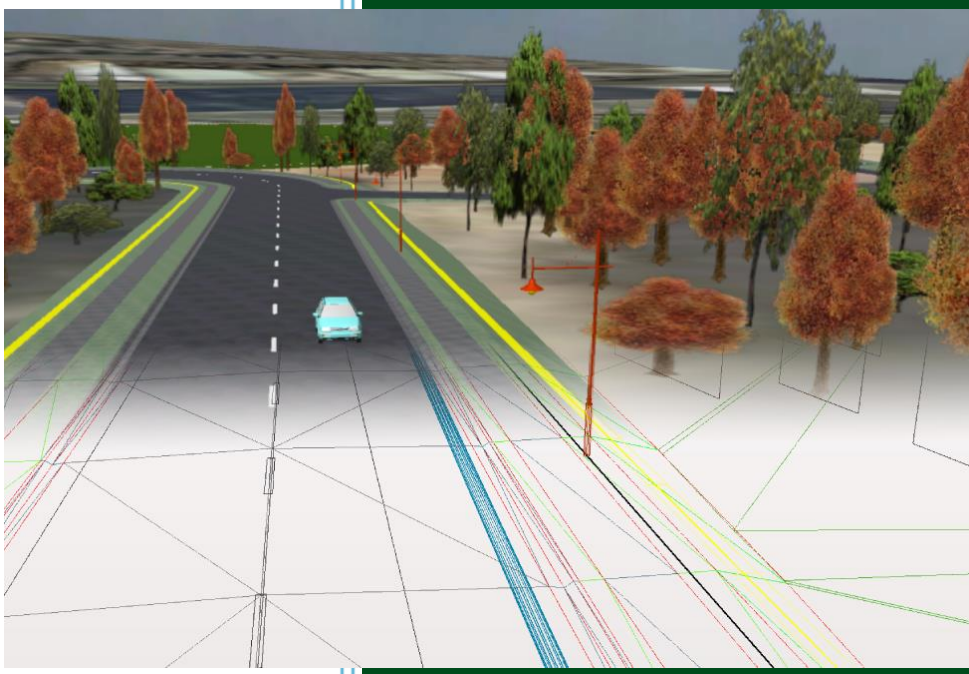
Civil Site Design for BricsCAD®



Quick Start Tutorial

civil survey solutions

civil survey solutions

Authorised Developer
Authorised Training Centre

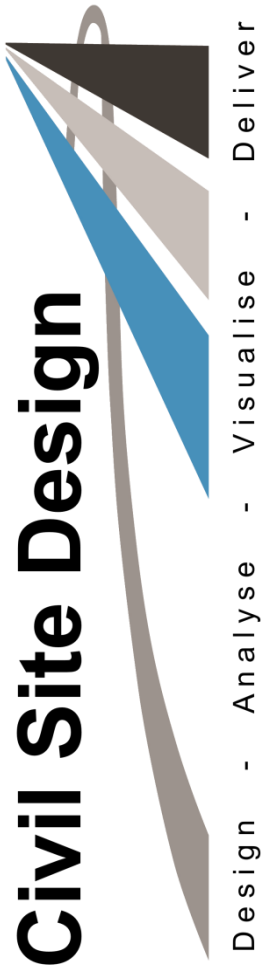
Civil Site Design

Civil Design Inside AutoCAD

Civil Site Design provides a template and string based design environment for creating Roads and land re-grading inside AutoCAD, as well as pipe design including stormwater drainage and sewer.

Here's how Civil Site Design can benefit you:

- ✓ Integrates design and drafting on your AutoCAD based platform
- ✓ One change – multiple updates. Modern software with dynamic relationships between objects. Automate revision control by keeping your objects co-ordinated
- ✓ All-in-One Program – Get Surface, Alignment, Road and Pipe tools in the one program
- ✓ Easy to learn – leverage your AutoCAD skills
- ✓ Low cost – resides on top of your AutoCAD
- ✓ Intuitive commands follow workflow process
- ✓ You never leave the AutoCAD environment using Civil Site Design.
- ✓ Design aids include road, drainage, sewer and grading design.
- ✓ Works inside Autodesk's AutoCAD, MAP or Civil 3D.



Quick Start Tutorials

We welcome you to try it out! See below for some quick start tutorials that will showcase the fundamentals of Civil Site Design for delivering a total design solution for all your surface, road and pipe design needs.

Table of Contents

Developed by:

civil survey solutions

Sales: sales@civilsurveysolutions.com.au
Ph: 1300 254 004 (Australia)

Quick Start Data

Before you Start – Please start BricsCAD and open the **Quick Start Tutorial.dwg** training file.

*Note: The training files can be found by running the **General ➢ Open Tutorial Folder** **Command**.*

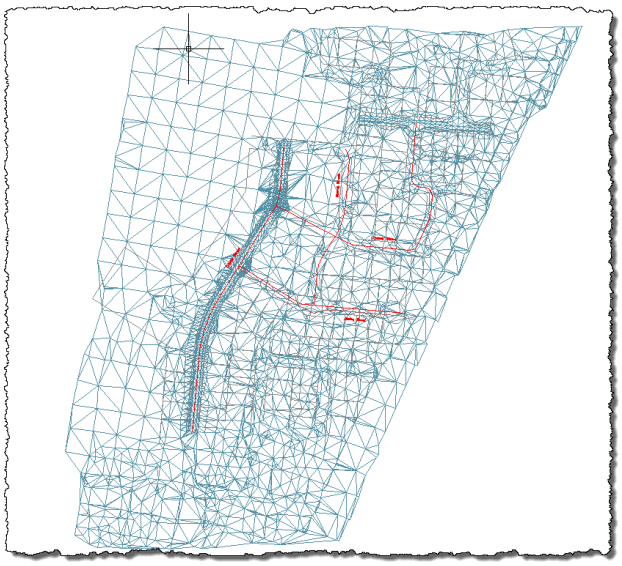
Surfaces

You can create Surfaces (also known at DTMs and TINs) from multiple inputs such as 3D faces, polylines, BricsCAD points, point files and more. Surface display is easily managed, directly in the drawing, with user control over contours, triangles and shading display.

Let's have a look at creating a surface from 3D faces.

Creating a Surface

Let's do a quick review of the current drawing:



On the right you can see that there are triangles (3D faces) in the drawing – these represent the surveyor's information. The first step is to convert these into a TIN Surface. The 3D faces are located on the layer SURF

Also included are layers for other objects as follows:

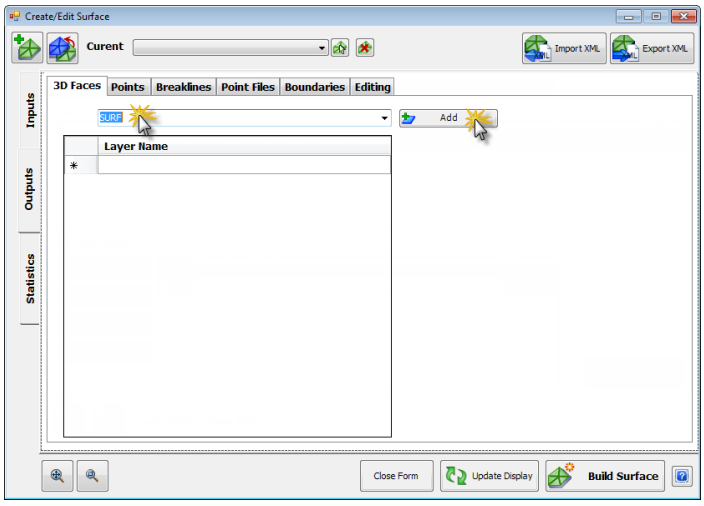
- Alignments – ALIGN
- Drainage – PIPES-DRAINAGE
- Sewer – PIPES-SEWER

Let's get started and make a surface from the 3D faces

Step 1: From the menu, click on ***Surfaces*** ➢ ***Create Surface.***
Step 2: At the dialogue box to **Select Required Drainage Settings** file, just click **OK**.
Step 3: At the prompt for a surface name, type **NS** and click **OK**. The Create/Edit Surface form will display:

This form consists of three tabs:

- ***Inputs***: use the tabs to select input data for the surface
- ***Outputs***: manages the display of contours, contour labels, slope shading, direction and height shading
- ***Statistics***: lists the statistics of the surface – you can copy this information across to use as inputs elsewhere

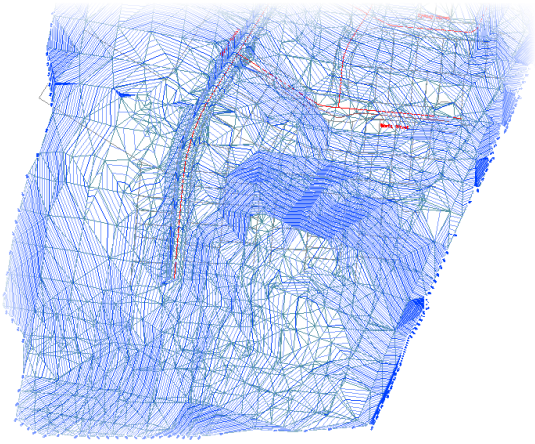


Note: This form is 'modeless' – you can position it where you want, leave it open and still work in BricsCAD.

Step 4: From the **3D Faces** tab, click on the pick box and select the layer **SURF** from the list (the software filters to display layers containing 3D faces)

Step 5: Click on the **Add** button to add this layer of data to the inputs list.

Step 6: Click on **Build Surface** – the software will immediately build a surface from the inputs and display in the drawing.



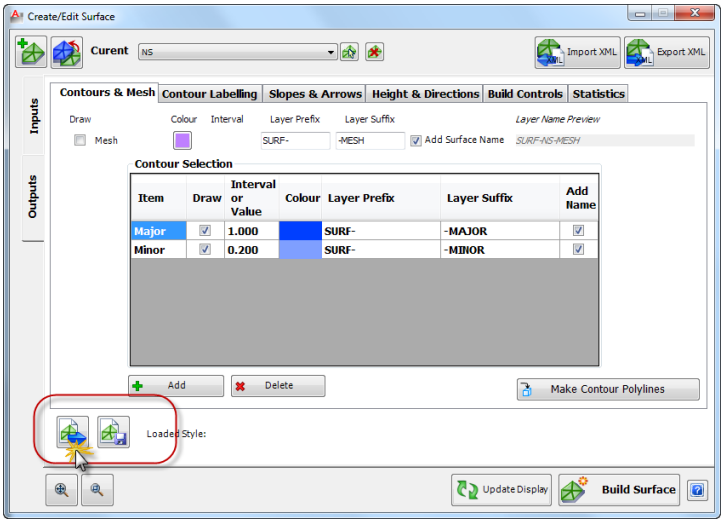
Note: You can turn off the triangles (3D faces) now, for better clarity in the drawing.

The surface is a single object in the drawing – there are export tools to generate base BricsCAD entities of the triangles and contours, for final presentation.

Step 7: Click on the **Inputs > Boundaries** tab. From here you can apply any closed 2D polyline as an outer boundary for the surface, as well as hiding internal triangles. Do not add a boundary in this case.

Surface Display Controls

Click on the **Outputs** tab – from here you can manage how the surface displays. You control what displays as well as what layers are used – ARD will create the layers as required and you can manage the available layers from the **ARD Layer List** found under settings in the alignments tab.



All of the display controls can be saved to the ARD Settings folder and then be recalled on any project. Let's start by changing the display using a saved Style:

Step 8: Click on **Load Style**.

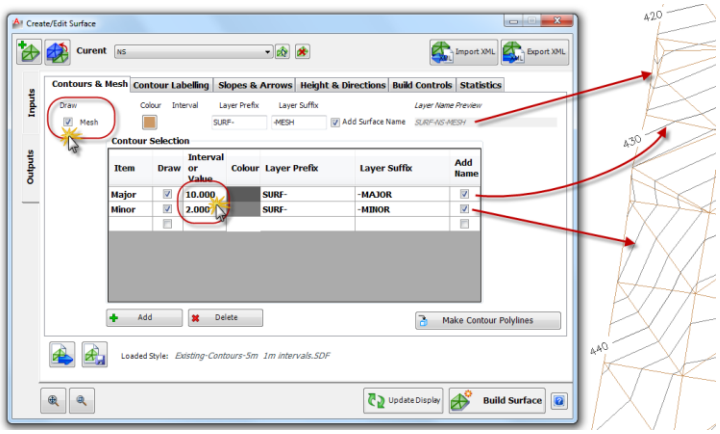
Step 9: In the list of available styles, select **Existing-Contours-5m & 1m intervals.SDF** and click **OK**

Surface Contour Display

The surface display immediately changes in the drawing. Let's have a quick tour of the user controls

Step 10: The **Contours & Mesh** tab allows you to set the layer and mesh (triangle) display. Make the following changes:

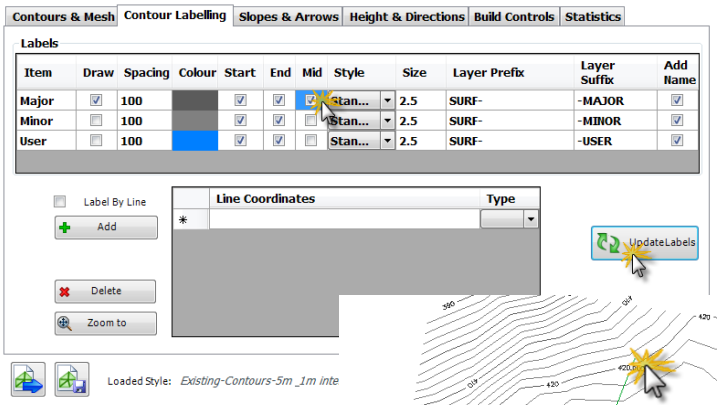
a. Tick on the Mesh
b. Change the Major Contours to 10
c. Change the Minor Contours to 2
d. Change the colours as desired



As changes are made, the surface display automatically updates. Click on **Update Display** if desired.

Contour Labels

Contour labels can be applied at the edges of each contour, at regular intervals internal to the contour line, and along user selected lines. You get to control colour, layer, text style and text size.



Step 11: Click on the **Outputs > Contour Labelling** tab

Step 12: For the Major contour labels, tick ON the **Mid** labels and then click **Update Labels**. Note extra labelling at regular intervals along the major contours – the **Spacing** parameter controls the distance between the contour labels.

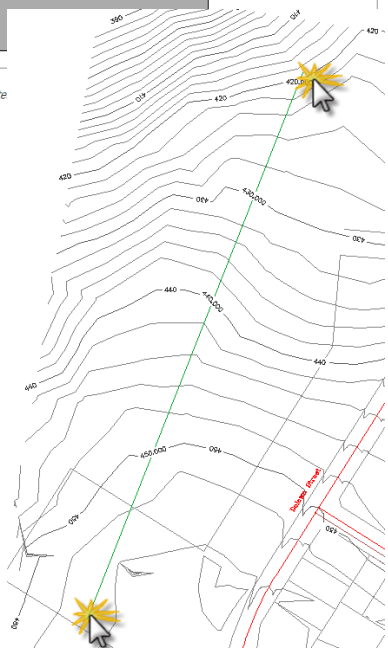
To add your own contour labels where you want:

Step 13: Tick on the option **Label by Line**, then click on the **Add** button.

Step 14: In the drawing, click on two locations inside the surface – contour labels will be added along the line between the two points.

Labels will be created along the line.

This line displays when the Create Surface form is displayed – moving the line using BricsCAD grips will move the contour label locations.

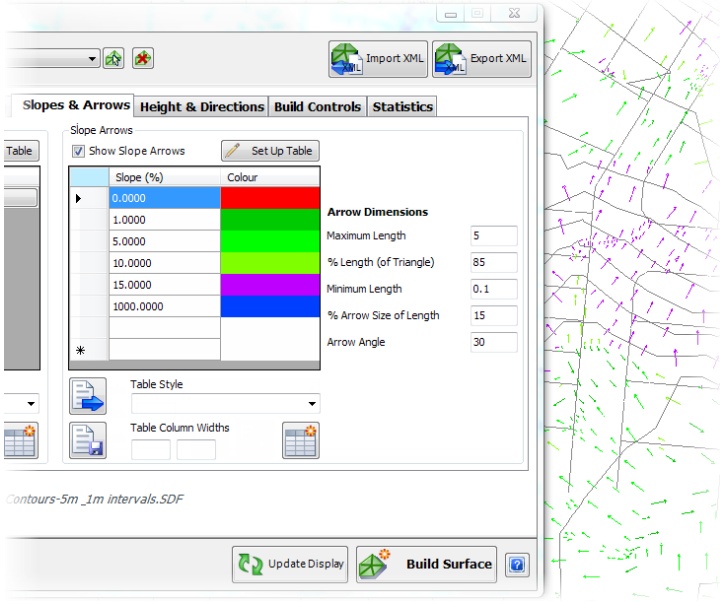


*Notes: You can change the Type and click on **Update Labels** to update the labels in the drawing.*

Contour lines will not be hidden by labels in BricsCAD.

Slope Arrows

Sometimes it's helpful to see the direction of flow as well as see 'at a glance' the steep and flat areas across the surface – this is done by displaying slope Arrows



Step 15: Click on the **Outputs > Slopes & Arrows** tab and tick on **Show Slope Arrows**

Step 16: Click on the **Update Display** button and note the slope arrows display in the drawing

Step 17: Type in different slope ranges, change range colours (by clicking on the colour swatch) and click **Update Display**

Step 18: Untick **Show Slope Arrows** and click on **Update Display**.

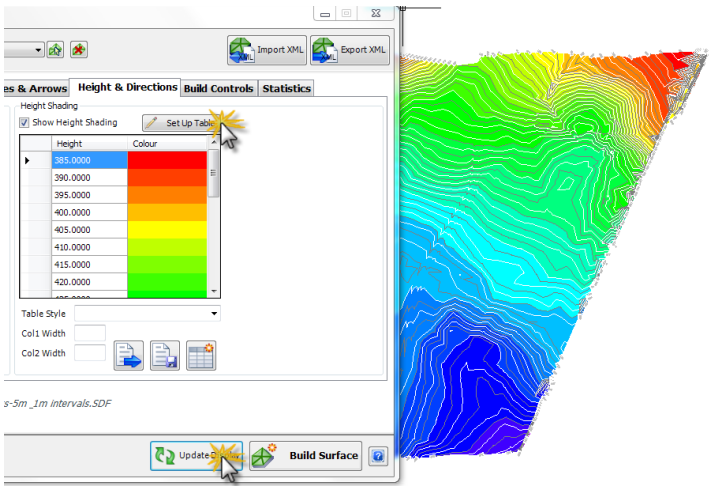
You can also select a BricsCAD table style, set the column widths and click on the **Create Table** button to add a BricsCAD table in the drawing

Note: In BricsCAD, you will need to manually merge the Title row to span the full table column extents by selecting the table row and selecting the Merge button in the table editor form.

Height Shading

It is often helpful to display the surface with different colours representing elevation ranges.

Step 19: Click on the **Outputs > Height & Directions** tab and tick on **Show Height Shading**



Now to set up the height ranges – the Set Up Table button allows you to do this easily.

Step 20: Click on the **Set up Table** button and set the following:

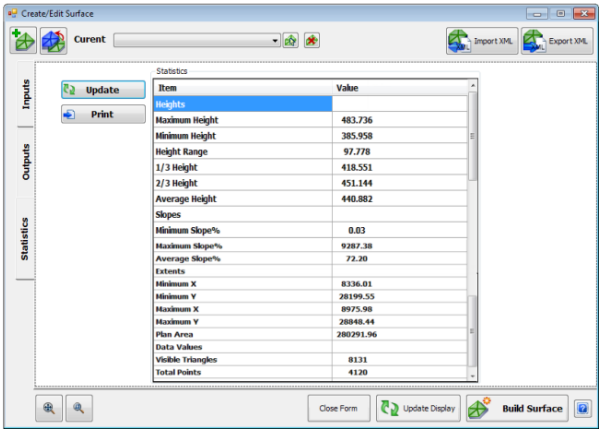
a. Under **Region Split Metho**, tick on the option **By Increment** and set the increment to **5**.
b. Click on the colour swatch and set the colour to **Red**.
c. Click OK to apply the colour ranges, and then click on **Update Display**.

Step 21: Untick **Show Height Shading** and click on **Update Display**.

Surface Statistics

Step 22: Click on the **Statistics** tab

Step 23: Click on the **Update** button and review the outputs. The **Print** button will generate a text file output



That concludes this quick tour of surfaces.

Step 24: Tick on the cross at the top of the form to close the Create Surface form

Alignments

In Civil Site Design, alignments are created from polylines. The alignment creation/editing form includes full IP editing tools to add and edit curves (with or without spirals). Labelling occurs as the design changes – you get to control the layer, text size, orientation and spacings for labelling. You can immediately create BricsCAD tables of the alignment from the same form from which you create or edit the alignment.

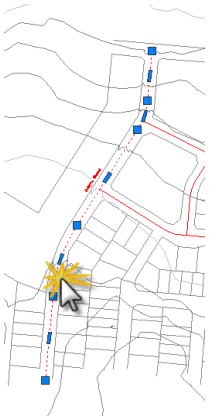
In ARD, every alignment can be given vertical geometry (turned into a String object). The software also understands how to manage particular strings in a road network – these are Roads, Kerb Returns, Cul-de-sacs and Knuckles. These are special strings you can make from the alignments.

Let's start by creating an alignment.

Step 1: From the menu, click on ***Alignments*** ➢ ***Create Alignment***.

At the prompt, click on the red polyline (on layer ALIGN) on the left side of the surface. The alignment runs north-south.

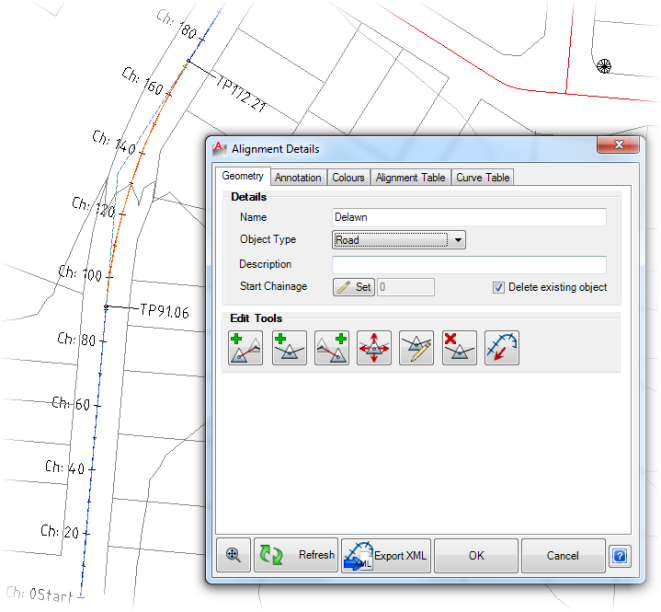
Upon selection of the polyline, the Create/Edit Alignment form will display. From here we can name the alignment, control the display and generate output tables.



Geometry Edits

Step 2: In the Geometry tab, fill out the following:
a. Name: **Delawn**
b. Object Type: **Road**

Leave **Delete Existing Object** ticked on to replace the polyline with an alignment.



The **Edit Tools** allows you to immediately edit the geometry of the alignment – move IP's around, add IP's, delete IP's, and change curve radii using the editing tools.

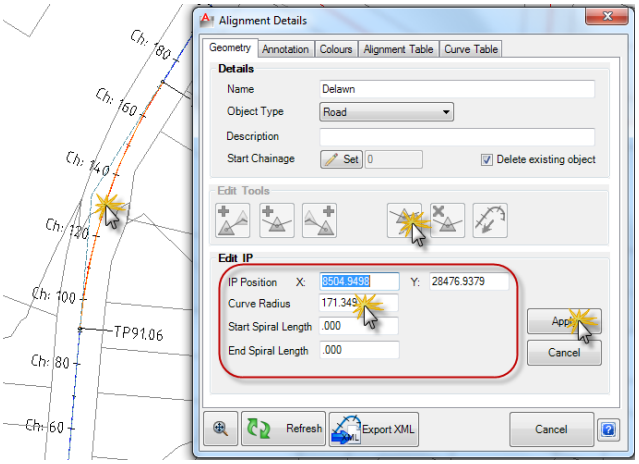
Let's change the first curve radius.

Step 3: From the Edit Tools frame, click on **Edit Curve** and click on the first curve on the alignment.

The Edit Tools panel will now change to display the details of the alignment IP selected and to allow editing of the IP details:

Step 4: Click on **Curve Radius** - change the radius to **250** and click **Apply**

The alignment will immediately update in the drawing – the new alignment geometry will be drawn and the labelling will update to suit. The original alignment geometry will display in a different colour, and will be removed upon exit from the alignment form.



Step 5: If desired, click on **Move an IP** from the Edit Tools
a. Click to select the first IP at chainage zero
b. Click to select a new IP position – note that the curve updates to maintain tangency

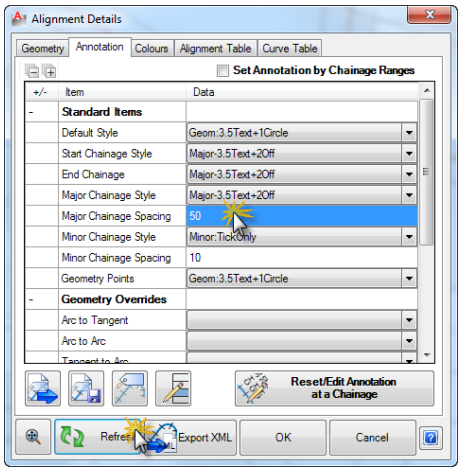
Let' review the annotation of the alignment

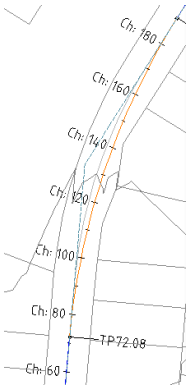
Alignment Annotation

Step 6: Click on the **Annotation** tab

The Annotation tab allows you to fully control the alignment labelling. Different geometry along the alignment can be controlled by assigning Annotation Styles – these set out the text positions, orientation, layer, number of decimal places and more. You can make your own annotation styles using the Annotation Style Editor.

Let's keep it simple by changing the Major Chainage interval:





Step 7: Click on the **Major Chainage Spacing** value and change it to **50**
Step 8: Click on the **Refresh** button – the labelling updates in the drawing

On the left is the labelling at 20m spacings – 50m spacings shown on the right.

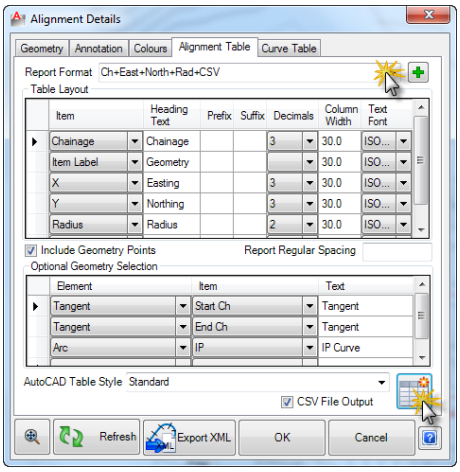
You can save the annotation configuration as an Alignment Annotation Set and apply it to any alignment.

Alignment Display - Layers

Step 9: Click on the **Colours** tab. Here you can set the layers (and colours and linetypes) to apply to the different geometric components of the alignment – lines, curves and spirals. When you are editing the alignment, you can set different layers to apply to the edited alignment – this allows you to see the original alignment geometry when edits have been applied.

The Alignment Colour Style enables the layers to be assigned to the alignment. The Alignment and Code Layer List allows you to set up layers for ARD to create an use (in both the Alignment Colour Style and in the display of drafting in plan)

Alignment Tables



Alignment Tables can be readily created and customised by you. Alignment Tables are created as BricsCAD tables in the drawing and get updated whenever you click on the **Create Table** button.

Step 10: Click on the **Alignment Table** tab
Step 11: From the **Report Format** pick list, select **Ch+East+North+Rad+CSV**

Below the pick list is the table configuration – you can edit this as desired to add more columns to report in the table and to customise the data and heading output.

You can add more Report Formats to establish your preferred tabular display.

Step 12: Click on the **Create Table** button and pick a location to the right of the surface. A BricsCAD table will be created in the drawing and will contain all of the alignment details.

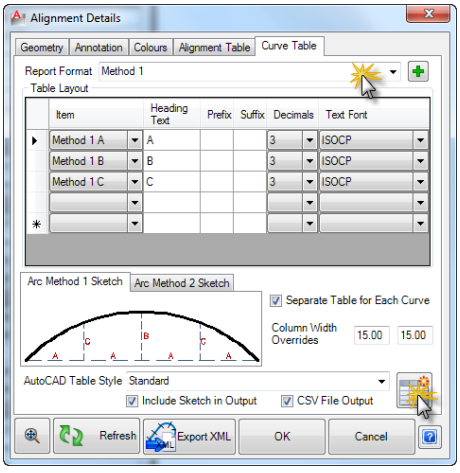
This is a BricsCAD table – you can customise the colours, layers, text size and text style by changing the BricsCAD Table Style.

Alignment DELAWN

Chainage	Geometry	Easting	Northing	Radius
0.000	Tangent	8493.445	28345.030	-
72.080	Tangent	8499.708	28416.837	-
131.277	IP Curve	8511.753	28474.655	250.00
190.475	Tangent	8537.025	28528.034	250.00
293.502	Tangent	8591.803	28615.292	-
310.077	IP Curve	8598.879	28630.239	70.00
326.653	Tangent	8602.251	28646.428	70.00
381.070	Tangent	8606.980	28700.639	-

Note: In BricsCAD, you will need to manually merge the Title row to span the full table column extents by selecting the table row and selecting the Merge button in the table editor form.

Curve Tables



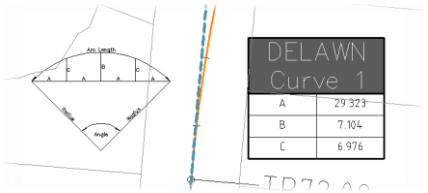
Some designs require curve tables to be created. Curve tables describe each curve on an alignment.

Step 13: Click on the **Curve Table** tab
Step 14: From the **Report Format** pick list, select **Method 1**

The Table Layout list allows you to select different parameters of the curve to display. A number of 'standard' arc setout methods are included and can be included in a curve table.

You can create one table for all the curves, or separate curve tables at each curve along the alignment.

Step 15: Click on the **Create Table** button – BricsCAD tables will be inserted at the start of each curve, as well as a block describing the 'method 1' curve setout parameters. Move the tables where you want them – clicking on the **Create Table** button again will update the tables where you have relocated them.



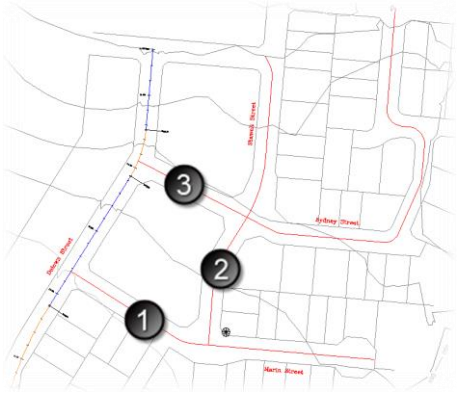
Step 16: Click on **OK** to create the Alignment.

You can click on the on ***Alignments*** ➢ ***Edit Alignment*** command to reopen the **Alignment Details** form and edit the labelling, geometry or table outputs.

Creating Multiple Alignments

Repeat the above process for creating alignments to create alignments from the other four polylines in the drawing (no need to edit the geometry or generate tables). An alternative to the below instructions is to use the Auto Create Alignments command in the Alignments menu. This will set automatic road names.

Step 17: From the menu, click on ***Alignments*** ➢ ***Create Alignment***.



Select each alignment from the drawing and assign the following from the **Geometry** tab:

1. Marin Street:
   a. Name: **Marin**
   b. Object Type: **Road**
2. Stawell Street
   a. Name: **Stawell**
   b. Object Type: **Road**
3. Sydney Street
   a. Name: **Sydney**
   b. Object Type: **Road**

Each alignment starts or ends at another alignment centreline, where tee intersections are to be formed. Stawell Street and Sydney Street cross each other, with the intention to form a crossing intersection.

There are now four alignments created in the drawing – the next step is to start making some Roads.

Roads

So far we've worked on building the horizontal geometry for creation of roads and intersections. The next step is to take the 2D plan layout geometry (the alignments) and turn them into Strings, with road cross sections assigned.

A String is an object that contains vertical geometry and (optionally) cross section geometry. It uses an alignment for horizontal geometry control. Cross sections are calculated at user defined intervals and at geometry points along the alignment – cross sections are joined together to form surface models.

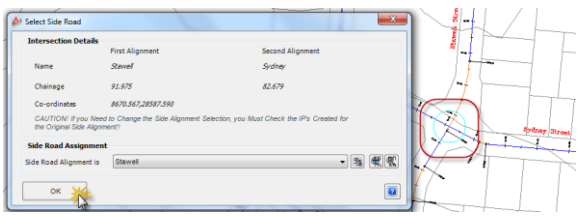
A Road is a particular type of String – as well as having its own vertical geometry and cross sections assigned at each sampled section, when one Road String meets another, or two Road Strings cross each other, they will share levels (elevations) where they intersect. In all cases of intersecting Roads, there is a 'Main' Road and a 'Side' Road. The Main Road does not get adjusted through the intersection zone; however the Side Road String vertical grading will adjust to adopt the design of the Main Road cross section, normally between the left and right edges of the Main Road. The cross section Code that is used to define the edge of Main Roads is controllable by you – the default is to look for the LEB and REB codes on the Main Road cross sections.

Typical Cross Sections - Templates

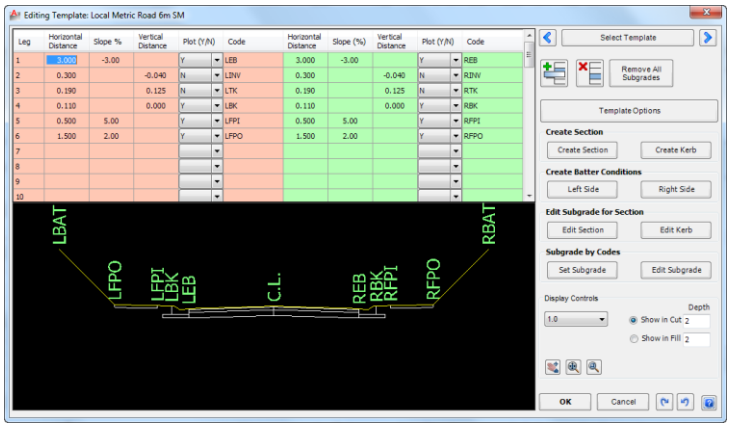
Let's start by looking at the Template Editor. The Template Editor allows you to create 'typical' cross sections to apply to a String. The 'template' normally provides a starting point for your design – after the template is applied and cross sections are created, users then edit the cross sections based on their design requirements

Step 1: From the menu, click on ***Roads*** ➢ ***Settings*** ➢ ***Create/Edit Section Templates***.

Step 2: Because two alignments are crossing, ARD needs to be told which one is the Side Road. Accept Stawell as the Side Road and click **OK**.



The **Template Editor** form will now display. The Template Editor consists of:

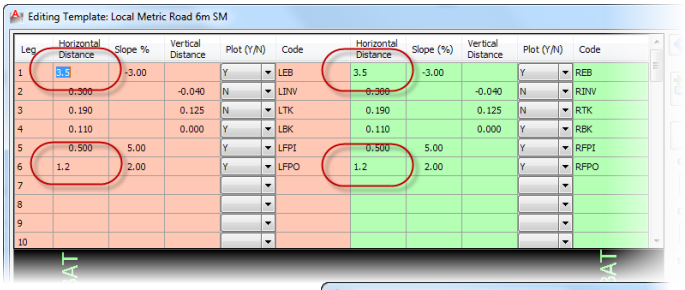


- A spreadsheet view at the top for you to create Sections (aka Legs) left and right of the String to which it will be attached. Each Section is defined by an offset distance, as slope/vertical and a Code (describing the end of the Section). The Plot option sets whether or not the Code will be textually described at the time of plotting cross sections.
- There is a visual display below the spreadsheet view – this updates as geometry is changed
- Function controls to the right
  - Use the ◀ ▶ arrows at the top to cycle through Templates
  - Click on **Template Options** to make new Templates
  - Click on Create Section or **Create Kerb** to **create Sections** (complete with subgrade layers)
  - Click on **Let Side** and **Right Side** to assign some starting batters.
  - Controls to set subgrades to be assigned if new Codes are inserted on the String.

Step 3: Set the current template to **Local Metric Road 6m SM**.

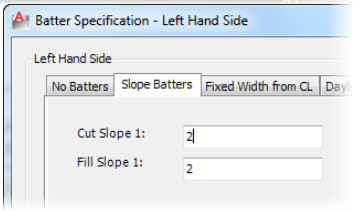
Step 4: Set the Width of Section (Leg) 1 to 3.2m on both sides – this changes the width of the road (LEB and REB codes)

Step 5: Set the width of Section (Leg) 6 to 1.2m on both sides – this changes the width of the footpath (LFPO and RFPO)



Step 6: Change the batter slopes – click on the button **Left Side** and change the Slopes to **1:2** cut and **1:2** fill

Step 7: Repeat for the right side – click on the button **Right Side** and change the Slopes to **1:2** cut and **1:2** fill

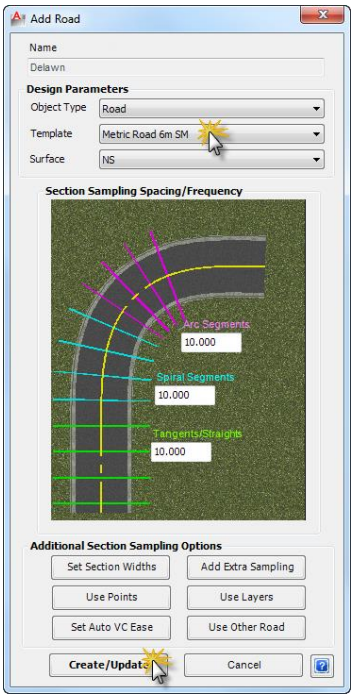


You can make as many Templates as you need by clicking on the **Template Options** button and selecting to **Create New Template in Local Library**.

We will use the **Metric Road 6m SM** Template for our Road cross sections.

Road Creation

Step 8: From the menu, click on ***Roads*** ➢ ***Create/Edit Road***.
Step 9: At the prompt, pick the **Delawn** alignment in the drawing. The Add Roads form displays:
Step 10: Set the **Template** to **Metric Road 6m SM** and click on **OK**.



In the drawing, the road linework will immediately display, representing the extrusion of the cross sections along the String vertical grading and the alignment horizontal geometry.

Note: The colours of the linework can controlled from the menu command ***Alignments > Settings >*** ***ARD Layer List***.

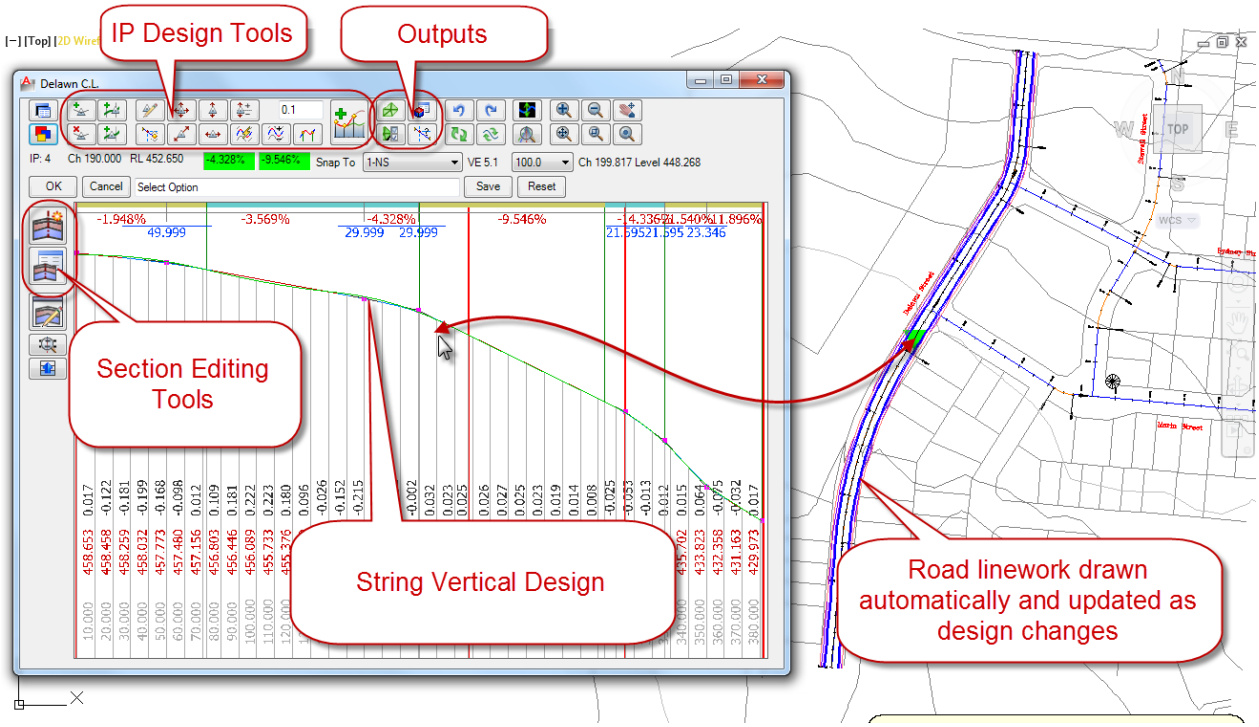
The Vertical Grading Editor Window will also display. The Vertical Grading Editor provides a comprehensive set of tools for designing the string vertical grading. Create/delete IP's, set IP levels/grades and assign vertical curves from the Vertical Grading Editor interface.

As well as managing the string vertical grading, the Vertical Grading Editor also allows the user to open cross section windows to review the cross sections as well as allowing cross sections to be edited.

The Design Data form allows you to fully customise your cross sections along your string, and can be accessed from the Vertical Grading Editor or directly from the menu or **Roads-Main** toolbar.

Vertical Grading

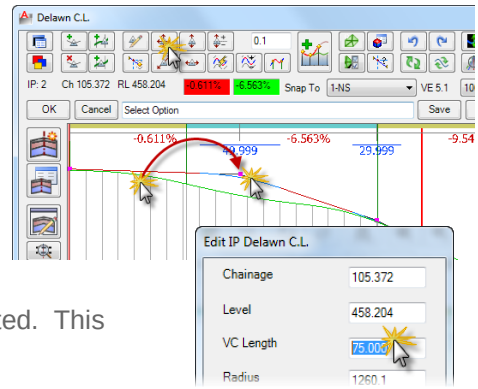
A view of the initial design and layout is shown below:



Move the Vertical Grading Editor (VGE) window where you want in the drawing and resize the window to suite. Note that the software has applied an automatic best fit design, creating multiple IP's inclusive of vertical curves. You are encouraged to edit the vertical design.

As the design changes, the linework in the drawing will update.

Step 11: In the Vertical Grading Editor, click on the **Delete IP** button up the top, near the left.
Step 12: Click in the display window of the Vertical Grading Editor at around chainage **150** – the vertical IP is removed and the design updated.



Step 13: Click on the **Move IP Anywhere** button and click on the IP located at around Chainage **50** in the VGE display window. Move the IP around and click to set a new position.

Upon selecting a new location the information updates in the VGE window and the linework in the drawing updates – have a look at the new batter offsets.

Step 14: Click on the **Edit IP** button and click on the IP just edited. This opens the IP Editor form.

Step 15: Change the **VC Length** to **75**m and click **OK**.

You can click on the **Add IP** button to create more IP's. The 'snap' IP buttons (and) will create IP's that are snapped onto the sampled (existing) surface.

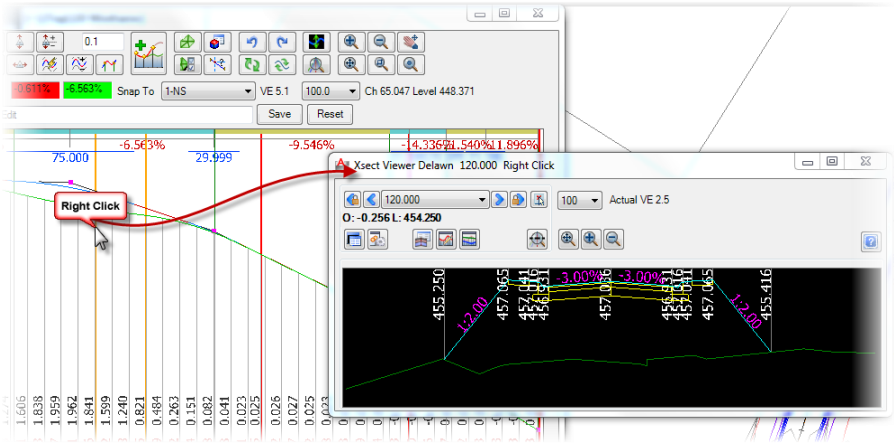
The **Raise Lower All IP's** icon allows you to set a cut/fill volume outcome – once you click OK the software raises and lowers the string to achieve the required volume result.

Note: Use the middle scroll button to zoom and pan in the Vertical Grading Editor display window. Roll the middle mouse to zoom and hold it down to pan.

There are many more IP editing tools – hover over each item to read what it does.

Cross Section Display

You can display as many cross section windows as you like, sized and positioned to suit. The quickest way to display these is directly from the Vertical Grading Editor window.



Step 16: Hover in the Vertical Grading Editor window at around the location of the IP you edited and press the right mouse button

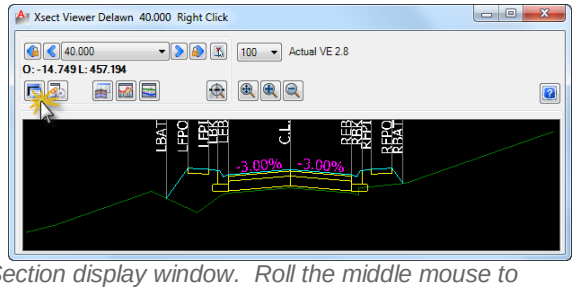
Step 17: Resize and reposition the cross section window.

You can show more cross section windows by clicking on the **Display Cross Section Window** from the VGE and then clicking a chainage to display in the VGE display.

Step 18: Click on the **Move IP Anywhere** button and click on the IP you have previously edited. Move the IP around and watch the cross section window/s. All open cross section windows automatically update as design changes are made.

Step 19: Change the display of information by clicking on the **Display Settings** button

Step 20: In the **Settings** form, untick Show Levels and tick **On** Show Code. Click **OK** to exit the form - the cross section window now displays code information, instead of design levels.



Use the arrows to navigate to different chainages or use the pick list to change the chainage to display. Right clicking on the VGE will reset the chainage for the cross section window opened with the right click.

Note: Use the middle scroll button to zoom and pan in the Cross Section display window. Roll the middle mouse to zoom and hold it down to pan.

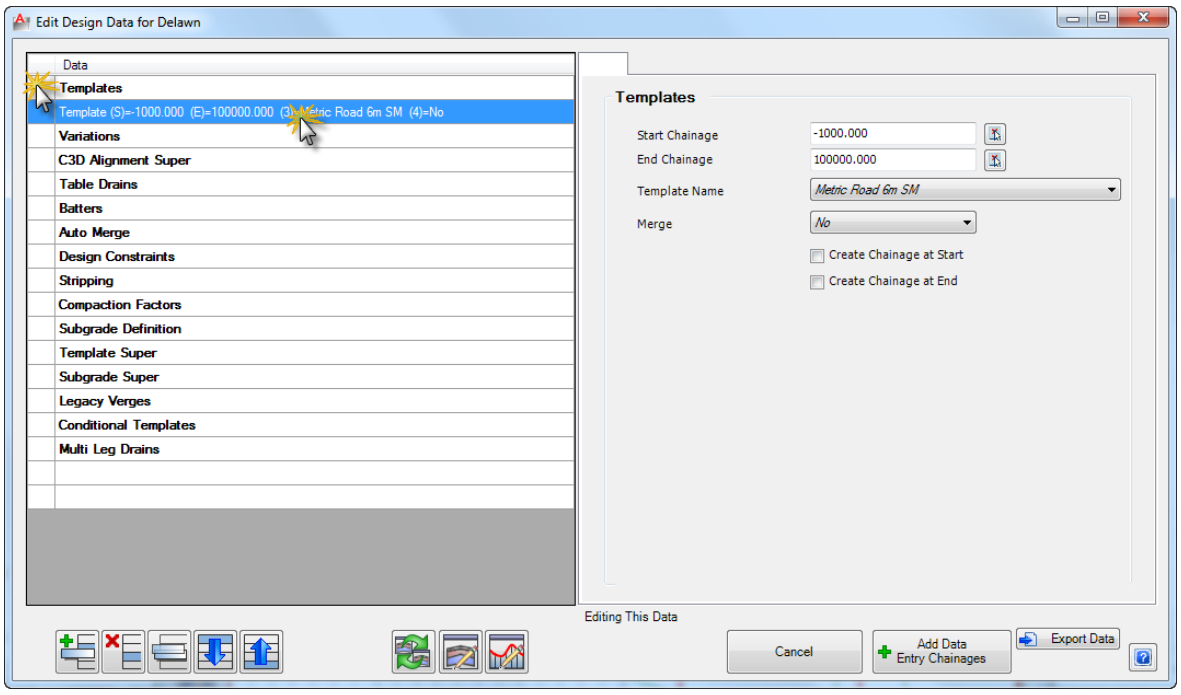
Close and open cross section windows by clicking on the X top right.

Cross Section Editing

You can edit the applied Templates directly from the VGE – just click on the **Create/Edit Template** button on the left of the VGE.

Editing cross sections along the length of the String is made from the **Design Data Form**.

Step 21: From the VGE, click on the **Design Data Form** button.



The left side panel lists all the aspects of the cross section that can be edited – anything with a + next to it can be expanded to show the inputs/edits assigned.

Step 22: Expand **Templates** and click on the template entry – the details show on the right. The **Metric 6m SM** Template is applied for the full length of the alignment.

You could completely change your cross sections by picking a different Template and clicking to apply the changes.

The **Add Entry** button is used to add more controls – this allows you to change Templates along the Road string.

The **Variations** section allows you to edit your cross sections on a code-by-code basis. With the Variations controls you apply the following edits, and more:

- Delete a Code over a chainage range
- Insert a new Code over a chainage range
- Vary the width and/or slope of a Code over a chainage range
- Set the offset of a Code to match a selected alignment
- Set the offset and level of a Code to match a selected String

The **Batters** section allows you to override the batter conditions anywhere along the Road string.

Step 23: Close the **Design Data Form** window by clicking on the X at the top right of the form.

Surface Creation

As well as creating a surface for the current String, you can create an 'Automatic Road Network Surface' model – this type of surface model automatically includes the Road, Kerb Return, Cul-de-sac and Knuckle Strings in one trimmed surface model. To set this type of surface up:

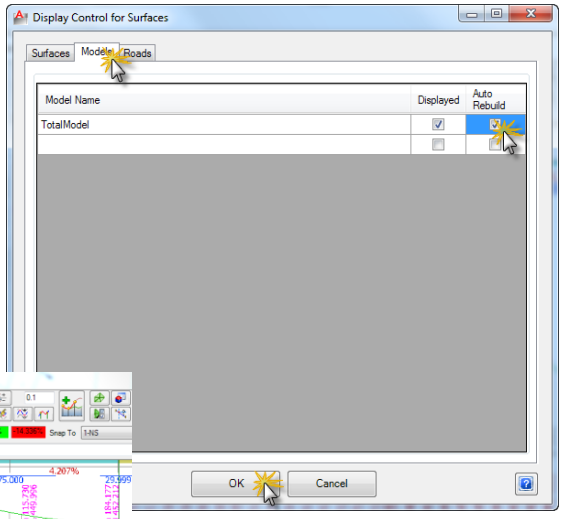
Step 1: From the menu, click on ***Roads*** ➢ ***Auto Model***. *A surface model of the current Road string is automatically created.*

Note: Use the ***Surfaces*** ➢ ***Create Surface*** *command to edit the display of the surface, or edit some aspects of the display directly from the VGE*

Step 2: From the VGE window, click on the **Control Surface Display** button

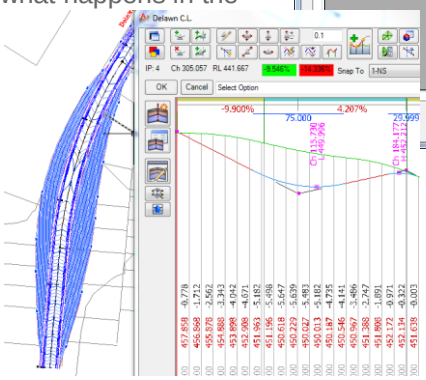
Step 3: On the Surfaces tab, you can quickly turn the mesh or contours on/off, and change contour intervals. Click on the **Models** tab

Step 4: For the **TotalModel**, tick on the option **Auto Rebuild** and click **OK**



This sets ARD to automatically update the surface as changes are made in the VGE.

You can test this by using the **Move IP Anywhere** function to move an IP and reviewing what happens in the drawing as each edit is made.



Quick Volumes

At any time during the design of any String, you can extract a quick volume summary of your string design.

Step 1: From the VGE, click on the button **Compute Volume Summary Report**. A text file will display with summary volumes for the String.

Step 2: Close the VGE window for **Delawn** by clicking on the **OK** button.

Plotting and Outputs

Let's generate some long and cross sections for the **Delawn** Road string.

Long Sections

Step 1: From the Menu, click on ***Roads*** ➢ **Plot Long Sections**. Click on the Delawn alignment in the drawing.

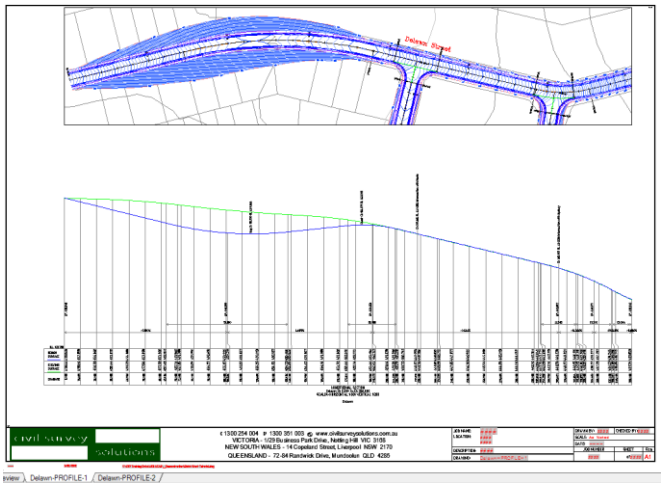
Step 2: In the **Select Sections to Plot** form, click on **Selected Sampled Sections**.

A preview long section displays in a new Layout and a form displays for you to edit and manage the display of your long section. From here you can set up the layers, scales and assign a title block. You can also fully customise the data displayed on the long section and in the rows at the bottom.

Saving styles provides a quick means of recalling long section output displays, ready for immediate plotting output.

Step 3: Click on **Load Style**, select the style **QS A1 Title** and click OK. The long section presentation will immediately update.

Step 4: Click on **Plot to Layout** to create new BricsCAD Layouts in the drawing – the long section will span multiple sheets (layouts) as required. A model view can also be included in the output.



Go back to the Model tab.

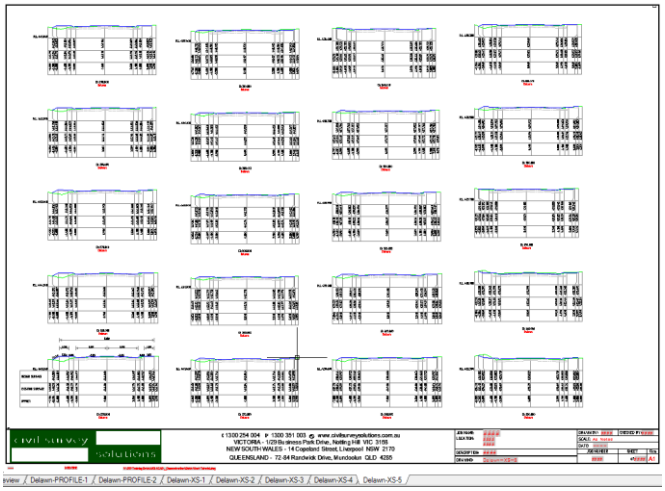
Cross Sections

Step 5: From the menu, click on ***Roads*** ➢ **Plot Cross Sections**. Click on the Delawn alignment in the drawing.

Step 6: In the **Select Sections to Plot** form, click on **Selected Sampled Sections**.

Step 7: Click on **Load Style**, select the style **QS A1 Title** and click OK. The cross section presentation will immediately update.

Step 8: Click on **Plot to Layout** to create new BricsCAD Layouts in the drawing – the cross sections will span multiple sheets (layouts) as required.



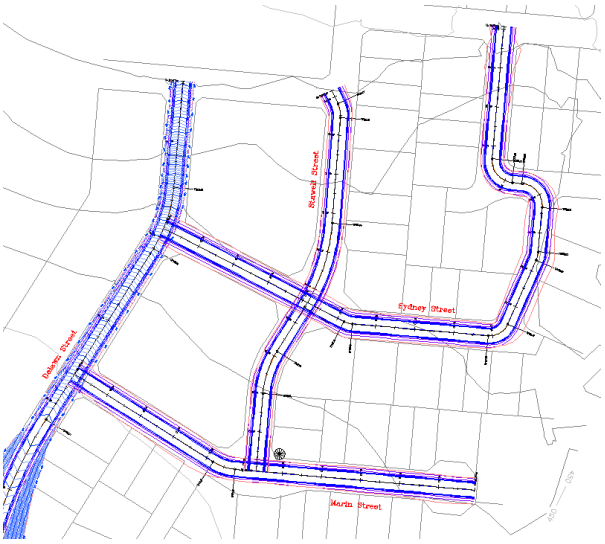
Creating Multiple Roads

The ***Create/Edit Road*** can be used to create the other Road Strings, one at a time. You can however automate the process by using the **Auto Road Creation** command.

Step 9: From the menu, click on ***Roads*** ➢ **Auto Creation Tools** ➢ ***Auto Road Creation***.

The **Add Road** form will display as before, with some command buttons removed (the ones that are unique to individual Road Strings).

Step 10: Click on **OK** (**Create/Update**) to create all the Roads.



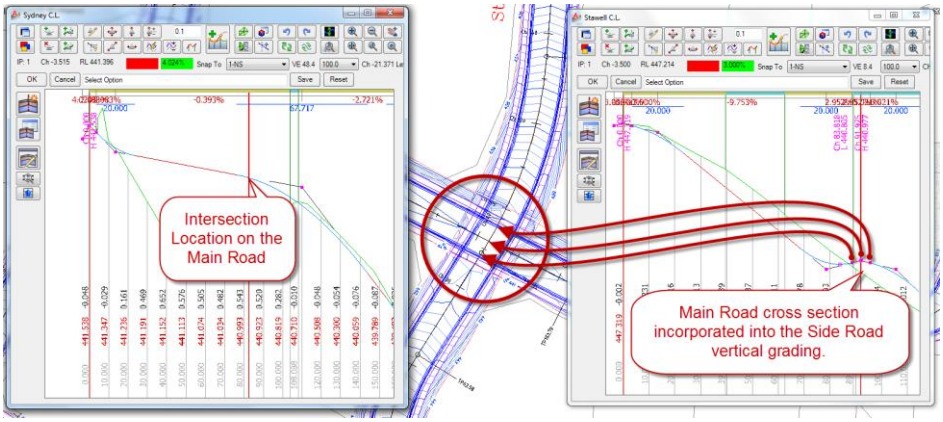
Now, let's review **Stawell** Street (This is the Road running North South in the middle of the drawing) in the Vertical Grading Editor to see the interaction with both Marin Street at the southern end and Sydney Street in the middle of the String.

Note: Until we create Kerb returns, the automatic surface model that updates will be incorrect at the intersections – as we make kerb returns the software will neatly trim up the intersections to get rid of overlapping cross sections

Step 11: From the menu, click on ***Roads*** ➢ ***Select/Open/Edit*** ➢ ***Open Vertical Grading Window.***
Step 12: At the command prompt, click on **Stawell** Street in the drawing (or press [Enter] and select **Stawell** from the list of strings).
Step 13: Repeat the above two steps and open **Sydney** Street (This is the road that forms a cross road with Stawell).

Stawell Street and Sydney Street both contain IP's at the start of each String, with the IP's describing the cross section of the intersecting 'main' Road. Stawell Street also has IP's midway along that describe the cross section of Sydney Street where it crosses Stawell Street.

Move Sydney Road up and down by editing the IP closest to the vertical red line located around chainage 80 – the vertical grading of Stawell Street will immediately update to match the new cross section of Sydney Street.



Automatic vertical curves have been created either side of the intersection – you edit these when you create or resample the road cross sections (they are not editable in the VGE window).

Intersection connectivity is automatic.

The surface will rebuild as the vertical grading of any string is edited – it's now time to create kerb returns.

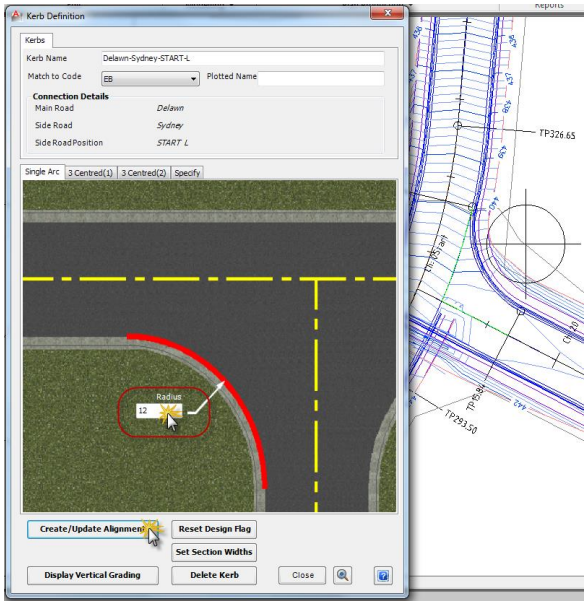
Step 14: Close all open VGE windows.

Kerb Returns

Kerb Return Strings are a special type of String – as well as supporting a vertical grading design and cross sections, kerb return strings understand that they should connect to two Road strings, specifically to particular Codes on the road strings. By default, these Codes are LEB and REB (representing left edge of bitumen and right edge of bitumen).

Step 1: From the menu, click on ***Roads*** ➢ ***Create/Edit Kerb Return*** and click on the northern side of the intersection between Delawn Street and Sydney Street.
Step 2: From the **Single Arc** tab, type **12** for the **Radius**
Step 3: Click on the button **Create/Update Alignment**

In the drawing, a new 'kerb' alignment is created as well as a Kerb String – the intersection is automatically remodelled to include the removal of overlapping Codes from the Main and Side Roads and the surface rebuilt.



The cross section for the kerb return string is automatically constructed from the matching Codes on the Main and Side Road cross sections – when these codes are dissimilar, users can specify a Template for the kerb return cross sections.

Step 4: If desired, change the **Radius** to **15** and click on **Create/Update Alignment**. The kerb return updates both the horizontal alignment as well as the String vertical grading.
Step 5: Click on **Display Vertical Grading**, click **OK** on a message if it displays.
Step 6: Click **Close** to close the **Kerb Definition** form.

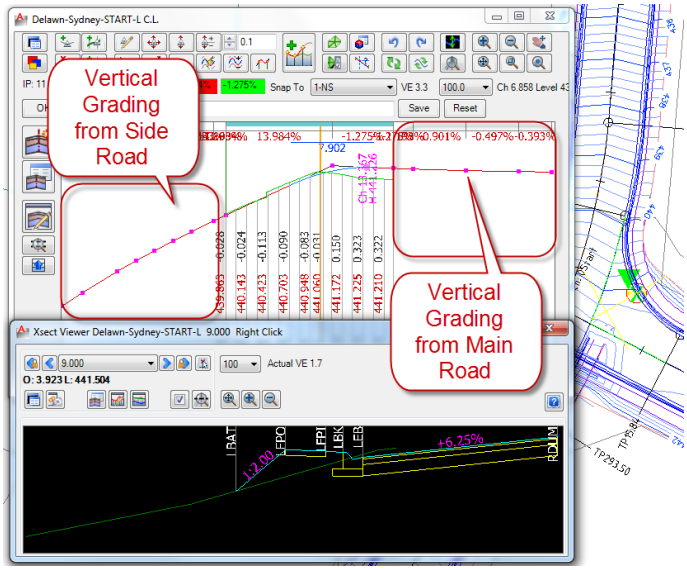
The Vertical Grading of the Kerb Return will display. The incoming IP's and outgoing IP's from the kerb return are matched to the Road cross section Code levels.

If desired, open the VGE of the Main and Side Road – edit the Main Road IP's and watch both the side road and the kerb return VGE update.

The cross sections of the kerb return are 'built' from the main and side road cross sections.

Step 7: Click **OK** to close the kerb return VGE.

*Note: Use the **Kerb Return** command to edit any kerb return string in the drawing.*



Multiple Kerb Returns

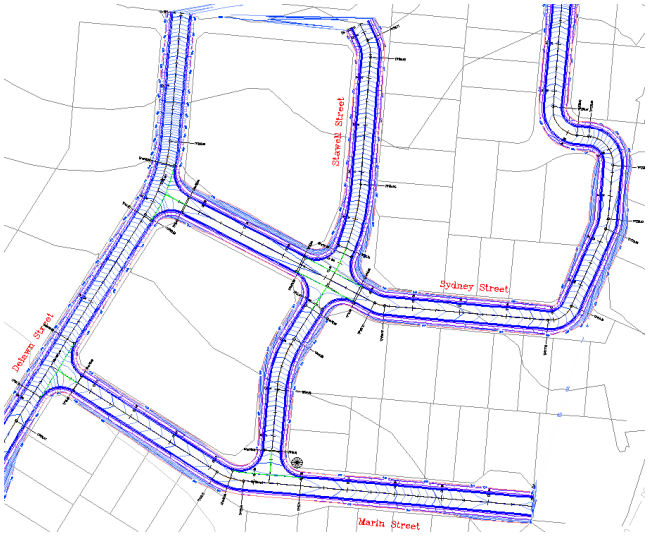
You can automate the kerb creation process.

Step 8: From the menu, click on ***Roads*** ➢ ***Auto Creation Tools*** ➢ ***Auto Kerb Returns***

Step 9: At the form that displays, type **10** for the kerb return radius and click **OK**.

The software sweeps through all the Road intersections and creates kerb return Strings at each quadrant. The surface model and the road linework immediately updates, including automatic trimming of the main and side roads to include the kerb returns.

*Note: Use the **Kerb Return** command to edit the radius of any kerb return string.*



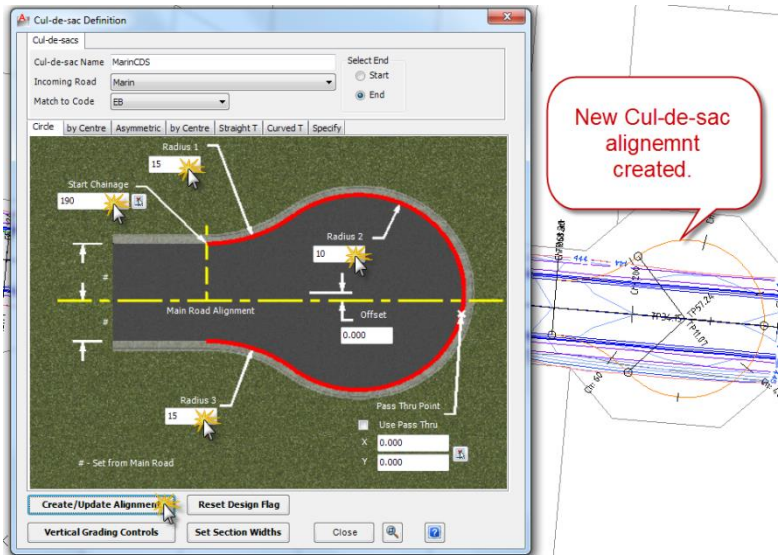
Cul-de-sacs

Cul-de-sac Strings are a special type of String – as well as supporting the vertical grading design and cross sections, cul-de-sac strings understand that they should connect to the start or end of a Road string, specifically to particular Codes on the road strings.

Step 1: From the menu, click on ***Roads*** ➢ ***Cul-de-sac*** and press [Enter] at the command prompt – this will initiate the process for creating a new Cul-de-sac

Step 2: Fill in the following:

- Cul-de-sac Name: **MarinCDS**
- Incoming Road: **Marin**
- Match to Code: **EB**
- Start Chainage: **190**
- Radius 1: **15**
- Radius 2: **10**
- Radius 3: **15**



Step 3: Click on the **Create/Update Alignment** button.

The alignment creates in the drawing.

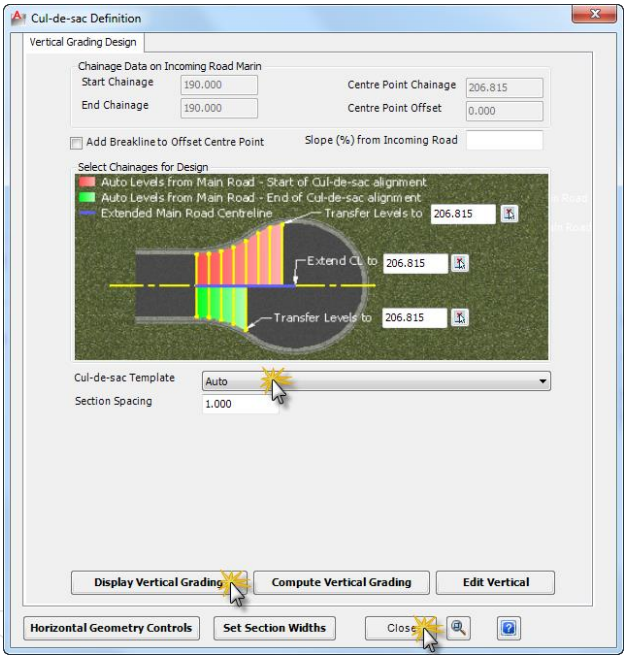
Now to establish the vertical grading controls and the modelling controls.

Step 4: Click on the **Vertical Grading Controls** button to display the vertical grading design controls.

Step 5: For the **Cul-de-sac Template** pick **Auto**

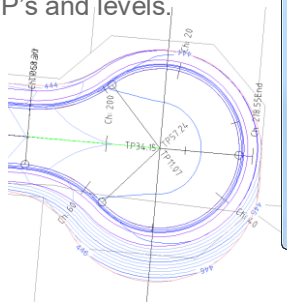
Step 6: Click on the button **Display Vertical Control** – the VGE for the MarinCDS String will display

Step 7: Click on **Close** to close the cul-de-sac form.



The cul-de-sac VGE shows the incoming and outgoing Road levels along the edge of the Road – you get to design the internal IP's and levels.

Note: The Transfer Levels options allow you to set automatic IP's to be included in the cul-de-sac vertical grading, with the elevations being calculated using the road crossfalls.



The Surface automatically rebuilds to include the cul-de-sac design.

*Note: Use the **Cul-de-sac** command to edit the cul-de-sac horizontal and vertical geometry of the cul-de-sac alignment, unless you have manually created the alignment using the Specify tab.*

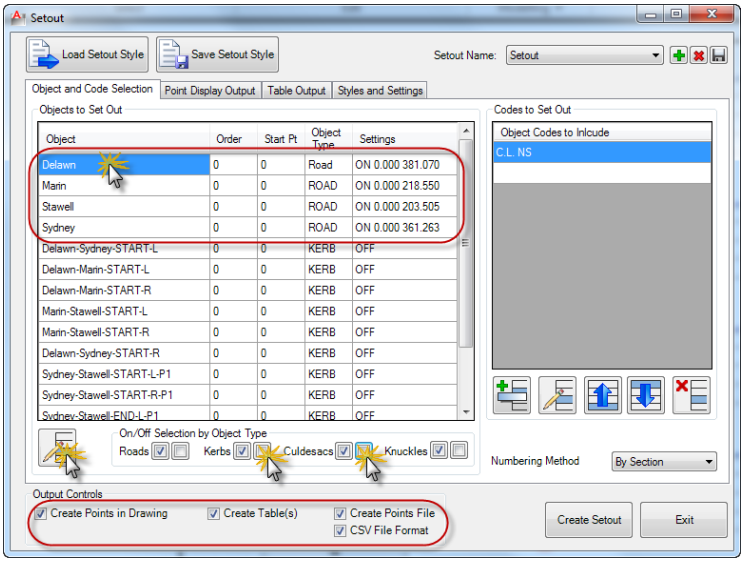
Setout

Step 15: Go to the Model Tab and from the menu, click on ***Roads*** ➢ **Multi Object Setout**.

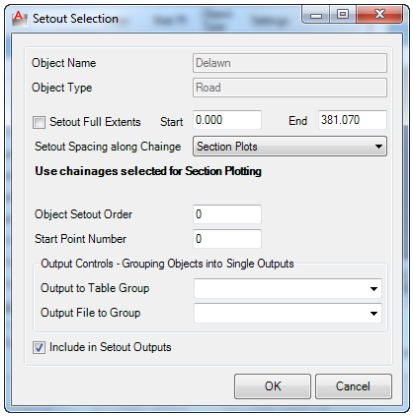
Step 16: Click **OK** at the next form to accept the name for the setout.

Step 17: Click on **Load Style**. Pick **CL Code - P (no circle) dwg + PENZD table.setOutStyle** and click OK. This will establish some Settings output controls

Step 18: From the **Object and Code Select** tab, tick the Off buttons for the **Kerbs** and **Cul-de-sacs** so we just set out the Road centrelines.



You can pre-set which sampled cross sections are used in the creation of setout points. By default, the Roads will setout every sampled section – it's often preferred to limit this to the same sampling as the cross sections.



Step 19: In the **Objects to Set Out** list, click on **Delawn**, then click on the **Edit** button.

Step 20: In the **Setout Spacing along Chainage**, use the pick list to set the spacings to **Section Plots**. This sets the setout spacings to match the cross section plot sampling.

You can also set a starting point number for each Road.

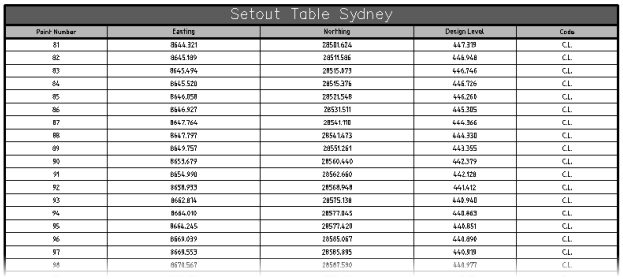
Step 21: Click **OK**.

From the **Setout** form, click on the **Create Setout** button.

At the prompts, click on screen to set locations for the tables.

Point number text will be displayed in the drawing and BricsCAD tables of the setout will be displayed in the drawing.

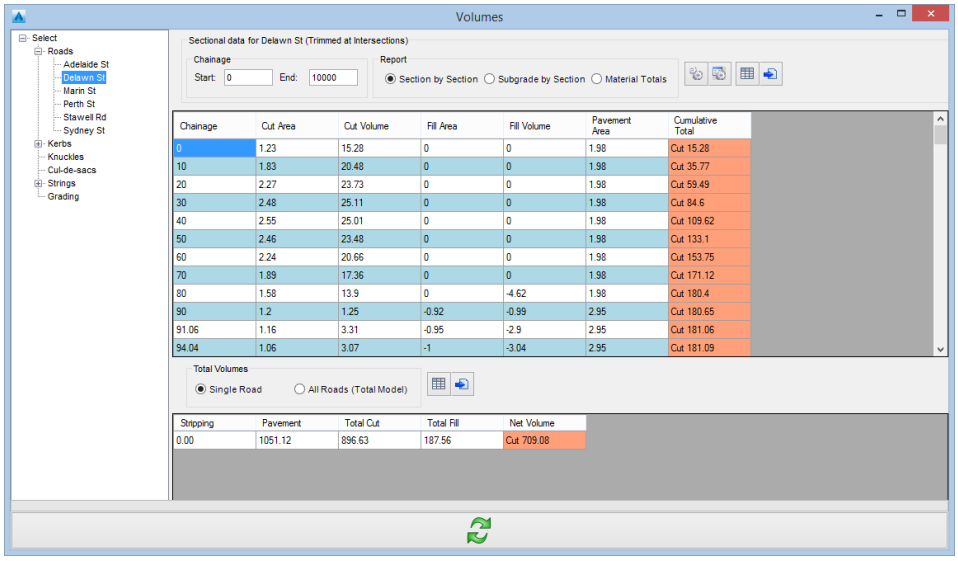
Note: In BricsCAD, you will need to manually merge the Title row to span the full table column extents by selecting the table row and selecting the Merge button in the table editor form.


Setout Table Sydney

Point Number	Easting	Northing	Design Level	Code
81	8644.321	28581.624	447.378	CL
82	8645.589	28515.586	448.948	CL
83	8645.496	28515.373	448.744	CL
84	8645.520	28515.376	448.726	CL
85	8646.058	28521.548	448.260	CL
86	8646.927	28531.511	445.385	CL
87	8647.764	28541.118	444.366	CL
88	8647.797	28541.473	444.338	CL
89	8649.757	28551.261	443.355	CL
90	8653.679	28560.440	442.379	CL
91	8654.990	28562.660	442.128	CL
92	8658.933	28568.948	441.412	CL
93	8662.874	28575.138	440.948	CL
94	8664.010	28577.865	440.863	CL
95	8666.245	28579.428	440.851	CL
96	8669.039	28585.067	440.896	CL
97	8669.553	28585.895	440.939	CL
98	8671.567	28587.590	441.977	CL

Volume Reports

You can generate volume reports for each String, as well as a summary volume report across the entire road network (this includes trimming of the roads and inclusion of the kerb returns, cul-de-sacs and knuckles)



Step 1: From the menu, click on ***Roads*** ➢ ***Reports*** ➢ ***Volume Report***

Step 2: From the Volume Calculations form, set the following:

Step 3: Road Selection: **Delawn**

Step 4: Select **Calculation Settings**

a. Set the Left side extent to LEB
b. Set the Right side extent to RBK

c. Click Save Settings

Step 5: Select **Report Settings**
a. Under Section by Section tab, tick on the Strip Area & Design Level columns
b. Click Save Settings

Step 6: Select **AutoCAD table**
a. When prompted, pick a point on screen for the location of the AutoCAD table

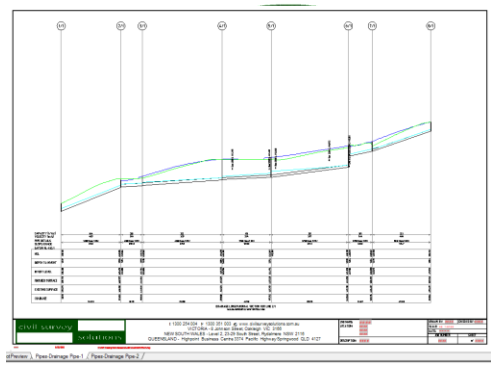
Step 7: Select **Saveas CSV file**
a. When prompted, save the CSV file in a location of your choosing

Other Road Design Stuff

Much of the functionality of the software isn't covered here – some topics include:

- String control, which can be applied to any cross sections
- Construction of your own model from any strings and codes
- Superelevation automatically applied at each curve
- Knuckle design tools
- Roundabout tools
- Intelligent (logic driven) batter conditions
- Slope pattern block insertion
- Road reconstruction (cross section editing) tools

Pipes - Drainage



Quickly create, edit and plot pipe networks using ARD.

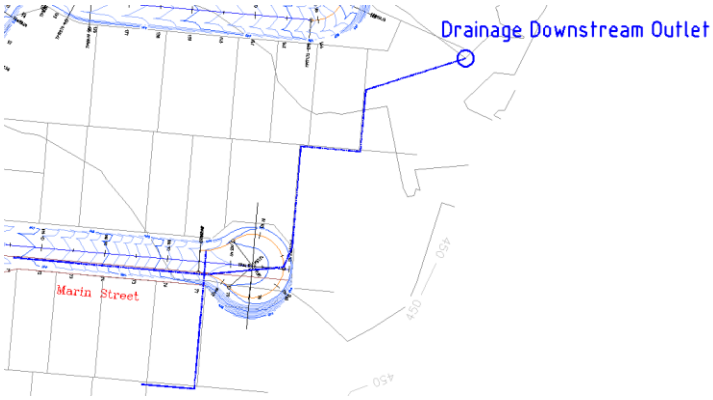
ARD Pipes supports the creation and layout of stormwater drainage pipes (Rational Method), sewer pipes, service obstruction and general pipe networks.

Plotting and report outputs are made directly into the drawing, so revision control is made simple.

Like Roads, the pipes module supports the display of multiple branches along any network in separate design windows (pipe Vertical Grading Editor windows) from which you can readily edit pipe sizes, invert/obvert levels, insert and delete pits.

Let's have a look a creating a quick drainage network from some polylines in the drawing.

In the drawing command line, type **LAYON** and press [Enter}. All layers will turn on – there are layers in the drawing representing pipe and pit locations. ARD can create pipes and pits directly from polylines, converting each polyline vertex into a pit.



Note the following layers in the drawing:

- PIPES-DRAINAGE

This layer contains the polylines that represent the desired locations for pipes and pits.

Pipe Settings

There a number of pipe settings that control the list of pits, pipes, design controls of cover and slope, minimum and maximum flow velocities, freeboard, rainfall locations, design storm ARI's and more. Let's have a look at some of the critical items for drainage design:

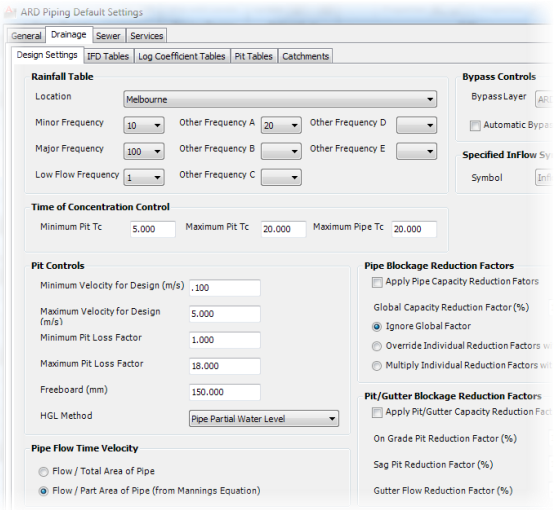
Step 1: From the menu, click on ***Pipes*** ➢ ***Settings*** ➢ ***Active Network***

Step 2: From the **General Tab** then click on the **Defaults Tab**, set the **Design Surface** to be **TotalModel**. Top of pit levels are initially set to match the design surface

Step 3: Click on the **Pipe Tables** tab and review the list of Pipes – pipes are set by Pipe Class and Pipe Type. The pipe type list allows the user to type in pipe ID's, set pipe sizes and other geometric and hydraulic controls.

Step 4: Click on the **Drainage Tab** then click on the **Design Settings** tab. Items to confirm/edit include:

a. Location: **Melbourne**
b. Minor Frequency: **10**yr ARI
c. HGL Method: **Pipe Partial Water Level**
d. Pipe Flow Time Velocity: **Flow/Part Area of Pipe**
e. Automatic Bypass to next Pit: Tick **ON**



Step 5: Click on the **IFD Tables** tab. Here is where you can create new IFD tables to use.

Step 6: Click on the **Pit Tables** tab. Here you can set up Pit Classes and Pit Types – for each Pit Type you establish the performance of the pit (eg: for on-grade pits, you specify gutter flow vs inflow)

Step 7: Click on the **Services Tab** – here you create service obstruction types and set the pipe size/shape and the clearances required around the pipe

Step 8: Click **OK** to save and exit the settings.

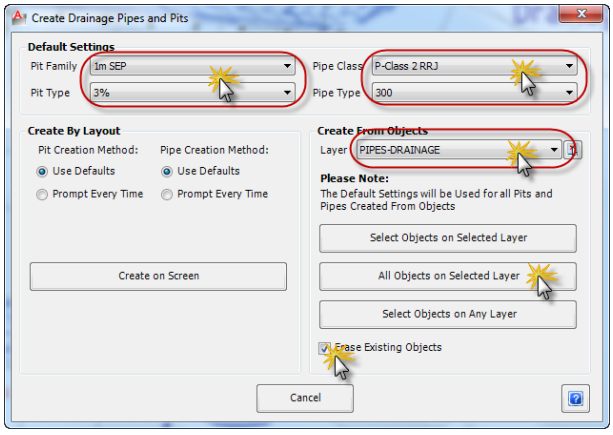
Pipe and Pit Creation

Step 9: From the menu, click on ***Pipes*** ➢ ***Create Drainage*** ➢ ***Pipes/Pits***

Step 10: In the Create Drainage Pipes and Pits dialogue, fill in the following:

- Pit Family: **1m SEP**
- Pit Type: **3%**
- Pipe Class: **P-Class 2 RRJ**
- Pipe Type: **300**
- Layer**: PIPES-DRAINAGE**
- Erase Existing Objects: Tick **ON**

Step 11: Click on **All Objects on Selected Layer**.

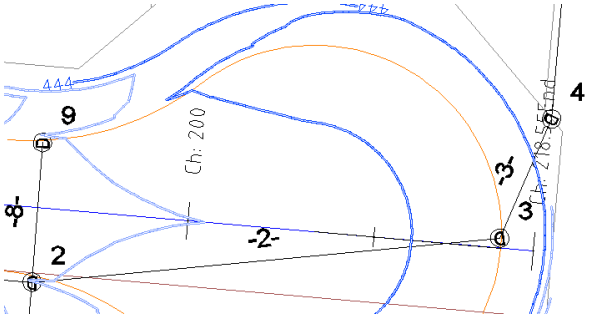


All polylines on the layer PIPES-DRAINAGE are used to create pipes and pits, including pit junctions where polyline vertices intersected.

Note: To remove pipes or pits, the ARD Pipe Editing tools must be used – deleting them in BricsCAD does not remove them from the ARD pipes/pits list.

Unless you tell it not to, when you make the pipes and pits into a network and assign/edit catchments, ARD will automatically resize the pipes and change the invert levels to achieve the required design hydraulic grade line (HGL).

Let's have a quick look at one of the editing tools.

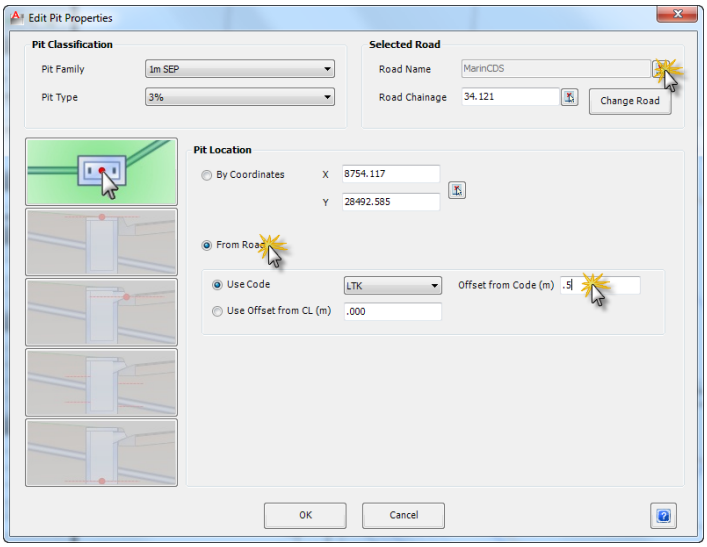


Step 12: From the menu, click on ***Pipes*** ➢ ***Edit*** ➢ ***Edit Pit Properties***.

Step 13: At the prompt, click on Pit number 3 (located at the head of the court bowl)

Step 14: In the Pit Properties form, note the icons on the left allowing control of:

a. Pit location in plan
b. Top of pit level
c. Surcharge level
d. Pipe constraints
e. Bottom of pit level



If desired, set the pit location based on the court bowl. The process is:

Step 15: Next to Road Name at the top right, click on the Pick icon. Pick the cul-de-sac alignment in the drawing, then pick for the chainage

Step 16: In the **Pit Location** frame, click on **From Road**. Pick **LTK** as the Code and type in **0.5** as the **Offset from Code**.

Step 17: Click **OK** to exit the form – the pit location (and optionally levels) is adjusted).

Now the plan location of the pit is set to be a fixed offset from the cul-de-sac design.

Other useful edits to review include **Move Pit**, **Edit Pipe Properties** and **Insert Pit**.

Assigning Catchments

Skip this step if you are doing a general pipe design, or you know your pipe sizes and levels.

Stormwater catchments can be created from polylines in the drawing or can be assigned areas directly. You set the parameters of the catchment.

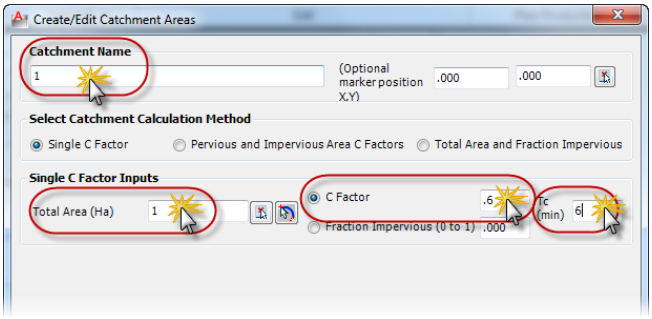
Step 1: From the menu, click on ***Pipes*** ➢ ***Create Drainage*** ➢ ***Catchment Area***.

Step 2: At the prompt, click on the leftmost upstream pit (pit number 1)

Step 3: In the **Add/Edit Catchment Areas** form, click on the **Add** button to make a new catchment.

Step 4: In the **Create/Edit Catchments** form, set the following:

a. Catchment Name: **1**
b. Method: **Single C Factor**
c. Total Area: **1** Ha
d. C Factor: **0.6**
e. Tc (min): **6**



Step 5: Click **OK** to create the catchment.

The catchment is created and a block is inserted in the drawing to represent the catchment. You are returns to th e**Add/Edit Catchment Areas** form to add more catchments to the pit.

Step 6: Click **Close** to stop adding catchments to this pit.

Step 7: At the prompt, select the northern pit at the head of the court bowl (pit 9)

Step 8: In the **Add/Edit Catchment Areas** form, click on the **Add** button to make a new catchment.

Step 9: In the **Create/Edit Catchments** form, set the following:

a. Catchment Name: **2**
b. Method: **Single C Factor**
c. Total Area: **1** Ha
d. C Factor: **0.7**
e. Tc (min): **6**

Step 10: Click **OK** to create the catchment.

Step 11: Click **Close** to stop adding catchments to this pit and then press [Esc] to stop assigning catchments.

Creating a Network

Currently you have a collection of connected pipes and pits. You now need to make them into a Network by specifying a single downstream (outlet) pit and assigning the outlet water level. Once this is done, ARD will route the flows through the network and determine the HGL outputs and assign pipe sizes and levels to deal with the Minor Frequency storm.

Step 1: From the menu, click on ***Pipes*** ➢ ***Create/Update Network***.
Step 2: At the prompt, select the most eastern (rightmost) pit (pit 8) as the downstream outlet pit
Step 3: In the **Create Drainage Network** form, type in **Drainage 1** for the network name and then click **OK**.
Step 4: For the **Downstream Water Level** select **Top of Pipe** and click **OK**.

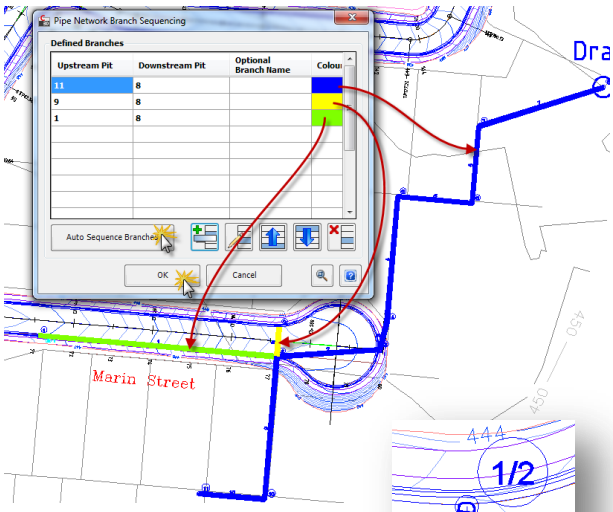
The drainage network is now formed – the pits will be circled and directional flow arrows will be presented on each pipe, with the directions pointing toward the outlet.

The pipes and pits have now been designed based on the catchments and can be reviewed in the pipe Vertical Grading Editor (VGE) windows.

Branch Sequencing

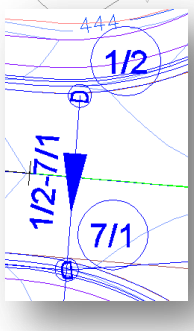
It's important to tell ARD how you want to plot the pipe runs (in what order) – to do this you apply Branch Sequencing. After you have sequenced the network you can also renumber the pits based on the branch numbers/names.

Step 5: From the menu, click on ***Pipes*** ➢ ***Branch Sequence***.
Step 6: At the prompt, select a pipe on the network and click **OK** to confirm the selection
Step 7: In the Pipe Network Branch Sequencing form, click on **Auto Sequence Branches** to have the software automatically establish branch sequencing – note that the pipes in the drawing highlight based on the branch order assigned. You can edit the branch sequencing manually as well as establish branch names.
Step 8: Click OK to save the branch sequencing and exit.



Now to establish pipe names based on the branch sequencing:

Step 9: From the menu, click on ***Pipes*** ➢ ***Network Labelling Settings***.
Step 10: From the Network Labelling Settings form, set the following:
  a. Network Labelling Style: **Pit Number/ Line Number**
  b. **Defaults for Pipe Labels and Pit Labels:** type a hyphen (**-**) in the **Middle** cell
Step 11: Click **OK** to apply the adjusted pipe and pit labels based on the branch sequencing.



Note: You can make your own labelling style for the pits.

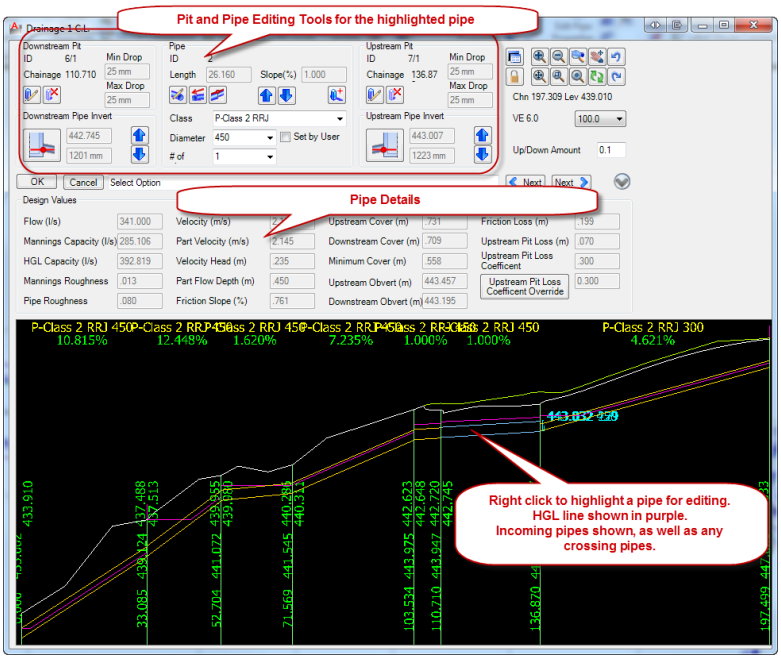
Editing Pipes in a Network

One of the most powerful tools for quickly editing your pipe network is via the Pipe Vertical Grading Editor. The Vertical Grading Editor allows you to view and edit a pipe long section from any selected pit to the downstream outlet – you can delete pits, insert pits, change pipe sizes/types and edit pipe invert levels directly from the Vertical Grading Editor, and see what happens to the hydraulic grade line (HGL) as you edit the network.

Step 1: From the menu, click on ***Pipes*** ➢ ***Vertical Grading Editor***.

Step 2: Click on the most upstream pit on the southern side of Marin Street. All pipes from this pit to the outlet will be highlighted for editing in the Pipes Vertical Grading Editor. Click **OK** to confirm selection.

The Pipes Vertical Grading Editor window opens. Move the Vertical Grading Editor (VGE) window where you want in the drawing and resize the window to suite. Right click on a pipe to select it for editing. Pipes are automatically sized and graded to meet design requirements. A tracker in the drawing shows the plan location while editing in the pipes VGE. The Hydraulic Grade Line (HGL) displays and updates as pipes and pits are edited.



Step 3: Right click on a pipe and note the Design Values and details at the top of the form

Step 4: For the **Downstream Pit**, click on the **Lower** button to lower the end of the pipe. Note the padlock display – this indicates that you have manually edited the pipe downstream invert.

Step 5: Right click on another pipe. For the **Pipe**, click on the **Lower** button to lower the whole pipe. Minimum pit drops will be preserved, unless you have manually edited the invert of the next upstream or downstream pipe.

Step 6: Click in the **Diameter** pick box and pick a bigger pipe size – all downstream pipes will update so they don't downsize. Un-tick **Set by User** to restore the pipe size to the 'design' size.

From the Pipes VGE you can insert pits, delete pits and set the pipe slope, as well as other edits.

Note: Use the Next buttons to cycle through the pipes and use the various zoom buttons to zoom to particular pipes along the VGE.

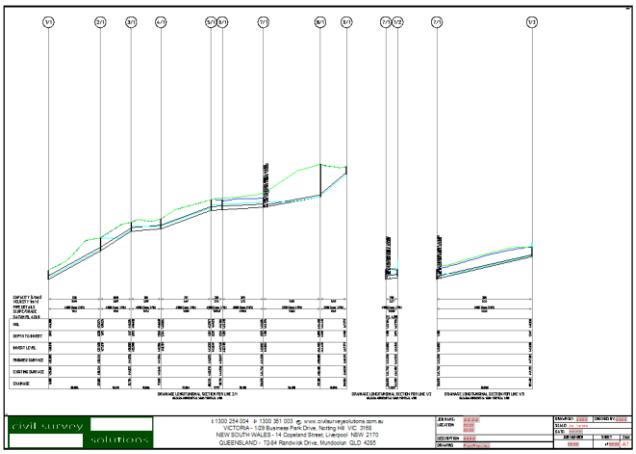
There are more pipe and pit editing tools – hover over each item to read what it does.

Plotting and Reports

Long Sections

Step 1: From the menu, click on ***Pipes*** ➢ **Plot Long Section**. Click on one of the pipes you created and then click **OK** to confirm the selection.

A preview long section displays in a new Layout and a form displays for you to edit and manage the display of your long section. From here you can set up the layers, scales, assign a title block and customise the data displayed on the long section and in the rows at the bottom.

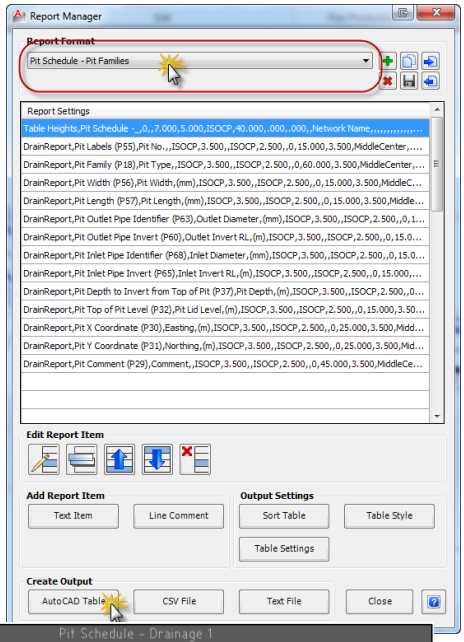


Step 2: Click on **Plot to Layout** to create new BricsCAD Layouts in the drawing – the long section will span multiple sheets (layouts) as required. The HGL, incoming pipes and (when created) crossing pipes will display.

Go back to the Model tab.

Reports

In the software you can develop your own reports to generate tables – you pick the property to report in each column, as well as set the heading and column widths. A BricsCAD table is created in the drawing – there is a command line entry to force all tables to update after you make a change to the network.



Step 3: From the menu, click on ***Pipes*** ➢ ***Reports*** ➢ **General**. Click on one of the pipes you created and then click **OK** to confirm the selection.

Step 4: From the **Report Format** pick box, select **Pit Schedule – Pit Families**. The report settings list will update – each line is setting a column to display in the table output.

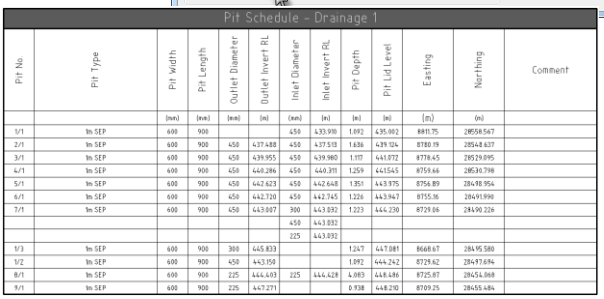
Step 5: Click on the **BricsCAD Table** button and select a location in the drawing. A BricsCAD table will be created.

Step 6: Repeat the above two steps to make the following reports:

a. Report Format: **Drainage – HGL Report**
b. Report Format: **Drainage – Pipe Report**
c. Report Format: **Drainage – Pit Report**

Note: When you elect to plot a report that contains data that is dependent on the storm frequency (ARI), you will be prompted to select the storm frequency/ies to output. A table will be created for each frequency.

Note: In BricsCAD, you will need to manually merge the Title row to span the full table column extents by selecting the table row and selecting the Merge button in the table editor form.

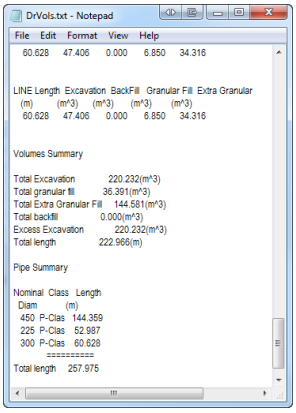


Trench Volumes

Step 7: From the menu, click on ***Pipes*** ➢ ***Reports*** ➢ **Trench Volumes**. Click on one of the pipes you created and then click **OK** to confirm the selection.

Step 8: Accept the default Subgrade Depth (this accounts for pipes that run under the road – you only backfill to the underside of the Road pavement) and click **OK**.

A text report is created, listing the volume of material removed and backfilled. At the bottom of the report is a summary of the volumes as well as a summary of the pipes used.



Note: You can change the backfill conditions for the pipes by using the ***Set Pipe Backfill Lengths*** *command.*

Plan Drafting

Step 9: From the menu, click on ***Pipes*** ➢ **Label Network**. Click on one of the pipes you created and then click **OK** to confirm the selection.

Step 10: Accept the defaults in the **Select Plot Items in Plan** form and click **OK**.

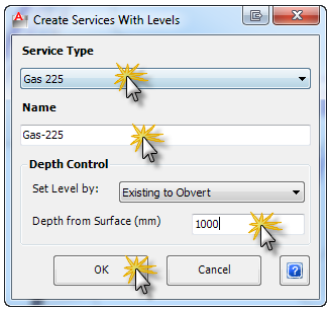
Text is added at each pit and along each pipe.

Data Share

You can share your network (including catchments and bypass) geometry to Watercom Drains or PC Drains directly. With Watercom Drains there is also the capacity to receive the results from the Drains analysis and plot the pipe, pit and HGL changes using ARD Pipes.

Service Obstructions

Crossing services are a significant design consideration in any pipe network. In ARD, you create service pipe networks directly from polylines – these will show up on your drainage, sewer and other service obstruction networks when viewed in the Pipe Vertical Grading Editor – the pipes are colour discriminated to highlight clashes and insufficient pipe clearances



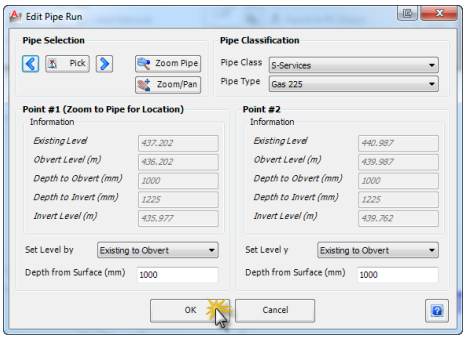
Step 1: From the menu, click on ***Pipes*** ➢ ***Create Service*** ➢ **Pipes/Pits with Levels**.

Step 2: In the **Create Service with Levels** form, set the following:

- **a.** Service Type: **Gas 225**
- **b.** Name: **Gas-225**
- **c.** Depth from Surface (mm): **1000**

Step 3: Click **OK**. At the command prompt, select the polyline on layer **Pipe-Service** (the polyline crosses the last pipe in the network).

Step 4: In the Edit Pipe Run form, you can set the levels of each polyline segment to establish pipe levels along the service network.



Step 5: Accept the defaults and click **OK**.

A service obstruction network will be created. This can be viewed and edited in the Pipe Vertical Grading Editor.

Pipe Clash Detection

Step 7: From the menu, click on ***Pipes*** ➢ ***Vertical Grading Editor***.

Step 8: Click on pit **6/1** of the Drainage network (pit is located at the head of the court bowl) and click **OK** to confirm the selection.

Step 9: In the Pipes VGE for the drainage network, note the conflict with the crossing Gas pipes.

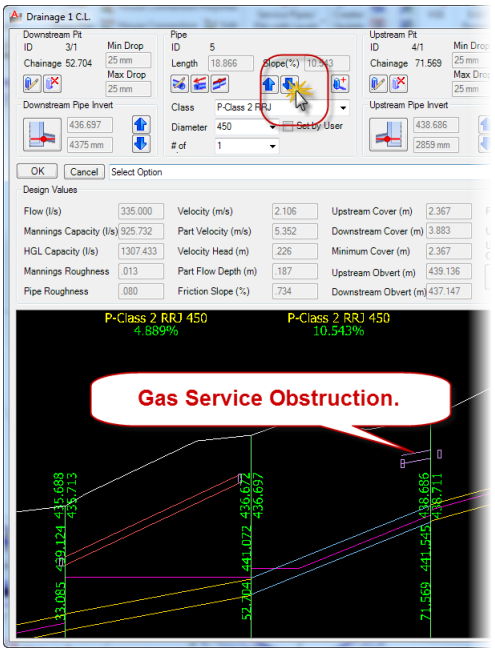
Step 10: Right click on a pipe where the crossing gas is in conflict.

Step 11: For the **Pipe**, click on the **Lower** button to lower the whole pipe. Minimum pit drops will be preserved, to lower upstream and downstream pipes

Step 12: As required, right click on another and lower the pipe to avoid the services

Note: Service Pipes change colour when they are outside the clearance of both pipes.

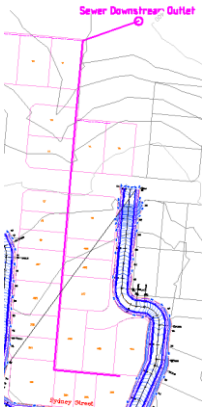
Crossing services (and in fact any crossing pipes of any network) display on the long sections.



Pipes - Sewer

Sewer pipe design processes parallel those of drainage – the process for creating pipes and pits, editing pipes and pits, creating networks, editing in the Pipe Vertical Grading Editor and plotting are the same.

Where sewer differs from drainage is in regards to lot control (house connections) – for sewer, the levels of the sewer mains are driven by the lowest connection levels from adjoining properties.



Step 1: ARD can create pipes and pits directly from polylines, converting each polyline vertex into a pit.

Note the following layers in the drawing:

- PIPES-SEWER – polyline representing the desired location for pipes and pits
- LOTS-SEWER – closed polylines for each property

This layer contains the polylines that represent the desired locations for pipes and pits.

Pipe Settings

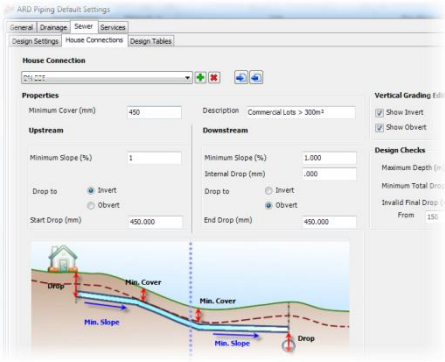
There are specific settings for Sewer design – these relate to pit drop controls, pipe slop controls and house connection controls.

Step 2: From the menu, click on ***Pipes*** ➢ ***Settings*** ➢ ***Active Network***

Step 3: Click on the **Sewer Tab** then click on the **House Connections Tab**. You can make new House Connections and set the slope, cover and drop behaviour of the house connection – this is used to establish the initial house connection levels and set the sewer pipe levels.

Step 4: Click on the **Design Tables** tab. The **Direction Change Pit Drops** sets the pit drops to be assigned based on the direction change through a pit. The **Junction Pit Drops** tab sets the pit drops to apply at junctions, based on the worst angle change for incoming pipes.

Step 5: Click **OK** to save and exit the settings.



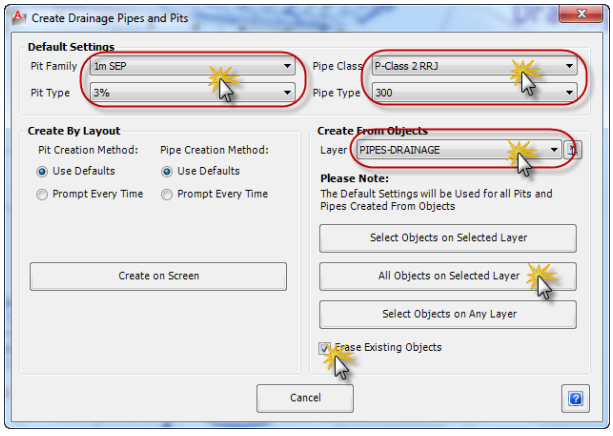
Pipe and Pit Creation

Step 6: From the menu, click on ***Pipes*** ➢ ***Create Sewer*** ➢ ***Pipes/Pits***

Step 7: In the Create Sewer Pipes and Pits dialogue, fill in the following:

- Pipe Class: **P-Class 2 RRJ** or **P-PVC**
- Pipe Type: **300** or **225** or **150**
- Layer**: PIPES-SEWER**
- Erase Existing Objects: Tick **ON**

Step 8: Click on **All Objects on Selected Layer**.



All polylines on the layer PIPES-SEWER are used to create pipes and pits, including pit junctions where polyline vertices intersected.

Note: To remove pipes or pits, the ARD Pipe Editing tools must be used – deleting them in BricsCAD does not remove them from the ARD pipes/pits list.

As discussed in the Drainage Pipes exercises, a range of editing tools are included to enable you to control the position, types, sizes and levels of pipes and pits. Some key editing tools include ***Edit Pit Properties***, **Move Pit**, **Edit Pipe Properties** and **Insert Pit**.

Creating a Network

Step 9: From the menu, click on ***Pipes*** ➢ ***Create/Update Network***.
Step 10: At the prompt, select the most north eastern (rightmost) pit (pit 21) as the downstream outlet pit
Step 11: In the **Create Drainage Network** form, type in **Sewer 1** for the network name, set the **Sewer Type** to **Default** and then click **OK**.

The sewer network is now formed

Branch Sequencing

Step 12: From the menu, click on ***Pipes*** ➢ ***Branch Sequence***.
Step 13: At the prompt, select a pipe on the network and click **OK** to confirm the selection
Step 14: In the Pipe Network Branch Sequencing form, click on **Auto Sequence Branches** and click **OK**
Step 15: From the menu, click on ***Pipes*** ➢ ***Network Labelling Settings***.
Step 16: From the Network Labelling Settings form, set the following:
a. Network Labelling Style: **MH Line Number/ Pit Number**
b. **Defaults for Pipe Labels and Pit Labels:** type a hyphen (**-**) in the **Middle** cell
Step 17: Click **OK** to apply the adjusted pipe and pit labels based on the branch sequencing. Pits will be numbered MH Branch#-Pit#.

Note: You can make your own labelling style for the pits.

Creating House Connections (Lot Control)

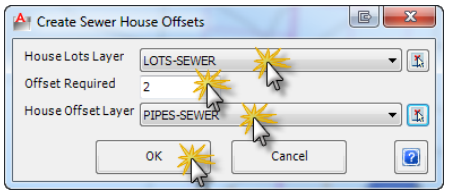
House connections (lot controls) set minimum pipe level controls required for the sewer network.

You can draw any 2D polyline that connects to the sewer pipe and turn it into a House Connection – the software will then set levels for the house connection network and use these to drive the permissible sewer pipe levels.

You can also use property boundaries in the drawing (each property being a closed polyline) to draw polylines that can be used as house connections. The tools for this include making an offset boundary (for the easement within which the house connection line can reside), putting levels at each corner (so you can see the high and low points) and drawing some 'worst case' polylines that you could immediately turn into house connections. Let's have a look at these tools.

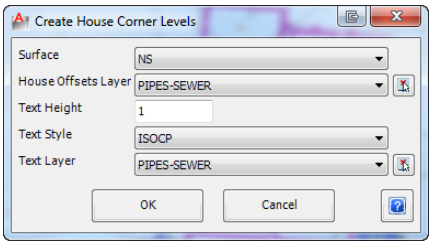
Generating some Polylines for House Connections

Step 18: From the menu, click on ***Pipes*** ➢ ***Create Sewer*** ➢ ***House Offsets***
Step 19: Use the 'pick' icon to select objects in the drawing as desired. Set the following:
a. House Lots Layer: **LOTS-SEWER**
b. Offset Required: **2**m
c. House Offset Layer: **PIPES-SEWER**

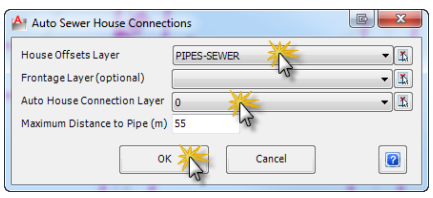


Step 20: Click **OK**. Each lot on the layer LOTS-SEWER will now have an offset polyline created.

Step 21: From the menu, click on ***Pipes*** ➢ ***Create Sewer*** ➢ ***House Corner Levels***
Step 22: Set the following:
d. Surface: **NS**
e. House Offset Layer: **PIPES-SEWER**
f. Text Height: **1**
g. Text Style: **ISOCP**
h. Text Layer: **PIPES-SEWER**



Step 23: Click **OK**. Each corner of the offset polyline will now have corner levels shown.
Step 24: From the menu, click on ***Pipes*** ➢ ***Create Sewer*** ➢ ***House Connection Polylines***.
Step 25: Set the following:
i. House Offset Layer: **PIPES-SEWER**
j. Auto House Connection Layer: **0**
Step 26: Click **OK**. Polylines are drawn – these can be used for creating House Connections as required.



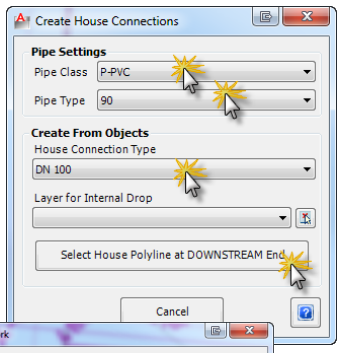
Generating House Connections

Step 27: Find Lot 10 in the drawing – it is located at the northern end of the network, on the horizontally branching pipe

Step 28: From the menu, click on ***Pipes*** ➢ ***Create Sewer*** ➢ ***House Connection***

Step 29: In the **Create House Connections** form, set the following:

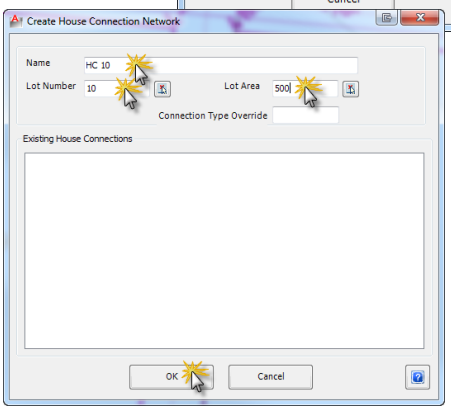
k. Pipe Class: **P-PVC**
l. Pipe Type: **90**
m. House Connection Type: **DN 100**



Step 30: Click **Select House Polyline at DOWNSTREAM End**. In the drawing, pick the polyline that runs around the western edge of lot 10 and hits the sewer line (pipe MH3-1-MH2-1). **Click on the polyline at the end closest to the sewer line**.

Step 31: Next, give the house connection a name and (optionally) at lot nubmer and lot area. Set the following:

n. Name: **HC-10**
o. Lot Number: **10**
p. Lot Area: **500**



Step 32: Click **OK** to create the house connection.

A house connection network is created – you can edit this in the Pipe VGE as desired (or to review the assigned levels)

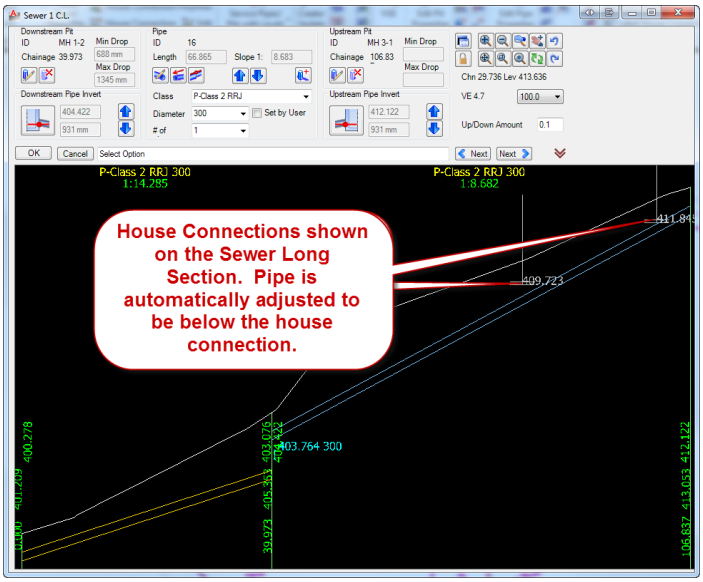
Step 33: Repeat the above steps to create a House Connection network for Lot 9, and more as desired.

Editing Pipes in a Network

Step 34: From the menu, click on ***Pipes*** ➢ ***Vertical Grading Editor***.

Step 35: Click on Pit **MH 3-1**, which the most upstream (and western) pit located on the western branch. Click **OK** to confirm selection.

The Pipes Vertical Grading Editor window opens. Pipes levels are automatically adjusted to accommodate house connection outlets. Moving the sewer pipes above the house connections will change their display colour.



Step 36: Edit the pipe levels as desired.

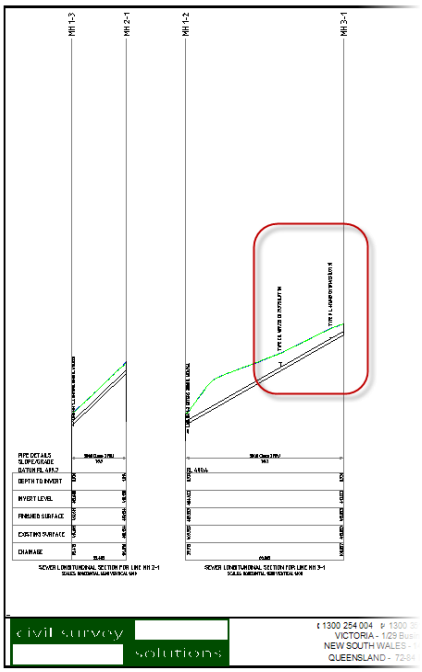
Step 37: Click OK to exit the Pipes Vertical Grading Editor

Plotting and Reports

Long Sections

Step 38: From the menu, click on ***Pipes*** ➢ **Plot Long Section**. Click on one of the pipes you created and then click **OK** to confirm the selection.

Step 39: Click on **Plot to Layout** to create new BricsCAD Layouts in the drawing – the long section will span multiple sheets (layouts) as required. The House Connections, incoming pipes and (when created) crossing pipes will display.



Go back to the Model tab.

Reports

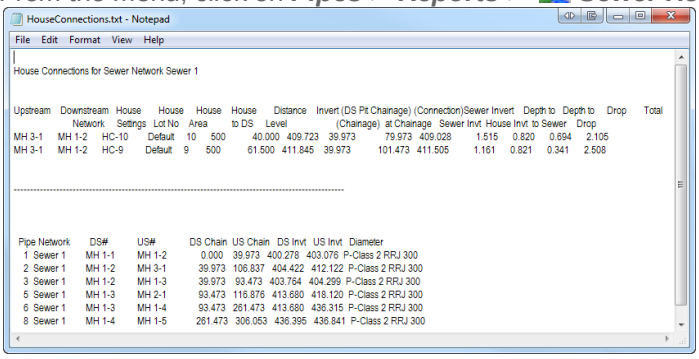
In the software you can develop your own reports to generate tables – you pick the property to report in each column, as well as set the heading and column widths. A BricsCAD table is created in the drawing – there is a command line entry to force all tables to update after you make a change to the network.

Step 40: From the menu, click on ***Pipes*** ➢ ***Reports*** ➢ **General**. Click on one of the pipes you created and then click **OK** to confirm the selection.

Step 41: From the **Report Format** pick box, select **Sewer – Pit Schedule**. Click on the **BricsCAD Table** button and select a location in the drawing. A BricsCAD table will be created.

There is a specific report for the house connections.

Step 42: From the menu, click on ***Pipes*** ➢ ***Reports*** ➢ **Sewer House Connections**.



HouseConnections.txt - Notepad

File Edit Format View Help

House Connections for Sewer Network Sewer 1

Upstream	Downstream Network	House Settings	House Lot No	House Area	House to DS	Distance Level	Invert (DS Pit Chainage) (Chainage)	(Connection) at Chainage	Sewer Invt	Depth to House Invt	Depth to to Sewer	Drop	Total
MH 3-1	MH 1-2	HC-10	Default	10	500	40.000	409.723 39.973	79.973	409.028	1.515	0.820	0.694	2.105
MH 3-1	MH 1-2	HC-9	Default	9	500	61.500	411.845 39.973	101.473	411.505	1.161	0.821	0.341	2.508

Pipe Network	DS#	US#	DS Chain	US Chain	DS Invt	US Invt	Diameter
1 Sewer 1	MH 1-1	MH 1-2	0.000	39.973	400.278	403.076	P-Class 2 RRJ 300
2 Sewer 1	MH 1-2	MH 3-1	39.973	106.837	404.422	412.122	P-Class 2 RRJ 300
3 Sewer 1	MH 1-2	MH 1-3	39.973	93.473	403.764	404.299	P-Class 2 RRJ 300
5 Sewer 1	MH 1-3	MH 2-1	93.473	116.876	413.680	418.120	P-Class 2 RRJ 300
6 Sewer 1	MH 1-3	MH 1-4	93.473	261.473	413.680	436.315	P-Class 2 RRJ 300
8 Sewer 1	MH 1-4	MH 1-5	261.473	306.053	436.395	436.841	P-Class 2 RRJ 300

Trench Volumes

Step 43: From the menu, click on ***Pipes*** ➢ ***Reports*** ➢ **Volumes**. Click on one of the pipes you created and then click **OK** to confirm the selection.

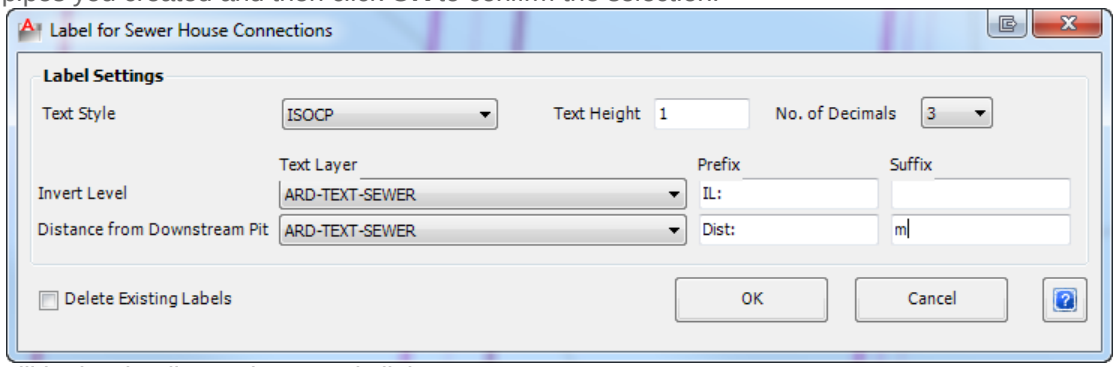
Step 44: Accept the default Subgrade Depth and click **OK**.

A text report is created, listing the volume of material removed and backfilled. At the bottom of the report is a summary of the volumes as well as a summary of the pipes used.

Plan Drafting

As well as labelling the pipes and pits, as was done with the Drainage Network, you can also label the house connections.

Step 45: From the menu, click on ***Pipes*** ➢ **Label Sewer House Connections**. Click on one of the pipes you created and then click **OK** to confirm the selection.



Step 46: Fill in the details as above and click **OK**.

Text is added at each House Connection.

HEC-RAS Support

You can generate sections in ARD and pass them to HEC-RAS. At the time of passing across the data you can include ineffective areas, houses and skewed sections. HEC-RAS results can be imported back into the drawing as 3D polylines (flooding extents) and a 'water' surface can be created.

Notes:

Notes:

Notes: