Project Boulder for InfraWorks 360 is a free limited technology preview that matches the simplicity of two-dimensional (2D) flood simulations together with the immersive model. As collaborative effort between Autodesk and Hydronia, the InfraWorks 360 user is provided the ability to simulate, visualize, and animate 2D flood events directly in the Autodesk InfraWorks 360 model environment while utilizing the Hydronia RiverFlow2D computation model.

Features:

- Create simulation area & model inflow and outflow
- Automatic creation of simplified surface mesh
- Establish a uniform manning's n value for the model
- Determine the simulation time and increment
- Utilize the Hydronia RiverFlow2D model to perform the simulations
- Animate the resulting flood surface by elevation, depth and velocity in the model.

Feedback on this preview is a critical element of the Autodesk© //Labs_ process. Your feedback drives what features graduate, and the priority of new or expanded functionality in the eventual product.

Getting Started

1. Download and Install Project Boulder from the Autodesk© //Labs_ website on a Windows 7 or 8 PC. This installs as a plug-in available on the Autodesk© InfraWorks 360 2016 release of the product. (Version 16.1)
2. Download and Install Hydronia RiverFlow2D from the Hydronia InfraWorks website. Follow the links on the Autodesk© //Labs_ Project Boulder website to the Hydronia website and follow their instructions for the installation of RiverFlow2D.
3. Once completed, Launch InfraWorks 360.
4. Open a model. From any tool icon in the Intelligent Tools– expand it and select the Analysis and look for Flood Simulation icons.
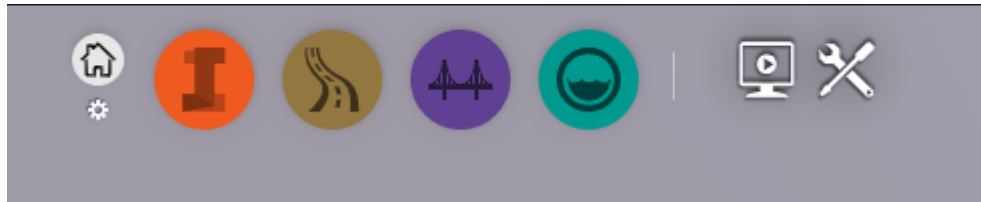
Figure 1 InfraWorks 360 Intelligent Tools

Launching Flood Simulation

5. Once the Tool is opens, select the Analysis action.

Figure 2 Intelligent Tool Actions - Analysis

6. The Project Boulder consists of two commands, each command found under the Intelligent Tools > Analysis action as shown in Figure 1. The first Icon launches begins the process and the second icon is allows the user to turn off and on a snapshot of the simulation. Both commands will be discussed later in this document.

Figure 3 Project Boulder commands

7. To perform a flood simulation using Hydronia RiverFlow2D, click the Flood Simulation icon shown above.

InfraWorks 360 will prompt the user to define a boundary. This boundary limits the size of the simulation study area to a specific region of interest within the InfraWorks 360 model. This area is where InfraWorks will re-mesh the terrain surface, assign the inflow and outflow boundaries and pass this information to RiverFlow2D for computations.

Define Study Area

8. Click and draw a polygon boundary around the area of interest. A double-click will complete the polygon.
9. Once the Boundary is complete, define the inflow boundary condition location using a click-drag-click approach along the originally defined polygon simulation boundary.

AUTODESK.

10. Repeat this step to define the outflow location boundary.

Each boundary condition can be modified by simply selecting the carpet and adjusting the end rollers by selecting and dragging to the desired locations.

Figure 4 Inflow and Outflow Boundary Conditions ready for edit

AUTODESK.

Simulation Conditions

11. The Flood Simulation Dialog box allows the user to specify a single steady flow that enters the model, a uniform Manning's n value and the simulation time and interval values.
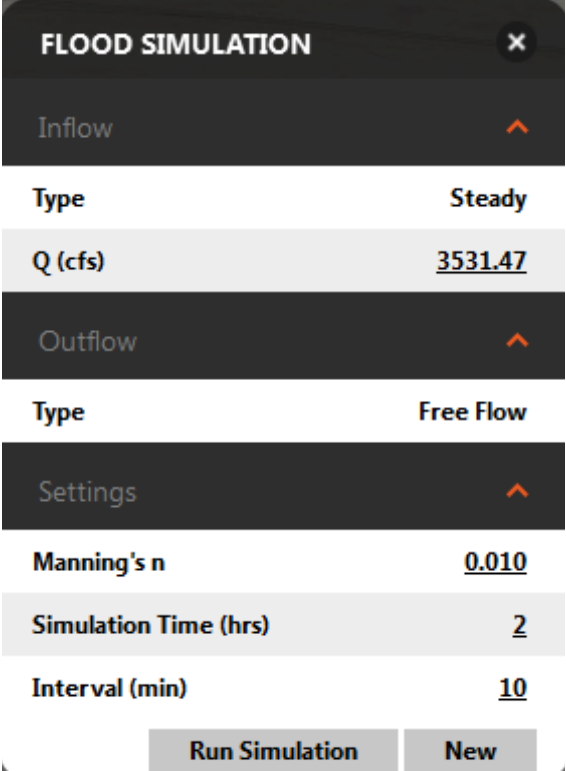

Figure 5 Simulation Settings

12. The Run Simulation button launches RiverFlow2D using the simulation settings above together with a generalization of terrain mesh designed to keep the simulation times quick and the animation manageable.
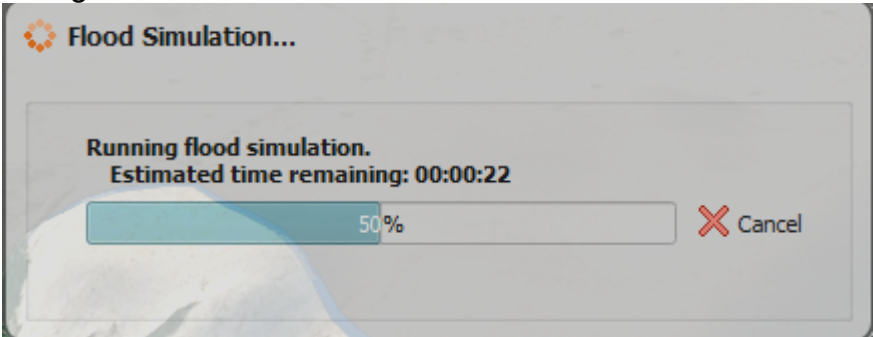

Figure 6 Simulation running

13. The Simulation runtime estimation becomes a function of simulation time, time interval and the size of the model. The model runs with a minimum of 2000 triangles, but a significantly large model utilizes model triangles with a maximum triangle edge of about 20m.

AUTODESK.

Animation Player

14. Once the simulation is complete, the Animation Player appears. This allows the user to visualize the simulation over the simulation time, at the interval time step using either a Water Surface Elevation, Depth or Velocity theme.
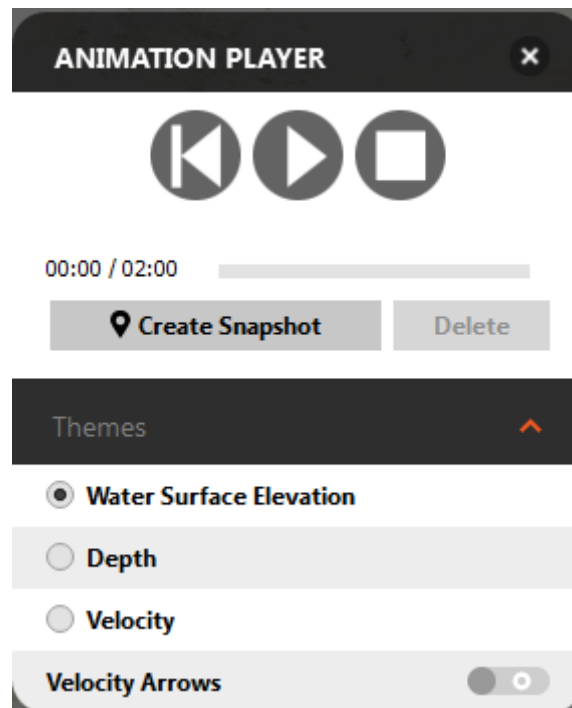

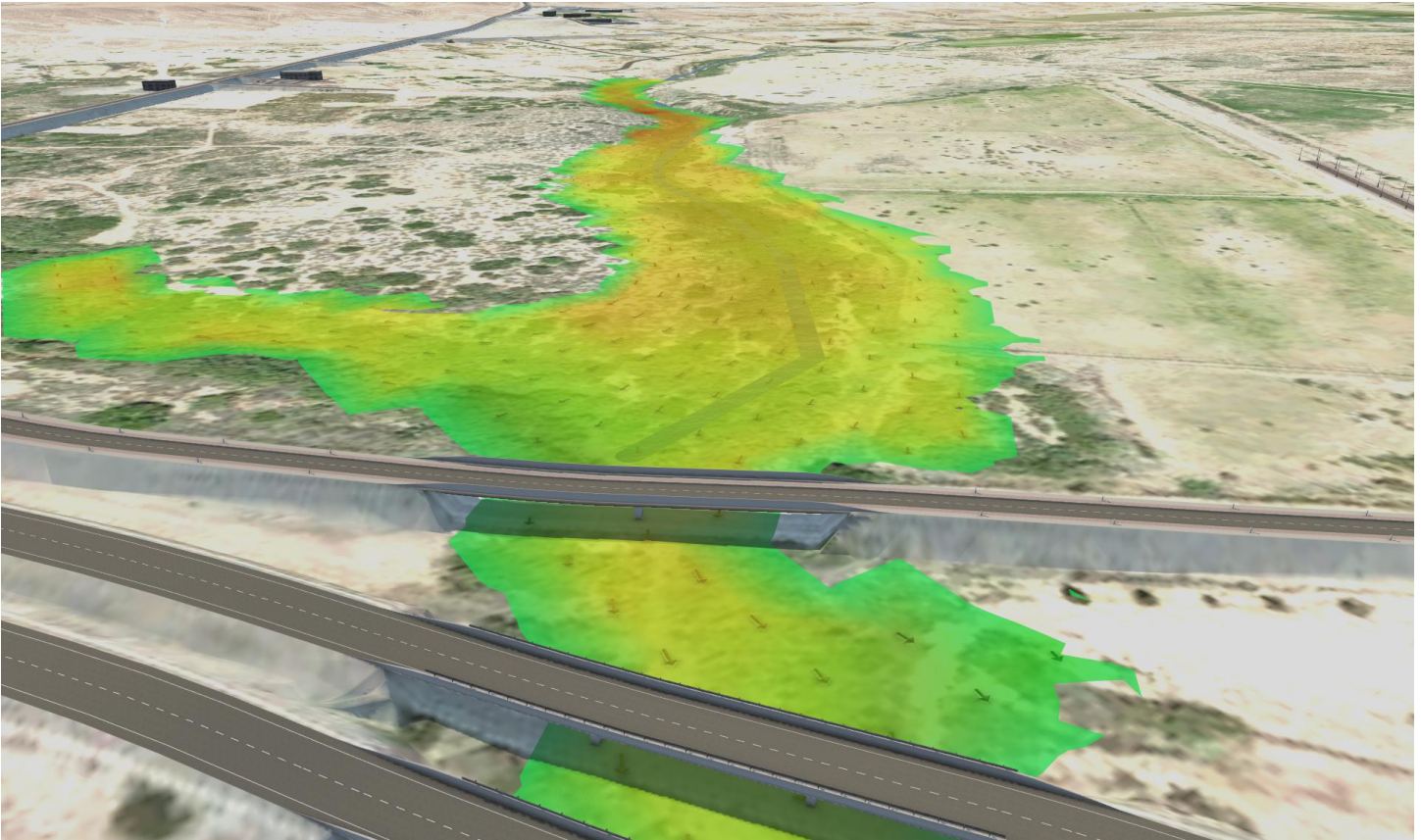**Figure 7 Animation Player & Visual Themes**

Figure 8 Sample Visual

AUTODESK.