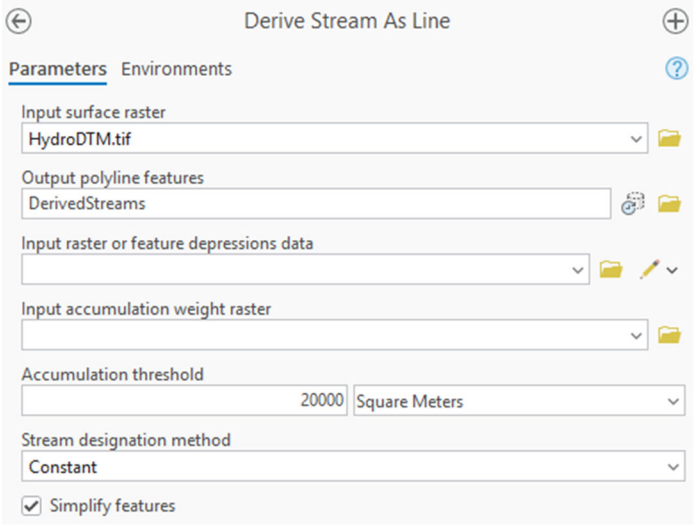
HYDRO MODELING TOOLS ARCGIS PRO

This document describes the general process to develop a hydrology model representing culverts crossing main roads for your area of interest (AOI). This workflow uses a succession of ArcGIS Pro tools that need to be executed in order to obtain a correct final result.

Before starting development of a hydrology model for your AOI, spend some time getting to know the tools that will be used. The first tool that will be used is `Derive Stream as Line`. This tool creates a polyline feature class representing all possible streams found in the area. The next tool used is `Locate Depression`. This geoprocessing tool creates a polygon feature class representing depression areas detected in the input digital terrain model (DTM). The third tool is `Generate Breach Line` which identifies missing upstream connection points and creates depression breach lines. It uses output from the locate depression layer to identify areas that need to be connected by culverts to facilitate water flow. Finally, the last tool is `Adjust Raster to Stream`. This tool modifies the DTM initially used to "carve out" where culverts are predicted to be located.

Let's start the hydrology model:

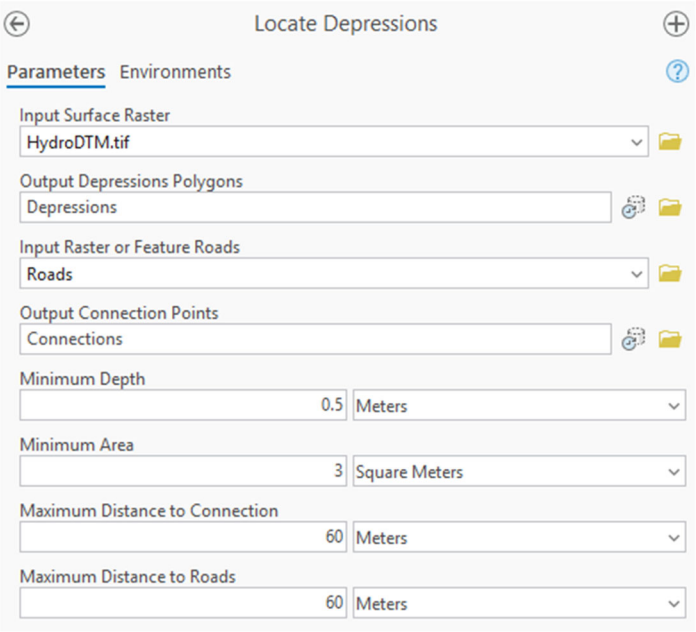
1. Before getting starting the geoprocessing workflow, we need to make sure we have all the necessary input data.
    a. DTM (NOTE: ideally, a pit-filled bare earth model derived from current lidar will be available)
    b. Road line feature class
    c. AOI polygon feature class. (NOTE: For better results, use the AOI polygon to clip the DTM raster layer and roads line layer. This will also allow shorter processing time when the tools are used).
2. Using the geoprocessing pane, find and open the `Derive Stream as Line` tool.
    a. The DTM will be used as the `Input surface raster`.
    b. Provide a meaningful name for the `Output polyline features`
    c. `Input raster or feature depressions data` can be left blank
    d. `Input accumulation weight raster` can be left blank
    e. `The Accumulation threshold` should not be left blank. Very good results have been obtained using 20,000 m^2. The value needs to reflect the complexity of the terrain in the AOI and the size of the contributing watershed basin. If the threshold is equal to 20,000 m^2 (20 hectares), only raster cells receiving 20 or more hectares of upstream flow will be used to define a stream.
    f. `Stream designation method` is constant
    g. `Simplify features` is left checked.
    h. The completed tool dialog should look similar to the following.



i. Click Run.

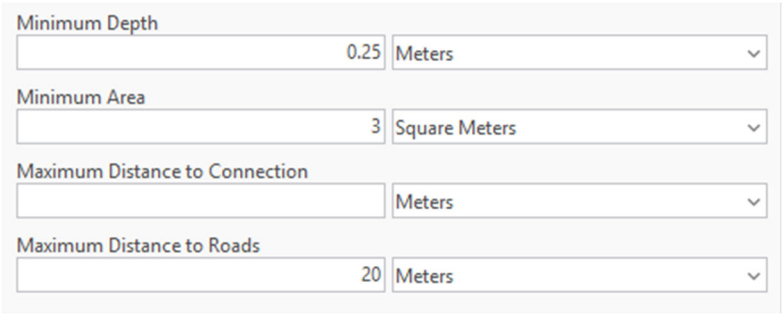
Processing will take some time and is a function of the size of the AOI and level of terrain relief within the AOI. Processing a 250 mi^2 area in eastern Idaho required six minutes to complete.

3. Take a few minutes to examine the results relative to the terrain, various base maps (topographic and aerial imagery), as well as the roads network.
4. When completed, open the `Locate Depressions` tool from the geoprocessing pane.
   a. Complete the tool dialog as shown below.



   b. The `Input Surface Raster` is the DTM
   c. `Output Depression Polygons` is a new polygon layer with the name of choice. This is a feature class showing where water is most likely to accumulate.
   d. `Input raster or feature roads` is the road layer for the AOI (line or polyline)
   e. `Output Connection Points` is a point feature given a name of your choice

f. `Minimum Depth` determines or defines the depth of depressions. For this scenario it will be 0.5m. This parameter determines the *minimum* depth of depressions.
g. `Minimum Area` is set here to 3 m^2. This parameter determines the *minimum* area for the depressions. Minimum depth and minimum area effectively work together to define "depressions".
h. `Maximum Distance to Connection` is 60 meters for this case study. This parameter determines the *maximum* distance between stream connections.
i. `Maximum Distance to Roads` is 30 meters for this case study. This parameter determines the maximum distance between a stream and road.
j. Click Run. Using the settings shown above for the 250 mi^2 AOI required 20 minutes to complete. A second iteration using the parameters shown below required only 7 minutes to completed with results being similar.

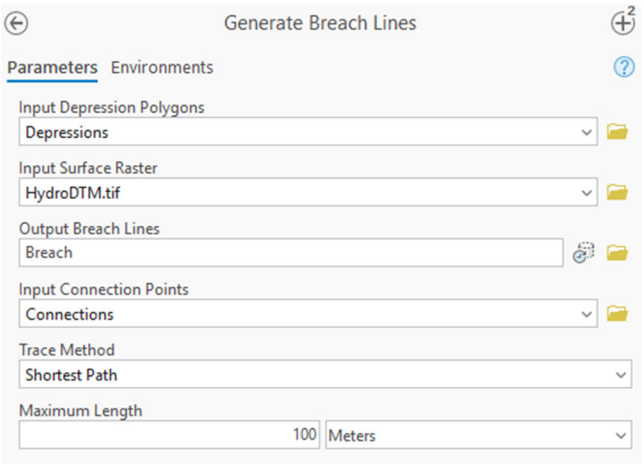


The last four variables are optional. You can use the default values, leave the values blank, or experiment with values depending on the needs of the user and the desired result. It is recommended to experiment with values and use those that best model reality.

5. Review the results and if necessary, run the locate depressions tool again using different minimum/maximum parameters.
6. Once you are satisfied with the results, open the `Generate Breach Lines` geoprocessing tool.

This tool will look for missing upstream connection points and create depression breach lines. It uses the locate depression layer output to determine areas that need to be connected by culverts to facilitate water flow.

a. `Input Depression Polygons` is the output polygon layer created by the previous locate depressions tool.
b. `Input Surface Raster` is the same DTM used in both previous tools.
c. `Output Breach Lines` is a line feature class with name of your choice
d. `Input Connection Points` is the point feature class created in the previous locate depressions tool output (i.e., connection points).
e. `Trace Method` is set to the `Shortest Path`. You are encouraged to experiment with other options.
f. `Maximum Length` is 100 meters (up to the user and needs)

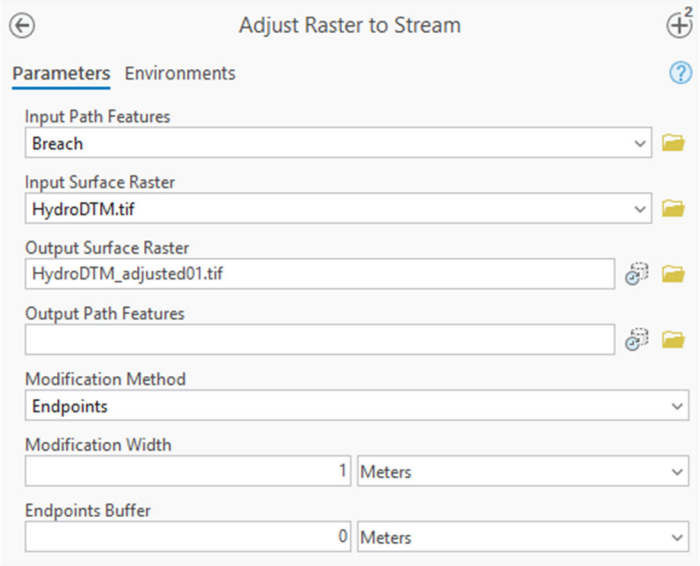


g. Click Run.

7. Review the resulting breach lines layer. The output will look similar to the figure below although results may vary broadly.



8. Last, open the `Adjust Raster to Stream` geoprocessing tool. This tool will create a new modified DTM by "carving out" where culverts are predicted to be located (or could be located to facilitate stream flow).
    a. `Input Path Features` is a line feature from the breach line tool output
    b. `Input Surface Raster` is the DTM
    c. `Output Surface Raster` is a new DTM created with "culverts/trenches" crossing the road features.
    d. `Output Path Features` can be left blank
    e. `Modification Method` is set to the default endpoints
    f. `Modification Width` is set to the default 1 meter (this parameter determines the width of the trench for the culvert on the DTM).
    g. `Endpoint buffer` is also set to the default 0 meter.



h. Click Run.

9. Once processing is complete, display the new DTM as a hillshade model. Here's how:
   a. Select the DTM layer from the Contents pane
   b. Click the Imagery menu
   c. From the Imagery ribbon, click Raster Functions

      A new `Raster Functions` pane will open on the right side of the screen.

   d. Expand the `Surface` section at the bottom of the pane.
   e. Click Hillshade
   f. From the `Hillshade Properties` pane, simply select the Raster layer from the drop-down list.
   g. Click the `Create New Layer` button.

10. The figure below shows before and after results. These results should be similar to your results.



END