Build Custom Parts for Use in AutoCAD® Civil 3D and InfraWorks®

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Description
Discover the new Infrastructure Parts Editor software and how it can help you build custom parts for your AutoCAD® Civil3D and InfraWorks® models. This new software, formally known on the Autodesk Labs as Project Kameleon, enables the authoring of Inlets, Manholes, Culverts, Frames, Grates, End Treatments, and more. Learn how to use the Parts Editor to use a built-in shape library of parametric shapes to create custom parts. Discover how to use Inventor® and the Inventor® Plug In, Infrastructure Part Shape Utilities, to extend the built-in part library with custom parametric models, and lastly add part sizes through the Infrastructure Parts Editor. Finally, learn the steps and workflow of bringing these parts into AutoCAD® Civil 3D software and InfraWorks® software for use on an existing model.

Speaker(s)
Steve Hill is an Autodesk™ AutoCAD® / AutoCAD® Civil3D Certified Professional who has eighteen years of professional experience in Autodesk™ products and site civil design with an array of project types including subdivision design, road reconstructions, streetscapes, shopping centers, rail spurs, natural gas wells/pipelines, and landfill design. His project design experience includes various stages from overall planning to the detailed construction including site design, site grading, complex volume calculations, utility and pond sizing, soil erosion control measures, and details. Hill also maintains a strong background in computers and resolving issues on multiple operating platforms. With his LLC, Red Transit Consultants, he is passionate about developing applications and workflow management tools for AutoCAD®, Civil 3D, and Map 3D® as well as small Windows stand-alone applications with various programming languages.

Learning Objectives
- Discover the new Infrastructure Parts Editor software and how it can be used to help advance your models
- Learn how to use existing built-in parts to create custom infrastructure model content
- Learn how to model your very own parametric shape content to build custom infrastructure model content
- Discover how to use Inventor® and the Inventor® Plug In, Infrastructure Part Shape Utilities, to extend the built-in part library with custom parametric models
The new Infrastructure Parts Editor Software

The Infrastructure Parts Editor (IPE) is a parametric shape editing tool for creating parts catalogs that are compatible with Autodesk InfraWorks® and AutoCAD® Civil 3D. (Autodesk, 2017). IPE graduated this year from Autodesk Labs previously known as Project Kameleon.

Terminology

Part Catalog: A collection of part families containing the definitions of all the parts that can be inserted into a model or drawing. (Autodesk | Civil 3D, 2017)

Part Family: A group of similar parts that vary by size. Each part in the part family uses the same general shape, with both geometric and non-geometric properties varying across the size range. (Autodesk, 2017)

Where to Find the Software

The Infrastructure Parts Editor is installed with AutoCAD® Civil 3D and InfraWorks® (minimum 2018.1 is required). The software installs as a standalone application that can be opened without having AutoCAD® Civil 3D or InfraWorks® open, however, there are buttons in both software that allow for launching IPE. Here are the three ways to launch the program

Windows Start:
1. Click Start
2. Go to All Programs, Autodesk, Autodesk Project Infrastructure Parts Editor; alternatively, you can type in Infrastructure Parts Editor to find it.
3. IPE launches and opens to the Catalog tab.

Autodesk InfraWorks®:
1. Open Autodesk InfraWorks®
2. Open an Autodesk InfraWorks® model from InfraWorks® Home
3. Click
4. Click
5. IPE launches and opens to the Catalog tab.

AutoCAD® Civil 3D:
1. Open AutoCAD® Civil 3D 2018 (minimum 2018.1 installed)
2. Open a drawing file.
3. On the home tab, go to the Create Design Panel, and select the drop down
4. Click the Infrastructure Parts Editor button
5. IPE launches and opens to the Catalog tab.
Overview of Infrastructure Parts Editor
Upon opening, you'll see a screen like below. In the upper left, there are three main tabs, Catalog, Parts, and Publish.

Catalog tab is for opening catalogs via file or model, and saving catalogs.

Parts tab is for editing the parts within an open catalog.

Publish tab is for publishing the edited catalog.

Catalog Tab

IPE can be utilized to manage part catalogs for both Pipe Networks and Pressure Pipe Networks. It can be also used for managing Bridge components and other shapes for Autodesk InfraWorks®.

In the screenshot below, notice the two domain categories shown are for Drainage Structure and Piping. In my opinion, this is slightly confusing from AutoCAD® Civil 3D terminology as currently in IPE, you cannot build custom pipes for Pipe Networks. It is better to think of the Drainage Structure domain as Pipe Networks (structures only), and the Piping domain as Pressure Pipe Networks (fittings, appurtenances, and piping).

From the Autodesk Help:
Drainage Structure catalogs are composed of Inlet or Access Structures, Grates or Covers, Underground Structures, and Pipes or Culvert Barrels.

Piping catalogs are composed of Olets, Fasteners, Flanges, Instruments, Fittings, Pipes, Valves and Operators, and Miscellaneous parts.

For this class and sake of time allowed, we'll only be looking at the Drainage Structures.
Note the help button on the left – this will open up an internet page that has a growing amount of information about IPE. This will be helpful in the future.

**Parts Tab**

After opening up the Generic Drainage Structure – Imperial catalog, notice the tab is now switched to the Parts Tab, as the parts are stored inside a catalog. Here you’ll find a Navigation Panel on the left and an Overview of images on the right, like the image shown below.

![Image](image_url)

In the Navigation Panel, at the bottom there are three primary buttons that control the display of the Navigation Panel: Assembly, Structure, and Culvert. Structure is the default selection when entering the Parts tab.

With Structure category selected, in the Navigation panel under Part Families, notice that the part families are divided into three classes: Surface Structures, Underground Structures, and Grates or Covers. It’s important to understand how these classes are used.

**Surface Structures** are the primary controlling feature in a part family and is the portion of the structure used to place a part on a surface. Surface structures include the frame in which the grate or cover would site on and parts of the surface structure would be visible.
from the surface after constructed. These are the primary components that drive in creating the overall assembly.

Select this category to see images of what’s available:

Underground Structures are the portion of the structure that are underground and would not be seen after constructed.

Select this category to see images of what’s available:

Grates or Covers are only the grate or cover access into the structure and are typically visible from the surface after constructed. These do not include the frame that the grate or cover sits on.

Select this category to see images of what’s available:

These three components can be combined together to create an Assembly – items in the Structure category currently can be used in both AutoCAD® Civil 3D and
InfraWorks®. Selecting any of the part families within these classes will bring up the details about that item for editing. For more on this functionality, see the section in this document Using Existing Built-in Parts to Create Custom Infrastructure Model Content.

With Culvert category selected, in the Navigation panel under Part Families, notice that the parts are divided into two classes: Pipes or Culvert Barrels and Culvert End Treatments. It’s important to understand how these classes are used.

**Pipes or Culvert Barrels** are simple pipes or culverts. The out of the box installation comes with Box Pipe and Circular Concrete Pipe.

Select this category to see images of what’s available:

![Image of available Culvert Barrels](image1)

**Culvert End Treatments** are head walls, flared end sections, wing walls, and energy dissipaters that attach to the end of a pipe.

Select this category to see images of what’s available:

![Image of available Culvert End Treatments](image2)

Like the structure components, these two components can be combined together to create an Assembly, however, it’s important to note that currently these can only be used in InfraWorks®. Selecting any of the part families in these two classes will bring up the details about that item for editing. For more on this functionality, see the section in this document Using Existing Built-in Parts to Create Custom Infrastructure Model Content.
With Assembly category selected, in the Navigation panel under Part Families, notice that the parts are divided into two base types: Inlets or Access Structure and Culvert.

Inlets or Access Structures, as previously mentioned, are the combined assembly of a Surface Structure, Underground Structure, and a Grate or Cover. The items in this list are the part families that would be published in the part catalog and would be available for use in AutoCAD® Civil 3D or InfraWorks®.

Culverts, as previously mentioned, are the combined assembly of a Pipe or Culvert Barrel (or multiple) and a Culvert End Treatment. The items in this list are the part families that would be published in the part catalog and would be available for use only in InfraWorks®.

Selecting any of the part families in these two base types will bring up the details about that part family for editing. For more on this functionality, see the section in this document Using Existing Built-in Parts to Create Custom Infrastructure Model Content.

Publish Tab

The publish tab is strictly used when a catalog is ready to be published. Selecting the Publish button launches a selection dialog to all for publishing all or specific Assemblies from the Parts tab.
After selecting Assemblies (Part Families) to publish from the list, choosing Next will bring up the following dialog. Here you can choose to publish to InfraWorks® or AutoCAD® Civil 3D or both at the same time.

A few key points to note about this dialog:

1. The errors at the top – click the blue arrow to read information about any present errors.

2. For InfraWorks®, we can publish to the shared content folder or directly to a specific model.

3. For AutoCAD® Civil 3D, we can publish our catalog to a different directory other than the provided out of the box directory – we just need to remember to set the Pipe Network Catalog to the proper directory to find the parts.
Using Existing Built-in Parts to Create Custom Infrastructure Model Content

Create a New Part Catalog
If you’d like to begin with an empty catalog and add in desired Part Families you can do so by creating a new part catalog. This could be helpful if you’d like to only have a catalog of specific parts for a project. To create the new Part Catalog:

1. On the Catalog tab, Click New.
2. Select the Drainage Structure or Piping Domain.
3. Click Create.
4. Specify the Catalog name, storage path, Description, and units of measurement, then click ok.

Note: Catalogs must have a specific unit and cannot contain part families of mixed units.

Add or Modify a Part Family
Once a Catalog is open, you can add or modify a Part Family as needed.
1. Select either Structure or Culvert button at the bottom of the Navigation pane, depending on the type of part family you plan to add or modify:

   1. Select a part family and the area right of the navigation pane will show tabs with various properties of the part family.

   2. Or, Select the class type (Surface Structures, Underground Structures, Grates or Covers), then click the new button; the part family will be created and the area right of the navigation pane will show tabs with various properties of the part family.
Add or Modify a Part Family Properties
When a part family is selected or newly created, on the Part Family Properties tab, we can verify the Part Category and the Part Class – but cannot edit these values as they are preset based on what class was selected when the part family was created. We can however control the part family name. It’s recommended to have a unique name that correlates to the shape of the part and the range of sizes offered. This can be edited at a later time if needed.

Review or Modify Shape Details
With the Shapes tab selected, here we can specify the Parametric shape. By default, the parametric shape begins to pull from the library of shapes we have. Hover the cursor over a shape to display an enlarged preview of the shape. You may select one of the shapes already available, or choose to define a New Shape Template.

To define a New Shape Template, choose New Shape Template and locate the Inventor® .IPT file from your local system. A preview will appear with the shape and the Parameters key dimensions exposed from the .IPT file.
Modify Detailed Part Family Properties
With the Detailed Properties tab selected, a number of attribute fields will appear depending on which type of part family that is selected. The Part Family Global Name is the only item that is required, all others are optional.

Add or Modify Part Sizes
On the Part Editing tab, the Part Editing table allows for adding and removing size variations, represented in the table as rows. Each column adjacent to the Part Size Local Name represents dimensions for that part. Dimensions with a single asterisk, such as *SPDiameter, are required parameters.

Optionally, we can Export the table to Excel and modify the data there. When finished and saved, we can Import the Excel table back and the Part Editing table will update accordingly.
**Validate Part Family Sizes**

On the Size Validation tab, we have a Size Table and a Size View. As each row in the table is highlighted, we can visually see the model in the Size View adjust to the dimensions entered for that row on the Part Editing tab. This allows for validation that the sizes entered were entered accurately. The Size View can be selected and we are able to orbit the view to see the part from various angles.

![Size Validation screenshot](image)

**Add or Modify Assemblies**

Once a Catalog is open and the catalog contains at least one part family each of the class components for either the Structure or Culverts domain, an Assembly can be created.

To create an assembly, you must begin with a base part family. The base part serves as the master size table for the assembly, controlling the size selection, orientation, size matching requirements, and insertion positioning for the assembled part. For Inlets and Access Structures, the base part type is a Surface Structure. For Culverts, the base part is Pipe Or Culvert Barrel.

To Add a new Assembly:
1. Select the domain Inlet or Access Structures or Culver
2. Click the New button

To modify an existing Assembly, select the part family you wish to modify.
Choose the Base Part
With the Base Part tab current, Select the Base Part Type – this should default to the correct selection based on what domain was selected. Choose the graphic of the base part family you desire to use. Enter in the Assembly Name (part family name).

Choose the Components
With the Components tab selected, drag parts from the Matching Parts section and drop them over to the corresponding location in the Assembly View section. Warning symbols that appear on the matching parts signify that there may be part sizes that do not match the offered base part sizes – it does not mean that the part will not match up. Additionally, it’s important to note that this view will not tell you if a specific part will not match a base part.
Modify the Detailed Properties
On the Detailed Properties tab, edit the Part Family global name, if desired and a short
description about the part family. If the part family is an Inlet, check the box, otherwise it
will be placed in an Access structure folder on export. Also, if the part family has a sump,
check the box to allow for that property to be provided.

Validate the Part Family Sizes
The Size Validation tab shows the sizes from each Assembly matched item and the
sizes it found to match. Sizes that do not match will be shown in red. Select a size to see
the Assembly view show a preview graphic of the part at that selected size.
Modeling Custom Parametric Shape Content to Create Custom Infrastructure Parts

One of the great features of Infrastructure Parts Editor is the ability to generate your own parametric content catalogs for use in AutoCAD® Civil 3D or InfraWorks®. However, in order to do so, there are some important items discussed in this section to get started.

Autodesk Inventor®
In order to create parametric models for importing into Infrastructure Parts Editor, you will need to have Autodesk Inventor® installed with the free Infrastructure Part Shape Utilities add on that comes with Infrastructure Parts Editor. Currently, Autodesk Inventor® is not included in any of the Infrastructure Suites or AEC Collections. Autodesk Inventor® is currently available stand-alone or as part of the Product Design & Manufacturing Collection. Additionally the software can be rented on a monthly basis.

Basics of Autodesk Inventor®
In order to use Autodesk Inventor®, one will need to have a basic understanding of 3D modeling and understand some key basic concepts of Autodesk Inventor®. Some of these key concepts are listed below with a general explanation of each.

Cartesian Coordinate System
Autodesk Inventor® uses a Cartesian coordinate system in a three-dimensional space “in which the location of a point is given by coordinates that represent its distances from perpendicular lines that intersect at a point called the origin.” (Dictionary.com, 2017)
Work Planes
Work planes are a key feature to help constrain your model to part of the Cartesian coordinate system or another object. When starting a file, we are provided with three work plans tied to the coordinate system: YZ Plane, XZ Plane, and XY Plane. All other work planes and model features will be rooted from these planes.

Sketches
Sketches are used to define a 2D shape for extrusion and must be started from the face of an object or a work plane.

Parametric Constraints
Parametric constraints are added to sketches or extrusions to define how various components can be adjusted within the model and how they should interact with other components.

3D Objects
3D objects are created by selecting the profile from a sketch and then defining the length, rotation, and type of extrusion that should take place.

Infrastructure Parts Editor Autodesk Inventor® Template
Currently, a specific template for using Autodesk Inventor® to model parts for Infrastructure Parts Editor is not provided. There are key items that could be added to a template prior to starting modeling to help streamline the process – they are listed below:

Autodesk Inventor® UCS
By default, the top view is set to look down on the Y axis, this is backwards for Infrastructure parts as objects we should be looking down on the Z axis.

Top View – Wrong
Top View – Right
To set the view correctly:

1. Rotate your view to the correct image shown on previous page – in the Standard.ipt template this is set as Front View.
2. Select the drop-down arrow by the view cube

3. Then choose Set Current View as Top

Parameter Names
If desired, one could enter in all the appropriate parameter names for use into the Parameters window and save it to a template. The preferred parameter names are listed on the next page.
List of Preferred Parameter Names to Use
This list was provided on one of the most recent Civil Community Presentations and lists the preferred parameter names for Structure modeling.

<table>
<thead>
<tr>
<th>Class Type</th>
<th>Parameter</th>
<th>Value Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface Structure</td>
<td>FlangeTh</td>
<td>Base Flange Thickness</td>
</tr>
<tr>
<td></td>
<td>WallTh</td>
<td>Wall thickness of Inlet</td>
</tr>
<tr>
<td></td>
<td>CrossSlope</td>
<td>Roadway Cross Slope</td>
</tr>
<tr>
<td></td>
<td>GutterSlope</td>
<td>Roadway Gutter Slope</td>
</tr>
<tr>
<td></td>
<td>SSHeight</td>
<td>Height of Inlet</td>
</tr>
<tr>
<td></td>
<td>SSDiameter*</td>
<td>Diameter of Inlet</td>
</tr>
<tr>
<td></td>
<td>SSSWidth**</td>
<td>Width of Inlet</td>
</tr>
<tr>
<td></td>
<td>SSSLength**</td>
<td>Length of Inlet</td>
</tr>
<tr>
<td></td>
<td>SPDiameter*</td>
<td>Structure Port Diameter</td>
</tr>
<tr>
<td></td>
<td>SPWidth**</td>
<td>Structure Port Width</td>
</tr>
<tr>
<td></td>
<td>SPLength**</td>
<td>Structure Port Length</td>
</tr>
<tr>
<td></td>
<td>CGDiameter*</td>
<td>Diameter of Cover</td>
</tr>
<tr>
<td></td>
<td>CGWidth**</td>
<td>Width of Cover</td>
</tr>
<tr>
<td></td>
<td>CGLength**</td>
<td>Length of Cover</td>
</tr>
<tr>
<td></td>
<td>CGHeight</td>
<td>Height of Cover</td>
</tr>
<tr>
<td>Underground Structure</td>
<td>WallTh</td>
<td>Wall Thickness of Catchment Structure</td>
</tr>
<tr>
<td></td>
<td>BaseTh</td>
<td>Base Thickness of Catchment Structure</td>
</tr>
<tr>
<td></td>
<td>USHeight</td>
<td>Height of Catchment Structure</td>
</tr>
<tr>
<td></td>
<td>USBaseHeight</td>
<td>Base Height (region where pipes connect)</td>
</tr>
<tr>
<td></td>
<td>USDiameter*</td>
<td>Total Diameter of Structure</td>
</tr>
<tr>
<td></td>
<td>USWidth**</td>
<td>Total Width of Structure</td>
</tr>
<tr>
<td></td>
<td>USLength**</td>
<td>Total Length of Structure</td>
</tr>
<tr>
<td></td>
<td>SPDiameter*</td>
<td>Structure Port Diameter</td>
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<tr>
<td></td>
<td>SPWidth**</td>
<td>Structure Port Width</td>
</tr>
<tr>
<td></td>
<td>SPLength**</td>
<td>Structure Port Length</td>
</tr>
<tr>
<td>Cover or Grate</td>
<td>CGDiameter*</td>
<td>Diameter of Cover</td>
</tr>
<tr>
<td></td>
<td>CGWidth**</td>
<td>Width of Cover</td>
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<tr>
<td></td>
<td>CGLength**</td>
<td>Length of Cover</td>
</tr>
<tr>
<td></td>
<td>CGHeight</td>
<td>Height of Cover</td>
</tr>
</tbody>
</table>

*For round objects only

**For rectangular objects only

Bold parameters above are used to match up the assembly in Infrastructure Parts Editor
Infrastructure Part Shape Utilities

Infrastructure Parts Editor automatically will create your model assemblies and it is able to do this because of the tools provided with the Infrastructure Part Shape Utilities.

Assign Cover/Grate allows for applying an image for cover or grate objects. Start the command, select a face, choose Cover, Grate, or Remove, and choose the rotation angle.

Pipe Connector allows for adding connection points where a pipe would connect. This tool is helpful for designing Pressure Pipe Fittings and defining where they connect to a pipe.
Assembly Connector allows for adding connections to drainage structure parts so they can be assembled together properly. The dialog has four port names listed below, only two are actually for drainage structures, italicized below.

Connector
GrateOrCoverPort
PipeOrCulvertPort
UndergroundStructurePort

Reference Point allows for adding additional Reference points to parts so that they can be adjusted and snapped to at these locations within AutoCAD® Civil 3D or InfraWorks®.

Export Template command exports the part along with the necessary images and XML files in order for Infrastructure Parts Editor to process the part.
Model Oil Water Separator

1. Open Autodesk Inventor® and start with a template setup with at least the UCS settings explained above, ensuring that your top view is looking down on the Z axis onto the XY plane.

2. On the 3D Model tab choose Start 2D Sketch.

3. Select the XY plane to connect the sketch to.

4. On the Sketch tab, choose Rectangle Two Point Center.

5. Snap to the yellow center point and drag out diagonally. Before clicking a second point, type in “SPLength = 78”, hit the TAB key and enter “SPWidth = 48”. Hit ENTER when done.

   If you accidentally, clicked and place the rectangle, you’ll see green lines (non-constrained). You can do the following to constrain them:

   a. From the Sketch tab, select the Dimension button, click on the longer green line and click again somewhere to drop the dimension. Then type in “SPLength = 78”.

   b. From the Sketch tab, select the Dimension button, click on the shorter green lines and click again somewhere to drop the dimension. Then type in “SPWidth = 48”.

6. Next, from the Sketch tab, select the Offset button. Select the line of the rectangle and drag inward. Before clicking, type “WallTh = 4” and hit ENTER.

7. Once you are complete and the sketch looks like below, click Finish Sketch.
8. Next, double click the middle mouse button to Zoom Extents, and orient your view so you can see the sketch on an angle.


10. Select the profile button and choose the gap from the above image that represents the 4” wall thickness. See screenshot below for the direction button to click.

11. Change the extrude direction to downward so you are extruding towards the bottom view. See screenshot below for the direction button to click.

12. In the distance text box, clear out the value there and type “USHeight = 60”. Click ok.

The model should now look something like this:
13. Rotate you view so you are looking at the bottom of the structure. From the 3D Model tab, choose Plane Offset from Plane.

14. Select the bottom face of the model in the area of the 4” Wall thickness. Leave the offset value as 0 and click the green check mark.

15. Next, from the 3D Model tab, choose Start 2D Sketch and select the work plane created in the step above.

16. From the Sketch tab, choose the Project Geometry button. Hover over the 4” Wall thickness area until all the lines of the model turn white, then left click. The lines will turn yellow, right click and select Ok. Click Finish sketch when done.

17. Next, we'll Extrude the Floor of the structure. From the 3D Model tab click Extrude. For the Profile, select both the Wall face and the center void area of the structure. In the Distance text box type “BaseTh=7”. Click ok. Your model should now look something like below:
18. Next, we need to model the baffle and weir walls inside the structure. To do so, we can edit our view to help sketch the objects. In the model browser, make sure the Origin folder is open, then from the 3D model tab, click Start 2D Sketch. Select the XZ plane from the model browser. You should now be oriented to the front view and look something like this:
19. Click on the View tab and click the button Slice Graphics. Your view should now look like the image below, so you can easily draw the walls in.

20. Next, from the sketch tab, we'll draw three rectangles representing the baffle and weir walls. The two baffle walls we will draw first – these can be two floating rectangles not touching any walls – use the Sketch tab, Rectangle Two Point and draw something like below.
21. Next, draw the weir wall like shown below – except snap to the top of the floor as this wall should be connected to the floor.

![Diagram of weir wall]

22. Next, we'll want to constrain each of these walls – but we'll want to do so from the center of the walls. To do so, we need to add construction lines. From the Sketch tab, on the Format panel, choose the construction line format button. See screen shot below:

![Construction lines format button]

23. With the construction lines format selected, use the Line command from the Sketch tab and snap to the center of each top of wall and snap perpendicular to the bottom of wall. Right click and choose ok when done.

24. Next, let's constrain the thickness of each of the walls to the Wall Thickness parameter created previously. To do this, click the Dimension button from the sketch tab, select the top line of a wall, pull off the dimension and drop it. In the Edit Dimension dialog, use the arrow button on the right side of the text box, choose List Parameters, and select "WallTh". You could also type this in if preferred. Click the green check mark – repeat for the other two walls.
25. We need to constrain the horizontal position of the walls in relationship to the inside walls of the overall structure. Let’s begin with the longer baffle walls and use the Dimension button again and select an inside end wall and one of the construction lines, pull off the dimension and enter in 14”. Do the same for the weir wall except use 30” for the value. Your sketch should look similar to below:

![Dimension sketch diagram]

26. Next, we need to constrain the top and bottoms of each wall. For the weir wall, we only need to constrain the top as the bottom was snapped to the top of the floor. Use the dimension command to constrain the top of the baffle walls 12” from the top of the structure, the bottom of the baffle walls 12” from the top of the floor of the structure, and the top of the weir 24” from the top of the floor. Your sketch should look similar to below:
27. Click Finish sketch. Orient your model so you are looking into the top of the structure and can see the three walls.

28. Use the Extrude button to extrude the walls. For the profile, choose each of the walls. For the Direction, choose the Symmetric button. For the distance, enter in the following formula “SPWidth - (2*WallTh)”. Click Ok. Your model should look something like below:

29. The model is now complete, and we need to prepare for export. In the Quick Access Toolbar location, find the fx button, this will open the Parameters dialog. For each of the
named parameters, check the Key and Export Parameters column. Click Done when complete.

30. Next, assign the structure material. On the Quick Access Toolbar, click the pull down that says “Generic”. At the bottom, check the box Autodesk Material Library. Click the pull down again and choose Concrete, Precast. The model will now take on that material.

31. In the model browser, turn off any work planes from visibility by selecting them, right clicking, and unchecking visibility.

32. Next, we need to define the structure connection points with the Infrastructure Part Shape Utilities. On the Environments tab click Infrastructure Part Shape Utilities – a new ribbon tab will appear. Choose Assembly connector. In the Port Name text box type “U” – UndergroundStructurePort will appear. Use the Select button and start by selecting a work plane from the model browser – expand Origin and select the XY plane. Next choose the top wall outline of the structure – a green cylinder arrow should appear pointing upwards. Click Ok.

33. Lastly, from the Infrastructure Part Shape Utilities tab, we need to use the Export Template button. Here you can enter a part name and choose how the model visually appears in the graphic. This graphic will be what is used in Infrastructure Parts Editor. Click Export button when ready.

34. The Underground Structure is now complete.
References:

Autodesk | Civil 3D. (2017, 10 22). *About the Part Catalog and Parts Lists for Pipe Networks.* Retrieved from Autodesk Knowledge Network:


http://www.dictionary.com/browse/cartesian-coordinate-system