
Command Reference: ReadTimeSeriesList()

Read one or more time series using location identifiers from a table

Version 10.21.00, 2013-05-17

The `ReadTimeSeriesList()` command reads one or more time series using location identifiers from a table, an example of which is shown below as a comma-separated value file:

```
# Example list file.  Comments start with the # character.
# Column headings can be specified in the first non-comment row using quotes.
"Structure ID","Structure Name"
500501,Ditch 501
500502,Ditch 502
# Invalid ID (see IfNotFound parameter)
509999,Ditch 9999
```

The command typically is used when reading time series from a single source and can streamline processing in the following situations:

- A list of identifiers may have been generated from a database query
- A list of identifiers may have been extracted from a model data set

TSTool uses the location identifiers in the table with the command parameters and internally creates a list of time series identifiers. The time series are of the standard form:

```
Location.DataSource.DataType.Interval[.Scenario]~DataStore[~InputName]
```

where the brackets indicate optional information. TSTool then queries each time series, which can be processed further by other commands. See also the `ReadTimeSeries()` command, which performs essentially the same functionality but only reads one time series.

Although it is possible to specify a datastore (or “input type”) that reads from files by also using the `InputName`, this is not generally recommended because the `ReadTimeSeriesList()` command can only specify one input file name and the file will be reopened for each time series read. Instead, read commands for specific file formats should be used because these commands are typically optimized to read multiple time series from the files. Use the `SetInputPeriod()` command to set the period to read.

The following dialog is used to edit the command and illustrates the syntax of the command.

Edit ReadTimeSeriesList() Command

Read a list of time series using location identifiers in a table.
 The information specified below is used with the location identifiers to create time series identifiers, which are then used to read the time series.
 The time series identifiers (TSIDs) are of the form:
 LocationID.DataSource.DataType.Interval.Scenario~DataStore~InputName
 The term "DataStore" is used generically to mean a database, web service, or file supplying time series data (also called "Input Type" elsewhere).
 Use the SetInputPeriod() command to specify the period to read.

Table ID: Required - table containing list of location IDs.

Location ID column: Required - name of column containing location IDs.

Data source: Optional or required depending on datastore.

Data type: Optional or required depending on datastore.

Data interval: Required - data interval (time step) for time series.

Scenario: Optional.

Datastore: Required - needed to identify input database, file, etc.

Input name: Optional - file name if required for datastore.

If time series not found?: Required - how to handle time series that are not found.

Default units: Optional - units when IfNotFound=Default.

Command:
`ReadTimeSeriesList (TableID="StationList",LocationColumn="TSID",DataSou
 rce="NWIS",DataType="00060-00003",Interval="Day",DataStore="DateValue"
 ,InputName="Data/testdata.dv",IfNotFound=Warn)`

ReadTimeSeriesList() Command Editor

The command syntax is as follows:

```
ReadTimeSeriesList (Parameter=Value, ...)
```

Command Parameters

Parameter	Description	Default
TableID	The identifier for the table that provides the list of location identifiers.	None – must be specified.
LocationColumn	The column in the table containing the location identifiers to use in time series identifiers.	None – must be specified.
DataSource	The data source in the time series identifier. For example, if using the State of Colorado's HydroBase, USGS indicates that data are from the United States Geological Survey. See the datastore and input type appendices for more information on available data types.	May or may not be required, depending on the datastore or input type. Refer to the input type appendices.
DataType	The data type in the time series identifier.	Usually required. Refer to the

Parameter	Description	Default
	For example, if using the State of Colorado's HydroBase, DivTotal is used for diversion totals. See the input type appendices for more information on available data types.	datastore and input type appendices.
Interval	Data interval in the time series identifier, using standard values such as 15Minute, 6Hour, Day, Month, Year.	None – must be specified.
Scenario	Scenario in the time series identifier.	Usually not required.
DataStore	The data store (or input type) in the time series identifier. Refer to the datastore and input type appendices or the TSTool main GUI for options.	None – must be specified.
InputName	The input name in the time series identifier, when a file name is required.	
IfNotFound	Indicates how to handle missing time series, one of: <ul style="list-style-type: none"> Warn – generate fatal warnings and do not include in output. Ignore – generate non-fatal warnings and do not include in output. Default – generate non-fatal warnings and create empty time series for those that could not be found. This requires that a SetOutputPeriod() command be used before the command to define the period for default time series. 	Warn
DefaultUnits	Default units when IfNotFound=Default.	Blank – no units.

A sample command file to process monthly diversion data from the State of Colorado's HydroBase database is as follows:

```
# Read monthly diversion total from HydroBase for the structures in the list
# file. The data source is set to DWR because data source is saved in
# HydroBase.
ReadTimeSeriesList (TableID="Diversions.csv",LocationColumn="WDID",
    DataSource=DWR,DataType=DivTotal,Interval=Month,InputType=HydroBase,
    IfNotFound=Default)
```