Command Reference: ReadTimeSeriesList()

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

Version 10.23.00, 2013-09-10

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 (tables can also be read from Excel or a datastore):

```
# 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

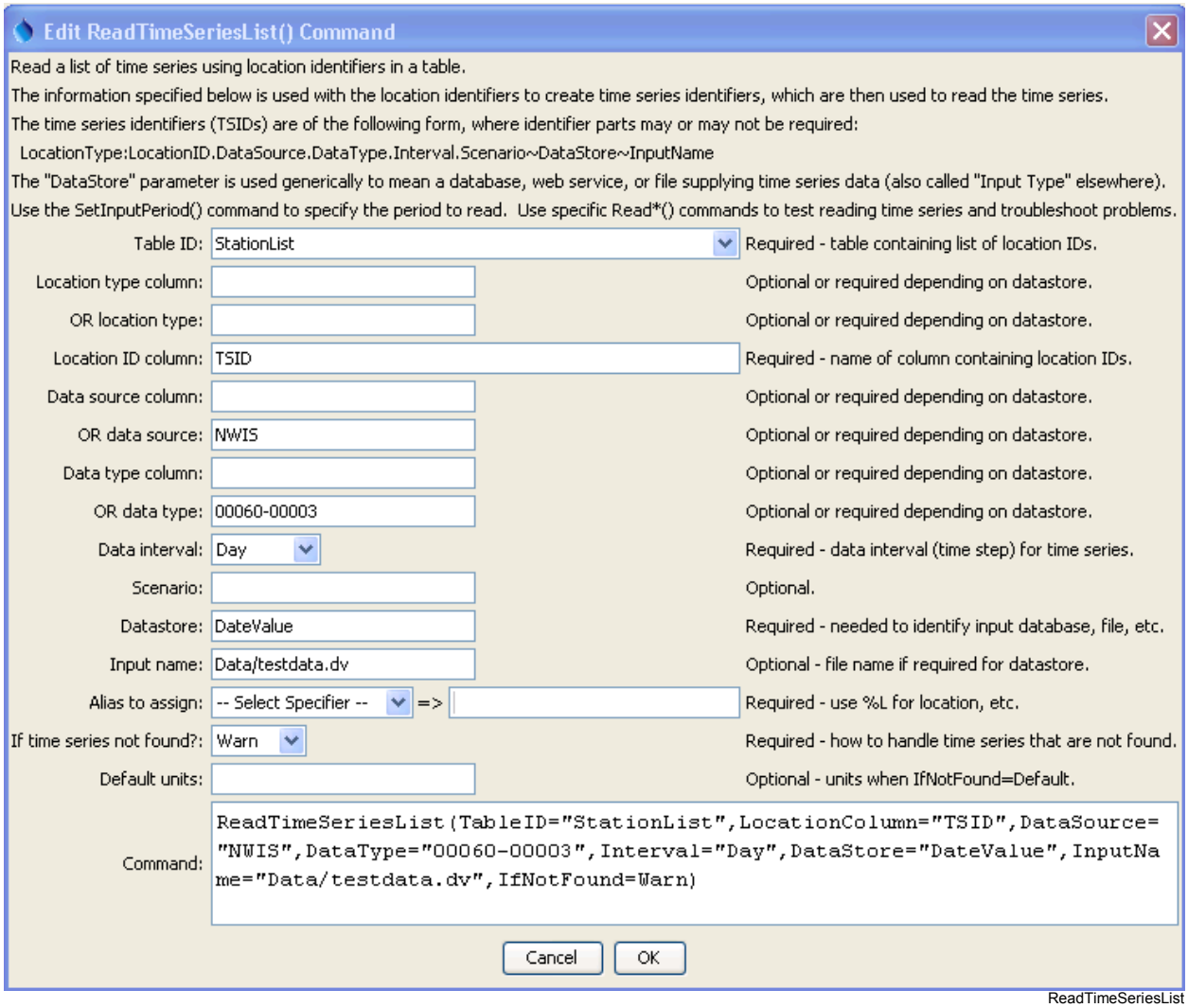
TSTool uses the location identifiers in the table with the command parameters and internally creates a list of time series identifiers. The time series are of the standard form (information in brackets is optional for basic use):

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

TSTool then queries each time series using the time series identiifier See also the `ReadTimeSeries()` command, which performs essentially the same functionality but only reads one time series. Refer to the appendices for each datastore and file input type to understand specific time series identifier conventions.

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 typically are 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.



ReadTimeSeriesList

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.
`LocationType Column`	The column in the table containing the location type to use in time series identifiers. Specify `LocationTypeColumn` or `LocationType`.	May or may not be required, depending on the datastore or input type.
`LocationType`	The location type in the time series	May or may not be required,

Parameter	Description	Default
	identifier. Specify `LocationTypeColumn` or `LocationType`.	depending on the datastore or input type.
`LocationColumn`	The column in the table containing the location identifiers to use in time series identifiers.	None – must be specified.
`DataSource Column`	The column in the table containing the data source to use in time series identifiers. Specify `DataSourceColumn` or `DataSource`.	May or may not be required, depending on the datastore or input type.
`DataSource`	The data source(s) in the time series identifier, separated by commas. For example, if using the State of Colorado's HydroBase, `USGS` indicates that data are from the United States Geological Survey and `DWR` are from the Division of Water Resources. If multiple data sources are specified, each will be tried until a time series is found. This is enabled because sometimes gages change ownership. Specify `DataSourceColumn` or `DataSource`.	May or may not be required, depending on the datastore or input type
`DataType Column`	The column in the table containing the data type to use in time series identifiers. Specify `DataTypeColumn` or `DataType`.	Data type is often required
`DataType`	The data type in the time series identifier. For example, if using the State of Colorado's HydroBase, `DivTotal` is used for diversion totals. Specify `DataTypeColumn` or `DataType`.	Data type is often required
`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.	Generally only required when reading from a file.
`IfNotFound`	Indicates how to handle missing time series, one of: • `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	`Warn`

Parameter	Description	Default
	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.	
`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)
```