Select end point on frontage:
```

NOTES

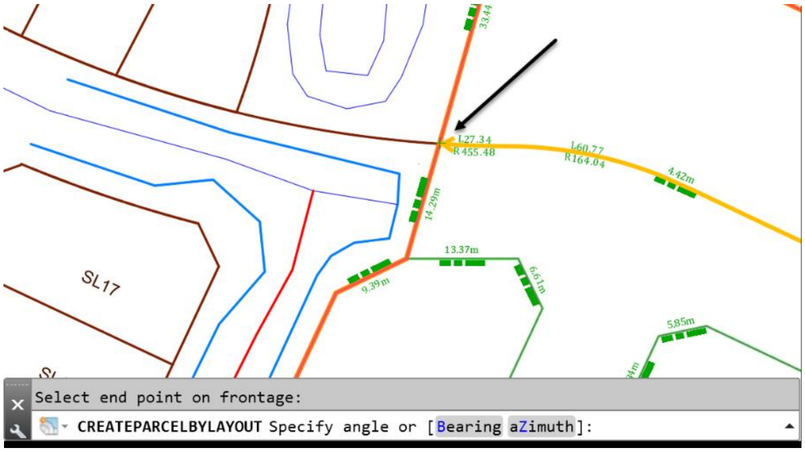
8. What you need to do is slowly slide westward along the south frontage of **lot 3**. Notice how the orange frontage line slides with you.




9. When you get to the western end of the frontage, click to tell Civil 3D that you are done specifying the frontage.

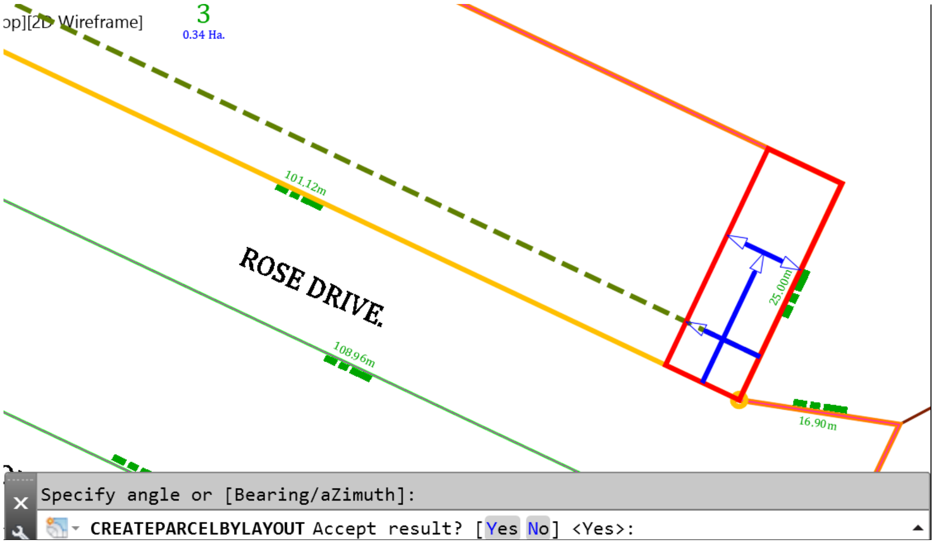



10. An arrow is, then, added to the frontage line to indicate the direction in which lots will be created. You are then prompted to specify the angle of parcel segments, compared to the frontage.



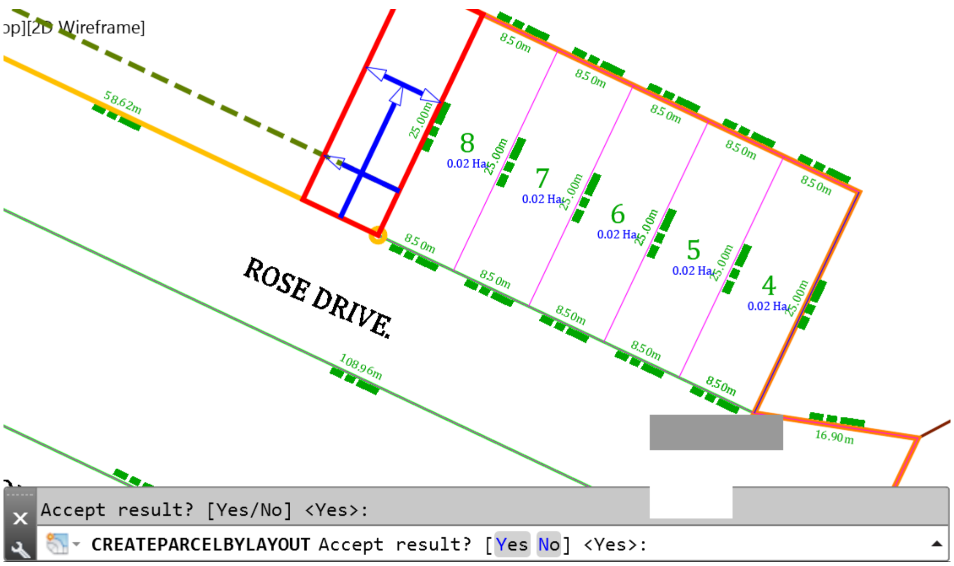

11. **Enter** at the command line to accept the default 90-degree angle. The first parcel is now created; you are prompted to **Accept** or reject the result.

NOTES



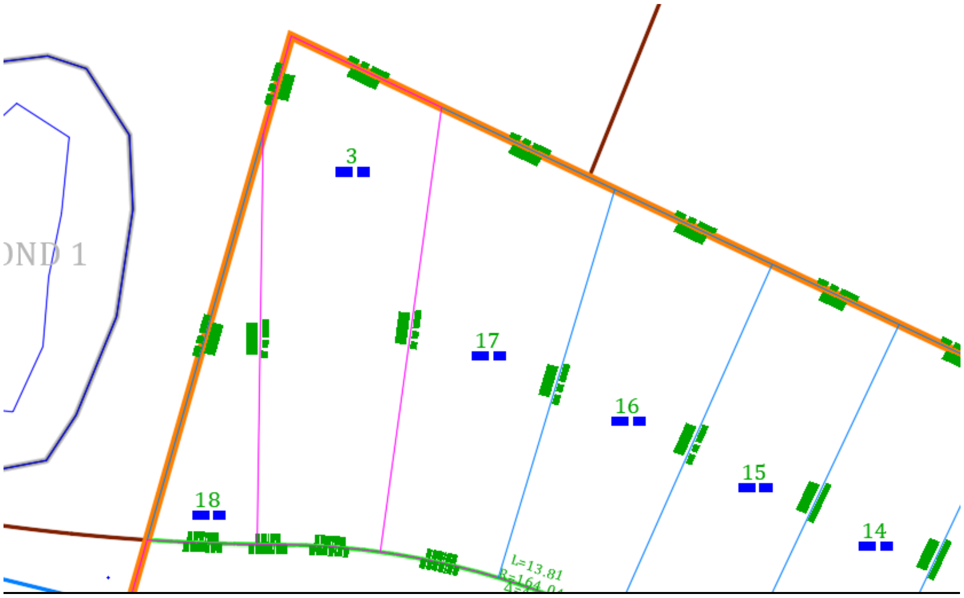

12. **Let's** keep pressing **Enter** to accept, while the lots are being created. In any case, we have already set our design criteria, so there little that we should worry about for now.

13. Notice how new lots are being created, numbered and parcel and segment labels are added automatically?

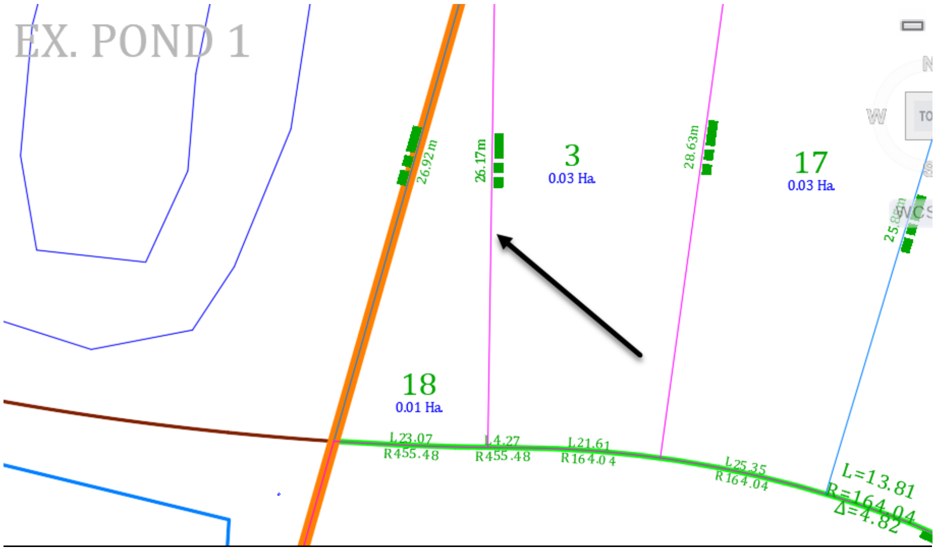



14. Keep pressing **Enter** until you get to the final lot

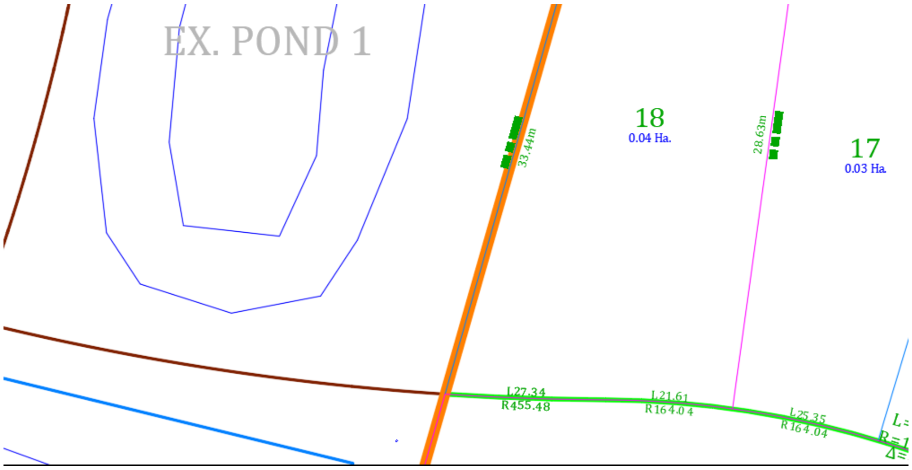
15. Press **Escape** to end the command




16. Because we have turned off the automatic mode, the last small parcel (**Lot #18**) is not redistributed and merged with **Lot #3**. Let's do a manual redistribution by selecting and deleting the shared parcel segment between the two lots.



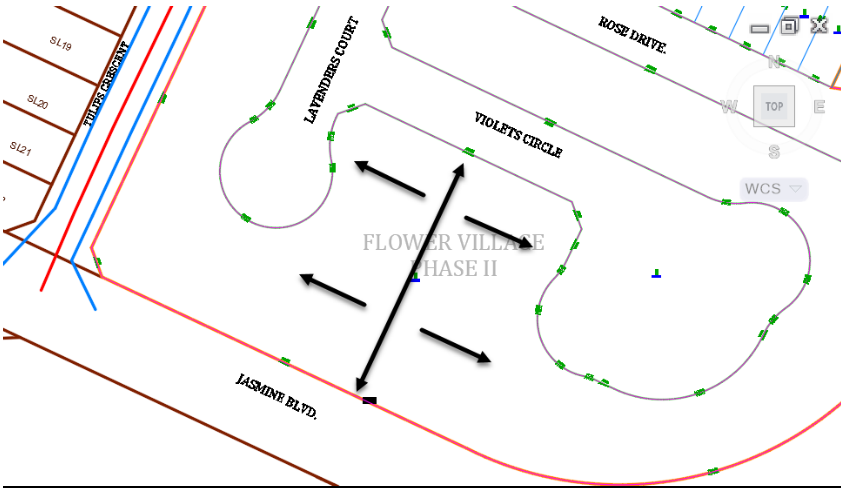

17. The two lots are now merged to create a bigger **Lot #18**.



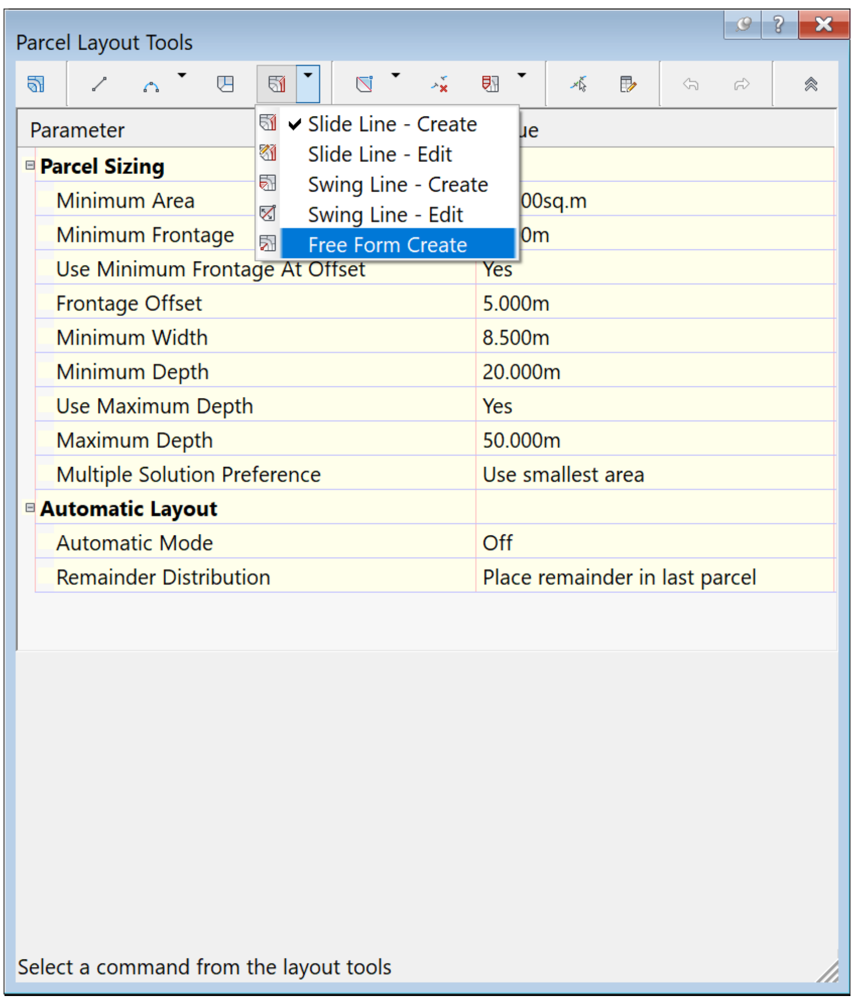

NOTES

### 6.4.2 Free Form Create

1. Now let's introduce another common lot creation command, Free Form Create. Looking at the preliminary lot layout, we may need to split the cul-de-sac areas, to the south of **Violets Circle.** This will allow us to have back-to-back abutting lots. If not, we will have lots that are too deep to maximize our land.

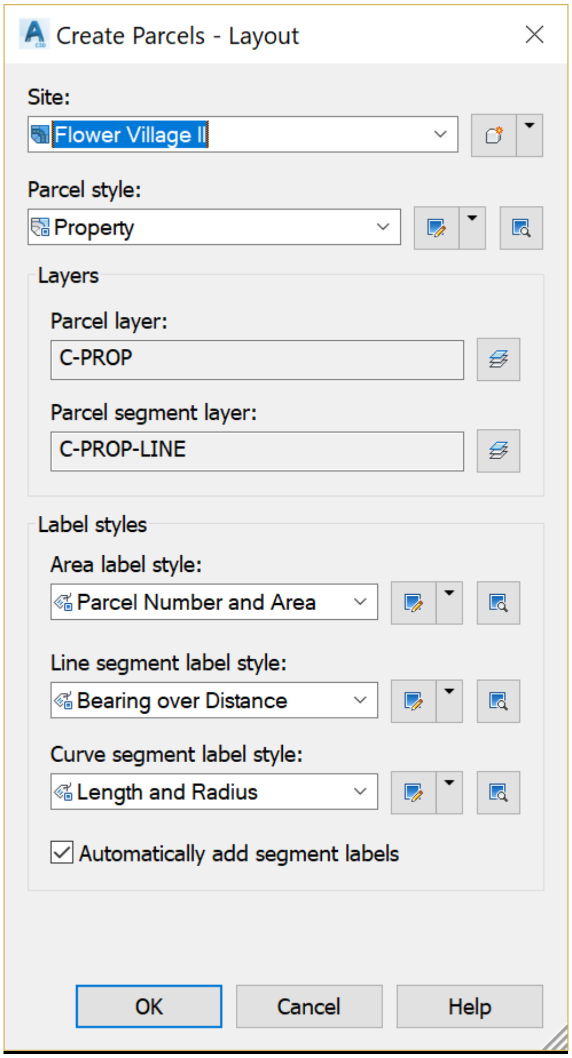



2. In case you have closed it, relaunch the **Parcel Creation Tool**. From there, launch the **Free Form Create** command.

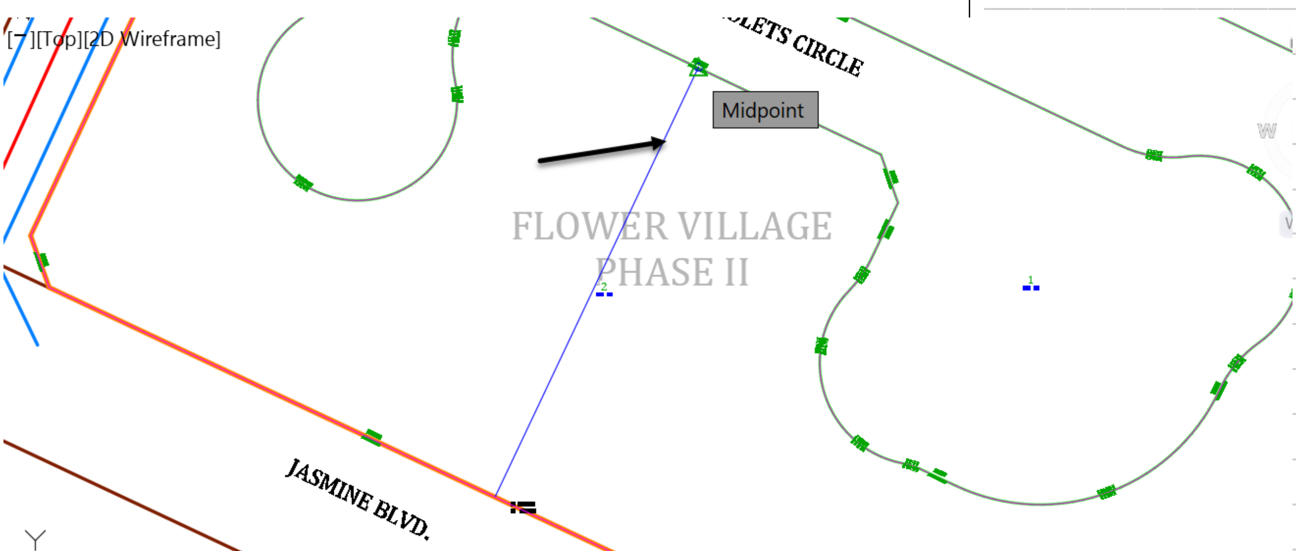



NOTES

3. Use the same settings we previously had when creating the parcels along **Rose Drive** and click **OK**.



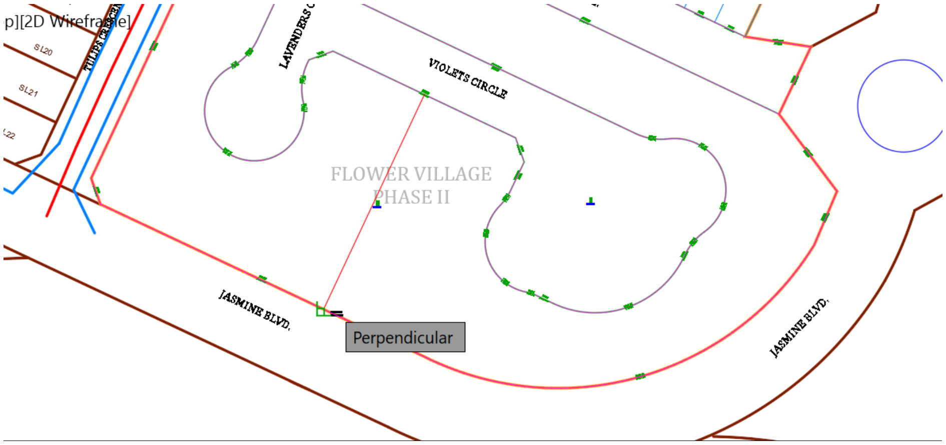

4. Adjust the Swing line to be perpendicular to **Violets Circle**. Then bring your cursor slightly close to the southern right-of-way line.




NOTES

5. Now, either activate the mid-point object snap from **Osnap** settings or type **Mid** at the command line and press enter. This will position the swing line exactly at the middle of the south right-of-way line.

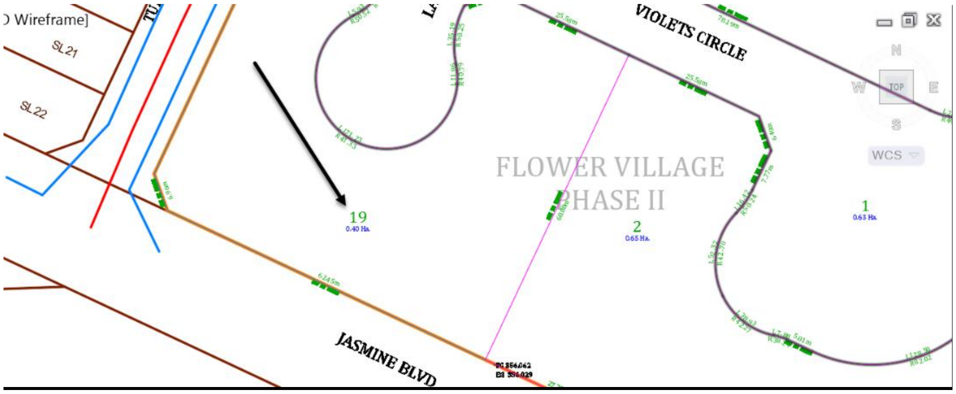
6. To specify the second point, either activate the **Perpendicular** from the **oSnap** settings or simply type **PER** at the command line. Then move the cursor closer to the south property line.




7. When the perpendicular snap sign appears, click on the line. The new segment line is created.

8. Press escape to exit the command.

9. If you zoom closer, you will notice that another parcel is added.

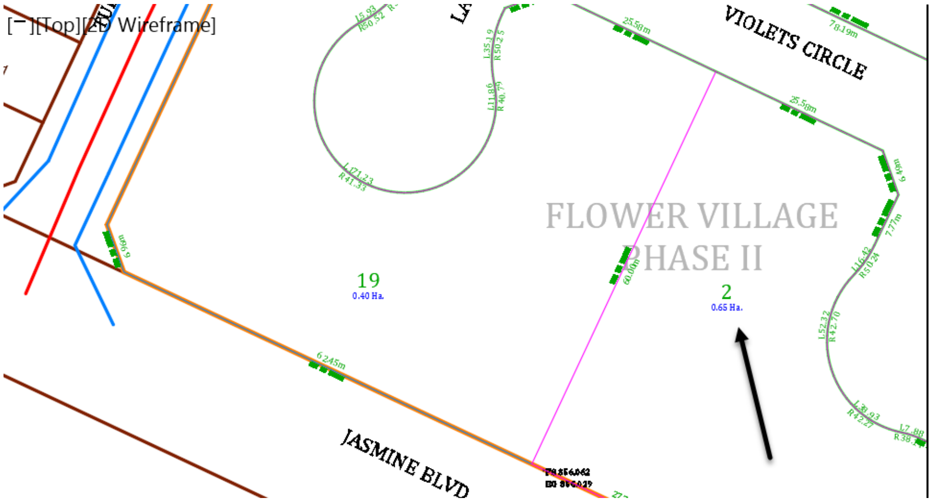



10. Now re-open, the **Parcel Layout Tools**. We have our last settings unchanged and ready to be utilized again. Let's make a minor adjustment and change the **Minimum Depth** to **15m** or **45ft,** instead of **20m** or **70ft** as previously set.

11. Next, launch the **Slide Line – Create** command.

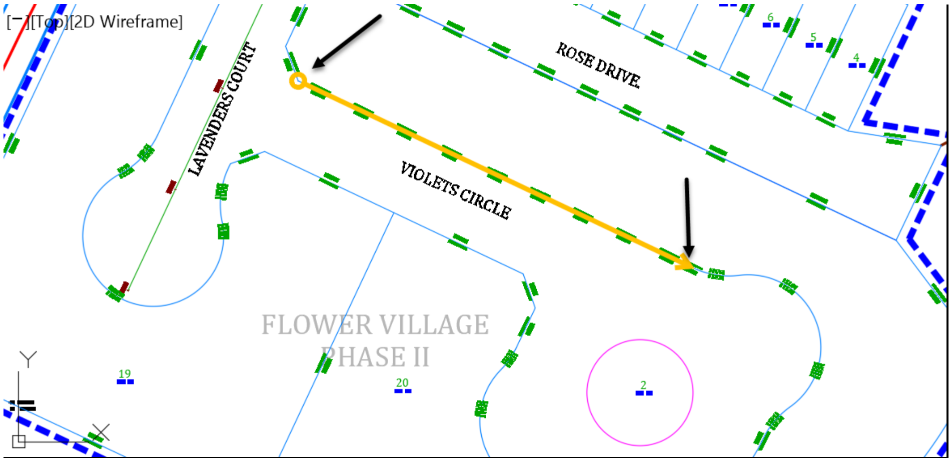
NOTES

12. For parcel to subdivide, click on parcel **Lot 2**.

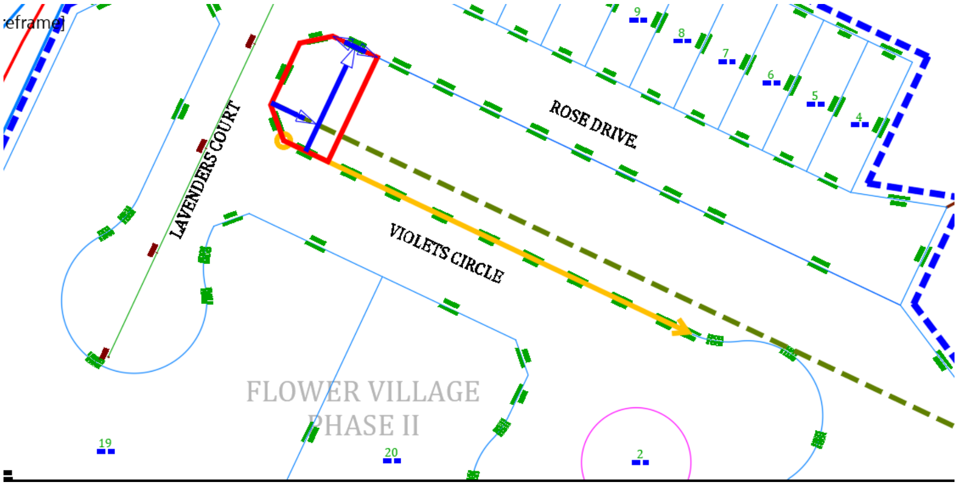



13. For the start point, click on the northwest corner of **Violets Circle**.

14. Next, run the frontage line along the **Violets Circle** right-of-way, from the top left corner to the start of the cul-de-sac.



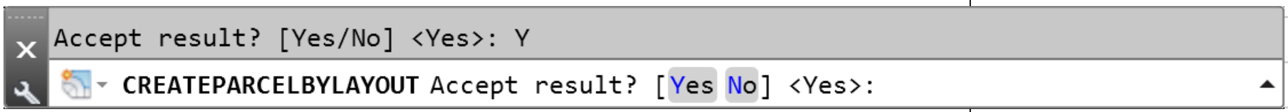

15. Press enter to accept a 90-degree **default angle**.



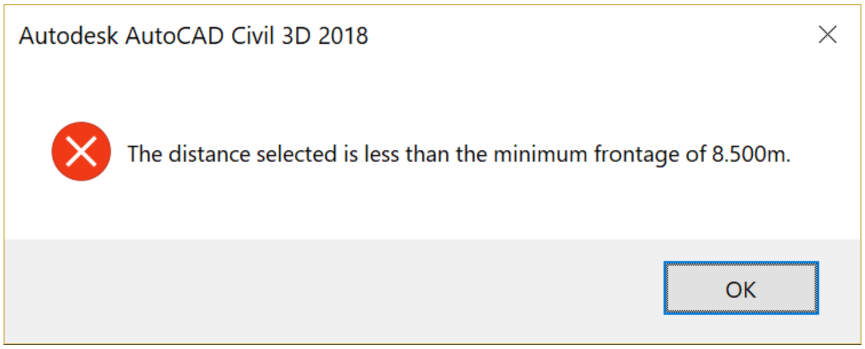

NOTES

NOTES

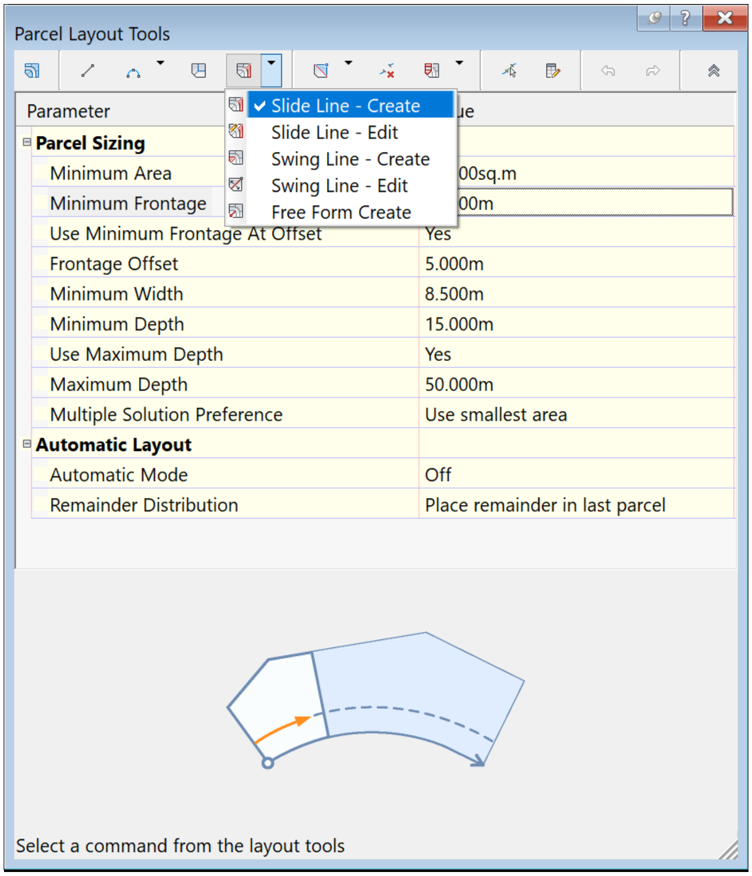
16. When prompted to accept the result, keep pressing **Enter** to accept.




17. When you have less than **8.5m** or **25ft** of frontage, you will receive a warning. If that happens, click **OK** to end the parcel creation and exit the current command.



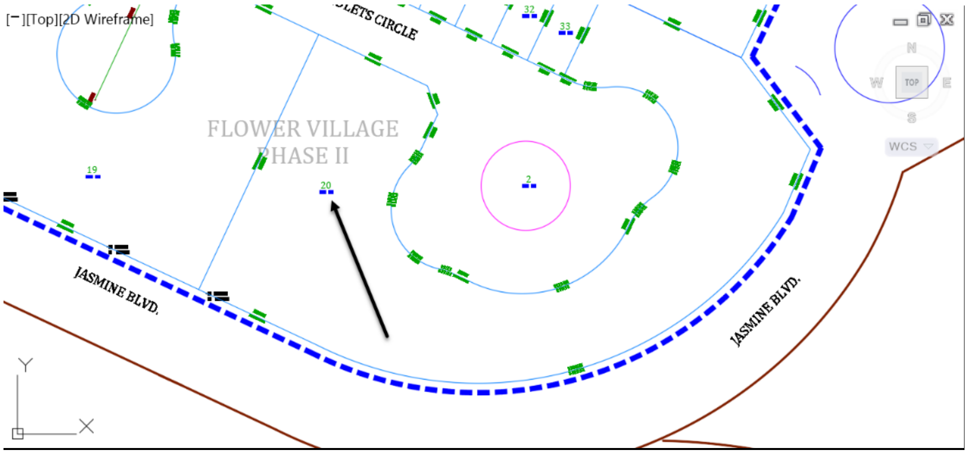

18. Now, we need to subdivide the cul-de-sac area. We want to create a few high-end single-family units. So, we will make the lot a little wider and bigger overall. To do this, we need to adjust our parameters. Return to the **parcel creation tools** or reopen it, if it's closed. Now, change the minimum width to **20m** or **65ft**, and the depth to **15m** or **50ft**. Then, run the **slide line – create** command



NOTES

19. Accept the previous settings for parcel creation.

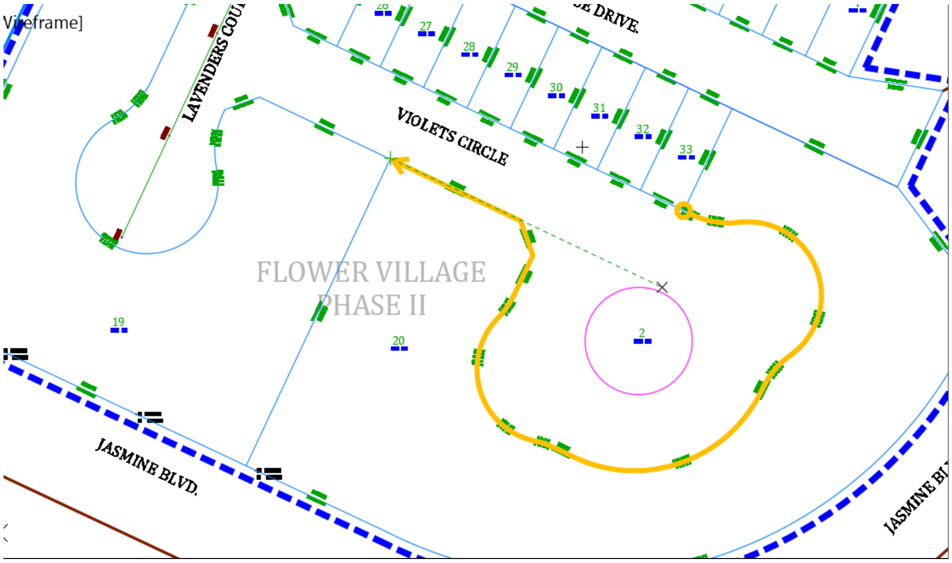
20. When prompted to select the parcel to be subdivided, select the big parcel around the cul-de-sac.

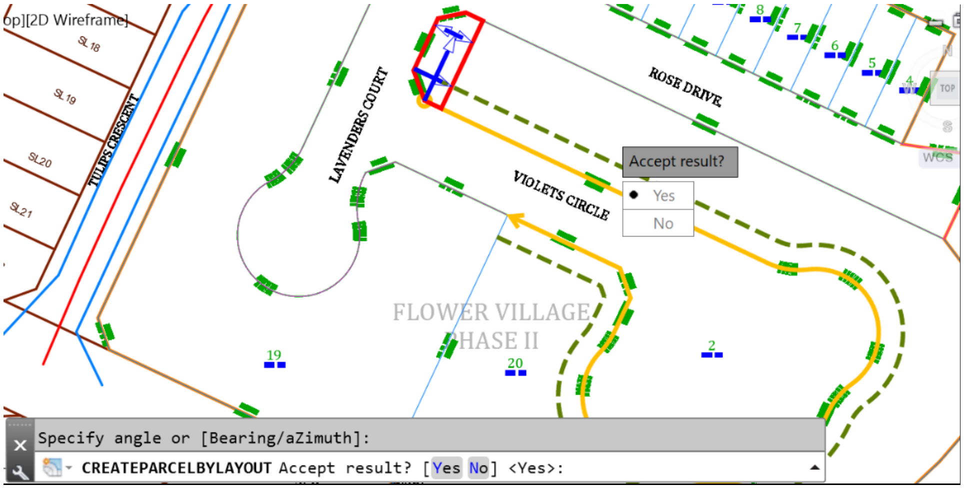



21. A new parcel is created. When you are prompted for angle, press **Enter** to accept perpendicular lot segments.

22. Then,

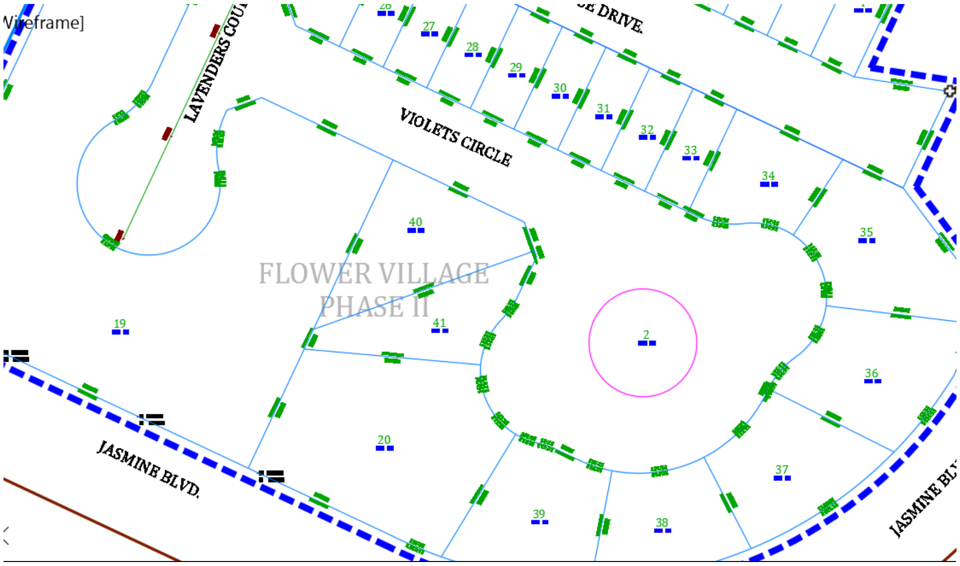
- click at the start of the cul-de-sac, where we ended the previous lots before changing the parameters,
- next, click at the end of the parcel to be subdivided.






23. Press enter to accept the default 90-degree angle

24. Keep pressing **Enter** all the way, until the last parcel is created.



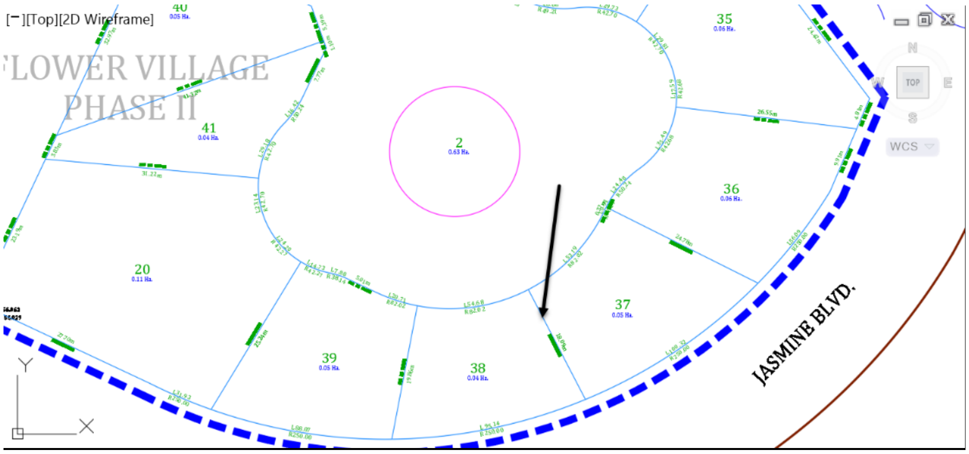

NOTES

NOTES

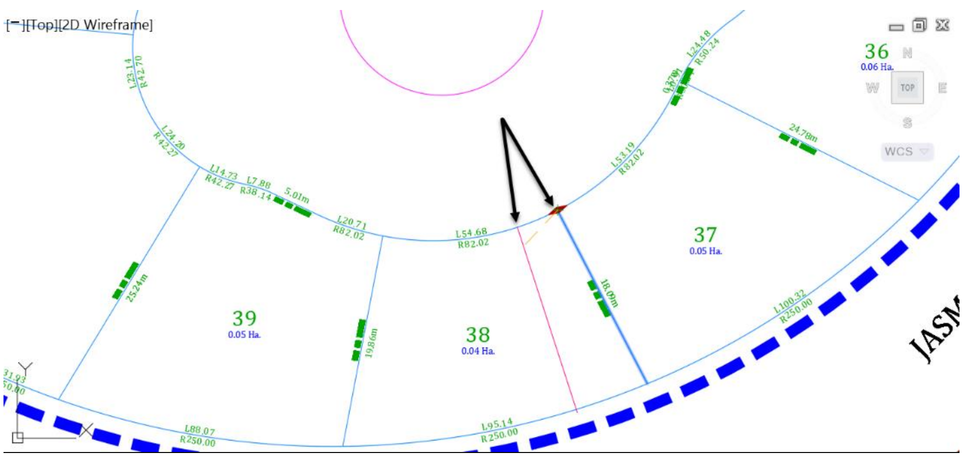
### 6.4.3 Parcel Adjustment

1. We have subdivided the **Violets Circle** cul-de-sac. However, we need to adjust a couple of the parcels in the cul-de-sac, to make it a little wider to accommodate a bigger house.

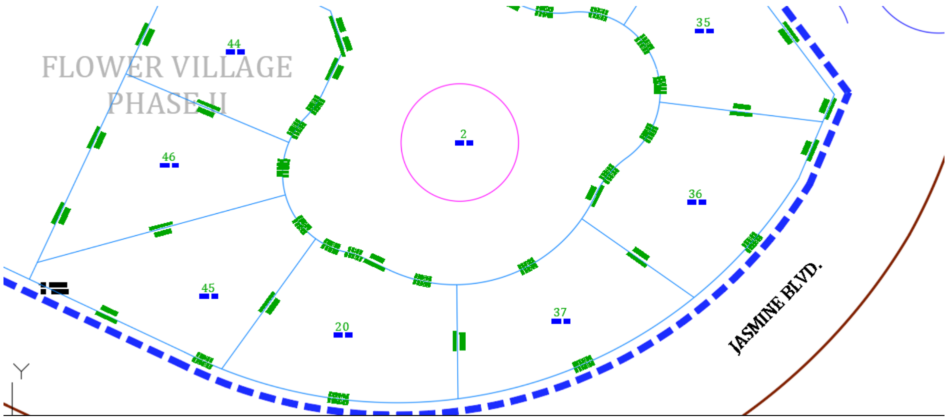
2. Click on the lot line for the smaller cul-de-sac lots.



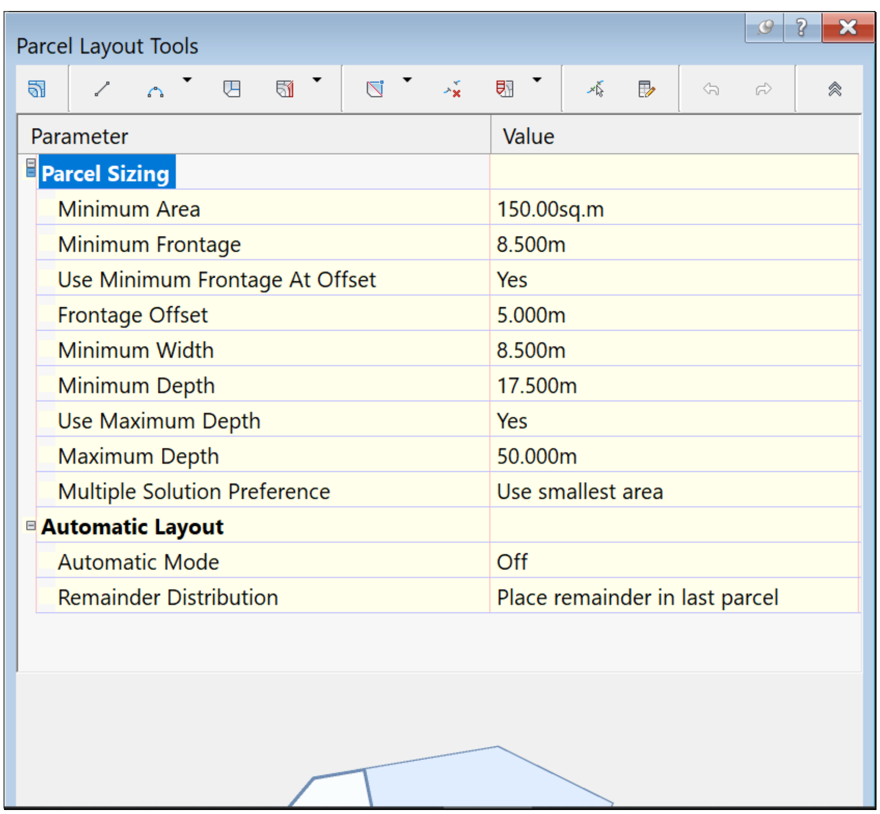

3. A blue handle will appear at the start of the lot segment. What we need to do here is to click on the small blue grip and gently slide it in the direction we want to widen the lot in. Don't worry; the line will always stay perpendicular to the frontage.



4. Use this manual adjustment to adjust the lots in the cul-de-sac. A good hint is to move the lot segment until the lots are around 0.06 – 0.07 Hectares or 0.15-0.17 Acres. When it's all said and done, we should have the single-family lots around the cul-de-sac, a little bigger than the townhouses on the streets. Note that you may need to delete the last parcel segment line in of the corner lot. The lot numbers are not currently in sequential order. However, don't worry about it, as we will see how to re-order the parcels and put everything in a neat sequential order.




5. Now let's start creating the parcels fronting **Lavender Court**. Launch the **Parcel Creation Tools**. Reset the minimum frontage to **8.5m or 30ft**. Because of the cul-de-sac ahead, we will also set the minimum depth to **17.5m or 60ft** instead of the typical **20m or 65ft.**


Parcel Layout Tools

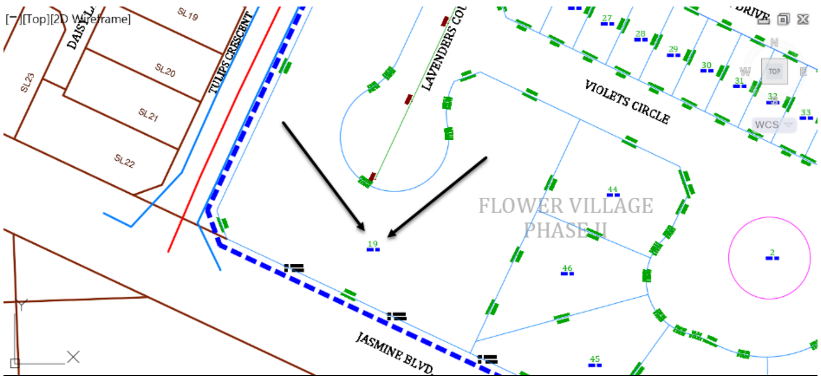
| Parameter | Value |
|---|---|
| **Parcel Sizing** | |
| Minimum Area | 150.00sq.m |
| Minimum Frontage | 8.500m |
| Use Minimum Frontage At Offset | Yes |
| Frontage Offset | 5.000m |
| Minimum Width | 8.500m |
| Minimum Depth | 17.500m |
| Use Maximum Depth | Yes |
| Maximum Depth | 50.000m |
| Multiple Solution Preference | Use smallest area |
| **Automatic Layout** | |
| Automatic Mode | Off |
| Remainder Distribution | Place remainder in last parcel |

NOTES

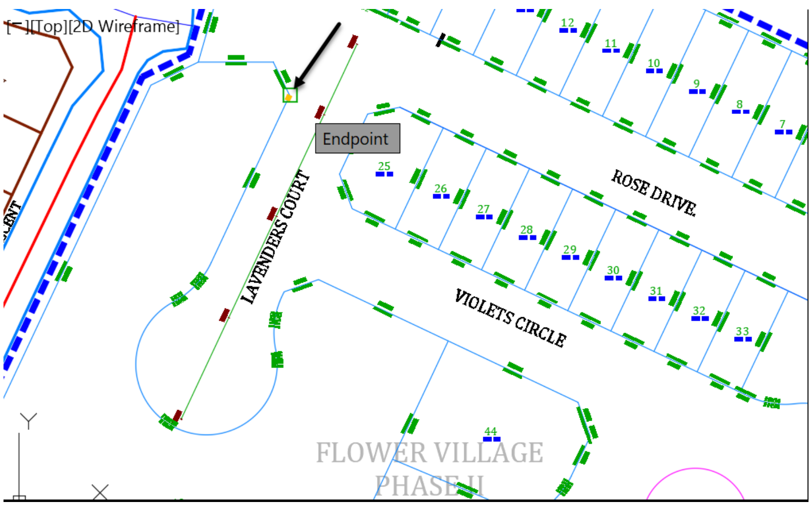
NOTES

6. Accept the default settings in the **Create Parcel – Layout** window and click **OK**

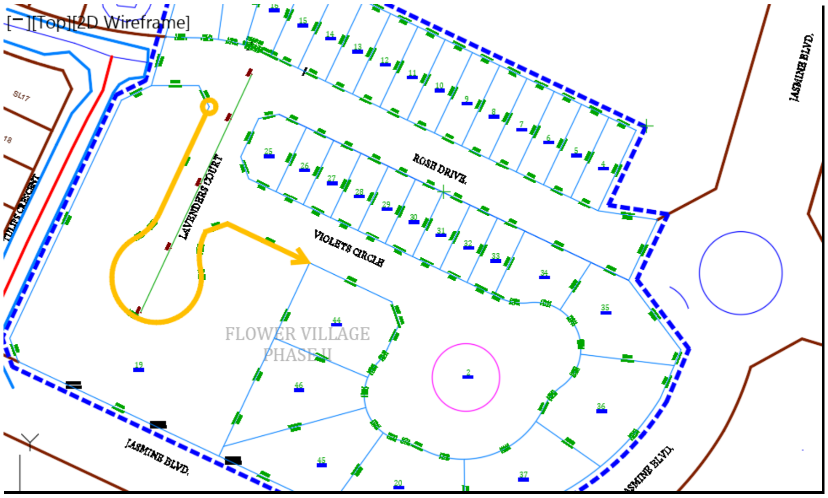
7. When prompted to choose the parcel to be subdivided, click on the remaining big lot.




8. For the start point of the frontage, click on the northwest start point, in the vicinity of the intersection with **Rose Drive**.




9. Slowly slide the frontage line southward, then around the cul-de-sac, all the way to the last parcel created, on **Violets Circle**.



NOTES

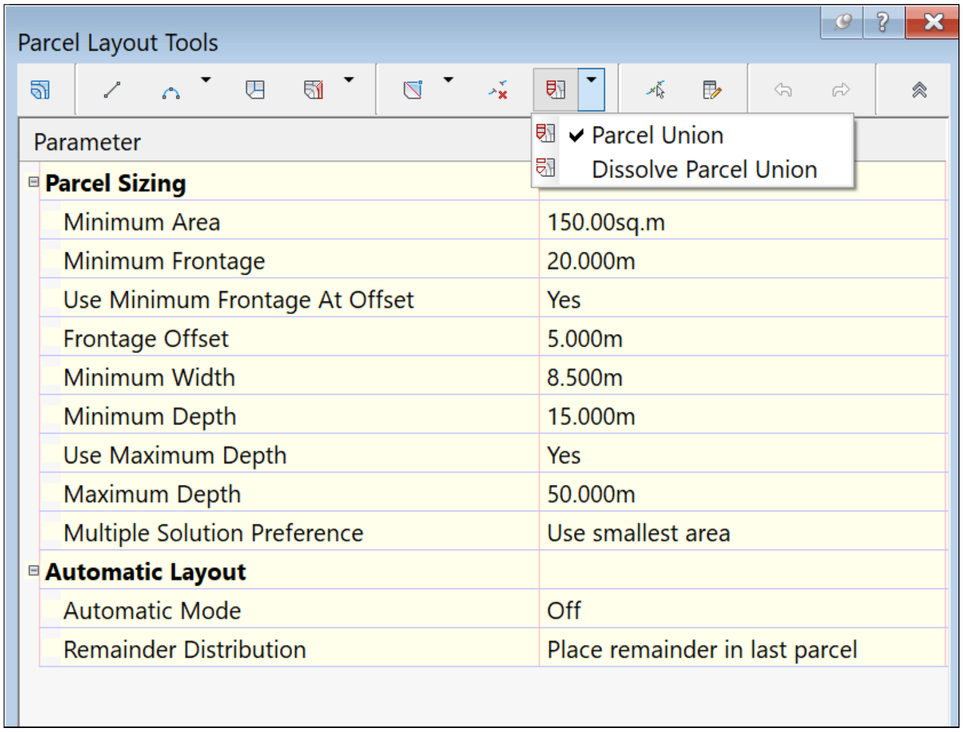
10. When prompted to specify the angle, press **Enter**. The first lot, a corner lot, is created in the northwest corner. Keep pressing enter until you create the last parcel, or you get a notification that no more room is left to add another parcel.

11. The newly created parcels are shown. Select to delete or adjust, by using the handles on any parcel that is too big, too small, irregularly shaped or located on the wrong side of the road frontage. In the end, you should be able to obtain ordinary shapes parcels with areas around **0.05-hectares** or **0.12 acres**.

NOTES

## 6.5 Parcel Union and Dissolve

1. When two adjacent parcels are too small or need to be combined for any reason, we can merge them. We can achieve that by either deleting the common lot line or using the parcel Union command from the **Parcel Layout Tools**. If for some reason, we changed our mind, we can easily **Dissolve a Parcel Union** and separate them.

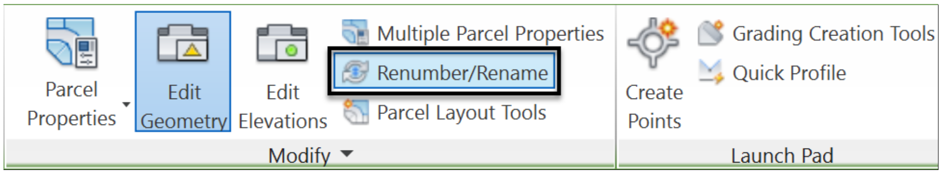



## 6.6 Parcel renumbering

To ease plan readability, it is important to number the parcels in sequential order. Once we are satisfied with the size of the parcels and the layout of the segments, we can renumber the parcels.

To renumber the parcels, in an ascending or descending order we can,

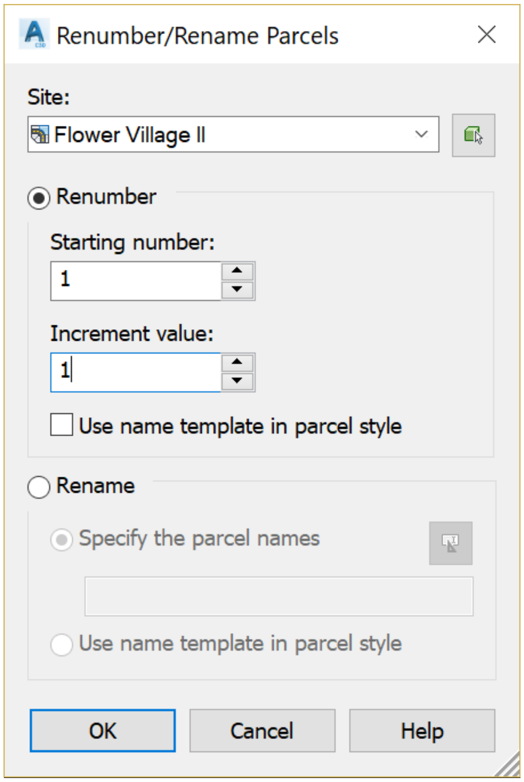
1. First, click on any parcel. We want to bring up the parcel tools on the contextual ribbon. Then run the **Rename/Renumber** command.




2. In the **Rename/Renumber Parcels** windows reset the **Starting number** to **1** and use an increment value of **1**. You can use an increment value of **2** if you are looking to use odd numbers on one side of the street and even numbers on the other side. In that

NOTES

case, you would need to restart the number with **2**, when you are renaming the even numbers.



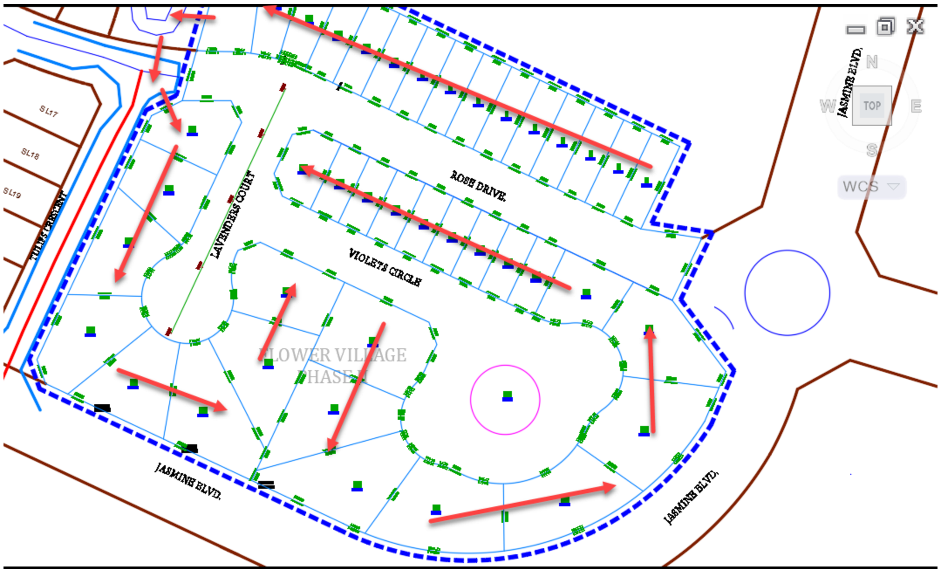

3. Click **OK.**

4. Click inside of the first lot, the one to the northeast of **Rose Drive**.


5. Continue to click inside the parcels, going westward on **Rose Drive**, then south and cross the lots west of **Lavender Court.** Then go around the two cul-de-sacs and finally along the north right-of-way of **Violets Circle**. When doing this, you need to be careful not to cross any street. Otherwise, the right-of-way parcel is going to be numbered as well. When it's all said and

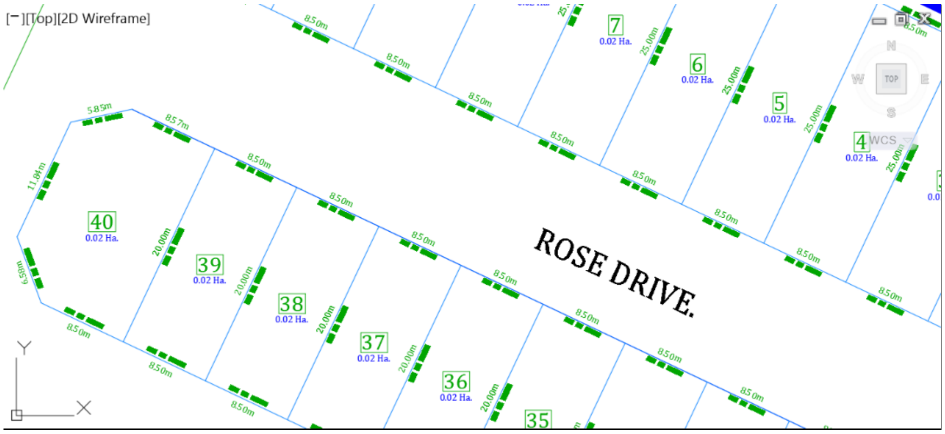
NOTES

done, your parcels should be numbered from **1** to **40**, give or take, depending on your adjustments.



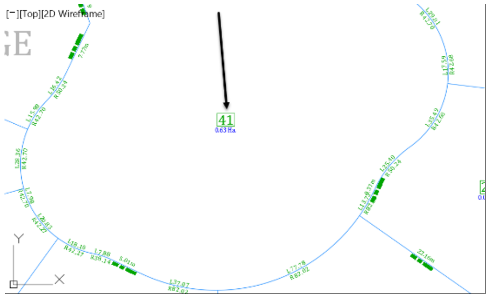

6. Once you reach the last parcel, the one in the northeast corner of **Violets Circle**, **Enter** twice at the command line.

7. Subsequently, the parcels are renumbered in a neat order.

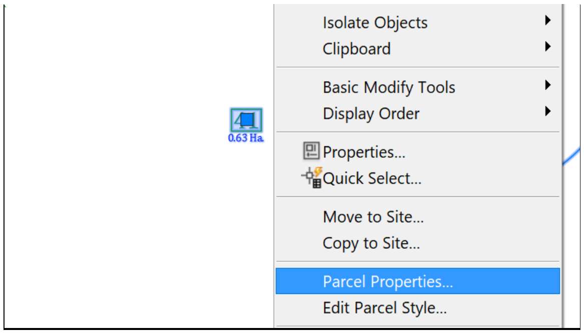


## 6.7 Parcel Styles

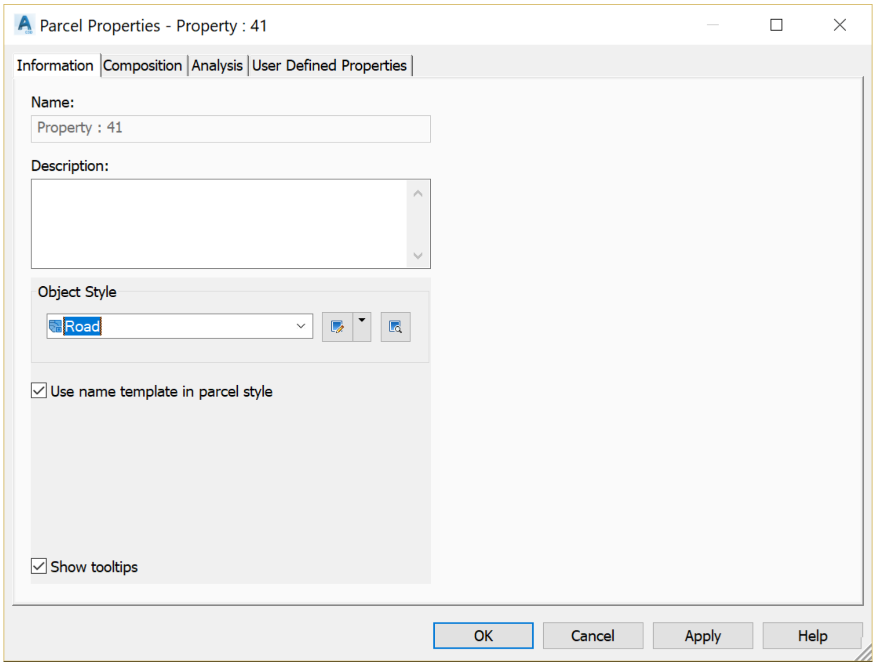
1. Now, let's assign a different parcel style to the road right-of-way. Most of the time we don't necessarily need to show road areas on a plan, much less number them. Select the right-of-way, in the cul-de-sac of **Violets Circle.**



2. Next, **Right-click** on the right-of-way parcel, the big one in the **Violet Circle** cul-de-sac area, and select **Parcel Properties.**




3. Change the parcel **Object Style** to **Road.** That will apply a bolder line style to make the roadway areas stand out from residential lots.



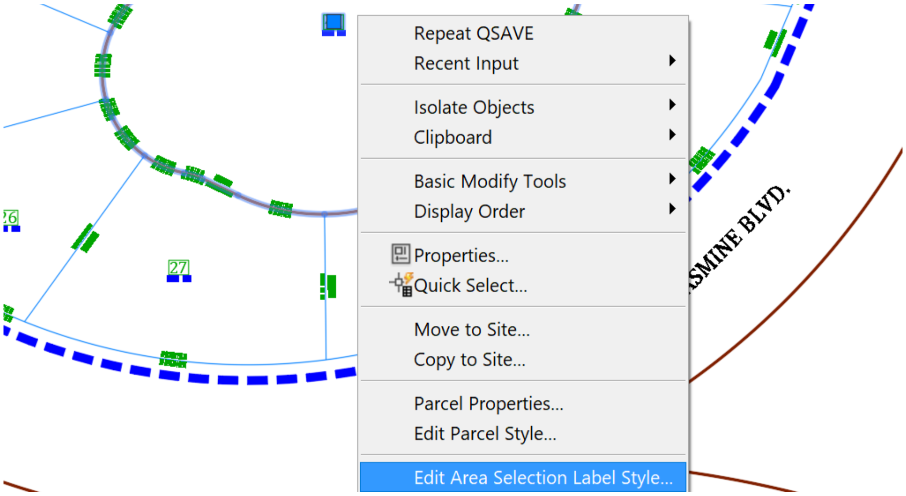

4. Click **OK.**

NOTES

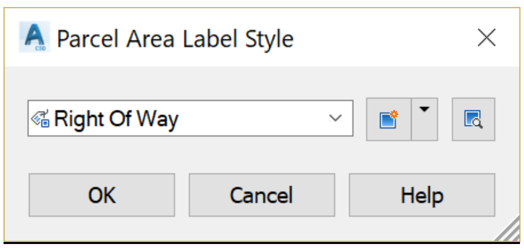
NOTES

## 6.8 Parcel Label Style

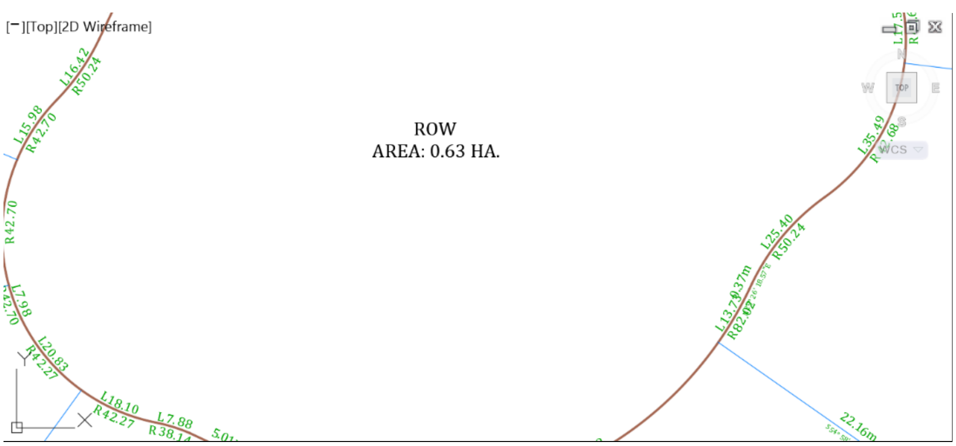
1. Now, we need to change the label style, the annotation that displays the road area information. Select the road parcel, right-click, and click **Edit Area Selection Label Style**.




2. Change the label style to **Right Of Way.**




3. We have now applied a **right-of-way** label style to the street area.



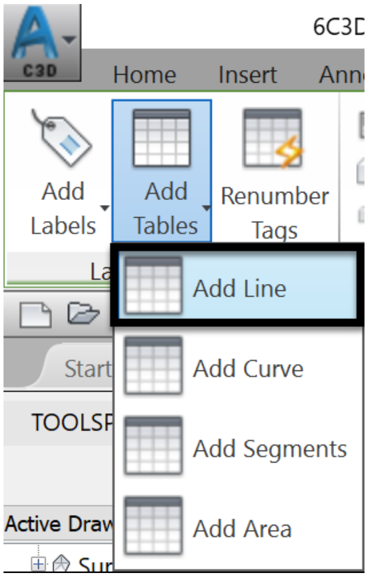
NOTES

## 6.9 Parcel Tables

At some point along the development process, we will be required to create a subdivision plat. In addition, we may also be asked to provide a real estate agent with a list of parcels, along with areas and lot lines. To do that, we need to create parcel tables.

1. In the drawing area, select a parcel.

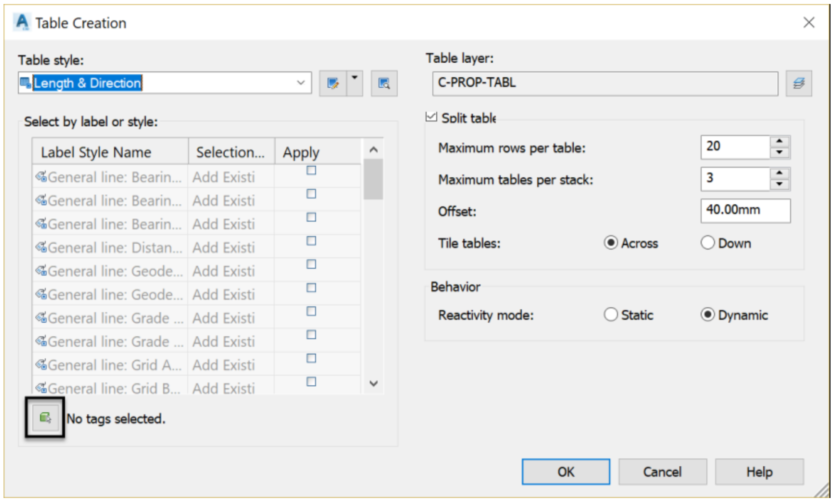
2. From the ribbon, click on **Add tables**, then **Add Line.**




3. We have roughly 40 plots. We want to organize our table in four columns of 8 lines. To do that, in the **Table Creation** windows we need to split the table as needed.

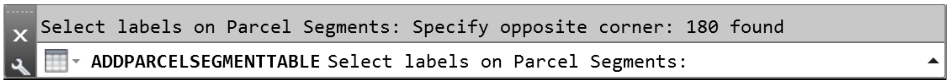
4. We also need to keep the tables **Dynamic**. That option automatically updates the tables, should anything change with the parcels.

5. We also want to convert the segment labels to tags. That's a very good thing to do since it improves plan readability and presentation. Instead of having all the bearing, distance and radius information overcrowd the drawing area, we can just convert them to tags. We can then reference them in a table. To do that, click on the **No Tag selected** button.



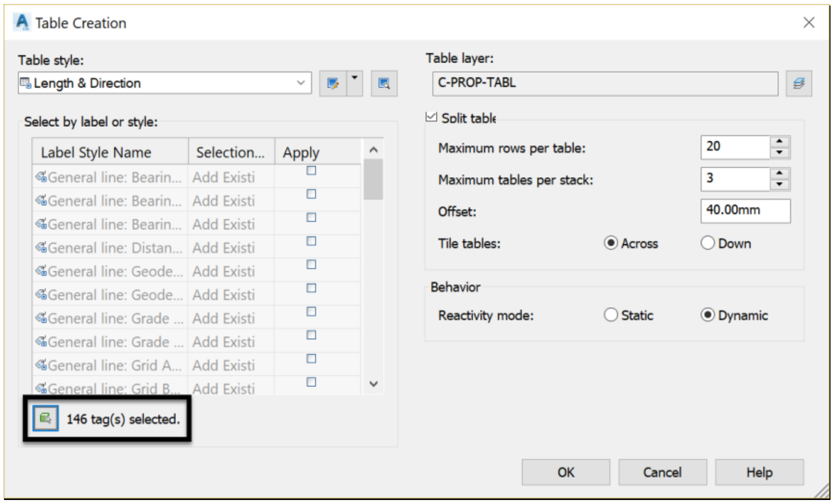

6. Zoom out, in the drawing, and make a rectangular selection. The lower left and the upper right corners of the selection should completely cover the proposed project area.

7. You will be notified at the command line of the total number of objects selected.




8. Then press **Enter** at the command line to accept the selection. In the new window, click on **Convert all selected label styles to tag mode**. It's self-explanatory, but this will effectively convert all labels to tag mode.

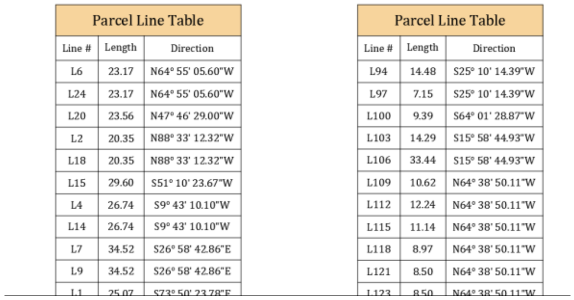
9. We are now brought back to the **Table Creation** window, where we can notice the total number of labels added.




NOTES

10. Click **OK.**

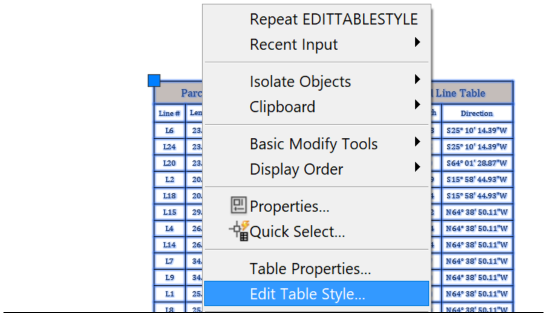
11. Click on an empty spot in the drawing to view a complete list of parcel segment tables.



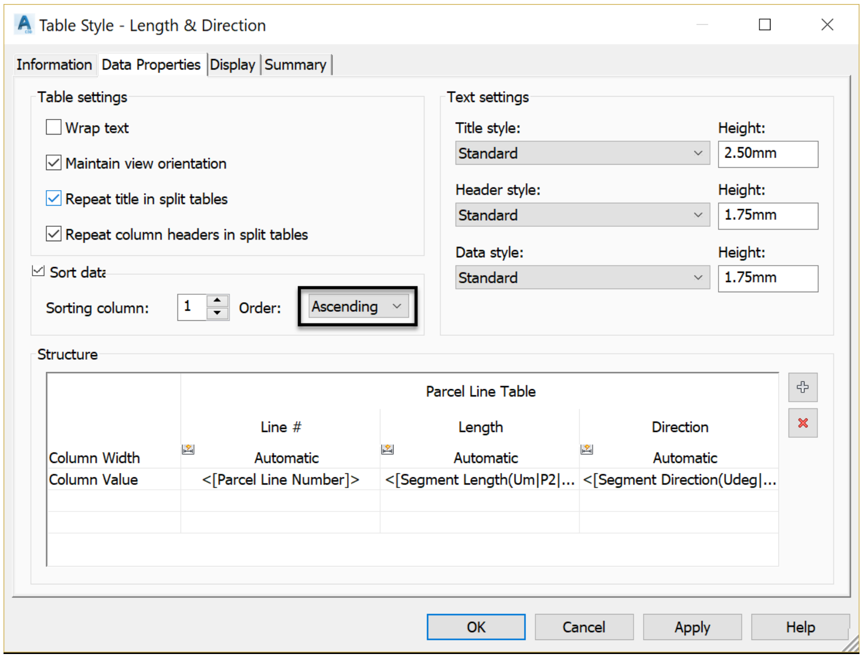
| Parcel Line Table | | |
|---|---|---|
| Line # | Length | Direction |
| L6 | 23.17 | N64° 55' 05.60"W |
| L24 | 23.17 | N64° 55' 05.60"W |
| L20 | 23.56 | N47° 46' 29.00"W |
| L2 | 20.35 | N88° 33' 12.32"W |
| L18 | 20.35 | N88° 33' 12.32"W |
| L15 | 29.60 | S51° 10' 23.67"W |
| L4 | 26.74 | S9° 43' 10.10"W |
| L14 | 26.74 | S9° 43' 10.10"W |
| L7 | 34.52 | S26° 58' 42.86"E |
| L9 | 34.52 | S26° 58' 42.86"E |
| L1 | 25.07 | S73° 50' 23.78"E |

| Parcel Line Table | | |
|---|---|---|
| Line # | Length | Direction |
| L94 | 14.48 | S25° 10' 14.39"W |
| L97 | 7.15 | S25° 10' 14.39"W |
| L100 | 9.39 | S64° 01' 28.87"W |
| L103 | 14.29 | S15° 58' 44.93"W |
| L106 | 33.44 | S15° 58' 44.93"W |
| L109 | 10.62 | N64° 38' 50.11"W |
| L112 | 12.24 | N64° 38' 50.11"W |
| L115 | 11.14 | N64° 38' 50.11"W |
| L118 | 8.97 | N64° 38' 50.11"W |
| L121 | 8.50 | N64° 38' 50.11"W |
| L123 | 8.50 | N64° 38' 50.11"W |

12. To make the tags easier to reference between the drawing and the tables, we can sort the **Tables** in an ascending or descending order. This can be done by right-clicking first on the table, then select **Edit Table Styles**.




13. In the next window, make sure the **Data Properties** tab is activated. Then, check **Sort Data**. After that, choose to sort by **Ascending** order. You can also change the **Text settings** and adjust styles and heights for the table title, header or cells.




NOTES

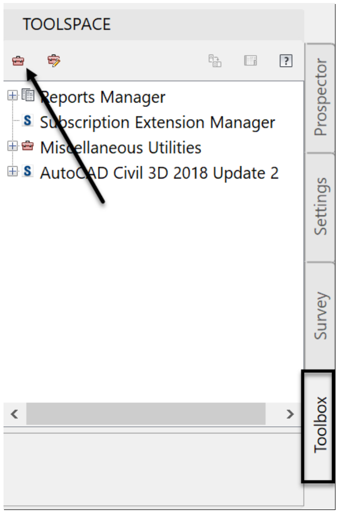
NOTES

14. Click **OK.** Note also that the table styles and colouring schemes can be changed on the display tab.

15. Additionally, we can generate specific reports for team members who do not use Civil 3D in their daily work. A listing of the subdivision plat information can be created in the form of excel, text document or even HTML files.

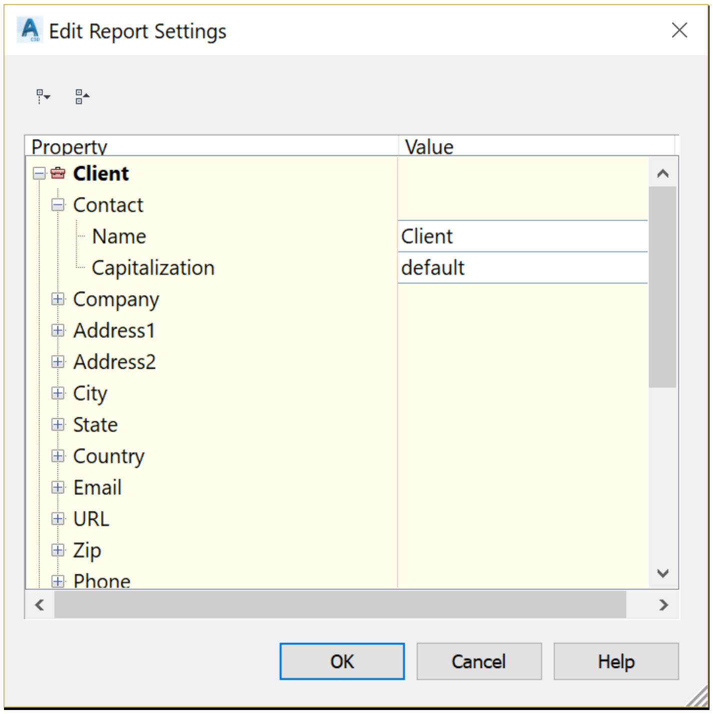
16. Simply click on the **Toolbox**, not to be confused with the **Toolspace**. A quick reminder that the **Toolbox** is part of the **Toolspace**.

17. Click the **Report Settings** button to adjust your preferences.

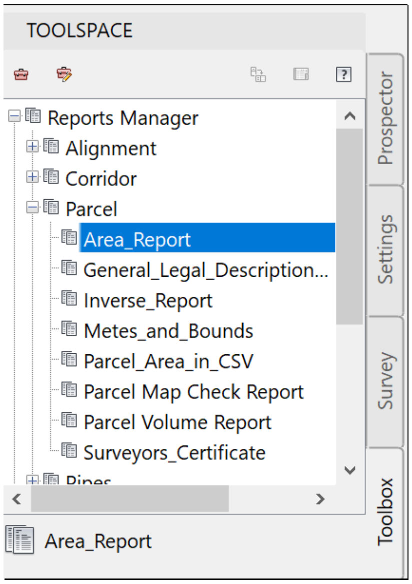



18. Adjust the necessary information such as the client's name and contact data.

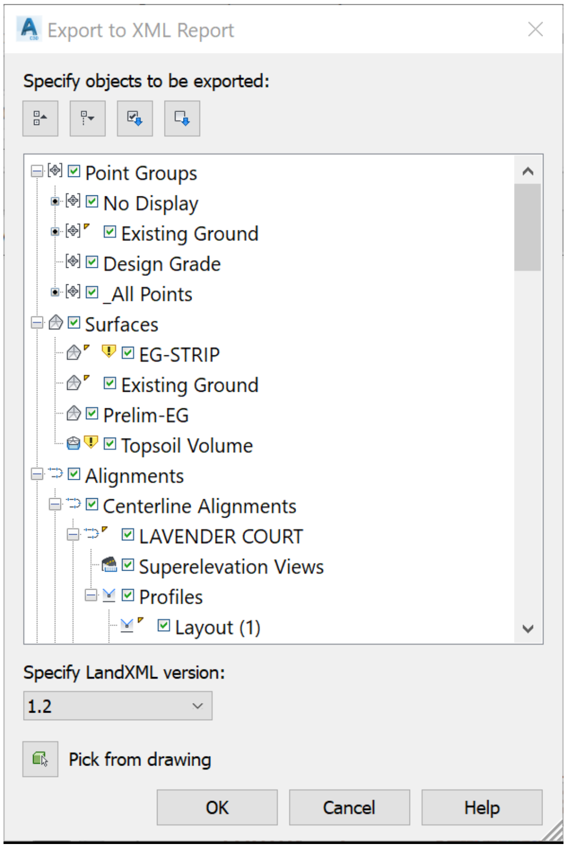
19. When done, click **OK.**



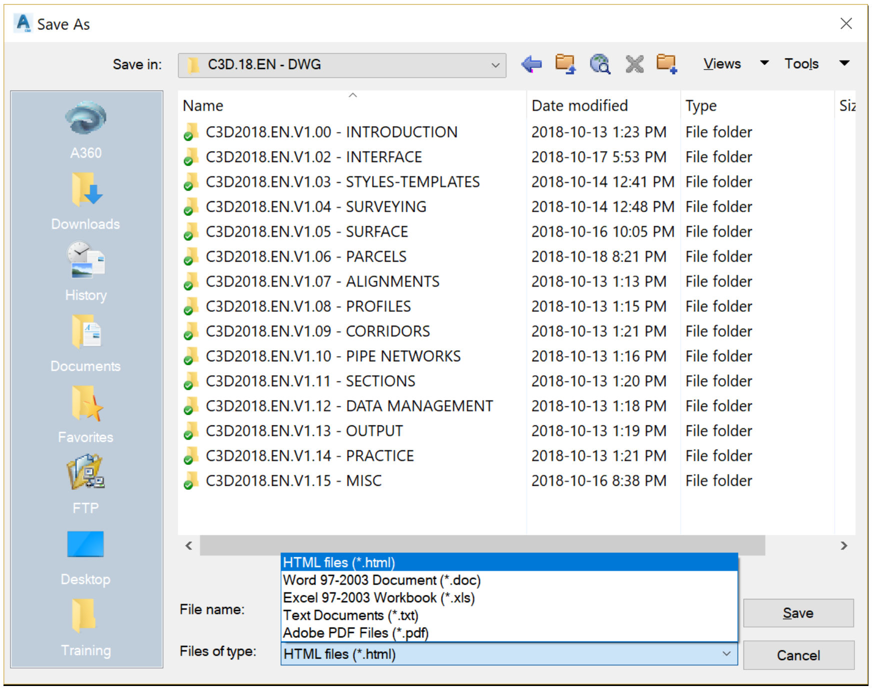
20. Make sure the **Report Manager** tree structure is open by clicking on the "**+**" sign. Then, navigate to the parcel line and click on the "**+**" sign. Afterward, double-click on **Area Report**, for instance.



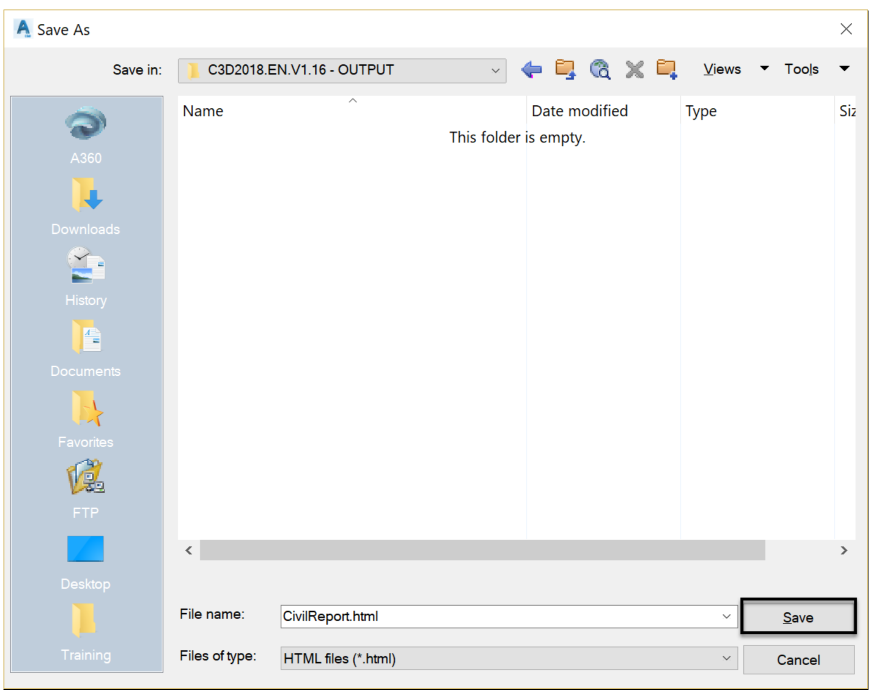

21. In the new window that opens, you will be presented with options to export information to an XML file. Don't be overwhelmed by the high number of boxes you can check or uncheck. You have already specified that you want a parcel **Area Report**. So, go ahead and click **OK**.



22. Choose a directory to export the Output file. Before saving, you have an option to change the file format to either HTML, Word document, Excel sheet, text or pdf.




23. Click to **Save** the report in the project folder.




NOTES

24. We now have an HTML or CSV report that we can attach to a development site plan.

# Your Company Name

## 123 Main Street

## Suite #321

## City, State 01234

**Parcel Area Report**

**Client:** Client Company

**Project Name:** C:\Users\Infratech.Civil\Dropbox_INFRATECH\Training\AutoCAD Civil 3D\C3D18\C3D18-EN\C3D.18.EN - DWG\C3D2018.EN.V1.06 - PARCELS\6C3D2018.EN.V1.06 - Parcels by Layout.dwg

**Project Description:**

**Report Date:** 2018-10-19 8:08:46 AM

**Prepared by:** Preparer

| Parcel Name | Square Meters | Hectares | Perimeter (m) |
|---|---|---|---|
| Property : 1 | 212.500 | 0.021 | 67.000 |
| Property : 2 | 212.500 | 0.021 | 67.000 |
| Property : 3 | 212.500 | 0.021 | 67.000 |
| Property : 4 | 212.500 | 0.021 | 67.000 |
| Property : 5 | 212.500 | 0.021 | 67.000 |
| Property : 6 | 212.500 | 0.021 | 67.000 |
| Property : 7 | 212.500 | 0.021 | 67.000 |
| Property : 8 | 212.500 | 0.021 | 67.000 |
| Property : 9 | 212.500 | 0.021 | 67.000 |

NOTES

NOTES

# 7 ALIGNMENTS

## 7.1 Introduction

Alignments allow us to represent or design linear objects such as roads, sidewalks, railway tracks or survey baselines.

They are also useful for the creation of design items such as profiles, road corridors, cross sections, or earthworks volumes.

An alignment is typically composed of lines, curves, spirals or a combination of these elements. They can be created from an already existing entity (mostly lines, arcs or polylines) or by using the Civil 3D **Alignment Layout tools**.

This chapter kicks off the process of the detailed design of the subdivision. Processing the survey data, creating alignments and laying out parcels sets up the table for the ensuing design. The process of designing a subdivision street will include, among other components:

1. The creation of **Alignments.** They provide the two-dimensional information needed to determine the path that the road will take.
2. The design of **Profiles.** They give us the 3rd dimension, the elevation information at each point along the alignment.
3. And, the design of a **Typical Cross-section.** It tells us the different sub-entities, such as pavement, sidewalks, and curbs, that compose the road.

Now suppose that the project is located in the municipality of **Flower Bay**. We must then design the road following the local standards as required for a **14m** or **46ft** local road right-of-way. That includes, on both sides,

- A **7m** or **25ft** road travel way or pavement.
- A **20cm** or **6in** curb.
- A **1.5m** or **5ft** sidewalk.
- A **1.8m** or **6ft** green space.

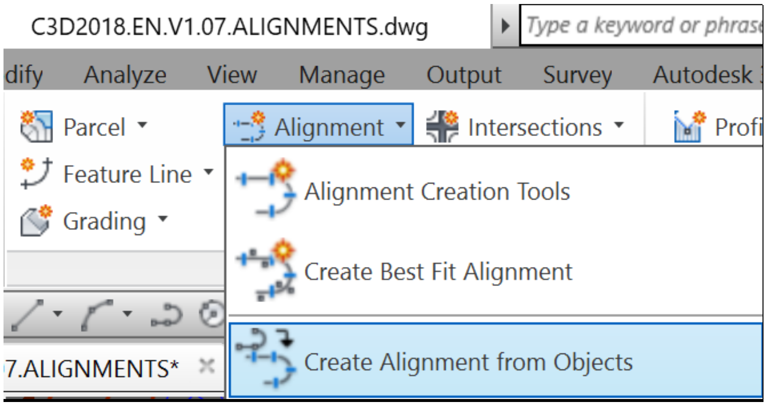
In Civil 3D, we have two main ways to create an alignment: From **existing objects** (lines and polyline) or by layout (using the **Alignment Creation tools)**. Creating civil 3D components **from object** or **by creation tools** is a recurring theme. As we have done for surfaces, we will use these two methods for alignments, profiles, pipe networks, and feature lines.

## 7.2 Alignment from objects

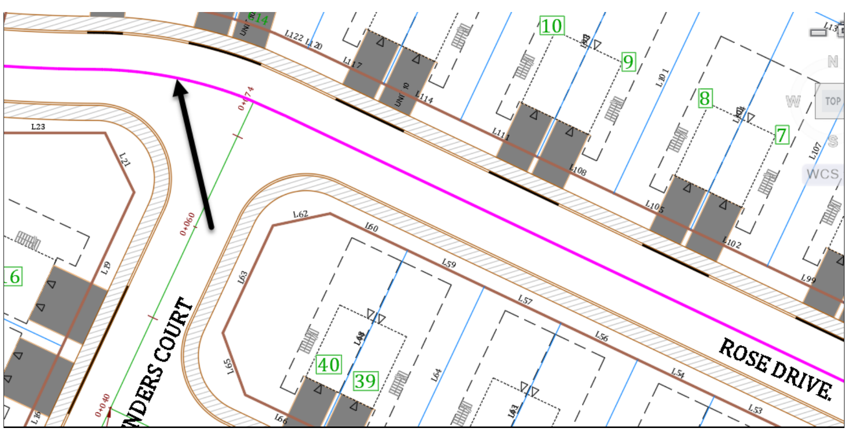
Let's get started with creating alignments from objects.

1. Open the **07.01-Alignments dwg** file in **Lesson 07** practice folder.

2. From the **Ribbon**, select **Create an alignment from objects**.



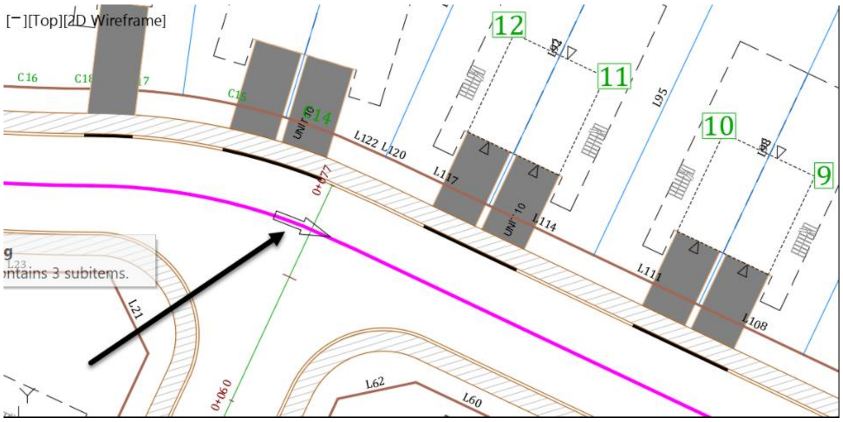

3. Select the magenta polyline at the centerline of **Rose Drive**. The main site access road and the baseline were already established during **Phase 1** of the project. Click towards the western end of the polyline so that the orientation of alignments stations is from West to East. It is always recommended to have alignments run West to East and South to North, to make plans and profiles more readable in layouts.



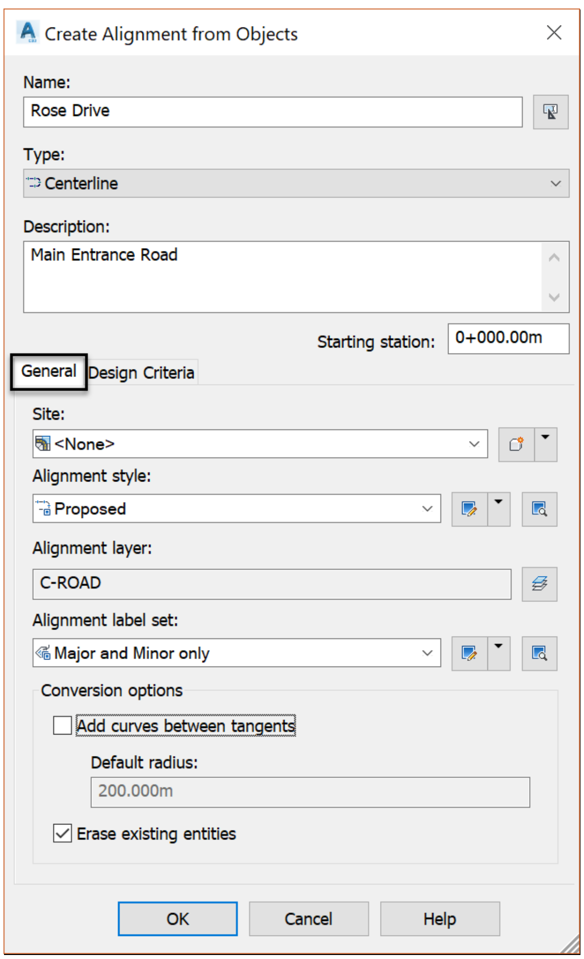

NOTES

NOTES

4. Afterward, press **Enter** at the command line twice. The second time would be to accept the alignment direction. If you missed it or have the alignment in the wrong direction, don't worry and keep working. We will be able to change the direction of the alignment.




5. Next, in the new window,

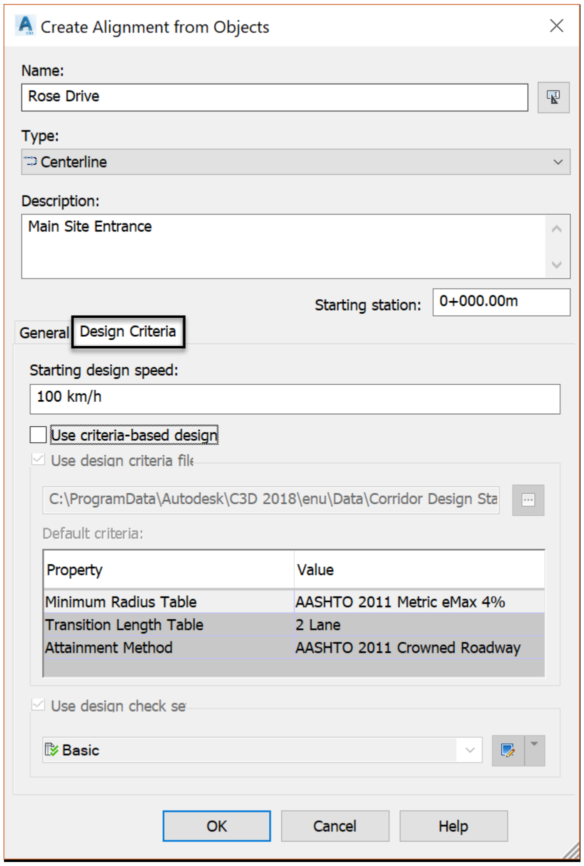



- Name the alignment **Rose Drive.**

- Then, choose a **Centerline** alignment type. Alignments can be either of type centerline, offset, curb return, rail, or miscellaneous. Alignments are categorized based on their function. You can use centerline types for a road centerline,

the rail type for a railway, or the miscellaneous type for other uses, such as a utility conduit. Also note that after creating the alignment, you can change it by using the **alignment properties** window.

- Do not specify a site as we are not looking to have any interaction with other items.
- After that, use the major and minor label set, to display tick marks at minor stations and values at major stations.
- Next, make sure to uncheck the **Add Curves between tangents** checkbox. Use this option in situations where we want to create curves between adjacent lines. It is obvious that the curves are already created in our case when we first created the baseline. Consequently, we don't need to create any more curves.
- Finally, check the **erase the existing entities** checkbox to erase the original polylines. Once you create the alignment, the polyline is not needed anymore and can be disregarded. If for any reason we need it, we can simply offset the alignment to recreate it.




- Now, switch to the **Design Criteria** tab. First, we can specify a **Design Speed** at the alignment starting station or other stations. This option is mostly used for the design of major roads when calculations such as super elevations are needed. To make these calculations, we can perform a

NOTES

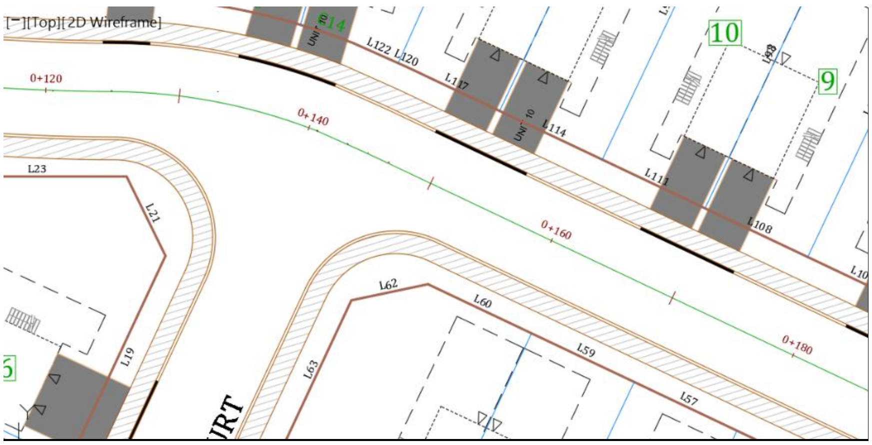
NOTES

**Criteria-Based Design.** We can use this method to meet a certain criterion when a standard file is supplied by a department of transportation. In this case, the default file is the **Civil 3D Metric (2011) Roadway Design Standards** XML file. To specify a design criteria file, click on the small button with the three dots . Then, you are directed to the default criteria folder where you can choose the criteria file you would like to apply. You can make a copy of any of these files, adjust them to your own local criteria requirements, by using a text editor and save it in this same folder.

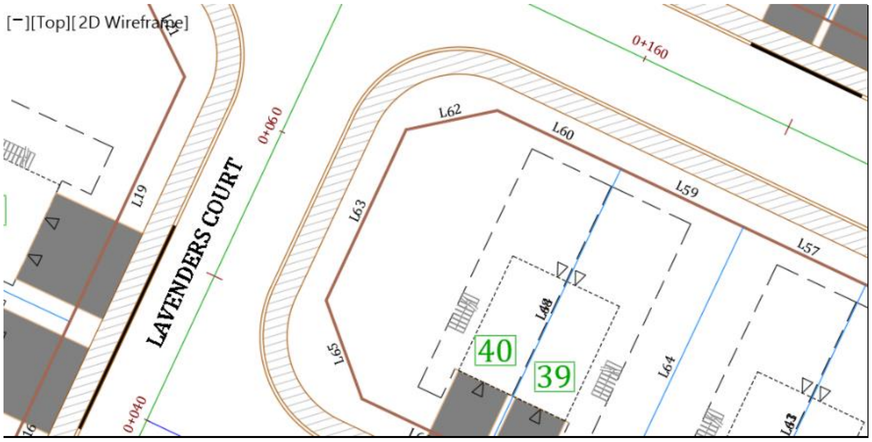
- Finally, you can specify a **Design Check Set List** to **display** design checks and receive messages when a design criterion is not met.

- Before closing this window, make sure the **Use criteria-based design** is unchecked.

- To finish, Click **OK** to create the **Rose Drive** alignment. It is then shown with stations showing every **20ms** or **50ft**, depending on the units of your practice drawing.




6. Use the same previous steps to create an alignment for **Lavender Court** using the magenta line in the centerline. In the end, you should have an alignment running south to north, with stations showing every **20m** or **50ft** depending on your practice file units.

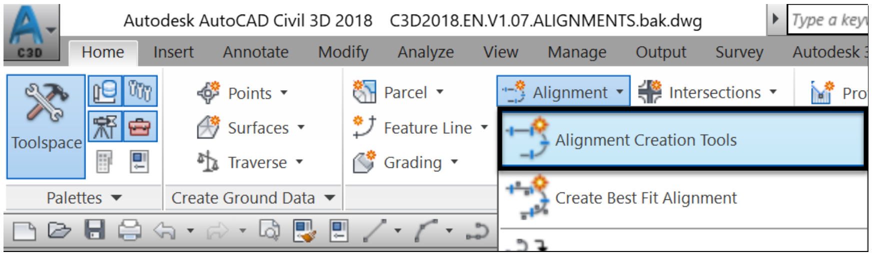


NOTES

## 7.3 Creating an alignment by Layout (creation tool)

The second way to create an alignment is to use the **Alignment Creation Tools.** These tools allow you to create an alignment using specific design conditions. It would otherwise be very difficult to achieve without these tools. They can also be used to modify an alignment once it's already created. Let's see a few situations where the **Alignment Creation Tools** are very handy.

1. First, to launch the tools, from the ribbon, select the **Alignment Creation Tools**.




2. Next, enter **Violets Circle** for the **name** of the alignment.

3. The type will be **Centerline** since this will represent the centerline of a street.

4. For **description,** enter a text describing the purpose of the alignment.

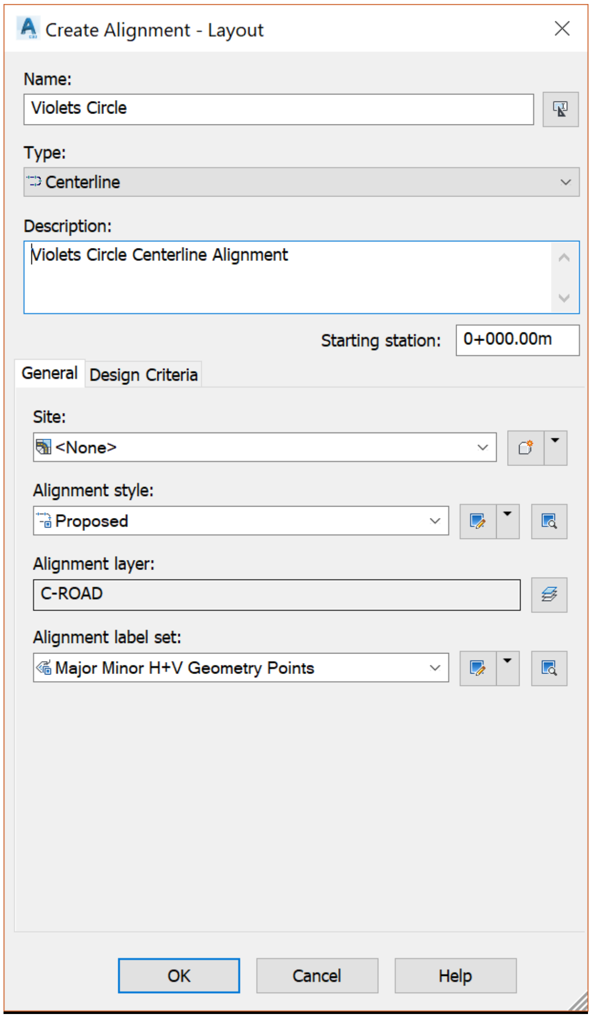
5. Then, do not specify any site, as we don't need any interaction between the alignment and other design items.

6. Afterward, for **style**, use **Proposed** to differentiate between proposed and existing alignment styles.

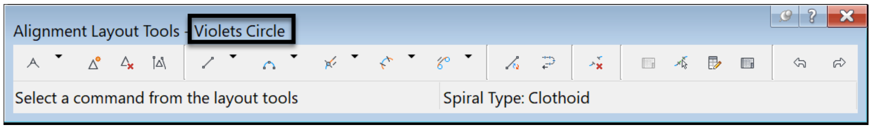
7. The layer is set by default from the template settings, and there's no need to change it.

8. Use the **Major Minor H+V Geometry points** to show major and minor stations, and horizontal and vertical geometry points.

9. Finally, no need to change anything on the **Design Criteria** tab.




1. The **Alignment Layout Tools** toolbar now opens. This toolbar will also indicate the name of the alignment you are currently working on. A close attention needs to be paid to this as sometimes, in the heat of the action, you may select the wrong alignment to edit.



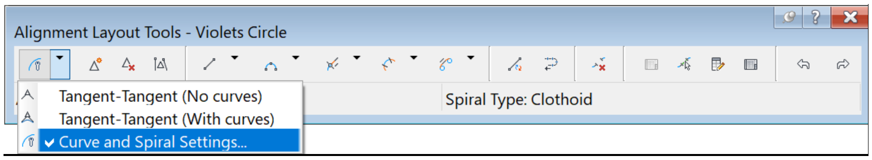

2. This toolbar has several tools to perform site-specific design. For example, you can:

    - design tangents, curves, and spirals, with specifics tangency requirements.
    - add or remove **PI**s (Point of Intersections),
    - convert AutoCAD lines or arcs to alignment entities,
    - reverse or edit created entities,
    - delete or select sub-entities, and
    - display a tabular view of design elements.

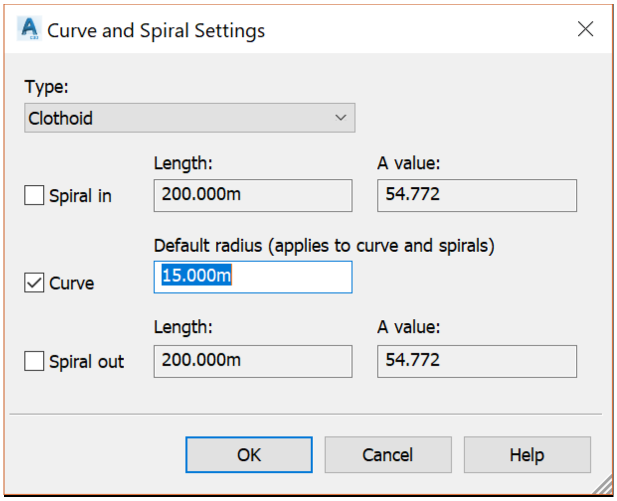
Now, let's explore a few of these tools.

NOTES

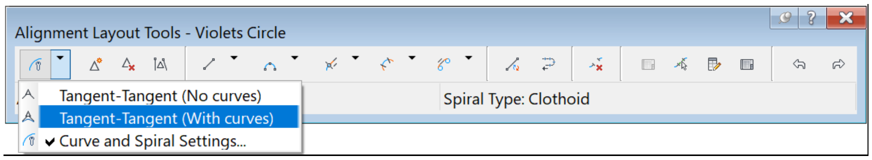
3. First, we will be creating an alignment around a cul-de-sac with a **15m** or **50ft** radius requirement. So, let's set that parameter. On the toolbar, click on **Curve and spiral settings**.




4. In the next window, accept **Clothoid** as the default type of spiral. We are not creating any spiral in this case, so this is irrelevant. Check the **Curve** checkbox and put **15m** or **50ft** for the default curve value and click **OK.**



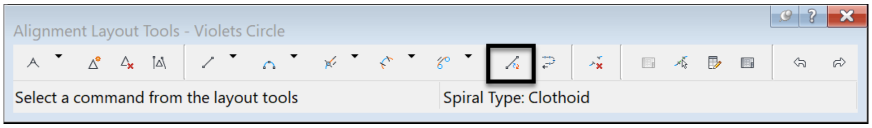

5. Now, start designing the alignment by running the **Tangent-Tangent (With curves)** command. It will automatically create a curve with a radius value. Exactly like we specified in the **Curve and Spiral Settings**, whenever we create two subsequent tangents. As recommended earlier, whenever possible, always design your alignment from west to east and south to north. In this case, the alignment should start at the intersection of **Lavender Court** and **Violets Circle**. That point is pre-determined for us with the blue line centerline representing **Violet Circle**'s centerline. You can also create this line using multiple AutoCAD drafting options. One is to create two lines perpendicular to the curb line at each end of the street.Then, draw a line from the midpoint to the midpoint of these two lines. Or, you can use the AutoCAD constraints options to draw the centerline. But, this is not an AutoCAD course. Therefore, we will focus on Civil 3D functionalities.



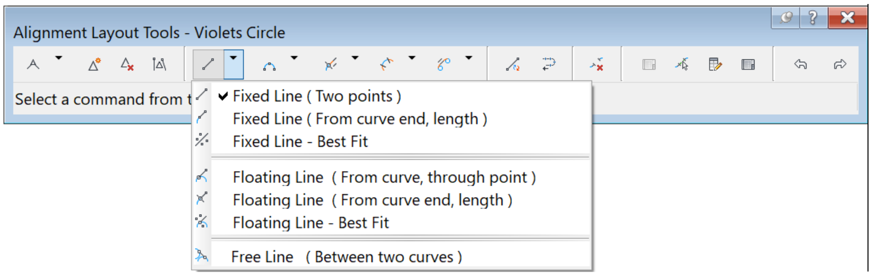

NOTES

NOTES

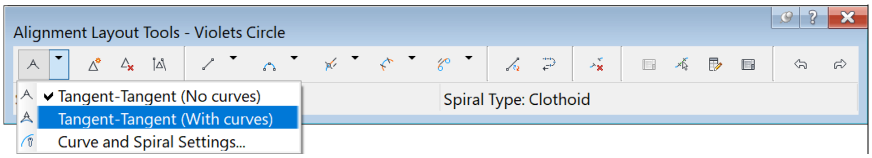
6. Now that you are set up, you can use the **Convert AutoCAD line and Arcs** command to convert AutoCAD entities. This will create tangents and curves for the alignment.




7. Alternatively, you can use any of the line creation tools, if you know your start and end points.

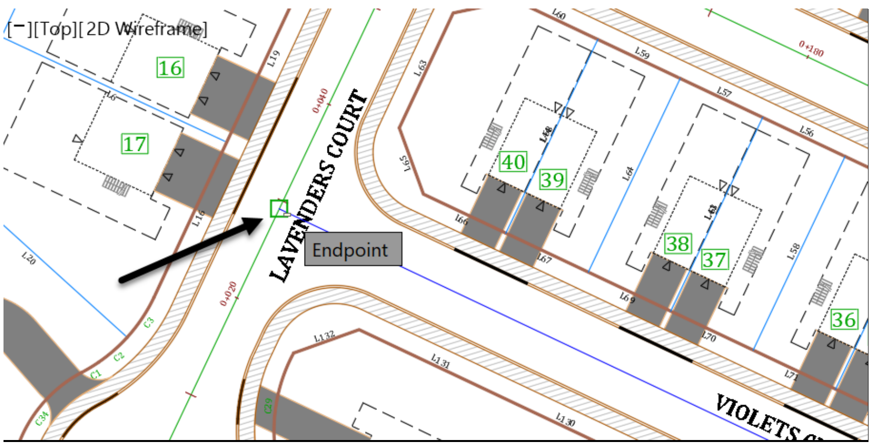



8. Or, the **Tangent-Tangent (No Curves)** command.

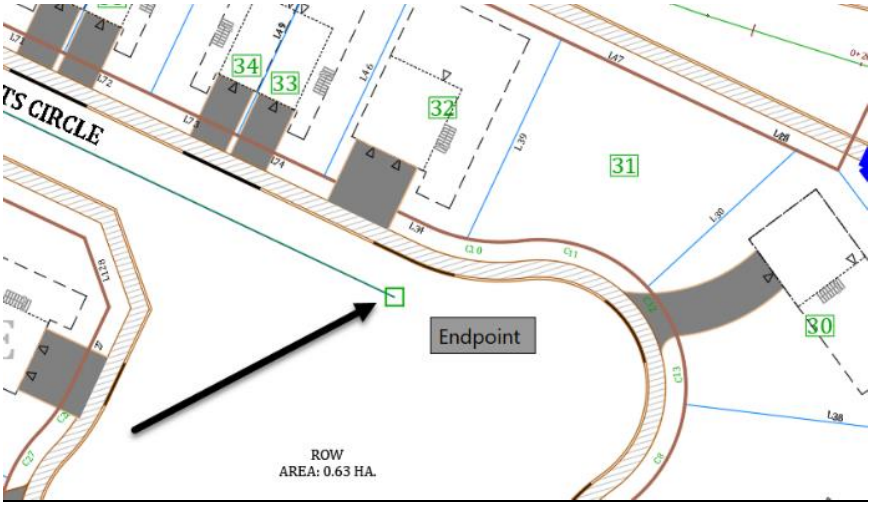



9. The advantage of the **Tangent-Tangent,** (with or without curves), is that you can keep designing without exiting the active command.

10. Let's use the **Tangent-Tangent (With curves)** command. Click at the intersection of the two streets, with your **End** object snap mode activated.




11. Then, click on **Point A**, at the east end of the polyline on **Violets Circle**.



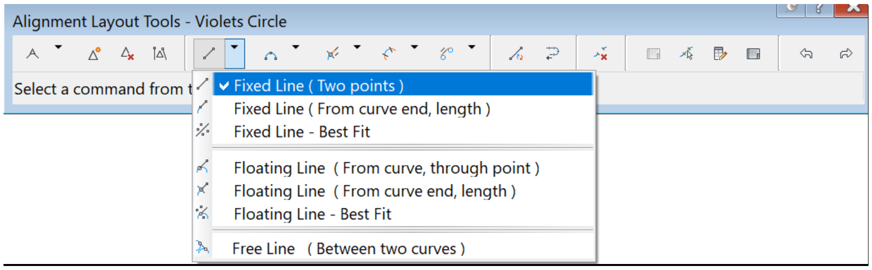

12. Now, we are going to learn how to use **fixed** and **floating** line and curve entities.

    - **Floating** entities are constraint-based alignment geometries. They will maintain tangency between adjacent objects.
    - On the other end, **fixed** entities are fixed in their position, and must be edited directly. They are not affected by the geometry of adjacent entities and do not maintain tangency.
    - The third type of entity is the **free** entity, line or curve. It is defined by not only one, but two entities, on which the free entity is dependent on them to define its geometry. An instance of this is a fillet between two curves.

13. Let's start with a line entity between point A and B. If you have closed the **Alignment Layout tools** by mistake; you can always reopen it. To reopen it, selecting an alignment, then right-click and select **Edit Alignment Geometry**. Then you can, once again, edit the alignment or add additional data.

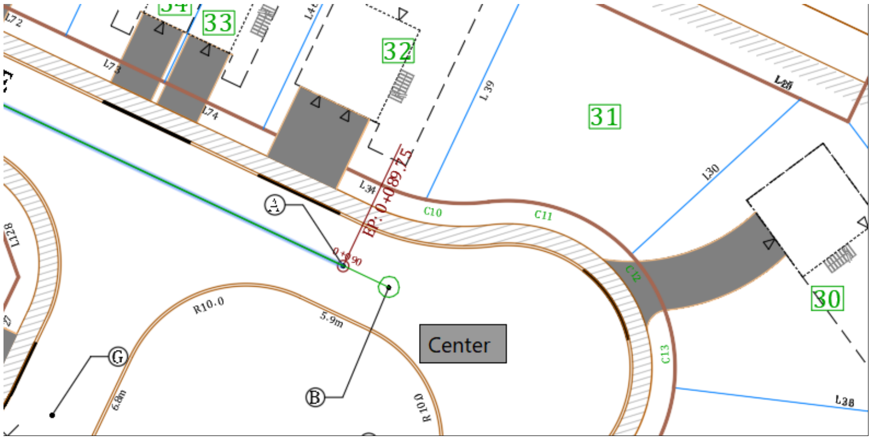
14. After clicking a button on the **Alignment Layout Tools**, follow the prompts at the command line.

15. What we want to do next is use the **Fixed Line (Two points)** command to add a tangent. Clicking on **Point A** then **Point B**. At each click, make sure the **Center** object snap mode is used.



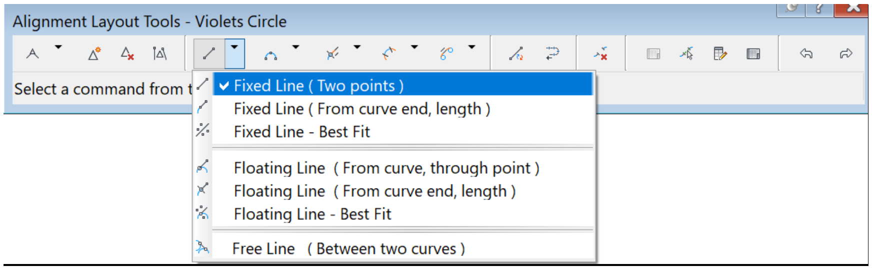

NOTES

NOTES




16. Now what we want to do is create the centerline alignment of **Violets Circle**, in the cul-de-sac area. An island with inside curb line has already been created for us during the conceptual phase of the project. We need to create the centerline with a minimum travel way width of **3.5m** or **12.5ft**, on both ends of the centerline. A few reference points, from **A** to **G**, have been created for us to use.

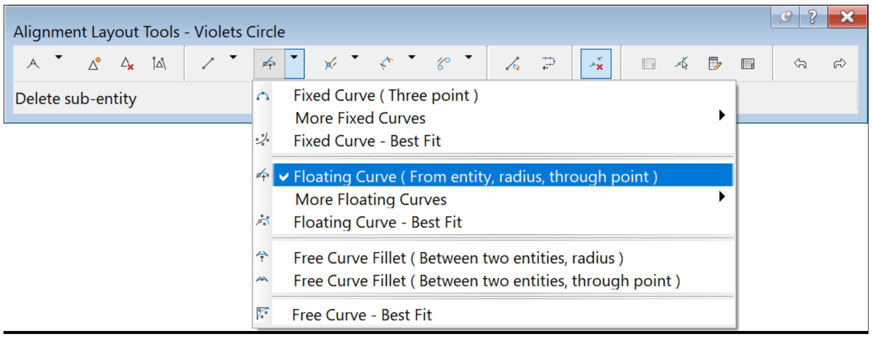
17. First, we need to connect **Point A** to **Point B**. For that to happen, you can use any of the alignment line creation commands. They can be **Fixed Line (two points)**, **Floating line** or **tangent-tangent.**



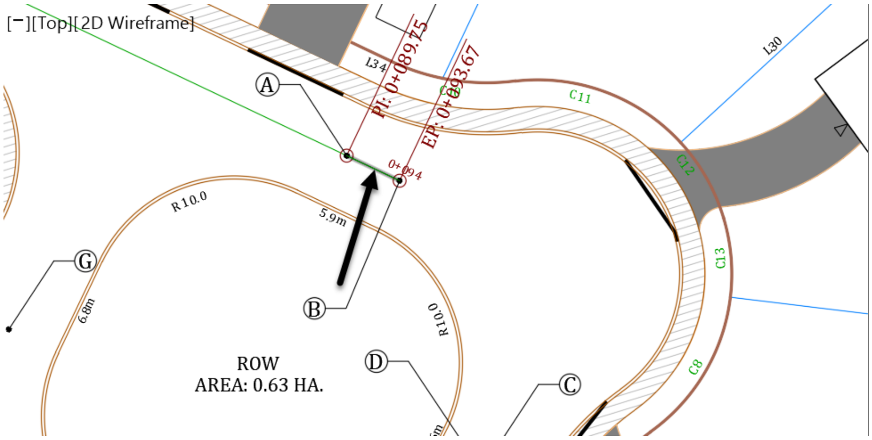

18. Next, we will go from **Point B** to **C**. Between these two points, the outside radius of the face of the curb is **10m** or **35ft**. Therefore, we need to create a centerline radius of **13.5m** or **45ft**. The new curve entity between **B** and **C** will also need to maintain a tangency, going from **A** and **B.**

19. The command that allows us to do that by using **Floating Curve (From Entity end, radius, through point)**. Once you run the command, you are prompted to select the entity to attach the new curve to. Click on the line between **A** and **B**.

NOTES




```
End of selected entity already has an attachment.
Select entity to attach to:
```




20. Enter **13.5m** or **45ft,** when prompted to specify a **radius**, at the command line**.**

```
Select entity to attach to:
Specify radius or [Degree of curvature] <15.000m>: 13.5
```

21. For the **curve solution angle**, press **Enter** to accept a value **less than 180-degrees**.

```
Specify radius or [Degree of curvature] <15.000m>: 13.5
Is curve solution angle [Greaterthan180 Lessthan180] <Lessthan180>:
```

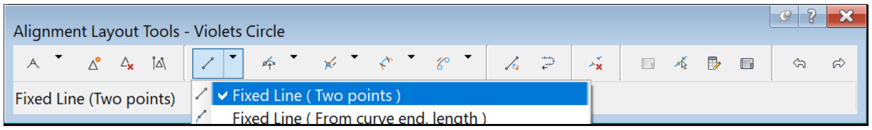
22. Then, when prompted to specify the **Endpoint** of the entity, click on **Point C.**

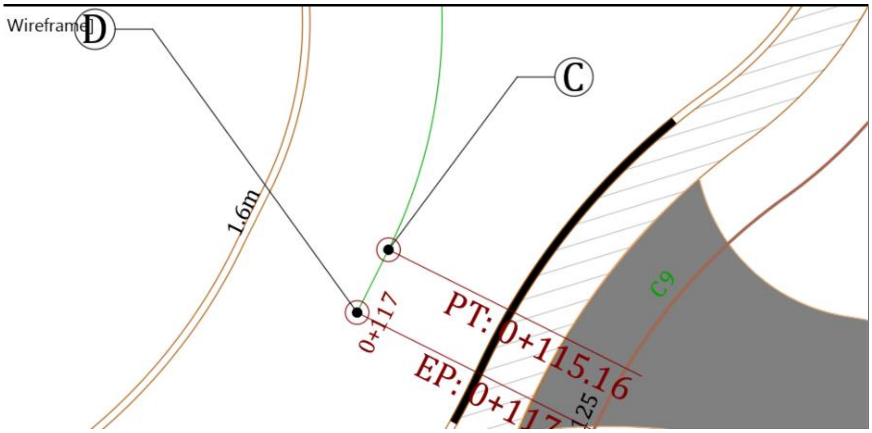
```
Is curve solution angle [Greaterthan180/Lessthan180] <Lessthan180>:
Specify end point:
```

23. We have now created a curve between **B** and **C.** Also, it is perfectly tangent to the previous line entity. Creating tangent and floating entities is one of the main benefits of the **Alignment layout tools.** It's crucial to use these tools when highly accurate alignments are required. Especially when designing rail tracks, highways or airfields.

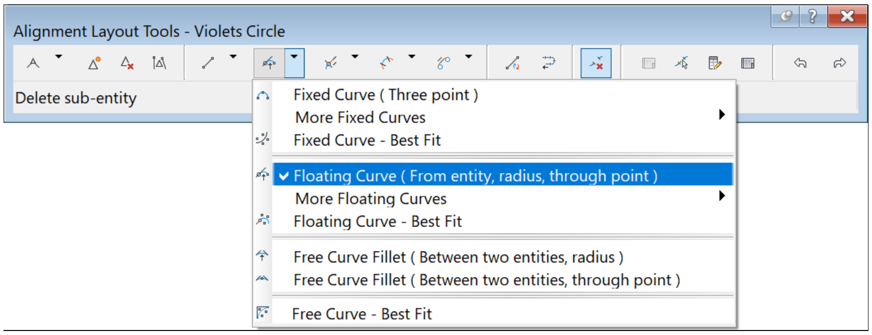
24. Next, we need a line entity from **Point C** to **Point D.** Simply run a **Fixed Line (Two points)** command and click on the start and

end points. Make sure the Object Snap setting is active, to connect the two points.

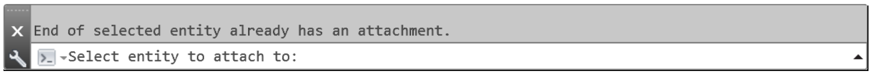


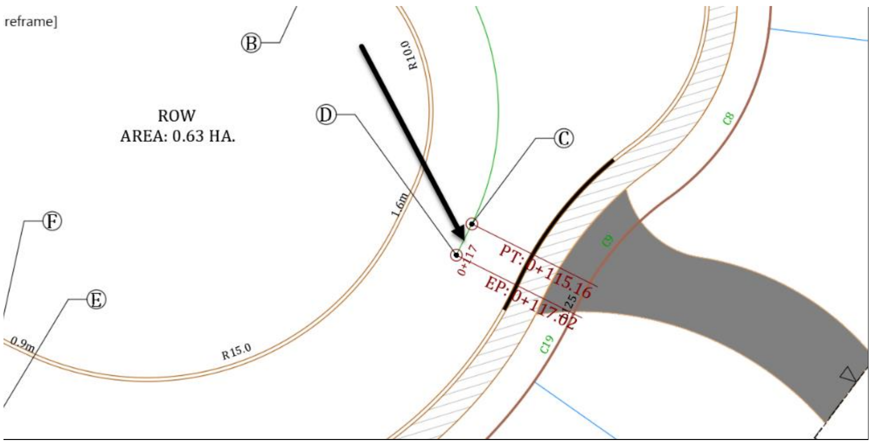




25. Now that **Point C** and **Point D** are connected, we need to tie **Point D** to **Point E**, with an **18.5m** or **60ft** radius curve. Run **Floating curve (From entity, radius, through point)** command.




26. Select the line between **Point C** and **Point D**, when prompted for the entity to attach to.




NOTES

27. When asked to specify a **radius**, type **18.5m** or **60ft**

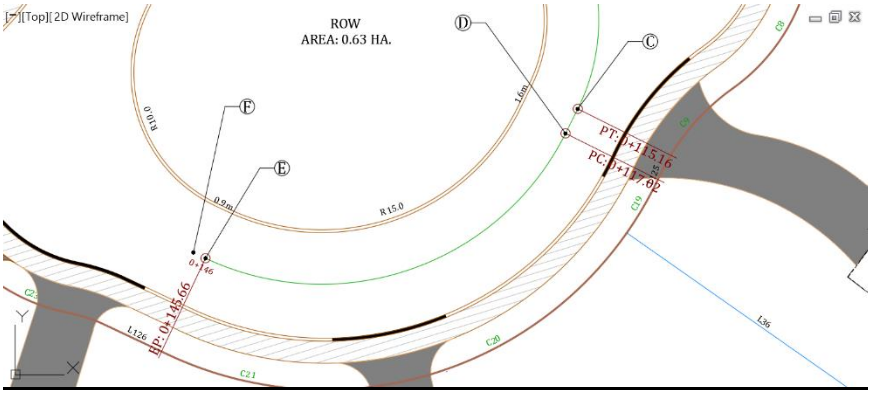
28. For the solution angle, press **Enter** to accept an angle that is **less than 180-degrees**

```
Specify radius or [Degree of curvature] <15.000m>: 13.5
>_ -Is curve solution angle [Greaterthan180 Lessthan180] <Lessthan180>:
```

29. Then, when asked to **Specify Endpoint**, click on **Point E**.

```
Is curve solution angle [Greaterthan180/Lessthan180] <Lessthan180>:
>_ -Specify end point:
```

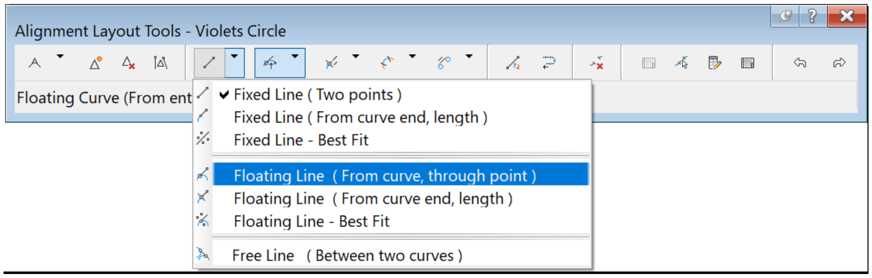
30. We now have a curve created between points **D** and **E**. Moreover; it is perfectly tangent to the previous line entity.




31. Next, create a tangent from **E** to **F**.

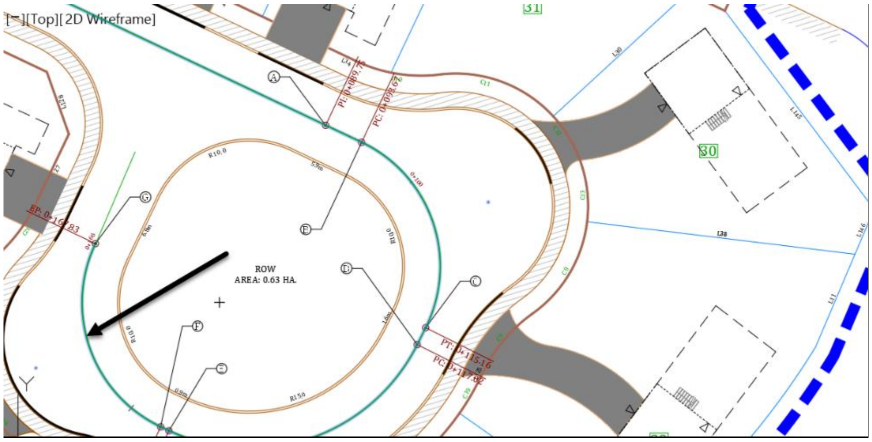
32. Then a floating curve from **F** to **G**, with a **13.5m** or **40ft** radius.

33. Last, create a **floating line**. We could have created a **fixed line** if we knew the exact location of the start and end points. We are going to demonstrate how to use **floating lines** that always stay tangent to a curve, whatever the circumstances. Run the **Floating Line (From curve, through point)** command.



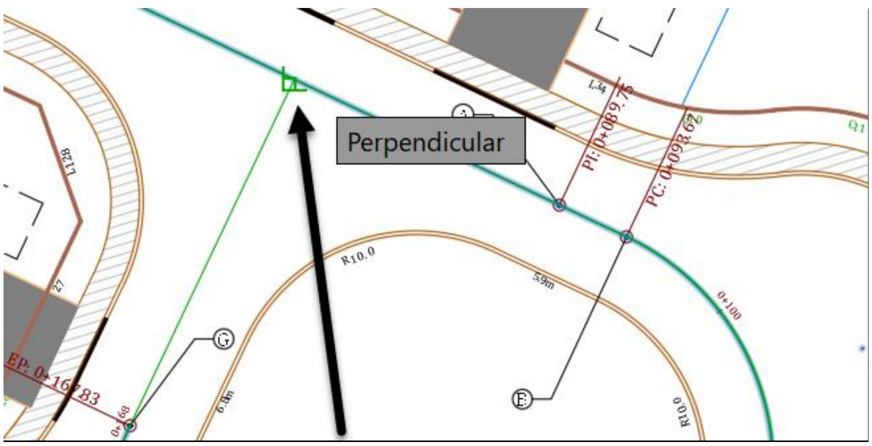

34. Select the entity to attach to, this time the last curve we created.

NOTES

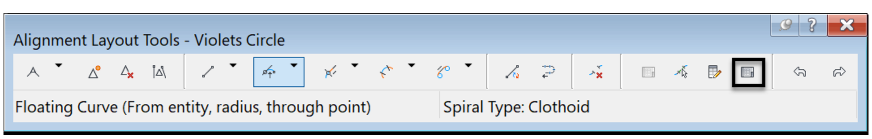



35. Next, you are prompted for **Endpoint**. Type **PER**, an alias for **perpendicular**, at the command line and press **Enter**.


36. Move the cursor closer to the first tangent of the alignment. Once you see the perpendicular sign, click on the alignment to snap the new tangent in place.

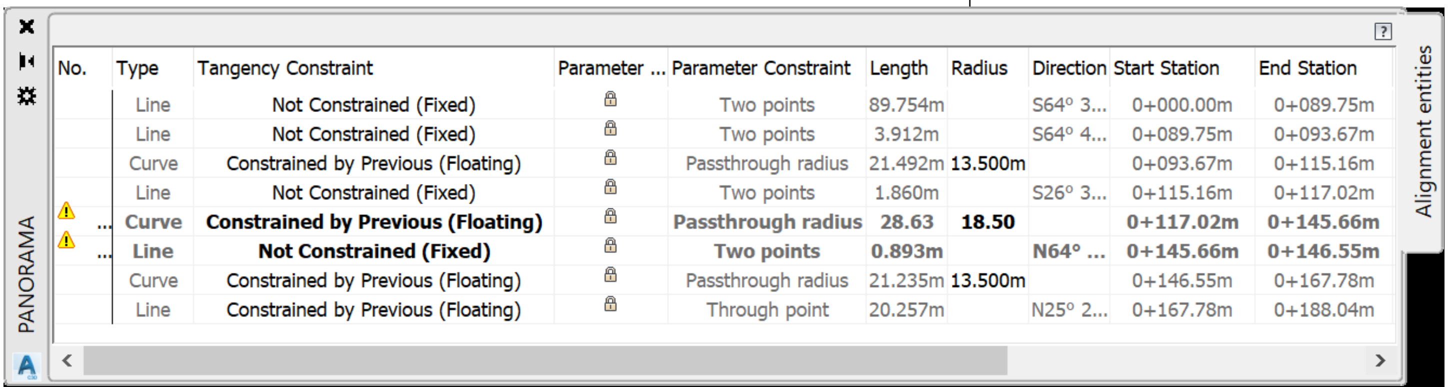



37. Now, let's explore a couple of the **Alignment Layout Tools**. The first thing we need to do is verify our design through a tabular view. On the **Alignment Layout Tools**, click on the table icon to the far right .




NOTES

38. The **panorama** window is displayed. It is used in Civil 3D to display many types of data, such as the Point Editor, the Alignment Entities or other vistas. In this case, it displays the alignment entities.

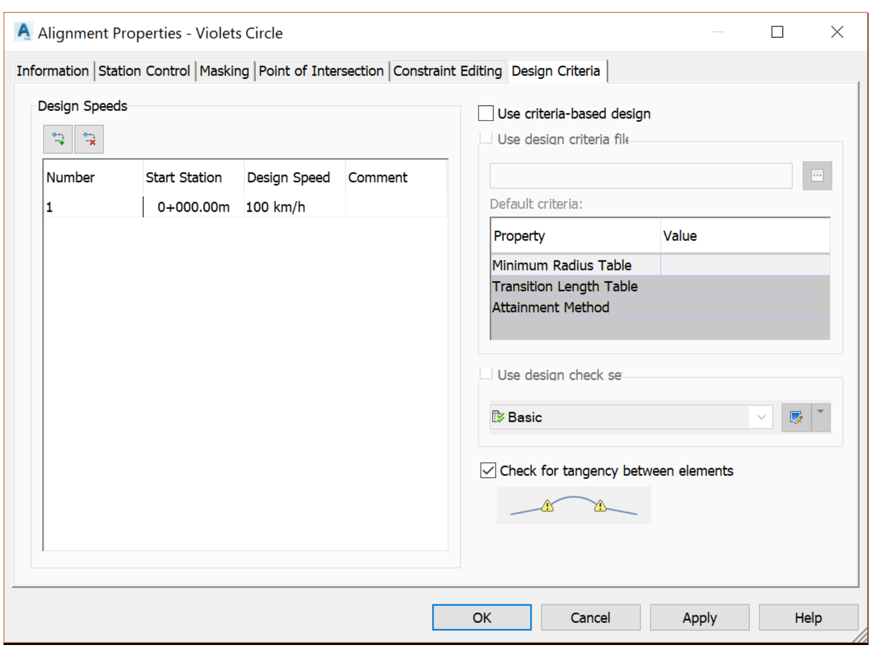


| No. | Type | Tangency Constraint | Parameter ... | Parameter Constraint | Length | Radius | Direction | Start Station | End Station |
|---|---|---|---|---|---|---|---|---|---|
| | Line | Not Constrained (Fixed) | | Two points | 89.754m | | S64° 3... | 0+000.00m | 0+089.75m |
| | Line | Not Constrained (Fixed) | | Two points | 3.912m | | S64° 4... | 0+089.75m | 0+093.67m |
| | Curve | Constrained by Previous (Floating) | | Passthrough radius | 21.492m | 13.500m | | 0+093.67m | 0+115.16m |
| | Line | Not Constrained (Fixed) | | Two points | 1.860m | | S26° 3... | 0+115.16m | 0+117.02m |
| ... | **Curve** | **Constrained by Previous (Floating)** | | **Passthrough radius** | **28.63** | **18.50** | | **0+117.02m** | **0+145.66m** |
| ... | **Line** | **Not Constrained (Fixed)** | | **Two points** | **0.893m** | | **N64° ...** | **0+145.66m** | **0+146.55m** |
| | Curve | Constrained by Previous (Floating) | | Passthrough radius | 21.235m | 13.500m | | 0+146.55m | 0+167.78m |
| | Line | Constrained by Previous (Floating) | | Through point | 20.257m | | N25° 2... | 0+167.78m | 0+188.04m |



39. What jumps at us are the warning signs. They represent the sub-entity that violates the specified design criteria. If you were not using a **design criteria**, you would not have noticed and move along. But since you have applied a design check, violations will be highlighted for you. You have not purposely set the check. It's a setting that came with this specific template and it is activated anytime you create an alignment **by layout**. To turn it off,

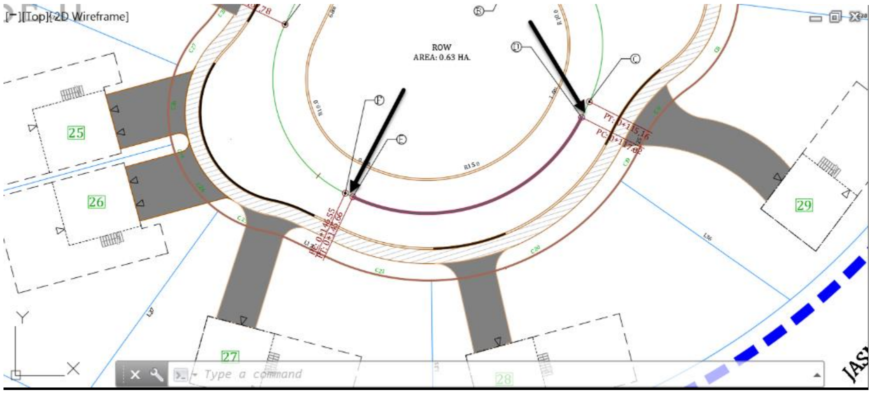
- select the alignment,
- right click and select **Alignment properties**
- Then, go to the **design criteria** tab. On there, you can uncheck the design criteria checkbox. We recommend that you leave it checked. This is just so you know where the error warnings are coming from.




40. Now let's fix the warnings. The two culprits are two fixed lines, sandwiching a curve. You can identify them by clicking them in the panorama and watching them getting highlighted in the

NOTES

drawing. As we've mentioned before, **Fixed Lines** do not always maintain tangency when edited. In a case like this, using **floating** entities, when possible, is always recommended. To delete the two lines, on the **Alignment Layout Tools**, run the **delete sub-entity command** and click on the two fixed lines.




41. Once you delete the lines, replace them with floating lines by using the command **Floating Line (From curve, through point)**.

42. After that, create the two **floating lines** and check back in the panorama window. The warning signs are now gone, and everything looks good. In this table, we can check our radiuses, entities directions, start and end stations, and so forth.

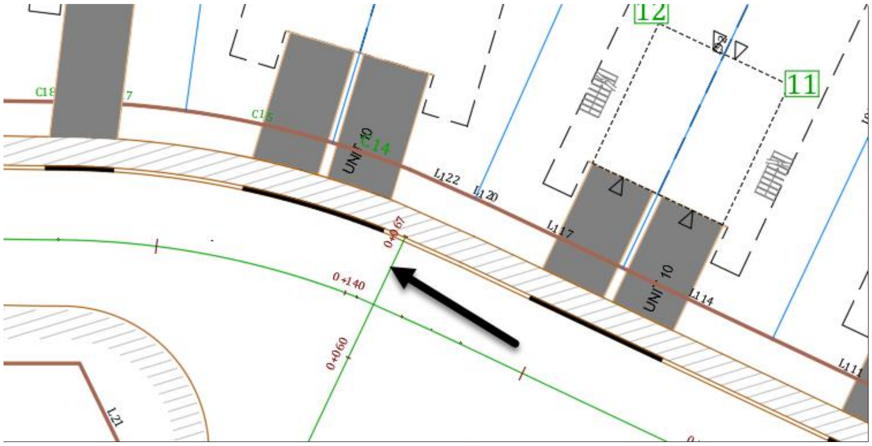
| No. | Type | Tangency Constraint | Parameter ... | Parameter Constraint | Length | Radius | Direction | Start Station | End Station |
|---|---|---|---|---|---|---|---|---|---|
| | Line | Not Constrained (Fixed) | | Two points | 89.754m | | S64º 3... | 0+000.00m | 0+089.75m |
| | Line | Not Constrained (Fixed) | | Two points | 3.912m | | S64º 4... | 0+089.75m | 0+093.67m |
| | Curve | Constrained by Previous (Floating) | | Passthrough radius | 21.492m | 13.500m | | 0+093.67m | 0+115.16m |
| | Line | Constrained by Previous (Floating) | | Through point | 1.860m | | S26º 3... | 0+115.16m | 0+117.02m |
| | Curve | Not Constrained (Fixed) | | Three points | 28.641m | 18.500m | | 0+117.02m | 0+145.66m |
| | Line | Constrained by Previous (Floating) | | Through point | 0.907m | | N64º 4... | 0+145.66m | 0+146.57m |

PANORAMA

Alignment entities

NOTES

43. Now, zoom to the intersection of **Lavender Court** and **Rose Drive**. Let's say you want to keep things tidy and do not want to have the alignment of **Lavender court** cross over to the other side of **Rose Drive** centerline.




44. What you can do is insert a **PI (Point of Intersection**) at the intersection and delete the unwanted excess entity. To do that, use the **Alignment Layout Tools.** On this toolbar, let's mention a few useful tools:

- The **Insert PI** command used to insert **point of intersections**
- The **Delete PI** tool, to remove **point of intersections**
- The **Convert line and arcs** tool, to convert AutoCAD entities
- The **Reverse sub-entity** command to change the direction of a sub-entity
- The **Select sub-entity** command to select sub-entities for editing
- And the **Undo** and **Redo** commands to undo or redo edit operations

NOTES

NOTES

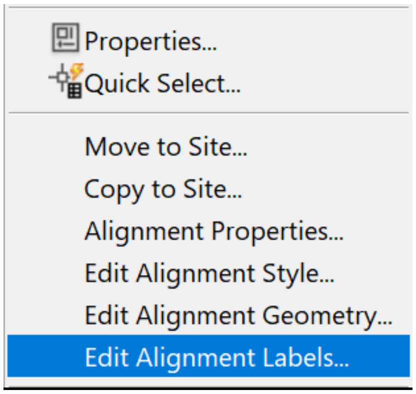
## 7.4 Alignment Labels

Alignment labels allow us to display information about the alignment. That includes stations, offsets, geometry points, profile elevations, and more.

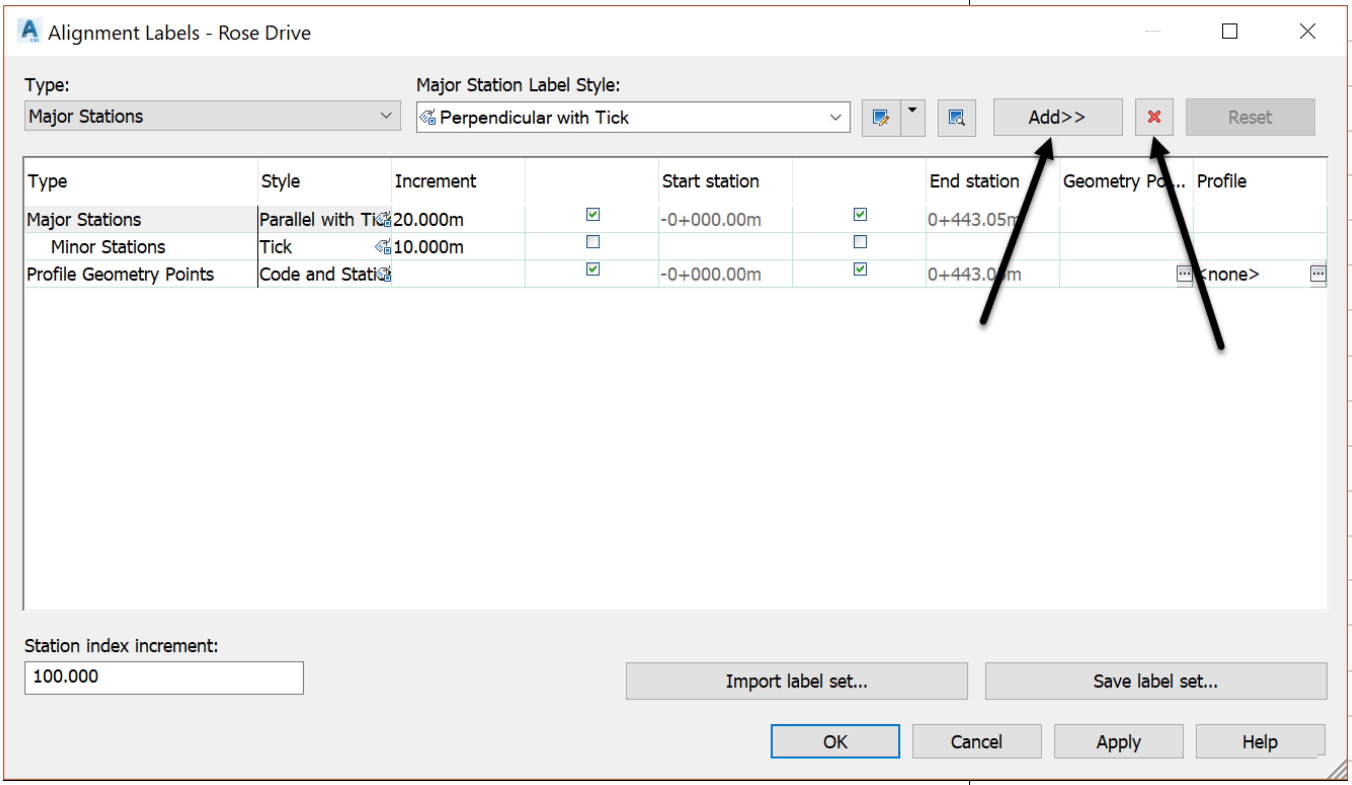
To create an alignment label:

1. First, select the alignment, let's say **Rose Drive**;

2. Then, right-click and select Edit Alignment Labels .



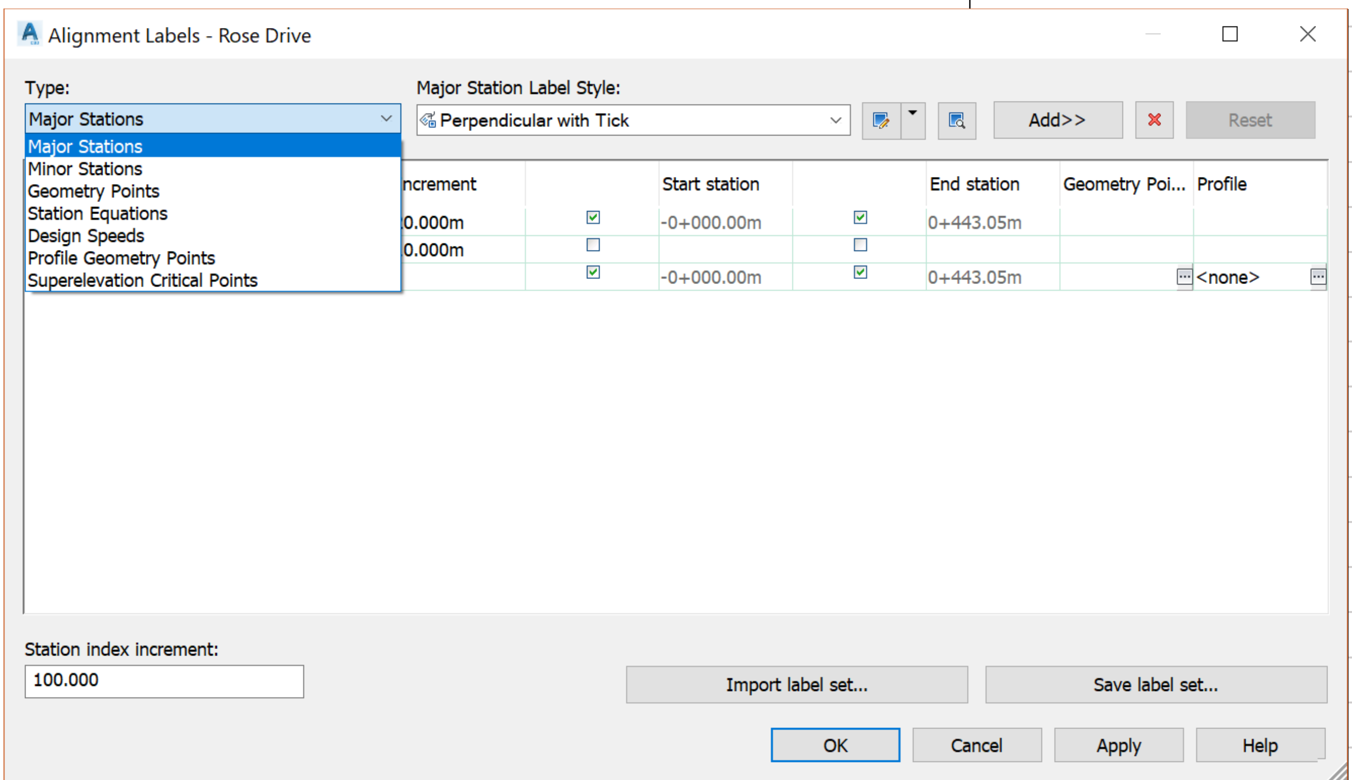

3. In the **Alignment Labels** window, **Add** or **Remove** alignment labels. You can also change the station labeling frequency.




4. Among the types of labels you can add are:

   - Major and Minor labels.

- Geometry points like the start of curves, end of curves, **PIs,** and the like.
- Station equations, for alignments that have them.
- Design speeds, if you are using a criteria base design.
- Profile geometry points labels, that allow you to reference and specifies profile information, including grade breaks, PVIs, vertical curves values, and much more.
- Finally, label superelevation critical points.




5. If needed, you can also **Import a label set** or **save** one for later use.

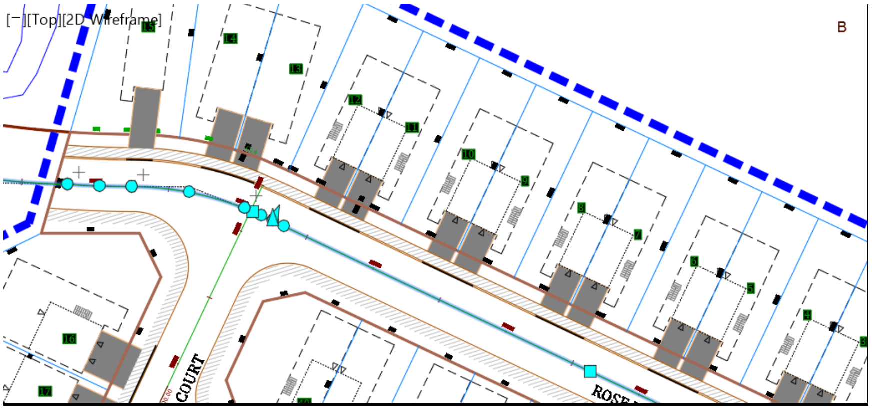
6. Finally, click **OK** to close the label creation window.

NOTES

### 7.4.1 Labels and Alignment Tables

We can display the alignment entities information in a tabular manner. This is particularly important for field stakeout. Let's assume a surveyor has requested the centerline information of the new **Rose Drive.** To create that information, we can

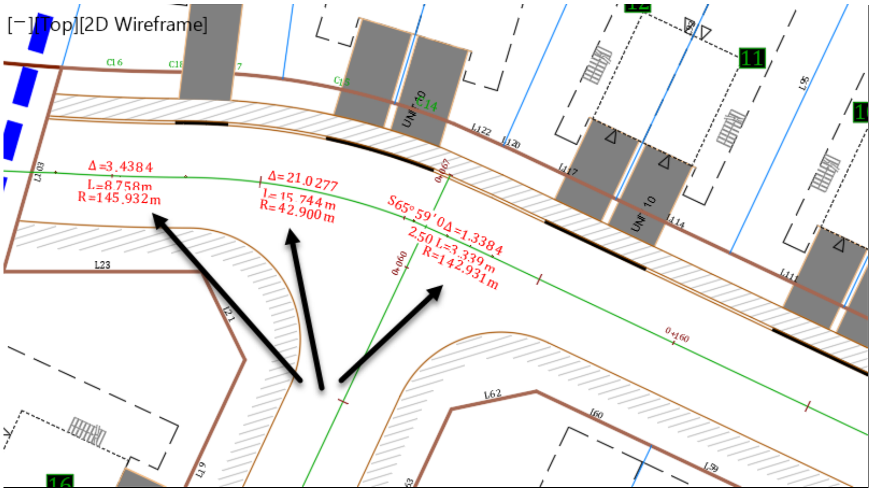
1. Select the alignment from the drawing.




2. then, run the **Add Labels** command and select **Multiple Segments,** from the ribbon, in the upper left corner.

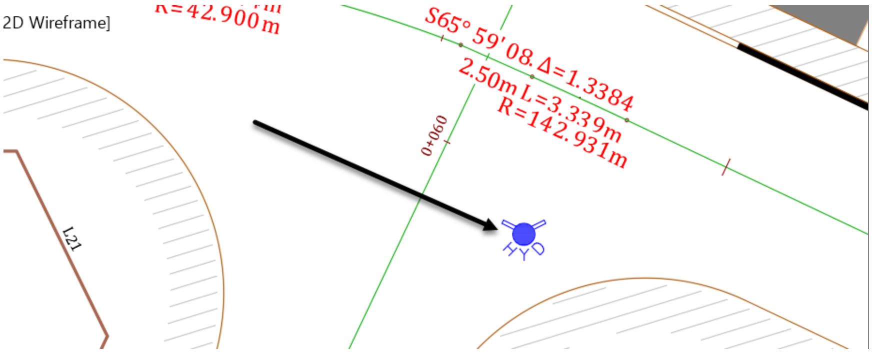
3. Click on the **Rose Drive** alignment, when prompted to specify an alignment to label.

4. The alignment labels now appear in the drawing.

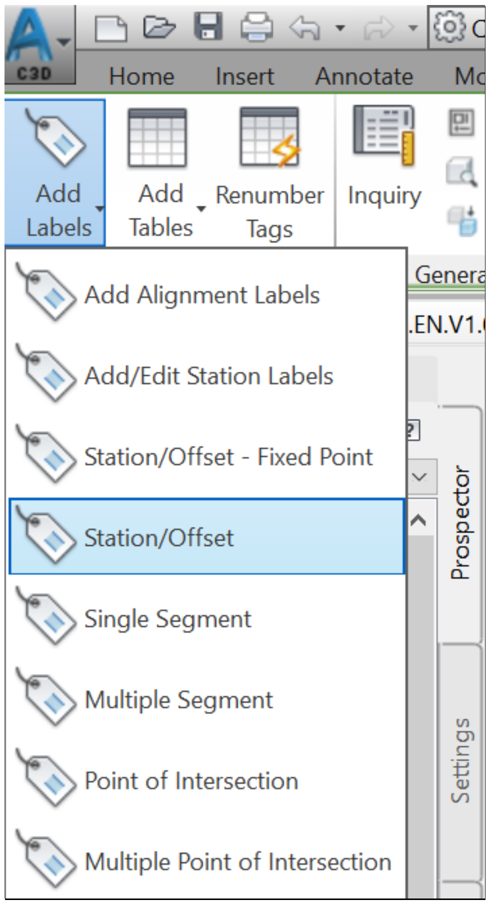



5. Another type of label we can create are **Station Offsets** at selected points along the alignment. **Station Offsets** are useful in referencing objects in relation to a survey baseline for instance. Let's say we are required to provide the **station/offset** information of the proposed Fire Hydrant in the intersection area.

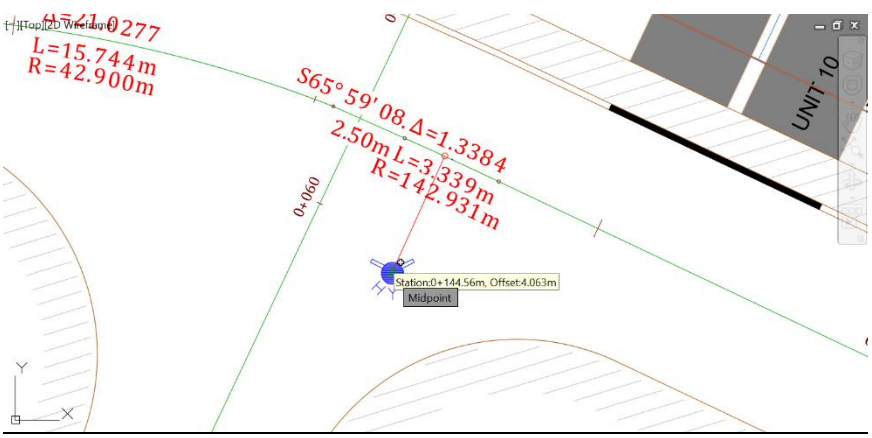
NOTES




6. Select the **Rose Drive** alignment. Next, run the **Station/Offset** command.



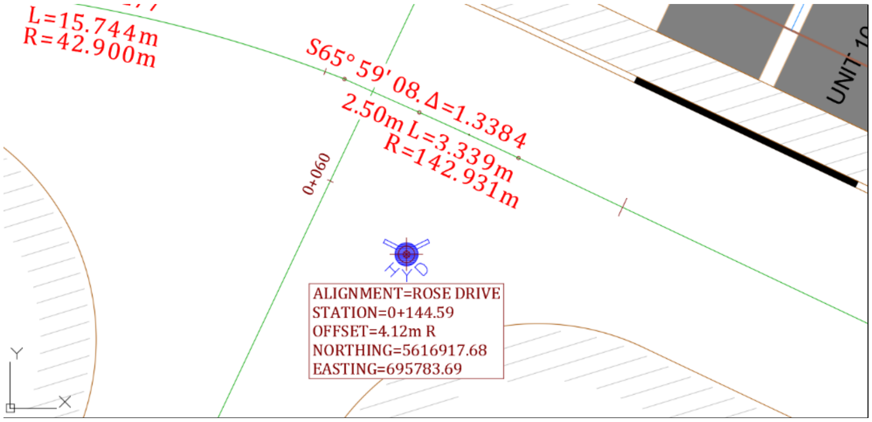

7. Then click twice on the **fire hydrant**. The first time to determine the station and a second time to specify the offset.



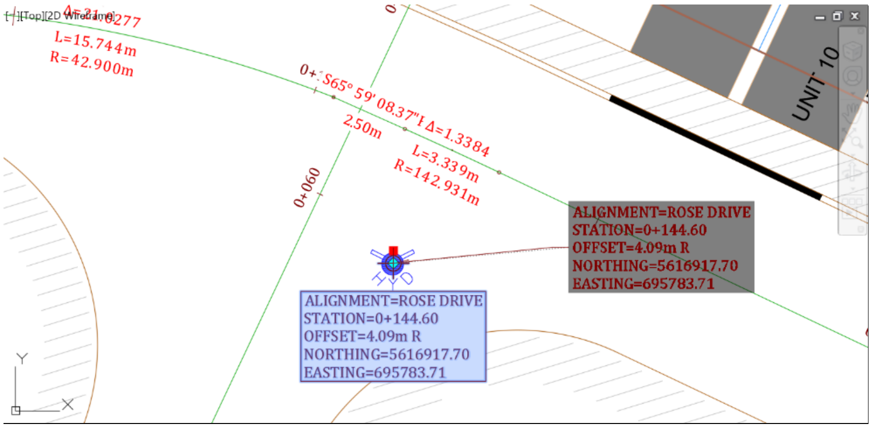

NOTES

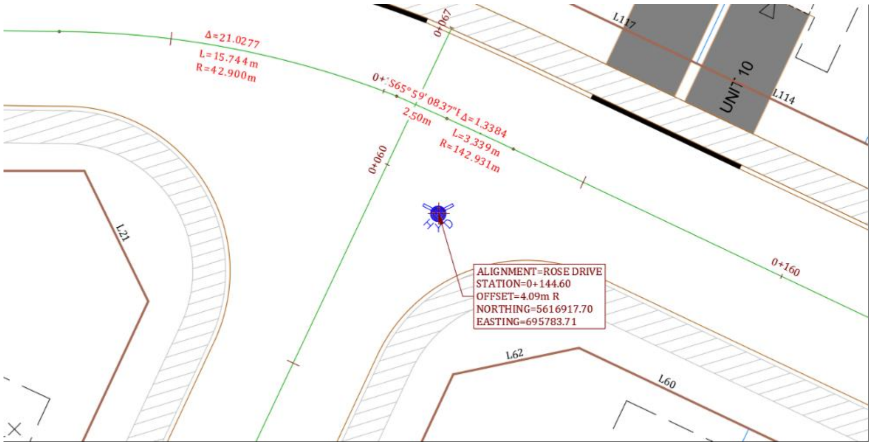
NOTES

8. The station/offset label information for the **fire hydrant** now will appear in the drawing.




9. If needed, you can move the labels to make the plan more readable.



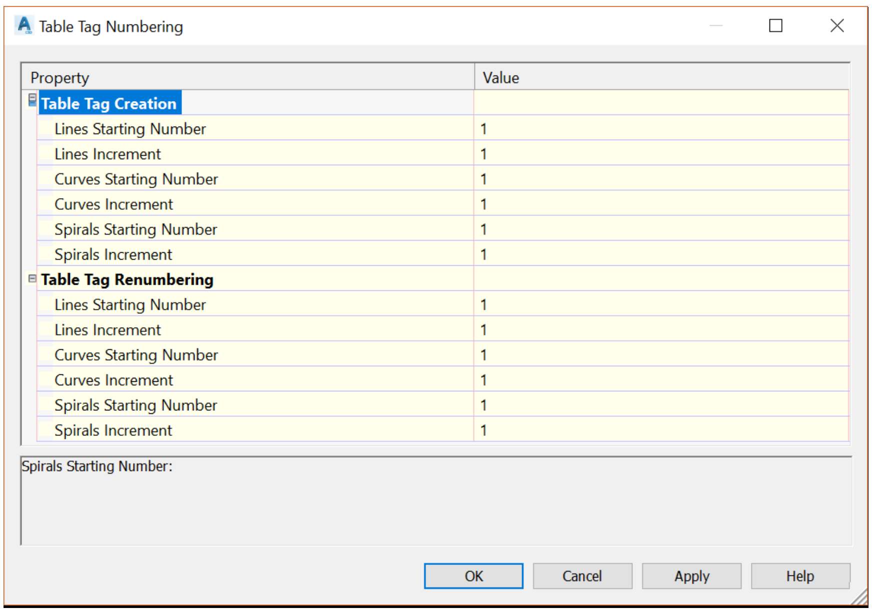



NOTES

## 7.5 Alignments Tables

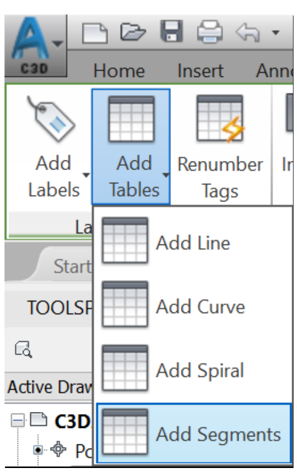
Let's see how to create alignment label tables. They help us make construction plans more readable by not overloading the drawing with information. Instead of displaying every construction note, we can simply display smaller tags and reference them in a table for more details. Let's practice that.

1. First, let's set up the table before we create the tags. We want to specify our label numbering scheme. Switch to the **Settings** tab of the **Toolspace**. Right-click on the name of the drawing and select **Table Tag Numbering;** this brings up the **Table Tag Numbering** window. Reset all the numbers to **1.** So that all lines, curves, and spirals start with the number **1.** We also want to increment them by **1.**

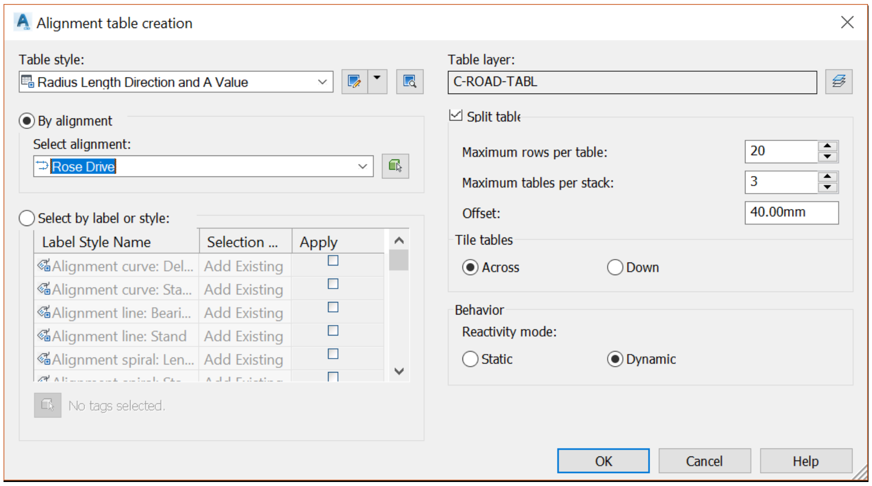



2. Click **OK** to close the window.

3. Next, select the **Rose Drive** alignment and then run the **Add Segments** command in the **Add Tables** drop-down menu.



4. In the **Alignment Table Creation** window, keep all default styles and layers. Then choose to create the table **by alignment**. Then select the **Rose Drive** alignment.

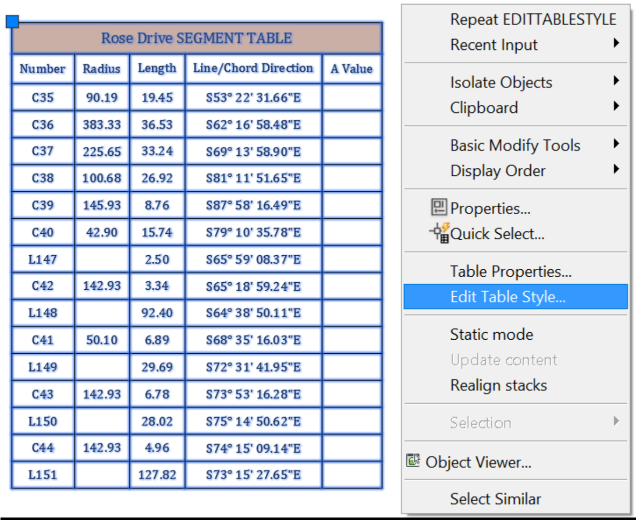



5. Click **OK** and select an empty area in the drawing to insert the table.

| Rose Drive | | | | |
|---|---|---|---|---|
| Number | Radius | Length | Line/Chord Direction | A Value |
| C35 | 90.19 | 19.45 | S53° 22' 31.66"E | |
| C36 | 383.33 | 36.53 | S62° 16' 58.48"E | |
| C37 | 225.65 | 33.24 | S69° 13' 58.90"E | |
| C38 | 100.68 | 26.92 | S81° 11' 51.65"E | |
| C39 | 145.93 | 8.76 | S87° 58' 16.49"E | |
| C40 | 42.90 | 15.74 | S79° 10' 35.78"E | |

6. You will notice that our table doesn't necessarily start with label **C1** for curves or **L1** for line segments. That is simply because these labels were already used for the parcel segment labels, so the table uses the next available numbers.

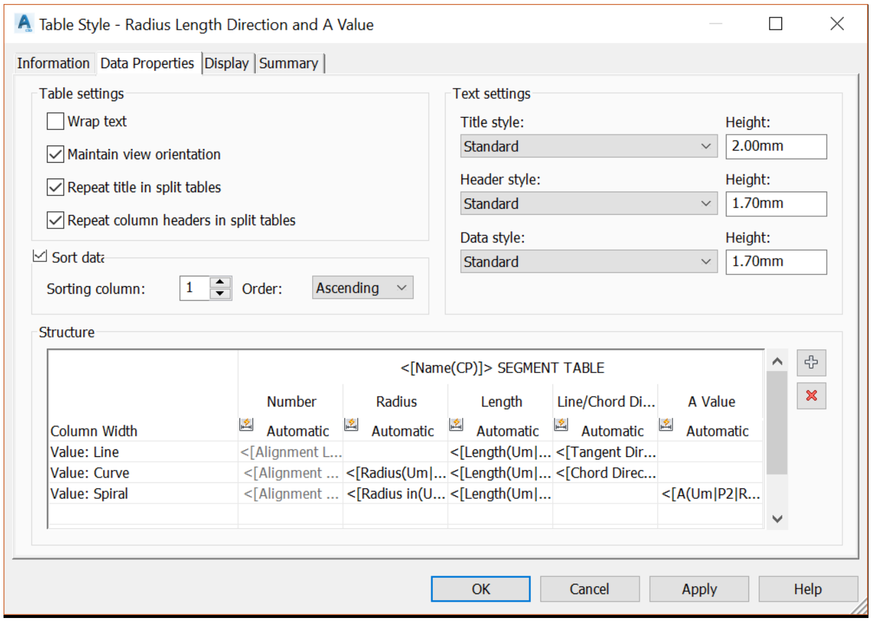
7. Next, to make the table easier to read, sort it in ascending tag numbers. Select the table, right-click and choose **Edit Table Style**.



| Rose Drive SEGMENT TABLE | | | | |
|---|---|---|---|---|
| Number | Radius | Length | Line/Chord Direction | A Value |
| C35 | 90.19 | 19.45 | S53° 22' 31.66"E | |
| C36 | 383.33 | 36.53 | S62° 16' 58.48"E | |
| C37 | 225.65 | 33.24 | S69° 13' 58.90"E | |
| C38 | 100.68 | 26.92 | S81° 11' 51.65"E | |
| C39 | 145.93 | 8.76 | S87° 58' 16.49"E | |
| C40 | 42.90 | 15.74 | S79° 10' 35.78"E | |
| L147 | | 2.50 | S65° 59' 08.37"E | |
| C42 | 142.93 | 3.34 | S65° 18' 59.24"E | |
| L148 | | 92.40 | S64° 38' 50.11"E | |
| C41 | 50.10 | 6.89 | S68° 35' 16.03"E | |
| L149 | | 29.69 | S72° 31' 41.95"E | |
| C43 | 142.93 | 6.78 | S73° 53' 16.28"E | |
| L150 | | 28.02 | S75° 14' 50.62"E | |
| C44 | 142.93 | 4.96 | S74° 15' 09.14"E | |
| L151 | | 127.82 | S73° 15' 27.65"E | |

NOTES

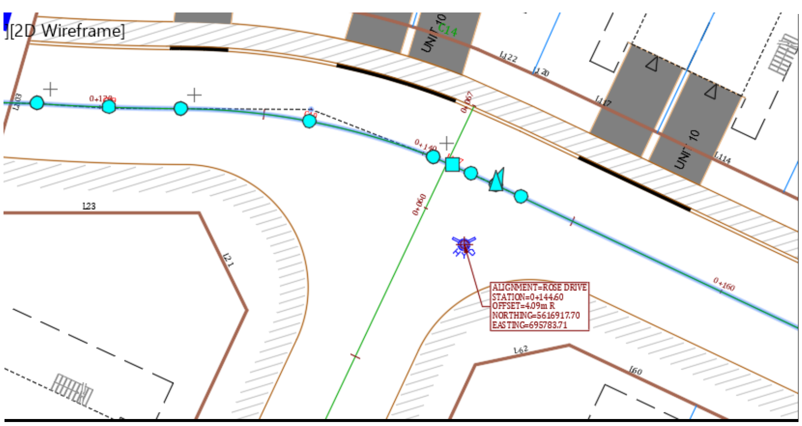
8. In the style editor window, on the **Data Properties** tab, check the **Sort Data** case and choose by **Ascending** order.



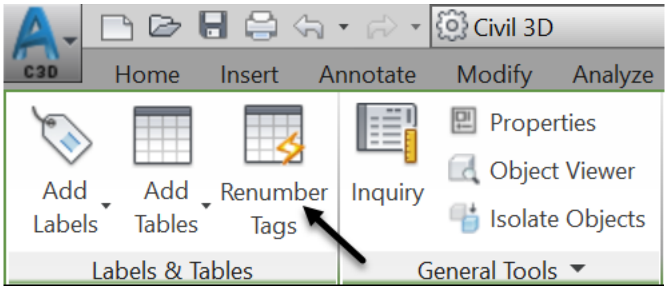

9. Next, click **OK** to close the table style editing window. The curves and tangents are now sorted and displayed separately.

| Rose Drive | | | | |
|---|---|---|---|---|
| Number | Radius | Length | Line/Chord Direction | A Value |
| C35 | 90.19 | 19.45 | S53° 22' 31.66"E | |
| C36 | 383.33 | 36.53 | S62° 16' 58.48"E | |
| C37 | 225.65 | 33.24 | S69° 13' 58.90"E | |
| C38 | 100.68 | 26.92 | S81° 11' 51.65"E | |
| C39 | 145.93 | 8.76 | S87° 58' 16.49"E | |
| C40 | 42.90 | 15.74 | S79° 10' 35.78"E | |
| C41 | 50.10 | 6.89 | S68° 35' 16.03"E | |
| C42 | 142.93 | 3.34 | S65° 18' 59.24"E | |
| C43 | 142.93 | 6.78 | S73° 53' 16.28"E | |
| C44 | 142.93 | 4.96 | S74° 15' 09.14"E | |
| L4 | | 2.50 | S65° 59' 08.37"E | |
| L18 | | 92.40 | S64° 38' 50.11"E | |
| L24 | | 29.69 | S72° 31' 41.95"E | |
| L78 | | 28.02 | S75° 14' 50.62"E | |
| L79 | | 127.82 | S73° 15' 27.65"E | |

10. If needed, the tags can be renumbered according to your preference. To **renumber** them, simply select the alignment in the drawing area.



11. Then, on the ribbon, run the **Renumber Tags** command. Renumbering tags is typically done when all of the design is completed, and all tags in the drawings should be numbered in ascending order.




NOTES

NOTES

# 8 PROFILES

## 8.1 Introduction

In the previous lesson, we have learned how to create **alignments**. They give us 2D information along a path. Profiles are the tools that give us the third dimension or elevation along the alignment. For this reason, profiles are often called **vertical alignments.** Whereas the alignment itself is called **horizontal alignment**.

Therefore, a profile must always be associated with an alignment.

In general, we will have two main types of profiles: a **surface profile**, created from a pre-existing surface and a proposed or **layout profile**. Both profiles can be superimposed in a common view. This gives us an overall view of existing and proposed conditions. Surface profiles can be dynamic (with automatic updates that reflect changes made to the associated surface). In some rare circumstances, we may also choose to make them static, without any updates.

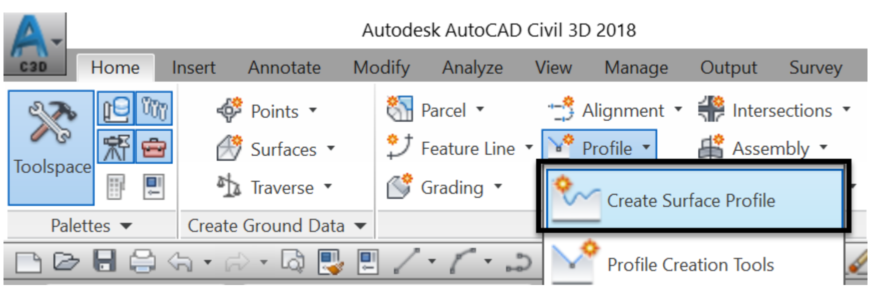
## 8.2 Surface Profiles

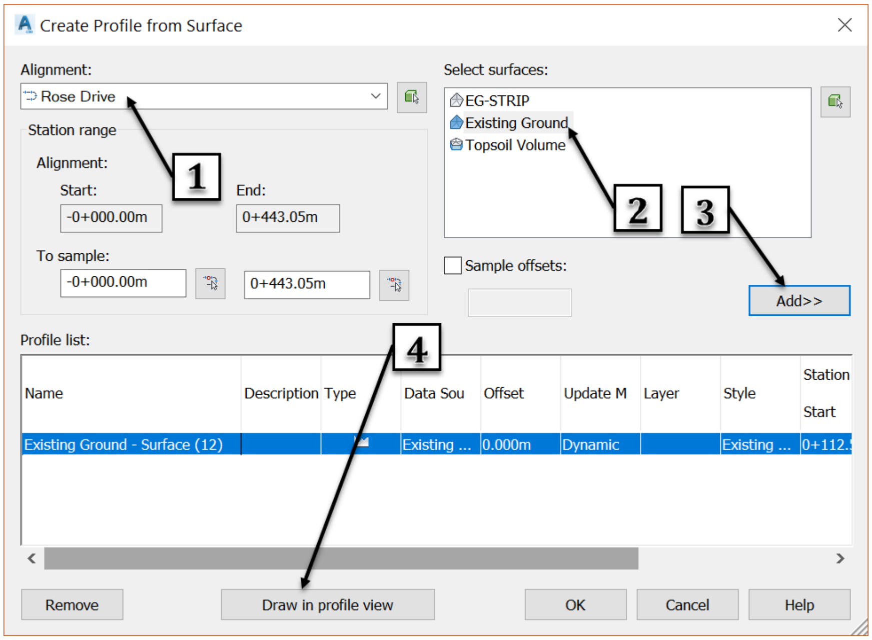
Let's practice how to create profiles.

1. First, Open the **08.01-Profiles dwg** file in **Lesson 08** practice folder.

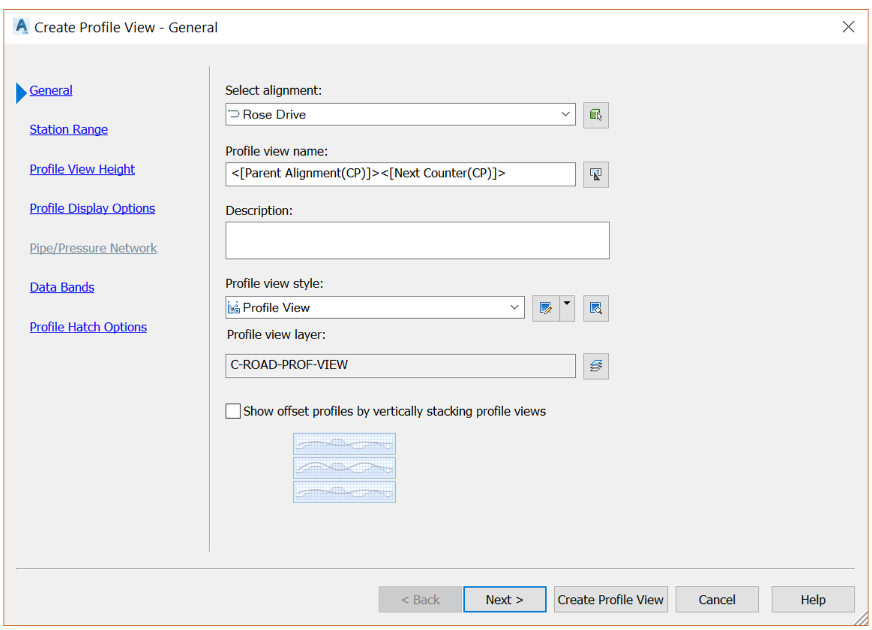
2. In this file, a few surfaces, **EG-Strip**, **Existing Ground** and **Top Soil Volume** have already been created during the previous lessons.

3. From the ribbon, create a **Surface Profile** for the **Existing Ground** using the **Rose Drive** alignment.



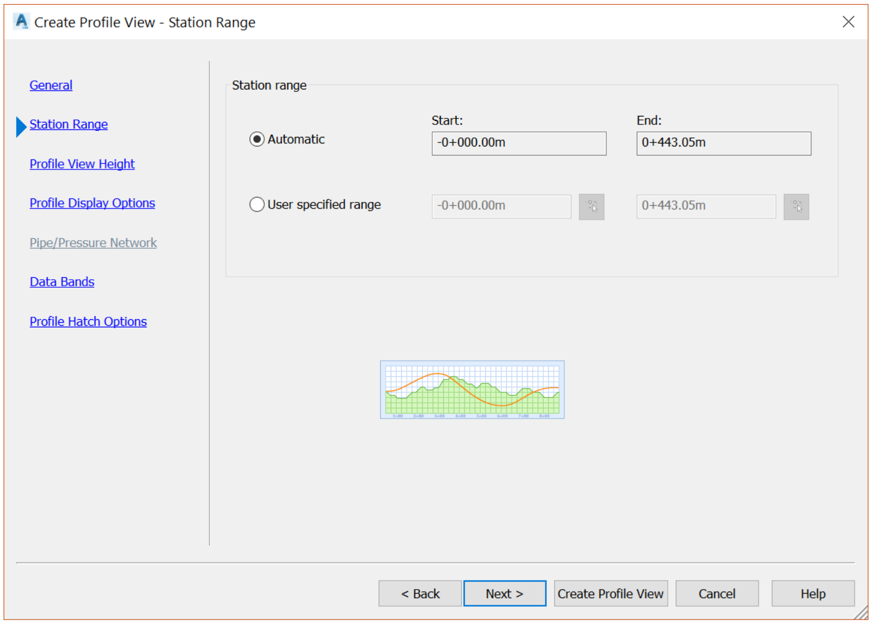



4. On the **General** page, you can specify basic information about the profile view. This includes the parent alignment, the profile view name, description, style, and layer. Click **Next** to accept. Later, you will be able to change any parameter you wish to adjust.

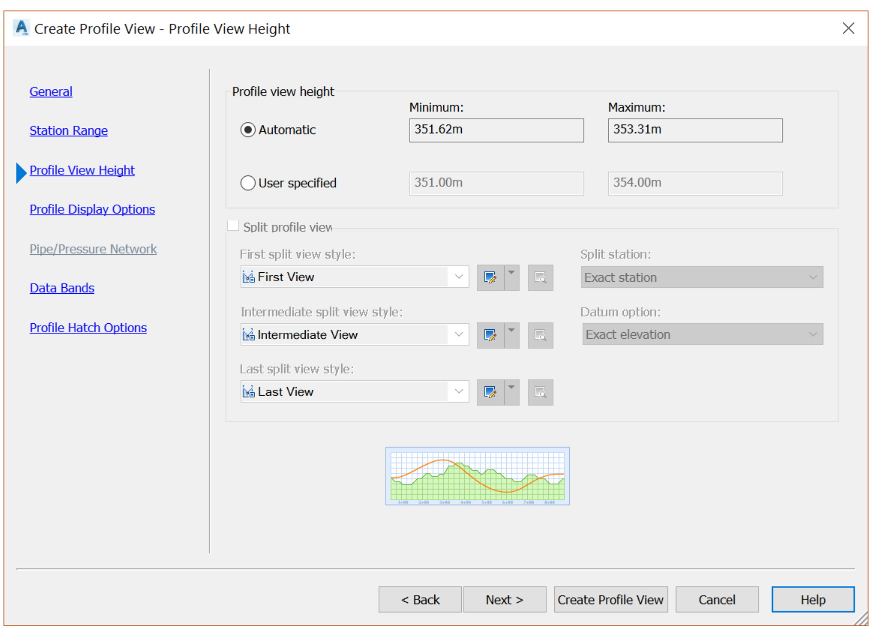



5. On the **Station Range** page, you can specify the station range to which the profile view is drawn. The **Automatic** option specifies the full station range for the horizontal alignment. Whereas, **User Specified Range** limits the station range in the profile view to your specified values. Click **Next** to accept the current values. Later, you will be able to change any parameter you wish to adjust.

NOTES



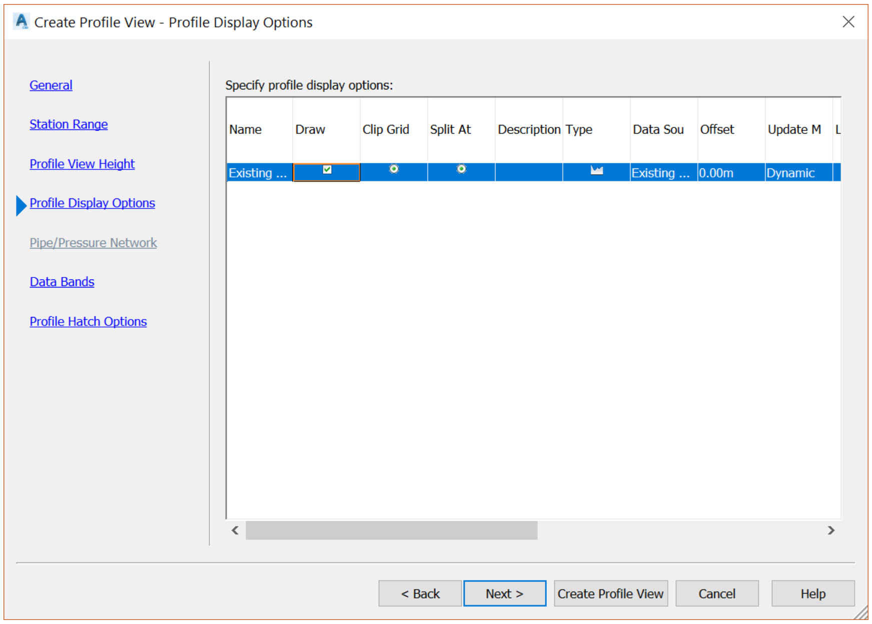

6. Next is the **Profile View Height** page. It specifies the profile view height and any split profile view settings. The **Automatic** option means the full height of the highest surface profile. The profile view includes a buffer region above the maximum and below the minimum elevations. We will see how to set it in the profile styles. The **User Specified** option means you are specifying the height to which the profile view is drawn. If a surface profile extends beyond the user-specified value, it may be clipped. So, you need to have an idea of existing elevations before setting a **User-Specified** elevation. Click **Next** to accept current values. Once again, later, you will be able to change any parameter you wish to adjust.




7. Next, we have the **Profile Display Options** page. Here we can view and change the settings for all the profiles associated with the parent alignment of the profile view. Click **Next** to accept

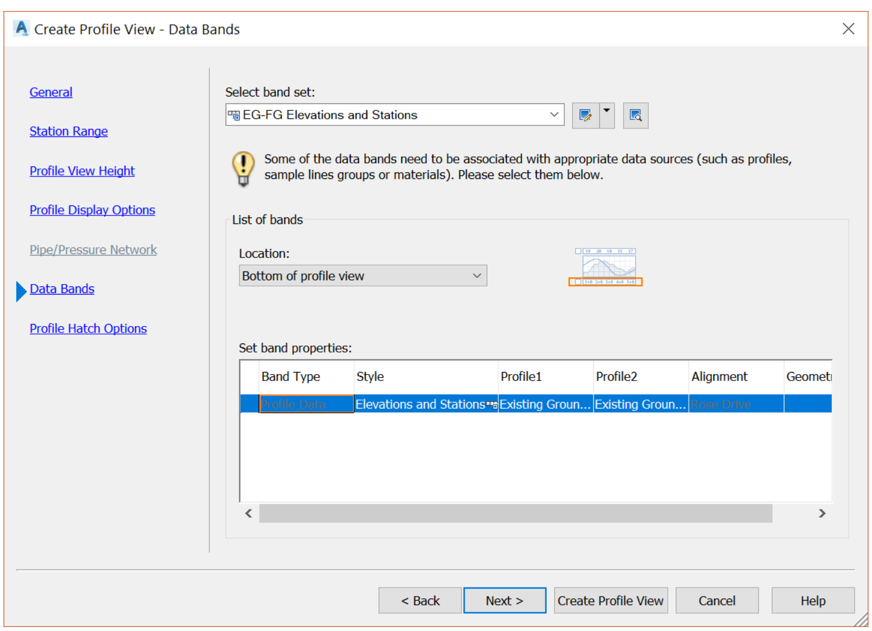
NOTES

current values. As usual, later, you can make changes to any parameter if you wish to adjust.




8. We currently don't have any **Pipe Network** in the drawing. So, the Pipe and Pressure Network page is grayed out and skipped.

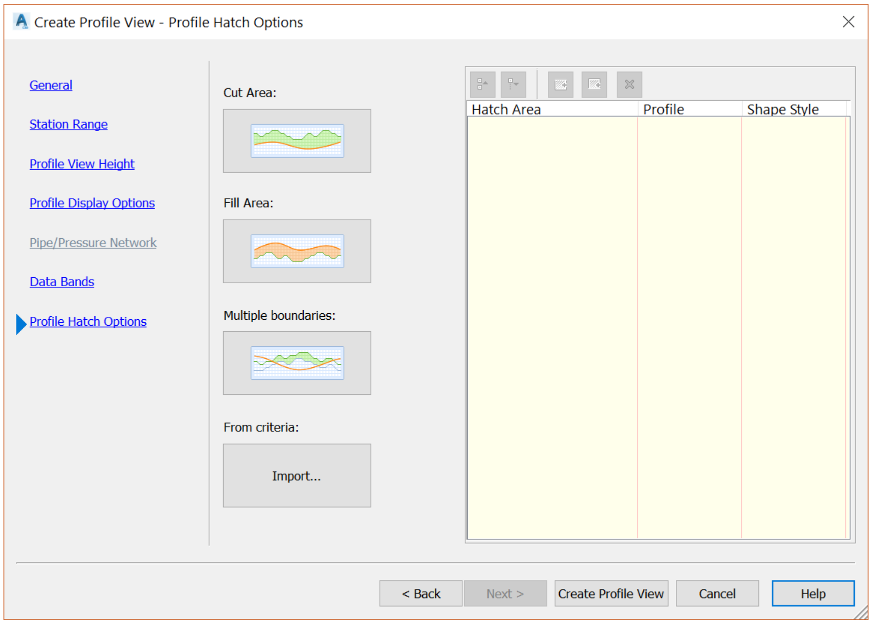
9. On the **Data Bands** page, we can specify the properties of the data bands associated with the profile view. For example, we have by default added a **Profile Data Band Type.** It will display **Profile1** and **Profile2** elevations, at major profile stations. We can assign **Profile1** to **Existing Ground** and **Profile2** to **Design Ground** when we create the design grades later.



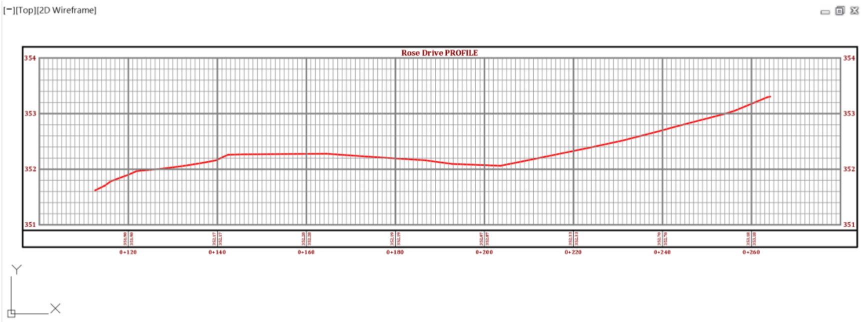

NOTES

NOTES

10. Finally, on the **Profile Hatch Options** page, we can add hatch areas to display cut and fill regions in a profile view.




11. Click **Create Profile View** to select a point in the drawing area where the lower left corner of the profile view will be inserted.



NOTES

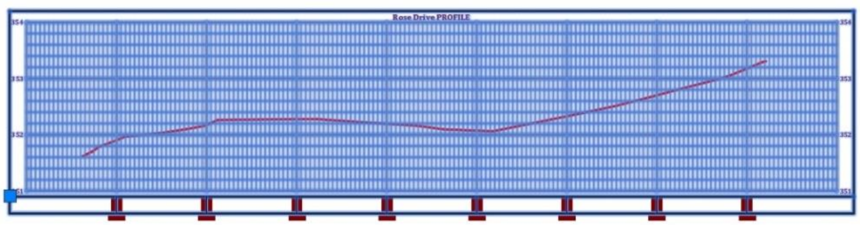
## 8.3 Proposed Profiles (Layout Profile)

The next step is to create the design of the road. Unlike a **Surface profile,** which represents a pre-existing surface, the proposed profile must be designed. This is done according to established standards. This includes horizontal and vertical geometry design for safe driving conditions.

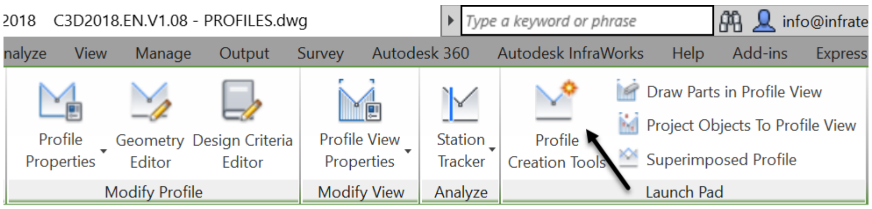
Minimum requirements for roadway design are usually spelled out in design manuals. They are typically published by the federal transportation authority. Sometimes, a stricter version of the requirements is enforced at the local level. Among such requirements are speed limits, safe driving sight, and the like.

To create the design profile (also called proposed profile), we need to proceed as follows:

1. First, select the **Rose Drive** profile view we have just created.

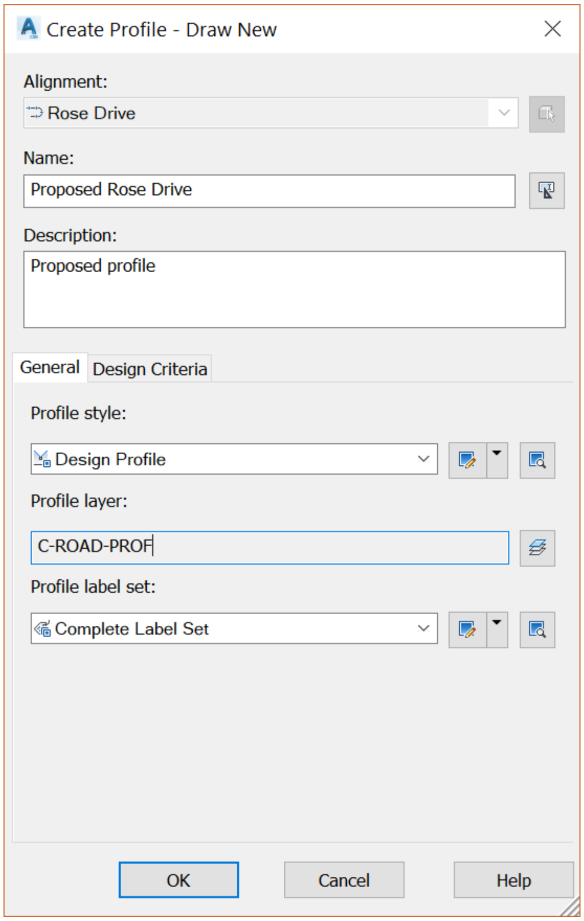


2. Then, from the Ribbon, launch the **Profile Creation tools.**

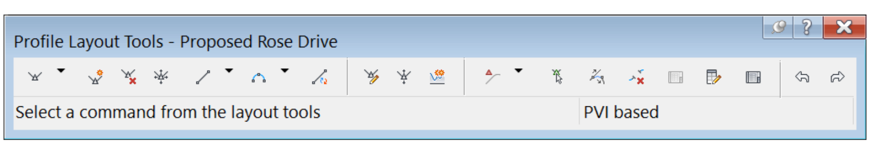


3. Afterward, enter the profile creation settings in the **Create Profile** dialog box. In particular, let's specify,

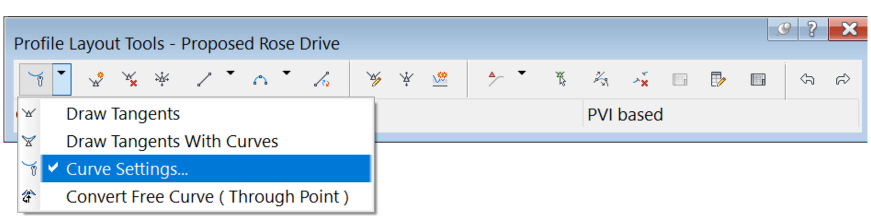
- The **Name** of the profile
- The **Profile Style**
- and the **Label Set** to use




4. Click **OK,** and the **Profile Creation Tools** will appear on the screen. If it looks familiar, it is because we have just used something similar, the **Alignment Creation Tools**, which has analogous commands.

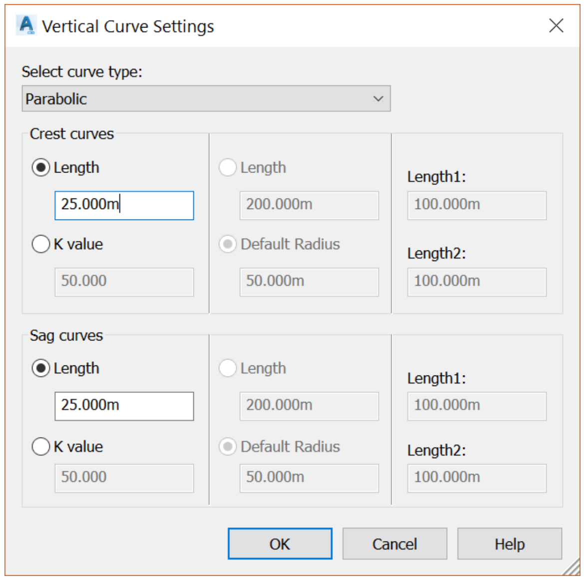



5. Now, let's set curves and tangents design parameters.

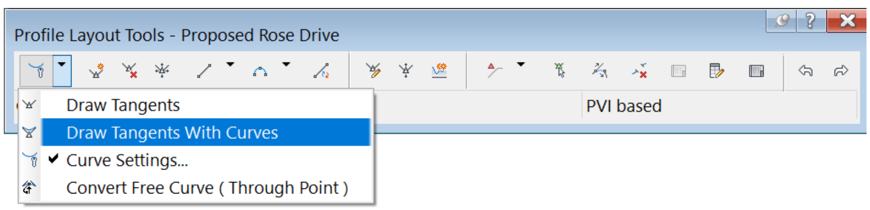



NOTES

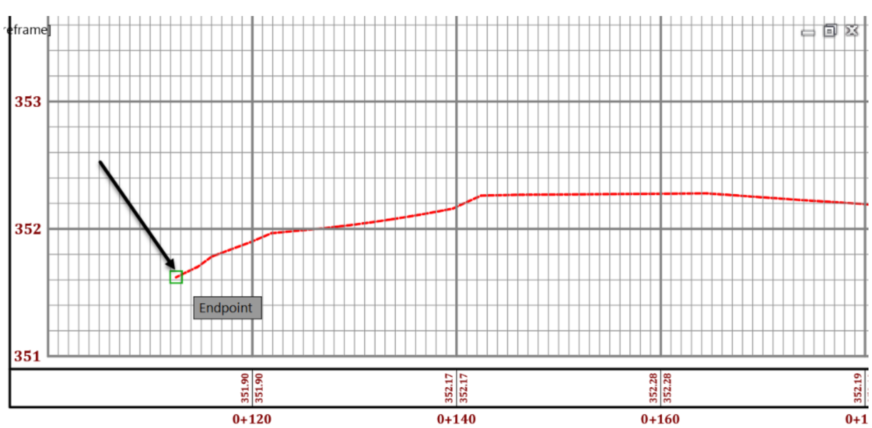
6. In the **Vertical Curve Settings** dialog box, use **25m** or **75ft** for Crest and Sag Vertical Curves. Alternatively, we can use the **K** value, which is mathematically linked to the curve length.




7. Now let's create tangents and curves for the proposed profile.



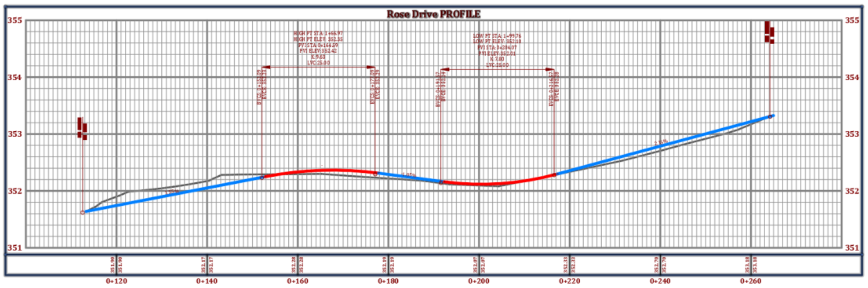

8. Click the start point of the existing ground profile. Make sure you have your **End** object snap option activated or simply type the alias **end** at the command line. We are clicking on the existing ground because we are trying to match the existing ground at the tie-in point. Ideally, we would like to have the road grade at that location so that we can carry it out a few tens of meters or feet before any change of slope. For now, let's assume that the slope that we are creating is the same on the other side of the project.




NOTES

NOTES

9. Next, click on a few points (4 or 5) along the existing ground profile. During this process, try to minimize cut and fill volumes, meaning we don't want to be too high or too low compared to the existing ground. We want to have something like this.

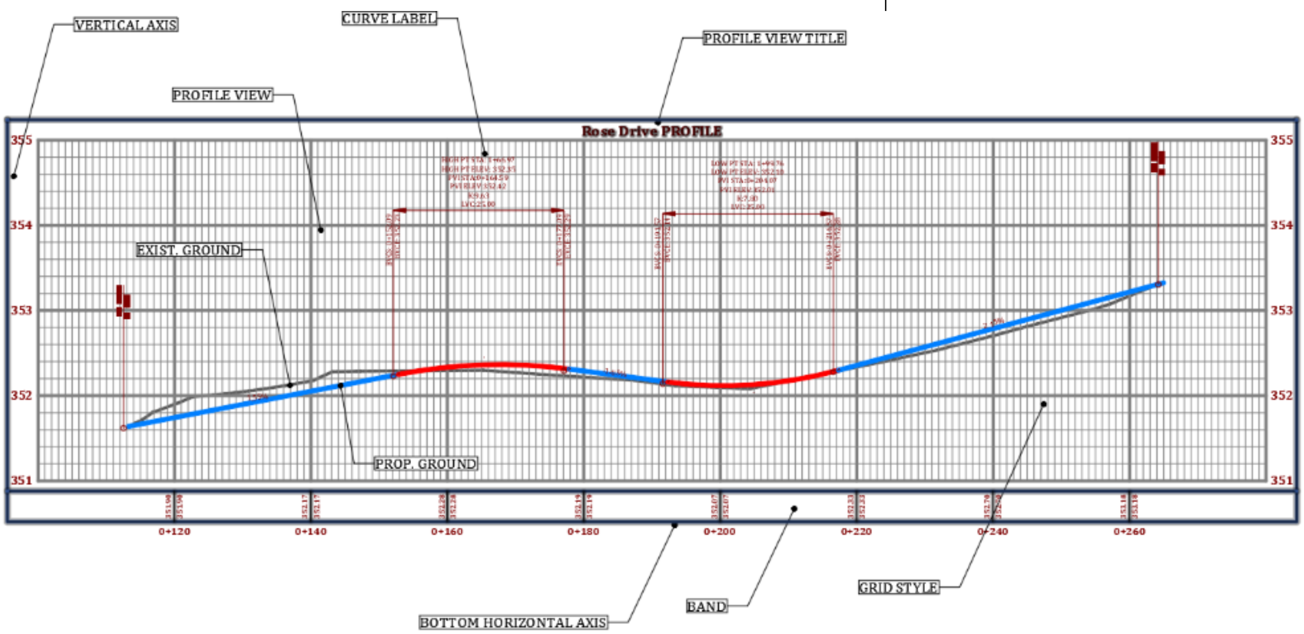



## 8.4 Profile Styles

Like most of Civil 3d entities, profiles and profile views are controlled by styles. The **Profile styles** set the visibility and format of profile components, such as lines, curves, arrows, and line Extensions. To access the profile style editor:

10. Select the profile in the profile view

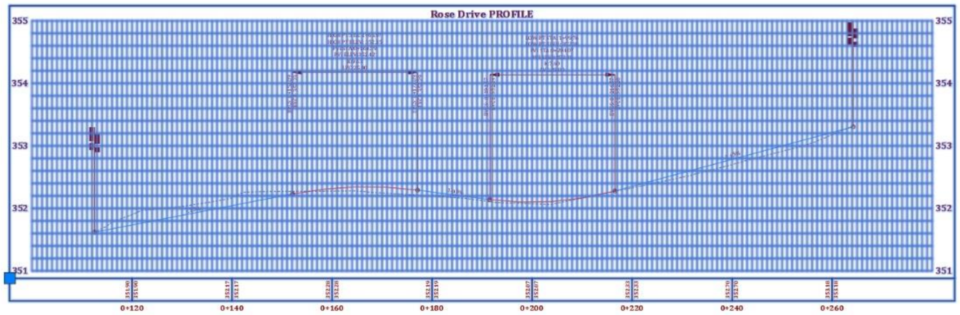
11. Right-click and select **Edit profile style**



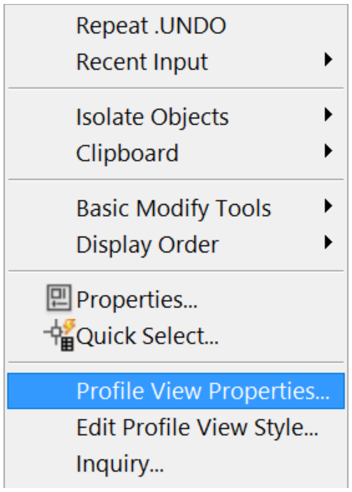
## 8.5 Profile Views

To assign or edit the style of the profile view we need to:

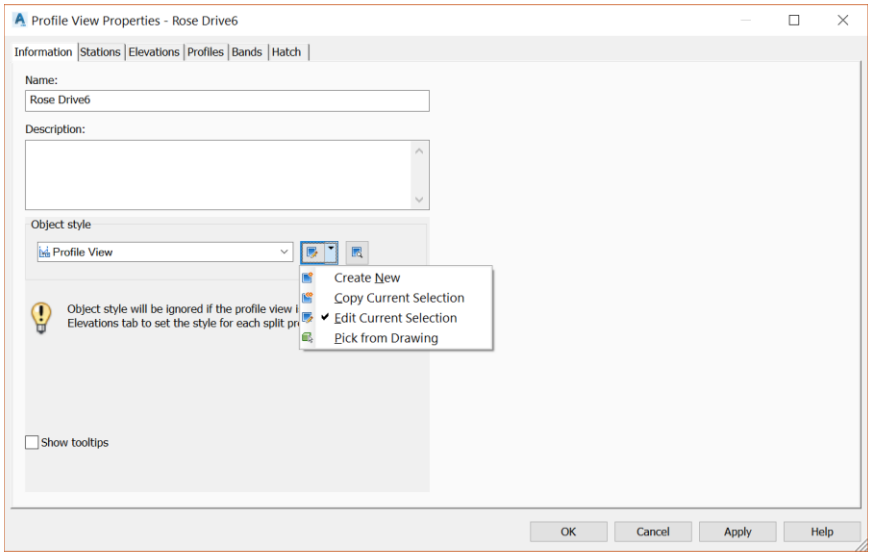
1. First, select the profile view in the drawing.




2. Right-click and choose **Profile View Properties.**

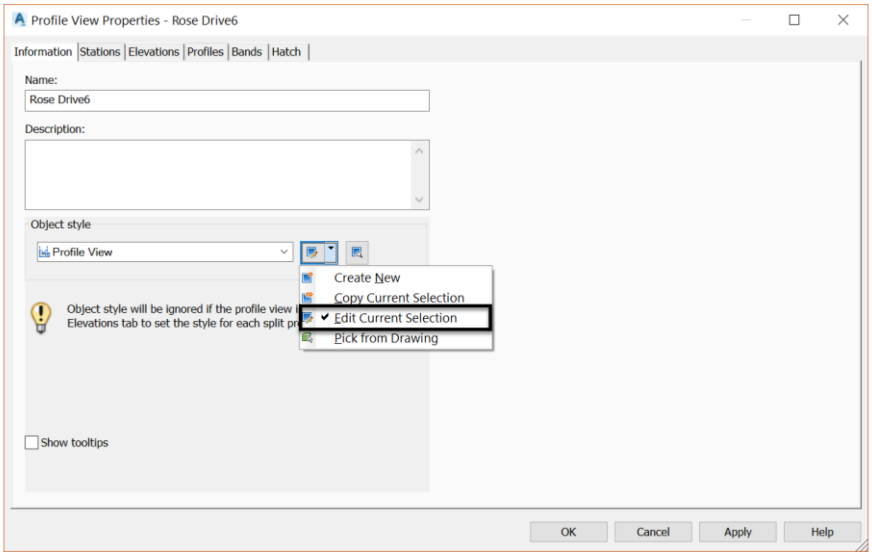



3. On the **Information** tab, we can either assign a pre-existing style, create a new one, or copy another style, then modify it to fit our needs.



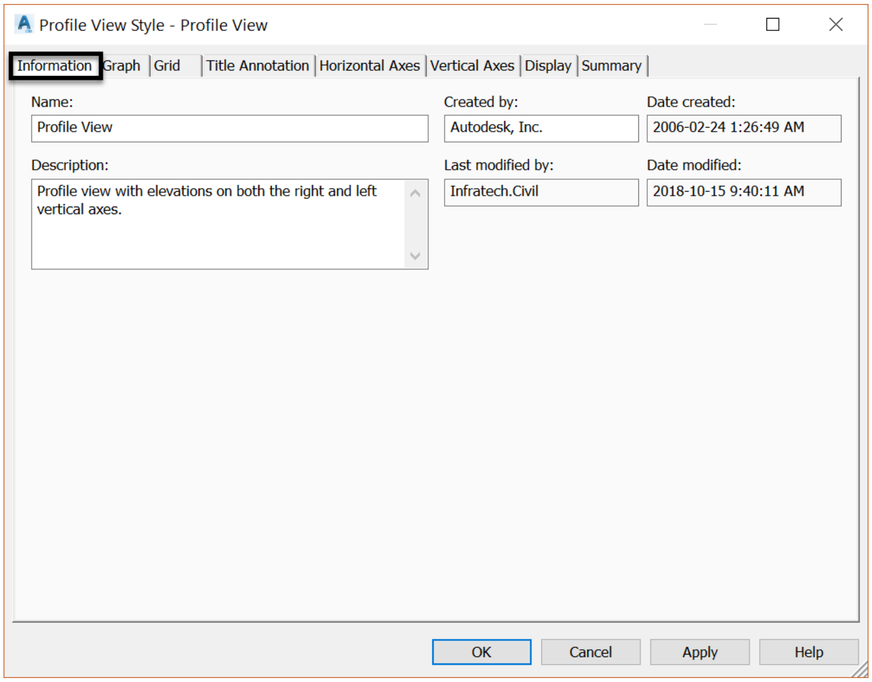

NOTES

4. Let's choose to **Edit current selection** and modify the current style.



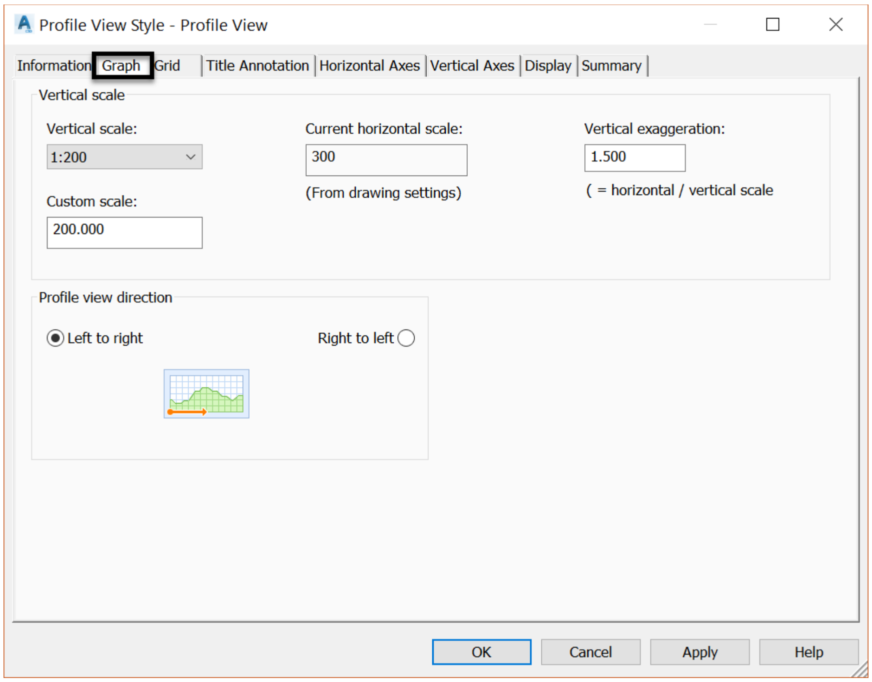

5. The **Profile style Editor** opens, and we can make all needed modifications to display the profile view as we need.

6. The first item is the **Information** tab. Here we can record basic information about the profile view style. That includes the name of the person who created the style, the date of creation and modification, etc.

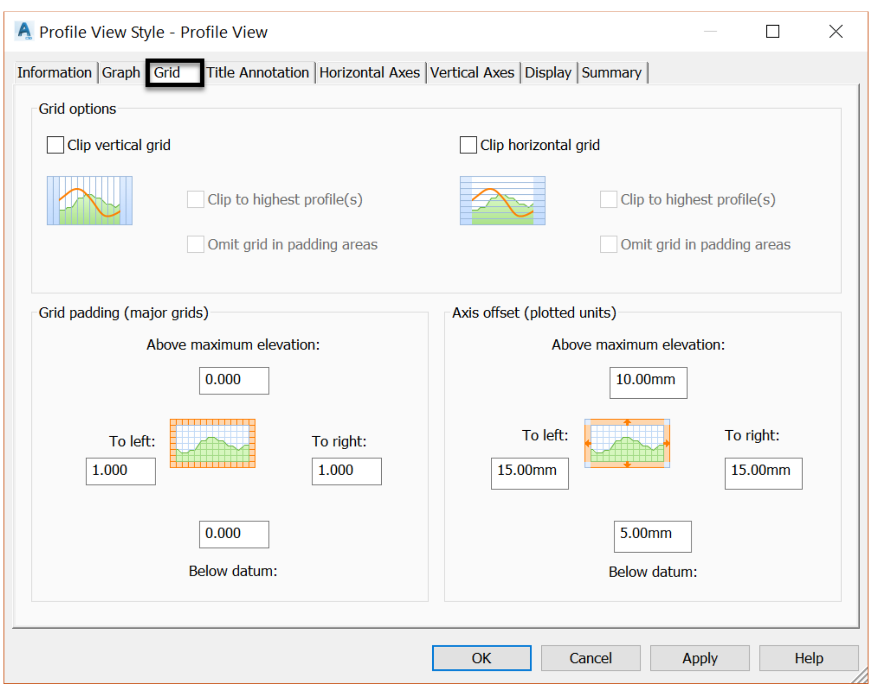



NOTES

7. **Next** is the **Graph** tab: you can use this tab to set the vertical scale and direction of the profile view. The vertical scale controls how much you want to increase the elevation to improve visibility in the profile view. This does not affect elevation data per se. It is strictly a visibility feature. This can be done either by vertical exaggeration or simply by entering a scale value. When one is used, the other is calculated automatically. Profiles are typically drawn from left to right, but situations may arise where you need to go from right to left.

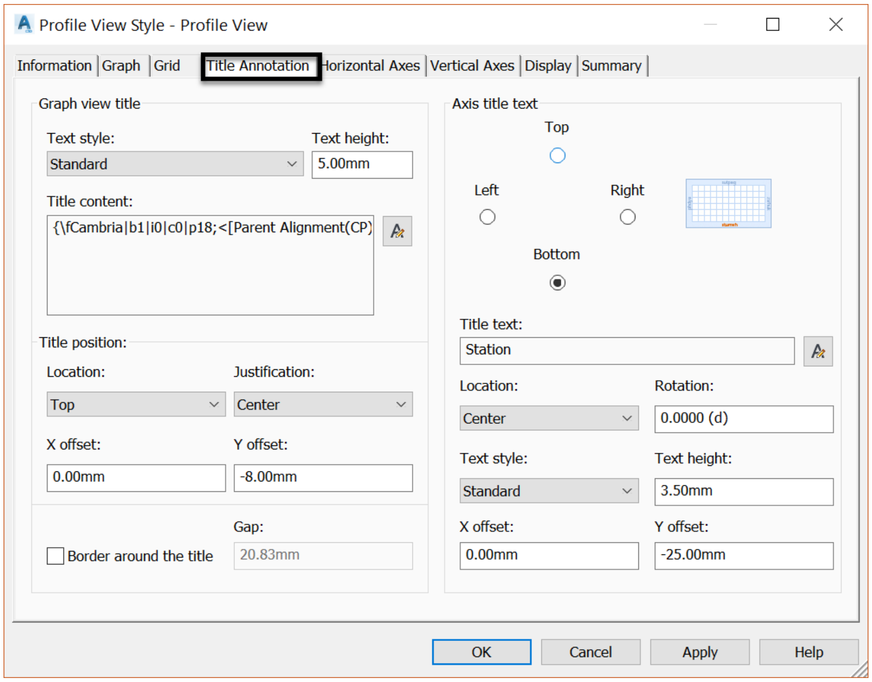



8. The **Grid tab** mainly controls the clipping, padding, and axis offset options on the profile view grid.

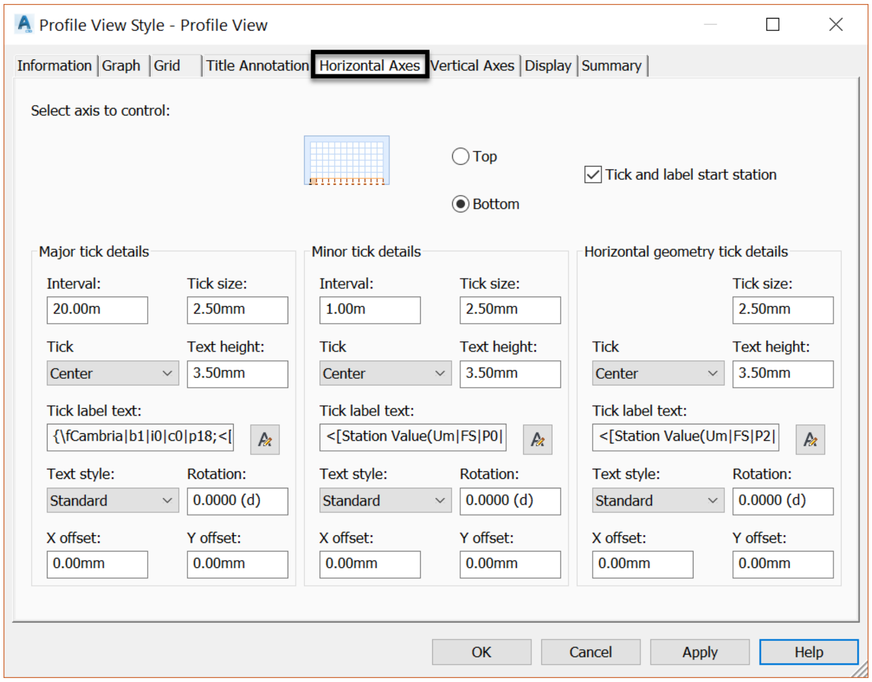



NOTES

9. The next tab is **Title Annotation.** Here we can specify settings for the graph view, including graph and axis titles with options. That includes text content, height, style, rotation, offset and border style.



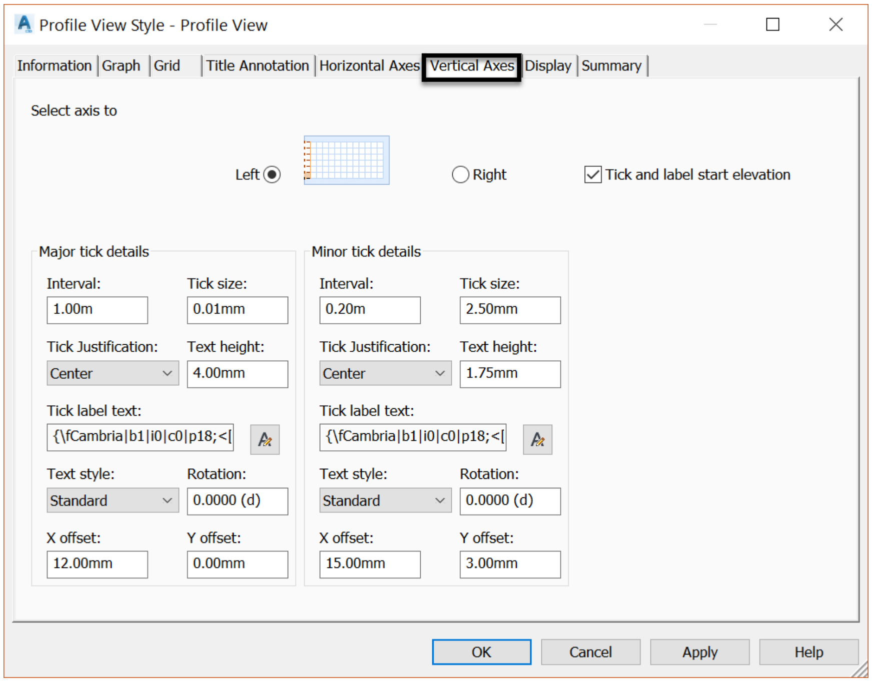

10. Next, we have the **Horizontal Axes** tab. We can use it to specify the settings for the major and minor stations such as the text and tick marks on the horizontal axes of the profile view.




NOTES

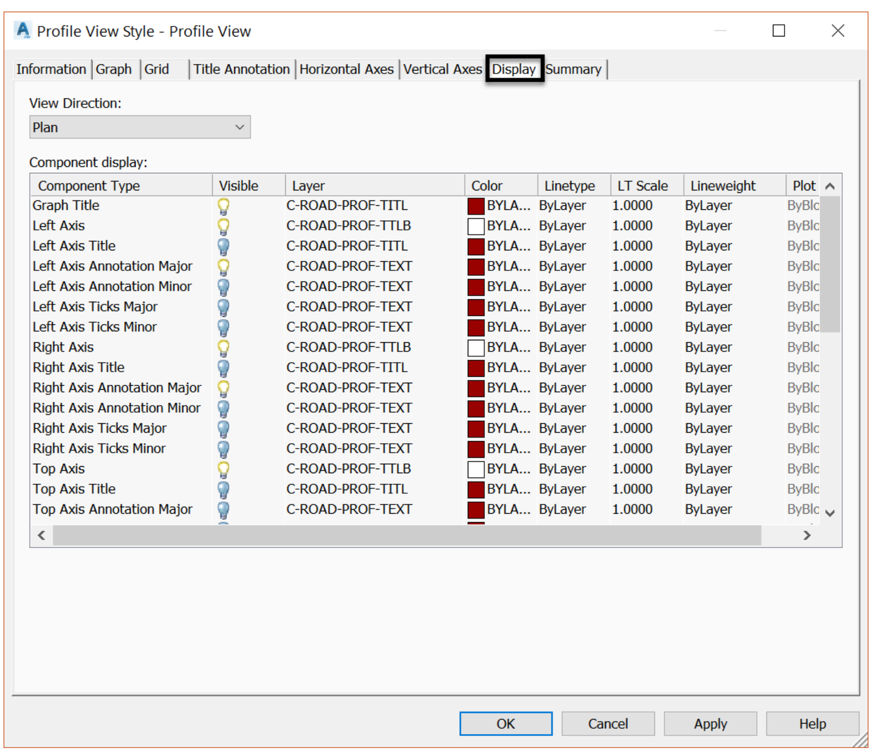
NOTES

11. The next tab is **Vertical Axes**, where we can specify the settings for the text and tick marks on the vertical axes of the profile view. We can pick left or right vertical axis to adjust the Major and Minor elevation Details.




12. Next to last is the **Display** tab where we can manage the display of profile view components in 2D and 3D. On this tab, we can turn on or off most of the settings that we saw in the last few tabs.

13. Finally, we have the **Summary** tab where we can review and adjust all of the settings we have seen for the profile view style.

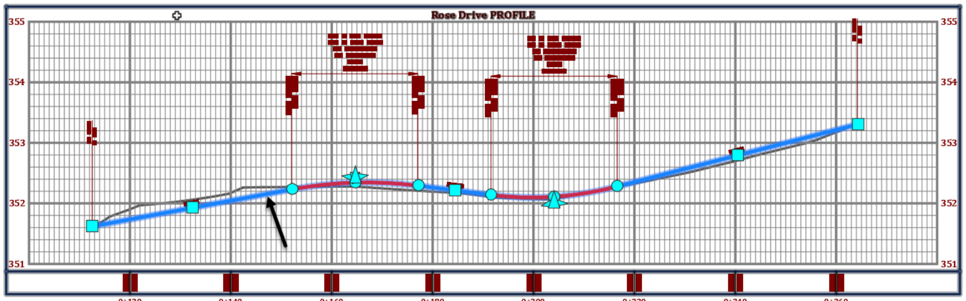


## 8.6 Profile Style

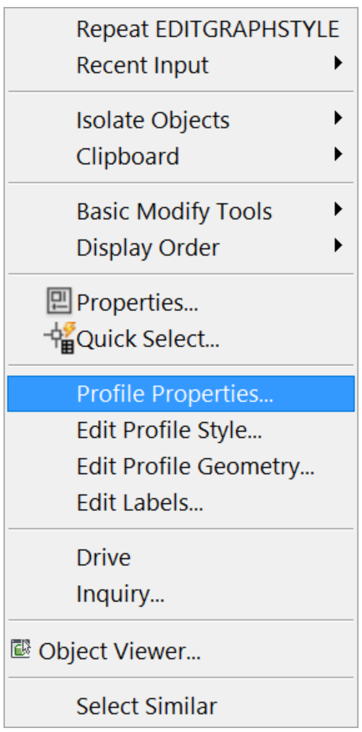
The profile is the line representing the vertical alignment. The profile style is therefore different from the profile view style we just talked about.

To assign or modify the style of a profile,

1. First, select the profile in the view.



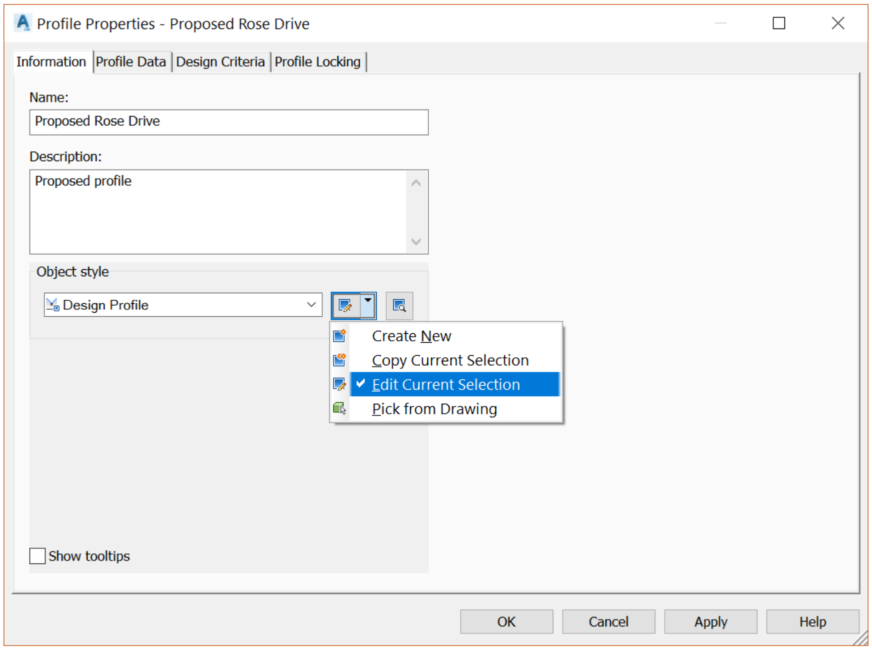

2. Right-click and select the **Profile Properties**



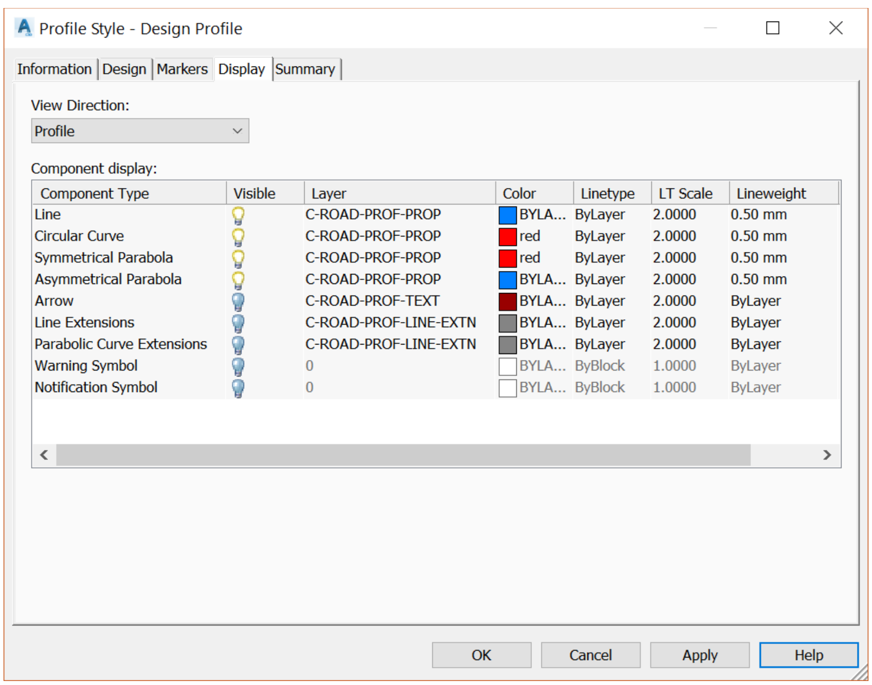

NOTES

NOTES

3. On the **Information** tab, in the object style section, we can apply or modify a pre-existing style, create a new one or apply a style from the drawing. Let's try to **Edit the Current Selection**.




4. On the **Display tab**, we can set the visibility and format of profile components, such as lines, curves, arrows, warning symbols, and all the others.

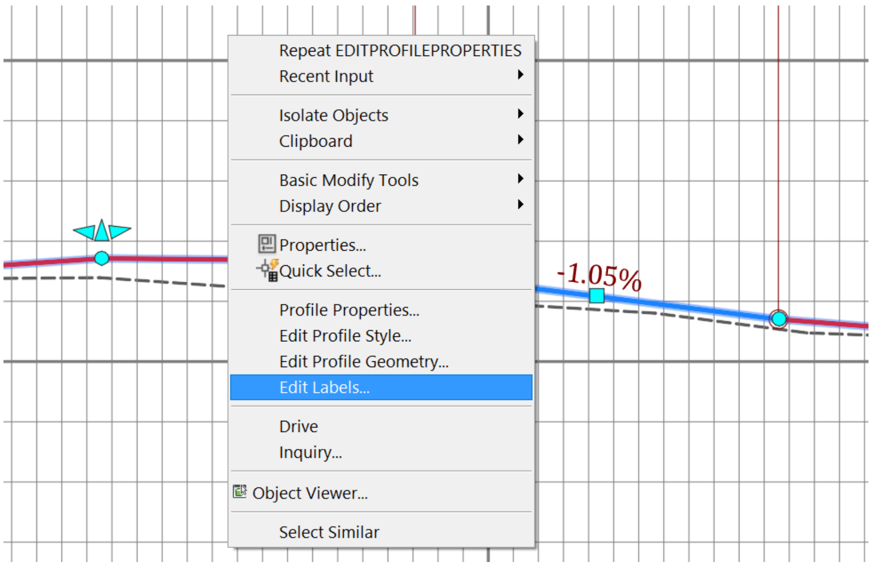


## 8.7 Profile Label Styles

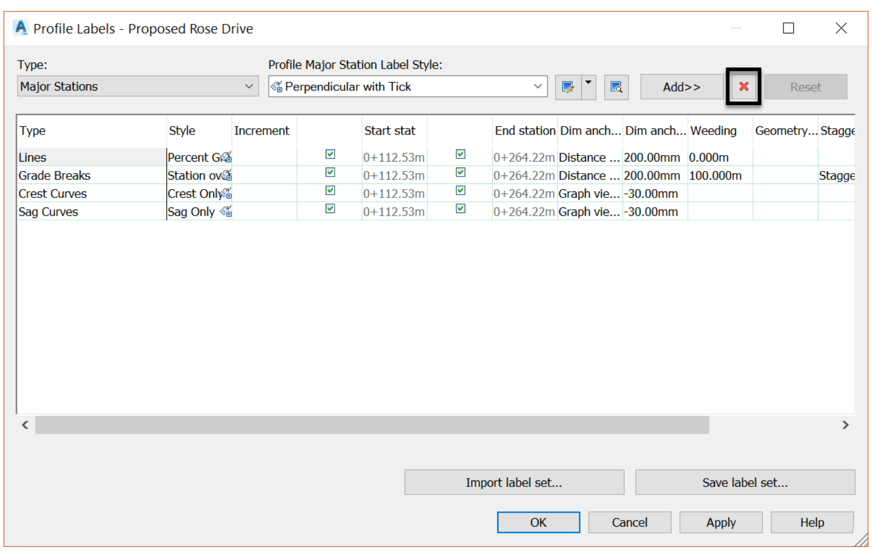
When creating profiles, we have the option to create label styles for the profiles. If for some reason we have not attached labels to the profile, we can still add them later if needed.

Since we already added ours, lets remove them and show how to create them had they not been added at the profile creation stage.

1. To remove the profile labels, right-click on the profile line and select **Edit Labels**.

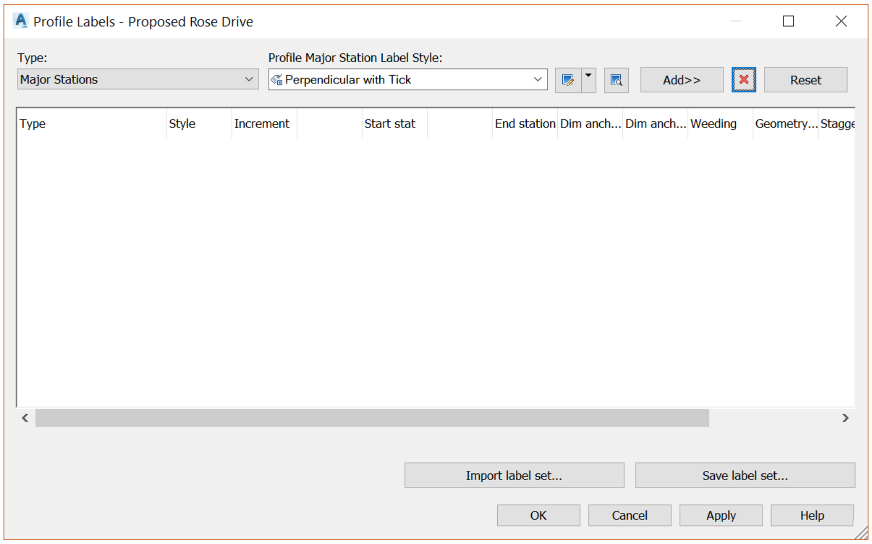



2. Click on the **X** sign as many times as necessary to delete all the labels present.

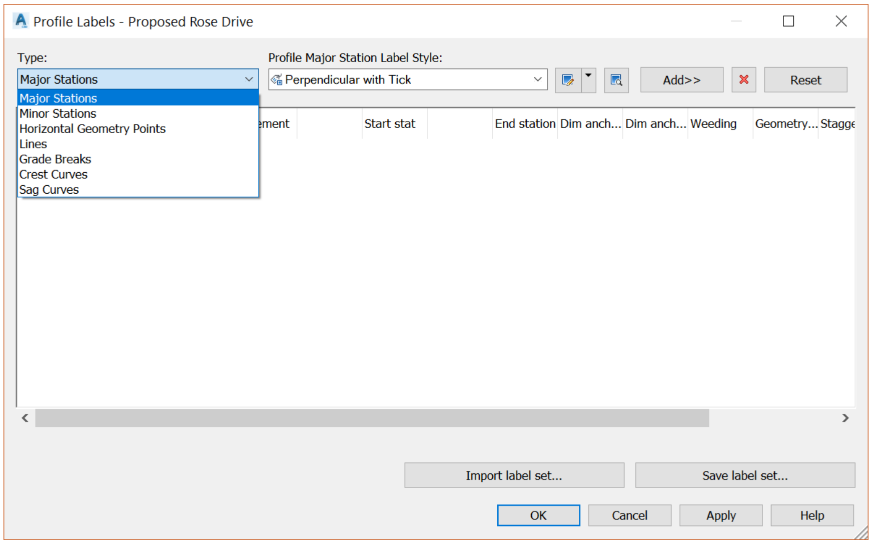



NOTES

3. When you are done, the **Profile Labels** window should look completely blank.



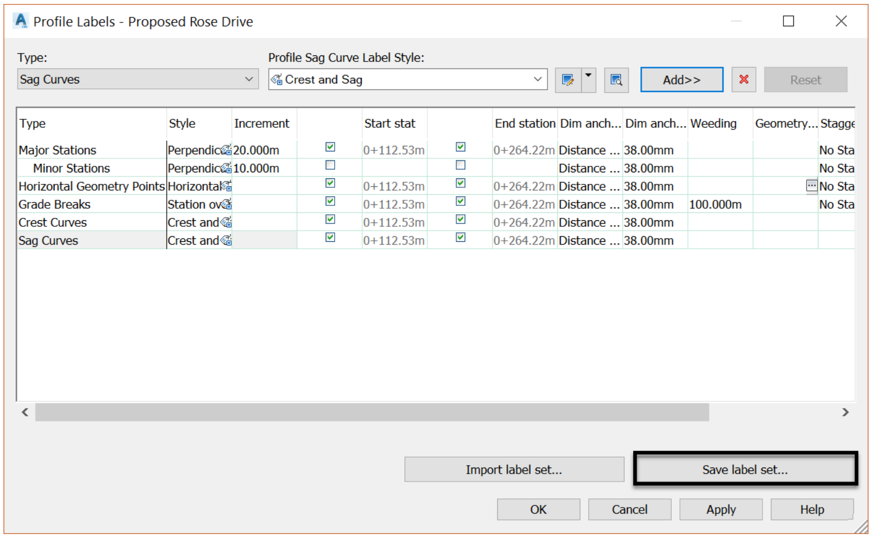

4. Now, in the **Type** drop-down box, we can specify the type of label to add to the label set. Once we pick a type of label, we can later define the properties for the label style. These properties vary depending on the type of label. We may have properties like stations, geometry points, lines, grade breaks, and vertical curves.



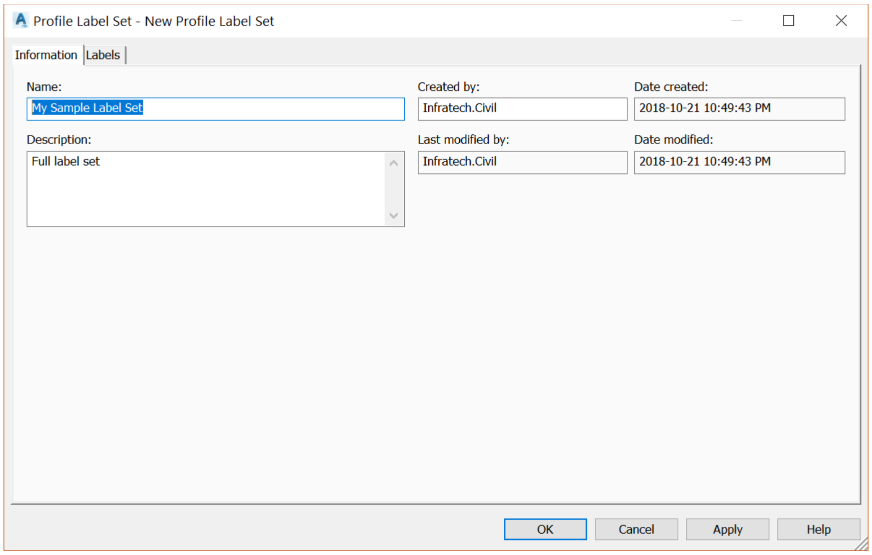

5. Let's go ahead and set a **Major Station** at every **20m** or **50ft** increment, minor stations at every **10m** or **25ft**. Let's also add labels at horizontal geometry points, grade breaks and curves.

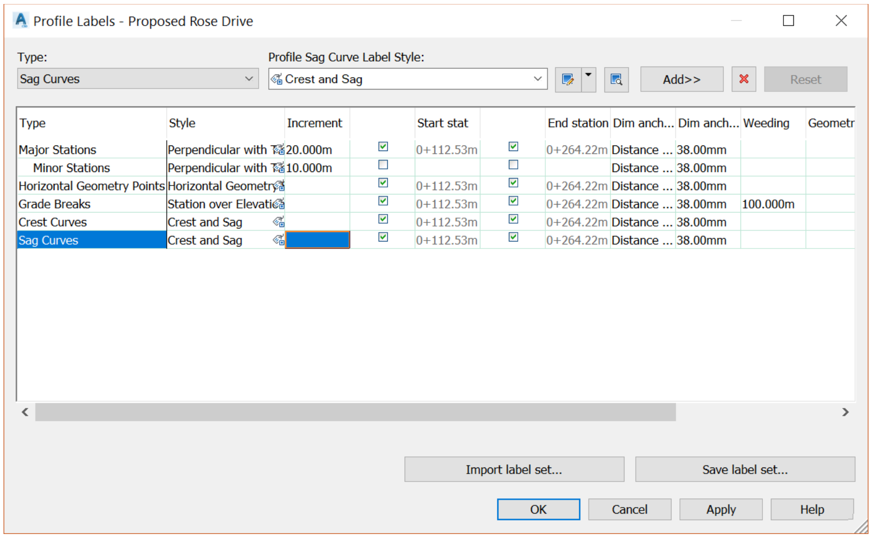
NOTES

6. If you are happy with your **label set** setup, you can even save it and use it on future profiles. To do that, simply click on **Save Label Set.**




7. Give it a name and click **OK.**



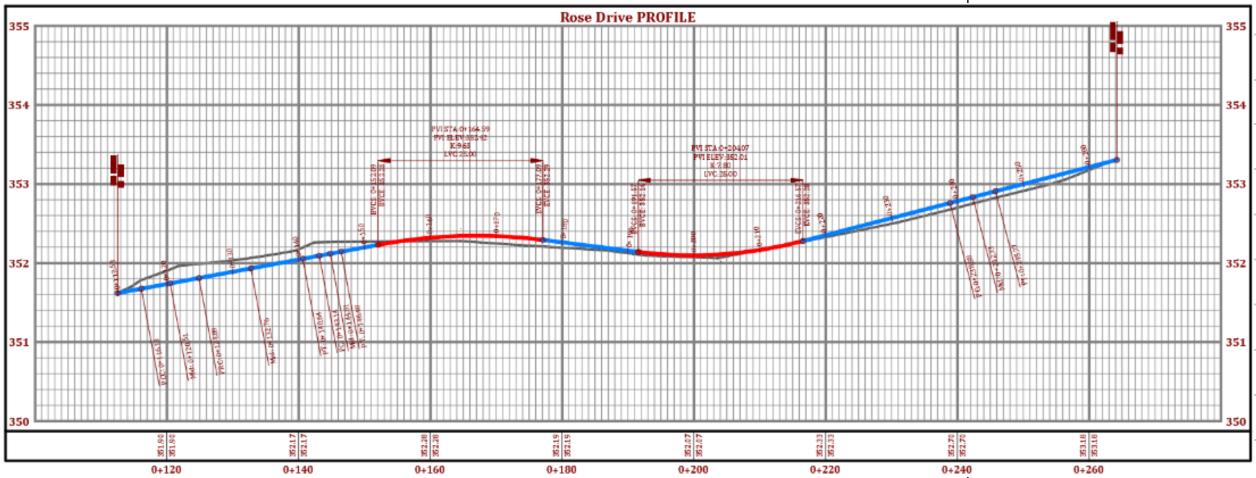




NOTES

NOTES

8. Click **OK** again to exit the **Profile Labels** window.

9. When you are done, you should have a profile view and profile labels that look as shown.



# 9 CORRIDORS

## 9.1 Introduction

We are often challenged with creating 3D models to make a flexible and advanced representation of our design. Civil 3D corridors allow us to create these models for design projects such as roadways, highways, railways and more.

The corridor brings together various Civil 3D components and integrates them into one monolithic 3D model. The three most basic constituents for a corridor are:

- an alignment or baseline,
- a profile, sometimes substituted by a feature line,
- a 2D cross-section of the entity being modeled, represented by an assembly.

Other Civil 3D objects and data can also contribute to the creation of a more complex and robust corridor model. Among others, we can use polylines, surfaces and profiles to improve a corridor.

Corridors are not typically printed on construction plans unless elaborate styles and code sets are used. However, they can be utilized to extract invaluable data. That includes points, surfaces for volume calculations, feature lines for grading, and many more.

## 9.1 Cross-section or Assembly

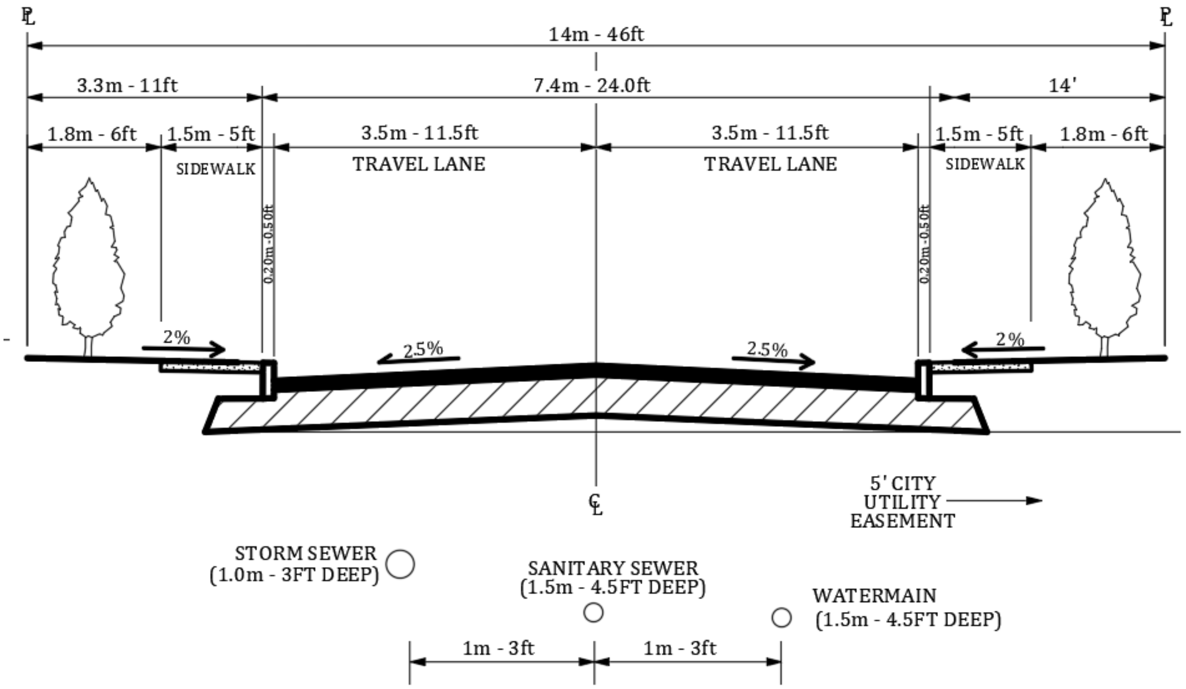
The assembly represents the cross-section of a given right of way. A typical section is almost always provided in the **design manual** by the permitting agency. Civil 3D provides us with tools called **subassemblies.** They can be leveraged to represent the components of a typical cross-section such as travel lanes, curbs, and sidewalks. We have recreated and will show later, the typical cross-section requirement for the municipality of **Flower Bay**. For a local minor road, like the ones in our subdivision, the requirements can be as follows:

- A travel lane width of **3.5m or 11.5ft** on both sides of the road centerline, at **2.5%** slope, followed by a **20centimeter or 6in** straight curb.
- Then, an abutting **1.5m** or **5ft** sidewalk, at a **2%** slope.
- Finally, a **1.8m** or **6ft**, at **2%** landscaping strip, extending to the property line.

In addition, we must also meet underground utilities depth and separation requirements.

NOTES

NOTES

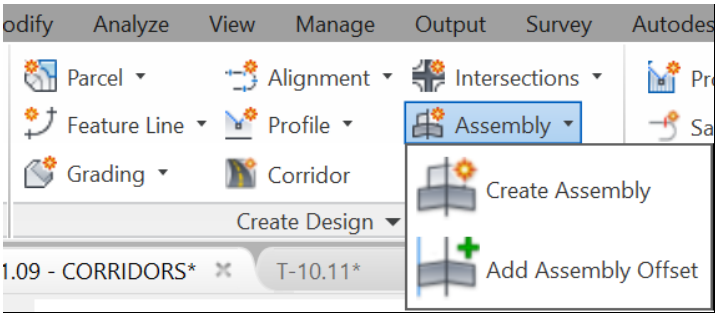



Once again, let's summarize. To create a corridor we need: a baseline or **alignment**, a feature line or **profile** and a typical cross-section or **assembly**.

So far, we have created the first two. Now, let's see how to create the typical cross-section, or an **assembly,** in Civil 3D parlance.

1. To begin, open the **09.01-Corridors dwg** file in **Lesson 09** practice folder.

2. Next, from the **Home** tab, on the **Create Design** panel, launch the **Create Assembly** command.



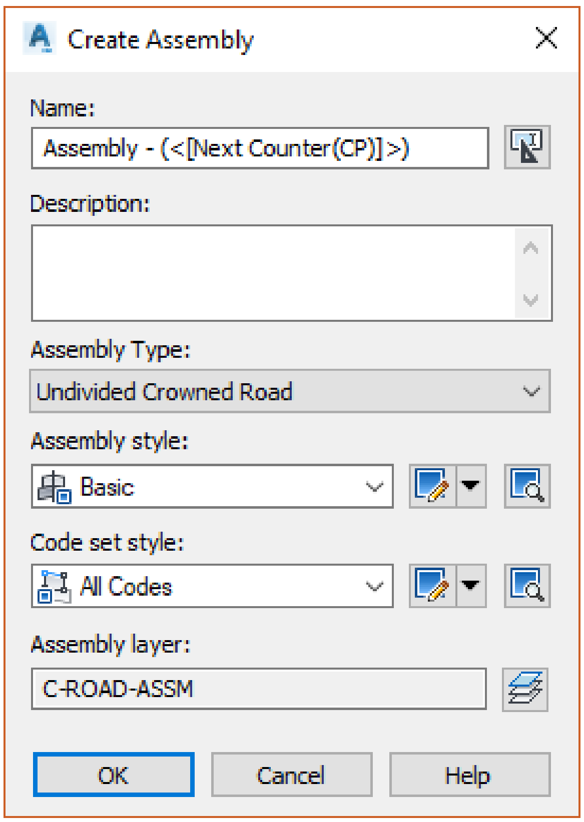

3. Then, give the assembly a name that indicates the situation in which it will be used. For example, call it **City of Flower Bay Local Street.** In the next window you can,

    - First, give an optional description of the assembly.
    - Then, specify the type of roadway that the assembly will support. For example, we may wish to apply the calculation of superelevation for the axis of rotation. Therefore, the assembly type must match the **Roadway Type,** on the superelevation calculation page. For the

NOTES

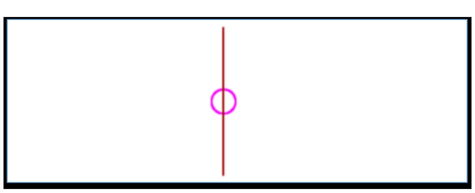
current project, we will not need any of that, since we are not creating any superelevation. Nonetheless, let's choose the **Undivided Crowned Road** type.

- Afterward, specify an **assembly Style** like we assigned an object style to the Civil 3d objects we have previously seen.
- Then, you have the **Code Set Style**, which will determine the styles of the different components of the corridors, like points, lines, and areas. This is different from the assembly style, which once again creates a style for the cross-section. We will better understand the difference in a moment when we see them in action.
- Next thing you can do is choose an **Assembly Layer**, which we will leave to the template to assign for us, by default.

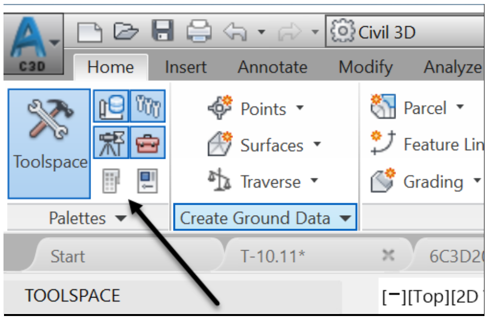



4. Press **Ok** to exit the **Create Assembly** dialog box.

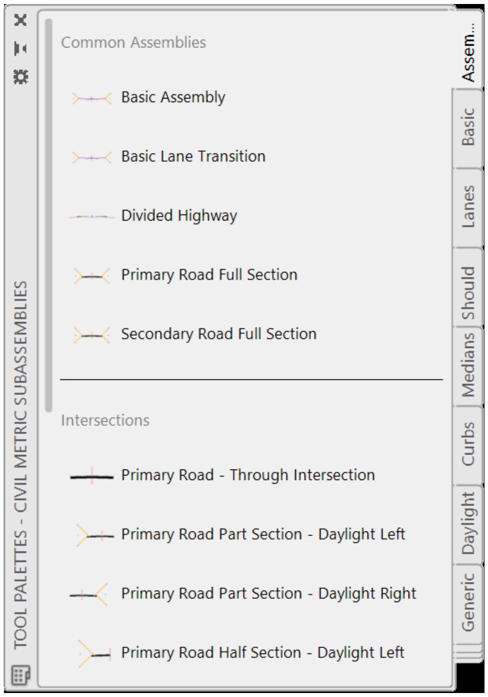
5. In an empty area of the screen, choose the insertion point of the **Assembly**. How the assembly appears (brown line with a magenta circle) is decided by the **Assembly Style.** Now we know what the assembly style we chose means. We can always right-click on it to edit the style to our liking.



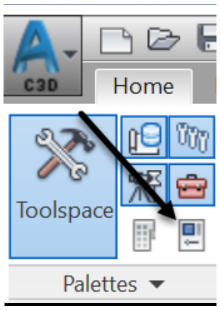
6. Once the assembly is created, we need to attach the **sub-assemblies** to it. They are the different elements that make up the cross-section. To create **sub-assemblies**, click on the **Tool palettes** (this is different from the **Toolbox** that is part of the prospector).




7. The **Tool palettes** opens:



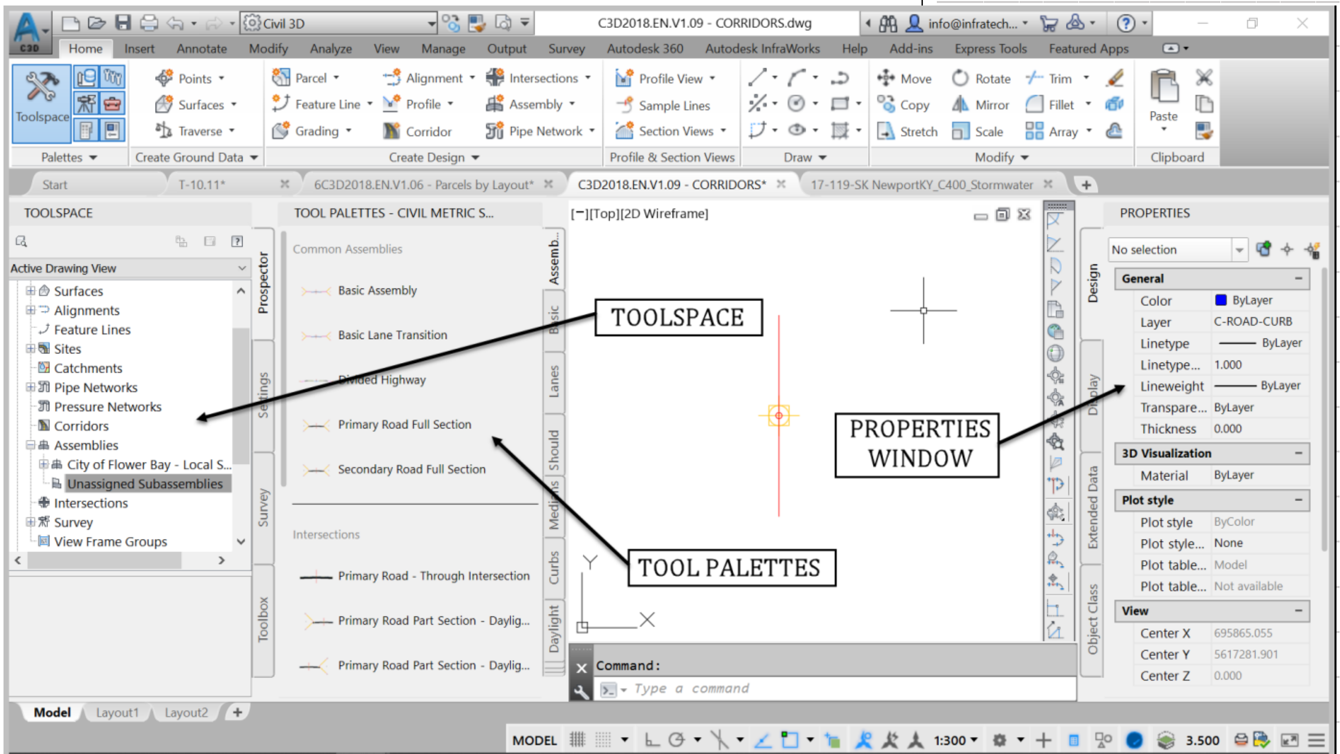

8. Now, display the **Properties window**, if it is not already opened.



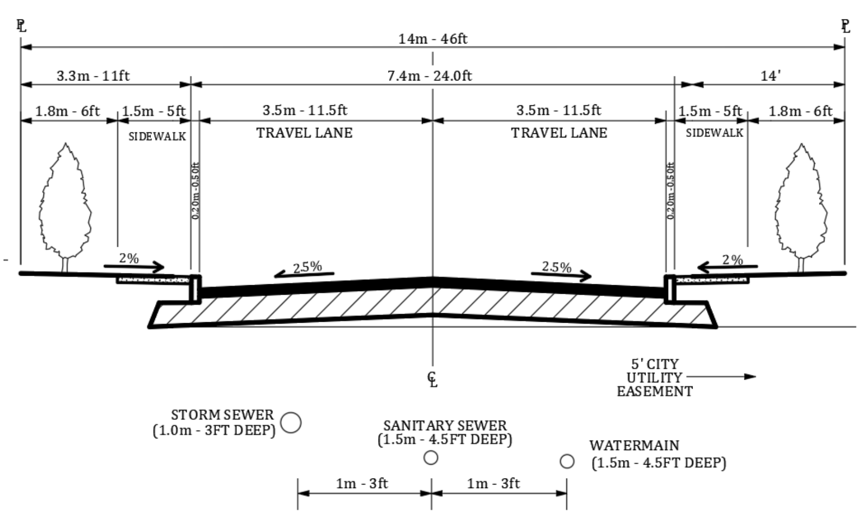

NOTES

NOTES

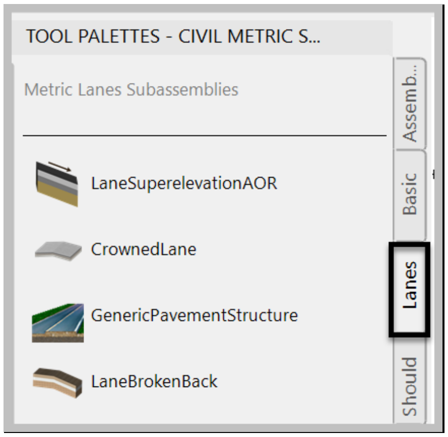
9. Then, you need to organize your work environment in such a way that you can easily access your different windows and palettes. For example, place the **Properties Window** on the right, and the **Tool Palettes** to the left. An ideal Civil 3D setup would be to have at least two monitors and distribute the windows for easy access.



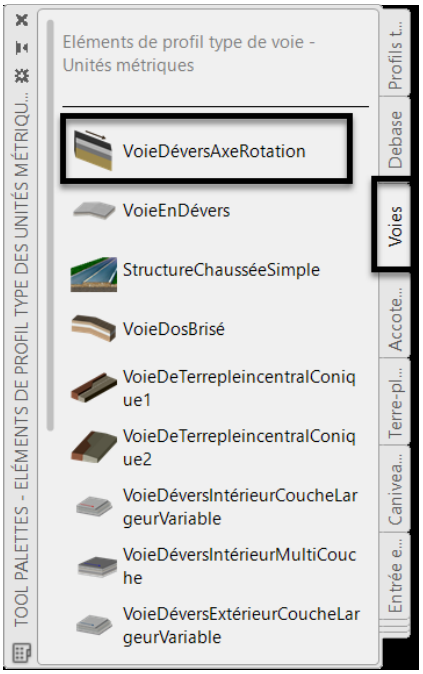

10. Once the **Tool palettes** are displayed, you can start creating the subassemblies. Just remember that we are creating an assembly per our design requirements.



11. Let's start from the centerline and work our way out to the edges. On the **Tool Palettes,** click the **Lanes** tab.



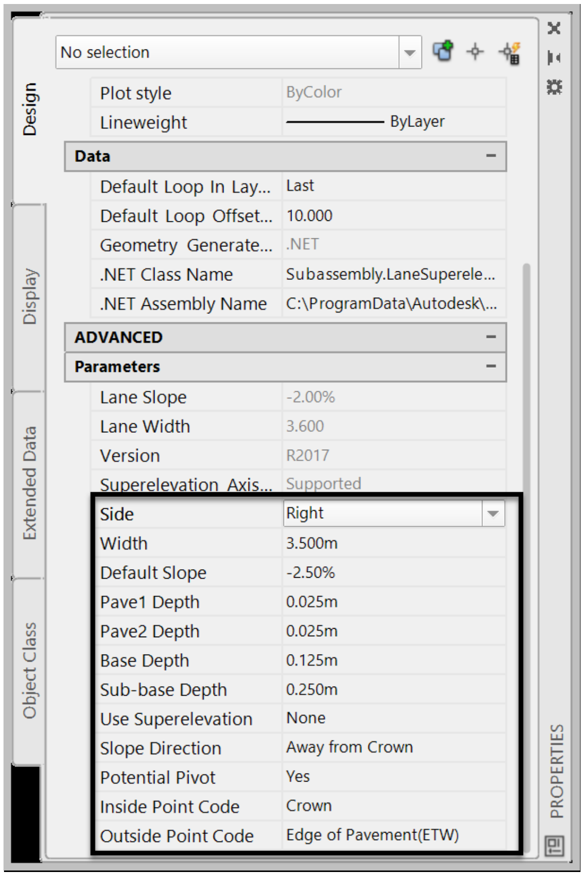

12. Click on **LaneSuperelevationAOR**. This is a general and versatile lane subassembly because it can be applied for Outside or Inside Lane superelevation slopes. It can also be used for divided, undivided, crowned and broken-back roads. Furthermore, it offers up to four layers for the pavement structure. Most of the time, this would be the only **Lane subassembly** you will need for local streets.




13. Now, we are going to apply our specifications to this subassembly. Once you clicked on it, you will see that the properties window already took notice and displayed all the parameters for this subassembly. In the **Properties Window**, scroll down to the **Parameters** section and enter the following values:

NOTES

- **3.5 meters** or **11.5ft** for **Width;**
- **-2.5 %** for **Default Slope;**
- **0.050m** or **0.15ft** for **Pave1 Depth.** This will be the first lift of asphalt;
- **0.050m** or **0.15ft** for **Pave2 Depth.** That is for the second lift of asphalt;
- **0.125m** or **0.40ft** for **Base Depth**, for your granular base material;
- And finally, **0.250m** or **0.80ft** for **Sub-base Depth**, the road sub-base material;
- Before moving to the next step, notice the line that says **Side**. You can change it to **the left** or **right**. That means that any subassembly that we select will be inserted on the side that we chose. So, we need to be mindful of that.

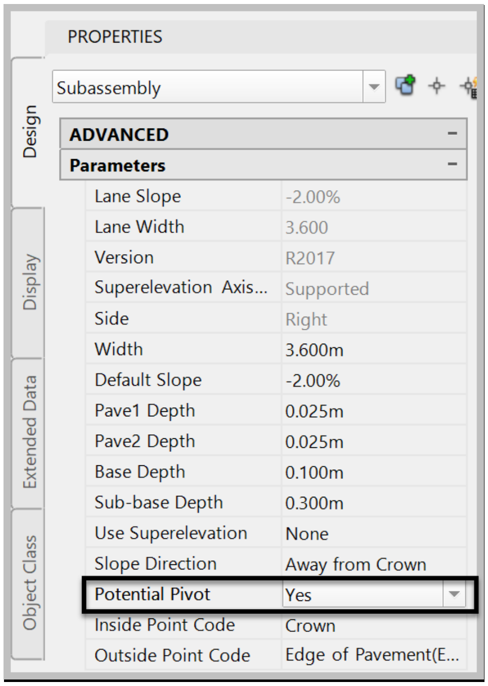



14. Click on the circle of the cross-section.

15. At this point, our right lane has been placed.

NOTES

16. Since we are designing a subdivision street without any need to create a superelevation, we are going to turn off the options to pivot the lane. Select the **LaneSuperelevationAOR** subassembly. Then in the **Properties Window**, in the **Parameters** section, find the **Potential Pivot** line and change it to **No.**



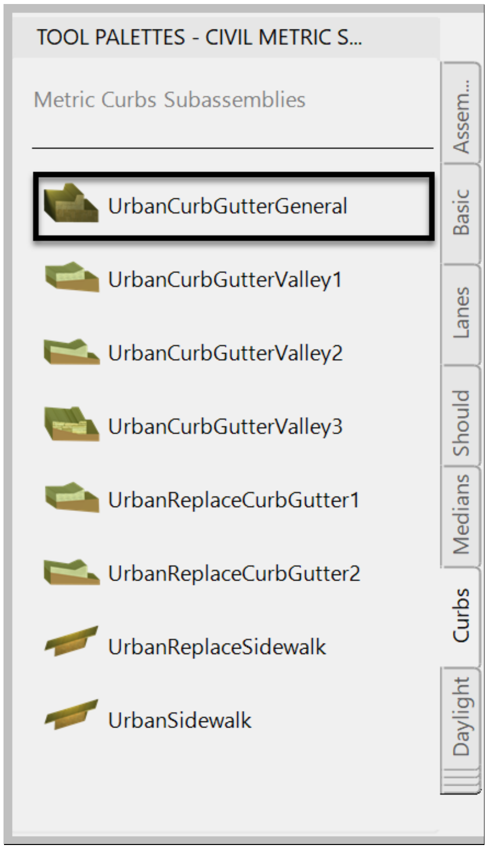

17. Next, we need to create the straight face curb abutting the lane. Our curb will have a **0.15m** or **6in** height of curb and a **0.20m** or **0.5ft** wide back of curb. To create the curb,

- We can either use the **BasicCurb** on the **Basic** tab of the **Tool Palettes**, which works in this case since our curb is a basic one. Or,
- You can switch to the **Curbs** tab of the **Tool Palettes** and choose a more advanced one, depending on your situation. To demonstrate how to customize the curbs, let's use one of the advanced ones.

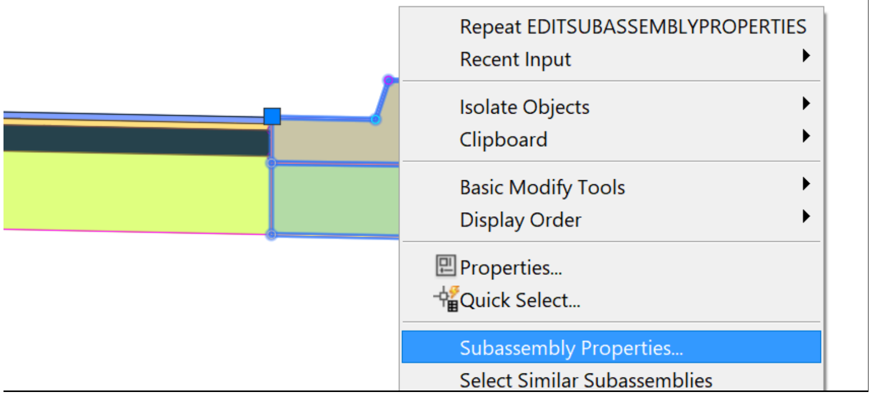
NOTES

NOTES

- On the **Curbs** tab, click on **UrbanCurbGutterGeneral**.

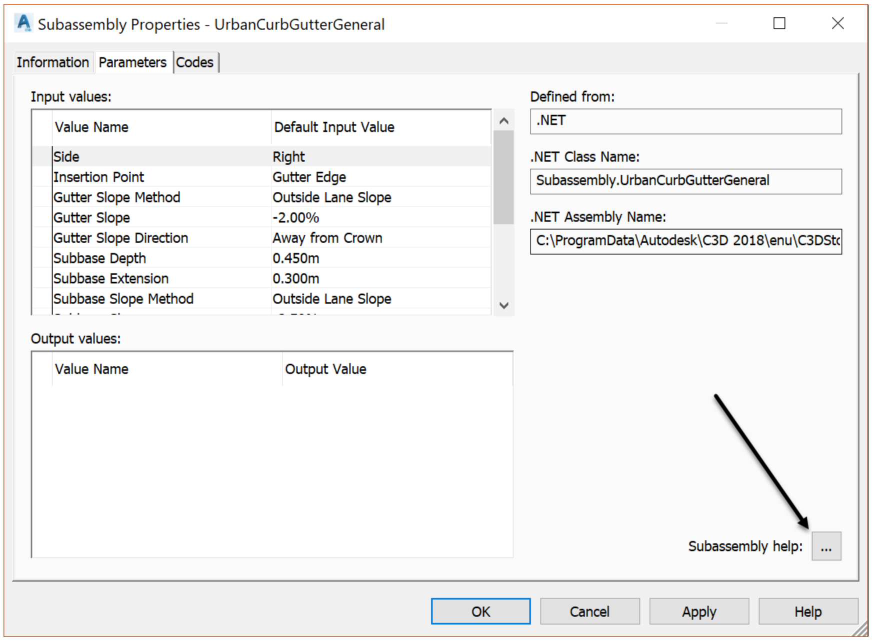



18. Next, we need to specify the parameters of the curb and its attachment point. Before entering the parameters, we first need to understand what the different values mean. For that, we can call the **Subassembly help** for assistance. Select the curb subassembly, right-click and select **Subassembly Properties**.

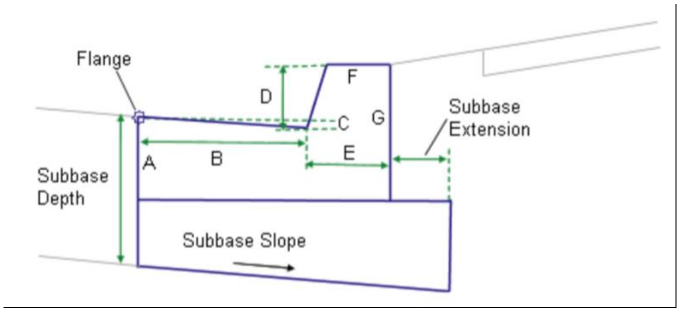


NOTES

19. In the next window, activate the **Parameters** tab and click on **Subassembly Help**.



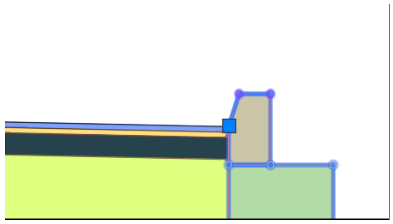

20. The figure displayed explains the required values in the parameters section of the property window.




21. Now, keep the help window opened, maybe on a second screen if you have one, and click **OK** on the **subassembly Properties** window to return to the drawing. In the Properties window, while you have the curb selected, adjust the following parameters.

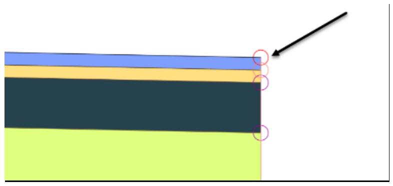
   - **Dimension B** (in millimeters or inches) which represents the **Width from the flange point to the gutter flowline.** This is typically for a road with a curb and gutter. Since we don't have a gutter, you are probably thinking "we should just put zero." Unfortunately, Civil 3D will not accept that. So, we need to trick it and put a microscopic value, say 0.001. This will not make any difference in real life. Besides, this is to show how to use the subassemblies and adjust them to your needs. Now, the subassembly appears without a gutter.

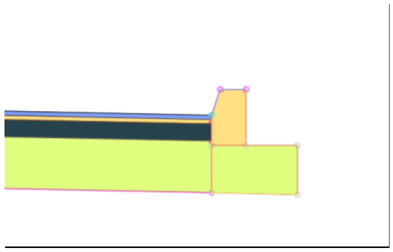
NOTES



22. Next, we need to make sure that **Dimension D** (**0.15m** or **6in**) and **dimension E** (**0.20m** or **8in)** meet our requirements, which they do by default. We also have a **Sub-base** meeting our requirements. Notice that we have changed the slope of the face of the curb. If it is absolutely required to have it at a 90 degrees angle, we can simply use the basic curb. There are also Civil 3D tools such as **Subassembly from Polyline** or the **Subassembly Composer** we can use to meet unique design conditions. We refer you to Infratech's **Advanced Road Design** course for more on these tools. For now, the only thing left is to change the **Sub-base slope** of the curb to **2.5%,** and we are all set.

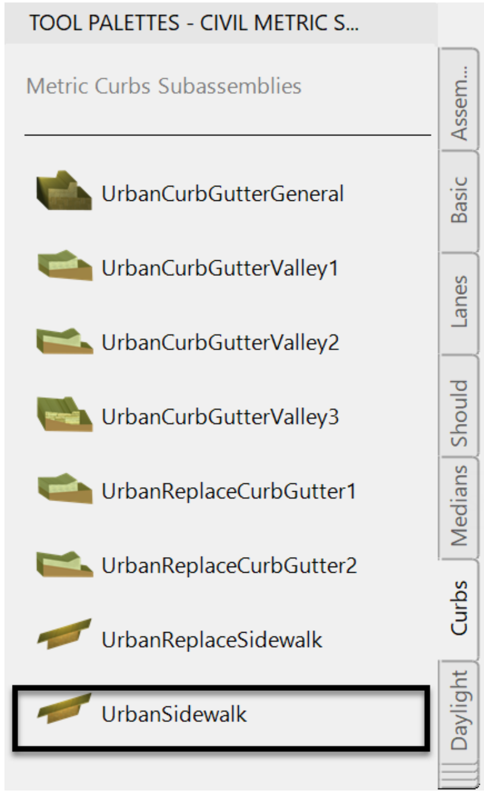
23. Next, we need to specify the attachment point of the curb. For that, click on the upper circle of the right edge of the lane. Note that we can also first specify the attachment point of the subassembly and then adjust the parameters as needed.

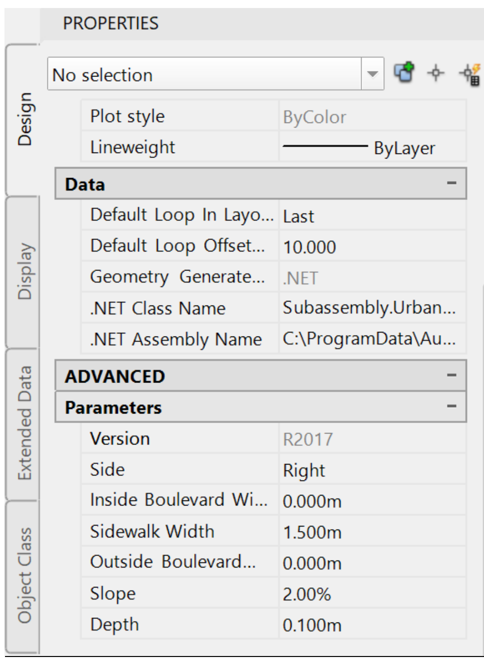




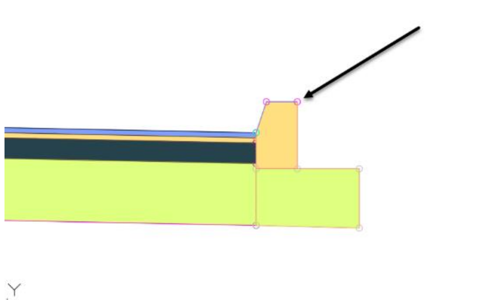
24. After the curb, we now need to specify the parameters of the sidewalk. Here, we also have two options. We can either use the **basicsidewalk** on the Basic tab of the Tool Palettes or use more advanced ones like the **UrbanSidewalk** from the **curbs** tab. We are required to have a **2.5%** slope for pavement, base and subbase, which the basic sidewalk cannot do. So, we are going to click on **UrbanSidewalk** on the Curbs tab.

    - Then, adjust the **Sidewalk Width** parameter to **1.5m** or **5ft**,
    - the **inside** and **outside boulevards** to **0,** and
    - the Slope to **2%**, and
    - leave the other parameters to their default values.



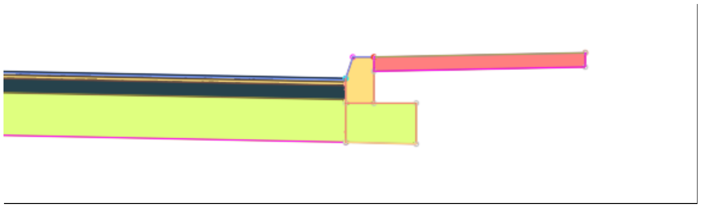




25. Next, click on the back of the curb to attach the sidewalk.

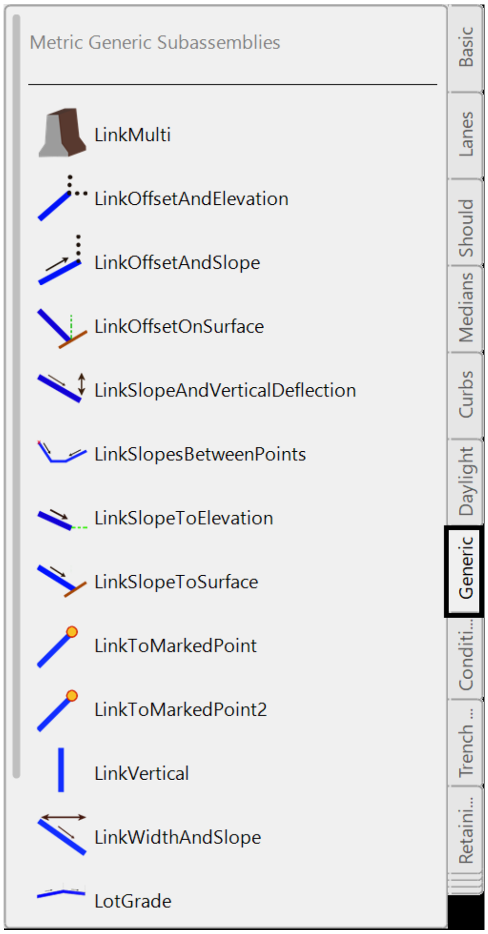


NOTES

26. The sidewalk is then attached to the curb.



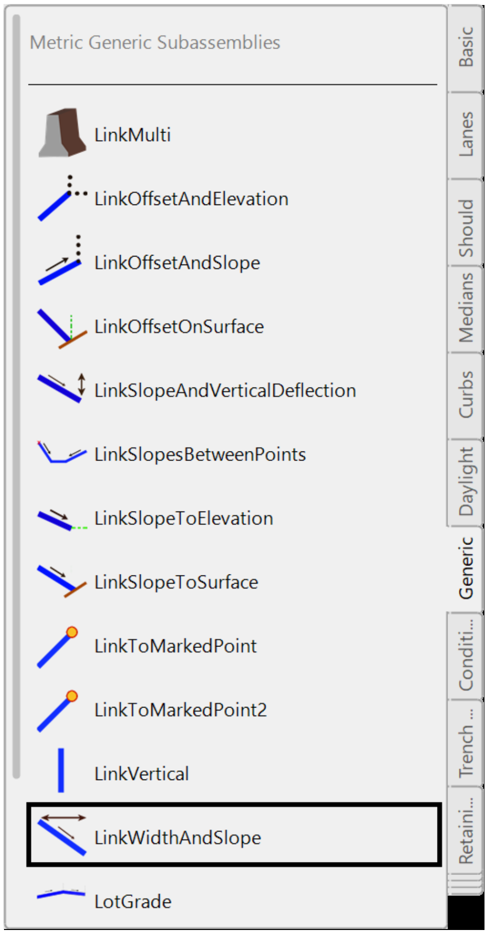
27. We have now designed most of the right side of the road. We can leave it as-is and grade inside the lots later. To grade roads and street on the edges, Civil 3D offers a set of very versatile tools: Let us introduce the **Generic** subassemblies! These subassemblies offer different options to create links. This is helpful when we need to connect to a point that is at a specific distance, elevation or even on a given surface. A connection can also be made at a specified slope or offset. It's easy to see situations where these subassemblies will be useful. That could be when we are daylighting to an existing ground surface or simply grading the front yard of a parcel at a given slope.



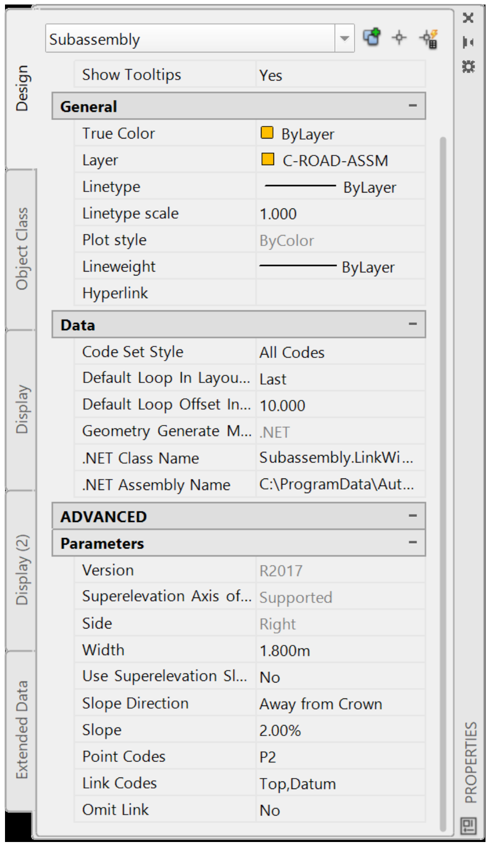

NOTES

NOTES

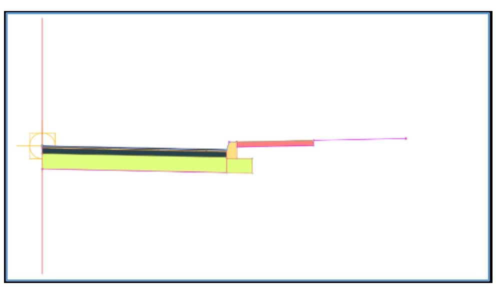
28. Next, we need to create the **1.8m** or **6ft** landscaping strip from the end of the sidewalk to the property line. For that, we will use the **LinkWidthAndSlope** generic subassembly. It allows us to go from the attachment point to a specified distance, at a specific grade.




29. In the property window, we need to enter the **2%** slope and the **1.8m** or **6ft** landscaping strip.



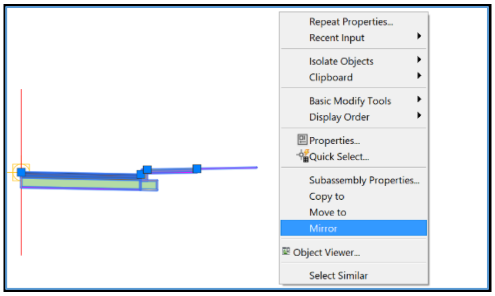

30. After clicking on the top point of the sidewalk, we would have completed the right side of the road cross-section.



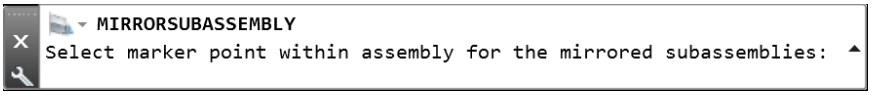
31. We can repeat everything we just did on the right, for the left side of the cross-section. But it would be much easier if we can simply copy what has already been done. Well, we have a command that can do just that. The **mirror** command can create a mirror image from the road center for us. Note: this is not your basic AutoCAD mirror command. To access it,

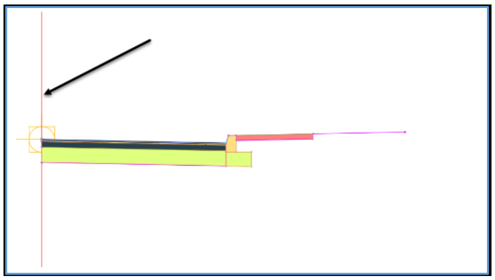
    - We need to select all the subassemblies to the right. Be careful to not include the assembly baseline (vertical line) at the center, in your selection.
    - Once all assemblies to the right are selected, right-click and select **mirror**.

NOTES

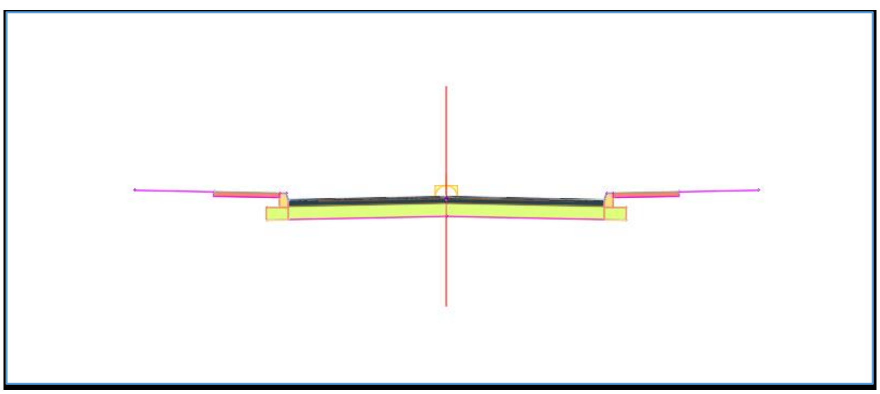



32. Then, when asked to select a marker to mirror from, click on the vertical center line.



33. We have now completed the full road section. We could have graded a little further inside the property. We could use another generic subassembly, maybe the **LinkSlopeToSurface** one, to tie the road to the existing ground at a gentle slope. This would especially be the case if we were only building the road and develop the lots later. But, since we are also doing the lot level grading, we will leave our cross-section as-is for now and do some lot grading later.

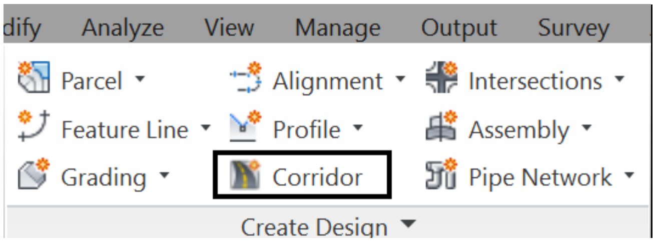


NOTES

## 9.2 Corridor Creation (proposed road)

Now that we have the three required basic elements (alignment, profile, and cross-section), we can create a road model or corridor. To do that:

1. Click the **Corridor Creation** command, on the ribbon, in the **Create Design** Panel of the **Home** tab.



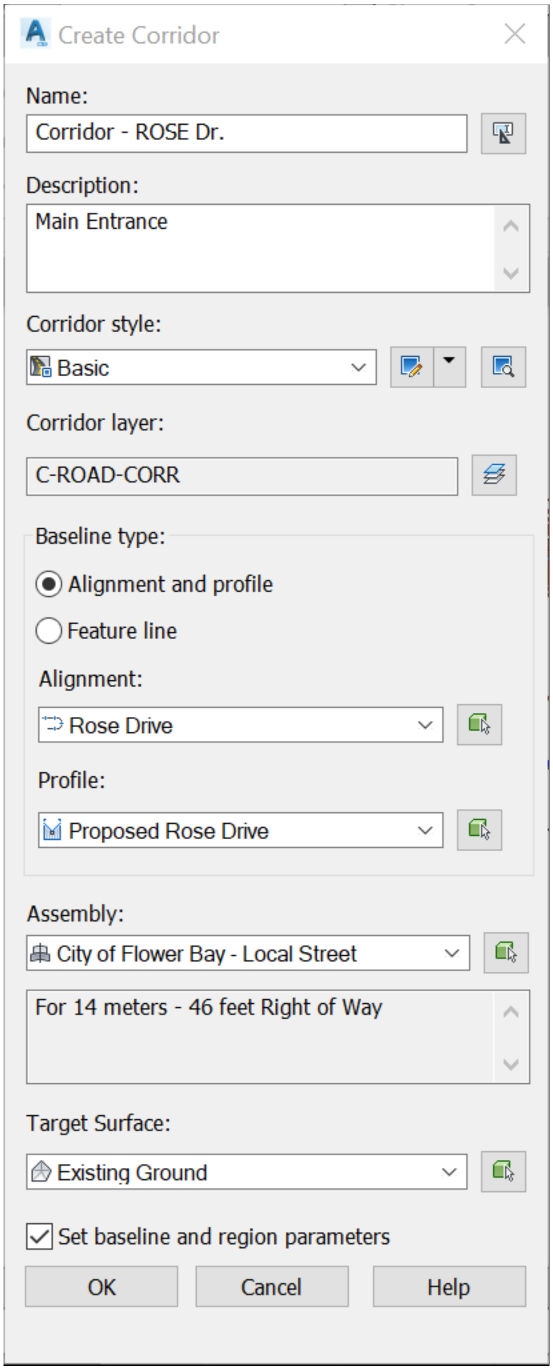

2. In the next window, define the following parameters for the creation of the corridor:

    - First, assign a **name** to the corridor. Call it **Corridor - Rose Drive**, for example;
    - Then, enter the **description**: even though not necessary, it builds a good habit to always enter descriptions;
    - For the style, let's go with the **basic style**. The style displays the appearance of the corridor components. Including boundaries and assembly insertion stations;
    - Next, leave the layer to the **default layer** assigned in the template;
    - The corridor elevations can be built either from profiles or Feature lines. When possible, always use **alignment and profile.** You can, later, use them to create road sheets and other construction plan items.
    - For the profile, choose **Proposed Rose Drive.** Don't worry if there is a number in the bracket after the name of the profile. That simply indicates the number of times the profile has been created in this file.
    - For assembly, pick the **City of Flower Bay – Local Street** typical section.
    - Next, pick **Existing Ground** as the target surface. We don't really need one in this case since we are leaving the road unfinished at the property line.
    - Finally, uncheck **set baseline and region parameters.** We will come back later to the **Baseline and Region Parameters** dialog box. It will allow us to specify

NOTES

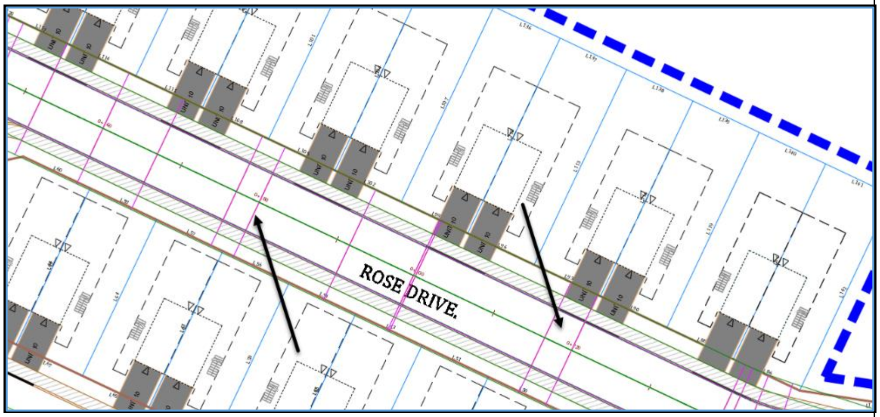
parameters such as baselines, regions, targets, frequencies, and much more. This is the main corridor setup and edit window.

- Lastly, click **OK.**

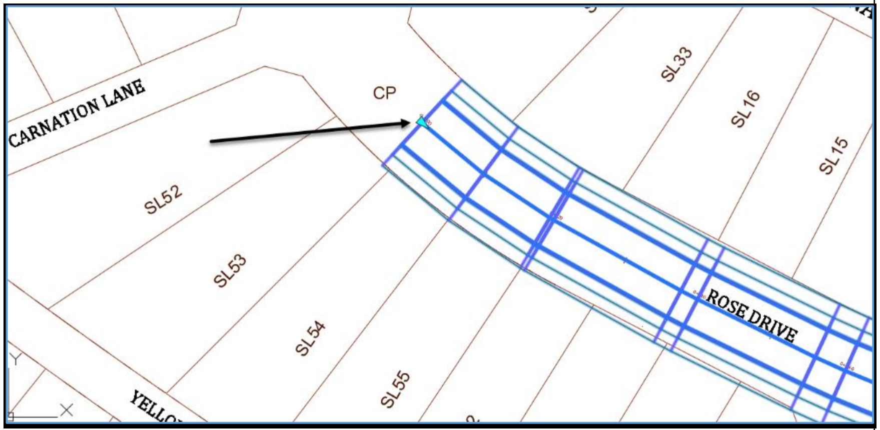



3. If you zoom to the **Rose Drive** road area, you will notice that the corridor has now been created. It extends further than needed, to the previous phase of the project to the west and to undeveloped lands to the east. This is simply because we haven't set any parameter when creating the corridor, so it is simply defaulted to the full length of the alignment.

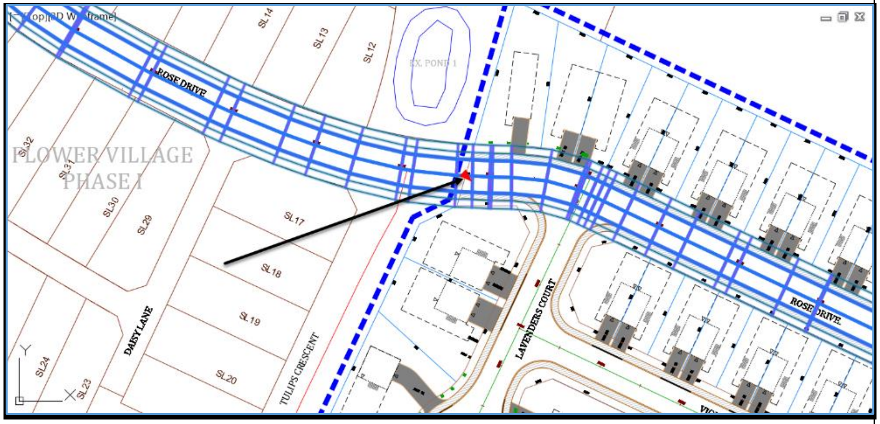
NOTES




4. Now, zoom to the west end of the corridor. When you select it, you will notice a cyan grip triangle shape at the end.



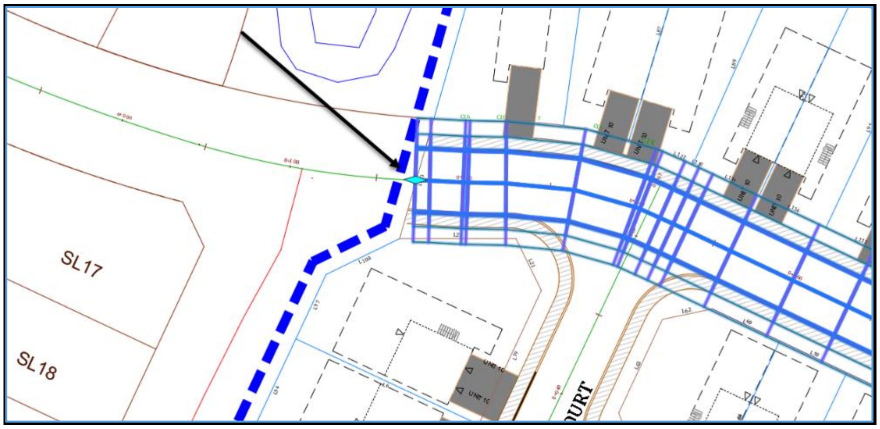

5. Then, grab it and slide slowly eastward, all the way to the start of the current phase of the project and click at the site boundary line.



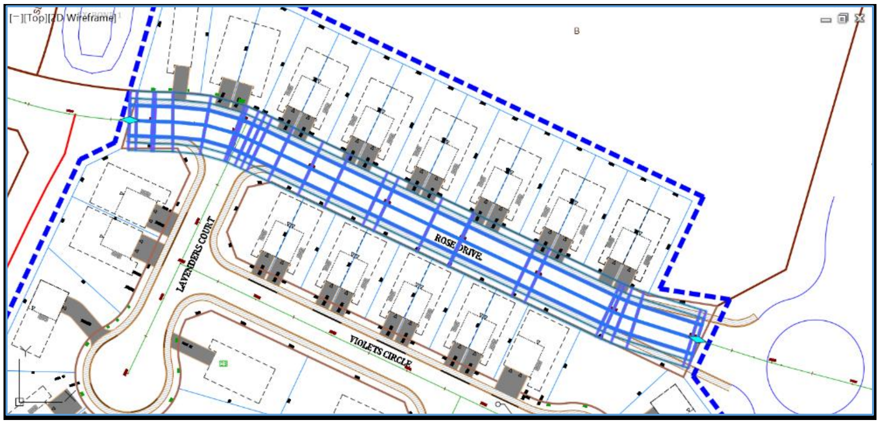

6. The west limit of the corridor is now at the site boundary. You will notice that the triangle is now a diamond shape, which means we can extend it in either direction.

NOTES

NOTES




7. Finally, repeat the same previous steps and move the east corridor limit to the east site boundary line, just before the roundabout. The corridor is now contained within the site boundaries.

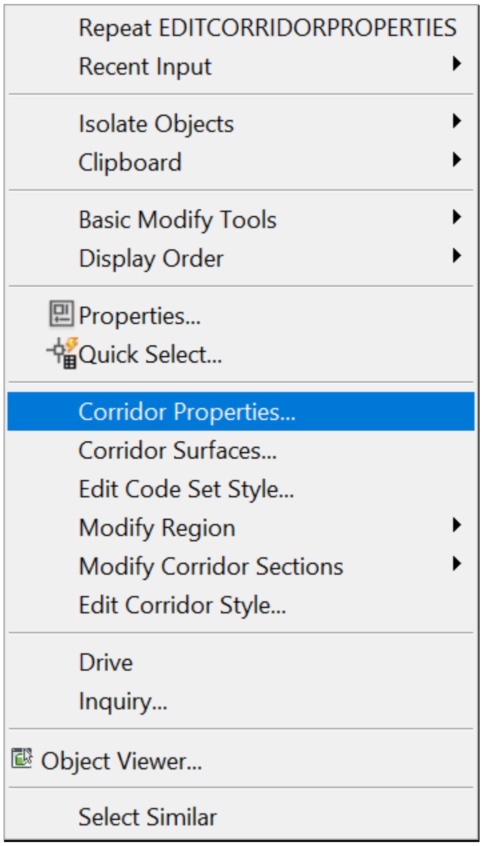


## 9.3 Modifying a corridor

1. If you are following along, keep working from the current file. If not, open the **09.02-CORRIDOR – Modifications dwg** file in the corridor working folder. This should bring you up to speed, and you can go back to practice the previous steps when you find time to do so.

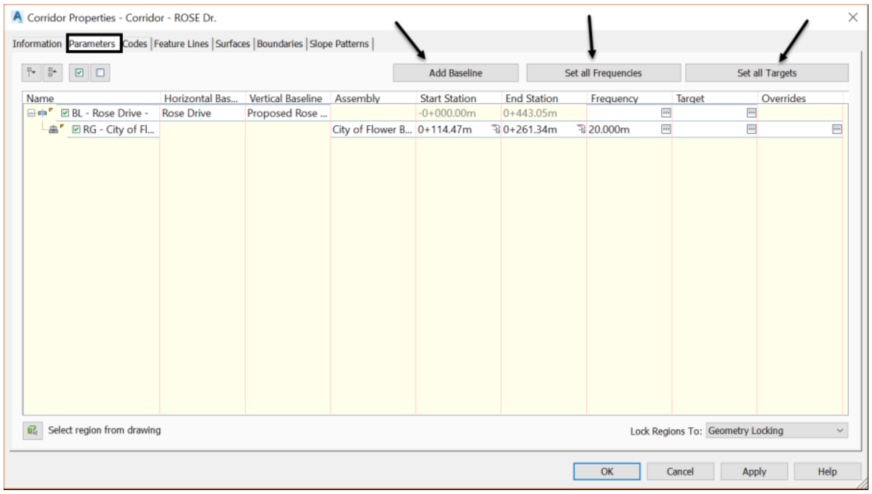
2. Once the corridor is created, it needs to be fine-tuned to meet our design requirements. For that, we can go to the corridor properties window and see potential adjustments we can make. This is the same window we would be presented with; had we checked the **Baseline and Region Parameters** checkbox in the corridor creation dialog box.

3. In the drawing or the prospector, select the newly created Corridor (**Corridor – Rose Drive**). Then, right-click and select **Corridor Properties**.




NOTES

4. Switch to the **Parameters** tab in this window for more options for modifying the corridor.




5. On this tab, we have three main types of items we can manage:

- **Baselines**: which are basically alignments we can add to the Corridor definition. This can be a bike path, a secondary road, a ditch, a stream or simply another street that is not parallel to the main road alignment.
- **Frequencies**: they specify the insertion frequency of the assembly along the corridor. The corridor is essentially constructed by inserting typical sections along the alignment. They are inserted at the elevations of the profile, at each insertion station. Generally, the smaller the frequency, the more precise the corridor is, especially in the curved regions of the corridor.
- **Targets**: enable us to extend the corridor and match specific offsets, elevations or surfaces. For instance, one can have a road that narrows or widens. In these cases, the paving of the road must be defined to target the variations of the widening or narrowing of the pavement. This may be for a bus stop or a roadway tapering.

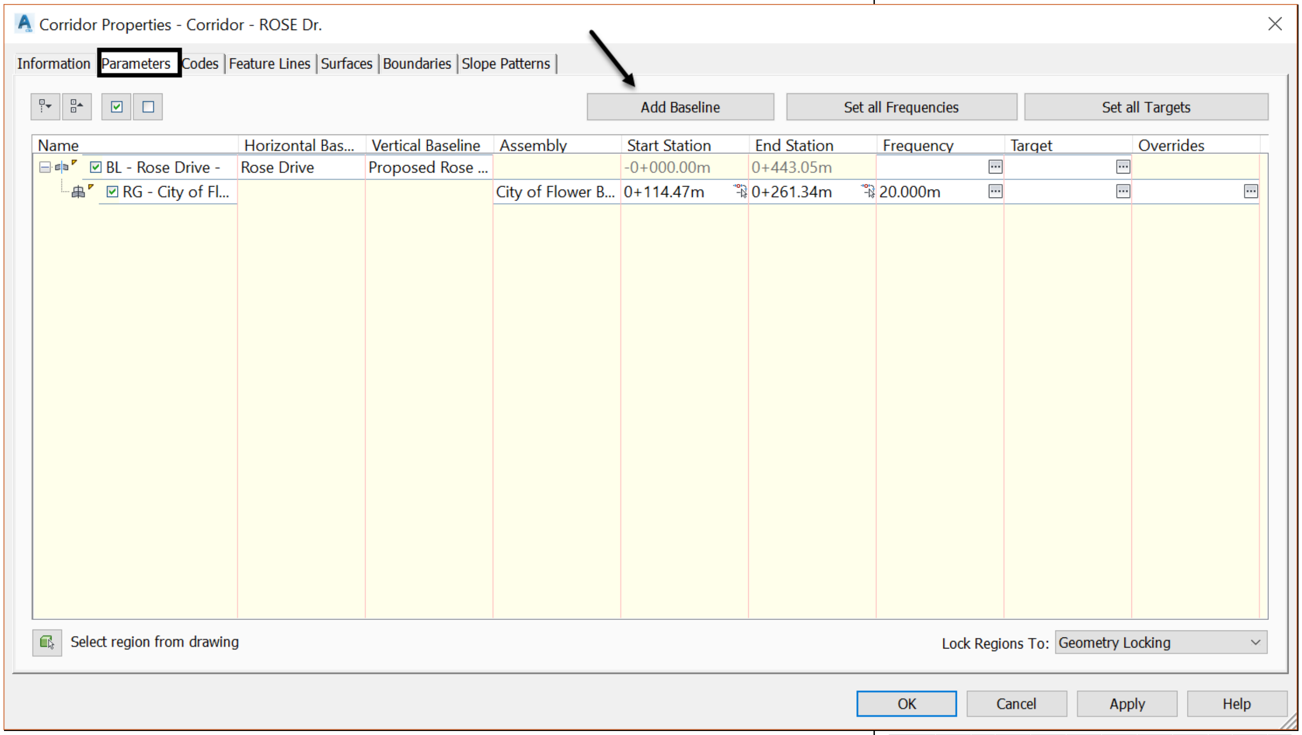
6. Now, let's see these options in practice.

NOTES

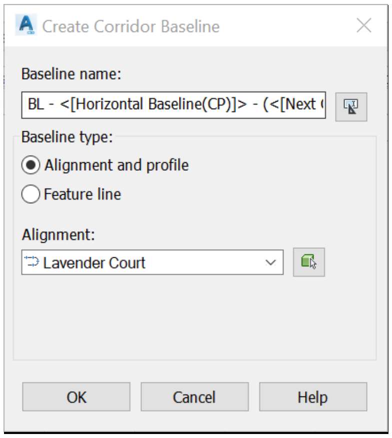
NOTES

### 9.3.1 Baselines

1. First, click on **Add Baseline** on the parameters tab.

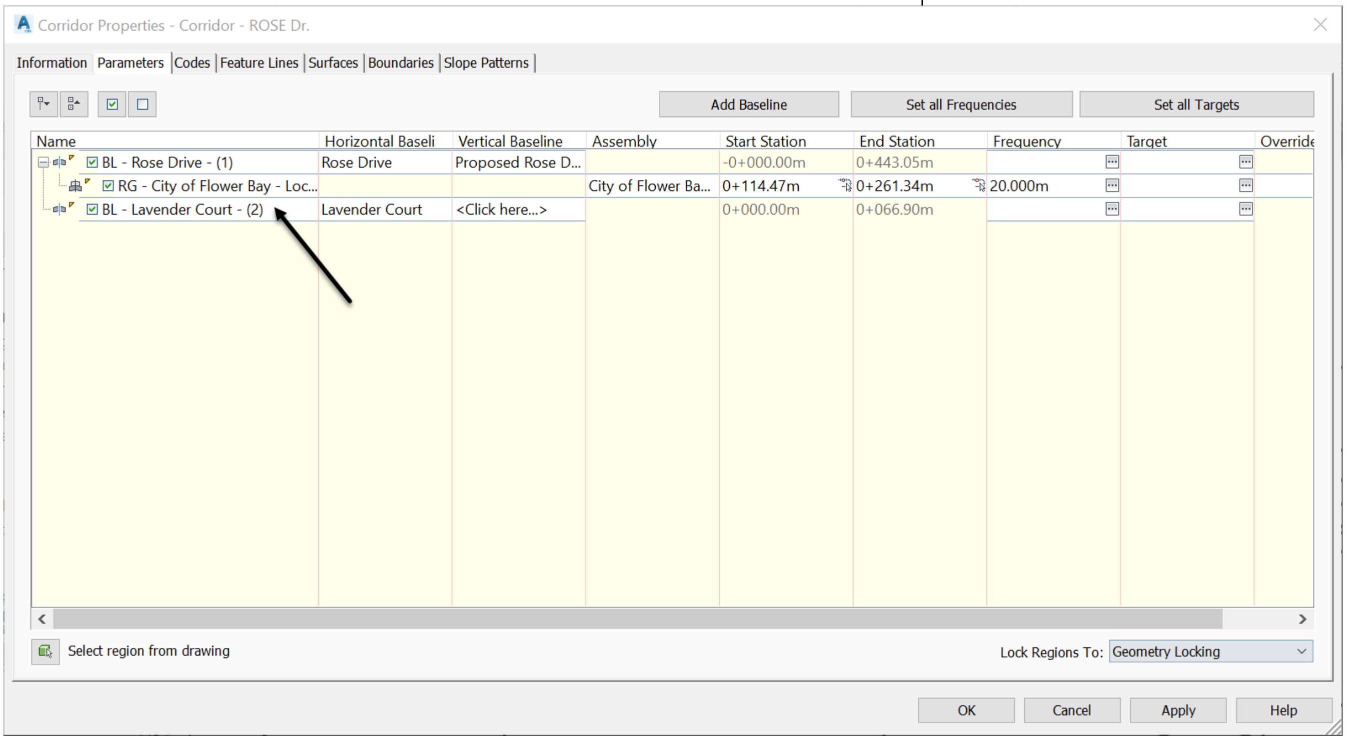



2. We are in a residential neighborhood with straight roads and associated infrastructures. For that reason, the only type of baselines we have are streets. So, select the baseline type of **Alignment and Profile** and pick **Lavender Court** for alignment.

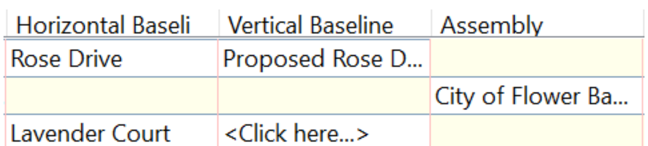



3. Click **Ok,** and you will notice that a new baseline is added to the corridor.

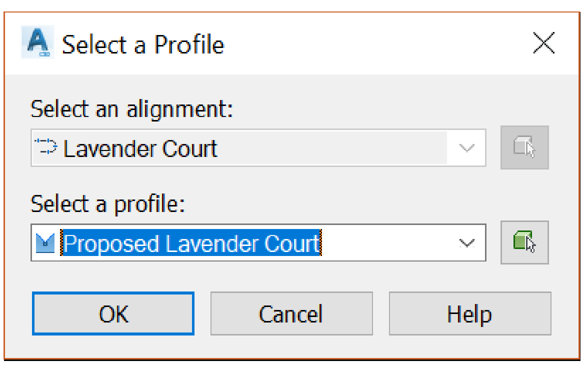
NOTES



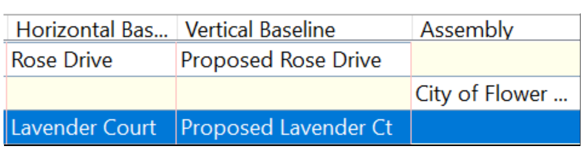

4. Now we need to specify the profile associated with the baseline. Click on **<Click Here>** in the vertical baseline column.



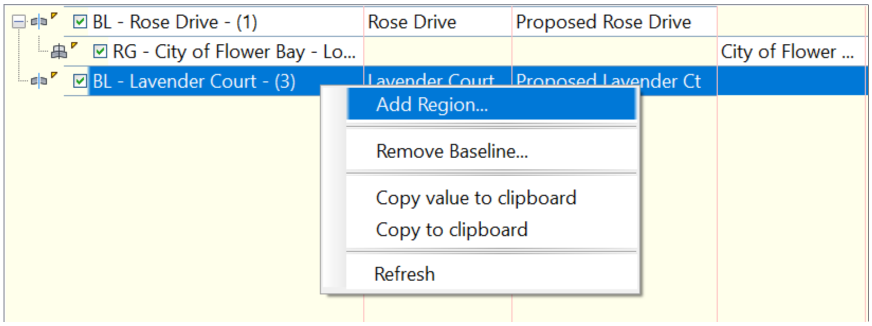

5. In the next dialog box, select the **Proposed Lavender Court** profile and click **OK.**



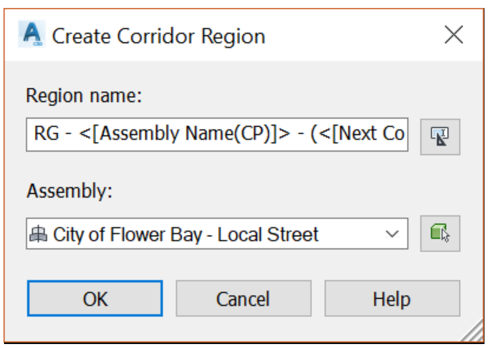

6. The new **Vertical baseline** is added to the corridor.




7. Next, we need to specify **regions**, meaning the segment of the road for which we are creating the corridor baseline. To specify the baseline regions, right click on the newly created baseline and select **Add Region**.




8. Next, we need to decide what cross-section to apply in that region of the corridor. One road can have many cross-sections. For instance, think of a project with local roads, major and minor collectors. The **Create Corridor Region** dialog box would be where we decide which cross-section to use for which part of the corridor. In the current window, select the **Local Street** assembly. And, frankly, that's the only one we have created for this project so far. Additionally, it would be nice to name each region and baseline manually, instead of the automated names. But, those names will be good enough since corridors have features to track down regions and baselines.




9. Click on **OK** to apply the assembly to the region;

10. Make sure you click the "**+**" sign to the left of the baseline name to expand the list of regions and their information. The **Start Station** and **End Station** columns will apply the region range. To provide that information; you can type the station values if you know them. Or, click on the little station symbol to the right to specify the station on screen. For the start station, since we know that the alignment starts in the center of the cul-de-sac, accept or type **0+000** for the start station. For the end station, let's use the symbol option since we have no clue what the exact end station value is. So, click on the little icon, then in the drawing, click on the west end of the curb return. For precision design purposes, activate your **end** object snap option to click on the exact point of the end of the curve.

NOTES

11. We have now set the start and end of the **Lavender Court** region. Don't worry about the yellow warning to the left of the region. It is just a sign that the corridor has changed and needs to be rebuilt. If you don't have the warning sign, it means that you have **automatic rebuilt** activated on the corridor, in the prospector.

| Start Station | End Station |
|---|---|
| -0+000.00m | 0+443.05m |
| 0+114.47m | 0+261.34m |
| 0+000.00m | 0+066.90m |
| 0+000.00m | 0+049.38m |

12. Now, click on **OK** to look at the changes in the drawing.


13. As expected, we have two regions that extend within the specified stations.

NOTES

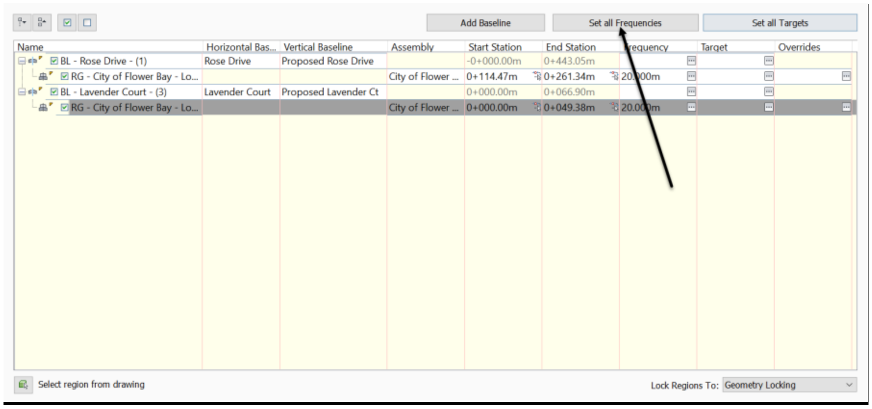
## 9.3.2 Corridor Frequencies.

After the baselines, the next main parameters we can change are the **Frequencies**. As explained before, frequencies dictate how often cross-sections are inserted. This is particularly important at curves and vertical geometry locations. To manage **frequencies**, select the corridor and return to the **Corridor Properties** window.

1. At this point, we may need to change the corridor's name from **Rose Drive** to maybe the project name, since the corridor includes other streets. On the Information tab, change the name to **Corridor – Flower Village**;

2. Then, switch to the **Parameters Tab**;

3. We can set the frequency individually for each region. For that, just click on the icon with three little dots, in the **Frequency** column, on each frequency line. Or, we can do it globally, for the entire corridor, by clicking on the **Set all Frequencies** button. Let's choose the latter option.



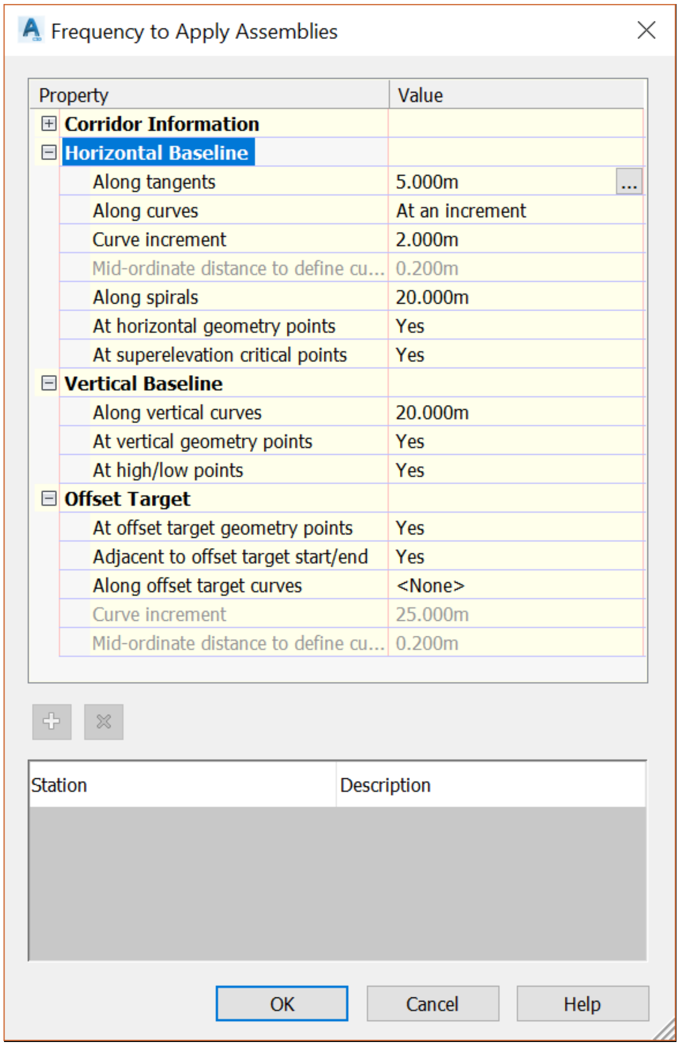

4. In the **Frequency to Apply Assembly** dialog box, apply the following values.

   - ☐ For horizontal baseline let's go **10m or 30ft** along the tangents. That would be the straight line portions of the road.
   - ☐ In the curves, choose a **2m or 5ft** increment.
   - ☐ Then, accept the default choices to insert cross-sections at horizontal geometries. They are point like curve-segment intersections, change of tangents directions,

   etc.

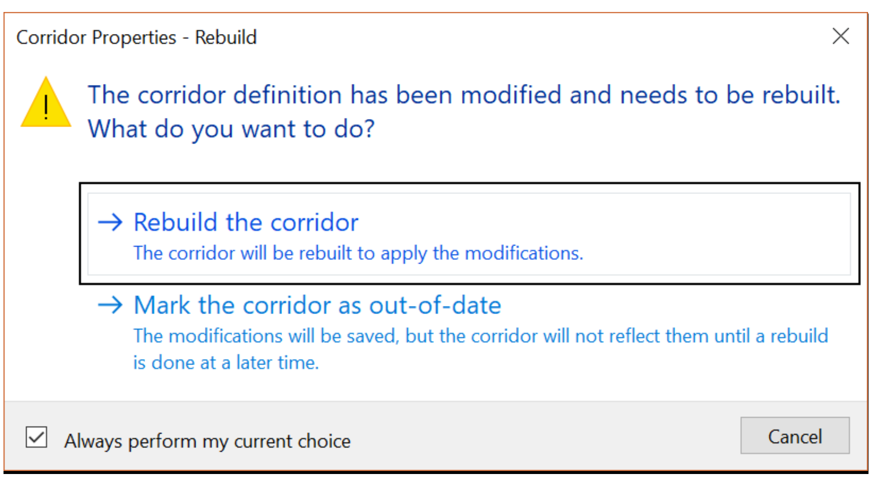
   - ☐ Also, accept default value for **vertical baseline** (profiles) and **Offset Targets**.

NOTES




5. Click **OK** in the next two windows to accept the new frequency settings.

6. When presented with the **Corridor Properties – rebuild** window, choose the option to **rebuild the corridor**. Activate the checkbox at the bottom of the window to automatically pick this option and avoid this window all together next time.



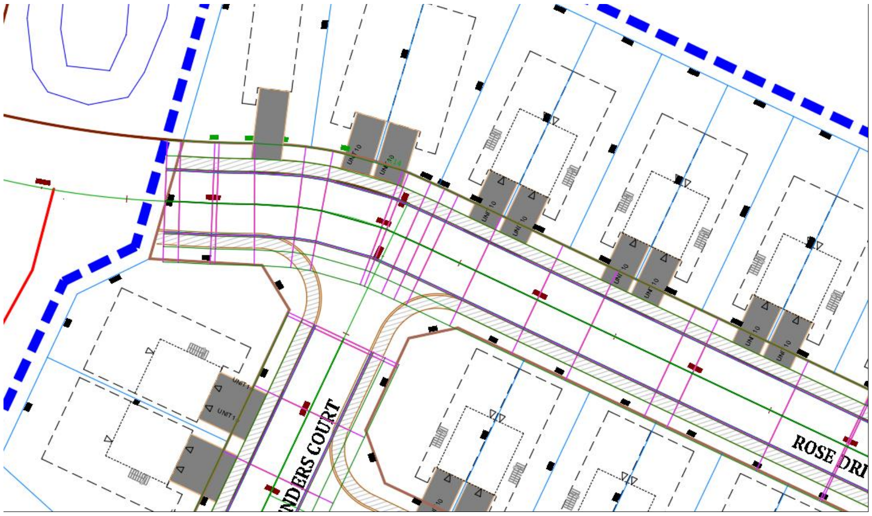

We can observe that changing the frequencies in the curves added more sections to the corridor and created better accuracy and visual appeal.

NOTES

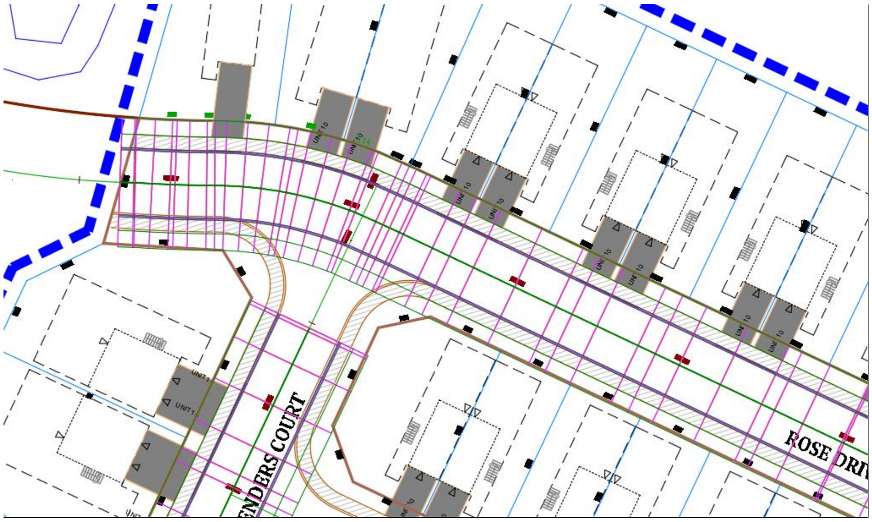
NOTES

Well, maybe that last point is currently debatable, but we will have the evidence when we create intersections and cul-de-sacs.

Before (frequency at **10meters** or **30ft**)



After (frequency at **2meters** or **5ft**)

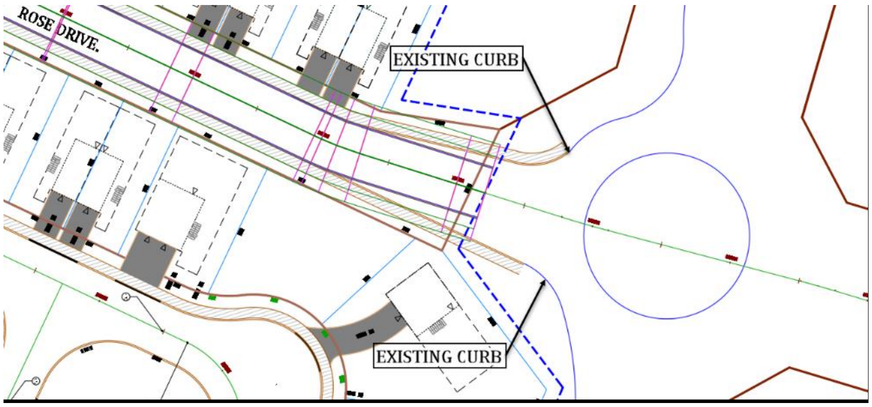


NOTES

### 9.3.3 Corridor targets

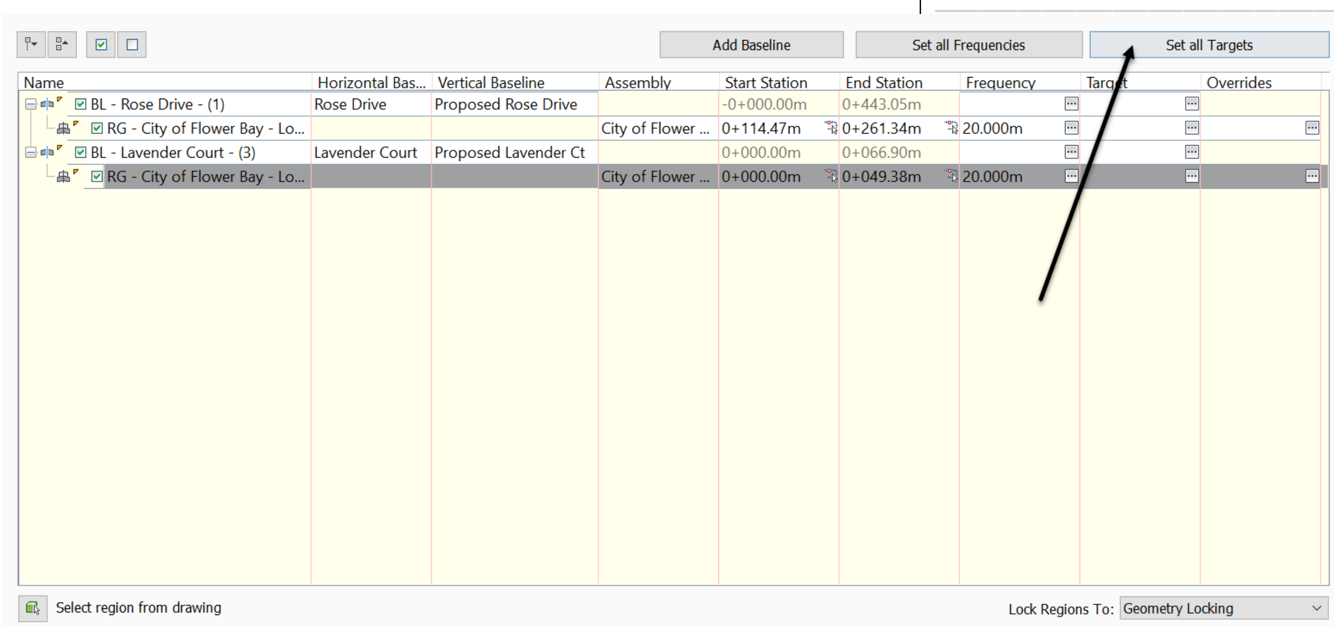
After **baselines** and **frequencies**, the third type of parameter we can manage are **targets**. As a reminder, targets enable us to connect to existing features, either by offset, elevation or projection to a surface. In this project, we have an existing curb and pavement that we need to tie to. But, the existing and proposed pavements don't necessarily have the same width.

For that reason, our street needs to widen to match the existing road.



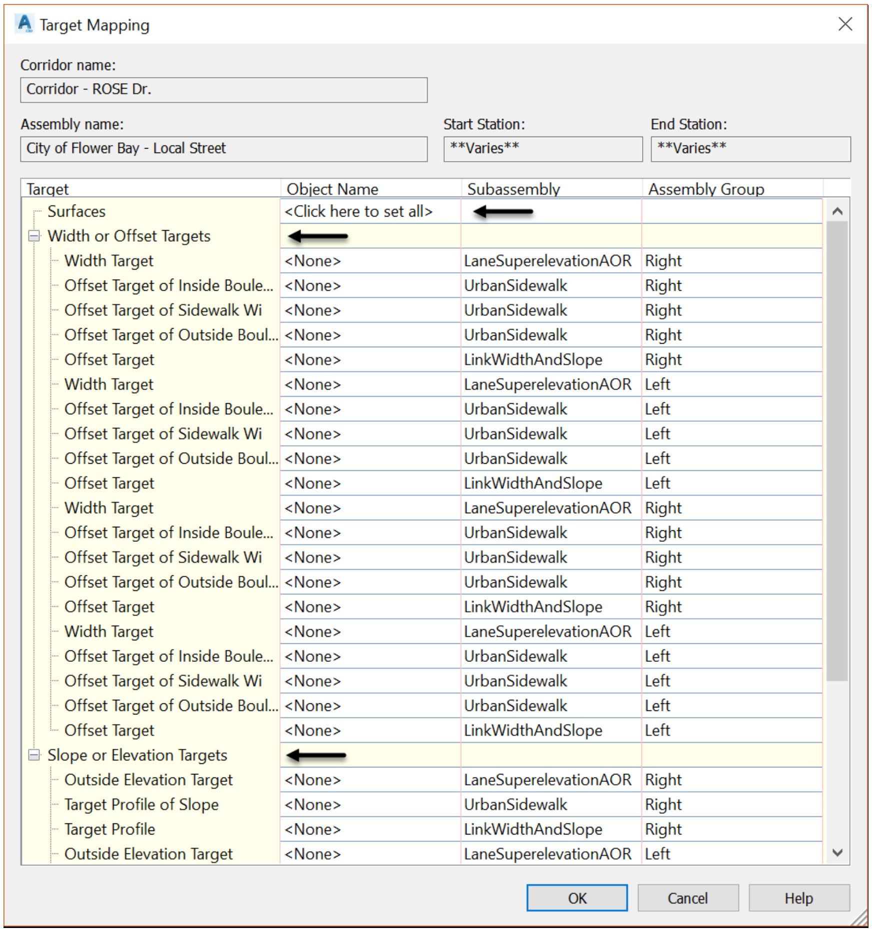

1. To widen the road to match the existing pavement, we need to apply the target parameters of the corridor. Return to the **Corridor Properties** window.

2. Click on **Set All Targets** on the **Parameters** window.

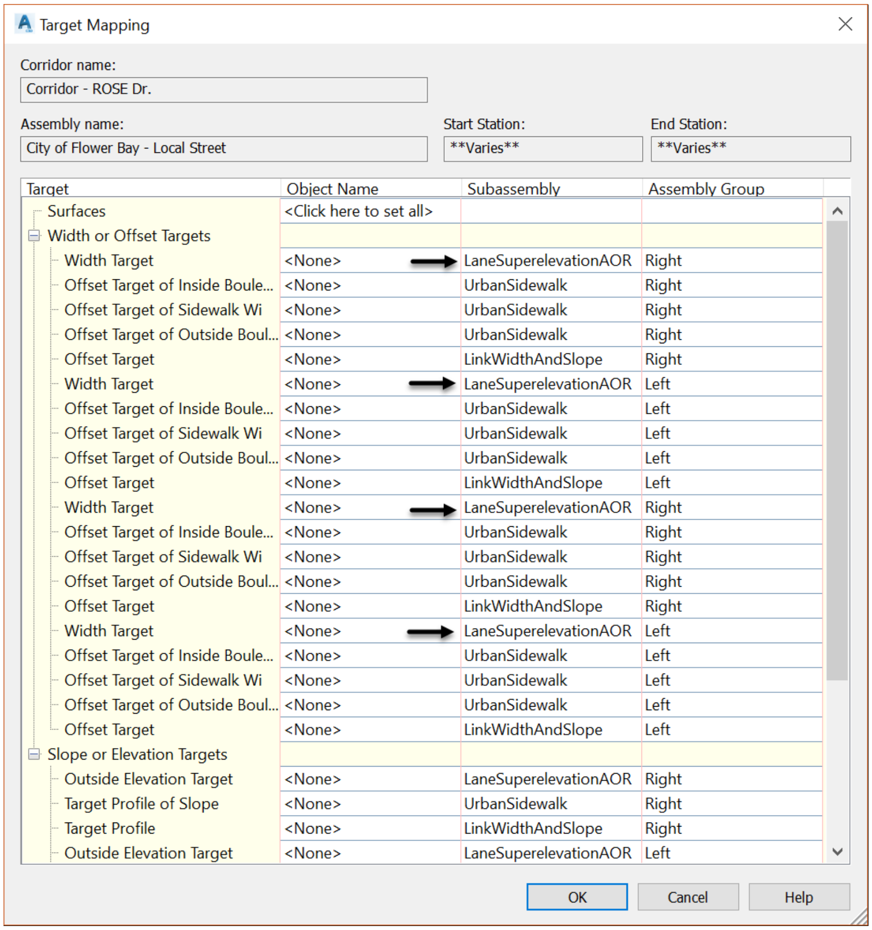


NOTES

3. On the **Target Mapping** dialog box, we can set the three types of targets we've mentioned before: **Surface**, **Width or Offset** and **Slope or Elevation**.

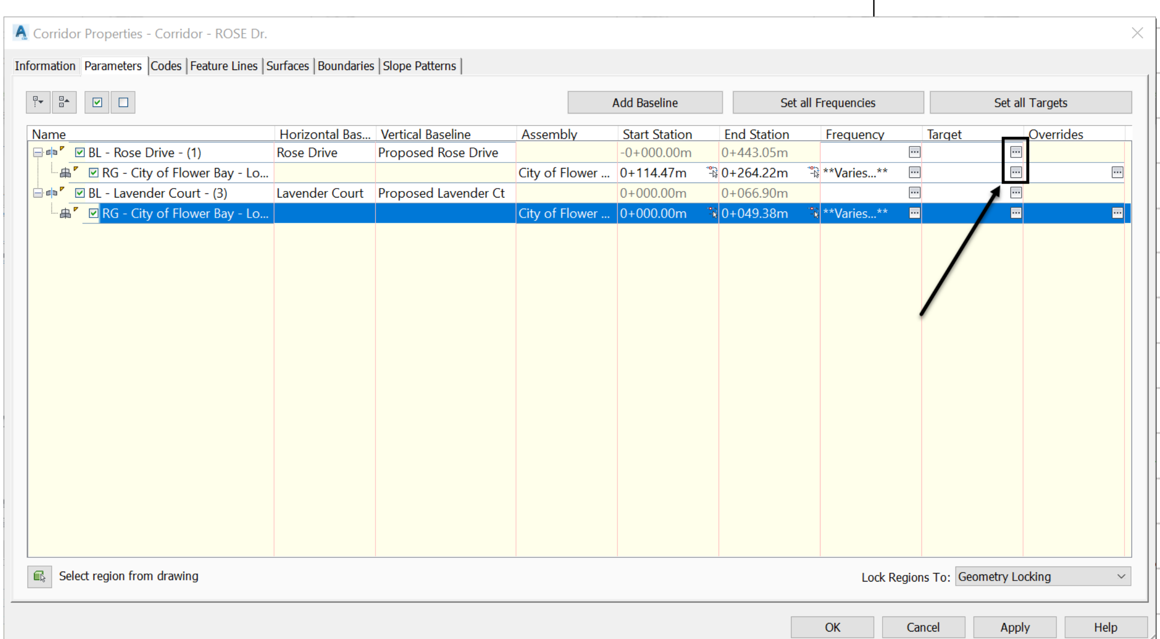



4. We have no options to choose in the surface section because we have not used any subassembly that can target a surface. We are mostly interested in the width target in this case. The item that needs to be extended is the road pavement. It is represented by the **LaneSuperelevationAOR** subassembly. Scroll through the **width or Offset target** section. You will notice that the **LaneSuperelevationAOR** subassembly is on four lines.



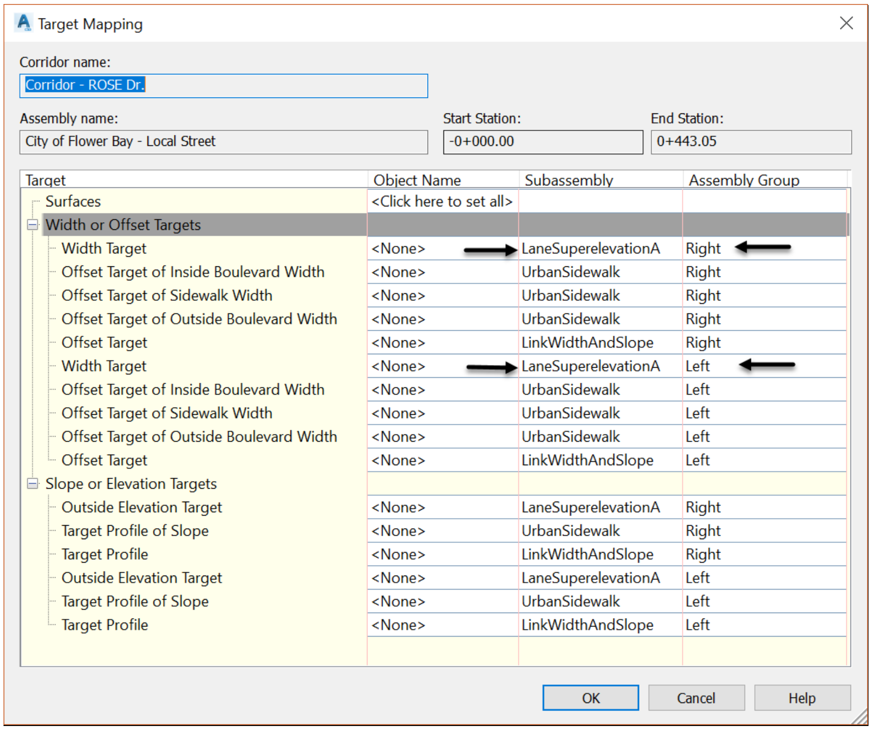

5. So, what we are going to do is back up a little bit and return to the previous window, to pick where we would like to apply the widening. Click on **OK** or **Cancel** to close this window.

6. Now, go back to the corridor **Parameters** tab, in the **Target** column. Click on the icon with the three dots on the **Rose Drive Baseline** or **RG – City of Flower Bay** region. Choosing **Baseline** will apply the target to the whole baseline while choosing a region will apply the target only to that specific region.

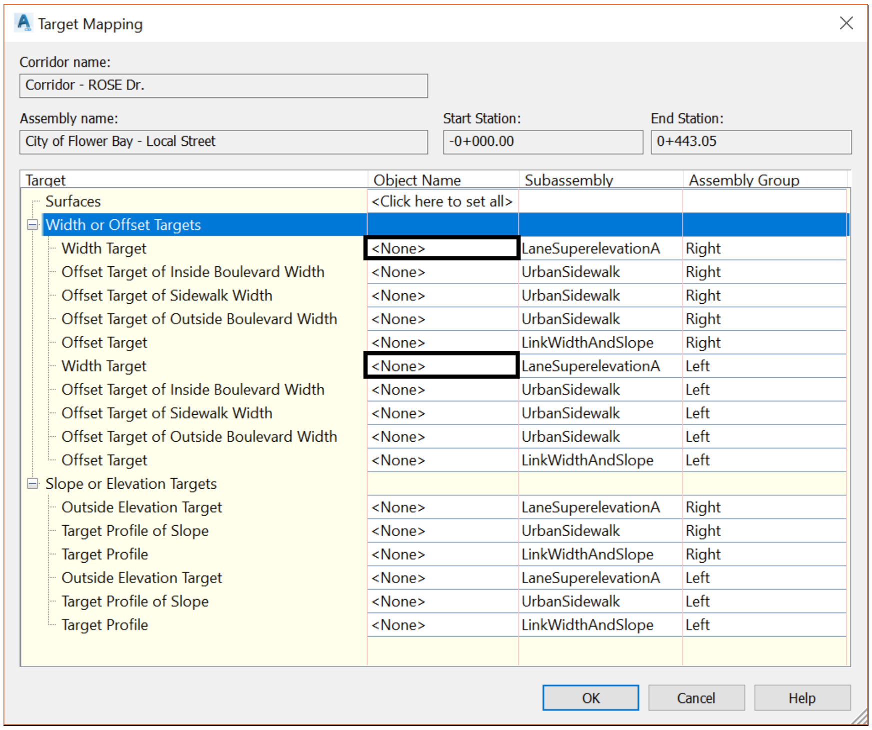



NOTES

7. Let's go with the entire baseline icon, the top one. Once you clicked on that, the same previous **Target Mapping** window opens. But, this time with much less data, because we chose to only work with the **Rose Drive** baseline. Consequently, we only have two lanes to work with. The left and right sides of the road.

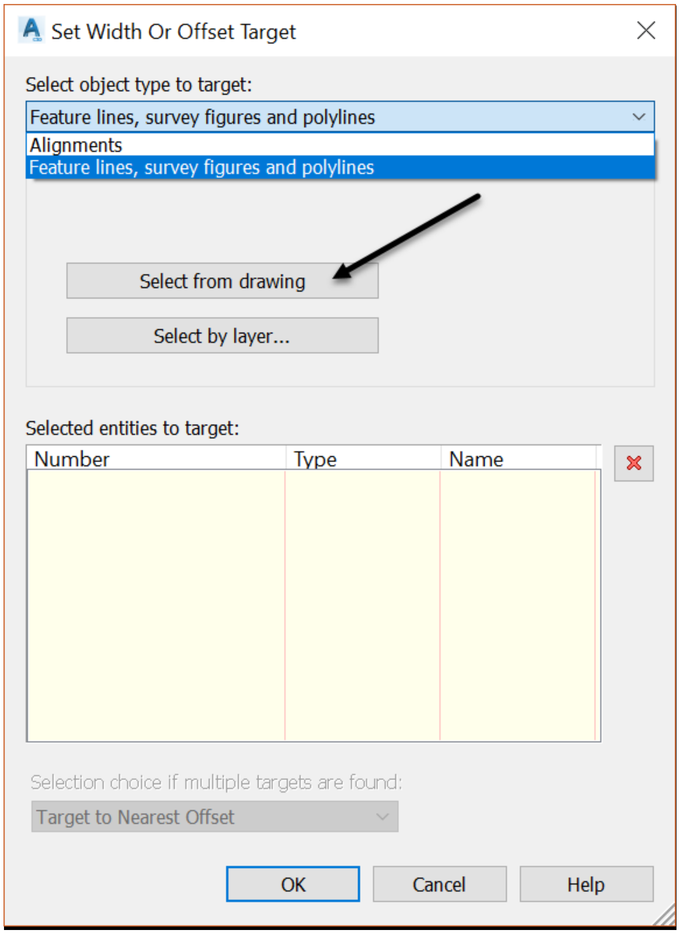



8. On each of the two **LaneSuperelevationAOR** lines, click on **<None>** in the **Object Name** column. Before clicking on each, make a note of which side, left or right, you are working on from the **Assembly Group** column.

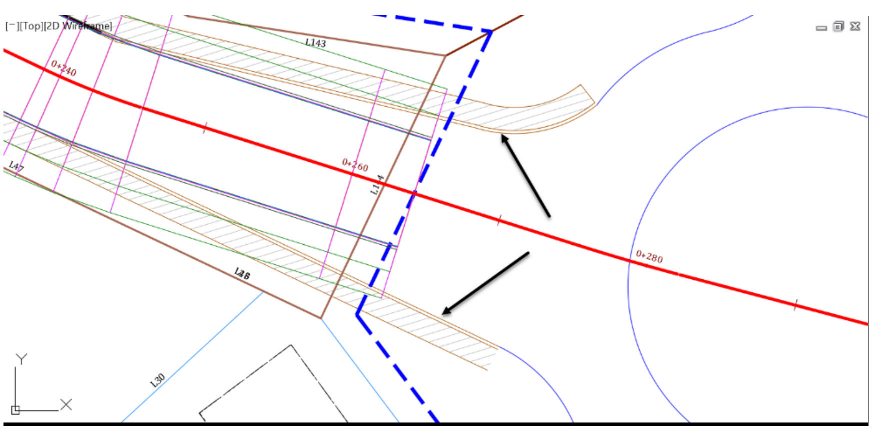



NOTES

9. In the new **Set Width or Offset Target** window, change the object type to **Feature Lines, survey figures and polylines** and click on **select from drawing**.



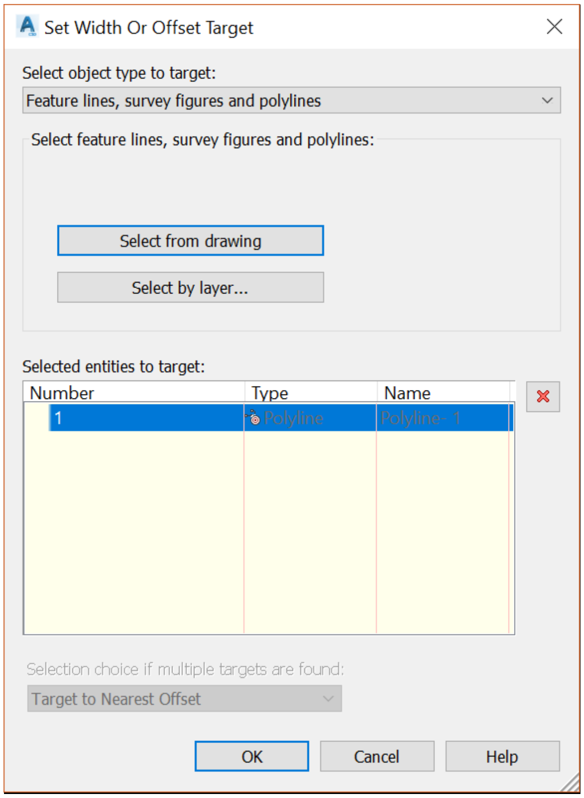

10. In the drawing, for the **left-side** lane, click on the north face of the curb polyline, and for the **right-side** lane, click on the south face of the curb polyline.



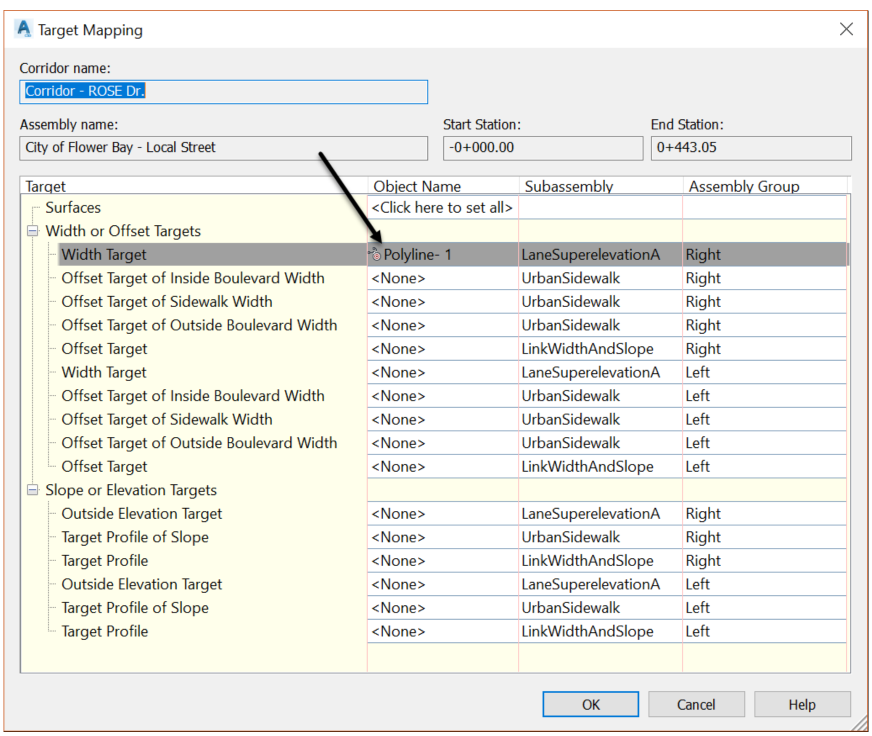
NOTES

NOTES

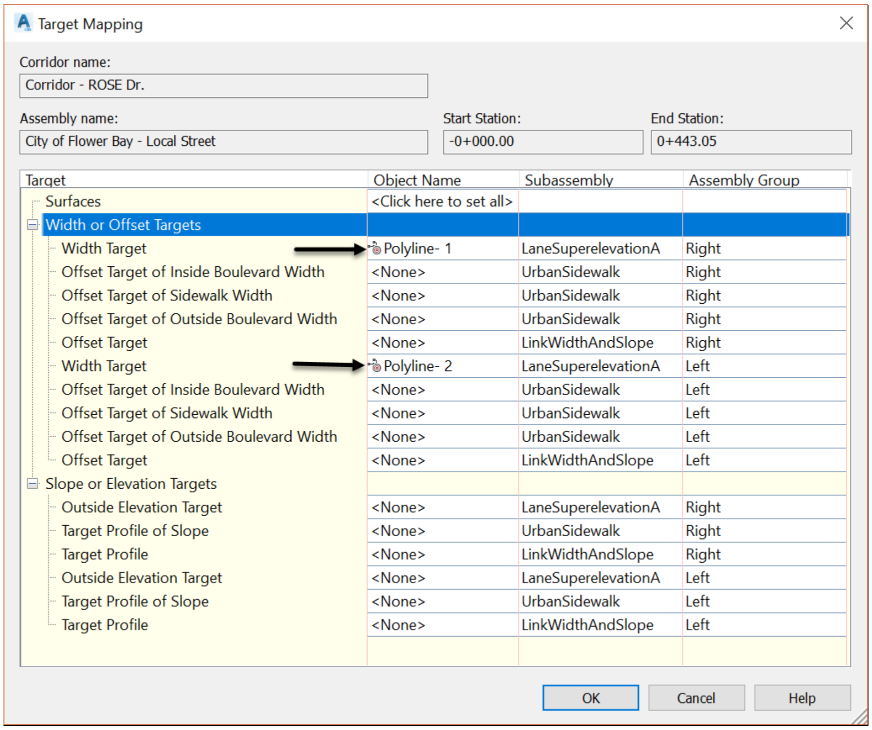
11. Once a polyline is selected, it will automatically appear in the **Selected entities to target** list.




12. Once you click **OK** again, the selected polyline will appear in the **Object Name** column.



13. Repeat the same steps for the left side of the road, by picking this time the north face of the curb polyline. Once done, you should have something like the right side, with a new name.



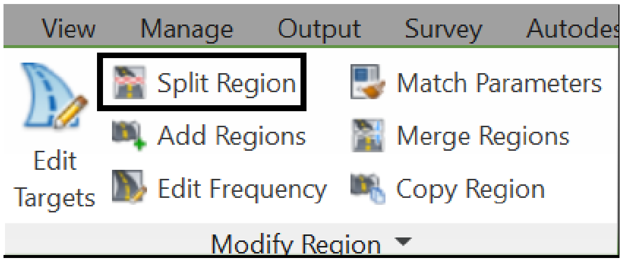

14. Click **OK** to close the **Target Mapping** window. In the drawing, notice how the road widens to match the existing pavement.

NOTES

### 9.3.4 Splitting a corridor

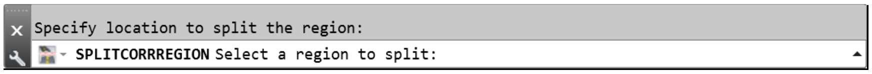
Once the corridor is created and the parameters are adjusted, we may still need to make modifications to the corridor. If you zoom to the intersection of **Rose Drive** and **Lavender Court,** you will notice that the corridor is a little messed up. First, the right-side lane is still trying to target the south face of the curb. Besides, we also need to provide a more detailed design for the intersection. So, we are going to split the corridor and make some room in the intersection area.


1. To split the corridor, select it, then from the ribbon, run the **Split Region** command.



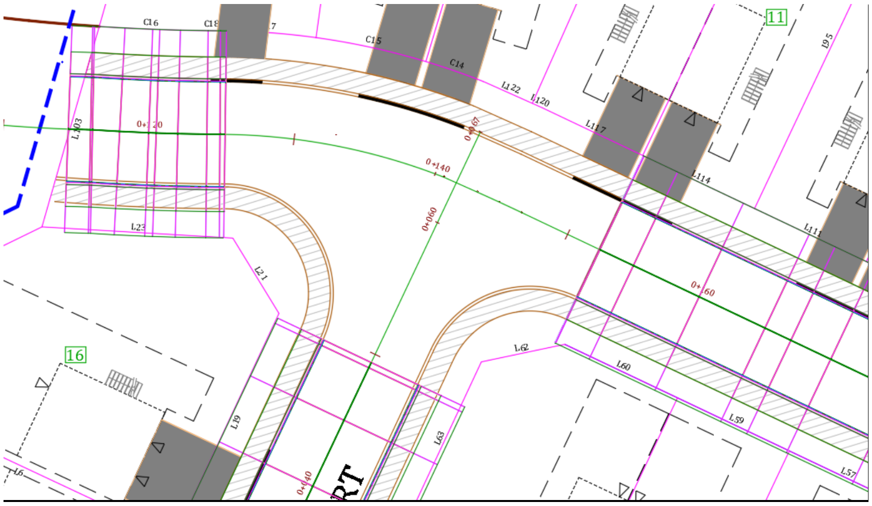

2. Now, click twice inside **Roses Drive,** in the vicinity of station **0+140** and hit **Enter**.

3. When prompted to select another region to split, press **Esc**.




4. In the corridor, you will notice three small grips, two triangles, and a diamond. Each triangle enables you to move a region in one direction, while the diamond allows you to move the split point in either direction. Use the left and right triangles to move the split regions on each side to, approximately, the start of the south curb returns. You can use the **end** object snap option to snap exactly where to the end curves. When done, we should have the intersection area free and clear of any corridor section.

NOTES



C16
C18
C15
C14
L122
L120
L117
L114
11
16
0+120
0+140
0+060
0+160
L23
L21
L19
L62
L60
L59
L57
L63
0+040

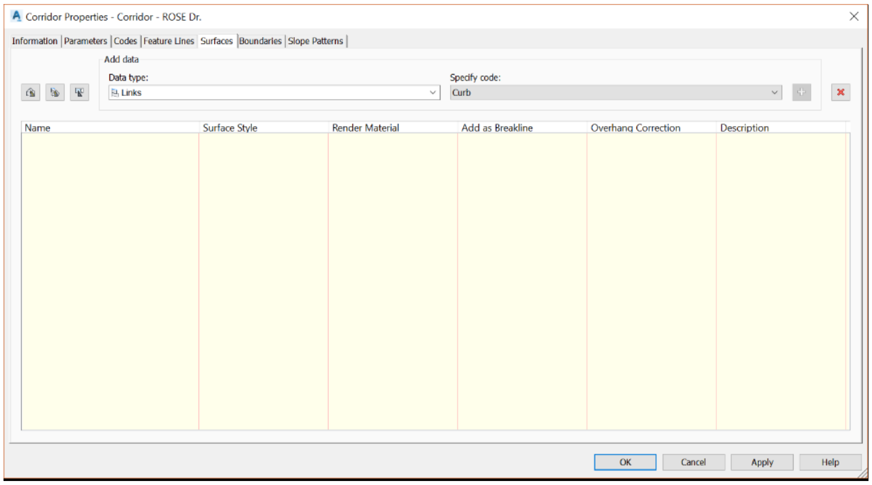

NOTES

NOTES

## 9.4 Creating a corridor Surface

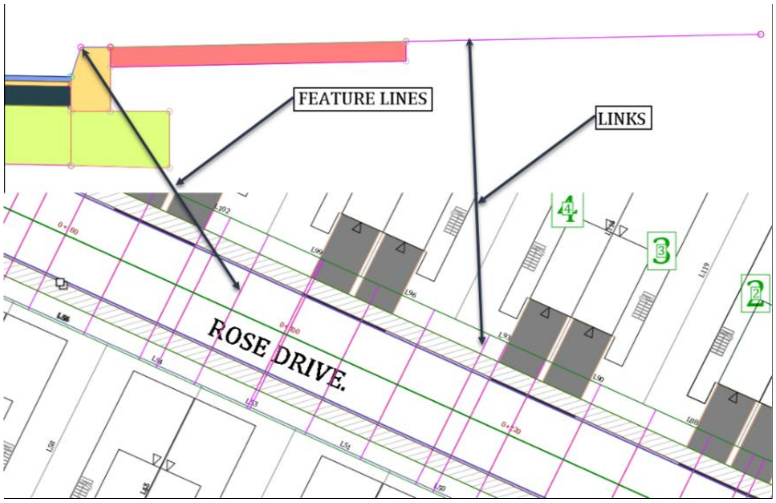
One of the best utilization of a corridor is that we can use it to generate a surface. The corridor surface can then be used or combined with other design elements to create a final ground surface. In can also help compute earthworks quantities or be exported for field stakeout, for example. Corridor surfaces remain dynamically linked to the original corridor. They will be updated if the corridor changes. Like any other type of surface, a corridor surface will also appear in the surface collection of the prospector. There, you can change its style, label it, or use it for a surface analysis.

1. The corridor surface is created using the **Surfaces** tab of the **Corridor Properties** dialog box.




2. As shown in the **data type** drop-down box, two types of corridor elements can be used to create a corridor surface:

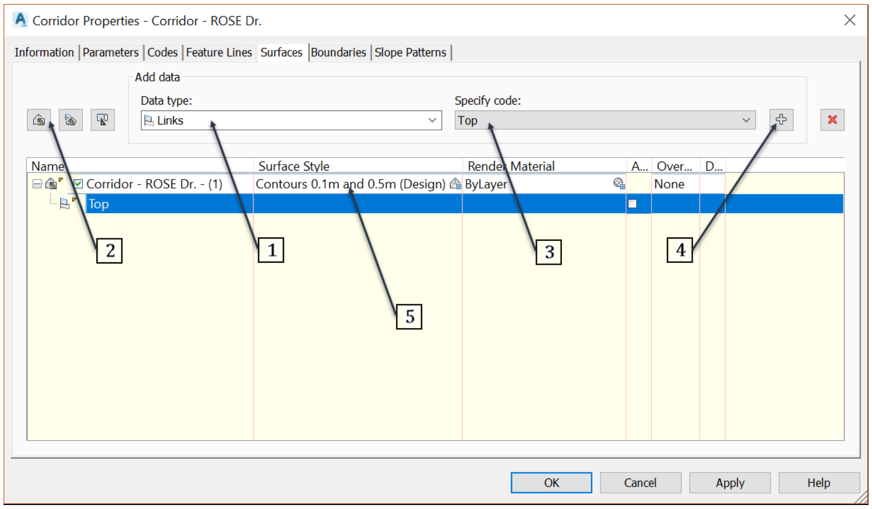
   - **Links**, which are the linear objects in the assembly like the lane, curbs or sidewalks, and
   - **Feature lines**, which represent the point objects such as road crowns, curb tops or backs, in the assembly. These point objects are converted to lines in plan view of the corridor.



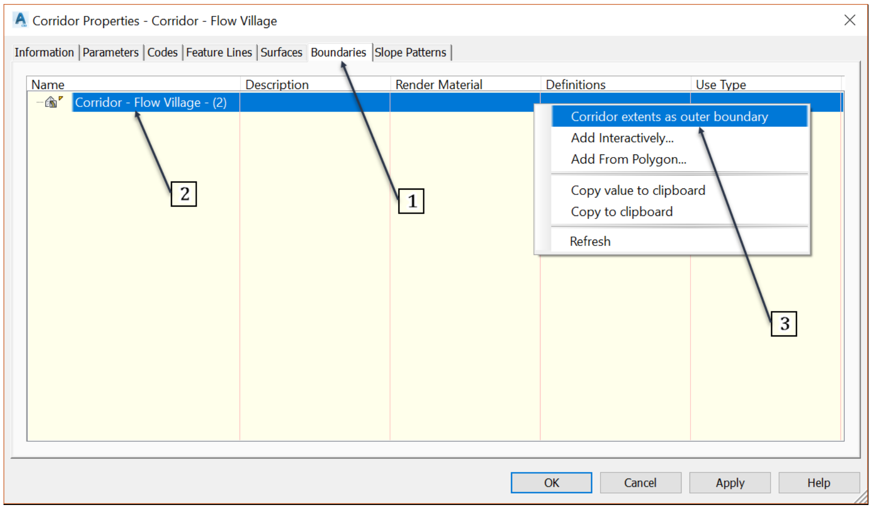
NOTES

3. Let's use the **Links** method to extract a surface from a set of link codes. Since we are looking to create a finished ground surface, we will use the **Top** link code. This code is included in all superficial points such as lane crowns, the top edge of pavements, face and back of curbs, top of sidewalks and generic links. If we needed to create a base or subbase surface, we would just use the links with codes including **the base** or **subbase**. To create the top surface,

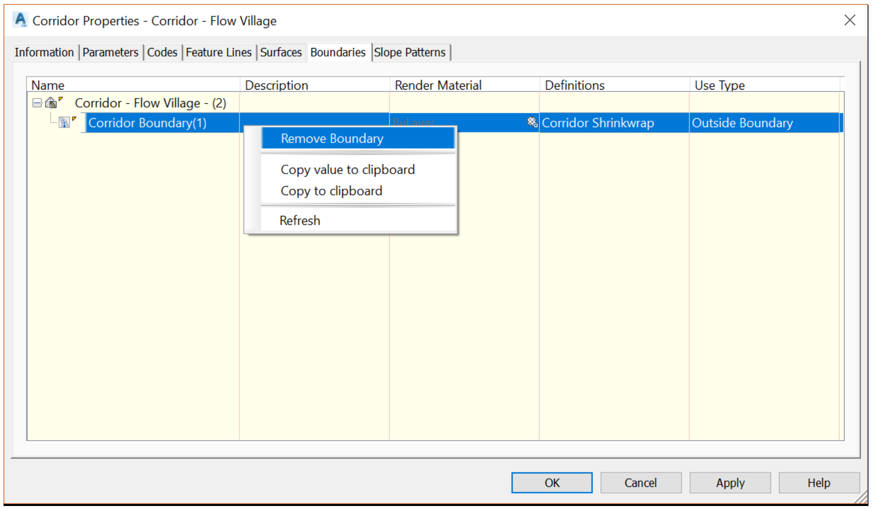
    - click on the **create a corridor surface** command,
    - specify a **Top** code,
    - then click on the "+" button to add the surface definition and choose a Surface Style.
    - Since this is a relatively flat road surface, we will choose a dense contour style, like the **0.1m** or **0.5ft** minor contours.




4. Once the surface has been defined, next we need to specify a boundary. Due to the triangulation between cross sections, corridor surfaces tend to extend beyond the roadway area. To force the surface to stay within the road right of way, we must define a corridor boundary. To do that, switch to the **Boundaries** tab, then right-click on the surface name and choose to add the **Corridor extents as outer boundary**. That adds the boundary automatically from the corridor extents. This option is most adequate for a corridor with many baselines, such as ours. When dealing with single baseline corridors, it would be better to add a boundary from <code name>. An example of sample codes is **ETW** - Edge of Travel Way or **P2** for generic assembly codes.




5. Optionally, we can also add boundaries interactively by clicking on the drawing. We can also select the **Polygon** option and click on a pre-existing closed polyline.
6. If you ever need to remove a corridor surface boundary, simply select it, right-click and select **remove boundary.** We will not do that in this case, since we need the corridor boundary.



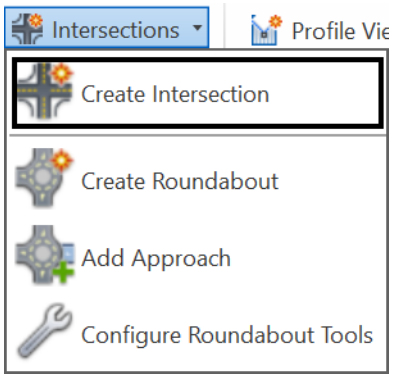

7. Now, click **OK** to close the corridor properties dialog box. You will notice that a corridor surface is created and displayed in the drawing area. Furthermore, it is bounded to the street right of way.


NOTES

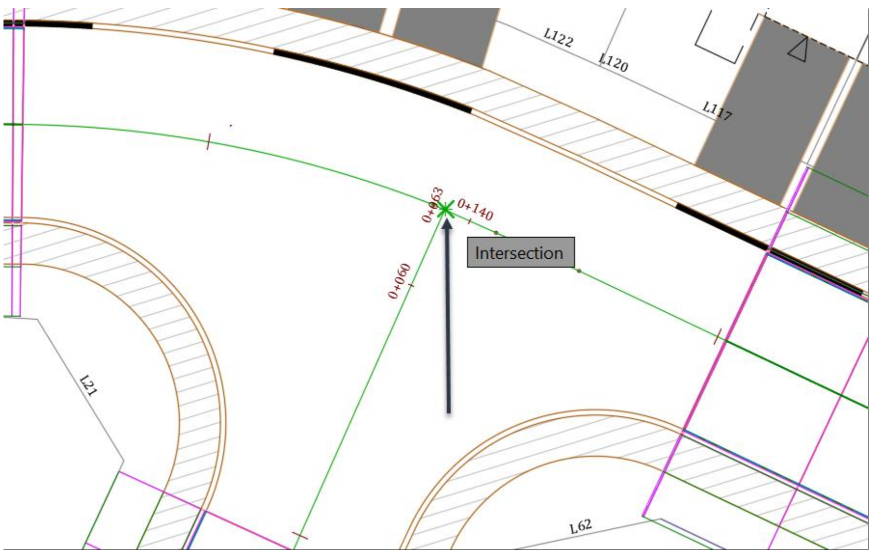
NOTES

8. Now, we need to create the intersection to fill the void we created by splitting the corridor and moving the regions. An intersection can be created automatically or manually by using assemblies and targets. Let's use the automatic method.

9. On the ribbon, run the **Create intersection** command.




10. Select the point by clicking on the intersection of **Rose Drive** and **Lavender Court**.

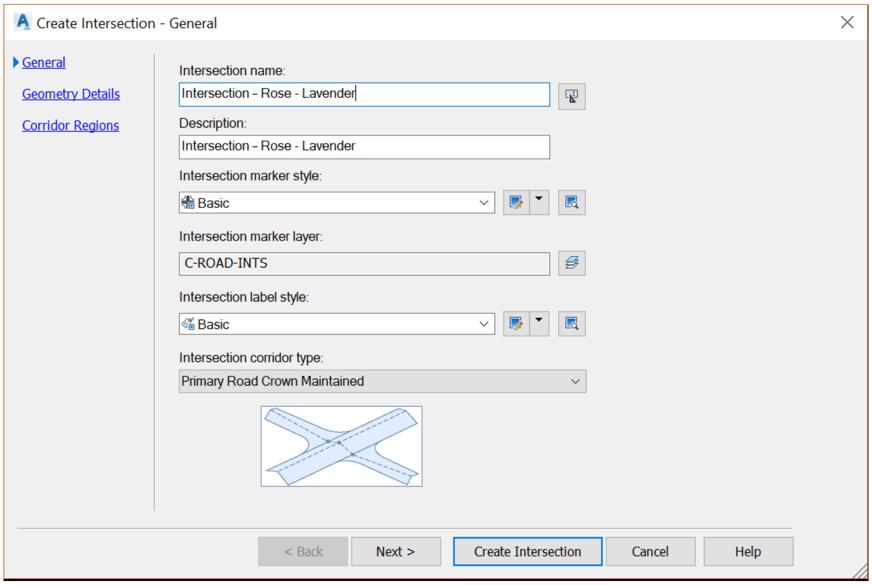



11. That opens the **Create Intersection** wizard. When you use two crossing alignments, you should specify the principal road and the secondary one. For a "**T**" type intersection, the continuous street (**Rose Drive**) will automatically be designated as the principal road. The discontinued street (**Lavender Court**) will be the secondary road.

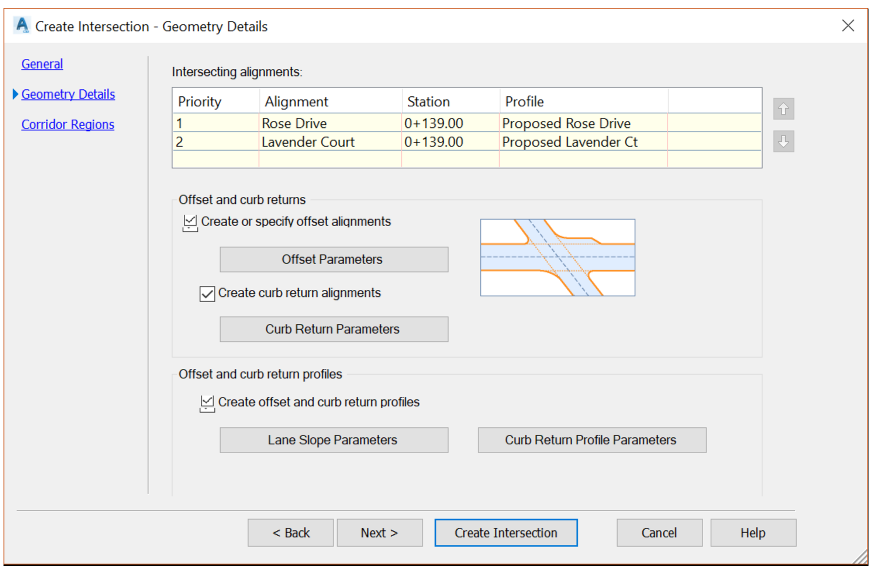
12. On the General page, we can specify general details for the intersection object, such as name, styles and corridor type.

    - Let's name it **Intersection – Rose - Lavender**.
    - For description we will use the same as the name, meaning **Intersection – Rose – Lavender.**
    - For the styles and layer, leave the default values.
    - Finally, for corridor type choose **Primary Road Crown Maintained.** With this option, the crown of the primary

road is maintained, while the secondary road is adjusted to match the primary road at the edges.




13. Click **Next** to move to the **Geometry Details** page. It is on this page that we specify most of the design parameters.



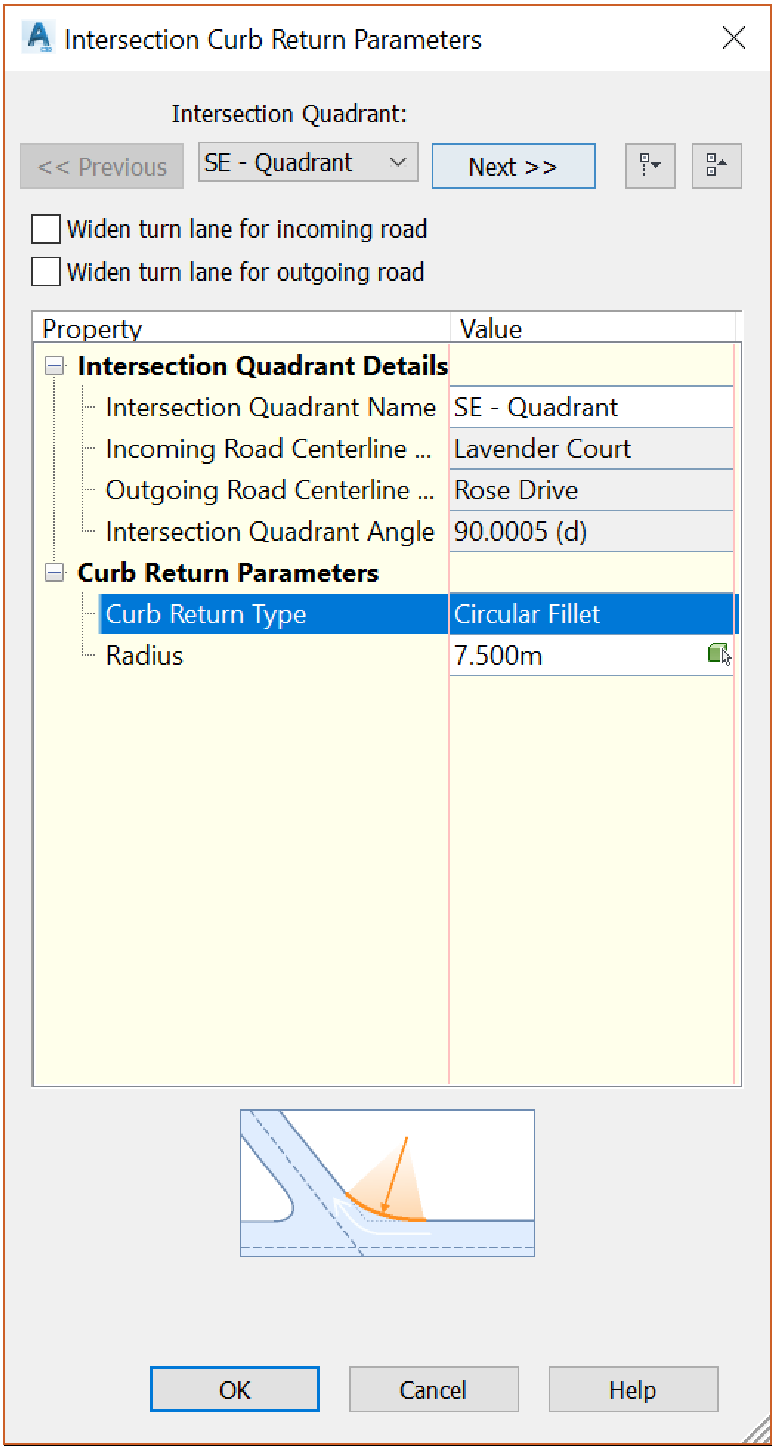

14. The first type of parameters we need to adjust is the street widths.

- Click on **Offset parameters.**
  - ☐ Change the **offset value** for each of the four street lanes to **3.5m or 11ft.** Notice how the image at the bottom adjusts to give us some visual cues on which element of the intersection is being adjusted.
  - ☐ Then, click **OK** to close the **Offset Parameters** window.

15. Next, we need to adjust the **curb returns** on the **Geometry Details** page.

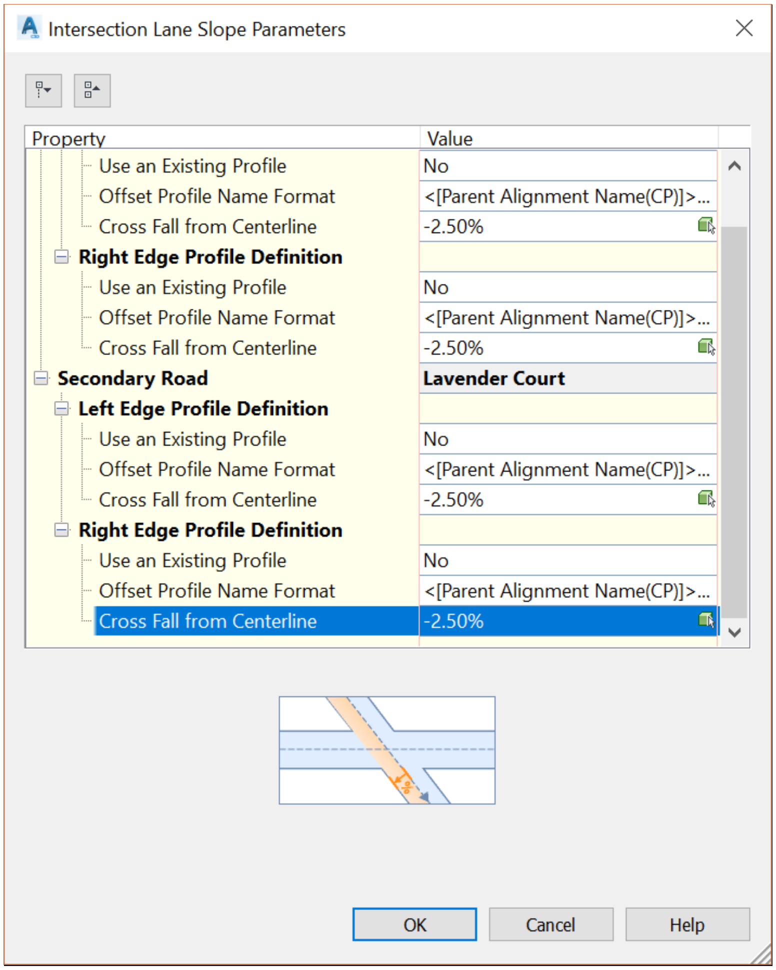
NOTES

- Click on **Curb Return Parameters**. The values we need to change in this window are the radiuses.
  - ☐ For **Curb Return Type**, keep the default **Circular Fillet;** and
  - ☐ Enter **7.5m** or **25ft** for radiuses.
  - ☐ Once you changed a value, click on the **Next** button to move to the next quadrant. In this case, we have two quadrants. For an "**X**" or "**+**" type intersection, you would have four quadrants.
  - ☐ After you are done adjusting the two quadrants, click on **OK** to close the **curb return parameters** window.




NOTES

- Next, we need to adjust the **lane slopes**.
  - ☐ This is done by first clicking on the **Lane Slope Parameters** button,
  - ☐ Then by adjusting the **cross fall from centerline** slopes.
  - ☐ Change the values to **-2.5%** for the four lanes, then
  - ☐ Click **OK**.

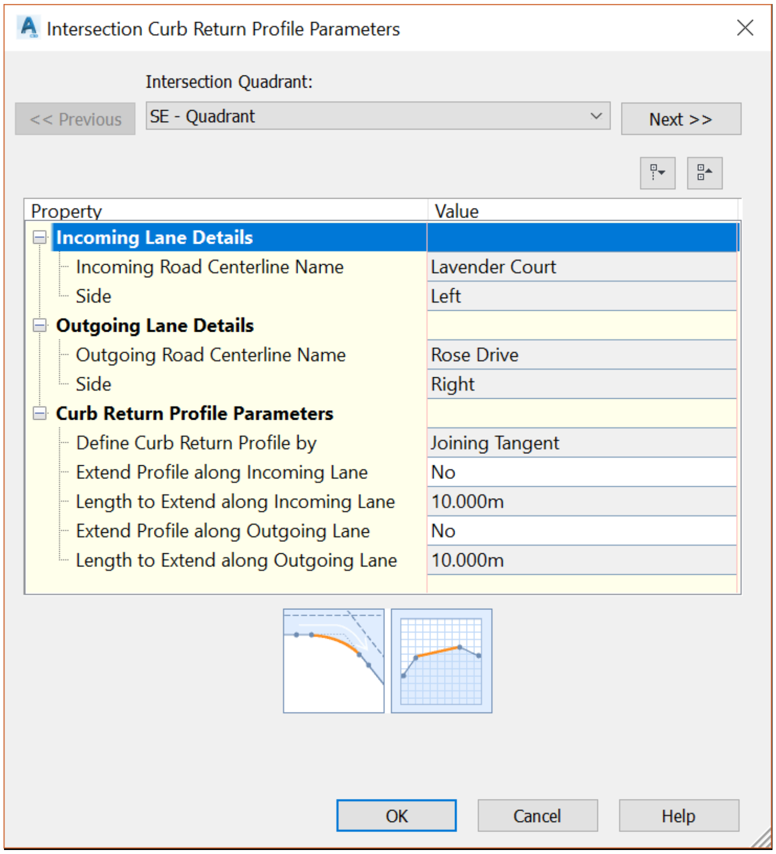



NOTES

NOTES

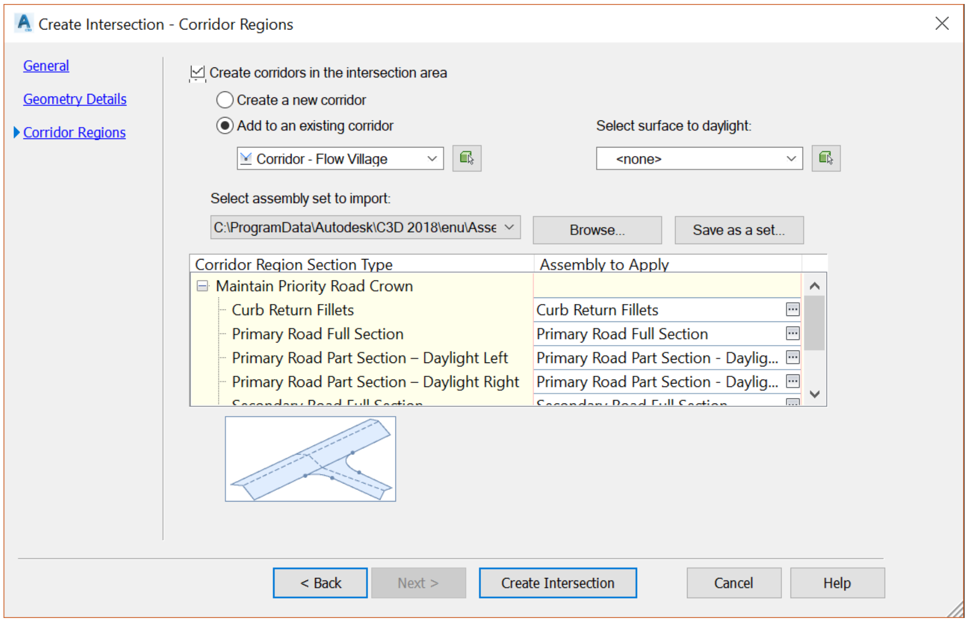
16. The last parameters we need to adjust on the **Geometry Details** page are the **curb profiles.**

- Click on **Curb Return Profile Parameters.** Here you can view and edit the parameters to extend the curb return profiles into incoming and outgoing lanes.
- Let's change the values to **1.5m** or **5ft**. We don't need to extend the intersection too far into the other streets.
- Make sure this adjustment is made for the two quadrants.
- Last, click on **OK**.




17. Next, click on next to move to the **Corridor Regions** page.

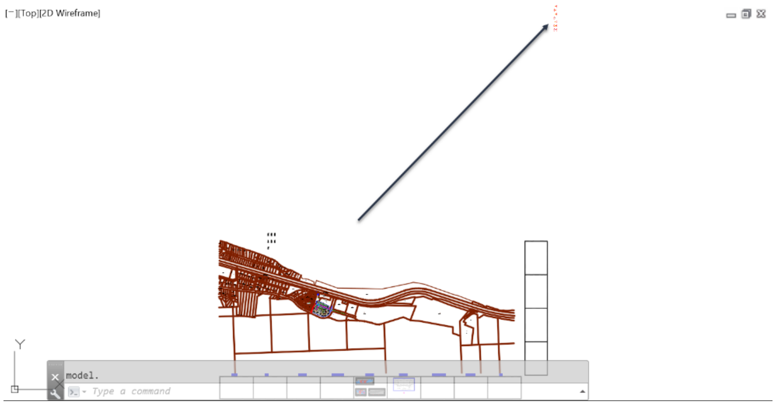
- Choose to **create corridors in the intersection area**, then add it to the existing **Corridor – Flower Village**.
- In the **Select surface to daylight** dropdown box, choose **none**. We will later daylight to the existing ground with our parameters. But daylighting with this particular set of assemblies may produce unexpected results. Because they are not set for our specific street.
- Leave the **select assembly set to import** to the default. These are default Civil 3d assemblies. We will modify them later to fit our requirements.




18. Click **Create an intersection**. The new intersection is created as expected.

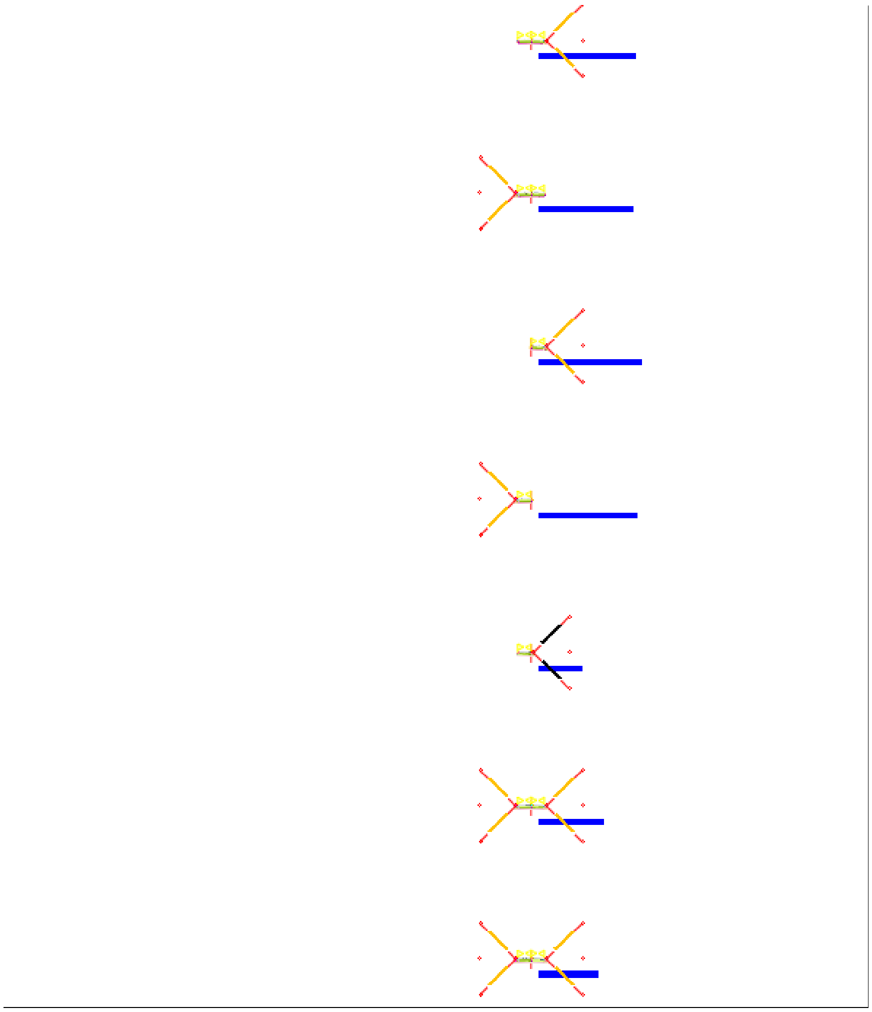
19. If you only see the intersection labels, but not the intersection itself, you may need to type **regenall,** at the command line**.** As an option, use the alias **REA** at the command line to regenerate the drawing and show the intersection.

20. Now that the intersection is created, we need to adjust it to use our typical road cross section. Remember, we have used a set of subassemblies that comes with Civil 3D, out-of-the box. If you do a zoom extent (alias **ZE** at the command line), you will notice them in an empty area, far from the project site.

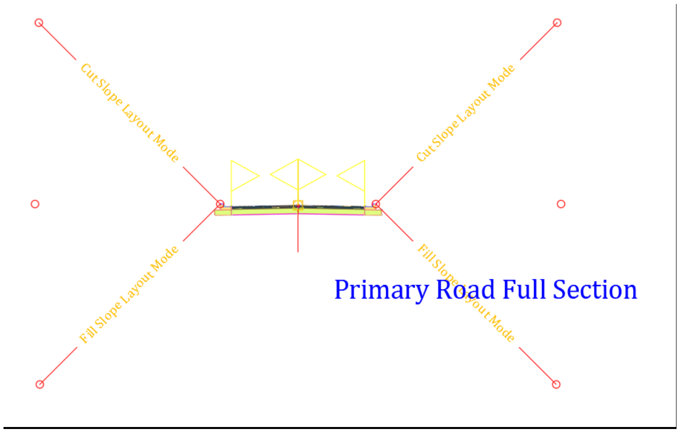



NOTES

21. Zoom closer and you have more information on these subassemblies.

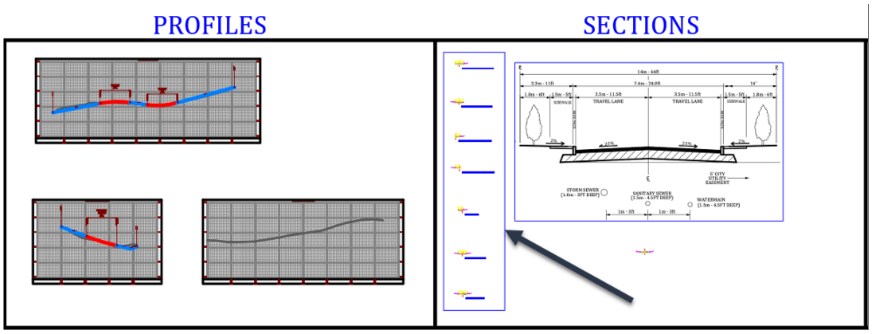


22. Zoom a little bit closer...

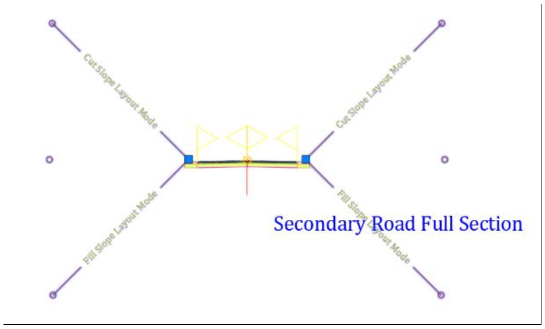



NOTES

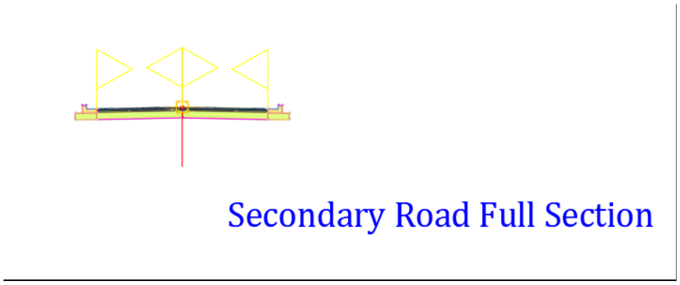
23. Next, let's move them closer to our drawing objects, in a spot we can find them easily, compared to the desert they are currently located in. A good spot may be the **Sections** rectangle at the bottom of the drawing. Simply select them and move them like here, for example.




24. Each one of these subassemblies is applied to the specific region of the intersection, indicated by their names. Next, our goal is to modify them to fit our specifications. So, we need to change the lane widths, curb types, add a sidewalk and landscaping strip, and delete unneeded subassemblies. Let's start by deleting all the **BasicSideSlopeCutDitch** subassemblies, in all assemblies of the imported set.

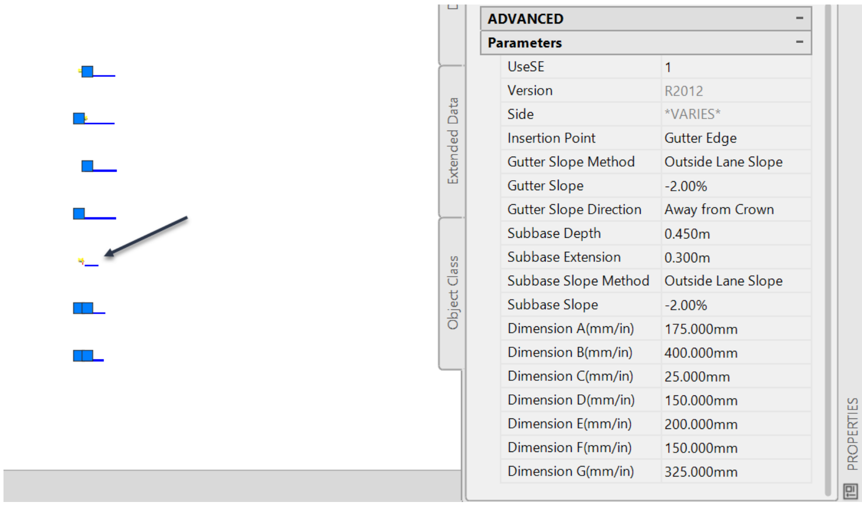



25. After deleting the **BasicSideSlopeCutDitch** subassembly, the secondary road should look as displayed.

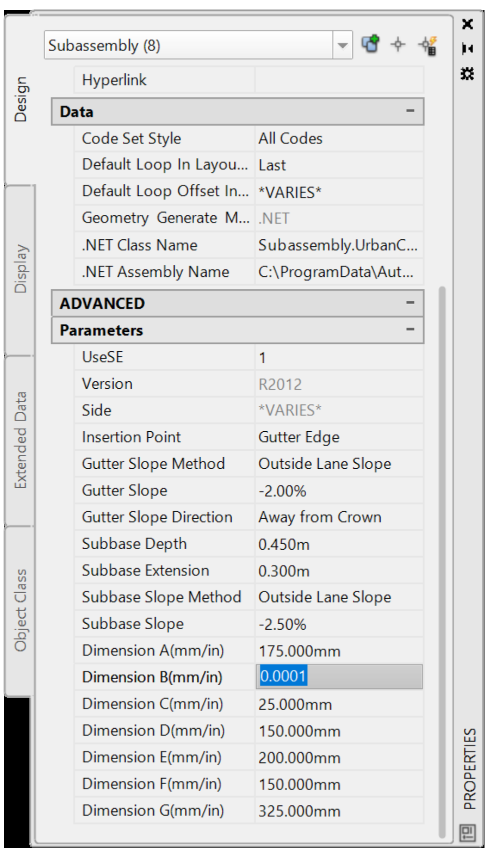



NOTES

26. Next, select all the curbs, except the one on the **Curb Return Fillets** assembly. The reason we are not selecting it is that for some reason Autodesk used an **R2010 version** of curbs while all other curbs are **R2012**. So, if we select it with the other curbs, we will not have the list of parameters we want to edit, in the properties window globally. We will make this point clear in a minute. For now, just proceed and select all curbs, except for this one.

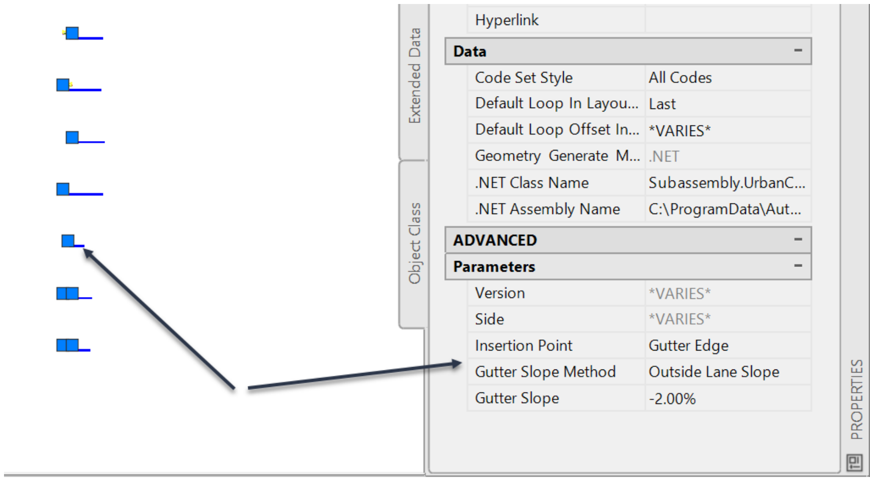



27. Now, we can globally edit the parameters of the curb to our specifications, as we have previously done. Don't forget to set the **B dimension** to 0.0001.

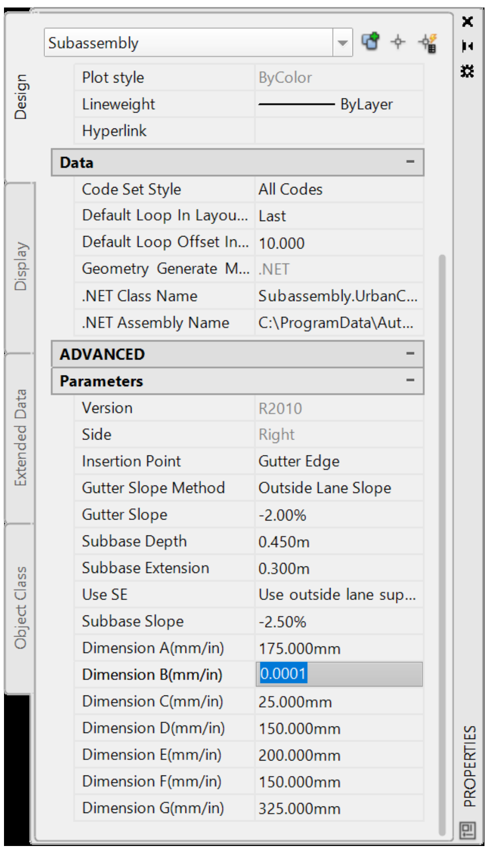



NOTES

28. Now, let’s add the curb on the **Curb Return Fillets** assembly to the selection set, to illustrate what we were mentioning. Click on it to add it to the selection set. Notice how the parameters list changes and that we don’t have access to the same list of parameters. That's because all the curbs are a version R2012, except the one used for the **Curb Return Fillets** assembly.

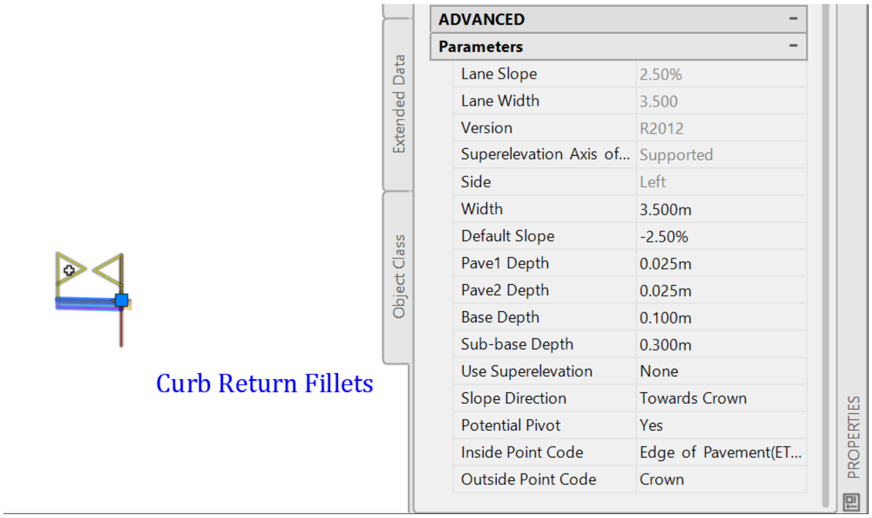



29. Now, press on **Esc** to deselect the subassemblies. Then, select that curb individually and adjust its parameters.

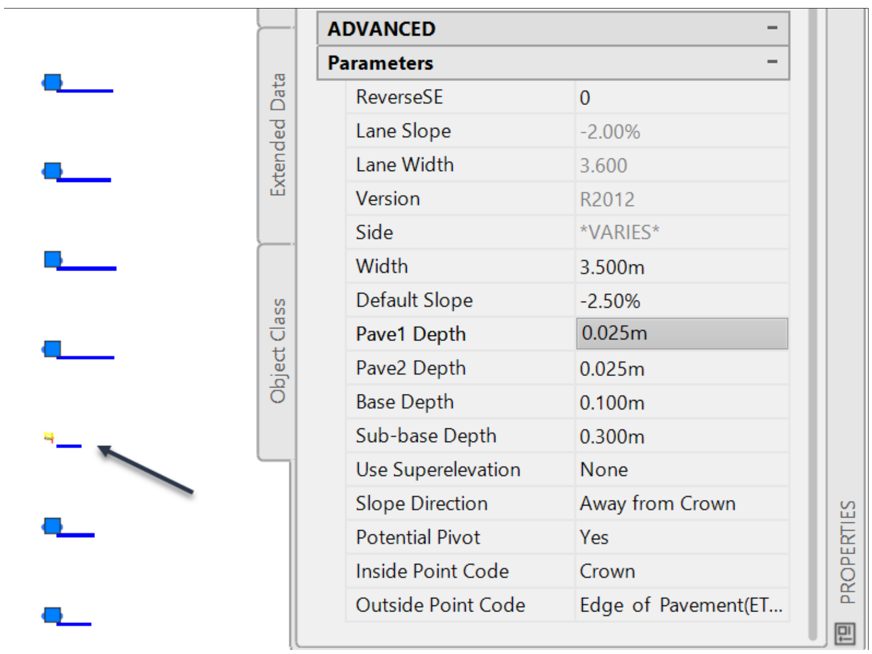



NOTES

30. While we are still at the curb return assembly, select the lane and adjust its parameters, mainly the **Slope** and **Width**.

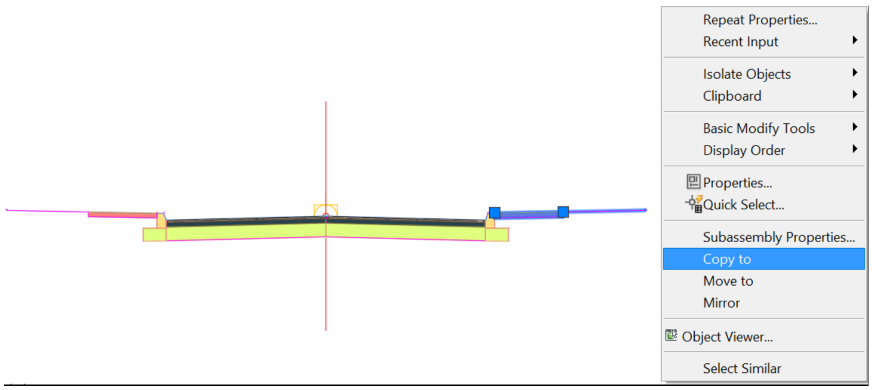



31. Now select all the remaining lanes, except the one on the **curb return assembly** and adjust their parameters.

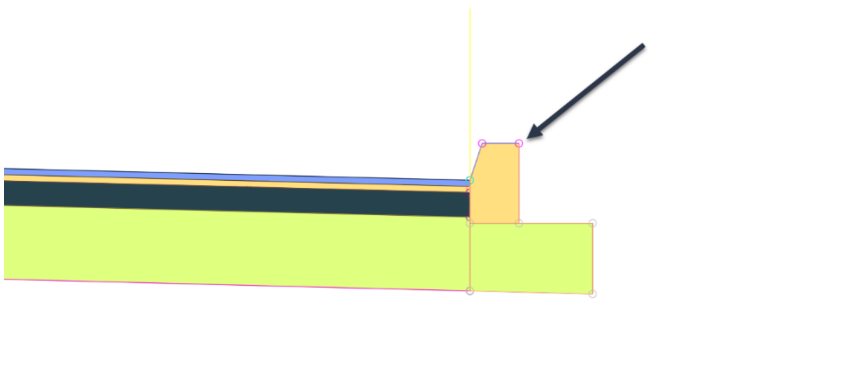


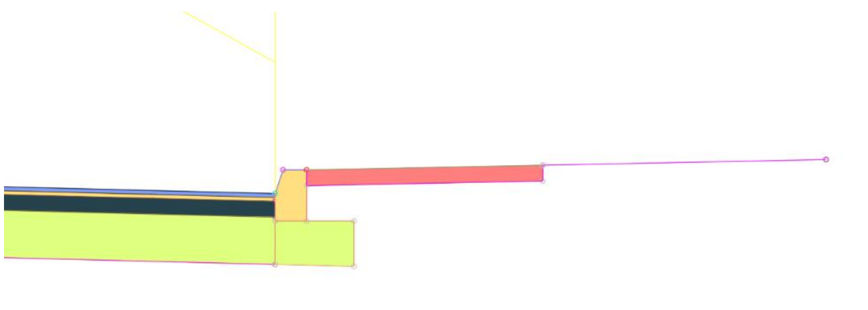

NOTES

32. Next, we need to add the sidewalks and the landscaping strip subassemblies to the assemblies. We can simply recreate them or copy them over from our original assembly. To copy the sidewalk and generic subassemblies, zoom to the original **City of Flower Bay - Local Street** assembly that we created. Then, right-click and select **Copy to**




33. Next, zoom to the set of assemblies and click on the top back of curb point to attach the two subassemblies we have copied. You need to be mindful of the side, left or right, you have copied the subassemblies from. Do not copy subassemblies to the opposite side.



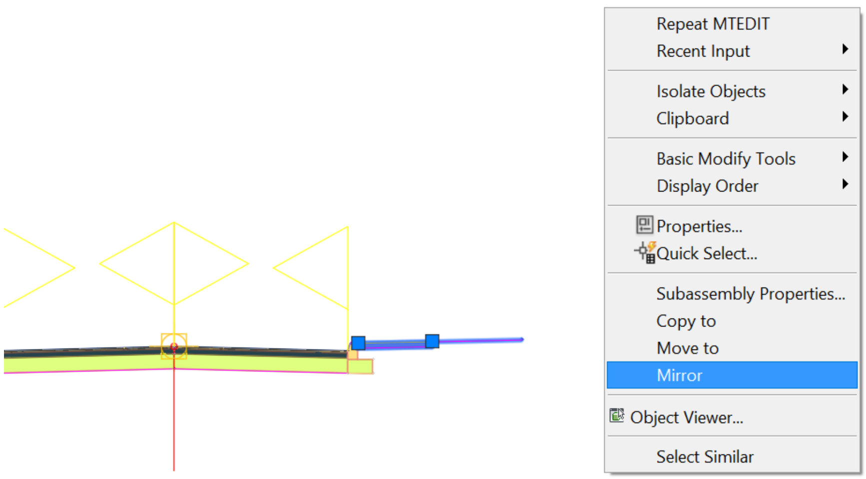


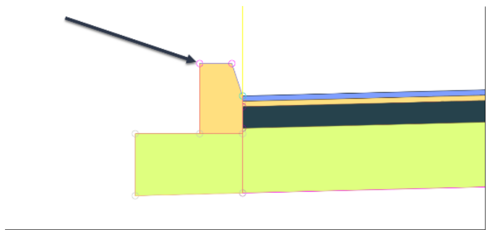
34. Repeat the process and duplicate the assemblies you have just copied to all other curbs on the same side. On the right side, we will copy them to the following assemblies: **Secondary Road Half Section - Daylight Right**, **Curb Return Fillets**, **Secondary Road Full Section** and **Primary Road Full Section**.

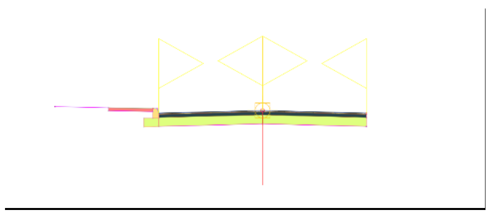
NOTES

NOTES

35. For the left side, select two subassemblies (the **sidewalk** and the **generic** one) on the right, **right-click** and select **Mirror**. Then click on the top back of curb points, on the opposite side.





36. Repeat the same process,

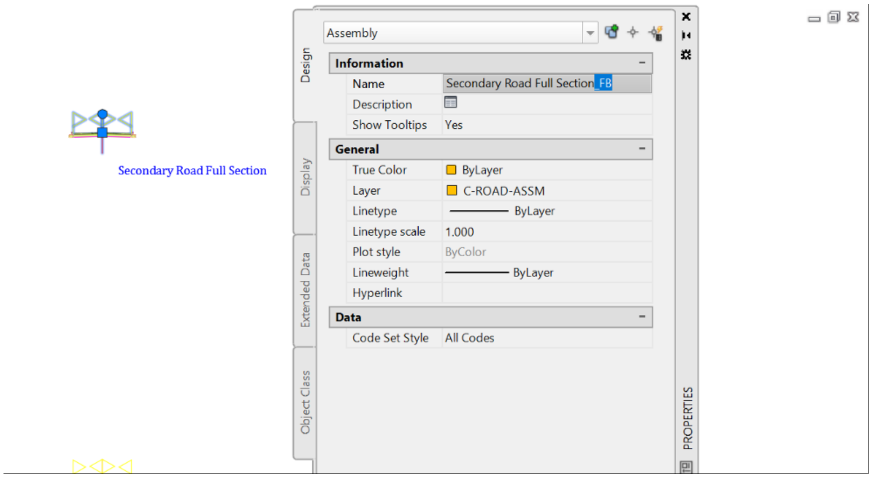
   - by mirroring from the right; or
   - by copying the sidewalk and generic assembly to the curbs on the left.

37. When done, you would have attached them on the left to **Primary Road Part Section - Daylight Left**, **Secondary Road Half Section - Daylight Left**, **Secondary Road Full Section**, and **Primary Road Full Section.**

38. We will now rename the typical sections to create a set that can be reused to create an intersection later in this project or another one. To do that,

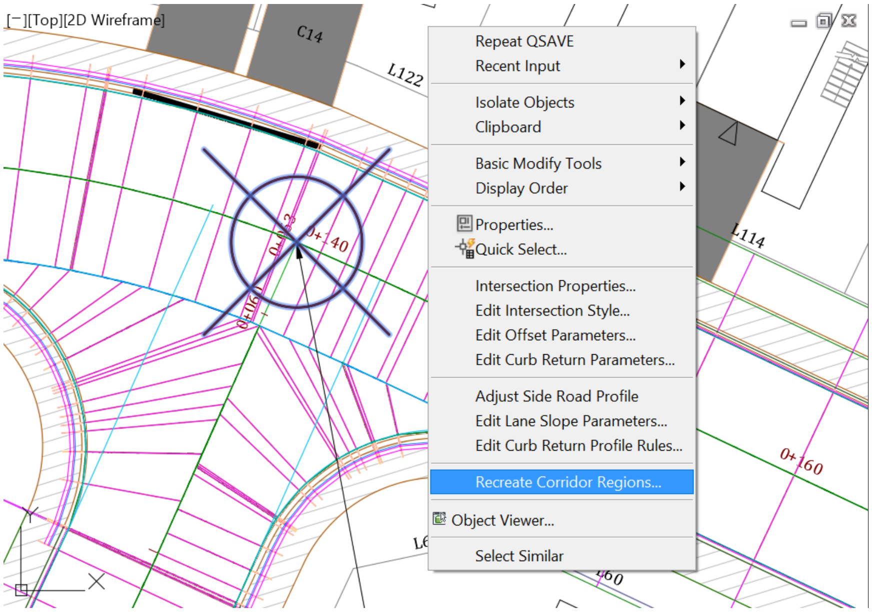
NOTES

39. Add a suffix or prefix of your choice to each assembly. For example, we can add the name of the municipality at the end of the assembly, such as adding the suffix **FB** for **Flower Bay**.

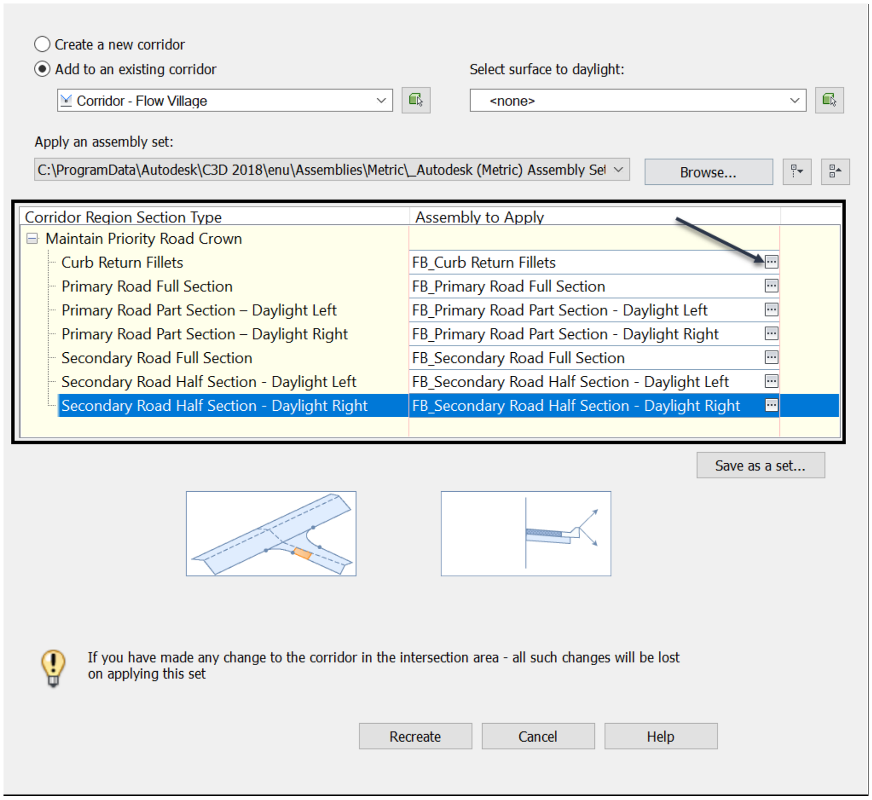



40. Repeat this process for each of the assemblies. Select them and add a suffix.

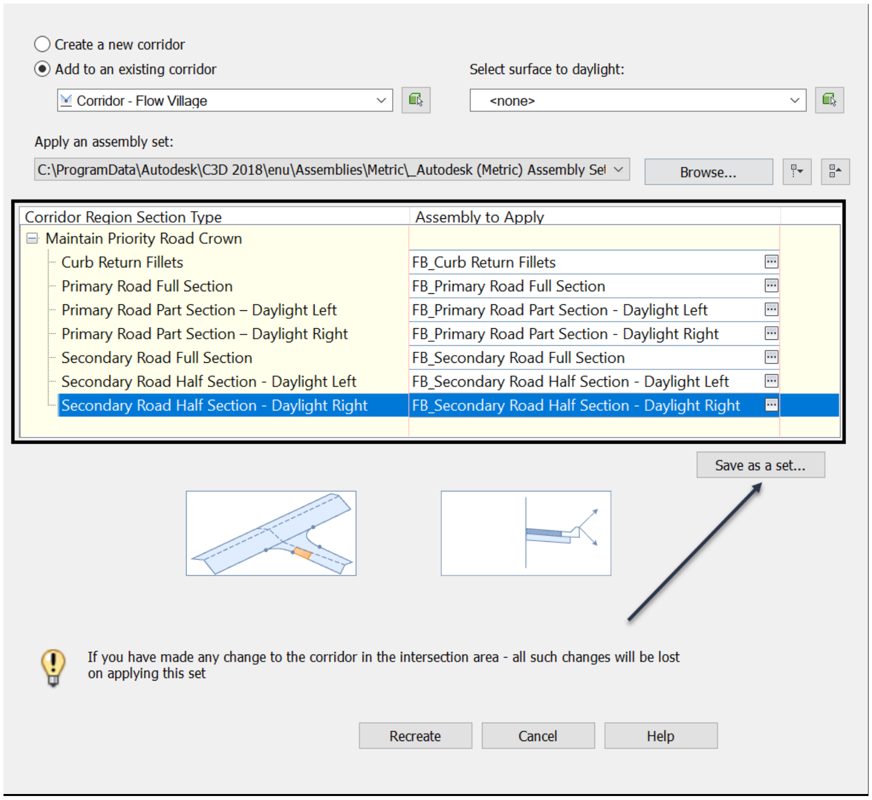
41. Now, let's create a set of assemblies for future use. Select the intersection symbol. Right-click and select **Recreate Corridor Regions**.



42. Replace each subassembly with the renamed ones, by clicking on the **three dots icon** to the right of each assembly.

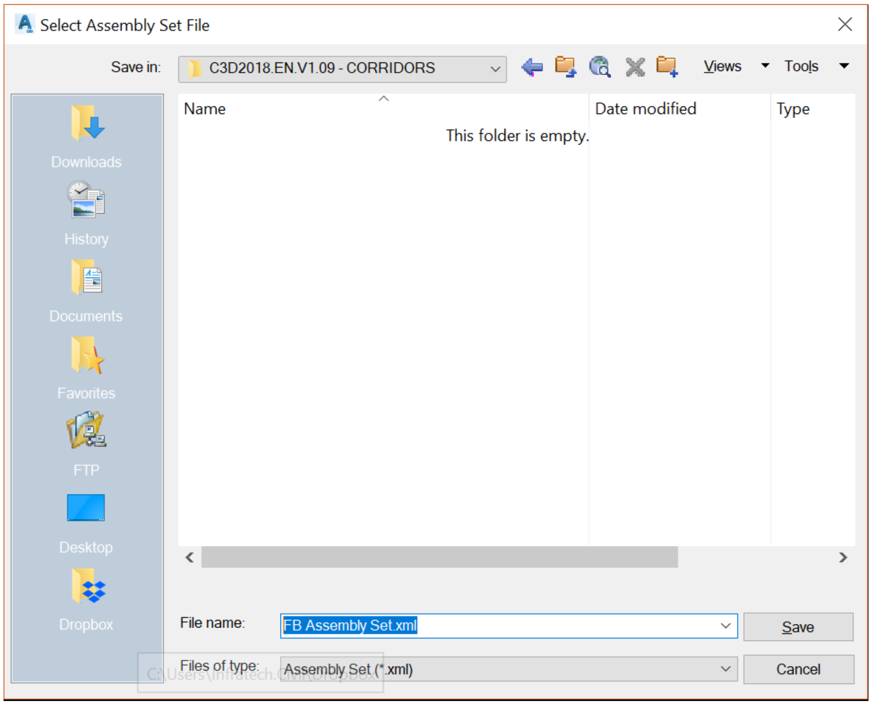



43. Once all subassemblies have been replaced, save the set under a new name in an accessible folder for future use.




NOTES

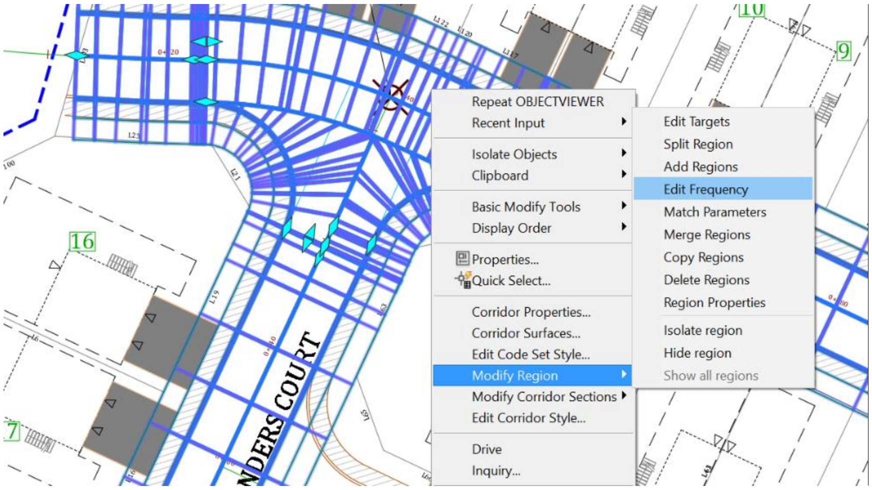
44. Now, browse to specify the folder to save the assembly set, in our corridor practice folder, for example.




45. Once you are back to the **Intersection Creation** window, click on **Recreate**. The intersection is now re-created with the new assemblies and their specifications.

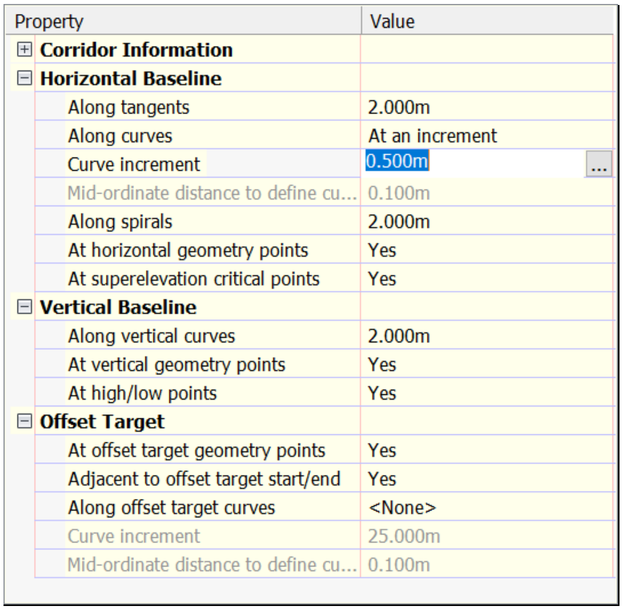

NOTES

46. We can now adjust the intersection by tightening the frequencies. To do so, select the intersection, right-click and select **Modify region**, then **Edit Frequency**.




47. Next, click inside one of the corridor quadrants, the southeast one for example.


48. Afterward, in the frequency editor change the **Curve Increment** to **0.5m or 1.5ft**.

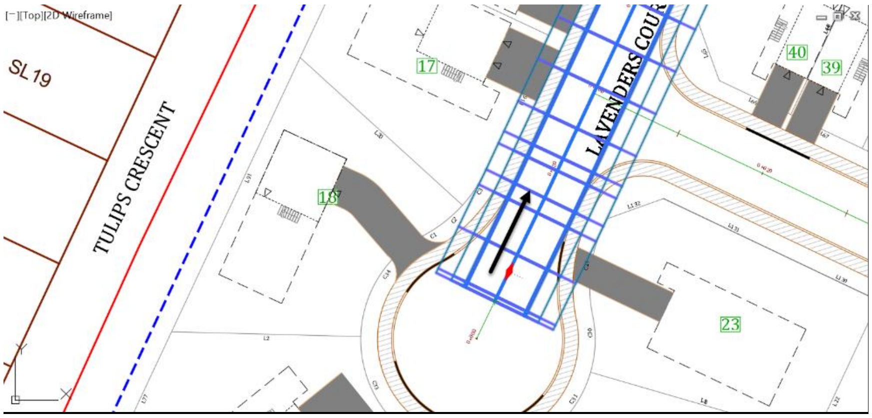


| Property | Value |
|---|---|
| ⊞ Corridor Information | |
| ⊟ Horizontal Baseline | |
| Along tangents | 2.000m |
| Along curves | At an increment |
| Curve increment | 0.500m |
| Mid-ordinate distance to define cu... | 0.100m |
| Along spirals | 2.000m |
| At horizontal geometry points | Yes |
| At superelevation critical points | Yes |
| ⊟ Vertical Baseline | |
| Along vertical curves | 2.000m |
| At vertical geometry points | Yes |
| At high/low points | Yes |
| ⊟ Offset Target | |
| At offset target geometry points | Yes |
| Adjacent to offset target start/end | Yes |
| Along offset target curves | <None> |
| Curve increment | 25.000m |
| Mid-ordinate distance to define cu... | 0.100m |

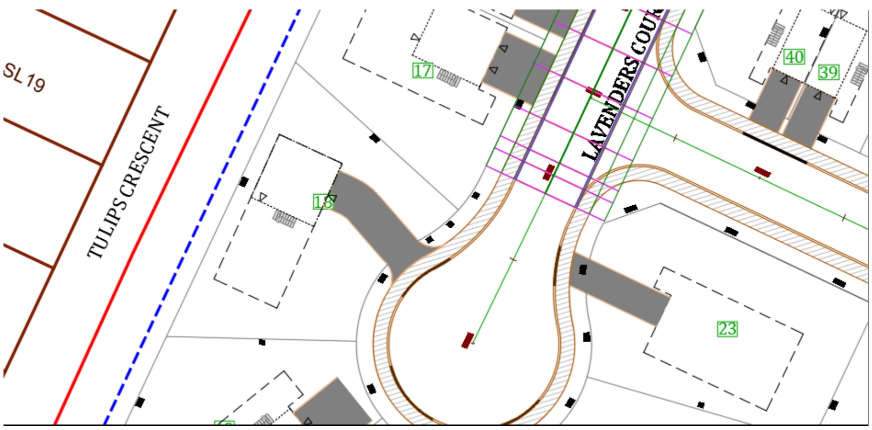
NOTES

## 9.5 Cul de Sac Design

1. The next step in the corridor creation process is designing the cul-de-sacs. Let's start with the one on **Lavender Court**. If your corridor extends into the cul-de-sac area, simply select the zone and move it a bit north, past the start of both cul-de-sac curves. This will create some space for us to work with. Don't worry about the exact location for now. We will tie the cul-de-sac and the tangent zones of the corridor later. For instance, the initial position may look like this.




2. And the final position, after dragging will look like this.



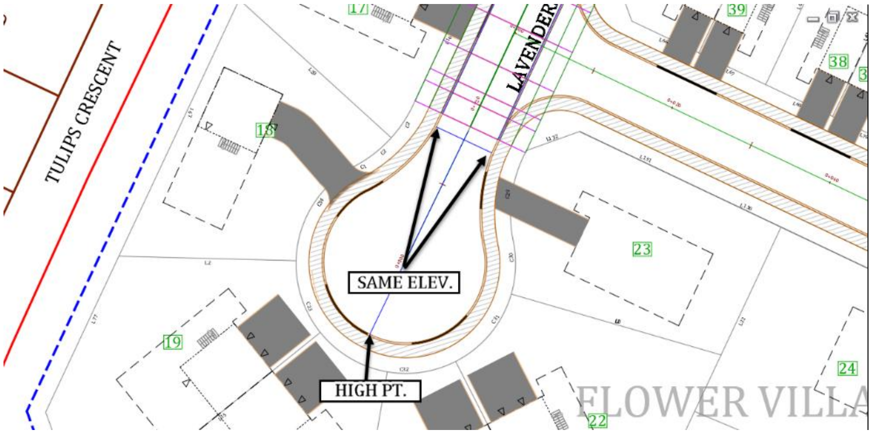

3. The bad news is that we don't have an automatic tool like the intersection wizard to create cul-de-sacs. But the good news is that with a little bit of creativity, we can create cul-de-sacs as easily as intersections, if not easier. Besides, if we master this process, we will be able to create corridors in many other circumstances.

4. The idea here is to consider the cul-de-sac as a curb return as we did for the intersection. That means, instead of working from the centerline outward to the curb, we need to work from the curb inward to the road centerline. So, we must have an alignment and profile for the curb line and apply the curb return subassembly to create a corridor.

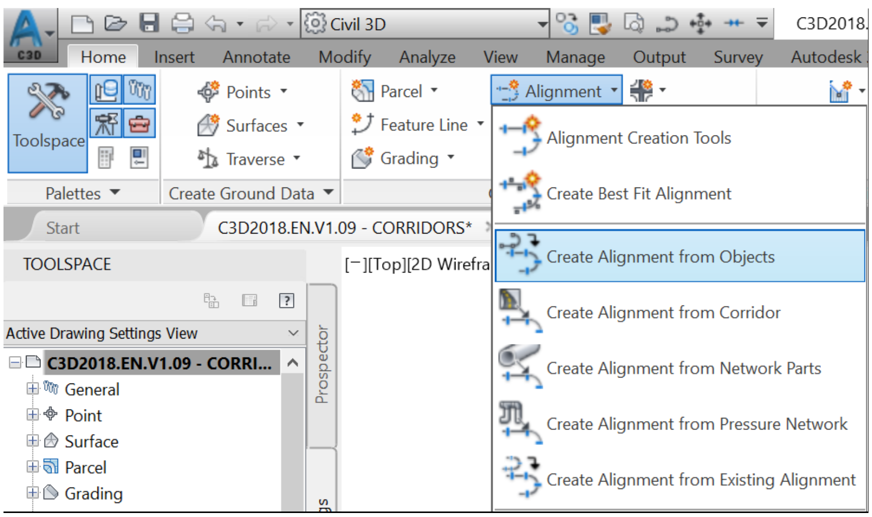
NOTES

NOTES

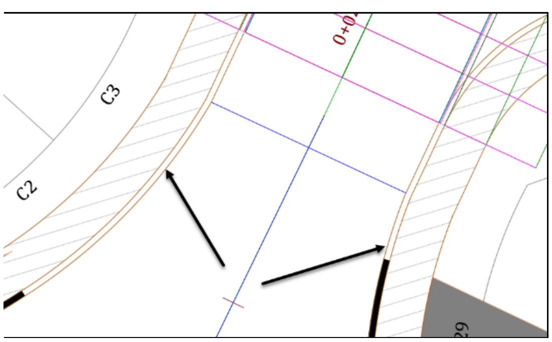
5. When creating the curb profile, we want to have the same elevations at the start and end of the zone and set a high point at the end of the cul-de-sac.




6. Now, let's start creating the curb profile. First, from the ribbon, use the **Create Alignment from Objects** command.

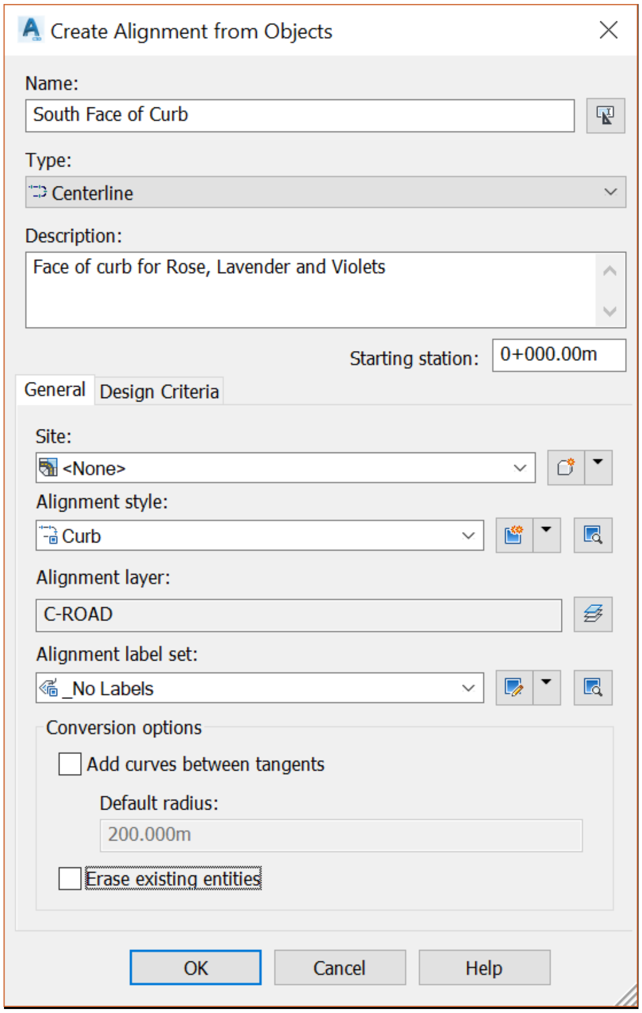



7. Select the curb line.



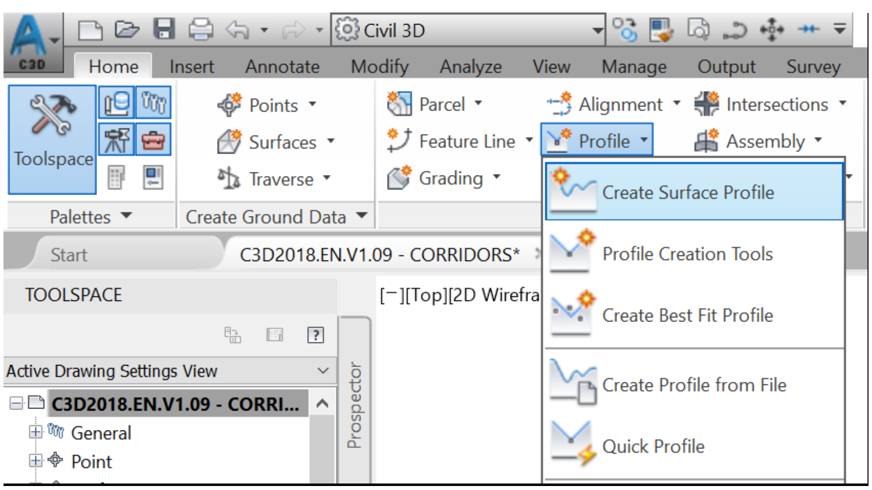

8. At the command line, press **Enter** twice, first to confirm the polyline, then a second time to accept the direction of the alignment.

9. In the **Create Alignment from Objects** window, assign the name, description and alignment style which we will set to **Curb**. To avoid cluttering the drawing, we will not assign any label. For **Conversion Options**, we do not want to **Add curves between tangents** nor do we want to **Erase the existing entity**.

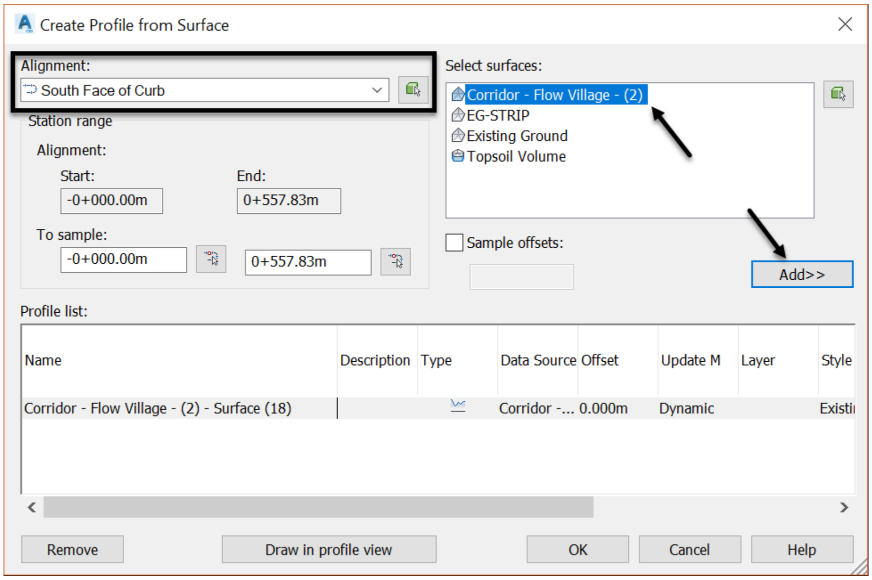



10. Click **OK** to create the alignment.

11. Next, we need to associate a profile to the alignment. Create a profile using the corridor surface, from the **Create Surface Profile** command on the ribbon.

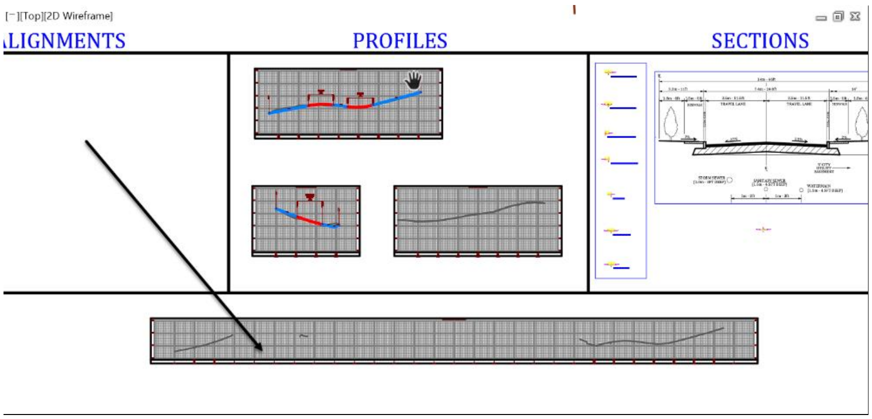



NOTES




12. After you've added the **Flower Village Corridor surface** using the **Add** button, click on **Draw in profile view**. We will accept the default settings for the profile creation because we have already used them and can change them if we need to.

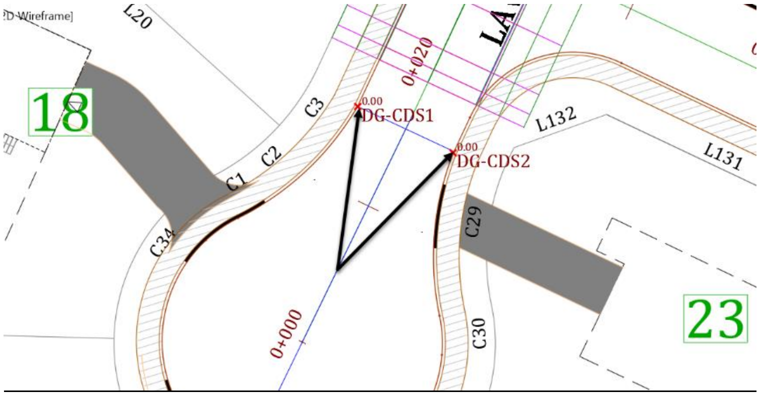
13. Next, click somewhere in the drawing area, under the existing profile views, to specify the bottom left corner of the profile view.



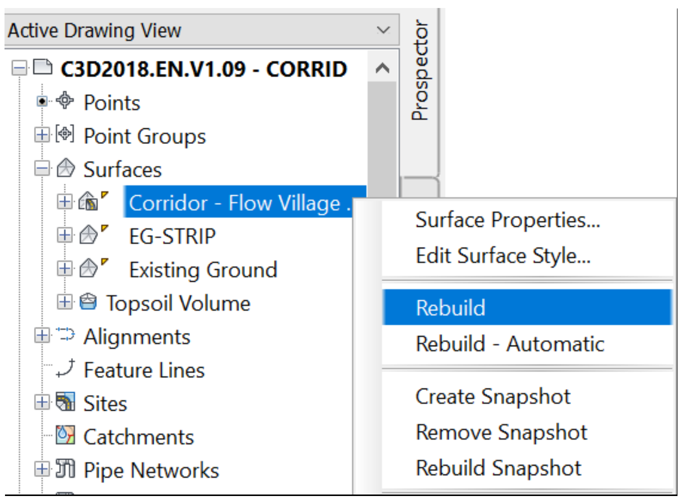

14. This profile represents the entire length of the curb. We need to identify where our cul-de-sac is located, in the profile view. We have multiple options to do that. One of the options is to: create points at the start and end of the cul-de-sac, then project them to the profile view.

    - Using the **manual** point creation command create points at the two locations mentioned. Let's give them the descriptions DG-CDS1, DG-CDS2, just to indicate that they are design points for the Cul-De-Sac.

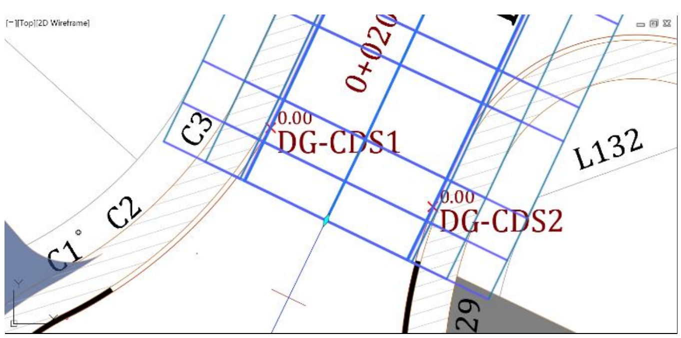
NOTES




- Next, we need to assign the point elevations. We want DG-CDS1 and DG-CDS2 to have the same elevations as the existing corridor.
- To do that, first, temporarily drag the corridor south, to cover the points area.
- Also, make sure the corridor surface is updated. In the prospector, select the Corridor – Flower Village's surface, right-click and rebuild the surface.



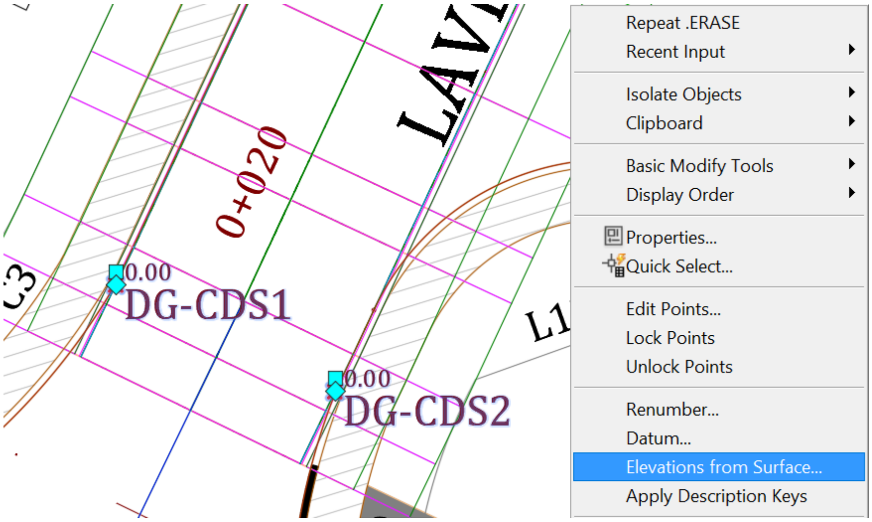

15. Once the corridor is dragged south, zoom to the 2 points and make sure they are on the pavement, not on the curb. If you need to, move them slightly to the pavement area.



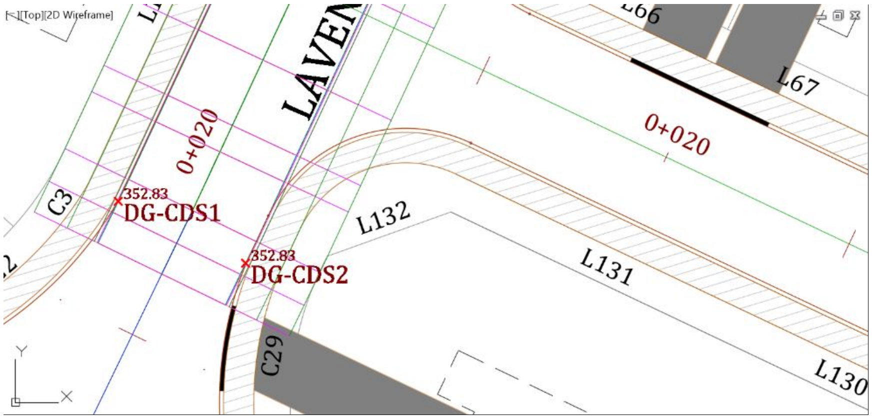

NOTES

NOTES

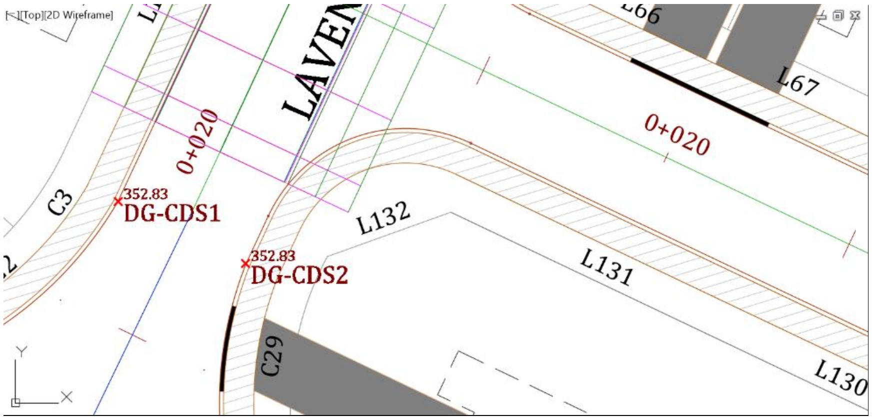
16. Next, select the two points and elevate them to the corridor surface, using the **Elevations from Surface** command.




17. In the **Select surface** window, select **Corridor – Flower Village** and click **OK**. The points are now assigned to the corridor elevations.

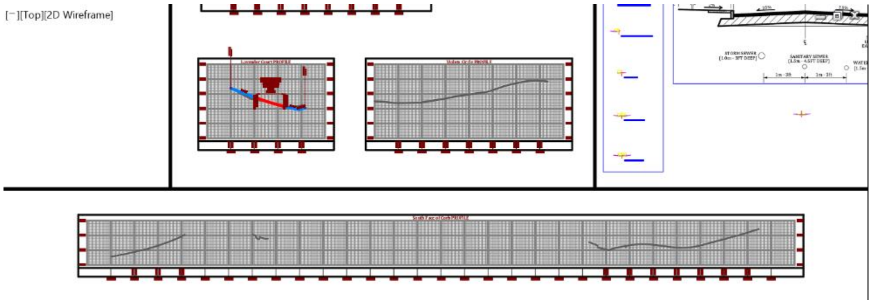



18. Now, we can move the corridor back, as it has served its temporary purpose of providing the needed elevations.

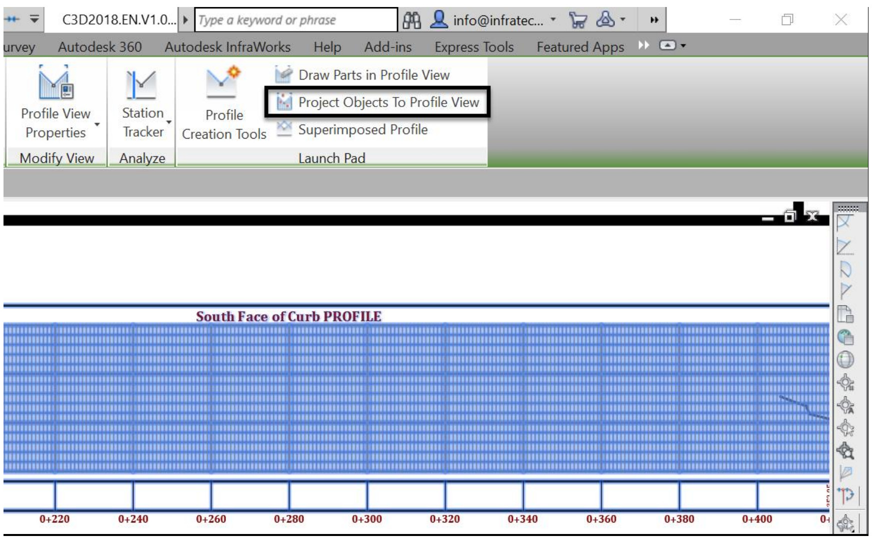



19. Next, we will use the two points to create a profile for the curb return. The points will also help us locate the cul-de-sac area on the profile view.

20. Now, **zoom** to the curb profile view.

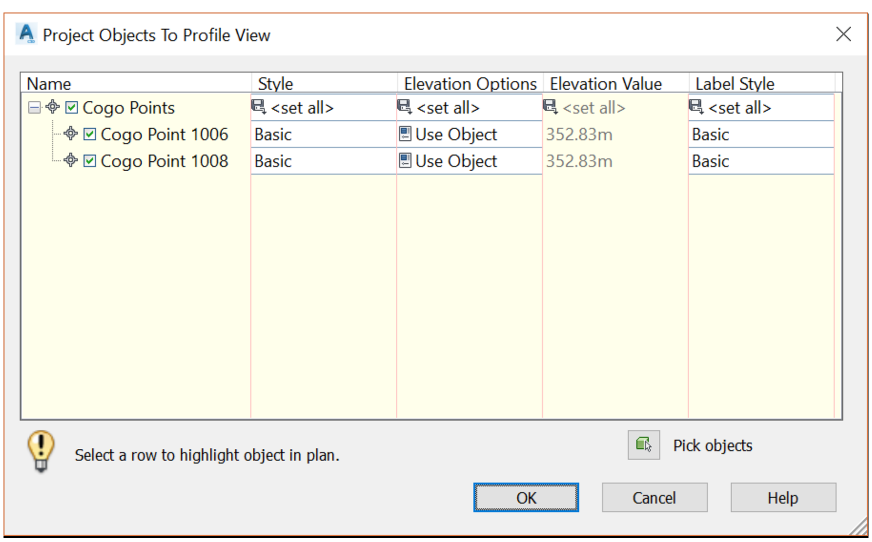


21. Then, **Select** it and on the ribbon, run the command **Project Objects To Profile View**.




22. In the drawing, select the two points and press enter at the command line to project them to the curb profile view.

23. In the next window, we can change the projected object styles, label styles, and elevation options. We will leave them as-is, for now, since we want to use the original elevations obtained from the corridor.

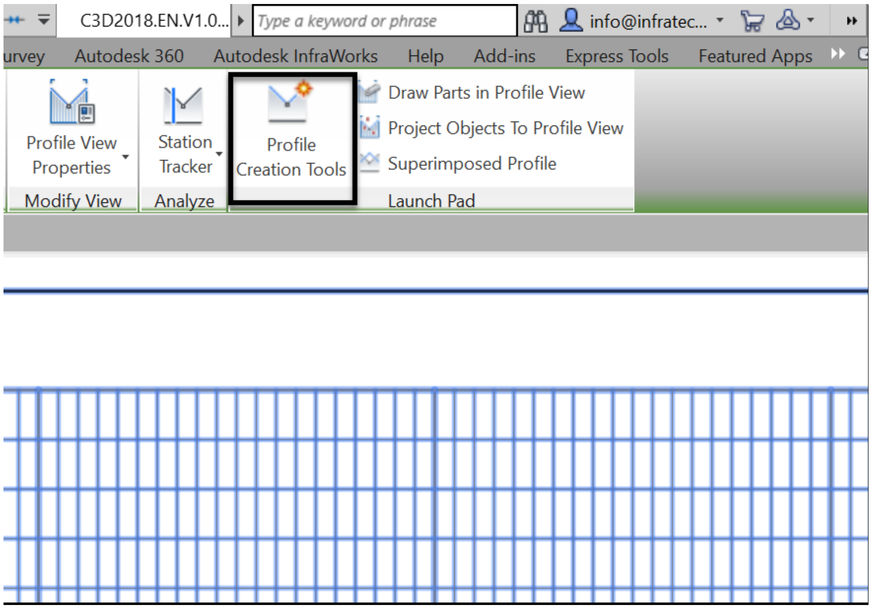



NOTES

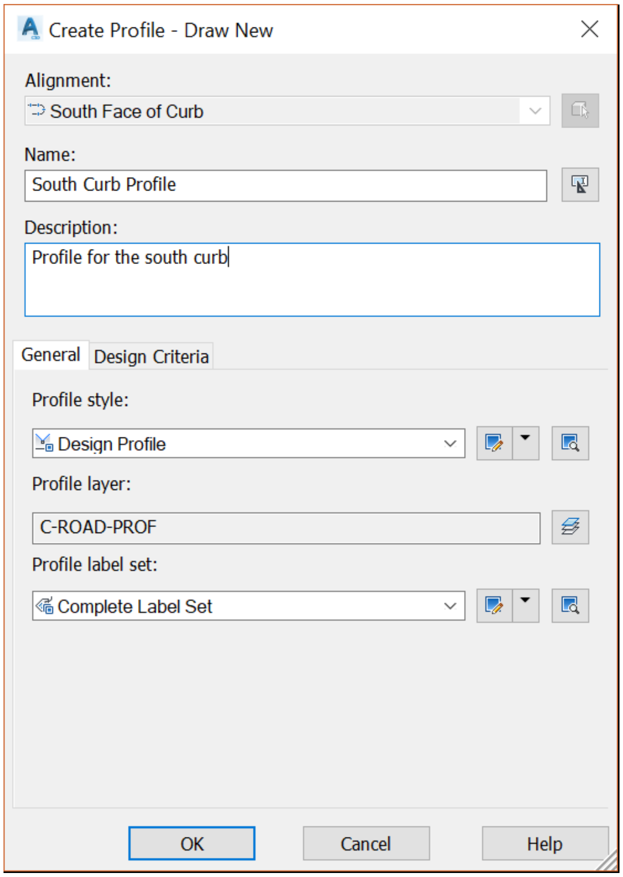
NOTES

24. Click **OK** to close the project window. Once you zoom back to the profile view, you will notice the projected points. This gives us a clue about the location of the cul-de-sac on the alignment.

25. We can easily create the profile now. Select the profile view and run the **Profile Creation Tools** command from the ribbon.

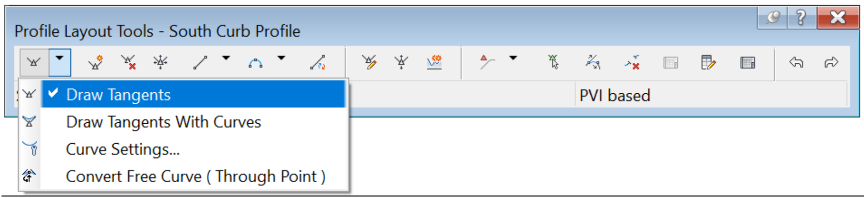


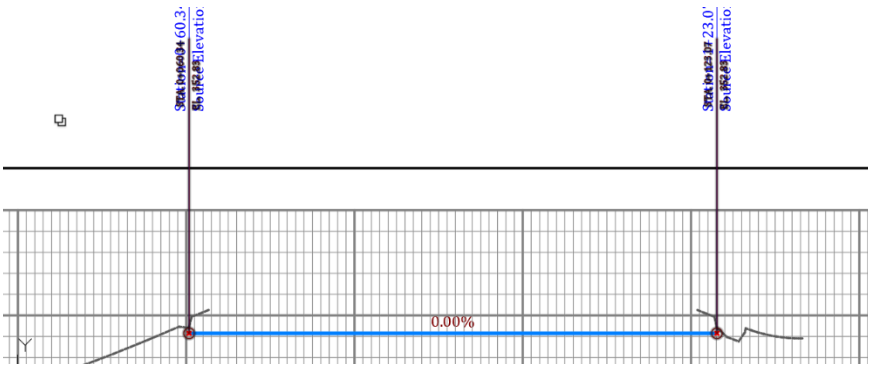

26. In the new window, assign the name to the profile, give it a description and choose a label set. This will not be a profile that we will print, so we don't need to worry about appearance here.



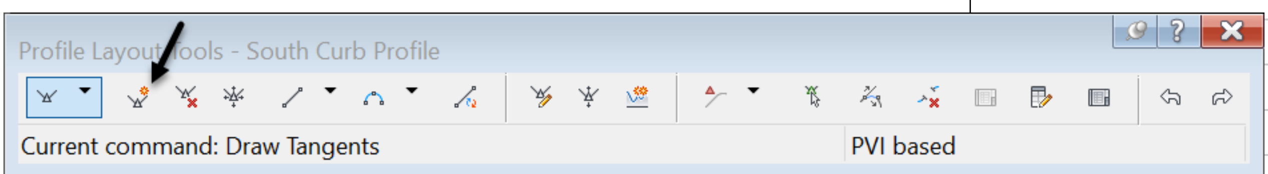
NOTES

27. Click **OK.** In the **Profile Layout Tools**, draw a tangent between the two projected points. You may need to use the node object snap mode to draw with precision.

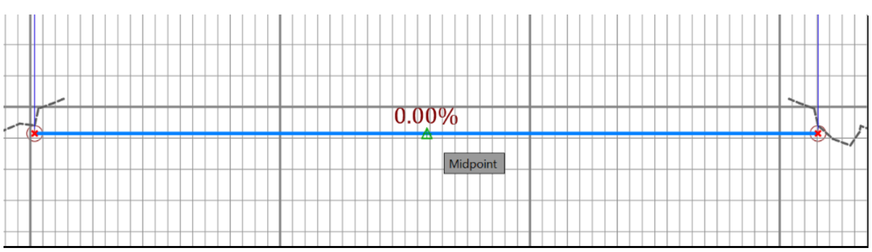


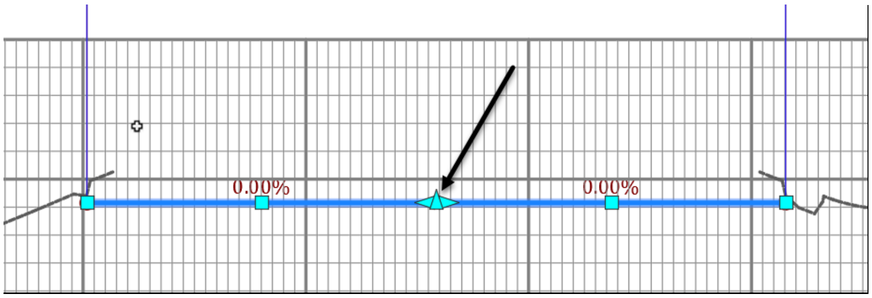

28. Next, we need to create the high point of the cul-de-sac. Use the **Profile Layout Tools** to create a **PVI** (Point of Vertical Intersection) in the middle of the tangent.



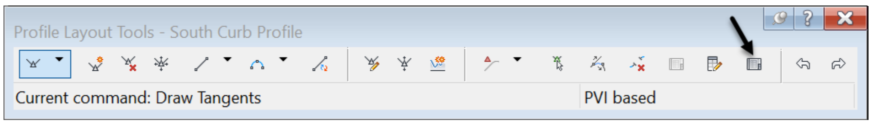

29. Next, with your middle **object snap** activated, click on the middle of the profile tangent, around the **0.00%** slope label, to create the **PVI** for the high point.



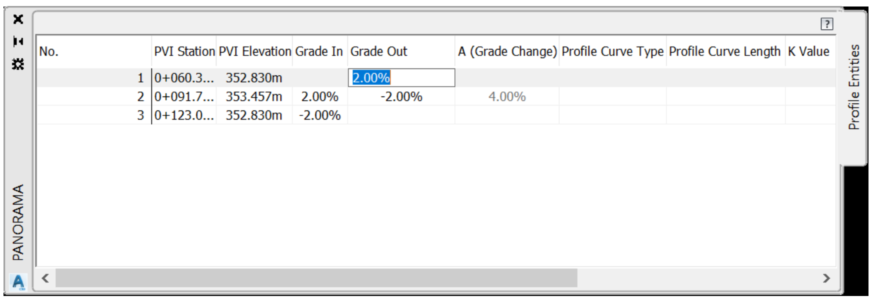

30. All that is left for the profile creation is to set the slopes. You can do it manually by grabbing the **PVI** point and slide it up, with your **Ortho** mode activated.



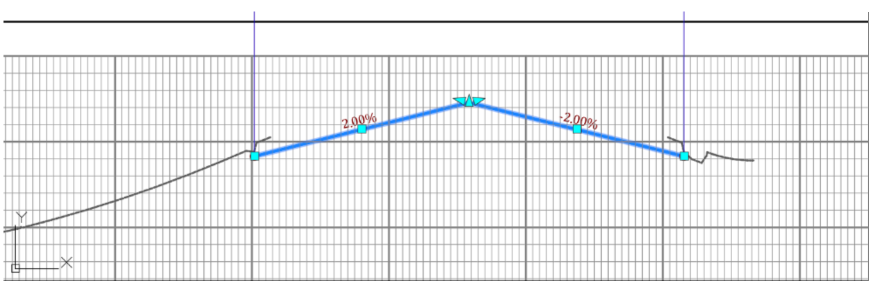

31. Or, you can set it with precision by using the tabular editing option of profiles. To do that, click on the **Grid View** command of the layout tools.




32. In the **Panorama**, set the grade out to **2%**.




33. That adjusts the profile automatically and creates a high point, with a **2%** grade.



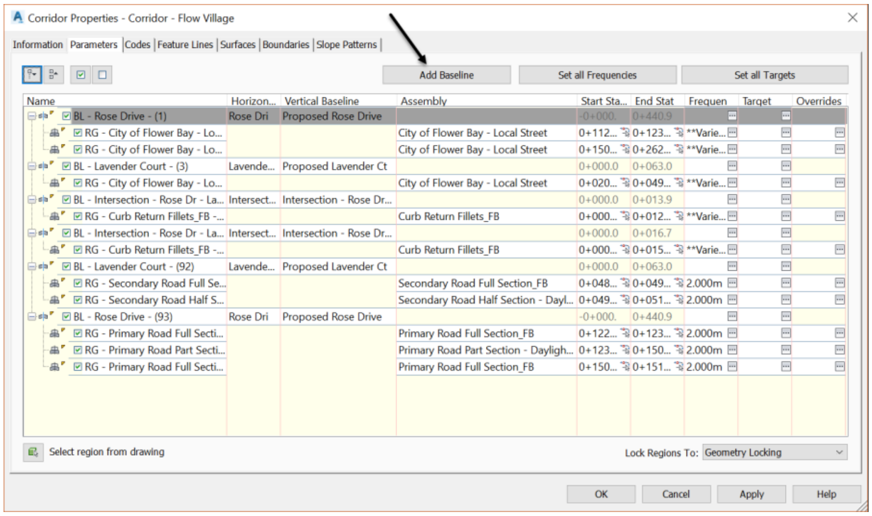

34. Now, we have an alignment, a profile, and subassembly. We are thus ready to create a new corridor for the cul-de-sac. Or, we can add a new baseline, to the existing corridor, to represent the cul-de-sac.

35. Let's go with the latter option. Select the corridor. Right-click, then select **Corridor Properties** and click on the **Parameters** tab.

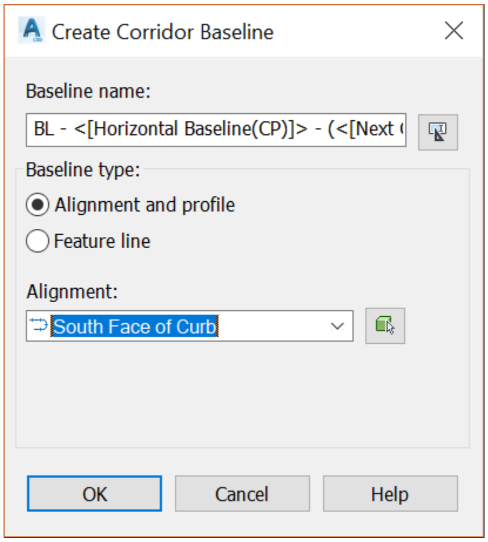
NOTES

NOTES

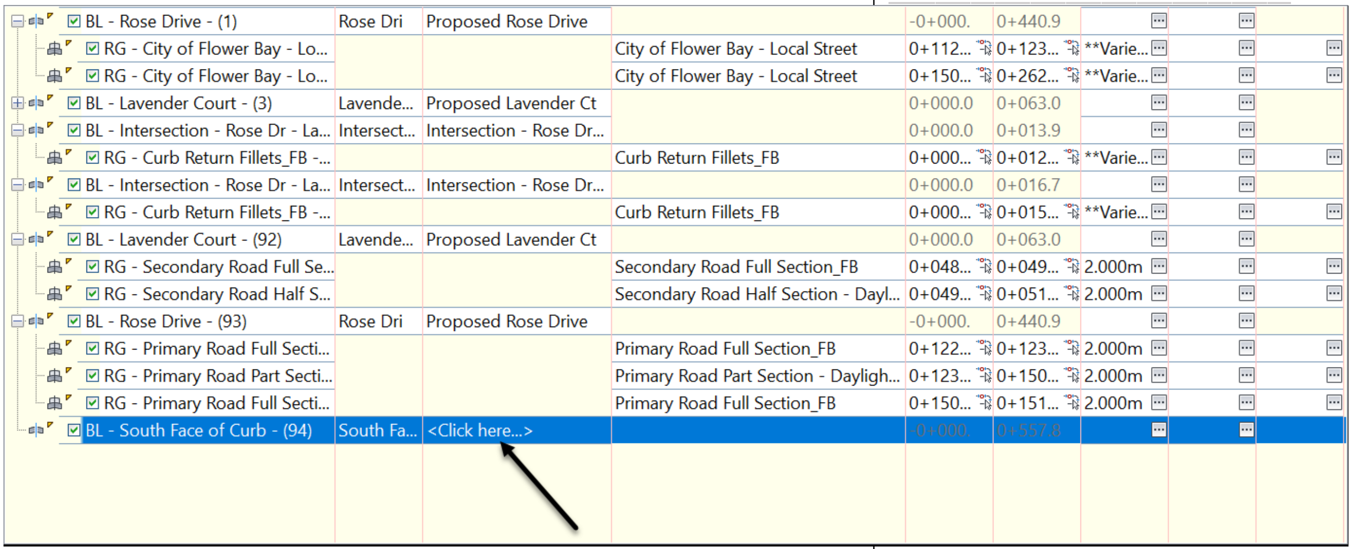
36. Next, click on **Add Baseline**.




37. Select the face of curb alignment and click **OK**

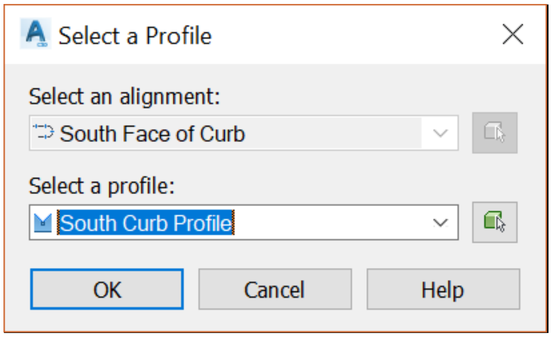



38. Then, click to specify the profile to use for the baseline.

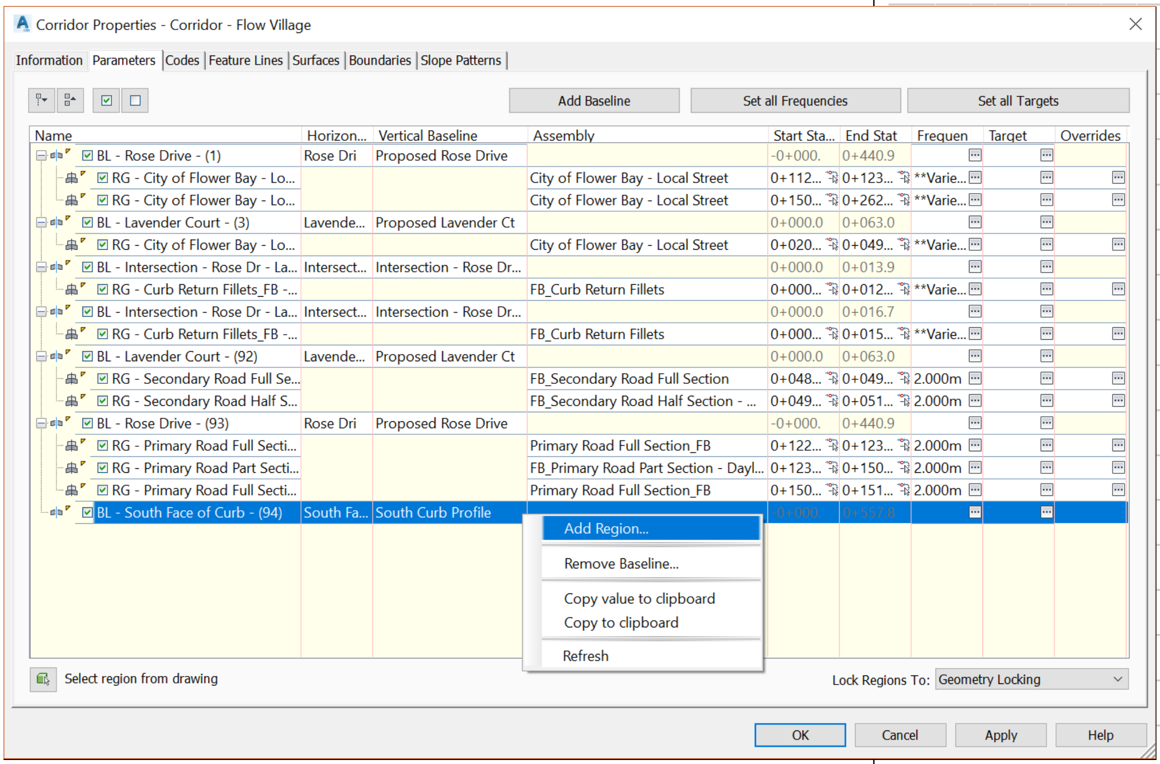


NOTES

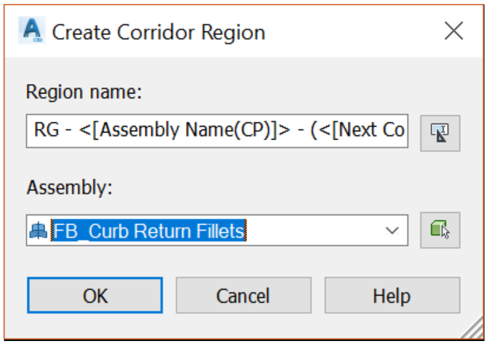
39. In the profile selection window, choose the curb profile we have just created and click **OK**




40. Now, we need to specify, along the baseline, where to add the cul-de-sac portion of the corridor. This is done by creating a **Region** and specifying a start and end station for the **Region**. Select the new baseline, right-click and select **Add Region**.

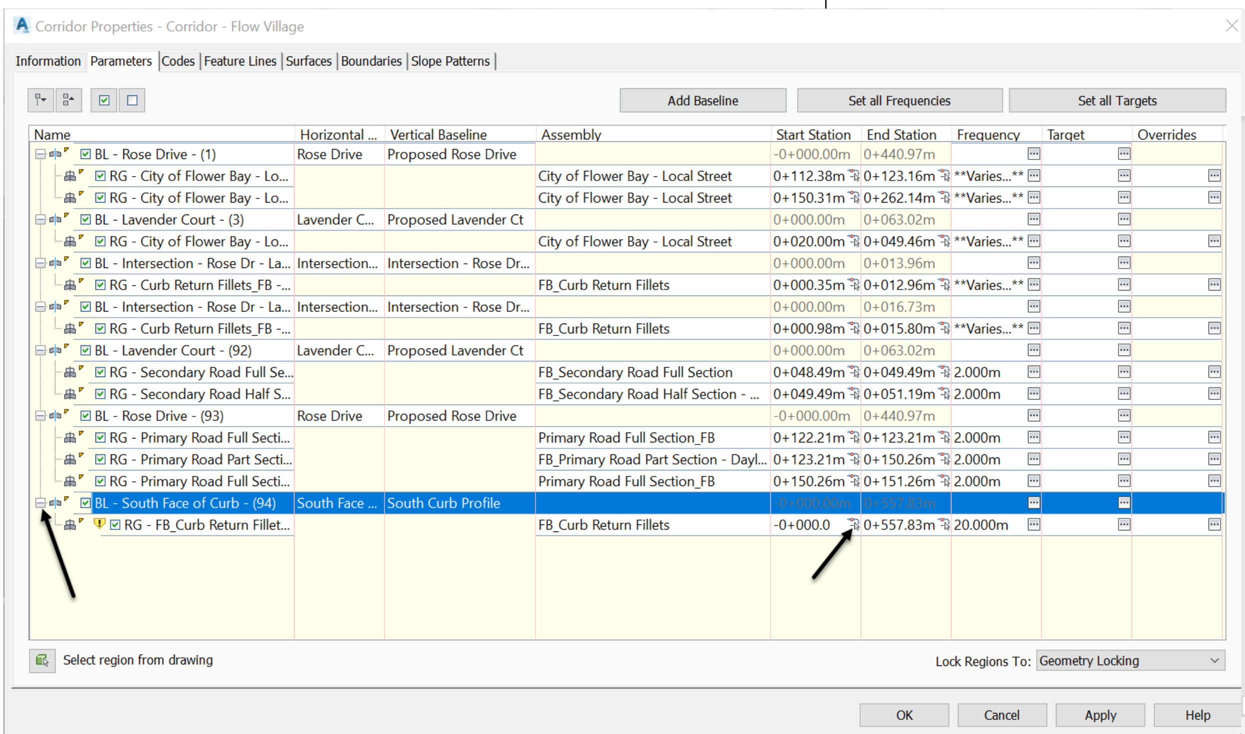



41. In the **Create Corridor Region** window, keep the default Region name. Then, select **Curb Return Fillets -FB** assembly, and click **OK** to close the window.



42. The new region is added. By default, it stretches the entire length of the baseline, which is not representative of what we are trying to do. So, we need to specify a start and end station.

- Click on the "**+**" sign to the left of the baseline name,
- then click on the small arrow icon to the right of the start station.
- This will take you to the drawing area to click on the point you would like to use as a start point. Of course, in this case, it is going to be **DG-CDS1**. You may need to have the **Node** mode object snap activated.




43. Once in the drawing, click on **DG-CDS1**. This will automatically pick up the corresponding station.

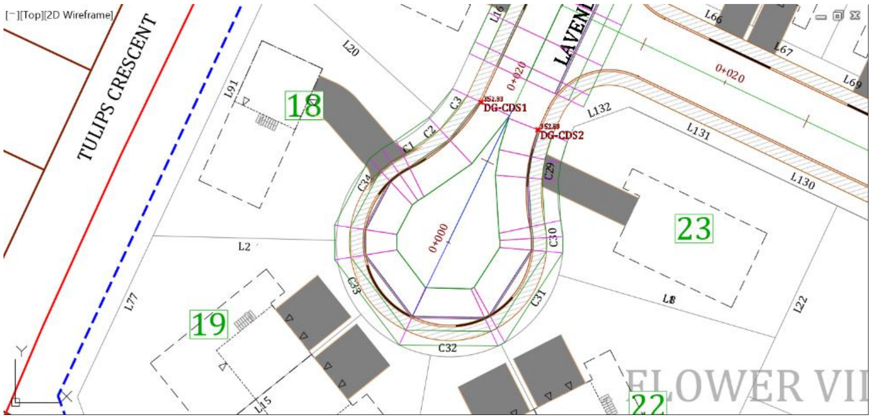
44. Repeat the previous steps for the **End Station** and this time click on **DG-CDS2.**

45. We have now both the start and end station specified, along with a profile and an assembly. This should give us everything we need to create a corridor region. So, let's click **OK** to close this window and check it out.

NOTES

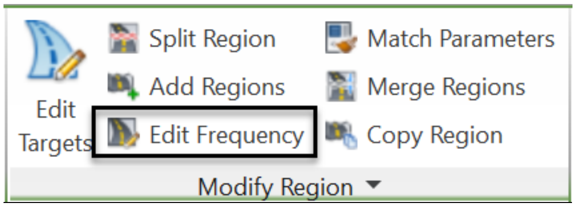
NOTES

46. Now, it looks like we have created a cul-de-sac.

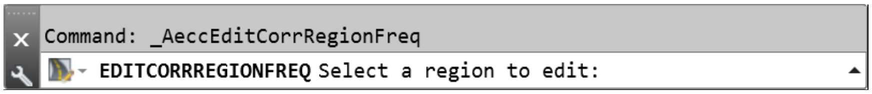



47. However, it just looks like we can do better than this. The cul-de-sac bulb is not filled, and the lines look like something is off. Well, that's because we have not inserted enough cross-sections and haven't set a target to fill out the void in the middle.

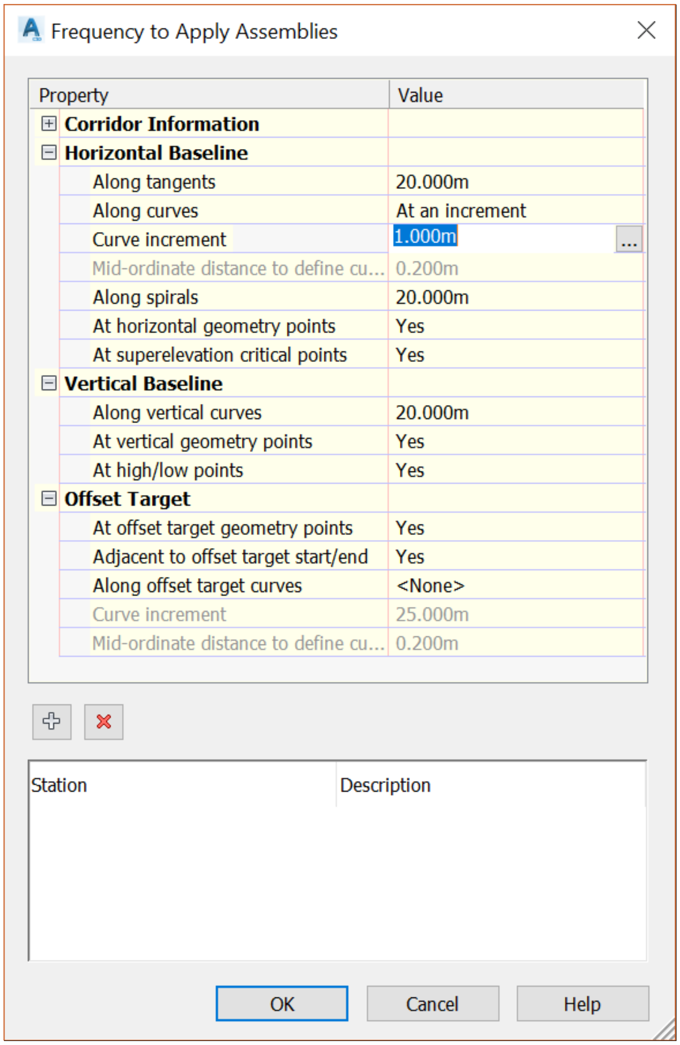
48. First, let's insert more sections. This is accomplished by using tighter frequencies. Select the corridor, on the ribbon, and run the **Edit Frequency** command.



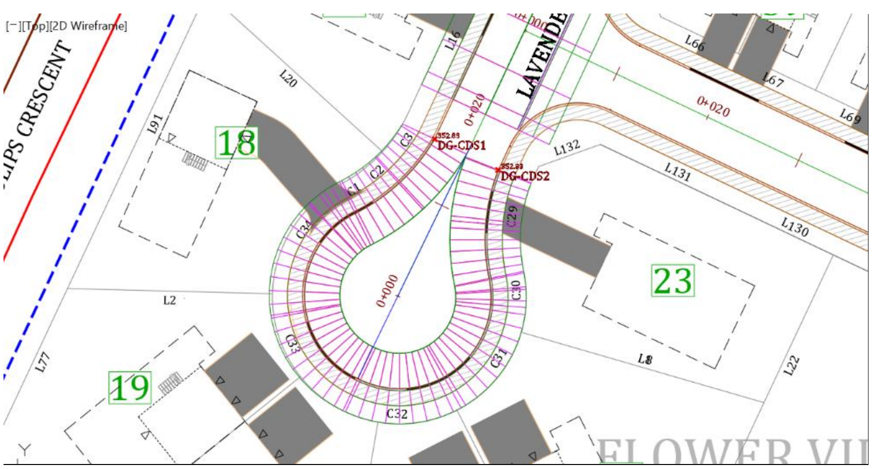

49. When prompted at the command line to specify a region to edit, click inside the corridor, around the cul-de-sac area.



50. In the next window, set the curve increment to **0.5m** or **2ft**. We only have to worry about the curves, since there is no tangent in the cul-de-sac region.




51. Click **OK** to exit the frequency window.

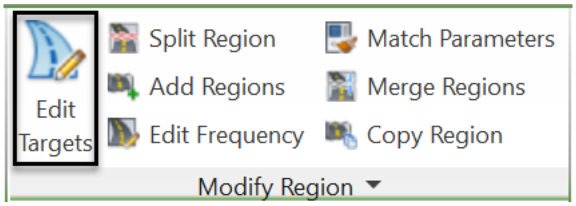



52. Applying a tighter frequency makes the corridor look much smoother.

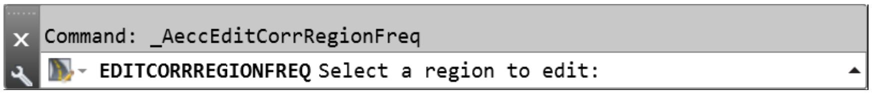
NOTES

NOTES

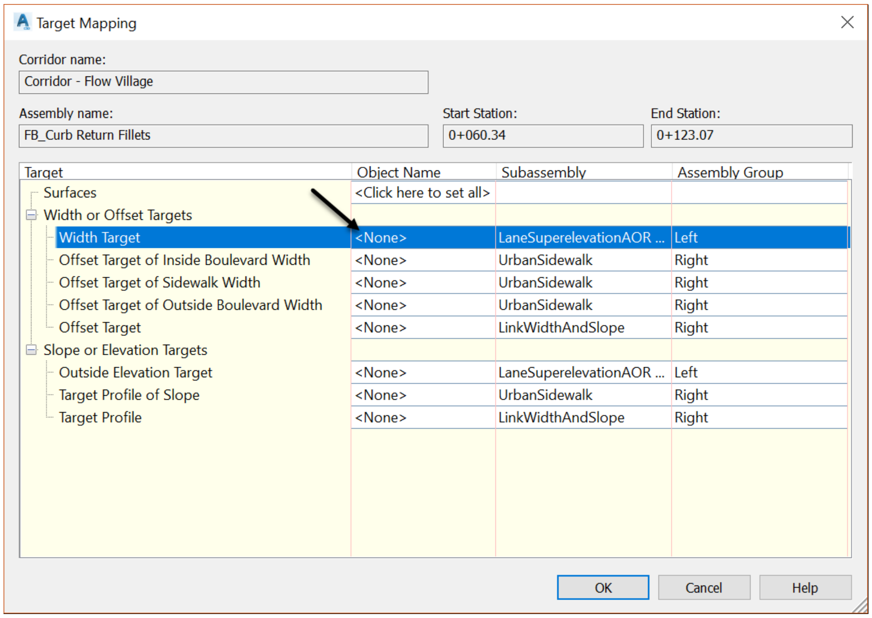
53. Now, unless we are required to create an island in the center of the cul-de-sac, we must pave the void in the middle. To do that, we need to use targets and extend the asphalt to the center of the pavement. Select the corridor, on the ribbon run the **Edit Targets** commands.




54. When prompted at the command line to specify a region to edit, click inside the corridor, around the cul-de-sac area.

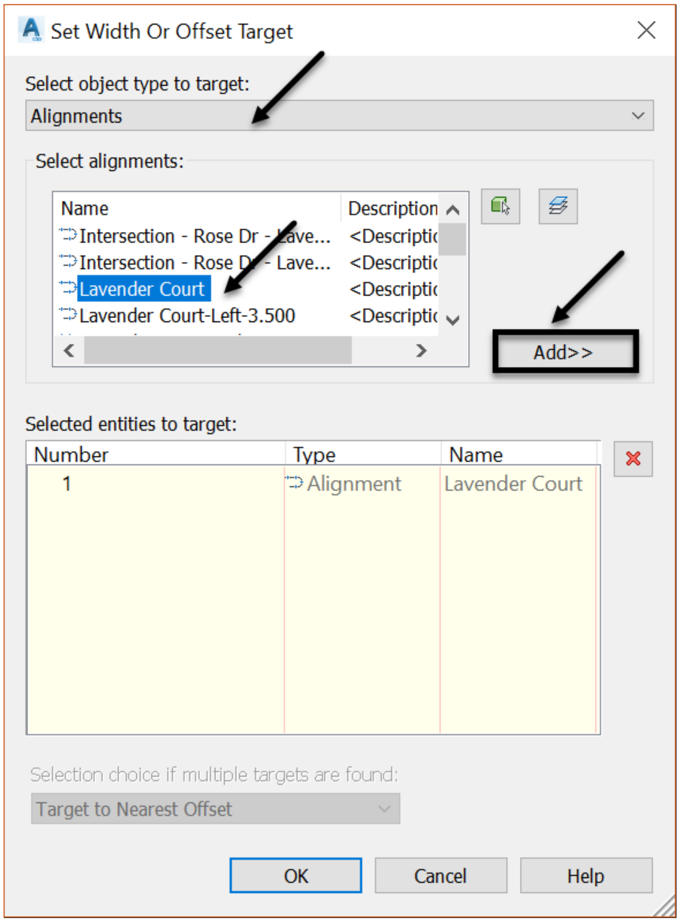



55. As we've seen before, we can assign three types of targets to a corridor: a **surface**, an **offset or width,** and a **slope or elevation**. To extend the cul-de-sac pavement, we need to use the **width** and **elevation** target types to extend to the centerline alignment and profile. So, in the new window, on the **Width Target** line, in the **Object Name** column, click on **<None>** to specify the width to target.

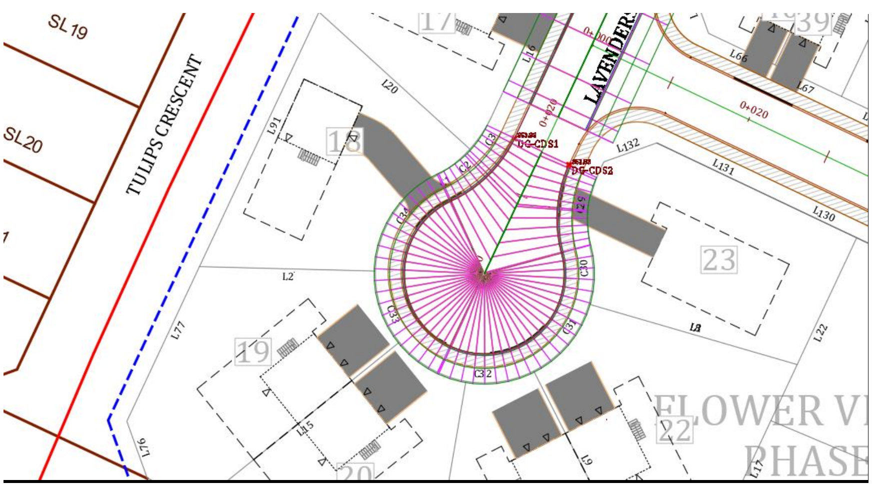


NOTES

56. Then in the **Set Width or Offset Target** window, specify the alignment to target. In here, we can target alignments or other objects such as polylines or feature lines. In this case, choose the **Lavender Court** alignment.




57. Click **OK** twice to preview the changes made by targeting the center line.

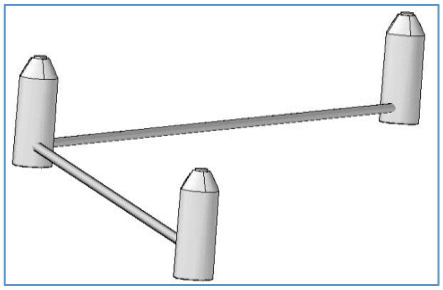



58. Next, we can add the **Lavender Court** road centerline profile as a second target. But, if the cul-de-sac is perfectly symmetrical, there will be no need to do that. The uniform **2%** slope will be carried throughout the cul-de-sac, from the edge of pavement to the centerline.

# 10 PIPE NETWORKS

## 10.1 Introduction

The land development engineer usually designs the superficial infrastructure. However, he or she may also be required to design the underground wet system, especially the wet utilities like watermains, sanitary and stormwater sewers. As usual, the design should also be done in accordance with specific municipal by-laws. An example is illustrated by the municipality of **Flower Bay**'s typical street cross-section.



Civil 3D has a set of tools called **Pipe Network Creation Tools.** They enable us to design these utilities and meet the requirements, including maximum pipe lengths, minimum covers, minimum and maximum slopes, and flow velocities. Each flow type system, in Civil 3D, is categorized under a separate **Pipe Network**, with interlocking entities. A pipe network is composed of two main components: **Pipes** and **Structures.**

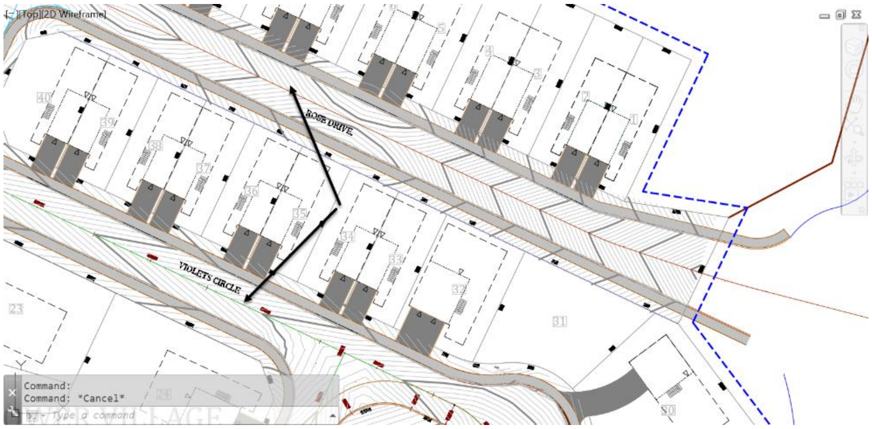
**Pipes** are the linear parts in the network and are made up of different type of materials (Concrete, PVC, HDPE, metal, etc.). Two of their main characteristics are the diameter and slope, which combine to determine the amount of flow a pipe can carry. That amount is typically estimated in terms of rate, such as **cubic meters** or **cubic feet per second**, rather than volume.


**Structures** are the elements that connect the pipes. Due to the maximum allowable length of pipes, **Structures** must be installed at regular intervals. We also need structures at the change of slopes, change of directions or when water is captured from the street surface.

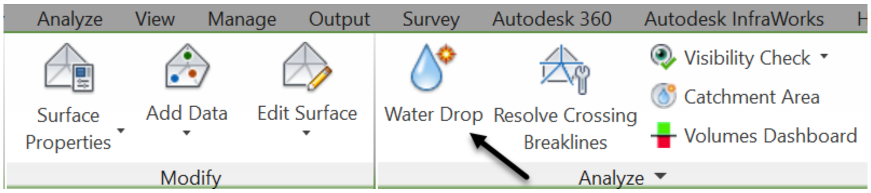
## 10.2 Laying out a Storm Sewer network.

Pipe networks can be created using existing Civil 3d objects like 3D polylines and feature lines. We can also design them by layout using the Pipe Network Creation Tools. First, let's create the storm sewer network with the by object method.

1. Open the **10.01-Pipes** dwg file in **Lesson 10** practice folder.
2. In previous lessons, we have already designed a proposed surface for Roses Drive, using a proposed profile. The profiles will also allow us to determine low and high point locations. As a requirement of the stormwater system design, we need to make sure that we put catch basin inlets at every low point, on the street. We can thus collect rainfall runoffs. To identify these points, we can use the profile and create low point labels on the street alignment. But an easier and more visual way is to use the Water Drop tool we talked about in the lesson on surfaces. Let's run the Water Drop command by selecting our design surface. It's worth mentioning that the design surface is still temporary as we haven't done any grading at the lot level yet. However, we are done with the street portion of the surface. You can see the finalized corridor by turning on the C-Road-Corridor layer, from the layer manager. Look at it and turn the C-Road-Corridor layer back off.
3. Let's check out the locations of these low points on the plan. Each time we locate a low point, we will create a circle to mark the location temporarily. In the drawing, select the corridor surface.




4. On the ribbon, run the Water Drop command.

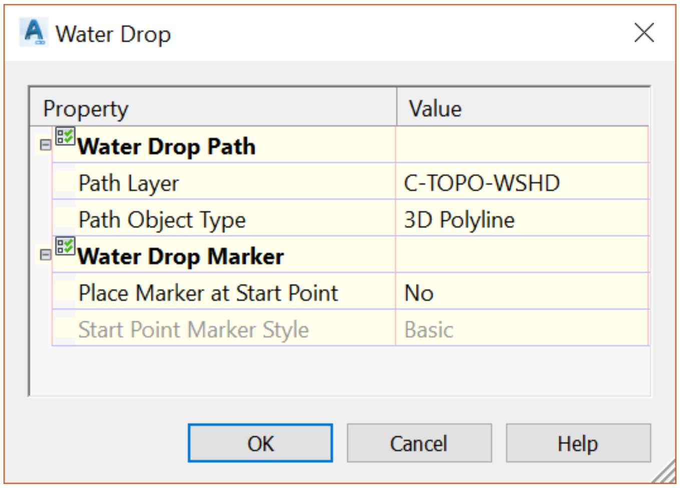




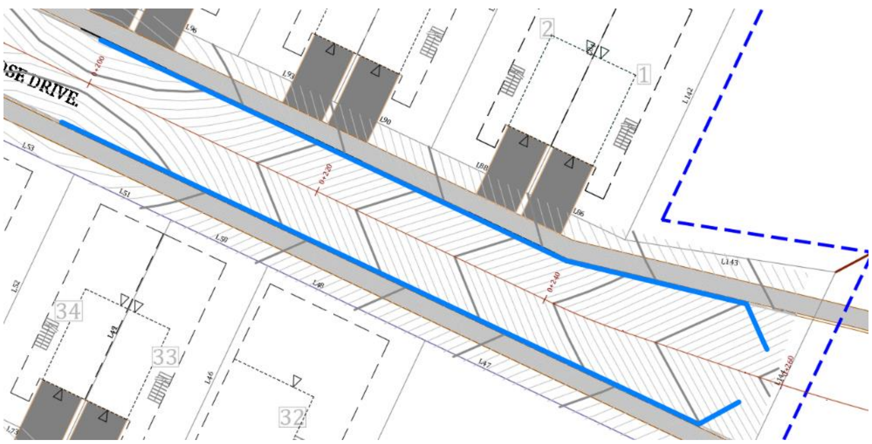

NOTES

NOTES

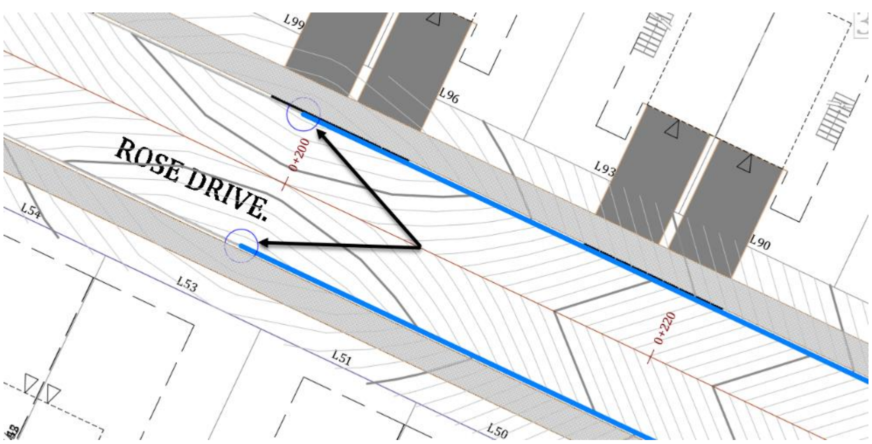
5. In the next window, assign a layer, probably the C-TOPO-WSHD watershed layer and a Path Object Type. Then click on OK.




6. Next, click on the eastern end of Rose Drive, on both sides of the centerline, around station 0+260m or 0+850'. You will notice a flow line running along the curb line on both sides of the road. These represent the path that a water drop would take along the street all the way to the lowest point in the area.

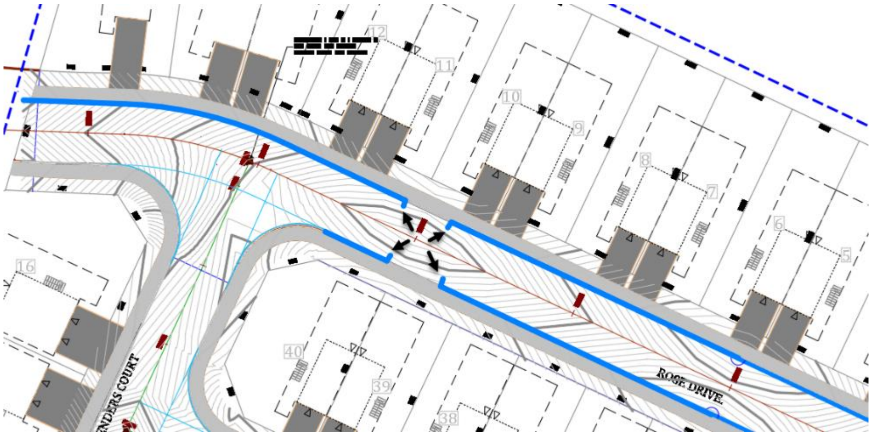



7. To add a low point label, Create two circles at the ends of the flow paths.



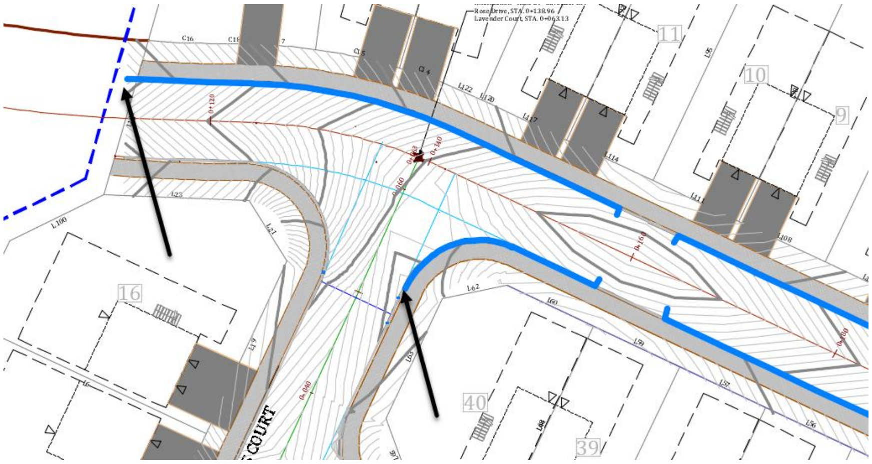
NOTES

8. Next, create a water drop flow path on the other side of the street. Looking at the contour lines, there is a high point around station 0+160m or 0+525ft. Let's click on both sides of the high point and both sides of the street centerline.



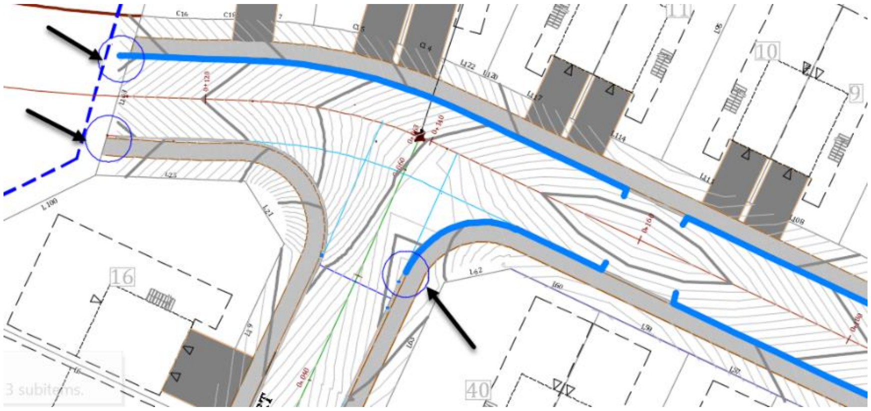

9. By examining the flow paths from the last couple of points, we can conclude that:

    - On the north side of **Rose Drive** centerline, water is flowing toward the western end of the site. Therefore, a catchbasin needs to be installed at that station on the north and south side of the street centerline, exactly at the site boundary. On the south side of **Rose Drive**, the flow path is stopping approximately at the start of the southeast quadrant of the intersection.

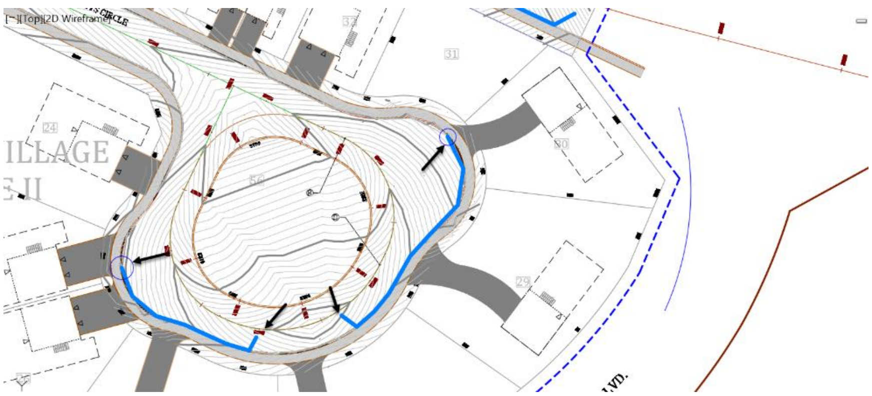


NOTES

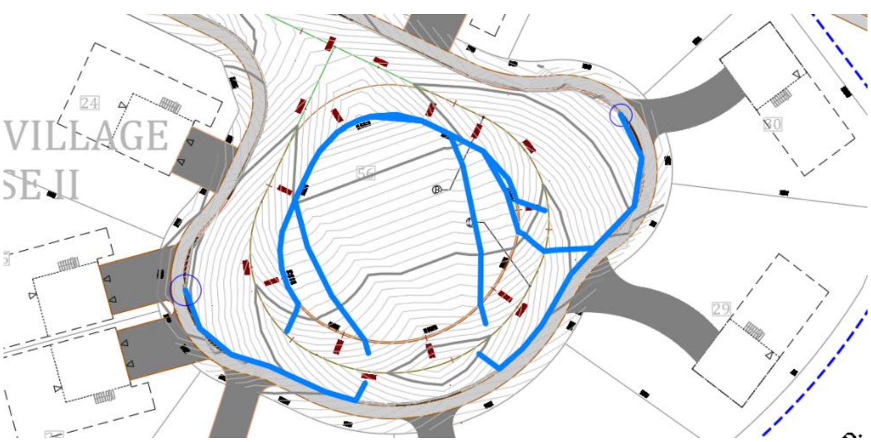
10. Now, create the three circles at the low points for future catchbasins.




11. Next, repeat the same process for **Lavender Court** and **Violets Circle**, starting at the end of the cul-de-sacs. For example, for **Violets Circle**, let's put a couple of water drops on each side of the high points around station **0+132m** or **0+430ft.** You will notice the two low points in the cul-de-sac.

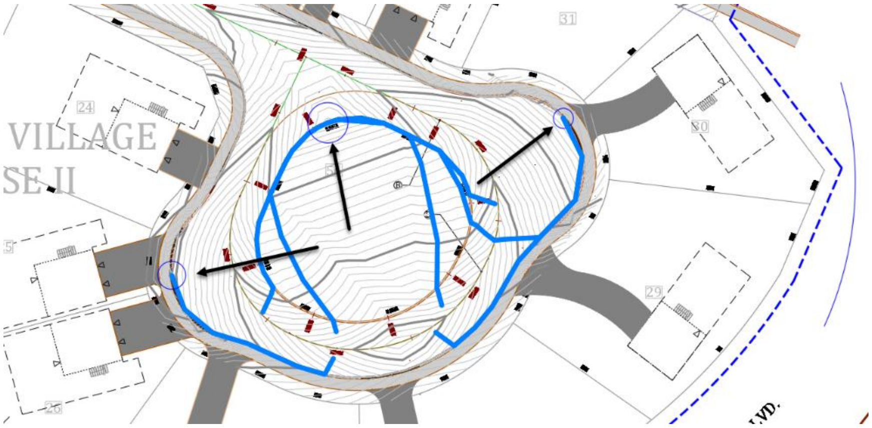



12. If you create a drop on the north side of these two locations, you will notice a flow path going through the island, to the low point on the other side. That's because we have not refined the elevations enough for the curbs to have enough effect and curb the flow. However, the grading is looking good for now, as we can clearly identify the low points.

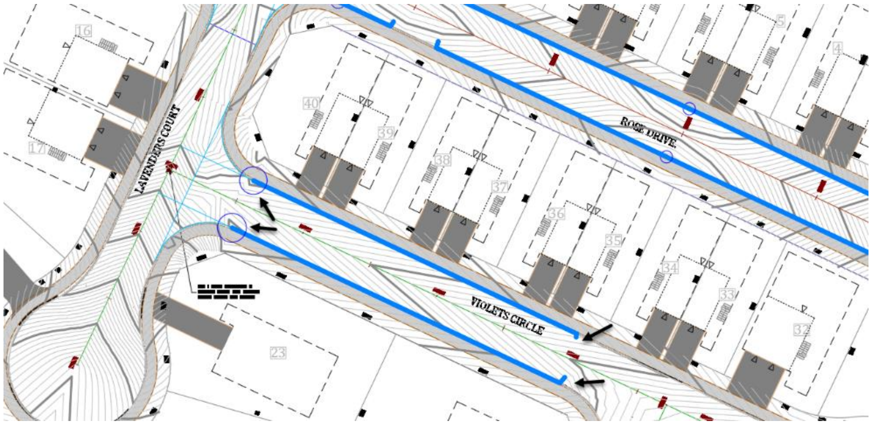


NOTES

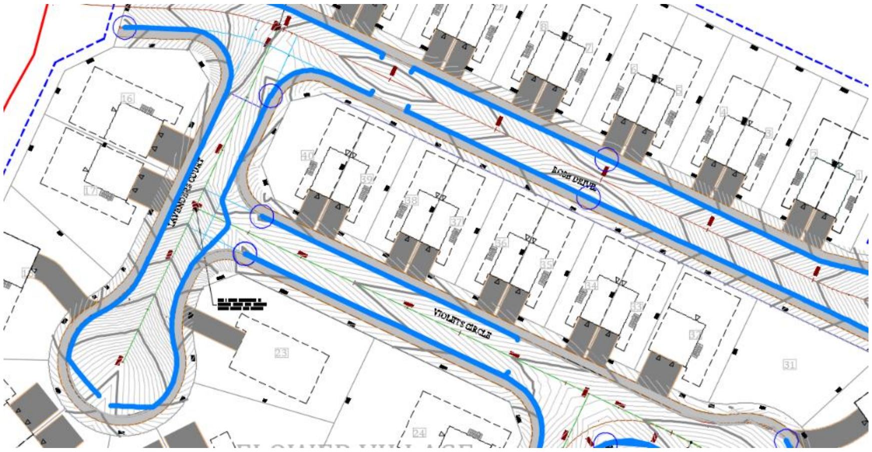
13. Now, create the third circle in the cul-de-sac for the grading.



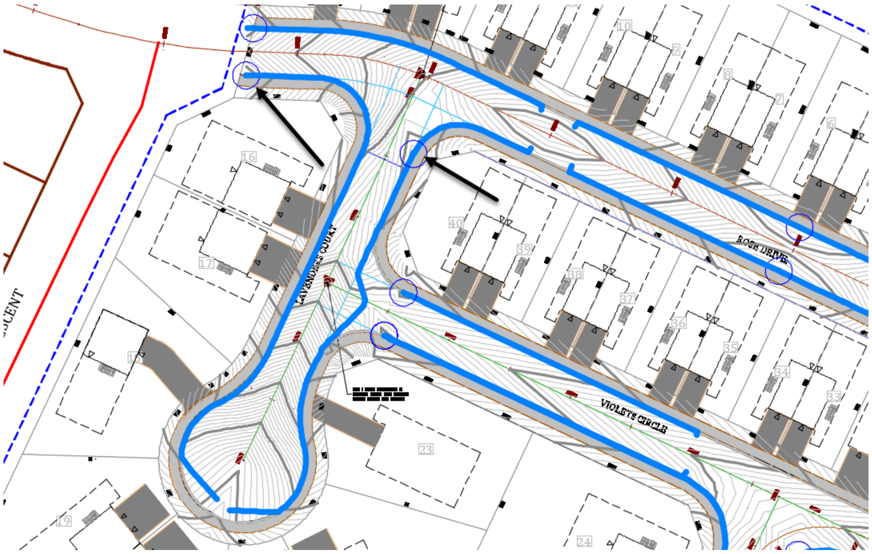

14. Next, create the **Water Drops**, on the linear section of **Violets Circle**. The low points are located all the way to the start of the curve at the intersection with **Lavender Court**. These will be two ideal locations to install catchbasins and catch all the flow coming from up top.



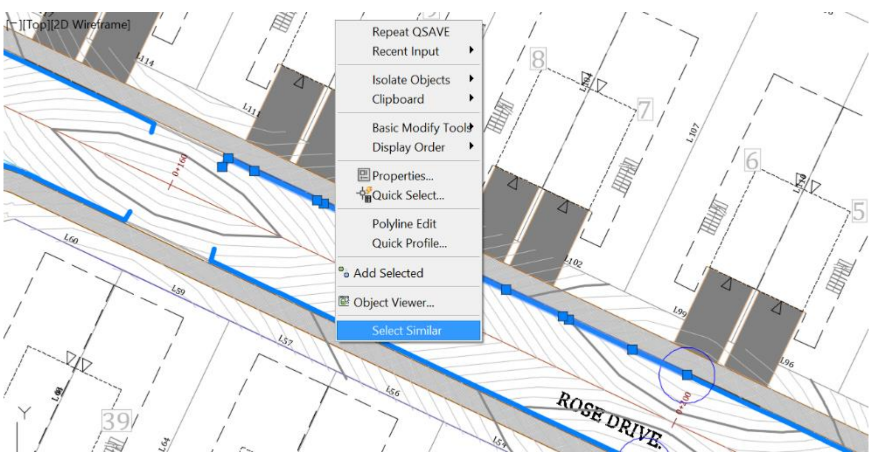

15. Finally, let's create the water drops in the cul-de-sac area of the **Lavender Court**. Create two water drops on each side of the high point located at the top of the cul-de-sac.



16. You'll notice that the rainfall runs off from the cul-de-sac all the way to the last manhole on the west of **Rose Drive,** on one side. And on the other side, to a low point, at the start of the curb return in the southeast quadrant of the **Rose Drive – Lavender Court** intersection.



17. This process gives us a preliminary idea of how to lay out our storm sewer system. Before moving on to the next thing, let's clean up the clutter we have created with the water drop lines. **Select** one of the lines, **right-click**, **select similar** and **delete** from the keyboard. The select similar allows selecting entities sharing similar properties. We facilitated their selection by creating the water drop lines using 3D polylines and putting them on the watershed layer. Few other objects have these properties. On the other hand, we will not delete the circles, as we still need them to place our catchbasins.

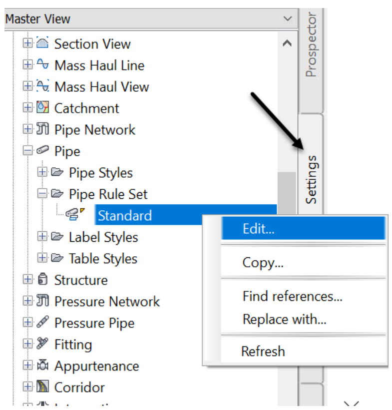



NOTES

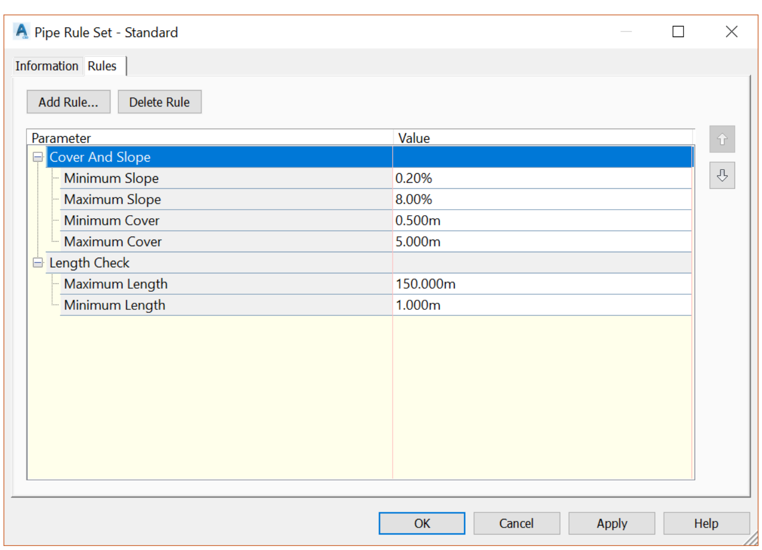
## 10.3 Pipe Rules


We can now start laying out the system and adjust as needed. Before we do that, we have one more "table set up" item we need to take care of: the pipe network **Pipe rules**. Pipe rules allow us to establish some basic criteria that must always be met during the layout of the pipe network. Examples of pipe rules can include the minimum or maximum pipe slope, minimum cover, minimum or maximum pipe length, etc.

1. From the **Settings** tab of the **Toolspace**, browse to the Pipe category, then open the **Pipe Rule Set** item. We currently only have one rule set, **Standard**. Let's use it and customize it to our needs. Select it, right-click and **Edit**.




2. In the **Pipe Rule Set** window, enter the values for **Minimum and Maximum Slopes**, **Minimum and Maximum Cover**, **Minimum** and **Maximum length.** Note that you can make the maximum slope a little higher if you are dealing with very steep grounds. Before clicking **OK** to close the **Pipe Rule Set** window, notice that you can also create or delete rules here.



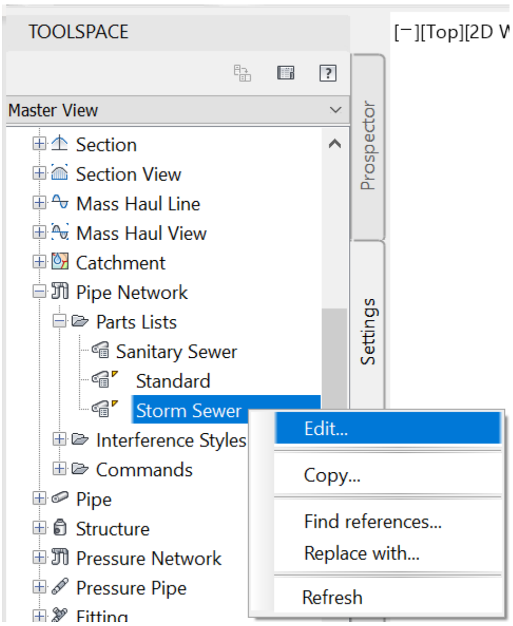

NOTES

## 10.4 Pipe Parts List

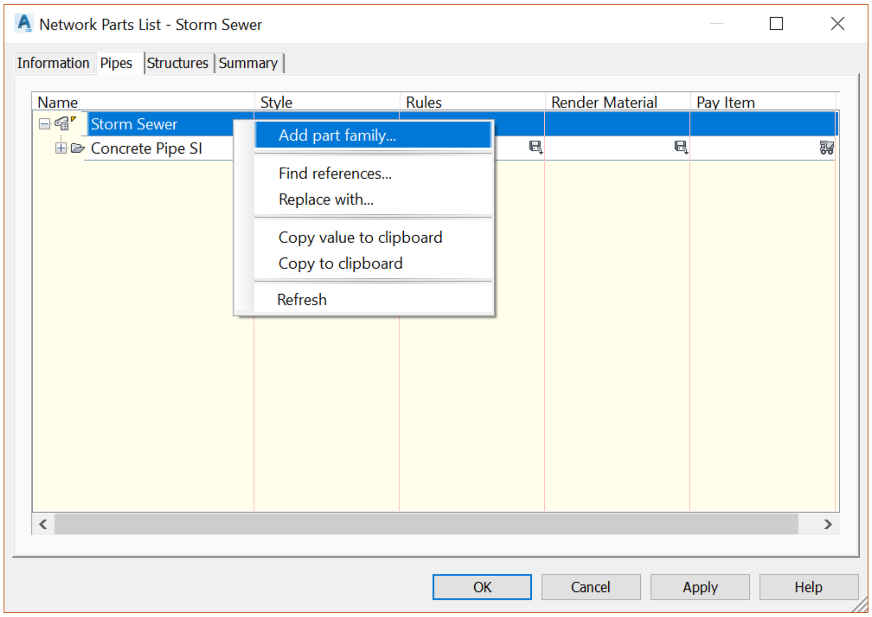
The final "table setup item" is to set the **Parts List** for the network. For example, we need to decide the different pipe or manhole materials, shapes and diameters that we will use in the project.

To specify a **Part List**,

1. Switch to the **Settings** tab of the prospector.
2. Browse to the **Pipe Network** section.
3. Select **Storm Sewer**, **right-click** and choose **Edit**. Here, we can create, delete or modify existing part lists.

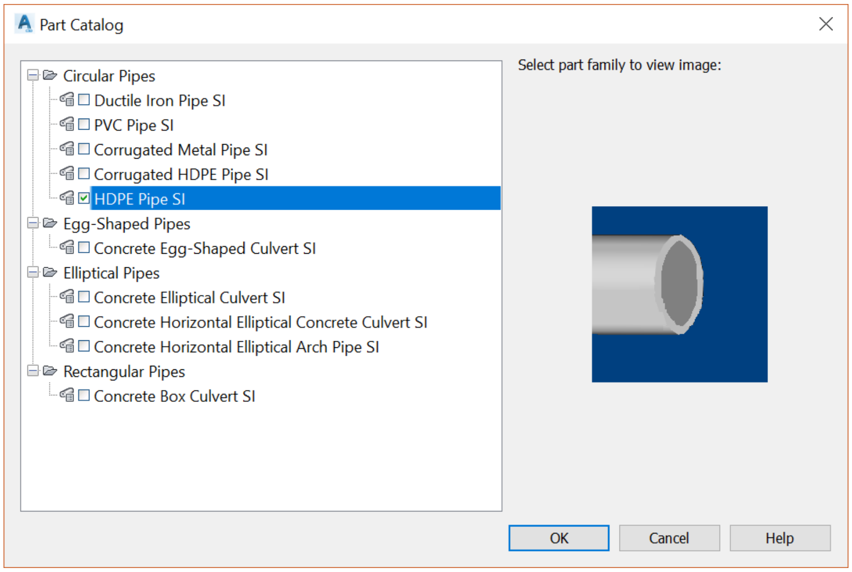



4. Activate the Pipes tab. Collapse the **Concrete Pipe** list. In this project, we will be using **HDPE** pipes. So right-click on **Storm Sewer** and **Add part family**.

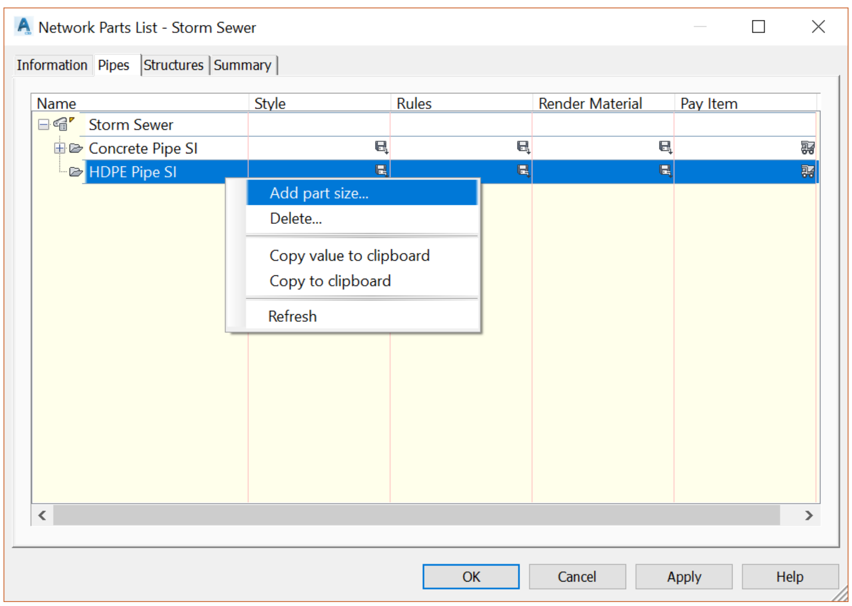



NOTES

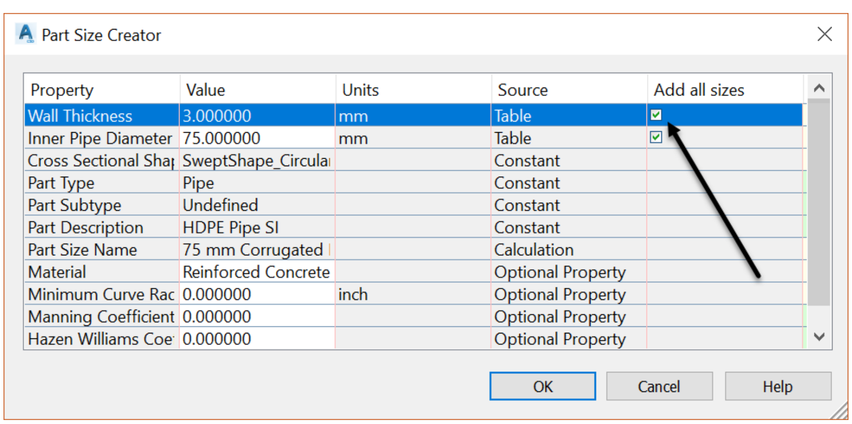
5. In the **Part Catalog**, check **HDPE Pipe** and click **OK.**




6. Now, select the new family we have just added and **right-click** and **add part size.**




7. To simplify things, even though we will not use all of them, let's go ahead and **add all sizes.**



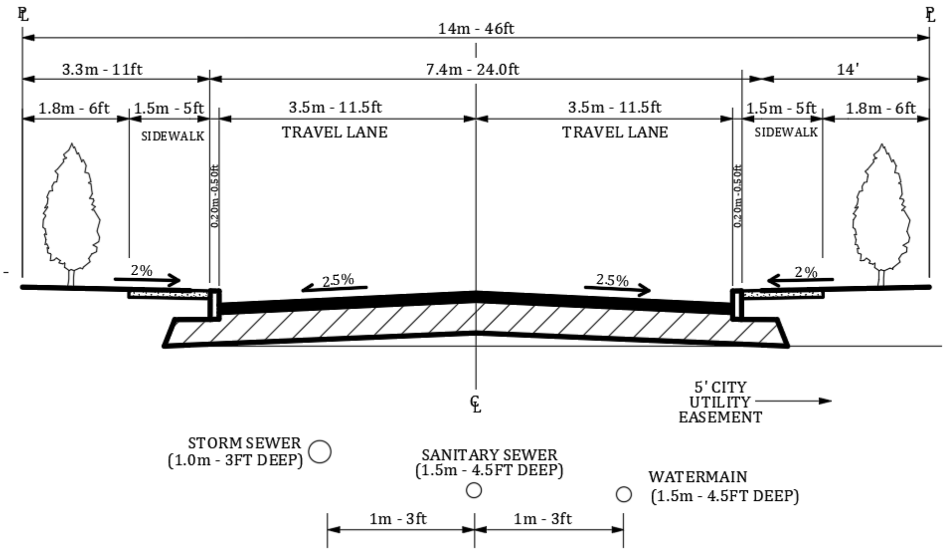

NOTES

8. We can do similar operations on the **Structures** tab by adding different sort of structures and sizes. For now, click on **OK** and close the window.

NOTES

## 10.5 Creating a network by objects

Now is a good time to refresh ourselves with the requirements from the municipal code regarding utility layout.

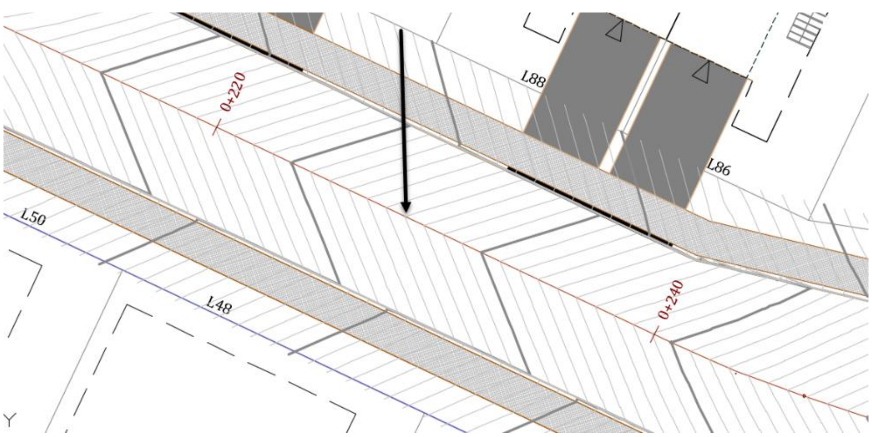



Looking at the typical section, storm sewer pipes need to be at a **1m** or **3ft** depth offset from the centerline, with a minimum of **1m** or **3ft** cover.

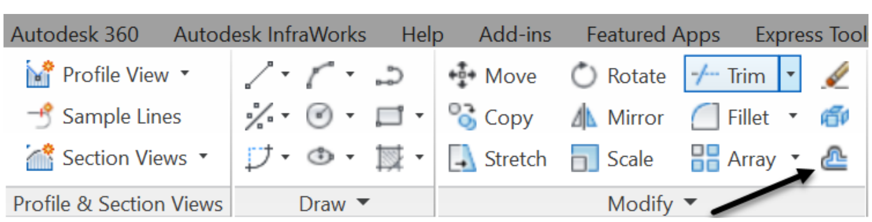
**Pipe networks** can be created **By Object** or by using the **Pipe Network Creation Tools**.

Let's create the network using the object-based method first. To do that,

1. Select the centerline of the **Rose Drive** alignment.

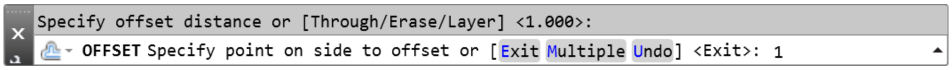



2. Offset it **1m** or **3ft** to the north by using the AutoCAD offset command or typing the alias **O** at the command line.

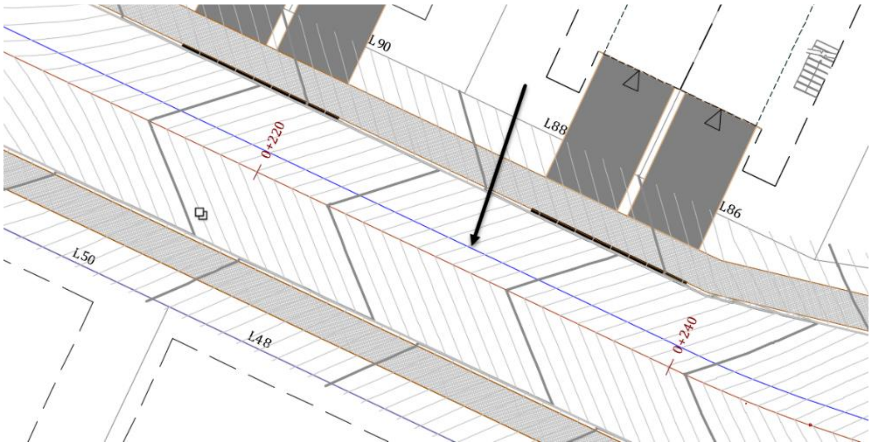



NOTES

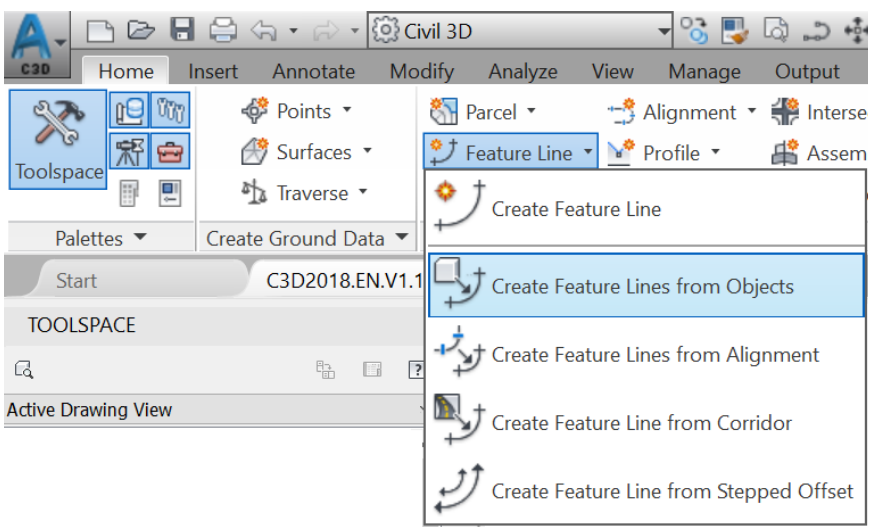
3. When prompted to **Specify Offset distance** at the command line, enter **1m** or **3ft** and press **Enter**.




4. When prompted to specify a side, click north of **Rose Drive** Alignment.




5. The new line will represent the centerline of the stormwater. Therefore, we need to assign elevations to the line, so that we can set the pipe invert elevations. For that, we need to convert the polyline to a more powerful type of line: **Feature lines.** They are predominantly grading entities that can interact with other objects. In this case, we use them due to the ease of use presented by their visual editor and geometry editing tools. To create a **Feature line** from the offset centerline, run the **Create Feature Lines from Objects** command from the ribbon.



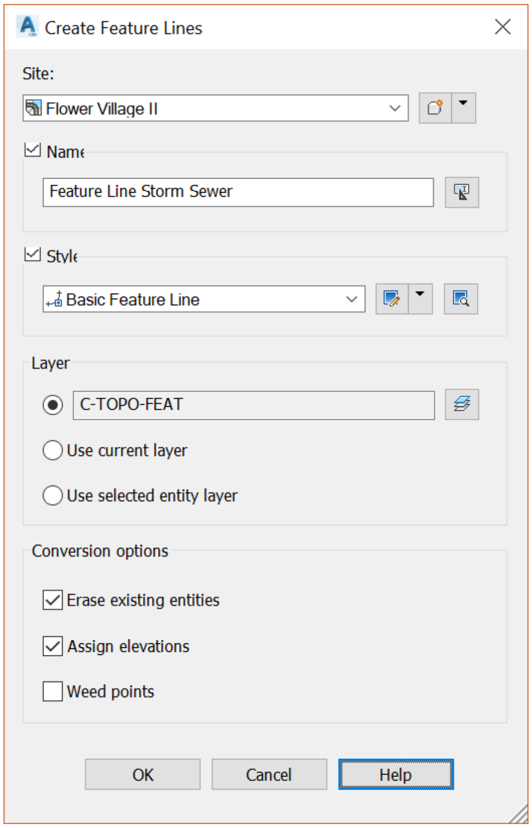

6. Select the polyline created by offset and press **Enter** at the command line.
7. In the **Create Feature Lines** window,

   - First, we need to assign a **Site**. We talked about sites before, they are topological containers where we can put

NOTES

entities that can have an interactive relationship. In this project, we only have one site, so our feature line will reside in the **Flower Village II** site.

- Next, name the feature line and select a style, say **Basic Feature Line** style.
- We will leave the layer to the default one.
- Last for the **Conversion Options**; we will **erase existing entities** and **assign elevation**. However, there is no need to **weed points** on the polyline. The polyline has a fairly reasonable number of points and is on an almost straight path.
- Click **OK**.



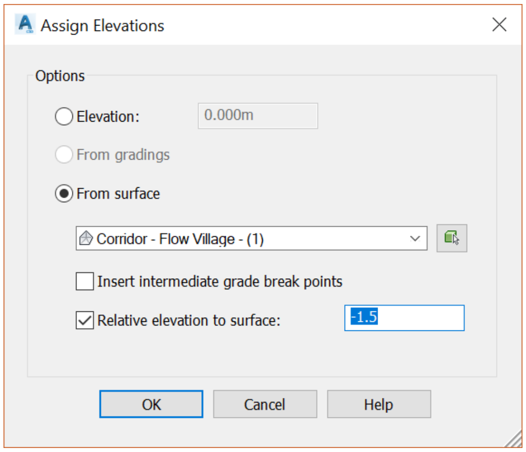

8. In the **Assign Elevations** window,

- Assign elevation from the **corridor surface,**
- Set a relative elevation of **-1.5m** or **-5ft** for the pipe inverts. We are required to have a minimum cover of **1m** or **3ft**, but we still need to account for the pipe diameter. So, we are adding an additional **0.5m** or **2ft**. If we need to add or reduce the pipe cover, we can always raise or lower the feature line by selecting it and using the **raise/lower** command. At this stage, we still don't know the final sizes of the pipes. This will be established in the stormwater modeling phase of the design. Please refer to the **Advanced Civil 3D – Stormwater design** module of our courses.

NOTES

NOTES

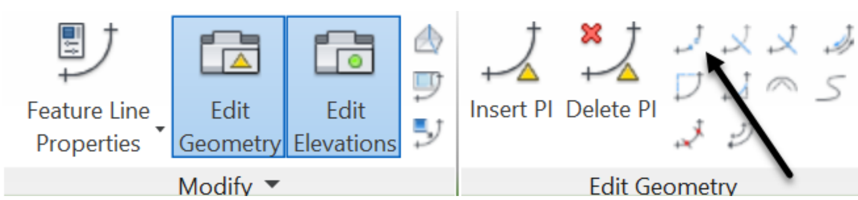
- Now, click on **OK**.



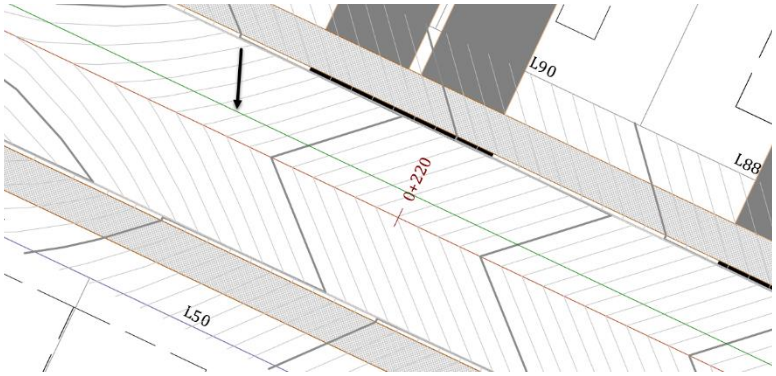

9. The polyline is now converted to a **Feature Line.** Once it is selected, the contextual ribbon displays all feature lines related commands.


10. Let's use a few of the **Feature Line** editing tools.

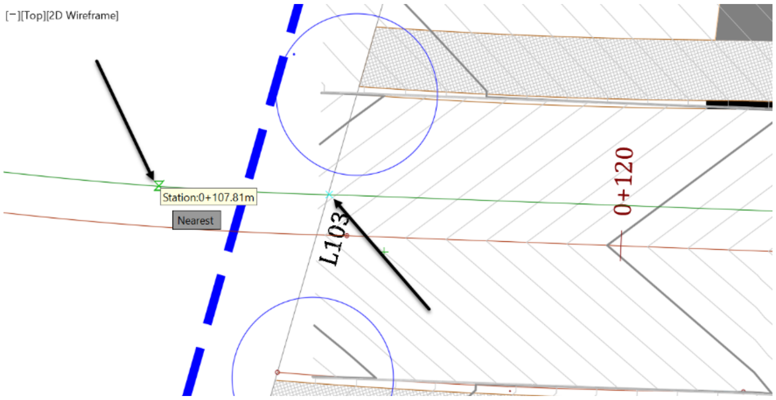
  - First, **Break** the feature line to contain it inside the project. It's worth mentioning that regular AutoCAD commands such as **break** and **trim** will not work on feature lines. They have their own set of commands. We can run them from the **Edit Geometry** or **Edit Elevations** panels of the contextual feature line ribbon. So, while you have the feature line selected, click on the **Break** command in the **Edit Geometry** panel.



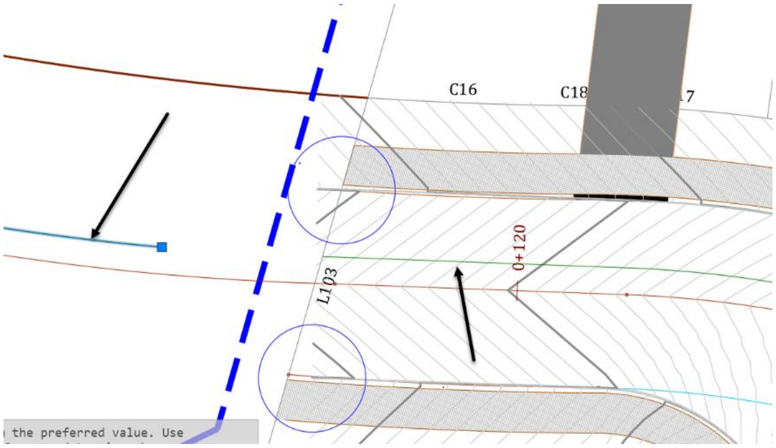
- When prompted to **Select an object to break** at the command line, click on the **Feature Line**, at the eastern or western end of the site boundary.




- When prompted to specify the second breakpoint, click outside the site area on the feature line.



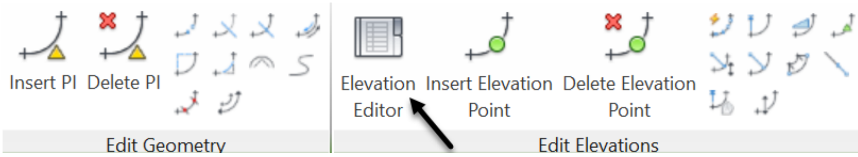

- The feature line is now broken in two. We can simply delete the part that falls outside of the site.



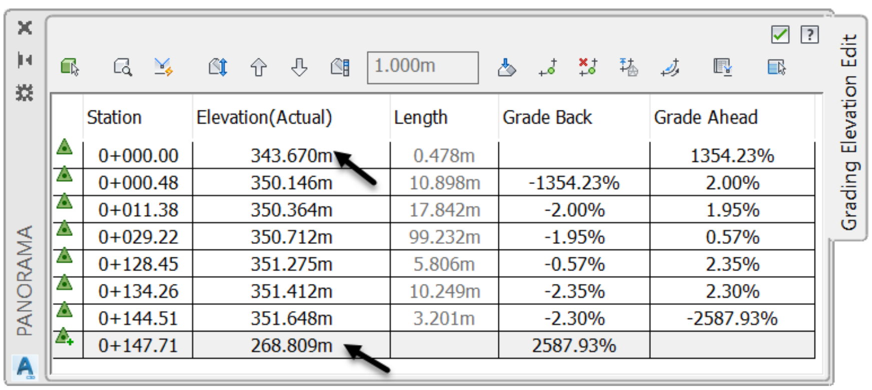

- Now repeat the same operations, break and delete, on the other side of the project.

NOTES

11. Because we are never too sure, let's check the elevations of the **Feature Line** before using it to create a pipe network. Select the feature line and in the **Edit Elevations** panel, display the **Elevation Editor**. You can also display it by selecting the feature line, right-clicking, and choosing **Elevation Editor**.




12. Once the **Elevation Editor** is displayed, in the **panorama** window, you have all the vertices of the feature line in a tabular view. You can also see their associated stations, elevations, distances and grades between points. All data in this window, except the lengths, can be manually adjusted. Depending on where you broke the feature line you may need to adjust the elevation of the points at the extremities. If the surface falls out of the **Corridor Surface** we assigned our elevations from; their elevations are not going to be correct.



| Station | Elevation(Actual) | Length | Grade Back | Grade Ahead |
|---|---|---|---|---|
| 0+000.00 | 343.670m | 0.478m | | 1354.23% |
| 0+000.48 | 350.146m | 10.898m | -1354.23% | 2.00% |
| 0+011.38 | 350.364m | 17.842m | -2.00% | 1.95% |
| 0+029.22 | 350.712m | 99.232m | -1.95% | 0.57% |
| 0+128.45 | 351.275m | 5.806m | -0.57% | 2.35% |
| 0+134.26 | 351.412m | 10.249m | -2.35% | 2.30% |
| 0+144.51 | 351.648m | 3.201m | -2.30% | -2587.93% |
| 0+147.71 | 268.809m | | 2587.93% | |



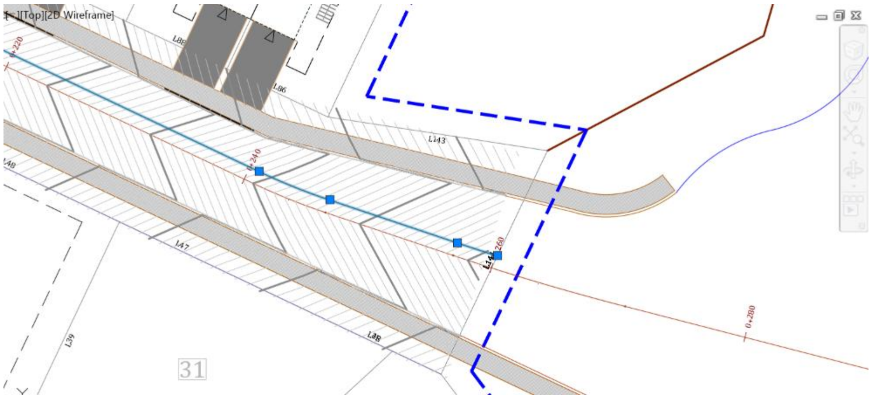
13. In this case, to fix the elevations at the two extremities of the feature line, we will simply assign a **2% Grade Ahead** and **-2% Grade back**. This allows us to change the elevations only at the extremities.

14. Click on the **green checkmark** to dismiss the Panorama vista. Your changes will apply to the feature line.

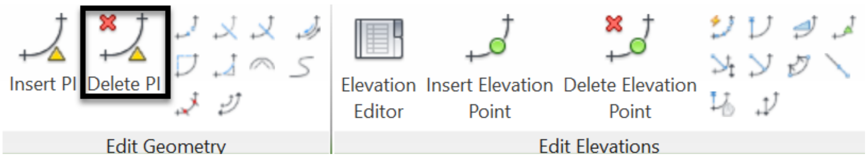
NOTES

NOTES

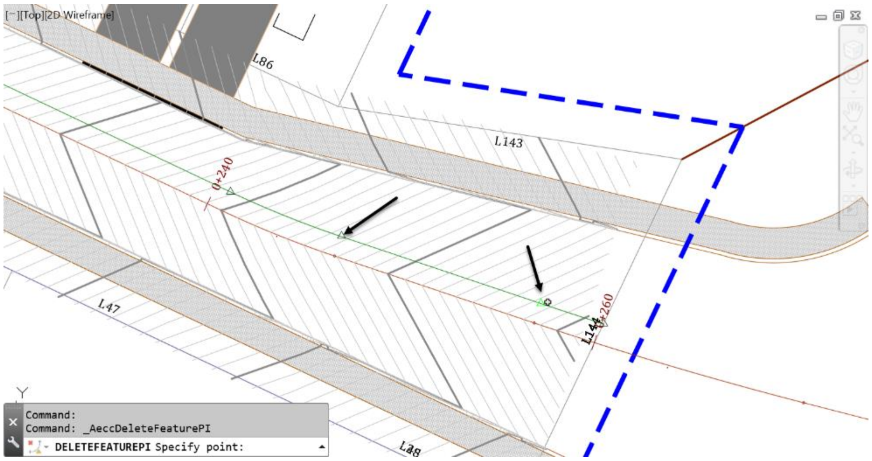
15. Next, we need to delete the unnecessary vertices, because stormwater structures are going to be created at all vertices. Note that we can go ahead and create the network and adjust it later to remove the extra parts. But we have an opportunity here to learn more on the feature line editing tools, and we don't want to miss it. So, select the feature line and zoom to the east end of the site, just before the roundabout.



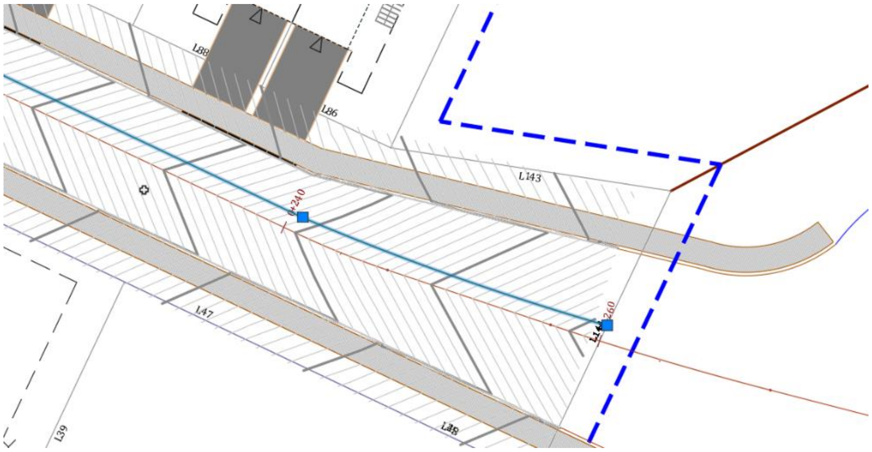
16. We obviously don't need manholes at all the shown vertices. So, delete the two middle vertices. We need to keep the westernmost one to meet the maximum pipe length of **120m** or **400ft.** We also need the easternmost one to allow future connection to our project by others. To delete feature line vertices, first, run the **Delete PI** command on the ribbon.



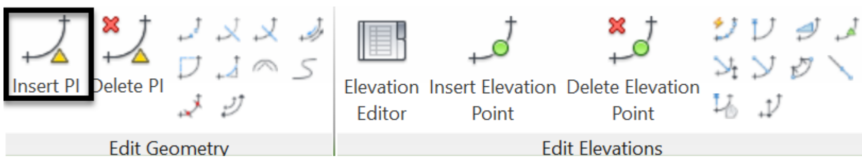

17. When prompted to specify the point to delete, click on the two middle points.



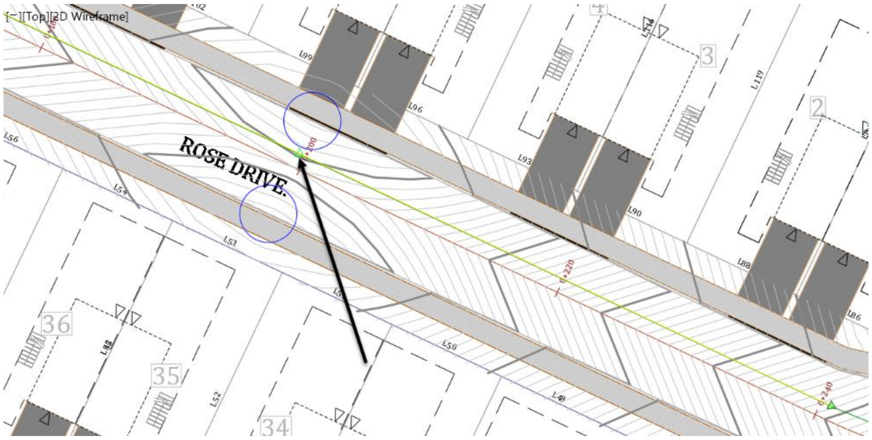
18. We now have two locations for future manholes.



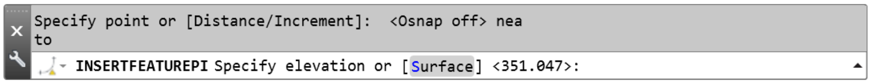
1. Moving westward, we see the two spots we reserved for catchbasins. That means we need a manhole in the vicinity to tie the catchbasins to. So, this time instead of deleting, we need to insert a vertex.
2. Run the **Insert PI** command.




3. Then, click approximately between the two future catchbasins, on the feature line.



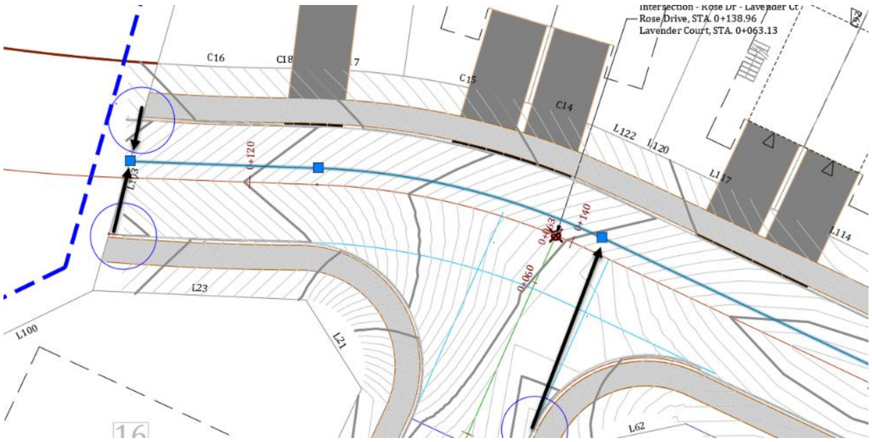

4. The elevation is automatically calculated using the feature line slopes. Press **Enter** to accept it. You can change it if needed, but there is no point in this case since these are just temporary elevations. We just need something reasonable to lay out the pipe network.




NOTES

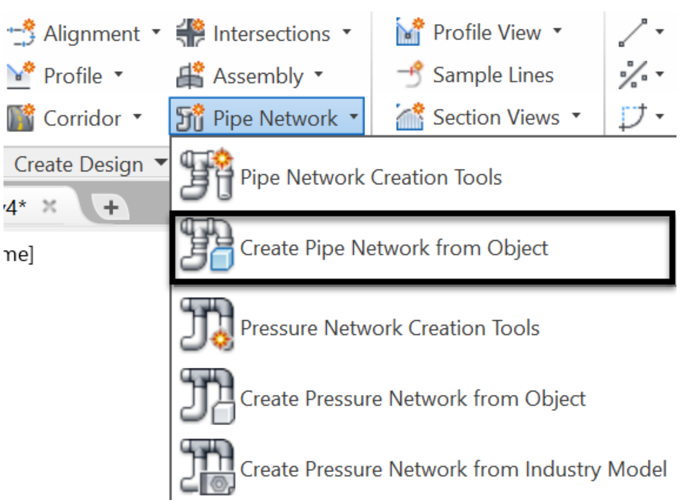
NOTES

5. Next, press **Escape** to exit the **PI** creation command and move down the road to the intersection area. Here, we need to get rid of two more vertices. However, we need the structure at the end of the curve to keep the nice smooth curve. We can have a curved pipe if we are using a flexible pipe that can be bent. If not, we then need to convert that segment of the feature line to a straight line, using the **Edit Curve** command. So, let's only remove the second to last vertex towards the end. This way, we can tie the end catchbasins to the last vertex and the intersection's catchbasin to the first vertex. The remaining vertex can also be a catchbasin, or a null structure. If a catchbasin is not needed there, when we make our final decision, we can remove it, using the **Delete PI**.



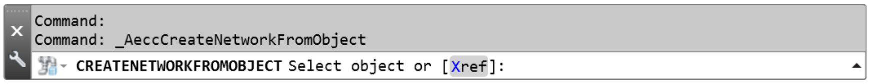

6. For now, our feature line is set up, and we can create the pipe network.

7. From the **Home** tab of the ribbon, run the **Create Pipe Network from Object** command.

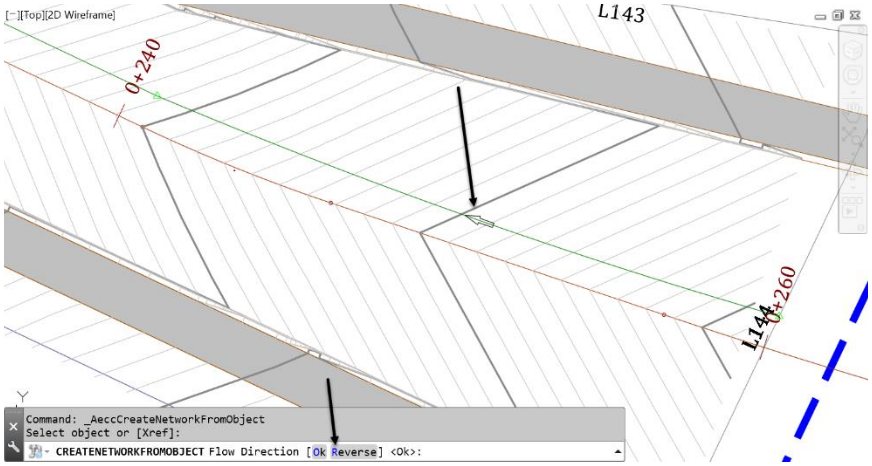


NOTES

8. When prompted to select an object, click on the feature line towards its eastern end.




9. This will show the flow direction from east to west as we needed. If you have the flow in the opposite direction, you can always type **R** at the command line to **Reverse** it.

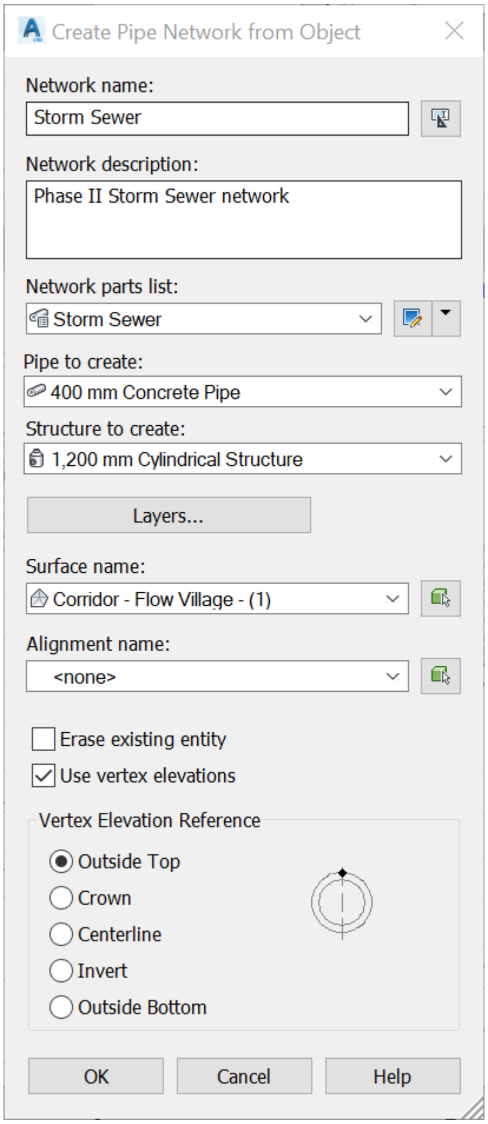



10. Press **Enter** to accept the flow direction.

11. Now, we need to enter the network details in the **Create Pipe Network from Object window**.

- For the name, we will go with **Storm Sewer**.
- Use a description of your choice, like **Phase 2 Storm Sewer network.**
- Then, assign the Network parts list to use to **Storm Sewer**.
- For Pipe to create, choose a **450mm** or **18in** size pipe. This should be good size pipes, due to the high slopes we have. As mentioned before, the final sizing will be done in **Storm and Sanitary Analyses**, or **SSA** for short. It is the pipe modeling software that comes embedded in Civil 3D.
- For Structure to create, select a **1,200mm** or **48in** diameter, **cylindrical Structure.**
- Leave the **layers** to default.
- For **surface**, select the road **corridor surface.** This is the reference surface from which **pipe rules** will be applied.
- For **Alignment name**, choose **Rose Drive**. This will be the alignment from which structure stations and offset will be referenced.
- We will not **Erase the existing entity**, just in case we need it later.

- Check the box to **Use the vertex elevations**. After all, this is the whole point of the lengthy process of creating feature lines and assigning elevations to them.
- Finally, check the **Outside Top** to use as feature line elevations. This will be the point from where the cover is estimated from the final ground. Remember, we added an extra cushion of **0.5m** or **1.5ft** when we were creating the feature line. The reason was to account for the pipe diameter. That will not be needed if we are going to use the **Outside Top**. Nonetheless, let's accept the extra cushion. If in the end, we have enough room, we can raise the whole network to bring it to within the required **1.5m** or **5ft** cover.

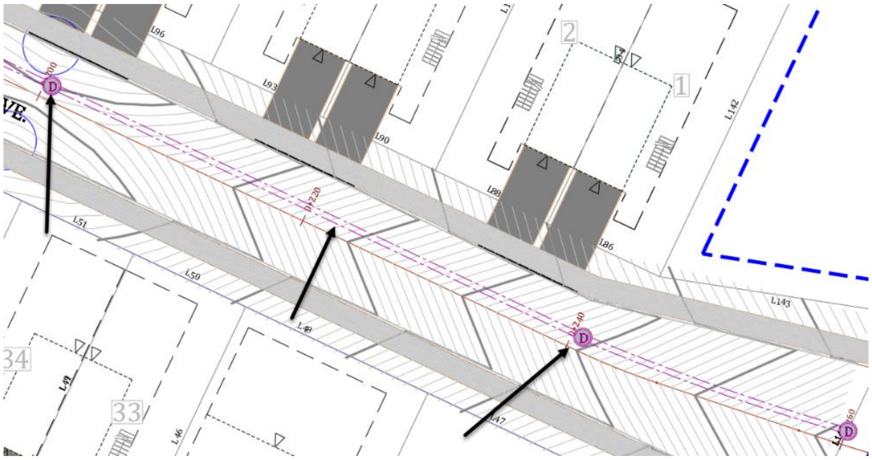



12. Now, click on **OK.**

NOTES

NOTES

13. The pipes and structures are now created and will appear in the drawing area, and also in the prospector. Next, we will see how to create and modify pipes using the **Pipe Creation Tools**.



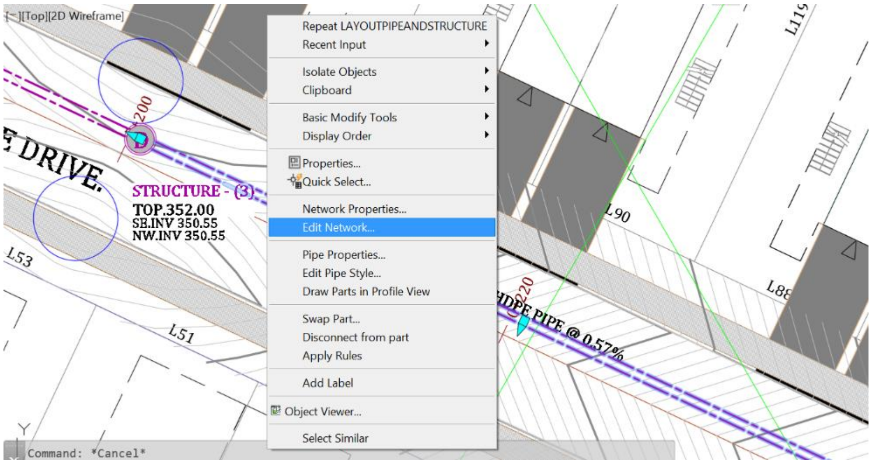
## **10.6** Pipes Creation Tools

The second option to create and edit pipe networks is by using the **Pipe Creation Tools.** Using the tools, we can modify an existing network or create a new one or set initial pipe network creation parameters. We will have options to assign names, descriptions, layers, parts and much more. Just like we did with the creation **by object** method.

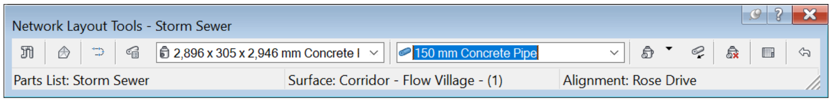
Let's use the tools to modify an existing system, the **Storm Sewer** network, and create a new one, the **Sanitary Sewer**.

We will modify the **Storm Sewer** network by adding the catchbasins and their leads to the manholes.

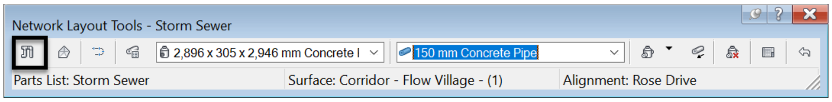
1. First, select any part belonging the **Storm Sewer**, right click and select **Edit Network.**



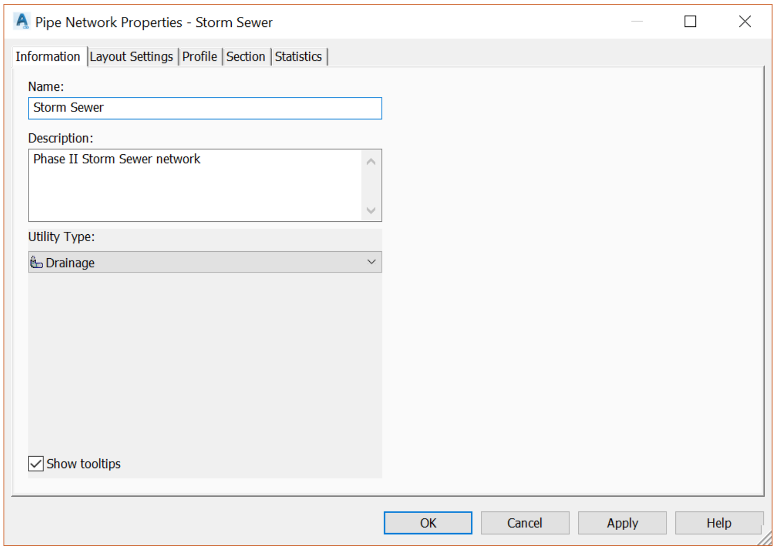
2. The **Network Layout Tools** opens and comes already set with the network parameters like the surface, the alignment, and the part family to use. We can still change them if needed.



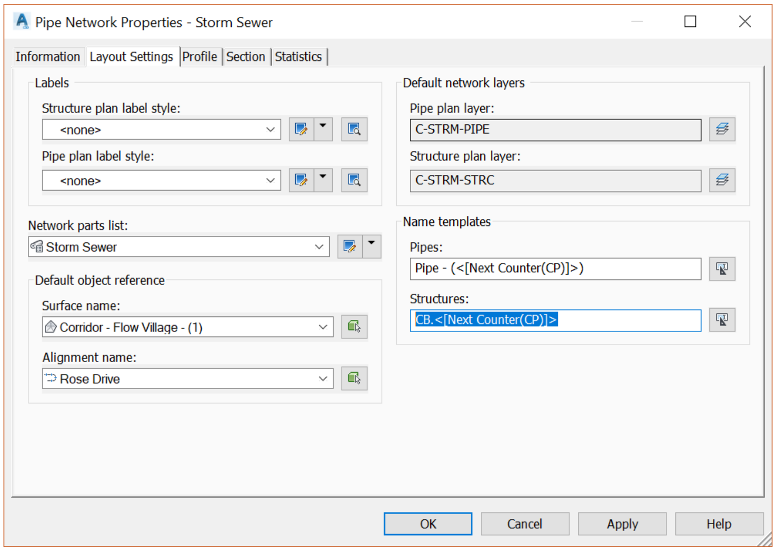

3. Let's explore a few of the layout tools. First is the Pipe Network Properties.




4. Once you click on it, you can change the Network name and description from the Information tab.



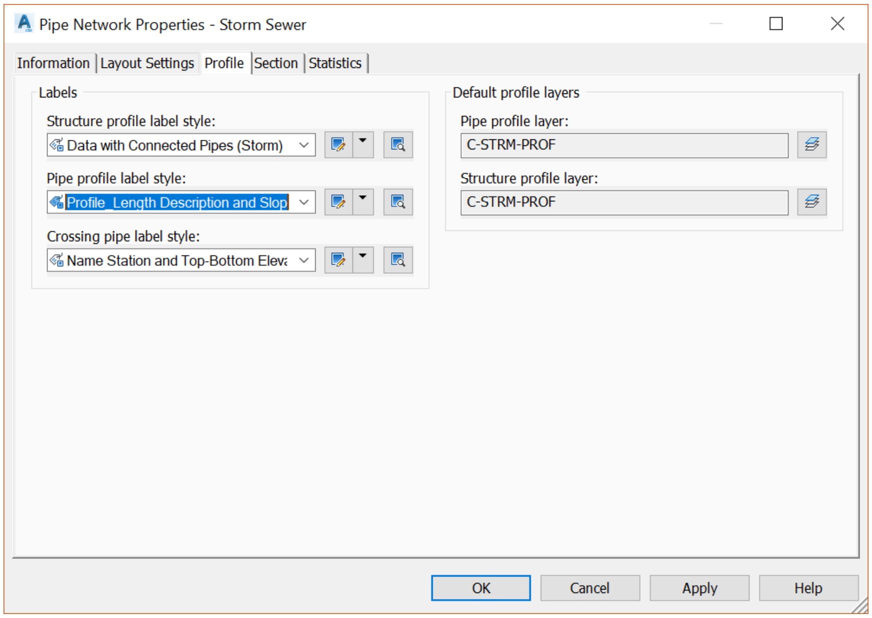

5. On the **Layout Settings** tab, you can change previous parameters. That includes reference surfaces, alignments, default layers, and **Name templates**. For example, let's name all catchbasins **CB.XXX**, where **XXX** is a number.




NOTES

NOTES

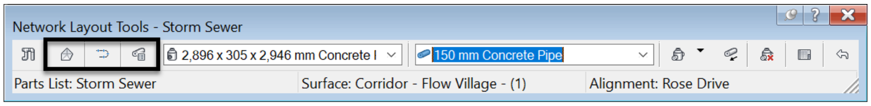
6. On the **Profile** tab, we can specify the default label and layers to assign to the network parts in **profile view**.



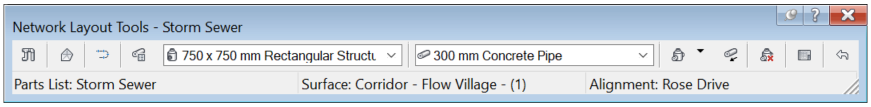

7. On the **Section** tab, we can set the default layers in section view. Meanwhile, **Statics** tab gives us a summary of all network parts.

8. Click **OK** to close the **Pipe Network Properties** window.

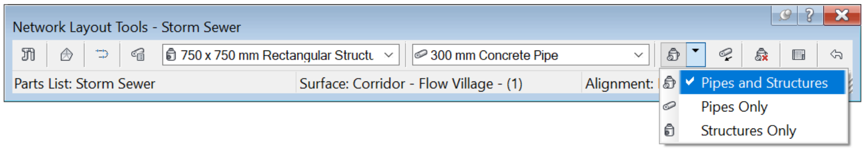
9. The next three items allow us to set the reference surface alignment and the part list to use.



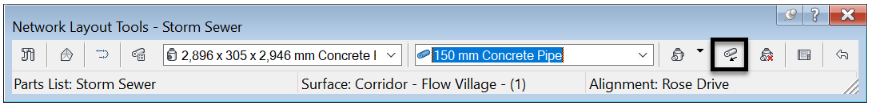

10. Next, we can choose, depending on the part list we are using, the specific structure and pipe we are creating. Since we are creating catchbasins, let's go with a rectangular junction box. Let's also choose a default **300mm** or **12in** pipe.



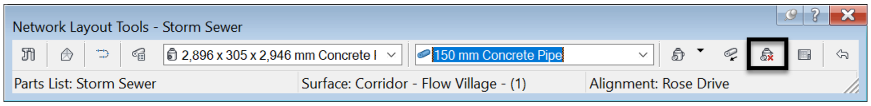

11. Next, we can decide to create **Pipes and Structures**, **Pipes Only** or **Structures Only.** That depends on our current needs. In this case, we are creating more catch basins and leads, so we need both.



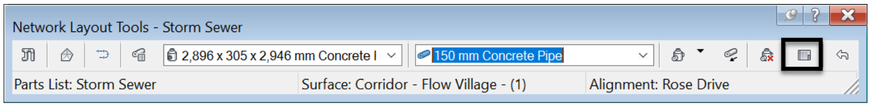
12. The next command allows us to **Toggle Upslope/Downslope**. That means we can specify whether the slope of the pipe network is upward or downward, in order to apply the proper pipe rules.



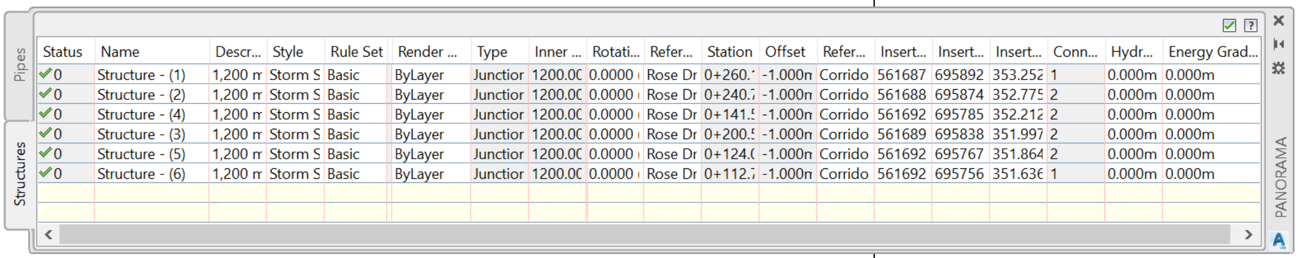

13. The next command allows you to delete network parts. You can do the same in the drawing by simply selecting a set of parts and hitting **delete** on the keyboard.




14. Next, we have the familiar **panorama** vista, which gives us access to a tabular view of the network parts and data.



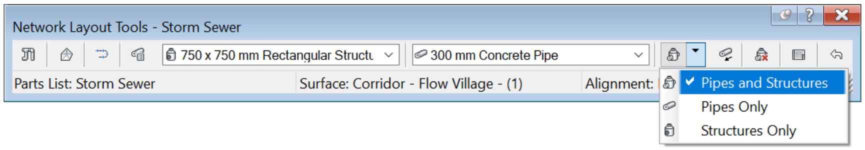

15. On the left side of the vista, we have two important tabs that give us access to **Structures** and **Pipes.** We may also have a third one giving access to the **Grading Editor** if a feature line is currently in editing mode.



| Status | Name | Descr... | Style | Rule Set | Render ... | Type | Inner ... | Rotati... | Refer... | Station | Offset | Refer... | Insert... | Insert... | Insert... | Conn... | Hydr... | Energy Grad... |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 0 | Structure - (1) | 1,200 m | Storm S | Basic | ByLayer | Junctior | 1200.0C | 0.0000 | Rose Dr | 0+260.1 | -1.000m | Corrido | 561687 | 695892 | 353.252 | 1 | 0.000m | 0.000m |
| 0 | Structure - (2) | 1,200 m | Storm S | Basic | ByLayer | Junctior | 1200.0C | 0.0000 | Rose Dr | 0+240.7 | -1.000m | Corrido | 561688 | 695874 | 352.775 | 2 | 0.000m | 0.000m |
| 0 | Structure - (4) | 1,200 m | Storm S | Basic | ByLayer | Junctior | 1200.0C | 0.0000 | Rose Dr | 0+141.5 | -1.000m | Corrido | 561692 | 695785 | 352.212 | 2 | 0.000m | 0.000m |
| 0 | Structure - (3) | 1,200 m | Storm S | Basic | ByLayer | Junctior | 1200.0C | 0.0000 | Rose Dr | 0+200.5 | -1.000m | Corrido | 561689 | 695838 | 351.997 | 2 | 0.000m | 0.000m |
| 0 | Structure - (5) | 1,200 m | Storm S | Basic | ByLayer | Junctior | 1200.0C | 0.0000 | Rose Dr | 0+124.0 | -1.000m | Corrido | 561692 | 695767 | 351.864 | 2 | 0.000m | 0.000m |
| 0 | Structure - (6) | 1,200 m | Storm S | Basic | ByLayer | Junctior | 1200.0C | 0.0000 | Rose Dr | 0+112.7 | -1.000m | Corrido | 561692 | 695756 | 351.636 | 1 | 0.000m | 0.000m |

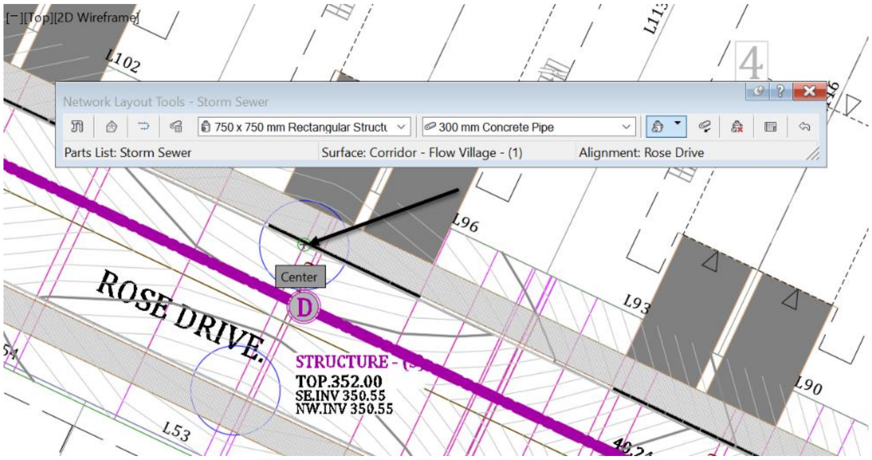


16. Now that we have perused the layout tools let's use them to create pipes and structures. Run the **Pipes and Structures** creation command.

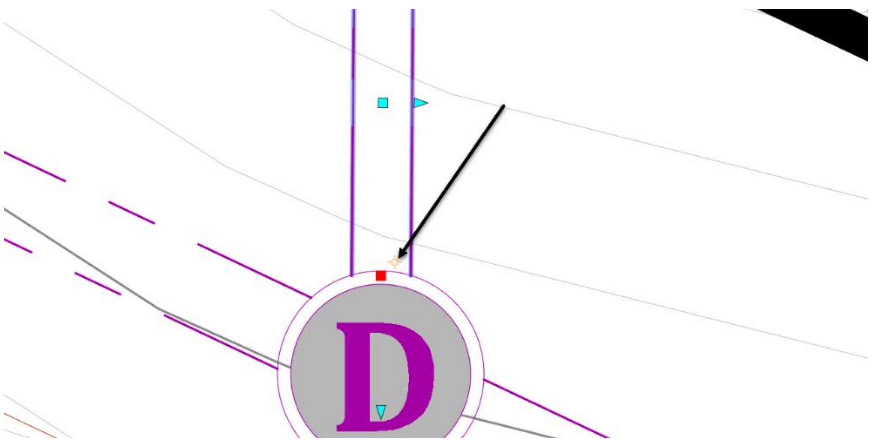



NOTES

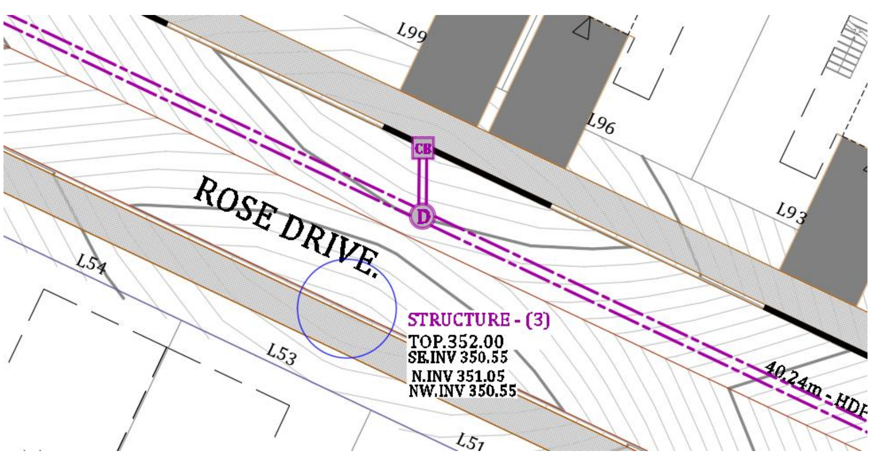
17. Click in the middle of the first catch basin circle. Make sure you have your circle object snap activated or use the **CEN** command line alias.




18. Once you click, the catchbasin is added. Next, bring your cursor close to an adjacent manhole, then click once you notice the small yellow symbol. That means that your pipe is being connected to that structure.




19. The catchbasin and lead are now created and tied to the connection manhole.



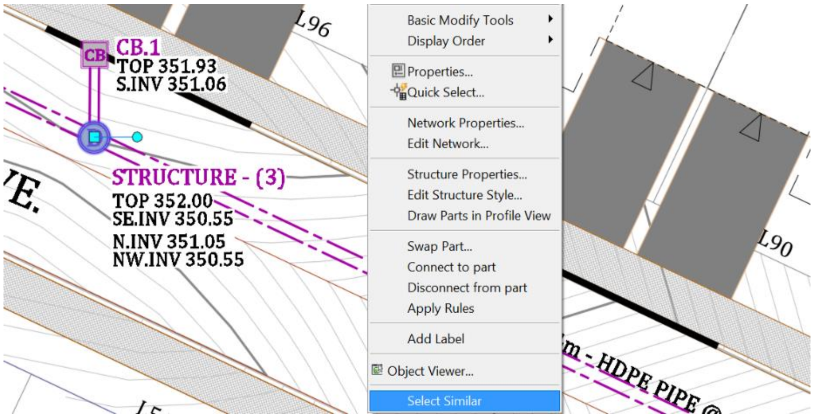

NOTES

## 10.7 Renaming Pipes and Structures

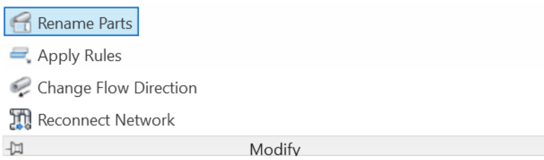
Before proceeding further, let's rename our storm structures to make them more explicit. Instead of **Structure – (X)**, let's use names like **DM.XXX** (for Drainage Manholes), **CB.XXX** (for catchbasins), and **SS.XXX** (for Sanitary Sewers). That will make the structures much easier to identify.

To rename the storm manholes:

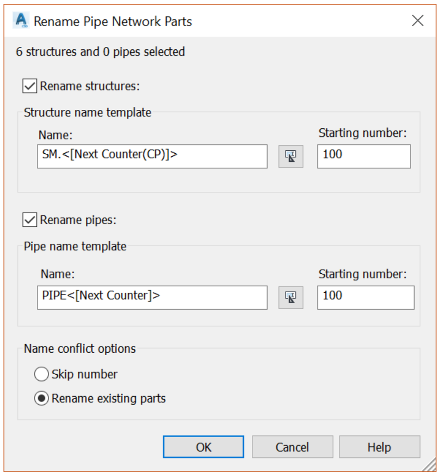
1. First, select one of the Drainage manholes, right-click and select similar.




2. Next, from the ribbon, expand the **Modify** panel and run the **Rename Parts** command.

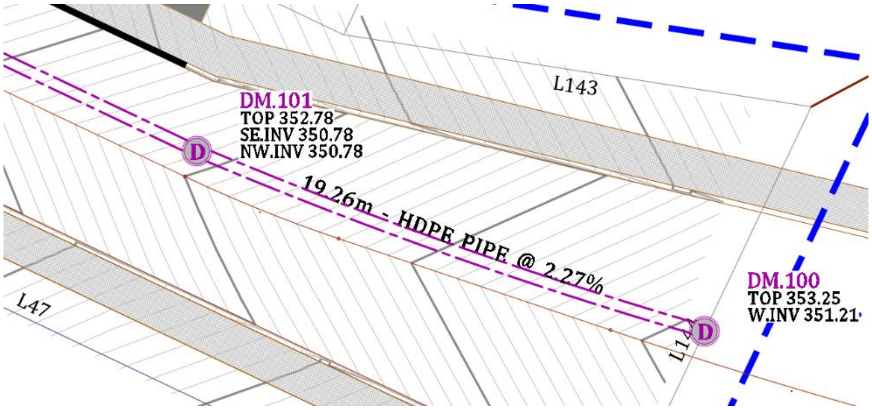



3. In the **Rename Pipe Network Parts** window, specify the name templates, starting numbers and increments. In this case, we would like to start numbering at **100** and increment by **one.** We are starting at 100 because the previous phase was assigned numbers from 0 to 99. We also want to **Rename existing parts**.




NOTES

4. Click on **OK**.
5. If you zoom in the drawing, you will notice that all **Drainage Manholes**, with the **D** symbol, have been renamed to the format **DM.XXX**




6. Next, use the layout tools to:

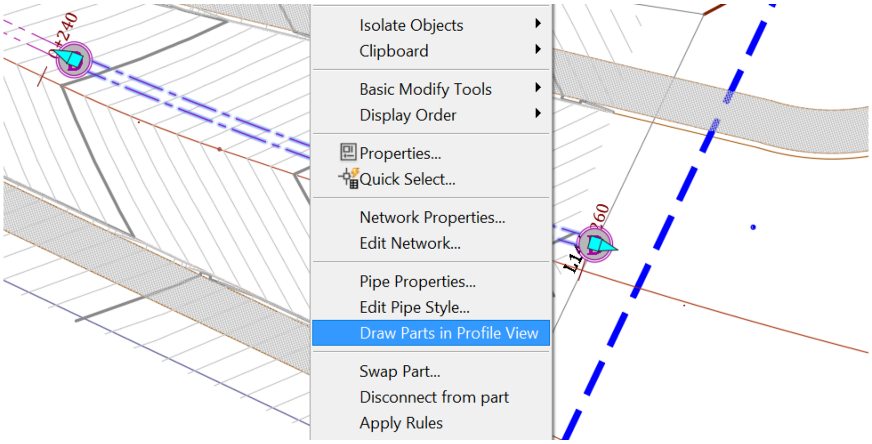
   - First, create the rest of the **CB**s or catchbasins. Make sure the incoming catchbasins inverts are at least at the same elevations as the sewer pipes and that they have a positive slope.
   - Then, create a completely new pipe network for the **Sanitary Sewer**. Among the steps we used for the **Storm Drainage** system, you can apply the following steps to create the **Sanitary Sewer** network.
     - Use the existing street centerlines to create manholes and pipes.
     - Insert a manhole at each major junction.
     - Make sure each lot can be connected to a sanitary sewer main.
     - Label each pipe and manholes. Use the **SS.XXX** format to renumber manholes**.**
     - Set the **Sanitary Sewer** network layout settings, to reference the corridor surface. Each part also needs to reference the appropriate alignment. We can also set default layers and name templates.
     - Set the sanitary manholes to concentric structure types. That will differentiate them from storm structures, which are using a cylindrical style.
     - Set the pipe diameters to **200mm** or **8in** for sanitary mains.

NOTES

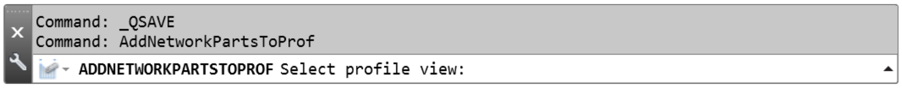
**10.8** Projecting Pipes in Profiles.

Now, let's project the pipe network in profile view, for better visualization and annotation.

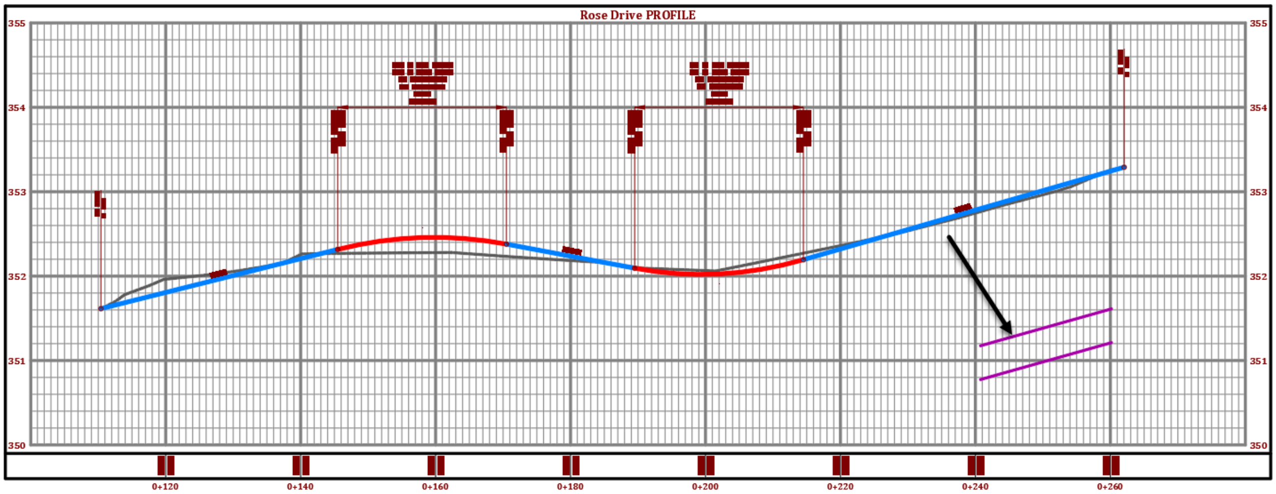
1. Right-click on one of the pipes or structures and select **Draw Parts in Profile View**.




2. When prompted to select a profile view, select the **Rose Drive** profile view. We can find it in the profiles area of the drawing.

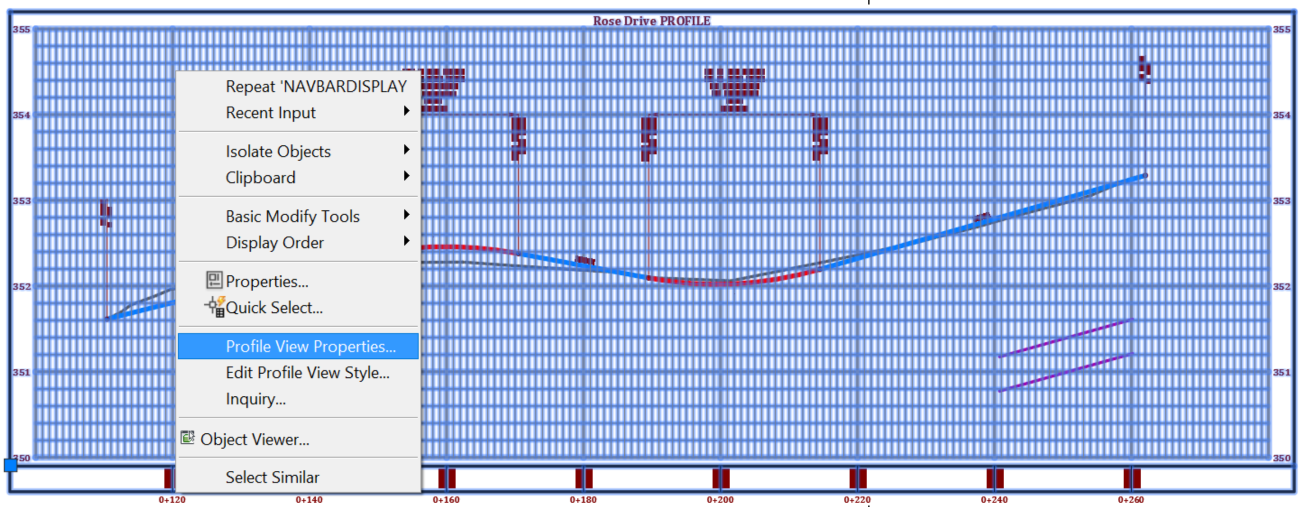



3. Notice that the pipe lines will appear in the profile view.

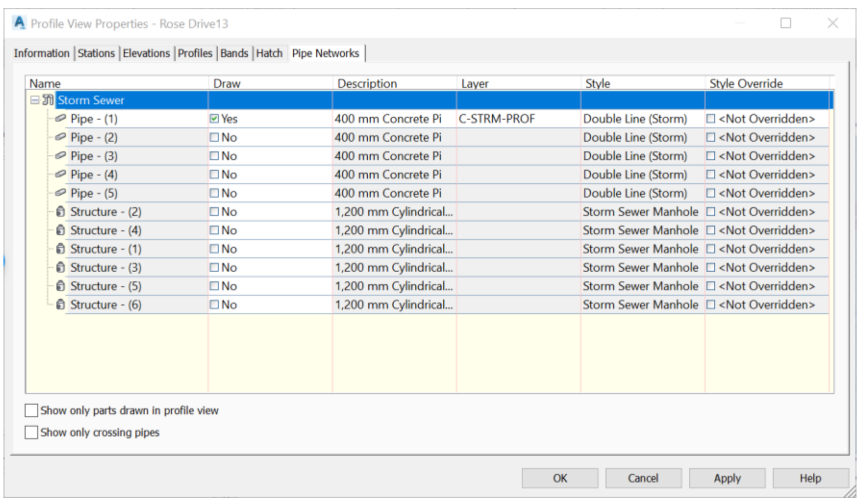


NOTES

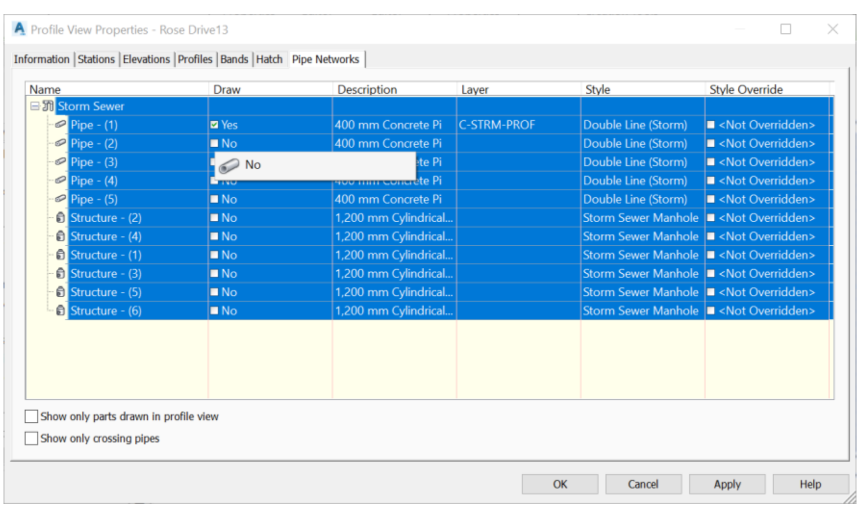
4. Now, let's project all pipes and structures in the profile view. To do so, **right-click** on the grids of the profile view and select **Profile View Properties.**



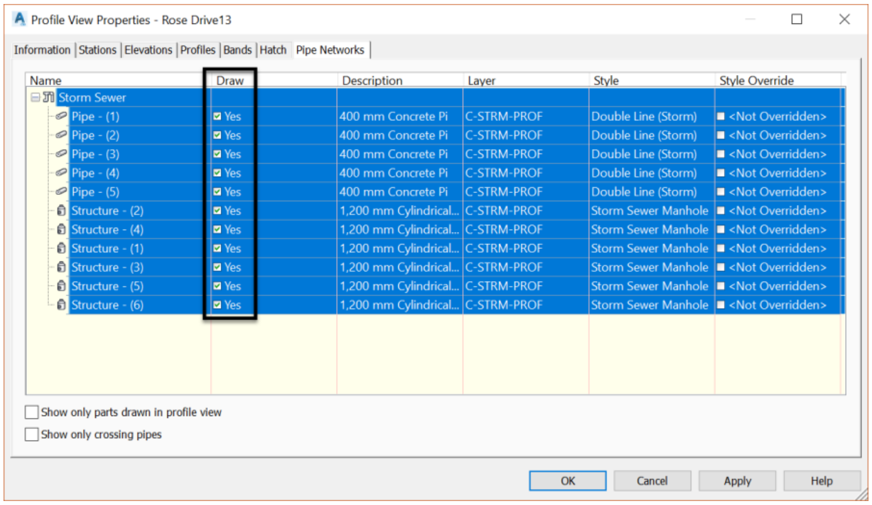

5. Activate the **Pipe Networks** tab. Currently only the projected pipes or structures have the **Yes/No** option checked to **Yes.**



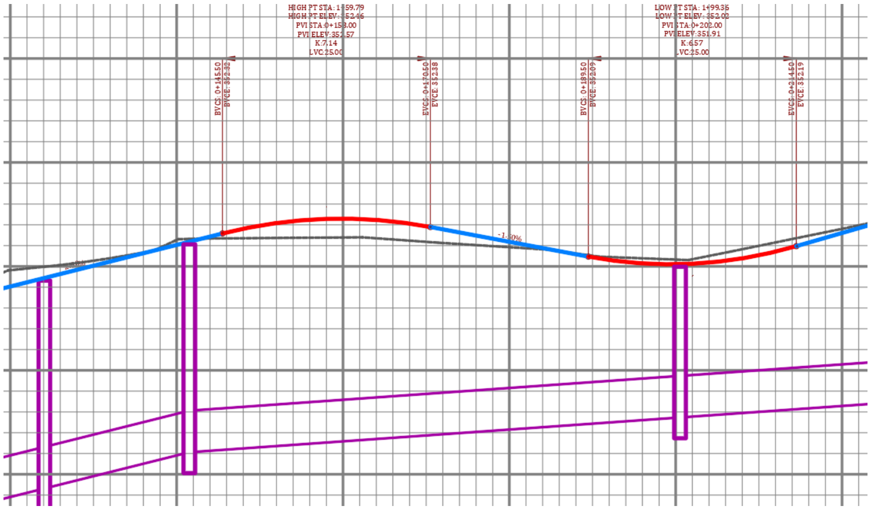

6. To project a part to the profile view, we need their status to be a **Yes.** To do that, select all of them by simultaneously pressing **CTRL + A** on the keyboard to select all of them.



7. Click on one of the **No** options in the **Draw** column to convert all options from **No** to **Yes.**




8. Click **OK,** and all parts should now appear in the profile view.

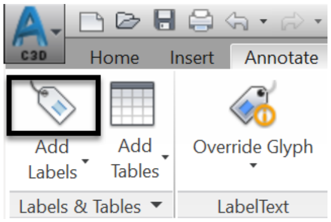


NOTES

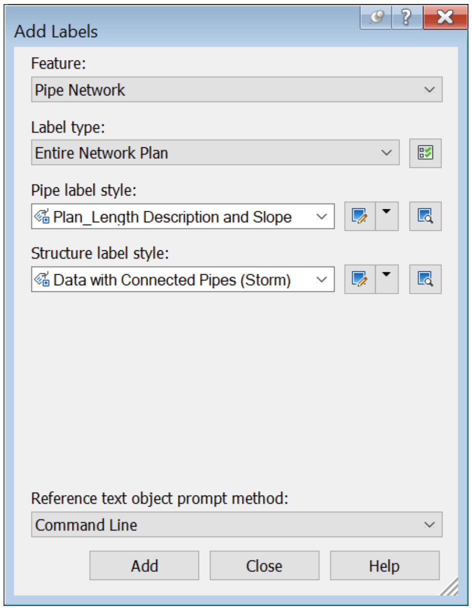
## **10.9** Labeling a Pipe Network

We want to make construction plans more readable. So, after creating the pipe network, we should add labels to the plan and profile network parts. To do that,

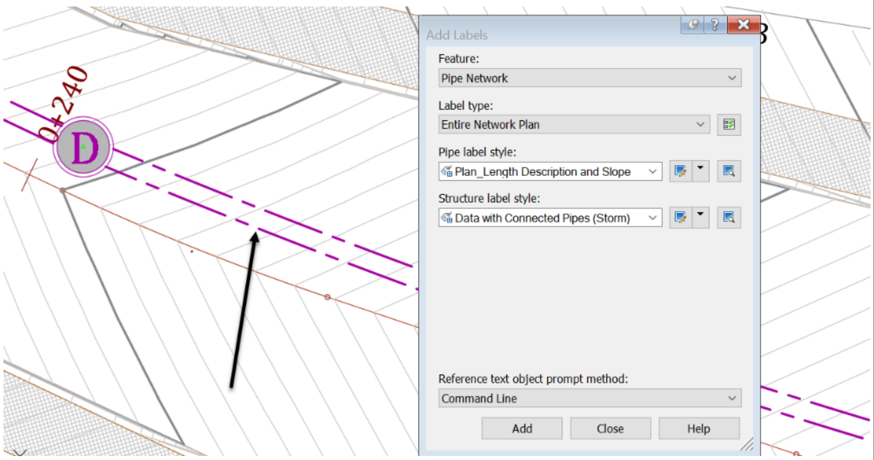
1. First, reopen the **Add Label** dialog box, from the **Annotate** tab of the ribbon, then run the **Add labels** command.



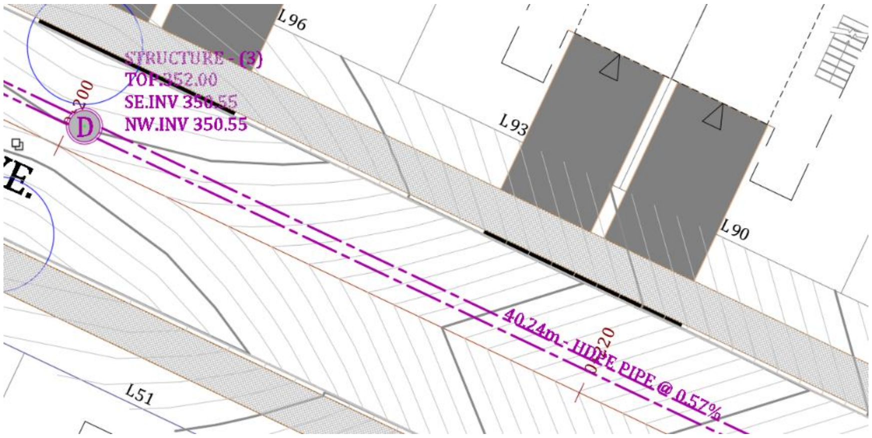

2. In the **Add Labels** dialog box, choose the options to label the **Entire Network Plan.** Then choose the plan label styles for pipes and structures and click on **Add**.



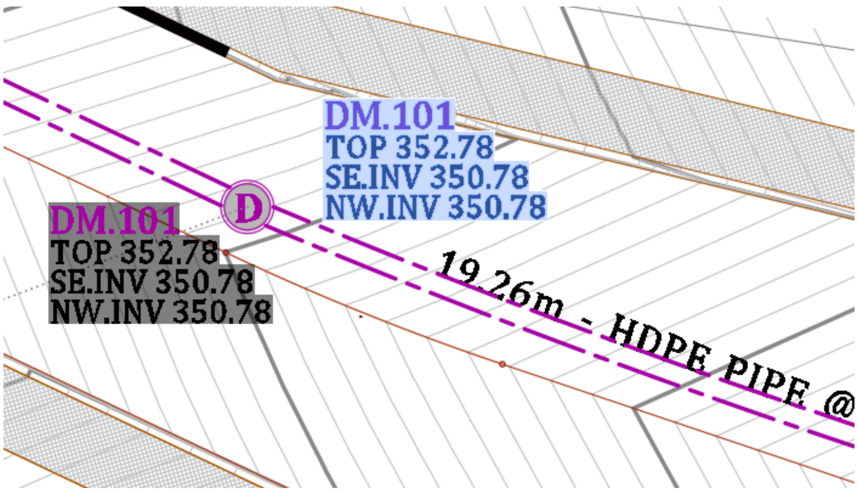

3. Click on a network part, pipe or structure, in plan view.



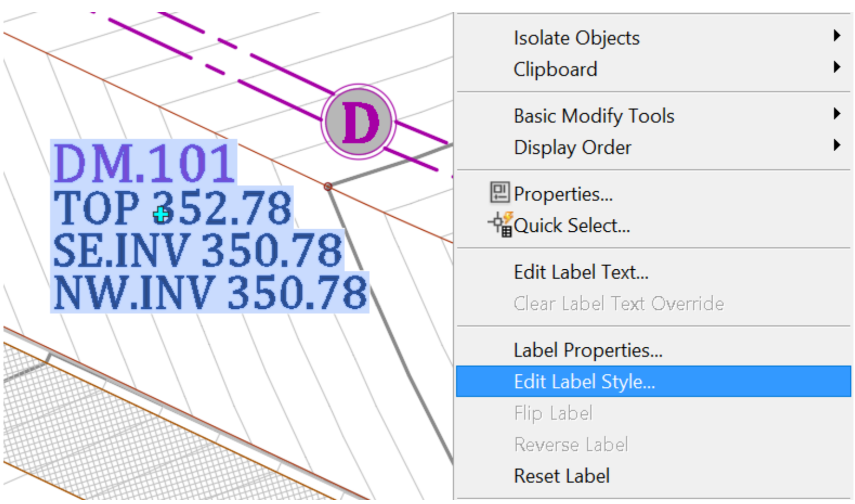

NOTES




4. If the labels overlap other objects, you can move and position them manually to a better spot.



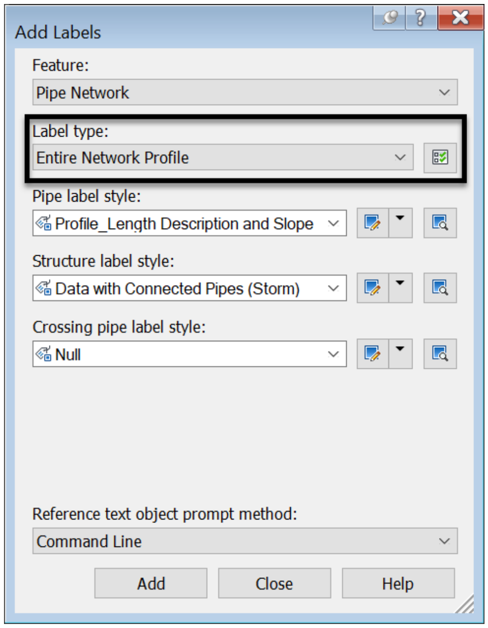

5. You can also edit label styles just like any other Civil 3D object.



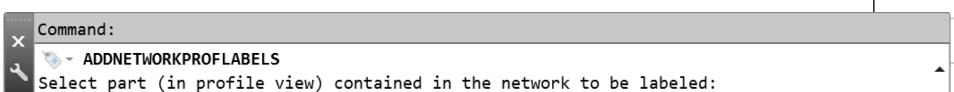

NOTES

NOTES

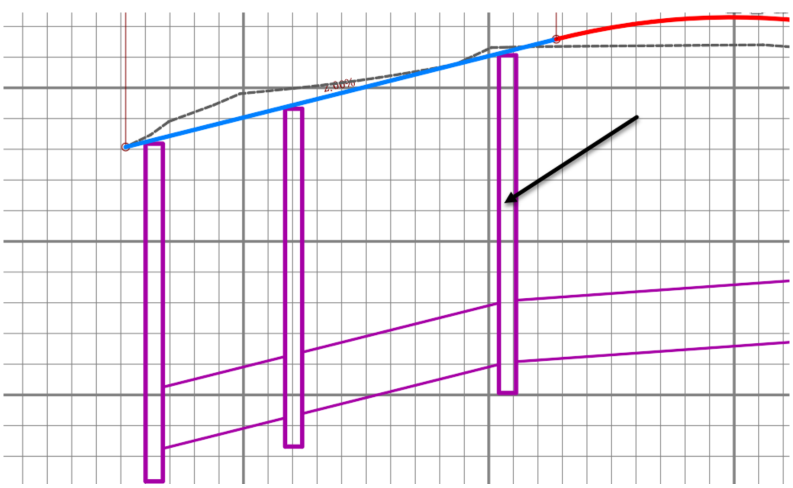
6. Next, let's add labels to the network in profile view. Return to the **Add Label** dialog box. This time, for **Label Type**, select the **Entire Network Profile** and click **Add**.




7. You will be prompted to select the parts to label.



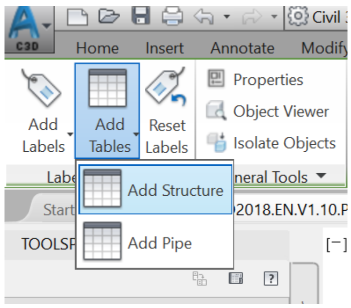

8. Go to the **Rose Drive** profile and click on one of the parts.



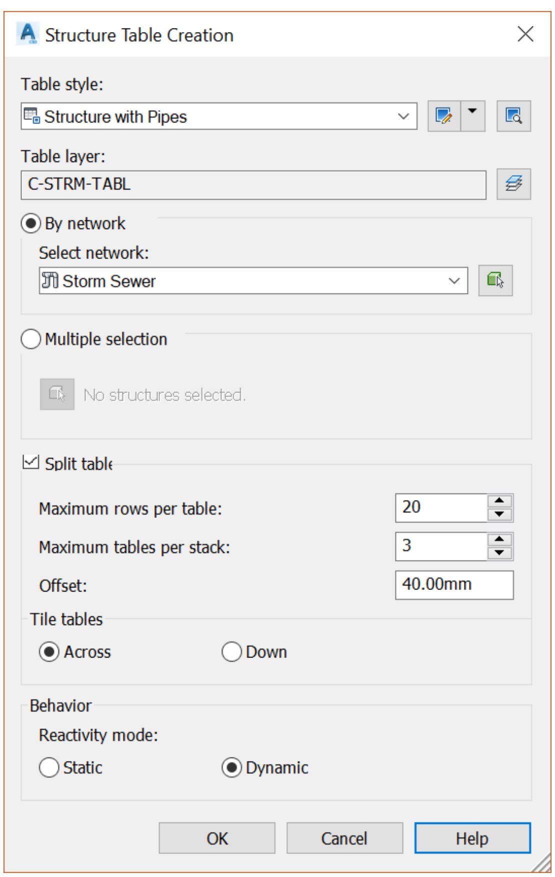
## **10.10** Creating pipe and structure tables

As the pipe network grows, it will be harder to keep an inventory of all parts included in the network. A table would be a good option to provide more information on the network. We can include parts information, including connections, inverts, slopes, descriptions, coordinates, and more. To add a pipe network table, follow these steps:

1. Select a part of the network (pipe or structure).
2. Let's start with a structure table. On the ribbon, run the **Add Structure** command.




3. The **Structure Table Creation** dialog box presents the table creation generic options. Among the choices, we have **Table Style**, **Table Layer**, **Split Table**, and **Static** or **Dynamic Behaviors**. The only thing new here is the option to create a table **By network** or by **Multiple selections**. Pick **By network** to create a table for the entire **Storm Sewer** network. Then, click **OK** to create the table.



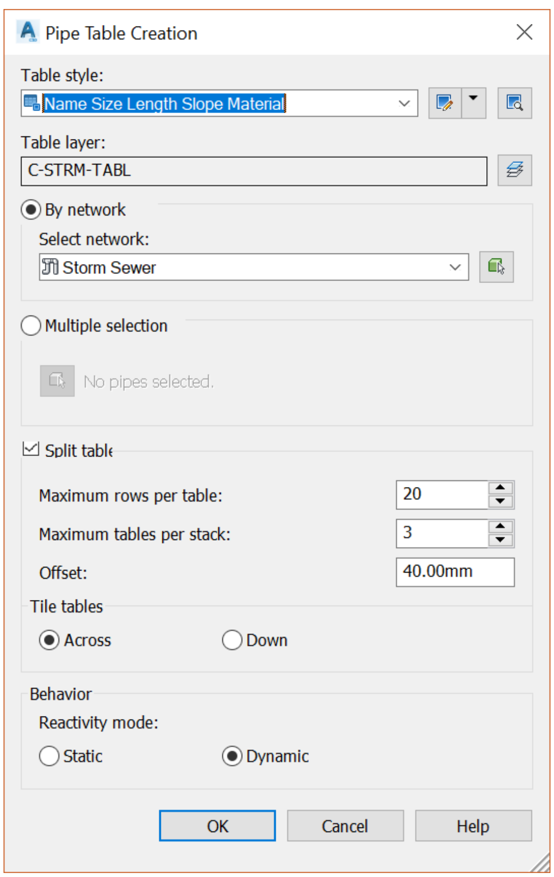

NOTES

4. Click somewhere close to the other tables in the drawing to insert the **Structure Table**.

| Parcel Line Table | | |
|---|---|---|
| Line # | Length | Direction |
| L44 | 20.00 | N25° 21' 13.60"E |
| L45 | 22.99 | S64° 38' 59.58"E |
| L46 | 20.00 | N25° 21' 13.60"E |
| L47 | 22.99 | S64° 38' 59.58"E |
| L48 | 13.81 | S64° 38' 59.58"E |
| L49 | 20.00 | N25° 21' 13.60"E |
| L50 | 8.50 | S64° 38' 59.58"E |
| L51 | 8.50 | S64° 38' 59.58"E |
| L52 | 20.00 | N25° 21' 13.60"E |
| L53 | 8.50 | S64° 38' 59.58"E |
| L54 | 8.50 | S64° 38' 59.58"E |
| L55 | 20.00 | N25° 21' 13.60"E |
| L56 | 8.50 | S64° 38' 59.58"E |
| L57 | 8.50 | S64° 38' 59.58"E |
| L58 | 20.00 | N25° 21' 13.60"E |
| L59 | 8.50 | S64° 38' 59.58"E |
| L60 | 8.57 | S64° 38' 59.58"E |
| L61 | 20.00 | N25° 21' 13.60"E |
| L62 | 5.85 | N78° 07' 41.02"E |
| L63 | 11.84 | N25° 04' 54.40"E |

| STRUCTURE TABLE | | | |
|---|---|---|---|
| STRUCTURE NAME: | DETAILS: | PIPES IN: | PIPES OUT |
| DM.100 | 1200 mm<br>RIM = 353.25<br>SUMP = 351.11<br>INV OUT = 351.212 | | 400 mm W.INV 351.21 |
| DM.101 | 1200 mm<br>RIM = 352.78<br>SUMP = 350.38<br>INV IN = 350.775<br>INV OUT = 350.775 | 400 mm SE.INV =350.78 | 400 mm NW.INV 350.78 |
| DM.103 | 1200 mm<br>RIM = 352.21<br>SUMP = 349.81<br>INV IN = 350.212<br>INV OUT = 350.212 | 400 mm SE.INV =350.21 | 400 mm NW.INV 350.21 |
| DM.102 | 1200 mm<br>RIM = 352.00<br>SUMP = 350.15<br>INV IN = 350.547<br>INV IN = 351.052<br>INV OUT = 350.547 | 400 mm SE.INV =350.55<br>300 mm N.INV =351.05 | 400 mm NW.INV 350.55 |
| CB.1 | ??? mm<br>RIM = 351.93<br>SUMP = 349.06<br>INV OUT = 351.058 | | 300 mm S.INV 351.06 |
| DM.104 | 1200 mm<br>RIM = 351.86<br>SUMP = 349.46<br>INV IN = 349.864<br>INV OUT = 349.864 | 400 mm E.INV =349.86 | 400 mm W.INV 349.86 |
| DM.105 | 1200 mm<br>RIM = 351.64<br>SUMP = 349.24<br>INV IN = 349.636 | 400 mm E.INV =349.64 | |

5. Use the same previous steps, used for the structures, to insert the **Pipe Table**.




NOTES

6. Click in the drawing area to insert the **Pipe Table.**

| Pipe Table | | | | | |
|---|---|---|---|---|---|
| NAME | SIZE | LENGTH | SLOPE | START STRUCTURE | END STRUCTURE |
| Pipe - (1) | 400 mm | 19.3 m | 2.27% | Structure - (1) | Structure - (2) |
| Pipe - (2) | 400 mm | 40.2 m | 0.57% | Structure - (2) | Structure - (3) |
| Pipe - (3) | 400 mm | 59.0 m | 0.57% | Structure - (3) | Structure - (4) |
| Pipe - (4) | 400 mm | 17.8 m | 1.95% | Structure - (4) | Structure - (5) |
| Pipe - (5) | 400 mm | 11.4 m | 2.00% | Structure - (5) | Structure - (6) |

NOTES

# 11 SECTIONS

## 11.1 Introduction

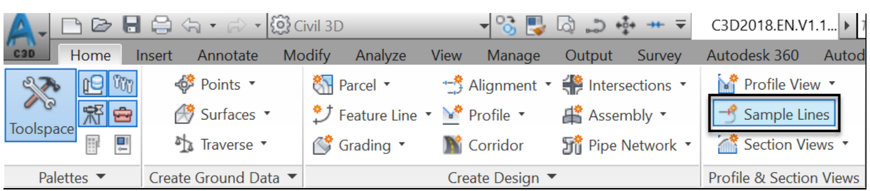
Cross sections enable us to examine existing and proposed surface elevations. In Civil 3D, section data is defined and displayed using sample lines grouped in a collection called a **sample line group**.

The Civil 3d sample line creation wizard allows us to create cross-sections. We can create them at given intervals, and at a specified left and right distance from the alignment centerline.

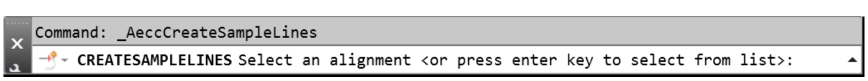
In this project, we will create cross-sections at **20m** or **50ft** intervals along **Rose Drive**. To do this, we need to create sample lines that will represent the stations where the sections will be cut.

## 11.2 Creating sample lines

1. Open the **11.01-Sections dwg** file in the **Lesson 11** practice folder**.** You will notice that we did a little more work in the drawing from where we left in the last lesson. In particular, we created the sanitary and water network. We have also projected these two networks in their respective profiles and added labels in plan and profile views.
2. To create sample lines, launch the **Sample Lines** creation command.

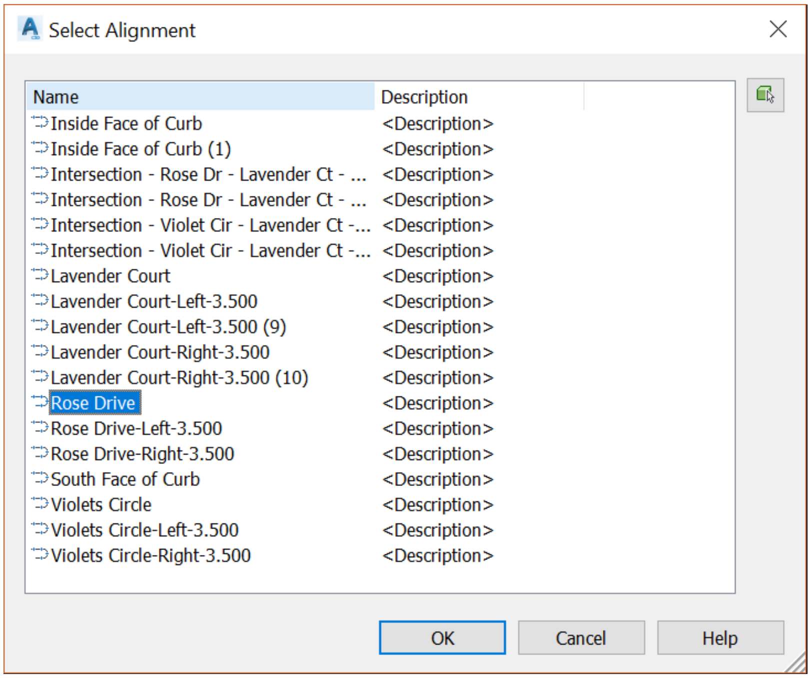



3. When prompted to select an alignment to create sample lines for, press **Enter** at the command line. Note that you can directly click on the alignment in the drawing to select it. However, clicking in the drawing is not recommended when you have a huge number of overlapping alignments. We could potentially click on the wrong alignment.

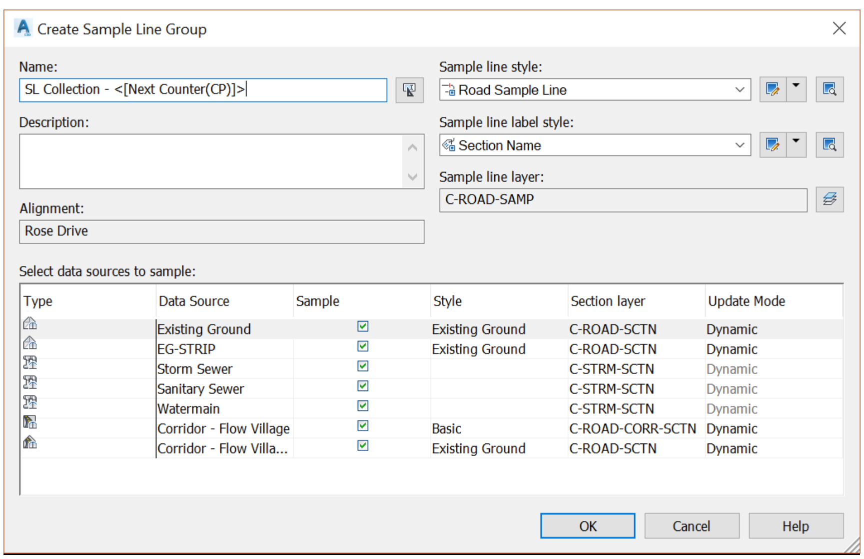



NOTES

4. In the **Select Alignment** window, select **Rose Drive** and click **OK**.



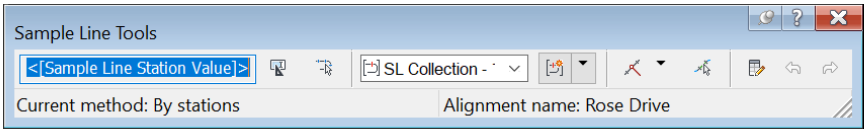

5. In the new **Create Sample Line Group** dialog box, select a Civil 3D object that can be sampled, such as surfaces, corridors, and pipe networks. In the current situation, we will choose all existing surfaces and the corridor**.** In this same window, we can specify the sample line styles, as well as the layer. Let's leave all the options to the default ones and click on **OK**.



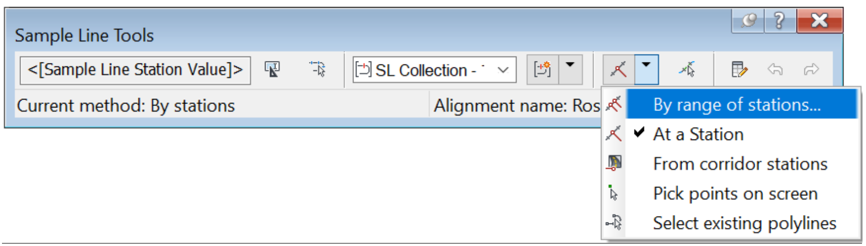

NOTES

NOTES

6. Next, we are presented with the **Sample Line Tools** that will enable us to create sample lines according to our specifications. It's not the most obvious thing to notice. But, the sample line creation method and the working alignment are displayed at the bottom of the toolbar. In this case, we can see that the current method is **By Station**.

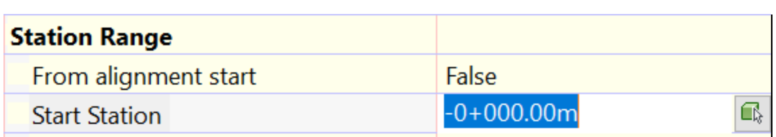



7. Let's go ahead and provide more details for the sample lines creation. On the **Sample Line Tools**, create new sample lines **By range of stations**.

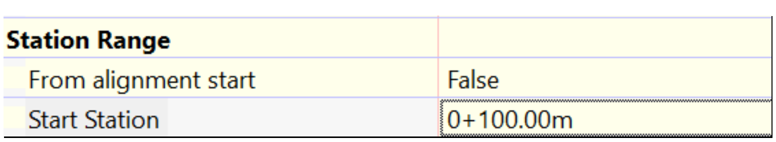



8. Then, in the **Create Sample Lines–By Station Range** window, we will proceed as follows:

9. First, set **From Alignment Start** and **From Alignment End** values to false. This means that we don't want to create sample lines, thus sections, on the entire length of the alignment. Therefore, we need to specify where we want to start and end. Click inside the text box to the right of **Start Station**. To the right of the **-0+000.00**, click on the small green cube and white arrow icon. Next in the drawing, click on **Rose Drive,** just before the start of our project, on the west end. Maybe, at station **0+100m** or **0+320ft.** Alternatively, if you know the exact station where you want your sections to start, you can just type in the station values.




10. Our start is now set.



NOTES

11. Repeat the same process to set the **End Station.** This time, to specify the station, click on the alignment, just before the roundabout at the east end, something like station **0+280.00m** or **0+920.00ft.**

| Station Range | |
|---|---|
| From alignment start | False |
| Start Station | 0+100.00m |
| To alignment end | False |
| End Station | 0+280.00m |

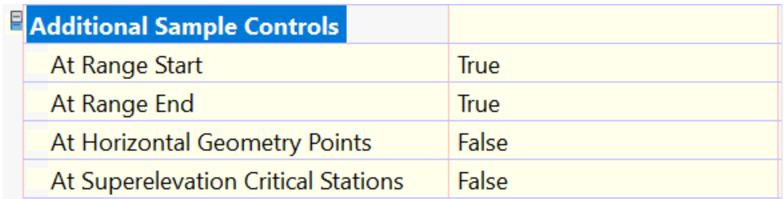
12. Next, we need to specify the **Swath Widths**. This means how far left or right of the alignment centerline we want to sample. We want to cover the entire right of way and maybe a few meters or feet inside the lots. So **15m** or **50ft** on both sides will be more than enough. Note that we have an option to **Snap to an alignment** for the definition of the swath width. This option extends the sample lines out to an alignment, an offset alignment for instance, rather than a fixed width. This scenario does not apply to us, so we will simply choose **False** on both accounts.

| Left Swath Width | |
|---|---|
| Snap to an alignment | False |
| Alignment | Rose Drive |
| Width | 15.000m |
| **Right Swath Width** | |
| Snap to an alignment | False |
| Alignment | Rose Drive |
| Width | 15.000m |

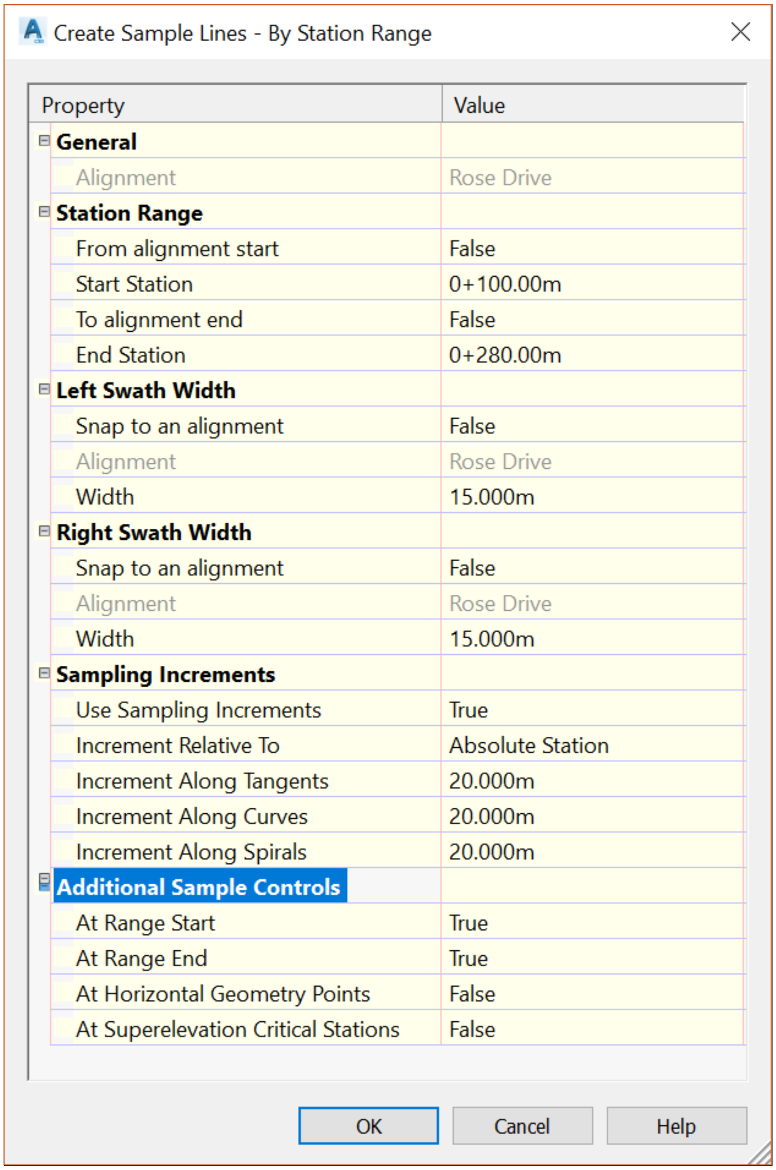
13. Next is another important item on the list, the **Sampling Increments**. To use regular increment values, we need to set the **Use Sampling Increments** line value to **True**. We will enter **20m** or **50ft** along both tangents and curves. We don't need to worry about **spirals** since we don't have any on this street.

| Sampling Increments | |
|---|---|
| Use Sampling Increments | True |
| Increment Relative To | Absolute Station |
| Increment Along Tangents | 20.000m |
| Increment Along Curves | 20.000m |
| Increment Along Spirals | 20.000m |

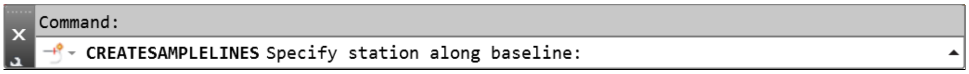
14. Finally, we need to set some **Additional Sample Controls**, to create, or not, more sample lines at specific locations along the alignment. Let's go ahead and answer **True** for **At Range Start** and **At Range End**. This is specifically important when calculating material quantities. They are, most of the time, computed using the average area method between two sample line stations. This is so that you don't miss quantities by not creating sample lines at the start and end of the project.




15. Once you are done with the **Create Sample Lines** window, click **OK** to create the lines.




16. When requested to choose additional locations to create sample lines at the command line, hit **Enter** or **Escape**.




NOTES

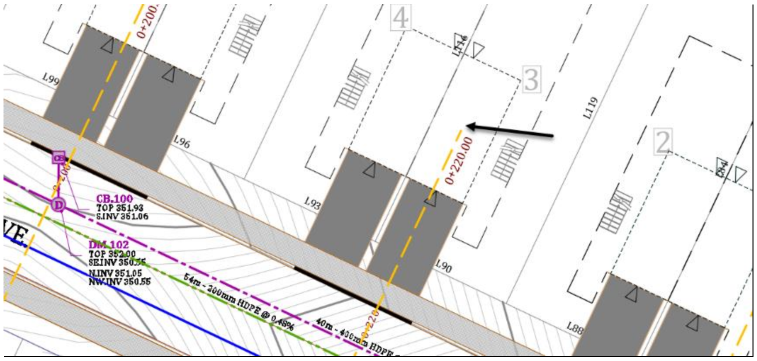
17. Observe the sample lines created at each major station every **20m** or **50ft**.


NOTES

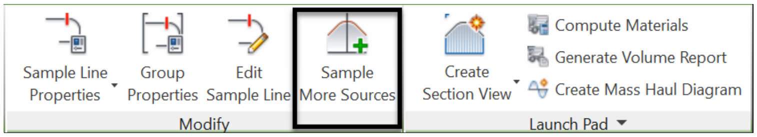
## **11.3** Checking sampled sources

Before creating the sections, you may need to check if all the surfaces have been properly sampled. Remember that, if a surface is created after the sample lines, the surface will not be included in the set of samples. So, we need to make the **Sample Lines Group** aware of newly available sources. To do that,

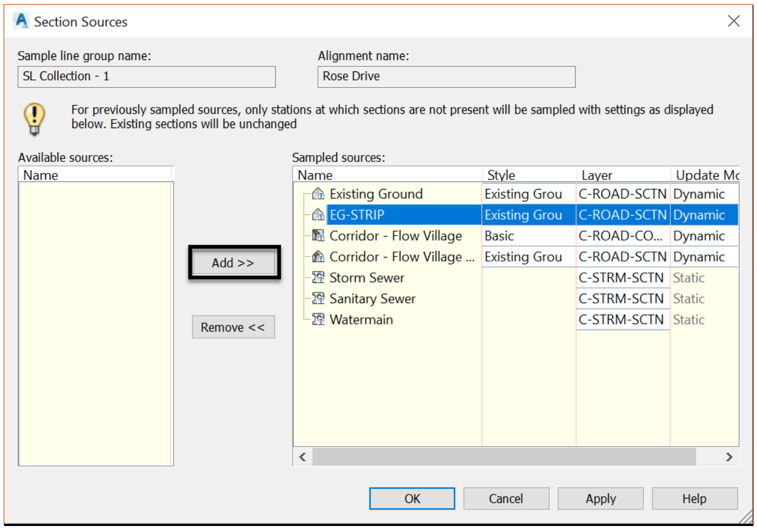
1. First, **right-click** on one of the sample lines.



2. Select **Sample Additional Sources** from the ribbon.

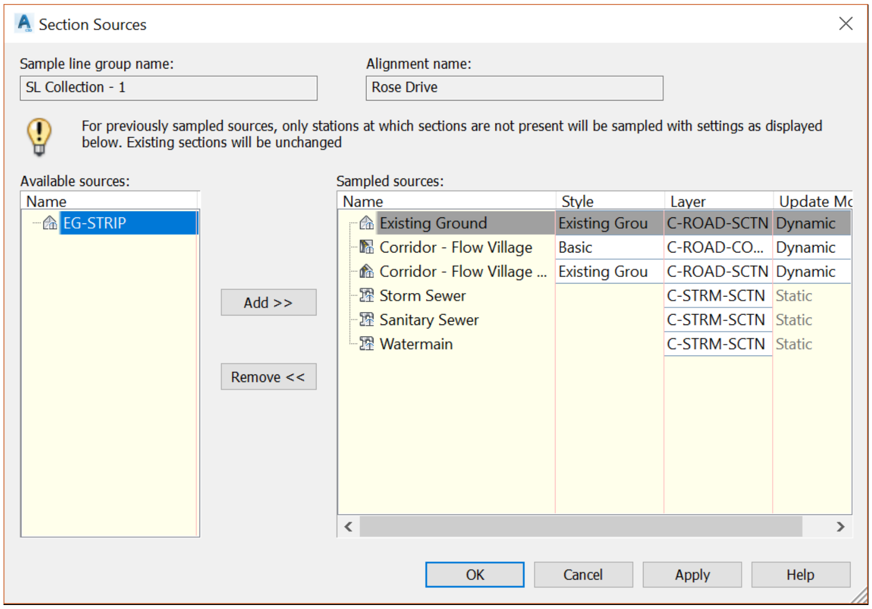



3. If prompted to, click on one of the sample lines again.
4. In the **Section Sources**, on the left side, there's a section called **Available sources**. On the right side, we have a list of already **sampled sources**. If we had created any new source, surface or pipe network, after the sample lines, they would be shown in the left section. In our case, we have sampled all the sources. On the other hand, maybe we don't want to use certain sources in the sections. Case in point, we don't need to show the **EG-STRIP** surface. To remove it from the sampled list, select it and click on the **Remove** button.




NOTES

5. The **EG-STRIP** is now removed from the list.



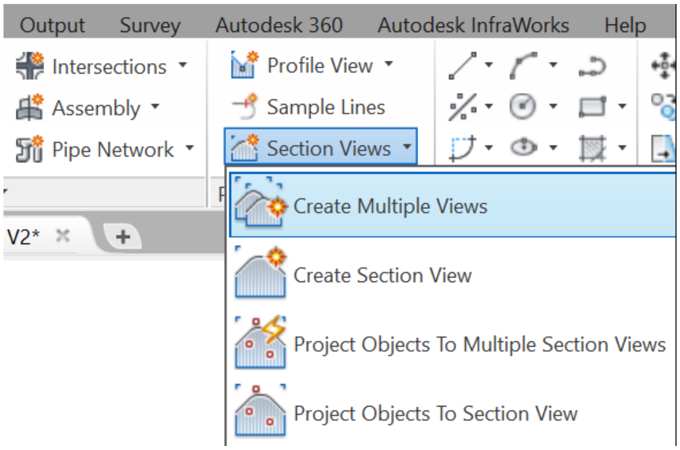

6. Click **OK** to exit the **Section Sources** window.

NOTES

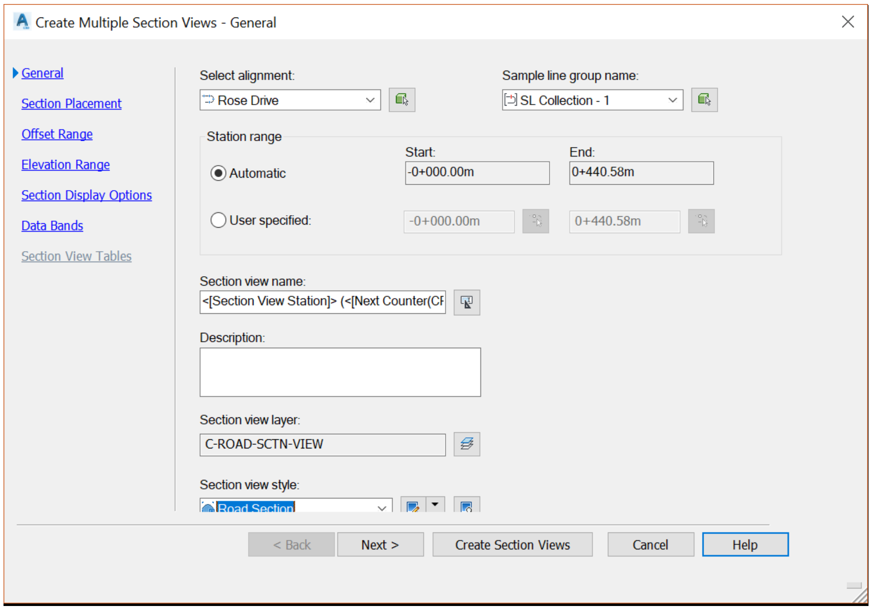
## 11.4 Creating Cross Sections

To create the cross-sections,

1. From the ribbon, on the **Home** tab, run the **Create Multiple Views** command.

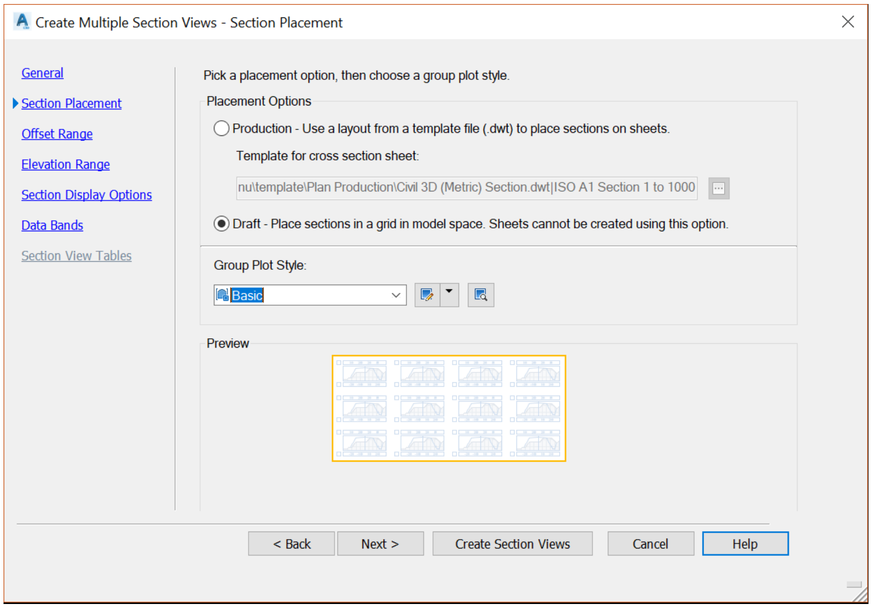



2. On the **General** page, we can specify the parent alignment, sample line groups, stations, description, style and more information. Select the **automatic** option and click on **Next**.

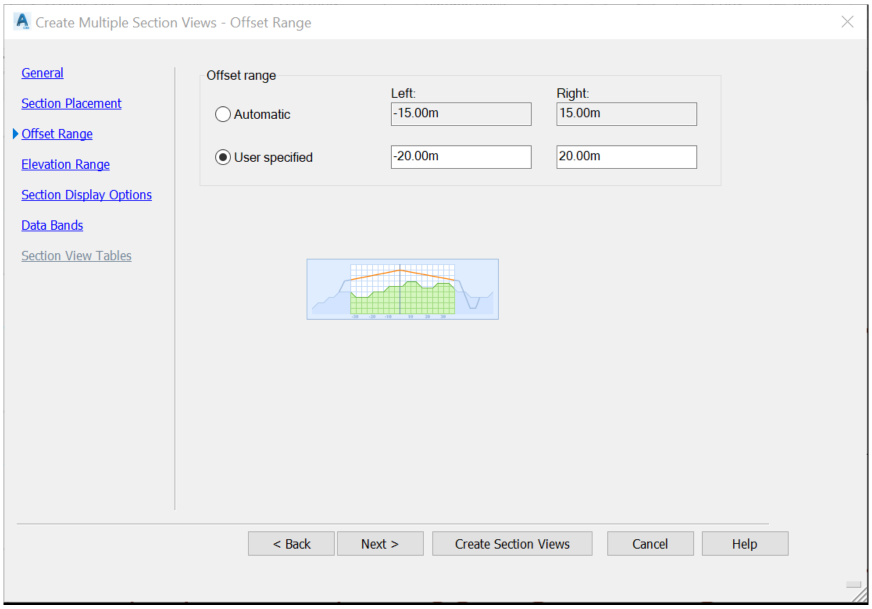



NOTES

3. On the **Section Placement** page, we have two options. We can either create the sections with a pre-existing drawing template or place them temporarily in the drawing area. For now, let's place them as a **draft** in the model and click on **Next**.

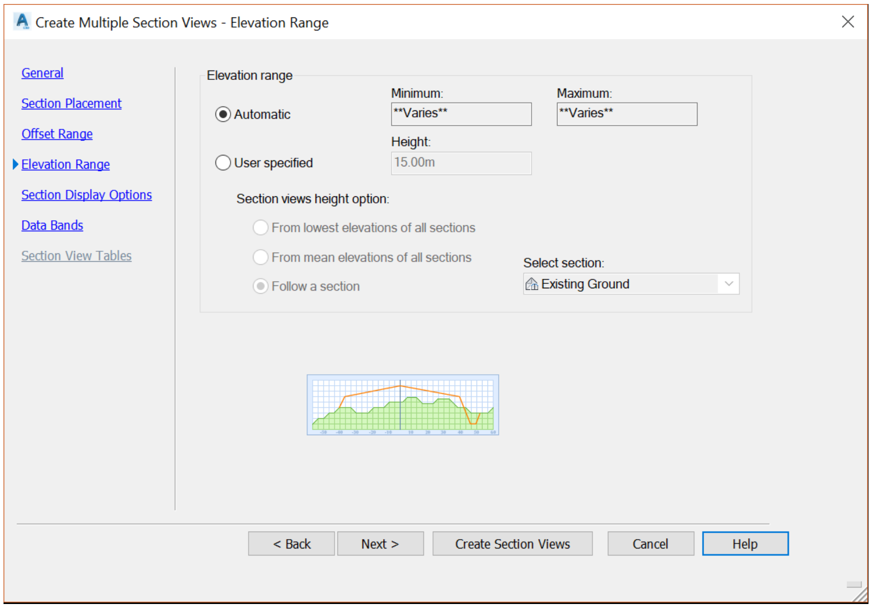



4. On the **Offset Range**, we can specify the offset range of the section views. We have sampled **15m** or **50ft** on each side, but let's create the view for **20m** or **75ft** on each side and click on **Next**.

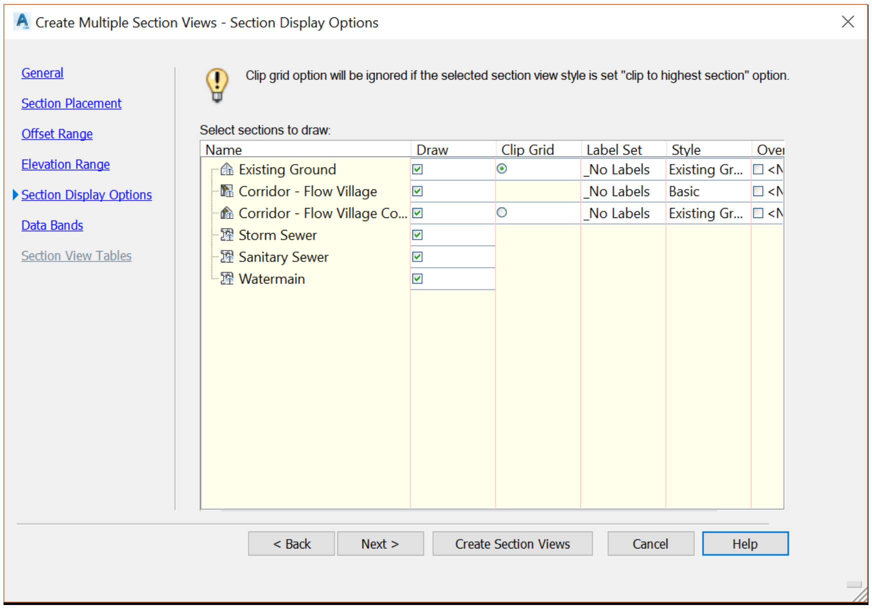



NOTES

5. On the **Elevation Range** page, specify the elevation range in the section view. Let's go with the automatic range option and click on **Next**.



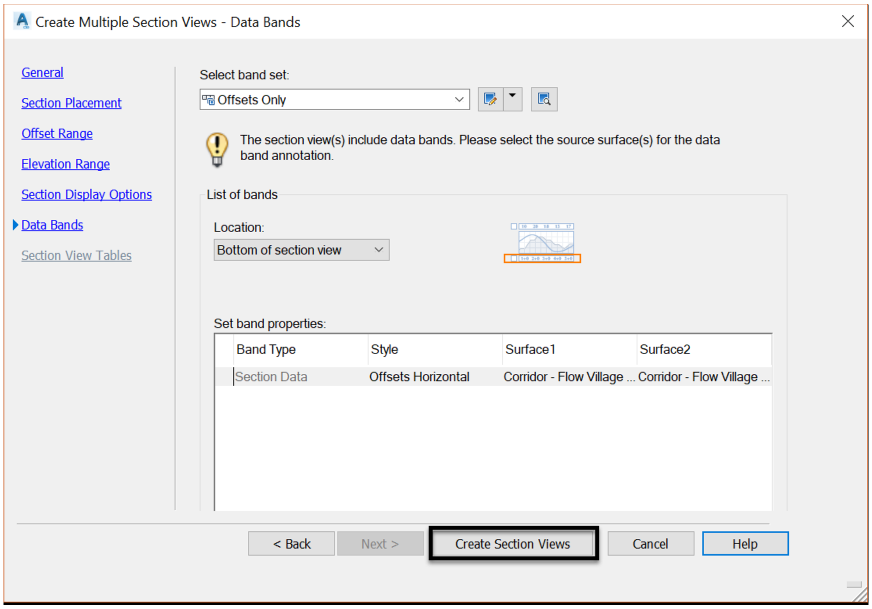

6. On the **Sections Display Options** page, specify the sampled items we would like to display in the section views. Let's choose to display all of them and click on **Next**.




NOTES

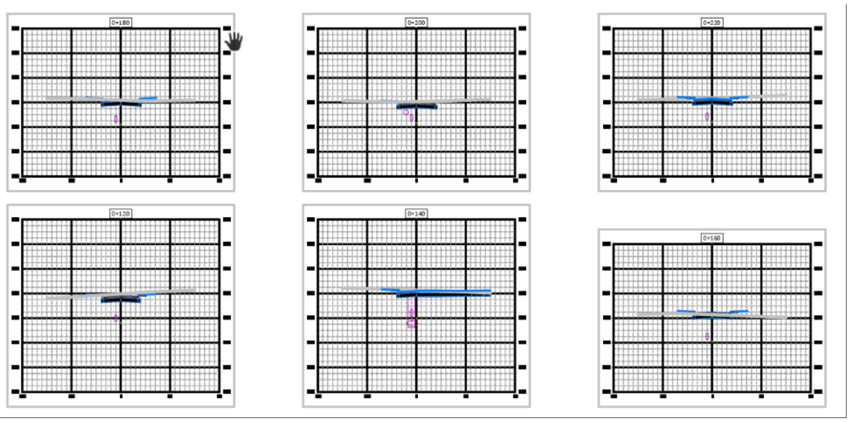
NOTES

7. On the **Data Bands** page, define the properties and locations of the data bands to attach to each section view. This can be information such as surface elevations, pipe data, and the like. The last page would have been the **Section View Tables** where we could set volume table properties for the section view. However, since we haven't calculated the volumes yet, that option is greyed out. So, click on **Create Section Views** to create the sections.




8. Next, we are prompted at the command line to specify the origin of the section views.


9. Click somewhere on the screen to create the section views. Here, we see an example of sections of the existing and final surfaces:

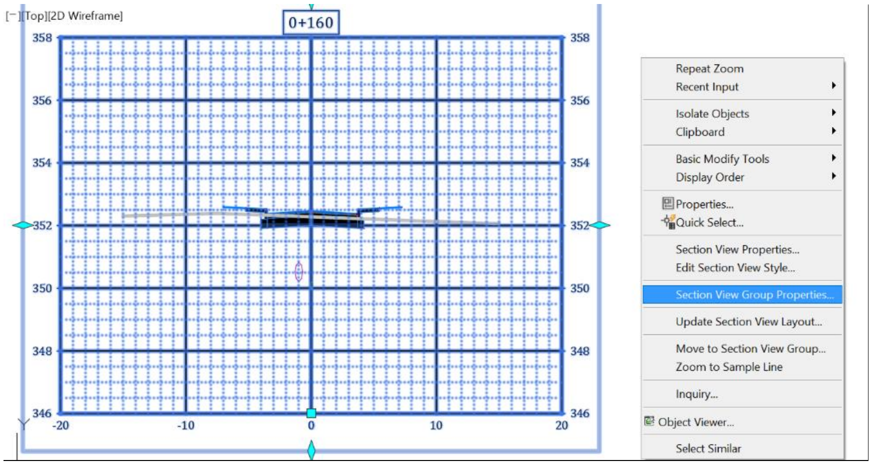


## 11.5 Sections view group properties

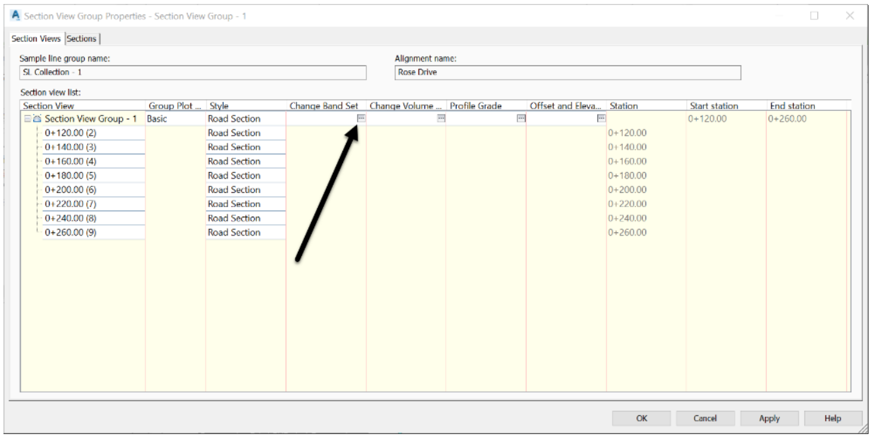
## 11.6 Bands

It would be convenient to show some information along the sections. That's where **Bands** come in. Most of the time, we attach them as tables associated with each section. We can use them to display design information, including profile elevations, cut and fill data, pipe networks, and much more.

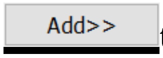
1. Let's add some bands to show the elevation information for the proposed and existing surface along with cut and fill depths. To do that,
2. Select a cross-section view,
3. Right click on **Section View Group Properties**.



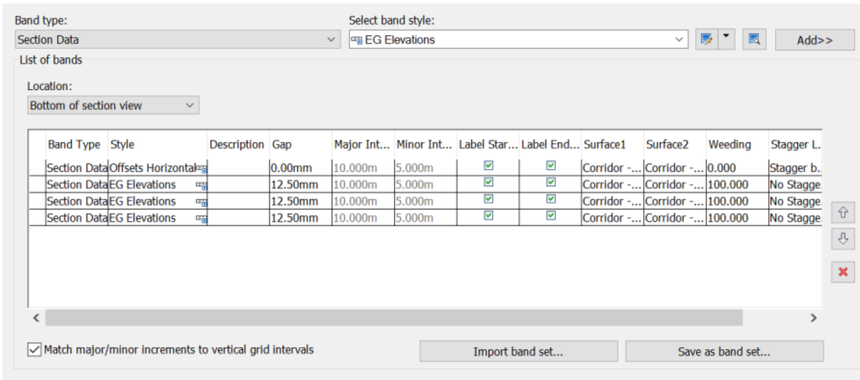

4. The **Section View Group Properties** window is the main place where you edit section view settings and styles. Here, you can create or modify Group plot styles, band sets, profile styles, view offsets, elevations, and sections styles. For example, let's see how to work with bands. Let's click on **Change Band Set**.



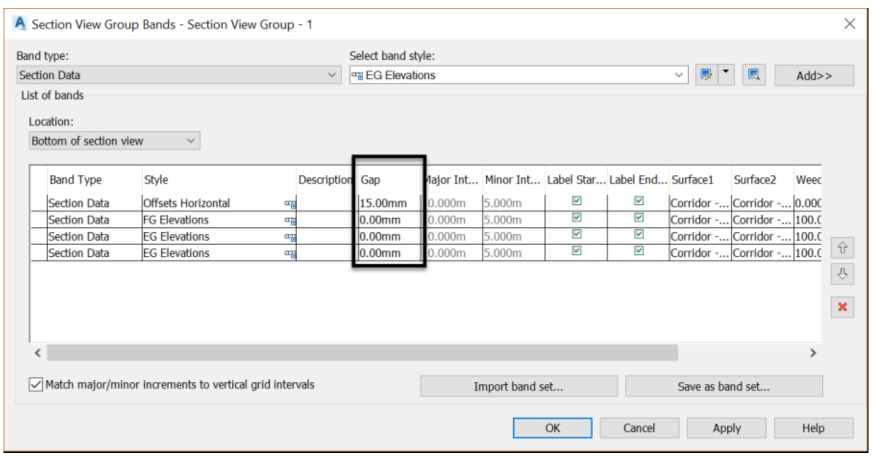

NOTES

5. Click on Add 

three times to add bands. We will have four in total, one for the stations, one for the existing ground, another one for the proposed ground and the last one for cut and fill depths.



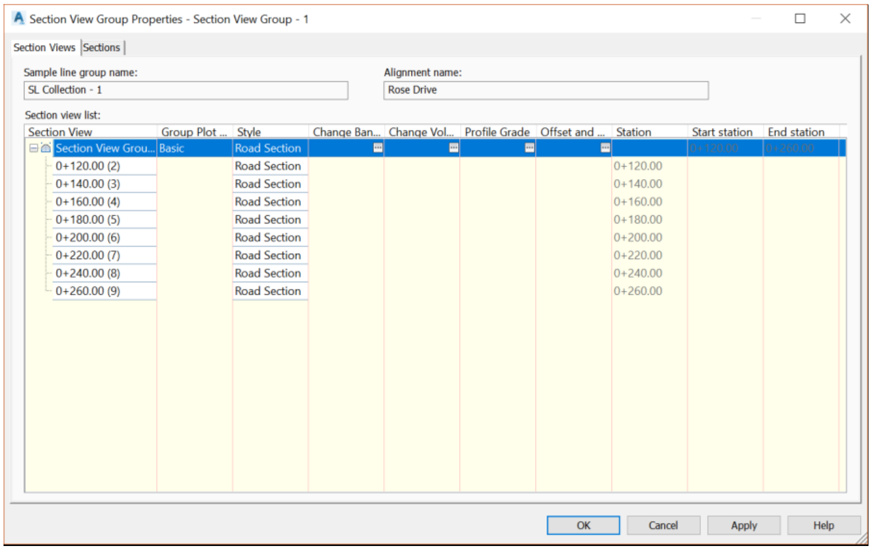

6. In the **Gap** column, change values to zero for all bands, except the offset horizontal.  We will set it at **15mm** or **0.6in**, to put it below the section border margin.



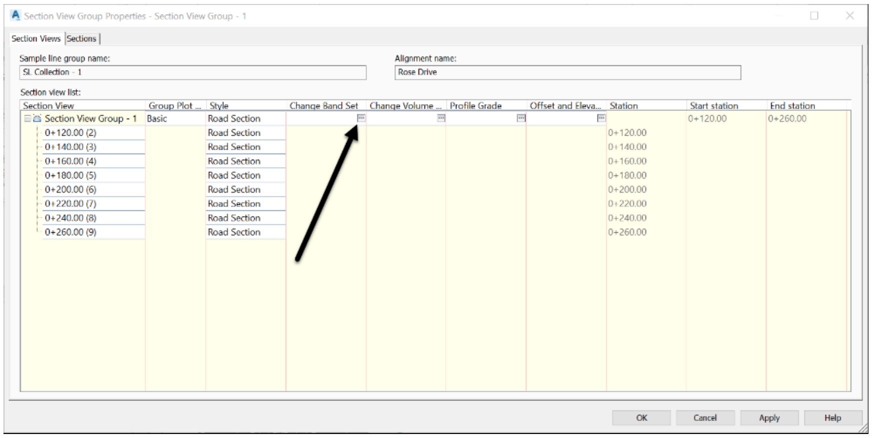

7. Click **OK** twice to preview the bands. You will notice that more adjustments are needed to the bands. To do that, we have several options such as changing text sizes, widths or heights of the bands, or increasing the spacing between sections.

NOTES

8. Let's start by adjusting the width of the bands. Return to the **Section View Group Properties** window.




9. Open the **Change Band Set** page.

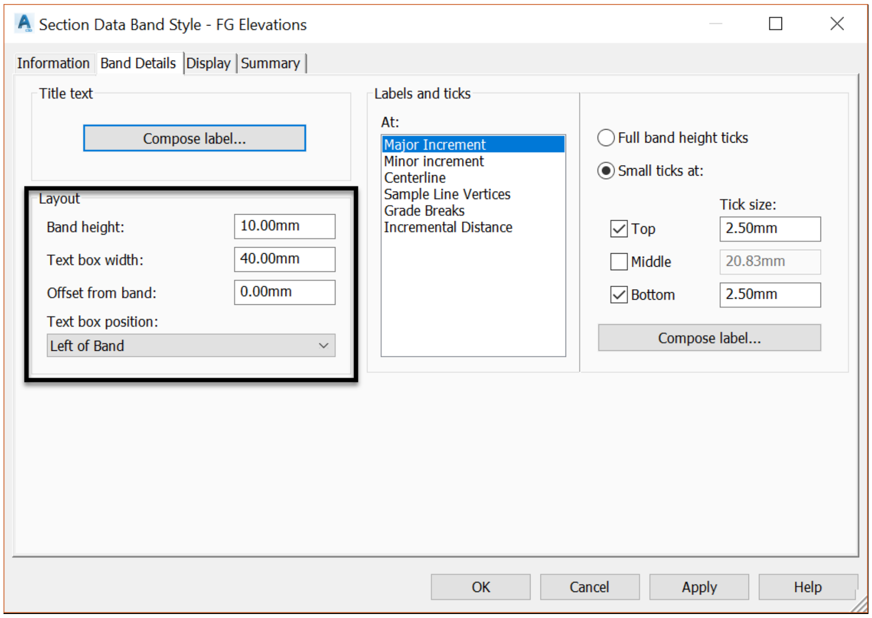



10. To adjust the widths and heights of a band, for example, the **Offsets Horizontal** one, click on the small band style symbol to its right and edit it in the **Pick Band Style** dialog box.

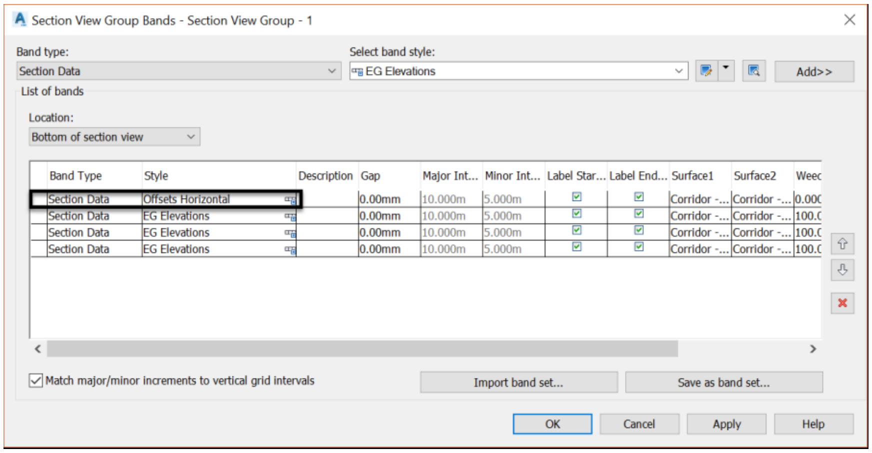

NOTES

11. On the **Band Details** tab, in the **Layout** section, assign

- ☐ a height of **10mm** or **0.4in,**
- ☐ a text box width of **40mm** or **1.6in** and
- ☐ a **0** for **offset from band**.

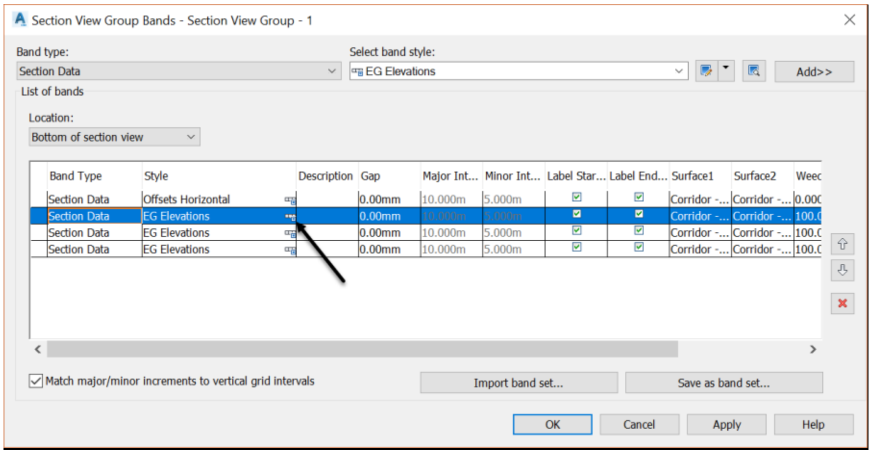



12. The first band style is already set to **Offsets Horizontal**. So, we will leave that one as-is. It will represent the station values of the section view.

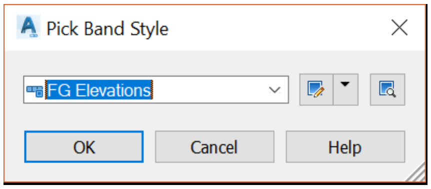



NOTES

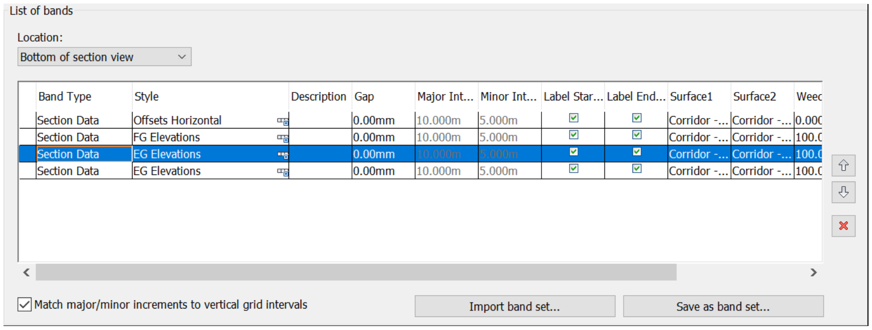
13. Next, change the **EG Elevations** style to **FG Elevations** by clicking on the small band style symbol to its right.




14. Then pick the **FG Elevations** Style in the **Pick Band Style** dialog box.

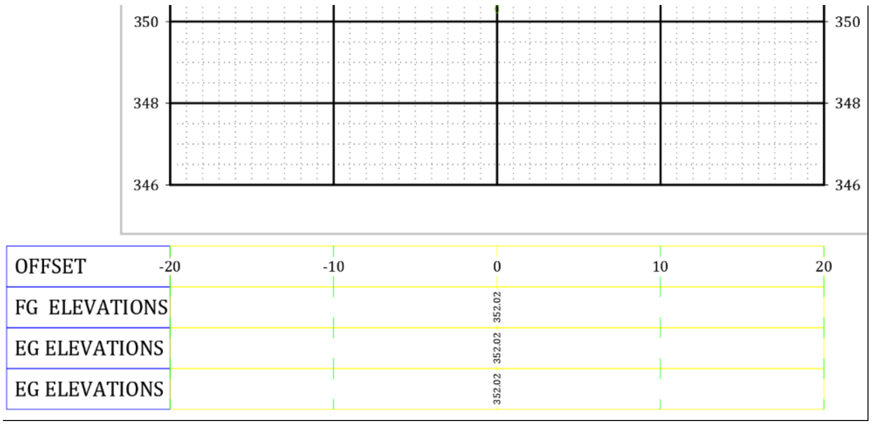



15. On the third line, we are already set with the **EG Elevations** styles, which we will use for the existing.

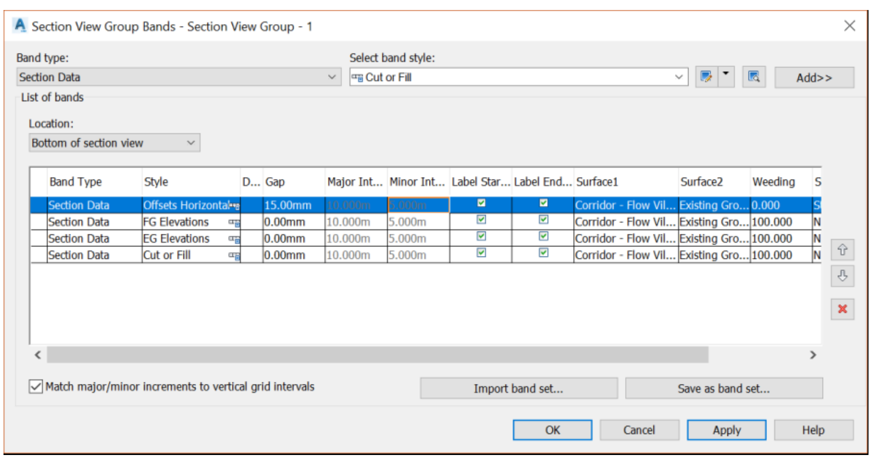



NOTES

16. On the last line, select the **Cut or Fill** band style.
17. Click **OK** on all the windows and observe the changes. We already see an improvement in the bands.

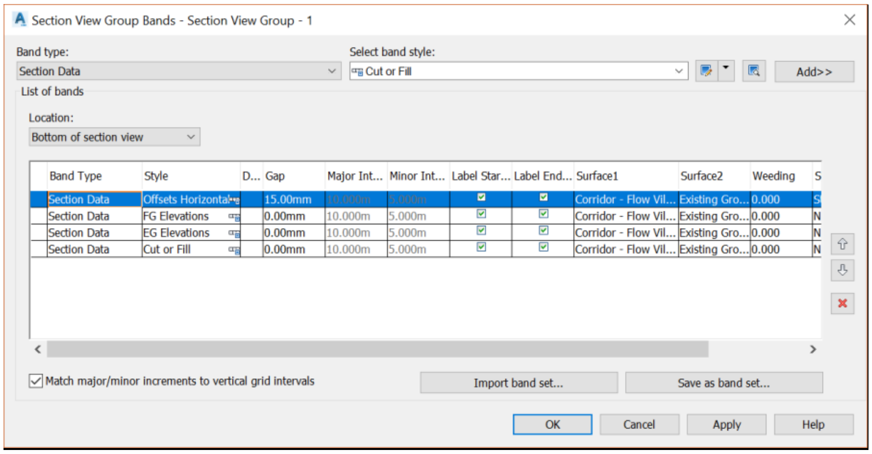



18. However, we still need to populate the band with elevation information at the major stations. Also, the line styles could also use some improvements. To populate the elevation data, we simply need to assign surfaces to the different bands, so they know which elevation to display. We also need to verify the **weeding** factor. Sometimes, it prevents data to be displayed. A quick note that the **weeding** option is used when we don't want to display the information at intervals too close. So, if we choose a weeding of **20m** or **75ft,** for example, the labels will be shown at a minimum every **20m** or **75ft**. All labels between this interval will be weeded out.
19. Return to the **Section View Group Properties** and open the **Bands** window.

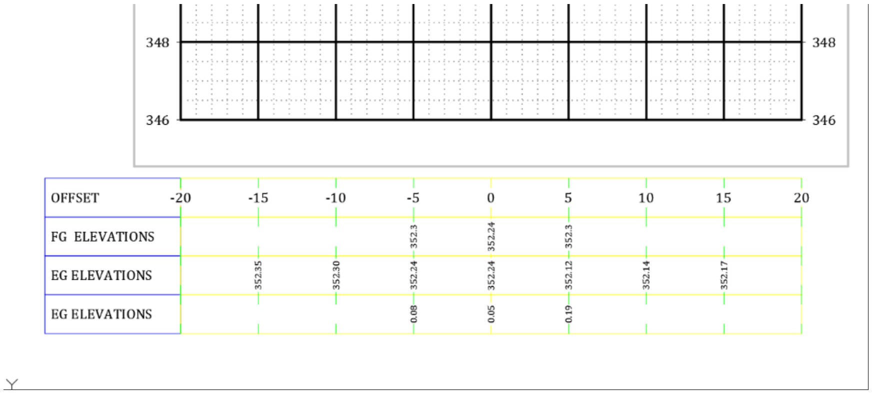



20. As expected, we need to assign the proper surfaces and also change the **weeding** from the default values to **zero**. Make sure all surfaces in the **Surface1** column are set the **Corridor Surface,** and the ones in the **Surface2** column are set to the **Existing Ground** surface.

NOTES




21. Click **OK** to close the **Section View Groups Bands** window. The bands are now filled with data, at all offsets, where we have an existing or proposed ground elevation.

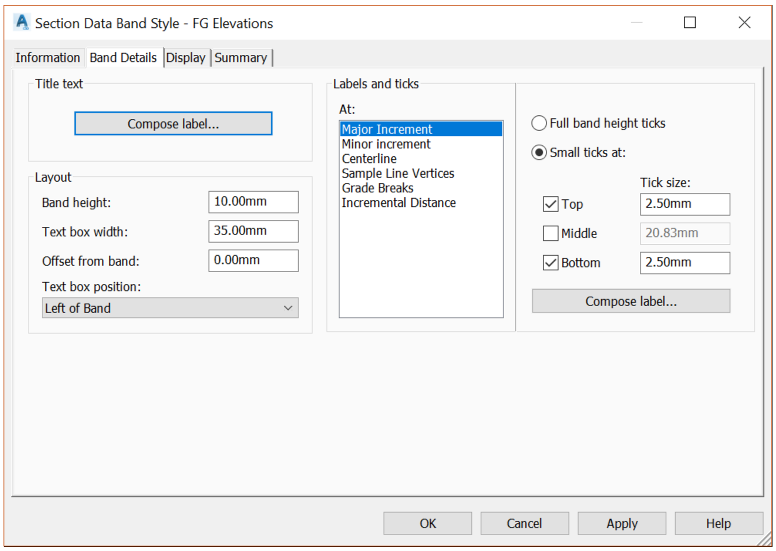



NOTES

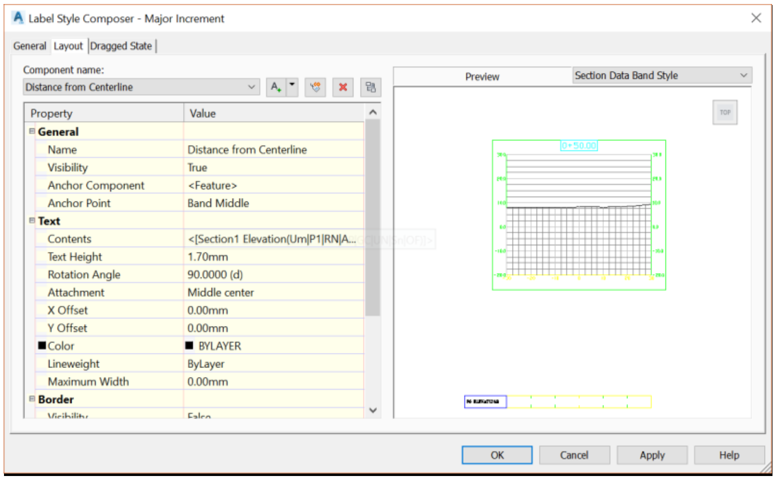
NOTES

22. Now let's go a little deeper in the band's rabbit hole and explore where they are set up to display the information they show. Return the **Section View Groups Properties** window and from there, open the Section **View Group Bands** window. Let's edit the **FG Elevations** style by clicking on the symbol to its right and edit the style.

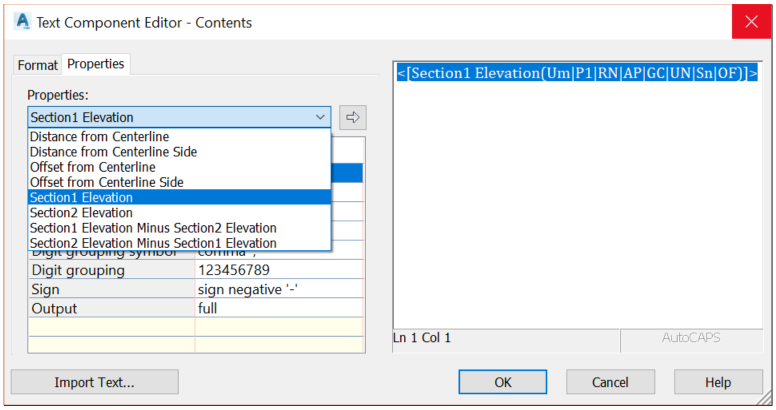
- On the **Information** tab, we have the usual style records.
- The **Band Details** tab is where most of the setup is done. The **Title text** section is where we set up the band title, the information that is displayed to the left of the band data. In this case, we have **Offsets**, **FG Elevations**, **EG elevations**, and so forth. To change it, simply click on **Compose label**; and change the content in the **Label Style Composer**.




- The **Label and ticks** area is where we decide the data to display, at specific locations. That can be **Major and Minor stations** or increments, centerline, sample line vertices, grade breaks, and incremental distances. Our labels are displayed at major stations. So, let's select **Major Increment** and click on **Compose label** to set the label content.



- In the **label style composer** window, the most important line is the **Contents**. So, click on there to edit the content of the label. In the **Text Composer Editor** window, we can choose different properties to display, including distances, offsets, **Section 1** (our proposed ground) or **Section 2** (the existing ground) and elevations. We can even estimate cut and fill by choosing to display the difference between Section 1 and 2.




- Click **OK** twice to return to the **Section Data Band Style** window. On the **Display** tab, we can edit the visibility and format settings for the section view component styles, including borders, titles, and labels.
- Click OK four times to close all the windows.

NOTES

# 12 VOLUMES

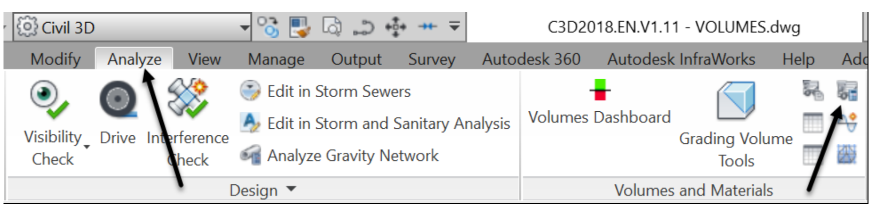
## 12.1 Introduction

When a design is complete, we need to estimate material quantities for construction cost estimation.

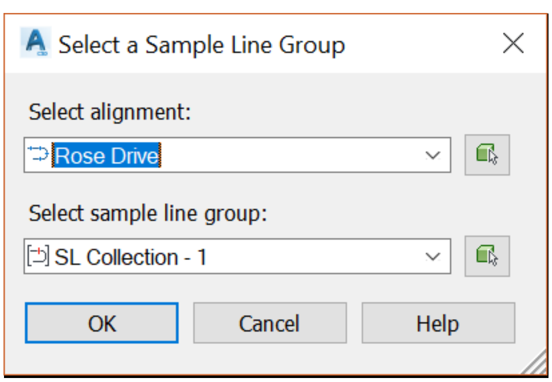
The quantity of each of the components and their management on-site will impact the bottom line of the project. So, it's in the designer's best interest to provide an accurate quantity estimation. In addition, operation, maintenance, and management of materials can also add a cost. For example, identifying cut and fill material locations along a road would enable better earthworks cost management. For instance, we can identify borrow and storage locations. Civil 3D provides tools to both estimate quantities and locate them, including asphalt, concrete, gravel base, subbase, earthworks, and more. This is done by using surfaces or subassembly structures.

## 12.2 Material List

1. Open the **12.01-Volumes dwg** file in the **Lesson 12** practice folder.
2. Activate the **Analyze** tab and run the **Compute Material** command.



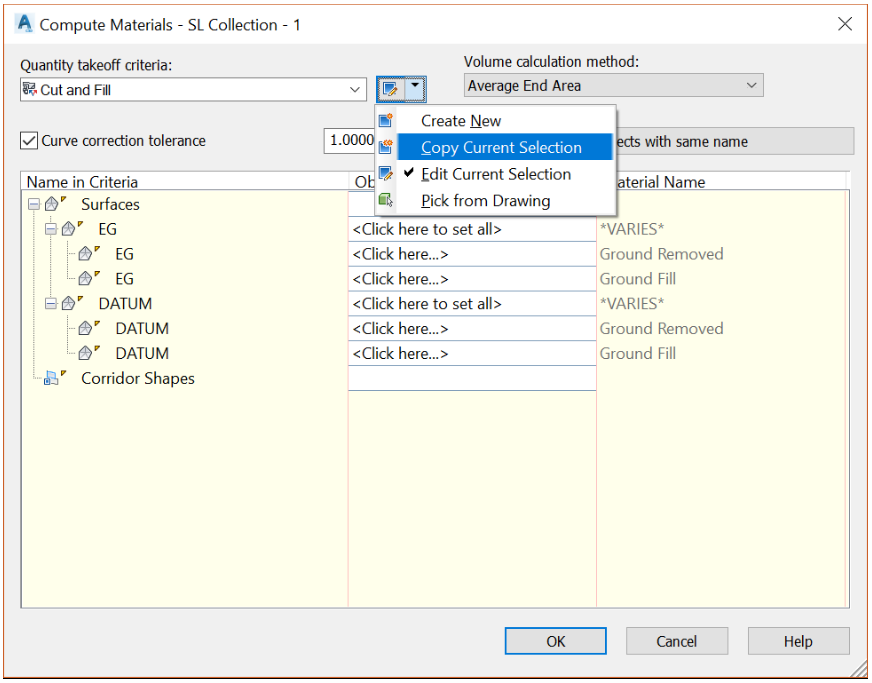

3. In the **Select Sample Line Group** window, select the **Rose Drive** alignment and the associated sample line group, then click on **OK.**



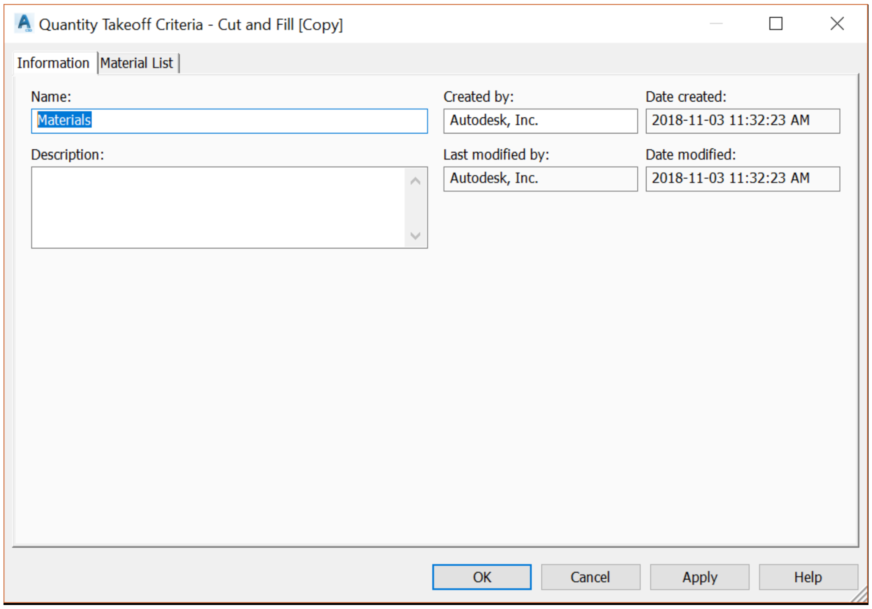

NOTES

NOTES

4. In the **Compute Material** window, we are provided with a default material list called Cut and Fill, courtesy of our default template. However, let's copy and modify it to suit our needs.

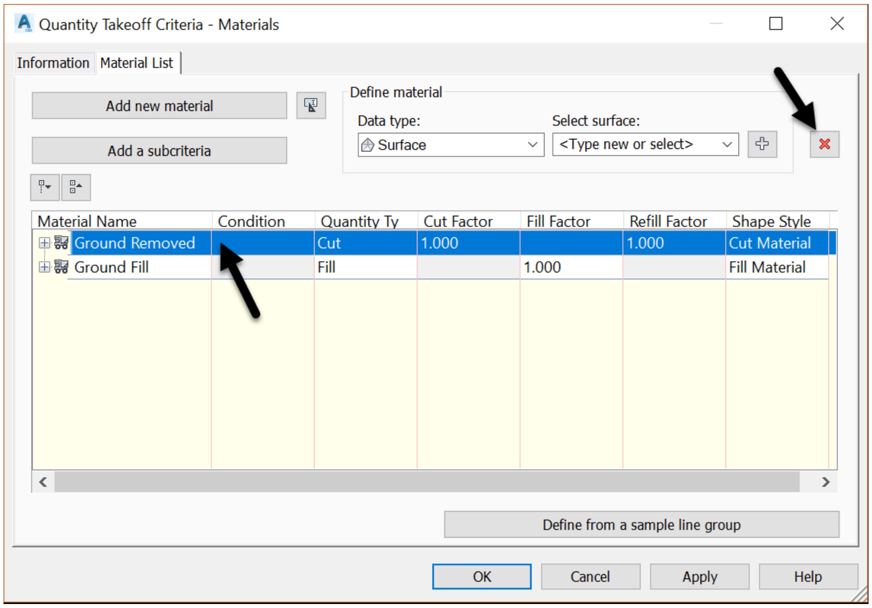



5. First, let's change the name, as we are not only interested in Cut and Fill material. We would also like to estimate other materials. So, in the **Quantity Takeoff Criteria** window, switch to the **Information** tab and change the **Name** simply to **Materials.**

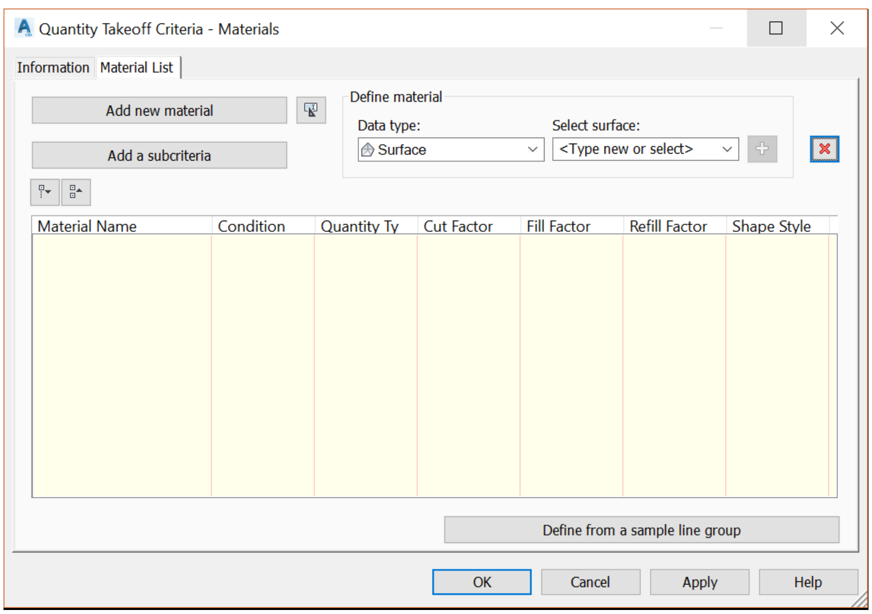


NOTES

6. Now switch to the **Material List** tab. Remove the materials already on the list. We need to learn how to create them by ourselves. Select each line item and click on the delete icon to remove it.

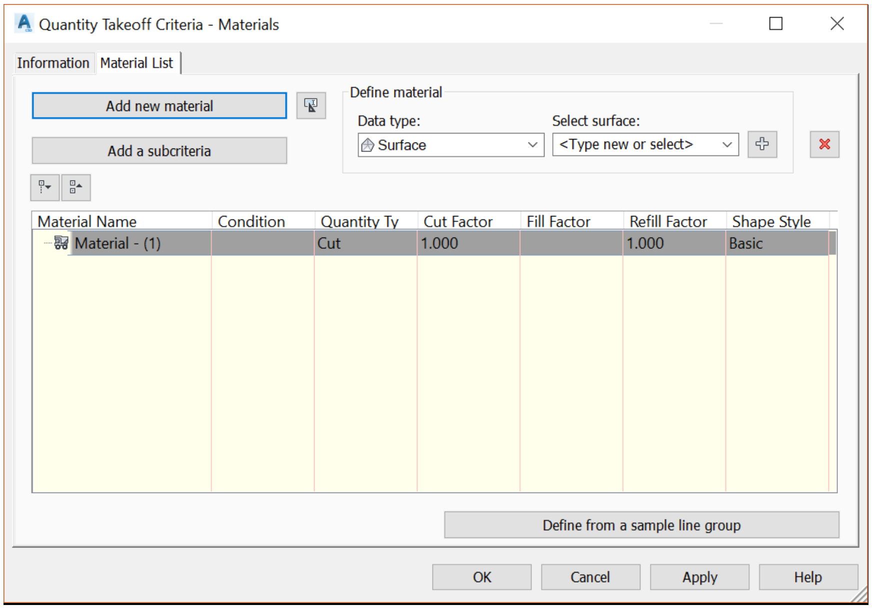



7. Now, the list of materials must be completely empty.

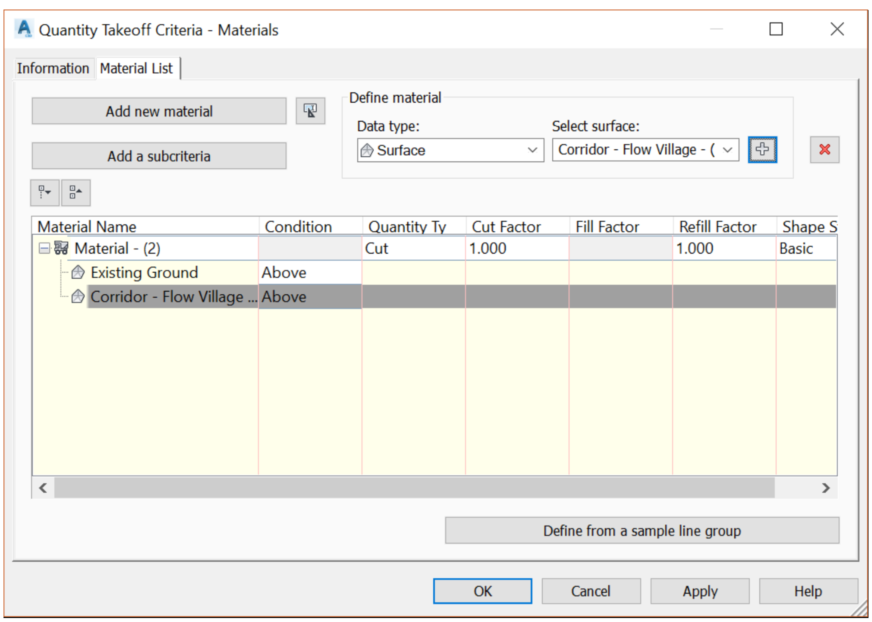


NOTES

8. Next, let's start creating our list. Click on **Add new material.**




9. Then choose **Surface** in the **Data Type** drop-down and choose the **Existing Ground** in the **Select a surface** drop-down menu. Next, click on the “**+**” icon to add the surface to the material we have just created. Repeat this process by choosing, this time, the **Corridor Flower Village** Surface.



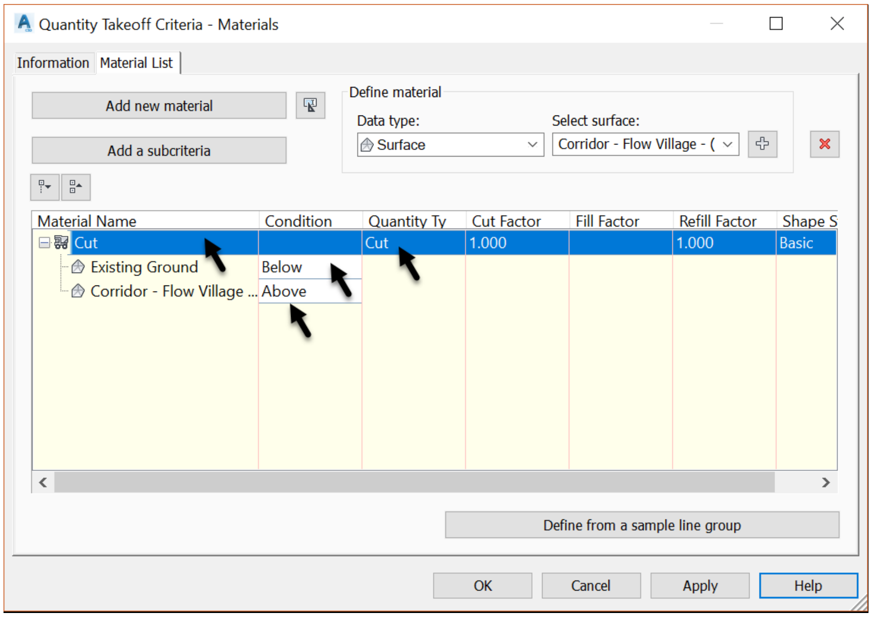

10. Now, we need to define the material by specifying the **Condition** that defines it. The condition is the criteria on which to base the calculation. Among the choices of condition, we have the following options:

- **Above:** which means that an area above this surface is included in the calculation.
- **Below:** means that the area below this surface is included in the material definition.

NOTES

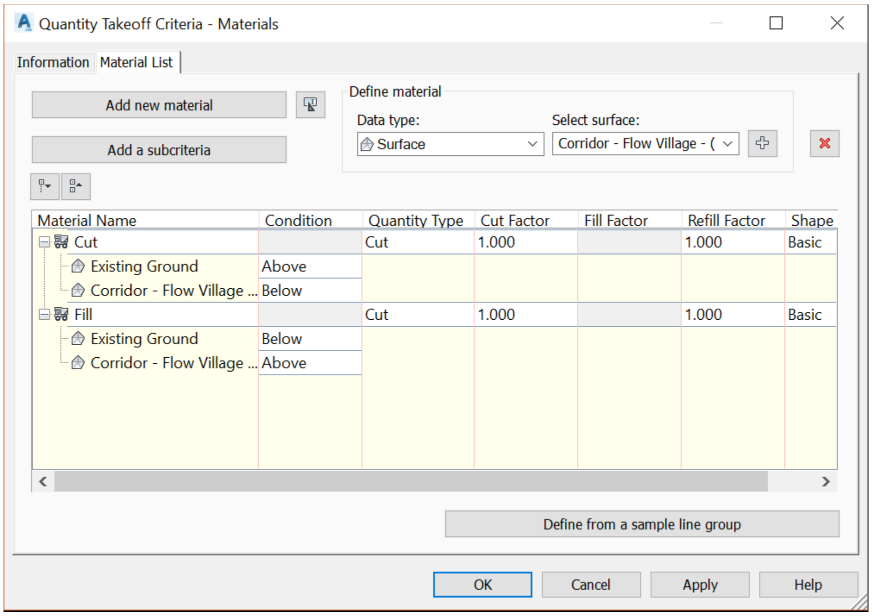
11. Let's use the condition definitions to create a **Cut material**. We have a **Cut material** whenever a material needs to be removed, meaning the proposed surface is below the existing surface. So, the condition is that the material is **below** the **Existing surface** and **above** the **Propose Surface**. Using the definition of **Conditions**, we can define the **Cut** material.

   - We first need to **add a new material** and call it **Cut**.
   - Then, we will add the two surfaces, proposed and existing.
   - Next, we need to change conditions to put:
     1. the **existing** ground as **Below;** and
     2. **proposed** surface as **Above.**
   - Lastly, we need to change the **Quantity Type** to **Cut**.

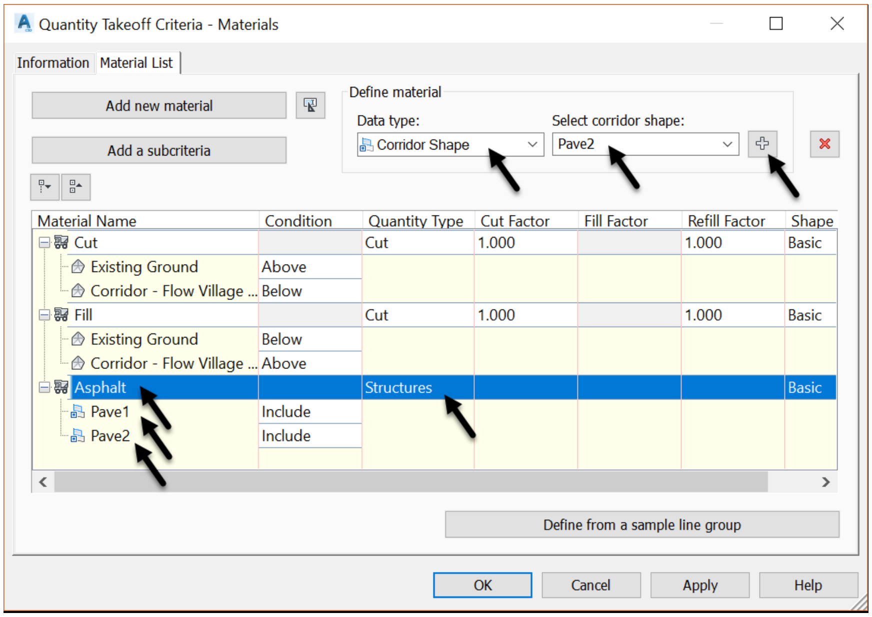


12. Use the previous steps to add the definition for the **Fill material.** We have fill when the proposed surface is higher than the existing ground and dirt must be hauled in to fill the additional void. Using the definition of **Conditions**, we can define the fill material.

- First, **add a new material** and call it **Fill**.
- Then, add the same two surfaces, proposed and existing.
- Next, we need to change conditions to the contrary of the cut material, and
- lastly, we need to change the **Quantity Type** to **Fill**.



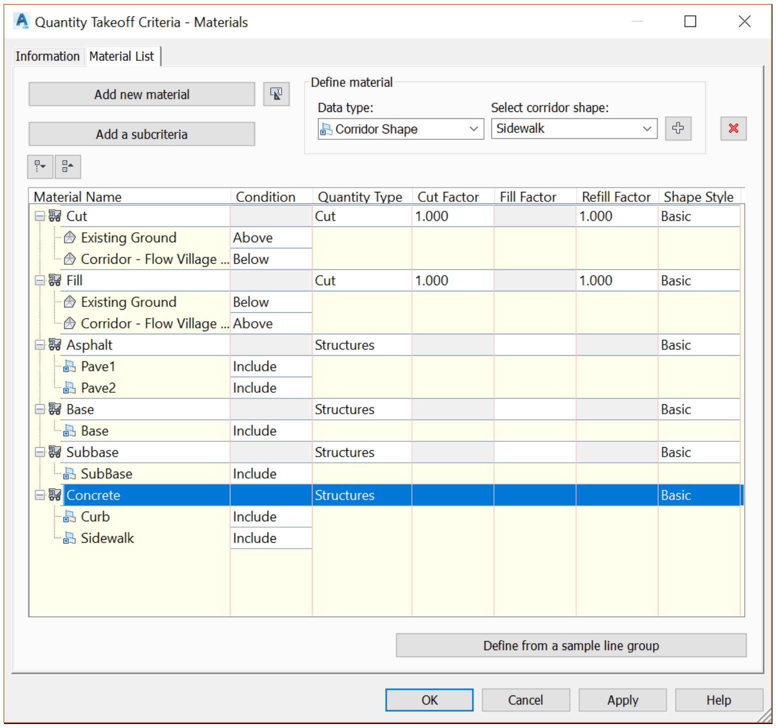

13. Now that we finished the earthworks materials, we need to define the structure type materials. That will include asphalts, concretes, and others. Let's start with the asphalt. We have two lifts of asphalts, **pave 1** and **pave 2**. We can estimate the materials separately or use the include option to combine them.




NOTES

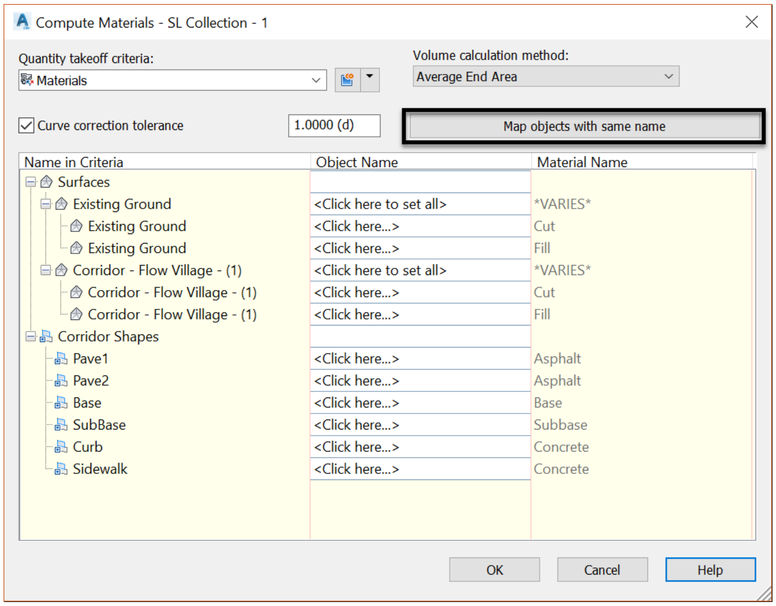
14. Use the previous steps to add the following list of materials, in addition to the **Cut** and **Fill**, that we already did:

- Asphalt, which will have two lifts, pave 1 and pave 2
- The granular base
- The subbase material
- And the concrete composed with curb and sidewalk materials.

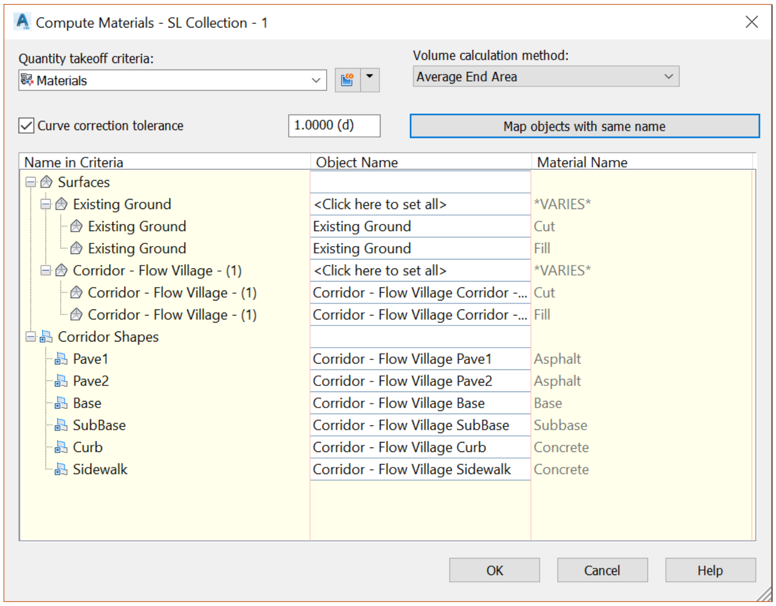



15. Click **OK** to close the **Quantity take Off Criteria** window.

16. Next, on the **Compute Materials** page, we need to **Map Objects with name,** to match the criteria and the drawing's surface or corridor shape names.



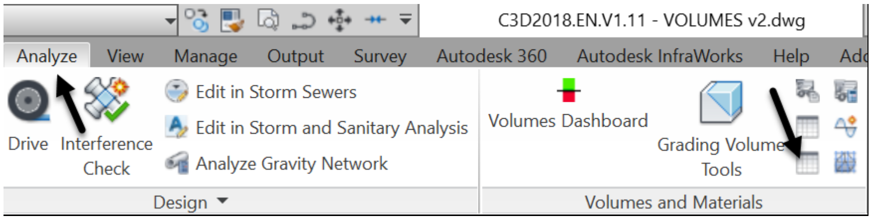

NOTES


Compute Materials - SL Collection - 1
Quantity takeoff criteria:
Materials
Volume calculation method:
Average End Area
Curve correction tolerance
1.0000 (d)
Map objects with same name
Name in Criteria
Object Name
Material Name
Surfaces
Existing Ground
<Click here to set all>
*VARIES*
Existing Ground
Existing Ground
Cut
Existing Ground
Existing Ground
Fill
Corridor - Flow Village - (1)
<Click here to set all>
*VARIES*
Corridor - Flow Village - (1)
Corridor - Flow Village Corridor -...
Cut
Corridor - Flow Village - (1)
Corridor - Flow Village Corridor -...
Fill
Corridor Shapes
Pave1
Corridor - Flow Village Pave1
Asphalt
Pave2
Corridor - Flow Village Pave2
Asphalt
Base
Corridor - Flow Village Base
Base
SubBase
Corridor - Flow Village SubBase
Subbase
Curb
Corridor - Flow Village Curb
Concrete
Sidewalk
Corridor - Flow Village Sidewalk
Concrete
OK
Cancel
Help

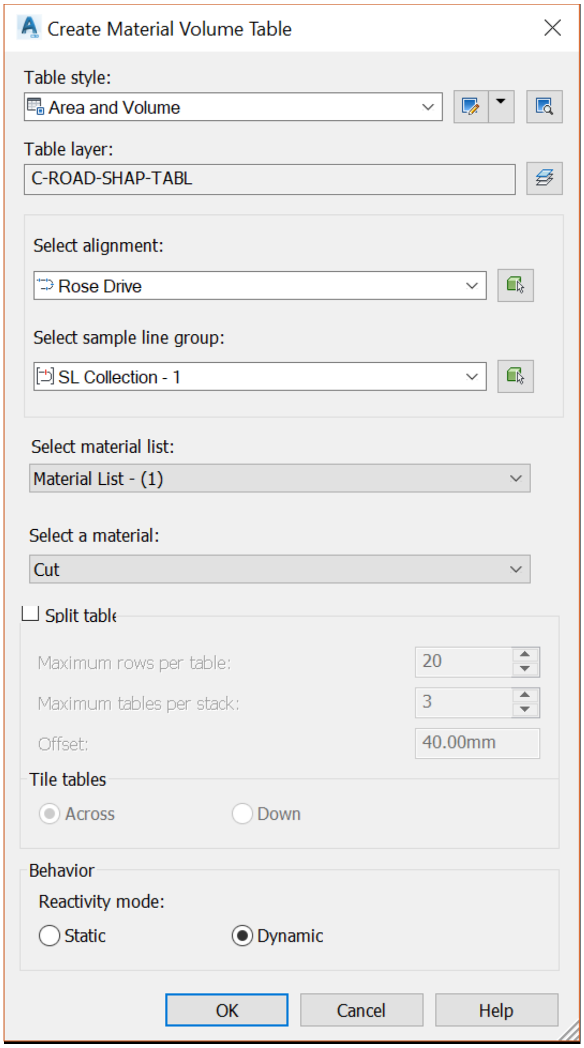
NOTES

## 12.3 Material Tables

1. Click **OK** to close the window.
2. The volumes have now been computed. All we have left to do is display them. Let's generate the material tables. On the **Analyze** tab, run the **Material Table** command.




3. Let's first display the **Cut** materials table.




NOTES

4. Click **OK** and when prompted, click somewhere on the screen to create the material table.

| Material Table | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 0.26 | 2.46 | 2.46 |
| 0+140.00 | 1.06 | 11.23 | 13.69 |
| 0+160.00 | 2.90 | 39.33 | 53.02 |
| 0+180.00 | 1.11 | 40.10 | 93.12 |
| 0+200.00 | 0.24 | 13.52 | 106.64 |
| 0+220.00 | 0.40 | 6.41 | 113.04 |
| 0+240.00 | 0.90 | 12.97 | 126.02 |
| 0+260.00 | 0.00 | 8.85 | 134.87 |
| 0+280.00 | 0.00 | 0.00 | 134.87 |

5. Use the previous steps to produce the list of materials for **Fill, asphalt, base, subbase,** and **concrete**.

| Material Table | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 0.26 | 2.46 | 2.46 |
| 0+140.00 | 1.06 | 11.23 | 13.69 |
| 0+160.00 | 2.90 | 39.33 | 53.02 |
| 0+180.00 | 1.11 | 40.10 | 93.12 |
| 0+200.00 | 0.24 | 13.52 | 106.64 |
| 0+220.00 | 0.40 | 6.41 | 113.04 |
| 0+240.00 | 0.90 | 12.97 | 126.02 |
| 0+260.00 | 0.00 | 8.85 | 134.87 |
| 0+280.00 | 0.00 | 0.00 | 134.87 |

| Material Table | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 2.10 | 21.24 | 21.24 |
| 0+140.00 | 0.58 | 26.28 | 47.52 |
| 0+160.00 | 0.00 | 5.87 | 53.39 |
| 0+180.00 | 0.08 | 0.82 | 54.21 |
| 0+200.00 | 0.68 | 7.59 | 61.80 |
| 0+220.00 | 0.73 | 14.03 | 75.83 |
| 0+240.00 | 0.16 | 8.90 | 84.72 |
| 0+260.00 | 0.00 | 1.65 | 86.38 |
| 0+280.00 | 0.00 | 0.00 | 86.38 |

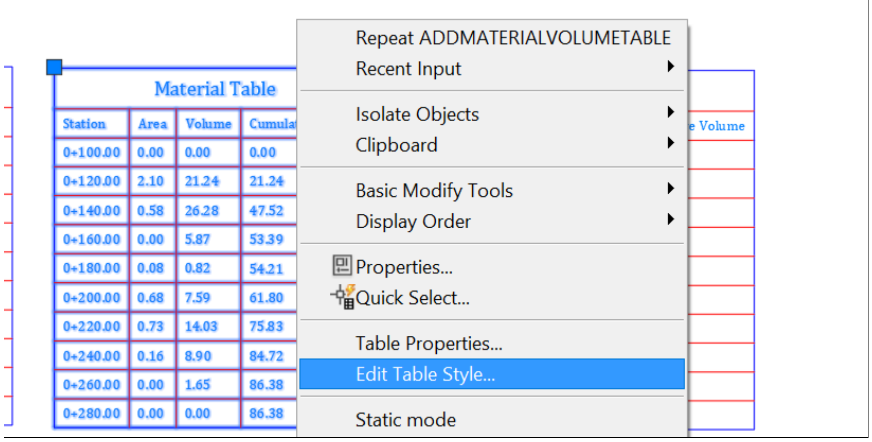
| Material Table | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 0.35 | 3.50 | 3.50 |
| 0+140.00 | 0.92 | 11.76 | 15.26 |
| 0+160.00 | 0.36 | 12.75 | 28.00 |
| 0+180.00 | 0.36 | 7.20 | 35.20 |
| 0+200.00 | 0.36 | 7.20 | 42.40 |
| 0+220.00 | 0.36 | 7.20 | 49.61 |
| 0+240.00 | 0.36 | 7.20 | 56.81 |
| 0+260.00 | 0.00 | 3.60 | 60.41 |
| 0+280.00 | 0.00 | 0.00 | 60.41 |

| Material Table | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 0.70 | 7.00 | 7.00 |
| 0+140.00 | 1.85 | 23.51 | 30.51 |
| 0+160.00 | 0.72 | 25.49 | 56.01 |
| 0+180.00 | 0.72 | 14.40 | 70.41 |
| 0+200.00 | 0.72 | 14.40 | 84.81 |
| 0+220.00 | 0.72 | 14.40 | 99.21 |
| 0+240.00 | 0.72 | 14.40 | 113.62 |
| 0+260.00 | 0.00 | 7.21 | 120.82 |
| 0+280.00 | 0.00 | 0.00 | 120.82 |

| Material Table | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 0.42 | 4.23 | 4.23 |
| 0+140.00 | 0.21 | 6.50 | 10.73 |
| 0+160.00 | 0.42 | 6.35 | 17.08 |
| 0+180.00 | 0.42 | 8.45 | 25.53 |
| 0+200.00 | 0.42 | 8.45 | 33.98 |
| 0+220.00 | 0.42 | 8.45 | 42.43 |
| 0+240.00 | 0.42 | 8.45 | 50.88 |
| 0+260.00 | 0.00 | 4.23 | 55.11 |
| 0+280.00 | 0.00 | 0.00 | 55.11 |

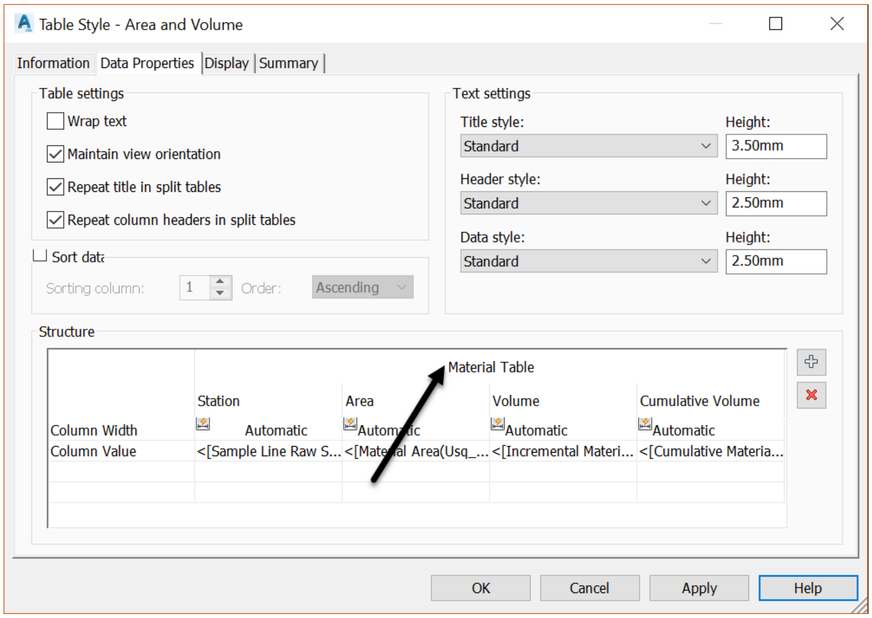
| Material Table | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 0.35 | 3.50 | 3.50 |
| 0+140.00 | 0.92 | 11.76 | 15.26 |
| 0+160.00 | 0.36 | 12.75 | 28.00 |
| 0+180.00 | 0.36 | 7.20 | 35.20 |
| 0+200.00 | 0.36 | 7.20 | 42.40 |
| 0+220.00 | 0.36 | 7.20 | 49.61 |
| 0+240.00 | 0.36 | 7.20 | 56.81 |
| 0+260.00 | 0.00 | 3.60 | 60.41 |
| 0+280.00 | 0.00 | 0.00 | 60.41 |

6. Next, like most Civil ivil 3d entities, you can change the display of the table by editing their style. It would be neat, for example, to display the name of each material in the table. To do that, select one of the tables, right-click and **Edit Table Style**.

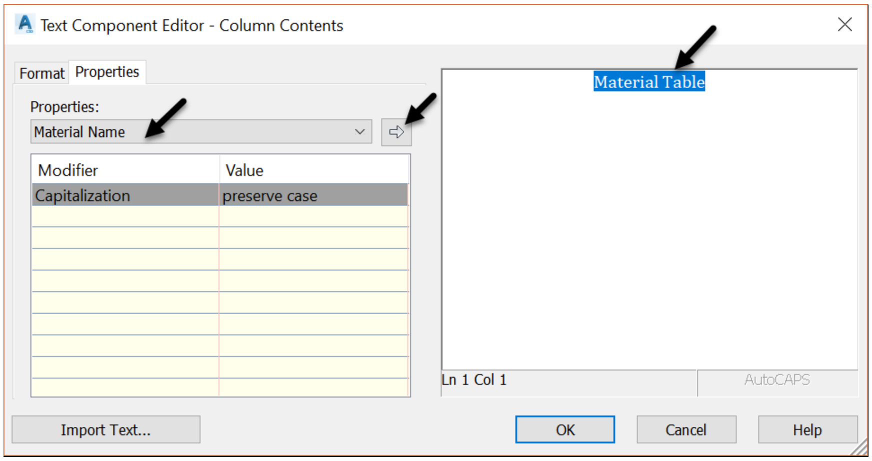


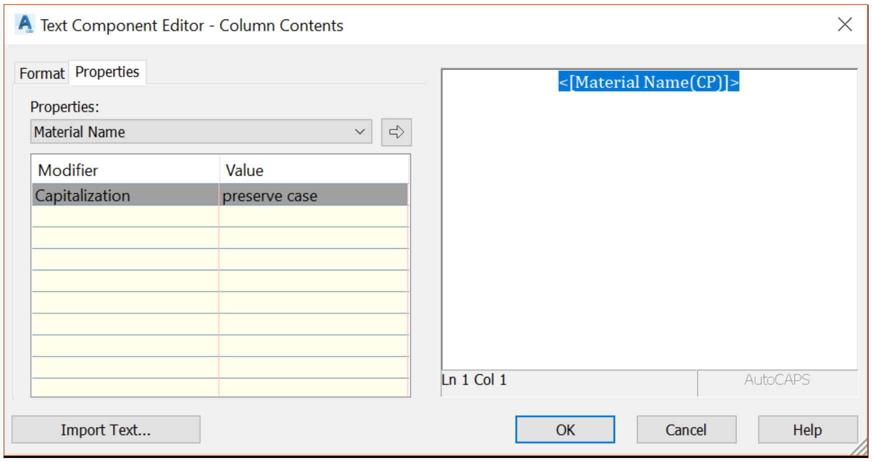

NOTES

7. In the **Table Style** editor, you can modify the content and appearance of the table. **Double-click** on the title of the table to open the **Text Component Editor**.




8. In the editor, highlight **Material Table**, then select **Material Name** in the **Properties** drop-down menu. Next, click on the horizontal arrow, to overwrite the current content.



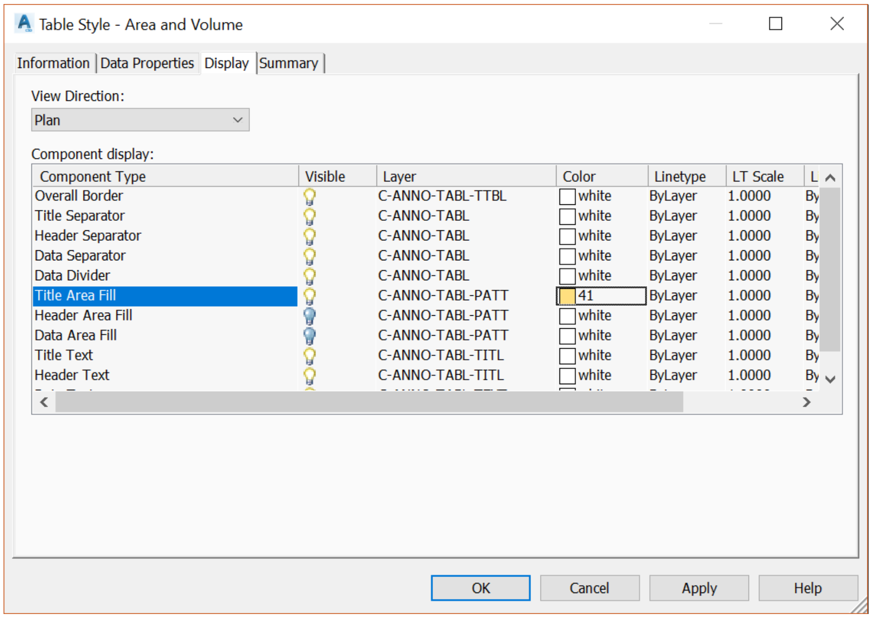




NOTES

9. Click to close the **Text Component Editor** window**.**

10. Now, switch to the **Display** tab.

- Select all the components by clicking on the first one, using the **Shift key** on the keyboard and clicking on the last one.
- Next, change all colours to White.
- Then, turn on **Title Area Fill** component and change its colour to 41, for example.




11. Now in the drawing, all tables have changed to display the name of the materials.

| Cut | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 0.26 | 2.46 | 2.46 |
| 0+140.00 | 1.06 | 11.23 | 13.69 |
| 0+160.00 | 2.90 | 39.33 | 53.02 |
| 0+180.00 | 1.11 | 40.10 | 93.12 |
| 0+200.00 | 0.24 | 13.52 | 106.64 |
| 0+220.00 | 0.40 | 6.41 | 113.04 |
| 0+240.00 | 0.90 | 12.97 | 126.02 |
| 0+260.00 | 0.00 | 8.85 | 134.87 |
| 0+280.00 | 0.00 | 0.00 | 134.87 |

| Fill | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 2.10 | 21.24 | 21.24 |
| 0+140.00 | 0.58 | 26.28 | 47.52 |
| 0+160.00 | 0.00 | 5.87 | 53.39 |
| 0+180.00 | 0.08 | 0.82 | 54.21 |
| 0+200.00 | 0.68 | 7.59 | 61.80 |
| 0+220.00 | 0.73 | 14.03 | 75.83 |
| 0+240.00 | 0.16 | 8.90 | 84.72 |
| 0+260.00 | 0.00 | 1.65 | 86.38 |
| 0+280.00 | 0.00 | 0.00 | 86.38 |

| Asphalt | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 0.35 | 3.50 | 3.50 |
| 0+140.00 | 0.92 | 11.76 | 15.26 |
| 0+160.00 | 0.36 | 12.75 | 28.00 |
| 0+180.00 | 0.36 | 7.20 | 35.20 |
| 0+200.00 | 0.36 | 7.20 | 42.40 |
| 0+220.00 | 0.36 | 7.20 | 49.61 |
| 0+240.00 | 0.36 | 7.20 | 56.81 |
| 0+260.00 | 0.00 | 3.60 | 60.41 |
| 0+280.00 | 0.00 | 0.00 | 60.41 |

| Base | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 0.70 | 7.00 | 7.00 |
| 0+140.00 | 1.85 | 23.51 | 30.51 |
| 0+160.00 | 0.72 | 25.49 | 56.01 |
| 0+180.00 | 0.72 | 14.40 | 70.41 |
| 0+200.00 | 0.72 | 14.40 | 84.81 |
| 0+220.00 | 0.72 | 14.40 | 99.21 |
| 0+240.00 | 0.72 | 14.40 | 113.62 |
| 0+260.00 | 0.00 | 7.21 | 120.82 |
| 0+280.00 | 0.00 | 0.00 | 120.82 |

| Concrete | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 0.42 | 4.23 | 4.23 |
| 0+140.00 | 0.21 | 6.50 | 10.73 |
| 0+160.00 | 0.42 | 6.35 | 17.08 |
| 0+180.00 | 0.42 | 8.45 | 25.53 |
| 0+200.00 | 0.42 | 8.45 | 33.98 |
| 0+220.00 | 0.42 | 8.45 | 42.43 |
| 0+240.00 | 0.42 | 8.45 | 50.88 |
| 0+260.00 | 0.00 | 4.23 | 55.11 |
| 0+280.00 | 0.00 | 0.00 | 55.11 |

| Asphalt | | | |
|---|---|---|---|
| Station | Area | Volume | Cumulative Volume |
| 0+100.00 | 0.00 | 0.00 | 0.00 |
| 0+120.00 | 0.35 | 3.50 | 3.50 |
| 0+140.00 | 0.92 | 11.76 | 15.26 |
| 0+160.00 | 0.36 | 12.75 | 28.00 |
| 0+180.00 | 0.36 | 7.20 | 35.20 |
| 0+200.00 | 0.36 | 7.20 | 42.40 |
| 0+220.00 | 0.36 | 7.20 | 49.61 |
| 0+240.00 | 0.36 | 7.20 | 56.81 |
| 0+260.00 | 0.00 | 3.60 | 60.41 |
| 0+280.00 | 0.00 | 0.00 | 60.41 |

NOTES

NOTES

12. What is even more interesting about the tables is that we can modify the profile and observe instant updates to the volume calculations. That's because the profiles were used to create the corridor surface. Which in turn was used to estimate the volumes. For example, if there is too much cut material, you can incrementally raise the profile, at strategic stations, until the cut and fill volumes balance out.