Literal View Options

This topic covers literal view options, which allow you to directly specify the request/message body.

The Literal view contains the following options:

- Text
- File
- Data Source

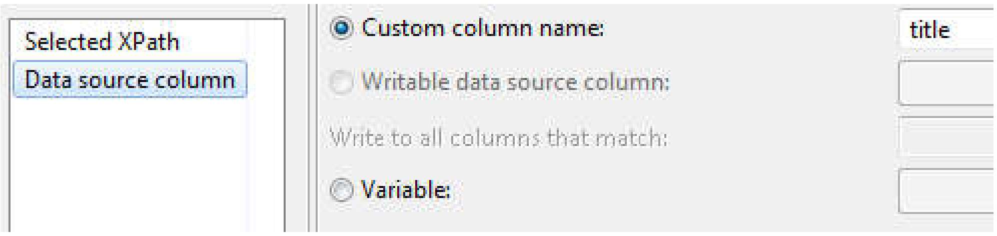
Text

Use this option if you want to type or copy a message into the UI. Select the appropriate **MIME type**, then enter the message in the text field below the **Text** radio button.

- You can parameterize attributes and simple elements that contain text content by accessing data source values (from data sources or data banks) by using the {$} syntax. If parameterizing with a column from a repository data source, be sure to reference a primitive column or a primitive list column, not a record list column.

  For example, use ${books:publisher} to reference the publisher column from the books sheet of an Excel data source. Use ${title} to reference the title column from a data source or the data bank tool.

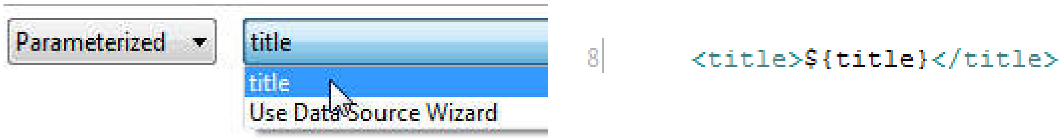
  If you use ${} format variables such as `${value}`, those variables will be resolved before processing. To prevent variables from being processed, escape `${}` variables using a backslash. For example, the value "`\${HOST}`" will be interpreted as "`${HOST}`"; the variable will not be resolved.



Selected XPath
Data source column

● Custom column name: title
○ Writable data source column:
Write to all columns that match:
○ Variable:

  Data source settings from Form views will be carried over into Literal view (and from Literal view to Form views).



Parameterized ▾ | title

```
<title>${title}</title>
```

- You can parameterize complex XML elements from a Parasoft Data Repository data source column, which may reference zero, one, or multiple hierarchical record values. For details, see “Parameterizing with Hierarchical Data from a Data Repository Data Source”, page 1110.

- You can beautify XML or JSON by right-clicking within the **Text** field and selecting **Beautify** from the shortcut menu.
- For SOAP, you can choose elements from a schema for both the body and the header by right-clicking within the **Text** field and selecting **Import Schema Element**. After selecting this option, a dialog appears from which you can load declared elements from a schema location. After loading elements, you can select multiple elements for the SOAP header. Once you click **OK**, a SOAP Envelope will be created based on the chosen element definitions.

Parameterizing with Hierarchical Data from a Data Repository Data Source

You can parameterize complex XML and JSON elements from a Parasoft Data Repository data source column, which may reference zero, one, or multiple hierarchical record values. To achieve this, you add a Parasoft Repository Data source as described in “Working with Large, Hierarchical Data Sets”, page 613 (specifically, “Creating a Repository Data Source”, page 635), then you use "ParasoftColumn" to specify which Data Repository column should be used to parameterize that level in the hierarchy of the message. See “Hierarchical Parameterization with Literal View”, page 639 for details.

File

If you already have a file that specifies the message, use this option to indicate the location of that file.

- Check the **Persist as Relative Path** option if you want the path to this file to be saved as a path that is relative to the current configuration file. Enabling this option makes it easier to share tools across multiple machines. If this option is not enabled, the path to this file will be saved as an absolute path.
- Check the **Resolve Environment Variables** option if your file is text and has environment variables that you want resolved. Otherwise, leave this option disabled to improve performance. This option must be disabled if file is binary.

Data Source

If you already have a file containing the various messages you want to use, setup a File data source for that file (as described in “Configuring a File Data Source”, page 374), then use the **Data source name column** box to indicate which column of data you want to use.

2. Use the **Rows** controls to indicate the range of rows you want to use.
   - If you only want to use selected rows, click the **Range** button, then enter the desired range (assuming a one-based index) by typing values into the **From** and **To** fields. For example, to use only the first 10 rows, enter 1 in the **From** field and 10 in the **To** field. To use only the fifth row, enter 5 in the **From** field and 5 in the **To** field.
3. Choose the desired data sources from the **Available** box and click the **Add** button to add them to the **Selected** box.
   - The **Available** box contains all of the data sources added to the test suite. After selecting and adding the desired data sources to the **Selected** box, the column names contained in the added data sources display in the **Columns** box.

Configuring a Data Group Data Source that Lets You Switch Between Multiple Data Sources

A data group can include a number of similar data sources that have at least one column in common; it allows you to select which data group should be applied at any given time. This is especially useful if you want to dynamically specify different data sources for the same tool. You can quickly switch which data source is used at runtime without having to edit tools or data sources. To use data groups, you group together data sources with shared columns and then specify which data source should be active.

For a discussion of how data groups compare to aggregate data sources, see “Understanding Data Groups and Aggregate Data Source”, page 368.

To combine multiple data sources into a data group:

1. (Optional) Change the data source label in the **Name** field of the Data Source configuration options.
2. Use the **Rows** controls to indicate the range of rows you want to use.
   - If you only want to use selected rows, click the **Range** button, then enter the desired range (assuming a one-based index) by typing values into the **From** and **To** fields. For example, to use only the first 10 rows, enter 1 in the **From** field and 10 in the **To** field. To use only the fifth row, enter 5 in the **From** field and 5 in the **To** field.
3. Choose the desired data sources from the **Available** box and click the **Add** button to add them to the **Selected** box.
   - The **Available** box contains all of the data sources added to the test suite.
   - If a data source does not have any columns in common with the currently selected data sources, it cannot be added.
   - If adding a data source will reduce the number of shared columns across data source, a warning will be displayed.
   - After you select and add the desired data sources to the **Selected** box, the common columns that occur in *all* the selected data sources will be shown in the **Columns** box.
4. Under **Active Data Source**, specify which data source you want to be active. If you want to specify the active data source "on the fly" at runtime, you can configure the Active Data Source field with an environment variable using the ${var_name} notation, then specify the desired environment at runtime (from the command line interface or the UI—see “Configuring Testing in Different Environments”, page 394 for details).

Configuring a File Data Source

To configure a File data source:

1. (Optional) Change the data source label in the **Name** field of the Data Source configuration options.
2. Use the **Rows** controls to indicate the range of rows you want to use.
    - If you only want to use selected rows, click the **Range** button, then enter the desired range (assuming a one-based index) by typing values into the **From** and **To** fields. For example, to use only the first 10 rows, enter 1 in the **From** field and 10 in the **To** field. To use only the fifth row, enter 5 in the **From** field and 5 in the **To** field.
3. Specify the file or directory to import files from. All of the files available in the specified location will display in the table. Right-click options allow you to cut, copy, and paste values in as well.
    - To filter which files are used by the File Data Source, enter a string in the **File Filter** field. For example:
        - ***** = wild card for any string
        - ***.*** = all files (this is the default)
        - ***.txt** = all text files
        - **data*** = all files whose files names begin with "data"
        - **data*.txt** = all text files whose files names begin with "data"
        - ***data*** = all files whose file names contain the string "data" somewhere
4. Enable the **Refresh Dynamically Based on Last Import** option if you want the file data source to use all files that are in the import directory at the time the file data source is used. If you do not enable this option, the file data source will only use the files that are listed in the table in the file data source—even if the import directory has additional files. This option must be enabled in the following cases to ensure the portability of the .tst file:
    - When the .tst file needs to be executed on multiple machines.
    - When the .tst file will be used in a load test that uses multiple Load Test generator machines.

At runtime, SOAtest will use the contents of each file as a data source value.

Configuring a Writable Data Source

To configure a writable data source that captures runtime data for reuse in subsequent tests:

1. (Optional) Change the data source label in the **Name** field of the Data Source configuration options.
2. Use the **Rows** controls to indicate the range of rows you want to use.
    - If you only want to use selected rows, click the **Range** button, then enter the desired range (assuming a one-based index) by typing values into the **From** and **To** fields. For example, to use only the first 10 rows, enter 1 in the **From** field and 10 in the **To** field. To use only the fifth row, enter 5 in the **From** field and 5 in the **To** field.
3. Specify your preferred writing mode for writing to the data source. You can choose one of the following options:
    - **Set-Up test mode> Append:** Choose this option if you want set-up tests to append new data after any previously written values. This is the default mode. *Only set-up tests can write to the data source in this mode.*
    - **Standard test mode> Overwrite:** Choose this option if you want standard tests (non set-up tests) to overwrite any previously written values with new data. *Only standard tests can write to the data source in this mode*.