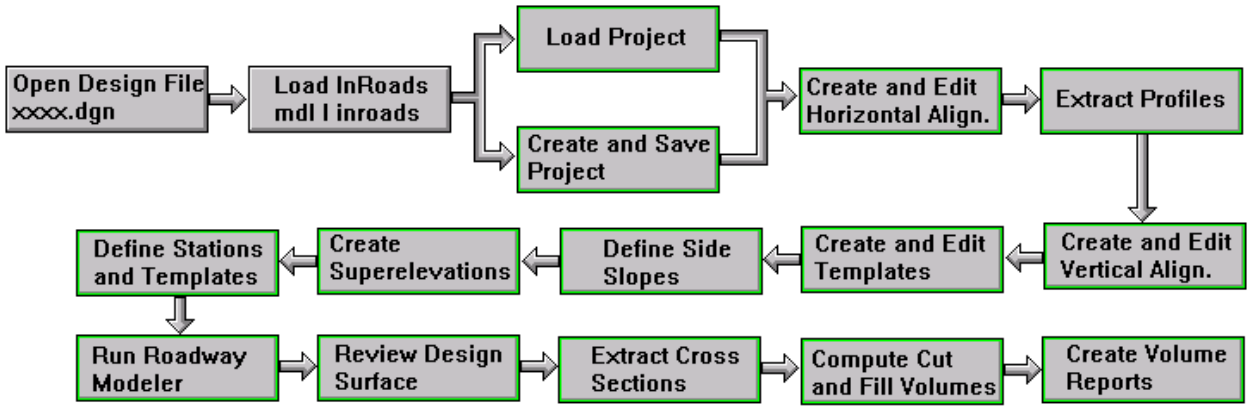
# SD DEPARTMENT OF TRANSPORTATION
# OFFICE OF ROAD DESIGN

## CADD Procedures Manual

## Section B - Design

# Table of Contents

## InRoads Master Workflow




## Getting Started

- If the project folder has not already been created, create the folder as u:/rd/prj/COUNPCEM using NT Explorer.
- Create a shortcut for the project (see A-3) and use it to start InRoads.
- Create a new Microstation 3D graphics file, saving it as ePCEM.dgn in the project folder.
- Attach the topography file as a reference file from the u:\regionXX\prj\COUNPCEM folder.
- Execute MicroStation's **File** > **Save Settings** command.
- Toggle on the **Write Lock**, the **Pen Lock** and the **Delete Ink Lock**.

If you want graphics saved every time they are drawn toggle **Delete Ink** OFF.

## Open InRoads Files

See Appendix A for instructions on creating an InRoads Project file. An InRoads Project file allows the user to simultaneously open selected InRoads design files. This file does can be created at any time,.

Original Ground Surface

Open the original ground DTM from the REGION folder (u:\regionXX\prj\COUNPCEM\PCEMorg.dtm).

Geometry Project

- **File > New.**
- Under the *Geometry* tab select Geometry Project as the geometry type and enter ePCEM as the project name. Add pertinent information in the *Description* field.
- Save the design geometry to the corresponding project folder as (ePCEM). The save process will default to "Geometry Projects (*.alg)".

## Create and Edit Horizontal Alignments

Open the geometry project (ePCEM.alg) from the project folder.

- **File > New.**

- Under the *Geometry* tab select Horizontal Alignment as the geometry type and enter mainline as the alignment name. Add pertinent information in the description window.

See CADD procedures Section A regarding proper naming convention for non-mainline alignments.

Create PI's – There are three separate and independent methods for creating horizontal alignments:

- **Geometry > Horizontal Curve Set > Add PI –** Identify points for PI's using graphics and/or key-ins. (Example key-ins: ne = northing & easting coordinates; di = distance & direction).
- **Geometry > Horizontal Elements**
- **Geometry > Utilities > Create/Edit Alignment by COGO Points –** The COGO point names must be numeric.
- The PI's can be displayed by using **Geometry > View Geometry > Horizontal Alignment**, and toggling on either **Off-Alignment Points**, or **Tangents.**

Computations are planer even if the horizontal alignment includes several different elevations.

Add Horizontal Curves (Horizontal Curve Set method)

- **Geometry > Horizontal Curve Set >Define Curve**
- Identify the curve set by clicking <Next> until the desired curve set is highlighted. Fill in the *Horizontal Curve Set Editor* dialog fields or use *Curve Calculator* to obtain the desired values.

or, to eliminate a short tangent between Horizontal Curves…

- Turn off Accusnap
- Lay out Horizontal PI's as described above.
- **Geometry > Horizontal Element > Add Floating Horizontal Curve**
- Set *Mode* to Point & Direction. Be sure the *Point* checkbox is toggled on.
- Create the 1st Horizontal Curve.
  - Select the *Point* target and snap to the PRC (midpoint of the ahead tangent is a good place to snap to, but any point along that tangent will work).
  - Select the *Direction* target and identify the agead tangent of the 1st curve.
  - Select <Apply> and select the back tangent of the first curve. Data point to accept.
- Create the 2nd Horizontal Curve.
  Select <Apply> and slelect the ahead tangent of the 2nd curve. Data point to accept.
- Delete the middle tangent.
  - **Geometry > Horizontal Element > Delete Element**
  - Set *Inclusion Mode* to Selected Element Only.
  - Select <Apply> and select the middle tangent. Data point to accept.

**SAVE !! When the information is worth saving – Save It.**

Review the Horizontal Alignment

- **Geometry > Review Horizontal**, or…
- Move the mouse pointer over the desired alignment, Right click and select *Review*

Annotating the Horizontal Alignment

- To annotate the Bearings, PC's, PT's and ends of the alignment:
  - **Geometry > View Geometry > Horizontal Annotation**
  - Set *Apply Style* to Alignment.
  - Select the appropriate *Preference* for the type of project being designed.

    - Enter the alignment to be annotated and <Apply>.
- Stationing:
    - **Geometry > View Geometry > Stationing**
    - Select the appropriate *Preference* for the type of project being designed.
    - Select the *Horizontal Alignment* to be stationed and <Apply>.
- PI curve data:
    - **Geometry > View Geometry > Curve Set Annotation**
    - Select the *Horizontal Alignment* to be annotated and <Apply>.

In general, the beginning station of horizontal alignments should be 0+00. The work should begin at 10+00 for all scales. This allows for work to begin previous to 10+00, if needed, without changing the stationing of the design data and notes. Setting the beginning station to the necessary "begin station" can be accomplished by the following:

- **Geometry > Horizontal Curve Set > Stationing**
- Select the *Horizontal Alignment*
- Enter the desired starting station in the *Starting Station* and <Apply>.

**It is very important that all the points defining the Horizontal Alignment be at the same elevation (1500.00).** If all the same elevations are not used, the plan sheets may be cut incorrectly. Review the horizontal alignment - **Geometry > Review Geometry Points >** *Report***.**

If the alignment points are on different elevations, they can all be moved to one elevation by choosing "All Points" under "Mode", typing the desired elevation and <Apply>.

If superelevation has been applied and the horizontal alignment requires modification or restationing such that the ending station is larger than the original ending station, the superelevation will have to be recalculated and reapplied (Rate Calculator and Build Application Stations). Otherwise, you will not be able to view the cross sections at the end of the alignment.
Horizontal Alignment Reports. You can also create reports to review your alignment.

- **Tools > Reports > Geometry**
- Select the *Geometry* tab.
- *Report Type* – select Horizontal Alignment.
- *Report Library* – select the SD DOT horizontal and vertical database (C:\Program Files\Bentley\Civil\data\sd_hor&vert.dba).
- *Report Template* – select either SDHORIZONTAL or SDHORIZONTAL1.
- Toggle on *Text to Screen*.
- Enter the desired alignment name in the *Report On* window.
- <Apply>.

## Extract Profiles

The purpose of this process is to create a profile for displaying the existing ground profile, and from which the new grade line may be established.

- Open the geometry project file and set active the appropriate horizontal alignment.
- Open and set active the original ground dtm (PCEMorg.dtm) from the respective region project folder.

Before creating the existing ground profile, verify the existing ground elevation limits (**Surface > Surface Properties**) and adjust the profile limits accordingly to allow for the display of design vertical PI's.

- Set **Station Lock** and **Write Lock** to ON.

- **Evaluation > Profile>Create Profile**
- Select the appropriate *Preference* for the type of project being designed (Rural, Suburban, Urban).
- Toggle the PCEMorg surface under *Symbology*.
- Toggle *Alignment* for the *Source* parameter (this will draw the original ground along the active horizontal alignment) and select the active horizontal alignment as the alignment.
- Under the *Controls* tab enter the minimum and maximum elevation limits as explained above. Likewise, *Station* Start and Stop limits should be entered at this time as necessary.
- <**Apply>** and then place a Data point to locate the lower left-hand corner of the profile. The profile grid will be generated with the execution of the **Create Profile** command.

## Create and Edit Vertical Alignments

Open the geometry project (ePCEM.alg) from the project folder.

- Set the Horizontal Alignment to mainline.
- **File > New.**
- Under the *Geometry* tab select Vertical Alignment as the geometry type and enter mainline as the alignment name. Add pertinent information in the description window.
- Graphically lay out vertical alignment PI's
- **Geometry > Vertical Curve Set > Add PI**
- Toggle on Elevation, Station or Grade locks as desired in **Dynamics** dialog box
- PI's can also be added by keying in station and elevation using **se=** (station and elevation)

View Alignment

**Geometry > View Geometry > Active Vertical**

Add vertical curves

- **Geometry > Vertical Curve Set**
- Select **Define Curve…**
- Identify the vertical PI by clicking <Next> until the desired PI is highlighted.
- Enter vertical curve length in **Length** field
- <Apply >

**SAVE !! When the information is worth saving – Save It.**

Before exporting/importing horizontal and vertical alignments, change all curve lengths to 0 feet. This will allow the alignment to be defined with PI's rather than components.

Vertical Alignment Reports

- **Tools >Reports>Geometry**
- Create the vertical alignment report following the same procedure as outlined above for horizontal alignments, substituting vertical for horizontal.

## Copy Templates and Decision Tables

Create a template library as follows:

- **File > New > Typical Section Library**
- Enter the internal name of the template library in the *Name* field (PCEM)
- Save the template library to the corresponding project folder as (PCEM). The save process will default to "Typical Section Libraries (*.tml)".
- Copy the desired template and decision tables from the template library sddot.tml. The SDDOT template library includes three templates (rural, urban, and interstate) and six decision tables (standard ditch left, standard ditch right, sloped ditch left, sloped ditch right, urban ditch left, and urban ditch right).

If you are using a standard or sloped ditch, you will probably want to copy all four of the decision tables pertaining to those respective ditches.

- **Modeler > Copy Typical Sections**
- <OK>
- <Browse> from the *From Library File* field to find U:\rd\Bentley\Civil\data\English\sddot.tml.
- Select the desired template and/or decision tables from the list.
- In the *To Library File* field enter in or <Browse> to select u:\rd\prj\COUNPCEM\PCEM.tml.
- <Apply>

Now you can edit the templates and decision tables in your template library as necessary.

Do not save your copied templates until the template library has been re-opened or your copied templates will be lost.

## Edit Templates

Open the typical section library (PCEM.tml) from the project folder.

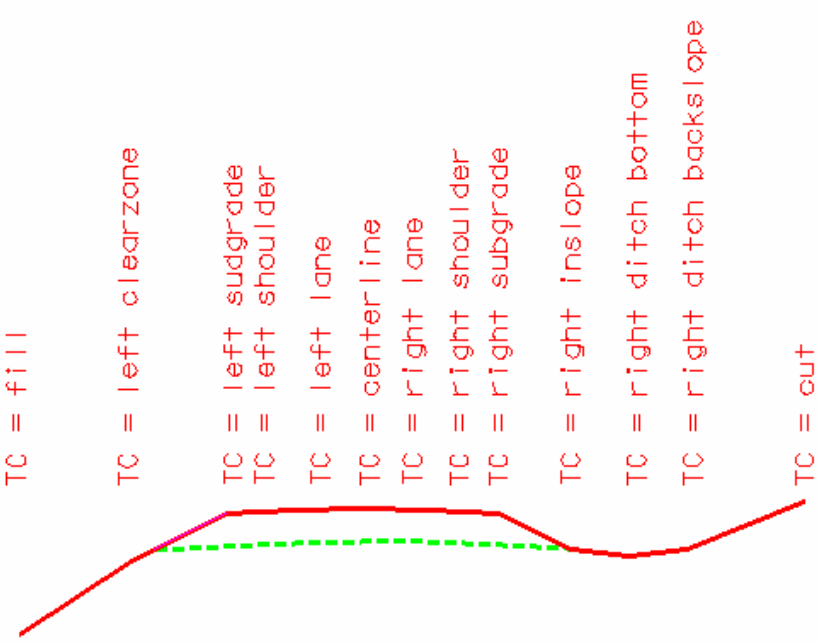
- **Modeler > Define Typical Sections**
- Select the *Template* tab.
- Select the appropriate template and <Edit>.

A mainline template will include the standard mainline Layers, subgrade and undercut. See CADD procedures Section A regarding proper naming convention for non-mainline Layers.

When creating your own templates, it should be noted that a decision table will be connected to the first layer you input regardless of naming conventions.

- When adding any layers, make sure the layer is shown on all templates if the layer is going to be used in more then one location on the project. For Example: If you have undercut on the first 1/4 of a project and the last 1/4 of a project, you must have undercut on the template in the middle 1/2. Otherwise, the software will transition the undercut through the middle 1/2 from the first 1/4 of the project to the last 1/4 of the project. The template will need to be adjusted to match the design summary. Make the necessary modifications to lane, shoulder, and subgrade widths.
- Under the *Layer* tab, select the desired layer to be modified. To edit a segment, click on the *Segment* tab. Change *Edit Mode* to *Global*, change the desired parameters, tabbing between each parameter. To add a segment, change Edit Mode to either *Add Before* or *Add After*. Type in desired parameters and <New>.
- <Next> to move you from one segment to the next.

Each segment in the template is named with a transition control name (Example: the segment to the right of centerline on the rural template is “right lane” - See Example: rural that follows). These TC Names are to be left as they are unless it is deemed necessary to change them. TC point names should remain consistent even when transitioning from one template to another.




Example: rural

When transitioning from one template to another, the templates should have the same number of segments. Where the width is 0 feet enter a “dummy” segment (0.01ft).

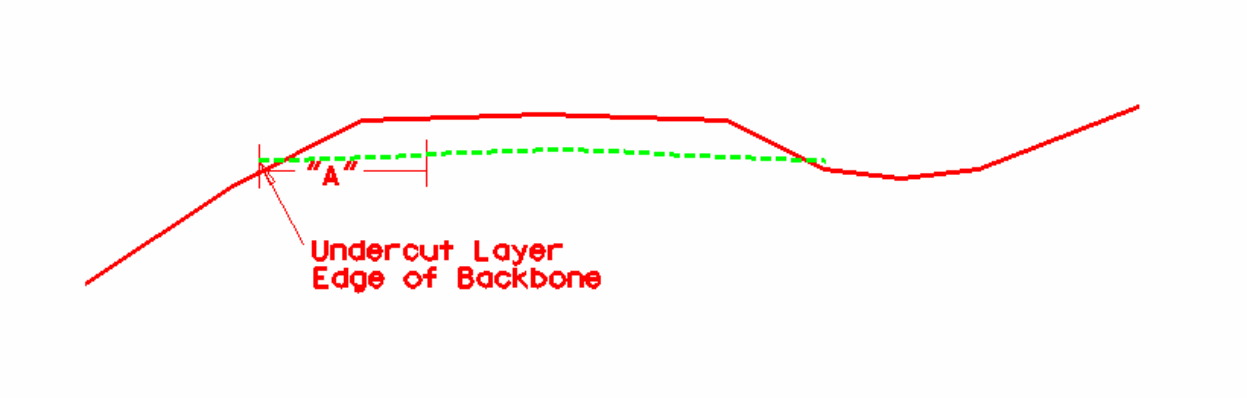
New TC names can be created or existing TC names modified in the same box. At this time the name, description, and feature style of the corresponding TC line can be set.

All TC lines are on level 56. Colors of the TC lines are as follows: ditch lines=red(2), fill lines=blue(5), lane lines=green(3), shoulder lines=purple(6), and subgrade lines=yellow(4).

The default undercut layer is set assuming 3 ft of undercut with a 4:1 inslope rural and a 6:1 inslope interstate.

The undercut layer segment TC names should be the same as the corresponding subgrade layer TC names (i.e., the undercut segment for the segment directly right of centerline should be named “right lane”, to correspond with the subgrade segment directly right of centerline). Likewise, if a segment in the subgrade is changed, the corresponding segment will need to be changed in the undercut layer.

Segment “A” in the following diagram will be set as follows:




| Shoulder Width (ft) | Undercut Depth (ft) | | |
|---|---|---|---|
| | 1 | 2 | 3 |
| 6 | 10.35 | 14.70 | 19.05 |
| 8 | 12.35 | 16.70 | 21.05 |
| 10 | 14.35 | 18.70 | 23.05 |
| 12 | 16.35 | 20.70 | 25.05 |

Table of “A” segment length (ft) in relation to Undercut Depth and Shoulder Width

**When the modifications to the template are complete, save the template library.**

- To view the template graphically:
  - **Modeler > Define Typical Sections**
  - Select the *Templates* tab.
  - Select the appropriate template
  - <Display>
  - Data a location for the template to be displayed.

The superelevation will be set for the default parameters as described in the superelevation portion of this document. If you need to adjust the superelevation range points and they are not at the outside edges of your backbone, only two segments outside of the superelevation range can be superelevated. This would be done by using the shoulder rollover locks discussed in the superelevation portion of this document.

When working with templates and superelevation dialog boxes, it may be desirable to set the slope format to 0.50 so cross slopes read as 0.02 versus 50%.

- **Tools > Options**
- Select the *Units and Format* tab.
- Under *Format* Select the *Slope* drop down menu and select 0.50.
- <Apply>

## Edit Decision Tables

Decision Tables are the method to be used by the Road Design office for defining the roadway section beyond the subgrade limits. This can define standard cut and fill sections as well as urban cut and fill sections. It is flexible in that any number of situations can be coded into a particular Decision Table to define special criteria such as a special ditch or flatter than standard fill slope.

Open the template library (PCEM.tml) from the project folder.

- **Modeler > Define Typical Sections**
- Select the *Decision Tables* tab.
- Typically decision tables will be used to define the ditch and fill sections of the roadway.
- Set the original ground surface PCEMorg active.
- Select the appropriate decision table and then select the <Edit> button.
- The default target for any decision with the table is the original ground surface PCEMorg. Within each of the decision tables previously copied from template library sddot.tml will need to be edited to target the the original ground surface.
    - Within the Edit Decision Table dialog box, select the first entry that targets "PCEMorg".
    - <Edit Record>
    - Verify that the Target *Type* has been set to *Surface* and set the Target *Surface* to PCEMorg.
    - <Apply>
    - Repeat steps 1 through for each entry that targets "PCEMorg".
- The decision table will need to be adjusted to match the scope document, as well as any special conditions that are project specific. To do this, make the necessary modifications for the ditch configuration and clearzone width.

**When the modifications to the decision table are complete, save the template library.**

To view the decision table graphically, use the same procedure as with displaying templates.

## Special Ditches

Special Ditches are ditches whose grade line does not parallel the roadway grade line. When special ditches are required, additional vertical alignments need to be created under the appropriate horizontal alignment.

- Create the vertical alignments **special ditch lt** and/or **special ditch rt** using the steps outlined above for creating vertical alignments.
- Set the newly created special ditch alignment active.
- **Geometry > Vertical Element > Add Fixed Line…**
- Set *Mode* to *By Two Points.*
- Toggle on each of the two *Station / Elevation* sets.
- Fill in the desired station and elevation for each segment, verifying that the stations are 0.1' before the desired beginning station and 0.1' after the desired ending station.
- <Apply> and data to accept.
- Repeat as necessary to complete the special ditch vertical alignment.
- **Geometry > Vertical Element > Check Integrity** must be executed to verify that any added vertical elements are in the correct order, by station.

### Decision Tables

In projects where backslopes or parameters in the template do not change often, it might be useful to use a combined decision table to represent the roadway. Create a decision table in which a special ditch is being searched. If a special ditch alignment is not found, the table will go to the next entry, which is a standard ditch, followed by entries for a fill section. In doing this, a special ditch can be created without changing the decision table each time. The following table may be useful:

| Index | Target | Start TC | End TC | Slope | Width | Seek | Con. Pt | Att. | After | Repeat | Target Type | Elev. Adj. |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 0 | mainline , special | left subgrade | left inslope | -4.00:1 | 100.00 | * | * | | | | Alg. Z | 0.50 |
| 1 | 5798org | left inslope | left special ditch | -20.00:1 | 10.00 | | * | | * | | DTM | 0.00 |
| 2 | | left special ditch | left ditch bottom | 20.00:1 | 10.00 | | * | | * | | | |
| 3 | | left ditch bottom | Cut | 7.00:1 | 100.00 | * | * | | * | 5 | | |
| New Group | | | | | | | | | | | | |
| 4 | 5798org | left subgrade | left inslope | -4.00:1 | 14.00 | | * | | | | DTM | 0.00 |
| 5 | | left inslope | left ditch bottom | -20.00:1 | 10.00 | | * | | | | | |
| 6 | | left ditch bottom | left ditch backslop | 20.00:1 | 10.00 | | * | | | | | |
| 7 | | left ditch backslope | Cut | 7.00:1 | 100.00 | * | * | | | 5 | | |
| 8 | 5798org | left subgrade | left inslope | -4.00:1 | 14.00 | * | * | | | | DTM | 0.00 |
| 9 | | left inslope | left ditch bottom | -20.00:1 | 10.00 | | * | | | | | |
| 10 | | left ditch bottom | left ditch backslop | 20.00:1 | 10.00 | | * | | | | | |
| 11 | | left ditch backslope | Cut | 5.00:1 | 100.00 | * | * | | | 5 | | |
| New Group | | | | | | | | | | | | |
| 12 | 5798org | left subgrade | left clearzone | -4.00:1 | 18.00 | * | * | | | | DTM | 0.00 |
| 13 | | left clearzone | Fill | -4.00:1 | 40.00 | * | * | | | | | |
| 14 | | left clearzone | Fill | -3.00:1 | 100.00 | * | * | | | 5 | | |
| New Group | | | | | | | | | | | | |
| 15 | 5798org | left subgrade | left inslope | -4.00:1 | 14.00 | | * | | | | DTM | 0.00 |
| 16 | | left inslope | Cut | -20.00:1 | 30.00 | * | * | | | | | |
| 17 | | left subgrade | left inslope | -4.00:1 | 14.00 | | * | | | | | |
| 18 | | left inslope | left ditch bottom | -20.00:1 | 10.00 | | * | | | | | |
| 19 | | left ditch bottom | Cut | 20.00:1 | 30.00 | * | * | | | | | |

**Note:** The above table is an example only. Different situations may require some parameters to be adjusted.

The special ditch vertical alignments must be fully created prior to running Roadway Modeler. Roadway Modeler will not recognize an empty alignment targeted in the decision tables. If the exact requirements for a special ditch vertical alignment are unknown, enter a “dummy” segment of at least 20’ into the special ditch vertical alignment.

**Annotating:**

The workflow for annotating special ditches in the cross sections is the same as for annotating continuous alignments. Once the cross sections have been annotated, verify that the special ditch has been annotated at the beginning and ending of the special ditch.

## Creating a Roadway Library

Open the template library (PCEM.tml) from the project folder.

Create a new Roadway Library.

- **File > New**
- Select the *Roadway Library* tab
- Enter the name of the roadway library as (PCEM) and enter pertinent information in the *Description* field.
- Save the roadway library to the corresponding project folder as (PCEM). The save process will default to “Roadway Libraries (*.rwl)”.

Define roadway.

- **Modeler > Define Roadway > New**
- Enter the name of the Roadway run as mainline for mainline, xrsta for cross roads, etc. and enter pertinent information in the *Description* field.
- <Apply>

Creating Station and Template Entries.

- Select the newly crated Roadway definition and select <Edit>.
- <New>
- In the *Station* field, enter the station where the desired template use will begin.
- Set *Mode* to Left and Right. This will allow each side of the roadway to be edited separately.
- In the *Interval* field, input the interval by which InRoads will "drop" a template for modeling (typically 50 feet for rural and/or urban projects).
- In the *Template* field, select the appropriate, predefined template (typically the same for Left and for Right) from the dropdown list.
- Set the *Catch Point* field to Decision Table.
- In the *Name* field select the appropriate, predefined decision table from the dropdown list.
- <Apply>
- Repeat this process as necessary for each variation to the desired template use and/or the desired decision table use.
- <Close> when the Roadway Entry process is complete.

**When the creation of the Roadway is complete, save the Roadway Library.**

## Superelevation

Superelevation is stored with the horizontal alignment in the geometry project.

- Set the appropriate Horizontal and Vertical Alignments active.
- **File > New**
- Select the *Geometry* tab and set *Type* to Superelevation.
- Enter the appropriate Superelevation name and enter pertinent information in the *Description* field.
- <Apply> and <Close>

Define Superelevation on the Template

The pivot and range points on the templates from the sddot.tml template library have been preset, however if changes need to be made refer to the following:

- **Modeler > Define Typical Sections**
- Select the *Templates* tab
- Select the template and <Edit>
- Select the *Superelevation* tab
- Set the *Zone* to either Left or Right Backbone
- <Set Range>
- Assign the Range Points to the template (outside edges of backbone)
- <Set Crown>
- <Set Pivot> under *Superelevation One* to the center of the roadway
- <Set Pivot> under *Superelevation Two* is to be used when modeling a divided highway with 2 roadways.

Toggle "Do Not Superelevate Segment" if a selected template segment (i.e. a vertical segment transitioning from 4' of undercut to 3' of undercut) is not to be superelevated.

Compute Superelevation Rates.

- **Modeler > Superelevation > Rate Calculator**
- Set *Method* to Table Method.
- Open the *Rate Table* that corresponds with the design speed as defined in the scope document from u:\rd\Bentley\Civil\data\English.
- Set the *Column from Rate Table* to the appropriate lane configuration
- Set the *Preferred Maximum Rate* to 0.06% (16.6667:1).
- Select the curves in the alignment that have the design speed set in the Rate Table or all if applicable (<All>).
- <Compute>

If different design speeds are required when applying superelevation, then change the Rate Table to the desired design speed and select only the curves applicable to that speed <Compute>. Continue for other speeds and curves as needed.

Review / Edit Superelevation Rates.

- <Report…>
- Verify the superelevation calculations.

The rates computed in this command become the input for *Build Transitions*.

Build Transitions

- **Modeler > Superelevation > Build Transitions**
- Select the *Controls* tab
  - Toggle *Percent Total Transition on Tangent* and set to 80.00.
  - *Tangent Length* and *Runoff Length* are project specific (See Chapter 5 – Horizontal Alignment for information on separation distances between curves).

Tangent length is the minimum length such that when the tangent distance between reverse curves is less than this value, InRoads will not attempt to achieve normal crown between curves. Instead, the road will transition directly from one full superelevation to the next.

Runoff length is the minimum superelevation runoff length (L) for transitions.

- Select the *Main* tab
  - Set *Horizontal Alignment, Vertical Alignment, Superelevation* and *Roadway Definition* to the appropriate settings.
  - Set *Show* to Superelevation Finished Grade.
  - <Compute>
  - Utilize <Report…> and the information displayed in the *Review/Edit* window to review the transitions. The information can also be saved to a file from the <Report…> dialog box.
  - <Add…>, <Edit…> or <Delete> the transitions at this point.

**SAVE !! When the information is worth saving – Save It.**

If superelevation has been applied and the horizontal alignment has then been modified or restationed such that the ending station is larger than the original ending station, superelevation will have to be reapplied to the superelevation transitions. If the superelevation is not reapplied the cross sections at the end of the alignment will not be viewable. After recomputing the superelevation transitions, remodel the entire alignment.

## Roadway Modeler

Open the original ground surface (PCN#org.dtm), the geometry project (ePCN#.alg), the template library (PCN#.tml) and the roadway library (PCN#.rwl). These files can be opened automatically with the creation of an InRoads project file (PCN#.rwk). See Appendix A for instructions on creating the InRoads Project file.

Verify that the *Station Lock* is toggled on.

- **Modeler > Roadway Modeler**
- Set *Horizontal Alignment, Vertical Alignment* and *Superelevation* to the appropriate settings.
- Select the appropriate *Roadway Definition*.
- Select the appropriate *Original Surface.*
- <Apply>

**When the roadway modeling is complete, save the newly created surfaces.**

Under the *Advanced* tab, the settings have been preset for modeling a typical roadway project. These settings may be changed to meet project specific criteria.

When modeling a roadway or other alignment that does not have superelevation, the user may wish to toggle *Superelevation / Apply to* off. This will eliminate the need to create a *Superelevation* geometry type for that particular alignment.

Toggling on *Transition Control Lines / Display* may help the user to determine problem areas in the decision tables or templates. Keep in mind that the transition control lines will be graphically recreated with each application of Roadway Modeler. In other words if the Roadway Modeler command is executed three times with the *Transition Control Lines / Display* toggled on, three sets of transition control lines will be created in the dgn, unless the first two sets have been deleted or undone.

## Review Surfaces

Open the appropriate surface (*.dtm) files. (The surface files can also be included to open automatically with the Project (*.rwk) file.

**Do not triangulate surfaces. This has already been done. Use the View Triangles command to view them.**

If you need to save the display of triangles or contours, create a separate graphics file (cPCEM.dgn), which can then be attached as a reference file as needed.

- **Surface > View Surface > Contours**
- Select the appropriate *Surface*.
- For large projects, establish a fence and set *Fence Mode* to the appropriate setting. The fence must be established prior to executing the View Surface command.
- Set *Interval* and *Minors per Major* to the desired settings.
- Edit the <Preferences…> and *Symbology* as needed.
- <Apply>

And/or

**Surface > View Surface > Triangles**
Select the appropriate *Surface, Fence Mode, Symbology* and <Preferences…>.

If a surface is viewed temporarily in a file, delete contours or triangles when done and compress the graphics file.

The fence mode is useful for large projects because it can limit the contours or triangles to an area inside the fence.

## Create Cross Sections

Cross sections will be drawn in a separate graphics file (xPCEM.dgn).

Prior to creating or displaying cross sections, the correct surface symbology should be assigned to each surface. Assigning symbology to the surfaces can be accomplished at any time, however it will prove more convenient to assign the symobology before executing the Create Cross Section command.

- **Surface > Surface Properties…**
- Select the *Advanced* tab.
  - Select the appropriate *Surface.*
  - Select the corresponding *Cross Sections Symbology.*
  - Select the corresponding *Profiles Symbology* (this will be useful when creating approach pipe sections).
- <Apply>

Two separate cross section sets will be created for two separate purposes:

1. The Plans Cross Section Set – This set will be created for plotting and will include annotation and a plans border. The Undercut surface will not be displayed in this set of cross sections unless requested by the Area Office.
2. The Volume Cross Section Set – This set must show the entire range of cross sections and all surfaces necessary for volume computations. The actual earthwork volumes are calculated from the elements created for each surface in the graphics file.

Open the original ground surface (PCEMorg.dtm), the geometry (ePCEM.alg) and the appropriate design surfaces (subgrade, undercut, etc.)

- **Evaluation > Cross Section > Create Cross Section…**
- <Preferences…>

  - Plans Cross Section Set – Set the appropriate preferences to 10_20_portrait, 20_40_portrait, 10_20_landscape or 20_40_landscape.
  - Volume Cross Sections Set – Set the preference to *volumes*
- Select the *Main* tab.
  - *Set Name* will default to the horizontal alignment name, and will change by an increment of 1 (mainline_1, mainline_2, etc.) for each subsequent cross section set created. Modify this entry to (volume) for the Volume Cross Section Set.
  - Toggle the *Source* to *Alignment* and select the appropriate horizontal alignment.
  - The *Offsets* will be preset according to the preference selected, however they may be adjusted as necessary (the sum of the modified left and right offsets cannot exceed the sum of the preset offsets) to properly display all cross sections.
  - Select the appropriate *Symbology* surfaces to be displayed on the cross sections. The order in which the surfaces are loaded no longer impacts which symbology is assigned to the surfaces. Surface symbology is assigned as previously outlined.

When the Undercut surface is selected, it is drawn on every cross section, both cuts and fills. However, when volumes are run, MDC is only calculated when the undercut surface is actually in cut.

- Select the *Features* tab.
  - Toggle off *Include Features.*

See Appendix A for a thorough discussion on Features, and their uses.

- Select the *Controls* tab and modify the *Limits* settings as needed. (The *Station* limits will be ignored if entries are created under the *Custom* tab). Verify that *Window Clearance / Apply* has been toggled on, and that the *Top:* and *Bottom:* variables are each set to 10.00.
- Select the *Spacing* tab. This step is required to compensate for an out of range error that occurs with the InRoads Civil version 8.04 (Service Pack 1) package. No settings need to be modified. The tab simply has to be selected.

See Cross Section Reports later in this document for details on creating a cross section report during the Create Cross Sections process.

- Return to the *Main* tab.
- <Apply> and data point a position in the graphics file.

**The cross section *Interval* must be at the same interval (or multiple of interval) that the template was dropped. If it is not, the design surface may not tie to the original ground on the cross sections. The cross section Interval shall be set to 100.00 feet for rural projects and to 50.00 feet for urban projects or mountainous terrain.**

## Cross-Section Viewer

- **Evaluation > Cross Section > Cross Section Viewer**
- Select the appropriate *Cross Section Set.* A fence with the active MicroStation properties will display around the selected cross section set.
- Modify the *Zoom Factor* as needed.
- <Run> and data the appropriate view (*ESC* to stop the display while running).

## Cross Section Reports

- **Create Cross Section > Report** tab
- Toggle on *Generate Binary Report.*
- Enter crosssub.bin in the *File Name:* field and <Browse…> to verify the file will be saved to the project folder.
- Set *Report:* to All Points.

Following the Create Cross Section process:

- **Tools > Reports >** *General* tab
- *Report Library* – select the SD DOT cross section database from C:\Program Files\Bentley\Civil\data\xsec.dba.
- *Report*– select either X_SECT_Multi.
- *Binary File* – select crosssub.bin from the project folder.
- Toggle on *Text to Screen.*
- <Apply>

From the *Results* dialog box the cross section information can be saved as crosssub.rpt to the project directory.

## Update Cross Sections

This command allows the user to update the surface graphical elements following the execution of the Roadway Modeler command, without having to recreate cross sections.

**Features must first be removed from the cross section set before surfaces are refreshed. To remove features, follow the steps listed below, selecting *Display Off* instead of *Refresh* and the appropriate features listed in the *Object / Feature* window in conjunction with the appropriate surface.**

- **Evaluation > Cross Sections > Update Cross Section**
- Select the appropriate *Cross Section Set.*
- Toggle *Refresh* to update the surface graphical elements.
- Set *Limits / Station Range* as needed.
- Select the appropriate surfaces under *Object / Surface*.
- <Apply>

This command is also used to toggle on or off features for annotating the cross sections.

## Annotating Cross Sections

- **Evaluation > Cross Section > Update Cross Section**
- Select the appropriate *Cross Section Set.*
- Toggle *Mode: / Display On*
- Toggle *Object / Feature* and select the appropriate *Surface.*
- <All> to select all of the features.
- <Apply> and <Close>

- **Evaluation > Cross Section > Annotate Cross Section**
- Select the appropriate *Cross Section Set.*

- <Preferences…> and load the appropriate preference.
- Select the Feature(s) corresponding with the point on the cross section to be annotated.
- <Apply>

Exterior Boundary, Left Subgrade and Right Subgrade are annotated using the 10_20_scale or 20_40_scale preferences. Special Ditches are annotated using the Left Special Ditch and Right Special Ditch preferences. Note that the Special Ditch preferences are configured for the 20_40 scale cross section sets.

Once the cross section set has been annotated return to the Update Cross Section command, Toggle *Mode: / Display Off* and follow the remainder of the steps for displaying features outlined above to "turn off" the features. This will remove the feature graphical elements from the dgn.

## Plotting Cross Sections

Batch plotting of the individual plan sheets contained within the cross section set can be achieved by using the Interplot Organizer software.

- Start the Interplot Organizer software from the shortcut icon on the Windows desktop or from the programs menu found by depressing <Start>.
- **File > Create Plots…** or the Create Plots icon located on the toolbar.
- <Add…>
- Select the xPCN#.dgn file from the project folder.
- <Browse…> and select Xsect.set from C:\dot\rd\settings.
- <OK>
- **File > Print…** or the Print icon located on the toolbar.
- Select the appropriate printer.
- Toggle *Print Range / All* or *Print Range / Selection* as is appropriate.
- <OK>
- Save the Interplot Organizer Plot Set to the project folder for later use.

## Compute Cut and Fill Volumes using End Area Volumes

Open the cross section file (xPCN#.dgn), the appropriate geometry project (*.alg) file and surface (*.dtm) files.

- **Evaluation > Volumes> End-Area Volumes**
- Select the appropriate *Cross Section Set.*
- Select the *Original Surface* (PCN#org).
- Select the appropriate subgrade *Design Surface.*
- *Subgrade Surface* must be toggled off.
- Set *Units* to Cubic Yards.
- Toggle *Output / Binary:* and set the file name to endvol.bin, pathed to the project folder.
- Toggle *Output / Mass-Haul Data File:* and set the file name to haul.mhd, pathed to the project folder.
- Toggle *Method / Correct for Curvature* on.
- Enter in the appropriate value for *Volume Factors / Fill.*
- Select the *Advanced* tab.
    - Toggle *Moisture Density Control (MDC) / Use* on and set the *MDC Surface:* to the appropriate undercut surface.
- Select the *Adjustments* tab.

- *Adjust For / Added Quantities:* This option must be selected to enter additional cut and fill quantities. Quantities such as borrow, muck and unstable, salvage and entrance volumes are entered here. Shrinkage values must also be entered with each added fill quantity entry. *Added Quantities* are project specific.
- *Adjust For / Volume Exceptions:* This option allows the user to select a portion of the model, which will not be included in the volume quantity (grading exception). When using this option for excluding bridges, the start and stop stationing must be the toe of the fill slopes. *Volume Exceptions* are project specific.
- *Adjust For / Station Cut/Fill Factors:* This option allows the user to input shrinkage or swell factors and overrides the default cut and fill factors entered under the *Main* tab. Toggle *Use Station Cut/Fill* Factors on. *Station Cut/Fill Factors* are project specific.

- Select the *Annotation* tab.
  - Toggle on the appropriate *Symbology* options as necessary to annotate the volume cross section set. This step is a user preference and is not required for calculating volumes.

To save the values entered under the *Adjustments* tab enter Added Quantities, pathed to the project folder, in the *Control File / File Name:* field. This will create a (.txt) file. This file does not save the toggled features, so check to make sure that everything is toggled on all of the tabs.

Added Quantities can also be edited/added by opening the mainline.txt file mentioned above in a text editing software program such as Notepad or Wordpad. Extreme caution should be taken when working with the Added Quantities in this manner, as the entries need to be in a specific order and numbered in a specific manner.

## Volume Reports

- **Tools > Reports**
- *Report Library* – select the SD DOT volumes database from C:\Program Files\Bentley\Civil\data\sd-vol.dba.
- Set *Report:* to *EWORK-E(SDDOT).*
- In the *Input / Binary File:* field <Browse> to select endvol.bin from the project folder.
- Toggle *Output / Text to Screen* on.
- <Apply>
- <Save As…> to save this report results to the project directory as u:\rd\prj\COUN\PCN#\endvol.txt. From this file a hard copy of the earthwork report can be created.

## Mass Haul Diagram

Open the geometry project (ePCN#.alg) from the project folder, and set the appropriate horizontal alignment active.

- **Evaluation > Volumes> Mass-Haul Diagram**
- Select the *Main* tab
- Set the *Mass-Haul Data File:* field to haul.mhd, pathed to the project directory.
- Toggle *Direction / Left to Right* on.
- *Exaggeration / Horizontal:* should be set to 0.50 and *Exaggeration / Vertical:* should be set to 0.02.
- <Apply >

# Pipe Cross Sections

### Mainline Pipe Cross Sections

Open the geometry project (ePCEM.alg) from the project folder, and set the appropriate horizontal alignment active.

- Place horizontal event points at all proposed mainline pipe locations.
  - **Geometry > Horizontal Curve Set > Events…**
  - Set *Define By:* to *Single Station.*
  - Toggle *Add As / Station and Offset*.
  - Enter the pipe station in the *Locate By / Station* field, leaving the *Offset* field set to 0.00.
  - <Apply>
  - Repeat until all pipe locations have been entered.
  - <Close>
- **Modeler > Roadway Modeler…**
- Create a new Microstation 3D graphics file, saving it as pPCEM.dgn in the project folder.
- **Evaluation>Cross Section>Create Cross Section**
  - Set the *Preference* to 10_20_Landscape.

Create a separate pipe cross section set for the 20_40_Landscape preference as needed.

- Select the *Features* tab.
  - Toggle *Include Features:* off.
- Select the *Controls* tab.
  - Toggle *Window Clearance / Apply* on and set the *Top:* and *Bottom:* clearances to 20.00.
- Select the *Custom* tab.
  - Select *Perpendicular* or *Skewed* as the *Type*.
  - Enter the pipe *Station:* and *Skew Angle:* as appropriate as well as the *Left Offset:* and *Right Offset:* corresponding with the previously selected preference.

The *Skew Angle:* is positive for Left Hand Forward and negative for Right Hand Forward.

  - <Add>
  - Repeat until all pipe sections have been entered.
  - Save the custom pipe section set as pipe10_20.xsc (or pipe20_40), in the project folder**.**
- <Apply>

The custom pipe section set can be recreated at any time, by loading the .xsc file from the project folder (*Control File / File Name)*.

Following the creation of the pipe cross sections, load the XPIPE MDL application to draw each individual pipe. (MicroStation Utilities > MDL Applications).

### Approach Pipe Cross Sections

- Open pPCEM.dgn.
- Load the XSECT cell library and place the cell ENTL or ENTP. If you are using the ENTL cell, the station range is 320 feet. If you are using the ENTP cell, the station range is 200 feet.

- **Evaluation > Profile > Create Profile…**
- Set *Preferences* to *appr_pipe.*
- Select the *Main* tab.
    - Toggle the *Symbology* surface *Subgrade* on (*PCEMorg* if the approach pipe section is in a fill section).
- Select the *Controls* tab.
    - Toggle *Limits* / *Station* on and select a start station that is ½ the cross section limit ahead and a stop station that is ½ the cross section limit back of the entrance location.
    - Toggle *Direction* / *Left to Right* on for entrances on the left and *Direction* / *Right to Left* for entrances on the right.
- Select the *Offsets* tab.
    - Set the appropriate *Surface.*
- Toggle *Symbology* Offset 1 on.
- <Surface Properties…>
    - Select the *Advanced* tab.
    - Set the appropriate *Surface*.
    - Set the appropriate *Profiles* / *Symbology* to Subgrade.
    - Set the pipe *Offset* / *Distance* from mainline (-) for left and (+) for right.
    - Set the *Offset* / *Symbology* to Appr_Pipe.
    - <Apply>
- <Apply>
- Data a green point on the left side of ENTL or ENTP for an approach pipe left of mainline or data a green point on the right side of ENTL or ENTP for an approach pipe right of mainline.

Following the creation of the pipe cross sections, load the XPIPE MDL application to draw each individual pipe. (MicroStation Utilities > MDL Applications).

# Appendix A

## *Developing an InRoads Project*

An InRoads Project (*.rwk) file can be created, that can be opened at the beginning of each InRoads session. This file gives the user the ability to open each desirable component of the modeling process without having to open each component individually.

- Open the Surface files (*.dtm) for the project, beginning with PCEMorg.dtm (from the REGION project folder), and proceeding with subgrade.dtm and then undercut.dtm files.
- Open the Geometry project file (ePCEM.alg).
- Open the Typical Section Libraries file (PCEM.tml).
- Open the Roadway Libraries file (PCEM.rwl).
- Open, if applicable, the Drainage file (PCEM.sdb).
- InRoads **File > Save As**
- <Options…>
  - Select the *Surfaces* tab.
    - ✓ Toggle *Add* for the PCEMorg surface. (Road Design does not have access rights to the Region project folders, thus *Update* must not be toggled on for this surface, as Update also saves any modifications to the component file, when the InRoads session is ended or the Project file is saved).
    - ✓ Toggle *Add* and *Update* for the remainder of the surfaces.
  - Select the *Geometry Project* tab.
    - ✓ Toggle *Add* and *Update* for the Geometry Project.
  - Select the *Typical Section Library* tab.
    - ✓ Toggle *Add* and *Update* for the Typical Section Library.
  - Select the *Roadway Library* tab.
    - ✓ Toggle *Add* and *Update* for the Roadway Library.
  - Enter PCEM in the *File Name:* field.
- <OK>

The InRoads Project file can also be created by copying u:\rd\Bentley\Civil\data\English\rd.rwk to the appropriate project file. Once the copy has been completed utilize a text editor program to edit the path for each respective component of the file.

## *Feature Codes*

The following discussion focuses on the use of Feature Codes with cross sections. Feature Codes can also be used with profiles, however Road Design does not utilize Feature Codes with profiles at this time.

InRoads Feature Codes are a valuable tool for the user. However, as much as they are valuable, they are detrimental to the cross section design file. Each displayed feature code consists of two MicroStation line elements. Consequently, the more feature codes that are displayed, the larger the cross section design file becomes, and the longer it takes InRoads to create the cross section set. The user should take every opportunity to remove feature codes from the cross section design file when they are not needed.

Feature codes must also be removed from the cross section set prior to remodeling the roadway. Once Roadway Modeler has been completed, the existing feature codes are no longer valid, and InRoads will not recognize them.

The two most common uses for Feature Codes is for annotating cross sections and for attaching cells (such as the curb & gutter cells) to cross sections.

As directed earlier in the Create Cross Sections section of this document, toggle *Include Features* off when creating a cross section set. If it is desirable to include the features, steps can be taken to reduce the number of features displayed by the use of filters.

- Toggle on the **Filters Lock**.
- **Evaluation > Cross Sections > Create Cross Section**
- Select the *Features* tab.
- <Filter…>
- Set the *Filter Name* to *No Existing Ground*. This will remove the features associated with the existing ground from the list of features to be displayed.
- <OK>
- Proceed with the remainder of the create cross section process.

To attach cells to features

- **Tools > Symbology Manager**
- Select the appropriate feature code from the list displayed in the dialog box.
- <Edit>
- Select *Cross Section Point*.
- <Edit>
- Set *Display As* to *Cell*.
- Set *Level* to the appropriate level.
- Set *Cell Name* to the appropriate cell to be associated with the feature code.

The proper cell library must be attached to the MicroStation design file prior to executing the Symbology Manager command.

- Verify that *X Scale, Y Scale* and *Z Scale* are properly set. Some cells require that the *Y Scale* be set to 0.50, while the other scales remain at 1.00.
- <OK>
- <Apply> and <Close>
- <Close>

These cells will now be displayed with the corresponding feature code, when the feature codes are displayed in the cross section set.

Cells associated with feature codes will be removed or added from the display in the cross section set, as their parent feature code is removed or added.

## ***Independent Transition Control***

Independent control allows for the flexibility to model a roadway with varying geometry, without creating additional templates. In those locations where a turn lane, mailbox turnout, historical marker turnout or guardrail widening are called for, Independent Transition Control should be considered for the varying geometry.

The first step in setting up independent control is to modify the templates to allow the appropriate segments to vary to meet the needs of the required geometry.

- Open the template library (PCEM.tml) from the project folder.
- Follow the steps defined earlier in this document for a template editing session.
- Set the *Edit Mode* to *Global* and select the appropriate segment to be modified.
- Set the *Fixity* to *Variable Width*. Exercise caution when using the *Variable Slope* and *Variable Slope and Width* options. If the *Fixity* is set to *Fixed*, Horizontal transition control can still be applied, however the transition control will only be able to lengthen the segment. InRoads will not shorten a defined segment if its *Fixity* is set to *Fixed.*
- <Update>
- Repeat the process to modify the Fixity on other segments as needed.

InRoads will automatically adjust the location of those segments located outside a segment that varies with independent transition control.

**SAVE !! When the information is worth saving – Save It.**

- Set the appropriate horizontal and vertical alignments active.
- **Modeler>Define Roadway**
- Select the appropriate roadway definition and <Edit>.
- Select the *Horizontal and Vertical Controls* tab
- Select the appropriate *Transition Control Name* (the segment that is to vary to meet the needs of the geometry).
- Toggle *Use Horizontal Control* on.
- Toggle *User Vertical Control* off.
- Select the appropriate *Horizontal Alignment* (this will be other than the active horizontal alignment if a separate alignment has been created for the sole purpose of utilizing Horizontal Control).
- Input the appropriate *Station Limits / Start* and *Stop* stationing.
- Input the appropriate *Offsets / Start* and *Stop* distance from the horizontal alignment. Keep in mind that this distance is to be the true offset from the alignment, not the distance the Transition Control is to vary. Also, if a separate horizontal alignment is being used for Horizontal Control, the offset distances will be 0.00'.
- <New>
- Repeat the process as necessary for each additional entry.

**SAVE !! When the information is worth saving – Save It.**

# Appendix B - InRoads Storm & Sanitary Workflow, v8.5

## Reference Documents

The following documents, provided by Intergraph, are useful tools for using the software.

u:\doc\ SelectCAD\Ver8.02_doc \Tutorials\
**Working With Storm & Sanitary SelectCAD.pdf** – contains information on getting started, common design workflows, and a tutorial and sample data set that provides step-by-step instructions for learning how to create a storm drainage system. **Chapter 5 of the above document provides a good workflow, which was followed to create this document.**

u:\doc\SelectCAD\Ver8.02_doc\Reference\
**Storm & Sanitary Design Procedures.pdf** - is a technical reference document that details the industry standards, computations, and methodologies that the software utilizes for hydraulic/hydrology calculations.

u:\doc\InRoads\Ver8.04.00.00\Product_suite_ref_guide\
**DAA022620v2.pdf** – Provides explainations of each command and input in InRoads. *Drainage* begins on page 1259, and *Tools>Drainage Options* begins on page 2324 (DAA022620v3.pdf)

## Starting Inroads Storm & Sanitary

Need to contact the HELP desk to have BIT install the software on your computer.

Loading Inroads Storm & Sanitary after software installed
Inroads toolbar: **Tools > Application Add-Ins >** check **InRoads Storm & Sanitary** box**, OK**
This will add **Drainage** to the Inroads toolbar and also a Drainage under Tools.
Save settings to have this automatically load for this project, but should toggle off when not in use since it checks out a license every time it is loaded.

Create Drainage (.sdb) file when first beginning
**File > New > Drainage Tab >**
Name: **PCEM** (will automatically add the extension .sdb)

Save Drainage (.sdb) file when needed

Open Drainage (.sdb) file
**File > Open >**
set Files of type: to Drainage (*.sdb), then select **PCEM.sdb**
Can also open the PCEM.sdb by adding to the PCEM.rwk file and opening only that.

Be sure to have the project geometry and surface files loaded for InRoads Storm & Sanitary to reference from.

**HINT**: Be sure you have run **dot.bat,** so your computer has the latest SDDOT standard drainage files loaded. Recommend checking that the correct i_structure.dat file is loaded everytime InRoads Storm & Sanitary is used by looking under **Tools > Drainage > Options > Inlet Tab**, and that Type B is an option under class: when the type: is set to combination.

Recommend creating **sPCEM.dgn** as a 3D graphics file in your PROJECT folder using microstation.
This file will be used to design the storm sewer system.

## Storm Sewer Design Procedure

### 1. Set preferences and symbology for network structures.

Most of the following are typical project settings that should automatically be loaded from the default preference, which is in civil.ini and loaded when dot.bat is run. These settings should be double checked before placing structures and computing to meet project specific needs.

**Tools > Drainage > Options**

**General Tab:**

Drainage Structures File: i_structures.dat (the program seeks this file from C:\program files\bentley\civil\data\imperial\ which is loaded when dot.bat is run)
Area Units: ac
Discharge Units: cfs
Status: Fixed for inlets, Resize for pipe (See following hint)

**HINT:** Under the General Tab, Status: **Fixed** needs to be toggled while placing inlets, but should be set back to **Resize** when placing pipes to have the software resize pipes during the analysis. If any inlet in the network is toggled to resize, the computer will lock up when analyzing the network. It may be easiest to leave as Resize and edit the inlets after they are placed. To change this after an inlet is placed, go to **Drainage > Edit/Review, select the inlet, toggle on Fixed at the bottom of the Inlet Tab, Apply**.

**Structure IDs Tab:**

Toggle on Use Prefixes

| Prefix: | | |
|---|---|---|
| | Pipe: | P |
| | Channel: | CH |
| | Culvert: | CV |
| | Manhole: | MH |
| | Inlet: | IN |
| | Pump: | PM |
| | Area: | A |
| | Zone: | Z |
| | Utility: | U |

**Gutter Section Tab:**

| | |
|---|---|
| Type: | Composite |
| Gutter Width: | 2.00 ft |
| Side Slope: | 20.00:1 (.05 ft/ft) |
| Longitudinal Slope: | Compute from DTM |
| Transverse Slope: | Compute from DTM (usually 50:1 or .02) |
| Roughness: | 0.013 |
| Maximum Spread | 8.00 ft |

**Design Tab:**

Design Equation: Manning

Structure: Set parameters for each type of structure to be placed as follows:

**Pipe:**

| | |
|---|---|
| Minimum Height: | 18 in. |
| Maximum Height: | 200 in. |
| Minimum Velocity: | 2 ft/s |
| Maximum Velocity: | 40 ft/s |
| Depth to Height Ratios: | (Used only when creating a Sanitary flow system) |

| For Height up to: (in.) | Use Ratio of: |
|---|---|
| 15 | 0.50 |
| 150 | 0.75 |
| 999 | 0.93 |

**Inlet:** The following parameters are applied to all the appropriate inlets during the design analysis of the network, as performed in step 4, and cannot be changed for different inlets during a single analysis. The following values listed are for most typical uses of the standard inlet types, except a Type B-in sump. To check a Type B-in sump, input the values listed farther below, and run a separate design analysis just for the non-typical Type B-in sump inlet results.

**Typical Inlet Uses (Type B-on grade, all Type S & all Type C):**

| | | |
|---|---|---|
| Curb Height: | 6.0 in | For typical curb & gutter sections like Type F & B. Does not include inlet depression. |
| Curb Opening Height: | 6.1 in | The value shown is for a Type S inlet. This parameter is only used for curb opening (Type S) and Combination (Type B) inlets in sump This value should be the total opening height with depression below the gutter subtracted out (6.1"-3.4"=2.7"). However, spread does not appear to compute correctly when depression is provided, so input a depression of 0" for Type S-in sump under the inlet tab. Hopefully this gets corrected in next version. |
| Curb Length: | 2.96 ft | Only used for Combination Inlets (Type B). Normally this length should be the same as the grate length, so the curb opening capacity is neglected, as is standard design according to HEC22. If the curb length is longer than the grate, then the extra length is assumed upstream of the grate, and flow is taken into only the extra curb opening length and then the grate. |
| Orifice Depth: | 0.52 ft | This value is for a Type C inlet, Neenah R-3457-C2 B, and is the head where flow changes from weir to transitional flow on the Neenah website (www.nfco.com). This parameter is used for all inlets except curb openings (Type S). Determines whether a grate is in weir or orifice flow in a sump location. This is also depth where curb opening orifice capacity is added with the grate's for a combination inlet (Type B). This depth is compared to the gutter depth plus depression, however, depression cannot be input for grates in sump locations in version 8.5. |

**Type B-in Sump:** (will need to run a separate analysis with these parameters)

| | | |
|---|---|---|
| Curb Opening Height: | 4.5 in | See discussion above. This value should be 4.5"-1.5"=3". However, depression (1.5") cannot be input for combination inlets (Type B) in sump, so 0" depression is assumed and the total opening of 4.5" is input. |
| Orifice Depth: | 0.49 ft | From Neenah R-3067-V V. See discussion above. |

## 2. Lay out network structures

**Drainage > Lay Out**

**HINTS**: Place junction structures first such as inlets and manholes, then connect with pipes.

When placing inlets be sure the subgrade surface is active, so the gutter section uses the correct transverse (typically 2%) and longitudinal slope, if computing slope from DTM. These slopes are important when computing flows and spread.

When placing Type S inlets, be aware that the offset to the center of the inlet is behind the curb & gutter, so the subgrade surface is much higher than that below the roadway surfacing. This may cause the program to set the initial pipe elevation too high at this inlet when it maintains the 1 ft. minimum subgrade cover over the pipe. You will need to adjust the pipe elevation.

**Inlet Tab:**

| | |
|---|---|
| Options… Button at bottom: | Select and set the parameters needed as shown below under **Options…** before placing each inlet. |
| Inlet ID: | Set the number as needed or use automatic number. |
| Location: | Set as needed |
| Angle: | Will come in as set in **Drainage Options, Inlet, Orientation** |
| Long. Slope: | Will come in as set in **Drainage Options, Gutter Section** |
| Transverse Slope: | Will come in as set in **Drainage Options, Gutter Section** |
| Connecting Pipes: | Add connecting pipes if needed, not typically done. |

**Options…** Button at bottom: Takes you to **Drainage Options, Inlet**:
(Can also get to this by Tools > Drainage > Options, Inlet Tab)
Input the type of inlet you want to place with the different options and parameters listed as follow:

**Type B Inlet (On-Grade):**

| | |
|---|---|
| Type: | Combination |
| Class: | Type B |
| Grate Size (LxW): | 2.96 x 1.48 (Choose the one that gives the vault size needed, as will show below, to fit pipe in) |
| Vault Shape: | Box |
| Vault Size (LxWxT): | 3 x 2 x 6 ftxftxin (Change to size needed to fit pipe) |
| Placement Offset: | 0 |
| Location: | **On-Grade** |
| Connection Point: | Inside (pipe connects to center of inside wall) |
| Depression: | --- in (This parameter cannot be input) |
| Drop Across: | 0.0 ft |
| Maximum Depth: | 10 ft (According to standard plate) |
| Depth below Invert: | 0.0 ft |
| Grate Cover: | 45 % (Percent flow is restricted by bars of grate. Only used for orifice flow in sump) |
| Clogging: | 0 % (Assume no clogging of inlets) |
| Intercept Flow: | 60 % (Ignored since inlet size is set to fixed) |
| Orientation: | Parallel to Alignment (Change as needed) |

**Type B Inlet (in Sump):** These inlets are typically not used in sump conditions, unless the gutter flow is 1.0 cfs or less.
Input the same parameters as above Type B Inlet (On-grade), except as follows:

| | |
|---|---|
| Location: | **Sump** |
| Stand Height: | 0.0 ft |

Intercept Flow cannot be input

**Type S Inlet (in Sump):**

| | | |
|---|---|---|
| Type: | Curb Opening | |
| Class: | Type S | |
| Opening Length: | 5 or 10 | (Choose the one that gives the vault size needed, as will show below, to fit pipe in. 5' lengths are generally used in place of Type B inlets when requested by the city.) |
| Vault Shape: | Box | |
| Vault Size (LxWxT): | 10 x 3 x 6 ftxftxin | (Change to size needed to fit pipe) |
| Placement Offset: | 0 | |
| Location: | **Sump** | |
| Connection Point: | Inside | (pipe connects to center of inside wall) |
| Depression: | 0.0 in | (This should be 3.4", which is the depression below the gutter bottom at the front of the inlet curb opening, to be added to the curb opening height of 2.7" for calculations. However, spread does not appear to compute correctly when depression is provided, so use 0" until fixed. |
| Drop Across: | 0.0 ft | |
| Maximum Depth: | 6.75', 8.75' or 11.75' | (Varies depending on vault size. Depths shown are from the top back of curb. From standard plates - 6.75' for 3x5 & 3x10, 8.75' for 4.5x10 & 11.75' for 6x10) |
| Stand Height: | 0.0 ft | |
| Depth below Invert: | 0.0 ft | |
| Orientation: | Parallel to Alignment | (Change as needed) |

**Type S Inlet (On-Grade):** These inlets are typically not used on-grade.
Input the same parameters as above Type S Inlet (in Sump), except as follows:

| | | |
|---|---|---|
| Location: | **On-Grade** | |
| Depression: | 1.5 in | (This is the depression below the gutter bottom just upstream of the inlet opening) |
| Stand Height is not input | | |
| Intercept Flow: | 60 % | (Ignored since inlet size is set to fixed) |

**Type C Inlet (in Sump):**

| | | |
|---|---|---|
| Type: | Median Drop | (this type uses 4 weir sides vs. 3 for grates) |
| Class: | Type C | |
| Grate Size (LxW): | 4 x 3 | (Choose the one that gives the vault size needed, as will show below, to fit pipe in) |
| Vault Shape: | Box | |
| Vault Size (LxWxT): | 4 x 3 x 6 ftxftxin | (Change to size needed to fit pipe) |
| Placement Offset: | 0 | |
| Location: | **Sump** | |
| Drop Across: | 0.0 ft | |
| Maximum Depth: | 10 ft | |
| Stand Height: | 0.0 ft | |
| Depth below Invert: | 0.0 ft | |
| Grate Cover: | 50 % | (Percent flow is restricted by bars of grate Only used for orifice flow in sump) |
| Clogging: | 0 % | (Assume no clogging of inlets) |
| Orientation: | Parallel to Alignment | (Change as needed) |

**Type C Inlet (On-Grade):** These inlets are typically not used on-grade.
Input the same parameters as above Type C Inlet (in Sump), except as follows:

| | | |
|---|---|---|
| Location: | **On-Grade** | |
| Stand Height is not input | | |
| Intercept Flow: | 60 % | (Ignored since inlet size is set to fixed) |

**Pipe Tab:**

Options… Button at bottom: Select and set the parameters needed as shown below under **Options…** before placing each pipe.

| | |
|---|---|
| Pipe ID: | Set the number as needed or use automatic number. |
| Location: | Set as needed |
| Use Soffit Elevations: | Toggle on to align inside top of pipe, resulting in abrupt drops in flowline. If not toggled, pipe flowlines will be aligned. See Design Manual, Ch.11 for best method to use. Generally, hydraulic characteristics are better when top of pipes are aligned. Aligning pipe at flowlines is typically done in flat terrain. This toggle will be used by the software to initially set the pipe elevations. When individuals adjust the pipe elevations manually, as is typically always needed, they will need to match soffits themselves, if so desired. |
| Invert In, Invert Out & Slope: | Set these 3 parameters as desired. The software analyzes the pipe at these elevations, and does not automatically adjust during the design analysis. Users will need to review the initial elevations set by the software, and adjust as needed for better flow characteristics and meet cover requirements. One parameter will need to be fixed. To fix one correctly: toggle it off so it can be edited, type in what you want to be set, then toggle it on. Edit one of the other parameters as desired, and it will automatically update the last. |

**Options…** Button at bottom: Takes you to **Drainage Options, Pipe**:
(Can also get to this by Tools > Drainage > Options, Pipe Tab)
Input the type and size of pipe you want to place with the parameters listed as follows:

| | | |
|---|---|---|
| Shape: | Circular, Box, Elliptical or Pipe-Arch | (select from available) |
| Material: | RCP, CMP or PVC | (select from available) |
| Size (_x_xT): | (select desired size from those available, listed in inches) | |
| Min. Slope: | 200:1 | |
| Max. Slope: | 5:1 | |
| Min. Cover: | 1.0 ft | |
| Max. Cover: | 20 ft | (May need to check max. cover from Ch. 11) |

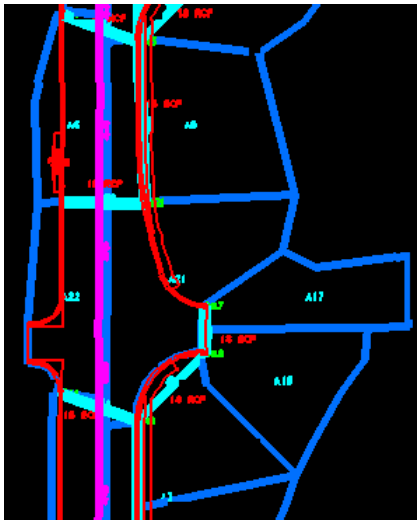
**HINT:** The network should be designed with all round pipe. If an arch pipe is placed in the network, it will change all the round pipe in the network to the same diameter as the rise of the arch pipe, or it may resize the arch pipe incorrectly. It typically resizes the arch pipe to be the maximum size available or the last one listed in the i_structures.dat file. To size an arch pipe correctly, all pipe in the network need to be designed as arch pipe. After the network is designed and pipe sizes and types determined, then edit the pipe sizes where needed, so they will be drawn correctly on the profile. Hopefully, this will not be a problem in the next version of the software.

After drainage structures are placed they can be changed by:

**Drainage > Edit/Review** , then select structure to be edited
(or can double click on the structure ID under the SelectCAD window)

## 3. Compute flow and attach drainage areas.

Create shapes of the drainage areas flowing to each inlet in the network using microstation tools such as Place Smartline.



**Tools > Drainage > Options**

**Area Tab:**

| | |
|---|---|
| Rainfall Method: | IDF File (See below) |
| Frequency: | 10 yr |
| Time of Concentration: | Kirpich: with 0.0078 as the regional constant |
| Runoff Coefficient: | Can input a value or hit the Compute… button to determine a value that will be used as the default for all the areas in the network. Typically the Runoff Coefficient will be input or computed separately for each drainage area under *Drainage>Flows>Compute Flow* and not at this location. |

**IDF File:** The appropriate idf file (rainfall intensity duration frequency data), used to compute the flows by the rational method, will need to be set in the project defaults by the following:

**File > Project Defaults**, with the cursor in **Rainfall Data**, select Browse…, go to C:\dot\rd\data\ to select the correct CITY.idf file based on the location in the state. See the Drainage Ch. 11 of the Road Design Manual for the city to use.

**HINT**: Whenever flows are to be computed, double check that the correct rainfall data file (City.idf) is selected in the project defaults.
The project default should keep the same rainfall city until it gets changed.

**NOTE:** If the designer wishes to calculate flows for a storm frequency other than what was originally specified (i.e. 100 yr storm vs. 10 yr storm) the following steps need to be followed.

1. Under the Tools > Drainage > Options > Area tab change the frequency to the desired storm.
2. Under <u>each</u> of the established drainage areas (Edit/Review Area), toggle from the Time of Concentration data box to the Intensity data box. Storm intensities will update with this step.

*Simply changing the storm frequency under the Tools > Drainage > Options > Area tab will not change the rainfall intensities for the drainage areas. Each and every established Area data set will need to be updated as listed above.*

**Drainage > Flows > Compute Flow** - Computes flow from drainage areas and inputs into the network.

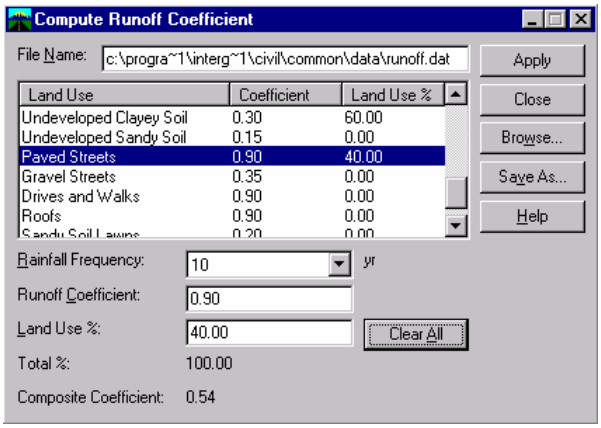
**Options… Button** can set the parameters as explained previously for *Tools>Drainage>Options.*
Recommend double checking that the correct city.idf file is used.

**Modified Rational Tab:** (Q=CIA)

| | |
|---|---|
| Area ID: | Use automatic number or input a number.<br>May want to match the Area ID to the corresponding Inlet ID. |
| Drainage Area: | Click the Locate button and select graphic shape of the drainage area as created above, or input area in acres. |
| Runoff Coefficient: | Compute based on land use values and % of use, as shown below under **Compute Runoff Coefficient**, or input a value. |
| Time of Concentration: | Toggle on and typically input the minimum 15 minutes or, **Compute Time of Concentration** as shown below for larger areas if the resulting time is greater than 15 minutes. |
| Intensity: | Don't toggle on.<br>The intensity from the specified City.idf file will be used. |
| Peak Flow: | Shows computed value based on above parameters. |
| Attach To: | Click the Locate button and select the inlet or pipe that this drainage area will flow to. |

**Compute Runoff Coefficient:**

| | |
|---|---|
| File Name: | c:\Program Files\Bentley\Civil\data\runoff.dat |
| Rainfall Frequency: | 10 yr |
| Runoff Coefficient: | Shows the coefficient of the highlighted land use.<br>Note: The coefficients change for different frequency. |
| Land Use %: | Input the percentage of each type of land use in the drainage area, until there is a total of 100% below. |
| Total %: | Automatically adds up all the Land Use % input.<br>Needs to be 100% before computes composite coeff. |
| Composite Coefficient: | Computed runoff coefficient based on land use %. |




**Compute Time of Concentration:** Use if the following Result is greater than 15 min.

| | |
|---|---|
| Method: | Kirpich |
| Kirpich: | 0.0078 |
| Length: | Input distance from most distant point in drainage area to inlet. |
| Slope: | Input average slope of the above drainage basin flow length. |

**Drainage > Flows > Inject Flow** (or under the **Drainage > Edit/Review > Flow Tab, Injected Flow:**)
Injects flow into a network where needed instead of computing flow from an area.

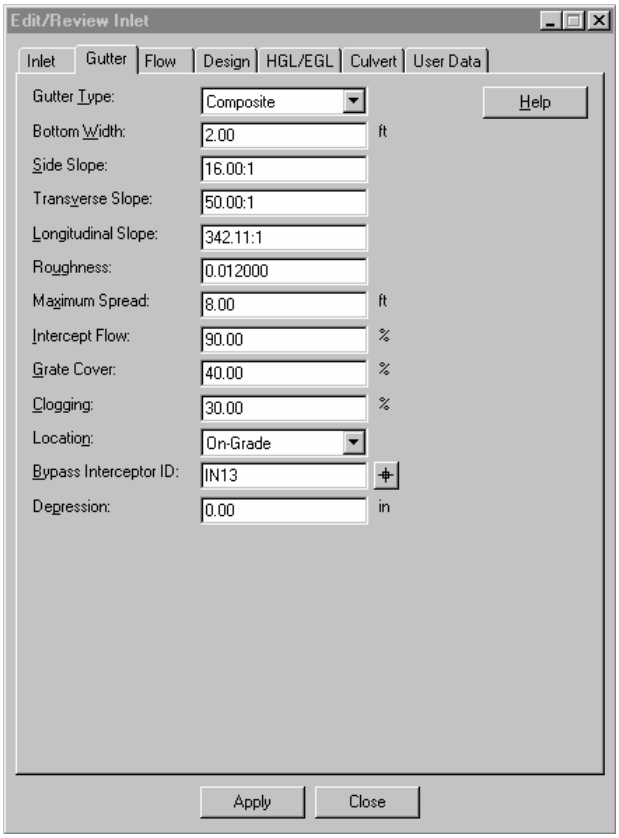
**Note:** When injecting flow into an inlet, the flow will first go into the gutter and spread will be computed from the injected flow. To inject flow directly into the inlet without first going into the gutter, then create a small pipe into the inlet and inject flow into the pipe.

**Bypass flow interceptor** needs to be set for each On-Grade inlet in the network by the following:

**Drainage > Edit/Review**, select structure to be edited,
or can type in the structure ID in the Microstation key-in window.

**Gutter Tab:**

Bypass Interceptor ID: Select structure that the bypass gutter flow will flow to.
Apply




**HINT:** Recommend double checking all the inlet, gutter and flow parameters before the next step of designing the network.

## 4. Design the network

**Drainage > Network > Design**

Toggle on "Tree Network Containing:" under Structures, and select any structure in the network to be analyzed.

Toggle on "Generate Design Log"

Note: Do not toggle "Enable Time of Concentration" when using 15 minute minimum time of concentration. 15 minutes will then be used to size all the pipe in the network. When toggling this, additional time for flow passing through the upstream pipes will be added for sizing pipes. This will result in smaller pipes. Toggle on if computing time of concentrations or using smaller minimum time of concentrations. The minimum time of concentration and this toggled item will be reviewed later after recommendations are received from the consultant for the new drainage manual.

"Capture all flow to inlet, ignoring Inlet capacity calculations":
The network should be analyzed twice, first with this toggled off and then with it on. This follows the method recommended in HEC22.

Toggled Off analysis is needed to accurately compute gutter flow spreads at the inlets. By not toggling this, the inlet gutter bypass flows will be computed and added to the downstream inlet gutters, resulting in more accurate spread computations.

Toggled On analysis is needed to compute pipe sizes. Pipes should be sized assuming all flow enters the inlets and no bypass flows are computed. This may result in slightly larger pipe at the upstream of a network.

Note: Do not toggle "Use Depth to Height Ratios". These are used for sanitary flows. This could be used for storm sewer if there was a special need to limit the depth of water in the pipes. If so, these ratios are set in the Drainage Options > Design tab.

Note: Peaking Factor Method: is not applicable, since it only pertains to sanitary systems.

Toggle on Generate HGL and EGL.

Toggle on "Use Water Depth" under Outfall Water Level, unless the outfall water level is known.

Toggle on "Greatest Flow" under Trunk Line Path

Ke for Outlet Control Pipes from Drop Manholes: 0.5

**Options…** button: to easily change design settings or double check that the settings are correct. The settings should be double checked, especially the rainfall city.idf file.

Apply

Save Results (Design Log)

**Notes**: InRoads Storm & Sanitary results do not appear to be totally accurate for inlet computations (or spread analysis), in that they do not match HEC22 results. Comparisons of Type B inlets indicate that InRoads was 6% to 7% less efficient than HEC22. Comparison of a Type S inlet on-grade with 1.5" depression was found to be 9% less efficient in InRoads than HEC22. This may be due to InRoads using slightly different exponents than listed in HEC22 for the computation of the required curb opening length (Lt). Type S inlet in sump computations are questionable, since the spread gets very small when any depression is input. Therefore, a depression of 0" for a Type S in sump is recommended. Depression should not be utilized in the computations anyway until depth of flow puts it in orifice flow, which is rare. These discrepancies will be logged with Bentley and hopefully resolved in future versions. Using InRoads appears to give slightly conservative spread rates. If more accurate spread rates are desired then consider using other HEC22 based software.

## 5. Create a profile

**Evaluation > Profile > Create Profile**

Load the **Storm** Preference

**Main Tab:**

| | |
|---|---|
| Create: | Window and Data |
| Set Name: | Input the name for the profile |
| Source: | Network |
| Symbology: | Select surface(s) you want to see in relation to the pipe and water profile. |

**Network Tab:**

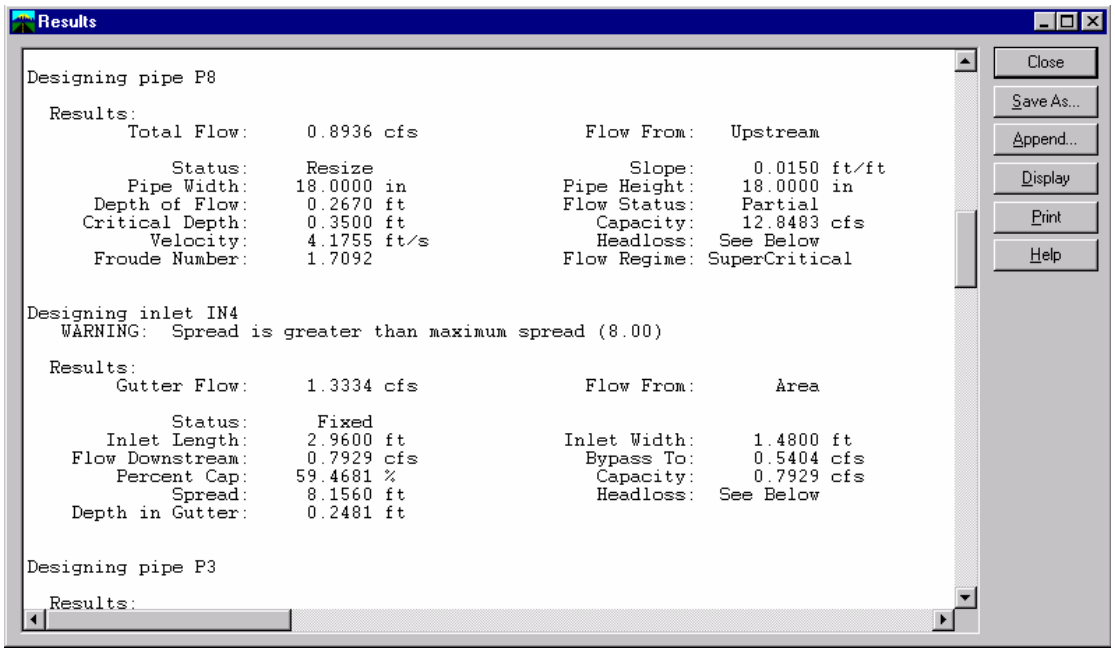
| | |
|---|---|
| Network Structures: | Enter begin and end structures of the network for the profile. |
| Symbology: | Should be set with typical settings from storm preference. |

## 6. Analyze the network

Analyze the network by reviewing the design log and network profile that were created as above.

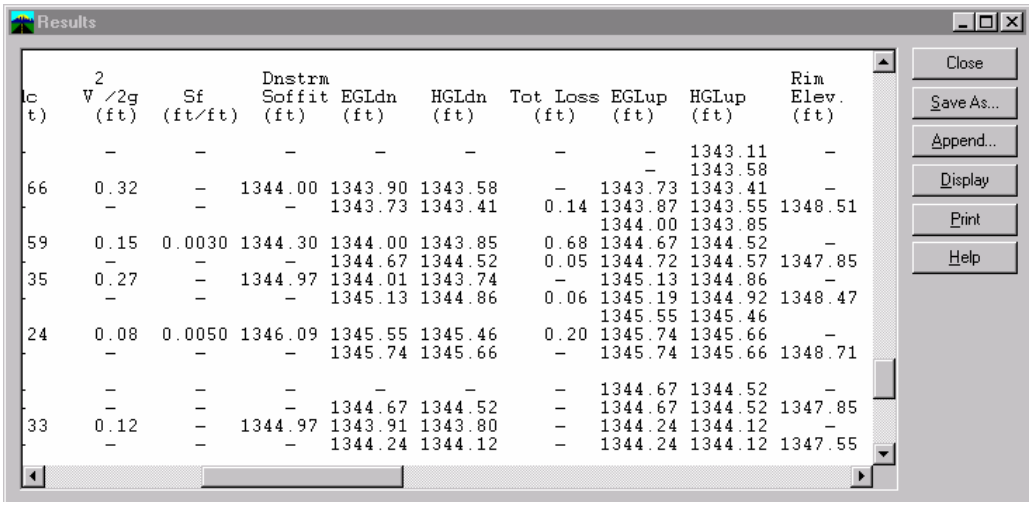
**Design Log:** The design log was created as above by toggling on Generate Design Log when designing the network under **Drainage > Network > Design**.

Review the design log for warning messages such as: spread greater than maximum spread.




```
Designing pipe P8

  Results:
        Total Flow:      0.8936 cfs              Flow From:    Upstream

            Status:      Resize                      Slope:      0.0150 ft/ft
       Pipe Width:     18.0000 in              Pipe Height:     18.0000 in
     Depth of Flow:      0.2670 ft             Flow Status:     Partial
    Critical Depth:      0.3500 ft               Capacity:     12.8483 cfs
          Velocity:      4.1755 ft/s             Headloss:   See Below
     Froude Number:      1.7092                Flow Regime: SuperCritical


Designing inlet IN4
   WARNING:  Spread is greater than maximum spread (8.00)

  Results:
        Gutter Flow:      1.3334 cfs              Flow From:        Area

            Status:       Fixed
       Inlet Length:      2.9600 ft             Inlet Width:      1.4800 ft
   Flow Downstream:      0.7929 cfs               Bypass To:      0.5404 cfs
       Percent Cap:     59.4681 %                 Capacity:      0.7929 cfs
            Spread:      8.1560 ft                 Headloss:   See Below
   Depth in Gutter:      0.2481 ft


Designing pipe P3

  Results:
```

The bottom of the design log shows HGL/EGL Computation Results. Review these and the profile to see that the HGL & EGL meet criteria discussed in the Road Design Manual, Drainage Chapter 11. Some criteria to meet include: keeping the HGL below the finished roadway surface and having correct values for the outfall.




```
      2                  Dnstrm                                                Rim
ic    V /2g     Sf       Soffit EGLdn    HGLdn   Tot Loss EGLup   HGLup      Elev.
t)     (ft)   (ft/ft)    (ft)   (ft)     (ft)      (ft)    (ft)    (ft)       (ft)

       -       -         -       -        -         -        -    1343.11       -
                                                                 -    1343.58
66     0.32      -    1344.00 1343.90  1343.58      -    1343.73  1343.41       -
       -       -         -    1343.73  1343.41    0.14   1343.87  1343.55  1348.51
                                                          1344.00  1343.85
59     0.15  0.0030   1344.30 1344.00  1343.85    0.68   1344.67  1344.52       -
       -       -         -    1344.67  1344.52    0.05   1344.72  1344.57  1347.85
35     0.27      -    1344.97 1344.01  1343.74      -    1345.13  1344.86       -
       -       -         -    1345.13  1344.86    0.06   1345.19  1344.92  1348.47
                                                          1345.55  1345.46
24     0.08  0.0050   1346.09 1345.55  1345.46    0.20   1345.74  1345.66       -
       -       -         -    1345.74  1345.66      -    1345.74  1345.66  1348.71

       -       -         -       -        -         -    1344.67  1344.52       -
       -       -         -    1344.67  1344.52      -    1344.67  1344.52  1347.85
33     0.12      -    1344.97 1343.91  1343.80      -    1344.24  1344.12       -
       -       -         -    1344.24  1344.12      -    1344.24  1344.12  1347.55
```

## 7. Redesign the network and recreate a profile if needed.

Make changes to the network to correct problems found in the design log and profile.

## 8. View Network Layout

**Drainage > View > Drainage**

Load the appropriate preference

**Design** preference – use during design. Shows inside and outside pipe walls to help determine if pipe will fit in inlet.

**Final** preference – use on plan sheets. Shows single thick line for pipe.

## 9. Annotate the network structures.

Do this for your information only during design. Not to be shown on plan sheets.

**Drainage > View > Annotate Structures**

## 10. Display Network on Final Profiles

For this step, you must receive permission from the drafter to use the dPCEM.dgn. You also need to open the dPCEM.alg in order to annotate the profiles in the d file.

**Evaluation > Profile > Create Profile**

Load the **Storm** preference.

**Main Tab:**

Create: **Data Only**
Source: **Network**

**Network Tab:**

Under **Network Structures** enter the From: and To: of the entire network that will be displayed on the profiles.

In the **Symbology** box toggle:
- Pipe Inside
- Pipe Outside
- Inlet
- Manhole
- Branch Lines

Click <**Apply**>

You will need to select each individual profile that you want to show the storm sewer on.

**Note:** More often than not, the branch lines will not show up on the profile. The branch lines that do not show up will need to be drawn separately by selecting the network structures From: and To: for the branch line only.

**Hint:** Once you have the storm sewer on each profile, and you make changes, you can use the **update drainage profile** command. This will update all the profiles at once with the changed data.

**Profile > Update Drainage Profile**

## 11. Annotating Drainage Profiles

For this step, you still need to be in the dPCEM.dgn.
In order to annotate the slope of the pipe properly you need to have the proper slope option on.

**Tools > Options**
**Units and Format Tab:**
Slope: **50%**

**Evaluation > Profile > Annotate Drainage Profile**
There will be two preferences to choose from. Load the **Lt to Rt** preference if the drainage network is flowing left to right on the profile. Load the **Rt to Lt** preference if the network is flowing right to left on the profile.

Click <**Apply**> and then select each individual profile.

**Hint:** Once all the annotation is done, you may need to clean up the profiles, because some of the annotation may be overlapping.

Now that the drainage profile is drawn and annotated you can leave the d file.

## 12. Storm Sewer on Cross Sections

**Drainage > View > Drainage**
Load the **Final** Preference
Apply

In order to view on the cross sections you need to create features out of the network. One way is to import a surface. A storm sewer surface will need to be created called **Storm.dtm**. This surface will contain the storm sewer features.

**File > Import > Surface**

**From Graphics Tab:**

| | |
|---|---|
| Surface: | **Storm** |
| Load From: | **Single Element** |
| Elevations: | **Use Element Elevations** |
| Seed Name: | **18"RCP** or whatever size you need |
| Feature Style: | Select style that matches the pipe size |
| Point Type: | **Breakline** |

Duplicate Names: Toggle on **Rename**
**Toggle on Exclude from Triangulation**

Click <**Apply>** then click on every 18" Pipe. You will need to change the Seed Name and Feature Style with each new pipe size.

**Hint:** If you make a mistake, you can delete a feature under **Surface > Edit Surface > Delete Feature**

**Evaluation > Cross Section > Create Cross Section**

Toggle on the new **Storm** surface under Symbology.

Highlight all of the new **Storm** features under the Features tab. Now when the cross sections are created they will have the proper pipe cell at the correct elevation.

**Hint:** Remember that when the drainage network is updated, that the pipe features also need to be updated. Which would mean deleting the changed feature and importing it again.

## 13. Generate reports

**Tools > Drainage > Reports**