AUTODESK **UNIVERSITY** 2015

CI10693

Tips & Tricks - 10 Ways to Increase Your Productivity with Autodesk InfraWorks 360

Mathews Mathai
Autodesk,Inc.

Learning Objectives

- Advanced techniques to import and visualize various types of data in InfraWorks 360
- Represent complex transportation features such as bridges, overpasses or intersections
- Move data between Autodesk InfraWorks 360 and other Autodesk design tools
- Create compelling content or visuals to communicate design intent

Description

Don’t we all want to get the most out of Autodesk InfraWorks 360 with a goal of minimizing effort and time, cutting down overall project costs and going beyond what we ever imagined doing? This session will show you 10 ways you can realize this. From powerful advanced techniques for creating, editing and analyzing models to advanced tips for rendering and visualization, creating custom schemas, representing transportation features like intersections, bridges and overpasses or creating custom scripts that create new or enhance existing capabilities of InfraWorks 360, this is a must attend course for anyone wanting to take Autodesk InfraWorks 360 to the next level.

Your AU Expert

Mathews Mathai has been in the Civil Infrastructure arena for the last 23 years and has worked in various roles including software development, application engineering and product management. After completing his Master's degree in Computer Science from the Florida Institute of Technology, Mathews worked for CAiCE Software Corporation developing survey and road design applications before joining Autodesk in 2002. Mathews is currently a Business Consultant with Autodesk Global Services responsible for customizing, implementing and deploying Autodesk infrastructure and planning solutions all over North America.

Contents

Advanced techniques to import and visualize various types of data in InfraWorks 360

Extending the InfraWorks 360 Database Schema

InfraWorks 360 comes with a default database schema that describes the entire configuration of the InfraWorks 360 database, including all of its tables, relations, and user definable attributes. Schema templates allow you to set up custom classes, attributes, and categories when a model is first created.

Why Modify the Schema
A user may need to work with infrastructure and building data that are currently not part of the standard out-of-the-box InfraWorks 360 database schema. For example, the user may have an ESRI shape file containing transportation data (for example, road type and condition index) or building asset data (for example, building type, floor area and annual energy usage) that they need to use in a conceptual design.

The Autodesk InfraWorks 360 schema definition per default is incorporated in the model itself. To modify or change the default schema definition, you need to create a file called **im_schema.json** and place it in the ***<model.files>/unver*** folder.

Helpful Tips

- Where possible start with an existing schema definition file and modify as needed. A sample schema file can be found at ***…/CI10693/Data/Schema Definition***
- After a schema definition file is created or modified and placed in the ***<model.files>/unver*** directory, close and restart the InfraWorks 360 project for the schema changes to take effect.

Schema JSON File Structure

There are two main sections in the schema JSON file, one to define the "Classes" and one to define the "Display"

"**Classes**": These define new user-defined categories derived from base classes that you want to use in your project and will display both in the Model Explorer and the feature's Properties.

The standard available InfraWorks 360 "base" classes from which new classes can be created are:

- BARRIERS
- BUILDINGS
- CITY_FURNITURE
- COVERAGES
- LAND_AREAS
- PIPELINES
- PIPE_CONNECTORS
- POIS

- RAILWAYS
- ROADS
- TREES
- WATER_AREAS

To create two new classes called CITY_BUILDINGS and CITY_ROADS from the base class BUILDINGS and ROADS respectively, enter

```
"Classes": [
        {
             "name": "CITY_BUILDINGS",
             "base": "BUILDINGS",
             "Attributes": []
        },
        {
             "name": "CITY_ROADS",
             "base": "ROADS",
             "Attributes": []
        }
```

Note:

- The closing parenthesis of the last Class entry is NOT followed by a comma

Next we need to define the related attributes associated with the newly created custom classes. Each attribute can only be one of the following types; Integer, Double, DateTime, Boolean, or String:

- To create 4 attributes for the new CITY_BUILDINGS class called BUILDING_TYPE, YEAR_BUILT, FLOOR_AREA and ANNUAL_ENERGY_USE, in the Class Attributes section (between the [ ]) enter

```
{
 "name": "BUILDING_TYPE",
 "type": "String"
},
{
  "name": "YEAR_BUILT",
  "type": "Integer"
},
{
  "name": "FLOOR_AREA",
  "type": "Double"
},
{
  "name": "ANNUAL_ENERGY_USE",
  "type": "Double"
}
```

- To create 2 attributes for the new CITY_ROADS class called ROAD_TYPE and CONDITION_INDEX, enter

```
{
 "name": "ROAD_TYPE",
 "type": "String"
},
{
 "name": "CONDITION_INDEX",
 "type": "Integer"
}
```

Note:

- The closing parenthesis of the last Attribute entry in each Class definition is NOT followed by a comma

Next, we need to define how we want these custom attributes to "Display" within the InfraWorks 360 Model Explorer and Feature Properties dialog by defining the style, order and naming of the attributes. Enter the following in the "Display" section

```
"Display": {
      "en": [
            {
            "name": "BUILDING_TYPE",
             "displayName": "Building Type",
             "category": "City Building",
             "priority": 500
            },
            {
            "name": "YEAR_BUILT",
             "displayName": "Year Built",
             "category": "City Building",
             "priority": 501
            },

            {
             "name": "FLOOR_AREA",
             "displayName": "Gross Floor Area",
             "category": "City Building",
             "priority": 502
            },
            {
             "name": "ANNUAL_ENERGY_USE",
             "displayName": "Annual Energy Consumption (kWh)",
             "category": "City Building",
             "priority": 503
            },
            {
             "name": "ROAD_TYPE",
             "displayName": "Road Type",
             "category": "City Road",
             "priority": 506
            },
```

```
    {
     "name": "CONDITION_INDEX",
     "displayName": "Condition Index",
     "category": "City Road",
     "priority": 507
    }
]
```

Note:

- The closing parenthesis of the last Display entry is NOT followed by a comma
- The priority defines the display order starting with the low values first.

Lastly, after you have completed your editing, save the file and store it in the **"<model.files>/unver**" folder and then close and reopen the model.

To verify that the new schema has taken effect, select and edit a building or road feature to view its properties. You should now see a section called CITY_BUILDINGS and CITY_ROADS with the new custom properties.

Using Scripts to Import Custom Data

Frequently, you will come across the need to import and use data in your model that resides in a format not supported by InfraWorks 360. For example, you may have spreadsheet information that has been exported to a CSV file that you would like to import and use in your model.

Why Script

Scripting is typically used to simplify workflows, automate workflows, extend the existing capabilities of InfraWorks 360 or integrate *with* other applications using external Application Programming Interfaces (*APIs)*. APIs are able to do this by "exposing" some of a program's internal functions to the outside world in a limited fashion.

Types of Scripts

There are two types of scripts that can be created and executed within the Autodesk InfraWorks 360 environment:

Project Specific Scripts

These scripts are project specific and are used to format and filter imported data with JavaScript during Data Source Configuration.

1. Click  .
2. Import a file or database data source.
3. Double click the data source in the **Data Sources** panel to configure the data.

4. Click the **Scripts** tab in the **Data Source Configuration** dialog box.
5. Select **Edit**.
6. Using JavaScript, enter your changes or add new scripts.

   The example below shows a simple script that varies the style for streets, based on their values for the property "Elevation."

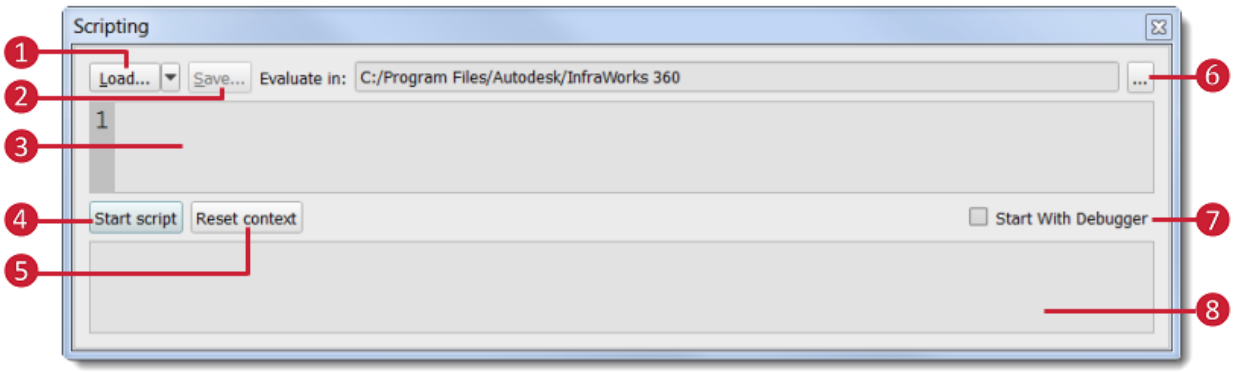
```
function Process() {
   ROADS.ELEV_FROM = SOURCE.ElevStart;
   ROADS.ELEV_TO = SOURCE.ElevEnd;
   ROADS.LANES_BACKWARD = SOURCE.LanesTo;
   ROADS.LANES_FORWARD = SOURCE.LanesFrom;
   ROADS.NAME = SOURCE.Name;
   if ((ROADS.ELEV_FROM > 0) || (ROADS.ELEV_TO > 0)){
      ROADS.RULE_STYLE = "DefaultStreetStyles:Bridge0";
   if ((ROADS.ELEV_FROM > 0) || (ROADS.ELEV_TO > 0)){
      ROADS.RULE_STYLE = "DefaultStreetStyles:Tunnel0";
   } else {
      ROADS.RULE_STYLE = "DefaultStreetStyles:Street0";
   }
}
```

   Note that once the **Edit** tab is selected to create or modify a script, the **Common** tab becomes unavailable and all processing of the selected data must be handed by the script. To activate or access the **Common** tab, the **Revert** option must be selected. This will result in the default script being reinstated.

Standalone Scripts

A standalone script is not project specific and can be created using JavaScript to simplify workflows, add new workflows or extending out of the box capabilities

Click  ➤  ➤  to open the Scripting Console.



1. Load a JavaScript file to use.
2. Save the current script.
3. Click in the Scripting Area and type or paste in a script.
4. Click to start the script.
5. Reset the context of the script.
6. Browse to a different location to evaluate the script in. This is the directory where the script will be run.
7. Select Start with Debugger to open the debugger window once a script is started.
8. Script results are returned here.

Scripting Language

Autodesk InfraWorks 360 uses JavaScript as its scripting language. Being a C-based language, if a user is familiar with C, C++, or C# and has had any development experience in the past, scripting using Autodesk InfraWorks should not be too difficult at all. There is also lots of help on the internet on JavaScript.

A very good resource for JavaScript tutorials and examples can be found at www.w3schools.com/js

Documentation on the InfraWorks JavaScript API is provided as part the Class Files provided as part of this session and can be found at **…*CI10693*\Data\InfraWorks JS Documentation\html.** To view the JavaScript API document, extract the contents of the file and select the **index.html**

Important: It should be noted that the current JavaScript is currently “un-supported” by Autodesk. This means that the APIs are being enhanced and have not been locked down. Any scripts that are developed using this “un-supported” API may require some minor changes to execute when the final API is released at some point in the future.

We will demonstrate how to use scripting to import building data from a CSV file exported from an EXCEL spreadsheet into a model using the new database schema created earlier in the session

Sample CSV File:

BldgID,Name,Height,Type,Year Built,Annual Energy Usage
23,City Hall,65.5,City,1956,15575,5890
45,Post Office, 18,Federal,1992,5790,1600
67,Fedex,10,Private,1500,2007,600
….

The Javascript script is located at **…*CI10693*\Data\Javascript Example**

Using Model Explorer Subsets to Build Selection Sets

InfraWorks 360 users can use the Model Explorer functionality to select an existing feature class (base or custom) and create an SQL expression that specifies a subset of that feature class. You can use subsets to compare data within a feature class or create simple of complex select sets to assist with planning and analysis.

Feature classes can be filtered in the Model Explorer using an expression, which creates a subset of the feature class based on the conditions you specified.

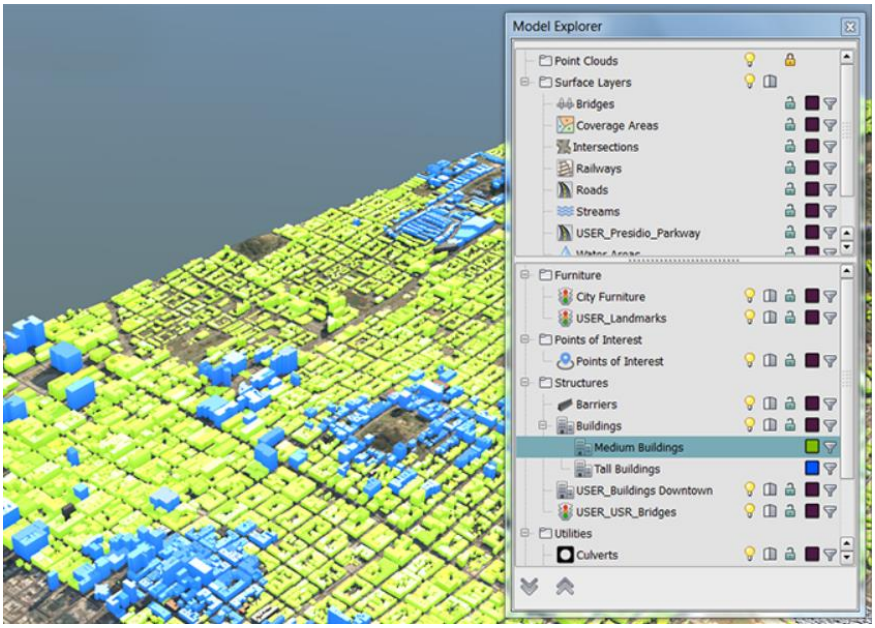
There are 3 types of filters:

- **Simple Filter**: Specifies a property, an operator, and a value. For example, ROOF_HEIGHT > 24. The subset defined by that expression includes only buildings whose roof height value is more than 24 feet.

- **Multiple-Condition Filter**: Combines two or more simple filters. For example, ROOF_HEIGHT > 24 AND BUILDING_TYPE = 'Office'. The subset defined by that expression includes only buildings whose roof height value is more than 24 feet and that are Office buildings**.**

- **Location Filter**: Specifies an area in the model. Features within this area are included in the subset, while those outside the area are not. For example, you can include features within a radius from a known location in the model.

For example, to visually compare buildings of different roof heights, create a subset for each height and assign a different highlight color to each one. Then turn the highlighting on and off for the different subsets to compare their number, location, and so on. A single feature class can have multiple subsets.



Workflow:

1. If you are filtering by location, fit the model to the extents of the feature class or data source.
2. Click  in the **Utility** Bar.
3. In the **Model Explorer** panel, for the Building feature class to filter, click  .
4. In the **Create New Subset** dialog box, create an expression
5. When you have finished defining the subset, click **OK** to apply the filter. InfraWorks 360 runs a validation to check the syntax of the expression. If the validation finds an error, an error message will display in the bottom of the Expression Area.

   To re-define the subset expression at any time, right-click the subset and click **Edit Subset**. To make a copy of the subset (and its expression), right-click the subset and click **Copy Subset**. You can then edit the copy.

6. In **Model Explorer**, enter a name for the subset. To rename the subset later, right-click it and click **Rename Subset**.

Example: We shall create a subset that uses a **Multiple-Condition** and **Location** filter to select all buildings and roads within a 500 feet radius of a known location where the buildings have a roof height of at-least 50 feet and the road condition index is > 7.5 (where 1 is poor and 10 is excellent).

Applying User-Defined / Custom Styles to New InfraWorks 360 Models

Typically on a project, the styles you see in the style palettes of a model are a combination of the InfraWorks base styles plus user created custom styles. However, when a new InfraWorks 360 model is created, either with Model Builder or using specific data sources, only the default styles are currently available. The user has to then either create new styles or import custom styles from one or several existing projects which can be time-consuming. How can all newly created InfraWorks 360 models automatically include custom styles that have been created earlier?

Workflow 1: (***Non-Model Builder Generated Models***)

1. Create an empty model and use it to store all your custom styles.
2. Duplicate this model to start a new model and name it accordingly.

Workflow 2: ***(Model Builder Generated Models)***

1. Copy your custom styles from a model into the local library where the base styles are stored.
   - Styles are stored within existing models here:
     - ***<InfraWorks Model directory>\<model name>.files\unver\Content\Styles\<style type>*** where the InfraWorks model directory is typically: ***%USERPROFILE%\Documents\Autodesk InfraWorks Models***
     - For each style there is an **ACItem** file and a corresponding image file.

   - The local Styles library can be found here:
     - ***%ProgramData%\Autodesk\InfraWorks 360\Resources\LocalLibrary\Styles\<style type>***

   Note that styles added to the local library will only be seen on the local machine. You also want to be careful doing this since you would be modifying out of the box style files if you modified an existing style and did not name it differently.

2. Copy the custom styles into the local library to be able to see them from any model.

3. Name your custom styles so they can be differentiated from shipping styles in case you want to export them for sharing.

Represent complex transportation features such as bridges, overpasses or intersections

Use Custom Profiles in Road and Railway Styles

A custom profile is a 2D cross section of linear features such as barriers, pipelines, guardrails, and fences that can be created in an external application (such as AutoCAD Map 3D) in SDF or SQLite format. The 2D representations are then brought into InfraWorks 360 and attached to a road or railway style. InfraWorks 360 will then extrude the 2D cross section along the length of the selected road or railway track so that it can continuously hug the road or railway and smoothly follow its assigned track around curves.

For example, the image below shows a road style using a custom profile (created in AutoCAD Map 3D) as a crash barrier

Custom Profile

Application of the Custom Profile

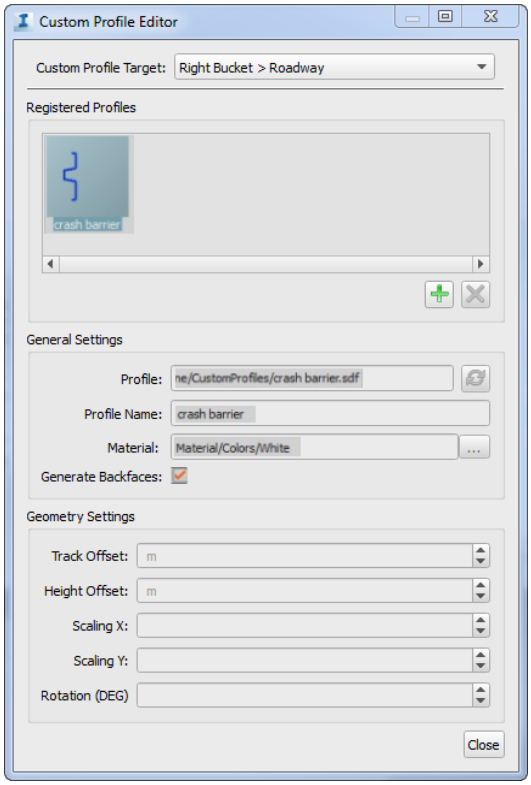
Custom profiles allows user to create any number of helpful features, such as guardrails, barriers, or fencing. More advanced uses of custom profiles (in combination with Decorations) can also help you visualize features such as ski lifts, gondolas, even power cables.

Here is the basic workflow of using Custom Profiles in InfraWorks 360:

1. Click  .
2. Select the **Road** or **Railway** tab in the **Style Palette**.
3. Double-click the style you want to edit.
4. Click  to open the **Custom Profile Editor**.
5. Select the track that you want to attach a custom profile to as your **Custom Profile Target**.

   For example, you can choose to add a custom profile to a road's median, bikeway, curb, green-space, or sidewalk, and so forth.

6. Select a profile style from the **Registered Profiles** pane, or click  to choose a SDF or SQLite file from your local system to add as a new profile. The custom profile appears in the preview area of the editor.



7. Specify settings:

   General Settings:

   - **Profile**: The file path for the selected custom profile. Refresh if you have updated the source data.
   - **Profile Name**: Give the custom profile a name.
   - **Material**: Choose a material color or texture for the profile.
   - **Generate Backfaces:** Check this setting to generate front and back faces for the profile, creating closed triangles.

   Geometry Settings:

   - **Track Offset:** The horizontal offset of your custom profile, relative to the centerline of the track you specified as the custom profile target.
   - **Height Offset:** The vertical offset of your custom profile relative to the centerline of the custom profile target track.
   - **Scaling X:** Scales the road or railway horizontally relative to the profile points.
   - **Scaling Y:** Scales the road or railway vertically relative to the profile points.
   - **Rotation (DEG):** Rotates (in degrees) the road or railway relative to the profile points.

8. Click **Close** in the **Custom Profile Editor**, then click **OK** in the **Configure Road/**... or **Configure Railway/...** dialog box. Drag and drop your updated road or railway style onto the desired road or railway to display its custom profile.

   Note: In the case of intersections between roads that have custom profiles, the custom profiles will connect smoothly around corners if the intersecting roads are using the same style. However, railways do not support custom profiles for intersections.

   Several sample custom profiles are located at **…*CI10693*\Data\Custom Profiles**

Creation and Visualization of Intersections

Creation of intersections with dedicated turn lanes for visualization purposes has been a challenge in the past due to:

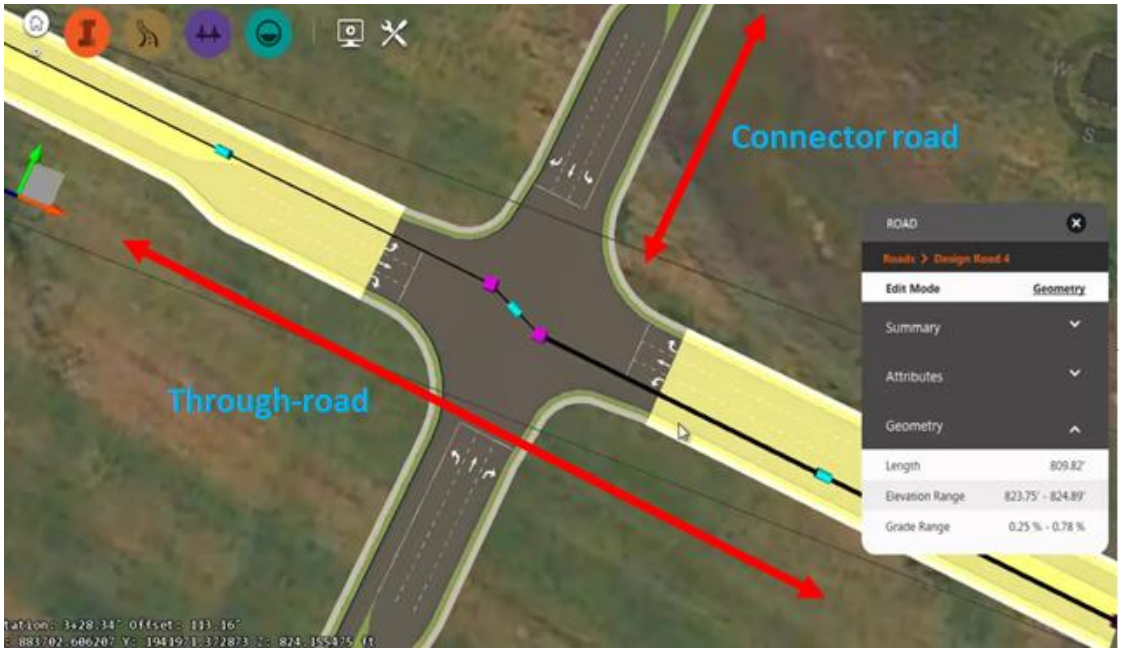
- Difficulty lining up the through lanes at an intersection when there are left, through and right turn lanes
- InfraWorks limitation where it is not possible to have 4 disjointed roads create an intersection

InfraWorks 360 now supports offset intersections (i.e. intersections with offset approaches), so with the use of both right and left turn lane styles and an intersection object, you can create an intersection with multiple turn lanes for visualization purposes. With the additional tweaking of styles and alignments, other simple and complex intersection configurations can be achieved a similar workflow.

Let us see what the desired end result looks like below where the intersection has 4 approaches where each one has its own left and right turn lane as well as a through lane

Fundamental Approach

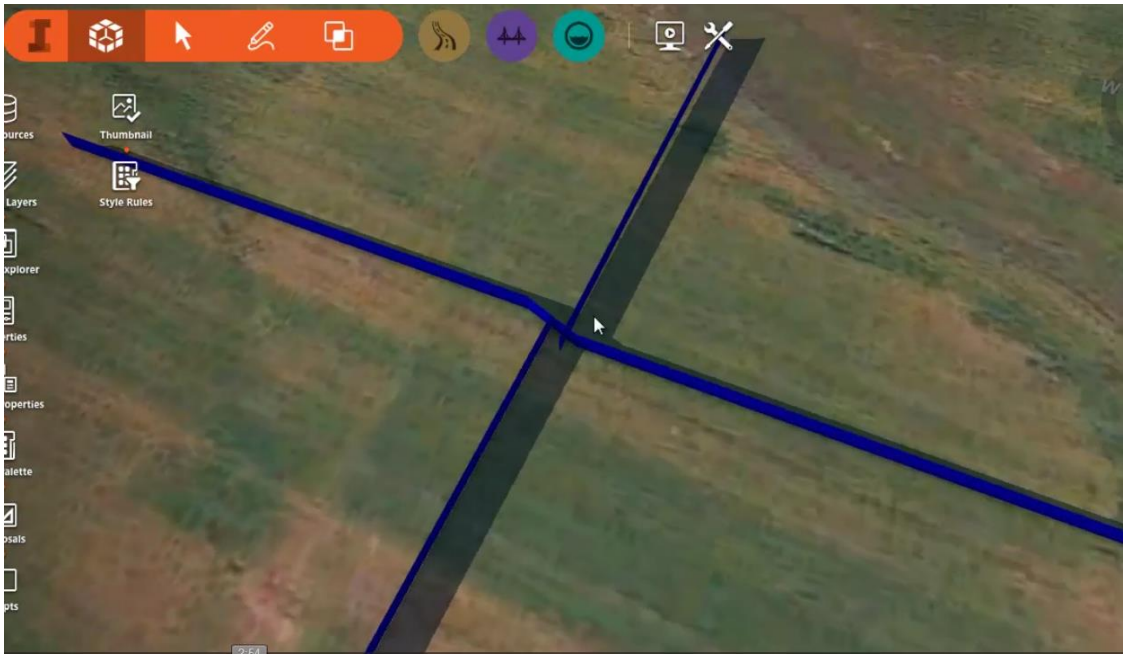
Create a single alignment for the through road with a kink in the alignment and separate alignments for the connector roads. The kink in the through-road measures the width of the lane and allows the through-lane to line up across the intersection.



Note: While an intersection such as this can be created using free style sketching it will requires a bit of experimentation as the grip locations are unnatural. So we will use a template to help make this process more repeatable and easier to deploy. The template will help the planner/designer sketch in the through and connector roads so that the grip locations are inserted in the right location and then allows the user to make minor adjustments if needed.

Workflow:

1. Create a new InfraWorks 360 Style Catalog called **Intersection** and import the 3 styles located at **…*CI10693*\Data\Multi-turn Intersection Layout.** The styles are:
    - Traditional roadway with green-space and sidewalk
    - Asymmetrical road with a median on the right side of the centerline
    - Asymmetrical road with median on the left side of the centerline
2. Insert the intersection template by inserting the FBX template file located at **…*CI10693*\Data\Multi-turn Intersection Layout**
3. Configure this 3D Model FBX file as a **City Furniture** feature and use **Interactive Placing** to place it at the desired location on the model. This template will show us where we need to place the grips as we sketch/design the alignments. See image below of the template

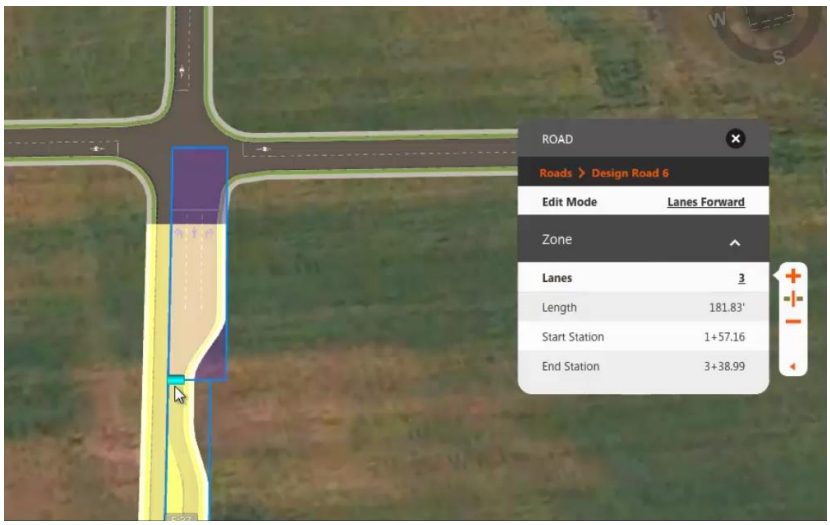


4. Use the InfraWorks 360 **Collector Road** feature type to lay the road where the template is meeting the ground for both the through-road and connector road

   Note: We are using a **Collector Road** feature because it has a high design speed and will not allow curves to be inserted between PIs as we draw in the road components. Hold down the **Ctrl key** to connect the connector road endpoint to the template endpoint at the intersection as it will tend to otherwise gravitate towards the through road centerline PI location.

5. Remove the intersection template once the through and connector roads are inserted. We should now see our offset intersection.
6. Create and apply a **Style Zone** on the southern approach using the **Median RT** style.

7. Create and apply a forward lane zone and specify 3 lanes (left, through and right)



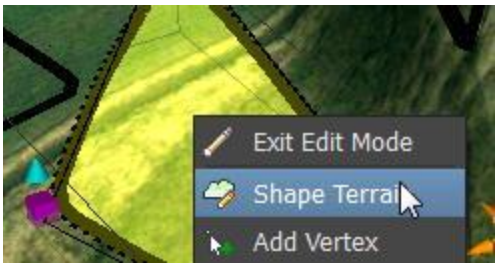
8. Repeat the same process for the opposite connector road approach road.
9. Follow the same process for the through road except create a lanes forward zone that ends in the middle of the kink and then set the start point.

   Note: If you go too far into the intersection to set the end point, the lane will end up spilling over into the other side of the intersection. If you go too short, the intersection will end up looking like a knot. So you may have to slide the end grip and move it back and forth to find the right location.

Manipulating Coverage Areas

Terrain Shaping

The "Shape Terrain" Coverage Area feature lets you flatten a coverage area and set the elevation of that coverage, either by dragging the vertical arrow grip (with a preview) to the desired elevation or by typing the desired value. All vertices on the coverage boundary are then set to this value.



Controlling Coverage Area Corner Radii

The InfraWorks 360 defaults generate cornered coverage area features. Although there is no built-in option from any menu that allows you to control this setting, there is a variable within the InfraWorks resource files (.JSON) that controls the value for corner radii.

The value is called **SmoothParam**, and can be found in the .JSON files for the respective elements. Building settings are in the ***Buliding.JSON*** file and Water settings are in the ***Water Area.JSON*** file. The files themselves are saved as part of the individual InfraWorks model's backup files, so you may have to do some digging to get to the folder.

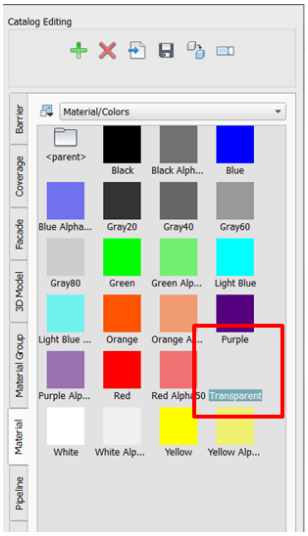
A typical path would be: ***<InfraWorks Models Folder>\<Model Name>\.files\unver\DrawTools***

Once you find these files, open the one you want using **Notepad**, and modify the **SmoothParam** value to adjust the corner settings for those elements. When working with .JSON data, there are some important things to know.

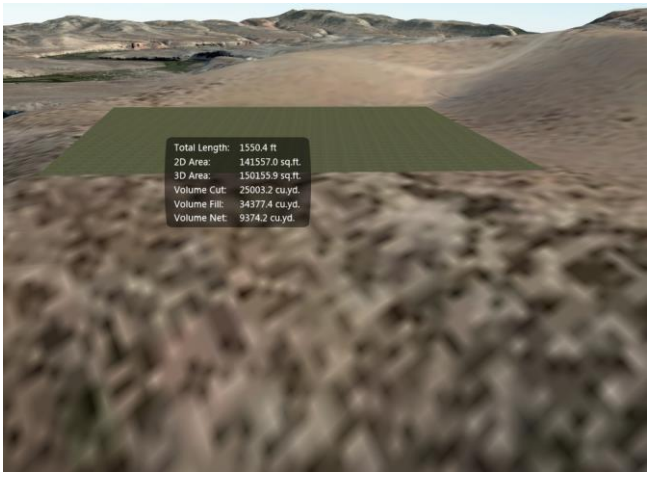
1. .JSON files are generated for the model when the elements are added in InfraWorks 360. For example, if your model does not include (and never has) any water features, the ***Water Areas.JSON*** will not exist. You'll need to create an element of that type first so the .JSON file is generated, then you can edit the setting.
2. The .JSON files are referenced by the model when the model is originally opened. That being said, any changes you make to variables within .JSON files require you to close and re-open the model before those changes take effect.
3. The default values to use for **SmoothParam** are between 0.00 and 0.499. The default value for Buildings is 0.00 to create corners, and 0.499 for Water Areas to create fillets.
4. You can use any number 0.00 or higher. You'll get some funky shapes if you get anywhere near or above 1.0, but it is worth checking out just to see what happens.

Controlling Coverage Area Slopes
If you edit a Coverage Area and look at its properties you can see a field called **Smooth Radius**. The Smooth Radius value can be modified to change the slope of the rounding. A value of 0 will get a near straight vertical wall.

Applying Transparent Styles to Coverage Areas
Frequently, after a coverage area has been shaped, the materials associated with the coverage area remains. Assign a **Transparent** color to the Coverage Area to see through to the underlying terrain texture or imagery.



Generating Cut and Fill Volumes using Coverage Areas
The **Terrain Statistics** feature provides the ability to calculate and display the surface volume information (cuts and fills) resulting from a coverage area being adjusted for location and /or height.



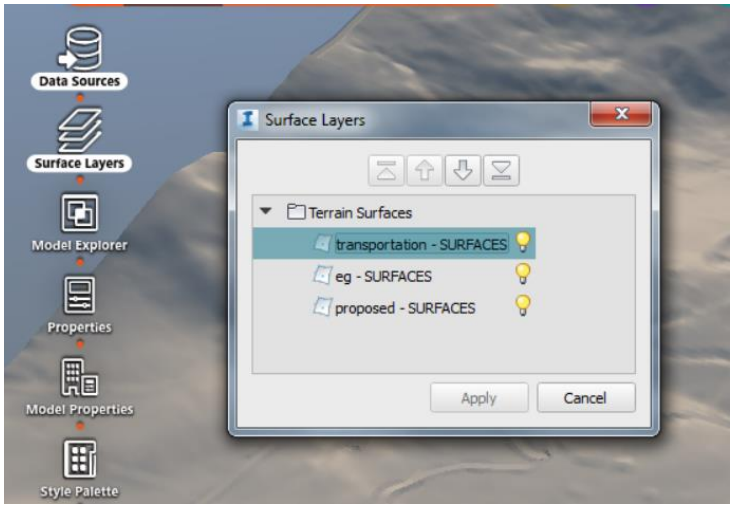
Move data between Autodesk InfraWorks 360 and other Autodesk design tools

Layering Multiple Civil 3D Surfaces in InfraWorks 360

InfraWorks 360 currently handles a single terrain surface at a time. When multiple surfaces are imported into the model a combined/merged surface is created where the last one imported is the priority surface. Frequently, a user may, for example, need to visualize a Civil 3D phasing plan for a landfill in InfraWorks 360 and will be starting with existing topography which will have multiple 'phases' or fill sequences. The following workflow describes how the user can layer their design over the existing topography and orthoimagery.

Civil 3D will export all surfaces in a drawing as a single InfraWorks surface when exported to an .IMX file.

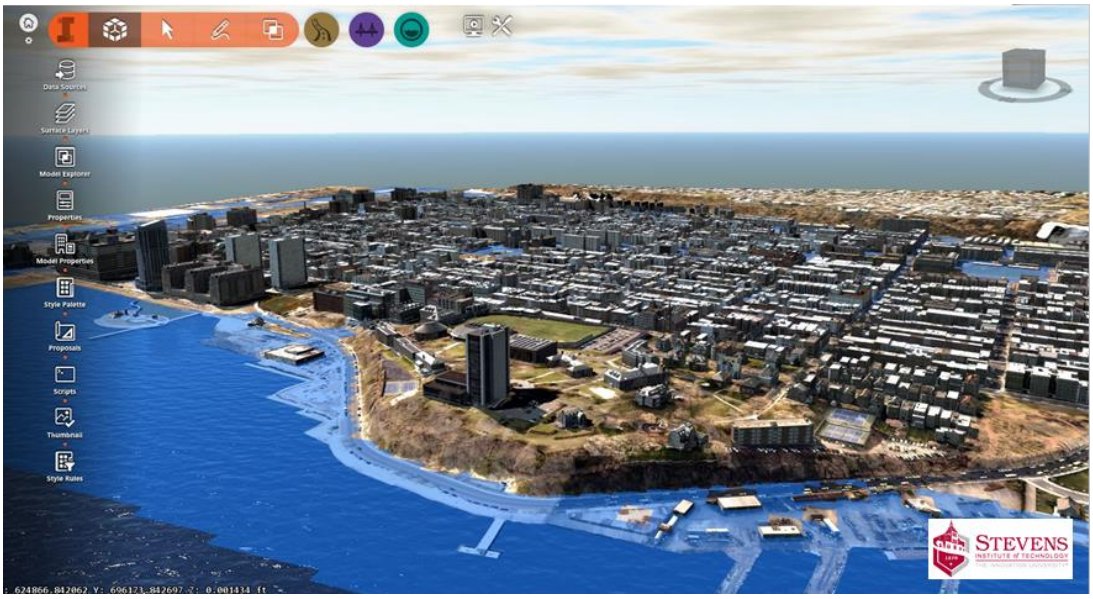
1. Import the IMX file into InfraWorks 360 using the **Data Sources** feature
2. A list of all surfaces in the IMX is displayed. Select the first surface to be imported.
3. Continue the same process to import the other IMX surfaces to create separate surfaces in InfraWorks 360. Each surface is placed on a separate layer
4. Control the display order and visibility of each of the imported surfaces using the **Surface Layers** tool.



Creating and Visualizing a Flood Surface Using ESRI GIS Data and Civil 3D Surface

InfraWorks 360 can be used to bring in a variety of different data types to communicate and visualize interesting concepts and events. In this example, Autodesk worked with Stevens Institute of Technology in Hoboken, New Jersey to create a representation of the flood that occurred during super storm Sandy in the City of Hoboken at 21:30 EDT, Oct 29, 2012, when the Hudson River was at its highest off the Castle Point.

The flood surface at the specific time was provided using an ESRI GRID file.



Flood Simulation at 21:30 EDT, Oct 29, 2012, when the Hudson River was at its highest off the Castle Point.

Workflow:

1. Import the flood surface model (for example ESRI GRID (.adf) file) into Civil 3D as a surface
2. Assigned a water material to the created surface
3. Save the surface as an FBX model file from Civil 3D
4. In InfraWorks 360, import the FBX file as a 3D Model (under **Data Sources**) and assign it as a "**Points of Interest**" feature type. Make sure you assign the appropriate Coordinate System.
5. **Close and Refresh**.
6. If you want to turn off the model from viewing, go to **Model Explorer** and turn of the light bulb next to **Points of Interest**

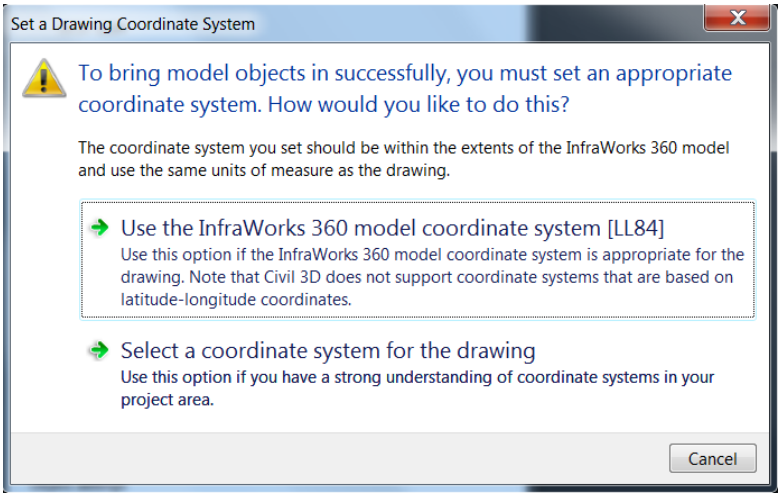
   Note you could potentially bring in multiple hydrodynamic flood models as Points of Interests showing the flood situation at different times of the day and turn anyone of them on or off for visualization

How to use InfraWorks Model Exchange with Civil 3D

Many users are developing preliminary design concepts using InfraWorks 360 and then need to take the model into a design application like Civil 3D for detailed design. Once the detailed design is complete, they want to bring it back into InfraWorks 360 to visual and help determine if further refinements are needed on the design side. These users also want a workflow that is streamlined, automated and easy to use and avoids cumbersome exporting and mapping workflows.

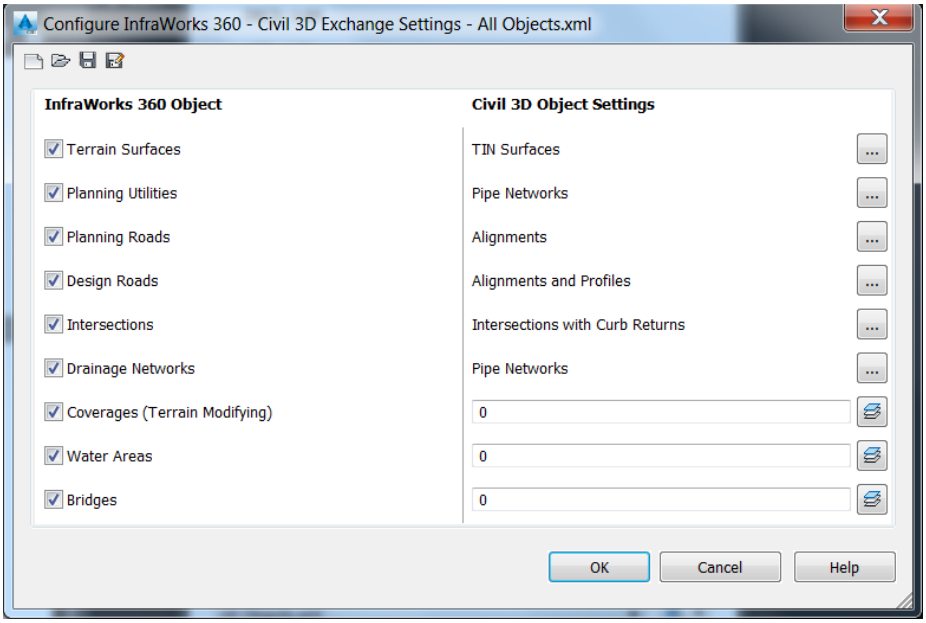
Use Case 1: Preliminary Design in InfraWorks 360, Detailed Design in Civil 3D

1. Create a new drawing in Civil 3D using an existing template. There is no need to set any coordinate systems when setting up the new drawing.
2. On the Civil 3D 2016 **Insert** menu, select **Open InfraWorks 360 Model**
3. Select the InfraWorks model (.sqlite) that you want to use. Civil 3D will look for the coordinate system used by the InfraWorks 360 model and load it in the dialog.
4. Set a coordinate system for the empty Civil 3D drawing by selecting **Set a Coordinate System…**
5. You are presented with a couple of options for setting the Civil 3D drawing coordinate system:
    a. Use the model coordinate system used by the InfraWorks 360 model
    b. Select a coordinate system using the Civil 3D **Drawing Settings** dialog box



6. Bring in either the entire InfraWorks 360 model (or the model within a defined Area of Interest).
7. In the **Object Settings** section, there are 3 out-of-the-box setting configurations you can choose from to determine how the InfraWorks 360 objects will be mapped to Civil 3D objects when they are brought into Civil 3D. These are very similar to a style and can be edited as needed.

    Note: You can also access this functionality directly using the **Configure InfraWorks 360 - Civil 3D Exchange Settings** in the **InfraWorks 360** Civil 3D menu
8. Open the settings configurations file ***All Objects.xml.***



9. Select the necessary mapping and then save as a new custom configuration file
10. Use the **Refine Selection Set…** to further refine what you need to bring in from InfraWorks 360 (Terrain Surfaces, Planning Roads, Planning Utilities, Design Roads, Intersections, Drainage Networks with same part sizes, etc.)
11. Click on **Open Model**

Use Case 2: Moving a Civil 3D Design into InfraWorks 360 for Visualization and Analysis

1. Save the Civil 3D drawing containing the detail design you want to bring in and visualize using InfraWorks 360
2. Create a new InfraWorks 360 proposal using the existing preliminary design as the base proposal.
3. Select **Data Sources** and select the **Autodesk AutoCAD Civil 3D DWG** option
4. Pick the Civil 3D DWG drawing you want bring into InfraWorks 360. The drawings will be examined to determine which objects InfraWorks 360 will be able to consume before they are brought in
5. Turn off all the surfaces that originated from the InfraWorks 360 preliminary design. These would typically be the following:
    - AIW_Existing_Ground
    - AIW_Existing_Transportation
    - AIW_Proposed_Ground

    Note: Ensure that the **Roads**, **Corridor Surface** and **Coverage Area** options are turned on.

6. Configure the **Roads** feature.

   By default, InfraWorks 360 will attempt to bring back all the roads in the drawing that originated from InfraWorks 360. Exclude all the roads except the centerlines of the design corridors you want. **Close and Refresh**
7. Bring in the **Corridor Surface** and **Coverage Areas**
   Coverage areas are created to represent the different object types that were brought in as part of the corridor. For example, there is a separate coverage area for the pavement, the different elements of the curb, sidewalk, etc.

Note: If any modifications are made in Civil 3D (such as extending the corridor length) and the drawing changes saved, selecting the **Civil 3D Data Sources** entry in InfraWorks 360 and clicking **Reimport**, the model will be automatically updated to reflect the design changed made.

Create compelling content or visuals to communicate design intent

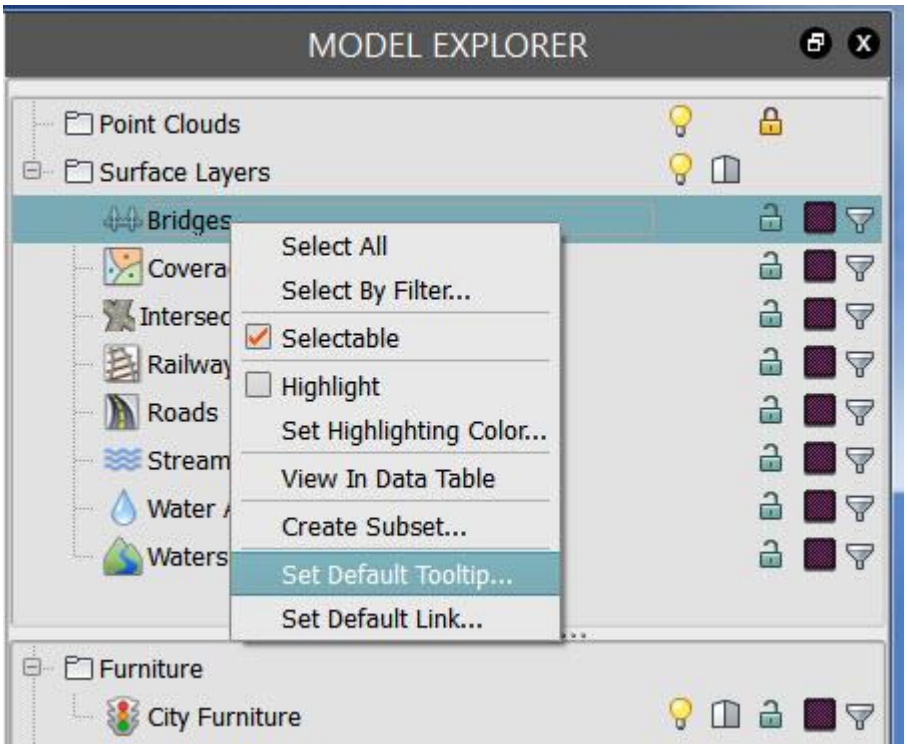
Using Tooltips to View Asset and Performance Data

The InfraWorks 360 feature tooltip feature can be used in very interesting and compelling ways to quickly get access to feature data as well as calculated and external data.

The example below shows how you can use Tooltips to display building asset information, performance characteristics, historical data and links to reference files all by hovering the mouse over the specific building asset.

You start by creating default tooltips for feature classes using the Model Explorer. Asset or subset selection-based tooltips created from the Properties palette override default feature class tooltips created from the Model Explorer.

1. Click  in the **Utility** Bar to display the **Model Explorer**.
2. In the **Model Explorer**, right-click the row for the feature class whose tooltip you are creating.
3. Click **Select All** to apply the **Tooltip** to all objects in the feature class.
4. Right-click the feature class row again and click **Set Default Tooltip** to open the **Edit Tooltip** window.



5. Enter Tooltip content.
    - Optionally, click **Preview** to see what the tooltip will look like. Click **Return to Editor** to revise or to finish creating tooltip.
6. Click **OK**.
7. Click **Update** in Properties Palette to replace existing Tooltip content, if any.

Sample code for a tooltip displaying building asset information is provided at
…*CI10693*\Data\Tooltips

InfraWorks 360 Troubleshooting and Performance Tips

In this section we will look at how to effectively troubleshoot your InfraWorks models if a problem arises. Issues that may arise occasionally include:

- The model has crashed several times and now the user is unable to create new proposals.
- Although Autodesk® InfraWorks™ launches successfully, the project crashes each time you try to load it.
- When opening a project, the user sees an error message that the database schema is unknown.
- The project cannot be opened after upgrading the model to a later version.

Recovery of InfraWorks 360 Models
There are currently two options available to try and recover your InfraWorks 360 project/project data. The first option is the simpler of the two and has less potential for data-loss. The processes described below do not guarantee to restore all InfraWorks 360 projects that may have been corrupted but has been success on most projects where some anomaly occurred. It is strongly advised to create backups for every folder or file that is being edited or deleted.

Workflow 1: (Simple and Less Potential for Data Loss)

1. Close InfraWorks 360
2. Navigate to the **TileCache** folder found at:
   C:\Users\<current user>\AppData\Local\Autodesk\Autodesk InfraWorks\cache\16.3 [this is the default location]
3. Empty the contents of the **TileCache** folder without deleting the folder itself. This will cause the project to rebuild once you try and open it again.
   Note: You may need to change your folder settings to show hidden files and folders.
4. If steps 1-3 do not resolve the issue, continue with the following steps to try to recover your data.
5. Reboot your machine. This ensures that no stray 'git' processes are still running. Perform this step even after exiting or ending the InfraWorks 360 desktop process.)

6. Browse to the two hidden ***.git*** file folders in the project directory [you may need to turn on hidden files/folders in Windows Settings].
   Project files location is ***C:\Users*current user*\Documents\Autodesk InfraWorks Models*.***
   The 2 ***.git*** folder locations are:
   …\Autodesk InfraWorks Models\<modelname>.files\.git
   …\Autodesk InfraWorks Models\<modelname>.files\ \unver\.git
7. Check for and delete the ***index.lock*** files in each of the above folders.
   Note**:** If the index file does not contain a ***.lock*** file, do not delete the index files. The ***.lock*** extension only appears when the project crashes. Deleting the Index file will not help in the recovery of your project.
8. Launch InfraWorks 360 and attempt to load your project.

If you still cannot open your project, please reboot your machine again proceed with Workflow 2.

Workflow 2: Remove the InfraWorks database's uncommitted modification history to remove any "toxic" changes that may have occurred.

PLEASE NOTE: The following steps will result in converting the currently open proposal/master to be the master, and all other proposals [including the original master] will be lost. You will also lose any history from the project.

Overview
InfraWorks 360 models consist of a SQLite file and a <model>.files folder containing model data and two GIT repositories – one for feature data (the "feature repository") and one for non-feature data (the "meta repository"). Fortunately this results in some data redundancy which can help with recovery. At the lowest level we have the GIT repository which stores a history of all changes to the model data. If the GIT repository is intact then we can restore the model data to any revision in the history. At the next level we have the model data, which corresponds to the active revision of the GIT repository. If any model data files are corrupt then we can use the GIT repository to recover them. Finally we have the SQLite file, which is simply a SQLite version of the feature model data. If the SQLite is corrupt then it can be recreated from the feature model data.

The general recovery process works like this:

- perform any recovery of the GIT repositories (both feature and meta repositories)
- recreate the model data by checking out valid revisions of the GIT repositories
- perform additional manual cleanup of the model data
- recreate the GIT repositories using the current set of model data
- recreate the SQLite using the feature model data

GIT Repository Recovery

The first step to model recovery is to test the integrity of the GIT repositories. This can be done using the following GIT command (running it from the same folder as the GIT repository). The GIT.exe can be found at ***C:\Program Files\Autodesk\InfraWorks 360\Git\bin***

```
git fsck
```

Here's a sample output showing a database that had numerous errors in both its feature and meta repositories:

```
bad sha1 file: .git/objects/0d/3ff31b1e959df784193fb70eafc792e0e2f564 (1)
...
bad sha1 file: .git/objects/ff/44e35a18192359d3f9ed60bec784f2f4f04d16 (1)
Checking object directories: 100% (256/256), done.
broken link from    tree 476c22ab9729b8415f1fb0f840b9fa467a1b6477
              to    blob 5cc368661cf59648b69af6bf9a4f985f8bcb683e
broken link from    tree f96bf89d74650caeeb9cdf2078a5aecdf15897b3
              to    blob f0e26b40d69872ca2f19a372cae642f127429f63
dangling commit 7be0ee3c883864cf945cc67231933a6613556192
missing blob f0e26b40d69872ca2f19a372cae642f127429f63
missing blob 5cc368661cf59648b69af6bf9a4f985f8bcb683e
dangling commit 0186cd4503e40af6d08f968e758835a00411b676
dangling blob 25877e481b9ae3d875273de7908dba21924fe56b
missing blob f04e699e1c0a48ae4623657eac9fa7bdcb766632
...
```

The bad sha1 files are files ending with " (1)". These are duplicates of existing files, so no data was lost and they can be ignored. Dangling commits / blobs should also not cause any data loss. Missing blobs, however, are problematic and the only way to recover these is to get access to an earlier / uncorrupted copy of the model that has these files. But even if you can't gain access to these, it might still be possible to recover an older revision of the model data if it doesn't depend on any of the missing blobs.

Recreate GIT Repositories

Assuming the model data was successfully recreated, the next step for recovery is to recreate the GIT repositories. In some cases this step can be skipped. In particular, if the current GIT repositories have no corruption and if no attempts were made to upgrade the model then the current repositories should work fine. But if there was corruption or if you reset model data to a revision other than the latest commit then the repositories should be recreated. The steps to recreate a GIT repository are straightforward:

- change to the directory containing the .git folder for the repository you wish to recreate
- delete the .git folder
- run the following commands:

```
git init
git add -all
git commit -m "Recreate repository"
```

Note that the **config** commands ensure that the repository is configured the way InfraWorks 360 expects.

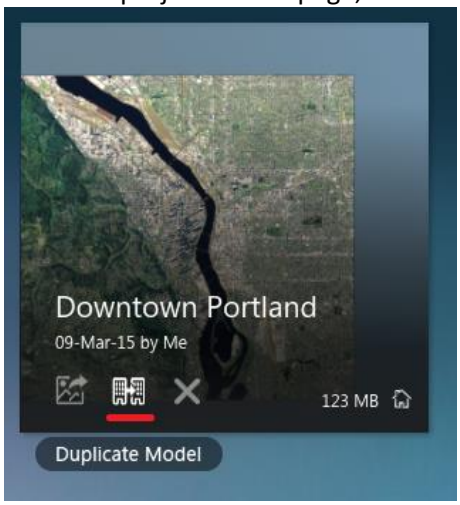
Recreate SQLite

Now that the model data and GIT repositories have been recreated, the final recovery step is to recreate the SQLite file. As mentioned earlier, the SQLite file stores a SQLite database version of the feature model data. If you made any corrections to the feature model data (e.g. by resetting the branch head) then the SQLite file may no longer be in sync with that data, and you should recreate the SQLite file.

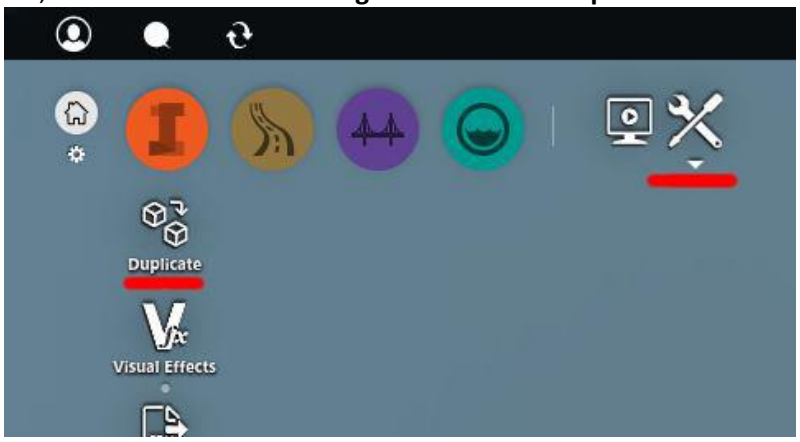
Note: GIT is using internally and so might be subject to change in the future

If you are still able to open the project, you can also try to 'refresh the model' by creating a copy of your project.

1. From the project launch page, find the **Duplicate Model** option:



2. OR, once in the model: **Settings and Utilities> Duplicate**



Improving Model Performance

Occasionally**,** a user may experience drastic performance issues when opening, creating or regeneration a model or scenario where it may takes up to several minutes for the model or scenario to create, open or for the regeneration to complete. While there may be various reasons for this, some of the most common causes are:

1. The excessive use of coverage areas in the model which have a lot of vertices.

    - Review the number of vertices of the data used to create the coverage features
    - Perform the following steps to speed up the model regeneration:
        - Select all the **Coverage Areas** using the Model Explorer
        - Right mouse click and **Select All**. This selects all the imported and sketched coverage areas.
        - Right click in the canvas and select **Properties**
        - In the **Geometry** section enter a value for **0.075m** in the **Generalization** field.
        - Click **Update** to apply the change.

          Note: The generalization simplifies the polygons. You may now see some sharper points appearing on the coverage curves. You can experiment with this number until you get a representation that is acceptable.

        - The model should now regenerate much faster.

2. Large amount of tree objects being used in the model to display large treed areas. These tree areas are challenging for InfraWorks 360 to render due to the sheer number of trees that are necessary to make the model look appropriately irregular and dense.

    - Replace the regular InfraWorks 360 tree objects with a low polygon tree 3D model which uses an image based object and has been seen to perform well in these situation as it reduces the polygon count to speed up performance. A sample 3DS Max (.3ds) low polygon tree model is provided at **…*CI10693*\Data\Low Polygon Tree Model**

3. **Optimizing a Model or Scenarios for Mobile/Web Use**

Due to physical and network bandwidth limitations it is not practical to publish a complete, large model from the InfraWorks 360 desktop environment to the InfraWorks 360 app or web browser.

The best practice to publish models for online viewing is to define scenarios for areas of interest and publish these. The scenarios should not cover the whole model if it is large, but embrace the areas containing the preliminary designs. This approach ensures that users get a smooth and practical viewing experience on InfraWorks 360 web and mobile app.

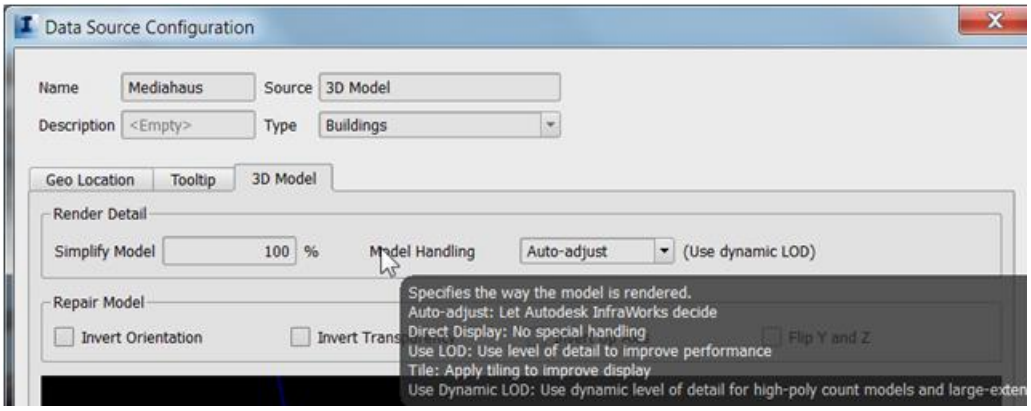
Potential Issues

If a scenario does not publish successfully, its status will display 'problem'. Here are possible reasons why this could have happened, mostly related to scenario size and complexity:

- Size of Area of Interest may be too large. Change it to a smaller area in the Scenario Editor.
- There may be too many 3D models (such as Revit or 3ds models) in the scenario.
- The model may have too many complex 3D models significantly increasing the mesh complexity.
- The Tile or Texture Size values may be too high.

Potential Solutions

- Update to the latest InfraWorks 360 version. Multiple improvements have been added over the last year that may significantly improve performance.
- Limit the Area of Interest to what is really needed to be reviewed for your project.
  - Only use the "entire model" option for the Area of Interest of your scenario with precaution, especially for large models
- While InfraWorks 360 native features like buildings or roads are displayed with high performance, complex 3D models slow it down.
  - Limit the number of 3D models in the scenario. For example, do not include a large number of 3D models of buildings, trees or cars if it is not needed.
- Reduce the complexity of included 3D models (mesh complexity). You can do that upfront in Autodesk Revit or Autodesk 3ds Max, and during import to InfraWorks 360 with the options in the Data Configuration dialog shown below.

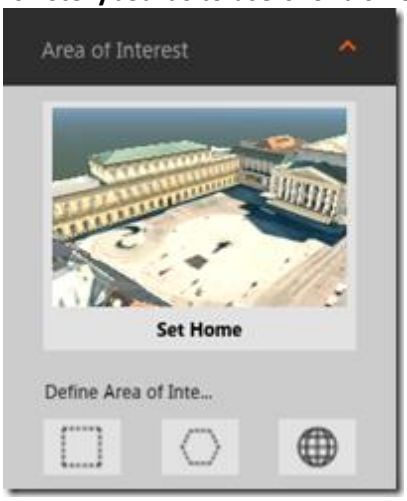


Example of a Revit model (9MB):
By setting the **Simplify Model** percentage to 50%, you can quickly load the model in web/mobile viewer without losing any important components of the building.

Figure 1: Not much information lost by simplifying the model at importing by 50%

General Tips for Scenarios Creation and Viewing in InfraWorks 360 Viewer

- Switching between proposals can be realized by publishing a scenario for each proposal
- Tooltips are available for the POI (Point of Interest) layer.
- Additionally you can publish **storyboards** to users for a smooth 'fly through' your scenario



- Older scenarios (prior to InfraWorks 2014 R4) should be republished with the latest InfraWorks 360 version to get thumbnails and storyboards in the viewer
- To get a smooth navigation experience the LOD setting in the Web browser jumps back to a lower level if the scenario is too complex or too large. A reduction of complexity in the

model is necessary before publishing. The default option for LOD is medium. Higher LODs are only recommended for smaller scenarios.

- If you do not want to simplify your imported 3D models in your desktop model, create a proposal and simplify them there.

For Experts: Hidden Options to Optimize Scenario Publishing

You can optimize the scenario by editing some settings in the **.json** file of the scenario. It is recommended to create a backup copy of that file before making any changes.

1. Close your model
2. Determine the location of the model on the local system
3. Go to the repository "***unver/Scenarios/***"
4. Open the **.json** file with a text editor tool
5. Edit and save your changes (see the available options below)
6. Open the model
7. Sync it to the cloud

You can edit the following options:

- "Level of details"
  - Options: Low/Medium/High
  - Set to Medium by default
- "Tile size"
  - Displayed in meters
  - Default = -1
    - -1 is an option to ask the scenario generator to automatically calculate the best-fit tile-size based on the extent of the scenario area of interest (AOI). This can avoid situations where it takes more than an hour to generating a large-scaled scenario with small tile-size. If the ratio of the scenario extents to **Tile Size** is very large, the scenario generator may fail by exceeding a timeout condition
    - The optimized value for the scenario tile size will be auto-calculated. For numbers larger than 0, it's the real tile size. This value should not go lower than 200.

- "Texture size"
  - Improves the quality of the rendering
  - Defines the size of textures in pixels and influences the quality of textures but does not influence rendering time.

- "Ambient occlusion"
  - Options: true/false
  - Ambient occlusion for terrain and adds rendering time if set to true.