Overview of batch processing:

In ArcGIS, batch processing means to execute a single tool multiple times with different inputs without your intervention. Suppose you had several datasets with the projection undefined. Instead of opening the Define Projection tool many times and defining the projection for each dataset, you can use the batch option and fill out a simple form once to have the Define Projection tool process automatically.

To determine which tools have a batch option, right-click the tool in ArcToolbox. For example, to project four individual elevation grids you can use the Define Projection tool in batch mode.

Using the Define Projection Tool in Batch mode

1. Open Arc Toolbox.
2. Expand the Data Management Tools and Projections and Transformations tool boxes.
3. Right-click the Define Projection Tool and click Batch.
4. Click the plus sign [+] three times to add three more rows.
5. Right click in the empty box below the column heading Input Dataset or Feature Class and click Browse.
6. Navigate to your local directory and select your first elevation grid. Continue to the next rows and add the second dataset. [Alternatively, click and drag the datasets from the Table of Contents in the rows.
7. Right click in the box below the column heading Coordinate System and click Open.
8. In the Define Projection dialog box click the button on the right-hand side to select a coordinate system. [In this example, the elevation data came with a .prj file which stated that the projection is UTM NAD83 Zone 17N.]
9. In the Spatial Reference Properties window, open the Projected Coordinate Systems folder, the UTM Folder, the NAD 1983 folder and select NAD 1983 UTM Zone 17N. Click OK, and OK again to close the Define Projection dialog box.
10. Right click in the box containing the new coordinate system and select the option **Fill**. The row will automatically be populated with the same coordinate system.

11. To execute the batch processing click **OK**.

**Why is Batch Processing Useful?**

One classic scenario for batching is using the **Clip** tool to clip multiple input datasets to a predetermined study area polygon. Suppose you had 20 datasets to clip. Obviously, you could open the **Clip** tool dialog 20 times, fill out the parameters, and execute. By the time you're done, a half-hour may have passed. By using the batch option, you can spend just a few minutes filling out a simple form (the batch grid) specifying the 20 input datasets, then have **Clip** automatically execute 20 times, freeing you to do other work.