Munsys 15.1

DRAINAGE USER MANUAL

Munsys® Drainage User Manual

Visit Open Spatial on the internet: http://www.openspatial.com

Contents

Chapter 5 Maintaining drainage data

Chapter 1
Introducing the Munsys Drainage User Manual

Welcome to Munsys Drainage

Munsys Drainage is used to capture and maintain stormwater networks and drainage objects such as rivers, floodlines and dams. It is a user-friendly, easy to use geographic information management tool, which does not require GIS expertise to capture and manipulate data.

Munsys Drainage forms part of the Munsys product range, which comprises the following applications:

- Munsys Cable Route
- Munsys Cable Fiber
- Munsys Cadastral
- *Munsys Drainage*
- Munsys Electricity
- Munsys Roads
- Munsys Sewer
- Munsys Water
- Munsys Map Books
- Munsys Spatial Data Manager
- Munsys Management Console
- Munsys Query
- Munsys Lineage
- Munsys Scheduled Tasks

About this manual

The *Munsys Drainage User Manual* enables users to easily find their way around Munsys Drainage, and provides a conceptual overview of the functionality used in Munsys Drainage. For the purpose of this manual, we assume that you are familiar with:

- The business rules of the application
- AutoCAD Map
- Common GIS terminology

What's in this manual

The *Munsys Drainage User Manual* consists of the following chapters:

- Chapter 1 – Introducing the Munsys Drainage User Manual, gives an overview of this manual, and provides links to additional reading material.
- Chapter 2 – Getting acquainted with Munsys Drainage, gives an overview of Munsys Drainage and its various components.
- Chapter 3 – Querying drainage data from the database, describes how to query drainage data from the database.
- Chapter 4 – Creating drainage data, shows the user how to capture a stormwater network, add additional drainage objects, and post drainage data to the database.
- Chapter 5 – Maintaining drainage data, describes how to maintain existing drainage data.

Additional reading material

Before you start using this manual, we suggest that you first read the Munsys Concepts User Manual, which contains the following information:

- the generic functionality of the various Munsys applications, including the Query Palette and the Info Palette
- the generic query functionality that is used to query spatial data from the Oracle® database
- how to structure a query through GSC settings
- how to view spatial data
- how to work with Munsys Objects
- extras such as annotation, reporting and legend options

Conventions in this manual

The following table lists the typographical conventions used in this manual.

Text element	Example
Keys you press on the keyboard	CTRL, ENTER, DEL
Screen buttons	Click Close.
Folder paths	C:\Program Files\Open Spatial
Menu paths	choose Query > Clear Basemap.
Hypertext links to more information	http://www.openspatial.com
Text displayed/typed on the command line	Munsys Drainage
Dialog box/screen names	The Drainage Settings dialog box
Application functions	The Integrity Check function

Table 1 Munsys typographical conventions

Finding the information you need

To get help on

- general issues, select Help from the Munsys menu bar.
- an operation in progress, click the Help button on the dialog box.
- the latest support options, visit http://www.openspatial.com

Chapter 2
Getting acquainted with Munsys Drainage

Welcome to Munsys Drainage

Munsys Drainage is designed for the capture and maintenance of stormwater networks and additional drainage objects. Munsys Drainage caters for stormwater objects such as pipes, channels, and culverts, and for stormwater nodes such as curb inlets, grid inlets, and manholes. Additional drainage objects that can be captured along with the stormwater network include rivers, floodlines, dams and catchment areas.

Objects are typically placed along cadastral boundaries and network connectivity is generated after the objects have been captured. The Drainage toolbars and menus enable fast and efficient data capture and modification. Munsys Drainage also checks the integrity of spatial and attribute data before the data is posted to the database.

Launching Munsys and Munsys Drainage

To launch Munsys, do one of the following:

- Double-click the **Munsys Applications 15.1** icon on the Windows desktop.

- Choose **Start** > **Program Files** > **Open Spatial** > **Munsys 15.1** > **Munsys Applications 15.1**

Connecting to the Oracle database

Munsys uses Oracle as its data store. The advantage of using the Oracle Spatial or locator technology is that spatial and attribute data are captured and managed in a single database. This reduces processing overheads and eliminates the complexity of coordinating and synchronizing different sets of data.

The Munsys Applications support multiple Logons which permit users to logon to different databases such as Training, Test or Production databases by selecting a Logon Profile from a drop-down list. When the Connect function is selected, the logon credentials (excluding the password) are pre-populated based on last successful database connection.

The Logon Profile details are customizable and stored in the Current User's Registry Keys. By default, three Logon Profiles are defined, namely Logon1, Logon2 and Logon3. The Logon Profile descriptions can be changed from the default description to be more meaningful one by simply clicking in the text box next to the Logon Profile and overwriting the default value.

Important You have to be connected to the Oracle database before you can start working with Munsys Cadastral. If you try to launch any application without being connected to the database, you will be prompted to connect to the database first.

To connect to the Oracle database

1 For first time connection do one of the following:

- Choose **File** > **Database** > **Connect...**
- Click the **Connect to Database** button on the Munsys standard toolbar.

The Connect to Database dialog box is displayed.

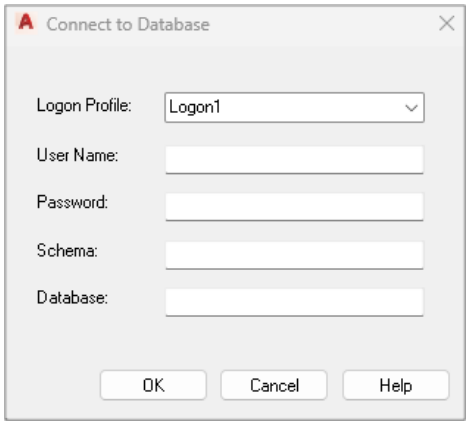


Figure 1 The Connect to Database dialog box

2 In the **Logon Profile** box, select a profile from the drop-down list.

Note The Logon Profile can be changed to a more meaningful description by clicking in the Logon Profile text box and overwriting the default description with a value not exceeding 25 characters, for example: TRAINING, TEST, STAGING or PRODUCTION.

3 In the **User Name** field, enter your user name.

4 In the **Password** field, enter your password.

5 In the **Schema** field, enter the schema name.

6 In the **Database** field, enter the database name.

7 Click **OK**.

On successful connection to the database, the last Logon Profile, User Name, Schema and Database details are saved in your Current User's Registry Key where they are used to pre-populate the Connect to Database dialog-box on next logon. Your password is never saved, and you will always be prompted to enter your Password.

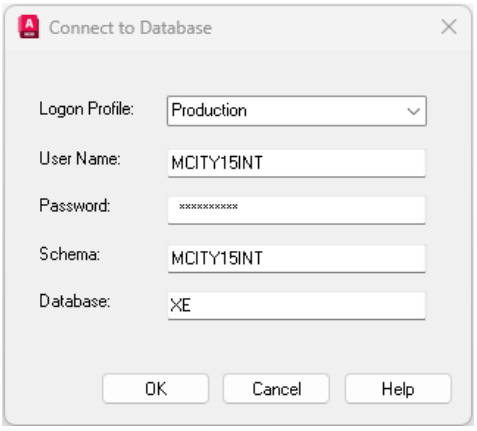


Figure 2 The Connect to Database dialog box with Logon Profile

Note The last used Logon Profile details, and the various Logon descriptions are stored in the registry key **Computer\HKEY_CURRENT_USER\Software\Open Spatial\Munsys 15.1\Applications\Logons**,

whilst the registry key **Computer\HKEY_CURRENT_USER\Software\Open Spatial\Munsys 15.1\Applications\Logons\[Logon1]** saves the individual User Name, Schema and Environment Name details per Logon Profile.

The administrator assigns you user rights to log on to the database, and will inform you what your user name, password, schema name, and database are. A successful connection to the database activates the appropriate functions on the menu bar and toolbars.

To launch Munsys Drainage

When you launch Munsys for the first time, the configured base map automatically loads. When you launch Munsys Drainage, the Munsys Drainage functions menu bar and toolbars are loaded.

1 To launch Munsys Drainage, do one of the following:

- Choose **File** > **Munsys Applications**.
- Click the **Munsys Applications** button on the Munsys standard toolbar.

The Available Applications dialog box is displayed.

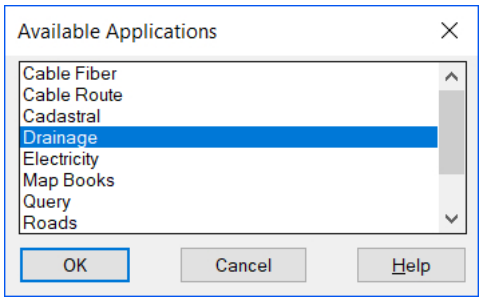


Figure 3 The Available Applications dialog box

2 From the list of available applications, select **Drainage,** and then click **OK**.

The Munsys Drainage functions are loaded.

The Munsys Drainage interface

The Munsys Drainage interface consists of the following:

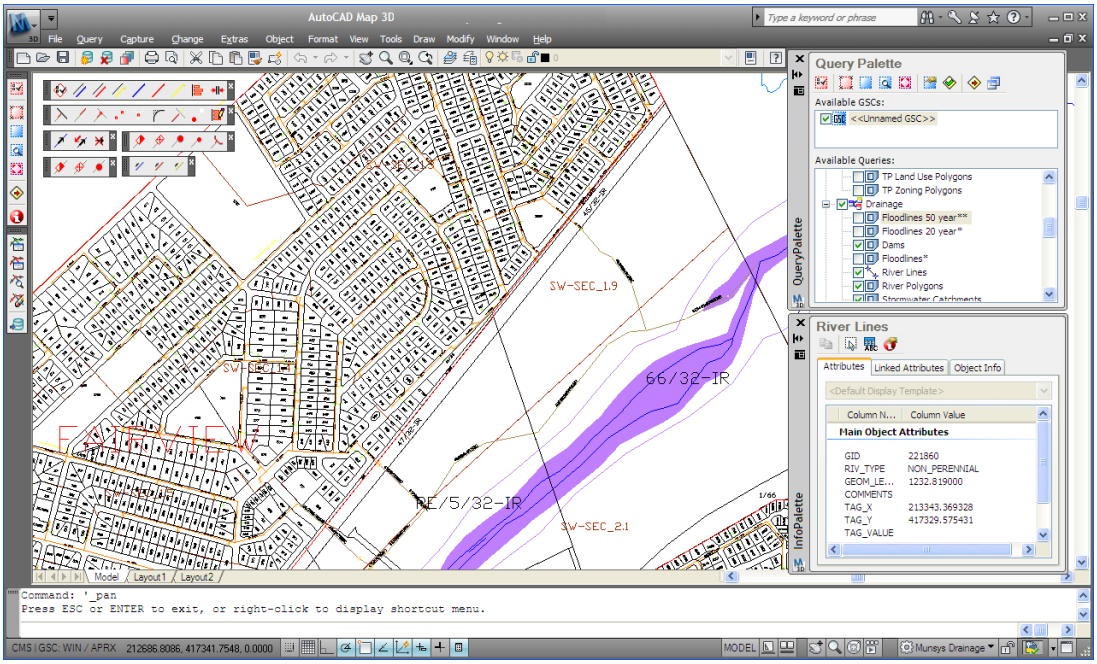


Figure 4 The Munsys Drainage interface

Munsys menu bar – contains the Munsys and AutoCAD Map menus. Munsys menus are defined by a menu file, which automatically loads when the Munsys application is launched.

Standard toolbar – contains frequently used AutoCAD Map buttons, standard Microsoft® buttons and Munsys buttons for connecting to the database and launching the various Munsys applications.

AutoCAD Map drawing area – this area is used to display Munsys data that is extracted from the database and AutoCAD Map drawings.

AutoCAD Map command line – displays prompts and messages.

AutoCAD Map status bar – displays information/drawing aids.

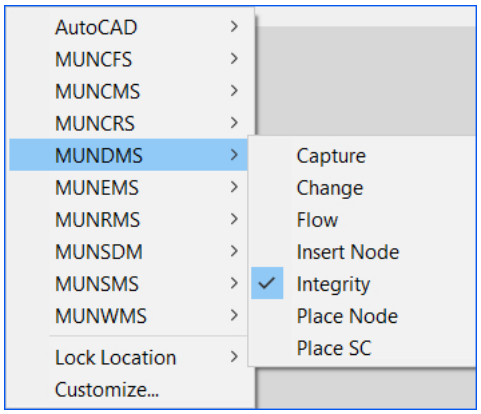
Munsys main toolbar – contains frequently used Munsys functions.Munsys Integrity toolbar – contains the Integrity Check and Post to Database functions.

Use this button	When you want to...
	...check drainage object integrity
	...check drainage network integrity
	...browse integrity markers

Use this button	When you want to...
	...erase integrity markers
	...post drainage data to the database

Table 5 The Munsys Drainage Integrity Toolbar

Munsys Drainage toolbars – Munsys Drainage has six application-specific toolbars that display when activated. To display the Drainage toolbars, right-click in the grey area on the right-hand side of the Munsys Standard toolbar.



Point to MUNDMS, and then select the toolbars that you want displayed one by one. The toolbars are displayed in the drawing area.

- The Munsys Drainage Capture toolbar contains the following buttons:

Use this button	When you want to...
	...specify drainage capture settings.
	...draw an offset pipe.
	...draw an offset culvert.
	...draw an offset channel.
	...draw a freehand pipe.
	...draw a freehand culvert.
	...draw a freehand channel.
	...place a drainage note.

Use this button	When you want to...
	...place dimension arrows.

Table 6 The Munsys Drainage Capture toolbar

- The Munsys Drainage Change toolbar contains the following buttons:

Use this button	When you want to...
	...extend a drainage object to a boundary.
	...extend a drainage object by distance.
	...extend and break a drainage object.
	...break a drainage object.
	...change a drainage object.
	...fillet a drainage object.
	...trim a drainage object.
	...move a node.
	...change a drainage note.

Table 7 The Munsys Drainage Change toolbar

- The Munsys Drainage Flow toolbar contains the following buttons:

Use this button	When you want to...
	...show flow direction.
	...change flow direction.
	...clear direction arrows.

Table 8 The Munsys Drainage Flow toolbar

- The Munsys Drainage Place Node toolbar contains the following buttons:

Use this button	When you want to...
	...place an endpoint curb inlet.
	...place an endpoint drop inlet.
	...place an endpoint manhole.
	...place an endpoint junction.
	...place an endpoint wingwall.

Table 9 The Munsys Drainage Place Node toolbar

- The Munsys Drainage Insert Node toolbar contains the following buttons:

Use this button	When you want to...
	...insert a nearest curb inlet.
	...insert a nearest drop inlet.
	...insert a nearest manhole.

Table 10 The Munsys Drainage Insert Node toolbar

The Munsys Drainage Label toolbar contains the following buttons:

Use this button	When you want to...
	...place a pipe label.
	...place a culvert label.
	...place a channel label.

Table 11 The Munsys Drainage Label toolbar

About drainage objects

The connection of stormwater pipes and nodes are used to build converging stormwater networks. Stormwater pipes, culverts and channels can be constructed at a specified offset distance from cadastral boundaries, by coordinate, or freehand. Stormwater pipe diameter increases as stormwater networks develop, for example, to reservoirs, rivers or dams. Catchment areas are bounded by a watercourse, draining into a river or dam.

Stormwater networks contain different types of objects:

- Stormwater pipes – pipes that run in the road reserves.
- Culverts – usually covered channels that carry stormwater below ground level to catchment areas.
- Channels – open waterways that carry stormwater to catchment areas.
- Curb inlets – an inlet on the side of the road, which gathers flowing rainwater.
- Drop inlets – an inlet that is level with the road surface, also known as a grid inlet.
- Manholes – found at every bend point in stormwater pipes and used for access to pipes or culverts.
- Junctions – bend points in the network.
- Wingwalls – an extension of a channel or pipe constructed to stop soil erosion around the inlet or outlet.

With Munsys Drainage, you can also capture the following drainage objects:

- Floodlines – an estimate of the level to which water may rise on average (for example, every 20 or 50 years).
- Catchments – an area from which rainfall drains into a river, reservoir or dam.
- Rivers – captured as lines or polygons.
- Dams – captured as polygons.
- Drainage notes – supply additional information about plans and drawings.
- Dimensions – supply additional information in terms of the location of a drainage object, relative to the cadastral boundary.
- Labels – contain dynamic text information that represent a specified column in the database.

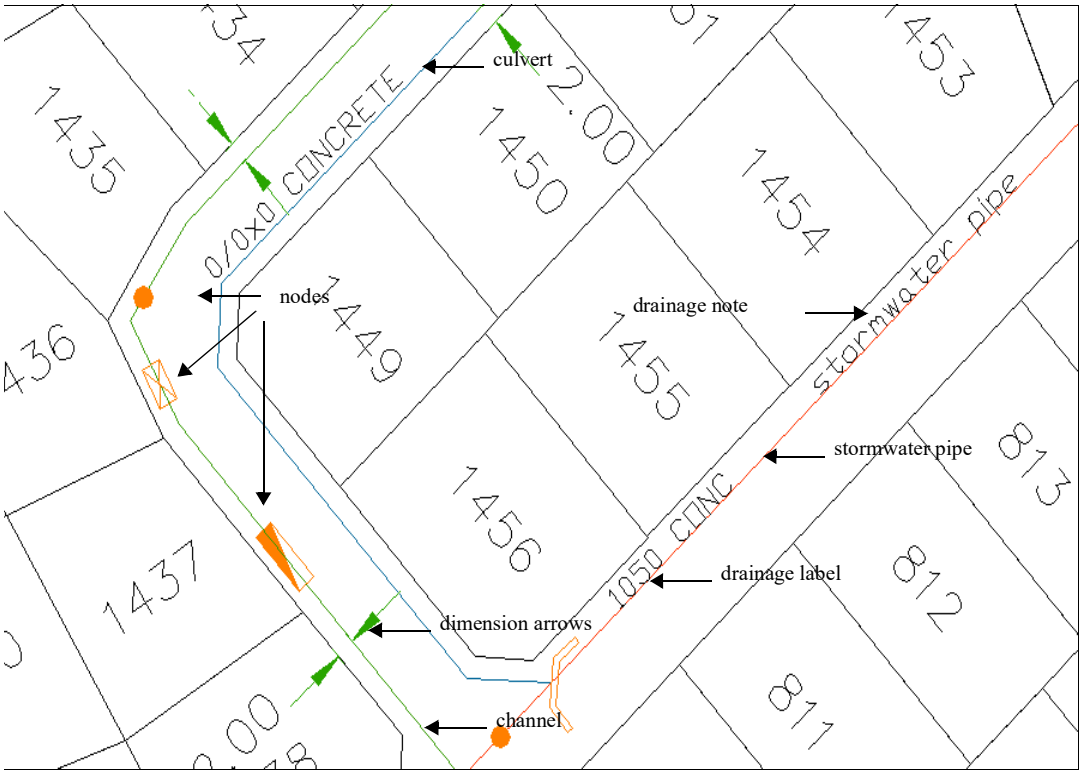


Figure 12 Drainage objects

Chapter 3
Querying drainage data from the database

Introduction

This chapter describes how to query drainage data from the database. Stormwater pipes, culverts, channels, service connections, nodes, notes, labels and dimensions are the stormwater objects that contribute to a stormwater network. Additional drainage objects include catchments, rivers, floodlines and dams. In this chapter, you will learn how to retrieve drainage data from the database with the Query menu. You can also use the Munsys Query Palette to assist in retrieving drainage objects from the database. The Query Palette is also used to create custom (user-defined) queries. You can also query a map page grid, which can be used to locate services based on the map page number, and is represented by a layout of polygons.

For more information about the Munsys Query Palette, refer to the *Munsys Concepts User Manual*.

Querying all drainage objects

This query retrieves all the drainage objects according to the current GSC from all the drainage tables in the database. The queried objects are formatted according to their system query settings, and each object is retrieved onto its respective layer, for example stormwater pipes are retrieved onto the SWPIPE layer. The Query All Drainage Objects function retrieves the following drainage objects:

Drainage object	Layer	Munsys object type
Stormwater pipe	SWPIPE	MunLine
Culvert	SWCULVERT	MunLine
Channel	SWCHANNEL	MunLine
Node	SWNODE	MunLine
Service Connections	SWSERV	MunLine
Symbols	SWSYM	MunLine
Pipe label	SWLABEL_PIPE	MunLabel
Culvert label	SWLABEL_CULVERT	MunLabel
Channel label	SWLABEL_CHANNEL	MunLabel
Dimension note	SWNOTE_NOTE_TYPE_DIM	MunLabel
Dimension arrow	SWDIM	MunPoint
Stormwater catchment	SWCATCH	MunPolygon
River – line	RIVERLINE	MunLine
River – polygon	RIVERPOLY	MunPolygon
Floodline	FLOODLINE	MunPolygon
Dam	DAM	MunPolygon
Drainage note	SWNOTE	MunLabel

To query all the drainage objects from the database, do the following:

- Choose **Query** > **All Drainage Objects**.

The command line indicates how many pipes, channels, culverts, rivers, floodlines, dams, catchments, nodes, service connections, symbols, labels, notes, and dimensions respectively have been retrieved from the database.

Figure 1 Query all drainage objects

Querying stormwater network objects

This group of queries retrieves all of the stormwater objects related to the stormwater network according to the current GSC, and onto their respective layers. Stormwater network objects that can be retrieved are stormwater:

- pipes - retrieves all stormwater pipes that form part of the stormwater network onto the SWPIPE layer. Stormwater pipes are MunLine objects.
- culverts - retrieves all the culverts that form part of the stormwater network onto the SWCULVERT layer. Culverts are MunLine objects.
- channels - retrieves all the channels that form part of the stormwater network onto the SWCHANNEL layer. Channels are MunLine objects.
- nodes - retrieves all the stormwater nodes that form part of the stormwater network onto the SWNODE layer. The nodes are formatted according to their system query settings, which are determined by the database administrator.
- notes - retrieves all the drainage notes onto the SWNOTE layer.
- symbols - this retrieves all the stormwater symbols on the SWSYM layer.
- service connections - retrieves all the service connection objects.
- dimensions (notes and arrows) - retrieves all the drainage dimension arrows and dimension notes. Drainage dimension arrows and dimension notes are used to supply additional information on plans and drawings.
- labels - this retrieves all the drainage labels onto the SWLABEL layer.

To query stormwater network objects, do the following:

1 Choose **Query** > **Stormwater Network**

The command line displays the number of objects that were retrieved.

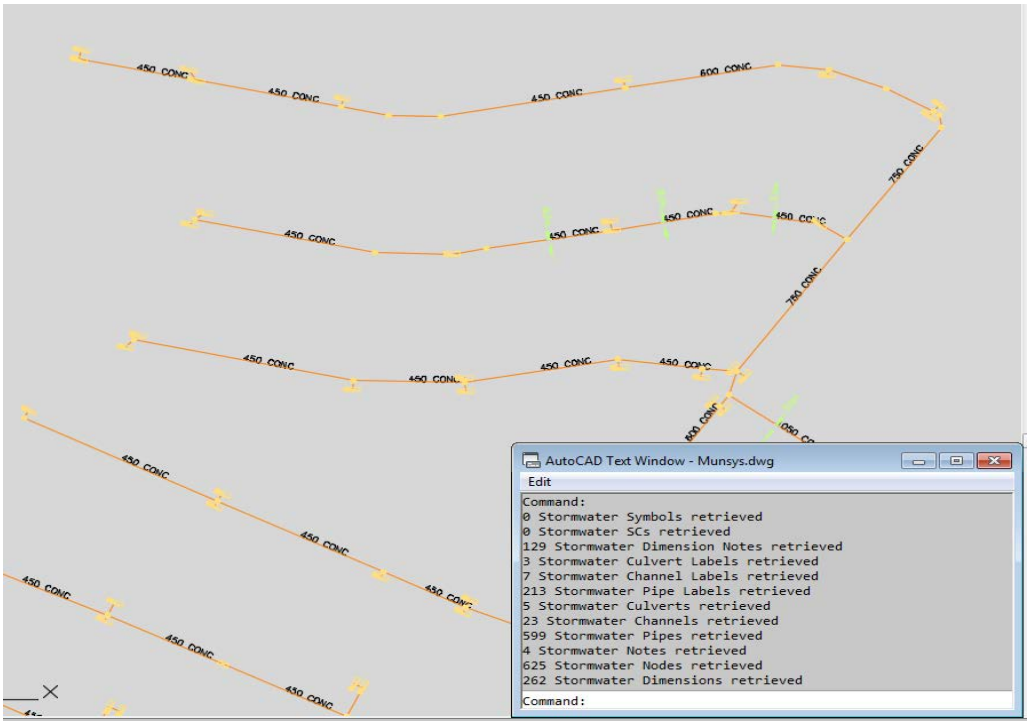


Figure 2 Querying the stormwater network

Querying additional drainage objects

With this query, you can retrieve any of the following drainage objects from the database, according to the current GSC:

- rivers
- floodlines
- dams
- catchments

To query additional drainage objects, do one of the following:

1 Choose **Query** > **Drainage Objects** > **Rivers**.

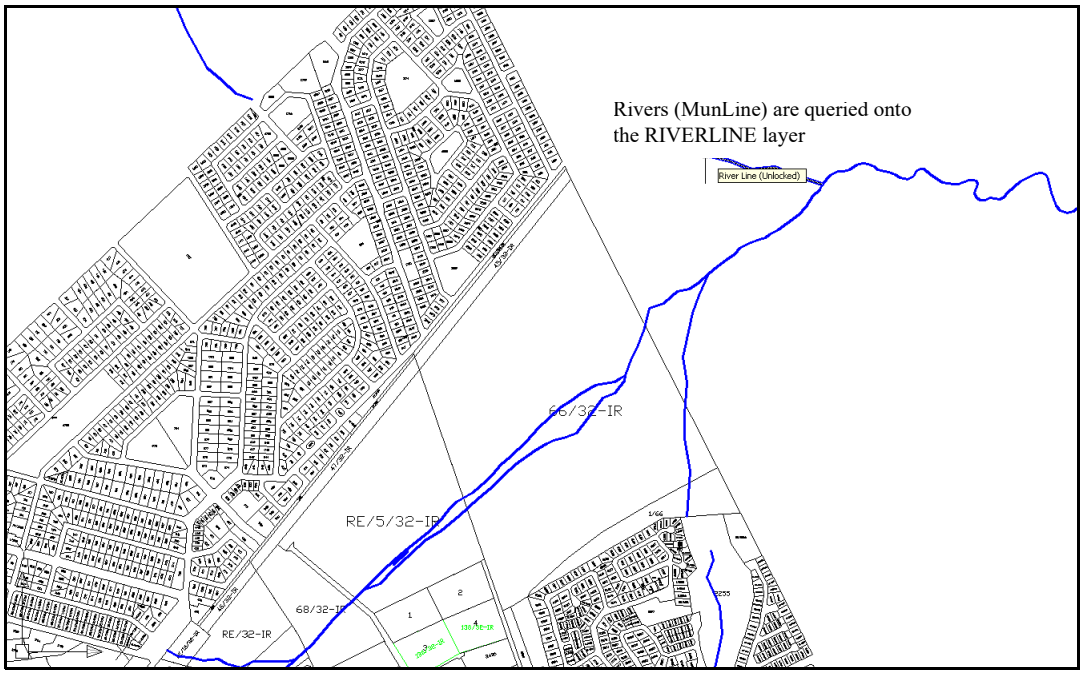


Figure 3 Querying rivers

The command line indicates how many rivers have been retrieved from the database.

2 Choose **Query** > **Drainage Objects** > **Floodlines**.

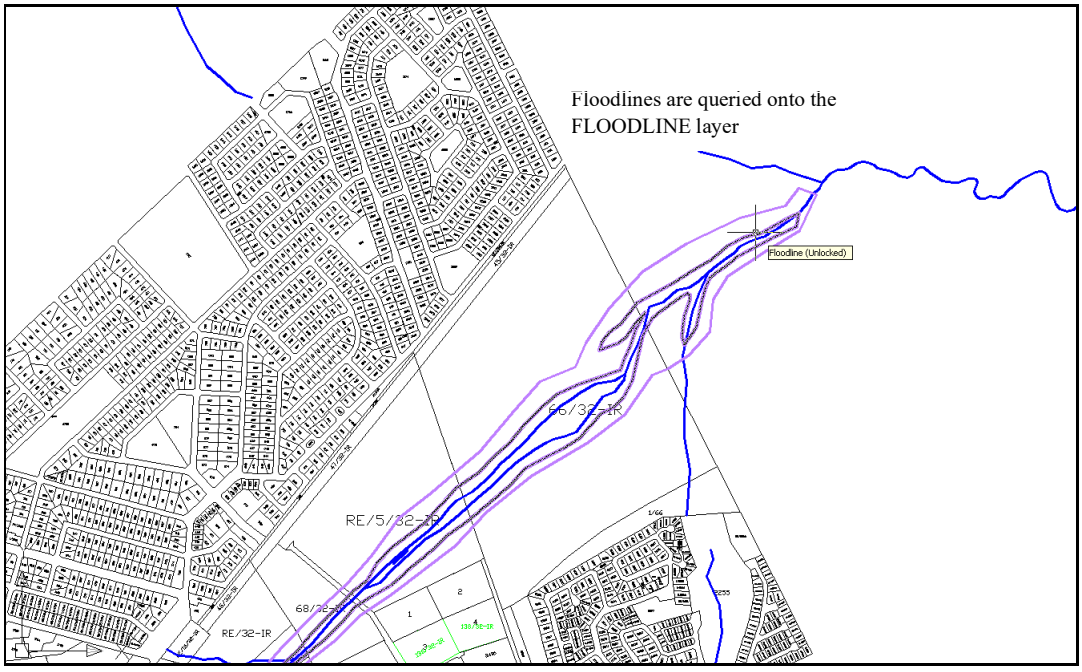


Figure 4 Querying floodlines

The command line indicates how many floodlines have been retrieved from the database.

3 Choose **Query** > **Drainage Objects** > **Dams**.

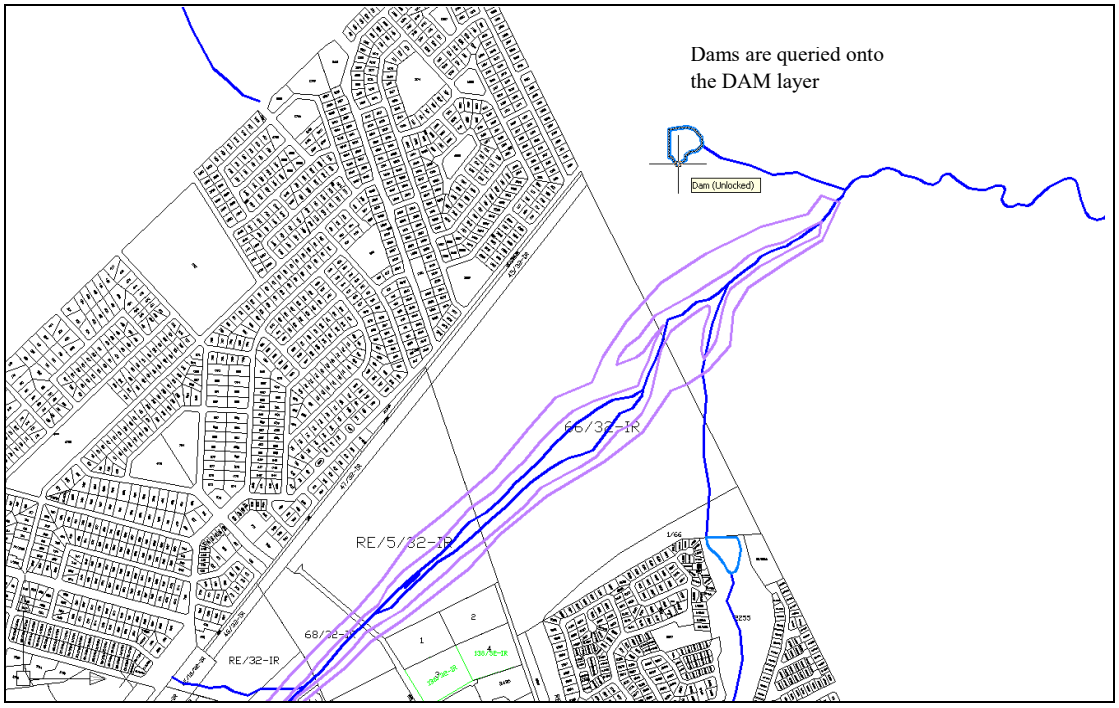


Figure 5 Querying dams

The command line indicates how many dams have been retrieved from the database.

4 Choose **Query** > **Drainage Objects** > **Catchments**.

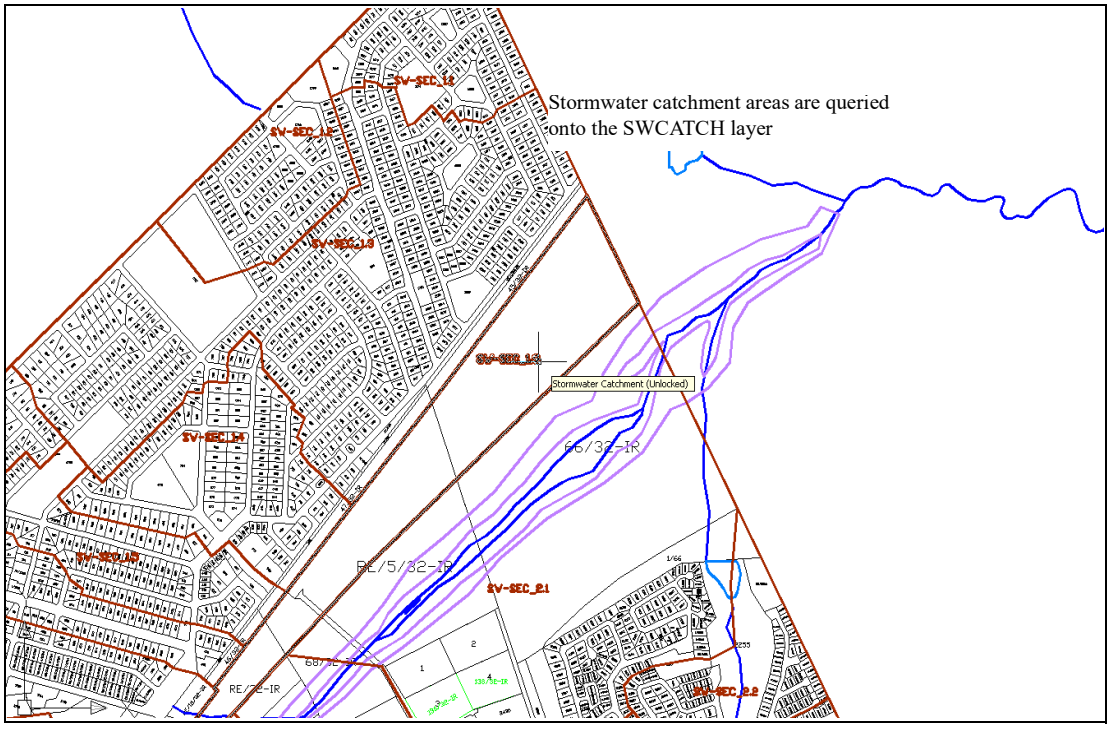


Figure 6 Querying catchments

The command line indicates how many catchments have been retrieved from the database.

Querying a map page grid

This query retrieves a map page grid from the database. The map page grid can be used to locate services based on the map page number, and is represented by a layout of polygons similar to the sheet layout used in Munsys Map Books. The map page grid can be modified by using the Object > Generate MunPolygons and Object > Edit MunPolygon menu items to capture and change the map page grid objects. The drainage map page grid is retrieved onto the SWMAPPAGE layer.

To query a map page grid

- Choose **Query** > **Map Page Grid**.

The map page grid is retrieved to the extents of the database.

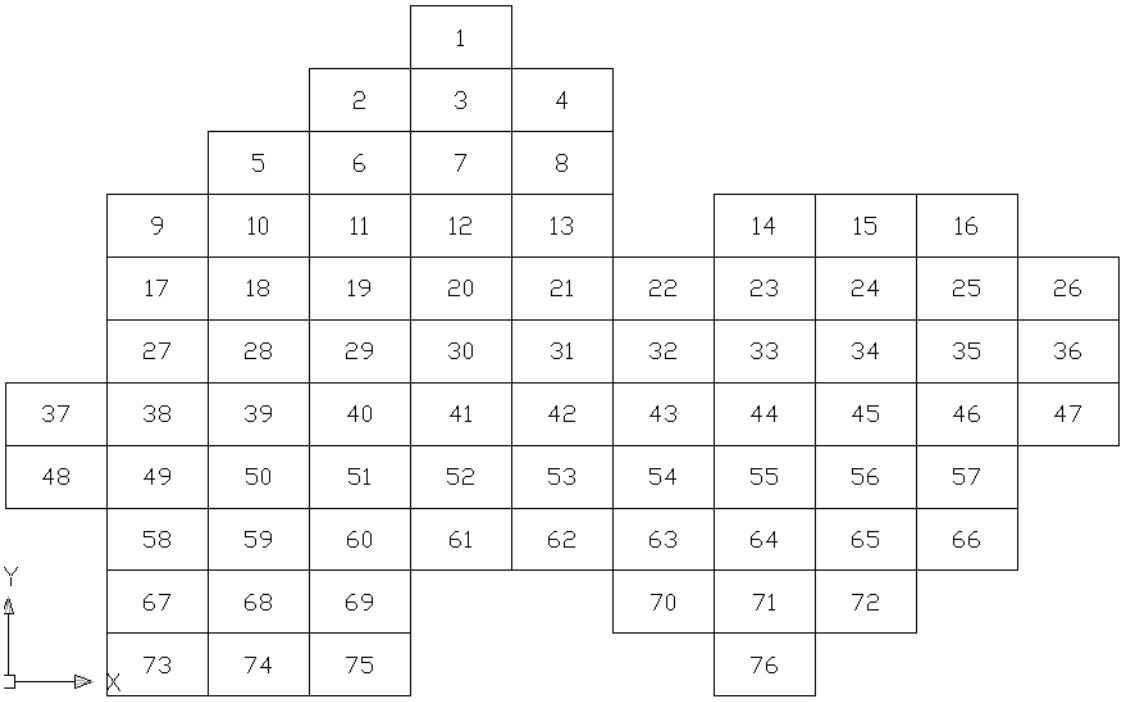


Figure 7 Querying a map page grid

Chapter 4
Creating drainage data

Capturing drainage data

This chapter describes how to capture a stormwater network and additional drainage objects, and how to post the captured data to the database.

A stormwater network gradually converges into watercourses, using natural slopes to allow the flow of water. The smallest diameter stormwater pipe is at the start of the stormwater network. As the stormwater pipes start converging, the pipe diameter becomes larger to accommodate the increase of water flow.

Stormwater pipes, culverts and channels rely on gravity or natural slope to allow the flow of water.

Capturing a stormwater network

Before you start capturing a stormwater network, you have to query the related cadastral data from the database to make certain that existing stormwater objects locate geographically correct. Existing stormwater objects need to be retrieved to determine the relationship between existing and new stormwater objects, to ensure that data is not duplicated.

The capture process is started by first placing all the stormwater pipes, culverts and channels. In residential areas, the stormwater network starts with the stormwater pipes, which are usually placed in the road reserves. Stormwater pipe diameter increases as the network develops to the catchment areas. To ensure that pipes are broken at pipe intersections, they have to be cleaned up.

After stormwater pipes, culverts or channels have been captured, stormwater nodes are placed followed by service connections.

Next, the manholes and junctions are aligned onto the stormwater pipes and culverts. Curb and drop inlets are placed in the road reserve, and wingwalls at the inlet and outlet of channels and pipes, where appropriate.

The stormwater integrity check must be run to check the stormwater objects against the business rules built into the capture and modify routines. Stormwater objects that pass the integrity check can be posted to the database.

A unique GID (Geographic Identifier) is automatically assigned to each stormwater object for identification purposes.

Drainage capture settings

Each Munsys application has its own default capture settings that are set by the database administrator in the Munsys Management Console.

In Munsys Drainage, default capture settings are available for:

- catchments – the catchment tag height
- channels – bottom width, depth, label height, material, offset distance, top width and type.
- culverts – depth, label height, material, offset distance, tag height, top width and type
- dams – tag height and type
- dimensions – dimension note height
- file – the drawing directory
- floodlines – tag height and type
- general – construction color, include network in object integrity, database extents display resolution, integrity circle size, tag and symbol scale and rotation if coordinate transformation is done, snap tolerance, tag angle, font, height and justification
- integrity – short objects, stormwater node, pipe, service connection and search tolerance
- nodes – tag height and stormwater symbol scale factor
- stormwater symbols – the stormwater symbol scale
- notes – tag height
- pipes – label height, tag height, diameter, material, type, offset distance
- rivers – tag height and type
- service connections – the default length and service connection type

You can change the current settings on the Drainage Settings dialog box to speed up the capture process. Capture settings only apply to *new* drainage objects, and if you change the default settings, the changes are only valid for the current session.

Settings are changed by highlighting the current value, and then entering a new value, or by selecting the new value from a drop-down list.

The user cannot change all of the settings. Some settings may only be changed by the GIS administrator to enforce consistency and integrity on a higher level.

Tip A short description of a value is displayed when it is highlighted.

To specify drainage capture settings

1 Do one of the following:

- Choose **Capture** > **Drainage Settings...**
- Click the **Drainage Settings** button on the Munsys Drainage **Capture** toolbar.

The Drainage Settings dialog box is displayed, showing the default drainage capture settings.

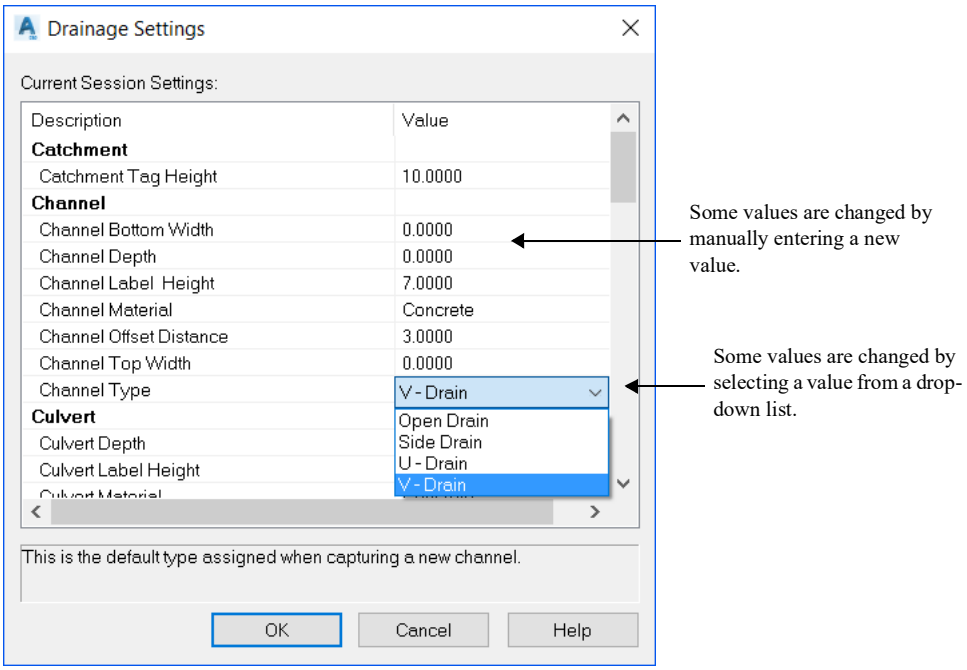


Figure 1 The Drainage Settings dialog box

2 To change a value, do one of the following:

- Highlight the current value, and then enter a new value.
- Highlight the current value, and then choose the new value from the drop-down list.

3 Click **OK** to apply the new settings.

The settings will apply until you change them again, or until the current session is terminated, or until you switch to a different application.

Capturing stormwater pipes, culverts and channels

Stormwater pipes, culverts and channels can be captured using two methods:

- In residential areas, the network can be captured at a specified offset distance from the cadastral boundaries.
- In other areas, the network can be captured by selecting points (freehand), or by entering coordinates on the command line.

Stormwater pipes, culverts and channels are usually placed in the road reserves in residential areas. As they start converging, the diameter becomes larger to accommodate the increase in water flow.

Stormwater pipes, culverts and channels are captured on their appropriate layers, to be verified when the integrity check is run.

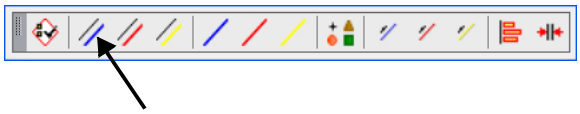
Drawing offset stormwater pipes, culverts and channels

Offset stormwater pipes are drawn by specifying multiple points along a boundary (for example parcel boundaries), specifying the offset segment, and then specifying the offset distance. Offset channels and culverts are drawn by specifying points along a boundary, or by selecting segments to offset the channel or culvert from.

To draw an offset stormwater pipe

1 Do one of the following:

- Choose **Capture** > **Draw Offset** > **Pipe**.
- Click the **Draw Offset Pipe** button on the Munsys Drainage **Capture** toolbar.



2 Specify an approximate start point for the pipe.

3 Specify an approximate endpoint for the pipe.

4 Select the segment to offset the pipe from.

5 Specify the side for offset.

6 On the command line, specify the offset distance, or press **ENTER** to accept the default offset distance.

The command line displays a message confirming that the pipe was created successfully.

To draw an offset culvert

A culvert can be drawn as a polyline entity between two stormwater nodes and is composed of a series of straight-line or arc segments.

1 Do one of the following:

- Choose **Capture** > **Draw Offset** > **Culvert**.
- Click the **Draw Offset Culvert** button on the Munsys Drainage **Capture** toolbar.

The command line prompts you to specify points, or to select a segment to offset the culvert from.

2 To draw an offset culvert by specifying points, do the following:

- Specify the first point
- Then specify next point of the culvert drawn as a straight-line segment or change to draw an arc segment [**A**rc/**H**alfwidth/**L**ength/**U**ndo/**W**idth].

By selecting A, the polyline linear list allows you to continue capturing offset culverts as a series of arcs. If selecting the Arc option, the polyline linear list changes with more selection options [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width].

Select L to return to drawing the culvert as series of straight-line segments.

- Press **ENTER** when you have specified all the points.Specify a point on the side to offset the culvert.
- On the command line, specify the offset distance, or press **ENTER** to accept the default offset distance.

The command line displays a message confirming that the culvert was created successfully.

3 To draw an offset culvert by selecting segments, do the following:

- On the command line, type **O**, and then press **ENTER**.
- Select a segment to offset the culvert from.
- Specify a point on the side to offset the culvert.
- On the command line, specify a multiplication factor for the default offset distance, or press **ENTER** to accept the default offset distance.
- Continue selecting segments until all the required segments have been selected, and then press **ENTER**.
- Press **ENTER** to confirm that you have completed selecting segments.

The command line displays a message confirming that the culvert was created successfully.

To draw an offset channel

A channel can be drawn as a polyline entity between two stormwater nodes and is composed of a series of straight-line or arc segments.

1 Do one of the following:

- Choose **Capture** > **Draw Offset** > **Channel**.
- Click the **Draw Offset Channel** button on the Munsys Drainage **Capture** toolbar.

The command line prompts you to specify points, or to select a segment to offset the channel from.

2 To draw an offset channel by specifying points, do the following:

- Specify the first point
- Then specify next point of the channel drawn as a straight-line segment or change to draw an arc segment [**A**rc/**H**alfwidth/**L**ength/**U**ndo/**W**idth].

By selecting A, the polyline linear list allows you to continue capturing offset channels as a series of arcs. If selecting the Arc option, the polyline linear list changes with more selection options [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width].

Select L to return to drawing the channel as series of straight-line segments.

- Press **ENTER** when you have specified all the points.Specify a point on the side to offset the channel.
- On the command line, specify the offset distance, or press **ENTER** to accept the default offset distance.

The command line displays a message confirming that the channel was created successfully.

3 To draw an offset channel by selecting segments, do the following:

- On the command line, type **O**, and then press **ENTER**.
- Select a segment to offset the channel from.
- Specify a point on the side to offset the channel.
- On the command line, specify a multiplication factor for the default offset distance, or press **ENTER** to accept the default offset distance.
- Continue selecting segments until all the required segments have been selected, and then press **ENTER**.
- Press **ENTER** to confirm that you have completed selecting segments.

The command line displays a message confirming that the channel was created successfully.

Drawing freehand stormwater pipes, culverts and channels

Freehand stormwater pipes, culverts and channels are drawn by selecting points in the drawing, or by entering coordinates on the command line. The command line prompts you for endpoints, allowing a sequence of objects to be constructed. Stormwater pipes, culverts and channels are captured on their respective layers (SWPIPE, SWCHANNEL and SWCULVERT), to be verified when the integrity check is run.

To draw a freehand pipe

1 Do one of the following:

- Choose **Capture** > **Draw Freehand** > **Pipe**.
- Click the **Draw Freehand Pipe** button on the Munsys Drainage **Capture** toolbar.

2 Specify a start point for the pipe, or enter coordinates on the command line.

3 Specify an end point for the pipe, or enter coordinates on the command line.

4 Press **ENTER** to complete.

Tip To construct a sequence of pipes, continue to specify endpoints, and then press **ENTER**.

The command line displays a message confirming that the pipe was created successfully.

To draw a freehand culvert

1 Do one of the following:

- Choose **Capture** > **Draw Freehand** > **Culvert**.
- Click the **Draw Freehand Culvert** button on the Munsys Drainage **Capture** toolbar.

- Specify the first point for the culvert
- Then specify next point of the culvert drawn as a straight-line segment or change to draw an arc segment [**A**rc/**H**alfwidth/**L**ength/**U**ndo/**W**idth].

By selecting A, the polyline linear list allows you to continue capturing offset culverts as a series of arcs. If selecting the Arc option, the polyline linear list changes with more selection options [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width].

- Select **L** to return to drawing the culvert as series of straight-line segments.
- Press **ENTER** to complete.

The command line displays a message confirming that the culvert was created successfully.

To draw a freehand channel

1 Do one of the following:

- Choose **Capture** > **Draw Freehand** > **Channel**.
- Click the **Draw Freehand Channel** button on the Munsys Drainage **Capture** toolbar.

- Specify the first point for the channel
- Then specify next point of the channel drawn as a straight-line segment or change to draw an arc segment [**A**rc/**H**alfwidth/**L**ength/**U**ndo/**W**idth].

By selecting A, the polyline linear list allows you to continue capturing offset channels as a series of arcs. If selecting the Arc option, the polyline linear list changes with more selection options [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width].

- Select **L** to return to drawing the channel as series of straight-line segments.
- Press **ENTER** to complete.

The command line displays a message confirming that the channel was created successfully.

The following figure gives an example of a stormwater pipe, culvert and channel:

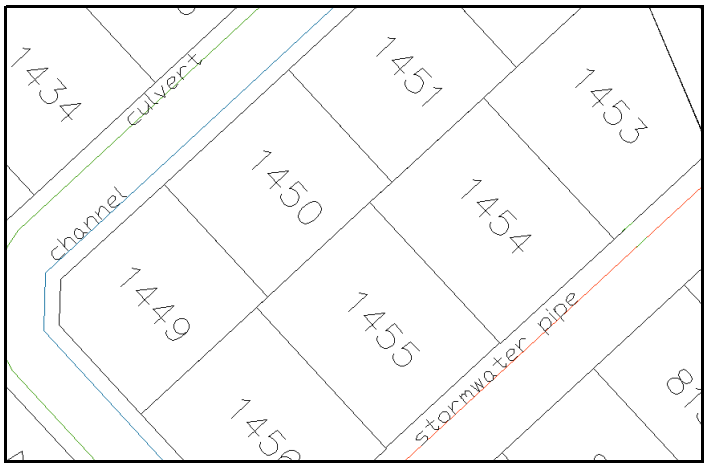


Figure 2 Stormwater pipe, culvert and channel

Showing flow direction

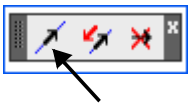
When you have captured and edited all the pipes, culverts and channels in the stormwater network, you have to check the flow direction of these objects to verify that it is correct. This can be done for both new and existing objects.

Important Make certain that the **OSnap** settings are set to **None** before showing or changing flow direction.

To show flow direction

1 Do one of the following:

- Choose **Capture** > **Show Flow Direction**.
- Click the **Show Flow Direction** button on the Munsys Drainage **Flow** toolbar.



2 Select the appropriate stormwater objects (pipes, channels or culverts), and then press **ENTER**.

A *direction arrow* on each selected object indicates the flow direction, as shown in the following figure:

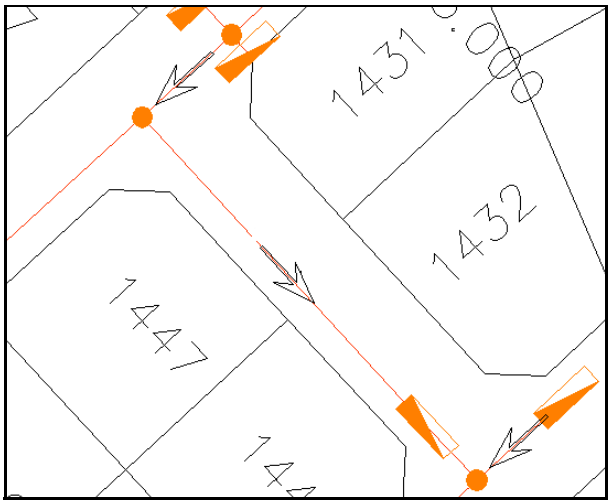


Figure 3 Showing stormwater flow direction

To change flow direction

This function is used to reverse the flow direction of stormwater pipes, culverts and channels. This enables users to construct the network without taking direction into account. Once the construction is complete, the flow direction can be changed easily.

1 Do one of the following:

- Choose **Capture** > **Change Flow Direction**.
- Click the **Change Flow Direction** button on the Munsys Drainage **Flow** toolbar.

2 Select the objects (pipes, culverts or channels) of which you want to change the flow direction, and then press **ENTER**.

The direction is reversed automatically.

Important The integrity check does not verify flow direction; therefore it is very important to make certain that the flow direction is correct.

To clear direction arrows

When you are satisfied that the direction of all the stormwater pipes, culverts and channels is correct, the direction arrows can be cleared from the drawing.

1 Do one of the following:

- Choose **Capture** > **Clear Direction Arrows**.
- Click the **Clear Direction Arrows** button on the Munsys Drainage **Flow** toolbar.

The direction arrows are removed from the drawing.

Capturing stormwater nodes

Stormwater nodes are captured by placing them either at the endpoint, or at a nearest point on a pipe, culvert or channel. Nodes are captured as MunPoint objects, and are displayed as symbols.

Attribute information, providing more descriptive information about nodes, is associated with every node. A stormwater network contains the following types of nodes:

Node	Description
Curb inlet	An inlet on the side of the road that gathers flowing rainwater
Drop inlet	An inlet that is level with the road surface; also known as a grid inlet.
Manhole	Found at every bend point in the stormwater network, and used to gain access to stormwater pipes and culverts
Junction	A bend point in the stormwater network that does not provide access to pipes or culverts
Wingwall	An extension of a channel or pipe, constructed to prevent soil erosion around the inlet or outlet.

Table 4 Stormwater nodes

Placing endpoint nodes

Endpoint nodes are placed at the end point of a stormwater pipe, culvert or channel, closest to the point where it was selected. The following endpoint nodes can be placed:

- curb inlets
- drop inlets
- manholes (cannot be placed on channels)
- junctions
- wingwalls

To place an endpoint node

1 Do one of the following:

- Choose **Capture** > **Place Endpoint Node** > *Node Name*.
- Click the appropriate button on the Munsys Drainage **Place Node** toolbar.

2 Select a point close to the endpoint on the stormwater pipe, culvert or channel where you want to place the node.

The node is placed on the SWNODE layer, to be verified when the integrity check is run.

Inserting nearest nodes

With this function, you can insert a manhole, curb inlet, or drop inlet at the nearest selected point on a pipe, culvert or channel. The underlying object is broken at the selected point when the node is inserted.

You can use this function when constructing a single long pipe, culvert or channel, inserting the required nodes at regular intervals. It is also used in cases where new nodes are added to an existing stormwater network.

To insert a nearest node

1 Do one of the following:

- Choose **Capture** > **Insert Nearest Node** > *Node Name*.
- Click the appropriate button on the Munsys Drainage **Insert Node** toolbar.

2 Select any nearest point on the stormwater pipe, culvert or channel where you want to insert the node.

The node is placed on the SWNODE layer, to be verified when the integrity check is run.

Capturing stormwater service connections

Service connections connect land parcels to the stormwater network.

The Munsys Drainage Place SC toolbar contains the functions that are used to capture different types of service connections.

A service connection is captured as a freehand connection, or at 45° or 90° as a single or double service connection, at either the endpoint or a nearest position on a gravity pipe.

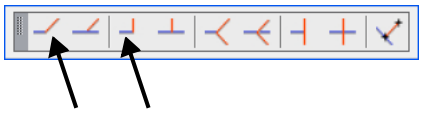
Placing single service connections

Single service connections are placed nearest to the selection point, or at the endpoint of a gravity pipe, at either 45° or 90° angles to the pipe, using the default service connection length and service connection type as defined in the Drainage Settings dialog box.

To place a single service connection at the endpoint of a gravity pipe

1 Do one of the following:

- Choose **Capture** > **Place Single SC** > **45 Deg Endpoint** or > **90 Deg Endpoint**.
- Click the **Place Single SC 45 Deg Endpoint** or **90 Deg Endpoint** button on the Munsys Drainage **Place SC** toolbar.



2 Select a point close to the endpoint of the gravity pipe where you want to place the service connection.

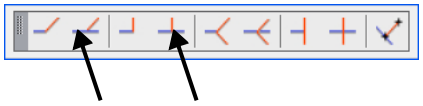
3 Rotate your mouse in a clockwise or counter-clockwise direction, and then click to display the service connection vector at angles of 45 or 90 degrees.

The service connection is placed on the SWSERV layer.

To place a single service connection at a nearest point on a gravity pipe

1 Do one of the following:

- Choose **Capture** > **Place Single SC** > **45 Deg Nearest** or > **90 Deg Nearest**.
- Click the **Place Single SC 45 Deg Nearest** or **90 Deg Nearest** button on the Munsys Drainage **Place SC** toolbar.



2 Select a point on the gravity pipe where you want to place the service connection.

3 Rotate your mouse in a clockwise or counter-clockwise direction, and then click to display the service connection vector at angles of 45 or 90 degrees.

The service connection is captured on the SWSERV layer.

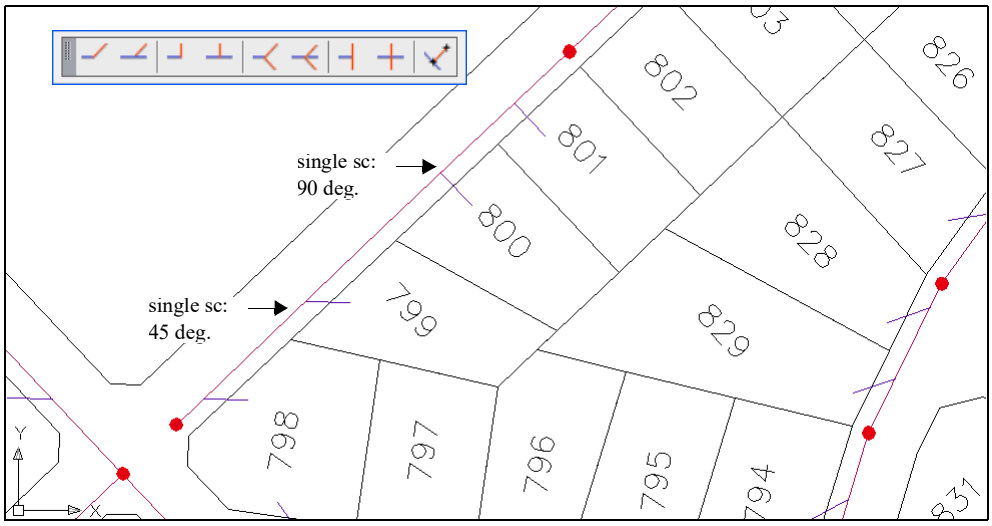


Figure 5 Single service connections

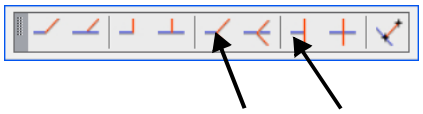
Placing double service connections

Double service connections are placed nearest to the selection point on, or at the endpoint of a gravity pipe, at either 45° or 90° angles to the pipe, using the default service connection length and service connection type as defined in the Drainage Settings dialog box.

To place a double service connection at the endpoint of a gravity pipe

1 Do one of the following:

- Choose **Capture** > **Place Double SC** > **45 Deg Endpoint** or > **90 Deg Endpoint**.
- Click the **Place Double SC 45 Deg** or **90 Deg Endpoint** button on the Munsys Drainage **Place SC** toolbar.



2 Select a point close to the endpoint of the gravity pipe where you want to place the service connection.

3 Rotate your mouse in a clockwise or counter-clockwise direction, and then click to display the service connection vector at angles of 45 or 90 degrees.

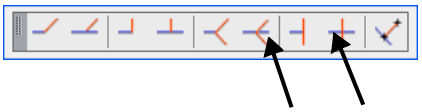
4 Repeat **Step 3** to place the second segment of the service connection.

The service connection is captured on the SWSERV layer.

To place a double service connection at a nearest point on a gravity pipe

1 Do one of the following:

- Choose **Capture** > **Place Double SC** > **45 Deg Nearest** or **90 Deg Nearest**.
- Click the **Place Double SC 45 Deg** or **90 Deg Nearest** button on the Munsys Drainage **Place SC** toolbar.



2 Select a point on the gravity pipe where you want to place the service connection.

3 Rotate your mouse in a clockwise or counter-clockwise direction, and then click to display the service connection vector at angles of 45 or 90 degrees.

4 Repeat **Step 3** to place the second segment of the service connection.

The service connection is captured on the SWSERV layer.

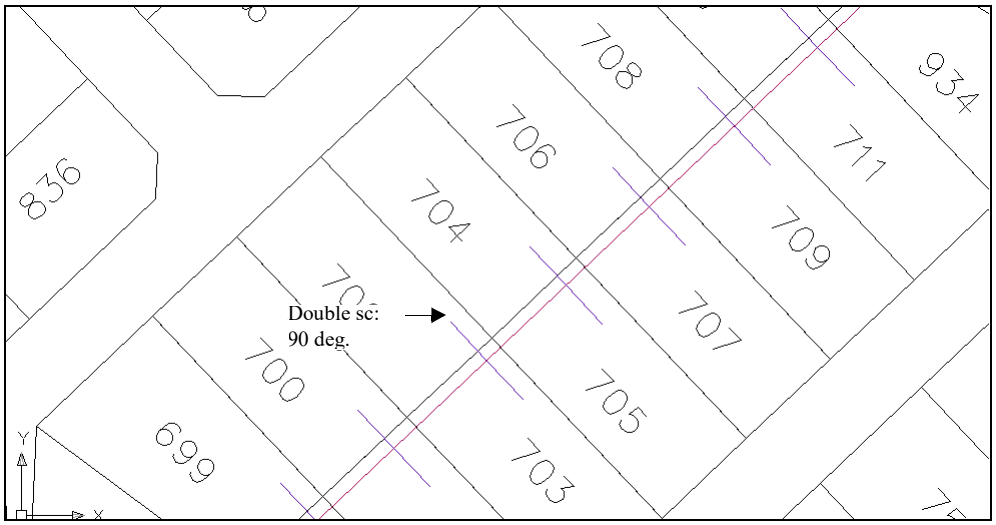


Figure 6 Double service connections

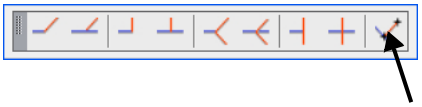
Placing freehand service connections

With this function, you can place a single freehand service connection that is longer than the default service connection length specified on the Drainage Settings dialog box, and that does not tie into a stormwater pipe at the standard 45° and 90°. The service connection is captured with the default service connection type specified on the Drainage Settings dialog box.

To place a freehand service connection

1 Do one of the following:

- Choose **Capture** > **Place Freehand SC**.
- Click the **Place Freehand Service Connection** button on the Munsys Drainage **Place SC** toolbar.



2 Select the gravity pipe where the service connection should be connected, and then drag and click to place the service connection inside the parcel boundary.

The service connection is captured on the SWSERV layer.

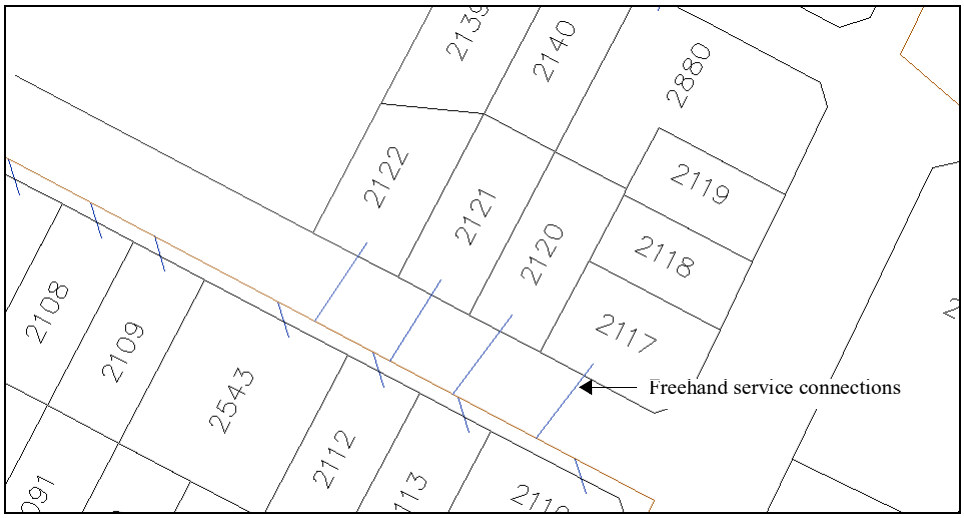


Figure 7 Freehand service connections

Capturing rivers, floodlines and dams

Rivers can be drawn freehand, at a specified offset distance from a selected segment, or drawn as a polygon. Floodlines and dams are captured as polygons. Rivers, floodlines and dams do not form part of the stormwater network.

Drawing rivers

To draw a freehand river

Freehand rivers are drawn by specifying consecutive points in the drawing, or by entering coordinates on the command line.

To draw a freehand river, do the following:

1. Choose **Capture** > **River-Line** > **Draw Freehand River**.
2. Specify the start point river by selecting a point, or enter coordinates on the command line.
3. Specify the next points for the river, or select A for Arc[**A**rc/**H**alfwidth/**L**ength/**U**ndo/**W**idth], or enter more coordinates on the command line.

   You can construct the river line as a series of straight-line or arc segments. By selecting A, the polyline linear list allows you to continue capturing river lines as a series of arcs. If selecting the Arc option, the polyline linear list changes with more selection options [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]. Select L to return to drawing the river line as series of straight-line segments.
4. Press **ENTER** when you have specified all the points to create the river.

   The river is created on the RIVERLINE layer, to be verified when the integrity check is run.

To draw an offset river

Offset rivers are captured at a specified offset distance from segments. The default offset distance can be changed in the Drainage Settings dialog box.

1. Choose **Capture** > **River-Line** > **Draw Offset River**.

   The command line prompts you to specify points, or to select a segment to offset the river from.
2. To draw a river by specifying points, do the following:
   - Specify the first point, and then specify next points for the river.
   - Then specify next point of the river drawn as a straight-line segment or change to draw an arc segment [**A**rc/**H**alfwidth/**L**ength/**U**ndo/**W**idth].

   By selecting A, the polyline linear list allows you to continue capturing the river as a series of arcs. If selecting the Arc option, the polyline linear list changes with more selection options [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]. Select L to return to drawing the river as series of straight-line segments.
   - Press **ENTER** when you have specified all the points.
   - Specify a point on the side to offset the river.
   - On the command line, specify the offset distance.

   The command line displays a message confirming that the river was created successfully.

3 To draw a river by selecting segments, do the following:

- On the command line, type **O**, and then press **ENTER**.
- Select a segment to offset the river from.
- Specify a point on the side to offset the river.
- On the command line, specify a multiplication factor for the default offset distance, or press **ENTER** to accept the default offset distance.
- Continue selecting segments until all the required segments have been selected, and then press **ENTER**.
- Press **ENTER** to confirm that you have completed selecting segments.

The command line displays a message confirming that the river was created successfully.

Drawing a polygon river

A polygon river is created by selecting consecutive points to form a polygon. Next, a tag (the river name) is assigned to the river. The polygon builder uses the lines and tag of the newly created river to build the river polygon, and verifies that the polygon forms a closed area. The integrity check still needs to verify the standard business rules for drainage objects before the river can be posted to the database. River polygons are generated on the RIVERPOLY layer.

To draw a river boundary

1 Choose **Capture** > **River-Polygon** > **Draw River Boundary**.

2 Specify the first polygon point.

3 Specify the next points, and then press **ENTER**.

4 Press **ENTER** to close the polygon, or type **N** on the command line to specify more points.

Munsys closes the polygon automatically by snapping to the first specified point.

To place a river tag

1 Choose **Capture** > **River-Polygon** > **Place River Tag**.

2 Specify the insertion point for the tag.

3 On the command line, specify the tag height, or press **ENTER** to accept the default height.

4 To indicate the tag angle, do one of the following:

- Indicate the tag angle with your mouse.
- On the command line, type **A** to align the tag to an object, press **ENTER**, and then select the object to which you want to align the tag.

5 Type the tag text on the command line, and then press **ENTER**.

The tag is inserted as specified.

To generate a river polygon

1. Choose **Capture** > **River-Polygon** > **Generate River Polygons**.
2. Select all the lines that form the river polygon, and then press **ENTER**.
3. Select the river tag, and then press **ENTER**.

   The command line indicates that the polygon has been built successfully.

Drawing floodlines

A floodline boundary is drawn by selecting consecutive points to form a polygon. Munsys closes the polygon automatically by snapping to the first specified point. Next, a tag is assigned to the floodline. The polygon builder uses the lines and tag of the newly created floodline to build the floodline polygon.

To draw a floodline boundary

1. Choose **Capture** > **Floodline Boundary** > **Draw Floodline Boundary**.
2. Specify the first polygon point.
3. Specify the next points, and then press **ENTER**.
4. Press **ENTER** to close the polygon, or type **N** on the command line to specify more points.

   Munsys closes the polygon automatically by snapping to the first specified point.

To place a floodline tag

1. Choose **Capture** > **Floodline Boundary** > **Place Floodline Tag**.
2. Specify the insertion point for the tag.
3. On the command line, specify the tag height, or press **ENTER** to accept the default height.
4. To indicate the tag angle, do one of the following:
   - Indicate the tag angle with your mouse.
   - On the command line, type **A** to align the tag to an object, press **ENTER**, and then select the object to which you want to align the tag.
5. Type the tag text on the command line, and then press **ENTER**.

   The tag is inserted as specified.

To generate a floodline polygon

1. Choose **Capture** > **Floodline Boundary** > **Generate Floodline Polygons**.
2. Select the lines that form the floodline polygon, and then press **ENTER**.
3. Select the floodline tag, and then press **ENTER**.

   The command line indicates that the polygon has been built successfully.

Drawing dams

A dam is drawn by selecting consecutive points to form a polygon. Munsys closes the polygon automatically by snapping to the first specified point. Next, a tag (text) is assigned to the dam. The polygon builder uses the lines and tag of the newly created dam to build the dam polygon.

To draw a dam boundary

1. Choose **Capture** > **Dam** > **Draw Dam Boundary**.
2. Specify the first polygon point.
3. Specify the next points, and then press **ENTER**.
4. Press **ENTER** to close the polygon.

   Munsys closes the polygon automatically by snapping to the first specified point.

To place a dam tag

1. Choose **Capture** > **Dam** > **Place Dam Tag**.
2. Specify the insertion point for the tag.
3. On the command line, specify the tag height, or press **ENTER** to accept the default height.
4. To indicate the tag angle, do one of the following:
   - Indicate the tag angle with your mouse.
   - On the command line, type **A** to align the tag to an object, press **ENTER**, and then select the object to which you want to align the tag.
5. Type the tag text on the command line, and then press **ENTER**.

   The tag is inserted as specified.

To generate a dam polygon

1. Choose **Capture** > **Dam** > **Generate Dam Polygons**.
2. Select the lines that form the dam polygon, and then press **ENTER**.
3. Select the dam tag, and then press **ENTER**.

   The command line indicates that the polygon has been built successfully.

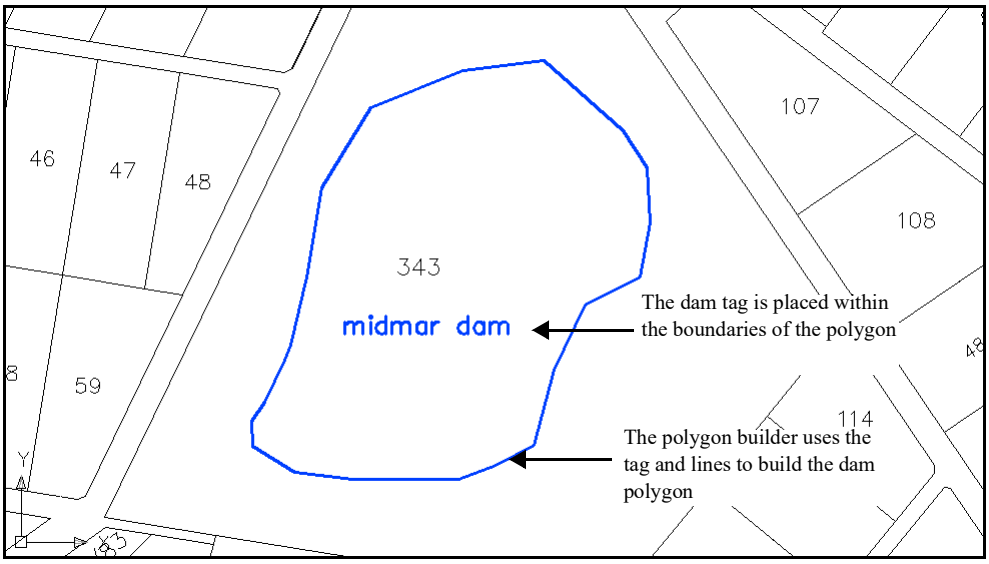


Figure 8 A dam polygon

Capturing stormwater catchments

A stormwater catchment area is created by selecting consecutive points to form a polygon. Munsys closes the polygon automatically by snapping to the first specified point. Next, a tag is assigned to the catchment. The polygon builder uses the lines and tag of the newly created catchment to build the catchment polygon, and verifies that the polygon forms a closed area. The integrity check still needs to verify the standard business rules for stormwater before the catchment can be posted to the database. Catchment polygons are generated on the SWCATCH layer.

To draw a catchment boundary

1. Choose **Capture** > **Catchment** > **Draw Catchment Boundary**.
2. Specify the first polygon point.
3. Specify the next points, and then press **ENTER**.
4. Press **ENTER** to close the polygon, or type **N** on the command line to specify more points.

   Munsys closes the polygon automatically by snapping to the first specified point.

To place a catchment tag

1. Choose **Capture** > **Catchment** > **Place Catchment Tag**.
2. Specify the insertion point for the tag.
3. On the command line, specify the tag height, or press **ENTER** to accept the default height.
4. To specify the tag angle, do one of the following:
   - Indicate the tag angle with your mouse.
   - On the command line, type **A** to align the tag to an object, press **ENTER**, and then select the object to which you want to align the tag.

5 Type the tag text on the command line, and then press **ENTER**.

The tag is inserted as specified.

To generate a catchment polygon

1 Choose **Capture** > **Catchment** > **Generate Catchment Polygons**.

2 Select all the lines on the catchment polygon, and then press **ENTER**.

3 Select the catchment tag, and then press **ENTER**.

The command line indicates that the polygon has been built successfully.

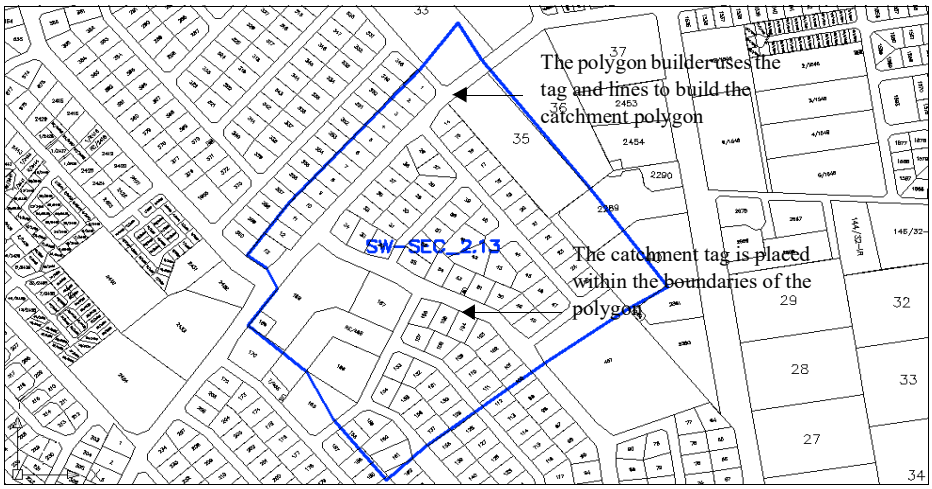


Figure 9 A stormwater catchment area

Placing stormwater symbols

This function is used to place stormwater symbols in the current drawing. Stormwater symbols represent additional annotation in the form of a symbol, and do not form part of the stormwater network. Stormwater symbols can be placed anywhere without any restriction. Stormwater symbols are captured on the SWSYM layer.

Note The database administrator can add additional stormwater symbols to the lookup table **LU_DMS_SWSYM** as required. This table contains a number of default symbols, but can be customized.

To place stormwater symbols

1 Do one of the following:

- Choose **Capture** > **Place Stormwater Symbol...**
- Click the **Place Stormwater Symbol** button on the Munsys Drainage **Capture** toolbar.

2 The **Stormwater Symbols** dialog box is displayed, showing a list of available stormwater symbols.

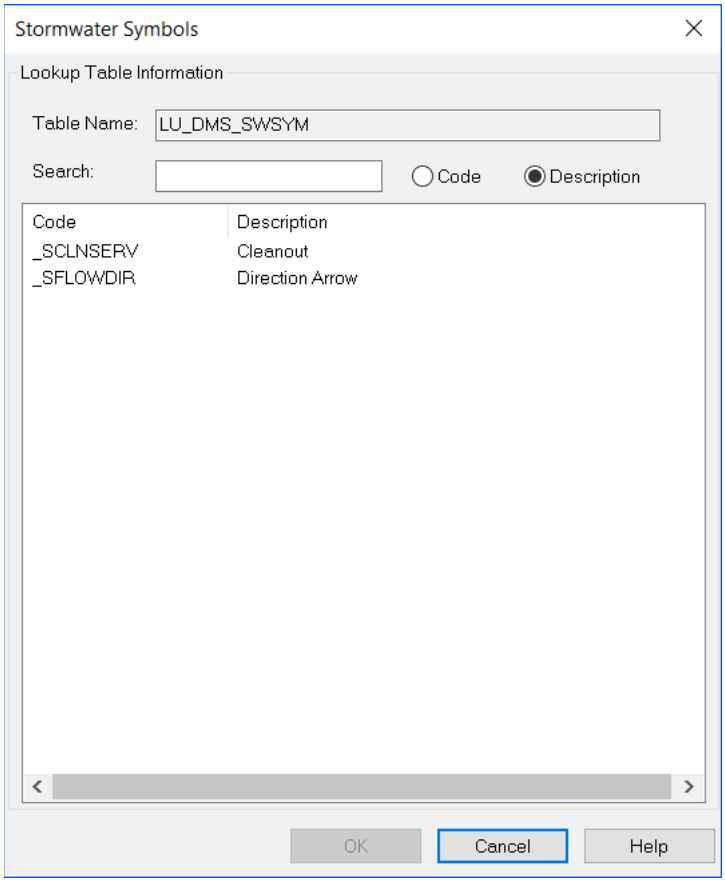


Figure 10 The Stormwater Symbols dialog box

3 From the list, select the symbol that you want to place, and then click **OK**.

4 Specify an insertion point for the symbol.

5 Specify a rotation angle for the symbol by doing one of the following:

- Enter the rotation angle on the command line, or press **ENTER** to accept the default angle of zero.
- Indicate the angle with your mouse pointer by picking two points in the drawing.
- On the command line, type **AL** to align the symbol to an object, press **ENTER**, and then select the object to which you want to align the symbol.

The symbol is placed at the insertion point and angle that you specified.

Tip Press **ENTER** or the **Space Bar** to continue placing consecutive symbols of the same type, without having to select a symbol type from the list every time.

Placing drainage labels

In Munsys, labels contain text information that represent a specified column in the database. These labels are dynamic and change according to the changes in the database. Labels can be created for stormwater pipes, culverts and channels.

To place drainage labels

The following information is displayed on drainage labels:

- pipe – diameter and pipe material
- culvert – width, depth and material
- channel – top width, bottom width, material and depth

1 To place a drainage label, do one of the following:

- Choose **Capture** > **Place Drainage Label** > *Label Name*.
- Click the appropriate button on the Munsys Drainage **Label** toolbar.

2 Select the pipe, culvert or channel to which you want to attach the label.

3 Specify an insertion point for the tag.

4 Specify the tag height, or press **ENTER** to accept the default tag height.

5 To indicate the tag angle, do one of the following:

- Indicate the tag angle with your mouse.
- On the command line, type **A** to align the tag to an object, press **ENTER**, and then select the object to which you want to align the tag.

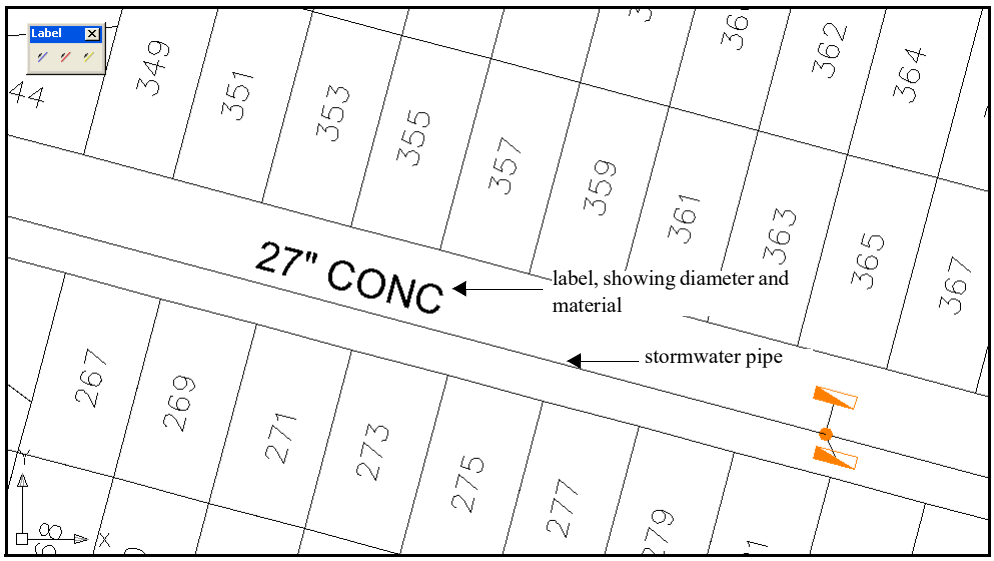


Figure 11 A stormwater pipe label

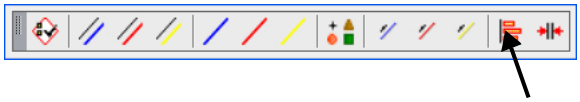
Placing drainage notes

For map production purposes, information about a spatial object is added in the form of a note. Drainage notes are placed on the SWNOTE layer.

To place a drainage note

1 Do one of the following:

- Choose **Capture** > **Place Drainage Note**.
- Click the **Place Drainage Note** button on the Munsys Drainage **Capture** toolbar.



2 Specify the insertion point for the note.

3 On the command line, specify the note height, or press **ENTER** to accept the default note height.

4 To specify the note angle, do one of the following:

- Indicate the angle with your mouse.
- On the command line, type **A** to align the note to an object, press **ENTER**, and then select the object to which you want to align the note to.

5 Type the note value on the command line, and then press **ENTER**.

The note is inserted as specified.

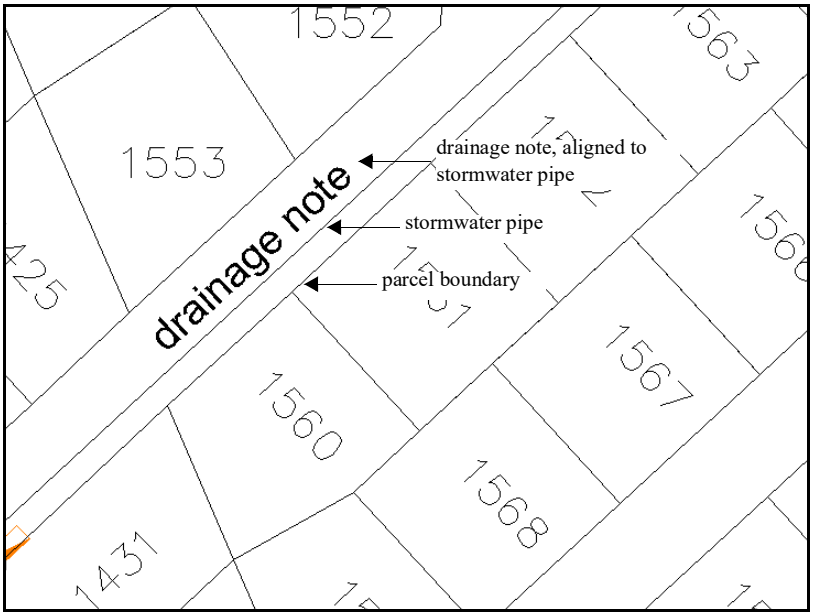


Figure 12 A drainage note

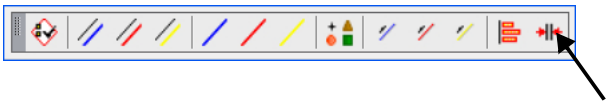
Placing dimension arrows

Map production is a reduced version of reality and it is therefore virtually impossible to capture stormwater pipes at the true offset distance from a parcel boundary. Dimension arrows are used to indicate the offset distance of a stormwater pipe, culvert or channel from a parcel boundary.

To place dimension arrows

1 Do one of the following:

- Choose **Capture** > **Place Dimension Arrow**.
- Click the **Place Dimension Arrow** button on the **Munsys Drainage Capture** toolbar.



2 Specify the first segment for dimension.

The dimension arrows will be placed perpendicular to this segment.

3 Specify the second segment for dimension.

The dimension arrows are aligned to the point selected on this segment.

4 On the command line, enter the offset distance, or press **ENTER** to accept the default offset distance as determined in the Drainage Settings, or press **A** to place the actual calculated distance.

The dimension text is placed at the dimension arrows, indicating the distance between the two dimension lines.

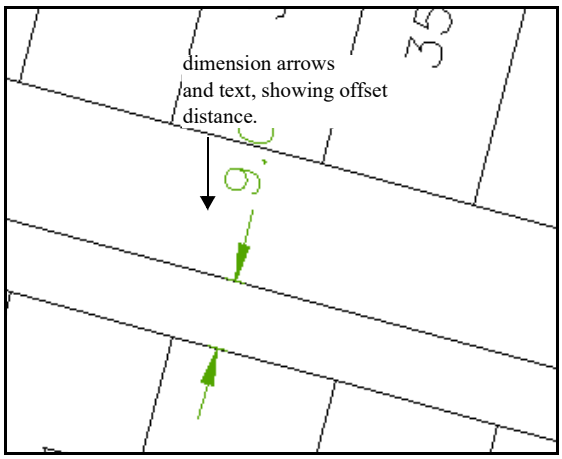


Figure 13 Dimension arrows

Capturing pipe, channel, culvert and node levels

With Munsys Drainage, you can capture pipe, culvert and channel levels by entering ground and invert levels for their respective start and end nodes. The length and gradient are calculated automatically. Node levels (depth, ground level, length and width) can be captured as well.

To attach pipe levels

1 Choose **Capture** > **Pipe Levels...**

2 Select the stormwater pipe that you want to attach the information to.

The Pipe Information dialog box is displayed.

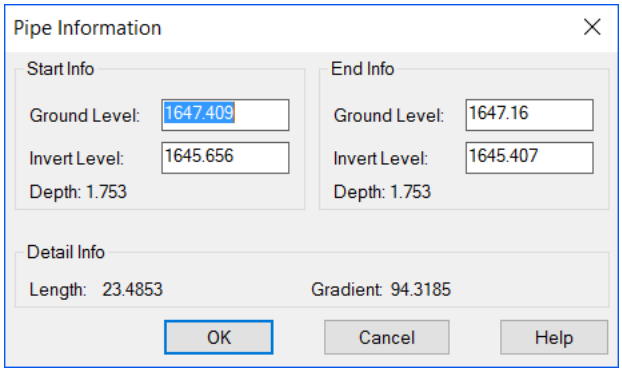


Figure 14 The Pipe Information dialog box

3 From the **Start Info** group, enter the following information:

- Ground level
- Invert level

The Depth value is calculated automatically.

4 From the **End Info** group, enter the following information:

- Ground level
- Invert level

The Depth value is calculated automatically.

The pipe length and gradient values in the Detail Info group are calculated automatically from the Start Info and End Info groups.

5 Click **OK**.

The attribute changes are accepted.

To attach culvert levels

1 Choose **Capture** > **Culvert Levels...**

2 Select the culvert that you want to attach the information to.

The Culvert Information dialog box is displayed.

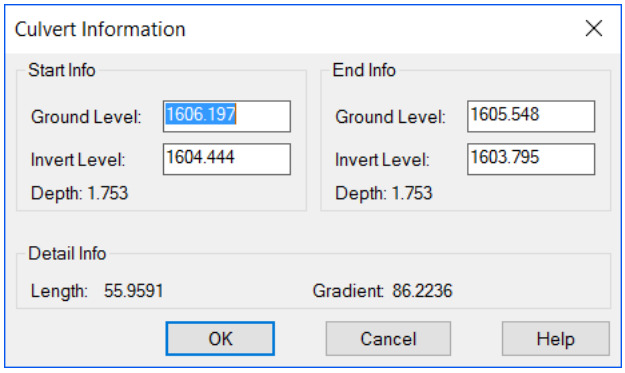


Figure 15 The Culvert Information dialog box

3 In the **Start Info** group, enter the following information:

- Ground level
- Invert level

The depth value is calculated automatically.

4 From the **End Info** group, enter the following information:

- Ground level
- Invert level

The Depth value is calculated automatically.

The culvert length and gradient values in the Detail Info group are calculated automatically from the Start Info and End Info group.

5 Click **OK**.

The attribute changes are accepted.

To attach channel levels

1 Choose **Capture** > **Channel Levels...**

2 Select the channel that you want to attach the information to.

The Channel Information dialog box is displayed.

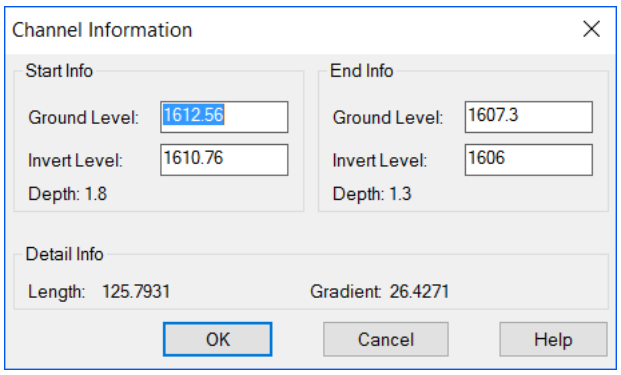


Figure 16 The Channel Information dialog box

3 In the **Start Info** group, enter the following information:

- Ground level
- Invert level

The Depth value is calculated automatically.

4 From the **End Info** group, enter the following information:

- Ground level
- Invert level

The Depth value is calculated automatically.

The channel length and gradient values in the Detail Info group are calculated automatically from the Start Info and End Info group.

5 Click **OK**.

The attribute changes are accepted.

To attach node levels

1 Choose **Capture** > **Node Levels...**

2 Select the appropriate stormwater node.

The Stormwater Node Information dialog box is displayed.

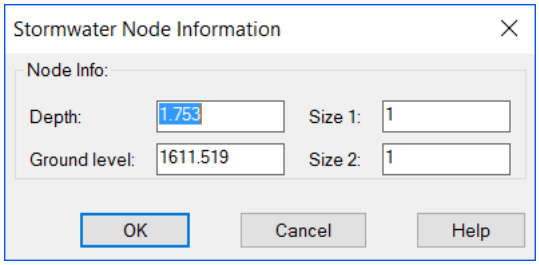


Figure 17 The Stormwater Node Information dialog box

3 Enter a value in each of the following fields:

- Node Depth (m)
- Ground Level
- Size 1 – length
- Size 2 – width

4 Click **OK** to attach the node levels for the selected node.

Checking drainage integrity

The Integrity Check provides a set of rules to validate spatial objects. All new or modified spatial objects have to be verified against integrity rules built into the capture and change routines of every application before they can be posted to the database. New and changed objects contain an internal status that requires the validation process.

The Integrity Check is run from the Capture menu, or by clicking the appropriate button on the Integrity toolbar. The way in which the integrity check is executed depends on the preferences/options specified in the Munsys Options dialog box. The various options available on this dialog box are discussed in detail in the Munsys Concepts User Manual: Chapter 4: Munsys Options.

The integrity check is split into two separate functions to facilitate flexibility and ease of use:

- Validate Object Integrity – checks and validates the spatial and/or attribute data of an object, according to the options specified in the Munsys Options dialog box:
  - When the Validate Geometry: Modified objects option has been selected, the geometry of only new or modified objects will be validated.
  - When the Validate Geometry: All objects option has been selected, all the objects in the selection set will have their geometry validated. This includes locked, unlocked, modified and non-modified objects.
  - When the Validate Attributes: Modified attributes option has been selected, the attributes of only new or modified objects will be validated.
  - When the Validate Attributes: All attributes option has been selected, all the objects in the selection set will have their attributes validated. This includes locked, unlocked, modified and non-modified objects.

  When the object integrity check has completed and changes in objects have been encountered that may affect network integrity, a warning message to this effect is displayed if the option was specified in the Munsys Options dialog box.
- Validate Network Integrity – validates the rules that, together with the objects, make up a network, and according to the Network Integrity options specified in the Munsys Options dialog box. Network validation includes the topology and connectivity aspects, where topology is resembled in the geometry of objects and connectivity by means of attributes attached to objects.

  This function is dependent on the relationships between objects, and therefore requires more objects to be included as part of the validation process than only those that were modified. The Validate Network Integrity function allows non-validated objects to be selected, but does not attempt to reset any integrity flags. It only places integrity markers when errors are encountered in the network. If the selection set contains objects that have not been validated by the Validate Object Integrity function, an Object Integrity Warning is displayed, recommending that object integrity has to be checked before network integrity.

Note The *INTEG_NETWORK_AUTCHECK* application setting (Include Object in Network Integrity Check), which is set by the database administrator in the Munsys Management Console, allows a network integrity check to be performed automatically whenever an object integrity check is run. The same network

rules and settings still apply. When the integrity check is completed, the **Integrity Results** dialog box displays a combined list of both object and network validations that were performed during the integrity check. When this setting is enabled, the object integrity check cannot be integrated with the database posting function, and the **Automatically perform Object Integrity** option on the **Munsys Options** dialog box **Preferences** tab will be unavailable.

When drainage object integrity is checked, the following is verified:

Validation check/error condition	Description
Unlocked objects not rectified	Indicates that an object could have been rectified but, because it was unlocked, the integrity check failed to rectify the object and it was left as is.
Short spatial objects	Applies to any object with a length shorter than the Munsys Drainage database tolerance setting.
Duplicate points in spatial objects	Applies to redundant coordinates being removed from an object. If the coordinates are within *SNAP* tolerance, they are discarded.
Objects with incomplete geometry	Refers to objects that do not have geometry, for example: tags that were placed without using the polygon builder process do not have a geometry.
Objects outside database extents	Checks if the objects selected are within the confines of the geographic extents of the database.
Attribute rules	Checks for the attribute rules as specified in the MUNSYS_INTEG_ATTR table, for example NULL values, etc.
Attribute data types	Checks the data type to be consistent with what is in the database; i.e. attached attributes should not exceed the limits of the table's column definition, for example: a tag may not be longer than the table's TAG_VALUE column width.

Table 18 The drainage object integrity check

When drainage network integrity is checked, the following is verified:

Validation check/error condition	Description
Unlocked objects not rectified	Indicates that an object could have been rectified but, because it was unlocked, the integrity check failed to rectify the object and it was left as is.
Pipes within snap tolerance	Checks for pipes, culverts and channels that can snap to other pipes, channels or culverts that are within the snap tolerance.

Pipes without nodes	Checks whether each pipe, channel or culvert is connected to a node at both ends of the pipe.
Duplicate pipes	Checks for two different pipes, culverts or channels that are connected to the same nodes.
Nodes within snap tolerance	Checks for nodes that can be snapped to pipes, culverts or channels that are within the snap tolerance.
Isolated nodes	Checks for nodes that are not connected to any pipes, culverts or channels.
Nodes with too many pipes	Checks for nodes that are connected to too many pipes, culverts or channels.
Nodes with too few pipes	Checks for nodes that are connected to too few pipes, culverts or channels.
Duplicate nodes	Checks for nodes that have been captured on top of one another.

Table 19 The drainage network integrity check

Note The integrity check will transfer pipe levels from a node or pipe, culvert or channel, depending on the existence of level attributes in the object. If the node does not have level attributes set but the pipe, culvert or channel does contain level attributes, the node cover level and depth will be calculated from the pipe, culvert or channel to the node. If the pipe, culvert or channel does not have levels set but the node contains levels, the pipe, culvert or channel levels will be calculated from the node to the pipe, culvert or channel.

Checking drainage object integrity

1 Do one of the following:

- Choose **Capture** > **Data Integrity** > **Validate Object Integrity...**
- Click the **Validate Object Integrity** button on the **Integrity** toolbar.

2 Select the objects that you want to validate, and then press **ENTER**.

The Integrity Results dialog box is displayed, providing summarized information and validation results encountered during the object integrity check.

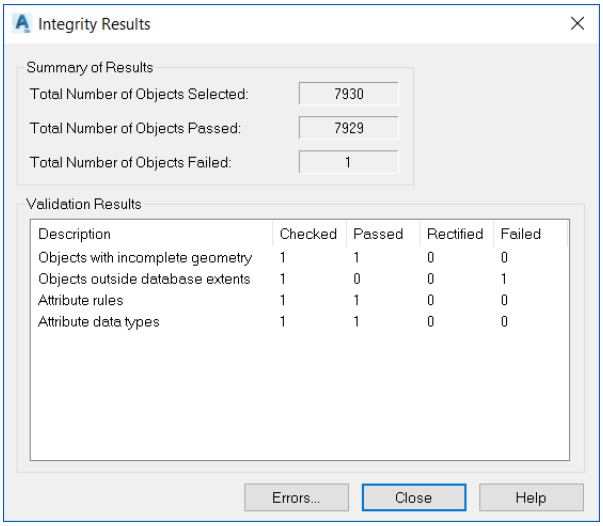


Figure 20 The Integrity Results dialog box

3 If you selected the **Notify when objects require network validation** option on the Munsys Options dialog box, and if objects were encountered during the integrity check that may affect network integrity, the following message is displayed:

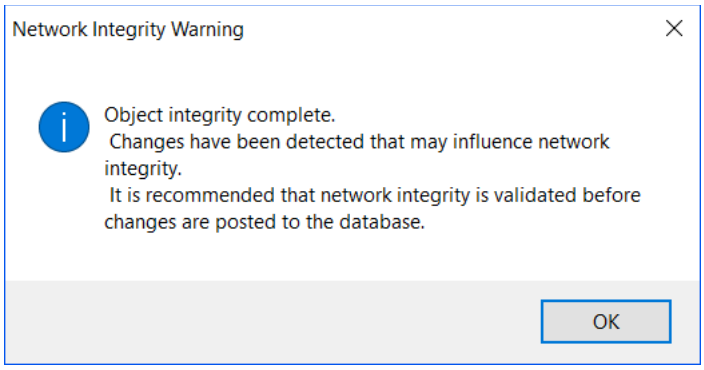


Figure 21 Network Integrity Warning

Checking drainage network integrity

1 Do one of the following:

- Choose **Capture** > **Data Integrity** > **Validate Network Integrity...**
- Click the **Validate Network Integrity** button on the **Integrity** toolbar.

2 Select the objects that you want to validate, and then press **ENTER**.

Note If you selected objects that have not been validated by the **Validate Object Integrity** function, the following message is displayed:

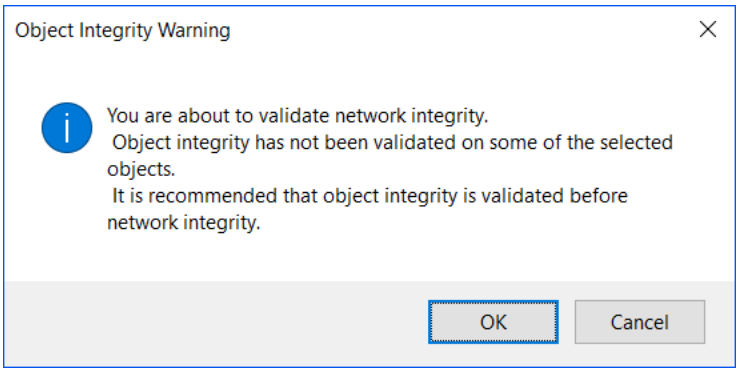


Figure 22 Object Integrity Warning

3 When the integrity check has completed, the **Integrity Results** dialog box is displayed, providing summarized information and validation results encountered during the network integrity check.

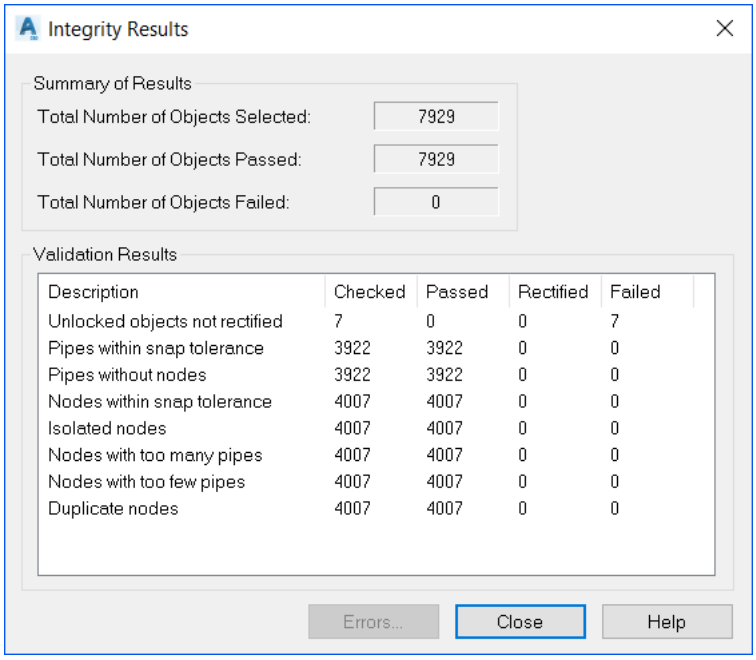


Figure 23 The Integrity Results dialog box

Browsing integrity markers

You can review the integrity markers resulting from the integrity check with the Browse Integrity Markers dialog box. This dialog box displays the complete list of errors encountered during the integrity check. The list consists of the error type, the object affected, a description of the error, and the GID of the spatial object. From this dialog box, you can locate and remove specific integrity markers in the drawing.

To browse integrity markers

1 Do one of the following:

- Choose **Capture** > **Browse Integrity Markers...**
- Click the **Browse Integrity Markers** button on the **Integrity** toolbar.
- Click the **Errors...** button on the **Integrity Results** dialog box.

The Browse Integrity Markers dialog box is displayed.

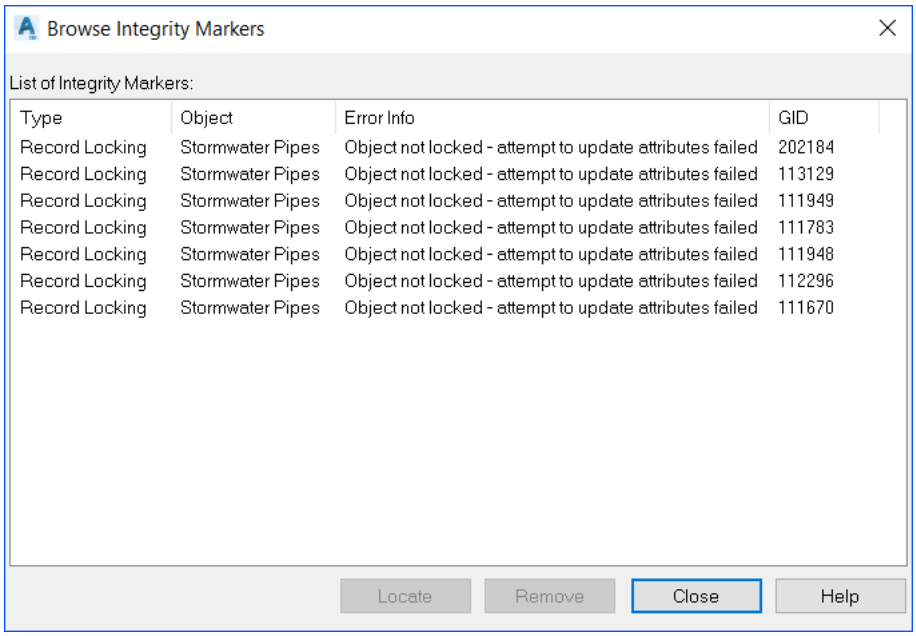


Figure 24 The Browse Integrity Markers dialog box

2 To locate an error, select the appropriate integrity marker, and then click the **Locate** button.

Munsys zooms to the extent of the spatial object to which the integrity marker refers.

3 To remove integrity markers from the list and from the drawing, select one or more integrity markers from the list, and then click the **Remove** button.

The integrity markers are removed from the drawing and from the list of markers.

Erasing integrity markers

With this function, you can erase all the integrity markers. Integrity markers are stored on the INTEG layer.

To erase integrity markers

1 Do one of the following:

- Choose **Capture** > **Erase Integrity Markers**.
- Click the **Erase Integrity Markers** button on the Integrity toolbar.

The command line prompts you for confirmation to erase all the integrity markers.

2 Press **ENTER** to erase the integrity markers.

The integrity markers are erased.

Posting drainage data to the database

New or modified drainage data that has passed the integrity check is posted to the database.

To post drainage data to the database

1 Do one of the following:

- Choose **Capture** > **Post to Database...**
- Click the **Post to Database** button on the **Integrity** toolbar.

The Database Posting Summary dialog box is displayed.

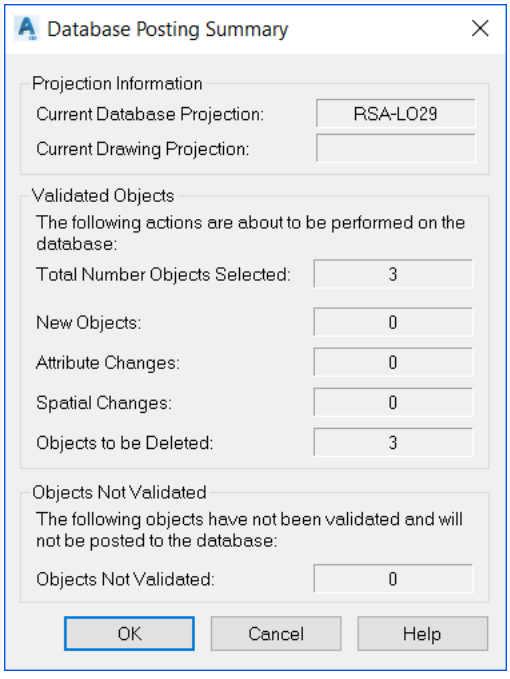


Figure 25 The Database Posting Summary dialog box

This dialog box displays the following:

- The total number of spatial objects selected for posting.
- The number of new objects created.
- The number of attribute changes made.
- The number of spatial changes made.
- The number of objects marked for deletion.
- The number of objects that have not been validated and that will not be posted to the database.

2 Click **OK** to update the database.

The Database Posting Results dialog box is displayed.

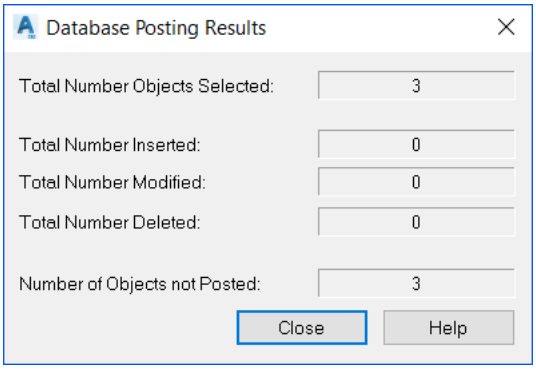


Figure 26 The Database Posting Results dialog box

This dialog box displays the following:

- The total number of objects selected for posting to the database.
- The total number of spatial objects inserted into the database.
- The total number of spatial objects modified in the database.
- The total number of spatial objects deleted from the database.
- The total number of objects not posted to the database.

3 Click **Close** to exit the **Database Posting Results** dialog box.

Note Once you have posted objects to the database, you will not be able to undo this function, which will avoid duplicate objects and keep the content of the drawing synchronized with the database.

Validating object integrity and posting data at the same time

If you select the Automatically perform Object Integrity option on the Munsys Options dialog box Preferences tab, you can perform an object integrity check and database posting operation at the same time. The following rules will be applied when the objects integrity phase of the operation is executed:

- Only modified objects will be selected for object integrity
- The integrity flag will be reset automatically if the object passes object integrity
- Integrity markers will be placed accordingly where errors occur, as happens when a normal object integrity check is run from the Capture menu
- If an error occurs, the integrity flag will not be reset
- Only objects related to the current application will be included in the object integrity check (i.e. if you are working in the Drainage application, only drainage objects will be checked)

If errors occurred during the object integrity check, you can do one of the following:

- Ignore the errors that occurred and continue to post the objects that passed the integrity check to the database
- Cancel the database posting operation and return to the map interface
- View the errors that occurred with the Browse Integrity Markers dialog box (this option also cancels the posting operation, allowing you to correct the errors first before attempting to post the objects to the database once again)

To validate object integrity and post objects at the same time

1 On the **Munsys Options** dialog box **Preferences** tab, select the **Automatically perform Object Integrity** check box in the **Database Posting Preferences** group.

2 Choose **Capture** > **Post to Database.**

An object integrity check is run on the all the new or modified objects in the drawing.

3 If any errors occurred, the following message is displayed:

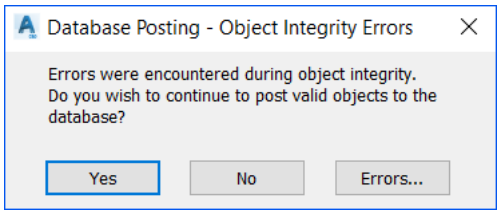


Figure 27 Database Posting – Object Integrity Errors

4 Do one of the following:

Click Yes to ignore the errors that have occurred and post the objects to the database that have passed the integrity check

Click No to cancel the posting operation and return to the drawing

Click Errors... to view the errors that occurred using the Browsing Integrity Markers dialog box.

Drainage network quality reporting

The Network Quality Report function on the Extras menu is used to validate the quality of the entire drainage network. The network quality report complements the network integrity check. Although it is recommended that a network integrity check is always done before objects are posted to the database, some problems might still be present in the quality of a network. This could, for example, be due to the fact that the network integrity check might have been done on a small area of the network. The network quality check uses all the objects in the database to validate the quality of the entire network. The validation is done at attribute level, without having to validate the associated spatial data. Specific columns in each of the spatial tables are used to build up the topology of the network, for example the start and end nodes of a stormwater pipe. The network quality report will validate these values based on drainage-specific rules.

Any network errors that have been found are populated to the MUNSYS_INTEG_RESULTS table, together with the appropriate error message related to each spatial object where an error occurred. The MUNSYS_INTEG_RESULTS table can be used to set up a query, which will use this table as a linked table with a condition based on the SP_TABLE column that matches the spatial table name of the query.

If a single spatial object contained more than one error, a record will be stored for each error that occurred within that object. You can also have the errors populated to the COMMENTS column of the object(s) in which errors occurred, if you do not traditionally use this column for other purposes. If more than one error has occurred in an object, the COMMENTS column is populated with the last error that is found. The following table shows the drainage network errors that may occur:

Spatial Table	Error	Description
SP_SWPIPE	NODE MISSING	The START_NODE/END_NODE value is NULL, or the node ID was not found in the SP_SWNODE table
SP_SWCHANNEL	NODE MISSING	The START_NODE/END_NODE value is NULL, or the node ID was not found in the SP_SWNODE table
SP_SWCULVERT	NODE MISSING	The START_NODE/END_NODE value is NULL, or the node ID was not found in the SP_SWNODE table
SP_SWNODE	ID NULL	The value of the NODE_ID column is NULL.
SP_SWNODE	SEQUENCE EXCEEDED	The value in the NODE_ID column exceeds the current value of the DMS_SWNODE sequence
SP_SWNODE	ID NOT UNIQUE	The NODE_ID column contains a non-unique value
SP_SWNODE	NODE ISOLATED	The node ID was not found in either the START_NODE or END_NODE columns of the SP_SWPIPE, SP_SWCHANNEL or SP_SWCULVERT tables

Figure 28 Drainage network quality errors

To validate drainage network quality

1 Choose **Extras** > **Network Quality Report...**

The command line displays the message: Generating network quality report

2 If errors were found in the network, the following message is displayed:

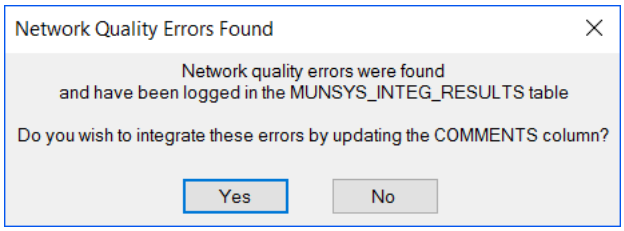


Figure 29 Network Quality errors found

3 Click **Yes** if you want the **COMMENTS** column updated, or **No** if you do not want the column updated.

The Save Results Summary As dialog box is displayed.

4 Save the report to a location of your choice.

The report is opened in Windows Notepad, and contains the following information:

- The number of objects that were checked
- The number of objects found that violated the network rules
- The number of endpoints that were found in the network
- The number of errors that were found, grouped by error type

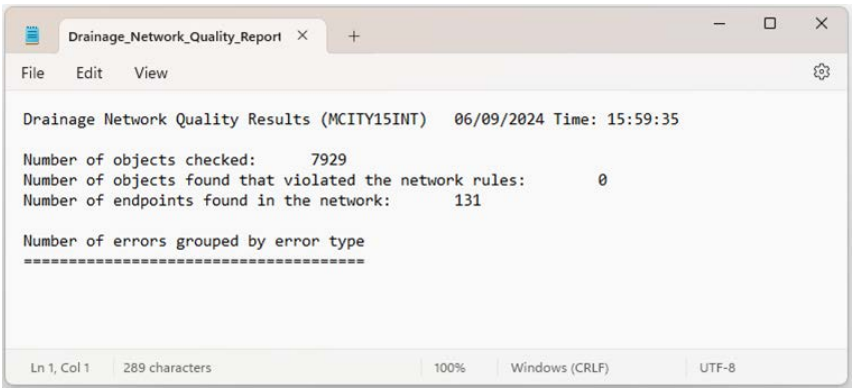


Figure 30 A drainage network quality report

Generating drainage data status reports

Data status reports provide summarized information about spatial objects that are currently stored in the database. Once a report has been generated, it is saved to a comma delimited file, and then displayed in Notepad for easy viewing. The following reports can be generated for drainage objects:

- a summary of all the drainage objects in the database
- a summary of the total length and number of stormwater pipes by diameter
- a summary of the total length and number of stormwater pipes by material
- a summary of the total length and number of stormwater channels by dimension
- a summary of the total length and number of stormwater channels by material
- a summary of the total length and number of stormwater culverts by dimension
- a summary of the total length and number of stormwater culverts by material
- a summary of the number of stormwater nodes, grouped by node type
- a summary of the total length and number of service connections by type
- a summary of length and number of stormwater pipes by diameter and material
- a summary of length and number of stormwater channels by dimension and material
- a summary of length and number of stormwater culverts by dimension and material

To generate a drainage data status report

1 Choose **Extras** > **Data Status Report...**

The Data Status Report dialog box is displayed, showing a list of all the reports that can be generated.

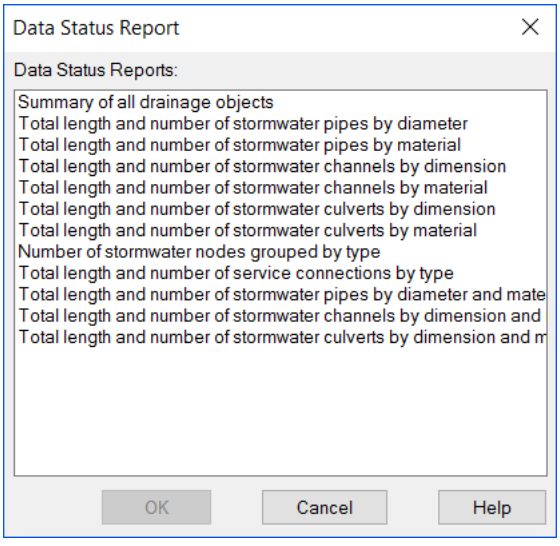


Figure 31 The Data Status Report dialog box

2 Select one or more reports to generate from the list, and then click **OK**.

3 Save the file to a location of your choice.

The file is opened in Windows Notepad.

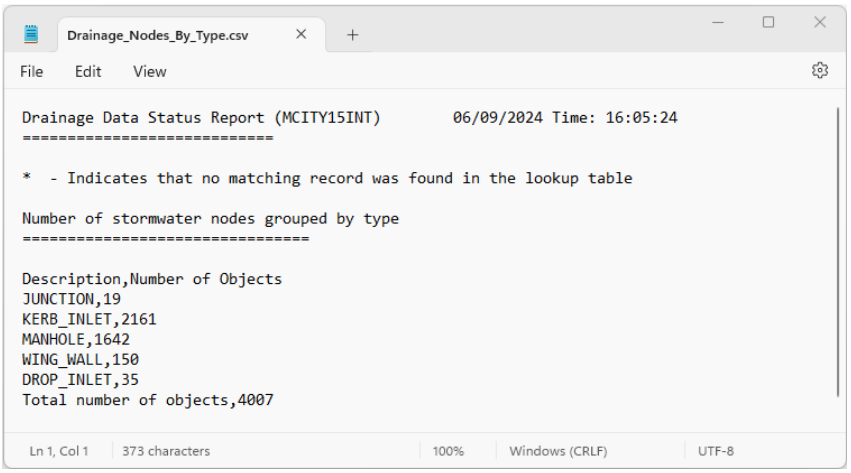


Figure 32 Data status report for stormwater nodes

Converting AutoCAD entities to drainage objects

With Munsys, you can convert selected AutoCAD entities to Munsys Drainage objects. Block entities can be converted to stormwater nodes (curb inlets, drop inlets, manholes, junctions, and wingwalls) or stormwater symbols (cleanout or direction arrow). Line entities can be converted to service connections, pipes, culverts, channels or rivers, text can be converted to tags or drainage notes, and polygons can be converted to rivers, floodlines, dams or catchments.

Although some basic attributes are assigned as part of the conversion process, you will need to assign attributes to the objects created with the conversion process.

To convert block entities to stormwater nodes

With this function, you can convert AutoCAD block entities to any of the following stormwater nodes:

- curb inlets
- drop inlets
- manholes
- junctions
- wingwalls

To convert block entities to stormwater nodes, do the following:

1 Choose **Extras** > **Convert Block to** > ***Node Name***.

2 Select the block entities that you want to convert, and then press **ENTER**.

One stormwater node is created for each block entity that you selected, and the drawing is updated accordingly.

To convert block entities to stormwater symbols

With this function, you can convert AutoCAD block entities to any of the available stormwater symbols.

To convert block entities to stormwater symbols, do the following:

1 Choose **Extras** > **Convert Block to** > **Stormwater Symol**.

2 Select the block entities that you want to convert, and then press **ENTER**.

The Stormwater Symbols dialog box is displayed, showing a list of available stormwater symbols.

3 From the list, select the symbol that you want to place, and then click on **OK.**

One stormwater symbol is created for each block entity that you selected, and the drawing is updated accordingly.

To convert line entities to drainage objects

With this function, you can convert line entities to any of the following drainage objects:

- pipes, culverts and channels
- rivers
- service connections

1 To convert line entities to drainage objects, do one of the following:

- Choose **Extras** > **Convert Line to** > **Pipe**.
- Choose **Extras** > **Convert Line to** > **Culvert**.
- Choose **Extras** > **Convert Line to** > **Channel**.
- Choose **Extras** > **Convert Line to** > **River**.
- Choose **Extras** > **Convert Line to** > **Service Connection**.
  - When converting a line entity to a service connection (SC), the SC must also be linked to the parcel before the PIPE_ID and the PRCL_GID will be added to the respective SC columns. To link a SC to a parcel use the menu item: **Change > Link SC's to Parcels.**

2 Select the line objects that you want to convert to stormwater objects, and then press **ENTER**.

The selected line objects are converted and the drawing is updated accordingly.

To convert text to drainage notes or tags

With this function, you can convert AutoCAD text to the following:

- floodline tags
- dam tags
- catchment tags
- drainage notes

1 To to convert text to drainage tags or notes, do one of the following:

- Choose **Extras** > **Convert Text to** > **River Tag**.
- Choose **Extras** > **Convert Text to** > **Floodline Tag**.
- Choose **Extras** > **Convert Text to** > **Dam Tag**.

- Choose **Extras** > **Convert Text to** > **Catchment Tag**.
- Choose **Extras** > **Convert Text to** > **Drainage Note**.

2 Select the objects that you want to convert, and then press **ENTER**.

The entities are converted to the drainage objects that you specified, and the drawing is updated accordingly

To convert polygons to drainage objects

With this function, you can convert polygons to any of the following drainage objects:

- rivers
- floodlines
- dams
- catchments

1 Do one of the following:

- Choose **Extras** > **Convert Polygon to** > **River**.
- Choose **Extras** > **Convert Polygon to** > **Floodline**.
- Choose **Extras** > **Convert Polygon to** > **Dam**.
- Choose **Extras** > **Convert Polygon to** > **Catchment**.

2 Select the polygons that you want to convert, and then press **ENTER**.

The polygons are converted to the drainage objects that you specified, and the drawing is updated accordingly.

Exporting drainage objects to LandXML

Munsys Drainage provides tools for the engineers and technicians to exchange Munsys storm water data with other engineering software that supports LandXML format. With Munsys Drainage, you can export stormwater node, pipe and culvert object data to LandXML data format.

The exported LandXML data includes pipe and culvert attributes such as material, diameter, gradient, length and unique GID, while the node attributes include the unique Node ID, node type, cover and invert levels and connecting Pipe GIDs.

The XML export function in Munsys Drainage is available for storm water pipes, storm culverts and all drainage nodes. The pipe, culvert and node spatial and attribute data can then be imported into other applications.

LandXML data file Elements

The XML export file consists of 5 main sections, namely:

- Header
- Units
- Project Name
- Application version
- Pipe Networks
  - Node Structures
  - Pipe Structures

Header

The file header records the LandXML language, version number, the date and time that the XML file was created. the file header also references the XML websites displaying XML definitions.

```
<LandXML xmlns="http://www.landxml.org/schema/LandXML-1.1"
xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
xsi:schemaLocation="http://www.landxml.org/schema/LandXML-1.1 http://www.landxml.org/schema/LandXML-1.1/LandXML-1.1.xsd" date="2022-09-11" time="07:45:33" version="1.1" language="English" readOnly="false">
```

Units

The Units for the XML export are defined for both *Imperial* and *Metric*. If the locale for the schema is set to US the Imperial units are applied, whereas if the locale for the schema is set to INT, AUS or AUS2, the Metric units are applied. The Imperial units are used as the default if no locale is set.

```
<Units>
<Imperial areaUnit="squareFoot" linearUnit="foot" volumeUnit="cubicYard" temperatureUnit="fahrenheit" pressureUnit="inchHG" diameterUnit="inch" angularUnit="decimal degrees" directionUnit="decimal degrees" />
</Units>
```

Example of Units for US schemas

<Units>
<Metric areaUnit="squareMeter" linearUnit="meter" volumeUnit="cubicMeter" temperatureUnit="celsius" pressureUnit="mmHG" diameterUnit="millimeter" angularUnit="decimal degrees" directionUnit="decimal degrees" /></Units>

Example of Units for INT, AUS or AUS2 schemas

The following table denotes the different Imperial and Metric units referenced in the XML export file.

Unit Type	Imperial	Metric
Area	squareFoot	squareMeter
Linear	foot	meter
Volume	cubicYard	cubicMeter
Temperature	fahrenheit	celsius
Pressure	inchHG	mmHG
Diameter	inch	millimeter
Angular	decimal degrees	decimal degrees
Direction	decimal degrees	decimal degrees

Figure 33 Imperial and Metric Units

Project Name

The Project information in the XML file refers to the folder name and the AutoCAD drawing name from which the storm water data was exported.

<Project name="C:\Munsys15.1\Munsys.dwg" />

Application version

The Application data creates information pertaining to the Munsys version used to generate the XML file. The Author tag stores the username of the person who created the XML file.

<Application name="Munsys" desc="Munsys" manufacturer="Open Spatial Pty Ltd" version="15.1" manufacturerURL="www.openspatial.com" timeStamp="2022-09-10T08:03:03">
<Author createdBy="MCITY15US" createdByEmail="" company="" companyURL="" timeStamp="2022-12-10T08:03:03" />
</Application>

Note The application version information is not editable from within Munsys.

Node Structures

The Drainage Nodes are exported with the following XML data elements:

```
<Struct name="MUNID-1472" desc="MANHOLE" elevRim="482.268" elevSump="481.268">
<Center>415223.28079906 218553.75873292</Center>
<CircStruct diameter="48.0" thickness="0.5" />
<Invert elev="479.807" flowDir="Out" refPipe="MUNID-113158" />
</Struct>
```

The following table shows the storm water node attributes exported to LandXML format:

Elements	Example	Description
Struct name	"MUNID-1472"	This value refers to the unique node ID (NODE_ID) in SP_SWNODE.
desc	"MANHOLE"	This value refers to the node type (NODE_TYPE) in SP_SWNODE.
elevRim	"482.268"	This value refers to the ground level (NODE_GL) in SP_SWNODE.
elevSump	"481.268"	This value is calculated by subtracting the node depth value (NODE_DEPTH) from the ground level value (NODE_GL) in SP_SWNODE.
Center	415223.28079906 218553.75873292	This value refers to the physical insertion point of the node (GEOMETRY) in SP_SWNODE.
CircStruct diameter	"48.0"	This value is the default diameter assigned to the node.
thickness	"0.5"	This value is the default thickness assigned to the node.
Invert elev	"479.807"	This value is the invert level of the connected pipe or culvert. (END_INVELEV from SP_SWPIPE if *flowDir* is "IN" or START_INVELEV from SP_SWPIPE if *flowDir* is "OUT").
flowDir	"In" or "Out"	This value determines if the pipe or culvertis incoming or outgoing.
refPipe	"MUNID-113158"	This value is the pipe GID of the connected pipe in SP_SWPIPE or the GID of the connected culvert in SP_SWCULVERT.

Figure 34 LandXML Node attributes

Pipe Structures

The Storm Water Pipes are exported with the following XML data elements:

```
<Pipe name="MUNID-113158" refEnd="MUNID-4304" refStart="MUNID-1472" desc="CONC"
length="130.946000" slope="0.03758038">
<CircPipe diameter="33.00" thickness="0.25" />
</Pipe>
```

Example of Storm Water Pipe for US schemas

The following table shows the storm water pipe attributes exported to LandXML format:

Elements	Example	Description
Pipe name	"MUNID-113158"	This value refers to the unique pipe GID in SP_SWPIPE .
refEnd	"MUNID-4304"	This value refers to the end node ID (END_NODE) in SP_SWPIPE.
refStart	"MUNID-1472"	This value refers to the start node ID (START_NODE) in SP_SWPIPE .
desc	"CONC"	This value refers to the pipe material (PIPE_MATRL) in SP_SWPIPE.
length	"130.94600"	This value refers to the length of the pipe (GEOM_LENGTH) in SP_SWPIPE.
slope	"0.03758038"	This value refers to the pipe gradient (PIPE_GRADIENT) in SP_SWPIPE.
CircPipe diameter	"33.00"	This value is the pipe diameter (PIPE_DIA) in SP_SWPIPE.
thickness	"0.25"	This value is the default pipe thickness used to represent the pipe for US schemas.

Figure 35 LandXML Pipe attributes

Culvert Structures

The Storm Water Culverts are exported with the following XML data elements:

```
<Pipe name="MUNID-202298" refEnd="MUNID-4388" refStart="MUNID-4387" desc="CONC"
length="360.889000" slope="0.02727154">
<RectPipe height="2.00" width="60.00" thickness="0.25" />
</Pipe>
```

Example of Storm Water Culvert for US schemas

The following table shows the storm water culvert attributes exported to LandXML format:

Elements	Example	Description
Pipe name	"MUNID-202298"	This value refers to the unique culvert GID SP_SWCULVERT.
refEnd	"MUNID-4388"	This value refers to the end node ID (END_NODE) in SP_SWCULVERT.
refStart	"MUNID-4387"	This value refers to the start node ID (START_NODE) in SWCULVERT.
desc	"CONC"	This value refers to the culvert material (CLVT_MATRL) in SP_SWCULVERT.
length	"360.889000"	This value refers to the length of the culvert (GEOM_LENGTH) in SP_SWCULVERT.
slope	"0.02727154"	This value refers to the culvert gradient (CLVT_GRADIENT) in SP_SWCULVERT.
RectPipe height	"2.00"	This value is the culvert depth (CLVT_DEPTH) in SP_SWCULVERT.
width	"60.00"	This value is the culvert width (CLVT_WIDTH) in SP_SWCULVERT.
thickness	"0.25"	This value is the default culvert thickness used to represent the culvert for US schemas.

Figure 36 LandXML Culvert attributes

Exporting drainage nodes, pipes and culverts to LandXML

Before you export the storm water nodes, pipes and culverts to LandXML format you will first query the drainage network into the AutoCAD drawing area using the Munsys Query functionality. For more information about querying data from the database, please consult the *Munsys Concepts User Manual.* You can select in which projection the data is queried into by selecting the Extras > Set Coordinate System menu option.

To export the drainage network objects, do the following:

1. Choose **Extras** > **Export Objects to** > **Land XML**.
2. Select the drainage entities that you want to export, and then press **ENTER**.

   The command indicates the number of pipes and nodes selected.
3. The **Save File As** dialog box is displayed.
4. On this dialog box, specify the filename and folder where the .xml file will be saved to. The default destination is **C:\Munsys15.1\Munsys.xml**

Figure 37 Example of the LandXML export file

Importing drainage objects from LandXML

By importing XML data you can easily update existing drainage objects' attributes with new values that have been calculated from 3rd party design and analysis applications, such as AutoCAD Civil 3D.

If the Munsys drainage objects have already been queried into the current drawing when the XML file is imported, drainage objects with matching IDs will automatically be updated with the attributes from the XML file. The nodes, pipes and culverts will immediately change to the Munsys Integrity color indicating that attribute values have changed and must be checked before posting to the database.

If the objects being imported from the XML file cannot find existing Munsys objects with matching IDs in the current drawing, new CAD objects are created. The nodes are imported onto a default layer STORM-NODE-MUNSYS, while the pipes and culverts are imported onto the default layer STORM-PIPE-MUNSYS. Using the conversion tools available in Munsys Drainage, these new objects can be converted from CAD entities to drainage objects ready for checking and posting to the database.

To import drainage objects from LandXML

Note The XML width and depth attributes are ignored for updates to culverts when the XML data is imported.

To import the XML data, do the following:

1. Choose **Extras** > **Import Objects from** > **LandXML**.
2. The **Select LandXML to Import** dialog box is displayed.
3. On this dialog box, specify the filename and folder where the .xml file will be imported from. The default folder is **C:\Munsys15.1**
4. Once the filename is selected choose the **Open** button.

   Existing Munsys drainage objects are automatically updated with the XML attribute data, and their color is changed to the Munsys Integrity color.

Exporting drainage objects to CZML

Munsys Drainage provides the tools for users to take the 2 ½D Munsys storm water data and recreate as full 3D geospatial data using the pipe and node invert levels and depths to generate accurate, open source CZML files. CZML is a JSON format for describing graphical data in a 3D context or primarily for display Digital Twin environments such as Cesium or Terria.io, as well as CSIRO's Data61 open data platform. These 3D data files can then be displayed in Digital Twin Environments such as Cesium virtual globe or Terria.io where global 3D terrain, imagery and building infrastructure are used to provide full 3D geospatial visualization in context and integrated with active environments. With Munsys Drainage, you can export stormwater node and pipe object data to CZML data format.

The way the selected data is displayed in the web browser is defined in the configuration file called xtrExportCZML_Def.ini. If you used the default installation path when installing the Munsys applications, the .ini file is located in C:\Program Files\Open Spatial\MunApps15.1. It is recommended that this file be copied to another folder before it is edited.

The configuration file is also used to specify which attributes to display, to set the color and opacity for solid filled objects, to set line widths and colors for line and polyline objects, and to define which object type and color to represent node type objects.

The resultant CZML file contains a single JSON array for each object selected in the drawing, where each object-literal element in the array is a CZML Packet. A CZML Packed describes the graphical properties for a single object in the scene, such as a drainage node and pipe.

The user must first query the drainage nodes and pipes into the drawing that they want to include in the CZML file. The definition file must be configured to include the attribute parameters to be exported with the spatial object. Drainage culverts and channels are not supported for the CZML export process.

CZML Export configuration file

The configuration file (xtrExportCZML_Def.ini) is installed as part of the Munsys Applications and stores all the parameters and settings required to interpret data which is "converted" to a CZML format.

The default configuration file *xtrExportCZML.ini* resides in the application folder *C:\Program Files\Open Spatial\MunApps15.1* is populated with "example" parameters. It is recommended that this configuration file be copied to *C:\Munsys15.1* before it can be edited.

The following parameters are required to export storm water pipes to CZML format:

- **ID_DESCRIPTION**: Indicates the object is a drainage pipe i.e., Pipe
- **NAME_DESCRIPTION**: Populates the Name variable in the CZML packet header i.e. Stormwater Pipe.
- **MUN_ID:** The Unique identifier (MUN_ID) which references the spatial table within the Munsys schema from which the spatial and attribute data is extracted, i.e., 65 = SP_SWPIPE
- **START_HEIGHT**: Reads the Start Invert Level value from the column START_IL.
- **END_HEIGHT**: Reads the End Invert Level value from the column END_IL.

- **PIPE_DIA**: Reads the culvert width value from the column PIPE_DIA.
- **COLOR_RGBA**: Sets the color used to represent the storm water pipe using the RGBA format [255,255,255,255]
- **ATTRIBUTE_COLUMN_NAMES**: List of all attribute columns to be included in the CZML export file and there may not be spaces between the comma separators, i.e., PIPE_-TYPE,PIPE_MATRL,PIPE_DIA,START_GL,END_GL.
- **ATTRIBUTE_COLUMN_DESCRIPTIONS**: List of all attribute column descriptions to be included in the CZML export file. The order of the column descriptions must match the order of the column names, and there may not be spaces between the comma separators, i.e., Pipe Type,Pipe Material,Pipe Diameter,Start Ground Level,End Ground Level.
- **CZML_OUTPUT_TYPE**: The Object type used to represent the spatial object. Storm Pipes can be represented as a PIPEVOLUME output type.

```
[STORMWATER PIPE - Gravity]
ID_DESCRIPTION=Pipe
NAME_DESCRIPTION=Stormwater Pipe Gravity
MUN_ID=65
START_HEIGHT=START_IL
END_HEIGHT=END_IL
PIPE_DIA=PIPE_DIA
COLOR_RGBA=0,255,0,255
ATTRIBUTE_COLUMN_NAMES=PIPE_TYPE,PIPE_MATRL,PIPE_DIA,START_GL,END_GL
ATTRIBUTE_COLUMN_DESCRIPTIONS=Pipe Type, Pipe Material,Pipe Diameter,Start Ground
Level,End Ground Level
CZML_OUTPUT_TYPE=PIPEVOLUME
```

Figure 38 Example of a CZML export file for Storm water pipes

The following parameters are required to export storm water manholes to CZML format:

- **ID_DESCRIPTION**: Indicates the object is a storm water node i.e., Node
- **NAME_DESCRIPTION**: Populates the Name variable in the CZML packet header i.e. Stormwater Manhole.
- **MUN_ID:** The Unique identifier which references the spatial table within the Munsys schema from which the spatial and attribute data is extracted, i.e., 63 = SP_SWNODE
- **NODE_TYPE:** Filters the selection based on the node type value from the column NODE_TYPE i.e: MANHOLE
- **NODE_HEIGHT:** Reads the Ground Level value from the column NODE_GL.
- **NODE_DEPTH:** Reads the Node Depth value from the column .NODE_DEPTH.
- **COLOR_RGBA:** Sets the color used to represent the storm water node using the RGBA format [255,255,255,255] i.e., 128,255,128,255
- **ATTRIBUTE_COLUMN_NAMES**: List of all attribute columns to be included in the CZML export file and there may not be spaces between the comma separators, i.e. NODE_ID,NODE_FUNC,NODE_REF.
- **ATTRIBUTE_COLUMN_DESCRIPTIONS**: List of all attribute column descriptions to be included in the CZML export file. The order of the column descriptions must

match the order of the column names, and there may not be spaces between the comma separators, i.e. Node ID,Node Function, Node Referrence.

- **CZML_OUTPUT_TYPE**: The Object type used to represent the spatial object. Manholes can be defined as a TUBE.
- **TUBE_RADIUS**: Sets the radius for the TUBE object type.
- **COLOR_RGBA:** Sets the color used to represent the storm water manhole using the RGBA format [255,255,255,255]

```
[STORMWATER NODE - Manhole]
ID_DESCRIPTION=Node
NAME_DESCRIPTION=Stormwater Manhole
MUN_ID=63
NODE_TYPE=MANHOLE
NODE_HEIGHT=NODE_GL
NODE_DEPTH=NODE_DEPTH
ATTRIBUTE_COLUMN_NAMES=NODE_ID,NODE_FUNC,NODE_REF
ATTRIBUTE_COLUMN_DESCRIPTIONS=Node ID,Node Function,Node Reference
COLOR_RGBA=0,0,255,255
CZML_OUTPUT_TYPE=TUBE
TUBE_RADIUS=1.5
```

Figure 39 Example of a CZML export file for Storm water manholes

The following parameters are required to export storm water curb/kerb inlets to CZML format:

- **ID_DESCRIPTION**: Indicates the object is a storm water node, i.e. Node
- **NAME_DESCRIPTION**: Populates the Name variable in the CZML packet header i.e. Stormwater Kerb Inlet.
- **MUN_ID:** The Unique identifier which references the spatial table within the Munsys schema from which the spatial and attribute data is extracted, i.e., 63 = SP_SWNODE
- **NODE_TYPE:** Filters the selection based on the node type value from the column NODE_TYPE, i.e: KERB_INLET
- **NODE_HEIGHT:** Reads the Ground Level value from the column NODE_GL.
- **NODE_DEPTH:** Reads the Node Depth value from the column NODE_DEPTH
- **COLOR_RGBA:** Sets the color used to represent the storm water node using the RGBA format [255,255,255,255] i.e., 128,255,128,255
- **ATTRIBUTE_COLUMN_NAMES**: List of all attribute columns to be included in the CZML export file i.e., NODE_ID,NODE_FUNC,NODE_REF
- **ATTRIBUTE_COLUMN_DESCRIPTIONS**: List of all attribute column descriptions to be included in the CZML export file. The order of the column descriptions must match the order of the column names i.e., Node ID,Node Function,Node Reference.
- **CZML_OUTPUT_TYPE**: The Object type used to represent the spatial object. Kerb Inlets can be defined as a BOX.

- **BOX_SIZE**: Sets the Length and With dimensions for the BOX object type which can be read from the Node Sizes defined for the Kerb Inlet i.e. NODE_SIZE1,NODE_-SIZE2

```
[STORMWATER NODE - Kerb Inlet]
ID_DESCRIPTION=Node
NAME_DESCRIPTION=Kerb Inlet
MUN_ID=63
NODE_TYPE=KERB_INLET
NODE_HEIGHT=NODE_GL
NODE_DEPTH=NODE_DEPTH
COLOR_RGBA=128,255,128,255
ATTRIBUTE_COLUMN_NAMES=NODE_ID,NODE_FUNC,NODE_REF
ATTRIBUTE_COLUMN_DESCRIPTIONS=Node ID,Node Function,Node Reference
CZML_OUTPUT_TYPE=BOX
BOX_SIZE=NODE_SIZE1,NODE_SIZE2
```

Figure 40 Example of a CZML export file for Storm water Kerb inlets

Exporting drainage nodes and pipes to CZML

Before you export the storm water nodes and pipes to CZML format you will first query the drainage network into the AutoCAD drawing using the Munsys Query functionality. For more information about querying data from the database, please consult the Munsys Concepts User Manual. When the data is exported to CZML format, the geometry for each object is transformed to WGS84 (X;Y;Z values).

To export the drainage network objects, do the following:

1 Choose **Extras > Export Objects to CZML**.

2 The **CZML ini File** dialog box is displayed.

Navigate to the folder where the modified configuration file was copied to i.e. *C:\Munsys15.1*, else navigate to *C:\Program Files\Open Spatial\MunApps15.1*.

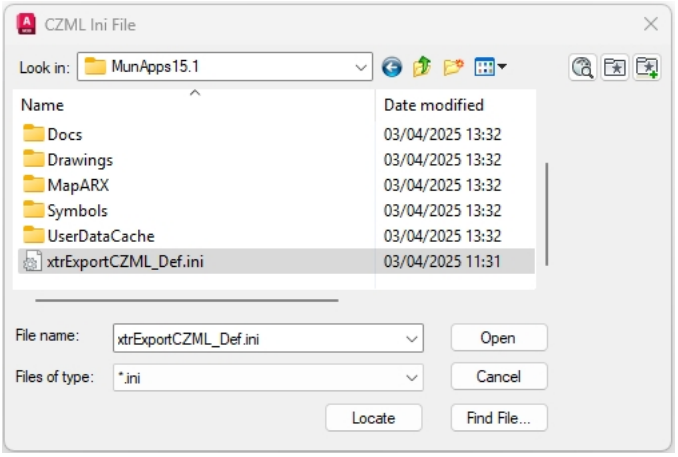


Figure 41 Selecting the Drainage Configuration file

3 Select the configuration file to apply to the export, then select **OPEN**.

4 The **Save File As** dialog box is displayed and defaults to the folder *C:\Munsys15.1*.

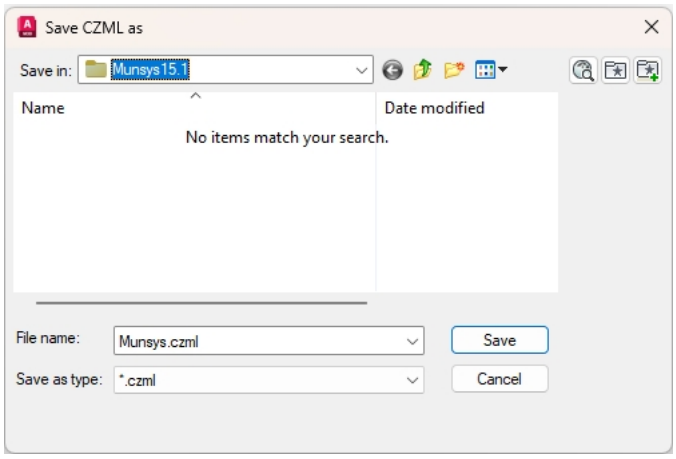


Figure 42 Entering the Drainage Export File name

5 On this dialog box, specify the file name and folder where the resultant .czml file will be saved to. The default destination folder is **C:\Munsys15.1**.

6 In AutoCAD, at the command line, you are prompted to select the drainage entities to export, and then press **ENTER**.
The command indicates the number of objects selected.

7 At the command line the date and time stamp of when the export started and ended is displayed.

```
Select objects:
"28/03/2024 Time: 09:55:30"
Export Completed...
"28/03/2024 Time: 09:55:31"
Command:
```

8 Opening the CZML file in a text editor displays the file contents which contains a single JSON array for each object selected in the drawing where each object-literal element in the array is a CZML Packet.

```
[
{
"id": "document",
"name": "CZML Geometries: Polyline",
"version": "1.0"
}
,
{
"id": "Pipe5",
"name": "Stormwater Pipe Gravity",
"description": "<table><tr><td>Pipe Diameter</td><td>600</td></tr><tr><td>Pipe Type</td><td>D50</td></tr><tr><td> Pipe Material</td><td>CONC</td></tr><tr><td>Start Ground
Level</td><td>1616.484</td></tr><tr><td>End Ground Level</td><td>1616.293</td></tr></table>",
"polylineVolume": {
"positions": {
"cartographicDegrees": [28.25111955063326,-26.07565894609591, 1614.731, 28.25122277785745,-26.07564467872116, 1614.540]
},
"shape": {
"cartesian2":
[0.283655572679795,-0.097670446337147,0.2331437884370913,-0.1887961173149511,0.1545114224730162,-0.2571501902106336,0.05724269861296327,-0.2944881550342992,-0.046930339512061
931,-0.2963065021785412,-0.1454428860739012,-0.2623859121418186,-0.2264128740668315,-0.1968177086971522,-0.2800741279491605,-0.1075103848635901,-0.2999543085469173,-0.005235
721931184958,-0.283655572679795,0.09767044633714696,-0.2331437884370912,0.1887961173149512,-0.1545114224730162,0.2571501902106336,-0.05724269861296344,0.2944881550342992,0.0
4693033951206927,0.2963065021785413,0.1454428860739011,0.2623859121418187,0.2264128740668315,0.1968177086971521,0.2800741279491605,0.1075103848635901,0.2999543085469173,0.0052
35721931185053]
},
"cornerType": "BEVELED",
"material": {
"solidColor": {
"color": {"rgba": [0, 255, 0, 255]
}
}
}
,
"clampToGround": true
}
}
,
{
"id": "Node3",
"name": "Kerb Inlet",
"description": "<table><tr><td>Node ID</td><td>451</td></tr><tr><td>Node Function</td><td></td></tr></table>",
"position": {
"cartographicDegrees": [28.25144415448063,-26.07581552020803, 1614.796]
},
"box": {
"dimensions": {
"cartesian": [NODE_SIZE1, NODE_SIZE2, 1.753]
},
"material": {
"solidColor": {
"color": {"rgba": [128, 255, 128, 255]
}
}
}
}
}
,
```


Figure 43 Example of the CZML export file for a Storm water pipe and node object

Chapter 5
Maintaining drainage data

Modifying drainage data

Spatial data is stored in spatial and attribute tables as records. Users can edit spatial data by retrieving it from the database, and then changing characteristics or attributes using the Change menu and toolbar. Drainage objects are retrieved onto their respective layers, for example, SWPIPE, SWNODE or SWNOTE. Attribute data can be edited directly in its table format.

Once stormwater pipes, culverts, channels and nodes have been placed, they have to be manipulated to make certain that they are joined correctly, for example with no overlaps or undershoots. Each pipe has to have a start and end node.

Attribute values linked to drainage objects can be modified easily from the Change menu, for example pipe diameter, width and depth, and river, floodline and dam types.

Drainage objects that have been changed need to be validated against the applicable business rules before they can be posted to the database.

Changing drainage objects

To extend a drainage object to a boundary

Drainage objects (stormwater pipes, culverts, channels and rivers) can be extended to a boundary object by first indicating the boundary object, and then selecting the object to extend. The boundary object must be able to intersect with the drainage object. This function is often used to extend pipes, channels or culverts to create Tjunctions during capture. The end point of the object is then changed to intersect the boundary object.

1 Do one of the following:

- Choose **Change** > **Extend Drainage Object to Boundary**.
- Click the **Extend Drainage Object to Boundary** button on the Munsys Drainage **Change** toolbar.

2 Select the boundary object where the object has to extend to.

3 Select the object that needs to extend.

The object is extended as indicated.

To extend a drainage object by distance

This function extends a drainage object (pipe, culvert, channel or river) with a specified distance at the endpoint closest to a selected point on the object. You are prompted for the distance to extend the object with. You can also use this function to shorten a drainage object by entering a negative distance, for example -50.

1 Do one of the following:

- Choose **Change** > **Extend Drainage Object by Distance**.
- Click the **Extend Drainage Object by Distance** button on the Munsys Drainage **Change** toolbar.

2 Select the object that you want to extend.

3 Enter a distance on the command line.

The drainage object is extended as indicated.

To extend a drainage object and break it at the intersection

This function extends a drainage object (pipe, culvert, channel or river) to a boundary pipe, culvert, channel or river that is then broken at the intersection. The object to be broken is selected first, and then the object to extend. The object to be broken must be able to intersect with the second object.

1 Do one of the following:

- Choose **Change** > **Extend and Break Drainage Object**.
- Click the **Extend and Break Drainage Object** button on the Munsys Drainage **Change** toolbar.

2 Select the drainage object that you want to break.

3 Select the object that you want to extend.

The objects are extended and broken respectively, as indicated.

To break a drainage object

This function breaks a drainage object (pipe, culvert, channel or river) nearest to a selected point on the object.

1 Do one of the following:

- Choose **Change** > **Break Drainage Object**.
- Click the **Break Drainage Object** button on the Munsys Drainage **Change** toolbar.

2 Select the drainage object that you want to break.

3 Specify a break point on the object that you selected.

The drainage object is broken at the break point that you specified.

To change a drainage object

With this function, you can move the endpoint of one or more selected drainage objects (pipes, culverts or channels) to a new location.

1 Do one of the following:

- Choose **Change** > **Change Drainage Object**.
- Click the **Change Drainage Object** button on the Munsys Drainage **Change** toolbar.

2 Select the drainage object that needs to move.

3 Specify the destination point.

The end point of the object is moved to the destination point that you specified.

To fillet a drainage object

This function connects two drainage objects (pipes, channels, culverts or rivers) at an apparent intersection.

1 Do one of the following:

- Choose **Change** > **Fillet Drainage Object**.
- Click the **Fillet Drainage Object** button on the Munsys Drainage **Change** toolbar.

The command line will prompt you to indicate the two drainage objects that need to be extended or trimmed.

2 Select the first object.

3 Select the second object.

The objects are connected as indicated.

To join drainage objects

This function is used to join two drainage objects (stormwater pipes, culverts or channels), creating a single object. The attributes from the first object selected remain the attributes for the joined object. If there is a gap between the two objects to be joined, they are joined with a line segment. A joined object is created from the geometry of the first object selected, and the second object selected is moved to the DELETED layer. The first selected object, now the new joined object, is flagged as a change to be verified when the integrity check is run.

1 Choose **Change** > **Join Drainage Object**.

2 Select the first drainage object (the object that you want to join to).

3 Select a point close to the endpoint of the second object (the object that you want to join).

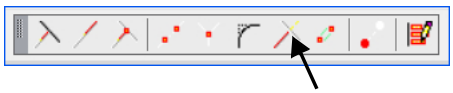
The objects are joined as indicated, and the attributes from the first object selected remain the attributes for the joined object.

To trim a drainage object

You can trim a section of a drainage object (pipe, culvert, channel or river) by first selecting the cutting edge, and then selecting the section of the object to be removed. The cutting edge must be a line that intersects the drainage object.

1 Do one of the following:

- Choose **Change** > **Trim Drainage Object**.
- Click the **Trim Drainage Object** button on the Munsys Drainage **Change** toolbar.



2 Select the object that you want to trim to.

3 Select a point on the side of a second object that needs to be trimmed.

Munsys trims the last indicated drainage object to the first one.

To change the vertex of a drainage object

With this function, you can remove, move, add or insert new vertices to segments in a stormwater pipe, culvert, channel or river.

To add a vertex to a drainage object

1. Choose **Change** > **Change Drainage Object Vertex** > **Add**.
2. Select the applicable drainage object.
3. Select the segment endpoint.
4. Select the position for the new point.

   A vertex is added at the new point that you selected.

To move a drainage object vertex

1. Choose **Change** > **Change Drainage Object Vertex** > **Move**.
2. Select the applicable drainage object.
3. Specify a point closest to the vertex that you want to move.
4. Specify the point that you want to move the vertex to.

   The vertex is moved to the point that you specified.

To remove a drainage object vertex

1. Choose **Change** > **Change Drainage Object Vertex** > **Remove**.
2. Select the applicable drainage object.
3. Select the vertex that you want to remove from the drainage object.

Note A vertex can only be removed from a drainage object containing more than two vertices.

To insert a vertex into a drainage object

1. Choose **Change** > **Change Drainage Object Vertex** > **Insert**.
2. Select the applicable drainage object.
3. Select the segment for the inserted point.

   Select the position for the new point.

   The vertex is inserted in the position you selected.

Moving a node

With this function, you can move a node to a new location by first selecting the node, and then specifying the point where you want to place it. This function is often used to move a node to a new pipe, channel or culvert endpoint.

To move a node

1 Do one of the following:

- Choose **Change** > **Move Node**.
- Click the **Move Node** button on the Munsys Drainage **Change** toolbar.

2 Drag-and-click the node to the desired destination point.

The node is moved to the destination point you indicated.

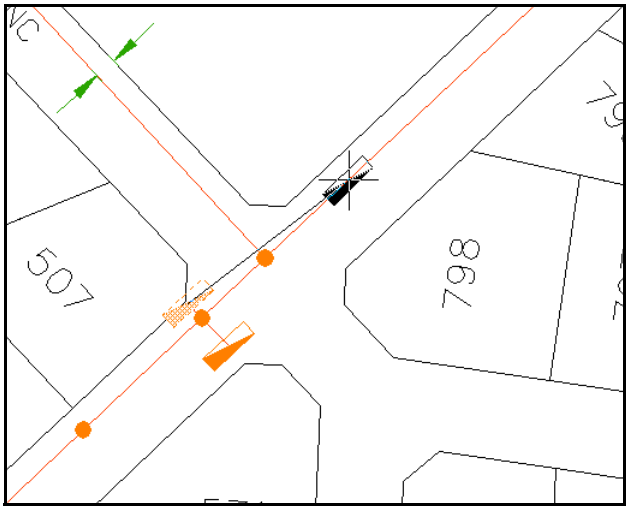


Figure 1 Moving a node

Rotating a node

With this function, you can rotate one or more selected nodes. The rotation angle is specified by typing the angle on the command line, indicating the angle with the mouse pointer or specifying an object to align the node with.

To rotate a node

1 Do one of the following:

- Choose **Change** > **Rotate Node**.
- Click the **Rotate Node** button on the Munsys Drainage **Change** toolbar.

2 Select the node that you want to rotate.

3 To specify the rotation angle, do one of the following:

- Enter the rotation angle on the command line, or press **ENTER** to accept the default angle.
- Indicate the angle with your mouse pointer by picking two points in the drawing.
- On the command line, type **A** to align the node to a segment, press **ENTER**, and then select the object to which you want to align the node.

The node is rotated as specified.

Snapping a node to the endpoint of a drainage object

With this function, you can snap a node to the endpoint of a drainage object. The node is moved to the end point closest to the point selected on the drainage object.

To snap a node to the endpoint of a drainage object

1 Do one of the following:

- Choose **Change** > **Snap Node to Endpoint**.
- Click the **Snap Node to Endpoint** button on the Munsys drainage **Change** toolbar.

2 Select the applicable node.

3 Select the drainage object close to the endpoint that you want to snap the node to.

The node is snapped to the endpoint of the drainage object.

Snapping a node to the endpoint of a drainage object, breaking the drainage object

With this function, you can break a drainage object and snap a node to the endpoint of the broken drainage object

To snap a node and break a drainage object

1 Do one of the following:

- Choose **Change** > **Snap Node and Break Drainage Object**.
- Click the **Snap Node and Break Drainage Object** button on the Munsys drainage **Change** toolbar.

2 Select the applicable node.

3 Select the drainage object that you want to snap the node to.

4 Specify a break point on the drainage object.

The node is snapped to the endpoint of the broken drainage object.

Changing a node type

With this function, you can change the node type of one or more selected nodes. The new node type is selected from a list. The symbol name and the NODE_TYPE column are updated according to the new node type selected.

To change a node type

1 Choose **Change** > **Change Node Type...**

2 Select the node(s) that you want to change, and then press **ENTER**.

The Node Types dialog box is displayed.

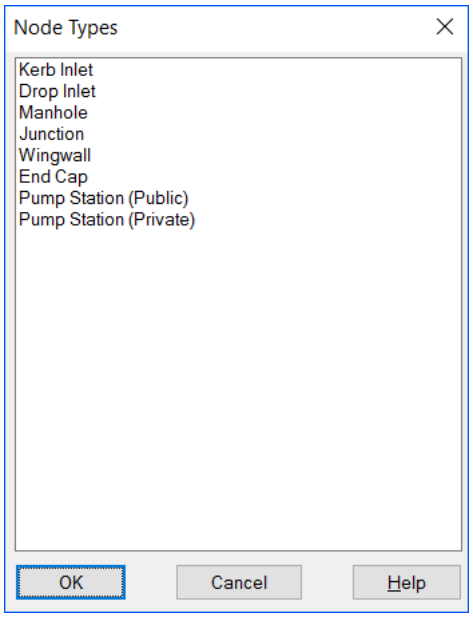


Figure 2 The Node Types dialog box

3 Select the new node type from the list, and then click **OK**.

The node type is changed as indicated. It is flagged as a change to be verified with the integrity check, and the new node is displayed in the integrity color.

Changing drainage attribute data

With Munsys Drainage, you can change the physical attributes of stormwater objects, for example material, diameter, type, depth, etc. You can also change pipe, culvert, channel and node levels. You can also change a service connection type and link service connections to their respective parcels.

Editing drainage object attributes

The Edit Attributes function is used to edit the attributes of one or more selected drainage objects that belong to the same object type. The current application determines what object types may be selected; for example, if you are working in Munsys Drainage, only drainage objects may be selected. If you select more than one object type (for example pipes and nodes), you will be required to choose a single object type to edit.

Objects that are selected for editing are locked (if record locking is enabled in the database). Once the objects have been selected and an object type to edit has been specified, the Edit Attributes dialog box is displayed. This dialog box contains various options that can be used to edit the attributes of the spatial objects:

- Attribute templates – attributes can be edited using an attribute template. The attributes that will be displayed when an attribute template has been selected will depend on the attributes that have been specified on the template, as well as the formatting parameters that apply to each column selected as part of the attribute template. If no attribute template has been selected, all the attributes belonging to the spatial object type are displayed.
- Editing multiple objects simultaneously – if you select this option, attributes of multiple objects are displayed simultaneously. Where the attributes of all the selected objects are the same, a value is displayed. Where attributes of the various objects that were selected differ, a value of *VARIES* is displayed. When a change is made to a value, the new value is applied to all the selected objects. If you do not select the Edit multiple objects simultaneously check box, you can edit the selected objects one by one. The values that are changed are only applied to the object that is currently selected.
- AutoZoom – this option is only available when the Edit multiple objects simultaneously option is not selected, i.e. when you are going to edit the objects one by one. If the AutoZoom to object option is selected, Munsys will zoom to each object in the drawing as it becomes the current object, and highlight it. The object navigation buttons are used to move from one object to the next.

The various attributes are displayed on the Edit Attributes dialog box in three columns: Description, Value and Column Name. You can resize the dialog box for easier viewing. Values that may not be edited are unavailable. Attributes are edited in the *Value* column, either by typing a new value or by selecting a value from a drop-down list. Changes that are made are applied to objects in the drawing. These changes will only be updated in the database when the object integrity check has been run and the objects have been posted to the database.

When editing attributes, mandatory fields are displayed with an "!" character prefixing the attribute column in the Attributes list on the Edit Attributes dialog box. If the mandatory attribute is not captured for new or modified objects, the Object Integrity Check will fail based on the attribute rules setup. Using the Browse Integrity Markers option, the Error Info describes which column may not be NULL.

To edit drainage objects

1 Choose **Change** > **Edit Attributes...**

2 Select the objects that you want to edit, and then press ENTER.

If you selected more than one object belonging to different object types, the Spatial Object Identification dialog box is displayed.

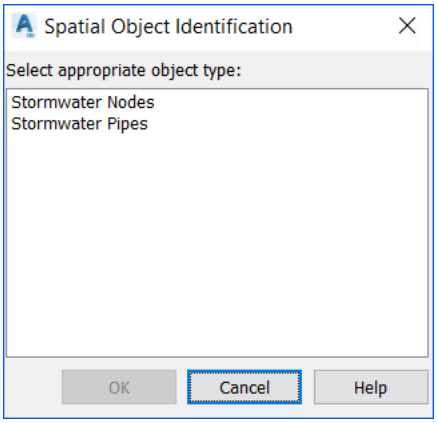


Figure 3 The Spatial Object Identification dialog box

3 Select a single spatial object type to edit, and then click OK.

The Edit Attributes: [*Object Type*] dialog box is displayed.

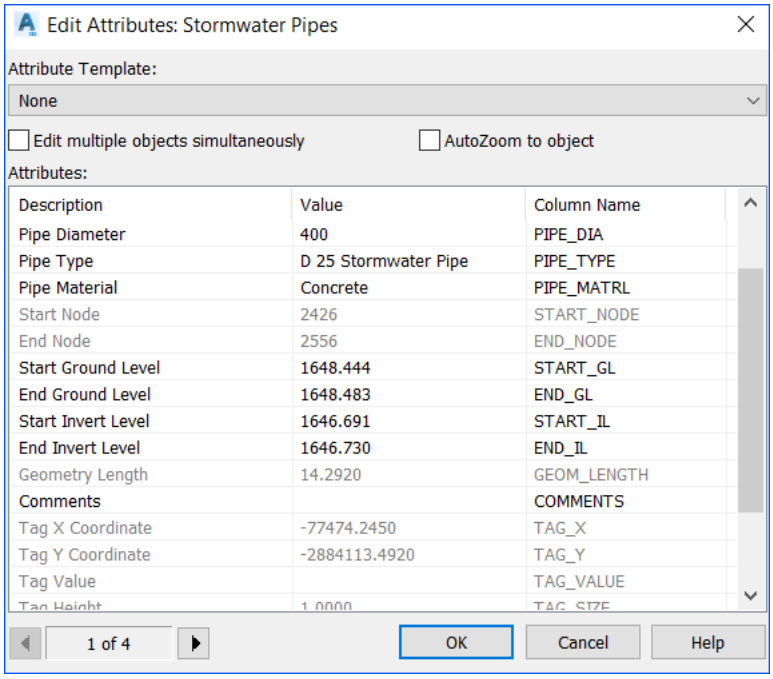


Figure 4 The Edit Attributes dialog box

4 If you want to edit the objects using an attribute template, select the appropriate attribute template from the **Attribute Template** list. If you do not want to make use of an attribute template, select **None**.

5 If you want to edit the attributes of all the selected objects at the same time, select the **Edit multiple objects simultaneously** check box.

Similar values are displayed, while values that differ between objects are displayed as *VARIES*, as seen in the following figure:

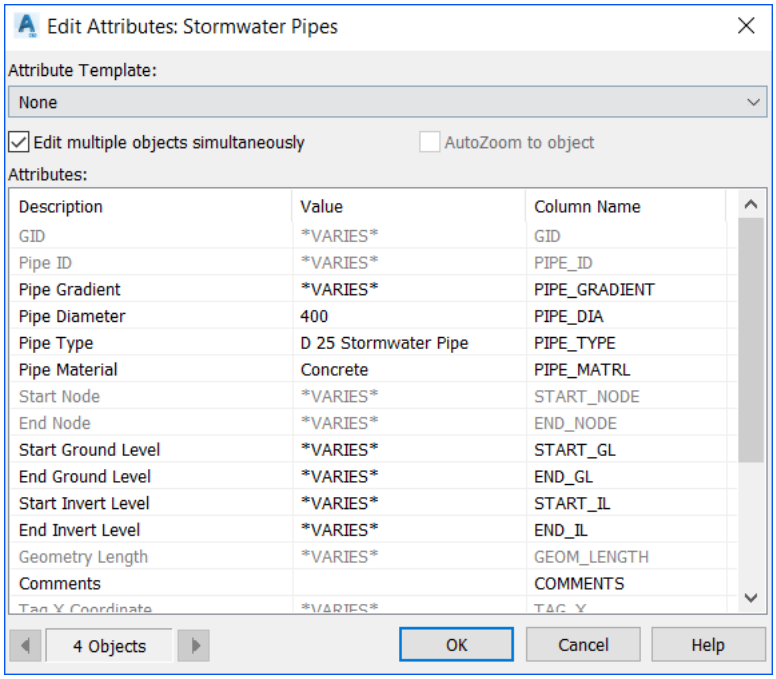


Figure 5 The Edit Attributes dialog box: Edit multiple objects simultaneously

6 If you want to edit the selected objects one by one, clear the **Edit multiple objects simultaneously** check box. Select the **AutoZoom to object** check box to zoom to the objects one by one and highlight them. The object navigation buttons are used to move from one object to the next.

7 To edit an attribute, do one of the following:

- Highlight a value, and then choose a new value from the drop-down list in the **Value** column.
- Highlight a value, and then enter a new value in the **Value** column

Note Mandatory attributes are displayed with an "**!**" character preceding the Attribute Description. Failure to capture mandatory attributes for new or modified objects results in the object failing the Object Integrity Check.

8 Click **OK** to apply the new value(s) to the object(s).

9 The values are applied to the objects, to be verified with the object integrity check.

Editing linked table attributes

This function is used to edit spatial object attributes that exist in linked tables. Linked table attributes are updated directly to the database. The current application determines what object types may be selected; for example, if you are working in Munsys Drainage, only drainage objects may be selected. If you select more than one object type (for example pipes and nodes), you will be required to choose a single object type to edit. You can only select objects that have previously been posted to the database.

The link tables that can be edited using this function must be defined through an attribute template. The applicable attribute template is selected from a list, which is created by filtering attribute templates according to the following characteristics:

- Only attribute templates associated with the current spatial object type, being edited are included – the primary table in the attribute template is the same table from which the spatial objects were queried.
- Only attribute templates that contain linked tables are included
- The linked table specified may not be a Munsys spatial table
- The user must have edit privileges to the linked table
- The link column used to describe the relationship with the primary table must reference a true column in the linked table. This may not be an expression.

Attributes are displayed on the Linked Table Attributes dialog box in three columns: Description, Value and Column Name. You can resize the dialog box for easier viewing. Values that may not be edited are unavailable. Attributes are edited in the Value column by typing in a new value. Navigation buttons are used to move between the various records that are available and to move from one object to the next. You can also insert records into, or delete records from a linked table, depending on the privileges you have on the link table.

The procedure for editing linked table attributes is described in detail in the *Munsys Concepts User Manual*.

Changing pipe, culvert, channel and node levels

This function is used to change ground and invert levels for the start and endpoint of a stormwater pipe, channel or culvert during editing. Ground and invert levels are stored with these objects to cater for a drop manhole, where incoming pipes, channels or culverts have a higher invert level than the outgoing ones.

To change pipe levels

1 Choose **Change** > **Pipe Levels...**

2 Select the appropriate stormwater pipe.

The Pipe Information dialog box is displayed.

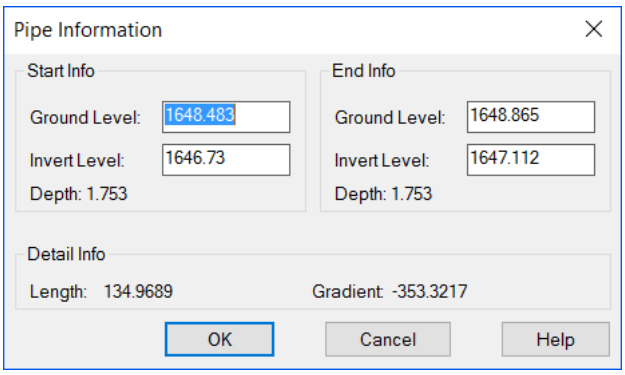


Figure 6 The Pipe Information dialog box

3 From the **Start Info** group, change the following:

- Ground level
- Invert level

The Depth value is updated automatically.

4 From the **End Info** group, change the following:

- Ground level
- Invert level

The Depth value is updated automatically.

The pipe length and gradient values in the Detail Info group are calculated automatically from the Start Info and End Info group.

5 Click **OK** to accept the changes you have made.

The attribute changes are accepted.

To change culvert levels

1 Choose **Change** > **Culvert Levels...**

2 Select the appropriate culvert.

The Culvert Information dialog box is displayed.

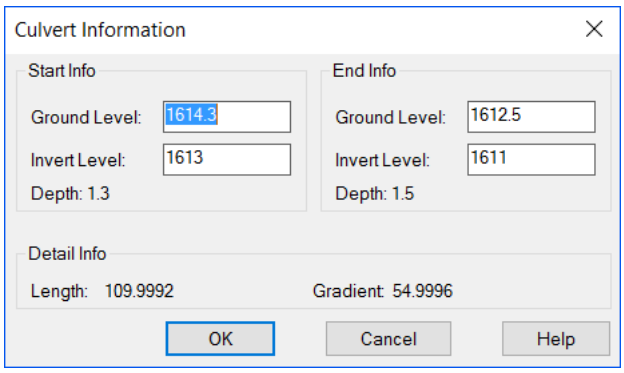


Figure 7 The Culvert Information dialog box

3 From the **Start Info** group, change the following:

- Ground level
- Invert level

The Depth value is updated automatically.

4 From the **End Info** group, change the following:

- Ground level
- Invert level

The Depth value is updated automatically.

The culvert length and gradient values in the Detail Info group are calculated automatically from the Start Info and End Info group.

5 Click **OK** to accept the changes you have made.

The attribute changes are accepted.

To change channel levels

1 Choose **Change** > **Channel Levels...**

2 Select the appropriate channel.

The Channel Information dialog box is displayed.

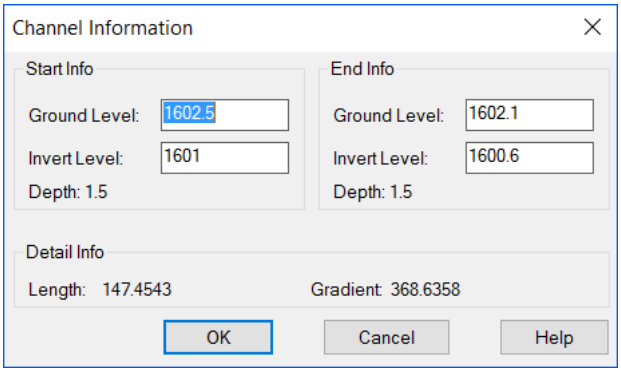


Figure 8 The Channel Information dialog box

3 From the **Start Info** group, change the following:

- Ground level
- Invert level

The Depth value is updated automatically.

4 From the **End Info** group, change the following:

- Ground level
- Invert level

The Depth value is updated automatically.

The channel length and gradient values in the Detail Info group are calculated automatically from the Start Info and End Info group.

5 Click **OK** to accept the changes you have made.

The attribute changes are accepted.

To change node levels

1 Choose **Change** > **Node Levels...**

2 Select the appropriate stormwater node.

The Stormwater Node Information dialog box is displayed.

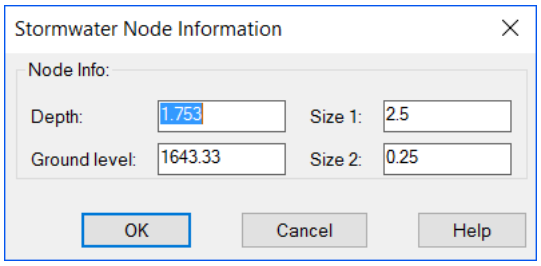


Figure 9 The Stormwater Node Information dialog box

3 Enter a new value for one or all of the following:

- Depth
- Ground Level
- Size 1 – length
- Size 2 – width

4 Click **OK** to apply the changes.

The attribute changes are accepted.

Updating Levels

There is no direct corrulation between the drainage node's cover level and depth, and the storm pipes' ground level , invert level and gradient. The association between the nodes and pipes is the connectivity, which is defined in the database design.

So when one of the values is changed for either the storm water node or pipe, channel or cluvert, the associated information is not automatically updated to the connected object, for example:

The node depth value stored with the storm water nodes is not automatically updated when the cover or ground level or invert levels for the attached pipes, channels or culverst are updated.

or

The start and end ground levels are not updated when the connected node's ground level is updated.

For this reason, menu items to update the associated nodes, pipes, channels, culverts and gradients, based on their connectivity, can be updated or recalculated using the following menu items:

- Change > Update Levels > Node to Pipe - Updates the connecting pipes' start/end cover elevations and the start invert elevation using the connecting nodes' cover/ground level and depth.
- Change > Update Levels > Pipe to Node - Updates the connecting nodes' cover elevation and depth using the start/end cover elevations and start/end invert elevations from the connecting pipes.
- Change > Update Levels > Recalculate Gradients - recalculates the gradient for selected pipes, channels and culverts.

Typical Drainage As Constructed detail plan

In the typical As Constructed Design plan example below there are drainage pipes running downstream between manholes MH106, MH107 and MH108.

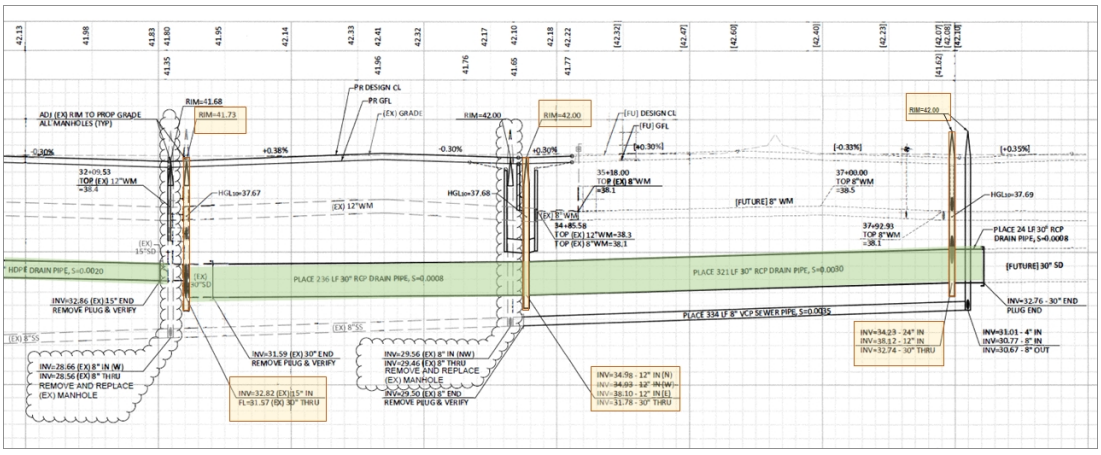


Figure 10 Typical As Constructed Design Plan

To update pipes, culverts or channels levels with connecting nodes

In the example of updating Node to Pipe, Manholes MH106, MH107 and MH108 have the following Node Depth and Ground Level values captured in the SP_SWNODE table:

Attribute Column Name	MH106	MH107	MH108
NODE_REF	MH106	MH107	MH108
NODE_DEPTH	11.26	10.22	10.11
NODE_GL	44.00	42.00	41.68

Figure 11 SP_SWNODE captured values

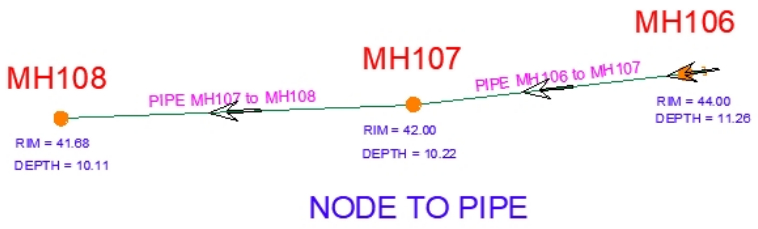


Figure 12 Drainage nodes Cover Elevations and Depths simplified

To execute the Node to Pipe function, do the following:

1 Choose **Change** > **Update Levels > Node to Pipe**

2 Select the appropriate stormwater node(s) and then press **Enter**.

Figure 13 The storm water nodes are selected

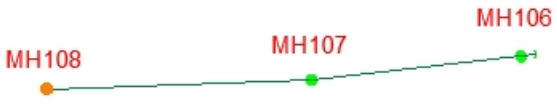


Figure 14 The storm water pipe attributes are updated

The command line indicates the number of pipe(s), channel(s) and culverts updated.

The storm water pipe, channel and/or culvert color is changed to the Munsys Integrity color, indicating that the Object Integrity check is required before posting the updates to the database.

After completing the Node to Pipe update function, the following values are calculated for the Drainage Pipes in the SP_SWPIPE table:

Attribute Column Name	Pipe from MH106 to MH107	Pipe from MH107 to MH108
PIPE_GRADIENT - calculated	61.704	362.848
START_GL - calculated	44.00	42.00
END_GL - calculated	42.00	41.680
START_IL - calcualted	32.74	31.780
END_IL	[not calculated]	[not calculated]

Table 15 SP_SWPIPE calcualted values

The downstream end invert level values of both pipes were intentionally not calcualted since they cannot be assumed from the downstream node depth, especially in the case of drop manholes.

To update node levels with connecting pipes, culverts or channels

In the example of updating Pipe to Node, Pipe MH106 to MH107 and Pipe MH107 to MH108 have the following Ground Level and Invert Level values captured in the SP_SWPIPE table:

Attribute Column Name	Pipe from MH106 to MH107	Pipe from MH107 to MH108
START_GL	44.00	42.00
END_GL	42.00	41.680
START_IL	32.740	31.780
END_IL	31.780	31.570

SP_SWPIPE captured Ground and Invert Levels

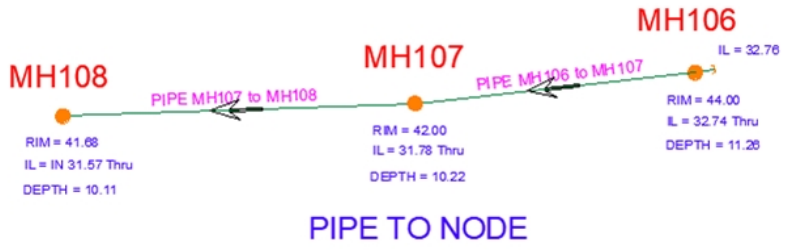


Figure 16 Drainage Pipe Ground and Invert Levels simplified

To execute the Pipe to Node function, do the following:

1 Choose **Change** > **Update Levels > Pipe to Node**

2 Select the appropriate stormwater pipes(s), channel(s) and/or culvert(s) and then press **Enter**.

Figure 17 The storm water pipes, channels and/or culverts are selected

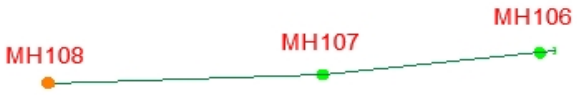


Figure 18 The storm water node attributes are updated

The command line indicates the number of nodes updated.

The stormwater nodes' color is changed to the Munsys Integrity color, indicating that the Object Integrity check is required before posting the updates to the database.

After completing the Pipe to Node update function, the following values are calculated for the Drainage Nodes in the SP_SWNODE table:

Attribute Column Name	MH106	MH107	MH108
NODE_DEPTH - calculated	11.26	10.22	[not calculated]
NODE_GL - calculated	44.00	42.00	[not calculated]

Table 19 SP_SEWGPIPE calcualted values

The depth of the node is calcualted form the start Invert Level of the downstream pipe and not the end Invert Level of the upstream pipe.

To recalculate pipe, culvert or channel gradients

1 Choose **Change** > **Update Levels > Recalculate Gradients**

2 Select the appropriate storm water pipe(s), channel(s) and/or culvert(s) and then press **Enter**.

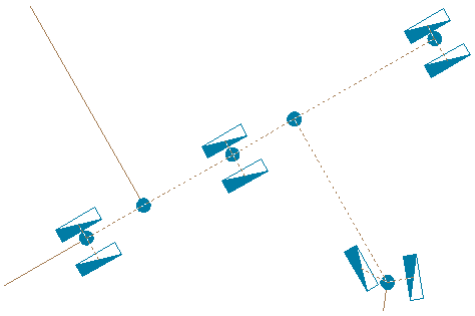


Figure 20 The storm water pipes, channels and/or culverts are selected

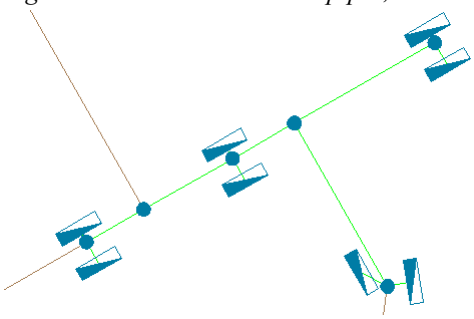


Figure 21 The storm water gradients are recalculated

The command line indicates the number of pipe(s), channel(s) and/or culvert(s) updated.

The storm water pipe, channel and/or culvert's color is changed to the Munsys Integrity color, indicating that the Object Integrity check is required before posting the updates to the database.

Changing stormwater pipe attributes

You can change the default pipe diameter, material or type that was assigned to a stormwater pipe during the capture process. The new value is selected from a list to ensure data accuracy.

Tip If you construct the stormwater pipes with the default diameter, type or material set to the one used most commonly, only pipes that have different attributes need to be changed, reducing the amount of data to be captured.

To change stormwater pipe diameter

1 Choose **Change** > **Change Pipe Attributes** > **Diameter...**

2 Select the appropriate stormwater pipe(s), and then press **ENTER**.

The Select Stormwater Pipe Diameter dialog box is displayed, highlighting the diameter of the pipe that you selected. If you selected multiple pipes, no current diameter is highlighted.

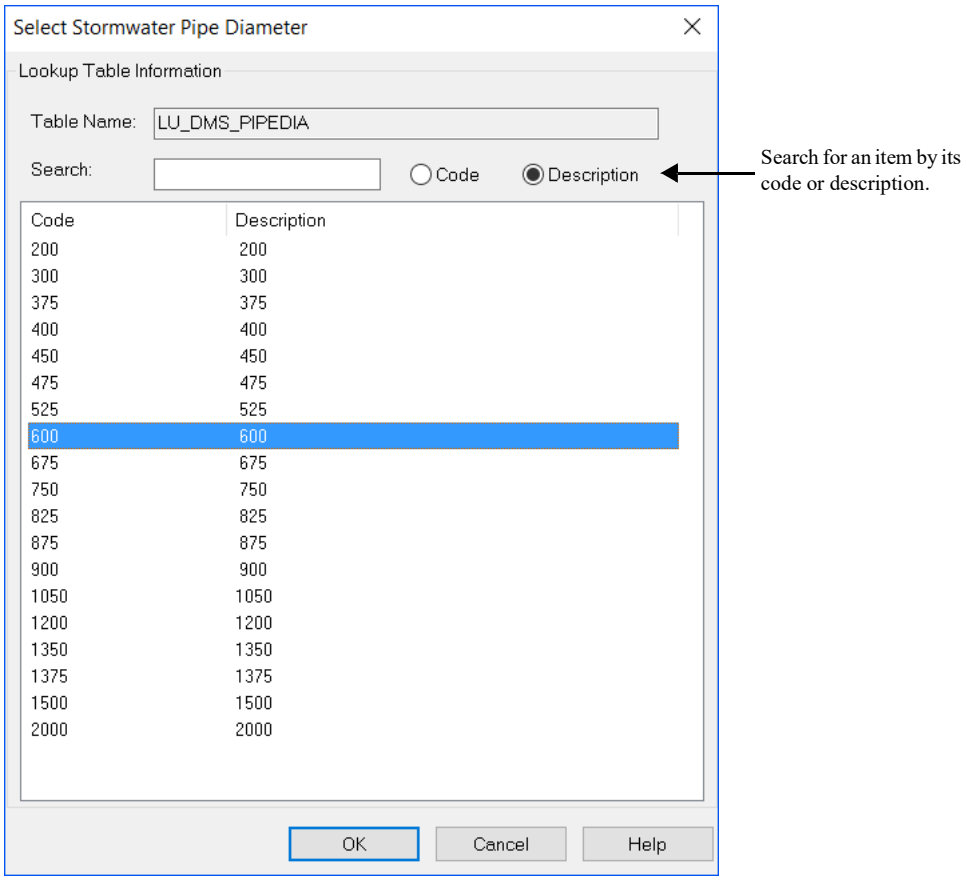


Figure 22 The Select Stormwater Pipe Diameter dialog box

3 Select the appropriate diameter, and then click **OK**.

To change stormwater pipe material

1 Choose **Change** > **Change Pipe Attributes** > **Material...**

2 Select the appropriate stormwater pipe(s), and then press **ENTER**.

The Select Stormwater Pipe Material dialog box is displayed, highlighting the material of the pipe that you selected. If you selected multiple pipes, no current diameter is highlighted.

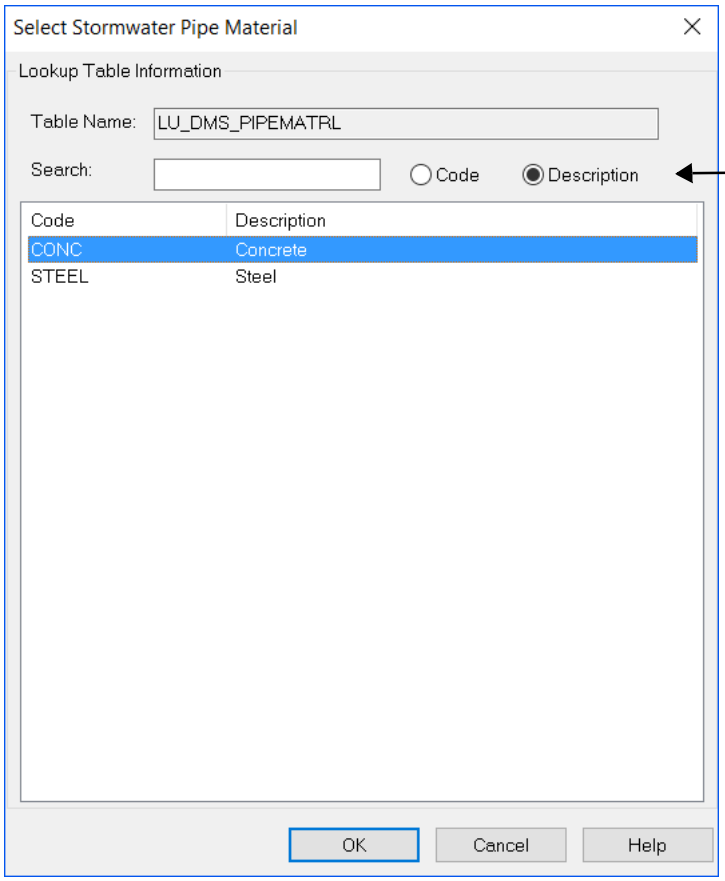


Figure 23 The Select Stormwater Pipe Material dialog box

3 Select the appropriate pipe material from the list, and then click **OK**.

To change a stormwater pipe type

1 Choose **Change** > **Change Pipe Attributes** > **Type...**

2 Select the appropriate stormwater pipe(s), and press **ENTER**.

The Select Stormwater Pipe Type dialog box is displayed, highlighting the type of the pipe that you selected. If you selected multiple pipes, no current type is highlighted.

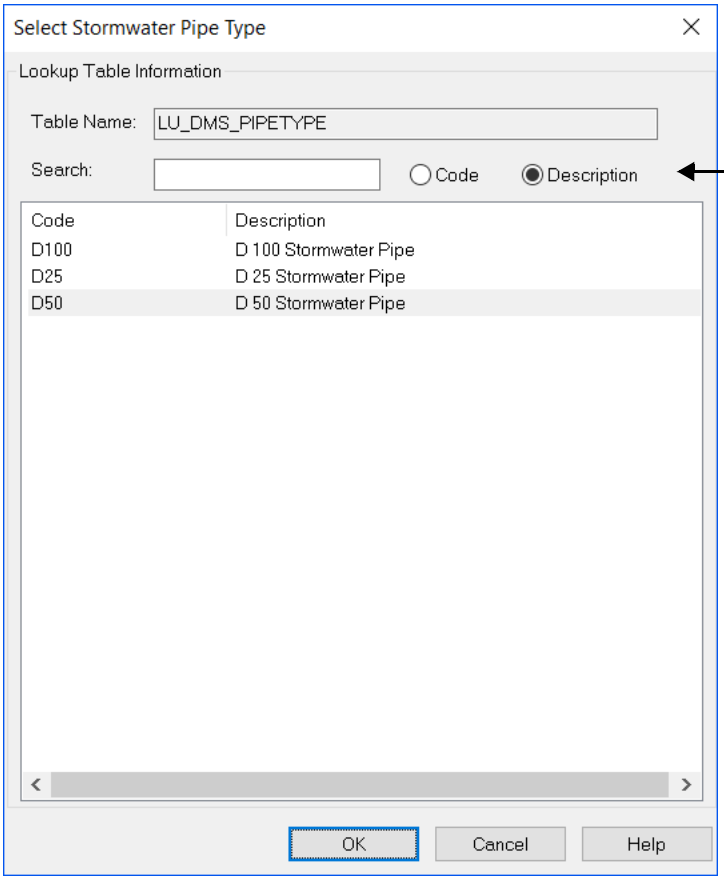


Figure 24 The Select Stormwater Pipe Type dialog box

3 Select the appropriate pipe type, and then click **OK**.

Changing culvert attributes

You may change the default width, depth, material or type that was assigned to a culvert during the capture process. The new value is selected from a list to ensure data accuracy.

Tip If you construct the culverts in the stormwater network with the default width, depth, material or type set to the one that is used most commonly, only culverts that have different attributes need to be changed, reducing the amount of data to be captured.

To change culvert width

1 Choose **Change** > **Change Culvert Attributes** > **Width...**

2 Select the appropriate culvert/s, and then press **ENTER**.

The Select Culvert Width dialog box is displayed, highlighting the width of the culvert that you selected. If you selected more than one culvert, no current width is highlighted.

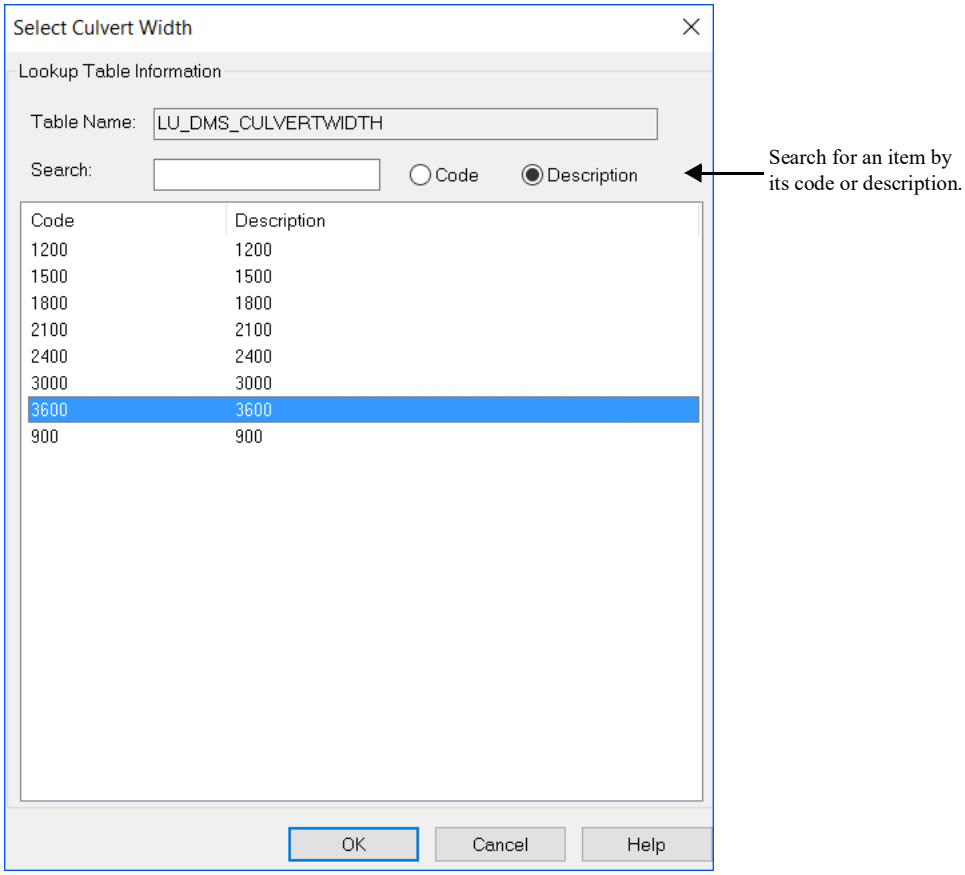


Figure 25 The Select Culvert Width dialog box

3 From the **Select Culvert Width** dialog box, select the appropriate width, and then click **OK**.

The attribute changes are accepted.

To change culvert depth

1 Choose **Change** > **Change Culvert Attributes** > **Depth...**

2 Select the appropriate culvert/s, and then press **ENTER**.

The Select Culvert Depth dialog box is displayed, highlighting the depth of the culvert that you selected. If you selected more than one culvert, no current depth is highlighted.

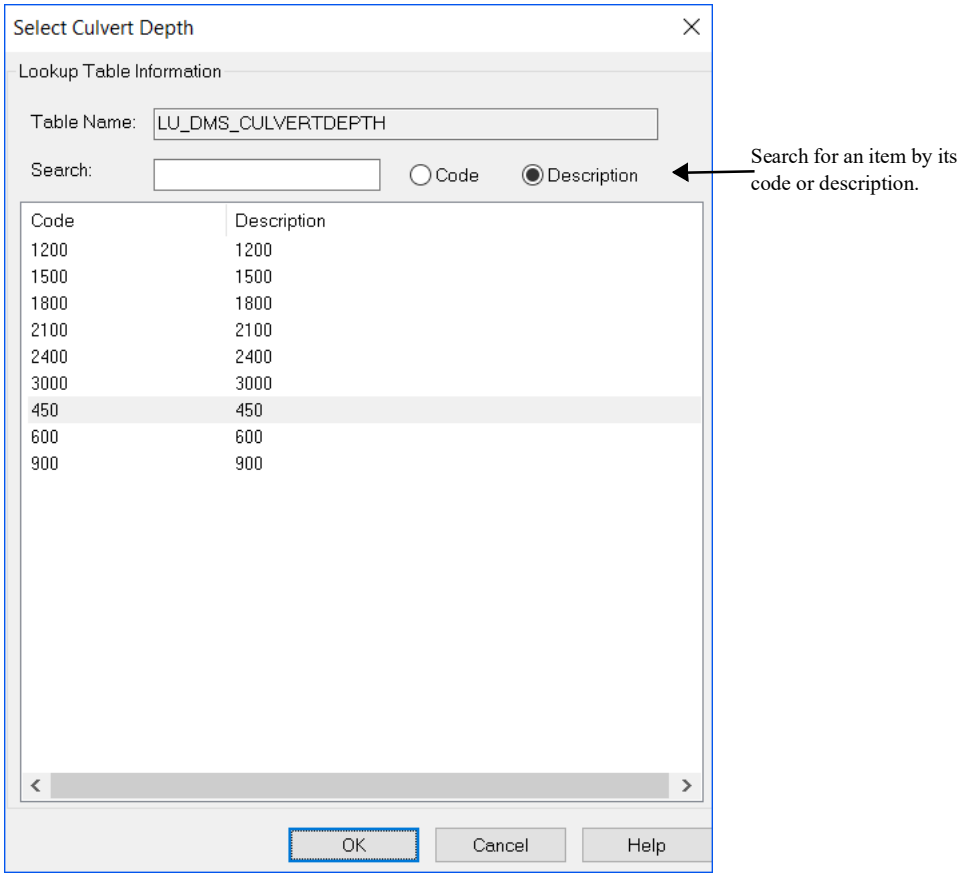


Figure 26 The Select Culvert Depth dialog box

3 From the **Select Culvert Depth** dialog box, select the appropriate depth, and then click **OK**.

The attribute changes are accepted.

To change culvert material

1 Choose **Change** > **Change Culvert Attributes** > **Material...**

2 Select the appropriate culvert/s, and then press **ENTER**.

The Select Culvert Material dialog box is displayed, highlighting the material of the culvert that you selected. If you selected more than one culvert, no current material is highlighted.

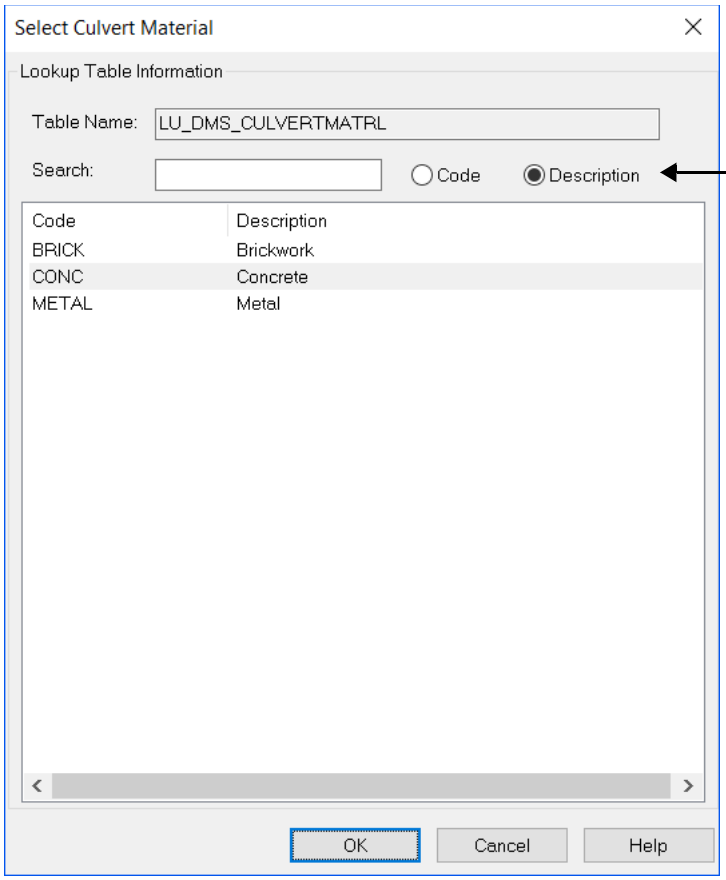


Figure 27 The Select Culvert Material dialog box

3 From the **Select Culvert Material** dialog box, select the appropriate material, and then click **OK**.

The attribute changes are accepted.

To change a culvert type

1 Choose **Change** > **Change Culvert Attributes** > **Type...**

2 Select the appropriate culvert/s, and then press **ENTER**.

The Select Culvert Type dialog box is displayed, highlighting the type of the culvert that you selected. If you selected more than one culvert, no current type is highlighted.

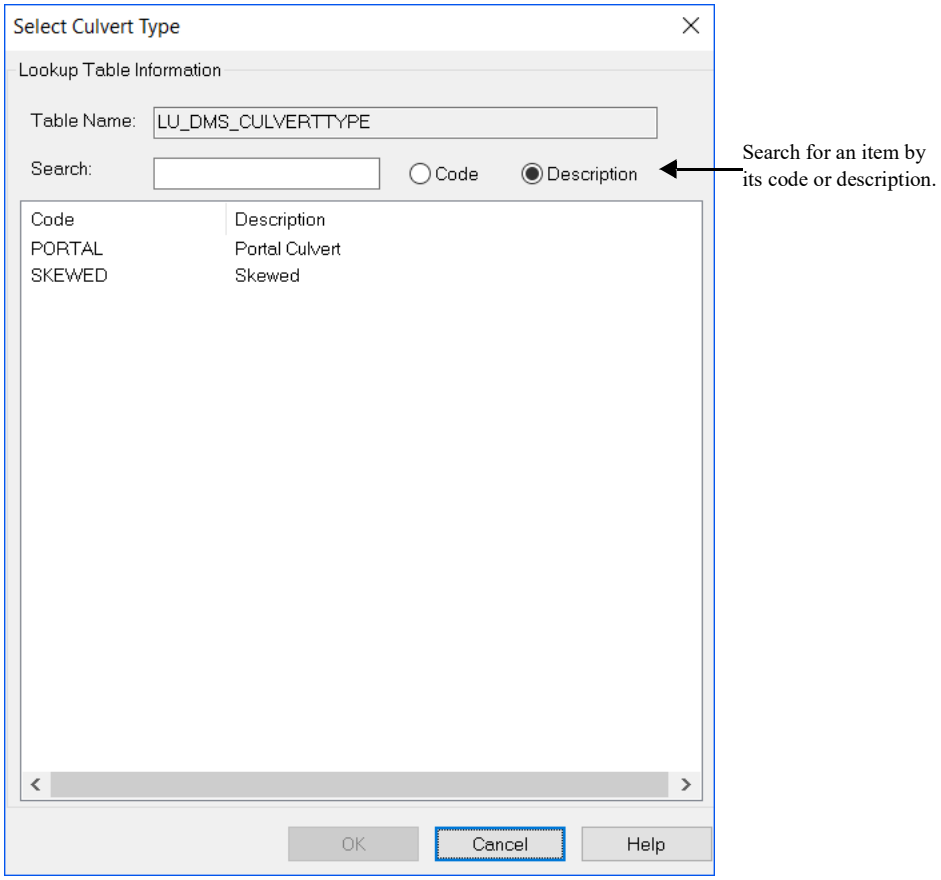


Figure 28 The Select Culvert Type dialog box

3 From the **Select Culvert Type** dialog box, select the appropriate type, and then click **OK**.

The attribute changes are accepted.

Changing channel attributes

You may change the following attributes that were assigned to a channel during the capture process:

- top width
- bottom width
- depth
- material
- type

Tip If you construct the channels in the stormwater network with the default attributes set to the ones that are used most commonly, only channels that have different attributes need to be changed, reducing the amount of data to be captured.

To change the top width of a channel

1 Choose **Change** > **Change Channel Attributes** > **Top Width...**

2 Select the appropriate channel/s, and then press **ENTER**.

The Channel Top Width text box is displayed, showing the top width of the channel that you selected. If you selected more than one channel, the text box will be empty.

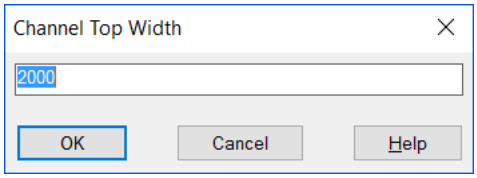


Figure 29 The Channel Top Width text box

3 In the text box, type the new top width, and then click **OK**.

The attribute change is accepted.

To change the bottom width of a channel

1 Choose **Change** > **Change Channel Attributes** > **Bottom Width...**

2 Select the appropriate channel/s, and then press **ENTER**.

The Channel Bottom Width text box is displayed, highlighting the bottom width of the channel that you selected. If you selected more than one channel, the text box will be empty.

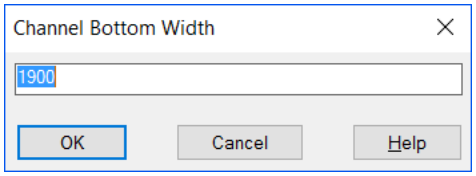


Figure 30 The Channel Bottom Width text box

3 In the text box, type the new bottom width, and then click **OK**.

The attribute changes are accepted.

Changing channel depth

1 Choose **Change** > **Change Channel Attributes** > **Depth...**

2 Select the appropriate channel/s, and then press **ENTER**.

The Select Channel Depth text box is displayed, highlighting the depth of the channel that you selected. If you selected more than one channel, the text box will be empty.

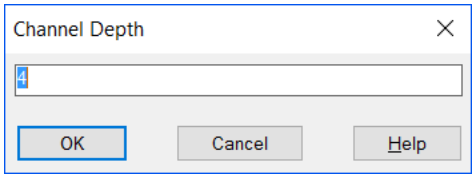


Figure 31 The Select Channel Depth text box

3 In the text box, type the new depth, and then click **OK**.

The attribute changes are accepted.

To change channel material

1 Choose **Change** > **Change Channel Attributes** > **Material...**

2 Select the appropriate channel/s, and then press **ENTER**.

The Select Channel Material dialog box is displayed, highlighting the material of the channel that you selected. If you selected more than one channel, no current material is highlighted.

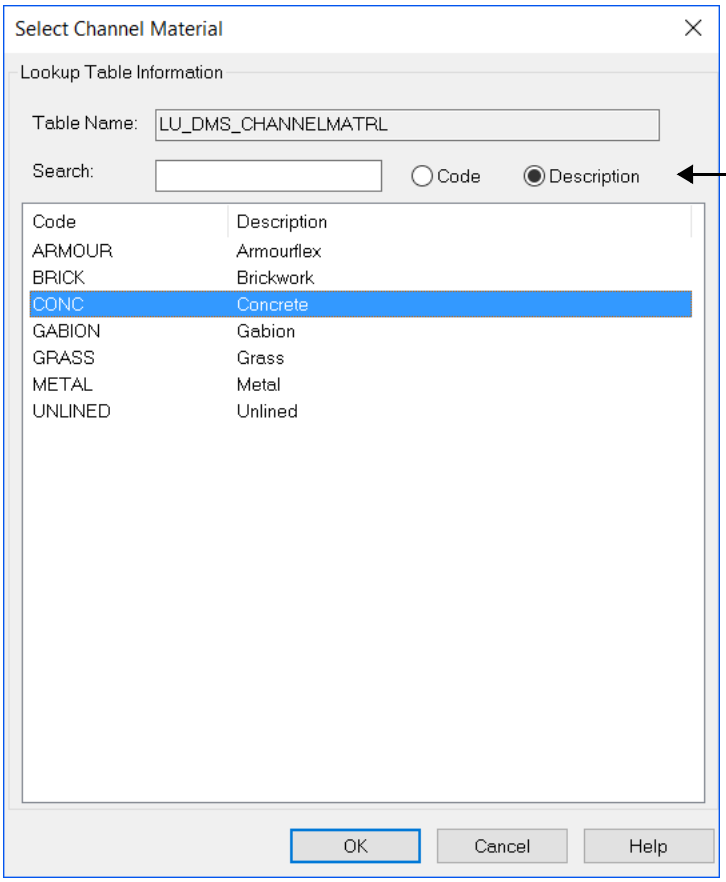


Figure 32 The Select Channel Material dialog box

3 From the **Select Channel Material** dialog box, select the appropriate material, and then click **OK**.

The attribute changes are accepted.

To change a channel type

1 Choose **Change** > **Change Channel Attributes** > **Type...**

2 Select the appropriate channel/s, and then press **ENTER**.

The Select Channel Type dialog box is displayed, highlighting the type of the channel that you selected. If you selected more than one channel, no current type is highlighted.

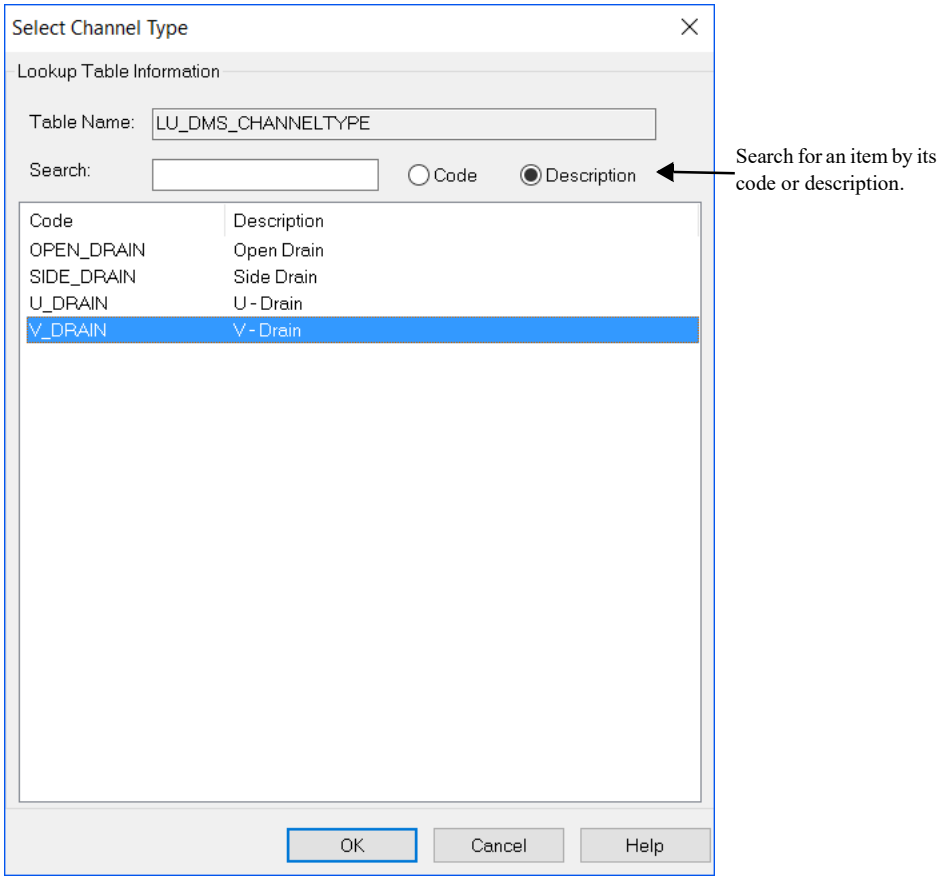


Figure 33 The Select Channel Type dialog box

3 From the **Select Channel Type** dialog box, select the appropriate type, and then click **OK**.

The attribute changes are accepted.

Changing river attributes

You may change the default type that was assigned to a river during the capture process. The new river type is selected from a list to ensure data accuracy.

Tip If you capture all the rivers with the default type set to the one that is used most commonly, only rivers that have a different type need to be changed, reducing the amount of data to be captured.

To change a river type

1 Choose **Change** > **Change River Attributes** > **Type...**

2 Select the appropriate river/s, and then press **ENTER**.

The Select River Type dialog box is displayed, highlighting the type of the river that you selected. If you selected more than one river, no current type is highlighted.

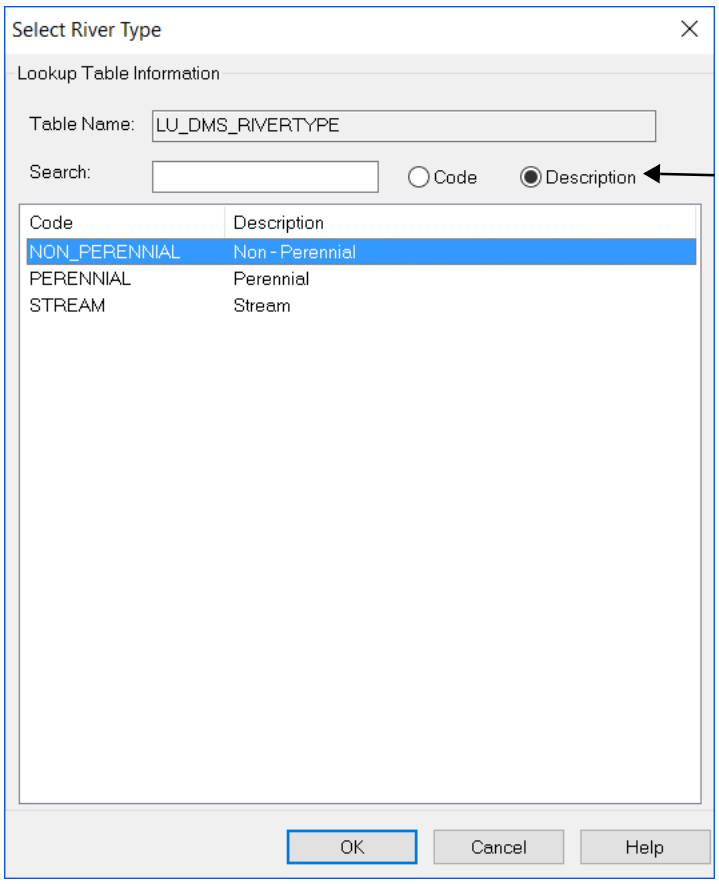


Figure 34 The Select River Type dialog box

3 From the **Select River Type** dialog box, select the appropriate river type, and then click **OK**.

The attribute changes are accepted.

Changing floodline attributes

You may change the default type that was assigned to a floodline during the capture process. The new floodline type is selected from a list to ensure data accuracy.

Tip If you capture all the floodlines with the default type set to the one that is used most commonly, only floodlines that have a different type need to be changed, reducing the amount of data to be captured.

To change a floodline type

1 Choose **Change** > **Change Floodline Attributes** > **Floodline Type...**

2 Select the appropriate floodline/s, and then press **ENTER**.

The Select Floodline Type dialog box is displayed, highlighting the type of the floodline that you selected. If you selected more than one floodline, no current type is highlighted.

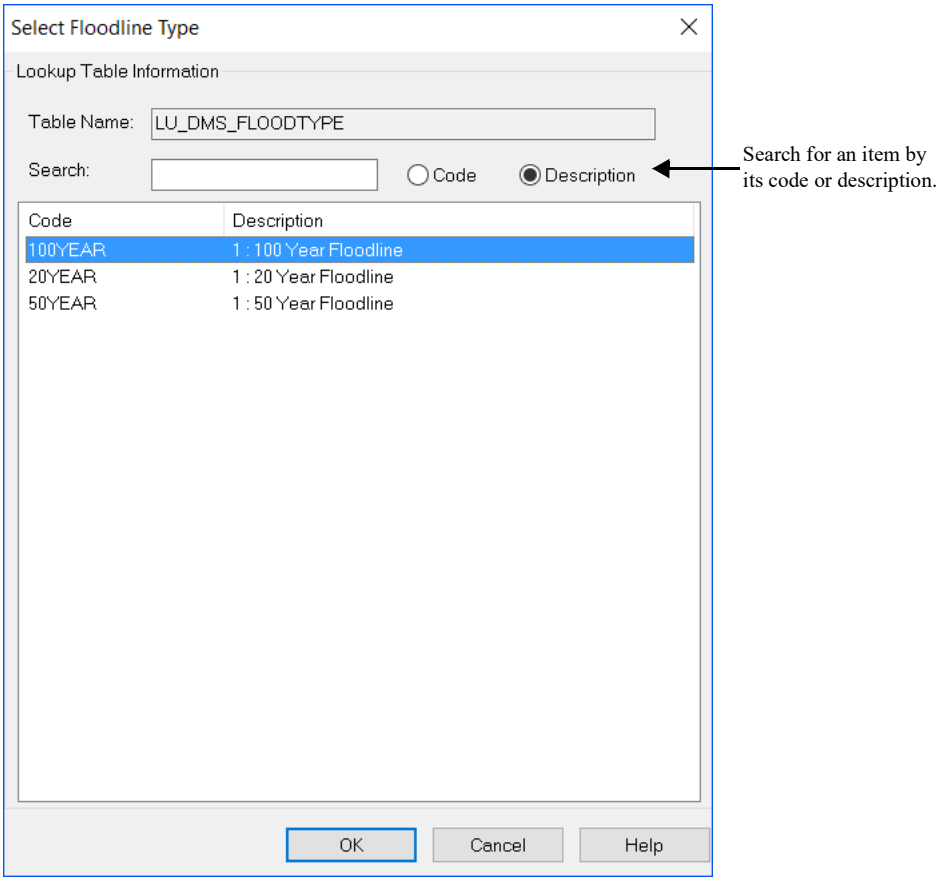


Figure 35 The Select Floodline Type dialog box

3 From the **Select Floodline Type** dialog box, select the appropriate floodline type, and then click **OK**.

The attribute changes are accepted.

Changing dam attributes

You may change the default type that was assigned to a dam during the capture process. The new dam type is selected from a list to ensure data accuracy.

Tip If you capture all the dams with the default type set to the one that is used most commonly, only dams that have a different type need to be changed, reducing the amount of data to be captured.

To change a dam type

1 Choose **Change** > **Change Dam Attributes** > **Dam Type...**

2 Select the appropriate dam/s, and then press **ENTER**.

The Select Dam Type dialog box is displayed, highlighting the type of the dam that you selected. If you selected more than one dam, no current type is highlighted.

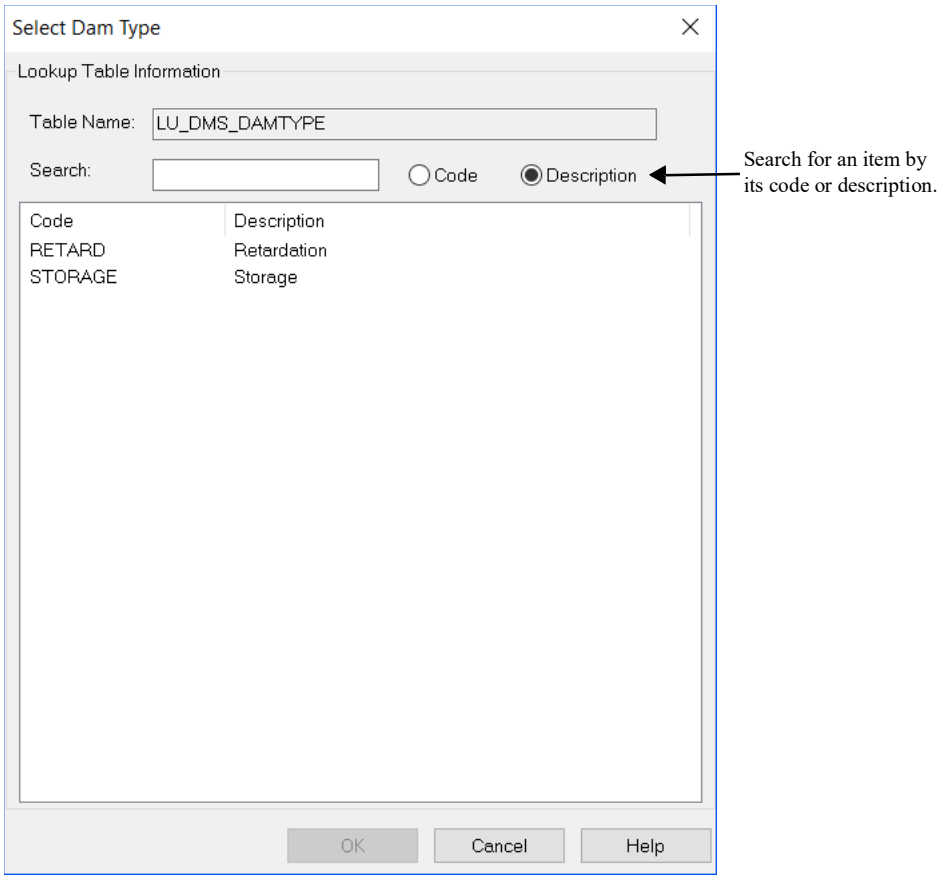


Figure 36 The Select Dam Type dialog box

3 From the **Select Dam Type** dialog box, select the appropriate dam type, and then click **OK**.

The attribute changes are accepted.

Changing a service connection type

With this function, you can change a service connection type from the default that was entered when the service connection was captured. The new service connection type is selected from a list to ensure data accuracy.

To change a service connection type

1 Choose **Change** > **Change SC Type...**

2 Select the appropriate service connection(s), and then press **ENTER**.

The Select Service Connection Type dialog box is displayed, showing the type of the service connection that you selected. If you selected multiple service connections, no current type is highlighted.

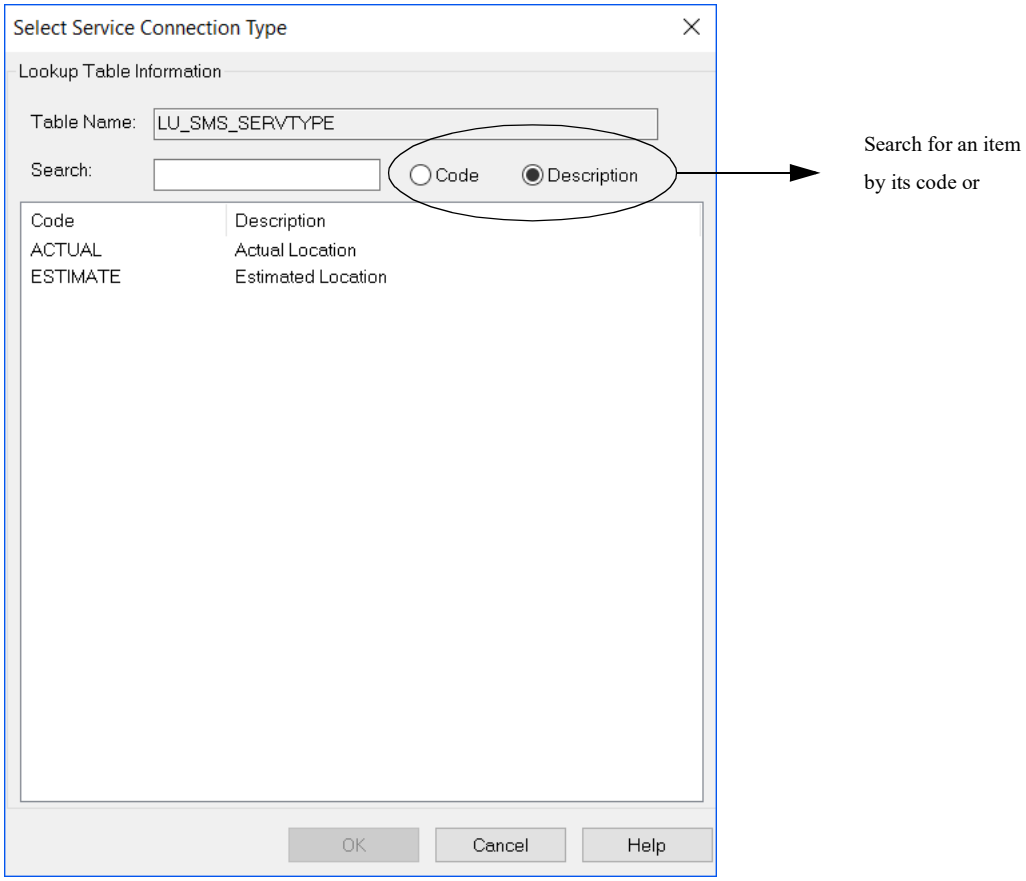


Figure 37 The Select Service Connection Type dialog box

3 Select the appropriate service connection type from the list, and then click **OK**.

Linking service connections to parcels / buildings

This function is used to link selected service connections to their respective parcels or buildings.

The parcels and buildings have to be queried from the database.

To link service connections to parcels

1 Choose **Change** > **Link Service connections**.

2 Select the appropriate service connections, and then press **ENTER**.

The selected service connections are linked to their respective parcels.

Note If the endpoint of the service connection falls inside a parcel boundary the PRCL_GID column is updated with the GID value from SP_PARCEL.

Note If the endpoint of the service connection falls inside a parcel boundary and a building polygon the PRCL_GID column is updated with the GID value from SP_PARCEL, and the LINK_ID column is updated with the GID value from SP_BUILDING

Changing drainage notes

With this function, you can change the text of an existing drainage note.

To change a drainage note

1 Do one of the following:

- Choose **Change** > **Change Drainage Note...**
- Click the **Change Drainage Note** button on the Munsys Drainage **Change** toolbar.

2 Select the drainage note that you want to change.

The Drainage Note text box is displayed, showing the current note.

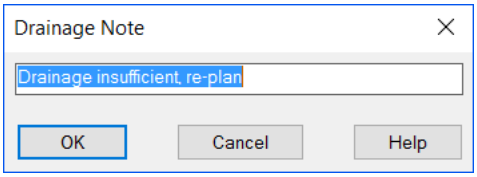


Figure 38 The Drainage Note text box

3 Change the note as required, and then click **OK**.

Adding comments

This function is used to assign descriptive comments to one or more selected drainage objects.

To add comments

1 Choose **Change** > **Add Comment...**

2 Select the drainage object(s) that you want to add a comment to.

The Drainage Comment text box is displayed.

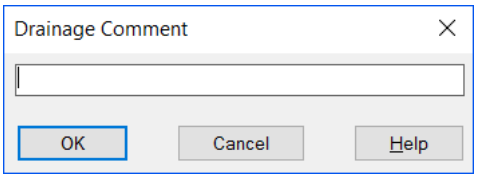


Figure 39 The Drainage Comment text box

3 In the text box, add the comment, and then click **OK**.

The comment is assigned to the selected object(s).

Deleting and undeleting drainage objects

With Munsys Drainage, existing drainage objects can be deleted from the database. Multiple objects can be selected for deleting. Drainage objects that form part of a redundant network have to be deleted. Deleted drainage objects are moved to a frozen layer DELETED, which is removed from the database when changes are posted. With the Undelete function, you can restore drainage objects that have been moved to the DELETED layer and that are marked for deletion.

To delete drainage objects

1 Choose **Change** > **Delete Drainage Object**.

2 Select the drainage objects that you want to delete.

The command line indicates how many drainage objects you have selected, and how many have been filtered out.

3 Press **ENTER** to delete the selected drainage objects.

The drainage objects are flagged for deletion and moved to the DELETED layer. These changes are applied to the database when changes are posted.

To undelete drainage objects

1 Choose **Change** > **Undelete Drainage Objects**.

The drainage objects that you deleted are displayed in the drawing.

2 Select the drainage objects that you want to restore.

The Delete flag is no longer set for the selected objects. In the event of spatial or attribute changes before the objects were flagged for deletion, these changes are applied.

Index

E

F

I

J

L

M

N