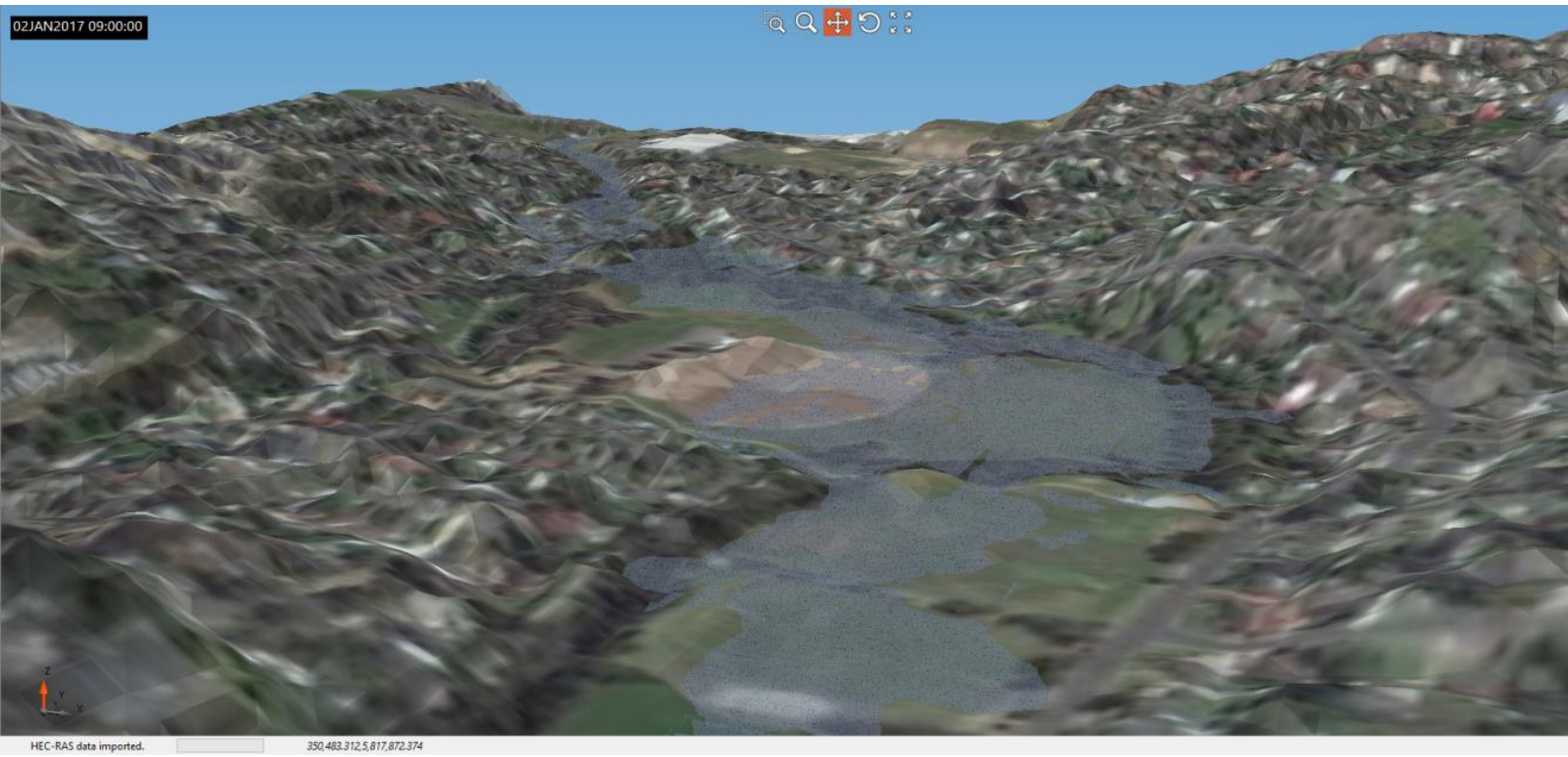
Civil Site Design

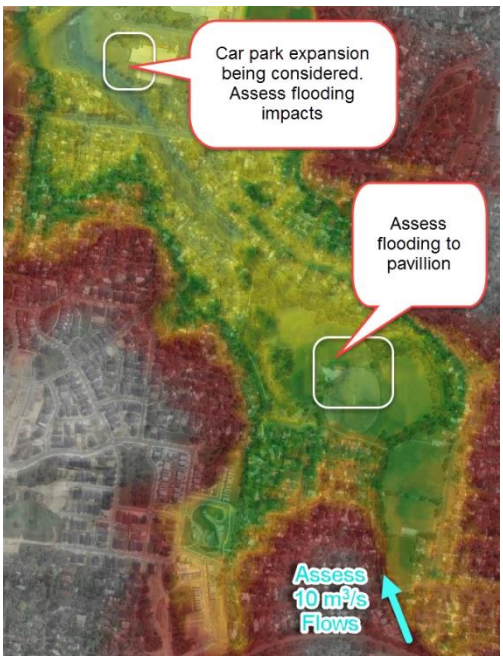
DESIGN • ANALYSE • VISUALISE • DELIVER

HEC-RAS 2D Flood Modelling Tutorial



Civil Site Design and HECRAS 2D Flood Modelling

HECRAS version 5 and later includes functionality to analyse water flows moving across a surface. this is known as 2D flood modelling and provides more accurate modelling of water movement across a surface than 1d (or section based) flood modelling.



If you are using Civil Site Design, you now have functionality to work with HECRAS input and output files via the Export HECRAS 2D and Import HECRAS 2D commands.

For this example, we'll use HECRAS to undertake a 2D flow analysis of an existing creek abutting an existing car park being considered for expansion. The existing creek flow performance is to be maintained following the new works, and enhancement of the creek flow may be required if flooding is occurring downstream.

A peak flow of 10m^3/s must be managed in the creek.

The Tutorial Data Set

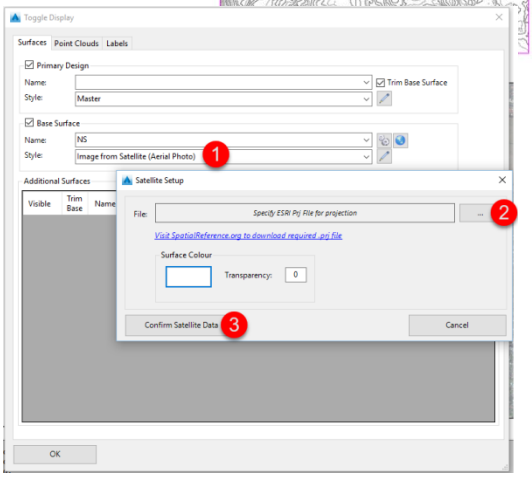
The tutorial data is installed with the software installation to the following location (Note: the HEC-RAS menu must be ticked on for inclusion during install):

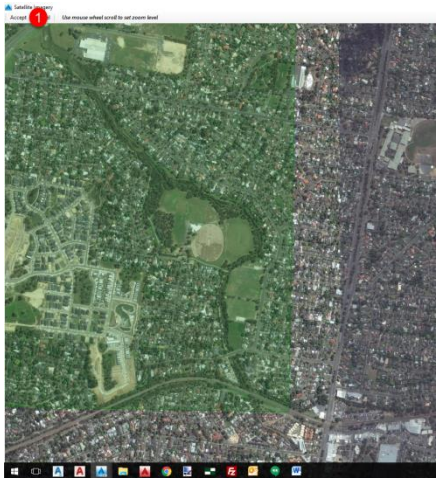
Civil 3D: *C:\CSS Training Data\Civil Site Design\HEC-RAS\HECRAS 2D Tutorial – Start.dwg*
CAD: *C:\CSS Training Data\Civil Site Design\HEC-RAS\HECRAS 2D Tutorial – Start - CAD.dwg*

This drawing file includes a surface and is located in the MGA-55 Coordinate system.

At this stage, there is no surface associated with the data set. This can be remedied using Model Viewer. To do this:

- Start the ribbon command Roads Tab > Select Panel > Model Viewer. This will start Model Viewer and populate the existing surface name into the Toggle Display form
- Under Base Surface, click on the picklist dropdown for Style and select Image from Satellite (Aerial Photo)

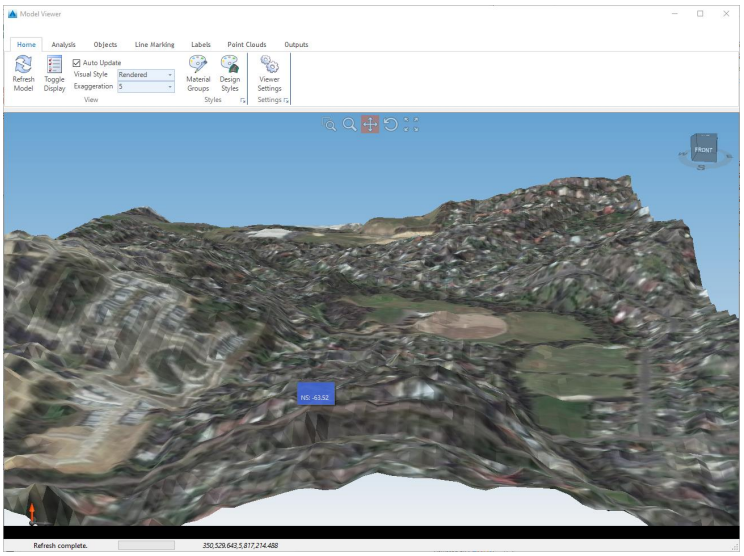




- The Satellite Setup form will display. Firstly click on the … button, select Australia MGA-55.prj from the list and click OK.
- Click on Confirm Satellite Data to see the imagery available under the surface location.
- Roll the mouse to zoom out but 3 steps only (the more zoom out, the less quality) and click the Accept button top left.
- Click OK to exit the Toggle Display form. An image is associated with the surface in Model Viewer.

For AutoCAD and Civil 3D users, an image is also inserted in the drawing.

For BricsCAD users, an image is saved to the –Data folder. When inserted into the drawing it will automatically position itself correctly.



Exporting Terrain Data to HECRAS

The Export HECRAS 2D command will take the combination of your design and existing surfaces and share them across to HECRAS as a single geotiff file. The goetiff describes the surface extents and elevations for HECRAS. The command can be run directly from the ribbon interface, or can be run from inside Model Viewer.

If you opt to run the command from the CAD ribbon interface, the Toggle Display form will display for you to confirm what surface data to bring across to HECRAS - you can opt to just include the existing surface conditions, or you could send across a combined surface containing the existing surface as well as your design surfaces. This would enable you to analyse pre and post development conditions. If you run the command from inside Model Viewer, you get the benefit of directly reviewing the surface before export across to HECRAS.

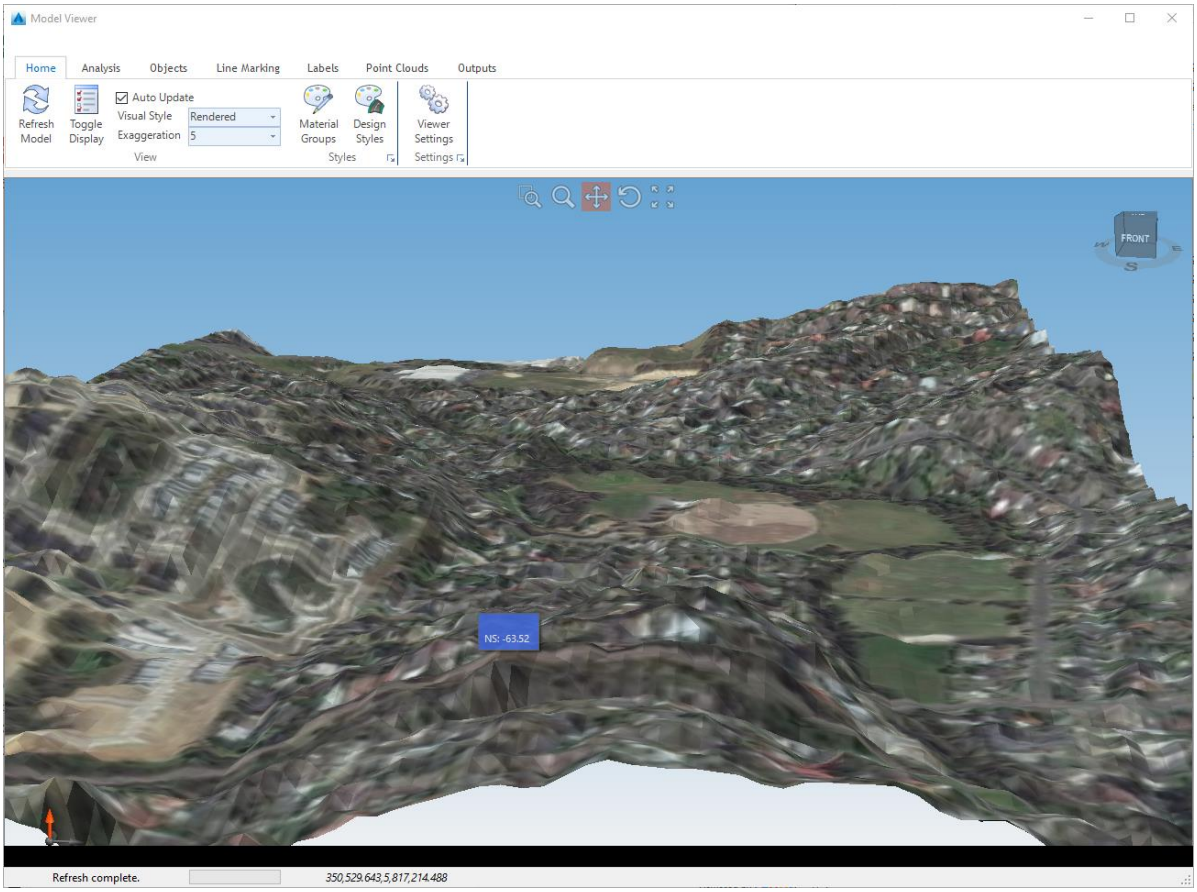
In the Export HECRAS 2D command you need to specify a location and name for the geotiff you are exporting, as well as a grid spacing. Set a grid spacing that matches the accuracy of the data you have, to ensure you don't lose information and, conversely, you don't include unnecessary data.

Once you have a geotiff file you can use this to generate terrain data in HECRAS.

Steps:

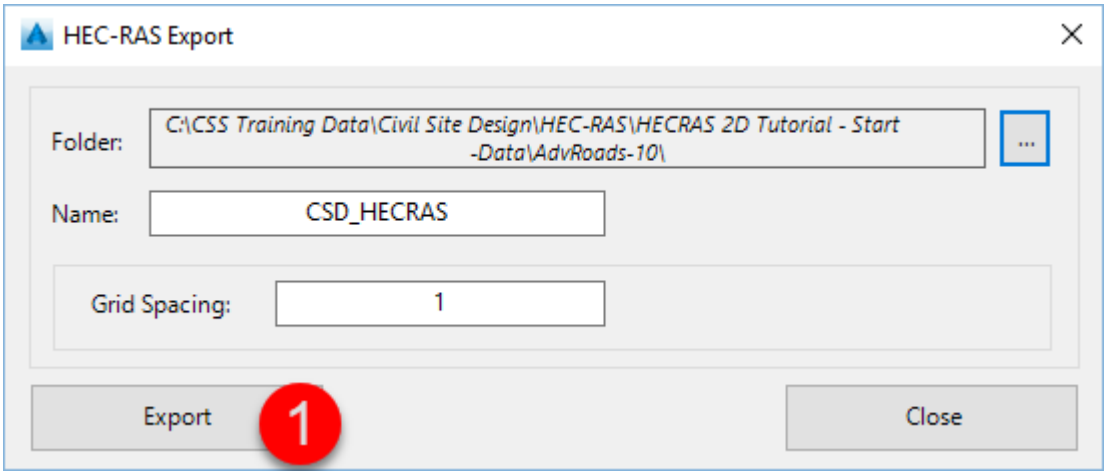
1. If you closed Model Viewer, click on the ribbon command Roads Tab > Select Panel > Model Viewer. This will start Model Viewer.

2. If required, confirm the settings via Toggle Display. If the previous steps are complete, the surface and image required to be analysed in HECRAS should already by shown.



Note: in the above image the exaggeration has been set to 5 to accentuate the shape of the terrain. Exaggeration must be set to 1 before exporting

3. Click on the Analyse Tab > HECRAS 2D Panel >  Export button



In the Export form you confirm the location, name and grid size of the file to export.

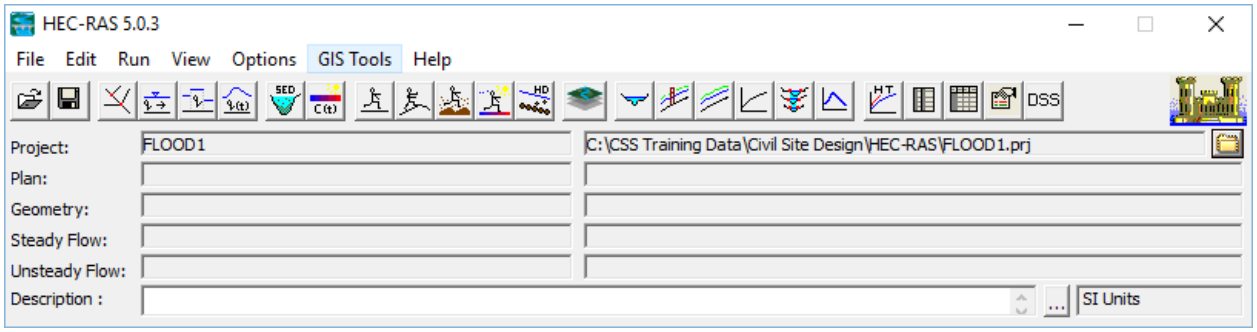
4. Leave the Grid Spacing to the default of 1
5. Note the path to the output HECRAS file, then click Export to create a HECRAS input file.

Import and Analysis in HECRAS

After starting your HECRAS project and establishing the units and setup, you need to open RAS Mapper and create a terrain. You will firstly set up a projection, or coordinate system, for your data.

HECRAS Project Setup

6. Click on File > New. Select a location and name for the new project.



7. Click on Options > Unit System to confirm the units (this example is using Metric units).

HECRAS RAS Mapper

8. From the HEC-RAS interface, select GIS Tools > RAS Mapper… or else click on the RAS Mapper button.

The RAS Mapper form will display with a menu at the top and a list of layers along the side. The first task is to populate the Terrain layer using the .tif created by Civil Site Design. Optionally, a Map Layer can be included for the aerial photo.

Assigning a Projection

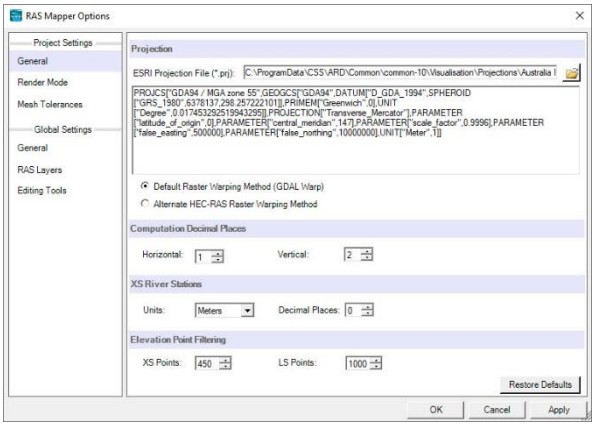
Before creating a terrain, a projection must be set. This is controlled by a projection file, extenslon .prj. Some typical Australian and other country specific .prj files are contained in the Civil Site Design Settings folder, in the Visualisation and then Projections folders. There is also a website where you can generate .prj files by typing in your location.

9. From the Ras Mapper form, click on Tools > Set Projection for Project.

For this example, a .prj has been provided in the same directory as the tutorial drawing: C:\CSS Training Data\Civil Site Design\HEC-RAS

10. Navigate to the above folder location and open Australia MGA-55.prj. This assigns the Projection.

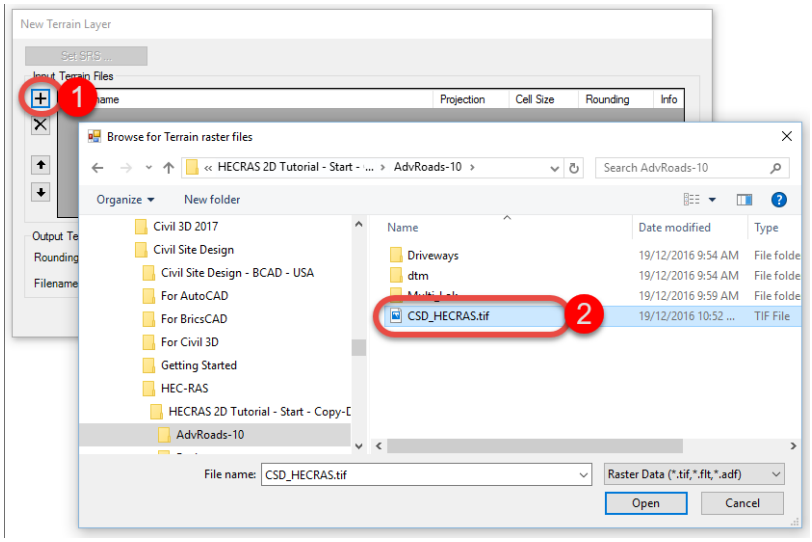
    After selecting the .prj file click OK to apply the projection.



Adding a Terrain

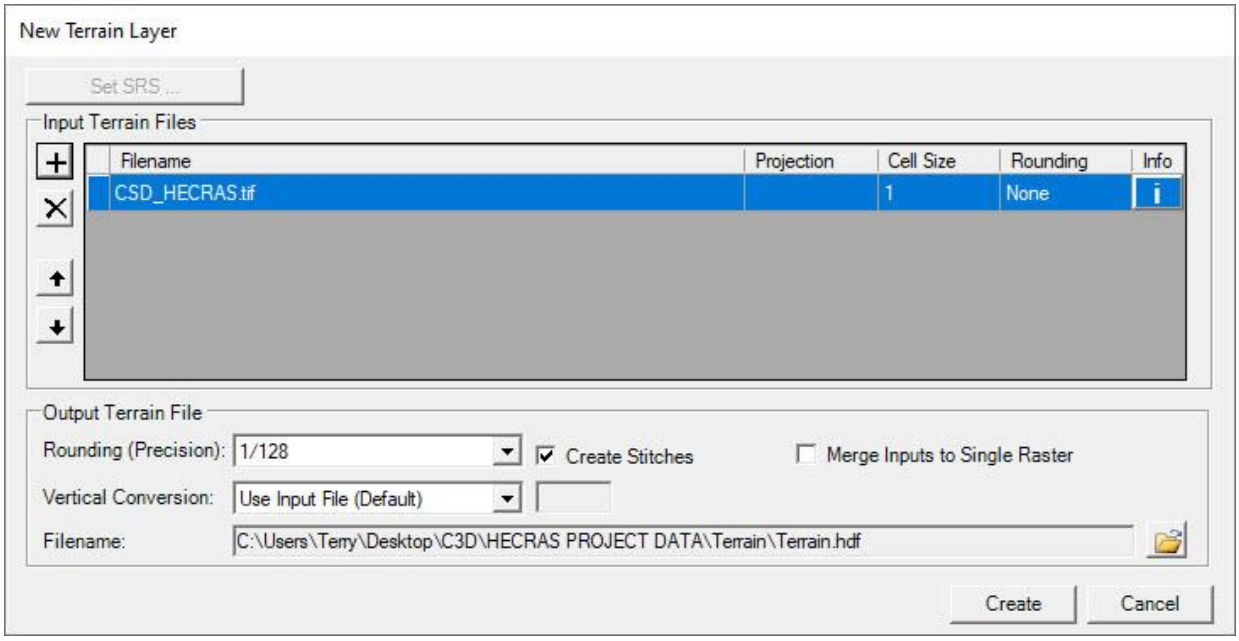
When you open the Terrain form and create a new Terrain, you will have an opportunity to load a terrain file - here you can use the exported file from Civil Site Design to generate the terrain data for HECRAS.

11. Click on Tools > New Terrain

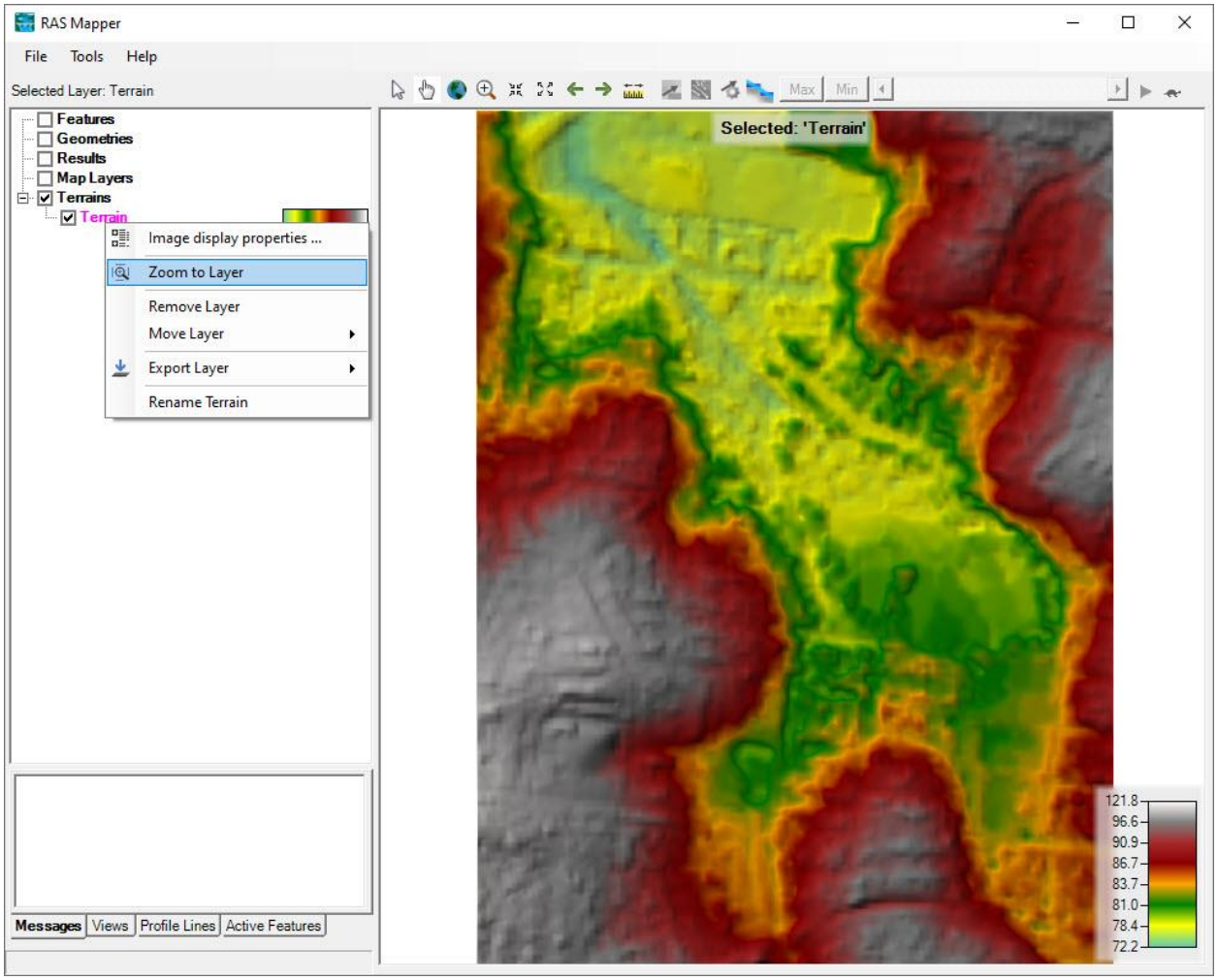


12. Click on + to add a new data source for the terrain. Navigate to the output location of the .tif and open that file

This data will be used to create a Terrain – in HECRAS the terrain is saved as a file with extension .hdf.

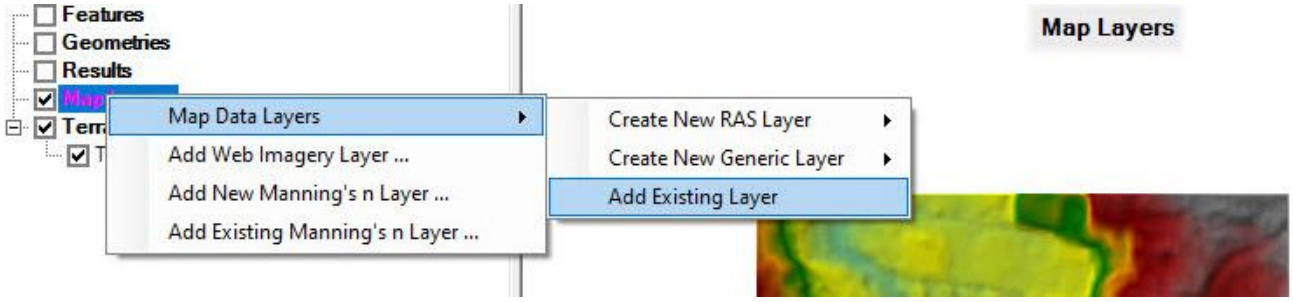


13. Click on Create to import the Terrain.
14. Note under the Terrains layer is a new layer. Make sure that Terrains is ticked on, then right click on Terrain and choose Zoom to Extents to see the full extents of the terrain model in RAS Mapper.

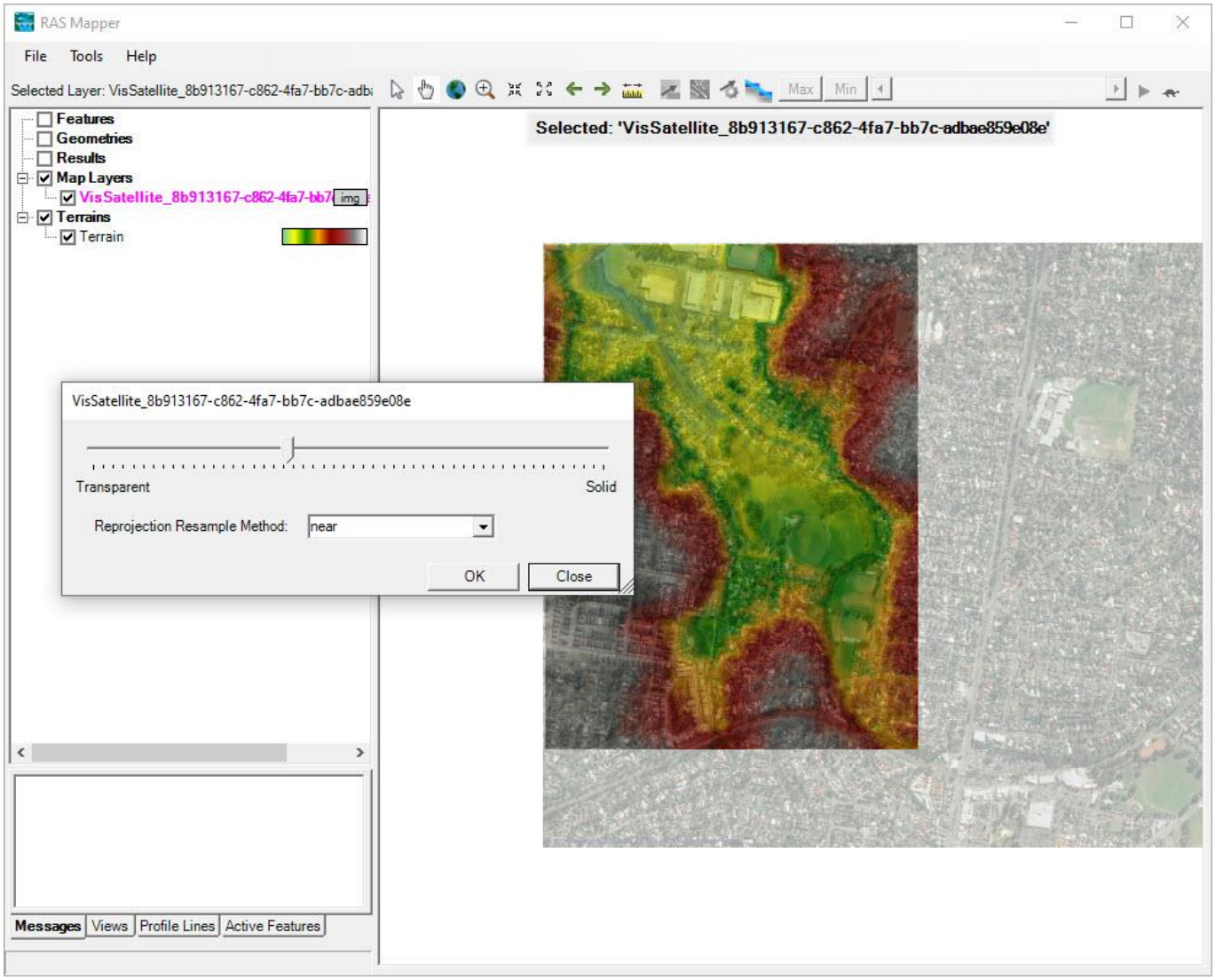


Adding an Aerial Photo

With RAS Mapper you can also include imagery and other geospatial data.



15. Right click on Map Layers and follow the Map Data Layers flyout to select Add Existing Layer. When prompted to select a file, use the file type dropdown at the bottom right to select Other (*.*). Selecting a .jpg file will then insert it into HECRAS so long as it is accompanied by a world file (.jgw file contains the spatial location for a .jpg file).
16. Select the image file output during step 5 above or *C:\CSS Training Data\Civil Site Design\HEC-RAS\VisSatellite.jpg*
    Tick on the image to display it in RAS Mapper. Right click on the image name to edit the Image Display Properties – a transparency can be assigned here.



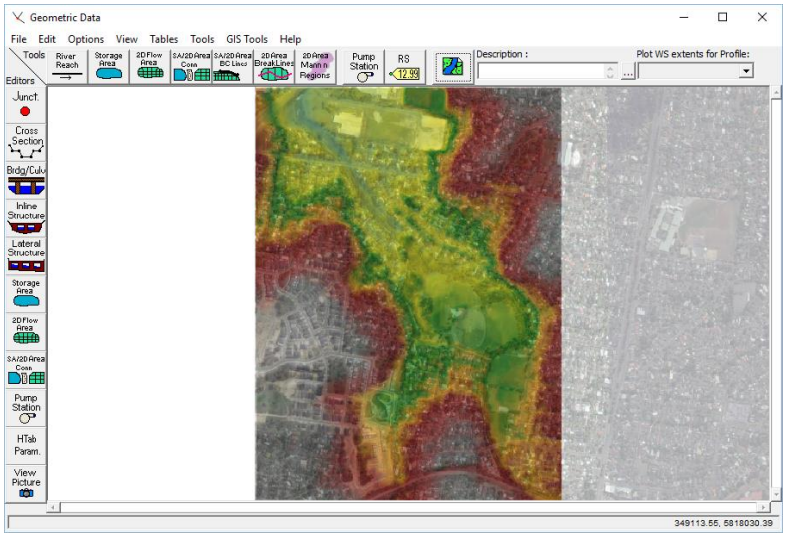
After importing data into RAS Mapper, it will become available in the HECRAS Geometry Editor. In here you can assign flood data along the boundaries, establish the model grid size for flood analysis, edit the surface breaklines, assign different surface Manning's n values and more.

Close RAS Mapper.

Creating the 2D Flow Area

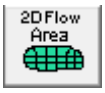
17. From the HEC-RAS interface, click on View/Edit Geometric Data 

A new form will display entitled Geometric Data. The surface will initially display with the colours representing elevations. To show the aerial photo, click on  and tick on to display the Map Layer you added.

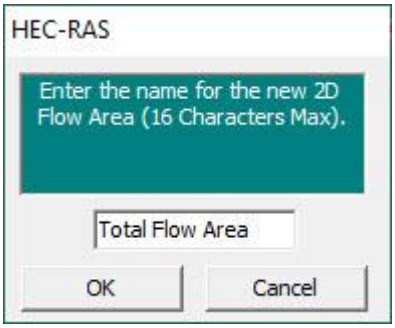


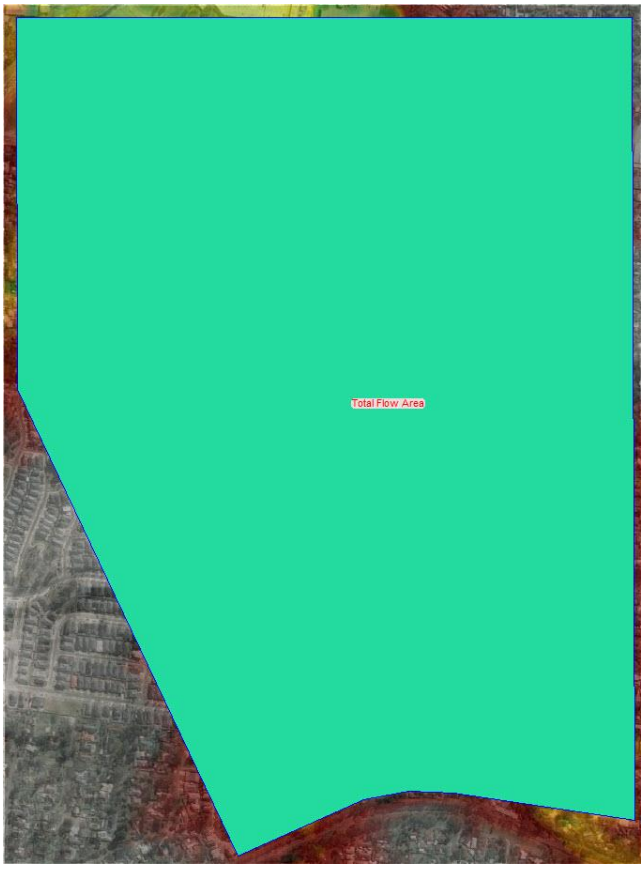
The linework in your original drawing can be included in the Geometric data as a layer, after exporting it out from the drawing in an appropriate format for HECRAS.

Now to create a 2D flow area for analysis. To keep it simple, select the extents of the terrain as the boundary. This is a relatively well defined basin, so the extents of the boundary are typically much higher than the creek (excepting the inflow and outflow locations north and south). When you select the area you can establish a calculation mesh across the surface. In our example, a 10m mesh is suitable and matches with the existing surface triangulation.

18. Click on the 2D Flow Area 

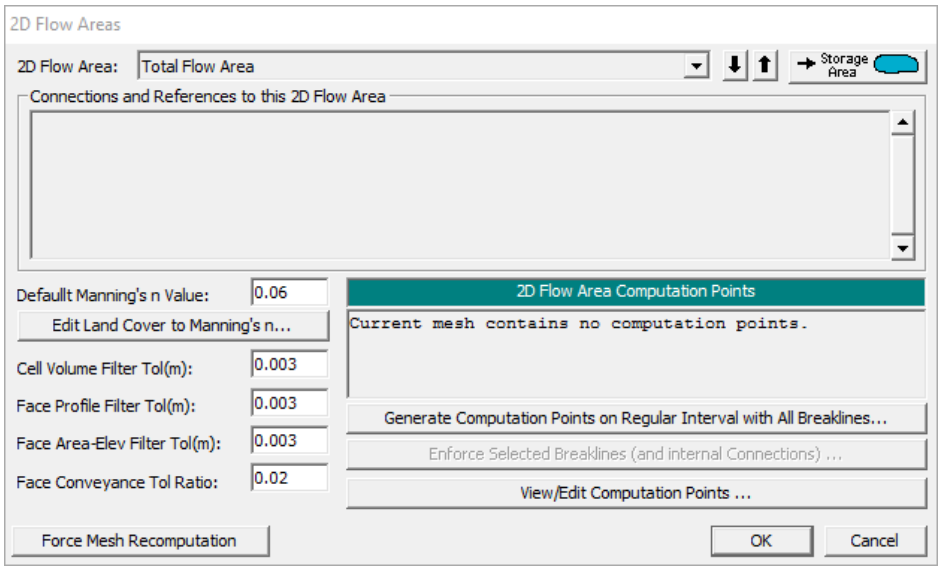
button. Use the left mouse button to sketch the boundaries of a flow area. Double click on the start of the first line drawn to close the area.
19. At the prompt, name the area Total Flow Area and click OK.



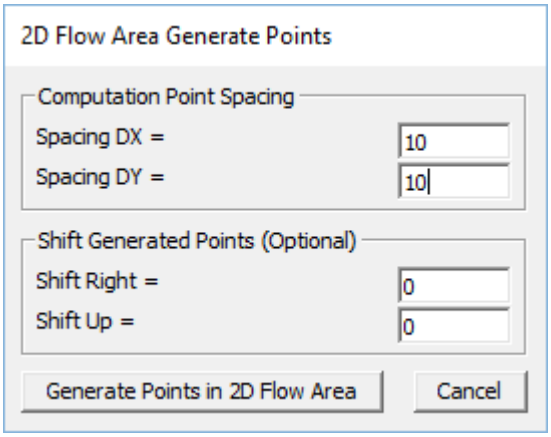


20. Left click on the new area and select Edit 2D Flow Area …

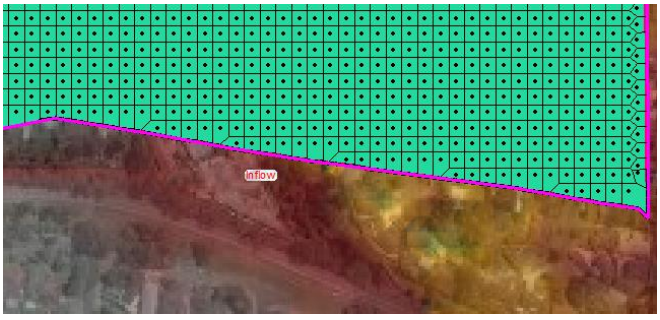
In the 2D Flow Areas form you apply a computational mesh across the surface, as well as establish Manning's n values to apply for the default area and for any other Land Cover areas added.



21. Click on Generate Computation Points on Regular Interval with All Breaklines…
22. Set DX and DY both to 10 and click Generate Points in 2D Flow Area



23. Click OK to close the 2D Flow Areas form.
    The pattern on the 2D flow area in the Geometric Editor will change – zooming in will expose the individual mesh areas. Zoom in to view the mesh shape. If any are red in colour then you will need to use the Edit > Add Points command to add more mesh point or else adjust the boundary shape.



Note: if it is required to edit the outline of the 2D flow area, use the menu command Edit > Move Points/Objects or other edit tool in the Edits menu.

The 2D Area Breaklines 

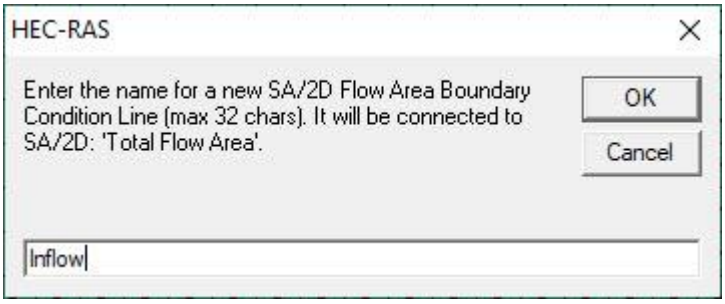
command can be used to incorporate breaklines into the flow area to better represent ridge and valley lines for the surface.

Setting Boundary Conditions (for Incoming and Outgoing flows)

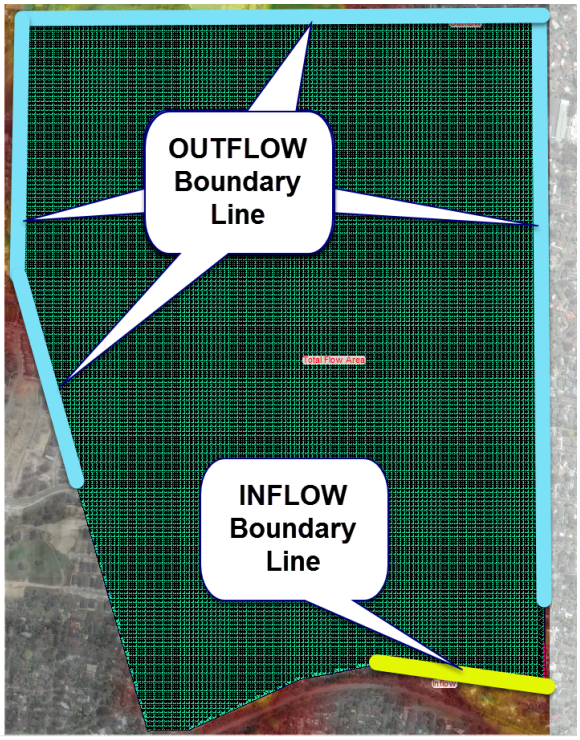
Flow comes from the ponding areas along the southern boundary of the surface, so a boundary condition needs to be set along this edge. We can then assign inflow along this boundary to spill across the surface. The low side of the surface is along the northern boundary, so we can set a boundary condition along there for how water can exit along that boundary.

24. Click on the SA/2D Area BC Lines button 

25. Click <u>inside the 2D Flow Area</u> along the southern boundary to set an incoming flow line. Flow will come in uniformly along this line. Double click to finish the boundary



26. Name the boundary Inflow and click OK to create.
27. Repeat the above 3 steps to create an 'outflow' boundary. Run the boundary just <u>outside the 2D Flow Area</u> along the west, north and eastern boundary where the properties are and name the flow area boundary Outflow. The outputs should look similar to below:



Note that in the geometry editor you can incorporate both 1D and 2D flow zones into a single model for analysis. Additionally, boundary conditions can be set within the 2D area to represent different hydraulic behaviours. After setting up the surface and boundary conditions, you can move onto assigning the flow conditions.

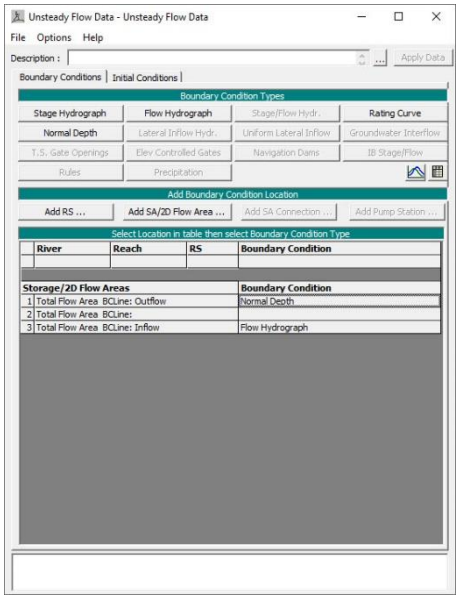
28. Click on File > Save Geometry Data.
29. Name the file Existing Conditions and click OK

Note: Where a boundary is not specified, water is assumed to well up along the boundary as if the boundary edge was a wall.

Establish Flow Conditions along the Boundaries

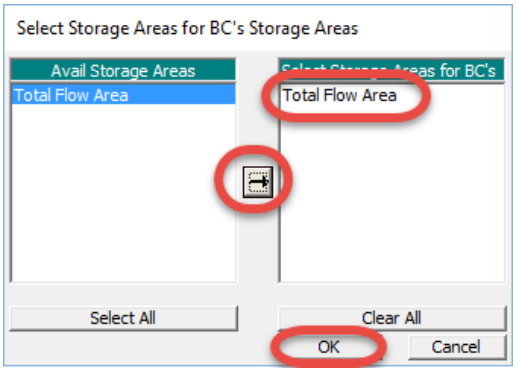
In the Flow Conditions form, we can set an inflow for the Inflow incoming boundary along the southern boundary. In this example, we're working with a peak flow of 10 m^3/s and describing the storm shape as a simple bell type curve. You can copy and paste from spreadsheet input tools, such as Excel, to quickly create the flow information for HEC-RAS. For the outgoing boundary conditions, we set for a normal depth and apply a friction slope to describe how the exiting flow is handled.

30. From the HECRAS menu click on View/Edit Unsteady Flow data 



A new form will display entitled Unsteady Flow Data. The form will list the two Boundary Conditions set for the 2D area.

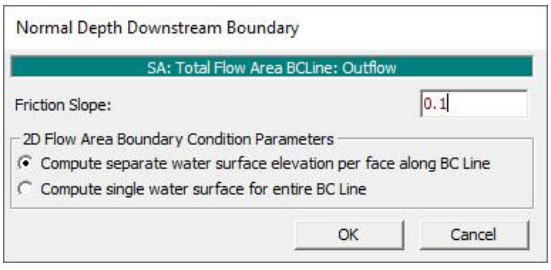
31. Under Add Boundary Condition Location, click on **Add SA/2D Flow Area…** to include the Total Flow area



32. Click on the -> arrow to include the ***Total Flow Area*** for analysis and click OK.

Next, we set the outflow conditions along the northern boundary line we've created.

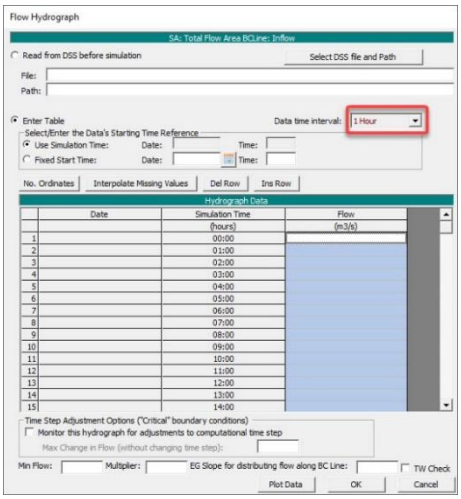
33. Under Storage/2D Flow Area, click the flow area entitled ***Outflow***
34. Click on Normal Depth at the top to establish the outflow conditions along this line
35. Type in a Friction Slope of 0.1 and click OK



The Boundary Condition for the Inflow will be set to Normal Depth.

Now we need to assign incoming flows into the system.

36. Under Storage/2D Flow Area, click on the flow entitled ***Inflow***
37. Click on Flow Hydrograph to assign a time dependent flow to this inflow line

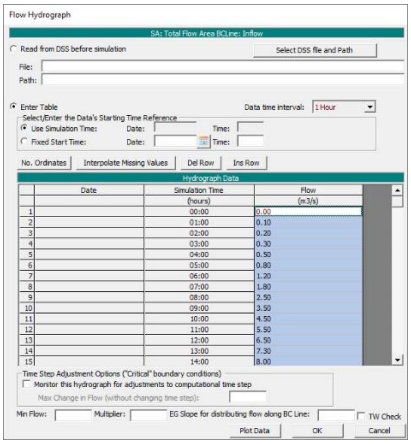


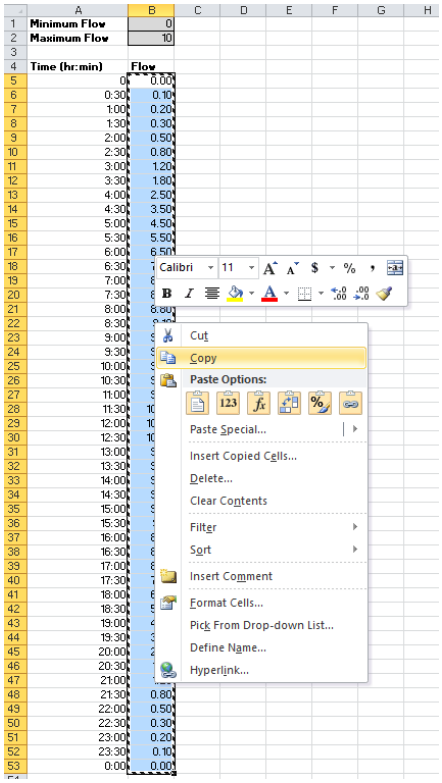
For Data Time Interval, set the value to 1 hr using the pick list

To add flows you can type in a Flow for each time step (0 to 48hrs) and set for a build up to a peak flow of 10m^3/s, however it is often quicker to spreadsheet out the required flow characteristics, copy them from Excel, and use the paste command to paste them into the Flow column.

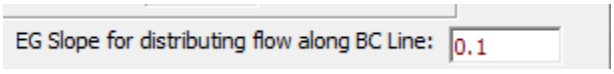
For this purpose, you may wish to open the file Storm Input Calculator.xls file located in the same directory as the source drawing. In this file, you can type in the Minimum and Maximum flow values at the top of the form and it will populate values for 1hr increments.

Set the Maximum Flow to 10m^3/s (or higher if you wish). Then highlight the values in the Flow column and copy to clipboard (Ctrl + C or right click and select Copy). In the Flow Hydrograph form, highlight all Flow cells from 0:00 to 48:00 and press Ctrl+V





38. For EG Slope for distributing flow along BC Line, type in 0.1



39. Click OK to assign the flow conditions to the boundaries specified
40. Click on File > Save Unsteady Flow Data. Name the file Flows and click OK
41. Click on File > Exit to close the flow data form.

Re-establish the Area Conditions

You may receive an error if you move directly to analysing the flows, without first returning to the Geometric Data Editor form and re-establishing the grid calculations to ensure they are correctly saved following the flow assignment. To do this:

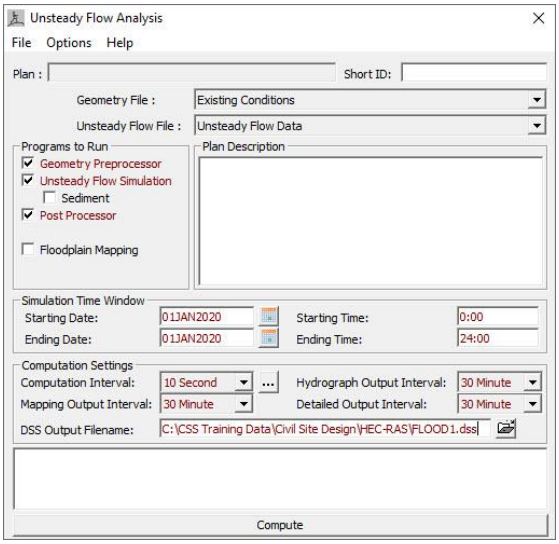
42. From the HEC-RAS interface, click on View/Edit Geometric Data 
43. Click on the 2D area and select Edit 2D Flow Area…
44. In the form that displays click on Generate Computation Points on Regular Interval with All Breaklines…
45. Accept the default spacings you previously applied and click Generate Points in 2D Area. Accept the warning message and click Yes.
46. Click OK to close this form then click on File > Save Geometry Data
47. Close the Geometry Editor form.

Analyse the Flows

When you analyse the system you set up time stamps for calculating flows, reporting times and total analysis duration. RAS Mapper is then used to review the flood extents over time, assess the peak extents, flow depths and flow velocities. As part of this analysis process, HECRAS has created a file to store this storm flow information. Civil Site Design can import this file and visualise it in 3D in Model Viewer.

The Computation Interval is very important – if it is too large or too small you will generate instability in your model or miss important calculation outcomes. Because we are on a steep site with fast flow in sections, a small Computation Interval is required – we'll be going for 3 seconds.

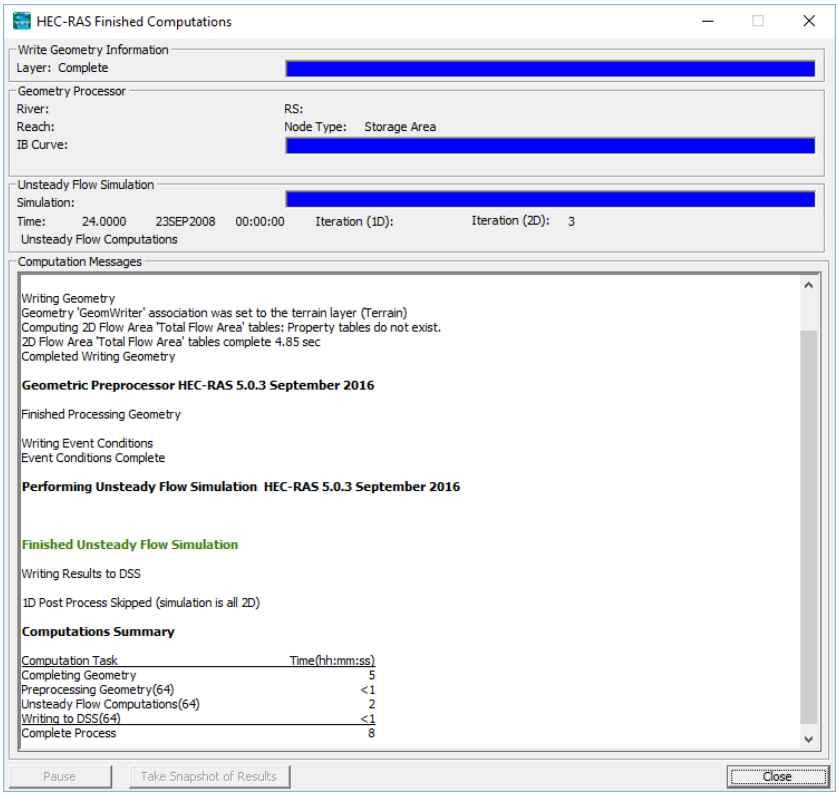
48. From the HECRAS menu click on Perform and Unsteady Flow Simulation  button.



49. Fill out the form as follows:
    a. Tick ON
        i. Geometry Preprocessor
        ii. Unsteady Flow Simulation
        iii. Post Processor
    b. Set the Simulation Time Window
        i. Pick a Starting Date (default selection shown) and Ending Date (same as starting)
        ii. Set the starting time to 0:00
        iii. Set the ending time to 24:00 (for 48hrs of flow calculation time, you would also change the Ending Date to be one day past the Starting Date)

c. Set the Computational Settings
    i. Set the Computation Interval (how often it calculates levels) to 10 seconds
    ii. Set the Hydrograph Output Interval to 30 minutes
    iii. Set the Mapping Output Interval (time increments displayed in RAS Mapper) to 30 minutes
    iv. Set the Detailed Output Interval to 30 minutes

50. Click on Compute



If you receive any information in red text, it is likely that some data has not been saved, data hasn't been fully added or there is an error in the mesh or data input.
Review the above steps, ensuring you (again) Save Geometry Data and Save Unsteady Flow Data, and try computing again.

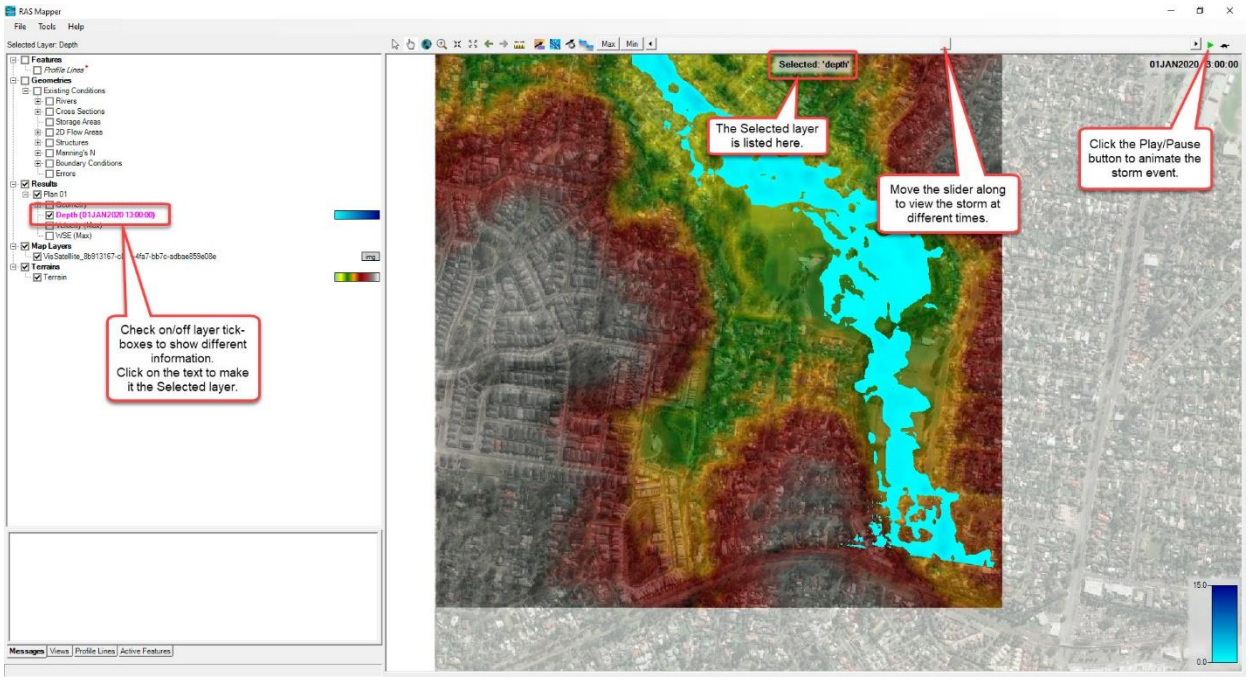
You'll have to wait a little while as it processes – 4-6 minutes for a 24 hr calculation period.

51. Close the computations form
52. Click on File > Save Plan to save the applied flow analysis inputs
53. Click Close

You can view the results directly inside HECRAS via RAS Mapper. As well, you can generate lines for the flood extents and other flow outcomes and export this to a geospatial file format suitable for import using the Map 3D tools in Civil 3D.

To check out the results in RAS Mapper, do the following:

From the HECRAS interface, click on the RAS Mapper button.



Tick on the Depth layer to show the depth of water. Click on the Depth (Max) text to make it the Selected layer and use the slider at the top to change the flow time to see development of flow over time. The play button will animate the flow.

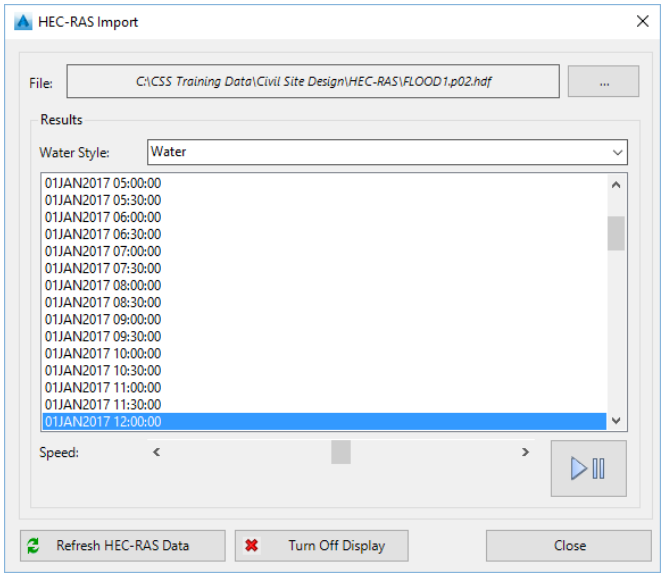
3D Review and Analysis using Model Viewer

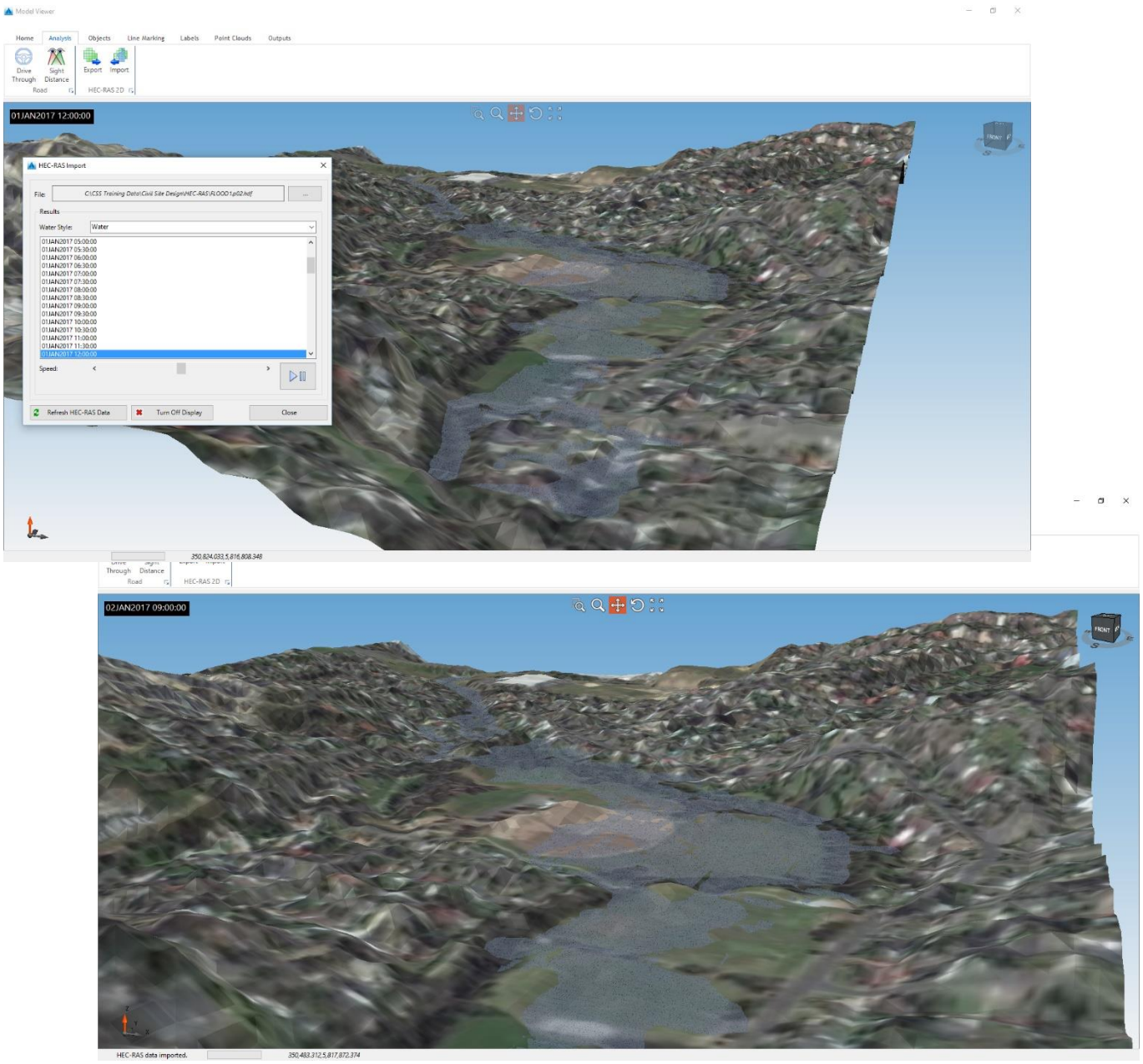
The Import HECRAS 2D command lets you import the HECRAS output file (which is a file with extension p##.hdf). After running the command and selecting the file, a new form displays with the calculated time stamps included.

54. Open Model Viewer
55. Click on Analyse > Import HEC-RAS
56. Click on the ... button to select an output file. Output files have the name of the HECRAS project file with .p##.hdf, where ## is a 2 digit number. Locate the project folder you saved the HEC-RAS analysis to and you will find the output .hdf file/s there. Select and Open.

57. Select a time stamp and see the extents of flooding at that time.

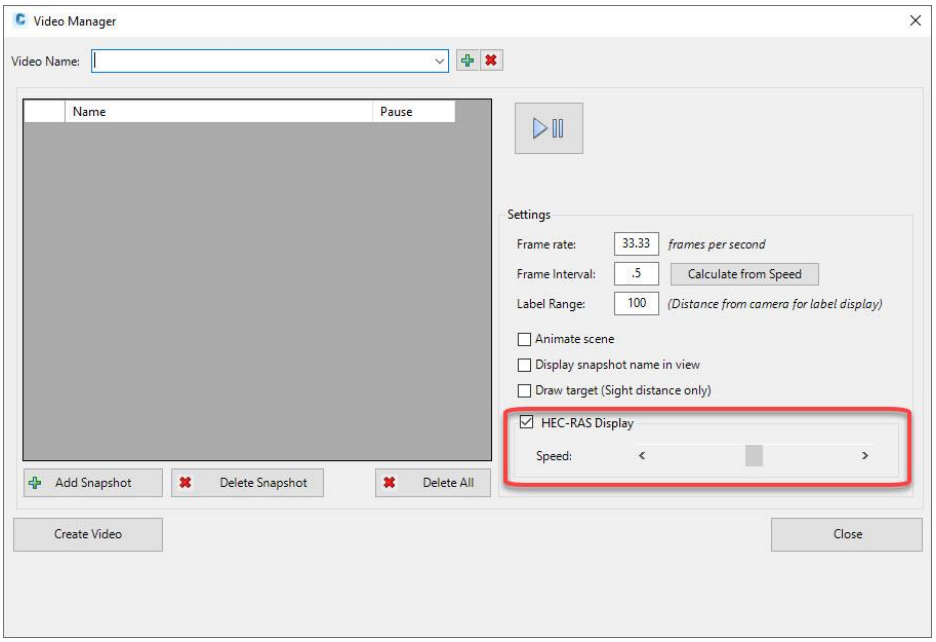
    You can click the play/pause button to  see the development of flooding over time and assess, in 3D, how the flooding may impact downstream properties and infrastructure.





Video Output

From the Outputs tab, click on  Record Video to create an animated video file (.avi) of the water.



Adding the Flood Extents Linework into your Drawing

You will need to create an export file from HEC-RAS in order to import it into your drawing as linework.

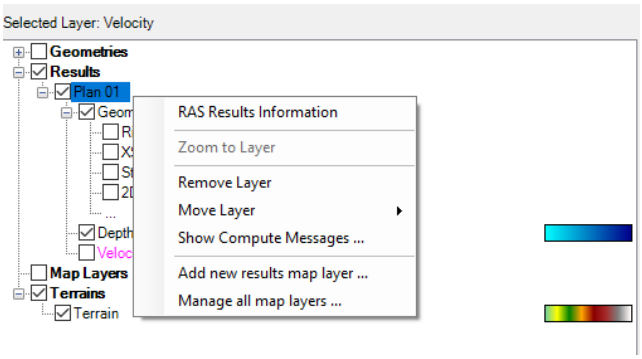
Since HEC-RAS interfaces with geospatial systems, you will be generating a geospatial file (ESRI .shp) describing the linework. This needs to then be imported into your drawing.

Civil 3D and Map 3D include tools to read .shp files and create objects in the drawing from the file/s. AutoCAD and BricsCAD do not natively have functionality to interface with geospatial files such as .shp. There are third party applications, such as Spatial Manager, that deliver functionality to import, export and interrogate geospatial data.

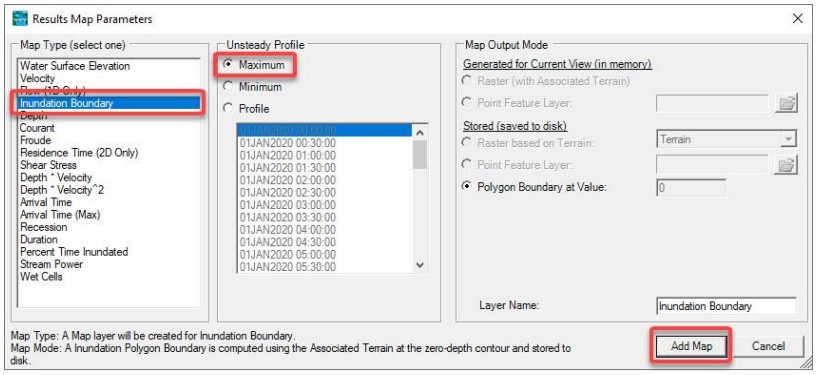
Output from HEC-RAS

To generate the .shp file of the water extents you need to:

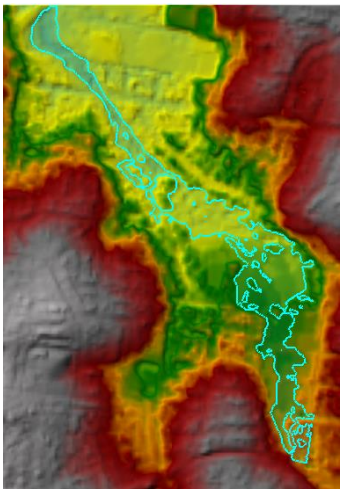
1. Open HEC-RAS and go into RAS Mapper. You should see all the results in RAS mapper from your analysis.



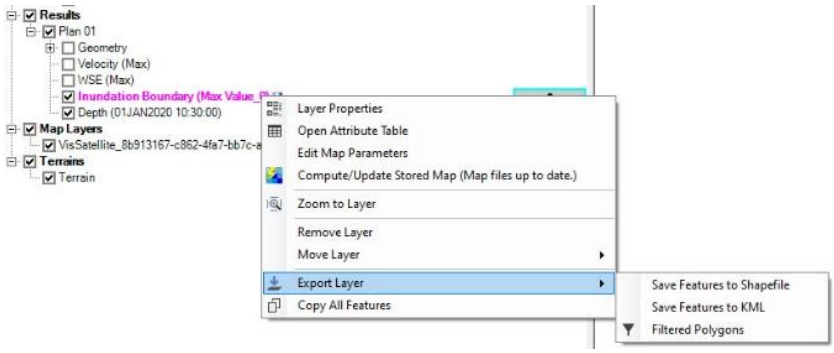
2. In the Selected Layer list, right click on the Results Layer (Plan 01) and select **Add new results map layer…** The following form will display:



3. You can create a new layer displaying any of the Map Type information, at any particular time. To generate the maximum inundation
    a. Select the Map type **Inundation Boundary**
    b. Under Unsteady Profile, click on **Maximum**
    c. Confirm to create a Polygon Boundary
    d. Click on **Add Map**
    e. Right click on the new layer and select **Compute/Update Stored Map**.
    The new layer shows the inundation extents



4. Right click on the new layer and select Export Layer > Save Feature to Shapefile (or KMZ if viewing in Google Earth)



5. Save the file and then it can be imported into your drawing.

Import into the Drawing

There are different options for importing the .shp file into the drawing, depending on the CAD platform.

AutoCAD/BricsCAD users

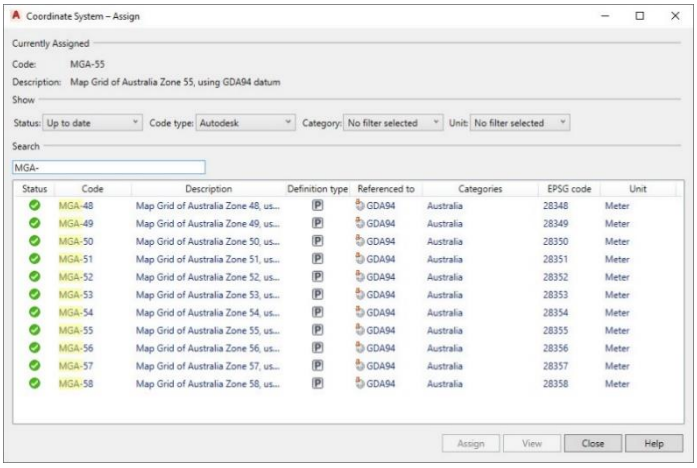
If using AutoCAD or BricsCAD you have some options to include the output .shp file in the drawing:

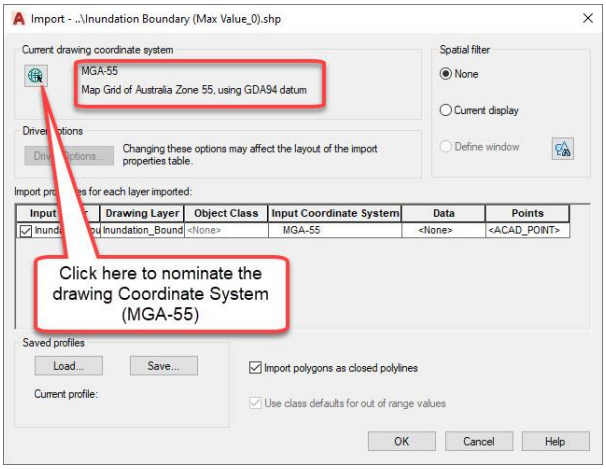
1. Obtain Spatial Manager. After installation, additional geospatial tools are added to the ribbon to enable reading and writing ESRI .shp files (as well as numerous other formats) directly in the drawing
2. Download and install QGIS. This is a free standalone geospatial application allowing users to add a layer of data using the .shp file and saving that data out to a .dxf format (which can be opened in AutoCAD/BricsCAD)

Civil 3D/Map 3D Users

If using Civil 3D (or AutoCAD Map 3D), the command MAPIMPORT can be used to include the linework from the file.

Note for Civil 3D/Map 3D users: If not already set, you may wish to specify the Coordinate System of the drawing by either clicking the button or typing MAPCSASSIGN in the Civil 3d command line. Use the Search function and then click the Assign button. For this example, we are using MGA-55.





After importing the layer, polylines (or hatching) will be included in the drawing: